

CNC Programming Handbook

Third Edition

A Comprehensive Guide to Practical CNC Programming

Peter Smid

Industrial Press, Inc.

989 Avenue of the Americas

New York, NY 10018, USA

<http://www.industrialpress.com>

Library of Congress Cataloging-in-Publication Data

Smid, Peter

CNC Programming Handbook: Comprehensive Guide to Practical CNC Programming/
Peter Smid.

p. cm.

ISBN 0-8311-3347-3

1. Machine-tools--Numerical Control--Programming--Handbooks, manuals, etc.,...I.
Title.

TJ1189 .S592 2000

621.9'023--dc21

00-023974

Third Edition

CNC Programming Handbook

Industrial Press, Inc.
989 Avenue of the Americas
New York, NY 10018, USA

Copyright © 2007. Printed in the United States of America.

All Rights Reserved.

This book or parts thereof may not be reproduced, stored in a retrieval
system, or transmitted in any form without the permission of the publishers.

TABLE OF CONTENTS

1 - NUMERICAL CONTROL	1	Quadrants	16
DEFINITION OF NUMERICAL CONTROL	1	Right Hand Coordinate System	17
NC and CNC Technology	1	MACHINE GEOMETRY	17
CONVENTIONAL AND CNC MACHINING	2	Axis Orientation - Milling	17
NUMERICAL CONTROL ADVANTAGES	2	Axis Orientation - Turning	18
Setup Time Reduction	3	Additional Axes	18
Lead Time Reduction	3	5 - CONTROL SYSTEM	19
Accuracy and Repeatability	3	GENERAL DESCRIPTION	20
Contouring of Complex Shapes	3	Operation Panel	20
Simplified Tooling and Work Holding	3	Screen Display and Keyboard	21
Cutting Time and Productivity Increase	4	Handle	22
TYPES OF CNC MACHINE TOOLS	4	SYSTEM FEATURES	22
Mills and Machining Centers	4	Parameter Settings	22
Lathes and Turning Centers	5	System Defaults	23
PERSONNEL FOR CNC	5	Memory Capacity	24
CNC Programmer	5	MANUAL PROGRAM INTERRUPTION	25
CNC Machine Operator	6	Single Block Operation	25
SAFETY RELATED TO CNC WORK.	6	Feedhold	25
2 - CNC MILLING	7	Emergency Stop	25
CNC MACHINES - MILLING	7	MANUAL DATA INPUT - MDI	26
Types of Milling Machines	7	PROGRAM DATA OVERRIDE	26
Machine Axes	8	Rapid Motion Override	26
Vertical Machining Centers	8	Spindle Speed Override	27
Horizontal Machining Centers	10	Feedrate Override	27
Horizontal Boring Mill	10	Dry Run Operation	27
Typical Specifications	10	Z-Axis Neglect	28
3 - CNC TURNING	11	Manual Absolute Setting	28
CNC MACHINES - TURNING	11	Sequence Return	28
Types of CNC Lathes	11	Auxiliary Functions Lock	28
Number of Axes	11	Machine Lock	28
AXES DESIGNATION	11	Practical Applications	29
Two-axis Lathe	12	SYSTEM OPTIONS	29
Three-axis Lathe	12	Graphic Display	29
Four-axis Lathe	13	In-Process Gauging	30
Six-axis Lathe	13	Stored Stroke Limits	30
FEATURES AND SPECIFICATIONS	13	Drawing Dimensions Input	30
Typical Machine Specifications	13	Machining Cycles	30
Control Features	14	Cutting Tool Animation	30
4 - COORDINATE GEOMETRY	15	Connection to External Devices	30
REAL NUMBER SYSTEM	15	6 - PROGRAM PLANNING	31
RECTANGULAR COORDINATE SYSTEM.	15	STEPS IN PROGRAM PLANNING.	31
Axes and Planes	16	INITIAL INFORMATION	31
Point of Origin	16	CNC MACHINE FEATURES	31
		Machine Type and Size	31
		Control System	32
		PART COMPLEXITY	32
		MANUAL PROGRAMMING	32

Disadvantages	32
Advantages	32
CAD/CAM AND CNC	33
Integration	33
Future of Manual Programming	33
TYPICAL PROGRAMMING PROCEDURE	33
PART DRAWING	34
Title Block	34
Dimensioning	34
Tolerances	36
Surface Finish	36
Drawing Revisions	36
Special Instructions	37
METHODS SHEET	37
MATERIAL SPECIFICATIONS	37
Material Uniformity	37
Machinability Rating	37
MACHINING SEQUENCE	38
TOOLING SELECTION	38
PART SETUP	39
Setup Sheet	39
TECHNOLOGICAL DECISIONS	39
Cutter Path	39
Machine Power Rating	40
Coolants and Lubricants	40
WORK SKETCH AND CALCULATIONS	41
Identification Methods	41
QUALITY IN CNC PROGRAMMING	42
 7 - PART PROGRAM STRUCTURE	 43
BASIC PROGRAMMING TERMS	43
Character	43
Word	43
Block	44
Program	44
PROGRAMMING FORMATS	44
WORD ADDRESS FORMAT	44
FORMAT NOTATION	45
Short Forms	45
Milling System Format	46
Turning System Format	46
Multiple Word Addresses	47
SYMBOLS IN CNC PROGRAMMING	47
Plus and Minus Sign	48
TYPICAL PROGRAM STRUCTURE	48
Program Structure Benefits	49
PROGRAM HEADER	50
Tool Comments	50
 8 - PREPARATORY COMMANDS	 51
DESCRIPTION AND PURPOSE	51

APPLICATIONS FOR MILLING	51
APPLICATIONS FOR TURNING	53
G-CODES IN A PROGRAM BLOCK	54
Modality of G-codes	54
Conflicting Commands in a Block	54
Word Order in a Block	55
GROUPING OF COMMANDS	55
Group Numbers	55
G-CODE TYPES	56
G-Codes and Decimal Point	56
 9 - MISCELLANEOUS FUNCTIONS	 57
DESCRIPTION AND PURPOSE	57
Machine Related Functions	57
Program Related Functions	57
TYPICAL APPLICATIONS	58
Applications for Milling	58
Applications for Turning	58
Special MDI Functions	58
Application Groups	58
M-FUNCTIONS IN A BLOCK	59
Startup of M-Functions	60
Duration of M-Functions	60
PROGRAM FUNCTIONS	60
Program Stop	60
Optional Program Stop	61
Program End	62
Subprogram End	62
MACHINE FUNCTIONS	62
Coolant Functions	62
Spindle Functions	63
Gear Range Selection	64
Machine Accessories	64
 10 - SEQUENCE BLOCK	 65
BLOCK STRUCTURE	65
Building the Block Structure	65
Block Structure for Milling	65
PROGRAM IDENTIFICATION	66
Program Number	66
Program Name	66
SEQUENCE NUMBERS	67
Sequence Number Command	67
Sequence Block Format	67
Numbering Increment	68
Long Programs and Block Numbers	68
END OF BLOCK CHARACTER	68
STARTUP BLOCK OR SAFE BLOCK	69
PROGRAM COMMENTS	70
CONFLICTING WORDS IN A BLOCK	70
MODAL PROGRAMMING VALUES	71
EXECUTION PRIORITY	72

11 - INPUT OF DIMENSIONS	73	MAXIMUM FEEDRATE	95
IMPERIAL AND METRIC UNITS	73	Maximum Feedrate Considerations	95
Comparable Unit Values	74	FEEDHOLD AND OVERRIDE	95
ABSOLUTE AND INCREMENTAL MODES	74	Feedhold Switch	95
Preparatory Commands G90 and G91	75	Feedrate Override Switch	95
Absolute Data Input - G90	76	Feedrate Override Functions	96
Incremental Data Input - G91	76	E ADDRESS IN THREADING	96
Combinations in a Single Block	76	14 - TOOL FUNCTION	97
DIAMETER PROGRAMMING	77	T-FUNCTION FOR MACHINING CENTERS	97
MINIMUM MOTION INCREMENT	77	Tool Storage Magazine	97
FORMAT OF DIMENSIONAL INPUT	77	Fixed Tool Selection	98
Full Address Format	78	Random Memory Tool Selection	98
Zero Suppression	78	Registering Tool Numbers	98
Decimal Point Programming	79	Programming Format	99
Input Comparison	80	Empty Tool or Dummy Tool	99
CALCULATOR TYPE INPUT	80	TOOL CHANGE FUNCTION - M06	99
12 - SPINDLE CONTROL	81	Conditions for Tool Change	99
SPINDLE FUNCTION	81	AUTOMATIC TOOL CHANGER - ATC	100
Spindle Speed Input	81	Typical ATC System	100
DIRECTION OF SPINDLE ROTATION	81	Maximum Tool Diameter	101
Direction for Milling	82	Maximum Tool Length	101
Direction for Turning	82	Maximum Tool Weight	101
Direction Specification	83	ATC Cycle	102
Spindle Startup	83	MDI Operation	102
SPINDLE STOP	84	PROGRAMMING THE ATC	102
SPINDLE ORIENTATION	84	Single Tool Work	102
SPINDLE SPEED - R/MIN	85	Programming Several Tools	103
SPINDLE SPEED - SURFACE	85	Keeping Track of Tools	103
Material Machinability	85	Any Tool in Spindle - Not the First	103
Spindle Speed - Imperial Units	86	First Tool in the Spindle	104
Spindle Speed - Metric Units	86	No Tool in the Spindle	105
CONSTANT SURFACE SPEED	86	First Tool in the Spindle with Manual Change	105
Maximum Spindle Speed Setting	88	No Tool in the Spindle with Manual Change	106
Part Diameter Calculation in CSS	89	First Tool in the Spindle and an Oversize Tool	106
Cutting Speed Calculation	90	No Tool in the Spindle and an Oversize Tool	106
13 - FEEDRATE CONTROL	91	T-FUNCTION FOR LATHES	107
FEEDRATE CONTROL	91	Lathe Tool Station	107
FEEDRATE FUNCTION	91	Tool Indexing	107
Feedrate per Minute	91	TOOL OFFSET REGISTERS	108
Feedrate per Revolution	92	Geometry Offset	108
FEEDRATE SELECTION	92	Wear Offset	109
ACCELERATION AND DECELERATION	92	Wear Offset Adjustment	110
Exact Stop Command	93	The R and T Settings	110
Exact Stop Mode Command	93	15 - REFERENCE POINTS	111
Automatic Corner Override	93	REFERENCE POINT GROUPS	111
Tapping Mode	93	Reference Point Groups Relationship	112
Cutting Mode	94	MACHINE REFERENCE POINT	112
CONSTANT FEEDRATE	94	Return to Machine Zero	113
Circular Motion Feedrates	94	PART REFERENCE POINT	113
		Program Zero Selection	113
		Program Zero - Machining Centers	114
		Program Zero - Lathes	116
		TOOL REFERENCE POINT	116

16 - REGISTER COMMANDS 117

POSITION REGISTER COMMAND	117
Position Register Definition	117
Programming Format	117
Tool Position Setting	118
MACHINING CENTERS APPLICATION	118
Tool Set at Machine Zero	118
Tool Set Away from Machine Zero	118
Position Register in Z-axis	119
Programming Example	119
LATHE APPLICATION	119
Tool Setup	120
Three-Tool Setup Groups	120
Center Line Tools Setup	120
External Tools Setup	121
Internal Tools Setup	121
Corner Tip Detail	121
Programming Example	121

17 - POSITION COMPENSATION 123

DESCRIPTION	123
Programming Commands	123
Programming Format	123
Incremental Mode	124
Motion Length Calculation	124
Position Compensation Along the Z-axis	126
Using G47 and G48	126
Face Milling - One Possible Application	126

18 - WORK OFFSETS 127

WORK AREAS AVAILABLE	127
Additional Work Offsets	128
WORK OFFSET DEFAULT AND STARTUP	128
Work Offset Change	129
Z-axis Application	130
HORIZONTAL MACHINE APPLICATION	131
EXTERNAL WORK OFFSETS	132
LATHE APPLICATIONS	132
Types of Offsets	132
Geometry Offset	132
Wear Offset	133
Tool and Offset Numbers	133
TOOL SETUP	133
Center Line Tools	133
Turning Tools	134
Boring Tools	134
Command Point and Tool Work Offset	134

19 - TOOL LENGTH OFFSET 135

GENERAL PRINCIPLES	135
Actual Tool Length	135
Gauge Line	135
Table Top Face	136

TOOL LENGTH OFFSET COMMANDS	136
Distance-To-Go in Z-axis	136

TOOL LENGTH SETUP	137
On-Machine Tool Length Setting	137
Off-Machine Tool Length Setting	138
Tool Length Offset Amount Register	138

Z-AXIS RELATIONSHIPS.	138
Preset Tool Length	139
Tool Length by Touch Off	139
Using a Master Tool Length	140
G43-G44 Difference	140

PROGRAMMING FORMATS.	141
Tool Length Offset not Available	141
Tool Length Offset and G92	142
Tool Length Offset and G54-G59	143
Tool Length Offset and Multiple Tools	143

CHANGING TOOL LENGTH OFFSET	144
--	------------

HORIZONTAL MACHINE APPLICATION	145
---	------------

TOOL LENGTH OFFSET CANCEL	145
--	------------

20 - RAPID POSITIONING 147

RAPID TRAVERSE MOTION	147
G00 Command	147

RAPID MOTION TOOLPATH.	148
Single Axis Motion	148
Multiaxis Motion	148
Straight Angular Motion in Rapid Mode	150
Reverse Rapid Motion	150

TYPE OF MOTION & TIME COMPARISON	150
---	------------

REDUCTION OF RAPID MOTION RATE	151
---	------------

RAPID MOTION FORMULAS	151
--	------------

APPROACH TO THE PART	152
---------------------------------------	------------

21 - MACHINE ZERO RETURN 153

MACHINE REFERENCE POSITION	153
Machining Centers	153
Lathes	154
Setting the Machine Axes	154
Program Commands	155
Command Group	155

RETURN TO PRIMARY MACHINE ZERO	155
---	------------

Intermediate Point	155
Absolute and Incremental Mode	156
Return from Z-depth Position	157
Axes Return Required for the ATC	159
Zero Return for CNC Lathes	159

RETURN POSITION CHECK COMMAND	160
--	------------

RETURN FROM MACHINE ZERO POINT	161
---	------------

RETURN TO SECONDARY MACHINE ZERO	162
---	------------

22 - LINEAR INTERPOLATION 163

LINEAR COMMAND	163
---------------------------------	------------

Start and End of the Linear Motion	163	GENERAL RULES	184
Single Axis Linear Interpolation	163	ABSOLUTE AND INCREMENTAL INPUT	185
Two Axes Linear Interpolation	164	INITIAL LEVEL SELECTION	185
Three Axis Linear Interpolation	164	R-LEVEL SELECTION	186
PROGRAMMING FORMAT	164	Z-DEPTH CALCULATIONS	186
LINEAR FEEDRATE	165	DESCRIPTION OF FIXED CYCLES	187
Feedrate Range	165	G81 - Drilling Cycle	187
Individual Axis Feedrate	165	G82 - Spot Drilling Cycle (Drilling with Dwell)	188
PROGRAMMING EXAMPLE	166	G83 - Deep Hole Drilling Cycle - Standard	188
23 - BLOCK SKIP FUNCTION	167	G73 - Deep Hole Drilling Cycle - High Speed	188
TYPICAL APPLICATIONS	167	G84 - Tapping Cycle - Standard	190
BLOCK SKIP SYMBOL	167	G74 - Tapping Cycle - Reverse	190
CONTROL UNIT SETTING	167	G85 - Boring Cycle	191
BLOCK SKIP AND MODAL COMMANDS	168	G86 - Boring Cycle	191
PROGRAMMING EXAMPLES	169	G87 - Backboring Cycle	192
Variable Stock Removal	169	G88 - Boring Cycle	192
Machining Pattern Change	170	G89 - Boring Cycle	193
Trial Cut for Measuring	172	G76 - Precision Boring Cycle	193
Program Proving	173	FIXED CYCLE CANCELLATION	194
Barfeeder Application	174	FIXED CYCLE REPETITION	194
Numbered Block Skip	174	The L or K Address	194
24 - DWELL COMMAND	175	L0 or K0 in a Cycle	195
PROGRAMMING APPLICATIONS	175	RIGID TAPPING	195
Applications for Cutting	175	Comparison - Standard vs. Rigid Tapping	195
Applications for Accessories	175	Rigid Tapping - Fixed Cycles	195
DWELL COMMAND	175	Rigid Pecking Cycle	196
Dwell Command Structure	176	Cancellation	196
DWELL TIME SELECTION	176	26 - MACHINING HOLES	197
SETTING MODE AND DWELL	177	SINGLE HOLE EVALUATION	197
Time Setting	177	Tooling Selection and Applications	197
Number of Revolutions Setting	177	Program Data	200
MINIMUM DWELL	177	DRILLING OPERATIONS	200
Practical Considerations	177	Types of Drilling Operations	200
NUMBER OF REVOLUTIONS	178	Types of Drills	200
System Setting	178	Programming Considerations	201
Time Equivalent	178	Nominal Drill Diameter	201
LONG DWELL TIME	179	Effective Drill Diameter	201
Machine Warm-Up	180	Drill Point Length	201
X Axis is the Dwelling Axis	180	Center Drilling	202
Safety and Dwell	180	Spot Drilling	203
FIXED CYCLES AND DWELL	180	Blind Hole Drilling	203
25 - FIXED CYCLES	181	Through Hole Drilling	204
POINT-TO-POINT MACHINING	181	Flat Bottom Holes	204
Single Tool Motions vs. Fixed Cycles	182	Indexable Insert Drilling	205
Basic Concept	182	PECK DRILLING	206
FIXED CYCLE SELECTION	182	Typical Peck Drilling Application	206
PROGRAMMING FORMAT	183	Calculating the Number of Pecks	207
		Selecting the Number of Pecks	207
		Controlling Breakthrough Depth	207
		REAMING	208
		Reamer Design	208
		Spindle Speeds for Reaming	209
		Feedrates for Reaming	209
		Stock Allowance	209
		Other Reaming Considerations	209
		SINGLE POINT BORING	209

Single Point Boring Tool	209	Milling Mode	237
Spindle Orientation	210	Number of Cutting Inserts	238
Block Tools	211	PROGRAMMING TECHNIQUES	238
BORING WITH A TOOL SHIFT	211	Single Face Mill Cut	239
Precision Boring Cycle G76	211	Multiple Face Mill Cuts	240
Backboring Cycle G87	211	USING POSITION COMPENSATION	241
Programming Example	212		
Precautions in Programming and Setup.	212	29 - CIRCULAR INTERPOLATION	243
ENLARGING HOLES	212	ELEMENTS OF A CIRCLE	243
Countersinking	213	Radius and Diameter	243
Maximum Spot Drill Depth for CSINK	214	Circle Area and Circumference	244
Counterboring.	214	QUADRANTS.	244
Spotfacing	214	Quadrant Points	244
MULTI Z-DEPTH DRILLING	215	PROGRAMMING FORMAT	245
WEB DRILLING	216	Arc Cutting Direction	245
TAPPING	216	Circular Interpolation Block	245
Tap Geometry	217	Arc Start and End Points.	246
Tapping Speed and Feedrate	218	Arc Center and Radius	246
Pipe Taps	219	Arc Center Vectors	246
Tapping Check List	219	Arc in Planes	247
HOLE OPERATIONS ON A LATHE	220	RADIUS PROGRAMMING	248
Tool Approach Motion	221	Blend Radius	248
Tool Return Motion.	221	Partial Radius	248
Drilling and Reaming on Lathes	222	FULL CIRCLE PROGRAMMING	248
Peck Drilling Cycle - G74.	222	Boss Milling	250
Tapping on Lathes	223	Internal Circle Cutting - Linear Start	251
Other Operations	224	Internal Circle Cutting - Circular Start.	251
27 - PATTERN OF HOLES	225	Circle Cutting Cycle	252
TYPICAL HOLE PATTERNS	225	ARC PROGRAMMING	253
RANDOM HOLE PATTERN	225	FEEDRATE FOR CIRCULAR MOTION	253
STRAIGHT ROW HOLE PATTERN	226	Feedrate for Outside Arcs	254
ANGULAR ROW HOLE PATTERN	226	Feedrate for Inside Arcs	254
Pattern Defined by Coordinates	226	30 - CUTTER RADIUS OFFSET	255
Pattern Defined by Angle	227	MANUAL CALCULATIONS	255
CORNER PATTERN	228	Cutter Path Center Points	256
GRID PATTERN	228	Cutter Radius	257
Angular Grid Pattern	229	Center Points Calculation	257
ARC HOLE PATTERN.	230	COMPENSATED CUTTER PATH.	258
BOLT HOLE CIRCLE PATTERN	231	Types of Cutter Radius Offset	258
Bolt Circle Formula	232	Definition and Applications	258
Pattern Orientation	232	PROGRAMMING TECHNIQUES	259
POLAR COORDINATE SYSTEM	233	Direction of Cutting Motion	259
Plane Selection	234	Left or Right - not CW or CCW	259
Order of Machining	234	Offset Commands	260
28 - FACE MILLING	235	Radius of the Cutter	260
FACE MILL SELECTION	235	History of Offset Types	261
Basic Selection Criteria	235	Programming Format	261
Face Mill Diameter	235	Address H or D ?	261
Insert Geometry	236	Geometry and Wear Offsets	262
CUTTING CONSIDERATIONS	236	APPLYING CUTTER RADIUS OFFSET	262
Angle of Entry.	237	Startup Methods.	262
		Offset Cancellation.	264
		Cutter Direction Change.	265

HOW CUTTER RADIUS OFFSET WORKS	265		
Look-Ahead Offset Type	265	In and Out Ramping	289
Rules for Look-Ahead Cutter Radius Offset	266	Direction of Cut	289
Radius of the Cutter	267	Width and Depth of Cut	290
Radius Offset Interference	268	CORNER RADIUS CALCULATION	290
Single vs. Multiaxis Startup	269		
OVERVIEW OF GENERAL RULES	270	33 - SLOTS AND POCKETS	291
PRACTICAL EXAMPLE - MILLING	271	OPEN AND CLOSED BOUNDARY	291
Part Tolerances	271	Open Boundary	291
Measured Part Size	271	Closed Boundary	291
Programmed Offsets	272	PROGRAMMING SLOTS	291
D-offset Amount - General Setting	272	Open Slot Example	291
Offset Adjustment	273	Closed Slot Example	293
Offsets for External Cutting	273	POCKET MILLING	294
Offsets for Internal Cutting	273	General Principles	294
One Offset or Multiple Offsets?	274	Pocket Types	295
Preventing a Scrap	274	RECTANGULAR POCKETS	295
Program Data - Nominal or Middle?	274	Stock Amount	296
TOOL NOSE RADIUS OFFSET	275	Stepover Amount	296
Tool Nose Tip	275	Length of Cut	297
Radius Offset Commands	275	Semifinishing Motions	297
Tool Tip Orientation	275	Finishing Toolpath	297
Effect of Tool Nose Radius Offset	276	Rectangular Pocket Program Example	298
Sample Program	276	CIRCULAR POCKETS	299
Minimum Clearance Required	277	Minimum Cutter Diameter	299
Retraction from a Bored Hole	277	Method of Entry	299
Change of Motion Direction	278	Linear Approach	299
		Linear and Circular Approach	300
		Roughing a Circular Pocket	301
31 - PLANE SELECTION	279	CIRCULAR POCKET CYCLES	302
WHAT IS A PLANE?	279		
MACHINING IN PLANES	279	34 - TURNING AND BORING	303
Mathematical Planes	279	TOOL FUNCTION - TURNING	303
Machine Tool Planes	280	T-Address	303
Program Commands for Planes Definition	280	LATHE OFFSETS	304
Default Control Status	281	Offset Entry	304
STRAIGHT MOTION IN PLANES	281	Independent Tool Offset	304
CIRCULAR INTERPOLATION IN PLANES	281	Tool Offset with Motion	305
G17-G18-G19 as Modal Commands	282	Offset Change	305
Absence of Axis Data in a Block	282	MULTIPLE OFFSETS	305
Cutter Radius Offset in Planes	283	General Approach	306
PRACTICAL EXAMPLE	283	Diameter Tolerances	306
FIXED CYCLES IN PLANES	284	Shoulder Tolerances	307
		Diameter and Shoulder Tolerances	307
32 - CONTOUR MILLING	285	OFFSET SETTING	308
END MILLS	285	FUNCTIONS FOR GEAR RANGES	308
High Speed Steel End Mills	286	AUTOMATIC CORNER BREAK	309
Solid Carbide End Mills	286	Chamfering at 45 Degrees	309
Indexable Insert End Mills	286	Blend Radius at 90 Degrees	311
Relief Angles	286	Programming Conditions	311
End Mill Size	286	Programming Example	311
Number of Flutes	287	ROUGH AND FINISHED SHAPE	312
SPEEDS AND FEEDS	287	Rough Operations	312
Coolants and Lubricants	288	Finish Operations	313
Tool Chatter	288	Stock and Stock Allowance	313
STOCK REMOVAL	289		
Plunge Infeed	289		

PROGRAMMING A RECESS	314
SPINDLE SPEED IN CSS MODE	315
LATHE PROGRAM FORMAT	316
Program Format - Templates	316
General Program Format	316
Approach to the Part	316

35 - LATHE CYCLES 317

STOCK REMOVAL ON LATHES	317
Simple Cycles	317
Complex Cycles	317
PRINCIPLES OF LATHE CYCLES	317
G90 - STRAIGHT CUTTING CYCLE	318
Cycle Format	318
Straight Turning Example	319
Taper Cutting Example	319
Straight and Taper Cutting Example	321
G94 - FACE CUTTING CYCLE	322
Cycle Format	322
MULTIPLE REPETITIVE CYCLES	323
General Description	323
Cycle Format Types	323
Cutting Cycles and Part Contour	323
Chipbreaking Cycles	323
CONTOUR CUTTING CYCLES	324
Boundary Definition	324
Start Point and the Points P and Q	324
TYPE I AND TYPE II CYCLES	325
Programming Type I and Type II Cycles	325
Cycle Formatting	325
G71 - STOCK REMOVAL IN TURNING	325
G71 Cycle Format - 6T/10T/11T/15T	326
G71 Cycle Format - 0T/16T/18T/20T/21T	326
G71 for External Roughing	326
G71 for Internal Roughing	327
Direction of Cutting in G71	327
G72 - STOCK REMOVAL IN FACING	327
G72 Cycle Format - 6T/10T/11T/15T	327
G72 Cycle Format - 0T/16T/18T/20T/21T	328
G73 - PATTERN REPEATING CYCLE	328
G73 Cycle Format - 6T/10T/11T/15T	328
G73 Cycle Format - 0T/16T/18T/20T/21T	329
G73 Example of Pattern Repeating	329
G70 - CONTOUR FINISHING CYCLE	330
G70 Cycle Format - All Controls	330
BASIC RULES FOR G70-G73 CYCLES	331
G74 - PECK DRILLING CYCLE	331
G74 Cycle Format - 6T/10T/11T/15T	331
G74 Cycle Format - 0T/16T/18T/20T/21T	331
G75 - GROOVE CUTTING CYCLE	332
G75 Cycle Format - 6T/10T/11T/15T	332
G75 Cycle Format - 0T/16T/18T/20T/21T	332
BASIC RULES FOR G74 AND G75 CYCLES	332

36 - GROOVING ON LATHES 333

GROOVING OPERATIONS	333
Main Grooving Applications	333
Grooving Criteria	333
GROOVE SHAPE	333
Nominal Insert Size	334
Insert Modification	334
GROOVE LOCATION	334
GROOVE DIMENSIONS	334
Groove Position	335
Groove Depth	335
SIMPLE GROOVE PROGRAMMING	335
PRECISION GROOVING TECHNIQUES	336
Groove Width Selection	337
Machining Method	337
Finishing Allowances	338
Groove Tolerances	339
Groove Surface Finish	340
MULTIPLE GROOVES	340
FACE GROOVES	340
Radial Clearance	341
Face Grooving Program Example	341
CORNER GROOVES / NECK GROOVES	342
GROOVING CYCLES	342
G75 Cycle Applications	342
Single Groove with G75	343
Multiple Grooves with G75	343
SPECIAL GROOVES	343
GROOVES AND SUBPROGRAMS	344

37 - PART-OFF 345

PART-OFF PROCEDURE	345
Parting Tool Description	345
Tool Approach Motion	346
Stock Allowance	347
Tool Return Motion	347
Part-off with a Chamfer	347
Preventing Damage to the Part	348

38 - SINGLE POINT THREADING 349

THREADING ON CNC LATHES	349
Form of a Thread	349
Threading Operations	350
TERMINOLOGY OF THREADING	350
THREADING PROCESS	351
Steps in Threading	351
Spindle Synchronization	351
Individual Threading Motions	351
Thread Start Position	353
Thread Cutting Diameter and Depth	353
Number of Threading Passes	353
Thread Depth Calculation	354

Thread Depth Constants	354	39 - SUBPROGRAMS	383
Calculation of Threading Passes	355	MAIN PROGRAM AND SUBPROGRAMS	383
Thread Cutting Motion	356	Subprogram Benefits	383
Retract from Thread	357	Identification of Subprograms	384
Return to Start Position	357	SUBPROGRAM FUNCTIONS	384
THREADING FEED AND SPINDLE SPEED	357	Subprogram Call Function	384
Threading Feedrate Selection.	358	Subprogram End Function	384
Spindle Speed Selection.	359	Block Number to Return to.	385
Maximum Threading Feedrate	359	Number of Subprogram Repetitions	385
Lead Error	360	L0/K0 in a Subprogram Call	386
Rounding Error	360	SUBPROGRAM NUMBERING	387
E-address	361	Organized Approach	388
TOOL REFERENCE POINT	361	Protected Subprograms	389
BLOCK-BY-BLOCK THREADING.	361	SUBPROGRAM DEVELOPMENT	389
BASIC THREADING CYCLE - G92	362	Repeating Pattern Recognition	389
MULTIPLE REPETITIVE CYCLE - G76.	363	Tool Motion and Subprograms	390
G76 Cycle Format - One Block Format	364	Modal Values and Subprograms	391
G76 Cycle Format - Two Block Format	364	MULTI LEVEL NESTING	392
One-Block vs Two-Block Format.	365	One Level Nesting	392
Programming Examples	365	Two Level Nesting	392
First Thread Diameter Calculation	366	Three Level Nesting	393
THREAD INFEEED METHODS.	366	Four Level Nesting	393
Radial Infeed (Straight Infeed)	367	Nesting Applications	393
Compound Infeed (Flank Infeed)	367	CONTOURING WITH A SUBPROGRAM.	394
Thread Insert Angle	367	TOOL CHANGE SUBPROGRAM.	395
Thread Cutting Type - Address P	368	100 000 000 HOLE GRID	395
COMPOUND INFEEED CALCULATIONS	368	40 - DATUM SHIFT	397
Initial Considerations	368	DATUM SHIFT WITH G92 OR G50	397
Z-axis Start Position Calculation	369	Program Zero Shift	397
THREAD RETRACT MOTION	370	LOCAL COORDINATE SYSTEM	399
Thread Pullout Functions	370	G52 Command	400
Single Axis Pullout	371	MACHINE COORDINATE SYSTEM	400
Two-Axis Pullout	371	DATA SETTING	402
HAND OF THREAD	371	Data Setting Command	402
Configurations for Hand of Thread.	372	Coordinate Mode	402
THREADING TO A SHOULDER	373	WORK OFFSETS	402
Insert Modification	373	Standard Work Offset Input	402
Program Testing	375	Additional Work Offset Input	403
OTHER THREAD FORMS.	375	External Work Offset Input.	403
Thread Depth - ACME example	375	TOOL LENGTH OFFSETS	403
TAPERED THREAD	376	Valid Input Range	404
Depth and Clearances	376	CUTTER RADIUS OFFSETS	404
Taper Calculation	376	LATHE OFFSETS	404
Block by Block Taper Thread	377	MDI DATA SETTING	405
Tapered Thread Using a Simple Cycle	378	PROGRAMMABLE PARAMETER ENTRY	405
Tapered Thread and G76 Cycle	378	Modal G10 Command	405
MULTISTART THREAD	379	Parameters Notation	406
Threading Feedrate Calculation	380	Program Portability	406
Shift Amount	380	Bit Type Parameter	407
Application Example	381	Effect of Block Numbers	408
Quality of Multistart Threads	381		
THREAD RECUTTING	382		
Before Part Removal	382		
After Part Removal	382		

41 - MIRROR IMAGE 409

BASIC RULES OF MIRROR IMAGE 409

Toolpath Direction	409
Original Toolpath	410
Mirror Axis	410
Sign of Coordinates	410
Milling Direction	411
Arc Motion Direction	411
Program Start and End	411

MIRROR IMAGE BY SETTING 411

Control Setting	411
Programming - Manual Mirror Setting	412

PROGRAMMABLE MIRROR IMAGE 412

Mirror Image Functions	412
Simple Mirror Image Example	413
Complete Mirror Image Example	414

MIRROR IMAGE ON CNC LATHES 414

42 - COORDINATE ROTATION 415

ROTATION COMMANDS 415

Center of Rotation	415
Radius of Rotation	415
Coordinate Rotation Cancel	417
Common Applications	417

PRACTICAL APPLICATION 417

43 - SCALING FUNCTION 421

DESCRIPTION 421

Scaling Function Usage	421
----------------------------------	-----

PROGRAMMING FORMAT 421

Scaling Center	422
Scaling Factor	422
Rounding Errors in Scaling	423

PROGRAM EXAMPLES 423

44 - CNC LATHE ACCESSORIES 425

CHUCK CONTROL 425

Chuck Functions	425
Chucking Pressure	426
Chuck Jaws	426

TAILSTOCK AND QUILL 426

Tailstock Body	426
Quill	427
Center	427
Quill Functions	427
Programmable Tailstock	427
Safety Concerns	427

BI-DIRECTIONAL TURRET INDEXING 428

Programming Example	428
-------------------------------	-----

BARFEEDER ATTACHMENT 429

Bar Stopper	429
-----------------------	-----

ADDITIONAL OPTIONS 430

Part Catcher or Part Unloader	430
Continuous Operation	430
Parts Counter	431
Pull-Out Finger	431

PROGRAMMING EXAMPLE 431

45 - HELICAL MILLING 433

HELICAL MILLING OPERATION 433

Helical Interpolation	433
Programming Format	433
Arc Vectors for Helical Interpolation	434
Applications and Usage	434

THREAD MILLING 434

Applying Thread Milling	434
Conditions for Thread Milling	434
Thread Milling Tool	435
Premachining Requirements	435
Clearance Radius	435
Productivity of Thread Milling	435

THE HELIX 435

THREAD MILLING EXAMPLE 437

Straight Thread	437
Initial Calculations	437
Starting Position	438
Motion Rotation and Direction	438
Lead-In Motions	440
Thread Rise Calculation	441
Milling the Thread	441
Lead-Out Motions	442
Complete Program	442
External Thread Milling	442
Tapered Thread Milling	442
Further Considerations	443

THREAD MILLING SIMULATION METHOD 443

HELICAL RAMPING 444

46 - HORIZONTAL MACHINING 445

INDEXING AND ROTARY AXES 445

INDEXING TABLE (B-AXIS) 445

Units of Increment	445
Direction of Indexing	446
Table Clamp and Unclamp Functions	446
Indexing in Absolute and Incremental Mode	446

B-AXIS AND OFFSETS 447

Work Offset and B-axis	447
Tool Length Offset and B-axis	448

RETURN TO MACHINE ZERO 450

INDEXING AND A SUBPROGRAM 450

COMPLETE PROGRAM EXAMPLE 452

AUTOMATIC PALLET CHANGER - APC 453

Working Environment	454
Types of Pallets	454
Programming Commands	455
Pallet Changing Program Structure	455

HORIZONTAL BORING MILL 455**47 - LIVE TOOLING ON LATHES 457****TURN-MILL OR MILL-TURN 457**

Programming Issues	457
General Terms.	457

MACHINE DESIGN 458

Features	458
Benefits	458

C-AXIS PROGRAMMING 458

XZ + C Axes	458
-----------------------	-----

FIXED CYCLES 459

M-functions and C-axis	460
Drilling on Face	460
Drilling on Diameter	461
General Considerations	461

Y-AXIS PROGRAMMING. 462

Plane Selection	463
Additional Axes	463

POLAR COORDINATE INTERPOLATION . . . 463

Polar Coordinate Mode	463
Approximation Method	464

CYLINDRICAL INTERPOLATION 465

Z + C-axis	465
Practical Example	465

48 - WRITING A CNC PROGRAM 467**PROGRAM WRITING. 467**

Legibility of Handwriting.	468
Programming Forms	468
Confusing Characters.	468

PROGRAM OUTPUT FORMATTING. 469**LONG PROGRAMS 471**

Program Length Reduction.	471
Memory Mode and Tape Mode	472

49 - PROGRAM DOCUMENTS 473**DATA FILES 473****PROGRAM DOCUMENTATION 474**

External Documentation.	474
Internal Documentation	474
Program Description	475

SETUP AND TOOLING SHEETS. 475

Setup Sheet	476
Tooling Sheet	476
Coordinate Sheet	477

DOCUMENTATION FILE FOLDER 477

Identification Methods	477
Operator's Suggestions	478
Filing and Storage	478

50 - PROGRAM VERIFICATION 479**DETECTION OF ERRORS 479**

Preventive Measures	479
Corrective Measures	479

GRAPHIC VERIFICATION 480**AVOIDING ERRORS 480**

Syntax Errors	481
Logical Errors	481

COMMON PROGRAMMING ERRORS 482

Program Input Errors	482
Calculation Errors	482
Hardware Errors	482
Miscellaneous Errors	482

51 - CNC MACHINING 483**MACHINING A NEW PART 483**

Setup Integrity	484
Program Integrity	484

RUNNING THE FIRST PART 485**PROGRAM CHANGES 486**

Program Upgrading	486
Program Updating	487
Documentation Change	487

ALTERNATE MACHINE SELECTION 487**MACHINE WARM UP PROGRAM 488****CNC MACHINING AND SAFETY. 488****SHUTTING DOWN A CNC MACHINE 489**

Emergency Stop Switch.	489
Parking Machine Slides	490
Setting the Control System	490
Turning the Power Off	490

EQUIPMENT MAINTENANCE 490**52 - INTERFACING TO DEVICES 491****RS-232C INTERFACE. 491****PUNCHED TAPE 492**

Tape Reader and Puncher	492
Leader and Trailer	494
Tape Identification	494
Non-printable Characters	494
Storage and Handling.	494

DISTRIBUTED NUMERICAL CONTROL . . . 494**TERMINOLOGY OF COMMUNICATIONS . . 495**

Baud Rate	495
Parity	495
Data Bits	495
Start and Stop Bits	495

DATA SETTING 495**CONNECTING CABLES 496**

Null Modem	496
Cabling for Fanuc and PC	496

53 - MATH IN CNC PROGRAMMING 497**BASIC ELEMENTS 497**

Arithmetic and Algebra	497
Order of Calculations	497

GEOMETRY 498

Circle	498
PI Constant	499
Circumference of a Circle	499
Length of Arc	499
Quadrants	499

POLYGONS 500**TAPERS 501**

Taper Definition	501
Taper Per Foot	502
Taper Ratio	502
Taper Calculations - Imperial Units	502
Taper Calculations - Metric Units	502

CALCULATIONS OF TRIANGLES 503

Types of Angles and Triangles	503
Right Triangles	504
Similar Triangles	504
Sine - Cosine - Tangent	505
Inverse Trigonometric Functions	506
Degrees and Decimal Degrees	506
Pythagorean Theorem	506
Solving Right Triangles	507

ADVANCED CALCULATIONS 508**CONCLUSION 508****54 - CNC AND CAD/CAM 509****PROGRAMMING MANUALLY ? 509**

CAM Software	509
Desktop Computer Programming	509

TOOLPATH GEOMETRY DEVELOPMENT 510**TOOL PATH GENERATION 510****COMPLETE ENVIRONMENT. 510**

Multi Machine Support	511
Associative Operations	511
Job Setup	511
Tooling List and Job Comments	511
Connection Between Computers	511
Program Text Editor	512
Printing Capabilities	512
Pen Plotting	512
CAD Software Access	512
Support for Solids	512
Software Specifications	513
Hardware Specifications	513
Typical Hardware / Software Requirements	514
Utilities and Special Features	514

POST PROCESSORS. 514

Customizing Post Processor	514
--------------------------------------	-----

IMPORTANT FEATURES. 515

Input from User	515
Machining Cycles	515
User Interface	515
CAD Interface	515

SUPPORT AND MANAGEMENT. 515

Training	516
System Management	516
Technical Support	516

THE END AND THE BEGINNING 516**A - REFERENCE TABLES 517****Decimal Equivalents 517****Imperial Threads - UNC/UNF 520****Straight Pipe Taps NPS 520****Taper Pipe Taps NPT. 521****Metric Coarse Threads 521****Metric Fine Threads 521****B - SPEEDS AND FEEDS 523****SURFACE SPEED AND SPINDLE SPEED 523****CHIPLOAD 524****BASIC FORMULAS 524****RELATED FORMULAS 524**

Tapping Feedrate	524
Z-depth for Spot Drilling	524
Z-depth for Drilling (Through Hole).	525
Dwell Time Calculation	525
Cutting Time	525

C - NCPlot FEATURES 527**Full Featured Text Editor 527****Program Formatting Tools 527****Program Translation Tools 527****Advanced G-code Backplotter 527****Flexible Viewport Controls 528****View Controls 528****Calc Tools 528****DXF Importing / Exporting 528****Other Tools 528****Index 529**

1

NUMERICAL CONTROL

Numerical Control technology as known today, emerged in the mid 20th century. It can be traced to the year of 1952, the U.S. Air Force, Massachusetts Institute of Technology in Cambridge, MA, USA, and the name of John Parsons (1913-2007), who is closely associated with the invention of numerical control. It was not applied in production manufacturing until the early 1960's. The real boom came in the form of CNC, around the year of 1972, and a decade later with the introduction of affordable micro computers. History and development of this fascinating technology has been well documented in many publications.

In manufacturing field, and particularly in the area of metal working, Numerical Control technology has caused something of a revolution. Even in the days before computers became standard fixtures in every company and many homes, machine tools equipped with Numerical Control system found their special place in many machine shops. The relatively recent evolution of micro electronics and the never ceasing computer development, including its impact on Numerical Control, has brought enormously significant changes to manufacturing sector in general and metal-working industry in particular.

DEFINITION OF NUMERICAL CONTROL

In various publications and articles, many descriptions have been used during the years, to define what Numerical Control actually is. It would be pointless to try to find yet another definition, just for the purpose of this handbook. Many of these definitions share the same idea, same basic concept, just use different wording.

The majority of all the known definitions can be summed up into a relatively simple statement:

Numerical Control can be defined as an operation of machine tools by means of specifically coded instructions to the machine control system

The '*specifically coded instructions*' are combinations of the letters of alphabet, digits and selected symbols, for example, a decimal point, the percent sign, or the parenthesis symbols. All instructions are written in a logical order and in predetermined form. The collection of *all* instructions necessary to machine a single part or operation is called an *NC Program*, *CNC Program*, or a *Part Program*. Such a program can be stored for future use and used repeatedly to achieve identical machining results at any time.

◆ NC and CNC Technology

In strict adherence to terminology, there is a difference in the meaning of abbreviations *NC* and *CNC*. The *NC* stands for the older and original *Numerical Control* technology, whereby the abbreviation *CNC* stands for the newer *Computerized Numerical Control* technology - a modern successor to its older relative. However, in everyday practice, *CNC* is the preferred abbreviation. To clarify the proper usage of each term, look at the major differences between *NC* and *CNC* systems.

Both systems perform the same tasks, namely manipulation of data for the sole purpose of machining a part. In both cases, the control system internal design contains all logical instructions that process the input data. At this point the similarity ends.

The *NC* system (as opposed to the *CNC* system) uses a fixed logical functions, those that are built-in and permanently wired within the control unit. These functions cannot be changed by the part programmer or the machine operator. Because of the fixed wiring of control logic, *NC* control system is synonymous with the term '*hardwired*'. The system can interpret a part program, but it does not allow any changes to the program at the control (using the control features). All required program changes must be made away from the control, typically in an office environment. Also, *NC* system typically requires the compulsory use of punched tapes for input of the program information.

The modern *CNC* system (but not the old *NC* system), uses an internal micro processor (*i.e.*, a computer). This computer contains memory registers storing a variety of routines that are capable of manipulating logical functions. That means the part programmer or machine operator can change any program at the control unit (at the machine), with instantaneous results. This flexibility is the greatest advantage of *CNC* systems and probably the key element that contributed to such a wide use of the technology in modern manufacturing. Typically, *CNC* programs and the logical functions are stored on special computer chips, as *software instructions*, rather than used by the hardware connections, such as wires, that control the logical functions. In contrast to the *NC* system, the *CNC* system is synonymous with the term '*softwired*'.

When describing a particular subject that relates to numerical control technology, it is customary to use either the term *NC* or *CNC*. Keep in mind that *NC* can also mean *CNC* in everyday talk, but *CNC* can never refer to the older

technology, described in this handbook under the abbreviation of NC. The letter 'C' stands for *Computerized*, and it is not applicable to the hardwired system. All control systems manufactured today are of the CNC design. Abbreviations such as C&C or C'n'C are not correct and reflect poorly on anybody that uses them.

CONVENTIONAL AND CNC MACHINING

What makes CNC machining methods superior to conventional methods? Are they superior at all? Where are the main benefits? While comparing CNC and conventional machining processes, common general approach to machining a typical part will emerge:

1. Obtain and study the engineering drawing
2. Select the most suitable machining method
3. Decide on the setup method (work holding)
4. Select cutting tools and holders
5. Establish spindle speeds and cutting feedrates
6. Machine the part

This general approach is the same for both types of machining. One major difference is *how* various data are input. A feedrate of 10 inches per minute (10 in/min) is the same in manual or CNC applications, but the method of applying it is not. The same can be said about a coolant - it can be activated by physically turning a knob, pushing a switch or programming a special code. All these actions will result in coolant rushing out of a nozzle. In both kinds of machining, a certain amount of knowledge by the user is required. After all, metal working, and metal cutting specifically, is mainly a skill, but it is also, to a great degree, an art and a profession of large number of people. So is the application of *Computerized Numerical Control*. Like any skill, or art, or profession, mastering it to the last detail is necessary to be successful. It takes a lot more than just technical knowledge to be a CNC machinist, operator or CNC programmer. Work experience, intuition, and what is sometimes called a '*gut-feel*', are much needed supplements to any skill.

In conventional machining, the operator sets up the machine and moves each cutting tool, using one or both hands, to produce the required part. Design of a manual machine tool offers many features that help the process of machining a part - levers, handles, gears and dials, to name just a few. The same body motions are repeated by the operator for every part machined. However, the word '*same*' in this context really means '*similar*' rather than '*identical*'. Humans are not capable to repeat every process exactly the same at all times - that is the job of machines. People cannot work at the same performance level all the time, without a rest. All of us have some good and some bad moments. Such moments, when applied to machining a part, are difficult to predict. There will always be some differences and inconsistencies within each batch of parts. Parts will not always be *exactly* the same. Maintaining dimensional tolerances and surface finish quality are the most

typical problems encountered in conventional machining. Individual machinists may have their own 'time proven' methods, different from those of their fellow colleagues. Combination of these and other factors create a large field of inconsistency.

Machining under numerical control does away with the majority of inconsistencies. It does not require the same physical involvement as manual machining. Numerically controlled machining does not need any levers or dials or handles, at least not in the same sense as conventional machining. Once the part program has been proven, it can be used any number of times over, always returning consistent results. That does not mean there are no limiting factors. Cutting tools do wear out, material blank in one batch is not identical to the material blank in another batch, setups may vary, etc. These factors should be considered and compensated for, whenever necessary.

Emergence of numerical control technology does not mean an instant - or even a long term - demise of all manual machines. There are times when a traditional machining method is preferable to a computerized method. For example, a simple one time job may be done more efficiently on a manual machine than on a CNC machine. Certain types of machining jobs will benefit from manual, semiautomatic or automatic machining, rather than machining under numerical control. CNC machine tools are not meant to replace every manual machine, only to supplement them.

In many instances, the decision whether certain machining will be done on a CNC machine or not is based on the number of required parts and nothing else. Although the volume of parts machined as a batch is always an important criteria, it should never be the only factor. Consideration should also be given to the part complexity, its tolerances, the required quality of surface finish, etc. Often, a single complex part will benefit from CNC machining, while fifty relatively simple parts will not.

Keep in mind that numerical control has never machined a single part by itself. Numerical control is only a *process* or a *method* that enables a machine tool to be used in a productive, accurate and consistent way.

NUMERICAL CONTROL ADVANTAGES

What are the main advantages of numerical control?

It is important to establish which areas of machining will benefit from it and which are better done the conventional way. It is absurd to think that a two horse power CNC mill will win over jobs that are currently done on a twenty times more powerful manual mill. Equally unreasonable are expectations of super improvements in cutting speeds and feedrates over a conventional machine. If the machining and tooling conditions are the same, the total cutting time will always be very close in both cases.

A list of some major areas where CNC users can and should expect improvement includes:

- Setup time reduction
- Lead time reduction
- Accuracy and repeatability
- Contouring of complex shapes
- Simplified tooling and work holding
- Consistent cutting time
- General productivity increase

Each area offers only a *potential* improvement. Individual CNC users will experience different levels of *actual* improvement, depending on the product manufactured, CNC machine used, setup methods applied, complexity of fixturing, quality of cutting tools, management philosophy and engineering design, experience level of the workforce, individual attitudes, and many others.

◆ Setup Time Reduction

In many cases, actual setup times for CNC machines can be reduced, sometimes quite dramatically. It is important to realize that setup is a manual operation, greatly dependent on the performance of CNC operators, the type of fixturing and general machine shop practices. Setup time is unproductive, but necessary - it is part of the overall costs of doing business. To keep setup time to minimum should be the primary consideration of any machine shop supervisor, programmer and operator.

Because of the design of CNC machines, real setup time should not be a major problem. Modular fixturing, standardized tooling, fixed locators, automatic tool changing, pallets, and other advanced features, make the setup time more efficient than a comparable setup of conventional machines. With good knowledge of modern manufacturing, productivity can be increased quite significantly.

The number of parts machined in a single setup is also important, in order to assess the actual cost of setup time. If a great number of parts is machined in one setup, the setup cost per part can be rather insignificant. A very similar reduction can be achieved by grouping several different operations into a single setup. Even if the setup time is longer, it may be justified when compared to the time required to setup several conventional machines and operations.

◆ Lead Time Reduction

Once a part program is written and proven correct, it is ready to be used again in the future, even at a short notice. Although the first run lead time is usually longer, it is virtually nil for all subsequent runs. Even if an engineering change of the part design requires program modification, it can be done usually quickly, reducing the lead time.

Long lead time, required to design and manufacture several special fixtures for conventional machines, can often be reduced by using simplified fixturing.

◆ Accuracy and Repeatability

The high degree of accuracy and repeatability of modern CNC machines has been the single major benefit to many users. Whether part program is stored on a disk or in the computer memory, or even on a tape (the original method, now obsolete), it always remains the same. Any program can be changed at will, but once proven, no changes are usually required any more. A given program can be reused as many times as needed, without losing a single bit of data it contains. True, program has to allow for such changeable factors as tool wear and operating temperatures, it has to be stored safely, but generally very little interference from the CNC programmer or operator will be required. The accuracy of modern CNC machines and their repeatability allows high quality parts to be produced consistently, time after time.

◆ Contouring of Complex Shapes

CNC lathes and machining centers are capable of contouring a large variety of different shapes. Many CNC users acquired their machines only to be able to handle complex parts. A good examples are CNC applications in the aircraft and automotive industries. Any use of some kind of computerized programming is virtually mandatory for any three dimensional tool path generation.

Complex shapes, such as molds, manifolds, dies, etc., can be manufactured without the additional expense of making a model for tracing. Mirrored parts can be achieved literally at the switch of a button. Storage of part programs is a lot simpler than storage of paper patterns, templates, wooden models, and other pattern making tools.

◆ Simplified Tooling and Work Holding

Non-standard and 'homemade' tooling that clutters the benches and drawers around a conventional machine can be eliminated by using standard tooling, specially designed for numerical control applications. Multi-step tools such as pilot drills, step drills, combination tools, counter borers and others, are replaced with several individual standard tools. These tools are often cheaper and easier to replace than special and non-standard tools. Cost-cutting measures have forced many tool suppliers to keep a low or even a nonexistent inventory, while increasing delivery time to the customer. Standard, off-the-shelf tooling can usually be obtained faster than non-standard tooling.

Fixturing and work holding for CNC machines have only one major purpose - to hold the part rigidly and in the same position for all parts within a batch. Fixtures designed for CNC work do not normally require special jigs, pilot holes and other hole locating aids.

◆ Cutting Time and Productivity Increase

Cutting time on a CNC machine is commonly known as the *cycle time* - and is always consistent. Unlike a conventional machining, where the operator's skill, experience and personal fatigue are subject to changes, CNC machining is under the control of a computer. Only a small amount of manual work is restricted to the setup and part loading and unloading. For large batch runs, the high cost of unproductive time is spread among many parts, making it less significant. The main benefit of a consistent cutting time is for repetitive jobs, where production scheduling and work allocation to individual machine tools can be done very efficiently and accurately.

One of the main reasons companies often purchase CNC machines is strictly economic - it is a serious investment with great potential. Also, having a competitive edge is always on the mind of every plant manager. Numerical control technology offers excellent means to achieve significant improvements in manufacturing productivity and increasing the overall quality of manufactured parts. Like any means to an end, it has to be used wisely and knowledgeably. When more and more companies use CNC technology, just having a CNC machine does not offer the extra edge anymore. Companies that grow and get forward are those where the use of technology is managed efficiently, with the goal to be competitive in the global economy.

To reach the goal of major increase in productivity, it is essential that users understand the fundamental principles on which CNC technology is based. These principles take many forms, for example, understanding the electronic circuitry, complex ladder diagrams, computer logic, metrology, machine design, machining principles and practices, and many others. Each discipline has to be studied and mastered by all persons in charge. In this handbook, the main emphasis is on topics relating directly to CNC programming and understanding the most common CNC machines - *Machining Centers* and *Lathes* (sometimes called *Turning Centers*). Part quality consideration should be very important to every programmer and machine operator and this goal is also reflected in the handbook approach as well as in numerous examples.

TYPES OF CNC MACHINE TOOLS

Different types of CNC machines cover rather large variety. Number of installations is rapidly increasing, and the technology development advances at a rapid pace. It is impossible to identify all possible applications, they would make a long list. Here is a brief list of some of the groups CNC machines can be part of:

- Mills and Machining centers
- Lathes and Turning centers
- Drilling machines

- Boring mills and Profilers
- EDM wire machines
- Punch presses and Shears
- Flame cutting machines
- Routers
- Water jet and Laser profilers
- Cylindrical grinders
- Welding machines
- Benders, Winding and Spinning machines, etc.

Without a doubt, CNC machining centers and lathes dominate the number of installations in industry. These two groups share the market just about equally. Some industries may have a higher need for a particular type of machines, depending on their needs. One must remember there are many models of lathes available and equally many different models of machining centers. However, the programming process for a vertical machine is similar to the one for a horizontal machine or even a simple CNC mill, for example. Even between different machine groups, there is a great amount of general applications, while the programming process is generally unchanged. For example, a contour milled with an end mill has a lot in common with a contour cut with a wire on an EDM machine.

◆ Mills and Machining Centers

Minimum number of axes on a milling machine is three - the X, Y and Z axes. Part set on a milling machine is always stationary, mounted on a moving machine table. The cutting tool rotates, it can move up and down (or in and out), but it does not physically follow the tool path.

CNC mills - sometimes called CNC milling machines - are usually small, simple machines, without a tool changer or other automatic features. Their power rating is often low. In industry, they are used for toolroom work, maintenance purposes, or small part production. They are usually designed for simple contouring, unlike CNC drills.

CNC machining centers are far more popular and efficient than drills and mills, mainly for their flexibility. The main benefit users get out of a CNC machining center is the ability to group several diverse operations into a single setup. For example, drilling, boring, counter boring, tapping, spot facing and contour milling can be incorporated into a single CNC program operation. In addition, the flexibility is enhanced by automatic tool changing, using pallets to minimize idle time, indexing to a different face of the part, using a rotary movement of additional axes, and number of other time saving features. CNC machining centers can be equipped with special software that controls cutting speeds and feeds, life of the cutting tool, automatic in-process gauging, broken tool detection, offset adjustment and other production enhancing and time saving devices.

There are two basic designs of a typical CNC machining center. They are *vertical* and *horizontal* machining centers. The major difference between the two types is the nature of work that can be done on them efficiently. For a vertical CNC machining center, the most suitable type of work are flat parts, either mounted to the table fixture, or held in a vise or a chuck. The work that requires machining on two or more faces (sides) in a single setup is more desirable to be done on a CNC horizontal machining center. A good example is a pump housing and other cubic-like shapes, often irregular. Some multi-face machining of small parts can also be done on a CNC vertical machining center equipped with a rotary table.

Programming process is the same, but an additional axis (usually B axis) is added to the horizontal version. This axis is either a simple table positioning axis (indexing axis), or a fully rotary axis for simultaneous contouring.

This handbook concentrates on CNC vertical machining centers applications, with a special section dealing with the uniqueness of horizontal setup and machining. Suggested programming methods are also applicable to small CNC mills or drilling and/or tapping machines, but the part programmer has to consider their often severe restrictions.

◆ Lathes and Turning Centers

A CNC lathe in its basic form is a machine tool with two axes, vertical X axis and horizontal Z axis. The main feature of a lathe that distinguishes it from a mill is that the part is rotating about the machine center line. In addition, the cutting tool is normally stationary, mounted in a sliding turret. Cutting tool follows the contour of the programmed tool path. Many modern CNC lathes are much more than just turning centers - with a simple milling attachment, the so called *live tooling*, the milling cutter has its own motor and rotates while the spindle is stationary. More complex designs incorporate off-center milling, double spindles, double turrets, part transfer, and many other efficiency improving features. These machines are generally called the *mill-turn* centers or sometimes the *turn-mill* centers.

Modern lathe design can be horizontal or vertical. Horizontal type is far more common than vertical type, but both designs have their purpose in manufacturing. Several different designs exist for either group. For example, a typical CNC lathe of the horizontal group can be designed with a flat bed or a slant bed, as a bar type, chucker type or a universal type. Added to these combinations are many accessories that make a CNC lathe an extremely flexible machine tool. Typically, accessories such as tailstock, steady rests or follow-up rests, part catchers, pullout-fingers and a third axis milling attachment are popular components of CNC lathes. A CNC lathe can be very versatile - so versatile in fact, that it is often called a *CNC Turning Center*. All text and program examples in this handbook use the more traditional term *CNC lathe*, yet still recognizing all its modern functions.

PERSONNEL FOR CNC

Computers and machine tools have no intelligence. They cannot think, they cannot evaluate a situation in a rational way. Only people with certain skills and knowledge can do that. In the field of numerical control, the skills are usually in the hands of two key people - one doing the *programming*, the other doing the actual *setup* and *machining*. Their respective numbers and duties typically depend on company preferences, its size, as well as the product manufactured there. However, each position is quite distinct, although many companies combine the two functions into a one, often called a *CNC Programmer/Operator*.

◆ CNC Programmer

CNC programmer is usually a person who has the most responsibility in the CNC machine shop. This person is often responsible for the success of numerical control technology in whole the plant. Equally, this person is also held responsible for problems related to CNC and related operations. Although duties may vary, the programmer is also responsible for a variety of tasks relating to the effective usage of one or more CNC machines. In fact, this person is often accountable for the production and quality parts from all CNC operations.

Many CNC programmers are experienced machinists who have had a practical, hands-on experience as machine tool operators. They know how to read technical drawings and they can comprehend engineering intent behind the design. This practical experience is the main foundation for the ability to '*machine*' a part in an off-machine environment. A good CNC programmer must be able to visualize all tool motions and recognize all restricting factors that may be involved. The programmer must be able to collect, analyze, process and logically integrate all collected data into a single, cohesive and safe part program. In simple terms, the CNC programmer must be able to decide upon the best manufacturing methodology in all respects.

In addition to machining skills, the CNC programmer has to have a good understanding of mathematical principles, mainly application of equations, solution of arcs and angles. Equally important is the knowledge of trigonometry. Even with computerized programming, the knowledge of manual programming methods is absolutely essential to deep and thorough understanding of computer output and to assure control over such output.

The last important quality of a truly professional CNC programmer is his or her ability to listen to other people - the engineers, CNC operators, managers. Good listening skills are the first prerequisites to become flexible. Any professional CNC programmer must be flexible in order to offer high quality in programming.

◆ CNC Machine Operator

The CNC machine tool operator is a complementary position to that of CNC programmer. The programmer and the operator may exist in a single person, as is common in many smaller shops. Although the majority of duties performed by a conventional machine operator has been transferred to CNC programmer, CNC operator also has many unique responsibilities. In typical cases, the operator is responsible for tool and machine setup, for changing of completed parts, often even for some in-process inspection. Many companies expect quality control at the machine - and the operator of any machine tool, manual or computerized, is also responsible for the quality of work done on that machine. One of the most important responsibilities of CNC machine operator is to report findings about each program to the programmer. Even with the best knowledge, skills, attitudes and good intentions, the 'final' program can always be improved. CNC operator, being the one who is the closest to actual machining, knows precisely what extent such improvements can be.

SAFETY RELATED TO CNC WORK

On the walls of many machine shops may hang a safety poster with a simple, yet very powerful message:

The first rule of safety is to follow all safety rules

The heading of this section does not indicate whether safety is oriented at the programming or the machining level. There is no reason for it - safety is totally *independent*. It stands on its own and it governs behavior and activities of everybody in machine shop and outside of it. At first sight, it may appear that safety is something related to machining and machine operations, perhaps to the machine setup as well. That is definitely true but hardly presents a complete picture.

Safety is the most important element in programming, setup, machining, tooling, fixturing, inspection, shipping, and *you-name-it* operation within a typical machine shop daily work. Safety should never be compromised and cannot be overemphasized. Companies talk about safety, conduct safety meetings, display posters, make speeches, call experts. This mass of information and instructions is presented to all of us for some very good reasons. Quite a few

are based on past tragic occurrences - many laws, rules and regulations have been written as a result of inquests and inquiries into serious accidents.

At first sight, it may seem that in CNC work, safety is a secondary issue, not as important as in manual machining. There is a lot of automation in CNC, a part program that is used over and over again, tooling that has been used in the past, a simple setup, etc. All this can lead to complacency and false assumption that safety is taken care of. This is a wrong view that can have serious consequences.

Safety is quite a large subject but a few points that relate to CNC environment are very important. Every machinist should know the hazards of mechanical and electrical devices. The first step towards a safe work place is with a clean work area, where no chips, oil spills and other debris are allowed to accumulate on the floor. Taking care of personal safety is equally important. Loose clothing, jewelry, ties, scarfs, unprotected long hair, improper use of gloves and similar infractions, is dangerous in any machining environment. Protection of one's eyes, ears, hands and feet is strongly recommended.

While a machine is operating, protective devices should be in place and no moving parts should be exposed. Special care should be taken around rotating spindles and automatic tool changers. Other devices that could pose a hazard are pallet changers, chip conveyors, high voltage areas, hoists, etc. Disconnecting any interlocks or other safety features is dangerous - and also illegal, without appropriate skills and authorization.

Modern technology has brought machines that may have nine or more axes, tight work areas, special tool indexing, part transfers, etc. While these features dramatically increase company productivity, they also require additional safety training - and practicing all safety rules.

In CNC programming (manual or computer based), observation of safety rules is equally important. A tool motion can be programmed in many ways. Speeds and feeds have to be realistic, not just mathematically 'correct'. Depth of cut, width of cut, various tool characteristics, they all have a profound effect on overall safety in the shop.

All these ideas are just a very short summary and a reminder that safety should be taken seriously at all times.

2

CNC MILLING

Many different types of CNC machines are used in industry - the majority of them are *CNC machining centers* and *CNC lathes*. They are followed by wire EDM, fabricating machines and machines of special designs. Although the focus of this handbook is on the two types that dominate the market, many general ideas can be applied to other CNC equipment.

CNC MACHINES - MILLING

Description of CNC milling machines is so large, it can fill a thick book all by itself. All machine tools from a simple knee type milling machine up to a five axis profiler can be included in this category. They vary in size, features, suitability for certain work, etc., but they do all have one common denominator - *their primary axes are X and Y axes* - and for this reason, they are called the XY machines.

In the category of XY machines are also wire EDM machine tools, laser and water jet cutting machines, flame cutters, burners, routers, etc. Although they do not qualify as milling type machine tools, they are mentioned because the majority of programming techniques applicable to milling can be applied to these machine types as well. The best example is a *contouring* operation, a process common to many CNC machines.

For the purpose of this handbook, a milling machine can be defined:

Milling machine is a machine capable of a simultaneous cutting motion, using an end mill as the primary cutting tool, along at least two axes at the same time

This definition eliminates all CNC drill presses, since their design covers positioning but not contouring. The definition also eliminates wire EDM machines and a variety of burners, since they are capable of a contouring action but not with an end mill. Users of these machine tools will still benefit from the many subjects covered here. General principles are easily adaptable to the majority of CNC machine tools. For example, a wire EDM uses a very small cutter diameter, in the form of a wire. A laser cutting machine uses laser beam as its cutter, also having a known diameter but the term *kerf* is used instead. The focus will be concentrated on metal cutting machine tools, using various styles of end mills as the primary tool for contouring. Since an end mill can be used in many ways, first look will be at the various types of available milling machines.

◆ Types of Milling Machines

Milling machines can be divided into three categories:

- By the number of axes - two, three or more
- By the orientation of axes - vertical or horizontal
- By the presence or absence of a tool changer

Milling machines where the motion of a spindle is *up and down*, are categorized as *vertical machines*. Milling machines where the spindle motion is *in and out*, are categorized as *horizontal machines* - see Figure 2-1 and 2-2.

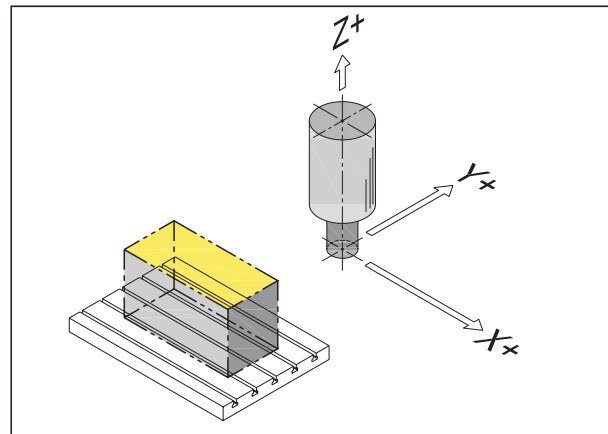


Figure 2-1

Schematic representation of a CNC vertical machining center

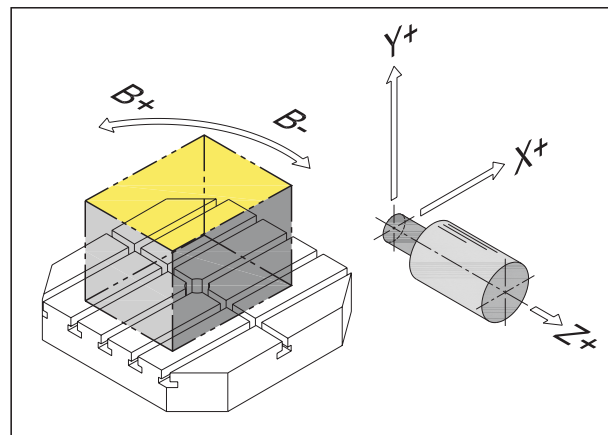


Figure 2-2

Schematic representation of a CNC horizontal machining center

These simplified definitions do not reflect reality of the current state of art in machine tool design. Machine tool industry is constantly changing. New and more powerful machines are designed and produced by many manufacturers in several countries, with more features and flexibility.

The majority of modern machines designed for milling are capable of doing a multitude of machining tasks, not only the traditional milling. These machines are also capable of many other metal removing operations, mainly drilling, reaming, boring, tapping, profiling, thread cutting and many others. They may be equipped with a multi-tool magazine (also known as a carousel), a fully automatic tool changer (abbreviated as ATC) and a pallet changer (abbreviated as APC), a powerful computerized control unit (abbreviated as CNC), and so on. Some machine models may have additional features, such as adaptive control, robot interface, automatic loading and unloading, probing system, high speed machining features, and other marvels of modern technology. The question is - can machine tools of these capabilities be classified as simple CNC milling machines? In two words - certainly not. Milling machines that have at least *some* of the advanced features built-in (usually many features), are known as a separate category of machines - they are called *CNC Machining Centers*. This term is strictly CNC related - a *manual machining center* is a description that does not exist.

◆ Machine Axes

Milling machines and machining centers have at least three axes - X, Y and Z. These machines become even more flexible if they have a fourth axis, usually an indexing or a rotary axis (A-axis for vertical models or B-axis for horizontal models). Even higher level of flexibility can be found on machines with five or more axes. A simple machine with five axes may be a boring mill that has three major axes, plus a rotary axis (usually B-axis) and an axis parallel to the Z-axis (usually W-axis). However, true complex and flexible five-axis profiling milling machine is the type used in aircraft industry, where a multi-axis, simultaneous cutting motion is necessary to machine complex shapes and reach cavities and various angles.

At times, the expression *two and a half axis* machine or a *three and a half axis* machine is used. These terms refer to those types of machines, where simultaneous cutting motion of all axes has certain limitations. For example, a four-axis vertical machine has X, Y and Z-axis as primary axes, plus an indexing table, designated as an A-axis. The indexing table is used for positioning, but it cannot rotate simultaneously with the motion of primary axes. That type of a machine is often called a '*three and a half axis*' machine. By contrast, a more complex but similar machine that is equipped with a fully rotating table, is designed as a true four-axis machine. Rotary table can move simultaneously with the cutting motion of the primary axes. This is a good example of a true '*four axis*' machine tool.

Each machining center is described by its specifications as provided by the machine tool manufacturer. Manufacturers list many specifications as a quick method of comparison between one machine and another. It is not unusual to find a slightly biased information in the descriptive brochure - after all, it is a sales tool.

In the area of milling systems, three most common machine tools are available:

- CNC Vertical Machining Center ... VMC
- CNC Horizontal Machining Center ... HMC
- CNC Horizontal Boring Mill

Programming methods do not vary too much for either type, except for special accessories and options. Some of the major differences will be orientation of machine axes, additional axis for indexing or full rotary motion, and the type of work suitable for individual models. Description of the most common type of a machining center - *Vertical Machining Center (VMC)* - presents a fairly accurate sample of describing other machines of the above group.

◆ Vertical Machining Centers

Vertical machining centers are mainly used for flat type of work, such as plates, where the majority of machining is done on only one face of the part in a single setup.

A vertical CNC machining center can also be used with an optional fourth axis, usually a rotary head mounted on the main table. Rotary head can be mounted either vertically or horizontally, depending on the desired results and the model type. This fourth axis can be used either for indexing or a full rotary motion, depending on the design purchased. In combination with a tailstock (usually supplied), the fourth axis in vertical configuration can be used for machining long parts that need support at both ends.

The majority of vertical machining centers most operators work with are those with an empty table and three-axes configuration.

From the programming perspective, there are at least two items worth mentioning:

- *ONE* - Programming always takes place *from the viewpoint of the spindle*, not the operator's. That means the view is as if looking straight down, at ninety degrees towards the machine table for development of the toolpath motion. *Programmers always view the top of part !*
- *TWO* - Various markers located somewhere on the machine show positive and negative motion of the machine axes. *For programming, these markers should be ignored!* These indicate operating directions, not programming directions. As a matter of fact, typically the programming directions are exactly opposite of the markers on the machine tool

Vertical and Horizontal Machining Center - Typical Specifications		
Description	Vertical Machining Center	Horizontal Machining Center
Number of axes	3 axes (XYZ)	4 axes (XYZB)
Table dimensions	780 x 400 mm 31 x 16 inches	500 x 500 mm 20 x 20 inches
Number of tools	20	36
Maximum travel - X axis	575 mm 22.5 inches	725 mm 28.5 inches
Maximum travel - Y axis	380 mm 15 inches	560 mm 22 inches
Maximum travel - Z axis	470 mm 18.5 inches	560 mm 22 inches
Spindle speed	60-8000 rpm	40 - 4000 rpm
Spindle output	AC 7.5/5.5 kW AC 10/7 HP	AC 11/8 kW AC 15/11 HP
Spindle nose-to-table distance - Z axis	150 - 625 mm 6 - 24.6 inches	150 - 710 mm 6 - 28 inches
Spindle center-to-column distance - Y axis	430 mm 17 inches	30 - 560 mm 1.2 - 22 inches
Spindle taper	No. 40	No. 50
Tool shank size	BT40	CAT50
Number of pallets	N/A	2
Pallet type	N/A	Rotary shuttle
Pallet change time	N/A	5 seconds
Pallet indexing angle	N/A	0.001 degree
Feedrate range	2 - 10000 mm/min 0.100 - 393 in/min	1 - 10000 mm/min 0.04 - 393 in/min
Rapid traverse rate	40000 mm/min (XY) - 35000 mm/min (Z) 1575 in/min (XY) - 1378 in/min (Z)	30000 mm/min (XY) - 24000 mm/min (Z) 1181 in/min (XY) - 945 in/min (Z)
Tool selection	Random memory	Random memory
Maximum tool diameter	80 mm (150 w/empty pockets) 3.15 inches (5.9 w/empty pockets)	105 mm 4.1 inches
Maximum tool length	300 mm 11.8 inches	350 mm 13.75 inches
Tool change time (chip-to-chip)	2.5 seconds	4 seconds

◆ Horizontal Machining Centers

Horizontal CNC Machining Centers are also categorized as multi-tool and versatile machines, and are used for cubical parts, where the majority of machining has to be done on more than one face in a single setup.

There are many applications in this area. Common examples are large parts, such as pump housings, gear cases, manifolds, engine blocks and so on. Horizontal machining centers always include a special indexing table and are often equipped with a pallet changer and other features.

Because of their flexibility and complexity, CNC horizontal machining centers are priced significantly higher than vertical CNC machining centers.

From programming point of view, there are several unique differences, mainly relating to the *Automatic Tool Changer*; the indexing table, and - in some cases - to the additional accessories, for example, pallet changer. All differences are relatively minor. Writing a program for horizontal machining centers is no different than writing a program for vertical machining centers.

◆ Horizontal Boring Mill

Horizontal boring mill is just another CNC machine. It closely resembles a CNC horizontal machining center, but it does have its own differences. Generally, a horizontal boring mill is defined by the lack of some common features, such as the *Automatic Tool Changer*. As the name of the machine suggests, its primary purpose is boring operations, mainly lengthy bores. For that reason, the spindle reach is extended by a specially designed quill. Another common feature is an axis parallel to the Z-axis, called the W-axis. Although this is, in effect, the fifth axis designation (X, Y, Z, B, W), a horizontal boring mill cannot be called a true five axis machine. Both the Z-axis (quill) and the W-axis (table) work in opposite directions - towards each other - so they can be used for large parts and most of *hard-to-reach* areas. It also means, that during drilling, the machine table moves against an extended quill. Quill is a physical part of the spindle. It is in the spindle where the cutting tool rotates - but the in-and-out motions are done by the table. Think of the alternate method offered on horizon-

tal boring mills - if the quill were to be very long, it would lose its strength and rigidity. The better way was to split the traditional single Z-axis movement into two - the quill extension along Z-axis will move only part of the way towards the table and the table itself - the new W-axis - will move another part of the way towards the spindle. They both meet in the area of the part that could be machined using all other machine tool resources.

Horizontal boring mill may be called a 3-1/2 axis CNC machine, but certainly not a 5-axis CNC machine, even if the number of programmed axes is five. Programming procedures for CNC boring mills are very similar to the horizontal and vertical CNC machining centers.

◆ Typical Specifications

On the preceding page is a comprehensive chart showing typical specifications of a *CNC Vertical Machining Center* and a *CNC Horizontal Machining Center*. These specifications are side by side in two columns, strictly for convenience, not for any comparison purposes. These are two different machine types and comparison is not possible for all features. In order to compare individual machine tools within a certain category, machine tool specifications provided by machine manufacturer often serve as the basis for comparison. These specifications are contained in a list of verifiable data, mainly technical in nature, that describes the individual machine by its main features. Machine tool buyers frequently compare many brochures of several different machines as part of the pre-purchase process. Managers and process planners compare individual machines in the machine shop and assign any available workload to the most suitable machine.

A fair and accurate comparison can be made between two vertical machining centers or between two horizontal machining centers, but cannot be done fairly to compare between any two different machine types.

In a typical machine specification chart, additional data may be listed, not included in the earlier chart, depending on the exact features. In this handbook, the focus is on only those specifications that are of interest to a CNC programmer and, to a large extent, a CNC operator.

3

CNC TURNING

CNC MACHINES - TURNING

Conventional engine lathe or a turret lathe is a common machine in just about every machine shop. A lathe is used for machining cylindrical or conical work, such as shafts, rings, wheels, bores, threads, etc. The most common lathe operation is removal of material from a round stock, using a turning tool for external cutting. A lathe can also be used for internal operations such as boring, as well as for facing, grooving, threading, etc., if a proper cutting tool is used. Turret lathes are usually weaker in machining power than engine lathes, but they do have a special holder that stores several mounted cutting tools. An engine lathe has often only one or two cutting tools mounted at a time, but has more machining power.

Typical lathe work controlled by a CNC system uses machines known in industry as the *CNC Turning Centers* - or more commonly - *CNC lathes*.

The term '*turning center*' is rather unpopular, but an accurate overall description of a computerized lathe (a CNC lathe) that can be used for a great number of machining operations during single setup. For example, in addition to the standard lathe operations such as turning and boring, CNC lathe can be used for drilling, grooving, threading, knurling and even burnishing. It can also be used in different modes, such as chuck and collet work, barfeeder, or between centers. Many other combinations also exist. CNC lathes are designed to hold several tools in special turrets, they can have a milling attachment (live tooling), indexable chuck, a sub-spindle, tailstock, steady rest and many other features not always associated with a conventional lathe design. Lathes with more than four axes are also common. With constant advances in machine tool technologies, more CNC lathes appear on the market that are designed to do a number of operations in a single setup, many of them traditionally reserved for a mill or machining center.

◆ Types of CNC Lathes

Basically, CNC lathes can be categorized by the *type of design* and by the *number of axes*. The two basic types are a *vertical* CNC lathe and a *horizontal* CNC lathe. Of the two, the horizontal type is by far the most common in manufacturing and machine shops. A vertical CNC lathe (incorrectly called a vertical boring mill) is somewhat less common but is irreplaceable for a large diameter work. For CNC programmer, there are no significant differences in programming approach between the two lathe types.

◆ Number of Axes

The most common distinction of different CNC lathes is by the number of programmable axes. Vertical CNC lathes have two axes in most designs available. The much more common CNC horizontal lathes, commonly designed with two programmable axes, are also available with three, four, six, and more axes, thus adding extra flexibility to manufacturing of more complex parts.

A typical horizontal CNC lathe can further be described by the *type* of engineering design:

- FRONT lathe ... an engine lathe type
- REAR lathe ... a unique slant bed type

Slant bed type is very popular for general work, because its design allows cutting chips to fall away from the CNC operator and, in case of an accident, forces the part to fall down into a safe area, towards chip conveyer.

Between the categories of flat bed and slant type lathes, front and rear lathes, horizontal and vertical lathe designs, there is another variety of a lathe. This category describes CNC lathes by the *number of axis*, which is probably the simplest and most common method of lathe identification.

AXES DESIGNATION

A typical CNC lathe is designed with two standard axes - one axis is the X-axis, the other axis is the Z-axis. Both axes are perpendicular to each other and represent the typical two-axis lathe motions. X-axis also represents *cross travel* of the cutting tool, Z-axis represents its *longitudinal motion*. All varieties of cutting tools are mounted in a turret (a special tool magazine) and can be external or internal. Because of this design, a turret loaded with all cutting tools moves along both X and Z axes, which means all tools are in the work area at all times.

Following the established standards of milling machines and machining centers, the only machine axis capable of making a hole by methods of drilling, boring, piercing or punching, is the Z-axis.

In CNC lathe work, the traditional axis orientation for a horizontal type of lathe is *upwards* and *downwards* motion for the X-axis, and *left* and *right* motion for the Z-axis, when looking from the machinist's position. This view is shown in the following three illustrations *Figure 3-1*, *Figure 3-2*, and *Figure 3-3*.

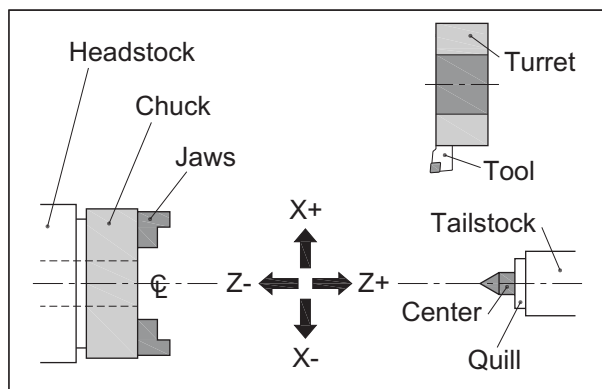


Figure 3-1

Typical configuration of a two axis slant bed CNC lathe - rear type

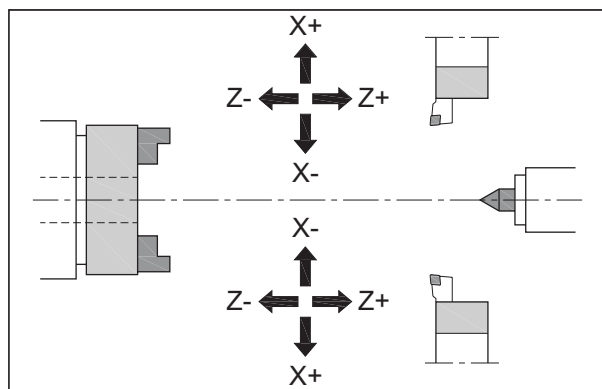


Figure 3-2

Typical configuration of a CNC lathe with two turrets

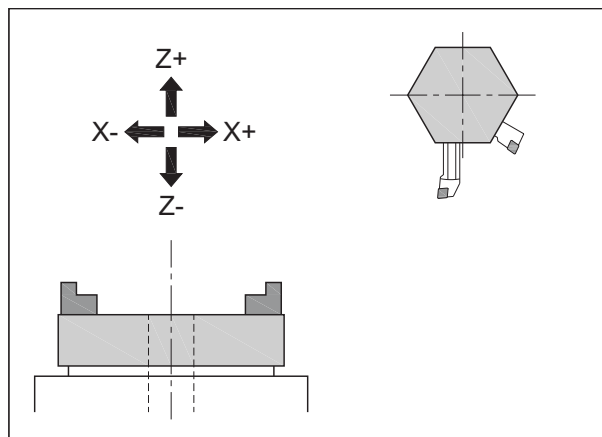


Figure 3-3

Schematic representation of a vertical CNC lathe

This is true for both front and rear lathes, and for lathes with three or more axes. Chuck face is oriented vertically to the horizontal spindle center line for all horizontal lathes. Vertical lathes, due to their design, are rotated by 90°, where the chuck face is oriented horizontally to the vertical spindle center line.

In addition to the X and Z primary axes, the multi-axis lathes have individual descriptions of each additional axis, for example, the C-axis or Y-axis, used for milling operations, using the so called *live tooling* (see page 457). More details on the subject of coordinate system and machine geometry are available in the next chapter.

◆ Two-axis Lathe

This is the most common type of CNC lathes. The work holding device, usually a chuck, is mounted on the left side of the machine (as viewed by the operator). Rear type, with slant bed, is the most popular design for general work. For some special work, for example in the petroleum industry (where turning tube ends is a common work), a flat bed is usually more suitable. Cutting tools are held in a specially designed indexing turret that can hold four, six, eight, ten, twelve and more tools. Many such lathes also have two turrets, one on each side of the spindle center line.

Advanced machine tool designs incorporate tool storage away from the work area, similar to design of machining centers. Tens and even hundreds of cutting tools may be stored and used for a single CNC program. Many lathes also incorporate a quick changing tooling system.

◆ Three-axis Lathe

Three-axis lathe is essentially a two-axis lathe with an additional axis. This axis has its own designation, usually as a C-axis in absolute mode (H-axis in incremental mode), and is fully programmable. Normally, the third axis is used for cross-milling operations, slot cutting, bolt circle holes drilling, hex faces, side faces, helical slots, etc. This axis can replace some simple operations on a milling machine, reducing the setup time for the job. Some limitations do apply to many models, for example, the milling or drilling operations can take place only at positions projecting from the tool center line to the spindle center line (within a machining plane), although others offer off-center adjustments.

The third axis has its own power source but the power rating is relatively lower when compared with the majority of machining centers. Another limitation may be the smallest increment of the third axis, particularly on the early three axis lathes. Smallest increment of one degree is certainly more useful than an increment of two or five degrees. Even better is an increment of 0.1°, 0.01°, and commonly 0.001° on the latest models. Usually, lathes with three axes offer a very fine radial increment that allows a simultaneous rotary motion. Those with low increment values are usually designed with an oriented spindle stop only.

From the perspective of CNC part programming, the additional knowledge required is a subject not difficult to learn. General principles of milling apply and many programming features are also available, for example, fixed cycles and other shortcuts.

◆ Four-axis Lathe

By design, a four-axis CNC lathe is a totally different concept than a three-axis lathe. As a matter of fact, to program a four-axis lathe is nothing more than programming *two two-axis lathes at the same time*. That may sound strange at first, until the principle of a four-axis CNC lathe becomes clearer.

There are actually two controls and two sets of XZ axes, one for each pair (set) of axes. Only one program may be used to do the *external* - or *outside* - diameter roughing (OD) and another program to do the *inside* - or *internal* - roughing (ID). Since a four-axis lathe can work with each pair of axes *independently*, the OD and ID can be machined at the same time, doing two different operations simultaneously. The main keys to a successful 4-axis lathe programming is coordination of the tools and their operations, timing of the tool motions and a generous sense of healthy compromise.

For several reasons, both pairs of axes cannot work all the time. Because of this restriction, special programming features such as synchronized waiting codes (typically *Miscellaneous Function*), the ability to estimate how much time each tool requires to complete each operation, etc., are required. There is a level of compromise here, because only *one* spindle speed can be used for both active cutting tools, although feedrate is independent for both pairs of axes. This means that some machining operations simply cannot be done simultaneously.

Not every lathe job benefits from 4-axis machining. There are cases when it is more costly to run a job on a 4-axis lathe inefficiently and it may be very efficient to run the same job on a 2-axis CNC lathe.

◆ Six-axis Lathe

Six-axis CNC lathes are specially designed lathes with a twin turret and a set of three axes per turret. This design incorporates many tool stations, many of them power driven, as well as back-machining capabilities. Programming these lathes is similar to programming a three-axis lathe twice. Control system automatically provides synchronization, when necessary.

A small to medium size six-axis CNC lathe is popular choice of screw machine shops and industries with similar small parts and large volume applications.

FEATURES AND SPECIFICATIONS

A look at a typical promotional brochure describing a CNC machine tool is very useful in many respects. In most cases, the artwork quality is impressive, the printing, photographs, paper selection and the use of colors is equally well done. It is the purpose of the brochure to make a good marketing tool and attract the potential buyer.

However, there is more in a promotional brochure than just attractive illustrations - in fact, a well designed brochure offers a wealth of technical information, describing the machine tool in many details. These are *features* and *specifications* the CNC machine tool manufacturer considers important to the customer.

In the majority of brochures, there are practical data that can be used in programming a particular CNC machine, a lathe in the example.

◆ Typical Machine Specifications

A typical horizontal CNC lathe, with two axes and a slant bed design, may have the following specifications (taken from an actual brochure):

Description	Specification
Number of axes	Two (X, Z) or three (X, Z and C)
Maximum swing over bed	560 mm 22.05 inches
Maximum turning diameter	350 mm 13.76 inches
Maximum turning length	550 mm 21.65 inches
Spindle bore	85 mm 3.34 inches
Bar capacity	71 mm 2.79 inches
Number of tools	12
Tool size square	25 mm 1 inch
Tool size round	Ø40 mm Ø1.57 inches
Indexing time	0.15 second
Axis travel in X-axis	222 mm 8.75 inches
Axis travel in Z-axis	635 mm 25 inches
Rapid traverse rate X-axis	16000 mm/min 629 in/min
Rapid traverse rate Z-axis	24000 mm/min 944 in/min
Cutting feedrate	0.01 - 500 mm/rev 0.0001 - 19.68 in/rev
Chuck size	254 mm 10 inches
Main spindle motor	AC 15/11kW AC 20/14.7HP
Spindle speed	35 - 3500 rpm
Minimum input increment	0.001 mm 0.0001 inch
Motorized head:	
Number of rotating tools	12
Rotating tool speed	30 - 3600 r/min
Milling motor	AC 3.7/2.2 kW AC 5/2.95 HP
Collet size	1 - 16 mm 0.04 - 0.63 inches
Tap size	M3 - M16 metric #5 - 5/8 inches

It is very important to understand all specifications and features of the CNC machine tools in the shop. Many features relate to the control system, many others to the machine tool itself. In CNC programming, many important decisions are based on one or several of these features, for example number of tool stations available, maximum spindle speed and others.

◆ Control Features

The last item in understanding the overall description of a CNC lathe is a look at some control features unique to lathes and how they differ from a typical milling control. The subject of control features is described in more detail, starting on *page 19*.

At this time, some features and program codes may not make much sense - they are included for reference only. Common and typical features are listed:

- X-axis represents a diameter, not a radius
- *Constant surface speed (CSS)* - also known as *cutting speed (CS)* - is standard control feature (G96 for cutting speed and G97 for spindle speed)
- Absolute programming mode is X or Z or C
- Incremental programming mode is U or W or H
- Thread cutting of various forms (including taper and circular) can be performed, depending on the control model
- Dwell can use the P, U or X address (G04)
- Tool selection uses 4-digit identification
- Feedrate selection (normal) is in mm/rev or in/rev
- Feedrate selection (special) is in m/min or in/min
- Rapid traverse rate different for X and Z axes
- Multiple repetitive cycles for turning, boring, facing, contour repeat, grooving, and threading are available
- Feedrate override is common from 0 to 200% in 10% increments (on some lathes only from 0 to 150%)
- X-axis can be mirrored
- Tailstock can be programmable
- Automatic chamfering and corner rounding using C and R (or I / K) addresses in G01 mode
- Thread cutting feedrate available with six-decimal place accuracy (for inch units)
- Thread cutting feedrate available with five-decimal place accuracy (for metric units) - seldom needed
- Least input increment in X-axis is 0.001 mm or 0.0001 inches on diameter - one half of that amount per side

4

COORDINATE GEOMETRY

One of the first major steps towards basic understanding of CNC principles and geometrical concepts is thorough understanding of a subject known in mathematics as the *system of coordinates*. System of coordinates is founded on a number of mathematical principles dating back over four hundred years. The most important of these principles are those that can be applied to CNC technology of today. In various publications on mathematics and geometry, these principles are often listed under the headings such as the *real number system* and the *rectangular coordinates*.

REAL NUMBER SYSTEM

One main key to understanding rectangular coordinates is understanding of basic math - arithmetic, algebra and geometry. The key knowledge in this area is knowledge of the *real number system*. Within the real number system, there are ten available numerals (digits), 0 to 9 (zero to nine), that can be used in any of the following groups:

- Zero integer ... 0
- Positive integers ... 1, 2, +3, 10, 12943, +45
(with or without sign)
- Negative integers ... -4, -381, -25, -77
(minus sign required)
- Fractions ... $1/8$, $3/16$, $9/32$, $35/64$
- Decimal fractions ... 0.185, 0.2, 0.546875, 3.5, 15.0

All groups are used on a daily basis. These groups represent the mainstream of just about all applications of numbers in modern life. In CNC programming, the primary goal is to use numbers to 'translate' engineering drawing - based on its dimensions - into a specific cutter path.

Computerized Numerical Control means control by the numbers using a computer. All drawing information has to be translated into CNC program, using primarily numbers. Numbers are also used to describe commands, functions, comments, and so on. The mathematical concept of a real number system can be expressed graphically on a horizontal or a vertical line, called the *number scale*, where all divisions have the same length - *Figure 4-1*.

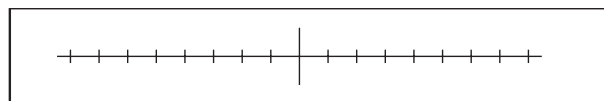


Figure 4-1

Graphical representation of the Number Scale

Length of each division on the scale represents the unit of measurement in a convenient and generally accepted scale. It may come as a surprise that this concept is used every day. For example, a simple ruler used in schools is based on the number scale concept, regardless of measuring units. Weight scales using tons, pounds, kilograms, grams and similar units of mass are other examples. A simple household thermometer uses the same principle. Other similar examples are available as well.

RECTANGULAR COORDINATE SYSTEM

Rectangular coordinate system is a concept used to define a planar 2D point (two dimensions), using the XY coordinates, or a spacial 3D point (three dimensions), using the XYZ coordinates. This system was first defined in the 17th century by a French philosopher and mathematician *Rene Descartes (1596-1650)*. His name is used as an alternative name of the rectangular coordinate system, it is called the *Cartesian Coordinate System* - see *Figure 4-2*.

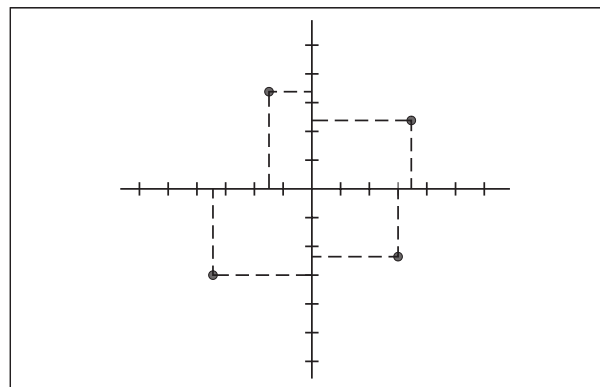


Figure 4-2

Rectangular coordinate system = Cartesian coordinate system

The concepts used in design, drafting *and* in numerical control are over 400 years old. A given point can be mathematically defined on a plane (two coordinate values) or in space (three coordinate values). The definition of one point is relative to another point as a distance parallel with one of three axes that are perpendicular to each other. In a plane, only two axes are required, in space, all three axes must be specified. In programming, point represents an exact location. If such a location is on a plane, the point is defined as a 2D point, along two axes. If the location is in space, the point is defined as a 3D point, along three axes.

When *two* number scales that intersect at right angles are used, mathematical basis for a *rectangular coordinate system* is created. Several terms emerge from this representation, and all have an important role in CNC programming. Their understanding is very important for further progress.

◆ Axes and Planes

Each major line of the number scale is called an *axis*. It could have either vertical or horizontal orientation. This very old principle, when applied to CNC programming, means that at least two axes - *two number scales* - will be used. This is the mathematical definition of an axis:

An axis is a straight line passing through the center of a plane or a solid figure, around which the parts are symmetrically arranged

This definition can be enhanced by a statement that an axis can also be a *line of reference*. In CNC programming, an axis is used as a reference all the time. The definition also contains the word '*plane*'. A *plane* is a term used in 2D applications, while a *solid object* is used in 3D applications. Mathematical definition of a plane is:

A plane is a surface in which a straight line joining any two of its points will lie wholly on the surface

From the *top viewpoint* of the observer, looking straight down on the illustration Figure 4-3, a *viewing direction* is established. This is often called *viewing a plane*.

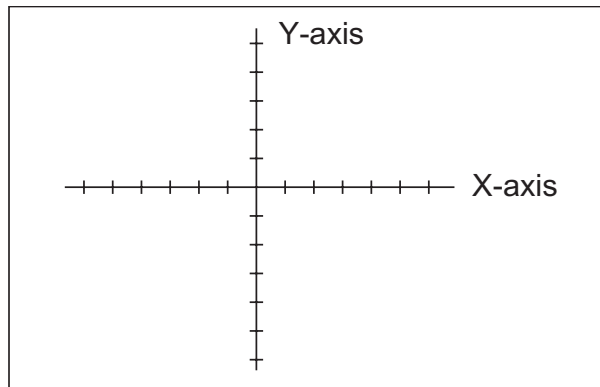


Figure 4-3

Axis designation - viewing a plane

Mathematical designation is fully implemented in CNC

A plane is a 2D entity - the letter X identifies its horizontal axis, the letter Y identifies its vertical axis. Such plane is called the *XY plane*. Defined mathematically, the horizontal axis is *always* listed as the *first* letter of the pair. In drafting and CNC programming, this plane is also known as the *Top View* or a *Plan View*. Other planes are also used in CNC, but not to the same extent as in CAD/CAM work.

◆ Point of Origin

Another term that emerges from the rectangular coordinate system is called the *point of origin*, or just *origin*. It is the exact point where the two perpendicular axes intersect. This point has a zero coordinate value in each axis, specified as planar X0Y0 and spacial X0Y0Z0 - Figure 4-4.

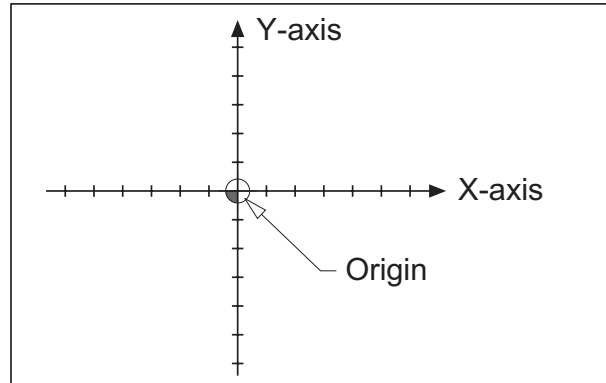


Figure 4-4

Point of origin - intersection of axes

This intersection called *origin* has a rather special meaning in CNC programming. This origin point acquires a new name, one typically called the *program reference point*. Other terms are also used: *program zero*, *part reference point*, *workpiece zero*, *part zero*, and probably a few others - all with the same meaning and same purpose.

◆ Quadrants

Viewing the two intersecting axes and the new plane, four distinct areas can be clearly identified. Each area is bounded by two axes. These areas are called *quadrants*. Mathematically defined,

A quadrant is any one of the four parts of the plane formed by the system of rectangular coordinates

The word *quadrant*, from Latin word *quadrans* or *quadrantis*, means *the fourth part*. It suggests *four* uniquely defined areas or *quadrants*. Looking down in the top view at the two intersecting axes, the following definitions apply to quadrants. They are mathematically correct and are used in all CNC/CAD/CAM applications:

Quadrant I	UPPER RIGHT
Quadrant II	UPPER LEFT
Quadrant III	LOWER LEFT
Quadrant IV	LOWER RIGHT

Quadrants are always defined in the *counterclockwise* direction, from the horizontal X-axis and the naming convention uses *Roman* - not *Arabic* - numbers normally used.

Numbering of quadrants always starts at the *positive* side of the *horizontal* axis. Figure 4-5 illustrates the definitions.

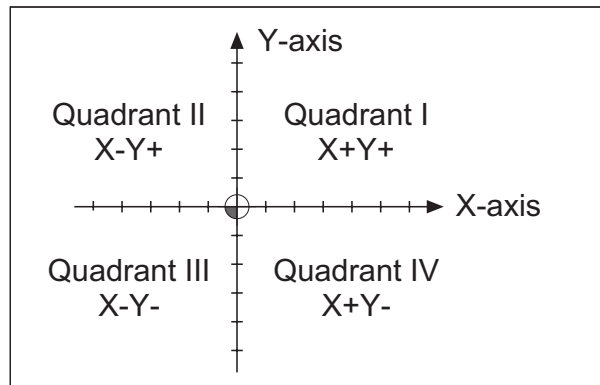


Figure 4-5

Quadrants in the XY plane and their identification

Any point coordinate value can be positive, negative or zero. All point coordinates are determined solely by their *location* in a particular quadrant and individual distances along an axis, again, relative to origin - Figure 4-6.

Point location	Coordinate	
	X-axis	Y-axis
Quadrant I	+	+
Quadrant II	-	+
Quadrant III	-	-
Quadrant IV	+	-

Figure 4-6

Algebraic signs for a point location in plane quadrants

■ IMPORTANT:

- ... If the defined point lies exactly on the X-axis, it has the Y value equal to zero (Y0)
- ... If the point lies exactly on the Y-axis, it has the X value equal to zero (X0)
- ... If the point lies exactly on both X and Y axes, both X and Y values are zero (X0 Y0).

X0Y0Z0 is the point of origin. In part programming, positive values are written *without* the plus sign - Figure 4-7.

◆ Right Hand Coordinate System

In all illustrations of *number scale*, *quadrants* and *axes*, the *origin* divides each axis into two portions. The zero point - *the point of origin* - separates the *positive* section of an axis from the *negative* section. In the right-hand coordinate system, the *positive* axis starts at origin and is directed towards the *right* for X-axis, *upwards* for Y-axis and *towards the perpendicular viewpoint* for Z-axis. Opposite directions are always negative.

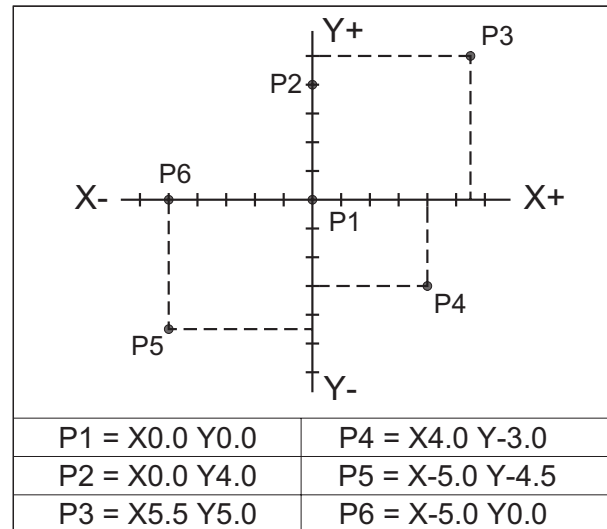


Figure 4-7

Coordinate definition of points within rectangular coordinate system
Point P1 = Origin = X0Y0

If these directions were superimposed over a human right hand, they would correspond to the direction from the root of thumb or finger towards its tip. Thumb would point in the X+ direction, index finger in the Y+ direction and middle finger in the Z+ direction.

CNC machines are normally programmed using the so called *absolute* coordinate method, that is based on the point of origin being X0Y0Z0. This absolute programming method follows very strictly the rules of rectangular coordinate geometry and all concepts covered in this chapter.

MACHINE GEOMETRY

Machine geometry defines the relationship of distances and dimensions between *fixed point of the machine* and *selectable point of the part*. Typical geometry of CNC machines uses the right hand coordinate system. Positive and negative axis direction is determined by an established viewing convention. The general rule for Z-axis is that it is always the axis along which a simple hole can be machined with a single point tool, such as a drill, reamer, wire, laser beam, etc. Figure 4-8 on the next page illustrates standard orientation of planes for XYZ type machine tools.

◆ Axis Orientation - Milling

A typical vertical machining center has three controlled axes, defined as X-axis, Y-axis, and Z-axis. X-axis is parallel to the longest dimension of machine table, Y-axis is parallel to the shortest dimension of the table and Z-axis is the spindle movement. On a vertical CNC machining center, X-axis is the table longitudinal direction, Y-axis is the saddle cross direction, and Z-axis is the *spindle* direction.

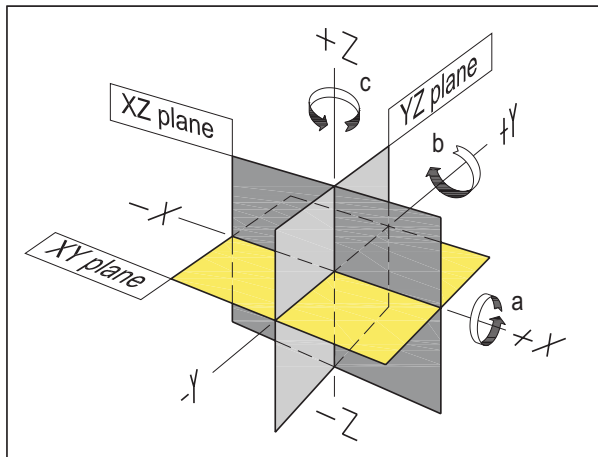


Figure 4-8
Standard orientation of planes and CNC machine tool axes

For CNC horizontal machining centers, the terminology is changed due to design of these machines. X-axis is the table *longitudinal* direction, Y-axis is the column direction and Z-axis is the *spindle* direction. Horizontal machine can be viewed as a vertical machine rotated in space by ninety degrees. Additional feature of a horizontal machining center is indexing *B-axis*. Typical machine axes applied to CNC vertical machines are illustrated in Figure 4-9.

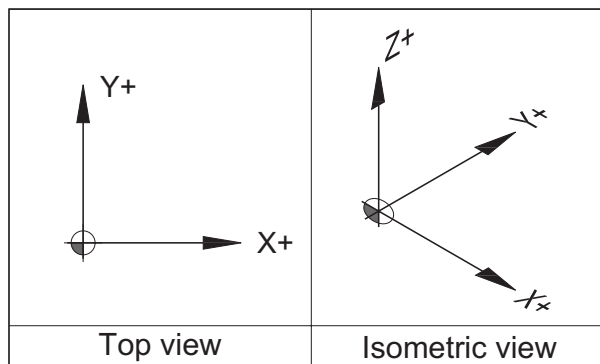


Figure 4-9
Basic axes of a typical vertical CNC machining center

◆ Axis Orientation - Turning

Standard CNC lathes have two axes, X and Z. More axes are available, but they are not important at this point. Special additional axes, such as C-axis and Y-axis, are designed for milling operations (live tooling) and require unique version of a standard CNC lathe.

What is much more common for CNC lathes in industry, is the double orientation of XZ axes. CNC lathes are separated as *front* and *rear* lathes. An example of a front lathe is similar to the conventional engine lathe. All slant bed lathe types are of the rear kind. Identification of axes in industry have not always followed mathematical principles.

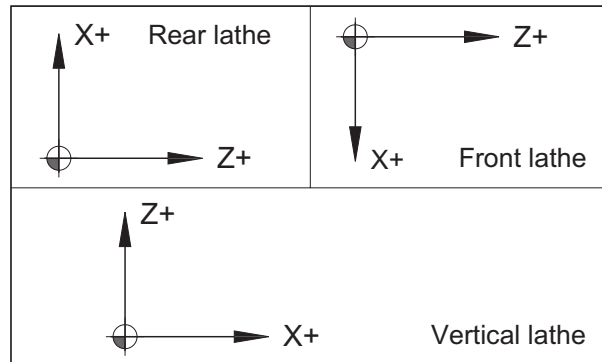


Figure 4-10
Typical machine axis orientation for various CNC lathes

Another lathe variety, a vertical CNC lathe, is basically a horizontal lathe rotated 90°. Typical axes for horizontal and vertical machine axes, as applied to turning, are illustrated in Figure 4-10.

◆ Additional Axes

A CNC machine of any type can be designed with one or more additional axes, normally designated as the secondary - or *parallel* - axes using the U, V and W letters. These axes are normally parallel to the primary X, Y and Z axes respectively. For a rotary or an indexing applications, additional axes are defined as A, B and C axes, as being rotated about the X, Y and Z axes, again in their respective order. Positive direction of a rotary (or an indexing) axis is the direction required to advance a right handed screw in the positive X, Y or Z axis. Relationship between the primary and supplementary axes is shown in Figure 4-11.

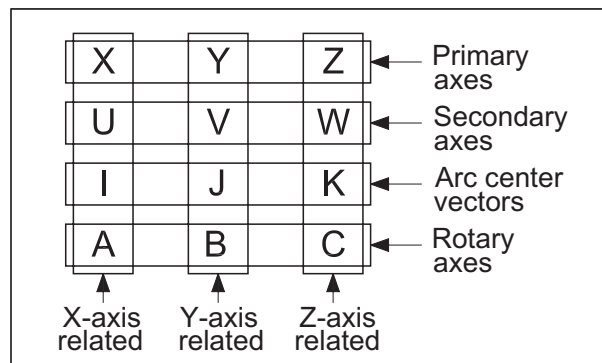


Figure 4-11
Relationship of primary and supplementary machine axes

Arc center modifiers (sometimes called arc center vectors) are not true axes, yet they are also related to primary axes XYZ. This subject will be described in the section on *Circular Interpolation*, starting on page 243.

5

CONTROL SYSTEM

A machine unit equipped with a computerized numerical control system is commonly known as a CNC machine. In an analogy of the machine tool being the *body* of a CNC machine system, the control unit is its *brain*, its nerve center. There are no levers, no knobs and no handles on a CNC machine the way they function on conventional milling machines and lathes. All machine speeds, feeds, axes motions and hundreds of other tasks are programmed by a CNC programmer and controlled by a computer that is major part of the CNC unit. To make a program for a CNC machine tool means to make a program for the control system. True, the machine tool is a major consideration as well, but it is the *control unit* that determines the program *format*, its *structure* and its *syntax*.

In order to fully understand CNC programming process, it is important to understand not only the intricacies of how to machine a part, what tools to select, what speeds and feeds to use, how to setup the job and many other features. It is equally important to know how the computer, the CNC unit, actually works without the need to be an expert in electronics or a computer scientist. *Figure 5-1* shows an actual Fanuc control panel.

Machine manufacturers add their own operation panel, with all switches and button needed to operate their CNC machine and all its features. A typical operation panel is illustrated in *Figure 5-2*. Another item required for the system, the *handle*, will be described as well.

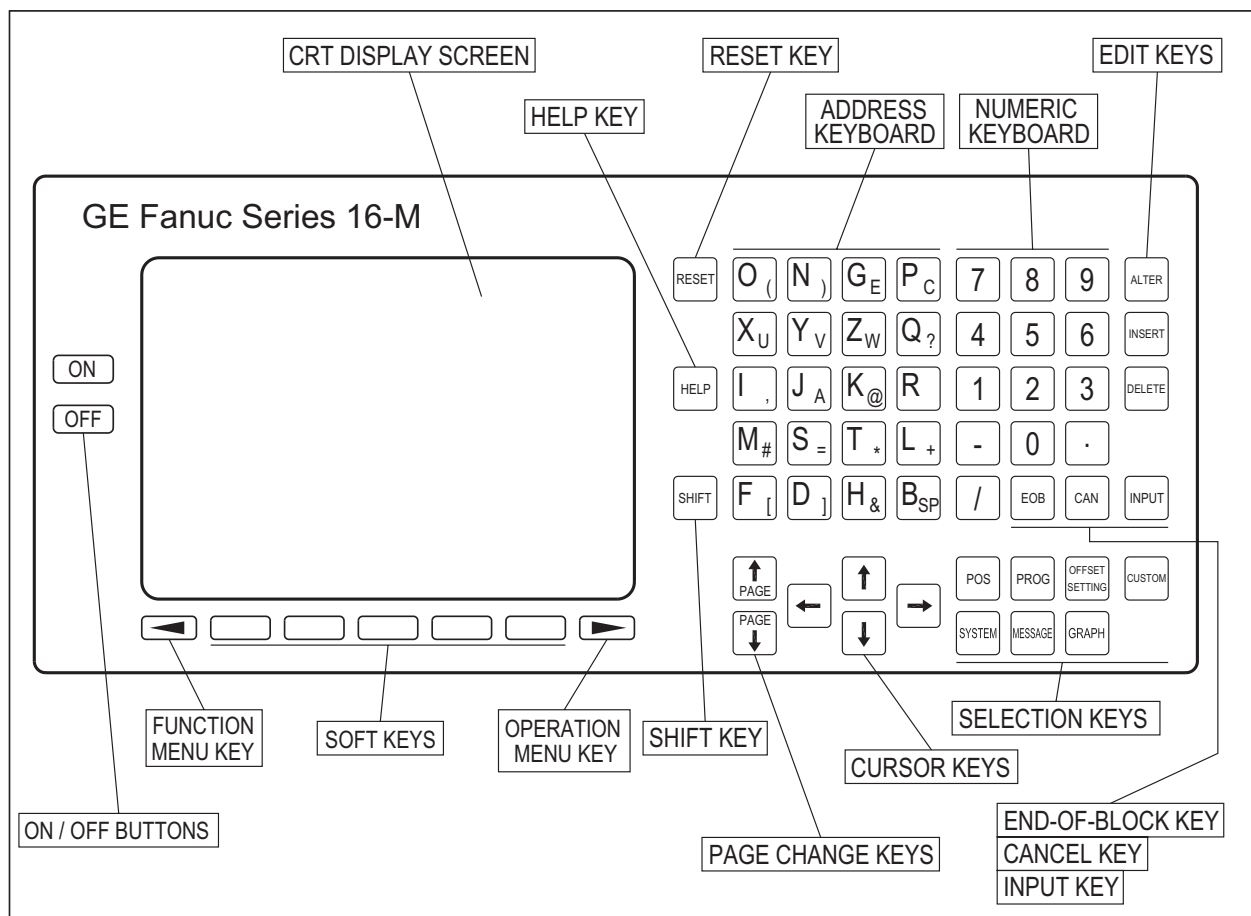


Figure 5-1

A typical example of a Fanuc control panel - actual layout and features will vary on different models (Fanuc 16M)

GENERAL DESCRIPTION

Even a brief look at any control unit reveals that there are two basic components - one is the *operation panel*, full of rotary switches, toggle switches and push buttons. The other component is the *display screen* with a keyboard or a keypad. A programmer who does not normally work on the CNC machine will seldom, if ever, have a reason to use either the operation panel or the display screen. They are available at the machine to the CNC machine operator, and used for machine setup as well as to control the activities of the machine.

Should the CNC programmer be at least interested in the machine operation? Is it necessary for the programmer to *know* and *understand* all functions of the control system? There is only one answer to both questions - *definitely yes*.

The control unit - *the CNC system* - contains features that only work in conjunction *with* the program, it does not do anything useful on its own. Some features can be used *only* if the program itself supports them. All switches and buttons and keys are used by the machine operator, to exercise control over program execution and machining process.

◆ Operation Panel

Depending on CNC machine type, the following table covers the most typical and common features found on modern operation panel. There are some small differences for operation of a machining center and a lathe, but both operation panels are similar. As with any general reference book, it is always a good idea to double check with the manufacturer specifications and recommendations. It is common that many machines used in the shop have some special features.

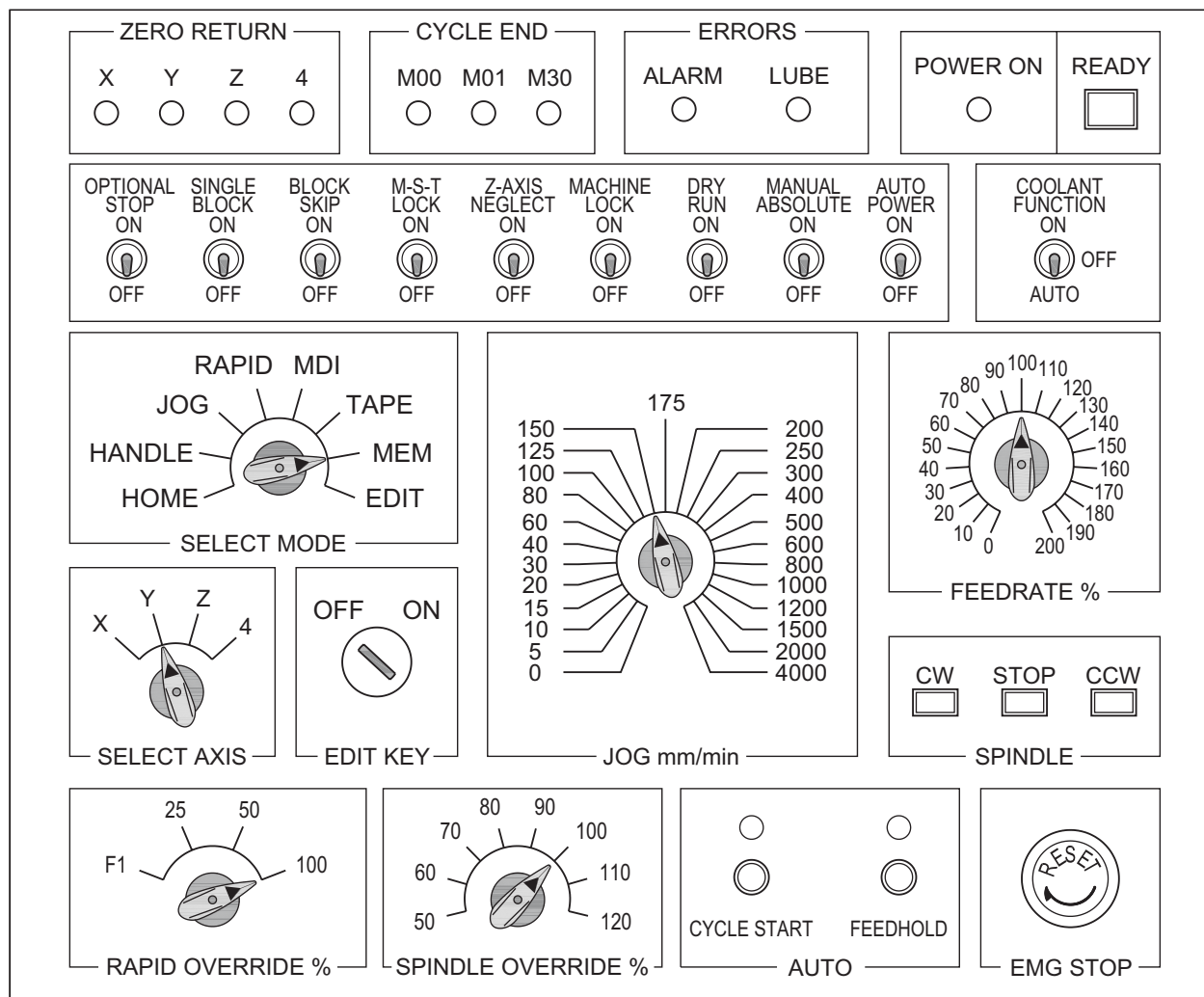


Figure 5-2

A typical operation panel of a CNC machining center - actual layout and features will vary on different models

Feature	Description
ON / OFF switch	Power and control switch for the main power and control unit
Cycle Start	Starts program execution or MDI command
Emergency Stop	Stops all machine activity and turns off power to the control unit
Feedhold	Temporarily stops motion of all axes
Single Block	Allows program run one block at a time
Optional Stop	Temporarily stops program execution (M01 required in program)
Block Skip	Ignores blocks preceded with a forward slash (/) in the program
Dry Run	Enables program testing at fast feedrates (without a mounted part)
Spindle Override	Overrides programmed spindle speed, usually within 50-120% range
Feedrate Override	Overrides programmed feedrate, usually within 0-200% range
Chuck Clamp	Shows current status of the chuck clamping (Outside / Inside clamping)
Table Clamp	Shows current status of table clamping
Coolant Switch	Coolant control ON / OFF / AUTO
Gear Selection	Shows current status of working gear range selection
Spindle Rotation	Indicates spindle rotation direction (clockwise or counterclockwise)
Spindle Orientation	Manual orientation of the spindle
Tool Change	Switch allowing a manual tool change
Reference Position	Switches and lights relating to setup of machine, from reference position
Handle (MPG)	Manual Pulse Generator (MPG), used for Axis Select and Handle Increment switches
Tailstock Switch	Tailstock and/or quill switch to manually position the tailstock
Indexing Table Switch	Manually indexes machine table during setup
MDI Mode	Manual Data Input mode

Feature	Description
AUTO Mode	Allows automatic operations
MEMORY mode	Allows program execution from memory of the CNC unit
TAPE / EXT or DNC mode	Allows program execution from an external device, such as a desktop computer (DNC) or a punched tape
EDIT mode	Allows changes to be made to a program stored in CNC memory
MANUAL Mode	Allows manual operations during setup
JOG Mode	Selects jog mode for setup
RAPID Mode	Selects rapid mode for setup
Memory Access	Key (switch) to allow program editing
Error Lights	Red light indicating an error

Even if some features may not be listed, virtually all of those in the table are somewhat related to CNC program. Many control systems have unique features of their own. These features must be known to the CNC operator. The program supplied to the machine should be *flexible*, not *rigid* - it should be 'user friendly'.

◆ Screen Display and Keyboard

Screen display is the 'window' to control operation. Any active program can be viewed, including the control status, current tool position, various offsets, parameters, even a graphic representation of the toolpath. On all CNC units, individual monochrome or color screens can be selected to have the desired display at any time, using the input keys (keyboard pads and soft keys). Setting for international languages is also possible.

Keyboard pads and soft keys are used to *input* instructions to the control. Existing programs can be modified or deleted, new programs can be added. Using keyboard input, not only the machine axes motion can be controlled, but the spindle speed and feedrate as well. Changing internal parameters and evaluating various diagnostics are more specific means of control, often restricted to service people. Keyboard and screen are used to set program origin and to hook up to external devices, such as a connection with another computer. There are many other options, particularly for multi axis machines. Every keyboard allows the use of *letters*, *digits* and *symbols* for data entry. Not every keyboard allows the use of *all* alphabet letters or *all* available symbols. Some control panel keys have a description of an *operation*, rather than a letter, digit or symbol, for example, *Read* and *Punch* keys or the *Offset* key.

◆ Handle

For setup purposes, each CNC machine has a rotary handle that can move one selected axis by as little as the least increment of the control system. The official Fanuc name for the handle is *Manual Pulse Generator*. Associated with the handle is *Axis Select* switch (often duplicated on the operation panel as well as on the handle) and the range of increment (that is the least increment X1, X10 and X100). The letter X in this case is the multiplier and stands for 'X times'. One handle division will move the selected axis by X times the minimum increment of active units of measurement. Handles with digital display on a small screen are also available. In Figure 5-3 and the following table are details of a typical traditional handle.

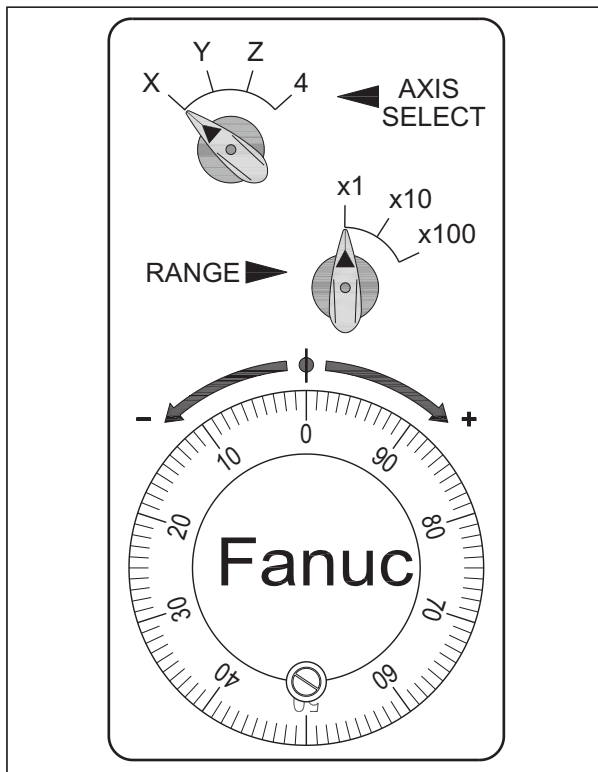


Figure 5-3

An example of a detached handle, called the *Manual Pulse Generator (MPG)*, with a typical layout and features. Layout and features may vary on different machine models.

Handle multiplier	One handle division motion is ...	
	for metric units	for imperial units
X1	0.001 mm	0.0001 inch
X10	0.010 mm	0.0010 inch
X100	0.100 mm	0.0100 inch

SYSTEM FEATURES

CNC unit is nothing more than a sophisticated special purpose computer. The 'special purpose' in this case is a *computer capable of controlling the activities of a machine tool*, such as lathe or machining center. It means the computer has to be designed by a company that has expertise in this type of special purpose computers. Unlike many business types of computers, each CNC unit is made for a particular customer. The customer is typically a machine manufacturer, *not* the end user. The manufacturer specifies certain requirements that the control system has to meet, requirements that reflect uniqueness of the machines they build. Basic control does not change, but some customized features may be added (or taken away) for a specific machine. Once a control system is sold to the machine manufacturer, more features are added to the whole system. They relate to the design and capabilities of the machine.

A good example is a CNC unit for two machines that are the same in all respects except one. One machine has a *manual* tool changer, the other has an *automatic* tool changer. In order to support the automatic tool changer, the CNC unit must have special features built-in, that are not required for a machine without the tool changer. The more complex a CNC system is, the more expensive it is. Users that do not require all sophisticated features, do not pay a premium for features they do not need.

◆ Parameter Settings

Information that establishes the built-in connection between the CNC control and the machine tool is stored as special data in internal registers, called *system parameters*. Some of the information in this handbook is quite specialized and listed for reference only. Programmers with limited experience do not need to know system parameters in a great depth. Original factory settings are sufficient for most machining jobs.

When the parameter screen is displayed, it shows the parameter number with some data in a row. Each row of numbers represents one *byte*, each digit in the byte is called a *bit*. The word *bit* is made from the words *Binary digIT* and is the smallest unit of a parameter input. Numbering of bits starts with 0, read from the right to the left:

Number	#7	#6	#5	#4	#3	#2	#1	#0
xxxx	0	1	1	0	1	0	0	1

Fanuc control system parameters belong to one of three groups, specified within an allowed range:

- Binary codes
- Units inputs
- Setting values

These groups use different input values. The *binary input* can only have an input of a 0 or 1 for the *bit* data format, 0 to +127 for the byte type. *Units input* has a broader scope - units can be in mm, inches, mm/min, in/min, degrees, milliseconds, etc. A *value* can also be specified within a given range, for example, a number within the range of 0-99, or 0-99999, or +127 to -127, etc.

A typical example of a *binary input* is a selection between *two options*. For instance, a feature called *dry run* can be set only as *effective* or *ineffective*. To select a preference, an arbitrary bit number of a parameter has to be set to 0 to make the dry run effective and to 1 to make it ineffective.

Units input, for example, is used to set the increment system - the dimensional units. Computers in general do not distinguish between imperial and metric units, just numbers. It is up to the user *and* the parameter setting, whether the control will recognize 0.001 mm or 0.0001 inches as the least increment. Another example is a parameter setting that stores the maximum feedrate for each axis, the maximum spindle speed, etc. Such values must never be set higher than the machine itself can support. An indexing axis with a minimum increment of 1°, will *not* become a rotary axis with 0.001° increment, just because the parameter is set to a lower value, even if it is possible. ***Such a setting is wrong and can cause serious damage!***

To better understand what CNC system parameters can do, here is an abbreviated listing of parameter classification for a typical Fanuc control system (many of them are meaningful to the service technicians only):

- Parameters related to Setting
- Parameters related to Axis Control Data
- Parameters related to Chopping
- Parameters related to the Coordinate System
- Parameters related to Feedrate
- Parameters related to Acceleration/Deceleration Control
- Parameters related to Servo
- Parameters related to DI/DO
- Parameters related to MDI, EDIT, and CRT
- Parameters related to Programs
- Parameters related to Serial Spindle Output
- Parameters related to Graphic Display
- Parameters related to I/O interface
- Parameters related to Stroke Limit
- Parameters related to Pitch Error Compensation
- Parameters related to Inclination Compensation
- Parameters related to Straightness Compensation
- Parameters related to Spindle Control
- Parameters related to Tool Offset
- Parameters related to Canned Cycle
- Parameters related to Scaling and Coordinate Rotation
- Parameters related to Automatic Corner Override
- Parameters related to Involute Interpolation
- Parameters related to Uni-directional Positioning
- Parameters related to Custom Macro (User Macro)
- Parameters related to Program Restart

Parameters related to High-Speed Skip Signal Input
Parameters related to Automatic Tool Compensation
Parameters related to Tool Life Management
Parameters related to Turret Axis Control
Parameters related to High Precision Contour Control
Parameters related to Service ... *and other parameters*

Quite a few parameters have nothing to do with daily programming and are listed only as an actual example. All system parameters should be set or changed only by a qualified person, such as an experienced service technician. A programmer or operator should not modify any parameter settings. These changes require not only qualifications but authorization *as well*. Keep the list of original parameter settings away from the control, in a safe place, just in case.

Take care when changing control system parameters !

Many parameters are periodically updated during program processing. CNC operator is usually not aware that this activity is going on at all. There is no real need to monitor this activity. The safest rule to observe is that once the parameters have been set by a qualified technician, any *temporary* changes required for a given work should be done through the CNC program. If *permanent* changes are required, an authorized person should be assigned to do them - *nobody else*. Some parameters may be changed - ***very carefully*** - through the program - *see page 405*.

◆ System Defaults

Many parameter settings stored in control at the time of purchase have been entered by the manufacturer as either the *only choices*, the *most suitable choices*, or the *most common selections*. That does not mean they will be the preferred settings - it means they were selected on the basis of their common usage. Many settings are rather conservative in their values, for safety reasons.

The set of parameter values established at the time of installation are called the *default* settings. The English word '*default*' is a derivative of a French word '*defaut*', that can be translated as '*assumed*'. When main power to the control is turned on, there are no set values passed to parameters from a program, since no program has yet been used. However, certain settings become active automatically, *without* an external program. For instance, a cutter radius offset is automatically canceled at the control system startup. Also canceled are the fixed cycle mode and tool length offset. The control '*assumes*' that certain conditions are preferable to others. Many operators will agree with most of these initial settings, although not necessarily with all of them. Some settings are customizable by a change of a parameter settings. Such settings will become permanent and create a new '*default*'.

Always document any changes made to parameters !

A computer is fast and accurate but has no intelligence. People are often slow and make errors, but have one unique ability - *they think*. A computer is just a machine that does not *assume* anything, does not *consider*, does not *feel* - computer *does not think*. A computer does not do anything that a human effort and ingenuity has not done during the design process, in form of hardware and software.

When a CNC machine is powered, its internal software sets *certain* existing parameters to their default condition, designed by engineers. Not *all* system parameters, only *certain* parameters can have an assumed condition - a condition that is known as the *default value* (condition).

For example, a tool motion has three basic modes - a *rapid* motion, a *linear* motion and a *circular* motion. The default motion setting is controlled by a parameter. Only one setting can be active at the startup. Which one? The answer *depends on the parameter setting*. Many parameters can be preset to a desired state. Only the *rapid* or *linear* mode can be set as default in the example. Since rapid motion is the first motion in most programs, it seems to make sense to make it a default - *but wait!*

Most controls are set to *linear* motion as the default (G01 command) to be in effect at the start - strictly *for safety reasons*. When machine axes are moved manually, the parameter setting has no effect. If a manual input of an axis command value takes place, either through the program or from the control panel, a tool motion results. If a motion command is not specified, CNC system will use the command mode that had been preset as the default in the parameters. Since the default mode is linear motion G01, the result is an *error condition*, faulting the system *for the lack of feedrate!* There is no cutting feedrate in effect, which the G01 requires. Had the default setting been rapid motion G00, a rapid motion would be performed, as it does not need programmed feedrate.

It is beneficial to know the default settings of all controls in the shop. Unless there is a good reason to do otherwise, defaults for similar controls should be the same.

◆ Memory Capacity

CNC programs can be stored in the control memory. Program size is only limited by the control internal capacity. This capacity is measured in variety of ways, originally as the equivalent *length of tape* in meters or feet, lately as the *number of bytes* or the *number of screen pages*. A common minimum memory capacity of a small CNC lathe control may be 20m of tape (66 ft). This is an old fashioned method that somehow persisted in staying with us. On CNC milling systems, the memory requirements based on the same criteria are generally larger and the typical minimum memory capacity is 80 m or about 263 ft. Optionally, larger memory capacity can be added to any control system. The minimum memory capacity of a control varies from one machine to another - always check control specifications carefully.

Modern methods of measuring memory capacity prefer to use *bytes* as the unit, rather than a length of an obsolete tape. A byte is the smallest unit of storage capacity and is very roughly equivalent to one character in the program.

Memory capacity of the control system should be large enough to store the longest CNC program expected on a regular basis. That requires some planning before the CNC machine is purchased. For example, in three dimensional mold work or high speed machining, the cost of additional memory capacity may be very high. Although any cost is a relative term, there are reliable and inexpensive alternatives, well worth looking into.

One alternative is running the CNC program from a personal computer. An inexpensive communication software and cabling is required to connect the computer with the CNC system. The simplest version is to *transfer* a CNC program from one computer to the other. More sophisticated possibility includes software and cables that can actually run the machine from a personal computer, without loading it into the CNC memory first. This method is often called '*drip-feeding*' or '*bitwise input*'. When operated from a personal computer, the CNC program can be as long as the storage device capacity, typically the hard drive.

Most CNC programs will fit into the CNC internal memory. Many controls use the number of available characters or the equivalent length of tape. Here are some formulas that can be used to get at least approximate memory capacity calculations:

➡ Formula 1 :

To find the program length in *meters*, when the capacity is known in *characters*, use the following formula:

$$S_m = N_c \times 0.00254$$

where ...

S_m = Storage capacity in meters
 N_c = Memory capacity (number of characters)

➡ Formula 2 :

To find the length of program in *feet*, when the capacity is known in *characters*, use the following formula:

$$S_f = \frac{N_c}{120}$$

where ...

S_f = Storage capacity in feet
 N_c = Memory capacity (number of characters)

➡ Formula 3 :

To find the number of *characters* in a given program, if the system memory capacity is known in *meters*:

$$C = \frac{m}{0.00254}$$

🔍 where ...

C = Number of available characters
m = Memory capacity in meters

Virtually the same results can be achieved by a slightly restructured formula:

$$C = \frac{m \times 1000}{2.54}$$

➡ Formula 4:

To find the number of *characters*, if the system memory capacity is known in *feet*, use the following formula:

$$C = f \times 120$$

🔍 where ...

C = Number of available characters
f = Memory capacity in feet

Latest Fanuc controls show the available memory as the number of *free screen display pages*. This type of data is not as easy to convert as the others.

In cases where the available memory capacity is too small to accept large programs, several techniques are available to minimize the problem, for example, eliminating leading zeros or removing all or some block numbers.

MANUAL PROGRAM INTERRUPTION

If a program needs to be interrupted in the middle of processing, control system offers several ways to do that, using the machine operation panel. The most common features of this type are toggle switches or push buttons for a *single block* operation, *feedhold* and *emergency stop*.

◆ Single Block Operation

The normal purpose of a program is to control the machine tool automatically and sequentially in a continuous mode. Every program is a series of formatted commands - or instructions - written as individual lines of code, called *blocks*. Blocks and their concepts will be described in the following chapters. All program commands in a single

block are processed as a *single instruction*. The blocks are received by the control system in sequential order, from top down and in the order they appear in the program. Normally, a CNC machine is run in a continuous mode, while all blocks are processed automatically, one after another. This continuity is important for production, but not practical when proving a new program, for example.

To disable continuous program execution, a *Single Block* switch is provided on the operation panel. In the single block mode, only one program block will be processed each time the *Cycle Start* key is pressed. On the operation panel, single block mode can be used separately or in combination with other settings that make program proving faster and more accurate.

◆ Feedhold

Feedhold is a special push button located on the operation panel, usually close to the *Cycle Start* button. When this button is pressed during a rapid, linear or circular axes motion, it will immediately stop the motion. Such action applies to all axes active at the time. This feature is convenient for machine setup or first part run. Some types of motion restrict the feedhold function or disable it altogether. For example, threading or tapping modes make the switch inoperative.

Activating feedhold at the machine will not change any other program values - it will only affect axis motion. Feedhold switch will be illuminated (in red light), as long as it is effective. CNC programmer can override feedhold from *within* the program, for special purposes.

◆ Emergency Stop

Every CNC machine has at least one special mushroom shaped push button, red in color, that is located in an accessible place on the machine. It is marked as the *Emergency Stop* or *E-stop*. When this button is pressed, *all machine activities will cease immediately*. The main power supply will be interrupted and the machine will have to be restarted. Emergency stop switch is a mandatory safety feature on all CNC machines.

Pressing the emergency stop button is not always the *best* or even the *only* way to stop a machine operation. In fact, the latest controls offer other features, far less severe, designed to prevent a collision between cutting tool and part or fixture. Previously discussed feedhold button is only one option, along with several other features. If the emergency stop must be used at all, it should be used only as the *last resort*, when any other action would require unacceptably longer time. There is no need for panic, if something does go wrong.

For some machine actions, the effect of *Emergency Stop* is not always apparent. For example, the spindle requires a certain time for deceleration to stop.

MANUAL DATA INPUT - MDI

A CNC machine is not always operated by the means of a program. During part setup, the CNC operator has to do a number of operations that require physical movements of machine slides, rotation of spindle, tool change, etc. There are no mechanical devices on a CNC machine. The handle (*Manual Pulse Generator*) is an electronic, not a mechanical unit. In order to operate a CNC machine without conventional mechanical devices the control system offers a feature called *Manual Data Input* - or MDI.

Manual Data Input enables the input of program data into the system *one program instruction* at a time. If too many instructions were to be input repeatedly, such as a long program, the procedure would be very inefficient. During setup and for similar purposes, one or a few instructions at a time will benefit from MDI.

To access MDI mode, the MDI key on the operation panel must be selected. That opens the screen display with the current system status. Not all, but the majority of programming codes are allowed in the MDI mode. Their format is identical to the format of a CNC program in written form. This is one area where CNC operator acts as a CNC programmer. It is very important that the operator is trained at least in the basics of CNC programming, certainly to the point of being able to handle the setup instructions for *Manual Data Input*.

PROGRAM DATA OVERRIDE

All CNC units are designed with a number of special rotary switches that share one common feature - they allow the CNC operator to *override* programmed spindle speed or programmed speed of an axis motion. For example, a 15 in/min feedrate in the program produces a slight chatter. A knowledgeable operator will know that by *increasing* the feedrate or *decreasing* the spindle speed, the chatter may be eliminated. It is possible to change the feedrate or the spindle speed by editing the program, but this method is not very efficient. A certain 'experimentation' may be necessary during actual cut to find the optimum setting value. Manual override switches come to the rescue, because they can be used by trial during operation. There are four override switches found on most control panels:

- **Rapid feedrate override** (rapid traverse)
(modifies the rapid motion of a machine tool)
- **Spindle speed override**
(modifies the programmed spindle r/min)
- **Feedrate override** (cutting feedrate)
(modifies the programmed feedrate)
- **Dry run mode**
(changes cutting motions to a variable speed)

Override switches can be used individually or together. They are available on the control to make work easier for both the operator and the programmer. Operator does not need to 'experiment' with speeds and feeds by constantly editing the program and the programmer has a certain latitude in setting reasonable values for cutting feedrates and spindle speed. The presence of override switches is not a licence to program unreasonable cutting values. Overrides are fine tuning tools only - part program must always reflect the machining conditions of the work. Usage of override switches does not make any program changes, but gives the CNC operator an opportunity to edit the program later to reflect all optimum cutting conditions. Used properly, override switches can save a great amount of valuable programming time as well as setup time at CNC machine.

◆ Rapid Motion Override

Rapid motions are selected in the CNC program by a preparatory command without a specified feedrate. If a machine is designed to move at 985 in/min (25000 mm/min) in the rapid mode, this rate will never appear in the program. Instead, the rapid motion mode is called by programming a special preparatory command G00. During program processing, all motions in G00 mode will be at the manufacturer's fixed rate. The same program will run faster on a machine with a high rapid motion rating than on a machine with a low rapid motion rating.

During setup, the rapid motion rate may require some control for program proving, when very high rapid rates are uncomfortable to work with. After a program had been proven, rapid rate can be applied at its maximum. CNC machines are equipped with a *rapid override switch* to allow temporary rapid motion settings. Located on the machine control panel, this switch can be set to one of four settings. Three of them are usually marked as the *percentage* of the maximum rate, typically as 100%, 50% and 25%. By switching to one of them, the rapid motion rate changes. For example, if the maximum rapid rate is 985 in/min or 25000 mm/min, the actual reduced rates are 493 in/min or 12500 mm/min at the 50% setting and 246 in/min or 6250 mm/min at the 25% setting. Each of the reduced rates is more comfortable to work with during setup.

The fourth position of the switch often has no percentage assigned and is identified as an F1 or by a small symbol. In this setting, the rapid motion rate is even slower than that of 25% setting. Why is it not identified as 10% or 15%, for example? The reason is simple - control system allows a *customized* selection as to what the setting will be. It may be a setting of between 0 and 100%. The default setting is also the most logical - usually 10% of the maximum rapid traverse rate. This setting should never be higher than 25% and can be done only through setting of a system parameter. Make sure that all persons who work on such a machine are aware of the changes.

◆ Spindle Speed Override

The same logic used for the application of rapid rate override can be used for the spindle speed override. The required change can be established during actual cutting by using the *spindle speed override switch*, located on the machine control panel. For example, if the programmed spindle speed of 1000 r/min is too high or too low, it may be changed temporarily by the switch. During actual cutting, the CNC operator may experiment with the spindle speed override switch to find the optimum speed for given cutting conditions. This method is a much faster than 'experimenting' with the program values.

Spindle speed override switch can be continuous on some controls or selectable in increments of 10%, typically within the range of 50-120% of the programmed spindle speed. A spindle programmed at 1000 r/min can be overridden during machining to 500, 600, 700, 800, 900, 1000, 1100 and 1200 r/min. This large range allows the CNC operator certain flexibility of optimizing spindle rotation to suit given cutting conditions. There is a catch, however. The optimized spindle speed change may apply to only one tool of the many often used in the program. No CNC operator can be expected to watch for that particular tool and switch the speed up or down when needed. A simple human oversight may ruin the part, the cutting tool or both. The recommended method is to find out the optimum speed for each tool, write it down, then change the program accordingly, so *all tools* can be used at the 100% spindle override setting for production.

Comparison of the increments on the spindle override switch with the increments on switches for rapid traverse override (described earlier) and the feedrate override (described next), offers much more limited range. The reason for the spindle speed range of 50% to 120% is safety. To illustrate with a rather exaggerated example, no operator would want to mill, drill or cut any material at 0 r/min (no spindle rotation), possibly combined with a heavy feedrate.

In order to change the selected override setting into 100% speed in the program, a new spindle speed has to be calculated. If a programmed spindle speed of 1200 r/min for a tool is *always* set to 80%, it should be edited in the program to 960 r/min, then used at 100%. The formula is quite simple:

$$S_n = S_p \times p \times 0.01$$

where ...

- S_n = Optimized - or NEW - r/min
- S_p = Originally programmed r/min
- p = Percentage of spindle override

Overriding the programmed spindle speed on CNC machines should have only one purpose - to establish spindle speed rotation for the best cutting conditions.

◆ Feedrate Override

Probably the most commonly used override switch is one that changes programmed feedrates. For milling controls, the feedrate is programmed in *in/min* or *m/min*. For lathe controls, the feedrate is programmed in *in/rev* or *mm/rev*. Feedrate per minute on lathes is used only in cases when the spindle is not rotating and the feedrate needs to be controlled.

The new feedrate calculation, based on the overridden feedrate setting, is similar to that for spindle speed:

$$F_n = F_p \times p \times 0.01$$

where ...

- F_n = Optimized - or NEW - feedrate
- F_p = Originally programmed feedrate
- p = Percentage of feedrate override

Feedrate can be overridden within a large range, typically from 0% to 200% or at least 0% to 150%. When the feedrate override switch is set to 0%, the CNC machine will *stop* the cutting motion. Some CNC machines do not have the 0% percent setting and start at 10%. This can be change by a system parameter. The maximum of 150% or 200% cutting feedrate will cut 1.5× or 2× faster than the programmed feedrate amount.

There are situations, where the use of a feedrate override would damage the part or the cutting tool - or both. Typical examples are various tapping cycles and single point threading. These operations require spindle rotation synchronized with the feedrate. In such cases, the feedrate override will become *ineffective*. Feedrate override *will* be effective, if standard motion commands G00 and G01 are used to program any tapping or thread cutting motions. Single point threading command G32, tapping fixed cycles G74 and G84, as well as lathe threading cycles G92 and G76 have feedrate override cancellation built into the software. All these and other related commands are described later in the handbook, in more detail.

◆ Dry Run Operation

Dry run is a special kind of override. It is activated from the control panel by the *Dry Run* switch. It only has a direct effect on the feedrate and allows much higher feedrate than that used for actual machining. In practice, it means the program can be executed much faster than using a feedrate override at the maximum setting. No actual machining takes place when the dry run switch is in effect.

What is the purpose of the dry run and what are its benefits? Its purpose is to test the program integrity *before* the CNC operator cuts the first part. Benefits are mainly in the time saved during program proving when no machining takes place. During dry run, part is normally *not* mounted

in the machine. If the part is mounted in a holding device and the dry run is used as well, it is very important to provide sufficient clearances. Usually, it means *moving the tool away from the part*. Program is then processed 'dry', without actual cutting, without a coolant, just in the air. Because of the heavy feedrates in dry run, no part can be machined safely. During dry run, the program can be checked for all possible errors except those that relate to the actual contact of cutting tool with the material.

Dry run is a very efficient setup aid to prove overall integrity of a CNC program. Once a program is proven during dry run, CNC operator can concentrate on those sections of the program that contain actual machining. Dry run can be used in combination with several other features of the operation panel.

Make sure to disable dry run before machining !

◆ Z-Axis Neglect

Another very useful tool for testing unproven programs on CNC machining centers (*not* lathes) is a toggle switch located on the operation panel called *Z-axis Neglect* or *Z-axis Ignore*. As either name suggests, when this switch is activated, any motion programmed for the Z-axis will *not* be performed. Why the Z-axis? Since the X and Y axes are used to profile a part shape (the most common contouring operations), it would make no sense to temporarily cancel either one of these axes. By temporarily neglecting, that is *disabling*, the Z-axis temporarily, CNC operator can concentrate on proving the accuracy of the part contour, without worrying about depth motions. Needless to say, this method of program testing must take place without a mounted part, and normally without a coolant as well. *Be careful here!* It is important to *enable* or *disable* the switch *at the right time*. If the Z-axis motion is disabled before the *Cycle Start* key is pressed, all following Z-axis commands will be ignored. If the motion is enabled or disabled during program processing, the position of Z-axis may be inaccurate.

Z-axis neglect switch may be used in both manual and automatic modes of operation. Just make sure that the motion along the Z-axis is changed back to the *enabled* mode, once the program proving is completed. Some CNC machines require resetting of the Z-axis position settings.

◆ Manual Absolute Setting

Some older CNC machines had a toggle switch identified as *Manual Absolute*, that could be set to ON or OFF position. If installed, its purpose is simple - if a manual motion is made during program processing, for example to move a drill to inspect a hole, work coordinates are updated if the switch is ON, but they are *not* updated if the switch is OFF. In practice, this switch should always be ON - and for that reason, most controls do not have this switch anymore.

◆ Sequence Return

Sequence Return is a special function controlled by a switch or a key on the control panel. Its purpose is to enable the CNC operator to start a program from the middle of an interrupted program. Certain programmed functions are memorized (usually the last speed and feed), others have to be input by the *Manual Data Input* key. Operation of this function is closely tied to actual machine tool design. More information on the usage can be found in the machine tool manual. This function is very handy when a tool breaks during processing of very long programs. It can save valuable production time, if used properly.

◆ Auxiliary Functions Lock

There are three functions available to the operation of a CNC machine that are part of '*auxiliary functions*' group. These functions are:

Miscellaneous functions lock	Locks 'M' functions
Spindle functions lock	Locks 'S' functions
Tool functions lock	Locks 'T' functions

As described later in this chapter, auxiliary functions generally relate to *technological* aspects of CNC programming. They control such machine functions as spindle rotation, spindle orientation, coolant selection, tool changing, indexing table, pallets and many others. To a lesser degree, they also control some program functions, such as compulsory or optional program stop, subprogram flow, program closing and others.

When auxiliary functions are locked, all machine related miscellaneous functions M, all spindle functions S and all tool functions T will be temporarily suspended. Some machine manufacturers prefer the name *MST Lock* rather than *Auxiliary Functions Lock*. MST is an acronym of the first letters from the words *Miscellaneous*, *Spindle* and *Tool*, referring to the program functions that will be locked.

Applications of these locking functions are limited to job setup and program proving only and are not used for production machining.

◆ Machine Lock

Machine Lock function is yet another control feature for program proving. So far, we have looked at the *Z-axis Neglect* function and locking of the auxiliary functions. Remember that the *Z-axis Neglect* function will disable the motion of Z-axis only and the *Auxiliary Functions Lock* (also known as *MST lock*) locks miscellaneous functions, spindle functions and tool functions. Another function, also available through the control panel, is called *Machine Lock*. When this function is enabled, motion of *all* axes is locked.

It may seem strange to test a program by locking all tool motions, but there is a good reason to use this feature. It gives the CNC operator a chance to test the program with virtually no chance of a collision.

When machine lock is enabled, *only the axis motion is locked*. All other program functions are available, including tool change and spindle functions. This function can be used alone or in combination with other functions in order to discover possible program errors. The most typical errors are syntax errors and the various tool offset functions.

◆ Practical Applications

Many of the control features described in this chapter, are used in *conjunction* with each other. A good example is *Dry Run* used in conjunction with the *Z-axis Neglect* or the *Auxiliary Functions Lock*. By knowing what function are available, CNC operator makes a choice to suit the needs of the moment. There are many areas of equal importance on which the CNC operator has to concentrate when setting up a new job or running a new program. Many features of the control unit are designed to make operator's job easier. They allow a focus on one or two items at a time rather than the complexity of the whole program. These features have been covered in a reasonable detail, now is the time to look at some practical applications.

During initialization of a new program run, a good CNC operator will take certain precautions as a matter of fact. For example, the first part of the job will most likely be tested with a rapid motion set to 25% or 50% of the available rapid rate. This reduced setting allows the operator to monitor the program integrity, as well as specific details. These details may include items such as a possibility of insufficient clearance between tool and stock, checking if the toolpath looks reasonable, and so on.

CNC operator will have a number of tasks to perform simultaneously. Some of these tasks include monitoring spindle speed, feedrate, tool motions, tool changes, coolant, etc. A careful and conscious approach results in building the confidence in the integrity of a CNC program. It may be the second or even the third part of the job when CNC operator starts thinking of the optimization of cutting values, such as spindle speed and cutting feedrates. This optimization will truly reflect the ideal speeds and feeds for a particular part under given setup.

Production supervisor should not arbitrarily criticize an override setting less than 100%. Many managers consider the CNC program as an unchangeable and perfect document. They take the attitude that what is written is infallible - which is not always true. Often, the CNC operator may have no other choice *but to override* programmed values. What is most important, is modification of the program that reflects the optimized cutting conditions.

Once the machine operator finds what values must be changed in the program itself, this program must be edited to reflect these changes. Not only for the job currently worked on, but also for any repetition of the same job in the future. After all, it should be the goal of every programmer and CNC operator, to run any job at one hundred percent efficiency. This efficiency is most likely reached as a combined effort of the operator and the programmer. Good CNC programmer will always make the *effort* to reach 100% efficiency at the desk and then improve the program even further.

SYSTEM OPTIONS

Optional features on a CNC system are like options on a car. What is an option at one dealership, maybe a standard feature at another. Marketing strategies and corporate philosophies have a lot to do with this approach.

Here is a look at some control features that may or may not be classified as optional on a particular system. But some important disclaimer first:

This handbook covers subject matter relating to the majority of control features, regardless of whether they are sold as a standard or an optional feature of the system. It is up to the user to find out what exact options are installed on a particular control system

◆ Graphic Display

Graphic representation of toolpath on the display screen is one of the most important, as well as sought after, control options. Do not confuse this option with any type of conversational programming, which also uses a graphic toolpath interface. In the absence of a computer assisted programming (CAM), graphic display on the control panel is a major benefit. Whether in monochrome or color, the convenience of seeing the tool motions *before* actual machining is much appreciated by CNC operators and programmers alike.

A typical graphics option shows machine axes and two cursors for zooming. When the toolpath is tested, individual tools are distinguished by different colors, if available, or different intensity. Rapid motions are represented by a dashed line type, cutting motions by a continuous line type. If the graphics function is applied during machining, tool motions can be watched on the display screen - very helpful for those CNC machines that have dirty, oily and often scratched safety shields.

Upwards or downwards scaling of the display allows for evaluation of a tool motion overall or for detail areas. Many controls also include actual toolpath simulation, where the part shape and the cutting tool can be set first, then seen on the screen.

◆ In-Process Gauging

During many unattended machining operations, such as in manufacturing cells or Agile manufacturing, a periodic checking and adjusting dimensional tolerances of the part is imperative. As the cutting tool wears out, or perhaps because of other causes, the dimensions may fall into the 'out-of-tolerance' zone. Using a probe device and a suitable program, *In-Process Gauging* option offers quite a satisfactory solution. CNC part program for the *In-Process Gauging* option will contain some quite unique format features - it will be written parametrically, and will be using another option of the control system - the *Custom Macros* (sometimes called the *User Macros*), which offer variable and parametric type programming.

If a company or machine shop is a user of *In-Process Gauging* option, there are good chances that other control options are also installed and available to the CNC programmer. Some of the most typical options are probing software, tool life management, macros, etc. This technology goes a little too far beyond standard CNC programming, although it is closely related and frequently used. Companies that already use the numerical control technology, will be well advised to look into these options to remain competitive in their field.

◆ Stored Stroke Limits

Definition of an area on CNC lathes or a cube on CNC machining centers that is safe to work within, can be stored as a control system parameter called *stored stroke limit*. These stored stroke limits are designed to prevent a collision between the cutting tool and a fixture, machine tool or part. The area (2D) or the cube (3D) can be defined as either *enabled* for the cutter entry or *disabled* for the cutter entry. It can be set manually on the machine or, if available, by a program input. Some controls allow only one area or cube to be defined, others allow more.

When this option is in effect and the CNC unit detects a motion in the program that takes place within the forbidden zone, an error condition results and the machining is interrupted. A typical applications may include zones occupied by a tailstock, a fixture, a chuck, a rotary table, and even an unusually shaped part.

◆ Drawing Dimensions Input

An option that seems somewhat neglected, is the programming method by using input of dimensions from an engineering drawing. The ability to input known coordinates, radiuses, chamfers and given angles directly from the drawing makes it an attractive option. This ability is somewhat overshadowed by poor program portability. Such an option must be installed on all machines in the shop, in order to use the programmed features efficiently. This is not a common feature and is not covered in this handbook.

◆ Machining Cycles

Both milling and turning controls offer a variety of machining cycles. Typical machining cycles for milling operations are called *fixed cycles*, also known as *canned cycles*. They simplify simple point-to-point machining operations such as drilling, reaming, boring, back boring and tapping. Some CNC systems also offer cycles for face milling, pocket milling, various hole patterns, etc.

CNC lathes also have many machining cycles available to remove material by automatic roughing, profile finishing, facing, taper cutting, grooving and threading. Fanuc controls call these cycles *Multiple Repetitive Cycles*.

All these cycles are designed for easier programming and faster changes at the machine. They are *built in* the control and *cannot be changed*. Programmer supplies the cutting values during program preparation by using an appropriate cycle call command. All processing is done automatically, by the CNC system. Of course, there will always be special programming projects that cannot use any cycles, at least not effectively, and have to be programmed manually or with the use of an external computer and CAM software.

◆ Cutting Tool Animation

Many of the graphic toolpath displays defined earlier, are represented by simple lines and arcs. Current tool position is usually the location of a line or arc endpoint on the screen. Although this method of displaying cutting tool motion graphically is certainly useful, there are two disadvantages to it. The cutting tool shape and the material being removed cannot be seen on the screen, although toolpath simulation may help a bit. Many modern controls incorporate graphic feature called *Cutting Tool Animation*. If available on the control, it shows the part blank, the mounting device and the tool shape. As the program is processed, CNC operator has a reasonably accurate visual aid in program proving. Each graphic element is identified by a different color, for even a better appearance. The blank size, mounting device and tool shape can be preset for exact proportions and a variety of tool shapes can be stored for repetitive use. This option is a good example of CAD/CAM-like features built into a stand-alone control system.

◆ Connection to External Devices

CNC computer (control) can be connected to an external device, usually another computer. Every CNC unit has one or more connectors, specifically designed for interfacing to peripheral devices. The most common device is called RS-232 (EIA standard), designed for communications between two computers. Setting up the connection with external devices is a specialized application. CNC operator uses such a connection to transfer programs and other settings between two computers, usually for storage and backup purposes. Devices other than RS-232 are also available - check with the machine vendor.

6

PROGRAM PLANNING

Development of any CNC program should always begin with a very carefully planned process. Such process typically starts with an engineering *drawing* (also called a *blueprint* or a *technical print*) of the required part, released for production. Before any machining process can be completed, several steps have to be considered and carefully evaluated. Placing a greater effort into program planning will yield better program and better machined part.

STEPS IN PROGRAM PLANNING

Individual steps required in program planning are generally determined by the nature of part to be machined. There is no magic overall formula available for all jobs, but some basic steps are quite common and should always be considered carefully:

- Initial information provided / Machine tools features
- Part complexity / Evaluation of machining features
- Manual programming / Computerized programming
- Typical programming procedure / Program structure
- Part drawing / Engineering data
- Methods sheet / Material specifications
- Machining sequence - Operations / Tool order
- Tooling selection / Cutting Holders / Inserts / HSS Tools
- Part setup / Part holding / Fixtures
- Technological decisions / Cutting conditions
- Work sketch and individual calculations
- Quality considerations in CNC programming

All steps in the list are suggestions only - they are *guidelines*. Individual steps should always be flexible, so they can be adapted for any job and its unique requirements.

INITIAL INFORMATION

The main purpose of most engineering drawings is to define the part shape, individual dimensions, and relationships between part features. Some drawings may also include data about the initial blank material (stock), such as type, size, and shape. In CNC programming, good familiarity with various materials is important. For programming purposes, materials used to machine a part are evaluated by their size, type, shape, condition, hardness, etc.

Part drawing and material data are the primary sources of information about a specific part to be machined. They define the starting point of program planning. The objective of such a plan is to collect all available data and use all initial information for one purpose - to establish grounds for the most efficient method of machining, along with all related considerations - mainly part accuracy, productivity, safety and convenience.

Drawing and material data provide much of initial information, but they are not the only source. A great part of what is needed to develop a part program is not found in the drawing directly, but in other documentation. For example, a process sheet (routing sheet) provides many engineering requirements *not covered* in the drawing, such as *pre-* and *post-* machining operations, grinding allowances, assembly features, requirements for hardening, next machine setup, and many others. Collecting relevant information from all available sources provides enough groundwork to start planning a CNC program development.

CNC MACHINE FEATURES

No amount of initial information is much useful if the selected CNC machine is not *suitable* for a particular job. During program planning, CNC programmer concentrates on a *particular machine tool with a particular CNC system*. These two major parts of a CNC machine are always connected and they must always be considered in any single CNC machine definition. It is just not enough to select a special fixture or a special setup - the CNC machine itself has to be suitable to handle any required setup.

Modern technology offers a large number of special features that can be purchased as options for the selected CNC machine. These options are too numerous to list, but any manufacturer's or dealer's web sites specify all details. When a CNC machine is purchased and delivered, the machine shop needs and requirements should be satisfied, at least for a few years. Very few companies go and buy a new CNC machine just to suit a particular job, although that is not an uncommon situation. Yet, such cases are rather rare and happen only if they make strong economic sense.

◆ Machine Type and Size

Possibly the two most important steps in program planning relate to the *type* and *size* of CNC machine, particularly its *work space* or *work area*. Other features, equally important, are machine power rating, spindle speed and

feedrate range, number of tool stations, tool changing system, available accessories, etc. Typically, small CNC machines have higher spindle speeds and lower power rating, large machines have lower spindle speeds available, but their power rating is higher.

◆ Control System

Control system is the heart of a CNC machine. Being familiar with all standard and optional control features available is a must. This knowledge enables using a variety of advanced programming methods, such as different machining cycles, subprograms, macros and other time saving features of a modern CNC system.

A programmer does not have to physically run a CNC machine. Yet, part programs will become better and more creative with good understanding of the machine and its control system. Program development reflects programmer's knowledge of various CNC machine operations.

One of the main concerns in program planning should be how the CNC operator *perceives* the part program. To a large degree, such perception is quite subjective, at least in the sense that different operators will express their personal preferences, often in opposite terms. On the other hand, every operator appreciates an error-free, concise, well documented and professionally prepared part program, written in a consistent way, one after another. A poorly designed program is disliked by any operator, regardless of personal preferences.

PART COMPLEXITY

When all initial information, such engineering drawing, material stock, available tools and CNC equipment are evaluated, the complexity of programming task becomes much clearer. How difficult is to program a given part manually? What are the machine capabilities? What are the costs? Many questions have to be answered before starting actual program development.

Many simple programming jobs may be assigned to a less experienced programmer or even to a CNC operator. It makes sense from the management perspective and it is also a good way to gain valuable experience.

Difficult or complex jobs will benefit from a computerized programming system. Technologies such as *Computer Aided Design* (CAD) and *Computer Aided Manufacturing* (CAM) have been a strong part of the manufacturing process for many years. The cost of a CAD/CAM system is only a fraction of what it used to be only a few years ago. Even small shops now find that benefits offered by modern technology are too significant to be ignored. Several programming systems are available and can handle virtually any job. For a typical machine shop, a Windows based programming software can be very beneficial. A typical exam-

ple of this kind of application is the very popular and powerful *Mastercam*™ or *EdgeCam*™. There are several others. All major software vendors offer three-axis milling, standard lathe, multi-axis milling, turning with milling axes (live tooling), five axis machining, wire EDM, etc.

MANUAL PROGRAMMING

Manual programming (without a computer) has been the most common method of preparing a part program for many years. The latest CNC controls make manual programming much easier than ever before by using fixed or repetitive machining cycles, variable type programming, graphic tool motion simulation, standard mathematical input and other time saving features. In manual programming, all calculations are done by hand, with the aid of a pocket calculator - no computer programming is used. Programmed data can be transferred to CNC machines using a *memory card* or via a *cable*, using an inexpensive desktop or a laptop computer. Either process is faster and more reliable than other methods. Short programs can also be entered manually, by keyboard entry, directly at the machine. A punched tape used to be common media of the past but has virtually disappeared from machine shops.

◆ Disadvantages

There are some disadvantages associated with manual programming. Perhaps the most common is the length of time required to actually develop a fully functioning CNC program. Manual calculations, verifications and other related activities in manual programming are very time consuming. Other disadvantages, also very high on the list, are a large percentage of errors, the difficulty in making changes to a program, a lack of tool path verification (simulators are available, such as *NCPlot*®), and many others.

◆ Advantages

On the positive side, manual part programming does have quite a few unmatched qualities. Manual programming is so intense that it requires total involvement of CNC programmer and yet offers virtually unlimited freedom in actual development. Programming manually does have its disadvantages, but it teaches a tight discipline and organization in program preparation. It forces the programmer to understand programming techniques to very last detail. In fact, many useful skills learned in manual programming are directly applied to CAD/CAM programming. Programmer has to know *what* is happening at all times and *why* it is happening. Very important is the *in-depth understanding* of every detail during the program development.

Contrary to many beliefs, a thorough knowledge of manual programming methods is absolutely essential for efficient management of CAD/CAM programming.

CAD/CAM AND CNC

The need for improved efficiency and accuracy in CNC programming has been the major reason for development of a variety of methods that use a computer to prepare part programs. Computer assisted CNC programming has been around for many years. First, in the form of language based programming, such as *APT™* or *Compact II™*. Since the late 1970's, CAD/CAM has played a significant role by adding visual aspect to the programming process. The acronym *CAD/CAM* means *Computer Aided Design* and *Computer Aided Manufacturing*. The first three letters - *CAD* - cover the area of engineering design and drafting. The second three letters - *CAM* - cover the area of computerized manufacturing, where CNC programming is only a small part. The whole subject of CAD/CAM covers much more than just design, drafting and programming. It is a part of modern technology also known under yet another acronym - *CIM* - *Computer Integrated Manufacturing*.

In the area of numerical control, computers have played a major role for a long time. Machine controls have become more sophisticated, incorporating the latest techniques of data processing, storage, tool path graphics, machining cycles, etc. Programs can now be prepared with the use of inexpensive computers, including sophisticated graphical interface. Initial cost should no longer be an issue, even small machine shops are able to afford a programming system in house. These systems are also popular because of their flexibility. Typical computer based CNC programming system does not have to be dedicated only to programming - all related tasks, often done by the programmer, can also be implemented on the same computer. For example, cutting tool inventory management, database of part programs, material information sheets, setup sheets and tooling sheets, etc. The same computer could also be used for uploading and downloading CNC programs.

◆ Integration

The key letter in acronym CIM is - *integration*. It means putting all elements of manufacturing together and work with them as a single unit and more efficiently. The main idea behind successful integration is to avoid duplication. One of the most important rules of using a CAD/CAM computer software (and CNC) is:

NEVER DO ANYTHING TWICE !

When a drawing is developed in CAD software (such as *AutoCAD*), then done again in CAM software (such as *Mastercam*), there is duplication. Duplication breeds errors. In order to avoid duplication, most of CAD systems incorporate a transfer method of the design to a selected CAM system to be used for CNC programming. Typical transfers are achieved through special DXF or IGES files. The *DXF* stands for *Data Exchange Files* or *Drawing Ex-*

change Files, and the *IGES* abbreviation is a short form of *Initial Graphics Exchange Specification* files. Once the geometry is transferred from CAD system to CAM system, only the tool path related process is added. Using a *post processor* (special kind of formatter), the computer software will prepare part program, ready to be loaded directly to the CNC machine.

In most cases, both DXF and IGES methods have been replaced by direct translators, when CAD/CAM vendors provide a facility in their software to import a native file, without converting it first.

◆ Future of Manual Programming

It may seem that manual programming is on the decline. In terms of actual use, this is quite likely true. However, it is necessary to keep in perspective that any computerized technology is based on the already well established methods of manual programming. Manual programming for CNC machines serves as the *source* of new technology - it is the very elementary concept on which computerized programming is based. This knowledge base opens doors for development of more powerful hardware and software applications.

Manual programming may be used somewhat less frequently today and eventually will be used even less - but knowing it well - *really understanding it* - is and always will be the key to harness the power of CAM software. Even computers cannot do everything. There will always be some special programming project that no CAM software, regardless of price, may handle to absolute satisfaction. If machine control system can handle such a project, manual programming is the way to have ultimate control, when any other methods may not be suitable or practical.

Even with a well customized and organized computerized programming system, how can the generated program output be exactly as intended? How can a CNC operator change any program entry at the machine, without knowing its rules and structure?

Successful use of computerized programming requires understanding of manual programming methods.

TYPICAL PROGRAMMING PROCEDURE

Planning a CNC program is not much different than any other planning - at home, at work, or elsewhere - it must be approached in a *logical and methodical* way. The first decisions relate to *what tasks* have to be done and *what goals* have to be achieved. All other decisions relate to *how* to accomplish these goals in an efficient and safe manner. Such a progressive method of approach not only isolates individual problems as they develop, it also forces their solution *before* any next step can be taken.

The list of following items forms a fairly common and logical sequence of tasks to be done in CNC programming. All items are listed in a *suggested order* only, offered for further and deeper evaluation. This - or any other - order should be changed to reflect special conditions or working habits. Some items may be missing from the list altogether, others may be somewhat redundant:

- Study of initial information (drawing and methods)
- Material stock (blank) evaluation
- Machine tool specifications
- Control system features
- Sequence of machining operations
- Tooling selection and arrangement of cutting tools
- Part setup on the machine
- Technological data (speeds, feedrates, etc.)
- Tool path determination
- Working sketches and mathematical calculations
- Program writing and preparation for transfer to CNC
- Program testing and debugging
- Program documentation

There is only one goal in CNC program planning - to provide all instructions in the form of a program that will result in an error-free, safe and efficient CNC machining. Individual steps in the suggested procedures may require some changes - for example, should cutting tools be selected *before* or *after* the final part setup is determined? Can manual part programming method be used efficiently? Are any working sketches necessary? Do not be afraid to modify any so called *ideal* procedure - either temporarily, for a given job, or even permanently, in order to reflect a particular CNC programming style.

Remember, there are no ideal procedures

PART DRAWING

Part drawing is the single most important document used in CNC programming. It visually identifies the shape, important dimensions, tolerances, surface finish and many other requirements for the completed item. Drawings of complex parts often require many sheets, showing different views, details and sections. CNC programmer first evaluates *all* drawing data first, then isolates those that are relevant for a particular program development. Unfortunately, many drafting methods do not reflect the actual CNC manufacturing process. They reflect the *designer's thinking*, rather than the method of manufacturing. Such drawings are generally correct in technical sense, but they are harder to study by the programmer and may need to be '*interpreted*' to be of any value in CNC programming. Typical

examples are methods of applying dimensions, absence of a datum point that can be used as a program reference point and the view orientation in which the part is drawn. In the CAD/CAM environment, the traditional gap between the design, drafting and CNC programming must be eliminated. Just as it helps the programmer to understand designer's intentions, it helps the designer to understand the basics of CNC programming. Both, the designer and the programmer have to understand each other's methods and find common ground that makes the whole process of design and manufacturing coherent and efficient.

◆ Title Block

Drawing title block - *Figure 6-1* - is typical to all professional drawings. Its purpose is to collect all descriptive information related to a particular drawing.

No.	Date	Revision	By
Part Name:			
Scale:		Material:	
Dr.:		Date:	
Chk.:		Drawing number:	
App.:			

Figure 6-1

A title block example of an engineering drawing

The size and contents of a title block vary between companies, depending on the type of manufacturing and internal standards. It is usually a rectangular box, positioned in the corner of a drawing, divided into several small boxes. Contents of the title block include such items as the part name and part number, drawing number, material data, revisions, special instructions, etc. Data in the title block supply crucial information for CNC programming and can be used for program documentation to make easier cross referencing. Not all title block information is needed in programming, but may be used for program documentation.

Revision dates in a drawing are associated with the title block. They are important to the programmer, as they indicate *how current* is the drawing version. Only the latest version of a part design is important to manufacturing.

◆ Dimensioning

Part drawing dimensions are either in *metric* or *imperial* units. Individual dimensions can be referenced from a certain datum point or they can be consecutive, measured from the previous dimension. Often, both types of dimensions are mixed in the same drawing. When writing a program, it may

be more convenient to translate all consecutive - or *incremental* - dimensions into datum - or *absolute* - dimensions. Most CNC programs benefit from drawings using *datum*, or *absolute dimensioning*. Similarly, when developing a sub-program for tool path translation, an *incremental* method of programming may be the right choice - and any choice depends on the application. The most common programming method for all CNC machines uses *absolute* dimensioning method - Figure 6-2, mainly because of editing ease within the CNC system (at the machine).

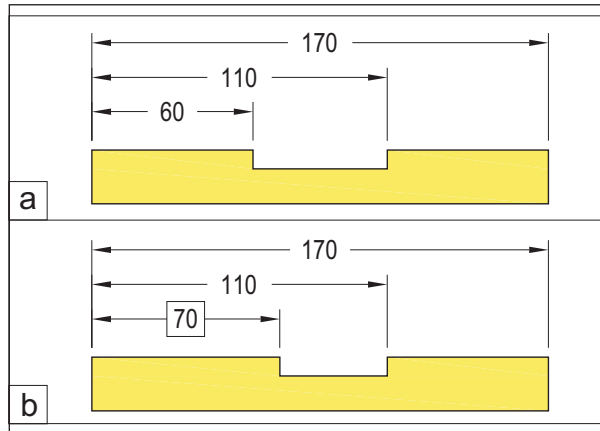


Figure 6-2

Program using **ABSOLUTE** dimensions
Only a single change in the program is necessary

With the absolute system of dimensioning, many program changes can be done by a single modification. Incremental method requires at least two modifications. The differences between the two dimensioning systems can be compared in Figure 6-2, using the *absolute* dimensioning method, and in Figure 6-3, using the *incremental* dimensioning method. The word *incremental* is more common in CNC, in drafting the equivalent word would be *relative*. Both illustrations show the a) figure before revision, and the b) figure after revision.

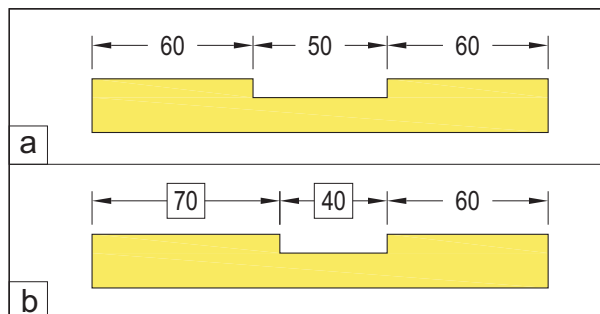


Figure 6-3

Program using **INCREMENTAL** dimensions
Two or more changes in the program are necessary

Fractions

Drawings (particularly older drawings) dimensioned in imperial units often contain fractions. A fractional dimension was sometimes used to identify a *less important* dimensional tolerances (such as ± 0.03 inches from the nominal size). The number of digits following decimal point often indicated a tolerance (the more digits specified, the smaller the tolerance range). These methods of using *implied tolerances* are not an ISO standard and are of no use in CNC programming. Fractional dimensions have to be changed into their decimal equivalents. The number of decimal places in a program is determined by the minimum increment of the control system. A dimension of $3\text{-}3/4$ is programmed as 3.75 without rounding, and a dimension of $5\text{-}11/64$ inches is programmed as 5.1719, its closest rounding. Many companies have upgraded their design standards to the ISO system and adhere to the principles of CNC dimensioning. In this respect, drawings using metric units are much more practical.

Some dimensioning problems are related to an improper use of CAD software, such as AutoCAD. Some designers do not change default settings for the number of decimal places and every metric dimension ends up with three decimal places and every imperial dimension with four decimal places. This is a poor practice and should always be avoided. The most professional approach is to specify dimensional tolerances for all dimensions that require them, and even use *Geometric Dimensioning and Tolerancing* standards (GDT or GD&T). GD&T establishes relationship between two or more features of a drawing. For example, GD&T dimension will define concentricity of a hole with a surface. Before GD&T became part of design drafting, written messages and instructions cluttered the drawing. These drawings do not conform to international ISO standards.

Degrees

Angular dimensions should always be specified in decimal degrees (DD), according to modern standards. Older drawings will often contain angular dimensions specified in *degrees-minutes-seconds* (DMS or D-M-S). To convert DMS to DD:

$$DD = D + \frac{M}{60} + \frac{S}{3600}$$

where...

- DD = Decimal degrees
- D = Degrees (as per drawing)
- M = Minutes (as per drawing)
- S = Seconds (as per drawing)

For more details, see page 506.

◆ Tolerances

For high quality precision machining work, many part dimensions are specified within a certain *range of acceptable deviation from their nominal size*. For example, imperial units tolerance of $+0.001/0.000$ inches will be different from metric tolerance of $+0.1/0.0$ mm. Dimensions of this type are usually critical dimensions and require special attention during CNC machining. It may be true in many companies that CNC operators are ultimately responsible for maintaining required part sizes within tolerances (providing the program is correct) - but it is equally true, that CNC programmers can make machine operator's task a bit easier. Consider the following lathe example:

- ➊ Drawing dimension specifies a certain size as $\varnothing 75 + 0.00/-0.05$ mm. **What actual dimension should appear in the program?**

Programmer has several choices here. High side of the dimensional range may be programmed as X75.0, low side as X74.95. A middle value of X74.975 is also an option. Of course, any other dimension between high and low value is also a possibility, although not a practical one. Each selection is mathematically correct. A creative CNC programmer looks not only for mathematical aspects, but for technical aspects as well. Cutting edge of a tool wears out with more parts machined. That means the machine operator has to *fine-tune* part sizes by using *tool wear offsets*, available on most CNC systems. Such a manual interference during machining process is acceptable, but when done too often, it slows down the production and adds to overall costs.

Properly focused programming approach can control frequency of such manual adjustments to a great degree. Consider the $\varnothing 75$ mm mentioned earlier. If it is an *external* diameter, the tool edge wear will cause the actual dimension during machining to become progressively *larger*. In case of an *internal* diameter, the actual dimension will become *smaller* as the cutting edge wears out. By programming X74.95 for the external diameter (bottom limit) or X75.0 for the internal diameter (top limit), the cutting edge wear will move *into* the tolerance range, rather than *away* from it. Manual tool offset adjustment at the machine may still be required, but less frequently. Another approach is to select *middle size* of the tolerance range - this method will also have a positive effect but more manual adjustments may be necessary during machining.

◆ Surface Finish

Precision machined parts always require a certain degree of surface finish quality. Related drawings indicate desired surface finish quality for various part features. Imperial drawings define surface finish quality in *micro inches*, where *1 micro inch* = 0.000001 ". Metric drawings use surface requirements expressed in *microns*, where *1 micron* = 0.001 mm. Symbol for a micron is the Greek letter μ . Some drawings use standard symbols - Figure 6-4.

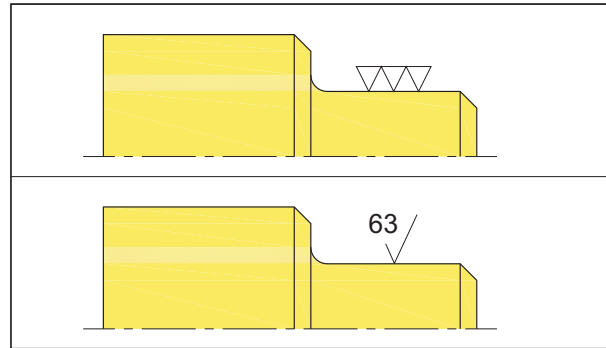


Figure 6-4

Surface finish marks in a drawing:
Metric (top) and imperial (bottom)

The most important factors influencing quality of surface finish are spindle speed, feedrate, cutting tool radius and the amount of material removed. Generally, a larger cutter radius and slower feedrates contribute towards *finer* surface finishes, while the opposite is equally true. Overall cycle time will be longer but can often be compensated for by elimination of any subsequent operations such as grinding, honing or even lapping.

◆ Drawing Revisions

Another important section of part drawings, often overlooked by CNC programmers, shows engineering changes (known as *revisions*) made on the drawing up to a certain date. Using reference numbers or letters (Figure 6-5), the designer identifies such changes, usually with both values - the previous and the new value - for example:

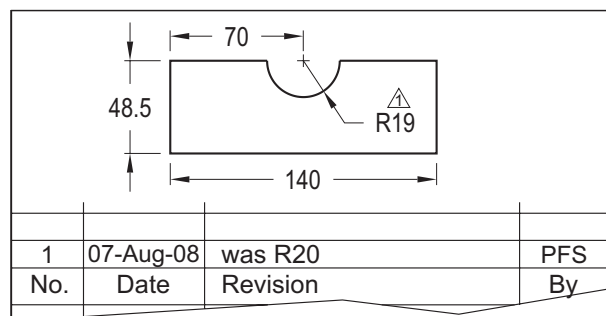


Figure 6-5

Drawing revision example

Only the *latest* changes for each dimension are important to CNC program development. Make sure such a program not only reflects the current (latest) engineering design, but also is identified in some unique way to distinguish it from any previous program versions. Many programmers keep a copy of the part drawing corresponding to the stored program, thus preventing possible misunderstandings later.

◆ Special Instructions

Many drawings also include special instructions and comments that cannot be expressed with established drafting symbols and are therefore spelled out independently, in words. Such instructions are very important for CNC program planning, as they may significantly influence the programming procedure. For example, a certain part element is identified as a *ground* surface or diameter. Drawing dimensions always show the *finished* size. During program development, this dimension likely has to be adjusted for any grinding allowance necessary. Actual amount of such allowance is selected by the programmer and should be written as a special instruction in the program. Another example of special instructions relates to any machining done during part assembly. For example, a certain hole should be drilled and tapped, and is dimensioned exactly as any other hole, but a special instruction indicates the drilling and tapping must be done when the part is handled during assembly. Operations relating to such a hole are *not* programmed and any overlook of small instruction such as this example, may result in an unusable part.

Many drawing instructions use a special pointer called a *leader*. Usually it is a line ending with an arrow, pointing towards the specific area that it relates to. For example, a leader may be pointing to a hole, with a worded caption:

Ø12 - REAM 2 HOLES - 20 DP

This is a requirement to ream 2 holes to the full depth of 20 mm with a reamer that has 12 mm diameter.

METHODS SHEET

Some companies have a staff of qualified manufacturing technologists or process planners responsible for determination of all manufacturing processes. Their responsibility is to develop a series of machining instructions, detailing the exact route of each part through the manufacturing process. They allocate work to individual machines, develop machining sequences and setup methods, select tooling, etc. Their instructions are written in a *methods sheet* (also called a *routing sheet*) that accompanies the part through all stages of manufacturing, typically in a plastic folder (or as a computer file). If such a sheet is available, its copy should become part of CNC documentation. One main purpose of a methods sheet is to provide CNC programmer with as much information as possible to shorten turnaround between programs. One great advantages of methods sheets in programming is their comprehensive coverage of all required operations, both CNC and conventional, thus offering a complete overview of the manufacturing process. High quality methods sheet will save a lot of decisions - if it is made by a qualified manufacturing engineer, who specializes in work detailing. The ideal methods sheet is one where all recommended CNC manufacturing processes closely match established part programming methods.

For various reasons, many CNC machine shops do not use methods sheets, routing sheets or similar documentation. CNC programmer acts as a process planner as well. Such an environment offers a certain amount of flexibility but demands a large degree of knowledge, skills and responsibility from one person at the same time.

MATERIAL SPECIFICATIONS

Also important consideration in program planning is evaluation of the *material stock*. Typical material is raw and unmachined (a bar, billet, plate, forging, casting, etc.). Some material may be already premachined, routed from another machine or operation. It may be solid or hollow, with a small or a large amount to be removed by CNC machining. The *size* and *shape* of material determines the setup mounting method. The *type* of material (steel, cast iron, brass, etc.) will influence not only the selection of cutting tools, but the cutting conditions for machining as well.

A program cannot be planned without knowing the type, size, shape and condition of the material

◆ Material Uniformity

Another important consideration, often neglected by programmers and managers alike, is the *uniformity* of material specifications within a particular batch or from one batch to another. For example, a casting or forging ordered from two suppliers to be used for the *same part* may have slightly different sizes, hardness and even shape. Another example is a material cut into single pieces on a saw, where the length of each piece varies beyond an acceptable range. This inconsistency between blank parts makes programming more difficult and time consuming. It also creates potentially unsafe machining conditions. If such problems are encountered, the best planning approach is to place emphasis on machining safety than on machining time. At worst, there will be some *air cutting* or slower than needed cutting feedrate, but no machining cuts will be too heavy for the tool to handle.

Yet another approach is to separate non-uniform material into groups and make separate programs for each group, properly identified. The best method is to cover all known and predictable inconsistencies under program control, for example, using the block skip function.

◆ Machinability Rating

Also very important aspect of material specification is its *machinability*. Charts with suggested speeds and feeds for most common materials are available from major tooling companies. These charts are helpful in programming, particularly when an unknown material is used. The suggested values are a good starting point, and can be optimized later, when the material properties are better known. See *page 523* for some very basic and general suggestions.

Machinability rating in the imperial units is given in units called *feet per minute* (ft/min). Often the terms *surface feet per minute*, *constant surface speed* (CSS), *cutting speed* (CS), *peripheral speed* or just *surface speed* are used instead. For metric designation of machinability rating, *meters per minute* (m/min) are used. In both cases, the *spindle speed* (r/min) for a given tool diameter (for a mill) or a given part diameter (for a lathe) is calculated, using common formulas. For imperial units, the spindle speed can be calculated in *revolutions per minute* (r/min):

$$r / \text{min} = \frac{12 \times \text{ft} / \text{min}}{\pi \times D}$$

For metric calculations, the formula is similar:

$$r / \text{min} = \frac{1000 \times \text{m} / \text{min}}{\pi \times D}$$

where ...

- r/min = Revolutions per minute (spindle speed S)
- 12 = Converts feet to inches
- 1000 = Converts meters to millimeters
- ft/min = Cutting speed in feet per minute
- m/min = Cutting speed in meters per minute
- π (pi) = Constant value of 3.141593...
- D = Tool diameter (milling) or
Part diameter (turning) - in inches or mm

MACHINING SEQUENCE

Machining sequence defines the order of machining operations. Technical skills and machine shop experience do help in program planning, but a good quantity of common sense approach is equally important. Machining sequences must have logical order - for example, drilling must be programmed before tapping, roughing operations before finishing, first operation before second, etc. Within this logical order, further specification of the order of individual tool motions is required for a particular tool. For example, in turning, a face cut may be programmed on the part first, then roughing all material on diameters will take place. Another method is to program a roughing pass for the first diameter, then face and continue with the remainder of the diameter roughing afterwards. In drilling, a spot drill before drilling may be useful for some applications, but in another program a center drill may be a better choice. There are no fixed rules, no absolutes, that determine which method is better - each CNC programming assignment has to be considered individually, based on the basic criteria of safety, quality, and efficiency.

Machining sequence must follow logical order

Basic approach for determining a machining sequence is careful evaluation of all related operations. In general, program should be planned in such a way that the cutting tool, once selected, will do as much work as possible, before a tool change. On most modern CNC machines, much less time is needed for positioning a tool than for a tool change. Another consideration is in benefits gained by programming all heavy operations first, then the lighter semi-finishing or finishing operations. It may mean an extra tool change or two, but this method minimizes any shift of material in the holding fixture while machining. Another important factor is the current position of cutting tool when a certain operation is completed. For example, when drilling a pattern of holes in the order of 1-2-3-4, the next tool (such as a boring bar, reamer or a tap) could be programmed in opposite order of 4-3-2-1 to minimize unnecessary tool motions - *Figure 6-6*.

T01 = Spot Drill	T02 = Drill	T03 = Tap
Hole 1	Hole 4	Hole 1
Hole 2	Hole 3	Hole 2
Hole 3	Hole 2	Hole 3
Hole 4	Hole 1	Hole 4

Figure 6-6

Typical machining sequence for three common hole operations

This machining sequence may have to be changed after final selection of tools and setup method. Although practical in many cases, the reverse machining sequence may not be practical when stored as subprograms. This important subject starts on *page 383*).

Program planning is not an independent execution of individual steps - it is a very interdependent and very logically and coherent approach to achieve a certain goal.

TOOLING SELECTION

Selecting tool holders and cutting tools is another important step in planning a CNC program. Category of tooling, their selection and usage, includes a lot more than just cutting tools and tool holders - it covers an extensive line of accessories, involving numerous vises, fixtures, chucks, sub-tables, steady-rests, tailstock, indexing tables, clamps, collets and many other holding and auxiliary devices. Cutting tools require special attention, due to the very large variety available and their direct effect in machining.

Cutting tool used for the job is usually the most important selection. It should be selected by two main criteria:

- Efficiency of usage
- Safety in operation

Many supervisors responsible for CNC programming try to make the existing tooling work at all times. Often they ignore the fact that a suitable new tool may do the job faster and more economically. A thorough knowledge of tooling and its applications is a separate technical profession - the programmer should know well all general principles of cutting tool applications. In many cases, a tooling company representative may provide additional valuable assistance.

Arrangement of tools in the order of usage is also a subject of serious consideration in CNC program planning. On CNC lathes, each cutting tool is assigned to a specific turret station, which requires consideration about the *distribution of tools* - how they are arranged between short and long tools (such as short turning tools versus long boring tools). This is important for prevention of a possible interference during cutting or tool changing. Another concern should be the *order* in which each cutting tool is called, particularly for machines that do not have a bi-directional tool indexing. Most machining centers use a random type tool selection, where the order of tools is unimportant, only the diameter of the tool and its weight has to be considered.

All tool offset numbers and other program entries should be documented in a form known as the *tooling sheet*. Such a document serves as a guide to the operator during job setup. It should include at least some basic documentation relating to the selected tool. For example, such documentation may include the tool description, its length and diameter, number of flutes, tool and offset numbers, speed and feeds selected for that tool and other relevant information.

PART SETUP

Another decision in program planning relates to the actual part setup - *how to mount* the raw or premachined material, what *supporting* tools and other devices should be used, how many *operations* are required to complete as many machining sequences as possible, where to select a *program zero*, etc. Setup is necessary and it should be done efficiently.

Some types of machines are designed to make the setup time more productive. Multi spindle machining centers or lathes can handle two or more parts at the same time. Special features, such as tool offset settings, barfeeder for a lathe, an automatic pallet changer or dual setup on the table, also help. Other solutions can be added as well.

◆ Setup Sheet

At this stage of program planning, once the setup is decided, making a *setup sheet* is a good idea, particularly for jobs with many tools and/or operations. A setup sheet can be a simple sketch, designed mostly for use at the machine, that shows part orientation when mounted in a holding device, tool offset numbers used by the program, datum points and, of course, all necessary identifications and descriptions. Other information in a setup sheet should relate

to some unique requirements established during planning stages of the program (such as position of clamps, bored jaws dimensions, depth of clamping, limits of tool extension, etc.). Setup sheet and tooling sheet can be combined into a single source of information. Most programmers use their own various versions.

TECHNOLOGICAL DECISIONS

The next stage of CNC program planning involves selection of spindle speeds, cutting feedrates, depth of cut, coolant application, etc. All of the already considered factors will have their influence. For example, the available range of spindle speeds is fixed for any CNC machine, size of the cutter and material type will influence speeds and feeds, power rating of the machine tool will help determine what amount of material can be removed safely, etc. Other factors that influence program design include tool extensions, setup rigidity, cutting tool material and its condition. Not to be overlooked is the proper selection of cutting fluids and lubricants - they, too, are important for overall part quality.

◆ Cutter Path

Main core of any CNC programming is the precise determination of cutter path - known as the *tool path* or *toolpath*. This process involves individual cutter movements in its relationship to the part.

In CNC programming, always look at the cutting tool as being moved around the work !!

THIS PRINCIPLE APPLIES TO ALL CNC MACHINE TOOLS

The key factor for understanding this principle is to always visualize the *tool* motion, *not* the machine motion. The most noticeable difference between programming a machining center as compared to a lathe program is the cutter rotation as compared to part rotation.

In both cases, the programmer always must think in terms of the *cutter moving around the part* - Figure 6-7.

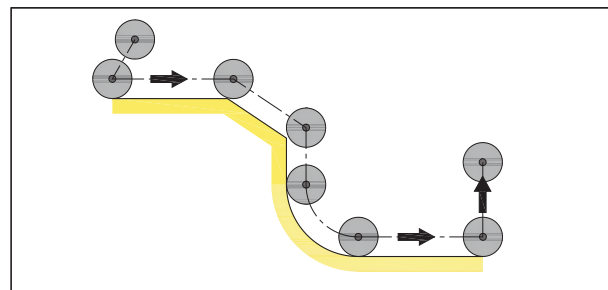


Figure 6-7

Contouring toolpath motion - as intended for milling or turning

Toolpath for all contouring tools always has to take into consideration the cutter *radius* (tool nose radius for turning and boring), either by programming an *equidistant tool-path* for the radius center or by using *cutter radius offset* (also known as *cutter radius compensation* or *tool nose radius compensation*). CNC machines for milling and turning are provided with rapid motion, linear interpolation and circular interpolation, all as standard features. To generate more complex paths, such as a helical milling motion or program rotation, a special option has to be available in the control unit. Two groups of typical toolpaths are used:

- Point-to-point *also called* Positioning
- Continuous *also called* Contouring

Positioning is used for a point location operations, such as drilling, reaming, tapping and similar operations; *continuous path* generates a contour (profile). In either case, all programmed data refer to the cutter position when a certain motion is completed.

This position is called *tool target position* - Figure 6-8.

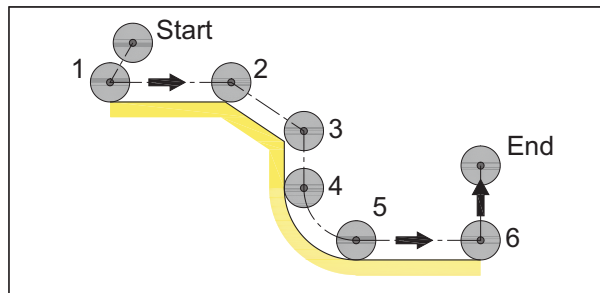


Figure 6-8

Contouring toolpath motion - target positions identified

The contour start and end positions are identified and so are the positions for each contour change. Each target position is called the *contour change point*, which has to be carefully calculated. The order of target locations in the program is very important. Based on the above illustration, it means that tool *position 1* is the *target* position commencing at the *Start* point, *position 2* is the *target* position beginning at point 1, *position 3* is the *target* from point 2 and so on, until the *End* position is reached. If the contour is for milling, these targets will be in X and Y axes. In turning, they will be in X and Z axes.

Most contouring operations require more than just one cutting motion, for example, roughing and finishing. Part of the programming process is to isolate areas that need roughing. Can one cutting tool do all operations? Can all tolerances be maintained? Is the tool wear a problem? Can the surface finish be achieved? When programming non-cutting rapid motions, take the same care as with cutting motions. A particular focus should be to minimize rapid tool motions and ensure safe clearances.

◆ Machine Power Rating

Machine tools are rated by their power, as one important specification for CNC programming. Laws of physics define that heavy cuts require more power than light cuts. A depth or width of a cut that is too large can break the tool and stall the machine. Such cases are unacceptable and must be prevented. CNC machine specifications list typically identifies power rating of the motor at machine spindle. This rating is either in *kW* (kilowatts) or *HP* (horsepower). Several formulas are available for calculations relating to various power ratings, establishing metal removal rates, tool wear factors, etc.

One of the more useful formulas is the comparison of *kW* and *HP* (based on 1 HP = 550 foot-pounds per second):

1 kW = 1.341022 HP	or	~ 1.3 HP
1 HP = 0.7456999 kW	or	~ 0.75 kW

The topic of power and forces in machining can be complex and is not always needed in everyday programming. There are publications devoted to this particular subject. Work experience is often a better teacher than formulas.

◆ Coolants and Lubricants

When cutting tool contacts the material for an extended period of time, great amount of heat is generated. Cutting edges get overheated, become dull and an insert or the whole tool may break. To prevent such possibilities, a suitable *coolant* must be used.

Water soluble, bio-degradable, oil is the most common coolant. A properly mixed coolant dissipates heat from the cutting edge and it also acts as a lubricant. The main purpose of lubrication is to reduce friction and make the metal removal easier. Flood of the coolant should aim at the tool cutting edge, with a flexible pipe or through a coolant hole built into the tool itself.

Never use plain water as coolant -
it may severely damage the machine tool

CNC operator is responsible for a suitable coolant in the machine. Coolant should be clean and mixed in recommended proportions. Water soluble oils should be biodegradable to preserve the environment and properly disposed of. CNC programmer decides when to program coolant and when not. Ceramic cutting tools are normally programmed dry, *without a coolant*. Some cast irons do not require flood coolant, but air blast or oil mist may be allowed. These coolant functions vary between machines, so check the machine reference manual for further details.

Flood coolant may be used to cool down the part and gain better tolerances. It can also be used to flush away chips from congested areas, such as deep holes and cavities.

The main benefits of cutting fluids far outweigh their inconveniences. Cutting fluids are often messy, the cutting edge cannot be seen during operation, operator may get wet and sometimes old coolant smells. With proper coolant management, all problems related to coolants can be controlled.

A coolant related programming issue is *when* to turn the coolant ON in the program. As the coolant ON function M08 only turns on the pump motor, make sure the coolant actually reaches the tool edge *before* contact with work. Programming coolant ON earlier is better than too late.

In summary, using coolant on CNC machining centers and lathes serves three major purposes:

- ... to dissipate heat from cutting edge and part
- ... to lubricate area between tool edge and part
- ... to flush away chips

WORK SKETCH AND CALCULATIONS

Manually prepared programs always require several mathematical calculations. This part of program preparation intimidates many programmers but is a necessary step, even for simple operations. Many complex contours will require more calculations, but not necessarily more complex calculations. Almost any mathematical problem in CNC programming can be solved by the use of arithmetic, algebra and trigonometry - that's it!. Advanced fields of mathematics - analytic geometry, spherical trigonometry, calculus, surface calculations, etc. - are required for programming complex molds, dies and similar shapes. In such cases, a CAD/CAM programming system is necessary.

Those who can solve a right angle triangle can make calculations for almost any CNC program. At the end of this handbook is an overview of some common mathematical problems and applications. When working with more difficult contours, it is often not the solution itself that is difficult, it is the ability to arrive at the solution. CNC programmer must have the ability to *see* exactly what triangle has to be solved, what formula to use, what approach to select. It is not unusual to do several intermediate calculations before the required final coordinate point can be established.

Calculations of any type often benefit from a visual (pictorial) representation. Such calculations usually need a working sketch. A simple sketch can be drawn by hand and should be done in an *approximate scale*. Larger sketch scales are much easier to work with. Making the sketch in at least approximate scale has one great advantage - *you can immediately see various relationships* - what dimensions should be smaller or larger than the others, the relationship of individual elements, the shape of an extremely small detail, etc. All that said, there is one purpose a sketch should *never* be used for, regardless how accurate it is:

Never use a scaled sketch to guess unknown dimensions !

Scaling a drawing is a poor and unprofessional practice, that creates more problems than it solves. It is a sign of either laziness or incompetence, commonly both.

◆ Identification Methods

A sketch used for calculations can be done directly in the drawing, or on paper or even using CAD software. Every sketch is associated with several mathematical calculations - they make the sketch necessary in the first place. Using color coding or point numbering as identification methods offers benefits and better organization. Rather than writing coordinates at each contour change point in the drawing, use point reference numbers and create a separate coordinate sheet form using the reference numbers, as illustrated in *Figure 6-9*. A printable sample of a coordinate sheet is also included on the enclosed CD.

Position	X-axis	Y-axis	Z-axis

Figure 6-9

Coordinate sheet example - blank form (no data)

Such a sheet can be used for milling or turning, by filling only the applicable columns. The aim is to develop a consistent programming style from one program to another. Fill-in all values, even those that do not change.

A completed coordinate sheet is a better programming reference, as shown in *Figure 6-10*.

Position	X-axis	Y-axis	Z-axis
START	X57.0	Y126.0	
P1	X45.0	Y105.0	
P2	X108.258	Y105.0	
P3	X146.156	Y79.6	
P4	X146.156	Y50.0	
P5	X173.156	Y23.0	
P6	X238.0	Y23.0	
END	X238.0	Y62.0	

Figure 6-10

Coordinate sheet example - filled form for a milling toolpath

QUALITY IN CNC PROGRAMMING

All parts machined on CNC equipment are evaluated by their quality - usually *after* they are machined. Quality inspectors check many features - are the dimensions within tolerances? Is the surface finish up to standard? Is there a consistency between parts, etc.? Modern CNC equipment provides optional inspection related features during machining process, for example, *in-process gaging*. Many machine shops require their CNC machinists to be quality inspectors while the parts are being made. One subject that is equally important, yet not very often mentioned, is quality in CNC programming.

CNC programming starts with a plan. Although number one quality of a CNC programmer is knowledge and skill, there are at least two related and equally important considerations in program planning - programmer's *personal approach* and *professional attitude*. How the CNC programmer approaches a certain job, assignment or project will have a great influence on the final outcome of parts produced by the CNC operator. Programmer's attitudes have significant influence on program development and final results. They also have significant influence on the CNC operator - it's just human nature.

Ask yourself some questions. As a CNC programmer - are you attentive to detail, are you precision minded, are you well organized, are you concerned if something is not done right? Do you '*cut corners*' just to have the job done? Can a program you just developed, or an existing program, be improved further to make it safer and more efficient?

CNC program quality is much more than writing an error free program - that is the absolute prerequisite and goes without saying. Quality in programming includes concern as how a program affects the CNC operator, machine setup, and actual part machining. Quality in programming means constant effort at improvement and desire to make the next program even better.

Consistency in programming is one the best ways to achieve high quality programs. Once a certain method or process been found superior to others, stick with it. Use the same method again and again. CNC operators like nothing less than programs that vary in structure.

Part complexity should never stand in the way - it is only related to *your* knowledge level and willingness to solve problems. It should be a personal goal to make a program - every program - the best program possible.

Set your quality standards high !

7

PART PROGRAM STRUCTURE

Typical CNC program is composed of series of sequential instructions related to machining a part. Each instruction is specified in a format the CNC system can accept, interpret and process. Each instruction must also conform to machine tool specifications. This program input method can be defined as *an arrangement of machining instructions and related tasks*, written in the format of a CNC system and aimed at a particular machine tool.

Various controls may have different formats, but most are similar. Subtle differences exist among CNC machines from different manufacturers, even those equipped with the same control system. This is common, considering the specific demands individual machine builders place upon the control manufacturer to accommodate many original and unique machine design features. Such variations are usually minor but still important for programming.

BASIC PROGRAMMING TERMS

The field of CNC has its own terminology, special terms and its jargon. It has its own abbreviations and expressions that only people in the field understand. CNC programming is only a small section of computerized machining and it has a number of its own expressions. The majority of them relate to the *structure* of part programs.

There are *four basic terms* used in CNC programming. They appear in professional articles, books, papers, lectures and so on. These words are the key to understanding general CNC terminology:

Character → Word → Block → Program

Each term is very common and equally important in CNC programming and deserves its own detailed explanation.

◆ Character

A character is the smallest unit of CNC program. It can have one of three forms:

- Digit
- Letter
- Symbol

Characters are combined into meaningful CNC *words*. This combination of digits, letters and symbols is called the *alpha-numerical* program input.

Digits

There are ten digits, 0 to 9, available for use in a program to create numbers. The digits are used in two modes - one for *integer* values (numbers without a decimal point), the other for *real numbers* (numbers with a decimal point). Numbers can have *positive* or *negative* values. On some controls, real numbers can be used with or without the decimal point. Numbers applied in either mode can only be entered within the range that is allowed by the control system.

Letters (Addresses)

All twenty six letters of the English alphabet are available for programming, at least in theory. Most control systems will accept only certain letters and reject others. For example, a two-axis CNC lathe control will reject the letter Y, as the Y-axis is typically unique to milling operations (milling machines and machining centers). On the other hand, many CNC lathes with milling capabilities will accept the letter (address) Y, if the Y-axis is available. Capital letters are normal designation in CNC programming, but some controls accept low case letters with the same meaning as their upper case equivalent. ***If in doubt, use capital letters.***

Every control accepts CAPITAL letters
but not all controls accept low case letters

Symbols

Several symbols are used for programming, in addition to the ten digits and twenty six available letters. The most common symbols are the *decimal point*, *minus sign*, *percent sign*, *parenthesis* and others, depending on the control options. Their use in a program is strictly defined. Decimal point is used for values expressed in mm, inches or degrees. Minus sign is used to identify a dimensional value as negative, percent sign is used for file transfers, and parentheses are used for program comments and messages.

◆ Word

A program *word* is a combination of *alpha-numerical* characters, creating a single instruction to the CNC. Each word begins with a capital letter, followed by a number representing a program code or actual value. Typical words indicate *axes position*, *feedrate*, *speed*, *preparatory commands*, *miscellaneous functions* and many others.

Word is the unit of instruction to the control system

◆ Block

Just like one word is used as a *single* instruction to the CNC system, program block is used as a *multiple* instruction. A program entered into the control system consists of individual lines of instructions, sequenced in a logical order of processing. Each line - called a *sequence block* or simply a *block* - is composed of one or several *words* and each word is composed of *two or more characters*.

In the control system, each block must be separated from all others. To separate blocks in the MDI (*Manual Data Input*) mode at the control, each block has to end with a special *End-Of-Block* code (symbol). This code is marked as EOB on the control panel. When preparing part program on a computer using a keyboard, using the *Enter* key will terminate the block (similar to the old *Carriage Return* on typewriters). When writing a program on paper first, each program block should occupy only a single line on the paper. Each program block contains a series of individual instructions that are executed simultaneously.

◆ Program

Part program structure may vary quite a bit for different controls, but the logical programming approach does not change from one control to another. CNC program usually begins with a *program number* or similar identification, followed by sequenced blocks of instructions in a logical order. The program ends with a *stop code* or a program termination symbol, such as the *percent sign* - %. Some controls also require the *stop code* at the program beginning. Internal documentation and messages to the CNC operator may be located in strategic places within the part program. Programming format has evolved significantly during the years and several formats have emerged.

PROGRAMMING FORMATS

Since the early days of numerical control, three formats had become significant in their time. They are listed in the order of their original introduction:

- Tab Sequential Format no decimal point available
... old NC systems only
- Fixed Format no decimal point available
... old NC systems only
- Word Address Format decimal point available
... typically used with
CNC systems

Only the *very early* control systems use the *tab sequential* or *fixed* formats. Both of them disappeared in the early 1970's and are now obsolete. They have been replaced by much more convenient *Word Address Format*. Its greatest benefit is using addresses for words and decimal point format when necessary.

WORD ADDRESS FORMAT

The *word address format* is based on a combination of one letter and one or more digits - *Figure 7-1*.

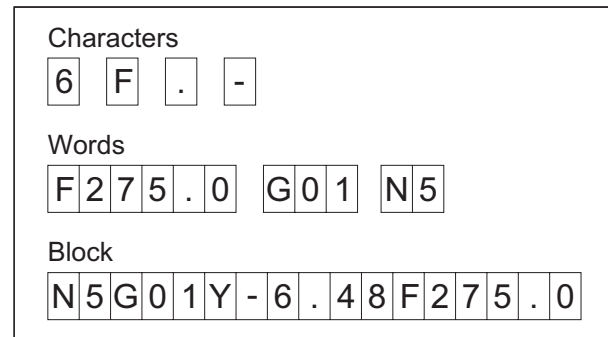


Figure 7-1

Typical word address programming format

In some applications, such a combination can be supplemented by a symbol, such as a minus sign or a decimal point. Each letter, digit or symbol represents one character in the program and in control memory. This unique alpha-numerical arrangement creates a *word*, where the letter is called the *word address*, followed by numerical data with or without symbols. The *word address* refers to a specific register of the control memory. Some typical words are:

G01 M30 D25 X15.75 N105 H01 Y0 S2500
Z-5.14 F12.0 T0505 T05 /M01 B180.0

The *address* (letter) in the block defines the word *meaning* and must always be written first. For example, X15.75 is correct, 15.75X is not. No spaces (space characters) are allowed *within* a word - X 15.75 is *not* correct - spaces are only allowed *before* the word, meaning before the letter, between words.

Data always indicate the word numerical assignment. This value varies greatly and depends on the preceding address. It may represent a sequence number N, a preparatory command G, a miscellaneous function M, an offset register number D or H, a coordinate word X, Y or Z, feedrate function F, spindle function S, tool function T, etc.

Any one *word* is a series of characters (at least two) that define a single instruction to the machine control unit. The above examples of typical words have the following meaning in a CNC program:

G01	Preparatory command
M30	Miscellaneous function
D25	Offset number selection - milling applications
X15.75	Coordinate word - as a positive value
N105	Sequence number (block number)
H01	Tool length offset number

Y0	<i>Coordinate word - as zero value</i>
S2500	<i>Spindle speed function</i>
Z-5.14	<i>Coordinate word - as a negative value</i>
F12.0	<i>Feedrate function</i>
T0505	<i>Tool function - turning applications</i>
T05	<i>Tool function - milling applications</i>
/M01	<i>Miscellaneous function w/block skip symbol</i>
B180.0	<i>Indexing table function</i>

Individual program words are instructions grouped together to form logical sequences of programming code. Each sequence that will process one series of instructions simultaneously forms a unit called a *sequence block* or *program block* or - simply a *block*. The series of blocks is arranged in a logical order that is required to machine a *complete* part or a *complete* operation is called a *part program* also known as a *CNC program*.

The next block shows a rapid tool motion to the absolute position of X13.0Y4.6 within current units setting and with a coolant turned on:

N25 G90 G00 X13.0 Y4.6 M08

where ...

N25	Sequence or block number
G90	Absolute mode
G00	Rapid motion mode
X13.0 Y4.6	Coordinate location
M08	Coolant ON function

The control will always process any single block as one complete unit - *never partially*. Most controls allow a random word order in a block, as long as the block number is specified first. Many programmers follow an unofficial - but recommended - order of various words in a block. For example, G-codes are listed first, followed by axes data, than all remaining instructions.

Block number must always be specified first

FORMAT NOTATION

Each program word can only be written in a specific way. The number of digits allowed in a word, depending on the address and maximum number of decimal places, is set by the control manufacturer. Not all letters can be used. Only letters with an assigned meaning can be programmed, except in a comment. Symbols can be used in only some words, and their position in the word is fixed. Some symbols are used only in custom macros - control limitations are important. Symbols supplement the *digits* and *letters* and provide them with an additional meaning. Typical programming symbols are the minus sign, decimal point, percent sign and a few others. All symbols are listed in a table on page 47.

◆ Short Forms

Control manufacturers often specify the input format in an abbreviated form - *Figure 7-2*.

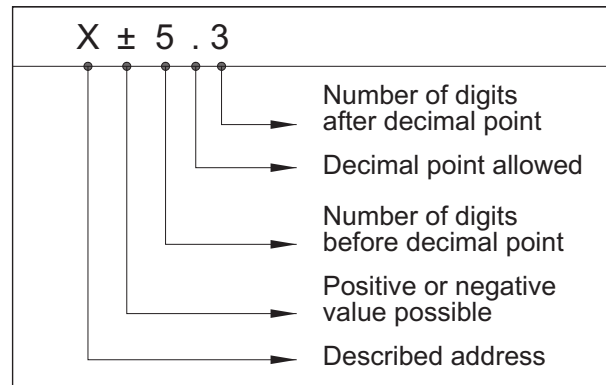


Figure 7-2

Word address format notation - X axis format in metric mode shown

Describing the full format description for each meaning would be unnecessarily too long. Consider the following complete and *not* abbreviated description of address X - as a coordinate word that is used in metric system:

Address X accepts positive or negative data with the maximum of five digits in front of a decimal point and three digits maximum after the decimal point - decimal point is allowed

Absence of decimal point in the notation means that decimal point is not used; the absence of a plus/minus (\pm) sign means that the address value cannot be negative - *a lack of sign means a positive value by default*.

These samples of format notation explain the shorthand:

G2 Two digits maximum, no decimal point or sign

N4 Four digits maximum, no decimal point or sign

S5 Five digits maximum, no decimal point or sign

F3.2 Five digits maximum, three digits maximum in front of the decimal point, two digits maximum behind the decimal point, decimal point is allowed, no sign is used

Be extra careful when evaluating shorthand notations from a control manual or other source. There are no industry standards and not all control manufacturers use exactly the same methods, so the meaning of short forms listed may vary significantly. Typical list of word addresses, their format notation and description is shown in the following tables. They contain address notations based on typical Fanuc control systems.

⚠ Note:

The presented format notation concept is often applied by CAD/CAM software developers for CNC program output using post processors

◆ Milling System Format

Address descriptions vary for many addresses, depending on the input units. The table below lists metric format descriptions (imperial format is in parenthesis, if applicable). Listed are format notations for milling units.

Some addresses have additional meaning when used in special applications
--

The first column is the address, second column is the format notation, third column is the description:

Address	Notation	Description
A	A±5.3	Rotary or Indexing axis - unit is degrees - used about the X axis
B	B±5.3	Rotary or Indexing axis - unit is degrees - used about the Y axis
D	D2	Cutter radius offset number (sometimes uses address H)
F	F5.3	Feedrate function - may vary
G	G2	Preparatory commands (G3 for extended G-code set)
H	H3	Offset number (tool position and/or tool length offset)
I	I±5.3 (I±4.4)	Arc center modifier for X axis Shift amount in fixed cycles (X) Corner vector selection for X axis (old type controls)
J	J±5.3 (J±4.4)	Arc center modifier for Y axis Shift amount in fixed cycles (Y) Corner vector selection for Y axis (old type controls)
K	K±5.3 (K±4.4)	Arc center modifier for Z axis
K (L)	K4 (L4)	Fixed cycle repetition count Subprogram repetition count
M	M2	Miscellaneous function (M3 for extended M-function set)
N	N5	Block number or sequence number (N4 for older controls)
O	O4 (O5)	Program number (EIA) or :4 or :5 for ISO - not common)
P	P4	Subprogram number call Custom macro number call
	P3	Work offset number - used with G10
	P53	Dwell time in milliseconds
	P5	Block number in main program when used with M99

Address	Notation	Description
Q	Q5.3 (Q4.4)	Depth of peck in fixed cycles G73 and G83
	Q±5.3 (Q±4.4)	Shift amount in fixed cycle G76 and G87
R	R±5.3 (R±4.4)	Retract point in fixed cycles Arc radius designation
S	S5	Spindle speed in r/min
T	T4	Tool function
X	X±5.3 (X±4.4)	X axis coordinate value designation
	X5.3	Dwell function with G04 only
Y	Y±5.3 (Y±4.4)	Y axis coordinate value designation
Z	Z±5.3 (Z±4.4)	Z axis coordinate value designation

◆ Turning System Format

Similar chart as for milling, this one is for lathe systems. A number of definitions are the same and are included only for convenience. Notation is in the metric format, imperial notation is in parenthesis, if applicable to the address.

Some addresses have additional meaning when used in multiple repetitive cycles of the G70 series or special applications
--

Address	Notation	Description
A	A3	Angle of thread for G76 Angle for direct drawing input
C	C±5.3 (C±4.4)	Chamfer for direct drawing input
	C±5.3	Absolute axis rotation in degrees
D	D4	Number of divisions in G73
	D53 (D44)	Depth of cut in G71 and G72 Relief amount in G74 and G75 Depth of first thread in G76
E	E3.5 (E2.6)	Precision feedrate for threading
F	F3.3 (F2.4)	Feedrate function - may vary
G	G2	Preparatory commands (G3 for extended G-code set)
H	H±5.3	Incremental axis rotation in degrees

Address	Notation	Description
I	I±5.3 (I±4.4)	Arc center modifier for X axis Taper height in X for cycles X axis relief in G73 Direction of chamfering Motion amount in X in G74
K	K±5.3 (K±4.4)	Arc center modifier for Z axis Taper height in Z for cycles Z axis relief in G73 Direction of chamfering Motion amount in Z in G75 Thread depth in G76
K (L)	K4 (L4)	Subprogram repetition count
M	M2	Miscellaneous function (M3 for extended M-function set)
N	N5	Block number or sequence number (N4 for older controls)
O	O4 (O5)	Program number (EIA) or :4 or :5 for ISO - not common)
P	P4 (P5)	Subprogram number call Custom macro number call Offset number with G10 Return to block number with M99 Start block number in G71 and G72
	P53	Dwell time in milliseconds
Q	Q4 (Q5)	End block number in G71 and G72
R	R±5.3 (R±4.4)	Arc radius designation Arc radius for corners
S	S5	Spindle speed in r/min or ft/min
T	T4	Tool function
U	U±5.3 (U±4.4)	Incremental value in X axis Stock allowance in X axis
	U5.3	Dwell function with G04
W	W±5.3 (W±4.4)	Incremental value in Z axis Stock allowance in Z axis
X	X±5.3 (X±4.4)	Absolute value in X axis
	X5.3	Dwell function with G04 only
Z	Z±5.3 (Z±4.4)	Absolute value in Z axis

◆ Multiple Word Addresses

One feature that is noticeable in both tables is the abundance of different meanings for some addresses. This is a necessary feature of the word address format. There are only twenty six letters in the English alphabet but more than that the number of commands and functions. As new control features are added due to technological advance-

ments, even more variations may be necessary in the future. Several addresses have such an established meaning (for example, X, Y and Z are coordinate words), that giving them any additional meaning would be confusing. Many letters, on the other hand, are not used very often and their multiple meaning is quite acceptable (typical addresses are I, J, K, P, Q, for example). In addition, the actual meaning of addresses varies between milling and turning systems and even between manufacturers.

The control system has to have some means of accepting a particular word with a precisely defined meaning in the program. In most cases, the preparatory command G will define its meaning, at other times it will be an M function or a setting of system parameters.

SYMBOLS IN CNC PROGRAMMING

In addition to the basic symbols, Fanuc can accept other symbols for different applications. The following table describes all symbols available on Fanuc and compatible control systems:

Symbol	Description	Comment
.	Decimal point	Fractional part of a number
+	Plus sign	Positive value or <i>addition</i> sign in Fanuc macros
-	Minus sign	Negative value or <i>subtraction</i> sign in Fanuc macros
*	Multiplication sign	<i>Multiplication</i> sign in Fanuc macros
/	Slash (front slash)	Block skip function symbol or <i>division</i> sign in Fanuc macros
()	Parenthesis	Program comments & messages
%	Percent sign	Stop code (end of program file)
:	Colon	Program number designation (rare)
,	Comma	Used only within comments
[]	Brackets	Calculations in Fanuc macros
;	Semicolon	Non programmable End-Of-Block symbol (screen display only)
#	Sharp sign	Variable definition or call in Fanuc macros
=	Equal sign	Equality symbol in Fanuc macros

The table above lists typical standard and special symbols available on most control systems.

Special symbols are used only with optional features, such as *custom macro* option. These symbols cannot be used in standard programming, as they would cause an error. Typical standard symbols are found on the computer keyboard. *Ctrl*, *Shift* and *Alt* character combinations are not allowed in CNC programming.

◆ Plus and Minus Sign

One of the most common symbols in mathematics is an algebraic sign - *plus* or *minus*. In CNC, any data in a motion command can be either positive or negative. Virtually all control systems allow for the omission of a plus sign for all positive values. This feature is sometimes called *positive bias* of the control system. This is a mathematical term indicating assumed positive value of a number, if no sign is programmed. In CNC, the implied positive sign is always *after* the address (letter) in a word:

X+125.0 *is the same as* **X125.0**

For negative numbers, the minus sign symbol must always be programmed. If minus sign is missing, the number automatically becomes positive, with an incorrect result value, in this case, a possible tool position:

X-125.0 *Negative value (- sign is necessary)*
X125.0 *Positive value (no sign means positive value)*
X+125.0 *Positive value (+ sign is ignored)*

Symbols supplement letters and digits and are important part of CNC program structure. Not all symbols are used in every program - in fact, only very few are normally used.

TYPICAL PROGRAM STRUCTURE

Although it may be a bit early to show a complete program, it will do no harm to look at a typical program structure. In the next two examples, a typical mill-type structure is shown - one for fixed cycles, one for other machining:

```
O0701 (ID MAX 15 CHARS)
(SAMPLE PROGRAM STRUCTURE FOR FIXED CYCLES)
(PETER SMID - 07-DEC-2008)

N1 G21
N2 G17 G40 G80 G49
N3 T01
N4 M06
N5 G90 G54 G00 X.. Y.. S.. M03 T02
N6 G43 Z25.0 H01 M08
N7 G99 G82 X.. Y.. R.. Z.. F..
(---- CUTTING MOTIONS WITH TOOL T01 ----)
...
N33 G80 Z25.0 M09
N34 G28 Z25.0 M05
N35 M01

N36 T02
N37 M06
N38 G90 G54 G00 X.. Y.. S.. M03 T03
N39 G43 Z25.0 H02 M08
N40 G99 G81 X.. Y.. R.. Z.. F..
(---- CUTTING MOTIONS WITH TOOL T02 ----)
...
N62 G80 Z25.0 M09
N63 G28 Z25.0 M05
N64 M01

N65 T03
N66 M06
N67 G90 G54 G00 X.. Y.. S.. M03 T01
N68 G43 Z25.0 H03 M08
N69 G99 G84 X.. Y.. R.. Z.. F..
(---- CUTTING MOTIONS WITH TOOL T03 ----)
...
N86 G80 Z25.0 M09
N87 G28 Z25.0 M05
N88 G28 X.. Y..
N89 M30
%
```

```
(BLANK LINE MAY BE NECESSARY - OR % SYMBOL)
(PROGRAM NUMBER AND ID)
(BRIEF PROGRAM DESCRIPTION)
(PROGRAMMER AND DATE OF LAST REVISION)
(BLANK LINE)
(UNITS SETTING IN A SEPARATE BLOCK)
(INITIAL SETTINGS AND CANCELLATIONS)
(TOOL T01 INTO WAITING POSITION)
(T01 INTO SPINDLE)
(T01 RESTART BLOCK - T02 INTO WAITING POSITION)
(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
(FIXED CYCLE - G82 USED AS AN EXAMPLE)

(CYCLE CANCEL - CLEAR ABOVE PART - COOLANT OFF)
(HOME IN Z ONLY-SPINDLE OFF)
(OPTIONAL STOP)
(— BLANK LINE —)
(TOOL T02 INTO WAITING POSITION - CHECK ONLY)
(T02 INTO SPINDLE)
(T02 RESTART BLOCK - T03 INTO WAITING POSITION)
(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
(FIXED CYCLE - G81 USED AS AN EXAMPLE)

(CYCLE CANCEL - CLEAR ABOVE PART - COOLANT OFF)
(HOME IN Z ONLY - SPINDLE OFF)
(OPTIONAL STOP)
(— BLANK LINE —)
(TOOL T03 INTO WAITING POSITION - CHECK ONLY)
(T03 INTO SPINDLE)
(T03 RESTART BLOCK - T01 INTO WAITING POSITION)
(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
(FIXED CYCLE - G84 USED AS AN EXAMPLE)

(CYCLE CANCEL - CLEAR ABOVE PART - COOLANT OFF)
(HOME IN Z ONLY - SPINDLE OFF)
(HOME IN XY ONLY - MAY BE OMITTED)
(END OF PROGRAM)
(STOP CODE - END OF FILE TRANSFER)
```

O0702 (ID MAX 15 CHARS)	(BLANK LINE MAY BE NECESSARY - OR % SYMBOL)
(SAMPLE PROGRAM STRUCTURE FOR MILLING)	(PROGRAM NUMBER AND ID)
(PETER SMID - 07-DEC-2008)	(BRIEF PROGRAM DESCRIPTION)
	(PROGRAMMER AND DATE OF LAST REVISION)
	(BLANK LINE)
N1 G21	(UNITS SETTING IN A SEPARATE BLOCK)
N2 G17 G40 G80 G49	(INITIAL SETTINGS AND CANCELLATIONS)
N3 T01	(TOOL T01 INTO WAITING POSITION)
N4 M06	(T01 INTO SPINDLE)
N5 G90 G54 G00 X.. Y.. S.. M03 T02	(T01 RESTART BLOCK - T02 INTO WAITING POSITION)
N6 G43 Z25.0 H01 M08	(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
N7 G01 Z.. F..	(FEED TO Z CLEARANCE OR DEPTH)
(---- CUTTING MOTIONS WITH TOOL T01 ----)	
...	
N33 G00 Z25.0 M09	(CLEAR ABOVE PART - COOLANT OFF)
N34 G28 Z25.0 M05	(HOME IN Z ONLY-SPINDLE OFF)
N35 M01	(OPTIONAL STOP)
	(— BLANK LINE —)
N36 T02	(TOOL T02 INTO WAITING POSITION - CHECK ONLY)
N37 M06	(T02 INTO SPINDLE)
N38 G90 G54 G00 X.. Y.. S.. M03 T03	(T02 RESTART BLOCK - T03 INTO WAITING POSITION)
N39 G43 Z25.0 H02 M08	(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
N40 G01 Z.. F..	(FEED TO Z CLEARANCE OR DEPTH)
(---- CUTTING MOTIONS WITH TOOL T02 ----)	
...	
N62 G00 Z25.0 M09	(CLEAR ABOVE PART - COOLANT OFF)
N63 G28 Z25.0 M05	(HOME IN Z ONLY - SPINDLE OFF)
N64 M01	(OPTIONAL STOP)
	(— BLANK LINE —)
N65 T03	(TOOL T03 INTO WAITING POSITION - CHECK ONLY)
N66 M06	(T03 INTO SPINDLE)
N67 G90 G54 G00 X.. Y.. S.. M03 T01	(T03 RESTART BLOCK - T01 INTO WAITING POSITION)
N68 G43 Z25.0 H03 M08	(TOOL LG OFFSET - CLEAR ABOVE WORK - COOLANT ON)
N69 G01 Z.. F..	(FEED TO Z CLEARANCE OR DEPTH)
(---- CUTTING MOTIONS WITH TOOL T03 ----)	
...	
N86 G00 Z25.0 M09	(CLEAR ABOVE PART - COOLANT OFF)
N87 G28 Z25.0 M05	(HOME IN Z ONLY - SPINDLE OFF)
N88 G28 X.. Y..	(HOME IN XY ONLY - MAY BE OMITTED)
N89 M30	(END OF PROGRAM)
%	(STOP CODE - END OF FILE TRANSFER)

The XY value in the block N88 should be the *current position* of the X and Y axes. If the absolute position is unknown, change the block to the incremental version:

N88 G91 G28 X0 Y0

If a tool has to be repeated, make sure not to include tool change block for the current tool. Many CNC systems will generate an alarm if the tool change command cannot find a particular tool in the magazine. In both preceding examples, tool repeat blocks will be N5, N38 and N67.

The program structure examples are for a vertical CNC machining center with random tool selection mode and typical Fanuc control system - some minor changes are to be expected. Study the *program flow* rather than its exact content. Note the repetitiveness of blocks for each tool and also note a blank line (empty block) between individual tools for easier orientation.

Each program block is identified with a comment as brief explanation. In practice, the two versions shown are combined in a single program. Actual meaning of the content will be explained in subsequent chapters in this handbook.

◆ Program Structure Benefits

Developing a solid program structure is absolutely essential for one simple reason - *it is going to be used all the time*. Practical benefits are numerous - they include ease of reading the program (on screen or on paper), starting with any tool is easier, important default settings are defined *after* tool change (for example, G90, G54, G00). All these (and other) benefits result in fewer errors and increased productivity at the CNC machine.

Well designed program structure
is worth the time spent developing it

PROGRAM HEADER

Examples above showed a typical program structure for milling machines and machining centers. At the beginning of a typical program, there is usually a block or two of comments that provide general information. Various comments and/or messages may be placed in the program, providing they are enclosed in parentheses. This kind of internal documentation is useful to both the programmer and operator.

A special series of comments at the program *top* is defined as the *program header*, where various program features are identified. The next example is a rather much exaggerated sample of items that *may* be used in a program header:

```
(
(FILE NAME ..... 01234.NC)
(LAST VERSION DATE ..... 07-DEC-2008)
(LAST VERSION TIME ..... 19:43)
(PROGRAMMER ..... PETER SMID)
(MACHINE ..... OKK - VMC)
(CONTROL ..... FANUC 30i)
(UNITS ..... METRIC)
(JOB NUMBER ..... 4321)
(OPERATION ..... DRILL-BORE-TAP)
(STOCK MATERIAL ..... H.R.S. PLATE)
(MATERIAL SIZE ..... 200 X 150 X 50)
(PROGRAM ZERO ..... X0 - LEFT EDGE)
(                               Y0 - BOTT EDGE)
(                               Z0 - TOP FACE )
(STATUS ..... NOT VERIFIED)
)
```

◆ Tool Comments

Some CNC programmers like to list all tools at the program beginning, also as comments. For example:

```
(== T01 - 100 MM DIA FACE MILL ==)
(== T02 - 25 MM DIA END MILL - 4 FLUTES ==)
(== T08 - 10 MM SPOT DRILL ==)
(== T15 - 7 MM DRILL - LONG SERIES ==)
```

Within the program, each tool may be identified as well, using the same comment at the beginning of each tool:

```
(== T08 - 10 MM SPOT DRILL ==)
N15 T08
N16 M06
N17 G90 G54 G00 X.. Y.. S.. M03 T15
N18 ...
```

Other comments and messages to the operator can be added to the program as required, such as operation description or special action required.

8

PREPARATORY COMMANDS

The program address G identifies a *preparatory command*, often called a *G-code*. This address has one and only objective - that is to *preset* or to *prepare* the control system to a certain desired *condition*, or to a certain *mode* or a *state* of operation. For example, address G00 presets a rapid motion mode for the machine tool but does not move any axis, address G81 presets the drilling cycle but does not drill any holes, etc. The common term *preparatory command* indicates its meaning - a G-code will *prepare* control unit to accept the programming instructions *following* the G-code, in a specific way.

DESCRIPTION AND PURPOSE

One block example will illustrate the *need for* preparatory commands in the following program entry:

N7 X13.0 Y10.0

Even a casual look at this block shows that the coordinates X13.0Y10.0 relate to the cutting tool *end position*, when block N7 is executed (*i.e.*, processed by the control). Block N7 does not indicate whether the coordinates are in absolute or incremental mode. It does not indicate whether X13.0Y10.0 are in metric or imperial units. Neither it indicates whether the actual motion to this specified target position is a rapid motion or a linear motion. If evaluation of such a block cannot establish the meaning of its contents, control system does not have enough information. The supplied information in such a block is *incomplete*, therefore unusable by itself. Some additional instructions for the coordinates are required that define their full purpose.

For example, in order to make the block N7 a tool motion in *rapid* mode (G00) using *absolute dimensions* (G90), *all* these instructions - or commands - must be specified *before* the block or *within* the block:

➡ Example A :

N7 G90 G00 X13.0 Y10.0

➡ Example B :

N3 G90
N4 ...
N5 ...
N6 ...
N7 G00 X13.0 Y10.0

➡ Example C :

N3 G90 G00
N4 ...
N5 ...
N6 ...
N7 X13.0 Y10.0

➡ Example D :

N2 G90
N3 G00
N4 ...
N5 ...
N6 ...
N7 X13.0 Y10.0

All four examples have the same machining result, providing that there is no change of any G-code mode between blocks N4 and N6 in the examples B, C and D. G-codes that can be written once and stay in effect until canceled or changed are *modal* G-codes, divided into logical groups.

One G-code in a modal group replaces another G-code from the same group

Modal and non-modal G-codes will be described shortly. Each control system has its own list of available G-codes. Many G-codes are very common and can be found on virtually all controls, others are unique to the particular control system, even the machine tool. Because of the nature of machining applications, the list of typical G-codes will be different for milling systems and turning systems. The same applies for other types of machines. Each group of G-codes must be kept separate.

Check machine documentation for available G codes !

APPLICATIONS FOR MILLING

The G-code table on the next page is a considerably detailed list of the most common preparatory commands used for programming CNC milling machines and CNC machining centers. All listed G-codes may not be applicable to a particular machine and control system, so consult your machine and control reference manual to make sure. Some G-codes listed are a special option that must be available for the CNC machine in the control system.

G code	Description
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise (CW)
G03	Circular interpolation counterclockwise (CCW)
G04	Dwell - as a separate block only
G09	Exact stop check - one block only
G10	Programmable data input - Data Setting
G11	Data Setting mode cancel
G15	Polar Coordinate Command cancel
G16	Polar Coordinate Command
G17	XY-plane designation
G18	ZX-plane designation
G19	YZ-plane designation
G20	Imperial units of input
G21	Metric units of input
G22	Stored stroke check ON
G23	Stored stroke check OFF
G25	Spindle speed fluctuation detection ON
G26	Spindle speed fluctuation detection OFF
G27	Machine zero position check
G28	Machine zero return (reference point 1)
G29	Return from machine zero
G30	Machine zero return (reference point 2)
G31	Skip function
G40	Cutter radius compensation cancel
G41	Cutter radius compensation - left
G42	Cutter radius compensation - right
G43	Tool length compensation - positive
G44	Tool length compensation - negative
G45	Position compensation - single increase
G46	Position compensation - single decrease
G47	Position compensation - double increase
G48	Position compensation - double decrease
G49	Tool length offset cancel
G50	Scaling function cancel
G51	Scaling function

G code	Description
G52	Local coordinate system setting
G53	Machine coordinate system
G54	Work coordinate offset 1
G55	Work coordinate offset 2
G56	Work coordinate offset 3
G57	Work coordinate offset 4
G58	Work coordinate offset 5
G59	Work coordinate offset 6
G60	Single direction positioning
G61	Exact stop mode
G62	Automatic corner override mode
G63	Tapping mode
G64	Cutting mode
G65	Custom macro call
G66	Custom macro modal call
G67	Custom macro modal call cancel
G68	Coordinate system rotation
G69	Coordinate system rotation cancel
G73	High speed peck drilling cycle (deep hole)
G74	Left hand threading cycle
G76	Fine boring cycle
G80	Fixed cycle cancel
G81	Drilling cycle
G82	Spot-drilling cycle
G83	Peck-drilling cycle (deep hole drilling cycle)
G84	Right hand threading cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G90	Absolute dimensioning mode
G91	Incremental dimensioning mode
G92	Tool position register
G98	Return to initial level in a fixed cycle
G99	Return to R-level in a fixed cycle

In case of inconsistency between the listed codes in this handbook and the control system manual, always select G-codes listed by the control manufacturer

APPLICATIONS FOR TURNING

Fanuc lathe controls use three G-code group types - A, B and C. *Type A* is the most common; in this handbook, all examples and explanations are *Type A* group, including the table below. Only one type can be set at a time. Types *A* and *B* can be set by control system parameter, but type *C* is optional. Generally, most G-codes are identical, only a few are different in the *A* and *B* type groups. More details on the subject of G-code groups, *see page 55*.

G code	Description
G00	Rapid positioning
G01	Linear interpolation
G02	Circular interpolation clockwise
G03	Circular interpolation counterclockwise
G04	Dwell (as a separate block)
G09	Exact stop check - one block only
G10	Programmable data input (Data Setting)
G11	Data Setting mode cancel
G20	English units of input
G21	Metric units of input
G22	Stored stroke check ON
G23	Stored stroke check OFF
G25	Spindle speed fluctuation detection ON
G26	Spindle speed fluctuation detection OFF
G27	Machine zero position check
G28	Machine zero return (reference point 1)
G29	Return from machine zero
G30	Machine zero return (reference point 2)
G31	Skip function
G32	Threading - constant lead
G35	Circular threading CW
G36	Circular threading CCW
G40	Tool nose radius offset cancel
G41	Tool nose radius offset left
G42	Tool nose radius compensation right

G code	Description
G50	Tool position register / Maximum r/min preset
G52	Local coordinate system setting
G53	Machine coordinate system setting
G54	Work coordinate offset 1
G55	Work coordinate offset 2
G56	Work coordinate offset 3
G57	Work coordinate offset 4
G58	Work coordinate offset 5
G59	Work coordinate offset 6
G61	Exact stop mode
G62	Automatic corner override mode
G64	Cutting mode
G65	Custom macro call
G66	Custom macro modal call
G67	Custom macro modal call cancel
G68	Mirror image for double turrets
G69	Mirror image for double turrets cancel
G70	Profile finishing cycle
G71	Profile roughing cycle - Z-axis direction
G72	Profile roughing cycle - X-axis direction
G73	Pattern repetition cycle
G74	Drilling cycle
G75	Grooving cycle
G76	Threading cycle
G90	Cutting cycle A (Group type A)
G90	Absolute command (Group type B)
G91	Incremental command (Group type B)
G92	Thread cutting cycle (Group type A)
G92	Tool position register (Group type B)
G94	Cutting cycle B (Group type A)
G94	Feedrate per minute (Group type B)
G95	Feedrate per revolution (Group type B)
G96	Constant surface speed mode (CSS)
G97	Direct r/min input (CSS mode cancel)
G98	Feedrate per minute (Group type A)
G99	Feedrate per revolution (Group type A)

Most of the preparatory commands are discussed under their individual applications, for example, G01 will be explained under *Linear Interpolation*, G02 and G03 under *Circular Interpolation*, etc. In this section, G-codes are described in general, regardless of the type of machine or control unit.

G-CODES IN A PROGRAM BLOCK

Unlike miscellaneous functions, known as the *M-functions* and described in the next chapter, several preparatory commands may be used in a single block, providing they are not in a logical conflict with each other:

```
N25 G90 G00 G54 X6.75 Y10.5
```

This method of program writing is several blocks shorter than its single block alternative:

```
N25 G90
N26 G00
N27 G54
N28 X6.75 Y10.5
```

Both methods will be identical during continuing processing. However, the second example, when executed in a *single block* mode, each block will require pressing the *Cycle Start* key to activate each block. The shorter method is more practical, not only for its length, but for the logical connection between individual addresses within the block.

Some rules of application and general considerations apply to G-codes when used with *other* data in a block. The most important of them is the subject of *modality*.

◆ Modality of G-codes

Earlier, the following example *C* was used to demonstrate the general placement of several G-codes into a program block:

➡ Example C - original :

```
N3 G90 G00
N4 ...
N5 ...
N6 ...
N7 X13.0 Y10.0
```

If the structure is changed slightly and filled with realistic data, these five blocks may be the result:

➡ Example C - modified (as programmed) :

```
N3 G90 G00 X5.0 Y3.0
N4 X0
N5 Y20.0
N6 X15.0 Y22.0
N7 X13.0 Y10.0
```

Note the rapid motion command G00 - how many times does it appear in the program? Just *once* - in block N30. In fact, so does the command for absolute mode, G90 - only once. The reason neither G00 nor G90 has been repeated in every block is because both commands remain active from the moment of their first appearance in the program. The term *modal* is used to describe this characteristic.

For a command to be modal, it has to remain in a certain mode until canceled by an equivalent mode

As most (not all) G-codes are modal, there is no need to repeat a modal command in every block. Using the earlier example *C* once more, control system will make the following interpretation during program execution:

➡ Example C - modified (as processed) :

```
N3 G90 G00 X50.0 Y30.0
N4 G90 G00 X0
N5 G90 G00 Y200.0
N6 G90 G00 X150.0 Y220.0
N7 G90 G00 X130.0 Y100.0
```

This program example does not represent any practical application by moving from one location to another at a rapid rate, but it demonstrates the modality of preparatory commands. The main purpose of modal values is to avoid unnecessary duplication of programming modes. G-codes are used so often, that writing them in the program can be tedious. Fortunately, the majority of G-codes can be applied only once, *providing they are modal*. In the control system specifications, preparatory commands are identified as modal and unmodal.

◆ Conflicting Commands in a Block

Preparatory commands are used in the program, to select from two or more possible modes of operation. If the rapid motion command G00 is selected, it is a specific command relating to a tool motion. As it is impossible to have a rapid motion and a cutting motion active *at the same time*, it is impossible to have G00 and G01 active at the same time. Such a combination creates a *conflict* in a block. If conflicting G-codes are used in the same block, the *latter* G-code will be used by the control (system error is also possible).

```
N74 G01 G00 X3.5 Y6.125 F20.0
```

In the example, the two commands G01 and G00 are in conflict. As G00 is the *latter* one in the block, it will become effective. The feedrate is ignored in this case.

```
N74 G00 G01 X3.5 Y6.125 F20.0
```

This is the exact opposite of the previous example. Here, the G00 is first, therefore G01 command will take precedence and the motion will take place as a cutting motion at the specified feedrate of 20.0 in/min.

◆ Word Order in a Block

G-codes are normally programmed *at the beginning of a block*, after the block number, before other significant data:

```
N40 G91 G01 Z-0.625 F8.5
```

This is a traditional order, based on the idea that if the purpose of G-codes is to *prepare* or *preset* control system to a certain condition, the preparatory commands should always be placed first. Supporting this argument is the fact that only non-conflicting codes are allowed in a single block. Strictly speaking, there is nothing wrong with rearranging the order to:

```
N40 G91 Z-0.625 F8.5 G01
```

Perhaps unusual, but quite correct. That is not the case with the next method of positioning a G-code in a block:

```
N40 Z-0.625 F8.5 G01 G91
```

Watch for situations like this! What happens in this case is that the cutting motion G01, feedrate F and depth Z will be combined and executed *using the current dimensional mode*. If the current mode is absolute, Z-axis motion will be executed as an absolute value, *not* an incremental value. The reason for this exception is that Fanuc allows to mix dimensional values in the *same* block. That can be a very useful feature, if used carefully. A typical *correct* application of this feature can be illustrated in this example:

```
(G20)
N45 G90 G00 G54 X1.0 Y1.0 S1500 M03          (G90)
N46 G43 Z0.1 H02
N47 G01 Z-0.25 F5.0
N48 X2.5 G91 Y1.5          (G90 MIXED WITH G91)
N49 ...
...
```

Blocks N45 through N47 are all in absolute mode. Before the block N48 is executed, absolute position of the axes X and Y is 1.0,1.0. From this starting position, the target location is the absolute position of X2.5 combined with the incremental motion of 1.5 inches along the Y-axis. The resulting absolute position will be X2.5Y2.5, making a 45° motion. In this case, G91 will remain in effect for *all* subsequent blocks, until G90 is programmed. Most likely, the block N48 will be written in absolute mode (still active):

```
...
N48 X2.5 Y2.5
...
```

Normally, there is no reason to switch between the two modes in the same block. It can result in some very unpleasant surprises, and some controls may not support this method. Yet, there are some occasions when this special technique brings benefits, for example, in subprograms.

GROUPING OF COMMANDS

Earlier example of conflicting G-codes in a single block brings one issue to the forefront. It makes sense, for example, that motion commands such as G00, G01, G02 and G03 cannot co-exist in the same block. Such distinction is not so clear for other preparatory commands. For example, can the tool length offset command G43 be programmed in the same block as the cutter radius offset command G41 or G42 - even in theory only? The answer is yes, but let's look at the reason why.

Fanuc control system recognizes preparatory commands by separating them into arbitrary *groups*. Each group, called the *G-code group*, has a Fanuc assigned arbitrary two-digit number. The main rule governing the co-existence of G-codes in a single block is very simple. If two or more G-codes from the *same group* are programmed in the same block, they are in conflict with each other.

◆ Group Numbers

G-code groups are typically numbered from 00 to 25. This range varies between different control models, depending on the features. It can even be higher for the newest controls or where more G-codes are offered by the machine control. One of these groups - the most unique one and perhaps the most important as well - is the *Group 00*.

All preparatory commands in Group 00 are *not* modal, sometimes using the descriptions *unmodal* or *non-modal*. They are only active in the block in which they were programmed. If unmodal G-codes are to be effective in several consecutive blocks, they *must* be programmed in *each* of those blocks. For majority of unmodal commands, this repetition will not be used very often.

For example, a dwell is a programmed pause measured in milliseconds. It is needed only for the duration within specified time, no longer. There is no logical need to program dwell in two or more consecutive blocks. After all, what is the benefit of the next three blocks?

```
N56 G04 P2000
N57 G04 P3000
N58 G04 P1000
```

All three blocks contain the same function - a dwell - one after another. The part program can be made much more efficient by simply entering the total dwell time into a single block with identical results:

```
N56 G04 P6000
```

The following groups are typical for Fanuc control systems. Applications for milling and turning controls are specially distinguished by the *M* and *T* letters respectively, in the *Type* column of the table:

Group	Description	G codes	Type
00	Unmodal G codes	G04 G09 G10 G11 G27 G28 G29 G30 G31 G37 G45 G46 G47 G48 G52 G53 G65 G51 G60 G92 G50 G70 G71 G72 G73 G74 G75 G76	M/T M/T M/T M/T M T T T
01	Motion Commands, Cutting Cycles	G00 G01 G02 G03 G32 G35 G36 G90 G92 G94	M/T T T
02	Plane Selection	G17 G18 G19	M
03	Dimensioning Mode	G90 G91 (U and W for lathes)	M T
04	Stored Strokes	G22 G23	M/T
05	Feedrate	G93 G94 G95	T
06	Units Input	G20 G21	M/T
07	Cutter Radius Offset	G40 G41 G42	M/T
08	Tool Length Offset	G43 G44 G49	M
09	Cycles	G73 G74 G76 G80 G81 G82 G83 G84 G85 G86 G87 G88 G89	M M M M
10	Return Mode	G98 G99	M
11	Scaling Cancel, Mirror Image	G50 G68 G69	M T
12	Coordinate System	G54 G55 G56 G57 G58 G59	M/T M/T
13	Cutting Modes	G61 G62 G64 G63	M/T M
14	Macro Mode	G66 G67	M/T
16	Coordinate Rotation	G68 G69	M
17	CSS	G96 G97	T
18	Polar Input	G15 G16	M
24	Spindle Speed Fluctuation	G25 G26	M/T

The group relationship makes a perfect sense in all cases. One possible exception is Group 01 for *Motion Commands* and Group 09 for *Cycles*. The relationship between these two groups is this - if a G-code from Group 01 is specified in any of the fixed cycle Group 09, the cycle is immediately canceled, *but the opposite is not true*. In other words, an active motion command is *not* canceled by a fixed cycle.

Group 01 is not affected by G-codes from Group 09. In a summary ...

Any G-code from a given group automatically replaces another G-code from the same group

G-CODE TYPES

Fanuc control system offers a flexible selection of preparatory commands. This fact distinguishes Fanuc from many other controls. Considering the fact that Fanuc controls are used worldwide, it only makes sense to allow the standard control configuration to follow established style of each country. A typical example is the selection of dimensional units. In Europe, Japan and many other countries, metric system is the standard. In North America, the common system of dimensioning still uses imperial units. As both markets are substantial in the world trade, a clever control manufacturer tries to reach them both. Almost all control manufacturers offer a selection of the dimensional system. But Fanuc and similar controls also offer selection of programming codes that were in effect *before* Fanuc reached the worldwide market.

The method Fanuc controls use is a simple method of parameter setting. By selecting a specific system parameter, one of two or three G-code types can be selected, the one that is typical for a particular geographical user. Although the majority of G-codes are the same for every type, the most typical illustration are G-codes used for imperial and metric selection of units. Many earlier US controls used G70 for imperial units and G71 for metric units. Fanuc system has traditionally used G20 and G21 codes for imperial and metric input respectively.

Setting up a parameter, the G-code type that is the most practical can be selected. Such a practice, if done at all, should be done only once and only when the control is installed, before any part programs have been written for it. Change of the G-code type at random is a guaranteed way to create organizational nightmare. Keep in mind that a change of one code meaning will affect the meaning of another code. Using the units example for a lathe, if G70 means an imperial input of dimensions, you cannot use it to program a roughing cycle. Fanuc provides a different code in that group. Always stay with the standard G-code type. All G-codes in this handbook use the default group of the *Type A*, and also the most common group.

◆ G-Codes and Decimal Point

Many latest Fanuc controls include a G code with a decimal point, for example, G72.1 (Rotation copy) or G72.2 (Parallel copy). Several preparatory commands in this group are related to a particular machine tool or are not typical enough to be described in this handbook.

9

MISCELLANEOUS FUNCTIONS

The address *M* in a CNC program identifies a *miscellaneous function*, sometimes called a *machine function*. Not all miscellaneous functions are related to the operation of a CNC machine - quite a few are related to the processing of part program. The more suitable term *miscellaneous functions* is used throughout this handbook.

DESCRIPTION AND PURPOSE

Within the structure of a CNC program, programmers often need some means of activating certain aspects of machine operation or controlling the program flow. Without availability of such means, any part program would be incomplete and impossible to run. First, let's look at those miscellaneous functions that relate to the operation of a CNC machine - the true *machine functions*.

◆ Machine Related Functions

Various physical operations of a CNC machine must be controlled by the program, to ensure fully automated machining. These functions generally use the M-address and include the following operations:

- | | |
|-----------------------------|---------------------|
| ■ Spindle rotation | CW or CCW |
| ■ Gear range change | Low / Medium / High |
| ■ Automatic tool change | ATC |
| ■ Automatic pallet change | APC |
| ■ Coolant operation | ON or OFF |
| ■ Tailstock or quill motion | IN or OUT |

These operations vary between machines, due to the different designs by various machine manufacturers. A machine design, from the engineering point of view, is based on a certain primary machining application. A CNC milling machine will require different functions related to this type of machine than a CNC machining center or a CNC lathe. A numerically controlled EDM wire cutting machine will have many special functions, typical to that kind of machining, including those not found on other machines.

Even two machines designed for the *same type* of work, for example, two kinds of CNC vertical machining centers, will have functions different from each other, if they have a different control system or significantly different options. Various machine models from the same manufacturer will also have certain unique functions, even if the CNC systems are identical.

All machine tools designed for metal removal by cutting have certain common features and capabilities. For example, spindle rotation can have three - *and only three* - possible selections in a program:

- Spindle normal rotation
- Spindle reverse rotation
- Spindle stop

In addition to these three possibilities, there is a function called the spindle orientation, also a machine related function. Another example is coolant. Coolant can only be controlled in a program as being either ON or OFF.

These operations are typical to most CNC machines. All are programmed with an M-function, typically followed by two digits, although some control models allow the use of a three digit M-function, Fanuc 16/18, for example.

Fanuc also uses three digit M-functions in several special applications, for example, for synchronization of two independent turrets on a multi-axis lathe. All these and other functions are related to the operation of CNC machines and belong to the group collectively known as *miscellaneous functions* or simply as the *M-functions* or *M-codes*.

◆ Program Related Functions

In addition to the machine functions, several M-functions are used to control the execution of a CNC program. An interruption of a program execution requires an M-function, for example, during the change of a job setup, such as part reversal. Another example is a situation where one program calls one or more subprograms. In such a case, each program has to have a program call function, the number of repetitions, etc. M-functions handle these requirements.

Based on previous examples, using miscellaneous functions falls into two main groups, based on a particular application:

- Control of machine functions
- Control of program execution

This handbook covers only the most common miscellaneous functions, used by the majority of controls. Unfortunately, there are many functions that vary between machines and control systems. These functions are called *machine specific functions*. For this reason, always consult documentation for the particular machine model and its control system.

TYPICAL APPLICATIONS

Before learning any M-functions, note the type of activity these functions do, regardless of whether such activity relates to the machine or the program. Also note the abundance of two way toggle modes, such as *ON* and *OFF*, *IN* and *OUT*, *Forward* and *Backward*, etc. Always check your manual first - for reasons of consistency, all M-functions in this handbook are based on the following table:

◆ Applications for Milling

Function	Description
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset, no rewind)
M03	Spindle rotation normal (CW for R/H tools)
M04	Spindle rotation reverse (CCW for R/H tools)
M05	Spindle stop
M06	Automatic tool change (ATC)
M07	Coolant mist ON <i>(usually an option)</i>
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)
M19	Spindle orientation
M30	Program end (always with reset and rewind)
M48	Feedrate override cancel OFF <i>(deactivated)</i>
M49	Feedrate override cancel ON <i>(activated)</i>
M60	Automatic pallet change (APC)
M78	B axis clamp <i>(nonstandard)</i>
M79	B axis unclamp <i>(nonstandard)</i>
M98	Subprogram call
M99	Subprogram end

◆ Applications for Turning

Function	Description
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset, no rewind)
M03	Spindle rotation normal (CW for R/H tools)

M04	Spindle rotation reverse (CCW for R/H tools)
M05	Spindle stop
M07	Coolant mist ON <i>(usually an option)</i>
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)
M10	Chuck open
M11	Chuck close
M12	Tailstock quill IN
M13	Tailstock quill OUT
M17	Turret indexing forward
M18	Turret indexing reverse
M19	Spindle orientation (optional)
M21	Tailstock forward
M22	Tailstock backward
M23	Thread gradual pull-out ON
M24	Thread gradual pull-out OFF
M30	Program end (always with reset and rewind)
M41	Low gear selection
M42	Medium gear selection 1
M43	Medium gear selection 2
M44	High gear selection
M48	Feedrate override cancel OFF <i>(deactivated)</i>
M49	Feedrate override cancel ON <i>(activated)</i>
M98	Subprogram call
M99	Subprogram end

◆ Special MDI Functions

Several M-functions cannot be used in a CNC program at all. This group is used in so called *Manual Data Input* mode exclusively (MDI). An example of such functions is a step by step tool change for machining centers, used for service purposes only, never in the program. These functions are outside the scope of this handbook.

◆ Application Groups

The two major categories, described earlier, can further be divided into several groups, based on the specific application of the miscellaneous functions within each group. A typical distribution list is contained in the following table:

Group	Typical M-functions
Program	M00 M01 M02 M30
Spindle	M03 M04 M05 M19
Tool change	M06
Coolant	M07 M08 M09
Accessories	M10 M11 M12 M13 M17 M18 M21 M22 M78 M79
Threading	M23 M24
Gear ranges	M41 M42 M43 M44
Feedrate override	M48 M49
Subprograms	M98 M99
Pallets	M60

This table does not cover all M-functions or even all possible groups. Neither does it distinguish between machines. On the other hand, it does indicate the *types* of applications miscellaneous functions are used for in everyday CNC programming.

Miscellaneous functions listed in this chapter are used throughout the book. Some of them appear more often than others, reflecting their general use in programming. Functions that do not correspond to a particular machine control system are either not used or not needed. However, the concepts for their applications are always similar for most control systems and CNC machines.

In this chapter, only the more general functions are covered in significant detail. Remaining miscellaneous functions are described in the sections covering individual specific applications. At this stage, the stress is on usage and behavior of the most common miscellaneous functions.

M-FUNCTIONS IN A BLOCK

If a miscellaneous function is programmed in a block by itself, with no other data supplementing it, only the function itself will be executed. For example,

N45 M01

is an *optional stop*. This block is correct - an M-function *can* be the only block entry. Unlike the preparatory commands (G codes), only *one* M-function is allowed in a block - unless the control allows multiple M-functions in the same block (some latest controls only), a program error will occur.

More practical method of programming certain miscellaneous functions is in a block that contains a *tool motion*. For example, turning the coolant on and - *at the same time* - moving the cutting tool to a certain part location may be required. As there is no conflict between these instructions, the actual program block may look something like this:

N56 G00 X252.95 Y116.47 M08

In this example of block N56, the precise *time* the M08 function will be activated is not very important. In other cases, the timing may be very important. Some M-functions *must* be in effect before or after certain action takes place. For example, look at this combination - a Z-axis motion is applied together with *program stop* function M00 in the same block:

N319 G01 Z-62.5 F200.0 M00

This is a far more serious situation and two answers are needed. One is *what exactly* will happen, the other is *when exactly* it will happen, when the M00 function is activated. There are three possibilities and three questions to ask:

1. Will the program stop take place immediately, when the motion is activated - at the start of block?
2. Will the program stop take place while the tool is on the way - during a motion?
3. Will the program stop take place when the motion command is completed - at the end of block?

One of the three options *will* happen - but which one? Even if a practical purpose of these examples may not be apparent at this stage, it is useful to know how the CNC system interprets blocks containing tool motion combined with a miscellaneous function.

Each M-function is logically designed - it also shows a great deal of *common sense*.

The actual startup of any M-function should be divided into *two* groups - not three, as the questions might suggest:

- M-function activates at the start of block (simultaneously with the tool motion)
- M-function activates at the end of block (when the tool motion has been completed)

No M-function will be activated *during* block execution, there is no logic to it. What is the logical startup of coolant ON function M08 in block N56 above? The correct answer is that the coolant will be activated at the *same time* as the tool motion *begins*. The correct answer for example block N319 is that the M00 program stop function will be activated *after* the tool motion has been completed. Makes sense? Yes, but what about the other functions, how do they behave in a block with motion?

Let's look at them next.

◆ Startup of M-Functions

Take a look at the list of typical M-functions. Add a tool motion to each and try to determine how the function is going to behave, based on the previous notes. A bit of logical thinking provides a good chance to arrive at the right conclusion. Compare the two following groups to confirm:

M-functions activated at the START OF A BLOCK	
M03	Spindle rotation normal (CW for R/H tools)
M04	Spindle rotation reverse (CCW for R/H tools)
M06	Automatic tool change (ATC)
M07	Coolant mist ON <i>(usually an option)</i>
M08	Coolant ON (coolant pump motor ON)

M-functions activated at the END OF A BLOCK	
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset, no rewind)
M05	Spindle stop
M09	Coolant OFF (coolant pump motor OFF)
M30	Program end (always with reset and rewind)
M60	Automatic pallet change (APC)

If there is any uncertainty about how an M-function will interact with tool motion, the safest choice is to program the M-function as a *separate block*. That way, it will always be processed *before* or *after* the relevant program block. For majority of applications this is a safe solution.

◆ Duration of M-Functions

Knowledge of *when* an M-function takes effect is logically followed by another question - *how long* such a function will be active. Some miscellaneous functions are active only in the block they appear. Others will continue to be in effect until canceled (replaced) by another miscellaneous function. This is similar to the modality of preparatory commands (G-codes), however the word *modal* is not usually used with M-functions. As an example of a function duration, take miscellaneous functions M00 or M01. Either one will be active for *one* block only. The coolant ON function M08, will be active until a *canceled* or an *altering* function is programmed. Keep in mind that any one of the following functions will also cancel coolant ON mode - M00, M01, M02, M09 and M30. Compare these two tables:

M-functions completed in ONE BLOCK	
M00	Compulsory program stop
M01	Optional program stop
M02	End of program (usually with reset, no rewind)
M06	Automatic tool change (ATC)
M30	Program end (always with reset and rewind)
M60	Automatic pallet change (APC)

M-functions active UNTIL CANCELED or ALTERED	
M03	Spindle rotation normal (CW for R/H tools)
M04	Spindle rotation reverse (CCW for R/H tools)
M05	Spindle stop
M07	Coolant mist ON <i>(usually an option)</i>
M08	Coolant ON (coolant pump motor ON)
M09	Coolant OFF (coolant pump motor OFF)

Above classification is logical and shows good design and common sense. There is no need to remember each M-function and its exact activities. The best place to find out for certain, is to study manuals supplied with the CNC machine and watch the program run right on the machine.

PROGRAM FUNCTIONS

Miscellaneous functions that control program processing can be used either to interrupt the processing temporarily (in the middle of a program) or permanently (at the end of a program). Several functions are available for this purpose.

◆ Program Stop

M00 function is defined as an *unconditional* or *compulsory* program stop. Any time CNC system encounters this function during program processing, all automatic operations of the machine tool will stop:

- Motion of all axes
- Rotation of the spindle
- Coolant function
- Further program execution

Control settings will *not* be reset when M00-function is processed. All significant program data currently active are still retained (feedrate, coordinate setting, spindle speed, etc.). Program processing can only be resumed by activating the *Cycle Start* key. M00 function cancels both spindle rotation and coolant function - either one or both have to be reprogrammed in subsequent blocks.

Miscellaneous function M00 can be programmed as an individual block or in a block containing other commands, usually axis motion. If M00 is programmed together with a motion command, the motion will be completed first, *then* the program stop will become effective:

➡ M00 programmed **after a motion command** :

```
N38 G00 X189.5
N39 M00
```

➡ M00 programmed **with a motion command** :

```
N39 G00 X189.5 M00
```

In both cases, any motion command will be completed first, *before* the program stop is executed. Actual difference between the two examples is apparent only in single block processing mode (for example, during a trial cut). There will be no practical difference in auto mode of processing (*Single Block* switch set to OFF position).

Practical Usage

Program stop function used in a program makes the CNC operator's job much easier. It is useful for many jobs. One common use is a part inspection at the machine, while the part is still mounted. In program stop mode, the part dimensions or tool condition can be checked. Chips accumulated in a bored or drilled hole can be removed, for example, before another machining operation can start. Blind hole tapping is a good example. Program stop function is also necessary to change the current setup before the program is completed, for example, to reverse a part. Any manual tool change also requires M00 function in the program.

Program stop function M00 is used only for a manual intervention during program processing

All control systems also offer an *optional* program stop M01, described next. The main rule of using M00 is the need of a *manual intervention* for *every* part machined. Manual tool change in a program qualifies for M00, because every part needs it. A dimensional check may not qualify, if it is infrequent. In this case, M01 function will be a better choice. Although the difference between both functions is slight, the actual difference in cycle time can be significant for large number of parts.

When using M00 in a program, always inform the operator *why* the function has been used and what its purpose is. Make your intent known to avoid a confusion. This intent can be made available to the operator in two ways:

- In setup sheet, refer to the block number that contains miscellaneous function M00 and describe any manual operation that has to be performed:

BLOCK N39 REMOVE CHIPS

- In the program itself, issue a comment section with the necessary information. Comment section must be enclosed in parentheses (three versions shown):

[A] N39 M00 (REMOVE CHIPS)

[B] N39 X189.5 M00 (REMOVE CHIPS)

[C] N39 X189.5 M00
(REMOVE CHIPS)

Any one of these methods will give the CNC operator all necessary information. From all options, the comment section [A] or [B] in a program is preferable. The built-in instructions can be read at the control panel display screen.

◆ Optional Program Stop

Miscellaneous function M01 is an *optional, conditional* program stop. It is similar to M00 function, with one difference. Unlike M00 function, when M01 function is read by the control, program processing will *not* stop without operator's interference. The *Optional Stop* toggle switch or a button key located on the operation panel can be set to either ON or OFF position. When M01 function in the program is processed, current switch setting will determine whether the program will temporarily stop or processing continues without interruption:

Optional Stop switch setting	Result of M01
ON	Processing will stop
OFF	Processing will not stop

In case there is no M01 function programmed, the setting of the *Optional Stop* switch is not important. Normally, it should be set to OFF position for production work.

When active, M01 function behaves exactly as the M00 function. Motion of all axes, spindle rotation, coolant function and any further program execution will be temporarily interrupted. Feedrate, coordinate settings, spindle speed setting, etc., are retained. Further processing of the program can only be resumed by pressing the *Cycle Start* key. All programming rules for M00 function also apply to M01 function.

A good idea is to program M01 as the only entry in the last block of each tool, followed by a blank line with no data. If there is no need for program interruption, the *Optional Stop* switch will be set to OFF position and no production time is lost. If there is a need to stop processing at the end of a tool, the switch will be set to ON position and program processing stops when M01 is processed. Any time loss is usually justified under the circumstances, for example, to change a cutting insert or to inspect important dimensions or surface finish quality.

◆ Program End

Every program must include a special function defining the *end* of active program. For this purpose, there are two M-functions available - the M02 and M30. Both are similar, but each has a distinct purpose. M02 function will terminate the program, but *will cause no return* to the first block at the program top. Function M30 will terminate the program as well but it *will cause a return* to the program top. The word '*return*' is often replaced by the word '*rewind*'. It is a leftover from the times when a reel-to-reel tape reader was common on NC machines. The tape had to be *rewound* when the program has been completed for each part. M30 function provided this *rewind* capability.

When control system reads the program end function M02 or M30, it cancels all axis motions, spindle rotation, coolant function and usually resets the system to the default conditions. *On some controls the reset may not be automatic and any programmer should be aware of it.*

If the program ends with M02 function (usually old programs only), the control remains at the program end, ready for the next *Cycle Start*. On modern CNC equipment there is no need for M02 at all, except for backward compatibility. This function was used in addition to M30 for those machines (mainly NC lathes) that had tape readers without reels, using a short loop tape. The tape trailer was spliced to the tape leader, creating a closed loop. When the program was finished, the start of tape was next to its end, so no rewind was necessary. Long tapes could not use loops and required reels and M30. So much for the history of M02 - just ignore its existence.

Is M02 the Same as M30 ?

On most modern controls, a system parameter can be set to make the M02 function with the same meaning as that of M30. This setting can give it the rewind capabilities, useful in situations where an old program can be used on a machine with a new control without changes.

In a summary, if the end of program is terminated by M30 function, the rewind *will* be performed; if the M02 function is used, the rewind *will not* be performed.

When writing a program, make sure the last block in the program contains nothing else but M30 as the preferred end (sequence block *is* allowed to start the block):

```
N65 . . .
N66 G91 G28 X0 Y0
N67 M30                                (END OF PROGRAM)
%
```

On some controls, the M30 function can be used together with the axes motion - *definitely NOT recommended !*:

```
N65 . . .
N66 G91 G28 X0 Y0 M30                (END OF PROGRAM)
%
```

Percent Sign

The percent sign (%) after M30 is a special *stop code*. This symbol terminates the loading of a program from an external device. It is also called the *end-of-file marker*.

◆ Subprogram End

The last M-function for a program end is M99. Its primary usage is in subprograms. Typically, the M99 function will terminate a subprogram and return to the processing of the previous program. If M99 is used in a standard program, it creates a program with *no end* - such a situation is often called an *endless program loop*. M99 should be used only in subprograms, not in the standard programs.

MACHINE FUNCTIONS

Miscellaneous functions relating to the operation of machine tool are part of another group. This section describes the most important of them in detail.

◆ Coolant Functions

Many metal removal operations require that the cutting tool is flooded with a suitable coolant. In order to control the flow of coolant in the program, there are three miscellaneous functions usually provided for this purpose:

M07	Mist ON
M08	Flood ON
M09	Mist or Flood OFF

Mist is the combination of a small amount of cutting oil mixed with compressed air. It depends on the machine tool manufacturer whether this function is standard for a particular CNC machine tool or not. Some manufacturers replace the mixture of oil and air with air only, or with oil shot only, etc. In these cases, it is typical that an additional equipment is built into the machine. If this option exists on the machine, the most common miscellaneous function to activate the oil mist or air is M07.

Function similar to M07 is M08, coolant *flood*. This is by far the most common method in CNC programming. It is standard for virtually all CNC machines. Coolant, usually a suitable mixture of soluble oil and water, is premixed and stored in the machine coolant tank. Some machines also need an M-function for coolant *through the spindle*. Flooding the tool cutting edge is important for three reasons:

- Heat dissipation
- Chip removal
- Lubrication

Primary reason to use coolant flood aimed at the cutting edge is to dissipate the heat generated there during cutting. The secondary reason is to remove chips from the cutting area, using coolant pressure. Finally, the coolant also acts as a lubricant to ease the friction between cutting tool and material. Lubrication helps to extend tool life and improves surface finish.

During initial tool approach towards the part or during final return to the tool change position, coolant is normally not required. To turn off the coolant function, use the M09 function - *coolant off*. M09 will turn off supply from coolant tank and nothing else. In reality, the M09 function will shut off the coolant pump motor.

Each coolant related function may be programmed in separate blocks or together with an axis motion. There are subtle but important differences in the order and timing of program processing. Following examples explain the main differences:

➡ Example A - oil mist is turned ON, if available :

N110 M07

➡ Example B - coolant is turned ON :

N340 M08

➡ Example C - coolant is turned OFF :

N500 M09

➡ Example D - axis motion and coolant ON :

N230 G00 X11.5 Y10.0 M08

➡ Example E - axis motion and coolant OFF :

N400 G00 Z1.0 M09

All examples show the differences in program processing. The general rules of coolant programming are:

- Coolant ON or OFF in a *separate* block becomes active in the block in which it is programmed (Examples A, B and C)
- Coolant ON, when programmed *with axes motion*, becomes active simultaneously with the axes motion (Example D)
- Coolant OFF, programmed *with axes motion*, becomes effective only upon completion of the axes motion (Example E)

The main purpose of M08 function is *to turn the coolant pump motor on*. It does not guarantee that the cutting edge receives any coolant immediately. On large machines with long coolant pipes, or machines with low coolant pump pressure, some delay is to be expected before coolant covers the distance between pump and cutting tool.

Coolant should always be programmed with two important considerations in mind:

- There will be no coolant splashing outside of the work area (outside of the machine)
- There will never be a situation when the coolant reaches a hot edge of the tool

The first consideration is relatively minor. If the coolant function is programmed in the 'wrong' place, the result may be just an inconvenience. Wet area around the machine may present unsafe working conditions and should be quickly corrected. Even more serious situation happens when the coolant suddenly starts flooding a cutting tool that has already entered the material. The change in temperature at the cutting edge may cause the tool to break and damage the part. Carbide tools are far more easily affected by temperature changes than high speed steel tools. Such a possibility can be prevented during programming, by using the M08 function *a few blocks ahead* of the actual cutting block. Long pipes or insufficient coolant pressure on the machine may delay the start of the actual flooding.

◆ Spindle Functions

Beginning on *page 81*, details of all aspects of controlling the machine spindle in a CNC program are covered. Miscellaneous functions that are available for spindle control are *spindle rotation* and *spindle orientation*.

Most machine spindles can rotate in both directions, *clockwise* (CW) and *counterclockwise* (CCW). The direction of rotation is always relative to a standard point of view. The viewpoint is established from the spindle side as the *direction along the spindle center line towards its face*. CW rotation in such a view is programmed as M03, CCW direction as M04, assuming the spindle can be rotated either way.

Both drilling and milling types of machines use this established convention quite commonly. The same convention is also applied to CNC lathes. On a CNC milling machine or a machining center, it is more practical to look towards the part from the spindle side rather than from the table side. On a lathe (slant bed horizontal type), the more practical view is *from the tailstock towards the spindle*, because that is the closest to how the CNC machine operator stands in front of the lathe. However, M03 and M04 spindle directions are established the same way as for machining centers. A further complication is the fact that left hand tools are used in lathe work more frequently than in the work for milling applications. Make an effort to study the instruction manual for a specific machine carefully - also see related details described on *page 82*.

Spindle function to program a spindle stop is M05. This function will stop the spindle from rotating, *regardless of the rotation direction*. On many machines, the miscellaneous function M05 must also be programmed *before* reversing the spindle rotation:

```

M03                                (SPINDLE CW)
...
<... Machining at the current location ...>
...
M05                                (SPINDLE STOP)
<... Usually a tool change ... >
M04                                (SPINDLE CCW)
...
<... Machining at the current location ...>
...

```

M05 function may also be required when changing gear ranges on CNC lathes (if available). A spindle stop programmed in a block containing an axis motion, will take place *after* the motion has been completed.

The last spindle control function is function M19, called the *spindle orientation*. Some control manufacturers call it the spindle key lock function. Regardless of the description, M19 function will cause the spindle to stop in an oriented position. This function is used mostly during machine setup, seldom in the program itself. Machine spindle must be oriented in two main situations:

- Automatic tool change (ATC)
- Tool shift during a boring operation (G76 and G87 boring cycles only)

When *Automatic Tool Change* (ATC) function M06 is used in the program, there is no need to program spindle orientation for the vast majority of CNC machining centers. This orientation is built into the automatic tool changing sequence and guarantees the correct positioning of all cutting tool holders. Some programmers do like to program M19 with machine zero return motion for the tool change position, to save a second or two of the cycle time.

Spindle orientation is necessary for certain boring operations on milling systems. To exit a bored hole with a boring tool away from the finished cylindrical wall, the spindle must be stopped first, the tool cutting bit must be oriented, and then the tool can be retracted from the hole. A similar approach is used for backboring operations. However, these special cutting operations use fixed cycles in the program, where the spindle orientation is built in. For more details, see pages 84 and 210.

In conclusion, M19 miscellaneous function is rarely used in the program. It is available as a programming aid and to the CNC operator for setup work, using MDI operations.

◆ Gear Range Selection

Virtually all programmable gear range selections apply to CNC lathes. On machining centers, spindle gear range is normally changed automatically. Most CNC lathes have two or more gear ranges available, some more powerful lathes are equipped with up to four selections. The basic programming rule is to select the gear range based on machining applications.

For example, most roughing operations require the *power* of the spindle more than the spindle *speed*. In this case, a low range is usually a better selection. For finishing work, a medium or high range is better, because high spindle rotation can be more beneficial to metal removing process.

Distribution of gear related miscellaneous functions depends entirely on the number of gear ranges the CNC lathe has available. Number of ranges is 1, 2, 3 or 4. The following table shows typical distribution of M-functions, but still check the actual commands in your machine tool manual.

Ranges	M function	Gear Range
1 available	N/A	None programmed
2 available	M41 M42	Low range High range
3 available	M41 M42 M43	Low range Medium range High range
4 available	M41 M42 M43 M44	Low range Medium range 1 Medium range 2 High range

Simple rule of thumb is that the higher the gear range, the more spindle speed is possible and less spindle power is required. The opposite is also true. Normally, the spindle rotation does not have to be stopped to change a gear, but consult the lathe manual anyway. If in doubt, stop the spindle first, change the gear range, then restart the spindle.

◆ Machine Accessories

The majority of miscellaneous functions is used for some physical operation of machine tool accessories. From this group, the more common applications have been already covered, specifically coolant control and gear changes. The remaining M-functions in this group are described in detail elsewhere in this handbook, so only a short description is offered here.

The most notable of machine related M-functions are:

M function	Description	Type
M06	Automatic tool change (ATC)	M
M60	Automatic pallet change (APC)	M
M23 M24	Thread gradual pull-out <i>ON / OFF</i>	T
M98 M99	Subprogram <i>call</i> / Subprogram <i>end</i>	M/T

10

SEQUENCE BLOCK

Each line of a CNC program is called a *block*. In the terminology established earlier, a *block* was simply defined as a *single instruction processed by the CNC system*.

A sequence block, a program block - or simply a *block* - is normally one hand written line of the program listing, or a line typed in a text editor and terminated by the *Enter* key. This line can contain one or more program words - words that result in the definition of a *single instruction* to the CNC system. Such a program instruction typically contains combinations of preparatory commands, coordinate words, tool functions and commands, coolant function, speeds and feeds commands, position registration, offsets of different kinds, etc. In plain English, the contents of one block will be processed as a single unit *before* the control processes any following block of instruction. While a CNC program is being processed, the control system will evaluate individual instructions (blocks) *as one complete machine operation step*. Each part program consists of a series of blocks necessary to complete a certain machining process. The overall program length will always depend on the total number of blocks and their size.

BLOCK STRUCTURE

A single block allows as many program words as necessary. Some controls impose a limit on the number of characters in one block. There is only a theoretical maximum for Fanuc and similar controls, irrelevant in practice. The only restriction is that *two or more duplicated* words (functions or commands) cannot be used in the same block (with the exception of G codes). For example, only one miscellaneous M function or one coordinate word for the X axis in a single block are allowed. Most recent controls allow up to three M-functions in a block, providing they are not in conflict with each other. The order of individual words within a block follows a fairly free format - that means the required words may be in any order, providing the sequence block (N address) is written *first*. Although individual words are allowed in a block to be in any order, it is a standard practice to place words in a *logical order* within a block. It makes the CNC program easier to read and understand.

A typical program block structure is very dependent on the control system and CNC machine type. A typical block may contain several instructions, suggested in the order of entry. Not all program data are necessary to be specified in each block, only when required (to switch from one mode to another, for example). A block typically includes:

- Block number N
- Preparatory commands G
- Auxiliary functions M
- Axis motion commands X Y Z A B C U V W ...
- Words related to axes I J K R Q ...
- Machine or tool function S F T

The contents of a program block will vary between machine tools of different types, but logically, general rules will always be followed, regardless of the CNC system or the machine tool.

◆ Building the Block Structure

Each block in a CNC program has to be developed with the same thought and care as any other important structure, for example a building, a car, or an aircraft. It starts with good planning. Decisions have to be made as to what will and what will not be part of the program block, similar to a building, car, aircraft or other structure. Also, important decisions have to be made as to what order of commands - or *instructions* - are going to be established within the block, and many other considerations.

The next few examples compare a typical structure of blocks for milling operations and blocks for turning operations. Each block is presented as a separate example.

◆ Block Structure for Milling

In milling operations, the structure of a typical program block will reflect the realities of a CNC machining center or a similar machine.

➡ Milling block examples:

N11 G43 Z10.0 S780 M03 H01 (EXAMPLE 1)

N98 G01 X237.15 Y45.75 F150.0 (EXAMPLE 2)

The first milling example in block N11, is a typical illustration of a tool length offset entry, applied together with the spindle speed and spindle rotation direction.

The second example (block N98), shows a typical programming instruction for a simple linear cutting motion, using the linear interpolation method and a suitable cutting feedrate. Both examples reflect metric setting.

☉ Turning block examples:

N67 G00 G42 X50.0 Z2.5 T0202 M08 (EXAMPLE 1)

N23 G02 X75.0 Z-28.0 R5.0 F0.125 (EXAMPLE 2)

In the lathe examples (also metric), block N67 illustrates a rapid motion to an XZ position, as well as several other commands - the tool nose radius offset startup G42, activation of the tool offset (T0202), and the coolant ON function M08. The example in block N23 is a typical circular interpolation block with a feedrate of 0.125 mm/rev.

PROGRAM IDENTIFICATION

A CNC program can be identified by its number and - on some controls - also by its name. Identification by the program number is necessary in order to store more than one program in the CNC memory. Program name or a brief description, if supported, can be viewed on the control screen display.

◆ Program Number

The first block used in any part program is commonly a program number, and must be specified in the program, if the control system requires it. Two addresses are available for the program number - the capital letter O for EIA format, and the older colon [:] for ASCII (ISO) format. In memory operation, the control system always displays program number with the letter O. Program number block is *not always* necessary to include in the CNC program and often it is better to let the CNC operator make the selection.

If a program does use program numbers, they must be specified within a certain range. Programs for older Fanuc controls must be within the range of O1 - O9999, program number zero (O0 or O0000) is not allowed. Newer controls allow 5-digit program numbers, in range of O1 - O99999. Neither a decimal point or a negative sign is not allowed in any program number. Leading zeros suppression is normal - for example, O1, O01, O001, O0001, and O00001 are all legitimate entries, in this case for a program number *one*.

◆ Program Name

On the latest Fanuc control systems, the program name can be included *in addition* to the program number, *not instead* of it. Program name (or a brief program description) can be up to sixteen characters long (spaces and symbols are counted). The program name must be *on the same line* (in the same block) as the program number:

O1001 (DWG. A-124D IT.2)

There is a distinct advantage of displaying the program number along with its description - it makes the directory listing more descriptive and useful.

Watch carefully *where* the description is written. If the program name is longer than sixteen characters allowed, no error is generated, but only the first sixteen characters will be displayed. Make sure to avoid program names that can be ambiguous when displayed. Consider these two program names, they both appear to be correct:

O1005 (LOWER SUPPORT ARM - OP 1)

O1006 (LOWER SUPPORT ARM - OP 2)

Since the control screen display can only show the *first sixteen* characters of program name, the program names will be ambiguous when displayed:

O1005 (LOWER SUPPORT AR)

O1006 (LOWER SUPPORT AR)

To eliminate this problem, use an abbreviated description that falls *within* the sixteen characters and still contains all significant information - for example:

O1005 (LWR SUPP ARM OP1)

O1006 (LWR SUPP ARM OP2)

If a detailed description is required, it has to be divided over one or more comment blocks:

O1005 (LWR SUPP ARM OP1)

(OPERATION 1 - ROUGHING)

Specified comments in the block or blocks following the program number will *not* appear on the directory screen listing, but still will be a useful aid to the CNC operator. They will be displayed during the program processing and, of course, listed in a hard copy printout.

Program *names* should be short and descriptive - their purpose is to assist the CNC operator when searching for programs stored in the control memory. Any suitable information could include a drawing number or a part number, shortened part name, operation, etc. Descriptions not suitable in such blocks include machine model, control system, programmer's name, dates and times, company or customer's name and similar data - they can be part of program header, described earlier - *see page 50*.

On most controls, when loading part program into the memory, CNC operator must specify the program number, regardless of what the actual number is in the program. *Program number entered at the control always supercedes the programmer specified number*. It can be a number that just happens to be available (unused), or it can be a number that has a unique meaning, for example, a unique group (for example, all programs that begin with the O10xx belong to the group associated with a single customer). Sub-programs are different - they must always be stored using the number specified by the CNC programmer. Innovative use of program numbers may also serve to keep track of programs developed for each machine or part.

SEQUENCE NUMBERS

Individual program sequence blocks can be referenced with a number for easier orientation within the program. The program address for a block number is the letter N, followed by up to five digits - from sequence 1 to sequence 9999 or 99999, depending on the control system. The block number range will typically be N1 to N9999 for older controls and N1 to N99999 for newer controls. Some rather old controls accept block number in the three digit range only, N1 to N999.

The N address must always be the *first word* in the block. For an easier orientation in programs that use subprograms, there should be no duplication of block numbers between both types. For example, a main program starting with N1 and a subprogram also starting with N1 may cause a confusing situation. Technically, there is nothing wrong with such a designation. Refer to *page 387* for suggestions on block numbering in subprograms.

N0 and N negative or with a decimal point are not allowed

◆ Sequence Number Command

In the table below, the first column represents sequence numbers used normally, the second column shows the sequence numbers required in a format acceptable to the machine control system, as applied to a CNC program:

Increment	First block number
1	N1
2	N2
5	N5
10	N10
50	N50
100	N100

Using sequence numbers (block numbers) in a CNC program offers two advantages and at least one possible disadvantage:

- On the positive side, block numbers will make any program search much simplified during editing or tool repetition at the machine. They also make the program much easier to read when displayed during processing or read on paper. They mean both *the programmer* as well as *the operator* benefit
- On the negative side, block numbers will *reduce* the available control system memory storage. That means a fewer number of programs can be stored in the memory, and long programs may not fit in their entirety

◆ Sequence Block Format

Block number program input format notation, using the N address, is N5 for the more advanced controls and N4 or even N3 for older controls. Block number N0 is not allowed, neither is a minus sign, a fractional number, or the use of a decimal point. Minimum block increment number must always be an integer - smallest integer allowed is one (N1, N2, N3, N4, N5, etc.). A larger increment is allowed and its selection depends on your personal programming style or standards established within company.

Typical sequence block increments other than one are:

Increment	Program example
2	N2, N4, N6, N8, ...
5	N5, N10, N15, N20, ...
10	N10, N20, N30, N40, ...
100	N100, N200, N300, N400, ...

Some programmers like to use increments of 5, 10 or even more. There is nothing wrong with this method, but the CNC program will become unnecessarily too long, too soon, and possibly difficult to manage. Block numbers do occupy memory space.

In all cases of block increments other than one, the intent is the same - to allow for additional blocks to be filled-in between existing blocks, if such a need arrives. The need may arise while proving or optimizing the program during test run, where an addition to the existing program will be required. Although all new blocks (the ones inserted) will *not* be in the order of an equal increment, at least they will be numerically ascending. For example, a face cut on a lathe with a single cut (*Example A*) was modified by the machine operator for two cuts (*Example B*):

➡ Example A - single face cut :

```
N40 G00 G41 X85.0 Z0 T0303 M08
N50 G01 X-1.8 F0.2
N60 G00 W3.0 M09
N70 G40 X85.0
```

...

➡ Example B - *Example A* modified for two face cuts :

```
N40 G00 G41 X85.0 Z1.5 T0303 M08
N50 G01 X-1.8 F0.2
N60 G00 W3.0
N61 X85.0
N62 Z0
N63 G01 X-1.8
N64 G00 W3.0 M09
N70 G40 X85.0
```

...

Note the change in block N40 and added blocks N61 to N64. Preference in this handbook is to program in increments of one and if an addition is needed, the added blocks will have *no block numbers at all* (check if the control system allows block numbers to be omitted, most do).

➡ Example C - single face cut :

```
N40 G00 G41 X85.0 Z0 T0303 M08
N41 G01 X-1.8 F0.2
N42 G00 W3.0
N43 G40 X85.0
```

➡ Example D - **Example C** modified for two face cuts :

```
N40 G00 G41 X85.0 Z1.5 T0303 M08
N41 G01 X-1.8 F0.2
N42 G00 W3.0
X85.0
Z0
G01 X-1.8
G00 W3.0
N43 G40 X85.0
```

Note that the program is a little smaller and the additional blocks are quite visual and noticeable when printed or displayed on the screen.

Leading zeros may (and should) be omitted in the block number - for example, N00008 should be written as N8. Omitting leading zeros will significantly reduce the overall program length. Trailing zeros must always be written, to distinguish for such similarities as N08 and N80.

Use of block numbers in a part program is optional, as shown in the earlier example. A program containing block numbers is easier to read. For a CNC operator, search and edit functions in program editing are also easier. *Note* - some programming applications depend on the block numbers, for example, lathe multiple repetitive cycles G70, G71, G72, G73. In this case, *at least* the significant blocks have to be numbered (*see page 324*).

◆ Numbering Increment

Fanuc block numbers in a program can be in any physical order - ascending, descending or mixed - they can also be duplicated or missing altogether. Some programming practices are established as preferable, because they are logical and make sense. Having a mixed order of sequence numbers in a program serves no useful purpose and neither do duplicated sequence numbers. If a program contains duplicate block numbers and a block number search is initiated, the control system will only search for the *first* occurrence of the particular block number, which may or may not be the one expected. Any further search will have to be repeated from the string found last. The reason for such a generous latitude in sequence block numbering is to offer flexibility to CNC operator *after* the program has been done and loaded into the control.

Block sequence number does not affect the order of program processing, regardless of its increment. Even if the blocks are numbered in a descending or mixed order, the part program will always be processed sequentially, on the basis of the block *contents*, not its number. The increment of 5 or 10 is the most practical, since it allows for insertion of up to 4 to 9 blocks respectively between any two original blocks. That should be more than sufficient for the majority of program modifications. Normally, this handbook uses block increment of one (N1, N2, N3, ...).

For those CNC programmers who use a computer based programming system, just a few words relating to the programming of sequence numbers. Although the computer programming system allows the start number of the block and its increment to almost any combination, adhere to *the start and increment numbers of one* (N1, N2, N3, ...). The purpose of a computer based programming is to keep an accurate database of the part geometry *and* the cutting tool path. If the CNC program is modified manually, the part computer database is not accurate any more. Any CNC program change should always be reflected in the *source* of the program, as well as its result - never in the result alone.

◆ Long Programs and Block Numbers

Long programs are always difficult to load into a CNC memory with limited capacity. In such cases, the program length may be shortened by omitting block numbers altogether or - even better - by programming them only where really needed - for all *significant blocks*. Significant blocks are those that have to be numbered for purposes of program search, tool repetition, or other procedure that depends on program numbers, such as machining cycles or tool changes. In these cases, select increments of two or five, for the operator's convenience. Even limited use of sequence numbers will increase the program length, but for a justifiable reason.

If *all* sequence block numbers have been omitted in the program, the search on the machine control will become rather difficult. CNC operator will have no other option but to search for the next occurrence of a particular address within a block, such as X, Y, Z, etc., rather than a sequence block number. This method of search may unnecessarily prolong the search time.

END OF BLOCK CHARACTER

Because of the control system specifications, individual sequence blocks must be separated by a special character, known as the *end-of-block* character or by its abbreviation EOB or E-O-B. On most computer systems, the EOB character is generated by the *Enter* key on the keyboard. When a program is input to the control by MDI (Manual Data Input), EOB character key on the control panel terminates the block. The end-of-block symbol on Fanuc controls appears as a semicolon [;].

The displayed semicolon symbol is only a *graphic representation* of the end-of-block character and is never entered literally in the CNC program. Under no circumstances it should be included in the program itself. Some older control systems have an asterisk [*] as the display symbol for the end-of-block, rather than the semicolon [;]. Many controls use other symbols, that also represent the end of block, for example, some use the dollar sign [\$]. In any case, remember such symbol is only a *representation* of the end-of-block character, *not* its actual character.

STARTUP BLOCK OR SAFE BLOCK

A *startup block* (sometimes called a *safe block* or a *status block*) is a special sequence block. It contains one or more modal words (usually preparatory commands of several G groups) that preset the control system into a desired *initial* or *default* state. This block is placed at the beginning of each program or even at the beginning of each tool and it is the *first block* processed during a repetition of a program (or a tool within a program). In CNC program, the startup block usually precedes any motion block or axis setting block, as well as the tool change or tool index block. This is the block to be searched for, if the program or the desired cutting tool is to be repeated during machine operation. Such a block will be slightly different for the milling and turning systems, due to the unique requirements of each control system.

Earlier in this handbook (*see page 23*) one topic covered state of the control system when the main power has been turned on - which activates the *system default* settings. A CNC programmer should never count on these default settings or conditions, since they can be easily changed by the machine tool operator, without programmer's knowledge. If such a change does happen, the programmed settings will *not* correspond to those suggested by the machine tool manufacturer or the engineers who designed the control system.

A careful CNC programmer should always adopt the attitude of a *safe* programming approach and will not leave anything to chance. Programmer will try to preset all required conditions under the program control, rather than counting on the defaults of a CNC system. Such an approach is not only much safer, it will also result in programs that are easy to use during setup, tool path verification and tool repetition due to tool breakage, dimensional adjustments, etc. It is also very beneficial to the CNC machine operators, particularly to those with limited experience.

In all applications listed, startup block will not increase the machining cycle time in any way. Another benefit of a startup block is that the program is more transportable from one machine tool to another, since it does not count on the default settings of a particular machine-control combination.

The name *safe block* - which is another name used for a startup block - does not become *safe* on its own - it must be *made* safe. Regardless of the name, this block should contain all control settings for the program or cutting tool that start the program in a 'clean' state. The most common entries that set this initial status are the dimensioning system (metric/imperial and absolute/incremental), cancellation of any active cycle, cancellation of active cutter radius offset mode, plane selection for milling, feedrate default selection for lathes, etc. Examples presented here show some startup blocks for both milling and turning controls.

At the program beginning (for milling), a startup block may be programmed with the following contents:

N1 G00 G17 G21 G40 G54 G64 G80 G90 G98

N1 block is the first sequence number, G00 selects the rapid mode, G17 establishes the XY plane selection, G21 selects metric units, G40 cancels any active cutter radius offset, G64 sets a continuous cutting mode, G80 cancels any active fixed cycle, G90 selects the absolute mode, and G98 will retract to the initial level in a fixed cycle. These conditions apply only when the startup block is processed as the *first major block* in CNC program - any subsequent program changes will become effective only with the block in which the change is applied. For example, if G01 command is effective by default, any subsequent usage of G00, G02, or G03 will cancel the G01 command.

At the beginning of a CNC lathe program, the startup block may contain these G codes:

N1 G21 G00 G40 G99

N1 is the first block number, G21 selects metric units, G00 selects the rapid mode, G40 cancels any active tool nose radius offset, and the G99 selects feedrate per revolution mode. Reference to absolute or incremental system is usually not required, since lathe controls use addresses X and Z for *absolute* dimensioning and addresses U and W for *incremental* dimensioning. For lathe controls that do not support the U and W addresses, the standard G91 code is used for incremental values in X and Z axes. As in the milling example, any of the words programmed in the safe block can be overridden by subsequent change of G-codes.

Some controls systems do not allow certain G-codes on the same line. For example, G20 or G21 may *not* be programmed with other G-codes. If you are not sure, place these G-codes in separate blocks. Instead of ...

N1 G21 G17 G40 G49 G80

... two or more blocks can be safely used:

**N1 G21
N2 G17 G40 G49 G80**

PROGRAM COMMENTS

One or more program comments and messages can be included within the program body as separate blocks, or as parts of an existing block, mostly in cases when the message is short. In either case, the message must be enclosed in parenthesis (for ASCII/ISO format):

➡ Example A :

N330 M00 (REVERSE PART)

➡ Example B :

N330 M00 (REVERSE PART / CHECK TOOL)

➡ Example C :

N330 M00
(REVERSE PART / CHECK TOOL)

The purpose of a message or comment in the program is to inform the machine operator of a specific task that must be performed *every time* program reaches the stage of processing where such message appears. Comments are also useful for understanding the program at a later date and can be used for documenting the program.

Typical messages and comments relate to information about setup changes, chip removal from a hole, dimensional check, cutting tool condition check and many others. A message or a comment block should be included only if the required task is *not clear* from the program itself - no need to describe what happens in each block. Messages and comments should be brief and focused, as they occupy a memory space in the CNC memory.

From practical perspective, short series of messages and comment blocks can be provided at the beginning of each program, to list all significant drawing information and cutting tools required for the job. This subject has been covered on *page 50* - here is just a reminder:

O1001 (SHAFT - DWG B451)
(SHAFT TOOLING - OP 1 - 3 JAW CHUCK)

(T01 - ROUGH TOOL - 1/32R - 80 DEG)
(T02 - FINISH TOOL - 1/32R - 55 DEG)
(T03 - OD GROOVING TOOL - 0.125 WIDE)
(T04 - OD THREADING TOOL - 60 DEG)

N1 G20 G99
N2 ...

If the available memory space of CNC unit is limited, using comment blocks in this manner may prove impractical. It will be better if the messages and comments are listed in proper setup and tooling sheets, with all required details.

CONFLICTING WORDS IN A BLOCK

All instructions in a program block must be logical and reasonable - not impossible. For instance, the first block of a program contains the following words:

N1 G20 G21 G17

What the block contains is simply not logically possible. It instructs the control to:

'Set the imperial system of dimensions, also set the metric system of dimensions and set the XY plane'.

Definitely not possible, but also not realistic - what will actually happen and how does the control interpret such a statement? The XY-plane is all right, but what about the selection of dimensions? Obviously, both selections are not possible - the block contains *conflicting* words - opposite dimensional units. Some controls may give an error message, Fanuc systems will not. What will happen? The control unit will evaluate the sequence block and check for any words within the same group. Distribution of G-code groups have been described in the section dealing with preparatory commands - G codes (*see page 55*).

If the computer system finds two or more words that belong to the same group, it will not return an error message, it will automatically activate the *last* word of the group. In the example of conflicting dimensional selection, it will be the preparatory function G21 - selection of metric dimensions - that becomes active. That may or may not be the selection required. Rather than counting on some kind of elusive luck, make sure there are no conflicting words in any program block.

In the example illustrating the metric and imperial selection, the preparatory command G was used. What would happen if, for example, the address X was used? Consider the following example:

N120 G01 X11.774 X10.994 Y7.056 F15.0

There are two X addresses in the same block. Control system *will not* accept the second X-value, but it will issue an alarm (error). Why? Because there is a great difference between programming rules for a G-code as such and for coordinate system words. Fanuc controls allow to place as many G-codes in the same block as needed, providing they are not in conflict with each other. But the same control system will not allow to program more than one coordinate word of the same address for each sequence block. Some other rules may also apply. For example, the words in a block may be programmed in any order, providing the block N-address is the first one listed. For example, the following block is legal (but very nontraditional in its order):

N340 Z-0.75 Y11.56 F10.0 X6.845 G01

As a matter of good programming practices, be sure to write the entries for each sequence block in a logical order. Block number must be the first word and is usually followed by G-code(-s), primary axes in their alphabetical order X., Y., Z.), auxiliary axes or modifiers (I., J., K.), miscellaneous functions and words, and the feedrate word as the last item. Select only those words needed for the individual block:

```
N340 G01 X6.845 Y11.56 Z-0.75 F10.0
```

Two other possibilities exist that may require a special attention in programming approach. For example, how will the following block be interpreted?

```
N150 G01 G90 X5.5 G91 Y7.7 F12.0
N151 (INCREMENTAL MODE IN EFFECT)
```

There is an apparent conflict between the absolute and incremental modes. Most Fanuc controls will process this block exactly the way it is written (check first). The X-axis target position will be reached in absolute mode, but the Y-axis will be an incremental distance, measured from tool current position. It may not be a typical approach, but it offers advantages in some cases. Remember - the sequence block following block N150 will be in *incremental* mode, since G91 is specified *after* the G90 command!

Another programming application to watch for, is in a block programmed while circular interpolation mode is active. Section dealing with this subject starts on *page 243*, and specifies that an arc or a circle can be programmed either with arc vectors I, J and K (depending whether a milling or a turning control system is used). It also specifies that a direct radius input, using the address R, can be used. Both of the following examples are correct, resulting in a 90° arc with a 1.5 inch radius:

➡ With I and J arc vectors :

```
N21 G01 X15.35 Y11.348
N22 G02 X16.85 Y12.848 I1.5 J0
N23 G01 ...
```

➡ With the direct radius R-address :

```
N21 G01 X15.35 Y11.348
N22 G02 X16.85 Y12.848 R1.5
N23 G01 ...
```

Now, consider how the control system will process block N22, if it contains *both*, the I and J vectors as well as the radius input:

```
N22 G02 X16.85 Y12.848 I1.5 J0 R1.5
```

or

```
N22 G02 X16.85 Y12.848 R1.5 I1.5 J0
```

The answer may be surprising - in *both cases*, the control will ignore the I and J values and *will only process the amount of radius R*. Actual order of address definition is irrelevant in this special case. The address R has a higher control priority than I and J addresses, if programmed in the same block. All examples assume that the control system supports R-radius input.

MODAL PROGRAMMING VALUES

Many program words are modal. The word *modal* is based on the word 'mode' and means that a specific command remains in this mode after it has been used in the program once. It can only be canceled by another modal command of the same group. Without this feature, a program using linear interpolation in absolute mode with a feedrate of 18.0 in/min, would contain absolute command G90, linear motion command G01 and feedrate F18.0 *in every block*. With modal values, the programming output is much shorter. Virtually all controls accept modal commands. The following two examples illustrate the differences:

➡ Example A - *without* modal values :

```
N12 G90 G01 X1.5 Y3.4 F18.0
N13 G90 G01 X5.0 Y3.4 F18.0
N14 G90 G01 X5.0 Y6.5 F18.0
N15 G90 G01 X1.5 Y6.5 F18.0
N16 G90 G01 X1.5 Y3.4 F18.0
N17 G90 G00 X1.5 Y3.4 Z1.0
```

➡ Example B - *with* modal values :

```
N12 G90 G01 X1.5 Y3.4 F18.0
N13 X5.0
N14 Y6.5
N15 X1.5
N16 Y3.4
N17 G00 Z1.0
```

Both examples will produce identical results. Compare each block of *Example A* with the corresponding block of *Example B*. Observe that the modal commands are *not* necessary to be repeated in CNC program. In fact, in everyday programming, many program commands used are modal. The exceptions are those program instructions, whose functionality starts and ends in the same block (for example dwell, machine zero return, certain machining instructions, such as tool change, indexing table, etc.). M-functions behave in a similar fashion. For example, if the program contains a machine zero return in two consecutive blocks (usually for safety reasons), it may look like this:

```
N83 G28 Z1.0 M09
N84 G28 X5.375 Y4.0 M05
```

G28 cannot be removed from block N84, because the G28 command is *not* modal and must be repeated.

EXECUTION PRIORITY

There are some special cases, mentioned earlier, where the *order* of commands in a block determines the priority in which the commands are executed. To complete the subject of a sequence block, let's look at another situation.

Here are two unrelated blocks used as examples:

N410 G00 X22.0 Y34.6 S850 M03

and

N560 G00 Z5.0 M05

In the block N410, rapid motion is programmed together with two spindle commands. What will *actually happen* during program execution? It is very important to know *when* the spindle will be activated in relationship to the cutting tool motion. On Fanuc and many other controls, the spindle function will take effect *simultaneously* with the tool motion.

In block N560, Z-axis (Z5.0) motion is programmed, this time together with the spindle stop function (M05). Here, the result will be different. Machine spindle will be stopped only when the motion is *one hundred percent completed*. For more details relating to M-functions used with tool motion, see tables on *page 60*.

Similar situations exist with a number of miscellaneous functions (M-functions), and any programmer should find out exactly how a particular machine and control system handle a motion combined with an M-function address in the same block. Here is a refresher in the form of a list of the most common results:

Functions that will be executed *simultaneously* with tool motion are:

M03	M04	M07	M08
-----	-----	-----	-----

Functions that will be executed *after* the tool motion has been completed are:

M00	M01	M05	M09	M98
-----	-----	-----	-----	-----

Be careful here - if in doubt, program it safe. Some miscellaneous functions require an *additional* condition, such as another command or function to be active. For example, M03 and M04 will only work if the spindle function S is in effect (spindle is rotating). Other miscellaneous functions should be programmed in separate blocks, many of them for logical or safety reasons:

M02	M06	M10	M11	M19	M30	M60	M99
-----	-----	-----	-----	-----	-----	-----	-----

Functions indicating the end of program or subprogram (M02, M30, M99) should stand on their own and not be combined with other commands in the same block, except in special cases. Functions relating to machine mechanical activity (M06, M10, M11, M19, M60) *should* be programmed without any motion in effect, for safety. In case of M19 (spindle orientation), the spindle rotation must be stopped first, otherwise machine may get damaged. Not all M-functions are listed in the above examples, but they should provide a good understanding of how they may work, when programmed together with a motion. The chapter describing the miscellaneous functions also covers the duration of typical functions within a program block.

It never hurts to play it safe and always program these possible troublemakers in a sequence block containing no tool motion. For mechanical M-functions, make sure the program is structured in such a way that it provides safe working conditions - these functions are oriented mainly towards the machine setup.

11

INPUT OF DIMENSIONS

Addresses in a CNC program that relate to the tool position at a given moment are called *the coordinate words*. Coordinate words always have a dimensional value, using currently selected units - *metric or imperial*. Typical coordinate words are X, Y, Z, I, J, K, R, etc. They are the basis of all dimensions in CNC programs. Tens, hundreds, even thousands of calculations may have to be made to develop a program that will do what it is intended to do - to *accurately machine a complete part*.

Dimensional entries in a program assume two attributes:

- Dimensional *units* ... Imperial or Metric
- Dimensional *references* ... Absolute or Incremental

Unit of dimensions in a program can be one of two kinds - *metric or imperial*. The reference of dimensions can be either *absolute or incremental*.

Fractional values, for example 1/8, are not allowed in a CNC program and have to be converted to their decimal equivalent. In metric format, *millimeters* and *meters* are used as units, in imperial format it is *inches* and *feet* that form the basis of units. Regardless of format selected, the number of programmed decimal places can be controlled, suppression of leading and trailing zeros can be set and decimal point can be programmed or omitted, as is applicable to a particular CNC system.

IMPERIAL AND METRIC UNITS

Drawing dimensions can be used in a program in either *metric or imperial* units. This handbook uses the combined examples of both the imperial (English) system, still common in the USA, to some extent in Canada, and one or two other countries. The metric system is common in the rest of the world. With economy reaching global markets, it is important to understand both systems. The use of metric system is on the increase even in countries that still use imperial units of measurement, mainly the United States.

Machines that come equipped with Fanuc controls can be programmed in either mode. Initial CNC system selection (known as the *default* condition) is controlled by a parameter setting of the control system, but can be overridden by a preparatory command written in the part program. Default conditions are usually set by the machine tool manufacturer or distributor. Often, it is based on engineering design decisions, as well as the demands of their customers.

During program development, it is imperative to consider the impact of control system default conditions on program execution. Default conditions come into effect the moment a CNC machine has been *turned on*. Once a command is issued, in MDI mode or in a program, many default values may be overwritten and will remain changed from that point on. Dimensional unit selection in the CNC program will change the *default* value - that is the internal control setting. In other words, if metric unit selection is made, the control system will remain in that mode until an imperial selection command is entered. That can be done either through the MDI mode, a program block, or a system parameter. This applies even for situations when the power has been turned *off* and then *on* again!

To select a specific dimensional input, regardless of default conditions, a preparatory G command is required at the beginning of CNC program:

G20	Selects imperial units (inches and feet)
G21	Selects metric units (millimeters and meters)

Without specifying the preparatory command in the program, control system will default to the status of current parameter setting. Both preparatory command selections are *modal*, which means the selected G code remains active until the opposite G code is programmed - so metric units will be active until the imperial system replaces it and vice versa.

This reality may suggest a certain freedom of switching between the two units anywhere in the program, almost at random and indiscriminately. *This is not true*. All controls, including Fanuc, are based on the *metric* system, partially because of the Japanese influence, but mainly because the metric system is more accurate. Any 'switching' by use of the G20 or G21 command does not necessarily produce any real conversion of one unit into the other, but merely *shifts* the decimal point, not the actual digits. At best, only some conversions take place, not all. For example, G20 or G21 selection will convert one measuring unit to another on *some* - but *not all* - offset screens. Many controls will convert all settings, but even then it is *not* recommended to mix the two unit systems in the same program. Following two examples will illustrate the *incorrect* result of changing G21 to G20 and G20 to G21 within the same program. Read the comments attached to each block - you may find a few surprises:

➡ Example 1 - from metric to imperial units :

G21 *Initial unit selection (metric)*
 G00 X60.0 *X value is accepted as 60 mm*
 G20 *Previous value will change into 6.0 inches*
 (real translation is 60 mm = 2.3622047 inches)

➡ Example 2 - from imperial to metric units :

G20 *Initial unit selection (imperial)*
 G00 X6.0 *X value is accepted as 6.0 inches*
 G21 *Previous value will change into 60 mm*
 (real translation is 6.0 inches = 152.4 mm)

Both examples illustrate the possible problem caused by switching between two dimensional units in the same program. For this reason, always use only one dimensional unit in a part program. If the program calls a subprogram, the rule extends to subprograms as well:

Never mix metric and imperial units in the same program

In fact, it is ill-advised to mix them, even if the results for the control system are predictable and full conversion is available. The selection of dimensional system will make a great difference how some control functions will work. The following functions *will be* affected by the change from one system of units to the other:

- Dimensional words (X, Y, Z axes, I, J, K modifiers, etc.)
- Constant Surface Speed (CSS)
(known as *cutting speed*, used by CNC lathes)
- Feedrate function (F-address)
- Offset values (H and D offsets for milling
and tool preset values for lathes)
- Screen position display (number of decimal places)
- Manual Pulse Generator - the HANDLE (value of divisions)
- Some control system parameters

The initial selection of dimensional units can also be done by a system parameter setting. Control status when the power has been turned on is the same as it was at the time of last power shut off. If neither G20 nor G21 is programmed, the control accepts dimensional units selected by a parameter setting. If G20 or G21 is included in the program, *the program command will always take a priority* over any control system parameter setting. Programmer makes all decisions - the control system is only interpreting them, but it does not mean it is always 'right'.

Dimensional units setting should always be in a separate block, *before* any axis motion, offset selection, or even setting of a coordinate system (G92, G50 and G54 to G59) - in other words, it should be in the first program block. Failure to follow this rule may produce incorrect results, particularly when frequently changing units for different jobs.

◆ Comparable Unit Values

There are many units available in both metric and imperial systems. In CNC programming, only a very small portion of them is used. Metric units are based on a *millimeter* or a *meter*, depending on application. Imperial units are based on *inches* and *feet*, again, depending on application. Common abbreviations for different units are:

Millimeter	mm
Meter	m
Inch	in
Foot	ft

Many programming terms use these abbreviations. The next table shows comparable terms between the two dimensional systems (older terms are in parentheses):

Metric	Imperial
m/min (also MPM)	ft/min (also FPM or SFPM)
mm/min	in/min (also IPM or ipm)
mm/rev	in/rev (also IPR or ipr)
mm/tooth	in/tooth (also IPT or ipt)
kW	HP

ABSOLUTE AND INCREMENTAL MODES

A dimension in either input units must have a *specified point of reference*. For example, if X35.0 appears in the program and currently selected units are millimeters, the statement does not indicate *where* the dimension of 35 mm has its origin. Control system needs more information to interpret dimensional values correctly, as intended.

There are two types of references in CNC programming:

- Reference to a *common point* on the part
... known as the origin for ABSOLUTE input
- Reference to a *previous point* on the part
... known as the last tool position for INCREMENTAL input

In the example, X35.0 dimension (and any other as well) can be measured from a selected fixed point of the part, called *origin*, or *program zero*, or *program reference point* - all these terms have the same meaning. The same value of X35.0 can also be measured from the *previous* position, which is always the *last* tool position. This position then becomes the *current* position for the *next* tool motion. Control system cannot distinguish one of the two possibilities from the X35.0 statement alone, and some other description must be added to the program.

All dimensions in a CNC program measured from the *common point* (origin) are *absolute* dimensions, as illustrated in Figure 11-1, and all dimensions in a program measured from the *current position* (last point) are *incremental* dimensions, as illustrated in Figure 11-2.

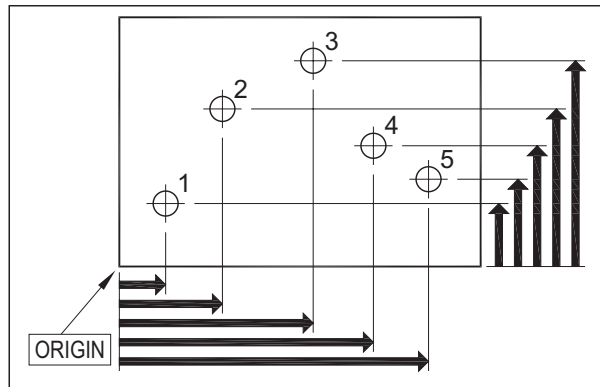


Figure 11-1

Absolute dimensioning - measured from part origin
G90 command will be used in the program

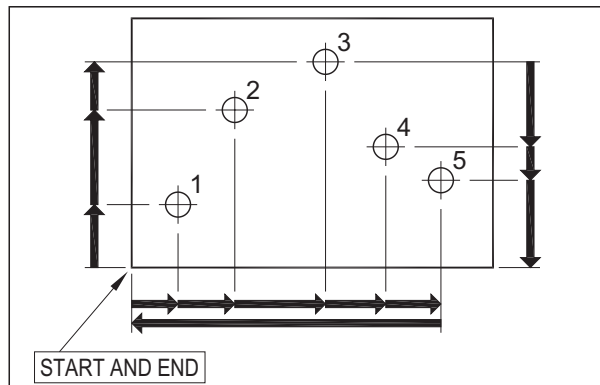


Figure 11-2

Incremental dimensioning - measured from the current tool location
G91 command will be used in the program

Absolute dimensions in a program represent target locations of the cutting tool from origin

Incremental dimensions in a program represent the actual amount and direction of the cutting tool motion from the current location

Since the dimensional address X, written as X35.0, is programmed the same way for *either* point of reference, some additional means must be available to the programmer. Without them, the control system would use default system parameter setting, not always reflecting the programmer's intentions. Selection of the dimensioning mode is controlled by two modal G commands.

◆ Preparatory Commands G90 and G91

There are two preparatory commands available for the input of dimensional values - G90 and G91 - to distinguish between two available modes:

G90	Absolute mode of dimensioning
G91	Incremental mode of dimensioning

Both commands are modal, therefore they will cancel each other. The control system uses an initial default setting when powered on, which is usually the *incremental mode*. This setting can be changed by a system parameter that pre-sets the computer at the power startup or a reset. For individual CNC programs, system setting can be controlled by including the proper preparatory command in the program, using either one of two available commands - G90 or G91.

It is a good programming practice to always include any required setting in the CNC program, and not to count on any default control system settings. It may come somewhat as a surprise that the common default control setting is the *incremental mode*, rather than the absolute mode. After all, absolute programming has a lot more advantages than incremental programming and is far more popular. In addition, even if incremental programming is used frequently, every program still starts up in the absolute mode. So, the question is *why* the incremental default? There is a good reason for it - as in many cases of defaults - *machining safety*. Follow this reasoning:

Consider a typical start of a new program loaded into the machine control unit. Control had just been turned on, part is safely mounted, current cutting tool is at the home position, offsets are set and the program is ready to start. Such a program is most likely written in more practical absolute mode. Everything seems fine, except that the absolute G90 command is missing in the program. *What will happen at the machine?* Think before an answer and think logically.

When the first tool motion command is processed, the chances are that its target values will be positive or have small negative values. Because any dimensional input mode is missing in the program, control system '*assumes*' the mode as being incremental, which is the normal default value stored in a system parameter. The active tool motion, generally in X and Y axes only, will take place to either the overtravel area, in the case of positive target values, or by a small amount, in case of negative target values. In either case, the chances are that no damage will be done to the machine or the part. Of course, there is no guarantee, so always program with safety in mind.

G91 is the standard default mode for input of dimensions

◆ Absolute Data Input - G90

In absolute programming mode, all dimensions are measured from the *point of origin*. Origin is the *program reference point*, also known as *program zero*. Actual motion of the machine is the real difference between the current absolute tool position and its previous absolute position. Algebraic signs [+] plus or [-] minus refer to the quadrant of rectangular coordinates, *not* the direction of motion - see page 16 for details. Positive sign does not have to be written for any address. All zero values, such as X0, Y0 or Z0 refer to the tool position at program reference point, *not* to the tool motion itself. Zero value of any axis must be written when such tool position is required.

The preparatory command G90, selected for absolute mode setting, remains modal until the incremental command G91 is programmed. In absolute mode, there will be no motion for any axis that is omitted in the program.

Probably the main advantage of absolute programming is ease of modification by either the programmer or CNC operator. A change of one dimension does not effect any other dimension in the program.

For CNC lathes with Fanuc controls, the common representation of the absolute mode is the axis designation as X and Z, *without* the G90 command. If available, rotation axis uses the address C in absolute mode. Some lathes may use the G90 command, but not those with Fanuc controls.

◆ Incremental Data Input - G91

In incremental mode of programming, also called a *relative* mode, all program dimensions are measured as departure distances into a specified direction (equivalent to 'distance-to-go' on the control system). Actual motion of the machine is the specified amount along each axis, with the direction indicated as positive or negative.

The signs + or - specify *direction* of the tool motion, *not* the quadrant of rectangular coordinates. Plus sign for positive values does not have to be written, but minus sign must be used. All zero input values, such as X0, Y0 or Z0 mean there will be *no tool motion along that axis*, and *do not* have to be written at all. If a zero axis value is programmed in incremental mode, it will be ignored. The preparatory command for incremental mode is G91 and remains modal until the absolute command G90 is programmed. There will be no motion of any axis omitted in the program block.

The main advantage of incremental programs is their portability between individual sections of a program. An incremental program can be called at different locations of the part, even in different programs. It is mostly used when developing subprograms or repeating an equal distance.

For Fanuc controlled CNC lathes, the most common representation of the incremental mode is the axis designation as U and W (for X and Z respectively), without the G91

command. If available, rotation axis uses the address H in incremental mode. Some lathes may use the G91 command, but not those with Fanuc controls.

◆ Combinations in a Single Block

On most Fanuc controls, absolute and incremental modes can be combined in a single program block for special programming purposes. This may sound rather unusual, but there are significant benefits in this advanced application. Normally, a block is in one mode only - *either* in absolute mode *or* in incremental mode. On many controls, for any change over to the opposite mode, motion command must be programmed in a separate block. Such controls, for example, do not allow programming incremental motion along one axis and absolute motion along the other axis *within the same block*.

Most Fanuc control systems *do* allow programming both modes *within the same block*. Just specify G90/G91 preparatory command *before* each significant address.

For lathe work, where G90 and G91 are not used, the change over is between X and U axes and Z and W axes. The X and Z contain absolute values, U and W are incremental values. Both types can easily be written within the same block. Here are two typical examples:

➡ Milling example (G21 in effect):

N68 G01 G90 X125.3 G91 Y45.15 F185.0

The milling example shows a motion where the cutter has to reach absolute position of X125.3 mm and - *at the same time* - has to move along the Y axis by specified distance of 4515 mm in the positive direction. Note the actual position of the G90 and G91 commands in the block - it is very important, but it may not work on all controls.

➡ Turning example (G20 in effect):

N60 G01 X13.56 W-2.5 F0.013

This example for a CNC lathe shows a tool path motion, where the cutting tool has to reach diameter of 13.56 inches and - *at the same time* - has to move 2.5 inches *into* the negative Z axis direction, represented by incremental designation address W. G90 or G91 is not normally used, when *Group A* of G codes is used (this is the most common G code group). G-code groups are described on page 55.

Anytime there is a switch between absolute and incremental mode in a CNC program, part programmer must be careful not to remain in the 'wrong' mode longer than needed. The switch between modes is usually temporary, for specific purpose and may affect one or more blocks. Always reinstate the original program setting. Remember that both absolute and incremental modes are *modal* - they remain in effect until canceled by the opposite mode.

DIAMETER PROGRAMMING

Every X-axis coordinate programmed for CNC lathes can be entered as a *diameter* value. This approach simplifies lathe programming and makes the program easier to read. Normally, the default of virtually all Fanuc controls is set to *diameter programming*. Control system parameter can be changed to interpret the X-axis as a radius input:

G00 X45.0 *Diameter dimension* ... when set by a parameter
G00 X22.5 *Radius dimension* ... when set by a parameter

Either value is correct, with the appropriate parameter setting (actual parameter number varies for different controls). Diameter programming is easier to understand by both the programmer and operator, because drawings use diameter dimensions for cylindrical parts and measuring diameters at the machine is equally common. Exercise certain caution - if the diameter programming is used, all tool offsets relative to X-axis must be treated as applicable to the *part diameter*, not to its single side (radius value).

Another important programming consideration is the selection of absolute or incremental mode of dimensional input. In diameter programming, where the X-axis amount represents part diameter, it is much more common to program in absolute mode. In those cases, when an incremental value is required, remember that all incremental dimensions in the program must also be specified *per diameter*, not per radius. In incremental mode, the intended X-axis motion will be programmed with the U address, specified as a distance and direction to travel on *diameter*.

For example, both sections of the following metric programs are identical - note that they both *start* in absolute mode and only the diameter inputs appear different:

➡ Example 1 - Absolute diameters :

```
G00 G42 X85.0 Z2.0 T0404 M08 (ABSOLUTE START)
G01 Z-24.0 F0.3
X95.0
Z-40.0
X112.0
Z-120.0
X116.0
G00 ..
```

➡ Example 2 - Incremental diameters :

```
G00 G42 X85.0 Z2.0 T0404 M08 (ABSOLUTE START)
G01 Z-24.0 F0.3
U10.0 (X95.0)
Z-40.0
U17.0 (X112.0)
Z-120.0
U4.0 (X116.0)
G00 ..
```

Absolute mode start is necessary - distance is not known.

MINIMUM MOTION INCREMENT

Minimum increment (also called the *least* increment) is the *smallest amount of an axis movement* the control system is capable of supporting. The minimum increment is the smallest amount that can be programmed within the selected dimensional input. Depending on the dimensional input selection, the minimum axis motion increment is expressed either in *millimeters* for the metric system or in *inches* for the imperial system.

Units system	Minimum increment
Metric	0.001 mm
Imperial	0.0001 inch

In the definition of minimum increment, the most common increments are 0.001 mm and 0.0001 inches for metric and imperial units respectively. For a typical CNC lathe, the minimum increment for the X-axis is also 0.001 mm or 0.0001 inches but is measured on the *diameter* - that means a 0.0005 mm or 0.00005 inches minimum increment per side. Fine tuning for machining precision is much more flexible and precise in metric system than in imperial system of units:

Minimum increment	Converted equivalent
0.001 mm	0.00003947 inches
0.0001 inches	0.00254 millimeters

For high precision work, using metric units should be the preferred dimensional system for part programming. In fact, the metric system is *154% more accurate* than the imperial system, which makes the imperial system almost *60.63% less accurate* than the metric system.

FORMAT OF DIMENSIONAL INPUT

The year of 1959 is often considered to be the first year when numerical control technology was first used in practical manufacturing. Since that time, several major changes have taken place that influenced programming format of dimensional input. Even to this day, dimensional data can be programmed in one of four possible ways:

- Full address format
- Leading zeros suppression
- Trailing zeros suppression
- Decimal point

In order to understand the format differences, looking back some years may be beneficial. Older control systems (mainly the old NC systems as compared to the modern CNC systems) were not able to accept the highest input level of dimensions - *decimal point format* - but all newest controls accept *all* earlier program formats, even when the decimal format is most common. The reason is compatibility with existing programs (old programs). Since decimal point programming method is the latest of four available, control systems that allow decimal point programming can also accept programs written many years earlier (assuming that the control and machine tool are also compatible). *The reverse is not true.*

This is a very important issue, because knowing how the control interprets a *number that has no decimal point* is critical for all tool motion commands and feedrates.

◆ Full Address Format

The full format of a dimensional address is described by the notation of ± 53 in metric units and ± 44 in imperial units. That means all eight available digits have to be written for the axis words X, Y, Z, I, J, K, etc. For example, metric dimension of 0.42 mm, when applied to the X axis, would be programmed as:

X00000420 ... eight digit format

Imperial dimension of 0.625, also when applied to the X axis, would be programmed as:

X00006250 ... eight digit format

The full format programming is applicable only to the *very early* control units, but is still correct even today. The programmed axis was usually written *without* the axis designation, which is determined by *position* of the dimension within the block. For modern CNC programming, the full format is obsolete and is used here only for reference and comparison. Yes, this format will work quite nicely in modern programs, but don't use it as a standard.

◆ Zero Suppression

Zero suppression concept is a great improvement over the full programming format. It was the adaptation of a new format that reduced the number of zeros in the dimensional input. Many modern controls still support the method of zero suppression, but only for reasons of compatibility with old and proven programs.

Zero suppression means that *either* the *leading* or the *trailing* zeros within the eight-digit dimensional input do not have to be written in the program. The result is a great reduction in program length. Any default setting is done by the control manufacturer, although it can be optionally set by a system parameter. *Don't make any changes without a valid reason!*

Since *leading* zeros suppression and *trailing* zeros suppression are mutually exclusive, which one is more practical for addresses without a decimal point? As it depends on parameter setting of the control system *or* status designation by the control manufacturer, the actual control status must always be known. This status determines which zeros can be suppressed. It may be zeroes at the *beginning* or zeros at the *end* of a dimension without a decimal point. These zeros are *before the first* significant digit or *after the last* significant digit. In the extremely unlikely event that a CNC system is equipped with zero suppression feature as the only mode, programming with decimal point will not be possible. To illustrate the results of zero suppression, earlier examples will be used.

The metric units input of 0.42 mm, also applied to the X axis, is programmed with leading zeros suppressed, as:

X420

The same dimension of 0.42 mm with trailing zeros suppressed, will appear in the program as:

X0000042

If the imperial input of 0.625 inches is to be programmed with *leading zero suppression* format in effect and applied to the X axis, it will appear in the program as:

X6250

The same dimension of 0.625 inches with *trailing zeros suppressed*, will appear in the program as:

X0000625

Although both examples above illustrate only one small application, the impression that leading zero suppression is more practical than trailing zero suppression is accurate. Many older control systems are indeed set arbitrarily to accept *leading zero suppression as the default*, because of its practicality. Here is the reason why - study it carefully, although today the subject is more trivial than practical. On the other hand, if even one decimal point is omitted (forgotten, perhaps) in the part program, this knowledge becomes very useful and the subject is not trivial any more.

Preference for Leading Zero Suppression

Minimum and maximum dimensional input the control system can accept consists of eight digits, without a decimal point, ranging from 00000001 to 99999999:

- Minimum: 0000.0001 inches *or* 00000.001 mm
- Maximum: 9999.9999 inches *or* 99999.999 mm

The decimal point is *not* written. If the program uses zero suppression of either type, a comparison of input values should be useful:

Input value comparison - Metric units (millimeters)		
Decimal point	Leading zeros suppression	Trailing zeros suppression
X0.001	X1	X00000001
X0.01	X10	X0000001
X0.1	X100	X000001
X1.0	X1000	X00001
X10.0	X10000	X0001
X100.0	X100000	X001
X1000.0	X1000000	X01
X10000.0	X10000000	X1

Leading zero suppression is more beneficial, because it supports numbers with the more practical *small* fractional part (fine sizes) rather than those with a large integer part.

For imperial input, the results will be similar:

Input value comparison - Imperial units (inches)		
Decimal point	Leading zeros suppression	Trailing zeros suppression
X0.0001	X1	X00000001
X0.001	X10	X0000001
X0.01	X100	X000001
X0.1	X1000	X00001
X1.0	X10000	X0001
X10.0	X100000	X001
X100.0	X1000000	X01
X1000.0	X10000000	X1

Even if programs always use decimal point, knowing the effect of zero suppression is important. For example, what will happen if the programmer forgets a decimal point in a motion address or the CNC operator forgets to punch it in for an offset entry? These are serious - *and common* - errors that can be avoided with some care and good knowledge.

To complete the section on zero suppression, let's look at a program input that uses an axis letter but *not as a coordinate word*. A dwell command is suitable for explanation. Starting on *page 175*, all details relating to dwell programming are covered. For now, just use the basic format and one second dwell time as the unit. The dwell format is specified by the dwelling axis X, as being X5.3. This for-

mat tells us that the dwell can be programmed with the X axis, followed by the maximum of eight digits, always positive. If the control system allows decimal point input, there is no confusion. If leading or trailing zeros have to be suppressed, the programmed input is very important.

For example, a program requires dwell lasting 0.5 of a second (one half of a second). In various formats, the program block containing the $\frac{1}{2}$ sec. dwell will be:

- Full format X00000500
- No leading zeros X500
- No trailing zeros X000005
- Decimal point X0.5 or X.5

Note that logic behind the format is the same for dwell as for the coordinate words. The programmed format will always adhere to the address notation. Incidentally, in several fixed cycles, the dwell is expressed by the P address, which *does not* take a decimal point at all and always must be programmed with leading zero suppression mode in effect. Half a second dwell will be equal to P500. More on using dwell in fixed cycles, *see page 183*.

◆ Decimal Point Programming

All modern programming will use the decimal point for dimensional and some other input. Programming decimal point, particularly for program data requiring very accurate measurement, makes the CNC program much easier to develop and to interpret at a later date.

From all available program addresses that can be used, not all can be programmed with the decimal point. Those that can are those that specify data in millimeters, inches, or seconds (some exceptions do exist).

The following two lists contain addresses where the decimal point is allowed in programs for both milling and turning controls:

➡ Milling control programs :

X, Y, Z, I, J, K, A, B, C, Q, R

➡ Turning control programs :

X, Z, U, W, I, K, R, C, E, F (also B and Y)

Control system that supports the decimal point method of programming, can also accept dimensional values *without* a decimal point, in order to allow compatibility with older programs. In such cases, it is important to understand the principles of programming format using leading and trailing zeros. If they are used correctly (see earlier explanations), there will be no problem to apply various dimensional formats to any other control system, old or new. If possible, always use decimal point as a standard approach.

This compatibility enables many long time users to load their old programs into newer CNC controls - but *not* the other way around - usually with only minor modifications, or no modifications at all.

Virtually all modern CNC units do not have the ability to accept a paper tape because they have no tape reader. To convert any tapes that contain good programs, there are two options - one, have someone to install a tape reader in the control, if possible and justified (probably not). The other method is to store the contents of a tape in a personal computer. This method is very inexpensive and offers much better storage options than a paper tape. With suitable software and a portable tape reader (that is the catch), the task is not impossible. Keep also in mind that there are few companies specializing in this kind of work.

Dimensional data in the metric system assume 0.001 mm minimum increment, while in the imperial system the increment is 0.0001 of an inch (leading zero suppression mode is in effect as a default) - therefore:

Y12.56 is **Y125600** ...for imperial units
Y12.56 is **Y12560** ...for metric units

Programmed values with and without a decimal point can be mixed together in the same block:

N230 X4.0 Y-10

This may be beneficial for extreme conservation of system memory. For example, the X4.0 word will require fewer characters than the word X40000 - on the other hand, the Y-10 is shorter than the decimal point equivalent of Y-0.001 (both examples are in imperial units). If all digits *before* or *after* the decimal point are zeros, they do not have to be written:

X0.5 = **X.5**
Y40.0 = **X40.**
Z-0.1 = **Z-.1**
F12.0 = **F12.**
R0.125 = **R.125** ... etc.

Any zero value *must* be written - for example, X0 cannot be written as X only. In this handbook, all program examples will use the decimal point format, whenever possible. Many programmers prefer to program zeros as in the left column of the example. This method does add a few characters into the system memory, but program data are much easier to read. They are also easier for learning.

◆ Input Comparison

Differences in the input format for both metric and imperial dimensioning can be seen clearly. One more time, the same examples will be shown, as before:

➡ Metric example - input of 0.42 mm :

Full format	X00000420
No leading zeros	X420
No trailing zeros	X0000042
Decimal point	X0.42 or X.42

➡ Imperial example - input of 0.625 inches :

Full format	X00006250
No leading zeros	X6250
No trailing zeros	X0000625
Decimal point	X0.625 or X.625

CALCULATOR TYPE INPUT

In some specialized industries, such as woodworking or fabricating, the majority of dimensions (especially metric) do not require decimal parts, only whole numbers. In these cases, the decimal point would always be followed with a zero. Fanuc provides a solution to such situations by a feature called the *calculator input*. Using this feature can shorten overall program size, often significantly.

Calculator type input requires setting of a system parameter. Once the parameter is set, neither the decimal point nor the trailing zeros have to be written - they will be assumed. For example, X25 will be interpreted as X25.0, *not* the normally expected X0.025 (mm) or X0.0025 (inches).

In case the input value does require decimal point, it can be written as usually. This means all values with a decimal point will be interpreted correctly and numbers without the decimal point will be treated as major units only (millimeters or inches). Here are some examples:

Standard Input	Calculator Input
X345.0	X345
X1.0	X1
Y0.67	Y0.67
Z7.48	Z7.48

Normally, the control system is set to leading zero suppression mode and non-decimal values are interpreted as the number of smallest units. For example, Z1000 in G21 mode will be equivalent to Z1.0 (mm), in G20 mode as Z0.1 (inches).

12

SPINDLE CONTROL

Both types of CNC machines, machining centers and lathes, use spindle rotation when removing excessive material from a part. Spindle rotation may be that of the cutting tool (milling) or the part itself (lathes). In both cases, activities of machine spindle and working feedrate of the cutting tool need to be strictly controlled by the program. These CNC machines require instructions that relate to the selection of a suitable *speed* of machine spindle and a cutting *feedrate* for a given job.

There is a number of ways to control machine spindle and cutting feedrate and they all depend mainly on the type of CNC machine and the current machining application. In this chapter, we look at the *spindle control* and its programming applications.

SPINDLE FUNCTION

Program command relative to spindle speed is controlled in the CNC system by the address *S*. Programming format of the *S* address is usually within the range of 1 to 9999 for standard machines and no decimal point is allowed:

S1 to S9999

For many high speed CNC machines is not unusual to have spindle speed available up to five digits, in the range of 1 to 99999, within the *S* address range:

S1 to S99999

Maximum spindle speed range available in the control must always be *greater* than the maximum spindle speed range of the machine itself. It is quite typical that virtually all control systems support a much greater range of spindle speeds than the CNC machine allows. In programming spindle speeds, the limitation is always caused by the machine unit, *not* by the control system.

◆ Spindle Speed Input

The address *S* relates to machine spindle function, and must always be assigned a specific numeric value in CNC programs. There are three alternatives as to what numeric value (input) of the spindle function may be:

- Code number .. old controls only - now obsolete
- Spindle speed .. r/min
- Cutting speed .. m/min or ft/min

On CNC lathes, both modern alternatives may exist, depending on the control system. For CNC milling systems, cutting speed is not applicable (except for calculations), but direct spindle speed is. Spindle speed selection by special code number is an obsolete concept, not required on modern controls and not discussed here.

Spindle speed designation *S* is not sufficient to be programmed by itself. In addition to the selected spindle speed address, certain additional attributes are necessary as well. These are attributes that control spindle function environment. For example, if the spindle speed is specified as *S400* in the program, programming instruction is not complete, because the spindle function stands by itself. It does not include *all* information the control system requires for spindle data. A spindle speed value that is set, for example, to 400 r/min or 400 m/min or 400 in/min or 400 ft/min (depending on actual machining application), still does not contain all necessary information, namely, the spindle rotation *direction*.

Most machine spindles can be rotated in two directions - *clockwise* or *counterclockwise*, depending on the type and setup of the cutting tool used. Spindle rotation has to be specified in the program, *in addition* to the spindle speed function. There are two miscellaneous functions provided by the control system that control direction of the spindle - *M03* and *M04*.

DIRECTION OF SPINDLE ROTATION

Thinking in terms of *right* and *left*, *up* and *down*, *clockwise* and *counterclockwise*, and similar directional terms, is thinking in terms that are *relative* to some known reference. To describe a spindle rotation as clockwise (CW), or as counterclockwise (CCW), some established and standard reference method is needed, in this case a reference point of view (reference viewpoint).

Direction of spindle rotation is always relative to the point of view that is established from the spindle side of the machine. This part of a machine that contains the spindle, and is generally called the machine headstock. Looking from the machine headstock area into the *direction along spindle center line and towards its face*, establishes the correct viewpoint for defining CW and CCW rotation of the spindle. For CNC drills, milling machines and CNC machining centers, the reference point of view is quite simple to understand. For CNC lathes, the rules are exactly the same, and will be described shortly.

◆ Direction for Milling

It may be rather impractical to look down along the center line of the spindle, perpendicularly towards the part. The common standard view is from the operator's position, facing the front of a vertical machine. Based on this view, the terms *clockwise* and *counterclockwise* can be used accurately, as they relate to the spindle rotation - *Figure 12-1*.

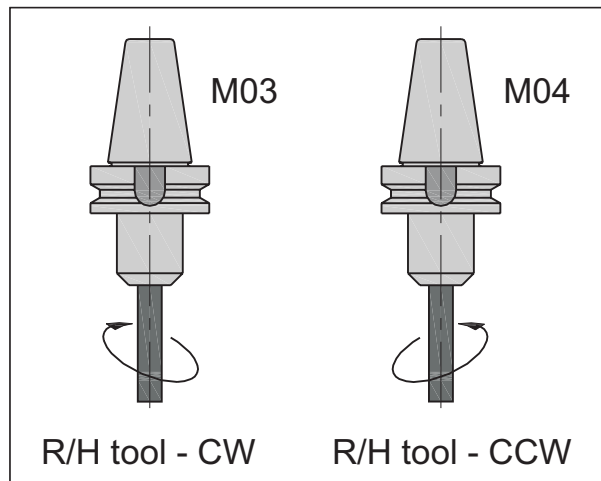


Figure 12-1
Direction of spindle rotation.
Front view of a vertical machining center is shown

◆ Direction for Turning

A comparable approach would also seem logical for CNC lathes. After all, the operator also faces the front of a machine, same as when facing a vertical machining center. *Figure 12-2* shows a front view of a typical CNC lathe.

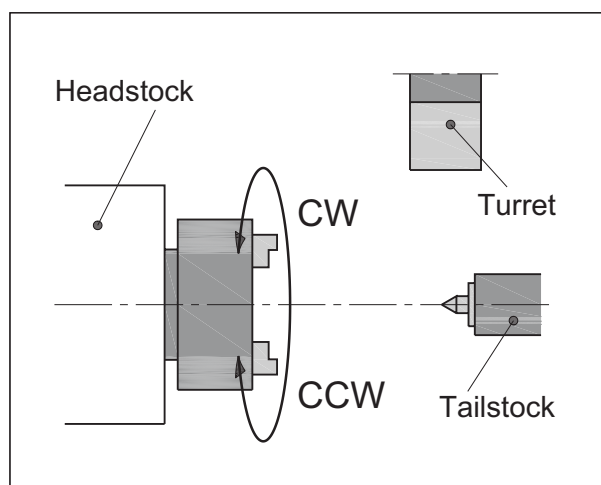


Figure 12-2
Typical view of a slant bed two axis CNC lathe.
CW and CCW directions only appear to be reversed

Although descriptions CW and CCW in the illustration appear to be opposite to the direction of arrows, they are correct. The reason is that there are two possible points of view, and they are both using spindle center line as the viewing axis. Only one of the viewpoints matches the standard definition and is, therefore, correct. The definition of spindle rotation for lathes is *exactly* the same as for machining centers.

To establish spindle rotation as CW and CCW,
look from the headstock area towards the spindle face

The first and proper method will establish the relative viewpoint starting *at the headstock* area of the lathe. From this position, looking towards the tailstock area, or into the same general area, clockwise and counterclockwise directions are established correctly.

The second method of viewing establishes the relative viewpoint starting *at the tailstock* area, facing the chuck. This is an *incorrect* view to establish spindle rotation!

Compare the following two illustrations - *Figure 12-3* shows the view from the headstock, *Figure 12-4* shows the view from the tailstock and arrows must be reversed.

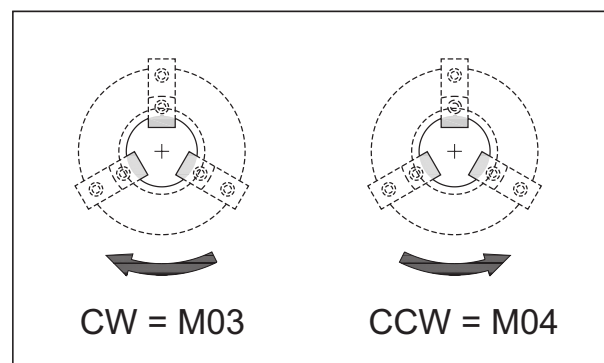


Figure 12-3
Spindle rotation direction as viewed from the headstock

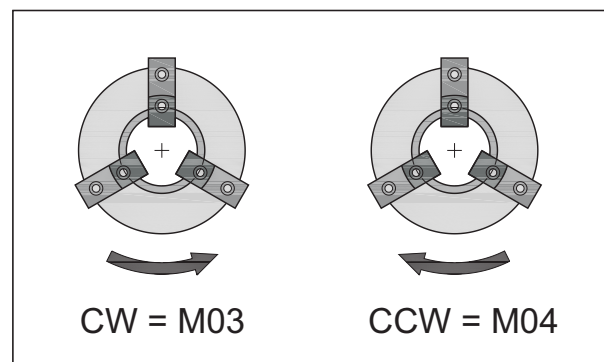


Figure 12-4
Spindle rotation direction as viewed from the tailstock

◆ Direction Specification

If the spindle rotation is clockwise, M03 function is used in the program - if spindle rotation is counterclockwise, M04 function is used in the program.

Since spindle speed S in the program is dependent on the spindle rotation function M03 or M04, their relationship in a CNC program is very important.

Spindle speed address S and spindle rotation function M03 or M04 must always be accepted by the control system together. One without the other will not mean anything to the control.

There are at least two correct ways to program spindle speed and spindle rotation:

- If spindle speed and rotation are programmed together in the same block, spindle speed and spindle rotation will start simultaneously
- If spindle speed and rotation are programmed in separate blocks, spindle will not start rotating until both the speed and the rotation commands have been processed

◆ Spindle Startup

The following examples demonstrate a number of correct starts for spindle speed and spindle rotation in a program. All examples assume that there is *no active setting* of spindle speed S, either through a previous program or through *Manual Data Input* (MDI). On CNC machines, there is no registered or default spindle speed when machine power is turned on.

➡ Example A - Milling application :

```
N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X14.0 Y9.5
N4 G43 Z1.0 H01 S600 M03 (SPEED WITH ROTATION)
N5 ...
```

This example is one of the preferred formats for milling applications. Both spindle speed and spindle rotation are set with the Z-axis motion towards the part. Equally popular method is to start the spindle with XY motion - block N3 in the example:

```
N3 G90 G00 G54 X14.0 Y9.5 S600 M03
```

Selection is a matter of personal preference. Units selection does not influence spindle speed.

➡ Example B - Milling application :

```
N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X14.0 Y9.5 S600 (SPEED ONLY)
N4 G43 Z1.0 H01 M03 (ROTATION STARTS)
N5 ...
```

This second example B is technically correct, but logically flawed. There is no benefit in splitting spindle speed and spindle rotation into two blocks. This method makes the program harder to interpret.

➡ Example C - Milling application :

```
N1 G20
N2 G17 G40 G80
N3 G00 G90 G54 X14.0 Y9.5 M03 (ROTATION SET)
N4 G43 Z1.0 H01 (NO ROTATION)
N5 G01 Z0.1 F50.0 S600 (ROTATION STARTS)
N6 ...
```

Again, example C is not wrong, but it is not very practical either. There is no danger, if the machine power has been switched on just prior to running this program. On the other hand, M03 *will* activate the spindle rotation, if another program was processed earlier. This could create a possibly dangerous situation, so follow a simple rule:

Program M03 or M04 together with or after the S address, not before

➡ Example D - Turning application with G50 :

```
N1 G20
N2 G50 X13.625 Z4.0 T0100
N3 G96 S420 M03 (SPEED SET - ROTATION STARTS)
N4 ...
```

This is the preferred example for CNC lathes, if the older G50 setting method is used. Because the spindle speed is set as cutting speed CSS - *Constant Surface Speed*, the control system will calculate the actual revolutions per minute (r/min) based on the CSS setting of 420 (ft/min) and the current part diameter at X13.625. The next example E is correct but not recommended (see caution box above).

➡ Example E - Turning application with G50 :

```
N1 G20
N2 G50 X13.625 Z4.0 T0100 M03 (ROTATION SET)
N3 G00 X6.0 Z0.1 (NO ROTATION)
N4 G96 G01 Z0 F0.04 T0101 S420 (ROTAT. STARTS)
N5 ...
```

➡ Example F - Turning application without G50 :

```
N1 G20 T0100
N2 G96 S420 M03 (SPEED SET - ROTATION STARTS)
N3 G00 ...
```

In this more contemporary example (G50 is not used as a position register command anymore), machine spindle speed will be calculated for a tool offset value stored in the *Geometry Offset* register of the control system. Control system will perform the calculation of actual r/min when the block N2 is executed.

These examples are only *technically correct* methods for a spindle start. All contain selected rotation at the beginning of a program and cover both, milling and turning applications. The example for *beginning* of a program has been selected intentionally, because for any first tool in the program, there is no active speed or rotation in effect (normally carried on from a previous tool). However, the control unit may still store spindle speed and rotation from the last tool of the *previous* job!

Any tool *following* the first tool will normally assume the programmed speed selection and rotation for the previous tool. If only the spindle speed command S is programmed for the next tool, *without* rotation direction, the tool will assume the *last* programmed rotation direction. If only the rotation direction code M03 or M04 is programmed, spindle speed S will be the same as for the previous tool.

Be careful if a program contains program stop functions M00 or M01, or the spindle stop function M05. Any one of them will automatically stop the spindle. It means to be absolutely sure as to *when* the spindle rotation will take place and what it will be. Always program spindle speed selection and its rotation in the same block and for *each* tool. Both functions are logically connected and placing them within a single block will result in a coherent and logical program structure.

SPINDLE STOP

Normally, most work requires a spindle that rotates at a certain speed. In some cases, a rotating spindle is not always desirable. For example, before programming a tool change or reverse a part in the middle of a program, the spindle must be stopped first. Spindle must also be stopped during a tapping operation and at the end of program. Some miscellaneous functions will stop spindle rotation automatically (for example, functions M00, M01, M02 and M30). Spindle rotation will also stop automatically during certain fixed cycles. For a total control of a program, spindle stop should always be specified in the program. Counting on other functions to stop the spindle is not a good programming practice. There is a special function available in programming, to stop the spindle.

In order to stop the spindle rotation, use function M05. M05 will stop both clockwise and counterclockwise spindle rotation. Because M05 does not do anything else (unlike other functions that also stop the spindle, such as M00, M01, M02, M30 and others), it is used for situations, where the spindle must be stopped *without* affecting any other programmed activities. Some typical examples include reversal in tapping, tool motion to the indexing position, turret change position, or after machine zero return, depending on application. Using one of the other miscellaneous functions that automatically stop spindle, the M05 function is not required. On the other hand, it does no harm to program exactly what is required, in a particular order.

This method may result in a slightly longer program, but it will be easier to read and maintain it, mainly by CNC operators with limited experience.

Spindle stop function can be programmed as a separate block as well, for example:

N120 M05

or in a program block containing a tool motion, such as in the next example:

N120 Z1.0 M05

Tool motion will always be completed first, *then* the spindle will be stopped. This is a safety feature built into the control system. Always remember to program M03 or M04 to reinstate spindle rotation.

SPINDLE ORIENTATION

The last M-function that also relates to spindle activity, is M19. This function is most commonly used to set a machine spindle into an oriented position. Other M-codes may be valid, depending on the control system, for example M20 on some controls. Spindle orientation function is a very specialized function, seldom appearing in the program itself. When M19 function is used, it is mainly during setup, in *Manual Data Input* mode (MDI). This function is generally common for milling systems, because only specially equipped CNC lathes will require it. M19 function can only be used when the spindle is stationary, usually after the spindle has stopped. When the control system executes M19 function, the following action will result:

The spindle will make a slight turn in both directions, clockwise and counterclockwise, and after a short period, the internal locking mechanism will be activated. In some cases, the locking that takes place is audible. The spindle will be locked in a *precise* position, and rotating it by hand, will not be successful. The exact locking position is fixed and is determined by the machine tool manufacturer, indicated by the setting angle - *Figure 12-5*.

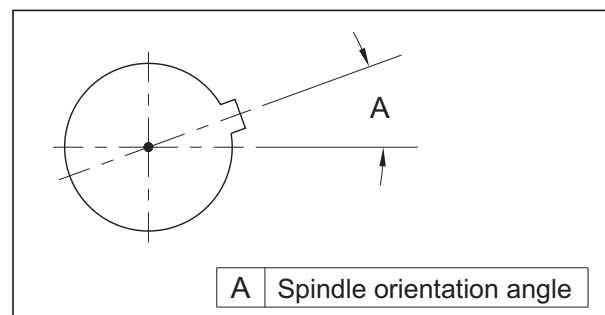


Figure 12-5

Spindle orientation angle is defined by the machine manufacturer and cannot be changed

In normal CNC operation, M19 function enables the machine operator to place a tool into the spindle manually and guarantees a proper tool holder orientation. Later chapters will provide more details about spindle orientation and its applications, for example, in single point boring cycles.

WARNING - An incorrect tool holder orientation may result in a damage to the part or machine

Many CNC machining centers (not all) use tool holders that can be placed into the tool magazine only one way. To achieve this goal, the tool holder has a special notch built-in, that matches the internal design of spindle - *Figure 12-6*. In order to find side of the holder that has the notch, usually there is a small dimple or other mark on the notch side. This design is intentional.

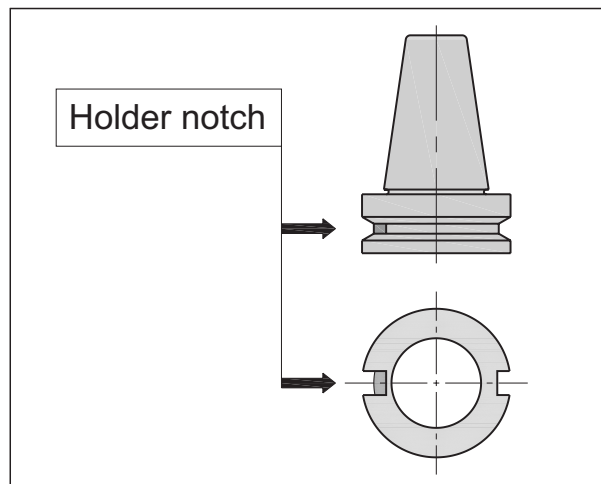


Figure 12-6

Built-in notch in a tool holder used for correct tool orientation in the spindle - not all machines require this feature

For tools with several flutes (cutting edges), such as drills, end mills, reamers, face mills, etc., the orientation of cutting edge relative to the stopped spindle position is not that important. However, for single point tools, particularly boring bars, orientation of the cutting edge during setup is extremely important, particularly when certain fixed cycles are used. There are two fixed cycles that use the *built-in* spindle orientation - G76 and G87 - in both, the tool retracts from the machined hole *without rotating*. In order to prevent damage to the just finished bore, the tool retraction must be controlled. Spindle orientation guarantees that the tool will shift away from the finished bore into a clear direction. An accurate initial setup is necessary!

Those machines that allow placing tool holder into the spindle either way still require proper setting of tools that shift when G76 or G87 fixed cycles are programmed.

SPINDLE SPEED - R/MIN

When programming CNC machining centers, designate the spindle speed directly in *revolutions per minute* (r/min). A basic block that contains spindle speed of 200 r/min, for example, will require this data entry:

N230 S200 M03

Such format is typical to milling controls, where no cutting speed (peripheral or surface speed) is used. There is no need to use a special preparatory command to indicate the r/min setting, it is the control default. The r/min setting must have a minimum increment of one. Fractional or decimal values are not allowed and the r/min must always be set within the range of any machine specifications.

A few machining centers may be equipped with the option of a *dual* spindle speed selection - *direct r/min* and *cutting speed*. In this case, as well as for all lathe programming, a proper preparatory command is used to distinguish which selection is active. G96 is used for cutting speeds, G97 for direct specification of r/min. The distinction between them is discussed next.

SPINDLE SPEED - SURFACE

Programmed spindle speed should be based on the machined *material* and the *cutting tool diameter* (machining centers), or the *part diameter* (lathes). General rule is that the larger the diameter, the slower the spindle r/min must be. Spindle speed should never be guessed - *it should always be calculated*. Such a calculation will guarantee that spindle speed is *directly proportional to the programmed diameter*. An incorrect spindle speed will have a negative effect on both tool and part.

◆ Material Machinability

To calculate spindle speed, each work material has a suggested machinability rating for a given tool material. This rating is either a percentage of some common material, such as mild steel, or a direct rating in terms of its *cutting (peripheral) speed* also known as *surface speed*. Surface speed is specified in *feet per minute (ft/min)* in imperial units, and in *meters per minute (m/min)* in metric system. An older abbreviation used for *ft/min* is *FPM*, meaning *Feet Per Minute*. Amounts of surface speeds indicate the level of machining difficulty with a *given tool material*. The lower the surface speed, the more difficult it is to machine the work material.

Note the words '*given tool material*'. To make all comparisons meaningful and fair, they must be made with the same type of cutting tool, for example, surface speeds for high speed steel tools will be much lower than those for cobalt based tools and, of course, for carbide tools.

Based on the surface speed and the cutter diameter (or part diameter for lathes), machine spindle speed can be calculated in *revolutions per minute*, using one mathematical formula for imperial units system and another when metric units are programmed.

Surface speed will increase for soft materials and decrease for hard materials

High speed steel tools will run slower than carbide tools in the same material

◆ Spindle Speed - Imperial Units

To calculate spindle speed in *r/min*, the *surface* speed of the material for the cutting tool type must be known, as well as the part or tool diameter:

$$r / \min = \frac{12 \times \text{ft} / \min}{\pi \times D}$$

where ...

r/min = Spindle speed in revolutions per minute
 12 = Multiplying factor - feet to inches conversion
 ft/min = Surface speed in **feet per minute**
 π = Constant 3.1415927
 D = Diameter in **inches** (cutter diameter for milling, or part diameter for turning)

➡ Example :

Surface speed for selected material is 150 ft/min, and the cutting tool diameter is 1.75 inches:

$$\begin{aligned} r / \min &= (12 \times 150) / (3.1415 \times 1.75) \\ &= 327.4 \\ &= 327 \text{ r/min} \end{aligned}$$

Many programming applications can use a shorter formula, without losing any significant accuracy:

$$r / \min = \frac{3.82 \times \text{ft} / \min}{D}$$

For less demanding calculations, the 3.82 constant may be rounded to 4 and used as an easier calculation without a calculator. The measuring units must be applied properly, otherwise the results will not be correct.

Never mix imperial and metric units in the same program !

◆ Spindle Speed - Metric Units

When metric system is used in a program, the logic of previous formula is the same, but the units are different:

$$r / \min = \frac{1000 \times \text{m} / \min}{\pi \times D}$$

where ...

r/min = Spindle speed in revolutions per minute
 1000 = Multiplying factor - meters to mm conversion
 m/min = Surface speed in **m/min**
 π = Constant 3.1415927
 D = Diameter in **mm** (cutter diameter for milling, or part diameter for turning)

➡ Example:

Given surface speed is 30 m/min and the cutting tool diameter is 15 mm:

$$\begin{aligned} r / \min &= (1000 \times 30) / (3.1415 \times 15) \\ &= 636.6 \\ &= 637 \text{ r/min} \end{aligned}$$

A shorter version of the formula is an acceptable alternative and almost as accurate as the precise formula:

$$r / \min = \frac{318.3 \times \text{m} / \min}{D}$$

Again, by replacing the constant 318.3 with constant 320 (or even 300), the *r/min* will be somewhat inaccurate, but most likely within an acceptable range.

CONSTANT SURFACE SPEED

On CNC lathes, machining process is different from milling process. Turning tool has no diameter and the diameter of a boring bar has no relationship to the spindle speed. It is the *part diameter* that is the diameter used for spindle speed calculations. As the part is being machined, its diameter changes constantly. For example, during a facing cut or during roughing operations the diameter changes - see illustration in *Figure 12-7*. Programming the spindle speed in *r/min* is not practical - after all, which of the numerous diameters should be selected to calculate the *r/min*? The solution is to use the *surface speed* directly, as an block entry in the lathe program.

To select surface speed is only a half of the procedure. The other half is to communicate this selection to the control system. Control has to be set to the *surface speed* mode, not the *spindle speed (r/min)* mode. Operations as drilling, reaming, tapping, etc., are common on a lathe and they require the direct *r/min* in the program. To distinguish between the two alternatives in lathe programming, the choice of *surface speed* or *revolutions per minute* must be specified. This is done with preparatory commands G96 and G97, *prior to the spindle function*:

G96 S . . M03 *Surface speed selected (m/min or ft/min)*
G97 S . . M03 *Spindle speed (r/min) selected*

For milling, this distinction normally does not exist and spindle speed in r/min is always assumed.

By programming the surface speed command G96 for turning and boring, the control enters into a special mode, known as the *Constant Surface Speed* or CSS. In this mode, actual spindle revolutions will *increase* and *decrease* automatically, depending on the actual diameter being cut (current diameter). Automatic *Constant Surface Speed* feature is built in the control systems available for all CNC lathes. It is a feature that not only saves programming time, it also allows the tool to remove *constant amount of material* at all times, thus saving the cutting tool from excessive wear and creating a better surface finish.

Figure 12-7 shows a typical example, when a facing cut starts at X6.2 ($\varnothing 6.2$), and faces the part to machine centerline or slightly below. G96 S375 was used and 6000 r/min was the maximum spindle speed of the lathe.

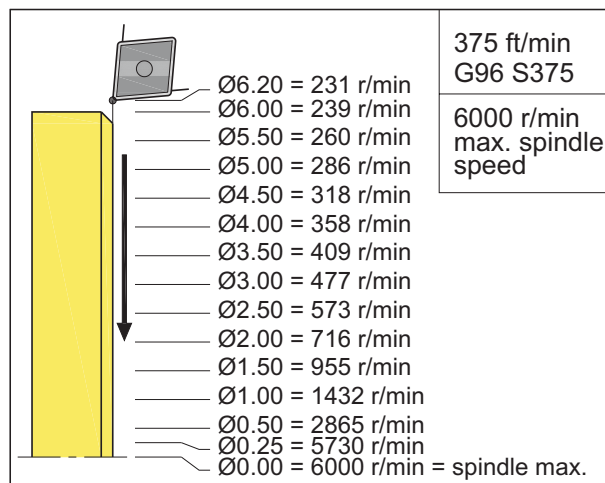


Figure 12-7

Example of a facing cut using constant surface speed mode G96

Although only selected diameters are shown in the illustration, along with their corresponding revolutions per minute, the updating process is always constant. Note the sharp increase in r/min as the tool moves closer to machine centerline. When the tool reaches X0 ($\varnothing 0.0$), the speed will be at its *maximum*, within the current gear range. As this speed may be extremely high in some cases, control system allows setting of a certain maximum, described later.

To program a surface speed for a CNC lathe, there are several options. In the following three examples, the most important ones will be examined. Gear change functions (if available) are omitted for all examples.

➡ Example 1 :

Surface speed is set right after the coordinate setting is read from *Geometry offset*:

```
N1 G20
N2 (GEOMETRY OFFSET SET TO X16.0 Z5.0) T0100
N3 G96 S400 M03
...
```

In this common application, the actual spindle speed will be based on the preset diameter of 16 inches, resulting in 95 r/min in block N3. In some cases, this will be too low. Consider another example:

➡ Example 2 :

On very large CNC lathes, geometry offset setting of the X-axis diameter is quite large, say $\varnothing 24.0$ inches. In the previous example, target diameter of the next tool motion was not important, but in this case it is. For example:

```
N1 G20
N2 (GEOMETRY OFFSET SET TO X24.0 Z5.0) T0100
N3 G96 S400 M03
N4 G00 X20.0 T0101 M08
...
```

In *Example 2*, the initial tool position is at X24.0 and the tool motion terminates at X20.0, both values are diameters. This translates to an actual motion of only 2.0 inches. At the X24.0, the spindle will rotate at 64 r/min, at X20.0 it will rotate at 76 r/min. The difference is very small to warrant any special programming. It is different, however, if the starting position is at a large diameter, but a tool moves to a much smaller target diameter.

➡ Example 3a :

From the initial position of $\varnothing 24.0$ inches, the tool will move to a rather small diameter of 2.0 inches:

```
N1 G20
N2 (GEOMETRY OFFSET SET TO X24.0 Z5.0) T0100
N3 G96 S400 M03
N4 G00 X2.0 T0101 M08
...
```

Spindle speed at the start of program (block N3) will be the same as in previous example, at 64 r/min. In the next block (N4), the speed calculated for $\varnothing 2.0$ inch will be 764 r/min, automatically calculated by the control. This rather large change in spindle speeds may have an adverse effect on some CNC lathe work. What may happen is that the cutting tool will reach the $\varnothing 2.0$ inch *before* the spindle speed fully accelerates to the required 764 r/min. The tool may start removing material at a speed much slower than intended. In order to correct the problem, data in CNC program need to be modified:

Example 3b :

Program modification takes place in block N3. Instead of programming constant surface speed mode, program direct *r/min* for the target of Ø2.0 inches, based on 400 ft/min surface speed. Actual *r/min* has to be calculated first, then the CSS setting will be programmed in a subsequent block:

```
N1 G20
N2 (GEOMETRY OFFSET SET TO X24.0 Z5.0) T0100
N3 G97 S764 M03
N4 G00 X2.0 T0101 M08
N5 G96 S400
...
```

In the example, at Ø24.0 (X24.0 offset), the actual speed would be only 64 r/min. At Ø2.0 (X2.0 in N4), the speed will be 764 r/min. Cutting tool *may* reach X2.0 position before the spindle speed has accelerated to full 764 r/min, *if it is not calculated and programmed earlier* - see block N3.

This technique is only useful if the CNC lathe does not support automatic time delay. Many modern lathes have a built-in timer, that forces the cutting tool to wait before actual cutting, until the spindle speed has fully accelerated.

Older CNC lathes used G50 position register command, and the initial position was part of the program. For example, instead of geometry offset set to X24.0 Z5.0, program would contain G50 X24.0 Z5.0. *Geometry Offset* setting is much more flexible, as it is done at the machine.

◆ Maximum Spindle Speed Setting

When CNC lathe operates in the *Constant Surface Speed* mode, the spindle speed is directly related to the current part diameter. The smaller the work diameter is, the greater the spindle speed will be. So, a natural question is - what will happen if the tool diameter is *zero*? It may seem impossible to ever program a zero diameter, but there are at least two cases when that is exactly the case.

In the first case, zero diameter is programmed for all *centerline operations*. All drilling, center drilling, tapping and similar operations are programmed at the zero diameter (X0). These operations are always programmed in the direct *r/min* mode, using G97 command. In G97 mode, the spindle speed is controlled directly, *r/min* does not change.

The second case of a zero diameter is when *facing off* a solid part all the way to the centerline. This is a different situation. For all operations at X0, the cutting diameter does not change, because a direct *r/min* is programmed. During a face cutting operation, the diameter changes all the time while material removal continues until the tool reaches the spindle centerline. No, don't reach for the formulas explained earlier. *Any calculation with a diameter in the formula being zero, will result in error!* Rest assured, there will *not* be 0 r/min at the spindle centerline - or an error - in G96 mode. Return to Figure 12-7 for illustration.

Whenever the surface speed spindle mode is active and tool reaches spindle centerline at X0, the result will normally be the *highest* spindle rotation possible, *within the active gear range*. It is paradoxical, but that is exactly what will happen. Such situation is acceptable when the part is well clamped, does not extend too far from the chuck or fixture, tool is strong and robust, and so on. When a part is mounted in a special fixture, or an eccentric setup is used, when part has a long overhang, or when some other adverse conditions are present, the maximum spindle speed at the centerline may be *too high* for operating safety!

There is a simple solution to this problem, using a programming feature available for Fanuc and other controls. Surface speed mode can be used with a *preset highest limit*, specified in *revolutions per minute*. Program function for maximum spindle *r/min* setting is normally G50 or G92 on some lathes. This maximum setting is sometimes called *maximum spindle speed clamping*. Do not confuse this G50/G92 with its other meaning, position register preset. Here is an example of G50 as a speed limiting command:

```
O1201 (SPINDLE SPEED CLAMP)
N1 G20 T0100
N2 G50 S1500 (1500 R/MIN MAX)
N3 M42 (HIGH SPINDLE RANGE)
N4 G96 S400 M03 (CSS AND 400 FT/MIN)
N5 G00 G41 X5.5 Z0 T0101 M08
N6 G01 X-0.07 F0.012 (BELOW CENTER LINE)
N7 G00 Z0.1
N8 G40 X9.0 Z5.0 T0100
N9 M01
```

What actually happens in program O1201? Block N1 selects imperial units of measurement and T01. The critical block N2 has a *simple meaning*:

G50 S1500
... means ...
Do not exceed 1500 r/min in G96 surface speed mode

Block N3 selects the spindle gear range; block N4 sets the surface speed mode, using 400 ft/min surface speed. Spindle rotation M03 is called in the same block. In block N5, the tool makes a rapid motion towards Ø5.5 and the part front face. During rapid motion, tool nose radius offset and coolant function are activated. Spindle speed at Ø5.5 will be 278 r/min, using formula described earlier in this chapter. Next block N6 is the actual facing cut. At the cutting feedrate of 0.012 in/rev, the tool tip faces off the blank part to the centerline. In reality, the cut end point is programmed on the other side of spindle center line (X-0.07).

Tool nose point radius size must be taken into consideration when programming with the tool nose radius offset (G41/G42) and to the machine centerline. A special section later explains what exactly will happen during this cut - see page 278 for further details.

Block N7 moves the tool tip 0.100 inches away from the face, at a rapid rate. In the remaining two blocks, the tool will rapid to the indexing position with a cancellation of radius offset in N8 and an optional program stop is provided in block N9.

Now, think of what happens in critical blocks N5 and N6. Spindle will rotate at the speed of 278 r/min at the Ø5.5. Since the CSS mode is in effect, as the tool tip faces off the part, its diameter is becoming smaller and smaller while the *r/min is constantly increasing*.

Without the maximum spindle speed limit in block N2, the spindle speed at the centerline will be equivalent to the *maximum r/min* available within M42 gear range. A typical range may be 4000 r/min or even higher.

With the preset maximum spindle speed limit of 1500 r/min (G50 S1500), the spindle will be constantly increasing its speed, but only *until* it reaches the 1500 preset r/min, then it will *remain at that speed* for the rest of cut.

At the control, CNC operator can easily change the maximum limit value, to reflect true setup conditions or to optimize cutting values.

Spindle speed is preset - or *clamped* - to the maximum *r/min* setting, by programming the S function together with the G50 preparatory command. If the S function is in a block not containing G50, the control will interpret it as a new spindle speed (CSS or r/min), that will be active from that block on. *This error may be very costly!*

Use caution when presetting maximum r/min of the spindle !

Maximum spindle speed can be clamped in a separate block or in a block that also includes the current tool coordinate setting. Typically, the combined setting is useful at the beginning of a tool, the separate block setting is useful if the need arises to change the maximum spindle speed in the middle of a tool, for instance, between facing and turning cuts using the same tool.

To program G50 command as a separate block, anywhere in the program, just issue the preparatory command combined with the spindle speed preset value. Such a block will have *no effect* whatsoever on any active coordinate setting, it represents just another meaning of G50 command. The following examples are all correct applications of G50 command for both, the old style coordinate setting and/or the maximum spindle speed preset:

N12 G50 X20.0 Z3.0 S1500 *Double meaning*

N38 G50 S1250 *Single meaning*

N15 G50 X8.5 Z2.5 *Single meaning*

N40 G50 Z4.75 S700 *Double meaning*

If the CNC lathe supports old G92 instead of G50, keep in mind that they have exactly the same meaning and purpose. On old controls, G50 command is more common than G92 command but programming method is the same.

◆ Part Diameter Calculation in CSS

Often, knowing at *what diameter* the spindle will actually be clamped can be a very useful information. Such knowledge may influence preset value of spindle speed clamping. To find out at what diameter the *Constant Surface Speed* will remain fixed, the formula that finds the r/min at a given diameter must be reversed:

$$D = \frac{12 \times \text{ft} / \text{min}}{\pi \times \text{r} / \text{min}}$$

where ...

D = Diameter where CSS stops (in inches)
 12 = Multiplying factor - feet to inches conversion
 ft/min = Active surface speed - feet per minute
 π = Constant 3.1415927
 r/min = Preset maximum spindle speed

➡ Example - Imperial units :

If the preset value in a program is G50 S1000 and the surface speed is set as G96 S350, the CSS will be clamped when it reaches the Ø1.3369 inches:

$$\begin{aligned} D &= (12 \times 350) / (\pi \times 1000) \\ &= 1.3369015 \\ &= \text{Ø}1.3369 \end{aligned}$$

The formula may be shortened, as before:

$$D = \frac{3.82 \times \text{ft} / \text{min}}{\text{r} / \text{min}}$$

Formulas based on the imperial system, can be adapted to a metric environment:

$$D = \frac{1000 \times \text{m} / \text{min}}{\pi \times \text{r} / \text{min}}$$

where ...

D = Diameter where CSS stops (in mm)
 1000 = Multiplying factor - meters to mm conversion
 m/min = Active surface speed - meters per minute
 π = Constant 3.1415927
 r/min = Preset maximum spindle speed

Just like in its imperial version counterpart, metric formula may be shortened as well:

$$D = \frac{318.3 \times \text{m / min}}{\text{r / min}}$$

➡ Example - Metric units :

If the preset value in a program is G50 S1200 and the surface speed is set as G96 S165, the CSS will be clamped when it reaches the Ø43.768 mm:

$$\begin{aligned} D &= (1000 \times 165) / (\pi \times 1200) \\ &= 43.767609 \\ &= \text{Ø}43.768 \text{ mm} \end{aligned}$$

◆ Cutting Speed Calculation

Constant Surface Speed (CSS) - cutting speed - is required for virtually all turning and boring operations on a CNC lathe. It is also the basic source of cutting data, from which the spindle speed is calculated for virtually all machining center operations. Now, consider a very common scenario:

CNC operator has optimized the current cutting conditions, including the spindle speed, so they are very favorable. Can these conditions be applied to subsequent jobs?

Yes, they can - and should - provided that certain critical requirements will be satisfied:

- Machine and part setup are equivalent
- Cutting tools are equivalent
- Material conditions are equivalent
- Other common conditions are satisfied

If these requirements are met, the most important source data is the spindle speed actually used during machining. Once the spindle speed is known, the cutting speed (CSS) can be calculated and used for any other tool diameter, providing the requirements above are met.

In a nutshell, the whole subject can be quickly summed up by categorizing it as a *cutting speed* calculation - calculation of *Constant Surface Speed (CSS)*, when the tool or part diameter *and* the spindle speed are known.

From there on, it is a simple matter of formulas:

To calculate *Cutting Speed* in imperial units:

$$\text{ft / min} = \frac{\pi \times D \times \text{r / min}}{12}$$

➡ EXAMPLE:

A Ø5/8 inch drill works very well at 756 r/min - what is its cutting speed in ft/min?

$$\text{ft/min} = (3.14 \times 0.625 \times 756) / 12 = 123.64$$

To calculate *Cutting Speed* in metric units:

$$\text{m / min} = \frac{\pi \times D \times \text{r / min}}{1000}$$

➡ EXAMPLE:

A Ø7 mm end mill works very well at 1850 r/min - what is its cutting speed in m/min?

$$\text{m/min} = (3.14 \times 7 \times 1850) / 1000 = 40.66$$

The major benefit of using this method is a significant reduction of time spent at the CNC machine, usually required to find and 'fine-tune' the optimum spindle speed during setup or part optimization. Knowing when a particular cutting speed can be applied is one of several optimizing methods available to CNC programmers.

13

FEEDRATE CONTROL

Feedrate is the closest programming companion to spindle function. While the spindle function controls spindle speed and spindle rotation direction, feedrate controls *how fast the tool will move*, usually to remove excessive material (stock). In this handbook, subject of *rapid positioning*, sometimes called a *rapid motion* or *rapid traverse motion*, is not considered a true feedrate and will be described separately, starting on page 147.

FEEDRATE CONTROL

Cutting feedrate is the speed at which the cutting tool removes the material by cutting action

A cutting action may be a rotary tool motion (drilling and milling, for example), rotary motion of the part (lathe operations), or other action (flame cutting, laser cutting, water jet, electric discharge etc.). Feedrate function is used in CNC program to select correct feedrate amount, suitable for the desired action.

Two feedrate types are used in CNC programming:

- Feedrate per minute - mm/min *or* in/min
- Feedrate per revolution - mm/rev *or* in/rev

The most common types of CNC machines, machining centers and lathes, can be programmed in either feedrate mode. In practice, it is much more common to use the *feedrate per minute* on machining centers and the *feedrate per revolution* on lathes.

There is a significant difference in G-codes used for machining centers and lathes.

FEEDRATE	Milling	Turning Group A	Turning Group B	Turning Group C
Per minute	G94	G98	G94	G94
Per revolution	G95	G99	G95	G95

G-code **Group A** is the most commonly used on Fanuc controls and also in this publication

Another type of special feedrate is called the *inverse time feedrate*. It is used in some rotary applications only.

FEEDRATE FUNCTION

Calling address for a feedrate word in a program is the address *F*, followed by a number of digits. The number of digits following address *F* depends on the feedrate mode selection and machine tool application. Decimal place is normally allowed for feedrate programming.

◆ Feedrate per Minute

For milling applications, all cutting feedrate in linear and circular interpolation modes is programmed in *millimeters per minute (mm/min)* or in *inches per minute (in/min)*. The amount of feedrate is the distance a cutting tool will travel in one minute (60 seconds). This value is modal and is canceled only by another F-address word. The main advantage of feedrate per minute is that it is *not* dependent on current spindle speed. That makes it very useful in milling operations, using a large variety of tool diameters. Standard abbreviations for feedrate per minute are:

- Millimeters per minute ... mm/min
- Inches per minute ... in/min (*or* older ipm / IPM)

The most typical format for feedrate per minute is F5.1 for the metric system and F5.1 for the imperial system.

For example, feedrate of 15.5 inches per minute, will be programmed as F15.5. In metric system, feedrate amount of 250 mm/min will appear in the program as F250.0. A slightly different programming format may be expected for special machine designs.

One important item to remember about feedrate is the *range* of available feedrate values. Feedrate range of the control system *always exceeds* that of machine servo system. For example, feedrate range of a CNC machine with Fanuc CNC system is typically described in metric units, for example, 30000 mm/min may be the maximum for a machine with 60000 mm/min rapid motion. Minimum cutting feedrate depends on the machine builder - it could be 0.1 mm/min or 0.1 in/min or a totally different amount.

In milling, programming command (G-code) for *feedrate per minute* is G94. For most machines, it is set automatically, by the system default and does not have to be written in the program. For lathe operations, feedrate per minute is used very seldom. In *Group A*, G-code for feedrate per minute is G98, for *Groups B* and *C* it is G94. CNC lathes use primarily *feedrate per revolution* mode.

◆ Feedrate per Revolution

For CNC lathe work, feedrate is not measured in terms of time, but as the actual distance the tool travels in one spindle revolution (rotation). This *feedrate per revolution* is common on lathes (G99 for Group A). Its value is modal and another feedrate function cancels it (usually the G98). Lathes can also be programmed in *feedrate per minute* mode (G98), to control feedrate when the spindle is stationary. Two standard abbreviations are used for *feedrate per revolution*:

- Inches per revolution in/rev (or older ipr)
- Millimeters per revolution mm/rev

The most programming typical format for feedrate per revolution is *three* decimal places in *metric* system, and *four* decimal places in *imperial* system. Metric feedrate example of 0.42937 mm/rev will be programmed as F0.429 on most controls. In imperial units, a feedrate of 0.083333 in/rev will be applied in the CNC program as F0.0833 on most controls. Many modern control systems accept feedrate of up to five decimal places for metric units and six decimal for imperial units.

Be careful when rounding feedrate values. For turning and boring operation, reasonably rounded feedrates are quite sufficient. Only in tapping and single point threading the feedrate precision is critical for a proper thread lead, particularly for long or very fine threads. Some Fanuc controls can be programmed with up to six decimal places feedrate precision for threading only.

Programming command for feedrate per revolution is G99. For most lathes, this is the system default, so it does not have to be written in the program, unless the opposite command G98 is also used.

It is relatively more common to program a *feedrate per minute* (G98) for a CNC lathe program, than it is to program a *feedrate per revolution* (G99) in a milling program. The reason is that on a CNC lathe, this command controls feedrate while the spindle is *not* rotating. For example, during a barfeed operation, a part stopper is used to 'push' the bar to a precise position in the chuck or a collet, or a pull-put finger to 'pull' the bar out. Rapid feed would be too fast and feedrate per revolution is not applicable. Feedrate per minute is used instead. In cases like these, G98 and G99 commands are used in a lathe program as required. Both commands are modal and one cancels the other.

FEEDRATE SELECTION

To select the best feedrate, one that is most suitable for a given job, some general knowledge of machining is useful. This is an important part of programming process and should be done carefully. A feedrate selection depends on many factors, most notably on:

- Spindle speed - in r/min - revolutions per minute
- Tool diameter [M] or tool nose radius [T]
- Surface requirements of the part
- Cutting tool geometry
- Machining forces
- Part setup
- Tool overhang (extension)
- Cutting length motion
- Amount of material removal (depth or width of cut)
- Method of milling (climb or conventional)
- Number of flutes in material (for milling cutters)
- Safety considerations

The last item is safety - always a programming responsibility number one, to assure the safety of people and equipment. Safe speeds and feeds are only two aspects of much wider safety awareness in CNC programming.

ACCELERATION AND DECELERATION

During a typical contouring operation, the direction of cutting motion is changed quite often. There is nothing unusual about it, with all the intersections, tangency points and clearances. In contouring, it means that in order to program a sharp corner on a part, the tool motion along X-axis in one block will have to change into a motion along Y-axis in the next block. To make the change from one cutting motion to another, control must *stop the X-motion* first, then *start the Y-motion*. Since it is impossible to *start* at a full feedrate instantly, without *acceleration*, and equally impossible to *stop* feedrate without *deceleration*, a possible cutting error may occur. This error may cause sharp corners on the part contour to be cut with an undesirable overshoot, particularly during very high feedrates or extremely narrow angles. It only occurs during a cutting motion in G01 mode, as well as in G02 and G03 modes, but *not* in rapid motion mode G00. During rapid motion, motion deceleration is automatic - and away from the part.

In routine CNC machining, there is a small chance of ever encountering such an error. Even if the error is present, it will likely be well within tolerances.

If the error does need correction, Fanuc controls provide two commands that can be used to correct the problem:

G09	Exact stop	(one block only)
G61	Exact stop mode	(modal)

Exact stops increase the cycle time. For programs used on older machines, they may be required in some cases.

◆ Exact Stop Command

The first of two commands that control the feedrate when machining around corners is G09 command - *Exact Stop*. This is an *unmodal* command and has to be repeated in every block, whenever it is required.

In program example O1301, there is no provision for acceleration and deceleration. That may cause uneven corners, due to the rather high feedrate of F90.0 (in/min):

O1301 (NORMAL CUTTING)

```
...
N13 G00 X15.0 Y12.0
N14 G01 X19.0 F90.0
N15 Y16.0
N16 X15.0
N17 Y12.0
...
```

By adding G09 exact stop command in the program, the motion in that block will be fully completed *before* the motion in the other axis will start - O1302:

O1302 (G09 CUTTING)

```
...
N13 G00 X15.0 Y12.0
N14 G09 G01 X19.0 F90.0
N15 G09 Y16.0
N16 G09 X15.0
N17 Y12.0
...
```

Example O1302 guarantees a sharp corner at all three positions of the part. If only one corner is critical for sharpness, program the G09 command in the block that terminates at that corner - O1303:

O1303 (G09 CUTTING)

```
...
N13 G00 X15.0 Y12.0
N14 G01 X19.0 F90.0
N15 G09 Y16.0
N16 X15.0
N17 Y12.0
...
```

G09 command is useful only if a handful of blocks require deceleration for a sharp corner. For a program where all corners must be precise, constant repetition of G09 is not very efficient.

◆ Exact Stop Mode Command

The second command that corrects an error at sharp corners is G61 - *Exact Stop Mode*. It is much more efficient than G09 and functions identically. The major difference is that G61 is a *modal* command that remains in effect until it is canceled by G64 cutting mode command. G61 shortens programming time, but not the cycle time. It is most useful when G09 would be repeated too many times in the same program, making it unnecessarily too long - O1304.

O1304 (G61 CUTTING)

```
...
N13 G00 X15.0 Y12.0
N14 G61 G01 X19.0 F90.0
N15 Y16.0
N16 X15.0
N17 Y12.0
N18 G64
...
```

Note that program example O1304 is identical in results to program O1301. In both cases, exact stop check applies to *all* cutting motions - unmodally in program O1301, modally in program O1304. Also note an additional block - N18. It uses G64 command - *normal cutting mode*. Normal cutting mode is the default setting when machine power is turned on and is not usually programmed. *Figure 13-1* illustrates tool motion *with* and *without* G09/G61 command. The large overshoot amount shown is exaggerated only for the illustration, in reality it is very small.

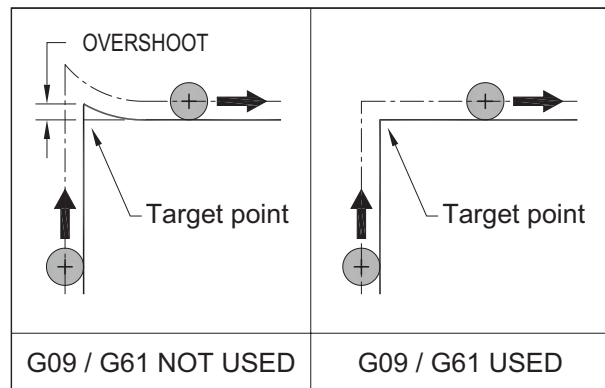


Figure 13-1

Feedrate control around corner - Exact Stop commands
The overshoot is exaggerated for clarity

◆ Automatic Corner Override

While a cutter radius offset is in effect for milling cutters, feedrate at the contour change points is normally *not* overridden. In a case like this, the preparatory command G62 can be used to automatically override cutting feedrate at the corners of a part. This command is active until G61 command (exact stop check mode), G63 command (tapping mode), or G64 command (cutting mode) is programmed.

◆ Tapping Mode

Programming in tapping mode G63 will cause the control system to ignore any settings of *feedrate override switch*, except the 100% setting. It will also cancel function of the feedhold key, located on control panel. Tapping mode will be canceled by programming G61 command (exact stop check), or G62 command (automatic corner override mode selection), or G64 command (cutting mode selection).

◆ Cutting Mode

When cutting mode G64 is programmed or is active by the system default, it represents *normal* cutting mode. When this command is active, exact stop check G61 will *not* be performed, neither will automatic corner override G62 or tapping mode G63. That means motion acceleration and deceleration will be done normally (as per control system) and feedrate overrides *will be* effective. This is the most common default mode for a typical control system.

Cutting mode can be canceled by programming the following commands: G61 (exact stop mode), G62 (automatic corner override mode) or G63 (tapping mode).

G64 command is not usually programmed, unless one or more of other feedrate modes are used in the same program. To better understand G62 and G64 modes, compare illustrations in Figure 13-2.

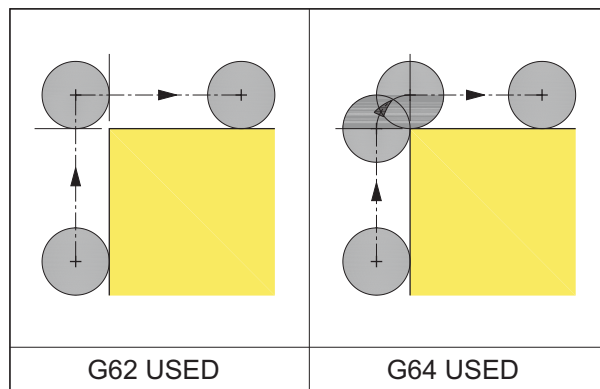


Figure 13-2

Corner override mode G62 and default G64 cutting mode

CONSTANT FEEDRATE

Beginning on page 243, the main topic is *circular interpolation*. In this chapter are detailed explanations and examples of maintaining a constant cutting feedrate for inside and outside arcs, from the *practical* point of view. At this point, the main focus is on actual *understanding* of the constant feedrate, rather than its *application*.

In programming, normal process is to calculate coordinate values for all contour change points, based on the part *drawing*. Cutter radius that produces the *centerline* of a tool path is typically disregarded. When programming arcs to drawing dimensions, rather than to the cutter centerline, feedrate applied to the programmed arc always relates to the *programmed* radius, *not* the actual radius cut at the cutter center.

When cutter radius offset is active and the arc tool path is offset by cutter radius, the *actual* arc radius that is cut can be either *smaller* or *larger*, depending on the preset offset amount for cutting tool motion.

It is very important to understand that *effective* cutting radius will *decrease* in size for all internal arcs and it will *increase* in size for all external arcs. Since the cutting feedrate does not change automatically during cutter radius offset mode, *it must be adjusted* in the program. Usually, this adjustment is not necessary, except in cases where the surface finish is of great importance or cutter radius is very large. This consideration applies only to circular motions, not to linear cutting.

◆ Circular Motion Feedrates

Setting feedrates for circular motions is generally the same as for linear feedrates. In fact, most programs do not change feedrate for linear and circular tool motions. If part surface finish is important, so called 'normal' feedrate must be adjusted *higher* or *lower*, with consideration of the cutter radius, the type of radius cutting (outside or inside arc) and cutting conditions. The larger the cutter radius, the more reason the cutting feedrate for programmed arcs will need some correction.

In case of arc cutting, the equidistant tool path - after applying cutter radius offset - may be *much larger* or *much smaller* than the arc programmed to drawing dimensions.

Feedrate for compensated arc motions is always based on the currently programmed linear motion feedrate. Look for a more detailed explanation on page 253, with an illustration and examples. First, here is the standard formula for calculating a linear feedrate:

$$F_l = r / \text{min} \times F_t \times n$$

where ...

- F_l = Linear feedrate (in/min **or** mm/min)
- r/min = Spindle speed (revolutions per minute)
- F_t = Feedrate per tooth (cutting edge)
- n = Number of cutting edges (flutes or inserts)

Based on the linear feedrate formula, arc feedrate adjustments are influenced by the *side* of machined arc - *outside* or *inside* arc. Linear feedrate should be *increased* for outside arcs and *decreased* for inside arcs.

For *outside* arcs, the feedrate is generally adjusted *upwards*, to a higher value:

$$F_o = \frac{F_l \times (R + r)}{R}$$

where ...

- F_o = Feedrate for outside arc
- F_l = Linear feedrate (normal feedrate for the tool)
- R = Outside radius of part
- r = Cutter radius

For *inside* arcs, the feedrate is generally adjusted *downwards*, to a lower value:

$$F_i = \frac{F_l \times (R - r)}{R}$$

where ...

- F_i = Feedrate for inside arc
- F_l = Linear feedrate (normal feedrate for the tool)
- R = Inside radius of part
- r = Cutter radius

MAXIMUM FEEDRATE

Maximum *programmable cutting feedrate* for CNC machines is determined by the machine manufacturer, not the control manufacturer. For example, maximum cutting feedrate on a particular machine may be 10000 mm/min, although the CNC system can support a feedrate several times greater. This is applicable to all controls, but there are additional programming considerations for CNC lathes, where *feedrate per revolution* is the main method of programming motion velocity of a cutting tool.

◆ Maximum Feedrate Considerations

Maximum *cutting* feedrate per revolution is always restricted by the *programmed spindle speed (r/min)* and *maximum rapid traverse rate* of a CNC lathe. It is quite easy to program feedrate per revolution too high without even realizing it. This problem is most common in single point threading, where cutting feedrates can be unusually high.

A CNC machine cannot deliver heavier feedrates than the maximum amount it was designed for - threading results would be unacceptable. When unusually heavy feedrates and fast spindle speeds are combined in the *same* program, it is advisable to check whether the active feedrate does not exceed the maximum feedrate allowed on a given machine. It can be calculated as the *maximum feedrate per revolution*, according to the following formula:

$$F_{\max} = \frac{R_{\max}}{r / \min}$$

where ...

- F_{\max} = Maximum feedrate per revolution (units/rev)
- R_{\max} = Lower of the maximum feedrate, selected between X and Z axes
- r/\min = Spindle speed in revolutions per minute

R_{\max} is set either in *mm/min* or *in/min*, depending on the input units selected. Page 359 covers more details relating to feedrate limits for threading.

FEEDHOLD AND OVERRIDE

While running a program, the programmed feedrate may be temporarily suspended or changed by using one of two available features of the control system. One is called a *feedhold* switch, the other is a *feedrate override* switch. Both switches are standard and allow the CNC operator to manually control programmed feedrate during program execution. They are located on the machine operation panel.

◆ Feedhold Switch

Feedhold is a push button that can be toggled between *Feedhold ON* and *Feedhold OFF* modes. It can be used for both feedrate modes, *feedrate per minute* or *feedrate per revolution*. On many controls, feedhold will stop not only a cutting feed with G01, G02, G03 in effect - it will also stop rapid motion G00. Other program functions will remain active during a feedhold state.

For certain machining operations, feedhold function is *automatically disabled* and becomes ineffective. This is typical for tapping and threading, using G84 and G74 tapping cycles on machining centers and threading operations using the G32, G92 and G76 on CNC lathes.

◆ Feedrate Override Switch

Feedrate override is normally controlled by means of a special rotary switch, located on the control panel of the CNC unit - Figure 13-3.

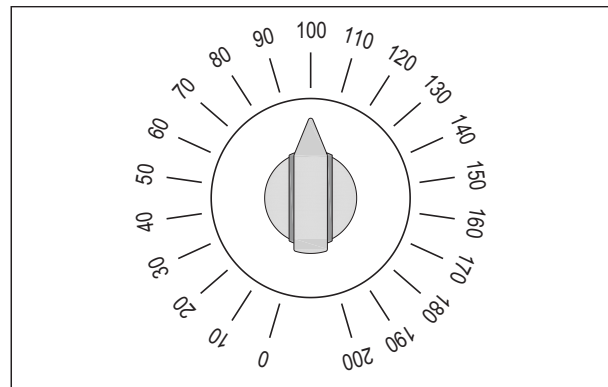


Figure 13-3

Typical feedrate override switch

This rotary switch has marked settings or divisions, indicating the *percentage of programmed feedrate*. A typical range of a feedrate override is 0 to 200% (or 0 to 150%), where 0 may be no motion at all or the slowest motion, depending on the machine design. 200% setting *doubles* all programmed feedrates. Programmed feedrate of 12.0 in/min (F12.0) is 100% feedrate. If the override switch is set to 80%, the actual cutting feedrate will be 9.6 in/min. If the switch is set to 110%, the actual cutting feedrate will be 13.2 in/min.

This simple logic applies to metric system as well. If the programmed feedrate is 300 mm/min, it becomes 100%. An 80% feedrate override results in 240 mm/min cutting feedrate and a 110% feedrate override setting is equivalent to 330 mm/min for the cutting tool.

Feedrate override switch works equally well for lathes and *feedrate per revolution*. For example, the programmed feedrate of 0.014 in/rev will result in the actual feedrate of 0.0126 in/rev with 90% feedrate override and 0.0182 in/rev with 130% feedrate override. If a very precise feedrate per spindle revolution is required, be careful with override settings. For example, a programmed feedrate is F0.012, in *inches per revolution*. A change by a single division on the override dial will either *increase* or *decrease* the programmed amount by a full 10 percent. Therefore, the feedrate will be 0.0108 at 90%, 0.0120 at 100%, 0.0132 at 110%, etc. In most cases, such precise feedrate is not required, but keep in mind that some feedrates will not be accessible, for example, a feedrate of 0.0115 in/rev, because of the fixed 10% increments on the override switch.

In single point threading mode G32, the feedrate override switch is *disabled*. Feedrate override is also disabled for tapping cycles G84 and G74 on machining centers, and for single point threading cycles G92 and G76 on lathes. If the tapping mode is used for milling systems, with the command G63, both *feedrate override* and *feedhold* functions are disabled - *through the program* !

Control system offers two feedrate override functions for cutting motions *other than* tapping or threading cycles. They are M48 and M49. These are *programmable* functions, but may not be available for all controls.

◆ Feedrate Override Functions

Although the feedrate function uses address F, two special miscellaneous functions M can be used in a program to set the *feedrate override ON* or *OFF*. On the machine operation panel, a switch is provided for feedrate override. If the CNC operator decides that programmed feedrate has to be temporarily *increased* or *decreased*, this switch is very handy. On the other hand, during machining operations, where the cutting feedrate *must* be used as programmed, the override switch has to be set to 100% only, not to any other setting.

A good example are special tapping operations *without* cycles, using G01 and G00 preparatory commands. Functions M48 and M49 are used precisely for such purposes:

M48	Feedrate override cancel function is OFF, which means feedrate override is <i>active</i>
M49	Feedrate override cancel function is ON, which means feedrate override is <i>inactive</i>

M48 function enables the CNC operator to use the feedrate override switch freely; M49 function will cause feedrates to be executed *as programmed*, regardless of the feedrate override switch setting on the machine panel. The most common usage of these two functions is for tapping or threading *without* a cycle, where the exact programmed feedrate *must* always be maintained. The following example shows a typical programming method:

```

N10 S500 M03                                (USING TAP 12 TPI)
...
N14 G00 X5.0 Y4.0 M08
N15 Z0.25
N16 M49                                (DISABLE FEEDRATE OVERRIDE)
N17 G01 Z-0.625 F41.0 M05
N18 Z0.25 M04
N19 M48                                (ENABLE FEEDRATE OVERRIDE)
N20 G00 X. . Y. . M05
N21 M03
...

```

The tapping motion occurs between blocks N16 and N19 and the feedrate override is disabled for these blocks.

E ADDRESS IN THREADING

Some CNC lathes with older controls (like Fanuc 6T) use feedrate address *E* for threading, rather than the more common address *F*.

Feedrate function *E* is similar to the *F* function. It also specifies the thread lead as *feedrate per revolution*, in mm/rev or in in/rev, but it has a *greater decimal place accuracy*. On older Fanuc control system model 6T, for example, the range for thread lead is:

➡ English - Fanuc 6T control :

```

F = 0.0001      to      50.0000 in/rev
E = 0.000001    to      50.000000 in/rev

```

➡ Metric - Fanuc 6T control :

```

F = 0.001       to      500.000 mm/rev
E = 0.0001      to      500.0000 mm/rev

```

On the newest control models, FS-0T to Fanuc 30T, the ranges are similar (there is no E address), but the safest way to find the available ranges is to lookup the specifications for your control system.

The *E* address is redundant on all newer controls and is retained only for compatibility with older programs that may be used on machines equipped with newer controls. The available threading feedrate ranges vary between different control systems, and depends on the type of feed screw and the input units used in the program. Virtually all controls now allow F-address with extended precision, when applied to threading.

14

TOOL FUNCTION

Each numerically controlled machine using an automatic tool changer must have a special tool function (T-function) that can be used in the program. This function controls the behavior of the *cutting tool*, depending on the type of machine. There are noticeable differences between T-functions used on CNC machining centers and those used on CNC lathes. There are also differences between similar controls for the same machine type. The normal programming address for the tool function uses the address T.

For CNC machining centers, T-function typically controls the *tool number* only. For CNC lathes, the function controls indexing to the *tool station number*, as well as the *tool offset number*.

T-FUNCTION FOR MACHINING CENTERS

All vertical and horizontal CNC machining centers have a feature called the *Automatic Tool Changer*, abbreviated as *ATC*. In the program or MDI mode on the machine, this feature uses the function T, where the T address refers to a tool number selected by the programmer. The subsequent digits describe the tool number itself. On CNC machines with a manual tool change, the tool function may not be required at all.

Before programming for a particular CNC machining center begins, the *type of the tool selection* for that machine must be known. There are two major types of tool selection used in automatic tool change process:

- Fixed type
- Random memory type

To understand the difference between them, the first step is to understand the general principles of tool storage and tool selection, available for many modern CNC machining centers.

◆ Tool Storage Magazine

A typical CNC machining center (vertical or horizontal) is designed with a special *tool magazine* (sometimes called a *tool carousel*), that contains all tools required by the program. This magazine is not a permanent storage for the tools, but many machine operators keep the commonly used tools there at all times, if possible. A typical 20-tool magazine is illustrated in Figure 14-1.

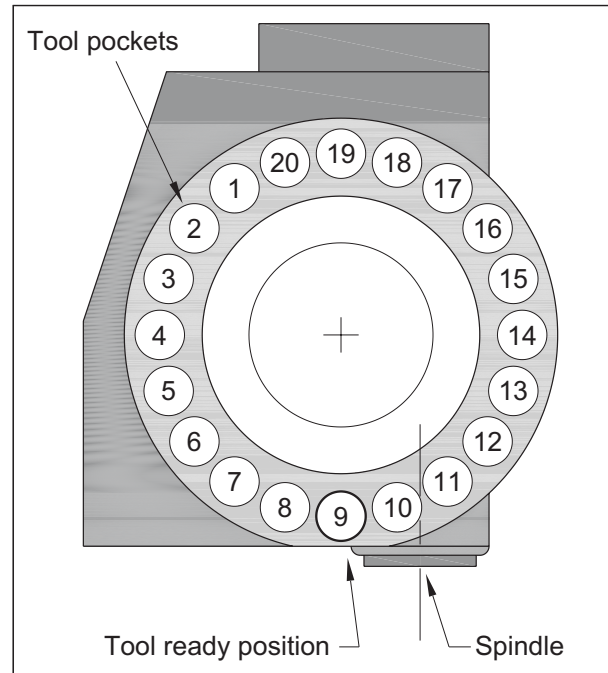


Figure 14-1
Typical side view of a 20-tool magazine

The capacity of such a magazine can be as small as ten or twelve tools and as high as several hundred tools on special machines. Typical medium size machining center may have between twenty and forty tools, larger machines will have more. Magazine is usually round or oval (larger capacity will be shaped in a zigzag form). It consists of a certain number of *pockets* - or *pots* - where the tool holder with a cutting tool is placed during setup. Each pocket is numbered in consecutive order. It is important to know that *pocket numbers are fixed* for each pocket. Magazine can be operated manually during setup, as well as automatically, through CNC program or MDI. The number of magazine pockets is the maximum number of tools that can be changed automatically on that machining center.

Within the travel of tool magazine is one special position, used for automatic tool change. This position is aligned with the tool changer and is commonly called the *waiting position*, the *stand-by position*, the *tool-ready position*, or just the *tool change position*.

◆ Fixed Tool Selection

A machining center that uses fixed tool selection requires the CNC operator to place all tools into magazine pockets that *match the tool numbers*. For example, tool number 1 (called as T01 in the program) *must* be placed into the magazine pocket number 1, tool number 7 (called as T07 in the program) must be placed into the magazine pocket number 7, and so on.

Magazine pocket is mounted on a side of the CNC machine, usually away from the work area (work cube). With fixed tool selection, the control system has no way of determining which tool number is in which magazine pocket number at any given time. CNC operator has to match the tool numbers with magazine pocket numbers during setup. This type of a tool selection is commonly found on many older CNC machining centers, or on some inexpensive machining centers, used for light production only.

Tool programming is quite easy - whenever T-function number is used in the program, that will be the tool number selected during a tool change. For example,

```
N67 T04 M06
```

or

```
N67 M06 T04
```

or

```
N67 T04
N68 M06
```

simply means to bring tool number 4 into the spindle (the last method is preferred). What will happen to the tool that is in the spindle at that time? M06 tool change function will cause the active tool to return to the magazine pocket it came from, before any new tool will be loaded. Usually, the tool changer takes the shortest way to select a new tool.

Today, this type of a tool selection is considered impractical and costly in a long run. There is a significant time loss during tool changes, because the machine tool has to wait until the selected tool is found in the magazine and placed into the spindle. Programmer can somewhat improve the efficiency by selecting tools and assigning tool numbers carefully, not necessarily in the order of usage. Examples in this handbook are based on a more modern type of the tool selection, called the *random memory*.

◆ Random Memory Tool Selection

This feature is most common on modern machining centers. It also stores all tools required to machine a part in the tool magazine pockets, away from machining area. CNC programmer identifies each tool by a T number, usually in the order of usage. Calling the required tool number by the program will physically move the tool to the *waiting* posi-

tion within tool magazine. This can happen simultaneously, while the machine is using another tool to cut a part. Actual tool change can take place anytime later. The is the concept of *next tool waiting* - where the T-function refers to the *next tool*, not the current tool. In the program, the next tool can be made ready (stand-by) by programming a few simple blocks:

```
T04                                (MAKE TOOL 4 READY)
...
<... Machining with previous tool ...>
...
M06                                (ACTUAL TOOL CHANGE - T04 IN SPINDLE)
T15                                (MAKE NEXT TOOL READY)
...
<... Machining with tool 4 - T04 ...>
...
```

In the first block, tool T04 was called into the waiting station of tool magazine, while the previous tool was still cutting. When machining has been completed, actual tool change will take place, where T04 will become the active tool. Immediately, the CNC system will search for the next tool (T15 in the example) and places it into the waiting position, while T04 is cutting.

This example illustrates that the T-function will *not* make any physical tool change at all. For that, the *automatic tool change function* - M06 - also described later in this section, is needed and must be programmed.

Do not confuse the meaning of address T used with fixed tool selection and the same address T used with random tool selection. The former refers to actual *number of magazine pocket*, the latter refers to *tool number of the next tool*. Tool call is programmed earlier than it is needed, so the control system can search for that tool while another tool is doing productive work.

◆ Registering Tool Numbers

Computers in general, and CNC systems in particular, can process given data very quickly and with the utmost precision. For CNC work, all required data must be input first, to make the computer work in our favor. In random tool selection method, CNC operator is free to place *any tool* into *any magazine pocket*, as long as the actual setting is registered into the CNC unit, in form of control system parameters. There is no need to worry too much about system parameters, just accept them as the collection of various system settings. Registering tool numbers has its own entry screen.

During machine setup, CNC operator will place the required tools into magazine pockets, writes down the numbers (which tool number is in which pocket number), and registers this information into the system. Such an operation is a normal part of the machine tool setup and various shortcuts can be used.

◆ Programming Format

Programming format for the T-function used on milling systems depends on the maximum number of tools available for the CNC machine. Most machining centers have number of available tools under 100, although very large machines will have more tool magazines available (even several hundred). In the examples, two-digit tool function will be used, covering tools within a range of T01 to T99.

In a typical program, T01 tool command will call the tool identified in the setup sheet or a tooling sheet as tool number 1; T02 will call tool number 2, T20 will call tool number 20, etc. Leading zeros for tool number designation may be omitted, if desired - T01 can be written as T1, T02 as T2, etc. Trailing zeros *must* always be written, for example, T20 must be written as T20, otherwise the system will assume the leading zero and call the tool number 2 (T2 equals to T02, not T20).

◆ Empty Tool or Dummy Tool

Often, an empty spindle, free of any tool, is required. For this purpose, an empty tool station has to be assigned. Such a tool will also have to be identified by a unique number, even if no physical tool is used. If a magazine pocket or the spindle contains no tool, an *empty* tool number is necessary for maintaining the continuity of tool changes from one part to another. This nonexistent tool is often called the *dummy* tool or the *empty* tool.

The number of an empty tool should be selected as higher than the maximum number of tools. For example, if a machining center has 24 tool pockets, the *empty* tool should be identified as T25 or higher. It is a good practice to identify such a tool by the largest number within the T-function format. For example, with a two digit format, the *empty* tool should be identified as T99, with a three digit format as T999. This number is easy to remember and is visible in the program.

As a rule, do not identify the *empty* tool as T00 - all tools *not assigned* may be registered as T00. There are, however, machine tools that do allow the use of T00, without possible complications.

TOOL CHANGE FUNCTION - M06

Tool function T, as applied to CNC machining centers, will *not* cause the actual tool change - miscellaneous function M06 must be used in the program to do that. The purpose of tool change function is to exchange the tool in the spindle with the tool in the waiting position. The purpose of T-function for milling systems is to rotate the magazine and place the selected tool into the waiting position, where actual tool change can take place. This *next tool search* happens while the control processes blocks following the T function call.

➡ Example :

```
N81 T01 ... makes T01 ready = loaded in the waiting position
N82 M06 ... brings T01 into the spindle
N83 T02 ... makes T02 ready = loaded in the waiting position
```

The three blocks appear to be simple enough, but let's explore them anyway. In block N81, tool addressed as T01 in the program will be placed into the waiting position. The next block, N82, will activate the actual tool change - tool T01 will be placed into the spindle, ready to be used for machining. Immediately following the actual tool change is T02 in block N83. This block will cause the control system to search for the next tool, T02 in the example, to be placed into the waiting position. The search will take place *simultaneously* with the program data following block N83, usually a tool motion to the cutting position at the part. There will be no time lost, on the contrary, this method assures that the tool changing times will be always the same (so called *chip-to-chip time*).

Some programmers prefer to shorten the program somewhat by programming the tool change command *together* with the next tool search in the same block. This method saves one block of program for each tool:

```
N81 T01
N82 M06 T02
```

The results will be identical - the choice is personal.

Some machine tools will not accept the shortened two-block version and the three-block version must be programmed. If in doubt, always use the three-block version

◆ Conditions for Tool Change

Before calling M06 tool change function in the program, always create safe conditions. Most machines have a light located on the control panel for visual confirmation that the tool is physically present at the tool change position.

Safe automatic tool change can take place only if these conditions are established:

- All machine axes had been zeroed
- Spindle must be fully retracted:
 - (a) In Z-axis at machine zero for vertical machines
 - (b) In Y-axis at machine zero for horizontal machines
- X and Y axis positions of the tool must be selected in a clear area
- The next tool must be previously selected by a T-function

A typical program sample illustrates tool change between tools in the middle of a program (from T02 to T03) - graphically illustrated in Figures 14-2 to 14-4:

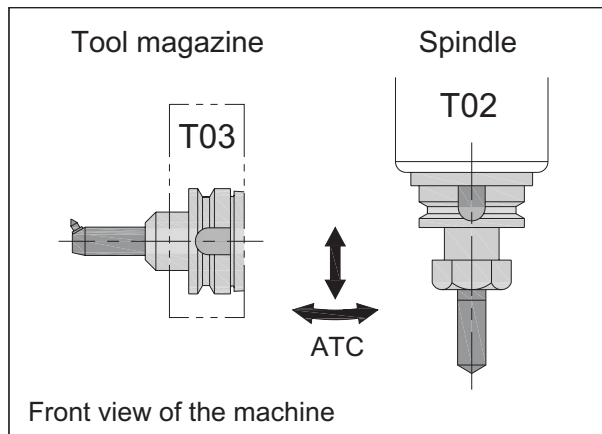


Figure 14-2

ATC example - Blocks N51 to N78 (current status)

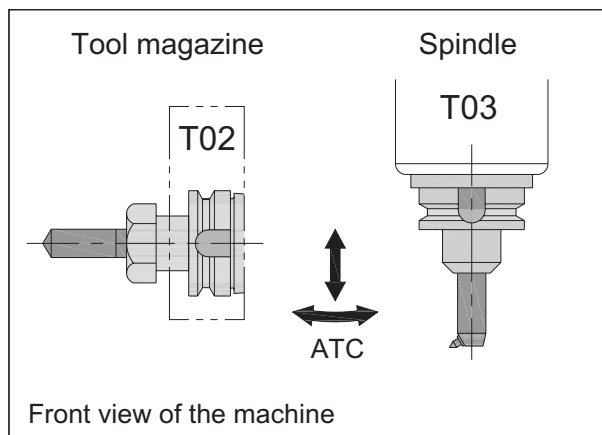


Figure 14-3

ATC example - Block N79 (actual tool change)

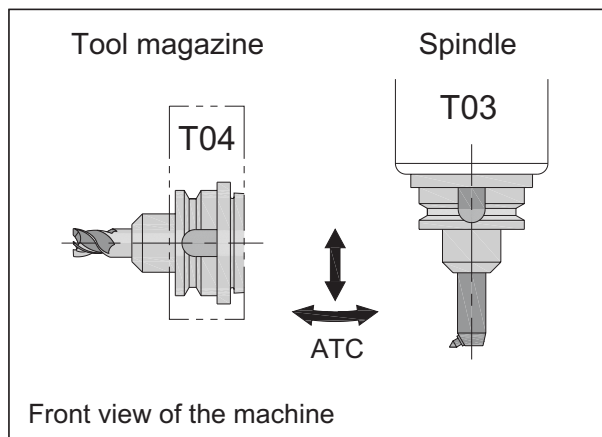


Figure 14-4

ATC example - Block N80 (new tool waiting = next tool)

➡ Example for illustrations :

```

N51 ...                               (... T02 IN SPINDLE)
N52 ... T03   (... T03 READY FOR TOOL CHANGE)
... .. (MACHINING WITH T02)
N75 G00 Z1.0 (RETRACT FROM DEPTH)
N76 G28 Z1.0 M05 (T02 COMPLETED)
N77 M01 (OPTIONAL STOP)
... .. (BLANK LINE BETWEEN TOOLS)
N78 T03 (T03 CALL REPEATED)
N79 M06 (T02 OUT - T03 IN THE SPINDLE)
N80 G90 G54 G00 X-18.56 Y14.43 S700 M03 T04
N81 ... .. (MACHINING WITH T03)
  
```

In the example, block N76 represents the end of machining, using tool T02, already in the spindle. It will cause tool T02 to move into Z-axis machine zero position, stopping spindle at the same time. Optional program stop function M01 follows in block N77.

In the following block N78, the call for T03 is repeated - this is not necessary, but may come very useful for repeating the tool later. Block N79 is the actual tool change. T02 in the spindle will be replaced with T03 that is currently in the waiting position.

Finally, in block N80, rapid motion in X and Y axes represents the first motion of T03, with spindle ON. Note the T04 at the block end. To save time, the next tool should be placed into the waiting position as soon as possible after any tool change.

Also note that when T02 is completed in block N77, it is *still in the spindle!* There are programmers who do not follow this method. If the tool change is included right *after* the G28 block (machine zero return) and *before* the M01 block, it will be more difficult for CNC operator to repeat the tool that just finished working, if it becomes necessary.

AUTOMATIC TOOL CHANGER - ATC

Several references to *Automatic Tool Changer* (ATC) were made in some examples. There are many designs of ATC's on various machines and they vary greatly from one machine manufacturer to another. Needless to say, the method of programming varies for different types, sometimes quite a bit. Machine tool changer, once it is setup, will automatically index the programmed cutting tool, in proper order. Everything will be under program control. Programmer and operator should be thoroughly familiar with the type of ATC on all machining centers in the shop.

◆ Typical ATC System

A typical *Automatic Tool Change* system may have a double swing arm, one for the incoming tool, another for the outgoing tool. It will likely be based on *Random Memory* selection (described earlier), which means the next tool can be moved to a waiting position and be ready for a tool

change, while the current tool works. This machine feature always guarantees the same tool change time. Typical time for tool changing cycle only can be very fast on modern CNC machines, often measured in fractions of a second.

The maximum number of tools that can be loaded into the tool magazine also varies greatly, from as few as 10 to as many as 400 or more. A small CNC vertical machining center may have typically 10 to 30 tools. Larger machining centers will have a greater tool capacity.

Apart of the tool changer features, programmer and machine operator should be also aware of other technical considerations that may influence tool change under program control. They relate to the physical characteristics of cutting tools when mounted in the tool holder:

- Maximum tool diameter
- Maximum tool length
- Maximum tool weight

◆ Maximum Tool Diameter

Maximum tool diameter that can be used without any special considerations is specified by the machine manufacturer. It assumes that a maximum diameter of a certain size may be used in *every* pocket of the tool magazine. Many machine manufacturers allow for a slightly larger tool diameter to be used, providing the *two adjacent* magazine pockets are empty (Figure 14-5).

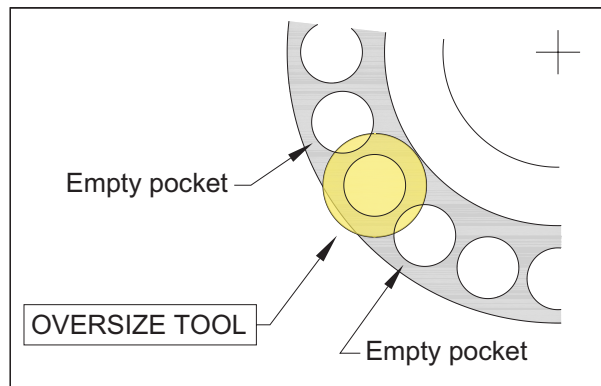


Figure 14-5

The adjacent pockets must be empty for a large tool diameter

For example, a machine description lists the maximum tool diameter *with* adjacent tools as 4 inches (100 mm). If both adjacent pockets are empty, the maximum tool diameter can be increased to 5.9 inches (150 mm), which may be quite a large increase. By using tools with a larger than recommended diameter, there is a *decrease* in the actual number of tools that can be placed in the tool magazine.

Adjacent pockets must be empty for oversize tools !

◆ Maximum Tool Length

Tool length in relation to ATC is the extension of a cutting tool from the spindle gauge line towards the part. The longer the tool length, the more important it is to pay attention to Z-axis clearance during tool change. Any physical tool contact tool with the machine, fixture or part is very undesirable. Such a condition could be very dangerous - there is not much that can be done to interrupt the ATC cycle, except pressing the *Emergency Switch*, which is usually too late. Figure 14-6 illustrates the concept of the tool length, as measured from the gauge line.

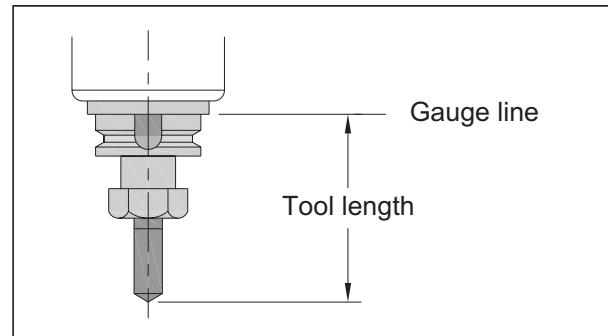


Figure 14-6

The concept of tool length

◆ Maximum Tool Weight

Most programmers will usually consider the tool diameter and its length, when developing a new program. However, some programmers will easily forget to consider the tool overall *weight*. Weight of a cutting tool does not generally make a difference in programming, because the majority of tools are lighter than the maximum recommended weight. Keep in mind that the ATC is largely a mechanical device, and as such has certain load limitations. The weight of a tool is always the *combined weight* of cutting tool *and* tool holder, including collets, screws, pull studs and similar parts that are subject to tool change.

Do not exceed the recommended tool weight during setup !

For example, a given CNC machining center may have the maximum recommended tool weight specified as 22 pounds or about 10 kg. If even a *slightly* heavier tool is used, for example 24 lb. (10.8 Kg), the ATC should not be used at all - use a manual tool change for that tool only. Machine spindle may be able to withstand a slight weight increase but the tool changer may not. Since the word '*slight*' is only relative, the best advice in this case is - *do not overdo it!* If in doubt, always consult manufacturer's recommendations. Examples in this chapter illustrate how to program such a unusual tool change, providing the tool weight is safe.

◆ ATC Cycle

Programmer does not have to know every detail related to the automatic tool changer actual operation. It is not a vital knowledge, although it may be quite a useful knowledge in many applications. On the other hand, a CNC machine operator should know *each and every step* of the ATC cycle inside out.

As an example, the following description is relevant to a typical CNC vertical machining center and may be a little different for some machines. Always study individual steps of the ATC operation - often, that knowledge will resolve a problem of tool jam during tool changing. This is a possible time loss that can be avoided. Some machines have a step-by-step cycle available with a special rotary switch, usually located near the tool magazine.

In the following example, a tool changer with a double arm swing system is used. It will take the cutting tool from the waiting position and exchange it with the tool currently in the machine spindle.

ATC is a process that will execute the following order of steps when the tool change function M06 is programmed. All steps described are quite typical, but not necessarily standard for every CNC machining center, so take them only as a close example:

1. Spindle orients
2. Tool pot moves down
3. Arm rotates 60 degrees CCW
4. Tool is unclamped (in the magazine and spindle)
5. Arm moves down
6. Arms rotates 180 degrees CW
7. Arm moves up
8. Tool is clamped
9. Arm rotates 60 degrees CW
10. The rack returns
11. Tool pot moves up

This example is only presented as general information - its logic has to be adapted to each machine tool. Instruction manual for the machine usually lists relevant details about ATC operations.

Regardless of the machine tool used, two conditions are always necessary to perform ATC correctly:

- Spindle must be stopped (with the M05 function)
- Tool changing axis must be at the home position (machine reference position)

For CNC vertical machining centers, the tool changing axis is the Z-axis, for the horizontal machining centers it is the Y-axis. M06 function will also stop the spindle, but never count on it. It is strongly recommended to stop the spindle with M05 function (spindle stop) *before* the tool change cycle is executed.

◆ MDI Operation

Incidentally, each step of the tool change cycle can usually be executed through MDI (*Manual Data Input*), using special M-functions. These functions are *only* used for service purposes, via MDI operation, and *cannot* be used in a CNC program. The benefit of this feature is that a tool changing problem can be traced to its cause and corrected from there. Check instructions for each machine to get details about these functions.

PROGRAMMING THE ATC

A number of possibilities exists in relation to automatic tool changer. Some of the important ones are the number of tools used, what tool number is registered to the spindle (if any) at the start of a job, whether a manual tool change is required, whether an extra large tool is used, etc.

In the next several examples, some typical options will be presented - these examples can be used directly, if the CNC machine tool uses exactly the same format, or they can be *adapted* to a particular working environment. For the following examples, some conditions must be established that will help to understand the subject of programming a tool change much better.

To program ATC successfully, all that is needed is programming format for *three tools* - the *first* tool used, the tool(-s) used in the *middle* of the program and the *last* tool used in the program. To make the whole concept even easier to understand, all examples will use only four tool numbers - each tool number will *represent* one of the four available programming formats:

- T01 ... Tool designation represents the first tool used in CNC program
- T02 ... Tool designation represents any tool in CNC program between the first and last tool
- T03 ... Tool designation represents the last tool used in CNC program
- T99 ... Tool designation represents an empty tool (dummy tool) as an empty tool pocket identification

In all examples, the first three tools will always be used, the empty tool only if required. Hopefully, these examples will illustrate the concept of many possible ATC applications. Another possible situation is in situations where only a *single tool* is used in the CNC program.

◆ Single Tool Work

Certain jobs or special operations may require only one tool to do the job. In this case, such tool is generally mounted in the spindle during setup and no tool calls or tool changes are required in the program:

```

O1401      (FIRST TOOL IN THE SPINDLE AT START)
N1 G20                      (INCH MODE)
N2 G17 G40 G80              (SAFE BLOCK)
N3 G90 G54 G00 X.. Y.. S.. M03 (TOOL MOTION)
N4 G43 Z.. H01 M08          (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09             (T01 MACHINING DONE)
N27 G28 Z.. M05             (T01 TO Z-HOME)
N28 G00 X.. Y..            (SAFE XY POSITION)
N29 M30                     (END OF PROGRAM)
%
```

Unless the selected tool is in the way of part changing, it remains in the spindle permanently for the whole job.

◆ Programming Several Tools

Machining a part using several tools is the most typical method of CNC work. Each tool is loaded into the spindle when required, using various ATC processes. From the programming viewpoint, various tool changing methods do not affect the program cutting section, only the tool *start* (before machining) or the tool *end* (after machining).

As already discussed, the required tool can be changed automatically, only if the Z-axis is at machine zero (for vertical machining centers) or the Y-axis is at machine zero (for horizontal machining centers). Tool position in the remaining axes is only important to the safety of tool change, so there is no tool contact with the machine, fixture, or part. All following examples are formatted for vertical machine models. Some programs use machine zero return for *all* axes at the end of last tool, for example:

```

...
N393 G00 Z.. M09            (CURRENT TOOL WORK DONE)
N394 G28 Z.. M05            (CURRENT TOOL TO Z HOME)
N395 G28 X.. Y..           (CURRENT TOOL TO XY HOME)
N396 M30                    (END OF PROGRAM)
%
```

Technically, there is nothing wrong with this practice, but it may cause a significant time loss for a large volume of parts. A preferred method is to either make the tool change *above* the last tool location, or to move the tool *away* from the part, to a *safe* location. This last method is illustrated in the examples that present various methods of program startup, as it relates to different methods of tool changing.

◆ Keeping Track of Tools

If the tool changing operation is simple, it should be easy to keep a track of where each tool is at any given moment. In later examples, more complex tool changes will take place. Keeping a visual track of which tool is waiting and which tool is in the spindle can be done with a 3 column table with block number, tool waiting and tool in the spindle.

Block Number	Tool Waiting	Tool in Spindle

To fill the table, start from the program top and find every occurrence of both, the T-address *and* the M06 function. All other data are irrelevant. In example O1402, the table will be filled as a practical sample of usage.

◆ Any Tool in Spindle - Not the First

This is the most common method of programming an ATC. CNC operator sets all tools in the magazine, registers the settings but leaves the tool measured last in the spindle. On most machines, this tool should *not* be the first tool. Programmer matches this tool changing method within the program. The following example is probably one that may be most useful for everyday work. All activities are listed in the comments.

```

O1402      (ANY TOOL IN SPINDLE AT START)
          (**** NOT THE FIRST TOOL ****)
N1 G20                      (INCH MODE)
N2 G17 G40 G80 T01          (GET T01 READY)
N3 M06                     (T01 TO SPINDLE)
N4 G90 G54 G00 X.. Y.. S.. M03 T02 (T02 READY)
N5 G43 Z.. H01 M08          (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09             (T01 MACHINING DONE)
N27 G28 Z.. M05             (T01 TO Z HOME)
N28 G00 X.. Y..            (SAFE XY POSITION)
N29 M01                     (OPTIONAL STOP)

N30 T02                     (T02 CALL REPEATED)
N31 M06                     (T02 TO SPINDLE)
N32 G90 G00 G54 X.. Y.. S.. M03 T03 (T03 READY)
N33 G43 Z.. H02 M08          (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09             (T02 MACHINING DONE)
N47 G28 Z.. M05             (T02 TO Z HOME)
N48 G00 X.. Y..            (SAFE XY POSITION)
N49 M01                     (OPTIONAL STOP)

N50 T03                     (T03 CALL REPEATED)
N51 M06                     (T03 TO SPINDLE)
N52 G90 G00 G54 X.. Y.. S.. M03 T01 (T01 READY)
N53 G43 Z.. H03 M08          (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09             (T03 MACHINING DONE)
N67 G28 Z.. M05             (T03 TO Z HOME)
N68 G00 X.. Y..            (SAFE XY POSITION)
N69 M30                     (END OF PROGRAM)
%
```

The filled-in table below shows the status of tools for the *first part only*. “?” represents *any* tool number.

Block Number	Tool Waiting	Tool in Spindle
N1	?	?
N2	T01	?
N3	?	T01
N4	T02	T01
T01 WORKING		
N30	T02	T01
N31	T01	T02
N32	T03	T02
T02 WORKING		
N50	T03	T02
N51	T02	T03
N52	T01	T03
T03 WORKING		

When the *second part* is machined or any other part after that, all tool tracking is simplified and consistent. Compare the next table with the previous one - there are no question marks. The table shows exactly where each tool is.

Block Number	Tool Waiting	Tool in Spindle
N1	T01	T03
N2	T01	T03
N3	T03	T01
N4	T02	T01
T01 WORKING		
N30	T02	T01
N31	T01	T02
N32	T03	T02
T02 WORKING		
N50	T03	T02
N51	T02	T03
N52	T01	T03
T03 WORKING		

Examples shown here use this method as is or slightly modified. For most jobs, there is no need to make a tool change at XY safe position, if the work area is clear of obstacles. Study this method before all others. It will help to see the logic of some more advanced methods a lot easier.

A few comments related to the O1402 example. Always program M01 optional stop *before* a tool change - it will make it easier to repeat the tool, if necessary. Also note beginning of each tool, containing the next tool search. Tool in the block containing the first motion has already been called - compare block N4 with N30 and block N32 with N50. The repetition of tool search at the start of each tool has two reasons. It makes the program easier to read (tool that is coming into the spindle will be known) and it also allows a *repetition* of this tool, regardless of which tool is currently in the spindle.

◆ First Tool in the Spindle

Program may also start with the first tool already being in the spindle. This is a common practice for ATC programming. The first tool in the program must be loaded into the spindle during setup. In the program, the first tool is called to the waiting station (ready position) during the *last* tool - *not the first tool*. Then, a tool change will be required in one of the last program blocks. The first tool programmed must be *first for all parts* within the job batch.

```

O1403          (FIRST TOOL IN SPINDLE AT START)
N1 G20          (INCH MODE)
N2 G17 G40 G80 T02          (GET T02 READY)
N3 G90 G54 G00 X.. Y.. S.. M03
N4 G43 Z.. H01 M08          (APPROACH WORK)
...
<... T01 working... >
...
N26 G00 Z.. M09          (T01 MACHINING DONE)
N27 G28 Z.. M05          (T01 TO Z HOME)
N28 G00 X.. Y..          (SAFE XY POSITION)
N29 M01          (OPTIONAL STOP)

N30 T02          (T02 CALL REPEATED)
N31 M06          (T02 TO SPINDLE)
N32 G90 G54 G00 X.. Y.. S.. M03 T03 (T03 READY)
N33 G43 Z.. H02 M08          (APPROACH WORK)
...
<... T02 working... >
...
N46 G00 Z.. M09          (T02 MACHINING DONE)
N47 G28 Z.. M05          (T02 TO Z HOME)
N48 G00 X.. Y..          (SAFE XY POSITION)
N49 M01          (OPTIONAL STOP)

N50 T03          (T03 CALL REPEATED)
N51 M06          (T03 TO SPINDLE)
N52 G90 G54 G00 X.. Y.. S.. M03 T01 (T01 READY)
N53 G43 Z.. H03 M03          (APPROACH WORK)
...
<... T03 working... >
...
N66 G00 Z.. M09          (T03 MACHINING DONE)
N67 G28 Z.. M05          (T03 TO Z HOME)
N68 G00 X.. Y..          (SAFE XY POSITION)
N69 M06          (T01 TO SPINDLE)
N70 M30          (END OF PROGRAM)
%
```

This method is not without a disadvantage. Since there is always a tool in the spindle, it may become an obstacle during setup or part changing. Solution here is to program the tool change in such a way that there is *no tool* in the spindle during part setup (*spindle empty* condition).

◆ No Tool in the Spindle

An empty spindle at the start and end of each machined part is less productive than starting with the first tool in the spindle. One extra tool change increases the cycle time. An empty spindle at start should only be used if the programmer has a valid reason, for example, to recover space above the part that would otherwise be occupied by the tool. Such recovered space may be useful for removing the part, for example, with a crane or a hoist. Programming format for this situation is not much different from the previous example - except that there is an *extra tool change* at the end of the program. This tool change brings the first tool back into the spindle, for consistent startup of each program run.

```
O1404          (NO TOOL IN SPINDLE AT START)
N1 G20          (INCH MODE)
N2 G17 G40 G80 T01 (GET T01 READY)
N3 M06          (T01 TO SPINDLE)
N4 G90 G54 G00 X.. Y.. S.. M03 T02 (T02 READY)
N5 G43 Z.. H01 M08 (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09          (T01 MACHINING DONE)
N27 G28 Z.. M05          (T01 TO Z HOME)
N28 G00 X.. Y..          (SAFE XY POSITION)
N29 M01                (OPTIONAL STOP)

N30 T02                (T02 CALL REPEATED)
N31 M06                (T02 TO SPINDLE)
N32 G90 G54 G00 X.. Y.. S.. M03 T03 (T03 READY)
N33 G43 Z.. H02 M08      (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09          (T02 MACHINING DONE)
N47 G28 Z.. M05          (T02 TO Z HOME)
N48 G00 X.. Y..          (SAFE XY POSITION)
N49 M01                (OPTIONAL STOP)

N50 T03                (T03 CALL REPEATED)
N51 M06                (T03 TO SPINDLE)
N52 G90 G54 G00 X.. Y.. S.. M03 T99 (T99 READY)
N53 G43 Z.. H03 M08      (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09          (T03 MACHINING DONE)
N67 G28 Z.. M05          (T03 TO Z-HOME)
N68 G00 X.. Y..          (SAFE XY POSITION)
N69 M01                (OPTIONAL STOP)
N70 M06                (T99 TO SPINDLE)
N71 M30                (END OF PROGRAM)
%
```

◆ First Tool in the Spindle with Manual Change

In the next example, the second tool represents *any middle* tool in the program that uses three or more tools. This tool may be too heavy or too long and cannot be indexed through the ATC cycle and must be loaded manually. This tool change can be done by the operator, but only if the *program supports manual tool change*. To achieve this goal is to use M00 program stop with an appropriate comment, describing the reason for the stop. Optional stop M01 is *not* a good selection - M00 is a much safer choice - it will *always* stop the machine without interference from the operator.

Follow the next example carefully, to understand how a manual tool change can be performed when the first tool is in the spindle. T02 in the example will be changed manually by CNC operator.

```
O1405          (FIRST TOOL IN SPINDLE AT START)
N1 G20          (INCH MODE)
N2 G17 G40 G80 T99 (GET T99 READY)
N3 G90 G54 G00 X.. Y.. S.. M03
N4 G43 Z.. H01 M08 (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09          (T01 MACHINING DONE)
N27 G28 Z.. M05          (T01 TO Z HOME)
N28 G00 X.. Y..          (SAFE XY POSITION)
N29 M01                (OPTIONAL STOP)

N30 T99                (T99 CALL REPEATED)
N31 M06                (T99 TO SPINDLE)
N32 T03                (T03 READY)
N33 M00                (STOP AND LOAD T02 MANUALLY)

N34 G90 G54 G00 X.. Y.. S.. M03 (NO NEXT TOOL)
N35 G43 Z.. H02 M08      (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09          (T02 MACHINING DONE)
N47 G28 Z.. M05          (T02 TO Z HOME)
N48 G00 X.. Y..          (SAFE XY POSITION)
N49 M19                (SPINDLE ORIENTATION)
N50 M00                (STOP AND UNLOAD T02 MANUALLY)

N51 T03                (T03 CALL REPEATED)
N52 M06                (T03 TO SPINDLE)
N53 G90 G54 G00 X.. Y.. S.. M03 T01 (T01 READY)
N54 G43 Z.. H03 M08      (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09          (T01 MACHINING DONE)
N67 G28 Z.. M05          (T03 TO Z HOME)
N68 G00 X.. Y..          (SAFE XY POSITION)
N69 M01                (OPTIONAL STOP)
N70 M06                (T01 TO SPINDLE)
N71 M30                (END OF PROGRAM)
%
```

Note the M19 function in block N49. This miscellaneous function will orient spindle to exactly the same position as if automatic tool changing cycle were used. CNC operator can then replace the current tool with the next tool and still maintain the tool position orientation. This consideration is mostly important for certain boring cycles, where the tool bit cutting edge has to be positioned away from the machined surface. If a boring bar is used, it is necessary to align its cutting tip.

◆ No Tool in the Spindle with Manual Change

The following program is a variation on the previous example, except that there is no tool in the spindle when program starts.

```
O1406          (NO TOOL IN SPINDLE AT START)
N1 G20          (INCH MODE)
N2 G17 G40 G80 T01      (GET T01 READY)
N3 M06          (T01 TO SPINDLE)
N4 G90 G54 G00 X.. Y.. S.. M03 T99 (T99 READY)
N5 G43 Z.. H01 M08      (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09      (T01 MACHINING DONE)
N27 G28 Z.. M05      (T01 TO Z HOME)
N28 G00 X.. Y..      (SAFE XY POSITION)
N29 M01            (OPTIONAL STOP)

N30 T99            (T99 CALL REPEATED)
N31 M06            (T99 TO SPINDLE)
N32 T03            (T03 READY)
N33 M00            (STOP AND LOAD T02 MANUALLY)
N34 G90 G54 G00 X.. Y.. S.. M03 (NO NEXT TOOL)
N35 G43 Z.. H02 M08      (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09      (T02 MACHINING DONE)
N47 G28 Z.. M05      (T02 TO Z HOME)
N48 G00 X.. Y..      (SAFE XY POSITION)
N49 M19            (SPINDLE ORIENTATION)
N50 M00            (STOP AND UNLOAD T02 MANUALLY)

N51 T03            (T03 CALL REPEATED)
N52 M06            (T03 TO SPINDLE)
N53 G90 G54 G00 X.. Y.. S.. M03 T99 (T99 READY)
N54 G43 Z.. H03 M08      (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09      (T03 MACHINING DONE)
N67 G28 Z.. M05      (T03 TO Z HOME)
N68 G00 X.. Y..      (SAFE XY POSITION)
N69 M01            (OPTIONAL STOP)
N70 M06            (T99 TO SPINDLE)
N71 M30            (END OF PROGRAM)
%
```

◆ First Tool in the Spindle and an Oversize Tool

Sometimes it is necessary to use a little larger diameter tool than the machine specifications allow. In that case, the oversize tool *must* return to the same pocket in the tool magazine it came from and the two adjacent magazine pockets must be empty. *Do not use a tool that is too heavy!* In example O1407, the large tool is T02. Again, follow the block comments carefully.

```
O1407          (FIRST TOOL IN SPINDLE AT START)
N1 G20          (INCH MODE)
N2 G17 G40 G80 T99      (GET T99 READY)
N3 G90 G54 G00 X.. Y.. S.. M03
N4 G43 Z.. H01 M08      (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09      (T01 MACHINING DONE)
N27 G28 Z.. M05      (T01 TO Z HOME)
N28 G00 X.. Y..      (SAFE XY POSITION)
N29 M01            (OPTIONAL STOP)

N30 T99            (T99 CALL REPEATED)
N31 M06            (T99 TO SPINDLE)
N32 T02            (T02 READY)
N33 M06            (T02 TO SPINDLE)
N34 G90 G54 G00 X.. Y.. S.. M03 (NO NEXT TOOL)
N35 G43 Z.. H02 M08      (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09      (T02 MACHINING DONE)
N47 G28 Z.. M05      (T02 TO Z HOME)
N48 G00 X.. Y..      (SAFE XY POSITION)
N49 M01            (OPTIONAL STOP)

N50 M06            (T02 OUT OF SPINDLE TO THE SAME POT)
N51 T03            (T03 READY)
N52 M06            (T03 TO SPINDLE)
N53 G90 G54 G00 X.. Y.. S.. M03 T01 (T01 READY)
N54 G43 Z.. H03 M08      (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09      (T03 MACHINING DONE)
N67 G28 Z.. M05      (T03 TO Z HOME)
N68 G00 X.. Y..      (SAFE XY POSITION)
N69 M01            (OPTIONAL STOP)
N70 M06            (T01 TO SPINDLE)
N71 M30            (END OF PROGRAM)
%
```

◆ No Tool in the Spindle and an Oversize Tool

This is another tool change version. It assumes no tool in the spindle at program start. It also assumes the next tool is *larger* than the maximum recommended diameter, within reason. In this case, the oversize tool must return to *exactly* the same pocket it came from. It is important that the adjacent magazine pockets are both empty.

Both adjacent magazines pockets
must be empty for oversize tool diameter !

In example O1408, T02 represents the large tool.

```

O1408                (NO TOOL IN SPINDLE AT START)
N1 G20                (INCH MODE)
N2 G17 G40 G80 T01    (GET T01 READY)
N3 M06                (T01 TO SPINDLE)
N4 G90 G54 G00 X.. Y.. S.. M03 T99 (T99 READY)
N5 G43 Z.. H01 M08    (APPROACH WORK)
...
<... T01 working...>
...
N26 G00 Z.. M09        (T01 MACHINING DONE)
N27 G28 Z.. M05        (T01 TO Z HOME)
N28 G00 X.. Y..        (SAFE XY POSITION)
N29 M01                (OPTIONAL STOP)

N30 T99                (T99 CALL REPEATED)
N31 M06                (T99 TO SPINDLE)
N32 T02                (T02 READY)
N33 M06                (T02 TO SPINDLE)
N34 G90 G54 G00 X.. Y.. S.. M03 (NO NEXT TOOL)
N35 G43 Z.. H02 M08    (APPROACH WORK)
...
<... T02 working...>
...
N46 G00 Z.. M09        (T02 MACHINING DONE)
N47 G28 Z.. M05        (T02 TO Z HOME)
N48 G00 X.. Y..        (SAFE XY POSITION)
N49 M01                (OPTIONAL STOP)

N50 M06                (T02 OUT OF SPINDLE TO THE SAME POT)
N51 T03                (T03 READY)
N52 M06                (T03 TO SPINDLE)
N53 G90 G54 G00 X.. Y.. S.. M03 T99 (T99 READY)
N54 G43 Z.. H03 M08    (APPROACH WORK)
...
<... T03 working...>
...
N66 G00 Z.. M09        (T03 MACHINING DONE)
N67 G28 Z.. M05        (T03 TO Z HOME)
N68 G00 X.. Y..        (SAFE XY POSITION)
N69 M01                (OPTIONAL STOP)
N70 M06                (T99 TO SPINDLE)
N71 M30                (END OF PROGRAM)
%
```

This and previous examples illustrate some of the ATC programming methods. Tool change programming is not difficult once all tool changing mechanics of the machining center are known.

T-FUNCTION FOR LATHES

So far, the tool function was covered as it applied to CNC machining centers. CNC lathes also use the tool function T, but with a completely different structure.

◆ Lathe Tool Station

A typical slant bed lathe uses a polygonal turret holding all external and internal cutting tools in special pockets or *stations*. These *tool stations* are similar to a tool magazine on machining centers. Typically, their design accepts 8, 10, 12 or more cutting tools - *Figure 14-7*.

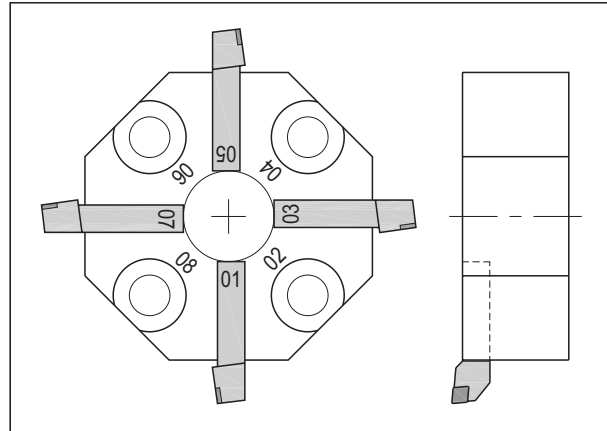


Figure 14-7

Typical view of an octagonal lathe turret (tools 1 to 8)

Some CNC lathe models start adopting the tool changer type similar to machining centers, with many more tools available and away from the work area.

Since all tools are normally held in a single turret, the one selected for cutting will always *carry along* all other tools into the work area. This may be a design whose time has passed but it is still very commonly used in industry. Because of a possible interference between a tool and the machine or part, care must be taken not only of the *active* cutting tool, but also of *all other tools* mounted in the turret, while considering all possible collision situations.

◆ Tool Indexing

To program a tool change, or rather to *index* a cutting tool into its active position, the T-function must be programmed using a proper format. For CNC lathe, this format calls for the address T, followed by *four* digits - *Figure 14-8*.

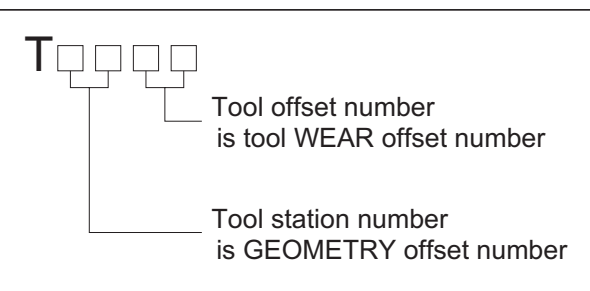


Figure 14-8

Structure of a 4-digit tool number for CNC lathes

It is important to understand this function well. Think about the four digits as *two pairs* of (two) digits, rather than four single digits. Leading zeros within each pair may be omitted. Each pair has its own meaning:

The *first* pair (first and second digits), control the tool index station and the geometry offset.

➡ Example :

T01xx - selects the tool mounted in position one and activates geometry offset number one

The *second* pair (third and fourth digits), control the tool wear offset number used with the selected tool.

➡ Example :

Txx01 - selects the wear offset register number one

It is customary, but not mandatory, to match both pairs, if possible. For example, tool function T0101 will select tool station number one, geometry offset number one and the associated tool wear offset register number one. This format is easy to remember and should be used every time, if only one offset number is assigned to the tool number.

If *two or more different wear offsets* are used for the same tool, it is not possible to match both pairs. In such a case, *two or more different* wear offset numbers must be programmed and applied to *the same* tool station number:

➡ Example :

T0101 for turret station 01,
geometry offset 01 and wear offset 01

➡ Example :

T0111 for turret station 01,
geometry offset 01 and wear offset 11

The first pair is always the tool station number *and* the geometry offset number. The latter example assumed that tool wear offset 11 is not used by another tool. If tool 11 is used with offset 11, another suitable wear offset number must be selected, for example 21, and program it as T0121. Most controls have 32 or more offset registers for geometry offsets and another 32+ wear offsets registers.

All offset values are applied to CNC lathe program by first registering their values into *offset registers*.

TOOL OFFSET REGISTERS

The word *offset* has been mentioned already several times with two adjectives - with the expression *geometry offset* and the expression *wear offset*. What exactly is an *offset*? What is the difference between one offset and the other?

On the OFFSET display of a typical Fanuc control, there is a choice of two screens, both *very* similar in appearance. One is called the *Geometry Offset* screen, the other is called the *Wear Offset* screen. *Figure 14-9* and *Figure 14-10* show examples of both screens, with typical (*i.e.*, reasonable) sample entries (imperial units input).

OFFSET - GEOMETRY				
No.	X-offset	Z-offset	Radius	Tip
01	-8.4290	-16.4820	0.0313	3
02	-8.4570	-14.7690	0.0000	0
03	-8.4063	-16.3960	0.0156	3
04	-8.4570	-12.6280	0.0000	0
05	-8.4350	-16.4127	0.0000	0
06	-9.8260	-13.2135	0.0313	2
07	0.0000	0.0000	0.0000	0

Figure 14-9

Example of the GEOMETRY offset screen display

OFFSET - WEAR				
No.	X-offset	Z-offset	Radius	Tip
01	0.0000	0.0000	0.0313	3
02	0.0000	0.0150	0.0000	0
03	0.0036	0.0000	0.0156	3
04	0.0000	-0.0250	0.0000	0
05	0.0010	-0.0022	0.0000	0
06	-0.0013	0.0000	0.0313	2
07	0.0000	0.0000	0.0000	0

Figure 14-10

Example of the WEAR offset screen display

◆ Geometry Offset

Geometry offset number *always* matches the turret station number selected. Machine operator measures and enters geometry offsets for all tools used in the program.

GEOMETRY offset amount is always measured from the machine zero position

Distance from machine zero position will reflect the distance from tool reference point to part reference point (program zero). *Figure 14-11* shows a typical measurement of geometry offset applied to a common *external* tool.

All X settings will normally have *diameter* values and are stored as *negative* for a typical rear lathe of a slant bed type. Z-axis settings will normally be also negative (positive values are possible but impractical). How to actually measure geometry offset is a subject of CNC machine tool operation training, not programming. Manual and automatic methods are available for this purpose.

Figure 14-12 shows a typical measurement of geometry offset applied to a common *internal* tool.

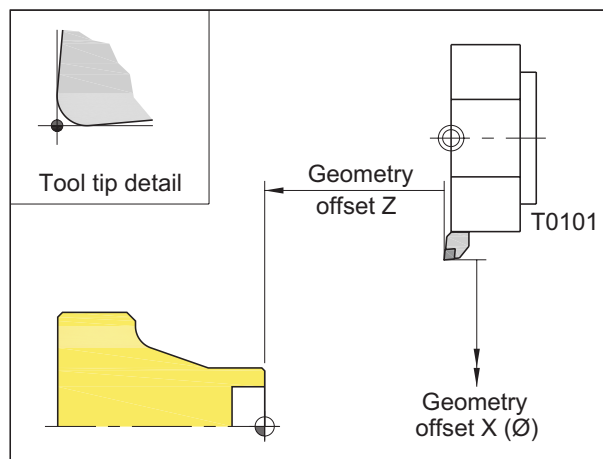


Figure 14-11

Typical geometry offset for external (turning) tools

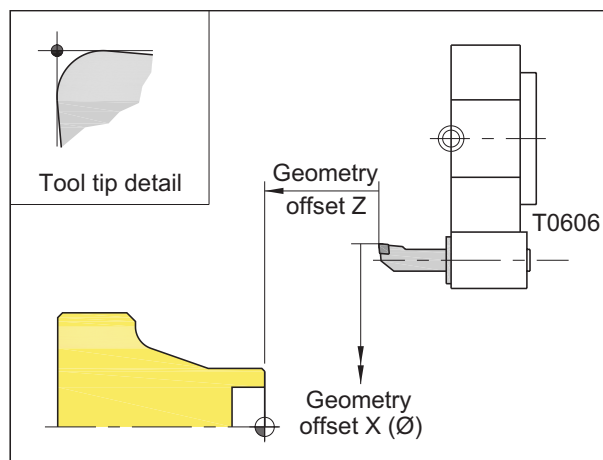


Figure 14-12

Typical geometry offset for internal (boring) tools

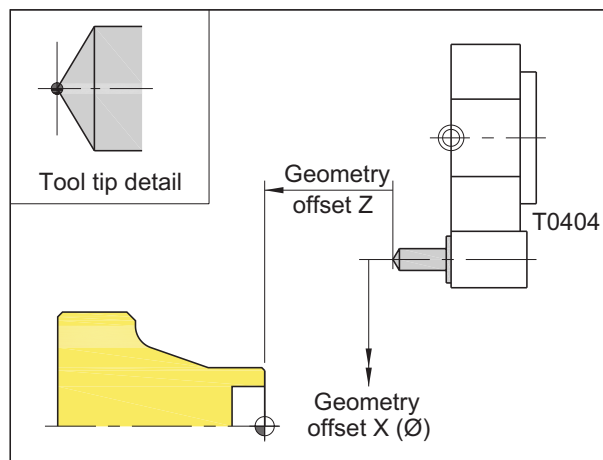


Figure 14-13

Typical geometry offset for center line (drilling) tools

One last possibility relating to geometry offset is illustrated in *Figure 14-13*. It shows geometry offset applied to any tool used on the spindle center line (always at X0 position). These tools include center drills, drills, taps, reamers, etc. Their X-offset setting will always be the same.

◆ Wear Offset

In CNC program development, many dimensions used will be those in the part drawing. For example, a dimension of 3.0000 inches, is programmed as X3.0. This figure does not reflect any implied dimensional tolerances. Program entries X3.0, X3.00, X3.000 and X3.0000 have *exactly* the same result. What is needed to maintain dimensional tolerances, particularly when they are tight? What has to be done with a worn out tool that is still good enough to cut a few more parts? The answer is that the programmed tool path must be adjusted, *fine-tuned*, to match the machining conditions. Program itself will not be changed, but a *wear offset* for the selected tool is applied.

WEAR offset amount is the difference between **programmed** value and the **actual** measured size of part

Figure 14-14 illustrates the principle of tool wear offset, although the scale is exaggerated for emphasis.

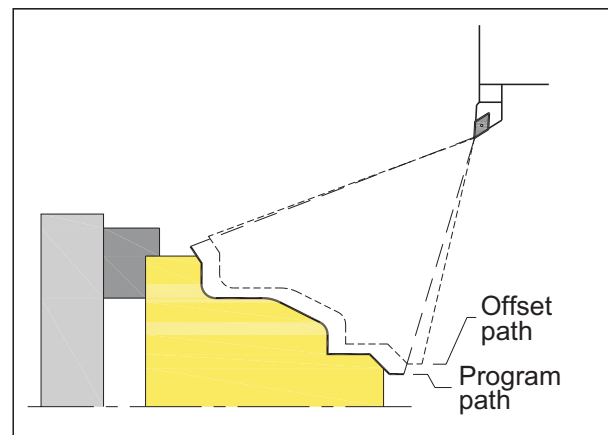


Figure 14-14

Programmed tool path and tool path with wear offset

Actual wear offset setting has only one purpose - it compensates between the programmed value, for example that of Ø3.0 inch, and the actual dimension *as measured* during inspection, for example, Ø3.004. The differential value of -0.004 is entered into the wear offset register. This is the offset number specified as the *second pair* of tool function used in the program. Since the program uses diameters for the X-axis, offsets will also be applied to diameter. Details on this kind of adjustment are more useful to CNC machine operator, but any part programmer will benefit from them as well.

◆ Wear Offset Adjustment

To illustrate the concept of wear offset adjustment on a rear type lathe, T0404 in the program will be used as an example. The goal is to achieve an outside diameter of 3.0 inches and tolerance of ± 0.0005 . Initial setting of the wear offset in register *Txx04* will be zero. Relevant section of the program may look something like this:

```
N31 M01
...
N32 T0400 M42
N33 G96 S450 M03
N34 G00 G42 X3.0 Z0.1 T0404 M08
N35 G01 Z-1.5 F0.012
N36 ...
```

When the machined part is inspected (measured), it can have only one of three possible inspection results:

- On-size dimension
- Oversize dimension
- Undersize dimension

If the part is measured *on size*, there is no need to interfere. Both tool setup and program are working correctly. If the part is *oversize*, it can usually be recut, if machining an *outside diameter*. For an *inside diameter*, the exact opposite will apply. Recut may damage the surface finish, which should be a concern. If the part is *undersize*, it becomes a scrap. The aim is to prevent all *subsequent* parts from being undersize as well. The following table shows inspection results for all three existing possibilities:

Measurement	External diameter	Internal diameter
ON size	Size OK	Size OK
OVER size	Recut possible	SCRAP
UNDER size	SCRAP	Recut possible

Let's go a little further. Whether the part will be oversized or undersized, something has to be done to prevent this problem from happening again. Common action to take is *adjusting the wear offset amount*. Again, the emphasis here is that this is an example of an *outside diameter*.

External diameter X3.0 in the example may result in 3.004 diameter measured size. That means it is 0.004 oversize - on diameter. CNC operator, who is in charge of offset adjustments, will change the currently set 0.0000 value in the X-register of the wear offset 04 to -0.0040. The subsequent cut should result in a part that will be measured within specified tolerances.

If the part in the example is undersize, say at 2.9990 inches, the wear offset must be adjusted by +0.0010 in the X-positive direction. Measured part is a scrap.

Principle of the wear offset adjustment is logical. If the machined diameter is *larger* then the drawing dimension allows, the wear offset is changed into the *minus* direction, towards the spindle center line, and vice versa. This principle applies equally to external and internal diameters. The only practical difference is that an oversized external diameter and undersize internal diameter *can* be recut (see the table above). Several practical examples of using the wear offset creatively are described on *page 306*.

◆ The R and T Settings

The last items are the R and T columns (*Geometry and Wear*). Both offset screen columns are only useful during setup. The *R column* is the radius column, *T column* is the tool tip orientation column (*Figure 14-15*).

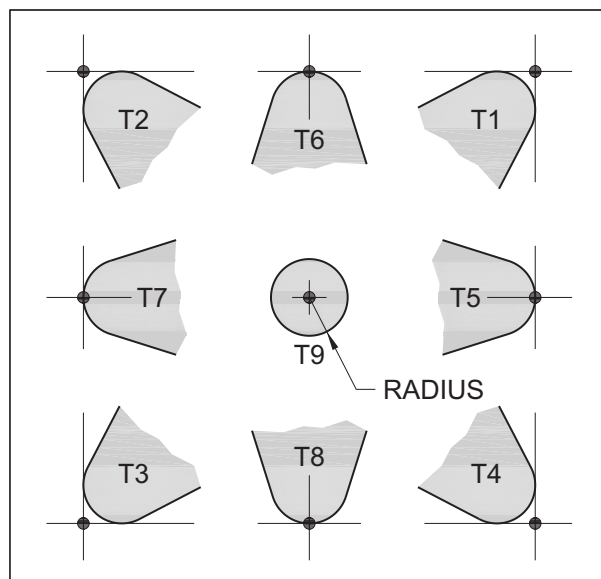


Figure 14-15

Arbitrary tool tip orientation numbers used with tool nose radius compensation (G41 or G42 mode)

The main rule of using R and T columns is that they are only effective in tool nose radius offset mode. If no G41 or G42 is programmed, settings in these two columns are irrelevant. If G41/G42 command is used, non-zero values for that tool must be set in both columns. The R-column requires *tool nose radius* of the cutting tool, the T-column requires *tool tip orientation number* of the cutting tool. Both are described on *page 275* in more detail. The most common tool nose radii for turning and boring are:

1/64 of an inch = 0.0156 or 0,4 mm
 1/32 of an inch = 0.0313 or 0,8 mm
 3/64 of an inch = 0.0469 or 1,2 mm

Tool tip numbers are arbitrary and indicate the tool orientation number used to calculate the nose radius offset, regardless of tool setting in the turret.

15

REFERENCE POINTS

In the previous chapters, the basic relationship between machine geometry and part setup was covered. CNC programmers work in a fairly precise environment, and several mathematical relationships are of extreme importance.

There are three major environments in programming that require an established mathematical relationship:

Environment	Relationship consists of ...
Machine	Machine tool + Control system (CNC unit)
Part	Workpiece + Drawing + Material
Tool	Holder + Cutting tool

Each environment by itself is independent of the other two. If the relationship is not apparent right away, consider the *sources* of each environment:

- **MACHINE TOOL** is made by a company specializing in machine tools, usually not controls or cutting tools
 ... *this environment is combined with* ...
- **CONTROL SYSTEM** is made by a company specializing in the application of electronics to machine tools. They do not normally manufacture machine tools or cutting tools
- **PART** (workpiece) is a unique engineering design developed in a company that does not manufacture machine tools, control systems, or cutting tools and holders
- **CUTTING TOOLS** are a specialty of tooling companies, which may or may not make cutting tool holders. These companies do not manufacture machine tools or CNC systems

These sources inevitably converge when customer buys a CNC machine. A certain engineering design (part), must be machined on a machine tool from one manufacturer, using control system of another manufacturer, cutting tools from yet another manufacturer, and tool holders from a fourth source. These sources are similar to a musical quartet of first class musicians who never played together. In both cases there is a need to create perfect harmony.

By itself, each environment is not very useful. A machine without tools will not yield any profit; a tool that cannot be used on any machine is not going to benefit the manufacturing either. A part cannot be machined without tools.

The common point here is that all three environments cannot be useful without some 'team work'. They have to work together - *they have to interact*.

For programming purposes, these relationships and interactions are based on one common denominator of *each* environment - *a reference point*.

Reference point is a fixed or selected arbitrary location on the machine, on the tool and on the part. A fixed reference point is a precise location along two or more axes, designed during manufacturing or setup. Some reference points are established by the programmer, during programming process. In these three environments, three reference points are needed - one reference point for each of the available groups:

- **Machine reference point** .. Machine zero *or* Home
- **Part reference point** .. Program zero *or* Part zero
- **Tool reference point** .. Tool tip *or* Command point

In a typical machine shop language, these reference points have somewhat more practical meaning. *Home position* or a *machine zero* are synonymous terms for *machine reference point*. *Program zero*, or *part zero*, or *part origin* are terms commonly used instead of the more official term *part reference point*. And the names *tool tip* and *tool command point* are commonly used for the *tool reference point*.

REFERENCE POINT GROUPS

The first group covers the *CNC machine tool* or *CNC machine* for short, which is the combination of machine proper and the control system. Numeric values that relate to the CNC machine tool include a variety of dimensions, specifications, parameters, ranges, ratings, etc. When a part is set in a fixture on the machine table or mounted into a lathe chuck, collet, face plate, or other work holding device, there is a second group of numbers to consider. They are *part considerations*, such as its size, its height, diameter, shape, etc., that are unique to each job. Finally, the third group of numbers relate to the *cutting tools*. Each cutting tool has its individual features, as well as features that are shared with many other cutting tools.

All available numeric values have a meaning - they are not merely numbers - they are actual *values* or *setting amounts* that programmers and operators have to work with individually as well as together.

◆ Reference Point Groups Relationship

The key to any successful CNC program is to make all three groups to work in a coordinated way. This goal can only be achieved by understanding the principles of reference points and how they work. Each reference point can have one of two characteristics:

- *Fixed* reference point
- *Flexible or floating* reference point

A fixed reference point is set by the machine manufacturer as part of the hardware design and cannot be physically changed by the user. A CNC machine has at least one fixed reference point. When it comes to deciding the reference points for the part or the cutting tool, CNC programmer has certain degree of freedom. Part reference point (program zero) is always a flexible point, meaning its actual position is in programmer's hands. Reference point for the mounted cutting tool can be either fixed or flexible, depending on machine design.

MACHINE REFERENCE POINT

Machine zero point, often called *machine zero*, *home position* or just *machine reference position*, is the origin of machine coordinate system. The location of this point may vary between machine manufacturers, but the most obvious differences can be found between individual machine types, namely vertical and horizontal models.

In general terms, a CNC machine has two, three, or more axes, depending on the type and model. Each axis has a maximum range of travel that is fixed by the manufacturer. This range is usually different for each axis. If the CNC operator exceeds the range on either end, an error condition known as *overtravel* will occur. Not a serious problem, but one that could be annoying. During machine setup, particularly after the power has been turned on, the position of all axes has to be preset to be always the same, from day to day, from one part to another. On older machines, this procedure is done by setting a grid, on modern machines, by performing a *machine zero return command*. Fanuc and many other control systems prevent automatic operation of a machine tool, unless the machine zero return command has been performed at least once - when the power to the machine has been turned on. A welcome safety feature.

On all CNC machines that use typical coordinate system, machine zero is located at the *positive* end of each axis travel range. For a typical three-axis vertical machining center, look at the part in the XY plane, that is straight down from the tool position (tool tip). Also look into the XZ plane (operator's front view of the machine), or into the YZ plane (operator's right-side view of the machine). These three planes are perpendicular to each other and together create so called *work cube* or *work space* - Figure 15-1.

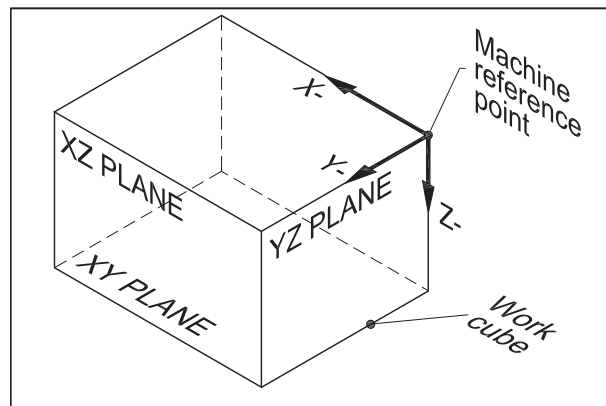


Figure 15-1

Machine reference point and axes orientation for a vertical machine

The cubical shape shown is useful only for overall understanding of machine work area. For programming and set-up, the majority of work is done with one or two axes at a time. To understand the work area and machine zero point in a plane, look at the machine from the top (XY machine plane) and from the front (XZ machine plane). Figures 15-2 and 15-3 illustrate both views.

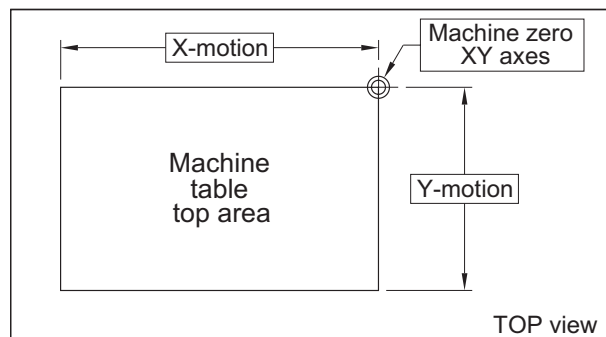


Figure 15-2

Top view of a vertical machine as viewed towards the table

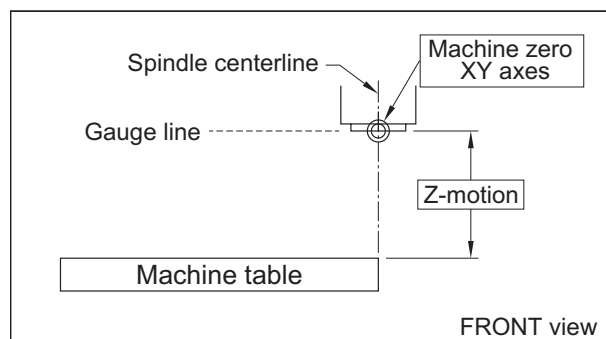


Figure 15-3

Front view of a vertical machine as viewed from the front

Compare the two views. In top view, the upper right corner is also the spindle center line shown in the front view.

Also note that in front view, there is a dashed line identified as the *gauge line*. This is an imaginary location for the proper fit of tool holder tapered body and is set by the machine manufacturer. Inside of the spindle is a precision machined taper that accepts tool holder with the cutting tool. Any tool holder mounted in the spindle will be in exactly the same position. The Z-motion illustrated will be shortened by the cutting tool projection. This subject of tool referencing is discussed later in this chapter.

◆ Return to Machine Zero

In manual mode, the CNC operator physically moves selected axes to machine zero position. The operator is also responsible to register this position into the control system, if necessary (older systems only). Never turn off power to the machine, while machine slides are *at* or very *close* to machine zero position. Being too close will make manual machine zero return more difficult later, after the power had been restored. A clearance of 1.0 inch (25.0 mm) or more for each axis *from* machine zero is usually sufficient. A typical procedure to physically reach the machine zero position will follow these steps:

1. Turn the power on (machine and control)
2. Select machine zero return mode
3. Select the first axis to move (usually Z-axis)
4. Repeat for all other axes
5. Check the lighted *in-position* indicators
6. Check the position screen display
7. Set display to zero, if necessary

Mainly for safety reasons, the first selected axis should be the Z-axis for machining centers and the X-axis for lathes. In both cases, either axis will be moving *away* from work, into a clear area. When the axis has reached machine zero position, a small indicator light on the control panel turns on to confirm that the axis actually reached machine zero. The machine is now at its reference position, at machine zero, or at the machine reference point, or at *home* - whichever term is used in the shop. Indicator light ON is the confirmation for each axis. Although the machine is ready for use, a good operator will go one step further. On the *position* display screen, the actual relative position should be set to zero readout for each axis, as a standard practice, if it is not set to zero automatically by the control. Typically, the POS button on the control panel selects position screen display.

PART REFERENCE POINT

A part ready for machining is located within the machine motion limits. Every part must be mounted in a device that is safe, suitable for the required operation and does not change position for any other part of the job run. Fixed location of the device is very important for consistent results and precision. It is also very important to guarantee that each part of the job is set the same way as the first part. Once the setup is established, part reference point can be selected.

This vital reference point will be used in a program to establish the relationship with machine reference point, reference point of the cutting tool and the drawing dimensions.

Part reference point is commonly known as *program zero* or *part zero*. Because the coordinate point that represents program zero can be selected by the programmer almost anywhere, it is not a fixed point, but a *floating* point. As this point is selectable, more details can be covered - after all, it is the programmer who selects part zero.

◆ Program Zero Selection

When selecting program zero, often in the comfort of programmer's office, a major decision is made that will influence the efficiency of part setup and its machining in the shop. Always be very attentive to all factors that are for and against program zero selection in a certain position.

In theory, the program zero point may be selected literally anywhere. That is not much of an advice, although true in mathematical terms. Within practical restrictions of the machine operations, only the most advantageous possibilities should be considered. Three such considerations should govern the selection of program zero:

- Accuracy of machining
- Convenience of setup and operation
- Safety of working conditions

Machining Accuracy

Machining accuracy is paramount - all parts must be machined exactly to the same drawing specifications. Accuracy is also important consideration in repeatability. All parts in the batch must be the same and all subsequent jobs must be the same as well.

Convenience of Setup and Operation

Operating and setup convenience can only be considered once the machining accuracy is assured. Working easier is everyone's desire. An experienced CNC programmer will always think of the effect the program has in machine shop. Defining program zero that is difficult to set on the machine or difficult to check is not very convenient. It slows down the setup process even more.

Working Safety

Safety is always important to whatever we do - machine and part setup are no different. Program zero selection has a lot to do with safety of the machining operation.

We look at the typical considerations of program zero selection for vertical machining centers and lathes individually. Differences in part design influence the program zero selections as well.

◆ Program Zero - Machining Centers

CNC machining centers allow a variety of setup methods. Depending on the type of work, some most common setup methods use vises, chucks, subplates and hundreds of special fixtures. In addition, CNC milling systems allow a multi-part setup, further increasing the available options. In order to select a program zero, all three machine axes must be considered. Machining centers with additional axes require zero point for each of these axes as well, for example, the indexing or rotary axes.

What are the most common setup methods? Most machining is done while clamped on machine table, in a vise or a fixture mounted on the table. These basic methods can be adapted to more complex applications.

CNC programmer determines the setup method for any given job, perhaps in cooperation with the machine operator. CNC programmer also selects the program zero position for each program. The process of selecting program zero starts with drawing evaluation, but two steps have to be completed first:

- Step 1. Study how the drawing is dimensioned, which dimensions are critical and which are not
- Step 2. Decide on the method of part setup and holding

Program zero almost presents itself in the drawing. In any setup, make sure all critical dimensions and tolerances are maintained from one part to another. Drawing dimensions not specified are usually not critical.

The simplest setup on a machine table involves support for the part, some clamps and locating surfaces. Locating surfaces must be *fixed* during job run and easy to be measured from. The most typical setup of this kind is based on the *three pin* concept. Two pins form a single row and the third pin is offset away at a right angle, creating a 90° angle setup corner as two locating surfaces - *Figure 15-4*.

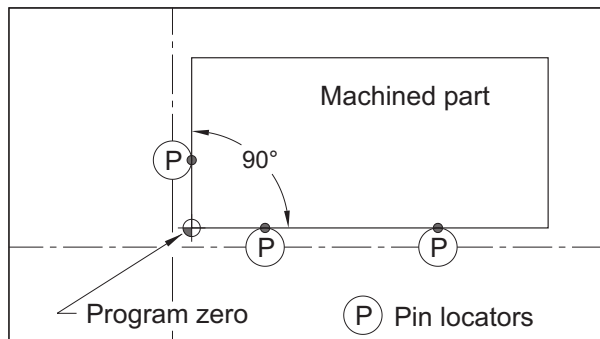


Figure 15-4

Three-pin concept of a part setup (all pins have the same diameter)

Since the part touches only one point on each pin, the setup is very accurate. Clamping is usually done with top clamps and parallels. The left and bottom edge of the part

are both parallel to machine axes and perpendicular to each other. Program zero (part zero) is at the intersection of the two locating edges.

The three-pin concept is common for virtually all setups, without using actual pins. If part is mounted in a vise, there are similarities. Vise jaws must be parallel to or perpendicular with machine axes and the fixed location must be established with a stopper or other fixed method.

Since a machine vise is the most common work holding device for small parts, let's use it as a practical example of how to select program zero. *Figure 15-5* illustrates a typical simple engineering drawing, with all expected dimensions, descriptions and material specifications.

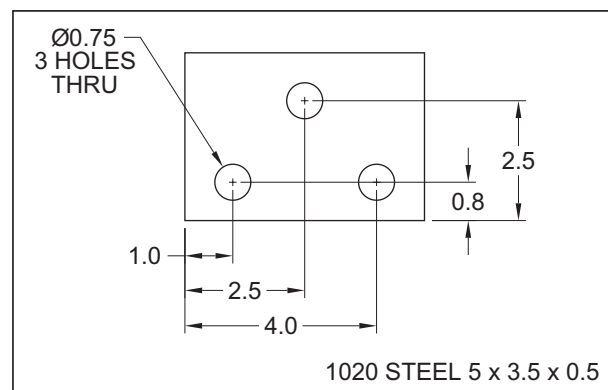


Figure 15-5

Sample drawing used for selecting program zero example

When selecting a program zero, first study the drawing dimensions. Designer's dimensioning style may have flaws, but it still is the engineering drawing used. In the example, dimensioning for all holes is from the lower left corner of the work. Does the program zero of the part suggest itself?

For this example, there should be no question about programming the reference point anywhere else except at the lower left corner of the part. This is the drawing origin and it will become the part origin as well. It also satisfies *Step 1* of the program zero selection process. The *Step 2*, dealing with work holding device selection is next. A typical setup in a special CNC machine vise could be the one illustrated in *Figure 15-6*.

In the setup identified as *Version 1*, the part has been positioned between vise jaws and a left part stopper. Part orientation is the same as per drawing, so all drawing dimensions will be applied in the program using these drawing dimensions. It seems that this is a winning setup - yet, this setup is actually quite poor.

What is missing in the decision is any consideration of the actual overall size of material. Drawing specifies a rectangular stock of 5.00 x 3.50. These are *open* dimensions - they can vary ± 0.010 or more and still be acceptable.

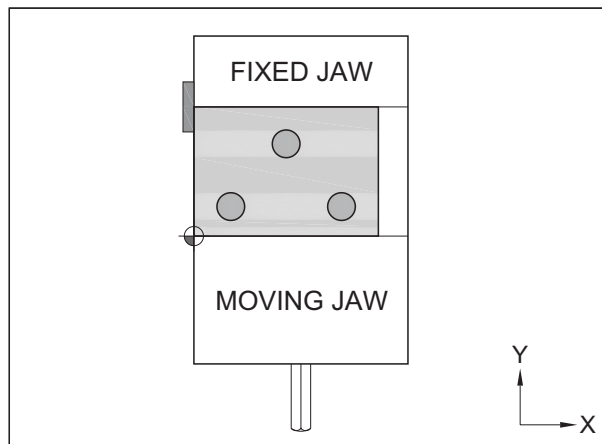


Figure 15-6

A sample part mounted in a machine vise - Version 1

Combine any acceptable tolerance with the vise design, where one jaw is a *fixed jaw* and the other one is a *moving jaw*, and the problem can be seen easily. The critical Y-axis reference is against a moving jaw!

Program zero edge should be the *fixed jaw* - a jaw that does not move. Many programmers incorrectly use a moving jaw as the reference edge. The benefit of programming in the first quadrant (all absolute values are positive) is attractive, but can produce inaccurate machining results, unless the blank material is 100% percent identical for all parts (usually not a normal case). *Version 1* setup can be improved significantly by rotating the part 180° and aligning a part stopper to the opposite side - *Figure 15-7*.

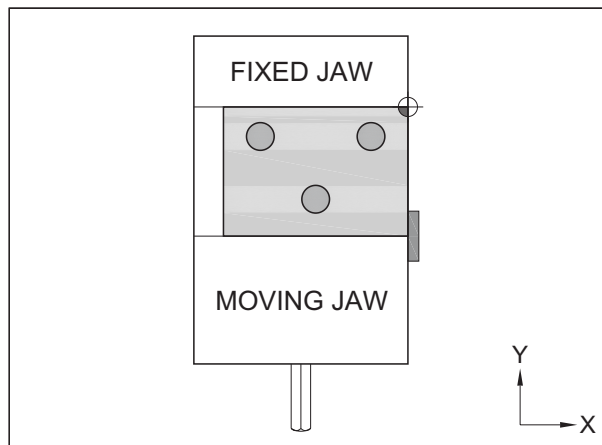


Figure 15-7

A sample part mounted in a machine vise - Version 2

In *Version 2*, results are consistent with the drawing. Part orientation by 180° has introduced another problem - *the part is located in the third quadrant!* All X and Y values will be *negative*. Drawing dimensions can be used in the program, but as negative. Just don't forget the minus signs.

If the choice is between *Version 1* and 2, select *Version 2* and make sure all negative signs are programmed correctly.

Is there another method? In most cases there is. The final *Version 3* will offer the best of both worlds. Part program will have all dimensions in the first quadrant, as per drawing. Also, the part reference edge will be against the fixed jaw! What is the solution? Rotate the *vise* 90° and position the part as shown - *Figure 15-8*, if possible.

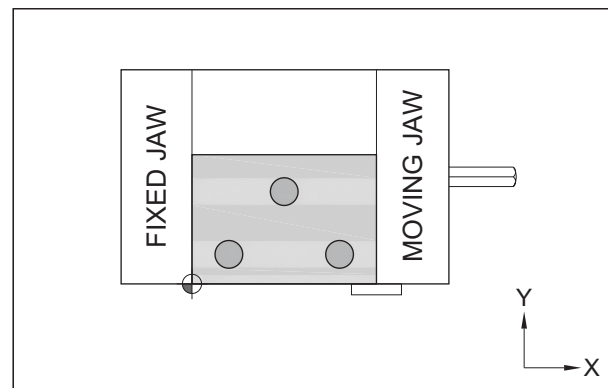


Figure 15-8

A sample part mounted in a machine vise - Version 3

To select program zero for the Z-axis, the common practice is to select the top face of finished part. That will make the Z-axis positive *above* the face and negative *below* the face. Another method is to select the bottom face of the part, where it is located in the fixture.

Special fixtures can also be used for part setup. In order to hold a complex part, a fixture can be custom made. In many applications of special fixtures, the program zero position may be built into the fixture, away from the part.

Selecting a program zero for round parts or patterns (bolt circles, circular pockets), the most useful program zero is at the center of the circle - *Figure 15-9*.

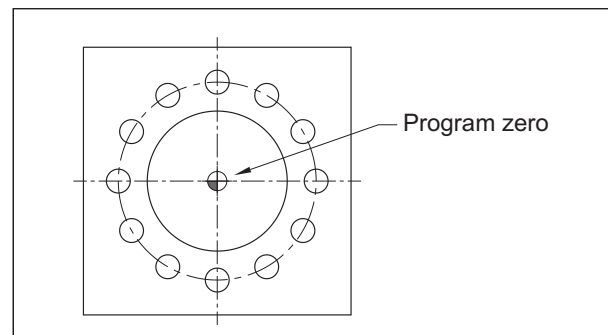


Figure 15-9

Common program zero for round objects is the center point

Page 399 describes the G52 command that may solve many problems associated with program zero at the center.

◆ Program Zero - Lathes

On CNC lathes, program zero selection is simple. There are only two axes to consider - the vertical X-axis and the horizontal Z-axis. Because of lathe design, the X-axis program zero selection is always the spindle center line.

On CNC lathes, program zero for the X-axis **MUST** be on the center line of the spindle

For the Z-axis, three popular methods are used:

- Chuck face . . . main face of the chuck
- Jaw face . . . locating face of the jaws
- Part face . . . front of the finished part

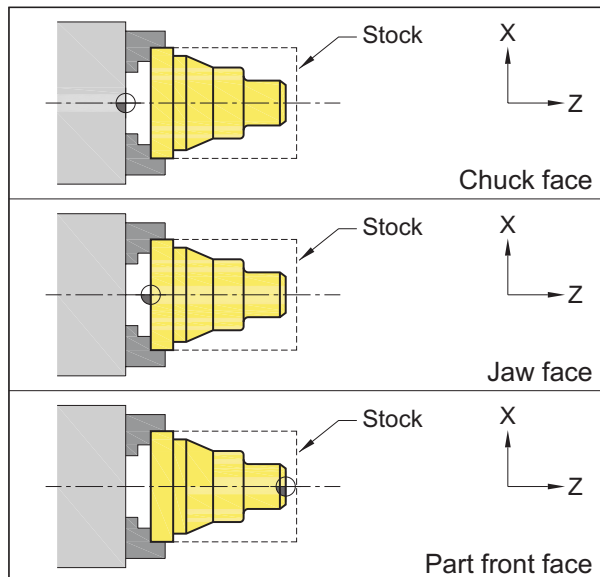


Figure 15-10

Common program zero options for a CNC lathe - center line is X0

Figure 15-10 illustrates the three options. In practical setup, a chuck face offers only one benefit - it can be easily touched with the tool edge, using feelers to prevent tool chipping. On a negative side, unless the part rests against chuck face, additional calculations are needed for the coordinate data and drawing dimensions cannot be used easily.

Jaw or fixture face presents more favorable situation. The face can also be touched with the tool and is consistent for all parts. This location may benefit machining irregular shapes, such as castings, forgings and similar parts.

Many lathe parts require machining at both ends. During the first operation, material stock for the second operation must always be added to every Z value. This is the main reason why CNC programmers keep away from program zero located on jaw or fixture face, except in special cases.

The most popular method is setting program zero on the *front face* of the *finished* part. This is not a perfect selection either but has many other advantages. The only disadvantage is that during setup, there is often no finished face. Many operators add the width of the rough face to the setup or cut a small face for the tool to touch.

What are the benefits of program zero at the front face? One benefit is that many drawing dimensions along Z-axis can be transferred directly into the program, normally with a *negative* value. A lot depends on the dimensioning method, but in majority of cases, CNC programmer benefits. Another benefit, probably the most important, is that a negative Z-value of a tool motion indicates the work area, a positive Z-value is in the clear area. During program development, it is easy to forget a minus sign for the Z-axis cutting motions. Such an error, if not caught in time, will position the tool *away* from the part, with tailstock as a possible obstacle. It is a wrong position, but a better one than hitting the part. Examples in this handbook use program zero at the *front finished face*, unless otherwise specified.

TOOL REFERENCE POINT

The last reference point is related to the tool itself. In milling and related operations, the tool reference point is usually the intersection of tool centerline and the lowest positioned cutting tip (edge).

In turning and boring, the most common tool reference point is an imaginary tool point of the cutting insert, because most tools have a cutting edge with a built-in radius.

For tools such as drills and other point-to-point tools used in milling or turning, the reference point is always the extreme tip of the tool, as measured along Z-axis. Figure 15-11 shows some common tool tip points.

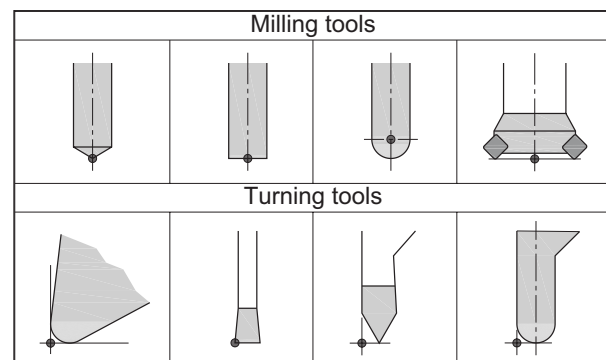


Figure 15-11

Typical tool reference points for various cutting tools

All three reference point groups are related. An error in one setting will have an adverse effect on another setting. Knowledge of reference points is important to understand register commands, offsets and machine geometry.

16

REGISTER COMMANDS

The three reference points available for CNC programming must be harmonized to work together correctly. Having the reference points for the *part* (i.e., program zero) and the cutting *tool* (i.e., tool tip) available, there has to be some means to associate them together, to fit them together. There must be some means to 'tell' the control system exactly where each tool is physically located, within the machine work area, *before* it can be used. The oldest method to do all this is to *register* the current tool position into the control system memory, through the program. This method required a command called the *Position Register*.

POSITION REGISTER COMMAND

Preparatory command for the tool position register is G92 for machining centers and G50 for (most) lathes:

G92	Position register command	(used in milling)
G50	Position register command	(used in turning)

Some CNC lathes also use the G92 command, but lathes supplied with Fanuc and similar controls normally use G50 command instead. In practical applications, both G92 and G50 commands have identical meaning and the following discussion applies to both commands equally. In the first part of this chapter, the focus will be on milling applications using G92 command, lathe applications using G50 command will be explained later.

In modern CNC programming, both position register commands were replaced by a much more sophisticated and flexible feature called the *Work Offsets* (G54 to G59), (see page 127 for details) and the *Tool Length Offset* (G43), (see page 135 for details). However, there are still quite a few older machine tools in shops that do not have the luxury of the G54 series of commands. There are also many companies using programs developed years ago, but still running on modern CNC equipment. In those cases, understanding the position registration command is an essential skill. This command has always been one that some programmers and operators found a little difficult to understand. In reality, it is a very simple command.

First, a look at some more detailed definition of this command. A typical description only specifies *Position Register Command*, which by itself is not very conclusive.

◆ Position Register Definition

A little more verbose definition of the position register command could be expressed this way:

Position register command sets the tool location as the distance and direction
FROM ... the program zero,
TO ... the tool current position,
measured along the axes

Note that this definition does not mention *machine zero* at all - instead, it mentions the *current tool position*. This is a very important distinction. Current tool position *may be* at machine zero, but it also *may be* somewhere else, within travel limits of the machine axes.

Also note the emphasis on *from-to* direction. By definition, the distance is unidirectional, between the program zero and the current tool location. The direction is always *from* program zero, *to* tool position, never reversed. In a program, the correct identification of each axis value (positive, negative, or zero) is always required.

Position register is only applicable in absolute mode of programming, while G90 command is in effect. It has no use in the incremental programming mode G91. In practical programming, virtually all programs written in incremental mode do begin in absolute mode, in order to reach the first tool location.

◆ Programming Format

As the command name suggests, tool position data associated with G92 command will be *registered* (i.e., stored) into the control system memory.

Format for G92 command is as follows:

G92 X.. Y.. Z..

In all cases, the address of each axis specifies the distance *from program zero to tool reference point* (tool tip). CNC programmer provides all coordinates based on the program reference point (program zero), discussed earlier. Any additional axis will also have to be registered with G92, for example the B-axis for the indexing table on horizontal machining centers.

◆ Tool Position Setting

The only purpose of G92 command is to *register the current tool position into the control memory* - nothing more!

No machine motion will ever occur
in a block containing the G92 command !

The effect of G92 can be seen on the *absolute position* screen display. At all times, absolute position display has some values for each axis, including zero. When G92 command is executed, all current values of the display will be *replaced* with the settings specified with G92. If an axis was not specified with G92, there will be no change of display for that axis. At the machine, the operator has a major responsibility - to match the actual tool setting with the settings specified in the G92 command.

MACHINING CENTERS APPLICATION

In programming for CNC machining centers *without* the *Work Coordinate System* feature (also known as *Work Offsets*), *Position Register* must be established for each axis and each tool. There are two methods:

- Tool position is set *at machine zero*
- Tool position is set *away from machine zero*

Which method is better? We look at both of them.

◆ Tool Set at Machine Zero

The first method requires that the machine zero position will also be the tool change position for all axes. This is not necessary and definitely *very* impractical. Consider it for a moment and think why it is impractical.

A program is usually done away from the machine, but the part position on the table must be specified:

G92 X12.0 Y7.5 Z8.375

Numbers in the example look innocent enough. But consider the CNC operator at the machine, trying to setup a part (without a special fixture), to be *exactly* 12.0 inches away from machine zero in the X-axis. At the same time, the operator must setup the same part *exactly* 7.5 inches away from machine zero for the Y-axis. The same effort has to be done for the Z-axis as well.

It is an almost impossible task, at least without some special fixtures. It is definitely an extremely *unproductive* task. There is no *need* for those numbers, they are strictly arbitrary - X12.0 could have easily been X12.5, with no added benefit whatsoever. All this difficulty is encountered only because the programmer has chosen the machine zero reference point for tool change position (mainly in the X and Y axes).

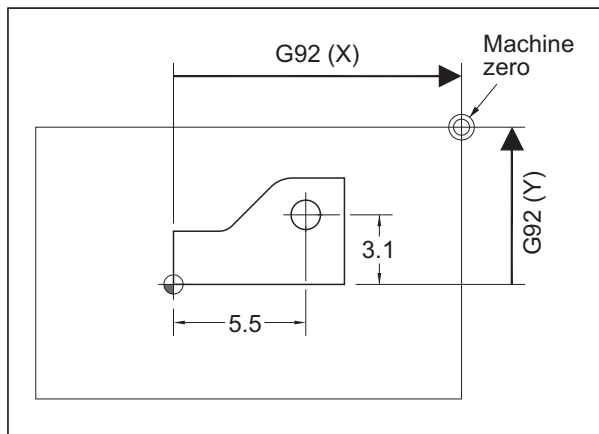


Figure 16-1

Current tool position register set at machine zero
(only XY axes shown)

Figure 16-1 shows a G92 setup based on the tool set at machine zero position. This method of starting program at machine zero is useful. There could be an advantage, for example, if a special fixture is permanently attached to the machine table. A subplate with a locator grid is a common example. Permanently set one or more vises may also benefit. There are numerous variations on this type of setup.

◆ Tool Set Away from Machine Zero

The second method eliminates the difficulty of the previous setup. It allows the programmer to set XY tool position *anywhere* within the machine travel limits (considering safety first) and use that position as the tool change position for XY axes. As there is no need for the machine zero itself, the CNC operator can setup the part anywhere on the table, in any reasonable position, within machine axes limits. Figure 16-2 shows an example of a tool set at a negative X-axis and positive Y-axis.

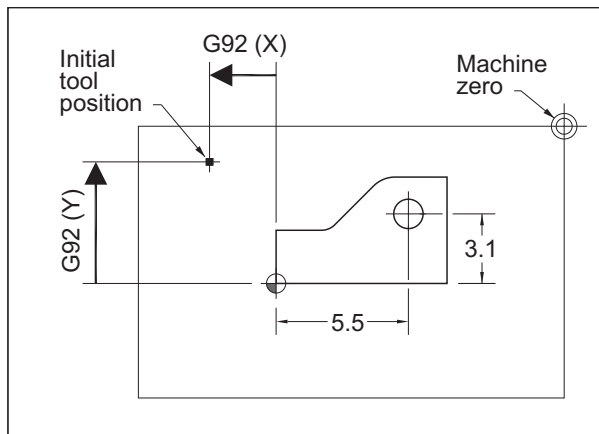


Figure 16-2

Current tool position register set away from machine zero
(only XY axes shown)

In order to place tool into the starting tool change position, the operator physically moves the tool *from program zero* by amounts specified in the G92 statement. This is a lot easier job and also much more efficient than restricting setup to the machine zero.

Once the tool change position is established, all tools in the program will return to this position for a tool change. Z-axis automatic tool change position on vertical machining centers must be programmed at machine zero as *the only* automatic tool change position. So the discussion really applies to XY axes only. Regardless of a tool position, the G92 setting will be the same for all tools, unless there is a good reason to change it.

The only major disadvantage of this method is that the new tool change position is only memorized by the control system while the power is on. When the machine power is turned off, the tool change position is lost. Many experienced CNC operators solve this problem by simply finding the actual distance from machine zero to the tool change position, register it once for each particular setup and then move the tool by that distance after restoring power, for example, at the start of a new day.

◆ Position Register in Z-axis

For a typical vertical machine, the Z-axis must be fully retracted to machine zero, in order to make automatic tool change. Position register setting is measured from program zero of the Z-axis (usually the top of finished face), to the tool reference tip, while the Z-axis is at machine zero position. There is no other option.

Normally, each tool will have a *different* Z setting of the G92 command, assuming tool length is different for each tool. As a rule, the XY settings will not change. *Figure 16-3* shows a typical setting for G92 command along Z-axis. Example O1601 illustrates the concept.

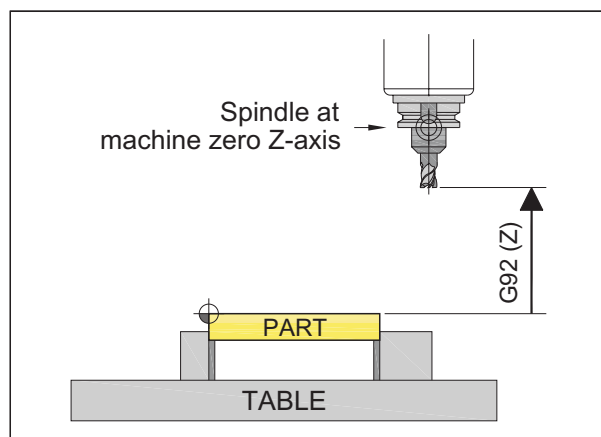


Figure 16-3

Current tool position register set at machine zero for the Z-axis (each tool will normally have a different setting)

◆ Programming Example

To illustrate how to use the position register command in a part program for vertical machining centers, certain rules have to be followed:

- Cutting tool should be changed first (in spindle)
- G92 must be established *before* any tool motions
- Tool *must* return to the G92 position when all cutting is completed

All three rules are followed in a sample program:

```
O1601                                (PROGRAM NUMBER)
N1 G20                                (SET ENGLISH UNITS)
N2 G17 G40 G80 G90 T01                (GET TOOL 1 READY)
N3 M06                                (TOOL 1 TO SPINDLE)
N4 G92 X9.75 Y6.5 Z11.0              (SET CURRENT XY)
N5 G00 X1.0 Y0.5 S800 M03            (MOVE TO POSITION)
N6 Z0.1 M08                          (MOVE TO CLEAR ABOVE)
N7 G01 Z-0.55 F5.0                  (FEED TO DEPTH)
N8 X3.0 Y4.0 F7.0                   (CUT A SLOT)
N9 G00 Z11.0 M09                    (RAPID TO Z MACHINE ZERO)
N10 X9.75 Y6.5 M05                  (RAPID TO XY SET POSITION)
N11 M01                             (OPTIONAL STOP FOR TOOL 1)
...
```

This is a simple example to write but more difficult to set-up on the machine. Don't worry about unknown program entries at the moment, the explanations should be clear.

Note - the Z-axis setting position must always be known - *at machine zero!* It does not matter whether the tool change in XY is made, at machine zero or away from it - the program format will be the same, just *meaning* of the values will be different. Only one tool is used, but normally, each tool will have a different Z setting as the position register, since each tool has a different length.

LATHE APPLICATION

For the CNC lathes with Fanuc and similar controls, G50 command is used instead of G92 command:

G50 X.. Z..

If G92 is used for a lathe, the command is similar:

G92 X.. Z..

Either command has exactly the same definition and rules as for milling - it indicates the distance *from* program zero, *to* the current tool position along axes.

No machine motion will occur
in a block containing G50 or G92 command !

Commands G50 and G92 are identical, except that they belong to two different G-code groups. Fanuc actually offers three G-code groups for lathe controls. Based on history, typical Japanese made controls use G50, whereby typical US made controls used G92. A cooperative US and Japanese venture known as *GE Fanuc* (*General Electric* and *Fanuc*) produces controls that are the most common in North American industry, and using the G50 command.

To program position register for lathe applications is very similar to that of G92 for mills. However, due to design of CNC lathes, where all tools are mounted in the turret, the *projection* of each tool (for both axes) from the turret holder must be considered. Not only that, possible interference must be prevented, because all mounted inactive tools move simultaneously along the one that is used for cutting. In milling, all non-active tools are safely out of way, placed in a tool magazine. Several new designs of CNC lathes are available, where tool changer on the lathe resembles the milling type.

◆ Tool Setup

The most important programming decision for lathe work relates to the setup. Although there are several options to select from, some are preferable to others.

Probably the most practical approach for lathe setup will be to have the tool change position for all tools corresponding to the machine zero position. This is a very easy position to move the turret to, just using the control panel switches. The position register measured to machine zero does have one major disadvantage - it may be *too far* for most jobs, particularly on larger lathes along the Z-axis. Just imagine a tool motion of 30 inches or more along the Z-axis only to index the turret and then the same 30 inch motion back to continue a cutting cycle. It is not efficient at all. There is a solution, however.

Much more efficient method is to select the tool indexing position for all tools as *close to the part* as possible. This position should always be based on the *longest* tool mounted in the turret (usually internal tools), whether such tool is used in the program or not. If there is enough clearance for the longest tool in the turret, there will also be enough clearance for all remaining shorter tools.

A possible compromise of the two methods described is to keep tool indexing position at the X-axis machine zero only (which is usually not too distant) and just establish the Z-axis position.

On a CNC lathe, do not forget to keep in mind the general layout of all tools in the turret, to prevent a collision with the part, the chuck, or the machine.

There are other, but less common, methods to setup a tool on the lathe using the G50 command.

◆ Three-Tool Setup Groups

On a typical slant bed CNC lathe, equipped with a polygonal turret (6 to 14 stations), all cutting tools reside in individual stations of the turret. During tool indexing, only the selected tool is in the active station. Upon evaluation of the type of tools used for CNC lathe operations, it will be clear that there are only three groups of cutting tools, based on the type of work they normally do:

- Tools working on the *part center line*
- Tools working *externally* on the part
- Tools working *internally* on the part

If the position register for each group is understood well, it will be easy to apply it to any tool within a group, regardless of the number of tools used.

◆ Center Line Tools Setup

Lathe tools classified as center line tools are typically center drills, spot drills, standard twist drills, indexable carbide drills, taps, reamers, and so on. Even an end mill can be used at the spindle center line. All tools in this group have a single common denominator, whereby the tool tip is always located on the spindle center line (X0), while they cut. These tools must always be setup exactly at 90° to the work face (parallel to the Z-axis).

Position register value in X-axis is from the spindle center line (X0) to the center line of the tool. For the Z-axis, the position register value is measured from program zero to the tool tip. Typically, center line tools will have a fairly large overhang - that means their G50 value along the Z-axis will be relatively small, when compared to external tools, which generally do not extend that much.

Figure 16-4 illustrates a typical setup for center line tools, using an indexable drill as an example.

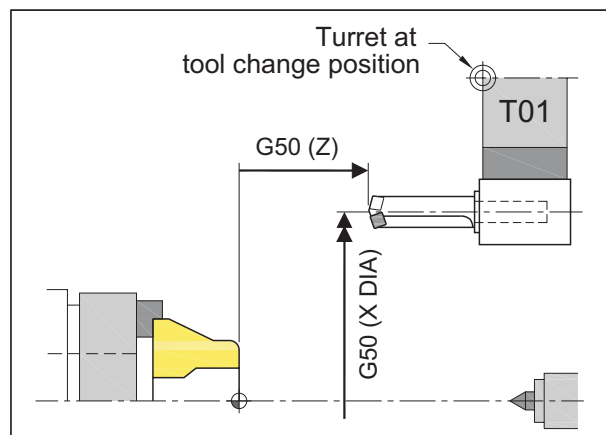


Figure 16-4

Typical G50 setting for center line lathe tools

◆ External Tools Setup

For external machining operations such as roughing and finishing outside diameters, taper cutting, grooving, knurling, single point threading, part-off and others, the cutting tool is rather small and approaches the part in an open space with generous clearances.

Position register value is measured from the program zero to the imaginary tool tip of the insert (see details at the end of this chapter). In case of tools like threading tool or a grooving tool, G50 amount is usually measured to the left side of the insert, for safety reasons.

Figure 16-5 illustrates a typical position register setup for an external tool (turning tool shown in the example).

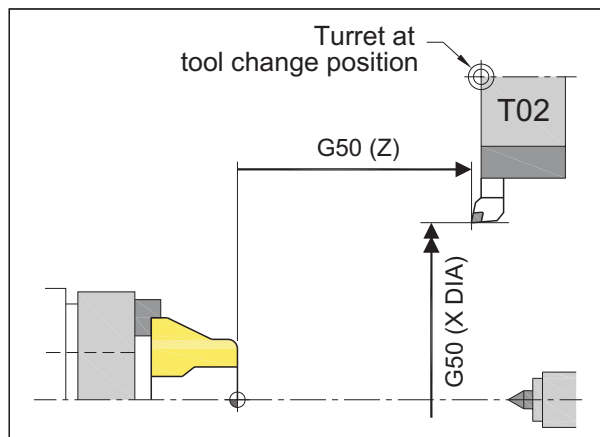


Figure 16-5

Typical G50 setting for external lathe tools

◆ Internal Tools Setup

Internal tools are all tools that do majority of their work inside of a part, in a premachined hole, core or other cavity. Typically, we may first think of a boring bar, but other tools can be used as well for various internal operations. For example, an internal grooving and internal threading are also common operations on CNC lathes. Setup rules for Z-axis apply in the same way for internal tools as for external tools of the same type.

Along the X-axis, tool position register setting must be made to the imaginary tip of the insert. Figure 16-6 shows a typical position register setup for an internal tool (boring bar shown in the example).

All three illustrations (Figures 16-4, 16-5 and 16-6) also show a possible order of the three operations (drill - turn - bore) for a typical CNC lathe job. Note that the turret position is identified as a *tool change position*, not necessarily as the machine zero position. That means G50 may be set anywhere within travel limits of the machine, even at machine zero.

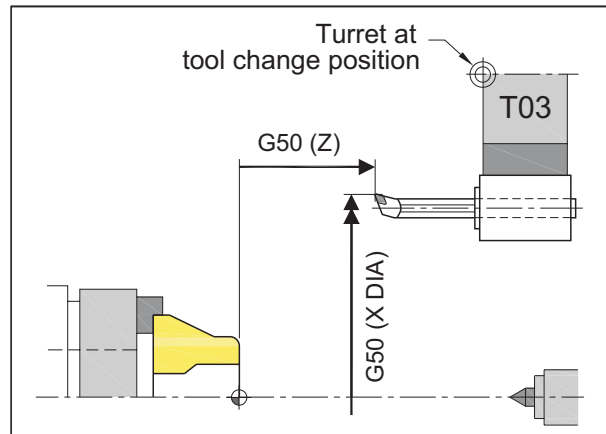


Figure 16-6

Typical G50 setting for internal lathe tools

For safety reasons, no tool should extend from a turret into Z negative zone - that is to the left of part front face. Many CNC lathes allow travel *beyond* the Z-axis machine zero (about 1-2 inches or 25-50 mm). Sometimes, this zone can be entered to make a safe tool change for very long tools. However, this is more advanced programming method and requires strict safety considerations. There is virtually no extended zone for X-axis above machine zero position (only about 0.02 inches or 0.5 mm).

Another safety concern relating to long tools is clearance in the part holding area, including chuck and jaws. Make sure to extend only those tools where the work requires it.

◆ Corner Tip Detail

Typical turning tool contains an indexable insert, with a corner radius for strength and surface finish control. When position register command is used for a tool that has a radius built-in, the programmer has to know (and also tell the CNC operator), *which edge* the G50 setting corresponds to. In many cases, the choice is simple. G50 value is measured from program zero to the imaginary intersection of tangential X and Z-axis. Depending on actual tool shape and its orientation in the turret, G50 setting will vary. Figure 16-7 on the next page shows several typical settings for the most common orientations of a tool with a corner radius, including two grooving tools.

◆ Programming Example

The example showing how to use position register command G50 on a lathe will be very similar to that of a machining center. First, the tool change is made, followed with G50 setting for the selected tool. When machining is done with that tool, tool has to return to the same absolute position as specified in the G50 block. The following simplified example is for two tools - the first tool is programmed to cut a face, the second tool is programmed to cut a 2.5 inch diameter:

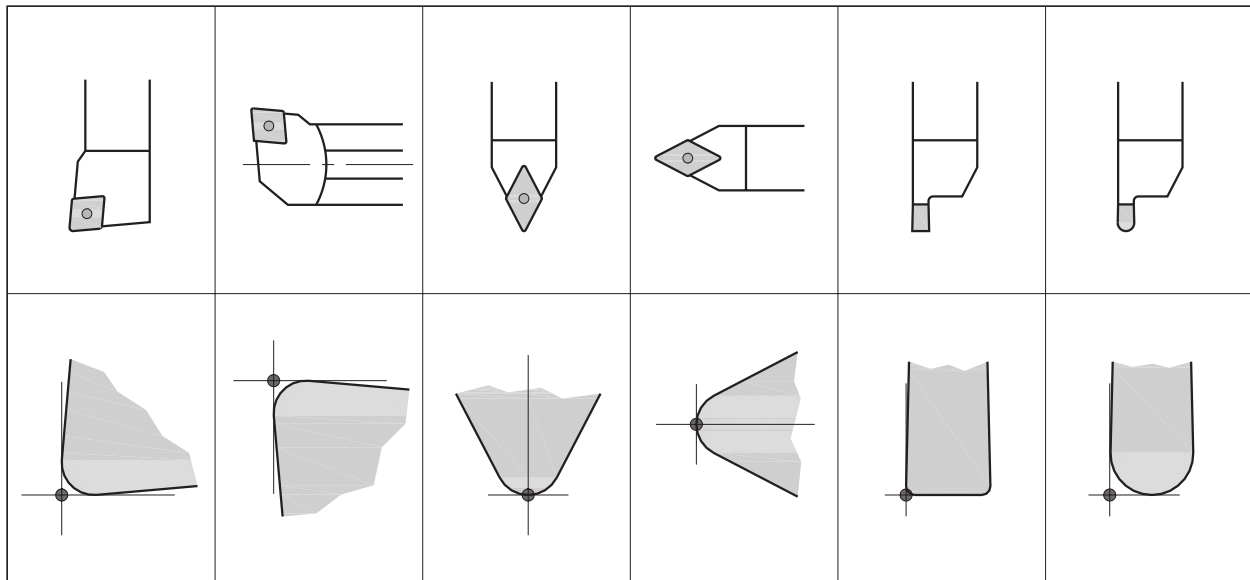


Figure 16-7

Position register setting G50 for common tool tip orientations - the heavy dot indicates XZ coordinates set by G50 X.. Z.. for the tool above

```

O1602
N1 T0100
N2 G50 X7.45 Z5.5
N3 G96 S400 M03
N4 G00 X2.7 Z0 T0101 M08
N5 G01 X-0.07 F0.007
N6 G00 Z0.1 M09
N7 X7.45 Z5.5 T0100
N8 M01

N9 T0200
N10 G50 X8.3 Z4.8
N11 G96 S425 M03
N12 G00 X2.5 Z0.1 T0202 M08
N13 G01 Z-1.75 F0.008
N14 G00 X2.7 M09
N15 X8.3 Z4.8 T0200
N16 M30
%
```

Note blocks N2 and N7 for the first tool, and N10 and N15 for the second tool. For each tool, these XZ pairs of blocks are exactly the same. What the program is 'telling' the control system here is that block N2 only registers the current tool position, but block N7 actually returns that tool to the same position it came from. For the second tool, block N10 registers the current tool position, block N15 forces the tool to return there.

Other important blocks to consider together are blocks N7 and N10. Block N7 is the *tool change position* for the first tool, block N10 is the *tool position register* for the second tool - *both tools are at the same physical position of the turret!* The difference in XZ values reflects the difference in the extension (length) of each tool from the turret station. All that is done with G50 command is telling the control *where the current tool tip is from program zero* - always keep that in mind!

17

POSITION COMPENSATION

Obsolete subject - chapter included for compatibility purposes

Relationships between various reference points in CNC programming are always expressed as preset numeric values. More often than not, these numbers, these specific values, are required well *before* actual machine setup takes place. During programming, many dimensions are known exactly, others are known approximately and there are also those that are not known at all. Some known dimensions are subject to variations between jobs. Without any corrective facility available to the programmer, it will be almost impossible to setup the machine precisely and efficiently. Fortunately, modern controls offer many features to make both programming and machine setup an easier, faster and more precise activity. A number of coordinate systems, offsets and compensations are typical support tools used in programming for corrective purposes.

One of the oldest programming methods available is called *position compensation*. As the name suggests, using position compensation functions, the *actual* tool position is compensated relative to its *theoretical* or *assumed* position.

It is only one of several corrective methods available to the programmer. On modern CNC systems, this method is available for compatibility with older programs. Today, this technique is obsolete. It has been replaced by the much more flexible *Work Offsets (Work Coordinate System)*, described in the next chapter. This chapter describes some typical programming applications that can benefit from using the old-fashioned position compensation method.

DESCRIPTION

The main purpose of position compensation is to correct any differences between machine zero and program zero tool positions. In practice, it is used in those cases, where the distance between two reference points is subject to some variations or is not known at all. For example, when working with castings, program zero taken from the cast surface will be subject to frequent changes. Using position compensation will eliminate the need to make constant program changes or realignment of the fixture setup. Normally, a part is mounted in a fixture on the table and its whole setup is compensated. For this reason, position compensation is sometimes called *fixture offset* or the *table offset*. The difference between an offset and a compensation is often very subtle, and for any practical purposes, the two terms are same.

In this handbook, each term is used in the same meaning as the majority of users interpret it. Position compensation can also be used for a very limited replacement of the cutter radius offset - this usage is not covered at all for its obsolescence. Instead, the emphasis will be on positioning of tool from machine zero towards the part.

Like several other functions, position compensation is a programming method that requires input of the CNC machine operator. Programmer specifies the type of compensation and the memory register number, CNC operator enters actual setting values at the machine, using appropriate display screens, during part setup.

◆ Programming Commands

On Fanuc and similar controls, there are four old preparatory commands (G-codes) available to program position compensation functions:

G45	<i>Single increase</i> in the programmed direction by the compensation amount
G46	<i>Single decrease</i> in the programmed direction by the compensation amount
G47	<i>Double increase</i> in the programmed direction by double the compensation amount
G48	<i>Double decrease</i> in the programmed direction by double the compensation amount

These definitions are based on *positive* compensation values stored in the control register. If the stored values are negative, the meaning of all definitions is valid only when the signs are inverted. None of these four preparatory commands is modal and all are valid only within the block in which they appear. If required in many blocks, they must be repeated in any subsequent block, if needed again.

◆ Programming Format

Each G-code (G45 to G48) is associated with a unique position compensation number, programmed with the address *H*. The *H* address points to the control system memory area storage number. On most Fanuc controls, the programmed letter can also be *D*, with exactly the same meaning. Whether the *H* or *D* address is used in the program, depends on the actual setting of a control system parameter.

A typical programming format for position compensation function is:

```
G91 G00 G45 X. . H. .
```

or

```
G91 G00 G45 X. . D. .
```

where the appropriate G-code (G45 through G48), is followed by the target position and number of the memory storage area (using *H* or *D* address).

Note that the example uses *incremental* and *rapid motion* modes and only *one axis*. Normally, the compensation has to be applied to both X and Y axes. However, only a single measured amount can be stored under either H or D number. Since it is most probable that the compensation value will be different for each axis, it must be specified on separate blocks, with two *different* offset numbers H (or offset numbers D), for example:

```
G91 G00 G45 X. . H31      (H31 STORES THE X VALUE)
G45 Y. . H32              (H32 STORES THE Y VALUE)
```

or

```
G91 G00 G45 X. . D31      (D31 STORES THE X VALUE)
G45 Y. . D32              (D32 STORES THE Y VALUE)
```

For the record, the H address is also used with another type of compensation, known as the *tool length offset* (or *tool length compensation*), described in Chapter 19. The D address is also used with another type of compensation, known as the *cutter radius offset* (or *cutter radius compensation*), described in Chapter 30.

The applicable preparatory G code will determine how the address H or address D will be interpreted. In the examples, more common address H will be used - Figure 17-1.

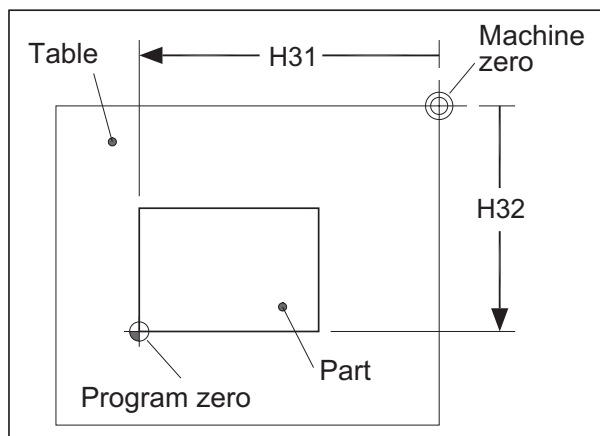


Figure 17-1

Position compensation - general concept

◆ Incremental Mode

A question may arise why the compensated motion is in incremental mode. Remember that the main purpose of position compensation is to allow a correction of the distance between machine zero and program zero. Normal use is when starting the tool motion *from* machine zero position. By default - and without any offsets - coordinate settings or active compensations, machine zero is the absolute zero, it is the *only* zero the machine control system 'understands' at the time.

Take the following example of several blocks, typically programmed at the beginning of a program with position compensation:

```
N1 G20
N2 G17 G80 T01
N3 M06
N4 G90 G00 G45 X0 H31      (NO X MOTION)
N5 G45 Y0 H32              (NO Y MOTION)
N6 . . .
```

This example illustrates a motion from machine zero (the current tool position), to program zero, which is the target position, along XY axes. Note the absolute mode setting G90 in block N4. Assume that the control system is set to H31 = -12.0000 inches. Control will evaluate the block and interpret it as programmer's intention to go to the absolute zero, specified by G90. It checks the current position, finds that it is at the absolute zero already and *does nothing*. There will be *no motion*, regardless of the compensation value setting, if the absolute motion is programmed to either X0 or Y0 target position. If G90 is changed to G91, from absolute to incremental mode, there *will* be a motion along the *negative* direction of X-axis, by the distance of exactly 12 inches and there will be a similar motion along Y-axis, in block N5. The conclusion? *Use position compensation commands in the incremental mode G91 only.*

◆ Motion Length Calculation

Look a little closer at how the control system interprets a position compensation block. Interpreting the way how the control unit manipulates numbers is important for understanding how a particular offset or compensation actually works. Earlier definition has stated that single increase is programmed with G45 command and single decrease with G46 command. Both G47 and G48 commands are of no consequence at the moment. Since both commands are tied up with a particular axis and with a unique H address, all possible combinations available must be evaluated:

- Either an *increase* or a *decrease* is programmed (G45 or G46)
- Axis target can have a *zero* value, or a *positive* value, or a *negative* value
- Compensation amount may have a *zero* value, or a *positive* value, or a *negative* value

In programming, it is important to set certain standards and consistently abide by them. For example, on vertical machining centers, compensation is measured *from* machine zero *to* program zero. That means a negative direction from the operator's viewpoint. The result is a logical decision to set negative compensation values as standard.

It is very crucial to understand *how* the control interprets information in a block. In position compensation, it evaluates the value stored in memory called by the address H (or D). If the value is zero, no compensation takes place. If the value of H is stored as a negative value, it *adds* this value to the value of axis target position and the result is the motion length and direction. For example, assume the memory register H31 stores the value of -15.0 inches, and the machine current location is at its zero position and the control axis setting is also set to zero. Then, the block

G91 G00 G45 X0 H31

will be interpreted as

$$-15.0 + 0 = -15.0000$$

resulting in the total motion of negative 15.0 inches along the X-axis.

If the setting of X-axis target position is a non-zero and positive, the same formula applies:

G91 G00 G45 X1.5 H31

will be interpreted as

$$-15.0 + 1.5 = -13.5000$$

However, the next example is *not* correct:

G91 G00 G45 X-1.5 H31

Here, the motion will try to go into the *positive* X-axis direction and the result will be X-axis overtravel. Since the value of X is negative, G45 command cannot be used and G46 command must be *used* instead:

G91 G00 G46 X-1.5 H31

will be interpreted as

$$-15.0 + (-1.5) = -15.0000 - 1.5 = -16.5000$$

G45 could have been left in the program and the negative offset value could have been changed to a positive value. This could be quite confusing and definitely not consistent, but it would work quite well. To see the different possibilities, program O1701 is not doing very much, except moving from machine zero to different positions and back to machine zero (G28 command refers to a machine zero return and is explained separately, starting on *page 153*).

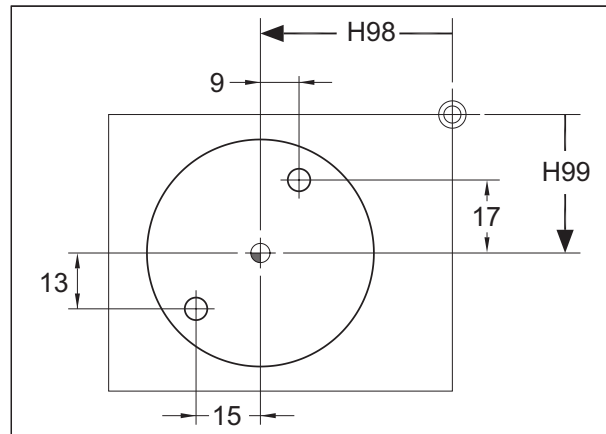


Figure 17-2

Position compensation applied to different target locations: zero, positive and negative - see O1701 program example

Figure 17-2 shows illustration for the following program example O1701. The logic applies to both X and Y axes exactly the same way. It is written in metric units and has been tested on Fanuc 11M, with the H address (D would work the same way). Position compensation values H98 and H99 were set to:

H98 = -250.000
H99 = -150.000

... for the X and Y axes respectively. Modal commands were not repeated:

```

O1701                                     (G45 AND G46 TEST)
N1 G21 G17
N2 G92 X0 Y0 Z0
N3 G90 G00 G45 X0 H98                   (ABS X0 TARGET)
N4 G46 Y0 H99                           (ABS Y0 TARGET)
N5 G28 X0 Y0

N6 G91 G00 G45 X0 H98                   (INC X0 TARGET)
N7 G46 Y0 H99                           (INC Y0 TARGET)
N8 G28 X0 Y0

N9 G90 G00 G45 X9.0 H98                 (ABS X+ TARGET)
N10 G46 Y17.0 H99                       (ABS Y+ TARGET)
N11 G28 X0 Y0

N12 G91 G00 G45 X9.0 H98                 (INC X+ TARGET)
N13 G46 Y17.0 H99                       (INC Y+ TARGET)
N14 G28 X0 Y0

N15 G90 G00 G45 X-15.0 H98              (ABS X- TARGET)
N16 G46 Y-13.0 H99                      (ABS Y- TARGET)
N17 G28 X0 Y0

N18 G91 G00 G45 X-15.0 H98              (INC X- TARGET)
N19 G46 Y-13.0 H99                      (INC Y- TARGET)
N20 G28 X0 Y0
N21 M30
%
```

Control system will process each motion block individually - either the way it was intended or the wrong way (symbol O/T means an *overtravel* condition, preceded with the axis and direction of the overtravel):

```

N3  G90  ->  G45  ->  0   .   .   .   no motion
N4  G90  ->  G46  ->  0   .   .   .   no motion
N6  G91  ->  G45  ->  0   .   .   .   X-250.0
N7  G91  ->  G46  ->  0   .   .   .   Y+ O/T

N9  G90  ->  G45  ->  +   .   .   .   X-241.0
N10 G90  ->  G46  ->  +   .   .   .   Y+ O/T
N12 G91  ->  G45  ->  +   .   .   .   X-241.0
N13 G91  ->  G46  ->  +   .   .   .   Y+ O/T

N15 G90  ->  G45  ->  -   .   .   .   X+ O/T
N16 G90  ->  G46  ->  -   .   .   .   Y-163.0
N18 G91  ->  G45  ->  -   .   .   .   X+ O/T
N19 G91  ->  G46  ->  -   .   .   .   Y-163.0

```

◆ Position Compensation Along the Z-axis

Position compensation feature usually applies only to X and Y axes and will not normally be used with the Z-axis. In most cases, the Z-axis has to be controlled by another kind of compensation - known as the *tool length offset*.

This very current and modern method is described in detail, starting on *page 135*. If the Z-axis is programmed with G45 or G46 commands, it will also be affected.

◆ Using G47 and G48

In the examples, position compensation feature was used only between the machine zero and program zero, as a method of determining where exactly is the part located on the table. Single increase using G45 and single decrease using G46 commands were used, because they were the only commands needed.

Commands G47 (double increase) and G48 (double decrease) are only necessary for a *very* simplified cutter radius offset and are not covered in this handbook because of their obsolescence. However, they can still be used.

◆ Face Milling - One Possible Application

➡ In a later section (*page 235*), the principles of face milling will be explained in more detail. In that chapter is a very good example of how to apply position compensation to offset diameter of the face mill in a clear position, regardless of its size. This is probably the only possible use of G45 and G46 commands in contemporary programming.

18

WORK OFFSETS

Using the method of *Work Offsets* for tool positioning based on machine zero is much faster and more efficient than using the older methods of position compensation functions G45 and G46 described in the previous chapter. Work offsets are also known as *Work Coordinate System*, or even as *Fixture Offsets*. Work offsets are much more efficient than using the position register commands G92 (milling systems) or G50 (turning systems). CNC programmers who do not know the meaning of *position compensation* functions or the meaning of *position register* commands, are most likely working with the most modern CNC machines only. However, there are many machines in industry that still require these rather obsolete functions. Knowing them well will increase the number of available programming tools.

This chapter describes the most modern methods to coordinate the relationship between machine zero reference position and the program zero reference point. Focus will be on the *Work Coordinate System* feature of a modern control, whether it is called the *Work Coordinate System* or the *Work Offsets*. The latter term seems to be more popular because it is a little shorter. Think of the work offsets as an *alignment* between two or more coordinate systems.

WORK AREAS AVAILABLE

Before some more detailed descriptions can be covered, just what *is* a work coordinate system - or a *work offset*?

Work offset is a method that allows the CNC programmer to program a part away from CNC machine, without knowing its exact position on the machine table. This is a very similar approach as in the position compensation method, but much more advanced and flexible. In work offset system, up to six parts may be set up on the machine table, each having a different work offset number. Programmer can move the tool from one part to another with absolute ease. To achieve this goal, a special preparatory command for the active work offset is needed in the program and the control system will do the rest. Control system will automatically make any adjustment for the difference between the two part locations.

Unlike position compensation function, two, three, or more axes may be moved simultaneously with work offsets, although the Z-axis for CNC machining centers is controlled independently, using G43 or G44 tool length offset commands. Commands relating to the Z-axis offset are fully described in the next chapter - see page 135.

In position compensation, to switch machining from one part to another within the same setup, the program has to contain a different compensation number from program zero of the previous part. Using the work offset method, *all program zeros are measured from machine zero position*, normally up to six, but more offsets are available.

The six work coordinate systems - or *work offsets* - that are available on Fanuc control systems are assigned the following preparatory commands:

G54	G55	G56	G57	G58	G59
-----	-----	-----	-----	-----	-----

When the control unit is turned on, the default coordinate system is normally G54, at least in most cases.

Basically, work offsets establish up to six independent work areas as a standard feature. Settings stored in the CNC unit are always distances measured *from machine zero to program zero*. As there are up to six work areas, up to six independent program zero positions can be defined. Figure 18-1 shows the basic relationships, using the default G54 setting.

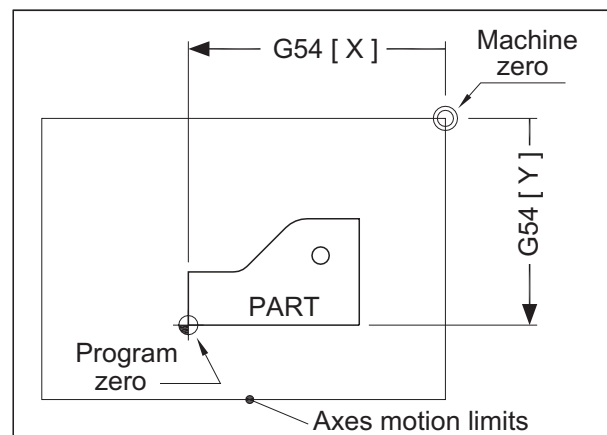


Figure 18-1
Basic relationships of the work offset method

The same relationships illustrated for default work offset apply exactly the same way for the other five available work offsets G55 to G59. Settings stored in the control system are always physically measured from machine zero position *to* program zero of the part, as determined by CNC programmer. Operator uses edge finders, corner finders and other devices to find these distances.

The distance *from* machine zero to program zero of each work area is measured separately along the X and Y axes and input into the appropriate work offset register of the control unit. Note that the measurement direction is *from* machine zero to program zero, never the other way around. If the direction is negative, the minus sign must be entered in the offset screen.

For comparison with the position register command G92, Figure 18-2 shows the same part set with the older method of G92 and machine zero as a start point. Note the *opposite* arrows designation, indicating the direction of measurement - *from* program zero to machine zero.

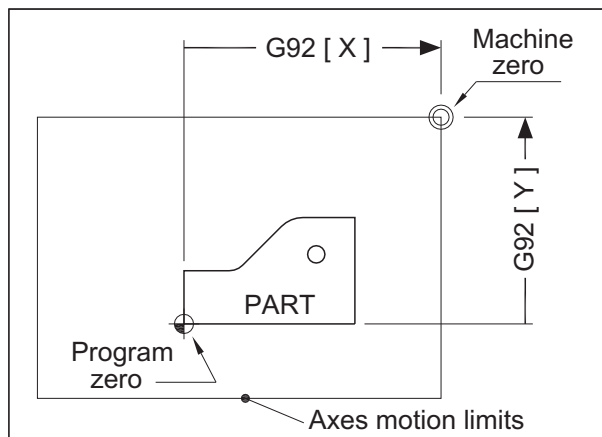


Figure 18-2

Basic relationships of the Position Register command G92

For work offsets G54 to G59, a typical entry into the coordinate offset position register will be the X-axis as a negative value, the Y-axis as a negative value and the Z-axis as a *zero* value, for the majority of vertical machining centers. This is done by CNC operator at the machine. Figure 18-3 shows an example of a typical control system entry.

01 (G54)
X -12.5543
Y - 7.4462
Z 0.0000

Figure 18-3

Typical data entry for the G54 work coordinate system

By using G54 to G59 settings in the program, control system selects the stored measured distances and the cutting tool may be moved to any position within the selected work offset simultaneously in both X and Y axes, whenever desired.

Part position on the machine table is usually unknown during programming process. The main purpose of work offset is to synchronize the actual position of the part as it relates to the machine zero position.

◆ Additional Work Offsets

The standard number of six work coordinate offsets is usually enough for most types of work. However, there are jobs that may require machining with more program reference points, for example, a multi-sided part on a horizontal machining table. What options do exist, if the job requires ten work coordinate systems, for example?

Fanuc offers - *as an option* - up to 48 additional work offsets, for the total of 54 (6+48). If this option is available on the CNC system, any one of the 48 work offsets can be accessed by programming a special G code:

G54.1 P..	Selection of additional work offset, where P = 1 to 48
-----------	--

➡ G54.1 P.. example :

G54.1 P1	Selection of additional work offset 1
G54.1 P2	Selection of additional work offset 2
G54.1 P3	Selection of additional work offset 3
G54.1 Px..	Selection of additional work offset x..
G54.1 P48	Selection of additional work offset 48

The utilization of additional work offsets in the program is exactly the same as that of standard commands:

N2 G90 G00 G54.1 P1 X5.5 Y3.1 S1000 M03

Most Fanuc controls will allow omission of the decimal portion of the G54.1 command. There should be no problem programming:

N2 G90 G00 G54 P1 X5.5 Y3.1 S1000 M03

The presence of P1 to P48 function within a block will select an *additional* work offset. If P1 to P48 parameter is missing, the default work offset command G54 will be selected by the control system.

WORK OFFSET DEFAULT AND STARTUP

If no work offset is specified in the program and the control system supports work offsets, the control will automatically select G54 - that is the *normal default selection*. In programming, it is always a good practice to program the work offset command and other default functions, even if the default G54 is used constantly from one program to another. Machine operator will have a better feel for the CNC program. Keep in mind that the control still has to have accurate work coordinates stored in the G54 register.

In the program, work offset may be established in two ways - either as a separate block, with no additional information, as in this example:

N1 G54

Work offset can also be programmed as part of a startup block, usually at the head of program or at the beginning of each tool:

N1 G17 G40 G80 G54

The most common application is to program the appropriate work offset G-code in the same block as the first cutting tool motion:

N40 G90 G54 G00 X5.5 Y3.1 S1500 M03

Figure 18-4 illustrates this concept. In the above block N40, the absolute position of the tool has been established as X5.5Y3.1, *within* the G54 work offset. What will *actually* happen when this block is processed?

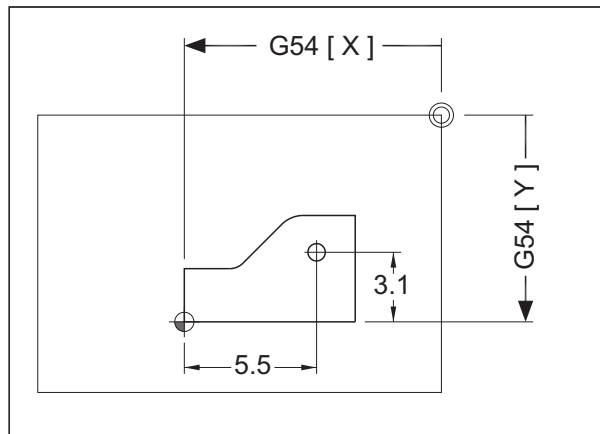


Figure 18-4

Direct tool motion to a given location using G54 work offset

Note that there are no X or Y values associated with G54 command in the illustration. There is no need for them. CNC operator places the part and fixture in *any suitable location* on the machine table, squares it up, finds how far is the program zero away from machine zero and enters these settings into the control register, under G54 heading. The entry could be either manual or automatic.

Assume for a moment, that after setup, the measured distances from machine zero to program zero were X-12.5543 and Y-7.4462. The computer will determine the actual motion by a simple calculation - it will always *add* the programmed target value X to the measured value X, and the programmed target value Y to the measured value Y.

Actual tool motion in the block N40 will be:

$$\begin{aligned} X &= -12.5543 + 5.5 = -7.0543 \\ Y &= -7.4462 + 3.1 = -4.3462 \end{aligned}$$

These calculations are absolutely unnecessary in everyday programming - they are only useful to the thorough understanding of how the control unit interprets given data.

The whole calculation is so consistent, it can be assigned into a simple formula. For simplicity, the settings of the EXT (external or common) offset are not included in the formula, but are explained separately, later in the chapter:

$$L = D + T$$

where ...

- L = Actual motion length (distance-to-go displayed)
- D = Measured distance from machine zero
- T = Programmed absolute target position (axis value)

Be very careful when adding a negative value - mathematically, double signs are handled according to the standard rules:

PLUS and PLUS <i>becomes</i> PLUS	$a + (+ b) = a + b$
PLUS and MINUS <i>becomes</i> MINUS	$a + (- b) = a - b$
MINUS and PLUS <i>becomes</i> MINUS	$a - (+ b) = a - b$
MINUS and MINUS <i>becomes</i> PLUS	$a - (- b) = a + b$

In the example, plus and minus combination creates a negative calculation:

$$-10 + (-12) = -10 - 12 = -22$$

If any other work offset is programmed, it will be automatically replaced by the new one, *before* the actual tool motion takes place.

◆ Work Offset Change

A single CNC program may use one, two, or all work offsets available. In all multi-offset cases, work offset setting stores *the distance from machine zero to program zero of each part in the setup*.

For example, if there are three parts mounted on the table, each individual part will have its own program zero position associated with one work offset G-code.

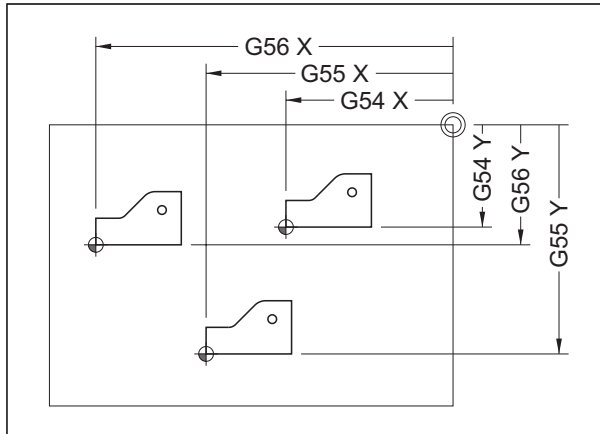


Figure 18-5

Using multiple work offsets in one setup and one program.
Three parts shown in the example.

Compare all possible motions in Figure 18-5:

G90 G54 G00 X0 Y0

... will rapid from the *current* tool position, to the program zero position of the *first* part.

G90 G55 G00 X0 Y0

... will rapid from the *current* tool position, to the program zero position of the *second* part.

G90 G56 G00 X0 Y0

... will rapid from the *current* tool position, to the program zero position of the *third* part.

Of course, the target position *does not* have to be part zero (program zero) as shown in the example - normally, the tool will be moved to the first cutting position right away, to save cycle time. The following program example will illustrate that concept.

In the example, a single hole will be spot drilled on each of the three parts to a calculated depth of Z-0.14 (program O1801). Study the simplicity of transition from one work offset to another - there are no cancellations - just a new G code, new work offset. The control will do the rest:

```
O1801
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X5.5 Y3.1 S1000 M03 (G54 USED)
N4 G43 Z0.1 H01 M08
N5 G99 G82 R0.1 Z-0.14 P100 F8.0
N6 G55 X5.5 Y3.1 (SWITCH TO G55)
```

```
N7 G56 X5.5 Y3.1 (SWITCH TO G56)
N8 G80 Z1.0 M09
N9 G91 G54 G28 Z0 M05 (SWITCH BACK TO G54)
N10 M01
...
```

Blocks N3 through N5 relate to the first part, within the G54 work offset. Block N6 will spot drill the hole of the second part of the same setup, within G55 work offset and block N7 will spot drill the hole of the third part of the same setup, within G56 work offset. Note the return to G54 work offset in block N9. Return to the default coordinate system is not required - it is only a suggested good practice when a particular tool operation is completed. Work offset selection is always modal - take care of the transitions between tools from one work offset to another. Bringing back the default offset G54 may always be helpful at the end of each tool, for safety.

If all these blocks are in the *same* program, the control unit will automatically determine the *difference* between the current tool position and the same tool position within the next work offset. This is the greatest advantage of using work offsets - an advantage over position compensation and position register alternatives. All mounted parts may be identical or different from each other, as long as they are in the same positions for the whole setup.

◆ Z-axis Application

So far, there was a conspicuous absence of the Z-axis from all discussions relating to the work offset. That was no accident - it was intentional. Although any selected work offset can apply to the Z-axis as well, and with exactly the same logic as for X and Y axes, there is a better way of controlling the Z-axis. The method used for Z-axis is in the form of G43 and G44 commands that relate specifically to *tool length compensation*, more commonly known as the *tool length offset*. This important subject is discussed separately in the next chapter. In the majority of programming applications, work offset is used only within the XY plane. This is a typical control system setting and may be represented by the following setup example of the stored values within the control register:

```
(G54) X-8.761 Y-7.819 Z0.000
(G55) X-15.387 Y-14.122 Z0.000
(G56) X-22.733 Y-8.352 Z0.000
(G57) ...
```

The Z0 offset entry is very important in the examples and in the machine control. Specified Z0 means that the coordinate setting for the Z amount (representing the height of part) does not change from one part to another, even if the XY setting does.

The only time there is a need to consider Z-axis within the work offset setting is in those cases, where the height of each part in the setup is *different*. So far, only the XY positions were considered, as they had been the ones changing.

If the Z amount changes as well, that change must be considered by modifying the coordinate register setting of the control. This is the responsibility of CNC operator, but the programmer can learn an important lesson as well.

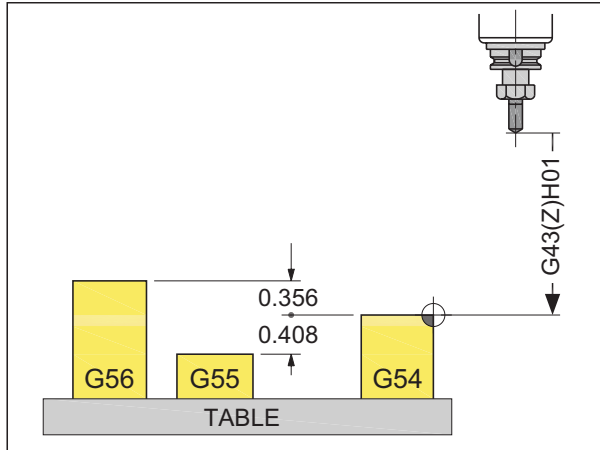


Figure 18-6

Setting of work offsets for a variable part height

Figure 18-6 shows some typical and common possibilities used for special parts that have a variable height within the same tool setup. The difference between part heights has to be always known, either from the part drawing specifications or from actual measurements at the machine.

If the previous multi-offset example for XY setting are also adapted for the Z-axis, the work offset can be set up for parts within the same setup, but with variable heights. This variable height is controlled by the Z-axis. Result of the setting will reflect the difference in height between the measured Z-axis surface for one part and the measured Z-axis surface for the other parts. Based on data in the previous example, combined with the Z values shown in Figure 18-6, the control system settings may look like this:

```
(G54) X-8.761 Y-7.819 Z0.000
(G55) X-15.387 Y-14.122 Z-0.408
(G56) X-22.733 Y-8.352 Z0.356
```

The important thing to know about control of the Z-axis within the selected work offset is that it works in very close *conjunction* with the tool length offset, discussed in the next chapter. Stored amount of the Z-axis setting within a work offset will be applied to the actual tool motion and used to *adjust* this motion, according to the setting of tool length offset. An example may help here.

For instance, if the tool length offset of a particular cutting tool is measured as Z-10.0, the actual motion of such a tool to program zero along Z-axis will be -10.0 inches within G54 work offset, -10.408 within G55 work offset, and -9.644 within G56 offset - all using the examples in the previous illustration, shown in Figure 18-6.

HORIZONTAL MACHINE APPLICATION

Machining several parts in a single setup is done quite frequently on CNC vertical machining centers. The multiple work offset concept is especially useful for CNC horizontal machining centers or boring mills, where many part faces may have to be machined during a single setup.

Machining two, three, four, or more faces of the part on a CNC horizontal machining center is a typical everyday work in many companies. For this purpose, the work offset selection is a welcome tool. For example, *program zero* at the pivot point of the indexing table can be set for the X and Y axes. Program setting of the Z axis may be in the same position (the pivot point of the indexing table) or it can be on the face of each indexed position - either choice is acceptable. Work offset handles this application very nicely, up to six faces with a standard range of G-codes.

There is no significant difference in the programming approach - the switch from one work offset to another is programmed exactly the same way as for vertical machining applications. The only change is that the Z-axis will be retracted to a clear position and the table indexing will usually be programmed between the work offset change.

Figure 18-7 illustrates a typical setting for four faces of a part, where Z0 is at the top of each part face. There could be as many faces as there are table indexing positions. In either case, the programming approach would be similar if Z0 were at the center of indexing table, which is also quite a common setup application. A separate chapter covers details relating to horizontal machining - see page 445.

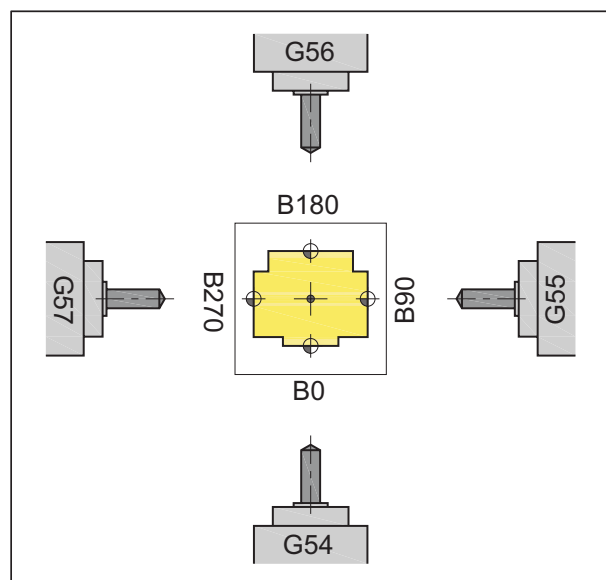


Figure 18-7

Example of work offsets applied to a horizontal machining center

EXTERNAL WORK OFFSETS

A careful look at a typical work offset screen display reveals one special offset that is identified by one of the following designations:

- 00 (EXT)
- 00 (COM)

The two zeros - 00 - indicate that this work offset is *not* one of the standard six offsets G54-G59. These offsets are identified by numbers 01 through 06. The 00 designation also implies that this is not a programmable offset, at least not by using standard CNC programming methods. *Fanuc Macro B* option does allow programming this offset.

The abbreviation *EXT* means *External*, and the abbreviation *COM* means *Common*. Machine control will have one or the other designation, but not both. As a matter of curiosity, the *COM* designation is found on older controls, whereby the *EXT* designation is more recent. The reason? With the explosion of personal computer market, the *COM* abbreviation has become de facto standard abbreviation for the word *communications*. As Fanuc controls also support several communication methods, including the connection with a personal computer, some time ago, the *COM* offset designation has been replaced with the designation *EXT*, to prevent possible confusion between the two abbreviations used in computing.

Either abbreviation refers to the same offset and has the same purpose. On the screen display, this special offset is usually located *before* or *above* the offset for G54, for example, as illustrated in *Figure 18-8*:

00 (EXT)		01 (G54)	
X	0.0000	X	-12.5543
Y	0.0000	Y	- 7.4462
Z	0.0000	Z	0.0000

Figure 18-8

Example of an *EXT* (external) work offset display (*EXT* = *COM*)

The major difference between an external or common work offset is that it is not programmable with any particular G-code. Its setting is normally set to zero for all axes. Any *non-zero* setting will activate this work offset in a very important way:

***** IMPORTANT *****
Any setting of external work offset will always affect ALL work offsets used in the CNC program

All six standard work offsets, as well as any additional work offsets, will be affected by the settings stored in the external work offset, based on the setting of each axis. Because all programmable coordinate systems will be affected, the name for this special offset is *Common Work Offset* or more often, the *External Work Offset*.

LATHE APPLICATIONS

Originally, work coordinate system was designed for CNC machining centers only. It did not take long to apply it to CNC lathes as well. The operation, logically and physically, is identical to that for machining centers. Using work offsets for CNC lathes eliminates the awkward use of G50 or G92 and makes CNC lathe setup and operation much faster and easier.

◆ Types of Offsets

The main difference in applying work offsets on a lathe is that seldom will there be a need for more than one work offset. Two work offsets are a possibility, three or more are used for some very special and complex setups. G54 to G59 commands are available on all modern CNC lathes and it is quite customary to ignore work offset selection in the program, unless more than one offset is used. That means the CNC lathe programmer depends on the default G54 setting as a rule.

Two special offset features found on the most latest control systems are *Geometry* and *Wear* offsets, either on the same screen display, or on separate screens, depending on the control model.

◆ Geometry Offset

Geometry offset is the equivalent of a work offset as it is known from milling controls. It represents the distance from tool reference point to program zero, measured from machine zero along a selected axis. Typically, on a slant bed CNC lathes, with tool turret *above* the spindle centerline, the geometry offset for both X and Z axes will be negative. *Figure 18-9* illustrates reasonable geometry values for a drill, turning tool and boring bar (T01, T02, T03).

GEOMETRY OFFSET				
No.	X-offset	Z-offset	Radius	Tip
01	-8.6330	-2.3630	0.0000	0
02	-8.6470	-6.6780	0.0469	3
03	-9.0720	-2.4950	0.0313	2
04	0.0000	0.0000	0.0000	0
05	0.0000	0.0000	0.0000	0

Figure 18-9

Typical data entries for a lathe tool *GEOMETRY* offset

◆ Wear Offset

Wear offset is also known and used on milling controls, but only for the tool length offset and cutter radius offset, not for work coordinate system (work offset).

On CNC lathes, the purpose of wear offset is identical to that for machining centers. This offset compensates for the tool wear and is also used to make fine adjustments to the geometry offsets. As a rule, once the geometry offset for a given tool is set, that setting should be left unchanged. Any adjustments and fine tuning of actual part dimensions should be done by the wear offset only.

WEAR OFFSET				
No.	X-offset	Z-offset	Radius	Tip
01	0.0000	0.0000	0.0000	0
02	-0.0060	0.0000	0.0469	3
03	0.0000	0.0040	0.0313	2
04	0.0000	0.0000	0.0000	0
05	0.0000	0.0000	0.0000	0

Figure 18-10

Typical data entries for a lathe tool WEAR offset

Figure 18-10 shows some reasonable sample entries in the wear offset registers. The tool radius and tip number settings appear in both displays and the display in both screens is automatic after the offset value input. Tool nose radius and tool tip orientation number are unique to CNC lathe controls.

◆ Tool and Offset Numbers

Just like tools on CNC machining centers have numbers, they have numbers on CNC lathes as well. Usually, only one coordinate offset is used, but different tool numbers. Remember, the tool number for a lathe has *four digits*, for example, T0404:

- The first two digits select the tool indexing station (turret station) *and* the geometry offset number. There is no choice here. Tool in station 4, for example, will also use geometry offset number 4
- The second two digits are for the wear offset register number only. They do not have to be the same as the tool number, but it makes sense to match the numbers, if possible

Depending on the control model and the display screen size, tool offset register may have a separate screen display (page) for geometry and wear offsets, or both offset types may be shown on the same screen display. Work offset settings (work coordinates) are always placed in the *Geometry* offset column.

TOOL SETUP

In the next three illustrations is a very similar layout as that shown on *page 119*, describing the use of G50 register method (position register command used in the program). *Compare the two illustrations!*

Setup of a CNC lathe is identical in both cases, except for the method and purpose of tool position measuring. All illustrations in the applications also match the reasonable data entered in tool geometry and tool wear offset screens of the control.

Typical settings along the X-axis are always negative (as shown in illustrations), typical settings along the Z-axis are usually negative. A positive value is also possible, but that means the tool is above work and tool changing can be very dangerous. *Watch out for such situations!*

The actual setting procedures are subject of a CNC machine operation training and not practical to cover in a programming handbook. There are additional methods, also part of machine training, that allow faster tool setting, using one tool as a master and setting all the remaining tools *relative to the master tool*.

◆ Center Line Tools

Tools that work on the spindle center line are tools that have their tool tip located on the center line during machining. This area covers all center drills, spot drills, various drills, reamers, taps, even end mills used for flat bottom holes. At the same time, it disqualifies all boring bars, since their tool tip does not normally lie on the spindle center line during machining. Center line tools are *always* measured from the center line of the tool to the center line of the spindle along the X-axis and from the tool tip to program zero along the Z-axis. Figure 18-11 illustrates a typical setting for center line tools.

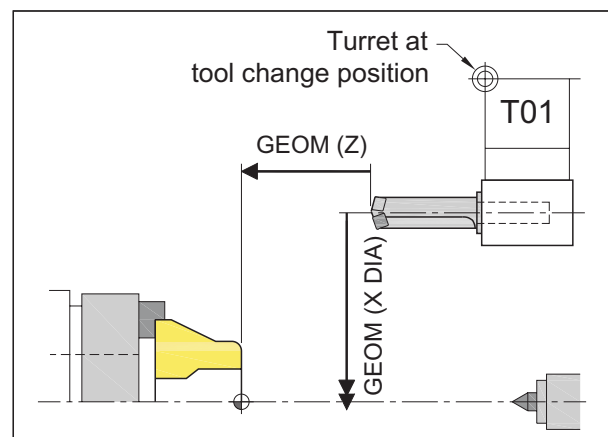


Figure 18-11

Typical geometry offset setting for CENTER LINE tools

◆ Turning Tools

Turning tools - or *external* tools - are measured from the imaginary tool tip to program zero, along the X-axis (as a negative diameter) and along the Z-axis, usually as a negative value as well. Keep in mind that if the cutting tool insert (for turning or boring) is changed from one radius to another radius in the *same* tool holder, the entered setup amount must also change. Such a change may be marginal, but even a marginal change is enough to cause a scrap, so a good care is needed. For turning, be extra careful for a tool nose radius that changes from a larger size to a smaller size, for example, from 3/64 (R0.0469) to 1/32 (R0.0313).

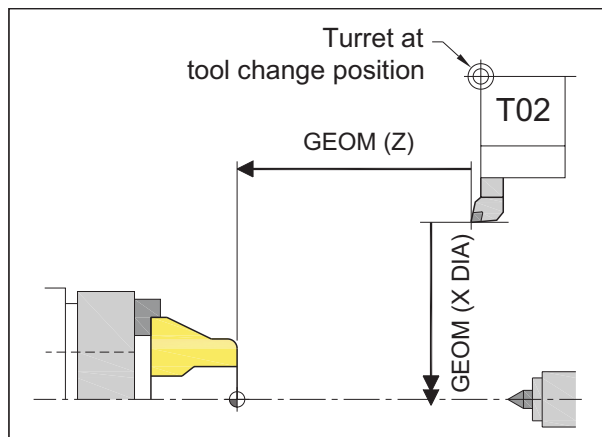


Figure 18-12

Typical geometry offset setting for EXTERNAL tools

Figure 18-12 illustrates a typical geometry setting for a turning (external) tool and Figure 18-13 illustrates a typical geometry setting for a boring (internal) tool.

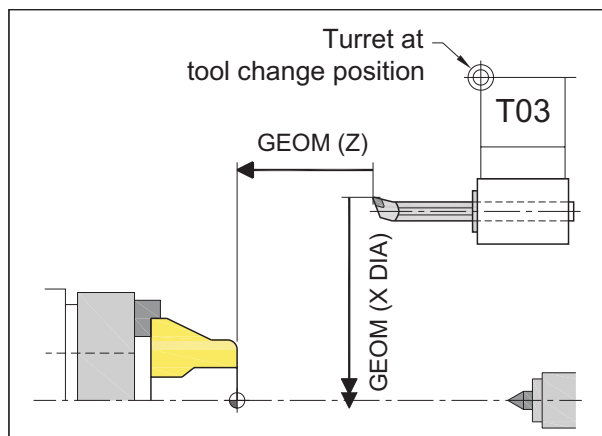


Figure 18-13

Typical geometry offset setting for INTERNAL tools

◆ Boring Tools

Boring tools - or *internal* tools - are always measured from the imaginary tool tip to program zero, along X-axis (typically as a negative diameter) and along Z-axis, typically as a negative value as well. In majority of cases, the X setting amount of a boring tool will be noticeably larger than that for a turning or other external tool.

For boring operations, same as for turning operations, also be extra careful for a tool nose radius that changes from a larger size to a smaller size. It is the same as for a turning tool. Scrap can be made very easily.

◆ Command Point and Tool Work Offset

For various reasons, it is quite common to change a cutting insert in the middle of work, primarily to maintain favorable cutting conditions and to keep dimensional tolerances within drawing specifications. Cutting inserts are manufactured to very high standards, but a certain tolerance deviation should be expected between inserts obtained from different sources. If changing an insert, it is advisable to adjust the wear offset for precision work, in order to prevent scrapping the part.

Tool holders accept inserts of the same shape and size but with a different nose radius. Always be cautious when replacing an insert with an insert that has a larger or smaller tool nose radius. The offset has to be adjusted in both axes, by the proper amount.

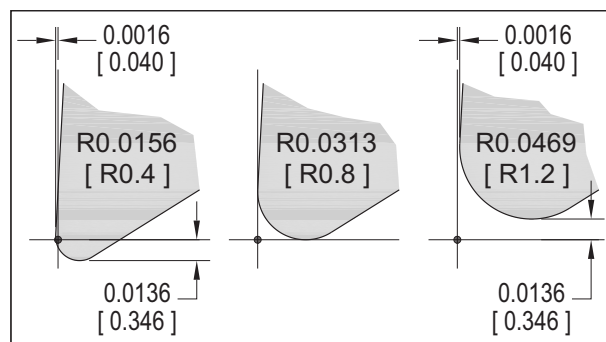


Figure 18-14

Setting error caused by a different insert radius in the same holder

Figure 18-14 example shows the most common standard setting for a 1/32 (0.0313) [0.8 mm] nose radius (middle), and the setting error for a radius that is smaller (left) and one that is larger (right). Dimensions indicate the amount of error for the particular insert shown in the example. Metric dimensions are shown in the brackets.

When changing an insert, adjust the required offset(-s)

19

TOOL LENGTH OFFSET

So far, we have looked at two methods of compensation for the actual position of *cutting tool* in relation to the *machine reference point*. One method was the older type, using *position compensation*, the other was the contemporary *work coordinate system* method (*work offset*). In both cases, the emphasis was only on the X and Y axes, not on the Z-axis. Although the Z-axis could have been included with either method, the results would not have been very practical. Main reason is the nature of the CNC work.

Normally, programmer decides on the setup of a part in the fixture and selects the appropriate location of XYZ program zero (part reference point or part zero). When using work offsets, XY axes are always measured from the *machine reference point* to the *program zero* position. By a strict definition, the same rule applies to the Z-axis as well. The major difference is that both measured XY values will *remain unchanged for all tools*, whether there is one tool used or one hundred tools. That is not the case with Z-axis.

The reason? Each tool has a *different length*.

GENERAL PRINCIPLES

Length of each cutting tool has to be accounted for in every program for a CNC machining center. Since the earliest applications of numerical control, various techniques of programming tool length have emerged. They all belong into one of two basic groups:

- Actual tool length is known
- Actual tool length is unknown

Needless to say, each group requires its own unique programming technique. To understand the concept of tool length in CNC programming, it is important to understand meaning of the phrase *actual tool length*. This length is sometimes known as the physical tool length or just tool length and has a very specific meaning in CNC programming and setup.

◆ Actual Tool Length

Let's evaluate a simple tool first. By holding a typical drill, we can determine its physical length with a measuring device. In human terms, a six inch long drill has a length of six inches, measured from one end to the other. In CNC programming that is still true, but not quite as relevant. A drill - or any other cutting tool - is normally mounted in a tool holder and only a *portion* of the actual tool projects

out, the rest is hidden in the holder. Tool holder is mounted in the spindle by means of a standardized tooling system. Tool designations, such as the common sizes HSK63, HSK100, BT40 and CAT50, are examples of established European and American standards. Any tool holder within its category will fit any machine tool designed for that category. This is just one more precision feature built into the CNC machine.

Length of a tool for the purposes of CNC programming must always be associated with the tool holder and in relation to the machine design. For that purpose, manufacturers build a precision reference position into the spindle, called the *gauge line* (or *gage line*).

◆ Gauge Line

When a tool holder with cutting tool is mounted in the spindle of CNC machine, its own taper is mounted against an opposite taper inside the spindle and held in tightly by a pullbar. Precision of manufacturing allows for a constant location of the tool holder (any tool holder) in the spindle. This position is used for reference and is commonly called the *gauge line*. As the name suggests, it is an imaginary reference line used for gauging - or *measuring* - along the Z-axis - *Figure 19-1*.

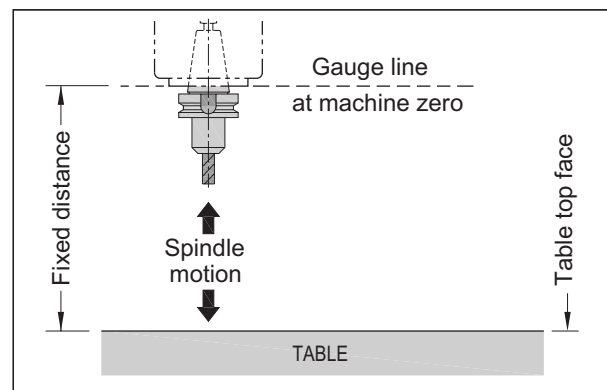


Figure 19-1

Typical front view of a CNC vertical machining center

Gauge line is used for accurate measuring of tool length and any tool motion along the Z-axis. Gauge line is determined by the machine manufacturer and is closely related to another precision face, called the *machine table*, actually, the *table top face*. Gauge line is one side of a plane that is parallel with another plane - the *table top face*.

◆ Table Top Face

Every CNC machining center has a built-in machine table on which the fixture and part are mounted. Top of the machine table is precision ground to guarantee flatness and squareness for the located part.

In addition, machine table is located a certain fixed distance from the gauge line. Just like the position of tool holder in the spindle cannot be changed, the position of table (even for a removable table using a palette system) cannot be changed either. Surface of the table creates another reference plane that is related to the gauge line and parallel to it as well. This arrangement allows to accurately program a tool motion along the Z-axis.

Tool length offset (compensation) can be defined:

Tool length offset is a procedure that corrects the difference between programmed tool length and its actual length

The most significant benefit of tool length offset in CNC programming is that it enables the programmer to design a complete program, using as many tools as necessary, *without actually knowing the actual length of any tool*.

TOOL LENGTH OFFSET COMMANDS

Fanuc systems and several other machine controls offer three commands relating to the tool length offset - all are preparatory G-commands:

G43 G44 G49

All three commands are only applicable to the Z-axis. Unlike work offset commands G54-G59, G43 or G44 cannot be used without additional specification. They can only be used with an offset number designated by the address H. Address H must be followed by up to three digits, depending on the number of offsets available within the control system:

G43	Positive tool length offset
G44	Negative tool length offset
G49	Tool length offset cancel
H00	Tool length offset cancel
H..	Tool length offset number selection

Tool length offset should always be programmed in the absolute mode G90. A typical program entry will be the G43 or G44 command, followed by the Z-axis target position (based on part zero for Z) and the H-offset number:

N66 G43 Z1.0 H04

This is also a convenient block to add coolant function M08 for the current tool, if desired:

N66 G43 Z1.0 H04 M08

Resulting motion in the example will be to 1.0 inch above the part zero. Control system will calculate the *distance to go*, based on the value of H-offset stored by the CNC operator during setup.

TOOL OFFSET (LENGTH)

No.	GEOMETRY	WEAR
001	-6.7430	0.0000
002	-8.8970	0.0000
003	-7.4700	0.0000
004	0.0000	0.0000
005	0.0000	0.0000
006	0.0000	0.0000
...

Figure 19-2

Typical tool length offset entry screen

Figure 19-2 shows a typical screen for tool length offset entry. Note that the actual display will vary from one control to another and the wear offset column may not be available on some controls. The wear offset (if available) is only used for *adjustments* to the tool length and is separate from the geometry offset.

G44 command is hardly ever used in a program - in fact, it has the dubious distinction of being the least used commands of all Fanuc G-codes. Its comparison with G43 is described later in this chapter.

Many CNC programmers and operators may not realize that the Z-axis setting in a work offset (G54-G59) is also very important for the tool length offset. The reason why will be clear in the coming descriptions of different methods of tool length offset setting.

Some programming manuals suggest the older G45 or G46 commands can also be used for tool length offset. Although this is still true today and may have had some advantages in the early days, it is best to avoid them. First, the position commands are practically not used anymore and, second, they can also be used with the X and Y axes and do not truly represent the Z-axis exclusively.

◆ Distance-To-Go in Z-axis

In order to interpret how the CNC system uses tool length command, the programmer or operator should be able to calculate the *distance-to-go* of the cutting tool. Logic behind the tool length offset is simple:

- Stored amount of the H-offset will be *added* to the target Z-position if G43 is used, because G43 is defined as the *positive* tool length offset
- Stored amount of the H-offset will be *subtracted* from the target Z-position if G44 is used, because G44 is defined as the *negative* tool length offset

Target position in both cases is the absolute Z-axis coordinate in the program. If the Z-axis setting of work offset (G54-G59), the length offset stored amount, and the Z-axis target are all known, *distance-to-go* can be accurately calculated. Control system will use this formula:

$$Z_d = W_z + Z_t + H$$

where ...

- Z_d = Distance-to-go along Z-axis (actual travel)
- W_z = Work coordinate value for Z-axis
- Z_t = Target position in Z-axis (Z-coordinate)
- H = Stored amount of the applied H-offset number

➡ Example - $W_z = 0$:

G43 Z0.1 H01 where:

G54 in Z is set to Z0, Z-axis target position is 0.1 and H01 is set to -6.743, then the distance-to-go Z_d will be:

$$\begin{aligned} Z_d &= 0 + (+0.1) + (-6.743) \\ &= 0 + 0.1 - 6.743 \\ &= -6.643 \end{aligned}$$

Displayed *distance-to-go* will be Z-6.643.

In order to make sure the formula is always correct, try to change a few values.

➡ Example - $W_z = 0.0200$:

In this example, the program contains block

G43 Z1.0 H03 where:

G54 in Z is set to 0.0200, Z-axis target is Z1.0 and the stored amount of H03 is -7.47:

$$\begin{aligned} Z_d &= (+0.02) + (+1.0) + (-7.47) \\ &= 0.02 + 1.0 - 7.47 \\ &= -6.45 \end{aligned}$$

The result is correct, tool will travel along the Z-axis, towards the part and *distance-to-go* will be Z-6.45.

In the last example, a negative target position is shown:

➡ Example - $W_z = 0.0500$:

Program block contains a negative Z-coordinate:

G43 Z-0.625 H07 where:

G54 along Z is set to 0.0500, Z-axis target is -0.625 and offset H07 stores -8.28 setting. *Distance-to-go* calculation uses the same formula, but with different values:

$$\begin{aligned} Z_d &= (+0.05) + (-0.625) + (-8.28) \\ &= 0.05 - 0.625 - 8.28 \\ &= -8.855 \end{aligned}$$

Again, the formula works correctly and can be used for *any* distance-to-go calculation along Z-axis. Experimenting with other settings may also be useful.

TOOL LENGTH SETUP

The length of a tool used for machining (consisting of cutting tool and tool holder), can be set directly on the CNC machine or away from it. These setup options are often called *on-machine* or *off-machine* tool length setups. Each option has an advantage and it has its corresponding disadvantage. They both share a certain relationship to the gauge line, as it applies to the tool length or its projection (extension). These two setup options are directly opposite to each other and often cause philosophical divisions (or at least some friendly disagreements) among CNC programmers. Evaluate each setup option and compare its advantages with its disadvantages. Which one appears to be somewhat better will depend on many additional factors as well.

Both options require involvement of two people, or at least two professional skills - typically CNC programmer and CNC operator. The question narrows down to *who is going to do what - and when*. To be fair, both sides have to do something. Programmer has to identify all selected tools by their number (the T-address) and assign tool length offset numbers, for example, G43 with the H-address for each tool. Operator has to physically set all tools into their holders and register measured amounts of H-addresses into the CNC system memory (offset registry).

◆ On-Machine Tool Length Setting

In technical terms, bulk of *on-machine* setting requires the work of a CNC operator. Typically, the operator places a tool into the spindle and measures distance the tool travels *from machine zero to part zero* (program zero). This work can only be done between jobs and is definitely nonproductive. It can be justified under certain circumstances, particularly for jobbing shops and short-run jobs or for machine shops with very few people. Although setting of a large number of tools will take much longer than setting of a few tools, there are setup methods available to the operator that allow reasonably speedy *on-machine* tool length setup, namely using the *master tool* method, described later in this section. One major benefit of this method is that it does not require the expense of additional equipment and a skilled person to operate it.

◆ Off-Machine Tool Length Setting

In technical terms, *off-machine* setting requires the work of a skilled tool setter or a CNC operator. Since such setting is done away from the machine, a special equipment is required, adding to the overall cost of manufacturing. This equipment can be a simple fixture with a height gage (even made *in-house*), or a more expensive, commercially available digital display device.

◆ Tool Length Offset Amount Register

Whichever method of tool length setting is used, it produces a measured amount that represents length of the selected tool. This amount is useless by itself and must be somehow supplied to the program, before the job is machined. CNC operator must register the measured amount into the system, under proper heading on the control panel.

Control system contains a special registry for tool length offset, usually under the heading of *tool length setting*, *tool length offset*, *tool length compensation* or just *offset*. Regardless of the exact heading, setting procedure is to make sure the measured length is entered into the control, so it can be used by the program during its processing. Measured length is always within the Z-axis machine travel limits, yet still allows for sufficient clearances for both part and tool changes.

To understand the tool length offset, try to fully understand the Z-axis motion and geometry of the CNC machine first. On vertical and horizontal machining centers, look at the XZ plane, which is the top of part for both. Principles for both machine types are identical, but primary focus will be on the vertical machining center layout.

Z-AXIS RELATIONSHIPS

To understand the general principles of tool length offset, look at a schematic illustration of a typical setup for a vertical machining center - *Figure 19-3*.

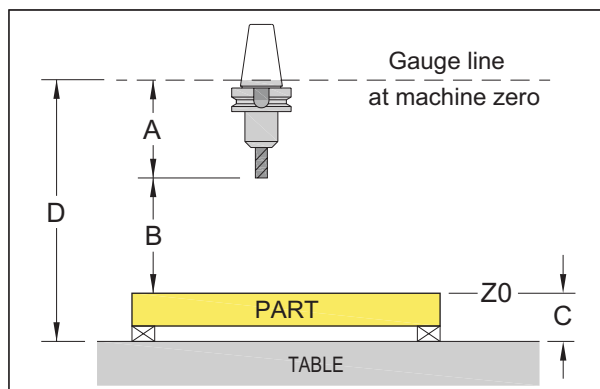


Figure 19-3

Z-axis relationships of the machine, cutting tool, table top face, and the part height

The illustration represents common setup of a CNC vertical machining center, looking from the machine front position, a typical operator's viewpoint. Spindle column is located at the machine zero position. This is the limit switch position for positive Z-axis travel and is necessary for automatic tool change on virtually all machining centers. All four illustrated dimensions are either known, can be found in various instruction or service manuals, or can be physically measured. They are always considered as *known* dimensions or *given* dimensions and used as *equally critical* for accurate machine setup:

- Distance between tool gauge line and tool cutting point
... *dimension A in the illustration*
- Distance between tool cutting point and Z0 (program zero of the part)
... *dimension B in the illustration*
- Height of part (distance between table top and Z0 of the part)
... *dimension C in the illustration*
- Total of all three previous dimensions (distance between tool gauge line and table top)
... *dimension D in the illustration*

It is rather rare that the programmer or operator would always know all four dimensions. Even if that were possible, some calculations would not be worthwhile doing. The reality is that only *some* dimensions are known or can be found out relatively easily.

In the illustration, dimension D is always known, because it is the distance determined by machine manufacturer. It may not be possible or practical to know the C-dimension (height of part with clearances), but with planning and a common setup, this dimension can be known as well.

That leaves dimension A - the distance between tool gauge line and its cutting point. There is no other method to find this dimension, but to actually measure it. In the earlier days of numerical control, this length A had to be always known and embedded in the program. Because of the inconveniences involved in finding this dimension, more practical methods have developed later.

Today, three methods are generally considered in programming tool length setup, including the original method:

- Preset tool method is the original method
... *it is based on an external tool setting device*
- Touch-off method is the most common method
... *it is based on measurement at the machine*
- Master tool method is the most efficient method
... *it is based relative to the length of the longest tool*

Each method has its benefits. CNC programmer considers these benefits and chooses one method over the other. Applications of these methods and operations do not relate to the programming process directly - they are methods of *physical setup* on the machine only. For proper understanding of the subject by CNC programmers, they are included here as well. Regardless of which setting method is chosen, include a reference to the selected setting in the program, in form of a comment or a message.

◆ Preset Tool Length

Some users prefer to *preset* the length of cutting tools *away* from the machine, rather than during machine setup. This has been the original method of setting tool lengths. There are some benefits in this approach - the most notable is elimination of non-productive time spent during setup. Another benefit applies to horizontal machining centers, where program zero is often preset to the center of rotary or indexing table. There are disadvantages as well. Presetting tool length away from machine requires an external device, known as the *tool presetter*, which could be a relatively expensive addition to CNC machine.

Using a tool presetter, all cutting tools are set at the external device, while CNC machine runs production job. There is no measuring on the machine when jobs do change. All the operator has to do, is to enter the previously measured offset amounts into their respective registers. Even that portion of setup can be done through the program by using the optional G10 command (if available).

This method also requires a qualified person responsible for presetting all cutting tools. A large number of small and medium users with vertical machining centers cannot afford the additional expenses and do the setting of cutting tools during part setup, mostly using the touch-off method. This method may also be suitable choice when small job runs are machined. The touch-off method of tool length offset setting is described in the next section.

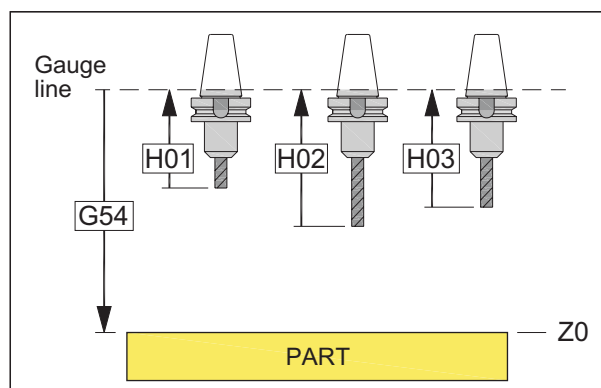


Figure 19-4

Tool length preset away from the machine (tool presetter method). Work offset (G54-G59) must be used

During tool length measurement process, distance from the tool cutting tip to spindle gauge line is accurately determined - *Figure 19-4*. Preset tools will reach the machine already mounted in a tool holder, identified by their tool numbers and with a list of measured (preset) tool lengths. All the CNC operator has to do, is to set the required tools into machine tool magazine and register each tool length in appropriate offset register, using provided offset number.

All preset dimensions have *positive* values, measured from the tool reference point to the gauge line. Gauge line of the machine is simulated in the presetting device and has to match. Each dimension will be entered as the H-offset amount in the tool length offset screen. For example, a tool length is preset to the amount of 8.5 inches, with the programmed offset number for this tool as H05. On the offset screen, under number 05, machine operator enters the measured length of 8.5000:

```
04 ...
05 8.5000
06 ...
```

◆ Tool Length by Touch Off

Tool length that uses touch-off method is very common, in spite of some time loss during setup. As the illustration in *Figure 19-5* shows, each tool is assigned a unique H-number (similar to the previous example), called the *tool length offset number*:

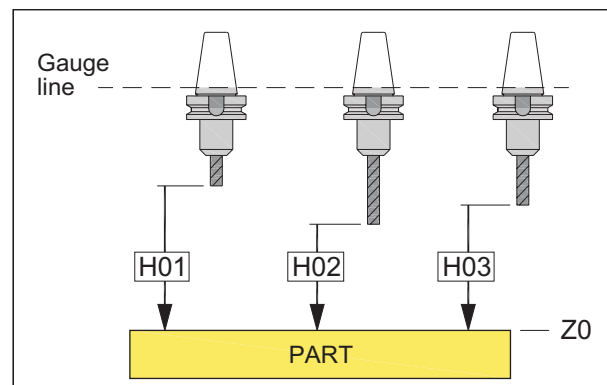


Figure 19-5

Touch-off method of the tool length offset setting

This number is programmed as the address H followed by the number itself. H-number usually corresponds to the tool number for convenience. Basic setup procedure is to measure the distance tool travels *from machine zero position (home) to program zero position (Z0)*. This distance is always negative and is entered into the corresponding H offset numbers under tool length offset menu of the control system. Most important notion here is that the Z-axis settings for any work offset G54-G59 and the *common* offset are normally set to Z0.0.

◆ Using a Master Tool Length

Using the touch-off method to measure tool length can be significantly speeded up by using a special method of a *master tool*, usually the longest tool. This tool can be a real tool or just a long bar with a rounded tip, permanently mounted in a spare tool holder. Within the Z-axis travel, this new 'tool' would usually extend out *more* than any anticipated tool that may be used.

Offsets G54 to G59 and the *external* work offset normally contain the Z-amount set to 0.0, when part touch-off method is used. This setting will change for the master tool length method. Master tool length measurement is very efficient and requires the following setup procedure. It provides suggested steps that may need some modification:

1. Take the master tool and place it in the spindle
2. Zero the Z-axis and make sure the read-out on the **relative** screen is Z0.000 or Z0.0000.
3. Measure the tool length for master tool, using the touch-off method described previously. After touching the measured face, **leave the tool in that position !**
4. Instead of registering the measured value to tool length offset number, register it into the common work offset or one of the G54-G59 work offsets under the Z-setting ! **It will be a negative value !**
5. While the master tool is touching measured face, set the relative Z-axis read-out to **zero !**
6. Measure every other tool, using the touch-off method. The reading will be from the master tool tip, **not** from machine zero
7. Enter the measured amounts under the H-offset number, in the tool length offset screen. It will always be a negative amount for any tool shorter than the master tool

➔ Note:

Master tool **does not** have to be the longest tool at all. Concept of the longest tool is strictly for safety - it means that every other tool will be shorter

Choosing any other tool as master tool, the procedure is logically same, except the H-offset entries will be *positive* for any tool that is *longer* than the master and they will be *negative* for any tool that is *shorter* than the master. In the rare case where the measured tool will have exactly the same length as master tool, the offset entry for that tool will be zero. Illustration in Figure 19-6 shows the concept of master tool settings.

After the master tool length is set and registered into the Z-axis of work offset, enter the distance from the tool tip of the new tool to the tool tip of master tool, and register it in the appropriate H-offset number. If the longest tool is an actual tool, rather than a plain bar used for setup, its H-offset amount must be always set to 0.0.

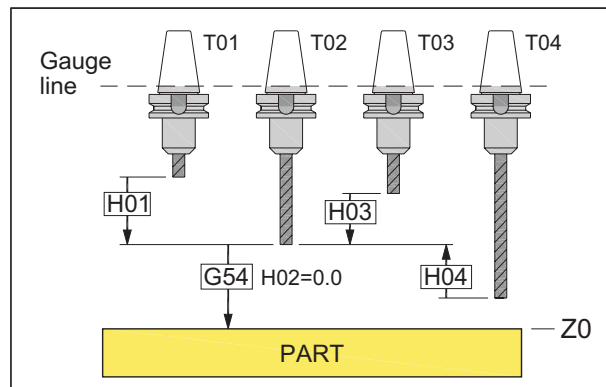


Figure 19-6

Tool length offset using the master tool length method.
T02 is the master tool, with setting of H02=0.0

The greatest benefit of this setting method is shortened setup time. If certain tools are used for many of jobs, only the master tool length needs to be redefined for any new part height while all other tools remain unchanged. They are all related to the master tool only.

◆ G43-G44 Difference

Initial description at the beginning of this chapter indicates that Fanuc and similar CNC systems offer *two* preparatory commands that activate the tool length offset. These two commands are G43 and G44. Most programmers use G43 command exclusively in the program and may have some difficulty to interpret the meaning of G44 command, because they have never used it. There is a good reason why G44 is a dormant command - not quite dead but barely breathing. Programmers would like to know *how* - and *when* - or even *if* - to use one over the other. Here is an attempt at explanation.

First, take a look at the definitions found in various CNC reference books and manufacturers' specifications sheets. In different versions of these publications, the following typical definitions are used - all are quoted literally and all are correct:

G43	Plus offset
G44	Minus offset
G43	Tool length offset positive
G44	Tool length offset negative
G43	Plus direction
G44	Minus direction

These definitions are correct only if taken within the context of their meaning into consideration. That context is not really clear from any of these definitions. *Plus to where? Positive of what?* To find the context, think about the use of tool length offset on a CNC machine. *What is the purpose of the tool length offset?*

The main and most important purpose of any tool length offset is to allow part program development *away* from the machine, *away* from tools and fixtures, and *without* knowing the actual cutting tool length during programming.

This process has two parts - one is in the program, the other at the machine. In the program, either G43 command or G44 command is required, together with proper H offset number - that is done by the programmer. At the machine, tool length offset can be set *on* or *off* the machine. Either way, the tool length is measured and the measured amount is entered into the control - that is the job of the operator. It is the *measuring at the machine* that has a number of variations - programmer has a choice of only two G-codes.

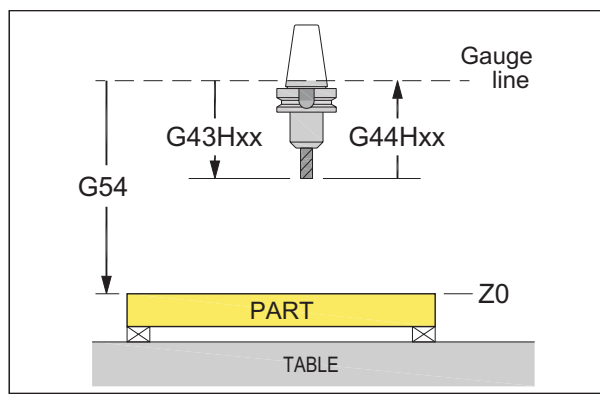


Figure 19-7

Less common method of using the tool length offset
Work offset (typically G54) must be set as well

Figure 19-7 illustrates one of two methods to set a tool length command - G54 or other work offset must be used.

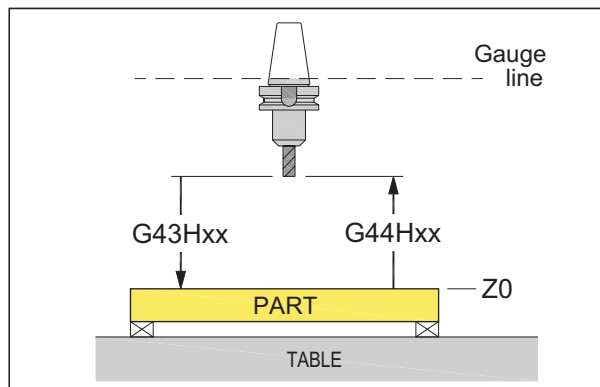


Figure 19-8

More common method of using the tool length offset
No work offset setting is required and G43 is the preferred choice

Figure 19-8 illustrates the other - and much more common - method. In this case, all work offset commands G54 to G59 will normally have the Z-axis setting of 0.0.

In either case, written program will be exactly the same (it is only the setting method that changes, *not* the programming method). Program will contain the tool length offset command (G43 or G44), followed by the target position along Z-axis and the H-offset number:

G43 Z1.0 H06 or G44 Z1.0 H06

Control system cannot offer any benefits, until the measured value for H06 is stored in the offset registers. For example, if the H06 has been measured as 7.6385, it will be entered as a *negative* value if G43 is used, and as a *positive* value if G44 is used (tool motions will be identical):

G43 Z1.0 H06 H06 = -7.6385

G44 Z1.0 H06 H06 = +7.6385

It is clear that the 'secret' of G43 versus G44 difference is nothing more than a sign reversal. Either command instructs the control as to *how* the actual Z-axis motion is calculated. Using G43, the H-offset amount will be *added* (+) in the calculation. Using G44, the H-offset amount will be *subtracted* (-). The actual Z-travel motion will be:

G43: Z + H06 = (1.0) + (-7.6385) = -6.6385

G44: Z - H06 = (1.0) - (+7.6385) = -6.6385

Tool length measuring method done *on* the machine (touch-off) will result in offsets with *negative* amounts. Setup process can automatically input all measured dimensions into the offset register, *as negative*. That is the reason why G43 is the standard command to program tool length offset. G44 is just not practical for everyday work.

PROGRAMMING FORMATS

Programming format for tool length offset is very simple and has been illustrated many times already. In the following examples are some general applications of various methods. The first one will show programming method if no tool length offset is available. Understanding the development of tool length offset over the years makes it easier to apply it in the program. Other example shows a comparison of programming methods for the older G92 programming style and the modern G54-G59 method. The last example shows the G54-G59 method applied to a simple program using three tools, as a typical way of programming today.

◆ Tool Length Offset not Available

In the early days of programming, tool length offset and work offsets were not available. G92 position register command was the only G-code used for setting the current tool position. Part programmer had to know *all and every* dimension specified by the machine manufacturer and *all and every* dimension of the job being setup, specifically the distance from Z0 to the tool tip.

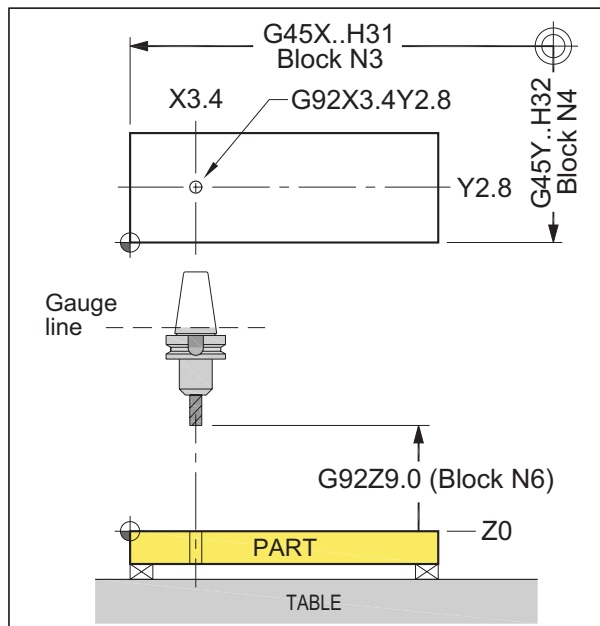


Figure 19-9

Setting tool length without tool length offset - program O1901

This early type NC program required the position compensation command G45 or G46 in XY axes and the position register command G92 in XYZ axes. Each part must start at machine zero - Figure 19-9:

```
O1901
N1 G20 (INCH MODE SELECTED)
N2 G92 X0 Y0 Z0 (MACHINE ZERO POSITION)
N3 G90 G00 G45 X3.4 H31 (X POSITION COMP)
N4 G45 Y2.8 H32 (Y POSITION COMP)
N5 G92 X3.4 Y2.8 (TOOL POS REGISTER XY)
N6 G92 Z9.0 (TOOL POS REGISTER Z)
N7 S850 M03 (SPINDLE COMMANDS)
N8 G01 Z0.1 F15.0 M08 (Z APPROACH MOTION)
N9 Z-0.89 F7.0 (Z CUTTING MOTION)
N10 G00 Z0.1 M09 (Z RAPID RETRACT)
N11 Z9.0 (MACHINE ZERO RETURN Z)
N12 X-2.0 Y10.0 (CLEAR POSITION XY)
N13 M30 (END OF PROGRAM)
%
```

◆ Tool Length Offset and G92

When the tool length offset became available, programming became much easier. Position compensation feature G45/G46 was still in use at the time and G92 had to be set for both X and Y axes. However, G92 setting for the Z-axis was replaced by G43 or G44 command, with an assigned H offset number - Figure 19-10.

Today, this method of combining the position compensation G45/G46 and tool length offset G43/G44 is considered obsolete, or at least quite old-fashioned. Only the G43H.. is used in modern programming, with the target position.

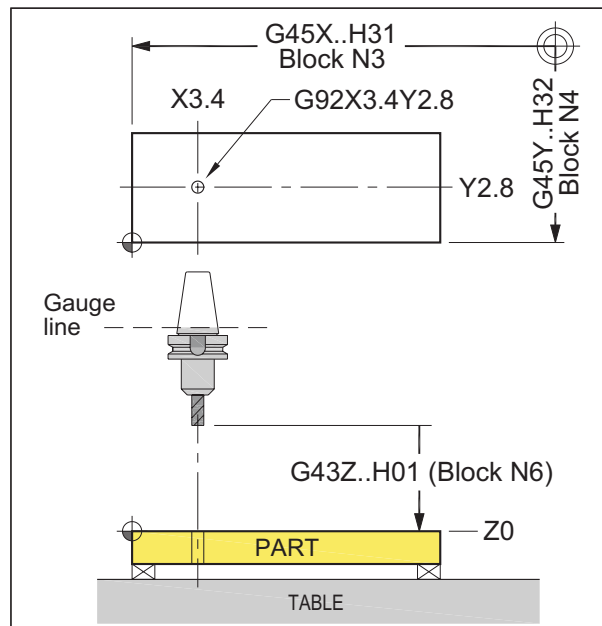


Figure 19-10

Setting tool length with G43 (Z) and G92 (XY) - program O1902

In an improved program, the tool length offset G43 is applied to the *first motion* command of the Z-axis (note that block N12 is not required - G28 cancels length offset):

```
O1902
N1 G20 (INCH MODE SELECTED)
N2 G92 X0 Y0 Z0 (MACHINE ZERO POSITION)
N3 G90 G00 G45 X3.4 H31 (X POSITION COMP)
N4 G45 Y2.8 H32 (Y POSITION COMP)
N5 G92 X3.4 Y2.8 (TOOL POSITION REGISTER)
N6 G43 Z1.0 H01 (TOOL LENGTH COMP Z)
N7 S850 M03 (SPINDLE COMMANDS)
N8 G01 Z0.1 F15.0 M08 (Z APPROACH MOTION)
N9 Z-0.89 F7.0 (Z CUTTING MOTION)
N10 G00 Z0.1 M09 (Z RAPID RETRACT)
N11 G28 X3.4 Y2.8 Z1.0 (MACHINE ZERO RETURN)
N12 G49 D00 H00 (OFFSETS CANCELLATION)
N13 M30 (END OF PROGRAM)
%
```

When a program is developed using G92, blocks N6 and N7 can be joined together for convenience, if preferred:

```
N6 G43 Z1.0 S850 M03 H01
N7 ...
```

This method has no effect on the tool length offset, only on the moment at which the spindle starts rotating. If used, position compensation and tool length offset *cannot* be programmed in the same block.

Note that the position compensation is still in effect in this example, due to the lack of work coordinate offset of the G54 to G59 series.

◆ Tool Length Offset and G54-G59

Modern programming has many commands and functions available and the G54-G59 series is one of them. G92 command has been replaced with work offset system in the range of G54 to G59 and, optionally, more. Normally, G92 is not used in the same program that contains any work offset selection G54 through G59 or the extended series.

Here is a program example of using tool length offset in a G54-G59 work offset environment:

```
O1903
N1 G20                                     (INCH MODE SELECTED)
N2 G90 G00 G54 X3.4 Y2.8                 (XY TARGET LOCATION)
N3 G43 Z1.0 H01                           (TOOL LENGTH COMP Z)
N4 S850 M03                               (SPINDLE COMMANDS)
N5 G01 Z0.1 F15.0 M08                    (Z APPROACH MOTION)
N6 Z-0.89 F7.0                           (Z CUTTING MOTION)
N7 G00 Z0.1 M09                           (Z RAPID RETRACT)
N8 G28 X3.4 Y2.8 Z1.0                     (MACHINE ZERO RETURN)
N9 G49 D00 H00                             (OFFSETS CANCELLATION)
N10 M30                                   (END OF PROGRAM)
%
```

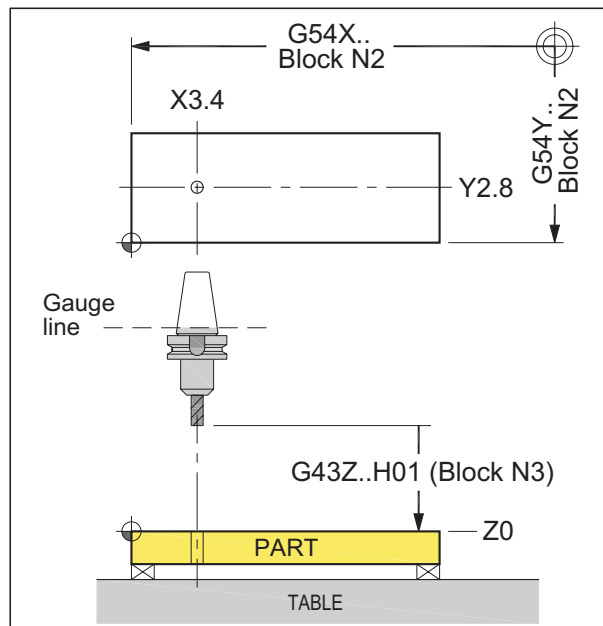


Figure 19-11

Setting tool length with G43 (Z) and G54-G59 (XY) - program O1903

In this example - Figure 19-11, using work offsets G54 through G59, the blocks N2, N3 and N4 can be joined together without a problem, perhaps to speed up processing:

```
N2 G90 G00 G54 G43 X3.4 Y2.8 Z1.0 S850 M3 H01
N3 ...
```

The G54 command will affect all axes, G43 with H01 will affect only the Z-axis. Tool must move in the clear.

◆ Tool Length Offset and Multiple Tools

The majority of CNC programs include more than one tool - in fact, most jobs will require several different tools. Our next example (independent of the previous drawings) illustrates a common method how the programmer enters tool length offset for *three* tools.

Three holes need to be spot-drilled, drilled and tapped. Drawing or explanation of the machining is not important at this time - just concentrate on the G43 tool length application. It is the program structure that is important now - note that there is *no change* in the program structure of any tool, only in the programmed values.

```
O1904
N1 G20
N2 G17 G40 G80 T01
N3 M06
N4 G90 G00 G54 X1.0 Y1.5 S1800 M03 T02
N5 G43 Z0.5 H01 M08 (TOOL LG OFFSET FOR T01)
N6 G99 G82 R0.1 Z-0.145 P200 F5.0
N7 X2.0 Y2.5
N8 X3.0 Y1.5
N9 G80 Z0.5 M09
N10 G28 Z0.5 M05
N11 M01

N12 T02
N13 M06
N14 G90 G00 G54 X3.0 Y1.5 S1600 M03 T03
N15 G43 Z0.5 H02 M08 (TOOL LG OFFSET FOR T02)
N16 G99 G81 R0.1 Z-0.89 F7.0
N17 X2.0 Y2.5
N18 X1.0 Y1.5
N19 G80 Z0.5 M09
N20 G28 Z0.5 M05
N21 M01

N22 T03
N23 M06
N24 G90 G00 G54 X1.0 Y1.5 S740 M03 T01
N25 G43 Z1.0 H03 M08 (TOOL LG OFFSET FOR T03)
N26 G99 G84 R0.5 Z-1.0 F37.0
N27 X2.0 Y2.5
N28 X3.0 Y1.5
N29 G80 Z1.0 M09
N30 G28 Z1.0 M05
N31 M30
%
```

This is a practical example of contemporary use of G43 tool length offset in a CNC program. In summary, G43 tool length offset requires target Z-position and the address H for each tool. The actual offset value is set at the control, during a given job setup. Two or more length offsets may be used for the same tool if necessary, but that is somewhat more advanced subject, described separately in the next section.

Also note that there is no tool length offset cancellation. Cancellation will also be explained later - see page 145.

CHANGING TOOL LENGTH OFFSET

Great majority of programming jobs requires only a single tool length offset command per cutting tool. Based on this principle, *Tool 1* (T01) has been associated with tool length offset H01, *Tool 2* (T02) with tool length offset H02, etc. However, in some very special circumstances, the tool length offset may have to be changed for the *same tool*. In those applications, there will be *two or more* tool length offsets for a single tool.

An example of a single tool length offset change would be any part that uses two or more drawing references along the Z-axis. *Figure 19-12* illustrates this concept with a groove dimensioned by its depth location for the top and bottom (groove width of 0.220 is the difference).

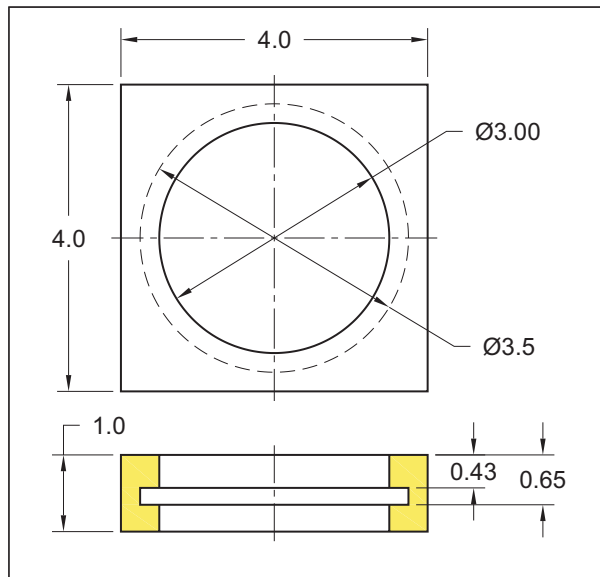


Figure 19-12

Example of programming more than one tool length offset for a single tool - program O1905

Based on the illustration, suitable cutting method must be determined first (pre-machining of the Ø3.00 hole is assumed). A 0.125 wide slot mill will be a good choice to profile the circle, using typical milling method for a full circle (see page 248). Program can be shortened by using a subprogram method (subject starts on page 383). Because the 0.220 groove width is larger than the cutter, more than one cut is needed - two in this case. For the first cut, tool is positioned at Z-0.65 depth (as per drawing) and makes the first cut at the groove bottom. Tool *bottom edge* will reach the Z-0.65 depth.

For the second cut, *top edge* of slotting mill is used and the tool makes a profile for the second groove (in fact, it will *widen* the first groove) at the depth of Z-0.43 (again, as shown in the drawing).

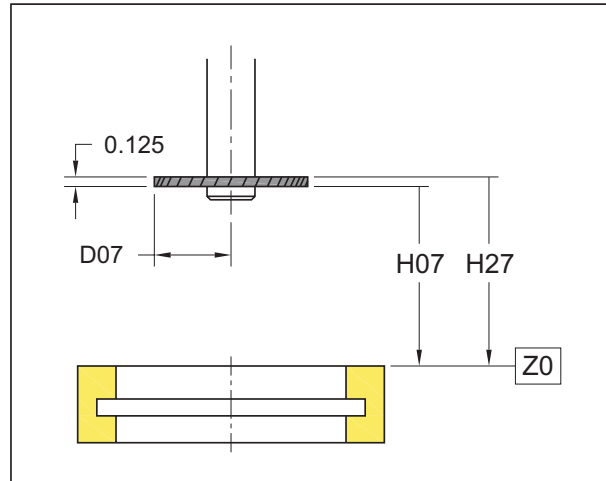


Figure 19-13

Setting of two length offsets for a single tool. The difference between H07 and H27 offsets is the width of slot (0.125 shown)

Note the words - *bottom edge* versus *top edge* of the slot mill. Which edge is programmed as a reference for the tool length? Bottom edge or top edge?

Figure 19-13 shows that *two* reference positions are used for the same tool. For this reason, the program requires *two* tool length offsets, H07 and H27 in the illustration. D07 is the cutter radius offset, and 0.125 is the slotting mill width.

Other methods of programming can be used, for example, calculating the difference manually, but the method using multiple tool length offsets is very useful during machining to allow fine groove width adjustments. It is shown in the following example - program O1905:

```
O1905
(TWO TOOL LENGTH OFFSETS FOR ONE TOOL)
N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X0 Y0 S600 M03
N4 G43 Z1.0 H07 M08 (ABOVE JOB CLEARANCE)
N5 G01 Z-0.65 F20.0 (CUTTER EDGE - BOTTOM)
N6 M98 P7000 (CUTTING GROOVE AT Z-0.65)
N7 G43 Z-0.43 H27 (CUTTER EDGE - TOP)
N8 M98 P7000 (CUTTING GROOVE AT Z-0.43)
N9 G00 Z1.0 M09
N10 G28 Z1.0 M05
N11 M30
%

O7000
(SUBPROGRAM FOR GROOVE IN O1905)
N1 G01 G41 X0.875 Y-0.875 D07 F15.0
N2 G03 X1.75 Y0 R0.875 F10.0
N3 I-1.75
N4 X0.875 Y0.875 R0.875 F15.0
N5 G01 G40 X0 Y0
N6 M99
%
```

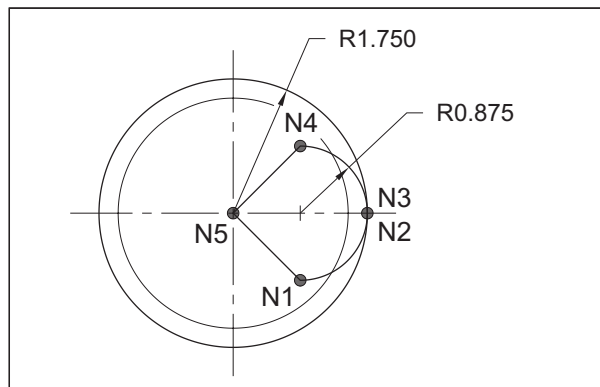


Figure 19-14

Full circle milling - subprogram O7000
Start and finish of cutting is at the center of the groove

In the example, tool length offset H07 is used for the bottom reference edge of the slotting mill and H27 is used for the top reference edge of the slotting mill. D07 is used for the cutter radius only. Figure 19-14 shows all XY tool motions used in subprogram O7000.

HORIZONTAL MACHINE APPLICATION

So far, all presented examples were aimed towards a CNC vertical machining center. Although the logic of tool length offset applies equally to *any* machining center, regardless of the Z-axis orientation, there are some noticeable differences in practical applications on horizontal machining centers (*subject starts on page 445*).

Horizontal machining center allows programming of a tool path on several faces of the part. Since each face has a different distance from the tool tip (along Z-axis), the tool length offset for each face will vary. It is common to program different work offsets and different tool length offsets for each face.

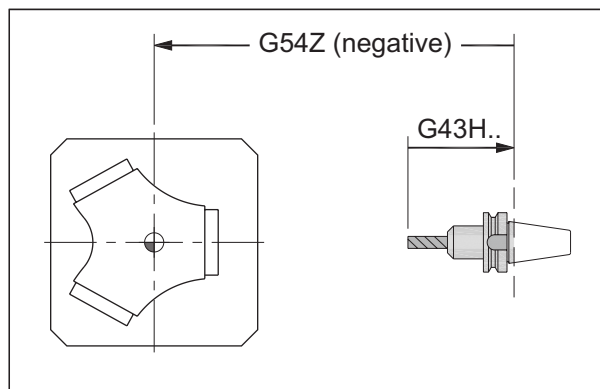


Figure 19-15

Typical tool length offset setting for a preset tool
Program zero is at the center of the table

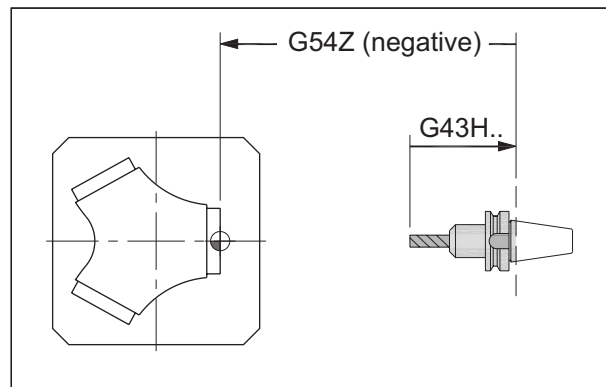


Figure 19-16

Typical tool length offset setting for a preset tool
Program zero is at the face of the part

The two related illustrations show typical setup of tool length offset for preset tools on horizontal machining centers. Figure 19-15 shows program zero at the center of machine table, Figure 19-16 shows the program zero at the face of part.

TOOL LENGTH OFFSET CANCEL

In programming, a well organized approach is always important. That means, a program command that is turned on when needed should also be turned off, when not needed anymore. Tool length offset commands are no exception.

Tool length offset cancellation may be included in the program. There is a special preparatory command available that cancels any selected method of the tool length offset, either G43 or G44. The command to cancel any active tool length offset in the program (or via MDI) is G49:

G49	Tool length offset cancel
-----	---------------------------

One method of using the G49 command is on its own - in a single block - just before returning to the machine zero in the Z-axis, for example:

```
N176 G49
N177 G91 G28 Z0
...
```

A similar method also cancels the offset numbers:

```
N53 G91 G28 Z0 H00
```

In this case, the G28 command is coupled with an H offset number zero - H00. Note, there is no G49 in the block and H00 does the job of cancellation. There is no setting for H00 on the control. It just means cancellation of the tool length offset.

A program may also be *started* with the tool length offset command canceled (under program control), usually in the safety line (safety block or initial block):

N1 G20 G17 G40 G80 G49

... or a variation of the same block:

N1 G20

N2 G17 G40 G80 G49

There is one more way to cancel the tool length offset - *do not program it at all*.

A strange suggestion, perhaps, but well founded. Most examples in this handbook *do not* use G49 command at all. *Why not? What happens at the end of each tool?*

Fanuc rule is quite explicit - any G28 or G30 commands (both execute tool return to the machine zero) will cancel the tool length *automatically*.

The meaning is simple - programmer may take advantage of this rule and does not need to specifically cancel the tool length offset, if machine returns to the tool change position. This is normal for all machines with an automatic tool changer. This approach is illustrated in many examples included in this handbook.

Any one of the methods will guarantee that the active tool length offset will be canceled. There may be some differences between machine manufacturers and consulting the machine manual will always be the responsible approach.

There are some machines that require using G49 for every tool.
Always check manufacturer's instructions !

20

RAPID POSITIONING

A CNC machine tool does not always cut material and 'make' chips. From the moment the cutting tool becomes active in a part program, it goes through a number of motions - some are productive (cutting), others are nonproductive (positioning).

Positioning motions are necessary but nonproductive. Unfortunately, these motions cannot be totally eliminated and have to be managed as efficiently as possible. For this purpose, the CNC system provides a feature called the *rapid traverse* motion. Its main objective is to shorten the positioning time between non-cutting operations, where the cutting tool is not in contact with the part. Rapid motion operations usually involve four types of motion:

- From the tool change position towards the part
- From the part towards the tool change position
- Motions to bypass obstacles
- Motions between different positions on the part

RAPID TRAVERSE MOTION

Rapid traverse motion, sometimes called a *positioning motion*, is a method of moving the cutting tool from one position to another position at a *rapid rate of the machine*. The maximum rapid rate is determined by CNC machine manufacturer and takes place within the machine travel limits.

The common rapid rate for many larger CNC machines is about 25000 mm/min (985 in/min). Medium and smaller machines can go much higher, offering rapid motion up to 75000 mm/min (2950 in/min) or even more, particularly for smaller machines. Machine manufacturer determines the rate of rapid motion for each of the machine axes. The motion rate can be the same for each axis or it can be different. A different rapid rate is usually assigned to the Z-axis, while X and Y axes have the same rapid motion rate.

Rapid motion can be executed as a single axis motion, or as a compound motion of two or more axes simultaneously. It can be programmed in absolute or incremental mode of dimensioning and it can be used whether the spindle is rotating or stationary. During program execution, CNC operator may temporarily interrupt the rapid motion by pressing feedhold key on the control panel, or even setting the feedrate override switch to zero or a decreased rate. Another kind of rapid rate control can be achieved by the *dry run* function, usually during setup.

◆ G00 Command

Preparatory command G00 is required in CNC program to initiate rapid motion mode. Feedrate function F is *not required* with G00 and, if programmed, will be ignored during rapid motion (in G00 mode). Such a feedrate will be stored in memory and becomes effective beginning with the first occurrence of any cutting motion (G01, G02, G03, etc.), unless a new F-function is programmed together with the cutting motion:

➡ Example A (feedrate in in/min) :

```
N21 G00 X24.5 F30.0
N22 Y12.0
N23 G01 X30.0
```

In block N21, only the rapid motion will be executed. Feedrate of 30.0 in/min will be ignored in this block, but stored for later use. Motion in the block N22 will also be in rapid positioning mode, since G00 is a modal command. The last block, N23, is a linear motion (cutting motion), that requires a feedrate. As there is no feedrate assigned to the motion in this block, the *last programmed* feedrate will be used. That was specified in block N21 and it will become the *current* feedrate in block N23, as F30.0.

➡ Example B (feedrate in in/min) :

```
N21 G00 X24.5 F30.0
N22 Y12.0
N23 G01 X30.0 F20.0
```

In block N21, command G00 becomes modal, which means that it remains in effect until it is canceled by another command of the same group. In the example (B), command G01 in block N23 cancels rapid motion and changes rapid mode to a linear mode. Also, the feedrate is reprogrammed and will be at 20.0 in/min starting at block N23. In fact, the feedrate F30.0 in block N21 has never been used in the program - it is harmless but redundant and should be removed.

Rapid traverse motion is measured as the *distance in current units traveled in one minute* (measured in mm/min or in/min). The maximum rate is always set by the machine manufacturer, *never* by the control system or the program. A typical limit set by the machine builder is a rate between 7500 and 75000 mm/min (~300-3000 in/min), and even higher. Since motion per time is independent of the spindle rotation, it can be applied at any time, regardless of the last spindle rotation function mode (M03, M04, M05).

Depending on the CNC machine design, rapid motion rate can be the same for all axes, or each axis can have its own maximum rate. Maximum rapid rates for a typical machining center may be 30000 mm/min (1180 in/min) for the X and Y axes and about 24000 mm/min (945 in/min) for the Z-axis. For a CNC lathe, the rates are very similar, for example, 25000 mm/min (985 in/min) for the X-axis, and 30000 mm/min (1180 in/min) for the Z-axis. Emerging technologies allow for much higher rapid rates.

RAPID MOTION TOOLPATH

Every motion in G00 mode is a rapid, non-circular motion - circular and helical motions cannot normally be made at a rapid rate. The actual linear motion of a tool between two points is not necessarily the shortest toolpath in the form of a straight line, although many latest machines do provide such feature. On many CNC machines, a programmed toolpath and the resulting actual toolpath will be different, depending on several factors:

- Number of axes programmed simultaneously
- Actual length of motion for each axis
- Rapid traverse rate of each axis

Since the only purpose of rapid motion is saving unproductive time (motion from the current tool position to the target tool position), toolpath itself is irrelevant to the shape of the machined part. Always be aware of the actual rapid toolpath motion for reasons of safety, particularly when two or more axes are programmed at the same time. No physical obstacles must be in the way of a toolpath motion.

If there is an obstacle between any two points of the toolpath, such obstacle *will not be* automatically bypassed by the control for one very simple reason - the control has no way of detecting any obstacle. It is the programmer's responsibility to assure that any tool motion (rapid motion included) occurs without any obstacles in its way.

Some typical examples of physical obstacles that can interfere with a tool motion are:

- **FOR MACHINING CENTERS :**
Clamps, vises, fixtures, rotary or indexing table, machine table, part itself, etc.
- **FOR LATHES :**
Tailstock quill and body, chuck, steady rest, live center, face plate, fixture, other tool, part itself, etc.

Additional obstacles during a tool motion may be caused by special types of setup, machine design, tool mounting method, etc.

Always watch for obstacles during rapid motion

Although an obstacle may be in the way of cutting motions in G01, G02 or G03 mode (for example a face turning towards a tailstock on a lathe), the most problems occur during rapid motions G00, G28, G29, G30 and with fixed cycles G81 to G89, G73, G74 and G76. During rapid motion, the actual toolpath on many machines is much less predictable than during cutting motions. Keep in mind that the only purpose of rapid motion is to get from one part location to another location *fast* - but not necessarily straight.

In order to bypass obstacles and still assure a safe rapid motion in the program at all times, let's take a closer look at the available options while programming a rapid motion.

◆ Single Axis Motion

Any tool motion programmed specifically for only one axis at a time is always a straight line along the selected axis. In other words, each rapid motion that is parallel to one of the available axes, must be programmed in a separate block. The resulting motion is always equivalent to the shortest distance between the start and end points of this motion - *Figure 20-1*.

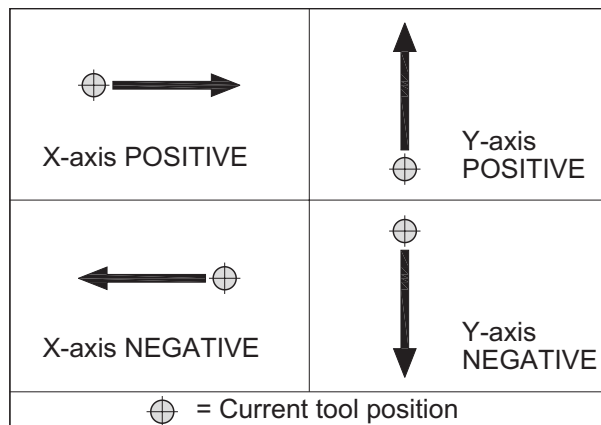


Figure 20-1

Single axis motion for a machining center application (XY shown)

Several consecutive program blocks, each containing only a single axis motion, can be included in the program to bypass obstacles to machining. This method of programming is preferable in cases where only the *exact* or *approximate* position of certain obstacles (such as clamps or fixtures) is known during part program preparation.

◆ Multiaxis Motion

Cutting tool moved at rapid rate always uses the G00 command. If this motion is a motion of two or more axes simultaneously, the programmed rapid path between two points and the actual rapid path may not always be identical. The result is a compound motion that can be - and often is - much different from the theoretical programmed motion - the motion originally intended.

In theory, the motion along any two axes is equivalent to a *straight diagonal motion*. The real motion, however, may or may not be a straight diagonal tool path at all. Consider the following example in *Figure 20-2*.

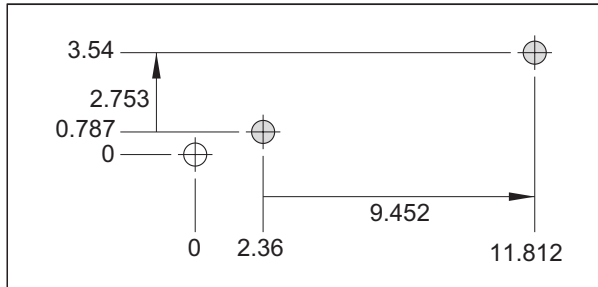


Figure 20-2

Drawing sketch for rapid motion examples

Current tool position (start point) is at X2.36 Y0.787 coordinate location. The tool motion terminates at X11.812 Y3.54 location. In terms of incremental motion, the cutting tool has to travel 9.452 inches along the X-axis and 2.753 inches along the Y-axis.

If rapid rate for both axes is the same (XY rapid motion rates usually are), such as 985 in/min, it will take

$$(9.452 \times 60) / 985 = 0.576 \text{ seconds}$$

to complete the X-axis motion - but only

$$(2.753 \times 60) / 985 = 0.168 \text{ seconds}$$

is required to complete the Y-axis motion. Since this motion is not completed until *both* axes reach the end point, it is logical that the actual tool path will be *different* from the programmed toolpath.

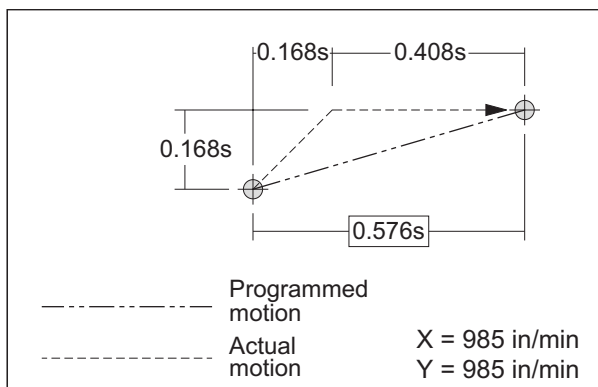


Figure 20-3

Rapid motion deviation - same rapid rate for each axis

Figure 20-3 shows a combination of an angular and a straight motion as the actual toolpath. Tool departs at the rate of 985 in/min (25000 mm/min) simultaneously in *both*

axes, with a resulting 45° motion. The total time required to reach the end position is 0.576 seconds, which is the longest time required for *either* axis to reach its target position. After the elapsed time of 0.168 seconds, the Y-axis target position has been reached, but there is still another 0.408 seconds left to complete the X-axis motion. The target position must be reached in *both* axes, so the tool then continues along X-axis only (for the remaining 0.408 seconds), in order to reach the final target position.

Another example, using the same location coordinates as in *Figure 20-2*, illustrates a less common situation, when the rapid rate is *different* for each axis - *Figure 20-4*.

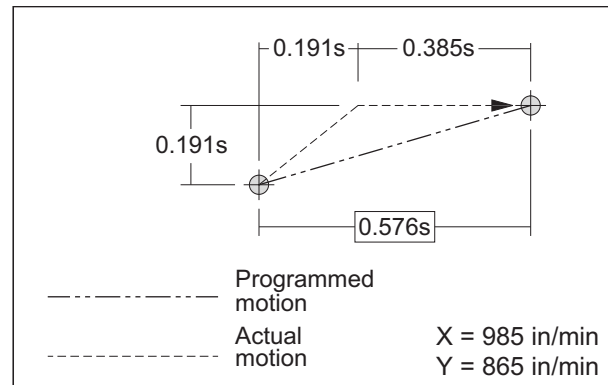


Figure 20-4

Rapid motion deviation - different rapid rate for each axis

In this not so common example, the X-axis rate is set to 985 in/min (25000 mm/min) but the Y-axis rate is set to 865 in/min (22000 mm/min). For the X-axis, it will then take

$$(9.452 \times 60) / 985 = 0.576 \text{ seconds}$$

to complete the X-axis motion (no change) - but only

$$(2.753 \times 60) / 865 = 0.191 \text{ seconds}$$

to complete the Y-axis motion. In this case, the resulting motion will also include an angular departure, but *not* at 45°, because of the different rating of rapid traverse rate for each axis. During the 0.191 seconds (which is the common time to both axes), the X-axis motion will travel

$$0.191 / 60 \times 985 = 3.136 \text{ inches}$$

but the Y-axis motion will be only

$$0.191 / 60 \times 865 = 2.753 \text{ inches}$$

The resulting motion is at 41.279° with a slight rounding applied. Actual departure angle is not always necessary to be known, but it helps to calculate it for rapid motions in some very tight areas of a part. It only takes a few simple trigonometric functions to make sure of the true rapid toolpath, provided the rapid rate per axis is known.

Both of the above examples illustrate an angular motion along two axes, followed by a straight single axis motion in the remaining axis. The graphical expression of these motions is a bent line, resembling a *hockey stick* or a *dog leg* shape, which are also very common terms applied to such a rapid motion.

Calculation of the actual motion shape, as was done earlier, is only seldom necessary. Taking some basic precautions, rapid motion can be programmed safely *without* any calculations. If no obstacle is within the work area (an imaginary rectangle created by the diagonally positioned start and end point), there is no danger of collision due to the diverted rapid tool path. On CNC milling systems, the third axis can also be used. The rectangle of the above example will be enhanced by the third dimension and a three dimensional space must be considered. In this case, no obstacle should be *within this space*, otherwise the same rules apply for a rapid motion along three axes as for a two-axis simultaneous rapid motion. Note that the rapid rate for the Z-axis on CNC machining centers is usually *lower* than the rapid rate for X and Y axes.

◆ Straight Angular Motion in Rapid Mode

In some uncommon circumstances, the theoretical rapid toolpath will correspond to the actual toolpath (with no bent line as a result). This will happen if the simultaneous tool motion has the same length in each axis and the rapid rates of all axes are identical. Such an occurrence is rather rare, although not impossible. Some machine manufacturers provide this feature as a standard and the programmer should know whether the machining center does have that feature or not. Another situation where the resulting motion is a straight angle, is when the rapid rating varies for each axis, but the required length of motion just 'falls' into the range that results in a straight angular motion.

Both of these occurrences are rare (more or less a case of good luck) and in actual programming will seldom happen. To be on the safe side, never take any chances - it is always more practical to program the rapid motion without the actual calculation of the toolpath but with safety as a primary consideration.

◆ Reverse Rapid Motion

Any rapid motion must be considered in terms of *approach* towards a part and the *return* back to the tool changing position. This is the way a cutting tool is normally programmed - it starts at a certain position and then returns there, when all cutting activity for the tool is completed. It is not a mandatory method, but it is an organized method, it is consistent, and it makes programming much easier.

So far, the subject was a rapid motion *before* an actual cut, starting from the tool change position. When the tool cutting function is completed, a rapid motion is required to return back to the tool change position.

This consideration is more important in turning applications than in milling, due to the nature of programming for the two types of machines. In turning, the approach motion may be along the Z-axis first, to avoid a collision with the tailstock, and *then* along the X-axis. The reverse motion should be along the X-axis first, *then* along the Z-axis motion, in order to achieve the same safety goal when returning to the tool change position.

A typical application of this programming technique may be useful after using a machining cycle (such as turning, boring, facing, threading, etc.), where the cycle starting point is also its end point.

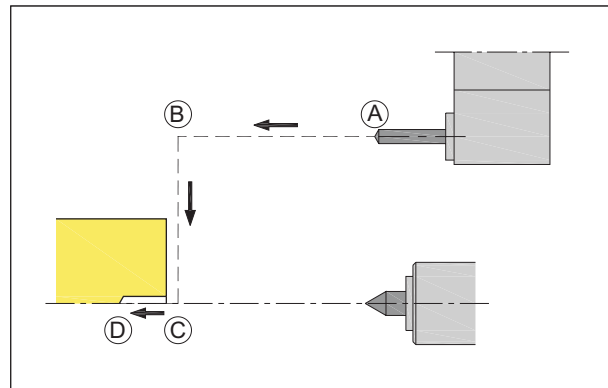


Figure 20-5

Typical example of a reversed rapid motion on a CNC lathe, used to bypass obstacles, for example, a tailstock

As Figure 20-5 shows, rather than programming a direct motion from the turret position to the cutting position (which would be directly from point A to point C), the tool motion was split. The approach towards the part will be in the order of A to B to C, at a rapid rate. From point C to point D, the actual cutting takes place. When cutting is done, tool will rapid in reverse order, back to the starting position. Rapid motion will be from D to C to B to A. This is a necessary precaution to bypass a potential obstacle, for example, the tailstock.

TYPE OF MOTION & TIME COMPARISON

Technique of programming each axis separately in individual blocks of the program, is recommended only for the purpose of bypassing possible obstacles during the tool path - and strictly for safety. This method of programming requires a slightly longer cycle time than the simultaneous multiaxis rapid motion. To compare the difference, consider a three axis rapid motion, such as a typical tool approach in milling.

As an example, an old machine has rapid rate is of only 394 in/min (10000 mm/min) for each axis. Motion takes place between coordinate location of X2.36 Y0.787 Z0.2 (start point) and X11.812 Y3.54 Z1.0 (end point).

The required time for tool travel along each axis can be easily calculated (*based on the initial Figure 20-2*):

■ X-axis time:

$$((11.812 - 2.36) \times 60) / 394 = 1.439 \text{ sec.}$$

■ Y-axis time:

$$((3.54 - 0.787) \times 60) / 394 = 0.419 \text{ sec.}$$

■ Z-axis time:

$$((1.0 - 0.2) \times 60) / 394 = 0.122 \text{ sec.}$$

If all three axes are moved simultaneously, the total time for positioning is 1.439 seconds, which is the longest time required for any axis to reach the end point. The program block will be:

G00 X11.812 Y3.54 Z1.0

If this motion were to be separated into three individual program blocks, the total time would be the result of individual times added together:

$$1.439 + 0.419 + 0.122 = 1.980 \text{ seconds}$$

which is about 37.5% longer. The percentage will vary, depending on the rapid motion rate and the rapid travel length, measured along each machine axis. Program blocks will be written separately:

**G00 X11.812
Y3.54
Z1.0**

Note that the modality of G00 rapid motion command does not require its repetition in the subsequent blocks.

REDUCTION OF RAPID MOTION RATE

During part setup or while proving a new program on the machine, CNC operator has an option to select a *slower rapid traverse rate* than the maximum established by machine manufacturer. This adjustment is done by the means of a special *rapid override switch*, located on the control system panel. This switch has typically four selectable positions, depending on the machine brand and the type of control system - *Figure 20-6*.

The second, third and fourth positions on the rapid motion override switch are rated as the *percentage of the actual rapid rate* - 25%, 50%, 100% respectively. They are set by the machine manufacturer. The first setting, sometimes identified by F0 (or F1) is a rapid motion rate set through a *control system parameter*. The F0 (F1) setting should always be slower than any other setting, typically less than the lowest setting of 25%, such as 10%, for example.

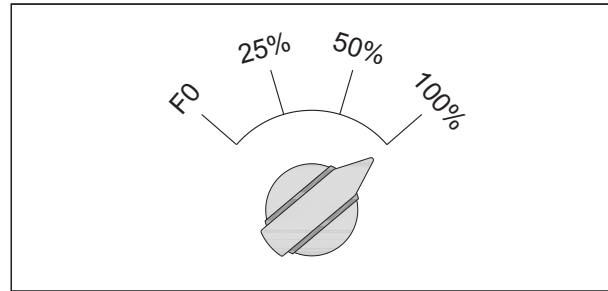


Figure 20-6

Rapid motion override switch set to 100% of rapid rate

Configuration of the rapid override switch varies between machines from different manufacturers. On some machines, the rapid motion may be stopped altogether, on others, tool will move at the slowest percentage and cannot be stopped with the override switch alone.

During actual production, after the program has been verified and optimized for best tool performance and productivity, the override switch should be set to the 100% pointer, to shorten overall cycle time.

RAPID MOTION FORMULAS

Calculations relating to the rapid tool motion can be expressed as formulas and used quickly at any time by substituting all known parameters. Relationships between rapid traverse rate, length of motion and the elapsed time can be expressed in the following three formulas:

$$T = \frac{L \times 60}{R}$$

$$R = \frac{L \times 60}{T}$$

$$L = \frac{T \times R}{60}$$

where ...

- T = Required time in seconds
- R = Rapid traverse rate per minute
for the selected axis - **mm/min** or **in/min**
- L = Length of motion - **mm** or **inches**

Units applied to the formulas must always be consistent within the selected system of measurement in the program. *Millimeters* and *millimeters per minute (mm/min)* must be used in the metric system. *Inches* and *inches per minute (in/min)* must be used with the imperial system. For any calculation relating to the rapid traverse time, the measuring units cannot be mixed.

APPROACH TO THE PART

Previous *Figure 20-5* had an illustration of a safe tool approach as applied to a CNC lathe. For CNC machining centers, the safety of part approach should be considered with equal care. Keep in mind that the general principles of rapid tool motion have to be considered for any machine. When approaching a part at a rapid rate, the cycle time can be somewhat shortened by keeping part clearances to the smallest safe minimum. Let's have a look at some potential problems.

In the following milling example (*Figure 20-7*), an approach to the part is made along the Z-axis, with a clearance of 0.05 inches (1.27 mm) in block N315:

```
N314 G90 G54 G00 X10.0 Y8.0 S1200 M03
N315 G43 Z0.05 H01
N316 G01 Z-1.5 F12.0
```

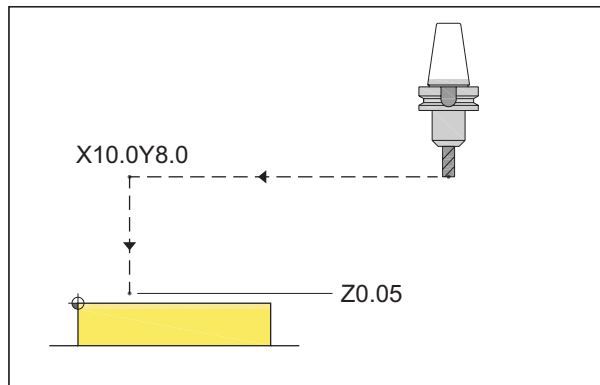


Figure 20-7

XY and Z axes approach to minimum clearance directly

There is nothing wrong with such a method of programming, providing the cutting tool is properly set and the part height is consistent from one part to another, as it should be. The clearance of 0.05 inches (1.27 mm) allows very little amount of unproductive cutting. On the other hand, an inexperienced CNC operator may not feel quite comfortable with such a small clearance, particularly during the early training stages. If the operator's convenience is considered as a significant factor contributing to the overall

productivity, it might be a reasonable compromise to split the Z-axis motion into two separate motions (*Figure 20-8*):

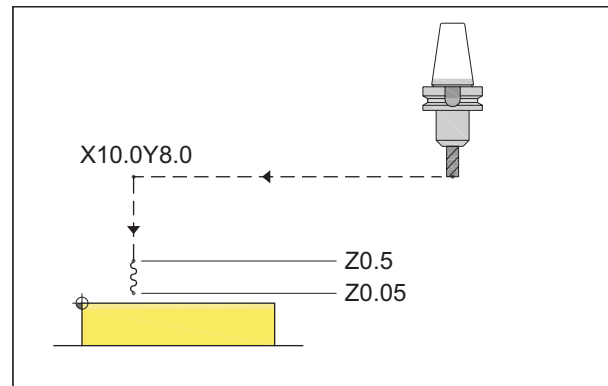


Figure 20-8

XY and Z axes approach to minimum clearance in a split motion

```
N314 G90 G54 G00 X10.0 Y8.0 S1200 M03
N315 G43 Z0.5 H01
N316 G01 Z0.05 F100.0
N317 Z-1.5 F12.0
```

In this method, the rapid motion has been first programmed to a much more comfortable position of 0.500 inches above the part (N315). Then, the motion continued to the cutting start point, using *linear* interpolation G01 in block N316. Since this is still a motion in the air, therefore not productive, a relatively heavy feedrate was used at the same time. As may be expected in such situations, there is a trade off.

Although the cutting time was slightly increased, at the same time, the CNC operator has been given an opportunity to use the feedrate override switch for testing the first part (used perhaps in a single block mode). Once the program is verified and debugged, the heavy feedrate in the non-cutting motion will speed up the operation and at the same time provide an extra safety clearance. The program with the split Z-axis motion can always be optimized later, although this may not be the best approach for repetitive jobs, since the setup is always 'new' for any repetition at a later date. However, it may be very useful when running lots of large numbers (several thousands, for example).

21

MACHINE ZERO RETURN

The ability of a control system to return a cutting tool from any position to the machine reference position is a critical feature of all modern CNC systems. Programmers and operators understand the term *machine reference position* as synonymous with the *home position* or *machine zero position*. This is *the position of all machine slides at one of the extreme travel limits of each axis*. Its exact position is determined by the machine manufacturer and is *not* normally changed during the machine working life. Return to that position is automatic, on request from the control panel, in MDI operation, or via a program.

MACHINE REFERENCE POSITION

The existence of machine reference position is for referencing purposes. In order that a CNC machine is a true precision machine tool, it requires more than just high quality components, it requires some unique location that can be considered the machine origin point - a zero position - a home position. Machine reference position is exactly such a point.

Machine zero is a fixed position on a CNC machine that can be reached repeatedly, on request, through the control panel, MDI, or program code execution

◆ Machining Centers

Although design of CNC machining centers varies for different models, there are only four possible locations for machine zero, within the XY view:

- Lower left corner of the machine
- Upper left corner of the machine
- Lower right corner of the machine
- Upper right corner of the machine

It is quite common, in fact normal, to start the first part of a new program from machine zero position. Often, it is also necessary to make a tool change at machine zero position and return there when the program execution is completed. So, several of the four alternatives are not very convenient for a part setup on the machine table and its removal when the machining is done.

The most common and standard machine reference position for vertical machining centers is at the *upper right corner* of the machine, looking perpendicularly towards the XY plane - *Figure 21-1*.

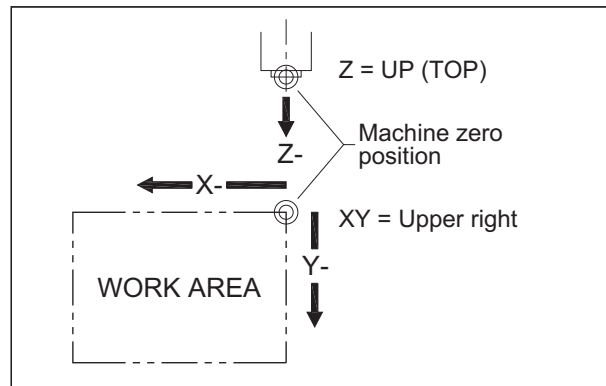


Figure 21-1

Machine zero position located at the upper right XY corner of a CNC vertical machining center

So far, any reference to the Z-axis in the description was quite intentional. Z-axis machine zero position for a vertical machining center is always where the *Automatic Tool Change (ATC)* takes place. This is a built-in location, normally placed at a safe distance from the machine table and the work area. For most machines, standard machine zero of CNC machining centers is at the extreme travel limit of each axis in the *positive* direction. There are exceptions, as may be expected.

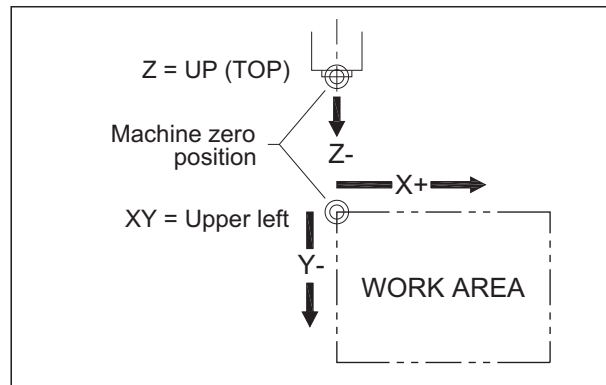


Figure 21-2

Machine zero position located at the upper left XY corner of a CNC vertical machining center

As *Figure 21-2* illustrates, some CNC vertical machining centers have the machine zero position at the *upper left* corner of the XY plane.

In both illustrations, the arrows indicate tool motion direction *towards the work area*. Moving the tool from machine zero into the opposite direction will result in a condition known as *overtravel* - compare the two possibilities:

- Tool motion from machine zero, if machine zero is located at the upper *right* corner:

X+ Y+ Z+ ... tool motion will overtravel

- Tool motion from machine zero, if machine zero is located at the upper *left* corner:

X- Y+ Z+ ... tool motion will overtravel

The other two corners (lower left and lower right of the XY view) are not used as machine zero.

◆ Lathes

The machine reference position for two axis CNC lathes is logically no different from the reference position of machining centers. An easy access by CNC operator to the mounted part is the main determining factor. Both, the X and Z axes have their machine reference position at the furthest distance from the rotating part, which means away from the headstock area, consisting of the chuck, collet, face plate, etc.

For the X-axis, machine zero reference position is always at the extreme limit of travel *away from the spindle center line*. For the Z-axis, machine reference position is always at the extreme travel *away from the machine headstock*. In both cases, it normally means a positive direction towards the machine zero - the same as for machining centers. The illustration in Figure 21-3 shows machine zero for a typical CNC lathe (rear type).

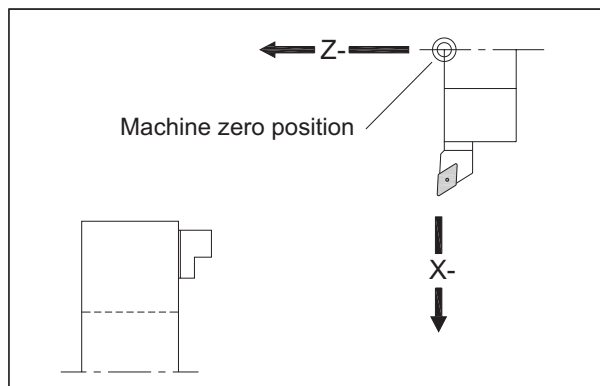


Figure 21-3

Machine zero position for a typical CNC lathe (rear type)

In the illustration, two arrows indicate tool motion direction towards the work area. Moving the tool from machine zero into the opposite direction will result in *overtravel* in the particular axis:

- Tool motion from machine zero of a typical rear lathe:

X+ Z+ ... tool motion will overtravel

◆ Setting the Machine Axes

From the earlier sections, remember that there is a direct relationship between the CNC machine, the cutting tool and the part itself. Work reference point (program zero or part zero) is always determined by the CNC programmer, the tool reference point is determined by the tool length at its cutting edge, also by the programmer.

Only the machine reference point (home position) is determined by the machine manufacturer and is located at a *fixed* position. This is a very important consideration.

Fixed machine zero means that all other references are dependent on this location

In order to physically reach machine reference position (home) and set the machine axes, for example, during part or fixture setup, there are three methods available to the CNC operator:

- *Manually* - using the control panel of the system

Machine operator will use the XYZ (machining centers) or the XZ (lathes) switches or buttons available for that purpose. One or more machine axes can be activated simultaneously, depending on the control unit

- *Using the MDI* - Manual Data Input mode

This method also uses the control panel. In this case, machine operator sets the MDI mode and actually programs the tool motion, using suitable program commands (G28, G30)

- *In the CNC program* - during a cycle operation

Using the same program commands as for MDI operation, CNC programmer, not the machine operator, includes machine zero return command (or commands) in the program, at desired places

When the CNC operator has performed the actual machine zero return, it is always a good idea to set the relative and absolute positions to zero on the display screen. Keep in mind that the relative display can only be set to zero from the control panel and the absolute display can only be changed through a work offset, MDI mode, or the part program. This topic is normally a part of CNC machine operation training, directly at the machine.

For the last two methods of a machine zero return, CNC system offers four specific preparatory commands.

◆ Program Commands

There are four preparatory commands relating to the machine zero reference position:

G27	Machine zero reference position return <i>check</i>
G28	Return to the <i>primary</i> machine zero reference position
G29	Return <i>from</i> the machine zero reference position
G30	Return to the <i>secondary</i> machine zero reference position (more than one is possible)

Of the four listed commands, G28 is used almost exclusively in two and three axis CNC programming. Its only purpose is to return the current tool to the machine zero position and do it along one or more axes specified in the G28 program block.

◆ Command Group

All four preparatory commands G27 to G30 belong to the group 00 of the standard Fanuc designation that describes *non modal* or *one-shot* G-codes. In this designation, each G code of the 00 group must be repeated in every block it is used in. For example, when G28 command is used in one block for Z-axis and then it is used in the next block for X and Y axes, it has to be repeated in *each* block as needed:

N230 G28 Z.. (MACHINE ZERO RETURN Z-AXIS)
N231 G28 X.. Y.. (MACHINE ZERO RETURN XY AXES)

The G28 command in block N231 *must* be repeated. If this command is omitted, the *last motion* command programmed will be effective, for example, G00 or G01!

RETURN TO PRIMARY MACHINE ZERO

Any single CNC machine may have more than one machine zero reference point (home position), depending on its design. For example, many machining centers with a pallet changer have a *secondary* machine reference position, that is often used to align both the left and right pallets during pallet changing. The most common machine tool design is the one that uses only a single home position. To reach this primary home position, preparatory command G28 is used in the program and can also be used during MDI control operation.

G28 command moves the *specified* axis or axes to the home position, always at a rapid traverse rate. That means G00 command is assumed and does not have to be programmed. Axis or axes of the desired motion (with a value) must *always* be programmed. Only the programmed axes will be affected.

For example,

N67 G28

shows G28 programmed by itself in the block - this is an *incomplete* instruction. At least one axis must be specified with the G28 command, for example,

N67 G28 Y..

which will only send the Y-axis to machine zero reference position, or ...

N67 G28 Z..

will only send the Z-axis to machine zero reference position, and ...

N67 G28 X.. Y.. Z..

will send all three specified axes to machine zero reference position. Any multi-axis motion requires caution - *watch for the infamous 'hockey stick' motion.*

◆ Intermediate Point

One of the elementary requirements of programming is the alpha numerical composition of a word. In the program, every letter must be followed by one or more digits. The question is what *values* will the axes in G28 have? They will be the *intermediate* point for machine zero return motion. The concept of an intermediate motion in G28 or G30 is one of the most misunderstood programming features.

Commands G28 and G30 must always contain the *intermediate point* (tool position). By Fanuc design and definition, G28/G30 commands have a *built-in* motion to an intermediate point, *on the way* to machine zero. An analogy can be made to an airplane flight from Los Angeles, USA to Paris, France, that temporarily stops over in New York City. It may not be the most direct route, but it serves a certain specific purpose, for example, to refuel the aircraft.

Coordinate values of axes associated with G28 and G30 commands always indicate an intermediate point

The purpose of the intermediate point, or position, is to shorten the program, normally by one block. This reduction is so marginal that the philosophy behind the design may be debated. Here is how the concept of intermediate point (position) works.

When G28 or G30 command is used in a program, *at least one axis must be specified* in the block. The value of that axis is the intermediate point, as interpreted by the control system. Absolute and incremental modes G90 and G91 make a great difference in interpretation of the G28 or G30 behavior, and will be described shortly.

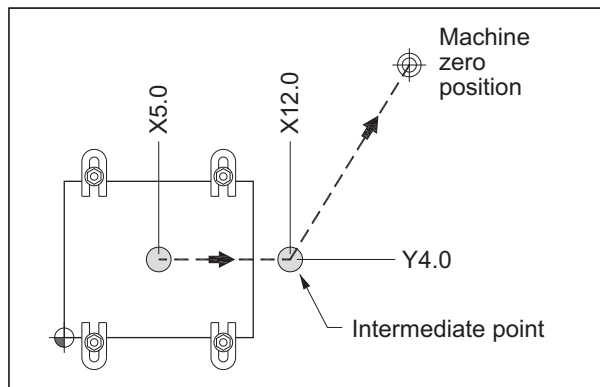


Figure 21-4

Intermediate point for machine zero return - XY axes shown

Tool motion in Figure 21-4 is from the central hole of the part. During such a motion, the tool can collide with the upper right clamp on its way to machine zero, if the motion to the home position were programmed directly. Only the X and Y axes are considered in the illustration. An *intermediate point* can be programmed in a safe location, without making the program any longer. Program *without* an effective intermediate point can be constructed as:

```
G90
...
G00 X5.0 Y4.0                (MACHINED HOLE)
G28 X5.0 Y4.0                (MACHINE ZERO MOTION)
...
```

The same program *with* an actual intermediate point at a safe location will change slightly:

```
G90
...
G00 X5.0 Y4.0                (MACHINED HOLE)
G28 X12.0 Y4.0              (MACHINE ZERO MOTION)
...
```

Earlier examples have shown the reason behind this double motion. It is very simple - only to *save a single program block* - that is all. Its intended purpose is to use one block of program to achieve two motions, that would otherwise require two blocks. A safe program could also be:

```
G90
...
G00 X5.0 Y4.0                (MACHINED HOLE)
X12.0                        (SAFE LOCATION)
G28 X12.0 Y4.0              (MACHINE ZERO RETURN)
...
```

to produce the same final result, but *with an extra block*.

For example, using the intermediate position, a tool can be programmed to avoid an obstacle on the way to machine zero. If programmed with care, the intermediate position may be quite useful. Normally, it is more practical to make

the intermediate motion equal to zero and move the cutting tool to machine zero directly. This is done by specifying the intermediate point as *identical* to the current tool position in *absolute* mode - or - by specifying a zero tool motion in *incremental* mode.

◆ Absolute and Incremental Mode

There is a major difference in programming machine zero return command G28 or G30 in *absolute* and *incremental* modes. Remember two basic differences between two similar statements:

G90 G00 X0 Y0 Z0 *and* G91 G00 X0 Y0 Z0

Each coordinate statement X0Y0Z0 is interpreted by the control system differently. To review, an address followed by a zero, for example X0, means *position at the program reference point*, if the mode is *absolute*, using the G90 command. If the mode is *incremental*, using the G91 command, the X0 word means *no motion* for the specified axis.

Most CNC lathes use the U and W addresses for incremental motion (based on absolute X and Z axes respectively), with the same logical applications. Absolute axes coordinates will be interpreted as the *programmed tool position*, incremental coordinates indicate the *programmed tool motion*.

Compare the two program examples below - they are the same - they are *identical* in terms of the actual tool motion:

```
(---> G28 USED IN ABSOLUTE MODE G90)
G90
...
N12 G01 Z-0.75 F4.0 M08
...
N25 G01 X9.5 Y4.874
N26 G28 Z-0.75 M09          (G28 IN ABSOLUTE MODE)
...
```

```
(---> G28 USED IN INCREMENTAL MODE G91)
G90
...
N12 G01 Z-0.75 F4.0 M08
...
N25 G01 X9.5 Y4.874
N26 G91 G28 Z0 M09         (G28 IN INCREMENTAL MODE)
...
```

Which method is better? Since both methods produce identical results, the choice is based on a given situation or personal preference. To switch to the incremental mode has its benefit, because the current tool location may not always be known. One disadvantage of this method is that G91 is most likely a temporary setting only and must be reset back to G90 mode, used by most programs.

A failure to reinstate the absolute mode may result in an expensive and possibly serious error

Absolute mode of programming specifies the current tool position from program zero - *always* and *at all times*. Many examples presented here use the absolute programming mode - after all, this is - or it should be - the standard programming mode, for majority of programs.

There is one time, where incremental mode of machine zero return has some very practical advantages. It happens in those cases when the current tool position *is not known* to the programmer. Such a situation typically happens when using subprograms, where incremental mode is used repeatedly to move a tool incrementally to different XY locations. For instance - where *exactly* is the cutting tool located when the drilling cycle is completed in block N35 of the following example?

```
G90
...
N32 G99 G81 X1.5 Y2.25 R0.1 Z-0.163 F12.0
N33 G91 X0.3874 Y0.6482 L7      (REPEAT 7 TIMES)
N34 G90 G80 Z1.0 M09           (CANCEL CYCLE)
N35 G28 (X???? Y????) Z1.0    (UNKNOWN POSITION)
...
```

Is it worth the extra effort to find the absolute location at all costs? Probably not. Let's look at some other examples. While in the absolute mode G90, axis coordinate settings define the intermediate point *location*. When incremental mode G91 is programmed, the coordinate settings define the actual distance and direction of the intermediate *motion*. In both cases, the intermediate tool motion will be performed first. Then - and only then - the final return to the machine zero reference position will take place.

Take the current tool position as X5.0 and Y1.0 (absolute position). In the program, the XY amounts of the G28 command that follows the position block are very important:

```
G90
...
N12 G00 X5.0 Y1.0
N13 G28 X0 Y0
...
```

In this example, the G28 command specifies that cutting tool should reach the machine zero position - identified as X0Y0 in block N13. Since G28 command relates to machine zero exclusively, it would be reasonable to assume that X0Y0 relates to machine zero, rather than the part zero. *That is not correct.*

X0Y0 refers to the XY point through which the tool will reach the machine zero position. That is the defined point already known to be the *intermediate position* for the machine zero return command. This intermediate point is assigned coordinates relating to the part (in absolute mode). In the example, cutting tool will move to program zero *before* continuing to the machine zero, resulting in a single block definition of two tool motions. This, of course, is not likely to be the intended motion.

The above example can be changed, so the intermediate motion is eliminated - or - *defined as the current tool position*. Intermediate motion can never be eliminated, but it can be programmed as a physical travel distance of zero.

```
G90
...
N12 G00 X5.0 Y1.0
N13 G28 X5.0 Y1.0
...
```

By this modification, the intermediate point becomes the *current tool position*, which results in *direct* motion to the machine zero. The reason is that the intermediate tool position *coincides* with the current tool position. This programming format has nothing to do with modal values of axes. In the part program, X5.0Y1.0 in block N13 must be repeated, while the absolute mode G90 is still in effect.

In cases when the current tool position is *not* known, the machine zero return has to be done in *incremental mode*. In this case, change temporarily to incremental mode and program a zero length motion for each specified axis:

```
G90
...
... (X??? Y???) (ANY UNKNOWN XY POINT)
N13 G91 G28 X0 Y0
N14 G90 ...
...
```

Again, an important remainder is in place here - always remember to switch back to absolute mode as soon as possible, in order to avoid misinterpreting the consecutive program data.

In a brief overview, the intermediate point cannot be eliminated from the G28/G30 block. If situation demands a return to machine zero without going through a separate intermediate point, use a zero tool motion towards the intermediate point. This method depends on the active G90 or G91 mode at the time:

- In G90 absolute mode motion to machine zero, the current tool coordinate location must be repeated for each axis specified with G28 command
- In G91 incremental motion to machine zero, the current tool motion must be equal to zero for each axis specified with the G28 command

◆ Return from Z-depth Position

One common example of using the intermediate tool position in a program block, is the return from a deep hole or a cavity to machine zero. In the following example - and solely for the purpose of better explanation - regular tool motions are used rather than a drilling cycle, to retract the tool from the hole depth. In the example, the current XY position is X9.5Y4.874, and a peck drilling operation will be simulated in separate blocks:

```

...
N21 G90 G00 G54 X9.5 Y4.874 S900 M03
N22 G43 Z0.1 H01 M08
N23 G01 Z-0.45 F10.0
N24 G00 Z-0.43
N25 G01 Z-0.75

```

In block N25, tool is at the bottom of the hole, at a current tool position of X9.5 Y4.874 Z-0.75 absolute coordinates. All the cutting is done and the tool has to be returned home in all three axes. For safety reasons, the Z-axis must retract first. Several options can be selected, but three of them are the most common:

- Retract the Z-axis above work in one block, then return XYZ axes to machine zero
- Retract the Z-axis all the way to machine zero, then return XY axes in the next block
- Return XYZ axes to machine zero directly from the current tool position (at the depth)

The *Figure 21-5* shows the available options.

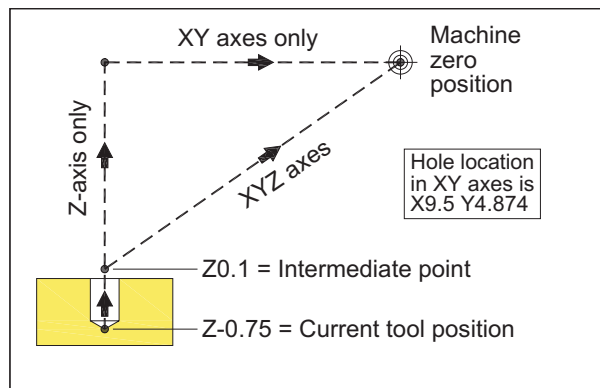


Figure 21-5

Machine zero return from a hole depth - milling

➡ Option 1

To retract the Z-axis above work in one block first, then return XYZ axes to the machine zero position, would be the 'normal' method, commonly used:

```
N26 G00 Z0.1 M09
```

This block must be followed by a return to the home position, along the Z-axis:

```
N27 G28 Z0.1 M05
```

Complete program segment for *Option 1* will be:

```

...
N21 G90 G00 G54 X9.5 Y4.874 S900 M03
N22 G43 Z0.1 H01 M08
N23 G01 Z-0.45 F10.0

```

```

N24 G00 Z-0.43
N25 G01 Z-0.75
N26 G00 Z0.1 M09
N27 G28 Z0.1 M05
N28 G28 X9.5 Y4.874
N29 M01

```

➡ Option 2

To retract the Z-axis all the way to machine zero first and then return XY axes in the next block, is a variation on *Option 1*. First, return the Z-axis to machine zero:

```
N26 G28 Z-0.75 M09
```

Then, return the XY axes to machine zero as well:

```
N27 G28 X9.5 Y4.874
```

Complete program segment for *Option 2* will be:

```

...
N21 G90 G00 G54 X9.5 Y4.874 S900 M03
N22 G43 Z0.1 H01 M08
N23 G01 Z-0.45 F10.0
N24 G00 Z-0.43
N25 G01 Z-0.75
N26 G28 Z-0.75 M09
N27 G28 X9.5 Y4.874 M05
N28 M01

```

➡ Option 3

To return all three axes XYZ to machine zero directly from the current tool position (while the tool is still at the hole full depth), only *one* zero return block will be needed:

```
N26 G28 X9.5 Y4.874 Z0.1 M09
```

This is the intended method of programming, as Fanuc controls are designed. Some programmers may disagree with Fanuc on this issue, but that is how it works.

Here is the complete program segment for *Option 3*:

```

...
N21 G90 G00 G54 X9.5 Y4.874 S900 M03
N22 G43 Z0.1 H01 M08
N23 G01 Z-0.45 F10.0
N24 G00 Z-0.43
N25 G01 Z-0.75 M09
N26 G28 X9.5 Y4.874 Z0.1 M05
N27 M01

```

The motion to machine zero will take two steps:

- Step 1: Z-axis will rapid to Z0.1 position
- Step 2: All axes will return to machine zero

Also note the rearrangements of M09 and M05 miscellaneous functions. Turning the coolant off first is more practical than stopping the spindle.

Although this is a matter of opinion, the choice of many programmers is to move the tool out of a cavity or hole first, then call the machine zero return command. If there is any justification for this preference, it is the perceived safety the CNC programmer puts into program design. To be fair here, there is absolutely nothing wrong with the alternate method, if it is used with care. Comparing individual options with each other does offer some valuable conclusions:

■ **OPTION 1 ...**

... is only reasonably safe, but quite efficient in terms of cycle time. There may be a possibility of an obstacle within the three-axis motion to machine zero

■ **OPTION 2 ...**

... is somewhat less efficient than the previous option, but definitely the safest one of all three

■ **OPTION 3 ...**

... is the most efficient in terms of program cycle time, but any error in position could result in a collision

◆ **Axes Return Required for the ATC**

If the only purpose of machine zero return is to make an automatic tool change, only certain axes must be moved for that purpose. For a vertical machining center, only the Z axis is required to make the tool change:

G91 G28 Z0 M06

Horizontal machining centers require only the Y-axis to reach its reference position for automatic tool change. For additional safety and extra convenience, Z-axis is usually programmed as well, along with Y-axis, to prevent a collision with an adjacent tool in the magazine:

G91 G28 Y0 Z0 M06

In both examples, the tool change function M06 will *not* be effective, until the machine zero reference position has been physically reached. M06 tool change function can be programmed in a separate block later, if desired.

Indexing or rotary axes also have their own reference point and are used with G28 command the same way as linear axes. For example, B-axis will return to the machine zero reference position in the following block:

G91 G28 B0

If it is safe, the B-axis may be programmed simultaneously with another axis:

G91 G28 X0 B0

Absolute mode designation follows the same rules for a rotary or indexing axis, as for the linear axes.

◆ **Zero Return for CNC Lathes**

For CNC lathe work, the G28 command may also be used, usually for setup. Common application of the machine zero return is also used, when at least one axis starts and ends at the machine zero position. This is quite often true of the X-axis but not of the Z-axis, which may be too far away on some larger lathe models.

Typically, a CNC lathe program will be designed in such a way, that machining of the first part will start from machine zero, but any subsequent part will be machined from a safe tool change position. This method is only practical if the program uses geometry offset, rather than the older G50 setting. The most common method of machine zero return on CNC lathes is the direct method, without an intermediate point, because no G91 is required, therefore, an error is more difficult to make:

N78 G28 U0

N79 G28 W0

These two blocks will return the cutting tool to machine zero in incremental mode, there is no intermediate motion applied. It is safer to move the X-axis first, using the incremental mode U, then the Z-axis, using the incremental mode W. If the work area is clear (watch for tailstock), both X and Z axes can be returned to machine zero at the same time:

N78 G28 U0 W0

Figure 21-6 illustrates a typical withdrawal of a boring bar from a hole, when the machining is completed.

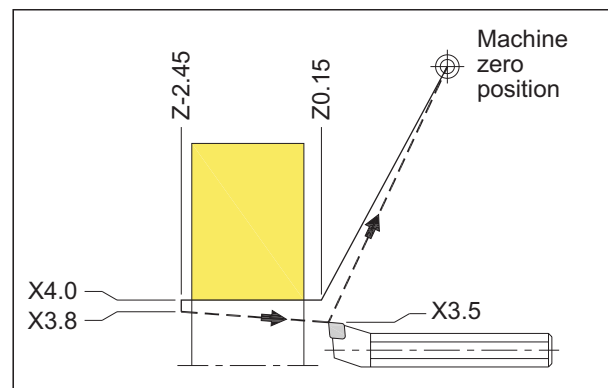


Figure 21-6

Machine zero return from a hole depth - turning application

When using position register command G50, the XZ setting must always be known for this command. In this case, programming rules for machine zero return are very similar. Assuming that the machine zero position is at the coordinate position X10.0 Z3.0, program for a boring tool can be written in two ways - one *without* using the G28 command, the other one *with* the G28 command.

☛ Example 1 :

The first example does not use G28 machine zero return command at all:

```
N1 G20 (EXAMPLE 1)
...
N58 G50 X10.0 Z3.0 S1000 (OLDER METHOD ONLY)
N59 G00 T0300 M42
N60 G96 S400 M03
N61 G00 G41 X4.0 Z0.15 T0303 M08
N62 G01 Z-2.45 F0.012
N63 X3.8 M09
N64 G00 G40 X3.5 Z0.15 M05
N65 X10.0 Z3.0 T0300
N66 M01
```

☛ Example 2 :

The second example will use G28 machine zero reference command, to achieve the same target position:

```
N1 G20 (EXAMPLE 2)
...
N58 G50 X10.0 Z3.0 S1000 (OLDER METHOD ONLY)
N59 G00 T0300 M42
N60 G96 S400 M03
N61 G00 G41 X4.0 Z0.15 T0303 M08
N62 G01 Z-2.45 F0.012
N63 G40 X3.8 M09
N64 G28 X3.5 Z0.15 M05 T0300
N65 M01
```

Most CNC programmers will likely feel more comfortable with the first example and saving one program block. The second example (*Example 2*) can be programmed in incremental mode as well, using the U and W addresses, but it would not be too practical.

RETURN POSITION CHECK COMMAND

Much less common preparatory command G27 performs a checking function - and nothing else. Its only purpose is to check (which means to *confirm*), if the programmed position in the block containing G27 is at the machine zero reference point or not. If it is, the control panel indicator light for each axis that has reached the position will go on. If the reached position is not at the machine zero, program processing is interrupted by an error condition displayed on the screen as an alarm.

If the tool starting position is programmed at the machine zero reference (home), it is a good practice to return there as well, when machining with that cutting tool is fully completed. This is quite commonly done for CNC lathes, where the tool change (indexing) normally takes place in the same position, although this position does not always have to be machine zero. Usually, it is a safe position near the machined part.

Format for G27 command is:

G27 X. . Y. . Z. .

where at least one axis must be specified.

When used in a program, the cutting tool will automatically rapid (no G00 necessary) to the position as specified by the axes in G27 block. The motion can be either in absolute or incremental mode. Note that no G28 command is used in the program.

```
N1 G20
N2 G50 X7.85 Z2.0 (OLDER METHOD ONLY)
N3 G00 T0400 M42
N4 G96 S350 M03
N5 G00 G42 X4.125 Z0.1 T0404 M08
N6 G01 Z-1.75 F0.012
N7 U0.2 F0.04
N8 G27 G40 X7.85 Z2.0 T0400 M09
N9 M01
```

In the example, block N8 contains G27, but no G00 or G28. This block instructs the CNC machine to return to position X7.85 Z2.0 and check, upon arrival to this target position, if that position is the machine zero in *all* specified axes (two axes in the example). Confirmation light will turn on, if the machine zero position has been reached. If the position is not confirmed, program will not proceed any further until the cause (misposition) is eliminated.

Compare the starting position in block N2 and the return position in block N8. Assuming that this position is at machine zero reference point in both X and Z axes, the above example will confirm OK position in the N8 block. Now, suppose that a small error has been made while writing block N8, and the X amount was entered as X7.58 rather than the expected X7.85:

```
N8 G27 G40 X7.58 Z2.0 T0400 M09
```

In this case, the control system will return an error condition. The error is displayed automatically on the control screen (as an alarm). Control system will *not* process the remainder of the program, until any error is corrected. The light indicating *Cycle Start* condition will turn off and the problem *source* has to be found. When looking for source of the problem, always check *both positions*, the start position block, as well as the end position block. The error is quite easy to make in either block. Also note that any axis *not* specified in the block will not be checked for its actual (current) position.

Another important point is the cancellation of cutter radius offset and tool offset. G27 preparatory command should always be programmed with G40 command and *Txx00* in effect (G49 or H00 for milling). If the tool offset or cutter radius offset is still in effect, the checking cannot be done properly, because the tool reference point is displaced by the offset value.

Here is how the first program (*Example 1*) listed earlier, can be modified to accept G27 command. Note that G27 will only move to the coordinates specified, *not* to any intermediate or other point. Block N65 will become the actual check block. The control system will move the machine axes to X10.0 Y3.0 and checks (confirms) whether this position is in fact the machine zero reference point. This is the reason *Example 1* could be modified, but not the second *Example 2*.

```

N1 G20
...
N58 G50 X10.0 Z3.0 S1000    (OLDER METHOD ONLY)
N59 G00 T0300 M42
N60 G96 S400 M03
N61 G00 G41 X4.0 Z0.15 T0303 M08
N62 G01 Z-2.45 F0.012
N63 X3.0 M09
N64 G00 G40 X3.5 Z0.15 M05
N65 G27 X10.0 Z3.0 T0300
N66 M01

```

Machine reference point return check can be done in either absolute or incremental mode. Absolute coordinates in block N65 (in the last example) can be replaced with incremental version:

```
N65 G27 U6.5 W2.85 T0300
```

There is a drawback to this command. A small price to pay when using this checking command is a slight cycle time loss. Because deceleration of a tool motion is built into the command by the control system, about one to three seconds may be lost when G27 command is implemented. This may be a significant loss if a large number of tools use G27 check in every program.

G27 command is seldom used with *geometry offset* setting of the tools, which is the current modern method. The G50 command is older and not used anymore on the newest CNC lathes, but many lathes are still used in industry that do need the G50 setting.

RETURN FROM MACHINE ZERO POINT

Preparatory command G29 is the exact opposite of G28 or G30 command. While G28 will automatically return the cutting tool to machine zero position, G29 command will return the tool to its *original* position - also *via an intermediate point*.

In normal programming usage, the command G29 usually follows G28 or G30 command. Rules relating to the absolute and incremental axis designation are valid for G29 in exactly the same respect as to G28 and G30. All programmed axes are moved at the rapid traverse rate to the intermediate position first, defined by the preceding G28 or G30 command block. An example for a lathe application illustrates the concept:

(LATHE EXAMPLE)

```

...
T0303
...
G28 U5.0 W3.0
G29 U-4.0 W2.375

```

G29 command should always be issued in the canceled mode of both cutter radius offset (G40) and fixed cycles (G80), if either is employed in the program. Use the standard cancellation G-codes - G40 to cancel cutter radius offset and G80 to cancel a fixed cycle, before G29 command is issued in the program.

A schematic sketch of the tool motion is illustrated in Figure 21-7.

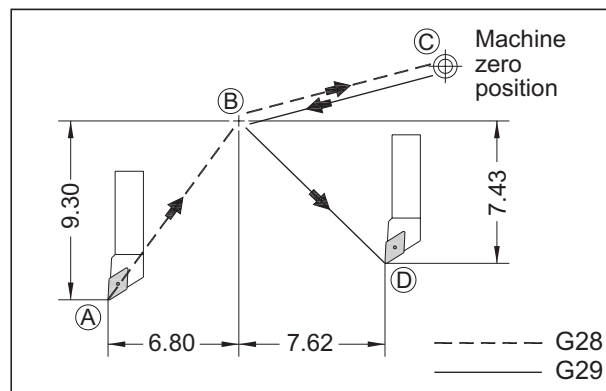


Figure 21-7
Automatic return from machine zero position

The illustration shows a tool motion from point A to point B first, then to point C, back to point B, and finally, to the point D. Point A is the starting point of the motion, point B is the intermediate point, point C is the machine zero reference point, and point D is the final point to reach - the actual target position.

Equivalent program commands, starting at the current tool position, which is point A, and resulting in the A to B to C to B to D tool path are quite simple:

```

G28 U18.6 W6.8
...
G29 U-14.86 W7.62

```

Of course, there would be some appropriate action programmed between the two blocks, for example, a tool change or some other machine activity.

Similar to G27 command, there is only a weak support for G29 among CNC programmers. It is one of the commands that can be very useful in some very rare cases, but virtually unnecessary for everyday work. However, it is always an advantage to know what 'tools of trade' are available in CNC programming. They may come handy.

RETURN TO SECONDARY MACHINE ZERO

In addition to the G28 machine zero command, specific CNC machines also have the G30 command. In this chapter, and the handbook generally, many examples apply equally to G28 and G30 commands and were sometimes identified as G28/G30 to cover both. So what is different in G30 and why is this command needed in the first place?

By definition, G30 preparatory command is a machine zero return command to the *secondary machine zero position*. That position must be available on the machine at the time of purchase. Note the descriptive word is *secondary*, not *second*. In virtually all respects, G30 is identical to the G28, except that it refers to a *secondary* program zero.

This secondary program zero can be the physical second, third, or even fourth reference point, as specified by the machine manufacturer. Not every CNC machine has a secondary machine zero reference position, and not every CNC machine even needs one. This secondary machine reference point serves only some very special purposes, mainly for horizontal machining centers, pallets, and similar equipment.

Programming format for G30 command is similar to the G28 command, with an addition of the P-address:

G30 P.. X.. Y.. Z..

where ...

- G30 = Indicates the selection of a secondary reference position
- P = Can be P2, P3 and P4 to identify the secondary position (2-4)
- XYZ = Is the intermediate point definition (one axis minimum must be specified)

The most common use of a secondary machine zero reference point in CNC programming is for pallet changing or multi spindle machines. In the control unit parameter setting, the distance of the secondary reference point is set from the primary reference point and is not normally changed during the working life of the machine and the pallet changer (or other machine feature).

To distinguish between multiple secondary machine zero positions, address P is added in the G30 block (there is no P address used for G28). If the CNC machine has only a single secondary machine reference position, the address P is usually *not* required in the program, and P1 is assumed in such a case:

G30 X. . Y. .

is the same as

G30 P1 X.. Y..

In this case, setting of the second reference point is within preset parameters of the control system. In respect to other programming considerations, G30 command is used in exactly the same way as the much more common G28 machine zero return command.

22

LINEAR INTERPOLATION

Linear interpolation is closely related to rapid positioning motion. While rapid tool motion is meant to be used from one position of the work area to another position *without cutting*, linear interpolation mode is designed for actual *material removal*, such as contouring, pocketing, face milling and many other cutting motions.

Linear interpolation is used in part programming to make a straight cutting motion from the cutter start position to its end position. It always uses the *shortest distance* a cutting tool path can take. The motion programmed in linear interpolation mode is always a straight line, connecting the contour start and end points. In this mode, the cutter moves from one position to another by the shortest distance between the end points. This is a very important programming feature, used mainly in contouring and profiling. Any angular motion (such as chamfers, bevels, angles, tapers, etc.) must be programmed in this mode to be accurate. Three types of motion can be generated in the linear interpolation mode:

- Horizontal motion ... single axis only
- Vertical motion ... single axis only
- Angular motion ... multiple axes

The term *linear interpolation* means that the control system is capable to calculate thousands of intermediate coordinate points between the start point and end point of the cut. Result of this calculation is the shortest path between the two points. All calculations are automatic - the control system constantly coordinates and adjusts the feedrate for all cutting axes, normally two or three.

LINEAR COMMAND

G01	Linear interpolation (linear motion)
-----	--------------------------------------

In G01 mode, the feedrate function F must be in effect. The first program block that starts linear interpolation mode must have a feedrate in effect, otherwise an alarm will occur during the first run, after power on. Command G01 and feedrate F are modal, which means they may be omitted in all subsequent linear interpolation blocks, once they have been designated, and providing the feedrate remains unchanged. Only a change of coordinate location is required for the axis designation in a program block. In addition to a single axis motion, linear motion along two or three axes may be also programmed simultaneously.

◆ Start and End of the Linear Motion

Linear motion, like any other motion in CNC programming, is a motion between two end points of a contour. It has the *start* position and the *end* position. Any *start* position is often called the *departure* position, the *end* position is often called the *target* position. Start of a linear motion is defined by the current tool position, its end is defined by the target coordinates of the current block. It is easy to see that the end position of one motion will become the start position of the next motion, as the cutter moves along the part, through all contour change points.

◆ Single Axis Linear Interpolation

Programmed tool motion along any single axis is always a motion parallel to that axis, regardless of the motion mode. Programming in either G00 or G01 mode will result in the same programmed end point, but at different feedrates and with different results. Evaluate Figure 22-1 for comparison of the two motion modes.

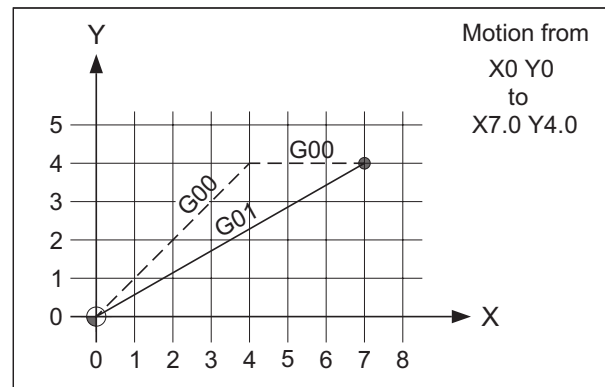


Figure 22-1

Comparison of the rapid mode and the linear interpolation mode

For CNC machining centers and the related machines, all tool motions that are parallel to the table edges are single axis motions. On CNC lathes, many external and internal operations, such as facing, shoulder turning, diameter turning, drilling, tapping and others, are programmed as single axis motions. In all cases, a single axis motion can be along either the *vertical* or the *horizontal* axis, within the current (working) plane. A single axis motion can never be an angular motion, which requires two or three axes. Another name for a motion that is parallel to a machine axis is *orthogonal* - horizontal or vertical only.

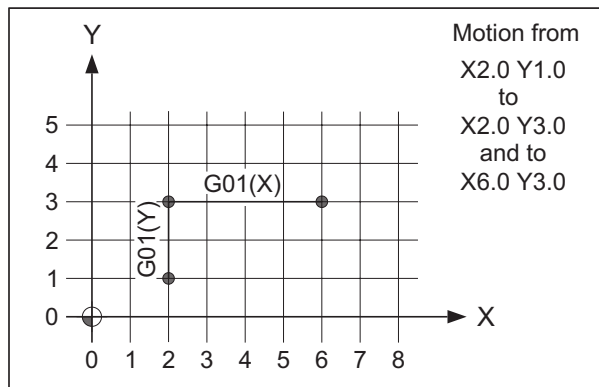


Figure 22-2

Single axis linear interpolation motion

Figure 22-2 illustrates a single axis linear interpolation motion, one along X-axis and the other along Y-axis.

◆ Two Axes Linear Interpolation

Linear motion can also be programmed along *two axes* simultaneously. This is a very common situation when the start point of linear motion and its end point have at least two coordinates that are different from each other, while in linear interpolation mode G01. The result of this two-axis motion is a straight tool motion *at an angle*. Such motion will always be the shortest distance between the start point and end point and results in a straight line at an angle calculated by the control - Figure 22-3.

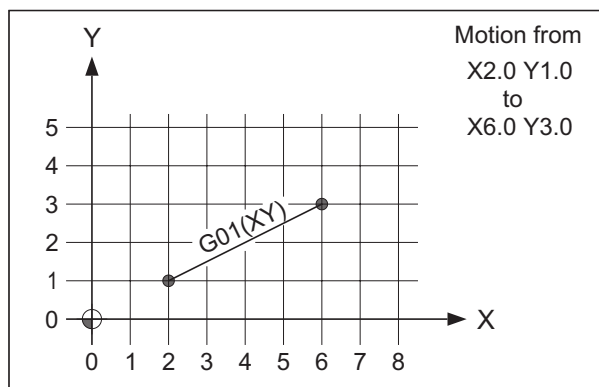


Figure 22-3

Two axes simultaneous linear interpolation motion

◆ Three Axis Linear Interpolation

Linear motion that takes place along three axes at the same time, is called the *three axis linear interpolation*. A simultaneous linear motion along three axes is possible on virtually all CNC machining centers. Programming a linear motion of this kind is not always easy, particularly when working with complex parts. Due to many difficult calculations involved in this type of tool motion, the manual pro-

gramming method is not efficient enough. Such programming projects more than justify an investment into a professional computer based programming system, such as the very powerful and widely used *Mastercam™*, that is based on modern computer technology combined with machining know-how. This type of programming is using desktop computers and is affordable by virtually all machine shops. Computer based programming is not a subject of this handbook, but its general concepts are discussed briefly in the last chapter of the handbook.

Three-axis (XYZ) simultaneous linear motion is illustrated in Figure 22-4.

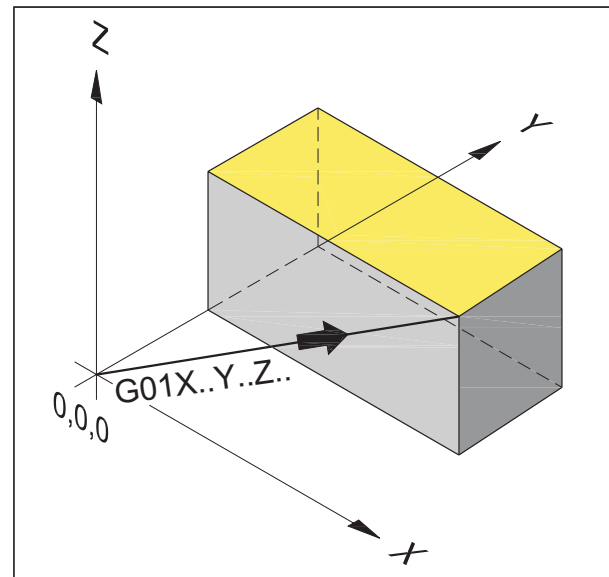


Figure 22-4

Three axes simultaneous linear interpolation motion

PROGRAMMING FORMAT

In order to program a tool motion in linear interpolation mode, use preparatory command G01 along with one, two, or three axes of tool motion, as well as a cutting feedrate (F address) suitable for the job at hand:

G01 X.. Y.. Z.. F..

All entries in linear motion block are modal and need to be programmed only if they are new or changed. Only the block instruction (word) that is affected by the change needs to be included in the program block.

Depending on which programming method is selected, linear interpolation motion may be programmed in absolute or incremental mode, using G90 and G91 preparatory commands for milling, and incremental addresses U and W for turning.

LINEAR FEEDRATE

The actual cutting feedrate for a defined tool motion can be programmed in two modes:

- ... per time mm/min *or* in/min
- ... per spindle revolution mm/rev *or* in/rev

The selection depends on the machine type and dimensional units used. Typically, CNC machining centers, drills, mills, routers, flame cutters, laser profilers, wire EDM, etc., use *feedrate per time*. CNC lathes and turning centers typically use *feedrate per revolution*.

◆ Feedrate Range

Every CNC system supports cutting feedrate only within a certain range. For linear interpolation in milling applications, the typical lowest feedrate is 0.0001, either as *in/min*, *mm/min* or *deg/min*. The lowest feedrate for linear interpolation in turning is dependent on the minimum increment of the coordinate axes XZ. The following two tables point out typical ranges a normal CNC system can support. The first table is for milling, the second table is for turning. All units used in part programming are represented.

Minimum motion increment	MILLING
0.001 mm	0.0001 - 240000.00 mm/min
0.001 degree	0.0001 - 240000.00 deg/min
0.0001 inch	0.0001 - 240000.00 in/min

Minimum motion increment	TURNING
0.001 mm	0.00001 - 500.00000 mm/min
0.001 degree	0.00001 - 500.00000 deg/min
0.0001 inch	0.000001 - 50.000000 in/min

It may appear that the maximum feedrate that can be used is unusually high. For actual cutting, that is true. However, keep in mind that these ranges are relative to the control system, *not* to the machine. Machine manufacturer will always limit the maximum feedrate, according to the machine design and its capabilities. Control system only provides the *theoretical range*, that is more for the benefit of machine designers than the actual user. The intent in this case is to allow machine manufacturers flexibility within current technological advances. As technology changes, the control system manufacturers will have to respond to these changes as well, by increasing the available ranges.

◆ Individual Axis Feedrate

The subject of actual cutting feedrate per axis is not crucial in programming at all. It is included here for the mathematically oriented and interested readers only. There is no need to know the following calculations at all - the CNC system will do them every time, all the time, accurately and automatically. On the other hand, here it is anyway.

In order to keep linear motion as the *shortest* motion between two points, the CNC unit must always calculate the feedrate for *each axis individually*. Depending on the direction of linear motion (its angular value), the computer will 'speed up' one axis and 'hold back' the other axis *at the same time*, and it will do it constantly during every cut. The result is a straight line between the start and end points of the linear contour. Strictly speaking, it is not a straight line but a jagged line, with edges so diminutive in size that they are virtually impossible to see, even under magnification. For all practical purposes, the result *is* a straight line.

Calculations are done by the CNC system, according to the following entries, as illustrated in *Figure 22-5*.

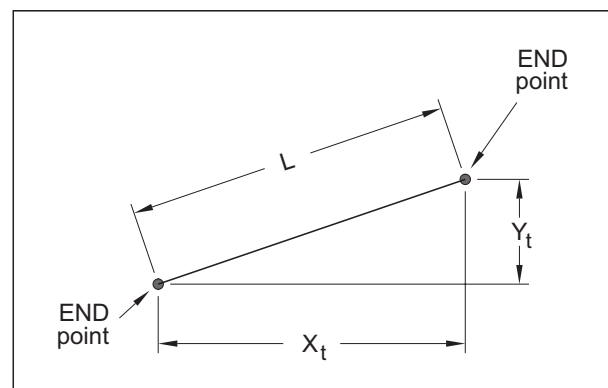


Figure 22-5

Data for the calculation of individual axis linear feedrate

Evaluate the following example of linear motion and try to apply the formulas listed afterwards:

```
G00 X10.0 Y6.0          (START POINT)
G01 X14.5 Y7.25 F12.0   (END POINT)
```

This linear motion takes place between two end points, from the starting point at X10.0 Y6.0 to the end point at X14.5 Y7.25 - feedrate is programmed at 12.0 in/min as F12.0. That means the actual travel motion along each axis is either known or it can be calculated:

$$\begin{aligned} X_t &= 14.5 - 10.0 = 4.5 \\ Y_t &= 7.25 - 6.0 = 1.25 \\ Z_t &= 0 \end{aligned}$$

The length L of tool total motion (as illustrated) is the actual compound motion, and can be calculated by using the well known *Pythagorean Theorem* (see page 506):

$$L = \sqrt{X_t^2 + Y_t^2 + Z_t^2}$$

The above formula is quite common, based on the square root of the total sum of squares of sides, that will result in the value of 4.6703854 as travel length in the example:

$$L = \sqrt{4.5^2 + 1.25^2 + 0^2} = 4.6703854$$

Control system will *internally* apply the formulas and calculate the actual motion along X-axis (4.25), as well as along Y-axis (1.25), plus length of the motion itself, which is 4.6703854. From these values, the computer system will calculate the X and Y axis feedrate - there is no motion that takes place along the Z-axis:

$$F_x = \frac{X_t}{L \times F}$$

$$F_x = 4.5 / 4.6703854 \times 12 = 11.562215$$

$$F_y = \frac{Y_t}{L \times F}$$

$$F_y = 1.25 / 4.6703854 \times 12 = 3.2117263$$

$$F_z = \frac{Z_t}{L \times F}$$

$$F_z = 0 / 4.6703854 \times 12 = 0.0$$

In this example, there is no Z-axis motion. If Z-axis were part of the tool motion, for example, during a simultaneous three dimensional linear motion, the procedure will be logically identical, with the inclusion of Z-axis in the calculations.

PROGRAMMING EXAMPLE

In order to illustrate the practical use of linear interpolation mode in a CNC program, here is a simple example, shown in *Figure 22-6*.

For even more comprehensive understanding, the example will be presented twice. One tool motion will start and end at the P1 location and will be programmed in the order of P1-P2-P3-P4-P5-P6-P7-P8-P1. The other program example will start at the same P1 location, but will continue in the opposite direction P1-P8-P7-P6-P5-P4-P3-P2-P1.

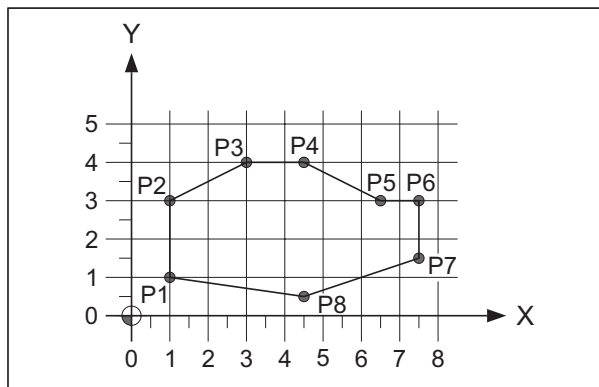


Figure 22-6

Example illustration for a simple linear interpolation

➡ Example 1 :

('CLOCKWISE' DIRECTION FROM P1)

```
G90 ... (ABSOLUTE MODE)
G01 X1.0 Y3.0 F.. (P1 TO P2)
X3.0 Y4.0 (P2 TO P3)
X4.5 (P3 TO P4)
X6.5 Y3.0 (P4 TO P5)
X7.5 (P5 TO P6)
Y1.5 (P6 TO P7)
X4.5 Y0.5 (P7 TO P8)
X1.0 Y1.0 (P8 TO P1)
...
```

➡ Example 2 :

('COUNTERCLOCKWISE' DIRECTION FROM P1)

```
G90 ... (ABSOLUTE MODE)
G01 X4.5 Y0.5 F.. (P1 TO P8)
X7.5 Y1.5 (P8 TO P7)
Y3.0 (P7 TO P6)
X6.5 (P6 TO P5)
X4.5 Y4.0 (P5 TO P4)
X3.0 (P4 TO P3)
X1.0 Y3.0 (P3 TO P2)
Y1.0 (P2 TO P1)
...
```

Linear interpolation provides means of programming all orthogonal (*i.e.*, vertical and horizontal) motions, as well as angular tool motions as the shortest linear distance between two points. Cutting feedrate must be programmed in this mode, for proper metal removal. Note that the coordinate location that has *not* changed from one point to the next - one block to the next - is not repeated in the subsequent block or blocks.

23

BLOCK SKIP FUNCTION

In many control and machine manuals, block skip function is also called the *block delete* function. The expression '*block delete*' offers rather a misleading description, since no program blocks will actually be deleted during processing, but only *skipped*. For this good reason, the more accurate description of this function is the *block skip* function, a term used in the handbook. This function is a standard feature of virtually all CNC controls. Its main purpose is to offer the programmer some additional flexibility in designing a program for no more than *two conflicting possibilities*. In the absence of a block skip function, the only alternative is to develop two individual part programs, each covering one unique possibility.

TYPICAL APPLICATIONS

To understand the idea of two conflicting possibilities, consider this programming application. The assignment is to write a program for a facing cut. The problem is that the blank material for parts delivered to the CNC machine is not consistent in size. Some blanks are slightly smaller in size and can be faced with a single cut. Others are larger and will require two facing cuts. This is not an uncommon occurrence in CNC shops and is not always handled efficiently. Making two inefficient programs is always an option, but a single program that covers both options is a better choice - but only if the *block skip function* is used in such a program and machine operator understands it.

This challenge illustrates a situation, where two conflicting options are required in a program at the *same* time. The most obvious solution would be to prepare two separate programs, each properly identified as to its purpose. Such a task can be done quite easily, but it will be a tedious, time consuming and definitely an inefficient process. The only other solution is to write a *single* program, with tool motions covering facing cuts for *both* possibilities. To avoid air cutting for those parts that require only one cut, a block skip function will be provided in the program and applied to all blocks relating to the *first* facing cut. The 'second' cut will always be required!

Other common applications of the block skip function include a selective ON/OFF status toggle, such as the coolant function, optional program stop, program reset, etc. Also useful are applications for bypassing a certain program operation, applying or not applying a selected tool to a part contour and others. Any programming decision that requires a choice from *two* predetermined options is a good candidate for the block skip function.

BLOCK SKIP SYMBOL

To identify block skip function in a program, a special programming symbol is required. This block skip function symbol is represented by a forward slash [/]. Control system will recognize the slash as a code for block skip. For most of CNC programming applications, the slash symbol is placed as the *first* character in a block:

➡ Example 1 :

```
N1 ... (ALWAYS PROCESSED)
N2 ... (ALWAYS PROCESSED)
N3 ... (ALWAYS PROCESSED)
/ N4 ... (PROCESSED ONLY IF BLOCK SKIP IS OFF)
/ N5 ... (PROCESSED ONLY IF BLOCK SKIP IS OFF)
/ N6 ... (PROCESSED ONLY IF BLOCK SKIP IS OFF)
N7 ... (ALWAYS PROCESSED)
N8 ... (ALWAYS PROCESSED)
```

On *some* control systems, the block skip code can also be used selectively for certain addresses *within* a block, rather than at its beginning. Check control manual if such a technique can be used - it can be very powerful:

➡ Example 2 :

```
N6 ...
N7 G00 X50.0 / M08
N8 G01 ...
...
```

In those cases, when the control system does allow the block skip *within* a programmed block, all instructions *before* the slash code will be executed, regardless of the block skip toggle setting. If the block skip function is turned ON (block skip function is active), only instructions *following* the slash code, will be skipped. In the *Example 2*, coolant function M08 (block N7) will be skipped. If the block skip function is turned OFF (block skip function is not active), the whole block will be executed in *Example 2*, including the coolant function.

CONTROL UNIT SETTING

Regardless of the slash code position within a block, the program will be processed in two ways. Either in its entirety, or any instruction following the slash within a block will be skipped (ignored). The final decision whether or not to use block skip function is made during actual machining,

by the operator, depending on the machining requirements. For this purpose, a push button key, a toggle switch, or a menu item selection is provided on the control panel of the CNC unit. Selection of block skip mode can be either as *active* (ON) - or *inactive* (OFF).

Most programs will not require any block skip functions. In such cases, the setting mode for block skip function on the control panel is irrelevant, but OFF mode is strongly recommended. The switch setting becomes *very* important, if the program contains even a *single* block containing the slash symbol. The active setting ON will cause all instructions in a block following the slash code to be ignored during program processing. The inactive setting OFF will cause the control to ignore the slash code and process all instructions written in the program block.

Block skip function set to ON position means
'Ignore all block instructions following the slash'

Block skip function set to OFF position means
'Process all block instructions'

In the *Example 1* listed earlier, the contents of blocks N4, N5 and N6 will be ignored, if the block skip function is ON. They will be processed, if the switch setting is OFF. The *Example 2*, also listed earlier, contains a slash in block N7. The slash symbol precedes miscellaneous function M08 (coolant ON). If the skip function switch is ON, the coolant will be ignored; if it is OFF, the coolant function will be effective. This application may be useful in a dry run mode, to bypass coolant flood during program verification, if no manual override is available.

Not all controls allow the slash code in any other block position, except as the first character in the block: /N..

BLOCK SKIP AND MODAL COMMANDS

To understand the way how modal values work with skipped blocks, recall that modal commands can be specified only once in a program, in the block where they occur first. Modal commands are not repeated in the subsequent blocks, as long as they remain unchanged.

In programs where the block skip function is not used at all, there is nothing to do. When the block skip function is used, watch carefully all *modal* commands. Remember that a command established in a block using the slash code will not always be in effect. It depends on the setting of block skip switch. Any modal command that has to be carried over from a section with slash codes to the section without slash codes may be lost if the block skip function is used. Overlooking modal commands when programming block skip function can result in a program with serious errors.

A very simple programming solution to avoid this potential problem is available. Just *repeat* all modal commands in the program section that will not be affected by the block skip function.

Compare the following two examples:

➡ Example A - Modal commands **are not** repeated :

```
N5 G00 X10.0 Y5.0 Z2.0
/ N6 G01 Z0.1 F30.0 M08
N7 Z-1.0 F12.0          (G01 AND M08 MISSING)
N8 ...
```

➡ Example B - Modal commands **are** repeated :

```
N5 G00 X10.0 Y5.0 Z2.0
/ N6 G01 Z0.1 F30.0 M08
N7 G01 Z-1.0 F12.0 M08
N8 ...
```

In both examples A and B, the program block containing slash code indicates intermediate Z-axis position as Z0.1. This position may be required only in certain cases during machining and the operator will decide whether to use it or not, and also when to use it.

The critical block, identified in the examples as N6, contains several modal functions. Commands G01, Z0.1, F30.0 and M08 will all remain in effect, unless they are canceled or changed in any following block. From block N7 it is apparent that only the Z-coordinate position and the cutting feedrate value have changed. However, the G01 and M08 commands are not repeated in the example A and will *not* be in effect, if the block skip switch is set ON.

Both examples A and B will produce identical results, but only if the block skip function is inactive (OFF). The control system will then execute the instructions in *all* blocks, in the order of programming sequence.

The processing result will be different for each programming example shown. If the block skip function is active (ON) - block instructions following the slash code will *not* be processed. The next example A yields an unacceptable result, with a fairly possible collision. Example B uses careful and thoughtful approach with very little extra work. These are the results when block N6 is skipped:

➡ Example A - Modal commands **are not** repeated :

```
N5 G00 X10.0 Y5.0 Z2.0          (RAPID MOTION)
N7 Z-1.0 F12.0                  (RAPID MOTION)
N8 ...
```

➡ Example B - Modal commands **are** repeated :

```
N5 G00 X10.0 Y5.0 Z2.0          (RAPID MOTION)
N7 G01 Z-1.0 F12.0 M08          (FEEDRATE MOTION)
N8 ...
```

Note that the linear motion G01, feedrate F30.0 and coolant M08 are all skipped in the example A. The X and Y axes have not been updated in either example and will remain unchanged. The conclusion is that example A will result in a Z-axis rapid motion in two consecutive blocks, causing a potentially dangerous situation. In the correct version, listed as example B, the programmed repetition of all commands - G01, F12.0 and M08 - assures the program will always be processed as intended. In the next section of this chapter, principles of program design for different practical applications will be presented.

In the summary, there is one basic rule for developing CNC programs with blocks using the block skip function:

Always program ***all*** instructions, even if it means repeating some program values and commands that have to be preserved

The slash symbol can be placed into the program *after* the program has been designed for *both* options. Just place the slash in those blocks that define the optional skip of all selected program blocks. *Always check program!*

Any CNC program containing block skip function should be checked at least twice - in ON and OFF mode

The result of this double check must be *always* satisfactory, whether processed with block skip in effect or without it. If an error is detected, *even a very minor error*, correct it first! After the correction, check the program at least *twice* again, covering both types of processing. Main reason for this special double check is that a correction made for one type of processing may cause (in some cases) a different error for the other type of processing.

PROGRAMMING EXAMPLES

Block skip function is very simple, often neglected, yet, it is a powerful programming tool. Many programs can benefit from a creative use of this feature. Type of work and some thinking ingenuity are the only criteria for its successful implementation. In the following examples, some practical applications of the block skip function are shown. Use these examples as start points for general program design or when covering similar machining applications.

◆ Variable Stock Removal

Removal of excessive stock material is typical during a rough cutting. When machining irregular shapes (castings, forgings, etc.) or rough facing on lathes, it may be difficult to determine the number of cuts. For example, some castings for a given job may have only the minimum excessive material, so one roughing or facing cut will be sufficient. Other castings for the same job may be larger and two roughing or facing cuts are needed.

If the program is designed in such a way that there is only *one* roughing or facing cut, problems may occur during machining of heavy stock. Programming *two* cuts for all parts produces a safer program, but will be inefficient for parts with a minimum stock. There will be too many tool motions known as 'cutting air', when the stock is minimal.

➡ Example - Variable stock face :

A face cutting of a stock that varies in size is a common problem in CNC work. A suitable solution is identical for turning and milling - the program should include tool motions for *two* cuts and the block skip function will be used on all blocks relating to the *first* cut.

Here is a lathe example of typical face cut, when the facing stock varies between 0.08 (2 mm) and 0.275 (7 mm). After considering several machining options, the programmer decides that the reasonable maximum stock that can be faced in a single cut will be 0.135 (3.5 mm) - Figure 23-1.

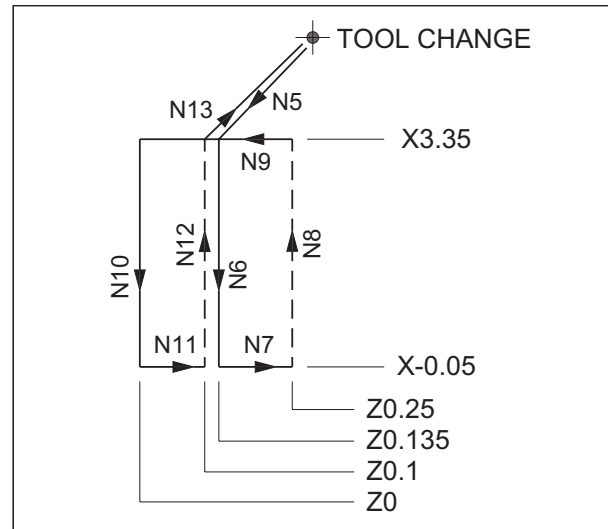


Figure 23-1

Variable stock for facing in a turning application - program O2301

```
O2301 (TURNING)
(VARIABLE FACE STOCK)
N1 G20 G40 G99
N2 G50 S2000
N3 G00 T0200 M42
N4 G96 S400 M03
N5 G41 X3.35 Z0.135 T0202 M08
/ N6 G01 X-0.05 F0.01
/ N7 G00 Z0.25
/ N8 X3.35
N9 G01 Z0 F0.05
N10 X-0.05 F0.01
N11 G00 Z0.1
N12 X3.5
N13 G40 X12.0 Z2.0 T0200
N14 M30
%
```

Block N5 contains the initial tool approach motion. The next three blocks are preceded by a slash. In N6, the tool cuts off the front face, at Z0.135; N7 moves the tool away from the face, block N8 is a rapid motion back to the initial diameter. There are no other blocks to be skipped after the block N8. N9 block contains a feedrate to the front face Z0, N10 is the front face cutting motion, N11 is the clearance motion, followed by standard final blocks.

Evaluate the example not once but at least *twice* - it shows what exactly happens. During the *first* evaluation, read *all* blocks, including those with slash. During the *second* time, ignore all blocks containing the slash code. There will be identical results when compared with the first evaluation. The only difference will be the number of actual cuts - *one*, not two. In milling, the procedure is very similar.

An example for a milling application uses a Ø5 inch face mill. The excessive material stock to be faced varies between 0.120 and 0.315. The largest reasonable depth of cut selected will be 0.177 (4.5 mm) - Figure 23-2.

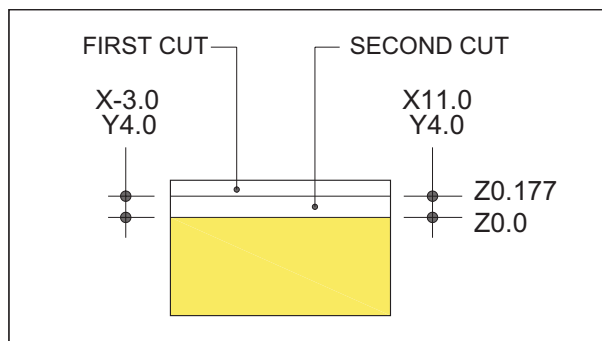


Figure 23-2

Variable stock for facing in a milling application - program O2302

```
O2302 (MILLING)
(VARIABLE FACE STOCK)
N1 G20
N2 G17 G40 G49 G80
N3 G90 G00 G54 X11.0 Y4.0
N4 G43 Z1.0 S550 M03 H01
N5 G01 Z0.177 F15.0 M08
/ N6 X-3.0 F18.0
/ N7 Z0.375
/ N8 G00 X11.0
N9 G01 Z0
N10 X-3.0 F18.0
N11 G00 Z1.0 M09
N12 G28 X-3.0 Y4.0 Z1.0
M13 M30
%
```

Block N5 in the example contains Z-axis approach to the first cut, at Z0.177 level. The next three blocks can be skipped if necessary. In N6 block, the face mill actually cuts at Z0.177 position, N7 is the tool clearance motion after the cut, and N8 returns the tool to the initial X-position. There are no other blocks to be skipped after block N8.

Block N9 does not need a feedrate for a good reason - it will be *either* F15.0 or F18.0, depending on whether blocks N6 to N8 were skipped or not. The feedrate is very important in block N10. Such repetition guarantees the required feedrate is in the critical block, when actual cutting takes place.

Both lathe and mill examples should offer at least some basic understanding of the *logic* used in program development, using the block skip function. Exactly the same logical approach can be used for more than two cuts and can also be applied to operations other than face cutting.

◆ Machining Pattern Change

Another application, where block skip function may be used efficiently, is a simple family programming. The term *family programming* means a programming situation where there may be a slight difference in design between two or more parts. Such a small variation between *similar* parts is often a good prospect for block skip function. A minor deviation in a machining pattern from one drawing to another can be adapted in a single program using the block skip function. Following two examples show typical possibilities of programming a *change of tool path*. In one example, the emphasis is on a skipped machining location. In the other, the emphasis is on the pattern change itself. Both examples are in metric and illustrate a simple grooving operation. For the lathe example, Figure 23-3 is related to program O2303.

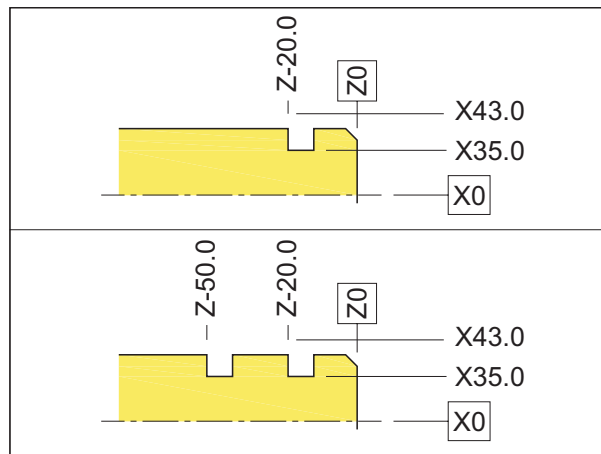


Figure 23-3

Variable machining pattern - turning application - O2303

The upper image shows machining result with block skip function set ON, the lower image shows machining result with block skip function set OFF, using the *same* program.

```
O2303 (LATHE EXAMPLE)
N1 G21
...
N12 G50 S1800
N13 G00 T0600 M42
N14 G96 S100 M03
```



```

N15 X43.0 Z-20.0 T0606 M08
N16 G01 X35.0 F0.13
N17 G00 X43.0
/ N18 Z-50.0
/ N19 G01 X35.0
/ N20 G00 X43.0
N21 X400.0 Z45.0 T0600 M01

```

Program O2303 demonstrates a single program for two parts with similar characteristics. One part requires a single groove, the other requires two grooves on the same diameter. In the example, both grooves are identical - they have the same width and depth and are machined with the same tool. The only difference between the two examples is the number of grooves and the second groove position. Machining the part will require block skip function set ON or OFF, depending on drawing requirements.

Evaluate the more important blocks in the program example. Block N15 is the initial tool motion to the start of the first groove at Z-20.0. In the next two blocks, N16 and N17, the groove will be cut and the tool returns to the clearance diameter. The following three blocks will cut the second groove, if it is required. That is the reason for block skip function. In block N18, tool moves to the initial position of groove 2 at Z-50.0, in block N19 the groove is cut. In block N20, tool retracts from the groove to a safe clearance position.

Milling example shown in *Figure 23-4*, also in metric, is represented in program O2304. This program handles two similar patterns that have four identical holes for both parts and two missing holes in the second part only. This is a good example of similar parts program, using block skip.

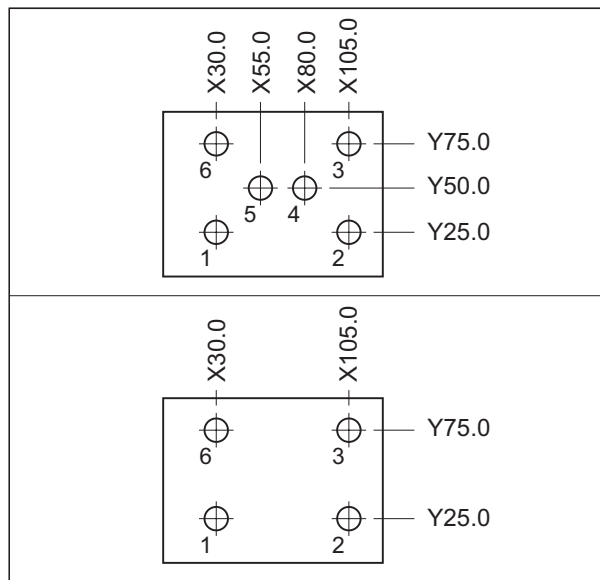


Figure 23-4

Program O2304 - variable machining pattern for a milling application - result with block skip OFF (top) and ON (bottom)

Both variations of program O2304 machine a hole pattern with either 6 or 4 holes. Block skip function has been used to make a single program covering *both* patterns. Top of *Figure 23-4* shows the hole pattern when block skip function is set OFF, the bottom shows the hole pattern when block skip mode is set ON.

O2304 (MILLING EXAMPLE)

```

N1 G21
...
N16 G90 G00 G54 X30.0 Y25.0 M08
N17 G43 Z25.0 S1200 M03 H04
N18 G99 G81 R2.5 Z-4.0 F100.0      (HOLE 1)
N19 X105.0                        (HOLE 2)
N20 Y75.0                          (HOLE 3)
/ N21 X80.0 Y50.0                  (HOLE 4)
/ N22 X55.0                        (HOLE 5)
N23 G98 X30.0 Y75.0                (HOLE 6)
N24 G80 G28 X30.0 Y75.0 Z25.0
N25 M01

```

Blocks N18 to N20 will drill holes 1, 2 and 3. Hole 4 in N21 and hole 5 in N22 will be drilled only if the block skip function is set to inactive mode (OFF), but neither one will be drilled when block skip setting is active (ON). Block N23 will always drill hole number 6.

A variation of this application is program O2305. There are *five* hole positions, but the block skip function is used *within* a block, to control only the Y-position of the hole. Top of *Figure 23-5* shows the hole pattern when block skip function is set OFF, the bottom shows the hole pattern when block skip mode is set ON. The middle hole will have a different Y-axis position, depending on the setting of the block skip function during machining.

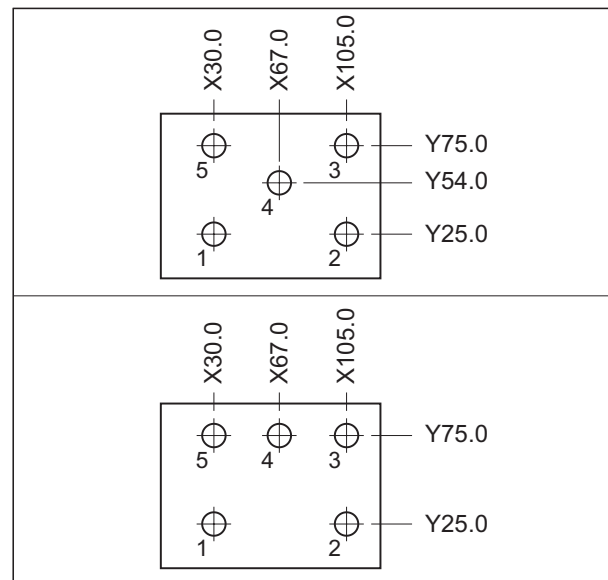


Figure 23-5

Program O2305 - variable machining pattern for a milling application - result with block skip OFF (top) and ON (bottom)

O2305 (MILLING EXAMPLE)

N1 G21

...

N16 G90 G00 G54 X30.0 Y25.0 M08

N17 G43 Z25.0 S1200 M03 H04

N18 G99 G81 R2.5 Z-4.0 F100.0

N19 X105.0

N20 Y75.0

N21 X67.0 / Y54.0

N22 G98 X30.0 Y75.0

N23 G80 G28 X30.0 Y75.0 Z25.0

N24 M01

(HOLE 1)
(HOLE 2)
(HOLE 3)
(==> HOLE 4)
(HOLE 5)

Hole 4 in block N21 will be drilled at the location of X67.0Y75.0, if the block skip mode is ON. The address Y54.0 in block N21 will not be processed. If the block skip mode is OFF, hole 4 will be drilled at coordinate location of X67.0Y54.0. In that case, the Y75.0 position from block N20 will be overridden. In order to guarantee proper drilling at position 5, the coordinate Y75.0 in block N22 must be written. If it is omitted, Y54.0 from block N22 will take precedence in block skip OFF mode.

Using block skip feature is the simplest way of designing a family of similar parts. These applications are a bit limited with block skip function alone, but they offer the fundamentals of a powerful programming technique and an example of logical thinking. Many detailed explanations and examples of programming complex families of parts can be found in a special *Custom Macro* option Fanuc offers on most control systems.

◆ Trial Cut for Measuring

Another useful application of block skip is to provide the machine operator with means of measuring the part *before* any final machining has been done. Due to various dimensional imperfections of a cutting tool combined with other factors, the completed part may be slightly outside of the required tolerance range.

The following method of programming is very useful for programming parts requiring very close tolerances. It is also a useful method for those parts, where the part shape is difficult to measure after all machining is completed, for example conical shapes, such as tapers. The same method is also quite useful for parts where the cycle time of an individual tool is relatively long and all tool offsets have to be fine tuned *before* production machining.

This approach to part programming is more efficient, as it eliminates a recut, increases surface finish, and can even prevent a scrap. In either case, *trial cut* programming method that employs the block skip function is used. Setting the block skip mode OFF, machine operator checks the trial dimension, adjusts the suitable offset, if necessary, and continues machining with block skip set ON.

All general concepts described in example O2306 are equally applicable to turning and milling - *Figure 23-6*.

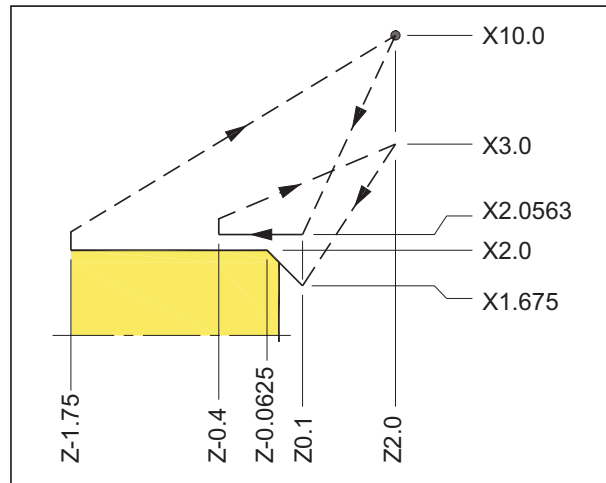


Figure 23-6

Application of a trial cut for measuring on a lathe - program O2306

O2306

(TRIAL CUT - LATHE)

N1 G20

...

N10 G50 S1400

N11 G00 T0600 M43

N12 G96 S600 M03

/ N13 G42 X2.0563 Z0.1 T0606 M08

/ N14 G01 Z-0.4 F0.008

/ N15 X2.3 F0.03

/ N16 G00 G40 X3.0 Z2.0 T0600 M00

/ (TRIAL DIA IS 2.0563 INCHES)

/ N17 G96 S600 M03

N18 G00 G42 X1.675 Z0.1 T0606 M08

N19 G01 X2.0 Z-0.0625 F0.007

N20 Z-1.75

N21 X3.5 F0.01

N22 G00 G40 X10.0 Z2.0 T0600

N23 M01

When program O2306 is processed with block skip set OFF, all blocks will be executed, including the trial cut and finish profile. With block skip set ON, the only operation executed will be finishing to size, *without* the trial cut. In this case, all significant instructions are retained by repetition of the key commands (block N18). Such a repetition is very crucial for successful processing in both modes of the block skip function. M00 function in N16 always stops the machine and enables a dimensional check.

Selecting trial diameter of 2.0563 in the example may be questioned. What is the logic for it? The trial diameter can be other reasonable size, say 2.05. That would leave 0.025 stock per side for the finish cut. It is true that a different diameter *could* have been selected. The four decimal number was only selected for one reason - *to psychologically* encourage the operator to maintain accurate offset settings. Feel free to disagree - programmers may prefer a three or even two decimal number instead - the options are open.

In the next example, another trial cut will also be programmed before actual machining, but for a much different reason - *Figure 23-7*.

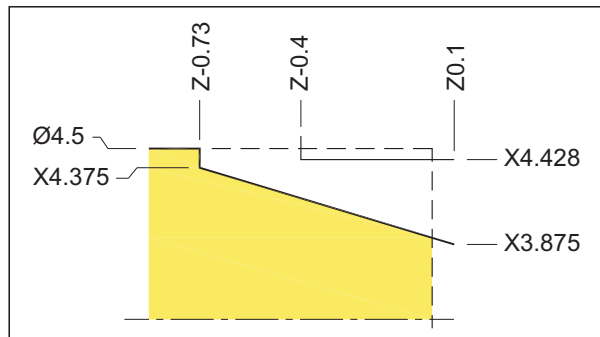


Figure 23-7

Trial cut for taper cutting on a lathe - program O2307

In program O2307, the part finished shape is a taper, a feature difficult to measure when completed. Adjusting the tool offset by trial and error is not the right solution. Programming a trial cut within the area of solid material, *along a straight diameter*, enables the operator to check the trial dimension comfortably and to adjust offset *before* cutting the finished taper.

```
O2307
(TRIAL CUT FOR TAPER - ONE TOOL)
N1 G20 G99 G40
N2 G50 S1750 T0200 M42
N3 G96 S500 M03
/ N4 G00 G42 X4.428 Z0.1 T0202 M08
/ N5 G01 Z-0.4 F0.008
/ N6 U0.2 F0.03
/ N7 G00 G40 X10.0 Z5.0 T0200 M00
/ (TRIAL CUT DIA IS 4.428 INCHES)

/ N8 G96 S500 M03
N9 G00 G42 X4.6 Z0.1 T0202 M08
N10 G71 U0.15 R0.03
N11 G71 P12 Q14 U0.06 W0.005 F0.01
N12 G00 X3.875
N13 G01 X4.375 Z-0.73 F0.008
N14 X4.6 F0.012
N15 S550 M43
N16 G70 P12 Q14
N17 G00 G40 X10.0 Z5.0 T0200 M01
```

Program O2307 illustrates a common situation, where a single cutting tool is used for both roughing *and* finishing operations. It shows a logical way of using the block skip function, in a simple form. In most applications, separate tools for roughing and finishing may be needed, depending on the degree of required accuracy. When using two cutting tools, the trial cut dimension is usually more important for the *finishing* tool than for the roughing tool. In program O2308, the block skip function is illustrated using two cutting tools - T02 is for roughing, T04 is for finishing. Previous example in *Figure 23-7* is used.

```
O2308
(TRIAL CUT FOR TAPER - TWO TOOLS)
N1 G20 G99 G40
N2 G50 S1750 T0200 M42
N3 G96 S500 M03
/ N4 G00 G42 X4.46 Z0.1 T0202 M08
/ N5 G01 Z-0.4 F0.008
/ N6 U0.2 F0.03
/ N7 G00 G40 X10.0 Z5.0 T0200 M00
/ (T02 TRIAL CUT DIA IS 4.46 INCHES)

/ N8 G50 S1750 T0400 M43
/ N9 G96 S550 M03
/ N10 G00 G42 X4.428 Z0.1 T0404 M08
/ N11 G01 Z-0.4 F0.008
/ N12 U0.2 F0.03
/ N13 G00 G40 X10.0 Z5.0 T0400 M00
/ (T04 TRIAL CUT DIA IS 4.428 INCHES)

/ N14 G50 S1750 T0200 M42
/ N15 G96 S500 M03
N16 G00 G42 X4.6 Z0.1 T0202 M08
N17 G71 U0.15 R0.03
N18 G71 P19 Q21 U0.06 W0.005 F0.01
N19 G00 X3.875
N20 G01 X4.375 Z-0.73 F0.008
N21 X4.6 F0.012
N22 G00 G40 X10.0 Z5.0 T0200 M01

N23 G50 S1750 T0400 M43
N24 G96 S550 M03
N25 G00 G42 X122.0 Z3.0 T0404 M08
N26 G70 P19 Q21
N27 G00 G40 X10.0 Z5.0 T0400 M09
N28 M30
%
```

Example O2308 can be improved further by including the control of taper on the width, for example. Programming a trial cut is useful but often a neglected technique, although it does present many possible applications.

◆ Program Proving

Block skip function can also be useful to prove a new program on the machine, to check it against obvious errors. Operators with limited experience may be uneasy to run a program for the first time. One of their most common concerns is the initial rapid motion towards a part, particularly when clearances are small. Rapid motion rate of many modern CNC machines can be very high, well over 2000 in/min. At such high speeds, rapid approach to the cutting position on the part may not add to operator's confidence, particularly when the approach is programmed too close to material. On most controls, CNC operator can set the rapid override rate to 100%, 50%, 25% and slower. On older controls, the rapid rate override cannot be done.

The next two examples, O2309 and O2310, show a typical programming method to eliminate the problem during setup and program proving, yet maintain full rapid motion rate during repeated operations for productivity.

Block skip function in these examples takes a less usual role - it is used for a *section* of a block, rather than the whole block itself, *if the control supports such a method*.

O2309 (TURNING EXAMPLE)

```
N1 G20 G40 G99
N2 G50 S2000
N3 G00 T0200 M42
N4 G96 S400 M03
N5 G41 X2.75 Z0 T0202 M08 / G01 F0.1
N6 G01 X.. F0.004
N7 ...
```

O2310 (MILLING EXAMPLE)

```
N1 G20 G17 G40 G80
N2 G90 G00 G54 X219.0 Y75.0 M08
N3 G43 Z-1.0 S600 M03 H01 / G01 F30.0
N4 G01 X.. F12.0
N5 ...
```

Both examples show block skip used *in* a single block. Design of both programs takes advantage of *two conflicting commands within the same block*. If two conflicting commands exist in a single block, the *latter* command used in the block will become effective (exceptions exist).

In both examples, the *first* command is G00, the *second* one is G01. Normally, the G01 motion will take a priority. Because of the slash function, the control will accept G00, if block skip is set ON, but it will accept G01, if block skip is set OFF. When the block skip mode is OFF, both motion commands will be processed and the *second* command in that block becomes effective (G01 overrides G00). Watch for one drawback, already emphasized:

Block skip **within** a block may not work with all controls

During the first machine run, operator should set block skip OFF, making G01 command effective. The tool motion will be slower than in rapid mode, but much safer. Also, feedrate override switch of the control system will become effective, offering additional flexibility.

When program proving is completed and the safe tool approach is confirmed, block skip can be set ON, to prevent the G01 motion from being processed. Both O2309 and O2310 are typical examples of breaking with tradition to achieve a specific result.

◆ Barfeeder Application

On a CNC lathe, block skip function can be used in barfeeding, for continuous machining. If the barfeeder allows it, the techniques is quite simple. Typical program will actually have *two ends* - one will use the M99 function, the other end will use the M30 function. M99 block will be preceded by the block skip function [/] and will be placed *before* the M30 code in the part program. This special technique is detailed on page 430.

◆ Numbered Block Skip

For most machining, block skip function is set to either the ON or OFF position and remains in this mode for the *whole* program. If the ON setting is required for one section of the program, but not for another, the operator has to be informed, usually in program comments. This practice of changing block skip mode in the middle of a program can be unsafe and possibly create major problems.

An optional feature on some controls is a *selective* or a *numbered* block skip function. This option allows the operator to select which portions of the program require the ON setting and which portions require the OFF setting. Setting can be done *before* pressing the *Cycle Start* key to initialize program processing. This method also uses the slash symbol, but followed by an integer, within the range of 1 to 9. Actual selection of the mode is done on the control screen (*Settings*), under the matching skip number.

For example, a program may contain three groups, each expecting a different setting of the skip function. By using the switch number after the slash symbol, all groups are clearly defined and all the operator must do is to *match* the control settings with the required activity.

```
N1 ...
N2 ..
/1 N3 ... (BLOCK SKIP GROUP 1)
/1 N4 ... (BLOCK SKIP GROUP 1)
...
...
N16 ...
/2 N17 ... (BLOCK SKIP GROUP 2)
/2 N18 ... (BLOCK SKIP GROUP 2)
/2 N19 ... (BLOCK SKIP GROUP 2)
...
...
N29 ...
/3 N30 ... (BLOCK SKIP GROUP 3)
/3 N31 ... (BLOCK SKIP GROUP 3)
...
...
N45 ...
```

Identical rules apply for selective block skip function as for the normal version. Incidentally, the /1 selection is the same as a plain slash only, so blocks N3 and N4 above, could have also been written this way:

```
/ N3 ...
/ N4 ...
```

Numbered block skip function is not available on all controls

Programs using selective block skip function can be very clever and even efficient, but they may place quite a burden on the machine operator. For the majority of jobs, there will be a plenty of programming power available by using the standard block skip function.

24

DWELL COMMAND

Dwell is a technical name for a *pause* in CNC programs - it is an intentional time delay applied during program processing, at the machine. In this time period - specified in the program - any axis motion is stopped, while all other program commands and functions remain unchanged (function normally). When the specified time expires, control system resumes processing of program block immediately *following* the dwell command.

PROGRAMMING APPLICATIONS

Programming a dwell is very easy and can be quite useful in two main applications:

- *During actual cutting,*
when the tool is in contact with material
- *For operation of machine accessories,*
when no cutting takes place

Each application is equally important to programmers, although the two are not used simultaneously.

◆ Applications for Cutting

When cutting tool is removing material stock, it is in contact with the machined part. A dwell can be applied during machining for a number of reasons. If the spindle is running, the spindle speed (r/min) *is* very important.

In practice, the application of dwell during a cut may be used for breaking chips while drilling, grooving, counter-sinking, counterboring, or parting-off. Dwell may also be used while turning or boring, in order to eliminate any physical marks left on the part by end thrust of the cutting tool. This thrust is the result attributed to various tool pressures during cutting. In many other applications, the dwell function is useful to control cutting feed deceleration on a corner during very fast feedrates, for example. This use of dwell could be particularly useful for older control systems with a possible backlash problem. In both cases, specified dwell command 'forces' the machining operation to be *fully completed* in one block, *before* the next block can be executed. CNC programmer still has to supply the exact length of time required for the pause duration. This time has to be sufficient - neither too short nor too long.

Dwell command is always completed
before the next operation begins

◆ Applications for Accessories

The second common application of dwell is after certain miscellaneous functions - M functions. Several such functions are used to control a variety of CNC machine accessories, such as barfeeder, tailstock, quill, part catcher, custom features, and many others. Programmed dwell time must allow for the full completion of a certain procedure, such as the operation of a tailstock. During such a procedure, the machine spindle may be either stationary or rotating. Since there will be no contact of the cutting tool with part material in this situation, it is *not* important whether the machine spindle actually rotates or not.

On some CNC machines, the dwell command may also be needed when changing spindle speed, usually after gear range change. This applies mainly to CNC lathes. In these cases, the best advice as to *how* and *when* to program dwell time is to follow the CNC machine manufacturer's recommendation. Typical examples of a dwell used for lathe accessories are described in *Chapter 44*, covering the subject.

DWELL COMMAND

G04	Dwell command
-----	---------------

Standard preparatory command for dwell is G04. Like other G commands, G04 used by itself only will do nothing. It must always be used with another address, in this case, specifying the *length of time* to dwell (pause). Three addresses can be used for dwell - they are X, P or U (address U can only be used for CNC lathes). Actual time duration specified by the selected address is either in *milliseconds*, or in *seconds*, depending on which address was used. Some control systems may use a different address for programming dwell, but its main purpose as well as programming methods remain identical.

Several fixed cycles for CNC machining centers also use dwell. This dwell is programmed together with the cycle data, *not* in a separate block (G04 is not used). Only fixed cycles that require a dwell time can use it in the same block. For all other applications, the dwell command must be programmed as an *independent block*. Dwell will always remain active for one block only and does not carry over to the next block. Dwell is a one-block function - it is *not* modal. During dwell execution, there is no change in status of program processing, only the overall cycle time will be affected.

◆ Dwell Command Structure

The structure - or format - for the dwell time is:

X5.3 ... All machines, excluding fixed cycles
U5.3 ... Lathes only
P53 ... All machines, including fixed cycles

In any case, the typical representation is five digits *before* and three digits *after* the decimal point, although that may vary on different control systems.

Since milliseconds or even seconds can be used as units of dwell, their relationship can be established:

1s = 1000ms
1ms = 0.001s

☞ **where ...**

s = second
 ms = millisecond

Examples of practical application of the dwell format are:

G04 X2.0 ... preferred for long dwells - in seconds
G04 P2000 ... preferred for short or medium dwells - in ms
G04 U2.0 ... lathe only - in seconds

In these examples, the dwell is 2 seconds or 2000 milliseconds. All three formats are shown with identical results. The next example is similar:

G04 X0.5
G04 P500
G04 U0.5

This example illustrates a dwell of 500 milliseconds, or one half of a second. Again, all three formats are shown.

In a CNC program, the dwell function may be used in the following way - note the dwell is programmed in a separate block, between two motions:

N21 G01 Z-1.5 F12.0
N22 G04 X0.3 (DWELL COMMAND 0.3 SEC)
N23 Z-2.7 F8.0

Programs using X or U addresses may cause a possible confusion, particularly to new programmers. The X and U addresses may incorrectly be interpreted as an axis motion. *This will never be the case.* By definition, the X axis and its lathe application, the U axis, is the *dwelling* axis.

X axis is the only axis common to all CNC machines

No axis motion will take place when the X, P or U address is used with the dwell command G04

The control unit interprets commands X or U as dwell, *not as axis motion*. This is because of presence of the preparatory command G04, which establishes *meaning* of the address that follows it. If using the X or U address for dwell does not feel comfortable, use a third alternative - the address P. Keep in mind, that the P-address does *not* accept decimal point, so any dwell is programmed directly as the *number of milliseconds* in order to control the elapsed time. One millisecond is 1/1000th of a second, therefore one second is equivalent to 1000 milliseconds.

Both addresses X and U can also be programmed in milliseconds with G04, without a decimal point - for example,

G04 X2.0 is equal to **G04 X2000**

Leading zero suppression is assumed in the format without decimal point (trailing zeros are always required):

P1 = **P0001** ... 1 millisecond = 0.001 second
P10 = **P0010** ... 10 milliseconds = 0.01 second
P100 = **P0100** ... 100 milliseconds = 0.1 second
P1000 = **P1000** ... 1000 milliseconds = 1.0 second

While minimum dwell is more important than maximum dwell, each control has a limit for maximum dwell duration. For the format using five digits in front of a decimal point and three digits following it, the dwell range is between 0.001 and 99999.999 seconds. That is equivalent to a dwell range from the minimum of 1/1000th of a second, up to 27 hours, 46 minutes and 39.999 seconds.

Dwell programming applications using the X and P addresses are identical to both machining centers and lathes, but the U address can only be used in lathe programs. The selection of either metric or imperial dimensional units has no effect on the dwell function whatsoever, as time is not dimensional.

DWELL TIME SELECTION

Seldom ever the dwell time will exceed more than just a few seconds, most often much less than one second. Dwell is always a *nonproductive time* and it should be selected as the shortest time needed to accomplish the required action. Any time delay for completion of a particular machine operation or a special machine accessory is usually recommended by the machine manufacturer. Selecting dwell time for cutting purposes is always programmer's responsibility. Unfortunately, some programmers often specify the dwell time too high. After all, one second seems like a very short time, but think about a typical example:

In one program block, a dwell function is assigned for the duration of one second. The spindle speed is set to 480 r/min and the dwell is applied at 50 part locations, perhaps during a spot face operation. That means the cycle time for *each* part is 50 seconds longer *with* the dwell, then it would

be *without* the dwell. Fifty seconds may not seem too unreasonable, but are these precious seconds necessary? Give it a little thought or - even better - calculate it. If dwell must be used at all, make sure to calculate the *minimum* dwell that can do the job. It is easy to select dwell arbitrarily, by guessing and without much thinking. In the example, the minimum dwell required is only 0.125 seconds (!):

$$60 / 480 = 0.125$$

This minimum dwell is *eight times shorter* than the one second dwell programmed initially. If minimum dwell is programmed rather than the estimated dwell, the cycle time will increase by only 6.25 seconds, rather than the original 50 seconds - a significant improvement in programming efficiency and productivity improvement at the machine.

Minimum dwell calculation and other issues related to it are described shortly.

SETTING MODE AND DWELL

Most programs for machining centers will use feedrate per time (programmed in millimeters per minute - *mm/min* - or inches per minute - *in/min*). Lathe applications are normally programmed as feedrate per one spindle revolution, in millimeters per revolution - *mm/rev* - or inches per revolution - *in/rev*. On many Fanuc controls, a parameter setting allows programming a dwell in either as elapsed time in seconds or milliseconds - or the *number of spindle revolutions*. Each application has its practical uses and benefits. Depending on system parameter setting, dwell command will assume an incompatible meaning with each setting:

◆ Time Setting

This is the most common and practical default setting for virtually all CNC machines. For the *elapsed time* setting, the dwell is always programmed in seconds or milliseconds, within the range allowed by the control unit, for example, from 0.001 to 99999.999 seconds, a typical range for Fanuc and many similar control systems:

G04 P1000

... represents the dwell of one second, equivalent to 1000 milliseconds.

◆ Number of Revolutions Setting

For the number of *spindle revolutions* setting, the dwell is expressed as the number of times a spindle rotates, within the range of 0.001 to 99999.999 *revolutions*, for example:

G04 P1000

... represents the dwell duration of one *revolution* of the spindle (P1000 = 1.000).

MINIMUM DWELL

During a cut, that is for operations where cutting tool is in contact with the machined part, minimum dwell definition is important, but the setting mode is unimportant (time or number of revolutions).

Minimum dwell is the time required to complete one revolution of the spindle

Minimum dwell, programmed in seconds, can be calculated, using a simple formula:

$$\text{Minimum dwell (sec)} = \frac{60}{r / \text{min}}$$

➡ Example :

To calculate minimum dwell in seconds for spindle rotation of 420 r/min, divide the *r/min* into sixty (there are 60 seconds in one minute):

$$60 / 420 = 0.143 \text{ seconds dwell}$$

Format selection of dwell block in the program will vary, depending on the machine type used and a particular programming style. All following examples represent the same dwell time of 0.143 of a second:

G04 X0.143
G04 P143
G04 U0.143

Regardless which format is used, all dwell times in the example specify 143 milliseconds, which is 0.143 of a second. Dwell in *milliseconds* can be calculated directly:

$$\text{Minimum dwell (ms)} = \frac{60 \times 1000}{r / \text{min}} = \frac{60000}{r / \text{min}}$$

It is allowed to mix both formats in a program, but such a practice does not represent consistent programming style.

◆ Practical Considerations

For *practical* dwell applications, the calculated minimum dwell is only *theoretically* correct and may *not* be the most practical to use. It is always better to round off the calculated value of minimum dwell *upwards* to get *more* than one spindle revolution. For example,

G04 X0.143 may become G04 X0.2

If double time is used - then G04 X0.143 will become G04 X0.286, or even G04 X0.3 to round off the time.

The reasoning for such an adjustment takes into consideration machining realities. It is common that machine operator may be running a certain job with spindle speed in an override mode, perhaps even set at its lowest setting, at 50%. Since 50% spindle speed override is the usual minimum setting on most CNC controls, the *double* minimum dwell will *always guarantee at least one complete revolution*, without any loss of production time.

Is there a magic dwell that can be used for most jobs? Not really, but you will find that most dwells will be very short, usually within the range of 0.2 and 0.5 seconds. A reversal of the previous formula calculates r/min for a given dwell:

$$r / \text{min} = \frac{60}{\text{Dwell}_{\text{sec}}}$$

For example, if programmed dwell is 0.25 of a second is suitable for rather slow speed of 240 r/min. In fact, the higher the spindle speed, the shorter dwell is required.

Another example is based on a particular machining operation, for example, a spot drill. Let's say, a spot drill is usually programmed within the spindle speed range of 600 to 1500 r/min, depending on stock material. The double minimum dwell for 600 r/min is 200 ms, while for 1500 r/min it is only 80 ms. The practical benefit is that if you program 250 ms for *any* spindle speed below 1500 r/min, the double minimum dwell will always be guaranteed. It is a compromise, but for a few holes it may be the fastest way to program dwell for spot drilling or similar operations.

NUMBER OF REVOLUTIONS

In the other dwell mode (selected by a system parameter), the programming format will only *appear* to be the same, but its meaning will be much different. In some applications, it may be desirable to program a delay for a certain *number of spindle revolutions*, rather than *elapsed time*.

In lathe grooving, when the tool reaches groove bottom, dwell time may be required to clean up the groove bottom diameter. Calculating the time in seconds (see below) would solve the problem, but an alternative approach could be more attractive. Many controls offer programming the required number of spindle revolutions directly, providing a system parameter is set. For example, to dwell as long as it takes to complete *three spindle revolutions* can be programmed directly, regardless of the spindle r/min.

It is normal in CNC lathe programming to use *cutting speed* rather than *spindle speed* in grooving. This situation presents a small problem - the actual spindle speed is *not* known. Yes, it can be calculated, but the effort may not be worth it. Using number of revolutions will solve the problem, *even if the cutting speed is changed (optimized)*.

◆ System Setting

If the control system is set to accept dwell as the *number of spindle revolutions*, rather than as elapsed time in seconds or milliseconds, the programming is very direct. All that is needed is to call the dwell command G04, followed by the number of required revolutions:

G04 X3.0 ... three spindle revolutions - direct input
G04 P3000 ... three spindle revolutions - direct input
G04 U3.0 ... three spindle revolutions - direct input

Each format represents the same result - a dwell lasting three spindle revolutions. Can we tell from the program whether the value means *time* or *revolutions*? We cannot.

Control settings have to be known for dwell

We *have to know* the control settings. One clue may be somewhat large dwell input - three revolutions are usually much shorter than three seconds of dwell. Note that the decimal accuracy is still written or implied, to allow fractions of a revolution, such as one half or one quarter.

◆ Time Equivalent

The two dwell modes cannot be mixed in one program, and even between programs, such a mix is not practical. CNC machine system parameter can be set to only one dwell mode at a time. Since control parameters are normally set for dwell as elapsed time, rather than spindle revolutions, the equivalent time must be calculated. The spindle speed (*r/min*) must always be known in such case.

Equivalent dwell time in seconds can be calculated to be equal to the required number of spindle revolutions.

Use the following formula:

$$\text{Dwell}_{\text{sec}} = \frac{60 \times n}{r / \text{min}}$$

☞ where ...

60 = Number of minutes (conversion factor)
n = Required number of spindle revolutions
r/min = Current spindle speed (revolutions per minute)

➡ Example :

To calculate dwell time in seconds for *full three* spindle revolutions, at spindle speed of 420 r/min, the above formula can be applied:

$$\text{Dwell}_{\text{sec}} = 60 \times 3 / 420 = 0.429$$

Program block representing the required three spindle revolutions in terms of dwell *time* will take one of the following forms:

G04 X0.429 ... three spindle revolutions - time equivalent
 G04 P429 ... three spindle revolutions - time equivalent
 G04 U0.429 ... three spindle revolutions - time equivalent

It may also be a good idea to work backwards and calculate the *equivalent of dwell time*, represented as the number of spindle revolutions. Usually, the result will not be an integer number and will require rounding to the nearest value upwards. The above formula can be easily modified to:

$$\text{Dwell}_{\text{rev}} = \frac{\text{r / min} \times \text{Dwell}_{\text{sec}}}{60}$$

➡ Example :

To confirm that the formula is correct, use the value of 0.429 from the previous example and calculate the number of revolutions for a dwell of 0.429 seconds at 420 r/min:

$$\text{Dwell}_{\text{rev}} = 420 \times 0.429 / 60 = 3.003 \text{ revolutions}$$

Result confirms the formula is correct. It is quite likely that the calculation will start with a dwell that is already rounded, for example, to one half of a second:

$$\text{Dwell}_{\text{rev}} = 420 \times 0.5 / 60 = 3.5 \text{ revolutions}$$

The dwell time based on a required number of revolutions is mostly used for CNC lathe applications, especially when cutting with very slow spindle speeds. A slow spindle rotation does not have the latitude of faster speeds and does not allow for a large error in dwell calculation. Keep in mind that the goal is to get *at least one complete* part rotation in order to achieve desired machining results. Otherwise, why program dwell at all? Consider another example:

Dwell is programmed for *one half of a second* duration, with spindle rotation set to low 80 r/min. The number of revolutions for one half of a second will be:

$$80 \times 0.5 / 60 = 0.6666667$$

which is *less* than one complete spindle revolution. The reason for programming the dwell function in the first place is not honored here and the dwell time has to be increased. Dwell of 0.5 seconds is therefore not sufficient. The minimum dwell has to be *calculated*, using the formula presented earlier:

$$60 \times 1 / 80 = 0.75 \text{ seconds}$$

Generally, there is not much use for this type of calculations - most programming assignments can be handled very well with the standard *dwell per time* calculations.

LONG DWELL TIME

For general CNC machining purposes, an unusually long dwell time is neither required nor it is necessary. Does that mean long dwell times are not needed at all?

A long dwell time is elapsed time that is well *above the established average* for most normal applications. Seldom ever there is a need to program dwell time during a part machining in excess of one, two, three, or four seconds. The large range available on the control system (*over 27 hours*) is more important to the *maintenance personnel*, than to CNC programmers. As an example of typical application when an unusually long dwell *may* be beneficial, is a program developed by the maintenance technicians for testing the spindle functionality.

Consider carefully the following actual situation common to machine service - a spindle of a CNC machine has been repaired and must be tested before the machine can be released back to production. Testing will include running the spindle at various speeds, for a certain period of time of each speed selection.

In a typical example, the maintenance department requires a small CNC program, in which machine spindle will rotate for 10 minutes at 100 r/min, then for another 20 minutes at 500 r/min, followed by the spindle rotation at the highest rate of 1500 r/min for additional 30 minutes. Program development is not an absolute necessity, since the maintenance technician may do such a test by manual methods. The manual approach will not be very efficient but it will still serve the purpose of the maintenance test.

A better choice in these cases is to *store* the testing procedure as a small program, directly into CNC memory. The maintenance (service) program will be a little different for machining centers than for lathes but all main objectives will remain identical.

➡ Example - Machining Centers - Spindle test :

```
S100 M03          (100 R/MIN INITIAL SPEED)
G04 X600.0        (600 SECONDS IS 10 MINUTES)
S500              (SPEED INCREASED TO 500 R/MIN)
G04 X1200.0       (1200 SECONDS IS 20 MINUTES)
S1500             (SPEED INCREASED TO 1500 R/MIN)
G04 X1800.0       (1800 SECONDS IS 30 MINUTES)
M05               (SPINDLE STOP)
```

For machining centers, the program starts with 100 r/min spindle speed. This selection is followed by a 600 second dwell (10 minutes). Spindle speed is then increased to 500 r/min and dwell time to 1200 seconds (another 20 minutes). The last speed is 1500 r/min running for 1800 seconds, or another 30 minutes, before the spindle finally stops.

Note that in cases of especially long dwell, using *seconds* rather than *milliseconds* is more practical.

Example - Lathes - Spindle test :

```

M43                      (GEAR RANGE SELECTION)
G97 S100 M03             (100 R/MIN INITIAL SPEED)
G04 X600.0               (600 SECONDS IS 10 MINUTES)
S500                     (SPEED INCREASED TO 500 R/MIN)
G04 X1200.0              (1200 SECONDS IS 20 MINUTES)
S1500                    (SPEED INCREASED TO 1500 R/MIN)
G04 X1800.0              (1800 SECONDS IS 30 MINUTES)
M05                      (SPINDLE STOP)

```

This lathe example is similar to the previous one. Initial spindle speed setting includes gear range selection, for example, M43. Spindle rotation has been set to 100 r/min. Dwell of 600 seconds follows, leaving the spindle rotating for full 10 minutes. Then the speed is increased to 500 r/min and remains that way for another 20 minutes (1200 seconds). Before the spindle is stopped, one more change is done - the spindle speed increases to 1500 r/min and remains at that speed for another 30 minutes (1800 seconds).

Long dwells are for special applications only, and should never be used for other purposes.

Maintain all safety rules when using long dwell times !

Machine Warm-Up

Another program (typically a subprogram) that uses a long dwell time is favored by many CNC programmers and CNC machine operators, to 'warm-up' the machine before running a critical job. This machine warming activity takes place typically at the start of a morning shift during winter months or in a cold shop. This programming approach allows CNC machine to reach a certain ambient temperature before any precision components are machined. The same approach can also be used to gradually reach the maximum spindle speed for high-speed machining (8000 r/min and up). As always, all safety considerations must have a high priority in these cases.

X Axis is the Dwelling Axis

The control display screen shows how much time is still left before the dwell time expires. This can be viewed by looking at the X display of the *Distance-To-Go* indicator on the POS (position) screen of a typical Fanuc control. This display will always be displayed as X, since X axis is the only dwelling axis, regardless of control system, even if addresses P or U are programmed. Why the X axis has been selected as the dwelling axis and not any other axis? There is a simple reason - because *the X axis is the only axis common* to all CNC machine tools - i.e., drilling machines, mills, machining centers, flame cutters, waterjets, lasers, and so on. They all use XY or XYZ axes. Two-axis lathes use XZ axes (there is no Y axis) and wire EDM uses XY axes (there is no Z axis). Other machines are similar.

Safety and Dwell

A few safety reminders have been mentioned already - always exercise caution when using long dwells, particularly for service or maintenance purposes. No CNC machine should *ever* be left completely unattended - after all, a test can fail. For long testing times, proper warning signs should be prominently posted at the machine. If signs are not practical, personal supervision will be necessary.

One more note - the dwell function should never be programmed for the purpose of allowing the machine operator time to perform short manual operation while program is running. Manual jobs such as polishing, filing, deburring, part reversal, tool or insert change, chip removal, inspection, lubrication, etc., must always be done - if absolutely necessary during program execution - as a manual operation, *never under the program control!*

Never use dwell to perform manual operations on the machine !

FIXED CYCLES AND DWELL

Chapter 25 covers fixed cycles for CNC machining centers in significant detail. Descriptions of all cycles can be found there. For this chapter, here are some comments as how dwell is programmed with fixed cycles. Several fixed cycles can be programmed with a dwell:

- Normally, cycles G76, G82, G88, G89
- Also cycles G74 and G84, only by parameter setting

Dwell address in fixed cycles is always P, to avoid duplication of X address in the same block. Neither address U nor command G04 are *ever* programmed in fixed cycles - dwell function is 'built' into all cycles that allow it (technically all cycles do). Rules for calculating dwell time are the same for cycles, as for any other machining application.

Example (spot drilling) :

```
N9 G82 X30.0 Y16.0 R2.0 Z-3.75 P300 F150.0
```

In the example, dwell of 300 milliseconds - 0.3 seconds - is specified by the address P. Dwell will become effective upon completion of motion along the Z axis (actual cutting motion), but *before* the rapid return motion.

If a G04 P. is programmed as a separate block in fixed cycle mode, for example, between G82 block and G80 block, *no cycle* will be executed in that block and the value of P in the fixed cycle definition is *not* updated. On the latest Fanuc controls, a system parameter setting enables or disables this usage. If this rare method is used at all, the command G04 P. will be active *before* the tool rapid motion from the location just completed. The dwell function will always be executed while the cutting tool is out of a hole, in the clear space. *This feature is seldom required.*

25

FIXED CYCLES

Machining holes is probably the most common operation, mainly done on CNC milling machines and machining centers, as well as CNC lathes with milling axes. Even in industries traditionally known for their complex parts, such as aircraft and aerospace components manufacturing, electronics, instrumentation, optical or mold making industries, machining holes is a vital part of the manufacturing process.

When we think of what machining holes actually means, we probably think first of such operations as center drilling, spot drilling, and standard drilling, using common tools. However, this category is much wider. Many related operations also belong to this group of machining holes. Standard center drilling, spot drilling and drilling are used together with related operations such as reaming, tapping, single point boring, boring with block tools, countersinking and counterboring, spotfacing and even backboring.

Machining one simple hole may require only one tool but a precise and complex hole may require several tools to be completed. Number of holes required for a given job is important for selection of proper programming approach.

Even holes machined with the same tool may be different. Holes having the same diameter may have a variable depth, they may even be at different depths of the part. If all possible combinations are considered, it is easy to realize that making one hole may be a simple matter, but making a series of many different hole operations in a single program requires a well planned and organized approach.

In majority of programming applications, hole operations offer a great number of similarities from one job to another. Hole machining is a reasonably predictable operation and any operation that is predictable is an ideal subject to be handled very efficiently by a computer. For this reason, virtually all CNC control manufacturers have incorporated several ingenious programming methods for machining holes into their control systems. These methods use so the called *canned cycles* or - more commonly - the *fixed cycles*.

POINT-TO-POINT MACHINING

Machining holes is generally not a very complicated procedure. There is no contouring required and there is no multiaxis cutting motion. The only motion when actual cutting takes place is along a single axis - for machining centers, it is virtually always the Z-axis. This type of machining is commonly known as *point-to-point machining*.

Method of point-to-point hole machining is a method of controlling the motions of a cutting tool along X and Y axes at a *rapid* machine rate, and along Z-axis at a *cutting* feedrate. Some motions along Z-axis may also include rapid motions. All this means is that in standard XY plane, there is no cutting along XY axes for holes operations. When cutting tool completes all motions along the Z-axis and returns from the hole to a clearance position, motions along the X and Y axes resume and proceed to a new part location. There, all Z-axis motions are repeated. Usually, this sequence of motions occurs at many locations. The hole shape and diameter is controlled by cutting tool selection, cutting depth is controlled by the part program. This method of machining is typical to fixed cycles for drilling, reaming, tapping, boring and related operations.

Elementary programming structure for point-to-point machining can be summed up into four general steps (typical drilling sequence is shown in the example):

- Step 1: Rapid motion to the hole location
... along X and/or Y axis
- Step 2: Rapid motion to the starting point of the cut
... along Z-axis
- Step 3: Feedrate motion to the specified depth
... along Z-axis
- Step 4: Retract (return) to a clear position
... along Z-axis

These four steps also represent the *minimum* number of blocks required to program one single hole, using manual programming method, *without* using fixed cycles. If there is only one or two holes in a part drawing and machining operation is nothing more than a simple center drilling or drilling, program length is of no significant importance. Normally, that is not the common case - there are many holes in a part and several tools have to be used to complete each hole to engineering specifications. Such a program could be extremely long and very difficult to interpret and even to make a small change, if necessary. In fact, it may even be too long to fit into standard CNC memory.

Possibly the most time consuming task in programming point-to-point operations is the amount of *repetitive* information that has to be written in a CNC program. This problem was solved by the introduction of fixed cycles. These cycles are also known as *canned cycles* because a lot of repetitive information is *canned* (or fitted) into a relatively small space of a computer chip.

◆ Single Tool Motions vs. Fixed Cycles

Two following examples compare the differences of programming a hole pattern as individual blocks (O2501), where each tool path step must be programmed as a single motion block, and the same pattern of holes using a fixed cycle (O2502). No explanations to the programs are given at this stage and the comparison is only a visual illustration between two distinct programming methods. It shows an application of a $\varnothing 5$ mm standard drill that is used to cut a full blind depth of 16 mm. Only three holes are used for the example, illustrated in Figure 25-1.

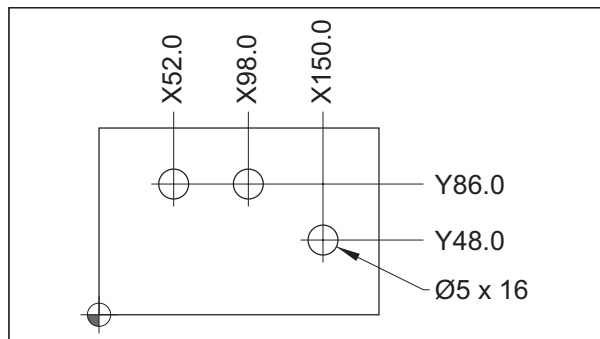


Figure 25-1

Simple hole pattern - programs O2501 and O2502

O2501 (EXAMPLE 1 - WITH INDIVIDUAL BLOCKS)

```

N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X52.0 Y86.0 S900 M03
N4 G43 Z10.0 H01 M08
N5 Z2.5
N6 G01 Z-17.5 F100.0
N7 G00 Z2.5
N8 X98.0
N9 G01 Z-17.5
N10 G00 Z2.5
N11 X150.0 Y48.0
N12 G01 Z-17.5
N13 G00 Z2.5 M09
N14 G28 X150.0 Y48.0 Z2.5
N15 M30
%
```

Second example listed in program O2502 uses the same hole pattern, but a fixed cycle offers many benefits.

O2502 (EXAMPLE 2 - WITH A FIXED CYCLE)

```

N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X52.0 Y86.0 S900 M03
N4 G43 Z10.0 H01 M08
N5 G99 G81 X52.0 Y86.0 R2.5 Z-17.5 F100.0
N6 X98.0
N7 X150.0 Y48.0
N8 G80 G28 X150.0 Y48.0 Z2.5 M09
N9 M30
%
```

For three holes only, program O2501 required total of 15 blocks. Program O2502 used a fixed cycle, and only nine blocks were needed. The shorter program O2502 is also easier to interpret, there are no repetitious blocks. Hole related modifications, updates and other changes can be done quickly, whenever required.

◆ Basic Concept

Comparing the two previous examples suggests the basic concept of any fixed cycle - *store repetitive data only once, and use them as many times as needed*. Figure 25-2 shows this basic concept applied to the first hole:

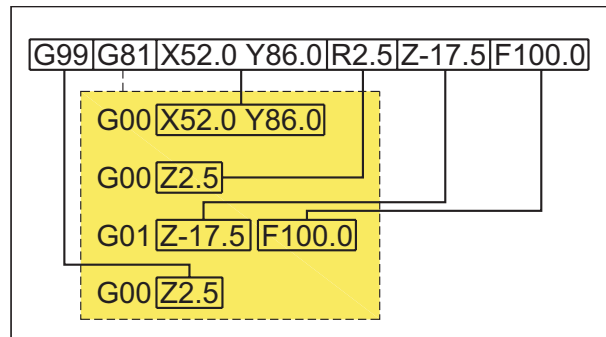


Figure 25-2

Basic concept of fixed cycles applied to examples O2501 and O2502

Once a cycle is called, only data that change are programmed - XY position is the most common change. As a general rule, fixed cycles should be used for all hole operations, even if a single hole is machined.

FIXED CYCLE SELECTION

Fixed cycles have been designed by control manufacturers to eliminate *repetition* in manual programming and allow an easy *program data changes* at the machine.

For example, a number of identical holes may share the same start position, same depth, same feedrate, same dwell, and other features. Only X and Y axes locations are often different for each hole machined. The main purpose of fixed cycles is to allow for programming necessary values only once - *for the first hole of the arrangement*. Specified values become modal for the duration of the cycle only and do not have to be repeated, unless and until one or more of them change. This change is usually for the XY location of a new hole, but other values may be changed for any hole at any time, particularly for more complex holes. A fixed cycle is called in the program by a special preparatory G command.

Fanuc and similar control systems support the following fixed cycles, often considered standard by other control manufacturers:

G73	High speed peck drilling cycle
G74	Left-hand tapping cycle
G76	Precision boring cycle
G80	Fixed cycle cancellation (any cycle)
G81	Drilling cycle
G82	Drilling cycle with dwell
G83	Peck drilling cycle
G84	Right-hand tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle

The list is only general and indicates the most *common* use of each cycle, not always the *only* use. For example, certain boring cycles may be quite suitable for reaming, although there is no reaming cycle directly specified. The next section describes programming format and details of each cycle and offers suggestions for their proper applications. Think of fixed cycles in terms of their *built-in* capabilities, not their general description.

PROGRAMMING FORMAT

General format for a fixed cycle is a series of parameter values specified by a unique address (not all parameters are used for every available cycle):

N.. G.. G.. X.. Y.. R.. Z.. P.. Q.. I.. J.. F.. L.. (or K..)

Explanation of the addresses used in fixed cycles (in the order of their typical programmed order):

N = Block number

Within the range of N1 to N9999 or N1 to N99999, depending on the control system

G (first G command) = G98 or G99

- G98 returns tool to the initial Z position
- G99 returns tool to the point specified by address R

G (second G command) = Cycle number

- Only one of the following G commands can be selected:

G73 G74 G76 G81 G82 G83
G84 G85 G86 G87 G88 G89

X = Hole position in X axis

X value can be an absolute or incremental value

Y = Hole position in Y axis

Y value can be an absolute or incremental value

R = Z axis start position = R level

- Position at which the cutting feedrate is activated

R level position can have an absolute value or an incremental distance and direction.

Z = Z axis end position = Z depth

- Position at which the feedrate ends

Z depth position can have an absolute value or an incremental distance and direction.

P = Dwell time (pause)

- Programmed in milliseconds (1 second = 1000 ms)

Dwell time (pause of tool) is practically applicable only to G76, G82, G88 and G89 fixed cycles. It may also apply to G74, G84 and other fixed cycles, depending on the control system parameter setting.

- Dwell time can be within a range of 0.001 and 99999.999 seconds, programmed as P1 to P99999999

Q = Address Q has two meanings

- When used with cycles G73 or G83, it means a *depth of each peck*
- When used with cycles G76 or G87, it means *amount of shift for boring*

Addresses I and J may be used instead of address Q, depending on the control system parameter setting.

I = Shift amount

- Must include the X-axis shift direction for boring cycles G76 or G87

I shift may be used instead of Q setting - see above

J = Shift amount

- Must include the Y-axis shift direction for boring cycles G76 or G87

J shift may be used instead of Q setting - see above

F = Feedrate specification

- Applies to actual cutting motion only

This value is expressed in *mm/min* or *in/min*, depending on the dimensional input selection.

L (or K) = Number of cycle repetitions

- Must be within the range of L0 and L9999 (K0 and K9999) L1 (K1) is the default condition

GENERAL RULES

Programming is a very controlled discipline - it means that there are rules, there are strict conditions, there are limitations, and there are restrictions. CNC programming is not a language programming but shares a lot with it. We talk about a *Fanuc* or *Siemens* programming, a *Cincinnati* programming, a *Mitsubishi* or *Mazatrol* programming, for example. In essence, fixed cycles are miniature programs.

Consider fixed cycles as a set of small condensed modules - modules that contain a step-by-step series of *preprogrammed* machining instructions (as in *Figure 25-2*). The cycles are called 'fixed', because their internal format cannot be changed by the CNC programmer. These program instructions relate to the specific kind of predictable tool motions that repeats from job to job.

Various basic rules and restrictions relating to fixed cycles can be summed up in the following items:

- Absolute or incremental mode of dimensioning can be established *before* a fixed cycle is programmed or anytime *while* a fixed cycle is active
- G90 must be programmed to select absolute mode, G91 is required to select incremental mode
- Both G90 and G91 modes are modal !
- If one of the X and Y axes is omitted in fixed cycle mode, the cycle will be executed at the specified location of one axis and the current location of the other axis
- If both X and Y axes are omitted in fixed cycle mode, the cycle will be executed at the current tool position
- If neither G98 nor G99 command is programmed for a fixed cycle, control system will select the default command as set by a system parameter (usually G98 command)

- Address P for dwell time designation cannot use a decimal point (G04 is not used) - dwell is always programmed in milliseconds
- If L0 (K0) is programmed in a fixed cycle block, control system will store the block data for later use, but will *not* execute them at the current coordinate location
- Command G80 will always cancel any active fixed cycle and will cause a rapid motion for any subsequent tool motion command. No fixed cycle will be processed in a block containing G80

➡ Example :

```
G80 Z1.125          ... is the same as ...
G80 G00 Z1.125      ... or ...
G00 Z1.125          ... OK but not recommended !
```

Preparatory G codes of Group 01, namely G00, G01, G02, G03 and G32 commands, are the main motion commands and will also cancel any active fixed cycle. For more information about G-code groups, *see page 55*.

Caution: In case of combining a fixed cycle command and a motion command of *Group 01* in a block, the *order* of programming such commands is very important:

```
G00 G81 X.. Y.. R.. Z.. P.. Q.. L.. F..
```

fixed cycle *is* processed, while in

```
G81 G00 X.. Y.. R.. Z.. P.. Q.. L.. F..
```

fixed cycle is *not* processed, X and Y motions will be performed, other data will be ignored, with the exception of feedrate F, which is stored. *Always void such situations!*

In this chapter, all fixed cycles are described in detail and each cycle has an illustration of its structure. Illustrations use shorthand graphic symbols, each with a specific meaning. *Figure 25-3* shows all symbols used in illustrations.

— — — — ➤	Rapid motion and direction
————➤	Cutting motion and direction
~~~~~➤	Manual motion and direction
▬➤	Boring bar shift and direction
⊙	Programmed coordinate
Q	Shift value / Depth of peck
d	Clearance value
CW / CCW	Spindle rotation
OSS	Oriented spindle stop
DWELL	Dwell function executed

Figure 25-3

Symbols and abbreviations used in fixed cycles illustrations

## ABSOLUTE AND INCREMENTAL INPUT

Like all machining processes, hole machining with cycles can use absolute programming mode G90 or incremental mode G91. Each selection will mainly affect the XY hole position, R-level, and Z-depth - Figure 25-4.

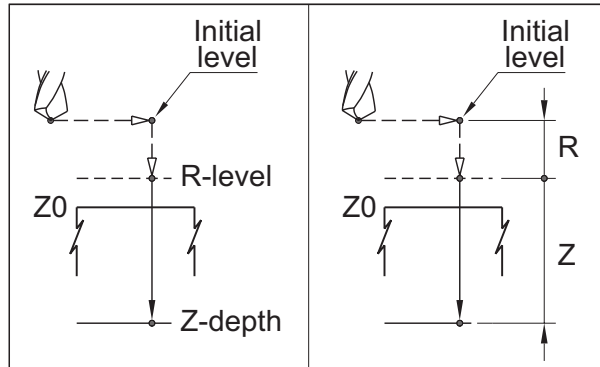


Figure 25-4

Absolute (left) and incremental (right) input values for fixed cycles

In absolute programming method, all values are related to the point of origin - often called the *program zero*. In incremental method, XY position of one hole is the distance from XY position of the previous hole. R-level is the distance from the *last Z* value, one established *before* calling a cycle, to the position where feedrate is activated. Z-depth value is the distance between R-level and the termination of feedrate motion. At the start of any fixed cycle, tool motion to the R-level will always be in rapid mode.

## INITIAL LEVEL SELECTION

There are two preparatory commands that control retract of the Z-axis when a fixed cycle is *completed*.

G98	... will cause the cutting tool to retract to the <i>initial</i> position = Z address designation
G99	... will cause the cutting tool to retract to the <i>R-level</i> position = R address designation

G98 and G99 commands are used for fixed cycles only and have no effect in other motion modes. Their main purpose is to bypass obstacles between holes within a machined hole pattern. Obstacles may include clamps, holding fixtures, protruding sections of the part, unmachined areas, accessories, etc. Without these commands, the cycle would have to be canceled and tool moved to a safe position. The cycle could then be resumed. With the G98 and G99 commands, such obstacles can be bypassed *without* canceling the fixed cycle, for more efficient programming.

Initial level is - by definition - absolute value of the *last Z* axis coordinate in the program - the last Z-address before a fixed cycle is called - Figure 25-5.

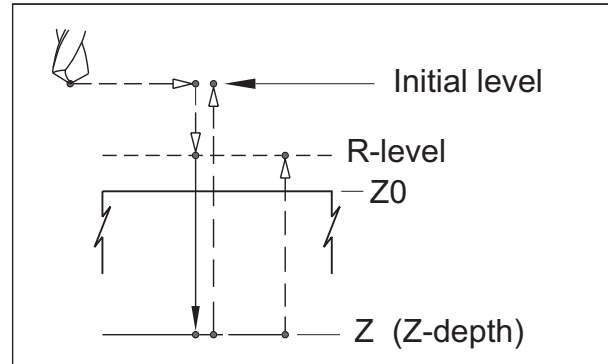


Figure 25-5

Initial level selection for fixed cycles

From practical point of view, always select this position as the *safe* level - not just anywhere and not without some prior thoughts. It is important that the level to which the tool retracts when G98 command is in effect is *physically above all obstacles*. Always use initial level with other precautions, to prevent a collision of the cutting tool during rapid motions. A collision occurs when the cutting tool is in an undesirable contact with the part, holding fixture, or the machine itself.

### ➡ Example of initial level programming :

Following program segment is a typical example of programming the initial level position (G21 mode):

```

...
N11 G90 G54 G00 X100.0 Y45.0 S1200 M03
N12 G43 Z20.0 H01 M08 (INITIAL LEVEL Z20.0)
N13 G98 G81 X100.0 Y45.0 R2.5 Z-19.5 F150.0
N14 ...
...
N20 G80
...
```

Fixed cycle (G81 in the example) is called in block N13. The last Z-axis value *preceding* this block is programmed in block N12 as Z20.0. This is setting of the initial position - *twenty millimeters above Z0* level of the part. The Z position can be selected at a standard general height, if the programs are consistent (for example 25 mm or 1 inch), or it may be different from one program to another. Safety is always the deciding issue here.

Once a fixed cycle is applied, initial Z-level cannot be changed, unless the cycle is canceled first with G80. Then, the initial Z-level can be changed and a required cycle be called. Initial Z-level is programmed as an absolute value, in G90 mode.

## R-LEVEL SELECTION

A particular cutting tool position from which the feedrate begins is also specified along the Z-axis. That means any fixed cycle block requires *two positions* relating to the Z-axis - one for the start point at which cutting begins, and another for the end point indicating hole depth. Basic programming rules do not allow the same axis to be programmed more than once in a single block. Therefore, some adjustment in the control design must be made to accommodate *both* Z values required for a fixed cycle. The obvious solution is that one of them must be *replaced with a different address (letter)*.

Since the Z-axis is closely associated with depth, it retains this meaning in all cycles. Replacement address is used for the tool Z position from which cutting feedrate is applied. This address uses the letter R. A simplified term of reference to this position is the *R-level*. Think of R-level in terms of 'Rapid to start point', where the emphasis is on the phrase 'Rapid to' and the letter 'R' - see Figure 25-6.

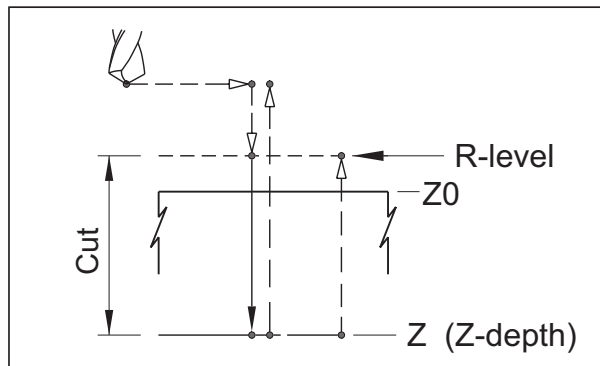


Figure 25-6

R level selection for fixed cycles

R-level is not only the start point of cutting feedrate, it is also the Z-axis position to which the cutting tool will retract upon cycle completion - *if* preparatory command G99 was programmed. If G98 was programmed, retract will be to the initial level. Later, G87 backboring cycle will be described as an exception, because of its purpose. This cycle does not use G99 retract mode, only G98! However, for all cycles, the R-level position must be selected carefully. The most common values are 1 to 5 mm (0.04 to 0.20 inches) above part face. Part setup has to be considered as well, and adjustments to the settings made, if necessary.

R-level that is programmed normally for drilling and similar operations is usually increased about three or four times for tapping operations using cycles G74 and G84. The main purpose of this additional clearance is to allow feedrate acceleration to reach its maximum *before* actual contact with material - see page 216 for more details on tapping.

### ➡ Example of R-level programming (G21) :

```
...
N29 G90 G00 G54 X67.0 Y80.0 S850 M03
N30 G43 Z25.0 H04 M08 (INITIAL LEVEL IS 25.0)
N31 G99 G85 R2.5 Z-23.0 F200.0 (LEVEL IS 2.5)
N32 ...
...
N45 G80
```

The initial level in the example is in block N30, set to Z25.0 (25 mm or ~1.0 inch above part). R-level is set in block N31 (cycle call block) as 2.5 mm (~0.1 inches). In the same block, G99 command is programmed and never changed during the cycle. That means the tool position will be 2.5 mm above part zero at the start *and* end of the cycle. When tool moves from one hole to the next, it moves along XY axes only at this Z height level of 2.5 mm above work.

R-level position is normally *lower* than the initial level position. If these two levels are the same (Initial level = R-level), cycle start and end points are equivalent to the initial position. R-level is commonly programmed as an absolute value, in G90 mode, but changed into an incremental mode G91, if the application benefits from such a change.

## Z-DEPTH CALCULATIONS

Each fixed cycle must include a *depth of cut*. This is the depth at which cutting tool *stops* feeding into the material. Depth is programmed by Z address when a cycle is called. The end point for depth cut is programmed as a Z value, normally *lower* than R-level and *initial* level. Again, G87 back boring cycle is an exception.

To achieve a program of high quality, always make a special effort to program calculated Z-depth accurately - *i.e., exactly*, without guessing its value or even rounding it off. For example, it may be tempting to round-off calculated depth of 0.6979 inches to 0.6980 or even to 0.70 - *avoid it!* It is not a question of triviality or whether one can get away with it. It is a matter of principle, programming consistency and discipline. With this approach and attitude, it will be much easier to retrace cause of a problem, should one develop later.

Z-depth calculation is based on the following criteria:

- Hole dimension in the drawing (diameter and depth)
- Absolute or incremental programming method
- Type of cutting tool used
- Added tool point length (for drills and other tools)
- Material thickness or full diameter depth of hole
- Selected clearances - above and below material (below material clearance for through holes)



On vertical machining centers, Z0 origin is typically programmed as the top of finished part face. In this case, the absolute value of Z address will always be programmed as a *negative* value. Recall that the *absence* of a sign in an axis address means positive value of that address. This method has one strong advantage. In case the programmer forgets to write the minus sign, actual depth value will automatically become a positive value. In that case, cutting tool will move *away* from the part, generally into a safe area. The part program will not be correct, but can be easily corrected, with only a loss of time.

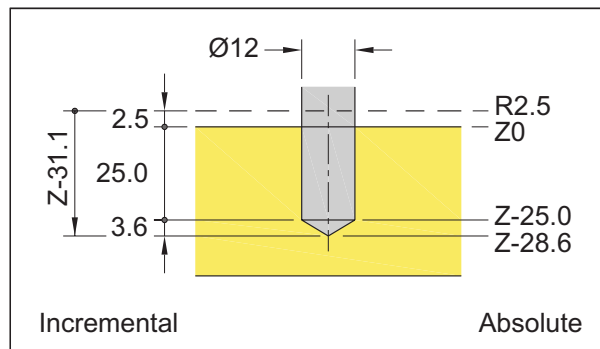


Figure 25-7

Z depth calculation for a drilling fixed cycle

#### ➡ Example of Z-depth calculation :

To illustrate a practical example of Z-depth calculation, consider the hole detail in *Figure 25-7*. A Ø12 mm drill is used to make the hole, with a full depth of 25 mm. If a standard twist drill is used, its tool tip length has to be taken into consideration. Standard drill design has a typical 118° to 120° point angle and an additional 3.6 mm has to be added to the specified depth:

$$12.0 \times 0.3 = 3.6 \quad \dots 0.3 \text{ is a constant}$$

$$25.0 + 3.6 = 28.6$$

Based on the result, the total Z-depth of 28.6 mm can be written in the CNC program:

G99 G83 X90.0 Y-40.0 R2.5 Z-28.6 Q8.0 F175.0

A peck drilling cycle G83 has been used in the example for best machining, although both R and Z values would be the same for G81, G82 or G73 cycles. The tool point length calculation using constant for various drill point angles is described in detail - *Chapter 26 - see page 201*.

There are some unique differences in calculating Z-depth for blind holes and through holes. *Chapter 26 (Machining Holes)* covers all major details.

## DESCRIPTION OF FIXED CYCLES

In order to understand how each fixed cycle works, it is important to understand the internal structure of each cycle and all details of its programming format. In the following descriptions, each fixed cycle will be evaluated in detail. Each cycle heading indicates the basic cycle programming format, followed by explanation of exact operational sequences. Common applications of each cycle will also be described.

All these details are important and should be helpful in understanding the nature of each cycle, as well as help in their selection - which cycle to select for the best machining results. In addition, knowledge of the internal cycle structure will help in designing any unique cycles, particularly in the area of higher level of CNC programming - using *custom macros*.

### ◆ G81 - Drilling Cycle

G98 (G99) G81 X.. Y.. R.. Z.. F..	
Step	Description of G81 Cycle
1	Rapid motion to XY position
2	Rapid motion to R-level
3	Feedrate motion to Z-depth
4	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to R-level (with G99)

#### WHEN TO USE G81 CYCLE - Figure 25-8 :

Mainly for drilling and center drilling, where a dwell (pause) at the bottom of hole (Z-depth) is not required.

If used for boring, G81 cycle will produce a helical scratch mark on the hole cylinder during retract.

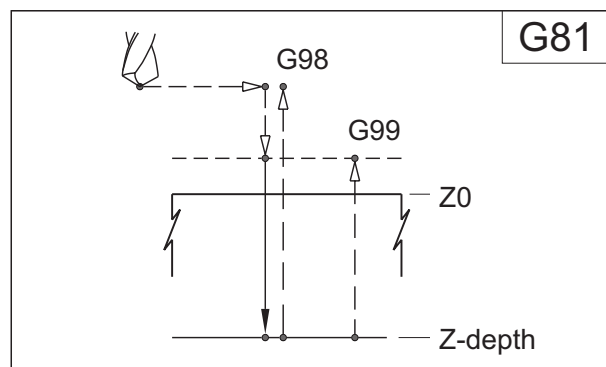


Figure 25-8

G81 fixed cycle - typically used for drilling

### ◆ G82 - Spot Drilling Cycle (Drilling with Dwell)

G98 (G99) G82 X.. Y.. R.. Z.. P.. F..	
Step	Description of G82 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Dwell at the depth - in milliseconds only (Pxxx)
5	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to <i>R-level</i> (with G99)

#### WHEN TO USE G82 CYCLE - Figure 25-9 :

Drilling with dwell - tool pauses at the hole bottom. Used for center drilling, spot drilling, spotfacing, countersinking, etc. - anytime a smooth finish is required at the hole bottom. Often used when slow spindle speed needs to be programmed.

If used for boring, G82 cycle will produce a helical scratch mark on the hole cylinder during retract.

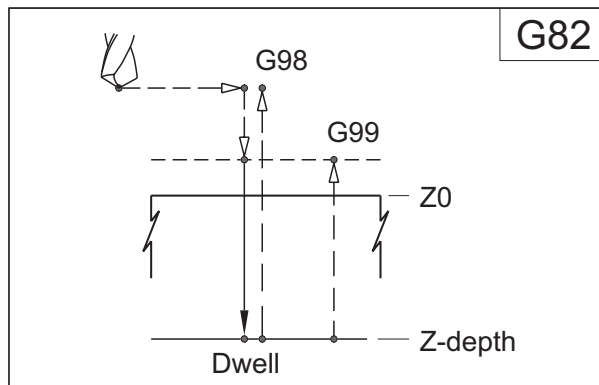


Figure 25-9

G82 fixed cycle - typically used for spot drilling

### ◆ G83 - Deep Hole Drilling Cycle - Standard

G98 (G99) G83 X.. Y.. R.. Z.. Q.. F..	
Step	Description of G83 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i> by <i>Q value</i> amount
4	Rapid retract to <i>R-level</i>
5	Rapid motion to the previous depth less a clearance (clearance 'd' is set by a system parameter)
6	Items 3, 4, and 5 repeat until <i>Z-depth</i> is reached
7	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to <i>R-level</i> (with G99)

#### WHEN TO USE G83 CYCLE - Figure 25-10 :

For deep hole drilling, also known as peck drilling, where the drill has to be retracted above the part (to a clearance position) after drilling to a certain depth. Compare this cycle with the high speed deep hole drilling cycle G73.

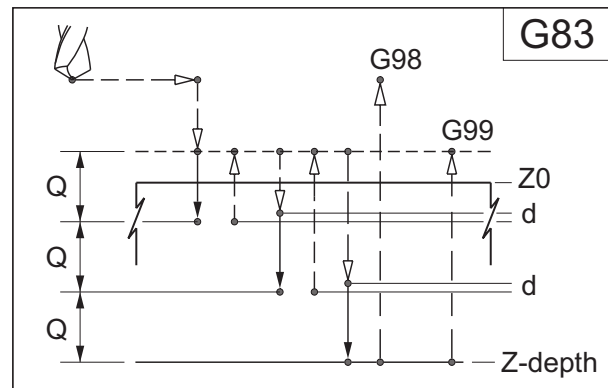


Figure 25-10

G83 fixed cycle - typically used for deep hole drilling  
- tool retracts to R-level after each peck

### ◆ G73 - Deep Hole Drilling Cycle - High Speed

G98 (G99) G73 X.. Y.. R.. Z.. Q.. F..	
Step	Description of G73 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i> by <i>Q value</i> amount
4	Rapid retract by a clearance value (clearance 'd' is set by a system parameter)
5	Feedrate motion in Z axis by the <i>Q amount</i> plus clearance 'd'
6	Items 4 and 5 repeat until <i>Z-depth</i> is reached
7	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to <i>R-level</i> (with G99)

#### WHEN TO USE G73 CYCLE - Figure 25-11 :

For deep hole drilling, also known as peck drilling, where chip breaking is more important than full retract of the drill. G73 cycle is often used for long series drills, when a full retract is not very important.

G73 fixed cycle is slightly faster than G83 cycle, hence its name 'high speed', because of time saved by **not** retracting to the R level after each peck. Compare this cycle with the standard deep hole drilling cycle G83.

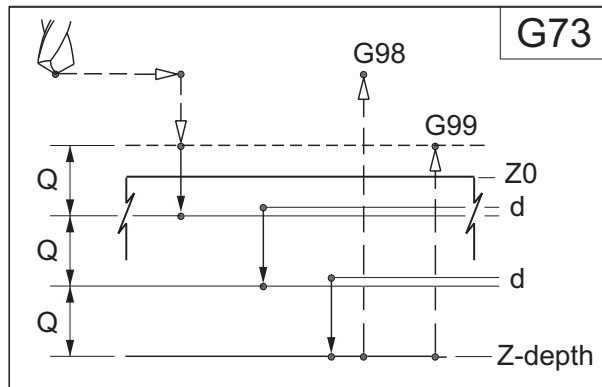


Figure 25-11

G73 fixed cycle - typically used for deep hole drilling or chip breaking  
- tool retracts to R-level only after the last peck

### Number of pecks calculation

When programming fixed cycles G83 and G73, always have at least a reasonable idea about how many pecks the drill will make - or should make - in each hole. Unnecessary peck drilling of hundreds or even thousands of holes can add up to significant lost time. Try to avoid too many pecks for each hole. For more predictable results, the number of pecks should always be *calculated*.

Actual number of pecks calculation applies equally to both G83 and G73 fixed cycles. Calculation of the number of pecks in these two fixed cycles is based on programmed amount of Q-depth and the total distance between R-level and Z-depth - *this calculation is not applied from the top of part (usually Z0)! Dividing this distance by programmed Q-amount will produce a number of pecks the drill will make at each hole location. Number of pecks in a cycle is always an integer and fractional calculations (if any) must always be rounded upwards, if necessary - here are some practical examples of various calculations:*

#### ➡ Example 1 - Metric units data - **no** rounding :

G90 G99 G73 X.. Y.. R2.5 Z-42.5 Q15.0 F..

In this metric example, the distance between R-level and Z-depth is exactly 45 mm (2.5 + 42.5) and the Q amount is set to 15 mm. Number of pecks will be 45 divided by 15, which equals exactly three. No rounding is necessary in this case and the number of pecks executed per hole will be three pecks of equal length.

In order to **increase** the number of pecks,  
change the current Q value to a **smaller** number.

In order to **decrease** the number of pecks,  
change the current Q value to a **larger** number.

Setting the Q amount is more accurate, if it is actually calculated rather than just estimated. To achieve desired number of pecks, divide the total distance between R-level and Z-depth by the required number of pecks. The result will be a Q amount programmed for the selected number of pecks. If rounding is necessary, always round *upwards*, otherwise the number of pecks may increase by one, without receiving any cycle time benefits - see the following example:

#### ➡ Example 2 - Metric units data - **with** rounding :

In this second example (independent of the first one), the distance between R-level and Z-depth is 56 mm, and exactly three pecks are required. Calculation of each peck depth is a simple division:

$$56 / 3 = 18.666667$$

Result is not an integer and *must* be rounded - to either 18.667 or 18.666 in metric. Although it looks that only *one* micron (0.001 mm) is at stake, it will make a big difference which way the rounding is done. If only three pecks are required, round off *upwards*, to Q18.667 (as a minimum):

Cut 1 18.667  
Cut 2 18.667  
Cut 3 18.666

Total 56 mm

If the result is rounded downwards, to Q18.666, the number of pecks will be four and practically no cutting will take place during the last peck:

Cut 1 18.666  
Cut 2 18.666  
Cut 3 18.666  
Cut 4 0.002 ... depth of the last peck is two microns !

Total 56 mm

#### ➡ Example 3 - Imperial units data - **no** rounding :

An example using imperial units - total distance between R-level and Z-depth is 2.5 inches with four pecks required:

$$Q = 2.5 / 4 = 0.625$$

In this case, no rounding is necessary, and Q0.625 will result in *exactly* four pecks, each of identical depth.

#### ➡ Example 4 - Imperial units data - **with** rounding :

G90 G98 G83 X.. Y.. R0.1 Z-1.4567 Q0.45 F..

In the second example, distance between the R-level and Z-depth is 1.5567 inches, Q value is 0.45, so the number of pecks can be calculated the same way as before:

$$1.5567 / 0.45 = 3.4593333$$

In this case, the result has too many decimal places and cannot be used as is, because most controls only accept *four* decimal places for imperial units (*three* decimal places for metric units). The result must be correctly *rounded upwards!*

As the nearest higher integer is four, each hole will require *four* pecks. Hole depth itself cannot be changed, so two available methods to change the number of pecks are:

- Change the R-level
- Change the peck depth

R-level is usually as close to the top part face as is practical, so there is not much that can be done there. That leaves the Q setting amount - the depth of each peck. By increasing this value, the total number of pecks will be fewer, by decreasing the Q value, the total number of pecks will be higher.

From the four examples shown, it is essential to understand the importance of calculations. Some additional observations are also important.

Peck drilling amount of Q can be changed for each hole, but that is seldom necessary. All pecks in a hole will always have an equal length, with a possible exception of the *last* peck. If the last peck amount is greater than remaining distance to the programmed *Z depth*, only that distance will be drilled:

No peck depth will ever exceed the **Z depth** coordinate position

Programmed Q amount can be adjusted in some creative ways for special purposes. By changing the Q peck amount skillfully, particular results can be achieved, such as an exact position of a tool tip during material breakthrough. This method is described in detail in *Chapter 26 - see page 206*.

To determine the 'best' peck depth, consider overall operating conditions for the job. Setup rigidity, part fixturing, cutting tool design, material machinability and other factors contribute to what a particular cutting tool can withstand while cutting. Past experience is also a good guide.

The main goal in peck drilling is to drill deep holes programmed in an efficient manner under safe conditions. That means programming the deepest Q amount that is reasonable and practical for a particular job and its setup.

Always keep in mind that there are two fixed cycles related to peck drilling - the standard G84 cycle and a similar, often neglected, G73 cycle.

#### ◆ G84 - Tapping Cycle - Standard

G98 (G99) G84 X.. Y.. R.. Z.. F..

Tapping sequence of G84 fixed cycle is based on *normal* initial spindle rotation - specified by M03.

Tap design must be in *right hand* orientation for G84 cycle with M03 spindle rotation in effect.

Step	Description of G84 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Spindle rotation stop
5	Spindle reverse rotation (M04) and feedrate back to <i>R-level</i>
6	Spindle rotation stop
7	Spindle rotation normal (M03) and retract to <i>initial level</i> (with G98) <i>or</i> remain at the <i>R-level</i> (with G99)

#### WHEN TO USE G84 CYCLE - Figure 25-12 :

Only for tapping a right hand thread. At the start of cycle, **normal** spindle rotation M03 must be in effect.

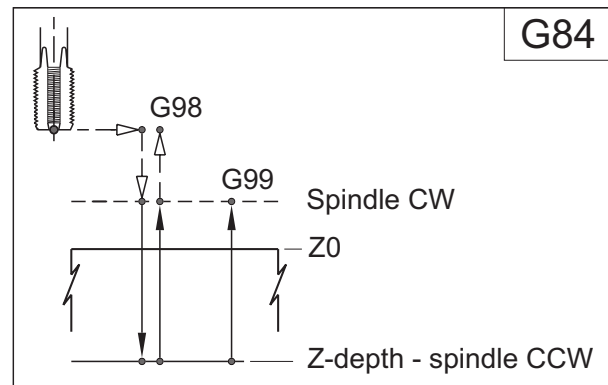


Figure 25-12

G84 fixed cycle - exclusively used for right hand tapping

#### ◆ G74 - Tapping Cycle - Reverse

G98 (G99) G74 X.. Y.. R.. Z.. F..

Tapping sequence of G74 fixed cycle is similar to that of G84 but based on *reverse* initial spindle rotation - specified by M04 spindle rotation function.

Tap design must be in *left hand* orientation for G74 cycle with M04 spindle rotation in effect.

Step	Description of G74 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>

4	Spindle rotation stop
5	Spindle normal rotation (M03) and feedrate back to <i>R-level</i>
6	Spindle rotation stop
7	Spindle rotation reverse (M04) and retract to <i>initial level</i> (with G98) or remain at the <i>R-level</i> (with G99)

#### WHEN TO USE G74 CYCLE - Figure 25-13 :

Only for tapping a left hand thread. At the start of cycle, **reverse** spindle rotation M04 must be in effect.

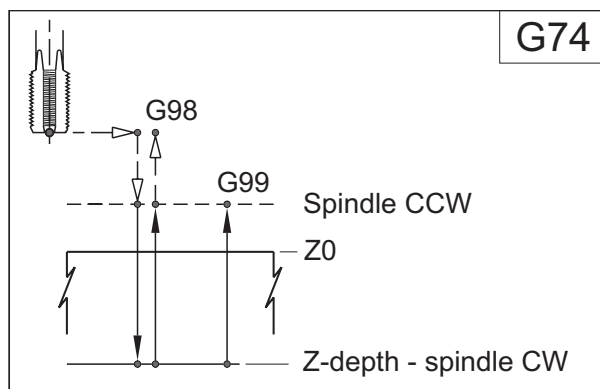


Figure 25-13

G74 fixed cycle - exclusively used for left hand tapping

Chapter 26 describes various techniques of hole machining, including tapping in significant detail.

Following comments cover only some more important tapping issues as they relate to programming and apply equally to both G84 and G74 tapping cycles:

- R-level should be higher for tapping cycle than for other cycles to allow for stabilization of cutting feedrate, due to acceleration
- Feedrate selection for the tap is very important. In tapping, there is a direct relationship between spindle speed and lead of the tap - this relationship must be maintained at all times
- Override switches on the control panel, used for spindle speed and feedrate, are ineffective during G84 or G74 cycle processing
- Tapping motion (in or out of the part) will be completed even if feedhold key is pressed during tapping cycle processing, for safety reasons

#### ◆ G85 - Boring Cycle

G98 (G99) G85 X.. Y.. R.. Z.. F..	
Step	Description of G85 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Feedrate motion back to <i>R-level</i>
5	Rapid retract to <i>initial level</i> (with G98) or remain at <i>R-level</i> (with G99)

#### WHEN TO USE G85 CYCLE - Figure 25-14 :

G85 boring cycle is typically used for boring and reaming operations. This cycle is used in cases where tool motion **into** and **out of** holes should improve the hole surface finish, its dimensional tolerances and/or its concentricity, roundness, etc. If using G85 cycle for boring, keep in mind that on some parts a tiny amount of stock may be removed while the cutting tool feeds backwards. This physical characteristics is due to relaxed tool pressure during retract. If hole surface finish gets worse rather than improves, try using another boring cycle.

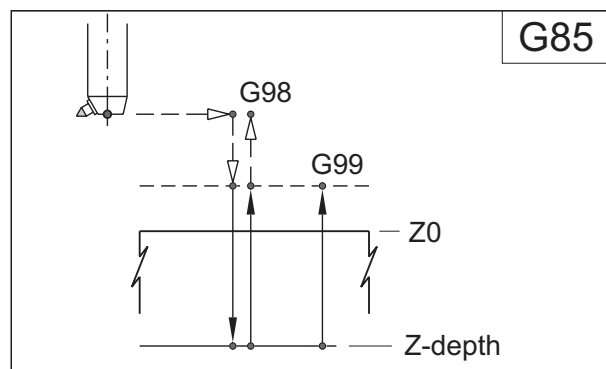


Figure 25-14

G85 fixed cycle - typically used for boring and reaming - no dwell

#### ◆ G86 - Boring Cycle

G98 (G99) G86 X.. Y.. R.. Z.. F..	
Step	Description of G86 cycle
1	Rapid motion to XY position with spindle on
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Spindle rotation stop
5	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to <i>R-level</i> (with G99)



**WHEN TO USE G86 CYCLE - Figure 25-15 :**

For boring rough holes or holes that require additional machining operations. This fixed cycle is very similar to G81 drilling cycle. Main difference is the spindle stop at the hole bottom.

**NOTE** - Although this cycle is somewhat similar to G81 cycle, it has characteristics of its own. In the standard drilling cycle G81, tool retracts while the machine spindle is rotating, but the spindle is stationary in G86 cycle. Never use G86 fixed cycle for drilling - for example, to save time - since any deposits of material on the drill flutes may damage the drilled surface or the drill itself.

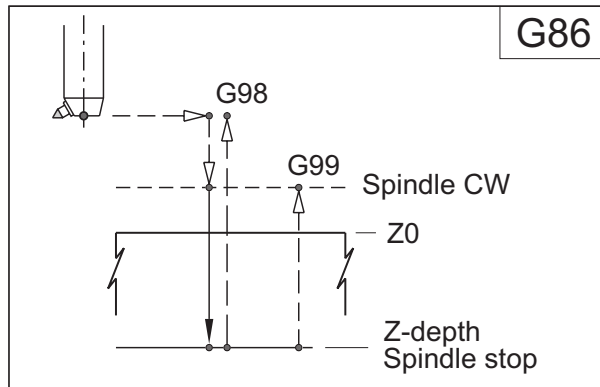


Figure 25-15

G86 fixed cycle - typically used for rough and semifinish boring operations

**◆ G87 - Backboring Cycle**

There are two programming formats available for the backboring fixed cycle G87 - the first one (using Q) is much more common than the second one (using I and J):

G98 G87 X.. Y.. R.. Z.. Q.. F..	
G98 G87 X.. Y.. R.. Z.. I.. J.. F..	
Step	Description of G87 cycle
1	Rapid motion to XY position
2	Spindle rotation stop
3	Spindle orientation
4	Shift <i>out</i> by the Q value or shift by the amount and direction of I and J
5	Rapid motion to R-level
6	Shift <i>in</i> by the Q value or shift back in the opposite direction of I and J
7	Spindle rotation on (M03)
8	Feedrate motion to Z-depth

9	Spindle rotation stop
10	Spindle orientation
11	Shift <i>out</i> by the Q value or shift by the amount and direction of I and J
12	Rapid retract to initial level
13	Shift <i>in</i> by the Q value or shift back in the opposite direction of I and J
14	Spindle rotation on

**WHEN TO USE G87 CYCLE - Figure 25-16 :**

This is a special cycle with rather limited use. It can only be used for some (not all) **backboring** operations. Its practical usage is not common, due to special tooling and setup requirements. Use G87 cycle only if all costs can be justified economically. In most cases, reversal of the part in a secondary operation is more practical.

**NOTE** - Boring bar used for this cycle must be set very carefully. It must be preset to match the diameter required for **backboring**. Its cutting bit must be set in oriented spindle mode, facing opposite direction than the shift direction.

This is also a cycle that has the most steps - fourteen. That means **each** backbored hole would require **fourteen** blocks using long-hand programming method.

G99 is **never** used with the G87 cycle

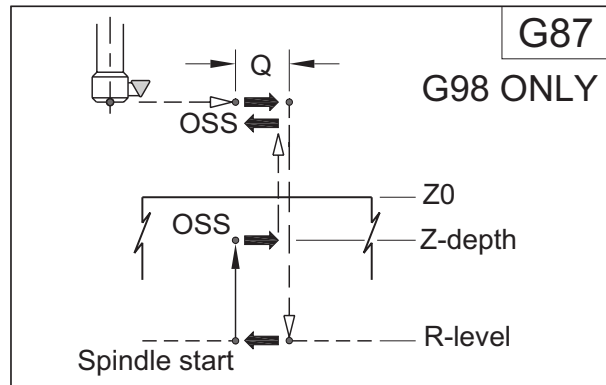


Figure 25-16

G87 fixed cycle - exclusively used for backboring - always with G98

**◆ G88 - Boring Cycle**

G98 (G99) G88 X.. Y.. R.. Z.. P.. F..	
Step	Description of G88 cycle
1	Rapid motion to XY position
2	Rapid motion to R-level
3	Feedrate motion to Z-depth

4	Dwell at the depth - in milliseconds (Pxxx)
5	Spindle rotation stop (feedhold condition is generated and CNC operator switches to manual operation mode and performs a manual task, then switches back to memory mode). CYCLE START will return to normal cycle
6	Rapid retract to <i>initial level</i> (with G98) or Rapid retract to <i>R-level</i> (with G99)
7	Spindle rotation on

#### WHEN TO USE G88 CYCLE - Figure 25-17 :

The G88 cycle is very rare. Its use is limited to boring operations with special tools that require **manual interference** at the **bottom** of a hole. When such an operation is completed, tool is moved out of the hole for safety reasons. This cycle may be used by some tool manufactures for certain operations.

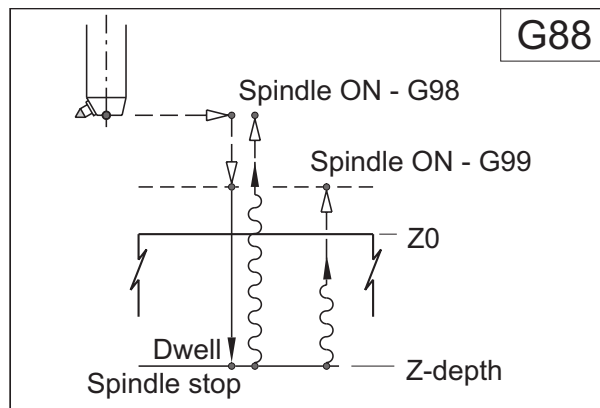


Figure 25-17

G88 fixed cycle - used when manual operation is required - rare

#### ◆ G89 - Boring Cycle

G98 (G99) G89 X.. Y.. R.. Z.. P.. F..	
Step	Description of G89 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Dwell at the depth - in milliseconds (Pxxx)
5	Feedrate motion to <i>R-level</i>
6	Rapid retract to <i>initial level</i> (with G98) or remain at <i>R-level</i> (with G99)

#### WHEN TO USE G89 CYCLE - Figure 25-18 :

For boring operations, when feedrate is required for both **in** and **out** motions, with a specified dwell at the hole bottom. Dwell distinguishes G89 cycle from G85 cycle.

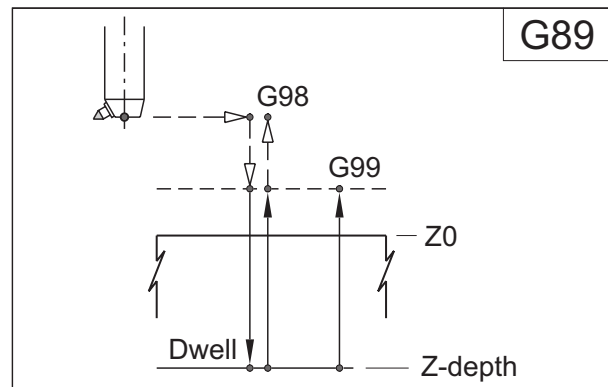


Figure 25-18

G89 fixed cycle - typically used for boring and reaming - with dwell

#### ◆ G76 - Precision Boring Cycle

This is probably the most useful cycle for high quality bored holes - no surface scratches on retract. There are two programming formats available for precision boring fixed cycle G76 - the first one (using Q) is much more common than the second one (using I and J):

G98 (G99) G76 X.. Y.. R.. Z.. P.. Q.. F..

G98 (G99) G76 X.. Y.. R.. Z.. P.. I.. J.. F..

Step	Description of G76 cycle
1	Rapid motion to XY position
2	Rapid motion to <i>R-level</i>
3	Feedrate motion to <i>Z-depth</i>
4	Dwell at depth - in milliseconds (Pxxx) ( <i>if used</i> )
5	Spindle rotation stop
6	Spindle orientation
7	Shift <i>out</i> by the Q value or shift by the amount and direction of I and J
8	Rapid retract to <i>initial level</i> (with G98) or remain at <i>R-level</i> (with G99)
9	Shift <i>in</i> by the Q value or shift back in the opposite direction of I and J
10	Spindle rotation resumes

#### WHEN TO USE G76 CYCLE - Figure 25-19 :

Boring operations, usually those for hole finishing, where quality of the completed hole is **very** important. Quality may be determined by the hole dimensional accuracy and with high surface finish. G76 cycle is also used to make holes cylindrical and with centerline parallel to a machine axis.

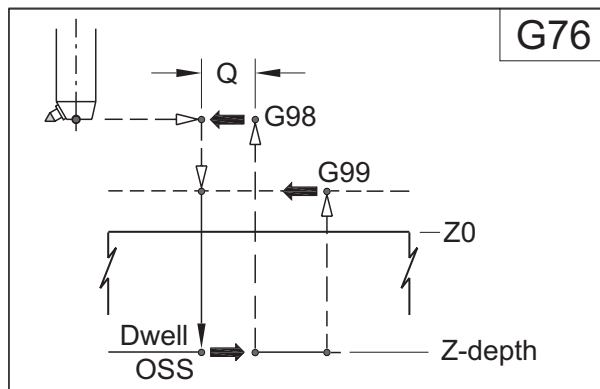


Figure 25-19

G76 fixed cycle - typically used for high quality boring

## FIXED CYCLE CANCELLATION

Any active fixed cycle can be canceled with G80 command. Control mode is automatically transferred to a rapid motion mode G00 when G80 is programmed:

```
N34 G80
N35 X15.0 Y-5.75
```

Block N35 does not specify rapid motion directly, it only *implies* it. This is a normal approach, but programming the G00 command as well may be a personal choice, although it is not necessary:

```
N34 G80
N35 G00 X15.0 Y-5.75
```

Both examples will produce identical results. The second version may even be a better choice. Combination of both examples offers another good programming method:

```
N34 G80 G00 X15.0 Y-5.75
```

In all three cases, the differences appear rather small, but they are very important to *understanding* fixed cycles. Although G00 *without* G80 would also cancel a cycle, it is a poor programming practice that should always be avoided.

## FIXED CYCLE REPETITION

When selected fixed cycle is programmed for many identical holes, this cycle is processed only once at each hole location within a part. This is normal condition, based on the assumption that most holes require only one cycle per tool. In CNC program, there is no *self-evident* special command that would indicate how many times to process a particular fixed cycle. That is true, such command is not evident, but it does exist. In fact, the assumption is that selected fixed cycle is to be done just *once* - i.e., not repeated.

Normally, the control system will execute a fixed cycle only once at a given location - in this case, there is no need to program the number of executions, since the system defaults to one automatically. To repeat the fixed cycle several times (more than once), program a special command that 'tells' the CNC system *how many times* you want the fixed cycle to be executed.

### ◆ The L or K Address

The command that specifies the number of repetitions (sometimes called *loops*) is programmed with the address L or K for some controls. The L or K address for the fixed cycle repetition is assumed to have a value of one, which is equivalent to a program statement L1 or K1. The L1 or K1 address does not have to be specified in the program.

For example, the fixed cycle call of the following drilling sequence,

```
N33 G90 G99 ...
N34 G81 X17.0 Y20.0 R0.15 Z-2.4 F12.0
N35 X22.0
N36 X27.0
N37 X32.0
N38 G80 ...
```

is equivalent to:

```
N33 G90 G99 ...
N34 G81 X17.0 Y20.0 R0.15 Z-2.4 F12.0 L1 (K1)
N35 X22.0 L1 (K1)
N36 X27.0 L1 (K1)
N37 X32.0 L1 (K1)
N38 G80 ...
```

Both examples will provide the control system with instructions for drilling four holes in a straight row - one at the location of X17.0 Y20.0, the other holes at locations X22.0 Y20.0 and X27.0 Y20.0, and X32.0 Y20.0 respectively - all to the depth of 2.4 inches.

If the L or K value in the second example is increased (or rather added to the first example), for instance, from L1 to L5 (or K1 to K5), the fixed cycle will be repeated five times at *each* hole location! There is no need for this type of machining. By changing the format only a little, the fixed cycle repetition can be used as a benefit - to make the program more powerful and efficient:

```
N33 G90 G99 ...
N34 G81 X17.0 Y20.0 R0.1 Z-2.4 F12.0
N35 G91 X5.0 L3 (K3)
N36 G90 G80 G00 ...
```

With that change, the advantage of a feature 'hidden' in the first example is emphasized - the *equal increment* between holes being exactly 5.0 inches. By using the incremental mode, on a temporary basis in block N35 and employing the power of the repetitive count L or K, the CNC

program can be shortened dramatically. This method of programming is very efficient for a large number of hole patterns in a single program. A further enhancement is to combine the L or K count with subprograms or macros.

#### ◆ L0 or K0 in a Cycle

In previous discussions, the default for a fixed cycle repetition was specified as L1 or K1, that does not have to be specified in the program. Any L or K value other than L1 or K1 must always be specified, within the allowable range of the L or K address. That range is between L0 and L9999 or K0 and K9999. The lowest L/K word is L0 or K0 - not L1 or K1! Why would we ever program a fixed cycle and then say *'do not do it'*. The address L0 or K0 means exactly that - *'do not execute this cycle'*. The full benefit of the L0/K0 word will be apparent in the examples listed under the section for subprograms, in *Chapter 39*.

By programming the L0 or K0 in a fixed cycle, what we are really saying is *not 'do not execute this cycle'*, but *'do not execute the cycle yet, just remember the cycle parameters for future use'*.

For most machining, fixed cycles are quite simple to learn. They do, however, have some complex features, waiting to be discovered and used in an efficient manner - even for a single hole.

## RIGID TAPPING

Discussion of fixed cycles would not be complete, unless another tapping method was also included. The two modes of tapping using fixed cycles were G84 (right hand tapping with M03) and G74 (left hand tapping with M04), already covered. Most modern CNC machining centers have a feature called *rigid tapping*. Yes, this method also requires a fixed cycle - but first, some technical details.

Rigid tapping has been available on CNC machining centers since the early 1990's. First, as an option, but more and more it is now offered as a standard machine feature. It is available under other names, such as *synchronized tapping*, for example. Regardless of the name, the main purpose of this modern tapping method must have some significant improvement over the more traditional method.

Incidentally, the description *'rigid'* does not refer to the rigidity of a CNC machine, but to the fact that a tap can be mounted into the same type of holder as an end mill, which is a rigid holder by design - the tool is *rigidly* mounted. That could also mean a collet or, more likely, a dedicated end mill holder with virtually no run-off. Rigid tapping uses spindle functions only and does not rely on any compensating mechanism, inherent to floating tap holders.

#### ◆ Comparison - Standard vs. Rigid Tapping

To compare the two methods, a simple table may be the best platform:

Operation / Function / Feature	Standard tapping	Rigid tapping
Floating tap holder required	YES	NO
Programming cycle available	YES	YES
Spindle / Feedrate synchronization	YES	NO
Peck tapping possible	NO	YES
Thread recutting	Difficult	Easy
Blind hole tapping	Moderate	Easy
Material consideration	High	Moderate
Hard materials tapping	High	Moderate
Spindle power requirements	High	Low
Z-depth adjustment possible	NO	YES
Overall costs	High	Low

As the table indicates, rigid tapping is the king of all tapping - if the CNC machine supports this feature. This is a factory installed machine feature and cannot be added later. Fortunately, it is a standard feature on virtually all CNC machining centers, but checking first with the vendor may prove to be useful.

#### ◆ Rigid Tapping - Fixed Cycles

This section title indicates that there is more than a single fixed cycle available for rigid tapping. In reality, that is not true, as it suggests a choice. Although the CNC programmer does have certain initial choice, once that is selected, another choice is not a practical option. Fanuc controls can have various settings, so checking a particular machine manual is always important.

Some modern controls can be set to Fanuc 15 mode, to allow older programs to be run on newer machines. Format uses familiar G84 cycle, but with a significant change:

**G84.2 X.. Y.. R.. Z.. F.. L.. (M03 MODE)**

where all other data have the same meaning as for standard G84 cycle. G98/G99 work the same way, and dwell P address can be added, if required. Repetition address L (loop) is for the number of repeats, also if required. As expected, this cycle will tap only a *right hand* thread. Surprisingly, there is no G74 version - instead, program G84.3:

**G84.3 X.. Y.. R.. Z.. F.. L.. (M04 MODE)**

Except the cycle number, all other block data are the same for both right hand and left hand rigid tapping.

For newer controls, such as models 16, 18, 21 and similar, there is another method. For a right hand tap, a familiar cycle can be used for rigid tapping:

```
G84 X.. Y.. R.. Z.. F.. K.. (M03 MODE)
```

For the left hand tap, the format is also the same:

```
G74 X.. Y.. R.. Z.. F.. K.. (M04 MODE)
```

Yes, these are exactly the same cycles as for standard mode tapping. The repeat address K (instead of L) is related to newer controls generally, and is *not* specific to rigid tapping. Although this format preserves the same cycle functionality, it also asks a natural question - *how does the control 'know' whether the tapping uses standard mode or rigid mode?*

Answer to this question is - it depends. It depends on whether the programmer's intention is to use rigid tapping exclusively or switch between standard and rigid modes, depending on the job. In the exclusive mode - rigid tapping only - a system parameter can be set, for example, parameter (G84) number 5200 bit #0 is set to 1 (for 16-18-21 control models).

For situations where some mode switching will be required, for example to work with old and new programs, some program modification will be necessary, using a miscellaneous function M29:

```
M29 S....
```

Typically, the M29 function is programmed together with the spindle speed command (r/min). There should be no axis movement between M29 and G84/G74 blocks, otherwise an alarm condition is generated. Look for the differences in these two examples (G21 mode - 0.75 mm pitch):

```
G00 X150.0 Y85.0 M03
S800
G84 R5.0 Z-16.5 F600.0
..
G80 ..
```

The same thread, using function M29:

```
G00 X150.0 Y85.0 M03
M29 S800
G84 R5.0 Z-16.5 F600.0
..
G80 ..
```

Fanuc allows machine manufacturers to change the M29 function to a different function. For programming, it means M29 is not necessarily standard.

### ◆ Rigid Pecking Cycle

This is a cycle unique to rigid tapping mode. It allows peck tapping, very much the same way as G83 (standard) or G73 (high speed) allows peck drilling. All that is needed is to include the Q-address in the cycle. Q-address is the depth of each peck:

```
G84 X.. Y.. R.. Z.. Q.. F.. K.. (M03 MODE)
```

or

```
G74 X.. Y.. R.. Z.. Q.. F.. K.. (M04 MODE)
```

Note that the examples distinguish right/left hand of tapping, *not* the standard/high speed mode like G83/G73.

Although it is possible to use standard and high speed mode in rigid peck tapping, it takes a system parameter setting to select one mode over the other mode. Again, for the 16-18-21 control models, parameter number 5200 is used, this time bit #5 (PCP) is selected. If set to 0, high speed peck drilling is performed, if set to 1, standard peck drilling is performed. This is a bit more cumbersome, but the choice is there.

### ◆ Cancellation

Just like any fixed cycle is cancelled by the G80 command, rigid tapping in any mode is cancelled by the same command.



## 26

## MACHINING HOLES

When programming CNC machining centers, there is always a hole or two that has to be drilled, bored, reamed, tapped, etc. From a simple spot drill to reaming, tapping and a complex backboring, the field of hole machining is very large. This chapter looks at several available programming methods for machining holes and presents techniques used. Various drilling and boring operations, as well as reaming, tapping and single point boring are covered.

As may be expected, the most common type of hole machining on CNC machining centers is in the area of drilling, tapping, reaming and single point boring. A typical machining procedure may be to center drill or spot drill a series of holes, then drill them, then tap or bore them. Machining one or more holes will benefit from using fixed cycles - G81 to G89, G73, G74 and G76, all described in previous *Chapter 25*.

## SINGLE HOLE EVALUATION

Before machining even a single hole on a CNC machine, all required tool paths have to be programmed. Before that, cutting tools have to be selected, speeds and feeds applied, the best setup determined and many other related issues must be resolved. Regardless of the exact approach, always start with a thorough *evaluation of the given hole*.

Evaluation step number one relates to part drawing. It will define the material to be machined, exact hole location and its dimensions. Holes are often *described*, rather than *dimensioned* and CNC programmer has to supply the missing details. *Figure 26-1* shows a medium complexity hole that can be machined using a CNC machine.

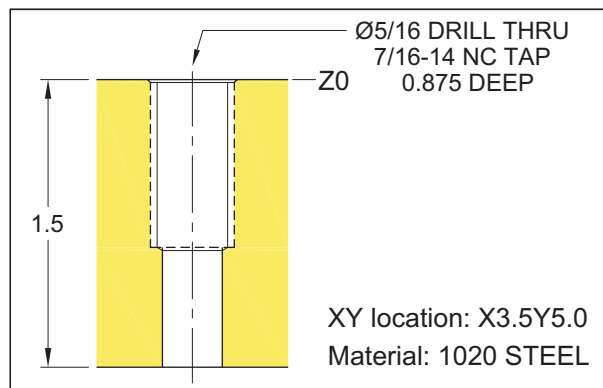


Figure 26-1

Evaluation of a single hole - programming example O2601

All relevant hole information is in the drawing, but some searching for details and other requirements is also needed. Hole location X3.5Y5.0 was specified in the drawing, as well as the material to cut - mild steel. Z-axis program zero will be assigned to the top face of part. Drilling and tapping operations are obvious, but is that all there is to know?

How many tools will be needed? What about center drilling to get exact location of the hole? Is spot drill a better choice? What about chamfering the drilled hole for tapping? What about hole tolerances and its surface finish? What about ...? Many questions have to be asked, and they all have to be answered.

### ◆ Tooling Selection and Applications

Based on drawing information alone, it may seem only two tools will be needed to program this hole. In reality, the implied information must be *interpreted* - it is not the drawing purpose to describe *how* to machine a hole - only the hole requirements related to its functionality and purpose. Good CNC programmer will most likely select *four* tools for best machining results. If four tools are selected, the first tool could be a 90° *spot drill*, followed up by a *tap drill*, then a *through-the-hole drill* and finally, a *tap*. A standard center drill may be used instead of a spot drill, but an additional tool will be required to chamfer the hole diameter at Z0 (top of part). All choices have to be sorted.

For this example, the following four tools are used:

- Tool 1 - T01 - 90° spot drill (+ chamfer)
- Tool 2 - T02 - Letter 'U' tap drill (Ø0.368)
- Tool 3 - T03 - Ø5/16 drill = Ø0.3125 - through the part
- Tool 4 - T04 - 7/16-14 UNC tap (Ø0.4375)

#### Tool 1 - 90° Spot Drill

The first tool is a 90° spot drill. Its purpose is dual - it will act as a centering drill and starts up the hole at very accurate XY location. A center drill or spot drill are much more rigid tools than a twist drill and either one will *start up* the hole, so the drill that follows will not deviate from its path - basic hole location and concentricity requirements are guaranteed. The second benefit of a spot drill is its chamfering capabilities. Design of this tool allows a chamfer to be made at the top of hole, providing the spot drill diameter is *larger* than chamfer diameter required. In this case, Ø5/8 spot drill is used, suitable to chamfer Ø7/16 hole.

Drawing does not specify a chamfer or its exact size, but a good machinist will always make a small chamfer, sometimes called a broken corner, unless there is a different requirement. A suitable chamfer here will be  $0.015 \times 45^\circ$ .

Once a suitable spot drill is chosen, its cutting depth has to be calculated - yes, *calculated*, not estimated. To make a  $0.015 \times 45^\circ$  chamfer for tap size  $\varnothing 7/16$  ( $\varnothing 0.4375$ ), the tap diameter has to be *enlarged* by 0.015 per side (0.03 on diameter), to 0.4675 chamfer diameter. Figure 26-2 shows hole relationship with the tool used - to calculate depth.

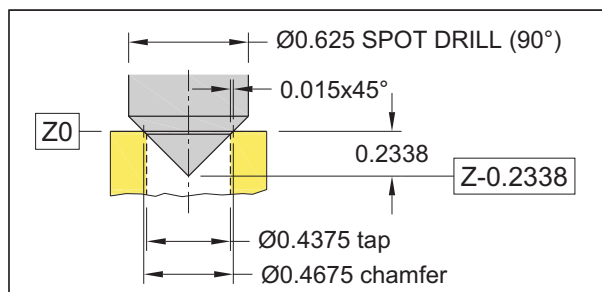


Figure 26-2

Spot drill operation detail - T01 in program O2601

Note, that for a  $90^\circ$  spot drill, depth of cut will always be *exactly* one half of the chamfer diameter ( $\varnothing \times 0.5$ ):

either  $0.4675 / 2 = 0.23375$   
 or  $0.4675 \times 0.5 = 0.23375$   
 for final ... **Z-0.2338**

Spot drill point length is discussed later in this chapter (see page 203).

#### Tool 2 - Tap Drill

Logically, the second tool will have to be a drill. In the example, *two* drills have to be used for the job - one for the through hole ( $\varnothing 5/16 = \varnothing 0.3125$ ), another one for the tap (letter U drill =  $\varnothing 0.368$ ). The question is - which one first? Does it really matter?

It certainly *does* matter which drill is programmed first. Main key here is the *difference* between the two drill diameters. It is a very small difference, only 0.0555 measured on diameter, in fact. From a machining point of view, it makes sense to use the *larger drill first*, than the smaller drill. Tap drill is larger than the through hole drill, so T02 will be the tap drill. If the smaller drill is programmed first, the larger drill that follows may produce an inaccurate hole, due to a very small amount of material to remove.

Now comes the question of first drill *size*. The *drill* in question is called a *tap drill*. It is a drill that will create round hole of proper size (*diameter* and *depth*) that can be used for the tapping operation that follows. Since machining operation calls for tapping, it makes a big difference for

*what purpose* is the tap used. Not all tapped holes can be done the same way. Some jobs require a loose fit, others a tight fit. The tap fit is determined by the size of tap drill. Most tapping applications fall into the 72-77% full thread depth category. In this case, T02 (letter U drill) will yield approximately 75% full thread depth. Percentage of the thread depth can be found in catalogues of all tap manufacturers. For example, these are some possible choices for the 7/16-14 tap:

Drill $\varnothing$	Decimal Value	Full Thread %
T	0.3580	86%
23/64	0.3594	84%
U	0.3680	75%
3/8	0.3750	67%
V	0.3770	65%

A metric drill between  $\varnothing 9.25$  and  $\varnothing 9.4$  can also be used. In general terms, for thin material stock, 75 to 80% full thread depth is recommended, for very thin stock even 100%. A thread that has 53% depth will, in most cases, break the bolt before it strips it. A full 100% thread is stronger by only 5% than a 75% thread, but the machine *power* required for tapping is three times higher.

Programmed Z-depth of the tap drill has to be deep enough to guarantee the required full thread depth of 0.875. That means *full* diameter of the drill has to reach a little deeper, for example, to 0.975 depth. That allows the tap end chamfer length to be *below* the full tap depth of 0.875, specified in the drawing. Figure 26-3 shows the tap drill values visually.

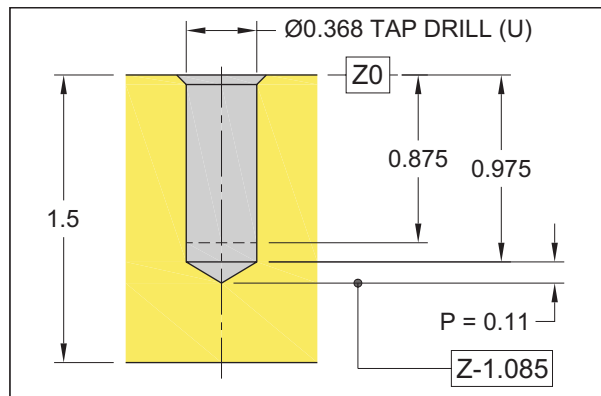


Figure 26-3

Tap drill operation detail - T02 in program O2601

Actual programmed depth for tap drill will have to take into consideration one more factor - the *drill point length*. Drill or - tool - point length is sometimes abbreviated as DPL, TPL or just by the letter P.

This chapter contains a table showing various mathematical constants to calculate drill point length - the most common constant uses drill diameter multiplied by 0.300, for a 118° drill point angle:

$$P = 0.368 \times 0.300 = 0.1104 = 0.11$$

Adding both calculations ( $0.975+0.11$ ), will provide the programmed Z-depth of Z-1.085.

*Tool 3 - Through Drill*

Next tool is a tool that drills the hole through the material. In the example, it is T03 (tool 3), a  $\varnothing 5/16$  standard drill.

As for the cutting depth of through drill, some simple calculations are needed. To make the calculations, hole depth has to be known, which is 1.5 inches in the example. Then, the calculated drill point length can be added to the required drill depth, usually with an extra clearance.

Relevant calculations for this through drilling operation are illustrated in *Figure 26-4*.

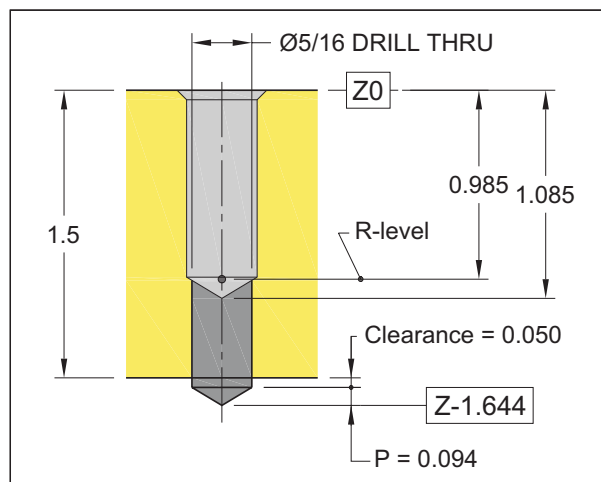


Figure 26-4

Through drill operation detail - T03 in program O2601

First, evaluate the drill point length P. It is calculated from a relationship of two given values - the drill *diameter* and the drill point *angle*.

For a standard Ø5/16 drill = Ø0.3125 that has 118° drill point angle, the 0.300 constant is used again, and the length of drill point P is:

$$P = 0.3125 \times 0.300 = 0.09375 = 0.0938 = 0.094$$

For the through hole in the example, drawing depth of 1.5 inches *plus* the calculated depth of 0.094 *seems to be* sufficient to drill this hole using selected tap drill.

In most through-hole applications, this value will *not* be sufficient - some extra clearance has to be added, applied to the tool penetration (breakthrough), say fifty thousands of an inch (0.050). Programmed value for the total drill depth (absolute Z value in the program) is the *sum* of nominal hole length, plus the tool point angle length, plus the selected clearance. In the program example, amount for through drill depth will be:

$$\text{Depth} = 1.5 + 0.094 + 0.05$$

$$= 1.644 \text{ or } Z-1.644 \text{ in the program}$$

One last calculation for this tool still has to be made. Remember that the previous tool had been used to predrill an opening? That means a smaller tool of  $\varnothing 0.3125$  is placed into an existing  $\varnothing 0.368$  hole. The drilling can start from inside of hole, rather than from a clearance above the part. In the program, R-level value is used and selected at R-0.985, which applies 0.100 clearance above the bottom of the existing hole. Both blind and through hole calculations are described later - *starting on page 203*.

*Tool 4 - Tap*

There is one more tool left to complete this example. It will be used for tapping the 7/16-14 thread. The thread size as specified in the drawing is 7/16 nominal diameter with 14 threads per inch ( $1/14 = 0.0714 = \text{pitch}$ ). Anytime a tapping tool is used in a program, watch its programmed depth along the Z-axis, particularly in a blind or semi-blind hole. The example shows a semi-blind hole, because the through hole is smaller than the tapped hole. If there were no through hole, it would be a blind hole with solid bottom, and if the through hole were the same size as tap drill, it would be a 100% through hole.

A through hole is the most forgiving for Z-depth calculation, closely followed by the semi-through hole. A blind hole has very little latitude, if any, and has to be programmed with maximum care.

The example drawing for the hole calls for tap depth of 0.875 inches. This is the *full* thread depth. Full depth of a thread is the actual distance a screw or a nut must travel before stopping (before retract). Programmed depth is, in fact, an *extended* depth, which must be *greater* than the theoretical depth, in order to achieve this goal. To calculate the length of extended depth, evaluate the tap end chamfer design (its type and length), described in more detail in the tapping section of this chapter.

A reasonable Z-depth is Z-0.95 (about one pitch over the depth) and can be optimized after actual machining. This is not really a calculation but an 'intelligent guess' - there is not much else that can be done and extensive experience helps. This completes the section on tooling application for a typical hole and provides enough data to write the actual program. Some of the procedures used in this example will now be explained in more detail.

### ◆ Program Data

In the example, only one hole is machined. If more holes are needed, they can be added by modifying the following program. For one hole used in the example, program includes all considerations for all four tools selected earlier. Spindle should be empty at the beginning of program:

```
O2601 (SINGLE HOLE EXAMPLE)
(T01 - 5/8 DIA - 90 DEGREE SPOT DRILL)
N1 G20
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X3.5 Y5.0 S900 M03 T02
N5 G43 Z0.1 H01 M08
N6 G99 G82 R0.1 Z-0.2338 P250 F4.0
N7 G80 Z1.0 M09
N8 G28 Z1.0 M05
N9 M01

(T02 - LETTER U DRILL - 0.368 DIA DRILL)
N10 T02
N11 M06
N12 G90 G54 G00 X3.5 Y5.0 S1100 M03 T03
N13 G43 Z0.1 H02 M08
N14 G99 G83 R0.1 Z-1.085 Q0.5 F8.0
N15 G80 Z1.0 M09
N16 G28 Z1.0 M05
N17 M01

(T03 - 5/16 DRILL THROUGH - 0.3125 DIA)
N18 T03
N19 M06
N20 G90 G54 G00 X3.5 Y5.0 S1150 M03
N21 G43 Z0.1 H03 M08
N22 G98 G81 R-0.985 Z-1.644 F8.0
N23 G80 Z1.0 M09
N24 G28 Z1.0 M05
N25 M01

(T04 - 7/16-14 TAP)
N26 T04
N27 M06
N28 G90 G54 G00 X3.5 Y5.0 S750 M03 T01
N29 G43 Z0.4 H04 M08
N30 G99 G84 R0.4 Z-0.95 F53.57 (F = S x LEAD)
N31 G80 G00 Z1.0 M09
N32 G28 Z1.0 M05
N33 G00 X-1.0 Y10.0 (PART CHANGE POSITION)
N34 M30
%
```

This detailed example shows that even a simple single hole requires a lot of thought and a significant amount of programming and machining skills.

## DRILLING OPERATIONS

Example O2601 provides a good illustration of what kind of programming and machining conditions are necessary for a typical hole. Next, let's look at the details of drilling operations in general, as they relate to various tools.

Drilling is one of the oldest operations in a typical machine shop. By definition, drilling is a removal of solid material to form a circular hole of the same diameter as the cutting tool (drill). Material removal is achieved by either rotating the drill (on milling systems) or by rotating the part itself (on turning systems). In either case, a vertical or horizontal machining application is possible. In a rather loose sense of the word, drilling operations also cover the extended areas of reaming, tapping and single point boring. Many programming principles that apply to drilling operations, can be equally applied to all related operations.

### ◆ Types of Drilling Operations

Drilling operation is determined by either the *type of hole* or the *type of tool*:

By the type of tool :	By the type of hole :
Center drill	Through hole
Spot drill	Chamfered hole
Twist drill (HSS, cobalt, etc.)	Semi-blind hole
Spade drill	Blind hole
Carbide indexable drill	Premachined hole
Special drill	...

### ◆ Types of Drills

Drills are categorized by their *design* and by their *size*. The oldest and most common design is a twist drill, usually made of high speed steel. Twist drill can also be made of cobalt, carbide and other materials, coated or uncoated. Other drill designs include spade drills, center drills, spot drills and indexable insert drills. Distinction in size is not only between metric and imperial drills, but also a finer distinction within the category using imperial units. All metric drills are designated in millimeters. Since the imperial dimensioning is based on inches (which is rather a large dimensional unit), finer distinctions are necessary. Inch dimensions of standard drills in imperial units are divided into three groups:

#### ■ FRACTIONAL SIZES :

1/64 minimum, in diameter increments of 1/64

#### ■ NUMBER SIZES :

Drill size #80 to drill size #1 ( $\varnothing 0.0135$  -  $\varnothing 0.228$ )

#### ■ LETTER SIZES :

Drill size letter A to drill size letter Z ( $\varnothing 0.234$  -  $\varnothing 0.413$ )

Metric sizes do not need any special distinctions. For imperial sizes, a listing of standard drills and their decimal equivalents is available from many sources.

### ◆ Programming Considerations

A standard drill has, regardless of size, two important features - the *diameter* and the *point angle*. Diameter is selected according to the drawing requirements, tool point angle relates to material hardness. They are both closely connected, since diameter determines the drilled hole size, tool point angle determines its depth. A smaller consideration is the number of flutes, which is normally two.

### ◆ Nominal Drill Diameter

The major consideration for a drill is always its diameter. Normally, the drill diameter is selected based on drawing information. If the drawing calls for a hole that needs only drilling and does not need any additional machining, drill to use is a standard drill. Its diameter is equivalent to the size specified in the drawing. A drill size of this kind is called a *nominal* or 'off-the-shelf' size.

Most applications involve holes that require other specifications in addition to their diameter - they include tolerances, surface finish, chamfer, concentricity, etc. In those cases, a single regular drill cannot be used alone and still satisfy all requirements. A *nominal* drill alone, even if the size is available, will not guarantee a high quality hole, due to machining conditions. Choosing a *multitool* programming method to machine such hole is a better choice. The normal practice in those cases is to use a drill size a bit smaller than the final hole diameter, then use one or more *additional* tools, which are capable of finishing the hole to drawing specifications. These tools cover boring bars, reamers, chamfering tools, end mills and others. Using these tools does mean more work is involved, but quality of the finished part should never be traded for personal conveniences.

### ◆ Effective Drill Diameter

In many cases, a drill is used to penetrate its *full* diameter through the part. In many other cases, only a *small portion* of the drill end point is used - a portion of the angular drill tip - *Figure 26-5*.

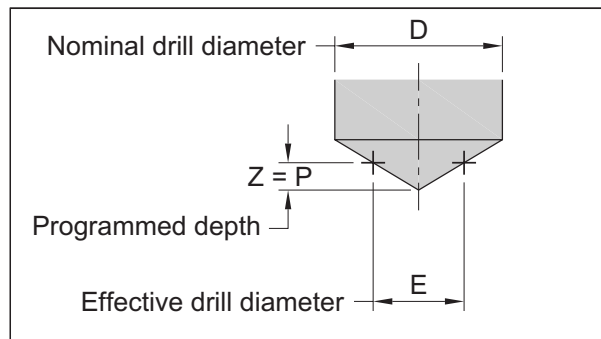


Figure 26-5

Nominal and effective drill diameters (twist drill shown)

During a cut, drill angular end will be gradually entered into the part, creating an increasingly larger hole diameter, yet still smaller than the drill diameter. At the end, the largest machined diameter will be equivalent to the *effective* diameter of the drill used. *Effective* drill diameter defines the actual hole diameter created *within* the zone of drill end point - it can also be the nominal drill diameter. Typical use is a spot drill operation for chamfering. Both spindle speed and feed must be calculated according to the *effective* drill diameter, not the full diameter, unless they are equal. The *r/min* and feedrate for effective diameter could be different than corresponding values for a nominal drill size. For this kind of jobs, selection of a short drill for rigidity is advised.

### ◆ Drill Point Length

The second important consideration is the drill point *length*. This length is very important to establish cutter depth for the full diameter. With the exception of a flat bottom drill, all twist drills have an angular point whose angle and length must be known in programming. These angles are largely standard and point length P must be calculated rather than estimated, because of its importance to the accurate hole depth - *Figure 26-6*.

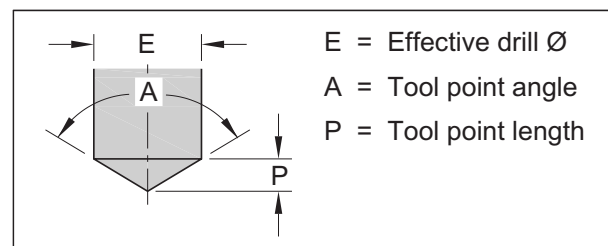


Figure 26-6

Tool point length data for a standard twist drill

On indexable insert drills this length is different, due to the drill construction. Indexable drill is not flat and its drill point length must also be considered in programming. A tooling catalogue shows all important dimensions.

Drill point length can be found quite easily, providing the drill diameter (nominal or effective) and the drill point angle are known. From the following formula and table of constants, required drill point length for standard drills can be calculated. Basic formula is:

$$P = \frac{\tan \left( 90 - \frac{A}{2} \right)}{2} \times E$$

where ...

- P = Drill point length **or** effective drill point length
- A = Drill point included angle
- E = Actual drill diameter **or** effective drill diameter



This rather long formula can be simplified and used with a mathematical constant (fixed for each drill point angle):

$$P = E \times K$$

where ...

- P = Drill point length
- E = Actual drill diameter **or** effective drill diameter
- K = Constant (see following table)

The most common constants  $K$  are listed in this table:

Tool Point Angle	Exact Constant	Practical Constant (K)
60°	0.866025404	0.866
82°	0.575184204	0.575
90°	0.500000000	0.500
118°	0.300430310	0.300
120°	0.288675135	0.289
125°	0.260283525	0.260
130°	0.233153829	0.230
135°	0.207106781	0.207
140°	0.181985117	0.180
145°	0.157649394	0.158
150°	0.133974596	0.134

Exact constant in the formula is rounded, but its practical value is sufficient for all programming applications. The value of constant  $K$  for 118° drill angle is 0.300, its real value is 0.300430310. Constant value has the main advantage of being easy to memorize and there is no formula to solve. For most jobs, only three constants are needed. For 90° (spot drilling and soft materials), 118° (standard materials), and 135° (hard materials). They are easy to learn:

- 0.5 ... for a 90° drill angle
- 0.3 ... for a 118° - 120° drill angle
- 0.2 ... for a 135° drill angle

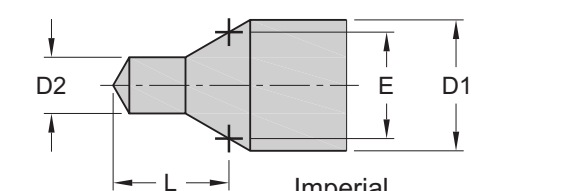
### ◆ Center Drilling

Center drilling is a machining operation that provides a small, concentric opening for tailstock support or pilot hole for a larger drill. Hole chamfering is not recommended with a center drill, because of the 60° angle of the tool.

Never center a hole to be drilled with indexable insert drills !

The most common tool for center drilling is a standard center drill (often called a *combined drill and countersink*), producing a 60° angle. Established North American industrial standards use a numbering system from #00 to #8 (plain type) or #11 to #18 (bell type) for center drills. In the metric system, center drills are defined by the pilot diameter, for example, a 4 mm center drill will have the pilot diameter of 4 mm. In both cases, the higher the number, the larger the center drill diameter. For some pre-drilling operations, such as chamfering, a tool with a 90° point angle, called a *spot drill*, is a much better choice.

Many programmers only estimate the depth of a center drill, rather than calculate it. Perhaps a calculation is not necessary for a temporary operation. What is a reasonable compromise between guessing and calculating is a data table, similar to that in *Figure 26-7* and *Figure 26-8*.

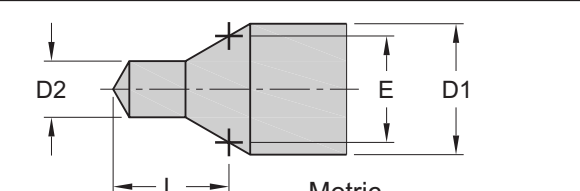


Imperial

Number	D1	D2	E	L
# 1	0.125	0.047	0.100	0.106
# 2	0.188	0.078	0.150	0.163
# 3	0.250	0.110	0.200	0.219
# 4	0.312	0.125	0.250	0.269
# 5	0.438	0.188	0.350	0.382
# 6	0.500	0.218	0.400	0.438
# 7	0.625	0.250	0.500	0.538
# 8	0.750	0.312	0.600	0.651

Figure 26-7

Standard center drill cutting depth table - #1 to #8 plain type  
 $L$  is the depth of cut for an arbitrary effective diameter  $E$  - imperial



Metric

Number	D1	D2	E	L
# 1	3.175	1.194	2.600	2.700
# 2	4.775	1.981	3.800	4.100
# 3	6.350	2.794	5.100	5.600
# 4	7.925	3.175	6.400	6.800
# 5	11.125	4.775	8.900	9.700
# 6	12.700	5.537	10.200	11.000
# 7	15.875	6.350	12.700	13.700
# 8	19.050	7.925	15.200	16.500

Figure 26-8

Same set of standard #1 to #8 center drills for metric applications

In both tables, there are necessary dimensions for standard imperial size center drills #1 to #8 (including their metric applications). The most important of them is cutting depth  $L$ . Its calculation has been based on a practical - but *arbitrary* - selection of the effective chamfer diameter  $E$ .

For example, #5 center drill has the depth value  $L$  that is listed as 0.382, based on an arbitrarily selected chamfer diameter  $E$  of 0.350 inches. These values can be modified as desired or a different table can be made. A similar table can be developed for true metric center drills (ISO).

### ◆ Spot Drilling

In the example O2601, the first tool used was a *spot drill*. A 90° spot drill has been chosen as an alternative to center drill, for three reasons:

- It provides the same hole positioning accuracy
- It can be programmed to make a chamfer on the hole
- It can be used for a large variety of hole diameters

Spot drill is unlike any other drill - its only cutting area is the angular tool point end, *never the body diameter*. Its web is much thinner than standard drills, and there is no clearance land on its edges. Programming the required depth is easy, because of its 90° point. It can be used for any hole that is smaller than its body diameter, with or without making a chamfer. The two following formulas show spot drill calculations; first, one *without* chamfer - Figure 26-09:

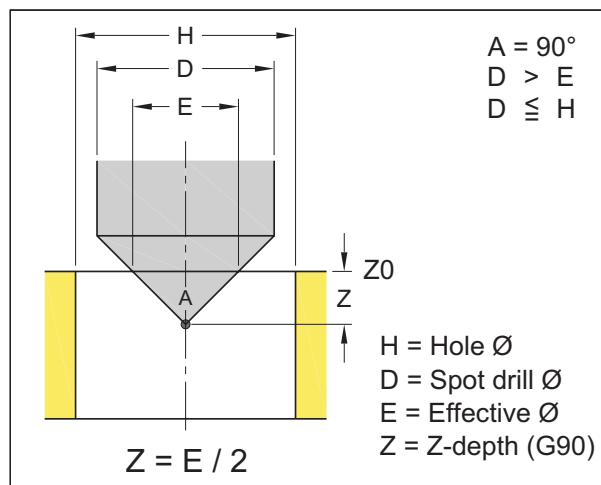


Figure 26-9

Spot drilling calculation for large holes - hole chamfer not required

This calculation is based on the size of effective diameter. Depending on the size of drill that follows, smaller or larger effective diameter may be considered. Use this calculation for locations of large holes only - holes that are larger than spot drill and when chamfer is not required.

The second calculation is similar and much more common in CNC programming. It shows a spot drill application that is used not only for precise hole location, but also to form a *specific chamfer*. Figure 26-10 shows the details:

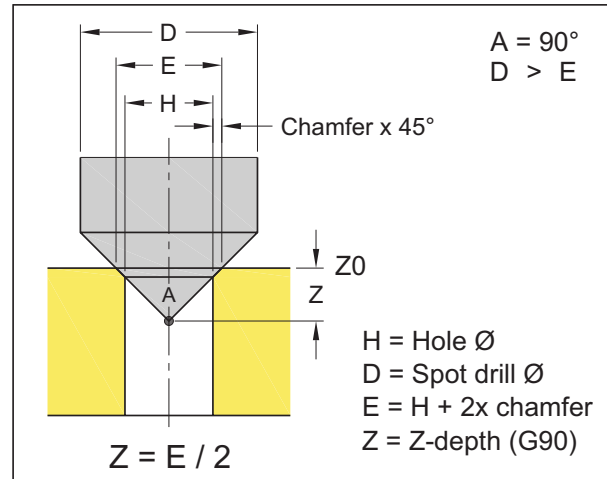


Figure 26-10

Spot drilling calculation for small holes - hole chamfer is required

Calculating the spot drill depth for holes with chamfer is simple - just take the effective diameter  $E$  - it may be called the *chamfer diameter* in this case, and divide it by two.

### ◆ Blind Hole Drilling

The major difference between drilling a blind hole and a through hole is that for blind holes, the drill does *not* penetrate material. Blind hole drilling should not present any more problems than a through hole drilling, but using a peck drilling method for deep holes may be worth considering in such cases. Also, a choice of a different drill geometry or even its design may improve overall machining.

In a typical shop drawing, depth of a blind hole is given as the *full diameter* depth. Drill point length is *not* normally considered to be part of the specified depth - it is in *addition* to any depth in the drawing.

As an example, if a standard Ø3/4 drill (Ø0.750) is used to drill a full diameter hole depth of 1.25 of an inch, the programmed depth will be:

$$1.25 + (0.750 \times 0.300) = 1.4750$$

In the part program (G20), corresponding block will be:

N93 G01 Z-1.475 F6.0

or - in case of a fixed cycle,

N93 G99 G85 X5.75 Y8.125 R0.1 Z-1.475 F6.0

Metric holes (in G21 mode) are treated exactly the same way. For example, a  $\varnothing 16$  mm drill is used to machine full diameter depth of 40 mm. Calculation uses the same constant K as for the sizes in imperial units:

$$40 + (16 \times 0.300) = 44.8$$

Depth specified in the drawing will have to be *extended* by the calculated drill point length. The new block will have the Z-axis value equal to the total of 40 mm depth, *plus* 4.8 mm calculated point length:

**N56 G01 Z-44.8 F150.0**

If the calculated depth appears in a fixed cycle, the same depth value will be used, although in a different format:

**N56 G99 G81 X215.0 Y175.0 R2.5 Z-44.8 F150.0**

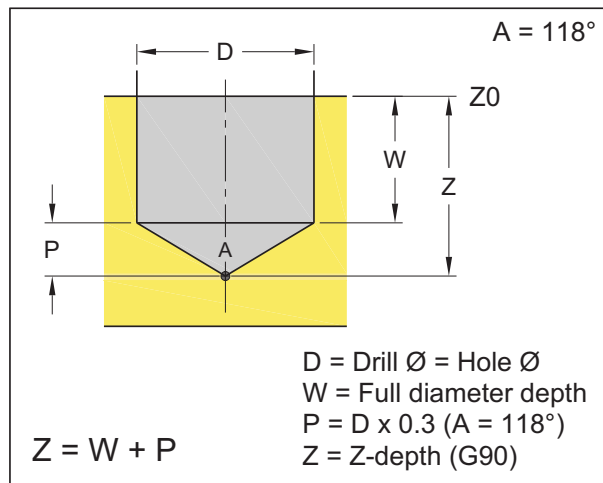


Figure 26-11

Blind hole drilling calculations - full diameter depth is known

In Figure 26-11, absolute Z-depth of a blind hole will be the *sum* of full diameter depth W, plus tool point length P. The previous example was based on these calculations.

When machining blind holes, cutting chips may accumulate and clog one or more holes, particularly at the hole bottom. This may cause a serious problem, especially if there is one or more subsequent operations on the same hole, for example, reaming or tapping. Make sure to include a program stop code M00 or M01 before such operation. For the purpose of hole clean up, M00 is a better choice, particularly if each hole has to be cleaned *every time* the program is executed. Otherwise, the much more efficient optional *program stop* M01 is quite sufficient. The major difference is how much actual space is left between initial drilling and any subsequent operation.

### ◆ Through Hole Drilling

Drilling a hole through the material is an equally common operation. It requires the Z-depth to include material thickness, drill point length and an extra clearance *beyond* the actual drill penetration point, also known as the *break-through* point or clearance.

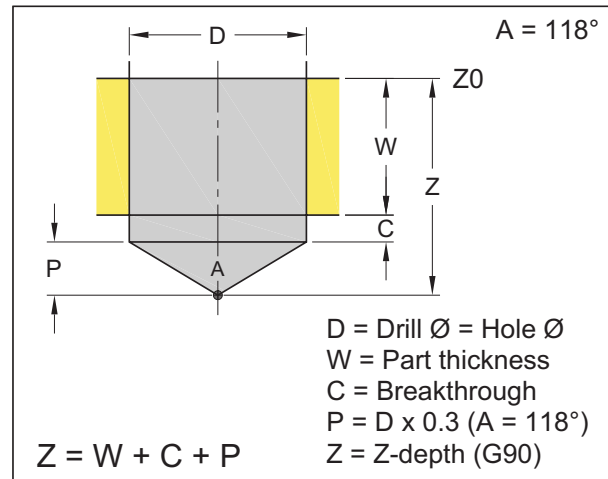


Figure 26-12

Through hole drilling calculations - with breakthrough clearance C

In Figure 26-12 it is shown that programmed depth for a through hole is the *sum* of material thickness that is equivalent to the full diameter depth W, plus the breakthrough clearance C, plus the tool point length P.

For example, if material thickness is one inch and the standard drill diameter D is  $\varnothing 5/8$  ( $\varnothing 0.625$ ) of an inch, the programmed depth, including a 0.050 clearance C, will be:

$$1 + 0.050 + 0.625 \times 0.300 = 1.2375$$

Pay attention to any possible obstructions (machine table, vise, parallels, fixture, machine table, etc.), when programming tool breakthrough clearance C. There is usually very little space below the part bottom face.

### ◆ Flat Bottom Holes

Flat bottom hole is a blind hole with a bottom at 90° to the spindle centerline (A=180°). The best practice is to use a standard drill to start the hole, then use a suitable size *slot drill* (generally known as a *center cutting end mill*), without predrilling. This is the best method, but some odd tool sizes may not be available. Using a flat bottom drill is usually not recommended.

To program a flat bottom hole using a slot drill is quite simple. For example - a  $\varnothing 10$  mm hole should be 25 mm deep (at the flat bottom). Using a  $\varnothing 10$  mm slot drill, the program is quite short (tool in spindle is assumed):

## O2602 (FLAT BOTTOM - VERSION 1)

```

N1 G21
N2 G17 G40 G80
N3 G90 G54 G00 X.. Y.. S850 M03
N4 G43 Z2.5 H01 M08
N5 G01 Z-25.0 F200.0
N6 G04 X0.5
N7 G00 Z2.5 M09
N8 G28 Z3.0 M05
N9 M30
%
```

A fixed cycle could be used for multiple holes and other improvements added, but the program is correct as is.

In the next example, the same hole will be machined a little differently. This program will use *two* tools rather than just one - a  $\varnothing 10$  mm standard drill (T01) and a  $\varnothing 10$  mm center cutting end mill (T02). Required finished depth is Z-25.0, at the flat bottom - see Figure 26-13:

## O2603 (FLAT BOTTOM - VERSION 2)

(T01 - 10 MM STANDARD DRILL)

```

N1 G21
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X.. Y.. S850 M03 T02
N5 G43 Z2.5 H01 M08
N6 G01 Z-24.9 F200.0 (TOOL TIP AT Z-24.9)
N7 G00 Z2.5 M09
N8 G28 Z2.5
N9 M01
```

(T02 - 10 MM CENTER CUTTING END MILL)

```

N10 T02
N11 M06
N12 G90 G54 G00 X.. Y.. S700 M03 T01
N13 G43 Z2.5 H02 M08
N14 G01 Z-21.0 F300.0 (1 MM CLEARANCE)
N15 Z-25.0 F175.0 (FULL DEPTH COMPLETED)
N16 G04 X0.5
N17 G00 Z2.5 M09
N18 G28 Z2.5 M05
N19 M30
%
```

There are three blocks of special interest in program O2603. The first block is N6, indicating the standard drill depth. Selected drill stops short of the full depth by 0.1 mm. Z-24.9 is programmed instead of the expected Z-25.0. A little experiment as to *how* short may be worth it. A reason for *not* drilling to the full depth with the standard drill is to prevent possible dimple mark at the hole center.

Two equally important blocks appear in the second tool of the program - blocks N14 and N15. In block N14, the end mill feeds at a heavier feedrate to the depth of only 21 mm, providing a 1 mm clearance. That makes sense, as there is nothing to cut for the end mill for full 22 mm. Follow calculation process of the 21 mm intermediate depth from this procedure - the illustration shows all important details.

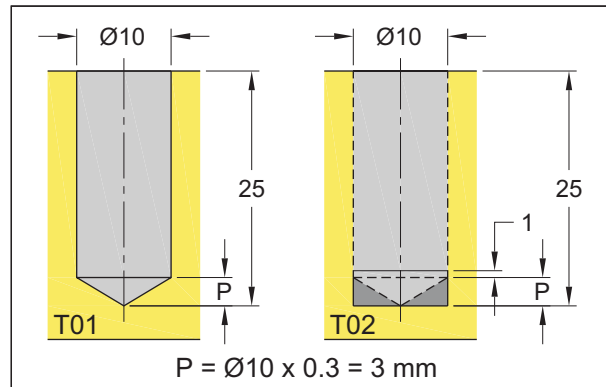


Figure 26-13

Illustration for program O2603 - two tools used

From the total depth of 25 mm cut by standard drill T01, subtract the length of the tool point P. That is 3 mm for a  $118^\circ$  drill point angle and  $\varnothing 10$  mm drill. The result is 22 mm. From the result, subtract 1 mm for clearance, and the result is Z-axis value of Z-21.0. In block N15, the end mill removes all excessive material left by T01, at a suitable cutting feedrate, usually programmed at a slower rate.

From machining viewpoint, programming a center drill or a spot drill first, to start up the hole, will add important positioning accuracy. This extra operation will also guarantee concentricity for both the standard drill and end mill. Another option is to use a flat bottom drill, but this is not a suitable CNC tool. Properly selected end mill is generally more rigid and can do the job much better.

## ◆ Indexable Insert Drilling

One of the great productivity improvement tools in modern machining is an indexable insert drill. This drill uses carbide inserts, just like many other tools for milling or turning. It is designed to drill holes in a *solid* material. It does *not* require center drilling or spot drilling, it is used with high spindle speeds and relatively slow feedrates and is available in a variety of sizes (metric or imperial). In most cases, it is used for *through* holes, although blind holes can be drilled as well. This type of a drill can even be used for some light to medium boring or facing.

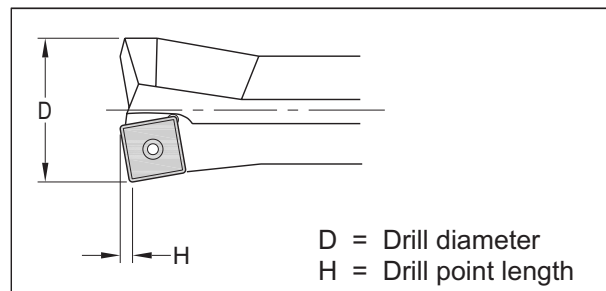


Figure 26-14

Cutting end of a typical indexable insert drill

Design of an indexable insert drill is very precise, assuring constant tool length, as well as elimination of re-grinding dull tools. *Figure 26-14* shows the cutting portion of a typical indexable drill.

In the illustration, drill diameter  $D$  controls hole size. Drill tip point length  $H$  is defined by the drill manufacturer and its amount is listed in tooling catalogues. For example, an indexable drill with the  $D$  diameter of 1.25, may have the  $H$  tip length 0.055. Indexable drills can be used for rotary and stationary applications, vertically or horizontally, on machining centers or lathes. For best performance, the coolant should be pressure fed *through* the drill, particularly for tough materials, long holes, and horizontal operations. Coolant not only disperses generated heat, it also helps flush out chips. When using an indexable insert drill, make sure there is enough power at the machine spindle. The power requirements at spindle increase proportionally with increased drill diameters.

On a machining center, indexable drill is mounted in machine spindle, therefore it becomes a rotating tool. In this setup, the drill should be used in a rigid spindle that runs true - no more than 0.010 inch (0.25 mm) of TIR (*Total Indicator Reading*). On spindles that have a quill, try to work with the quill *inside* the spindle, or extend it as little as possible. Coolant provisions may include an internal coolant, and special adapters are available for through the hole cooling, when the drill is used on machining centers.

On lathes, indexable drilling tool is always stationary. Correct setup requires that the drill is positioned on the center and be concentric with spindle centerline. The concentricity should not exceed 0.005 inch (0.127 mm) of TIR.

Always exercise care when drilling operation starts on a surface that is *not* flat - *Figure 26-15*. For best results, use indexable drills on surfaces that are  $90^\circ$  to the drill axis (flat surfaces). Within certain limits, the drill can also be used to

enter or exit an inclined, uneven, concave, or convex surface quite successfully. Cutting feedrate may need to be reduced for duration of any interrupted cut. Illustration shows areas where the feedrate should be slower. The letter  $F$  identifies an area that is machined using *normal* feedrate (normal entry/exit), and  $F/2$  indicates an area that requires a *reduced* feedrate. For reduced feedrate, programming one half of normal feedrate is usually sufficient.

In the illustration, frame  $a$  shows a tilted (inclined) surface entry,  $b$  frame shows an uneven surface, and frames  $c$  and  $d$  show convex and concave surfaces respectively.

An indexable drill should always be used  
in a fully protected machining area

Always watch for a flying disc when machining through holes !

## PECK DRILLING

Peck drilling is also called *interrupted cut drilling*. It is a drilling operation, using fixed cycles G83 (standard peck drilling cycle) or G73 (high speed peck drilling cycle). Difference between the two cycles is tool retraction method. In G83, the retract after each peck will be to the R-level (usually above the hole), in G73, there will only be a small retract (between 0.5 and 1 mm or 0.02 and 0.04 inches).

Peck drilling is often used for holes that are too deep to be drilled with a single tool motion. Peck drilling methods also offer several opportunities to improve any standard drilling techniques as well. Here are some possible uses of peck drilling methods for machining holes:

- Deep hole drilling
- Chip breaking and for short holes in tough materials
- Cleanup of chips accumulated on drill flutes
- Frequent cooling and lubricating of the drill cutting edge
- Controlling drill penetration through the material

In all cases, drilling motions of the G83 or G73 cycle produce an interrupted cut that can be programmed very simply by specifying a  $Q$ -amount in the cycle. This value specifies the actual depth of each peck. The smaller the  $Q$  value, the more pecks will be generated and vice versa. For most deep hole drilling jobs, the *exact* number of pecks is not important, but there are cases when either pecking cycle needs to be controlled.

### ◆ Typical Peck Drilling Application

For the majority of peck drilling applications, peck drilling depth  $Q$  needs to be only a *reasonable* depth. For example, a deep hole (with the depth at  $Z$ -2.125 inches at tool tip) is drilled with a 0.250 diameter drill and 0.600 peck depth. The G83 cycle may be programmed like this:

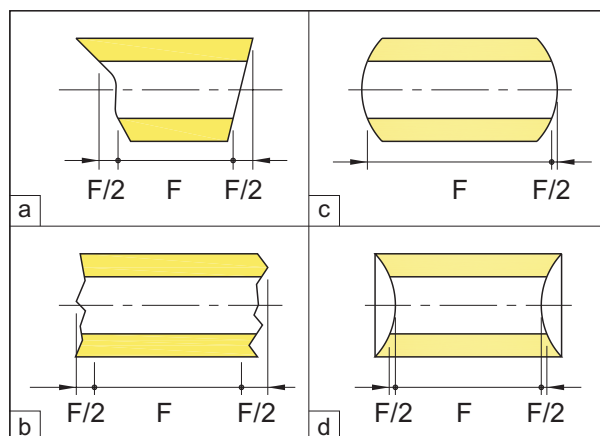


Figure 26-15

Uneven entry or exit surface for indexable drills feedrate:  
 $F$  = normal feedrate,  $F/2$  = reduced feedrate (one half of  $F$ )



N137 G99 G83 X.. Y.. R0.1 Z-2.125 Q0.6 F8.0

These programming values are *reasonable* for the job at hand - and that is all that matters. For most jobs, the number of pecks is usually not too important.

### ◆ Calculating the Number of Pecks

If the number of pecks G83 or G73 cycle will generate is important, it has to be *calculated*. Knowledge of how many pecks will result with a certain Q value for a given total depth is usually not important. If the program is running efficiently, there is no need for any modification. To find out how many pecks G83 or G73 cycle will generate, it is important to know the exact *total distance* the drill travels between R-level and Z-depth, as an incremental value. It is equally important to know the peck depth Q. This Q amount divided into the travel distance is the number of pecks:

$$P_n = \frac{T_d}{Q}$$

where ...

$P_n$  = Number of pecks  
 $T_d$  = Total tool travel distance  
 $Q$  = Programmed peck depth

For example, in the following G83 cycle (G20 mode),

N73 G99 G83 X.. Y.. R0.125 Z-1.225 Q0.5 F12.0

the total drill travel distance is 1.350, divided by 0.500, which yields 2.7. Since the number of pecks can only be a whole number, the nearest *higher* integer will be the actual number of pecks, in this case 3.

### ◆ Selecting the Number of Pecks

Much more common is programming of a *desired* number of pecks, particularly by experienced programmers. If a specific number of pecks will do a job efficiently, the amount of Q has to be calculated accordingly. Since the Q value specifies the *depth of each peck* and *not* the number of pecks, some simple math will be needed to select the depth Q, so it corresponds to the desired number of pecks.

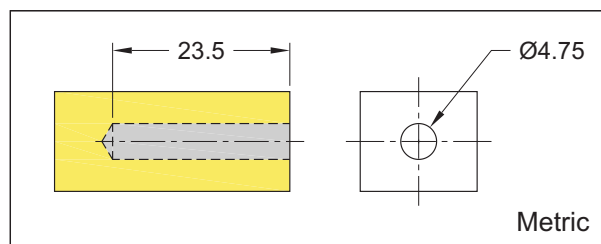


Figure 26-16

Peck drilling example - selected number of pecks

For example, *Figure 26-16* shows a simplified metric drawing for 4.75 mm drill. If *three pecks* are required in the following cycle - what will the Q depth be?

Drill point length is  $\varnothing 4.75 \times 0.3 = 1.425$  mm, so the final Z-depth will be Z-24.925, R-level will be 3 mm:

N14 G99 G83 X.. Y.. R3.0 Z-24.925 Q?? F120.0

The total drill travel from the R-level to the Z-depth is  $3.0 + 24.925 = 27.925$ . To calculate the peck depth Q, a new formula is similar to the previous one:

$$Q = \frac{T_d}{P_n}$$

where ...

$Q$  = Programmed peck depth  
 $T_d$  = Total tool travel distance  
 $P_n$  = Number of required pecks

Using the formula, result of  $27.925/3$  is 9.308333. With correct mathematical rounding to three decimal places, the Q depth is 9.308. Now, follow the individual peck depths to see what will happen:

Peck 1	9.308	accumulated depth	...	9.308
Peck 2	9.308	accumulated depth	...	18.616
Peck 3	9.308	accumulated depth	...	27.924
Peck 4	0.001	accumulated depth	...	27.925

There will be *four pecks* and the last one will only cut one micron - 0.001 mm (!) - or practically nothing at all. In those cases, where the last cut is very small and inefficient, always round the calculated Q value *upwards*, in this case to the minimum of 9.309 or higher:

N14 G99 G83 X.. Y.. R3.0 Z-24.925 Q9.309 F120.0

Always remember, the cutting tool will *never* go past the programmed Z-depth, but it could reach this depth in a very inefficient way that should always be corrected.

### ◆ Controlling Breakthrough Depth

Less frequent programming method, also very powerful, is to use peck drilling cycle to control the *breakthrough* of the drill through material, regardless of its drill size or material thickness. Here is some background. In many tough materials, when a drill starts penetrating the part bottom face (for a through hole), it creates potentially difficult machining conditions. The drill has a tendency to *push* the material out rather than cut it. This is most common when the drill is a little dull, material is tough, or the programmed feedrate is fairly high. These adverse conditions are also the result of heat generated at the drill edge, lack of lubrication reaching the drill cutting edge, worn-off flutes and several other factors.

A possible solution to this problem is to relieve the drill pressure when it is *about half way* through the hole, but *not* yet completely through - Figure 26-17.

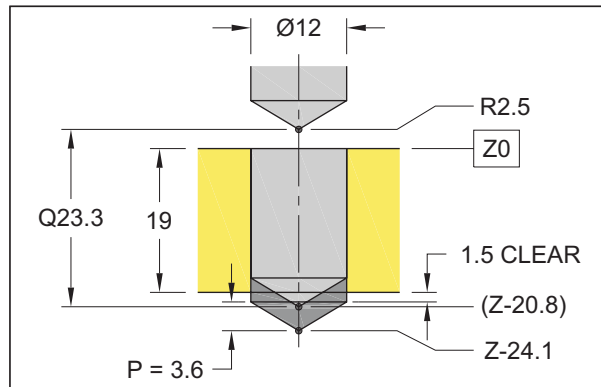


Figure 26-17

Controlled drill breakthrough using a peck cycle

Peck drilling cycle G83 is an excellent cycle for such use, but the Q depth calculation is extremely important. Actual number of pecks is not important, only the last two are critical for this purpose. To control a problem associated with the drill breakthrough, only two peck motions are needed. The illustration shows these two positions for a Ø12 mm drill drill through a 19 mm thick plate.

For most jobs, such a hole requires no special treatment. Just one cut through (using G81 cycle) and no peck drilling. Let's evaluate the solution to this particular situation:

A Ø12 mm drill has a point length of  $12.0 \times 0.3 = 3.6$  mm. Take one half (1.8 mm) of the drill point length as the first penetration amount (it does not have to be one half), which will bring the drill 1.8 mm *below* the 19 mm plate thickness, to absolute Z-depth of Z-20.8.

#### *This depth is controlled by the amount of Q depth*

Keep in mind that Q depth is an *incremental* value, measured from the R-level, in this case R2.5. That makes the Q-depth as Q23.3 (2.5 mm above and 20.8 mm below Z0). Programmed Z-depth is the final drill depth. If 1.5 mm clearance is added below the plate thickness, the final Z-depth will be sum of the plate thickness (19 mm), breakthrough clearance (1.5 mm) and the drill point length (3.6 mm), for the programmed value of Z-24.1:

G99 G83 X.. Y.. R2.5 Z-24.1 Q23.3 F..

This rather uncommon technique does not only solve a particular job related problem, it also shows how *creativity* and *programming* are complementary terms.

## REAMING

Reaming operations are very close to drilling operations, at least as far as the programming method is concerned. While a drill is used to *make* a hole (to open up the hole), a reamer is used to *enlarge* an existing hole.

Reamers are either cylindrical or tapered, usually designed with more than two flutes of different configurations. Reamers made of high speed steel, cobalt, carbide and with brazed carbide tips. Each reamer design has its advantages and disadvantages. Carbide reamer, for example, has a very high resistance to wear, but may be not economically justified for every hole. A high speed steel reamer is economical, but wears out much faster than a carbide reamer. Many jobs do not accept any compromise in selection of tools and the cutting tool has to be selected correctly for a given job. Sizing and finishing tools, such as a reamer, have to be selected even more carefully.

Reamer is a *sizing* tool and is not designed for removal of heavy stock. During a reaming operation, an existing hole will be *sized* - reamer will *size* an existing hole to close tolerances and add a high quality surface finish. Reaming itself will *not* guarantee hole concentricity. For holes requiring *both* high concentricity *and* tight tolerances, center drill or spot drill the hole first, then drill it the normal way, then rough bore it and only then finish it with a reamer.

A reaming operation will require a coolant to help make a better quality surface finish and to remove chips during cutting. Standard coolants are quite suitable, since there is not very much heat generated during reaming. Coolant also serves in an additional role, to flush away chips from the part and to maintain surface finish quality.

### ◆ Reamer Design

In terms of design, there are two features of a reamer that have a direct relationship to CNC machining and programming. The first consideration is the *flute design*.

Most reamers are designed with a left-hand flute orientation. This design is suitable to ream *through* holes. During a cut, the left-hand flute design 'forces' chips to the bottom of the hole, into an empty space. For blind holes that have to be reamed, a left-hand type of a reamer may not be suitable.

The other factor of reamer design is the *end chamfer*. In order to enter an existing hole that may still be without a chamfer, a lead-in allowance is required. The reamer end provides that allowance. Some reamers also have a short taper at their tip, for the same purpose. The chamfered lead is sometimes called a '*bevel lead*' and its chamfer an '*attack angle*'. Both have to be considered in programming.

### ◆ Spindle Speeds for Reaming

Just like for standard drilling and other operations, the spindle speed selected for reaming must be closely related to the type of material being machined. Other factors, such as part setup, its rigidity, its size and surface finish of the completed hole, etc., each contributes to the spindle speed selection.

As a general programming rule, the spindle speed for reaming will be reasonable if you use a modifying factor of 0.660 (2/3), based on the drilling speed used for the *same* material. For example, if a speed of 500 r/min produced good drilling conditions, the two thirds (0.660) of that speed will be *reasonable* for reaming:

$$500 \times 0.660 = 330 \text{ r/min}$$

Do not program a reaming motion in the reversed spindle rotation - the cutting edges may break or become dull.

### ◆ Feedrates for Reaming

The reaming feedrates are generally programmed higher than those used for drilling. *Double* or *triple* increases are not unusual. The main purpose of high feedrates is to force the reamer *to cut*, rather than to rub the material. If the feedrate is too slow, the reamer wears out rapidly. Slow feedrates cause heavy pressures as the reamer actually tries to enlarge the hole, rather than remove the stock.

### ◆ Stock Allowance

Stock is the amount of material left for finishing operations. A hole to be reamed must be smaller (*undersize*) than the pre-drilled or pre-bored hole - a logical requirement. Programmer decides *how much* smaller. A stock too small for reaming causes the premature reamer wear. Too much stock for reaming increases the cutting pressures and the reamer may break.

A good general rule is to leave about 3% of the reamer diameter as the stock allowance. This applies to the *hole diameter - not per side*. For example, a Ø3/8 inch reamer (Ø0.375), will work well in most conditions if the hole to be reamed has a diameter close to 0.364 inches:

$$0.375 - (0.375 \times 3 / 100) = 0.36375 = 0.364$$

Most often, a drill that can machine the required hole diameter exactly will not be available. That means using a boring bar to *presize* such hole before reaming. It also mean an extra cutting tool, more setup time, longer program and other disadvantages, but the hole quality will be worth the effort. In these cases, for tough materials and some 'space age' materials, the stock allowance left in the hole for reaming is usually decreased.

### ◆ Other Reaming Considerations

General approach for reaming is no much different than for other hole operations. When drilling a blind hole, then reaming it, it is inevitable that some chips from the drilling remain in the hole and may prevent a smooth reaming operation. Using program stop function M00 *before* the reaming operation allows the operator to remove all chips first, for a clear entry of the reamer.

The reamer size is always important. Reamers are often made to produce either a *press fit* or a *slip fit*. These terms are nothing more than machine shop expressions for certain tolerance ranges applied to a reamed hole.

Programming a reaming operation requires a fixed cycle. Which cycle will be the most suitable? There is no *reaming cycle* defined directly. Thinking about traditional machining applications, the most accepted reaming method is the *feed-in and feed-out* method. This method requires a feedrate motion to remove the material from the hole, but it also requires a feedrate motion back to the starting position, to maintain hole quality - its size and surface finish. It may be tempting to program a rapid motion out of a reamed hole to save cycle time, but often at the cost of quality. For the best machining, *feeding out* of a reamed hole is necessary. Suitable fixed cycle available for the Fanuc and similar controls is G85, which permits *feed-in* and *feed-out* motions, with no dwell at the bottom. If dwell at the hole bottom is required, G89 cycle is the choice. Cutting feedrate of the cycle will be *the same* for both motions. Any feedrate change will affect both motions - *in* and *out*.

## SINGLE POINT BORING

Another sizing operation on holes is called *boring*. Boring holes is a point-to-point operation along the Z axis only, typical to CNC milling machines and machining centers. It is also known as '*single point boring*', because the most common tool is a boring bar that has only *one* cutting edge. Boring on CNC lathes is considered a *contouring* operation and is not covered in this chapter (see *Chapters 34-35*).

Jobs requiring precision holes that have previously been done on a special jig boring machine can be done on a CNC machining center, using single point boring tools. Modern CNC machines are manufactured to high accuracy, particularly for positioning and repeatability - a proper boring tool and its application can produce high quality holes.

### ◆ Single Point Boring Tool

As for its practical purpose, a single point boring is a *finishing*, or at least a *semifinishing*, operation. Its main job is to enlarge - or *to size* - a hole that has been drilled, punched or otherwise cored. Boring tool works on diameter of the hole and its purpose is to produce the desired hole diameter being square and round in relation to other part features.

Although there is a variety of designs of boring tools on the market, the single point boring tool is usually designed for *cartridge* type inserts. These inserts are mounted at the end of the holder (*i.e.*, a boring bar) and usually have a built-in micro adjustment for fine tuning the effective boring diameter - *Figure 26-18*.

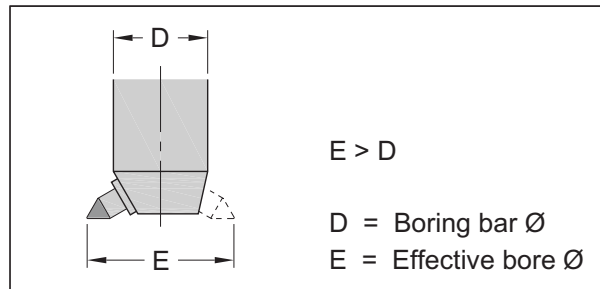


Figure 26-18

Effective diameter of a single point boring bar

The same programming techniques are applied to the boring bars of other designs, for example, a *block tool*. A block tool is a boring bar with *two* cutting edges, 180° apart. If the diameter adjusting mechanism is *not* available on the tool holder, the effective boring diameter must be *preset*, using either a special equipment or the slow but true and tried *trial-and-error* method. This trial and error setup is not that unusual, considering the setup methods that are available for a single point boring bar.

Just like any other cutting tool, a single point boring bar achieves best cutting results if it is short, rigid and runs concentric with the spindle centerline. One of the main causes of poorly bored holes is *boring bar deflection*, applying equally to milling and turning. The tool tip (usually a carbide bit), should be properly ground, with suitable cutting geometry and overall clearances. Position of the boring bar in the spindle - or its orientation - is also very important for many boring operations on machining centers.

### ◆ Spindle Orientation

Any round tool, such as a drill or an end mill, can enter or exit a hole along the Z axis, with little programming considerations for the final hole quality. Neither of these tools is used for holes that demand high quality surface finish and close tolerances. With boring, the hole surface integrity is *very important*. Many boring operations require that a cutting tool does not damage the hole surface during retract. Since retracting from a hole almost always leaves some marks inside the hole, special methods of retract must be used. There is one such method - it uses cycle G76 (or rarely G87) with *spindle orientation* feature of the machine and a *shift* of the boring tool away from finished surface. Spindle orientation feature was already described earlier (*page 84*), so this is just a reminder now.

The sole purpose of spindle orientation is to replace one tool holder for another in exactly the same position after each tool change. Without spindle orientation, the tool tip will stop at a random position of its circumference. Orienting the spindle for boring purposes is only one half of the solution. The other is setting position of the boring bit. This is usually a responsibility of CNC operator, since it has to be done during setup at the machine. Boring bar cutting bit must be set in such a way that when the shift takes place in fixed cycle G76 or G87, it will be into the direction *away* from the finished hole wall, ideally by XY vectors relative to the spindle orientation angle - *Figure 26-19*.

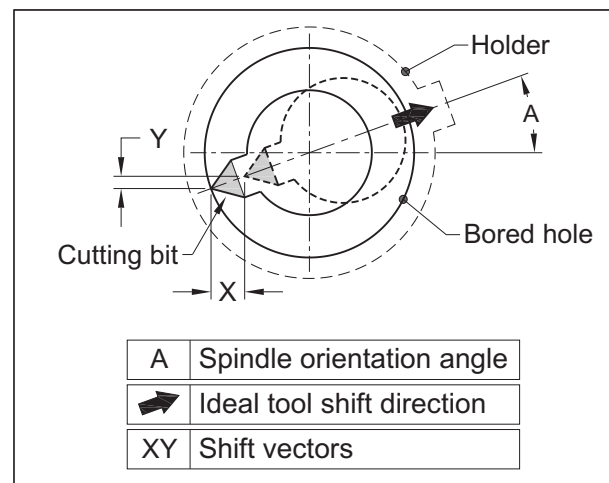


Figure 26-19

Single point boring bar, shift vectors, and spindle orientation angle

Spindle orientation is always factory designed and *fixed*. CNC programmer must always consider shift amount and its direction.

When machine spindle is oriented, it must be in a stopped mode. The spindle cannot rotate during any machining operation that requires a spindle shift. Review descriptions of the precision boring fixed cycle G76 (*page 193*) and the backboring cycle G87 (*page 192*). Machine operator *must* always know which way the spindle orients *and* into which direction the tool shift actually moves. Also keep in mind that the programmed shift can use either the Q-amount *or* the XY vectors, but not both. Programming selection is based on a system parameter setting.

Programming a bored hole that will be reamed later requires the boring bar only to assure concentricity and straightness of the finished hole. Surface finish of the bored hole is not too important. If boring is the *last* machining operation in the hole, the chances are that surface finish *will* be very important. It is difficult to retract a boring tool without leaving drag marks on the hole cylindrical surface. In that case, select a suitable fixed cycle - the precision boring cycle G76 may be the best choice.



### ◆ Block Tools

When using a *single* point boring bar for roughing or semifinishing operations, there is an option that is more efficient. This option also uses a boring tool, but one that has *two* cutting edges (180° opposite) instead of one - it is called a *block tool*. Block tools cannot be used for fine finishing operations, because they cannot be *shifted*. The only way of programming a block tool is within the 'in-and-out' tool motion. Several fixed cycles support this kind of motion. All motions *into the hole* are at a specified feedrate. On the way *out of the hole*, some motions are feedrates, others are rapid, depending on the cycle selection. The cycles that can be used with block tools are G81 and G82 (feed-in-rapid-out), as well as G85 and G89 that feeds in and feeds out while the machine spindle is rotating and another cycle, G86, when the tool retracts while the spindle is not rotating.

The greatest advantage of a block tool is the increased feedrate that can be programmed for this tool. For example, if the feedrate for a single point tool is 0.18 mm (0.007") per flute, for a block tool it will be at least double, 0.36 mm (0.014") per flute or more. Block tools are generally available in diameters from about Ø20 mm (0.75-0.80") and up.

## BORING WITH A TOOL SHIFT

There are two fixed cycles that require the boring tool to shift away from the centerline of current hole to avoid contact with the material. These cycles are boring cycles G76 and G87. G76 is by far the most useful and both are used together in program example O2604.

### ◆ Precision Boring Cycle G76

The G76 cycle is commonly used for holes that require high quality of size and surface finish. The operation boring itself is quite normal, however, tool retract from the hole is rather special. When the boring bar stops at the hole bottom in oriented position, it shifts away by the Q amount programmed in the cycle and retracts back to the starting position, where it shifts back to its normal position.

G76 cycle has been described in detail in the previous chapter (page 193). This chapter contains a practical programming example, showing a single hole XY location - in Figure 26-20 - Ø25 mm is the first bore size.

From the drawing, only the 25 mm hole is considered at the moment - program input will be quite simple:

```
N.. G99 G76 X0 Y0 R2.0 Z-31.0 Q0.25 F125.0
```

A hole bored with G76 cycle will have a high quality, but actual tool setting and supplied hole data must also be well selected. Note the Q-amount is rather small at 0.25 mm (0.01") - compare it with the Q-amount for the next tool.

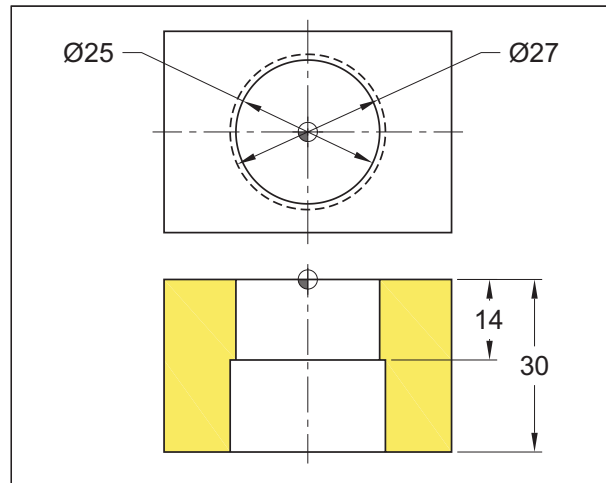


Figure 26-20

Sample drawing for G76 and G87 programming example O2604

### ◆ Backboring Cycle G87

Although the backboring cycle has some applications, it is not a common fixed cycle. As the name suggests, it is a boring cycle that works in the *reverse* direction than all other cycles - *from the back of the part*. Typically, the backboring operation starts at the hole bottom, which is the 'back face of the part', and the boring proceeds from the bottom upwards, in Z positive direction.

The G87 cycle has been described in the previous chapter. The Figure 26-20 also shows a diameter of 27 mm, which will be bored during the same setup as the 25 mm hole. This larger diameter is at the 'back side of the part', and it will be backbored, using the G87 cycle.

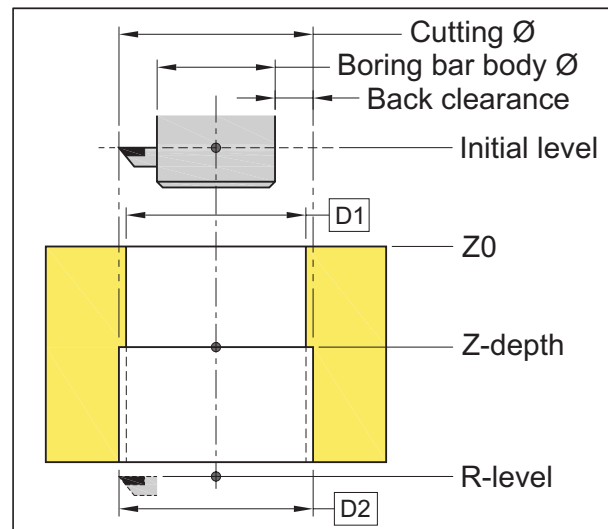


Figure 26-21

Setup considerations for a backboring tool (program O2604)



Figure 26-21 shows the boring bar setup that will backbore the 27 mm hole, from the hole bottom, upwards. Pay a close attention to individual descriptions.

In the illustration, D1 represents the smaller hole diameter (25 mm), and D2 represents the diameter of the hole to be backbored (27 mm). Within certain limits, D2 must always be larger than D1, otherwise no backboring can take place. Always make sure there is enough clearance for the body of the boring bar within the hole and at the hole bottom. *Backboring tool must always fit into an existing hole.*

### ◆ Programming Example

In order to show a complete program for reference, four tools will be used - *spot drill* (T01), *drill* (T02), *standard boring bar* (T03) and a *back boring bar* (T04). Program number is O2604.

```
O2604 (G76 AND G87 BORING)
(T01 - 15 MM DIA SPOT DRILL - 90 DEG)
N1 G21
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X0 Y0 S1200 M03 T02
N5 G43 Z10.0 H01 M08
N6 G99 G82 R2.0 Z-5.0 P100 F100.0
N7 G80 Z10.0 M09
N8 G28 Z10.0 M05
N9 M01

(T02 - 24 MM DIA DRILL)
N10 T02
N11 M06
N12 G90 G54 G00 X0 Y0 S650 M03 T03
N13 G43 Z10.0 H02 M08
N14 G99 G81 R2.0 Z-39.2 F200.0 (2 MM BELOW)
N15 G80 Z10.0 M09
N16 G28 Z10.0 M05
N17 M01

(T03 - 25 MM DIA STANDARD BORING BAR)
N18 T03
N19 M06
N20 G90 G54 G00 X0 Y0 S900 M03 T04
N21 G43 Z10.0 H03 M08
N22 G99 G76 R2.0 Z-31.0 Q0.25 F125.0 (25 DIA)
N23 G80 Z10.0 M09
N24 G28 Z10.0 M05
N25 M01

(T04 - 27 MM DIA BACK BORING BAR)
N26 T04
N27 M06
N28 G90 G54 G00 X0 Y0 S900 M03 T01
N29 G43 Z10.0 H04 M08
N30 G98 G87 R-32.0 Z-14.0 Q1.3 F125.0 (27 DIA)
N31 G80 Z10.0 M09
N32 G28 Z10.0 M05
N33 G28 X0 Y0
N34 M30
%
```

Make sure to follow all rules and precautions when programming or setting up a job with G76 or G87 fixed cycles in the program - Many of them are safety oriented

### ◆ Precautions in Programming and Setup

Main precautions for boring with tool shift relate to a few special considerations that are necessary for successful application of the two cycles - G76 and G87. The following list sums up the most important precautions:

- Through boring must be completed before backboring
- The first boring cycle (G76) must be programmed all the way through the hole, never partially
- For G76 cycle, only a minimum Q value is required (ex., 0.25 mm or 0.01 inches)
- For G87 cycle, the Q value must be *greater* than one half of the difference between the two diameters:  

$$(D2 - D1) / 2 = (27 - 25) / 2 = 1$$
*plus* the standard minimum Q value (ex., 0.3 mm)
- Always watch for the boring bar body, so it does not hit the hole surface during the shift. This can happen with large boring bars, small holes, or a large shift amount
- Always watch the boring bar body, so it does not hit an obstacle *below* the part. Remember that the tool length offset is measured to the cutting edge, not to the actual bottom of the boring tool
- G87 is always programmed in G98 mode, *never* in G99 mode !!!
- Always know the shift direction and set the tool properly

## ENLARGING HOLES

Any existing hole can also be enlarged from the top of part. To enlarge an existing hole at its top, we can use one of three methods that will enlarge previously machined hole - these methods are common in every machine shop. They are:

- Countersinking C'SINK or CSINK on drawings
- Counterboring C'BORE or CBORE on drawings
- Spotfacing SF, S.F., or S/F on drawings

All three machining methods will enlarge an existing hole, with one common purpose - they will allow the fitting part to be accurately seated in the hole by creating a clean surface. For example, a bolt head that has to be seated on a flat surface will require countersinking or spotfacing operation. All three operations require a perfect alignment with the existing hole (concentricity). Programming technique is basically the same for all three operations, except for the actual tool used. Speeds and feeds for these tools are usually lower than for drills of equivalent size. Any hole to be enlarged must exist prior to these operations.

### ◆ Countersinking

Countersinking is an operation that enlarges an existing hole in a conical shape, to a required depth. Countersinking is used for holes that have to accommodate a conical bolt head. From all three similar operations, countersinking requires the most calculations for precision depth. Typical countersinks have three angles:

- 60 degrees
- 82 degrees - this is the most common CSINK angle
- 90 degrees

Other angles are also possible, but less frequent.

To illustrate the programming technique and the required calculations, the cutting tool used must be known first. *Figure 26-22* shows a typical countersinking tool.

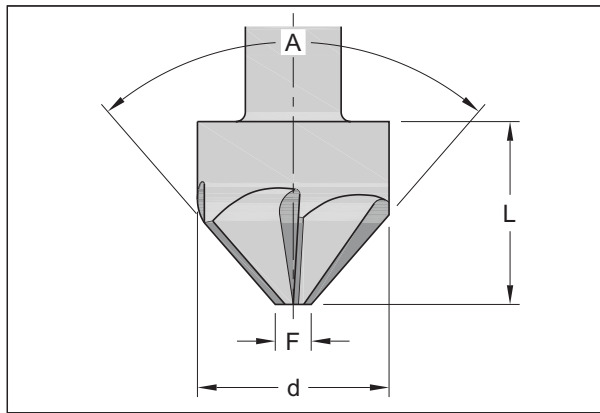


Figure 26-22

Typical dimensions of a countersinking tool

In the illustration,  $d$  is the countersink body diameter,  $A$  is the countersink angle,  $F$  is the tool flat diameter (equal to zero for a sharp end),  $L$  is the body length.

Programming a countersink operation requires specific given data in the drawing. This information is often provided through a description (leader/text), for example:

**0.78 DIA CSINK - 82 DEG**  
**13/32 DRILL THRU**

There is one programming issue - the specified CSINK diameter must be accurate. That is the  $\varnothing 0.78$  in the description, with countersink angle of  $82^\circ$ . A precise diameter can only be *created* by carefully calculating the Z-depth. Previously discussed constant values  $K$  for tool point length can be used (*see page 202*). Then, calculate the cutting depth in a manner similar to drills. One problem here is that the constant  $K$  for a drill point always assumes a sharp point at the tool tip. Countersinking tools do not always have a sharp point (except for some small sizes). Instead, they have a flat diameter  $F$ , normally specified in tooling catalogues.

*Figure 26-23* illustrates an example of countersink data, as shown in a typical drawing.

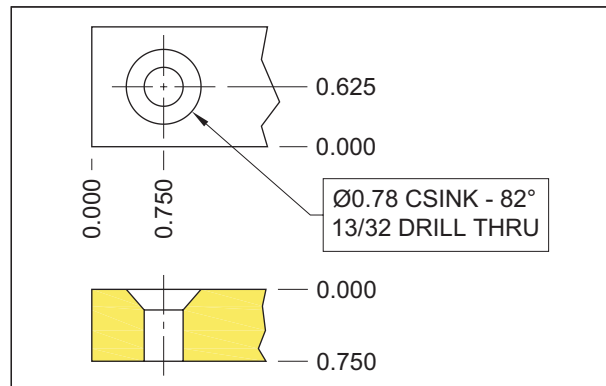


Figure 26-23

Programming example for a standard countersinking operation

*Figure 26-24* shows provided dimensions  $E$ ,  $A$ , and  $F$  and unknown  $P$  and Z-depth countersinking dimensions required for depth programming of a countersinking tool.

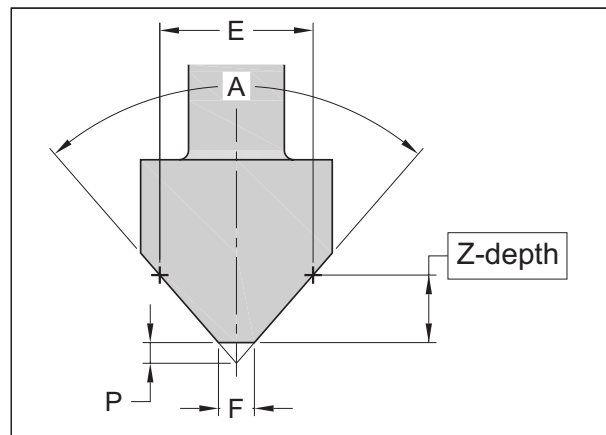


Figure 26-24

Z-depth calculation for a typical  $82^\circ$  countersink +  $P$  height  
Effective CSINK diameter  $E$ , angle  $A$ , and flat  $\varnothing F$  are known

The actual process of calculation is simple enough. First, determine the height  $P$  for a given flat diameter  $F$ . Use the standard constants as applied to a drill point length:

$$\begin{aligned} 60^\circ &= 0.866 \\ 82^\circ &= 0.575 \\ 90^\circ &= 0.500 \end{aligned}$$

In the illustration,  $E$  is the effective countersink diameter,  $A$  is the countersink angle,  $F$  is the flat diameter,  $P$  is the height of sharp point, and the Z-depth is the main goal - programmed tool depth. In this case, angle  $A$  is  $82^\circ$ , and flat diameter  $F$  as per tooling catalogue is  $3/16$  (0.1875). The height of sharp end  $P$  can now be calculated:

$$P = \varnothing 0.1875 \times 0.575 \quad (\text{K for } 82 = 0.575)$$

$$P = 0.1078$$

Actual programmed Z-depth for a countersink *with*  $F = 0$  (sharp point end) will ignore the  $P$  height and will be:

$$\text{Z-depth} = \varnothing 0.78 \times 0.575 = 0.4485$$

Since this depth *includes* the sharp point height  $P$ , all that has to be done to find out the real programmed Z-depth, is to *subtract* the  $P$  amount from the theoretical Z-depth:

$$\text{Z-depth} = 0.4485 - 0.1078 = 0.3407 = \text{Z-0.3407}$$

This is the final *programmed* Z-depth - program block for the countersink example may look something like this:

N35 G99 G82 X0.75 Y0.625 R0.1 Z-0.3407 P200 F8.0

Incidentally, the R-level could be *lowered* a bit, since there is a through hole already machined in the previous operation. Be careful here, the R-level will most likely be negative. Always program the G98 command and a small initial level, for example, Z0.1:

N34 G43 Z0.1 H03 M08 (0.1 IS INITIAL LEVEL)  
N35 G98 G82 X0.75 Y0.625 R-0.2 Z-0.3407 P200 F8.0

DO NOT make the R-level amount too deep !

### ◆ Maximum Spot Drill Depth for CSINK

Many holes to be countersunk will not start with a fully drilled hole - they will start with a  $90^\circ$  *spot drill* first - in this case, not for its chamfering capabilities, but to establish accurate XY location. Considering the dimensional differences between these two tools - particularly the difference in their included tool tip angle  $A$  - it is important to understand one important observation:

Spot drill with a larger tip angle may cut deeper than a countersink with a smaller tip angle

This is an important consideration for spot drilling holes that are to be countersunk. Since the spot drilling tool has a larger tip angle -  $90^\circ$  compared to  $82^\circ$  - it must *never* cut deeper than the result of a simple calculation:

$$D_s = E / 2$$

where...

$D_s$  = Spot drill depth  
 $E$  = Effective countersink diameter

Failure to adhere to this reality will result in an unwanted chamfer - *on the countersink!*

### ◆ Counterboring

Counterboring is an operation that enlarges an existing hole in a cylindrical shape to the required depth. Counterboring is used for holes that have to accommodate a round bolt head. It is often used on uneven or rough surfaces, or surfaces that are not at  $90^\circ$  to the bolt assembly. As for the proper tool selection, use a counterboring tool specially designed for this type of machining, or a suitable end mill instead. In either case, the program uses G82 fixed cycle. Since the counterbore depth is always given, there are no extra calculations required. Figure 26-25 shows a typical counterboring description.

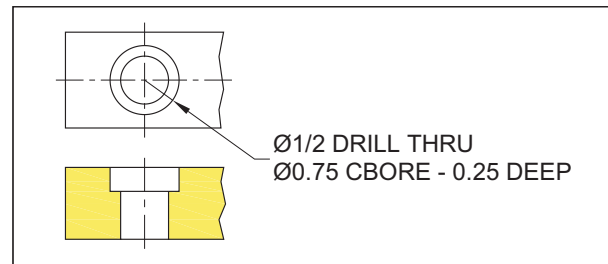


Figure 26-25

Programming example of a counterboring operation

For the example, the  $\varnothing 1/2$  inch hole had been machined earlier. Counterbore program block will be quite simple:

N41 G99 G82 X... Y... R0.1 Z-0.25 P300 F5.0

In counterboring, if a relatively slow spindle speed and fairly heavy feedrates are used, make sure the dwell time  $P$  in G82 cycle is sufficient. The rule of thumb is to program double value of minimum dwell. Minimum dwell  $D_m$  is:

$$D_m = \frac{60}{r / \text{min}}$$

For example, if the spindle speed is programmed as 600 r/min, the minimum dwell will be  $60/600=0.1$ , and doubled to 0.2 in the program, as P200. Doubling the minimum dwell value guarantees that even at 50% spindle speed override, there will be *at least* one full spindle revolution that cleans up the bottom of the counterbored hole. Some programmers choose to use a slightly longer dwell time, for more than one or two revolutions at the bottom.

### ◆ Spotfacing

Spotfacing is virtually identical to counterboring, except that the depth of cut is very minimal. Often, spotfacing is called *shallow counterboring*. Its purpose is to remove just enough material to provide a flat surface for a head of a bolt, a washer, or a nut. Programming technique is exactly the same as that for counterboring.

## MULTI Z-DEPTH DRILLING

On many occasions, the same cutting tool has to be programmed to move up and down between different heights (steps on a part) - it has to frequently change Z-depth and even R-level. For example, a drill will cut holes that have the same depth, but start at different heights.

This kind of programming requires two major conditions - the tool *should be* programmed *efficiently* (no time loss) and *must be* programmed *safely* (no collision).

Handling such programming problem is not difficult, once all available options are evaluated. These options are two preparatory commands - G98 and G99, used with fixed cycles exclusively. G98 command will cause the cutting tool to return to the *initial level*, while the G99 command will cause the cutting tool to return to the *R-level*. In practical programming, G98 command is used only in cases where an *obstacle between holes* has to be bypassed.

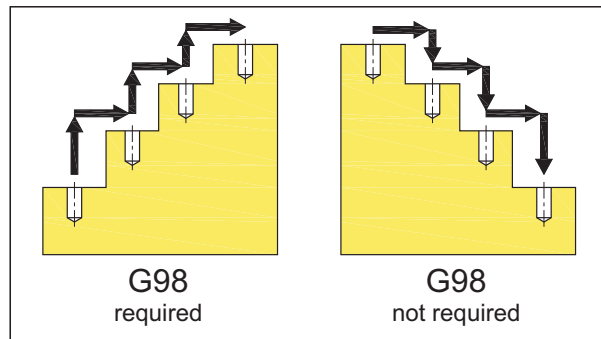


Figure 26-26

Tool motion direction between holes at different Z-depths

Figure 26-26 illustrates two programming possibilities, in a symbolic representation. The front view of a stepped part shows the Z-motion tool direction between holes. On the left, tool motion from one hole to the next could cause a collision with the wall and G98 is required for safety. On the right - with no obstacles - G98 is not required, and G99 can be used. Setting for the initial level is usually done in the G43 block, where the Z-amount must represent a clear tool location above *all* obstacles. A different, but much more practical example of this technique is illustrated in Figure 26-27 along with a corresponding program O2605 (G21 mode).

Only two tools are used for this example - T01 is a 90° spot drill that also makes a chamfer of 0.75 mm × 45° - it will be cutting to the depth of 5.75 mm below *each step face* ( $\varnothing 10 \text{ mm} / 2 + 0.75 \text{ mm} = 5.75$ ). T02 is a  $\varnothing 10 \text{ mm}$  drill *through* the hole, programmed to the absolute depth of Z-49.5 (45 mm thickness + 1.5 mm breakthrough +  $\varnothing 10 \text{ mm} \times 0.3 \text{ point length} = 49.5$ ). R-level is 2.5 mm above *any* actual face for both tools:

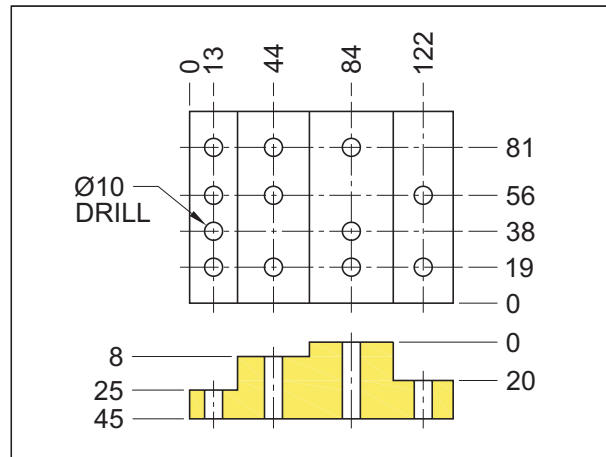


Figure 26-27

Multi Z-depth drilling - drawing for program example O2605

### O2605 (MULTI Z-DEPTH EXAMPLE)

```

N1 G21
N2 G17 G40 G80 T01
N3 M06 (T01 - 20 MM SPOT DRILL)
N4 G90 G54 G00 X13.0 Y19.0 S1200 M03 T02
N5 G43 Z10.0 H01 M08
N6 G99 G82 R-22.5 Z-30.75 P200 F200.0
N7 Y38.0
N8 Y56.0
N9 G98 Y81.0
N10 G99 X44.0 R-5.5 Z-13.75
N11 Y56.0
N12 G98 Y19.0
N13 G99 X84.0 R2.5 Z-5.75
N14 Y38.0
N15 Y81.0
N16 X122.0 Y56.0 R-17.5 Z-25.75
N17 Y19.0
N18 G80 Z10.0 M09
N19 G28 Z10.0 M05
N20 M01

N21 T02
N22 M06 (T02 - 10 MM DRILL THRU)
N23 G90 G54 G00 X122.0 Y19.0 S900 M03 T01
N24 G43 Z10.0 H02 M08
N25 G99 G83 R-17.5 Z-49.5 Q15.0 F250.0
N26 G98 Y56.0
N27 G99 X84.0 Y81.0 R2.5
N28 Y38.0
N29 Y19.0
N30 X44.0 R-5.5
N31 Y56.0
N32 Y81.0
N33 X13.0 R-22.5
N34 Y56.0
N35 Y38.0
N36 Y19.0
N37 G80 Z10.0 M09
N38 G28 Z10.0 M05
N39 M30
%
```

Study the program carefully. Watch the direction of tools - T01 starts at the lower left hole and ends at the lower right hole, in a zigzag motion. T02 starts at the lower right hole and ends at the lower left hole, also in a zigzag motion. Note there is one more G98 command for the first tool than for the second tool because of the two steps 'up'. In multi Z-depth hole machining you should understand three areas of program control, all used in program example O2605:

- G98 and G99 control
- R-level control
- Z-depth control

## WEB DRILLING

Web drilling is a common term for a drilling operation that takes place between two or more parts, separated by an empty space. Programming challenge in this case is to machine such holes *efficiently*. It would be easy to program just one motion through all separate parts and include the empty spaces in the cut. For many holes, this approach would prove to be very inefficient. Evaluate the front view of a web drilling example shown in Figure 26-28.

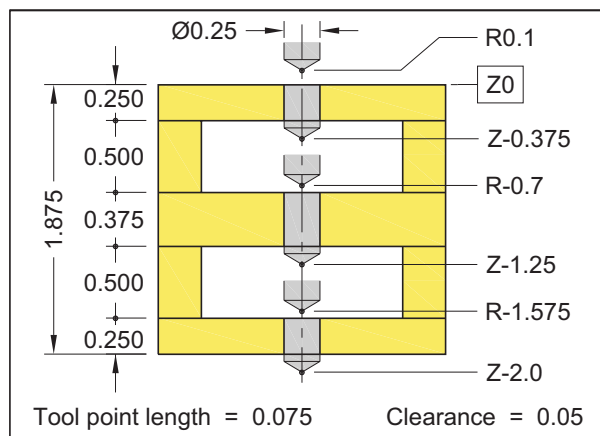


Figure 26-28

Web drilling example - front view - program O2606

The program uses X1.0Y1.5 as the hole position (G20). Both R-levels and Z-depths will be calculated. Clearances above and below each face are 0.05, the first R-level is R0.1. Drill point length of Ø1/4 drill is  $0.3 \times 0.25 = 0.075$ .

### O2606 (WEB DRILLING)

```

N1 G20
N2 G17 G40 G80 T01
N3 M06 (T01 - 90-DEG SPOT DRILL - 0.5 DIA)
N4 G90 G54 G00 X1.0 Y1.5 S900 M03 T02
N5 G43 Z1.0 H01 M08
N6 G99 G82 R0.1 Z-0.14 F250 F7.0
N7 G80 Z1.0 M09
N8 G28 Z1.0 M05
N9 M01

```

```

N10 T02
N11 M06 (T02 - 1/4 DIA DRILL)
N12 G90 G54 G00 X1.0 Y1.5 S1100 M03 T01
N13 G43 Z1.0 H02 M08
N14 G99 G81 R0.1 Z-0.375 F6.0 (TOP PLATE)
N15 R-0.7 Z-1.25 (MIDDLE PLATE)
N16 G98 R-1.575 Z-2.0 (BOTTOM PLATE)
N17 G80 Z1.0 M09
N18 G28 Z1.0 M05
N19 M30
%
```

Note that one single hole has required *three* blocks of the program, rather than the usual one. Each block represents only one plate in the part. Also note the G98 in block N16. Only one hole is done in the example, so the G98 is not really needed. The cycle cancellation command G80 with a return motion in block N17 would take care of the tool retract from the hole. However, if more holes are machined, move the tool to the new XY position *before* the G80 is programmed. In this case, the G98 is needed when the drills penetrates the last plate of the part. This example is not a perfect solution to web drilling cuts, as there is still some wasted motion. The only efficient programming method is to use the optional custom macro technique and develop a unique and efficient web drilling cycle.

## TAPPING

Tapping is second only to drilling as the most common hole making operation on CNC machining centers. As it is very common to tap in many milling applications, *two* tapping fixed cycles are available for programming on most control systems. The most common one is the G84 cycle for normal right hand tapping (R/H), the other one is the G74 cycle for reverse left hand tapping (L/H):

G84	Normal tapping - for <i>right hand</i> threads with M03 spindle rotation - right hand tap
G74	Reverse tapping - for <i>left hand</i> threads with M04 spindle rotation - left hand tap

The following example shows that programming a tapping operation for a hole is similar to other fixed cycles. All tool motions, including spindle stop and reversal at the hole bottom are included (built-in) in the fixed cycle:

```

...
N64 G90 G54 G00 X3.5 Y7.125 S600 M03 T06
N65 G43 Z1.0 H05 M08
N66 G99 G84 R0.4 Z-0.84 F30.0
N67 G80 ...

```

*Is it possible to tell the tap size used for the program?* Only four blocks are shown, but they can be a goldmine of information for a CNC operator who has to interpret an existing part program. *Think first.*



All information is there - a bit hidden, but it is there. The example shows a standard, 20 TPI thread - meaning *twenty threads per inch* - plug tap. XY coordinates are missing in G84 cycle (N66), because the current tool position has been established in earlier block N64 and it also corresponds to the tapped hole position as well. R-level is the standard start position for feedrate and the Z-depth is programmed as absolute depth of thread. The last address in this block is feedrate in inches per minute (*in/min*), programmed with the F-address (F30.0) - **a critical entry!**

Note that the R-level of R0.4 has an amount that is somewhat *higher* than might be used for drilling, reaming, single point boring and similar operations. Also, the programmed feedrate *appears* to be unreasonably high when compared to other tools. There is a good reason for these settings - yes, they are both *correct* and selected *intentionally*.

First, the R-level - the higher clearance for R-level allows *acceleration* of feedrate from 0 to 30 inches per minute to take place *in the air*. When tap contacts the part, cutting feedrate should be at its full programmed value, *never less*. A good rule of thumb is to program the tapping clearance about *two to four times* the thread pitch. This higher clearance will guarantee the programmed feedrate to be fully effective when actual tapping begins. Try to experiment with a slightly smaller number, to make the program more efficient. The main purpose of this method is to eliminate feedrate problems associated with motion acceleration.

Another subject is the programmed feedrate. The relatively high value of 30 in/min (F30.0) has also been carefully calculated. Any cutting feedrate for tapping *must* always be synchronized with the programmed *spindle speed* - the *r/min* programmed as the S-address. Keep in mind that tap is basically a *form* tool and its thread size and shape are built into it. Later in this chapter, the relationship between spindle speed and feedrate is explained in more detail (*see page 218*). Cutting feedrate *F* in the program example was calculated by multiplying the thread *lead* by the spindle *speed* given as *r/min*:

$$F = 1 / 20 \text{ TPI} \times 600 \text{ r/min} = 30.0 \text{ in/min}$$

Another way to calculate feedrate is to divide the *spindle speed* (r/min) by the number of *threads per inch* (TPI):

$$F = 600 \text{ r/min} / 20 \text{ TPI} = 30.0 \text{ in/min}$$

Overall quality of the tapped hole is also important, but it is not influenced solely by correct selection of speeds and feeds, but by several other factors as well. Material of the tap, its coating, its geometry, flute clearances, helix configuration, type of the start-up chamfer, material being cut - *and the tap holder itself* - all have a very profound effect on the final quality of any tapped hole. For best results in tapping, a floating tap holder is mandatory, unless the CNC machine supports more advanced *rigid tapping*. Floating tap holder design gives the tap a 'feel', similar to the feel

that is needed for manual tapping. A floating tap holder is often called a *tension-compression* holder and its applications are the same for both milling *and* turning operations. This type of holder allows the tap to be pulled out of a hole or pushed into a hole, within a certain range. One noticeable difference is the tool mounting method (tool orientation) in the machine (can be vertical or horizontal). High end floating tap holders also have an adjustable *torque*, which can change the feel of tap and even the range of tension and compression.

Tapping applications on CNC lathes are similar to those on machining centers. A special *tapping cycle* for a lathe control is not needed, as only one tap size can be used per part. Normally, each tapping motion is programmed with G32 command and block-by-block method.

CNC lathe tapping is different but not more difficult than tapping for CNC machining centers. Because it does not use fixed cycles (unless in turn-mill mode), programmers often make some common errors. This chapter uses examples for tapping on CNC lathes in a sufficient depth.

### ◆ Tap Geometry

There are literally dozens of tap designs, developed by many companies, that are used in CNC programming applications. A whole book could easily be filled with details of tapping tools and their applications. For CNC programming, only the *core basics* of tap geometry are important.

There are two main considerations in tap design that directly influence programming and data input values:

- Tap *flute* geometry
- Tap *chamfer* geometry

#### Tap Flute Geometry

Typical *flute* geometry of a tap is described in tooling catalogues in terms such as 'low helix', 'high helix', 'spiral flute', and others. These terms basically describe *how* the tap cutting edges are ground into the tap body. When programming a tapping operation, the effectiveness of flute geometry is always tied to *spindle speed*. Experimenting with tapping feedrate is limited by tap *lead* (usually just tap *pitch*), but there is a greater latitude with spindle speed selection. Part material *and* the tap flute geometry *both* influence machine spindle speed. Since almost all tool designs (not exclusively limited to taps) are results of corporate policies, engineering decisions and philosophies, various trade names and marketing strategies, there is not one way of saying 'use this tool' or 'use that tool' for a CNC program. Tooling catalogue from a tool supplier is the best source of technical data, but catalogue from another supplier may provide a better solution to a particular problem. Information gathered from a catalogue is a very good starting base for data in the CNC program. Keep in mind that all taps share some common characteristics.

### Tap Chamfer Geometry

Tap *chamfer geometry* relates to the tap end configuration. For CNC programming, the most important part of tap end point geometry is the *tap chamfer*.

In order to program a desired hole correctly, suitable tap must be selected according to specifications of the hole being tapped. For blind hole tapping, a different tap will be required than for tapping a through hole. There are three types of taps, divided by their end geometry configuration:

- Bottoming tap
- Plug tap
- Taper tap

Major difference between taps is the *tap chamfer length*. Figure 26-29 shows how the characteristics of a drilled hole influence programmed depth of the selected tap.

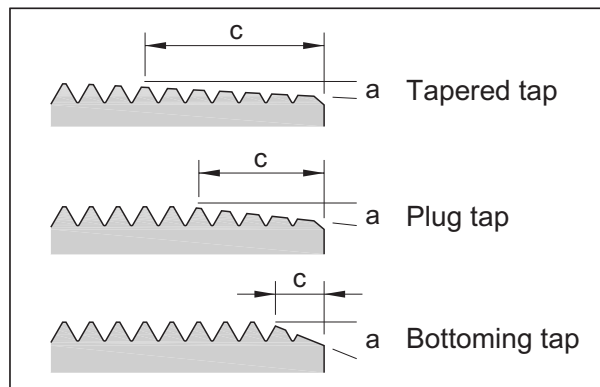


Figure 26-29

Typical tap ends - chamfer geometry configuration

Tap chamfer length *c* is always measured as the number of full threads. A typical number of full threads for a normal *tapered* chamfer is 8 to 10, for a *plug tap* 3 to 5, and for a *bottoming* tap 1 to 1.5. Angle of chamfer *a* also varies for each type - typically 4-5° for the tapered tap, 8-13° for the plug tap, and 25-35° for the bottoming tap.

A blind hole will almost always require a bottoming tap, a through hole will require a plug tap in most cases or a taper tap in some extremely rare cases. In a different way of explanation - the golden rule is - the larger the tap chamfer, the greater depth allowance required for each drilled hole.

### ◆ Tapping Speed and Feedrate

Relationship of programmed spindle speed (r/min) and programmed cutting feedrate is extremely important for programming cutting motion in feedrate *per time* mode. *Per time* mode is always programmed as *mm/min* (millimeters per minute) in programs using metric units, and *in/min* (inches per minute) for imperial units programming.

A specified *per minute* mode is typical to CNC milling machines and machining centers, where virtually all work is done either in *mm/min* or *in/min*. For tapping operations, regardless of machine tool, always program cutting feedrate as the *linear distance* the tap must travel during *one spindle revolution*. This distance is always equivalent to the tap *lead*, which is the same as the tap *pitch* (for tapping only), because taps are normally used to cut a single start thread only.

When using *feedrate per revolution* mode, mode that is typical to CNC lathes, the tap lead is always equivalent to the feedrate. For example, lead of 1.25 mm (0.050") will result in 1.25 mm/rev feedrate (0.050 in/rev), or F1.25 (F0.05) in the program.

On CNC machining centers, typical feedrate mode is always *per time*, measured in *per minute* mode, and the feedrate is calculated by one of the following formulas:

$$F_t = \frac{r / \text{min}}{\text{TPI}}$$

where ...

$F_t$	=	Feedrate per time (mm/min <b>or</b> in/min)
$r/\text{min}$	=	Spindle speed
TPI	=	Number of threads per inch

A similar formula will produce an identical result:

$$F_t = r / \text{min} \times F_r$$

where ...

$F_t$	=	Feedrate per time (mm/min <b>or</b> in/min)
$r/\text{min}$	=	Spindle speed in revolutions per minute
$F_r$	=	Feedrate per revolution

As feedrate for metric threads is much simpler to calculate, let's look at first at calculations for imperial threads. For example, a 20 TPI thread lead for a mill will be:

$$1 / 20 = 0.0500 \text{ inches}$$

and the programmed feedrate has to take into consideration the machine spindle speed, for example, 450 r/min:

$$F = 450 \times 0.05 = 22.5 = \text{F22.5 (in/min)}$$

or

$$F = 450 / 20 = 22.5 = \text{F22.5 (in/min)}$$

A metric tap on a lathe uses the same logic, but uses only one formula. For example, a tap of 1.5 mm lead (pitch) using 500 r/min is programmed with the feedrate of 750 mm/min, based on this calculation:

$$F = 500 \times 1.5 = 750.00 = F750.0 \text{ (mm/min)}$$

The key to successful tapping using a standard tapping head is to maintain relationship of the *tap lead* and the *spindle speed*. If spindle speed is changed, the feedrate per time (*in/min* or *mm/min*) must be changed as well. For many tension-compression type tap holders, adjustment of the feedrate *downwards* (so called underfeed) by about three to five percent may yield better results. This is because the tapping holder tension is more flexible than compression of the same holder.

If spindle speed in the above example is changed from S450 to S550 (tap size is *unchanged* at 20 TPI), the spindle speed change must be reflected in a *new* tapping feedrate:

$$F = 550 \times 0.05 = 27.50 = F27.5 \text{ (in/min)}$$

In the program, the new tapping feedrate requires a reduction of five percent, the feedrate *F* will be:

$$F = 27.5 - 5\% = 26.125$$

The actual feedrate value could be F26.1 or even F26.0. It is easy to change spindle speed of the tool in the program, or even directly on the CNC machine, then forget to modify feedrate for the tapping tool operation. This mistake can happen during program preparation in the office or during program optimization at the machine. If this change is small, there may be no damage, more due to luck than actual intent. If the change of spindle speed is major, tap will most likely break in the part.

### ◆ Pipe Taps

Pipe taps are similar in design to standard taps. They belong to two groups:

- Taper taps                                      NPT and API
- Straight taps (parallel)                      NPS

Their size designation (nominal size), is *not* the size of the tap, but the size of the pipe fitting. *American National Standard* pipe taper (NPT) has a taper ratio of 1 to 16, or 3/4 inch per foot (1.78991061° per side) and the tap chamfer is 2 to 3-1/2 threads - see *page 501* for various taper calculations.

Programming for pipe taps follows the usual considerations for standard threads. The only common difficulty is how to calculate the Z-depth position at least as a reasonable one, if not exactly. The final depth may be a subject of some experimentation with a particular tap holder and typical materials.

A proper tap drill size is very important. It will be different for tap holes that are only drilled and for tap holes that are drilled and reamed (using a 3/4 per foot taper reamer).

The following is a table of taper pipe thread sizes for NPT group and recommended tap drills, data that is useful for CNC programming:

NPT Group		Drilled Only		Taper Reamed	
Pipe Size	TPI	Tap Drill	Dec. Size	Tap Drill	Dec. Size
1/16	27	D	0.2460	15/64	0.2344
1/8	27	Q	0.3320	21/64	0.3281
1/4	18	7/16	0.4375	27/64	0.4219
3/8	18	37/64	0.5781	9/16	0.5625
1/2	14	45/64	0.7031	11/16	0.6875
3/4	14	29/32	0.9062	57/64	0.8906
1.0	11-1/2	1-9/64	1.1406	1-1/8	1.1250
1-1/4	11-1/2	1-31/64	1.4844	1-15/32	1.4688
1-1/2	11-1/2	1-47/64	1.7344	1-23/32	1.7188
2.0	11-1/2	2-13/64	2.2031	2-3/16	2.1875

For the straight pipe thread sizes (NPS), the following tap drills are recommended:

Pipe Size	TPI	Tap Drill	Decimal Size
1/16	27	1/4	0.2500
1/8	27	11/32	0.3438
1/4	18	7/16	0.4375
3/8	18	37/64	0.5781
1/2	14	23/32	0.7188
3/4	14	59/64	0.9219
1.0	11-1/2	1-5/32	1.1563
1-1/4	11-1/2	1-1/2	1.5000
1-1/2	11-1/2	1-3/4	1.7500
2.0	11-1/2	2-7/32	2.2188

The tapping feedrate maintains the same relationships for pipe taps as for standard taps.

### ◆ Tapping Check List

When programming a tapping operation, make sure the program data reflect the true machining conditions. They may vary between setups and machines, but the majority of them are typical to *any* tapping operations on *any* type of CNC machine.

Following short list identifies some of the most important items that relate *directly* to tapping operations in CNC programming.

- Tap cutting edges (have to be sharp and properly ground)
- Tap design (has to match the hole being tapped)
- Tap alignment (has to be aligned with tapped hole)
- Tap spindle speed  
(has to be reasonable for individual cutting conditions)
- Tap feedrate (has to be related to the tap lead and machine spindle speed)
- Part setup  
(rigidity of machine setup and tool is important)
- Drilled hole must be premachined correctly  
(tap drill size is very important)
- Clearance for tap start position  
(allow clearance for acceleration)
- Cutting fluid selection
- Clearance at the hole bottom  
(depth of thread must be guaranteed)
- Tap holder torque adjustment (ease of cutting)
- Program integrity (no errors)

Many designs of tap holders have their own special requirements, which may or may not have any effect on the programming approach. If in doubt, always check with the tap holder manufacturer for the suggested operation.

With modern CNC machines, the method of *rigid tapping* has become quite popular. There is no need for special tapping holders, such as the *tension compression* type - regular end mill holders or strong collet chucks can be used, saving the cost of tool holders. However, the CNC machine and its control system must support the rigid tapping feature. To program rigid tapping, there is a special M code available - check the machine documentation.

Rigid tapping mode must be supported by the CNC machine before it can be used in a program

## HOLE OPERATIONS ON A LATHE

Single point hole operations on a CNC lathe are much more limited than those on a CNC machining center. First, the number of holes that can be drilled or tapped in a single operation on a lathe is only one per part operation (two are rare), while the number of holes for a milling application may be in tens, hundreds and even thousands. Second, the boring (internal turning) on a lathe is a *contouring* operation, unlike boring on a milling machine, which is a *point-to-point* operation.

All point-to-point machining operations on a CNC lathe are limited to those that can be machined with a cutting tool positioned at the spindle *centerline*. Typically, these operations include center drilling, standard drilling, reaming and tapping. A variety of other cutting tools may also be used,

for example, a center cutting end mill (slot drill) to open up a hole or to make a flat bottom hole. An internal burnishing tool may also be used for operations such as precise sizing of a hole, etc. To a lesser degree, other operations, such as counterboring and countersinking may be programmed at the lathe spindle centerline, with a special point-to-point tool - *not a contouring tool*. All operations in this group will have one common denominator - they are all used at the spindle centerline and programmed with the X position as X0 in the program block.

Spindle speed for *all* centerline operations on a CNC lathe must be programmed in actual revolutions per minute (*r/min*), *not* in the constant surface speed mode (*CSS*), also known as *CS* - cutting speed. For that reason, G97 is used - for example,

**G97 S575 M03**

will assure the required 575 r/min at the normal spindle rotation (at 100% spindle speed override).

What will happen if CSS mode is used with G96 command, rather than the proper G97 command? CNC system will use the given information, spindle speed address S, in the program (given in peripheral - or surface - speed per minute, as *m/min* or *ft/min*). The system will then calculate the required spindle speed in *r/min* for the use by the machine tool.

Such a calculation is based on standard mathematical formula that relates to part *diameter*. If the diameter is zero - which is *exactly* what it is at the spindle centerline - the spindle revolutions will always be the *highest r/min* that is available within currently selected spindle gear range. This calculation is an exception to standard *r/min* calculation formula, where spindle speed at the centerline (diameter zero) would be zero - yes, *0 r/min*!

For example, if the cutting (surface) speed for a given material is 450 ft/min, the *r/min* at a Ø3.0 inches (X3.0) for the same material will be approximately:

$$S = (450 \times 3.82) / 3 = 573 \text{ rpm}$$

If the same speed of 450 ft/min is applied to the diameter zero (X0 in the program), the formula does not change, but the resulting action does:

$$S = (450 \times 3.82) / 0 = \text{results in error}$$

Although the spindle might be expected to stop control issue an error condition, it will do the *exact opposite* (because of the control design). Spindle speed will reach the maximum *r/min* that the current gear range will allow. Be very careful here - make sure that all centerline operations on a CNC lathe are always done in G97 (*r/min*) mode and *not* in G96 mode (*CSS*) mode.



### ◆ Tool Approach Motion

A typical geometry offset configuration setup (or the old G50 values) on a CNC lathe often have a relatively large X value and relatively small Z value. For example, metric geometry offset for a tool may be X-300.0 Z-25.0 (or programmed as G50 X300.0 Z25.0). This location indicates a suitable tool change position applicable to a drill. What does it mean to the tool motion for a drilling operation?

It means that machine rapid motion will complete the Z-axis motion long before completing the X-axis motion (with the quite common hockey-stick motion of the rapid command). Resulting tool motion is located *very close* to the part face (G21 mode):

```
N36 T0200 M42
N37 G97 S700 M03
N38 G00 X0 Z2.5 T0202 M08
N39 ...
```

To avoid a potential collision during tool approach towards the part, use one of the following methods:

- Move the X-axis first to the spindle centerline, then the Z-axis, directly to the start position for drilling
- Move the Z-axis first to a clear position, then the X-axis to spindle centerline, then complete the Z-axis motion into the start position for drilling

The first method may be practical only in those cases when the tool motion area is absolutely clear and has no obstacles in the way (do not count on such a situation). The second method, and probably the most common in programming, will first move the Z-axis close (but not too close) to the part, say 15 mm in the front (Z15.0). Motion that follows is X-axis motion only - directly to the centerline (X0). At this point, the cutting tool (such as a drill) is far from the Z-axis face. The last approach motion will be to the Z-axis start position, closer to part face, where the actual drilling cut begins. This method eliminates (or at least minimizes) any possibility of collision with obstacles along the way. The obstacles are - or at least could be - tailstock, parts catcher, steadyrest, fixture, face plate, etc. An example of this programming tool path method is the previous example, modified (G21 mode):

```
N36 T0200 M42
N37 G97 S700 M03
N38 G00 X0 Z15.0 T0202 M08
N39 Z2.5
N40 ...
```

This programming method splits the tool approach along Z-axis into *two* tool positions - one is a safe clearance for approach, the other one is a safe clearance position for the drill start. There is a minor alternative to this motion - the last Z-axis approach will be at a cutting feedrate G01, rather than at a rapid motion rate G00:

```
N36 T0200 M42
N37 G97 S700 M03
N38 G00 X0 Z15.0 T0202 M08
N39 G01 Z2.5 F2.0
N40 ... F.. (CUTTING FEEDRATE)
```

For the last approach motion, Z-axis motion has been changed to a linear motion, with a relatively high feedrate of 2 mm/rev), followed by cutting feedrate. Feedrate override can be used for setup, to control the approach rate. During actual production, there will be no significant loss in the cycle time.

### ◆ Tool Return Motion

The same logical rules of motion in space that apply to tool approach, apply also to tool return motion. Remember that the first motion *from* a hole must always be along the Z-axis (G20 mode):

```
...
N40 G01 Z-0.8563 F0.007
N41 G00 Z0.1
...
```

In block N40, the actual drill cutting motion takes place. When the cut is completed, block N41 is executed. The drill will rapid out of the hole to the same position it started from (Z0.1). It is not necessary to return to the same position, but it makes programming style more consistent. Once the cutting tool is safely out of the hole, it has to return to a tool changing position. There are two methods:

- Simultaneous motion of both axes
- Single axis at a time

Simultaneous motion of the X and Z axes does not present the same problem as it did on approach - on the contrary. Z-axis will complete the motion first, moving away from the part face. Also, there is no reason to fear a collision during a return motion if the approach motion was successful and programming style was consistent:

```
...
N70 G01 Z-0.8563 F0.007
N71 G00 Z0.1
N72 X11.0 Z2.0 T0200 M09
...
```

If in doubt, or if an obstacle *is* expected to be in the way of a tool motion, for example a tailstock, program a single axis at a time. In most cases, that will move the positive X axis first, as most obstacles would be to the right of the part:

```
...
N70 G01 Z-0.8563 F0.007
N71 G00 Z0.1
N72 X11.0
N73 Z2.0 T0200 M09
...
```



This programming example illustrates return motion with the X-axis programmed first. The fact that the tool is 0.100 off the front face is irrelevant - after all, the tool started cutting *from* that distance without a problem.

Other, less traditional, methods for tool motion towards and away from the part are also possible.

### ◆ Drilling and Reaming on Lathes

Drilling on a CNC lathe is also quite common operation, mainly as means for a hole opening to be used with other tools, such as boring bars. There are three basic kinds of drilling, typical to a CNC lathe machining:

- Center drilling and spot drilling
- Drilling with a twist drill
- Indexable insert drilling

Each method follows the same programming techniques as those described in the milling section earlier, except that there are no fixed cycles of the milling type used for the lathe work. Keep in mind that on a CNC lathe, the part is rotating, whereby the cutting tool remains stationary. Also keep in mind that most lathe operations take place in a horizontal orientation, causing fewer concerns about coolant direction and chip removal.

### ◆ Peck Drilling Cycle - G74

On Fanuc and compatible controls, there is a multiple repetitive cycle G74 available, that can be used for two different machining operations:

- Simple roughing with chip breaking
- Peck drilling (deep hole drilling)

In this section, the peck drilling usage of G74 cycle is described. Roughing application of G74 cycle is a contouring operation and is very seldom used.

In peck drilling, just like in any ordinary drilling, select spindle speeds and cutting feedrates first, then determine the hole starting position and finally, its depth position. In addition, establish (or even calculate) the depth of each peck. Lathe cycle G74 is rather limited in what it can do, but it has its uses. Its format for peck-drilling is:

**G74 X0 Z. . K. .**

where ...

- G74 = Peck drilling cycle selection
- X0 = Indicates cutting on centerline
- Z = Specifies the end point for drilling
- K = Depth of each peck (always positive)

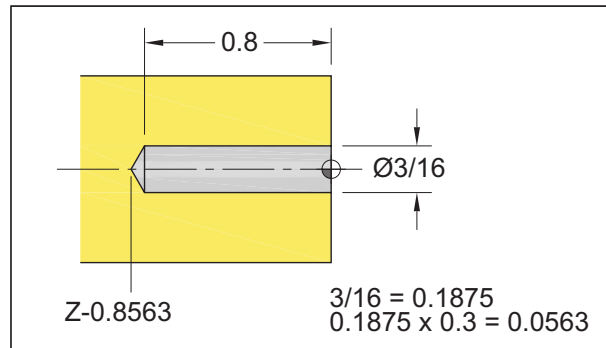


Figure 26-30

Peck drilling example - sample hole

The following program uses illustration in Figure 26-30, and shows an example of drilling a 3/16 hole ( $\varnothing 0.1875$ ) with a peck drill depth of 0.300 inches:

```
...
N85 T0400 M42
N86 G97 S1200 M03
N87 G00 X0 Z0.2 T0404 M08
N88 G74 X0 Z-0.8563 K0.3 F0.007
N89 G00 X12.0 Z2.0 T0400 M09
N90 M01
```

Peck drilling motion will start from Z0.2 position in block N87 and continue to Z-0.8563 position in block N88. That results in a 1.0563 long cut. Calculation of the number of pecks is the same as in milling.

With 0.300 length of each peck, there will be total of three *full* length pecks and one partial length peck, at the following Z-axis locations:

```
Z-0.1
Z-0.4
Z-0.7
Z-0.8563
```

Although the first three pecks are 0.300 deep each, the first one starts at Z0.2 and ends at Z-0.1. That will result in two thirds of the cut being in the air. Programmer has to decide when this approach is an advantage and when another method would be more suitable. At the end of each peck motion using G74 cycle, the drill will make a small retract by a *fixed* distance. This distance is set by a parameter of the control system and is typically about 0.020 inches (0.5 mm). A full retraction after each peck out of the hole (similar to G83 cycle for milling controls) is not supported by the G74 cycle.

Note that there is *no* programmed motion *out* of the hole when the peck drilling cycle is completed. This return motion is built-in within G74 cycle. If a tool motion such as G00 Z0.2 M05 follows block N88, no harm is done. It may give the operator extra confidence when running the job.

### ◆ Tapping on Lathes

Tapping on CNC lathes is a common operation that follows the same machining principles as tapping on machining centers. The major difference for lathes is the absence of a tapping cycle. There is no real need for a tapping cycle on a lathe, since most of lathe tapping operations machine only *one hole of the same type*. The absence of a tapping cycle may present some unexpected difficulties. Unfortunately, they are more common among programmers with limited experience. Before evaluating some of these difficulties closer, it is important to know the actual tool that holds the tap (*tap holder*) and the tapping process on CNC lathes in general.

The selected tap should always be mounted in a special tapping holder; the best type is one with tension and compression features, known as *tap floating holder*. Never use a drill chuck or a similar solid device - it will break the tap quickly and possibly damage the part as well.

Since there is no fixed cycle for tapping on a typical CNC lathe, each tool motion is programmed as a separate block. To do that, and to find out how to tap properly, let's first evaluate the process for a typical right hand tap in general, applied to a lathe operation:

- Step 01 Set coordinate position XZ
- Step 02 Select tool and gear range
- Step 03 Select spindle speed and rotation
- Step 04 Rapid to centerline and clearance with offset
- Step 05 Feed-in to the desired depth
- Step 06 Stop the spindle
- Step 07 Reverse the spindle rotation
- Step 08 Feed-out to clear of the part depth
- Step 09 Stop the spindle
- Step 10 Rapid to the start position
- Step 11 Resume normal spindle rotation or end program

Translated into a CNC program carefully, this step by step procedure can be used in everyday programming as a general guide to tapping on CNC lathes.

Figure 26-31 shows layout of a part and tool setup, used for program example O2607. Program example O2607 follows the eleven steps described above literally and is based on a very solid foundation. Technically, the program O2607 is correct - but only in theory, not practically.

Are there possible problem situations in example O2607 - some thoughts on what can go wrong?

#### O2607 (TAPPING ON LATHES) (CORRECT VERSION - IN THEORY ONLY)

```
...
(T02 - TAP DRILL 31/64)
...
...
N42 M01

(T03 - 9/16-12 PLUG TAP)
N43 T0300 M42
N44 G97 S450 M03
N45 G00 X0 Z0.5 M08 T0303
N46 G01 Z-0.875 F0.0833
N47 M05
N48 M04
N49 Z0.5
N50 M05
N51 G00 X12.0 Z2.0 T0300 M09
N52 M30
%
```

A brief look at the program O2607 does not show that anything is wrong. After all, the program covers all necessary motions and is, therefore, correct.

*Yet, this program contains major flaws !*

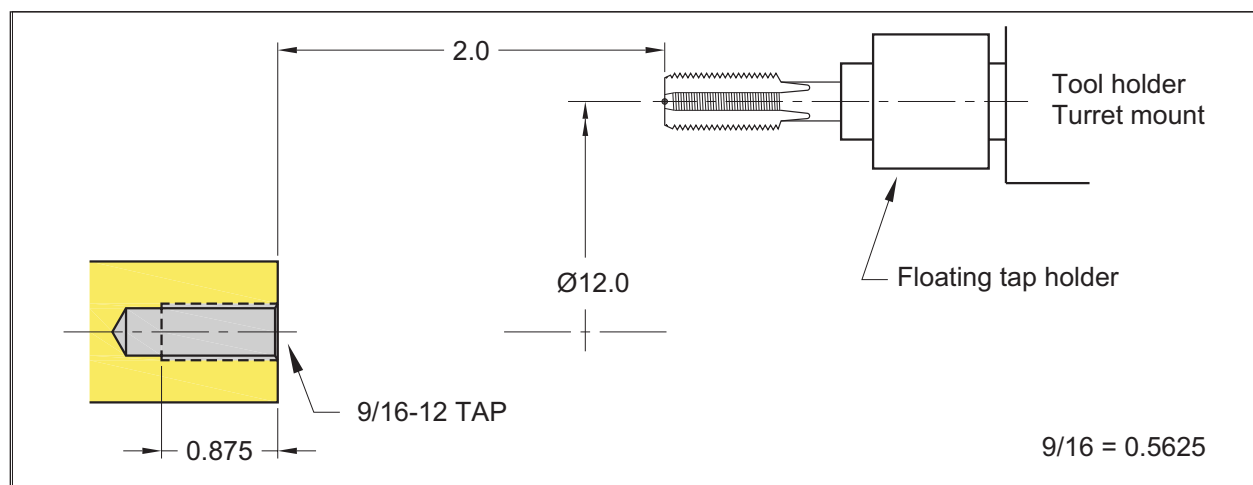


Figure 26-31

Typical setting of a tapping tool on CNC lathe - program examples O2607 and O2608

All earlier tapping steps have been carefully followed. Conducting a more in-depth study of the program will reveal two areas of potential difficulty or even danger. The first problem may arise if the machine feedrate override setting switch is *not* set to 100%. Remember, the tapping feedrate is always equal to the thread lead (F0.0833 is the feedrate for 12 TPI). If the override switch is set to any other value but 100%, the thread will be stripped at best and the tap broken at worst with related part damage.

The other problem will become evident only in a single block mode run, during setup or machining. Look at blocks N46 and N47. In the N46 block, tap reaches the end of Z-axis - *while the spindle is still rotating!* True, the spindle will be stopped in block N47, but in single block mode it will be too late. A similar situation will happen during the feed-out motion. Spindle reverses in block N48, but does not move until block N49 is processed. Therefore, the program O2607 is a *very poor* example of tapping on CNC lathes.

These are some details usually not considered for a fixed cycle application (such as the G84 tapping cycle), when used for milling programs. For milling, all tool motions are built-in, so they are contained *within* the fixed cycle. To eliminate the first potential problem of feedrate override setting, programming M48/M49 functions will temporarily disable the feedrate override switch. Even better way is to replace the feed-in and feed-out tap motion command from the current G01 mode to G32 mode (G33 on some controls). G32 command is normally used for single point threading. Two major results will be achieved with G32 command - the *spindle will be synchronized*, and the *feedrate override will be ineffective* by default (automatically). That solves the first problem. The second problem will be solved if the spindle M-functions are programmed in the *same block* as the tool motion. That means joining the block N46 with N47, and block N48 with N49.

This much improved version of the tapping example is in a new program - O2608:

```
O2608 (TAPPING ON LATHES)
(CORRECT VERSION IN PRACTICE)
...
(T02 - TAP DRILL 31/64)
...
N42 M01

(T03 - 9/16-12 PLUG TAP)
N43 T0300 M42
N44 G97 S450 M03
N45 G00 X0 Z0.5 M08 T0303
N46 G32 Z-0.875 F0.0833 M05
N47 Z0.5 M04
N48 M05
N49 G00 X12.0 Z2.0 T0300 M09
N50 M30
%
```

The block (N48 in the example) containing spindle stop function M05, is not required if the tap is the *last* tool programmed, although it does no harm in any other program. Compare this newer program O2608 with program O2607. Program O2608 is a great deal more stable and the possibility of any significant problem is virtually eliminated.

### ◆ Other Operations

There are many other programming variations relating to machining holes on CNC machining centers and lathes. This chapter has covered some of the most important and most common possibilities.

Some less common applications, such as machining operations using tools for backboring, or block boring tools, tools with multiple cutting edges and other special tools for machining holes may be quite infrequent in programming. However, programming these *unusual* operations is no more difficult than programming the ordinary everyday tool motions, using ordinary everyday tools.

The real ability of a CNC programmer is measured in terms of applying the past knowledge and experience to a new situation. It requires a thinking process and it requires a degree of ingenuity and hard work.

## 27

## PATTERN OF HOLES

In point-to-point machining operations, those consisting of drilling, reaming, tapping, boring, etc., we are often required to machine either a single hole or a series of holes with the same tool, usually followed by other tools. In practice, several holes are much more common than a single hole. Machining several holes with the same tool means machining a *pattern* of holes or a *hole pattern*. An English dictionary defines the word 'pattern' as a '*characteristic or consistent arrangement or design*'. Translated to hole machining terms, any two or more holes machined with the same tool establish a pattern. The desired hole pattern is laid out in the part drawing either randomly (*characteristic arrangement or design*) or in a certain order (*consistent arrangement or design*). Dimensioning of a hole pattern follows standard dimensioning practices.

This chapter describes some typical hole patterns laid out on a flat part and the various methods of their programming. To make matters simple, all programming examples related to hole patterns will assume a center drilling operation, using a #2 center drill, with chamfer diameter 0.150, to the depth of 0.163 (programmed as Z-0.163 - see page 202). Program zero (program reference point Z0) is the top face of part and the tool is assumed to be already in the spindle. For the purposes of clarity, no hole diameters or material size and thickness are specified in the examples.

From the dictionary definition above, we have to establish what makes a hole pattern *characteristic* or *consistent*. Simply, any series of holes that are machined with the same tool, one hole after another, usually in the order of convenience. That means all holes within a single pattern have the same nominal diameter. It also means that all machining must start at the same R-level and end at the same Z-depth. Overall, it means that all holes within a pattern are machined the same way for any single tool.

## TYPICAL HOLE PATTERNS

Hole patterns can be categorized into several typical groups, each group having the same character. Every hole pattern encountered in CNC programming belongs into one of the following pattern groups:

- Random pattern
- Straight row pattern
- Angular row pattern
- Corner pattern

- Grid pattern
- Arc pattern
- Bolt circle pattern

Some groups may be divided further into smaller sub-groups. A thorough understanding of each pattern group should help you to program any similar hole pattern.

There are several control systems available that have a built-in hole pattern programming, for example for a *bolt circle pattern*. These programming routines simplify the hole pattern programming quite substantially, but the program structure is usually unique to that particular brand of control and cannot be applied to other controls.

## RANDOM HOLE PATTERN

The most common pattern used in programming holes is a *random pattern*. Random pattern of holes is a pattern where all holes share the same machining characteristics, but the X and Y distances between them are inconsistent. In other words, holes within a random pattern share the same tool, the same nominal diameter, usually the same depth, but a variable distance from each other - *Figure 27-1*.

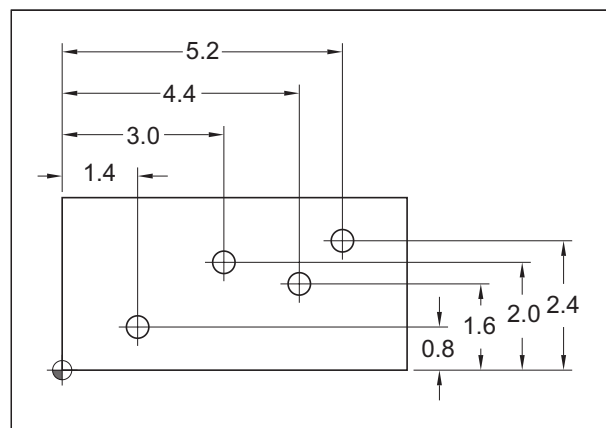


Figure 27-1

Random pattern of holes - program example O2701

There are no special time saving techniques used in programming a random pattern - only a selected fixed cycle used at individual hole locations. All XY coordinates within this hole pattern have to be programmed manually; the control system features will be no help here at all:

```

O2701 (RANDOM HOLE PATTERN)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.4 Y0.8 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 X3.0 Y2.0
N7 X4.4 Y1.6
N8 X5.2 Y2.4
N9 G80 M09
N10 G28 Z0.1 M05
N11 G28 X5.2 Y2.4
N12 M30
%
```

## STRAIGHT ROW HOLE PATTERN

Hole patterns parallel to the X or Y axis with an equal pitch is a *straight row* pattern. Figure 27-2 shows a 10 hole pattern along the X-axis, with a pitch of 0.950 inch.

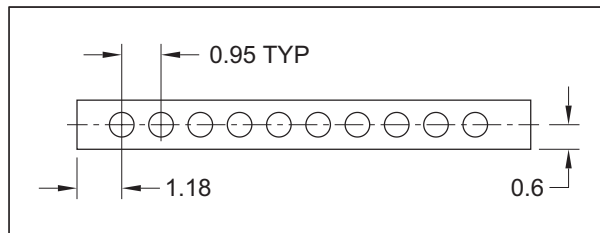


Figure 27-2

Straight row hole pattern - program example O2702

Programming approach takes advantage of a fixed cycle repetition feature, using the K or L address. It would be inefficient to program each hole individually. As always, the tool will be positioned at the first hole in G90 mode, then the cycle will machine this hole in block N5.

For the remaining holes, G90 mode must be changed to incremental mode G91, which instructs the control to machine the remaining nine holes incrementally, along the X-axis only. The same logic would also apply for a vertical pattern along the Y-axis. In that case, the pitch increment would be programmed along the Y-axis only. Note that the repetition count is always equal to the *number of spaces*, not the number of holes. The reason? The first hole has already been machined in the cycle call block.

```

O2702 (STRAIGHT ROW HOLE PATTERN)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.18 Y0.6 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 G91 X0.95 L9
N7 G80 M09
N8 G28 Z0 M05
N9 G28 X0 Y0
N10 M30
%
```

Two features of program O2702 should be emphasized. In block N6, the dimensioning mode was changed from absolute G90 to incremental G91 mode, to take advantage of the equal pitch distance. When all ten holes have been machined, the program has to include return to machine zero position motion, in the example, along all three axes. However, without a calculation, *we do not know* the absolute position at the tenth hole for the X-axis (the Y-axis remains unchanged at the position of 0.60 inches = Y0.6). To solve this 'problem', cancel the cycle with G80, leave G91 mode in effect and move to machine zero position in the Z-axis first (for safety reasons). Then - still in the incremental mode G91 - return both X and Y axes to machine zero simultaneously.

Normally, this first tool of the example would be followed by other tools to complete the hole machining. To protect the program and machining from possible problems, make sure that the G90 absolute command is reinstated for every tool that follows.

## ANGULAR ROW HOLE PATTERN

Pattern of holes in a row at an angle is a variation of a straight line pattern. The difference between them is that the incremental pitch applies to both X *and* Y axes. A hole pattern of this type will be established on the part drawing as one of two possible dimensioning methods:

- X and Y coordinates are given for the first and last hole

In this method, the pattern angular position is not specified and no pitch between holes is given.

- X and Y coordinates are given for the first hole only

In this method, pattern angular position is specified and the pitch between holes is given.

In either case, all necessary X and Y dimensions are available to write the program. However, the programming approach will be different for each method of drawing dimensioning.

### ◆ Pattern Defined by Coordinates

This method of programming is similar to the straight row pattern. Since the pitch between holes is not given, the increment between holes along each of the two axes must be calculated. This axial distance is commonly known as the *delta distance* (delta X is measured along the X-axis, delta Y is measured along the Y-axis). Such calculation can be done in two equally accurate ways.

The first calculation method can use a trigonometric method, but it is much easier to use the ratio of sides instead. In Figure 27-3, the pattern length along X-axis is 10.82 and along Y-axis it is 2.0:  $(2.625 - 0.625 = 2.0)$



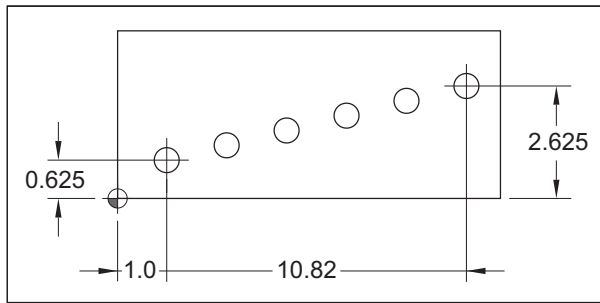


Figure 27-3

Angular hole pattern with two sets of coordinates - program O2703

Pattern of this kind has all holes spaced by equal distances along X and Y axes. As all holes are equally spaced, the ratio of sides for individual holes is identical to the ratio of whole pattern. When expressed mathematically, the increment between holes along the X-axis is equal to the overall distance of 10.82 divided by the number of X-axis spaces; the increment along the Y-axis is equal to the overall distance of 2.0 divided by the number of Y-axis spaces. Number of spaces for a six hole pattern is five, so the X-axis increment (delta X) is:

$$10.82 / 5 = 2.1640$$

and the Y-axis increment (delta Y) is:

$$2.0 / 5 = 0.4$$

The other calculation method uses trigonometric functions, which may also be used as a confirmation of the first method, and vice versa. *Both results must be identical*, or there is a mistake somewhere in the calculation. First, establish some temporary values:

$$A = \tan^{-1}(2.0 / 10.82) = 10.47251349^\circ$$

$$C = 2.0 / \sin A = 11.00329063$$

$$C1 = C / 5 = 2.20065813$$

Now, the actual increment along the two axes can be calculated, using C1 dimension as the distance between holes:

$$X \text{ increment} = C1 \times \cos A = 2.1640$$

$$Y \text{ increment} = C1 \times \sin A = 0.4000$$

Both calculated increments match, calculation is correct, and can now be used to write the program (O2703) - block N6 contains the values:

```
O2703 (ANGULAR ROW 1)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.0 Y0.625 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
```

```
N6 G91 X2.164 Y0.4 K5 (L5)
N7 G80 M09
N8 G28 Z0 M05
N9 G28 X0 Y0
N10 M30
%
```

Note that the program structure is identical to the example of the straight row pattern, except the incremental move with K5 (L5) address is along two axes instead of one.

### ◆ Pattern Defined by Angle

An angular line pattern can also be defined in the drawing by the X and Y coordinates of the first hole, number of equally spaced holes, distance between holes and the angle of pattern rotation - Figure 27-4.

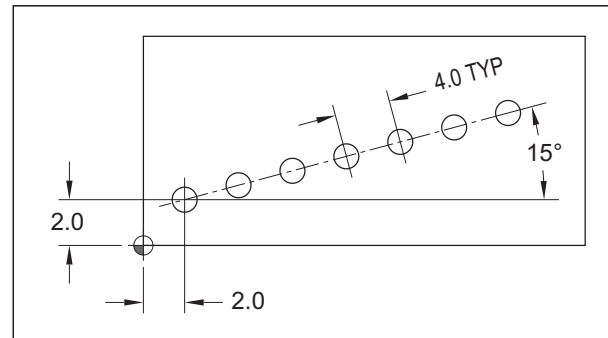


Figure 27-4

Angular hole pattern with coordinates, pitch and angle - O2704

In order to calculate the X and Y coordinate values, use trigonometric functions in this case:

$$X = 4.0 \times \cos 15 = 3.863703305$$

$$Y = 4.0 \times \sin 15 = 1.03527618$$

Program can be written after you round off the calculated values - program O2704:

```
O2704 (ANGULAR ROW 2)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X2.0 Y2.0 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 G91 X3.8637 Y1.0353 K6 (L6)
N7 G80 M09
N8 G28 Z0 M05
N9 G28 X0 Y0
N10 M30
%
```

Since the calculated increments are rounded values, a certain accumulative error is inevitable. In most cases, any error will be well contained within the required drawing tolerances. However, for projects requiring the highest precision, this error may be important and must be taken into consideration

To make sure all calculations are correct, a simple checking method can be used to compare all calculated values:

### ➡ Step 1

Find the absolute coordinates XY of the *last* hole:

$$\begin{aligned} X &= 2.0 + 4.0 \times 6 \times \cos 15 \\ &= 25.18221983 = \underline{X25.1822} \end{aligned}$$

$$\begin{aligned} Y &= 2.0 + 4.0 \times 6 \times \sin 15 \\ &= 8.211657082 = \underline{Y8.2117} \end{aligned}$$

### ➡ Step 2

Compare these new XY coordinates with the previously calculated increments as they relate to the last hole of the pattern (using rounded values):

$$\begin{aligned} X &= 2.0 + 3.8637 \times 6 = \underline{25.1822} \\ Y &= 2.0 + 1.0353 \times 6 = \underline{8.2118} \end{aligned}$$

Note that both X and Y values are accurate. When rounding, particularly when a large number of holes is involved, the accumulative error may cause the hole pattern out of tolerance. In that case, the only correct way to handle programming is to calculate the coordinates of each hole as absolute dimensions (that means from a *common* point rather than a *previous* point). Programming process will take a little longer, but it will be much more accurate.

## CORNER PATTERN

Pattern of holes can be arranged as a corner - which is nothing more than a pattern combining the straight and/or angular hole patterns - Figure 27-5.

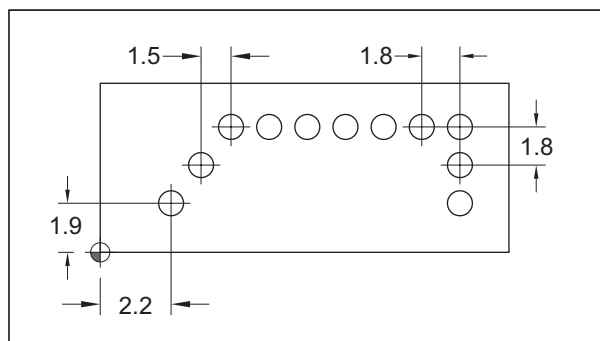


Figure 27-5

Corner pattern of holes - program example O2705

All rules mentioned for the straight and angular hole patterns apply for a corner pattern as well. The most important difference is the corner hole, which is *common* to two rows. A corner pattern can be programmed by calling a fixed cycle for each row. Soon, it will become apparent that each

corner hole will be machined *twice*. Visualize the complete process - the *last* hole of one row pattern is also the *first* hole of the next pattern, duplicated. Creating a special custom macro is worth the time for many corner patterns. The normal solution is to move the tool to its first position, call the required cycle and *remain within* that cycle:

```
O2705 (CORNER PATTERN)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X2.2 Y1.9 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 G91 X1.5 Y1.8 K2 (L2)
N7 X1.8 K6 (L6)
N8 Y-1.8 K2 (L2)
N9 G80 M09
N10 G28 Z0 M05
N11 G28 X0 Y0
N12 M30
%
```

This program offers no special challenges. In block N6, the angular row of holes is machined, starting from the lower left hole, in N7 it is the horizontal row of holes, and in N8 the vertical row of holes is machined, for continuous order. Just like in the earlier examples, keep in mind that the repetition count K or L is for the number of moves (spaces), not the number of holes.

## GRID PATTERN

Basic straight grid pattern can also be defined as a set of equally spaced vertical and horizontal holes, each row having equally spaced holes. If spacing of all vertical holes is the same as spacing of all horizontal rows, the final grid pattern will be a square. If spacing of all vertical holes is not the same as spacing of all horizontal rows, the resulting grid pattern is a rectangle. A grid pattern is sometimes called a *rectangular hole pattern* - Figure 27-6.

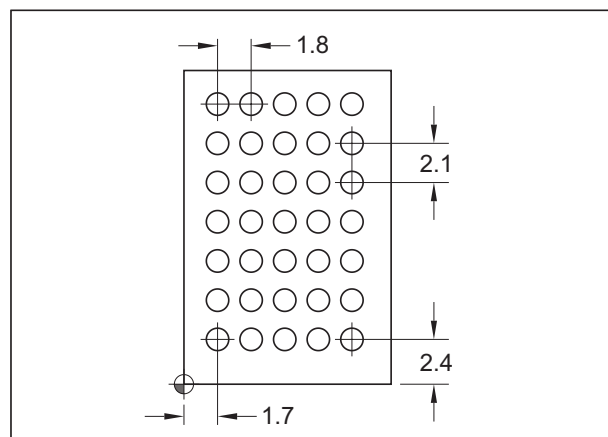


Figure 27-6

Rectangular grid hole pattern - program example O2706

A grid pattern is very similar to a series of corner patterns, using similar programming methods. One major consideration for a grid pattern programming is in its efficiency. Each row can be programmed as a single row pattern, starting, for example, from the left side of each row. Technically, that is correct, although not very efficient due to loss of time, when the tool has to travel from the last hole of one row, to the first hole of the next row.

More efficient method will look like a *zigzag* motion. To program a zigzag motion, program the first row or column starting at any corner hole. Complete that row (column), then jump to the nearest hole of the next row (column) and repeat the process until all rows and columns are done. The wasted time of the rapid motion is kept to the minimum.

#### O2706 (STRAIGHT GRID PATTERN)

```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.7 Y2.4 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 G91 Y2.1 K6 (L6)
N7 X1.8
N9 Y-2.1 K6 (L6)
N10 X1.8
N11 Y2.1 K6 (L6)
N12 X1.8
N13 Y-2.1 K6 (L6)
N14 X1.8
N15 Y2.1 K6 (L6)
N16 G80 M09
N17 G28 Z0 M05
N18 G28 X0 Y0
N19 M30
%
```

Two features of the program are worth noting - one is the jump from one row of the pattern to another - it has no repetition address K or L, which is the same as K1 (L1), because only *one* hole is being machined at that location. The second feature may not be so obvious right away. To make the program a bit shorter, start along the axis that contains the *larger* number of holes (Y-axis in the program example O2706). This example is a variation on the previous examples and also adheres to all the rules established so far. A special subprogram made for a grid pattern is also a common programming approach and can be used as well.

### ◆ Angular Grid Pattern

Straight grid pattern is the most common pattern for square and rectangular hole arrangement. A grid pattern may also be in the shape of a parallelogram (hexagon, for example), called an *angular grid pattern* - Figure 27-7.

Again, programming approach remains the same as for previous rectangular grid pattern, the only extra work required is the calculation of angular increments, similar to methods shown:

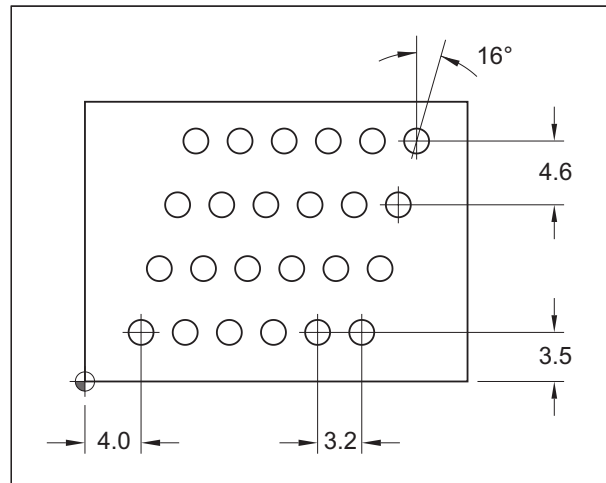


Figure 27-7

Angular grid hole pattern - program example O2707

Unknown increment in the drawing is the distance measured along the X-axis, from a hole in one horizontal row to the next hole in the following horizontal row:

$$X = 4.6 \times \tan 16 = 1.319028774 \text{ (X1.319)}$$

Program can be written in a similar way as for the straight row grid, except the extra 'jump' between rows will take place along both axes:

#### O2707 (ANGULAR GRID)

```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X4.0 Y3.5 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 G91 X3.2 K5 (L5)
N7 X1.319 Y4.6
N8 X-3.2 K5 (L5)
N9 X1.319 Y4.6
N10 X3.2 K5 (L5)
N11 X1.319 Y4.6
N12 X-3.2 K5 (L5)
N13 G80 M09
N14 G28 Z0 M05
N15 G28 X0 Y0
N16 M30
%
```

Many experienced programmers will consider even more efficient way of approaching the programs for grid patterns by using subprograms or even *User Macros*. Subprograms are especially useful for grid patterns consisting of a large number of rows or a large number of columns, as well as several tools. The subject of subprograms, including a practical example of a really large grid pattern, starts on page 383. The subject of *user macros* is not covered in this handbook, but *Fanuc CNC Custom Macros* book is available from Industrial Press, Inc. ([www.industrialpress.com](http://www.industrialpress.com)).

## ARC HOLE PATTERN

Another quite common hole pattern is a set of equally spaced holes arranged along an arc (not a circle). Such an equally spaced set of holes along any portion of a circle circumference creates an *arc hole pattern*.

Basic approach to programming an arc hole pattern should be the same as if programming any other pattern. Select the first hole that is most convenient. Is it the first hole or the last hole on the arc that is easier to find the coordinates for? Perhaps starting at 0° (3 o'clock or East position) would be a better choice? Illustration in *Figure 27-8* shows a typical layout of an arc hole pattern.

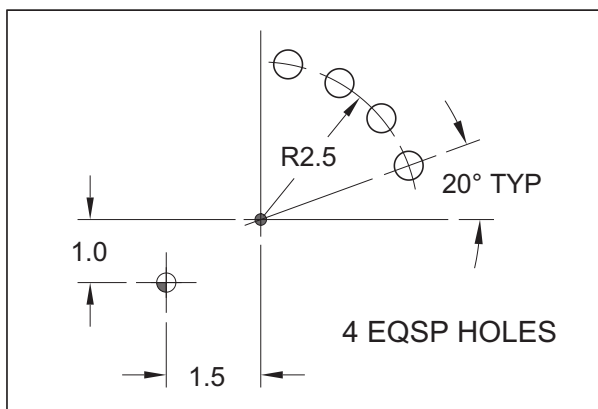


Figure 27-8

Arc hole pattern - program O2708

In the pattern, arc center locations are known, so is the arc radius, angular spacing between holes and the number of equally spaced (EQSP) holes along the circumference.

A number of calculations is needed to find the X and Y coordinates for each hole center location within the bolt hole pattern. Procedure is similar to that of an angular line in a grid pattern, but with several more calculations. These calculations use trigonometric functions applied to each hole separately - all necessary data and other information are listed in the drawing.

For any number of holes, exactly the double number of calculations will be required to get the coordinates for *both* axes. In the example, there are four holes, therefore eight calculations will be necessary. Initially, it may seem as a lot of work. In terms of calculations, it is a lot of work, but keep in mind that only two *trigonometric* formulas are involved for any number of holes, so all calculations will become a lot more manageable. Incidentally, this observation can be applied to just about any other similar programming application.

The best way to illustrate arc pattern programming, is to use the drawing example above. First, the programming task will be split into four individual steps:

### ➡ STEP 1

Start with calculation of a hole that is nearest to 0° location (3 o'clock position or East direction), then continue for other holes in the counterclockwise direction of the arc.

### ➡ STEP 2

Use trigonometric functions to calculate X and Y coordinates of the *first* hole:

Hole #1 - at 20 degrees

$$X = 1.5 + 2.5 \times \cos 20 = 3.849231552 \quad (X3.8492)$$

$$Y = 1.0 + 2.5 \times \sin 20 = 1.855050358 \quad (Y1.8551)$$

### ➡ STEP 3

Use the same trigonometric formulas as in *Step 2* and calculate XY coordinates for the 3 *remaining* holes. For each hole in the pattern, increase the included angle by 20°, so the second hole angle will be 40°, the third 60°, and so on:

Hole #2 - at 40 degrees

$$X = 1.5 + 2.5 \times \cos 40 = 3.415111108 \quad (X3.4151)$$

$$Y = 1.0 + 2.5 \times \sin 40 = 2.606969024 \quad (Y2.607)$$

Hole #3 - at 60 degrees

$$X = 1.5 + 2.5 \times \cos 60 = 2.750000000 \quad (X2.75)$$

$$Y = 1.0 + 2.5 \times \sin 60 = 3.165063509 \quad (Y3.1651)$$

Hole #4 - at 80 degrees

$$X = 1.5 + 2.5 \times \cos 80 = 1.934120444 \quad (X1.9341)$$

$$Y = 1.0 + 2.5 \times \sin 80 = 3.462019383 \quad (Y3.462)$$

### ➡ STEP 4

If the XY coordinates are calculated in the same order as they will appear in the CNC program, the listing of all hole locations can be used in that order (CCW shown):

Hole #1: X3.8492 Y1.8551  
Hole #2: X3.4151 Y2.6070  
Hole #3: X2.7500 Y3.1651  
Hole #4: X1.9341 Y3.4620

Now, program for the hole arc pattern can be written, using the XY coordinates for each hole location from the established calculations - program O2708:

```
O2708 (ARC PATTERN)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X3.8492 Y1.8551 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 X3.4151 Y2.607
N7 X2.75 Y3.1651
N8 X1.9341 Y3.462
```

```

N9 G80 M09
N10 G28 Z0.1 M05
N11 G28 X1.9341 Y3.462
N12 M30
%
```

There are two other methods (perhaps more efficient) to program an arc hole pattern. The first method will take an advantage of the local coordinate system (G52), described on *page 399*. The second method will use polar coordinate system (optional on most controls), described later in this chapter - in program O2710 (*page 233*).

## BOLT HOLE CIRCLE PATTERN

A pattern of equally spaced holes along the circumference of a circle is called a *bolt circle pattern* or a *bolt hole pattern*. Since the circle diameter is actually pitch diameter of the pattern, another name for a bolt circle pattern of holes is a *pitch circle pattern*. Programming approach is very similar to any other pattern, particularly to the arc hole pattern and mainly depends on the way the bolt circle pattern is oriented and how the drawing is dimensioned.

A typical bolt circle in a drawing is defined by XY coordinates of the circle center, its radius or diameter, the number of equally spaced holes along the circumference, and angular orientation of holes, usually in relation to the X-axis (that is to the zero degrees).

A bolt circle can be made up of any number of equally spaced holes, although some numbers are much more common than others, for example: 4, 5, 6, 8, 10, 12, 16, 18, 20, 24.

In later examples, the 6-hole and the 8-hole patterns (and their multiples) have *two* standard angular relationship to the X-axis at zero degrees

*Figure 27-9* is a typical bolt circle drawing. Programming approach for a bolt circle is similar to that of arc pattern.

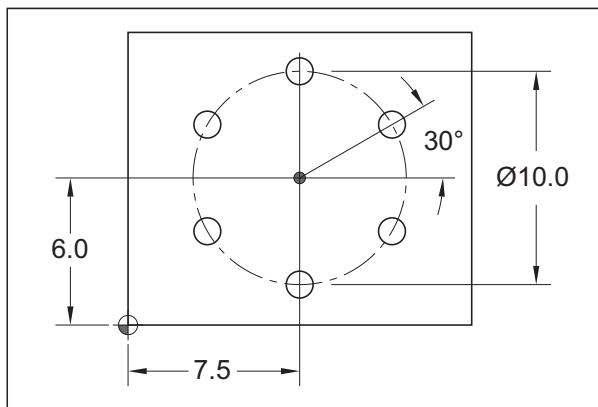


Figure 27-9

Bolt circle hole pattern - program O2709

First, select the machining location to start from, usually at program zero. Then find the absolute XY coordinates for the center of given circle. In the illustration, bolt pattern center coordinates are X7.5Y6.0. There will be no machining at this location, but the circle center will be the starting point for calculations of all holes on the bolt circle. When the circle center coordinates are known, write them down. Each hole coordinate located on the circumference must be adjusted by one of these values. When all calculations for the first hole are done (based on circle center), continue to calculate the X and Y coordinates for all other holes on the circle circumference, in an orderly manner.

In example O2709 are 6 equally spaced holes on the bolt circle diameter of 10.0 inches. That means there is a 60° increment between holes (360/6=60). The most common starting position for machining is at the boundary between quadrants. That means the most likely start will be at a position that corresponds to the 3, 12, 9 or 6 o'clock on the face of an analog watch. In this example, start will be at the 3 o'clock position. There is no hole at the selected location, the nearest one will be at 30° in the counterclockwise direction. A good idea is to identify this hole as a hole number 1. Other holes may be identified in a similar way, preferably in the order of machining, relative to the first hole.

Note that each calculation uses the same format. Other mathematical approach can be used as well, but watch the consistency of all calculations - *only the angle changes*:

Hole #1 - at 30 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 30 = 11.830127 & (X11.8301) \\
 Y &= 6.0 + 5.0 \times \sin 30 = 8.500000 & (Y8.5)
 \end{aligned}$$

Hole #2 - at 90 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 90 = 7.5000000 & (X7.5) \\
 Y &= 6.0 + 5.0 \times \sin 90 = 11.0000000 & (Y11.0)
 \end{aligned}$$

Hole #3 - at 150 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 150 = 3.16987298 & (X3.1699) \\
 Y &= 6.0 + 5.0 \times \sin 150 = 8.50000000 & (Y8.5)
 \end{aligned}$$

Hole #4 - at 210 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 210 = 3.16987298 & (X3.1699) \\
 Y &= 6.0 + 5.0 \times \sin 210 = 3.50000000 & (Y3.5)
 \end{aligned}$$

Hole #5 - at 270 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 270 = 7.50000000 & (X7.5) \\
 Y &= 6.0 + 5.0 \times \sin 270 = 1.00000000 & (Y1.0)
 \end{aligned}$$

Hole #6 - at 330 degrees

$$\begin{aligned}
 X &= 7.5 + 5.0 \times \cos 330 = 11.830127 & (X11.8301) \\
 Y &= 6.0 + 5.0 \times \sin 330 = 3.500000 & (Y3.5)
 \end{aligned}$$



Once all coordinates are calculated, program is written in the same way as for other patterns, shown already:

```
O2709 (BOLT CIRCLE PATTERN)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X11.8301 Y8.5 S900 M03
N4 G43 Z1.0 H01 M08
N5 G99 G81 R0.1 Z-0.163 F3.0
N6 X7.5 Y11.0
N7 X3.1699 Y8.5
N8 Y3.5
N9 X7.5 Y1.0
N10 X11.8301 Y3.5
N11 G80 M09
N12 G28 Z0.1 M05
N13 G91 G28 X0 Y0
N14 M30
%
```

It would be more logical to select the bolt circle center as program zero, rather than the part lower left corner. This method would eliminate modifications of bolt circle center position for each coordinate value and perhaps reduce a possibility of an error. At the same time, it would make it more difficult to set work offset (G54) on the machine. The best solution is to use G52 *local coordinate offset* method. This method is especially useful for those jobs that require translation of the bolt circle pattern (or any other pattern) to other locations of the same part setup. For details on local coordinate offset and the G52 command, see *page 399*.

### ◆ Bolt Circle Formula

In the previous calculations, there are many repetitious data. Basic methods are the same, only the angle changes. This type of calculation offers an excellent opportunity for creating a common formula that can be used, for example, as the basis of a computer program, calculator data input, etc. *Figure 27-10* shows the basic data for such a formula.

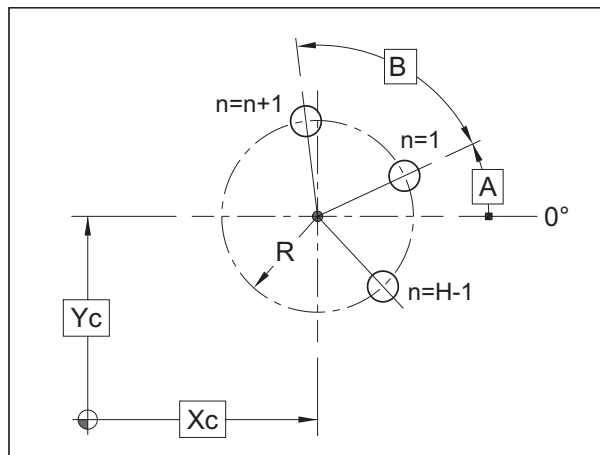


Figure 27-10

Basic data for a formula to calculate bolt hole pattern coordinates

Using following explanations and formulas, coordinates for any hole in any bolt circle pattern can be calculated easily. The formula is similar for both axes - study it well:

$$X = \cos ((n-1) \times B + A) \times R + X_c$$

$$Y = \sin ((n-1) \times B + A) \times R + Y_c$$

where ...

- X = Hole X coordinate
- Y = Hole Y coordinate
- n = Hole number counter - CCW from 0°
- H = Number of equally spaced holes
- B = Angle between holes = 360 / H
- A = First hole angle - from 0°
- R = Bolt circle radius **or** bolt circle diameter/2
- X_c = Bolt circle center from the X origin
- Y_c = Bolt circle center from the Y origin

### ◆ Pattern Orientation

Bolt circle pattern orientation is specified by the angle of the first hole from 0° of the bolt circle.

In daily applications, bolt circle patterns will have not only different number of holes, but different orientations as well. Bolt circles most commonly affected are those whose number of equally spaced holes is based on the multiples of six (6, 12, 18, 24, ...) and multiples of eight (4, 8, 16, 24, 32, ...). This relationship is important, since the orientation of the first hole will influence the position of all other holes in the bolt circle pattern.

*Figure 27-11* shows relationship of the first hole position to the 0° location of a bolt circle. 0° location is equivalent to the 3 o'clock position or the East direction.

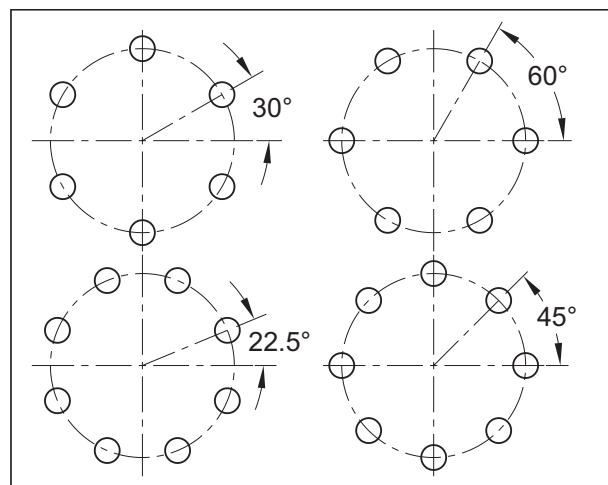


Figure 27-11

Typical orientations of a six and eight hole bolt circles

## POLAR COORDINATE SYSTEM

So far, all mathematical calculations relating to the arc or bolt circle pattern of holes have been using lengthy trigonometric formulas to calculate each coordinate. This seems to be a slow practice for a modern CNC system with a very advanced computer. Indeed, there is a special programming method available (usually as a control option) that takes away all tedious calculations from an arc or bolt circle pattern - it is called the *polar coordinate system*. There are two polar coordinate functions available, always recommended to be written as a separate block:

G15	Polar coordinate system cancel	OFF
G16	Polar coordinate system	ON

Program input values for bolt hole or arc patterns may be programmed with the polar coordinate system commands. Check first options of the control before using this method. Programming format is similar to that of programming fixed cycles. In fact, the format is identical - for example:

N.. G9.. G8.. X.. Y.. R.. Z.. F..

Two factors distinguish a standard fixed cycle from the same cycle used in polar coordinate mode.

The first factor is the initial command G that precedes the cycle - no special G-code is required for a standard cycle. For any cycle programmed in polar coordinate system mode, the preparatory command G16 must be issued to activate polar mode (ON mode). When polar coordinate mode is completed and no longer required in the program, command G15 must be used to terminate it (OFF mode). Both commands must be in a separate block:

```

N.. G16                                (POLAR COORDINATES ON)
N.. G9.. G8.. X.. Y.. R.. Z.. F..
N.. ...
N.. ...                                (MACHINING HOLES)
N.. ...
N.. G15                                (POLAR COORDINATES OFF)

```

The second factor is *meaning* of the X and Y words. In a standard fixed cycle, XY words define the hole position in rectangular coordinates, typically as an absolute location. In polar mode and G17 in effect (XY plane), both words take on a totally different meaning - specifying a *radius* and an *angle*:

- X-word becomes radius of the bolt circle
- Y-word becomes angle of the hole, measured from 0°

Figure 27-12 illustrates all three basic input requirements for a polar coordinate system.

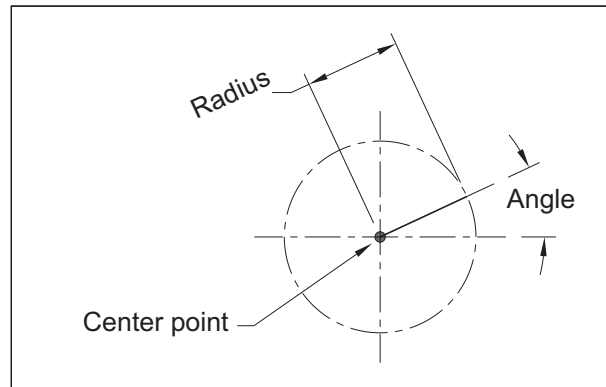


Figure 27-12

Three basic characteristics of polar coordinates

In addition to the X and Y data, polar coordinates also require the *center* of rotation (pivot point). This is the *last point* programmed *before* G16 command. Earlier, data in program O2708 and Figure 27-8 were calculated using trigonometric functions. With the polar coordinates control option, final program can be much simplified - O2710:

```

O2710 (ARC PATTERN - POLAR)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X1.5 Y1.0 S900 M03 (PIVOT POINT)
N4 G43 Z1.0 H01 M08
N5 G16                                (POLAR COORDINATES ON)
N6 G99 G81 X2.5 Y20.0 R0.1 Z-0.163 F3.0
N7 X2.5 Y40.0
N8 X2.5 Y60.0
N9 X2.5 Y80.0
N10 G15                               (POLAR COORDINATES OFF)
N11 G80 M09
N12 G91 G28 Z0 M5
N13 G28 X0 Y0
N14 M30
%

```

In the next program O2711, holes are equally spaced on bolt circle circumference. Dimensions in Figure 27-13 are applied to the polar coordinate programming method.

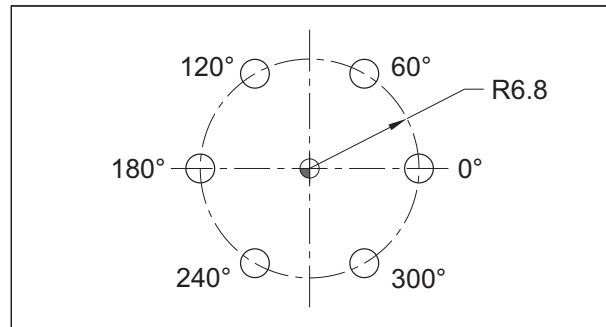


Figure 27-13

Polar coordinate system applied to bolt hole circle - program O2711

```

O2711 (G15-G16 POLAR EXAMPLE)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S900 M03      (PIVOT POINT)
N4 G43 Z1.0 H01 M08
N5 G16                                (POLAR COORDINATES ON)
N6 G99 G81 X6.8 Y0 R0.1 Z-0.163 F3.0
N7 X6.8 Y60.0
N8 X6.8 Y120.0
N9 X6.8 Y180.0
N10 X6.8 Y240.0
N11 X6.8 Y300.0
N12 G15                                (POLAR COORDINATES OFF)
N13 G80 M09
N14 G91 G28 Z0 M05
N15 G28 X0 Y0
N16 M30
%
```

Note that the center of polar coordinates (also called *pivot* point) is defined in block N3 - it is the *last X and Y location* programmed *before* the polar command G16 is called. In the program example O2711, the center is at X0Y0 location (block N3) - compare it with program O2710.

Both, the radius and angle values, may be programmed in either absolute mode G90 or incremental mode G91.

If a particular job requires many arc or bolt hole patterns, polar coordinate system option will be worthy of purchase, even at the cost of adding it later. If the Fanuc *User Macro* option is installed, macro programs can be created *without* having polar coordinates on the control and offer even more programming flexibility.

### ◆ Plane Selection

Subject of planes is briefly described on *page 247*, and in detail as a separate chapter, starting on *page 279*.

There are three mathematical planes, used for variety of applications, such as polar coordinates.

G17	XY plane selection
G18	ZX plane selection
G19	YZ plane selection

Selection of a correct plane is *extremely critical* to the proper use of polar coordinates. Always make it a habit to program the necessary plane, even the default G17 plane.

G17 plane is known as the XY plane. If working in another plane, make double sure to adhere to the following rules:

The first axis of selected plane  
is programmed with the **arc radius** value

The second axis of selected plane  
is programmed as the **angular position** of the hole

In a table format, all three plane possibilities are shown. Note, that if no plane is selected in the program, the control system defaults to G17 - the XY plane.

G-code	Selected plane	First axis	Second axis
G17	XY	X = radius	Y = angle
G18	ZX	Z = radius	X = angle
G19	YZ	Y = radius	Z = angle

Most polar coordinate applications take place in the default XY plane, programmed with G17 command.

### ◆ Order of Machining

The order in which holes are machined can be controlled by changing the *sign* of the angular value, while the polar coordinate command is in effect. If the angular value is programmed as a positive number, the order of machining will be counterclockwise, based on 0° position. By changing the value to a negative number, the order of machining will be clockwise (reversed).

This feature is quite significant for efficient programming approach, particularly for a large number of various bolt hole patterns. For example, a center drilling or spot drilling operation can be programmed very efficiently with positive angular values (counterclockwise order). Start will be at the first hole and, after the tool change, drilling can continue in reverse order, starting with the last hole. All angular values will now be negative, for the clockwise order of a subsequent tool. This approach requires a lot more work in standard programming, when polar coordinates are not used. The polar coordinate application using G16 command eliminates all unnecessary rapid motions, therefore shortening the overall cycle time.

Face milling is a machining operation that controls the height of a machined part. For most applications, face milling is a relatively simple operation, at least in the sense that it usually does not include any special contouring motions. Cutting tool used for face milling is typically a multi tooth cutter, called a *face mill*, although end mills may also be used for certain face milling operations, usually within small areas. Top surfaces machined with a face mill are generally perpendicular to the facing cutter axis. In CNC programming, face milling operations are fairly simple, although two important considerations are rather critical:

- Face mill diameter selection
- Tool initial starting position in relation to the part

It helps to have some experience and knowledge of face milling principles, such as the right cutter and insert selection, distribution of cuts, machine power consumption, and several other technical considerations. Some of the basic ones are covered in this chapter, but manufacturers' tooling catalogues and various technical references will be a more in-depth source.

## FACE MILL SELECTION

Like all milling operations, face milling employs a cutting tool that rotates while the machined part remains stationary. Face milling requires that a specific amount of material be removed from the top of part, at one or several depth levels, in a single cut or multiple cuts. Programming for face milling is so effortless that, in fact, many programmers do not pay sufficient attention to proper selection of the face milling cutter, proper inserts, and they do not even consider machine requirements and capabilities.

A typical face mill for CNC work is a multi tooth cutter with interchangeable carbide inserts. High speed steel face mills are not recommended for CNC work, although an HSS end mill can be a suitable choice to face mill small areas or areas hard to get to in any other way. Typical to a face milling operation is the fact that not all inserts of the milling cutter are actually working at the same time. Each insert works only within a part of one complete revolution. This reality may be an important consideration when trying to establish an optimum tool life for a face milling cutter. Face milling *does* require significant power resources from the machine tool. For insert setup in the cutter body, it is very important to have all inserts properly mounted and aligned.

### ◆ Basic Selection Criteria

Based on the job to be machined, selection of a face mill cutter has to take into account several possible situations:

- CNC machine specifications and condition
- Part material to be machined
- Setup method and work holding integrity
- Method of mounting
- Cutter overall construction
- Face mill diameter
- Number of inserts and insert geometry

The last two items, cutter diameter and insert data, will influence actual program development the most, although other items are important as well.

### ◆ Face Mill Diameter

One of the most important considerations for face milling operations is the cutter size selection. For a single face cut, the ideal width of a face mill cutter should be about 1.3 to 1.6 times larger than the material width. For example, a single 2.5 inches wide cut will benefit from a Ø4.0 face mill as a suitable size. This suggested 1.3 to 1.6 times ratio will assure a good formation of chips and their clearout from the part. For multiple cuts, always select the largest diameter cutter that can be used for the job, always considering the machine power rating, cutter and insert geometry, setup rigidity, depth and width of each cut, and other machining related factors.

The basic purpose of face milling is to machine off the top of a part to specified height (thickness). For this type of machining, select a reasonable face mill diameter size, which often means to use relatively large diameter tools. Sizes in the range of 50 to 300 mm (2 to 12 inches) are not unusual, depending on the CNC machine and the job.

One important consideration in face milling is the relationship between the tool *diameter* and the actual cutting *width*. Take, for instance, a Ø5.0 inch face mill. All tooling catalogues list the *nominal* size of the face mill (5 inches in the example), although the body diameter can be found in the catalogue as well. Nominal diameter always refers to the *full cutting width*. There is no way to tell the actual diameter of a tool holder body from the nominal size alone, it has to be looked up in a tooling catalogue. Normally, the actual cutter body size is not needed, except in those cases

where face milling takes place close to part walls or other obstacles. Size of the cutter body may prevent access to some areas of the part and may interfere elsewhere as well. The *Figure 28-1* shows some typical configurations.

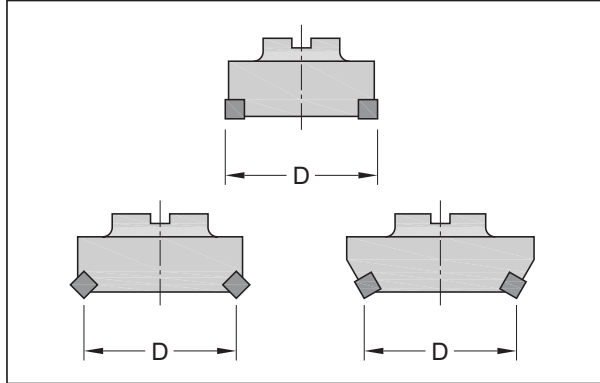


Figure 28-1

Nominal diameter 'D' of various face mill cutters

### ◆ Insert Geometry

Learn and become familiar with the basic terminology of milling cutters in order to understand various terms used in programming. Many tooling companies have available catalogues and technical booklets for their cutters and inserts that explain how the cutter work as well as all related terms. This information can be downloaded from their web sites or made available in printed form, on request. Keep in mind that cutting tool technology does change quite rapidly and constant improvements are being made. For programming purposes in this chapter, the brief look is only at the very basic items of insert geometry for face milling cutters.

Insert geometry and methods of insert mounting are determined by original design that controls the insert activity in the material during a cut. These factors strongly influence quality of the cutting. There are typically *three* general categories, based on the face mill cutting rake angle (also known as the *rake angle*):

- Positive geometry ... single / double
- Negative geometry ... single / double
- Combination of both ... positive / negative

Any detailed variations are too numerous to list, but a short overview offers at least some basis for further studies.

#### Positive Geometry

*Positive geometry* milling cutters require less machining power than negative cutters, so they may be more suitable on CNC machines that have limited power rating, usually small machines. They offer a good chip breaking characteristics and are a good choice for machining steel materials when the cutting load is not too heavy. Positive inserts are generally single sided, therefore less economical.

#### Negative Geometry

*Negative geometry* face mills offer very high insert edge strength and usually require a heavy duty machine and a fairly robust overall setup. Its side effects are poor chip formation for steel but not necessarily for some kinds of cast irons, where there is hardly any curling effect during chip formation. Their main benefit is the insert economy, since negative inserts are generally double sided, offering up to eight cutting edges for a single square insert, inserted in pockets of a face mill.

#### Double Negative Geometry

*Double negative geometry* can be used only if the CNC machine has sufficient power rating - and - both the cutting tool and the machined part are firmly mounted within a strong and rigid setup. Cast iron or certain hard materials will usually benefit from using double negative geometry. Chips do have the tendency to concentrate towards the machined part and do not fly away from the part with ease, possibly causing a chip jamming against the cutting insert or wedging themselves in confined areas. *Positive/negative* inserts should eliminate this clogging problem.

#### Positive / Negative Geometry

*Positive / Negative geometry* is most beneficial to those face milling operations where chip clogging could become problematic. This dual geometry design offers strength of the negative insert with the capability of 'chip curling' into a spiral shape. This design is usually most suitable for full width face milling.

As is recommended with any tools, always consult supplied technical specifications of the cutting tool manufacturers and compare several products before deciding on the most suitable choice for a particular work. Face mills and their inserts come literally in hundreds of varieties and each manufacturer claims superiority over the competition.

## CUTTING CONSIDERATIONS

To program a cutting motion for a face mill, it is important to understand how a face mill works best under different conditions. For example - unless a specially designed face milling cutter and proper insert geometry, shape and grade are used, try to avoid face milling a part width that is equal to, or only a slightly larger than, the face mill diameter. Full width face milling cut may cause the insert edge to wear out prematurely and the chip to 'weld' itself to the insert. Not only the insert suffers in form of a wear out, the actual part surface finish suffers as well. In some more severe cases, the cutting insert may have to be discarded prematurely, increasing the overall machining costs.

*Figure 28-2* shows desirable and undesirable relationship between cutter diameter and part width during face milling.



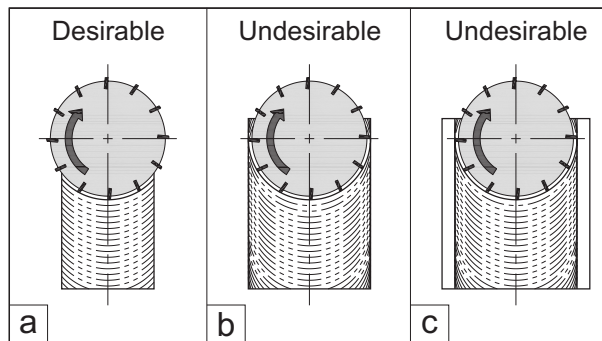


Figure 28-2

Schematic relationship of the cutter diameter and the part width. Only the cutter size (a) is desirable, although not its position.

All three illustrations show only relationship of the cutter diameter to the part width - they do *not* suggest the actual method of cutter entry into the material. One of the most important considerations for CNC programming of a face mill is the *angle* the milling cutter enters into the material.

### ◆ Angle of Entry

A face mill entry angle is determined by position of the cutter centerline relative to the part edge. If a part can be milled with a single cut, avoid situations where the cutter centerline position matches the part centerline. This *neutral* position causes a chatter and poor finish. Offset the cutter *away* from part centerline, either for a *negative* cutter entry angle, or a *positive* cutter entry angle. Figure 28-3 shows both types of entry angles and their effects.

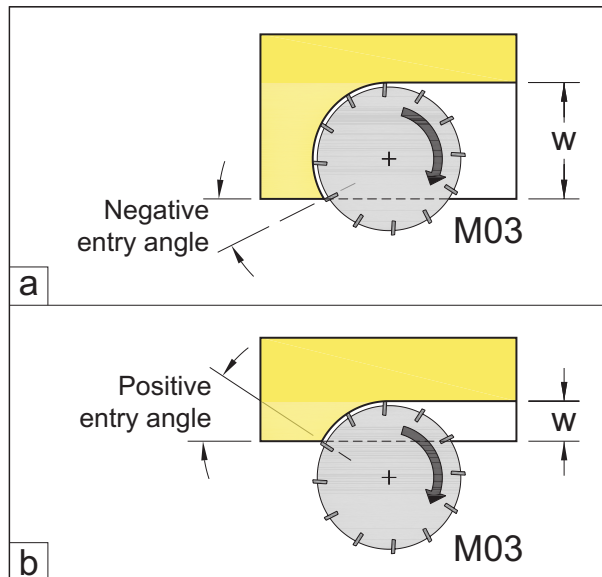


Figure 28-3

Insert entry angle into the part.  $W$  = width of cut  
(a) at the strongest insert point - negative entry angle  
(b) at the weakest insert point - positive entry angle

A neutral angle of entry (not shown) has the cutter centerline coincident with the part edge. Needless to say, when the cutting insert enters material, a certain force is required. During positive entry angle, the weak cutting edge has to absorb most of the forces. Since insert edge it is the weakest part of the insert itself, a positive entry angle may cause a breakage or at least some insert chipping. Normally, this entry method is *not* recommended.

Negative entry insert angle absorbs the entry force in the middle - at the strongest point of the insert. This is the preferred method, as it increases insert usability. It is always a good idea to keep the face mill center *within* the part area, rather than away from it. That way, the cutting insert will always enter at the preferred negative angle.

All these examples assume a solid part material being machined. If the face mill has to travel over some *empty spaces*, the cut will be interrupted. The *entry into* and *exit from* the part during interrupted cut will cause the cutter entry angle to be variable, not constant. As many other factors have to be considered in face milling, take these recommendations and suggested preferences only as guidelines. Always consult a tooling representative on the best method of handling a particular face milling job, particularly for materials that are difficult to machine.

### ◆ Milling Mode

In milling, the programmed cutting direction, relative to the table motion direction, is always very important. In fact, this factor so important that it is discussed in several sections of this handbook and covers a subject called the *milling mode*.

Traditionally, there are *three* milling mode possibilities available in milling operations:

- Neutral milling mode
- Conventional milling mode
- Climb milling mode

A neutral milling mode is a situation where the cutter follows the center line of a slot or a face, climb milling on one side and conventionally milling on the other side of center line. The *conventional* milling mode is also called the '*up*' mode and the *climb* milling mode is also called the '*down*' mode. These are all correct terms, although the terminology may be a little confusing. The terms *climb milling* and *conventional milling* are more often used with peripheral milling than with face milling, although exactly the same principles do apply for all milling. For most face milling cuts, climb milling mode is the best overall choice.

In Figure 28-4, example (a) illustrates the neutral *cutting* mode, example (b) shows the so called *down* cutting mode (or *climb* milling mode) and example (c) shows the so called *up* cutting mode (or *conventional* milling mode).

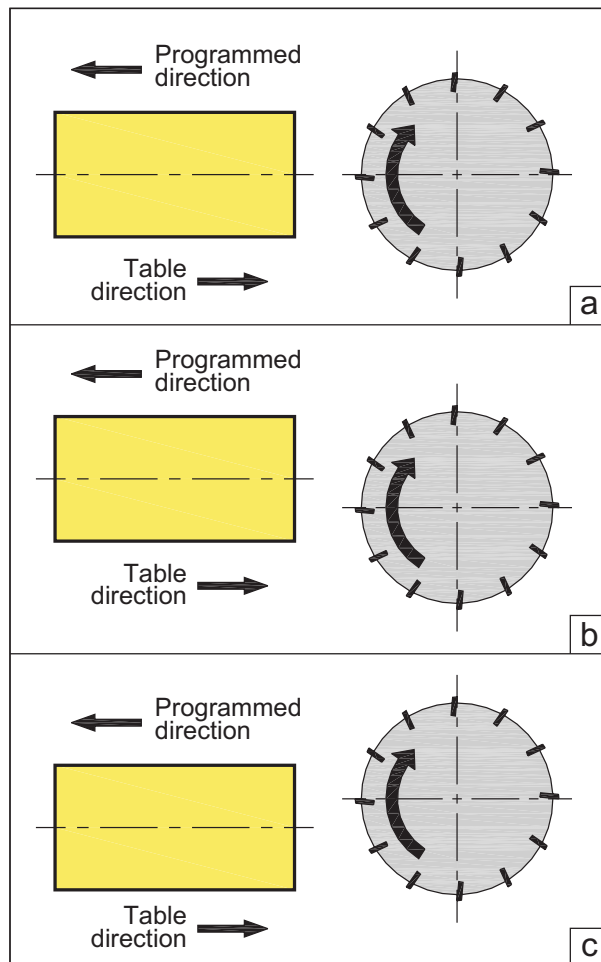


Figure 28-4

Face milling modes:

(a) Neutral milling mode

(b) Climb or 'down' milling mode

(c) Conventional or 'up' milling mode

### ◆ Number of Cutting Inserts

Depending on the face mill size, the common tool is a multi tooth cutter. A traditional tool called *fly-cutter* has usually only a single cutting insert and is not a normal tool of choice in CNC. Relationship of the number of inserts in the cutter to the effective cutter diameter is often called the *cutter density* or *cutter pitch*.

Typical face mills will belong into one of these three categories, based on the cutter density:

- Coarse density ... coarse pitch of inserts
- Medium density ... medium pitch of inserts
- Fine density ... fine pitch of inserts

As an overall general type, a coarse density cutter is usually a suitable choice. The more cutting inserts are engaged in material simultaneously, the more machining power will be required. Regardless of the insert density, it is important to have sufficient cutting clearances - the chips must not clog the cutter, but fly out freely.

At all times, at least one cutting insert must be in contact with the material, which will prevent heavy interrupted cut, with the possible damage to the cutter and to the machine. This situation may occur if a large face mill diameter is used for a very narrow part width.

## PROGRAMMING TECHNIQUES

Although defined earlier as a relatively simple operation, face milling can be programmed much better if some common sense points are observed. Since face milling often covers a large cutting area, it is important to consider carefully the actual tool path from its start position to its end position. Here is a list of some points that should be evaluated for any face milling operation:

- Always plunge-in to the required depth away from the part (in the air)
- If surface finish is important, change the cutter direction away from the part (in the air)
- Keep the cutter center within the part area for better cutting conditions
- Typically, select a cutter diameter that is about 1.5 times larger than the intended width of cut

Figure 28-5 shows a simple plate used for examples.

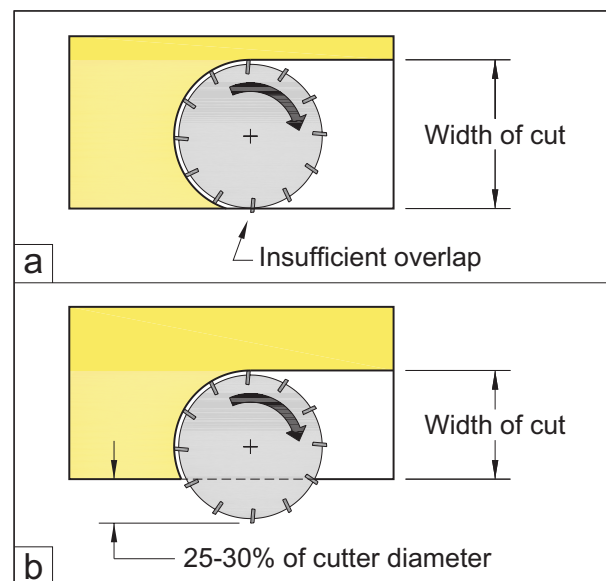


Figure 28-5

Width of cut in face milling - (b) is the recommended method

Figure 28-5a illustrates the *incorrect* and Figure 28-5b the *correct* width of a face mill cut. In example (a), the cutter is engaged in the part with its full diameter, causing friction at the cutting edge and decreasing tool life. Example (b) keeps only about 2/3 of the cutter diameter in the work, which causes a suitable chip thickness, as well as favorable angle of insert entry into the material.

### ◆ Single Face Mill Cut

The first face mill programming example (in G20 mode), uses a 5×3 plate (1 inch thick) that has to be face milled along the whole top surface to the final thickness of 0.800. Figure 28-6 shows this simple drawing.

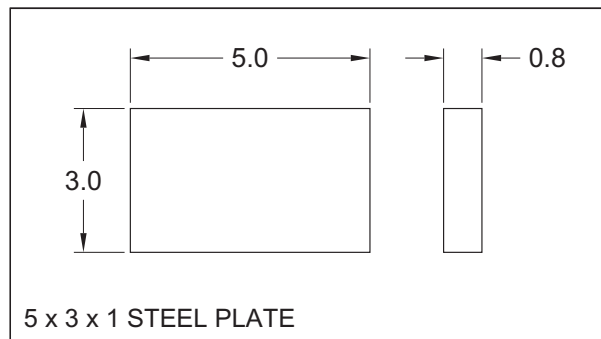


Figure 28-6

Example of a single face mill cut - program O2801

From the drawing is apparent that the face milling will take place *along* the part, so the X-axis horizontal direction will be selected. Before the program can be started, there are two major decisions to be made:

- Face mill diameter
- Start and end position of the cut

There are other important decisions to make, but these two are the most critical.

The part is only 3 inches wide, so a face mill that is *wider* than 3.0 inches should be selected. Although a Ø4.0 inch face mill seems like a natural choice, let's see if it conforms to the conditions that have been established earlier. The cutter diameter should be 1.3 to 1.6 larger than the width of cut. In this case,  $3 \times 1.3 = 3.90$  and  $3 \times 1.6 = 4.80$ . With a Ø4.0 face mill, that means only 1.33 times larger. Considering the need for the cutter to overlap both edges of the part, selection of a *five* inch face mill diameter is better.

Once the face mill diameter has been finalized, concentrate on the *start* and *end* positions. For safety reasons, plunging to depth has to start *away* from the part, in the air. Decision to cut along the X-axis (horizontally) has been made, so the question is whether from left to right or from right to left. It really does not matter, except for the direction of chip flow, so selection from right to left is probably a better choice.

Part X0 Y0 is at the lower left corner. To establish the starting X-position, consider the part length of 5.0 inches, the cutter radius ( $5/2=2.5$ ) and some clearance (0.25). The start X-axis position will be the sum of these values, X7.75. For the Y-axis start position calculation, consider the overhangs on both edges and select climb milling mode at the same time. Actually, the climb milling will be combined with a little of conventional milling, which is quite normal for face milling operations. Figure 28-7 shows the cutting tool start position at X7.75 Y1.0, and its end position at X-2.75 Y1.0, as well as the details of calculations.

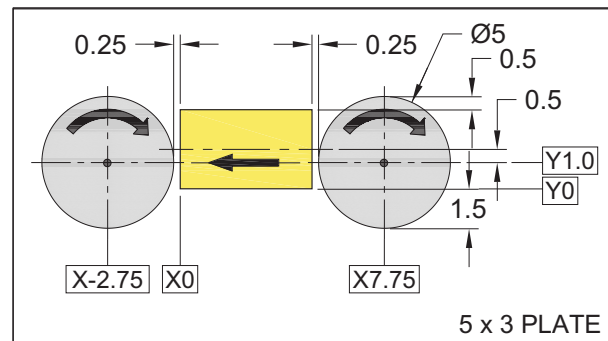


Figure 28-7

Face mill positions for a single face mill cut example

Position Y1.0 was based on the desire to have about one quarter to one third of the cutter diameter overhang at the part edge, for best insert entry angle. As 1.5 inch overhang is 30% of the cutter diameter, programmed absolute Y-position was established at a convenient Y1.0.

Now, part program for the single face milling cut can be written, with the top of finished face as program zero (Z0). Only one face cut is used - program example O2801.

```
O2801
(SINGLE FACE MILLING CUT)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X7.75 Y1.0 S344 M03
N4 G43 Z1.0 H01
N5 G01 Z0 F50.0 M08
N6 X-2.75 F21.0
N7 G00 Z1.0 M09
N8 G28 X-2.75 Y1.0 Z1.0
N9 M30
%
```

Spindle speed and feedrate are based on 450 ft/min surface speed, 0.006" per tooth and 8 cutting inserts, used only as reasonable values. Note the Z-axis approach in block N4. Although the tool is well above an empty area, the rapid motion is split between blocks N4 and N5, for safety reasons. With increased confidence, rapid motion to Z0 directly may be an option, if desired. This example shows the program Z0 at top of the *machined* part, not the stock face.

### ◆ Multiple Face Mill Cuts

General principles applying to a single face cut do apply equally to multiple face cuts. Since a face mill diameter is often too small to remove all material in a single pass on a large material area, several passes must be programmed at the same depth.

There are several cutting methods for a large area to be face milled and each may produce good machining conditions under certain circumstances. The most typical methods are *multiple unidirectional* cutting and *multiple bidirectional* cutting (called zigzag) - at the same Z-depth.

**Multiple unidirectional** cuts start from the same position in one axis, but changes the position in the other axis, above the part. This is a common method of face milling, but it lacks efficiency, because of frequent rapid return motions.

**Multiple bidirectional** cuts, often called zigzag cutting, are also used frequently; they are more efficient than the unidirectional method, but cause the face mill to change the climb milling method to the conventional method and vice versa. This method may work for some jobs, but may not be suitable for all materials.

In the next two illustrations, *Figure 28-8* shows schematically a unidirectional milling face. *Figure 28-9* shows a bidirectional face milling.

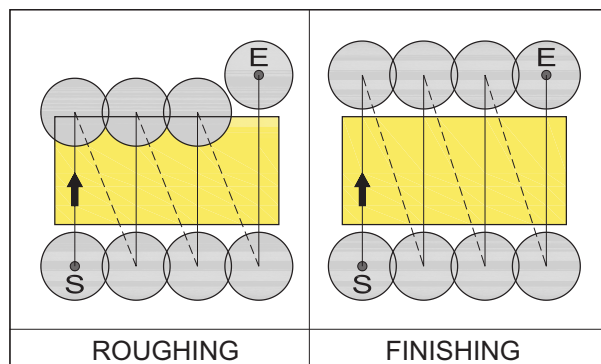


Figure 28-8

Unidirectional approach to a multiple face cut for rough and finish face milling

Compare the XY motions of these two methods. In addition, a tool path difference (cutter position) between roughing and finishing is also shown. The cutting direction may be either along the X or along the Y-axis, but the principles of cutting motion will remain the same.

Note the *start* position (S) and the *end* position (E) in the two illustrations. They are indicated by the heavy dot at the center of cutter. Regardless of the cutting method, face milling cutter is always in a clear position at the start and end of cutting, mainly for safety reasons.

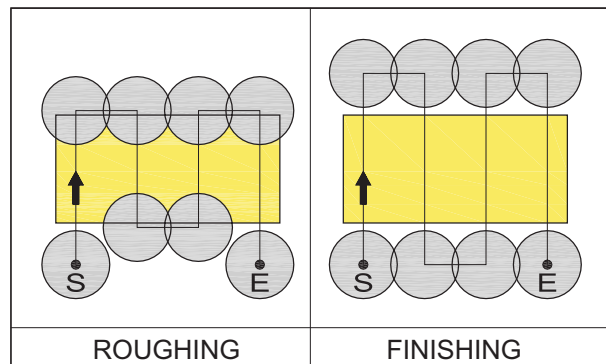


Figure 28-9

Bidirectional approach to a multiple face cut for rough and finish face milling

There is another fairly efficient method that cuts only in one mode, normally in climb milling mode. This method may remind of a circular or a spiral motion (along the XY axes) and is the most recommended method. It combines both previous methods and is illustrated in *Figure 28-10*.

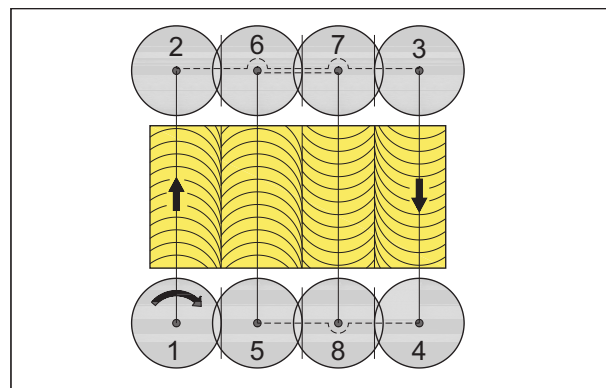


Figure 28-10

Schematic tool path representation for the climb face milling mode, applied to a unidirectional cutting

The illustration shows the *order* and *direction* of all individual tool motions. Main idea is to make each cut approximately the same width, with only about 2/3 of the diameter cutting at any time, and always in climb milling mode.

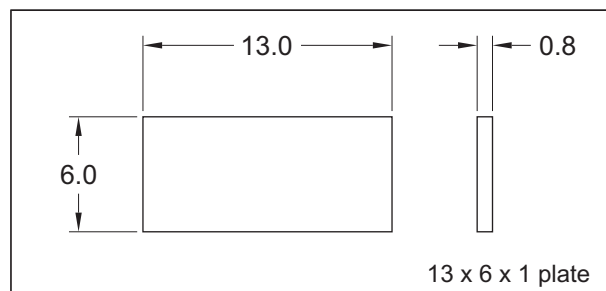


Figure 28-11

Example of a multiple face mill cut - program O2802

A programming example for multiple face milling cuts is based on the drawing shown in *Figure 28-11*. The previously discussed basics are applied and should present no difficulty in understanding the program.

```
O2802
(MULTIPLE FACE MILLING CUTS)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0.75 Y-2.75 S344 M03 (POS 1)
N4 G43 Z1.0 H01
N5 G01 Z0 F50.0 M08
N6 Y8.75 F21.0 (POS 2)
N7 G00 X12.25 (POS 3)
N8 G01 Y-2.75 (POS 4)
N9 G00 X4.0 (POS 5)
N10 G01 Y8.75 (POS 6)
N11 G00 X8.9 (POS 7 - 0.1 OVERLAP)
N12 G01 Y-2.75 (POS 8 - END)
N13 G00 Z1.0 M09
N14 G28 X8.75 Y-2.75 Z1.0
N15 M30
%
```

In the program O2802, all relevant blocks are identified with tool positions corresponding to the numbers in the earlier *Figure 28-10*.

The 13.0 inch part width was separated into four equal cutting widths of 3.25 each, which is a little less than 2/3 of a  $\varnothing 5.0$  cutter, its usable width of cut. Clearances of 0.25 off the part are the same as for the single face cut example. One major deviation from the norm was the motion to position number 7 in *Figure 28-10* and block N11 in the program. The last cutting motion is from position 7 to position 8. In order to make the surface finish better, the expected cut was overlapped at X9.0 by 0.100 to the programmed value of X8.9. In *Figure 28-12*, the schematics for O2802 program are shown, including block number references.

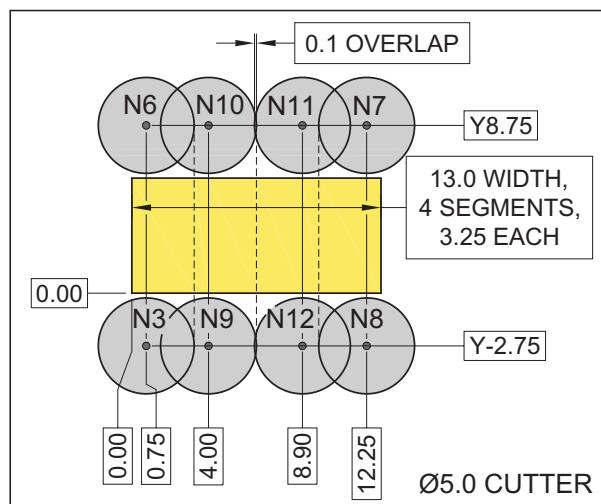


Figure 28-12

Multiple face milling details for program example O2802

Some of the examples could have been done in a shorter way along the X-axis, resulting in a smaller program. However, for the purpose of example illustrations, using the Y axis was more convenient.

## USING POSITION COMPENSATION

In both previous examples, the face mill XY start position has been calculated, considering its diameter and a suitable clearance. To use earlier O2801 program as an example, the starting position was X7.75 Y1.0. The part was 5.0 inches, plus a clearance of 0.25, plus the 2.5 inches cutter radius - total of X7.75 absolute value of the cutter center. One big disadvantage of this method is apparent when using a face mill that has a *different diameter* than the one expected by the program. A last minute change of the face mill at the machine may cause problems. Either there will be *too much* clearance (if the new tool is smaller) - or worse - there will be *not enough* clearance (if the tool is larger). There is another way to solve this problem.

As the title of this section suggests, the solution is to use somewhat 'obsolete' *Position Compensation* feature of the control system, already described in *Chapter 17*. It is probably the *only* practical application of position compensation on modern CNC machining centers.

Revisit the example O2801 and refer to earlier *Figures 28-6* and *28-7*. Illustrations show a need to face (with a single cut) a 5×3 plate, using a  $\varnothing 5.0$  inch face mill. In order to adhere to the safety rules in machining, face mill cutter has to be positioned in an open area, away from the part. In order to keep its cutting edges away from the part by one quarter of an inch, clearance of 0.25 inches has to be incorporated *with the radius of the face mill*, which is 2.5 inches, in order to achieve the actual tool starting position for the face milling cutter.

In a face milling program, this situation will take on one of the following forms:

- Face mill radius is programmed using actual dimensions
- Position compensation method is used

In the first case, program O2801 may be the result, with the following content:

```
O2801
(SINGLE FACE MILLING CUT - NO COMPENSATION)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X7.75 Y1.0 S344 M03
N4 G43 Z1.0 H01
N5 G01 Z0 F50.0 M08
N6 X-2.75 F21.0
N7 G00 Z1.0 M09
N8 G28 X-2.75 Y1.0 Z1.0
N9 M30
%
```





## 29

## CIRCULAR INTERPOLATION

In the majority of CNC programming applications, there are only two types of tool motions related to contouring. One is *Linear Interpolation*, discussed earlier, the other one is *Circular Interpolation*, covered in this chapter. Programming method of controlling a toolpath along an arc is similar to the method of programming a toolpath along a line. The method of circular contouring is called *circular interpolation*. It is commonly used in profiling on CNC vertical and horizontal machining centers, as well as on lathes and many other CNC machines, such as simple milling machines, routers, burners, water jet and laser profilers, wire EDM, and others.

Circular interpolation is used for programming arcs or complete circles in such applications as outside and inside radiuses (blend and partial), circular pockets, spherical or conical shapes, radial recesses, grooves, corner breaks, helical cutting, even large counterbores, etc. CNC unit will interpolate a defined arc with a very high precision, if the necessary information is given in the program.

## ELEMENTS OF A CIRCLE

To understand the principles of programming various circular motions, it helps to know something about the basic geometrical entity known as the *circle*. As an entity that is quite common in everyday life, a circle has various properties that are strictly mathematical, only considered in specialized disciplines, such as Computerized Numerical Control, motion control and automation.

The following definition of a circle and several other definitions that are related to a circle are based on some common dictionary definitions - *Figure 29-1*.

A circle is defined as a closed curve on a plane, where all points have the same distance from an internal point called the circle center point

There are other similar definitions of a circle that can be found in dictionaries and mathematical books. General understanding of a circle and its various properties as described in this handbook, provides sufficient knowledge for most aspects of CNC programming. Additional knowledge will be needed for some specialized or complex programming applications. At this time, become at least reasonably familiar with basic geometrical and trigonometric relationships for arcs and circles.

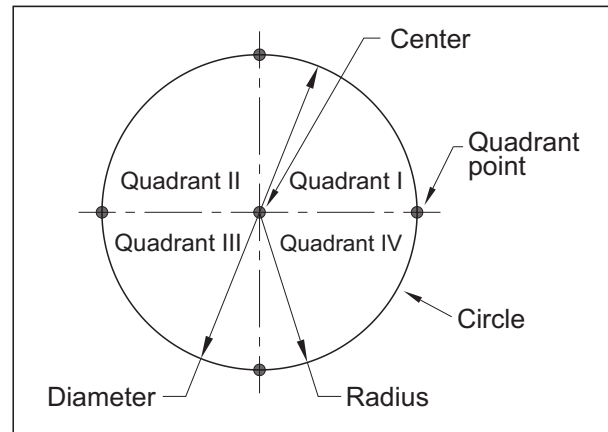


Figure 29-1  
Basic elements of a circle

### ◆ Radius and Diameter

In the simplest mathematical terms, a circle is defined by its center point and its radius. Two of the most important elements of a circle used in part programming are the circle *radius* and the circle *diameter*:

**RADIUS** of a circle is the line segment from the center point to any point on the circle

**DIAMETER** of a circle is the line segment through the center point of the circle and having both end points on the circle

Center point location of the circle is also important for CNC programming. The plural form of the word *radius* is *radii*, although the word '*radiuses*' has been accepted as a colloquial term and is considered correct. In CNC programming, radiuses and diameters are used all the time, on a daily basis for almost all contouring machines. Drawings used in machine shops use radius and diameter dimensions for even the simplest parts, with an almost unlimited number of possible applications.

Radiuses and diameters are also used in relation to the cutting tool insert designation, they are used for measuring and gauging (inspections), as well as in trigonometric calculations and various auxiliary sketches. In programming, the actual application of an arc or circle is not important, only its mathematical characteristics.

### ◆ Circle Area and Circumference

The *area* of a circle is defined by this formula:

$$A = \pi \times R^2$$

where ...

- A = Area of the circle
- R = Circle radius
- $\pi$  = Constant (3.1415927)

The *circumference* of a circle is the *length* of a circle if it were a straight line:

$$C = \pi \times D$$

where ...

- C = Circumference of the circle
- D = Circle diameter
- $\pi$  = Constant (3.1415927)

It is important to note that both the area and circumference of a circle (its actual length) are seldom used in CNC programming, although understanding their concepts presents a rather useful knowledge.

## QUADRANTS

A *quadrant* is a major property of a circle and can be defined mathematically:

A quadrant is any one of the four parts of the plane formed by the system of rectangular coordinates

It is to every programmer's benefit to understand the concept of quadrants and their applications for circular motions in milling and turning programs.

A circle is programmed in all four quadrants, due to its nature, while most arcs are programmed within one or two quadrants. When programming the arc vectors I, J and K (described later), the angular difference between the arc start and end points is irrelevant. The only purpose of arc vectors is to define a *unique* arc radius between two points.

For many arc programming projects, the direct radius can be used with the R-address, available for majority of control systems. In this case, the angular difference between the start and end points is very important, because the computer will do its own calculations to find the arc center. Arc with the angular difference of  $180^\circ$  or less, measured between the start and end points, uses an *R-positive* amount. Arc in which the angular difference is *more than*  $180^\circ$ , uses an *R-negative* amount. There are two possible choices and the radius amount alone cannot define a unique arc.

Also worth mentioning is a *mirrored* toolpath and its relationship to the quadrants. Although it is not a subject of the current chapter, mirroring and quadrants must be considered together. What happens to the toolpath when it is mirrored is determined by the quadrant where the mirrored toolpath is positioned. Details about mirror image as a programming subject start on *page 409*. For now, it should be adequate to cover a very brief overview only.

For example, if a programmed toolpath in *Quadrant I* is mirrored to *Quadrants II* or *IV*, the cutting method will be reversed. That means a *climb* milling will become *conventional* milling and vice versa. The same rule applies to a programmed toolpath in *Quadrant II* as it relates to *Quadrants I* and *III*. This is a very important consideration for many materials used in CNC machining, because climb milling in *Quadrant I* will turn into conventional milling in *Quadrants II* and *IV* - a situation that is not always desirable. Similar changes will occur for other quadrants.

### ◆ Quadrant Points

From the earlier definition should be clear that quadrants consist of two perpendicular lines that converge at the arc center point and an arc that is exactly one quarter of a circle circumference. In order to understand the subject deeper, draw a line from the center of an arc that is parallel to one of the axes and is longer than the arc radius. The line created an *intersection* point between the line and arc. This point has a special significance in part programming. It is often known as the *Quadrant Point* - or the *Cardinal Point* - although the latter term is not used very often, except in some mathematical applications. There are four quadrant points on a given circle, or four intersections of the circle with its axes. Quadrant points locations can be remembered easier by associating them with the dial of a common compass or a standard watch with an analog dial:

Degrees	Compass direction	Watch direction	Located between quadrants
$0^\circ$	EAST	3 o'clock	IV and I
$90^\circ$	NORTH	12 o'clock	I and II
$180^\circ$	WEST	9 o'clock	II and III
$270^\circ$	SOUTH	6 o'clock	III and IV

At this point of learning, it may be a good idea to refresh some terms relating to angle *direction* definition. Established industry standard (mathematics, as well as CAD, CAM and CNC) defines an absolute angular value as being *positive* in the *counterclockwise* direction and *always starting from zero degrees*. From the above table, zero degrees correspond to the *East* direction or *three o'clock* position of an analog clock - *Figure 29-2*.

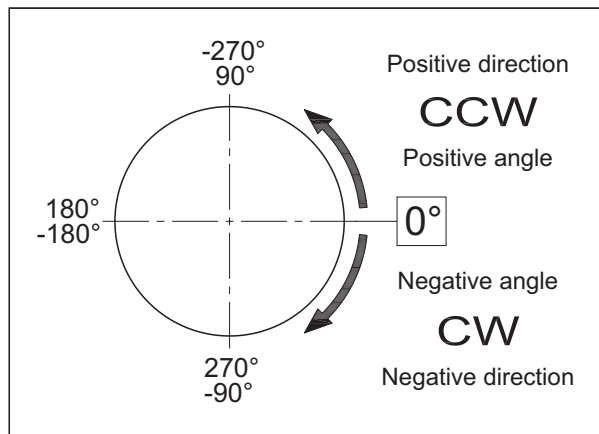


Figure 29-2

Mathematical definition of arc direction

There is another reason why quadrant points are important in CNC programming. In some cases, one or more quadrant points will be used as the arc *end points*, even if the circular cut covers more than one quadrant. This is particularly true for many older control systems, where crossing the quadrants in a single block is not allowed. Modern controls can generate arc of any length in a single block, up to 360°, with very few restrictions.

## PROGRAMMING FORMAT

Programming format for a circular interpolation toolpath must include several specifications, without which the task of cutting an arc would be impossible. The most important specifications are defined as:

- Arc cutting direction (CW or CCW)
- Arc start and end points
- Arc center and radius value

Cutting feedrate must also be in effect, discussed in more detail later in this chapter. Special modal G-codes are used for circular motion programming and additional specifications related to the circle radius are also required.

### ◆ Arc Cutting Direction

A cutting tool may move along an arc in two directions - *clockwise* (CW) or *counterclockwise* (CCW). These two terms are assigned by convention. On most machines, the motion direction is determined by looking perpendicularly at the plane in which the circular motion is programmed. The motion *from* the plane vertical axis *towards* the plane horizontal axis is clockwise, reverse motion is counterclockwise. This convention has *mathematical* origins and does not always match machine axes orientation. Machining in planes is described as a separate subject, starting on page 279; this chapter will only take a brief look.

In a typical program format, the first statement of a circular cutting motion block is the *cutting direction*. This direction defines cutting tool motion along the programmed arc, which is either clockwise or counterclockwise. The motion direction along an arc is programmed using one of two preparatory commands in the block.

### ◆ Circular Interpolation Block

All controls offer two preparatory commands associated with programming an arc direction:

G02	Circular motion clockwise	<i>CW</i>
G03	Circular motion counterclockwise	<i>CCW</i>

Both the G02 and G03 commands are modal, therefore they remain in effect until the end of program or until canceled by another command from the same G-code group, usually by another motion command.

Preparatory commands G02 and G03 are the key words used in programming to establish circular interpolation mode. Coordinate words following the G02 or G03 command are always designated within a selected plane. The plane is normally based on the available axes combinations of XY, ZX and YZ for milling or similar applications. Normally, there is no plane selection on a lathe, although some control indicate it as G18, the ZX plane.

Plane selection and the combination of circular motion parameters with the arc cutting direction determine the *arc end point*, and the R-amount specifies the *arc radius*. Special arc center *vectors* (also known as *modifiers*) are also available, if the programmer requires them.

When G02 or G03 command is activated by a CNC program, any currently active tool motion command is automatically canceled. This canceling motion is typically G00, G01 or a cycle command. All circular toolpath motions must be programmed with a cutting feedrate in effect, applying the same basic rules as for linear interpolation. That means the feedrate F must be programmed *before* or *within* the cutting motion block. If feedrate is not specified in the circular motion block, control system will automatically search for the *last* programmed feedrate. If there is no feedrate in effect at all, many controls usually return an error message (an alarm) to that effect. Cutting feedrate may be specified in one of two ways. Either directly, within the arc cutting block only or indirectly, by assuming the last active feedrate. Circular motion in rapid mode is generally not possible. Also not possible is a simultaneous three axes *circular* motion. For more details on this subject, look up the subject describing helical milling (see page 433).

On the majority of older controls, direct radius address R cannot be specified and the arc center vectors I, J and K must be used instead:

```

G02 X.. Y.. I.. J..      Milling program - CW
G02 X.. Z.. I.. K..      Turning program - CW
G03 X.. Y.. I.. J..      Milling program - CCW
G03 X.. Z.. I.. K..      Turning program - CCW

```

Control systems supporting the arc radius designation by address R will *also* accept the IJK vectors, but the reverse is *not* true. If *both* arc vectors IJK and radius R are used in the same block, the *radius* amount takes priority, *regardless of the order of words*:

```

G02 (G03) X.. Y.. R.. I.. J..
G02 (G03) X.. Y.. I.. J.. R..

```

Controls that accept *only* the vectors IJK will return an error message in case the circular interpolation block contains R-address (an unknown address).

### ◆ Arc Start and End Points

The *start point* of an arc is the point where circular interpolation *begins*, as determined by the cutting direction. This point *must* be located on the arc and it will always be an endpoint - tangency point or an intersection - resulting in a blend radius or a partial radius respectively. The instruction contained in the start point block is sometimes called the *departure command* - Figure 29-3.

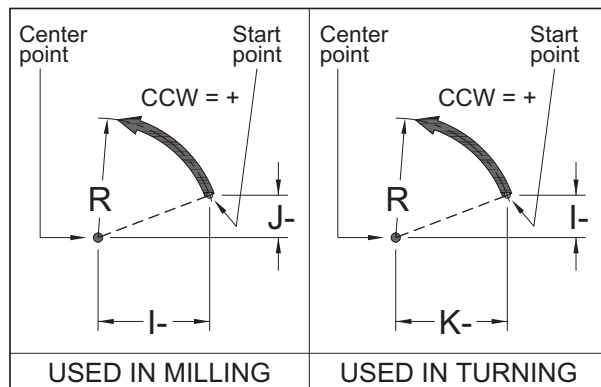


Figure 29-3

Center point and start point of an arc

Arc start point is always relative to the cutting motion direction and is represented in the program by coordinates in the block *preceding* the circular motion. In terms of a definition,

Start point of an arc is the last position of cutting tool before circular interpolation block

Here is an example:

```

N66 G01 X5.75 Y7.5      (START POINT OF ARC)
N67 G03 X4.625 Y8.625 R1.125      (ARC)
N68 G01 X.. Y..

```

In the example, block N66 represents the *end* of a contour, such as a linear motion. It also represents the *start* of the arc that follows next. In the following block N67, the arc is machined, so the coordinates represent the end of arc and start point of the next toolpath element. The last example block N68 represents the end point of the element that started from the arc. The *end point* of the arc is the coordinate point of any two axes, where the circular motion *ends*. This point is sometimes called the *target* position.

### ◆ Arc Center and Radius

The *radius* of an arc can be designated with the address R or with arc center vectors I, J and K. The R-address allows programming the arc radius directly, IJK arc center vectors are used to actually define the physical (actual) arc center position. Most modern control systems support R-address input, older controls require the arc center vectors only. Basic programming format will vary only slightly between milling and turning systems, particularly when the R-address version is used:

```

G02 X.. Y.. R..      Milling program - CW
G02 X.. Z.. R..      Turning program - CW
G03 X.. Y.. R..      Milling program - CCW
G03 X.. Z.. R..      Turning program - CCW

```

Why is the arc center location or the arc radius needed at all? It would seem that the end point of an arc programmed in combination with a circular interpolation mode should be sufficient. This is never true. Always keep in mind that *numerical control* means control of the toolpath by *numbers*. In this case, there is an *infinite number* of mathematical possibilities where all may correspond to this incomplete definition. There is virtually an unlimited number of arc radiuses that will fit between the programmed start and end points and still maintain the cutting direction.

Another important concept to understand is that the cutting direction CW or CCW has nothing to do with the arc center or the radius. Control system needs more information than direction and target point in order to cut the desired arc. This additional information must contain a definition that defines a programmed arc with a *unique* radius.

This unique radius is achieved by using the R-address for direct radius input, or using the IJK arc center vectors. Address R is the actual radius of the toolpath, usually the radius taken from part drawing.

### ◆ Arc Center Vectors

Figure 29-4 shows the signs of arc vectors I and J in all possible orientations. In different planes, different pairs of vectors are used, but the logic of their usage remains exactly the same.

Arc vectors I, J and K are used according to the following definitions (only I and J are shown in the illustration):



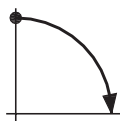
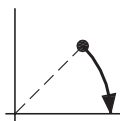

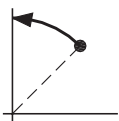
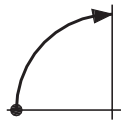
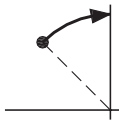


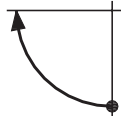
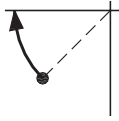

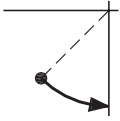
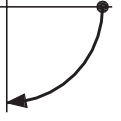
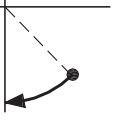
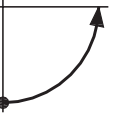
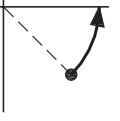
	G02		G03	
Quadrant I				
	I0 J-	I- J-	I- J0	I- J-
Quadrant II				
	I+ J0	I+ J-	I0 J-	I+ J-
Quadrant III				
	I0 J+	I+ J+	I+ J0	I+ J+
Quadrant IV				
	I- J0	I- J+	I0 J+	I- J+

Figure 29-4

Arc vectors *I* and *J* (also known as arc modifiers) and their sign designation in different quadrants (XY plane shown)

Arc center vector *I* is the distance, with specified direction, measured from the start point of the arc, to the center of the arc, parallel to the X-axis

Arc center vector *J* is the distance, with specified direction, measured from the start point of the arc, to the center of the arc, parallel to the Y-axis

Arc center vector *K* is the distance, with specified direction, measured from the start point of the arc, to the center of the arc, parallel to the Z-axis

The distance between the arc start point and the arc center point (as specified by IJK vectors) is almost always measured as an *incremental* distance between the two points. Some control systems, for example some older Cincinnati designs, use the absolute designation to define an arc center. In those cases, the arc center is programmed as an *absolute* value from *program zero*, not from the arc center. Always make sure how each of the control systems in the machine shop handles these situations. Generally, it can be said that:

Arc center is measured **from** arc start point **to** arc center

The lack of a suitable standard in this respect creates a major difference in programming format, so be careful to avoid a possible error. An error can be particularly likely in those cases where both types of controls are installed in the machine shop. There is *no compatibility* between programs using absolute and incremental method of the arc center.

The specified *direction* applies only to the incremental designation of arc center. It is the definition of relative position of the arc center from the start point, programmed with a directional sign - absence of the sign always assumes a positive direction, minus sign indicates a negative direction and must always be written. Arcs using absolute center definition follow standard rules of absolute dimensioning.

#### ◆ Arc in Planes

For machining centers, programming an arc in any one of the three geometrical planes is allowed - Figure 29-5. The correct arc vectors must be used for each selected plane:

G17 G02 (G03) X.. Y.. R.. (or I.. J..)  
 G18 G02 (G03) X.. Z.. R.. (or I.. K..)  
 G19 G02 (G03) Y.. Z.. R.. (or J.. K..)

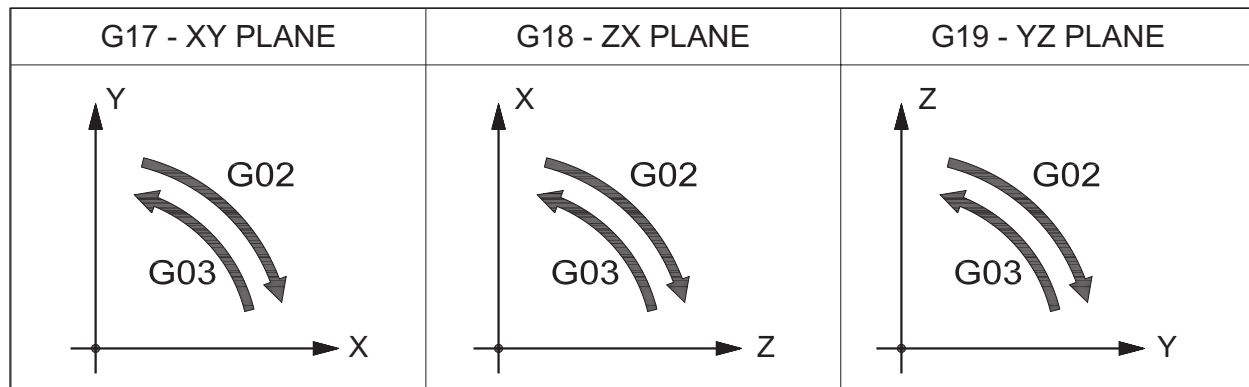


Figure 29-5

Arc cutting direction in three planes - the axis orientation is based on mathematical - not machine - planes

If the programmed plane is not aligned with the machine axes, or if the axes used in the program are selected without a plane designation, circular motion will take place according to the *axis selection in the program*. Always watch when the modal axis motion is omitted. The safest method of avoiding this potentially harmful problem is to follow a simple precaution - *never count on modal settings*.

In nonstandard planes, the circular program block should always contain specifications for *both axes*, as well as both arc vectors *or* the R radius. Such a block is *complete* and will always be executed on the basis of axes designation priority. This method is preferable to the selection of a previously defined plane. Even if the plane designation is incorrect, the resulting tool motion *will* always be correct.

## RADIUS PROGRAMMING

Programming arc is very common. By definition, an arc is only a *portion* of a circle and there are many ways to program an arc. If the arc is 360°, it must be programmed with the cutting start position being the same as its end position. In this case, a full circle is the result. If only a portion of the circle is programmed, only a radius is programmed. Two kinds of radii are used in CNC programming:

- Blend radius
- Partial radius

Each radius may be programmed in CW or CCW direction and each may be external or internal, as well as in any orientation that the cutting tool can handle.

### ◆ Blend Radius

A point of tangency between an arc and its adjacent element creates a *blend radius*. Blend radius is defined as a radius tangent between a *line and an arc*, an *arc and a line*, or between *two arcs*. A blend arc creates a smooth transition between one contour element and another. The point of tangency is the only contact point between the two elements.

The simplest form of a blend radius is between two perpendicular lines that are parallel to the machine axes. Calculation of the start and end points requires only a few additions or subtractions. A little more complex calculations are required when even one line is at an angle. In this case, trigonometric functions are used to calculate the start or the end point, or both. Similar calculations are required for blends between other entities as well. A blend arc is also known as a *fillet arc* or a *fillet radius*.

### ◆ Partial Radius

The opposite of a blend arc is a *partial arc* - there is no smooth blend between two contour elements, instead, there is an intersection. Mathematically, there are always two possible selections, however, the part drawing should be quite clear as to the shape of any partial radius. Partial radius can also exist between *two lines*, *one line and an arc*, or between *two arcs*. Partial radius can be defined as a radius where either the start point or the end point is *not* tangent to the adjacent element, but intersects it in two places. Actual calculation of point coordinates for the arc start or end point is about the same as that for a blend arc, depending which method of dimensioning had been used in the part drawing.

## FULL CIRCLE PROGRAMMING

All Fanuc systems and many other controls support a full circle programming. Full circle is an arc machined along 360°. Full circle cutting is possible on the lathes in theory only, since the type of work does not allow it. For milling work, full circle programming is fairly routine and is often required for many common operations, such as:

- Circular pocket milling
- Spotface and counterbore milling
- Helical milling (with linear axis)
- Milling a cylinder, sphere or cone

A *full circle cutting* is defined as a circular tool motion that completes 360° between the start and end points, resulting in identical coordinates for the start and end tool positions. This is a typical application of one block programming of a full circle - *Figure 29-6*.

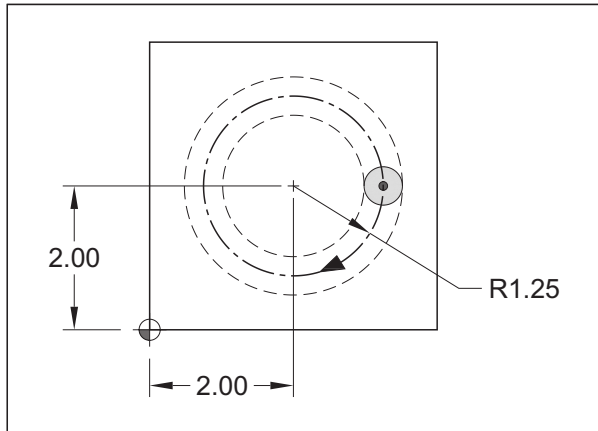


Figure 29-6

Full circle programming using one block of program entry

```
...
G90 G54 G00 X3.25 Y2.0 S800 M03
G01 Z-0.25 F10.0
G02 X3.25 Y2.0 I-1.25 J0 F12.0 (FULL CIRCLE)
G00 Z0.1
...
```

Older controls do not allow a circular interpolation motion in more than one quadrant per block. In this case, the circular motion has to be divided among four or even five blocks, depending on the starting tool position. Using the previous drawing example, resulting program will be a little longer, but with the same results - *Figure 29-7*:

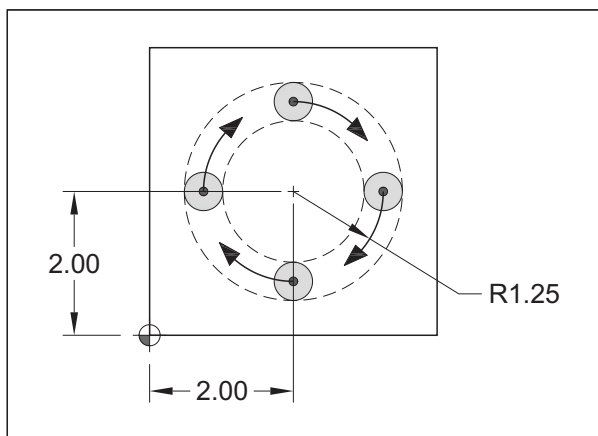


Figure 29-7

Full circle programming using four blocks of program entry

```
...
G90 G54 G00 X3.25 Y2.0 S800 M03
G01 Z-0.25 F10.0
```

```
G02 X2.0 Y0.75 I-1.25 J0 F12.0 (BLOCK 1 OF 4)
G02 X0.75 Y2.0 I0 J1.25 (BLOCK 2 OF 4)
G02 X2.0 Y3.25 I1.25 J0 (BLOCK 3 OF 4)
G02 X3.25 Y2.0 I0 J-1.25 (BLOCK 4 OF 4)
G00 Z0.1
...
```

This is an example of a four block programming that covers a full circle cutting. The arc start and end points are both located at a *quadrant* point of the axis line, which is an important programming consideration. The *quadrant* point in the example is equivalent to 3 o'clock position (0°). Note that G02 is repeated in each block only for the example emphasis and does *not* need to be repeated in a production program. The same applies to the occurrences of I0 and J0 - vectors do *not* have to be written if they are equal to zero.

Try not to make the job more difficult by establishing starting position of the cut away from any of the four *quadrant* points, which are at 0°, 90°, 180° and 270°. For example, if the cutting starts at 33°, there will be *five* circular blocks, not *four*; and the XY coordinates of the start point of the arc (shown as  $x_s$  and  $y_s$  distances), will have to be calculated using trigonometric functions - *Figure 29-8*:

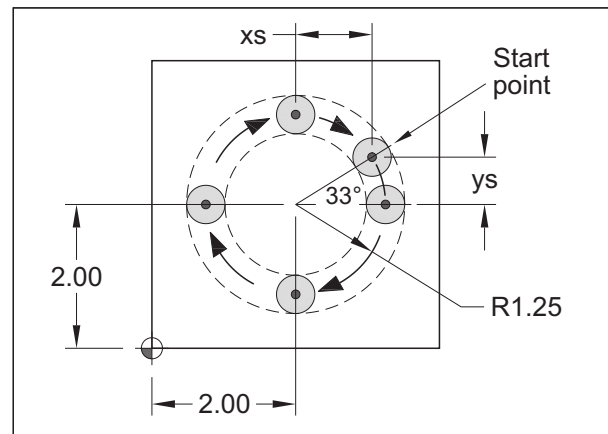


Figure 29-8

Full circle programming using five blocks of program code

```
...
G90 G54 G00 X3.0483 Y2.6808 S800 M03
G01 Z-0.25 F10.0
G02 X3.25 Y2.0 I-1.0483 J-0.6808 (BLOCK 1 OF 5)
G02 X2.0 Y0.75 I-1.25 J0 (BLOCK 2 OF 5)
G02 X0.75 Y2.0 I0 J1.25 (BLOCK 3 OF 5)
G02 X2.0 Y3.25 I1.25 J0 (BLOCK 4 OF 5)
G02 X3.0483 Y2.6808 I0 J-1.25 (BLOCK 5 OF 5)
G00 Z0.1
...
```

Values  $x_s$  and  $y_s$  were calculated by the following trigonometric functions:

$$x_s = 1.25 \times \cos 33 = 1.0483382$$

$$y_s = 1.25 \times \sin 33 = 0.6807988$$

From the results, the start point of the cut can be found:

$$X = 2 + x_s = 3.0483382 = X3.0483$$

$$Y = 2 + y_s = 2.6807988 = Y2.6808$$

If the control system supports a full circle program input in *one block*, the output program will be shorter, but *will require* the I and J arc center vectors only - direct R-radius *cannot* be used in this case. The reason is that both I and J vectors are *always* unique in their meaning, whereas radius R designation can be ambiguous. The following example is correct, using the I and J arc vectors:

```
...
G90 G54 G00 X3.0483 Y2.6808 S800 M03
G01 Z-0.25 F9.0
G02 X3.0483 Y2.6808 I-1.0483 J-0.6808
G00 Z0.1
...
```

The I and J modifiers cannot be arbitrarily replaced with the address R. The next example is *not* correct:

```
...
G90 G54 G00 X3.0483 Y2.6808 S800 M03
G01 Z-0.25 F9.0
G02 X3.0483 Y2.6808 R1.25 F12.0  (== WRONG ==)
G00 Z0.1
...
```

The reason? Mathematically, there are *many* options for a full circle programming. If an R-radius is programmed for a 360° arc, no circular motion will take place and such a block will be ignored by the control. This is a precaution built into the control software, to prevent from cutting an incorrect arc because of the infinite possibilities. In *Figure 29-9*, only a handful of the possible arcs is shown. The circles share the same cutting direction, start point, end point, and radius - *they do not share center points*.

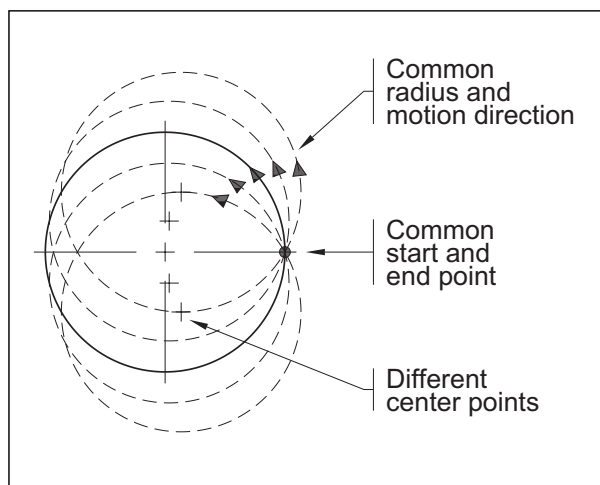


Figure 29-9

Many mathematical possibilities exist for a full circle cutting with R

### ◆ Boss Milling

As an example of a full circle cutting, a simple boss will be used, as illustrated in *Figure 29-10*.

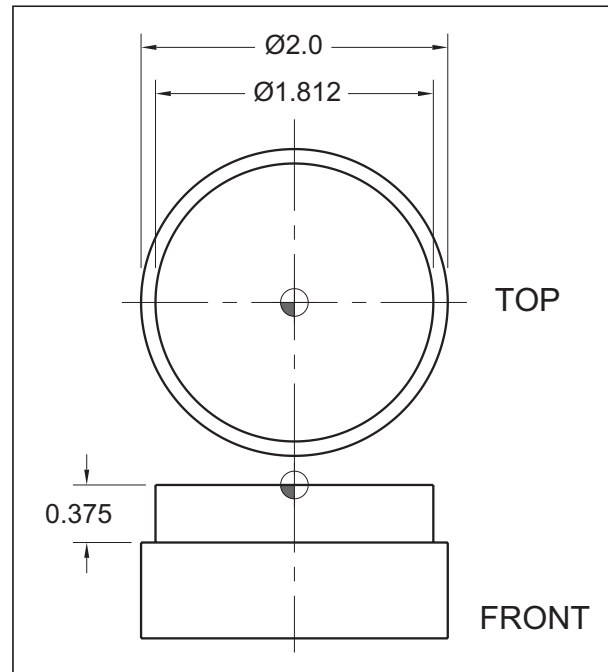


Figure 29-10

Boss milling example - drawing for program O2901

Boss or spigot milling are terms used for *external* milling of a full circle. The opposite is an *internal* milling of a full circle, where a *circular pocket* is most typical. The cutter used is Ø0.75 inch end mill programmed at Z-0.375 depth:

```
O2901
(0.75 DIA END MILL)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-1.0 Y1.5 S750 M03
N4 G43 Z0.1 H01
N5 G01 Z-0.375 F40.0 M08
N6 G41 Y0.906 D51 F20.0
N7 X0 F14.0
N8 G02 J-0.906
N9 G01 X1.0 F20.0 M09
N10 G40 Y1.5 F40.0 M05
N11 G91 G28 X0 Y0 Z2.0
N12 M30
%
```

In program O2901, tool moves first to the XY position and depth, then the cutter radius offset is started. When reaching the cutting depth, the tool made a straight climb milling motion to the top of boss. Then it swept around the circle to the same point, moved away straight, and by reversing the initial motions, it returned to its Y-axis start point - *Figure 29-11* shows the block numbers.

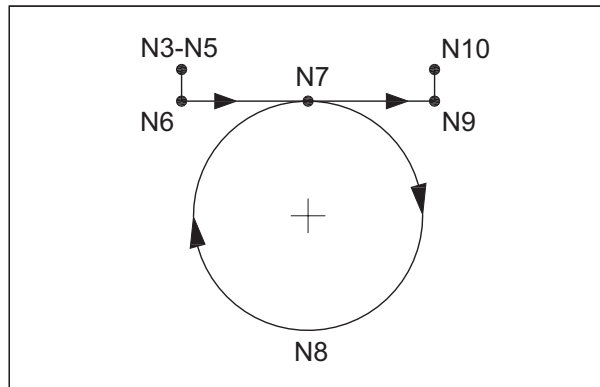


Figure 29-11

Boss milling example - tool motions for program O2901

Alternate applications may include operations such as multiple roughing passes, a semifinishing pass, two cutting tools and other selections related to machining.

#### ◆ Internal Circle Cutting - Linear Start

Internal full circle cutting is common and has many applications, such as circular pockets or counterbores. In an example, a  $\varnothing 1.25$  circular step is to be machined to the depth of 0.250 inch, in program O2902. A simple linear motion will be used for the startup, where the entry point blend is not too important. Cutting tool is a center cutting end mill (also known as a slot drill) - Figure 29-12:

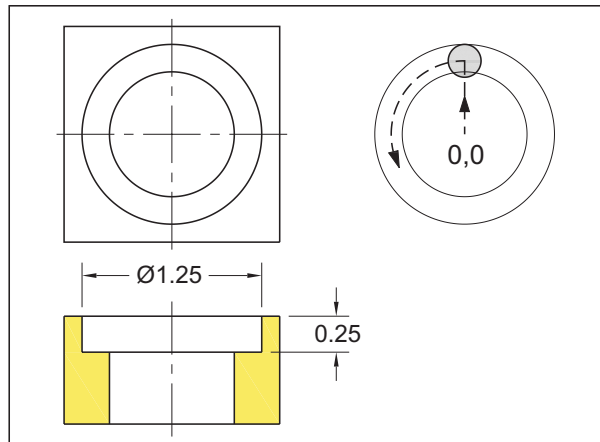


Figure 29-12

Internal circle cutting - linear approach only

O2902

(0.5 DIA CENTER END MILL)

N1 G20

N2 G17 G40 G80

N3 G90 G54 G00 X0 Y0 S900 M03

N4 G43 Z0.1 H01

N5 G01 Z-0.25 F10.0 M08

N6 G41 Y0.625 D51 F12.0

N7 G03 J-0.625

(LEAD-IN LINE)

(FULL CIRCLE)

N8 G01 G40 X0 F20.0 M09  
N9 G91 G28 X0 Y0 Z2.0 M05  
N10 M30  
%

(LEAD-OUT LINE)

Program O2902 shows both the arc start point and end point at  $90^\circ$ , programmed at 12 o'clock position. Cutter radius offset started during the motion from the arc center.

Cutter radius offset cannot start or end in a circular mode

This is true for almost any circular application, except the very few that use a special cycle.

#### ◆ Internal Circle Cutting - Circular Start

The simple linear approach programming method in the last example will not be practical when smooth blend between the approach and the circular cut is required. To improve surface finish, the start position of circular motion can be reached on an arc. The usual startup is from the center, first at a  $45^\circ$  linear motion, to apply the cutter radius offset, then on an arc that blends with the full circle. The Figure 29-13 illustrates the principle and program O2303 shows the complete program using lead-in/lead-out arc.

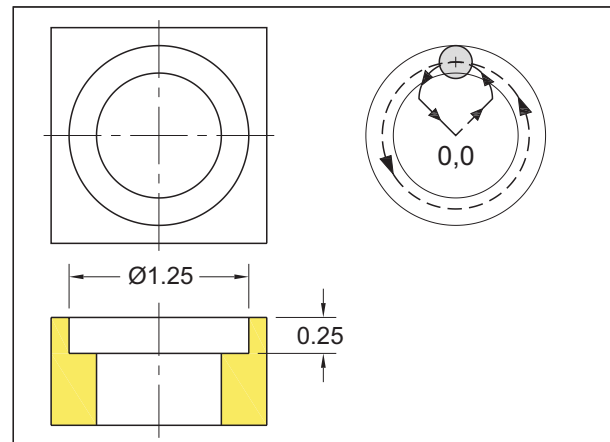


Figure 29-13

Internal circle cutting - linear and circular lead-in / lead-out

O2903

(0.5 DIA CENTER END MILL)

N1 G20

N2 G17 G40 G80

N3 G90 G54 G00 X0 Y0 S900 M03

N4 G43 Z0.1 H01

N5 G01 Z-0.25 F10.0 M08

N6 G41 X0.3125 Y0.3125 D51 F12.0 (LEAD-IN LINE)

N7 G03 X0 Y0.625 R0.3125 (LEAD-IN ARC)

N8 J-0.625 (FULL CIRCLE)

N9 X-0.3125 Y0.3125 R0.3125 (LEAD-OUT ARC)

N10 G01 G40 X0 Y0 F20.0 M09 (LEAD-OUT LINE)

N11 G91 G28 X0 Y0 Z2.0 M05

N12 M30

%



This programming method is slightly longer, but the surface finish quality with a circular approach is much better than with the linear approach.

If a control systems has the *User Macro* option and many circular pockets are required, the O2903 example could also be adapted to a macro - see page 302 for more details. Some controls have a circular pocket milling cycle built-in.

### ◆ Circle Cutting Cycle

Several controls, for example some Yasnac or Mitsubishi, but not Fanuc, have a built-in routine (cycle) to cut a full internal circle using special preparatory commands, typically G12 and G13. These cycles are very convenient programming aid and to the surprise of many programmers, Fanuc dropped this feature many years ago.

There is a logical relationship between G02 and G12, as well as between G03 and G13:

G12	Full circle cutting cycle	CW
G13	Full circle cutting cycle	CCW

A typical programming format for using these two special commands is quite simple:

G12 I.. D.. F..      Full circle CW  
G13 I.. D.. F..      Full circle CCW

In this format description, the I address is the *radius* of finished circle and is programmed as an incremental value with a sign. If the sign is positive (plus sign is assumed), the start point of the cut will be at 0°, which is equivalent to the 3 o'clock position or East direction. If the sign is negative, the start point of the cut will be at 180° position, which is equivalent to the 9 o'clock position or West direction. The command cannot be forced to a start in the Y-axis direction.

Programmed D address is the control register number for the cutter radius offset and F is the feedrate address. There are alternate versions of this cycle on some controls, but all very similar in nature.

Other conditions must also be accepted for successful usage of this programming shortcut. The cutting tool must *always* start at the center of a circular pocket, cutting plane must be set to the XY plane and the arc starting position is usually built-in, generally at 0° or 180° (Y-axis start is not possible). There is also a *built-in* cutter radius offset (G12 to the right, G13 to the left). Never program the commands G41 and G42 when using G12 or G13 command. If the cutter radius is in effect, it will be overridden by the selection of G12 or G13. The safest approach is to program these two cycles in G40 mode (cutter radius offset canceled) at all times.

What is not true in any other circular application, is true in this situation. In normal programming of arcs and circles, a cutter radius offset cannot start in an arc tool motion. In G12/G13 programming mode, the start motion from the uncompensated circle center position is *circular* to the compensated start point on the arc circumference. This is all built into the control and there no choice is offered. Consider this situation as a special case, definitely not as a rule.

On some Yasnac CNC models, there is an additional parameter in the G12/G13 format - the radius parameter, or the R parameter. This indicates special rapid motion portion, designed to reduce air cutting time.

As an example of G12/G13 programming, the earlier circular pocket, illustrated in *Figure 29-14*, will be used.

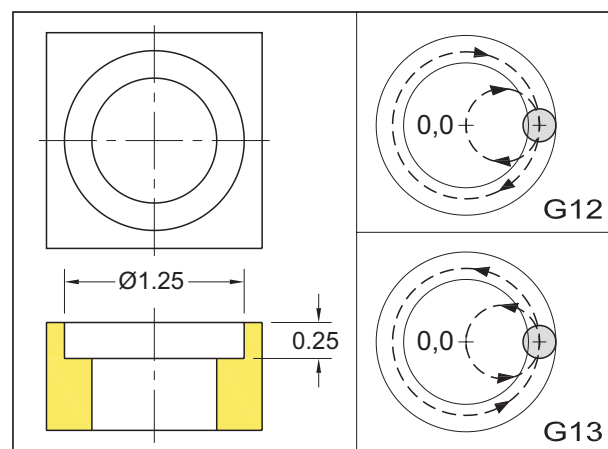


Figure 29-14

Full circle cutting using G12/G13 cycles - program O2904

```
O2904
(0.5 DIA CENTER CUTTING END MILL)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S900 M03
N4 G43 Z0.1 H01
N5 G01 Z-0.25 F10.0 M08
N6 G13 I0.625 D51 F12.0 M09      (IF AVAILABLE)
N7 G91 G28 X0 Y0 Z2.0 M05
N8 M30
%
```

The program is only two blocks shorter, but it is much simpler to develop. The cutter radius offset is automatic (built-in) and editing at the machine is much easier. There is also an additional bonus - since the start point on the circle is not a result of a straight line, but a lead-in arc, the surface finish quality will be better than using other types of tool approach. This is a preferable method when the machined surface quality is important. There is also a built-in lead-out arc in the cycle, similar to the lead-in arc, that is effective when the circle cutting is completed.

## ARC PROGRAMMING

With a full arc cutting, which means the *complete* 360° motion, the R-address cannot be used at all. Arc center vectors I and J have to be applied, even on latest controls.

What if the circle is 359.999°? Well, at first, circle *must* have 360°, therefore the word 'circle' is incorrect. Even a small difference of 0.001° does make a difference between a circle and an arc. Although this difference is much more important mathematically than for practical programming, the distinction is very important. In circular interpolation terms, an *incomplete circle* is nothing more than an *arc*. Look at this arc a little differently. If a 90° arc is made, the R-address (direct radius) can be programmed, for example:

```
G01 X2.0 Y5.25 F12.0
G02 X3.75 Y7.0 R1.75
```

If an arc that covers *exactly* 180° is programmed, the program will not be much different:

```
G01 X2.0 Y5.25 F12.0
G02 X5.5 Y5.25 R1.75
```

Note that the Y-coordinate is the same for the arc start and end position. Y amount in the circular motion block does not have to be repeated, it is used here only for illustration.

Another example shows programming an arc of 270°, still using the R-address. Are the following blocks correct?

```
G01 X10.5 Y8.625 F17.0
G02 X13.125 Y6.0 R2.625
```

The blocks appear to be correct. All calculations, format, individual words, they all appear to be right. Yet, the program is *wrong!* Its result will be a 90° arc, not 270°.

Study the illustration in *Figure 29-15*. It shows that there is not just one, but *two* mathematical possibilities when the R-address is used for arcs. The solid contour shown is the toolpath, dashes identify the two possible radiuses.

Programmers do not normally think of these mathematical alternatives, until they program arcs *larger* than 180° (or scrap a part). This is a similar situation to that of a full circle, described earlier. Although the I and J vectors can be used to remedy this problem, a different remedy may be a preferred choice. R-address can still be used in the program, but with a *negative* sign for any arc that is greater than 180°. For arcs smaller than 180°, the usual positive R radius remains in effect. Recall from some earlier explanations that if there is no sign with the R word (or any other word), the word assumes a *positive* value. Compare the two programming examples:

```
G01 X10.5 Y8.625 F17.0
G02 X13.125 Y6.0 R2.625      (90 DEGREES)
```

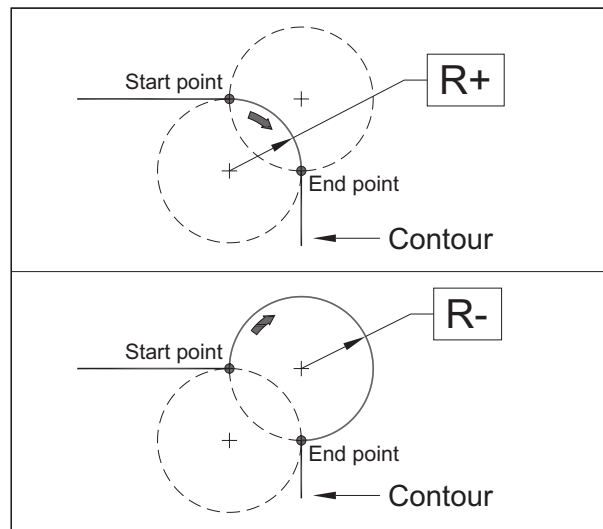


Figure 29-15

Sign of R address for circular cutting - only the center is different

The following example is identical to the previous one, except for the R-address sign.

```
G01 X10.5 Y8.625 F17.0
G02 X13.125 Y6.0 R-2.625      (270 DEGREES)
```

If frequently programming arcs that cover more than 180°, establish a particular programming style. If the style is well thought out, it will avoid the costly mistakes associated with the R-address sign error.

## FEEDRATE FOR CIRCULAR MOTION

In most programs, the feedrate for circular interpolation is determined the same way as feedrate for linear interpolation. Cutting feedrate for arcs is based on established machining conventions. They include the work setup, material machinability, tool diameter and its rigidity, programmer's experience and other factors.

Many programmers do not consider the machined radius when selecting the cutting feedrate for the tool. Yet, if the machined surface finish quality is really important, always consider the size of every radius specified in the part drawing. Perhaps the same feedrate for linear and circular motions programmed so far may have to be adjusted - either upward or downward.

In lathe programming, there is no reason to distinguish between linear and circular tool motions, regardless of the radius size. Tool nose radius is usually small, only averaging 0.0313 inches (or 0.8 mm) and the equidistant toolpath is close to the programmed toolpath, taken from a drawing. This is not the case for contour programming, where large tool radiuses are normal and common.

An adjusted arc feedrate is not required in every program. If the cutter center toolpath is close to the part drawing contour, no adjustment is needed. On the other hand, when a large diameter cutter is used to contour a small outside radius, a problem that affects the surface finish may occur. In this case, the tool center path generates a much longer arc than the one in the drawing. In a similar situation, if a large cutter diameter is used for an inside arc, the equidistant path will be much shorter than the original arc.

In typical programming, the linear feedrate is used for arcs as well, as determined by machinability rating for the given material. Formula for linear feedrate is:

$$F_L = r / \text{min} \times F_T \times n$$

where ...

$F_L$  = Linear feedrate (in/min or mm/min)  
 $r/\text{min}$  = Spindle speed  
 $F_T$  = Feedrate per tooth  
 $n$  = Number of cutting edges

A linear feedrate for 1000 r/min, 0.0045 in/tooth load and two cutting edges, the feedrate is 9.0 in/min. Using a relatively large cutter diameter, such as Ø0.625 (15.875 mm) or even larger, linear feedrate adjustment up or down for circular motion may be necessary to maintain good finish.

The elementary rule of feedrate adjustment for arcs is that the normally programmed linear feedrate is *increased for outside arcs* and *decreased for inside arcs* - Figure 29-16.

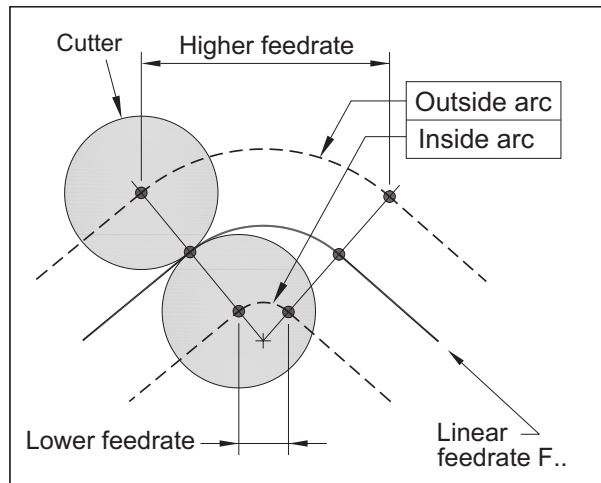


Figure 29-16

Feedrate adjustments for circular tool motion

An outside arc of a toolpath is **longer** than the drawing arc

An inside arc of a toolpath is **shorter** than the drawing arc

Two formulas provide the tools to find the adjusted arc feedrate, mathematically equivalent to the linear feedrate. Both formulas are recommended for external or internal contouring only, *not* for rough machining of solid material.

### ◆ Feedrate for Outside Arcs

For outside arcs, the adjusted feedrate will be *higher* than the linear feedrate, calculated from this formula:

$$F_o = \frac{F_L \times (R + r)}{R}$$

where ...

$F_o$  = Feedrate for OUTSIDE arc  
 $F_L$  = Linear feedrate  
 $R$  = Outside radius on the part  
 $r$  = Cutter radius

Based on a 14 in/min linear feedrate, an outside 0.375 arc radius requires an *upward* adjustment for a Ø0.50 cutter:

$$F_o = 14 \times (0.375 + 0.25) / 0.375 = 23.333333$$

The result is a major increase, to F23.3 in the program. Consider the same example with 0.75 cutter radius (Ø1.5):

$$F_o = 14 \times (0.375 + 0.75) / 0.375 = 42.0$$

Now, the feedrate changed from 14 in/min to 42 in/min - a three times increase. Use previous experience to determine whether the adjustment is justified or not.

### ◆ Feedrate for Inside Arcs

For inside arcs, the adjusted feedrate will be *lower* than the linear feedrate, calculated from this formula:

$$F_i = \frac{F_L \times (R - r)}{R}$$

where ...

$F_i$  = Feedrate for INSIDE arc  
 $F_L$  = Linear feedrate  
 $R$  = Inside radius on the part  
 $r$  = Cutter radius

Based on the linear feedrate of 14 in/min, the feedrate for 0.8243 inch inside radius with Ø1.25 cutter, must be adjusted *downward*:

$$F_i = 14 \times (0.8243 - 0.625) / 0.8243 = 3.384932$$

The result is a feedrate of 3.38 in/min. In the program, this will be the applied feedrate for the F-address.

## 30

## CUTTER RADIUS OFFSET

Contour of a part - also known as a profile - is normally programmed for milling applications by establishing the depth in Z-axis first, then moving the cutting tool individually along the X-axis, Y-axis, or both axes simultaneously. For turning applications, either the X-axis or the Z-axis, or both axes can be used to face, turn or bore a contour. For both types of machining, each contour element requires a single block of cutting motion in the program. These motions between contour change points can be programmed in millimeters or inches and they can use absolute value positions or incremental distances. In either case, keep in mind that this type of programming always uses the spindle centerline as the X-Y or X-Z tool movements. Although centerline programming is a very convenient method for program development, it is also a method unacceptable for machining. During contact with material, the cutting tool edge must be in contact with the programmed part contour - not its centerline.

Actual toolpath for all contouring operations is always equivalent to the cutting tool motion. Whether used on a CNC machining center or on a CNC lathe, the *cutting tool edge must always be tangent to the contour*, which means the tool motion has to create a path where the center point of the cutter is *always at the same distance* from the contour of the part. This is called the *equidistant* tool path.

The illustration in Figure 30-1 shows two types of a tool path. One is *not compensated*, the other is *compensated*. Both are applied to a particular contour, with the cutter diameter shown as well, including its positions.

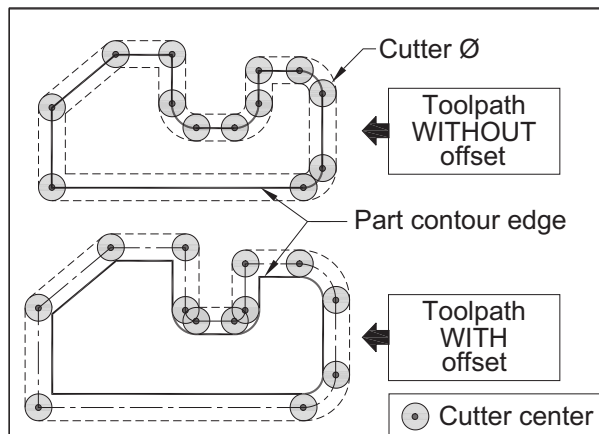


Figure 30-1

Tool path not compensated (above) and compensated (below), by the specified cutter radius

## MANUAL CALCULATIONS

Some basic realities should become evident by evaluating Figure 30-1. The most distinct observation is that the machined contour must always take place with toolpath *compensated by its radius*, which means its center point must be located in positions shown in the lower example. This machining necessity is not matched by the reality of engineering drawings. In a drawing, all dimensions refer to the actual *part* contour, *not* the contour of tool center. In fact, engineering drawings have nothing to do with tool locations or machining - it is not their purpose. If a cutter is used based on drawing dimensions alone, it will follow the toolpath illustrated in the upper example. Since the lower example represents the desired toolpath, the obvious question is how to change tool center positions from the edge of the part to the edge of the cutter?

The answer is - they have to be *calculated*. Actually, they do not have to be, if the CNC system is equipped with an advanced built-in feature called the *cutter radius compensation* or *cutter radius offset*. On CNC turning systems, this feature is called the *tool nose radius compensation* or *tool nose radius offset*. This mostly standard control feature enables CNC programmer to apply an offset command, program the part contour *as per drawing* dimensions and let the control system do all necessary calculations and adjustments automatically.

From now on, this chapter could continue and concentrate only on the automatic method of programming, using this powerful feature. After all, modern CNC machines do have cutter radius offset built-in. Once several basic rules are followed, cutter radius offset is easy to use.

Some background is necessary to fully understand the radius offset concept. If something is automated already, the knowledge of *how* it works makes the job so much easier, particularly when encountering a difficulty that has to be resolved very quickly. To really understand cutter radius offset - many programmers and machine operators do not - it is important to understand the basic principles - principles that are very much based on common mathematical calculations, including sometimes unpopular trigonometry. A simple drawing is shown in Figure 30-2 for that purpose.

Part program zero is at the lower left corner of part. Cutting will be external, in climb milling mode, and tool starts cutting in Y-direction first. The start and end tool positions are not important now, only calculations of individual contour points at intersections and tangency points.

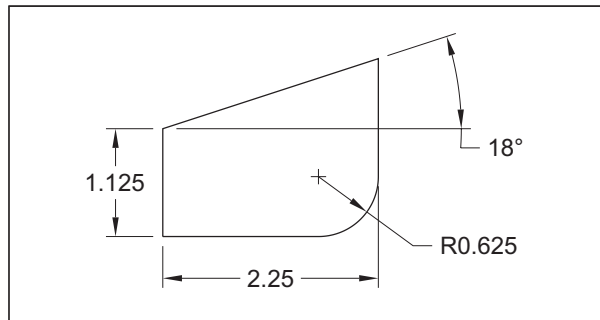


Figure 30-2

Sample drawing for manual calculations (examples)

Note that there are five points on the drawing, one at each contour change. In closed contours, these points are either *intersections* or points of *tangency* - they all are **endpoints**. As each point has two coordinates, total of *ten* coordinate values will be required. In addition, in open contours, each contour end with an *endpoint*.

Endpoints in CNC toolpath are called **Contour Points**

Most drawings always offer at least some contour points that need no calculations. It is a good idea to get well organized and mark these points from the drawing first. Then, make a chart in the order of actual cutter path. Study *Figure 30-3* carefully - it shows all five points and all coordinate values that need no calculation, with the exception of some additions and subtractions. Cutting toolpath is in the order of points P1-P2-P3-P4-P5-P1. That is also the order these coordinates will be entered in CNC program.

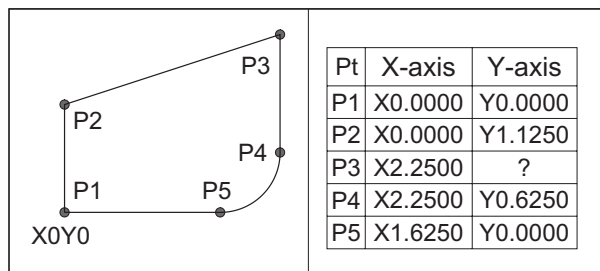


Figure 30-3

Contour change points required by the cutter path

In this simple contour, out of the ten coordinate values required, nine of them are provided. The missing Y-coordinate for P3 is the only exception - it is not expected on the drawing and must be calculated. Regardless of whether cutter radius offset is used or not, some calculations will always be necessary and this is one of them. After all, *manual* programming is done *by hand*.

*Figure 30-4* shows the trigonometry method used and all five points can be summed up in a small table or chart.

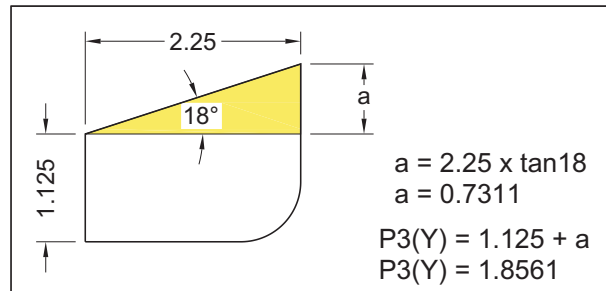


Figure 30-4

Trigonometric calculations to find unknown Y coordinate

Point No.	X coordinate	Y coordinate
P1	X0	Y0
P2	X0	Y1.125
P3	X2.25	<b>Y1.8561</b>
P4	X2.25	Y0.625
P5	X1.625	Y0

Once all coordinates are known, there is enough data to start the cutter path, but only *if* the cutter radius offset feature is used. However, that is not the intention at this moment. *To illustrate why, a whole new set of points has to be found - coordinates for the cutter center!*

### ◆ Cutter Path Center Points

A cutting tool for milling applications is always round. An end mill, for example, has a diameter of a certain size. Even tools used for turning and boring have a round end (called the *tool nose radius*), even if it is relatively small. Of course, we all know that any round object has a center. Milling cutter or a lathe tool tip are round objects, so they have a center. This evaluation may sound a bit too elementary and it is, but it is also the basis - the key element, the whole concept - of *cutter radius offset*. Every control system offers features related to cutter radius offset.

Take, for example, an electric router tool to cut a certain shape out of wood - how is it used? Using a pencil outline of the desired shape, the router bit is placed into the tool and starts cutting. Where? *It starts cutting outside of the outlined shape, otherwise the piece cut will be either too large or too small!* The same procedure is used when cutting a board with a hand or electric saw - the amount of saw width has to be compensated.

This activity is so simple, it might have been even done automatically, without serious thinking. The router bit *radius* or the saw width was *compensated* for before and during the cut. Just like the wood outline is followed, the outline of the machined part, *outline that is offset by the cutter radius is followed as well*.



The tool path (cutter path) generated by the cutting tool center *always* keeps the same distance from the part contour (part outline). There is even a special name for this type of tool path - it is called the *equidistant* tool path, which means 'distant by the same amount'. Figure 30-5 shows the same sample drawing with applied equidistant tool path.

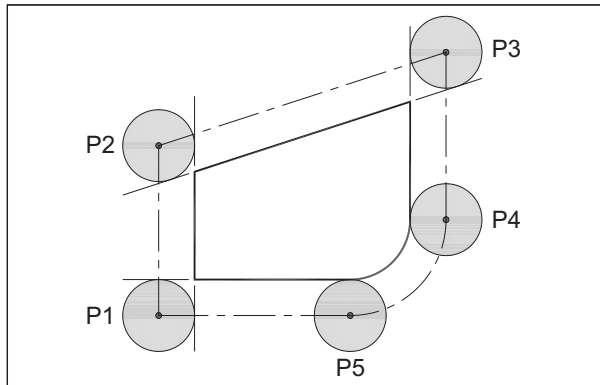


Figure 30-5

Equidistant tool path - cutter center coordinates required

The question now is - what to do about the point coordinates that have just been calculated and stored in the above table. Are they useful? Can they be used in a program? *Yes* to the first question, but *not yet* to the second. A few additional conditions have to be taken into consideration.

In the program, the original - *old* - set of points will be used to calculate a *new* set of points. Again, try to see which points are easy to calculate and establish them first.

For example, what are the XY coordinates of point P1? They are X0Y0. It is easy to see that the new and compensated P1 will have the value of cutter radius in the X minus and also the value of cutter radius in the Y minus direction, from the old P1. The actual coordinate for *any* point cannot be calculated at all, *without knowing the cutter radius first*.

### ◆ Cutter Radius

The nominal radius of a cutter is always known on new tools, or tools that have been physically measured. For high precision work, the cutter radius must be known almost 100%, say within 2.5 microns (0.025 mm = 0.0001"). That is not always possible for reground tools, tools previously used, slightly worn out tool, or tools that are either under-size or over-size for some other reason. All this means that programming the cutter centerline requires the exact tool radius to be known at the time of programming - in one and every case.

➡ Incidentally, at one time, this was the only method of programming a contour - no computer assistance, no CAD/CAM, just manual calculator, a pencil and a paper, and a 'five pound' eraser, as the old joke goes

### ◆ Center Points Calculation

Coordinate points illustrated earlier in Figure 30-5, represent the center of a cutter radius at each contour change point. Now, another requirement can be brought into the picture - the cutter radius *size*. Then, a new coordinate set of five points can be developed. For the example, a new end mill cutter of Ø0.750 will have 0.375 radius.

Which points can be 'read' from the illustration directly, without any trigonometric calculations? Look at and evaluate Figure 30-6. Out of the ten coordinate values required, only eight have been identified, using addition and subtraction only - but also realize that the previous ten calculations had to be done earlier as well, adding to the overall programming effort.

In order to finish the discussion on cutter center programming, the two Y values for P2 and P3 have to be calculated. Let's start with point P2.

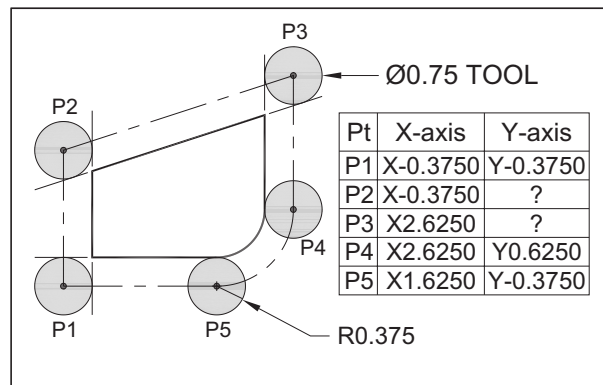


Figure 30-6

Contour change points for the cutter center path

Figure 30-7 shows all details of point P2 calculation. The trigonometry method itself is a subject programmers have to know how to work with - it is part of mathematics, extended to CNC programming.

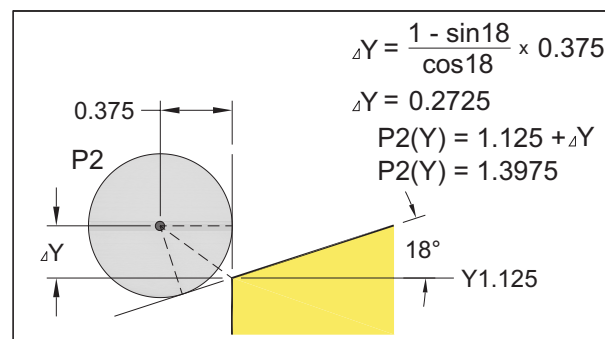


Figure 30-7

Calculation of P2 for the cutter center point

Point P3 calculation is similar, and shown in Figure 30-8.



Some applications may not be too clear at the moment, but with increased knowledge of this topic, it will be easier to understand the subject. Provided suggestions are only some of the possibilities automatic cutter radius offset offers. Now let's look at its actual use in CNC programming.

## PROGRAMMING TECHNIQUES

In order to program cutter radius in a compensated mode, the three items mentioned earlier must be known:

- Contour points based on the drawing
- Specified direction of the cutter motion
- Cutter radius amount stored in the control system

These items supply actual *data sources* for calculations. Computers only work with data and the data has to be supplied by the user. For the purposes of this chapter, assume that all data for the contour change points are based on the drawing as *XY drawing coordinates*.

### ◆ Direction of Cutting Motion

Regardless of whether an external or an internal tool path is programmed for a contour, there will always be a choice of *two* directions in any plane. For now only, the directions can be called *clockwise* and *counterclockwise* direction around the part contour. This incorrect, yet common terminology is often used when describing tool path around a contour. This general motion direction is compounded by the reality that there is a specific motion of the machine table (in milling), or motion of the tool (in turning). These are two very separate groups that need to be clarified - which one to consider - motion of the machine table or motion of the tool? Keep in mind, that *regardless of the CNC machine type used*, it is imperative to follow one fundamental rule of CNC programming:

In CNC programming, always consider the cutting tool moving around the part, never the other way around

This statement is true for CNC lathes, where it is obvious, but *it is also true for CNC machining centers*, where it is not so obvious. It is also true for other types of CNC machines, such as wire EDM, laser cutting machines, waterjet cutters, flame cutters, etc. When it comes to the so called direction *clockwise* versus *counterclockwise* describing a cutting direction around the part, a closer look is necessary.

### ◆ Left or Right - not CW or CCW

The first thing to take care of is to eliminate the misleading terms *clockwise* and *counterclockwise*. These terms are reserved exclusively for circular interpolation and have no place in discussion of the cutter radius offset. Instead, the more accurate terms *Left* and *Right* are used for clarity.

Even these two terms may be confusing sometimes - try to face somebody and talk about a position of an object. What's to the left for you is to the right for your opponent.

Just like in everyday situations, when faced with the *directional* terms *left* and *right*, we determine the correct position of an object with respect to a certain previously established *viewing direction*. A moving object is said to be to the left or to the right of a stationary object, depending on the *direction of its movement*.

In CNC programming, there is no difference. The compensated cutter tool path is positioned to the *left* or to the *right* of a stationary contour, when looking into the cutter path direction, as illustrated in Figure 30-9.

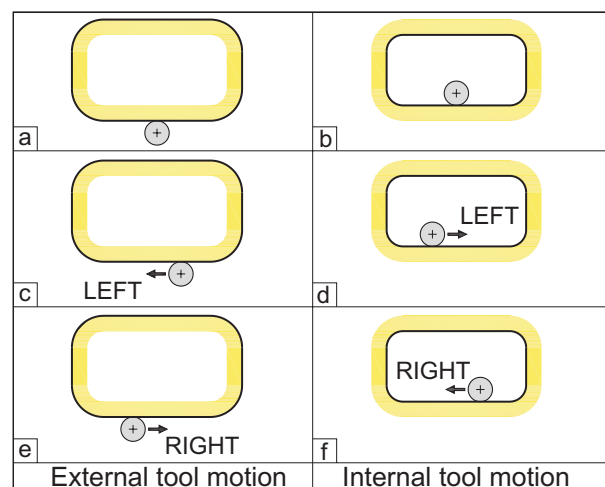


Figure 30-9

Cutter path direction as it relates to a stationary part contour:  
 (a - b) No motion direction shown - left and right is unknown  
 (c - d) Cutter positioned to the LEFT of the contour  
 (e - f) Cutter positioned to the RIGHT of the contour

The illustration shows all three options - a cutter without a direction, a cutter with direction specified and positioned to the *left of the contour*, and a cutter with direction specified and positioned to the *right of the contour*.

Out of the two compensation options, which one is better? Compensation *to the left* is preferred on CNC machining centers, because it produces a *climb milling* mode of cutting, assuming that a standard right-hand tool is used with M03 rotation. There might be a case for the compensation to the right, causing so called *conventional milling* mode of cutting. This mode should be used only in special cases, after consultation with a tooling specialist. This only applies to milling systems, not to turning.

Selection of *left* or *right* direction for milling a contour is not a matter of programming importance as such - it is selection based on *established machining principles*.

### ◆ Offset Commands

In order to program one or the other mode of cutting (based on cutting direction), control systems offer two preparatory commands to select the desired cutter radius offset direction mode:

G41	Offset (compensation) of the cutter radius is to the LEFT of contouring direction
G42	Offset (compensation) of the cutter radius is to the RIGHT of contouring direction

When either G41 or G42 mode is no longer required, it must be canceled by the G40 command:

G40	Cutter radius offset mode CANCEL
-----	----------------------------------

Figure 30-10 shows all three radius offset commands:

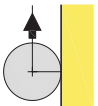
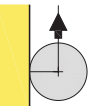
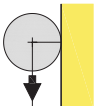
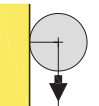
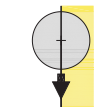
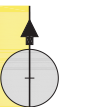
	G41 LEFT		G42 RIGHT
	G42 RIGHT		G41 LEFT
	G40 - NONE -		G40 - NONE -

Figure 30-10

Application of G41, G42, and G40 to a cutter path

Keeping in mind that the terms *climb* and *conventional* refer to milling only and are relative to spindle rotation and the hand of milling cutter. By this definition, the G41 command is applied for *climb* milling mode, G42 command is applied for *conventional* milling mode:

Tool Type	Tool Rotation	Cutting Direction	G-code G41 or G42	Type of Milling
Right Hand	M03	Left	G41	Climb
		Right	G42	Conventional
Left Hand	M04	Left	G41	Conventional
		Right	G42	Climb

The table shows all relationships between tool type, tool rotation, and cutting direction.

There is no cutter radius offset applied when G40 command is in effect.

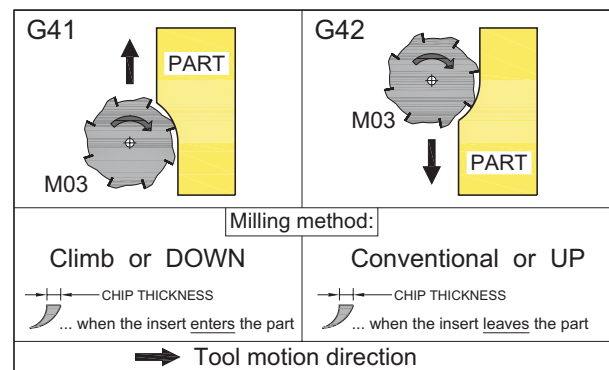


Figure 30-11

Climb milling and Conventional milling mode for a right hand cutter and spindle rotation mode M03

Figure 30-11 shows the G41 command as a climb milling mode and the G42 command as a conventional milling mode. Climb milling mode is the most common in CNC milling, particularly in contour milling.

### ◆ Radius of the Cutter

A common question relates to the cutter radius - *if it is ignored in the program, how does the control 'know' its size?* First, the radius is *not* ignored in the program - programmer still selects a tool by its diameter and provides sufficient clearances, based on that diameter. The only thing the program does *not* contain is the actual cutter radius size. As this is the case, *where is the radius size specified?* First, look at Figure 30-12 - it illustrates the effect of a different cutter radius applied to the same part contour.

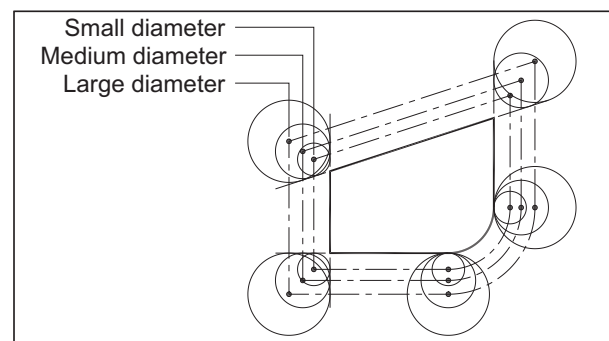


Figure 30-12

Cutter radius size and its effect on the actual tool path

Actual radius size is stored at the machine - *in the control system area called offset settings*. These offset areas (*offset screens on control unit*) have been used for *Position Compensation* (see page 123), *Work Offsets* (see page 127) and *Tool Length Offset* (see page 135).

Cutter radius offset is another offset that both the operator and the programmer must thoroughly understand.



### ◆ History of Offset Types

Fanuc controls have developed over the years, and because of their popularity and high reliability, many of the older models are still in use by machine shops. To understand various offsets and their application, it is important to know what *type of offset* a particular Fanuc control has. The rule of thumb is - as expected - the lower level or the older the control is, the lower the flexibility, and vice versa. Notice the word *flexibility* - it is not the *quality* that is lower or higher - just the flexibility. All differences are categorized as *Offset Memory Types*. There are *three* memory types available on Fanuc systems:

- Type A - lowest level of flexibility
- Type B - medium level of flexibility
- Type C - highest level of flexibility

Do not confuse these tool offset memory types with the cutter radius offset types! These offset types determine how the *tool length offset* and the *cutter radius offset* will be entered into the control system - and nothing else. Work offsets G54 to G59 are not affected by *memory types*.

#### Tool Offset Memory Type A

*Type A* tool offset is the lowest level available. Its flexibility is very limited, because this offset type shares the tool length settings with the cutter radius settings in a *single column* of data entry. As two groups of offsets share the same offset registry, *Type A* called a *shared* offset. Addresses H and/or D are used in the program, with further details covered later. Controls equipped with this type of tool offset memory are the most economical type in their class.

#### Tool Offset Memory Type B

While the *Type A* offset has only a single screen column, *Type B* has *two columns*. Now - do *not* assume! These two columns are *not* separate columns for tool length and tool radius settings at all. They are separated for the *Geometry Offset* in one column and the *Wear Offset* in the second column. Apart of this distinction, *Type B* is still a shared type of offset for both, tool length and tool radius settings. Again, the program uses addresses H and/or D.

#### Tool Offset Memory Type C

*Type C* offset offers the most flexibility. It is the only offset that separates tool length settings from those of tool radius. It still keeps the distinction of *Geometry Offset* and *Wear Offset*, as the *Type B* does. It means the control display will now be 2+2 columns (*four columns*). Normally, addresses H and D are used in the program.

It is relatively easy to tell which offset type is available - just look at the control display. *Figure 30-13* shows the typical appearance of each *Offset Memory Type* (all shown with zero values). The actual appearance may be slightly different, depending on the control model.

		Offset No.	Offset
		01	0.0000
		02	0.0000
		03	0.0000
		...	...

		Offset No.	Geometry	Wear
		01	0.0000	0.0000
		02	0.0000	0.0000
		03	0.0000	0.0000
		...	...	...

Offset No.	H-offset		D-offset	
	Geometry	Wear	Geometry	Wear
01	0.0000	0.0000	0.0000	0.0000
02	0.0000	0.0000	0.0000	0.0000
03	0.0000	0.0000	0.0000	0.0000
...	...	...	...	...

Figure 30-13

Fanuc tool offset memory types A, B, C from the top down

### ◆ Programming Format

Minimum information supplied to the control system in a CNC program is the offset command G41 or G42, which must always be combined with the H or D address in effect. The command is applied during single axis motion, but multi-axis motion is allowed, if programmed carefully:

G41 X.. D.. or..

G42 X.. D.. or..

G41 Y.. D.. or..

G42 Y.. D..

Inclusion or exclusion of a tool motion and how many axes can be used at a time will be discussed in this chapter as well. First, let's resolve the question of which address to use and when. *H address or D address?*

### ◆ Address H or D ?

With the three types of *Tool Memory Offset*, it is reasonable to expect somewhat different programming methods for each type. Up to a point, this is true.

Both *Type A* and *Type B* are shared type offsets, with only a single register, where tool length offsets are stored along with cutter radius offsets.

Normally, *Type A* and *Type B* are associated with the address H only. That means the H address is used with the G43 command, *as well as* with the G41 or G42 commands. Many cutting tools do not require cutter radius offset in the program, but *all* cutting tools require tool length offset in the program.



If a particular cutter requires *both* tool length offset number *and* cutter radius offset number, *two different offset numbers* from the *same offset range* must be used in the program and stored in the control register. That is the reason these offsets are called *shared* offsets.

For example, programmed tool T05 requires both offsets, which obviously cannot have the same offset number. The solution is to use the tool number as the tool length offset number and increase that number by 20, 30, 40, or so, for the cutter radius offset. Entry for *Type A* in the offset screen could be similar to the one in *Figure 30-14*:

Offset No.	Offset
...	...
05	-8.6640
...	...
35	0.3750

Figure 30-14

Shared offset register screen for tool offset memory TYPE A

For offset *Type B*, there are *two* columns available, but it is still a shared offset. Entry for *Type B* in the offset screen will be similar to the *Type A*, shown in *Figure 30-15*:

Offset No.	Geometry	Wear
...	...	...
05	-8.6640	0.0000
...	...	...
35	0.3750	0.0000

Figure 30-15

Shared offset register screen for tool offset memory TYPE A

*Type C* offset will have *two pairs* of columns. Since both tool length and cutter radius have each their own columns, the *same offset number* can be used for both - there is no need for the 20, 30, 40 or so, increment. In this case, the H address will be reserved exclusively for tool length offset number and the D address will be reserved exclusively for the cutter radius offset number. *Figure 30-16* shows an input logically corresponding to the *Type A* and *Type B*:

Offset No.	H-offset		D-offset	
	Geometry	Wear	Geometry	Wear
...	...	...	...	...
05	-8.6640	0.0000	0.3750	0.0000
...	...	...	...	...

Figure 30-16

Unique offset register screen for tool offset memory TYPE C

*Most modern controls allow using D address for cutter radius offset, regardless of which offset type is used.*

## ◆ Geometry and Wear Offsets

Similar to the application of geometry and wear offsets for *tool length offset*, described in a separate chapter starting on *page 135*, virtually identical general rules can be used for cutter radius offset as well.

Offset settings entered in the *Geometry* offset column should only contain the nominal cutter radius. In the examples, a Ø0.750 cutter has been used, with the radius of 0.375. That is the nominal value and that would also be the typical value entered into the *Geometry* offset column. The *Wear* offset column should only be used for adjustments, or fine tuning, relative to the cutter nominal size, as required during setup and/or machining. There is no separate column for adjustment or fine tuning for the *Type A* offset. Adjustments can still be made, but the cutter nominal size will not be preserved - only the most recent change will be displayed as current.

## APPLYING CUTTER RADIUS OFFSET

All programming items required to successfully apply cutter radius offset in CNC programs have been identified. Actual applications, ways to use the offset in a CNC program, as well as various methods of its proper usage, will be discussed next.

There are *four major keys* to a successful use of cutter radius offset feature - they apply equally to all machine types:

1. To know how to **start** the offset
2. To know how to **change** the offset
3. To know how to **end** the offset
4. To know what to **watch** between the start and end

Each item is important and will be discussed in order.

## ◆ Startup Methods

Contrary to the belief of many inexperienced CNC programmers, starting up the cutter radius offset is much more than just using G41X..D.. block (or a similar entry) in the program. Starting up cutter radius offset means adherence to two fundamental rules and several important considerations and decisions. The fundamental rule number one is simple - it relates to the start position of the cutter:

Always select start position of the cutter away from the contour, in a clear area

The fundamental rule number two is also simple and is based on strict adherence to the first rule:

Always apply the cutter radius offset together with a tool motion

Although fundamental, these two rules are not arbitrary - rules can be broken. Practical suggestion offered here is to follow the rules until a better way is found. When selecting a startup tool position, a few questions are worth asking:

- What is the intended cutter diameter?
- What clearances are required?
- Which direction will the tool take?
- Is there no danger of collision?
- Can other diameter cutter be used if needed?
- How much stock is to be removed?

The same drawing example used earlier will be used for this example as well and cutter radius offset will be applied to the contour. To turn the offset on, *to make it effective*, the cutter will be *away* from the actual cutting area, in clear. The intended cutter size is  $\varnothing 0.750$  (R0.375), climb mill mode is desired, and 0.250 clearance is away from the contour. With these numbers, the start position is calculated at X-0.625 Y-0.625. Figure 30-17 shows the start position that satisfies all rules and answers the questions established earlier.

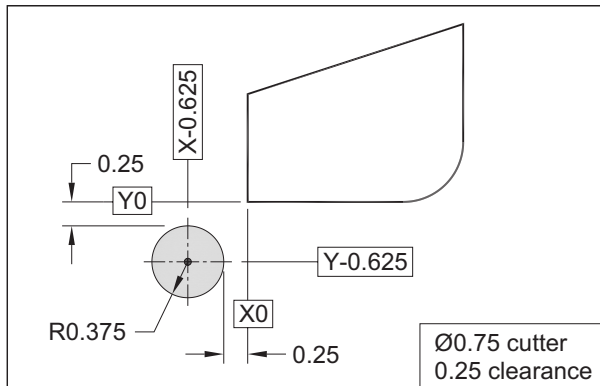


Figure 30-17

Off-contour start position of the cutter before radius offset is applied

Of course, the suggested location is not the only one suitable, but it is just as good as other possibilities. Note that the cutter located at position X-0.625 Y-0.625 is *not* compensated, both coordinates are measured to the *cutter center*: Once the start location is established, first few blocks of the program can be written:

```
O3001 (DRAWING FIGURE 30-2)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
N4 G43 Z1.0 H01
N5 G01 Z-0.55 F25.0 M08 (FOR 0.5 PLATE THICK)
N6 ...
```

For extra safety, cutter approach in blocks N4 and N5 to the depth of Z-0.55 (based on a 0.5 inch plate thickness) was split into two motions, although the cutter is still safely

above the clear area. Once the depth has been reached, the first motion can be programmed. Cutting direction is to the left of part (climb milling) and G41 command is applied. Moving tool around the part on the left side, means the first target point on the part has to be X0Y1.125 location. However, this position cannot be reached directly, because the left side of the part has to be machined as well. That means the tool has to reach the X0 position first. Next decision is selecting the Y position, to get the target point. Normally, this is done by programming a so called *lead-in* motion, or a cutter *entry* motion.

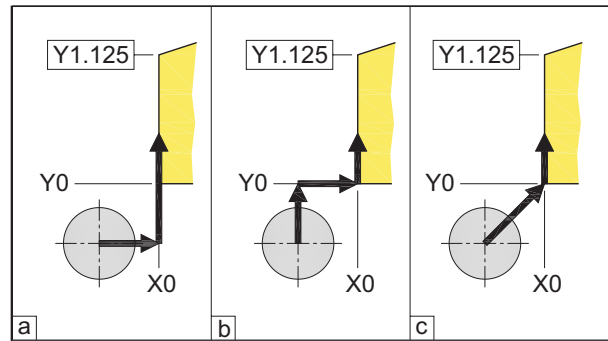


Figure 30-18

Possible lead-in motions used to activate cutter radius offset mode

Figure 30-18 shows some possibilities, all of them correct and all reaching the X0Y1.125 location eventually. Which one is the best? Here are some possible options:

The (a) option is simple - the tool moves towards the X0 first and the cutter radius offset is turned on *during* that motion. Then, the tool continues towards the first target point (Y1.125), already in compensated mode.

These two motions will appear in the program as:

```
N. . G01 G41 X0 D01 F15.0
N. . Y1.125 (P2)
N. . ...
```

The option [b] is technically correct, but requires three motions, whereas two motions are quite sufficient. This version will not be selected for the final program, although the program would still be correct:

```
N. . G01 G41 Y0 D01 F15.0
N. . X0
N. . Y1.125 (P2)
N. . ...
```

The last possibility - option [c] - is also simple and requires only two motions:

```
N. . G01 G41 X0 Y0 D01 F15.0
N. . Y1.125 (P2)
N. . ...
```

In all three versions, the cutter radius offset is started *together with the first motion*, while still away from the actual part contour. Because the option © actually ends *on* the part, selecting option (a) should be the preferred programming method of lead-in. Combination of (a) and © is also a good choice, with the Y-axis target in the negative area.

Once the offset has been turned on, all contour change points can be programmed along the part and the control computer will do its work by constantly keeping the cutter properly offset at all times. Program O3001 can now be extended up to point P5 in the original illustration:

```
O3001 (DRAWING FIGURE 30-2)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
N4 G43 Z1.0 H01
N5 G01 Z-0.55 F25.0 M08 (FOR 0.5 PLATE THICK)
N6 G41 X0 D01 F15.0 (START OFFSET)
N7 Y1.125 (P2)
N8 X2.25 Y1.8561 (P3)
N9 Y0.625 (P4)
N10 G02 X1.625 Y0 I-0.625 J0 (P5)
N11 G01 X. .
```

At block N10, cutter has reached the end of the 0.625 radius. Contouring is not yet finished - the bottom side has to be cut, along the X-axis. The question is - how far to cut and when to cancel the cutter radius offset?

This is the last cut on the part, so it still has to be machined *while the offset is in effect!* The cutter can end at X0, but that is not a practical position - it should move a bit farther, still along the X-axis only. How far is further? Why not to the same X-0.625 - the original start position? This is not the only clearance position available, but is the safest, most reliable and consistent. Block N11 will be:

```
N11 G01 X-0.625
```

Now the cutter has left the part contour area and cutter radius offset is not required anymore. It will be canceled shortly, but a little review of the startup may help.

Actual cutter radius was known for this job, which is not always the case. CNC programmer needs a suitable tool, because the cutting values depend on it. Within reason, a  $\varnothing 0.750$  or  $\varnothing 0.875$  cutter are not far apart - *except for clearances*. Clearance of 0.250 was selected for 0.375 cutter radius. That means the program is still good for cutters up to and including  $\varnothing 1.25$ . CNC operator has this freedom, because *the only change* is to the D01 offset amount in the control system offset registry. Speeds and feeds may have to be adjusted, if necessary. We will look later at what exactly happens when the cutter radius offset is applied.

The general rule in establishing a suitable start position is simple - it should always be selected with a clearance that is greater than the radius of the *largest* cutter that may be con-

sidered for cutting. This clearance may be increased for a large stock left on material or for a tool that is above average diameter. In order to complete the program, let's look at methods of canceling the cutter radius offset, when it is no longer needed.

### ◆ Offset Cancellation

A *lead-in* motion has been used at the startup of the cutter radius offset. To cancel the offset, a *lead-out* motion will be used. Length of the lead-out (just as the length of lead-in) has to be somewhat greater than or at least equal to the cutter radius. The lead-in and the lead-out motions are also called *ramp-in* and *ramp-out* motions.

The safest place to cancel cutter radius offset, for any machine, is away from the contour just finished. This should always be a clear area position. The start position can also be the end position. *Figure 30-19* shows the offset cancellation in the example. Program O3001 can be now be completed.

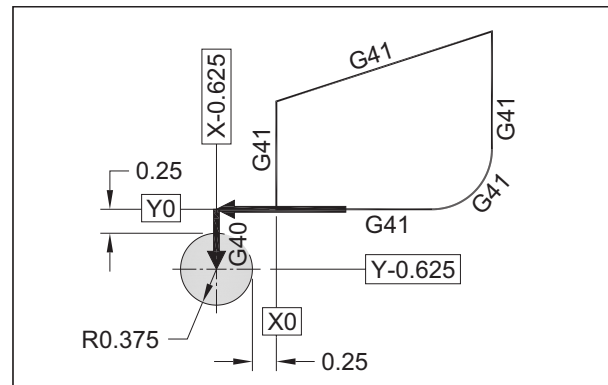


Figure 30-19

Cutter radius offset cancellation - program O3001

```
O3001 (DRAWING FIGURE 30-2)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
N4 G43 Z1.0 H01
N5 G01 Z-0.55 F25.0 M08 (FOR 0.5 PLATE THICK)
N6 G41 X0 D01 F15.0 (START OFFSET)
N7 Y1.125 (P2)
N8 X2.25 Y1.8561 (P3)
N9 Y0.625 (P4)
N10 G02 X1.625 Y0 I-0.625 J0 (P5)
N11 G01 X-0.625
N12 G00 G40 Y-0.625 (CANCEL OFFSET)
N13 Z1.0 M09
N14 G28 X-0.625 Y-0.625 Z1.0
N15 M30
%
```

Finally, program O3001 is ready use. There was no need for any change of the tool direction - such a change is rather a rare occurrence, at least for contouring operations using milling controls. Since some directional change may be useful in the future, some comments may also be useful.

### ◆ Cutter Direction Change

During a normal milling cut, there will seldom be a need to change the cutter offset direction from left to right or from right to left. If it does become necessary, the normal practice is to change from one mode to the other *without* canceling the G40 command. This practice is seldom used in milling, because change from G41 to G42 would also be a change from the preferred climb milling to the less preferred conventional milling. However, it is quite common in CNC lathe programming, with examples shown later.

## HOW CUTTER RADIUS OFFSET WORKS

Being able to program from given examples is certainly a good way to learn. Learning by a recipe or a sample does help in many cases, but it will not help much in cases where there is no sample, no recipe, and no example. In those cases, it is critical to really *understand* all principles behind the subject, such as principles of the cutter radius offset. The startup method is a good beginning. Next question is - what does happen during the tool motion in block N6?

N6 G41 X0 D01 F15.0

It is not as simple as it looks. We cannot evaluate just one block, such as N6, and know exactly what happens. The programmer has to understand what the control will do. Computers do not think, they only execute programmed instructions and follow these instructions very diligently. Block N6 is an instruction: *Move to X0, apply the radius setting stored in D01 to the left, during a linear motion at 15 in/min.* This is a program instruction to the control system. Where exactly does the tool stop? Look at Figure 30-20:

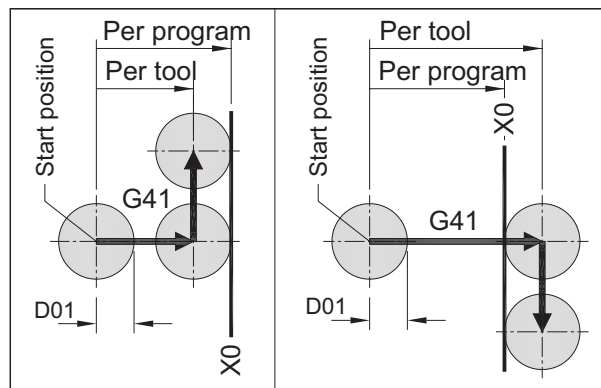


Figure 30-20

Ambiguous startup for a cutter motion in radius offset mode

Yes, there are *two* possibilities and they are both *correct*! Both versions compensate the cutter to the *left* of X0 target position. Conditions specified in block N6 have been fully satisfied - cutting tool moves to X0 as expected, offset is turned on to the *left* of part contour, *during* motion, using radius value stored in offset register D01. Looks good.

It may look good but the situation is *ambiguous*. There are *two possible outcomes*, while only one is required. Which one? For this job, the left one in the illustration, one where the tool moves along the Y+ direction next, after the radius offset has been applied. *This is the key!* Motion direction that follows G41 or G42 block *must* be known to the control. Look at two ways the program can be written:

#### ➡ Example 1 - Figure 30-21 (left) :

The next target position after N6 is Y *positive* direction:

```
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
...
N6 G41 X0 D01 F15.0 (START OFFSET)
N7 Y1.125 (POSITIVE Y-MOTION FOLLOWS)
```

#### ➡ Example 2 - Figure 30-21 (right) :

The next target position after N6 is Y *negative* direction:

```
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
...
N6 G41 X0 D01 F15.0 (START OFFSET)
N7 Y-1.125 (NEGATIVE Y-MOTION FOLLOWS)
```

In both cases, actual content of block N6 is the same, but the motion that follows block N6 is not - Figure 30-21.

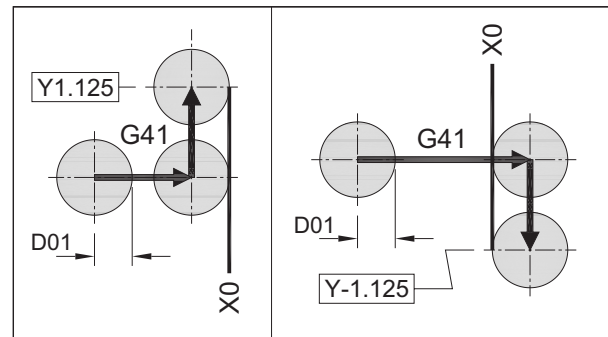


Figure 30-21

Importance of the next tool motion for cutter radius offset  
Y+ next direction on the left, Y- next direction on the right

### ◆ Look-Ahead Offset Type

While correct, the above block N6 alone does not contain sufficient amount of data to successfully apply the radius offset. The *next motion* - in fact, the *direction of the next motion* - must be known to the control system at all times!

How does the control handle such requirement? Controls using the cutter radius offset *Type C* have a built-in feature called the 'look-ahead' type of cutter radius offset.

*Look-ahead* feature is based on the principle known as *buffering* or *reading-ahead*. Normally, the control processor executes one block at a time. There will never be a motion caused by any buffered block (next program block).



In a short overview, this is the sequence of events:

- Control will first read the block containing startup of the cutter radius offset (that is block N6)
- Control detects an ambiguous situation, and does not process the block as yet
- Control advances the processing (buffering) to the next block (that is N7), to find out into which direction the tool will be programmed next
- During the 'next block reading', there is no motion at all - the control will only *register* the direction towards the target point and applies radius offset on the correct side of the part contour, during the startup block (N6 in the example)

This look-ahead type of cutter radius offset is very advanced internally in the software, but makes the contour programming so much easier on a daily basis. As maybe expected, there are some situations to be aware of.

### ◆ Rules for Look-Ahead Cutter Radius Offset

Look at the following sample program selection, not related to any previous examples:

☉ Example - single NO MOTION block :

```

N17 G90 G54 G00 X-0.75 Y-0.75 S800 M03
...
N20 G41 X0 D01 F17.0           (START OFFSET)
N21 M08                       (NO MOTION BLOCK)
N22 Y2.5                       (MOTION BLOCK)
...
```

What is the difference in program structure? Ignore the reason for the coolant ON function in block N21. If it can be justified, there is nothing wrong with it. The fact remains - there is *no axis motion* in block N21, which is the same block control system will *look ahead* to for the direction of next tool motion. Look at one more program selection - again, in a new example:

☉ Example - two NO MOTION blocks :

```

N17 G90 G54 G00 X-0.75 Y-0.75 S800 M03
...
N20 G41 X0 D01 F17.0           (START OFFSET)
N21 M08                       (NO MOTION BLOCK)
N22 G04 P1000                 (NO MOTION BLOCK)
N23 Y2.5                       (MOTION BLOCK)
...
```

Uncomfortable, perhaps - but not wrong - this time there are *two consecutive blocks* following the cutter radius offset - two consecutive blocks that *do not include any motion*.

Both examples present a program that might be fine if the radius offset were not applied. With offset in effect, such program structure can create problems. Controls with the 'look-ahead' feature can look ahead only so many blocks.

As most controls are of the look-ahead type, *one block look-ahead (buffering) is always guaranteed*. Many controls have *two* or more (even 1000) look-ahead blocks available. It all depends on actual control features - not all controls are the same. Here are some basic suggestions:

- If the control has a look-ahead type cutter radius offset feature, but the number of blocks that can be processed ahead is not known, assume it is only ONE block
- Make a test program to find out how many blocks the control can read ahead
- Once the cutter radius offset is programmed, try hard not to include any non-motion blocks - restructure the program, if necessary

Keep in mind that the control subjects any program input to the rules embedded in its software. Correct *input* must be provided first, in the form of an accurate part program.

What kind of a response can be expected if cutter radius offset is programmed wrong? Probably a scrap of the part. If control system cannot calculate the offset cutter position, it will act as if the offset were not programmed at all. That means, the initial tool motion will be towards X0 with the *cutter center*. When all necessary information is passed on to the control, offset will be applied, usually too late, after the cutter has entered the part. Scrap is the most likely result in this case. This *incorrect* program entry is graphically shown in Figure 30-22:

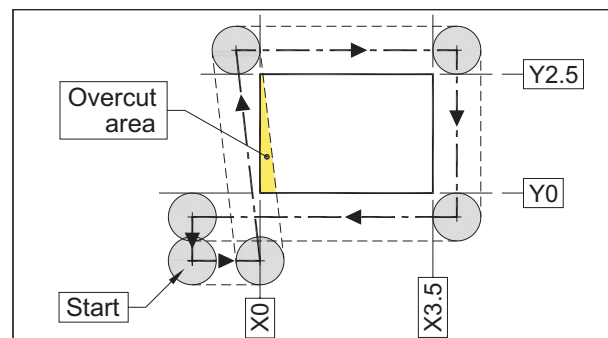


Figure 30-22

Tool path error due to wrong program structure - program O3002

There are two *no-motion* blocks in the program example O3002 that cause this error. They are after the cutter radius offset had been applied - in blocks N7 and N8. If the control system can look ahead only a single block, the program is wrong and the corrupted tool path shown in the above illustration will be the result - *severe overcutting*.

### O3002 (PROGRAM WITH RADIUS OFFSET ERROR)

```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-0.5 Y-0.5 S1100 M03
N4 G43 Z1.0 H01
N5 G01 Z-0.55 F20.0           (FOR 0.5 PLATE THICK)
```



```

N6 G41 X0 D01 F12.0          (START OFFSET)
N7 M08                      (NO MOTION BLOCK)
N8 G04 P1000                (NO MOTION BLOCK)
N9 Y2.5                     (MOTION BLOCK)
N10 X3.5                    (MOTION BLOCK)
N11 Y0                      (MOTION BLOCK)
N12 G01 X-0.5               (MOTION BLOCK)
N13 G00 G40 Y-0.5           (CANCEL OFFSET)
N14 Z1.0 M09
N15 G28 X-0.5 Y-0.5 Z1.0
N16 M30
%
```

Control that can read only *one* or *two* blocks ahead will *not* process program O3002 correctly - the next tool motion is in the *third* block when the offset is in effect. In order to avoid this incorrect motion, always avoid any program structure that contains more than one no-motion block in cutter radius offset mode.

### ◆ Radius of the Cutter

Every milling cutter has a diameter and one half of that diameter is the cutter radius. With new tools, the radius is always known and is sufficiently accurate. Accuracy of the radius depends on the cutter quality as well as on the way it is mounted in machine spindle. A run-off of 0.025-0.05 mm (0.001-0.002 inches) may not be a problem for roughing operations, but for precision finish, much higher accuracy is needed. Also needed is a way to *correct* for a tool wear and even slight tool deflection. All these adjustments are done through the D offset number, used as a pointer to the actual radius amount stored in the control register.

One simple - but very basic - rule should help to make sure the cutter radius offset will not fail:

Cutter radius should be smaller  
than programmed length of the tool travel

For example, in program O3001, the tool starting position is at X-0.625, the target position is X0. That means the programmed length of tool travel is 0.625. The radius selected was 0.375, which is smaller and adheres to the rule. There are two other possibilities.

- One possibility is where cutter radius is exactly the *same* as the programmed length of tool travel - this situation makes the starting position *equal* to the cutter radius
- Second possibility is where cutter radius is *larger* than the programmed length of tool travel - this situation makes the start position *smaller* than the cutter radius

Figure 30-23 shows a start position of a cutter that has the *same* programmed length of travel as the cutter radius. This is certainly allowed, but definitely not recommended. The reason is it limits the range of adjustments that can be made to the actual cutter radius during machining.

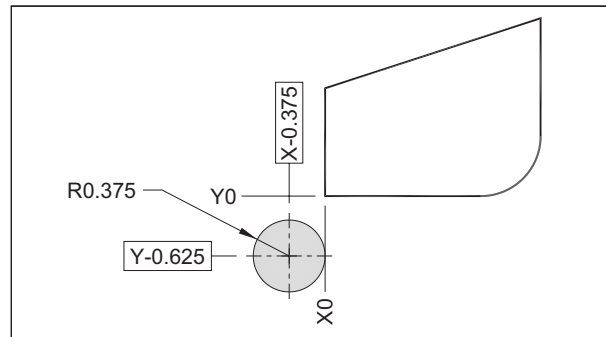


Figure 30-23

Cutter start position is equal to the cutter radius

The following program example shows 0.375 length of travel along the X-axis. If the D01 setting amount is *less* than 0.375, there will always be a motion toward X0, even if it is equal to the minimum motion amount (0.001mm or 0.0001 inch). If the D01 amount is *equal* to 0.375, the difference between the programmed length and the actual length is zero and there will *not be any motion* along the X-axis. In that case, the offset of the radius takes place without a movement and the subsequent motion to the target position Y1.125 will continue.

```

...
N3 G90 G00 G54 X-0.375 Y-0.625 S920 M03
...
N6 G41 X0 D01 F15.0          (START OFFSET)
N7 Y1.125                    (P2)
```

Always avoid a situation that provides zero tool length during offset activation - although logically correct, it does not provide any flexibility and can cause serious difficulties at some time in the future. For example, if D01 setting is changed from 0.375 to 0.376, the radius offset will fail - it is impossible to fit a large tool radius into a small area.

Figure 30-24 shows a start position where the cutter is partially on the *other side* of target position. This is definitely not allowed and control system will respond with an alarm warning - the infamous 'Cutter radius interference' alarm or 'CRC interference' message, like Alarm #041.

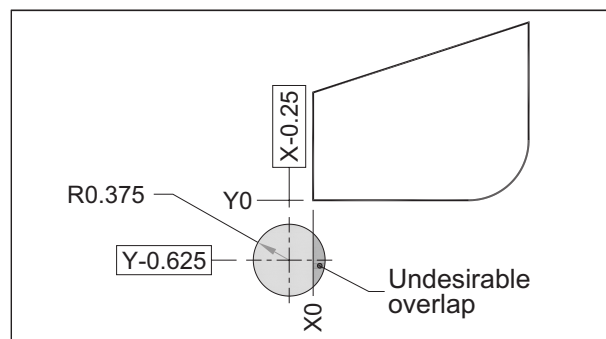


Figure 30-24

Cutter start position is smaller than the cutter radius

The following program sample is very similar to the previous examples, except the X-axis start position is too close to the target position, if the cutter radius is stored in the D01 register in the amount of 0.3750:

```
N3 G90 G54 G00 X-0.25 Y-0.625 S920 M03
...
N6 G41 X0 D01 F15.0          (START OFFSET)
N7 Y1.125                    (P2)
```

What will happen here? As usually, control system calculates the difference between programmed travel length of 0.25 and cutter radius 0.375. It will check the next travel direction as Y positive and determines that because the cutter is positioned to the left of the intended motion, it has to move 0.125 in the opposite way - in the X minus direction! That does not seem to be a problem, because there is a plenty of free space. *But there is a problem* - the control *does not recognize* the fact that there is a free space! Programmer knows it, but the control does not. Engineers who designed the software could have taken a number of actions - yet, they wisely decided to play it safe. They have decided to let the control system to reject this possibility and issue an alarm - an error condition. Depending on control system, the common alarm 'Overcutting will occur in cutter radius compensation C' or 'CRC interference' or a similar message will appear - the common alarm number for this error is *No. 041* on Fanuc control systems, for example. Many programmers, even those with long experience, have experienced this alarm. If not, they were either very fortunate or have never used cutter radius offset in the program.

Anytime the cutter radius interference alarm occurs, always look at the *surrounding blocks* as well, not just at the one where the control stops processing.

In the next section, the focus will be at the cutter radius interference that occurs *during* a tool motion, not just at the startup or termination of cutter radius offset.

### ◆ Radius Offset Interference

The last example illustrated only one of several possibilities, when the cutter radius offset alarm may occur. Another cause for this alarm is when a cutter radius is trying to enter an area that is *smaller* than the cutter radius, stored as the D-offset amount. To illustrate, evaluate the next program O3003, for a simple part shown in *Figure 30-25*.

```
O3003 (DRAWING FIGURE 30-25)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
N4 G43 Z1.0 H01
N5 G01 Z-0.55 F25.0 M08 (FOR 0.5 PLATE THICK)
N6 G41 X0 D01 F15.0      (START OFFSET)
N7 Y0.925
N8 G02 X0.2 Y1.125 I0.2 J0 (R0.2)
N9 G01 X1.0
N10 Y0.75
```

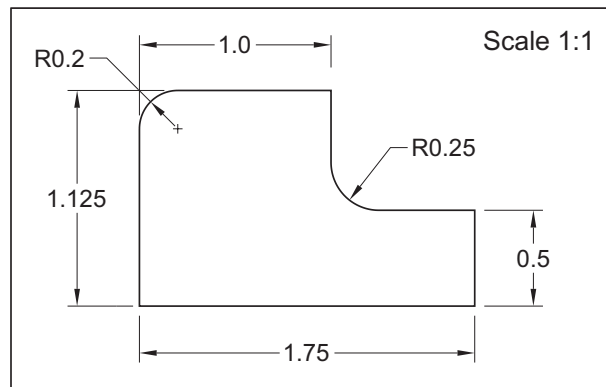


Figure 30-25

Simple drawing for program O3003

```
N11 G03 X1.25 Y0.5 I0.25 J0 (R0.25)
N12 G01 X1.75
N13 Y0
N14 X-0.625
N15 G00 G40 Y-0.625 (CANCEL OFFSET)
N16 Z1.0 M09
N17 G28 X-0.625 Y-0.625 Z1.0
N18 M30
%
```

The program is quite simple, it is correct and it follows all rules discussed so far. An important element is the cutter diameter selection - but even more important is the actual offset amount setting of the D address in the control offset register. Let's see what will happen - the same cutter is used as before, a  $\varnothing 0.750$  inch end mill. The amount of D01 stored in the control will be corresponding 0.3750.

Control unit will process information from the program combined with the offset amounts to determine the actual tool motion. Then, it executes programmed blocks as it moves the cutter along the part. Suddenly, at block N7 alarm *No. 041* occurs - *cutter radius interference problem*.

What has happened? There is nothing wrong with the program at all. Most CNC operators would look at the program and check it. After careful study, if they find it correct, the problem cause must be somewhere else, outside of program. Try not to blame the computer and don't waste any more time once you are satisfied the program is OK. Check the offset input in D01. The amount of 0.375 is stored there. That is also OK for the tool in the spindle. Check the drawing next - any errors? That is OK too. So while everything seems OK and there is still a radius offset alarm on the screen, do the next logical step.

Always evaluate three circular relationships between:

- Drawing dimensions ... *and* ... Program input
- Program input ... *and* ... Offset amounts
- Offset amounts ... *and* ... Drawing dimensions

This circular advice may take a while getting used to. It also requires a fair amount of experience as well. In the program example O3003, the problem is in a relationship of the stored offset amount and the drawing dimension.

Study the drawing carefully - there is an *internal* corner radius of 0.250 while the offset is set to the cutter radius of 0.375. This larger stored radius is expected to fit into the smaller 0.250 part radius. Obviously, that is not possible - hence the alarm.

Since the drawing dimension cannot be changed, the cutter diameter size must be changed, to a cutter diameter that is *less than* 0.500 inches. In summary, the problem was caused by selecting a cutter that is too large ( $\varnothing 0.75$ ). The other drawing radius of 0.200 is no problem, as *external* radiuses can have any size.

Fanuc controls (and similar control models) will *not* allow *gouging* in cutter radius offset *Type C*. This feature is built-in and there is no opportunity to see what would actually happen, if the protection were not there. Nobody wants to see gouging of the part, but *Figure 30-26* shows the same effect graphically. In fact, actual gouging (no warning) was a real problem in the earlier forms of cutter radius offsets *Type A* and *Type B*.

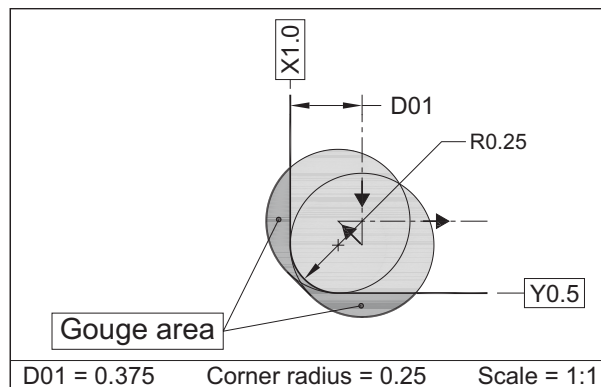


Figure 30-26

Effect of overcutting (gouging) in cutter radius offset mode  
Type C radius offset (look-ahead type) does not allow overcutting

### ◆ Single vs. Multiaxis Startup

There is another possible problem that can occur during cutter radius offset startup, particularly if programming the startup motion along *two axes*, rather than the suggested *single axis*. Earlier look at this possibility for external cutting has shown no problems. What about internal cutting?

Evaluate two different lead-in motions shown in *Figure 30-27*. Both are using a lead-in motion with G41 command towards an internal contour, for example, a wall of a pocket or similar contour. One motion is the recommended lead-in along a single axis, the other is a two-axis motion.

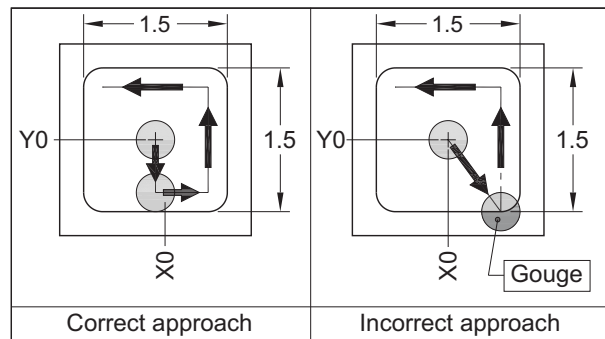


Figure 30-27

Possible problem in cutter radius offset mode during a lead-in with two axes simultaneously - internal cutting example

### ➡ Correct approach - SINGLE AXIS motion :

*Correct motion* approach shown on the left side of illustration contains the following relevant blocks:

```
N1 G20      (CORRECT APPROACH WITH A SINGLE AXIS)
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F6.0      (0.25 CONTOUR DEPTH)
N6 G41 Y-0.75 D01 F10.0 (START OFFSET)
N7 X0.75
N8 Y0.75
...
```

As corner radius of the part is the cutter radius, there is no internal radius in the program to worry about. Offset setting amount stored in register D01 does not have to consider it.

### ➡ Incorrect approach - MULTIAxis motion :

*Incorrect motion* approach shown on the right side of illustration contains the following relevant blocks:

```
N1 G20      (INCORRECT APPROACH WITH TWO AXES)
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F6.0      (0.25 CONTOUR DEPTH)
N6 G41 X0.75 Y-0.75 D01 F10.0 (START OFFSET)
N7 Y0.75
...
```

Why is there a severe overcut (gouge)? There is no way any control system can detect the contour bottom wall at Y-0.75. Lead-in motion for the offset is exactly the same as shown earlier for external cutting, but it's *not* in the clear.

Compare these two possible lead-ins for the drawing shown in *Figure 30-2*, at the beginning of this chapter. If the radius offset is started with a single axis motion, the result is shown at the left side illustration in *Figure 30-28*. If the offset is started with a two-axis motion, the result is shown at the right side illustration in *Figure 30-28*.

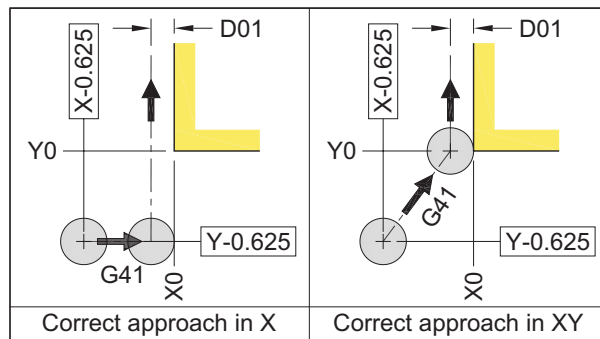


Figure 30-28

Startup of the cutter radius offset for external cutting:  
 - Single axis lead-in - shown on the left  
 - Two-axis lead-in - shown on the right

Here are the first few correct blocks of each method:

#### ➡ Correct approach - SINGLE AXIS motion :

```
N1 G20 (CORRECT APPROACH WITH ONE AXIS)
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
...
N6 G41 X0 D01 F15.0 (START OFFSET)
N7 Y1.125 (P2)
...
```

#### ➡ Correct approach - MULTIAXIS motion :

```
N1 G20 (CORRECT APPROACH WITH TWO AXES)
N2 G17 G40 G80
N3 G90 G54 G00 X-0.625 Y-0.625 S920 M03
...
N6 G41 X0 Y0 D01 F15.0 (START OFFSET)
N7 Y1.125 (P2)
...
```

Note that in cases of the cutter radius offset applied to an external contour, *both* programs listed are correct, because there is *no interference* with any section of the part. In reality, there is *the same* interference as in the internal milling example - the only difference is that this type of 'interference' is of no consequence in the majority of external contouring - the 'interference' is located in the air - below Y0 location in the XY motion.

There will always be problems that cannot be solved in any handbook, regardless of how comprehensive that book may be. Subjects and examples included in this handbook present some of the most important foundations for better understanding of the subject of cutter radius offset. With growing experience, such understanding becomes much deeper and easier.

Before going any further (yes, there is a lot more), let's review some general rules relating to the cutter radius offset feature.

## OVERVIEW OF GENERAL RULES

Reminders and rules are only important until a particular subject is fully understood. Until then, a general overview and some additional points of interest may come handy. Programming cutter radius offset is no different. Most of the following items apply to milling and turning, some are unique to milling only.

#### ➡ Milling only:

- Reach the Z-axis milling depth in G40 mode (cutter radius offset cancel mode)
- Do not forget the offset number D.. to include in the program - it is a small error that can cost a lot
- Retract from the depth (along Z-axis only) after the radius offset has been canceled
- Make sure the cutter radius offset corresponds to the work plane selected (chapter starts on page 279)

#### ➡ Milling and Turning:

- Never start or cancel radius offset in an arc cutting mode (with G02 or G03 in effect). Between the startup block and the cancel block, arc commands are allowed and normal, if the job requires them
- Make sure the cutter radius is always smaller than the smallest inside radius of the part contour
- In the canceled mode G40, move the cutter to a clear area. Always consider the cutter radius, as well as all reasonable clearances
- Apply cutter radius offset with G41 or G42 command, along with a rapid or a linear motion to the first contour element (only with G00 or G01 in effect)
- Use a single axis approach from the startup position
- Make sure to know exactly where the tool command point will be when the radius offset is applied along two axis
- In compensated mode (G41 or G42 in effect), watch for blocks that do not contain an axis motion. Avoid non-motion blocks if possible (missing X, Y and Z)
- Cancel cutter radius offset with G40 command, along with rapid or linear motion (G00/G01) only, preferably as single axis motion only
- G28/G30 machine zero return commands will not cancel cutter radius offset (but either one will cancel tool length offset)
- G40 command can be input through MDI to cancel any cutter radius offset (usually as a temporary or an emergency measure)

There are no unique turning requirements. At the end of this chapter, cutter radius offset for lathes will be covered separately - see page 275.



## PRACTICAL EXAMPLE - MILLING

The following in-depth example attempts to present a fairly complete and practical application of cutter radius offset for programming *and* machining purposes. It covers virtually all situations that can happen during the machining process and presents solutions to maintaining required dimensions of the part. When it comes to dimensions, the first subject that has to be thoroughly understood is the difference between *programmed* and *measured* part size.

### ◆ Part Tolerances

When machining is completed on a CNC machine or often even before that, the part has to pass through some inspection process. That means all drawing specifications have to be met for such part to pass inspection. One of the requirements is to maintain dimensional tolerances, either as *specified* in the drawing, or as *implied* in the drawing. Implied tolerances are often company established standards that are based on the number of decimal places used for dimensions (a method more and more on decline).

The next example focuses strictly on the effect of cutter radius offset on the part size in XY plane (top view). For that reason, only a simple application is presented, with the simplest tool path, but *not* necessarily the best machining method. Figure 30-29 shows the simple drawing.

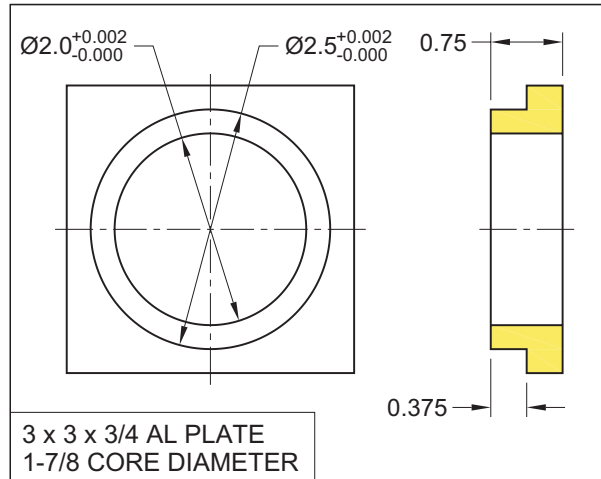


Figure 30-29

Drawing to illustrate practical cutter radius application  
- External and Internal

The main focus of the evaluation will be on the *specified* tolerance as being +0.002/-0.000, applied to dimensions of the two diameters - Ø2.5 external and Ø2.0 internal. Note that the *range* of all dimensional tolerances is the *same* for both diameters. This particular observation will be very important later.

### ◆ Measured Part Size

Every experienced machinist knows that the actual measured size of a part depends on many factors in addition to human factors. They include setup rigidity, cutting width and depth, material being used, cutting direction, the selection of tool, its exact size and quality, and so on.

When a part is inspected, its measured size can have only one of three possible outcomes:

- ON SIZE ... within specified tolerances
- OVERSIZE ... will be scrap for *internal* cutting
- UNDERSIZE ... will be scrap for *external* cutting

The first outcome is always ideal, regardless of whether external or internal cutting takes place. If the measured dimension is *on size* - that is *within* the specified tolerances - there is no need to do anything - *the part is good*. The second outcome (*oversize*) and the third outcome (*undersize*) have to be considered together - *the part is bad*.

In both latter cases, the measured dimension is *outside of specified tolerance* range. This situation requires a look at two additional items that also have to be considered:

- External cutting method ... known as OUTSIDE or OD
- Internal cutting method ... known as INSIDE or ID

Because the cutting tool approaches machined contour from different directions, the terms *oversize* and *undersize* are always relative to the *type of cutting*. The following table shows the most likely results:

Condition	External	Internal
OVERSIZE	Recut Possible	Scrap Likely
ON SIZE	No Action Required	No Action Required
UNDERSIZE	Scrap Likely	Recut Possible

Looking at the table, it is clear that no action is necessary if the measured part size is within tolerances, regardless of whether *external* or *internal* cutting took place. For the oversize or undersize results, a recut may be possible or a scrap will be the likely result.

#### EXTERNAL ...

A part machined *externally* (Ø2.500 inch OD in the example) that is determined to be *larger* than the allowed tolerance, can likely be recut, but a size that is *smaller* than the allowed tolerance will result in a scrap.

#### INTERNAL ...

A part machined *internally* (Ø2.000 inch ID in the example) that is determined to be *smaller* than the allowed tolerance, can likely be recut, but a size that is *larger* than the allowed tolerance will result in a scrap.



### ◆ Programmed Offsets

Probably the most attractive feature of cutter radius offset is that it allows a change of the actual tool size right on the machine - *by means of the offset register function D*. In the program example, only one tool is used - 0.750 inch diameter end mill - and one single cut for each contour (external and internal). Program X0Y0Z0 is at the center of the circles and the top of part:

```
O3004      (T01 - 0.75 DIA END FINISHING MILL)

(**** PART 1 - 2.5 DIA EXTERNAL CUTTING ****)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y2.5 S600 M03 (START POS.)
N4 G43 Z0.1 H01 M08 (CLEAR+TOOL LG.)
N5 G01 Z-0.375 F20.0 (DEPTH FOR 2.5 DIA)
N6 G41 Y1.25 D01 F10.0 (LEAD-IN MOTION)
N7 G02 J-1.25 (EXT. CIRCLE CUTTING)
N8 G01 G40 Y2.5 (LEAD-OUT MOTION)
N9 G00 Z0.1 (CLEAR ABOVE)

(**** PART 2 - 2.0 DIA INTERNAL CUTTING ****)
N10 Y0 (START POS. AT X0Y0)
N11 G01 Z-0.8 F20.0 (DEPTH FOR 2.0 DIA)
N12 G41 Y1.0 D11 F8.0 (LEAD-IN MOTION)
N13 G03 J-1.0 (INT. CIRCLE CUTTING)
N14 G01 G40 Y0 (LEAD-OUT MOTION)
N15 G00 Z0.1 M09 (CLEAR ABOVE)
N16 G28 Z0.1 M05 (Z AXIS MACHINE ZERO)
N17 M01 (OPTIONAL STOP)
...
```

Figure 30-30 shows tool path for the first half of the program - external diameter of 2.500 inches. Figure 30-31 shows tool path for the second half of the program - internal diameter of 2.000 inches.

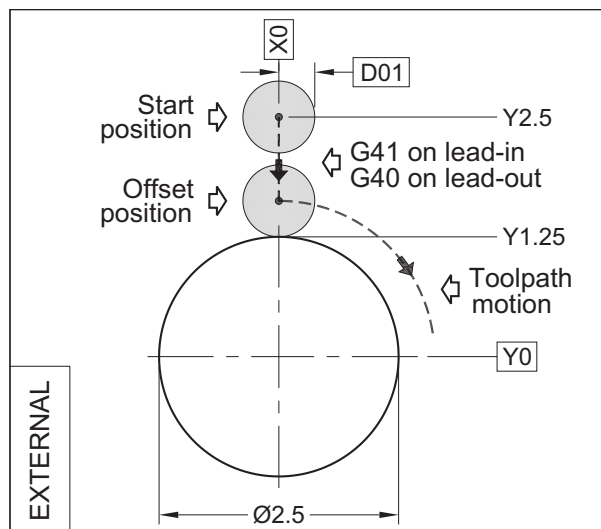


Figure 30-30

Detail for EXTERNAL tool path lead-in and lead-out - Example O3004

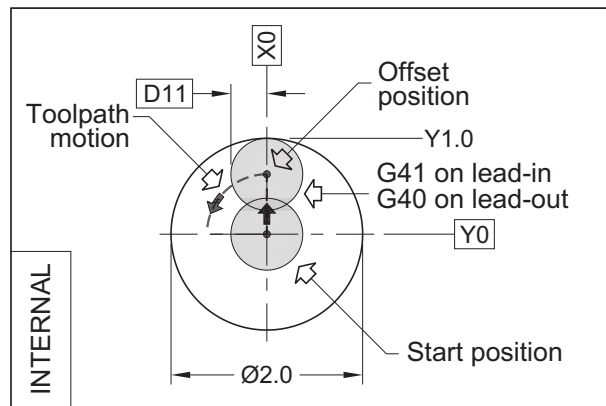


Figure 30-31

Detail for INTERNAL tool path lead-in and lead-out - Example O3004

As is customary in CNC programming, and also used in program O3004, the tool path uses *drawing* dimensions and other positions defined by the programmer. This is not only a standard method, but also the most convenient method to develop a CNC program. Such a program is easy to understand by the machine operator, drawing dimensions are easy to trace (if necessary) and changes can be made, if required. In plain language, the CNC programmer *ignores the cutter radius* and writes the program *as if the cutter were a point* - in effect, a 'special cutting tool' with a zero diameter.

### ◆ D-offset Amount - General Setting

One reality of machining is that a zero diameter cutter is not usually used, except for some engraving work. The majority of cutting tools do have specified diameters and these actual diameters have to be always considered - if not in the program, then on the machine.

One critical fact to be established first is that the CNC system always calculates a specified offset by its cutter *radius*, *not* by its diameter. It means the programmer provides the *cutter radius offset* in the form of a *D-address*. On the machine, programmed offset D01 will apply to the cutter radius registered in offset 1, D02 to the radius registered in offset 2, etc. What actual amounts are in these registers?

Since no radius of the cutter is included anywhere in the program itself, offset register D must normally contain the cutter radius *actual* value. Be careful - some machine parameters may actually be set to accept the cutter *diameter*, although all internal calculations are still set by the radius.

Evaluate program O3004; what will be the stored amount of D01? A Ø0.750 inch end mill is used, so D01 should be set to 0.375. This is correct in theory, but factors such as tool pressure, heat, material resistance, tool deflection, actual tool size, tool tolerances and other factors do influence the finished part size. Conclusion is that the D01 registered amount can be 0.375, but only under *ideal* conditions.

Ideal conditions are rare, of course. The same factors that influence machining will also have a significant effect on part dimensions. It is easy to see that any measured size that is not within tolerances can be only *oversize* or *undersize* and *external* and *internal* cutting method *does* make a difference as to how the offset can be adjusted.

Regardless of the cutting method, there is one major rule applied to the cutter radius offset adjustment in any control system - the rule has two equal parts:

POSITIVE increment to stored cutter radius offset will cause the cutting tool to move AWAY from the machined contour

NEGATIVE increment to stored cutter radius offset will cause the cutting tool to move CLOSER to the machined contour

Note the word '*increment*' - it means that the current radius offset amount will be *changed* or *updated* - but *not* replaced - with a new amount. Concept of '*moving away*' and '*moving closer to*' the part refers to a tool motion as viewed by the CNC operator. The part measured size can be controlled by *adjusting* the cutter radius offset amount in the control, programmed as the D address, according to these two rules. The most useful rule that applies *equally to the external and internal* adjustments has two alternatives:

To ADD more material TO the measured size,  
use LARGER setting amount of the D offset

To REMOVE material FROM the measured size,  
use SMALLER setting amount of the D offset

Experienced CNC operators can change offset settings at the machine, providing the program contains cutter radius offset commands G41 or G42 as well as the D address offset number - with the appropriate cancellation by G40.

Evaluating what *exactly* happens during tool motion for each cutting method (*external* or *internal*) offers certain options. In both cases, the cutting tool moves from its *starting* position, within a clear area, to the contour *target* position. This is the motion where cutter radius offset is *applied*, so this motion is very critical. In fact, this is the motion that *determines the final measured size* of the part. Each method can be considered separately.

### ◆ Offset Adjustment

Before any special details can be even considered, think about how the offset amount can be changed. In those cases where the part size is to be adjusted, *incremental* change of the offset value is a good choice. Incremental offset change means *adding to* or *subtracting from* the current offset amount (using the +INPUT key on a Fanuc screen) or storing the adjustment in *Wear* offset screen column. Changes to the actual program data should *never* be an option.

### ◆ Offsets for External Cutting

Evaluate the tolerance range for the *outside* circle Ø2.5. Tolerance for this diameter is +0.002/-0.0, so all sizes between 2.500 and 2.502 are correct. Any size *smaller* than 2.5 is *undersize* and a size *greater* than 2.502 is *oversize*.

There are three possible results of the measured size for *external* cutting. All examples are based on the expected middle size of 2.501 and D01 storing the amount of 0.375, which is the radius of a Ø0.750 milling cutter.

#### ➡ External - Example 1 :

2.5010 *measured, when setting of* D01 = 0.3750

This is the ideal result - no offset adjustment is necessary. The tool cutting edge touches the intended machining surface exactly. All is working well and the offset setting is accurate. Only standard monitoring is required. This is not such a rare situation as it seems - in fact, it is quite common with a new cutter, rigid setup and common tolerances.

#### ➡ External - Example 2 :

2.5060 *measured, when setting of* D01 = 0.3750

Measured diameter is 0.005 *oversize*. Tool edge has *not reached* the contour and *has to move closer* to it. The radius offset amount has to *decrease* by one half of the oversize amount, which is on the diameter (or width) but the offset amount is entered as a radius, per side. Offset D01 will be adjusted incrementally by 0.0025, to D01=0.3725.

#### ➡ External - Example 3:

2.4930 *measured, when setting of* D01 = 0.3750

Measured diameter is 0.008 *undersize*. Tool edge has *reached beyond* the programmed machining surface and *has to move away* from it. The radius offset amount has to be *increased* by one half of the undersize amount. The undersize is measured on the diameter (or width), but the offset amount is entered as a radius, per side only. Offset D01 will be adjusted incrementally by 0.004, to D01=0.3790.

### ◆ Offsets for Internal Cutting

Now is the time to look at tolerance range for the *inside* diameter of 2.0 inches. The tolerance range for this diameter is also +0.002/-0.0, so all part sizes between 2.000 and 2.002 will be correct. A size *smaller* than 2.000 will be *undersize* and a size *greater* than 2.002 will be *oversize*.

There are three possible results of the measured size for *internal* cutting. All examples are based on the expected middle size of 2.001 and D11 storing the amount of 0.375, the radius of a Ø0.750 cutter.

### ☉ Internal - Example 4 :

2.2010 *measured, when setting of D11 = 0.3750*

This is the ideal result - no offset adjustment is necessary. Edge of tool cutting touches the intended machining surface exactly. All is working well and the offset setting is accurate. Only normal monitoring is required.

### ☉ Internal - Example 5 :

2.0060 *measured, when setting of D11 = 0.3750*

Measured diameter is *0.005 oversize*. Tool edge has *reached beyond* the programmed machining surface and *has to move away* from it. The radius offset amount has to be *increased* by one half of the oversize amount. The oversize is measured on the diameter (or width), but the offset amount is entered as a radius, per side only. Offset D11 will be adjusted incrementally by 0.0025, to D11=0.3775.

### ☉ Internal - Example 6 :

1.9930 *measured, when setting of D11 = 0.3750*

Measured diameter is *0.008 undersize*. Tool edge has *not reached* the programmed machining surface and *has to move closer* to it. The radius offset amount has to be *decreased* by one half of the undersize amount. The undersize is measured on the diameter (or width), but the offset amount is entered as a radius, per side only. Offset D11 will be adjusted incrementally by 0.004, to D11=0.3710.

### ◆ One Offset or Multiple Offsets?

Program O3004 used D01 for the external diameter and D11 for the internal diameter. Only one tool was used and the goal was the middle tolerance of 2.501 external diameter and 2.001 internal diameter. Are two offsets in the program needed or will a single offset do?

The last examples evaluated only possibilities that were separate from each other. Program O3004 presents a common connection between the two diameters - the Ø0.750 end mill - it is used for cutting *both* diameters.

Assume for a moment, that only one offset is used, equal to D01=0.375, for example. External diameter will be measured as 2.501, as before. When the internal diameter of 2.000 inches is measured, its size is not 2.001 as expected, but only 1.999. This measurement is 0.002 undersize then the expected diameter - *both* diameters have a +0.002/-0.000 tolerance. Outcomes are different - +0.002 for the external diameter means oversize that can be recut, +0.002 for the internal diameter means oversize that is a scrap. Since one offset alone cannot be adjusted to meet the middle tolerance on *both* diameters, *two offsets* have to be used. It follows, that if D01=0.3750 and makes a perfect external diameter, D11 should have an equivalent setting amount of only 0.3730.

CNC operator should always be aware of all offsets used in the program and understand about the stored offset amounts, especially if more than one offset is used for one tool

In a setup or tooling sheet (or even in the program itself), the programmer should always list all offsets used in the program and suggest their starting values for each offset as a professional courtesy.

### ◆ Preventing a Scrap

When it comes to initial offset amounts, some creative techniques can be used. The goal is to use offset settings in such a way that the part will not likely be a scrap, *even with an unproven tool*. A good operator can prevent scraps caused by wrong offsets, at least to some degree. Main key is to create some *temporary* offset settings. The idea is to force a cut that is *oversize externally* or *undersize internally*, measure it, adjust it, *then* recut to the correct size.

Whether machining external or internal tool path, even the best setup will not guarantee that all part dimensions will be within tolerances. When machining an *external* contour, the diameter (or width) can be cut *intentionally larger* than required - in a controlled way. In this case, there is a risk that the diameter could be *too small*.

In *internal* contouring, the diameter can be cut *intentionally smaller* than required - in a controlled way. In this case, there is a risk that the diameter (or width) could be *too large*. Either case offers benefits but some drawbacks, too.

For the last few examples, the solution is to move the tool *away from* the programmed external machined surface by a *positive offset increment*. The increment amount must be greater than the expected error of the tool radius, as well as being suitable for a recut.

In both cases, when the test cut is made, measure the diameter and adjust the offset by one half of the difference between measured and intended diameters (widths). If only one side is cut, the difference is *not* halved.

### ◆ Program Data - Nominal or Middle?

Many coordinate locations in the program reflect actual dimensions that are taken from the drawing. The question is - what happens if the drawing dimension specifies a tolerance range? There are two varying opinions among CNC programmers. One opinion favors programming the *middle* value of tolerance range, the other prefers to program the *nominal* size and ignore all tolerances. Both opinions have some credibility and neither should be discarded. In this handbook, the preference is to use the *nominal* dimensional sizes and let the tolerances be handled by proper use of offsets - at the machine. Two reasons prevail. One is that a program using nominal dimensions is easier to read. Two, in case of drawing changes, they will affect the tolerances more often than nominal sizes.

## TOOL NOSE RADIUS OFFSET

All general principles and rules described so far also apply to the radius offset for a lathe contouring tool. There are few differences, mainly caused by the shape of the tool.

In milling, the cutting tool is always round. The cutter circumference (periphery) is the cutting edge and its radius is the nominal offset. Turning tools have a different design. The most common is a multi-sided carbide insert. An insert may have one or more cutting edges. For strength and longer insert life, the cutting edge has a relatively small corner radius. Typical radiuses for turning and boring tools are:

0.40 mm (metric) or  $1/64 = 0.0156$  (imperial)  
 0.80 mm (metric) or  $1/32 = 0.0313$  (imperial)  
 1.20 mm (metric) or  $3/64 = 0.0469$  (imperial)

Because lathe tool cutting edge is often called a *tool nose*, the term *tool nose radius offset* became common.

### ◆ Tool Nose Tip

Actual tool nose is usually a corner of the tool, where two cutting edges blend into a nose radius. Figure 30-32 shows typical corners of a turning tool and a boring tool.

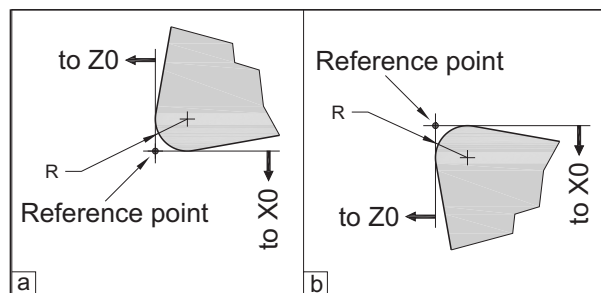


Figure 30-32

Tool reference point for typical turning (a) and boring (b) - rear lathe

The tool nose *reference point* in turning is often called the *command point*, *imaginary point* and, lately, even the *virtual point*. It is *this* point that is moved along the contour, because it is directly related to X0Z0 setting of the part.

### ◆ Radius Offset Commands

The same preparatory commands relating to cutter radius used in milling operations are also used for contouring on CNC lathes - Figure 30-33:

G41	Offset of tool nose radius to the LEFT of contouring direction
G42	Offset of tool nose radius to the RIGHT of contouring direction
G40	Offset of tool nose radius CANCEL

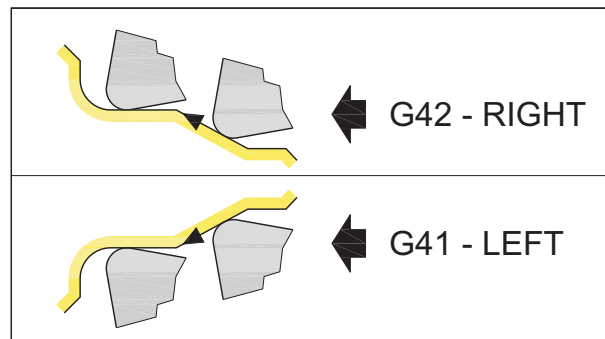


Figure 30-33

Tool nose radius offset applied to OD (G42) and ID (G41)

There is one major difference - for lathes, G-codes do not use the D-address - all actual offset values are stored in the *Geometry/Wear* offset. *Lathe tools have different cutting edges*, otherwise they are similar to milling.

### ◆ Tool Tip Orientation

The center of a circle symbolizing an end mill must be equidistant to the contour by its radius. In milling, cutting edges are part of the tool radius, on lathes, they are not. Lathe tools *do* have a radius but separate cutting edges. The nose radius center is also equidistant from the contour, and the edges change their orientation, even for the same insert. Additional definitions are needed in a form of a vector pointing from the *command point* towards the *radius center*. This vector is called the *tool tip orientation*, arbitrarily numbered. Control system uses this number to establish the nose radius center and its orientation from the command point. Figure 30-34 shows two tools and their tip orientation.

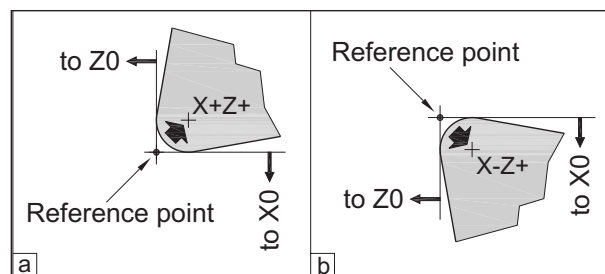


Figure 30-34

Relationship between tool nose reference point and radius center

Tool tip orientation is entered during the setup, according to some arbitrary rules. Fanuc controls require a fixed single digit number for each possible tool tip orientation. This number has to be entered into the offset screen at the control, under the T column heading. Value of the tool radius R must also be entered, as they both work together. If the tool tip is 0 (or 9), the control will compensate to the center. Figures 30-35 and 30-36 show the standard tool tip numbering for *rear orientation* CNC lathes - those with X+ up and Z+ to the right of origin.



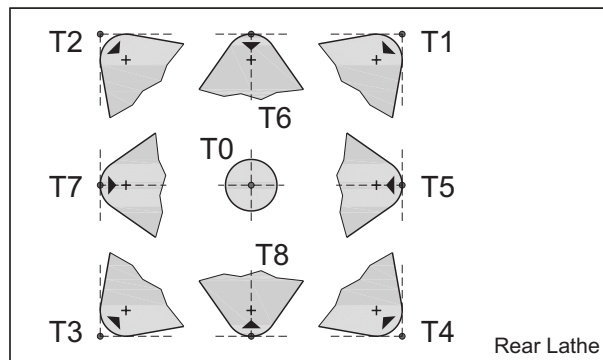


Figure 30-35

Arbitrary tool tip numbers for tool nose radius offset orientation

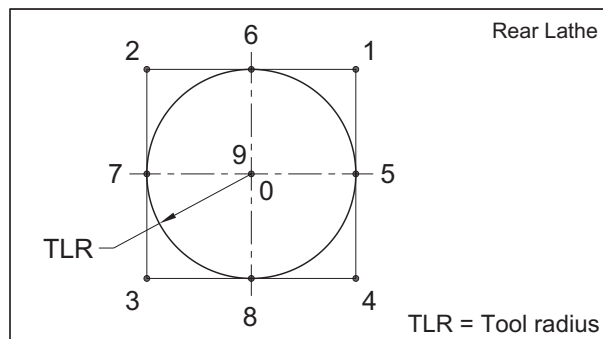


Figure 30-36

Schematic illustration of Fanuc type tool tip numbering

### ◆ Effect of Tool Nose Radius Offset

Some programmers do not bother using the tool nose radius offset. *That is wrong!* Study Figure 30-37 carefully first - explanations follow.

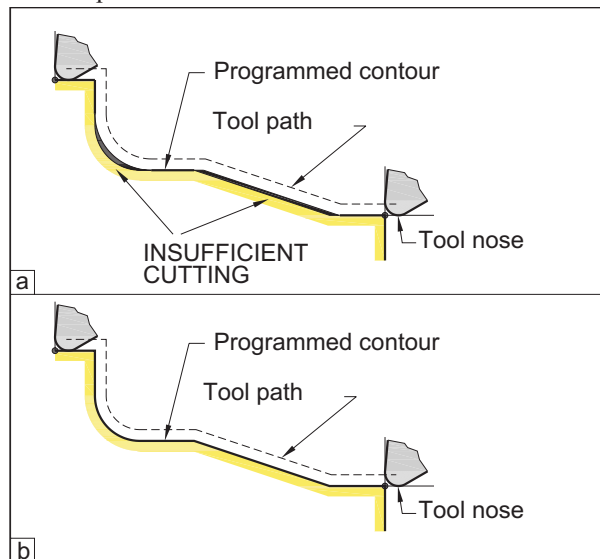


Figure 30-37

Effect of tool nose radius offset - (a) offset NOT used - (b) offset used

Theoretically, there is no need for the offset if only a single axis motion is programmed. However, single axis motions are typically part of a contour that also includes radii, chamfers and tapers. In such cases, tool nose radius offset is needed, otherwise all radii, chamfers and tapers will not be correct and part will be scrap.

The last illustration shows what areas of part would be *undercut* or *overcut*, if tool nose radius offset were *not* used during machining. Note that negative effect applies to a two-axis simultaneous motion only.

### ◆ Sample Program

The following program example O3005 shows a simple application of tool nose radius offset on an external and internal contour, based on the drawing in Figure 30-38. Only the finishing cuts are shown - roughing is also necessary, but would most likely use G71 multiple repetitive roughing cycle without radius offset (for G71 see page 325).

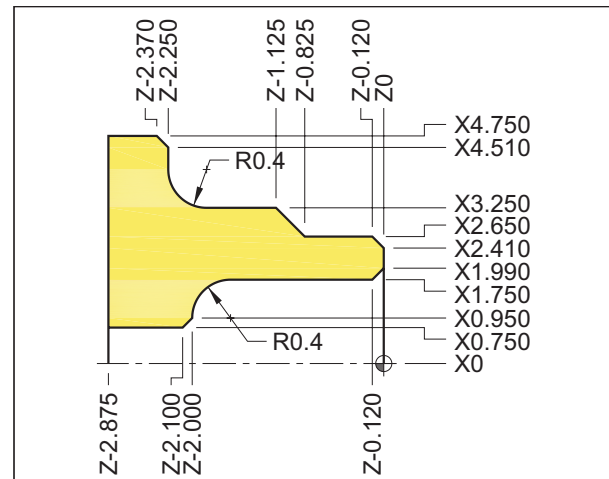


Figure 30-38

Simplified sample drawing for program example O3005

```

O3005
(G20) ...
N31 T0300                                     (EXTERNAL FINISHING)
N32 G96 S450 M03
N33 G00 G42 X2.21 Z0.1 T0303 M08
N34 G01 X2.65 Z-0.12 F0.007
N35 Z-0.825 F0.01
N36 X3.25 Z-1.125
N37 Z-1.85
N38 G02 X4.05 Z-2.25 R0.4
N39 G01 X4.51
N40 X4.8 Z-2.395
N41 U0.2
N42 G00 G40 X8.0 Z5.0 T0300
N43 M01

N44 T0400                                     (INTERNAL FINISHING)
N45 G96 S400 M03
N46 G00 G41 X2.19 Z0.1 T0404 M08
  
```



```

N47 G01 X1.75 Z-0.12 F0.006
N48 Z-1.6 F0.009
N49 G03 X0.95 Z-2.0 R0.4
N50 G01 X0.75 Z-2.1
N51 Z-2.925
N52 U-0.2
N53 G00 Z2.0
N54 G00 G40 X8.0 Z2.0 T0400
N55 M01
...

```

Note that both contour start and end positions are in a *clear area* - away from the part - for the same reason as in milling. Make sure to program sufficient clearance for both lead-in and lead-out motions. *Cutter radius compensation interference alarm* is always caused by insufficient clearance - cutter radius cannot fit into the space provided.

#### ◆ Minimum Clearance Required

As a rule, each physical clearance in the program should be large enough to accommodate **double** tool nose radius

The rule should be clear. *Figure 30-39* shows minimum clearances when set at the start and end of cut. Make sure the nose radius fits into this clearance two or more times. Symbols  $>TLR \times 2$  and  $\times 4$  mean the clearance should be greater than twice or four times the nose radius. Double radius per side becomes a quadruple radius on a diameter.

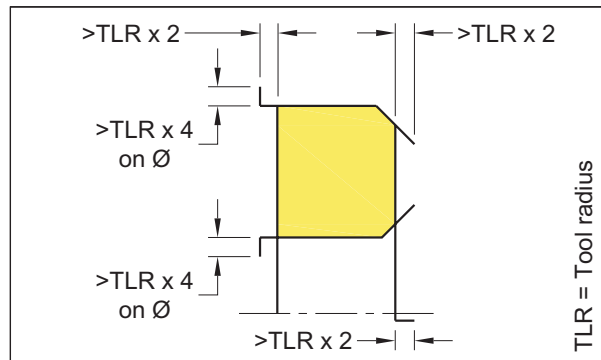


Figure 30-39

Minimum clearances for tool nose radius offset

	Nose radius	Per side	On diameter
mm	0.4	0.8	1.6
	0.8	1.6	3.2
	1.2	<b>2.4</b>	<b>4.8</b>
Inch	1/64 = 0.0156	0.0313	0.0626
	1/32 = 0.0313	0.0626	0.1252
	3/64 = 0.0469	<b>0.0938</b>	<b>0.1876</b>

The table shows minimum clearances required for different tool nose radius sizes. Note the four highlighted items.

The four highlighted items represent minimum clearances for the largest tool nose radius generally used in turning and boring. When they are rounded, they become easy to remember clearances - 2.5 or 5 mm and 0.1 or 0.2 inches. In tool nose radius offset, always programming the minimum clearance of at least 2.5 mm (0.100 inches) *per side*, will provide sufficient clearance for *all three standard tool nose radiuses* shown in the table.

There are two quite common programming errors when it comes to tool nose radius offset - one is for boring, the other for facing. Explanation of both errors will conclude this chapter.

#### ◆ Retraction from a Bored Hole

To illustrate this common problem in boring, a simple drawing example is presented in *Figure 30-40*:

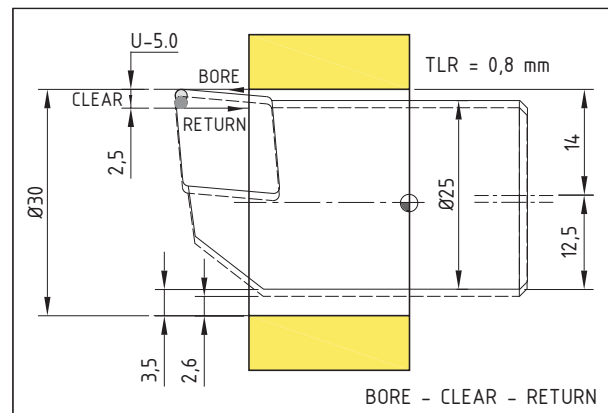


Figure 30-40

Retraction from a bore using sufficient clearance on diameter

Only a single boring cut is shown. The correct program for the bore is listed next:

```

(G21) ...
N21 T0700
N22 G96 S200 M03
N23 G41 X25.0 Z3.0 T0707 M08 (START POS.)
N24 G01 Z-30.0 F0.25 (NOT SHOWN ON DWG)
N25 U-5.0 (==> CRITICAL RETRACT AMOUNT)
N26 G00 Z3.0 (OUT OF HOLE FIRST)
N27 G40 X200.0 Z50.0 T0700 (THEN CLEAR+CANCEL)
N28 M01

```

The critical block is N25 - this is where most errors happen. Many programmers many think that 5 mm clearance is too large and the boring bar may hit the opposite part of the hole. First, 5 mm is on *diameter*, so it is only 2.5 mm per side. Second, into this 2.5 mm clearance a radius of 0.8 mm must fit - *twice*! That takes away another 1.6 mm from 2.5 mm, making the actual amount of tool motion only 0.9 mm (0.035"). The reason is that the tool radius has to be to the left of *all motions*, not just for the actual bore.

### ◆ Change of Motion Direction

The second common problem is when facing in tool nose radius offset mode. On CNC lathes, a change in cutting direction is used much more often than on machining centers.

The following example shows a facing cut on a solid face with G41 in effect, changing to a turning cut(-s) with G42 in effect - see *Figure 30-41*. Possible problem is discussed as well.

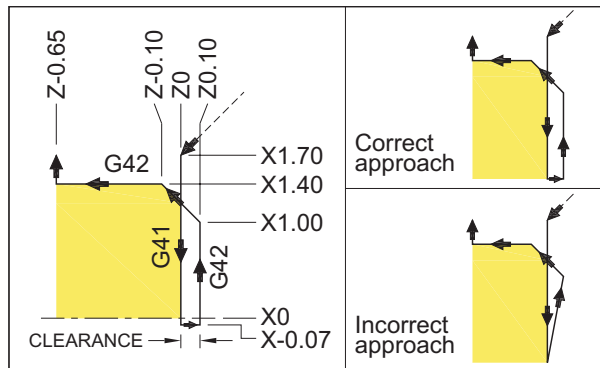


Figure 30-41

Tool nose radius offset change in facing for the same tool

```

...
N21 T0100                                (CORRECT APPROACH)
N22 G96 S400 M03
N23 G00 G41 X1.7 Z0 T0101 M08            (START)
N24 G01 X-0.07 F0.007                    (FACE OFF)
N25 G00 Z0.1                             (ONE AXIS ONLY)
N26 G42 X1.0                             (THEN COMPENSATION)
N27 G01 X1.4 Z-0.1 F0.012                (CONTOURING)
N28 Z-0.65
N29 X..

```

Face cutting is a single axis motion and the radius offset is used for consistency or it may be in effect for another reason. For solid parts, the face cut must end *below* the center line - X-0.07 in block N24 - at a diameter marginally larger than double tool radius. If the cut finishes at X0, the tool leaves a small unfinished tip at the center line and the face will not be flat. Also compare the *correct* and *incorrect* tool motions on the right side of the last illustration. If the above program is modified to the following version,

```

...
N21 T0100                                (INCORRECT APPROACH)
N22 G96 S400 M03
N23 G00 G41 X1.7 Z0 T0101 M08            (START)
N24 G01 X-0.07 F0.007                    (FACE OFF)
N25 G00 G42 X1.0 Z0.1                    (** WRONG **)
N26 G01 X1.4 Z-0.1 F0.012                (CONTOURING)
N27 Z-0.65
N28 X..

```

... the face will never be completed! Think about it.

## 31

## PLANE SELECTION

From all available machining operations, *contouring* or *profiling* is the single most common CNC application, perhaps along with hole making. During contouring, the tool motion is programmed in at least three different ways:

- Tool motion along a single axis only
- Tool motion along two axes simultaneously
- Tool motion along three axes simultaneously

There are additional axis motions that can also be applied (the *fourth* and *fifth* axis, for example), but on a CNC machining center, we *always* work with at least *three* axes, although *not always* simultaneously. This reflects the three dimensional reality of our world.

This chapter applies only to CNC milling systems, since turning systems normally use only two axes, and planes are therefore not required or used. Live tooling on CNC lathes does not enter this subject at this point.

Any absolute point in the program is defined by three coordinates, specified along the X, Y and Z axes. Programmed rapid motion G00 or a linear motion G01 can use *any number* of axes simultaneously, as long as the resulting tool motion is safe within the work area. No special considerations are required, no special programming is needed.

That is not the case for the following three programming procedures, where various considerations change quite significantly:

- Circular motion using G02 or G03 command
- Cutter radius offset using G41 or G42 command
- Fixed cycles using G81 to G89 commands, or G73, G74 and G76 commands

In all three cases - *and only in these three cases* - programmer has to consider a special setting of the control system - it is called a *selection of the machining plane*.

## WHAT IS A PLANE?

To look up a definition of a plane, research a standard textbook of mathematics or even a dictionary. From various definitions, plane can be described in one sentence:

A plane is a surface in which a straight line joining any two of its points will completely lie on that surface

Planes in the mathematical sense have their own properties. There is no need to know them all, but there are important properties relating to planes that are useful in CNC programming and in various phases of CAD/CAM work:

- Any three points that do not lie on a single line define a plane (these points are called *non-collinear* points)
- Plane is defined by two lines that intersect each other
- Plane is defined by two lines that are parallel to each other
- Plane is defined by a single line and a point that does not lie on that line
- Plane can be defined by an arc or a circle
- Two intersecting planes define a straight line
- Straight line that intersect a plane on which it does not lie, defines a point

These mathematical definitions are only included for reference and as a source of additional information. They are *not* required for everyday CNC programming.

## MACHINING IN PLANES

Path of a cutting tool is a combination of lines and arcs. A tool motion in one or two axes always takes place in a plane designated by two axes. This type of motion is a *two-dimensional* motion. In contrast, any tool motion that takes place in three axes at the same time is a *three-dimensional* motion.

### ◆ Mathematical Planes

In CNC machining, the only planes that can be defined and used are planes consisting of a combination of *any two* primary axes XYZ. Therefore, the *circular cutting motion*, *cutter radius offset* and *fixed cycles* can take place only in any one of three available planes:

XY plane	ZX plane	YZ plane
----------	----------	----------

The actual *order of axis designation* for a plane definition is very important. For example, the XY plane and the YX plane are *physically* the same plane. However, for the purposes of defining a relative tool motion direction (*clockwise* vs. *counterclockwise* or *left* vs. *right*), a clear standard must be established.

This international standard is based on the mathematical rule that specifies the *first* letter of the plane designation always refers to the *horizontal* axis and the *second* letter refers to the *vertical* axis when the plane is viewed. Both axes are always *orthogonal* (horizontal and vertical only) and *perpendicular* (at 90°) to each other. In CAD/CAM work, this standard defines the difference between the top and bottom, front and back, etc.

A simple way to remember *mathematical* designation of axes for all three planes is to write the alphabetical order of all three axes *twice* and isolate each pair with a space:

XYZXYZ ... becomes ... XY ZX YZ

In *mathematical* terms, the planes are defined as:

Plane	Horizontal Axis	Vertical Axis
XY	X	Y
ZX	Z	X
YZ	Y	Z

Note the emphasis on the word '*mathematical*'. The emphasis is intentional, and for a very good reason. As will soon be apparent, there is a great difference between the mathematical planes and the machine planes, as defined by viewing direction of the machine.

### ◆ Machine Tool Planes

A typical CNC machining center has three axes. Any two axes form a plane. A machine plane may be defined by looking at the machine from standard operating position. For a vertical machining center, there are three standard views, viewed perpendicularly (straight on):

- Top view ... XY plane
- Front view ... XZ plane
- Right side view ... YZ plane

Illustrations in *Figure 31-1* show the difference between the two definitions, caused by viewpoints that are *not* compatible.

It is clear that the XY plane and top view are the same in both definitions, and so is the YZ plane and the right side view. The ZX mathematical plane is *different* from the front plane on the machine, which is XZ, as shown in the middle illustration.

Mathematical plane defined as the ZX plane, where Z is the horizontal axis, is *reversed* on the machine plane for CNC machining centers. On the machine, this plane becomes the XZ plane, where the *X-axis* is the horizontal axis - a very important distinction.

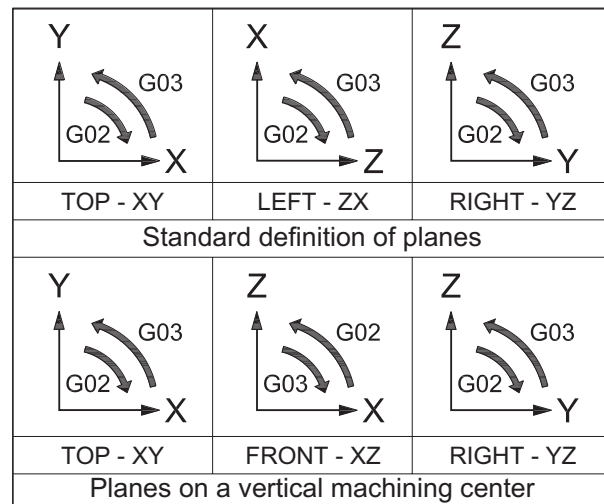


Figure 31-1

Comparison of standard mathematical planes (above), and planes as viewed on a CNC machining center (below)

In programming, the selection of planes is extremely important, yet often neglected and even misunderstood by programmers and operators alike. The main reason is that the majority of tool motions (particularly for contouring) are programmed and machined in the standard XY plane. On all CNC machining centers, the spindle is always perpendicular to the XY plane. Vertical and horizontal applications are exactly the same in this respect.

### ◆ Program Commands for Planes Definition

Selection of a plane for Fanuc and related controls adheres to the *mathematical* designation of planes, *not* the actual CNC machine tool planes. In a part program, each of the three *mathematical* planes can be selected by a special preparatory command - a unique G code:

G17	XY plane selection
G18	ZX plane selection
G19	YZ plane selection

For *all* rapid motions (programmed with G00) and all linear motions (programmed with G01), the plane selection command is totally irrelevant and even redundant. That is *not* the case for other motion modes, where the plane selection in a program is *extremely* important and must be considered carefully.

For machining applications using circular interpolation mode, with G02 or G03 commands (including helical motion), cutter radius offset with G41 or G42 commands and fixed cycles with G81 to G89 commands, as well as G73, G74, G76, the plane selection is very critical.

### ◆ Default Control Status

If a plane is not selected by the program, control defaults automatically to G17 - XY plane - in milling and G18 - ZX plane - in turning. If plane selection G-code is used, it should be included at the program beginning (top). Since the three plane commands only have affect on *circular motions, cutter radius offsets and fixed cycles*, the plane selection command G17, G18 or G19 can be programmed *before* any of these machining motions take place.

Always program the appropriate plane selection command.  
Never rely on the control settings !

Any plane selection change is programmed as desired, prior to actual toolpath change. Plane can be changed as often as necessary in a program, but only one plane can be active at any time. Selection of one plane cancels any other plane, so each of G17/G18/G19 commands cancels the other. Although true in an informative sense, it is most likely that any opportunities to mix all three plane commands in a single program are remote. From all three available motions, only the circular motion is affected by plane selection, but let's have look at the programming rapid and linear motions as well, at least for comparison purposes.

## STRAIGHT MOTION IN PLANES

Both rapid motions G00 and linear motions G01 are considered straight motions when compared with circular motions. Straight motions can be programmed for a single axis or as a simultaneous motion along two or three axes. The following examples only show typical unrelated blocks:

### ➡ Example - Rapid positioning - G00 :

G00 X5.0 Y3.0	<i>XY plane - 2D rapid motion</i>
G00 X7.5 Z-1.5	<i>XZ plane - 2D rapid motion</i>
G00 Y10.0 Z-0.25	<i>YZ plane - 2D rapid motion</i>
G00 X2.0 Y4.0 Z-0.75	<i>XYZ - 3D rapid motion</i>

### ➡ Example - Linear interpolation - G01 :

G01 X-1.5 Y4.46 F15.0	<i>XY plane - 2D linear motion</i>
G01 X8.875 Z-0.84 F10.0	<i>ZX plane - 2D linear motion</i>
G01 Y12.34 Z0.1 F12.5	<i>YZ plane - 2D linear motion</i>
G01 X6.0 Y13.0 Z-1.24 F12.0	<i>XYZ - 3D linear motion</i>

The examples refer to tool motion along the programmed axes. Plane selection command does not need to be used for any straight motion (along a single axis), unless the cutter radius offset or a fixed cycle is in effect. All tool motions will be interpreted correctly by the control, regardless of any plane in effect. The rules that apply to linear motions are not the same for circular motions.

## CIRCULAR INTERPOLATION IN PLANES

In order to complete a circular motion correctly, control system has to receive sufficient information from the part program. Unlike rapid positioning with G00 in effect or linear interpolation with G01 in effect, circular interpolation requires a programmed *direction* of motion. G02 is the command for CW arc direction and G03 is the command for CCW arc direction. According to general mathematical rules, *clockwise* direction is always viewed from the vertical axis towards the horizontal axis in any selected plane. *Counterclockwise* direction is always viewed from the horizontal axis towards the vertical axis.

When we compare the mathematical axes designation and the actual orientation of machine axes (based on a vertical machining center), the XY plane (G17) and the YZ plane (G19) correspond to each other. These two planes normally present no problems to CNC programmers. The ZX plane (G18) may cause a serious problem if not properly understood. Mathematically, the horizontal axis in G18 plane is the Z-axis and the X-axis is the vertical axis. On a vertical machining center, the order of machine axes orientation is *reversed*. It is important to understand that the clockwise and counterclockwise directions *only appear* to be reversed, but in reality, they are the same.

If the mathematical axes orientation is aligned with the machine axes, they *will* indeed match. *Figure 31-2* shows steps of aligning mathematical planes with machine planes:

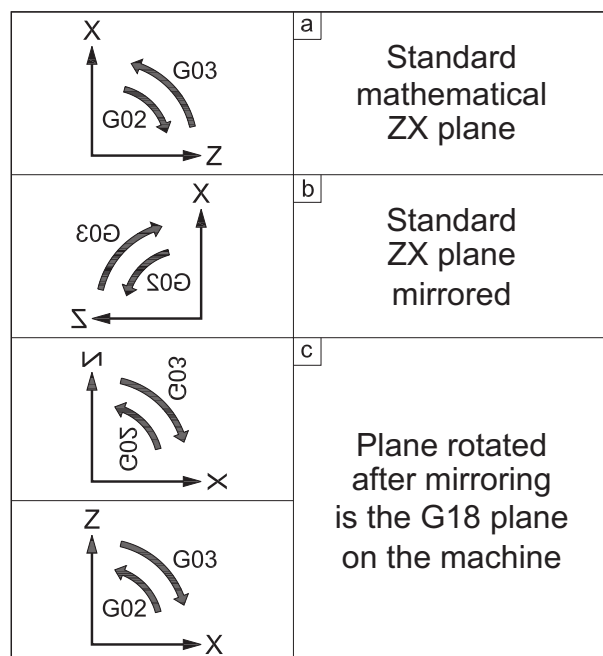


Figure 31-2

Progressive steps in aligning mathematical ZX plane with machine XZ plane, using G18 plane selection



Note that the G-code direction for arcs *does not* change either within the mathematical plane (a), or the mathematical plane mirrored (b), or even the mirrored plane rotated by negative 90° (c), even if the plane itself is changed. What occurred here is *not* a creation of any new plane (mathematical or otherwise). The view still represents a three dimensional object, viewed from a different directional viewpoint - from behind the machine and rotated.

On horizontal machining centers, the situation is similar. Both XY plane (G17) and ZX plane (G18) match between mathematical designation and the actual axes orientation. It is the G19 plane (YZ) that *appears* to be reversed and may cause some problems before the logical structure of the planes is well understood.

Correct selection of a machining plane will enable programming various contouring operations using circular and helical interpolation, cutter radius offset and fixed cycles, including use of right angle spindle accessories (heads). The most common applications of this type of machining are filleted (blend) radii, intersecting radii, circular pockets, profiled counterbores, cylinders, simple spheres and cones, and other similar shapes.

In order to understand the CNC applications of G02 and G03 commands in planes, the illustration in *Figure 31-3* should be helpful.

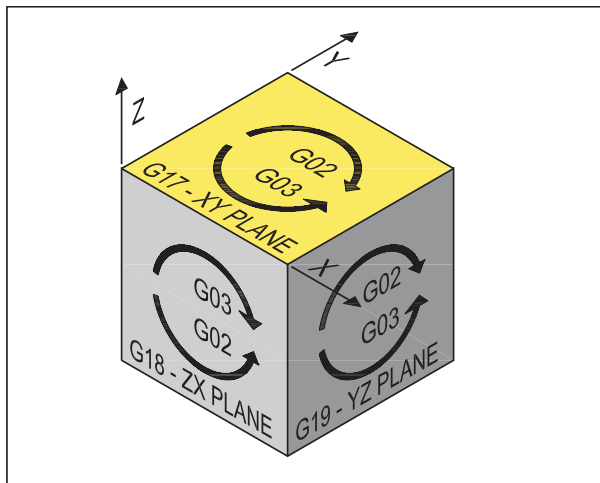


Figure 31-3

Actual circular tool path direction in all three machine planes.  
Note the apparent inconsistency for G18 plane

#### ◆ G17-G18-G19 as Modal Commands

The preparatory commands for a plane selection G17, G18 and G19 are all *modal* commands - programming any one of them will activate the selected plane only. Plane selection in the program will be in effect until canceled by another plane selection. The three plane related G-codes belong to the G code group number 02 exclusively.

The following format examples show some typical programming applications for circular interpolation:

G17 G02 X14.4 Y6.8 R1.4

G18 G03 X11.575 Z-1.22 R1.0

G19 G02 Y4.5 Z0 R0.85

Some older control systems do not accept the direct radius designation specified by the R-address. Instead, the arc center vectors I, J and K must be used. For programming circular motion within a selected plane, correct pair of arc center vectors IJK must be selected:

G17 G02 (G03) X.. Y.. I.. J..

G18 G02 (G03) X.. Z.. I.. K..

G19 G02 (G03) Y.. Z.. J.. K..

From previous topics, remember that ...

- XY axes - G17 plane - I and J arc center vectors
- XZ axes - G18 plane - I and K arc center vectors
- YZ axes - G19 plane - J and K arc center vectors

#### ◆ Absence of Axis Data in a Block

The programming format described here contains *complete data* for the end point of a circular motion. In practice, however, an experienced programmer does not repeat any modal values from one block to another. The major reason for this approach is saving programming time, shortening program length and increasing available memory space in the control system.

Portion of the following program example shows a typical application in a program where modal axes values are *not repeated* in subsequent blocks:

N.. G20	Imperial units
...	
N40 G17	XY plane selected
N41 G00 X20.0 Y7.5 Z-3.0	Start position of the tool
N42 G01 X13.0 F10.0	Plane selection irrelevant
N43 G18 G02 X7.0 R3.0	Z axis is assumed as absent
N44 G17 G01 X0	Plane selection irrelevant

Block N43 represents a contour of a 180° arc in the ZX plane. Because of the G18 command in N43, the control will correctly interpret the 'missing' axis as the Z-axis, and its value will be equal to the last Z-axis value programmed (Z-3.0). Also examine the G17 command in block N44. It is always a good practice to transfer the control status to its original plane selection as soon as the plane changes, although this is not absolutely necessary in the example.

Omitting the G18 command in block N43 will cause a serious program error. If G18 is omitted, the previously selected command G17 will still be in effect and circular interpolation will take place in the XY plane, instead of the intended ZX plane.

In this case, the axis assumed as ‘missing’ in the G17 plane will be the Y-axis and its programmed value of Y7.5. Control system will process such a block as if it were specified in a complete block:

```
N43 G17 G02 X7.0 Y7.5 R3.0
```

An interesting situation will develop if the plane selection command G18 in block N43 is absent, but the circular interpolation block contains *two axes* coordinates for the end point of the circular motion:

```
N43 G02 X7.0 Z-3.0 R3.0    G17 is still in effect
```

Although G17 is still the active plane, the specified arc will be machined correctly in G18 plane, even if G18 had not been programmed. This is because of the special control feature called *complete instruction* or *complete data priority*, provided in block N43 of the last example. The inclusion of two axes for the end point of circular motion has a higher priority rating than a plane selection command itself. A complete block is one that includes all necessary addresses without taking on modal values.

Two axes programmed in a single block override the active plane selection command

### ◆ Cutter Radius Offset in Planes

Plane selection for rapid or linear motions is irrelevant, providing that no cutter radius offset G41 or G42 is in effect. In theory, it means that regardless of the plane selection, all G00 and G01 motions will be correct. That is true, but seldom practical, since most CNC programs *do* use a contouring motion and they also use the cutter radius offset feature. As an example, evaluate the following blocks:

```
N1 G21
...
N120 G90 G00 X50.0 Y100.0 Z20.0
N121 G01 X90.0 Y140.0 Z0 F180.0
```

When the rapid motion programmed in block N120 is completed, the cutter will be positioned at the absolute location of X50.0 Y100.0 Z20.0. Absolute location of the cutting motion will be X90.0 Y140.0 Z0, after the block N121 is completed.

Adding a cutter radius offset command G41 or G42 to the rapid motion block, the plane selection will become extremely important. Radius offset will be effective only for those two axes selected by a plane selection command.

There will *not* be a 3-axis cutter radius offset taking place! In the next example, compare the absolute tool positions for each plane when the rapid motion is completed and cutter radius offset is activated in the program. Tool absolute position when the cutting motion is completed depends on the motion following block N121.

Radius offset amount of D25=100.000 mm, stored in the control offset registry, is used for the next example:

#### ➡ Example :

```
N120 G90 G00 G41 X50.0 Y100.0 Z20.0 D25
N121 G01 X90.0 Y140.0 Z0 F180.0
```

The compensated tool position when block N120 is completed, will depend on the plane G17, G18 or G19 currently in effect:

- If G17 command is programmed with three axes :

```
G17 X.. Y.. Z..    XY motion is compensated
```

- If G18 command is programmed with three axes :

```
G18 X.. Y.. Z..    ZX motion is compensated
```

- If G19 command is programmed with three axes :

```
G19 X.. Y.. Z..    YZ motion is compensated
```

The following practical programming example illustrates both circular interpolation and cutter radius offset as they are applied in different planes.

## PRACTICAL EXAMPLE

The example illustrated in *Figure 31-4* is a simple job that requires cutting the R0.75 arc in XZ plane. Typically, a ball nose end mill (also known as a spherical end mill) will be used for a job like this.

In the much simplified example, only two main tool passes are programmed. One pass is the left-to-right motion - across the left plane, over the cylinder, and over the right plane. The other pass is from right to left - across the right plane, over the cylinder, and across the left plane. A stepover for the tool is also programmed, between the two passes. Program of this type for the whole part could be done in the incremental mode and would greatly benefit from the use of subprograms.

*Figure 31-5* demonstrates tool motion for the two passes included in the program example. To interpret program data correctly, note that program zero is at the *bottom left corner* of the part. Both clearances off the part are 0.100 and the stepover is 0.050:

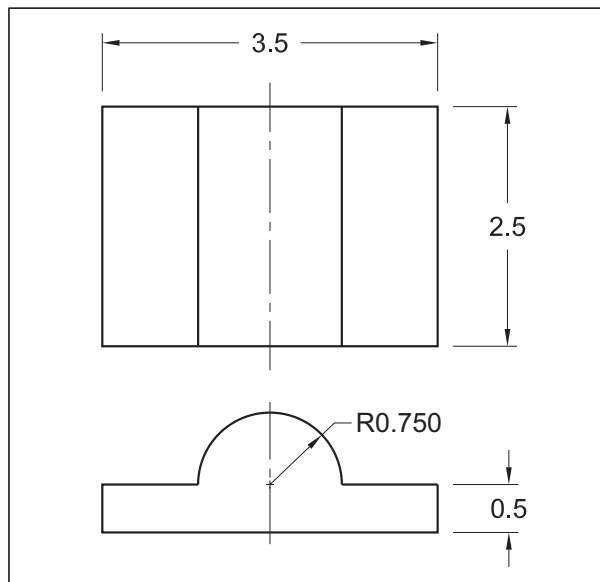


Figure 31-4

Drawing for the programming example O3101

```

O3101
N1 G20
N2 G18 (ZX PLANE SELECTED)
N3 G90 G54 G00 X-0.1 Y0 S600 M03
N4 G43 Z2.0 H01 M08
N5 G01 G42 Z0.5 D01 F8.0
N6 X1.0
N7 G03 X2.5 I0.75 (= G03 X2.5 Z0.5 I0.75 K0)
N8 G01 X3.6
N9 G91 G41 Y0.05
N10 G90 X2.5
N11 G02 X1.0 I-0.75 (= G02 X1.0 Z0.5 I-0.75 K0)
N12 G01 X-0.1
N13 G91 G42 Y0.05
N14 G90 ...

```

When working with this type of CNC program the first time, it may be a good idea to test the toolpath in the air, a little above the job. Errors can happen quite easily.

Three axes cutting motion is programmed manually only for parts where calculations are not too time consuming. For parts requiring complex motions calculations, a computer programming software is a better choice.

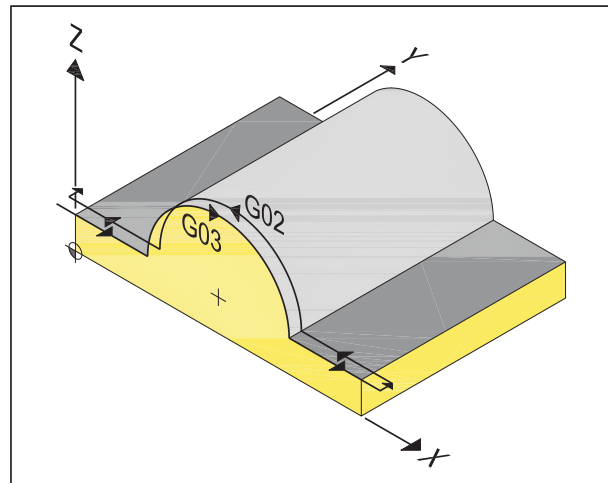


Figure 31-5

Tool path for programming example O3101

## FIXED CYCLES IN PLANES

The last programming item relating to plane selection is the application of planes in fixed cycles. For cycles in the G17 plane (XY hole locations), G17 is only important if a switch from one plane to another is contained in the same program. With special machine attachments, such as *right angle heads*, the drill or other tool is positioned *perpendicular* to the normal spindle axis, being in G18 or G19 plane.

Although the right angle heads are not very common, in many industries they are gaining in popularity. When programming these attachments, always consider the tool direction into the work (the *depth* direction). In the common applications of fixed cycles, G17 plane uses XY axes for the hole center location and the Z-axis for the depth direction. If the angle head is set to use the Y-axis as the depth direction, use G18 plane and the XZ axes will be the hole center positions. If the angle head is set to use the X-axis as the depth direction, use G19 plane and the YZ axes will be the hole center positions. In all cases, the R-level always applies to the axis that moves along the depth direction.

The difference between tool tip and centerline of spindle is the actual overhang. This extra overhang length must be known and incorporated into all motions of the affected axis not only for correct depths, but also for safety.

Even with the ever increasing use of carbide cutters for metal removal, the traditional HSS (high-speed steel) end mills still enjoy a great popularity for a variety of milling operations and even on lathes. These venerable cutters offer several benefits - they are relatively inexpensive, easy to find, and do many jobs quite well. The term *high speed steel* does not suggest much productivity improvement in modern machining, particularly when compared to carbide cutters. It was used long time ago to emphasize the benefit of this tool material to carbon tool steel. The new material of the day was a tool steel enhanced with tungsten and molybdenum (*i.e.*, hardening elements), and could use spindle speeds two to three times faster than carbon steel tools. The term high-speed-steel was coined and the HSS abbreviation has become common to this day.

The relatively low cost of high speed steel tools and their capability to machine a part to very close tolerances make them a primary choice for many milling applications. End mills are probably the single most versatile rotary tool used on a CNC machine.

Solid carbide end mills and end mills with replaceable carbide spiral flutes or inserts are frequently used for many different jobs and are on the rise. Most typical are jobs requiring a high metal removal rates and when machining hard materials. HSS end mill is still a common cutting tool choice for everyday machining.

Many machining applications call for a harder tooling material than a high speed steel, but not as hard as carbide. As tooling costs become an issue, the frequent solution is to employ an end mill with additional hardeners, for example a cobalt end mill. Such a tool is a little more expensive than a high speed steel tool, but far less expensive than a carbide tool. Cobalt based end mills have longer cutting tool life and can be used the same way as a standard end mill, with a noticeably higher productivity rate.

Solid carbide end mills are also available in machine shops and commonly used as regular small tools. Larger tools made of solid carbide would be too expensive, so special end mills with indexable inserts are the tools of choice. They can be used for both roughing operations and precision finishing work.

This chapter takes a look at some technological considerations when the CNC program calls for an end mill of any type or for a similar tool that is used as a profiling tool for peripheral cutting and contouring. This is an operation when the side of the cutter does most of work.

## END MILLS

End mills are the most common tools used for peripheral milling. There is a wide selection of end mills available for just about any conceivable machining application. Traditional end mills come in metric and imperial sizes, variety of diameters, styles, number of cutting flutes, numerous flute designs, special corner designs, shanks, and tool material compositions.

Here are some of the most common machining operations that can be performed with an end mill - HSS, cobalt, solid carbide or an indexable insert type:

- Contouring and peripheral end milling
- Milling of slots and keyways
- Channel grooves, face grooves and recesses
- Open and closed pockets
- Facing operations for small areas
- Facing operations for thin walls
- Counterboring
- Spotfacing
- Chamfering
- Deburring

End mills can be formed by grinding them into required shapes. The most common shapes are the *flat* bottom end mill (the most common type in machine shops), an end mill with a full radius (often called a *spherical* or a *ball nose* end mill), and an end mill with a corner radius (often called the *bull nose* end mill).

Each type of an end mill is used for a specific type of machining. Standard *flat end mill* is used for all operations that require a flat bottom and a sharp corner between the part wall and bottom. A *ball nose end mill* is used for simultaneous three dimensional (3D) machining on various surfaces. An end mill similar to a ball nose type is the *bull nose end mill*, used for either some 3D work, or for flat surfaces that require a corner radius between the part wall and bottom. Other shapes are also required for some special machining, for example, a center cutting end mill (called a slot drill), or a taper ball nose end mill.

Figure 32-1 shows the three most common types of end mills used in industry and the relationship of cutter radius to the cutter diameter.

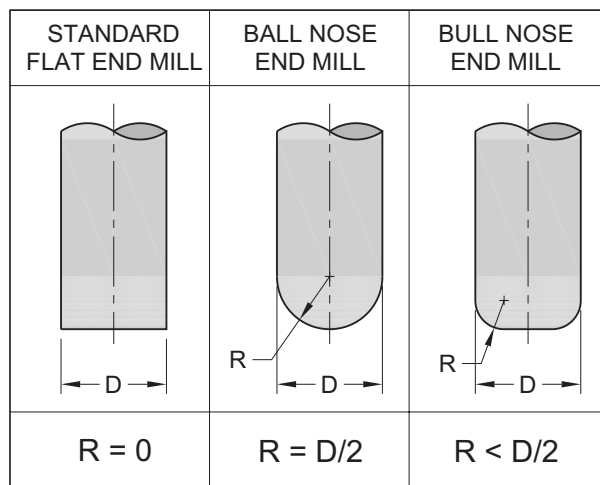


Figure 32-1

Basic configuration of the three most typical end mills

### ◆ High Speed Steel End Mills

True high speed steel end mills are the ‘old-timers’ in machine shops. They are manufactured either as a single end or a double end design, with various diameters, lengths and shank configurations. Depending on the cutting tip geometry, they can be used for peripheral motion (XY axes only), plunge motion (Z-axis only), or all axes simultaneously (XYZ axes). Either a single end or a double end can be used for CNC machining. When using a double end mill, make sure the unused end is not damaged in the tool holder, when mounted. On a CNC machine, all end mills are normally held in a collet type tool holder, providing the maximum grip and concentricity. Chuck type holders are not recommended for end mills of any kind.

### ◆ Solid Carbide End Mills

In essence, the solid carbide end mills have the same characteristics as HSS types and vary only in the type of material they are made of and some geometry adjustments. Using solid carbide end mill requires special machining circumstances. The tool itself is fairly expensive, and from metallurgical point of view, solid carbide is a brittle material that chips easily, particularly at sharp corners, or when it is dropped or improperly stored. When handled properly, it can remove metal with a great efficiency and produce superior surface finishes.

### ◆ Indexable Insert End Mills

Indexable insert end mills provide all the benefits of solid carbide end mills, but with the added convenience of replaceable carbide inserts. Many designs are available in this category as well. The holders for these tools match their internal diameter to the tool diameter. The tool has a ground flat area where the holder mounting screw prevents the tool from spinning.

### ◆ Relief Angles

It is always important to select proper tool relief angle for different materials being cut. Relief angle is also called the *clearance angle*. For HSS milling cutters, the recommended flute relief angles become larger for softer materials. For example, the primary relief angle for steel is  $3^\circ$  to  $5^\circ$ , whereby the primary angle for aluminum is  $10^\circ$  to  $12^\circ$ . Cutting tool representative will supply additional information for a particular machining application.

### ◆ End Mill Size

Three very important criteria relating to the size of an end mill have to be considered for CNC machining:

- End mill diameter
- End mill length
- Flute length

For CNC work, the end mill diameter must be very accurate. Nominal diameters are those that are listed in catalogues of various tooling companies. Non-standard size, such as reground cutters, must be treated differently for CNC work. Even with the benefits of cutter radius offset, it is not advisable to use reground end mills for precision machining, although they may do a very good job for emergency situations and for some roughing. That does not mean a reground cutter cannot be used for non-CNC work elsewhere in the shop or for less demanding CNC work.

Length of an end mill projected from the tool holder is also very important. A long projection may cause chatter that contributes to the wear of cutting edges. Another possible side effect for a long tool is deflection. Deflection will negatively influence size and surface finish quality of the finished part. Flute length is important for determination of the depth of cut.

Regardless of overall tool length (tool projection length from the spindle), the length of flutes determines the cutting depth. Figure 32-2 shows a useful ratio of width vs. depth of a rough side cut in milling:

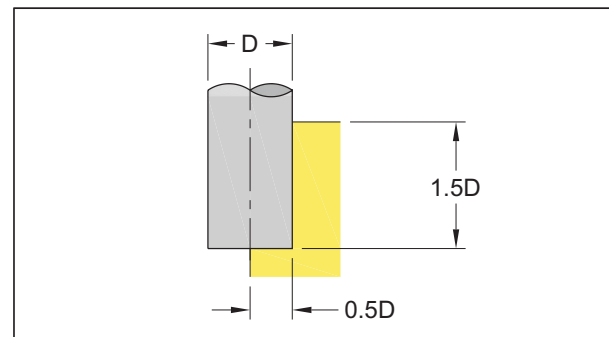


Figure 32-2

Relationship of the end mill diameter to the depth of cut for rough side cuts in milling



### ◆ Number of Flutes

When selecting an end mill, particularly for material of average hardness, the number of flutes should be the primary consideration. For profiling, many programmers select (virtually automatically) a four-flute end mill for any required tool size larger than Ø0.625 or Ø0.750. An end mill that has to *plunge-in* - that is - it has to cut *into* a solid material along the Z-axis - has normally only two flutes, regardless of diameter. This 'plunging-type' of end mill is also known under a more technical name as a *center-cutting end mill*, or under a rather old-fashioned name, a *slot drill*. The term slot drill has no relation to the tool called a drill, but to its machining action - just like a drill, a slot drill penetrates into a solid material, parallel to the Z-axis.

It is the area of small and medium end mill diameters that requires the most attention. In this size range, end mills come in two-, three-, and four-flute configurations. So what are the benefits of a two-flute versus a three-flute versus a four-flute design, for example? *Type of material* is the guiding factor here.

In this area, there is the expected give and take situation or a trade off. On the positive side, the fewer flutes an end mill has, the better conditions exist to avoid a chip buildup between the flutes during heavy cuts. Simply, there is more room. On the negative side, the fewer flutes that work in material, the slower feedrate has to be programmed as each flute works harder. When cutting soft, nonferrous materials, such as aluminum, magnesium, even copper, preventing a chip buildup is important, so a two-flute end mill type is a traditional choice, even if the feedrate has to be somewhat compromised.

A different scenario is presented for harder materials, because two other factors have to be considered - *tool chatter* and *tool deflection*. There is no doubt, that in ferrous materials, multi flute end mills will deflect less and chatter less than their two-flute counterparts.

What about the three-flute end mills? They seem to be - and in fact they are - a reasonable compromise between the two-flute and four-flute types. Three-flute end mills have never become a standard choice, even if their machining capabilities are often very good to excellent. Machinists have a difficulty to measure their diameter accurately, particularly with common machine shop tools such as a vernier or a micrometer. However, they do work very well in most materials.

Regardless of the number of flutes, an end mill with a larger diameter will deflect less than a similar end mill with a small diameter. In addition, the end mill effective length (measured as its overhang from the holder face) is important. The longer is the tool, the greater is its deflection - and that applies to all tools. Deflection pushes the tool away from its axis (center line). These are all results of common physical laws.

## SPEEDS AND FEEDS

In many other sections of the handbook, speeds and feeds are mentioned. *Also, see Appendix B (page 523) for basic suggestions.* Tooling catalogues have very good charts and recommendations on speeds and feeds for particular tools, used with different materials. However, one standard formula (imperial version) is used for calculating the spindle speed in *r/min* (revolutions per minute):

$$r / \min = \frac{12 \times \text{ft} / \min}{\pi \times D}$$

where ...

- r/min* = Spindle speed (revolutions per minute)
- 12 = Constant to convert **feet to inches**
- ft/min* = Surface speed in feet per minute
- $\pi$  = Constant for flat to diameter conversion
- D = Diameter of the tool in inches

For metric system, the formula is similar:

$$r / \min = \frac{1000 \times \text{m} / \min}{\pi \times D}$$

where ...

- r/min* = Spindle speed (revolutions per minute)
- 1000 = Constant to convert **mm to meters**
- m/min* = Surface speed in meters per minute
- $\pi$  = Constant for flat to diameter conversion
- D = Diameter of the tool in millimeters

Sometimes, there may be a benefit from the reverse formula - for example, when cutting at a certain spindle speed (*r/min*) that seems to be the perfect choice for a particular material. Next time a different diameter of the tool for that same material is used, just find out the *ft/min* rating for the material, which is applicable to any cutter size. The next formula will do exactly that (tool diameter is in inches):

$$\text{ft} / \min = \frac{\pi \times D \times r / \min}{12}$$

Metric formula is similar, but the tool diameter is in millimeters (mm):

$$\text{m} / \min = \frac{\pi \times D \times r / \min}{1000}$$

All entries in the formulas are based on previous explanations and should be easy to understand and apply.

To calculate cutting feedrate for any milling operation, the spindle speed in *r/min* must be known first. Also known has to be the number of flutes and chip load on each flute (suggested chip load is usually found in tool catalogues or in *Appendix B*). For imperial units, the chip load is measured in *inches per tooth* (a tooth is the same as a flute or an insert), with the abbreviation of *in/tooth*. The result is the cutting feedrate that will be in inches per minute - *in/min*.

For a lathe feedrate using standard turning and boring tools, the number of flutes is not applicable - the result is directly specified in inches per revolution (*in/rev*) or millimeters per revolution *mm/rev*.

$$\text{in / min} = \text{r / min} \times f_t \times N$$

where ...

in/min = Feedrate in inches per minute  
 r/min = Spindle speed in revolutions per minute  
 $f_t$  = Chip load in inches per tooth (per flute)  
 N = Number of teeth (flutes)

For metric system of measurement, chipload is measured in *millimeters per tooth* (per flute), with the abbreviation of *mm/tooth*. Metric formula is similar to the one listed for imperial units:

$$\text{mm / min} = \text{r / min} \times f_t \times N$$

where ...

mm/min = Feedrate in millimeters per minute  
 r/min = Spindle speed in revolutions per minute  
 $f_t$  = Chip load in millimeters per tooth  
 N = Number of teeth (flutes)

As an example of the above formulas, a  $\varnothing 0.75$  four flute end mill may require 100 ft/min in cast iron. For the same cutting tool and part material, 0.004 per flute is the recommended chip load. Therefore, the two calculations will be:

*Spindle speed:*

$$\begin{aligned} \text{r/min} &= (12 \times 100) / (3.14 \times 0.75) \\ \text{r/min} &= 509 \end{aligned}$$

*Cutting feedrate:*

$$\begin{aligned} \text{in/min} &= 509 \times 0.004 \times 4 \\ \text{in/min} &= 8.1 \end{aligned}$$

For safety reasons, always consider the part and machine setup, their rigidity, depth and/or width of cut and other relevant conditions very carefully.

Feed per tooth  $f_t$  (in *inches per tooth*), can be calculated as reversed values from the formula listed above.

Imperial units version of the formula is:

$$f_t = \frac{\text{in / min}}{\text{r / min} \times N}$$

Metric units formula is very similar, it calculates the feed per tooth  $f_t$  in *mm/tooth*:

$$f_t = \frac{\text{mm / min}}{\text{r / min} \times N}$$

When using carbide insert end mills for cutting steels, faster spindle speeds are generally better choice. At slow speeds, carbide cutter is in contact with a steel being cold. As spindle speed increases, so does the steel temperature at the tool cutting edge, producing lower strength of the material. That results in favorable cutting conditions. Carbide insert cutting tools can often be used three times and up to five times faster than standard HSS cutters. Two following basic rules establishing the relationship of tool material and spindle speed can be summed up:

High speed steel (HSS) tools will wear out very quickly, if used at high spindle speeds = high r/min

Carbide insert cutters will chip or even break, if the spindle speed is too low = low r/min

### ◆ Coolants and Lubricants

Using a coolant with high speed steel (HSS) cutter is almost mandatory for cutting all metals. Coolant extends the tool life and its lubricating attributes contributes to much improved surface finish. On the other hand, for carbide insert cutters, coolant *may not* be always necessary, particularly for roughing steel stock.

Never apply coolant on a cutting edge that is already engaged in the material !

### ◆ Tool Chatter

There are many reasons why a chatter occurs during peripheral (contour) milling. Frequent causes of chatter are weak tool setup, excessive tool length (overhang from tool holder), machining thin walls of material with too much depth or too heavy feedrate, etc. Cutter deflection may also contribute to chatter. Tooling professionals agree that well planned experiments with the combination of spindle speeds and cutting feedrates should be the first step. If chatter still persists, look at the machining method used and the overall setup integrity.

## STOCK REMOVAL

Although contour milling is mainly a semifinishing and finishing machining operation, end mills are also successfully used for roughing. The flute configuration (flute geometry) and its cutting edge are different for roughing and finishing. A typical roughing end mill will have corrugated edges - a typical example is a *Strasmann end mill*, named after the original designer and developer of roughing cutters and the trademarked name is now used as a generic description of this type of roughing end mill.

Good machining practice for any stock removal is to use large diameter end mill cutters with a short overhang, in order to eliminate, or at least minimize, the tool chatter and tool deflection during heavy cuts.

For deep internal cavities, such as deep pockets, it is a good practice to pre-drill to the full depth (or at least to the *almost* full depth), then use this new hole for an end mill that is smaller than the drilled hole. Since the end mill penetrates to the depth in open space, succeeding cuts will be mainly side milling operations, enlarging the cavity into its required size, shape and depth.

### ◆ Plunge Infeed

Entering a plunge-type end mill into part material along the Z-axis only is called *center-cutting*, *plunging* or *plunge infeed*. It is a typical machining operation and programming procedure to enter into an otherwise inaccessible area, such as a deep pocket, closed slot, or any other solid material entry. Not every end mill is designed for plunge cutting and the CNC operator should always make sure the right end mill is selected (HSS or carbide or indexable insert type of end mill). Programmer can make it easier by placing appropriate comments in the program.

### ◆ In and Out Ramping

Ramping is another process where the Z-axis is used for penetrating (entering) into solid part material. This time, however, the X-axis *or* the Y-axis are programmed simultaneously with the Z-axis. Depending on the end mill diameter, typical ramping angle is about 25° for a 1.0 inch cutter, 8° for a 2.0 inch cutter, and 3° for a 4.0 inch cutter. Ramping approach toward the part can be used for flat type, ball nose type, and bull nose type of end mills. Smaller end mills will use smaller angles (3°-10°). See Figure 32-3 for an illustration of a typical ramping motion.

Always be very careful from which XYZ tool position the cutting tool will start cutting at the top of part. Considering only the start and end points may not produce the best results. It is easy to have a good start and good end tool positions, but somewhere during the cut, an unwanted section of the part may be removed accidentally. A few simple calculations or a CAD system may help here.

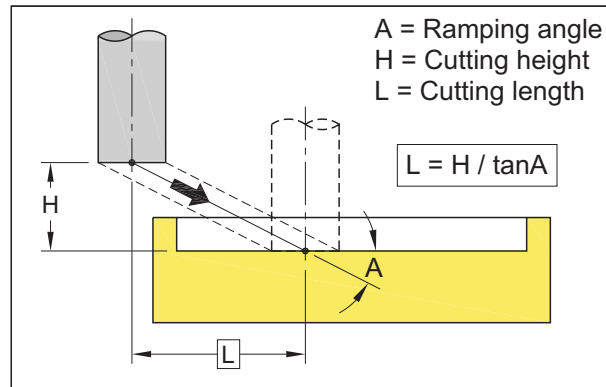


Figure 32-3

Typical entry angle for a ramping infeed into a solid material

### ◆ Direction of Cut

Direction of a cut for contouring operations is controlled by the programmer. Cutting direction of the end mill for peripheral milling will make a difference for most part materials, mainly in the area of material removal and the quality of surface finish. From basic concepts of machining, the cutting direction can be in two modes:

- Climb milling - also known as *DOWN* milling
- Conventional milling - also known as *UP* milling

Anytime the G41 command is programmed, cutter radius is offset to the left of part and the tool is *climb* milling. That assumes, of course, that the spindle rotation is normal, programmed with the M03 function, and the cutting tool is right hand. The opposite, G42 offset, to the right of the part, will result in *conventional* milling. In most cases, climb milling mode is the preferred mode for contour milling, particularly in finishing operations.

Figure 32-4 illustrates the two cutting directions.

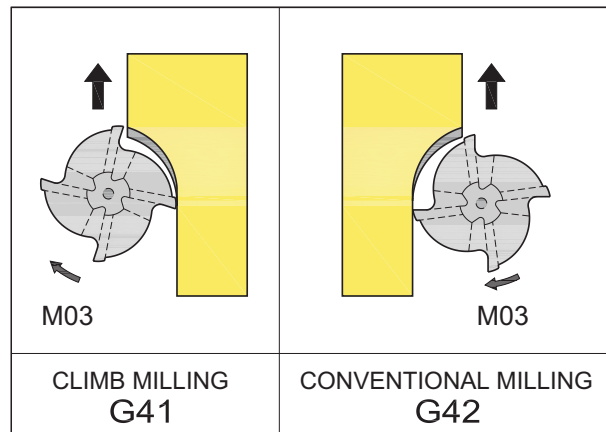


Figure 32-4

Direction of the cut relative to material, with M03 in effect

### Climb Milling

*Climb milling* - also called *down milling* - uses cutter rotation *in the feeding direction* and has the tendency to push the part *against* the table or fixture. Maximum chip thickness occurs at the beginning of the cut, and upon exit, the chip is very thin. Practical result is that most of the generated heat is absorbed by the chip, and hardening of the part is largely prevented.

Do not misunderstand the words **climb** and **down** describing the same machining direction

Both terms are correct, if taken in the proper context.

### Conventional Milling

*Conventional milling* - also called *up milling* - uses cutter rotation *against the feeding direction*, and has the tendency to pull the part *from* table or fixture. Maximum chip thickness occurs at the end of the cut, and upon exit, the chip is very thick. Practical result is possible hardening of the part, rubbing the tool into material, and a poor surface finish.

### ◆ Width and Depth of Cut

For good machining, the width and depth of cut should correspond to the existing machining conditions, namely

setup, type of material being machined and cutting tool used. Width of cut depends also on the number of flutes of the cutter that are actually engaged in the cut.

Approximately one third of diameter for the depth of cut is a good rule of thumb for small end mills, a little more for larger end mills.

Peripheral milling requires solid machining knowledge and certain amount of common sense. If a successful machining operation in one job is documented, it can be adapted to another job with ease.

## CORNER RADIUS CALCULATION

One of the most common mathematical calculations in contouring are tangency points of a blend (fillet) radius at a corner where an angular line intersects either a horizontal or a vertical line. As most data are generally provided, two situations can be covered in simple formulas - *Figure 32-5*.

Depending on drawing dimensions, sometimes the position of point *P* has to be calculated separately. This machine independent formula can be used equally for milling, turning and other contour applications, just make sure to fill-in the angle *A* amount that matches the angle orientation as shown.

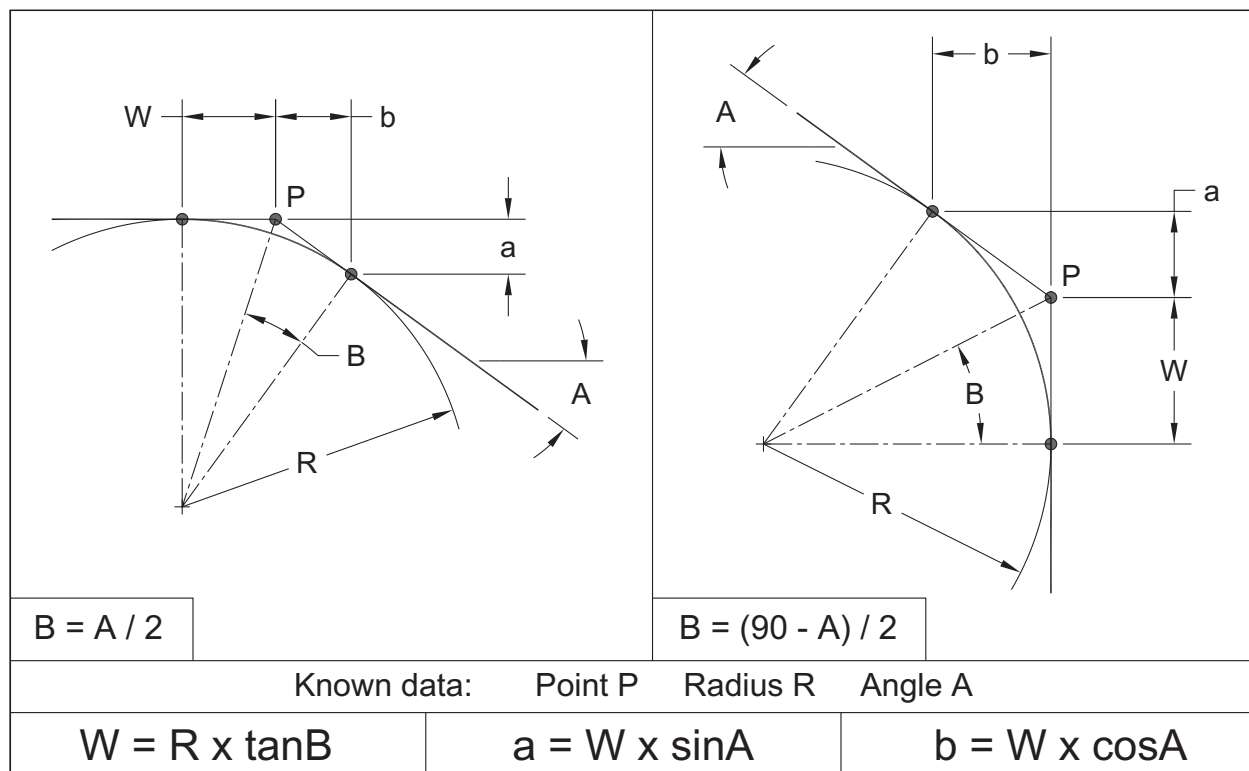


Figure 32-5

Formula to calculate points of tangency between an angle and a horizontal line (left), and an angle and a vertical line (right)

## 33

## SLOTS AND POCKETS

In many applications for a CNC machining center, the material has to be removed from the inside of a certain area, bounded by a contour and a flat bottom. This process is generally known as *pocketing*. To have a true pocket, the contour that defines the pocket boundary must be closed. However, there are many other applications, where the material has to be removed from an open area, with only a partial contour defined. An open slot is a good example of this type. This chapter looks at applications of closed pockets, partial pockets, slots and various programming techniques for internal material removal.

### OPEN AND CLOSED BOUNDARY

A continuous contour on which the start point and the end point are in different locations, is called an *open contour*. Continuous contour defined in the program that starts and ends at the same point location, is called a *closed contour*. From machining point of view, the major difference between an open and closed contour is *how the cutting tool reaches the contour depth*.

#### ◆ Open Boundary

An open boundary is not a true pocket, but belongs to a related category. Machining of this kind of a contour is quite flexible, as the tool can reach the required depth in an open space. Any good quality end mill in different varieties can be used to machine an open boundary.

#### ◆ Closed Boundary

Excessive material within a closed boundary can be removed in two ways, depending on the cutting operation. One way is to use an external tool and move it towards the outside of the boundary, another way is to use an internal tool and move it towards the inside of the boundary. In both cases, the actual machining follows. Cutting along the outside of a part is not considered pocketing but peripheral or contour milling (*page 285*). Cutting on the inside of a closed boundary is typical for machining pockets of various regular and irregular shapes. Some typical examples of regular shape pockets are closed slots, rectangular pockets, circular pockets, and so on. Irregular shape pockets can have any machinable shape, but they still use the same machining and programming methods as regular pockets.

One of the most commonly machined boundary shapes in manufacturing is milling of a simple cavity, usually quite small, called a *slot*.

### PROGRAMMING SLOTS

Slots are often considered as special types of grooves. These 'grooves' usually have one or two radial ends. If there are two ends, they are joined by a straight groove. A slot can be either open or closed, with the same size radius on both ends, two different radii, or one radius only. A typical slot that has only one end radius is called a *keyway*.

Slots can be open or closed, straight, angular, circular, with straight walls or shaped walls (using a tapering end mill). Programming slots with accuracy in mind usually requires a roughing operation and a finishing operation. Both operations can be made with the same tool or with two or more tools, depending on the part material, required dimensional tolerances, surface finish, and other conditions.

Certain slots, for example keyways, can be done with special cutters, called slotting cutters, rather than an end mill. To program a slotting cutter is usually a simple process of a linear motion - in and out. More complex - *and more accurate* - slots are machined with end mills, and the walls of the slot are contoured under program control.

Figure 33-1 shows a drawing of a typical open slot. This drawing will be used to illustrate the programming techniques of an open slot.

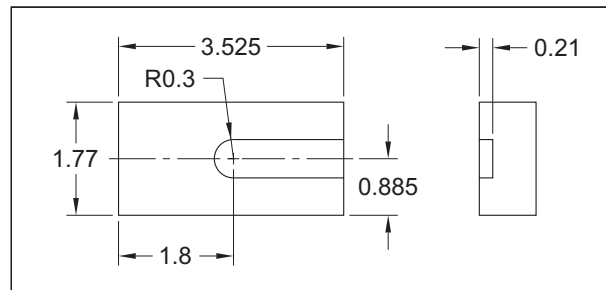


Figure 33-1

An open slot programming example O3301

#### ◆ Open Slot Example

Before programming any tool motion, study the drawing. That way, the machining conditions can be established, as well as setup and other requirements. Program zero can be determined quickly - dimensions are from the lower left corner (XY) and top (Z) of the part. That location will become the program zero.



Next considerations will relate to machining subjects:

- Number of tools
- Tool size
- Speeds and feeds
- Maximum cutting depth
- Method of cutting

#### Number of Tools

One or two tools can be used to cut the slot. If dimensional tolerances are very critical or the material is hard to cut, use two tools - one tool for roughing, another one for finishing. Both tools can have the same diameter or different diameters. For this example, only one tool will be used for both roughing and finishing.

#### Tool Size

Size of the cutting tool is mainly determined by the width of the slot. In the drawing, the slot has 0.300 radius, so the width is 0.600. There is no standard cutter of Ø0.600 - but - even if there were - would it be practical? What about a Ø0.500 inch cutter for 0.500 inch wide slot? It is possible, but the resulting cut would not be of the highest quality. Tolerances and surface finish would be hard to control. That means choosing a tool, preferably available off-shelf, that is a little smaller than the slot width. For the slot in the example, a Ø0.500 inch end mill is a suitable choice. When selecting the tool size, always calculate how much stock the tool will leave on the slot walls for finishing. Too much stock may require some additional semifinishing cuts. With the Ø0.500 cutter and the slot width of 0.600, the amount of stock left will be easy to calculate:

$$S = \frac{(W - D)}{2}$$

where ...

- S = Stock left on material  
 W = Width of slot ( = slot radius times two )  
 D = Cutter diameter

Stock left on the slot wall in the example will be:

$$S = (0.600 - 0.500) / 2 = 0.050$$

This is a suitable stock for finishing with a single cut.

#### Speeds and Feeds

Spindle speeds and cutting feedrates will depend on the exact situation at the CNC machine, so the example only uses a reasonable speed of 950 r/min and cutting feedrate of 8 in/min.

#### Maximum Cutting Depth

Drawing shows the slot depth as 0.210. Always check the depth - it may be too deep for a single cut, usually for small cutters or tough materials. Although a single cut can be used for the full depth, some stock at the slot bottom should be left for finishing.

#### Method of Cutting

Once all the other machining conditions are established, the method of cutting almost presents itself. Tool will be positioned above a clear position and at the slot center line. Then, the tool will be fed into the slot depth, leaving some material at the bottom, for finishing. In a linear motion, the tool will rough out material all the way to the slot radius center, then retract above the part. After that, it will be moved back to the original start position and at the full depth for slot contouring, in climb milling mode. In Figure 33-2, both XY tool motions and their program locations are shown as well as the stock left.

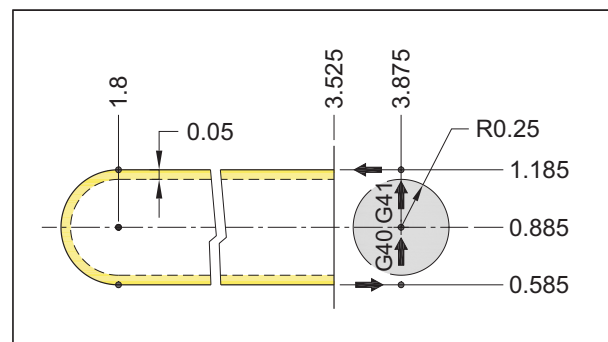


Figure 33-2

Contouring details for the open slot example O3301

To create the program is not difficult at all. The tool is in the spindle and all typical methods explained throughout the handbook are used.

```
O3301 (OPEN SLOT)
N1 G20 (INCH MODE)
N2 G17 G40 G80 (START UP SETTINGS)
N3 G90 G54 G00 X3.875 Y0.885 S950 M03 (START)
N4 G43 Z0.1 H01 M08 (START POSITION ABOVE)
N5 G01 Z-0.2 F50.0 (0.01 LEFT ON BOTTOM)
N6 X1.8 F8.0 (CUT TO SLOT RADIUS CNTR)
N7 G00 Z0.1 (RETRACT ABOVE WORK)
N8 X3.875 (RETURN TO START)
N9 G01 Z-0.21 F50.0 (FEED TO FULL DEPTH)
N10 G41 Y1.185 D01 F8.0 (APPROACH CONTOUR)
N11 X1.8 (CUT TOP WALL)
N12 G03 Y0.585 R0.3 (CUT SLOT RADIUS)
N13 G01 X3.875 (CUT BOTTOM WALL)
N14 G00 G40 Y0.885 (RETURN TO START POINT)
N15 Z1.0 M09 (RETRACT ABOVE WORK)
N16 G28 X3.875 Y0.885 Z1.0 M05 (M0 ZERO)
N17 M30 (END OF PROGRAM)
%
```

The example is quite self explanatory and the included block comments offer better understanding of the programming order and procedure. In this example, only one tool has been used. For high precision machining, using two tools will be better, even if it means a longer program.

### ◆ Closed Slot Example

Closed slot does not differ from an open slot that much. The greatest difference is in the tool entry into the material. There is no outside location - tool has to plunge into the material along the Z-axis, unless there is a predrilled hole. One method is to use a *center cutting end mill* (also known as *slot drill*). If this type of end mill is not available, or machining conditions are not suitable, tool will have to *ramp* into the material, as a second method. Ramping is a linear cutting motion, usually in the XZ, YZ, or XYZ axes.

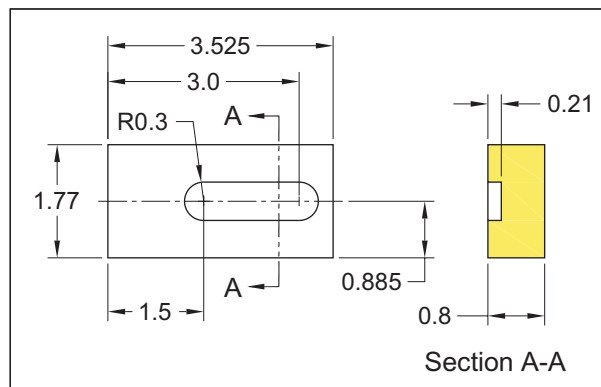


Figure 33-3

A closed slot programming example O3302

The second example is based on the drawing shown in Figure 33-3. This drawing is a slight modification of the open slot drawing. Many considerations established already will apply equally to the closed slot. A  $\varnothing 0.500$  inch end mill will be used, this time with a center cutting geometry that allows plunging into solid material.

Apart from different tool edge geometry required for the Z-axis plunging cut, only the actual method of cutting will change. For a closed slot (or a pocket), the tool has to move above work, to a certain XY start location. In the example, it will be the center of one of the slot radii. Portion of slot on the right is selected arbitrarily. Then, a plunge at a reduced feedrate will cut to the required depth (leaving 0.010 on the bottom) and, in a linear motion, the slot will be roughed out between the two centers - Figure 33-4.

Retracting the tool is not necessary, it can be fed into the final depth at the same tool location. The stock left is 0.050 all around the slot contour. At the final depth, and from the center location of the left part of the slot, finishing contour will start. Contouring will be more complex this time, because the tool is in a rather tight spot.

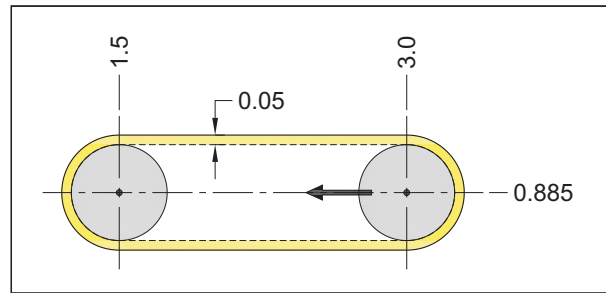


Figure 33-4

Roughing operation detail for a closed slot example O3302

### Internal Contour Approach

In the program, tool is now at the center of the left side of slot, ready to start the finishing cut. Climb milling mode has been selected and the contour approached in such a way that the tool motion continues to its left. One way is to make a straight linear cut from the current tool location at the center, to the 'south' position of the left arc (while applying the cutter radius offset).

This method works, but when approaching an inner contour it is better to use a tangential approach. An internal contour approached at a tangent requires an auxiliary approach arc (so called *lead-in arc*), since the linear approach towards the contour is not a choice.

Although the tangential approach using an arc improves the part surface finish, this preference creates another serious problem. *Cutter radius offset can never be started in circular interpolation mode!* Therefore, a non-circular motion has to be *added* - there will be *two* motions from the current tool location at the slot radius center to the contour start point:

- Motion 1 ... Linear motion with cutter radius offset
- Motion 2 ... Tangential approach arc motion

This technique is illustrated in Figure 33-5.

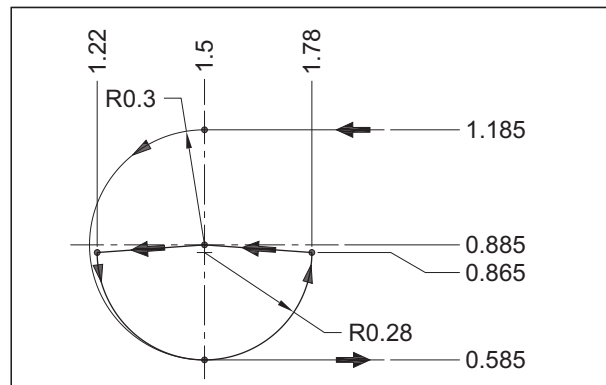


Figure 33-5

Detail of tangential approach towards an inner contour

Now, look closely at how the lead-in arc was created. The objective is to select the location and radius of the lead-in arc. Location selection is easy - the arc *must be tangent* to the contour. Lead-in radius dimension has to be selected with some logical thinking. When faced with an unknown dimension, always think of its purpose first. The purpose of lead-in arc is to move the cutting tool in a smooth curve towards the slot contour. That means the lead-in arc radius must be *larger* than the cutting tool radius. Finally, there is the slot radius itself, which is defined by the drawing. Relationship of all three radiuses can be put in perspective:

$$R_T < R_L < R_C$$

where ...

$R_T$	=	Tool radius	(T)	... selected
$R_L$	=	Lead-in radius	(L)	... calculated
$R_C$	=	Contour radius	(C)	... provided

Supply tool and contour radiuses and the lead-in radius can be calculated. The above formula shows mutual relationships. Slot contour radius ( $R_C$ ) is provided by the drawing. Once the cutting tool size is selected, that radius becomes fixed as well ( $R_T$ ). That leaves the lead-in approach radius ( $R_L$ ). That one has to be calculated and calculated *accurately, based on the other two dimensions*.

From the formula, it is clear that the lead-in radius arc must be greater than the cutter radius (0.250), while both must be smaller than the contour radius (0.300). That means the range (within three decimal places) is 0.251 to 0.299. If only rounded increments of 0.010 are considered, which one is better - 0.260 or 0.290? Well, the larger the better. With selection of rather a larger approach radius from the range, the arc tangential approach takes place at a smoother curve than with a smaller radius. The result is an improved surface finish. For program O3302, 0.280 is selected as the lead-in radius. This selection meets all the three relationships:

$$0.250 (R_T) < 0.280 (R_L) < 0.300 (R_C)$$

That is all information needed before writing the actual program. Note many programming similarities with the open slot, listed in program O3301.

```
O3302 (CLOSED SLOT)
N1 G20                                     (INCH MODE)
N2 G17 G40 G80                             (STARTUP SETTINGS)
N3 G90 G54 G00 X3.0 Y0.885 S950 M03        (START)
N4 G43 Z0.1 H01 M08                       (START POSITION ABOVE)
N5 G01 Z-0.2 F4.0                         (0.01 LEFT ON BOTTOM)
N6 X1.5 F8.0                             (CUT TO SLOT RADIUS CENTER)
N7 Z-0.21 F2.0                           (FEED TO FULL DEPTH)
N8 G41 X1.22 Y0.865 D01 F8.0 (LINEAR APPROACH)
N9 G03 X1.5 Y0.585 R0.28 (CIRCULAR APPROACH)
N10 G01 X3.0                             (CUT BOTTOM WALL)
N11 G03 Y1.185 R0.3 (CUT RIGHT SLOT RADIUS)
```

```
N12 G01 X1.5                             (CUT WALL TOP)
N13 G03 Y0.585 R0.3 (CUT RADIUS LEFT)
N14 X1.78 Y0.865 R0.28 (CIRCULAR DEPARTURE)
N15 G01 G40 X1.5 Y0.885 (LINEAR DEPARTURE)
N16 G00 Z1.0 M09 (RETRACT ABOVE WORK)
N17 G28 X1.5 Y0.885 Z1.0 M05 (MACHINE ZERO)
N18 M30 (END OF PROGRAM)
%
```

This program example is also a good illustration of how to approach *any* inside contour for finishing. Slots of other kinds (angular, circular, etc.), use the same principles illustrated in the last two examples.

## POCKET MILLING

Pocket milling is also a typical and common operation on CNC machining centers. Milling a pocket means to remove material from an enclosed area, defined by its boundary. This bounded area is further defined by its walls and bottom, although walls and bottom could be tapered, convex, concave, rounded, and have other shapes. Walls of a pocket create the boundary contour. Pockets can have square, rectangular, circular or undefined shape, they can be empty inside or they may have islands (areas that are not machined).

Programming pockets manually is usually efficient only for simple pockets, pockets of regular shapes, such as rectangular or circular pockets. For pockets with more complex shapes and pockets with many islands, the assistance of a computer is usually required.

### ◆ General Principles

There are two main considerations when programming a pocket for milling:

- Method of cutter entry
- Method of roughing

To open a space to start milling a pocket (into solid material), the cutter motion has to be programmed to enter along the direction of spindle (Z-axis), which means the cutter must be *center cutting* - to be able to *plunge* cut. In cases where the plunge cut is either not practical or not possible, a method called *ramping* can be used very successfully. This method is often used when center cutting tool is not available. Ramping requires the Z-axis to be used together with X-axis, Y-axis, or both. This motion will, of course, be a 2-axis or a 3-axis linear motion. All modern CNC machining centers support this type of motion.

Method of removing most of the material from the pocket is called *roughing*. Roughing method selection can be a little more complex. The location *where* to start the plunge or ramped cut is important, so is the *width* of cut. It may be difficult to do all roughing in climb milling mode. It may be difficult to leave exactly the same amount of stock for finishing everywhere in the pocket. Many cuts will be irregu-

lar and stock amount will not be even. For that reason, it is quite common to program a semifinishing cut of the pocket contour shape, before any finishing cut takes place. One or more tools may be used for this situation, depending on exact requirements.

Some typical methods for roughing a pocket are:

- Zigzag
- One direction - from inside of the pocket out
- One direction - from outside of the pocket in

In computer applications, other pocketing options are also possible, such as a true spiral, morph, one way, and others. In many cases, there is a choice of specifying the angle of cut, even a user selected point of entry and finishing leftovers. Manually, these more complex methods may be used as well, but it will be a very tedious work.

### ◆ Pocket Types

The most common pockets are also the easiest to program. They all have a regular shape, without any islands:

- Square pocket
- Rectangular pocket
- Circular pocket

Square and rectangular pockets are fundamentally the same and apart from their different length-to-width ratios, there is no major difference in programming.

## RECTANGULAR POCKETS

Rectangular and square pockets are quite easy to program, particularly if they are parallel to the X or Y axes. As an example of a rectangular pocket, the one illustrated in *Figure 33-6* will be used.

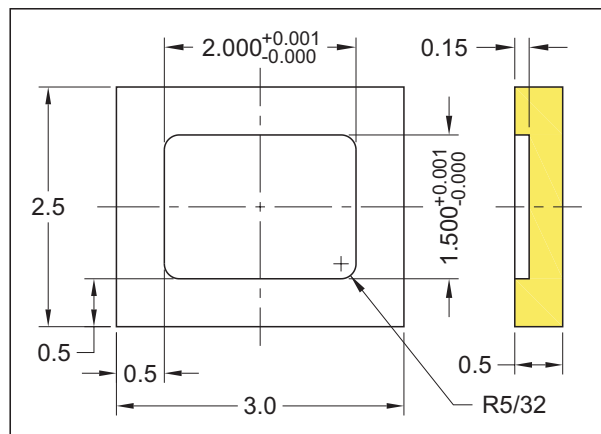


Figure 33-6

Sample drawing of a rectangular pocket - program O3303

To illustrate the complete pocket programming, starting with tooling selection is important. Material is also important and so are other machining decisions. Although rectangular pockets are often drawn with sharp corners, they will always have corners of the tool used, or larger, when machined. Corners in the drawing are 5/32 (0.1563), and can be made by a  $\varnothing 5/16$  center cutting end mill ( $\varnothing 0.3125$ ). For roughing, it may be a good choice, but for finishing, the radius should be a little smaller so the tool can actually *cut* in the corner, not just rub there. Selection of a  $\varnothing 0.250$  end mill is reasonable and will be used in the example.

Since all material in the enclosed area has to be removed (including bottom clean-up), think about all possible places where the cutting tool can enter into the depth by plunging or ramping. Ramping must always be done in a clear area, but plunging can be done almost anywhere. There are only two practical locations:

- Pocket center
- Pocket corner

There are some benefits to both selections and the inevitable disadvantages. Starting at the pocket center, the tool can follow a single directional path and, after the initial cut, can cut only in climb milling or conventional milling mode. There are slightly more math calculations involved in this method. Another method, starting at the pocket corner, is quite popular as well, but uses a zigzag motion, so one cut will be in a climb milling mode, the next cut will be in a conventional mode of machining. It is a little easier for calculations, however. In the example, a pocket corner will be used as the start location.

Any pocket corner is equally suitable for the start. In the program example O3303, the lower left pocket corner will be used.

There are three important factors the programmer has to consider when selecting start location for the cutting tool in an enclosed area:

- Cutter diameter (or radius)
- Amount of stock left for finishing
- Amount of stock left for semifinishing

There are also very important dimensions of the part, as defined in the drawing. They are the *length*, the *width*, and the *corner radius* (or radiuses) of the pocket - they must always be known, as well as the pocket position and its orientation relative to other features of the part.

In *Figure 33-7*, the starting point is identified as X1 and Y1 distance from the given corner (lower left), and all additional data are shown as well.

The letters are used to identify variable settings that must be done; programmer chooses their amounts, depending on a particular job.

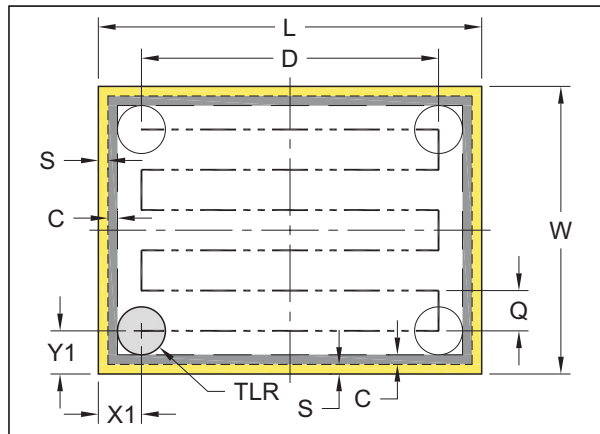


Figure 33-7

Pocket roughing start point in the corner - zigzag method

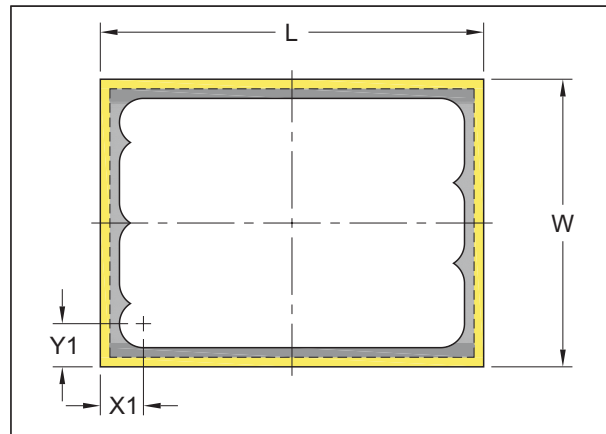


Figure 33-8

Result of a zigzag pocketing, without a semifinish cut

#### Meaning of variable data :

- $X_1$  = X-location of tool at start
- $Y_1$  = Y-location of tool at start
- TLR = Tool radius (cutter diameter / 2)
- L = Pocket length as per drawing
- W = Pocket width as per drawing
- Q = Calculated stepover between cuts
- D = Calculated length of actual cut
- S = Stock left for finishing
- C = Stock left for semifinishing (clearance)

#### ◆ Stock Amount

There are two stock amounts (values) - one relates to the *finishing* operation, usually done with a separate finishing tool, the other one relates to the *semifinishing* operation, usually done with a roughing tool. The cutter moves back and forth in a zigzag direction, leaving behind so called scallops. In 2D work, the word 'scallop' is used to describe uneven wall surface caused by the tool shape, and is similar in 3D cutting as well. The result of such a zigzag toolpath is generally unacceptable for finish machining, because of the difficulty of maintaining tolerances and surface finish while cutting uneven stock.

To avoid possible cutting problems at a later time, a secondary semifinishing operation is often necessary. Its purpose is to eliminate all uneven material. Choose semifinishing cut particularly for machining tough materials or when using small size diameter tools. Semifinishing stock allowance, marked as the C amount in the illustration, can also be equal to zero. If that is the case, it means no additional stock is required. Typically the stock allowance will have a small value.

Figure 33-8 illustrates the result of roughing operation of a rectangular pocket, *without* the semifinishing cut. Note the uneven stock (scallop) left for finishing tool. All high spots create the heaviest obstacles for a subsequent tool, so semifinishing tool path is highly recommended.

#### ◆ Stepover Amount

The actual pocket shape before semifinishing is always determined by the amount of *stepover*. A stepover in pocketing is just another name for the *width of cut*. This amount may be selected without actual calculation. A much better way is to *calculate* the stepover amount, based on the number of required cuts. That way, this amount will be equal for *all* cuts. Since it is quite common to think of a width of cut as some *percentage* of the cutter diameter, use this method for reference purposes only, and still *calculate* the cutting width and select one that will be *closest* to the cutter diameter percentage desired.

In the example, a rather larger than average stepover will be used, based on five required cuts (zigzag type). There is a substantial difference whether the number of cuts is selected as an *even number* or as an *odd number*:

- Even number of cuts will terminate roughing on the *opposite side* of the pocket relative to its start location
- Odd number of cuts will terminate roughing on the *same side* of the pocket relative to its start location

Practically, it does not matter which corner is selected to start at or in which direction the first cut begins. What matters is that the stepover is reasonable and, preferably, equal for all cuts. There is a simple way of calculating the stepover, based on a given number of cuts. If the calculated amount is too small or too large, just repeat the calculation with a different number of cuts N.

The calculation can be expressed in a formula:

$$Q = \frac{W - 2 \times \text{TLR} - 2 \times S - 2 \times C}{N}$$



In the formula, N is the number of selected stepovers and all other variables have the same meaning as before.

➡ Example :

In the example, five equal stepovers are needed, based on the pocket width of 1.500 inches, tool diameter 0.250 (TLR is 0.125), finishing stock S as 0.025 and semifinishing stock C as 0.010. The stepover size will be:

$$Q = (1.5 - 2 \times 0.125 - 2 \times 0.025 - 2 \times 0.01) / 5$$

$$Q = 0.2360$$

That may be a little too much for a  $\varnothing 0.250$  end mill, but it will make the example a bit shorter. Seven stepovers would result in a more reasonable amount of 0.1686 (rounding to three decimal places as 0.169 does no harm).

The above formula may also be modified to use pocket length, rather than pocket width. This may be a better choice if the pocket is narrower along X-axis, than it is along Y-axis.

◆ Length of Cut

Before any semifinishing, the cutting length - the incremental distance D of each cut - has to be calculated.

In many respects, formula to calculate the length of cut is very similar to the stepover calculation:

$$D = L - 2 \times \text{TLR} - 2 \times S - 2 \times C$$

In this example, the D amount will be:

➡ Example :

$$D = 2.0 - 2 \times 0.125 - 2 \times 0.025 - 2 \times 0.01$$

$$D = 1.6800$$

This is the *incremental* length of cut between stepovers (no cutter radius offset has been used).

◆ Semifinishing Motions

The only purpose of semifinishing motions is to eliminate uneven stock (scallops). Since the semifinishing will be normally done with the same tool as for roughing operation, the place to start semifinishing cuts is the last tool position of the roughing sequence. In this case, it was the pocket upper left corner. *Figure 33-9* shows linear motions from the *Start* to *End* of the semifinishing.

The length  $L_1$  and  $W_1$  are calculated, and the difference between the *Start* and *End* position is the C amount, constant along both axes.

Formula for the length and width of semifinishing cut, its actual cutting distance, is listed next:

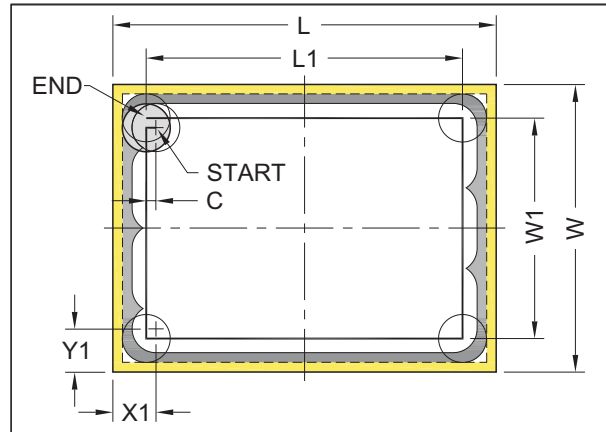


Figure 33-9

Semifinishing tool path begins at the last roughing location, and leaves equal stock for finishing operation

$$L_1 = L - 2 \times \text{TLR} - 2 \times S$$

$$W_1 = W - 2 \times \text{TLR} - 2 \times S$$

➡ Example :

$$L_1 = 2.0 - 2 \times 0.125 - 2 \times 0.025$$

$$L_1 = 1.7000$$

$$W_1 = 1.5 - 2 \times 0.125 - 2 \times 0.025$$

$$W_1 = 1.2000$$

◆ Finishing Toolpath

Once the pocket is roughed out and semifinished, another tool (or even the same tool in some cases) can be used to finish the pocket to its final size. This programmed tool path will typically provide offsets to maintain machining tolerances and speeds and feeds to maintain required surface finish. Typical starting tool position for a small to medium pocket is at its center, for a large pocket the starting position should be at the middle of the pocket, away from one of the walls, but not too far.

For the finishing cut, cutter radius offset should be in effect, mainly to increase flexibility in maintaining tolerances during machining. Since cutter radius offset cannot be started during an arc or a circular motion, linear lead-in and lead-out motions have to be added. In *Figure 33-10* is an illustration of a typical finishing toolpath for a rectangular pocket (with the start at the pocket center).

Some conditions do apply in these cases. One is that the lead-in arc radius must be calculated, using precisely the same method as for slots (same formula - different order):

$$R_L > R_T < R_C$$

where ...

$R_L$  = Lead radius  
 $R_T$  = Tool radius  
 $R_C$  = Corner radius

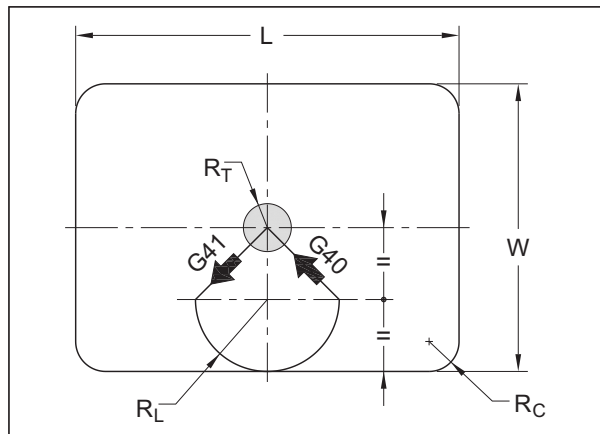


Figure 33-10

Typical finishing toolpath for a rectangular pocket

Milling mode is normally climb milling mode and the radius offset used will be G41, to the left side of the contour.

### Example :

To calculate the lead-in (approach) radius for the example drawing, start with the corner radius. Radius  $R_C$  is given as  $5/32$  (0.1563) and the tool radius  $R_T$  has been selected as 0.125, so the condition  $R_T < R_C$  is satisfied. In order to also satisfy the condition  $R_L > R_T$ , choose almost any lead-in radius larger than the tool radius, as long as it is reasonable. Both pocket length and width are also important, as always. If possible, choose the lead-in radius as *one quarter* of the pocket width  $W$ , for a little easier tool motion calculations. In the example,

$$R_a = W / 4 = 1.5 / 4$$

$$R_a = 0.375$$

Condition is satisfied, the lead-in radius is larger than the tool radius, and can be safely used in the program.

### ◆ Rectangular Pocket Program Example

Once all selections and decisions have been made, the pocketing program can be written, in example O3303. Two tools will be used, both  $\varnothing 0.250$  end mills, the roughing cutter must be able of center cutting. Program zero is the lower left corner of the part. All roughing and semi-finishing steps are documented in program comments.

```
O3303 (RECTANGULAR POCKET)
N1 G20
N2 G17 G40 G80 T01 (0.250 ROUGHING SLOT DRILL)
N3 M06
N4 G90 G54 G00 X0.66 Y0.66 S1250 M03 T02
N5 G43 Z0.1 H01 M08
N6 G01 Z-0.15 F7.0
  (-- ROUGHING START -----)
N7 G91 X1.68 F10.0                                (CUT 1)
N8 Y0.236                                           (STEPOVER 1)
N9 X-1.68 F12.0                                     (CUT 2)
N10 Y0.236                                          (STEPOVER 2)
N11 X1.68                                           (CUT 3)
N12 Y0.236                                          (STEPOVER 3)
N13 X-1.68                                          (CUT 4)
N14 Y0.236                                          (STEPOVER 4)
N15 X1.68                                           (CUT 5)
N16 Y0.236                                          (STEPOVER 5)
N17 X-1.68                                          (CUT 6)
  (-- SEMIFINISH START -----)
N18 X-0.01                                         (SEMIFINISH STARTUP X)
N19 Y-0.01                                         (SEMIFINISH STARTUP Y)
N20 Y-1.19                                         (LEFT Y- MOTION)
N21 X1.7                                           (RIGHT X+ MOTION)
N22 Y1.2                                           (UP Y+ MOTION)
N23 X-1.7                                           (LEFT X- MOTION)
N24 G90 G00 Z0.1 M09
N25 G28 Z0.1 M05
N26 M01

N27 T02                                           (0.250 FINISHING END MILL)
N28 M06
N29 G90 G54 G00 X1.5 Y1.25 S1500 M03 T01
N30 G43 Z0.1 H02 M08
N31 G01 Z-0.15 F12.0
  (-- FINISHING POCKET -----)
N32 G91 G41 X-0.375 Y-0.375 D02 F15.0
N33 G03 X0.375 Y-0.375 R0.375 F12.0
N34 G01 X0.8437
N35 G03 X0.1563 Y0.1563 R0.1563
N36 G01 Y1.1874
N37 G03 X-0.1563 Y0.1563 R0.1563
N38 G01 X-1.6874
N39 G03 X-0.1563 Y-0.1563
N40 G01 Y-1.1874
N41 G03 X0.1563 Y-0.1563 R0.1563
N42 X0.8437
N43 G03 X0.375 Y0.375 R0.375
N44 G01 G40 X-0.375 Y0.375 F15.0
N45 G90 G00 Z0.1 M09
N46 G28 Z0.1 M05
N47 X-2.0 Y10.0
N48 M30
%
```

Study the program carefully. It is not short but it follows all the decisions made earlier and offers many other details.

In the program, blocks N17 and N18 can be joined together into a single block. The same applies to blocks N19 and N20. They are only separated for the convenience of tracing the tool motions to match the illustrations. There is a slight benefit in using incremental mode of programming, but absolute mode would have been just as easy.

## CIRCULAR POCKETS

The other common types of pockets are so called *circular* or *round pockets*. Although the word *pocket* somehow implies a closed area with a solid bottom, the programming method relating to circular pockets can also be used for circular openings that may have a hole in the middle, for example, some counterboring operations.

To illustrate a practical programming application for a circular pocket, Figure 33-11 shows the typical dimensions of such a pocket.

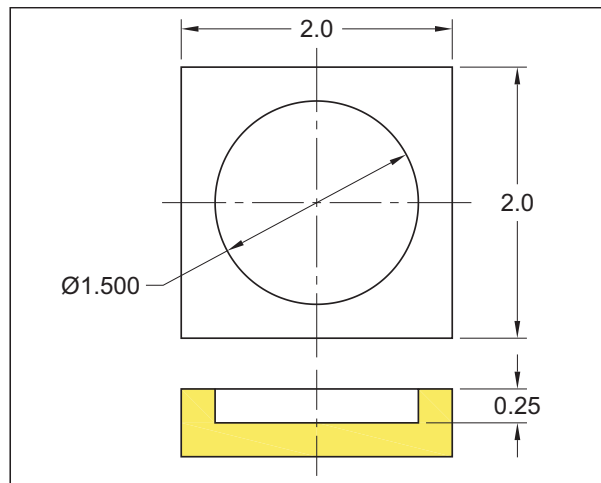


Figure 33-11

Sample drawing of a circular pocket (program examples O3304-06)

In terms of planning, the first thing to be done is selecting cutter diameter. Keep in mind, that in order to make the pocket bottom clean, without any residual material (uncut portions), it is important to keep the stepover from one cut to another by a limited distance that should be calculated. For circular pockets, this requirement influences the *minimum* cutter diameter that can be used to cut circular pocket in a single 360° sweep motion.

### ◆ Minimum Cutter Diameter

In the following illustration - Figure 33-12, relationship of the cutter diameter and the pocket diameter is shown. There is also a formula that will determine the minimum cutter diameter as *one third* of the pocket diameter. Milling will start at the pocket center, with a single 360° tool motion. In practical terms, selecting a cutter slightly larger than the minimum diameter is a much better choice. The major benefit of this calculation is in those cases when the pocket has to be done with only one tool motion around. The formula is still valid, even if cutting will be repeated several times around the pocket, by increasing the diameter being cut. In that case, the formula determines the maximum width of cut.

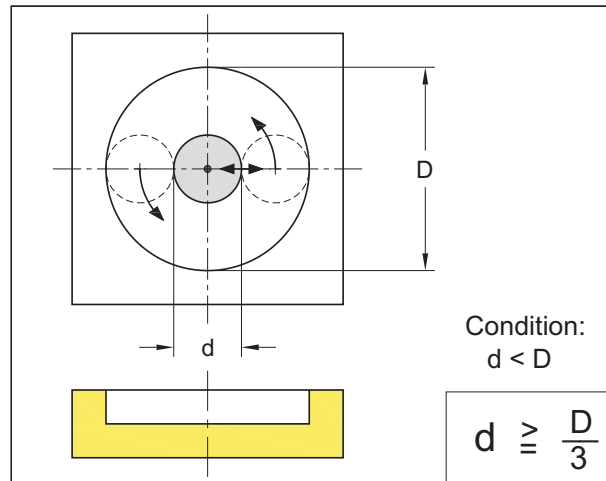


Figure 33-12

Relationship of the cutter diameter to the pocket diameter

For example, pocket diameter in the sample drawing is 1.5 inches. Using the one-third formula, select a plunging cutter (center cutting end mill), that has the diameter larger than  $1.5/3$ , therefore larger than 0.500. The nearest nominal size suitable for cutting will be Ø0.625 (5/8 slot drill).

### ◆ Method of Entry

The next step is to determine the tool entry method. In a circular pocket, the best place to enter along the Z-axis is at the pocket *center*. If the pocket center is also program zero X0Y0, and the pocket depth is 0.250, the program beginning may be similar to the following example (cutting tool already placed in the spindle is assumed):

```
O3304 (CIRCULAR POCKET - VERSION 1)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F8.0
N6 ...
```

In the next block (N6), the cutting tool will move from the pocket center towards the pocket diameter, and apply cutter radius offset along the way. This motion can be done in two ways:

- As a simple straight linear motion
- As a combined linear motion with a circular approach

### ◆ Linear Approach

The linear departure from pocket center can be directed into any direction, but a direction towards a quadrant point is far more practical. In the example, a motion along the Y-positive direction is selected, towards the 90° position.

Along the way, cutter radius offset for climb milling mode G41 is programmed, followed by the full 360° arc and another straight motion, back towards the center. During this motion, cutter radius offset will be canceled. *Figure 33-13* shows the toolpath.

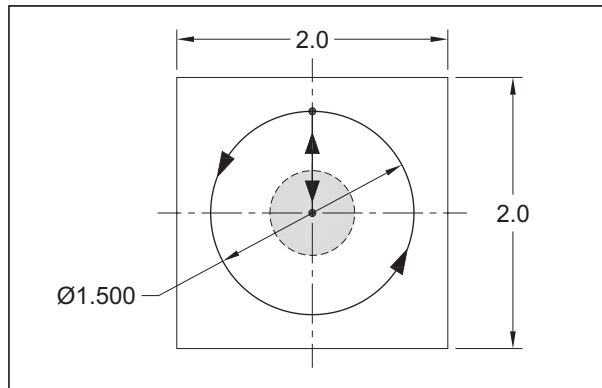


Figure 33-13

Linear approach for a circular pocket milling - program O3304

Graphic representation can be followed by a corresponding program segment - approach a quadrant point, profile the full arc, then return back to the center:

```
N6 G41 Y0.75 D01 F10.0
N7 G03 J-0.75
N8 G01 G40 Y0 F15.0
```

Now, the tool is back at the pocket center and the pocket is completed. Tool must also retract first, then move to machine zero (G28 motion is always in the rapid mode):

```
N9 G28 Z-0.25 M09
N10 G91 G28 X0 Y0 M05
N11 M30
%
```

This method is very simple, but may not always be the best choice, particularly for very close tolerances or high surface finish requirements. Better drawing tolerances may be achieved by roughing operations with one tool and finishing operations with one or more additional tools.

A possible surface tool mark, left at the contact point with the pocket diameter, is a distinct possibility in a straight approach to the pocket diameter. The simple linear approach is quite efficient when such pocket or counterbore is not too critical. Here is the complete listing for program O3304:

```
O3304 (CIRCULAR POCKET - VERSION 1)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F8.0
N6 G41 Y0.75 D01 F10.0
N7 G03 J-0.75
```

```
N8 G01 G40 Y0 F15.0
N9 G28 Z-0.25 M09
N10 G91 G28 X0 Y0 M05
N11 M30
%
```

Another programming technique for a circular pocket is much more practical - one that makes better surface finishes and also maintains tight tolerances required by many drawings. Instead of a single linear approach directly towards the pocket diameter, cutting tool can be programmed in a combined linear-circular approach.

#### ◆ Linear and Circular Approach

For this method, the cutting motion will be changed. Ideally, a small one half-arc motion could be made between the center and the pocket start point. That is possible only if the cutter radius offset is *not* used. As a matter of fact, some controls use a circular pocket milling cycle G12 or G13, doing exactly that (*see comments and an example on page 302*). If the Fanuc control has the optional *User Macros*, custom made G12 or G13 circular pocket milling cycle can be developed. Otherwise, a step-by-step method is the only way, one block at a time.

Since the radius offset is needed to maintain tolerances, and the offset cannot start on an arc, a linear approach will be programmed first with the cutter radius offset applied. Then, the circular lead-in motion is programmed. When the pocket is completed, the procedure will be reversed and the radius offset canceled during a linear motion back to the pocket center (lead-out). The approach radius calculation in this application is exactly the same as described for a slot finishing toolpath, earlier in this chapter. *Figure 33-14* shows the suggested toolpath.

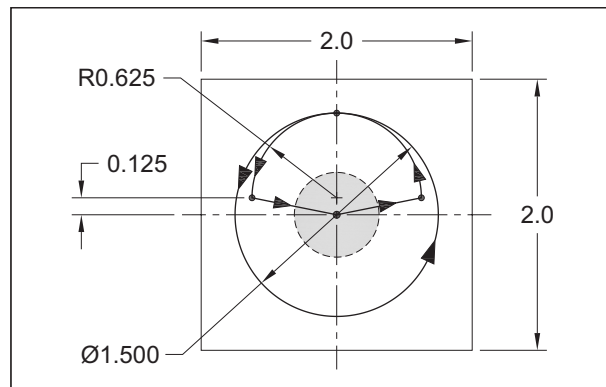


Figure 33-14

Combined linear and circular approach for a circular pocket milling - program example O3305

This example uses an lead radius of 0.625. Any radius that is *greater* than the cutter radius (0.3125) and *smaller* than the pocket radius (0.750) is correct. The final program O3305 complements the above illustration in *Figure 33-14*.

## O3305 (CIRCULAR POCKET - VERSION 2)

```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F8.0
N6 G41 X0.625 Y0.125 D01 F10.0
N7 G03 X0 Y0.75 R0.625
N8 J-0.75
N9 X-0.625 Y0.125 R0.625
N10 G01 G40 X0 Y0 F15.0
N11 G28 Z-0.25 M09
N12 G91 G28 X0 Y0 M05
N13 M30
%
```

This programming technique is by far superior to the straight linear approach. It does not present any additional programming difficulty at all, partly because of the symmetry of tool motions. In fact, this method can be - and should be - used for just about any approach towards an internal contour finishing (or even an external contour).

### ◆ Roughing a Circular Pocket

Often, a circular pocket is too large for a given tool to guarantee the bottom cleanup in a single cut around. In this case, the pocket has to be enlarged by roughing it first, in order to remove all excessive material, then the finishing toolpath can be applied. Some controls have special cycles, for example, a spiral pocketing. On Fanuc controls, custom cycles can be created with the *User Macros* option.

As an example, the same pocket drawing will be used as illustrated earlier in *Figure 33-11*, but machining will be done with a  $\varnothing 0.375$  cutter - *Figure 33-15*.

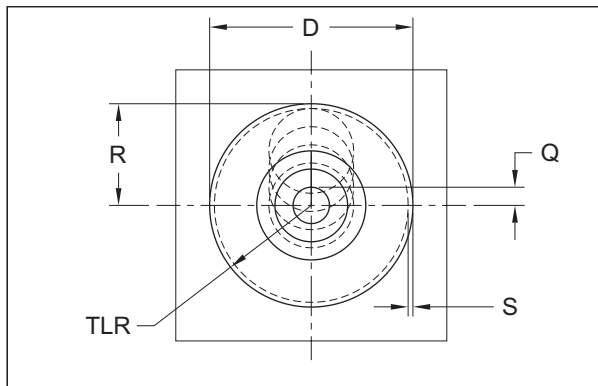


Figure 33-15

Roughing out a circular pocket - program O3306

The  $\varnothing 0.375$  end mill is a small tool that will *not* cleanup the pocket bottom using the earlier method. Schematic method of roughing is shown in *Figure 33-15*, and the value of  $Q$  is the equal stepover amount, calculated from the number of steps  $N$ , the cutter radius  $TLR$  and the stock amount  $S$ , left for the finishing toolpath.

This calculation is logically similar to the one for a rectangular pocket and the desired stepover amount can be accomplished by changing the number of steps.

Program example O3306 uses three stepovers, calculated from the following formula:

$$Q = \frac{R - TLR - S}{N}$$

where ...

- $Q$  = Calculated stepover between cuts
- $R$  = Pocket radius (pocket diameter  $D / 2$ )
- $TLR$  = Tool radius (cutter diameter  $/ 2$ )
- $S$  = Stock left for finishing
- $N$  = Number of cutting steps

In this application, the example calculations are:

➡ Example :

```

R = 1.5 / 2 = 0.75           Diameter D = 1.5
TLR = 0.375 / 2 = 0.1875
S = 0.025
N = 3
```

Using the above formula, the stepover amount  $Q$  can be found by calculation:

```

Q = (0.75 - 0.1875 - 0.025) / 3
Q = 0.1792
```

Final roughing program is quite simple and there is no cutter radius offset programmed or even needed. Note the benefit of incremental mode G91. It allows the stepover  $Q$  to be easily seen in the program, in G01 linear mode. Every following block contains the arc vector J, cutting the next full circle. Each circle radius (J) is increased by the amount of stepover  $Q$ :

## O3306 (CIRCULAR POCKET ROUGHING)

```

N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1500 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F7.0
N6 G91 Y0.1792 F10.0           (STEPOVER 1)
N7 G03 J-0.1792               (ROUGH CIRCLE 1)
N8 G01 Y0.1792               (STEPOVER 2)
N9 G03 J-0.3584               (ROUGH CIRCLE 2)
N10 G01 Y0.1792              (STEPOVER 3)
N11 G03 J-0.5376              (ROUGH CIRCLE 3)
N12 G90 G01 X0 F15.0
N13 G28 Z-0.25 M09
N14 G91 X0 Y0 M05
N15 M30
%
```



## CIRCULAR POCKET CYCLES

In the chapter that covered circular interpolation, two similar circular pocketing cycles were described briefly - see page 252. In this chapter, two more examples will provide additional details. Fanuc controls do not have the very useful G12 and G13 circular pocketing cycle as a standard feature. Controls that do have it, for example Yasnac, have a built-in macro (cycle), ready to be used. Fanuc users can create their own custom macro (as a special G-code cycle), with the optional *User Macro (Custom Macro)* feature, which can be developed to offer even more flexibility than a built-in cycle. See a special note at the end of this page.

The two G-codes are identical in all respects, except the cutting direction. Meaning of the two G-codes in a circular pocket cycle is:

G12	Circular pocket cutting CW
G13	Circular pocket cutting CCW

Either cycle is *always* programmed *without* cutter radius offset in effect - in G40 cancel mode - and has the following program format:

G12 I.. D.. F.. (CONVENTIONAL MILLING)

or

G13 I.. D.. F.. (CLIMB MILLING)

where ...

- I = Pocket radius
- D = Cutter radius offset number
- F = Cutting feedrate

Typically, either cycle is called in a program when cutting tool reaches the *center* and the *bottom* of a pocket to be machined. All cutting motions are arc motions, and there are three of them. There are no linear motions. The arbitrary start point (and end point) on the pocket diameter will usually be at 0° (3 o'clock) - Figure 33-16.

Previous example in Figure 33-11 can be used to illustrate the G12 or G13 cycle. For comparison, here is the program O3305, using a Ø0.625 end mill:

```
O3305 (CIRCULAR POCKET - VERSION 2)
N1 G20
N2 G17 G40 G80
```

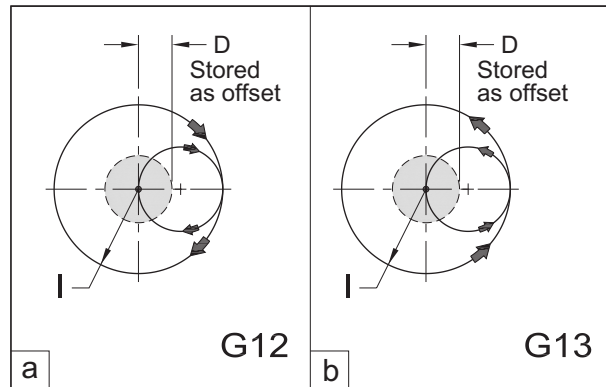


Figure 33-16

Circular pocket cycles G12 and G13

```
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F8.0
N6 G41 X0.625 Y0.125 D01 F10.0
N7 G03 X0 Y0.75 R0.625
N8 J-0.75
N9 X-0.625 Y0.125 R0.625
N10 G01 G40 X0 Y0 F15.0
N11 G28 Z-0.25 M09
N12 G91 G28 X0 Y0 M05
N13 M30
%
```

If G12 or G13 cycle or a similar macro is available, the following program O3306 can be written, using the same tool and climb milling mode:

```
O3306 (CIRCULAR POCKET - G13 EXAMPLE)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S1200 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.25 F8.0
N6 G13 I0.75 D1 F10.0 (CIRCULAR POCKET)
N7 G28 Z-0.25 M09
N8 G91 G28 X0 Y0 M05
N9 M30
%
```

Custom macros are extremely powerful programming tools, but their subject is quite large and beyond the limits of this handbook. **Special note:**

For those interested in actual step-by-step development of G12/G13 cycle, refer to my book *Fanuc CNC Custom Macros*, also published by Industrial Press, Inc.

## 34

**TURNING AND BORING**

There is so much information that can be covered in this section, that a whole book could be written just on the subject of turning and boring. Selected subjects are presented in this chapter, others are covered in chapters dealing with lathe cycles, grooving, part-off, single point threading, etc.

**TOOL FUNCTION - TURNING**

In terms of distinction, *turning* and *boring* are practically identical operations, except for the area of metal removal where the actual machining takes place. Often, terms *external turning* and *internal turning* are also used, meaning the same as turning and boring respectively. From programming perspective, the rules are virtually the same, and all significant differences will be covered as necessary.

CNC lathes require programming the selected tool by its tool number, using the T-address. In comparison with a CNC machining center, the tool function for lathes is more extensive and calls for additional details. One major difference between milling and turning controls is the fact that the T-address for CNC lathes *will make the actual tool change*. This is not the case in milling. No M06 function exists on a standard CNC lathe.

◆ **T-Address**

One difference from machining centers is that a tool defined as T01 in the program *must* be mounted in the turret station #1, tool defined as T12 must be mounted in turret station #12, etc. Another difference between milling and turning tools is in the *format* of the T-address. Format for turning system is T4, or more accurately, T2+2. The first two digits identify the turret station number and geometry offset, the last two digits identify the wear tool offset number for the selected tool station - *Figure 34-1*.

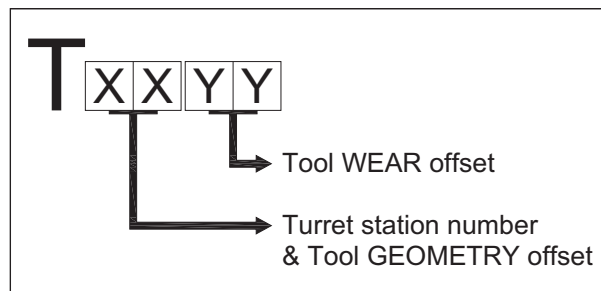


Figure 34-1

Typical tool function address for CNC lathes

*Txxyy* format represents tool station *xx* and wear offset number *yy*. For example, T0202 will cause the turret to index to the tool station #2 (first two digits) which will become the working station (active tool). At the same time, the associated tool wear offset number (the second pair of digits) will become effective as well, unless it is 00.

Selection of tool number (the first pair of digits), also selects the geometry offset on most modern CNC lathes. In that case, the second pair of digits will select tool wear offset number. Any tool station selected by the turret station number identification can be associated with any offset number within the available offset range. In most applications, only one tool offset number is active for any selected tool. In such a case, it is wise to program the offset number the same as the tool number. Such an approach makes the operator's job much easier.

Consider the following choices:

<b>G00 T0214</b>	<i>Tool station 02, wear offset 14</i>
<b>G00 T1105</b>	<i>Tool station 11, wear offset 05</i>
<b>G00 T0404</b>	<i>Tool station 04, wear offset 04</i>

Although all examples are technically correct, only the last example format is recommended. When many tools are used in a program, offset numbers for individual tools may be confusing, if they do not correspond to the tool station numbers. There is only one time when the offset number *cannot* be the same as the tool station number. That happens in those cases when *two or more offsets* are assigned to the same tool, for example T0202 for the first wear offset, T0222 for the second wear offset.

Leading zeros in the tool function can be omitted for the tool number selection, but not for selection of the wear offset number. T0202 has the same meaning when written as T202. Eliminating the leading zero for tool wear offset will result in an incorrect statement:

*T22* means *T0022*, which is an illegal format.

In summary, the active side of the turret (tool station) is programmed by the first pair of digits, the wear offset number is programmed by the last pair of digits in the tool function command:

**G00 T0404**

The most useful preference is to disregard the leading zero suppression and use the tool function in its full format, as shown above and in all examples in this handbook.

## LATHE OFFSETS

Although tool offset has been to some extent covered in the previous section describing the tool function, it is a very important feature for turning systems and some review will be beneficial.

Geometry offset is measured for *each tool* as the actual distance from the tool reference point to the program zero (Z-axis distance will be stored as a negative value and so will be the X-diameter) - Figure 34-2.

Geometry offset defines the tool position  
in relation to program zero

Wear offset is used for fine-tuning dimensions

The best way to illustrate the importance of tool wear offset, is to consider a program that does *not* use it. All programmed dimensions are ideal values, based on the drawing. Variable insert tolerances are not considered, neither is the tool wear. Any deviation from programmed dimensions caused by the actual tool size will produce an incorrect dimension when the part is machined, a very important concern for jobs with tight tolerances. Tool wear offset is used to 'fine tune' the actual machined dimensions against intended programmed dimensions.

The purpose of tool wear offset is to adjust any difference between the programmed dimensions and the actual tool position on the part contour. If the wear offset is not available to the control, all adjustments are made to the only offset available - that is to the geometry offset.

### ◆ Offset Entry

Tool offset number can be entered into the program in two different ways:

- As a command *independent* of the tool motion
- As a command applied *simultaneously* with a tool motion statement

### ◆ Independent Tool Offset

For an independent offset entry in the program, tool offset is applied *together with tool indexing*:

**N34 G00 T0202**

This command is usually programmed as the first block for each tool (in a clear position). If the older G50 position register is used, the offset is programmed together with, or immediately following, the coordinate register block. At this point, the tool is still at its indexing position. When tool offset is activated, it will cause a *physical motion* by the amount stored in the offset register. Note the preparatory command G00 before the tool function. This is a very important command, since it will enable the physical offset motion to actually take place. G00 is more important for the first tool, but should be programmed for any tool. On page 23, covering the control system, the initial control status (when the power is turned on) is described. Since the control system usually assumes the G01 command (linear interpolation) at the start up, feedrate would be required. However, it looks rather absurd to program T0202 F0.025, although it is correct. Rapid motion is far more practical and rather than depending on the current control status, programming the G00 command will *always* get the tool offset activated.

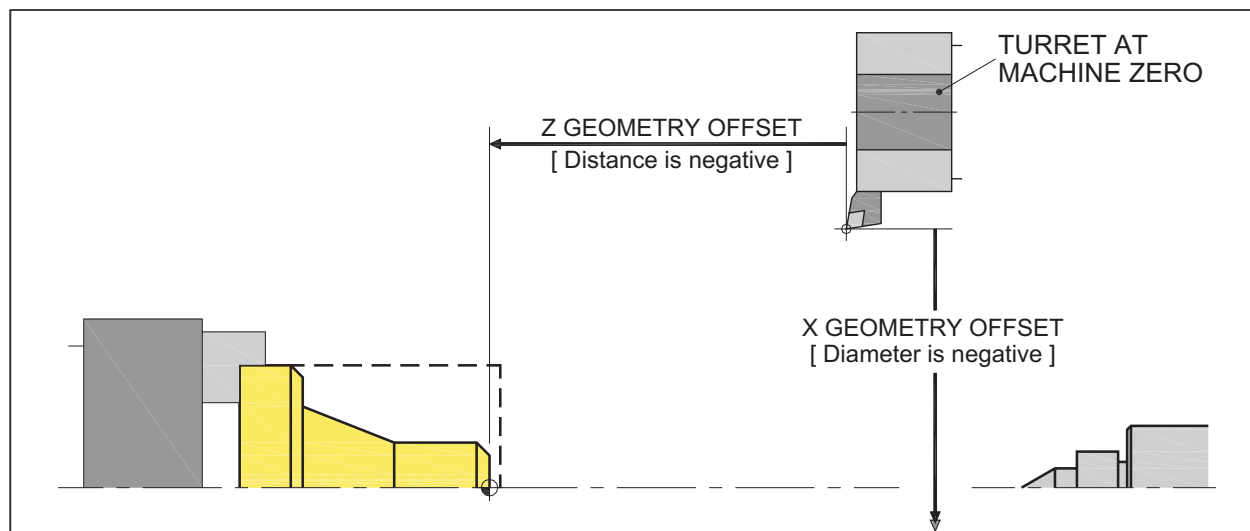


Figure 34-2

Geometry offset is the distance from tool reference point to program zero, measured along an axis from machine zero

### ◆ Tool Offset with Motion

The second method is to program the wear offset simultaneously with a cutting tool motion, usually during tool approach towards the part. This is the preferred method. The following two examples illustrate this recommended programming of the T-function for turning systems - offset is activated when the second pair of digits in a tool number call are equal to or larger than 01:

```
N1 G20 T0100
N2 G96 S300 M03
N3 G00 X.. Z.. T0101 M08
...
```

Tool change in the first block N1 uses no offset number (00) - just the tool number, that is also the geometry offset number. The offset is applied two blocks later in N3.

In most cases, it makes no difference, whether the offset is activated with or without a motion command. But some limitations are possible when programming the tool offset entry *without* a motion command. For example, if the wear offset value stored is unusually large and the tool starts from machine zero position, this type of programming may cause an overtravel condition.

Even in cases of a small offset value, there will always be a 'jump' motion of the turret when the offset is activated. Some programmers do not like this jumpy motion, although it will do no harm to the machine. In these cases, the best approach is to activate tool wear offset during the first motion, usually as a rapid approach motion towards the part. One consideration is very important when tool wear offset is activated together with a motion. Earlier in this chapter was a comment that the lathe tool function is also a function causing tool indexing. Without a doubt, the one situation to avoid is a simultaneous tool indexing and tool motion - it may have dangerous consequences.

In most cases, the best approach is to start each tool with tool indexing only, *without* any wear offset:

```
N34 T0200 M42
```

This example will register the coordinate setting for tool number 2, it will also index tool number 2 into working position, but it will *not* activate any offset (T0200 means index for tool 2 without tool wear offset). Gear range function may be added as well, if required. Such a block will normally be followed by the selection of spindle speed, and rapid approach to the first position, close to part. That is the block where the tool wear offset will be activated - *on the way* towards the first position:

```
N34 T0200 M42
N35 G96 S190 M03
N36 G00 G41 X12.0 Z0 T0202 M08
N37 G01 X1.6 F0.008
...
```

Also note that no G00 is required for a block containing tool indexing with zero wear offset entry. Main advantage of programming the tool offset simultaneously with a motion is the elimination of any jumpy motion; at the same time, no overtravel condition will result, even if the wear offset is unusually large. Wear offset amount will only *extend* or *shorten* the programmed rapid approach, depending on the actual offset amount stored.

Generally, the tool wear offset register number is entered *before* or *during* rapid approach motion.

### ◆ Offset Change

Most lathe programs require one offset for each tool. In some cases, however, the program can benefit if two or even more offsets are assigned to the same tool. Needless to say, only one offset can be active at one time. The current offset can be changed to another offset for the same tool to achieve additional flexibility. This is useful mainly in those cases when individual diameters or shoulder lengths must be machined to exact tolerances. Any new offset must be programmed *without* a cancellation of the previous one. In fact, this is the preferable method for changing from one offset to another. Reasoning is simple - remember that any offset change serves a purpose only during actual cutting. Offset cancellation could be unsafe if programmed during cutting motion. This is a very important - *and largely unexplored programming technique* - that some detailed examples are justified.

## MULTIPLE OFFSETS

A large number of jobs machined on CNC lathes requires very high precision. High precision often requires tolerance ranges as specified in the engineering drawing and these ranges may have quite a variety. Since a single offset per tool is often not enough to maintain these tolerances, two or more wear offsets are required for a single tool.

The following three examples are designed to present a complete understanding of the advanced subject covering multiple offsets. The same basic drawing will be used for all examples - only the tolerances will vary.

Project itself is very simple - program and machine three diameters as per drawing, and maintain tolerances at the same time. One rule at the beginning - the program *will not use* the middle tolerance of the X or Z value. This is an unfortunate practice that makes changes to the program much more difficult at a later time, if drawing tolerances are changed by engineers or designers.

In the drawings, the following tolerances can be found:

- Tolerances only on the diameter
- Tolerances only on the shoulders (faces)
- Tolerances on the diameters and shoulders

### ◆ General Approach

The tolerances in all three examples are for training purposes only and will be much smaller in reality. All chamfer tolerances are  $\pm 0.010$ , and non-specified tolerances are  $\pm 0.005$ . That will allow concentration on the project. Material is a  $\varnothing 1.5$  inch aluminum bar and three tools are used:

T01	For the face and rough contour
T03	For the finishing of contour to size
T05	0.125 wide part-off tool

Individual programmer's skill will determine the final result - how many offsets to program and where to enter them within the program. At the machine, CNC operator must store the correct setting amounts for each offset. In all cases, the main objective will be to aim for the middle tolerance in machining, *not in programming*.

### ◆ Diameter Tolerances

Drawing in Figure 34-3 shows the sample part with variable tolerances applied only on *diameters*.

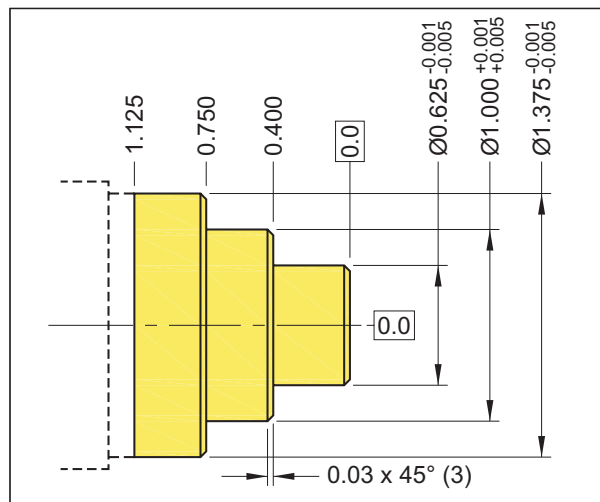


Figure 34-3

Multiple offsets - example for diameters only - O3401

Programming solution is to include *two* offsets for finishing, for example, T0313 and T0314. In the control, correct offset amounts have to be set before machining - the ideal amounts for middle tolerance are shown:

```
13 X-0.003 Z0.000
14 X+0.003 Z0.000
```

Note that the Z-offset (it controls shoulder lengths) must be the *same* for both wear offsets.

Here is the complete program - O3401:

```
O3401
(1.5 ALUMINUM BAR - EXTEND 1.5 FROM JAWS)

(T01 - FACE AND ROUGH TURN)
N1 G20
N2 G50 S3000 T0100
N3 G96 S500 M03
N4 G00 G41 X1.7 Z0 T0101 M08
N5 G01 X-0.07 F0.005
N6 Z0.1
N7 G00 G42 X1.55
N8 G71 U0.1 R0.02
N9 G71 P10 Q17 U0.04 W0.004 F0.01
N10 G00 X0.365
N11 G01 X0.625 Z-0.03 F0.003
N12 Z-0.4
N13 X1.0 C-0.03 (K-0.03)
N14 Z-0.75
N15 X1.375 C-0.03 (K-0.03)
N16 Z-1.255
N17 U0.2
N18 G00 G40 X5.0 Z5.0 T0100
N19 M01

(T03 - FINISH TURN)
N20 G50 S3500 T0300
(-- OFFSET 00 AT THE START OF THE TOOL -----)
N21 G96 S750 M03
N22 G00 G42 X1.7 Z0.1 T0313 M08
(-- OFFSET 13 FOR THE 0.625 DIAMETER -----)
N23 X0.365
N24 G01 X0.625 Z-0.03 F0.002
N25 Z-0.4
N26 X1.0 C-0.03 (K-0.03) T0314
(-- OFFSET 14 FOR THE 1.0 DIAMETER -----)
N27 Z-0.75
N28 X1.375 C-0.03 (K-0.03) T0313
(-- OFFSET 13 FOR THE 1.375 DIAMETER -----)
N29 Z-1.255
N30 U0.2
N31 G00 G40 X5.0 Z5.0 T0300
(-- OFFSET 00 AT THE END OF TOOL -----)
N32 M01

(T05 - 0.125 WIDE PART-OFF)
N33 T0500
N34 G97 S2000 M03
N35 G00 X1.7 Z-1.255 T0505 M08
N36 G01 X1.2 F0.002
N37 G00 X1.45
N38 Z-1.1825
N39 G01 X1.315 Z-1.25 F0.001
N40 X-0.02 F0.0015
N41 G00 X5.0
N42 Z5.0 T0500 M09
N43 M30
%
```

This is the complete program, using all three tools required. Since T01 and T05 do not change for any forthcoming examples, only T03 will be shown from now on.



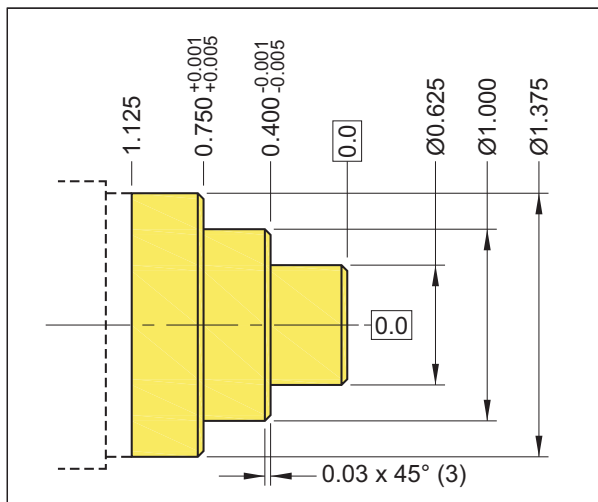


Figure 34-4

Multiple offsets - example for shoulders only - O3402

### ◆ Shoulder Tolerances

Example drawing shown in Figure 34-4 illustrates the same basic sample part with variable tolerances specified only on the *shoulders*.

Programming solution is to include *two* offsets for finishing cut, for example T0313 and T0314. In the control, their amounts have to be set before machining - the ideal amounts for middle tolerance are shown:

```
13 X0.0000 Z+0.0030
14 X0.0000 Z-0.0030
```

Note that in this case, the X-offset (it controls size of the diameters) must always be the same for both offsets. Here is T03 for program O3402:

O3402

```
...
(T03 - FINISH TURN)
N20 G50 S3500 T0300
(-- OFFSET 00 AT THE START OF TOOL -----)
N21 G96 S750 M03
N22 G00 G42 X1.7 Z0.1 T0313 M08
(-- OFFSET 13 FOR THE 0.4 SHOULDER -----)
N23 X0.365
N24 G01 X0.625 Z-0.03 F0.002
N25 Z-0.4
N26 X1.0 C-0.03 (K-0.03)
N27 Z-0.75 T0314
(-- OFFSET 14 FOR THE 0.75 SHOULDER -----)
N28 X1.375 C-0.03 (K-0.03)
N29 Z-1.255
N30 U0.2
N31 G00 G40 X5.0 Z5.0 T0300
(-- OFFSET 00 AT THE END OF TOOL -----)
N32 M01
...
```

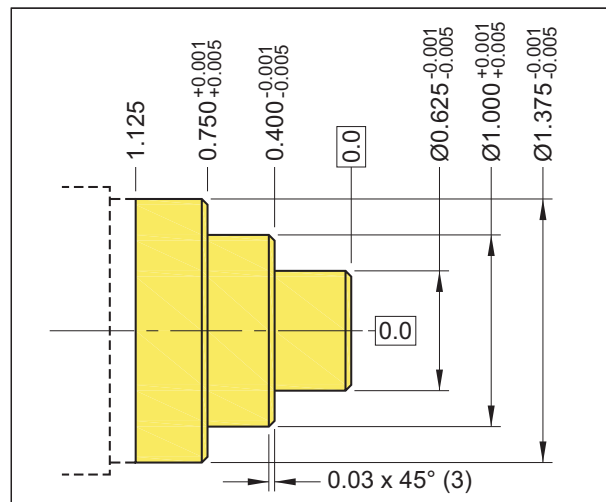


Figure 34-5

Multiple offsets - example for diameters and shoulders - O3403

### ◆ Diameter and Shoulder Tolerances

Example drawing shown in Figure 34-5 illustrates the same basic sample part with variable tolerances specified on both the *diameters* and *shoulders*.

Programming solution is to include *four* offsets for finishing, for example T0313, T0314, T0315 and T0316. In the control, their amounts have to be set before machining - the ideal amounts for middle tolerance are shown:

```
13 X-0.0030 Z+0.0030
14 X+0.0030 Z+0.0030
15 X+0.0030 Z-0.0030
16 X-0.0030 Z-0.0030
```

This is the most intensive version. Not only it is extremely important *where exactly* the offsets appear in the program, but their actual input amount is also critical.

Note that the four X-offsets (they control size of the diameters) tie up with the four Z-offsets (which control the length of shoulders). Here is T03 for program O3403:

O3403

```
...
(T03 - FINISH TURN)
N20 G50 S3500 T0300
(-- OFFSET 00 AT THE START OF TOOL -----)
N21 G96 S750 M03
N22 G00 G42 X1.7 Z0.1 T0313 M08
(-- OFFSET 13 FROM Z OVER TO Z UNDER ONLY ---)
N23 X0.365
N24 G01 X0.625 Z-0.03 F0.002
N25 Z-0.4
N26 X1.0 C-0.03 (K-0.03) T0314
(-- OFFSET 14 FROM X UNDER TO X OVER ONLY ---)
N27 Z-0.75 T0315
(-- OFFSET 15 FROM Z UNDER TO Z OVER ONLY ---)
```

```

N28 X1.375 C-0.03 (K-0.03) T0316
(-- OFFSET 16 FROM X OVER TO X UNDER ONLY ---)
N29 Z-1.255
N30 U0.2
N31 G00 G40 X5.0 Z5.0 T0300
(-- OFFSET 00 AT THE END OF TOOL -----)
N32 M01
...

```

CNC operator must always be aware of the existence of multiple offsets in a program as well as programmer's reason for using them. Initial settings are always critical, so are all changes during machining. As can be seen in the first two examples, programs O3401 and O3402, one group of offsets must always remain the same (*X or Z offsets*). For instance, in program O3401, diameters are controlled by offsets 03 and 13. That means the Z-offset setting must be the same - *always!* That also means, if there is a need to shift the shoulders 0.002 to the left, for example, *all shoulders must be shifted by the same amount:*

```

13  X-0.0030  Z-0.0020
14  X+0.0030  Z-0.0020

```

Failure to do that will result in inaccurate dimensions.

## OFFSET SETTING

The OFFSET screen selected by pressing a key on the control panel will initially display tool geometry and tool wear offsets. They are identical, except the title at the screen top. A typical display will resemble this screen layout (no offsets set):

OFFSET (GEOMETRY)				
NO.	X-AXIS	Z-AXIS	RADIUS	TIP
01	0.0000	0.0000	0.0000	0
02	0.0000	0.0000	0.0000	0
03	0.0000	0.0000	0.0000	0
...	...	...	...	...

X-axis and Z-axis are often shown just as X and Z, *Radius* is shown as R, and *Tip* is shown as T.

The NO. is the offset number, either the first pair of the T address - for the *Geometry* offset, or the second pair - for the *Wear* offset. X-axis and Z-axis are the columns where actual offset amounts are entered for each number, the *Radius* and *Tip* columns are only used if tool nose radius offset is programmed. In that case, *Radius* will be the tool nose radius and *Tip* will be an arbitrary number, as defined by Fanuc, specifying the tool tip orientation. This subject has been described starting on page 275.

## FUNCTIONS FOR GEAR RANGES

A number of CNC lathes are designed to work in several ranges of gear engagement. This machine feature enables the programmer to coordinate required spindle speed with specific power requirements of the machine. As a general rule, the higher the requirement for spindle speed, the lower the maximum available power rating will be, and vice versa. Ranges of spindle speed and power ratings for each range are determined by the machine manufacturer, and must never be changed.

Depending on the CNC lathe size, one, two, three, or four gear ranges may be available. Small lathes, or those designed with ultra high spindle speeds, may have no programmable gear range at all, which means only a single default gear range is available. Very large lathes may have all four gear ranges - and the maximum available spindle speed is usually low in comparison. The most common average is two gear ranges.

Miscellaneous functions for gear ranges may vary, but are typically M41, M42, M43 and M44, and assume the definition relative to the number of gear ranges available:

Range	Number of available ranges			
	1	2	3	4
Low	-	M41	M41	M41
Medium Low	-	-	-	M42
Medium	-	-	M42	-
Medium High	-	-	-	M43
High	-	M42	M43	M44

Once a certain gear range is selected, the spindle speed range is limited. If the exact range of spindle speed is important, always make an effort to find out the available spindle speeds in each range. Don't be surprised to find out that on most CNC machines, one rpm (1 r/min) is very rare. Typical lowest spindle speed may be around 20 to 30 r/min. Also, don't be surprised to find that there is an overlap, often quite large, for spindle speeds in two ranges. For example, if the *Gear 1* has a range 20 to 1400 r/min, *Gear 2* may have a range of 750 to 2500 r/min. When using spindle speeds available in either range, such as 1000 r/min, selection of gear range is not critical, but low gear range will produce more power.

Here is an actual, although unrelated, example:

```

Low gear range:    20 - 1075 r/min (M41)
High gear range:   70 - 3600 r/min (M42)

```

## AUTOMATIC CORNER BREAK

In CNC turning and boring, there are occasions where the cut from a shoulder to a diameter (or from a diameter to a shoulder) requires a corner break. Breaking a sharp corner is a common practice when machining between shoulders and diameters. Many engineering drawings specify that all sharp corners are to be broken, often without suggesting their size. It is up to the programmer to decide, usually within the range of 0.005 to 0.020 inches (0.125 to 0.500 mm). The required corner break may be either a *chamfer at a 45° angle*, or a *90° sweep blend radius* - both usually small. If the size of corner break is specified, then it must be applied. Corner breaking has three practical reasons:

- **Functionality**  
... for strength, ease of assembly, and clearances
- **Safety**  
... sharp corners are dangerous
- **Appearance**  
... the finished part looks better

In lathe work, many corner breaks apply to cuts between a shoulder and the adjacent diameter (the cut takes a 90° turn in one axis at a time). The start and end points calculation is not difficult but can be time consuming for some jobs, such as shaft turning with many different diameters.

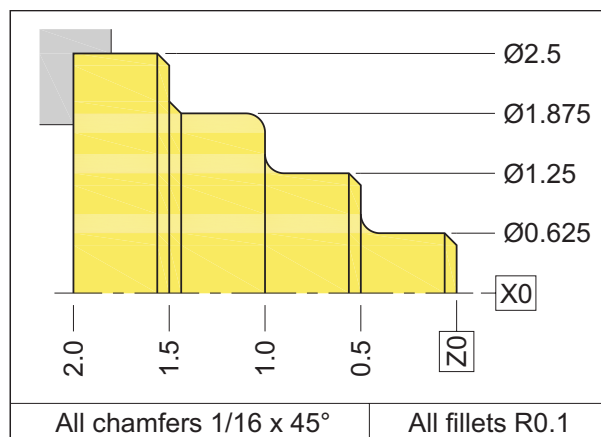


Figure 34-6

Example for an automatic corner break (chamfers and radii)

Drawing in *Figure 34-6* shows a simple external part that contains several corners that will benefit from automatic corner break programming feature - not all corners in the drawing qualify (!).

Compare the two methods, to better understand the differences applied in programming. If the programmer *does not* use automatic corner break, each contour change point must be calculated manually and the result will be rather long program O3404:

### O3404 (MANUALLY CALCULATED CORNER BREAK USED)

```

...
N51 T0100
N52 G96 S450 M03
N53 G00 G42 X0.3 Z0.1 T0101 M08
N54 G01 X0.625 Z-0.0625 F0.003
N55 Z-0.4
N56 G02 X0.825 Z-0.5 R0.1
N57 G01 X1.125
N58 X1.25 Z-0.5625
N59 Z-0.9
N60 G02 X1.45 Z-1.0 R0.1
N61 G01 X1.675
N62 G03 X1.875 Z-1.1 R0.1
N63 G01 Z-1.4375
N64 X2.0 Z-1.5
N65 X2.375
N66 X2.55 Z-1.5875
N67 U0.2
N68 G00 G40 X10.0 Z5.0 T0100
N69 M01

```

Only the finished contour is shown (no facing), starting at a *selected* clearance of Z0.1, with *calculated* X-diameter at X0.3. Each contour change point has to be carefully calculated. At the contour end, the last chamfer has been completed at a *selected* clearance of 0.025 above the top diameter, at X2.55, and *calculated* Z-axis at Z-1.5875.

As always in manual work, the possibility of errors can be significant. For example, one very common error in this type of programming is the target value of X-axis. In turning, it is easy to forget to double the chamfer or radius value (or half it for boring). The result is that block N56 may be:

N56 G02 X0.725 Z-0.5 R0.1 (ERROR IN X)

instead of the *correct* block

N56 G02 X0.825 Z-0.5 R0.1 (X IS CORRECT)

Remember: X = diameter. So what can be done in the program in order to implement the automatic corner break?

Fanuc control system offers two programming methods that relate to automatic corner breaking on lathes:

- Chamfering method ... for a 45° chamfer
- Blend radius method ... for a 90° blend

Both methods work in a very similar manner and certain rules have to be observed in both cases.

### ◆ Chamfering at 45 Degrees

Automatic corner chamfering will always take place in the G01 mode, and two special vectors I and K are available for this purpose or a C-vector on some models.

For automatic chamfer generation, the vectors I and K specify the *direction and the amount of cut for the required chamfer*:

**'I' VECTOR**

is used to create a chamfer starting from the X-axis,  
into the X+Z-, X-Z-, X+Z+, or X-Z+ direction

**'K' VECTOR**

is used to create a chamfer starting from the Z-axis,  
into the Z-X+, Z-X-, Z+X+, or Z+X- direction

The I and K vector definition is illustrated in *Figure 34-7*.

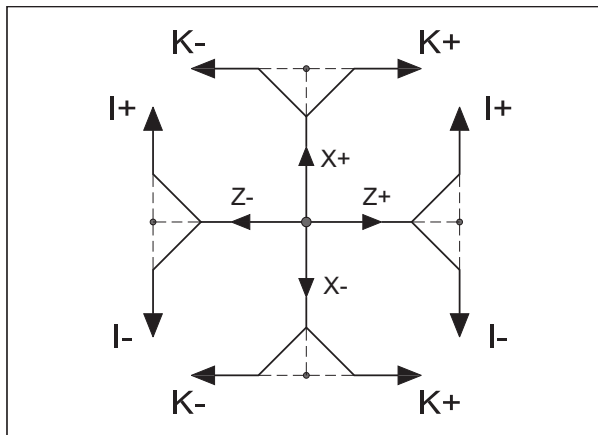


Figure 34-7

Vectors I and K for automatic corner chamfering

When the control system encounters a block containing chamfering vector I or K, it will automatically *shorten* the active programmed tool path length by the amount of the I or K vector, as specified in the program. If not sure whether the I or the K vector should be programmed for automatic chamfering, consult the above illustration, or apply the following rules:

*The vector I indicates the chamfering amount and motion direction when the tool motion is in the order of Diameter-Chamfer-Shoulder, which means tool is cutting along the Z-axis before the chamfer. The chamfer direction can only be from the Z-axis towards the X-axis, with the I vector programmed between:*

```
G01 Z-1.75 I0.125      (CUTTING ALONG Z AXIS)
X4.0                   (CONTINUING IN X AXIS AFTER CHAMFER)
```

*Vector K indicates the chamfering amount and motion direction when the tool motion is in the order of Shoulder-Chamfer-Diameter, which means tool is cutting along the X-axis before the chamfer. The chamfer direction can only be from the X-axis towards the Z-axis, with the K vector programmed between:*

```
G01 X2.0 K-0.125       (CUTTING ALONG X AXIS)
Z-3.0                  (CONTINUING IN Z AXIS AFTER CHAMFER)
```

In either case, the sign of I or K vector defines the chamfer cutting direction, within the coordinate system:

- Positive value of I or K vector indicates the chamfering direction into the plus direction of the axis *not* specified in the chamfering block
- Negative value of I or K vector indicates the chamfering direction into the minus direction of the axis *not* specified in the chamfering block

Values of I and K vectors are always single values (*i.e.*, radius values, *not* diameter values).

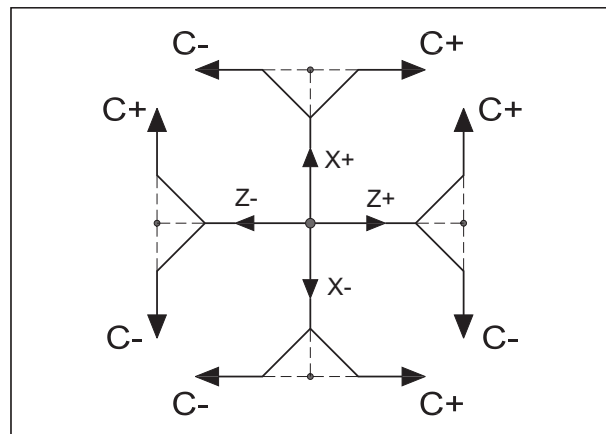


Figure 34-8

Vectors C for automatic corner chamfering

Many latest controls use vectors C+ and C- that replace the I+, I-, K+ and K- vectors - *Figure 34-8*. This is a much simpler programming method and its applications are the same as for the blend radius R, described shortly. There is no distinction between axes vector selection, just the specified direction:

- The C vector is used
  - ... to create a chamfer starting from the X-axis,  
into the X+Z-, X-Z-, X+Z+, or X-Z+ direction
  - or -
  - ... to create a chamfer starting from the Z-axis,  
into the Z-X+, Z-X-, Z+X+, or Z+X- direction

If the control unit allows C+ or C- vectors, the programming is much easier, as long as the motion direction is watched carefully. The two previous examples will be:

```
G01 Z-1.75 C0.125      (CUTTING ALONG Z AXIS)
X4.0                   (CONTINUING IN X AXIS AFTER CHAMFER)
```

```
G01 X2.0 C-0.125       (CUTTING ALONG X AXIS)
Z-3.0                  (CONTINUING IN Z AXIS AFTER CHAMFER)
```

As was the case with I and K vectors, the C vector is also specified as a single value *per side*, not per diameter.

### ◆ Blend Radius at 90 Degrees

A blend 90° sweep radius between a shoulder and diameter (or vice versa) is programmed in a similar way as the automatic 45° chamfer using the C vector. *It also takes place exclusively in the G01 mode!* Only one special vector R is used. For automatic blend radius, the vector specifies the *direction and amount of cut for the radius*:

- The R vector is used

... to create a blend radius starting from the X axis, into the X+Z-, X-Z-, X+Z+, or X-Z+ direction

- or -

... to create a blend radius starting from the Z axis, into the Z-X+, Z-X-, Z+X+, or Z+X- direction

The R vector definition is illustrated in *Figure 34-9*.

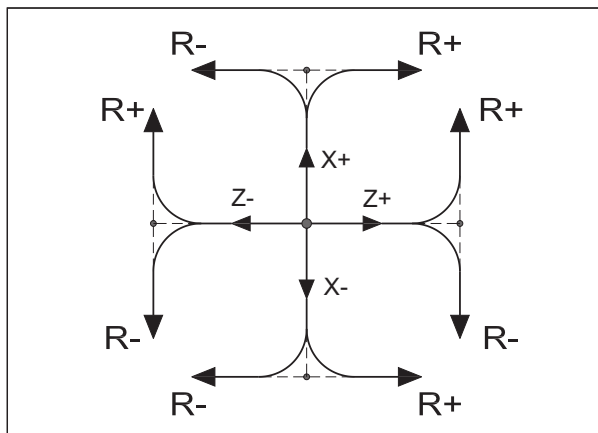


Figure 34-9

Vector R for automatic corner rounding (blend radius)

When the control system encounters a program block containing blend radius vector R, it will automatically *shorten* the current programmed tool path length by the R vector amount, as specified in the program. If not sure whether the R vector should be programmed for automatic blend radius, consult the above illustration or apply the following rule:

*Vector R* indicates the *radius amount and motion direction* when the tool motion is in the order of *Shoulder-Radius-Diameter*; which means tool is cutting along the X-axis before the radius. The same vector is also used when the *radius amount and motion direction* is in the opposite order of *Diameter-Radius-Shoulder*, which means cutting along the Z-axis before the radius.

Radius deviation can be from the X-axis towards the Z-axis, when the R vector is programmed:

```
G01 X2.0 R-0.125      (CUTTING ALONG X-AXIS)
Z-3.0                (CONTINUING IN Z AXIS AFTER RADIUS)
```

Radius direction can also be from the Z-axis towards the X-axis, when the R vector is programmed:

```
G01 Z-1.75 R0.125    (CUTTING ALONG Z AXIS)
X4.0                 (CONTINUING IN X AXIS AFTER RADIUS)
```

In either case, the sign of R vector defines the radius cutting direction, within the coordinate system:

- Positive value of R vector indicates the radius direction into the *plus* direction of the axis *not* specified in the radius block
- Negative value of R vector indicates the radius direction into the *minus* direction of the axis *not* specified in the radius block

### ◆ Programming Conditions

Breaking corners automatically makes programming for modern CNC lathes a lot easier, as only drawing dimensions are used and no external manual calculations are necessary. Regardless of whether program contains vectors I or K or C for chamfering, or vector R for blend radius corner, the basic conditions and general rules are very similar:

- Chamfer or radius must be fully contained in a single quadrant - 90° only
- Chamfers must have a 45° angle and radii must have a 90° sweep angle between a shoulder and diameter or a diameter and shoulder
- Amounts of chamfering vectors I and K or C, as well as radius vector R, are always single values - meaning *per side* values, not diameter values
- Direction of cut before corner rounding must be perpendicular to the direction of cut after rounding, along one axis only
- Direction of cut following the chamfer or radius must continue along a single axis only, and must have the length equivalent to at least the chamfer length or the radius amount - the cutting direction cannot reverse
- Both chamfering and blend radius corner breaking takes place in G01 mode (linear interpolation mode)
- When writing CNC program, only the known intersection from the drawing - *the sharp point* - is required. That is the point between shoulder and diameter, without the chamfer or radius being considered

These rules apply equally to turning and boring CNC lathe operations. Study them carefully to avoid problems at the machine later.

### ◆ Programming Example

Following program (O3405) combines both chamfering and blend radius vectors into a complete example. The same drawing is used for this version, as for the traditional method, illustrated earlier in *Figure 34-6*.



In order to fully appreciate all differences between the two programming methods (both are technically correct), compare the following program O3405 with the earlier program O3404. I and K vectors are used for chamfering, as they are more difficult than the C vectors:

#### O3405 (AUTOMATIC CORNER BREAKS USED)

```

...
N51 T0100
N52 G96 S450 M03
N53 G00 G42 X0.3 Z0.1 T0101 M08
N54 G01 X0.625 Z-0.0625 F0.003
N55 Z-0.5 R0.1
N56 X1.25 K-0.0625
N57 Z-1.0 R0.1
N58 X1.875 R-0.1
N59 Z-1.5 I0.0625
N60 X2.375
N61 X2.55 Z-1.5875
N62 U0.2
N63 G00 G40 X10.0 Z5.0 T0100
N64 M01

```

Although this program is shorter, the five blocks saved in the program offer only a small benefit. The real benefits mean no G02s, no G03s, no start/end points of chamfers or radiuses, no center point calculations.

Except for the contour beginning and end, this type of programming greatly enhances program development and allows for very fast and easy changes during machining, if necessary. If a chamfer or a blend radius is changed in the drawing, only a *single value* has to be changed in the program, without any recalculations. Of course, the rules and conditions mentioned earlier must be always observed. The main benefit of automatic corner breaking are the ease of changes and the absence of manual calculations.

## ROUGH AND FINISHED SHAPE

The vast majority of material removal on CNC lathe is done by using various cycles, described in detail in the next chapter. These cycles require input of data that is based on machining knowledge, such as depth of cut, stock allowance, speeds and feeds, etc.

Rough and finished shapes often require manual calculations, using algebra and trigonometry. These calculations should be done on separate sheets of paper, rather than in the drawing itself. That way, the work is better organized. Also, if there is a change later, for example, an engineering design change, it is easier to keep track of what is where.

### ◆ Rough Operations

A great part of lathe machining amounts to removal of excessive stock to create a part, almost completed. This kind of machining is generally known as *roughing*, rough turning, or rough boring. As a machining operation, rough-

ing does not produce a high precision part, that is not the purpose of roughing. Its main purpose is to remove unwanted stock efficiently, which means fast and with maximum tool life, and leave suitable all-around stock for finishing. Cutting tools used for roughing are strong, usually with a relatively large nose radius. These tools have to be able to sustain heavy depths of cut and high cutting feeds. Common diamond shaped tools suitable for roughing are 80° inserts (up to 2+2 cutting corners), and trigon inserts (up to 3+3 cutting corners). 2+2 or 3+3 means on 2 or 3 cutting edges on each side of the insert. Not all inserts can be used from both sides. Figure 34-10 shows some typical tools and orientation for rough turning and boring.

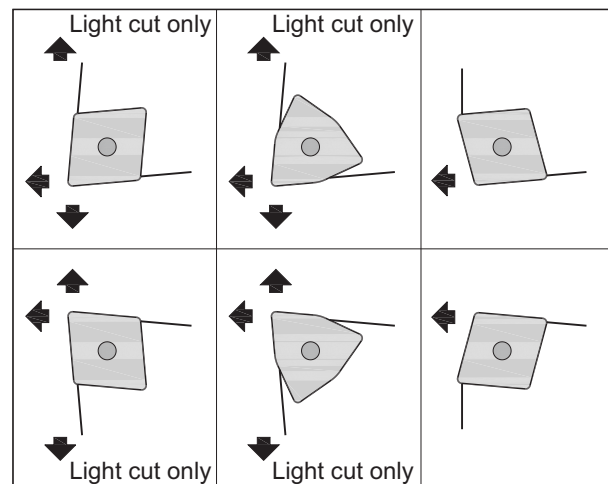


Figure 34-10

Tool orientation and cutting direction for roughing.  
Upper row shows external tools, lower row shows internal tools.

Although a number of tools can be programmed in several directions, some directions are not recommended at all, or only for light or medium light cuts.

In practice, always follow one basic rule of machining - this rule is valid for all types of machines:

Always do heavy operations before light operations

This basic rule means that *all* roughing should be done before the first finishing cut is programmed. The reason here is to prevent a possible shift of the material during roughing, after some finishing had already been done.

For example, the requirement is to rough and finish both external and internal diameters. If the above rule is applied to these operations, roughing outside of the part will be first, then roughing inside of the part, and only then applying the finishing cuts. It really does not matter whether the rough machining is done first externally or internally, as long as it gets done before any finish cuts, which also can be in either order.

Tool wear can be minimized if the depth of cut is sufficient and the cutting radius gets 'under the skin' of the material, usually during the first cut. Coolant is usually a must for most materials and should be applied *before* the tool actually contacts the part.

### ◆ Finish Operations

Finish operations take place as the final cutting motions, after most of the stock has been removed (roughed out), leaving only a small amount of overall stock for finishing. Finishing cutting tool can have smaller nose radius and, for even a better surface finish, higher spindle speeds and lower cutting feeds are typical.

Many different tools can be used for finishing operations as well, but the most typical finishing tools are two diamond shaped inserts, with a 55° and 35° insert angle. Their shape, common orientation and cutting directions are shown in *Figure 34-11*.

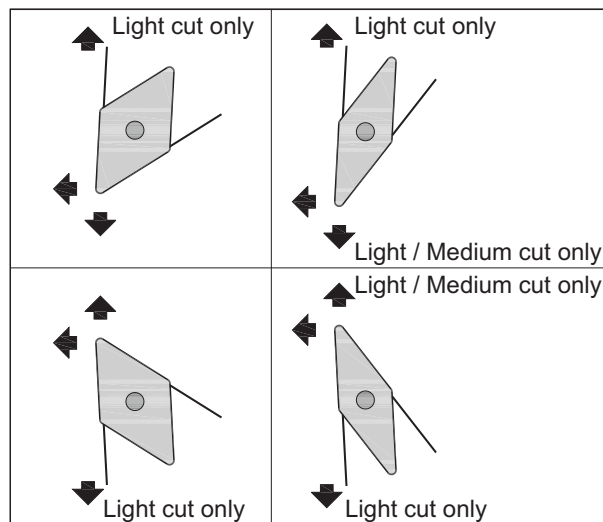


Figure 34-11

Tool orientation and cutting direction for finishing with common lathe tools. Upper row shows external tools, lower row shows internal tools.

Note that some cutting directions are only recommended for light or medium cuts. Why? The answer has a lot to do with the amount of material (stock) the tool removes in the specified direction.

### ◆ Stock and Stock Allowance

Material used for machining is often called *stock*. When the tool removes stock to cut a desired shape, it can only handle a certain amount of it at a time. Insert shape, its orientation toward the part and its cutting direction, the insert size and thickness, all have a profound effect on the allowable stock to be removed. This is especially important in semifinishing and finishing operations. Stock allowance

specifies the amount of material left for these fine operations. If too much material or too little material is left to be cut during finishing, part accuracy and surface finish quality will suffer. Also, carefully consider not just a stock allowance overall on the part, but *individual stock allowances* for the X and Z axes!

As before, there is a general rule of thumb, that on the X-axis, that is for cutting diameters, leave the stock *equivalent to or slightly larger* than radius of the subsequent finishing tool. For example, if a 0.031 inch tool nose radius (0.80 mm) is used for finishing, leave 0.030 to 0.040 inch stock (about 1 mm). That is the physical stock, the actual stock amount assigned *per side*, not on diameter!

The amount of stock left on Z-axis (typically for facing shoulders at 90°) is much more critical. If cutting along the positive X-axis only (for turning), or the negative X-axis only (for boring), with a tool that has a lead angle of 3° to 5°, do not leave more than 0.003 to 0.006 inch (0.080 to 0.150 mm) on any straight shoulder. *Figure 34-12* shows the effect of too much stock allowance for certain cutting directions and a method to eliminate it.

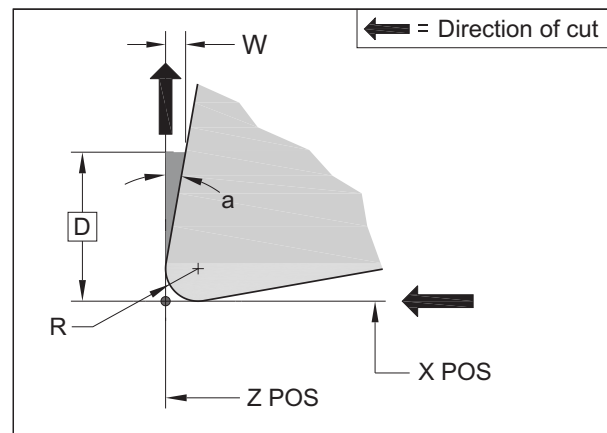


Figure 34-12

Effect of stock allowance W on depth of cut D

In the illustration, actual depth of cut D at the face Z POS, is determined by the amount of stock W. To calculate the depth D, use this formula:

$$D = \tan \frac{A}{2} \times R + \frac{W}{\tan A} + R$$

where ...

- D = Actual depth of cut at the face
- A = Lead angle of the insert
- R = Radius of the insert
- W = Stock left on face for finishing
- X POS = Target position for the X-axis
- Z POS = Target position for the Z-axis

Illustration applies equally to boring, when the X-axis direction is opposite the one shown. To understand better the consequences of a heavy stock left on the face, evaluate this example:

☛ **Example:**

Amount of stock left on face is 0.030, tool nose radius is 0.031, and tool lead angle is  $3^\circ$ :

$$W = 0.030, \quad R = 0.031, \quad A = 3$$

There is enough data available to calculate the unknown depth D, using the above formula:

$$D = \tan 3/2 \times 0.031 + 0.030 / \tan 3 + 0.031$$

$$D = 0.60425$$

For an insert with a  $\varnothing 0.500$  inch inscribed circle (such as DNMG-432, for example), the actual depth of cut at the face will be 0.60425 - *more than any reasonable amount!*

Since the earlier suggestion was no more than 0.006, recalculate the example for the largest depth, if the  $W=0.006$ :

$$D = \tan 3/2 \times 0.031 + 0.006 / \tan 3 + 0.031$$

$$D = 0.14630$$

That is a more reasonable depth of cut at the face, so the Z axis stock allowance of 0.006 can be used. For facing in the opposite X direction or for not unidirectional faces, leave stock much bigger, usually close to the tool radius.

## PROGRAMMING A RECESS

Another very important aspect of programming for CNC lathes is the change of cutting direction. Normally, program a tool motion in such a way that the motion direction from the starting point will be:

- Positive X direction for external machining  
... and / or ...  
Negative Z direction for external machining
- Negative X direction for internal machining  
... and / or ...  
Negative Z direction for internal machining

There are also *back turning* or *back boring* operations used in CNC programming, but these are just related and less common variations of the common machining. In the most common machining on CNC lathes, any change of direction in a single axis into the material constitutes an undercut, a cavity, or more commonly known - a *recess*.

A recess is commonly designed by the engineers to relieve - or undercut - a certain portion of the part, for example, to allow a matching part to fit against a shoulder, face, or surface of the machined part.

In CNC lathe programming, a recess can be machined very successfully with any tool that is used with the proper depth of cut, and a *suitable back angle clearance*. It is the second requirement that will be looked at next.

Figure 34-13 shows a simple drawing of a roller. In the middle of the object, there is an undercut (recess) between the  $\varnothing 1.029$  and the  $\varnothing 0.939$ . The objective is to calculate, *not to guess*, what is the maximum back angle tool that can be used for cutting this recess in a single operation.

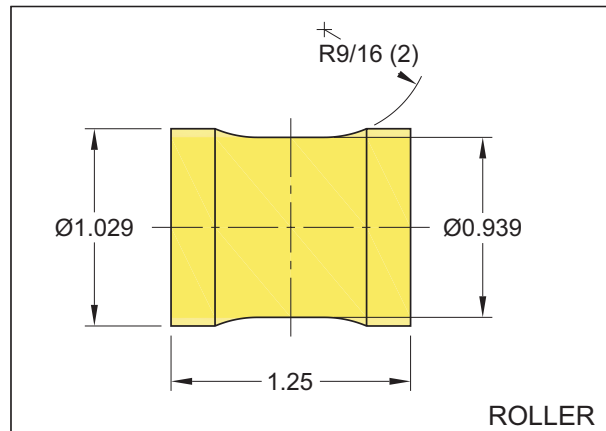


Figure 34-13

Back angle clearance calculation example

The first step is to consider the drawing - that is always the given and unchangeable source of data. Calculated difference between the diameters, and the recess radius will be required. Figure 34-14 illustrates the generic details of the *provided data* (except the angle  $b$ ) from the drawing.

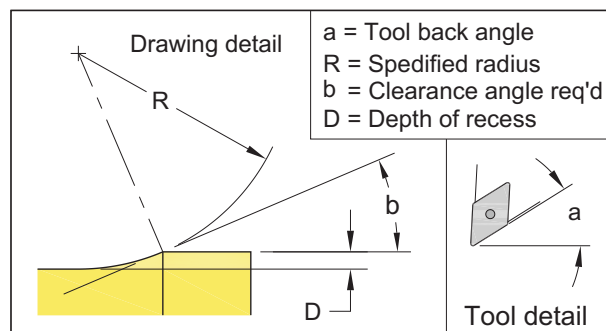


Figure 34-14

Data required to calculate angle 'b'

Formula required to calculate the angle  $b$  uses simple trigonometric formula. First, calculate the *depth* of the recess D, which is nothing more than one half of the difference between the two given diameters:

$$D = \frac{\text{LARGE DIA} - \text{SMALL DIA}}{2}$$

Once the recess depth  $D$  is known, formula to calculate the angle  $b$  can be used:

$$b = \cos^{-1} \left( \frac{R - D}{R} \right)$$

For the example, calculation will be:

$$b = \cos^{-1} \left( \frac{0.5625 - 0.045}{0.5625} \right) = 23.07392$$

For actual machining, select a tool with the back angle  $a$  greater than the calculated angle  $b$ . For the illustrated drawing ( $23.07^\circ$  required clearance), the selected tool could be either a  $55^\circ$  diamond shape (back angle clearance  $a$  is  $30^\circ$  to  $32^\circ$ ), or a  $35^\circ$  diamond shape (back angle clearance  $a$  is  $50^\circ$  to  $52^\circ$ ) - both are greater than the calculated minimum clearance. The actual angles depend on the tool manufacturer, so a tooling catalogue is a good source of data.

This type of calculation is important for any recesses, undercuts and special clearances, whether programmed with the help of cycles or developed block by block. The example only illustrates one possibility, but can be used for any calculations where the back angle clearance is required.

## SPINDLE SPEED IN CSS MODE

From several earlier topics, remember that the abbreviation *CSS* stands for *Constant Surface Speed*. This CNC lathe feature will constantly keep recalculating the actual spindle speed in *revolutions per minute* (r/min), based on the programmed input of surface speed. The surface speed is programmed in *feet per minute* - ft/min (imperial system) or in *meters per minute* - m/min (metric system).

In the program, the 'per minute' input uses preparatory command G96, as opposed to the direct *r/min* input using command G97.

*Constant Surface Speed* is a powerful feature of the control system and without it, we would look back many years. There is a rather small problem associated with this feature, often neglected altogether, or at least not considered important enough. This rather 'small problem' will be illustrated in a simple program example.

This program example covers only a few blocks at the beginning, when the cutting tool approaches the part. That is enough data to consider the question that follows.

```
O3406
N1 G20 T0100
N2 G96 S450 M03
N3 G00 G41 X0.7 Z0 T0101 M08
N4 ...
```

*The question is this:* What is the actual spindle speed (in *r/min*), when block N2 is executed? Of course, the spindle speed is unknown at the moment. It cannot be known, unless the current *diameter*, the diameter where the tool is located at that moment, is also known. Control system keeps track of the current tool position at all times. So, when block N2 is executed, the actual spindle *r/min* will be calculated for the current diameter, *as stored in the control*, specified in the geometry offset entry. For the example, consider that the current diameter is 23.5 or X23.5.

From the standard *r/min* formula, the spindle speed calculated for 450 ft/min and  $\varnothing 23.5$  as 73 r/min is rather slow, but correct. At the next block, block N3, the tool position is rather close to the part, at diameter of 0.700 (X0.7). From the same standard formula, the spindle speed can be calculated for that diameter as 2455 r/min - considerably fast but also correct. The problem? There may not be one for every machine, but if ever there is a problem, the following solution will eliminate it.

The possible problem can be linked to the rapid motion from  $\varnothing 23.5$  to  $\varnothing 0.700$ . The actual travel distance (per side of part) is  $(23.5 - 0.700)/2$ , which is 11.400. During rapid travel motion, the cutting tool has to move 11.400 inches and - *at the same time* - change the spindle speed from a slow 73 r/min, to a fast 2455 r/min. Depending on the control system and its handling of such a situation, the tool may actually start cutting *at a slower spindle speed than was originally intended*.

If such a situation does happen and presents a problem, the only step that can be done is to *preprogram* the expected spindle speed in r/min, *before* the cutting tool approach motion, *then* switch to the constant surface speed (CSS) mode and continue:

```
O3407
N1 G20 T0100
N2 G97 S2455 M03 (R/MIN PRESET)
N3 G00 G41 X0.7 Z0 T0101 M08
N4 G96 S450 M03
N5 ...
```

What had been done requires more evaluation. What had been done is that the spindle was started at the final expected *r/min*, before the tool reaches the part, in block N2. In block N3, tool moves to the start of cut, while the spindle is already at the peak of programmed speed. Once the target position along the X-axis has been reached (block N3), the corresponding CSS mode can be in effect for all subsequent cuts.

This is an example that does not necessarily reflect everyday programming of CNC lathes. In this situation, some additional calculations have to be done, but if they solve the problem - they are worth the extra effort! Some CAD/CAM system can be set to do exactly that, automatically. If the current X position of the tool is unknown, estimate it.

## LATHE PROGRAM FORMAT

In review of the already presented examples, a certain consistency can be seen in the program output. This may be called a style, a format, a form, a template, structure, as well as several other terms. Each programmer develops his or her own style over a period of time. A consistent style is important for efficient program development, program changes and program interpretation.

### ◆ Program Format - Templates

Most examples have followed a certain program format. Note that each CNC lathe program begins with the G20 or G21 command and perhaps some cancellation codes. The block that follows is a tool selection, next is spindle speed data, etc. This format will not basically change from one job to another - it follows a certain consistent pattern which forms the basic *template* for writing the program.

### ◆ General Program Format

To view the format often enough will forge a mental image in the programmer's mind. The details that are not understood yet will become much clearer after acquiring some general understanding of the relationships and details used in various programming methods. Here is a suggested template (structure) for a CNC lathe program.

#### ➡ General Program Pattern - Lathe :

```

O.. (PROGRAM NAME)
N1 G20 G40 G99          (PROGRAM START UP)
N2 T..00 M4..          (TOOL AND GEAR RANGE)
N3 G97 S.. M03          (STABILIZE R/MIN)
N4 G00 [G41/G42] X.. Z.. T.. M08 (APPROACH)
N5 G96 S..              (CUTTING SPEED)
N6 G01 [X../Z..] F..    (FIRST CUTTING MOTION)
N7 ...
...
...                      (MACHINING)
...
N.. G00 [G40] X.. Z.. T..00 (TOOL CHG POSITION)
N.. M01                 (OPTIONAL STOP)
...
N.. M30                 (PROGRAM END)
%
```

This generic structure is good for most lathe programs. Feel free to adjust it as necessary. For example, not every job requires spindle speed stabilization, so block N3 will not be necessary. It also means that M03 rotation has to be moved to block N5. Take the general program structure as an example only, not as a fixed format.

### ◆ Approach to the Part

An important part of any lathe program structure is the method of approaching a revolving part. If the part is concentric, the approach can be similar to the *A* option in *Figure 34-15*. Although a facing cut is illustrated, the approach would be logically the same for a turning or a boring cut. Keep the starting point *SP* well above the diameter, at least 0.100 (2.5 mm) *per side* and more, if the actual diameter is not known *exactly*. The *B* option of tool approach is two single axes at a time. It is a variation of the first example, and the X-axis motion can be further split into a rapid and cutting motion, if required. Finally, the *C* option uses clearance in the Z-axis, far from the front face. Again, final motion toward the face can be split into separate rapid and linear motions.

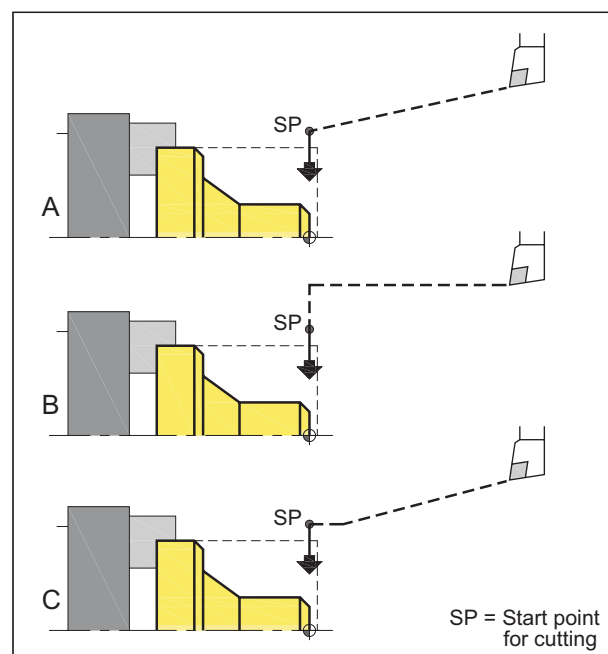


Figure 34-15

Safe approach to a part - example for a facing cut shown

There are many variations on these methods, too numerous to list. The main objective of considering approach towards the part in the first place is safety. A collision of a tool with a revolving part can have serious consequences.

Turning and boring is a large subject. Many other examples could have been included in this chapter. Other chapters in this book also cover turning and boring, but in a more specialized way, for example, turning and boring cycles. Examples that were presented in this chapter should be useful to any CNC lathe programming.



## 35

## LATHE CYCLES

In the last chapter, several lathe procedures described programming of a turning and boring tool path. A number of different techniques have been introduced, mainly describing the *finishing* tool path. Virtually no attention has yet been given to the removal of an excessive stock, in such operations as rough turning and rough boring. It is a subject in its own right and this chapter describes various methods of stock removal for roughing and finishing.

## STOCK REMOVAL ON LATHES

One of the most time consuming tasks in manual programming for a CNC lathe is the removal of an excessive stock, typically from a cylindrical material, known as rough turning or rough boring - or simply *roughing*.

To manually program a roughing tool path requires a series of coordinated rough passes, with one block of program for each tool motion. For roughing of a complex contour, such a method is extremely time consuming and very inefficient, as well as prone to errors. Some programmers try to sacrifice programming quality for speed, by leaving an *uneven* stock for finishing, causing the cutting tool to wear out prematurely. Surface roughness of the finished profile often suffers as well.

It is in the area of rough stock removal where the modern lathe controls are very useful and convenient. Almost all CNC lathe systems have a feature that allows the roughing tool path to be processed automatically, using *special cycles*. Roughing is not the only application for these cycles, there are also special cycles available for *threading* and *simple grooving*. The grooving and threading cycles are outside of this chapter, but will be covered in detail in the next three chapters.

### ◆ Simple Cycles

Fanuc and similar controls support a number of special lathe cycles. There are three rather simple cycles that have been part of Fanuc controls for quite a while. They first appeared with the early CNC units and were limited by the technological progress of the time. Various manuals and textbooks refer to them as the *Fixed Cycles* or *Simple Cycles* or even *Canned Cycles*, similar in nature to their cousins for drilling operations on CNC mills and machining centers. Two of these early cycles are used for turning and boring, the third cycle is a very simple threading cycle. This chapter covers the first two cycles.

### ◆ Complex Cycles

With the advancement of computer technology, control manufacturers have developed cutting cycles capable of very complex lathe operations and made them an integral part of the lathe control systems. These special cycles are called by Fanuc the *Multiple Repetitive Cycles*. Their major improvement over simple cycles is in their excellent flexibility. Some of these advanced cycles cover turning and boring, others grooving and threading.

Don't get misled by the description '*complex*' - these cycles are only complex in the mathematical sense and even then, only internally. They are complex within the control system only. In fact, these very advanced machining cycles are much easier to program than their simple predecessors. In addition, they can also be very easily changed at the machine control, to optimize them for best performance, right on the job.

## PRINCIPLES OF LATHE CYCLES

Similar to drilling operations for CNC machining centers, all cycles for lathes are based on the same technological principles. The programmer only enters the overall data (typically variable cutting parameters), and the CNC system will calculate the details of individual cuts. These calculations are based on the combination of the fixed and variable data. Return tool motions in all these cycles are automatic, and only the values to be changed are specified within the cycle call.

Simple cycles are designed exclusively to cut a straight cut, with no chamfers, tapers or radii and also with no undercuts. The simple cycles can only be used to cut vertically, horizontally, or at an angle, for taper cutting. These original cycles cannot do the same cutting operations as the more modern and advanced multiple repetitive cycles - for example, they cannot rough out a radius or change direction of the cutting. Simply, they cannot contour.

In the category of simple turning cycles, there are two that do allow removal of rough stock from a cylindrical or conical part. Each block of these cycles replaces four *regular* blocks of the part program. In the category of multiple repetitive cycles, there are several cycles designed for complex roughing, one for finishing, as well as cycles for grooving and threading. Multiple repetitive cycles are capable of some very complex contouring.

## G90 - STRAIGHT CUTTING CYCLE

Before going further, a reminder. Do not confuse G90 for lathes with G90 for machining centers. In turning, G90 is a lathe cycle, G90 is the absolute mode in milling:

G90 is absolute mode for milling,  
X and Z axes are absolute mode for turning

G91 is incremental mode for milling,  
U and W axes are incremental mode for turning

A cycle identified by G90 preparatory command (*Type A* group of G-codes) is called the *Straight Cutting Cycle (Box cycle)*. Its purpose is to remove excessive stock between the cutter start position and the coordinates specified by the X and the Z axes. Resulting cut is a straight turning or boring cut, *normally parallel* to the spindle centerline, and the Z-axis is the main cutting axis. As the cycle name suggests, G90 cycle is used primarily for removing stock in a rectangular fashion (box shape). The G90 cycle can also be used for a taper cutting. In *Figure 35-1*, the cycle structure and motions are illustrated.

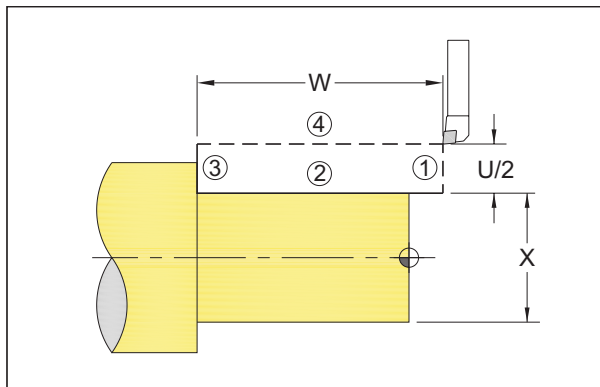


Figure 35-1

G90 simple cycle structure - straight cutting application

### ◆ Cycle Format

G90 cutting cycle has two predetermined programming formats. The first one is for straight cutting only, along the Z-axis, as illustrated in *Figure 35-1*.

#### ■ Format 1 :

G90 X(U).. Z(W).. F..

where ...

- X = Diameter to be cut
- Z = End of cut in Z position
- F = Cutting feedrate (usually *in/rev* or *mm/rev*)

The second format adds the parameter I or R to the block and is designed for taper cutting motions, with the dominance of the Z-axis - *Figure 35-2*.

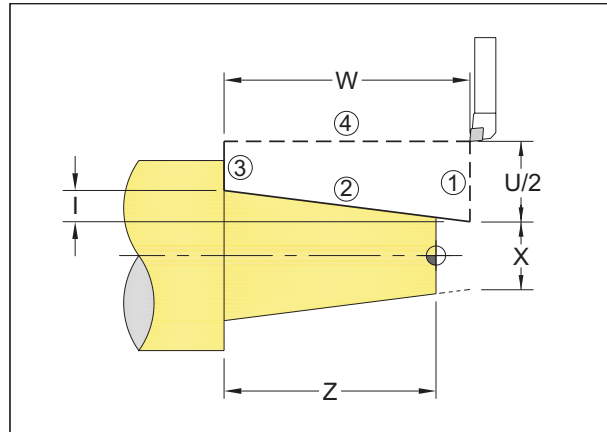


Figure 35-2

G90 cycle structure - taper cutting application

#### ■ Format 2 (two versions):

G90 X(U).. Z(W).. I.. F..  
G90 X(U).. Z(W).. R.. F..

where ...

- X = Diameter to be cut
- Z = End of cut in Z position
- I (R) = Distance and direction of the taper  
(I=0 or R=0 for straight cutting)
- F = Cutting feedrate (usually *in/rev* or *mm/rev*)

In both examples, the designation of axes as X and Z is used for absolute programming, indicating the tool position from program zero. The designation of axes as U and W is used for incremental programming, indicating actual travel distance of the tool from the current position. The F address is cutting feedrate, normally in *inches per revolution* or *millimeters per revolution*. The I address is used for taper cutting along the horizontal direction. It has an amount equivalent to *one half* of the distance from the diameter at taper end, to the diameter at taper beginning. The R address replaces the I address, and is available on *newer controls only* - its purpose is the same.

To cancel G90 cycle, all that is necessary to do is to use any motion command - G00, G01, G02 or G03. Commonly, it will be G00 rapid motion command:

G90 X(U) .. Z(W) .. I.. F..  
...  
...  
G00 ...

### ◆ Straight Turning Example

To illustrate a practical application of G90 cycle, study *Figure 35-3*. It shows rather a simple diameter turning, from a Ø4.125 inch stock down to a final Ø2.22 inch, over the length of 2.56 inches. There are no chamfers, no tapers, and no radii. This fact restricts the practical usefulness of the G90 cycle to a very simple roughing only, but still beats the manual alternative.

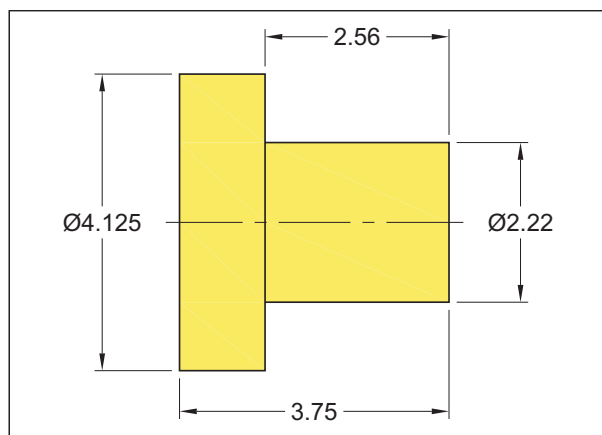


Figure 35-3

Example of G90 cycle in straight cutting - programs O3501 & O3502

Since G90 is a roughing cycle, the *depth* of each cut has to be selected first, then the *stock* amount left for finishing. To decide on the depth of each cut, find out how much stock is actually there to be removed from the diameter. The actual amount of stock is calculated *per side*, as a radius value, along the X-axis:

$$(4.125 - 2.22) / 2 = 0.9525$$

For a 0.030 stock per side for the finishing cut, the 0.030 will be subtracted from the total X-stock, so the total depth amount to remove will be 0.9225. Next is the selection of cut segmentation for the total depth. For five even cuts, each depth of cut will be 0.1845, for six cuts, 0.1538. Six cuts will be selected and 0.030 left per side, or 0.06 on the diameter - the first diameter will be X3.8175. Also, 0.005 stock allowance will be left on the face, so the Z-axis end of cut will be at Z-2.555. The actual clearance above the diameter and in front of the part will be the usual 0.100.

```
O3501
(G90 STRAIGHT TURNING CYCLE - ABSOLUTE)
N1 G20
N2 T0100 M41
N3 G96 S450 M03
N4 G00 X4.325 Z0.1 T0101 M08 (START POINT)
N5 G90 X3.8175 Z-2.555 F0.01 (PASS 1)
N6 X3.51 (PASS 2)
N7 X3.2025 (PASS 3)
N8 X2.895 (PASS 4)
N9 X2.5875 (PASS 5)
```

```
N10 X2.28 (PASS 6)
N11 G00 X10.0 Z2.0 T0100 M09
N12 M01 (END OF ROUGHING)
```

If preferred, use the incremental programming method. However, it is easier to trace the program progress with absolute coordinates than incremental distances. However, here is the incremental version:

```
O3502
(G90 STRAIGHT TURNING CYCLE - INCREMENTAL)
N1 G20
N2 T0100 M41
N3 G96 S450 M03
N4 G00 X4.325 Z0.1 T0101 M08 (START POINT)
N5 G90 U-0.5075 W-2.655 F0.01 (PASS 1)
N6 U-0.3075 (PASS 2)
N7 U-0.3075 (PASS 3)
N8 U-0.3075 (PASS 4)
N9 U-0.3075 (PASS 5)
N10 U-0.3075 (PASS 6)
N11 G00 X10.0 Z2.0 T0100 M09
N12 M01 (END OF ROUGHING)
```

This cycle is quite simple in both versions - all that is needed is to calculate the new diameter for each roughing cut. If the same roughing tool path had been programmed using the block-by-block method (without G90), the final program would be more than three times longer.

### ◆ Taper Cutting Example

The *Figure 35-4* shows a drawing similar to that used for the previous example. In this example, a taper will be cut, also using G90 simple cycle.

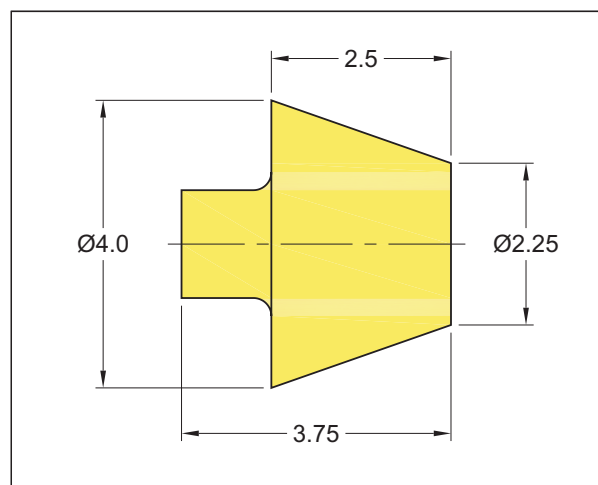


Figure 35-4

Example of G90 cycle in taper cutting - program O3503

In order to distinguish between straight cutting and taper cutting methods, using the same G90 cycle, there must be a way to distinguish these two kinds of cut, and there is one as part of the cycle parameters.

Main difference is the addition of an I(R) parameter to the cycle call, indicating the *taper amount* and its *direction per side*. This value is called a *signed radius value*. It is an I value because of its association with the X-axis. For straight cutting, the I value will always be zero and does not have to be written in the program. Its only significance is for taper cutting, in which case it has a non-zero value - Figure 35-5.

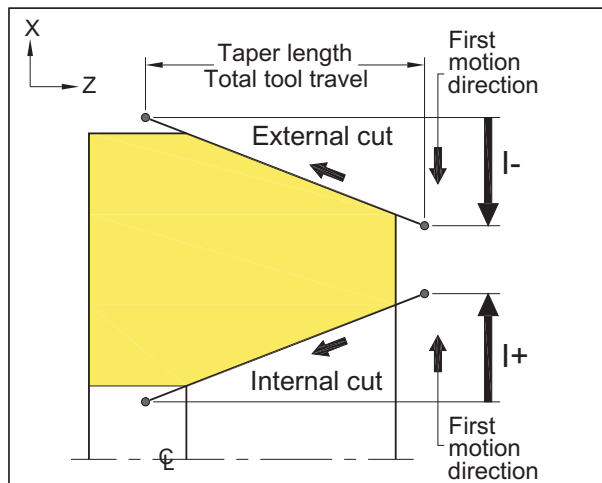


Figure 35-5

The I amount used for G90 turning cycle - external and internal

The illustration shows that the I amount is calculated as a single distance, *i.e.*, as per single side (a radius value), with specified direction, based on the *total* traveled distance and the direction of the first motion from the start position.

There are two simple rules for G90 taper cutting:

- If the direction of the first tool motion in X is negative, the I amount is negative
- If the direction of the first tool motion in X is positive, the I amount is positive

On a CNC lathe with the X-axis positive direction *above* the spindle center line, the typical I value will be *negative* for external taper cutting (turning) and *positive* for internal taper cutting (boring).

To program the part in Figure 35-4, keep in mind that the illustration represents the finished item and does not contain any clearances. Always add all necessary clearances first, then calculate the I amount.

In the example, a clearance of 0.100 will be added at each end of the taper, increasing its length along the axis from 2.5 to 2.7. The I amount calculation requires the *actual length* of tool travel, while maintaining the taper angle at the same time. Either the method of similar triangles or the trigonometric method can be used for such calculation (see pages 504 and 507 for details). Figures 35-6 and 35-7 illustrate details of the known and unknown dimensions for the I amount calculation.

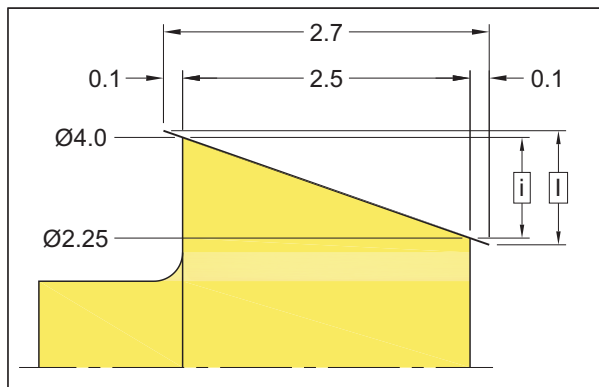


Figure 35-6

Known and unknown values for taper cutting - program O3503  
Amount 'i' is known, amount 'I' has to be calculated

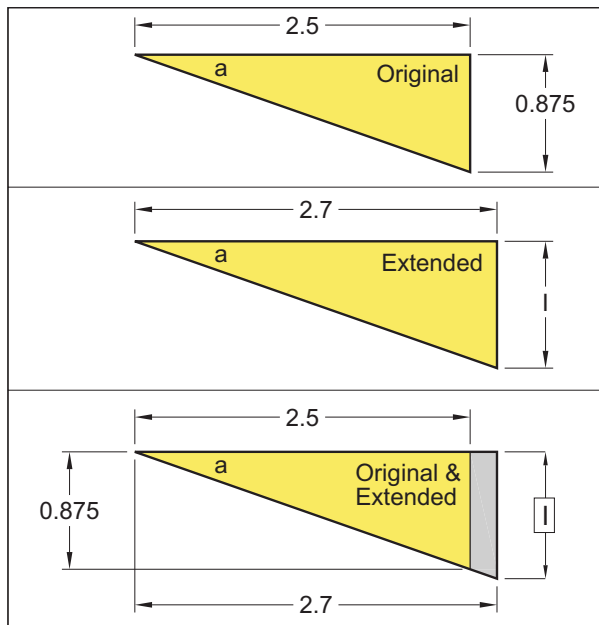


Figure 35-7

The I distance calculation using the similar triangles method

The example shown above almost suggests the simplest method of calculation, a method that is known in mathematics as the *law of similar triangles*. This law has several possible definitions, and the one that applies here is that ...

Two triangles are similar, if the corresponding sides of the two triangles are proportional

In programming, quite often there is a situation that can be solved by more than one method. Choose the one that suits better a certain programming style, then try the other method, expecting the same result. Both methods will be used here, to confirm the accuracy of the calculation.

### ➡ Using Similar Triangles Method

First, calculate the difference  $i$  between the two known diameters, as per drawing:

$$i = (4 - 2.25) / 2 = 0.875$$

therefore, the ratio of similar triangles will be

$$I / 2.7 = i / 2.5$$

We know  $i$  to be 0.875, so the relations can be modified by filling in the known amount:

$$I / 2.7 = 0.875 / 2.5$$

$$I = (0.875 \times 2.7) / 2.5$$

$$I = 0.945 \quad \dots \text{is the required amount for programming}$$

### ➡ Using Trigonometric Method

The second method of calculating the I amount requires trigonometry. At this point, it is known that

$$I = 2.7 \times \tan a$$

and the tangent value has to be calculated first:

$$\tan a = i / 2.5$$

$$\tan a = 0.875 / 2.5$$

$$\tan a = 0.350$$

The amount of I can be calculated using the result:

$$I = 2.7 \times 0.35$$

$$I = 0.945 \quad \dots \text{is the required amount for programming}$$

In both cases, the calculations have the same result, confirming accuracy of the process. The I amount calculation is shown in Figure 35-6 and detailed in Figure 35-7. Program O3503 is the final result - five cuts with 0.03 X-stock left:

```
O3503
(G90 TAPER TURNING EXAMPLE 1 - W/0.03 X-STOCK)
N1 G20
N2 T0100 M41
N3 G96 S450 M03
N4 G00 X4.2 Z0.1 T0101 M08 (START)
N5 G90 X3.752 Z-2.6 I-0.945 F0.01 (1)
N6 X3.374 (2)
N7 X2.996 (3)
N8 X2.618 (4)
N9 X2.24 (5)
N10 G00 X10.0 Z2.0 T0100 M09 (CLEAR POS.)
N11 M01 (END OF ROUGHING)
```

### ◆ Straight and Taper Cutting Example

Another variation of a taper is also common in CNC programming. The Figure 35-8 shows another simple drawing, this time with a taper and a shoulder.

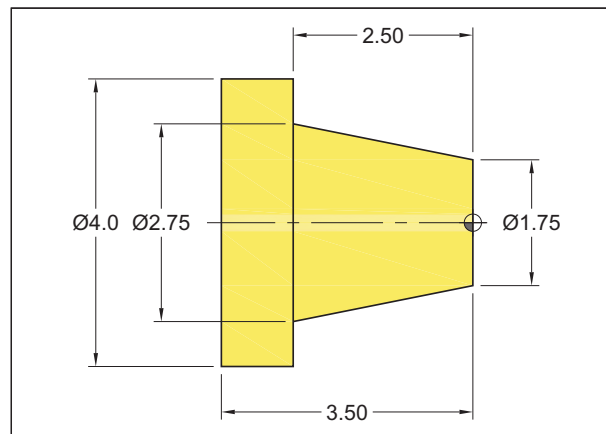


Figure 35-8

Example of G90 cycle used on a taper to a shoulder - O3504

Using the simple cycle G90, the machining requires a tapered cut towards a straight shoulder. A single G90 cycle can be used in this case as well, but could result in some *excessive or insufficient* cutting (too much or too little stock). The best approach is to use *two modes* of the cycle - one for the straight roughing, the other for tapered roughing.

Similar to the previous example, the I taper amount has to be calculated, using the same law of similar triangles as before. The height  $i$  of the original triangle over the length of 2.5 is calculated as one half of the difference between the Ø2.750 and the Ø1.750:

$$i = 2.75 - 1.75 / 2$$

$$i = 0.500$$

For the extended taper length, 0.005 stock amount is left at the shoulder for finishing and the taper is extended by 0.100 at the front face, for the total taper length of 2.595:

$$2.5 - 0.005 + 0.100 = 2.595$$

The I amount can now be calculated, based on the original and the extended values:

$$I / 2.595 = 0.500 / 2.5$$

$$I = (0.500 \times 2.595) / 2.5$$

$$I = 0.519 \quad \dots \text{negative direction}$$

For roughing, a 0.030 stock will be left per side along the X axis, which is 0.060 on diameter.

In roughing operations, it is always important to select a suitable depth of cut, with safety in mind, as well as the cutting conditions. In this example, the depth of cut selection will benefit from one simple programming technique. If the depth of cut is selected arbitrarily, the *last* depth will be whatever is left to cut. A better way is to select a calculated *number of equal cuts* - Figure 35-9.





## MULTIPLE REPETITIVE CYCLES

Unlike the fixed cycles for various drilling operations on machining centers, or the G90 and G94 simple cycles for turning, the advanced cycles for CNC lathe work are much more sophisticated. The major and most distinctive feature of these cycles is their departure from the sequential order of operations. Lathe work can be very complex and the modern control systems do reflect that need. Not only straight or tapered cuts can be programmed, but also radii, chamfers, grooves, undercuts, etc.; simply, several of these cycles are used for contouring. Tool nose radius offset may also be applied, if applicable to the job.

*Multiple repetitive cycles*, as these cycles are called, require a computer memory in order to be useful, so the old NC machines controlled by a punched tape, *cannot* benefit from them. In tape operation, the control unit reads the tape codes sequentially, in a forward direction only. A CNC control, on the other hand, is far more complex. It can read, evaluate and process information stored in the memory in both directions, forwards and backwards - at all times. It can process mathematical instructions internally in a split of a second, simplifying the programming effort.

### ◆ General Description

In total, there are seven multiple repetitive cycles available, identified by a preparatory command address G:

#### *Profile cutting cycles - Roughing:*

G71	Rough cutting cycle - <i>Horizontal emphasis</i>
G72	Rough cutting cycle - <i>Vertical emphasis</i>
G73	Pattern repeating roughing cycle

#### *Profile cutting cycles - Finishing:*

G70	Finishing cycle for G71, G72 and G73
-----	--------------------------------------

#### *Chipbreaking cycles:*

G74	Peck drilling cycle	in Z-axis - horizontal
G75	Peck grooving cycle	in X-axis - vertical

#### *Threading cycle:*

G76	Threading cycle - straight or tapered
-----	---------------------------------------

G76 threading cycle is described separately and in sufficient detail in the threading section (*see page 363*).

### ◆ Cycle Format Types

Each cycle is governed by very specific rules and has its *do's* and *don'ts*. The following sections describe each of them in detail, except the G76 threading cycle, which will be covered separately (*page 363*).

An important fact to take a note of, is that the format of programming for these cycles, the method of data input, is *different* for the lower level of Fanuc controls, such as the very popular 0T or the 16/18/20/21T series, than for the higher level, such as the 10/11T or the 15T series. These cycles, if they are available for the lower level controls, require their programming format in *two blocks*, not the normal *one block*. Check the parameter settings for each control, to find about compatibility issues. Description of both formats is also included in this chapter.

### ◆ Cutting Cycles and Part Contour

Probably the most common multiple repetitive cycles in turning and boring are those that are used for *profile cutting* or *contour cutting*. There are three cycles available within the roughing category:

#### ■ G71, G72 and G73

and one cycle is available for finishing:

#### ■ G70

The finishing cycle is designed to finish profile generated by *any* one of the three roughing cycles.

In some respects, there is an interesting situation in programming multiple repetitive cycles. So far, the emphasis was to program roughing cuts *before* finishing cuts. This approach makes perfect sense - it is also the only logical way from the *technological* point of view. Don't be surprised if this 'rule' is suddenly broken when computer calculations take over. Yes, the implication here is that when programming the three multiple repetitive roughing cycles, the *finished contour must always be defined first*, then its machining specifications can be applied to the roughing cycle. Sounds strange? At first, perhaps. When working with these cycles longer, it will be easy to see that it is actually quite a clever and ingenious method, although hardly a recent breakthrough.

### ◆ Chipbreaking Cycles

The two remaining 'chipbreaking' cycles are designed to produce an *interrupted cut*, either along the Z-axis (G75), or along the X-axis (G74). In practice, G74 cycle offers more practical applications than G75. G74 cycle allows to peck-drill on a CNC lathe. Although the need for peck drilling is much lower on a lathe than on a machining center, it is not that rare. However, G75 cycle, which is really a 'peck-grooving' cycle, is used rarely, as *it does not produce a precision groove*.

## CONTOUR CUTTING CYCLES

By far, the contour cutting cycles (profiling cycles), are the most common cycles in CNC lathe programming. They are used for external (turning) and internal (boring) material removal, along almost any machinable contour.

### ◆ Boundary Definition

The roughing cycles are based on the definition of *two* boundaries, typically called the *material* boundary, which is the outline of the blank, and the *part* boundary, which is the outline of the part contour. This is not a new concept at all, several early programming languages were using this method, such as the Compact II®, a very popular language based programming system of the 1970's.

The two defined boundaries create a fully enclosed area that defines the excessive material. From this isolated area, the material is removed in an orderly way, following specified machining parameters in the cycle call block or blocks. Mathematically, the minimum number of points that can define an area is *three*. These three points must be nonlinear (meaning not on the same line). *Figure 35-11* shows a simple boundary with only three points and a boundary consisting of many points.

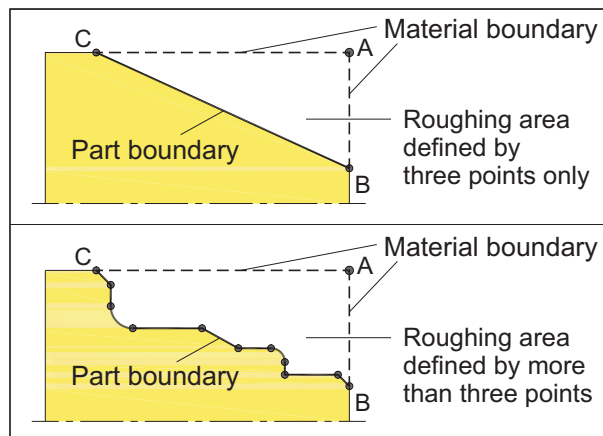


Figure 35-11

Material and part boundaries as applied to turning

In the contour cutting cycles, each point represents a tool position and the points A, B, and C represent the extreme corners of the selected (defined) machining area.

*Material boundary* is not defined directly, it is only implied. It lies between points A and B, and points A and C. *Material boundary can not contain* any other points; it must be a straight line, but not always a line parallel to an axis.

*Part boundary* is defined between points B and C, and may have any number of points between. For CNC programming, different descriptions will be used rather than the generic ABC points in *Figure 35-11*.

### ◆ Start Point and the Points P and Q

The illustrated point A is the *start point* of any contour cutting cycle. It can be defined in simple terms:

Start point is defined as the last XZ coordinate location of the tool, before the contour cutting cycle is called

Typically, this start point will be *closest* to the part corner where the rough cutting begins. It is important to select the start point very carefully, because it is more than 'just a start point'. In fact, this special point controls all approach clearances and the actual depth of the *first* roughing cut.

The generic points B and C in the last illustration will become points P and Q in the program, respectively:

Point P represents the block number of the first XZ coordinate of the finished contour

Point Q represents the block number of the last XZ coordinate of the finished contour

Other in-depth considerations relating to P and Q boundary points are equally important - there are quite a few of them:

- A number of points may be defined between the P and Q points, representing XZ coordinates of the finished contour. Contour is programmed using G01, G02, and G03 tool motions, as well as feedrates
- Material removal defined by the starting point and the P-Q contour must include all necessary clearances
- Tool nose radius offset should not be included between the P and Q points, but programmed *before* the cycle is called, usually during motion to the start point
- For roughing, material to be machined will be divided into a series of machinable cuts. Each roughing cycle accepts a number of user supplied cutting parameters
- For safety reasons, diameter of the start point should be *above* the stock diameter for external cutting, and *below* the core diameter for internal cutting
- Tool motion between P point and Q point must be steadily increasing for external cutting, or steadily decreasing for internal cutting
- Any change in direction between P and Q points is allowed only if Type II cycle is available *and* programmed, and then in one direction only - see the next section for details
- Blocks representing the first XZ coordinate of the contour P, and the last XZ coordinate of the contour Q, must contain a sequence number N, not duplicated anywhere else in the program

## TYPE I AND TYPE II CYCLES

In the initial versions of contour cutting cycles, a change of the contouring direction into the *opposite* direction along one axis was not allowed. That limited these cycles to some extent, because common undercuts or recesses were not possible to use in the program, yet they were common in machine shops.

Presently, this older method is called *Type I* repetitive cycles. The modern controls use many more advanced software features and the change in a *single* direction is now allowed. This newer method is now called *Type II*, allowing more programming flexibility when cutting recesses and cavities (undercuts). *Figure 35-12* compares the two types and shows a disallowed contour change in two directions, within a cycle. The example applies to G71 external cutting cycle, but can be modified for any internal cutting.

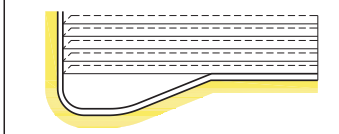
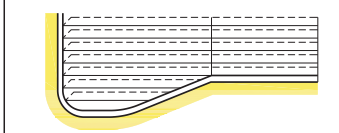

	Type I cycle ... is roughed out in a single depth
	Type II cycle ... is roughed out in several depths
	Bi-directional ... contour change in two axes is NOT allowed

Figure 35-12

Comparison of Type I and Type II cycles -  
- bi-directional change along two axes is not allowed

*Type I* allows a steadily *increasing* contour (for external cutting) or steadily *decreasing* contour (for internal cutting) from point P to point Q (typical cutting directions). On older controls, opposite X or Z direction is not allowed. Modern controls do allow an undercut to be machined with *Type I*, but the recess cutting will be done with a single pass. That may cause some heavy metal removal in certain cases. Make sure to know exactly which type the control system supports. Some experimentation may be necessary.

*Type II* allows a continually increasing profile or continually decreasing profile from point P to point Q. A change into the opposite direction is allowed for a *single axis only*, depending on the active cycle. Rough out process of an undercut will be a multiple tool path. The selection of *Type I* or *Type II* is applicable to the cycle, by programming *both axes* in the block represented by the P point. This is typically the block immediately following the cycle call in the program (after G71, G72, etc.).

### ◆ Programming Type I and Type II Cycles

If the control system supports *Type II* metal removal in turning and boring cycles, it also supports *Type I*, if it needs to be used for some special applications. That means, Fanuc has not replaced one type for another, it has added the *Type II*. Of course, the question is - how to distinguish between the two types in the program? Key to the type selection is in the contents of the block that *immediately follows the cycle call*:

- Type I ... only one axis is specified
- Type II ... two axes are specified

#### ➡ Example - Type I :

```
G71 U.. R..
G71 P10 Q.. U.. W.. F.. S..
N10 G00 X.. (ONE AXIS FOR TYPE I)
...
```

#### ➡ Example - Type II :

```
G71 U.. R..
G71 P10 Q.. U.. W.. F.. S..
N10 G00 X.. Z.. (TWO AXES FOR TYPE II)
...
```

If there is no motion along the Z-axis in the first block after the cycle call and *Type II* cycle is still required, just program W0 as the second axis.

### ◆ Cycle Formatting

On the next few pages is a description of the six turning cycles, covered in detail. It is important to understand the format of each cycle as it applies to a particular control. Several Fanuc control models are available and for the purposes of programming these multiple repetitive cycles, they can be separated into two groups:

- Fanuc system 0T, 16T, 18T, 20T, 21T
- Fanuc system 3T, 6T, 10T, 11T, 15T

Practically, it only means a change in the way the cycle is programmed, but the subject is also important for solving some incompatibility problems. Note that the tool function T is not specified in any of the examples, although it is also allowed as a parameter in all multiple repetitive cycles. Its only need maybe for a tool offset change.

## G71 - STOCK REMOVAL IN TURNING

The most common roughing cycle is G71. Its purpose is to remove stock by *horizontal* cutting, primarily along the Z-axis, typically from the right to the left. It is used for roughing out material out of a solid cylinder. Like all cycles, it comes in two formats - a one-block and a double block format, depending on the control system.

### ◆ G71 Cycle Format - 6T/10T/11T/15T

The one-block format for the G71 cycle is:

G71 P.. Q.. I.. K.. U.. W.. D.. F.. S..

where ...

- P = First block number of the finishing contour
- Q = Last block number of the finishing contour
- I = Distance and direction of rough semifinishing in the X-axis - per side
- K = Distance and direction of rough semifinishing in the Z-axis
- U = Stock amount for finishing on the X-axis diameter
- W = Stock left for finishing on the Z-axis
- D = Depth of roughing cut
- F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block
- S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

The I and K parameters are not available on all machines. They control the amount of cut for semifinishing, the last continuous cut before final roughing motions.

### ◆ G71 Cycle Format - 0T/16T/18T/20T/21T

If the control requires a double block entry for the G71 cycle, the programming format is:

G71 U.. R..  
G71 P.. Q.. U.. W.. F.. S..

where ...

#### First block:

- U = Depth of roughing cut
- R = Amount of retract from each cut

#### Second block:

- P = First block number of the finishing contour
- Q = Last block number of the finishing contour
- U = Stock amount for finishing on the X-axis diameter
- W = Stock left for finishing on the Z-axis
- F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block
- S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

Do not confuse address U in the first block, depth of cut per side, and address U in the second block, stock left on diameter. The I and K parameters may be used only on some controls and the retract amount R is set by a system parameter.

Example of external and internal use of the G71 cycle will use drawing data in *Figure 35-13*. The examples will use the two-block G71 method, as it is more common.

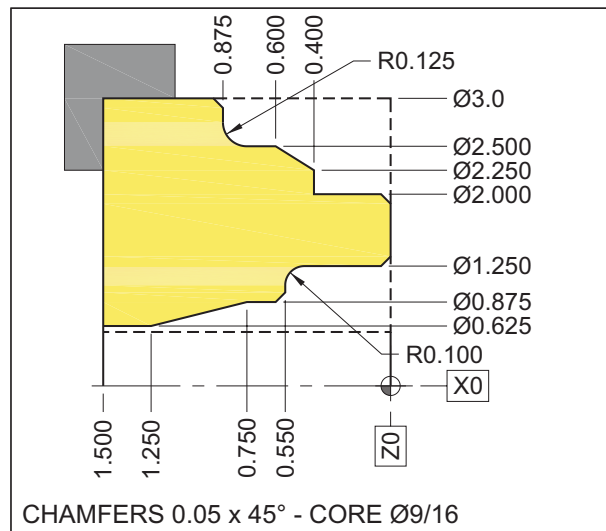


Figure 35-13

Drawing example to illustrate G71 roughing cycle - program O3505

### ◆ G71 for External Roughing

The stock material in the example has an existing hole diameter 0.5625 (Ø9/16). For external cutting of this part, a standard 80° tool will be used for a single cut on the face, as well as for roughing the outer contour. Program O3505 covers these operations.

```
O3505      (G71 ROUGHING CYCLE - ROUGHING ONLY)
N1 G20
N2 T0100 M41      (OD ROUGHING TOOL + GEAR)
N3 G96 S450 M03    (SPEED FOR ROUGH TURNING)
N4 G00 G41 X3.2 Z0 T0101 M08      (START FOR FACE)
N5 G01 X0.36      (END OF FACE DIA)
N6 G00 Z0.1      (CLEAR OFF FACE)
N7 G42 X3.1      (START POSITION FOR CYCLE)
N8 G71 U0.125 R0.04
N9 G71 P10 Q18 U0.06 W0.004 F0.014
N10 G00 X1.7      (P POINT = START OF CONTOUR)
N11 G01 X2.0 Z-0.05 F0.005
N12 Z-0.4 F0.01
N13 X2.25
N14 X2.5 Z-0.6
N15 Z-0.875 R0.125
N16 X2.9
N17 G01 X3.05 Z-0.95
N18 U0.2 F0.02      (Q POINT = END OF CONTOUR)
N19 G00 G40 X5.0 Z6.0 T0100
N20 M01
```

External roughing has been completed at this point in the program and internal roughing can be programmed for the next tool. In all examples that include a tool change between a short tool (such as a turning tool) and a long tool (such as a boring bar), it is important to move the short tool further from the front face. The motion should be far enough to accommodate the incoming long tool. The clearance is 6.0 in the above example (block N18 with Z6.0).



### ◆ G71 for Internal Roughing

The facing cut has been done with the previous tool and the roughing boring bar can continue machining:

```

N21 T0300                      (ID ROUGHING TOOL)
N22 G96 S400 M03              (SPEED FOR ROUGH BORING)
N23 G00 G41 X0.5 Z0.1 T0303 M08 (START POS.)
N24 G71 U0.1 R0.04
N25 G71 P26 Q33 U-0.06 W0.004 F0.012
N26 G00 X1.55                 (P POINT = START OF CONTOUR)
N27 G01 X1.25 Z-0.05 F0.004
N28 Z-0.55 R-0.1 F0.008
N29 X0.875 K-0.05
N30 Z-0.75
N31 X0.625 Z-1.25
N32 Z-1.55
N33 U-0.2 F0.02               (Q POINT = END OF CONTOUR)
N34 G00 G40 X5.0 Z2.0 T0300
N35 M01

```

Now, the part has been completely roughed out, leaving only the required stock on diameters and faces or shoulders. Finishing with G70 cycle, described later, is possible with the same tool, if tolerances and/or surface finish are not too critical. Otherwise, another tool or tools will be required in the same program, after a tool change.

At this stage, evaluate what has been done and why. Many principles that applied to the example are very common to other operations that also use the multiple repetitive cycles. It is important to learn them well at this point.

### ◆ Direction of Cutting in G71

Example O3505 shows that G71 can be used for roughing *externally* or *internally* with two important differences:

- Start point relative to P point (SP to P versus P to SP)
- Sign of U address for stock allowance on diameter

Control system will process the cycle for *external* cutting, if the X direction from the start point SP to the point P is *negative*. In the example, X start point is X3.1, P point is X1.7. The X direction is *negative* or decreasing and *external* cutting will take place.

Control system will process the cycle for *internal* cutting, if the X direction from the start point SP to the point P is *positive*. In the example, X start point is X0.5, P point is X1.55. The X direction is *positive* or increasing, and *internal* cutting will take place.

Figure 35-14 illustrates the concept of G71 cycle, as applied to both, external and internal cutting.

By the way, although the *sign* of the stock U value is very important for the final size of the part, it does *not* determine the mode of cutting. This concludes the section relating to the G71 multiple repetitive cycle. The face roughing cycle G72 is similar, and is described next.

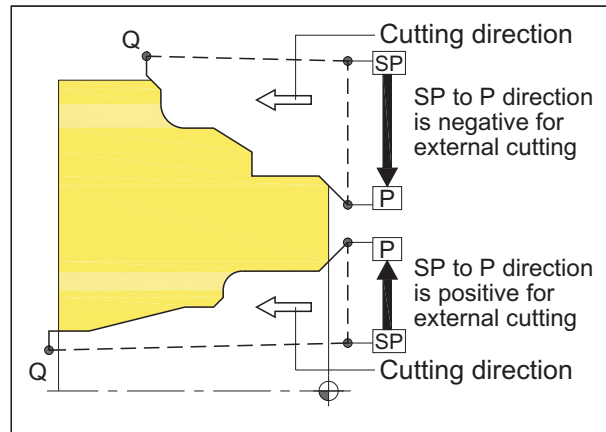


Figure 35-14

External and internal cutting in G71 cycle

## G72 - STOCK REMOVAL IN FACING

G72 cycle is identical in every respect to the G71 cycle, except the stock is removed mainly by *vertical* cutting (facing), typically from the large diameter towards the spindle center line X0. It is used for roughing of a solid cylinder, using a series of *vertical* cuts (face cuts). Like all other cycles in this group, it comes in two formats - a one block and a double block format, depending on the control system. Compare G72 with the G71 structure on examples in this chapter and you will find many similarities.

### ◆ G72 Cycle Format - 6T/10T/11T/15T

The one-block programming format for the G72 cycle is:

G72 P.. Q.. I.. K.. U.. W.. D.. F.. S..

where ...

- P = First block number of the finishing contour
- Q = Last block number of the finishing contour
- I = Distance and direction of rough semifinishing in the X-axis - per side
- K = Distance and direction of rough semifinishing in the Z-axis
- U = Stock amount for finishing on the X-axis diameter
- W = Stock left for finishing on the Z-axis
- D = Depth of roughing cut
- F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block
- S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

Meaning of each address is the same as for the G71 cycle. The I and K parameters are not available on all machines. These parameters control the amount of cut for semifinishing, which is the last continuous cut before final roughing motions are completed.

### ◆ G72 Cycle Format - 0T/16T/18T/20T/21T

If the control system requires a double block entry for the G72 cycle, the programming format is:

G72 W.. R.. G72 P.. Q.. U.. W.. F.. S..
--------------------------------------------

where ...

#### First block:

W = Depth of roughing cut  
R = Amount of retract from each cut

#### Second block:

P = First block number of the finishing contour  
Q = Last block number of the finishing contour  
U = Stock amount for finishing on the X-axis diameter  
W = Stock left for finishing on the Z-axis  
F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block  
S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

In G71 cycle for the double block definition, there were two addresses U. In G72 double block definition cycle, there are two addresses W. Make sure you do not confuse the W in the first block - depth of cut (actually it is a *width* of cut), and the W in the second block - stock left on faces. The I and K parameters may be available, depending on the control.

An example program O3506 for the G72 cycle uses the drawing data in Figure 35-15.

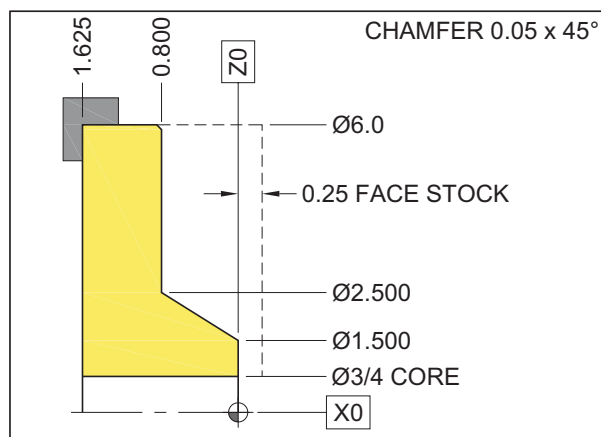


Figure 35-15

Drawing example to illustrate G72 roughing cycle - program O3506

In this facing application, all main data will be reversed by 90°, because the cut will be segmented along the X-axis. Roughing program using the G72 cycle is logically similar to the G71 cycle:

### O3506 (G72 ROUGHING CYCLE - ROUGHING ONLY)

```
N1 G20
N2 T0100 M41          (OD FACING TOOL + GEAR)
N3 G96 S450 M03       (SPEED FOR ROUGH FACING)
N4 G00 G41 X6.25 Z0.3 T0101 M08 (START POS.)
N5 G72 W0.125 R0.04
N6 G72 P7 Q13 U0.06 W0.03 F0.014
N7 G00 Z-0.875       (P-POINT = START OF CONTOUR)
N8 G01 X6.05 F0.02
N9 X5.9 Z-0.8 F0.008
N10 X2.5
N11 X1.5 Z0
N12 X0.55
N13 W0.1 F0.02       (Q-POINT = END OF CONTOUR)
N14 G00 G40 X8.0 Z3.0 T0100
N15 M01
```

General concept of G72 cycle is illustrated in Figure 35-16. Note the position of point P as it relates to the start point SP and compare it with the G71 cycle.

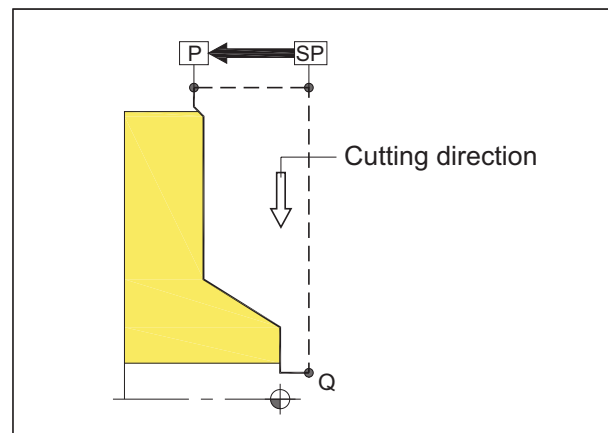


Figure 35-16

Basic concept of G72 multiple repetitive cycle

## G73 - PATTERN REPEATING CYCLE

The pattern repeating cycle is also called the *Closed Loop* or a *Profile Copying* cycle. Its purpose is to minimize the cutting time for roughing material of irregular shapes and forms, for example, forgings and castings.

### ◆ G73 Cycle Format - 6T/10T/11T/15T

The one-block programming format for G73 cycle is similar to both the G71 and G72 cycles:

G73 P.. Q.. I.. K.. U.. W.. D.. F.. S..
-----------------------------------------

where ...

P = First block number of finishing contour  
Q = Last block number of finishing contour  
I = X-axis distance and direction of relief - per side  
K = Z-axis distance and direction of relief  
U = Stock amount for finishing on the X-axis diameter

- W = Stock left for finishing on the Z-axis
- D = The number of cutting divisions
- F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block
- S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

### ◆ G73 Cycle Format - 0T/16T/18T/20T/21T

If your control system requires a double block entry for the G73 cycle, the programming format is:

```
G73 U.. W.. R..
G73 P.. Q.. U.. W.. F.. S..
```

where ...

#### First block:

- U = X-axis distance and direction of relief - per side
- W = Z-axis distance and direction of relief
- R = Number of cutting divisions

#### Second block:

- P = First block number of the finishing contour
- Q = Last block number of the finishing contour
- U = Stock amount for finishing on the X-axis diameter
- W = Stock left for finishing on the Z-axis
- F = Cutting feedrate (in/rev or mm/rev) overrides feedrates between P block and Q block
- S = Spindle speed (ft/min or m/min) overrides spindle speeds between P block and Q block

In the two-block cycle entries, do not mix up addresses in the first block that repeat in the second block (U and W in the G73 example). *They have a different meaning!*

### ◆ G73 Example of Pattern Repeating

Pattern repeating cycle G73 program example uses the drawing in Figure 35-17.

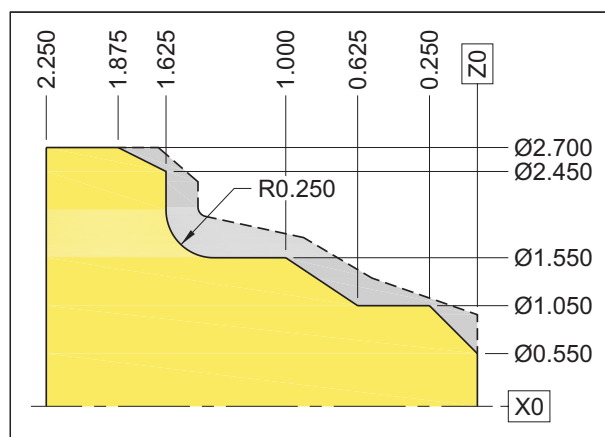


Figure 35-17

Pattern repeating cycle G73 - program example O3507

There are three important input parameters in G73 cycle - U/W/R (I/KD). One parameter seems to be missing - *there is no depth of cut specification!* In G73 cycle, it is not needed. The actual depth of cut is calculated automatically, based on these three parameters:

- U (I) ... amount of rough material to remove in X-axis
- W (K) ... amount of rough material to remove in Z-axis
- R (D) ... number of cutting divisions or repeats

Use this cycle with care - its design assumes an *equal amount* of rough stock to be removed along both the X and the Z axes. That is not the typical reality for forgings and castings, where the stock *varies* all over the material - see the illustration in Figure 37-17. The cycle can still be used with a reasonable efficiency, but some 'air' cutting may be an unwanted side effect for odd shaped parts.

In the example, the *largest expected* material amount per one side will be chosen as 0.200 (U0.2) and the *largest expected* material amount on the face as 0.300 (W0.3). The number of divisions could be either two or three, so the program will use R3. Some modification at the control may be necessary during actual setup or machining, depending on the exact condition and sizes of the casting or forging.

This cycle is suitable for roughing contours where the finish contour closely matches the contour of the casting or forging. Even if there is some 'air' cutting, this cycle may be more efficient than the selection of the G71 or G72 cycles where too much empty cutting would take place. The program O3507 shows roughing and finishing with the same tool (as an example):

#### O3507 (G73 PATTERN REPEATING CYCLE)

```
N1 G20 M42
N2 T0100
N3 G96 S350 M03
N4 G00 G42 X3.0 Z0.1 T0101 M08
N5 G73 U0.2 W0.3 R3
N6 G73 P7 Q14 U0.06 W0.004 F0.01
N7 G00 X0.35
N8 G01 X1.05 Z-0.25
N9 Z-0.625
N10 X1.55 Z-1.0
N11 Z-1.625 R0.25
N12 X2.45
N13 X2.75 Z-1.95
N14 U0.2 F0.02
N15 G70 P6 Q13 F0.006
N16 G00 G40 X5.0 Z2.0 T0100
N17 M30
%
```

The number of passes R3 may be necessary to accommodate some rotating eccentricity, normally associated with castings and forgings. On the other hand, R2 may be necessary for the heavier cut, to 'bite under the skin' of the material, for better tool life. Schematically, Figure 35-18 shows three programmed cutting divisions (R3).

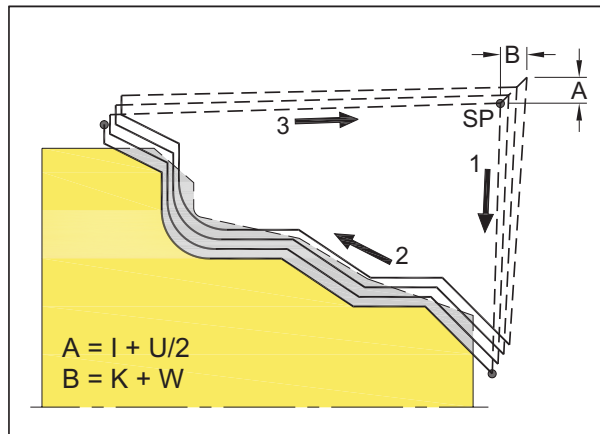


Figure 35-18

Schematic representation of G73 cycle

Note that the pattern repeating cycle does exactly that - it repeats the machining contour (pattern) specified between the P and Q points. Each individual tool path is offset by a calculated amount along the X and Z axes. On the machine, watch cutting progress with care - particularly for the first tool path. Feedrate override may come useful here.

## G70 - CONTOUR FINISHING CYCLE

The last of the contouring cycles is G70. Although it has a smaller G number than any of the three roughing cycles G71, G72 and G73, finishing cycle G70 is normally used *after* any one of these three roughing cycles. As its description suggests, it is strictly used for *finishing cut of a previously defined contour*.

### ◆ G70 Cycle Format - All Controls

For this cycle, there is no difference in the programming format for various controls - it is all the same, and the cycle call is a one-block command.

The programming format for G70 cycle is:

G70 P.. Q.. F.. S..

where ...

- P = First block number of the finishing contour
- Q = Last block number of the finishing contour
- F = Cutting feedrate (in/rev or mm/rev)
- S = Spindle speed (ft/min or m/min)

Cycle G70 accepts a previously defined finishing contour from either of the three roughing cycles, already described. This finishing contour is defined by the P and Q points of the respective cycles, and is normally repeated in the G70 cycle, although other block number references may be used - **BE CAREFUL HERE!**

For safety, always use the same start point for G70 as for the roughing cycles

Earlier roughing program O3505, using the G71 repetitive cycle for rough turning and rough boring, can be completed by using another two tools, one for external, one for internal finishing tool path:

(O3505 CONTINUED ...)

```
...
N36 T0500 M42          (OD FINISHING TOOL + GEAR)
N37 G96 S530 M03       (SPEED FOR FINISH TURNING)
N38 G42 X3.1 Z0.1 T0505 M08 (START POS.)
N39 G70 P10 Q18        (FINISHING CYCLE - OD)
N40 G00 G40 X5.0 Z6.0 T0500
N41 M01
```

```
N42 T0700              (ID FINISHING TOOL)
N43 G96 S475 M03       (SPEED FOR ROUGH BORING)
N44 G00 G41 X0.5 Z0.1 T0707 M08 (START POS.)
N45 G70 P26 Q33        (FINISHING CYCLE - ID)
N46 G00 G40 X5.0 Z2.0 T0700
N47 M30                (END OF PROGRAM)
%
```

Even for external finishing, the cutting tool is still programmed to start *above* the original stock diameter and off the front face, although all roughing motions have already been completed. A similar approach applies to the internal cut. For safety reasons, this is the recommended practice.

There are no feedrates programmed for G70 cycle, although the cycle format does accept a feedrate. The defined block segments P to Q for roughing tool already include feedrates. These programmed feedrates will be ignored in roughing mode and will become active only for G70 cycle, during finishing. If the finish contour did not include any feedrates, then program a *common feedrate* for finishing all contours during the G70 cycle processing. For example, program block

N39 G70 P10 Q18 F0.007

will be a waste of time, since the 0.007 in/rev feedrate will *never* be used. It will be *overridden* by the feedrate defined between blocks N9 and N17 of program O3505. On the other hand, if there is no feedrate programmed for the finishing contour at all, then

N.. G70 P.. Q.. F0.007

will use 0.007 in/rev exclusively for the finishing tool path.

The same logic described for G71 cycle, applies equally to G72 cycle.

Roughing program O3506, using G72 cycle for rough turning of the part face, can be completed by using another external tool for finishing cuts using G70 cycle:

(O3506 CONTINUED ...)

```

...
N16 T0500 M42           (OD FACING TOOL + GEAR)
N17 G96 S500 M03        (SPEED FOR FINISH FACING)
N18 G00 G41 X6.25 Z0.3 T0505 M08 (START POS.)
N19 G70 P7 Q13          (FINISHING CYCLE)
N20 G00 G40 X8.0 Z3.0 T0500
N21 M30
%
```

The rules mentioned earlier also apply for contour finishing defined by G72 cycle. Program O3507, using the G73 cycle, can be also be programmed by using another external tool for finishing, applying the same rules.

## BASIC RULES FOR G70-G73 CYCLES

In order to make the multiple repetitive stock removal cycles (contouring cycles) work properly and efficiently, observing the rules of their use is very important. Often a small oversight may cause a lengthy delay.

Here are the most important rules and observations as they relate to G71, G72 and G73 turning/boring cycles:

- Always apply tool nose radius offset before the stock removal cycle is called
- Always cancel tool nose radius offset after the stock removal cycle is completed
- Return motion to the start point is automatic, and must not be programmed
- The P block in G71 should not include the Z-axis value (Z or W) for cycle Type I
- Change of direction is allowed only for Type II G71 cycle, and along one axis only (W0)
- Stock allowance U is programmed on a diameter, and its sign shows to which side of the stock it is to be applied - sign is the X-direction relative to centerline:  
... U+ for turning - towards spindle centerline  
... U- for boring - away from spindle centerline
- Feedrate programmed for the finishing contour (specified between P and Q points) will be ignored during roughing
- D address in one-block format only, does not use decimal point, and must be programmed for leading zero suppression format:

D0750 or D750 is equivalent to 0.0750 depth

Only some control systems do allow a decimal point to be used for the D address (depth of cut) in one-block G71 and G72 cycles

## G74 - PECK DRILLING CYCLE

G74 cycle is one of two cycles usually used for non-contour work. Along with G75 cycle, it is used for machining an interrupted cut, such as chips breaking during a long cutting motion. *G74 cycle is used along the Z-axis.*

This is the cycle commonly used for an interrupted cut along the Z-axis. The name of the cycle is *Peck Drilling Cycle*, similar to G73 peck drilling cycle used for machining centers. For lathe work, G74 cycle application is a little more versatile than for its G73 equivalent on machining centers. Although its main purpose may be applied towards peck drilling, the cycle can be used with equal efficiency for interrupted cuts in non-contour turning and boring (for example, in some very hard materials), deep face grooving, difficult part-off machining, and many other applications.

### ◆ G74 Cycle Format - 6T/10T/11T/15T

The one-block programming format for G74 cycle is:

G74 X..(U..) Z..(W..) I.. K.. D.. F.. S..

where ...

X(U) = Final groove diameter to be cut  
 Z(W) = Z-position of the last peck - depth of hole  
 I = Depth of each cut (no sign)  
 K = Distance of each peck (no sign)  
 D = Relief amount at the end of cut  
       (must be zero for face grooving)  
 F = Groove cutting feedrate (in/rev or mm/rev)  
 S = Spindle speed (ft/min or m/min)

### ◆ G74 Cycle Format - 0T/16T/18T/20T/21T

The two-block programming format for G74 cycle is:

G74 R..  
G74 X..(U..) Z..(W..) P.. Q.. R.. F.. S..

where ...

#### First block:

R = Return amount (relief clearance for each cut)

#### Second block:

X(U) = Final groove diameter to be cut  
 Z(W) = Z-position of the last peck - depth of hole  
 P = Depth of each cut (no sign)  
 Q = Distance of each peck (no sign)  
 R = Relief amount at the end of cut  
       (must be zero for face grooving)  
 F = Groove cutting feedrate (in/rev or mm/rev)  
 S = Spindle speed (ft/min or m/min)



If both the X/U and P (or I) are omitted in the cycle, the machining is along the Z-axis only (peck drilling). In a typical peck drilling operation, only the Z, K and F values are programmed - see Figure 35-19.

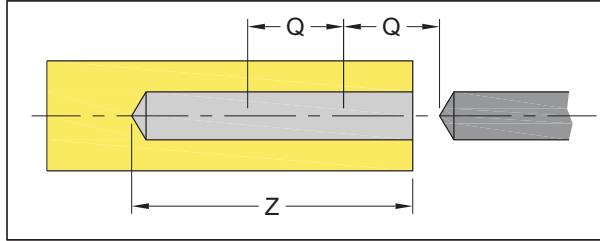


Figure 35-19

Schematic format for G74 cycle example (two block format)

The following program example illustrates G74 cycle:

```
O3507 (G74 PECK DRILLING)
N1 G20
N2 T0200
N3 G97 S1200 M03           (SPEED IN RPM)
N4 G00 X0 Z0.2 T0202 M08   (START POSITION)
N5 G74 R0.02
N6 G74 Z-3.0 Q6250 F0.012  (PECK DRILLING)
N7 G00 X6.0 Z2.0 T0200     (CLEAR POSITION)
N8 M30                     (END OF PROGRAM)
%
```

Drilling will take place to a three inch depth, in depth increments of 0.625 of an inch. Note the first depth peck is calculated from the start position. Programming of an interrupted groove is very similar in format.

## G75 - GROOVE CUTTING CYCLE

The G75 cycle is the other of two lathe cycles available for simple, non precision work. Together with the G74 cycle, it is used for operations requiring an interrupted cut, for example for breaking chips during a long or deep cutting motion. *G75 cycle is used along the X axis.*

This is also a very simple cycle, designed to break chips during a rough cut along the X axis - used mainly for a grooving operation. The G75 cycle is identical to G74, except the X axis is replaced with the Z axis.

### ◆ G75 Cycle Format - 6T/10T/11T/15T

The one-block programming format for G75 cycle is:

G75 X..(U..) Z..(W..) I.. K.. D.. F.. S..
-------------------------------------------

where ...

X(U) = Final groove diameter to be cut  
Z(W) = Z-position of the last groove  
(for multiple grooves only)

I = Depth of each cut (no sign)  
K = Distance between grooves (no sign)  
(for multiple grooves only)  
D = Relief amount at the end of cut  
(must be zero or not used for face groove)  
F = Groove cutting feedrate (in/rev or mm/rev)  
S = Spindle speed (ft/min or m/min)

### ◆ G75 Cycle Format - 0T/16T/18T/20T/21T

The two-block programming format for G75 cycle is:

G75 R..
G75 X..(U..) Z..(W..) P.. Q.. R.. F.. S..

where ...

#### First block:

R = Return amount (relief clearance for each cut)

#### Second block:

X(U) = Final groove diameter to be cut  
Z(W) = Z-position of the last groove  
P = Depth of each cut (no sign)  
Q = Distance between grooves (no sign)  
R = Relief amount at the end of cut  
(must be zero for face grooving)  
F = Groove cutting feedrate  
(usually in/rev or mm/rev)  
S = Spindle speed (usually ft/min or m/min)

If both the Z/W and Q (or K) are omitted in the cycle, the machining is along the X-axis only (peck grooving).

A practical example of G75 cycle will be presented in the next chapter.

## BASIC RULES FOR G74 AND G75 CYCLES

Several notes are common to both cycles:

- In both cycles, the X and Z values can be programmed either in the absolute or incremental mode.
- Both cycles allow an automatic relief.
- Relief amount at the end of cut can be omitted - in that case it will be assumed as zero.
- Return amount (clearance for each cut) is only programmable for the two-block method. Otherwise, it is set by an internal parameter of the control system.
- If the return amount is programmed (two-block method), and the relief amount is also programmed, the presence of X determines the meaning. If the X-value is programmed, the R-value means the relief amount.

## 36

## GROOVING ON LATHES

Groove cutting on CNC lathes is a multi step machining operation. The term *grooving* usually applies to a process of forming a narrow cavity of a certain depth, on a cylinder, cone, or a face of the part. The groove shape, or at least a significant part of it, will be in the shape of the cutting tool. Grooving tools are also used for a variety of special machining operations.

Grooving tool is usually a carbide insert mounted in a special tool holder, similar to any other tool. Designs of grooving inserts vary, from a single tip, to an insert with multiple tips. Inserts are manufactured to nominal sizes. Multi tip insert grooving tools are used to decrease costs and increase productivity.

## GROOVING OPERATIONS

Cutting tools for grooving are either *external* or *internal* and use a variety of inserts in different configurations. The most important difference between grooving and turning is the *direction of cut*. Turning tool can be applied for cuts in multiple directions, grooving tool is normally used to cut in a single direction only. A notable exception is an operation known as necking (relief grooving), which takes place at 45°, where the cutting insert angle and the infeed angle *must be identical* (usually at 45°). There is another application of a two axis simultaneous motion in grooving, a corner breaking on the groove. Strictly speaking, this is a turning operation. Although a grooving tool is not designed for turning, it can be used for some light machining, like cutting a small chamfer. During the corner breaking cut on a groove, the amount of material removal is always very small and the applied feedrate is normally low.

### ◆ Main Grooving Applications

Groove is an essential part of components machined on CNC lathes. There are many kinds of grooves used in industry. Most likely, programming will include many undercuts, clearance and recess grooves, oil grooves, etc. Some of the main purposes of grooving are to allow two components to fit face-to-face (or shoulder-to-shoulder) and, in case of lubrication grooves, to let oil or some other lubricant to flow smoothly between two or more connecting parts. There are also pulley or V-belt grooves that are used for belts to drive a motor. O-ring grooves are specially designed for insertion of metal or rubber rings, that serve as stoppers or sealers. Many industries use grooves unique to their needs, others use the more general groove types.

### ◆ Grooving Criteria

For a CNC programmer, grooving usually presents no special difficulties. Some grooves may be easier to program than others, yet there could be several fairly complex grooves found in various industries that may present a programming or machining challenge. In any case, before a groove can be programmed, have a good look at the drawing specifications and do some overall evaluations. Many grooves may appear on the same part at different locations and could benefit from a subprogram development. When planning a program for grooving, evaluate each groove carefully. In good program planning, evaluate the selected groove by at least three criteria:

- Groove shape
- Groove location on a part
- Groove dimensions and tolerances

Unfortunately, many grooves are not of the highest quality. Perhaps it is because many grooves do not require high precision and when a high precision groove has to be done, the programmer does not know how to handle it properly. Watch particularly for surface finish and tolerances.

## GROOVE SHAPE

The first evaluation before programming grooves is the groove shape. Shape is determined by the part drawing and corresponds to the groove purpose. Groove shape is the single most important factor when selecting a grooving insert. A groove with sharp corners parallel to the machine axes requires a square insert, a groove with radius requires an insert having the same or smaller radius. Special purpose grooves, for example an angular groove shape, will need an insert with angles corresponding to the groove angles as given in the drawing. Formed grooves require inserts shaped into the same form, etc. Some typical shapes of grooving inserts are illustrated in *Figure 36-1*.

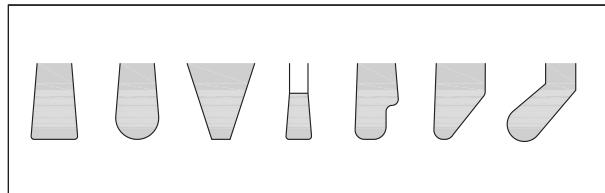


Figure 36-1  
Typical shapes of common grooving tools

### ◆ Nominal Insert Size

In many groove cutting operations, the groove width will be greater than the largest available grooving insert of a nominal size (*i.e.*, off the shelf size). Nominal sizes are normally found in various tooling catalogues and typically have widths like 1 mm, 2 mm, 3 mm or 1/32, 3/64, 1/16, 1/8 in inches, and so on, depending on the units selected.

For example, a groove width of 0.276 inches can be cut with a nearest *lower* nominal insert width of 0.250 inch. In such cases, the groove program has to include at least two cuts - one or more roughing cuts, in addition to at least one finishing cut. Another grooving tool may be used for finishing, if the tolerances or excessive tool wear make it more practical - *Figure 36-2*.

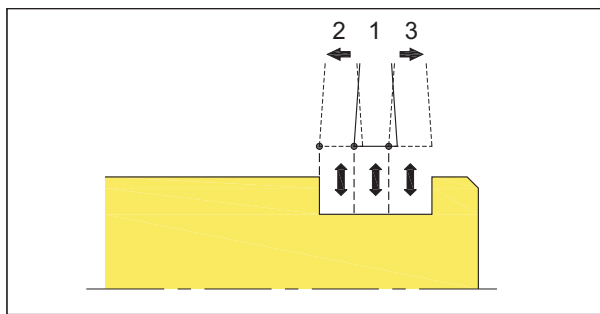


Figure 36-2

Cut distribution for grooves wider than the insert

### ◆ Insert Modification

Once in a while, programmers encounter a groove that requires a special insert in terms of its size or shape. There are two options to consider. One is to have a custom made insert, if it is possible and practical. For a large number of grooves, it may be a justified solution. The other alternative is to modify an existing insert in-house.

Generally, in CNC programming, off-the-shelf tools and inserts should be used as much as possible. In special cases, however, a standard tool or insert can be modified to suit a particular job. For grooving, it may be a small extension of the insert cutting depth, or a radius modification. Try to modify the groove shape itself only as the last resort. Modification of standard tools slows down the production and can be quite costly.

## GROOVE LOCATION

Groove location on a part is determined by the part drawing. The locations can be one of three groups:

- Groove cut on a cylinder ... diameter cutting
- Groove cut on a cone ... taper cutting
- Groove cut on a face ... shoulder cutting

Although some variations are possible, for practical purposes, only these three categories are considered. Each of the three locations may be either *external* or *internal*.

The two most common groove locations are on a cylinder, *i.e.*, on a straight outside - or *external* - diameter, or on a straight inside - or *internal* - diameter. Many other grooves may be located on a face, on a taper (cone), even in a corner. The illustration in *Figure 36-3* shows some typical locations of various grooves.

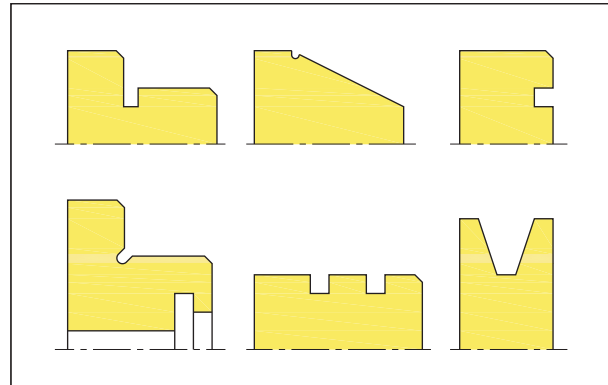


Figure 36-3

Typical groove locations on a part

## GROOVE DIMENSIONS

Dimensions of a groove are always important when selecting the proper grooving insert. Grooving dimensions include the *width* and *depth* of a groove, as well as corner specifications. It is not possible to cut a groove with an insert that is larger than the groove width. Also, it is not possible to feed into a groove depth that is greater than the depth clearance of the insert or tool holder. However, there is usually no problem in using a narrow grooving insert to make a wide groove with multiple cuts. The same applies for a deep cutting insert used to make a shallow groove. *Dimensions of a groove determine the method of machining.* A groove whose width *equals* the insert width selected for the groove shape, requires only *one* cut. Simple *feed-in* and *rapid-out* tool motion is all that is required. To program a groove correctly, the *width* and *depth* of the groove must be known as well as its *position* relative to a known reference position on the part. This position is the distance to one side - or one wall - of the groove.

Some extra large grooves require a special approach. For example, a groove that is 10 mm wide and 8 mm deep cannot be cut in a single pass. In this case, rough cuts for the groove will control not only its width, but also its depth. It is not unusual to even use more than one tool for such an operation. Grooving program may also need to be designed in sections. In case of an insert breakage, only the affected program section has to be repeated.

### ◆ Groove Position

In Figure 36-4 are shown two most common methods of dimensioning a typical groove. Groove width is given in both cases as dimension W, but the distance L from the front face is different in the example *a* and example *b*.

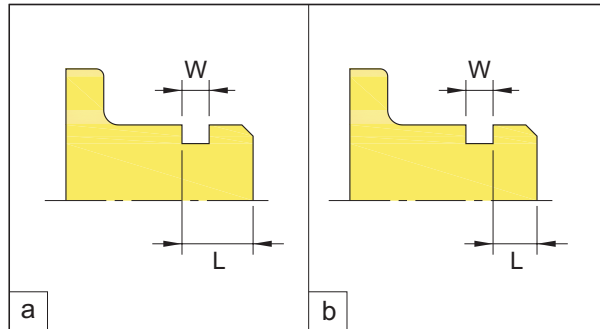


Figure 36-4

Groove position dimensioning - two common methods

In example 36-4a, the dimension L is measured to the *left* side of the groove. For programming purposes, this dimension is more convenient, because it will actually appear in the program as specified in the drawing. Normally, the standard tool reference point of a grooving tool is set to the *left* side of the grooving insert.

Figure 36-4b example shows the dimension L to the *right* side of the groove. The left side dimension can be found easily, by adding the groove width W. Programming considerations will be slightly different, particularly if the dimensional tolerances are specified. Always take a safer approach and presume that the specified dimension indicates *more important* dimension. If a tolerance range is specified for any dimension, this tolerance must always be maintained on the finished groove, and it will affect the overall programming method. A groove may also be dimensioned from another reference location, depending on its purpose.

### ◆ Groove Depth

In Figure 36-5, there are two typical methods of dimensioning the groove *depth*.

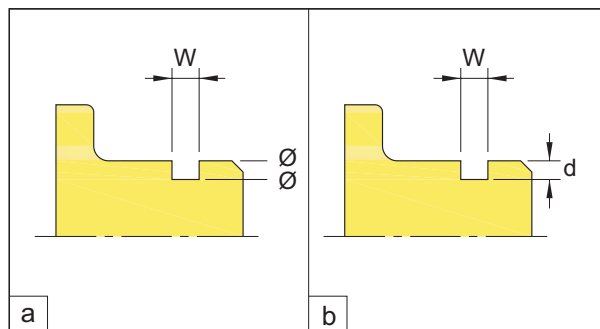


Figure 36-5

Groove depth dimensioning - two common methods

In the Figure 36-5a, the top and bottom groove diameter are both given. This method has a major benefit that the bottom groove diameter will actually appear as an X-axis amount in the program. A disadvantage is that the actual depth still has to be calculated and a proper grooving tool selected. Example in Figure 36-5b does show the groove depth but, most likely, the bottom diameter will have to be calculated manually. Both dimensioning examples are about equally common in CNC programming.

Deep grooves are usually grooves that have a reasonably normal width but also have a much deeper ratio between the groove top diameter and its bottom diameter.

## SIMPLE GROOVE PROGRAMMING

The simplest of all grooves is the one that has the same width and shape as the tool cutting edge - Figure 36-6.

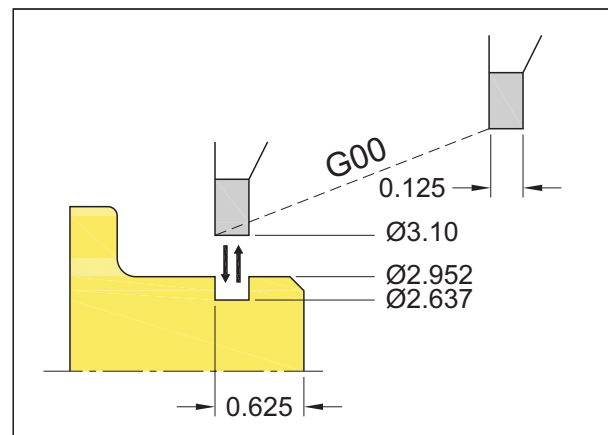


Figure 36-6

Simple groove example - program O3601  
Insert width is equal to the groove width

To program such a groove is very straightforward. In rapid mode, move the grooving tool to its starting position, feed-in to the groove depth, then rapid out back to the starting position, and - the groove is finished. There are no corner breaks, no surface finish control, and no special techniques used. Some will say, and no quality either. A dwell at the groove bottom may be the only improvement. True, quality of such groove will not be the greatest, but a groove it will be. Groove of this type is strictly a utility groove and is hardly ever required in precision manufacturing. At the same time, programming such grooves is a good start to learn more advanced techniques.

The following program O3601 uses standard 0.125 inch square grooving insert, for a groove of the same width. Groove depth is the single difference between the two diameters given in the drawing:

$$(2.952 - 2.637) / 2 = 0.1575$$

The sample program uses tool T08, as the last tool:

```
O3601 (SIMPLE GROOVE)
(G20)
...
N33 T0800 M42 (TOOL 8 ACTIVE)
N34 G97 S650 M03 (650 RPM SPEED)
N35 G00 X3.1 Z-0.625 T0808 M08 (START POINT)
N36 G01 X2.637 F0.003 (FEED-IN TO DEPTH)
N37 G04 X0.4 (DWELL AT THE BOTTOM)
N38 X3.1 F0.05 (RETRACT FROM GROOVE)
N39 G00 X6.0 Z3.0 T0800 M09 (CLEAR POSITION)
N40 M30 (END OF PROGRAM)
%
```

Program O3601 does the following. First, imperial mode of input is preset from the program beginning with G20. Blocks N33 and N34 are startup blocks for tool T08, with direct *r/min* selected. *Constant Surface Speed* (CSS) in G96 mode can be selected instead. N35 is a block where the tool moves to a position from which the groove will be started (start point). Clearance at this safe location is the clearance above the part diameter, which is 0.074 inches in the example:

$$(3.1 - 2.952) / 2 = 0.074$$

Coolant is applied in the same block, during tool motion. Block N36 contains the actual groove plunging cut, at a slow feedrate of 0.003 in/rev. Block N37 is a dwell time of 0.4 seconds, followed by tool return to the starting diameter and completion of the program.

Although this particular grooving example was very simple, evaluate the program a little more. It contains several important principles that can be applied to the method of programming any groove, where its precision and surface finish are very critical.

Note the clearance *before* groove cutting begins. Tool is positioned 0.074 inches above the part diameter, at Ø3.100. Always keep this distance to a certain safe minimum. Grooves are usually cut at a slow feedrate and it may take too much time just to cut in the air. Also note the actual cutting feedrate has increased from 0.003 in/rev in block N36 to a rather high feedrate of 0.050 in/rev in block N38. Rapid motion command G00 could have been used instead. Feeding out at a heavier feedrate (rather than using a rapid motion), may improve the groove surface finish by eliminating tool drag on the walls (sides).

Tool width of 0.125 never appears in the program, directly or indirectly. That means the insert shape and width will become the shape and width of the groove. It also means a different groove width, if another insert size is used, although the program structure remains unaffected. The structure will remain unaffected even if the grooving tool *shape* is changed. Combination of the shape *and* size change will offer endless opportunities, all of them being possible without a single change to the program.

## PRECISION GROOVING TECHNIQUES

A simple *in-out* groove will not be good enough. Its sides may have a rough surface, outside corners will be sharp and its width is dependent on insert width and its wear. For most of machining jobs, such a groove is not acceptable.

To program and machine any precision groove requires extra effort, but the result will be a high quality groove. This effort is not always justified, as high quality comes with a price. The next two illustrations show groove dimensions and program related details. Drawing in *Figure 36-7* shows a high precision groove, although its width is intentionally exaggerated for impact of the example.

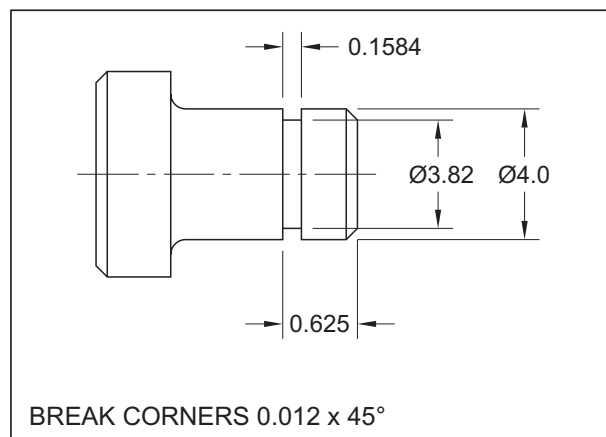


Figure 36-7

Drawing for a precision groove example O3602

What is the best cutting method? One plunge rough cut and two finish cuts, one for each wall, are reasonable; so is the 0.006 stock added to the bottom diameter. Also, sharp corners will be broken with a 0.012 chamfer at Ø4.0. *Figure 36-8* shows distribution of the cuts.

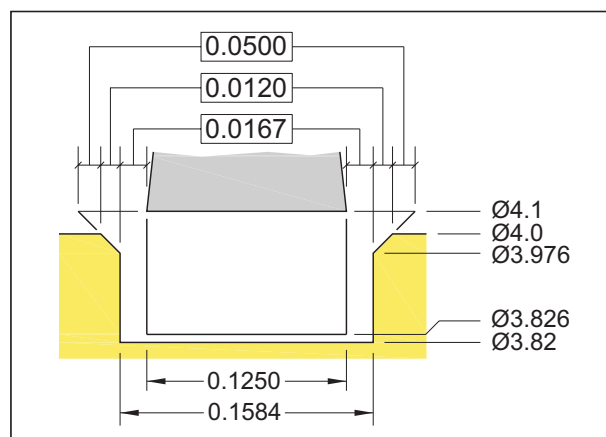


Figure 36-8

Precision groove - distribution of cuts for the example O3602



Before the first block can be programmed, selection of cutting tool and machining method is a sign of good planning. These are important decisions because they directly influence the final groove size and its condition.

### ◆ Groove Width Selection

Grooving tool selected for the example in program O3602 will be an external tool, assigned to tool station number three - T03. Tool reference point is selected at the *left edge* of the insert, which is a standard selection. Insert width has to be selected as well. Grooving inserts are available in a variety of standard widths, usually with an increment of 1 mm for metric tools, and 1/32 or 1/16 inch for tools in imperial system. In this case, non-standard groove width is 0.1584 inch. The nearest standard insert width is 5/32 inch (0.15625 inch). The question is - should we select the 5/32 inch insert width? In a short answer, no. In theory, this insert *could* cut the groove, but because the actual difference between the insert width and the groove width is so small (0.00215 inch over two walls), there is very little material to cut.

Dimensional difference would allow only slightly more than 0.001 per each side of the groove, which may cause the insert to rub on the wall rather than cut it. A better choice is to step down to the next lower standard insert width, that is 1/8th of an inch (0.1250). There is much more flexibility with 1/8 width than with 5/32 width. Once the grooving tool is selected, initial settings can be assigned - offset number (O3), spindle speed (400 ft/min), and gear range (M42) - and a note for the setup sheet:

- T0303 = 0.1250 SQUARE GROOVING TOOL

The first few program blocks can now be written:

```
O3602 (PRECISION GROOVE)
(G20)
```

```
...
N41 T0300 M42
N42 G96 S400 M03
...
```

### ◆ Machining Method

Once the grooving tool has been selected and assigned a tool station number (tool turret position), the actual *method* of machining the groove has to be decided. Earlier, the machining method has been described generally, now a more detailed description is necessary.

One simple programming method is *not* an option - the basic *in-out* technique used earlier. That means a *better* method must be selected, a method that will guarantee a high quality groove. The first step towards that goal is realization of the fact that a grooving insert with the width narrower than the groove width, will have to be plunged into the groove more than once. How many times? It is not difficult to calculate that a groove 0.1584 wide and machined

with a 0.1250 wide grooving insert, will need *at least* two grooving cuts. But what about a groove that is much wider than the groove in the example?

There is an easy way to calculate the *minimum number of grooving cuts* (or plunges), using the following formula:

$$C_{\min} = \frac{G_w}{T_w}$$

where ...

- $C_{\min}$  = Minimum number of cuts
- $G_w$  = Groove width for machining
- $T_w$  = Grooving insert width

Applying the formula to the example, the starting data are the groove width of 0.1584 of an inch and the grooving insert width of 0.1250 of an inch. That translates into the minimum of *two* grooving cuts. Always round upwards, to the nearest integer:  $0.1584/0.1250=1.2672=2$  cuts.

A possible decision could be to plunge once to finish the left side of the groove and, with one more plunge, to finish the groove right side. The necessary overlap between the two cuts is guaranteed and the only remaining operation is the chamfering. A groove programmed this way may be acceptable, but will *not* be of a very good quality.

Even if only an *acceptable* quality groove is produced during machining, such a result does not give the programmer much credit. What can be actually done to assure the *highest* groove quality possible?

In order to write first class programs, make the best efforts to deliver an exceptional quality at the **programming** level, in order to prevent problems at the **machining** level

How can this suggestion be used with the example? The main key is knowledge of machining processes. Machining experience confirms that removing an *equal* stock from each wall (side) of the groove will result in better cutting conditions, better surface finish control and better tool life.

If this observation is used in the current example, an important conclusion can be made. If two plunge cuts of uneven width will yield at least *acceptable* results, *three* cuts that are *equally distributed* should yield even better results.

If *at least three* grooving cuts are used to form the groove rather than the minimum two cuts, the CNC programmer will gain control of two always important factors:

- Control of the groove POSITION
- Control of the groove WIDTH

In precision grooving, these two factors are equally important and should be considered together.

Look carefully at how these factors are implemented in the example. The first factor applied under program control is the groove position. Groove *position* is given in the drawing as 0.625 inches from the front face of part, to the groove left side. There is no plus or minus dimensional tolerance specified, so the drawing dimension is used as arbitrary and is programmed directly. The second factor under program control is the groove *width*. That is 0.1584 of an inch on the drawing and the selected tool insert width is 0.1250. The goal is to program cutting motions in *three steps*, using the technique already selected:

#### ➔ STEP 1

Rough plunge in the groove middle, leaving an equal material stock on both groove faces for finishing - also leave small stock on the groove bottom

#### ➔ STEP 2

Program grooving tool operation on the groove left side, including the chamfer (corner break)

#### ➔ STEP 3

Program grooving tool operation on the groove right side, including the chamfer (corner break) and sweep groove bottom towards the left wall

The last two steps require chamfer cutting or a corner break. Width of the chamfer *plus* the width of any subsequent cut should never be larger than about one half to three quarters of the insert width. In the third step, sweeping of groove bottom is desired. That suggests the need to consider stock allowances for finishing.

### ◆ Finishing Allowances

During the initial step, the first plunge has to take place at the *exact* center of the groove. To calculate the Z-axis position for the start, first find the amount of stock on each wall that is left for finishing. The stock amount will be one half of the groove width minus the insert width - see details in the previous Figure 36-8:

$$(0.1584 - 0.1250) / 2 = 0.0167$$

The tool Z-position will be 0.0167 on the positive side of the left wall. If this wall is at Z-0.625, the grooving tool start position will be at Z-0.6083. When the tool completes the first plunge, there will be an equal amount of material left for finishing on both walls of the groove.

Do your best to avoid rounding off the figure 0.0167, for example, to 0.0170 inch. It would make no difference for the machining, but it is a sound programming practice to use *only* calculated values. The benefit of such approach is in eventual program checking, and also with general consistency in programming. Equal stock amounts offer this consistency; 0.0167 and 0.0167 is a better choice than 0.0170 and 0.0164, although the practical results will be the same.

Next look is at the X-axis positions. The first position is where the plunge will start from, the second position is the end diameter for plunging cut. A good position for the start is about 0.050 *per side* above the finished diameter, which in this case would be a clearance diameter calculated from the Ø4.0:

$$4.0 + 0.05 \times 2 = 4.1 \quad (\text{X4.1})$$

Do not start the cut with a clearance of more than 0.050 inch (1.27 mm) - with slow feedrates that are typical to grooves, there will be too much air to cut, which is not very efficient. End diameter is the groove bottom, given on the drawing as Ø3.82. Dimension of X3.82 could be programmed as the target diameter, but it does help to leave a very small stock, such as 0.003 per side (0.006 on diameter), to make a *sweep finish* of the groove bottom. That will add two times 0.003 to the 3.82 groove diameter, for the programmed X target as X3.826. Once the plunge is done, tool returns to the start diameter:

```
N43 G00 X4.1 Z-0.6083 T0303 M08
N44 G01 X3.826 F0.004
N45 G00 X4.1
```

Rapid motion back above the groove (N45) is a good choice in this case, because the sides will be machined later with the finishing cuts, so the surface finish of the walls is not critical at this moment. After roughing the groove, it is time to start the finishing operations.

All calculated amounts can be added to the previous Figure 36-8, and create data for a new Figure 36-9:

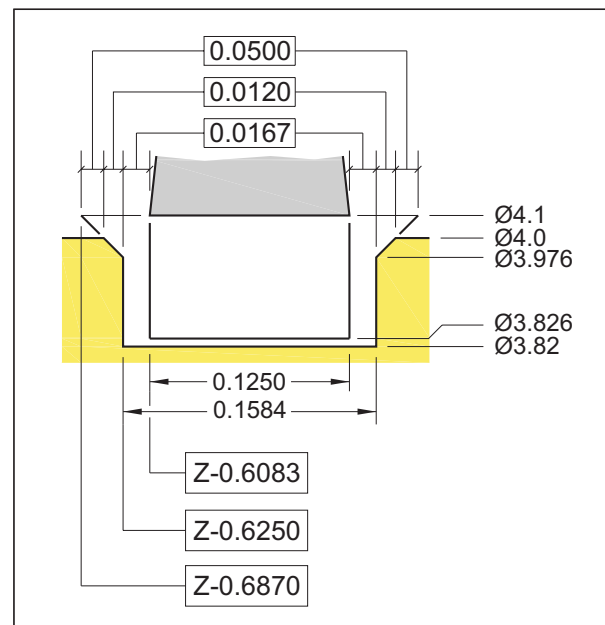


Figure 36-9

Precision groove - groove data used in program O3602

### ◆ Groove Tolerances

As in any machining, program for grooves must be structured in such a way, that maintaining tolerances at the machine will be possible. There is no specified tolerance in the example, but it is implied as very close by the four-decimal place dimension. A tolerance range, such as 0.0 to +0.001, is probably a more common way of specifying a tolerance. Only the dimensional value that falls *within* the specified range can be used in a program. In this example, the aim is the drawing dimension of 0.1584 (selected intentionally).

A possible problem often encountered during machining and a problem that influences the groove width the most, is a *tool wear*. As the insert works harder and harder, it wears off at its edges and actually becomes *narrower*. Its cutting capabilities are not necessarily impaired, but the resulting groove width may not fall within close tolerances. Another cause for an unacceptable groove width is the *insert width*. Inserts are manufactured within high level of accuracy, but also within certain tolerances. If an insert is changed, the groove width may change slightly, because the new insert may not have exactly the same width as the previous one. To eliminate, or at least minimize, the possible *out of tolerance* problem, use quite a simple technique - program an *additional* offset for finishing operations only.

Earlier, when the precision groove was planned, offset 03 had been assigned to the grooving tool. Why would an *additional* offset be needed at all? Assume for a moment, that all machine settings use just a single offset in the program. Suddenly, during machining, the groove gets narrower due to tool wear. What can be done? Change the insert? Modify the program? Change the offset? If the Z-axis offset setting is adjusted, either to the negative or positive direction, that will change the groove *position* relative to program zero but it will *not change the groove width!* What is needed is a second offset, an offset that controls the groove width only. In program O3602, the left chamfer and side will be finished with one offset (03), the right chamfer and side will use a second offset. To make the second offset easier to remember, number 13 will be used.

One other step has to be finished first - calculation of the left chamfer *start* position. Currently, the tool is at Z-0.6083 but has to move by the wall stock of 0.0167 and the chamfer width 0.012 as well as clearance of 0.050 - for a total travel of 0.0787, to Z-0.687 position. At a slow feedrate, the chamfer is done first and the cut continues to finish the left side, *to the same diameter* as for roughing, which is X3.826:

```
N46 Z-0.687
N47 G01 X3.976 Z-0.625 F0.002
N48 X3.826 F0.003
```

The next step is to return the tool above part diameter. This motion is more important than it seems. In the program, make sure the finished left side is not damaged when the tool retracts from the groove bottom. Also make sure

the grooving tool will not contact the right side wall stock. That means do not retract the tool further then the position of Z-0.6083. It also means do *not* rapid out, because of a possible contact during the 'dogleg' or 'hockey stick' motion, described on *page 149*. The best approach is to return to the initial start position at a relatively *high but non-cutting* feedrate:

```
N49 X4.1 Z-0.6083 F0.04
```

At this point, the left side wall is finished. To program similar motions for the right side wall, the tool has to cut with its *right* side (right edge). One method is to change the G50 coordinates in the program, if this older setting is still used, or use a different work coordinate offset. The method used here is probably the simplest and also the safest. All motions relating to the right chamfer and right side groove wall will be programmed in *incremental mode*, applied to the Z-axis only, using the W address:

```
N50 W0.0787 T0313
N51 X3.976 W-0.062 F0.002
```

In block N50, the tool travels the total distance equivalent to the sum of right wall stock 0.0167, chamfer of 0.012 and clearance of 0.050. In the same block, the second offset is programmed. This is the only block where offset 13 *should* be applied - one block before, it's too early, and one block, after it's too late.

Block N51 contains the target chamfer position and absolute mode for X-axis and is combined with incremental mode for Z-axis.

To complete the groove right side wall, finish the cut at the full bottom diameter, block N52, then continue to remove the stock of 0.003 from bottom diameter (block N53) - this is called *sweeping the groove bottom*:

```
N52 X3.82 F0.003
N53 Z-0.6247 T0303
```

Also look at the Z-axis end amount - it is a small value that is 0.0003 short of the 0.625 drawing dimension! The purpose here is to compensate for a possible tool pressure. *There will not be a step in the groove corner!* Because the sweep will end at the groove *left* side, the original offset (03) must be reinstated. Again, block N53 is the only block where the offset change is correct. Make sure not to change tool numbers - *the turret will index!*

The intended program O3602 can now be completed. All that remains to be done is return to the groove starting position, followed by program termination blocks:

```
N54 X4.1 Z-0.6083 F0.04
N55 G00 X10.0 Z2.0 T0300 M09
N56 M30
%
```

At this point, *complete* program O3602 can be written. Note program blocks where the offset has been changed, they are identified in the comment section:

```
O3602 (PRECISION GROOVE)
(G20)
...
N41 T0300 M42 (NO OFFSET)
N42 G96 S400 M03
N43 G00 X4.1 Z-0.6083 T0303 M08 (OFFSET 03)
N44 G01 X3.826 F0.004
N45 G00 X4.1
N46 Z-0.687
N47 G01 X3.976 Z-0.625 F0.002
N48 X3.826 F0.003
N49 X4.1 Z-0.6083 F0.04
N50 W0.0787 T0313 (OFFSET 13)
N51 X3.976 W-0.062 F0.002
N52 X3.82 F0.003
N53 Z-0.6247 T0303 (OFFSET 03)
N54 X4.1 Z-0.6083 F0.04
N55 G00 X6.0 Z3.0 T0300 M09 (NO OFFSET)
N56 M30
%
```

#### WARNING !

It is very important to use caution when a double tool offset for a single tool is used during machining ( this warning applies generally - not only for grooving )

Remember that the *purpose* of offset in the example is to control the groove *width*, not its diameter.

Always follow these precautions, based on the example program O3602:

- Start machining with identical initial amounts assigned to both offsets (the same XZ values for offsets 03 and 13).
- X-offset amounts of 03 and 13 must always be the same. If the X-setting of one offset is changed, setting of the other offset must be changed to the same amount. Adjust both X-offsets to control the groove depth tolerance
- If the groove width becomes too narrow and has to be adjusted, only the Z-offset amount is changed
- To adjust the groove left side wall position, change the Z-offset number 03
- To adjust the groove right wall position, change the Z-offset number 13
- Do not cancel the current offset - change from one to the other offset directly
- Make sure the tool number (the first two digits of the T address) does not change, otherwise, **THERE WILL BE A TOOL CHANGE!**

Other precautions can be added, depending on the exact conditions. Use common sense, and always check the program carefully, before it is released to production. If a tool number is changed in the program, **change all T-words!**

### ◆ Groove Surface Finish

Programming just about any precision groove should be fairly easy from now on. Only a few last notes on the subject of groove cutting as they relate to surface finish. Just by following the suggested methods of equal cut distribution, proper spindle speeds and feedrates, good condition of the cutting tool and insert, suitable coolant, and other techniques used in the example, surface finish will almost take care of itself.

Keep in mind, that the term '*precision groove*' does not only describe the precise groove position and its precise dimensions, it also means a high quality look, a look that often means much more than just a cosmetic feature.

## MULTIPLE GROOVES

Multiple grooving is a common term used for cutting the same groove at different positions of the same part. In these cases, the program will most likely benefit from developing a subprogram (subroutine) for multiple grooves, that will be called at various groove locations. Subprograms save valuable programming time, they are easily designed and easily edited. Although subprograms will be discussed in a separate chapter (*see page 383*), an example of a multiple groove programming using a subprogram is shown at the end of this chapter, at least for reference and basic introduction.

When cutting multiple grooves, more material will be removed. On external diameter grooves, there are no special considerations necessary, gravity will take care of the extra chips. This is not the same situation for internal grooves. The moment several grooves are machined internally, there is a small pile of cutting chips accumulated in the bored hole. These chips can be in the way of a smooth cutting operation and could damage the bored diameter and even the grooving tool itself. To solve this problem, consider machining of only a few grooves, move the tool out and blow out chips from the internal area. Using the optional stop M01 can be useful in this case. When all chips have been removed, continue with the same tool to cut more grooves.

## FACE GROOVES

Face grooving (sometimes incorrectly called *trepanning*) is a horizontal groove cutting process, with the tool moving along the Z-axis. Tool is programmed along the same principles as vertical grooving along the X-axis. Because of the nature of such a grooving cut, tool *orientation* presents the most important single consideration in face grooving. Main issue is the *radial clearance* of the cutting insert, during a cut. There is no need to worry too much about radial clearance for vertical grooving, because the insert cutting edge is on the same plane as machine center line. However, in horizontal grooving, the insert clearance along the cut radius is of utmost importance.



The next example shows how to program a typical face groove, and is illustrated in *Figure 36-10*.

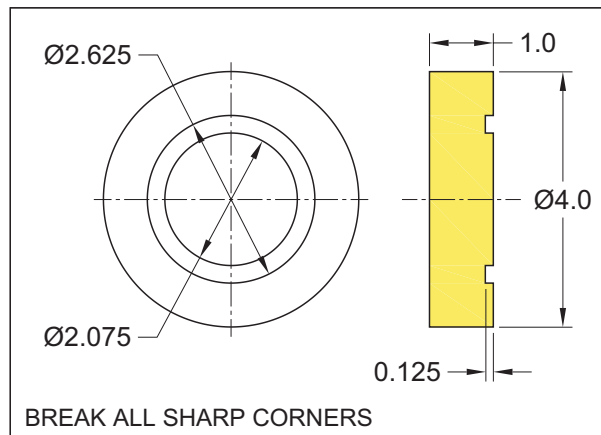


Figure 36-10

Face grooving example - program O3603

Although both external and internal groove diameters are engineering choices in the drawing, physical groove width is necessary for programming as well. To calculate the groove width, use a simple calculation - find one half of the difference between the two grooving diameters - that is:

$$(2.625 - 2.075) / 2 = 0.275$$

This is the actual groove width amount, 0.275 in the given example. Always keep in mind that the program will use a smaller 0.250 wide face grooving insert. Following the programming examples of precision groove listed earlier, three cuts will be made - one rough plunge in the middle of the groove, and two finishing cuts, with a small corner break. But first, let's look at the *radial clearance* of the grooving tool. This is a very important programming consideration; one that is unique to most face grooving operations, yet, it is also one that is easy to be overlooked.

#### ◆ Radial Clearance

Many grooving inserts are relatively high, in order to give them strength. Grooving insert for face grooving operations is mounted at 90° towards the part face (parallel to the spindle center line). A standard grooving insert has virtually no radial clearance and most likely will interfere with the part at its lower end - *Figure 36-11*.

From the illustration is obvious that the grooving insert cannot be used as is and has to be modified. Such a modification is usually done by grinding a suitable radial clearance, as illustrated in *Figure 36-12*.

This is a simple operation, providing the proper grinding tools are available. Make sure that the grinding does not affect the insert width and only minimum necessary material is removed, otherwise the tool loses strength.

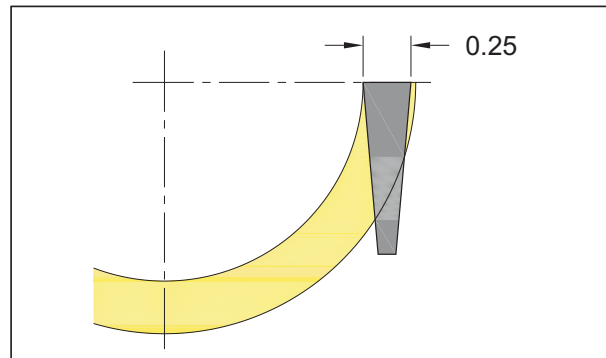


Figure 36-11

Interference of a standard grooving insert on a face groove

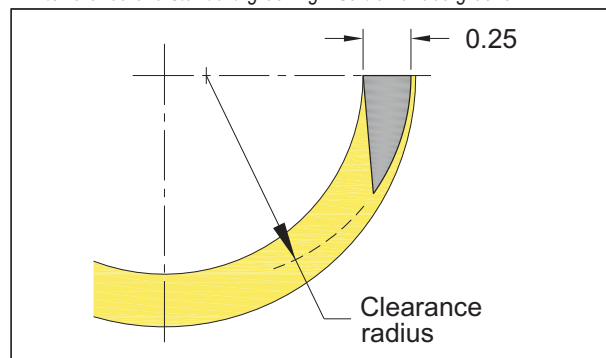


Figure 36-12

Standard grooving insert modified for face grooving

#### ◆ Face Grooving Program Example

Program O3603 uses modified insert and a 0.012 corner break, to eliminate sharp corners. Only one offset is used in the program. The tool set point is the lower edge of insert, corresponding to the Ø2.075. All calculations should be easy to retrace, they use exactly the same procedures as those described for vertical grooving:

O3603 (FACE GROOVE)  
(G20)

```

...
N21 T0400 M42
N22 G96 S450 M03
N23 G00 X2.1 Z0.05 T0404 M08
N24 G01 Z-0.123 F0.003
N25 Z0.05 F0.04
N26 X1.951
N27 X2.075 Z-0.012 F0.001
N28 Z-0.123 F0.003
N29 X2.1 Z0.05 F0.04
N30 U0.149
N31 U-0.124 Z-0.012 F0.001
N32 Z-0.125 F0.003
N33 X2.0755
N34 X2.1 Z0.05 F0.04 M09
N35 G00 X8.0 Z3.0 T0400
N36 M30
%
```



## CORNER GROOVES / NECK GROOVES

Corner grooving is also a grooving operation, one that uses a special grooving insert designed to cut along a 45° angular motion. Resulting groove can be square or with a radius, depending on the tool and insert used and design required. Grooving insert may also be a standard type insert, placed into a 45° angle tool holder. The purpose of this type of a groove is to allow machining of recesses and undercuts, usually in a corner of the part. It assures a shoulder match of two assembled components.

To program a corner groove (neck groove), the radius of the grooving insert must be known, 0.031 (1/32) of an inch in the example. Cutting depth is established from the drawing data. Normally, the corner groove is specified as a 'minimum undercut'. In this case, the center of undercut will be at the intersection of the shoulder and diameter. Cutting motion in and out of the groove must be at 45°, meaning an identical amount of travel in both X and Z axes. *Figure 36-13* illustrates a corner groove with a 0.031 radius minimum undercut.

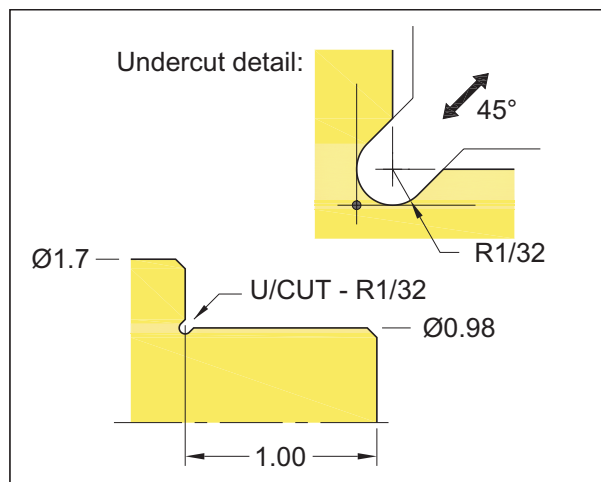


Figure 36-13

Corner groove - undercut program example O3604

The program itself has no hidden surprises and is not difficult to complete or interpret:

```
O3604 (CORNER GROOVE)
(G20)
...
N217 G50 S1000 T0500 M42
N218 G96 S375 M03
N219 X1.08 Z-0.95 T0505 M08
N220 G01 X0.918 Z-1.031 F0.004          (CUT IN)
N221 G04 X0.1                          (DWELL)
N222 X1.08 Z-0.95 F0.04                 (MOVE OUT)
N223 G00 X6.0 Z3.0 T0500 M09
N224 M30
%
```

Block N219 positions the tool in such a way that the insert center (as well as the setup point) is in on center line of the neck groove (0.050 clearance in X and Z axes). Blocks N220 and N222 are the two cutting motions - one into the groove in N220, the other out of the groove in N222. The amount of travel is exactly the same in either direction. dwell of 0.1 second is added for convenience at the neck bottom. Block N220 can also be programmed as an *incremental* motion for easier visualization:

```
N220 G01 U-0.162 W-0.081 F0.004
N221 G04 X0.1
N222 U0.162 W0.081 F0.04
```

## GROOVING CYCLES

Fanuc controls for lathes offer two multiple repetitive cycles G74 and G75 that can be used for an interrupted cutting along an axis. Programming formats for both cycles were described in the previous chapter (*page 332*). G74 cycle is used for cutting along the Z-axis and is used mostly for peck drilling, G75 is used for cutting in the X-axis, and is used mostly for simple grooving.

### ◆ G75 Cycle Applications

Although used mainly for grooving, G75 cycle can also be used for an interrupted cut in facing. This cycle is quite simplistic to be of any use for high quality surface finish, but it does have its benefits. Its main purpose is to break chips while cutting along the X-axis. This may be useful for some grooving and part-off operations, as well as face cutting. Another use is for roughing the core out of deep grooves, so they can be finished later by more precise methods.

In G75, the chip breaking is done by alternating between a cutting motion in one direction and a rapid retract motion in the opposite direction. This means that one cutting motion is always followed by an opposite rapid motion, on the basis of *feed-in-rapid-out* principle and a built-in clearance. *Figure 36-14* illustrates the concept.

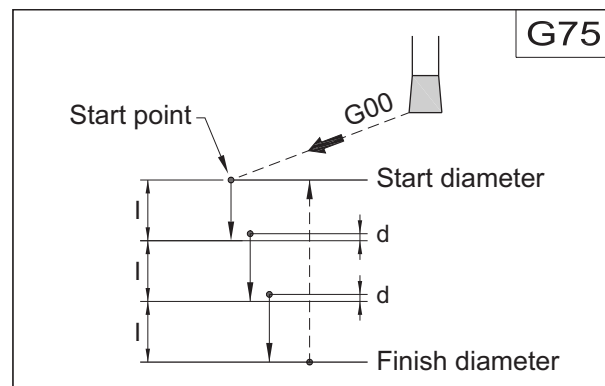


Figure 36-14

Schematic representation of the G75 cycle

Motion retract amount is built within the cycle and is set by an internal parameter of the control system. In *Figure 36-14* it is identified by the value  $d$ , (usually set to approximately 0.010 to 0.020 inches in the control). The next two examples illustrate the practical use of G75 grooving cycle.

#### ◆ Single Groove with G75

A single groove requires the X and Z coordinate of the starting point, the final groove diameter X, and the depth of each cut I. For a single groove, the Z-axis position and the K distance cannot be programmed. The Z position is given by the starting point and does not change.

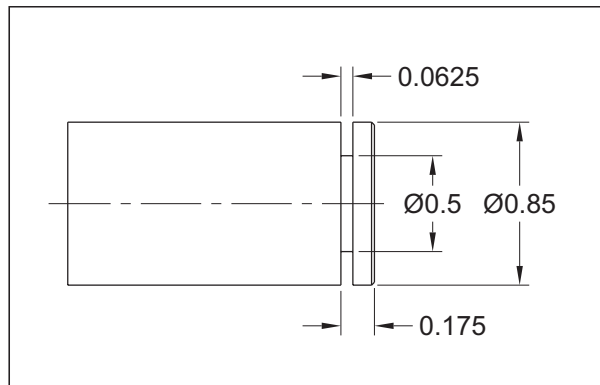


Figure 36-15

Single groove example using the G75 cycle - program O3605

The following program example O3605 cuts a single groove and is based on *Figure 36-15*.

```
O3605
(G75 SINGLE GROOVE)
(G20)
...
N43 G50 S1250 T0300 M42
N44 G96 S375 M03
N45 G00 X1.05 Z-0.175 T0303 M08
N46 G75 X0.5 I0.055 F0.004
N47 G00 X6.0 Z2.0 T0300 M09
N48 M30
%
```

Note that the I amount is 0.055. This is not a value without a meaning. In fact, it is a carefully calculated depth of each groove peck. The tool travel will be from Ø1.050 to Ø0.500, or 0.275 per side  $(1.05-0.50)/2=0.275$ . There will be exactly five grooving pecks of 0.055 each  $(0.275/5=0.055)$ .

#### ◆ Multiple Grooves with G75

It is possible to program multiple grooves very easily, using G75 cycle. In this case, the groove spacing, the pitch between grooves must always be equal, otherwise G75 cycle cannot be used. The clearance specification  $d$  in *Figure 36-14* is normally not programmed.

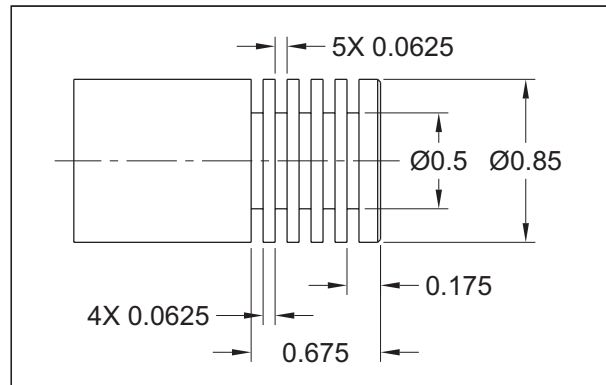


Figure 36-16

Multiple groove example using the G75 cycle - program O3606

Program example O3606 for multiple grooves, using G75 cycle, is based on *Figure 36-16*.

```
O3606
(G75 MULTIPLE GROOVES)
(G20)
...
N82 G50 S1250 T0300 M42
N83 G96 S375 M03
N84 G00 X1.05 Z-0.175 T0303 M08
N85 G75 X0.5 Z-0.675 I0.055 K0.125 F0.004
N86 G00 X6.0 Z2.0 T0300 M09
N87 M30
%
```

Setup and conditions for multiple grooves are identical to those for a single groove. The only difference is additional entries in the G75 cycle call.

This technique may be used not only for multiple grooves separated by solid material, but also for opening up a single groove that is much wider than the grooving insert. The only difference in programming will be the amount of K - the distance between grooves. If K is greater than the insert width, several individual grooves will be cut. If K is equal to or smaller than the insert width, a single wide groove will be cut. Experiment to find the best amounts.

## SPECIAL GROOVES

There are many more types of grooves than can be described in this handbook. They are mainly grooves of special shapes, used by specific industries - grooves that serve a certain purpose. The most typical grooves of this type are round grooves, pulley grooves, O-ring grooves, and several others. Certain grooves, usually those that conform to common industrial standards, can be machined with readily available inserts. A typical example of this kind of grooving is a pulley groove. The programming principles for 'nonstandard' grooves are no different than those described in this chapter.

## GROOVES AND SUBPROGRAMS

Programming multiple grooves with G75 cycle is usually not the preferred method for precision work. The two main drawbacks are the groove quality and the requirement for an equal spacing between grooves. There is another method to program multiple grooves - much more efficient method - one that uses subprograms.

Multiple grooves can be programmed very efficiently and with much increased precision by using the technique of subprograms, a subject starting on *page 383*. The guiding principle is to program all common groove motions in the subprogram and all motions that vary from groove to groove, in the main program. This way, the same groove can be repeated at fixed intervals or variable intervals, as needed.

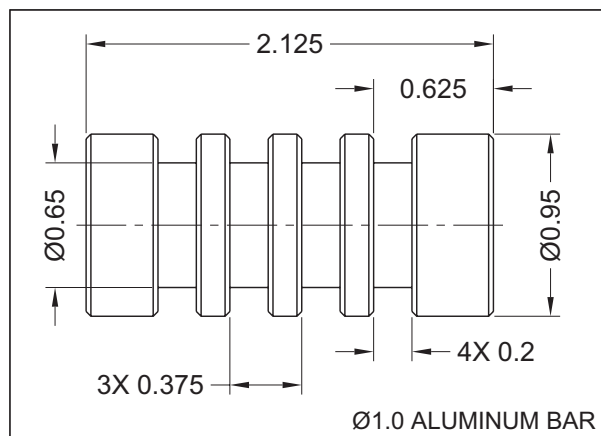


Figure 36-17

Multiple grooves programming using a subprogram -  
O3607 is the main program and O3657 is the subprogram

In *Figure 36-17* is a simple example of a multiple groove programming, using a subprogram. Only two cutting tools are used - a turning tool used for facing and turning and a 0.125 wide part-off tool that machines the four grooves, cuts the back chamfer and parts-off the finished part. Part-off operations are described in the next chapter. Note that all tool motions related to the groove position are programmed in the main program O3607, all tool motions related to the actual groove cutting are programmed in subprogram O3657. Equal spacing between grooves is used for the example.

```
O3607 (GRV W/SUB-PROG)
(T01 - 55 DEGREE DIAMOND INSERT)
N1 G20 T0100
N2 G96 S500 M03
N3 G00 X1.2 Z0 T0101 M08
N4 G01 X-0.07 F0.006 (FACE OFF)
N5 G00 Z0.1
N6 G42 X0.7 (START OF CHAMFER)
N7 G01 X0.95 Z-0.025 F0.003 (CHAMFER)
N8 Z-2.285 (TURN OD)
N9 U0.2 F0.03
N10 G00 G40 X4.0 Z4.0 T0100 M09
N11 M01
```

```
(T05 - 0.125 PART-OFF TOOL)
N12 G50 S2500 T0500
N13 G96 S500 M03
N14 G00 Z-0.5875 T0505 M08 (POS-GRV1)
N15 X1.0
N16 M98 P3657 (CUT GRV 1)
N17 G00 W-0.375 M98 P3657 (CUT GRV 2)
N18 G00 W-0.375 M98 P3657 (CUT GRV 3)
N19 G00 W-0.375 M98 P3657 (CUT GRV 4)
N20 G00 Z-2.285 (OPEN UP FOR PART-OFF)
N21 G01 X0.8 F0.006
N22 X1.1
N23 G00 X1.0 Z-2.2 (CHAMFER BACK START)
N24 G01 X0.9 Z-2.25 F0.003 (CHAMFER)
N25 X-0.02 F0.005 (PART-OFF)
N26 G00 X1.2 (CLEAR)
N27 G40 X4.0 Z4.0 T0500 M09
N28 M30
%
```

```
O3657 (SUB-PROG FOR O3607)
N101 G01 X0.66 F0.004 (FEED TO ROUGH OD)
N102 G00 X1.0 (CLEAR OUT)
N103 W-0.0875 (SHIFT TO LEFT CHFR)
N104 G01 X0.9 W0.05 F0.002 (LEFT CHFR)
N105 X0.66 F0.004 (FEED TO ROUGH OD)
N106 X1.0 W0.0375 F0.03 (BACK TO START)
N107 W0.0875 (SHIFT TO RIGHT CHFR)
N108 X0.9 W-0.05 F0.002 (RIGHT CHFR)
N109 X0.65 F0.004 (FEED TO FINISH OD)
N110 W-0.075 (SWEEP BOTTOM)
N111 X1.0 W0.0375 F0.03 (BACK TO START)
N112 M99 (RETURN TO MAIN)
%
```

This example completes the chapter related to grooving. Although grooving is a relatively simple machining operation, programming grooves can become a significant challenge in certain cases.

## 37

## PART-OFF

*Part-off*, sometimes called a *cutoff*, is a machining operation typical to lathe work, usually using a barfeeder attachment. During a part-off, the cutting tool (or part-off tool) separates the completed part from the bar stock. The completed part will fall off the bar, usually into a special bin to protect it from damage. Part catcher may also be used.

### PART-OFF PROCEDURE

Programming procedure for a part-off toolpath is very similar to the grooving procedure. In fact, part-off is an extension of grooving. The purpose of part-off is somewhat different, because its objective is to separate the completed part from the stock material, rather than create a groove of certain width, depth and quality. Material bar stock is usually a long round/hex rod that is 8, 10, 12 or more feet long.

Two most important considerations in part-off are the same as those for standard grooving. One is the chip control, the other is coolant application.

#### ◆ Parting Tool Description

Part-off uses a special cutting tool. Such a tool used for part-off is called a parting tool or a part-off tool. Sometimes the term *cutoff* is used for this kind of a tool, as well as the machining method; it has the same meaning as the term *part-off*. Part-off tool is similar in design to a grooving tool, with one major difference. The *length* of the cutting blade is much longer than that of a grooving tool, making it suitable for deep grooves. A typical example of a part-off tool is illustrated in *Figure 37-1*.

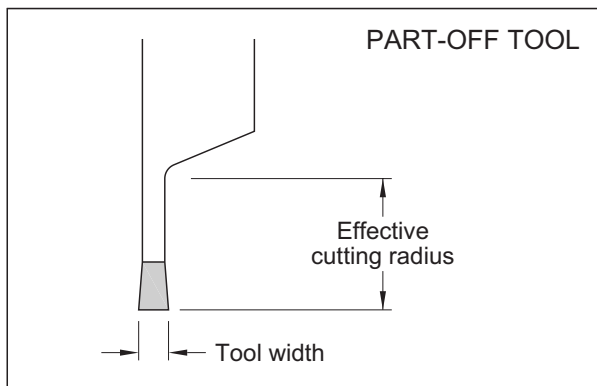


Figure 37-1  
Part-off tool - cutting end configuration

At the end of the metal blade is usually a carbide insert, with clearance angles on both sides. The tool cutting end is available in several different configurations, always at the end tip of the carbide portion. The most typical tool end configurations are shown in the following illustration - *Figure 37-2*:

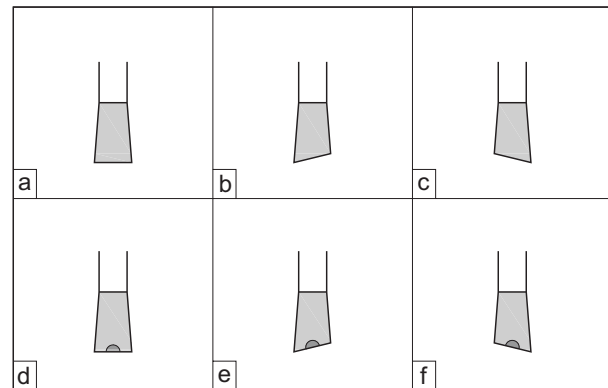


Figure 37-2  
Part-off tool - cutting tip configurations

Note the two kinds of each grooving insert design shown - the series without a dimple (items *a*, *b* and *c*), and the series with a dimple (items *d*, *e* and *f*). The dimple is an intentional dent pressed in the middle of the cutting edge that deforms the chip and helps in coiling it. The result is a chip that is narrower than the width of cut. Such a chip does not clog the generated groove and extends tool life, although it may cost a little more.

Also note a slight angle on *b*, *c*, *e* and *f* styles. This angle helps in controlling the size and shape of the stub left on the part when it is separated from solid bar. It also controls the rim size that is left over on the part when parting-off a tubular bar. Although all designs have their special applications, probably the most versatile choice would be the style *f*, particularly for large cutting diameters. Unlike in the other types of machining, the cutting chips for part-off should *coil*, not break. The cutting insert with a dimple or a similar design is best suited for that purpose.

It is a common practice amongst programmers to use only one parting tool for all the work. They select the parting tool long enough to accommodate the maximum bar diameter and leave it permanently mounted in a tool holder, even for small diameters. The reasoning for this approach is that it saves a setup time. That is true to some extent but has a downside as well. A long part-off tool usually has a

wider insert than a short tool, in order to compensate for strength and rigidity. When the bar size is large, a long part-off tool is necessary, with its relatively wide insert. If such a tool is used for short parts, such as rings or other tubular stock with thin walls, it is the wrong tool selection that also wastes material. A short part-off tool with a narrower insert will justify the setup change.

A generous supply of coolant should always be made available at the cutting edge, just like for grooving. A water soluble coolant is a good choice, since it offers both the cooling and lubricating qualities. A typical mixture would be one part of soluble oil for 15-20 parts of water or as recommended by the coolant manufacturer. Make sure the coolant is supplied at full pressure, particularly for larger diameters. High pressure helps the coolant to reach the cutting edge and flush off chips that may accumulate in the opening.

### ◆ Tool Approach Motion

The first step to program a part-off tool path is to select a part-off tool that has enough capacity to completely separate the part from a solid bar. The next decision is to select the insert width and location of the tool reference point. A part-off tool that is too short will not reach the spindle centerline safely. A tool too long may not be rigid enough, may cause vibrations, even break during the cut. The insert width is also important for good cutting conditions. Width of the tool is directly proportionate to the cutting depth capacity.

Selection of the part-off tool reference point follows the same rules as those for a grooving tool. It is definitely advantageous to have the same side for both, grooving and parting, to maintain setup consistency. The following programs illustrate the difference between the tool reference point on the *left* and on the *right* side of the tool tip - Figure 37-3 for program O3701 and Figure 37-4 for program O3702. In both cases, program zero is the front face of the finished part.

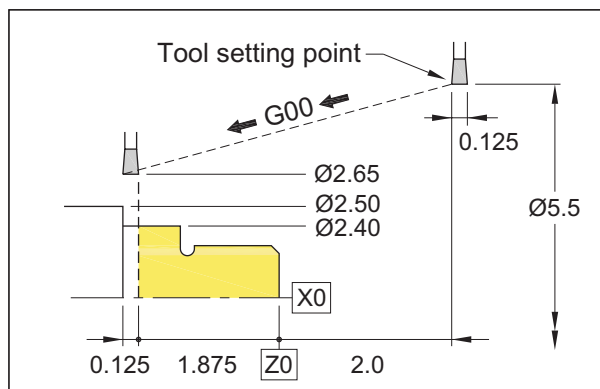


Figure 37-3

Part-off tool approach - left side tool reference - program O3701

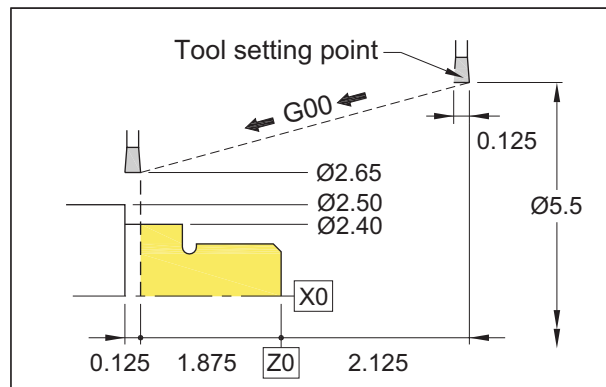


Figure 37-4

Part-off tool approach - right side tool reference - program O3702

In both examples, the tool change position and the final results are identical. Comparison of both programs shows dimensions of X-axis unchanged, but dimensions for the Z-axis are different (blocks N122 and N125). This reflects the cutting side of the tool tip.

#### O3701 (PART-OFF / LEFT SIDE TOOL EDGE)

```
...
N120 G50 S1250 T0800 M42
N121 G96 S350 M03
N122 G00 X2.65 Z-2.0 T0808 M08
N123 G01 X-0.03 F0.004
N124 G00 X2.65 M09
N125 X5.5 Z2.0 T0800
N126 M30
%
```

This example is consistent with the previous suggestions for precision grooving. Setting up the tool reference point on the *left* side of the tool is easier at the machine. If there is a good reason, set the tool reference point on the right side and program according to the Figure 37-4.

#### O3702 (PART-OFF / RIGHT SIDE TOOL EDGE)

```
...
N120 G50 S1250 T0800 M42
N121 G96 S350 M03
N122 G00 X2.65 Z-1.875 T0808 M08
N123 G01 X-0.03 F0.004
N124 G00 X2.65 M09
N125 X5.5 Z2.125 T0800
N126 M30
%
```

One weakness of the left side tool setting is that the insert width has to be always added to the Z position in the program. In the second example, the final length of the part is used directly, but a possible collision with the chuck or collet is strong. Take care selecting the X-axis tool approach position and program the tool *above* stock diameter, even if the previous turning operations removed most of the stock. Figure 37-5 shows correct and incorrect approaches of a part-off tool.



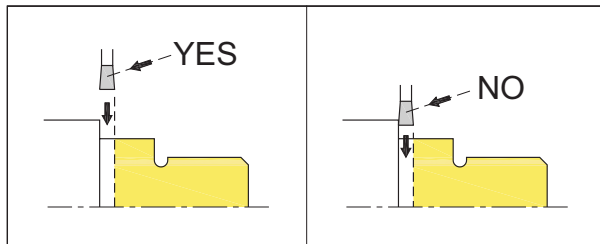


Figure 37-5

Correct and incorrect approach to stock diameter

### ◆ Stock Allowance

Part-off operation does not always mean all machining has been completed. Often, part-off may complete only the first operation and additional machining will be necessary on the machined part. In such an event, some extra material (stock) has to be left on the back face, for subsequent finishing. Leave a stock amount of about 0.010 to 0.020 inches (0.3 to 0.5 mm). In that case, the block N122 would be changed in both programs - for example, from Z-2.0 to Z-2.015 in the first program example O3701 and from Z-1.875 to Z-1.89 in the second program example O3702.

Another program entry that is important to look at is the programmed X amount in block N122 - it is X2.65 in the example. That will leave 0.125 inches actual clearance above the 2.4 cutting diameter. If that seems a little too much, think again. Always consider the *actual* bar stock diameter, for safety reasons. In the example, the bar stock diameter is 2.500 inches and the actual clearance will be a more reasonable 0.075 of an inch per side of the stock.

### ◆ Tool Return Motion

Another safety aspect of programming a part-off tool is the method of returning to the tool change position, when parting operation is completed. It may be very tempting to replace the two program blocks N124 and N125 with a single block, then return to the tool change position immediately after the part-off - for example:

```
N124 G00 X5.5 Z2.0 (or Z2.125) T0800 M09
```

After all, the part has just been separated, fallen into the bin and one block in the program can be saved. *Don't do this*, it could become a very hazardous procedure. The part *should* have been removed by the tool and it *should* have fallen into the bin - but has all this actually happened? A variety of reasons may cause an *incomplete* part-off. The result is a broken tool, scrapped part, possibly a damage to the machine itself.

Always return in the X-axis first,  
and always above the bar stock diameter

### ◆ Part-off with a Chamfer

Not always the machined part will be done during a secondary operation. When the machining has to be completed with a part-off tool, it will require the best quality overall finish possible. One requirement of a good surface finish is broken sharp corners. In the example, a sharp corner is at the intersection of X2.4 and Z-1.875. If the turning tool cannot cut the chamfer during turning operation, part-off tool can be a good choice. Most part-off tools are not designed for cutting sideways (along the Z-axis), but chamfering removes only a small amount of material that is within the tool capabilities. Avoid chamfers that are wider than about 75% of the insert width or take several cuts if needed. The chamfer has to be cut *before* part-off and it should be cut from outside in, *not* from inside out. Correct programming technique for machining a chamfer during part-off is summed up in the following steps:

- Position the tool further in the Z-axis than would be normal for regular part-off
- Start the part-off operation and terminate it just below the diameter where the chamfer will end
- Return to the starting diameter and move to the chamfer start position
- Cut the chamfer in one block and part-off in the subsequent block

To illustrate this programming technique, study the following program example O3703 and illustration shown in Figure 37-6 - tool reference point is on the left side, and the required chamfer is 0.020 inches at 45°:

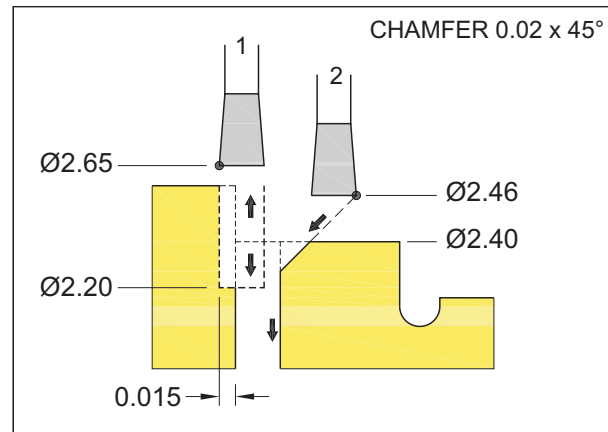


Figure 37-6

Corner breaking with a part-off tool - example O3703

O3703 (PART-OFF CHFR)  
(G20)

```
...
N120 G50 S1250 T0800 M42
N121 G96 S350 M03
N122 G00 X2.65 Z-2.015 T0808 M08
```

```

N123 G01 X2.2 F0.004
N124 X2.46 F0.03
N125 Z-1.95 (LEFT SIDE OF TOOL)
N126 U-0.1 W-0.05 F0.002
N127 X-0.03 F0.004
N128 G00 X2.65
N129 X5.5 Z2.0 T0800 M09
N130 M30
%
```

In block N122, the tool is positioned 0.015 past the position Z-2.0. Block N123 makes only a temporary groove - to Ø2.2. The next block (N124) is a motion out of the temporary groove, to the starting diameter of the chamfer Ø2.46. In the following block N125, tool shifts in the Z-axis, to the chamfer start position. The amount of 1.950 was calculated by additions and subtractions:

$$1.875 - 0.020 - 0.030 + 0.125 = 1.950$$

The amount of 1.875 is the part back face (as per drawing), 0.020 amount is the chamfer size; 0.030 is the clearance, and 0.125 is the insert width. Note the 0.125 tool width position adjustment, to maintain the tool reference point on the *left* side of the cutting edge while actually cutting with the right edge. Block N126 is the chamfer cutting, using incremental mode. Using incremental mode saves a few calculations. If using absolute mode, block N126 will be calculated:

```
N126 X2.36 Z-2.0 F0.002
```

Also note the decreased feedrate for the chamfer only, to assure a good finish. Feedrate decrease can be quite significant for very small chamfers. The program remaining blocks are unchanged.

In some cases, two tools can be justified for part-off operations. Setup of the two tools has to be accurate. A

small, rigid grooving tool can do the startup groove and the chamfer, then a part-off tool can do the rest. At the completion of the part-off, bar stock projecting from the spindle will have a small step (tip). Make sure to program a facing cut for each subsequent part to take this step (tip) into consideration.

### ◆ Preventing Damage to the Part

When the part is separated from the bar, it falls down. On impact, it may suffer a damage severe enough to make a good part a scrap. To prevent the possibility of a damage, CNC lathe operator may want to place a bucket filled with cutting coolant in the path of the falling part. Another method is to offset the part-off tool away from the centerline, just far enough that it does not separate the part. Then, when the machine is stationary (*i.e.*, not moving), CNC operator breaks the part manually.

In any case, always follow safety rules of the company - they exist for your protection.

Never touch the part while the program is in operation or the spindle is rotating

The best solution for part damage prevention is a CNC lathe equipped with a parts catcher, which is often a special machine option, ordered at the time of machine purchase.

For part-off, just like for grooving operations, always make sure there is an adequate supply of inserts on hand. Tools with sharp edges, or with very small radii, are generally weak tools, yet doing some very demanding work. Nobody wants to run out of tools in the middle of a very important rush job.

## 38

## SINGLE POINT THREADING

Threading is a machining process used to produce a helical groove of a given shape and size, usually on a cylinder (straight thread), or on a cone (tapered thread). The major purpose of threads is to connect two parts together without damage during joining and disjoining (assembly and disassembly). The most common applications of threading fall into four major categories:

- Fastening devices ... screws, bolts and nuts
- Measuring tools ... micrometer barrel
- Motion transmission ... lead screw, camera lenses
- Torque increase ... lifting or supporting jacks

Thread cutting is a very versatile manufacturing process. There are two main groups of thread production - metal cutting and plastic molding. It should not be a surprise that it is the plastic molding method that dominates manufacturing industry. Given the number of detergent bottles, pop bottles and other plastic products we consume, the number of threaded products using this method is astronomical.

In metalworking area of thread production - the primary group of interest - there are smaller several subgroups:

- Thread rolling
- Thread forming
- Thread grinding
- Tapping and die work
- Thread milling
- Single point thread cutting

For a typical CNC programmer, the areas of interest are usually confined to the last three - tapping (but no die work), thread milling and single point threading. Methods for tapping operations have been described in *Chapters 25* and *26*, thread milling will be described in *Chapter 45*. This chapter covers programming methods described as *single point threading*.

## THREADING ON CNC LATHES

In a *single setup*, CNC lathes can produce very high quality threads, in addition to turning, boring, grooving and other operations. This is a very attractive feature for manufacturers and many machine shops have purchased a CNC lathe for that reason alone. Any secondary operation requires additional setup, increasing the cost of production.

Single point thread cutting - typically known as a *single point threading* - uses a tool holder similar to other tool holders, but contains a special threading indexable insert, which may have one, two or three tips. Generally, the threading insert shape and size must match the shape and size of finished thread - *Figure 38-1*.

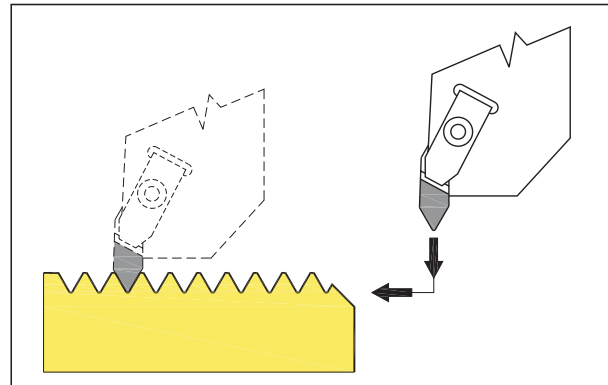


Figure 38-1

Comparison of the thread form and the threading tool shape

By definition, standard single point threading is a unique machining process of cutting a helical groove of a specific shape with uniform advancement per spindle revolution. The thread shape - or *form* - is mainly determined by the shape and mounting position of cutting tool. The uniformity of advancement is set by programmed feedrate.

### ◆ Form of a Thread

The most common thread form used in CNC programming is the familiar V-thread (in the shape of letter V) with a 60° included angle. There is a large variety of V-shape thread forms used in manufacturing, including metric and imperial threads. Other forms include trapezoidal shapes, such as metric trapezoid, Acme and worm threads, square and round threads, buttress threads and many others. In addition to these relatively common forms, there are threads specific to a particular industry, such as automotive, aircraft, military, and petroleum industries. To make matters even more interesting, the thread shape can be oriented on a cylindrical surface, a conical surface, it also can be external or internal. The thread can be cut on a face (scroll threads), even on circular surfaces. It can have a single or multiple starts, right or left hand orientation, constant or variable lead, etc. *Many programmers joke that CNC threading can be a breeze or a nightmare with nothing between.*

## ◆ Threading Operations

This section contains a detailed list of various threading operations that can be programmed for a common CNC lathe. Several operations require a special type of threading insert and some operations can only be programmed if the control system is equipped with special (optional) features:

- Constant lead threads
- Variable lead threads
- External and internal threading
- Cylindrical threads (straight threads)
- Tapered threads (conical threads)
- Right Hand (R/H) and Left Hand (L/H) threads
- Face threads (scroll threads)
- Single-start threads
- Multi-start threads
- Circular threads
- Multi-block threads

In spite of the seemingly endless possibilities and combinations in thread cutting, the programming knowledge and experience gained in one category will be indispensable in other categories. A good threading program is based on a sound knowledge of common threading principles.

## TERMINOLOGY OF THREADING

Threading is a relatively large subject, in fact, it is large enough to have a whole book dedicated to it. As subjects of this kind usually are, threading has its own technical terms. These terms appear in many books, articles, technical papers, manuals and other sources. To understand them is a requirement for any CNC programmer and operator.

Listed here are some of the most common terms used for threads and thread cutting:

### ■ ANGLE OF THREAD

... is the included angle between two thread sides, measured in an axial plane (along an axis)

### ■ CREST (also see ROOT)

... is the top surface of a thread that joins the two sides

### ■ DEPTH OF THREAD

... generally, the distance between the crest and root of the thread, measured normal to the axis (in programming, depth is considered as a measurable size per thread side)

### ■ EXTERNAL THREAD

... is a thread that is cut on the outside of machined part, for example, as a screw or a bolt

### ■ INTERNAL THREAD

... is a thread that is cut on the inside of machined part, for example as a nut

### ■ HELIX ANGLE

... is the angle made by helix of the thread at the pitch diameter with a plane perpendicular to the axis

### ■ LEAD

... is the distance the threading tool will advance along an axis during one spindle revolution. The lead always determines the threading feedrate and can have constant or variable form

### ■ MAJOR DIAMETER

... is the largest diameter of the thread

### ■ MINOR DIAMETER

... is the smallest diameter of the thread

### ■ MULTISTART THREAD

... is a thread with more than one start, shifted by the pitch amount

### ■ PITCH

... is the distance from a specified point of one thread to the corresponding point of the adjacent thread, when measured parallel to machine axis

### ■ PITCH DIAMETER

... on a straight thread, the pitch diameter is an imaginary diameter where both external and internal diameters contact each other (*i.e.*, diameter where they meet)

### ■ ROOT (also see CREST)

... is the bottom surface of a thread, joining the sides of two adjacent threads

### ■ SCROLL THREAD

... is also known as a face thread - it is a thread machined along the X axis, rather than the more common thread machined along the Z axis

### ■ SHIFT

... in multistart threading, it is a distance by which the cutting tool is displaced to cut another start; this distance is always equal to the thread pitch. The number of shifts is always one less than the number of starts

### ■ TAPERED THREAD

... is a thread on which the pitch diameter is increased or decreased by a constant ratio (such as a pipe thread)

### ■ TPI

... in imperial units of measuring, the number of threads counted over the length of one inch (1 / pitch) - metric thread is defined by its pitch (TPI equivalent is not applicable)

## THREADING PROCESS

Threading is one of the most automated programming tasks in modern machine shop, yet it could be one of the more difficult operations done on CNC lathes. At first look, it may seem an easy procedure to make a program for tool-path that has all cutting parameters clearly defined, such as threading. Practical applications, however, could present a big departure from theory. This comment may be arguable, at least until it is time to start searching for solutions to unusual threading problems or even regular threads that *'just don't seem to be coming out right'*. An experienced programmer should have the ability to think of yet another solution, when all other solutions seem to have been used up. Such approach is the key to any problem solving and applies equally to threading problems.

What often causes threading to be a difficult operation is the cutting tool application. *Single point threading tool is unlike any cutting tool*. Although threading tool holder is mounted in the turret just like other tools, thread cutting insert is unique. Threading tool does not only *cut*, it also *forms* the thread shape. Typically, any threading insert will have the shape of finished thread. Mounting a threading tool in the turret can be done at 90° *to* - or *parallel with* - the machine spindle centerline, *regardless* of the thread being cut. Decision which way to mount the tool is determined by the thread angle, relative to the spindle center line. It is important that the tool is mounted *square* with the turret face. Even a small angular deviation will have an adverse effect on the finished thread.

### ◆ Steps in Threading

When comparing a common V-type 60° threading insert with equally common 80° insert commonly used for rough turning/boring, some interesting results will emerge:

Description	Threading tool: 60° included angle	Turning tool: 80° included angle
Tool nose radius	Almost sharp tip	0.8 mm (0.0313")
Included tool angle	60° + WEAK support	80° + STRONG support
Typical cutting feedrate	Up to 6.5 mm/rev + (0.25 in/rev) +	0.4 - 0.8 mm/rev (0.015 - 0.03 in/rev)
Typical cutting depth	Very SMALL	MEDIUM to LARGE

Such comparisons are not fair and the table is definitely not scientific, but it proves one important conclusion for threading - *the weakest tool does the heaviest machining*. Yes, welcome to the world of single point threading.

Indirectly, the table shows that even a very fine pitch thread *cannot* be machined using a single threading pass only. A single pass would produce a thread of very poor quality at best and - most likely - a totally unusable thread. Tool life would also suffer greatly. Much better approach is to program the thread as a series of small cutting passes, where each pass increases the thread depth.

### ◆ Spindle Synchronization

Unlike any other CNC lathe machining operation, single point threading requires that the thread cutting tool starts machining at *exactly* the same circumferential point for each depth of cut. This requirement is called *spindle synchronization*. In programming, this is not an issue at all, neither it is an issue during setup and normal machining. When this knowledge *is* very important can be summed up in two points:

- Threading process - general understanding
- Thread recutting - rework

As for the first item, understanding any subject in depth is always rewarding, thread cutting included. All new knowledge leads towards change of thinking, hopefully for the better. This is true in both programming and machining - in threading, solid knowledge can be used to solve various problems that may occur.

It is the second item that every CNC programmer - and *particularly the CNC operator* - should fully understand - recutting a thread. The subject of *thread recutting* is covered at the end of this chapter. At this point, just keep in mind that once a threaded part is taken out of the holding device, it will be extremely difficult - if need be - to put it back for recutting into the same exact position as before.

With practice and a really 'good eye', it can be done. The best way to avoid recutting is prevention - measure the thread while it is still mounted, there are several methods.

### ◆ Individual Threading Motions

Previous paragraphs have established that for multi-pass single point thread cutting, machine spindle rotation must be synchronized for each threading pass, so each thread start point is located at the same position on the threaded cylinder. High quality thread will be completed when the last cutting pass produces proper thread size, shape, surface finish and tolerances. Since single point threading is a result of several passes cutting a single thread, programmers must understand each of these passes well - they constitute *individual threading motions*.

In programming, basic structure of each pass remains the same, only the actual thread data change from one pass to another. In a most elementary setup, there are at least four motions for each threading pass - the table example shows one-pass threading as applied to a standard straight thread:



Threading Motion	Motion description
1	From the starting position, move tool to the thread diameter in rapid motion mode
2	Cut the thread - one axis thread cut (at the feedrate equal to the lead)
3	Rapid retract from the thread
4	Rapid return to the starting position

Expanding on these brief descriptions, the four step tool motion process will typically include at least four following considerations that are critical to the CNC program.

#### Start Point for Threading

Before the actual first motion, the threading tool must move from its current position (for example, from indexing position) to a position close to the machined part. This is a rapid motion, in the air (G00 motion). Make sure to calculate the XZ coordinates for this position correctly. The coordinates are called the thread *start position*, because they define where the thread cut will start from and eventually return to. Such a start position must be defined *away* from the part, but still close to the thread, as an intersection of the X axis clearance and the Z axis clearance.

For the X-axis, selection of a clearance greater than the thread lead is usually recommended. This is a *physical* clearance, per side - it has to be doubled to define the actual coordinate. For the Z-axis (horizontal thread is assumed), there has to be a space before the tool contact part material. This clearance necessary is for acceleration and the general rule is for about three times the thread lead. More details follow in the next section.

#### Threading Motion 1 - Rapid mode

The first tool motion is directly related to the thread itself. It is a motion *from* the start position *to* the first thread cutting diameter. Since the thread cannot be cut at full depth in a single pass, the total depth must be divided into a series of more manageable depths. Each depth will depend on the type of tool, material used and overall setup rigidity. This approach motion is a rapid mode.

#### Threading Motion 2 - Threading mode

When the threading tool reaches selected cutting diameter for a given depth, the *second motion* becomes effective. This is the actual threading pass - it will be cut during this step, at a specified feedrate and only when machine spindle is *synchronized* with the threading feedrate. There is no need to take any special steps to maintain the synchronization - in any threading mode, it is automatic. Thread will be cut to the programmed thread end position, usually along the Z-axis.

#### Threading Motion 3 - Rapid mode

The *third threading motion* follows the actual thread cut - *after* the programmed diameter is fully completed. Then, the threading tool must retract from the thread - at machine rapid rate and to a specified X-axis clearance position. Normally, this should be the same position as X-axis *start position*. It is the most logical position and always provides expected and verified results. X-axis clearance is always a diameter programmed *outside* of the thread area.

#### Threading Motion 4 - Rapid mode

With the final *fourth motion*, one-pass threading process is completed. There is a very important difference from the X-axis clearance in this motion - as expected, in the fourth motion, the threading tool *must* return along the Z-axis - but where? *Always* to the Z-axis start position. *There is no choice*. Returning to any other point will result in a scrap, and, possibly, a broken tool. The reason? While the initial Z-position in *Motion 1* was selected with only acceleration in mind, it automatically became the control position for spindle synchronization. Therefore, the return to the same position is absolutely mandatory. Once correctly established, the same Z-axis start/end position *guarantees* the only way to maintain spindle synchronization that is necessary for threading. In fact, careful manipulation of this position is very useful in multi-start threading (described later in this chapter). *Motion 4* is always in rapid mode.

#### Remaining Passes

All remaining passes are calculated using the same approach, using any new thread diameter for exact thread depth control. Note that only *Threading Motion 2* will be programmed in threading mode, using a proper G code. Threading motions 1, 3 and 4 will be in rapid mode.

This typical description (illustrated in *Figure 38-2*), is only general in nature and usually not sufficient by itself for high quality thread cutting.

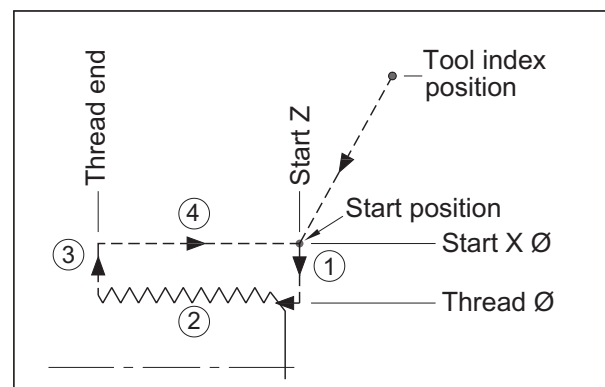


Figure 38-2  
Basic steps in single point thread cutting

### ◆ Thread Start Position

In reality, the tool start position is a specially selected *clearance position*. Unlike for turning or boring, this position has a special significance in threading. For a straight cylindrical thread, the minimum suitable clearance along the X-axis is about 2.5 mm (0.1 inches) per side, more for coarse threads, usually not smaller than the thread lead. For tapered threads, clearance calculation is the same, but applied over the *larger* diameter (external threads).

As for the clearance along Z-axis, some special considerations are necessary. When the threading tool comes into contact with material, it must be advancing at exactly 100% of the programmed feedrate. Since the cutting feedrate for threads is *always* equivalent to the thread lead, which may be quite large, certain amount of time will be required to reach the 100% of programmed feedrate. Just like a car needs some time to accelerate before reaching its cruising speed, threading tool has to reach full feedrate *before* it contacts any material. Real effect of acceleration must always be considered when selecting this clearance amount.

When programming coarse threads, the front clearance amount required will generally be much greater than a similar amount for fine or medium threads. For example, a common thread with 3mm pitch (8 TPI) will require feedrate of 3 mm/rev (0.125 in/rev). If the Z-axis clearance is too small, machine acceleration process will be incomplete upon the tool contact with material. Final result will be an imperfect and unusable thread. To avoid this very serious problem, a simple rule may help:

Z-axis start point clearance should be  
three to four times the length of thread lead

This is only a rule of thumb, a general rule; it works well in every day practice. Control manuals may offer a scientific way of calculating the minimum clearance, but this extra effort is seldom necessary.

#### *Obstacles for Clearance*

In some cases, the Z-axis clearance must be reduced because of space shortage, such as when thread cutting starts very close to a tailstock or machine limits. As acceleration time depends directly on the spindle speed (r/min), one common solution to avoid imperfect threads in this case is to *reduce* the spindle speed (r/min) - *feedrate must not be reduced* - it represents the thread lead. For more complex methods of infeed, Z-axis start position will be changed for each cut by a calculated amount.

Another method to increase Z-axis clearance is to modify the actual threading holder and/or insert, by grinding-off its less important parts. It may be a crude way, but it works in emergency.

### ◆ Thread Cutting Diameter and Depth

For cylindrical and conical thread cutting using the block method of programming (no cycles), each thread cutting diameter must be selected. From thread start position, the cutting tool will move *towards* spindle centerline for external threads and *away from* spindle centerline for internal threads. Actual cutting diameter for each pass must be selected not only with respect to the thread diameter, but also with respect to machining conditions.

In threading passes, the insert chip load becomes heavier as cutting depth increases. A damage to the thread, to the insert, or both, can be averted by maintaining a *consistent chip load* on the insert. One way to achieve consistency is to *decrease* each subsequent depth of the thread, another way is to apply a suitable *infeed method*. Commonly, both of these popular threading techniques are used simultaneously, for even better results.

To calculate depth of each threading pass, complex formulas are not required, just some common sense and a bit of experience. All threading cycles have an algorithm (special process) built in the control system that calculates each depth automatically. For manual calculations, the procedure follows a logical approach. *Total thread depth* (measured per side) must always be known - part programmer decides how many *threading passes* will be suitable for a particular thread.

Another threading amount to be decided is the *last* cut depth, a cut that actually finishes the thread to size. These amounts are usually very small and come from experience. In addition to the last cut, some threads may require a repetition of the last cut at the same depth - this cut is called the *spring pass*. Its main purpose is to remove any material left due to tool deflection caused by pressures. The rest is limited to mathematical calculations or available charts - and the control system.

When all three parameters (programmed amounts) have been firmly established, the total cutting depth must be distributed (divided) among individual threading passes, including the last pass depth. That will require a decision - how many threading passes are necessary?

### ◆ Number of Threading Passes

If each threading pass is programmed individually, rather than using threading cycles, programmer must decide how many passes are required for a given thread. This is a very important decision - too few or too many passes may cause severe problems at the machine. There is no easy way to change such a program - in almost all cases, an erroneous program will have to be re-written from scratch (at least its threading portion). The number of passes required is often listed in tooling catalogues and provides a good start. Otherwise, consider the tool material, material being cut, lubrication used, method of setup, length of part, rigidity, etc.

### ◆ Thread Depth Calculation

To fully understand how depth calculation works in CNC threading, two types of depth should be identified:

- Total thread depth
- Depth of each threading pass

**Total thread depth** is always measured 'per side' (radius value) and can be calculated from the *major* and *minor* diameters, if they are known:

$$\text{Thread Depth} = \frac{\text{Major } \varnothing - \text{Minor } \varnothing}{2}$$

More than likely, only the *major* diameter will be known, as per drawing definition. In this case, a common formula can be used to calculate thread depth - just keep in mind that there are actually *two similar* formulas - one for *external* and one for *internal* threading.

For the most common 60° V-thread in ISO M (metric) and UN (Unified National) forms - and - *external* threads only, the formula is:

$$D_{\text{EXT}} = \frac{0.61343}{\text{TPI}} = 0.61343 \times P$$

For the same group of threads applied *internally*, the formula is similar:

$$D_{\text{INT}} = \frac{0.54127}{\text{TPI}} = 0.54127 \times P$$

where ...

- $D_{\text{EXT}}$  = Single depth of external thread
- $D_{\text{INT}}$  = Single depth of internal thread
- TPI = Number of threads per inch
- P = Thread pitch in mm **or** 1/TPI for imperial

An example of actual usage is included in this chapter.

**Depth of each threading pass** should always be calculated for best results. In emergency, an approximation of individual cuts - if done properly - will also work well. The main goal here is to select the first cut depth, then *decrease* each depth after (for subsequent passes). Consider the reasoning here - the threading tool is triangular. When the first depth is cut, a certain area of the triangle will be cutting. As the insert is engaged deeper, more of its area will be included, potentially creating a situation when *too much* material is removed. Programming each 'next' cut with progressively decreasing depth is the main key to successful threading. These cuts take place between *major* and *minor* diameters, never deeper.

### ◆ Thread Depth Constants

The two common constants for calculating thread depth are **0.61343** for external threads, and **0.54147** for internal threads. Both are applied for any 60° V-thread ISO metric threads or UN imperial threads. Although it is not important to know the source of these constants, understanding their origin helps to better understand this most common thread form. It should be expected that these constants have been calculated from actual thread data, based on established international standards - *Figures 38-3 and 38-4*.

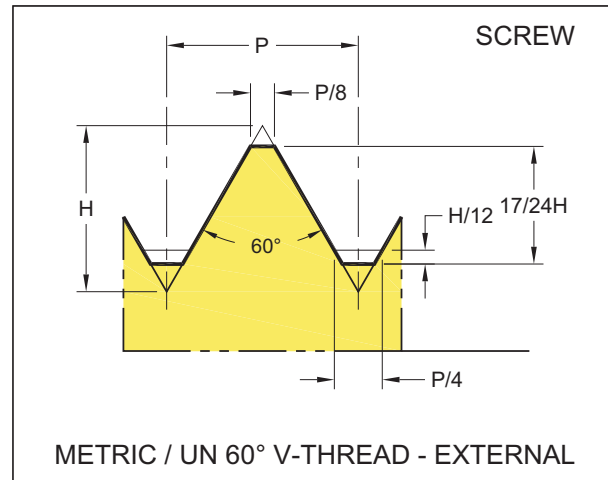


Figure 38-3

Standard ISO (M) and imperial (UN) 60° V-thread form - EXTERNAL

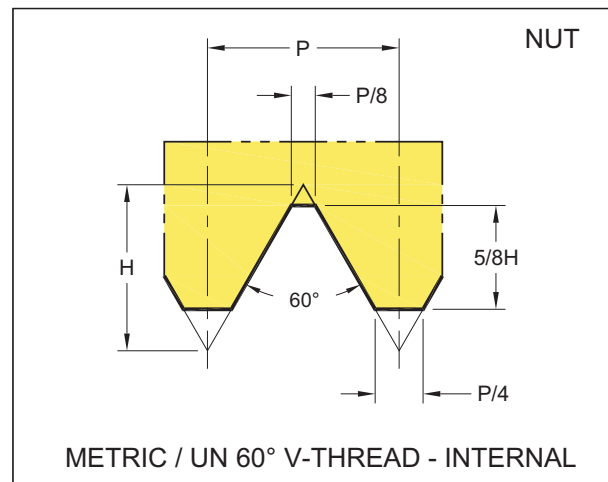


Figure 38-4

Standard ISO (M) and imperial (UN) 60° V-thread form - INTERNAL

As with many constants in mathematics, these two are also based on the main unit being 1 (one). In this case, since the thread depth will change with any pitch change, the pitch is assumed to have a value of 1. That means  $P=1$  mm or  $P=1$  inch. There is a reason why the unit of one has been used - it allows multiplication by *any* actual unit value to

produce the required result. The formula using such constant is usable for any pitch size. One other dimension is also important - the dimension H, overall thread height, measured between sharp points.

If pitch  $P = 1$ ,  
then ...  $H = 0.5 / \tan 30^\circ = 0.866025$

From here the actual calculations of thread depth can be made, based on the fractional dimensions of the standard:

$$\begin{aligned} 17/24H &= 17/24 \times 0.866025 \\ &= 0.708333 \times 0.866025 \\ &= \underline{0.613435} \quad \dots \text{EXTERNAL} \end{aligned}$$

$$\begin{aligned} 5/8H &= 5/8 \times 0.866025 \\ &= 0.625 \times 0.866025 \\ &= \underline{0.541266} \quad \dots \text{INTERNAL} \end{aligned}$$

#### American National Thread

Another standard - an older one - may still be used in various publications and drawing specifications, so it is important to understand this thread form and be able to calculate its thread depth. *American National* standard does not use the constant of 0.613 for external thread depth, but constant of 0.649 - see Figure 38-5.

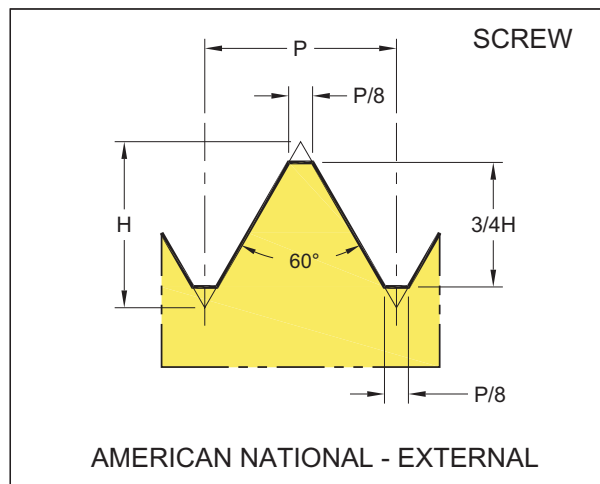


Figure 38-5

American National thread form

The main difference between the two forms is the specified height of  $3/4H$  compared to  $17/24H$ . The overall depth H is still the same - 0.866025.

$$\begin{aligned} 3/4H &= 3/4 \times 0.866025 \\ &= 0.75 \times 0.866025 \\ &= \underline{0.649519} \quad \dots \text{EXTERNAL} \end{aligned}$$

Constant for internal threads remains the same, at 0.541.

#### ◆ Calculation of Threading Passes

When individual threading passes are only approximated (meaning they are '*estimated*'), they may - or they may not - come out right. When the same passes are actually *calculated*, it does not automatically mean the thread will come out right either, but the chances are much better. There are many technological issues to deal with that cannot be included in any mathematical formula.

To illustrate calculations of individual passes, a typical example will be used - a simple external thread illustrated in Figure 38-6.

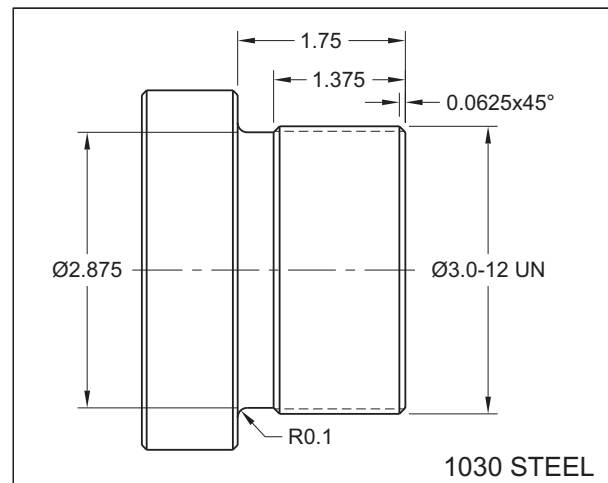


Figure 38-6

Sample part used for thread depth calculation  
- only important dimensions are shown

Calculating each threading pass is only necessary for the so called 'long hand' threading (where each motion has its own block), they are *not* needed for threading cycle G76.

As always in problem solving, the first step is to isolate *known data* relevant to the solution. In this case, they are:

- Major (given) diameter ... 3.0 inches
- Number of threads per inch ... 12 TPI
- Length of thread ... 1.5 (as reasonable)
- Material ... Mild steel

From the data that is known, some additional data can be calculated - *or* - selected:

- Thread depth ...  $0.61343 / 12 = 0.0511$
- Minor (final) diameter ...  $3 - 2 \times 0.0511 = 2.8978$
- First cutting depth ... 0.021 (selected)
- Cutting feedrate ... Thread lead = 0.0833

All these settings provide enough information to continue with program development.

Well, there is one more programming decision to make - yes, it is optional, but also strongly recommended - is to select a *last pass depth* - known as *finishing allowance* - for example, 0.004 for the following calculations. Now, all data are in, so the number of passes can be calculated:

$$n = \left( \frac{P - R}{Q} \right)^2$$

where ...

- n = Number of threading passes
- P = Depth of thread per side
- R = Finishing allowance amount (last pass depth)
- Q = Depth of the first threading pass

It appears that the selected letters do not suggest any specific meaning. That is true, but at this point, consider them to be just letters identifying known or unknown thread related values. Yet, they were selected carefully, with threading cycles in mind, covered later in this chapter. For the example, the calculated number of passes will be:

$$n = \left( \frac{0.0511 - 0.004}{0.021} \right)^2 = 5.03 = 5 \text{ passes}$$

Accepting that *five threaded passes* are suitable for this job (excluding the finishing allowance), they will be distributed between the specified diameter of 3 inches and the calculated final diameter of 2.8978 inches, with the first depth pass of 0.021. The depth of cut used for *calculation* has to be reduced by the finishing allowance, which will become the sixth pass. So, the usable depth is not 0.0511 but  $0.0511 - 0.004 = 0.0471$ . The 0.004 will be an *additional* pass, which is optional but has been selected here.

Pass 1 = $0.021 \times \sqrt{1} = 0.0210$	..cutting depth = 0.0210
Pass 2 = $0.021 \times \sqrt{2} = 0.0297$	..cutting depth = 0.0297
Pass 3 = $0.021 \times \sqrt{3} = 0.0364$	..cutting depth = 0.0364
Pass 4 = $0.021 \times \sqrt{4} = 0.0420$	..cutting depth = 0.0420
Pass 5 = $0.021 \times \sqrt{5} = 0.0470$	..cutting depth = 0.0470
Pass 6 = Pass 5 + 0.004	..cutting depth = 0.0510

Cutting depth shows *accumulated* depth. In order to calculate individual threading diameters, subtract each depth twice (double depth) from the major diameter of 3 inches:

Cutting diameter 1 ..	$3.0 - 2 \times 0.0210$	=	2.9580
Cutting diameter 2 ..	$3.0 - 2 \times 0.0297$	=	2.9406
Cutting diameter 3 ..	$3.0 - 2 \times 0.0364$	=	2.9272
Cutting diameter 4 ..	$3.0 - 2 \times 0.0420$	=	2.9160
Cutting diameter 5 ..	$3.0 - 2 \times 0.0470$	=	2.9060
Cutting diameter 6 ..	$2.906 - 2 \times 0.004$	=	<u>2.8980</u>

Note that this particular example shows almost perfect calculations (within 0.0002) - that is not always possible,

and some adjustments usually be necessary. Arbitrary selections or even manual adjustments of one or more depths may have to be made for best results. *Figure 38-7* shows - in scale - the distribution of threading cuts for the example.

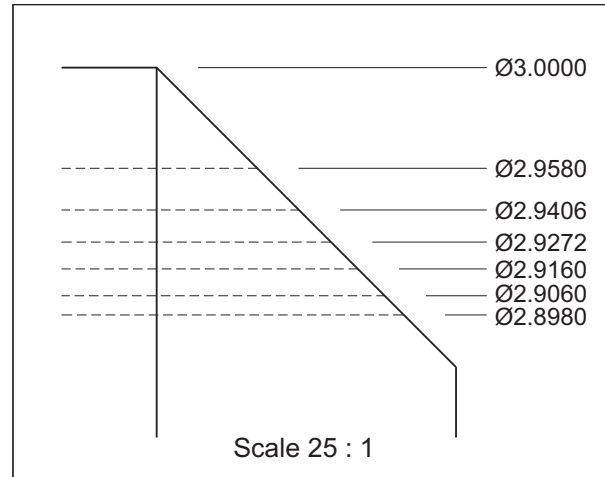


Figure 38-7

Distribution of individual threading passes based on a formula

All this theory has to make some practical sense - it is time to apply the theoretical principles into actual program.

### ◆ Thread Cutting Motion

When a threading tool reaches selected threading pass depth, it is ready to cut the thread itself. All thread cutting motions originate from the start position (start point). In fact, this start point should also be the *end point* for each threading pass, so it can become the consistent start point for the next pass. During thread cutting, a feedrate must be active. Although the threading cut is, in effect, a linear motion, do not *ever* use preparatory command G01 for threading. If G01 is used, the start point for each pass will *not* be synchronized with the previous thread start. Instead of G01 command, use a G code specifically designated for threading. Preparatory command G32 is the most common code used by Fanuc controls for threading without cycles. During a thread cutting motion with G32, control system automatically disables the feedrate override. CNC operator has to be extra careful to set the threading tool exactly, particularly when thread ends close to shoulders of the part or close to the chuck. To illustrate the programming process up to this point, here is a typical program section:

```
...
N61 G00 X3.3 Z0.25          (START POINT XZ)
N62 X2.958                  (THREAD DIA NO. 1 START)
N63 G32 Z-1.6 F0.0833       (THREAD TO END)
...
```

When block N63 is completed, threading tool is positioned at the end of the thread at X2.958 Z-1.6.



### ◆ Retract from Thread

Once the tool has reached its end position along Z axis, the tool must leave the material immediately, to avoid making any physical damage. This program block represents the *third* motion within the basic threading process. Retract motion can have two forms - straight out in one axis (normally along the X-axis), or a gradual pull-out along two axes (simultaneously along XZ axes) - *Figure 38-8*.

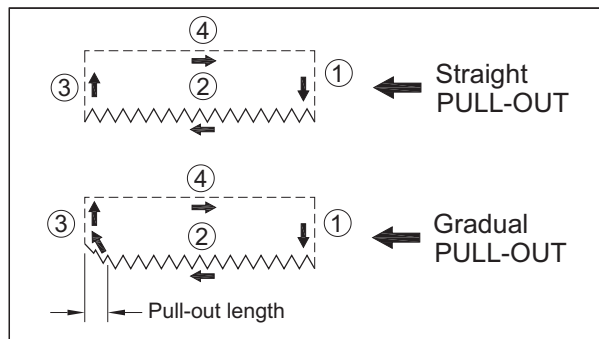


Figure 38-8

Exit options from a completed thread

Generally, the *straight pull-out* should be programmed whenever thread ends cutting in an *open space*, for example, in a relieve groove or a recess. For threads that do not end in an open area (some shaft applications, for example), the *gradual pullout* is a better choice. Gradual pull-out motion produces better quality threads and prolongs threading insert working life. In order to program a straight pull-out, the threading mode G32 must be canceled and replaced by a rapid motion mode, using the G00 command:

```
N64 G00 X3.3 (RAPID OUT)
```

For any gradual pullout, the threading G code and the feedrate *must remain in effect*. When the normal length of thread is completed - but *before* the tool is retracted - the threading tool moves in two axes simultaneously, ending *outside* of the thread. The normal length of the pullout is usually 1 to 1-1/2 times the lead (not the pitch), the suggested angle is 45°. It is also important to pay attention to the clearance diameter.

```
N64 U0.2 W-0.1 (GRADUAL PULLOUT-THREADING ON)
N65 G00 X3.3 (RAPID OUT)
```

For *external* threads, the rapid-out clearance diameter must always be *further* away from spindle center line than the last diameter of gradual pullout. For *internal* threads, the clearance diameter must be *closer* to spindle center line than the diameter of gradual pull-out.

In G32 mode, the gradual pull-out can be within the program length or can extend beyond it. The programmer has a choice here, it is based on the job requirements.

### ◆ Return to Start Position

Regardless of how the threading tool retraction is programmed - straight or gradual - the last step is always a return to the starting position. This tool motion is entirely in the open space, therefore programmed in the rapid motion mode G00. Normally, this return motion to the starting point is along one axis only, usually the Z axis. This is because in most programs, the tool retraction from the thread has already reached the X-axis diameter and only the Z-motion is left.

Here is a program segment showing gradual pull-out:

```
...
N61 G00 X3.3 Z0.25 (START POINT XZ)
N62 X2.958 (THREAD DIA NO. 1 START)
N63 G32 Z-1.6 F0.0833 (THREAD TO END)
N64 U0.2 W-0.1 (GRADUAL PULL-OUT)
N65 G00 X3.3 (RAPID OUT)
N66 Z0.25 (RETURN TO Z-START)
...
```

Returning to the start point makes sense - this point provides consistency in thread programming.

## THREADING FEED AND SPINDLE SPEED

Regardless of any active *cutting* motion, there is always a feedrate involved - this feedrate controls how effectively the tool removes material in various operations - drilling, contouring, facing, pocketing and many other machining operations. As threading operations are the main subject of this chapter, it is important to understand how feedrate is related to single point threading.

In threading, the choice of cutting insert selection, spindle speed and threading feedrate are always restricted - more so than for any other machining operation. Both, the threading tool and feedrate are determined by the engineering requirements - as drawing specifications. In terms of material removal, threading insert is one of the *weakest* tools used on CNC lathes - yet its applications demand some of the *heaviest* feedrates used in CNC lathe programming for any tool.

Other factors that can influence the final thread have to be dealt with as well, such as spindle speed (both minimum and maximum), the depth of each threading pass, the tool edge preparation, thread infeed method selected, setup of cutting tool and insert, plus many similar considerations. Often, a simple change of only one factor will correct a threading problem. At other times, many factors have to be considered - and possibly changed - to produce the best thread possible.

*Figure 38-9* compares feedrates for turning and threading - they are both in feedrate per revolution.

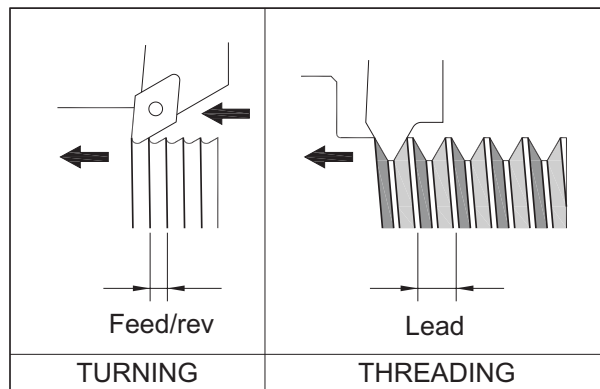


Figure 38-9

Comparison of feedrates between turning and threading

Next, we look at some practical consideration when selecting feedrates for threading.

### ◆ Threading Feedrate Selection

Selection of feedrate for general turning, boring, grooving, etc., is based on such factors as material type, tool nose radius, required surface finish, etc. In this sense, the 'correct' feedrate for such operations cover a large range. In threading, this flexibility is limited. Threading feedrate is *always* determined by the *lead* of the thread - *never* the pitch. In imperial units drawings, thread description is provided as the number of threads over one inch length, or TPI (TPI = threads per inch), and a nominal diameter. As an example, a thread that is described in the drawing as 3.75-8, means that the thread has 8 threads per inch, and the nominal diameter (for example, the major diameter) is Ø3.750. All single start *metric* threads have the pitch standardized, depending on the thread diameter. For instance, a thread described as M24×3 is a single start metric thread with the pitch of 3 mm on a 24 mm nominal diameter. A description M7×0.75 is a single start thread with three quarters of a millimeter pitch and nominal diameter of 7 mm.

Regardless of dimensional unit used, the most important terms for selecting the correct feedrate are the *lead of thread* and the *number of threading starts*.

It may help to review basic relationship between *thread lead* and *thread pitch* (see details in terminology of threading on page 350). In a common machine shop conversation (shop talk), the words *lead* and *pitch* are often used incorrectly. The reason is that for a single start thread, the amount of lead is *identical* to the amount of pitch. Since most machine shops work with a single start thread on a daily basis, misuse of the terms is seldom noticed. In addition, virtually all taps have a single start. What may be acceptable in a shop talk language has to be interpreted correctly in CNC programming. Each term has a very specific meaning in threading, so use them in the correct way:

$$\text{LEAD} = \frac{\text{No. of starts}}{\text{TPI}} = \text{Feedrate}$$

The programmed threading feedrate is always the lead of thread, never the pitch!

$$\text{PITCH} = \frac{1}{\text{TPI}}$$

From these two formulas it is easy to deduct that if the number of starts is *one*, both the lead and pitch will always have the same value.

The following formula should be applied for threading feedrate calculation:

$$F = L = P \times n$$

where ...

- F = Required feedrate (mm/rev **or** in/rev)
- L = Lead of the thread (mm **or** inch)
- P = Pitch of the thread (mm **or** inch)
- n = Number of starts (positive integer)

For example, a thread with a single start and the pitch of three millimeters (3 mm) will require feedrate of

$$3 \times 1 = \mathbf{F3.0}$$

For threading programs that use imperial units, the above formula is equally valid, since

$$P = \frac{1}{\text{TPI}}$$

where ...

- P = Thread pitch
- TPI = Number of threads per inch

As an example, the thread with one start and 8 TPI will require feedrate of

$$1/8 \times 1 = 0.125 \times 1 = \mathbf{F0.125}$$

Multistart threads are special in many ways, but the feedrate selection is also the *lead* - not the pitch of the thread. For example, a double start thread with 12 TPI will have the following feedrate:

$$1/12 \times 2 = 0.083333 \times 2 = \mathbf{F0.166667}$$

For most Fanuc controls, imperial threading feedrate can be programmed with six digits after the decimal place.

### ◆ Spindle Speed Selection

Spindle speed for thread cutting is always programmed in direct *r/min*, never as constant surface speed (CSS) (cutting speed). That means G97 preparatory command must be used with address S, specifying the number of revolutions per minute. For example, G97 S500 M03, will result in 500 *r/min* spindle speed. As it is true that single point threading takes place over several diameters between major and minor, G96 selection would seem logical. This is not the case. First, even for fairly deep coarse threads, the difference between the first and last diameter is insignificant. Second - *and this reason is even more important* - thread cutting requires a perfect spindle and feedrate *synchronization* at the start of each pass. Such synchronization can be more accurately achieved only with constant *r/min* rather than constant surface speed (CSS).

For majority of threads, the selection of *r/min* requires only consideration of general machining conditions, similar to turning operations. At the same time, selecting spindle speed should be considered along with the feedrate. Because of some heavy feedrates used for threading, there is a distinct possibility that certain threads cannot be cut at *any available spindle speed*. If this is confusing, keep in mind that feedrate is determined not only by the lead, but also by the *machine overall capability*. Every CNC lathe has a programmable feedrate value, specified in either *mm/min* or *in/min*, up to a certain maximum for each axis.

Take an example of a CNC lathe that has maximum programmable feedrate for X axis 7000 *mm/min* (275 *in/min*) and for Z axis 12000 *mm/min* (472 *in/min*). Consider the fact that there is a direct relationship between spindle speed and feedrate per revolution. Result of this relationship is actually *feedrate expressed in terms of time*, not per revolution. Feedrate per time is always result of the spindle speed in direct *r/min* multiplied by the feedrate per revolution in *mm/rev* or *in/rev*.

#### ➡ Metric units example :

$$700 \text{ r/min} \times 3 \text{ mm/rev} = 2100 \text{ mm/min}$$

#### ➡ Imperial units example :

$$700 \text{ r/min} \times 0.125 \text{ in/rev} = 87.5 \text{ in/min}$$

In CNC lathe programming generally, not only in threading, always make sure that feedrate per revolution combined with spindle speed will be *less than or equal to* the maximum *available* feedrate per time for the axis with the lower rating, which is usually the X axis.

Based on this simple rule, maximum spindle speed (in revolutions per minute) for a given lead can be selected according to the following formula:

$$R_{\max} = \frac{Ft_{\max}}{L}$$

where ...

$$\begin{aligned} R_{\max} &= \text{Maximum allowable spindle speed (r/min)} \\ Ft_{\max} &= \text{Maximum feedrate per time (X axis)} \\ L &= \text{Thread lead} \end{aligned}$$

#### ➡ Metric units example :

If thread lead L is 2.5 *mm* and the maximum feedrate for X axis  $Ft_{\max}$  is 7000 *mm/min*, then the maximum programmable threading spindle speed  $R_{\max}$  will be:

$$R_{\max} = 7000 / 2.5 = 2800 \text{ rpm}$$

#### ➡ Imperial units example :

If thread lead L is 0.125 *inches* and the maximum feedrate for X axis  $Ft_{\max}$  is 275 *in/min*, then the maximum programmable threading spindle speed  $R_{\max}$  will be:

$$R_{\max} = 275 / 0.125 = 2200 \text{ r/min}$$

The maximum allowable spindle speed (*r/min*) only reflects capabilities of the CNC machine. Feedrate actually used in a program must also take into account various machining and setup conditions, just like any other cutting operation. In practice, the majority of actual programmed spindle speed (*r/min*) will be well below the maximum capacity of CNC machine tool.

### ◆ Maximum Threading Feedrate

Selection of cutting feedrate in general was discussed earlier, starting on *page 91*. After studying the section on maximum *r/min* selection (spindle speed), it should not be surprising that similar limitations apply to the determination of *maximum threading feedrate* for a given spindle speed (programmed as *r/min*). Again, the limits of CNC machine tool are very important, so be aware of them when writing thread cutting programs.

Maximum programmable threading feedrate for a given spindle speed (in *r/min*) can be calculated from the following formula:

$$Fr_{\max} = \frac{Ft_{\max}}{S}$$

where ...

$$\begin{aligned} Fr_{\max} &= \text{Maximum feedrate per revolution} \\ &\quad \text{for a given spindle speed S} \\ Ft_{\max} &= \text{Maximum feedrate per time (X axis)} \\ S &= \text{Programmed spindle speed (r/min)} \end{aligned}$$

### ⊖ Metric units example :

In a metric example, the maximum machine feedrate for X-axis is 7000 mm/min and the programmed spindle speed S is selected as 1600 rpm. In this case, the maximum programmable threading feedrate will be (in mm/rev):

$$7000 / 1600 = 4.375 \text{ mm/rev}$$

Result shows the maximum lead that can be safely machined at 1600 r/min must be less than 4.375 mm.

### ⊖ Imperial units example:

If the maximum machine feedrate along X-axis is 275 in/min and spindle speed S is selected as 2000 r/min, then the maximum programmable feedrate will be (in in/rev):

$$275 / 2000 = 0.1375 \text{ in/rev}$$

Therefore, the maximum thread lead that can be cut at 2000 r/min is 0.1375 inches, which allows about 8 threads per inch or finer.

Changing the spindle speed (feedrate remains the same) will allow programming coarser threads on the same CNC lathe. For example, if only 1500 r/min is selected instead of the 2000, the maximum lead will increase to 0.1833 inches or about 6 threads per inch.

All calculated values only indicate the control and machine actual capabilities and do not guarantee a safe job setup or even suitable machining speed

### ◆ Lead Error

Normally, threading feedrate requires the address F, with up to three decimal place accuracy for metric threads (F3.3 format), and four decimal place accuracy for threads in imperial units (F2.4 format). The majority of threads are short and this accuracy is quite sufficient. There is never problem for metric threads, regardless of thread length, because the thread is defined by its lead already in the drawing. For threads programmed in imperial units, the thread lead must be calculated from drawing provided threads per inch (TPI). For many imperial threads, lead is accurately calculated within four decimals available for the F address. A 10 TPI requires programmed feedrate of F0.1, 16 TPI requires programmed feedrate of F0.0625, etc. These are threads that divide TPI into one, within the four decimal places accurately, such as 8, 10, 16, 20, 40 to name the most common number of threads. Not all threads fall into this rather convenient group. For many other threads, the calculated value must be properly rounded off.

Take a 14 TPI thread, for example. Ideal threading feedrate should be  $1/14=0.071428571$  inches per revolution. Rounded value used in the program should be F0.0714. Over a short thread length there is no noticeable error and the thread is well within all tolerances. That is not true if the

thread is unusually long or the rounded value has been improperly calculated. An *accumulative error*, known as the *thread lead error*, may result in possible scrap, due to incorrect thread. By using rounded value of 0.0714, the loss is 0.000028571 inches for *each* thread revolution. Lead error over one inch (or more) can be easily calculated:

$$L_e = (F_i - F_p) \times \text{TPI}$$

where ...

$L_e$  = Maximum lead error per inch  
 $F_i$  = Ideal feedrate (nine or more decimal places)  
 $F_p$  = Programmed rounded feedrate  
 TPI = Number of threads per inch

Over one inch, the error in the example will be 0.0004 of an inch, over fifty inches it will be full 0.0200 of an inch.

Thread length in inches	11.5 TPI		14 TPI	
	F0.0870	F0.086957	F0.0714	F0.071429
1.0	0.000500	0.000006	0.000400	0.000006
2.0	0.001000	0.000011	0.000800	0.000012
3.0	0.001500	0.000017	0.001200	0.000018
4.0	0.002000	0.000022	0.001600	0.000024
5.0	0.002500	0.000028	0.002000	0.000030
10.0	0.005000	0.000055	0.004000	0.000060
25.0	0.012500	0.000138	0.010000	0.000150
50.0	0.025000	0.000275	0.020000	0.000300

The table above shows accumulative error for 11.5 TPI and 14 TPI and compares two feedrates for each, as used in a program for threading.

### ◆ Rounding Error

Another example, somewhat more critical, is an incorrect rounding value. Ideally, a thread with 11.5 threads per inch should be programmed with the feedrate of 0.086956522. If this value is rounded to F0.0870, the accumulated error is 0.0005 per one inch and the error over 50 inches will be 0.0250 inches (see table). Even if the CNC machine does not allow six decimal places for threading feedrate, the correct rounding of the calculation is very important.

Compare the following rounded values and the errors they cause (11.5 TPI over 50 inches):

0.0869	. . .	error of 0.0325
0.0870	. . .	error of 0.0250
0.0871	. . .	error of 0.0825

*What a difference for only 1/1000th of an inch rounding.*



### ◆ E-address

As you see from the previous table, Fanuc engineers recognized this potential problem early and introduced the address E for threading feedrate on their earlier CNC controls (Fanuc 6, for example). The benefit of using E address for threading is that it allows programming with *six* decimal places instead of the standard *four* for imperial threads (increased accuracy allowed for metric threads using the E address is seldom used). With proper rounding, the accumulative error is virtually negligible.

Using the same illustration of 14 TPI over 50 inches, the error for the whole length will only be 0.0003 of an inch, if the F0.0714 is replaced by E0.071429. The second example, using a thread with 11.5 threads per inch, should be programmed with the feedrate of E0.086957. The accumulated error over fifty inches will be only 0.000275 inches, a negligible error. Note that the table shows *F* not *E* address:

The latest CNC systems allow use of six digit accuracy for the F-address as well

The lead error or rounding error are always potential problems when programming long thread leads. Depending on the kind of threading applications in machine shop, accumulative error of the thread lead may be critical or it may never be an issue to deal with.

## TOOL REFERENCE POINT

Good tool setup is critical to good machining environment. While a good setup is important to all tools, it is even more important to maintain good setup of threading tools, both external and internal. Tool cutting edge has to be correctly oriented, securely mounted in the insert pocket and it has to be the right type. Its reference point, used for setup, is also very critical.

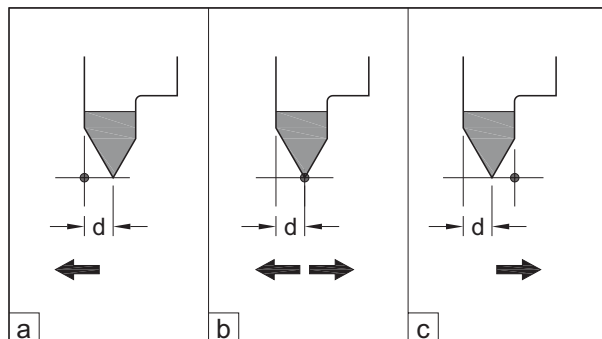


Figure 38-10  
Typical reference points for setup of threading tools

Threading tool reference point selection requires more considerations than for turning tools. Figure 38-10 shows three possibilities, in the order of programming frequency.

The third version 'c' is the rarest and offers virtually no benefit to the programmer except in some cases of left hand threading. For most left hand threads, one of the first two versions is also quite sufficient.

Threading insert setting as in Figure 38-10a is the most suitable for general use and for threads that end at a shoulder. Configuration in Figure 38-10b is suitable for threads that end in an open diameter (recess). The Figure 38-10c shows a possible setting for a left hand threading work.

Selection of tool reference point (G50 or geometry offset setting) as per illustration in Figure 38-10a is the most desirable one, when the intent is to *standardize* tooling setup for *any* type of thread. It is the most convenient setting, regardless of how a thread ends. It is also the safest method, as it is measured closest to the part. In some cases, an allowance must be made for the difference between the programmed edge and the actual edge. Tooling catalogs list this value precisely, or one half of the threading insert width (if applicable) can be used instead as general rule.

## BLOCK-BY-BLOCK THREADING

The oldest method of single point thread programming is to calculate each and every motion associated with the thread and write each motion as individual block of the program. This method is called *block-by-block* threading, long-hand threading, or just *block threading method*.

Each of the four basic motions occupies one block of program, resulting in the minimum of four blocks per each threading pass. If gradual pullout from the thread is used for thread cutting, there will be five blocks of program for each threading pass. When cutting coarse threads, threads in hard or exotic materials, even some multi start threads, this method often means quite a long program. The program length, difficulty in editing, high possibility of errors, and even small memory capacity of many older control systems, are negative sides of using this method.

On the positive side, CNC programmer has an *absolute programming control over the thread*. Such control placed into capable hands can often be applied to some special threading techniques, for example, cutting a thread shape with a threading tool much smaller than the thread itself or making large knuckle threads with a round grooving tool.

Thread programming method using the block technique for a constant lead thread is available on all CNC lathes that support threading.

G32	Thread cutting command (single point threading)
-----	-------------------------------------------------

Preparatory command for this type of threading is G32. Command G33 may exist on some controls, but G32 is the standard G-code for Fanuc and compatibles.



In an earlier example, a  $\varnothing 3.0$ -12 TPI external thread was used. All cuts were distributed in six passes, for the total depth of 0.0511 - here is a summary:

Pass #1 depth	=	0.0210	Total depth	0.0210
			Diameter	2.9580
Pass #2 depth	=	0.0087	Total depth	0.0297
			Diameter	2.9406
Pass #3 depth	=	0.0067	Total depth	0.0364
			Diameter	2.9272
Pass #4 depth	=	0.0056	Total depth	0.0420
			Diameter	2.9160
Pass #5 depth	=	0.0050	Total depth	0.0470
			Diameter	2.9060
Pass #6 depth	=	0.0040	Total depth	0.0510
			Diameter	2.8980

Always make sure all diameters are calculated carefully without errors. Small error can cause big scraps.

The threading operation in program O3801 will use the tool and offset number 5 (T0505), at 450 r/min spindle speed (G97 S450) at normal spindle rotation:

```
O3801 (G32 VERSION - BASIC)
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
```

Now, the thread start point has been reached. The next stage is to implement all four steps, one step per block, for the *first* pass:

```
N62 X2.958 (PASS 1)
N63 G32 Z-1.6 F0.0833 (or F/E0.083333)
N64 G00 X3.3
N65 Z0.25
```

The remaining six passes can be programmed next, just by changing the diameters. Note that the threading feedrate does *not* repeat - it is modal from block N63 on.

```
N66 X2.9406 (PASS 2)
N67 G32 Z-1.6
N68 G00 X3.3
N69 Z0.25
N70 X2.9272 (PASS 3)
N71 G32 Z-1.6
N72 G00 X3.3
N73 Z0.25
N74 X2.916 (PASS 4)
N75 G32 Z-1.6
N76 G00 X3.3
N77 Z0.25
N78 X2.906 (PASS 5)
```

```
N79 G32 Z-1.6
N80 G00 X3.3
N81 Z0.25
N82 X2.898 (PASS 6)
N83 G32 Z-1.6
N84 G00 X3.3
N85 Z0.25
```

Block N85 terminates the threading routines and the program can be closed as well, if there are no more tools used.

```
N86 X12.0 Z4.5 T0500 M09
N87 M30
%
```

What should strike as an odd occurrence in the example, is how many repetitions there are. Observe the three blocks following each new pass diameter - *they are always the same*. For a thread with many passes these repetitions will be numerous. This block-by-block method has one main benefit - is it under programmer's full control. Adjustments may be made to the number of threads and depth of each pass, but at the machine (at least not easily). Non-standard infeed method and a gradual pullout from the thread can also be added. Actual program editing after program has been completed is much more inconvenient.

## BASIC THREADING CYCLE - G92

Computerized control systems can perform many internal calculations and store their results in control memory for further use. This feature is especially useful for thread cutting, since multiple repetitions of block-by-block tool motions can be avoided and the program shortened quite significantly.

For better comparison and to illustrate a simple threading cycle, the same threading example that illustrated previous G32 command will be used again (12 TPI on a 3.000 inch external diameter), providing identical results. This cycle is usually called the G92 threading cycle on Fanuc or similar controls, also known as *box threading cycle*.

A word about *another* G92 command. Some programmers may be used to registering a current tool position with G92 command for milling applications. On CNC lathes, a command for the same purpose is G50, not G92. G92 used for threading has *nothing* to do with now virtually obsolete *position register* setting command. This applies for older control systems only - modern controls use advanced geometry offsets. Format for the G92 threading cycle is:

G92 X.. Z.. F..
-----------------

where ...

X = Current threading pass diameter  
 Z = Thread end position  
 F = Threading feedrate in in/rev

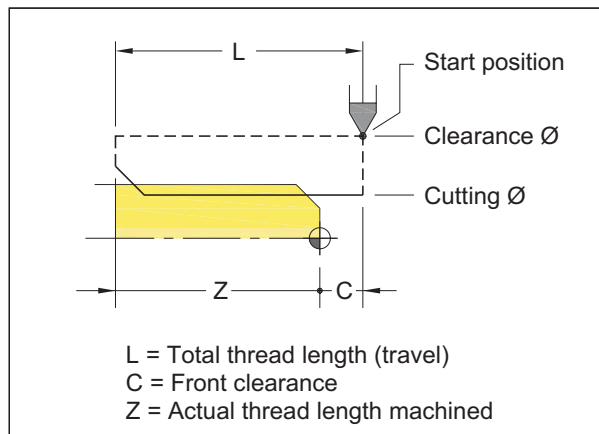


Figure 38-11

G92 - Simple thread cutting cycle, also called 'box threading cycle'

Schematic illustration of G92 straight thread cutting cycle is shown in Figure 38-11.

The program will do exactly the same job as with G32, except it will have a noticeably different structure.

Using G92 cycle, the following list shows all calculated diameters for each threading pass, as they will appear in the program (no change at this stage):

Pass #1	=	Ø 2.9580
Pass #2	=	Ø 2.9406
Pass #3	=	Ø 2.9272
Pass #4	=	Ø 2.9160
Pass #5	=	Ø 2.9060
Pass #6	=	Ø 2.8980

As before, the threading tool has been assigned a tool number and spindle speed - Tool 5 (T0505) and 450 r/min:

O3802 (G92 VERSION)

```
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
```

The first three blocks are identical to the block-by-block method of threading. Next, the threading tool will be positioned at the first pass diameter, cut the thread, retract and return to the starting position. The last three steps are always identical for each pass, and the main benefit of G92 threading cycle is that it eliminates such repetitive data and makes the program easier to edit.

G92 cycle will be called together with the first threading pass, in block N62. Note that X and Z axes input values and cutting feedrate are included in this block only:

```
N62 G92 X2.958 Z-1.6 F0.0833 (PASS 1)
```

Start position is defined by the last X and Z coordinate *before the cycle call*. In the example, start position is X3.3 Z0.25 (block N61). The remaining blocks include only data that have changed - which means only threading diameters. Five threading passes can be programmed just by changing the X-diameters. There is no need to repeat the Z location *or* feedrate.

```
N63 X2.9406 (PASS 2)
N64 X2.9272 (PASS 3)
N65 X2.9160 (PASS 4)
N66 X2.9060 (PASS 5)
N67 X2.8980 (PASS 6)
```

Block N67 is the last threading pass, followed by automatic return motion to the start position. From that position, program ends the same way as for G32.

```
N68 G00 X12.0 Z4.5 T0500 M09
N69 M30
%
```

One frequent programming mistake that can be made with this cycle is to omit G00 command in block N68. G92 cycle can be canceled only *by another motion command*, in this case by a rapid motion G00. If G00 is missing, the control system will expect that there are more threads to cut, which is not the case - undesired motions may happen.

Simple threading cycle G92 is just that - it is simple, without any frills. It does not have any special infeed methods, in fact, the only feeding method available is a straight plunge type (radial infeed). Later in this chapter, the plunge method of infeed will be described as *not* suitable for most threading operations. Automatic gradual pullout can be programmed with G92 by using M24 function prior to calling the G92 cycle, with examples later in this chapter.

Most controls support much more sophisticated threading method - also a cycle - called G76, described next.

## MULTIPLE REPETITIVE CYCLE - G76

In Chapter 35, various lathe cycles were the main subject, normally used for turning and boring. In this section, a similar look will focus on another multiple repetitive cycles, this time used for various threading applications.

In early stages of CNC development, simple G92 threading cycle was a direct result of computerized technology of its time. Computer technology has been rapidly advancing and many great new features have been offered to CNC programmers. These new features simplify program development. One of the major additions is another lathe cycle, used for threading - a multiple repetitive threading cycle G76. This cycle is considered a complex cycle - not because it is difficult to use (on the contrary) but because it has some very powerful internal features.

To fully appreciate the impact of G76 threading cycle, compare it with the original G32 threading method, and even G92 cycle just described. While a program using the G32 method requires four or even five blocks for each threading pass, and G92 cycle requires one block for each threading pass, G76 cycle will do *any single thread* in *one* or *two* blocks of program, depending on control model. With G76 cycle, *any number* of threading passes will still occupy only a very small portion of the program, making any editing (if necessary) very easy and fast.

There are two programming formats available, depending on the control model. This is similar to programming of the other lathe cycles.

#### ◆ G76 Cycle Format - One Block Format

Any threading cycle requires initial data input - information provided to the control that defines a particular thread in machining terms. Figure 38-12 illustrates G76 cycle for older Fanuc 10/11/15T controls. Straight thread is shown.

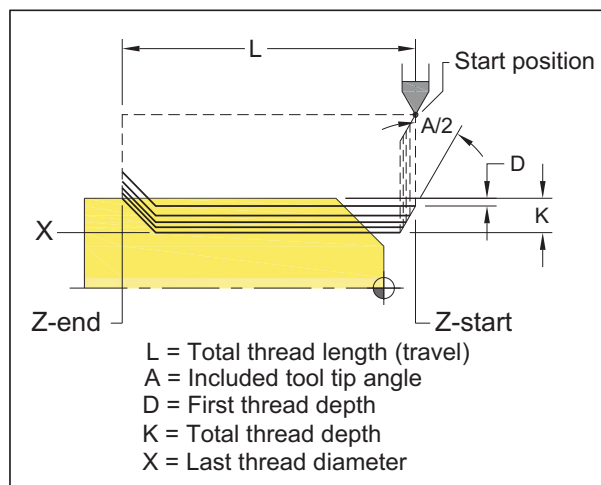


Figure 38-12

G76 multiple repetitive threading cycle for one block input - straight

The following parameters form the structure of G76 cycle applied as one-block (external *or* internal threads):

G76 X.. Z.. I.. K.. D.. A.. P.. F..

where ...

- X = Last pass thread diameter (external or internal)
- Z = Thread end position
- I = Taper amount over total length (I0 for straight threads)
- K = Actual thread depth per side - positive
- D = First threading pass depth - positive (no dec. point)
- A = Included insert angle - positive (six selections)
- P = Infeed method - positive (four selections)
- F = Feedrate (always the same as thread lead)

Observe differences in the format structure for multiple repetitive cycle G76 with simple G92 cycle. G76 cycle appears to be simple as well, but the control system must do a large number of calculations and checks. All these calculations need data, in the form of input parameters that establish required thread specifications. Yet, in spite of more input parameters, G76 cycle is very easy to use in CNC thread programming.

#### ◆ G76 Cycle Format - Two Block Format

On the later Fanuc controls 0T, 16T, 18T, 21T and others, format of G76 cycle is somewhat changed from older models. Its purpose and function remain the same, the difference is only in the way how program data input is structured. Fanuc 10/11/15T use a single line cycle input, described earlier. Fanuc 0/16/18T/21 and other models require a two line input. Programmer does not have a choice - each format depends on the control system.

One-block and two-block G76 cycles are not interchangeable

If the control system requires a two-block entry for G76 cycle, programming format must cover two blocks:

G76 P.. Q.. R..  
 G76 X.. Z.. R.. P.. Q.. F..

where ...

FIRST block - starts with G76:

- P = ... is a **six-digit** data entry in **three pairs**:  
 Digits 1 and 2 - number of finish cuts (01-99)  
 Digits 3 and 4 - number of leads for gradual pull-out (0.0-9.9 times lead), no decimal point used (00-99)  
 Digits 5 and 6 - angle of thread (tool tip angle) (00, 29, 30, 55, 60, 80 degrees only)
- Q = Minimum cutting depth (last depth of cut) (positive radial value - no decimal point)
- R = Fixed amount for finish allowance (decimal point allowed)

SECOND block - also must start with G76:

- X = a) Last thread pass diameter (absolute = X)  
 ... **or** ...  
 b) Distance from the start position to the last thread diameter (incremental = U)
- Z = Z-axis endpoint of thread (can also be incremental distance W)
- R = Radial difference between start and end thread positions at the final pass (R0 used for straight threads can be omitted)
- P = Single depth of thread (height of thread) (positive radial value - no decimal point)
- Q = First threading pass depth (largest cutting depth) (positive radial value - no decimal point)
- F = Feedrate (always the same as thread lead)

This format follows the logic of several lathe cycles described earlier in *Chapter 35*. Do not confuse P/Q letters of the first block with the P/Q letters of the second block. They all have their own meaning - within each block only! *Figure 38-13* shows some basic definitions of a straight two-block G76 threading cycle.

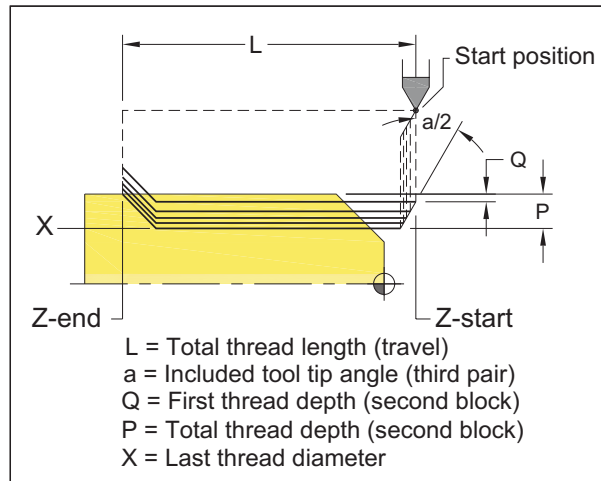


Figure 38-13

G76 multiple repetitive threading cycle for two block input - straight

### ◆ One-Block vs Two-Block Format

Some programmers may ask why Fanuc has made such a dramatic change from a relatively simple single block entry to a more cumbersome double block entry. The answer can be one word - *flexibility*. Compare features that can now be programmed, features that did not exist in the single block format. For example, the *last depth of cut* (minimum cutting depth). In a single block format, this amount was only stored as a control system parameter and could be changed internally only. Now, the double block format allows the same parameter to be changed externally, through the program. Other features follow the same reasoning.

As both formats will be described in more detail in the next section, here are two typical and independent entries using a two-block G76 cycle:

#### ➡ Example - Metric units ... **Internal M76×1.5 thread:**

```
N10 G76 P011060 Q050 R0.05
N11 G76 X76.0 Z-30.0 P812 Q250 F1.5
```

#### ➡ Example - Imperial units ... **External 1-11/16 thread with 20 TPI:**

```
N20 G76 P011060 Q005 R0.003
N21 G76 X1.6261 Z-1.5 P0307 Q0100 F0.05
```

### ◆ Programming Examples

The earlier threading example used for G32 and G92 was 3.0 external diameter with 12 TPI (*Figure 38-7, page 193*). It can be easily adapted to G76 programming method with a rather short output. Examples for both types of controls will be shown, using only the *minimum* number of program blocks (last tool shown in examples):

#### One-block G76 Cycle

##### O3803 (G76 VERSION - ONE BLOCK METHOD)

```
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 G76 X2.8978 Z-1.6 I0 K0.0511 D0210
      A60 P1 F0.0833
N63 G00 X12.0 Z4.5 T0500 M09
N64 M30
%
```

The fact that the entire threading section requires only six blocks is quite significant. Any programming change can be done by simple modification of selected data in block N62, which is the threading cycle. The most common change is the first threading pass depth, for example, from the current 0.0210 to 0.0180. All that has to be modified is address D0210 to D0180. As a result, more threading passes will be cut, presumably for a better quality thread. Feedrate can have up to six decimal places for long threads, and I0 can be omitted for straight threads.

#### Two-block G76 Cycle

##### O3804 (G76 VERSION - TWO BLOCK METHOD)

```
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 G76 P011060 Q004 R0.002
N63 G76 X2.8978 Z-1.6 R0 P0511 Q0210 F0.0833
N64 G00 X12.0 Z4.5 T0500 M09
N65 M30
%
```

Although only one block longer than the older method, possible changes offer much more flexibility at the machine. No more changing system parameters - several more options are now quickly available, if needed.

The comparison of the G76 cycle with G92 cycle is unfair, as each cycle is product of a different technological era. They coexist in the same control unit even now, mainly to be *downward compatible* with older programs. These two cycles provide good illustration of some significant differences between programming techniques.

For example, in the G92 threading cycle application, input of *each* thread pass diameter is important, in G76 cycle, only the *last* pass diameter input is important.

### ◆ First Thread Diameter Calculation

Control system calculates the actual first pass diameter in a very similar way as done manually in G32 or G92 threading modes. Calculation of the *first* threading diameter is based on supplied *external* thread information:

- Root diameter [ X address ]
- Total thread depth [ K or P address ]
- First thread depth [ D or Q address ]

For *one block* format, and based on supplied values, the first threading diameter  $T_f$  will be calculated as:

$$T_f = X + K \times 2 - D \times 2$$

In the example, X is 2.8978, K is 0.0511, the first threading depth D is 0.0210, entered in the program as D0210. Therefore, the first pass threading diameter  $T_f$  will be:

$$\begin{aligned} T_f &= 2.8978 + 0.0511 \times 2 - 0.021 \times 2 \\ T_f &= 2.9580 \end{aligned}$$

For *two block* format, and based on supplied values, the first threading diameter  $T_f$  will be calculated as:

$$T_f = X + P \times 2 - Q \times 2$$

In the example, X is 2.8978, P is 0511, the first threading depth Q is 0.021, entered in the program as Q0210. Therefore, the first pass threading diameter  $T_f$  will be:

$$\begin{aligned} T_f &= 2.8978 + 0.0511 \times 2 - 0.021 \times 2 \\ T_f &= 2.9580 \end{aligned}$$

Both P and Q of the two block example are amounts entered in the second G76 block. In either case, the result is the same diameter as in previous simpler examples, but this time it was calculated by the control unit. Both formulas can be adapted to calculate the first threading diameter for internal threading:

$$T_f = X - K \times 2 + D \times 2$$

$$T_f = X - P \times 2 + Q \times 2$$

For every day programming, the first diameter calculation is not necessary for either G76 cycle and is included here only to explain how the control system calculates it.

Note that the total thread depth uses 0.541 constant for internal threads rather than the 0.613 constant for external threads.

## THREAD INFEEED METHODS

Entry of a threading tool into material can be programmed in more than one way. One of the most important selections is a method that controls threading tool approach towards the thread, also known as *threading infeed*. This is a method detailing threading tool motions, using one of *two* basic methods of infeed, as illustrated in Figure 38-14.

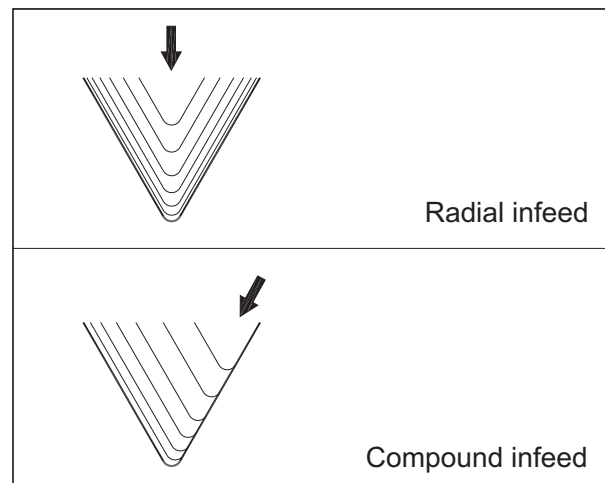


Figure 38-14

Radial and compound infeed for thread cutting

The simplest infeed method in thread programming is called the *radial* infeed method, also known as the *plunge*, *straight*, or *perpendicular* method. More common method is called an *angular* method, better known as a *compound* infeed or a *flank* infeed.

A need to control infeed direction in threading is to offer the best cutting conditions for cutting tool edge. Except for threads with very fine leads and some soft materials, the majority of threading cuts will benefit from a compound infeed (at an angle). Some threaded shapes are excluded for the reason of their geometry - for example, a square thread will always need a radial infeed (straight plunge infeed). The angle of infeed is programmed as the included tool tip angle in both types of G76 cycle.

Each infeed method has its variations, using the following features for one purpose - *the best chipload control*:

- Constant cutting amount
- Constant cutting depth
- One edge cutting
- Both edges cutting

There is a lesser known setting P in the one-block G76 cycle that selects each procedure - *see page 368*.



### ◆ Radial Infeed (Straight Infeed)

*Radial* infeed method is a common threading method for some jobs. For horizontal threads, it is used when a perpendicular motion towards the spindle centerline is required.

In G76 one block threading cycle, **A0** parameter is used for a radial infeed. In a two block threading cycle, the last two digits of the first block P-address will be two zeros, as in **P----00**.

In G32 block-by-block programming and G92 simple threading cycle, there is no parameter to program. The Z axis start position is the same for all thread diameters and is easier to program. The radial infeed is suitable for soft materials (brass, some aluminum, etc.), but it could damage threads cut in harder metals.

Main result of a radial infeed motion is that *both* insert cutting edges are removing material at the *same time*. Since both edges are opposite to each other, curling of the chips will also be opposite to each other. In many applications, this will cause high temperature and tool wear problems related to heat and may create a groove-like channel on the insert. Even decreasing depth for each infeed may not eliminate the problem. If radial infeed does not produce a high quality thread, a compound infeed approach will generally do a much better job.

### ◆ Compound Infeed (Flank Infeed)

Compound infeed method, also called a *flank* method, uses an *angular* cutting direction that moves pass by pass towards the final thread depth. Chip shape produced by compound threading method is similar to the shape of a chip produced by turning. Only one insert edge does the actual cutting, so most generated heat dissipates *away* from tool edge, while chips curl away, extending tool life. The chip depth can be heavier and fewer passes will be required for most threads. *Figure 38-14* shows compound infeed, where one cutting edge is in constant contact with a thread side. There is no cutting, only undesirable rubbing which may cause a poor surface finish and shorten tool life. To avoid this problem, calculate each threading pass position with a small angular clearance - *Figure 38-15*.

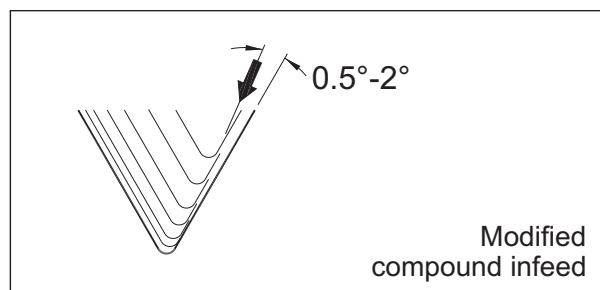


Figure 38-15

Modified compound infeed angle for better thread quality

For example, a typical V-thread, with 60° included angle has a flank angle of 30° - in this case, selected infeed angle should be a little less than that, say 29°, which provides a 1° clearance on one thread side. *Keep in mind* - the thread shape or geometry is not changed, as it is built into the physical shape of cutting insert. What is changing is the way *how* the insert will cut. An example using G32 threading method with modified compound infeed will be provided in the next section.

G76 threading cycle offers some very powerful input options, in forms of cutting parameters. There is no selection of infeed method available directly, but infeed can be influenced by other settings, particularly the tool tip angle. This setting is available for both types of G76 cycle.

### ◆ Thread Insert Angle

For compound infeed, a *non-zero* value is assigned to the parameter that represents tool angle, a value equal to the *included* angle of threading insert. Only the following six angle settings are allowed in a one-block G76 threading cycle:

A0	A29	A30	A55	A60	A80
A0	Radial (straight) infeed			ISO	
A29	ACME type of thread			ANSI	
A30	Metric Trapezoidal / Round thread			DIN 103 / DIN 405	
A55	Whitworth 55° thread			BSW, BSF, BSP	
A60	Standard 60° V-thread			M, UN, UNJ	
A80	German PG thread			Panserrohrgewinde	

For a two-block G76 cycle, the same six options are programmed as the last two digits of the *first* block P-address:

```
G76 P----00 Q. . R. .
G76 P----29 Q. . R. .
G76 P----30 Q. . R. .
G76 P----55 Q. . R. .
G76 P----60 Q. . R. .
G76 P----80 Q. . R. .
```

In theory, *any* of the six available options can be used for *any* thread, regardless of its actual tool tip angle. This may offer some creativity to find the 'best' infeed method.

In practical applications, *ACME* thread (29° included angle) is very common for transmission of motion, for example a tire changing jack. *Metric Trapezoid* thread is the metric version of ACME thread and has a 30° included angle. *Whitworth* thread has an included angle of 55°, with origin in Great Britain; it has limited usage, as metric threads become standard worldwide. *PG* thread is a special German pipe thread (*Panserrohrgewinde*), with 80° angle, not common in North America - it looks like a garden hose thread.

One-block G76 cycle (for older controls) offers a *P* parameter, that works very closely with the *A* parameter and defines the *thread cutting type*.

### ◆ Thread Cutting Type - Address P

In a one-block G76 threading cycle, threading infeed can be programmed with the address P, in addition to the address A. The purpose of threading parameter A is to control the threading infeed method - up to a certain extent - based on the *included angle* of threading insert. For a more controlled infeed method, a method that controls the threading *depth*, there is also parameter P, programmable in the G76 cycle format and available for the Fanuc controls that use one-block G76 input. It defines the *thread cutting type*, relating to the programmed thread depth.

In addition to radial infeed (straight or plunge), programmed with A0 parameter and compound infeed - a non-zero parameter A - there are two other main cutting types that can be used in programming a thread infeed - a *one side cut* and a *zig-zag cut*. These terms refer to the number of cutting edges employed at one time. The one side cut refers to cutting with *one edge*, the zig-zag cut refers to cutting with *two cutting edges*. Each of them can be used in conjunction with the selected A thread angle parameter and cutting depth - either as a *constant amount* or a *constant depth*.

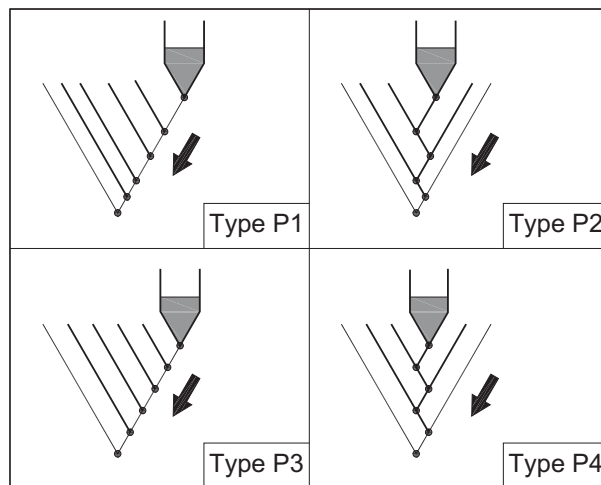


Figure 38-16

Cutting types for one-block G76 threading cycle

'P' parameter

Fanuc CNC lathe controls offer four methods of controlling the thread cutting depth infeed (Figure 38-16):

- |    |                       |                              |
|----|-----------------------|------------------------------|
| P1 | One edge cutting ...  | with constant cutting amount |
| P2 | Two edges cutting ... | with constant cutting amount |
| P3 | One edge cutting ...  | with constant cutting depth  |
| P4 | Two edges cutting ... | with constant cutting depth  |

On Fanuc lathe controls manufactured before model 10T, the P parameter in G76 cycle was not available. The equivalent of what is now the P1 parameter was the default. On controls that *do* support P parameter, if the P number is omitted in G76 cycle call, P1 cutting method is assumed as a default. This is the most common threading application and will be suitable for many jobs. It will apply *one cutting edge* of the threading tool and the *constant cutting amount*. That will result in equal chip volume removal. Feel free to experiment with the other three options as well.

## COMPOUND INFEEED CALCULATIONS

Compound (flank) infeed does not present programming problems when using the advanced G76 threading cycle. If a CNC system has G76 cycle available, it can be used for about 95% of all work. What about the remaining 5%? What if G76 cycle cannot be used and a program needs fully controlled compound infeed? How to control other infeed methods available for G76, without having the G76 cycle available or finding it impractical to use?

Unfortunately, there is only one way - take a pocket calculator and calculate each and every tool position and tool motion individually. Is it a lot of work? Yes. Is it worth doing? Absolutely. It has to be a really good job from the start, because even a slight modification at the machine could be very difficult. A top class programming job is always worth the extra time and effort when quality and precision of the final part depend on it. Quality is not instant - programmers (and machine operators) have to invest some extra work and time into it.

General principles of compound threading as applied to a block-by-block programming are simple, but the programming work may be tedious and editing on the machine may be impractical. Each threading pass has to be calculated in a different Z-axis start position. This is called the *shifted* position that must be calculated exactly, otherwise the program will fail. It also had better be right the first time, otherwise any changes could be long and costly. Again, in this example, the same thread will be used as in previous examples (Ø3.0-12 TPI external thread, introduced earlier, on page 355). Program will use G32 threading command, with a modified compound infeed at 29°.

### ◆ Initial Considerations

Thread used for the examples in this section is a Ø3.0-12 TPI external thread. All individual diameters for each threading pass had been calculated earlier, and all single depths for each pass had been established as well at the same time. These values will be used in this example as well. In total, there are six diameters and six depths to program the Z-value (total depth accumulation is 0.0511 per side), as illustrated in Figure 38-17.

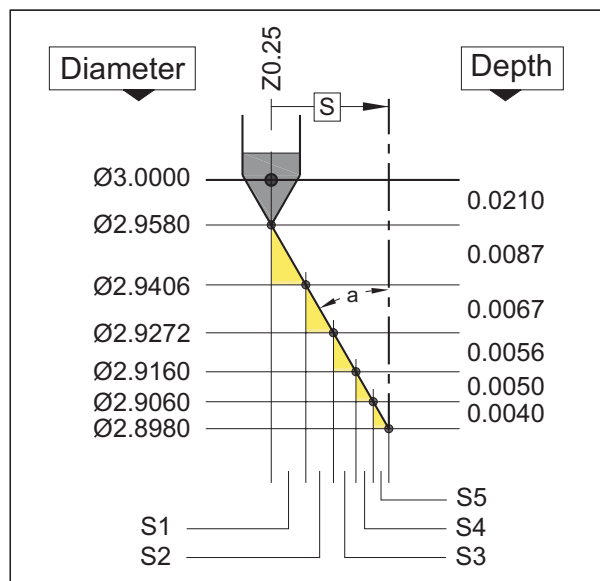


Figure 38-17

Calculations required for compound infeed used in G32 threading mode

The illustration shows distribution of each single threading depth for all six threading passes and matches the following summary:

Pass #1	single depth of 0.0210	at Ø 2.9580
Pass #2	single depth of 0.0087	at Ø 2.9406
Pass #3	single depth of 0.0067	at Ø 2.9272
Pass #4	single depth of 0.0056	at Ø 2.9160
Pass #5	single depth of 0.0050	at Ø 2.9060
Pass #6	single depth of 0.0040	at Ø 2.8980

In addition to individual diameters and depths, Figure 38-17 also shows combined *shift* of individual S1-S5 shift amounts as the *S* amount. When Z-start position is shifted, the shift distance must be calculated on the basis of compound angle *and* the threading pass single depth, but only for the *remaining* passes. That means for six threading passes, there will be five shifts. Every new calculation must always be based on the last calculation.

#### ◆ Z-axis Start Position Calculation

Illustrated distance *S* represents the *total* shift amount from the nominal Z-axis start position - in the example, this position is at Z0.25. Usually, the shift is to the Z-positive direction, mainly for reasons of increasing the clearance rather than decreasing it. Opposite direction is possible.

Theoretically, it makes no difference *which* direction the shift is programmed - *towards* the thread or *away from* the thread. Practically, it is always better to program the shift *away from* the thread, if possible - the tailstock may be in the way, so watch its position. This way, the distance for the feedrate acceleration will *increase*, rather than decrease.

Although another approach may also be chosen, the *S* distance will be calculated first, mainly for reference purposes. Only five shifts are required, so the total shift *S* must be calculated *after* the first pass is completed. It means from the first depth Ø2.958 to final depth Ø2.898:

$$(2.958 - 2.898) / 2 = 0.0300$$

Total thread depth after the first pass is 0.03 and the selected compound infeed angle *a* is 29°, so using a standard trigonometric formula will provide the *S* distance based on infeed angle *a*:

$$S = 0.03 \times \tan 29 = 0.016629272 = 0.0166$$

The *S* distance represents the *total* shift of a threading tool from the start position. Shift for each threading pass will be its relative share of the *S* value. Each share is identified as an *S_n* in the Figure 38-17, within 5 passes - S1 to S5.

$$S_n = D \times \tan 29$$

where ...

- $S_n$  = Current thread pass shift - incremental amount
- $n$  = Current pass number (shift count)
- $D$  = Single depth of current thread pass

Calculation for each pass uses the same basic formula, changing the *D* depth input. Keep in mind that the purpose of this process is to find a new Z-axis start position for *each* threading pass - i.e., the Z shift amount for a given thread diameter. Figure 38-18 illustrates the calculation.

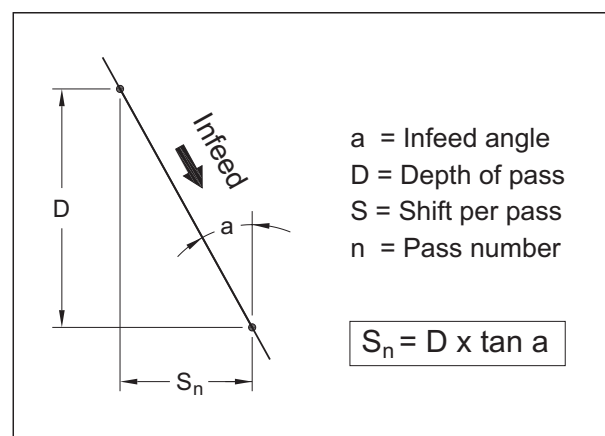


Figure 38-18

Calculation of thread Z-start position for individual passes

Once the Z-axis start position is established (Z0.25), it is used for the remaining five shifts. *There is no shift for the first pass.* The remaining five shifts are based on the initial position and can be calculated from the above formula:

$S1 = 0.0087 \times \tan 29 = 0.0048$   
 $S2 = 0.0067 \times \tan 29 = 0.0037$   
 $S3 = 0.0056 \times \tan 29 = 0.0031$   
 $S4 = 0.0050 \times \tan 29 = 0.0028$   
 $S5 = 0.0040 \times \tan 29 = 0.0022$   
**T O T A L** ..... = 0.0166

The five shifted positions for Z-axis start position can be calculated, based on the selected start position of Z0.25:

Pass #1 = Z0.2500 + 0 = Z0.25  
 Pass #2 = Z0.2500 + 0.0048 = Z0.2548  
 Pass #3 = Z0.2548 + 0.0037 = Z0.2585  
 Pass #4 = Z0.2585 + 0.0031 = Z0.2616  
 Pass #5 = Z0.2616 + 0.0028 = Z0.2644  
 Pass #6 = Z0.2644 + 0.0022 = Z0.2666

This example shows progressive shift from the initial position of Z0.25, away from the thread. Using this method offers a certain level of confidence that the original Z0.25 minimum clearance will never be smaller. Only the Z-axis value will change - other programmed values are not affected by this programming method at all.

The final program is not short, even for only six passes, which is typical with G32 programming, but it does illustrate the compound method of threading when no cycle is available or is not practical to use. Only the threading tool is shown in the example O3805.

#### O3805 (G32 THREADING - COMPOUND INFEEED)

```

...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 X2.958 (PASS 1)
N63 G32 Z-1.6 F0.0833 (or F/E0.083333)
N64 G00 X3.3
N65 Z0.2548 (SHIFT S1)
N66 X2.9406 (PASS 2)
N67 G32 Z-1.6
N68 G00 X3.3
N69 Z0.2585 (SHIFT S2)
N70 X2.9272 (PASS 3)
N71 G32 Z-1.6
N72 G00 X3.3
N73 Z0.2616 (SHIFT S3)
N74 X2.916 (PASS 4)
N75 G32 Z-1.6
N76 G00 X3.3
N77 Z0.2644 (SHIFT S4)
N78 X2.906 (PASS 5)
N79 G32 Z-1.6
N80 G00 X3.3
N81 Z0.2666 (SHIFT S5)
N82 X2.898 (PASS 6)
N83 G32 Z-1.6
N84 G00 X3.3
N85 Z0.25 M09
N86 X12.0 Z4.5 T0500
N87 M30
%
```

In program O3805, the thread infeed method is equivalent to the P1 parameter in G76 cycle (one-block method). This cutting type employs only a single edge of the threading insert, with a constant amount per each threading pass. It represents the most common programming method for threads and can be used as a sample for many other thread cutting applications. Block-by-block threads will be longer and will need to be checked for accuracy very carefully.

## THREAD RETRACT MOTION

As stated already, there are only two methods of retracting threading tool from the part - a straight motion along a *single axis*, and a gradual *simultaneous* motion along *two axes*. Both are used in single point thread programming. In fact, their frequent applications even justify special methods of program input.

The first method has actually been described already - see page 364 (G76 - first block). G76 cycle in its two-block format provides an input parameter that allows setting of the retract motion - the *gradual pull-out* from the thread. In this case, there is no need for any other programming interference - this feature is built in the two-block G76 cycle.

For older - and still very common controls - using the *one block* G76 entry, there is a different solution. There are two miscellaneous functions built into the control system as a standard feature. These thread retract functions are called the *thread chamfering functions* or *thread finishing functions*. Another description is a bit more accurate (or at least descriptive) than these two - *thread pullout functions*.

### ◆ Thread Pullout Functions

When using threading cycle G76 for CNC lathe work, the thread end position (Z-axis value) will either be in *solid* material or in an *open* space (a *groove recess*, for example, or an *undercut*). Programmed pullout can be programmed along a single axis, or along both axes simultaneously.

For older controls, typical Fanuc functions designed for this purpose are M23 and M24. They control the pullout of the threading tool at the thread end:

M23	Thread finishing ON	(two axes pull-out)
M24	Thread finishing OFF	(one axis pull-out)

Other machine controls may have similar functions. The purpose of these functions is to enable or disable the *automatic insertion* of a pullout retract between threading motion sequences 2 and 3, as described earlier in this chapter. Figure 38-19 illustrates the comparison of a threading motion *with* and *without* gradual pullout for both methods.

M23 and M24 functions apply only to G76 one block method



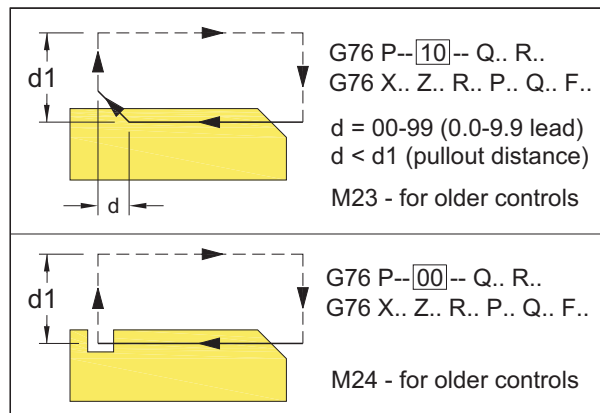


Figure 38-19

Typical gradual pull-off from thread - 1.0 lead shown in upper image  
Only older controls require M-functions M23/M24

### ◆ Single Axis Pullout

A single axis pullout (thread finishing OFF) is a simple rapid motion programmed at the end of threading pass as the *third motion* of the four threading sequences. Pullout direction is always in X-axis. For either threading cycle G92 or G76 (one-block method), this is the default condition, so M24 is not needed, unless M23 function is used as well, usually for another thread in the same program. These two functions cancel each other. If M24 function is used, it must be programmed *before* the threading cycle for which it has been applied. For example, threading program O3803 using the G76 one-block cycle will be slightly modified in O3806:

```
O3806 (G76 VERSION - ONE BLOCK METHOD)
...
N58 M24                      (THREAD PULLOUT OFF)
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 G76 X2.8978 Z-1.6 I0 K0.0511 D0210
    A60 P1 F0.0833
N63 G00 X12.0 Z4.5 T0500 M09
N64 M30
%
```

M24 function appears in block N58, the only block that was available without another M function. Example for a single axis pullout will be a modified program O3804:

```
O3807 (G76 VERSION - TWO BLOCK METHOD)
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 G76 P010060 Q004 R0.002
N63 G76 X2.8978 Z-1.6 R0 P0511 Q0210 F0.0833
N64 G00 X12.0 Z4.5 T0500 M09
N65 M30
%
```

Underlined item in N62 has been changed from 10 to 00.

### ◆ Two-Axis Pullout

Two-axis pullout is a gradual angular tool motion along two axes, away from the thread (thread finishing ON). The example O3808 is similar to the previous example:

```
O3808 (G76 VERSION - ONE BLOCK METHOD)
...
N58 M23                      (THREAD PULLOUT ON)
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.25 T0505 M08 (START POSITION)
N62 G76 X2.8978 Z-1.6 I0 K0.0511 D0210
    A60 P1 F0.0833
N63 G00 X12.0 Z4.5 T0500 M09
N64 M24                      (CANCEL M23)
N65 M30
%
```

In this case, M23 was applied in block N58 and an additional block N64 was used to cancel the pullout. This cancellation was not necessary in this program, but it is a good practice to cancel functions used only for specific purposes.

There are some conditions that apply to the M23 function. In Figure 38-19, the finishing distance  $d$  is set by the control parameter, within the range of  $0.100\times$  to  $12.700\times$  the thread lead. Normal control setting is equivalent to *one times* the thread lead ( $d=1.0$ ). Pullout angle from the thread is usually at  $45^\circ$ , or a little less because of a delay in the servo system. If the finishing distance  $d$  is greater than the pullout distance  $d1$ , the pullout will *not* be done.

Program example O3804 on page 365 shows a pullout of one lead ( $1.0 = 10$ ) in block N62:

```
N62 G76 P011060 Q004 R0.002
N63 G76 X2.8978 Z-1.6 R0 P0511 Q02100 F0.0833
```

In the G76 first block, the address  $P$  has six digits, separated into three pairs. Do not confuse the first pair with the second pair - they both indicate the amount of *one* with totally different meanings:

**P01xxxx** indicates *one finishing cut* within the range of 00 to 99 integers only.

**Pxx10xx** indicates *one lead* for pullout, within the range of 0.1 to 9.0. Real numbers are allowed, but the decimal point is not written.

## HAND OF THREAD

Any thread can be cut in either *right hand* or *left hand* orientation. Neither selection has any effect on the profile or thread depth, but other factors are important. The majority of threading applications use the right hand thread. *Right hand* and *left hand* terms relate to the helix of the thread - Figure 38-20.



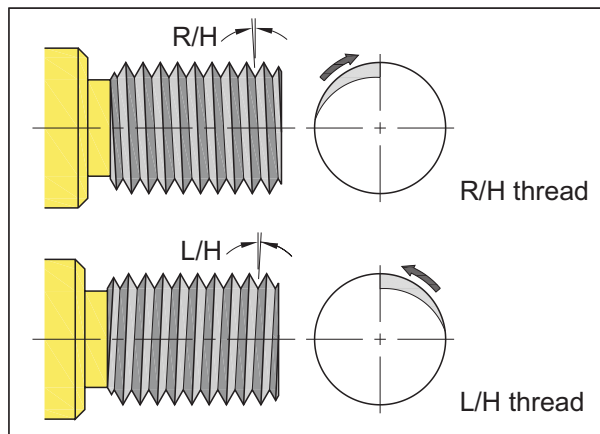


Figure 38-20

Right Hand and Left Hand threads shown on a screw

In CNC programming, the hand of thread is determined by two conditions:

- Cutting direction of the tool (Z+ or Z-)
- Direction of the spindle rotation (M03 or M04)

These conditions are used in various combinations in order to program a particular thread. Factors that influence the programming method for a R/H and L/H thread are:

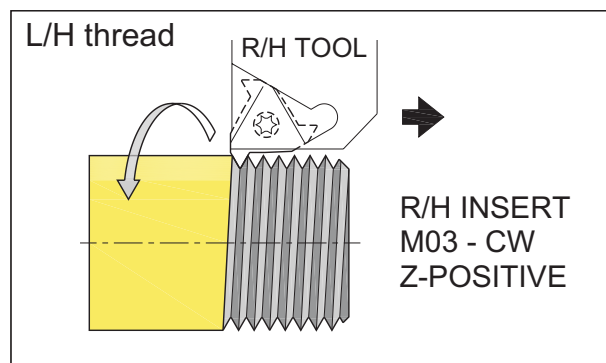
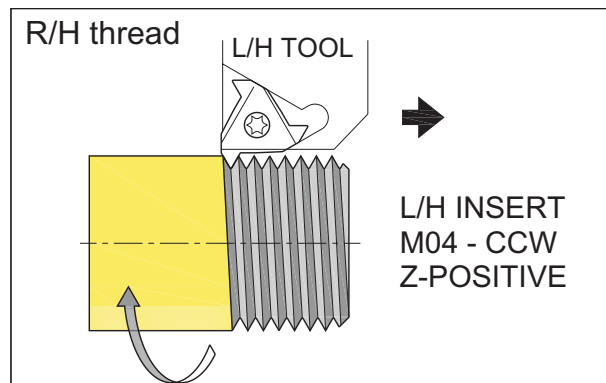
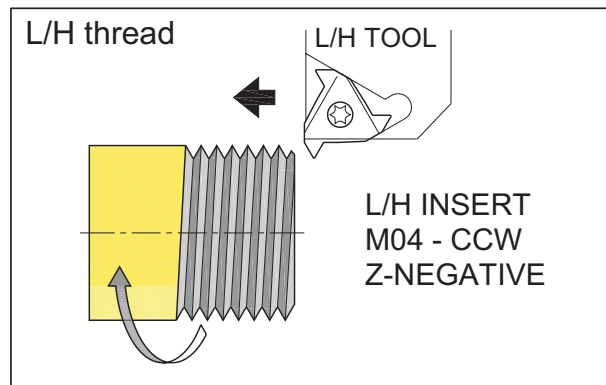
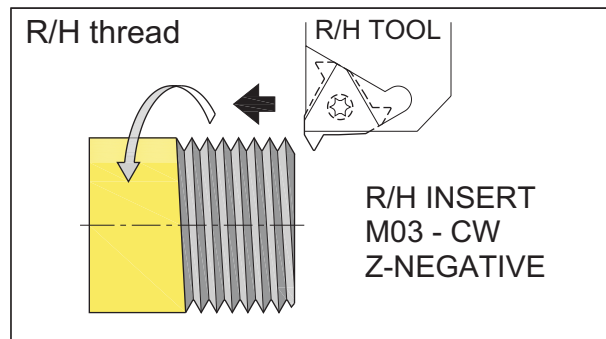
- Threading tool design - right hand or left hand
- External or Internal thread
- Front or Rear lathe design
- Spindle rotation direction - M03 or M04
- Cutting direction - Z+ or Z-
- Tool tip orientation in the turret

Theoretically, either hand of thread can be cut with *any* threading tool, but this approach is not right. A poor choice affects the thread quality, life of threading insert, additional costs are involved, etc. When a thread starts too close to a shoulder (in a recess), the clearance for acceleration is limited. The only method to prevent imperfect threads due to acceleration in a small area is to *decrease* the spindle speed.

#### ◆ Configurations for Hand of Thread

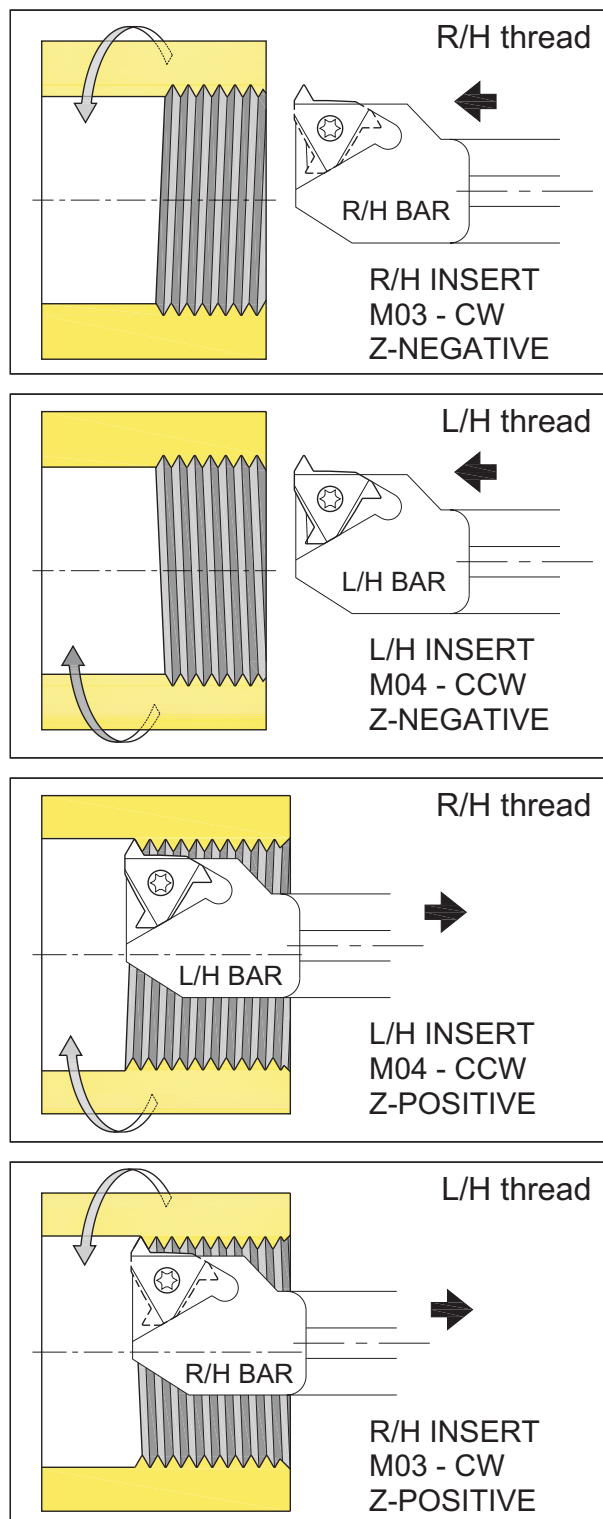
Often, it is difficult to visualize the actual machining, and making a mistake is quite easy. The most important consideration is that *all* factors involved are correct - even one incorrect factor will make a wrong thread.

Tooling manufacturers recognize this situation and provide graphical method similar to the ones shown here. All eight combinations cover R/H and L/H external and internal threads for a rear type lathe. Design of tool may influence programming - tools are shown at their start position - M03/M04 spindle rotation is described on *page 82*.



Figures 38-21, 38-22, 38-23, 38-24

Hand of external threads - REAR lathe applications



Figures 38-25, 38-26, 38-27, 38-28  
Hand of internal threads - REAR lathe applications

## THREADING TO A SHOULDER

Programming a thread that terminates at a shoulder presents a unique difficulty. This difficulty is a wall - better known as *shoulder* of the part. It is not enough to program the thread end point reasonably - it must be programmed *exactly*. Even then, a collision is possible if the tool setup is not accurate. There are three typical problems in this area of thread programming:

- Recess groove is too narrow or non-existent
- Threading insert is too wide
- Thread is too deep

The first problem of threading towards a shoulder, a narrow width of the recess groove, is easy to correct - just increase recess width in the program. The majority of recess grooves can be adjusted for threading tool, without damaging engineering intent behind the design. This may be a justified case of 'overruling' the drawing - but check first anyway!

The second and third problems may not be related, but their solution is usually the same. If the threading insert is too wide or thread is too deep, try to increase the recess width first, if at all possible. If the recess width cannot be increased, for whatever reason, then there is another choice - to *decrease* the threading insert width. One obvious solution is to change the current threading tool for a smaller one that can still cut the required thread depth. This may be an insert one size smaller, which usually requires a different tool holder as well.

If a smaller tool cannot be used, program for a *modified existing threading insert*. Modification in this case means physical grinding off the problematic portion of the insert, without disturbing the portion that actually removes material. Before deciding on any modification by grinding, consider other options carefully - altering standard tools designed for CNC work should always be the *last* resort, not an automatic first choice. A coated insert will lose its cutting advantages, if the coating is removed by grinding. Be careful not to grind off coating within the cutting section of the insert. In case the part program *does* use a modified threading insert, a few suggestions may help to do it with more knowledge.

Always use care with modified tools

### ◆ Insert Modification

There is a number of standard threading inserts in every tooling catalogue and chances of finding one suitable for the job at hand are very good. In case a standard threading insert needs modification, the following example illustrates a few programming methods - incidentally, it is irrelevant if there *is* or there *is not* a recess groove on the part.

To modify a standard threading insert, look at its normal configuration first. *Figure 38-29* shows a typical threading insert with the known width  $W$  and the angular length  $A$ , tip radius or flat  $R$ , and an unknown angular height  $H$ .

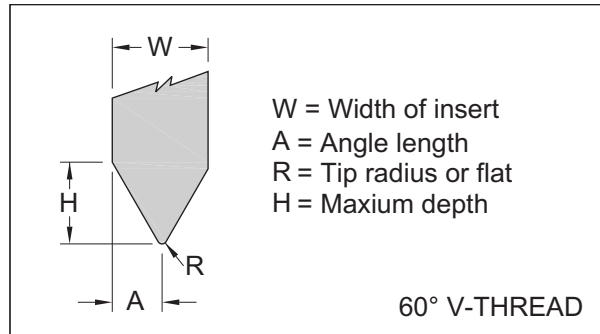


Figure 38-29

Essential dimensions of a typical threading insert

In the example, catalogue based insert dimension shown as  $W$  is 0.250 and  $A$  dimension is 0.130. Included angle of the threading insert is  $60^\circ$  and insert flat or tip radius  $R$  is 0.012, which is not relevant in this case. Dimension  $H$  indicates the maximum theoretical thread depth and is normally measured to the sharp point of insert tip. It is calculated using a trigonometric function:

$$\begin{aligned} H &= A / \tan 30 \\ H &= 0.130 / 0.577350269 \\ H &= 0.225166605 \\ H &= 0.2252 \end{aligned}$$

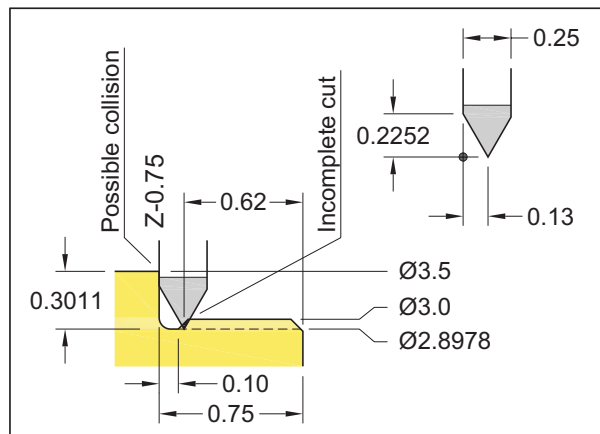


Figure 38-30

Threading insert before modification may cause problems

Two possible threading problems are illustrated above, in *Figure 38-30*. The example shows programming a thread with a 0.100 recess groove width, using an insert with angular length  $A$  of 0.130. This insert is *not* suitable for the job, as it cannot provide the minimum full depth thread length - the difference between the shoulder length and recess width:

$$0.750 - 0.100 = 0.650$$

The minimum thread length shown is only 0.620. There is no clearance and the thread length is too short. To solve this problem, select a smaller size threading insert if possible. If not, *modification* of a larger insert is the only way.

Such modification requires grinding of the insert in non-critical areas, to allow it to complete the minimum 0.650 thread length. In theory, the minimum amount to be ground off the insert is 0.030, difference between the required and actual thread lengths. This modification does not provide any *clearance* at the thread shoulder or at insert tip. These two clearances are essential for the best threading results. Even a minor setup error at the machine can cause some serious difficulty.

Always calculate modification amounts, never guess them

In the example, there are three dimensions that influence the actual amount of insert modification. Summing up all three will be the amount to be ground off the insert. One, the thread length has to be extended by 0.030 to achieve 0.650 minimum length. Two, clearance from the shoulder will also be 0.030, and three, clearance past the thread end will be 0.020. The last two clearances are arbitrary decisions by the programmer to suit the job. Final result is the total amount of insert modification as 0.080. It means the amount of 0.080 must be ground off the original larger threading insert. That will shorten the original angular length of 0.130 to the length of 0.050. Always make sure full depth of thread can be achieved with the modified insert. Part program will reflect the modification in Z-axis thread end position, which will be written as Z-0.8. Setup position of the insert - its *command point* - does not change. See *Figure 38-31* for complete solution.

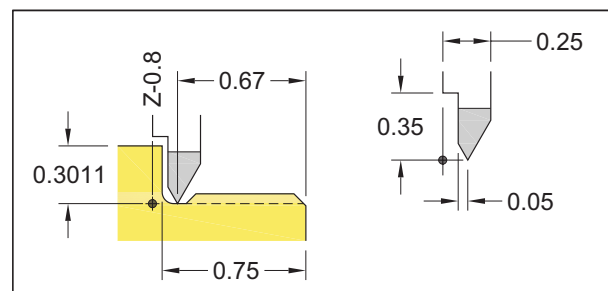


Figure 38-31

Modified threading insert provides sufficient clearance in the recess

In threading, thread length is *the actual length of the full depth thread*. Part design often allows a little longer thread, but not shorter. The shoulder height is also important. In the example, shoulder is only 0.3011 high, so insert modification was possible. A large threading insert may not always be modified and the only solution will be to use a smaller insert size.

### ◆ Program Testing

Whether a threading insert used is based on catalogue dimensions or a modified insert, threading to a shoulder presents a time of special caution by the CNC operator, when the first part is produced. Since feedrate override and feedhold switches are disabled during threading, program verification on the lathe will become more difficult. Even computer based graphic testing toolpath methods may not show all possible collisions.

A simple, yet very effective, thread program checking method is always available, right at the CNC lathe. This method requires a skilled CNC lathe operator, who does understand both the program and threading principles well. Knowledge of machine operation is also important.

This method employs several features found on contemporary CNC controls. Main purpose of the program test is to find out if the threading tool will collide with the shoulder *before* actual threading cut takes place.

The following steps are only general - adapt them to suit actual conditions when testing a thread cutting program:

- Use SINGLE BLOCK mode and step through the program until the thread start position is reached
- Switch from AUTO to MANUAL mode - spindle stops and the threading tool is in clearance area
- Select XZ screen display (absolute mode)
- Switch to HANDLE mode for the Z-axis
- While watching the XZ position display, move the handle in the same direction as the thread, until the tool reaches programmed Z-value, *or* it cannot move any further, whichever comes first
- If the tool reached target Z-position first, tool setup is safe for threading
- If the tool just about touched the part, but has not yet reached the target Z end position of the thread, tool setup needs adjustment by the difference between the programmed position and the actual position, plus some additional clearance

There are other testing methods available, for example, to use temporarily G01 *linear motion command*, instead of G32 *threading command*, without a part mounted in the spindle, of course.

In the non-threading mode, feedrate overrides are effective, whereas in threading mode, they are not !

By reading the current tool position on the screen display and comparing it with the programmed position, it will be possible to know whether a collision will happen or not. During the test, feedrate can be slowed down or stopped anytime. Main purpose of the program test is to establish safe working conditions *before* the threading takes place.

## OTHER THREAD FORMS

Although the standard V-shape thread with the 60° included tip angle is the most common thread form for both metric and imperial threads, it is by no means the only form. There are many thread forms and shapes programmers encounter in machine shops, too numerous to list.

As an example of a different threading form, look at an ACME thread as a subject for discussion. In metric, there is an equivalent thread, called the *Metric Trapezoidal* thread. From programming perspective, both threads are almost identical. ACME thread has a 29° included thread angle, metric trapezoidal thread has a 30° angle and somewhat different geometry definition. Different inserts are used.

Main application of a trapezoidal type thread is to *transmit a motion*, usually with a disengaging half-nut. Certain types of lead screws for conventional lathes use this type of thread as well. Programming trapezoid threads often requires a steady rest, since these threads may be quite long. An important consideration is the lead error accumulated over a long distance, discussed earlier - see *page 360*.

### ◆ Thread Depth - ACME example

Every thread has its formulas and mathematical relationships. There are two basic formulas relating to an ACME thread depth. One is for threads of 10 TPI and *coarser*, the other for threads of 12 TPI and *finer*. For ACME threads 10 TPI and coarser, the thread depth formula is:

$$T_d = 0.500 \times P + 0.010$$

For the ACME threads 12 TPI and finer, the thread depth formula is modified only slightly:

$$T_d = 0.500 \times P + 0.005$$

where ...

$$\begin{aligned} T_d &= \text{Thread depth} \\ P &= \text{Thread pitch} \end{aligned}$$

Other threads in the trapezoidal group are *Stub ACME* or a *60° Stub ACME*. Programming trapezoidal threads is no more difficult than programming any V-shape thread, providing thread formulas and geometric details of the thread design are known to the CNC programmer.

There are other threads that can be encountered outside of the 60° category - *Whitworth* threads, *Square threads*, *API* threads (used in the petroleum industry), *Buttress*, *Aero*, *Dardeclet* self locking threads, *Round* threads, *Lebus* threads (require special control features), and a number of others. Thread and threading data can be found in specialized tooling catalogues and technical publications.

## TAPERED THREAD

Programming procedure for a tapered thread is not significantly different than that for a straight thread. Tapered threading motion is along *two axes* simultaneously, rather than a single axis. The four basic motion steps are, therefore, almost identical to those for a straight thread:

- |          |                                                      |
|----------|------------------------------------------------------|
| Motion 1 | Rapid from the start position to the thread diameter |
| Motion 2 | Cut the thread (cutting along two axes)              |
| Motion 3 | Retract from thread                                  |
| Motion 4 | Return to the start position                         |

When compared to a straight thread, the only programming differences for a tapered thread are in the first two motions - *Motion 3* and *Motion 4* remain unchanged. In the *Motion 1*, the starting tool position is determined by the physical orientation of the threading tool - whether it is used for an external or an internal thread.

For *external* thread forms, the starting position of the threading tool must always be *above the largest diameter* of the thread. For *internal* thread forms, the starting position must always be *below the smallest diameter* of the thread. This is the same requirement as for a straight thread, but for a tapered thread it takes on an additional importance. For examples of a tapered thread, evaluate the simplified drawing in *Figure 38-32*.

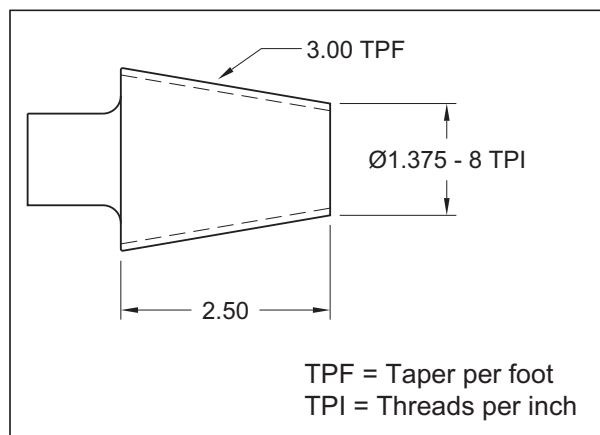


Figure 38-32

Basic drawing for a tapered thread example - program O3809

The thread is defined by its overall length (2.500), by its *front* diameter part (Ø1.375), by its taper angle (3.0 inches taper per foot) and by its pitch (8 TPI). It is a single start thread and the program zero is at the front face of finished part. All premachining operations have been done for this example. The first programming consideration for this type of machining should always be the *thread depth*.

### ◆ Depth and Clearances

From the previously established formula (*p. 354*), the external depth *D* of thread used in the program will be:

$$D = 0.61343 / 8 = 0.0766788 = 0.0767$$

Thread depth is measured axially and is not related to the thread angle (taper). Once the depth is established, clearances can be set - one in the front and one at the end of thread. Z-axis clearance amount will depend on the tool acceleration speed. Since threading feedrate will be programmed as F0.125, four times the lead rule of thumb would require a clearance to be 0.500 of an inch. Only 0.400 of an inch is sufficient with a relatively slow spindle speed of 450 r/min. End clearance can be smaller - there is enough open space at the thread end, and 0.200 is a reasonable clearance at this position, although a smaller clearance could be used as well.

For a tapered thread, always consider the *total* length of tool travel along each axis, *not* the actual thread length as per drawing - this is no different than for a single axis thread. Tool travel length in the example will be the combination of two selected clearances plus given thread length (along the Z axis):

$$0.400 + 2.500 + 0.200 = 3.100$$

The next step may not be always necessary, depending on actual method of programming. If a block-by-block G32 threading method is used, *both* start *and* end thread diameters will be needed for each pass. If a cycle G92 or G76 is used - the single distance between start and end diameter of a tapered thread is needed. This distance is programmed as a calculated dimension *I* of the one-block method or as a dimension *R* of the two-block threading cycle (second block). Previous examples were straight threads and I® values were zero (I0 or R0). I0 or R0 can be omitted.

### ◆ Taper Calculation

A thread taper has to be *calculated* to establish its start and end diameters. Any calculation method depends on how the taper is defined and dimensioned in the drawing. No part drawing will show the dimensions required for programming - they have to be calculated as part of the programming process, using one of two common methods.

One method uses the thread length and angle and can be calculated by applying standard trigonometric functions. The other method defines taper as a *ratio of its sides*. This method is often confusing to an inexperienced programmer. Typical ratios are defined in the part drawing directly, for example as 1:12, 1:16, etc., or indirectly, for example as the amount of *taper per foot* or, sometimes, as *taper per inch*. Keep one rule in mind:

Size of a taper is always measured on diameter



Standard North American pipe thread is a good example of a tapered thread. It is defined by a taper ratio of 1:16, which is equivalent to a  $\frac{3}{4}$  of an inch per foot taper, measured on diameter and perpendicular to an axis. Pipe thread may also be defined with a given angle per side - *one degree, forty seven minutes, twenty three seconds (plus some leftover)*, or 1.789910608 decimal degrees. For CNC programming, decimal degrees are preferred to degrees-minutes-seconds method (DMS or D-M-S), and many CAD drawings already reflect this preference. To understand all principles of a taper defined as the ratio of its sides, a definition followed by an example should help.

RATIO indicates the relationship between two values, expressed as a fraction

Both ratio values must be expressed in the same units and should be used in their lowest form of application ( $\frac{1}{4}$  instead of  $\frac{2}{8}$  or  $\frac{4}{16}$ ). For example, the ratio of 3 units to 4 units may have these forms:

$$3 : 4 = 3 / 4$$

In terms of a taper definition, it means that for every 3 units change along one axis, there will be 4 units change along the other axis.

TAPER PER FOOT indicates the difference between two diameters over the length of one foot or 12 inches

The drawing example of a 3 inch taper per foot is equivalent to a 1:4 taper ratio, because

$$3 / 12 = 1 / 4 = 1 : 4$$

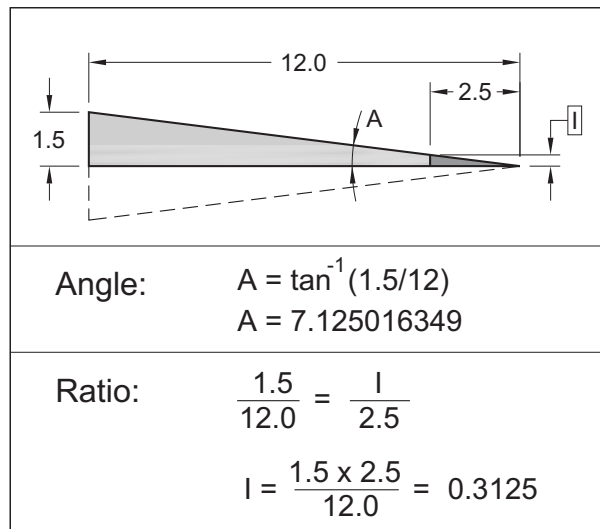


Figure 38-33

Taper thread calculations based on the example - clearances excluded

In CNC programming, we are only interested in calculating part diameters at the *beginning* and at the *end* of thread. These calculations can be done either by means of trigonometric functions, using the law of similar triangles, or by means of ratio calculations. Mathematical definitions relating to tapers start on page 501. Figure 38-33 shows angular calculation as well as a ratio calculation for the example.

Based on these principles, all required dimensions can now be calculated. Note that only *per side* or *radial* dimensions were used. In most programming applications, only one of the available methods will be required - either the angle *or* the ratio. There will always be an option to use the other method to verify accuracy of the calculations.

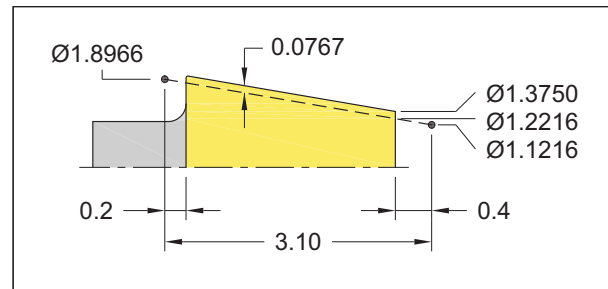


Figure 38-34

Calculated dimensions for taper thread example O3809

In Figure 38-34, the start and end diameters have been calculated using the angle and/or the ratio of sides method. Which results of the calculations will actually be used in the program will depend on the type of selected programming technique, such as using a block-by-block approach versus a cycle method. The details depend on the thread specifications and machine and control features.

### ◆ Block by Block Taper Thread

In block-by-block threading, taper thread programming is just as simple as programming a straight thread. To simplify the example, a straight infeed and nine threading passes will be used, for the total depth of 0.0767. The following nine depths must be applied at *both* ends of the thread. The first column lists depth of thread per pass, the second column lists front thread diameter, the third column is the end diameter. Front diameter is calculated at absolute Z-coordinate of Z0.4, the end diameter at Z-2.7:

Pass	Depth	Front Ø	End Ø
1	0.0165	1.2420	2.0170
2	0.0145	1.2130	1.9880
3	0.0120	1.1890	1.9640
4	0.0100	1.1690	1.9440
5	0.0080	1.1530	1.9280
6	0.0060	1.1410	1.9160
7	0.0040	1.1330	1.9080
8	0.0030	1.1270	1.9020
9	0.0027	1.1216	1.8966

All requirements are available to write program O3808:

```

O3809
...
(G32 - TAPERED THREAD)
N46 T0500 M42
N47 G97 S450 M03
N48 G00 X2.5 Z0.4 T0505 M08
N49 X1.242 (PASS 1)
N50 G32 X2.017 Z-2.7 F0.125
N51 G00 X2.5
N52 Z0.4
N53 X1.213 (PASS 2)
N54 G32 X1.988 Z-2.7
N55 G00 X2.5
N56 Z0.4
N57 X1.189 (PASS 3)
N58 G32 X1.964 Z-2.7
N59 G00 X2.5
N60 Z0.4
N61 X1.169 (PASS 4)
N62 G32 X1.944 Z-2.7
N63 G00 X2.5
N64 Z0.4
N65 X1.153 (PASS 5)
N66 G32 X1.928 Z-2.7
N67 G00 X2.5
N68 Z0.4
N69 X1.141 (PASS 6)
N70 G32 X1.916 Z-2.7
N71 G00 X2.5
N72 Z0.4
N73 X1.133 (PASS 7)
N74 G32 X1.908 Z-2.7
N75 G00 X2.5
N76 Z0.4
N77 X1.127 (PASS 8)
N78 G32 X1.902 Z-2.7
N79 G00 X2.5
N80 Z0.4
N81 X1.1216 (PASS 9)
N82 G32 X1.8966 Z-2.7
N83 G00 X2.5
N84 Z0.4
N85 G00 X12.0 Z4.5 T0500 M09
N86 M30
%
```

In the example, a radial infeed (plunge infeed) and pull-out is used for clarity. The program will not change much if a compound infeed is used and/or an angular pullout from the thread is applied. Of course, more calculations will be needed for the new Z-start position (shifted from Z0.4).

### ◆ Tapered Thread Using a Simple Cycle

In G92 threading cycle, thread taper is programmed as the radius *I* value, with specified direction *from* the end diameter *to* the start diameter:

G92 X.. Z.. I.. F..
---------------------

The X-address in the cycle represents current thread diameter at the *end of cut*, Z-address is the thread end position, *I* is the difference *per side* between thread diameter at the *end* and thread diameter at the *start*. The *I* value must include an algebraic sign (only minus sign must be written), specifying the *direction* of taper inclination, in this case a negative value. Program O3810 will cut a tapered thread using G92 threading cycle.

```

O3810
...
(G92 - TAPERED THREAD)
N46 T0500 M42
N47 G97 S450 M03
N48 G00 X2.5 Z0.4 T0505 M08
N49 G92 X2.017 I-0.3875 Z-2.7 F0.125 (PASS 1)
N50 X1.988 (PASS 2)
N51 X1.964 (PASS 3)
N52 X1.944 (PASS 4)
N53 X1.928 (PASS 5)
N54 X1.916 (PASS 6)
N55 X1.908 (PASS 7)
N56 X1.902 (PASS 8)
N57 X1.8966 (PASS 9)
N58 G00 X12.0 Z4.5 T0500 M09
N59 M30
%
```

Note that the *I* distance of taper inclination is the difference between the *end* diameter of 1.8966 and the *start* diameter of 1.1216, divided by 2. The result is:

$$(1.8966 - 1.1216) / 2 = 0.3875$$

This *I* value (0.3875) needs a directional sign, to indicate the taper direction (its inclination from end point). The *I* value in the example is negative because the taper start diameter is *below* the thread end diameter, as viewed on a typical rear lathe. In the program, the *I* entry will be I-0.3875.

### ◆ Tapered Thread and G76 Cycle

Both one-block and two-block multiple repetitive threading cycle G76 requires the taper inclination amount *not to be a zero*, if a tapered thread is machined. The *I* (one block method) or *R* (two block method - second line) value in the cycle specifies the difference *per side*, so called *radial distance*. It can be a positive or a negative value, indicating the direction between end and start diameters of the taper.

Remember that the X-diameter is always programmed at the *end* of thread and the *I* amount supplies the actual taper height as well as its inclination (taper ratio per side). On CNC lathes with the X+ axis direction upwards from the center line (rear lathes), an increasing taper diameter will require a negative *I* amount, and a decreasing taper will require a positive *I* amount. The programmed *I* amount is always a *single* value, measured on radius, not diameter - Figure 38-35 illustrates the concept for rear CNC lathes.

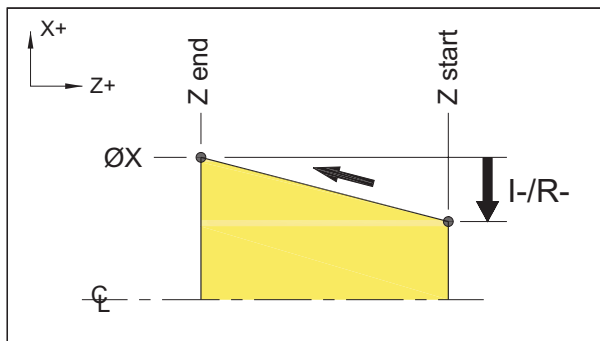


Figure 38-35  
Incline direction of an EXTERNAL tapered thread in G76 cycle

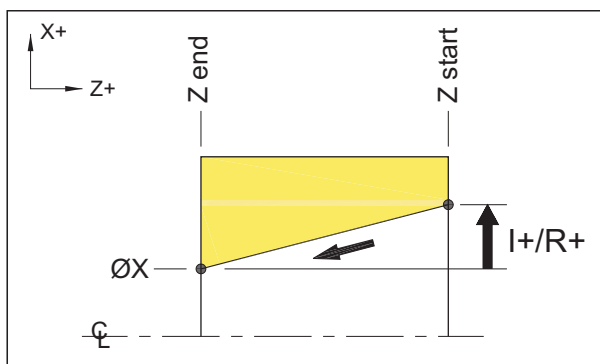


Figure 38-36  
Incline direction of an INTERNAL tapered thread in G76 cycle

Basic G76 cycle will be maintained but the *I* or *R* address will be added - a non-zero value must be programmed:

```
O3811 (G76 ONE BLOCK - TAPERED THREAD)
...
N46 T0500 M42
N47 G97 S450 M03
N48 G00 X2.5 Z0.4 T0505 M08
N49 G76 X1.8966 Z-2.7 I-0.3875 K0.0767 D0140
    F0.125
N50 G00 X12.0 Z4.5 T0500 M09
N51 M30
%
```

```
O3812 (G76 TWO BLOCKS - TAPERED THREAD)
...
N46 T0500 M42
N47 G97 S450 M03
N48 G00 X2.5 Z0.4 T0505 M08
N49 G76 P011060 Q004 R0.002
N50 G76 X1.8966 Z-2.7 R-0.3875 P0767 Q0140
    F0.125
N51 G00 X12.0 Z4.5 T0500 M09
N52 M30
%
```

If this method can be used for threading, G76 cycle is the best choice. It offers the fastest program generation as well as the best opportunities for on-machine editing.

## MULTISTART THREAD

Most threads have only one start, suitable for most applications. The most common purpose of a multistart thread is to transfer a *precision* motion very rapidly over a relatively long distance. Note the word *precision* - a coarse thread can be used to transfer motion rapidly, but with very little precision. Fine thread has the precision but transfers any motion slowly. An example of precision multistart threads are internal designs of some camera zoom lenses.

For programmers, there are some unique considerations for multistart threads. It is important that the start position for each thread is in such a location, that when viewed from the thread end, each start on the circumference will be divided in equal angular increments. Also important is to maintain equal thread profile when viewed from the thread cross section. To achieve these conditions, two programming tools are available:

- Controlled thread start position
- Controlled feedrate

Figure 38-37 shows symbolically views of the thread cross sections and end views.

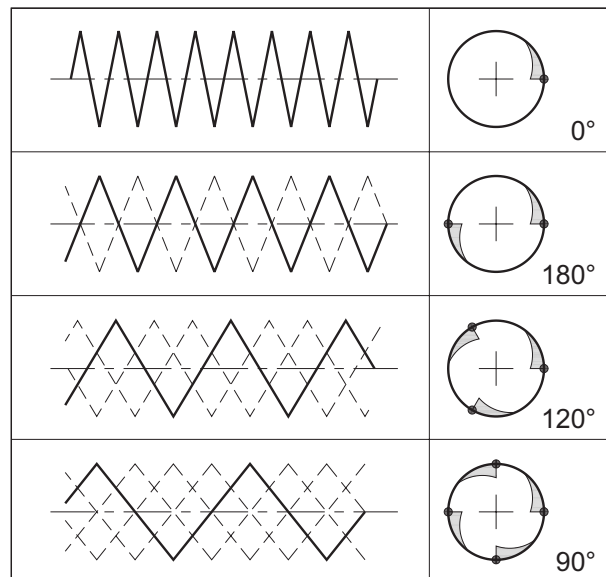


Figure 38-37  
Schematic representation of 1, 2, 3 and 4 start threads (dot = start)

In the illustration are four examples of the cross sections (left) and the end views (right) of a single start thread (top), double start (one below), triple start (two below) and a quadruple start (three below).

Even when the examples are represented only symbolically, specified thread *pitch* is maintained in all instances. Also note the *equal* distribution of each thread start.

Equal distribution is the key for successful multi-start threading, when the threaded part is viewed along its center line. Correct spacing is achieved by a programmed *shift amount* from one thread start to the next.

### ◆ Threading Feedrate Calculation

Threading feedrate is always the *lead* of thread, never the *pitch*. For a single start thread, lead and pitch have the same value - for a multistart thread, they do not. For example, take a single start thread of 16 TPI. Here, the lead and pitch are both 0.0625, so the feedrate is F0.0625. If the drawing specifies the thread as 16 TPI, but indicates a *double start*, (for example 3.0-16 TPI 2-START), that means the *pitch* of thread will remain unchanged (0.0625), but its *lead* will double to 0.1250. Therefore, programmed feedrate for double start thread with pitch of 0.0625 will be F0.125. The pitch multiplication will always depend on the *number of thread starts*. That means a triple start thread will have the feedrate three times the pitch, quadruple start thread four times, and so on.

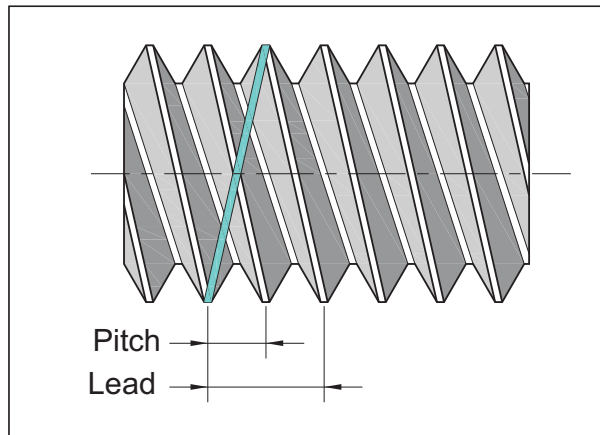


Figure 38-38

PITCH and LEAD relationship for a two start thread

In Figure 38-38, the relationship of pitch and lead of a double start thread is shown. The same logic that applies to a double start thread, also applies to triple, quadruple, etc., threads. Feedrate calculation is identical for *all* threads:

$$\text{Feedrate} = \frac{\text{Number of starts}}{\text{TPI}}$$

Figure 38-39 shows the relationships of pitch and lead for some more common multistart threads - the same pitch-lead relationship is maintained proportionately. Metric threads use only pitch, so the above formula can be changed to use pitch instead of threads per inch (metric *and* imperial):

$$\text{Feedrate} = \text{Number of starts} \times \text{Pitch}$$

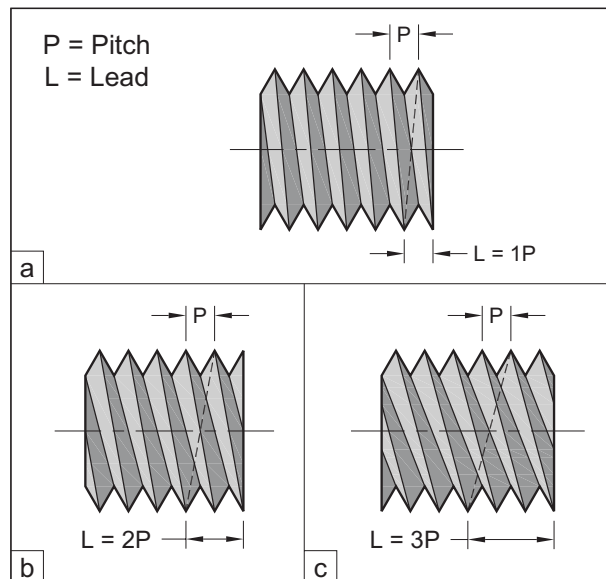


Figure 38-39

Multistart thread - relationship between pitch and lead:

- |                         |                         |
|-------------------------|-------------------------|
| (a) Single start thread | Lead = Pitch = $1P = P$ |
| (b) Double start thread | Lead = $2P$             |
| (c) Triple start thread | Lead = $3P$             |

### ◆ Shift Amount

Feedrate is not the only consideration for programming a thread with two or more starts. The other - equally important factor - is the programmed amount of *tool point shift*. This shift guarantees that each start will be in synchronized relationship to all other starts. When one thread is finished, the tool start position has to be *shifted* in Z-axis only and always by the *pitch* distance. Formula for the tool shift amount is always equal to the pitch:

$$\text{Shift amount} = \text{Pitch}$$

Shift has to be programmed *for each start after the first one* - the number of shifts is *one less than number of starts*:

$$\text{Number of shifts} = \text{Number of starts} - 1$$

Note - this formula is valid even for a single start thread, where no shift is required ( $1 \text{ start} - 1 = 0 \text{ shifts}$ ).

A few methods can determine *when* the tool shift is to be programmed. The first method, for a double start thread, is to program one thread to its full depth, then shift out and cut the second thread to its full depth. Another method, for the same thread, is to cut one pass of the first thread, shift *in*, cut the same pass for the second thread, shift *out*, cut the second pass for the first thread, shift *out* again and repeat the process until both threads are completed to the full depth. This approach applies to any number of starts.

Obvious advantage to the first method is programming ease. On negative side, if the tool cutting edge wears out on the first thread, second thread will not be as accurate. The advantage of the second method is that tool wear will be equally distributed over both threads, although programming will require a lot more effort, which presents the negative side. Additional problem is that in many hard materials, the thread edge life may suffer from extensive material removal.

### ◆ Application Example

To illustrate a sample multistart thread application, previous thread specifications are based on program O3804:

- Number of threads per inch is twelve (12 TPI)
- Number of starts is TWO (double start thread)
- Thread is cut as external at 3.000 nominal diameter
- Calculated thread depth is 0.0511 (0.61343 / 12)

Although the block-by-block programming method G32 can be used for special applications, acceptable results can be achieved in many threading applications by using the G92 or G76 cycles, with less programming, as well as the gain of easier editing at the machine. G76 two-block version is shown here, as it covers most modern controls. Although two versions will be presented, neither one may produced the best results in certain materials - cycles are always bit of a compromise between power and programming convenience.

The first program, O3813, shows the first thread fully completed *before* the second thread starts (cutting feedrate is F0.1667, *not* F0.0833):

```
O3813 (G76 - EACH START TO FULL DEPTH)
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.5 T0505 M08 (START POSITION)
N62 G76 P011060 Q004 R0.002
N63 G76 X2.8978 Z-1.6 R0 P0511 Q0210 F0.1667
N64 G00 X3.3 Z0.5833 (SHIFTED START POSITION)
N65 G76 P011060 Q004 R0.002
N66 G76 X2.8978 Z-1.6 R0 P0511 Q0210
N67 G00 X12.0 Z4.5 T0500 M09
N68 M30
%
```

Three major features of the above program should be mentioned. First, the original start point has been increased from Z0.25 to Z0.5, to allow feedrate acceleration in the air (block N61). Second, in blocks N62 and N63, the G76 cycle is used as in program O3804, with a major exception - the modal feedrate is *doubled* for a *double start* thread. Third, added block N64 - this block changes the start position for the *second* thread start. As the original start position was Z0.5, the new position must be increased by the pitch to Z0.5833. W0.0833 can be used instead of Z.

### ◆ Quality of Multistart Threads

One of the potential problems using G76 cycle for multiple threads is the *tool wear*. For soft materials, short threads and a few starts, the previous example is quite suitable to produce quality threads. Long multistart threads, threads with many starts, threads machined in hard materials, etc., present a situation where the threading tool may significantly wear out *before* all threads are machined to their full depth. G76 cycle itself does not provide for a situation such as this one, but a resourceful programmer should be able to find a suitable workaround.

The easiest method, is to use *two* identical tools - one for thread *roughing* and one for thread *finishing*. In program O3814, the main difference is the *shortened cutting depth*.

```
O3814 (G76 - ROUGH AND FINISH - TWO TOOLS)
...
N59 T0500 M42 (T05 = ROUGHING TOOL)
N60 G97 S450 M03
N61 G00 X3.3 Z0.5 T0505 M08 (START POSITION)
N62 G76 P011060 Q004 R0.002
N63 G76 X2.9098 Z-1.6 R0 P0451 Q0210 F0.1667
N64 G00 X3.3 Z0.5833 (SHIFTED START POSITION)
N65 G76 P011060 Q004 R0.002
N66 G76 X2.9098 Z-1.6 R0 P0451 Q0210
N67 G00 X12.0 Z4.5 T0500 M09
N68 M01

N69 T0700 M42 (T07 = FINISHING TOOL)
N70 G97 S450 M03
N71 G00 X3.3 Z0.5 T0707 M08 (START POSITION)
N72 G76 P011060 Q004 R0.002
N73 G76 X2.8978 Z-1.6 R0 P0511 Q0450 F0.1667
N74 G00 X3.3 Z0.5833 (SHIFTED START POSITION)
N75 G76 P011060 Q004 R0.002
N76 G76 X2.8978 Z-1.6 R0 P0511 Q0450
N77 G00 X12.0 Z4.5 T0700 M09
N78 M30
%
```

Roughing tool left 0.006 stock per side for finishing that is reflected in blocks N63 and N66 (underlined). Finishing tool uses the same program as before, but its Q-depth was increased to minimize air-cutting - most of the material has been removed with the roughing tool. **Note that all other data are identical for both tools!**

Although this method is the easiest to program, it requires much more effort during setup at the machine. Both tools have to be the same type, they have to be set very accurately, so they reach each start thread at the same location. Also, geometry and wear offsets have to be set correctly for both tools and maintained properly - if one wear offset for one tool is changed, it may often be necessary to change the other as well.

When applying this technique, tool wear will be evenly distributed between all threads and should result in a longer tool life and high quality multistart threads.



As an alternate method that also incorporates roughing and finishing cuts is a program that uses only *a single* tool, rather than two tools. This method is shown in the next program - O3815:

```
O3815 (G76 - ROUGH AND FINISH - ONE TOOL)
...
N59 T0500 M42
N60 G97 S450 M03
N61 G00 X3.3 Z0.5 T0505 M08 (START POSITION)
N62 G76 P011060 Q004 R0.002
N63 G76 X2.9098 Z-1.6 R0 P0451 Q0210 F0.1667
N64 G00 X3.3 Z0.5833 (SHIFTED START POSITION)
N65 G76 P011060 Q004 R0.002
N66 G76 X2.9098 Z-1.6 R0 P0451 Q0210
N67 G00 X3.3 Z0.5 (START POSITION)
N68 G76 P011060 Q004 R0.002
N69 G76 X2.8978 Z-1.6 R0 P0511 Q0450 F0.1667
N70 G00 X3.3 Z0.5833 (SHIFTED START POSITION)
N71 G76 P011060 Q004 R0.002
N72 G76 X2.8978 Z-1.6 R0 P0511 Q0450
N73 G00 X12.0 Z4.5 T0500 M09
N74 M30
%
```

Although this method makes setup much more efficient, it is not recommended for all multistart threading applications, as it does not fully solve issues associated with tool wear. For short job runs, it may work quite well and be a reasonable compromise.

This section has presented some special options that can be useful in certain thread cutting situations. Interpret all presented ideas as - *just ideas*. Threading jobs vary greatly and there is not a single solution to all threading problems.

## THREAD RECUTTING

When thread cutting is done, it should be checked for quality *before* the part is removed from the machine. Once the part is removed, any subsequent reclamping will need a great effort in order to recut the thread. While the part is clamped, the first threading pass will start at a random place of the cylinder circumference. Each subsequent pass will be automatically synchronized to start at the *same* position. As long as the threaded part remains clamped, this synchronization is assured.

### ◆ Before Part Removal

Prevention is always worth the extra effort necessary. There is one or two methods available to prevent thread recutting - just follow one important rule:

Do not unclamp part until the thread is checked

When threading operation is completed, check the thread *at the machine*, while the part is still clamped. This inspection can be done simply enough, if suitable thread gages are available. If a matching part is available, it may provide another checking method, although not one recommended by quality inspectors. Otherwise, you may have to go as far as using the rather awkward but reliable three-wire method.

### ◆ After Part Removal

Once the threaded part is removed from the holding device, its thread should be correct. In case it is not - and in spite of all prevention methods - what can be done? This subject is aimed mainly towards CNC lathe operators, but programmers can learn from it as well.

First, make sure to program a tool wear offset. Second, program M00 function at the end of each threading operation, before any other machining, even for the last tool. If the thread has to be recut after removal - while the part is still clamped, the CNC operator has to follow several steps:

1. Reclamp the threaded part to run concentric w/spindle
2. Set the X-axis offset large enough, so the threading tool moves above the thread (external threading) or below the thread (internal threading)
3. Visually align the threading tool tip with the thread already completed (only as accurate as one's eye)
4. Repeat the steps in the air while carefully adjusting the offset so the tool will eventually recut the thread

Thread recutting should - and could - be always prevented. The difficulty to reset a part precisely back to its original position is the major problem facing the final machining quality.

The length of a CNC program is usually measured in the number of characters such program contains. This number is similar to the number of bytes, if the program is stored on a computer disk. Physical length of a program is usually not an issue for most jobs. Program length will vary, depending on the complexity of work, number of tools used, method of programming and other factors. Generally, the shorter the program, the less time is needed to write it, and the less space it will occupy in CNC memory. Short programs also reduce the possibility of a human error, because they are easily checked, modified and optimized. Virtually all CNC systems offer features designed to shorten the length of a program to some extent and make the programming process easier, more efficient and less prone to errors. Typical examples of this type of programming are fixed cycles, multiple repetitive cycles and custom macros. This chapter describes the structure, development and applications of another method of efficient program preparation - the use of *subprograms*.

## MAIN PROGRAM AND SUBPROGRAMS

A CNC program is a series of instructions, assigned to different tools and operations. If such a program includes two or more *repetitive* instructions, its structure should be changed from a single long program to two or more separate programs. Each repetitive instruction is written only once and called when required. This is the main concept of subprograms. *Figure 39-1* shows a typical part layout repeated at different locations.

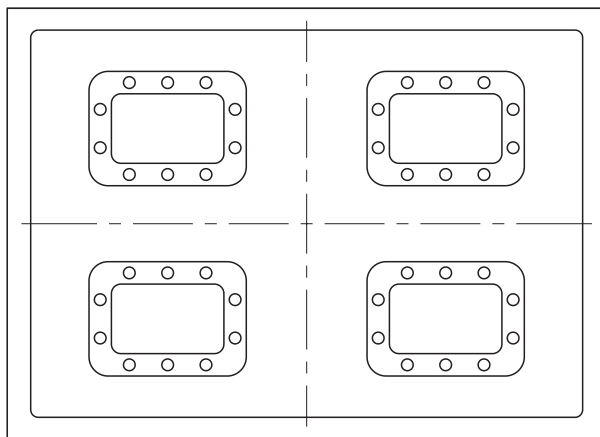


Figure 39-1

Example of a part requirement suitable to be used as a subprogram

Each program must have its own program number and is stored in the control memory. Programmer uses special M-function to call one program from another. The *first* program that calls another program is called the *main program*, all other programs are called *subprograms*. Main program is never called by a subprogram - it becomes the top level of all programs. Subprograms can also be called from other subprograms, up to a certain number of nesting levels. When a program containing subprograms is used, always select the main program, never the subprogram. The only time a subprogram is selected at the control is for editing purposes. In some reference materials, subprograms are also called *subroutines* or *macros*, but the term *subprogram* is used most often and the word *macro* could have a different meaning altogether.

### ◆ Subprogram Benefits

Any frequently programmed order of instructions or unchanging block sequences can benefit from becoming a subprogram. Typical applications for subprogram applications in CNC programming are:

- Repetitive machining motions
- Functions relating to tool change
- Hole patterns
- Grooves and threads
- Machine warm-up routines
- Pallet changing
- Special functions                      ... and others

Structurally, subprograms are similar to standard programs. They use the same syntax rules and look and feel the same. Often, it may not be easy to see the difference between a regular program and a subprogram at a casual glance. A subprogram can use absolute or incremental data input, as necessary. Subprograms are loaded into the CNC system memory just like other programs. When properly implemented, they offer several benefits:

- Program length reduction
- Program error reduction
- Programming time and effort reduction
- Quick and easy modifications

Not every subprogram will provide all benefits, but even one benefit should be a good reason to use subprograms.

### ◆ Identification of Subprograms

The first step towards a successful application of subprograms is the *identification and isolation of repetitive* programming sequences. For example, the next six program blocks represent a machine zero return for a typical horizontal machining center, at the start of program:

```
N1 G20
N2 G17 G40 G80           (STATUS BLOCK)
N3 G91 G28 Z0            (Z-AXIS RETURN)
N4 G28 X0 Y0             (X AND Y AXES RETURN)
N5 G28 B0                (B-AXIS RETURN)
N6 G90                   (ABSOLUTE MODE)
N7 ...
```

These blocks represent a typical sequence of commands that will be repeated *every time* a new program for that machine is written. Such a program may be written many times a week, each time repeating the same sequence of instructions. To eliminate any possibility of an error, frequently used order of blocks can be stored as a separate program and identified by a unique program number. Then, it can be called up at the top of any main program. This stored programming sequence will become a *subprogram* - a branch, or an extension, of the main program.

## SUBPROGRAM FUNCTIONS

A subprogram must be recognized by the control system as a *unique* type of program, *not as a main program*. This distinction is accomplished with two miscellaneous functions, normally applicable to subprograms only:

M98	Subprogram call function
M99	Subprogram end function

Subprogram *call function* M98 must always be followed by the subprogram number P--. Subprogram *end function* M99 terminates the subprogram and transfers processing back to program it originated from (a main program or a subprogram). Although M99 is mostly used to end a subprogram, it may also be rarely used in the main program, replacing the M30 function. In this case, the program will run 'forever', or until the *Reset* key is pressed.

### ◆ Subprogram Call Function

Function M98 calls up a previously stored subprogram from another program. If used only by itself in a block, it will result in an error. M98 is an incomplete function - it requires two *additional* addresses to become effective:

- Address P identifies the selected subprogram number
- Address L or K identifies the number of subprogram repetitions (L1 or K1 is the default)

For example, a typical subprogram call block includes the M98 function and the subprogram number:

**N167 M98 P3951**

In block N167, subprogram O3951 is called from CNC memory, to be repeated *once* - L1 (K1) counter is the default, depending on the control. Subprogram must be stored in the control before being called by another program.

M98 blocks that call subprograms may also include additional instructions, such as rapid tool motions, spindle speed, feedrate, cutter radius offset number, etc. On most controls, if included in the same block as subprogram call, the additional data will be passed to the contents of the subprogram. The following subprogram call block also contains a tool motion in two axes:

**N460 G00 X28.373 Y13.4193 M98 P3951**

This block executes the rapid motion first, *then* it calls the subprogram. The order of words in a block makes no difference to the block execution:

**N460 M98 P3951 G00 X28.373 Y13.4193**

results in the same machining order as if the tool motion preceded the subprogram call, but looks illogical.

### ◆ Subprogram End Function

When the main program and subprogram coexist in the control, they must differ by their program numbers. During processing, they will be treated as one continuous program, so a distinction must be made for the program end function as well. The end of program function is M30 or, less frequently, M02. Subprogram must be terminated by a different function. Fanuc uses M99 for that purpose:

```
O3951 (SUB-1)      Subprogram start
...
M99                Subprogram end
%
```

When a subprogram terminates, control returns program processing to the program of origin - it will *not* terminate the main program - that is the exclusive function of M30. Additional parameters may also be added to the M99 subprogram end, for example a block skip code [/], a block number to return to upon exit, etc. Note that the stop code symbol (the % sign) is used in the same manner for a subprogram, as for any main program. Subprogram termination is important and must always be done right. It sends two very important instructions to the control system:

- To terminate the subprogram
- To return to the block following subprogram call

Never use the program end function M30 (M02) to terminate a subprogram - it will *immediately cancel all program processing* and reset the control. Program end function does not allow program execution of any blocks beyond the block that contains it.

Normally, subprogram end M99 returns the processing to the block *immediately following* subprogram call M98. This concept is illustrated in Figure 39-2 (without block numbers) and described next.

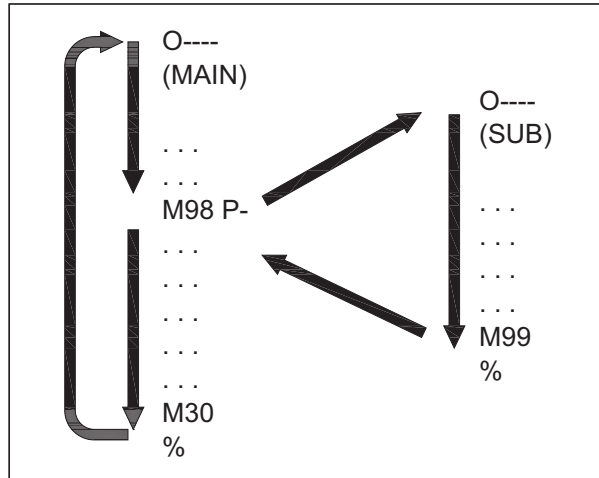


Figure 39-2

Flow of a program processing with a single subprogram

### ◆ Block Number to Return to

In most programs, M99 function is programmed by itself, as a stand alone entry, and as the *last* instruction in the subprogram. Usually, there are no other commands included with it in the same block. M99 function causes the subprogram to terminate and transfers its execution to the *next block* of the program it originated from. For example,

```
N67 M98 P3952          (SUBPROGRAM CALL)
N68 ...                (BLOCK TO RETURN TO FROM O3952)
N69 ...
N70 ...
```

executes block N67 by calling subprogram O3952. When subprogram O3952 is processed, control returns to the original program and continues processing instructions from the block N68, which is the *block to return to*.

### Special Applications

For some special applications, such as in barfeeding, it may be necessary to specify a *different* block number to return to, rather than using the *next block* default. If the programmer finds this option useful for some jobs and uses this technique, the P-address must be included with M99:

```
M99 P. .
```

In this format, the P-address represents the block number to return to - *from* the completed subprogram. This block number must be present in the program of origin. For example, if the main program contains these blocks,

```
(MAIN - PROGRAM)
...
N67 M98 P3952
N68 ...
N69 ...
N70 ...
```

and the subprogram O3952 is terminated by

```
O3952 (SUB)
...
...
M99 P70
%
```

the calling program processing will continue from block N70 (the main program in the example), *bypassing* blocks N68 and N69. This kind of application is not very common - it requires suitable type of work, in addition to the thorough understanding of sub-programming principles.

Address P has a different meaning  
when used with M98 and M99 functions

Daily applications of this powerful programming method are not common, but the feature is an item to be explored by inquiring programmers. Various associated applications will include other programming tools, such as a combination with a block skip function, using the slash code /.

### ◆ Number of Subprogram Repetitions

A very important subprogram call feature is the address L or K, depending on the control model. This address specifies the number of subprogram repetitions - *how many times* the subprogram has to be repeated before processing resumes in the original program. In most programs, a subprogram will be called only once, then the original program will continue.

Programs that require a *multiple* subprogram repetition before proceeding with the rest of the original program are common. For comparison, a single use of the subprogram O3952 could be called up from the program of origin as:

```
N167 M98 P3952 L1 (K1)
```

This is a correct program block, but the L1/K1 counter does not have to be programmed at all. It can be safely ignored - control unit always defaults to only *one* repetition.

If no address L/K is specified, the default value is always L1/K1

N167 M98 P3952 L1 (K1) is identical to N167 M98 P3952

**Note - In the following examples, substitute K for every L listed, if required by the control system**

Number of repetitions for some control models range between L0 and L9999 and the L address other than L1 must always be programmed. Some programmers write the full block, even for a single repetition, rather than counting on the default conditions of the control system. The choice is a personal preference.

#### Repetition Count Variation

Some Fanuc controls *do not* accept the L/K address as the number of repetitions and use a different format. On these controls, a single subprogram call is the same:

**N342 M98 P3952**

This block calls the subprogram only once, as no special request has been used. In order to repeat the subprogram call four times, instead of programming

**N342 M98 P3952 L4 (K4)**

program the requested number of repeats *directly after* the P address, in a single statement:

**N342 M98 P43952** *is the same as* **N342 P00043952**

The result is identical to the other version - subprogram will be repeated four times. *The first four digits are reserved for the number of repeats, the last four digits define the subprogram number.* For example,

**M98 P3950** *is the same as* **M98 00013950**

assumes a single repetition of subprogram O3950. In order to repeat O0050 subprogram 39 times, program

**M98 P390050** *or* **M98 P00390050**

The maximum number of repetitions does not change for the 0/16/18/20/21 controls - it is represented by the first four digits, to the maximum of 9999.

**M98 P99993952**

repeats the execution of subprogram O3952, nine thousand, nine hundred and ninety nine times, the maximum number of repetitions available (some old models may have the maximum of only 999 times).

#### ◆ L0/K0 in a Subprogram Call

There is no mystery in using the L/K counter greater than one to repeat a subprogram. This is a common application. Fanuc also offers a *zero* number of repetitions, in the form of L0/K0. When can the L0/K0 be programmed? Would anybody want to repeat a subprogram *zero times*?

There are some good reasons. Observe *Figure 39-3*. The five hole pattern has to be spot drilled, drilled and tapped.

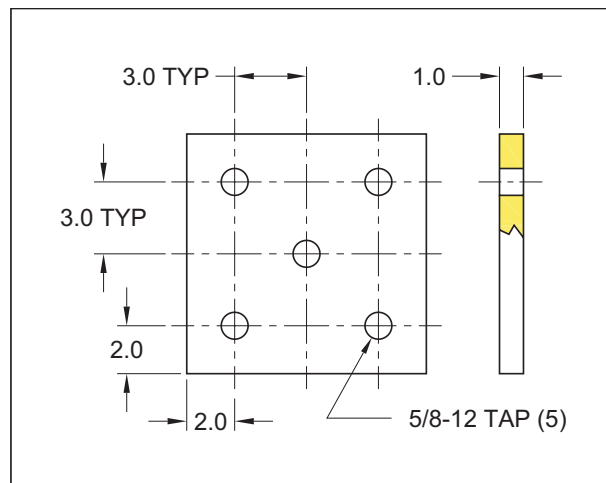


Figure 39-3

Sample drawing used for subprogram development  
Used in programs O3901, O3902 and O3953

For the spot drill ( $\varnothing 0.750$ ), G82 cycle is used with 0.2 seconds dwell to Z-0.3275 depth. For tap drill, G81 cycle is used, and for tapping 5/8-12 tap, G84 cycle is used. Spot drill prepares the hole for drilling and makes a 0.015 chamfer. Tap drill will be 35/64 drill ( $\varnothing 0.5469$ ), to open up the hole for 5/8-12 tap:

**O3901**

(TOOL 1 - 90-DEG SPOT DRILL -  $\frac{3}{4}$  DIA)

**N1 G20**

**N2 G17 G40 G80 T01**

**N3 M06**

**N4 G90 G00 G54 X2.0 Y2.0 S900 M03 T02**

**N5 G43 H01 Z1.0 M08**

**N6 G99 G82 R0.1 Z-0.3275 P200 F3.0** (LL HOLE)

**N7 X8.0** (LR HOLE)

**N8 Y8.0** (UR HOLE)

**N9 X2.0** (UL HOLE)

**N10 X5.0 Y5.0** (MIDDLE HOLE)

**N11 G80 Z1.0 M09**

**N12 G28 Z1.0 M05**

**N13 M01**

(TOOL 2 - 35/64 DRILL)

**N14 T02**

**N15 M06**

**N16 G90 G00 G54 X2.0 Y2.0 S840 M03 T03**

**N17 G43 H02 Z1.0 M08**

**N18 G99 G81 R0.1 Z-1.214 F11.0**

**N19 X8.0**

**N20 Y8.0**

**N21 X2.0**

**N22 X5.0 Y5.0**

**N23 G80 Z1.0 M09**

**N24 G28 Z1.0 M05**

**N25 M01**



```
(TOOL 3 - 5/8-12 TAP)
N26 T03
N27 M06
N28 G90 G00 G54 X2.0 Y2.0 S500 M03 T01
N29 G43 H03 Z1.0 M08
N30 G99 G84 R0.4 Z-1.4 F41.0
N31 X8.0
N32 Y8.0
N33 X2.0
N34 X5.0 Y5.0
N35 G80 Z1.0 M09
N36 G28 Z1.0 M05
N37 G28 X5.0 Y5.0
N38 M30
%
```

This type of program uses repeating XY coordinates for each tool (spot drilling, drilling, tapping). In order to make this program more effective, all *repeating* program blocks will be collected into a subprogram and used much more efficiently. Here is a pattern of holes separated from the long program that also includes G80Z1.0M09, as the standard end of *any* active fixed cycle (G20 mode):

```
X2.0 Y2.0
X8.0
Y8.0
X2.0
X5.0 Y5.0
G80 Z1.0 M09
```

Only a small effort is needed to reformat the existing program and separate it into a main program and a subprogram that stores the repeating machining pattern. Isolated XY coordinates of all five holes in the pattern are included:

```
O3953 (SUBPROGRAM)
(FIVE HOLE PATTERN)
N1 X2.0 Y2.0
N2 X8.0
N3 Y8.0
N4 X2.0
N5 X5.0 Y5.0
N6 G80 Z1.0 M09
N7 M99
%
```

This subprogram can be called from the main program, in this example, from a new program O3902. Programming L0 prevents double cutting of the first hole:

```
O3902 (MAIN PROGRAM)
(TOOL 1 - 90-DEG SPOT DRILL - 3/4 DIA)
N1 G20
N2 G17 G40 G80 T01
N3 M06
N4 G90 G00 G54 X2.0 Y2.0 S900 M03 T02
N5 G43 H01 Z1.0 M08
N6 G99 G82 R0.1 Z-0.3275 F200 F3.0 L0
N7 M98 P3953
N8 G28 Z1.0 M05
N9 M01
```

```
(TOOL 2 - 35/64 DRILL)
N10 M06
N11 T02
N12 G90 G00 G54 X2.0 Y2.0 S840 M03 T03
N13 G43 H02 Z1.0 M08
N14 G99 G81 R0.1 Z-1.214 F11.0 L0
N15 M98 P3953
N16 G28 Z1.0 M05
N17 M01
```

```
(TOOL 3 - 5/8-12 TAP)
N18 M06
N19 T03
N20 G90 G00 G54 X2.0 Y2.0 S500 M03 T01
N21 G43 H03 Z1.0 M08
N22 G99 G84 R0.4 Z-1.4 F41.0 L0
N23 M98 P3953
N24 G28 Z1.0 M05
N25 G28 X5.0 Y5.0
N26 M30
%
```

In the program, initial XY motion for each cutting tool will position the cutter at the *first* hole of the machining pattern. All fixed cycles used in the program will start at the first hole of the pattern. Since the first hole definition is included in the subprogram, *as well as in the main program*, programming L0 in the fixed cycle call is mandatory, else the first hole of the pattern will be machined twice. This is a classic application of L0/K0 as it relates to fixed cycles, but not to subprograms. Subprogram O3953 may also include standard machine zero return block G28Z1.0M05, as it repeats after each M98 call in the main program O3902. This practice is correct but *not* recommended, as it does not follow a clearly defined program structure.

## SUBPROGRAM NUMBERING

To keep track of subprograms is much more important than keeping track of regular programs. Always make sure to know *exactly* what subprograms are available and how they are used, what is their purpose. A single subprogram may be used in many other programs and proper subprogram identification technique is extremely important.

Control unit directory of programs does not distinguish between program numbers and subprogram numbers. The control system recognizes a subprogram call only by its programmed format, the miscellaneous function M98, followed by the P.. subprogram number statement.

All this means that subprogram number is assigned at the *programming* level, not at the *machine operation* level. It is the programmer's responsibility, *not* the CNC operator's, to assign subprogram numbers. Programmer has a great flexibility in organizing all subprograms and their identification - in fact, any programmer can design and set up certain basic rules and related standards. Many of the rules governing format of main programs also apply to subprograms. Remember these four main points:

- If used at all, program number is commonly specified by the letter O, followed by four or five digits, depending on the control system
- If at all, program number can be specified by the colon symbol, commonly : for the ISO format, followed by up to four or five digits, depending on the control system setting
- Main program number - O or : - cannot be negative or equal to zero
- Subprogram number cannot be negative or equal to zero

Within the allowed range, *any* number can be assigned to *any* main program or subprogram. Some programmers do not use program numbers at all. This approach is acceptable for most controls, but only if the application does not require subprograms. In most cases, main program numbers can be assigned by the machine operator. On the other hand, to maintain *control* of subprograms, program numbers become very important. The first step is to get organized. This is even more important if subprograms are designed to be called up by *many other programs at different times*. There is not one *best* method, but some proven suggestions offer an idea how to approach the subject of program numbering and develop a personal approach.

For example, in this handbook, all main programs are numbered consecutively, with the first two digits corresponding to the chapter number. In this chapter, the method also applies to subprograms, but the last two digits are arbitrarily increased by fifty, for example O3953 will be the third subprogram example in the chapter. Feel free to adapt this method to any reasonable format.

### ◆ Organized Approach

Suggested programming approach is based on the understanding that CNC memory is *not* used as a storage media for all part programs ever made. Control system memory capacity is *always* limited. At one point, this limit will be reached and there will be no more space left to accommodate more programs. Good program organization is one that uses CNC system memory storage only for the *current program*, perhaps a few more that are to be used soon.

If a unique program number is assigned by the *machine tool operator* during setup, this situation needs some good management as well. On some controls, main program number on the written copy will not always load automatically, so it is not really required. That means if an arrangement is made with the shop supervisor that the CNC operator stores all main programs using only *three* digits for regular program numbers 1-999; then there will be four digit numbers 1000-9999 available for subprograms. This available range is more than enough for most manufacturing applications. Such an approach presents a good management over those subprograms whose numbers selected. All four-digit subprogram numbers can be docu-

mented, logged and subsequently called from any program, main or another subprogram, without a fear of duplication or a part program number mismatch.

Subprograms should always be documented in some log book, complete with detailed descriptions, independently from all programs of origins. This way, such subprograms can be used when needed, often at a short notice, regardless of the program for which they have been originally written. Such a method allows to organize all subprograms by their series number (*i.e.*, 1000, 2000, 3000, etc., or 1100, 1200, 1300, etc.), for either the type of CNC machine, the type of subprogram, or the type of machining operation.

Individual subprograms have to have assigned program numbers that are unique. Program number assigned to a subprogram is called together with the M98 function and P address. Such a combination of the two words, M98 P., is the *minimum requirement* for a subprogram call from another program.

Using the example (early in this chapter) of the machine zero return sequence for a four axis vertical machining center, a subprogram can be created (with an assigned number O3954) for the blocks representing all needed commands - units selection G20 or G21 is *not* included:

```
O3954 (MACHINE ZERO RETURN)
N101 G17 G40 G49 G80
N102 G91 G28 Z0
N103 G28 X0 Y0
N104 G28 B0
N105 G90
N106 M99
%
```

Units selection should always be used in the main program, for flexibility. Once the machine zero return subprogram has been designed and stored in the memory, every main program can start by calling subprogram O3954:

```
O3903 (MAIN PROGRAM)
(PART ABC-123)
N1 G20
N2 M98 P3954
N3 G90 G54 G00 X. . Y. .
N4 ...
...
<... Machining ... >
...
...
N45 M30
%
```

*Units used for this program*  
*Subprogram O3954 call*  
*Normal program start*

*Main program end*

To visualize processing of the two programs by the CNC system, follow all operational steps in the order of program execution. During program O3903 execution, the control system will follow the following order of operations (instructions):

1. Set program number 03903 as the current program number
2. Display comment on the display screen
3. Set the units of measurement (inches in the example)
4. Branch out to the top of subprogram 03954
5. Execute all blocks in the subprogram 03954
6. When M99 is processed, the subprogram ends and returns to the main program
7. The main program is processed, beginning with the block N3
8. When M30 is processed, the main program ends and returns to the beginning
9. When the CYCLE START switch is activated, steps 1 to 8 are repeated

As the example shows, main program uses increments of 1, subprogram also uses increments of 1, but starting with N101 block number. There are two reasons for it. The first reason is that a properly designed subprogram will not likely be a subject to any major changes - there should be no need to add any extra blocks into the subprogram once it has been debugged. The second reason is even more important. The lack of duplicated sequence numbers will be visible on the control display screen. Display of active block numbers will quickly inform the CNC operator whether main program or subprogram is being processed. Fanuc controls are very forgiving about block numbers and allow identification of block sequences freely, within a specified range.

To illustrate the described concept, here is an example. In a simple application, where a main program calls a single subprogram, there should be no problem in block numbering. Even if the sequence numbers are duplicated in both main program and subprogram, it is not likely there will be any confusion. On the other hand, when several subprograms are called from the same main program, duplicated block numbers appear during main program processing, as well as when subprograms are processed. Such a situation may confuse the CNC operator to the extent of losing track of what is really happening at the control system at any given time.

To avoid this problem, consider assigning *unique block numbers* to each subprogram, thus preventing a duplication. One method is to identify the subprogram numbers in the high thousands series, for example O6100, O6200, O6300, etc. Then, block numbering in a subprogram can be based on the subprogram number. For example:

```
O6100 (SUB 1)
N6101 ...
N6102 ...
N6103 ...      ... and so on
```

```
O6200 (SUB 2)
N6201 ...
N6202 ...
N6203 ...      ... and so on
```

This method works only with the maximum of one hundred blocks, suitable for many subprograms. CNC operator finds it easy to monitor a program with several subprograms. This is not a foolproof method for all programs, but the idea will work for most jobs and may be expanded.

### ◆ Protected Subprograms

Subprograms are special programs designed to be used frequently. Special subprograms may be even stored in the system memory permanently, to be called by all or many other programs. Any interference with these subprograms, accidental or intentional, can prove to be disastrous. If only a single subprogram is lost from the memory, it may halt literally hundreds of programs that depend on the use of this ill fated subprogram.

Fanuc controls address this potential problem by allowing an assignment of certain specified series of program numbers that can be locked by a system parameter setting. As a typical example, a section of program number series 9000 (within the range of O9000-O9999), will not show on the control display, when locked by a system parameter. Also, programs in this series cannot be edited or printed out, etc. If the locking parameter is not set, programs of the 9000 series behave normally, like any other program. In order to take advantage of this feature to protect some important programs from unauthorized editing or even viewing, consult Fanuc documentation for further details.

## SUBPROGRAM DEVELOPMENT

Before a subprogram can be developed, it must be well thought out and planned. Since the most common application for subprograms is repetitive pattern of machining, the programmer should have the ability to *recognize* a machining pattern suitable to be used as a subprogram.

### ◆ Repeating Pattern Recognition

This ability to recognize a repeating pattern is a matter of experience. The first indications come when writing a conventional program block by block. Visually scan the written copy first. If there are *repeating* clusters of consecutive blocks containing the *same* data, it is a very good reason to evaluate the program more carefully and possibly develop a subprogram.

An experienced programmer will not write a program the long way first - that is a waste of time. Programming experience enhances the ability to recognize a potential for subprograms at the early stages of program planning. However, for a programmer with limited experience, there is no

damage done by developing a long program first. It takes more time and lacks efficiency. However, this is how professional experience is developed and cultivated over time. With limited experience, be willing to rewrite a program from a single long format to a main program and one or more subprograms. Programmer should be able to identify those sections of long programs that can qualify as subprograms. Once such a series of repetitive data is identified in a conventional program, it is only a matter of small adjustments to separate these repetitive clusters and define them as true subprograms.

### ◆ Tool Motion and Subprograms

One of the most common sub-programming applications is a tool path machined at different locations of the part. For example, a ten hole rectangular pattern needs to be programmed for blind holes - *Figure 39-4*.

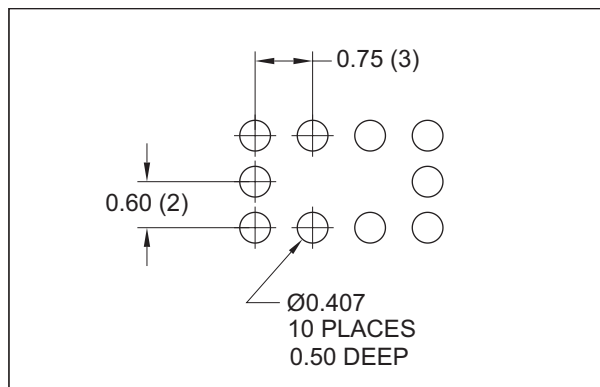


Figure 39-4

Detail of hole pattern used in program O3904

This hole pattern is repeated at four specified locations of the part, as illustrated in *Figure 39-5*.

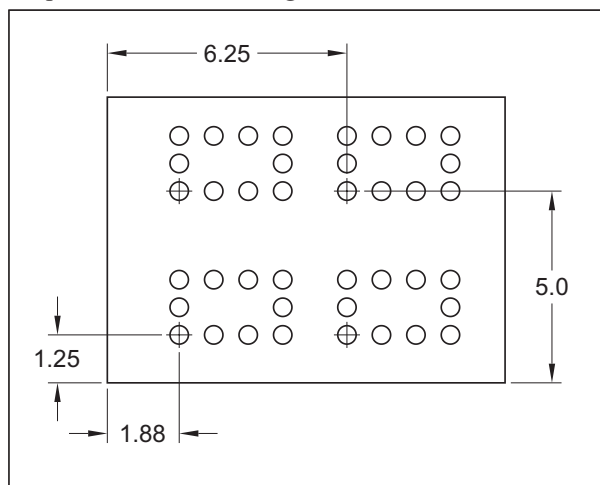


Figure 39-5

Hole pattern layout for program examples O3904 and O3905 (both using subprogram O3955)

Subprogram O3955 contains this pattern and uses L or K address to establish the number of fixed cycle repeats. In the first main program O3904, tool motion precedes the subprogram block. To start program development, concentrate on the hole pattern. First, select G91 incremental mode for the pattern. Then program X and Y incremental motions, starting from any hole, such as the lower left hand corner, and continue in one direction - *Figure 39-6*.

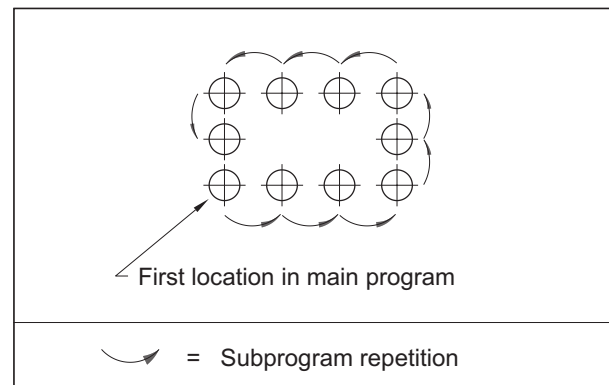


Figure 39-6

Subprogram O3955 processing flow

```
O3955 (SUBPROGRAM)
(FOUR-CORNER LOCATIONS)
N551 G91 X0.75 L3 (K3)
N552 Y0.6 L2 (K2)
N553 X-0.75 L3 (K3)
N554 Y-0.6
N555 M99
%
```

This subprogram is designed to machine *nine* holes in a rectangular pattern. The tenth hole - actually it is the first hole - is machined in a block with cycle call or rapid motion. The four pattern locations are not included in the subprogram - *they must be included in the main program*. Since the main program is using absolute mode G90, all individual pattern locations can be established:

```
O3904 (MAIN PROGRAM)
(FOUR-CORNER PATTERN - DRILLING BLIND HOLES)
N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X1.88 Y1.25
N4 G43 Z1.0 S350 M03 H01
N5 G99 G81 R0.1 Z-0.6221 F3.5 (LL HOLE 1)
N6 M98 P3955 (LL PATTERN)
N7 G90 X6.25 Y1.25 (LR HOLE 1)
N8 M98 P3955 (LR PATTERN)
N9 G90 X6.25 Y5.0 (UR HOLE 1)
N10 M98 P3955 (UR PATTERN)
N11 G90 X1.88 Y5.0 (UL HOLE 1)
N12 M98 P3955 (UL PATTERN)
N13 G80 G90 G28 Z1.0 M05
N14 G91 G28 X0 Y0
```

...



Only one cutting tool was used for this example, other tools will follow the same programming procedure. This method of the last example is more common - in absolute mode from main program, tool is positioned at the lower left hand corner of the pattern and the first hole is drilled at that location. Then the subprogram is called and the remaining nine holes are drilled, using incremental positioning commands and the number of cycle repetitions. Number of repetitions in the subprogram is always the number of spaces, *not* the number of holes.

A simpler way, particularly useful for a great number of pattern locations, is to *combine* the rapid motion to the pattern start point location with the subprogram call. This is acceptable for most control systems:

```
O3905 (MAIN PROGRAM)
(FOUR-CORNER PATTERN - DRILLING BLIND HOLES)
N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X1.88 Y1.25
N4 G43 Z1.0 S350 M03 H01
N5 G99 G81 R0.1 Z-0.6221 F3.5 M98 P3955
N6 G90 X6.25 Y1.25 M98 P3955
N7 G90 X6.25 Y5.0 M98 P3955
N8 G90 X1.88 Y5.0 M98 P3955
N9 G80 G90 G28 Z1.0 M05
N10 G91 G28 X0 Y0
...
```

The major advantage of O3905 is shortening the length of program O3904 - either method produces the same results and the selection is a matter of personal preference. Note the seemingly unnecessary repetitions of modal G90 and X and Y axes. Modal values have to be followed extra carefully for subprograms.

### ◆ Modal Values and Subprograms

All modal values in effect when the subprogram is called will remain in effect for that subprogram, unless changed within

In examples O3904 and O3905, note repetitions of G90, X6.25 and Y5.0. They are *very important*. Subprogram O3955 changes the control status to incremental mode G91 and the last hole of the ten hole pattern is *not* the same as the first one. The first hole of the pattern is machined when a rapid motion to that hole is completed in absolute mode. That happens in the main program, *not* within subprogram.

Here is another common problem. Finish contour subprogram uses cutter radius offset G41/G42 with D address. If the same subprogram is used for semifinishing and to leave some stock, for example, it will not work. The reason is that the D offset is fixed and is stored in the control as full cutter radius. Solution? Use *two* D offsets and remove the D address from subprogram, then call it together with M98:

M98 P.. D..

This way, the offset number D can be changed anytime the subprogram is called, without any change to the subprogram itself. This method is useful if the programmed contour requires *two* or *more different* offset settings, but it does not work on all controls. Here is the content of a simple contouring subprogram, with embedded D offset. D51 setting amount is equal to the cutter radius:

```
O3956 (CONTOUR SUBPROGRAM - A)
N561 G41 G01 X0 D51 F10.0      (D.. INCLUDED)
N562 Y1.75
N563 G02 X0.25 Y2.0 R0.25
N564 G01 X1.875
N565 Y0
N566 X-0.75
N567 G00 G40 Y-0.75
N568 M99
%
```

For contour finishing, this subprogram will be called by normal means, typically from the main program:

M98 P3956

The same subprogram can be used for finishing *as well* as for semifinishing, leaving some material stock, but two D offsets have to be used, such as D51 and D52. In this case, offset D51 stores the amount of cutter radius and contain the stock allowance (D51 = cutter radius + stock), D52 stores the finishing radius only (D52 = cutter radius). For a Ø0.500 end mill, the set amounts could be:

```
D51 = 0.250 radius + 0.007 stock = 0.257
D52 = 0.250 radius + 0.000 stock = 0.250
```

Next, the D.. has to be *removed* from the subprogram:

```
O3957 (CONTOUR SUBPROGRAM - B)
N561 G41 G01 X0 F10.0      (D.. NOT INCLUDED)
N562 Y1.75
N563 G02 X0.25 Y2.0 R0.25
N564 G01 X1.875
N565 Y0
N566 X-0.75
N567 G00 G40 Y-0.75
N568 M99
%
```

Control system *does* require the D offset but not necessarily in the same block as G41/G42. As long as the D address is specified *before* G41/G42, it can be passed on to a subprogram from the main program or even another subprogram, depending on operation:

```
M98 P3957 D51      ... for semifinishing
M98 P3957 D52      ... for finishing
```

This is a very powerful method of using subprograms for more than one operation, if the control system supports it.



### Return from a Subprogram

All current modal values should be cleared in the main program when a subprogram is completed. Settings that may have changed in the subprogram are absolute or incremental mode, motion command, coolant and others. Subprogram is always a special *branch* of another program - it is a *continuous extension* of the program of origin and it is its integral part. All modal values set anywhere in the program are valid until changed or canceled by a command of the same group. M99 subprogram end function will *not* cancel any modal values that are currently active.

As both O3904 and O3905 examples show, a fixed cycle is called from the main program only once. All modal cycle data are carried forward to the subprograms. Main program clearly shows current modal settings.

## MULTI LEVEL NESTING

The last example has shown the main program that calls only one subprogram and the subprogram does *not* call another subprogram. This is called *one level* nesting, or nesting at one level deep. Modern controls allow nesting up to four levels deep. That means, if the main program calls a subprogram number one, this subprogram can call a subprogram number two, that can call a subprogram number three, and that can call a subprogram number four. This is called a *four level* nesting. All four levels are rarely needed for any practical application, but these are the programming tools available, just in case. The following examples show program processing flow of each nesting level.

### ◆ One Level Nesting

One level nesting means that a main program calls only one subprogram and nothing more. Subprogram that is nested one level deep is the most common in CNC programming. Program processing starts at the top of the main

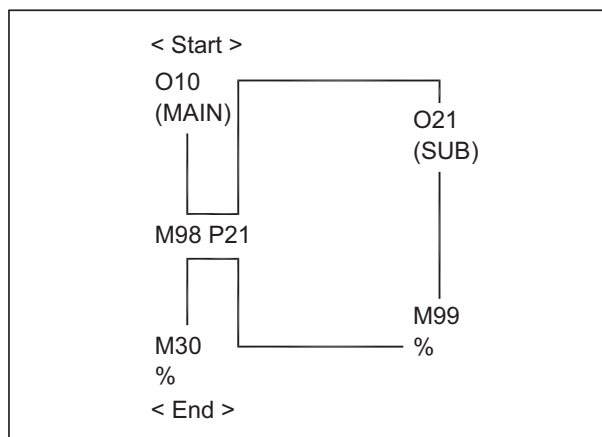


Figure 39-7

One level subprogram nesting

program. When a subprogram is called from the main program by M98 P. block, the control makes a forced branch to the beginning of called subprogram and processes its contents, then it returns to the main program to process all remaining blocks of the main program - *Figure 39-7*.

### ◆ Two Level Nesting

Processing of a subprogram that is nested two levels deep also starts at top of the main program. When the control encounters a subprogram call for the first level, it will branch from the main program and starts processing all blocks in the first subprogram, starting from its top. During processing of the first level subprogram, CNC system encounters a call for a second level subprogram.

At this point, processing of the first level is *temporarily* suspended and CNC system branches to the second level. Since there is no subprogram call from the second level, all subprogram blocks will be processed. Anytime the block containing M99 function is encountered, CNC system will automatically return to the program *it branched out of*. It will *resume* processing of that program, temporarily suspended before.

Return to the program of origin will normally be to the block immediately following the subprogram call block in that program. All remaining blocks in the first subprogram will be executed until another M99 function is encountered. When that happens, the control system will return to the program it branched out of (program of origin), in this case to the main program.

Since there are still some blocks left in the main program, they will be processed until the M30 function is encountered. M30 terminates the execution of the main program. *Figure 39-8* illustrates schematically the concept of a two level subprogram nesting.

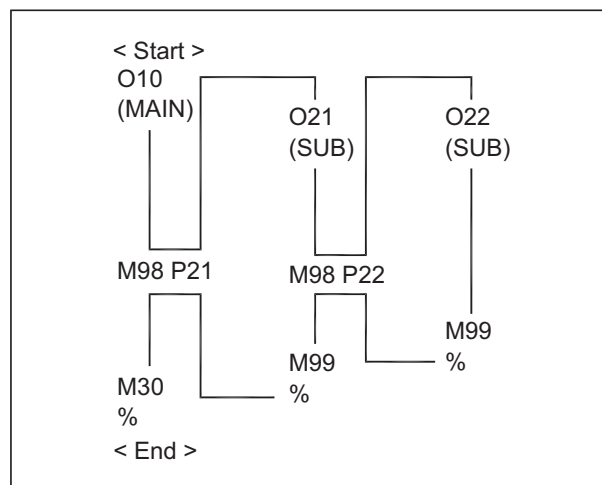


Figure 39-8

Two level subprogram nesting

### ◆ Three Level Nesting

Nesting up to three levels deep is the next logical extension of two level nesting. As before, starting at top of the main program (program O10 in the example illustrated in *Figure 39-9*), the first branch will be to the first level O21, another branch follows O22 and there is an additional branch to O23. Each subprogram is processed up to the next subprogram call, or the end of subprogram. Program processing will always return to the block following the subprogram call, ending in the main program.

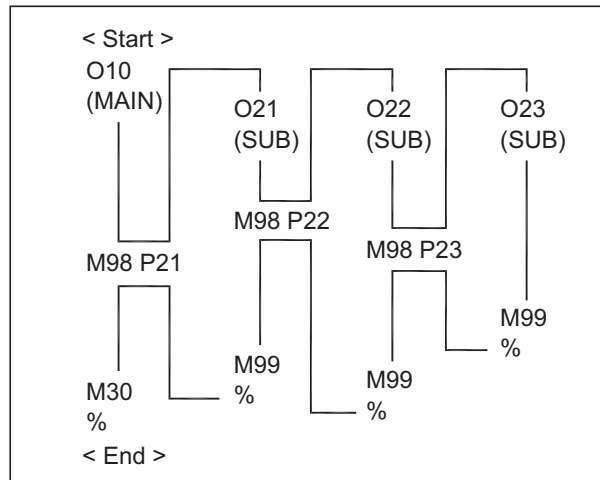


Figure 39-9

Three level subprogram nesting

### ◆ Four Level Nesting

The logic of multi level subprogram nesting should be pretty clear by now. Four level nesting is just a multiple extension of a single nesting and is logically identical to all previous examples.

Unnecessary addition of more branches for a multi depth subprogram nesting makes any programming application that much more complex and more difficult to master.

Programming subprogram nesting into the four level depth (or even the three level depth) will require a full understanding of program processing order - and having a suitable application for it. In a typical machine shop programming, there is seldom any need to use level three and level four nesting. If a good example of a four level nesting application is found, the typical program flow will conform to the format illustrated in *Figure 39-10*.

### ◆ Nesting Applications

Considering the reality that each subprogram can be repeated up to 9999 times in any program that calls it, shows the enormous programming power available to use and explore. Always be aware of potential difficulties, even dangers, when developing subprograms with several multi

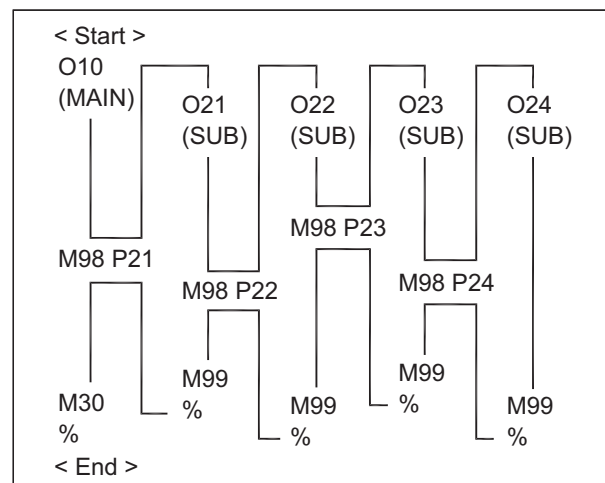


Figure 39-10

Four level subprogram nesting

nested subprograms. Such a programming approach may result in a short program, but at the cost of a long development time. Program preparation time, its development and debugging often take more time than writing conventional programs. Not only the logical development is complex and more time consuming, a significant portion of programming time must be spent on careful and thorough documentation of the process flow of all programs, setting up the initial conditions, checking validity of all data, etc.

There are many fairly experienced CNC programmers in the machining trades field, who try to use a multi level nesting at all costs, and the more levels, the better programmers they feel they are. These programmers, more often than not, use such complex programming technique as means of expressing their so called 'professional skill', usually measured against other programmers. Often, this is nothing more than a unnecessary contest, a frustration perhaps, and definitely an expression of a little ego trip.

When a programmer becomes obsessed with making the program as short as possible, *at any and all costs*, he or she is taking the wrong track. Such programs, even if they are technically flawless and logically correct, are not always very easy to use by the CNC operator. Machine operator with limited or no programming knowledge will find these programs extremely intimidating - even skilled and experienced operators will find them hard to read, hard to interpret, and most likely, they will be unable to make any substantial changes to them, in order to modify or optimize the programs for even a better performance.

A simple general rule for multi level nesting technique - use it only in those cases, when the frequency of their future deployment justifies the extra time spent for their development. Like anything else, many nesting levels offer certain advantages and many inevitable disadvantages.

## CONTOURING WITH A SUBPROGRAM

So far, a number of programming examples provided have been using a subprogram. They all related to machining holes and, hopefully, offered enough material to understand the concept of sub-programming (there will be one more - *a rather special one* - at the end of this chapter, so look for it). There are other examples found throughout the handbook that make generous use of subprograms.

Here is one more example relating to this chapter, this time applying a simple XY contouring work to a multiple Z depth - evaluate *Figure 39-11*.

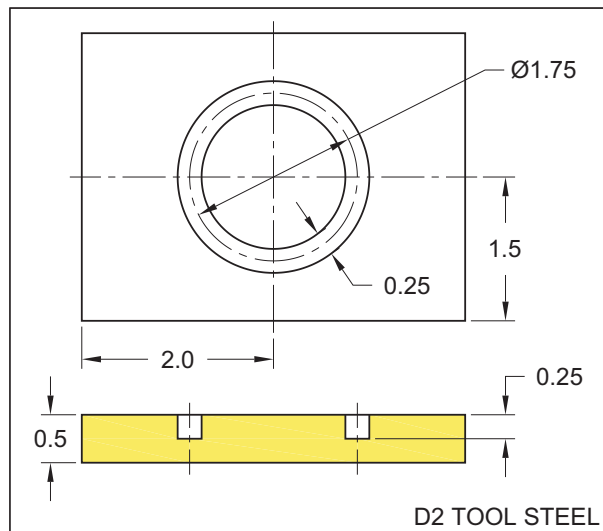


Figure 39-11

Main program O3906 using subprogram O3958

The job requires a groove with a Ø1.750 pitch to be machined to the depth of 0.250. It is a utility or a rough groove, so there is no need for precision tolerances, or even high quality of surface finish. All needed is a Ø0.250 center cutting end mill (slot drill), plunge to the depth, program a 360° circular tool path, and job is done. *Well, almost.*

Even in a material that cuts well, for example brass, splitting a single depth cut of 0.250 into two depth cuts of 0.125 may prove beneficial. Material for the example is *D2 tool steel*, rather a tough material. Tool will run at only 630 r/min and only plunge into the material 0.010 at a time, repeating

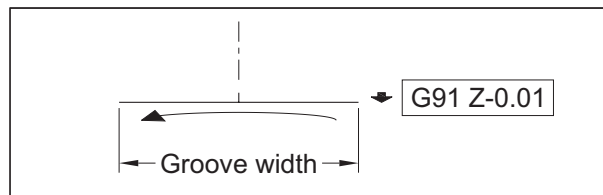


Figure 39-12

Detail of the subprogram O3958 - front view shown

the groove profile 25 times, for  $25 \times 0.010 = 0.250$  total required depth. Preference for a subprogram in such a case is without a question. Symbolic detail of the depth cut for a single increment is illustrated in *Figure 39-12*.

Subprogram O3958 contains only *tool motions common to all groove cuts* - that means the 0.010 incremental plunge cut and the 360° circular cut. All other motions will be included in the main program O3906. Note the word *incremental* for the plunge depth. The 0.010 depth must be programmed incrementally, otherwise it will only cut at the absolute depth of Z-0.01 - *all twenty five times!* Here is the complete main program O3906, followed by a single related subprogram O3958 (T01 is assumed to be in the spindle):

```
O3906 (MAIN FOR SIMPLE DEEP GROOVE)
(T01 - 0.250 DIA CENTER CUTTING END MILL)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X2.875 Y1.5 S630 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z0 F10.0 (START Z POSITION AT Z0 !)
N6 M98 P3958 L25 (CALL SUBPROGRAM 25 TIMES)
N7 G90 G00 Z1.0 M09
N8 G28 Z1.0 M05
N9 M30
%

O3958 (SUB FOR O3906)
N581 G91 G01 Z-0.01 F0.5 (INCREMENT BY -0.01)
N582 G03 I-0.875 F2.0 (FULL CIRCLE CONTOUR)
N583 M99
%
```

Intentionally, the program as presented is simple. It does show, however, two important considerations that have to be maintained in any subprogram development. These considerations relate to maintenance of a *continuous relationship between the main program and the subprogram*. They can be described as special requirements:

- ... to maintain a transfer from the main program to the subprogram (before subprogram is called)
- ... to maintain a transfer from the subprogram, back to the main program (after a subprogram is completed)

The first requirement is met in block N5. The Z-axis position *must be at Z0*, and nowhere else! Being at Z0, it will enable the tool to increment 25 times the distance of 0.010, resulting in 0.250 groove depth. Described differently, the tool start position before a subprogram is called must be at a position that results in a correct tool path (such as depth).

The second requirement is met in block N7. It is the G90 command that makes this block special. Why? Because the subprogram uses G91 command - incremental mode. When subprogram processing returns back to the main program, it no longer benefits from incremental mode, and G90 changes the incremental mode back to absolute mode.

## TOOL CHANGE SUBPROGRAM

Programming sequence for a typical ATC (automatic tool change) is usually short and simple. For CNC milling systems, M06 function will normally do the job and for CNC lathes, it is the T function that does the same thing. Tool change cannot be programmed without establishing certain conditions. Program functions relating to machine zero return, coolant cancellation, spindle stop and others, are all an integral part of the tool change routine. It may take three, four, five or more program blocks to establish the right conditions - *every* time the automatic tool change is programmed, which can be quite often. Even more significant is the fact that most blocks always have the same contents, regardless of the program being used.

As an example of this concept, consider the following sequence of operations - they are quite typical, required to program a tool change for several tools in a single program.

The example is based on a typical vertical CNC machining center, and uses automatic tool change function (ATC):

1. Turn off the coolant
2. Cancel fixed cycle mode
3. Cancel cutter radius offset mode
4. Turn off the spindle
5. Return to Z-axis machine reference position
6. Cancel offset values
7. Make the actual tool change

These seven individual operations occur in *every* program that requires this particular tool change and they will occur for *every* tool in *each* program. That is a lot of programming for a simple tool change sequence. To make such programming easier, develop a subprogram that includes all seven operations, then call it in the main program whenever a tool change is required:

### O3959 (TOOL CHANGE VERTICAL MACHINING CENTER)

```
N1 M09
N2 G80 G40 M05
N3 G91 G28 Z0
N4 G49 D00 H00
N5 G90 M06
N6 M99
%
```

This example can be easily modified for a different machine design or for a CNC horizontal machine. It may even include special requirements, such as specific manufacturer's options. Tool change may even be programmed at a certain machine table position. The only modification would be the addition of a G53X..Y.. block before the tool change block. Another example is a special code for tool change *and* the coolant ON function. Some machine manufacturers create a special M function, combining the two standard functions, for example, M16 - which is the combination of M06+M08 standard functions.

Also note the various cancellation functions - there are quite a few of them in subprogram O3959. When designing such a subprogram, the programmer has absolutely no idea when the program is loaded, whether the coolant will be ON or OFF; no idea if a fixed cycle or the cutter radius offset is active or not. Also, the programmer has no idea as to what the current status of G90 or G91 modes is.

Their *actual* status is really not that important. These cancellations are included in the subprogram, taking advantage of the fact that a cancellation of a function that is already canceled will be ignored by the control system. As the example shows, even a 'simple' tool change sequence requires some serious thinking.

## 100 000 000 HOLE GRID

In the last section of this chapter, perhaps a little deviation from the handbook seriousness will be tolerated. This section will look at subprograms from a different angle, but with a *real* example. The following exercise takes the sub-programming power to the very extreme. Although it is presented primarily on a light note, it does serve a very practical purpose - it shows the power of subprograms and, hopefully, makes a strong case for their use.

The example illustrates how one hundred million holes, (yes, *one hundred million* holes), can be spot drilled *and* drilled using a program of only 29 blocks for the two cutting tools. These 29 blocks even include program numbers and stop codes (% signs). Figure 39-13 shows a simple grid pattern of 10000 rows (X) and 10000 columns (Y).

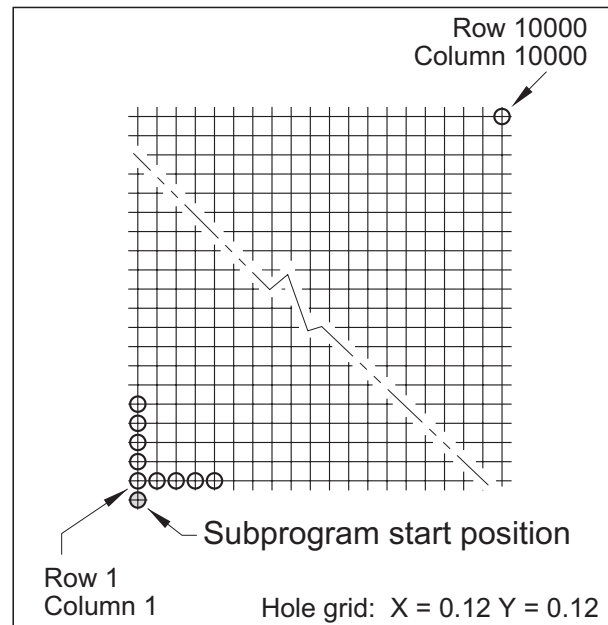


Figure 39-13

100 000 000 holes - rectangular grid pattern



To make the example reasonable, simple, and interesting at the same time, all holes are very small, only  $\varnothing 5/64$  (0.0781), with a pitch of 0.120 along each axis, resulting in a square grid pattern of holes very close to each other.

Only two tools are used, a *spot drill* with a 90° tool point angle to startup the hole for drilling and a  $\varnothing 5/64$  drill. Both cutting tools start machining from R0.06 cycle position above the plate to their respective depths: Z-0.04 for the spot drill and Z-0.215 for the drill.

From programming point of view, the program design is not difficult at all - it will use a main program and one subprogram. Programming procedure is the same for 100 000 000 holes, as if the grid were only 100 holes. The main program contains all standard settings and also calls the subprogram. This subprogram will repeat the active fixed cycle 9999 times, for *two rows*, one in each direction.

Start position for the first tool motion is at an arbitrary location X1.0Y1.0 (shifted by 0.120 along negative Y-axis). Fixed cycle drills the first hole, repeats itself 9999 times, shifts in the positive Y-axis once, drills a hole and repeats along the negative X-axis 9999 times again. This subprogram pattern repeats 5000 times - specified in the main program body:

```
O3960 (SUBPROGRAM)
N601 G91 Y0.12
N602 X0.12 L9999
N603 Y0.12
N604 X-0.12 L9999
N605 M99
%

O3907 (MAIN PROGRAM)
N1 G20
N2 G17 G40 G80 T01 (SPOT DRILL)
N3 M06
N4 G90 G00 G54 X1.0 Y1.0 S3000 M03 T02
N5 G43 Z1.0 H01 M08
N6 G99 G82 R0.06 Z-0.04 P30 F5.0 L0
N7 M98 P3960 L5000
N8 G90 G80 Z1.0
N9 G28 Z1.0
N10 M01

N11 T02 (5/64 DRILL)
N12 M06
N13 G90 G00 G54 X1.0 Y1.0 S3000 M03 T01
N14 G43 Z1.0 H02 M08
N15 G99 G81 R0.06 Z-0.215 F4.0 L0
N16 M98 P3960 L5000
N17 G90 G80 Z1.0
N18 G28 Z1.0
N19 G91 G28 X0 Y0
N20 M30
%
```

This program design takes an advantage of the subprogram nesting and the maximum number of repetitions.

What makes the program even more interesting is the estimate of *machining time*. This may go a little too far, but let's finish the fun. Before reading the whole page, make a guess - *how long will it take to machine all holes with the two tools?* Both speeds and feeds are reasonable for most materials, so are the clearances and dwell time for spot drilling. Rapid traverse of 475 in/min is assumed in all axes, not unreasonable. It is worth the few calculations? Motions between the machine zero and the first location are disregarded in both directions for convenience.

The first calculation finds the time it takes to make a rapid motion between all holes. One hundred million spaces (less one initial space), multiplied by 0.12 divided by 475 in/min is 25,263.1576 minutes. These motions will be multiplied by two, for two tools, therefore 50,526.3153 minutes.

Spot drill will move 0.060 from clearance to the top of part and another 0.040 depth of cut, for the total length of 0.100, multiplied by one hundred million holes at the rate of 5.0 in/min, therefore cutting time for spot drilling will be 2,000,000 minutes. Spot drill will rapid out of the hole one hundred million times the distance of 0.100, at the rate of 475 in/min, totaling 21,052.6316 minutes; the dwell time at each location is 0.030 seconds - translated into minutes, it will take another 50,000 minutes.

Actual drilling will take place to the depth of 0.215 from 0.060 clearance level, for the total travel of 0.275 at the rate of 4.0 in/min - which is another 6,875,000 minutes. Drill will rapid out of one hundred million times by the distance of 0.275, at the rate of 475 in/min, adding another time of 57,894.7368 minutes.

The grand total of all results is 9,054,473.6837 minutes, which is 150,907.8947 hours, which is 6,287.829 days, which is 17.2269 years. Believe it or not, it will take more than seventeen years of *uninterrupted* machining, to spot drill and drill one hundred million holes - and all that can be done with the main program and a subprogram totaling just over two dozen blocks of input.

Going into related details, size of the plate without margins would have to be 100 × 100 feet, so the actual machine travel would have to be greater than 100 feet along the X-axis *as well as* the Y-axis. Hardly any CNC machine on the market can handle this monstrous task. How would the plate be mounted, for example? That is another question.

To make the example even more fun for one last time, consider the time spent on programming, doing it *without* a subprogram and *without* the repetition count (L/K). Assuming that each block will take 6 seconds to write and 55 blocks will fit on a standard paper (hard copy), it would take about 19 years (yes, *nineteen years* !) just to write the program for the two tools (no interruptions, of course). As far as the paper is concerned, it would end up with 'only' 1,818,182 sheets, or a stack of approximately 705 feet (215 meters) thick. Enough of that - *subprograms do work*.



## 40

## DATUM SHIFT

The majority of CNC programs will be programs for a single job - a job that is relative to a specific machine available in the shop. Such a particular job will have its unique characteristics, its special requirements as well as its own toolpath. Toolpath is the most important of all features of a typical CNC program.

It is the CNC programmer's main responsibility to develop a functional toolpath for any given job, without errors and in the most efficient way. Toolpath development is very important, because it represents a machining pattern unique to the job at hand. In most programming jobs, this machining pattern is executed for the given job only and is irrelevant to any other CNC program. Often, programmers encounter opportunities, where an existing machining pattern can be used for many new jobs. This discovery will encourage development of all programs more efficiently and produce CNC programs for many additional applications and without errors.

Programming method that addresses this issue is known as *Translation of a Machining Pattern* or, more commonly, a *Datum Shift*. The most typical example of this method is a temporary change of program reference point (program zero) from the original position to a new position, so called *work shift*. Other programming methods include *Mirror Image*, described in the next chapter, *Coordinate Rotation* and *Scaling Function*, described in the chapters that follow.

This chapter describes in detail the advanced subject of *Datum Shift*, also known as *Machining Pattern Translation*. This is a basic feature of all CNC systems that can be applied in a variety of ways.

## DATUM SHIFT WITH G92 OR G50

In essence, datum shift is a temporary or permanent relocation of the part zero (program reference point) inside of active program. When this programming method is used, it relocates an existing machining pattern (toolpath) in the program at different locations within the CNC machine work area.

In an earlier section (*see page 117*), explanation of G92 (milling) and G50 (turning) commands was covered. Review these commands now, before continuing further. In particular, recall that these commands *do not* cause any direct tool motion, but they do influence any tool motion that *follows* it. Also keep in mind that the position register command G92 and G50 registers the *absolute* coordinates of

the current tool position and have no influence whatsoever on incremental dimensions, when using the G91 command for milling or U/W axes for turning. Its normal purpose is to 'tell' the control system the *current tool position*. This step is necessary at least once at the beginning of each tool to establish the relationship between the fixed program zero (part origin) and the actual position of the cutting tool. For example,

**G92 X10.0 Y6.5**

is 'telling' the control system that the cutting tool is set at positive 10.0 units away from program zero in the X-axis and positive 6.5 units away in the Y-axis.

What happens if a wrong position is registered? What if the preset values in G92 or G50 statement do not accurately reflect the *true*, the physical position of a cutting tool? As may be expected, the toolpath will occur at the wrong place and the result is quite likely a scrap of the machined part, tool breakage, even a damage to the machine itself. Certainly not a desirable situation.

A imaginative CNC programmer always tries to find ways and special methods that take advantage of the available programming tools. G92 and G50 commands are only two of many tools that offer a tremendous power to a creative CNC programmer. Although still available, they are considered obsolete for practical purposes.

For simple jobs, there is no need for special or creative manipulations. It is not very economical to invest precious time on adding features to the program that will never provide real advantages. If such a need is well justified, the program can be optimized later.

### ◆ Program Zero Shift

If the G92 command is used on machining centers or the G50 command for lathes at all, rather than the more current and very efficient G54 to G59 work offsets, only *one* G92 (G50) position register command is needed *for a single tool* - assuming that work offsets are not used.

Any occurrence of more than a single position register command per each tool in one program is called a *program zero shift*.

To illustrate the concept of program zero shift, a simple but relevant drawing will be used. This drawing is illustrated in *Figure 40-1*.

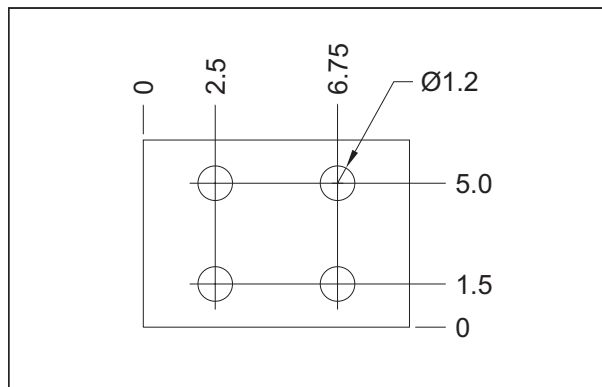


Figure 40-1

A sample drawing for zero shift illustration - program O4001

Based on this drawing, the four holes will be machined at *two independent* locations of the machine table setup, as illustrated in Figure 40-2.

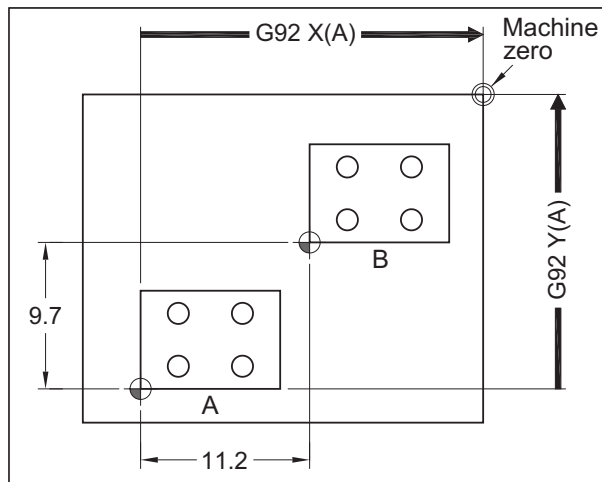


Figure 40-2

Program zero shift using G92 command for two parts - O4001

G92 X(A) indicates the X distance from part zero of *Part A* to machine zero, G92 Y(A) indicates the Y distance from part zero of *Part A* to machine zero. Note that the distances are *from* program zero *to* machine zero. They could terminate anywhere else if necessary, but *must* start from part zero. In order to use G92, the distances between both parts *must be known*. Simple values are used to simplify the example:

*Part A:* **G92 X22.7 Y19.5 Z12.5**

*Part B:* **X-11.2 Y-9.7 Z0** *from Part A*

Also note that the Z value is the *same* for both *Part A* and *Part B*, because the same tool is used for both parts. To spot drill the four holes at two locations, part program may be written this way - program O4001:

```
O4001
(G92 USED FOR TWO TABLE LOCATIONS)
N1 G20 G90
N2 G92 X22.7 Y19.5 Z12.5 (TOOL AT MACHINE ZERO)
N3 S1200 M03
N4 M08
N5 G99 G82 X2.5 Y1.5 R0.1 Z-0.2 P200 F8.0
N6 X6.75
N7 Y5.0
N8 X2.5 (TOOL AT LAST HOLE OF PART A)
N9 G80 Z1.0
N10 G92 X-8.7 Y-4.7 (SET AT LAST HOLE OF A)
N11 G99 G81 X2.5 Y1.5 R0.1 Z-0.2 P200
N12 X6.75
N13 Y5.0
N14 X2.5 (TOOL AT LAST HOLE OF PART B)
N15 G80 Z1.0
N16 G92 X-9.0 Y-4.8 (TOOL FROM MACHINE ZERO)
N17 G00 Z12.5 M09
N18 X0 Y0 (TOOL AT MACHINE ZERO)
N19 M30
%
```

Several blocks require clarification, namely blocks N2, N8, N10, N14, N16 and N18. Each of them relates to the current tool position in some way. *Be very careful here.* Not understanding the principles behind G92 calculations have caused programmers many troubles in the past.

Cutting tool starts from the machine zero position for each program execution. It is also mounted in the spindle before machining. In block N2, the part zero (reference point) for *Part A* is established. Cutting tool at this point is 22.7 inches from program zero along the X-axis, and 19.5 inches along the Y-axis. The coordinate setting in block N2 reflects this fact. In blocks N7 and N8, the tool has completed the last hole of *Part A* (at X2.5Y5.0) of the *current* G92 setting.

The next critical block is N10. At this point in the program, *Part A* is completed, but *Part B* has not yet been started. Think a little now and see where exactly the tool is *after* executing block N9. It is at the position of X2.5Y5.0 of *Part A*. If the tool has to move to the first hole of *Part B*, which is also the position of X2.5Y1.5, the program has to 'tell' the control where the tool is at that exact moment - but in relation to *Part B*! That is done by a simple arithmetic calculation:

$$\begin{aligned} \text{G92 (X)} &= 11.5 + 2.5 - 22.7 = -8.7 \\ \text{G92 (Y)} &= 9.8 + 5.0 - 19.5 = -4.7 \end{aligned}$$

Evaluate Figure 40-3 to visualize the calculation. Direction of arrows in the illustration is important for determining the axis sign in the G92 block.

Blocks N13 and N14 contain coordinates for the last tool location of *Part B*. From the illustration, it should be easy to understand meaning of the coordinate values in block N16. In order to complete the program, the cutting tool has to return to the home position (machine zero). This return

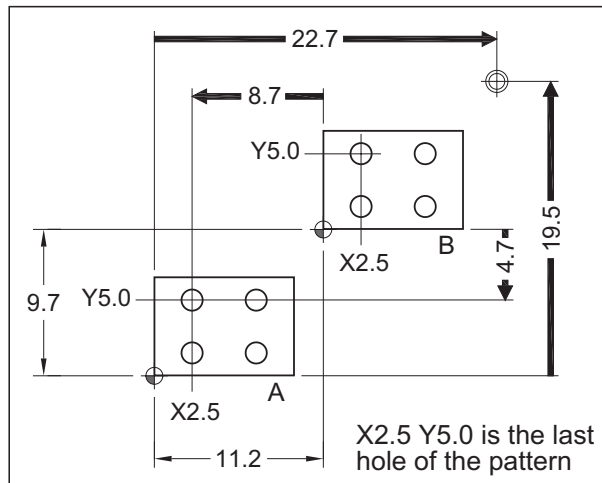


Figure 40-3

Calculations of G92 coordinates (XY) for program example O4001

will take place from X2.5Y5.0 of the Part B, which is 9.0 inches from machine zero along the X-axis and 4.8 inches along the Y-axis:

$$\text{G92 (X)} = 11.2 + 2.5 - 22.7 = -9.0$$

$$\text{G92 (Y)} = 9.7 + 5.0 - 19.5 = -4.8$$

Both programmed coordinates X and Y will be negative.

Once the current tool position is set at the last hole of Part B, return to machine zero can be made. This return is necessary, because it is the location of the *first* tool. The target position for machine zero is X0Y0 not because it is a machine zero, but because the G92 coordinates were measured from there! The actual X and Y motion to machine zero is programmed in block N18.

## LOCAL COORDINATE SYSTEM

The G92 command for position register is as old as absolute programming itself. In time, it has been supplemented by additional commands that control the system of coordinates. Work coordinate system (G54 to G59 work offsets) has been discussed and a suggestion made that G92 should not be used when any work offset is in effect. Such a situation prevents changing the program zero on the fly, when needed only temporarily. Fortunately, there is a solution in the form of a programmable *subset* of work coordinate system (work offsets) called the *local coordinate system* or the *child coordinate system*.

There are many cases, when a drawing is dimensioned in such a way that the work offsets G54 to G59 become somewhat impractical. A good example is a bolt hole pattern. If the overall machined component is round, chances are that the program zero will be selected at the *center* of the bolt hole pattern, which offers a certain benefit in calculations.

However, if the bolt pattern is within a rectangular area, for example, part zero maybe at the *edge* corner of the part.

Normally, absolute locations of the bolt holes will have to be calculated from program zero, unless either a *shift* of the program zero is used (using G92 described earlier), or a special coordinate system is selected.

When working with work offsets, three programming methods are available to make the job a lot more convenient and perhaps even less prone to miscalculations:

- Use the bolt circle center as program zero.  
This will be convenient for CNC programmer only, as it causes more work during setup
- Use two different work offsets in the program, for example, G54 for reference to the part edge and G55 for reference to the center of the bolt circle pattern
- Use local coordinate system, within the current work coordinate system (work offset) selected at the beginning of program

In all cases, one significant advantage has been gained - the programmer uses calculations relating to the bolt circle center coordinates, *directly in the CNC program*, without any need of extra additions and subtractions. This method may even simplify setup on the machine. Which method is better to select and when is addressed next.

The first method, programming to the bolt circle center, is a common method and no comments are necessary.

The second method, using changes from one work offset to another, is also quite common. Its usage is not difficult. One limitation of this method is the reality that only *six* work offsets are available as a standard feature on typical Fanuc controls - G54 to G59. If all six offsets are needed for some work, none is left as a 'spare', to use for situations such as a bolt circle pattern. There are additional work offsets available as an optional feature of the control system.

The third method, using *local coordinate system* method, has the main advantage that it allows the use of a *dependent* - also called a *child* - coordinate system *within* the current work offset - also called the *parent* work offset. Any number of local coordinate systems can be defined within any parent work offset. Needless to say, work is always done in one coordinate system at a time. **Note ...**

Local coordinate system is not a replacement for, **but an addition to**, the work coordinate system

Local coordinate system is a supplement, or a subset, or a 'child' of the current work offset. It must be programmed only when a standard or additional work offset has been selected. There are many applications that can take advantage of this powerful control feature.

### ◆ G52 Command

What exactly is the *local coordinate system*, and how does it work? Formally, it can be defined as a system of coordinates *associated* with the active work offset. It is programmed by the preparatory command G52.

G52	Local coordinate system
-----	-------------------------

G52 command is always complemented by the actual - known - work coordinates that set a new, that is *temporary*, program zero as illustrated in *Figure 40-4*.

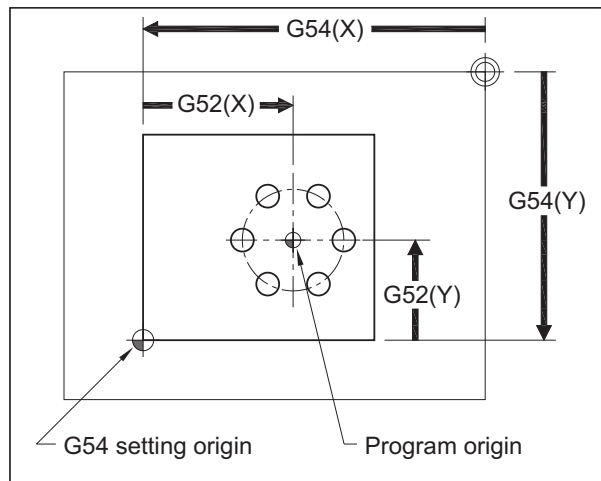


Figure 40-4

Local coordinate system definition using the G52 command

The illustration shows a bolt circle of six holes located in a rectangular plate. Typical program zero is at the lower left corner of plate and the bolt circle center is located X8.0 and Y3.0 inches from that corner, which will become the G52 shift amount. The bolt circle shown is  $\varnothing 4.5$  inches and the first hole is at  $0^\circ$  orientation. Subsequent holes are machined in the CCW direction as holes 2, 3, 4, 5 and 6.

What the program will do is to temporarily transfer the part zero from its current corner of plate to the bolt circle center, *in the program*. Using the illustration as a guide, follow the programming blocks, as they relate to bolt circle and in the logical order they would appear in a program:

```
G90 G54 G00 X8.0 Y3.0      (BOLT CIRCLE CENTER)
(-- WORK COORDINATE SYSTEM POSITION -----)

G52 X8.0 Y3.0
(-- NEW PROGRAM ZERO ESTABLISHED -----)

(G81) X2.25 Y0 (HOLE 1 LOCATION FROM NEW ZERO)
(-- COORDINATES FROM NEW ZERO -----)

G52 X0 Y0
(-- CANCEL LOCAL OFFSET AND RETURN TO G54 ---)
```

The modal G52 command is active until it is canceled in the program. To cancel local coordinate system and to return to the previously active work offset mode, all that has to be done is to program zero axis settings with G52:

G52 X0 Y0 ... last example

All tool motions that follow the cancellation will be relative to the original work offset, which was specified by the G54 selection earlier in the example.

Bolt circle program shown uses the techniques described. Think about the benefit of this type of programming, as opposed to letting the lower left corner be the only part zero.

First, a possible error by the CNC operator during setup has been minimized. True, the operator still has to set the G54 work offset reading at the lower left corner of the plate, but does not have to do any adjustments for the bolt circle center. Programming is also easier, because the coordinate values of the bolt circle originate from the center of the bolt circle, not from the plate corner.

#### O4002 (G54 AND G52 EXAMPLE)

```
N1 G20
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X8.0 Y3.0 S1200 M03 T02 (CNTR)
N5 G43 Z1.0 H01 M08
N6 G52 X8.0 Y3.0 (TEMP PRG ZERO AT BC CNTR)
N7 G99 G82 R0.1 Z-0.2 P100 F10.0 L0 (NO HOLE)
N8 X2.25 Y0 (HOLE 1)
N9 X1.125 Y1.9486 (HOLE 2)
N10 X-1.125 (HOLE 3)
N11 X-2.25 Y0 (HOLE 4)
N12 X-1.125 Y-1.9486 (HOLE 5)
N13 X1.125 (HOLE 6)
N14 G80 Z1.0 M09
N15 G52 X0 Y0 (RETURN TO G54 SYSTEM)
N16 G28 Z1.0 M05
N17 M01

N18 T02
N19 M06
N20 (... Machining continues ...)
```

## MACHINE COORDINATE SYSTEM

So far, the *work coordinate system*, using G54 to G59 work offsets, have been discussed, as well as the *local coordinate system* G52. They are both very powerful and extremely useful programming tools. Fanuc control system offers yet another coordinate system, not commonly used. It may be called the *third coordinate system*.

Selection of this coordinate system is exclusively with the machine coordinates and preparatory command G53.

G53	Machine coordinate system
-----	---------------------------



Machine coordinate system G53 uses the coordinates measured from machine zero as an input - *always!*

At first, benefits in using this unique coordinate system may not be too apparent. Before jumping to conclusions, evaluate the rules for machine coordinates system, perhaps some applications will become clear:

- Command G53 is effective only in the block where it is specified
- Programmed coordinates are always relative to machine zero position
- G53 is only used in the absolute mode (G90)
- Current work coordinate system (work offset) is not canceled by G53 command
- Cutter radius offset should always be canceled prior to G53 command

At least one possible usage emerges from these rules. Machine coordinate system can be used to guarantee tool changes at the *same machine table location* every time automatic tool change is programmed, regardless of which work is on the table and which work offset is active. This can be applied to a single program, or as a standard for all programs for a particular machine tool. Remember, the tool change position will always be determined by the actual tool distance from the machine zero position, *not* the program zero and *not* from any other position. On many machines, or during complex setups, it is advisable to establish a fixed tool change position, regardless of the part position. A good example is machining with a rotary table or any other permanent or semi-permanent fixture located on the machine table.

The following program illustrates the use of G53 command. It makes the tool change at a fixed position of the machine table, position that is *not* directly related to any particular program or any specific job - see *Figure 40-5*.

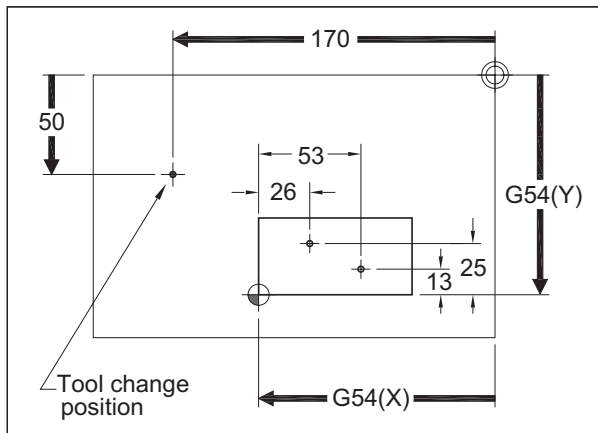


Figure 40-5

Machine coordinate system G53 - program example O4003

```
O4003 (G53 COMMAND USAGE - LARGE MACHINE)
N1 G20
N2 G17 G40 G80 T01
N3 G91 G28 Z0
N4 G90 G53 G00 X-170.0 Y-50.0 (TOOL CHG POS)
N5 M06 (ACTUAL TOOL CHANGE)
N6 G54 G00 X26.0 Y25.0 S1000 M03 T02
N7 G43 Z1.0 H01 M08
N8 G99 G82 R0.1 Z-0.2 P100 F8.0
N9 X53.0 Y13.0
N10 G80 G28 Z1.0 M05
N11 G53 G00 X-170.0 Y-50.0 (TOOL CHANGE POS)
N12 M01

N13 T02
N14 M06 (ACTUAL TOOL CHANGE)
N15 G90 G54 G00 X53.0 Y13.0 S780 M03 T03
N16 G43 Z1.0 H02 M08
N17 G99 G81 R0.1 Z-0.836 F12.0
N18 X26.0 Y25.0
N19 G80 G28 Z1.0 M05
N20 G53 G00 X-170.0 Y-50.0 (TOOL CHANGE POS)
N21 M01

N22 T03
N23 M06 (ACTUAL TOOL CHANGE)
...
%
```

The fourth item of the rules mentioned earlier states that the current work offset is *not* canceled by any active machine coordinate system command (G54-G59 work offset). Since the programming example O4003 does not illustrate this situation, the following sequence of tool motions (*not related to program O4003*) shows that G53 command is independent from G54 command:

```
N1 G21 (METRIC)
...
N250 G90 G54 G00 X17.7 Y35.3
N251 G01 Z-5.0 F200.0
N252 G00 Z500.0
N253 G53 X-400.0 Y-100.0 (FIXED POSITION)
N254 M00 (MANUAL TOOL CHANGE)

N255 S1200 M03
N256 X50.0 Y35.0 (IN ORIGINAL WORK OFFSET)
N257 (... Machining continues ...)
```

Programmed machining sequence is quite simple. Cutting tool moves to the XY part position in block N250, performs the required machining operation, such as drilling to depth in N251, rapids to a clear Z-position in N252, then moves to the fixed tool change position in block N253. In program stopped mode, CNC operator changes the tool manually in block N254, then the spindle speed and rotation are re-established in N255. In block N256 only the X and Y coordinate positions are specified. All other values are default values, including G54 work offset command. The previous block N256 has the same meaning as:

```
N256 G90 G54 G00 X50.0 Y35.0
```





Group L2 is a *fixed offset group number* that identifies the input as work offset setting. The P address in this case can have a value from 1 to 6, assigned to the G54 to G59 selection respectively:

P1 = G54	P2 = G55	P3 = G56
P4 = G57	P5 = G58	P6 = G59

for example,

**G90 G10 L2 P1 X-450.0 Y-375.0 Z0**

inputs X-450.0Y-375.0Z0 coordinates into the G54 work offset register (all examples for this section are metric).

**G90 G10 L2 P3 X-630.0 Y-408.0**

inputs X-630.0Y-408.0 coordinates into the G56 work coordinate offset register. Since the Z-amount has not been programmed, the current amount of the Z-offset is retained.

L2 = Standard work offsets
----------------------------

#### ◆ Additional Work Offset Input

In addition to the standard six work offsets for milling controls, Fanuc offers an optional set of additional offsets, G54.1P1 to G54.1P48. G10 command can also be used to input offset amounts to any one of the 48 additional work offsets while the command itself is very similar to the previous one:

**G10 L20 P.. X.. Y.. Z..**

Only the fixed offset group number has changed to from L2 to L20, which only selects the additional work offsets.

L20 = Additional work offsets
-------------------------------

#### ◆ External Work Offset Input

Another offset that belongs to the work coordinate system group is called *External* or *Common*. This special offset is not programmable - it cannot be programmed with any standard G-code and is only used to update all work offsets globally, affecting all work offsets in a program.

To input offset settings into the external offset, G10 uses the L2 offset group and P0 as the offset selection:

**G90 G10 L2 P0 X-10.0**

will place X-10.0 into the external work offset, while retaining all other settings (the Y-axis, the Z-axis and any additional axis as well). In practice, when using the shown setting, *each* work offset used in a particular program will be shifted by 10 mm into the X-negative direction.

## TOOL LENGTH OFFSETS

Tool length offset amount for milling controls can be programmed with the G10 command combined with the *Lxx* offset group. Depending on the *type* of control memory (see page 261), the L group will have different meanings.

There are three types of memory on Fanuc controls for the tool length and tool radius offsets:

Memory A - only one column for tool length offset
---------------------------------------------------

=> Input: Combined Geometry + Wear offset

Amounts: Amounts set by G10 L11 P.. R.. block

Memory B - two columns for tool offset
----------------------------------------

=> Input 1: Separate Geometry offset value

Amounts 1: Amounts set by G10 L10 P.. R.. block

=> Input 2: Separate Wear offset value

Amounts 2: Amounts set by G10 L11 P.. R.. block

Memory C - two columns for tool offset and two columns for radius offset
--------------------------------------------------------------------------

=> Input 1: Separate Geometry offset value

Used for: H-offset code

Amounts 1: Amounts set by G10 L10 P.. R.. block

=> Input 2: Separate Geometry offset value

Used for: D-offset code

Amounts 2: Amounts set by G10 L12 P.. R.. block

=> Input 3: Separate Wear offset value

Used for: H-offset code

Amounts 3: Amounts set by G10 L11 P.. R.. block

=> Input 4: Separate Wear offset value

Used for: D-offset code

Amounts 4: Amounts set by G10 L13 P.. R.. block

In all cases, the L number is an arbitrarily assigned offset group number by Fanuc and the P address is the offset register number in the CNC system. R setting is the amount of the offset to be set into the selected offset number. Absolute and incremental modes have the same effect on tool length programmed input as for work offsets (replacement in G90 mode, update in G91 mode).

As an example for a CNC machining center, the following block will input the amount of negative 468 mm into the tool length offset register number 5 (five):

**G90 G10 L10 P5 R-468.0**

If the offset has to be adjusted in order to make the cut 0.5 mm less deep for the tool length offset 5, change to the incremental mode G91 and program:

**G91 G10 L10 P5 R0.5**

Note the G91 incremental mode. If the last two examples are used in listed order, the final offset number 5 amount will be -467.5 mm.

Older Fanuc controls were using the address L1 instead of the newer L11. These controls did not have a wear offset as a separate entry. For a compatibility with the older controls, L1 is accepted on all modern controls in lieu of L11

### ◆ Valid Input Range

On most CNC machining centers, the range of tool length offset values is limited:

± 999.999 mm	Metric Geometry offset input
± 99.9999 inches	Imperial Geometry offset input
± 99.999 mm	Metric Wear offset input
± 9.9999 inches	Imperial Wear offset input

Physical number of available offsets is also limited, depending on the control model. There is always a minimum of 32 offset numbers available. Optionally, the CNC system can have up to 999 offsets available (and even more), most of them as a special option.

## CUTTER RADIUS OFFSETS

For the highest offset memory type C, the amount of cutter radius offset (D) may be input through the program, using G10 command with L12 and L13 offset groups:

**G90 G10 L12 P7 R5.0**

will input 5.000 radius value into the cutter radius geometry offset register number 7.

**G90 G10 L13 P7 R-0.03**

will input -0.030 radius amount into the cutter radius wear offset register number 7.

If the existing offset amount needs to be only *adjusted* or *updated*, use incremental programming mode. The last example of a wear offset will be updated by adding 0.01 mm:

**G91 G10 L13 P7 R0.01 (NEW SETTING IS 0.02 MM)**

Be careful with the G90 and G91 mode - remember to restore correct mode for subsequent sections of the program.

## LATHE OFFSETS

Tool length offset does not apply to lathe controls, because of a different offset structure. G10 command can be used to set offset data for a lathe control, using this format:

**G10 P. . X(U) . . Z(W) . . R©) . . Q. .**

The P address is either the *geometry offset* number or the *wear offset* number to be set. Addresses X, Z and R are absolute values, addresses U, W and C are their respective incremental equivalents. No G90 or G91 mode is available, when using the standard G-codes of the *A Group*.

To tell apart geometry and wear offsets, the geometry offset number must be increased by arbitrary 10000:

P10001 will be geometry offset number 1  
P10012 will be geometry offset 12 ... **and so on**

If the setting of 10000 is not added, the P-number will then become the wear offset number.

Here are some typical examples of offset data setting for a CNC lathe, along with expected results. All examples are *consecutive*, based on the order of input listed:

**G10 P10001 X0 Z0 R0 Q0**

... clears all geometry offset for *G 01* settings  
(Geometry offset register 1)

**G10 P1 X0 Z0 R0 Q0**

... clears all wear offset for *W 01* settings  
(Wear offset register 1)

Note - *Q0* also cancels setting of tool tip number in *G 01*

**G10 P10001 X-200.0 Z-150.0 R0.8 Q3**

... sets the contents of *G 01* geometry offset to:

**X-200.0 Z-150.0 R0.8 T3**

... also sets T3 in the wear offset - *automatically !!!*

**G10 P1 R0.8** *Current T setting assumed*

... sets R0.8 amount in *W 01* wear offset

Note, that it may be much safer to program:

**G10 P1 R0.8 Q3**     *Current setting not assumed*

**G10 P1 X-0.12**

... wear offset *W01* is set to X-0.12,  
regardless of its previous setting

**G10 P1 U0.05**

... updates X-0.12 by +0.05,  
to the new value of X-0.07

Note that the tool tip number (programmed in G10 application as the Q entry) will always change both geometry and wear offset simultaneously, whatever the amount or the offset type is. The reason is a control built-in safety that attempts to eliminate data entry error.

## MDI DATA SETTING

Programming various offset settings through part program data requires full understanding of the input format for any particular control system. It is too late when an incorrect setting causes a damage to the machine or part.

One method that can be used to make sure the offset data setting is correct, is a simple test. Test all G10 entries in MDI mode on the CNC unit first, and check the results:

- Select Program mode
- Select MDI mode
- Insert test data

For example, enter:

**G90 G10 L10 P12 R-106.475**

- Press INSERT
- Press CYCLE START

To verify input accuracy, check the tool length offset H12 - it should have the stored amount of -106.475.

While still in the MDI mode, insert another test data, for example:

**G91 G10 L10 P12 R-1.0**

- Press INSERT
- Press CYCLE START

Again, to verify, check the setting of tool length offset H12 - now, it should have the new amount of -107.475.

Develop other similar tests to follow the same routine. It is always better to start a program with confidence.

## PROGRAMMABLE PARAMETER ENTRY

This section covers yet another aspect of programming the G10 command - this time as a *modal* command. It is used to change a system parameter, through the program. This command is sometimes called the '*Write to parameter function*', and is definitely not very common in daily programming. Inexperienced programmers should skip this section altogether. It is very important to understand the concept of control system parameters, otherwise this section will not help much. Authorization to change parameters for the machine tool, regardless of other professional qualifications, is equally important to apply this section.

**WARNING!**  
Incorrect setting of CNC system parameters  
may cause irreparable damage to the CNC machine !

Typical uses of this command are common to changes of machining conditions, such as spindle and feedrate time constants, pitch error compensation data, etc. This command usually appears in optional *Custom Macros* (applied by the G65 command) and its purpose is to control certain machine operations. Concept and explanation of *Custom Macros* is not covered by this handbook. A separate publication by the author is available - *see introduction*.

### ◆ Modal G10 Command

When the G10 command was used for offset data setting earlier, it had to be repeated in each block. G10 for the offset entry can only be used as a *non-modal* command. Modern Fanuc controls also allow to do another type of change through the program - the change of *CNC system parameters* through a *modal* G10 command.

Many entries used in programs are automatically converted to a system parameter by the control system. For example, when programming G54, the set value is seen on the work offset screen. Yet, the actual storage of G54 setting takes place in a system parameter, identified by a certain parameter number. G54 work offset setting can be changed either through the offset data or through parameter change, when the parameter number must be known. Some system parameters cannot be changed as easily (and some cannot be changed at all), so the modal G10 command can be very useful for multiple changes. In fact, two related commands are required - G10 to start and G11 to cancel the setting:

**G10 L50**  
(... *data setting* ...)  
**G11**

The data setting block has three entries:

**G10 L50**  
.. **P.. R..**  
**G11**

In case of a modal G10 and G11 combination, the commands have this meaning:

G10	Data setting mode
L50	Programmable parameter entry mode fixed
.. P. R..	Data entry specification
G11	Data setting mode cancel

Between the block G10L50 and the block G11 is the list of system parameters that are to be set, one parameter per block. Parameter number uses the N-address and all data use P and R addresses. There are several types of parameter input:

Type of parameter input	Allowed input range
Bit type	0 or 1
Bit axis type	0 or 1
Byte type	0 to $\pm 127$
Byte axis type	0 to 255
Word type	0 to $\pm 32767$
Word axis type	0 to $\pm 32767$
Two word type	0 to $\pm 99999999$
Two word axis type	0 to $\pm 99999999$

Watch the bit types parameters - a single data number is always assigned 8 bits. Each bit has a different meaning, so exercise care when changing one bit but not another.

Word type is also called an *integer type* and the two-word type is also called a *long integer type*.

### ◆ Parameters Notation

Numbering of bit type and bit axis type parameters is standard from 0 to 7 (*computers start counting from zero not from one*), from *right to left*:

Number	#7	#6	#5	#4	#3	#2	#1	#0

where *Number* is the four digit parameter number and the #0 to #7 are individual bit positions - *note the order of numbering and the counting method*. Other, non-bit type parameters are input as a byte, word, or two word entries (there are axis and non-axis versions).

#### P Address

The P address is used only for parameters relating to axes (bit axis, byte axis, word axis and two word axis). If the parameter does not relate to an axis, the P address is redundant and does not have to be programmed.

If more than one axis is required to be set at the same time, use multiple .. P. R.. entries between G10 and G11 - see examples further in this section.

#### R Address

The address R is the new value to be registered into the select parameter number and must always be entered. The valid range listed above must be observed. Note the lack of decimal points in the entries.

### ◆ Program Portability

Programs containing even a single programmable parameter entry should be used only with the machine and control for which they were designed.

Use extreme care when a program that modifies system parameters is used on several machines

Parameter numbers and their meaning on different control models are not necessarily the same. The exact control model and its parameter numbers must be known during programming. For example, on Fanuc control Model 15, the parameter that controls the meaning of an address without a decimal point is number 2400 (Bit #0). The parameter that controls the same setting on Fanuc control Model 16 will use number 3401 (Bit #0).

The following examples illustrate various programmable parameter entries and have been tested on a Fanuc 16 Model B CNC control - lathe and mill version. Selected parameters are used for illustration only, not necessarily as typical applications. *Testing these parameters on the machine is not recommended!*

The first example changes the baud rate setting of an Input/Output device with RS-232 interface, if the *I/O Channel* is set to 0):

G10 L50  
N0103 R10  
G11

Parameter that controls the baud rate setting for a selected device has a number #103. From a table supplied by Fanuc, the R-value can be input:

Setting R-value	Description
1	50 baud
2	100 baud
3	110 baud
4	150 baud
5	200 baud



Setting R-value	Description
6	300 baud
7	600 baud
8	1200 baud
9	2400 baud
10	4800 baud
11	9600 baud
12	19200 baud

In the previous example,

```
G10 L50
N0103 R10
G11
```

4800 characters per second baud rate has been selected.

In another example, system parameter #5130 controls the chamfering distance for thread cutting cycles G92 and G76 (gradual pullout distance applicable to lathe controls only). The data type is a non-axis byte, unit of the data is 0.1 of a pitch and the range is from 0 to 127:

```
G10 L50
N5130 R1
G11
```

This program segment will change parameter #5130 to the value of 1. The chamfering amount will be equivalent to one pitch of the thread. Do not confuse byte with a bit - byte is a value 0 to 127 or 0 to 255 for the byte axis type, bit is a single state only (0 or 1, OFF or ON, DISABLED or ENABLED), offering selection of only one of two options available. The word BIT is actually an abbreviation of two words:

Bit = **Binary** digit ('binary' means *based on two*)

Another example is for the entry of a two word parameter type. It will change the work offset G54 to X-250.000:

```
G90
G10 L50
N1221 P1 R-250000
G11
```

Parameter #1221 controls G54, #1222 controls G55, and so on. P1 refers to the X-axis, P2 refers to the Y-axis, and so on, up to 8 axes on model 16. Because the valid range of a long integer (two word type) is required, a decimal point cannot be used. Since the setting is in metric system and one micron (0.001 mm) is the least increment, the amount of -250.000 will be entered as -250000. The following example is *NOT* correct and will result in an error:

```
G90
G10 L50
N1221 P1 R-250.0 (DECIMAL POINT NOT ALLOWED)
G11
```

Proper input is *without* the decimal point. An error condition (alarm or fault) will also be generated if the P address is not specified at all. For example,

```
G90
G10 L50
N1221 R-250000
G11
```

will generate an error condition. The next example is changed for a two axes input:

```
G90
G10 L50
N1221 P1 R-250000
N1221 P2 R-175000
G11
```

If this example is used on a lathe control, P1 is the X-axis, P2 is the Z-axis. On machining center, the P1 is the X-axis, P2 is the Y-axis and P3 will be the Z-axis, if required. In either case, the first two axes of G54 work offset setting will be -250.000 and -175.000 respectively.

Sometimes it is necessary to set all axes to zero. This may be done with a standard offset setting:

```
G90 G10 L2 P1 X0 Y0 Z0 (MILLING CONTROL)
```

or write to a parameter, also for a milling control:

```
G90
G10 L50
N1221 P1 R0 (SET G54 X COORDINATE TO 0)
N1221 P2 R0 (SET G54 Y COORDINATE TO 0)
N1221 P3 R0 (SET G54 Z COORDINATE TO 0)
G11
```

### ◆ Bit Type Parameter

The next example is quite harmless and may be used as a test, but be careful with any other parameters. Its only purpose is to set automatic block sequencing ON while entering a CNC program at the control. It also serves as an illustration of a bit type parameter and is a good example of some general thoughts and considerations that go into program preparation using programmable parameter mode.

On Fanuc 16 Model B (and most of the other models as well) is a feature that allows automatic entry of sequence numbers, if the program is entered from the keyboard. This feature is intended as a time saving device for manual entry of program data. In order to enable this feature, select the parameter that controls the ON and OFF status of this feature. On Fanuc 16 it is a parameter number 0000 (same as 0). This is a bit-type parameter, which means it contains

eight bits. Each bit has its own meaning. Bit #5 (SEQ) controls state of the automatic sequence numbering (ON or OFF is the same as 1 or 0, but only a number can be input). An individual bit cannot be programmed, only the single data number of all eight bits. That means all other bits must be known in order to change one. In this example, the current setting of parameter 0 is as follows:

			SEQ			INI	ISO	TVC
0000	#7	#6	#5	#4	#3	#2	#1	#0
	0	0	0	0	1	0	1	0

Specific meaning of the other parameters is irrelevant for this example and should be maintained. Bit #5 is set to 0, which means the automatic block numbering is disabled.

The following program segment will turn on the bit #5, without changing any other bits:

```
G10 L50
N0 R00101010
G11
```

The resulting entry in the parameter screen will reflect that change:

			SEQ			INI	ISO	TVC
0000	#7	#6	#5	#4	#3	#2	#1	#0
	0	0	1	0	1	0	1	0

Note that all bits had to be written. The job is not done yet, however. Fanuc offers an additional feature - the *increment* for the numbering can be selected as well, for example, selection of 10 will use N10, N20, N30 entries, selection of 1 will use N1, N2, N3, and so on. The example will select increments of five, for N5, N10, N15, etc. The increment has to be set - yes - by another parameter number. On Fanuc 16, the parameter number that contains the automatic numbering value is #3216. This is a word type parameter and the valid range is 0 to 9999. This parameter can only be activated by setting the bit #5 in parameter 0000 to 1. Program segment will look like this:

```
G10 L50
N3216 R5
G11
```

Once these settings are completed, there will be no need to enter block numbers in any program entered via the control panel keyboard.

Anytime the End-Of-Block key EOB is pressed, the N number will appear automatically on the screen, in the increments of five, saving keyboarding time during manual program input.

The idea behind G10 being modal in the programmable parameter entry mode is that more than one parameter can be set as a group. Since the two parameters are logically connected, a single program segment can be created, with the same final results as the two smaller segments earlier. The modal G10 command comes handy here:

```
G10 L50
N0000 R00101010
N3216 R5
G11
```

As neither parameter is the axis type, the address P was omitted. N0000 is the same as N0, and was used only for better legibility.

Note to Fanuc 15 users (Fanuc 15 system is *higher* than Fanuc 16) - the parameter number that selects whether the automatic sequencing will enabled is 0010, bit #1 (SQN). There is more flexibility on Fanuc 15 - the starting sequence number can be controlled with parameter #0031, and the parameter number that stores the increment amount is #0032, with the same program entry styles as shown. Also on Fanuc 15, the allowable range of sequence numbers is up to 99999. This is a typical example of a difference between two control models, even when they were produced by the same manufacturer.

### ◆ Effect of Block Numbers

Many programs include block numbers. It would be perfectly natural to assign block numbers to the last example:

```
...
N121 G10 L50
N122 N0000 R00101010
N123 N3216 R5
N124 G11
...
```

Will the program work? There are now *two* different N addresses in blocks N122 and N123. How does the control handle this situation? Rest easy - there will be no conflict. In case of two N addresses in a single block *within the G10 to G11 segment*, the first N address is the block number, the second one in the same block will be interpreted as the parameter number.

Always use extreme caution when working with system parameters - previous backup is strongly recommended

## 41

## MIRROR IMAGE

The main purpose of a CNC program development is to create a cutter toolpath in a specific location of the part or machine. If such toolpath requires both the right and left hand orientation, overall programming time can be shortened by using a feature called *Mirror Image*.

Any sequence of machining operations can be repeated symmetrically by using the *mirror image* feature of a control system. There is no need for new calculations, so this technique of programming reduces the programming time as well as the possibility of errors. Mirror image is sometimes called the *Axis Inversion* function. This description is accurate up to a point. Although it is true that in mirror image mode selected machine axes will be inverted, but several other changes will also take place at the same time. Therefore, the term *Mirror Image* is more accurate. Those who are familiar with a CAD system will find that the mirror image function in CNC is based on the same principles.

Mirror image is based on the principle of symmetrical parts, known as the *Right Hand* (R/H) and the *Left Hand* (L/H) parts (*Figure 41-1 - CAD mirror image used*).

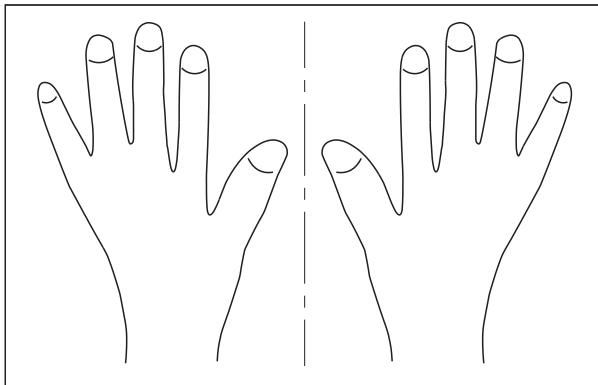


Figure 41-1

Right hand vs. Left hand as the principle of mirror image

Programming mirror image requires understanding of the basic rectangular coordinate system, particularly how it applies to quadrants. It also requires good grasp of circular interpolation and applications of cutter radius offset.

Earlier subjects established that there are four quadrants on any plane. The upper right area creates *Quadrant I*, the upper left area is *Quadrant II*, the lower left area is *Quadrant III*, and the lower right area is *Quadrant IV*. If program zero is at the lower left corner of the part, programming takes place in the first quadrant.

## BASIC RULES OF MIRROR IMAGE

The basic rule of a mirror image is based on the fact that machining a toolpath in one quadrant is not much different than machining the same toolpath in another quadrant. The main difference is the *reversal* of certain motion directions. That means a given part machined in one quadrant can be repeated in another quadrant *using the same program with the mirror image function in effect*.

Main principle of *Right Hand* vs. *Left Hand* orientation can be applied to a machined part orientation - *Figure 41-2*.

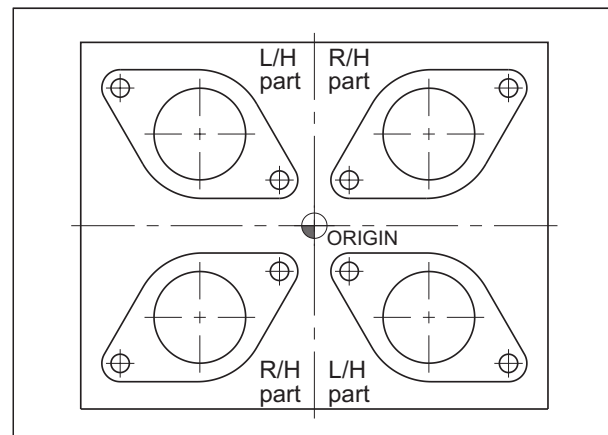


Figure 41-2

Right hand / left hand principle applied to a machined part

It was also established earlier that each quadrant requires unique signs of axes. Mirror image function allows the reversal of axes and other directional changes automatically.

## ◆ Toolpath Direction

Depending on the quadrant selected for mirror image, the toolpath directional change may affect *some* or *all* of these activities:

- Arithmetic sign of axis (plus or minus)
- Milling direction (climb or conventional)
- Arc motion direction (CW or CCW)

One or more machine axes may be affected. Normally, these axes are only the X and Y. Z-axis is generally *not* used for mirroring applications.

Not all activities are affected at the same time. If there is no circular interpolation in the program, there is no arc direction to consider. *Figure 41-3* shows the effect of mirror image on the toolpath, in all four quadrants.

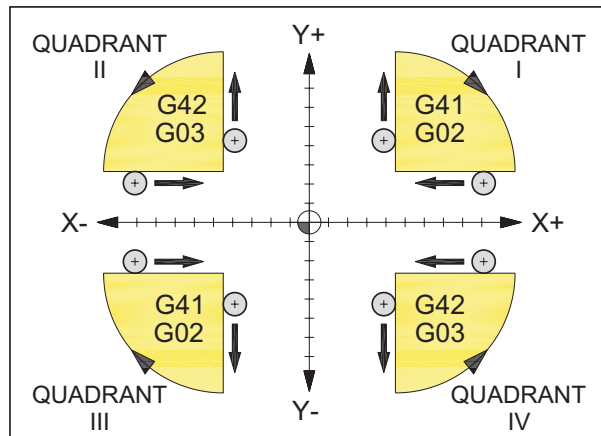


Figure 41-3

Effect of mirror image on toolpath in all four quadrants

### ◆ Original Toolpath

The original toolpath program may be developed in any quadrant. If there is no mirror image applied (default condition), any toolpath is machined in the defined quadrant only. This is how the majority of all applications is programmed. Once mirroring is started, it always mirrors the *original* machining pattern - the original toolpath - regardless in which quadrant it had been defined.

Mirroring will always transfer the machining pattern (the toolpath) to another quadrant or quadrants. That is the sole purpose of mirror image function. Programming mirror image requires that certain conditions are met. One of the conditions is definition of the mirror axis.

### ◆ Mirror Axis

Since there are *four* quadrants, they provide in fact *four* available machining areas. These areas are divided by *two* machine axes. Mirroring axis is the machine axis about which all programmed motions will 'flip' over. *Figure 41-4* shows the mirror axes and their effect on part orientation in quadrants. Mirror axis can be applied in two ways:

- At the machine ... by CNC operator
- Through the program ... by CNC programmer

The typical person who is responsible for the 'flip' is also listed. Either method allows one selection of the following possibilities:

1. Normal machining - no mirror image is set
2. Mirrored machining about the X-axis
3. Mirrored machining about the Y-axis
4. Mirrored machining about the X and Y axes

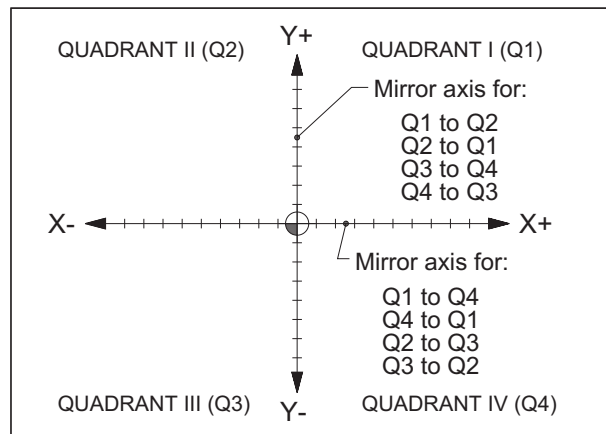


Figure 41-4

Mirror axis selection and its effect on part orientation

Programmable mirror image must be supported by the control system

*Normal* machining follows the program as is. For example, if the programmed path takes place in the second quadrant (using absolute mode G90), normal X coordinates will be negative and normal Y coordinates will be positive. The sign of coordinate points is always normal within the original quadrant programmed, when no mirror image is used. Once the machining takes place in a mirrored quadrant, one or both signs will change.

### ◆ Sign of Coordinates

The 'normal' sign depends on the quadrant of the coordinate system used in programming. If programming in the *Quadrant I*, both the X and Y axes have positive absolute coordinates. Here is the complete list of absolute coordinates when applied to all four quadrants:

Quadrant I	<b>X+ Y+</b>
Quadrant II	<b>X- Y+</b>
Quadrant III	<b>X- Y-</b>
Quadrant IV	<b>X+ Y-</b>

In mirroring programmed toolpath, the control system will *temporarily* change one or both signs, depending on the mirroring axis. For example, if the tool motion is programmed in Quadrant I (X+Y+), and is mirrored about the X-axis, it will assume the signs of Quadrant IV (X+Y-). Only the X-axis is the mirroring axis in this case. In another example, also based on the original program in Quadrant I, mirroring axis is the Y-axis. In this case, the temporary signs will be those of Quadrant II (X-Y+). If mirroring a program defined in Quadrant I along *both* axes, this program will be executed in Quadrant III (X-Y-).

### ◆ Milling Direction

Peripheral milling can be programmed in either *conventional* or *climb* milling mode. When looking at the original tool motion defined in climb milling mode within *Quadrant I*, the mirrored machining motions in the remaining quadrants will be as follows:

- Mirrored in Quadrant II ... Conventional mode
- Mirrored in Quadrant III ... Climb mode
- Mirrored in Quadrant IV ... Conventional mode

It is important to understand the machining mode when using mirror image. A conventional machining mode may not always yield good results. It may negatively affect surface finish of the part as well as dimensional tolerances.

### ◆ Arc Motion Direction

Another change to a toolpath that will happen only when a single axis is mirrored, is the rotation direction of an arc. Any clockwise (G02) arc programmed will become counterclockwise (G03) arc when mirrored along one axis, and vice versa. Here is the result of the arc motion direction; again, based on *Quadrant I*:

- Quadrant I - Original arc is CW :
  - Quadrant II - cutting CCW
  - Quadrant III - cutting CW
  - Quadrant IV - cutting CCW
- Quadrant I - Original arc is CCW :
  - Quadrant II - cutting CW
  - Quadrant III - cutting CCW
  - Quadrant IV - cutting CW

Control system will automatically perform G02 as G03 and G03 as G02 when required. For the majority of machining applications, arc motion direction change should not affect machining quality. For both the milling direction and the arc direction, refer again to the earlier *Figure 41-3*.

### ◆ Program Start and End

When a part is programmed with the intent to use mirror image, make sure to use a carefully thought out programming method, that uses a slightly different method than when programming in a single quadrant (without mirror image). During mirror image, all motions in the program, with the exception of machine zero return, will be mirrored, when the mirror image is turned *on*. That means the following considerations *do* matter and are significant:

1. HOW the program is started
2. WHERE the mirror image will be applied
3. WHEN the mirror image will be canceled

Start and end of a program that is to be mirrored is usually at the same location, typically at the part X0Y0.

## MIRROR IMAGE BY SETTING

Mirror image can also be set at the control unit. No special codes are required. Program is relatively short, since it contains tool motion for one quadrant only. Not every program can be mirrored without a good plan first - it must be structured with mirror image in mind from the beginning.

### ◆ Control Setting

Most controls have a screen setting or switches dedicated to mirror image set at the control. Both types allow the operator to set certain parameters in a friendly way, without the danger of overwriting other parameters by error. In case of a screen setting, a display similar to this will appear:

```
MIRROR IMAGE X-AXIS = 0 (0:OFF 1:ON)
MIRROR IMAGE Y-AXIS = 0 (0:OFF 1:ON)
```

This is the default display, where mirroring for both axes is turned off (cancel mode). To apply X-axis mirroring only, make sure the display shows

```
MIRROR IMAGE X-AXIS = 1 (0:OFF 1:ON)
MIRROR IMAGE Y-AXIS = 0 (0:OFF 1:ON)
```

To apply only the Y-axis mirror, the display must show

```
MIRROR IMAGE X-AXIS = 0 (0:OFF 1:ON)
MIRROR IMAGE Y-AXIS = 1 (0:OFF 1:ON)
```

And finally, in order to mirror about both axes simultaneously, the setting will be ON for both axes:

```
MIRROR IMAGE X-AXIS = 1 (0:OFF 1:ON)
MIRROR IMAGE Y-AXIS = 1 (0:OFF 1:ON)
```

To cancel any mirror image setting and return to normal program mode, the setting for both X and Y axes is zero:

```
MIRROR IMAGE X-AXIS = 0 (0:OFF 1:ON)
MIRROR IMAGE Y-AXIS = 0 (0:OFF 1:ON)
```

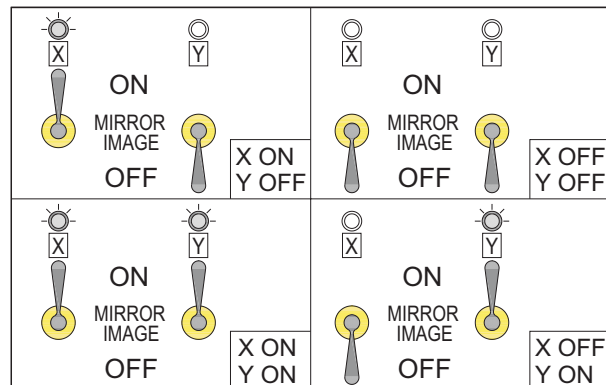


Figure 41-5

Toggle switches for manual setting of mirror image



Figure 41-5 shows mirror image settings using toggle switches ON/OFF mode. Most machines have a confirmation light that is turned ON for the currently mirrored axis.

### ◆ Programming - Manual Mirror Setting

Figure 41-6 is a drawing with three holes to be machined in all four quadrants. It will be used to illustrate the process of setting and programming mirror image.

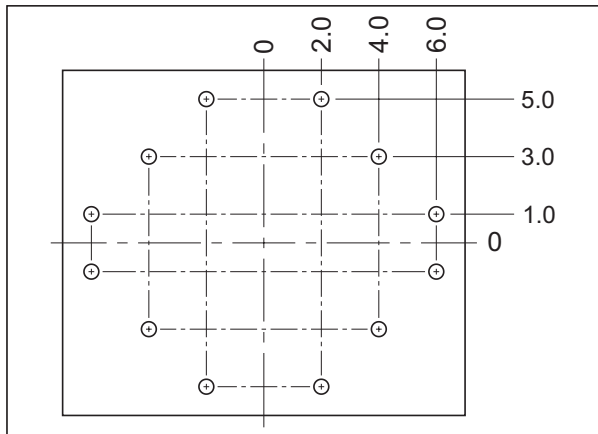


Figure 41-6

Drawing to illustrate manual mirror image programming

For a manual mirror image, the tool motion will be in one quadrant only - Figure 41-7, then mirrored into the other quadrants - Figure 41-8 and example O4101 match:

```
O4101 (CENTER DRILL THREE HOLES)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S900 M03          (X0Y0)
N4 G43 Z1.0 H01 M08
N5 G99 G82 X6.0 Y1.0 R0.1 Z-0.269 P300 F7.0
N6 X4.0 Y3.0
N7 X2.0 Y5.0
N8 G80 Z1.0 M09
N9 G28 Z1.0 M05
N10 G00 X0 Y0          (=> MUST RETURN TO X0Y0)
N11 M30
%
```

Look at the first tool motion in N3. It locates the cutting tool at X0Y0, where there is no hole! This is the most important block in the program for a mirror image, because it is this location that is *common to all four quadrants*!

## PROGRAMMABLE MIRROR IMAGE

Most controls have mirror image that can be set but not programmed. Mirror image activated by the control setting is done on CNC machine, not in the program. On the other hand, programmable mirror image uses M-functions (or sometimes G-codes) and almost always uses subprograms.

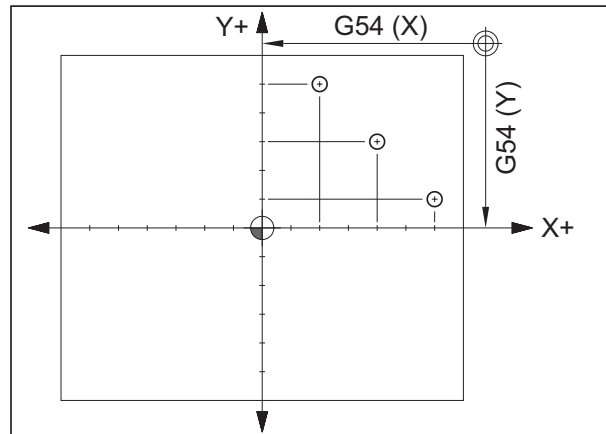


Figure 41-7

Programmed tool motion for the three holes located in Quadrant I

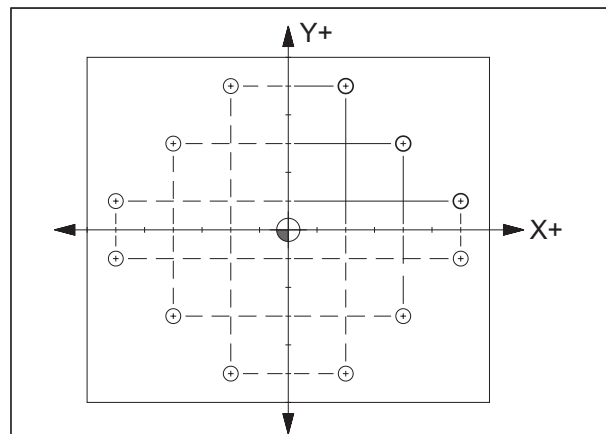


Figure 41-8

Resulting tool motion in all four quadrants using mirror image

Control settings are automatic by the program. The actual program codes for mirror image vary between machines, but the application principles are the same.

### ◆ Mirror Image Functions

In the examples, these functions will be used:

M21	Mirror image along the X-axis
M22	Mirror image along the Y-axis
M23	Mirror image cancel (OFF for either axis)

Mirror image is set for each axis by an M-function. If one function is in effect when another function is programmed, they will *both* be effective. To make only one axis effective, the mirror function must be canceled first.

Cancel mirror image mode when the tool motion is completed

### ◆ Simple Mirror Image Example

Program O4102 for the three holes in *Figure 41-6*, can be changed to the programmable mirror image. Holes absolute locations are stored in subprogram O4151:

#### O4151

```
N1 X6.0 Y1.0
N2 X4.0 Y3.0
N3 X2.0 Y5.0
N4 M99
%
```

Main program O4102 calls the subprogram O4151 in different quadrants, using mirror image functions.

Note that X0Y0 location is common to all four quadrants

```
O4102                                (MAIN PROGRAM)
N1 G20
N2 G17 G40 G80
N3 M23                                (=> MIRROR OFF)
N4 G90 G54 G00 X0 Y0 S900 M03        (X0Y0)
N5 G43 Z1.0 H01 M08
N6 G99 G82 R0.1 Z-0.269 P300 F7.0 K0 (L0)
N7 M98 P4151                          (QUADRANT I)
N8 M21                                (=> X-MIRROR ON)
N9 M98 P4151                          (QUADRANT II)
N10 M22                               (=> Y-MIRROR ON)
N11 M98 P4151                         (QUADRANT III)
N12 M23                               (=> MIRROR OFF)
N13 M22                               (=> Y-MIRROR ON)
N14 M98 P4151                         (QUADRANT IV)
N15 G80 Z1.0 M09                      (CYCLE CANCEL)
N16 M23                               (=> MIRROR OFF)
N17 G28 Z1.0 M05                      (Z MACHINE ZERO)
N18 G00 X4.0 Y6.0                    (CLEAR ATC LOCATION)
N19 M30                               (PROGRAM END)
%
```

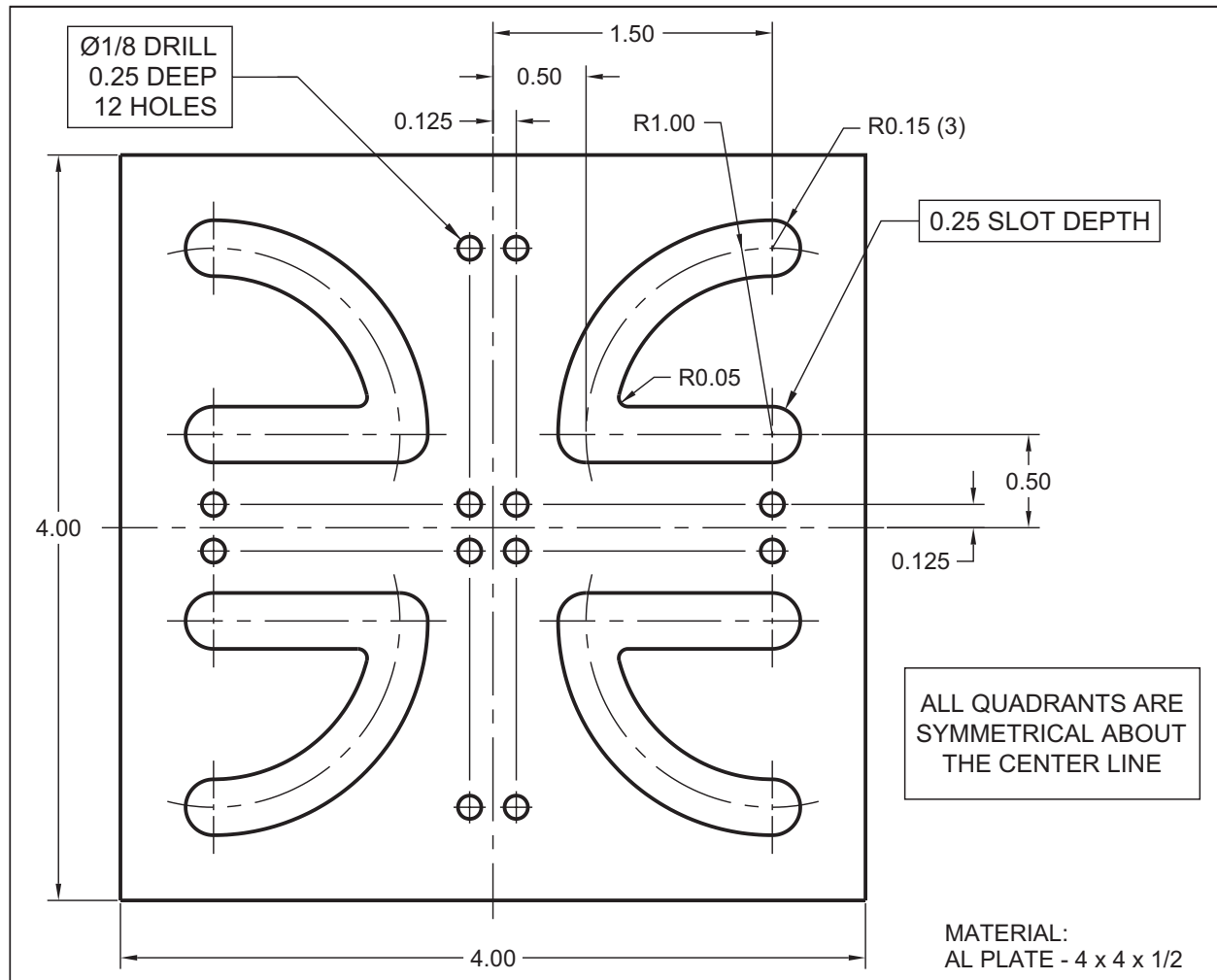


Figure 41-9

Comprehensive example of programmable mirror image. Uses main program O4103 and subprograms O4152 and O4153

### ◆ Complete Mirror Image Example

Complete example of a mirror image application with more elaborate tool motions will use two cutting tools to develop a program as per drawing in *Figure 41-9*. Program will also use coordinate shift G52, automatic tool change, a fixed cycle, interpolation motions and cutter radius offset. Two subprograms are needed - one for drilling the three holes in O4152, one for the slot milling in O4153.

```
O4152 (SUBPROGRAM - DRILLING)
N101 X0.125 Y0.125          (MIDDLE HOLE)
N102 X1.5                   (HOLE IN X)
N103 X0.125 Y1.5           (HOLE IN Y)
N104 X0 Y0 K0 (L0)         (NO HOLE AT PLATE CENTER)
N105 M99                   (SUBPROGRAM O4152 END)
%
```

Subprogram O4152 contains the 3 hole locations only, in *Quadrant I*. Cycle call is not included in the subprogram, and return to the center of the plate (N104) is still in a cycle mode but with the K0/L0 modifier.

```
O4153 (SUBPROGRAM - MILLING)
N201 G00 X1.5 Y1.5          (CENTER OF SLOT)
N202 G01 Z-0.25 F3.0
N203 G03 X0.5 Y0.5 I0 J-1.0 F5.0
N204 G01 X1.5
N205 G41 D01 X1.365 Y0.485   (SLOT START)
N206 G03 X1.5 Y0.35 I0.135 J0
N207 X1.65 Y0.5 I0 J0.15
N208 X1.5 Y0.65 I-0.15 J0
N209 G01 X0.7254
N210 G02 X0.6754 Y0.7 I0 J0.05
N211 X0.677 Y0.7125 I0.05 J0
N212 X1.5 Y1.35 I0.823 J-0.2125
N213 G03 X1.65 Y1.5 I0 J0.15
N214 X1.5 Y1.65 I-0.15 J0
N215 X0.35 Y0.5 I0 J-1.15
N216 X0.5 Y0.35 I0.15 J0
N217 G01 X1.5
N218 G03 X1.635 Y0.485 I0 J0.135
N219 G01 G40 X1.5 Y0.5      (SLOT END)
N220 G00 Z0.1
N221 X0 Y0                  (MOTION TO PLATE CENTER)
N222 M99                   (SUBPROGRAM O4153 END)
%
```

Quadrant I is also used in subprogram O4153 for one slot. Machining starts with the cutter at the slot centerline, roughing the radius and the walls. Then, cutter radius offset is used and slot is finished to size. The subprogram ends at the plate center in N221, the same as in drilling. Program O4103 uses both subprograms. If more tools are used, the programming method will not change.

```
O4103 (MAIN PROGRAM)
(USES SUBPROGRAMS O4152 AND O4153)
(X0 Y0 LOWER LEFT CORNER - Z0 TOP OF PART)
(M21 = X-MIRROR ON -----)
(M22 = Y-MIRROR ON -----)
(M23 = MIRROR OFF -----)
```

```
(T01 - 1/8 DIA SHORT DRILL)
N1 G17 G20 G40 G80 G49      (STARTUP BLOCK)
N2 T01 M06                  (TOOL CHANGE)
N3 G52 X2.0 Y2.0 M23        (MIRROR OFF)
N4 G90 G54 G00 X0 Y0 S1800 M03 T02
N5 G43 Z1.0 H01 M08
N6 G99 G81 R0.1 Z-0.269 F4.0 K0 (L0)
N7 M98 P4152                (QUADRANT I)
N8 M21                      (X-MIRROR ON)
N9 M98 P4152                (QUADRANT II)
N10 M22                     (Y-MIRROR ON)
N11 M98 P4152               (QUADRANT III)
N12 M23                     (MIRROR OFF)
N13 M22                     (MIRROR ON)
N14 M98 P4152               (QUADRANT IV)
N15 G80 M09                 (CYCLE CANCEL)
N16 M23                     (MIRROR OFF)
N17 G52 X0 Y0
N18 G28 Z0.1 M05
N19 G00 X4.0 Y6.0           (CLEAR ATC LOCATION)
N20 M01                     (OPTIONAL STOP)
```

```
(T02 - 1/4 DIA CENTER CUTTING END MILL)
N21 T02 M06                 (TOOL T02 TO SPINDLE)
N22 G52 X2.0 Y2.0 M23        (MIRROR OFF)
N23 G90 G54 G00 X0 Y0 S2500 M03 T01
N24 G43 Z0.1 H02 M08
N25 M98 P4153               (QUADRANT I)
N26 M21                     (X-MIRROR ON)
N27 M98 P4153               (QUADRANT II)
N28 M22                     (Y-MIRROR ON)
N29 M98 P4153               (QUADRANT III)
N30 M23                     (MIRROR OFF)
N31 M22                     (Y-MIRROR ON)
N32 M98 P4153               (QUADRANT IV)
N33 M23                     (MIRROR OFF)
N34 G52 X0 Y0 M09
N35 G28 Z0.1 M05
N36 G00 X4.0 Y6.0           (CLEAR ATC LOCATION)
N37 M30                     (PROGRAM END)
%
```

Note how the G52 command is used. In order to use mirror image correctly, the program zero must be defined on the mirror line (mirror axis). Since two lines (axes) are required for this project, the plate center plate *must be* the program zero. There is no need to return to the X and Y machine zero, either at the end of the tool or at the end of program. Location in a clear area for the tool change is all that is needed.

## MIRROR IMAGE ON CNC LATHES

*Mirror Image* function has its main application on a CNC machining center. On lathes, this application is limited to a lathe with two turrets, one on each side of the spindle center line. The actual mirroring will use the X-axis (the spindle center line) as the mirror axis and, in effect, allows the same programming method for both turrets.

Machining with mirror image can be used alone or combined with other time saving features, such as *Coordinate Rotation* and *Scaling Function*.

## 42

## COORDINATE ROTATION

A programmed tool motion that creates a pattern, contour or a pocket in orthogonal orientation can be rotated about a defined point and by specified angle. With this control feature, there are many opportunities to make CNC programming process easier, flexible and more efficient. This very powerful programming feature is usually a special control option, and is called the *Coordinate System Rotation*, or just *Coordinate Rotation*.

One of the most important applications of coordinate rotation is a program that is defined mainly vertically or horizontally (orthographic orientation) but machined at an angle specified by the drawing. Orthographic mode defines a tool motion as parallel to machine axes. To program orthographic mode is much simpler than calculating tool positions for many contour change points in true angular orientation. Compare the two rectangles shown in *Figure 42-1*.

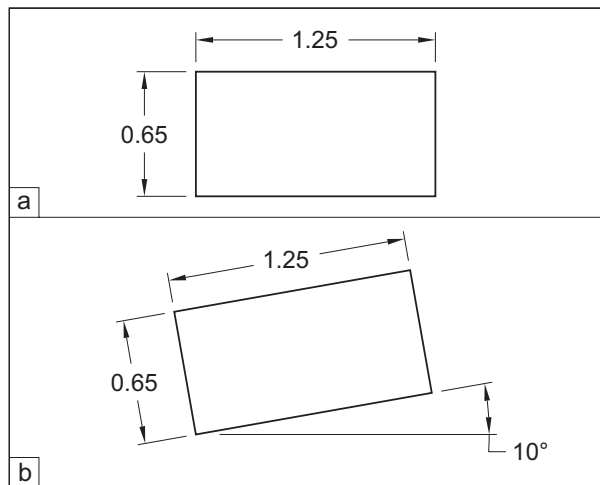


Figure 42-1

Original orthogonal object (a) and a rotated object (b)

The above figure (a) shows an orthogonal orientation of a rectangle, the figure below (b) shows the same rectangle, rotated by  $10^\circ$  in the counterclockwise direction. Manually, it is much easier to program the tool path for figure (a) and let the control system change it to a tool path represented in figure (b). Coordinate rotation feature is a special option and must be a part of the control system.

Mathematically, coordinate rotation is a feature that requires only three items to define a rotated part - the *center of rotation*, the *angle of rotation*, and the *tool path* to rotate.

## ROTATION COMMANDS

Coordinate rotation uses two preparatory commands to select this feature to ON or OFF mode. These two preparatory G commands that control coordinate rotation are:

G68	Coordinate system rotation ON
G69	Coordinate system rotation OFF

G68 command will *activate* coordinate system rotation, based on the *center of rotation* (also known as the *pivot point*) and the *degrees of rotation*:

G68 X.. Y.. R..

where ...

- X = Center of rotation defined by absolute X coordinate
- Y = Center of rotation defined by absolute Y coordinate
- R = Angle of rotation

### ◆ Center of Rotation

Both X and Y coordinates define the center of rotation (pivot point). This is a special point about which the rotation takes place - a point that can be defined by any two axes, depending on the selected working plane:

- XY coordinates define rotation point in G17 plane
- XZ coordinates define rotation point in G18 plane
- YZ coordinates define rotation point in G19 plane

Desired plane selection command G17, G18 or G19 must be entered into the program anytime *before* the rotation command G68 is issued. For more details relating to work planes, see *page 279*.

If center of rotation coordinates are not specified with the G68 command, the *current tool position* will be used as the *default* center of rotation. This method is neither a practical nor a recommended approach in any circumstances.

### ◆ Radius of Rotation

G68 angle representation is specified by the amount of R, using *degrees* as units, and measured from the *defined center*. Specified number of decimal places of R will become the angular amount. Positive R defines CCW rotation, negative R defines CW rotation - *Figure 42-2*.

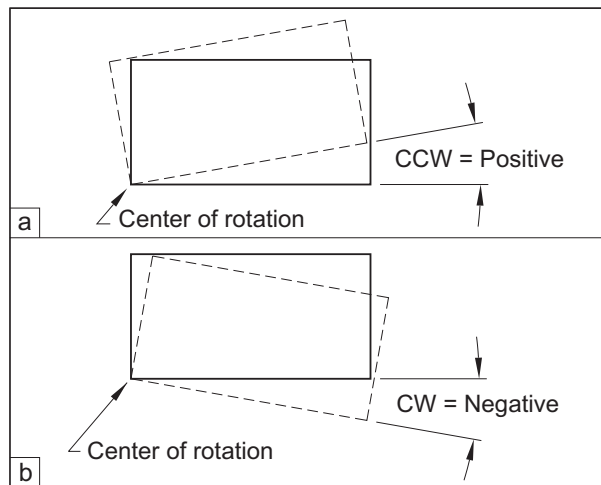


Figure 42-2

Direction of coordinate rotation, based on the center of rotation:

- (a) Counterclockwise direction has a positive angle  $R$
- (b) Clockwise direction has a negative angle  $R$

For basic programming example, a simple part shape that is easy to visualize will be used, such as a typical rectangular shape with a fillet corner radius - Figure 42-3.

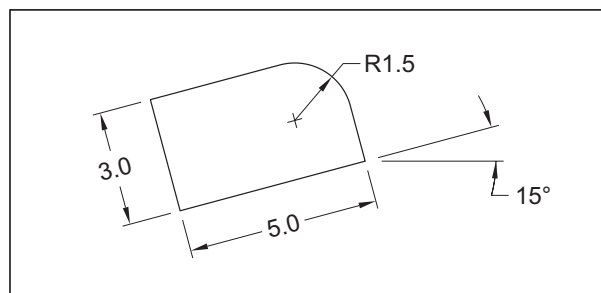


Figure 42-3

Part oriented as per engineering drawing specification

Actual tool path, including approach towards the part and departure from the part, is not included in the engineering drawing. *Be careful here* - if the approach and/or departure motions are included in the rotation, the *program zero* may also be rotated. In the Figure 42-4, the part orientation is 15° counterclockwise, based on its lower left corner.

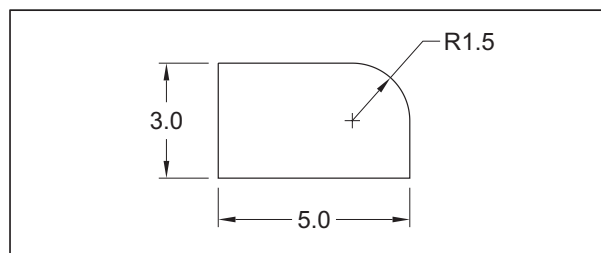


Figure 42-4

Part oriented as per program, using the G68 command

For a moment, ignore the rotation angle and program the part as if it were oriented in an orthogonal position, that is perpendicular to the axes, as shown in Figure 42-4.

For actual cutting path, decide whether the approach tool motions will be included in the rotation or not. This is a very important decision. In Figure 42-5 are two possibilities and the effect of coordinate rotation on program zero. In both cases, the approach tool path starts and ends at the same location of X-1.0 and Y-1.0 (clearance location).

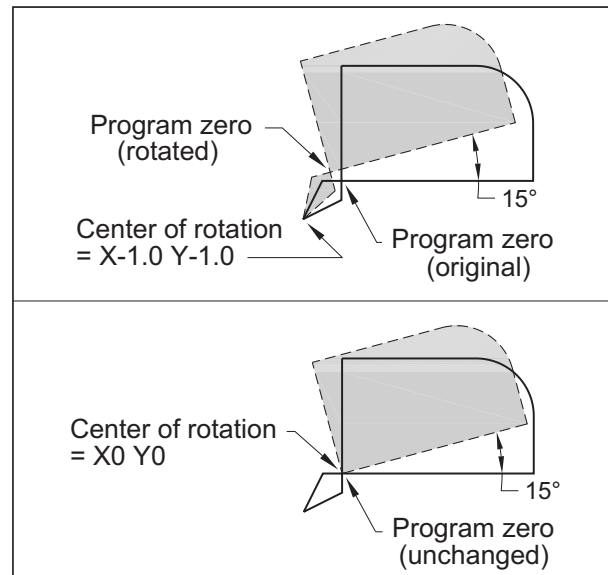


Figure 42-5

Comparison of the programmed tool path (solid line) and the rotated tool path (dashed line):

- (a) Program zero included in the rotation
- (b) Program zero not included in the rotation

Following program O4201 illustrates the above example (a) in Figure 42-5, which *does* include the program zero rotation. If the program zero is not to be rotated, include only the part profile tool path between G68 and G69 commands, and *exclude* the tool approach or departure motions. Also note the G69 in block N2 - coordinate mode cancellation is included there for added safety.

```
O4201
N1 G20
N2 G69 (ROTATION CANCELED IF NEEDED)
N3 G17 G80 G40
N4 G90 G54 G00 X-1.0 Y-1.0 S800 M03
N5 G43 Z0.1 H01 M08
N6 G01 Z-0.375 F10.0
N7 G68 X-1.0 Y-1.0 R15.0
N8 G41 X0 Y-0.5 D51 F20.0
N9 Y3.0
N10 X3.5
N11 G02 X5.0 Y1.5 I0 J-0.5
N12 G01 Y0
N13 X-0.5
```



```

N14 G40 X-1.0 Y-1.0 M09
N15 G69 (ROTATION CANCELED)
N16 G28 X-1.0 Y-1.0 Z1.0 M05
N17 M30
%
```

This simple program is developed for the orthogonal part orientation ( $= 0^\circ$  rotation), but machined at  $15^\circ$ , using the coordinate system rotation option.

In the example, block N8 contains cutter radius offset G41. Any tool offset or compensation programmed will be included when the coordinate rotation takes place.

### ◆ Coordinate Rotation Cancel

Command G69 *cancels* the coordinate rotation function and returns control system to its normal orthogonal condition. Always specify the G69 command in a separate block, as in the O4201 example.

### ◆ Common Applications

As mentioned already, the majority of CNC machines do *not* have coordinate rotation function available at all - *or* - they may have it available as an optional feature. Having this function can be very useful in two particular areas of machining applications:

- If the nature of work includes orthogonal parts machined at an angle (as per drawing requirement) - *the earlier example belongs to this category*
- If there is a short X and/or Y travel on the machining center and the part is positioned on the table at a known angle, because of limited machine travel

The second application in particular is a very useful example of coordinate system rotation, provided that two major conditions are satisfied:

- Rotated part must fit within the work area
- Angle of the setup must be known

In the *Figure 42-6*, a part cannot fit within the work area orthogonally, but it can fit when rotated.

This method is quite interesting but it is not always possible to be implemented. A hundred inch long part cannot be placed within the work area length of only 20 inches. However, there are cases when this programming technique can be very useful, even if it is not too common. The illustration only shows general principles of the application. If the positioning angle is not known, use an indicator at two locations of the mounted part and calculate it trigonometrically. In some cases, a special fixture may be required for such a setup.

Do not confuse terms **work area** and **table size**

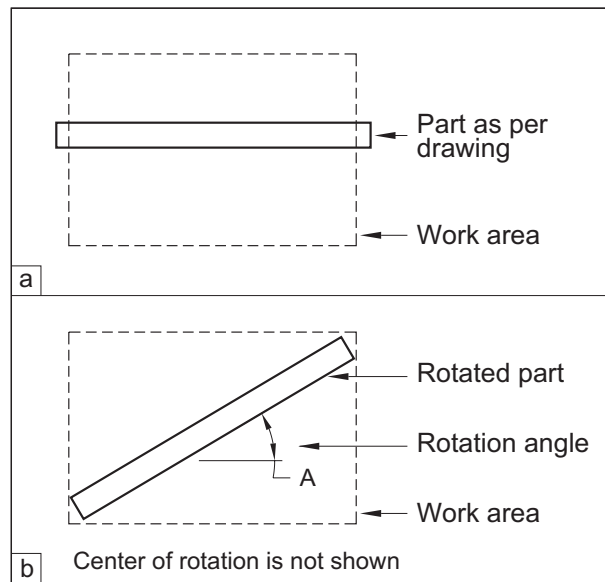


Figure 42-6

Coordinate rotation applied to fit a long part within the work area

Machine table size is always larger than the work area, to allow for setup and additional space. Work area is used for programming and often setup as well, and is always defined by the tool motion limits. Work area must be able to accommodate all programmed tool motions and clearances, including those with cutter radius offset in effect.

## PRACTICAL APPLICATION

In many cases, subprograms can be used very efficiently together with coordinate rotation. Applications such as milling polygonal shapes or machining at bolt circle locations are only typical possibilities. The following detailed example in *Figure 42-7* shows a part drawing that looks deceptively simple but involves quite a bit of programming.

All requirements and conditions for program development must be evaluated. The program core will machine all seven pockets with a  $\varnothing 0.25$  end mill (center cutting type). To make the program more realistic, rather than plunging to the full depth of 0.235, selection of 0.05 as the maximum depth of cut may be practical. Program will also leave some stock for finishing of the pocket walls (0.0075 per side). In addition, all sharp edges must be broken with a minimum chamfer. In all, only three tools will be used:

- Ø 3.0 FACE MILL
- Ø 1/4 CENTER CUTTING END MILL
- Ø 3/8 CHAMFERING TOOL

This is definitely a very advanced programming application. Not understanding the program at first is expected. With growing experience, it will be easier to interpret all program details. Hopefully, the enclosed notes will help.

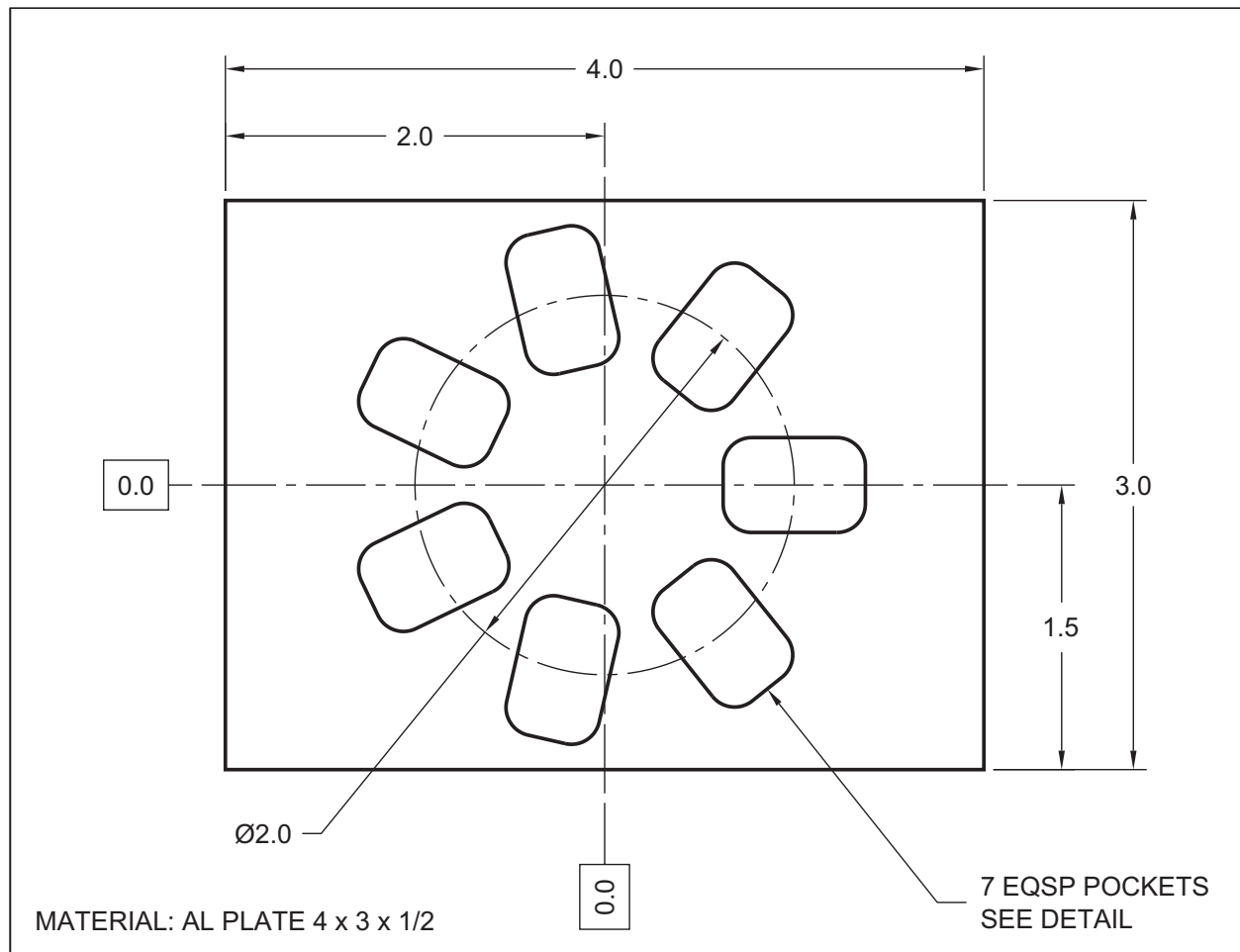


Figure 42-7

Comprehensive example of coordinate system rotation - program O4202

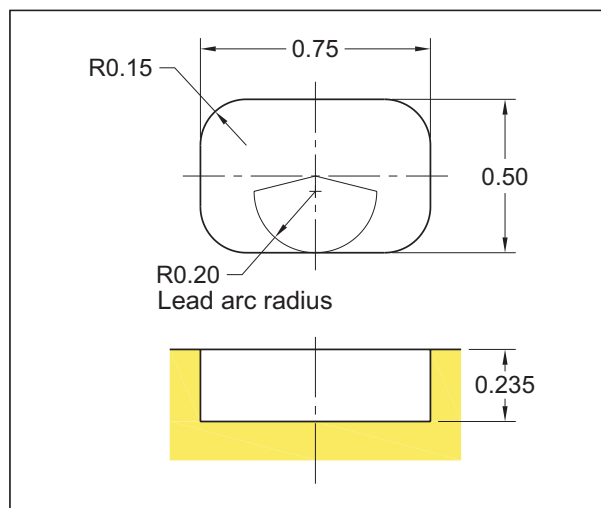


Figure 42-8

Top and front view of the pocket detail for program O4202

The main program O4202 will be developed with the aid of *four* subprograms. Although some parts may be a little difficult to understand, one key element is absolutely critical. In the two subprograms will be the following block:

**G91 G68 X0 Y0 R51.429**

Its purpose is to shift to the next pocket, at an angle. The X0Y0 remain the same - they will *always* be absolute, only the angle will be incremented, because of G91 command.

This example is not only a good illustration of coordinate system rotation, but also shows more advanced techniques of using subprograms as well as several additional features. Without any advanced programming techniques, the program could be done as well, but it would take much longer and it would be virtually impossible to optimize it at the machine. Complete program that follows - O4202 - is documented in detail and should present no problem to follow its progress and structure.

## O4202 (COORDINATE SYSTEM ROTATION)

```

(7 POCKETS - PETER SMID - VERIFIED ON FANUC 15M CNC SYSTEM)
(PARAMETER #6400 BIT #0 - RIN - MUST BE SET TO 1 TO ALLOW G90 AND G91)
(MATERIAL 4 X 3 X 1/2 ALUMINUM PLATE - HORIZONTAL LAYOUT)
(X0Y0 IS CENTER OF 2.0 DIA CIRCLE - Z0 AT THE FINISHED TOP OF THE PLATE)

(T01 ..... 3.0 DIA FACE MILL - SKIM CUT TO CLEAN TOP FACE)
(T02 ..... 1/4 DIA CENTER CUTTING END MILL - MAX DEPTH OF CUT 0.05)
(T03 ..... 3/8 DIA CHAMFERING TOOL - 90 DEGREES - MINIMUM CHAMFER)
(T02 / D51 - OFFSET FOR ROUGHING POCKET WALLS .... 0.140 SUGGESTED - 0.0075 PER SIDE)
(T02 / D52 - OFFSET FOR FINISHING POCKET WALLS ... 0.125 SUGGESTED)
(T03 / D53 - OFFSET FOR CHAMFERING ..... 0.110 SUGGESTED - TO BE ADJUSTED)
(INCREMENT OF ROTATION ..... 360/7 = 51.429 DEGREES)

(T01 - 3.0 DIA FACE MILL - SKIM CUT TO CLEAN TOP FACE)
N1 G20 (IMPERIAL UNITS)
N2 G69 (CANCEL COORDINATE ROTATION IF ACTIVE)
N3 G17 G40 G80 T01 (SEARCH FOR T01 IF NOT READY)
N4 M06 (T01 TO THE SPINDLE)
N5 G90 G54 G00 X-1.375 Y-3.25 S3500 M03 T02 (XY START POSITION FOR FACE MILLING)
N6 G43 Z1.0 H01 M08 (Z CLEARANCE FOR SETUP - COOLANT ON)
N7 G01 Z0 F30.0 (TOP OF FINISHED PART FOR FACE MILLING)
N8 Y3.125 F15.0 (FACE MILL LEFT SIDE)
N9 G00 X1.375 (MOVE TO THE RIGHT SIDE)
N10 G01 Y-3.25 (FACE MILL RIGHT SIDE)
N11 G00 Z1.0 M09 (Z AXIS RETRACT - COOLANT OFF)
N12 G28 Z1.0 M05 (Z AXIS HOME FOR TOOL CHANGE)
N13 M01 (OPTIONAL STOP)

(T02 - 1/4 DIA CENTER CUTTING END MILL - MAX DEPTH OF CUT 0.05)
N14 T02 (SEARCH FOR T02 IF NOT READY)
N15 M06 (T02 TO THE SPINDLE)
N16 G69 (CANCEL COORDINATE ROTATION IF ACTIVE)
N17 G90 G54 G00 X1.0 Y0 S2000 M03 T03 (XY START POSITION FOR THE CENTER OF POCKET 1)
N18 G43 Z1.0 H02 M08 (Z CLEARANCE FOR SETUP - COOLANT ON)
N19 G01 Z0.02 F30.0 (CONTROLS 0.005 LEFT ON THE POCKET BOTTOM)
N20 M98 P4252 L7 (ROUGH AND FINISH MILLING OF SEVEN POCKETS)
N21 G69 (CANCEL COORDINATE ROTATION IF ACTIVE)
N22 G90 G00 Z1.0 M09 (Z AXIS RETRACT - COOLANT OFF)
N23 G28 Z1.0 M05 (Z AXIS HOME FOR TOOL CHANGE)
N24 M01 (OPTIONAL STOP)

(T03 - 3/8 DIA CHAMFERING TOOL - 90 DEGREES)
N25 T03 (SEARCH FOR T03 IF NOT READY)
N26 M06 (T03 TO THE SPINDLE)
N27 G69 (CANCEL COORDINATE ROTATION IF ACTIVE)
N28 G90 G54 G00 X-2.5 Y-2.0 S4000 M03 T01 (XY START POSITION FOR PERIPHERAL CHAMFERING)
N29 G43 Z1.0 H03 M08 (Z CLEARANCE FOR SETUP - COOLANT ON)
N30 G01 Z-0.075 F50.0 (ABSOLUTE DEPTH FOR CHAMFERING Z-0.075)
N31 G41 X-2.0 D53 F12.0 (APPROACH MOTION AND RADIUS OFFSET)
N32 Y1.5 (CHAMFER LEFT EDGE)
N33 X2.0 (CHAMFER TOP EDGE)
N34 Y-1.5 (CHAMFER RIGHT EDGE)
N35 X-2.5 (CHAMFER BOTTOM EDGE)
N36 G00 G40 Y-2.0 (RETURN TO START POINT AND CANCEL OFFSET)
N37 Z0.1 (CLEAR ABOVE PART)
N38 X1.0 Y0 (MOTION TO THE CENTER OF POCKET 1)
N39 M98 P4254 L7 (CHAMFER SEVEN POCKETS)
N40 G69 (CANCEL COORDINATE ROTATION IF ACTIVE)
N41 G90 G00 Z1.0 M09 (Z AXIS RETRACT - COOLANT OFF)
N42 G28 Z1.0 M05 (Z AXIS HOME FOR TOOL CHANGE)
N43 X-2.0 Y8.0 (PART CHANGE POSITION)
N44 M30 (END OF MAIN PROGRAM O4202)
%
```



## 43

## SCALING FUNCTION

Normally, a programmed tool motion for a CNC machining center represents drawing dimensions, perhaps with cutter radius offset in effect. Occasionally, there may be a situation when the machining tool path that had already been programmed once must be repeated, but machined as *smaller* or *larger* than the original, yet still keep it proportional at the same time. To achieve this goal, a control feature called the *Scaling Function* can be used. Note the following two important items:

- Scaling function is an option on many controls and may not be available on every machine
- Some system parameters may be used for this function as well

For even greater flexibility in CNC programming, this optional scaling function can be used together with other programming functions, namely with *Datum Shift*, *Mirror Image* and *Coordinate System Rotation* - subjects described in the last three chapters.

## DESCRIPTION

CNC machine system can apply a specified scaling factor to all programmed motions, which means the programmed value of all axes will change. Scaling process is nothing more than multiplying the programmed axis value by the scaling factor, based on a defined scaling center point. CNC programmer must supply both the *scaling center* and the *scaling factor*. Through a control system parameter, scaling can be made effective or ineffective for each of the three main axes, but not for any additional axes. The majority of scaling is applied to the X and Y axes only.

It is important to realize that certain settings and preset amounts are *not* affected by the scaling function, namely various offsets. The following offset functions will *not* be changed if the scaling function is active:

- Cutter radius offset amount ... G41-G42 / D
- Tool length offset amount ... G43-G44 / H
- Tool position offset amount ... G45-G48 / H

In fixed cycles, there are two additional situations also *not* affected by the scaling function:

- X and Y shift amounts in G76 and G87 cycles
- Peck drill depth Q in G83 and G73 cycles
- Stored relief amount for G83 and G73 cycles

### ◆ Scaling Function Usage

In machine shop work, there are many applications for scaling an existing tool path. The result could be many hours of extra work saved. Here are some typical possibilities when a scaling function can be beneficial in CNC:

- Similar parts in terms of their geometry
- Machining with built-in shrinkage factor
- Mold work
- Imperial to metric and metric to imperial conversion
- Changing size of engraved characters

Regardless of application, scaling is used to make a new tool path larger or smaller than the original one. Scaling is therefore used for *magnification* (increasing size) or *reduction* (decreasing size) of an existing tool path - *Figure 43-1*.

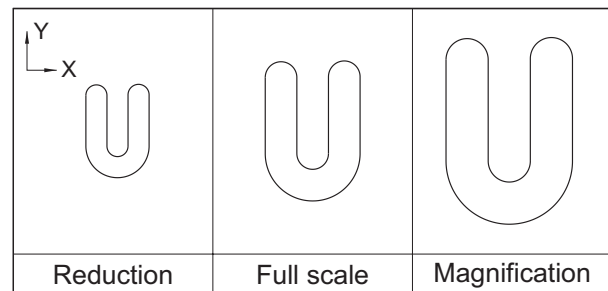


Figure 43-1

Comparison of a part reduction (left) and part magnification (right) also showing a part in full scale (middle)

## PROGRAMMING FORMAT

To supply the control unit with required information, programmer must provide the following data of information:

- Scaling center ... Pivot point (scaling origin)
- Scaling factor ... Reduction or Magnification

Two typical preparatory commands for scaling function are G51, canceled by another command, G50:

G50	Scaling mode cancel	Scaling OFF
G51	Scaling mode active	Scaling ON



Scaling function uses the following program format:

G51 I.. J.. K.. P..

where ...

- I = X coordinate of the scaling center (absolute)
- J = Y coordinate of the scaling center (absolute)
- K = Z coordinate of the scaling center (absolute)
- P = Scaling factor (0.001 or 0.00001 increment)

For best results, the G51 command should always be programmed in a separate block. Commands related to the machine zero return, namely G27, G28, G29 and G30 should always be programmed in scaling *OFF* mode. If G92 command is used for position register (older controls only), make sure it is also programmed in scaling *OFF* mode. Cutter radius offset G41/G42 should be canceled by G40 *before* scaling function is activated. Other commands and functions can be active, including all work offsets commands G54 through G59 as well as the additional ones.

### ◆ Scaling Center

Scaling center determines the location of the scaled tool path

Some high end Fanuc controls use I/J/K to specify the center point of scaling in X/Y/Z axes respectively. These values are always programmed as absolute values. As the center point controls the location of the scaled tool path, it is important to know one major principle:

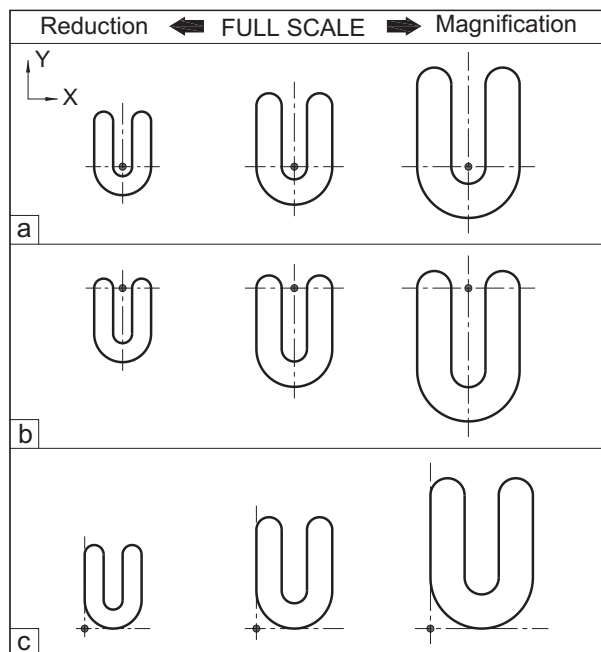


Figure 43-2

Comparison of scaled part location based on the scaling center

The scaled part will always *expand away from* and *reduce towards* the scaling point equally along all axes, as illustrated in Figure 43-2.

In order to understand a contour shape that is somewhat more complex, compare both the original and the scaled contours shown as an overlap in Figure 43-3. There are two machine tool paths (A and B) and the scaling center C. Depending on the scaling factor value, the result will be either tool path A1 to A8 or path B1 to B8.

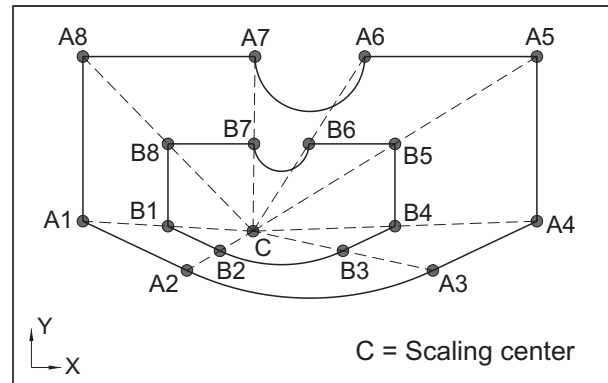


Figure 43-3

Effect of scaling point on the scaled part

Points A1 to A8 and points B1 to B8 in the illustration above represent contour change points of the tool path.

- ➡ If tool path A1 to A8 is the original path, then tool path B1 to B8 is the scaled tool path about center C, with a scaling factor LESS than 1
- ➡ If tool path B1 to B8 is the original path, then tool path A1 to A8 is the scaled tool path about center C, with a scaling factor GREATER than 1

All dashed lines connecting individual points are used for easier visualization of the scaling function. Starting from the scaling center C, each line always connects to the contour change point. The B point is always a midpoint between center point C and the corresponding point A. In practice, for example, it means that the distance between C and B5 and B5 and A5 is exactly the same.

### ◆ Scaling Factor

Scaling factor determines the size of scaled tool path

The maximum *scaling factor* is directly related to the smallest scaling factor. More advanced CNC systems can be set internally - through a system parameter - to preset the smallest scaling factor to either 0.001 or 0.00001. Some older models can only be set to 0.001 as the smallest scaling factor. Scaling factor is always independent of input units used in the program - G20 or G21.

When the smallest scaling factor is set to 0.001, the largest scale that can be programmed is 999.999. When the smallest scaling factor is set to 0.00001, the largest programmable scale is only 9.99999. Given the choice, the programmer has to decide between large scales, at the cost of precision or precision at the cost of large scales. For the majority of scaling applications, the 0.001 scaling factor is more than sufficient. Common terms for scaling factors are:

- Scaling factor > 1 ... Magnification
- Scaling factor = 1 ... No change - full scale
- Scaling factor < 1 ... Reduction

If the P address is not provided within G51 block, the system parameter setting will become effective by default.

### ◆ Rounding Errors in Scaling

Any conversion process should always be expected to return some inaccuracies, mainly due to rounding. For example, imperial-to-metric conversion uses the standard multiplying factor of 25.4, which happens to be an *exact* conversion factor. In order to convert a programmed value of 1.5 inches to its equivalent in millimeters, the value in inches must be multiplied by the constant of 25.4:

$$\text{mm} = 1.5 \text{ inches} \times 25.4 = 38.1 \text{ mm}$$

In this case, the conversion is 100 percent accurate. Now, convert the value of 1.5625 inches to millimeters:

$$\text{mm} = 1.5625 \text{ inches} \times 25.4 = 39.6875 \text{ mm}$$

So far, there is no problem - the resulting metric value as shown is also 100 percent accurate - *within the four decimal places* for normal programming in imperial units.

Scaling from millimeters to inches is much different. The scaling factor for millimeters to inches (within a nine place accuracy) is 0.039370079. However, scaling factor may only be programmed with a three or five decimal place accuracy. That means *rounding* the scaling factor will result in an inaccurate conversion. In many cases, the rounded result will be quite acceptable, but it is very important to consider the possibility of an error, in case it does matter.

Compare the error amount with different rounded scaling factors for 12.7 mm, which equals exactly to 0.500 inch:

➡ Using 0.001 minimum scaling factor:

$$\begin{aligned} \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.039 && \dots \text{preferred} \\ &= 0.4953 \text{ inches} && \dots \text{error of } 0.0047 \\ \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.038 && \\ &= 0.4826 \text{ inches} && \dots \text{error of } 0.0174 \\ \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.040 && \\ &= 0.5080 \text{ inches} && \dots \text{error of } 0.0080 \end{aligned}$$

➡ Using 0.00001 minimum scaling factor:

$$\begin{aligned} \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.03937 && \dots \text{preferred} \\ &= 0.499999 \text{ mm} && \dots \text{error of } 0.000001 \\ \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.03938 && \\ &= 0.500126 \text{ mm} && \dots \text{error of } 0.000126 \\ \text{mm} > \text{Inch} &= 12.7 \text{ mm} \times 0.03936 && \\ &= 0.499872 \text{ mm} && \dots \text{error of } 0.000128 \end{aligned}$$

These examples are rather extreme applications. If a 5% shrinkage factor is to be applied, for example, the scaling factor of 1.05 (magnification) or 0.95 (reduction) is well within the expected accuracy of the final part precision.

## PROGRAM EXAMPLES

This first scaling example is very simple - *Figure 43-4*.

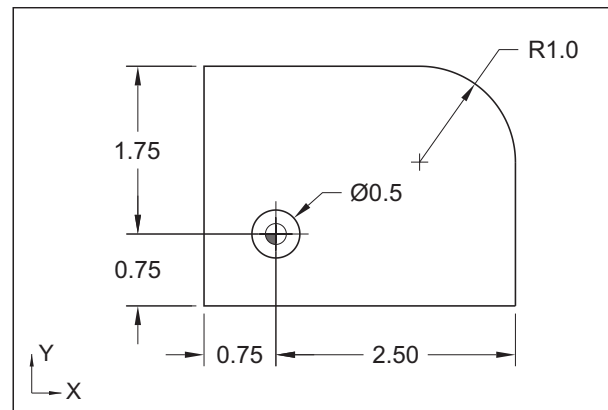


Figure 43-4

Drawing to illustrate scaling function - programs O4301 and O4302

Program O4301 is a basic contouring program, using a single cutting tool and only one cut around the part periphery. It is programmed normally, without any scaling.

### O4301 (BASIC PROGRAM USING G54 - NOT SCALED)

```

N1 G20
N2 G17 G40 G80
N3 G90 G00 G54 X-1.25 Y-1.25 S800 M03
N4 G43 Z1.0 H01 M08
N5 G01 Z-0.7 F50.0
N6 G41 X-0.75 D51 F25.0
N7 Y1.75 F15.0
N8 X1.5
N9 G02 X2.5 Y0.75 I0 J-1.0
N10 G01 Y-0.75
N11 X-1.25
N12 G40 Y-1.25 M09
N13 G00 Z1.0
N14 G28 Z1.0
N15 G28 X-1.25 Y-1.25
N16 M30
%
```

Program O4302 is a modified version of O4301. It includes a scaling factor value of 1.05 - or 5% magnification - and scaling center at X0Y0Z0. K0 can be omitted in G51.

```

O4302      (PROGRAM O4301 SCALED BY 1.05 FACTOR)
N1 G20
N2 G17 G40 G80
N3 G50                      (SCALING OFF)
N4 G90 G00 G54 X-1.25 Y-1.25 S800 M03
N5 G43 Z1.0 H01 M08
N6 G51 I0 J0 K0 P1.050      (FROM X0Y0Z0)
N7 G01 Z-0.7 F50.0
N8 G41 X-0.75 D51 F25.0
N9 Y1.75 F15.0
N10 X1.5
N11 G02 X2.5 Y0.75 I0 J-1.0
N12 G01 Y-0.75
N13 X-1.25
N14 G40 Y-1.25 M09
N15 G50                      (SCALING OFF)
N16 G00 Z1.0
N17 G28 Z1.0
N18 G28 X-1.25 Y-1.25
N19 M30
%
```

Program O4303 is more complex. Figure 43-5 represents the original contour. Figure 43-6 shows contour details with new scales and depth. Program starts with the smallest scale and works down. Note the very important blocks N712 and N713. Each contour must start from the original start point!

```

O4303 (MAIN PROGRAM)
(SCALING FUNCTION - VERIFIED ON YASNAC I80)
(T01 = 1.0 DIA END MILL)
N1 G20
N2 G50                      (SCALING OFF)
N3 G17 G40 G80 T01
N4 M06
N5 G90 G54 G00 X-1.0 Y-1.0 S2500 M03
N6 G43 Z0.5 H01 M08
N7 G01 Z-0.125 F12.0        (SET DEPTH)
N8 G51 I2.0 J1.5 P0.5       (0.5X AT Z-0.125)
N9 M98 P7001                (RUN NORMAL CONTOUR)
N10 G01 Z-0.25              (SET DEPTH)
N11 G51 I2.0 J1.5 P0.75     (0.75X AT Z-0.250)
N12 M98 P7001              (RUN NORMAL CONTOUR)
N13 G01 Z-0.35              (SET DEPTH)
N14 G51 I2.0 J1.5 P0.875    (0.875X AT Z-0.350)
N15 M98 P7001              (RUN NORMAL CONTOUR)
N16 M09
N17 G28 Z0.5 M05
N18 G00 X-2.0 Y10.0
N19 M30
%
```

```

O7001 (SUBPROGRAM FOR G51 SCALE)
(D51 = CUTTER RADIUS)
N701 G01 G41 X0 D51
N702 Y2.5 F10.0
N703 G02 X0.5 Y3.0 R0.5
N704 G01 X3.5
N705 G02 X4.0 Y2.5 R0.5
```

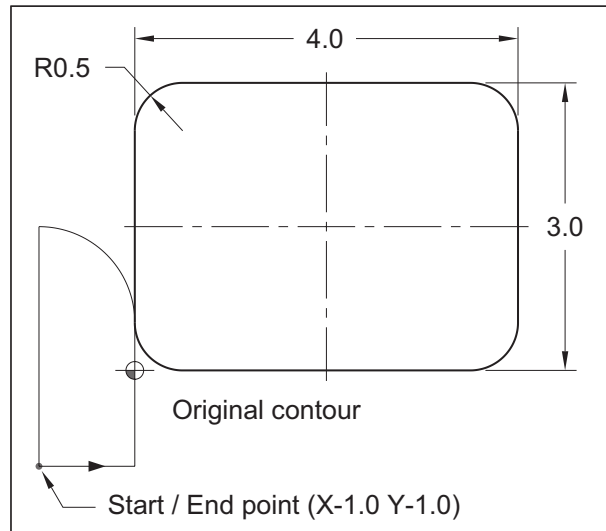


Figure 43-5

Original contour in full scale

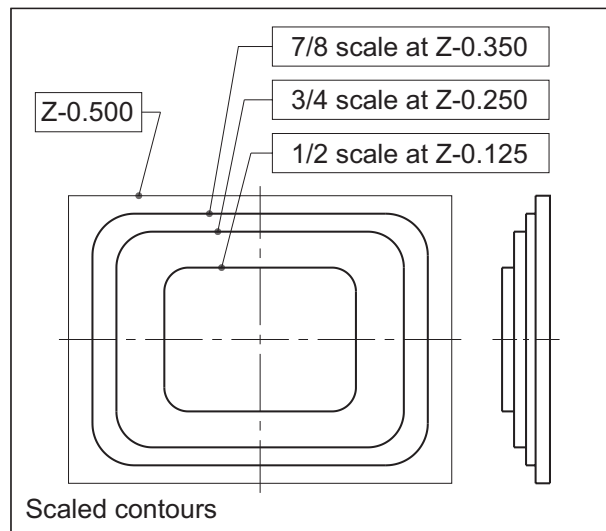


Figure 43-6

Scaled contours at three depths

```

N706 G01 Y0.5
N707 G02 X3.5 Y0 R0.5
N708 G01 X0.5
N709 G02 X0 Y0.5 R0.5
N710 G03 X-1.0 Y1.5 R1.0
N711 G01 G40 Y-1.0 F15.0
N712 G50                      (SCALING OFF)
N713 X-1.0 Y-1.0            (RETURN TO ORIGINAL START)
N714 M99
%
```

If available, scaling function offers many possibilities. Check all related control parameters and make sure the part program reflects the control settings. There are significant differences between various control models.

## 44

## CNC LATHE ACCESSORIES

Any CNC machine can be equipped with additional accessories, to make it more functional or functional in a particular way. In fact, most CNC machines have at least some additional accessories, either as a standard equipment or as an option. Machining centers have indexing and rotary tables, pallets, right angle heads, etc. All these are complex accessories and require a certain amount of time to understand them well. Apart from live tooling capabilities (described in a separate chapter), many CNC lathes are equipped with accessories that are usually quite simple to program. Some of the most noteworthy and typical programmable additions (or features) of this kind are:

- Chuck control
- Tailstock quill
- Bi-directional turret indexing
- Barfeeder

Other features may also be programmable as options:

- Parts catcher (unloader)
- Pull-out finger
- Tailstock body and quill
- Steady rest / follower rest
- Part stopper
- ... others as per machine design

Some of these accessories are fairly common, so it is worth looking at them in some detail and with a few examples of their programming applications.

## CHUCK CONTROL

In manual operations, a chuck, a collet or a special fixture mounted on the headstock of a lathe normally opens and closes when CNC operator presses a foot pedal. For safety reasons, a chuck that is rotating cannot be opened, because it is protected by an special safety interlock. Another important feature of chucks is that the terms *open* and *close* depend on the method of chucking - *external* or *internal*. A key switch is generally available to select one type of chucking or the other. *Figure 44-1* shows the difference.

Note that both terms *opened* and *closed* are relative to the setting of a toggle switch or a key switch, found on the machine itself, usually marked CHUCK CLOSED - that has two settings - IN and OUT.

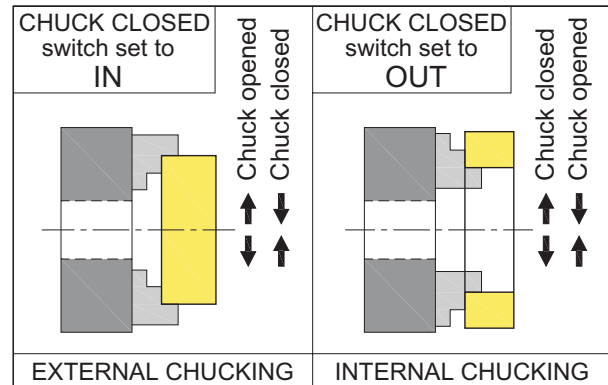


Figure 44-1

Part chucking - external and internal applications.  
Note the setting of the CHUCK CLOSED switch

In some applications, such as barfeeding, it is necessary to open and close the chuck under program control. Two M functions that control the chuck or collet opening and closing are normally available.

### ◆ Chuck Functions

Although the assigned numbers (normally miscellaneous functions) may vary for different machines, programming application is exactly the same. One function cancels the other. Typical M functions related to chuck control are:

M10	Open chuck
M11	Close chuck

➡ Example :

Typical programming procedure would include spindle stop and dwell:

```

M05          (STOP SPINDLE)
M10          (OPEN CHUCK)
G04 U0.1    (DWELL 1 SECOND)
M11          (CLOSE CHUCK)
M03          (RESTART SPINDLE)
  
```

This is a very simplified sequence, in which dwell is the time required for the bar (for example) to go through to the stop position. Some barfeeders do not require spindle to be stopped to feed the bar through and others have a special programming routine of their own.

M10 and M11 functions can also be used on the machine, during setup, using MDI setting mode in manual mode. Later in this chapter, M10 and M11 functions will be used for applications associated with barfeeding.

### ◆ Chucking Pressure

Amount of force required to clamp a part in the chuck is called the *chucking pressure*. On most CNC lathes the pressure is controlled by an adjustable valve, usually located in the tailstock area. Once the chuck pressure has been set, it is not changed very often. However, there are jobs that require the chucking pressure to be *increased* (tighter grip) or *decreased* (looser grip) frequently, usually within the same operation. Such special jobs will benefit from a programmable chuck pressure control.

Only a few CNC lathe manufacturers offer a programmable chucking pressure. If they do, it is in the form of two *non-standard* miscellaneous function, for example:

M15	Low chucking pressure
M16	High chucking pressure

Typically, part has to be reclamped in the chuck before either function can replace the other, which may disturb its position in the holding device. If chucking pressure feature is present on the lathe, read any related documentation supplied by the lathe manufacturer.

When changing chucking pressure  
- manually or through the program -  
always make sure the part is safely clamped.

### ◆ Chuck Jaws

This section is not directly related to programming, but covers a few tips useful to the programmer. Most chucks have three jaws, spaced 120° apart - see Figure 44-2.

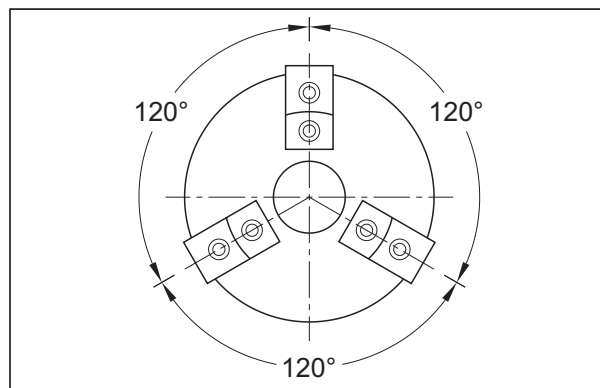


Figure 44-2

Typical three-jaw chuck for a CNC lathe

Chuck jaws may be *hard* (usually serrated for better grip) or *soft*, normally bored by CNC operator to suit the work diameter. Only soft jaws can be modified.

There is not much that can be done with hard jaws, except purchasing a suitable type for external or internal grip. Soft jaws are designed to be *bored* and the ability to do that is one of the basic skills a CNC operator must have. There are various techniques to bore soft jaws, all beyond the scope of this handbook. What is important is the understanding of what happens if jaws are not bored correctly.

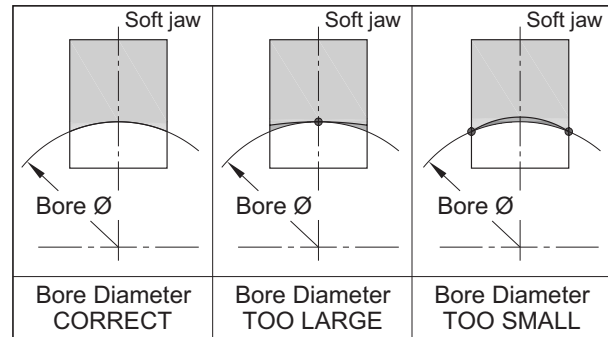


Figure 44-3

Soft jaws diameter bored correctly (left) and incorrectly

Figure 44-3 shows three versions - one correctly bored jaws and two incorrectly bored jaws. In both incorrect versions, the grip, the concentricity, or both, may suffer.

## TAILSTOCK AND QUILL

Tailstock is a very common accessory on a CNC lathe. Its main purpose is to support a part that is too long, too large, or needs to be pressed extra firmly against the jaws, for example, in some rough turning operations. A tailstock may also be used to support a finishing operation of a thin tubular stock, or to support a part that has a shallow grip in the jaws, to prevent it from flying out. On the negative side, tailstock is usually in the way of tool motions, so make sure to avoid a collision. A typical tailstock has three main parts:

- Tailstock body
- Quill
- Center

All parts are important in programming and setup.

### ◆ Tailstock Body

Tailstock body is the heaviest part of lathe tailstock. It is mounted to the lathe bed, either manually during setup, or through a programmable option, hydraulically. Truly programmable tailstock is normally available only as a factory installed option and must be ordered at the time of machine purchase.



### ◆ Quill

Quill is the shiny cylinder that moves in and out of the tailstock body. It has a fixed range of travel, for example, a 75 mm (3 inch) travel may be found on medium size lathes. When tailstock body is mounted to the lathe bed in a fixed position, quill is moved *out* to support the part, or *in*, to allow a part change. Part itself is supported by a center, mounted in the quill.

### ◆ Center

Center is a device that is placed into the quill with a tapered end, held by a matching internal taper and is physically in contact with the part. Depending on design, if the tailstock has an internal bearing, a dead center can be used. If tailstock has no internal bearing, a live center must be used instead. Machined part has to be pre-centered (on the CNC lathe or before), using the same tool angle as the tailstock center - normally 60°. A typical tailstock is illustrated in Figure 44-4.

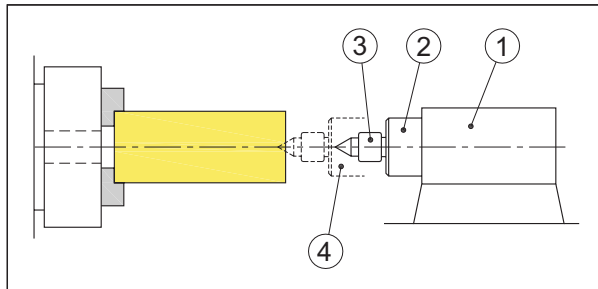


Figure 44-4

Typical tailstock for a CNC lathe:

- (1) Tailstock body (2) Quill - OUT (retracted for work change)  
(3) Center (4) Quill - IN (in work support position)

### ◆ Quill Functions

Programming tailstock quill motion only is just about the same for majority of CNC lathes. There are two miscellaneous functions that work the same way for a programmable and non-programmable tailstock body. These two typical functions are:

M12	Tailstock quill IN or ON = active
M13	Tailstock quill OUT or OFF = inactive

If quill is supporting the part, it is *in*, using the M12 function. If quill is not supporting the part, it is *out*, using the M13 function. For setup, the M12 and M13 functions may be used, and on many lathes, a toggle switch on the control is provided to operate the quill.

Spindle should be ON when the quill fully supports the part

### ◆ Programmable Tailstock

Tailstock body is normally not programmable (only the quill is), but this feature is available for many CNC lathes as a *factory installed option*. That means it has to be ordered when making the initial purchase; dealer cannot adapt such option to the machine at a later date. Many different types of programmable tailstocks are available, for example, a slide-type that moves left and right only, or a swing out type, that is out of the way when not needed.

A typical tailstock defined as *programmable* can be programmed using two *non-standard* M-functions. For the example, a CNC lathe may use these two M functions:

M21	Body of tailstock forward
M22	Body of tailstock backward

On some CNC lathes, there may also be two additional M functions available, one of them for *clamping* the tailstock, another for *unclamping* it. In many cases, the two tailstock functions have both clamp/unclamp functions built-in.

A typical programming procedure is to move tailstock towards the part, do some machining and move it back. Instead of presenting an actual programming example, let this procedure serve as a guide - fill-in any M-functions required for a particular CNC lathe:

1. Unclamp tailstock body
2. Move tailstock body forward
3. Clamp tailstock body
4. Move quill forward into the part
5. ... **do the required machining operations** ...
6. Move quill backward from the part
7. Unclamp tailstock body
8. Move tailstock backward
9. Clamp tailstock body

Some procedures take certain amount of time to complete, even if the time is measured in seconds. It is generally recommended to program a dwell function to guarantee completion of one step, before the next step starts. A review of Chapter 24 may help - see page 175.

### ◆ Safety Concerns

When programming a job that uses tailstock, safety is at least as important as for other operations. Tool motions programmed towards the part at start, and tool motions returning to tool change position later are both critical. The safest is an approach from tool change position towards the part along Z-axis first, *then* along X-axis. On return from a clear position close to work, reverse the order - first retract the X-axis above the part, *then* move Z-axis (both axes usually move to a safe tool change position).

## BI-DIRECTIONAL TURRET INDEXING

Another efficiency feature is a *bi-directional turret indexing*. Many CNC lathes have a so called *bi-directional indexing* built-in, that means an automatic method of turret indexing (control decides the direction). However, there is a certain benefit in having a *programmable* indexing direction. If that feature is available on a CNC lathe, there will be two miscellaneous functions available to program turret indexing. Both functions are *non-standard*, so check the machine tool manual, as always in such cases.

Typical M-functions for turret indexing are:

M17	Indexing forward: T01-T02-T03 . . .
M18	Indexing backward: . . . T03-T02-T01

Figure 44-5 shows an example of M17 and M18 functions for an eight-sided turret.

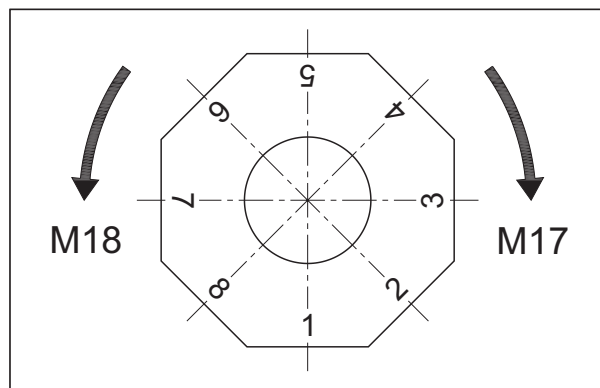


Figure 44-5

Programmable bi-directional turret indexing

In an example, programmer is working with a lathe that has an eight station turret. Tool T01 will be used first, then tool T08 and then back to tool T01 again. There is no problem to index from T01 to T08 *or* from T08 to T01, using the *automatic* turret indexing direction. It makes sense, that a bi-directional turret indexing should be used for efficiency. After all, T01 and T08 may be far apart in numbers but they are next to each other on a polygonal turret with eight stations. Control system will *always choose the shortest method*, in this case, from T01 to T08 in backward direction, then from T08 to T01 in forward direction.

If automatic bi-directional indexing is not built in the machine, it has to be programmed, assuming the control allows that. Otherwise, in normal programming, when going from T08 to T01, indexing motion will *pass all other six stations*, which is rather an inefficient method. The next example shows *how* and *where* to place the M-functions.

### ◆ Programming Example

This example is a complete program incorporating the bi-directional indexing and also shows how to use a fully programmable tailstock. All tool motions are realistic but not important for the example. *Order of numbering the tools in the turret may not be consistent* from one machine to another! Terms *forward* and *backward* are related to such order. M-functions described earlier are used here:

```

O4401 (BI-DIRECTIONAL INDEXING AND TAILSTOCK)
N1 G21 G99 M18 (SET INDEX BACKWARD)
N2 G50 S1200 (LIMIT MAX RPM)
N3 T0100 (SHORT FROM T02 TO T01 WITH M18)
N4 G96 S500 M03
N5 G00 G41 X98.0 Z5.0 T0101 M08
N6 G01 Z0 F0.75
N7 X-1.8 F0.18 (FACE CUT)
N8 G00 Z5.0
N9 G40 X250.0 Z150.0 T0100
N10 M01

N11 T0800 (SHORT FROM T01 TO T08 WITH M18)
N12 G97 S850 M03
N13 G00 X0 Z6.5 T0808 M08
N14 G01 Z-9.7 F0.125 (#5 CENTER DRILL)
N15 G04 U0.3
N16 G00 Z6.5
N17 X300.0 Z75.0 T0800
N18 M05 (SPINDLE STOP FOR TAILSTOCK)
N19 M01 (OPTIONAL STOP)

N20 M21 (TAILSTOCK FORWARD)
N21 G04 U2.0 (2 SECOND DWELL)
N22 M12 (QUILL IN)
N23 G04 U1.0 (1 SECOND DWELL)

N24 G50 M17 (NO MAX RPM - SET INDEX FORWARD)
N25 T0100 (SHORT FROM T08 TO T01 WITH M17)
N26 G96 S500 M03
N27 G00 G42 X79.5 Z2.5 T0101 M08
N28 G01 X87.0 Z-1.25 F0.2 (ROUGH CHAMFER)
N29 Z-65.0 F0.3 (ROUGH TURN)
N30 U0.2
N31 G00 G40 X250.0 Z150.0 T0100
N32 M01 (OPTIONAL STOP)

N33 T0200 (SHORT FROM T01 TO T02 WITH M17)
N34 G96 S600 M03
N35 G00 G42 X77.5 Z2.5 T0202 M08
N36 G01 X85.0 Z-1.25 F0.1 (FINISH CHAMFER)
N37 Z-65.0 F0.15 (FINISH TURN)
N38 U0.2 F0.4
N39 G00 G40 X300.0 Z150.0 T0200
N40 M05 (SPINDLE STOP FOR TAILSTOCK)
N41 M01 (OPTIONAL STOP)

N42 M13 (QUILL OUT)
N43 G04 U1.0 (1 SECOND DWELL)
N44 M22 (TAILSTOCK BACKWARD)
N45 G04 U2.0 (2 SECOND DWELL)
N46 M30 (END OF PROGRAM)
%
```

This example first uses T01 to face stock to the spindle center line. Then T08 comes in - center drill #5 - and makes a center hole. When the center drill moves into a clear position, tailstock body moves forward and locks, then quill moves into the work. T01 comes back to rough out the chamfer and diameter, after which T02 comes to finish them. When finishing operation is complete, spindle stops, quill moves out, then the tailstock body moves backward. CNC operator always sets the tailstock initial position.

At the job end, T02 is in active position. That means M18 has to be programmed at the program *beginning*, to get a short indexing from T02 to T01 - **a very important step!**

Noet how M17/M18 functions are programmed - their location in a particular block is very important. Either function programmed by itself will not cause turret to index - it only *sets* its direction! *Txx00 will make the actual indexing.*

All this leads to one question - can we find out if a particular CNC lathe has built-in *automatic* indexing direction (shortest direction) or *programmable* direction? A good chance is that on CNC lathes where only the *forward* direction takes place (no automatic indexing), there is also a feature called *programmable direction*, available in the form of M17 and M18 miscellaneous - or similar - functions.

Although the tendency on modern CNC lathes is to incorporate automatic turret indexing direction into the control system (which means the *control system is in charge*), there could be major benefits in having the programmable method available for special machining occasions. As an example, think of an oversize tool mounted in the turret. Such tool is perfectly safe to use, as long as it *does not* index the full swing of turret. Automatic indexing has *no provision* for such a situation!

On the other hand, with programmable indexing, CNC programmer has complete control. Programming a setup in a way that will never cause the turret to index full 360° at any time is possible. This may not always be a typical situation - it will take a few seconds extra time, but it can happen quite often for certain type of jobs.

## BARFEEDER ATTACHMENT

Barfeeder is an external attachment to a CNC lathe that allows small and medium cylindrical parts to be machined without interruption, up to a specified number that can be machined from a single bar of certain length. There are many advantages of using barfeeders, particularly those of modern hydrodynamic design type, rather than the old mechanical design. For example, sawing operations are eliminated (replaced with a much more precise part-off tool), no soft jaws to bore, unattended operation is possible (at least for an extended period of time), stock material economy and high spindle speeds can be achieved on many models with many other advantages.

Bars of material are stored in a special tube that guides the bar (by *pushing* it or *pulling* it) from the tube to the machining area. Two limitations are bar length and bar diameter. They are always specified by the barfeeder manufacturer and as spindle bore diameter of CNC lathe.

Many ingenious designs of barfeeders do exist nowadays and the programming method is heavily influenced by their design and features.

Functions controlling the chuck opening and closing, block skip function, M99 miscellaneous function and several other special functions, are typical features available for programming barfeeders. Many of these functions had been discussed earlier.

### ◆ Bar Stopper

Although a bar movement from the guide tube is controlled by the *chuck open* and *chuck close* functions (typically M10 and M11), the bar target position still has to be provided, in terms of how far it has to move out of the guide tube. This position should be lower than the bar diameter and on the positive side of Z axis (0.025 shown). This is the amount to be faced off (Z0 at the front face assumed). *Figure 44-6* shows such example.

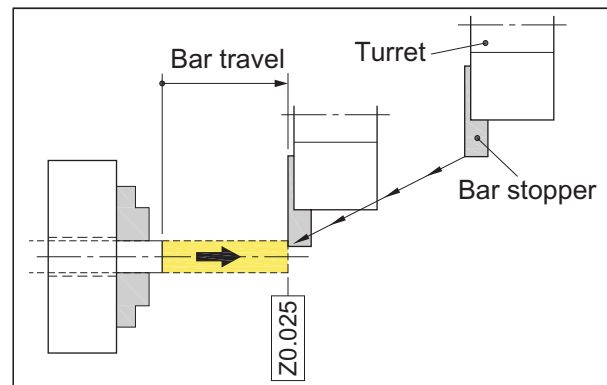


Figure 44-6

Bar stopper position for bar travel

Program itself is quite simple. It will use both M10 and M11 functions, but also another two functions that *may or may not be required* for a particular barfeeder. These *non-standard* miscellaneous functions are (in the example):

M71	Barfeeder ON - start
M72	Barfeeder OFF - stop

These functions are only examples and may be different for a certain barfeeding mechanism or unnecessary altogether. Here is the sample program with explanations:

```

O4402
N1 G21 T0100 M05          (T01 IS BAR STOPPER)
N2 G00 X3.0 Z0.6 T0101    (STOP POSITION)
N3 M10                    (CHUCK OPEN)
N4 G04 U1.0                (1 SECOND DWELL)
N5 M71                    (BARFEEDER ON)
N6 G04 U2.0                (2 SECOND DWELL)
N7 M11                    (CHUCK CLOSE)
N8 G04 U1.0                (1 SECOND DWELL)
N9 M72                    (BARFEEDER OFF)
N10 G00 X250.0 Z125.0 T0100 (CLEAR POSITION)
N11 M01                    (OPTIONAL STOP)

```

A few important notes relating to a bar stopper may be helpful to develop a better part program:

- Tool station #1 (T01) holds the bar stopper (N1)
- Initially, chucking of the bar (for each first piece from the bar) is done manually
- Spindle rotation must be stopped prior to chuck opening
- All miscellaneous functions related to barfeeding should be programmed as separate blocks
- Dwell should be sufficient for the task but not excessive

These are some general considerations for programming a bar stopper, but always check the recommended procedures for a particular barfeeder design.

## ADDITIONAL OPTIONS

There are many other options (non-standard features) on CNC lathes that will qualify as programmable accessories. Some maybe be rather unusual, such as a programmable *chip conveyor*, or programmable *tailstock pressure*, others may not be that rare, for example, a programmable *follower rest* (a moving version of a steady rest), used for ahead-of-tool support for long parts. Steady-rest and a follower-rest help prevent chatter or deflection on a relatively long part or a part with thin walls.

Another two common accessories that are also often related to each other - *and to barfeeding as well* - are:

- Part Catcher *also known as* Part Unloader
- Pull-Out Finger

Both are commonly used together with barfeeding operations and use two miscellaneous functions.

### ◆ Part Catcher or Part Unloader

A very common accessory for continuous machining, using a barfeeder, is *part catcher* or *part unloader*, as it is sometimes called. Its purpose is to catch the completed part after it had been parted-off. Instead of letting the completed part fall into the machine area and possibly cause damage,

this attachment will safely intercept each part and move it into a receiving box. Receiving box is often in the area of CNC lathe where machine operator can reach without danger, and without having coolant in the way. There are two *non-standard* miscellaneous functions for a part catcher:

M73	Part catcher advance	... in or forward
M74	Part catcher retract	... out or backward

Following program example illustrates how each function is programmed for a part-off tool.

```

O4403
N1 G21                      (TOP OF PROGRAM)
...
N81 T0700                  (PART-OFF TOOL ACTIVE)
N82 G50 S1500              (LIMIT MAXIMUM RPM)
N83 G96 S125 M03           (SPINDLE SPEED)
N84 G00 X53.0 Z-67.0 T0707 M08 (START POS.)
N85 M73                    (PART CATCHER ADVANCE)
N86 G01 X-0.25 F0.1        (PART-OFF CUTTING)
N87 G00 X53.0 M09          (MOVE ABOVE STOCK DIA)
N88 X250.0 Z125.0 T0700    (SAFE XZ POSITION)
N89 M74                    (PART CATCHER RETRACT)
N90 M01                    (OPTIONAL STOP)
/ N91 M30                  (CONTROLLED END OF PROGRAM)
N92 M99                    (RESTART FROM THE TOP OF PROGRAM)
%

```

T07 in the program is a 3 mm wide part-off tool, parting off a Ø50.0 stock diameter to 64 mm length, using standard simple process. Program shows special programming technique, relating to continuous operation. Concentrate on the last three blocks, N90, N91 and N92.

### ◆ Continuous Operation

Block N90 is an optional stop, typically used for setup and random checking. Block N91 contains M30 - end of program function. Note the slash symbol in front of the block. This is a *block skip* function, described earlier in *Chapter 23*. When block skip switch on the control panel is set to ON position, control system will *not* process the instructions in block N91. That means the program will not end there and processing will continue to block N92, where M99 is programmed.

Although M99 function is mainly defined as the end of a subprogram, it can also be used in the main program - as in this example. In that case, it causes a *continuous* processing loop. M99 function will make the program to return to the top, and - *without interruption* - repeat all processing again. Since the first tool will normally have a bar stopper programmed, barfeeder moves the stock out of the tube and the whole program repeats indefinitely - well, until the block skip switch is set to OFF position. Then, M30 function takes over and M99 in the subsequent block will *not* be processed, as the program has ended.



### ◆ Parts Counter

This kind of unattended lathe machining often uses another control system feature - *parts counter*. Parts may be counted via a program (usually a user macro), or by setting the number of required parts in control system memory. Counters may also be programmed by *non-standard* miscellaneous functions, for example:

M88	Count up	... <i>ascending order</i>
M89	Count down	... <i>descending order</i>

Preset number for the count is usually the bar capacity or the required number of parts from a single bar. Programming example at the end of this chapter will illustrate the counter function and some other features.

### ◆ Pull-Out Finger

As the name suggests, a pull-out finger is a device (CNC lathe accessory) that *grabs and pulls* the bar out of barfeeding guide tube (while the chuck is open). This is a typical method for barfeeders of the '*pull-type*'. Normally, pull-out finger is mounted in the turret, either as a separate 'tool', or as an add-on to an existing tool, in order to preserve the number of available tool stations. Since these activities cannot be used with spindle rotating, yet they often need a feedrate, they are programmed in the G98 mode - *feedrate per time* (mm/min or in/min).

Regardless of the exact pull-out finger model available, programming procedure is just about the same - no bar extends from the spindle longer than its face after part-off:

01. At a safe start position, index to the tool station where pull-out finger is mounted. Spindle must be stopped at this time with M05!
02. At a rapid rate, move to spindle centerline (X0), and a Z axis position about half-way of the overall bar projection
03. In 'feed-per-time' mode, feed-in towards the bar as extended after part-off
04. Dwell for about 0.5 second for the finger to catch the bar stock
05. Open the chuck with M10
06. Pull out bar stock from the guide tube
07. Dwell for about 0.5 second for the finger to complete the pull-out
08. Close the chuck with M11
09. Dwell for about 1 second to complete chuck closing
10. Move pull-out finger away from the bar stock
11. Return pull-out finger to the safe start position
12. Reinststate the 'feed-per-revolution' mode

In programming terms, the general structure will be similar to this format (item numbers correspond the list):

```
O4404
...
N.. ...
N.. Txx.. M05 (ITEM 01)
N.. G00 X0 Z.. (ITEM 02)
N.. G98 G01 Z.. F.. (ITEM 03)
N.. G04 U0.5 (ITEM 04)
N.. M10 (ITEM 05)
N.. G01 Z.. F.. (ITEM 06)
N.. G04 U0.5 (ITEM 07)
N.. M11 (ITEM 08)
N.. G04 U1.0 (ITEM 09)
N.. G00 Z.. (ITEM 10)
N.. X.. Z.. Txx00 (ITEM 11)
N.. G99 (ITEM 12)
N.. ...
```

Feel free to modify the program structure to suit the requirements of any unique setup in the machine shop.

## PROGRAMMING EXAMPLE

The following programming example illustrates a complete program for an unattended barfeeding operation, until the number of parts have been machined. CNC lathe operator sets the required number of parts when starting a new bar stock. This program requires a careful study. It does contain some very practical and advanced features, all of them already discussed, mostly in this chapter:

```
O4405
(N1 TO N18 FOR NEW BAR ONLY - 40 MM CUT-OFF)
N1 M18 (INDEX T03 TO T01)
N2 G21 T0100 M05 (T01 - BAR STOPPER)
N3 G00 X2.5 Z40.0 T0101 (NEW BAR OUT 40 MM)
N4 M10 (CHUCK OPEN)
N5 G04 U1.0 (1 SECOND DWELL)
N6 M71 (BARFEEDER ON)
N7 G04 U1.0 (1 SECOND DWELL)
N8 M11 (CHUCK CLOSE)
N9 X150.0 Z50.0 T0100 (CLEAR POSITION)
N10 M01 (OPTIONAL STOP)

N11 M17 (INDEX T01 TO T03)
N12 T0300 (T03 - 3 MM WIDE PART-OFF TOOL)
N13 G97 S1400 M03 (CUTTING RPM)
N14 G00 X32.0 Z0 T0303 M08 (START POSITION)
N15 G01 X-0.5 F0.1 (PART-OFF BAR BAD END)
N16 G00 X32.0 M09 (MOVE ABOVE BAR)
N17 X150.0 Z50.0 T0300 (CLEAR POSITION)
N18 M01 (OPTIONAL STOP)

N19 M18 (INDEX T03 TO T01)
N20 T0100 M05 (T01 - BAR STOPPER)
N21 G00 X2.5 Z1.0 T0101 (1 MM STOCK ON FACE)
N22 M10 (CHUCK OPEN)
N23 G04 U1.0 (1 SECOND DWELL)
N24 M71 (BARFEEDER ON)
N25 G04 U1.0 (1 SECOND DWELL)
```



N26 M11	(CHUCK CLOSE)	N41 T0300	(T03 - 3 MM WIDE PART-OFF TOOL)
N27 X150.0 Z50.0 T0100	(CLEAR POSITION)	N42 G97 S1400 M03	(CUTTING RPM)
N28 M01	(OPTIONAL STOP)	N43 G00 X32.0 Z-28.0 T0303 M08	(START POS.)
		N44 G01 X-0.5 F0.1	(PART-OFF TO 25 MM LENGTH)
N29 M17	(INDEX T01 TO T02)	N45 G00 X32.0	(MOVE ABOVE BAR)
N30 T0200	(T02 - FACE-CHAMFER-TURN OD)	N46 X150.0 Z50.0 T0300	(CLEAR POSITION)
N31 G96 S120 M03	(CUTTING SPEED)	N47 M01	(OPTIONAL STOP)
N32 G00 G41 X32.0 Z0 T0202 M08	(START FACE)	N48 M89	(UPDATE PART COUNTER BY 1)
N33 G01 X-1.8 F0.175	(FACE-OFF FRONT)	/ N49 M30	(CONTROLLED END OF PROGRAM)
N34 G00 Z2.5	(CLEAR Z+)	N50 M99 P19	(RESTART FROM BLOCK N19)
N35 G42 X17.5	(CHAMFER START)	%	
N36 G01 X24.0 Z-0.75 F0.08	(CUT CHAMFER)		
N37 Z-29.0 F0.12	(CUT DIAMETER TO LENGTH)		
N38 U5.0 F0.2	(CLEAR ABOVE BAR)		
N39 G00 G40 X150.0 Z50.0 T0200	(CLEAR POSITION)		
N40 M01	(OPTIONAL STOP)		

As it usually goes with accessories and options, the machine tool manufacturers use a number of M-functions to activate and deactivate a particular accessory. It is not possible to cover any specific procedures into a general reference material. Hopefully, ideas presented in this chapter will help to adapt to any manufacturer's recommendations and understand them better.

## 45

## HELICAL MILLING

Helical milling uses an optional control system feature called *helical interpolation*. In its simplest definition, helical interpolation is a machining operation where a circular interpolation uses *three axes simultaneously*. This could be a misleading statement because it implies a three dimensional arc or a circle. Such an arc or circle does not exist anywhere in the field of mathematics, it becomes a *helix*. Both G02 or G03 circular interpolation commands do use *all three axes* - for example,

G03 X.. Y.. Z.. .. F..

This type of operation is only available for CNC machining centers as an optional feature. Let's look at the subject of helical milling a little closer.

## HELICAL MILLING OPERATION

What exactly is helical milling? Essentially, it is a form of a circular interpolation - it is a programming technique to machine arcs and circles *combined* with a linear interpolation in the same block, during the same motion .

Previous topics that were related to circular interpolation presented one major feature of that subject. In circular interpolation, there are *two* primary axes used within the selected plane, with the intent to program an arc motion or a circular motion.

For example, in G17 XY plane (the plane that is most common), a typical format of circular interpolation will be in two forms:

➡ Using arc center vectors IJK for CW/CCW motion :

G02 X.. Y.. I.. J.. F..  
G03 X.. Y.. I.. J.. F..

➡ Using radius R for CW/CCW motion :

G02 X.. Y.. R.. F..  
G03 X.. Y.. R.. F..

Note that there is no Z-axis programmed. As a matter of fact, if the Z-axis were included in the same block as a circular milling, it will not work - *normally*. That means it will not work, *unless* the control system has a special feature called *helical interpolation* option.

### ♦ Helical Interpolation

Helical interpolation is usually a special control system option that is designed to be used for cutting a circle or an arc with a third dimension. This third dimension is always determined by the active plane:

- In G17 XY plane - the third dimension is the Z-axis
- In G18 ZX plane - the third dimension is the Y-axis
- In G19 YZ plane - the third dimension is the X-axis

In the active plane G17 (XY), the third dimension is the Z-axis. In the active plane G18 (ZX), the third dimension is the Y-axis and in the active plane G19 (YZ), the third dimension is the X-axis.

In all cases, the third dimension - *the third axis motion* - will *always* be a linear motion that is perpendicular to the active plane.

A more formal definition of a helical interpolation can be made, based on the previous statement:

Helical interpolation is a simultaneous **two-axis** circular motion in the working plane, with the linear motion along the **remaining** axis.

The resulting three axis motion is always synchronized by the control system and all axes always reach the target location at the same time.

### ♦ Programming Format

*General* formats for helical interpolation in a program are similar to formats available for a circular interpolation - plane selection is *particularly* important:

➡ Using arc center vectors IJK for CW/CCW motion :

G02 X.. Y.. Z.. I.. J.. K.. F..  
G03 X.. Y.. Z.. I.. J.. K.. F..

➡ Using radius R for CW/CCW motion :

G02 X.. Y.. Z.. R.. F..  
G03 X.. Y.. Z.. R.. F..

Plane selection programmed before helical interpolation block determines which axes will be active in the program and what their function will be.

### ◆ Arc Vectors for Helical Interpolation

Arc vector functions are programmed using the same principles as in circular interpolation but will be *different for each plane*. Here is a summary in a typical table:

Active Plane	Circular motion	Linear motion	Arc vectors
G17	X and Y	Z	I and J
G18	X and Z	Y	I and K
G19	Y and Z	X	J and K

Note that the arc vectors apply to the two axes that form the *circular motion* - linear motion has no influence whatsoever. If the control system supports the direct radius entry R (instead of the more traditional IJK vectors), the physical center of arc motion is calculated automatically, within the current plane.

### ◆ Applications and Usage

Although helical interpolation option is not the most frequently used programming method, it may be the only method available for a number of rather special machining applications:

- Thread milling
- Helical profiling
- Helical ramping

From these three groups, *thread milling* is by far the most common method of helical interpolation applied in industry and is described next in some practical detail. The last two applications are quite similar, although used less frequently and will be described later in this chapter as well.

## THREAD MILLING

There are two familiar methods of producing a thread on a CNC machine. On machining centers, the predominant method of thread generating is tapping, normally using fixed cycle G84 or G74. On CNC lathes, a tap is also used frequently (usually without the use of a cycle), but the majority of threads are machined by single point threading method, using block-by-block method of G32, simple cycle G92, and multiple repetitive cycle G76.

### ◆ Applying Thread Milling

There are many cases in manufacturing, where either tapping or single point threading method is impractical, difficult, or impossible in a given situation. Many of these difficulties can often be overcome by choosing *thread milling method* instead. Thread milling is definitely the most common industrial application of helical interpolation feature of the control system.

Thread milling can be used in programming to achieve special benefits. These benefits are quite numerous:

- A large thread diameter - virtually any diameter can be thread milled (with high concentricity)
- Smoother and more accurate thread generation (only thread grinding can be more accurate)
- Combination of thread milling within a single setup eliminates secondary operations
- Full depth thread can be cut
- Tap is not available
- Tapping is impractical
- Tapping is difficult and causes problems
- Tapping is impossible in hard materials
- Blind hole tapping causes problems
- Part cannot be rotated on a CNC lathe
- Left hand and right hand threading has to be done with one tool
- External and internal threading has to be done with one tool
- Thread deburring minimized or eliminated
- Gain of high surface finish quality, particularly in softer materials
- Extended life of the threading tool
- Elimination of expensive tapping heads
- Elimination of expensive large taps
- No need for spindle reversal (as in tapping)
- Better power rating of the tool versus the cut (about 1/5th is not unusual)
- One tool holder can accept inserts for different thread pitch size
- Reduction of overall threading costs

Thread milling only enhances other threading operations, it does not replace them. It uses special threading cutters, called *thread hobs*, or special multi-tooth *thread milling cutters*. In both cases, there is one common feature for both types of cutters - *pitch of the thread is built into the cutter*.

### ◆ Conditions for Thread Milling

For successful thread milling, three conditions must exist before writing a program:

- Control system must support the operation
- Diameter to be threaded must be premachined
- Suitable thread milling tool must be selected

All three conditions must exist simultaneously.

### ◆ Thread Milling Tool

Thread milling cutters are available in at least two varieties - some are made of solid carbide, some use carbide interchangeable inserts. In either design, the threading tool pitch must match the pitch of a thread required by the drawing. Tool has to be small enough to fit into the available internal space and large enough to guarantee suitable rigidity while cutting externally. For internal thread milling, cutters are available for thread milling in holes as small as 0.250 inch (6.35 mm).

Unlike a tap, thread milling tools do not have the helix angle built in, only the pitch. Helix angle is required for threading and is controlled during helical interpolation motion by linear movement. Typical thread milling tools are illustrated in *Figure 45-1*.

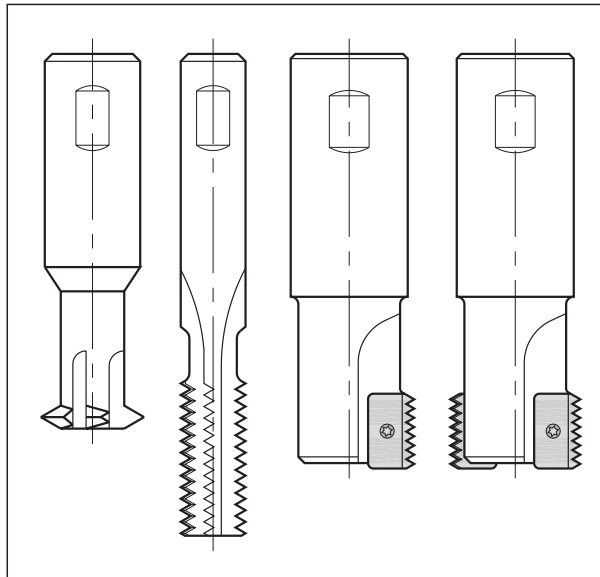


Figure 45-1

Typical thread milling cutters.

Solid carbide (left), single insert (middle) and a double insert (right)

### ◆ Premachining Requirements

A hole for a tap cannot have the same diameter as the tap itself. It has to be smaller to accommodate the *depth of the thread*. The same rule applies to helical milling:

- If a thread is milled on the inside diameter of the part (internally), the premachined diameter must be smaller than the nominal thread size
- If a thread is milled on the outside diameter of the part (externally), the premachined diameter must be equal to the nominal thread size

Either diameter (internal or external) may be slightly larger or slightly smaller than the 'normal' size, but this deviation is decided by the required 'fit' of the thread.

### ◆ Clearance Radius

Clearance radius protects the thread from damage by the cutting tool. Each cutting edge on the threading tool (hob) or indexable insert is ground with a *decreasing angle* in the direction of cut - this is called the *clearance angle*. This clearance angle guarantees smooth cutting conditions during thread milling.

### ◆ Productivity of Thread Milling

One of the reasons programmers choose thread milling operation could be the desire to improve machining productivity. There are many sizes of thread cutting tools available, with just about all pitch variations. In order to achieve the highest level of efficiency in thread milling, use a threading tool that is large enough to cut the required thread in a single revolution (in a 360° sweep). At the same time, the tool must have all necessary clearances.

A great deal of influence on thread milling productivity will be the total length of travel and the selection of cutting feedrates. A large diameter cutter can cut more efficiently (heavier feedrates), but cannot fit into confined areas. Small diameter cutter has the opposite effect - it can be used in a tight areas, but at lower feedrates. A smaller cutter may also be used with higher spindle speeds and the corresponding feedrate - the combined effect may shorten the cutting cycle time.

## THE HELIX

Words *helical* and *helix* are quite common in CNC programming and appear in this and other publications quite frequently. Perhaps it is time to look at the terms relating to thread milling in more detail.

The main word that is used in this context is the word *helix*. The word *helix* is based on the original Greek word for *spiral*. A dictionary definition gives us some clue as to its meaning - it suggests that a helix is anything in the shape of the thread of a screw. Helix is defined in the "*Machinery's Handbook*" by Industrial Press, Inc., New York, NY, USA, this way:

*"A helix is a curve generated by a point moving about a cylindrical surface (real or imaginary) at a constant rate in the direction of the cylinder's axis."*

This quite detailed definition means that the helix is a curve created by a circular motion of a point on a cylinder or a cone, combined with a simultaneous linear advance. A curvature of a common screw thread is a typical example of a straight helix.

A cutting tool motion based on the mathematical definition (using three axes), results in a helical motion, also known as helical interpolation.

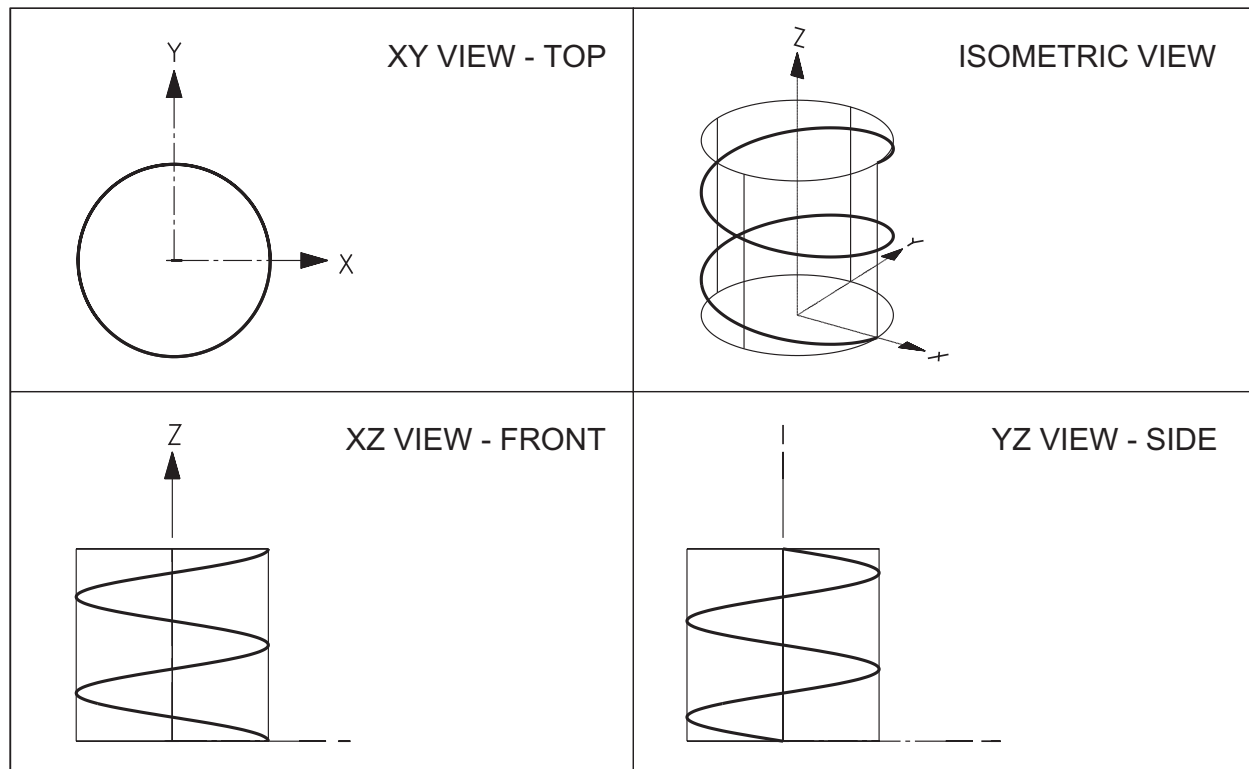


Figure 45-2

A typical helix shown in four standard views - two revolutions are shown between the top and the bottom of the helix

A helix is a machine motion that has *four* varieties:

- Clockwise circular cut with positive linear motion
- Clockwise circular cut with negative linear motion
- Counterclockwise circular cut with positive linear motion
- Counterclockwise circular cut with negative linear motion

Typical helix illustration in *Figure 45-2*, shows a common helix (which is a three-dimensional object) in four standard views. Helix is shown in these views:

- Top view (XY) shows only a circle
- Front view (XZ) shows the helix from the front
- Side view (YZ) shows the helix from the standard right side view
- Isometric view (XYZ) shows a three-dimensional appearance of a two-turn helix

Another view of a helix that is often very useful, is the *flat view* (also called the *flat layout*). This view is commonly used to illustrate a helix as a flat object that can wrap around a cylinder. *Figure 45-3* shows a flat layout of a right hand helix (one revolution).

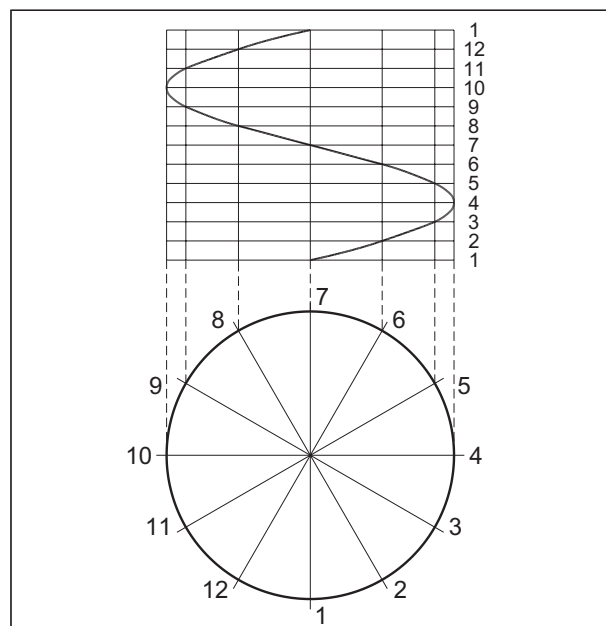


Figure 45-3

Flat view representation of a right-hand helix.  
One revolution of 360° is illustrated



## THREAD MILLING EXAMPLE

A thread milling operation on CNC machining centers can be programmed very efficiently by using helical interpolation feature of the control system. The easiest way to describe and explain the straight thread milling, is to show an illustrated example - *Figure 45-4*.

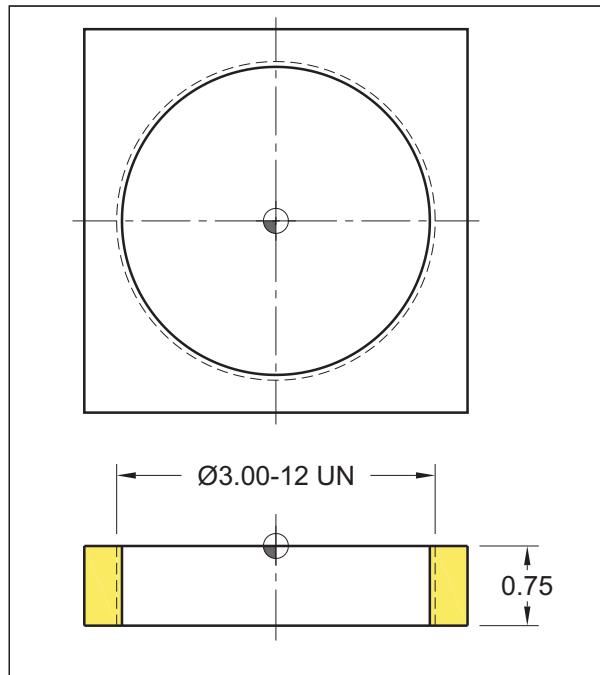


Figure 45-4

Internal thread milling example - program O4501

### ◆ Straight Thread

The following information contains collected initial data, based on the given drawing and the available tools:

1. Internal thread is 3.00 diameter - through the plate
2. Plate thickness is 0.75
3. 12 TPI = 12 threads per inch
4. 1.500 diameter thread hob
5. Tool T03 and offsets H03 and D03
6. Bored diameter is 2.9000 inches

This summary sets the stage for programming.

### ◆ Initial Calculations

In the example are six items to consider. Items 1, 2 and 3 were supplied by the drawing, but items 4, 5 and 6 were selected or calculated as part of the programming process. We look at the selected or calculated items individually.

*Item 4* is the threading cutter size. Two main characteristics of a threading cutter must be considered - its diameter and the *pitch* of cutting edges (teeth). Selecting a cutter diameter must be done carefully - it must be smaller than the *bored* diameter. Another challenge is to choose a thread milling cutter that has the correct number of teeth per inch (pitch). The thread mill diameter is more important for internal threads but the pitch of threading cutter must be maintained, regardless of whether the thread cutting is internal or external.

*Item 5* deals with the tool number and its related offset numbers. In this case, the tool number is 3, programmed as T03. Tool length offset number is H03 and cutter radius offset number is D03. D03 offset setting will contain the *radius* of threading cutter, in this case, its nominal value will be 0.6250. Offset numbers are arbitrary numbers for this example, others may be different. Just keep in mind that the diameter machined for an internal thread cutting *must* be smaller than the thread nominal size - just like predrilling a hole for tapping. That introduces the last item - *Item 6*.

*Item 6* lists the bored diameter as 2.9000 inches. Why this number and not other? Remember that the internal thread depth is established by a common formula. A generic formula to calculate the depth D of an internal thread multiplies the pitch by a constant (*see page 354* for details):

$$D = \text{PITCH} \times 0.54127$$

If the formula is applied to a Ø3.000, with 12 TPI thread ( $1/12 = 0.0833333$  pitch), the single depth of the thread is:

$$0.0833333 \times 0.54127 = 0.0451058$$

When the formula is applied to the calculation of a pre-bored diameter, this amount has to be *subtracted twice* for the required nominal diameter:

$$3.0000 - 2 \times 0.0451058 = 2.9097884$$

It follows that the bored diameter for the thread should be Ø2.9098 inches.

At this point, another consideration must be made. The threading *cutter* itself. Essentially, threading cutter is a forming tool. Its crest and its root will be formed on the finished thread. This feature presents a certain advantage. By programming the internal diameter a little smaller, the final size will be formed and result in a smooth surface finish. Leaving about 0.003 to 0.006 stock per side (0.07-0.14 mm) will do the trick. For the example, the 0.003 to 0.006 range is used and calculation of Ø2.9097884 can be rounded to an even Ø2.90, leaving only a small 0.0097884 stock on diameter, or 0.0048942 per side, for finishing. No doubt, the difference is reasonable, but it did take advantage of a rounding to a 'friendly' number, such as 2.9000.

### ◆ Starting Position

After all required data have been collected and properly calculated, another step can be made, this time to calculate the thread *start position*.

This is easy for X and Y axes - the center of thread diameter is as good start as any - better, in fact. In this example, and for simplicity, this XY position is also equivalent to the part origin X0Y0 position.

Start position of thread cutter measured along the Z-axis is much more important in helical milling than in any other type of milling. Z-axis start position must always be *synchronized* with the thread pitch, as the cutting will proceed in three axes simultaneously. Z-axis zero (Z0) will be at the top of part.

Z-axis start position is determined by several factors - *size of the thread mill* (in this case a tool with an indexable insert), *pitch of the thread* (in this case 0.0833333), *direction of the Z-axis motion* (up or down) and the *method of infeed* along XY axes.

When a thread is cut using the helical interpolation feature, all three axes used must be considered equally. Just like defining the approach arc for circular interpolation, the approach arc for a helical interpolation can be defined the same way - the procedure is exactly the same.

### ◆ Motion Rotation and Direction

In helical interpolation it is *extremely important* to coordinate, to synchronize, the following three program items:

- Spindle rotation
- Circular cutting direction
- Z-axis motion direction

Why are these three items so important? Why do they have to be coordinated at all? Evaluate them - one by one.

#### *Spindle Rotation*

Spindle rotation can be either M03 (clockwise) or M04 (counterclockwise).

#### *Circular Cutting Direction*

Circular direction follows the rules of circular interpolation - G02 is clockwise direction, G03 is counterclockwise direction.

#### *Z-Axis Motion Direction*

For vertical machining, Z-axis cutting direction may be along two directions:

- Up or positive
- Down or negative

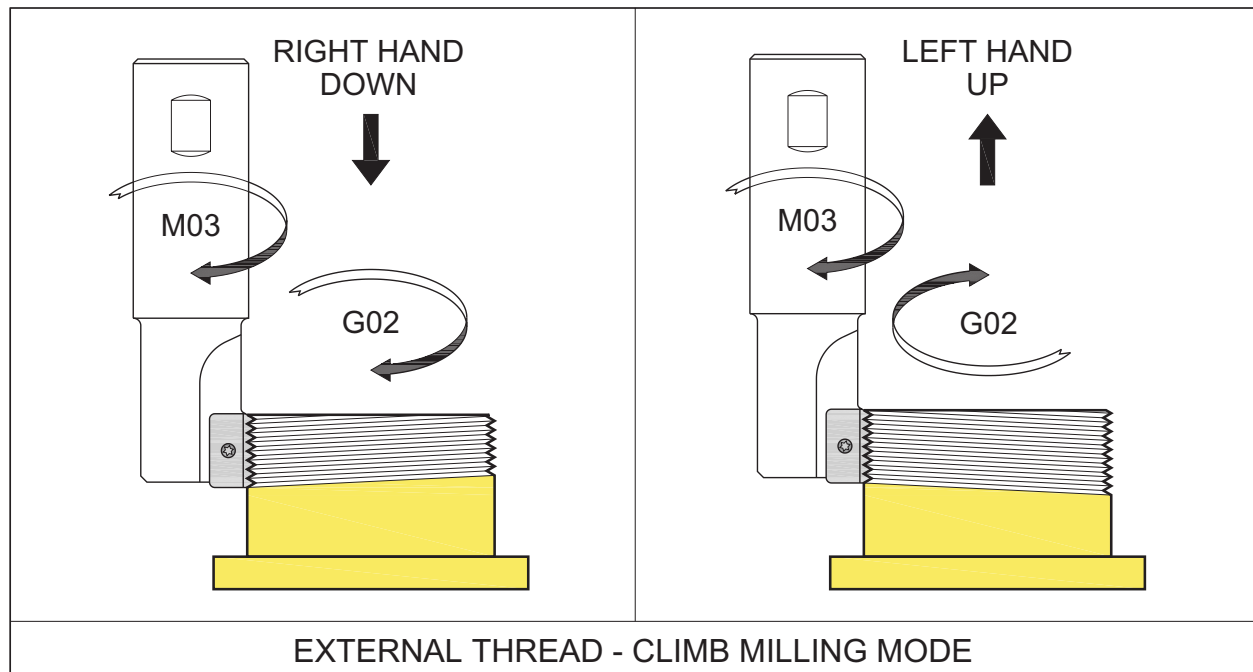


Figure 45-5

EXTERNAL thread milling using the climb milling mode - right and left hand threads, spindle rotation and cutter motions shown

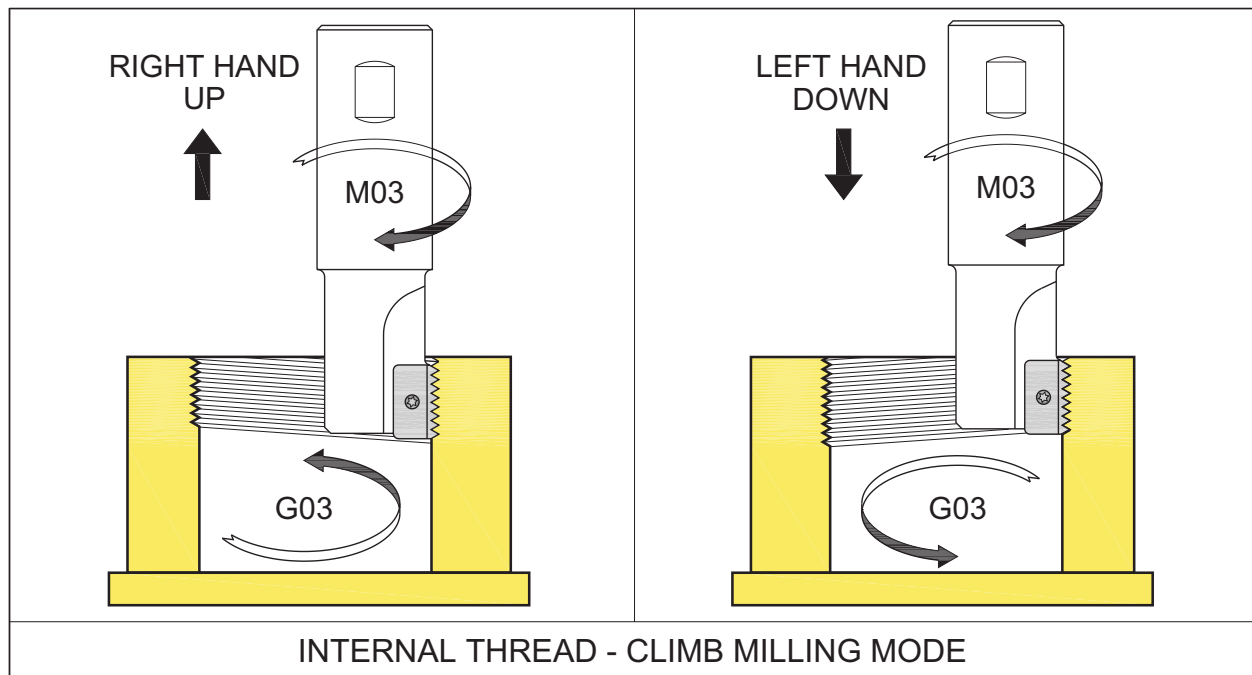


Figure 45-6

INTERNAL thread milling using the climb milling mode - right and left hand threads, spindle rotation and cutter motions shown

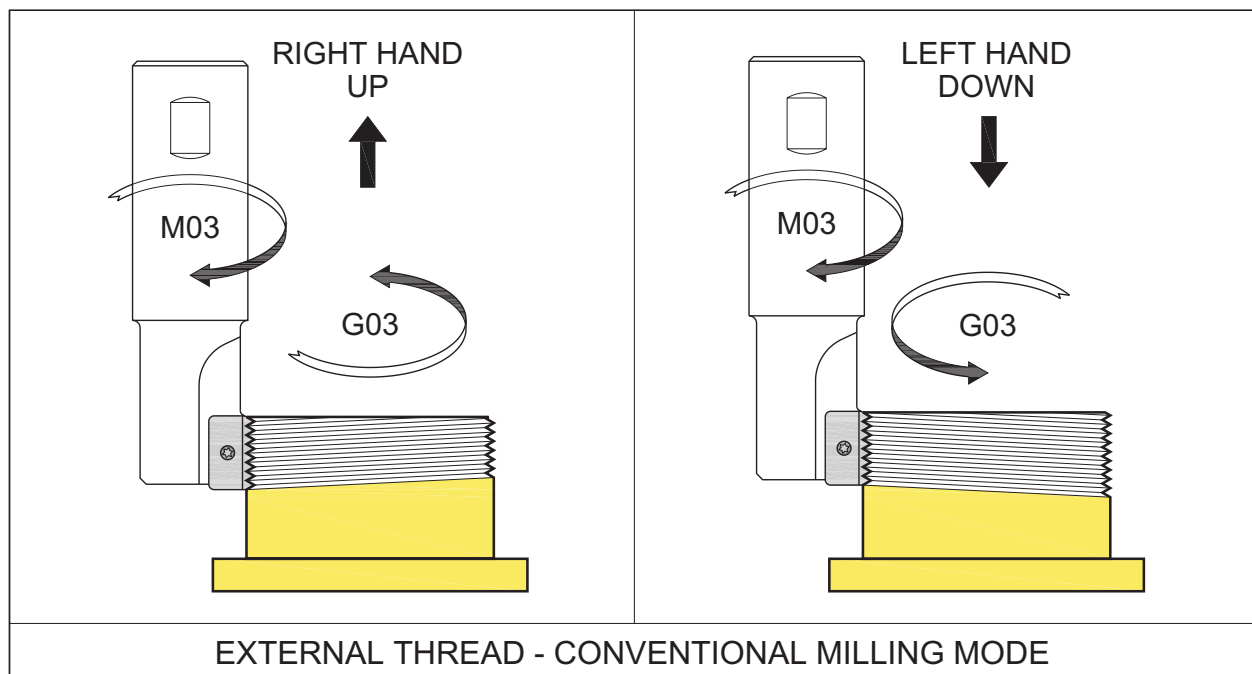


Figure 45-7

EXTERNAL thread milling using the conventional milling mode - right and left hand threads, spindle rotation and cutter motions shown

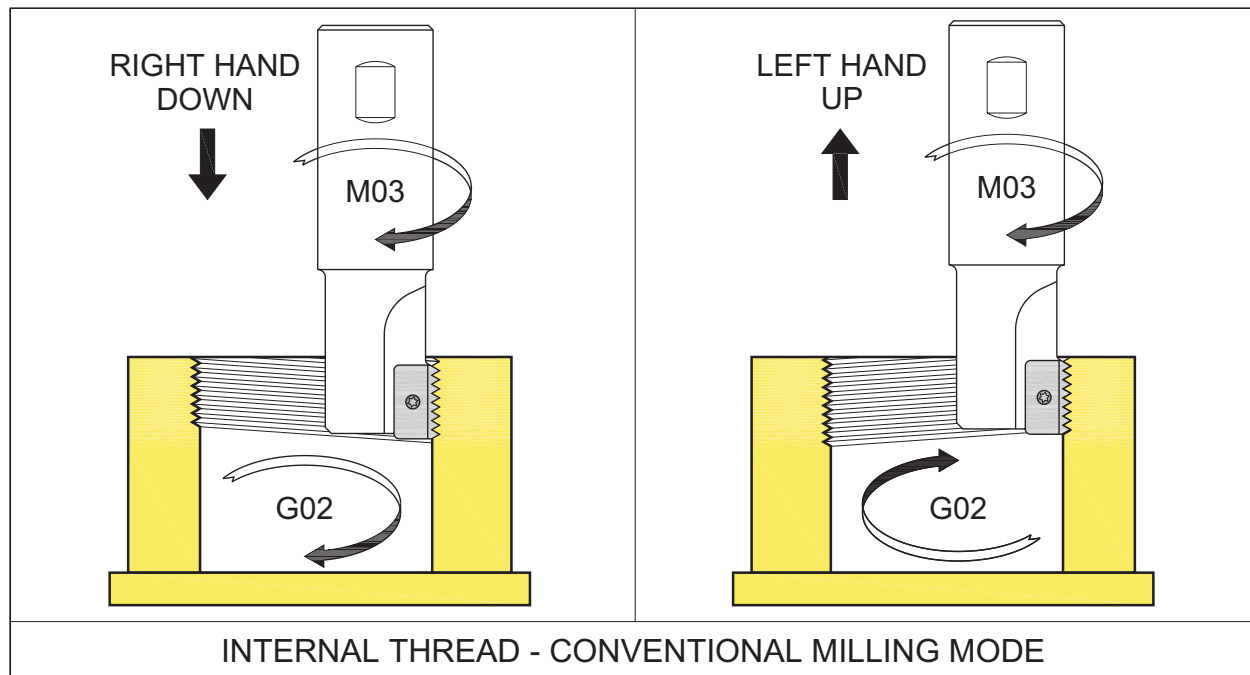


Figure 45-8

INTERNAL thread milling using the conventional milling mode - right and left hand threads, spindle rotation and cutter motions shown

Each motion item by itself is important, but it is the *coordination of all motions* that makes the thread to match engineering purposes. These motions together determine the *hand of thread* (left hand vs. right hand), and whether applied externally or internally. Figures 45-5, 45-6, 45-7, and 45-8 show all possibilities for various methods of thread cutting. Climb milling mode should always prevail.

### ◆ Lead-In Motions

In the example (Figure 45-4), thread to be milled is a *right hand thread* and an *internal* thread. Spindle rotation is normal (M03). Figure 45-6 (left) indicates that the thread has to be milled from the bottom upwards, with counter-clockwise (G03) tool motion.

There is one last consideration - the thread milling insert - mainly its height. Insert height determines how many revolutions are required to cut the thread at full depth. A single insert cutter will be used, and by consulting the tooling catalogue, determined that two revolutions will be sufficient to mill the required thread.

To start the thread milling example, the cutter has to be positioned at X0Y0 part origin and at a clear Z-depth. Since a multi-tooth insert cutter is used and there is space available, the start will be a little *below* the part bottom, say 0.200, at Z-0.95 (the plate thickness is 0.750, as per drawing). This extra clearance provides an even entry into the thread. The program start includes all current considerations:

### O4501 (INTERNAL RIGHT HAND THREAD MILLING)

```
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S900 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.95 F50.0
...
```

Similar to a program using circular interpolation, the next step to be done is determination of linear approach to the lead-in arc (in climb milling mode). This is also the motion that applies the cutter radius offset.

In helical milling, cutter radius offset applies only to the two axes of selected plane

In the example, radius offset is entered in block N6:

```
N6 G01 G41 X0.75 Y-0.75 D01 F10.0
```

The next block is a lead-in arc, with 0.750 approach radius. Only the motions along X and Y axes will be needed:

```
N7 G03 X1.5 Y0 R0.75 (or I0 J0.75)
```

Since only two axes are used, the motion is planar (on a plane). Figure 45-9 shows the tool motion. Note that the motion *appears* to be the same as for a circular interpolation (circular pocket application). This can be misleading - there is also the Z-axis involved!

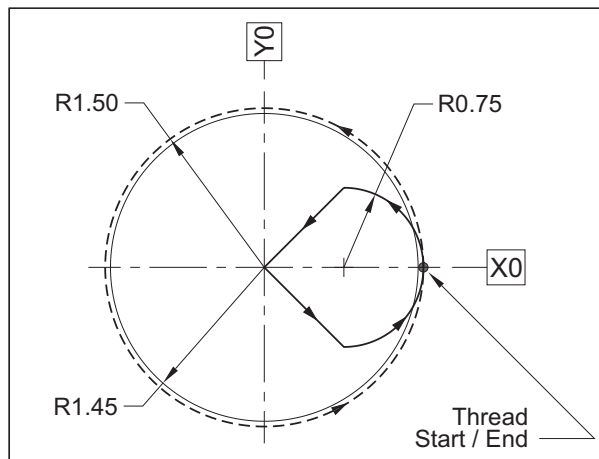


Figure 45-9

Lead-in and lead-out motions for thread milling example O4501  
(top view is shown)

However, this programming approach would bring the threading cutter into the material straight! Unlike a tap, thread milling cutter has all teeth *parallel*, so it would cut a series of grooves, not threads. This, of course, is an unacceptable situation.

To make a better cut, start with a *helical motion* for the lead-in arc. That means adding the Z-axis to the circular motion, in the upwards direction. Amount of the Z-target position must be *calculated*, not guessed. Helical approach has to consider both thread *pitch* and *degrees* of travel on the lead-in arc circumference.

Thread pitch in the example is:

$$1 / 12 = 0.0833333$$

and the degrees traveled on circumference total 90°, from X0.75Y-0.75 to X1.5Y0.

Considering that thread mill has to advance 0.0833333 inches for every 360°, it has to advance one quarter of that distance for each 90°.

The calculation of linear travel can be made from the following formula:

$$L_t = \frac{A \times P}{360}$$

where ...

- $L_t$  = Linear travel in helical interpolation
- $A$  = Amount of degrees interpolated (angle)
- $P$  = Thread pitch (1 / TPI)

Advance for 90° in the example will be:

$$L_t = 90 \times 0.0833333 / 360$$

$$L_t = 0.0208333 \quad (0.0208)$$

Cutting motion takes place along the positive Z-direction (up), so the target position absolute value will be *above* the start position, and a corrected block N7 can be written:

N7 G03 X1.5 Y0 Z-0.9292 I0 J0.75 (or R0.75)

At this point, the tool is in a position where the complete 360° helical motion can begin. Always try to start lead-in and lead-out arcs at quadrant positions (0°, 90°, 180° and 270°). These calculations are much easier to work with.

### ◆ Thread Rise Calculation

Some technical brochures or product catalogues may base their calculations on the thread insert helix angle, but one fact still remains unchanged. Thread milling cutter must always advance by the distance that is equivalent to the pitch amount in one revolution (360°). If a lead-in arc is used, only a portion of the pitch is programmed. The actual amount of linear travel has to be calculated as a ratio per degrees traveled (*see previous example*). Following formula is another version of the earlier one. It also calculates the amount of linear travel, this time based on the number of threads per inch (TPI):

$$L_t = \frac{A}{360 \times \text{TPI}}$$

where ...

- $L_t$  = Linear travel in helical interpolation
- $A$  = Amount of degrees interpolated (angle)
- TPI = Threads per inch

### ◆ Milling the Thread

Because of the cutter and thread size, *two full revolutions* have been selected to complete the specified thread. For each revolution, that is for each 360°, linear position of the cutter must be changed by the pitch amount. That is the 0.0833333 amount in the example. Thread motion is a helical milling motion and either absolute or incremental programming method can be used.

First, absolute dimensioning method will be selected, then the incremental method:

N8 G90 G03 X1.5 Y0 Z-0.8459 I-1.5 (TURN 1)  
N9 G03 X1.5 Y0 Z-0.7626 I-1.5 (TURN 2)

Repetitious data will not appear in the final program. For comparison, program the two motions incrementally:

N8 G91 G03 X0 Y0 Z0.0833 I-1.5 (TURN 1)  
N9 G03 X0 Y0 Z0.0833 I-1.5 (TURN 2)



When both helical motions are completed, the cutter had traveled 0.1666 along positive direction of the Z-axis and total of 720° (two revolutions). The program last part will be the cut ending position.

### ◆ Lead-Out Motions

For the same reason why the cutter approached the thread using helical interpolation over a 90° arc, the exit from the thread will be treated the same way. This departure from completed thread (a lead-out motion) will move the threading cutter *away* from the finished thread, again using one quarter turn motion that is still in the helical mode. Calculation is the same as before, and so is the amount:

$$L_t = 90 \times 0.0833333 / 360$$

$$L_t = 0.0208333$$

This incremental value will bring the tool up and away from the thread (programmed in absolute mode):

```
N10 G03 X0.75 Y0.75 Z-0.7418 I-0.75 J0
                        or (R0.75)
```

At this point, the cutter is in a position that is clear of the thread, so the linear motion can resume and cancel the cutter radius offset, then move back to the bore center, retract the tool above part, move to machine zero and terminate the program:

```
N11 G40 G01 X0 Y0
N12 G00 Z1.0 M09
N13 G28 X0 Y0 Z1.0 M05
N14 M30
%
```

At this point, the thread cutting job is done and the complete program can be written.

### ◆ Complete Program

Complete program that follows, combines all individual calculations and also includes all motions required for the threading cutter:

```
O4501 (INTERNAL RIGHT HAND THREAD MILLING)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S900 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.95 F50.0
N6 G41 X0.75 Y-0.75 D01 F10.0
N7 G03 X1.5 Y0 Z-0.9292 R0.75
N8 Z-0.8459 I-1.5
N9 Z-0.7626 I-1.5
N10 X0.75 Y0.75 Z-0.7418 R0.75
N11 G40 G01 X0 Y0
N12 G00 Z1.0 M09
N13 G28 X0 Y0 Z1.0 M05
N14 M30
%
```

(TURN 1)  
(TURN 2)

This program is only a small sample of one thread milling method. All calculations are logical and program code is clear. Reading various technical specifications for a thread milling cutter presents a wealth of information (including programming tips), suggested by a particular tool manufacturer. These recommendations always take on a more important role than any other method.

Figure 45-10 illustrates isometric view of the sample thread milling program O4501.

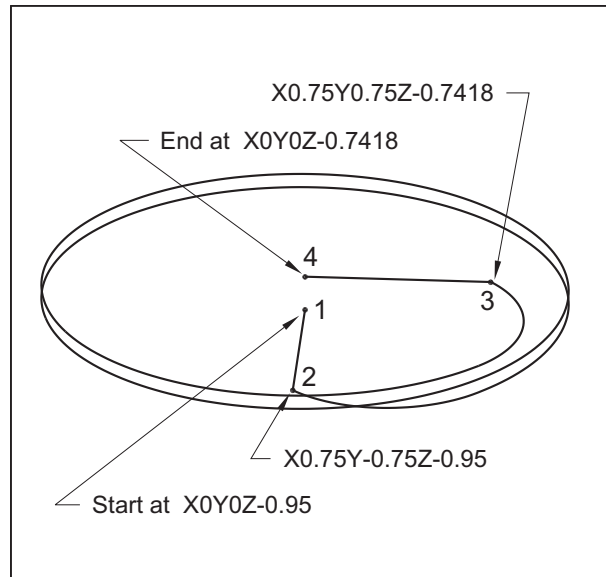


Figure 45-10

Isometric view of tool motions for the thread milling example

### ◆ External Thread Milling

External thread milling is often used for large threads with a carbide indexable threading insert. Both lead-in and lead-out motions are very important in this situation as well. Their calculations and those for the thread follow the same rules as for an internal thread. A straight linear lead-in and lead-out may be used, similar to the ones described for circular interpolation (*see page 251*). Otherwise, follow the motions shown in Figure 45-11.

### ◆ Tapered Thread Milling

It is possible, but much more difficult, to manually program a tapered thread (such as NPT or NPTF) using a thread milling cutter. For threads with a small pitch, soft material and very narrow taper angle, a tapered cutter may be used and programmed as if it were a straight cutter, in a single revolution. For larger threads, the only method is a *simulation* of helical milling (software is required in this case), using very small increments in *linear interpolation* mode only. Holders and inserts should be selected by the nominal size of the thread. Many tap manufacturers provide the necessary software.

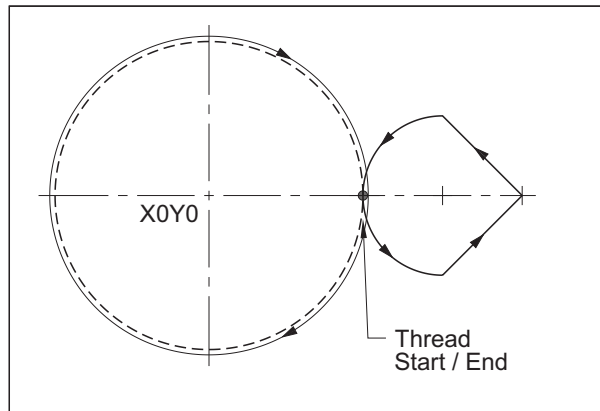


Figure 45-11

Lead-in and lead-out motions for external thread milling - top view

Tapered threads are sometimes called *conical* threads and will require different tool holders for right-hand threads and left-hand threads. This is a special application of helical interpolation that does not really belong in the manual programming area.

### ◆ Further Considerations

Two additional considerations are necessary to cover the subject of general thread milling in a reasonable depth. One is the application of *cutter radius offset* and the other one is the selection of *cutting feedrate*.

Cutter radius offset will only be active for the two axes selected by the active plane (for example, in G17, it will be X and Y axes). As a rule of thumb, always select climb milling cutting method, it is the preferred method for the majority of thread milling applications.

Feedrate selection is similar to feedrates for outside and inside arcs. This subject is discussed on *page 253*. Since a precision thread is the goal, the cutting feedrate will be 10 to 30 percent slower. A good start is at about 0.001 per tooth and higher, by experimenting.

## THREAD MILLING SIMULATION METHOD

There is an interesting way to mill a thread *without* the benefit of helical interpolation option available on the control system. This may be a case for many CNC machines, or in such cases where the machine shop needs to mill a thread only once in a while and the helical interpolation is not worth the cost of a control update.

To mill a thread (external or internal) under these conditions, a helical milling *simulation* will be used. Simulation of helical motions requires a simultaneous three-axis *linear* cutting motion, within acceptable tolerance of the thread. That means each motion will be a very small three-axis *linear* motion (using the X, Y and Z axes). The more accurate

thread is needed, the longer program will be generated. This method is practically impossible to do manually, as the development time could hardly be justified in any case. What is needed is a program software that will do the calculations in a matter of seconds. Many manufacturers of thread milling cutters provide such software free or for only a small cost and make it available at their web sites.

To illustrate this topic, the same thread will be used as in program O4701. Needless to say, a simulated program may be extremely long - *at least a few hundreds blocks*. Here is an example of such a program - it shows a few blocks of the beginning and a few blocks where the lead-in arc is completed. It only relates to the straight line and part of lead-in arc. Practically, this program is incorrect, because the tool radius is not compensated. Radius compensation would be done in the software used, not with G41 or G42 in the program - this is a *linear interpolation* in all three axes and cutter radius offset may not be used. The complete program had been done by using a CAD/CAM software, and was 463 blocks long, comparing to just 14 blocks for the equivalent complete program using helical interpolation.

```
G20
G17 G40 G80
G90 G54 G00 X0 Y0 S900 M03
G43 Z0.1 H01 M08
G01 Z-0.95 F10.0
X0.75 Y-0.75
X0.7846 Y-0.7492 Z-0.9494
X0.8191 Y-0.7468 Z-0.9488
X0.8536 Y-0.7428 Z-0.9482
X0.8878 Y-0.7373 Z-0.9476
X0.9216 Y-0.7301 Z-0.9470
X0.9552 Y-0.7214 Z-0.9464
X0.9883 Y-0.7112 Z-0.9457
...
...
...
X1.4967 Y-0.0697 Z-0.9304
X1.4992 Y-0.0350 Z-0.9298
X1.5000 Y0.0000 Z-0.9292
...
```

What the program output shows is a series of very small line segments, in a very precise order and increment. Follow at least a few blocks and visualize the actual motion. By the way, it took about three seconds to generate the 463 blocks of code in CAD/CAM. Knowing a high level language (such as Visual Basic®, Visual C++® and similar languages), writing similar utility software can be done very efficiently. Typically, when such a utility is executed, the user inputs the number of revolutions, the radius, thread lead and resolution. Program length can be shortened but the overall threading quality may not be acceptable.

Regardless of the method used to generate tool path for thread milling, this is a machining and programming area that deserves a lot more attention than it normally gets in many machine shops.

## HELICAL RAMPING

One very useful application of helical interpolation is *helical ramping*. It is used primarily as a replacement for plunge cut into solid materials. Roughing operations in enclosed areas (for example pockets), require cutting tool to reach a certain Z-depth *before* actual material removal. This Z-axis motion can be in an open space, if the material had been *predrilled*, for instance. Z-axis motion can also be cutting *straight into solid material*, with *center cutting* type tools (so called slot drills). *Helical ramping* is a method that allows any flat cutter to reach required Z-depth as a series of small helical cutting motions. Cutter can be *flat* and *non-center cutting*, as all cutting action is done by the cutter sides, not its bottom. Once the Z-depth has been reached, full circular interpolation is used to clean up. High level CAD/CAM software can do this very efficiently.

### Example :

For an example, a standard, flat bottom,  $\varnothing 0.500$  inch end mill is used (no need for a center cutting type) that will open the start hole to  $\varnothing 0.750$ . Pocket depth is 0.250 and in each helical motion the tool moves by 0.050. Pocket center is X0Y0 and Z-start position (clearance) is 0.050 above the top of part (Z-axis program zero). Total number of helical motions (revolutions) is six (one above top of work, plus another five below top of work).

For the depth, any increment can be chosen, depending on cutting conditions. Smaller increment results in more helical passes and longer cutting time.

Program can be in absolute or incremental mode - in this case, incremental mode is easier to program. Cutting will be done in climb milling mode - program O4502.

```
O4502 (HELICAL RAMPING)
N1 G20
N2 G17 G40 G80
N3 G90 G54 G00 X0 Y0 S700 M03
N4 G43 Z1.0 H01 M08
N5 G01 Z0.05 F50.0          (APPROACH TO Z-START)
N6 G41 X0.375 D01 F15.0     (START COMPENSATION)
N7 G91 G03 I-0.375 Z-0.05   (CUT ABOVE WORK)
N8 I-0.375 Z-0.05          (CUT 1 BELOW TOP FACE)
N9 I-0.375 Z-0.05          (CUT 2 BELOW TOP FACE)
N10 I-0.375 Z-0.05         (CUT 3 BELOW TOP FACE)
N11 I-0.375 Z-0.05         (CUT 4 BELOW TOP FACE)
N12 I-0.375 Z-0.05         (CUT 5 BELOW TOP FACE)
N13 I-0.375                (CIRCULAR BOTTOM CLEANUP)
N14 G90 G01 G40 X0         (RETURN TO XY START)
N15 G00 Z1.0 M09
N16 G28 Z1.0 M05
N17 M30
%
```

Two items are worth a note. Because incremental mode is used, Z-axis start is *extremely* important (block N4). Cutter radius offset is applied during a straight motion from the center to the start of first helical motion. *Figure 45-12* shows schematics of the program in four different views.

Helical interpolation is a powerful programming tool, often irreplaceable by any other method. Although it is a control option, its main benefit is short program output and the possibility of quick changes that may justify its extra cost.

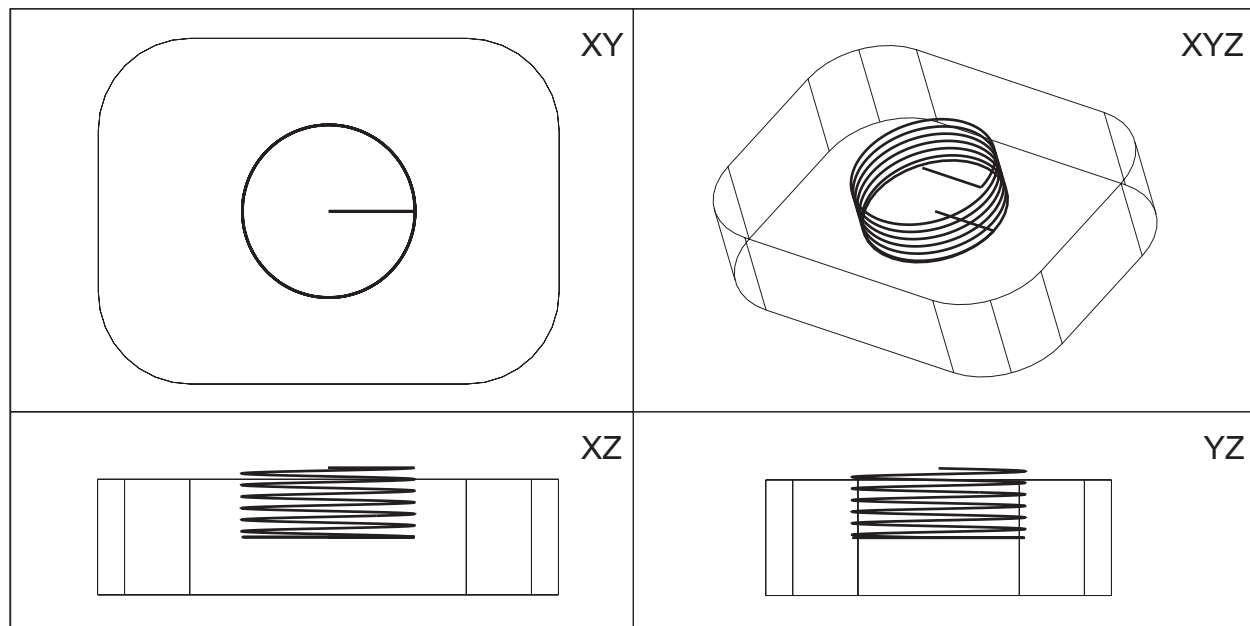


Figure 45-12

Schematic illustration of a helical motion used for ramping - program example O4502

## 46

## HORIZONTAL MACHINING

Throughout this handbook, there have been dozens of programming examples. They all shared one common feature - they were aimed at the CNC vertical machining centers. There was a reason for this approach. First, there are more vertical machining centers in machines shops overall, and mixing two different types of machines would make all reference material more complex. Second, almost every subject covered so far for vertical models is equally applicable to horizontal models. So what are the differences?

Horizontal machining center mainly differs from a vertical machining center in its general functionality. While a CNC vertical machine is mostly used for only one face type of work, a CNC horizontal machine is used for work on many faces of the same part during a single setup. This feature alone makes a horizontal machining center much more versatile machine - and also more expensive. Figure 46-1 shows comparison of axis orientation.

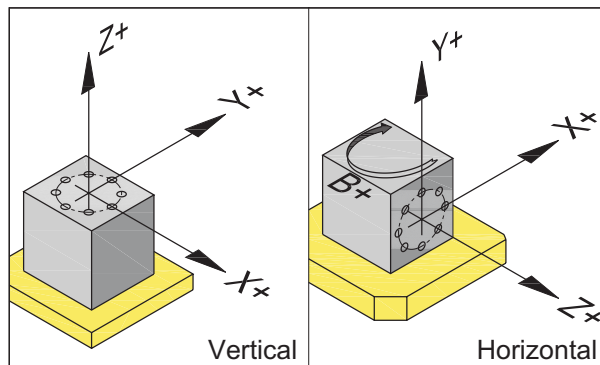


Figure 46-1

Axis orientation differences between vertical and horizontal machines

From the illustration is clear that the XY plane is used for the primary plane of work and the Z-axis is used to control cutting depth. There is no difference whatsoever between the two machine types in this respect.

Between programming and setup, there are three major differences on a horizontal machining center:

- Presence of a fourth axis, typically an indexing B-axis
- Presence of a pallet changer
- Richer variety of setup and offset settings

First, a brief look at the *fourth* axis of a typical CNC horizontal machining center.

## INDEXING AND ROTARY AXES

All programming concepts that have been discussed so far, apply equally to CNC horizontal machines. XY axes are used mostly for drilling and contouring operations, the Z-axis controls the cutting depth.

Horizontal machining centers differ from vertical machining centers not only in axes orientation and the type of work that can be machined. One of the major differences is an *additional* axis.

This is an *indexing* or a *rotary* axis, usually designated as the *B-axis*. Although the two terms are often used interchangeably, there is a difference between them.

- An *indexing* table will rotate the part that is mounted on it, but it cannot be used simultaneously with any kind of cutting motion. This type supports a *positioning* motion.
- A *rotary* table will also rotate the part that is mounted on it, but a simultaneous cutting action is possible. This type supports a *contouring* motion.

The most common fourth axis on a horizontal machining center is the indexing type, programmed as the B-axis.

## INDEXING TABLE (B-AXIS)

Indexing axis, as the name suggests, is used to index a table, if the machine is equipped with this feature. Horizontal machining centers and boring mills have indexing table as a standard feature. A full rotary table is an option on both types of machining centers.

### ◆ Units of Increment

Indexing axis is programmed as the number of degrees that is required by the job. For example, to index a table to a 45° absolute position, program:

G90 G00 B45.0

Minimum increment depends on the machine design. For indexing, a typical minimum unit of increment could be 1 degree or even 5 degrees. However, for more flexibility - and for rotary machining - much finer increment is required. Most machine manufacturers offer 0.1, 0.01 and 0.001 of a degree as the minimum indexing increment. In all cases, the programming of indexing motions can be done in two directions.

### ◆ Direction of Indexing

The B-axis can be programmed to index either *clockwise* or *counterclockwise*, looking from the top down to the table, which is the XZ-plane - Figure 46-2.

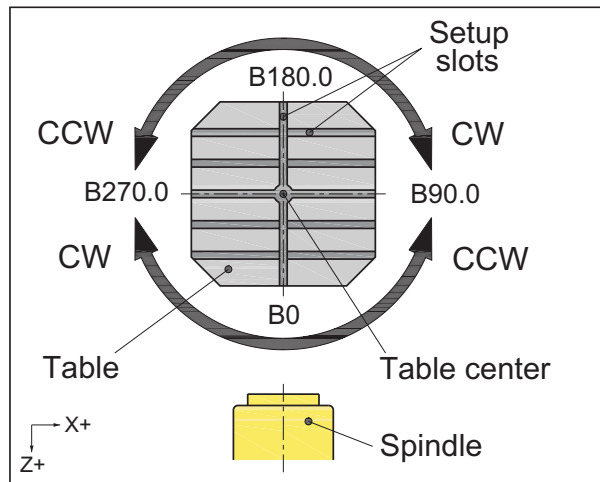


Figure 46-2

B-axis direction and general descriptions

The table size, including the size of corners, is important to determine clearances before indexing.

### ◆ Table Clamp and Unclamp Functions

In order to maintain a rigid setup, the indexing table must be clamped to the main body of machine during a cut. For indexing motions, the table must be unclamped. This is true of most machining centers. For this purpose, manufacturers offer special miscellaneous functions - two functions will be used in the examples:

- Table Clamp ... for example M78
- Table Unclamp ... for example M79

Actual function numbers may vary greatly with different machine designs, so check the manual for proper coding.

Normally, the unclamp function is programmed before the indexing, followed by the B-axis motion and another block containing the clamp function:

**M79**                      *Unclamp table*  
**G00 B90.0**            *Index table*  
**M78**                      *Clamp table*

Some designs require other M-functions, for example to control the clamping pin or a table ready confirmation.

B-axis is programmed logically the same way as the linear axes, including the mode of dimensioning. Either absolute or incremental mode can be used for indexing, using standard G90 and G91 commands respectively.

### ◆ Indexing in Absolute and Incremental Mode

Just like any other axis, B-axis can be programmed in absolute mode or incremental mode, with the same behavior as the linear axes.

The following example is in absolute mode, showing two table columns. The first column is the programmed indexing motion in G90 mode, the second column shows the actual resulting indexing motion (called *Distance-To-Go*) and its direction. All rotational directions are based on the perpendicular view towards the XZ-plane.

#### ➡ Absolute Mode - consecutive indexes :

Programmed motion in G90	Actual indexing motion
G90 G28 B0	Machine B-zero position
G00 B90.0	CW 90 degrees
B180.0	CW 90 degrees
B90.0	CCW -90 degrees
B270.0	CW 180 degrees
B247.356	CCW -22.644 degrees
B0	CCW -247.356 degrees
B-37.0	CCW -37 degrees
B42.0	CW 79 degrees
B42.0	No motion (0 degrees)
B-63.871	CCW -105.871 degrees

The next table is quite similar. The first column is the programmed indexing motion in G91 mode, the second column shows motion directions and the actual resulting absolute position. All rotational directions are based on the perpendicular view towards the XZ-plane.

#### ➡ Incremental Mode - consecutive indexes :

Programmed motion in G91	Actual absolute position
G90 G28 B0	Machine B-zero position
G91 G28 B0	Machine zero - no motion
G00 B90.0	CW 90.000
B180.0	CW 270.000
B90.0	CW 360.000
B270.0	CW 630.000
B0	No motion
B125.31	CW 755.310
B-180.0	CCW 575.310
B-75.31	CCW 500.000
B-75.31	CCW 424.690
B-424.69	CCW 0.000



Study both tables block by block, in the listing order. The results are always important for good understanding. Note the B-37.0 in the first table - exactly the same result could be achieved if the block read B323.0 as a positive value.

In the second table, the first block is in absolute mode to guarantee a start at B0. One occurrence that is interesting - when the rotation in the same direction reaches 360° (a full circle), it continues to increase. It does not become a zero degrees again. That is something to watch. If indexing (in incremental mode) takes place twice around, the absolute table position will be 720.000°. Indexing twice will also be necessary in the opposite way in order to reach absolute zero again. A small example is illustrated in *Figure 46-3*.

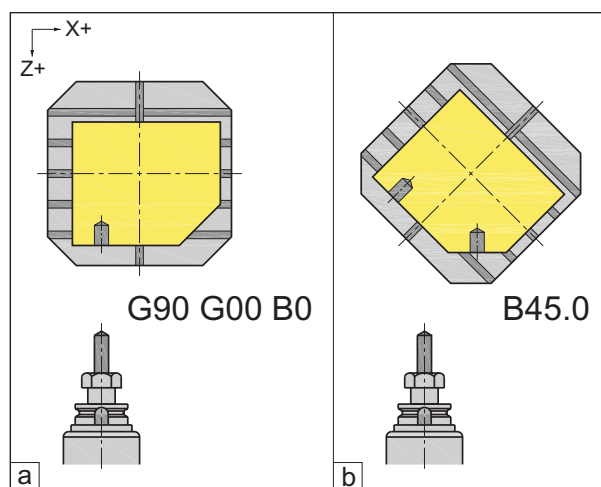


Figure 46-3

B-axis direction from B0 to B45.0 in absolute mode - O4601

To program the two positions shown above, typical block sequence would be, for example program sample O4601:

```
O4601
G90 G54 G00 X.. Y.. Z..
M79
B0
M78
...
< DRILL HOLE AT B0 >
...
G90 G55 G00 X.. Y.. Z..
M79
B45.0
M78
...
< DRILL HOLE AT B45.0 >
...
```

Dimensions relating to the drilling are not important for this example, but they are important for complete program.

Always observe safe clearances when indexing the B-axis

## B-AXIS AND OFFSETS

One of the most important differences between vertical and horizontal machining centers is the way the two major offsets are programmed and set:

- Work offset (G54-G59 + optional extended set)
- Tool length offsets (G43 H..)

Cutter radius offset is *not* affected by the B-axis and is programmed the same way as in vertical machining.

The relationship of offsets to the machined face of a part is very important and is also more complex than for vertical approach.

### ◆ Work Offset and B-axis

Work offset is measured the same as always - from machine zero to program zero. What is different now is the reality that there are *several* faces used for machining rather than just one. That means the toolpath for each face has to have its own program zero, therefore its own work offset. *Figure 46-4* shows a typical setting, looking at the part from the direction of spindle.

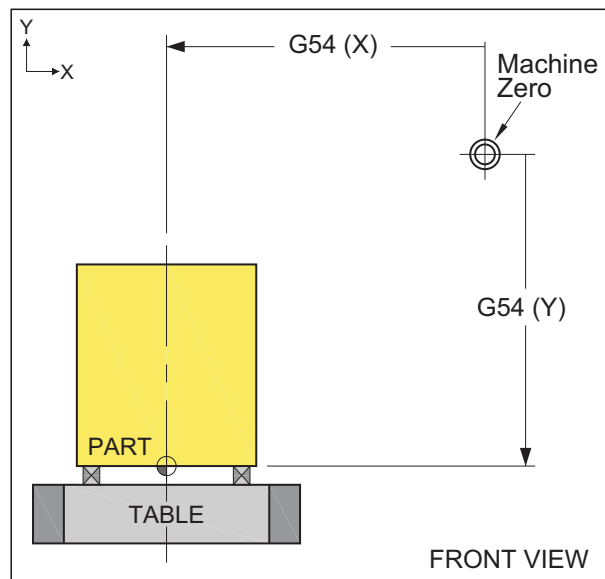


Figure 46-4

Work offset for a horizontal application - front view shown

Although the illustration shows part zero at the center of the indexing table, part zero may as well be at the top of each part face or even elsewhere. There are benefits in either approach and there is no 'one best' method. Often, it is the specified requirement of a job, fixture design, nature of the work and - of course - programmer's preferences.

When changing from one face to another, remember to change the work offset as well. For example, if there are four faces to machine, each face will have its own work offset, such as G54, G55, G56 and G57. B-axis is usually not dependent on work offset, so program the new offset during the first rapid motion. The previous short example illustrates this method. The next section describes work offset setting for the Z-axis as well as tool length offset.

### ◆ Tool Length Offset and B-axis

It should be easy to understand the concept of multiple work offsets for multiple faces. Setting the tool length can be quite complicated, depending on many factors that influence the decision. The first factor is the method of setting tool length. There are at least two methods to set the tool length offset. Both have already been covered, starting on page 135, but now they take on a new significance.

#### Touch-Off Method

One method is to *touch-off* Z0 of the machined face and register the distance from tool tip as a negative length offset. This has been a preferred method for vertical machines. Touch-off method may be acceptable for a small number of tools and indexes. Although it is possible to select the center of indexing table as Z0, it is not always a practical solution. Figure 46-5 shows the principle of touch-off setup in general terms, and Figure 46-6 shows a practical example. Note that, logically, the setup is exactly the same as for vertical machining. A program block ...

**G43 Z2.0 H01**

... will move the tool to Z-298.0, if H01 is set to -300.0.

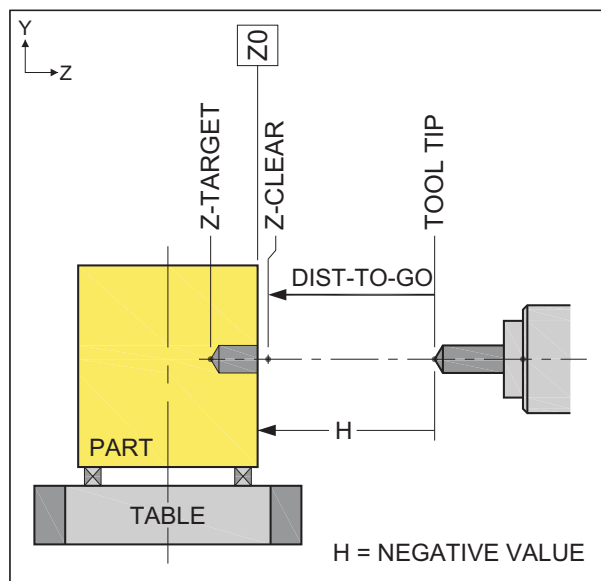


Figure 46-5

Touch-off tool length offset method - layout with H as negative

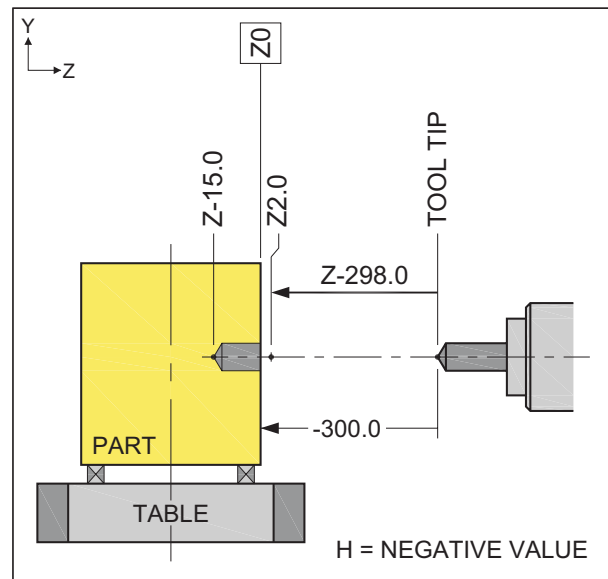


Figure 46-6

Touch-off tool length offset method - example with H as negative

#### Preset Method

Tool length set on vertical machining centers is often a touch-off method but it could also be a *preset* method. *Preset* method uses a special tool length presetter device and is done off-machine. There is a good reason why the preset method is much more practical for horizontal machining than for vertical machining.

Recall that a single tool normally requires single tool length offset. Now, consider a very typical situation for a horizontal machining - a single tool has to machine six faces, followed by other four tools that also do machining on the same six faces. Each of the five tools requires a unique tool length for each face - *for the total of 30 different length offsets!* This is not an isolated example at all, and there are several solutions to such a situation.

All solutions use the *preset tool* length measurement and *one additional setting*. Tool mounted in a holder is placed in the presetting device. Through a computerized optical reader, the presetter is calibrated to match the machine gauge line. Then, the tool length is accurately measured. It is a positive value representing the actual tool length from its tool tip to the machine gauge line. This is the amount that will be input into the corresponding tool length offset register. There is only one problem - *where is the relationship of this measured amount to the part position?* In the touch-off method, tool touches the part and the relationship is direct. The preset method has no contact - *one additional setting* mentioned earlier has to be made.

This setting is an entry of the distance between machine gauge line and Z0 of the current work offset Z-address - see Figures 46-7 and 46-8.

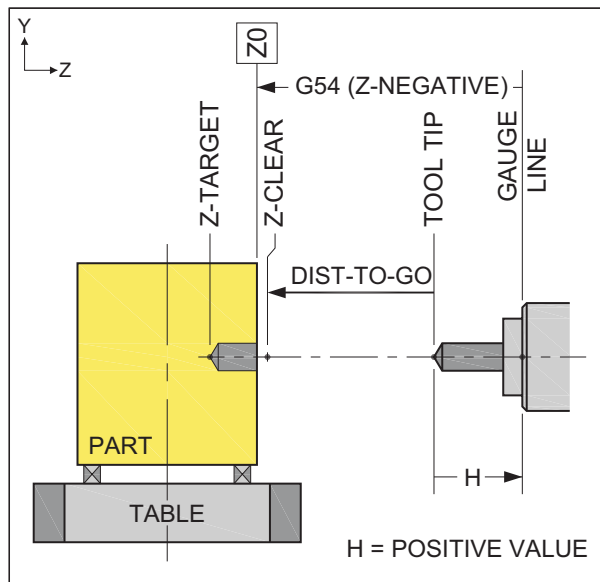


Figure 46-7

Preset tool length offset to Z0=face - layout with H as positive

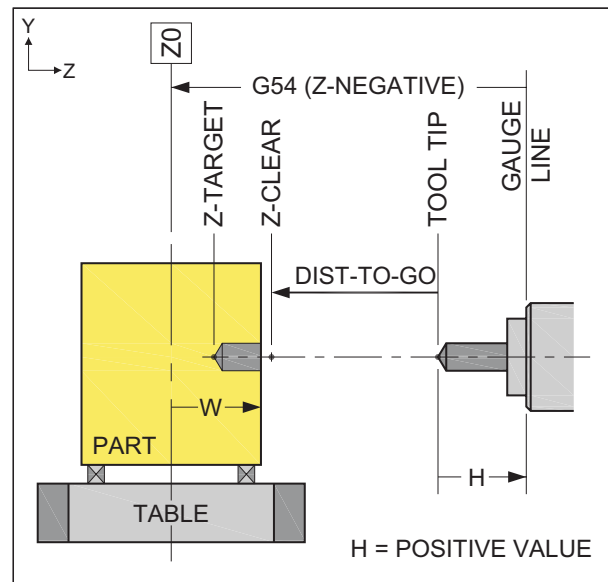


Figure 46-9

Preset tool length offset to Z0=center - layout with H as positive

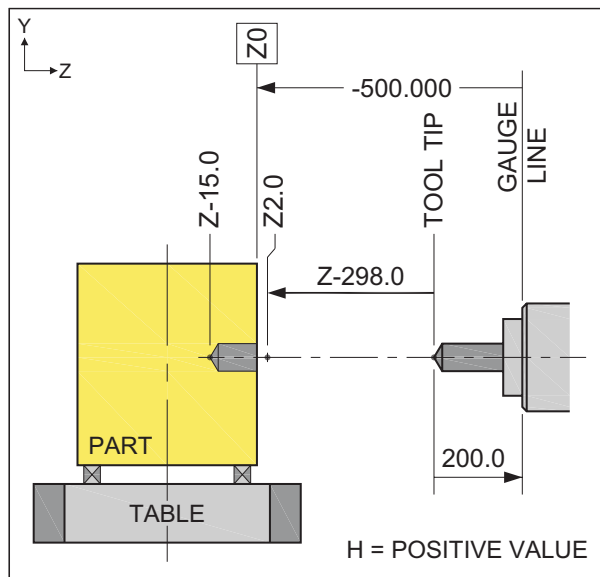


Figure 46-8

Preset tool length offset to Z0=face - example with H as positive

The illustration shows the offset amount entered into the Z-register of G54 work offset as -500.0. This is the distance from gauge line to part zero. To prove the method works, use H01=200.0 and the Z clear position as G43Z2.0H01. The distance-to-go is calculated the same way as always:

$$\begin{aligned} \text{G54 (Z)} + \text{Z clear} + \text{H01} &= \\ &= -500.0 + 2.0 + 200.0 \\ &= -298.0 \end{aligned}$$

The tool then continues normally to the Z-15.0 depth.

The last example was measured to Z0 position at the part front face. Another option exists if Z0 is set as the center of indexing table. In fact, it is only the *perception* of a change, the example is the same in reality. Figures 46-9 and 46-10 show the apparent change from the last two figures.

The setup has changed only because of the additional dimension *W*, specifying the distance from program zero to the part face. Z-axis values in the program will also change, as all dimensions are taken from Z0 at the table center, not the part face.

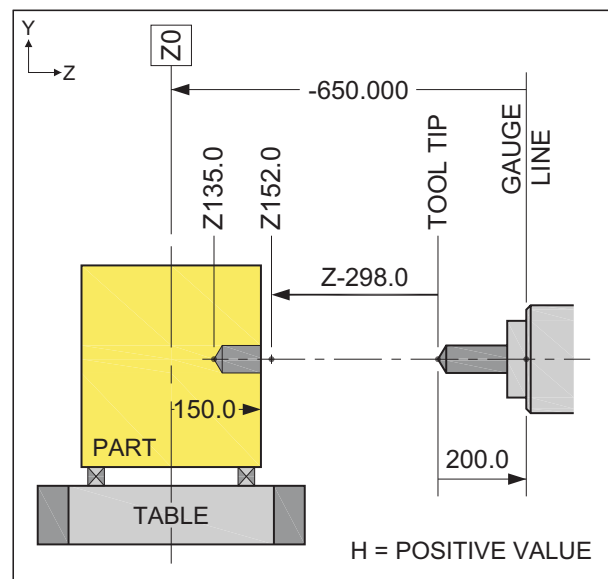


Figure 46-10

Preset tool length offset to Z0=center - example with H as positive

In the program is a block that moves the tool to clear Z position - G43Z152.0H01. To calculate the *distance-to-go* in this case, include the *W* distance that must always be known during setup (fixture drawing or actual measurement). That makes the *W*=150.0, no change for the length offset H01=200.0, but an important change to the G54 - now it is measured from the table center (Z0). The Z-clear position includes the *W* length and the physical clearance of 2 mm, same as in the previous case. In this example, the amount of Z-650.0 for G54 is used:

$$\begin{aligned} \text{G54(Z)} + \text{Z clear} + \text{H01} &= \\ &= -650.0 + 152.0 + 200.0 \\ &= -298.0 \end{aligned}$$

The tool then continues normally to the Z-135.0 depth. Overall, this setup application is exactly the same as the previous one. The operator must know where Z0 is located for every job. This information always originates from the CNC programmer in the form of a program comments or - even better - through a visual setup sheet.

## RETURN TO MACHINE ZERO

In CNC vertical machining, the return to machine zero has been programmed after every tool in majority of cases. Machine return was along the Z-axis only. The reason was simple - on a vertical machining center, Z-axis machine zero is synchronized with the automatic tool changer. This is *not* the case on a horizontal machining center.

Due to its design, the required machine zero return motion before each tool change is along the *Y*-axis. In all other respects, programming of machine zero commands is exactly the same for both types of machines.

Here is a simple comparison of a typical ending before a tool change for the two machine types:

Vertical:           G91 G28 Z0  
Horizontal:       G91 G28 Y0 Z0

If Y replaced Z as the axis to move, the question is what is the Z-axis return doing in the block when only the Y-return is required. The answer is *safety*. Although only the Y-axis is required to make successful automatic tool change, the tool has to be *away* from the part at the same time. Machine zero return along Z-axis makes it easier. Of course, programming only a sufficient clearance in the Z-axis would also achieve the same goal. That may prove more difficult than it appears. With the table in an index position other than zero, long and short tools, different part faces, fixture in the way, etc., it may be difficult to always know exactly how far to retract the Z-axis. That is why a simple rule is worth remembering when programming G28 return:

Return in Y-axis because it is necessary and in Z-axis for safety

## INDEXING AND A SUBPROGRAM

To describe all combinations of various setup methods and their influence on the program format is virtually impossible. The subject of horizontal machining, particularly its setup portion, can be quite complex and requires some experience. The general layout presented in this chapter should offer at least some basic understanding of the subject. A suitable programming example may also help.

To illustrate how indexing can be used in an efficient way, the program example in this section will spot drill and drill 612 holes on a cylinder - *Figure 46-11*. The spot drill will also break a chamfer of  $0.400 \times 45^\circ$ , measured from the high spot of the cylinder. All depth calculations are real.

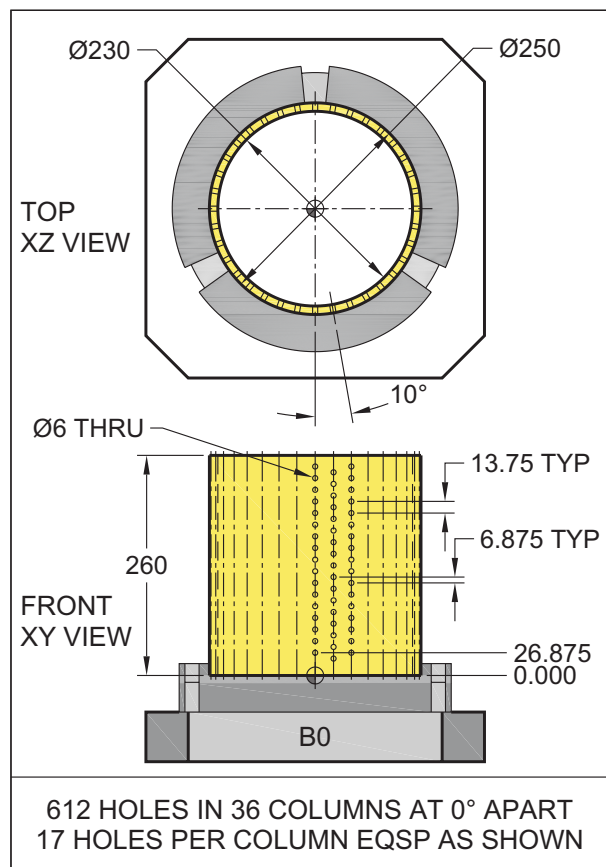


Figure 46-11

Practical example for indexing using subprograms - example O4602

Don't get discouraged by the large number of holes required. Using a subprogramming approach will minimize the program length. Program does not use any clamp and unclamp sequences, which is typical to the rotary type B-axis. If the machine requires unclamping the table before indexing and clamping it after indexing, use suitable M-functions for clamp and unclamp the table.

Before getting into the program itself, tools and their use need to be established. Only two tools will be required, a 10 mm spot drill and a 6 mm drill. Figure 46-12 shows the critical positions of the two tool tips.

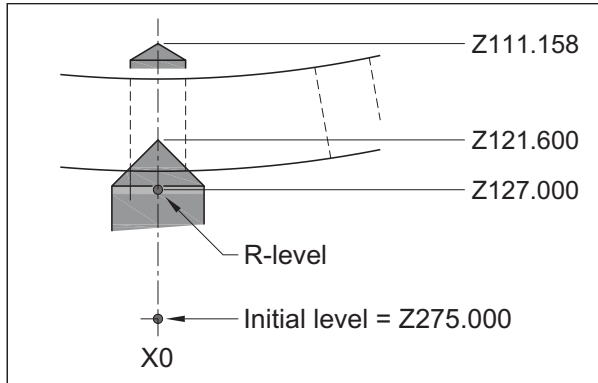


Figure 46-12

Detail of tool data used in program O4602

The R-level is the same for both tools and the spot drill depth also includes a small chamfer to deburr the holes. Drilling depth guarantees a full drill breakthrough. Actual calculations are not important here, but they do follow the same rules established in earlier chapters.

Development of the subprogram needs some work. Two subprograms will be used. They are virtually the same, except for the fixed cycle selection. Several other methods could have been also used, but this chapter concentrates on the indexing table only. Both subprograms will start at the bottom of the pattern, at B0 location (0°). This hole will be used as the start position only but will not be drilled until all other holes have been done. The hole is not drilled yet, but the 10° indexing has to be included in the subprogram. That is the reason for starting one column away. Two columns are part of each subprogram with a 10° index between them. Comments in the subprograms explain the process. Note the area marked in Figure 46-13, indicating the subprogram contents. *This example requires careful study.*

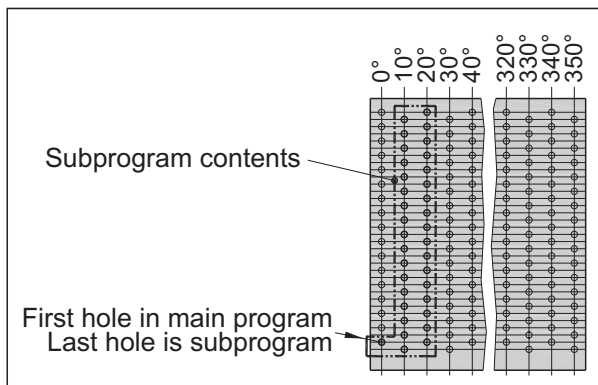


Figure 46-13

Flat cylinder layout - both ends shown for subprogram development

O4602 (MAIN PROGRAM)  
(START FROM MACHINE ZERO - T01 IN THE SPINDLE)  
(X0Y0 = FIXTURE CENTER / Z0 = BOTTOM OF PART)  
(T01 - 10 MM DIA SPOT DRILL)  
(T02 - 6 MM DIA DRILL THRU)

```
N1 G21
N2 G17 G40 G80
/N3 G91 G28 Z0
/N4 G28 X0 Y0
/N5 G28 B0
N6 G90 G54 G00 X0 Y26.875 S1000 M03 T02
N7 G43 Z275.0 H01 M08
N8 M98 P4651 L18
N9 G28 Y0 Z0
N10 G28 B0
N11 M01

N12 T02
N13 M06
N14 G90 G54 G00 X0 Y26.875 S1250 M03 T01
N15 G43 Z275.0 H02 M08
N16 M98 P4652 L18
N17 G28 X0 Y0 Z0
N18 G28 B0
N19 M06
N20 M30
%
```

O4651 (SUBPROGRAM FOR SPOT DRILL)

```
N101 G91 G80 Y-6.875 (MOVE DOWN BY PITCH)
N102 G90 Z275.0 (CLEAR Z)
N103 G91 B10.0 (ROTATE BY 10 DEGREES)
N104 G99 G82 R-148.0 Z-5.4 P200 F120.0 (DRL)
N105 Y13.75 L16 (16 MORE HOLES IN Y-PLUS)
N106 G80 G00 Y6.875 (MOVE UP BY PITCH)
N107 G90 Z275.0 (CLEAR Z)
N108 G91 B10.0 (ROTATE BY 10 DEGREES)
N109 G99 G82 R-148.0 Z-5.4 P200 (1 HOLE)
N110 Y-13.75 L16 (16 MORE HOLES IN Y-MINUS)
N111 M99 (END OF SUBPROGRAM O4651)
%
```

O4652 (SUBPROGRAM FOR 6MM DRILL)

```
N201 G91 G80 Y-6.875 (MOVE DOWN BY PITCH)
N202 G90 Z275.0 (CLEAR Z)
N203 G91 B10.0 (ROTATE BY 10 DEGREES)
N204 G99 G83 R-148.0 Z-15.84 Q7.0 F200.0 (DRL)
N205 Y13.75 L16 (16 MORE HOLES IN Y PLUS)
N206 G80 G00 Y6.875 (MOVE UP BY PITCH)
N207 G90 Z275.0 (CLEAR Z)
N208 G91 B10.0 (ROTATE BY 10 DEGREES)
N209 G99 G83 R-148.0 Z-15.84 Q7.0 (1 HOLE)
N210 Y-13.75 L16 (16 MORE HOLES IN Y MINUS)
N211 M99 (END OF SUBPROGRAM O4652)
%
```

Initial level of Z275.0, used in all three programs, is reasonable for safe indexing. Selection of suitable Z-axis clearance is important and knowing the indexing table size and size of its corners is imperative. For the record, the table for this job will be 400 × 400 mm square with 50 × 50 mm corners. Part setup is concentric with indexing rotation and there are no interfering elements. Loop L.. can be K..



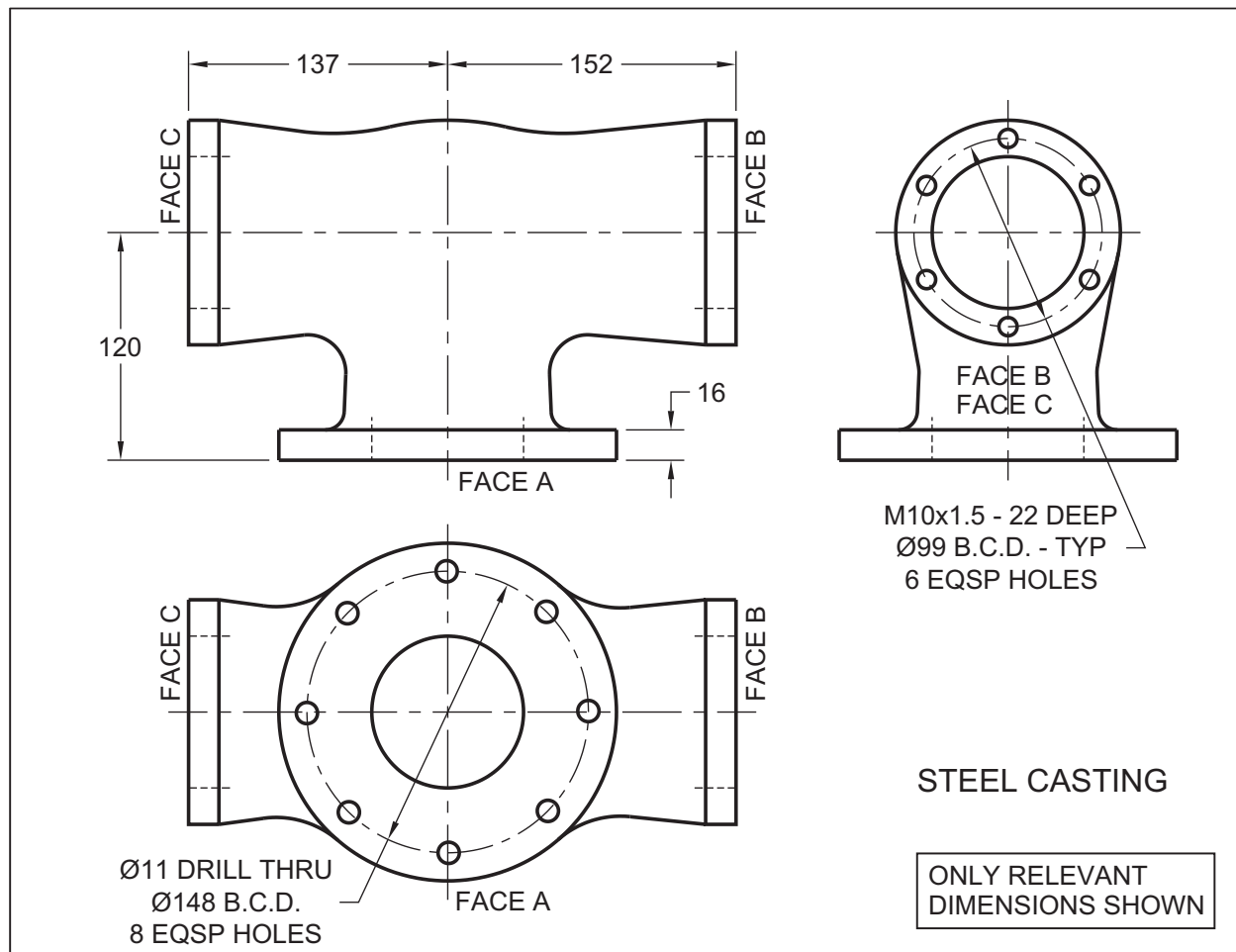


Figure 46-14

A typical multi-sided part suitable for a horizontal machining operation - program O4603 (subprograms O4653 and O4654)

## COMPLETE PROGRAM EXAMPLE

A typical part for horizontal machining center requires material removal from several sides in the same setup. Such a part, type of a housing, is shown in *Figure 46-14*.

For the example, only the holes will be machined at three different faces. As for tools, a spot drill, two drills and a tap will be used. The first step is to decide where to locate program zero. For the ease of programming and setup, the center of each bolt circle and the front of each face (Z) are good selections. Each face will have its own work offset - G54 for face A, G55 for face B and G56 for face C. The second step is to develop two subprograms for the hole locations. All dimensions have been calculated accurately but no details are necessary. First tool is in the spindle at startup. Part is located in a fixture mounted on the indexing table. Pallet changing has been omitted from the example, but is explained in the section that follows. The two subprograms contain bolt pattern coordinates.

O4653 (SUBPROGRAM FOR 8 HOLES AT 148 MM BCD)

```
N101 X74.0 Y0
N102 X52.326 Y52.326
N103 X0 Y74.0
N104 X-52.326 Y52.326
N105 X-74.0 Y0
N106 X-52.326 Y-52.326
N107 X0 Y-74.0
N108 X52.326 Y-52.326
N109 M99
%
```

O4654 (SUBPROGRAM FOR 6 HOLES AT 99 MM BCD)

```
N201 X49.5 Y0
N202 X24.75 Y42.868
N203 X-24.75 Y42.868
N204 X-49.5 Y0
N205 X-24.75 Y-42.868
N206 X24.75 Y-42.868
N207 M99
%
```

```

O4603 (MAIN PROGRAM)
(FACE A = G54 = B0 = 8 HOLES)
(FACE B = G55 = B90.0 = 6 HOLES)
(FACE C = G56 = B270.0 = 6 HOLES)

(T01 - 15 MM DIA SPOT DRILL)
(T02 - 8.4 MM TAP DRILL)
(T03 - M10 X 1.5 TAP)
(T04 - 11 MM DIA DRILL)

(T01 - 15 MM DIA SPOT DRILL - ALL HOLES)
N1 G21
N2 G17 G40 G80
/N3 G91 G28 Z0
/N4 G28 X0 Y0
/N5 M79
/N6 G90 G28 B0
/N7 M78
N8 G90 G54 G00 X74.0 Y0 S868 M03 T02
N9 G43 Z10.0 H01 M08
N10 G99 G82 R2.0 Z-5.8 P200 F150.0 L0
N11 M98 P4653 (SPOT DRILL FACE A)
N12 G80 Z300.0
N13 M79
N14 B90.0
N15 M78
N16 G55 X49.5 Y0 Z10.0
N17 G99 G82 R2.0 Z-5.3 P200 L0
N18 M98 P4654 (SPOT DRILL FACE B)
N19 G80 Z300.0
N20 M79
N21 B270.0
N22 M78
N23 G56 X49.5 Y0 Z10.0
N24 G99 G82 R2.0 Z-5.3 P200 L0
N25 M98 P4654 (SPOT DRILL FACE C)
N26 G80 Z300.0 M09
N27 G91 G28 Y0 Z0 M05
N28 M01

(T02 - 8.4 MM TAP DRILL)
N29 T02
N30 M06
N31 G90 G56 G00 X49.5 Y0 S1137 M03 T03
N32 G43 Z10.0 H02 M08
N33 G99 G83 R2.0 Z-24.8 Q6.0 F200.0 L0
N34 M98 P4654 (TAP DRILL FACE C)
N35 G80 Z300.0
N36 M79
N37 B90.0
N38 M78
N39 G55 X49.5 Y0 Z10.0
N40 G99 G83 R2.0 Z-24.8 Q6.0 L0
N41 M98 P4654 (TAP DRILL FACE B)
N42 G80 Z300.0 M09
N43 G91 G28 Y0 Z0 M05
N44 M01

(T03 - M10 X 1.5 TAP)
N45 T03
N46 M06
N47 G90 G55 G00 X49.5 Y0 S550 M03 T04
N48 G43 Z10.0 H03 M08
N49 G99 G84 R5.0 Z-23.0 F825.0 L0
N50 M98 P4654 (TAP FACE B)

N51 G80 Z300.0
N52 M79
N53 B270.0
N54 M78
N55 G56 X49.5 Y0 Z10.0
N56 G99 G84 R5.0 Z-23.0 L0
N57 M98 P4654 (TAP FACE C)
N58 G80 Z300.0 M09
N59 G91 G28 Y0 Z0 M05
N60 M01

(T04 - 11 MM DIA DRILL)
N61 T04
N62 M06
N63 M79
N64 B0
N65 M78
N66 G90 G54 G00 X74.0 Y0 S800 M03 T01
N67 G43 Z10.0 H04 M08
N68 G99 G81 R2.0 Z-20.3 P200 F225.0 L0
N69 M98 P4653 (DRILL FACE A)
N70 G80 Z300.0 M09
N71 G91 G28 X0 Y0 Z0 M05
N72 M30
%
```

Only a few comments to the example. Both the main program and the two subprograms are quite plain. Compared to vertical machining applications, the Z-axis safety clearance may seem a little too high with Z300.0 programmed before each indexing. Large clearances are for safety - they allow both the part and indexing table to index within a safe area, without any obstacles in the way. It is not convenient to actually calculate the minimum Z-clearance, but it is important to select it far enough for *all* faces. A CAD software can help here quite a bit. Other features and programming techniques are the same as used elsewhere in the handbook.

## AUTOMATIC PALLET CHANGER - APC

One of the greatest concerns in CNC machining is the unproductive time required for initial part setup and remounting the part when running a batch job. Many features incorporated in the control system or machine design itself can shorten unproductive time to a great degree. They include tool length offset, work offsets, cutter radius offset, etc. However, none of them solves the problem of time used up when mounting individual parts on the table. Probably the major breakthrough was an introduction of a pallet table to the CNC machine. Pallets are not a new idea in machining. For horizontal machining centers, interfaced pallets have become very practical feature to minimize setup time.

Traditionally, one machine has one work table. Such a design of a machine tool has one major flaw - while the machine is working (and the CNC operator is virtually idle), no other work can be performed. That means a setup for the next part is done at the expense of the machine being idle, resulting in unproductive time.

By definition, an automatic pallet is a work table that can be moved into and out of the machining position by program commands. If a purpose of such design is to shorten nonproductive setup time, it is necessary to have at least *two* independent pallets available - while the part on one pallet is being machined, the other pallet is available for changing setup for the next job or for unloading and loading individual parts. In this way, machining and setup can be done simultaneously, shortening or even totally eliminating unproductive time.

Although a two pallet system is the most customary for horizontal machining centers, designs with up to twelve pallets are not uncommon.

### ◆ Working Environment

For a typical dual pallet changer, two major areas should be distinguished:

- Machining area ... within the machine
- Setup area ... outside of the machine

One pallet is normally located in the machining area, the other in the setup area. When a program starts, it normally starts with Pallet #1 (with part) located in the machining area and Pallet #2 (with no part) in the setup area. There are many designs of pallets, but they all share three major parts:

- Pallet
- Machine locator
- Transfer System

*Pallet* is a portable work table with a ground surface to which fixtures and parts are mounted. Pallet table can have T-slots, tapped holes or both.

*Machine locator* (also known as a receiver) is a special device located inside of the machine. Its purpose is to accept and firmly hold the pallet loaded with a part ready for machining. Its design must be very robust and accurate at the same time.

*Transfer system* (also known as a pallet loader) is the system that transfers pallets between the load area and the machine work area.

Often the terms *load* and *unload* are used with pallets. *Load* means to move a pallet into the machining area, *unload* means to move a pallet into the setup area. Transfer system determines the type of pallet.

### ◆ Types of Pallets

There are two general types of pallets, based on their transfer system:

- Rotary type
- Shuttle type

The popular *rotary* type works on the principle of a turntable, where one pallet is outside of the machine, the other pallet is inside of the machine. The pallet change command rotates both pallets 180° and its programming is very simple. *Figure 46-15* illustrates typical rotary type.

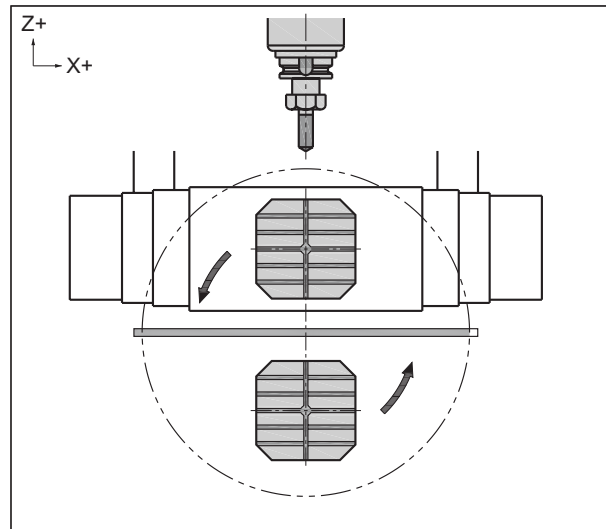


Figure 46-15

Typical rotary type of a pallet changer

Also popular is the *shuttle* type. This design incorporates double rails between the load area and the receiver inside the machine - *Figure 46-16*. Its programming is still simple but more involved than for the rotary type.

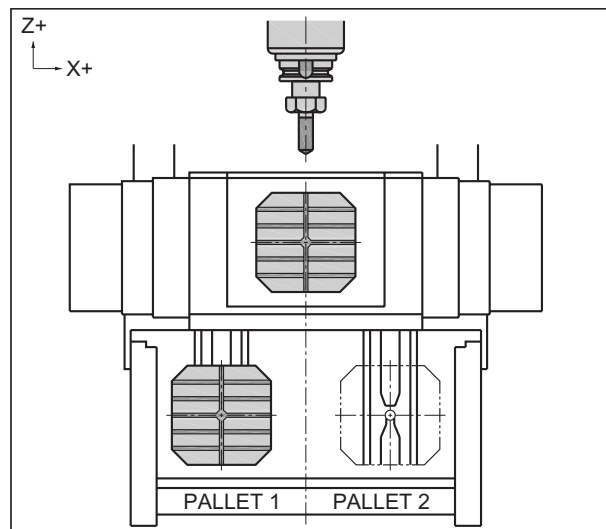


Figure 46-16

Typical shuttle type of a pallet changer

Both pallet types are loaded from the machine front area. Other pallet types are also available for some special machining applications.

### ◆ Programming Commands

The fairly standard miscellaneous function for automatic pallet changing is M60.

M60	Automatic Pallet Changer (APC)
-----	--------------------------------

This command works properly only when the pallet position is at one of two machine reference points:

G28	Machine return to the primary reference point
G30	Machine return to the secondary reference point

G28 command is a familiar command. G30 command is used exactly the same way, except it moves one or more selected axes to the secondary machine reference position.

### ◆ Pallet Changing Program Structure

The following program segment emphasizes the pallet changing on a typical shuttle pallet system. It can be easily adapted to a rotary pallet system. In both cases, one pallet is in the machining area, the other is in the setup area.

```
O4604
G91 G28 X0 Y0 Z0
G28 B0
M60                                (LOAD PALLET 1)
<... machining on Pallet 1 ...>
G91 G28 X0 Y0 Z0
G28 B0
M60                                (UNLOAD PALLET 1)
G30 X0
M60                                (LOAD PALLET 2)
<... machining on Pallet 2 ...>
G30 X0
M60                                (UNLOAD PALLET 2)
M30
%
```

## HORIZONTAL BORING MILL

A chapter on CNC horizontal machining would not be complete without at least some comments relating to the machine called a *horizontal boring mill*. A CNC boring mill is similar to a CNC horizontal machining center, usually larger in size. It may or may not have an automatic tool changer, and usually has the spindle motion split into two axes - *Z* and *W*. The following is a typical setup of a 4-axis horizontal boring mill with an indexing B-axis and a Fanuc or similar control with:

- Six work offsets ... G54 to G59
- Two machine reference points ... G28 and G30

Although there are five actual axes available, a horizontal boring mill is still only a four axis machine. The axes are:

- X-axis ... table longitudinal
- Y-axis ... column
- Z-axis ... spindle quill
- W-axis ... table traverse
- B-axis ... indexing or rotary table

Settings are similar to a machining center, plus the *W* axis. During setup, typical work offset values will be set as:

- G54 X = Negative  
Y = Negative  
Z = Zero  
W = Negative  
B = Zero

As many horizontal boring mills do not have an automatic tool changer, G30 position should be set conveniently for CNC operator to do a tool change manually (X,Y,W axes). This position is set by a system parameter. Z-axis value is the length of quill travel out of the spindle.

Programming format is based on the principle that all motions *into* the depth are done in the W-axis, rather than the Z-axis. Spindle quill that is controlled by the Z-axis, is pulled out only for clearance purposes; its extension from spindle must be long enough to guarantee enough clearance for the shortest tool used in the program.

Typical programming format is followed by a more detailed explanation. [nn] lines are for reference only and match the comments that follow:

```
[01] O4605 (PROGRAM NAME)
[02] (MESSAGE OR COMMENT)
[03] N1 G21
[04] N2 G91 G30 W0
[05] N3 G90 S.. M03
[06] N4 G54 G00 X.. Y..
[07] N5 G30 Z0
[08] N6 G43 W.. H..
[09] N70 G01 W.. F..
[10] . . .
[11] . . .
[12] . . . < machining >
[13] . . .
[14] . . .
[15] N35 G00 W..
[16] N36 M05
[17] N38 Z0
[18] N39 G30 G49 W0
[19] N40 G91 G30 X0 Y0
[20] N41 M06
[21] . . .
[22] . . .
[23] . . .
[24] N60 M30
[25] %
```

All following comments reference the line identification numbers in the previous example:

- [ 01 ] Program number (name up to 16 characters)
- [ 02 ] Message to the operator - only between parenthesis
- [ 03 ] Metric or imperial units selection
- [ 04 ] W-axis moves to a tool change position  
(incremental motion for safety)
- [ 05 ] Selection of absolute mode and spindle functions
- [ 06 ] Rapid motion to the starting position in XY within  
the G54 work coordinate system
- [ 07 ] Quill pulls out by the value of parameter 1241 (Z)
- [ 08 ] Tool length offset (set from the tool tip to the  
program zero) and motion to the clearance level
- [ 09 ] Feedrate motion to the required depth
- [ 10 ] ...
- [ 11 ] ...

- [ 12 ] ... ( machining the part ) ...
- [ 13 ] ...
- [ 14 ] ...
- [ 15 ] Rapid motion back to the clearance level (see [ 08 ])
- [ 16 ] Spindle stop
- [ 17 ] Rapid motion of the quill back to the spindle
- [ 18 ] Rapid motion to the tool change position along the  
W-axis and cancellation of tool length offset
- [ 19 ] Rapid motion to the tool change position along the X  
and Y axes - in incremental mode for safety
- [ 20 ] Manual tool change
- [ 21 ] ...
- [ 22 ] ... (additional machining, following the above format . . )
- [ 23 ] ...
- [ 24 ] End of program
- [ 25 ] End of record (stop code)



## 47

## LIVE TOOLING ON LATHES

Only a couple of decades or so back - in the late nineteen eighties and even in the early nineteen nineties - just considering a simple milling operation to be done on a CNC lathe was rare and virtually unheard of. Much has changed since these early days and modern CNC lathes today - *the true turning centers* - are unrecognizable from their early predecessors. That is not to say that the ubiquitous *two axis rear slant bed CNC lathe* is obsolete - on the contrary - it is still the main machine tool of choice for general round and conical machining. What has changed does not defy the original concept at all - in fact, *it adds to it*. The concept itself has always been quite simple for machine design engineers - *'let's design a CNC lathe in such a way that it can handle a few simple milling operations and save at least one machine setup and related handling'*. With technology advancing at a very rapid rate, it became possible to provide not only some rather simple milling capabilities, but also many very complex machining processes as well.

There are many possible combinations of turning machines equipped with milling capabilities, using so called *live tooling* - an independently powered non-turning tools. Descriptions such as *'rotary tools'* and *'driven tools'* are also frequently used. In terms of number of axes, having six, seven, nine or even more axes is not that unusual anymore on what is based on a standard two-axis CNC lathe. Combine all these axes with some additional features, and the amount of knowledge required to succeed is quite overwhelming, to say the least.

## TURN-MILL OR MILL-TURN

This widely used modern technology is known as the *turn-mill* or *mill-turn* operation with live tooling. Depending on a particular description, there is no consistency in industry as to which term is more accurate - they are both used about equally. For this handbook, the term **'turn-mill'** will be used, as it reflects the actual cutting activities more accurately - *'turning machine with milling capabilities'* or *'CNC lathe with live tooling'*, is more logical than the opposite definition. Actual use of the term may be a subject of some dispute, but regardless of which term is used, machining and programming processes are both the same.

Live tooling availability on a CNC lathe is the first step towards multi-process machining, that incorporates various turning and many milling applications, multiple turrets and chucks, subspindles, automated part reversal and part transfer and many other features. The result is a competitive edge for any machine shop.

### ◆ Programming Issues

The more machine axes are to be programmed, the more human effort is required. Doing all this work manually defeats all computer innovations seen in the last several years. Yes, the bottom line is simple - use an established software, such as Mastercam™, Edgecam™, and several others.

This comprehensive handbook has been all about manual programming - *why is there an apparent shift now?* Well, there is no shift. The main purpose of this handbook has always been to provide knowledge for programming manually, yes - but - to apply that knowledge in more advanced environments as well. Explaining more than a few machine axes in a publication such as this would become confusing and counterproductive. Fully featured CAM software can take care of many axes much easier than any explanation of manual methods. At the same time, some general concepts should be explained in detail, to form the basis of general understanding.

One important aspect of programming for this group of machine tools is that it combines skills of CNC lathe programmers with those of CNC mill programmers. Many machine shops do have their 'specialists' for each machine group. They may need to revisit that approach and train their personnel to be able to handle both groups simultaneously, with equal skill.

Before any serious programming effort is made, there is a need to understand some new words and expressions related to this modern technology.

### ◆ General Terms

Any new technology brings new words and phrases into existence. Milling capabilities of a CNC lathe are no different. Apart from the already mentioned terms turn-mill and mill-turn, other special terms emerged as well. For example, *multi-function* or *multi-tasking* machine tools are just two of them. Another popular term is *multi-process* machining. Regardless of the actual description, their single purpose is to do as much machining in a single setup as possible. That includes both turning and drilling/milling operations. It is also very important to understand some basic terminology that is different from standard machines.

- **Motorized tool** - also known as a *rotary tool* or a *driven tool* - is any tool that has its own power source and can rotate even if the lathe spindle is stationary. Power driven tools are mounted in the turret, together with other tools

- **Motorized tool rotation** - in order to distinguish between rotating spindle and rotating tool, M03/M04 is used for spindle rotation, while M13/M14 or M103/M104 is used for control of rotating tool - M05/M105 is a common command to stop *either* rotation
- **Polar Coordinate Interpolation** - typically programmed with G12.1/G13.1 or G112/G113 is sometimes called *notching* - do not confuse this feature with *Polar Coordinates*, typically programmed with G15/G16
- **Cylindrical Interpolation** - machining (typically grooving) on the part circumference, with a given depth. Typically uses G107 command
- **C-axis** - allows rotation of the spindle, usually in increments of  $0.001^\circ$  - machining can only be done at machine centerline - address C is used for absolute data, address H is used for incremental data input
- **C-axis ON/OFF** - a special M-function (or G-code) that allows switching from normal spindle function to a spindle acting as a rotary axis
- **C-axis Clamp ON/OFF** - may be required by some controls, particularly for heavier cuts, or it may be automatic. If programmed, a special M/G code is required
- **Y-axis** - allows machining off-centerline of the machine
- **B-axis** - allows angular positioning of the turret (if available)

More complex machine tools will have additional features, therefore more terms to understand. They will become familiar after studying the actual machine design.

## MACHINE DESIGN

Any major change in machine design will influence the programming methods. New features and design changes (as significant as milling on CNC lathes) will bring different programming methods and the inevitable challenges associated with it.

### ◆ Features

Whether one or two turrets or two spindles are available, live tools can be mounted parallel with, or perpendicular to, the machine centerline. Although there are major design differences between standard lathes and automatic or Swiss type lathes, the basic principles remain the same.

### ◆ Benefits

The main design feature of any multi-process machine tool is focus on reducing setup time. Once the initial setup has been made, complete parts can be machined without operator's interference. As expected, the main benefits include single setup for multiple operations.

From programming perspective, the first item of machine design to consider is the orientation of machine axes.

## C-AXIS PROGRAMMING

The first major difference in programming for live tooling, as compared with standard turning and boring, is addition of the C-axis to the standard X and Z lathe axes.

### ◆ XZ + C Axes

In this basic configuration, the addition of C-axis and live tools to a two-axis slant bed is the first step into the world of multi-tasking machines. Of course, it is not just turning and boring that can take place - such operations as slot milling and grooving, keyways, face drilling and milling, drilling on part circumference, and similar operation are also possible in the same setup. C-axis allows rotational position of the spindle. Its main purpose is to rotate the spindle in direct relationship with the motions of live tools.

Each live tool (*i.e.*, tool under its own power) is mounted vertically or horizontally in the lathe turret. If a turret has 12 stations, all can be occupied by live tools, or - more likely - the combination of standard tools and live tools. Figures 47-01 and 47-02 show different tool mounting.

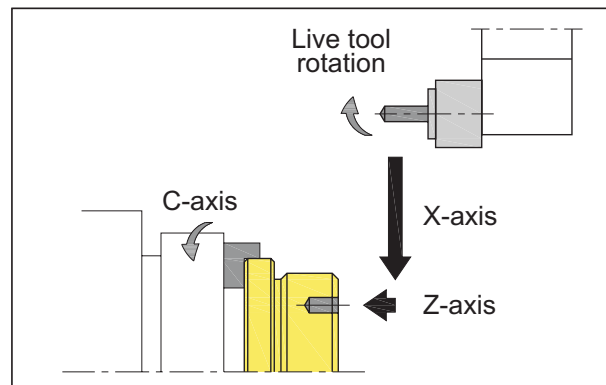


Figure 47-1

C-axis and live tool horizontal orientation - for cutting on faces

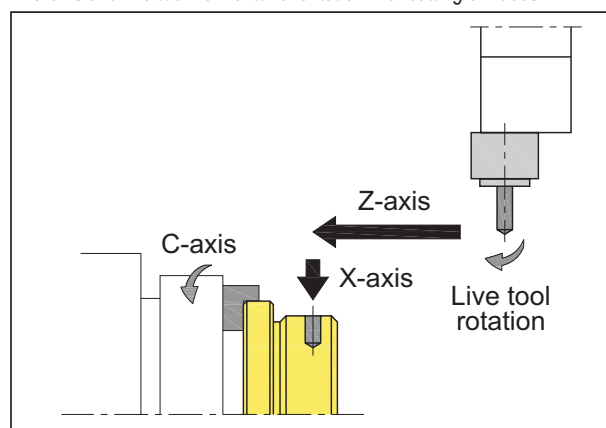


Figure 47-2

C-axis and live tool vertical orientation - for cutting on diameters

Live tools are interchangeable but once selected, they have a fixed diameter and are always programmed in G97 mode (r/min) - constant spindle speed mode. Feedrate is always programmed in units per time (mm/min *or* in/min), *not* in revolutions per minute. Since there is no spindle rotation, it makes sense. In case the tool called in the program does not move, check its programmed feedrate mode - the tool will either not move in units per revolution mode or the control will generate an alarm.

### Two Spindle Modes

In order to change the lathe spindle from its traditional function into a true C-axis, a special programming code is set by the machine manufacturer. Typical definition of such a code may include at least two descriptions:

- C-axis ON / OFF
- Normal spindle mode ON / OFF

Both of them serve the same purpose - to switch from one spindle mode to the other. Based on the selection, the programmed spindle speed is applied to either the spindle *or* the rotary tool.

### Machining Holes

One of the most common machining operations benefiting from C-axis and live tooling are holes. Fanuc controls provide a series of special fixed cycles that can be used for this purpose, either on the part diameter or part face. In both cases, drilling of each hole is combined with spindle rotation. It is important to understand that this is an indexing operation, not a rotary application. The differences are well defined - indexing operation does not allow any machining during spindle rotation. True rotary application (if available) is used for a different purpose - in this case, machining and spindle rotation can be applied simultaneously.

As machining holes using C-axis is the current topic, it is important to realize that spindle indexing takes place first, followed by the selected hole machining cycle. There will be no machining while the spindle indexing takes place.

### Indexing Increment

Once the spindle is in C-axis mode, it can be rotated (or indexed) by any desired amount in degrees (360° for one full turn). Depending on the control system, the smallest increment could only be 1° (360 positions), or it could be very fine 0.001° (360 000 positions). Typical program entry uses the C-address combined with specified degrees, for example,

**N185 C30.0**

*... will index the spindle to 30° absolute position.* For incremental positioning, use the H-address instead. C/H address designations have the same relationship as X/U and Z/W addresses.

## FIXED CYCLES

Fixed cycles for machining holes using driven tools are similar - *but not exactly the same* - as fixed cycles used on machining centers. Generally, there are three cycles in two groups available:

- Three cycles for drilling along the Z-axis: G83, G84, G85
- Three cycles for drilling along the X-axis: G87, G88, G89
- G80 cancels any active cycle

The list of cycles for a particular machine can be summed up in a simple table:

Cycle	Cutting direction	Description
G83	Z-axis	Drilling cycle
G84	Z-axis	Tapping cycle
G85	Z-axis	Boring cycle
G87	X-axis	Drilling cycle
G88	X-axis	Tapping cycle
G89	X-axis	Boring cycle
G80	Fixed cycle cancel	

General format for each cycle of G83-G85 group:

G83-G85 X(U).. C(H).. Z(W).. R.. Q.. P.. F.. K..

where ...

G83-G85 Cycle selection

X(U) Hole position

C(H) Hole position

Z(W) Tool position at hole bottom (depth)

R **Incremental** distance from initial level to R-level  
Specified as radius value

Q Peck depth - positive value - no decimal point

P Dwell time at hole bottom in milliseconds (ms)

F Feedrate

K Number of repetitions (may also be L)

General format for each cycle of G87-G89 group:

G87-G89 Z(W).. C(H).. X(U).. R.. Q.. P.. F.. K..

where ...

G87-G89 Cycle selection

Z(W) Hole position

C(H) Hole position

X(U) Tool position at hole bottom (depth)

R **Incremental** distance from initial level to R-level

Q Peck depth - positive value - no decimal point

P Dwell time at hole bottom in milliseconds (ms)

F Feedrate

K Number of repetitions (may also be L)

### ◆ M-functions and C-axis

Probably the most noticeable difference between different machines equipped with C-axis are various miscellaneous functions, particularly machine related M-functions. Although they perform essentially the same activities, there are no industry-wide standards. In this section, the following M-functions will be used for consistency in examples:

Spindle rotation CW	M03
Spindle rotation CCW	M04
Spindle rotation stop	M05
Driven tool rotation CW	M103
Driven tool rotation CCW	M104
Driven tool rotation stop	M105
C-axis clamp	M12
C-axis unclamp	M13
C-axis mode ON	M14 = Spindle mode OFF
C-axis mode OFF	M15 = Spindle mode ON

Keep in mind that these are only sample functions - consult your machine manual for equivalent M-functions as well as for any additional machine functions. Be particularly careful with functions such as M13-M14-M15 - they may be related to driven tool spindle rotation.

### ◆ Drilling on Face

Once major difference from machining centers application is that the R-level is specified incrementally. A typical program format for one hole - drilling only:

```

M14                                (C-AXIS MODE ON)
T0101                             (TOOL SELECTION)
G97 S700 M103                     (DRIVEN TOOL SPINDLE SPEED)
G98 G00 X40.0 Z10.0               (FEED/MIN+START POSITION)
C90.0                             (HOLE ORIENTATION)
G83 Z-15.0 R-8.0 F150.0           (DRILL HOLE)
G80 G99 X200.0 Z100.0 M105       (INDEX POSITION)
M15                                (SPINDLE MODE ON)
  
```

Figure 47-3 illustrates the concept:

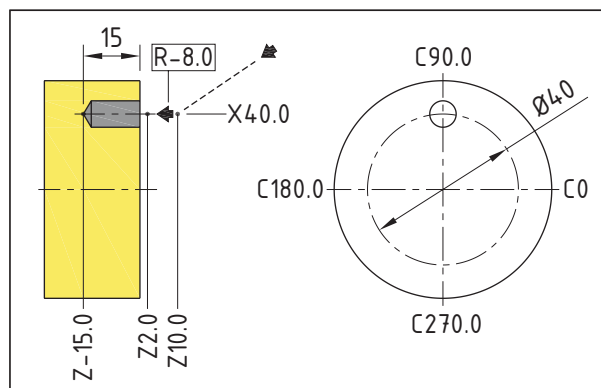


Figure 47-3

Concept of drilling cycle for face drilling using C-axis

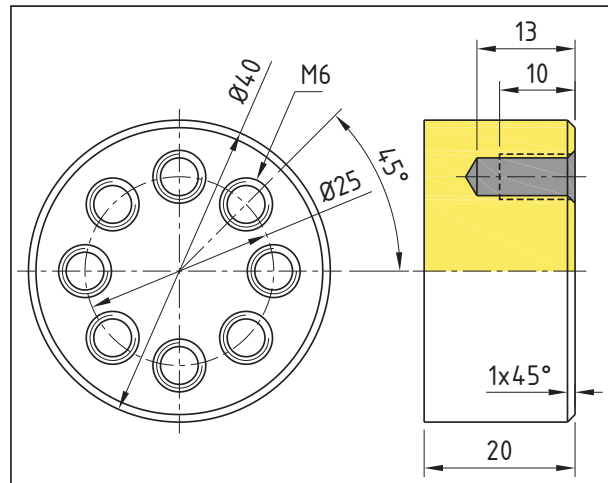


Figure 47-4

Drawing for program example O4701 and subprogram O4751

This simple concept can be applied to a complete example - see drawing in Figure 47-4. Three tools are used.

This example shows a typical C-axis programming when cutting direction is along the Z-axis. Three tools will be used - spot drill, drill and tap. G98 is feedrate per minute, G99 is feedrate per revolution. G83 cycle will be used as a spot drill (T01) with dwell of 300 ms (P300), and as a peck drill with 5 mm peck depth (Q5000). M6 metric tap size is the same as M6x1.

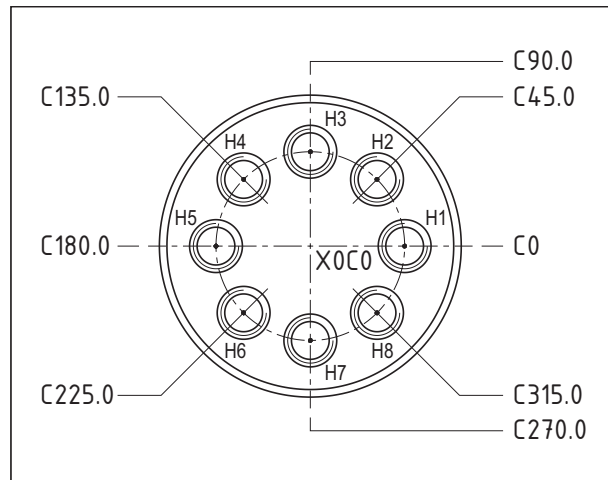


Figure 47-5

Hole positions on a face using C-axis (O4701 and O4751)

All holes start at C0 - hole H1 is machined at that position first - by all tools. Holes H2 through H8 are stored in the subprogram O4751. Another method could store all holes in the subprogram and call L0 or K0 in the cycle.



Note the clamping and unclamping of the C-axis. M29 is used in the example to select rigid tapping mode. This function can be replaced by another M-code or G-code.

```
O4701 (DRILLING ON FACE SURFACE)
N1 G21 (METRIC UNITS)
N2 G18 G40 G80 G98 (INITIAL SETTINGS)
N3 G54 (WORK OFFSET)
N4 G00 X200.0 Z100.0 (START POSITION)
N5 T0101 (T01 - DRIVEN SPOT DRILL - 10 MM DIA)
N6 G97 S1000 M103 (DRIVEN TOOL ROTATION CW)
N7 M14 (C-AXIS MODE ON)
N8 G00 Z5.0 M08 (INITIAL LEVEL)
N9 G28 C0 (ROTATE SPINDLE TO H1)
N10 G83 X25.0 Z-4.0 R-3.0 P300 F200.0 M12 (H1)
N11 M98 P4751 (H2-H3-H4-H5-H6-H7-H8)
N12 G00 X200.0 Z100.0 (INDEX POSITION)
N13 M01 (OPTIONAL STOP)
```

```
N14 T0303 (T03 - DRIVEN TAP DRILL - 5 MM DIA)
N15 G97 S1500 M103 (DRIVEN TOOL ROTATION CW)
N16 G00 Z5.0 M08 (INITIAL LEVEL)
N17 G28 C0 (ROTATE SPINDLE TO H1)
N18 G83 X25.0 Z-14.5 R-3.0 Q5000 F200.0 M12 (H1)
N19 M98 P4751 (DRILL HOLES H2 THROUGH H8)
N20 G00 X200.0 Z100.0 (INDEX POSITION)
N21 M01 (OPTIONAL STOP)
```

```
N22 T0505 (T05 - DRIVEN RIGID TAP - M6 X 1)
N23 G97 S250 M103 (DRIVEN TOOL ROTATION CW)
N24 G00 Z5.0 M08 (INITIAL LEVEL)
N25 G28 C0 (ROTATE SPINDLE TO H1)
N26 M29 (TURN RIGID TAPPING ON)
N27 G84 X25.0 Z-11.0 R-3.0 F250.0 M12 (H1)
N28 M98 P4751 (H2-H3-H4-H5-H6-H7-H8)
N29 M15 (C-AXIS MODE OFF)
N30 G00 G99 X200.0 Z100.0 (INDEX POSITION)
N31 M30 (END OF MAIN PROGRAM)
%
```

```
O4751 (SUBPROGRAM)
N101 M13 (C-AXIS UNCLAMP)
N102 C45.0 M12 (C-AXIS CLAMP + HOLE H2)
N103 M13 (C-AXIS UNCLAMP)
N104 C90.0 M12 (C-AXIS CLAMP + HOLE H3)
N105 M13 (C-AXIS UNCLAMP)
N106 C135.0 M12 (C-AXIS CLAMP + HOLE H4)
N107 M13 (C-AXIS UNCLAMP)
N108 C180.0 M12 (C-AXIS CLAMP + HOLE H5)
N109 M13 (C-AXIS UNCLAMP)
N110 C225.0 M12 (C-AXIS CLAMP + HOLE H6)
N111 M13 (C-AXIS UNCLAMP)
N112 C270.0 M12 (C-AXIS CLAMP + HOLE H7)
N113 M13 (C-AXIS UNCLAMP)
N114 C315.0 M12 (C-AXIS CLAMP + HOLE H8)
N115 G80 (FIXED CYCLE CANCEL)
N116 M13 M105 (DRIVEN TOOL - STOP ROTATION)
N117 M99 (END OF SUBPROGRAM)
%
```

Some machines have automatic clamping of the C-axis, and special M-functions are not necessary in such cases.

### ◆ Drilling on Diameter

Fixed cycles G87, G88 and G89 work on the same principle as cycles G83-G85, but X-axis is the drilling axis. In Figure 47-6 are four holes that will be drilled on diameter. This method of drilling is also called *circumferential hole machining*.

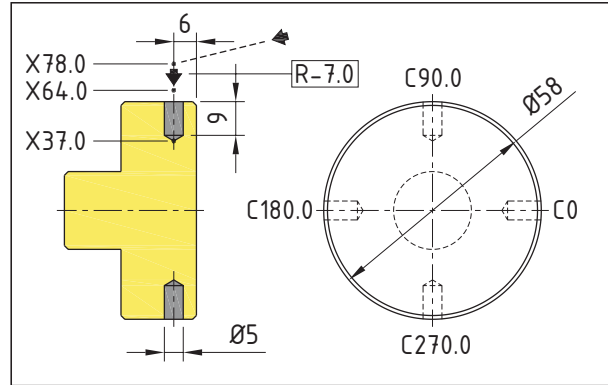


Figure 47-6

Holes drilled on diameter using C-axis - circumferential machining

A single tool program for the schematic drawing will be very similar to program showing face drilling:

```
M14 (C-AXIS MODE ON)
T0303 (TOOL SELECTION)
G97 S700 M103 (DRIVEN TOOL SPINDLE SPEED)
G98 G00 X78.0 Z-6.0 (FEED/MIN+START POSITION)
C0 (H1 ORIENTATION)
G87 X37.0 R-7.0 F150.0 (DRILL HOLE H1)
C90.0 (DRILL HOLE H2)
C180.0 (DRILL HOLE H3)
C270.0 (DRILL HOLE H4)
G80 G99 X200.0 Z100.0 M105 (INDEX POSITION)
M15 (SPINDLE MODE ON)
...
```

This example is shown without C-axis lock/unlock functions. *Note that the R-level is programmed as a radius value (per side)!*

### ◆ General Considerations

Here are some general comments to consider:

- Driven tool should be rotating before cycle is called
- All data in fixed cycles are modal, which means that repetitive data do not have to be written
- Although any command of G-code Group 1 (see page 55) will cancel the cycle, it is recommended to specify G80 as the best choice
- If Q-amount is set for peck drilling, specify it for each hole
- If P-dwell is set for the cycle, specify it for each hole



## Y-AXIS PROGRAMMING

Drilling or milling operation using the C-axis alone can only be done on a plane that intersect the machine centerline, following the XZ motions of standard lathe tools. When the C-axis is combined with the Y-axis (machine option), drilling and milling operations can be done off machine centerline. This machine specific feature greatly enhances machining flexibility. Compare the following two drawings - Figure 47-7 and Figure 47-8.

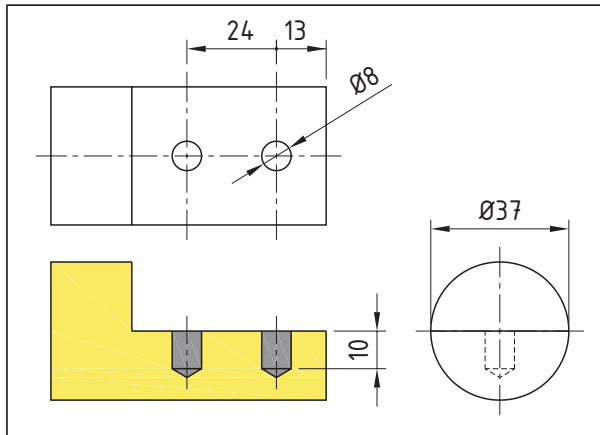


Figure 47-7

These holes can be machined with C-axis only

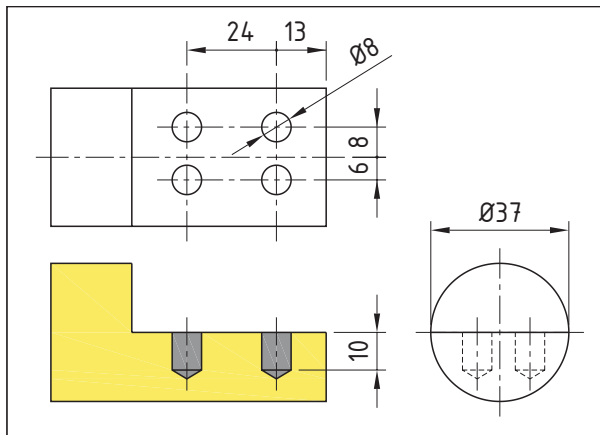


Figure 47-8

These holes can only be machined when Y-axis is also available

Y-axis motion is perpendicular to both X and Z axes, similar to standard vertical machining center. While the C-axis alone can be functional, Y-axis cannot exist without C-axis, which orients angular position of the spindle, therefore the part. C-axis can be locked in a specific angular position or unlocked for continuous rotary motion. This flexibility goes beyond simple machining, as it offers true contouring in four axes.

Figure 47-9 shows relationship of all four axes XYZC:

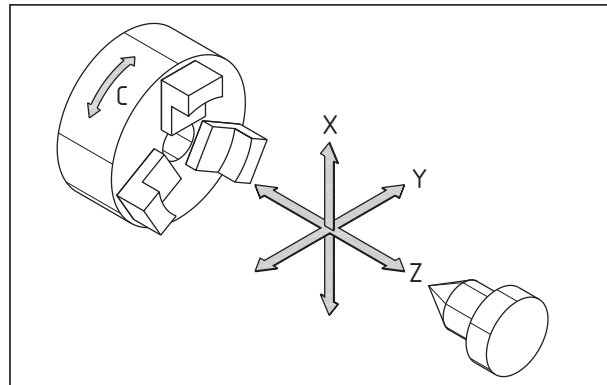


Figure 47-9

Axis orientation on a lathe with X, Y, C, and Y axes

Axis identification as Y-axis assumes absolute values, whereby identification of V-axis assumes incremental values. For example, one of the most common applications of the incremental mode is sending the Y-axis to its machine zero position, with no intermediate point:

N. . G28 V0

If not used, the Y-axis is locked at its Y0 position for regular turning and boring. When Y-axis is applied in the program, it must be unlocked (unclamped). Two machine specific functions are provided as examples for this purpose (check machine manual for actual codes):

☞ M470 - Y-axis lock  
M471 - Y-axis unlock

Simplified program for Figure 47-8 shows the technique (the flat already exists, and yes, it could also use the Y-axis):

```

M14                                (C-AXIS MODE ON)
T0505                              (TOOL SELECTION)
G97 S850 M103                      (DRIVEN TOOL SPINDLE SPEED)
M471                                (Y-AXIS UNLOCK)
G98 G00 X50.0 Z10.0                (FEED/MIN+START POSITION)
C0 Y8.0 Z-13.0                     (PART ORIENTATION)
G87 X-22.8 R-23.0 F150.0           (DRILL UR HOLE)
Y8.0 Z-37.0                        (DRILL UL HOLE)
Y-6.0 Z-37.0                      (DRILL LL HOLE)
Y-6.0 Z-13.0                      (DRILL LR HOLE)
G80 G99 G28 U0 W0 M105             (HOME POSITION XZ)
G28 V0                             (HOME POSITION Y-AXIS)
M470                                (Y-AXIS LOCK)
M15                                (SPINDLE MODE ON)

```

There are several similarities of programming the Y-axis on a turn/mill machine with machining center applications.

If necessary, C-axis can be programmed CCW (positive direction) or CW (negative direction). For some machines, the direction is controlled by an M-function.



### Polar Coordinate Interpolation - Example

This program matches dimensions in the last two illustrations. The depth of 15 mm is achieved by repeating the subprogram three times (5 mm incremental depth). Note the start position of Z-axis in block N11 - *it must be Z0*.

```
O4702
N1 G21                                     (METRIC MODE)
N2 G18 G40 G80 G98 (INITIAL SETTINGS+FEED/MIN)
N3 G54                                     (OFFSET SETTING)
N4 G00 X200.0 Z100.0 (INDEX POSITION)
N5 T0404 (T04 - DRIVEN END MILL - 16 MM DIA)
N6 G97 S1600 M103 (DRIVEN TOOL ROTATION CW)
N7 M14                                     (C-AXIS MODE ON)
N8 G00 Z5.0 (FRONT FACE CLEARANCE)
N9 X100.0 M08 (P0 - START DIAMETER)
N10 G28 C0 (C-AXIS - HOME POSITION RETURN)
N11 Z0 (POSITION FOR SUBPROGRAM)
N12 G112 (POLAR COORDINATE INTERPOLATION ON)
N13 M98 P4752 L3 (REPEAT SUBPROGRAM 3 TIMES)
N14 G113 (POLAR COORDINATE INTERPOLATION OFF)
N15 M15 (C-AXIS MODE OFF)
N16 G99 (FEED/REV)
N17 G00 X200.0 Z100.0 (INDEX POSITION)
N18 M105 (DRIVEN TOOL ROTATION STOP)
N19 M30 (END OF PROGRAM)
%
```

```
O4752 (SUBPROGRAM)
(X IS DIAMETER - C IS RADIUS)
N101 G01 G91 G94 Z-5.0 F1000.0 (DEPTH)
N102 G90 G41 X70.0 C8.66 F400.0 (P1)
N103 C-25.981 X30.0 (P2)
N104 X-30.0 (P3)
N105 X-60.0 C0 (P4)
N106 X-30.0 C25.981 (P5)
N107 X30.0 (P6)
N108 X70.0 C-8.66 (P7)
N109 G40 X100.0 C0 (P0)
N110 M99
%
```

There are no arcs in this example, but if an arc is machined in polar coordinate interpolation mode, the arc center is defined by the I-vector for X-axis and J-vector for C-axis. Address R can also be used in circular motions.

### ◆ Approximation Method

What are the options when polar coordinate interpolation is not available? There is one other method. This rather lengthy method is called *notching* and uses short program segments, working on the same principle as approximating special curves by using small increments in G01 mode.

Machining a polygonal shape is easy for manual programming, but for more complex shapes, a CAM software is highly recommended. Regardless of the programming method, a single segment of the hexagon can be programmed by dividing the flat into very small linear segments, each at a given degree increment.

The resulting program will be very large. For reference purposes only, the following is a typical appearance of such program generated by a CAM software for 1 inch flat on hexagon, with a lead-in - only one flat is shown (schematic illustration in Figure 47-12):

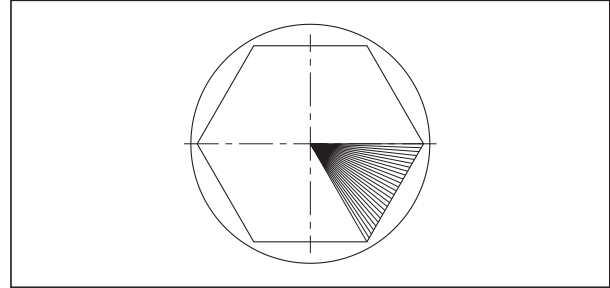


Figure 47-12

Schematic illustration of notching - segmented machining of a flat

```
G20
G28 U0 W0 H0
G98 T0808
G00 X4.6037 Z0.1
C5.343
G97 S650 M103
G01 Z-0.5 F8.0
X4.5017 C5.082 F40.11
X4.4012 C4.729 F54.28
X4.3027 C4.279 F69.09
X4.2069 C3.728 F84.56
X4.1141 C3.073 F100.63
X4.0251 C2.309 F117.18
X3.9404 C1.436 F134.07
X3.8606 C.452 F151.12
X3.7863 C-.643 F168.1
X3.7181 C-1.847 F184.74
X3.6566 C-3.154 F200.73
X3.6024 C-4.559 F215.73
X3.556 C-6.053 F229.36
X3.4792 C-9.083 F240.88
X3.4129 C-12.223 F250.99
X3.3574 C-15.462
X3.3132 C-18.786 F267.94
X3.2804 C-22.177
X3.2595 C-25.617 F278.76
X3.2503 C-29.083
X3.2531 C-32.554
X3.2679 C-36.007
X3.2943 C-39.42
X3.3324 C-42.774
X3.3817 C-46.049 F264.48
X3.442 C-49.23
X3.496 C-51.627 F247.64
X3.5561 C-53.947
X3.5825 C-55.012
X3.6038 C-56.098
X3.6199 C-57.201
X3.6306 C-58.316
X3.636 C-59.438
...
```

There is yet another application of the C-axis ...

## CYLINDRICAL INTERPOLATION

C-axis can also be used together with the Z-axis for simultaneous machining. This provides synchronized motion of the linear Z-axis with C-axis rotation of the part (spindle). The most common use for this type of machining is cylindrical grooving for such many applications as cam guide grooves. In order to program cylindrical grooves with lathe live tools, the control has to have a feature called cylindrical interpolation.

### ◆ Z + C-axis

To use cylindrical interpolation option in a program, one of two G-codes are provided:

G07.1	Cylindrical Interpolation
G107	Cylindrical Interpolation

For the purposes of the example, G107 will be used, although both have the same programming format. Before evaluating a simple but typical example, keep in mind that there are some rules and specifications of using G107 or G07.1 commands:

- C-axis represents radius (for example, G107 C76.9) (usually at the bottom of groove)
- Z-axis represents linear motion (mm or inches)
- X-axis controls the depth diameter (specify before)
- Rapid motion G00 cannot be used
- Fixed cycles cannot be used
- For arc movements, I/K vectors cannot be used - use radius R only (mm or inches)
- Working plane is defined by axes Z and C (ZC plane, using G19, if necessary)

Engineering drawing of the required groove is provided as a flat representation of the groove. This shape will be 'rolled around' - or 'wrapped' - around the machine cylinder. This is similar to a flat piece of dough that is rolled around a rolling pin. If the drawing was developed on a CAD system, designers may also include an isometric view of the part, but for programming purposes, the flat representation is enough.

### ◆ Practical Example

To illustrate the concept of cylindrical interpolation, a 10 mm wide groove will be machined all the way around a cylinder, at 152 mm bottom diameter. The groove changes direction at 90° for 50 mm, and returns back to its original track at 215°. All four corners of the centerline have a 5 mm radius. Figure 47-13 provides typical engineering data, which are generally not to scale.

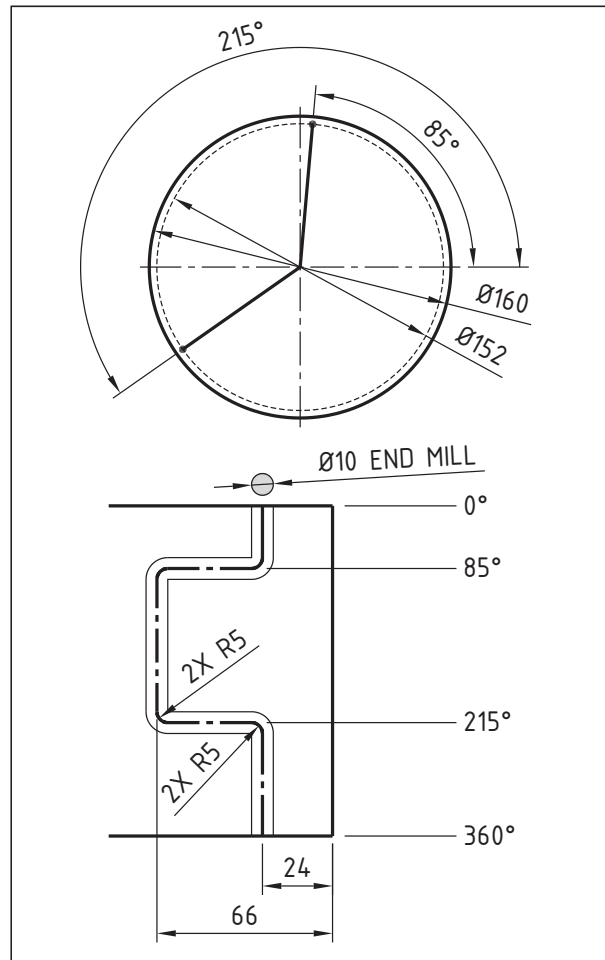


Figure 47-13

Typical flat layout of a cylindrical groove - program O4703

Program requires ZC coordinate locations for all contour change points of the center line. While the Z-coordinates are all known, C-axis requires additional angles for the R5 radius. This is usually the more difficult part of any job.

All C-axis data are angles of rotation in degrees. In order to convert flat (planar) data to radial data, some mathematical calculations will be necessary. The most important one is calculation of circle circumference, sometimes called the length of circle:

$$\text{Circumference} = \pi \times D$$

where  $\pi$  is constant of 3.14159 and D is diameter

For the example, D is the bottom diameter of the groove, specified in the drawing as 152 mm. Therefore,

$$\pi \times \text{Ø}152 = 477.5221 \text{ mm circumference}$$

Good planning makes a big difference - start with points.

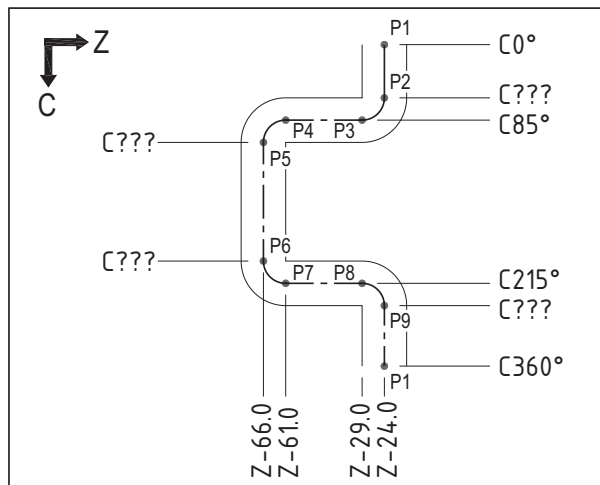


Figure 47-14

Flat layout with known and unknown values - program O4703

Figure 47-14 shows all known and unknown data. All Z-axis positions are known, but there are four radial positions (angles) that have to be calculated.

For the Z-axis, the end points of all four arcs are known in millimeters. In order to do a conversion, all units have to be the same, so the first step is to convert the two given angles (85° and 215°) to length in millimeters using the previously calculated circumference of 477.5221 mm:

$$85 / 360 \times 477.5221 = 112.7483 \text{ mm}$$

$$215 / 360 \times 477.5221 = 285.1868 \text{ mm}$$

Once these dimensions are known, the linear equivalent of all unknown points can be calculated by adding or subtracting radius R5:

$$\text{For P2 ... } 112.7483 - 5 = 107.7483 \text{ mm}$$

$$\text{For P5 ... } 112.7483 + 5 = 117.7483 \text{ mm}$$

$$\text{For P6 ... } 285.1868 - 5 = 280.1868 \text{ mm}$$

$$\text{For P9 ... } 285.1868 + 5 = 290.1868 \text{ mm}$$

There is one last step - to convert these calculations back to angular values that will be programmed as C-axis along with the two angular values provided initially. The key to this conversion is the following relationship:

$$360 / \text{Circumference} = \text{Angle} / \text{Length}$$

where angle is unknown and length is one of the values calculated for the four unknown points. The formula can be reversed to calculate the angle of rotation:

$$\text{Angle} = 360 / \text{Circumference} \times \text{Length}$$

For the example, previously calculated circumference is 477.5221 mm, so  $360 / 477.5221 = 0.75389$ , therefore:

$$\text{C-axis angle for P2} = 0.75389 \times 107.7483 = 81.231^\circ$$

$$\text{C-axis angle for P5} = 0.75389 \times 117.7483 = 88.769^\circ$$

$$\text{C-axis angle for P6} = 0.75389 \times 280.1868 = 211.231^\circ$$

$$\text{C-axis angle for P9} = 0.75389 \times 290.1868 = 218.769^\circ$$

Now, all C and Z data are known and program can be written from the new layout - Figure 47-15.

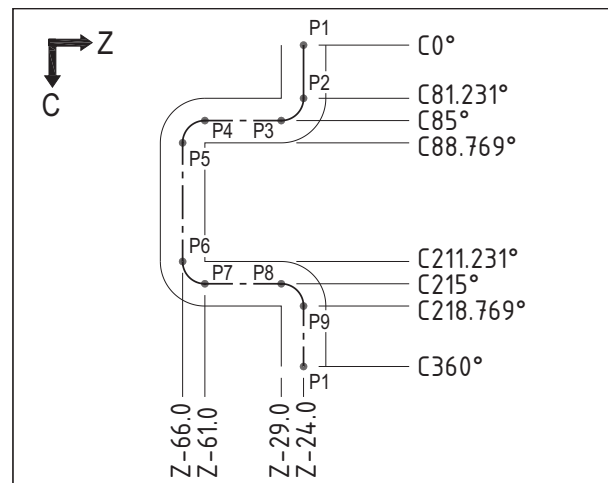


Figure 47-15

Flat layout with all values known - ready for program O4703

```

O4703
N1 G21 (METRIC MODE)
N2 T0606 (T06 - DRIVEN END MILL - 10 MM DIA)
N3 G97 S725 M103 (DRIVEN TOOL ROTATION CW)
N4 M14 (C-AXIS MODE ON)
N5 G107 H0 (SAFETY SETTING)
N6 G00 X170.0 Z-24.0 (X-CLEAR ABOVE START)
N7 G98 G01 X152.0 F75.0 (GROOVE BOTTOM - P1)
N8 G107 C76.0 (INTERPOLATION ON - PART RADIUS)
N9 C81.231 (POSITION P2)
N10 G02 Z-29.0 C85.0 R5.0 (POSITION P3)
N11 G01 Z-61.0 (POSITION P4)
N12 G03 Z-66.0 C88.769 R5.0 (POSITION P5)
N13 G01 C211.231 (POSITION P6)
N14 G03 Z-61.0 C215.0 R5.0 (POSITION P7)
N15 G01 Z-29.0 (POSITION P8)
N16 G02 Z-24.0 C218.769 R5.0 (POSITION P9)
N17 G01 C360.0 (POSITION P1)
N18 G107 C0 (CYLINDRICAL INTERPOLATION OFF)
N19 X170.0 (CLEAR ABOVE PART)
N20 G99 G00 X200.0 Z100.0 (INDEX POSITION)
N21 M15 (C-AXIS MODE OFF)
N22 M30 (END OF PROGRAM)
%
```

This simple example covered the basic approach to programming cylindrical interpolation. Like all subjects in this chapter, programming is very dependent on the actual machine design. For that reason, always consult your machine documentation for any special information.



## 48

# WRITING A CNC PROGRAM

Writing a CNC program is the final result of manual programming. This last step requires a sheet of paper, or many sheets of paper, that contain the part program. Program itself is composed of individual instructions related to machining and arranged in a series of sequential blocks. Writing does not mean using only a pen or pencil, on the contrary. Instead of paper, modern writing methods employ a computer and a text editor, but the result is still a written and or printed copy of a manually generated part program.

Manual program development is the result of a lot of hard work. A short program with a few lines of code may be as easily entered into the control directly as to be written down on paper. However, the written copy will often be required for documentation and other reference purposes.

The need to program by hand seems somewhat backwards in the age of computers, printers and other hi-tech wonders, but it is a method that will not disappear any time soon, at least not for simple programs. Writing a part program manually requires time and is always subject to errors. Manual work means work by hands, so it seems that a need for special computer skills is not required. Is that a correct assessment?

In the traditional way, program can be written with a pencil and a paper (and a five pound eraser, as an old cartoon claimed). Its final form is transferred to the control unit, a short program may be keyed into the system directly, by pressing various keyboard keys. For long programs, this approach is a waste of time. Modern alternative to a pencil is a computer keyboard, using a simple text editor to make a plain ASCII text file, with no formatting. Computer software creates a CNC program as a file stored on the hard drive. This file can be printed or send directly to CNC machine. The only difference is that computer keyboard has replaced the pencil and editing features of a text editor have replaced the eraser. Even today, a large amount of manual programming work in is still done in writing, using a devices such as pens, pencils, calculators and erasers.

Regardless of media used, learn how the computer - the control system - interprets a written program, what syntax to use, what to avoid and what format is correct. Even if not programming manually at all, it is important to know the principles of program writing techniques, in order to make changes or customize output that was developed by a CAD/CAM system, if necessary.

CNC program should be written in such a way that it can be interpreted without a difficulty

## PROGRAM WRITING

Writing all collected data into a final version of CNC part program is one of the last items of programming process. To get to this stage requires hard work through all other stages - but only when all thoughts have been collected, all decisions have been made and a certain level of comfort has settled in. In the previous chapters, emphasis was on program development as a logical process. Now, the focus will shift to the actual method of writing CNC program, following this logical process.

Writing a part program is based on two initial factors:

- Corporate standards ... company decides
- Personal style ... you decide

Both factors can be adapted simultaneously in a single program - they are fully compatible. It is unreasonable to expect any *industry or world-wide standards* relating to various techniques of developing a program. It may be even less reasonable to set any *company based standards*, unless there is a general set of rules and recommendations already in existence.

The final result is that the first guiding factor - company standards - is replaced by the second factor - personal style. From an objective point of view, there is nothing wrong at all with a personal style of programming. If the program works, who cares how it was done. From a revised point of view, it needs to be acknowledged that a CNC programmer can never succeed in isolation. Programming involves at least one user of the final program - CNC operator - and that makes programming, in effect, a true team work.

The most common problem with uncontrolled personal style is *inconsistency*. Any CNC machine shop that employs - or plans to employ - more than one programmer, should establish certain minimum standards for preparation of a part program. Adherence to these standards allows any team member to pick up where another member has left. Often, personal style of the first programmer in the company will carry on and on and eventually becomes 'the' company standard, for better or worse. Such a situation may well be very positive, but in most cases it needs revaluation or at least a bit of modernizing.

To define a company standard, first evaluate some suggestions and practical observations that may be helpful to prepare a program efficiently for any style that may be suitable to follow and useful in the future.

### ◆ Legibility of Handwriting

Writing a program without any assistance of a computer and a text editor, means writing a CNC program in pen or pencil, by hand. A hand written program is easier to correct without a mess and it should be double or even triple spaced when written on a sheet of paper. Individual words in a program block should be separated by a space, to further enhance legibility. This way, any additions or future changes (if they become necessary) can be made quite easily, yet still keeping the overall appearance of paper copy clean and neat. Problems with legibility of a manually generated program are much less of a factor, if the program is typed directly into a computer text file or using a CNC editor, such as the one included with NCPlot™ software. Even in those cases, the printed copy may be illegible for technical reasons, such as a low printer toner, for example.

### ◆ Programming Forms

In the early years of numerical control, special programming forms were popular with pre-printed columns for each address in a block. Those were the days when only numerical values were entered into the appropriate column and the column position determined the address meaning. These programming forms were often issued by control and machine manufacturers, as an aid to program writing and a little promotion on a side. Today, a ruled sheet of a standard size paper is fully sufficient. No special columns are needed and if a column or two is justified, it can be drawn easily enough. Modern programs use alphanumeric representation, writing the whole word - alpha characters as well as numeric characters and special symbols. This process is much more economical - hardly any machine manufacturers provide programming forms any more.

Some CNC programmers do not necessarily do the final program version themselves. Some shops consider such work a secretarial responsibility. That means somebody else (an assistant) will read the hand written copy and has to be able to read it correctly, the way it was intended. Such a person may have no knowledge of CNC programming and may not be able to detect even simple syntax errors.

### ◆ Confusing Characters

Legibility of programmer's handwriting is very important. Make a special effort when writing certain characters (alphabetical or numeric) that can be interpreted in more than one way. Depending on personal handwriting, some characters can be confusing to the reader. For example, a handwritten letter O and digit 0 can look the same. Digit 2 and the letter Z can be also confusing. The letter I and the digit 1 as well as a low case letter l are other examples. These are only some of the most obvious examples, but many other characters can also be confusing, depending on each person's handwriting. Try to develop a consistent writing technique to distinguish potentially confusing characters (confusing is a relative term, of course).

For instance, all computers and printers (even the old tape preparation systems) use a special type style to distinguish individual characters on screen and in print. In this handbook there is an obvious difference between a wide letter O (as in **O1111**) and a narrow digit 0 (as in **O0001**).

The same technique should be applied to personal handwriting. Take advantage of the fact that there is no letter O used on most controls anywhere, except as a program number and in a comment section, where a misprint will not create a problem anyway. If preferred, find a special designation only for the letter O and the rest is all digits 0 by default - unless hundreds of zeros are identified specifically and in a unique way in every part program.

O	or	Ø	_____	Digit zero
0			_____	Letter O
1			_____	Digit ONE
I			_____	Letter I
2			_____	Digit TWO
Z			_____	Letter Z

Figure 48-1

Special form of characters written to prevent ambiguity

Illustration in Figure 48-1 shows some *suggested* methods of common character distinction in handwriting. Find a personal way to write any characters that may improve handwriting legibility. Regardless of which method is selected as personally preferable, adhere to it - *be consistent*. There is nothing worse than adopting different 'standards' for every new program. Programmer or the person who prepares the program final version, will be more than confused and eventually may make a serious error.

Handwritten method can be bypassed entirely by keying in the program data via control keyboard. Such program can then be optimized, part machined and verified program sent out when the job is finished. This procedure may tie up the machine for a while and is not recommended as an everyday method of inputting programs into the control system. The best and fastest method is to prepare part program in a text editor on a computer and send it directly to the CNC machine, through cable connection or memory card.

Only a small minority of CNC users today use punched tape, and even then, it is usually for old machines only. More modern methods are available, such as disk storage of a desktop or laptop computer. Through an interface between computer and the machine, data can be transferred either way very reliably, thus eliminating punched tape and other methods altogether. Whichever method is selected as preferable, the program still has to be properly formatted.

# PROGRAM OUTPUT FORMATTING

Those who followed this handbook from its beginning, chapter by chapter, should be well familiar with programming by now. This section deals with the actual program format - not *its contents*, but how it *appears* on the printed paper or screen of the computer. It will evaluate four versions of the *same* program. Identical in every respect, except its outward appearance. Feel free to be the judge as to which of the four format versions is the most suitable and why. A rather long program is presented - intentionally - this is an actual program. It is not important what it does, only how it looks when printed or displayed. Each new version presented improves the previous version.

➡ Program **Version 1** - Imperial units used :

```
G20
G17G40G80G49
T01M06
G90G54G00X-32500Y0S900M03T02
G43Z1.0000H01M08
G99G82X-32500Y0R1000Z-3900P0500F80
X32500Y32500
X0
X-32500
Y0
Y-32500
X0
X32500
G80G00Z1.0000M09
G28Z1.0000M05
M01
T02M06
G90G54G00X-32500Y0S750M03T03
G43Z1.0000H02M08
G99G81X-32500Y0R1000Z-22563F120
X32500Y32500
X0
X-32500
Y0
Y-32500
X0
X32500
G80G00Z1.0000M09
G28Z1.0000M05
M01
T03M06
G90G54G00X-32500Y0S600M03T01
G43Z1.0000H03M08
G99G84X-32500Y0R5000Z-13000F375
X32500Y32500
X0
X-32500
Y0
Y-32500
X0
X32500
G80G00Z1.0000M09
G28X32500Y-32500Z1.0000M05
M30
%
```

This is the most primitive way of writing a program. Although it may offer some doubtful benefits, it is the least friendly version. Good program, yes, but with a very poor appearance. It is extremely difficult for CNC operator to read this mess.

➡ Program **Version 2** - Imperial units used :

```
N1G20
N2G17G40G80G49
N3T01M06
N4G90G54G00X-3.25Y0S900M03T02
N5G43Z1.0H01M08
N6G99G82X-3.25Y0R0.1Z-0.39P0500F8.0
N7X3.25Y3.25
N8X0
N9X-3.25
N10Y0
N11Y-3.25
N12X0
N13X3.25
N14G80G00Z1.0M09
N15G28Z1.0M05
N16M01
N17T02M06
N18G90G54G00X-3.25Y0S750M03T03
N19G43Z1.0H02M08
N20G99G81X-3.25Y0R0.1Z-2.2563F12.0
N21X3.25Y3.25
N22X0
N23X-3.25
N24Y0
N25Y-3.25
N26X0
N27X3.25
N28G80G00Z1.0M09
N29G28Z1.0M05
N30M01
N31T03M06
N32G90G54G00X-3.25Y0S600M03T01
N33G43Z1.0H03M08
N34G99G84X-3.25Y0R0.5Z-1.3F37.5
N35X3.25Y3.25
N36X0
N37X-3.25
N38Y0
N39Y-3.25
N40X0
N41X3.25
N42G80G00Z1.0M09
N43G28X3.25Y-3.25Z1.0M05
N44M30
%
```

This is definitely a much improved version of the same program. Look at what a simple block numbering and inclusion of decimal points does to overall program legibility. This program version is far from being final, but it does offer some very tangible improvement. Decimal points in programs are of course standard, except for old controls.

The next program version applies all improvements done so far and addresses some additional issues.

☐ Program Version 3 - Imperial units used :

```

N1 G20
N2 G17 G40 G80 G49
N3 T01 M06
N4 G90 G54 G00 X-3.25 Y0 S900 M03 T02
N5 G43 Z1.0 H01 M08
N6 G99 G82 X-3.25 Y0 R0.1 Z-0.39
N7 X3.25 Y3.25
N8 X0
N9 X-3.25
N10 Y0
N11 Y-3.25
N12 X0
N13 X3.25
N14 G80 G00 Z1.0 M09
N15 G28 Z1.0 M05
N16 M01
N17 T02 M06
N18 G90 G54 G00 X-3.25 Y0 S750 M03 T03
N19 G43 Z1.0 H02 M08
N20 G99 G81 X-3.25 Y0 R0.1 Z-2.2563 F12.0
N21 X3.25 Y3.25
N22 X0
N23 X-3.25
N24 Y0
N25 Y-3.25
N26 X0
N27 X3.25
N28 G80 G00 Z1.0 M09
N29 G28 Z1.0 M05
N30 M01
N31 T03 M06
N32 G90 G54 G00 X-3.25 Y0 S600 M03 T01
N33 G43 Z1.0 H03 M08
N34 G99 G84 X-3.25 Y0 R0.5 Z-1.3 F37.5
N35 X3.25 Y3.25
N36 X0
N37 X-3.25
N38 Y0
N39 Y-3.25
N40 X0
N41 X3.25
N42 G80 G00 Z1.0 M09
N43 G28 X3.25 Y-3.25 Z1.0 M05
N44 M30
%
```

This version is much improved. It uses all improvements of the previous version, yet adds a significant improvement - *spaces between words*. Still, it is difficult to visually identify the start of each tool. The next version will add a blank line between tools. Spaces do not impose an extra drain on CNC memory, yet the program is much easier to read.

☐ Program Version 4 - Imperial units used :

```

(DRILL-04.NC)
(PETER SMID - 07-DEC-08 - 19:43)

(T01 - 1.0 DIA - 90DEG SPOT DRILL)
(T02 - 11/16 TAP DRILL - THROUGH)
(T03 - 3/4-16 TPI PLUG TAP)
```

```

(T01 - 1.0 DIA - 90DEG SPOT DRILL)
N1 G20
N2 G17 G40 G80 G49
N3 T01 M06
N4 G90 G54 G00 X-3.25 Y0 S900 M03 T02
N5 G43 Z1.0 H01 M08 (INITIAL LEVEL)
N6 G99 G82 X-3.25 Y0 R0.1 Z-0.39 P0500 F8.0
N7 X3.25 Y3.25 (HOLE 1)
N8 X0 (HOLE 2)
N9 X-3.25 (HOLE 3)
N10 Y0 (HOLE 4)
N11 Y-3.25 (HOLE 5)
N12 X0 (HOLE 6)
N13 X3.25 (HOLE 7)
N14 G80 G00 Z1.0 M09 (HOLE 8)
N15 G28 Z1.0 M05
N16 M01
```

```

(T02 - 11/16 TAP DRILL - THROUGH)
N17 T02 M06
N18 G90 G54 G00 X-3.25 Y0 S750 M03 T03
N19 G43 Z1.0 H02 M08
N20 G99 G81 X-3.25 Y0 R0.1 Z-2.2563 F12.0
N21 X3.25 Y3.25 (HOLE 1)
N22 X0 (HOLE 2)
N23 X-3.25 (HOLE 3)
N24 Y0 (HOLE 4)
N25 Y-3.25 (HOLE 5)
N26 X0 (HOLE 6)
N27 X3.25 (HOLE 7)
N28 G80 G00 Z1.0 M09 (HOLE 8)
N29 G28 Z1.0 M05
N30 M01
```

```

(T03 - 3/4-16 PLUG TAP)
N31 T03 M06
N32 G90 G54 G00 X-3.25 Y0 S600 M03 T01
N33 G43 Z1.0 H03 M08
N34 G99 G84 X-3.25 Y0 R0.5 Z-1.3 F37.5 (HOLE 1)
N35 X3.25 Y3.25 (HOLE 2)
N36 X0 (HOLE 3)
N37 X-3.25 (HOLE 4)
N38 Y0 (HOLE 5)
N39 Y-3.25 (HOLE 6)
N40 X0 (HOLE 7)
N41 X3.25 (HOLE 8)
N42 G80 G00 Z1.0 M09
N43 G28 X3.25 Y-3.25 Z1.0 M05
N44 M30
%
```

The final version (*Version 4*) may be a luxury for some users, but it is the most elegant of all four. It adds initial descriptions and messages to the operator. It includes programmer's name and date of the last update. It also includes description of all tools at the program beginning. It uses the same tool descriptions for individual tools, at the beginning of each tool section, where it matters most.

Some lower level controls do not accept comments in the program. If there are comments in the program, such a control system will strip them automatically during loading.



## LONG PROGRAMS

Those who ever worked with a punched tape, used to run the program directly in a reel-to-reel operation. The maximum program length equaled the maximum length of tape that fitted on the reel, about 900 feet - or 275 meters - or 108000 characters. With today's modern equipment, there is no need for a tape anymore, most part programs will run from memory of the CNC system. Unfortunately, even that memory capacity is finite as well, often well below what the tape capacity used to be. It all means that a situation may arise, when a particularly long program will not fit into the memory. In addition to a good directory cleanup, there are two other possibilities to eliminate this problem.

### ◆ Program Length Reduction

The simplest way to reduce a program length is to eliminate all *unnecessary* characters from such program. Since the problem is related to a *long program*, the reduction in length will be much greater than can be illustrated here. There is a number of areas that should be considered before taking the proverbial red pen and starting the changes:

- Eliminate all unnecessary leading or trailing zeros (G00 = G0, X0.0100 = X.01, ...)
- Eliminate all zeros programmed for convenience (ex.: X2.0 = X2.)
- Eliminate all or most block numbers
- If using block numbers, increments by one will make a shorter program
- Join several single tool motions into a multiaxis tool motion, if safety allows
- Use default control settings, but check them first
- Do not include program comments and messages to the CNC operator
- Use comments and various descriptions on a separate piece of documentation

Organizing the programming process will definitely help - for example, include as many instructions in a single block as possible, rather than dividing them into many individual blocks. Use subprograms if possible, use fewer tool changes, even fewer tools, if that is possible, etc. At the same time, watch for undesirable side effects when eliminating or deviating from an established program format.

There is no doubt that many of these measures will result in some compromise between convenience and necessity. When thinking well ahead and organizing the work properly, the results will be worth the effort.

These methods are shortcuts and should be used for emergency situations only, not as standard programming procedures

To illustrate at least some shortcut methods, compare the two following examples - both will have identical results - well, *almost* identical results:

O4801 (TYPICAL PROGRAM - METRIC)  
N10 G21 G17 G40 G80 G90  
N20 G54 G00 X120.0 Y35.0  
N30 G43 Z25.0 H01  
N40 S500 M03  
N50 M08  
N60 G99 G81 X120.0 Y35.0 R3.0 Z-10.0 F100.0  
N70 X150.0  
N80 Y55.0  
N90 G80 G00 Z25.0  
N100 M09  
N110 G28 X150.0 Y55.0 Z25.0  
N120 M30  
%

A grand total of 194 characters have been programmed. The condensed version of this program needs only 89 characters, with a minor compromise. Program in this form is more memory efficient but much harder to read - remember this is only a short sample, not a real long program in its entirety, where the difference would be more impressive:

O4802  
G90 G0 X120. Y35.  
G43 Z25. H1 S500 M3  
M8  
G99 G81 R3. Z-10. F100.  
X150.  
Y55.  
G80 Z25. M9  
G91 G28 X0 Y0 Z0  
M30  
%

Total of 54.12% of program length have been saved in a rather very short program. Shortening program length may become useful in some cases, so here are several methods that have been used in the above example:

- Program description has been eliminated
- Block numbers have been eliminated
- G21, G17 and G54 have been eliminated (correct settings assumed on the control - *be careful!*)
- Zeros following a decimal point in full number have been canceled
- Some blocks were joined together
- G80 G00 has been replaced by G80 only (G00 is redundant, although commonly used)
- Leading zeros in G00, M08, M09, H01, and M03 have been removed
- Machine zero return has been changed from absolute mode to incremental mode
- ... *Keep in mind, this is a no-frills program only!*



Although both program examples will result in a part machined according to drawing specifications, some programming instructions will be processed differently. A very important change can be achieved in tool approach towards the part. In the first example, usually a standard version, the motion command positions the X and Y axes first, with the Z-axis motion following in a separate block. In the shorter example, the same order of motions has been preserved for safety reasons. If machining conditions allow, these two motions can be combined into a single motion. G43 and G54 commands can work quite well together in the same block, without a problem:

```
G90 G0 G43 G54 X120. Y35. Z25. H1 S500 M3
```

Always be careful to consider machine setup first and guarantee a safe approach towards or away from the machined part. If any obstacles come in the way because of any shortcut, the condensed example would be a wrong programming method and should be avoided.

Program preparation and its actual writing will become a daily routine very soon after establishing a personal programming style. If using a computer, learn how to write the program directly from the keyboard - it is a waste of time to write it by hand first. It may take a little time getting used to, but it is well worth it.

### ◆ Memory Mode and Tape Mode

Most CNC system have a special *Mode Switch* selector to choose from at least two options - MEMORY mode and TAPE mode. *Memory mode* is used most frequently - program is loaded into the CNC memory, it is edited from the memory, and is run from the same memory. Tape mode is, of course, to run a program from a tape and many users ignore the possibilities this mode offers. Even if not using punched tapes in the machine shop anymore, (majority of

companies do not), tape mode can be used to *emulate* a tape with many added benefits. Tape mode is not to be taken literally any more. Think of this mode as an *external* mode (DNC, for example), not in its actual old fashioned sense.

In order to use this external mode requires a little extra hardware and software. On the hardware side, only a reliable personal computer is needed, with a fair size hard disk storage capacity. A properly configured cable that will connect the computer with the CNC is also required. The computer may even be an 'abandoned' slower computer from the office - all that is needed is a minimum configuration, nothing fancy. On the software side, an inexpensive communications software is necessary, to send a program from the computer to the CNC and back.

Once everything is configured to work together, store the CNC program or programs on the hard disk of a personal computer, load the software and work with the CNC system as usually! The major difference is in editing. Since the program actually resides on the hard disk of a PC, use the computer and a text editor to edit a CNC program in that environment, not at the control system. The capacity of current hard drives is *x-times* more than will ever be needed. Aircraft companies, many mold shops, tool and die shops and other industries that require extremely long programs have embraced this technology long time ago, and very successfully, too.

Also consider this method for the *High Speed Machining* programs. This relatively new technology uses very high spindle speeds and feedrates but very small depths of cut. This combination means extremely long programs, many that will not fit into any memory configuration of any CNC system. So before investing into rather expensive control memory updates, investigate this method of running a part program from a personal computer, if the transfer speed is fast enough.

## 49

# PROGRAM DOCUMENTS

During program preparation, quite a number of individual pieces of documentation will accumulate. All sketches, calculations, setup sheets, tooling sheets, job descriptions, instructions to the operator and related notes contain valuable information. This information should be stored as part of the program documentation folder. Any changes to the finished program at a later date, for whatever reason, can be done much easier if the documentation is complete, well organized and stored in one place. A good documentation makes a review of the program at a later date much easier. If somebody else has to review a program, the documentation will save much of the valuable time. The way programmers document programs reflects not only their personal programming style, it also becomes a reliable indication of their sense of discipline and organizational capabilities.

A simple definition relating to program documentation can be presented:

Program documentation is a set of all records necessary to retrace the program development

Many CNC programmers, even machine shop supervisors, underestimate the importance of good program documentation. Their main arguments are that any paperwork is not worth the time, that it takes too long to collect all documents and prepare the documentation, that it is essentially a nonproductive effort, etc. These arguments are true, up to a point - in order to make a good documentation, yes, some time will be required. Not an excessive amount of time, but enough time to do a good job. If there are prepared blank forms available, they just need to be filled. It does not take any more time than writing the same information on any other piece of paper - it can actually take a lot less time. If a CAD system is available, use it to develop a customized tooling library and setup sheet. A variety of blank forms can be predefined, then filled quickly whenever they are needed. CAD system will save time, it makes the program documentation neat, and every sketch drawn in scale can be easily retraced. Using a word processing or a spreadsheet software is another way to save time required for documentation.

In essence, the purpose of a program documentation is to communicate programmer's ideas to somebody else or to review them at a later date. Creating documentation is not a directly productive work, and does not require to take too much extra time. Documentation may be a good investment in time management, it can save a lot of time over a few months.

## DATA FILES

A complete part program is not only a hard copy of the program or program data (usually stored on a disk). Examples of the important documents mentioned here are all a vital part of the program. They create a set of all files used for programming, called the *data files*.

All of these files are useful to the programmer, but only some are important to the CNC machine operator or the setup person. A number of files are only for reference, and are not normally sent to the machine shop. Two basic rules for data files can be established:

- Programmer keeps all the files
- Machine operator gets copies of relevant files only

These two rules guarantee that the ultimate responsibility for program control remains with CNC programmer. There is no need to duplicate every piece of documentation for the machine shop - only those items that relate to the actual machining have a place in the shop. Unnecessary duplication is counterproductive and should be avoided. The only items of documentation needed in the machine shop are:

- Part drawing
- Program printout
- Setup sheet
- Tooling sheet

*Part drawing* serves as a reference for comparison of the intended shape, dimensions, tolerances, etc., with the actual product. Only the drawing version that was actually used for programming should be considered. *Program printed copy* is the program listing made available to the machine operator, if required. Normally, it is the printed output of the program itself. Two remaining items, *setup sheet* and *tooling sheet* describe programmer's decisions relating to the part setup and the cutting tools selection. Each is also a complement to the program itself. In some cases, other (special) documentation has to be included as well, not mentioned here. Any piece of paper or a note that is considered important should be included.

Those who have written programs in a high level language C++, *Visual Basic*, etc) or in older languages such as *Basic*, *Pascal* or even *AutoLISP* (the original programming language for AutoCAD), know that they can add comments within the body of the program.

These comments are usually terse, just long enough to remind the user of what is happening in the program. If more information about a specific program is necessary, most likely there would be additional instructions, even a user's manual. This kind of external and internal program documentation applied in software development, is also adaptable to a CNC program.

## PROGRAM DOCUMENTATION

The difference between an external and internal program documentation deserves some explanation. Is one better than the other? Which one should be used?

The best documentation is one that combines both types for maximum effect. To distinguish between the two types, let's evaluate them individually.

### ◆ External Documentation

External documentation of a CNC program generally consists of several items and always of their latest version - this last statement is very important.

The following items are typical to any program documentation. They can be used (or ignored) as desired:

- Program copy printout
- Methods sheet, if available
- Part drawing
- Working sketches and calculations
- Coordinate sheet
- Setup sheet
- Tooling sheet
- Program data (disk or other media)
- Special instructions

Program copy printout is the final version of the programming process. It should be the *exact* contents of the program stored on a disk or other media. In those machine shops that use routing or methods sheets, the programmer should make it a policy to include a copy of any methods sheet in the documentation as well. Part drawing (or its copy) is extremely important to be kept together with the program. It is the ultimate reference source for the future. All sketches and calculations, together with a sheet of coordinates, are also useful at a later date, particularly if the program has to be revised for some reason. Setup sheet and tooling sheet will be discussed shortly.

That only leaves the program data source (usually stored on a disk or similar media) to be included in the documentation folder and any special instructions that may be required by the programmer, CNC machine operator or somebody else.

### ◆ Internal Documentation

Program internal documentation is contained *within the body of a program* itself. When writing a program, make an effort to strategically place comments into the program. Such messages are integral part of the program and are categorized as *internal program documentation*. These messages are either separate blocks of a program or additions to individual blocks (delimited by parentheses) and can be actually seen on the display screen during program execution (on most controls). They are also printed in a copy of the program. The greatest advantage of internal documentation is the convenience offered to CNC machine operator. The only disadvantage is that when loaded into the control memory, comments do occupy memory space. If the available memory is scarce, be modest with program comments and place more emphasis on external instruction methods. All program comments, messages, directions and instructions must be enclosed in parentheses:

(THIS IS A COMMENT, MESSAGE OR INSTRUCTION)

Parentheses are the required program formatting. Either comment, message or instruction can be an individual block in the program or it can be part of a program block. Control system will ignore all characters between the parenthesis. To avoid long descriptions internally, use pointers to external documentation instead - for example:

```
N344 ...
N345 M00 (SEE ITEM 4)
N346 ...
```

ITEM 4 in the program comment section will be a detailed description that relates to block N345, somewhere else in the program documentation, such as in a setup sheet. This kind of reference is useful when the message or comment would be too long to be stored in the program body.

For example, CNC operators may find the referenced ITEM 4 in the setup sheet, under a unique heading called *Special Instructions*:

```
...
ITEM 4. - Remove part, clean jaws, reverse and
clamp on the 120 mm diameter
...
```

Properly prepared internal documentation should always briefly describe each cutting tool used:

```
N250 T03
N251 M06
(T03 = 1 INCH DIA 4-FLT E/M)
N252 ...
```

Note that T03 is the *current* tool. This designation will vary greatly, depending on tool changing methods of a particular machine tool builder. Also note the use of abbreviations in the program comment - **4-FLT E/M** is a short form for a *4-flute end mill*.

Every time *Program Stop* command M00 is used in the program, document the reason why it is used:

```
N104 G00 Z1.0
N105 M00 (CHECK DEPTH = 0.157 INCHES)
```

N106 ...

For longer messages, make the comment a separate block:

```
N104 G00 Z1.0
N105 M00
(DEPTH TO SHOULDER MUST BE 0.157 INCHES)
```

N106 ...

Comments can be in the same block as program data:

```
N12 G00 X3.6 Z1.0
N13 M05 (CLEAN CHIPS FROM THE HOLE)
N14 ...
```

If the comment is written as a separate line, block number for the comment block is usually not used. Instructions that are part of the documentation should be clearly understood. Enigmatic or cryptic messages will not do. Distinct messages translate into time saving by the CNC operator or the setup person and they contribute to a quicker turnaround between individual jobs.

### ◆ Program Description

On many Fanuc systems, program *description* can also be documented. This is a special kind of a comment, also included in parentheses. There are some conditions that make the program description special.

- Description must be included in the same block as program number
- Description must have no more than 15 characters
- Low case characters will not be accepted

A typical example of a program description may include drawing name and/or number in the comment section:

```
O4901 (FLANGE-DWG. 42541)
```

Once these conditions are followed, program number can be viewed along with its description right on the directory screen (program listing) of the control system.

If an additional description that does not fit the 15 characters limit is needed, enter more comments in subsequent blocks. They will not be seen on the directory screen, but can still be handy for internal program documentation. They will be displayed during program processing on all controls that accept comments. The length of these comments is not usually limited to 15 characters:

```
O4902 (RING-OP.1)
(DWG. A-8462 REVISION D)
(PETER SMID - 07-DEC-08)
N1 ...
```

The main purpose of program documentation is to transfer all important decisions and ideas from the developer to the user. In CNC programming environment, documentation transfers all ideas from the programmer's desk to the machine shop. It serves as an important link within the overall communication process.

## SETUP AND TOOLING SHEETS

Apart from program printed copy and part drawing, *setup sheet* and *tooling sheet* are the other two most important pieces of good program documentation. The major difference between a setup sheet and a tooling sheet is the subject emphasized. Setup sheet is a sketch or a drawing that shows the part layout and orientation on the machine table or in a fixture, possibly even the description of individual operations. Tooling sheet usually lists only cutting tools and their mounting positions, along with spindle speeds, feedrates and offsets for each tool. Examples of both types are shown in this chapter.

One ongoing question many programmers always have is the reminder of a *Catch 22* variation:

*'Do I make the setup sheet and tooling sheet before or after writing the CNC program?'*

As is usual in many programming applications, adherents and opponents can be found on both sides of the issue. The favorite reasoning for making documentation *before* writing a single line of a program is quite simple. Setup sheet and tooling sheet are the guiding forces for writing the program, the forces of being well organized. Adhering to this method implies a well organized programmer or a team, it implies that everything is under control. It also suggests that all fixtures and tools and holders and inserts and other tools are already available in the machine shop, ready and waiting to be used. No doubt, and if at all possible, always make both setup sheet and tooling sheet, *before* starting actual program development. Logic behind the reasons for this method is very strong indeed.

Logic itself, however, does not take into consideration machine shop realities, even in cases when they are essentially not the best. A small conflict between departments, a delay in material delivery, tool on back order and similar problems, all contribute to the frustrations of CNC programmers in collecting initial data. Being under pressure from all sides, the programmer has no choice but to *improvise*, even in times of crisis. Programmer has to *compromise* - in terms of setup/tooling sheet development, it also means a certain downgrading. If there is no choice, always try to find a *reasonable* compromise, but never as an excuse for being sloppy.



The freedom in program development is considerable but it is not unlimited. Normal part program cannot be written without knowing the machine setup and tooling to be used. In many cases, the nature of the job offers many solutions. Even if the *exact setup*, or *exact tool* to be used are not known, think of other ideas, have some opinions - but have ideas and opinions based on some experience (yours or of others). Compromise does not rest with the 'now or later' situation, it rests in the selection of the most *likely* possibility. If something has to be changed, make sure the changes will be minimal. In any case, it is quite possible that the setup sheet and/or the tooling sheet will have to be modified *after* the program has been proven and optimized.

### ◆ Setup Sheet

In many shops, setup sheets are luxury. It is a simple statement of fact, but many setup sheets are quite poorly prepared if they are prepared at all. Often, they do not reflect the latest program changes and adjustments, they are not consistent between individual machines and even programmers. Although the time spent on preparing a setup sheet is considered nonproductive from the cost angle, it is a time far from being wasted. Setup process can be organized, certain rules can be set and adhered to and they can be applied to the preparation of a good setup sheet.

The golden rule of a good setup sheet is *to always make it in scale*. Setup sheet using an outline of the material, fixtures layout, finished shape, tool path, etc., should always be done in scale. Scale, even an *approximate* scale, is very important for visual comparison. Clamps and other mounting devices should be drawn in positions corresponding to the actual setup. Tool change location should be marked accurately, different views shown, if necessary. Critical positions should be dimensioned, indicating the maximum or minimum distances.

If cutter radius offset is used, speeds and feeds reflect a certain *nominal* cutter radius. At the discretion of machine operator, cutter radius may be changed within a reasonable range. This range should appear in the setup sheet, including a note on the adjustment of speeds and feeds.

In cases when the cutter exceeds a certain length, it may interfere with the part or other tools. In these cases, setup sheet should include the maximum cutter length allowed within that setup. For chuck work on a lathe, the maximum material jaw grip should be specified in the setup sheet.

Setup sheet has one purpose - to document details of how a particular part is mounted on the machine. It has to cover part holding method and reference point relationships (part, machine, and cutting tool). It has to describe positions of auxiliary devices used, such as tailstock, barfeeder, vise, face plate, hard and soft jaws, and others. A setup sheet template may be useful for each machine. A simple setup sheet is shown in *Figure 49-1*. Feel free to improve it as necessary.

Setup Sheet			
Program reference: ANGLE BRACKET	Date: 07-Dec-08	Programmer: Peter Smid	Sheet: 1 of 1
TOP		FRONT	

Figure 49-1

Simple setup sheet form - only basic data shown

A well designed setup sheet should also include information about the material (stock) used for machining, stock the program is based on. Not only the type of stock, also its rough dimensions, amount left for machining, its condition, and other features that are important to include in program documentation. This information is very valuable at its conception and will be even more valuable in the future, mainly for repeated jobs. Many times, a program is made when the stock is not yet available. If the programmer finds out later that there is too much deviation from the estimated conditions, the necessary changes are easier to make with good program documentation.

Although not a strict requirement, some programmers include cutting time for each machining operation on the setup sheet. When a job is run for the first time, the actual cutting time is unknown. As the program is used and optimized, it becomes proven and eventually finalized, the cutting time becomes known with more precision. Knowing the overall cutting time may help in planning the load work on the CNC machine. The most useful cutting time for an individual part is the *chip-to-chip* time that includes all supplementary times (for example tool change time, part replacement time, etc.), not only the cutting time itself.

### ◆ Tooling Sheet

Although tooling is really part of the setup, it requires a separate set of data, that may or may not fit on the setup sheet itself. If setups and tools used are constantly simple, it may be more convenient to have only one sheet, describing them both. However, for large or complex setups, making a separate tooling sheet is more practical. Both, the setup sheet and tooling sheet, are part of the same documentation and complement - *not replace* - each other.

Machine unit and its CNC system influence the contents of a tooling sheet. Tooling sheet for a lathe will be different than tooling sheet for a machining center. Data gathered for either machine will have some similarities and some unique items. A sample contents of a typical tooling sheet will include description of the following items:



- Machine and program identification
- Cutting tool type
- Tool coordinate data
- Tool diameter
- Insert radius and the tip number
- Offsets associated with the tool
- Tool length
- Tool projections from the holder
- Block number of the tool being indexed
- Brief description of the tool operation
- Basic speed and feed of the tool
- Tool holder description
- Tool number and/or tool station number
- Special instructions

In addition to the most common items, also include any unique information in the tooling sheet, for example, to inform the operator about non-standard tools, tools that require modification, premachined condition of the material, etc. An example of a simple tooling sheet is in *Figure 49-2*.

[illegible]

Figure 49-2

Simple tooling sheet form - only basic data shown

### ◆ Coordinate Sheet

The idea of a coordinate sheet is not new. It has been used in programming from its very beginning and it has been mentioned in this handbook many times already. A simple printed form containing the X, Y and Z axes can be used for both machining centers and lathes. *Figure 49-3* shows a good example of a simple coordinate sheet.

The Z-axis column will be usually blank for machining centers and Y-axis column will be blank for lathe programs. Modify the sheet to add additional axes or make separate sheets for each machine type.

*Printable forms are included on the CD-ROM.*

[illegible]

Figure 49-3

Simple coordinate sheet form - only basic data shown

**DOCUMENTATION FILE FOLDER**

All records that have been collected during program preparation are quite likely to be important enough to be kept for future reference. They may be stored all over the place, sometimes very hard to find. So, now is the time to put them all together and organize them. It is time to make a file folder, identify it, fill it up and store it properly.

## ◆ Identification Methods

Before some better methods of identifying program documentation can be suggested, think about a very popular, yet quite an impractical method. Some programmers use the program number as a reference for all related material. The basic thinking behind this idea is that the available program number range between 1 and 9999 (or even 1 to 99999) will take forever to use up, therefore becomes very useful for some identification purposes. This is a shortsighted thinking, usually by not a very busy programmer who has only one or two machines to take care of.

Look at possible problems with this way of thinking. True, to make almost ten thousand programs (or more) for one machine, it would take 'almost forever'. Even if more machines are available, at a rate of 25 programs a week, numbers will run out in a little more than 7 years. Is that the time to scrap the machine and buy a new one? And if 25 programs a week seems a bit steep, remember that *each* program will have to have a number. That may be three or more separate operations *for a single job*, there may be dozens and dozens of subprograms that also need their own program number. So the figures are not so unreasonable after all, and some better method should be sought from the beginning. It could be a manually generated method, or a comprehensive computerized database.

A few basic questions remain - what is the best method to employ available program numbers? Is it necessary to employ them at all? Is there an alternative?

The point of this evaluation is that all program number assignment (with the exception of subprograms), should be left exclusively to the *CNC operator*; if at all possible. That means, another way has to be found to identify the documentation containing all records.

One of the first decisions is the program name selection. Regardless of the number of machining operations or included subprograms, there should be only one folder for one job and only one name for one folder. The name of one folder should share the common denominators with any other folder. Try to make such a name *meaningful*.

With an access to a personal computer, the chances are that all files relating to each program are stored in the computer. In that case, the only limiting factor is software format structure to name the files. For example, the old DOS software files accepted up to eight alpha numeric characters for the file name and another three alpha numeric characters for the file extension. Since Windows 95, long file names are allowed, up to 255 characters plus extension - try to take advantage of this feature. Regardless of which CNC system is used, establish a file naming convention conforming to any possible restrictions. There are several methods for this approach.

One is a totally independent, sequential order. In this simplest form, all documentation related to the first program would be - for example - P0000001, the next program would be P0000002, etc. If the zeros are removed from this format, the files will *not* be listed in a correct alphabetical order on a computer display. No practical limit is imposed here. Another approach would be to use the current drawing number as the basis of documentation identification. This may a good method for the many companies that manufacture their own products, but not much suitable to various jobbing shops. Dealing with many different customers also means dealing with many different types of drawing numbers. The variety may be so great that it is almost impossible to find some common ground for standardization. Another variation on the same theme is a *job number*, rather than a drawing number. In many jobbing shops, each job gets a number assigned the moment the order is received. This *Job Number* is always unique, therefore a good candidate to be used as the number identifying program documentation.

Hopefully, the presented ideas will stimulate many additional ideas that will suit a particular work environment. There are no given rules on the methods of identifying an individual program; there are no rules governing the standard of part program documentation. The old reliable rule is always use the old fashioned *common sense* that is often not so common. Common sense and foresight help any standardization effort. The quality of any standard is measured by its usefulness in the future. For the longer period of time a particular standard can be useful, the better quality of thought has gone into its development.

### ◆ Operator's Suggestions

When CNC machine operator runs the job, he or she may have comments, ideas, corrections and variety of other suggestions that come from observation of how the program works. It may be a good idea to consider establishing a log book, a card system, a computer database, or a similar method of communicating the operator's ideas back to the CNC programmer. Whatever system may be selected, it should be available at the machine, so the CNC operator has a primary access to it. The main benefit of such a system is that all communication goes into one source and is easier to keep under control.

Apart from the nature of a particular comment or idea, the log should have the operator's name, current date, perhaps even current time, machine and job description, as well as any other details that may be relevant and useful anytime in the future, particularly when the part is run again.

### ◆ Filing and Storage

Physical program file folders can be quite bulky, particularly when they contain computer media, such as disks, large size paper drawings, long program printouts, etc. The storage of file folders is usually confined to standard office steel filing cabinets, which should be accessible to *every work shift*, although only qualified and authorized persons should be given the actual access.

If using any kind of computer media for storing the part programs, make sure they are safely stored in a separate container, rather than the file folder itself. Magnetic devices are particularly sensitive to adverse conditions and should be stored away from any heat source and magnetic field (including a telephone, for example). They should be kept in a dry and dust free environment. Keeping duplicates (or even triplicates) in a separate place is also a good and safe procedure.

Much more practical - and much less bulky - than physical paper is storage of proven programs is storage on a CD (Compact Disc) or a DVD (Digital Versatile Disc), using a standard hardware and software. Although these disks still require to be stored away from all heat sources, they are not a subject of magnetic fields and take much less space. Incorporating Adobe PDF (*portable document format*) files is another reliable option that saves space.

In physical storage, individual sheets or pages of the part program documentation should be either numbered consecutively, or have a reference number on each page. Drawers of the filing cabinets should be identified as to their contents. These are common enough requirements, but very often ignored all together, usually *because there is no time*. The main philosophy behind an orderly filing system is the speedy access to a required program that provides instant and accurate information.

## 50

**PROGRAM VERIFICATION**

When a CNC program is completed, there will be a written copy or a file copy stored somewhere on a computer. At this point, the program development is fully completed. It may be a perfect program with no errors. Of course, that was the intent from the beginning - to make an error free program. What happens if - *in spite of the best efforts* - there is an error? Even a small typing error can cause a severe problem when the program runs on the machine. Could an error be prevented? And if so, how?

Every program should be checked against all predictable errors *before* it reaches the machine. Checking can be quite simple, such as a visual comparison of the written copy and the printed copy. The main purpose of a program check is to detect obvious mistakes - mistakes that can be seen by *concentrating* and *looking for them*. The kind of errors detected first are mostly syntax errors. Of course, there is no guarantee that the program is error free, but when it does leave the programmer's desk, the effort should be to *make it* error free. All programs arriving at the machine should gain the confidence of CNC operator. Operator should be free to concentrate all efforts on proving the program sequences and run the first part. CNC operator has no time to check for program errors that could - and should - have been detected in the 'office'. To do all program proving on the machine is nonproductive and should be avoided.

**DETECTION OF ERRORS**

Before an error of any kind can be corrected, it must be discovered first. In any type of programming, errors can be found either *before* or *after* the program leaves programmer's desk. Intentions of any professional programmer are undoubtedly to discover any errors *before* they are detected during program execution, at the CNC machine. This is a *preventive* effort. If an error has to be corrected at the machine, during actual program run, CNC operator has to do something that should not normally be part of his or her job. Whatever action it is the operator must take, it is always a *corrective* action. Therefore, there are two measures that can be taken to help eliminate errors in CNC programs:

- Preventive measures      ... *proactive measures*
- Corrective measures      ... *reactive measures*

Preventive measures are those that all parties involved should participate in with suggestions and constructive criticism. On the other hand, corrective measures require certain skills, knowledge, and even some authority.

**◆ Preventive Measures**

All errors should be detected and corrected by the programmer, who has taken a certain amount of preventive measures. The first preventive measure is to *get organized*. Set up *procedures*, set up *standards*, set up *rules*. Then, follow them diligently. Errors that can be found before a program is used on the machine are numerous. Yet, it takes unique techniques to become successful in their detection.

The first method any programmer should use is simple - *check your own work*. Read the program and *evaluate* it. If the rules of consistency had been followed, the error check is easy. Programmers know the appearance of a program, its structure, the established standards, the order of various commands at the beginning and end of each tool. Checking the program itself should not take much time at all.

The second method can be successfully applied when working with other programmers or skilled operators. Ask a co-worker to look through any new or changed program. Do not be surprised to see mistakes such a check can reveal. A fresh, detached, and impartial look can be very productive. Sometimes, even taking a little break or fresh air first will rejuvenate the brain cells.

With a computer and a special simulation software, such as the enclosed *NCPlot™* (on CD), there is a third option - make a graphic representation of the toolpath on the screen.

A major part of preventive measures is finding *syntax errors*. A syntax error is one that can be detected by the control unit. For example, if a dollar sign appears in the program, the control will reject it as illegal. The control returns an error message or an '*alarm*'. If the digit 2 is entered in the program instead of the intended digit 7, that is *not* a syntax error. That is a *logical error*; since both are legitimate characters the control can accept.

**◆ Corrective Measures**

If an error is discovered at the control, the preventive check did not uncover it. An error that is found at the machine slows down production. It forces the machine operator to take corrective measures and eliminate such error. Operator can take one of two actions. One will be to return the program to the programmer, another action will be to correct the error at the machine. Which choice is better depends on the error seriousness. An error can be *soft* or *hard*. A soft error is one that does not require stopping the program from being processed by the CNC system.

For example, a missing *coolant* function M08 in the program can be switched on manually at the machine, without interrupting program processing. That is an example of a soft error - it is still an error, but classified as a *minor* error.

A hard error occurs when the program processing must be stopped by the operator, *as the only available choice*, and without doing a damage to the machine, cutting tool, part, or all of them. A common example of a hard error is a programmed tool motion that cuts in the wrong direction. Program itself is wrong and must be corrected. This is an example of a hard error, classified as a *major* error.

Most CNC operators do not like delays, especially delays caused by somebody else. A dedicated machine operator will do anything possible to correct a problem without any assistance. For program errors, the operator will try to take corrective measures to eliminate the problem. Not every operator is *qualified* to do even a simple change to the program. On the other hand, some qualified operators may not be *authorized* to do program changes as a matter of policy.

Every company benefits greatly, if the CNC operator has *at least* a basic training in CNC programming. The purpose of such a training is not to make the machine operator a fully qualified CNC programmer. Its purpose is to highlight how a part program influences CNC machining, the setup, tooling and all the other relationships between programming and machining. Its purpose is to offer the operator tools that can be used for minor program changes, etc. Such a training, if it is designed and delivered in a professional manner, is always a worthwhile investment. It may be a relatively short training that will pay for itself very quickly. Time delays on CNC machines are costly and the sooner the program is made functional, the less damage to the production control has been done.

Whenever a program has been changed at the machine, the program documentation must reflect these changes, particularly if they are permanent. Even a small permanent change should be always be documented in *all* copies of program documentation.

## GRAPHIC VERIFICATION

Programming errors can be costly, even if their cause is a minor human error. Omitted minus sign, a misplaced decimal point, an illegal character - all are minor oversights that cause major errors. Although a visually checked program should be error free, that may not always happen. The human eye is weaker when it evaluates non-graphic elements.

One of the most reliable methods of part program verification is a graphic display of the toolpath as it appears in the program. Almost all errors relating to the toolpath can be detected early, by one of three available graphic verification methods.

One method of graphic verification of a CNC program is a screen plot. This *optional* control feature will show all programmed tool motions on the control screen. Tool motions will be represented as lines and arcs. Feedrate motions will appear as a solid line of the selected color, rapid motions will appear as a dashed line. Display of the toolpath will appear on the control screen.

Many controls offer a graphic simulation option, where the toolpath is simulated on the screen. Each cutting tool can be shown by a different color or density, making the visualization easier. Some graphic simulation uses actual tool shape and the part for a realistic display. The negative part of any graphic verification is that it can only be used when the program is loaded into the control.

The second verification method is much older than the first. It is a hard copy plotted representation of all cutting tool motions. Hard copy plotting has been available in computer programming for a long time. To get the benefits of hard copy plotting, a pen plotter and a suitable software will make it work. Plotter is seldom a problem in companies using CAD software but may not be available to small machine shops. Required software is also part of a large computer based programming system and can be quite expensive. A simple version of a pen plotted toolpath is a screen dump, usually to a printer. Pen plotting in CNC is not a common activity.

There is a third method of graphic verification and can be off-machine. It uses a computer software designed to read a manually generated program, then displaying it on the computer screen. Typical software of this kind is *NCPlot™* - software with many advanced features that is included on the CD-ROM as a shareware.

## AVOIDING ERRORS

The goal of every programmer is to write error free programs. That is almost impossible, since any human activity is subject to occasional errors. Programmers with all levels of experience make mistakes, at least once a while.

Since prevention of errors should be the main goal of any programmer, this section looks at the subject in more depth. Some most common mistakes will be evaluated, along with suggestions to prevent, or at least to minimize, their happening. First, what exactly is a program error?

Program error is the occurrence of data in a program that will cause the CNC machine to work contrary to the intended plan or not to work at all

All errors can be classified into two groups:

- Syntax errors
- Logical errors



Although the average distribution of programming errors could be generally split at 50/50 between syntax and logical errors, certain conditions may swing the balance. A programmer with limited experience will make all kinds of errors. An experienced programmer makes more syntax errors. Let's look at each error group.

### ◆ Syntax Errors

Errors in this group are usually easy to deal with, once they are identified. Syntax error is simply one or more characters in the program that are either misplaced or do not belong there. This error covers program entries that do not conform to programming format (known as *syntax*) of the control system. For example, a two-axis lathe control systems do not accept the character Y. If such control encounters the letter Y in a lathe program, it will reject it as a syntax error and the program won't run. The same result will happen when the letter U is programmed for most milling controls. Some other letters cannot be used with either system - for instance the letter V - it is an illegal character for most milling and turning controls. Yet, it is very legal character in a four axis wire EDM control.

Syntax errors also occur if valid characters are used in combination with an option not supported by the control. A good example is a custom macro, an option on most Fanuc controls. Custom macros use a number of standard letters, digits and symbols, but also a number of special symbols, for example a sharp sign #, brackets [ ], asterisk *, etc.

Macros also use some special words, such as COS, SIN, GOTO and WHILE, words not allowed in a standard program. Macro symbols or words in a non macro program result in a syntax error. The error will also occur when the custom macro option is available, but some of the symbols are used incorrectly or the special words are misspelled.

Control system handles syntax errors arbitrarily - *it simply rejects them*. This rejection is displayed as an error message on the screen. Program processing will stop. Syntax errors are irritating and embarrassing, but almost harmless. Scrap as a result of a syntax error is possible but rare. The second group - the *logical* errors - is much different.

### ◆ Logical Errors

Logical errors are more serious than syntax errors. A logical error is defined as an *error causing the machine tool to act in a way contrary to the programmer's intentions*. If a motion is programmed to the absolute coordinate of X1.0, but program states X10.0, the control will go ahead but the tool position will be wrong. The same error will happen when Z10.0 is programmed, although the intent was X10.0. The control *does not* and *cannot* have any built-in protection against logical errors. Programmer has the responsibility to exercise all necessary care and caution. Logical errors can be serious - they may not only result in a scrap, they can damage the machine and even harm the operator.

Logical errors cover an unlimited number of possibilities. For example, the following lathe program *is wrong*:

```
O5001
(EXAMPLE WITH ERRORS)
N1 G20 G40 G99
N2 G50 S2500 T0400 M42
N3 G96 S530 M03
N4 G00 G41 X12.0 Z0.1 M08
/ N5 G01 X-0.06 F0.012
/ N6 G00 Z0.2
/ N7 X12.0
N8 Z0
N9 G01 X-0.06
N10 G00 Z0.1 M09
N11 X20.0 Z5.0 T0400 M01
...
```

There are three errors in the O5001 example. Try to identify them before reading any further.

The first error should be easy - a *tool offset* is missing. In block N2, tool T0400 is selected without an offset. This block is correct. Block N11 is the return to the indexing position, with expected tool offset cancellation, which was never programmed. The error is in block N4 - it should be:

```
N4 G00 G41 X12.0 Z0.1 T0404 M08
```

The second error is rather hidden and requires a trained eye to spot it. Note that a block skip symbol was used in blocks N5, N6 and N7. When running the program with the block skip setting ON, cutting feedrate will be missing in block N8. In this case, the control would issue an error message, but only for the *first time* this program is processed. The correct block N8 should be:

```
N8 Z0 F0.012
```

The third error is the missing cutter radius offset cancel in block N11. This block should correctly be written as:

```
N11 G40 X20.0 Z5.0 T0400 M01
```

An error of this kind may have a serious implications for the next tool. Even worse, this error may not be discovered during the first part run. The correct program is O4902:

```
O5002
(EXAMPLE WITHOUT ERRORS)
N1 G20 G40 G99
N2 G50 S2500 T0400 M42
N3 G96 S530 M03
N4 G00 G41 X12.0 Z0.1 T0404 M08
/ N5 G01 X-0.06 F0.012
/ N6 G00 Z0.2
/ N7 X12.0
N8 Z0 F0.012
N9 G01 X-0.06
N10 G00 Z0.1 M09
N11 G40 X20.0 Z5.0 T0400 M01
...
```



After evaluating these logical errors, what chances are there that the control will detect them and return an error message? *Nil, zero, zilch*. All errors in the example are good illustrations of logical errors. They may not always be easy to find, but they can create additional problems if not found early.

## COMMON PROGRAMMING ERRORS

Strictly speaking, there are no '*common*' programming errors. Every programmer makes some unique mistakes. It is difficult to list any errors as being more common than others. It is also true, that some mistakes are made more frequently than others and in that sense they are more common. Focusing on this group should be beneficial.

Both syntax and logical errors share the same cause - *the person who writes the program*. The most important step towards eliminating errors is the identification of a problem - ask yourself '*what mistake do I do repeatedly ?*' Everybody makes some 'favorite' mistakes, the solution lies in the correct answer to this simple question.

Most errors are results of insufficient program planning and a lack of precise programming style. Planning offers a sense of direction, style offers tools and organization.

The simplest - *and the most frequent* - error is an omission of some fundamental instruction. It may be a coolant function, program stop, a missing minus sign and others. Even the whole block may get lost, mainly when preparing the program from poor sources. Many errors are caused by the programmer's inability to *visualize* what will exactly happen when the program is processed. To this category belong all errors relating to setup, tooling and machining conditions - cuts that are too heavy or too light, insufficient clearances and depths, incorrect spindle speeds and cutting feedrates, even the selection of wrong tools for a given job.

### ◆ Program Input Errors

Most programs are hand written or typed and have to be transferred to the control system or a computer file. Many errors are caused by the *incorrect input of intended data*. Keep in mind that if somebody else is using the program, its legibility (if hand-written) and syntax is very important.

Input errors also include errors caused by forgetting to input significant characters in the program. These strings can be almost anything and can cause a serious problem. A missed coolant function is not likely to cause a big problem; a missed decimal point or a wrong tool retraction will. Other errors are insufficient tool clearances, a depth that is too shallow or too deep, errors relating to cutter radius offset (this is always a big group). Be also careful when canceling or changing modal program values. One common error is to cancel one kind of motion by replacing it with another type of motion in one block, then forgetting to reinstate the previous motion later.

### ◆ Calculation Errors

Using math functions and formulas is an integral part of developing CNC programs manually. Calculation errors include a wrong numeric input, even when a pocket calculator is used. Keying a wrong formula, wrong arithmetic sign or placing parentheses in a wrong position, all represent a serious error.

#### *Rounding Error*

A special type of an error is caused by *incorrect rounding*. This error is an accumulative error that results from too many dependent calculations. A rounded value used in other calculations may lead to an error. In many cases the error will be too small to cause any problems, but never count on it. It may become a very bad habit.

#### *Calculations check*

To prevent math errors when using formulas for calculations, it is a good idea to check the calculated result once more, using a *different* formula. Math is a generous subject and more than one calculation method is usually possible.

### ◆ Hardware Errors

The last type of program errors is by the *malfunction of a hardware element* of control system or machine. In CNC, even a bug in the software is possible. Their occurrence is rare, as modern controls are very reliable. When encountering an error, don't blame the control or the machine *as the first and only possible cause*. It shows a degree of ignorance, arrogance, and unwillingness to address the problem responsibly. Before calling for a service, make sure to exhaust all other possibilities of error detection first.

### ◆ Miscellaneous Errors

Some errors can be traced to the part drawing. An error in a drawing is possible, but first make sure to interpret any drawing correctly. Drawing errors include too many or too few dimensions, poor tolerances, non-matching dimensions, etc. Also make sure to use the latest drawing version.

Other errors may be caused by the wrong setup, tooling or material. These are not programming errors, but they have to be considered as possibilities. With some common sense and suitable precautions, many programming problems can be eliminated. For example, to prevent an unproven program to be processed as a proven program, just mark it as unproven. Mark it at the *beginning* of the program and leave it there until the program is checked.

A complete elimination of errors is not realistic. Mistakes do not happen - *but mistakes are always caused*. Inexperience, negligence, lack of concentration, poor attitudes, are just some causes. Always program with the attitude to eliminate programming errors altogether. That will be the first step to making fewer errors.

# 51

## CNC MACHINING

When a part program is fully completed and sent to the machine shop, programming process is over. All calculations have been done, program has been written and well documented, the program file is on the way to the CNC machine. Is the programmer's job really finished? Is there some reason that could bring the program back, perhaps with operator's comments, suggestions, or even criticism?

If the delivered part program is *perfect*, programmer will not hear a word from any direction. No doubt, programmer will hear negative comments from *all* directions. The question is - when is the programmer's responsibility *really* over? At what point in the process of manufacturing can the programming results be evaluated? When can the program qualify as a *good* program?

Probably the fairest and the most reasonable answer would be *whenever the part has been machined under the most optimized working conditions*. This means that the programming responsibility *does not* end with the program and documentation delivery to the shop. The program at this stage is still very much in the development process. It still has to be loaded to the CNC system, the machine has to be set up, cutting tools mounted and measured and a variety of small jobs done before the first part can be started. True, all these tasks are the responsibility of CNC operator, so there is no need for the programmer to care what happens during machining, right?

*Wrong!* Every CNC programmer should make an effort to be in constant touch with the actual production. In the field of business software development, it is quite normal to have a team of people to work on a certain large programming project. After all, most programming ideas come from talking to colleagues and the actual users of the program or particular software. The same is true for CNC programs used in machine shops. The users of programs are typically CNC machine operators - they can be a gold mine of constructive ideas, improvements and suggestions. Talk to them, ask questions, make suggestions, and - most important - *listen* to what they have to say. Programmers who never put their foot in the machine shop or go there reluctantly, programmers who may go there with their eyes closed and ears plugged, programmers who take the attitude that they are always right, are all on the wrong track. Exchanging ideas with machine operators, asking questions and seeking answers is the only way to be fully informed about what is actually going on in the machine shop. It is in programmer's interest to know how the CNC operators *feel* about the program, the programming style and the approach to programming overall. *Do* exchange

ideas and *do* communicate with each other - that is the best advice for becoming a better CNC programmer. Machine shop offers tremendous resources, take advantage of them.

CNC technology is an instrument to improve productivity with a minimal human involvement, measured at least by the physical level. As any other technology, it must be managed intelligently and by qualified people with experience. Without a firm grip and good control, without good management, the technology will not yield the expected results - in fact - in will become counterproductive.

The function and responsibilities of a CNC programmer has been covered. Now, let's look at what happens when the completed program and related material actually reach the machine shop.

### MACHINING A NEW PART

The most expensive part done on a CNC machine is always the *first one* of the batch. After the machine setup is completed, CNC operator is ready to test the program and machining conditions. Setup time is always non productive and testing a program is non productive as well. It takes quite a bit of time and effort, even if a good part comes out of the first run, as it should. These activities are necessary and must be done, but doing too many 'first' parts for one batch is not productive either.

Generally, there are two groups of CNC programs, each having a different effect on program proving. The first group covers all programs that have *never* been used on the CNC machine. These programs must be proved for accuracy, as well as optimized for best performance. The second group covers the *repetitive* jobs - programs that have been used at least once before and have been proved to be correct in all respects. Programs in this group have most likely been optimized for the best performance under certain given conditions. In both cases, the CNC operator must take a good care when running the first part of a batch. However, there are differences between a *new job run* versus a *repetitive job run*.

In either case, two qualities relating to the part program have to be established first:

- Setup integrity
- Program integrity

These two considerations are equally important - if only one of them is weak, the final result is not satisfactory. Always aim at the highest level in either category. Also keep in mind that the setup integrity has to be established again with each run in the future. Program integrity has to be established correctly only once.

### ◆ Setup Integrity

Machine setup is only a general description of the type of work actually done to get the CNC production going. The whole process covers setup of cutting tools, as well as part setup and many related tasks. No single check list can ever cover all points that have to be considered during a CNC machine setup. The major focus here is at the most important considerations, in form of a brief check list. Adjust the individual points according to the machines and CNC systems in your shop. Adjust the list to reflect personal working methods and/or programming style. The main purpose of this check list, or any other for that matter, is to cover as many details as possible and not to omit an important item, operation, procedure, and so on. Even a small omission may cause an accident and part damage or even a scrap due to a faulty machine tool setup.

#### *Cutting Tools Check*

■ Are the tools properly mounted in holders
■ Are the proper inserts used (radius, grade, chipbreaker, coating)
■ Are all the tools the right size
■ Are the tools placed in the proper magazine station
■ Are the offsets set correctly (set zero to unused offsets)
■ Is there an interference between individual tools
■ Is the boring bar properly oriented (milling)
■ Are all the tools sharp

#### *Part Setup Check*

■ Is the part mounted safely
■ Is the part properly oriented on the table (milling)
■ Is the projection of the part from the chuck safe (turning)
■ Is the part lined up for squareness (milling)
■ Are the clearances sufficient
■ Are all the clamps away from the cutting path
■ Is the machine at its start (home) position before you press Cycle Start
■ Does the tool change take place in a clear area

#### *Control Settings Check*

■ Is the coordinate setting registered (for G54 to G59)
■ Are all the offsets entered correctly
■ Is coolant necessary
■ What is the status of the BLOCK SKIP switch
■ Is the optional program stop M01 active (ON)
■ Is the DRY RUN off if the part is mounted
■ Do you start with a SINGLE BLOCK mode set to ON
■ Do you start with spindle speed and feedrate overrides set to LOW
■ What is the status of MANUAL ABSOLUTE switch (if applicable)
■ Has the position read-out on the screen been set from zero (origin preset)

#### *Machine Tool Check*

■ Is the slide lubrication container filled with proper type of oil (lubricant)
■ Is the coolant tank filled
■ Is the chuck and tailstock pressure set correctly (turning)
■ Has the machine been zeroed before running a job - is the read-out set to zeros
■ Is there enough pressure for the air attachments (air hose, etc.)

### ◆ Program Integrity

Any new and unproved program is a potential source of problems. In manual CNC programming, mistakes are a lot more common than in CAD/CAM programs. A good way to look at a new program is through the machine operator's eyes. Experienced CNC operators take a direct approach when running a new program - *they take no chances*. That does not mean the CNC programmer is not to be trusted - it simply reflects the fact that the machine operator is ultimately responsible for the expected quality of work and is aware of it. He or she has a sense of great responsibility. Whether the damage to the part or even scrap is caused by the program or for some other reason is a little consolation when the work is rejected.

What does the CNC operator look for in a new part program? Most machine operators would agree that the first and the most important thing is *consistency* in programming approach. For example, are all tool approach clearances the same way as always? If not - is there a reason? Is the basic programming format maintained from one program to another program and from one machine to another? A good operator scans the written program *twice* - once on the paper copy, the second time when the program

is loaded into the control system. It is surprising what can be seen on the screen that was not seen during the paper copy check. Reverse is also true. Common mistakes such as a missing minus sign or an address, a misplaced decimal point or a programmed amount extra large or extra small can usually be detected on the screen easier than on paper. If using a computer for manual programming, print out the program and check it visually. Using a double check, many costly mistakes can be prevented. There is software available to graphically check the program on a computer, using toolpath simulation. *NCPlot™* is such software and is included as a shareware on the CD in this handbook.

Consistency in programming style is very important and cannot be overstated. Consistency is an important way to gain confidence of CNC operators in program integrity.

## RUNNING THE FIRST PART

CNC machine operator usually starts a new job by studying the documentation included with the program, mainly drawing, setup sheet and tooling sheet. The next few steps describe a standard setup procedures that will vary occasionally, but they will remain the same for most jobs.

### ■ STEP 1 - Set the cutting tools

This first step uses a tooling sheet or tooling information from the part program. CNC operator sets the cutting tools into their holders and respective tool stations and registers all tool numbers into the control memory. Make sure the tools are sharp and mounted properly in the holders.

### ■ STEP 2 - Set the fixture

A fixture that holds or supports the part is mounted on the machine, squared and adjusted, if necessary, but the part itself is not mounted at this point. Setup sheet serves as the documentation, particularly for complex setups. A fixture drawing may often be required as well.

### ■ STEP 3 - Set the part

Locate the part into the fixture and make sure it is safely mounted. Check for possible interferences and obstacles in the setup. This step represents the end of most initial steps of CNC machine operation.

### ■ STEP 4 - Set the tool offsets

Depending on the type of machine, this step takes care of setting the tool geometry and wear offsets, tool length offset and cutter radius offset, if applicable. One of the most important parts of this step is the setting of work coordinate system (work offsets G54 to G59) or the tool position registers (G92 or G50) for old controls, but *not* both, even if they are available. Work offset setting is by far the best and most convenient selection of modern CNC machine tool setup.

### ■ STEP 5 - Check the program

This step is the first evaluation of the part program. The part may be removed from the fixture temporarily. Since all offsets are already set in the control, program is checked accurately, with all usual considerations. Program override switches on the control panel may be used, if required. Watch for tool motions in general and be sure to watch for tool indexes specifically. Repeat this step, if not absolutely sure with any aspect of the programmed toolpath.

### ■ STEP 6 - Reset the part

If the part was removed in the previous step, now is the time to mount it in the fixture again. Successful completion of all previous steps allows continuation with proving the first part. At this point, check the tooling once more, also check the oil and air pressure, clamps, offsets, switch settings, chucks, and any other important machine feature, just to be sure.

### ■ STEP 7 - Make a trial cut

An actual trial cut may be required in order to establish whether the programmed speeds and feeds are reasonable or not and if the various offsets are set properly. Trial cut is a temporary or an occasional cut that is designed to identify minor deviations in actual offset settings and allows their change. Make sure the trial cut leaves enough material for actual machining. Trial cut also helps to establish tool offsets to keep dimensional tolerances within limits.

### ■ STEP 8 - Adjust the setup

At this point, any necessary adjustments are finalized in order to fine tune the program before production begins. This step includes final offset adjustment (usually a wear offset). It is also a good time to adjust spindle speeds and feedrates, if necessary.

### ■ STEP 9 - Start the production batch

A full batch production can start now. Again - a quick second double check may prove to be worth the time.

The ideal way to run a new program is to run it first through the control graphic display, if available. It is fast and accurate, and offers a lot of confidence before actual machining. This test can be done with a variety of override modes in effect, for example, *Machine Lock* or *Single Block*. Do not underestimate features such as *Zero Axis Neglect* and *Dry Run*, when testing a new program.

If using graphic options of the control system, most likely, there will be two kinds of graphic representation of the tool path:

- Tool path simulation
- Tool path animation

Both features have been described in the last chapter.



The first type of graphic representation, *tool path simulation*, shows the outline of a finished part and the tool motions. Part outline is identified by a single color, the tool motions are identified by a dashed line (rapid motion) and a solid line (cutting motion). During program processing, the order of machining is shown on the display screen as either dashed lines or solid lines, depending on the motion type. The solid area of the part is not shown, neither are the tools, chuck or tailstock for lathe applications. With a color screen, the colors can be preset for each tool to further enhance the flexibility of graphics display.

The more descriptive method of verifying the toolpath prior to machining is *tool path animation*. In many respects similar to the toolpath simulation, the toolpath animation offers a few additional benefits. Part can be seen as a shaded form, rather than an outline only. The tool shape can be preset and seen on the screen display; the chuck shape and size can also be preset, as well as the outline of tailstock, fixture, etc., all in shaded form. The result is a very accurate representation of the actual setup conditions. As an additional benefit, the display is also proportional in scale. During the actual cut, material can be seen as being 'machined', right on the screen. Toolpath animation is a significant improvement over toolpath simulation.

Do not expect 100% accurate display of any toolpath. No graphic display can show every single detail and no simulation will show flying chips. What it does show is quite impressive, however. For CNC machining centers, the control with graphics can be set to one of several selectable views. More than one view can be set at the same time on the display screen, using a split screen method, also called *windows* or *viewports*. Many CNC operators run the graphic display twice, especially for milling systems - once in the XY view, the second time in the ZX or YZ view and even isometrically (XYZ view). Make sure the rapid motion display is turned on. Display can be run in a single step mode, areas that are either too small or especially critical can be enlarged (or reduced for large parts) with zooming features. Cutter radius offset, tool length offset and other functions can be turned on or off for the graphic display. Make sure the simulated conditions are as close to real conditions as possible. Also, do not forget to have all tools and offsets set *before* the program is tested. Unfortunately, these graphic options also add quite a bit to the overall cost of the control system and many companies choose not to purchase them.

Many programming instructions cannot be tested by using the graphics only. On most controls, there will be no clamps, no spindle speed or feedrates. Many other important activities cannot be seen, but what does show will make the actual cutting so much easier. Since all motions have been tested in the graphic mode of the control, all that has to be done during the actual run, is to concentrate on those details that could not be seen on the display. The tasks to be checked have been narrowed down and the part program is easier to follow.

## PROGRAM CHANGES

Even if a part program is proven, tested and the first part made and inspected, a good CNC operator looks at ways of improvement. Some improvements may be done immediately on the machine, before the whole job is completed. Some improvements may require a different setup, tooling or fixturing. Often, it would not be practical or even possible to implement those changes on the current job, but they should be applied the next time the same job is done. Some changes to the part program are result of a design modification and have nothing to do with program optimization. Others are strictly steps taken for the best productivity results. Regardless of the reason, virtually any change required by the machine shop involves the CNC programmer who has to apply any new changes to the new program.

All changes to a program should be for the better, they should *improve* the program. Often a major change will require a complete program rewrite, but more likely, a program can be modified to a reasonable extent. When a program is changed for the better, it is said to be *optimized*, it is upgraded. This method can be compared to another type of program change - *program update*.

### ◆ Program Upgrading

Upgrading a CNC program means to *strengthen* it, to *enrich* it, to make it *better* than it was before. It means to change it in a way that the cost of part production is significantly decreased. The cost reduction must be achieved with no compromise in quality of the part or machining safety.

The most common form of program upgrading (optimization) is minor changes to spindle speeds and feedrates. The process is called the *cycle time optimization*. Milling operations may require a different approach than turning operations. Jobs that are repeated frequently, as well as large size lots, should be scrutinized with even more care. Keep in mind that only *one second* saved on a cycle time will save one hour for each batch of 3600 pieces, half an hour for each 1800 pieces, and so on.

In the following check list are some major points to consider when optimizing a CNC program. The list is far from complete, but it should serve as a guide to what areas can be looked into and be explored. Some items in the list apply only to the milling operations, others only to turning. There are also some items that apply to both systems. Several of them require a special option of the control system or the machine tool to be available.

- |                                               |
|-----------------------------------------------|
| ■ Fine-tune the spindle speed and/or feedrate |
| ■ Choose the heaviest depth of cut possible   |
| ■ Choose the largest tool radius possible     |
| ■ Experiment with new cutting materials       |



■ Rearrange tool order for faster tool changes
■ Program bi-directional turret rotation
■ Let one tool do as much work as possible
■ Use M01 rather than M00 whenever possible
■ Avoid excessive dwell times
■ Eliminate 'air cutting' situations
■ Shorten rapid motions where applicable
■ Use multiaxis motion whenever safe
■ Apply fewer passes for threading
■ Look for block skip applications
■ Avoid spindle direction change
■ Shorten the tailstock travel distance
■ Do not return to machine zero after each piece
■ Program tool changes close to part
■ Reassess the setup and/or design a new one
■ Re-evaluate your knowledge and skills
■ Consider upgrading the CNC system

This check list is a typical sample only, although developed from experience. Many more items can be added to this list and many item can be modified in their description. Even programs that are to be used only once should be carefully audited. There may be an improvement that can be applied to a different job, sometimes in the future.

### ◆ Program Updating

In contrast to program upgrading (optimization), the reason for program updating has nothing to do with decreasing the part cost. In the end, the part *may* cost less, due to a change in engineering design or similar interventions, but not because of a program change. A program needs to be updated *after any change is made in the drawing that affects the CNC machining*. Even programs that have been previously upgraded may still have to be updated.

Engineering changes in part design are more common in companies that manufacture their own product line. In a job shop, the design changes are typically initiated by the customer, but have the same overall effect. The only difference is in the source and origin of such change.

A specific change that will affect the upgrade of a CNC program may be as small as a change in a single dimensional tolerance or as large as a complete part redesign. Personal experience may be somewhere between the two. An upgraded CNC program will reflect the magnitude of the change - whether it is a minor correction or a complete program rewrite.

### ◆ Documentation Change

The documentation that is associated with a particular part program is not much useful if it does not reflect all program changes done during part machining. Just like a well documented engineering drawing or other important data source, all revisions, updates, upgrades and many other changes should be recorded. Changes in mathematical calculations should be especially well documented and supplemented with formulas and sketches if possible. If there are several existing copies of the documentation, they too, should be replaced to make them current and up to date. The programmer's name, the nature of the change, the date, even the time of the day, should be used to indicate *when* such a change took place. Keeping the old version for reference (at least for a while) may also be a good idea. Sometimes one or two experiments may be necessary before deciding on the best documentation, on the final documentation suitable to particular needs.

## ALTERNATE MACHINE SELECTION

Even with the best planning, things can go wrong, at least occasionally. What happens in a machine shop when the only CNC machine is suddenly out of commission? Of course, this never happens, except when a rush job is just about to be set up on that very machine. It usually happens when it is expected the least.

Every production manager has to have an alternate plan of action. One of the most common actions is to do the job on another CNC machine. Of course, such a machine has to be available, but there is more to consider.

Usually a job is programmed for a specific machine and a CNC system. If two or more such machines have been installed in the shop, the program can be executed on any one of them. Comparably, if two or more machines and/or controls are totally incompatible, programs are not transferable and a new program must be developed. The best opportunity for compromise exists if two machines are different in size, but with the same control type. The existing program may be usable as is, or with only very minor modifications.

The major considerations for alternate machine selection involve tooling and setup. First, the cutting tools and holders, as well as fixtures, must be available. Tools must all be the same size, even if holders are different. Part position on the table, clamps locations, data holes, clearance areas, etc., must also be the same. In addition to these general considerations, specific conditions such as spindle speed, feed-rates, power rating of the machine tool and other factors must also be carefully examined. Overall accuracy and rigidity of the alternate machine is also very important.

Those machine shops where many part programs have to be portable have adapted various standards for both programming and setup operations.

## MACHINE WARM UP PROGRAM

Any precision equipment is guaranteed by its manufacturer to work accurately not only if it is handled properly, but also if it operates within a certain environment. Computers - CNC systems included - are particularly sensitive to rapid changes in temperature, humidity, dust level, external vibrations, etc. All potential hazards are clearly specified in the manufacturers' literature. Every CNC operator knows from experience that the part precision depends a great deal on the spindle temperature. Some ultra high precision machines even have an internal cooling system to keep the spindle temperature constant. In cold climates, on a cold morning in the winter, when the machine was sitting all night in an unheated shop, experienced CNC operator turns the spindle on for a few minutes, to let it warm up. At the same time, in order to make the slide lubricant freely moving along the guide ways, the operator makes a few free motions in both directions of all axes. If this process is repeated every day in cold winter months, it may be worth to automate it. A short program will do the job.

To write such a program is simple, but there are several important points to consider. First, make sure that the machine motions will always be in the area where there is no possibility of a collision. This program will be used with many jobs and modifying it every time a new job is set up is not an option. Another point to consider is the spindle speed in r/min. Avoid programming an excessively high r/min - a tool mounted in the spindle for the warm up could have a small or large diameter. To make the program to repeat itself indefinitely, use M99 function at the end. Program also function M30 for the program end, but with a block skip symbol [ / ]. When the warm up is to terminate, simply turn the block skip switch off. All machine motions will be completed and the program will end naturally.

Example O5101 is a typical warm up program for a milling system and uses imperial units. The program can be easily adapted to any other machine:

```
O5101 (WARM-UP FOR A MILL)
N1 G20
N2 G40
N3 G91 G28 Z0
N4 G28 X0 Y0
N5 S300 M03
N6 G00 X-10.0 Y-8.0
N7 Z-5.0
N8 S600
N9 G04 P2000
N10 X10.0 Z5.0
N11 Y8.0
N12 S750
N13 G01 X-5.0 Y-3.0 Z-2.5 F15.0
N14 X-2.0 Y-2.0
N15 Z-2.0 S800
N16 G04 P5000
N17 G28 Z0 M05
```

```
N18 G28 X0 Y0
/ N19 M30
N20 G04 P1000
N21 M99 P5 (REPEAT FROM BLOCK 5)
%
```

This example is simple in structure, yet well thought out. There are several intentional programming techniques contained in the sample program:

- Whole program is in incremental mode
- First motions are to the machine zero
- Z-axis motion is the first motion
- Spindle speed is increased gradually
- Dwell is used to lengthen the current action
- End tool motion is to the machine zero
- End of program M30 is 'hidden' by a block skip function
- Each program repetition starts at block N5

Several program versions can be developed, depending on the machine and the type of work expected on that machine. For example, if developing a warm up program for a CNC lathe, incorporate functions that are typical to a CNC lathe - for example, changing the gear range, moving the tailstock in and out, opening and closing the chuck jaws, doing a tool change, etc. On a horizontal machining center, include the indexing table motion; on a boring mill, the in and out spindle quill motion may also be programmed. Modify the program to suit any particular purpose, but keep in mind its aim - to warm up a machine that had been idle for a relatively long period of time in a cold temperature. Also keep in mind the safety of operations - the goal is *a generic program for a specific machine type*, a program that can be used with all jobs, without modifications.

## CNC MACHINING AND SAFETY

Machine shop safety is everybody's responsibility. Some basic safety issues have already been introduced in the first chapter of this handbook. CNC programmer has to apply safety at the programming level, CNC operator at the machine level, and so on. Many companies have established numerous safety rules and procedures that work well. Follow them and try to improve them.

Generally, the safety concerns of a machine operator are almost the same as those operators running conventional equipment. Safety starts with a clean work place and organized approach to programming, setup and machining. Many do's and don'ts can be itemized, but no list will satisfy all the safety concerns. Here is an attempt at a typical list of safety concerns in CNC shop. There are several general groups in the incomplete list. Many suggestions can be in different groups.

### ➡ Personal Safety

- Wear suitable clothing (tucked-in shirt, buttoned-up sleeves)
- Remove watches, rings, bracelets, and similar jewelry before machine operation
- Keep long hair under a net or tied up
- Protect your feet by wearing approved safety shoes
- Protect your eyes - wear approved safety glasses with protective side shields at all times
- Wear an approved safety helmet if that is the company policy
- Always protect your hands - never reach towards the part while the spindle is rotating
- In some cases, protection may also be needed for head and ears, perhaps even nose
- Never remove cutting chips by hand, with or without gloves on
- Do not use rags or gloves around moving or rotating objects
- When lifting heavy objects, ask for help, use a crane or do not lift

### ➡ Machine Environment Safety

- Make sure the floor is swept, free from oil, water, chips, and other hazards
- Check the walkways, so they are not blocked from any direction
- See whether all the material is safely stored and finished parts are in proper containers

### ➡ Machine Tool Safety

- Do not remove guards and protective devices
- Read and follow operating manuals
- Check fixtures and tools before they are used
- On the machine, make sure all the tools are tight in the holders, that the tools are sharp and selected properly for the job on hand
- Stop all machine motions when measuring or inspecting finished work
- Do not leave objects on top of machines
- Use only a suitable coolant mixture, and keep the coolant tank clean at all times
- Never use a file for breaking corners or a sand paper for surface polishing during the program execution
- Deburr sharp edges before handling a part
- Stop all machine power for maintenance
- Do not operate a faulty machine

- Do not alter design or functionality of the machines or controls
- Electrical or control maintenance should be done by authorized personnel
- Do not use a grinding machine near the CNC machine slides
- Do not use a welding equipment on CNC machine under power
- Behave responsibly - do not engage in pranks and horseplay around machinery

These are only some common sense suggestions, not a comprehensive list for CNC machining safety.

Always observe the company safety policies, as well as safety laws of a particular jurisdiction

## SHUTTING DOWN A CNC MACHINE

When the CNC machine is not used for an extended period of time, it should be shut down. Many users assume that shutting down a CNC machine means just to turn the power off. There is more than that to shutting down a machine tool with a power switch.

### ◆ Emergency Stop Switch

The purpose of emergency switch (E-stop) is to *stop all machine motions immediately*, regardless of the current operational mode. When pressed, the switch will lock in place and must be rotated manually in the opposite direction to be released. It should be used sparingly and only in real emergencies, such as when:

- An imminent situation that is unsafe to the human being is about to occur
- An imminent collision of the machine tool elements is about to occur

In certain situations, it is possible to cause damage to the machine and tooling when pressing the *Emergency Switch*. Depending on the machine design, there may be several emergency stop switches available, located at convenient places. CNC operator should always know the locations of each emergency stop switch. Emergency switch is also called the *E-stop* or *E-switch*.

### WARNING !

Although the emergency stop switch disconnects all power to the machine axes, electrical power is still supplied to the CNC machine

For a complete safety shut-down, always follow proper procedures as enacted by company policies.

When *Emergency Stop* switch is released or unlocked, the machine does not restart automatically. Machine setup conditions and other conditions have to present before the automatic start can be selected. This condition is usually achieved by pressing the *Power On* switch.

### ◆ Parking Machine Slides

Several chapters have mentioned a comment that a CNC program cannot be executed unless the machine had been zeroed first. Recall that zeroing a CNC machine while the machine slides are at - or almost at - the machine zero, is impractical and may result in an overtravel. Machine zero return needs about *one inch* minimum (or 25 mm), to be *away* from the machine zero position in each axis. This position is often easier to reach at the *end* of work than at its beginning. A practical CNC machine operator knows that to shut off the machine when all slides are at the machine zero position causes the subsequent start up to take a little more time.

To avoid any potential problems in the future, some programmers make a small program to bring the machine slides into a safe position at the end of work, before the power is turned off. Although the idea is good, the solution to one problem may cause another problem. If the machine slides are 'parked' repeatedly at the *same* position for a lengthy period of time, various dirt deposits will collect *under* the slides, possibly causing staining or even rusting in and around the 'parking' area. A better way is to let the CNC operator do the positioning of the slides manually. It does not take any more time and the slides will never be too long at any single position. All that is needed is a motion of one axis at a time, to a different position every time. Since it is done manually, there is a better chance that the machine position will be always different.

### ◆ Setting the Control System

Control panel of a CNC unit has many switches set to a certain state at the time of a shut down. Again, variations exist as to what the proper procedure is, but a good CNC operator will leave the control system in such a state that it does minimize a potentially dangerous situation, when used by the next person. Here are only some possibilities to apply before leaving the control system for a break, or a complete shut down:

- Turn down the feedrate override switch to the lowest setting
- Turn down the rapid override switch to the lowest setting
- Set mode to JOG or HANDLE
- Set the handle increment to X1 (minimum)

- Set the Single Block switch ON
- Set the Optional Block switch ON
- Set operation mode to MDI
- If available, remove the Edit key from the lock

Several other precautions could be also be used, but the ones listed are the most typical and should ensure reasonable safety precautions.

### ◆ Turning the Power Off

Procedures vary from one machine to another, so always consult the machine manual first. However, there are some procedures pretty common to all machines. General rule is to *reverse* the procedure of turning the power on. For example, if the procedure to turn the power on is

1. Main switch ON
2. Machine switch ON
3. Control switch ON

then the power off procedure will be

1. Control switch OFF
2. Machine switch OFF
3. Main switch OFF

Note that in either case, there is not one switch to do all work. This is for the safety of sensitive electronic system of the CNC unit. Also check the exact function of the emergency switch (described earlier), as it relates to the machine shut down procedure.

## EQUIPMENT MAINTENANCE

To maintain a CNC equipment is a professional discipline of its own. In general, it is better to leave any kind of maintenance to qualified technicians and engineers. CNC machine operator should only be concerned with the basic preventive maintenance, just by taking care of the machine in general. Modern control systems require very little maintenance, usually consisting of some air filter change and similar simple tasks.

Manufacturers of CNC units and machine tool builders supply reference manuals, including special ones for maintenance, with their products. These publications should be a compulsory study reading for any person involved with maintaining machine tools in working order, electrical, electronic, or mechanical. Many machine manufacturers, and even dealers, also offer training courses in maintenance and general troubleshooting.



## 52

## INTERFACING TO DEVICES

A completed CNC program, debugged and optimized for best performance, should be stored for *future use* or *reference*. Before such a program can be stored, it must be first loaded into the CNC memory, tested and optimized. There are many ways of loading a completed part program into the CNC memory. The most basic, and also the most time consuming, method is to simply key-in the program at the machine directly, using the control panel and keyboard. Without a doubt, this is also very inefficient method, prone to errors. It is true, that Fanuc controls offer a feature called *Background Edit*. This is a standard feature on most controls that allows the CNC operator to key-in (and/or edit) one program, while the control runs the machining operations for another program. In practice, however, many operators simply don't take advantage of this feature for various reasons.

In order to load a part program into the CNC memory or unload a program from the CNC memory, a hardware connection called a *data interface* is needed. An interface is usually an electronic device that is designed to communicate with the computer of the CNC unit.

Typical interfaces and storage media are:

- Tape reader and tape puncher (obsolete)
- Data cassettes (obsolete)
- Data cards (obsolete)
- Bubble cassettes (obsolete)
- Floppy disks (obsolete but still around)
- Hard (fixed) disks
- Removable devices (flash and USB storage)
- ROM (read-only-memory) devices
- ... and others

Many of these devices are proprietary, many require not only special cabling, but also software drivers that can run these devices. The focus of this chapter will be on the connections that can be easily assembled and those that use standard configurations. There is one industrial standard many of these devices have in common - a standard called an *RS-232C* interface. Well - almost a standard. There is a number of variations that follow the standard in principle, but deviate from it to some extent. This handbook is not an in-depth discussion of CNC communications, it only does an overview of the standard as a guideline, not as a solution to all CNC communications.

## RS-232C INTERFACE

Data transfer between two electronic devices (computers and controls) requires a number of settings that use the same rules for each device. Since each device may be manufactured by a different company, there must be a certain independent standard that all manufacturers adhere to. The RS-232C is such a standard - the letters *RS* stand for '*Recommended Standard*'. Almost every CNC system, a computer, a tape puncher and tape reader, has a connector (known as a *port*) that is marked RS-232C or similar. This port exists in two forms, one with a 25 *pin* configuration, the other with a 25 *socket* configuration. One with the pins is known as the DB-25P connector, one with the socket as DB-25S connector (male/female respectively). *Figure 52-1* illustrates the typical layout.

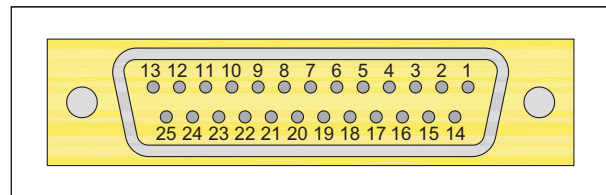


Figure 52-1

Typical 25-pin RS-232C port - DB type

RS-232C port on the CNC unit is usually a standard feature and uses the DB-25S type (the letter *S* means it is a socket type). An external computer, usually a desktop computer or a laptop, together with a suitable cable and communications software is also needed to transfer CNC programs. External devices use mainly the DB-25P type connector (the letter *P* means it is a pin type). The price tag for such a setup (hardware and software) is well below the cost of any suitable alternative, with the exception of flash cards and USB. It is also a very convenient method. CNC program is sent to the system memory and is stored there as long as needed to run the job. CNC operator usually makes some changes and when the job is completed, all changes that are to remain permanent are sent back to the desktop computer or a laptop computer and stored on hard disk. This method works well with a single CNC machine as well as several machines.

Although terms such as *Transmit* (or *Send*) and *Receive* are more common in software, some CNC systems use the terms *Punch* (which is equivalent to *Send*) and *Read* (which is equivalent to *Receive*). These terms go back to the days of punched tape.



To make this very popular method of communications work, only a suitable cable has to be installed between the computer port and the CNC system port. Loading and configuring a communications software that runs the complete operation also has to be done first. In addition, both devices must be set in a way they can ‘talk’ to each other.

Later in this chapter will be a few notes relating to the basic principles of using a personal computer as an interface with the CNC system. First, a short look at the original interface device - the venerable *punched tape* - as a media used for many years but rarely used anymore.

## PUNCHED TAPE

Since the beginning of numerical control technology, a punched tape has been the primary media for sending part program instructions to the control system. In the late 1980’s, punched tape has lost almost all its splendor and has been replaced by desktop and laptop computers loaded with inexpensive software.

Punched tape is fragile and often bulky. It can get dirty easily, but it had been very popular. It is economical to use and is still available (although the price per roll could be high). The majority of new CNC machines do not have tape reader any more. Used older machines may still have it. Many of these old controls accept tape only as an input device, not to run a job from the tape. Tape only loads the CNC memory. Changes to the program can be done through CNC and a corrected tape may be punched out later.

### ◆ Tape Reader and Puncher

One of the original facilities for data transfer was a tape reader built into now old NC and CNC machines. Its function on a CNC machine is quite different than on the early non-CNC equipment. Rather than using tape reader as the source for running the program, tape reader on a CNC machine is used to *load* the program stored on a paper tape *into* the system memory. Once loaded, the program is executed *from* the memory, in *Memory mode* setting, and the paper tape is no longer needed. There is one great weakness with this method. Working on a CNC machine often means some inevitable changes to the program *after* it had been loaded. Since these changes cannot be readily reflected on the tape, there can be confusion at a later date, possibly when the job is repeated. This is an organizational problem and can be resolved relatively easily.

One option is to make all necessary changes and corrections on the CNC unit, then punch out a new tape, using the RS-232C port. The difficulty of this approach is that while a built-in tape reader was common, a built-in tape puncher was virtually non-existent. A significant amount of money had to be spent on an external portable tape puncher, that usually incorporates the tape reader anyway, and causing duplication.

Modern machine shops do not use tapes, punchers and readers of any kind anymore. These once powerful tools have been replaced by inexpensive microcomputer technology and inexpensive communication software.

Even if the punched tape technology as such is obsolete by any modern standards, it may justify a short sideline for those who still use it and also for those who are interested in the ‘historical’ aspects of numerical control.

### Tape Media

Punched tape is the oldest media for storing programs. The tape is made of good quality, enforced paper. Punched tape is 1.0000 inch wide (25.4 mm) and about 900 feet long (about 274 meters) on a single roll, manufactured to exact standards. The most useful descriptions and dimensions are illustrated in *Figure 52-2*.

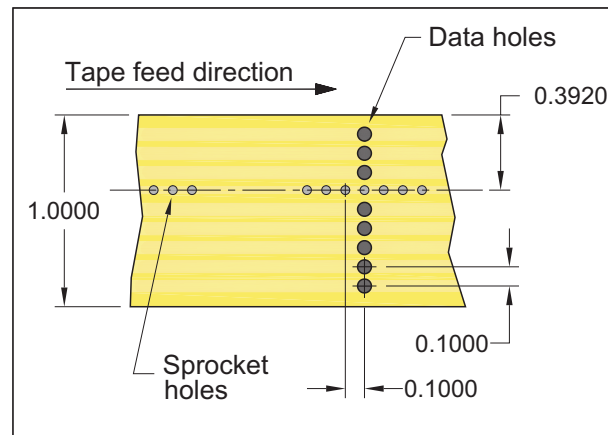


Figure 52-2

Punched tape detail - basic dimensional standards

Paper tape is generally available in black color, as its 100% opacity was required by most tape readers. Tape materials other than paper were also used, such as Mylar®, which is a paper tape sandwiched between two layers of strong plastic. Plastic makes the tape stronger, which is an important factor when such tape is to be used constantly, for example, in a reel-to-reel operation. Aluminum - or metal - tapes were also available. Both types of tape, Mylar® and metal, were and are relatively expensive and are used only for critical jobs and long programs. This was typical to aircraft, defense, nuclear and mold industries. Many companies also used the durable tape material for storing programs that have been proven on the machine.

Punched tape is generally available in a roll form, although a folded strips version may still be available. It has two main purposes:

- To store program data for use at a later date
- To serve as a media for transferring program data into the control system via a tape reader

### Tape Coding

Punched tape consists of a series of holes, laid across the tape width, where each row represents one character of the program - *a character is the smallest unit of input*. Punched characters are transferred through the tape reader to the control system in a form of electric signals. Each character can be composed of up to eight signals, represented by a unique combination of holes punched across the width of the tape in 0.1000 (2.54 mm) increments. A character can be any capital letter of the English alphabet, any digit, plus some symbols, such as a decimal point, minus sign, slash, and others.

### ISO and EIA Tape Format

When preparing the tape, try to understand two methods of standard tape coding - one, which employs the *even* number of punched holes, and the other, that uses the *odd* number of punched holes. Technical terms for these two systems are *Even Parity*, when a character is composed of 2, 4, 6 or 8 punched holes, and *Odd Parity*, when the character is composed of 1, 3, 5 or 7 punched holes. There is also coding that is a mixture of the two, called *No Parity*, that has no application for machine tools. For illustration of a partial tape coding, see Figure 52-3.

ISO CODE									EIA CODE								
8	7	6	5	4	3	2	1		8	7	6	5	4	3	2	1	
●	●	●	●	●	●	●	●	7					●	●	●	●	7
●	●	●	●	●	●	●	●	8					●	●	●	●	8
		●	●	●	●	●	●	9				●	●	●	●	●	9
	●				●		●	A		●	●			●		●	A
	●				●		●	B		●	●			●		●	B
●	●				●		●	C		●	●			●		●	C
●	●				●		●	D		●	●			●		●	D
●	●				●		●	E		●	●			●		●	E
●	●				●		●	F		●	●			●		●	F
●	●				●		●	G		●	●			●		●	G
●	●				●		●	H		●	●			●		●	H
●	●				●		●	I		●	●			●		●	I

Figure 52-3

Tape coding standards

Even parity (ISO) on the left, odd parity (EIA) on the right

Even parity of the punched tape corresponds to the *International Standards Organization* coding, called ISO in a common abbreviation, formerly known as the ASCII code (*American Standard Code for Information Interchange*). Odd parity is the standard of the *Electronic Industries Association*, EIA in short, that is slowly on the decline, mostly due to the limited number of available characters.

Even parity format ISO is also known as the standard DIN 66024 (ISO) or RS-358 (EIA) or ISO code R-840. Odd EIA format is the standard number RS-244-A.

Most modern numerical controls, providing they have a tape reader interface, will accept either tape coding automatically, based on the parity of the *first end-of-block* character punched on the tape.

### Parity Check

While punching a tape, make sure that the process is consistent for the whole length of program tape. Mixing ISO and EIA codes on any one tape will result in a rejection by the control tape reader. Such a fault is normally called a *parity error*. System check for correct parity is automatically performed by the control unit, when punched tape is loaded into the CNC memory or processed in a reel-to-reel operation. Control will check for the occurrence of *odd* characters in ISO tape and of *even* characters in EIA tape. Main purpose of such a check is to detect malfunction of punching or reading equipment, which can be very costly if it causes a character of one coding to become a character of the other coding.

### Control In and Out

On ISO tapes (even format), a pair of punched codes representing *parenthesis* identifies a section that is *not* to be processed by the control system. Whatever information is contained between the parenthesis will be ignored by the control. This is a section that may include program comments; they will appear in the hard copy printout, but will not be processed when the tape is read.

### Blank Tape

Blank tape is the tape purchased and is completely free of any holes. Often, it may be overprinted with directional arrows, to indicate the feeding direction or the top of tape. New blank tape was sometimes called a *virgin tape*.

Blank tape can also be one that has only sprocket holes punched but no holes representing individual program characters. Sprocket holes are small size holes, located between the third and the fourth channel of the tape and run along the tape length. Blank section of a tape is used at the beginning (leader) and at the end (trailer) of tape, to make it easier to handle. The blank section also provides protection to the coded section when the tape is stored rolled up.

### Significant Section

The section of punched tape that contains program data is often called the *significant data section*. Another term used in conjunction with significant data section is a *label skip function*. It means that everything up to the first EOB (end-of-block) character, that is punched on the tape will be ignored. That also means the significant data section of a tape is the section following the first EOB character.

The first occurrence of a carriage return (caused by the *Enter* key on a computer keyboard) is the first occurrence of the end-of-block character. This signal identifies beginning of the *significant data section* - section where the actual program is stored. Significant data section is terminated by a stop code, identified usually by a percent sign, acting as the *end-of-file* character. When the stop code is read by the reader, tape reading is completed. That is why no information is ever placed past the percent sign.

### ◆ Leader and Trailer

The *blank* sections of a punched tape are used as a leader and a trailer. Blank section preceding the coded program data (significant data section) is called the *leader*; section following the data is called the *trailer*. Suitable length of the leader or trailer is usually about 10 inches (250 mm) for memory operation (without reels), but should be about 60 inches (1500 mm) when the tape is on reels. For small diameter reels, the leader and trailer section can be shorter than for large reels. Sometimes the length of leader section must be extended to allow space for tape identification. Stickers or bright pencils can be used to supply information about the tape in its leader section.

### ◆ Tape Identification

Each punched tape should be identified as to its contents. Hand written data, adhesive labels or readable characters can be used within the leader section of the punched tape. Adhesive labels may not be a good choice because of their tendency to peel and fall off. Hand written notes may present difficulty when writing on black background. The identification usually contains program or tape number, drawing number and the part name - other information may also be included.

So called readable characters - *Figure 52-4* - seem to be the best solution, since they can be generated on the majority of tape preparation equipment.

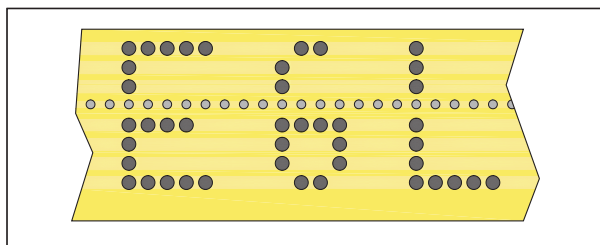


Figure 52-4

Example of readable characters on a punched tape

These special characters are actual punched holes representing real characters, namely letters, digits and symbols, rather than tape codes. An end-of-block character or the stop code may not be used in the readable section, if that section will go through the tape reader.

### ◆ Non-printable Characters

Most program characters stored on a punched tape will print normally. They are called the *printable* characters and include all capitals A to Z, numerals 0 to 9, and most symbols. Although alpha-numerical characters are printable, the following symbols cannot be printed:

- Stop code in EIA format
- Delete character
- Carriage return (or Enter key)
- Line feed
- Tab codes

One character appears on the display screen as a semicolon (;). This is a symbol for the end-of-block character and is never written. It is a control system *representation* of the carriage return in the part program.

### ◆ Storage and Handling

Paper tape is punched in a tape puncher. Punchers come with only the basic features, some have advanced features such as keyboard, printer, tape reader, setting switches, Input/Output ports, etc. Additional equipment, such as a tape winder, splicer, digital tape viewer, etc., is also available.

Storage of tapes requires a fair amount of space which increases with more tapes. Tapes are normally stored in plastic boxes, small enough to fit in specially designed metal cabinets with dividers. Tapes can be transferred into computer files to save space and expensive cabinets.

If still using paper tapes, handle them carefully by the edges only. Insist on the same treatment by machine operator and others. Take special care for paper tapes, particularly when they are manipulated by winding or unwinding. In order to prevent curling, the tape should never be wound into a small tight roll, which is very tempting for saving storage space. Heat and direct sunlight are also enemies of the tape, as is water. A reasonable amount of moisture keeps the tape from becoming too dry.

Tapes can be damaged if placed into the tape reader incorrectly. Long tapes require more care than short tapes. Grease and dust are the worst enemies of paper tapes and should be guarded against. Any tape that is to be used many times over, should be duplicated or even triplicated.

## DISTRIBUTED NUMERICAL CONTROL

The Input/Output (I/O) port RS-232C on a CNC machine is used to send and receive data. The external sources are usually a hard disk or a paper tape. In many shops, programs are transferred through the means of DNC, which means *Distributed Numerical Control*. The control has features available to make data transfer possible.

To communicate between one CNC machine and one computer using the RS-232C port, all equipment required is a cable between the two devices and a software. To communicate with two or more machines, using the same single RS-232C port, each machine must be connected to a split box with a cable. Split box is available with two or more outlets, selectable by a switch. This is the simplest form of DNC. It requires well organized procedures to make it work efficiently. DNC is not a part of the control unit and is not covered here. Commercial DNC packages are available at various levels of sophistication and cost.

Some DNC software also allows a useful feature called 'drip-feeding', which is a method used when the program is too large to fit into the CNC memory.

## TERMINOLOGY OF COMMUNICATIONS

Communications have their own terminology. There are many terms, but five terms are commonly used in CNC:

- Baud Rate
- Parity
- Data Bits
- Start Bit
- Stop Bit

### ◆ Baud Rate

*Baud rate* is the data transmission speed. It is measured as the amount of data bits per second, written as *bps*. Baud rates are only available in fixed amounts. Typical rates for older Fanuc controls are 50, 100, 110, 200, 300, 600, 1200, 2400, 4800 and 9600 bps. Modern controls can have the baud rate set to 2400, 4800, 9600, 19200, 38400, 57600 and 76800 bps. In terms of time, the higher the rate, the faster the transmission. Single data bit transfer rate will be the result of one divided by the baud rate:

$$S_b = \frac{1}{B}$$

where ...

- $S_b$  = Time required to transfer a single bit in seconds
- $B$  = Baud rate in seconds

A single bit transferred at 300 bps will take 0.00333 of a second, but a single bit transferred at 2400 bps will take only 0.00042 of a second. In practice, it takes about 10 bits to transfer one character (see *Stop Bits* section below), so at 2400 bps setting, the transmission will be at a rate of about 240 cps (characters per second). 4800 bps is a good setting once everything is working well. Higher settings are necessary for 'drip-feed' methods.

### ◆ Parity

*Parity* is a method of checking that all transmitted data were sent correctly. Just imagine what would happen if some characters or digits of a CNC program were not transferred correctly or not transferred at all. Parity can be *even*, *odd*, or *none*, and *even* is the most common selection for CNC communications. This is similar to punched tape parity.

### ◆ Data Bits

A *bit* is an acronym for *Binary digit*, and is the smallest unit that can store information in a computer. Each binary digit can have a value of either one (1) or zero (0). One and zero represent the ON and OFF status respectively, so a *bit* is something like a toggle switch that can be turned on and off as needed. In the computer, every letter, digit, and symbol used in a CNC program is represented by a series of bits, eight bits to be precise, that create a unit called a *byte*.

### ◆ Start and Stop Bits

To prevent loss of data during communication, each byte is preceded by a special bit called the *start bit*, which is low in voltage level signal. This signal is sent to the data receiving device and informs it that a byte of data is coming next.

A bit similar to the start bit, but at the *end* of the byte, has exactly the opposite meaning. It sends a signal to the receiving device that the byte has ended or stopped being transmitted. This bit at the end of a byte is called the *stop bit*. Because the start and stop bits go together, they are often teamed up together as the stop bits and set the devices to *two stop bits*.

Many terms exist in communications. With growing interest, this is a very rich field to study.

## DATA SETTING

Data used for communications must be set properly before data transfer can begin. The setting at one end (computer or the CNC system) must match the setting at the other end. For baud rate, consult the machine manual - a good start is at 2400 bps. Newer models have a higher default. Typical software setting is done through the configuration at the computer end and through the CNC system parameters at the CNC end. Settings at both ends must match. Typical Fanuc settings are:

- 4800 bps baud rate
- Even parity
- 7 data bits (seven data bits)
- 2 stop bits (two stop bits)

Proper connection depends mainly on the configuration of connecting data cables.



## CONNECTING CABLES

The most common cable for communication between a CNC machine and a computer is a shielded and grounded cable, containing several small wires (at least eight), each one enclosed in a colored plastic sleeve. Main purpose of making a communication cable is to connect the CNC port (usually 25 sockets) with the computer port (usually 25 pins), using a properly configured cable. Always use a cable of high quality. Shielded cables can reach farther distances and are generally better choice to withstand interferences during data transmission. Wires are identified by their gauge value, for example a 22-gauge or a 24-gauge wire is a good choice for communications.

The 25-pin port has each pin or socket numbered (see the first page of this chapter) and individual wires of the cable have to be connected to proper numbers at each end. It is quite common to ‘cross’ the wires between each end. Typical crossing would be between the pin number 2 and the socket number 3, and a pin 3 and socket 2. Some numbered positions have to be connected *at the same end* of the cable. This is called ‘jumping’.

### ◆ Null Modem

A very common cable wiring that is used in general communications is called a *null modem*. Connection of the two ends follows a certain standard, shown in *Figure 52-5*. Each number represents the pin or socket on the DB-25 connector. Note the jumps between connections 6 and 8 at both ends. *Figure 52-6* shows the same null modem configuration in a graphic way. This is a very popular method showing cable configurations.

PIN DB-25P	SOCKET DB-25S
1	1
2	3
3	2
4	5
5	4
7	7
6 and 8	20
20	6 and 8

Figure 52-5  
Null modem pin connections

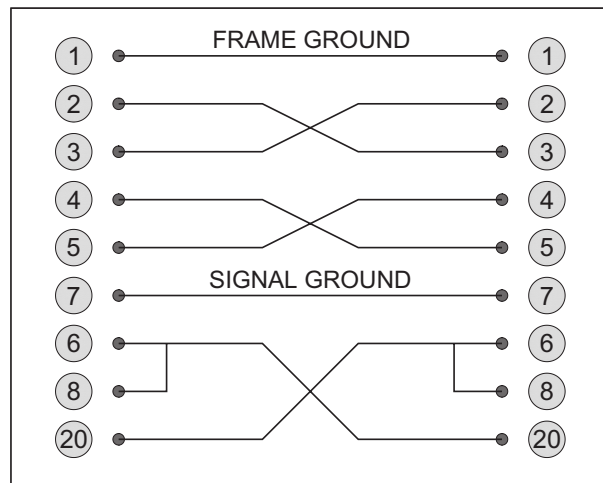


Figure 52-6  
Graphic representation of null modem connections

### ◆ Cabling for Fanuc and PC

As the most common cable communication will be between a Fanuc control and a desktop computer or a laptop, *Figure 52-7* illustrates a typical cable configuration. Note the similarity to the null modem configuration.

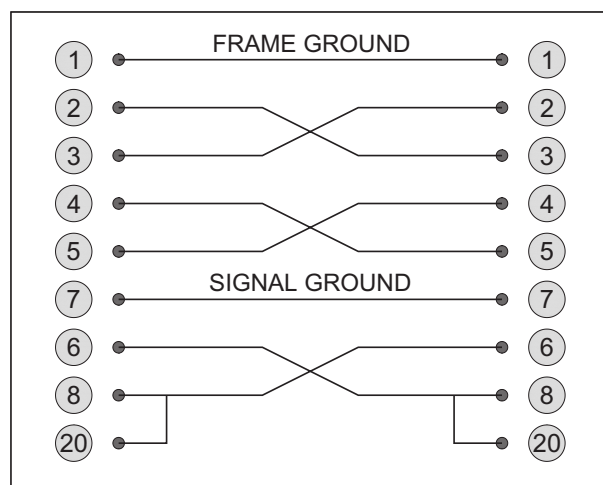


Figure 52-7  
Typical cable configuration for Fanuc controls

Regardless of what cable configuration will be used, a good communication software that will run the whole operation is also needed. Some companies use a software specially designed for CNC work, others purchase very inexpensive communications software, sold by the majority of computer stores.



## 53

**MATH IN CNC PROGRAMMING**

*Math in programming* - the single word 'math' often appears to be so powerful that it strikes a weak chord in many programmers. It is surprising how many new programmers, manual programmers in particular, are afraid of the often numerous calculations associated with CNC programming. This fear is really not substantiated. Let's look very briefly at what kind of mathematical knowledge is really necessary to handle typical programming calculations for manual program preparation.

First, the basic arithmetic functions - *addition, subtraction, multiplication* and *division* - are at the core of any mathematical activity. Going a bit further, the knowledge of common algebraic functions is definitely useful, mainly *square roots* and *powers of a number*.

Second, since CNC programming is based on the relationship of points within a system of rectangular or polar coordinates, a good knowledge of *basic geometry* is also imperative. The scope of this knowledge should cover understanding many principles of *angles*, the concept of *degrees* and their subsets, *tapers*, *polygons*, properties of an arc and a circle, the *pi* constant ( $\pi$ ), and other associated topics. Knowledge of *planes* and *axial orientations* is important in many cases as well.

Without a doubt, the most important part of geometry, one that absolutely *must* be mastered, is the solution of *right angle triangles*, using *trigonometric functions*. Very seldom there will be a problem or calculation that will require a solution using oblique triangles, although these problems may arise.

Knowledge of trigonometry is essential to any serious CNC programming

Most difficulties in solving trigonometric problems are not as much in the ability to use a specific formula and solve the triangle - but in the *inability to see* the triangle to be solved in the first place. Often, programming involves a drawing that is very complex in terms of part geometrical definitions. Such a drawing will have so many elements, that overlooking the obvious is possible, even likely.

Any specific knowledge of analytic and spacial geometry is not really required for a 2 and 2-1/2 axis work, but it is essential for a work in all three axes, particularly for complex surfaces, 3D tool path and multi surface machining or surface manipulation. However, this kind of programming is not done without a computer and CAD/CAM software.

There are several specific mathematical subjects to learn and to know in depth. All of them have been selected only for their importance in CNC programming and are described here in the necessary detail.

**BASIC ELEMENTS****◆ Arithmetic and Algebra**

The subject of *arithmetic* deals with handling numbers involving the four basic operations:

- Addition
- Subtraction
- Multiplication
- Division

*Algebra* is an extension of arithmetic and deals with handling numbers in terms of equations and formulas. Typical usage will involve:

- Square roots
- Powers of a number
- Trigonometric functions
- Solving formulas and equations
- Variable data

In algebra, typical work involves several known values and one or two unknown values. Using various formulas and equations, unknown values can be solved (calculated) to achieve the desired result.

**◆ Order of Calculations**

In the field of mathematics, there is a precisely defined order in which the calculations are performed. Every electronic calculator is based on these centuries old rules. In a combination of various algebraic operations, the order of calculations will follow these rules:

- Multiplications and divisions are always calculated first
- Additions and subtractions follow, order is not important
- Any roots, powers to a number, and operations within parentheses are always calculated before multiplications and divisions.

The following calculation will have the same result with or without parentheses:

$$3 + (8 \times 2) = 19$$

$$3 + 8 \times 2 = 19$$

The multiplication is always performed first, regardless of whether it is enclosed in parentheses or not. If addition must be done first, it must be enclosed within parentheses:

$$(3 + 8) \times 2 = 11 \times 2 = 22$$

These two examples show that even an innocently looking small omission may have significant consequences.

## GEOMETRY

For all practical purposes in manual CNC programming, there are only three entities in the engineering drawing:

- Points
- Lines
- Circles and Arcs

*Points* have no parts and are represented by the XY coordinates in a 2D plane or by XYZ coordinates in 3D space. Points are also created by an intersection of two lines, two circles or arcs, and a line and a circle or arc.

*Point* is also created by a line tangent to a circle or an arc, a circle or an arc tangent to another circle or an arc.

*Lines* are straight connections between two points creating the shortest distance between the points.

*Circles* and *Arcs* are curved elements that have at least a center and a radius.

Other elements such as *splines* and *surfaces* are too complex for manual programming, although they are also based on the same fundamental elements.

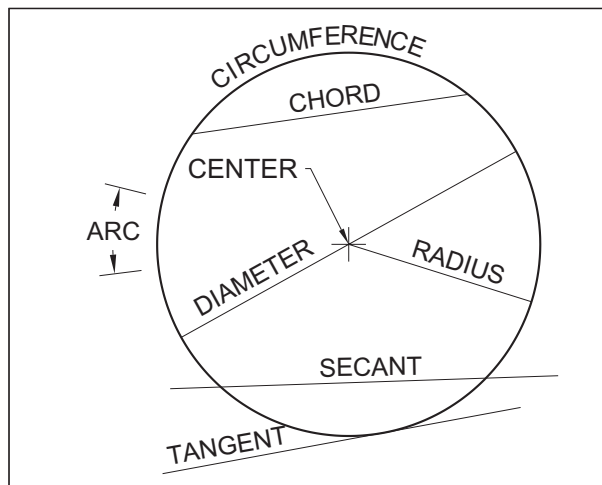


Figure 53-1

Basic elements of a circle

### ◆ Circle

*Circle* is mathematical curve, where every point on the curve has the same distance from a fixed point. This fixed point is called a *center*.

Several terms are directly related to a circle - *Figure 53-1*:

- **CENTER** - is a point from which a circle or an arc is drawn with a given radius
- **RADIUS** (radii or radiuses in plural) - is a line from the center to any point on the circumference of the circle
- **DIAMETER** - is a line through the center between two points on the circumference of the circle
- **CHORD** - is a straight line joining any two points on the circumference of the circle
- **ARC** - is any part of the circle between two points on the circumference of the circle
- **CIRCUMFERENCE** - is the length of the circle (length of the curve that bounds a circle)
- **TANGENT** - is a point where a line, an arc or another circle touches the circumference of the circle but does not cross it - this point is known as the point of tangency
- **SECANT** - is a straight line that passes through a circle and divides it into two sections

Two area sections of a circle have their own names. They are called the *sector* and the *segment* of a circle, and are shown in *Figure 53-2*:

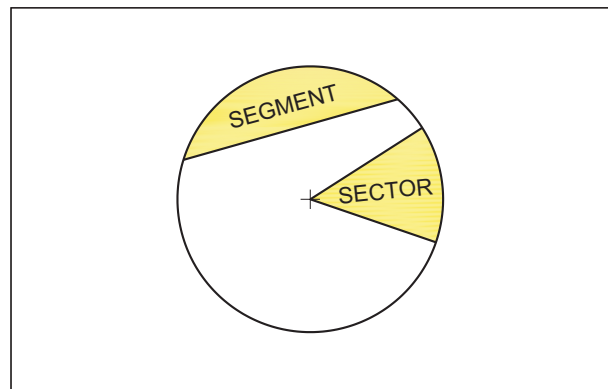


Figure 53-2

Segment and sector of a circle

- **SECTOR** - is an area within a circle formed by two radii and the arc they intercept
- **SEGMENT** - is an area within a circle formed by the chord and its arc

Neither the sector nor the segment of a circle play any significant role in CNC programming.

### ◆ PI Constant

$\pi$  is a Greek letter used in mathematics to represent the ratio of the circle circumference to the circle diameter. Its symbol is  $\pi$ , it is pronounced 'pie', and has the value of 3.141592654..., and regardless of how many decimal places will be used, it will always represent only an approximate value. For programming purposes, use the value returned by a calculator or computer, usually with six to nine decimal places. In both cases, the internal value is a lot more accurate than the displayed value. In many cases, the rounded value of 3.14 is sufficient for most results.

### ◆ Circumference of a Circle

The length of a circle - or its *circumference* - is seldom needed for programming and is included here only to enrich the general theory. It can be calculated from the following formula using the  $\pi$  constant:

$$C = 2 \times \pi \times r$$

or

$$C = \pi \times D$$

where ...

- C = Circle circumference
- $\pi$  = Constant 3.141592654...
- r = Circle radius
- D = Circle diameter

### ◆ Length of Arc

The length of an arc is also a rare requirement and can be calculated from the following formula:

$$C = \frac{2 \times \pi \times r \times A}{360}$$

where ...

- C = Circle circumference
- $\pi$  = Constant 3.141592654...
- r = Circle radius
- A = Arc angle

There are two other very important calculations relating to a circle. They are used in programming very often and should be understood well. One is based on the *chord* of a circle, the other on the *tangency* of a circle. As both calculations require the knowledge of trigonometry, they will be described later in the chapter (see page 508).

### ◆ Quadrants

Quadrant - is the part of a circle formed by the system of rectangular coordinates (described on page 15), where the axes pass through the center of the circle. There are four equal quadrants in a circle, identified by Roman numerals I, II, III and IV, starting at the upper right quadrant along the counterclockwise direction - Figure 53-3.

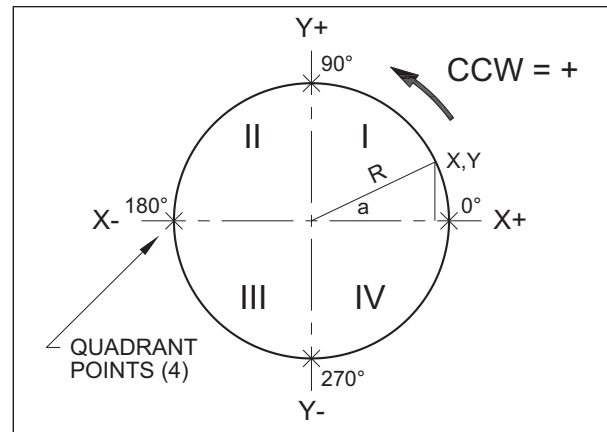


Figure 53-3

Quadrants of a circle and the mathematical definition of angular direction

Each quadrant is exactly 90°, crossing at circle quadrant points. Therefore, a circle has the sum of all four angles equal to 360°. Angles are always counted counterclockwise as positive, starting from zero degrees (0°).

Quadrant points (also known as *cardinal points*) are often compared to a hand direction on the face of an analogue clock or as a direction of a compass pointer. 0° is arbitrarily located at the equivalent position of *three o'clock* or *East* direction, 90° at *twelve o'clock* or *North* direction, 180° at *nine o'clock* or *West* direction, and 270° at *six o'clock* or *South* direction - Figure 53-4.

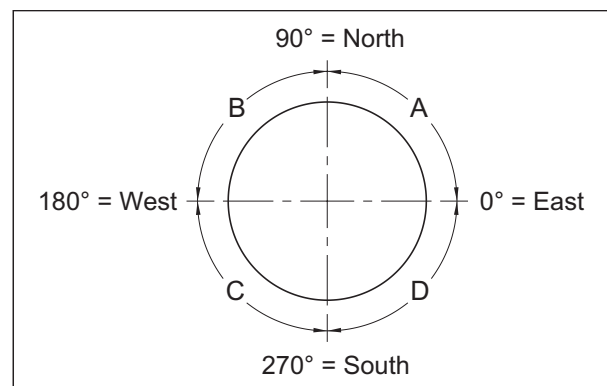


Figure 53-4

Angles and quadrants - 0° is East direction or 3 o'clock direction on the face of a standard analogue clock

## POLYGONS

Polygon is a common geometric element defined by a number of straight line segments that are joined at the end points. These line segments are the *sides* or *edges* of the polygon - Figure 53-5.

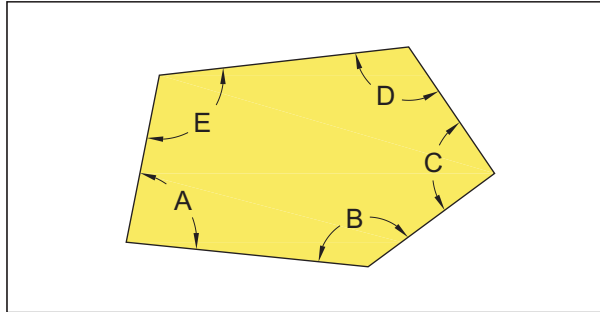


Figure 53-5  
Sum of angles in a polygon

The sum of all angles in a polygon can be calculated from the following formula:

$$S = (N - 2) \times 180$$

where ...

- S = Sum of the angles
- N = Number of sides in the polygon

For example, a five sided polygon shown in the illustration has the total sum of angles:

$$S = (5 - 2) \times 180$$

$$S = 540^\circ$$

There are several different polygons used in geometry, but only *one special kind* is of interest to CNC programming. This polygon is called a *regular polygon*, all others are irregular polygons. Regular polygon is a polygon where all sides are of equal length, called *equilateral sides*, and where all angles are also equal, called *equilateral angles* - Figure 53-6.

A single angle in a regular polygon can be calculated from this formula:

$$A = \frac{(N - 2) \times 180}{N}$$

where ...

- A = Single angle in degrees
- N = Number of sides in the polygon

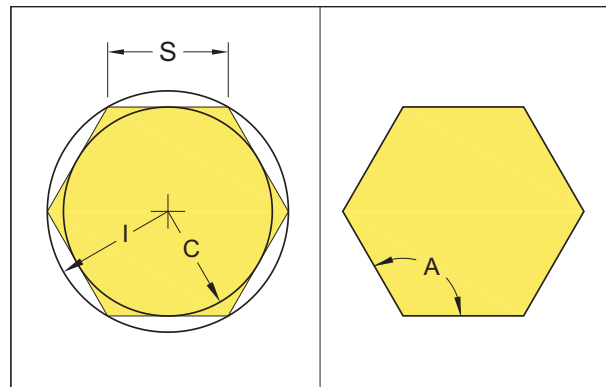


Figure 53-6  
Regular polygon  
Inscribed and circumscribed circles and a single angle

For example, a six sided polygon (commonly known as the *hexagon*) has a single angle of  $120^\circ$ :

$$A = (6 - 2) \times 180 / 6$$

$$A = 120^\circ$$

A regular polygon is quite often defined by the number of its sides and its center, located within an *inscribed* or *circumscribed* circle. Figure 53-6 above illustrates the concept of inscribed and circumscribed polygon, as it applies to a hexagon.

Although regular polygons may have virtually unlimited number of sides, some polygons are so common that they have a special descriptive mathematical name:

Number of sides	Common name
3	Triangle
4	Square (Quadrilateral)
5	Pentagon
6	Hexagon
7	Heptagon
8	Octagon
9	Nonagon
10	Decagon
12	Dodecagon
<i>n</i>	<i>n</i> -gon

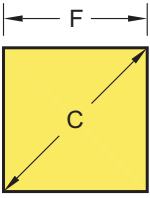
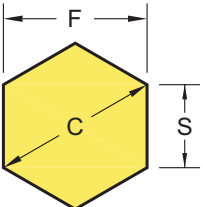
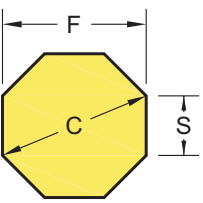
	$C = F \times \sqrt{2}$
	$C = F / \cos 30^\circ$ $C = 2 \times S$ $F = C \times \cos 30^\circ$ $F = S / \tan 30^\circ$ $S = F \times \tan 30^\circ$ $S = C / 2$
	$C = F / \cos 22.5^\circ$ $C = S / \sin 22.5^\circ$ $F = C \times \cos 22.5^\circ$ $F = S / \tan 22.5^\circ$ $S = F \times \tan 22.5^\circ$ $S = C \times \sin 22.5^\circ$

Figure 53-7

The most common regular polygons - square, hexagon and octagon

In Figure 53-7 are three most common regular polygons - a square, a hexagon and an octagon. Calculations of the distance between opposite corners  $C$ , the distance between flats  $F$  and the length of each side  $S$  are given. Note that a hexagon may have two different orientations (two horizontal sides or two vertical sides), which have no effect on the calculations. Hexagon orientation can be compared in Figure 53-6 with the hexagon orientation in Figure 53-7.

## TAPERS

All taper calculations are virtually confined to the lathe machining exclusively. Infrequently, tapers also appear in milling applications. All tapers in this section relate to the lathe applications (so called *circular tapers*), but can be modified to milling. The main purpose of tapers is to provide a match between assembled parts. By definition,

Taper is a uniformly created conical surface on a pin or in a hole

Many tapers are industry standards and are used for small tool holders (shanks), such as a Morse taper or a Brown and Sharpe taper. In addition, there standard tapered pins, machine spindle tapers, tool holder tapers, etc. In most cases, the taper is normally defined by the large end diameter, its length and a special note describing the taper.

The description varies between imperial and metric standards. For example, AMER NATL STD TAPER NO. 2 (American National Standard Taper number 2) is a specific taper description. Another common description in imperial units is a *taper per foot*. Metric system is much simpler, using only a ratio. Ratio is used in imperial drawings as well. In both measuring systems, there is one common rule:

Taper on diameter is the difference in diameter per unit of length

### ◆ Taper Definition

Most drawings define a taper in two common ways:

- One diameter and length with taper description or note
- Diameter at both ends and the length with taper description or note

If a single diameter is defined, it is often the larger one.

The taper description is a note with an arrow pointing towards the taper. In imperial measurements, the note may identify a standard taper or a taper per foot (TPF). In metric, the taper is always a ratio. Figures 53-8 and 53-9 show the differences between the two units, which is confined to the taper identification.

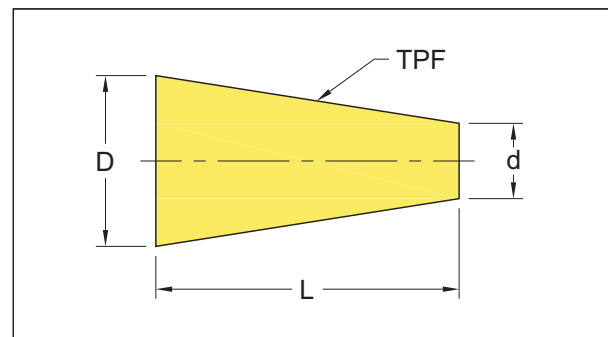


Figure 53-8

Circular taper - Imperial description

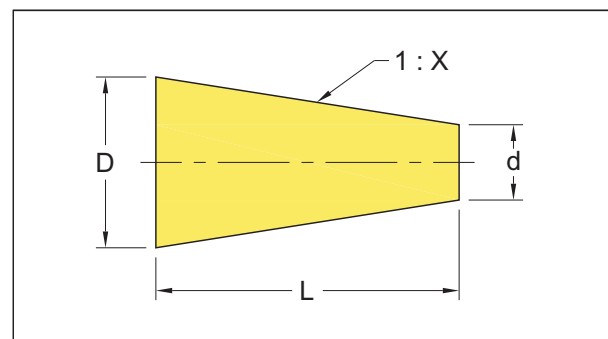


Figure 53-9

Circular taper - Metric description



In the *Figure 53-8*, showing imperial units method, the letters have the following meaning:

☞ Dimensions ...

D	=	Diameter at the large end in inches
d	=	Diameter at the small end in inches
L	=	Length of taper in inches
TPF	=	Taper per foot in inches
X	=	Ratio value 1 : X (not shown)

In the *Figure 53-9*, showing metric units method, the letters have the following meaning:

☞ Dimensions ...

D	=	Diameter at the large end in millimeters
d	=	Diameter at the small end in millimeters
L	=	Length of taper in millimeters
X	=	Ratio value 1 : X

All formulas in this section use these designations.

### ◆ Taper Per Foot

Taper per foot is defined as:

Taper per foot is the difference on diameter in inches over one foot of length

For example, a taper defined as 3.000 inches per foot, abbreviated as 3.0 TPF or 3 TPF in the drawing, is a taper that will change the conical diameter by 3 inches for every 1 foot of length.

### ◆ Taper Ratio

Metric definition of a taper is similar:

Taper is defined as the ratio of difference between large diameter and small diameter over a given length of the cone

The metric specification of a taper is the ratio X:

$$\frac{1}{X} = \frac{D - d}{L}$$

The ratio 1 : X means that over the length of X mm, the diameter of the cone will change (either as an increase or as a decrease) by 1 mm.

For example, a taper specified as 1 : 5 will increase 1 mm on diameter, every 5 mm of length.

For milling, the taper is defined as the difference in width over a given length (per side).

### ◆ Taper Calculations - Imperial Units

Missing drawing dimensions in *Figure 53-8* may be calculated from the given data. If the taper ratio is not specified (the normal case), but we want to know what the ratio is, the following formula will help. To calculate the taper ratio amount X, when D, d and L are known:

$$X = \frac{L}{D - d}$$

To calculate the small diameter d, with D, L and TPF:

$$d = D - \frac{L \times \text{TPF}}{12}$$

To calculate the large diameter D, with d, L and TPF :

$$D = \frac{L \times \text{TPF}}{12} + d$$

To calculate the length L, if D, d, and TPF are known:

$$L = (D - d) \times \frac{12}{\text{TPF}}$$

### ◆ Taper Calculations - Metric Units

Missing drawing dimensions in *Figure 53-9* may be calculated from the given data. In metric system, the taper ratio is normally known, other dimensions can be calculated.

To calculate the small diameter d, with D, L and X:

$$d = D - \frac{L}{X}$$

To calculate the large diameter D, with d, L and X:

$$D = d + \frac{L}{X}$$

To calculate the length L, if D, d, and X are known:

$$L = (D - d) \times X$$

To calculate the ratio X (if unknown), with d, D and L:

$$X = \frac{L}{D - d}$$

## CALCULATIONS OF TRIANGLES

The most common geometrical entity in programming is a triangle. All triangles are polygons, but not all triangles are regular polygons. All triangles have three sides, although not always of the same length. There is a number of different triangles in geometry, but only a handful are used in everyday CNC programming.

### ◆ Types of Angles and Triangles

Three main groups of triangles can be grouped together by their angles - *Figure 53-10*.


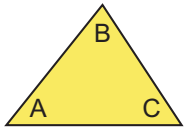
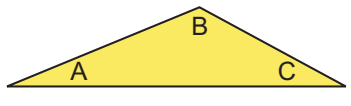
a		$A < 90^\circ$ $B < 90^\circ$ $C = 90^\circ$
b		$A < 90^\circ$ $B < 90^\circ$ $C < 90^\circ$
c		$A < 90^\circ$ $B > 90^\circ$ $C < 90^\circ$

Figure 53-10

Typical triangles a) Right triangle b) Acute triangle c) Obtuse triangle

Some more detailed definitions may be useful:

- **RIGHT** angle means that the given angle is equal to  $90^\circ$
- **ACUTE** angle means that the given angle is greater than  $0^\circ$  and smaller than  $90^\circ$
- **OBTUSE** angle means that the given angle is greater than  $90^\circ$  and smaller than  $180^\circ$
- A right triangle is also called a right angle triangle  
It defines a triangle that has one right angle ( $90^\circ$ )
- An acute triangle is also called an acute angle triangle  
It defines a triangle that has three acute angles
- An obtuse triangle is also called an obtuse angle triangle  
It defines a triangle that has one obtuse angle

In addition, there is also an *oblique* angle, which is not a new type of an angle, just a new definition:

- **OBLIQUE** angle can be either an acute or an obtuse angle, which means it cannot be  $90^\circ$  or  $180^\circ$

All triangles share a single feature - the *sum* of all angles in a given triangle is always equal to  $180^\circ$  - *Figure 53-11*.

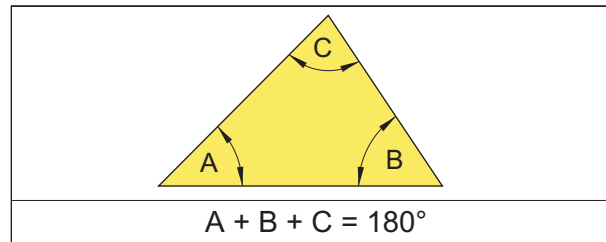


Figure 53-11

Sum of all angles in a triangle is always 180 degrees

The *oblique* triangle - and its close cousin the *isosceles* triangle - are types of triangles seldom ever needed in programming. However unlikely, it is always possible. These triangles can be solved only if at least three dimensions are known, and one of them must always be a side:

- One side and two angles must be known
- Two sides and the angle opposite one of them
- Two sides and the included angle
- Three sides

*Isosceles* triangle has two sides of equal length. Each side - or leg - is joined by a line called the base. The two angles at the base are always equal - *Figure 53-12*.

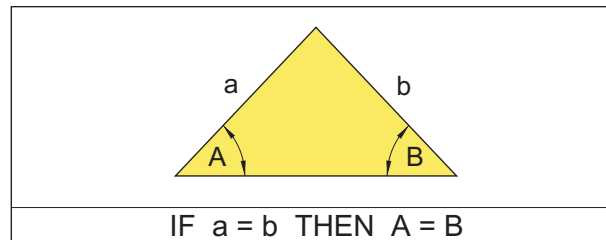


Figure 53-12

Isosceles triangle

A triangle that has all sides of equal length is called an *equilateral* triangle. An equilateral triangle is also always an *equiangular* triangle, because all internal angles are the same - each angle is  $60^\circ$  - *Figure 53-13*.

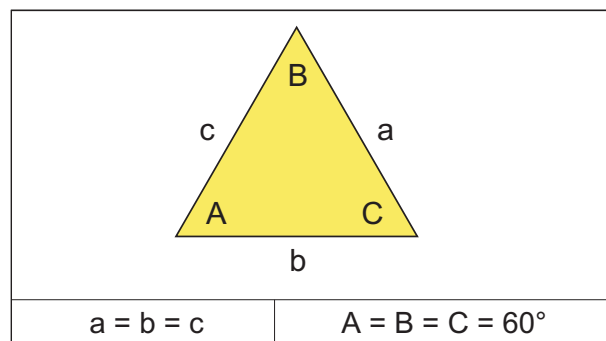


Figure 53-13

Equilateral triangle

### ◆ Right Triangles

A *right triangle* - or a *right angle triangle* - is triangle that has one angle equal to  $90^\circ$  (a triangle with two or more right angles is impossible). As there are  $180^\circ$  in any triangle (sum of all angles), that means the sum of the two remaining angles must also be  $90^\circ$ . There is a number of mathematical relationships that form the base of all calculations. Here is a look at those that are important in CNC programming. Learn these relationships well enough to be able to apply them to daily situations. Keep in mind that 99.9% of all triangles to be solved are right triangles.

The side of a right triangle that is opposite the right angle is called the *hypotenuse* and is always the longest side of the triangle. The other two sides are called *legs*. The illustration in Figure 53-14 shows a right triangle, where C angle is the right angle ( $90^\circ$ ) and the side  $c$  is the hypotenuse. The sides opposite to angles have a low case identification corresponding to the angles described in capital letters.

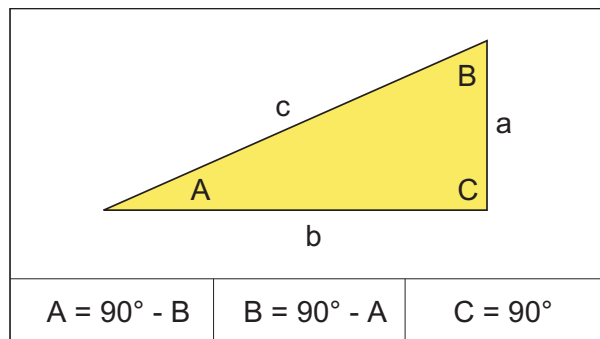


Figure 53-14

Right angle triangle and the relationship of angles

A circle drawn inside of a right triangle that is tangent to all three sides  $a$ ,  $b$ ,  $c$  - Figure 53-15 - has a diameter  $D$  calculated from this formula:

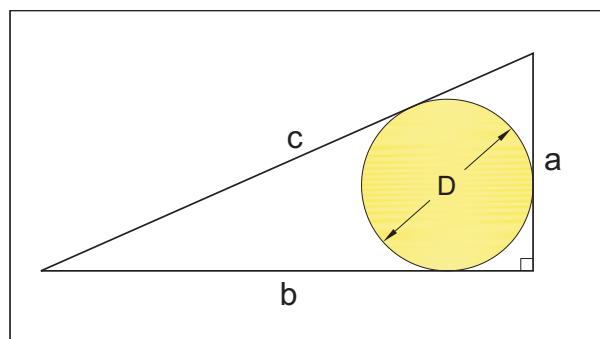


Figure 53-15

Circle inscribed in a right triangle

$$D = a + b - c$$

An inscribed angle in a semicircle is always  $90^\circ$ , as shown in Figure 53-16. Line AB is the circle diameter D.

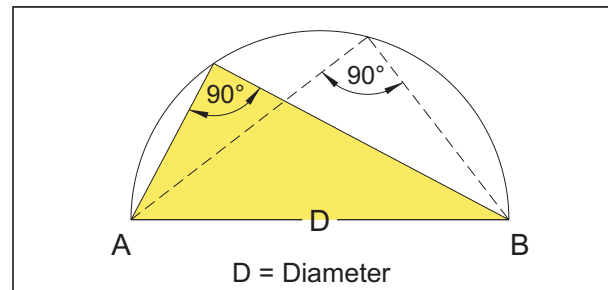


Figure 53-16

Inscribed angle in a semi-circle

In Figure 53-17 is a line from point A to the center of circle B. A line from point A to the tangency of the circle will create either a point C or point D. The angle  $a$  is created between lines AC and AD, where the line AB is a *bisector* of the angle  $a$ , creating two equal angles. The two angles  $a1$  and  $a2$  as well as triangles ABC and ABD are identical.

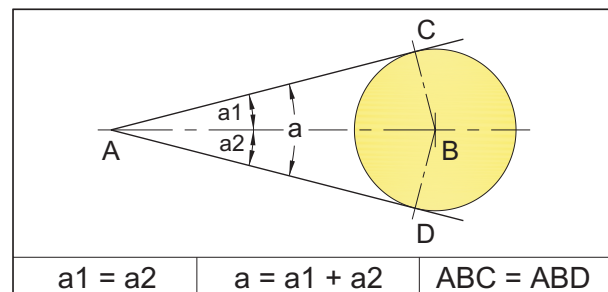


Figure 53-17

Bisector creates two equal angles

### ◆ Similar Triangles

Triangles are considered similar if they have their corresponding angles *equal* and their corresponding sides *proportional*. Any two triangles are similar, if:

- Two angles of one triangle are the same as two angles of the other triangle
- An angle of one triangle is the same as the angle of the other triangle and the including sides are proportional
- Both triangles are similar to another triangle
- The corresponding sides of the two triangles are proportional

In CNC programming, mathematical relationship of two triangles are used frequently - for example, when machining tapers or similar angular items. A good example is a taper specified in the drawing that must be extended at one or both ends, to allow for any necessary tool clearances.

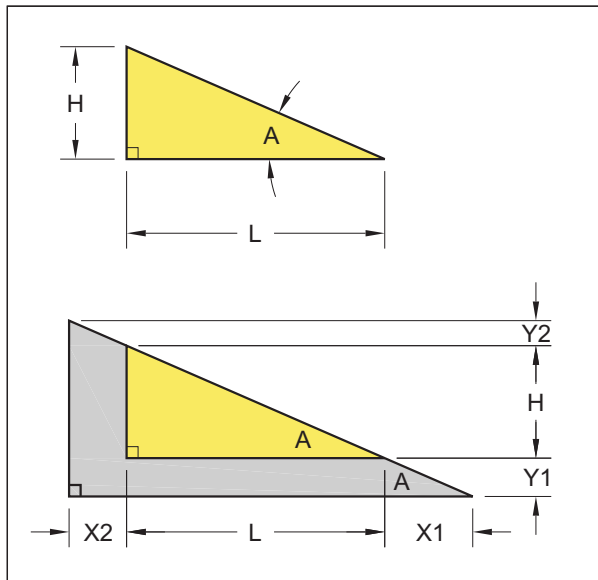


Figure 53-18  
Similar triangles - 1

The illustration in *Figure 53-18* shows the relationship between two *similar* triangles. The same illustration also includes several dimensions important for programming:

where ...

- L = Original length
- H = Original height
- A = Common (shared) angle
- X1 = Front clearance in the X axis
- X2 = Back clearance in the X axis
- Y1 = Front clearance in the Y axis
- Y2 = Back clearance in the Y axis

*Figure 53-19* shows the same two triangles in a simplified way. In the upper part, the values *X* and *Y* are sums of the extensions (clearances) from the previous example:

$$X = X1 + X2$$

$$Y = Y1 + Y2$$

The illustration bottom part shows the relationship of the opposite sides *H* and *U* to the adjacent sides *L* and *W*.

The relationship of sides as a formula is:

$$\frac{H}{U} = \frac{L}{W}$$

If *three* values are known instead of only two, the unknown value can be calculated using yet another formula. For example - the values *H*, *L* and *W* are known, while the *U* value has to be calculated, using these sample values - *H* is 13 mm, *L* is 45 mm and *W* is 57 mm. To calculate side *U*, the previous formula has to be reversed:

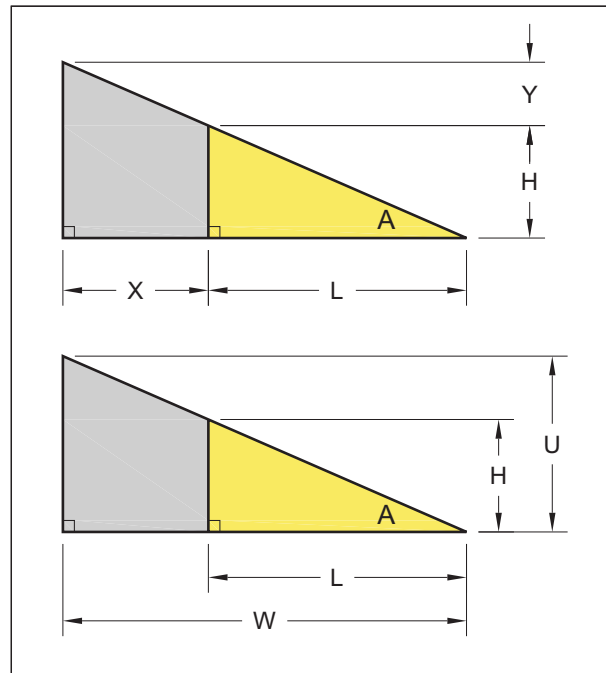


Figure 53-19  
Similar triangles - 2

$$U = \frac{W \times H}{L}$$

With known values entered, the *U* side can be calculated. If *U* is isolated on the left and the known values on the right of the equation, the calculation is simple:

$$U = (57 \times 13) / 45$$

$$U = 16.466667 \text{ mm} \sim 16.467 \text{ mm}$$

### ◆ Sine - Cosine - Tangent

*Figure 53-20* shows the most important relationships of sides and angles of a right triangle.

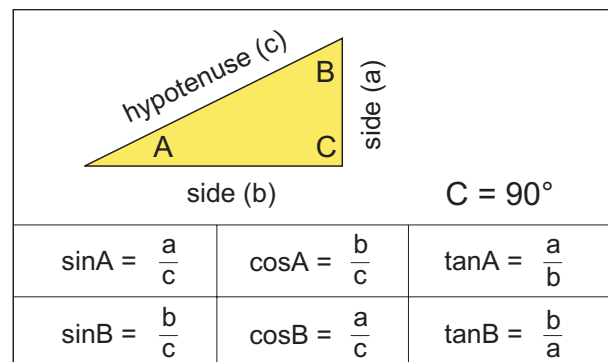


Figure 53-20  
Trigonometric functions - sine, cosine, and tangent

This relationship has its own terminology and is defined as a **ratio of sides**, using the *sine*, *cosine* and *tangent* functions of the given angle. Other available functions, namely *cotangent*, *secant*, and *cosecant*, are normally not used in CNC programming.

- Sine of an angle - abbreviated as *sin* - is a ratio of side opposite the angle to the hypotenuse of the triangle
- Cosine of an angle - abbreviated as *cos* - is a ratio of side adjacent to the angle to the hypotenuse of the triangle
- Tangent of an angle - abbreviated as *tan* - is a ratio of side opposite the acute angle to the side adjacent

### ◆ Inverse Trigonometric Functions

From all definitions, the value of sine, cosine and tangent is always expressed as a *ratio of two sides*. The angle that can be calculated from this ratio is the result of an *inverse trigonometric function*. An inverse function is sometimes associated with the word *arc*, as a prefix to any standard function. For example *arcsin* of an angle *A* is the *angle whose value* is the ratio of the side *a* and the hypotenuse *c*.

With most pocket calculators, the inverse function can be selected by pressing the **2nd** key + the standard function (*SIN*, *COS*, *TAN*) - often shown as the *power of minus 1*. Actual keystrokes vary from one calculator to another, but the result is always the same:

If ...  $\sin A = a / c$   
 Then ...  $A = \arcsin(a / c)$   
 Or...  $A = \sin^{-1}(a / c)$

If ...  $\cos A = b / c$   
 Then ...  $A = \arccos(b / c)$   
 Or...  $A = \cos^{-1}(b / c)$

If ...  $\tan A = a / b$   
 Then ...  $A = \arctan(a / b)$   
 Or...  $A = \tan^{-1}(a / b)$

While there is only a *single* result for any trigonometric function, there could be several results for the inverse function. For example, value of 0.707106781 is the *SIN* of 45°, as well as the *SIN* of 135°, and *COS* of 45°.

### ◆ Degrees and Decimal Degrees

Less common method of angle calculations in programming is *conversion of angles*. This method applies only to drawings using *minutes* and *seconds* that define precision of angles, rather than the more accurate *decimal degrees*. Older designations abbreviated as *DMS* or *D-M-S* (for *degrees-minutes-seconds*) dimensions are not suitable for program development and have to be converted to decimal degrees. Decimal degrees are **always** necessary for calculations of coordinate points, and the conversion is required.

A simple formula can be used to convert *degrees-minutes-seconds* to *decimal degrees*:

$$DD = D + \frac{M}{60} + \frac{S}{3600}$$

where...

DD = Decimal degrees  
 D = Degrees (as per drawing)  
 M = Minutes (as per drawing)  
 S = Seconds (as per drawing)

Therefore,

64°48'27" ... is equivalent to (parentheses are not required):  
 $64 + (48 / 60) + (27 / 3600) = 64.8075^\circ$

Abbreviations *DMS* or *D-M-S* and *DD* or *D-D* are common features of all scientific calculators. Interesting, but much less useful, is conversion to change *decimal degrees* to *DMS* ( $DD \Rightarrow DMS$ ). Such a conversion is not required in CNC programming, with the exception to perform a double check - to verify that the original converted result is correct. Calculation of *DD* to *DMS* is nothing more than isolating the decimal part of *DD* into three steps. For example, to convert 29.545021° to degrees-minutes-seconds (*DMS*) format, these three steps are required:

**[1]** The first step is to isolate the whole degrees amount from the decimal degrees:

$$29.545021 - 0.545021 = 29^\circ$$

**[2]** The seconds step is to take the decimal portion and multiply it by sixty, to get the minutes:

$$0.545021 \times 60 = 31.701126 = 32'$$

**[3]** The third and final step is to find seconds by taking the last decimal part and multiply it by sixty to get seconds:

$$0.701126 \times 60 = 42''$$

The final *DMS* value of the example will be 29°32'42'', with an insignificant rounding error.

### ◆ Pythagorean Theorem

The key work of the Greek mathematician Pythagoras (6th century B.C.) is known today as the *Pythagorean Theorem* - it is generally covered early in a high school mathematics curriculum. This theorem has become one of the core concepts of trigonometry - it states:

In any right triangle, the square of the hypotenuse is equal to the sum of squares of the other two sides



*Pythagorean Theorem* is used in CNC programming to find the length of *any* side in a right triangle, providing the *two other sides are known*. Figure 53-21 shows calculation of each side of a right triangle, based on two known sides:

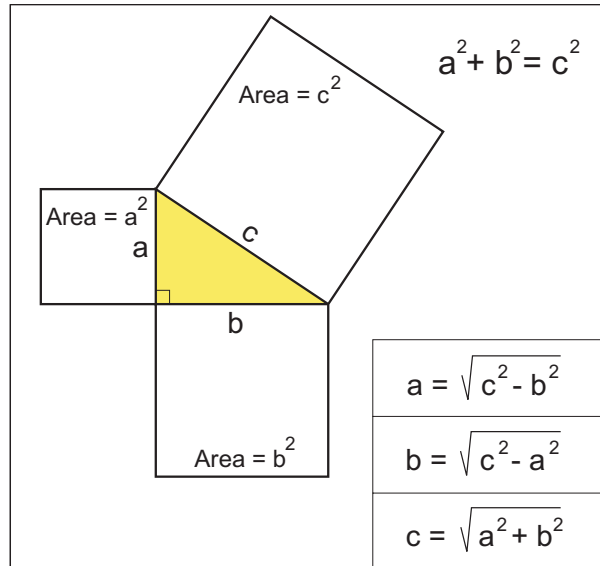


Figure 53-21  
Pythagorean Theorem

### ➡ Example ...

If the length of hypotenuse *c* is 30 mm and the side *b* is 27.5 mm, then the side *a* can be calculated as:

- ➔ *c* squared = 900.0 and *b* squared = 756.25  
... so the side *a* can now be calculated as well (symbol  $\sqrt{\quad}$  represents the square root):

$$\begin{aligned} a &= \sqrt{(30 \times 30 - 27.5 \times 27.5)} \\ a &= \sqrt{(900 - 756.25)} = \sqrt{143.75} \\ a &= 11.990 \end{aligned}$$

### ◆ Solving Right Triangles

Regardless of the actual method used, solving right triangles is a major part of manual programming. Calculations use the *sin*, *cos* and *tan* trigonometric functions, as well as the *Pythagorean Theorem*. As with any calculations, start with the known data. In trigonometry, any triangle can be solved, providing one of the two data sources is known:

- Two sides of a right triangle
- One side and one angle of a right triangle

The 90° angle is never used in calculations. Figure 53-22 shows trigonometric relations. If more than one solution is available, use both methods to double check the result.

			Trigonometry Relationships	
			SOH Sine = $\frac{\text{Opposite}}{\text{Hypotenuse}}$	
			CAH Cosine = $\frac{\text{Adjacent}}{\text{Hypotenuse}}$	
			TOA Tangent = $\frac{\text{Opposite}}{\text{Adjacent}}$	
$a = c \times \sin A$	$b = c \times \cos A$	$c = a / \sin A$		
$a = c \times \cos B$	$b = c \times \sin B$	$c = a / \cos B$		
$a = b \times \tan A$	$b = a \times \tan B$	$c = b / \sin B$		
$a = b / \tan B$	$b = a / \tan A$	$c = b / \cos A$		
$a = \sqrt{c^2 - b^2}$	$b = \sqrt{c^2 - a^2}$	$c = \sqrt{a^2 + b^2}$		
$A = 90^\circ - B$	$B = 90^\circ - A$	$C = 90^\circ$		

Figure 53-22  
Trigonometric functions - formulas for solving right angle triangles

## ADVANCED CALCULATIONS

The last two illustrations show formulas to calculate the chord  $C$  or the tangent  $T$  of a circle. Trigonometric formulas can be used as well in this case, but well developed formulas can make the same calculations faster. With only one exception, there are two solutions, dependent on the available data. The formulas can also calculate the radius  $R$ , angle  $A$  and the deviation  $d$ . Calculations relative to the **chord** of a circle are shown in Figure 53-23. Calculations relative to the **tangent** of a circle are shown in Figure 53-24.

## CONCLUSION

In this math for CNC oriented chapter, only the most important and commonly used mathematical subjects have been presented. Many more solutions and shortcuts are used by programmers and operators every day, showing their ingenuity in solving math problems. The author will appreciate any formula, shortcut or a practical solution to any programming problem - it will be seriously considered for the next edition of this handbook.

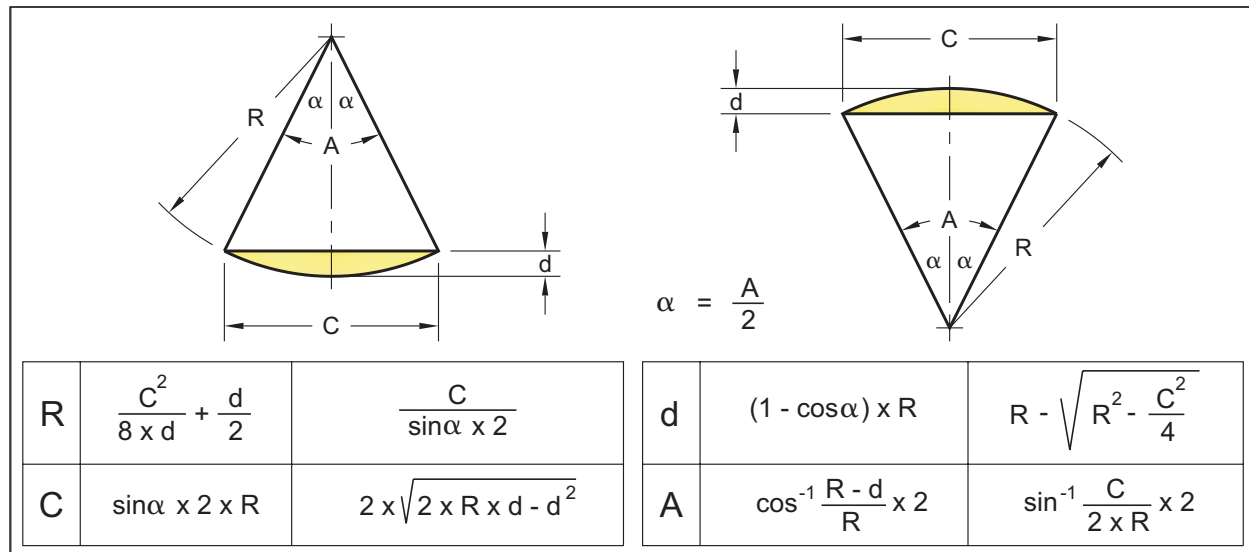


Figure 53-23

CHORD of a circle - calculations of chord, radius and deviation

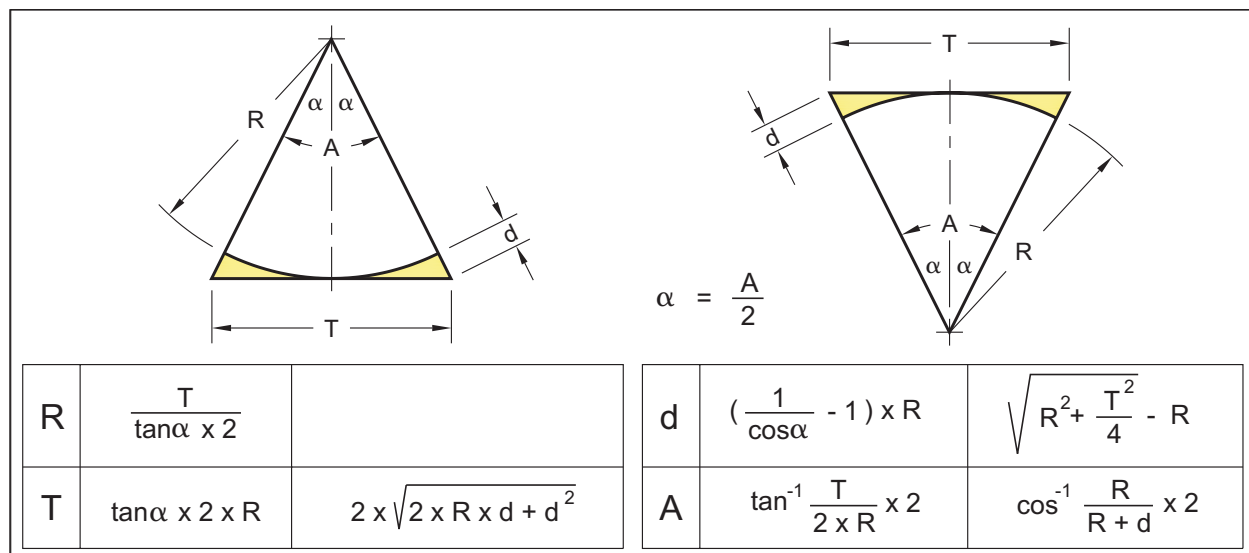


Figure 53-24

TANGENT of a circle - calculations of tangent, radius, angle and deviations

## 54

**CNC AND CAD/CAM**

Up to this point, all topics related to *manual* programming of CNC machines - all fifty-three chapters. In the last chapter, we look briefly at an area where manual programming is replaced by a computer, a suitable software and some *additional* skills. Note the word *additional*. Studying the handbook has certainly not been a waste of time. On the contrary - the handbook covers subjects that every CNC programmer should know, *regardless of the programming method used*. Programming with a computer is always desirable but to know the basic skills is the most important prerequisite. All basic skills are in understanding the manual process. All subjects and methods learned do not have to be applied by a pencil and paper. They could be applied by a CAD/CAM - or just CAM - programming. A simple statement may summarize it all:

Top class programming using CAM software requires solid knowledge of manual programming methods

**PROGRAMMING MANUALLY ?**

In the area of CNC programming application techniques, computers at all levels, from a personal computer to workstations are capable to produce most CNC machine programs in a time much shorter than any manual programming method. So, why is the high importance of manual programming methods so emphasized? Is the manual programming still alive, and if so, how healthy is it?

There are at least two important reasons why manual programming for CNC machines it is not dead yet and will not disappear anytime soon.

The first reason is that in manual programming, the programmer is able to do what computers cannot - and never will be - *programmers can think*. Manual programming teaches the invaluable lessons of discipline - a very important quality of a professional CNC programmer. Discipline means to concentrate, to constantly evaluate, to make decisions - to think all the time. In manual programming, there is a total, absolute and unequivocal control over the final product - the part program. Only a programmer can evaluate a given situation, analyze the problem and adapt to unforeseen circumstances. Only a programmer can feel that something may not be right. Only people use instruments known as thinking process, intelligence, instinct, gut feel, common sense and experience. Those are instruments inherent to humans, not computers. CNC programming is like the work of an artist - it can never be fully automated.

**◆ CAM Software**

Current CNC software, commonly known as *CAM software*, has many features that translate into a CNC program, corresponding to individual ideas of how the part program should be written. It can produce a program closely matching a particular direction of thinking, closely matching a particular programming style. But *closely* does not always mean close enough. Here comes the second reason.

The second reason is that when programming manually, the programmer *understands* the programming process and the resulting output. A program generated by a computer has to be in the format compatible with the CNC machine and its control system. If all goes well, there is no need to look at the program at all - it's there, in the files, ready to be loaded into the CNC machine. On the other hand, what if there is a problem - what then? Going back to the computer and reprogram a part may solve the problem on hand. The question is at what price. Ability to read the CNC program code, to really understand it, also means the ability to change it. Spending a valuable computer time just to add a forgotten coolant function seems excessive. Would it not be better just to edit the program by adding M08 function in the right place? Although the example is oversimplified, it also shows that real *understanding* of the programming process is very important. The best way to understand this process is to bypass the computer and get the same results. That can be achieved with manual programming.

It would be unfair to compare or promote manual programming against computer programming and vice versa. What is necessary to promote is the knowledge and understanding of manual programming principles. Without such knowledge, one can not become a good CNC programmer.

Most of the CNC programming can be done quite well on personal computers. The existing technology is progressing very rapidly and many 2D and 3D programming applications are available for a fraction of the cost when compared to just a few years ago. This trend will continue well into the future.

**◆ Desktop Computer Programming**

Complete computer system - that means the hardware, software and peripherals - suitable for CNC programming is changing at such a rapid pace that any in-depth discussion of the hardware would be obsolete in a matter of weeks. Almost the same speed of obsolescence applies to software as well. New features, new capabilities, new tools

are constantly arriving on the market and are offered to users in both areas. Because hardware and software has to be considered together, the first question is what to select first - the hardware or the software?

Such a decision must be based on the required application. What will the computer be used for? What kind of work needs to be computerized, automated? What results are expected? These are the *primary* considerations - *not* the kind of monitor or printer or the capacity of hard disk. They are also very important - but only *after* establishing the application needs.

Certain programming applications are typical to all machine shops. Others are unique to a particular type of manufacturing and the kind of work or the product manufactured. The following short list itemizes the major groups that a typical computer based CNC programming system should have:

- Tool path geometry creation environment
- Tool path generation
- Complete programming environment
- Post processing
- Training and technical support

It is important to understand *why* these features are important. Before investing into a technology that is all or partially new to the user, it helps to know *what* tools the software offers and *how* they can be used in everyday work.

## TOOLPATH GEOMETRY DEVELOPMENT

---

Most CNC programming systems require a toolpath geometry creation before the actual path of a cutting tool can be generated. The key words here are *toolpath geometry*. A common misconception among programmers is that they have to re-create everything in the original drawing. That is a wrong approach.

When it comes to toolpath geometry, two scenarios must be faced. One will be work from a paper drawing, the other from a CAD drawing stored in the computer. Although there are differences in approach, the fact remains that either a new geometry is created or an existing geometry is modified.

Modern CAM systems allow drawing in a CAD like manner - using editing features such as trim, fillet, break, copy, move, rotate, offset, mirror, scale, and so on.

Typically, programmer will define what is normally not on the drawing, at least not on a two dimensional representation of the part. Adding depth, separating entities by color, by levels (layers), adding clearances or a special lead-in and lead-out tool motion, and so on.

## TOOL PATH GENERATION

---

The key requirement of a CNC software is to produce a program of an accurate toolpath for a specific CNC machine. Toolpath creation, with all its calculations, is the most time consuming task in manual programming. It makes sense to make it the most important item to consider when planning to automate CNC programming process. Only high level CNC software supports a large variety of toolpaths. For example, helical milling or a full 3D machining are not always standard features in the software.

One mistake in software selection is to consider only the *existing* CNC machines and existing machining methods and practices. This rather narrowly focused approach is not always successful. Consider future plans in both strategies and capital investment. What about the product? How will the product change in five years? Knowing the philosophy and focus of the company, its policies and management strategies and yes - even its politics - will help to make a more accurate estimate of future needs.

Computer technology has grown a lot, yet it is so new that it is in the state of constant development. Nobody can predict with absolute accuracy what the future will offer in terms of CNC machining and CNC programming. If the current and the future needs are well established *before* purchasing a programming system, there is a good chance to beat obsolescence for a long time. CNC software developers offer periodical updates to their product, with more features added as computing power increases. The updates (new versions of the software), usually reflect developments of the technology, both on the hardware and software sides. It does not mean purchasing every new update offered, but it is important to select a CNC software developed by a solid and well established company that has the best chance to be still in existence when the need to update a system comes up. Computer industry is very active, and mergers, acquisitions and takeovers are as common as bankruptcies and failures.

## COMPLETE ENVIRONMENT

---

A typical high quality CNC programming software allows all programming and relating tasks to be done from a structured menu, using a mouse or similar pointing device. The important thing is that once the software is loaded, it can complete all tasks without returning to the operating system level. Some programming systems are based on modules and files that are not accessible from a menu, or they do not cover all the steps in programming.

The following list is meant only as a very brief guide to some of the main features that apply to CNC programming on personal computers. These are the expected features from any CAM software:

- Multi machine support (machining centers, lathes, EDM)
- Associative operations for flexible editing
- Job setup and material blank definition
- Tooling list and job comments (setup sheets)
- Connection between computers (communications feature)
- Program text editor (with CNC oriented features)
- Printing capabilities (text and graphics)
- Pen plotting (plotters)
- Interface with CAD software (DXF, IGES, STEP, STL, ...)
- Support for solid modeling (third party)
- Software specifications and features (including customizable post processing)
- Support for generally available hardware
- Utilities and special features, open architecture

Each described item will point out its significance. Although all items are useful in a programming system, it does not mean that all items are always necessary. Some features require an additional hardware equipment, such as a printer, plotter, cabling, small peripherals, etc.

#### ◆ Multi Machine Support

When it comes to support of different machine types and models, CNC software can be divided into two groups:

- Dedicated software
- Integrated software

*Dedicated* software supports only one kind of machines. For example, a software that is designed specifically to produce programs for CNC fabrication equipment, cannot be used for lathes, machining centers or EDM.

Dedicated software is often developed for a rather narrow and very specialized field of applications or when it applies to a particular machine only. CNC punching, forming and press brake equipment are good examples of such software.

*Integrated* software allows the programmer selection of several types of machine tools. Such a selection usually offers milling, turning and wire EDM. It is also common to use the software for machines such as burners, routers, laser cutters, waterjets, and profilers. For metal cutting, this is the preferred type of software.

Another reason also speaks clearly in favor of integrated software, and that is its *interface*. It is much easier to get used to one display for a lathe work and have almost the same display for milling work or EDM work. Software menus look the same, the navigational operations share common menu items, customization of the software (including post processors) is much simplified.

#### ◆ Associative Operations

When a toolpath is developed, it is attached to the previously defined toolpath geometry. For many reasons, it is not unusual to change the toolpath geometry later. The traditional method has been (and for many software vendors still is) to recreate the geometry, then recreate the toolpath.

Associative operation avoids creation of a new toolpath, it updates it automatically. It is fast and accurate. It works the other way as well - many tooling parameters can be changed quickly, on demand.

#### ◆ Job Setup

Job setup is a feature that describes the material blank of the part - its shape, dimensions, zero origin, and many other related items. Tools and related speeds and feeds can be often selected from the job setup, as well as various program parameters. Libraries that store common data for tools, materials and operations are also powerful software features.

#### ◆ Tooling List and Job Comments

CNC programming is a process covering several steps. Whether programming manually or with a computer, the selection of cutting tools is a manual task. Once selected, each tool is assigned its identifications, speed and feed values. Several tools can be grouped into a tooling library file and stored. Then, the order of their usage within the program is selected. Some parts require more than one machining operation. Complex setups require special instructions to the machine operator (setup sheet), describing the programmer's intents. All these programming decisions must be recorded and the documentation sent out to the machine shop. It is only reasonable to expect that any CNC programming software will support a tooling list, perhaps in a form of a tool library file and the process list. Material library file is also very useful, as it can store surface speeds for many materials and the programming software will calculate the exact spindle speed and feedrate, based on the tool selected. This is a good example of interaction between the tool library and material library.

#### ◆ Connection Between Computers

A programming system should also include a connection (communications option) between the personal computer and the CNC machine. This feature allows the program data exchange via a cable. Programs can be sent from the computer to the memory of the CNC machine and back.

An important point is that not all CNC machines have the port (outlet) and the capability to take advantage of direct connection. Even if all machines in the shop have this capability, it requires additional hardware and organizational discipline to make all elements work in harmony. The existence of a direct connection in a programming software is a must, even if it is not used immediately after the purchase.



### ◆ Program Text Editor

A CNC program generated by the software should be 100% complete and ready for use by the machine. The implication is that such a program is so perfect that it needs no further editing. This is the ideal way, the way it should happen. If a change in the program is needed, it should be done *within* the design of the part shape and that means through the CNC software - *not outside of it*. The reason is that any manual change to the generated program does not correspond to the program data as generated by the computer. In the environment where the data is shared by many users, such a practice will cause a lot of problems.

That brings up a question - why does a CNC software have a built-in text editor? There are two reasons. One, the editor can be used for creating or modifying various text files such as setup sheets, tooling sheets, operation data, post processor templates, configuration files, special instructions, procedures, etc. These files can be updated and otherwise modified as required, *without* a damage to the program database. The second reason is that in some *special* circumstances, a CNC program can be edited outside of the computer, providing such change does not modify significant data. For example, to add a missing *coolant* function M08 to the part program is much faster done in the text editor, than repeating the program generating process with the computer. Purists are right, it is not the right way of using a text editor, but at least the significant data (tool locations) are not tampered with and the database is otherwise completely accurate.

Many programmers use various external text editors or even word processors in text mode. These types of editors are not oriented towards the CNC programming, since they lack some features typical to the CNC program development. Only a CNC oriented text editors can handle automatic block number sequencing, removing the block numbers, adding cosmetic spaces in the program and other functions. The editor should be accessible from the main menu or from within the software.

### ◆ Printing Capabilities

Any text saved into a file, CNC programs included, can be printed using a standard printer. Paper copy is often necessary as a reference for the CNC operator, for stored documentation, or just for convenience. Printer does not need to be top of the line, just one with a standard paper width. Some programming software supports an option that is known as a printer plot or a hard copy. Hard copy is a graphic image of the screen transferred to the printer. The image quality is usually more than adequate. This hard copy is an excellent aid during program development stage. Better quality printer provides better quality print plot. The printer support is provided by the Windows environment, as most PC based CAM software is developed for the Windows operating system.

### ◆ Pen Plotting

Pen plot will usually produce image quality superior to the printer plot but for a CAM programming it is an unnecessary luxury. The only time when a pen plotter can be beneficial is for plotting to paper size that is not supported by standard printers. Other reasons will be the need for a color output, a special requirement by customers, or special documentation development. Before the graphics software appeared on the market, plotters were widely used to verify the tool path. Now, the toolpath is verified directly on the computer display screen, during interactive programming process, including different views and zooms. Printing can utilize monochrome or color laser printers. Most plotters are HPGL compatible. HPGL is an acronym for *Hewlett-Packard Graphics Language*, and is currently the most supported plot file exchange format.

### ◆ CAD Software Access

If an engineering drawing is generated by CAD software, all drawing information is stored in a computer database that can be accessed by other software, through a file format translation utility (more on the subject later). Once the CNC software accepted and processed the database from a CAD system, the CNC programmer can concentrate on generation of the toolpath itself, rather than defining toolpath geometry from scratch. Some modifications are usually necessary, so expect them. The most significant advantage of a quality CAD/CAM system is the avoidance of duplication. Without CAD system, CNC programmer has a lot of extra work to do, much of it is duplicated.

A high quality CNC software also allows the existing program file to be translated the other way, to a file that a CAD system can accept. This option is called *reversed processing*, and can be a benefit to companies that want to translate existing programs generated manually to an electronic form. Usually some additional work is required in these cases.

High level CNC software is a *stand alone* type. Stand alone software means that it does not need an access to a CAD system - toolpath geometry and toolpath itself can be developed from within the CAM software, independently of other software.

### ◆ Support for Solids

Solid modeling for 3D applications had been for a long time the domain of large computer systems. With the advance of powerful microcomputers, solid modeling is now an optional part of high level CNC software.

With solid models, the machining process of complex surfaces is much more streamlined. In addition, solid models offer the benefits of supplying engineering data, easier manipulation of objects, and many other features.

### ◆ Software Specifications

Another benefit of a high level CNC software is that it comes well supplied with a variety of useful features. What makes each system unique, is usually the method of how the programming process is executed. In the early years of development, programming was done by using special programming languages, such as APT™ or Compact II™. Some languages are still available but heavily on the decline. Modern interactive graphics programming has virtually eliminated the need for languages in just about all manufacturing fields. The more popular kind of programming is based on *interactive graphics*. Programmer defines geometry, typically as toolpath geometry, followed by the toolpath itself. Any error in the process is immediately displayed on the graphic screen and can be corrected before too much other work is done.

### ◆ Hardware Specifications

Specification of the software will determine the hardware selection. Hardware is a common term for the computer, monitor, keyboard, printer, modem, plotter, mouse, scanner, disk drive, storage media, CD/DVD writer, and many others. Hardware referred to in this chapter is based on the *Windows™* operating systems. Modern operating systems are based on a *graphical user interface (GUI)*. Some software can run under a different operating system, for example Unix (used mainly by large workstations) or different Windows versions. It is always to the user's advantage that the latest version of the operating system and CAM software is installed on the computer.

When thinking of purchasing a computer hardware, consider carefully at least three major criteria:

- Performance                   ... computer speed
- Data storage                   ... type and size
- Input / Output               ... ports

#### *Computer Speed*

*Performance* of the computer system is typically measured by the relative speed of the main processor. The higher the number, the faster the computer can process data. To make the comparison easier, the original IBM PC, model year 1983, had a 4.77MHz processor speed. Later model AT had 6MHz processor speed, improved further to 8 and 10MHz. Later, computers used the so called 386 microchip (generally Intel 80386 or 80486) and reached 25MHz, 33MHz and more. Pentium processors followed, and the process is ongoing. For serious CAD/CAM work, the latest fully featured processors should be used. Newest processors offer much higher processing speed, measured in GHz, and the more processing speed is available, the better performance of CNC programming system. Fast graphics card is also of the highest importance.

#### *RAM and Data Storage*

*Data* is stored in the computer in two forms - memory storage and disk storage (file). When an application such as CNC programming is started, CAM software is loaded into the computer memory. The more powerful the application software, the more memory it requires. This memory is known as *Random Access Memory*, usually called *RAM*. Every software specification identifies the minimum available RAM required. RAM of today high level computers around two to four gigabyte range is not uncommon. Any extra memory will speed up processing quite significantly. Data in the RAM is volatile, which means it is lost when the application is ended or the computer power is interrupted. To save important data from RAM into disk files, a hard disk or similar media can be used. For a micro computer CAD/CAM work, the absolute minimum requirement is high density removable drive and one large size hard drive. Floppy drives of any kind are not suitable.

The hard drive should have a fast access time and a high storage capacity. Another option is a tape drive, CD-R and CD-RW disks or recordable DVD disks for backup.

#### *Input and Output*

*Input and Output (I/O)* computer features, cover hardware items such as monitor, graphic card, keyboard, digitizer, scanner, printer and plotter. Monitor suitable for CAD/CAM work should be a large size color monitor providing very high resolution. The monitor and graphic card do relate to each other. The card must be able to generate the image, the monitor must be able to display the image. Speed of the video output is also very important.

A keyboard is a standard feature of a computer and serves as a basic input device. Mouse (or a digitizer on larger systems) are also input devices, but much faster than keyboard input. In CAD/CAM, where much work is done in graphic mode under a menu system, an item from the menu is user selected. In most cases it can be selected with a pointing device. User points at the menu item desired, presses a button on the device and the menu item is executed. Pointing device most suitable for CAM work in the Windows environment is a mouse with several buttons and a scroll wheel.

Both the printer and plotter are theoretically optional, but generally worth some consideration. For CNC work alone, a printer is more important than a pen plotter. If the setup is a true CAD/CAM, both peripheral devices may be needed.

All peripherals are interfaced with the computer using specially configured cables connected to the *Input/Output (I/O)* outlets called *ports*. Modem is normally not required for CNC programming, except for data exchange with a remote computer or Internet access. Laser or ink jet printers generally use a parallel interface known as the *Centronics* standard, but many other devices use a serial interface. There are also other popular I/O options, such as the USB (Universal Serial Bus) or Firewire connection.

### ◆ Typical Hardware / Software Requirements

Currently, the most popular hardware for CNC programming is the Windows based computer system. It is not possible to make a simple 'shopping list' for all hardware requirements that every CNC machine shop can use. Here are some rules applicable to any system and are not subject to becoming outdated very quickly. A typical list of minimum hardware requirements and options may be compiled:

- Hardware compatibility with IBM (Windows based)  
Apple computers have very limited CAD/CAM applications
- The latest version of the Windows operating system  
(must be supported by the CAM software)
- High central processor speed - higher = better  
(measured in GigaHertz units - GHz)
- Fast memory cache
- Requirement of a numeric (math) processor, which  
is normally part of most main processors
- Random Access Memory (RAM) - as much as possible
- Enough of hard disk space for program and data storage  
(measured in gigabytes or higher - with a fast access time)
- Backup system for data protection  
(tape cartridge, removable drive, CD, DVD, ...)
- High resolution graphics adapter (graphics card)  
(should have a rapid refreshing for the video output)
- Large high resolution color monitor - non-interlaced  
(measured in pixels - the more pixels per screen size,  
the finer the display, and the smaller the pixel size,  
the better the display)
- Pointing device - normally a mouse - is a current standard
- Pen plotter is required only in special circumstances  
(not needed for CNC work) - B size maximum is usually  
enough, if needed
- Working real time calendar clock (stamps all created  
files with the current date and time - standard feature)
- A good quality printer with a parallel or USB port  
(for hard copy documentation)
- CD or DVD drive & various multimedia features  
(sound card necessary)
- Access to additional global information  
(Internet, E-mail, user groups, newsgroups, ...)
- Two or more serial and USB ports
- Text editor - usually part of the software (or optional)

It is smart to keep abreast of the micro computer technology. It develops rapidly and even a few weeks may change some fundamental approaches and decisions. Following the development of computer technology creates awareness of the latest improvements, therefore a more educated user and/or buyer.

### ◆ Utilities and Special Features

Even the most updated version of the operating system is never as powerful and flexible as many users would like it to be. For that reason, many software developers came up with literally thousands of programs and utilities that supplement the readily available features. Many of these utilities are available as shareware or freeware from the Internet and other sources. Access to the Internet and the World Wide Web provides a great source of CNC and machine shop related topics and general information. These utilities are not necessary to use a CAM software, but they are a great time saver for many tasks associated with using a computer.

## POST PROCESSORS

---

CNC software must be able to output a program in a format unique to each control unit. The most important part of toolpath generation is data integrity. Computer generated program must be accurate and ready for the CNC machine. That means the completed program should require no editing, no optimization, no merging with other programs or similar manual activities. Such a goal can be achieved only by a well developed programming style - and a properly configured *post processor* for each different CNC machine.

A top quality post processor is probably the most important customized feature of a CNC software. When entering data into the software, settings describing the part shape, cutting values, spindle speeds, and many other data are stored for further processing. The software analyzes this data, sorts it and creates a database. This database represents the part geometry, toolpath geometry and other functions. CNC system cannot *understand* the data, regardless of its accuracy. To complicate things even more, every CNC system is different. Some program codes are unique to a single machine, some are quite common to many machines. The purpose of a post processor is to process the generic data and convert them to the machine code for individual control systems.

### ◆ Customizing Post Processor

Typically, a supplied post processor is more or less generic and has to be customized, at least to some extent. To develop a post processor in-house, usually means to customize the generic post processor supplied with the CAM software. Typical process depends on the type of post processor and its format. Small changes may take minutes, large changes days. Post processors can be very expensive.

CNC programmer must know the machine and control features extremely well. A deep and thorough knowledge of manual programming methods is a must - how else can a useful programming format be developed? Also important is the knowledge of machining methods. Finally, knowing any high level language can make the post processor development much more efficient and powerful.

## IMPORTANT FEATURES

---

There are several important features to look into when investing into a CNC programming software. They do have an impact on the final functionality of the program, at machine level. All these features are important and should be considered carefully.

### ◆ Input from User

One of the important features of a CAM programming software is its ability to handle input from the user. This input can be a special sequence of commands that cannot be handled by post processor at all, or would require too much effort. These commands are usually small in size and can be called and used in the graphics mode whenever required. Examples of such applications are a barfeeder sequence on a lathe or a pallet changing routine on a horizontal machining center. If the software supports some type of variable type of user commands, it adds an extra flexibility and power to the system.

### ◆ Machining Cycles

Another very important feature of a CAM software is its ability to generate a variety of fixed and repetitive cycles, that modern controls support. These cycles make a manual programming simpler and faster. Modern CNC systems take advantage of such cycles are available with a limited memory capacity. For that reason, support for such cycles is very important in CNC software, as it provides easy editing at the machine.

### ◆ User Interface

Customizing the display is also a useful feature. It is not as critical as others, but a facility to customize fonts, colors, toolbars, even menus adds extra power to the software. Colors are very important in CAD/CAM work. Color settings should be changeable to provide better distinction. Screen appearance may be changed by a different combination of colors for the foreground, background and text. The result is visual emphasis on what is important.

The last user interface feature is selection of verification options in the software. When the toolpath simulation is shown on the screen, a circle represents the tool diameter for milling applications, and the shape of a turning tool for CNC lathes. This tool image shows the current tool position, valid for the processed program section. Normally, the graphic image moves along the contour, without leaving any traces. A variation is that the tool will remain at the contour change points only, but nowhere else. This is called static display and is very important for some machining operations. Premium CAM software also allows to design a customized tool shape, including the tool holder and use it on the screen to simulate actual tool path. Shaded 3D tools add even more realism to program viewing.

Also important is the representation of the toolpath for lathe tools. Many cutting tools for CNC lathes have a back angle. A high quality software should also evaluate the tool back angle in its calculations and in the display.

### ◆ CAD Interface

A stand alone CNC programming system does not need a CAD software for geometry definitions. It can create its own. Yet, in a any CAD/CAM system is important to have the *option* of importing part geometry *from* a CAD system. Even if a company does not need CAD, it should be prepared to accept its files, perhaps from customers or company branch offices. Support for acceptance of native files is becoming the new standard.

Needless to say, if programming software cannot accept native drawing files, another solution must exist. Files from different vendors are proprietary and their structure is not a matter of public access. There is another way - *use a different file format*.

#### *File Exchange Formats*

The need to exchange design files between different software systems has always been a prime requirement. There are many competing formats of a neutral file format. The oldest of them is called *IGES (Initial Graphics Exchange Specification)*, originally developed to transfer complex design files from one software to another. Another format that is also used, is the *DXF* format by Autodesk™.

The *DXF (Drawing eXchange Format or Data eXchange Format)* is considered by many to be the standard of drawing file exchange between micro computers. It has been developed by Autodesk™, Inc., developers of the popular AutoCAD™, the most widely used PC based CAD in the world. DXF format is suitable only for common geometric elements, such as points, lines, arcs and a few others.

CNC software should also support an interface between the neutral files generated by a CAD system. Depending on the nature of a particular programming application, DXF interface may be needed for simpler jobs, and IGES for more complex geometries. High quality CNC software offers at least these two formats, usually many more. Keep in mind that the format and structure of the translators, such as DXF or IGES, is not in the hands of the CNC software developer, therefore it is a subject to change.

## SUPPORT AND MANAGEMENT

---

Hardware and software for CNC programming work can be costly. It can represent a significant investment of money and people and can become a total failure if it is not used properly. A failure is not the actual loss of the hardware and software cost. The real and heaviest loss is in the increased productivity, speed and quality that was expected but never materialized. Loss is also in the confidence the company



employees put into the technology. These losses can be high. To prevent such prospects, keep three key elements in mind when planning a CNC programming system:

- High quality training program for long term skills
- System management philosophy and strategies
- Technical support for hardware and software

No single item in the list is any more important than the others - they are *all equally* important.

### ◆ Training

Training should be planned, thorough, and professional. Many successful programs apply three levels of training. Some companies do not place enough emphasis on training, despite many studies and examples proving that good quality training does work. The lack of time and perceived high costs are often used as excuses. Training is a necessary investment for any company that wants to be competitive.

#### *Training Level 1*

The *first level* of training should be aimed at the person with *none* or *very little* computer experience. It should introduce CNC software to the programmer who programs manually. It should be an overall training, mainly general in nature, with the main emphasis on system features and capabilities - as they relate to the company where the software is installed. Typical general approach should be balanced by explaining philosophy behind software design, and structure of menus and commands. It is very important to show the student what the software can do in skilled hands. The first level should be done when the software is purchased. The objective is to give the programmer enough tools to *play* with the software, to grow into it. A simple way to achieve this goal is to try out simple projects while still programming manually more important jobs.

#### *Training Level 2*

The *second level* is most beneficial two or three weeks *after* the first level completion. It is the most critical level of the three. It should include a *systematic approach* to all software features, with special emphasis on features relating to machining operations used locally. This training level eliminates manual programming, and marks the beginning of a new era. Supervisors should evaluate the complexity of the first few jobs to be programmed and select, if possible, the less difficult jobs to build a little confidence.

#### *Training Level 3*

The *third level* is usually done 2-3 months later. It covers problems, questions, difficulties and concerns, introduces tips, shortcuts, etc. The purpose of this level is to create a long term confidence. At this stage, the programmer has many questions. Professional instructor can answer all questions, weed out bad habits, and offer further guidance.

### ◆ System Management

A reliable operation of all system elements is crucial to the success of CNC software. Use of any software requires good organization, it needs strategies, it needs focus, and it definitely needs a professional management. System management establishes standards and procedures for CNC and related operations. Concerns about people selection, data backup methods, confidentiality and security, work environment quality, etc., are not confined to a single discipline and should be important in the overall company culture.

### ◆ Technical Support

Technical support is an important part of system management. A service contract or a support package can be usually negotiated with the vendor, covering installation, hardware, update policies, new developments, information flow, etc. An important part of technical support is the speed and reliability of handling *emergency* situations. If a hard disk fails - and a data back up *does* exist - what can be done? CNC shop is waiting for the critical job, while the programmer cannot send program data to the machine, because an inexpensive hard disk failed. Support should cover both hardware and software. All support promised by the vendor should always be written down. Know *exactly* what the bill is for. *If something isn't in the contract, it usually isn't available.*

## THE END AND THE BEGINNING

What the future of CNC technology holds is always hard to predict. There are many indications where the technology will be going. System controls with more computing power, more standardized approach to programming, more solid modeling, more 3D, better storage methods, etc., are in the works. Changes are also inevitable in work skills.

Stand alone CNC machines will always be needed. On CNC machining centers, there will be much more emphasis on faster machining rates. On CNC lathes, the natural way of development would be to adapt the tool indexing techniques of machining centers. This would increase the number of cutting tools available and keep inactive tools away from the machining area. Also watch for features that eliminate secondary operations, such as complex milling features on lathes and built-in part indexing and reversal.

Predictions for computers are difficult, except that their power will increase. Hardware has developed faster than software and this will not change. CNC software is no exception. Winner of the competitive race will be the one that can combine hardware, software and people, makes a product for reasonable price and markets it around the world. Protectionist economy does not work and trading will not be confined to several 'local' blocks - it will be part of true global economy. Before too many personal opinions force their way out, it is time to end and say "*Learn, work, and then learn again*".



# About the Author

Peter Smid is a professional consultant, educator and speaker, with many years of practical, hands-on experience, in the industrial and educational fields. During his career, he has gathered an extensive experience with CNC and CAD/CAM applications on all levels. He consults to manufacturing industry and educational institutions on practical use of Computerized Numerical Control technology, part programming, CAD/CAM, advanced machining, tooling, setup, and many other related fields. His comprehensive industrial background in CNC programming, machining and company oriented training has assisted several hundred companies to benefit from his wide-ranging knowledge.

Mr. Smid's long time association with advanced manufacturing companies and CNC machinery vendors, as well as his affiliation with a number of Community and Technical College industrial technology programs and machine shop skills training, have enabled him to broaden his professional and consulting skills in the areas of CNC and CAD/CAM training, computer applications and needs analysis, software evaluation, system benchmarking, programming, hardware selection, software customization, and operations management.

Over the years, Mr. Smid has developed and delivered hundreds of customized educational programs to thousands of instructors and students at colleges and universities across United States, Canada and Europe, as well as to a large number of manufacturing companies and private sector organizations and individuals.

He has actively participated in many industrial trade shows, conferences, workshops and various seminars, including submission of papers, delivering presentations and a number of speaking engagements to professional organizations. He is also the author of articles, has a monthly CNC related column in ShopTalk Magazine, and many in-house publications on the subject of CNC and CAD/CAM. During his many years as a professional in the CNC industrial and educational field, he has developed tens of thousands of pages of high quality training materials.

Peter Smid is also the author of two other popular CNC books:

**CNC Programming Techniques**, *An Insider's Guide to Effective Methods and Applications*  
ISBN (0-8311-)3185-3

**Fanuc CNC Custom Macros**, *Practical Resources for Fanuc Custom Macro B Users*  
ISBN (0-8311-)3157-8

Both books have been published by Industrial Press. Inc. and are also available as eBooks.

The author welcomes comments, suggestions and other input from educators, students and industrial users.  
You can e-mail him through the **Main Menu** of the enclosed CD.

You can also e-mail him from the **CNC Programming Handbook** page at [www.industrialpress.com](http://www.industrialpress.com)

# Acknowledgments

In this third edition of the *CNC Programming Handbook*, I would like to express my thanks and appreciation to several people who provided valuable input into various subjects. My sincere thanks to:

- **Peter Eigler** for being the bottomless source of new ideas, knowledge and inspiration
- **Steve Gallant** and **Nick Eelhart** of Accucam Ltd., Cambridge, ON, Canada
- **Ferenc Szücs**, Application Engineer, DMG Trainer, Hungary
- **Wayne Pitlivka** of FerroTechnique Inc., Mississauga, ON, Canada
- **Marc Borremans**, Erasmus Hogeschool, Brussel, Belgium
- ... and many others

Even after three years of improving the *CNC Programming Handbook* and developing the enclosed compact disc, my wife Joan will always deserve my thanks and my gratitude. To my son Michael and my daughter Michelle - you guys have contributed to this handbook in more ways than you can ever imagine.

I have also made a reference to several manufacturers and software developers in the book. It is only fair to acknowledge their names:

- FANUC and CUSTOM MACRO or USER MACRO or MACRO B are registered trademarks of Fujitsu-Fanuc, Japan
- GE FANUC is a registered trademark of GE Fanuc Automation, Inc., Charlottesville, VA, USA
- Mastercam is a registered trademark of CNC Software Inc., Tolland, CT, USA
- EdgeCam is a registered trademark of Pathtrace, Inc.,
- NCPlot is a registered trademark of NCPlot LLC, Muskegon, MI, USA
- AUTOCAD is a registered trademark of Autodesk, Inc., San Rafael, CA, USA
- Kennametal is a registered trademark of Kennametal, Inc., Latrobe, PA, USA
- WINDOWS is a registered trademarks of Microsoft, Inc., Redmond, WA, USA

# README FILE

Peter Smid

## CNC PROGRAMMING HANDBOOK

Third Edition with CNC Projects CD-ROM v3.0

© 2000-2008 Industrial Press, Inc. - All Rights Reserved

The first and second editions of the **CNC Programming Handbook** have met with great success and each became an instant international bestseller. Many CNC programmers, machinists and operators, managers, supervisors, engineers, instructors, students and other readers have found the handbook an invaluable resource of information and illustrative examples.

This **Third Edition** of the **CNC Programming Handbook** goes another step further and provides enhanced chapters and illustrations, hundreds of practical examples (many new ones), solutions to problems, reference files, handouts and many other resources. Index has also been enlarged, for easier subject search. Within text, there are many references to actual page numbers, also to simplify search for related subjects. One of the most significant changes in this edition is the inclusion of a toolpath simulation software called **NCPlot** ([NCPlot website](http://www.ncplot.com)). This 15-day, fully functional shareware version, supports virtually all subjects covered in this handbook. All these additional resources are on the CD-ROM enclosed with the **CNC Programming Handbook**.

## CD-ROM

If every page of every file on the CD-ROM were to be printed, the number of sheets of paper would go well into many hundreds. This special CD offers a unique experience in finding any required material - the file search uses a logical menu structure with graphical preview, whenever applicable.

### For the instructors using the *First Edition* for training

The major addition to the *Second Edition* and *Third Edition* is the inclusion of the CD-ROM. The hardcover book did get a fresher look, but its contents were virtually intact in the *Second Edition*. There are no changes to the page numbers in the *Second Edition*. This *Third Edition* offers a significant change in forms of additions - no subject has been removed - which resulted in changes to page numbers.

To instructors, it means a little bit of work, because any training material already prepared that uses page numbers as references does need updating.

Inclusion of the CD with toolpath simulation will enhance the existing training, as the majority of projects are based on **CNC Programming Handbook** chapters and also follow the numbering of individual chapters.

### Why the CD-ROM?

The high item on the wish list from the users of the first edition was a desire to have not only the examples that are already in the book, but also actual programming projects and exercises. It would be virtually impossible to add even a few such projects to the book without making it too big and, therefore, more expensive. The author has chosen the approach of providing these projects on a CD-ROM, included with the *Second/Third Edition* and - at no additional cost. Another wish list that was high on the minds of many readers was the lack of toolpath simulation software. That has been fixed and NCPlot 15-day evaluation shareware is included on the CD. Special pricing for students enrolled in CNC classes may be available - check at [www.ncplot.com](http://www.ncplot.com).

## Automatic Startup and Setup

The enclosed CD-ROM is self-running, using the 'autorun' technology - when you place the CD into the CD-ROM drive of your computer, the operating system should automatically show *Setup Wizard*. In case this does not happen, chances are that your operating system is not configured to accept 'autorun' feature. Either enable 'autorun' in Windows or - from the Explorer, select **Setup.exe** on the CD.

## Installing to the Hard Drive

Even a fast CD-ROM is still much slower than modern hard drives. For this reason, when the CD-ROM is detected when placed into the drive, all files will automatically be installed to your hard drive and a shortcut icon will be placed on the desktop. If you do have to run from the CD, run file **CNC Projects.exe**

## Uninstalling the Software

All installed software can be uninstalled automatically. The suggested procedure is to uninstall from the Windows **Start** menu, by clicking **CNC Projects**, then selecting **Uninstall CNC Projects**. You may also uninstall from the **Control Panel** - select **Add/Remove Programs**, then select **CNC Projects**.

## Main Menu

The one second long welcome screen is the gateway to all other menu selections via the **Main Menu**. All other selections originate from the **Main Menu**. You can return to the **Main Menu** from any displayed page.

## Menu Structure

Menu selections follow established hierarchical menu development format. All files can be accessed with only a few mouse clicks. To help you understand each menu heading, a short explanation and/or information is given when the mouse is positioned over the menu item. Also, in most cases, a preview image will also appear, for additional convenience.

## Why are some menu items greyed out?

The majority of menus are developed along a logical hierarchical structure - the menu for the enclosed CD is no different. Such a structure allows a very efficient method to select the desired menu item with only a few mouse clicks. It also means that menu items have to be very tightly grouped and only so many can be placed on a single menu page. For the projects and exercises based on book chapters, the menu has been designed for twelve menu items maximum on a single menu page.

Ideally, if every one of the 53 chapters had 12 menu items (various programming projects and exercises), each menu would also have the same appearance. It is not necessary for every chapter to have so many exercises, so the problem arises when some chapters have only one or two menu items - the unavailable items would have to be blank and the overall appearance of the menu page would suffer.

Considering that any work dealing with CNC programming projects and exercises can really never be completed, the author has chosen to offer the available menu items to the **CNC Programming Handbook** users for suggestions and contributions. When you click on any dark grey menu item, you will receive a message encouraging you to email the author with a suggestion for another project or an exercise.

Although all chapters in the handbook are of equal importance, not every chapter requires a special project or an exercise. In this *Third Edition*, all major programming subjects are well covered with various programming projects and exercises. In the future editions, more projects may be added, as necessary.

## Opening Files

Any file listed on the menu should open when you click on the heading. Typically, **blue** color is used for listing, which will change to **orange**, when the mouse moves over the heading, then it changes to **green** as a confirmation that the click has been accepted.

## Acrobat Reader - Version 6+ Required

If you have a problem opening a PDF file, check if you have **Acrobat Reader** installed on your computer. If you do, and still have a problem opening files from the menu, check the **Acrobat Reader** version. If it is below version 6, **you must uninstall the old version**, and install the newest version - v8.1.0 is available from the **Utilities** menu. This version will open *all* PDF files that your old version opened, but also the newest files as well. You may also want to disable automatic update dialogue box that may interfere with a displayed page.

A small selection of files are **MS Excel** files (all under one menu), and can only be opened with *Microsoft Excel* installed on your system. All are compatible with MS Excel 2003 and 2007.

All PDF files can be printed directly from Acrobat Reader - they are all formatted to 8.5 x 11 inches and can be printed 100% size, with sufficient margins for A4 paper users and to punch holes for binders. For best results, **uncheck** 'Shrink oversized pages to paper size' in the printer dialogue box of Adobe Acrobat Reader.

## Acrobat Reader Splash Screen

As a default, when Acrobat Reader opens a file, it briefly shows its own splash screen, while opening the selected file. Some users prefer to turn off the splash screen for less clutter on the screen.

Follow this procedure to remove the Acrobat Reader splash screen:

- **Open Acrobat Reader**
- **Select *Edit* from the top menu**
- **Select *Preferences***
- **Scroll down to *Options***
- **From *Options*, select *Startup***
- **Uncheck the box next to *Display Splash Screen***

## Suggestions, Questions and Comments

The Author welcomes suggestions, questions and comments from the users. From the **Main Menu**, you can open the **INDUSTRIAL PRESS WEBSITE** or send email directly to the Author by selecting the item titled as **E-MAIL THE AUTHOR**. Internet connection is required for these two features to work. If you find an error either in the **CNC Programming Handbook** or in the CD-ROM files, feel free to inform the Author, so it can be corrected as soon as possible.

## Special Requirements

No special computer hardware is required, but the operating system must be Windows 98 or higher. The most important setting is the screen resolution. **The minimum setting is 800x600 pixels**, but higher settings will produce crisper appearance of the files, particularly drawings and other graphics. It is **strongly** recommended that you set your screen resolution at least one setting above the minimum, if possible.



In a summary, these three requirements are important:

- **Windows 98 or higher operating system (98SE, ME, NT/2000, XP, Vista)**
- **Acrobat Reader v6 or higher**
- **Screen resolution *minimum* 800x 600, preferably one or more steps higher**

Thank you for using ***CNC Programming Handbook and CNC Projects CD-ROM***.

## **Version History:**

### **Changes and Fixes (up to and including v3.0):**

- **NCPlot toolpath simulation software added (15-day shareware version)**
- **Updated interface and index**
- **Updated utilities**
- **Chapter 47 added**
- **Multipocket drawing - cross section corrected**
- **30-04 and 30-04 solutions enhanced**
- **34-07 and 34-08 solutions updated**
- **Fixed MS Excel file *Bolt Circle Coordinates.xls* (Y-coordinate other than zero now works)**
- **Replaced correct solution file for Project 01-02**
- **Fixed font related issues during installation**
- **Fixed minor bugs and spelling errors**
- **Version numbering changed to match the one on CD**

# Dedication

To my father František and my mother Ludmila,  
who taught me never to give up

# INDEX

---

## Index Terms

## Links

### A

Absolute data input	74	318	446
Acceleration and deceleration	92		
Accumulative error	482		
ACME thread example	375		
Additional axes	18		
Address format	46		
Air cutting	169		
Alpha-numerical input	43		
Americal national thread form	355		
Angle direction	244		
Angular dimensions	35		
APC	453		
Approach to the part	316		
ATC	8	99	
Automatic corner break	309		
Automatic corner override	93		
Automatic pallet change	453		
Automatic tool change	8	64	99
	159		153
ATC cycle	102		
First tool in the spindle	104		
Keeping track of tools	103		
Manual interference	105		
Maximum tool diameter	101		
Maximum tool length	101		
Maximum tool weight	101		
No tool in the spindle	105		
Programming ATC	102		
Several tools	103		
Using a single tool	102		
Using oversize tools	106		
Auxiliary functions lock	28		

## Index Terms

## Links

Axes and planes	16		
Axis inversion	409		
Axis orientation - milling	17		
Axis orientation - turning	18		
<b>B</b>			
Background edit	491		
Ball nose end mill	283		
Barfeeder	174	429	
B-axis	5	445	
Bitwise input	24		
Blend radius	311		
Block	25	44	65
Block format	67		
Block numbering	65	67	
Block numbers increment	68		
Block structure	65		
Conflicting words	70		
End-Of-Block (EOB)	44	68	
Sequence block	44	65	
Significant block	68		
Single block	25		
Status block	69		
Stop code (% symbol)	44		
Block skip	21	167	
Barfeeder application	174		
Numbered block skip	174		
Selective block skip	174		
Slash symbol	167		
Used for program proving	173		
Used for trial cut	172		
Used with modal commands	168		
Block tool	210		
Blueprint	31		
Bolt hole circle pattern	231		
Boring bar deflection	210		

**Index Terms****Links**

## Boring operations

Backboring cycle G87	211
Block tools	211
Boring tools	134
Precision boring cycle G76	211
Single point boring	209
Tool shift	211
Boss milling	250

**C**

CAD/CAM	33	509
CAM software	509	
Complete programming environment	510	
Desktop computer programming	509	
General features	515	
Geometry development	510	
Postprocessors	514	
Support and training	515	
Toolpath generation	510	
Calculation of triangles	503	
Calculations	41	
Calculator type input	80	
Canned cycles	181	324
Cardinal points	244	
Cartesian coordinate system	15	
C-axis	458	
Drilling on diameter	461	
Drilling on face	460	
Fixed cycles	459	
G83-G85 fixed cycles	459	
G87-G89 fixed cycles	459	
M-functions	460	
Z+C-axis	465	
Center cutting end mill	204	287
Centerline tools	133	
Chamfering	309	
Chamfer diameter	197	203
Character	43	



**Index Terms****Links**

Chipload	524	
Chipload control	366	
Chip-to-chip time	99	
Chord of a circle	508	
Chuck functions	425	
External and internal chucking	425	529
Open and close	425	
Chuck jaws	426	
Bored diameter	426	
Chucking functions		
Chuck pressure	426	
Circle cutting cycle	252	
Circular interpolation	243	
Arc center and radius	246	
Arc center vectors	246	
Arc cutting direction	245	
Arc in planes	247	281
Arc programming	253	
Blend radius	248	
Boss milling	250	
Circular motion direction	281	
Elements of a circle	243	
Feedrate for circular motion	253	
Full circle programming	248	252
Lead-in and lead-out path	251	
Partial radius	248	
Programming format	245	
Quadrants	244	
Radius programming	248	
Start point and end point	246	
Circumscribed circle	500	
Clamp and unclamp functions	446	
Climb milling	290	
CNC machine tools	4	
Four-axis lathe	13	
Horizontal boring mill	10	455
Horizontal machining center	10	
Lathe accessories	425	

**Index Terms****Links**

CNC machine tools ( <i>Cont.</i> )		
Lathe axes	11	
Machine axes	8	
Milling	7	
Six-axis lathe	13	
Three-axis lathe	12	
Turning centers and lathes	11	
Two-axis lathe	12	
Types of CNC lathes	11	
Typical specifications	9	13
Vertical machining center	8	
CNC machining	2	483
Alternate machine selection	487	
Machine shutdown	489	
New workpiece	483	
Preventive maintenance	490	
Program integrity	484	
Running the first part	485	
Safety issues	427	488
Setup integrity	484	
Warm-up program	488	
CNC programmer	5	
Conflicting commands	54	
Connection to external devices	30	
Contour change point	40	
Contouring		
Continuous path	40	
Equidistant path	40	
Control system	32	
Control panel	20	
Defaults	23	
Features	14	
Memory capacity	24	
Optional features	29	
Conventional machining	2	
Conventional milling	290	
Conversion of angles	506	

**Index Terms****Links**

Conversions			
HP to kW	40		
kW to HP	40		
Coolant functions	21	62	288
Coordinate geometry	15	17	
Coordinate system rotation	415		
Coordinate words	73		
Corner radius calculation	290		
Counterboring	212	214	
Countersinking	212		
CSS	86	90	315
Custom macros	30	187	
Cutter geometry			
Negative	236		
Positive	236		
Rake angle	236		
Cutter path determination	39		
Cutter radius offset	255	281	
Applying radius compensation	262		
Cutter direction change	265		
D amount setting	272		
Direction of motion	259		
Equidistant tool path	257		
Interference error	268	277	
Look-Ahead type	265		
Offset cancellation	264		
Offset commands G40-G42	260		
Practical example	271		
Programming format	261		
Programming techniques	259		
Startup methods	269		
Tool nose radius offset	275		
Toolpath center points	256		
Types of offset	258		
Cutting mode	94		
Cutting speed	178		
Cutting time	525		
Cutting tool animation	30		

**Index Terms****Links**

Cycle start	21	61	174
Cycle time	4		
Cylindrical interpolation	465		

**D**

Datum shift	397			
Cutter radius offset	404			
Data setting	402			
Lathe offsets	404			
Program zero change	397			
Tool length offset	403			
Using MDI control mode	405			
Using parameters	405			
Work offsets	402			
Decimal point format	78			
Decimal point programming	79			
Defaults	23			
Deflection	286			
Degrees and decimal degrees	506			
Degrees-minutes-seconds	35	506		
Delta increment	258			
Departure position	163			
Depth of cut	186			
Descartes, Rene	15			
Diameter programming	77			
Distance-To-Go	136	180	446	449
D-M-S	35	506		
DNC	494			
Drawing	31	34		
Changes and revisions	36			
Degrees	35			
Dimensioning methods	34			
Fractions	35			
GDT	35			
Special instructions	37			
Surface finish	36			
Title block	34			
Tolerances	36			

**Index Terms****Links**

Drawing dimension input	30		
Drilling operations	200		
Blind holes	203		
Center drilling	202		
Drill point length	201		
Drilling depth	525		
Effective drill diameter	201	214	
Flat bottom drilling	204		
Indexable drills	205		
Multilevel drilling	215		
Nominal drill diameter	201		
Peck drilling	206	222	
Reaming	208	222	
Spot drilling	214	524	
Through holes	203		
Web drilling	216		
Drip-feeding	24	495	
Driven tool	457		
Dry run	21	147	
Dummy tool	99		
Dwell command	175	183	427
As TAB alternative	180		
Dwell in number of revolutions	178		
Dwelling axis	180		
Long dwell time	179		
Minimum dwell	177	214	525
Safety issues	180		
Setting mode	177		
Time selection	176		
Used in fixed cycles	180		
DXF files	33		

**E**

Effective diameter	203		
Emergency stop	21	25	489
Empty tool	99		
End of block	68		
Endless program loop	62		



**Index Terms****Links**

Endpoints	256
Engineering drawing	31
English units	73
EOB	68
Equidistant tool path	257
Errors in programming	479
Calculation errors	482
Hardware errors	482
Input errors	482
Logical errors	481
Miscellaneous errors	482
Syntax errors	481
E-stop	489
E-switch	489
Exact stop check	93
Exact stop check mode	93
Execution priority	72

**F**

Face milling	126	235
Cutter selection	235	
Cutting considerations	236	
Programming techniques	238	
Using position compensation	241	
Facing on lathes	88	
Family part programming	170	
Feedrate control	91	
Chipload	524	
Circular cutting motion	253	
Circular motion feedrate	94	
Constant feedrate	94	
E address in threading	96	
Feedhold	21	25
Feedrate override	21	27
Feedrate per revolution	92	165
Feedrate per time	91	165
Feedrate range	165	
Inverse time feedrate	91	

**Index Terms****Links**Feedrate control (*Cont.*)

Maximum feedrate	95
Selection	92
Tapping feedrate	524

Fillet radius	248
---------------	-----

Fixed cycles	148	181	324
Absolute and incremental applications	185		
Basic concept	182		
Basic format structure	181		
Cancellation of a cycle	194		
Cycle repetition	184	194	
Detailed description	187		
General rules	184		
Initial level selection	185		
L0 parameter	195		
Plane selection	284		
Programming format	183		
R-level selection	186		
Selection	182		
Shift amount	183		
Z-depth calculations	186		
Fixed cycles for C-axis	459		
Floating point	113		
Fly-cutter	238		
Format notation	45		
Milling system format	46		
Turning system format	46		

**G**

G00 command	54	147	
G01 command	54	163	
G02-G03 commands	245		
G04 command	175	455	
G09 command	92		
G10 data setting command	402	405	408
G12-G13 cycle	252	302	
G15 command	233		
G16 command	233		

**Index Terms****Links**

G17-G19 commands	434			
G20 command	56	73		
G21 command	56	73		
G27 command	155	160		
G28 command	148	155	161	
G29 command	148	155	161	
G30 command	148	155	162	
G32 thread cutting command	361			
G40 command	260	275		
G41-G42 commands	260	275		
G43 command	127	136		
G44 command	127	136		
G45-G48 commands	123	126		
G49 command	136			
G50 command	83	117	397	
G50-G51 scaling commands	421			
G52 command	232	400		
G53 machine coordinates command	400			
G54.1 command	128			
G54-G59 work offsets	127	399		
G61 command	92			
G62 command	93			
G63 command	93			
G64 command	93			
G68-G69 commands	415			
G70 cycle	323	330		
G71-G73 cycles	323	325	327	
G72 command	327			
G73 peck drilling cycle	183	188		
G74 tapping cycle - left hand	183	190	196	216
G74-G75 lathe cycles	222	323		
G76 precision boring cycle	183	193	211	
G76 threading cycle	323	364	368	
G80-G89 commands	183			
G81 drilling cycle	187			
G82 spot drilling cycle	188			
G83 peck drilling cycle	188			
G84 tapping cycle - right hand	183	190	196	216

**Index Terms****Links**

G84.2 command	195			
G84.3 command	195			
G85 boring cycle	191			
G86 boring cycle	191			
G87 backboring cycle	192	211		
G88 boring cycle	192			
G89 boring cycle	193			
G90 absolute position command	75	156	164	401
	404			
G90 lathe cycle	318			
G91 incremental motion command	75	156	164	397
G92 position register command	117	128	397	399
G92 threading cycle	362	378		
G94 feedrate command	92			
G94 lathe cycle	322			
G95 feedrate command	92			
G96 command	315			
G97 command	88	315		
G98-G99 commands	185	215		
Gauge line	113	135		
G-codes	51			
Conflicting commands	54			
Group numbers	55			
Milling applications	51			
Modal commands	54			
Turning applications	53			
Types of G codes	56			
Using in a block	54			
With decimal point	56			
Gear ranges	21	64	308	
Geometric Dimensioning and Tolerancing (GDT)	35			
Geometry	498			
Geometry offset	108	132	161	262
Gouging	269			
Graphic display	29			
Groove recess	370			

## **Index Terms**

## **Links**

Grooving and part-off	333
Corner groove	342
Grooving applications	333
Grooving dimensions	334
Multiple grooves	340
Necking	333
Part-off	345
Precision groove	336
Relief grooving	333
Simple groove	335

## **H**

Handle	21		
Headstock	81		
Helical milling	433		
Helical interpolation	433		
Helix	435		
Ramping	444		
Thread milling	434		
Thread milling example	437		
Hockey stick motion	150	155	221
Home position	112	153	
Horizontal machining centers	131	445	

## **I**

IGES files	33			
Incremental data input	74	76	318	446
Indexing axis	446	450		
Indexing table	445			
Initial level selection	185			
In-process gauging	30			
Input of dimensions	73	445		
Input format	77			
Zero suppression	78			
Inscribed circle	500			
Interfacing to devices	491			
Connecting cables	496			
Data setting	495			



## **Index Terms**

## **Links**

### Interfacing to devices (*Cont.*)

DNC	494		
Punched tape	492		
RS-232C interface	491	493	495
Terminology of communications	495		
Intermediate point	155		
Interrupted cut	237		
Inverse time feedrate	91		

## **J**

John Parsons	1		
--------------	---	--	--

## **K**

Keyboard	21		
Keyway	291		

## **L**

Label skip function	493		
L-address	184	194	
Lathe cycles	317		
Contour finishing cycle	330		
Face cutting cycle	322		
Multiple repetitive cycles	323		
P and Q blocks	324	326	
Pattern repeating cycle	328		
Principle of cycles	317		
Profile cutting cycle	324		
Straight cutting cycle	318		
Lathe offsets	304		
Master tool setup	133		
Multiple offsets	305		
Offset change	305		
Offset entry	304	308	
Offsets and tool motion	305		
Lathe program format	316		
Law of similar triangles	320		
Lead-in	251		

**Index Terms****Links**

Lead-in motion	263	269	277	293
Lead-out	251			
Lead-out motion	264	277	293	
Least increment	77			
Linear interpolation	163	165	281	
Cutting feedrate	165			
Multiaxis motion	164			
Programming format	164			
Single axis motion	163			
Start and end of motion	163			
Live tooling	5	457		
C-axis	458			
Cylindrical interpolation	465			
Fixed cycles	459			
Polar coordinate interpolation	463			
Y-axis	462			
Live tooling on CNC lathes	457			
Local coordinate system	232	399		
Look-ahead radius offset	265			

**M**

M00 function	60	204		
M01 function	61	204		
M02 program end function	62			
M03 function	63	83		
M04 function	63	83		
M05 function	63	84		
M06 function	64	99		
M07 function	62			
M08 function	62			
M09 function	62			
M10-M11 functions	425			
M12-M13 functions	427			
M15-M16 functions	426			
M17-M18 functions	428			
M19 function	64	84	106	
M21-M22 functions	427			
M23-M24 thread finishing functions	64	370		

**Index Terms****Links**

M29 function	196			
M30 program end function	62			
M41-M44 functions	308			
M48-M49 functions	96	224		
M60 function	64	455		
M71-M72 function	429			
M73-M74 functions	430			
M88-M89 functions	431			
M98-M99 subprogram functions	62	64	384	391
	488			
Machinability rating	37	85		
Machine accessories	64			
Machine coordinate system	395	400		
Machine geometry	17			
Machine lock	28			
Machine warm-up	180			
Machine zero	112			
Absolute and incremental mode	156			
Intermediate point	155			
Machine zero return	153			
Primary machine zero	155			
Program commands	155			
Return from machine zero	161			
Secondary machine zero	162			
Zero position check	160			
Machine zero return	153	450		
Machining a workpiece	2			
Machining cycles	30			
Machining holes	197			
Drilling	200			
Hole enlargement	212			
Lathe applications	220			
Peck drilling	206	222		
Reaming	208	222		
Single point boring	209			
Tapping	216			
Tooling selection	197	199		
Web drilling	216			

**Index Terms****Links**

Machining sequence	38			
Macros	405			
Manual absolute	28			
Manual data input	26	58	83	405
Manual programming	32			
Massachusetts Institute of Technology	1			
Material specifications	31	37		
Math in programming	320			
Calculations of triangles	503			
Circle elements	498			
Order of calculations	497			
M-codes	57			
Machine related	57	62		
MDI functions	58			
M-function duration	60			
Milling applications	58			
Program related	57	60		
Turning applications	58			
MDI	21	26	58	83
	405			
Memory capacity	24			
Memory card	32			
Messages in a program	70	478		
Methods sheet	37			
Metric units	73			
Milling - G-codes	51			
Milling - M-codes	58			
Milling operations				
Contour milling	285	287	289	
Direction of cut	289			
End mills	285			
Peripheral milling	285			
Slots and pockets	291			
Speeds and feeds	287			
Stock removal	289			
Width and depth of cut	290			
Mill-turn	5			
Minimum axis increment	77			

**Index Terms****Links**

Mirror image	409			
Control setting	411			
Lathe applications	414			
Programmable setting	412			
Programming functions	412			
Miscellaneous functions	13	57		
Application in a block	59			
Description	57			
Machine related functions	57	62		
Program related functions	57	60		
Special MDI functions	58			
Modal commands	54	71	73	75
M-S-T lock	28			
Multilevel drilling	215			
Multiple repetitive cycles	317	363	366	

**N**

NCPlot toolpath simulation	468	479	527	
Numerical control	1			
Advantages	2			
Definition	1			
Hardwired controls	1			
Softwired controls	1			

**O**

Offsets	83	308	325	
Operation panel	20			
Optional stop	21	61		
Orthographic orientation	415			
Overtravel	112	125	154	

**P**

Pallet changer	445	453		
Pallet types	454			
Parameters	22	405	407	
Parsons, John	1			
Part catcher or unloader	430			



**Index Terms****Links**

Part complexity	32	
Part reference point	113	
Part setup	39	
Setup sheet	39	
Part unloader	430	
Part-off	345	
Parts counter	431	
Pattern of holes	225	
Angular row hole pattern	226	
Arc hole pattern	230	
Bolt hole circle pattern	231	
Corner pattern	228	
Grid pattern	228	
Random hole pattern	225	
Straight row hole pattern	226	
Typical patterns	225	
Peck drilling	206	222
Percent sign	44	62
Percent symbol (%)	44	
Peripheral spindle speed	85	
PI constant	499	
Pivot point	233	
Planes	279	
Absence of axis data	282	
Circular interpolation in planes	281	
Cutter radius offset	283	
Definition	16	279
Fixed cycles	284	
Machining in planes	279	
Mathematical planes	279	
Selection of planes	234	279
Plunge-type end mill	287	289
Pocket milling	291	
Circular pocket cycles	302	
Circular pockets	299	
Rectangular pockets	295	
Point of origin	16	
Point-to-point machining	181	

**Index Terms****Links**

Polar coordinate interpolation	463			
Polar coordinate system	233			
Position compensation	123			
Incremental mode	124			
Motion length calculation	124			
Programming commands	123			
Z-axis	126			
Position register commands	117			
Definition	117			
Lathes	119			
Milling	117			
Turning	117			
Z-axis setting	119			
Positive bias	48			
Post processing	33	514		
Power rating	40			
Preparatory commands	51			
Process sheet	31	37		
Program	44			
Program changes	486			
Program comments	70			
Program end	62			
Program header	48			
Program length reduction	471			
Program structure	48			
Program documents	473			
Documentation file folder	477			
Documenting a program	474			
Setup sheet	475			
Tooling sheet	475			
Program flow	49			
Program header	50			
Program identification	66			
Program name	66			
Program number	66			
Program planning	31	37	39	41
Program stop	60			

**Index Terms****Links**

Program verification	479		
Avoidance of errors	480		
Corrective measures	479		
Detection of errors	479		
Graphic method	480		
Input errors	482		
Preventive measures	479		
Thread cutting	375		
Program writing	467		
Confusing characters	468		
Long programs	471		
Output formatting	469		
Programming forms	468		
Program zero	16	113	283
Lathes	116		
Machining centers	114		
Selection methods	113		
Programmable quill	427		
Programmable tailstock	427		
Programming formats	44		
Format notation	45		
Word address format	44		
Programming procedure	33		
Programming terms	43		
Pull-Out finger	431		
Punched tape	492		
Pythagorean Theorem	165	506	

**Q**

Quadrants	16	244	
Quality in CNC programming	42		
Quill	427		

**R**

Radius programming	248		
Rapid positioning	147	281	304
Approach to the part	152		
Hockey stick motion	150	155	

**Index Terms****Links**Rapid positioning (*Cont.*)

Motion formulas	151			
Reduction of rapid rate	26	151		
Tool path motion	148			
Ratio of sides	506			
Real number system	15			
Reaming operations	208	222		
Recess programming	314			
Rectangular coordinate system	15			
Rectangular coordinates	15			
Reference points	111	497	507	
Fixed point	112			
Flexible point	112			
Machine zero	111	153	497	507
Machine zero (home)	112			
Part reference point	113			
Reference point groups	111			
Tool reference point	116	374		
Register commands	117			
Rene Descartes	15			
Return to machine zero	113			
Rigid tapping	195	217	220	
R-level selection	183	186		
Rotary tool	457			
Roughing and finishing	312	317		
Routing sheet	31	37		
RS-232C interface	30	491	494	

**S**

Safe block	69
Safety in CNC work	6
Scaling function	421
G50-G51 commands	421
Scaling center	422
Scaling factor	422
Screen display	21
Sequence block	65
Sequence return	28

**Index Terms****Links**

SETTINGS screen	174			
Setup sheet	39	475		
Shallow counterboring	214			
Significant digits	78			
Similar triangles	504			
Single block	21	25		
Single point threading				
Spindle synchronization	351			
Slash symbol	167			
Slot drill	204	287	293	
Slots and pockets	291			
Speeds and feeds	287	315	357	359
Speeds and feeds charts	523			
Spherical end mill	283			
Spindle control	81			
Basic formulas	524			
Constant surface speed	86			
Empty spindle	105			
Maximum spindle speed setting	88			
Spindle rotation	81			
Spindle speed - CSS	315	523		
Spindle speed - RPM	85	287	523	
Startup	83			
Spindle functions	21	63	81	
Spindle orientation	21	64	84	210
Spindle speed	178	523		
Spindle speed formulas	86			
Imperial version	38			
Metric version	38			
Spindle speed override	21	27		
Spindle stop	84			
Spindle synchronization	351			
Spot drill	178			
Spot drilling	203	214		
Spot drill depth	214	524		
Spotfacing	212	214		
Start point for threading	352			
Status block	69			



**Index Terms****Links**

Stock allowance	209	313
Stock removal	317	
Stop code (% symbol)	44	
Stored stroke limits	30	
Subprograms	383	
100 000 000 hole grid	395	
Development strategies	389	
Grooving applications	344	
Identification	384	
M98-M99 functions	384	
Multi-level nesting	393	
Numbering methods	387	
Protected subprograms	389	
Repetition	385	
Subprogram call	384	386
Subprogram end	62	384
Using B-axis	450	
Surface speed	523	
Symbols in programming	47	
System of coordinates	15	

**T**

Tailstock functions	426	
Body, quill and center	426	
Programmable quill	427	
Programmable tailstock	427	
Tangent of a circle	508	
Taper programming	319	321
Tapers	376	501
Tapping	216	
Check list	219	
Pipe taps	219	
Speeds and feeds	218	524
Tap chamfer geometry	218	
Tap flute geometry	217	
Tap geometry	217	
Tapping mode	93	
Tapping on lathes	223	

**Index Terms****Links**Tapping (*Cont.*)

Tension-Compression tap holder	217			
Target position	40	163		
Technical print	31			
Technological decisions	39			
Thread cutting tool	351			
Thread milling	434			
Example	437			
Simulation method	443			
Tapered thread	442			
Thread recutting	351			
Threading	349	353	355	373
	375	381		
ACME thread example	375			
Block-by-block threading	361			
Box threading cycle (G92)	362			
Chip load (chipload)	353			
Chipload control	366			
Depth of thread	353	375		
Forms of thread	349			
G76 cycle	363	366		
Hand of thread	371			
Imperfect threads	353			
Individual threading motions	351			
Infeed calculations	368			
Infeed methods	366			
ISO (M) metric thread form	354			
Lead error	360			
Maximum feedrate	359			
Multistart threads	379			
Number of threading passes	353			
Obstacles for start point	353			
One-block vs. two-block G76 cycle	365			
Pitch vs. lead	358			
Program testing	375			
Retract from thread	370			
Single point threading	349			
Speeds and feeds	357	359	380	

**Index Terms****Links**Threading (*Cont.*)

Spring pass	353
Start point	352
Tapered thread	376
Terminology	350
Thread depth constants	356
Thread forms	375
Thread recutting	382
Threading process	351
Threading to a shoulder	373
Tool wear	381
Tooling reference	361
UN (Imperial) thread form	354

## Tolerances

Implied tolerances	35
--------------------	----

Tool comments	50
---------------	----

Tool function	97
---------------	----

Chip-to-chip time	99
-------------------	----

Lathes	107
--------	-----

Machining centers	97
-------------------	----

Tool indexing - lathes	107
------------------------	-----

Tool length offset	130	135	403
--------------------	-----	-----	-----

Cancellation	145
--------------	-----

Change of offset	144
------------------	-----

Datum shift	403
-------------	-----

Distance-To-Go calculation	137
----------------------------	-----

G43-G44 difference	140
--------------------	-----

Horizontal application	145	448
------------------------	-----	-----

Offset commands	136
-----------------	-----

Used with G54-59	143
------------------	-----

Used with G92	142
---------------	-----

Tool length setup	137
-------------------	-----

Off-machine	138	448
-------------	-----	-----

Offset register	138
-----------------	-----

On-machine	137	448
------------	-----	-----

Preset tool length	139
--------------------	-----

Using master tool	140
-------------------	-----

Tool magazine	8	97
---------------	---	----

**Index Terms****Links**

Tool memory type		
Fixed type	98	
Random memory type	98	
Tool nose radius offset	275	
Change of motion direction	278	
Radius offset commands	275	
R-setting	110	
Tool tip orientation	275	
T-setting	110	
Tool number registration	98	
Tool offset registers	108	
Geometry offset	108	
Wear offset	109	
Tool position setting	118	
Tool setup groups	120	
Center line tooling	120	
External tooling	121	
Internal tooling	121	
Tooling selection	38	
Tooling sheet	39	475
Trial cut	172	
Trigonometry	503	
Turning - G-codes	53	
Turning - M-codes	58	
Turning and boring	303	
Turning tools	134	303
Turn-mill	5	
Turn-Mill	457	
Turret indexing	428	

**U**

U/W commands	164
Undercut	370
Undercut programming	314
Units conversion	73
USB	491
User macros	405

**Index Terms****Links****W**

W-axis	10		
Wear offset	109	133	262
Adjustment	110		
Web drilling	216		
Word address format	44		
Multiple addresses	47		
Order of words in block	55		
Word	43		
Work area	31		
Work coordinate system	127		
Work offsets	127	402	
Additional work offsets	128	403	
Common offset (COM)	132		
Datum shift	403		
Defaults	128	402	
External offset (EXT)	132	403	
G54-G59 commands	127	402	
Horizontal machine application	131	447	
Lathe applications	132		
Startup	128		
Work areas available	127		
Work offset change	129		
Z-axis application	130		
Work sketch	41		

**Z**

Z-axis neglect	28		
----------------	----	--	--





# REFERENCE TABLES

## Decimal Equivalents

The following chart lists fractional, wire gauge (number), letter and metric (mm) values for given decimal equivalents in inches.

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.0059		97	0.15
0.0063		96	0.16
0.0067		95	0.17
0.0071		94	0.18
0.0075		93	0.19
0.0079		92	0.20
0.0083		91	0.21
0.0087		90	0.22
0.0091		89	0.23
0.0095		88	0.24
0.0100		87	0.25
0.0105		86	
0.0110		85	0.28
0.0115		84	
0.0118			0.30
0.0120		83	
0.0125		82	
0.0126			0.32
0.0130		81	
0.0135		80	
0.0138			0.35
0.0145		79	
0.0150			0.38
0.0156	1/64		
0.0157			0.40
0.0160		78	
0.0177			0.45
0.0180		77	
0.0197			0.50
0.0200		76	
0.0210		75	
0.0217			0.55
0.0225		74	
0.0236			0.60
0.0240		73	
0.0250		72	
0.0256			0.65
0.0260		71	
0.0276			0.70
0.0280		70	
0.0292		69	
0.0295			0.75
0.0310		68	
0.0313	1/32		
0.0315			0.80
0.0320		67	
0.0330		66	
0.0335			0.85

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.0350		65	
0.0354			0.90
0.0360		64	
0.0370		63	
0.0374			0.95
0.0380		62	
0.0390		61	
0.0394			1.00
0.0400		60	
0.0410		59	
0.0413			1.05
0.0420		58	
0.0430		57	
0.0433			1.10
0.0453			1.15
0.0465		56	
0.0469	3/64		
0.0472			1.20
0.0492			1.25
0.0512			1.30
0.0520		55	
0.0531			1.35
0.0550		54	
0.0551			1.40
0.0571			1.45
0.0591			1.50
0.0595		53	
0.0610			1.55
0.0625	1/16		
0.0630			1.60
0.0635		52	
0.0650			1.65
0.0669			1.70
0.0670		51	
0.0689			1.75
0.0700		50	
0.0709			1.80
0.0728			1.85
0.0730		49	
0.0748			1.90
0.0760		48	
0.0768			1.95
0.0781	5/64		
0.0785		47	
0.0787			2.00
0.0807			2.05
0.0810		46	
0.0820		45	
0.0827			2.10
0.0846			2.15
0.0860		44	
0.0866			2.20
0.0886			2.25

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.0890		43	
0.0906			2.30
0.0925			2.35
0.0935		42	
0.0938	3/32		
0.0945			2.40
0.0960		41	
0.0965			2.45
0.0980		40	
0.0984			2.50
0.0995		39	
0.1015		38	
0.1024			2.60
0.1040		37	
0.1063			2.70
0.1065		36	
0.1083			2.75
0.1094	7/64		
0.1100		35	
0.1102			2.80
0.1110		34	
0.1130		33	
0.1142			2.90
0.1160		32	
0.1181			3.00
0.1200		31	
0.1220			3.10
0.1250	1/8		
0.1260			3.20
0.1280			3.25
0.1285		30	
0.1299			3.30
0.1339			3.40
0.1360		29	
0.1378			3.50
0.1405		28	
0.1406	9/64		
0.1417			3.60
0.1440		27	
0.1457			3.70
0.1470		26	
0.1476			3.75
0.1495		25	
0.1496			3.80
0.1520		24	
0.1535			3.90
0.1540		23	
0.1562	5/32		
0.1570		22	
0.1575			4.00
0.1590		21	
0.1610		20	
0.1614			4.10
0.1654			4.20
0.1660		19	
0.1673			4.25
0.1693			4.30
0.1695		18	
0.1719	11/64		
0.1730		17	
0.1732			4.40
0.1770		16	
0.1772			4.50

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.1800		15	
0.1811			4.60
0.1820		14	
0.1850		13	
0.1870			4.75
0.1875	3/16		
0.1890		12	
0.1910		11	
0.1929			4.90
0.1935		10	
0.1960		9	
0.1969			5.00
0.1990		8	
0.2008			5.10
0.2010		7	
0.2031	13/64		
0.2040		6	
0.2047			5.20
0.2055		5	
0.2067			5.25
0.2087			5.30
0.2090		4	
0.2126			5.40
0.2130		3	
0.2165			5.50
0.2188	7/32		
0.2205			5.60
0.2210		2	
0.2244			5.70
0.2264			5.75
0.2280		1	
0.2283			5.80
0.2323			5.90
0.2340		A	
0.2344	15/64		
0.2362			6.00
0.2380		B	
0.2402			6.10
0.2420		C	
0.2441			6.20
0.2460		D	
0.2461			6.25
0.2480			6.30
0.2500	1/4	E	
0.2520			6.40
0.2559			6.50
0.2570		F	
0.2598			6.60
0.2610		G	
0.2638			6.70
0.2556	17/64		
0.2657			6.75
0.2660		H	
0.2677			6.80
0.2717			6.90
0.2720		I	
0.2756			7.00
0.2770		J	
0.2795			7.10
0.2810		K	
0.2812	9/32		
0.2835			7.20
0.2854			7.25

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.2874			7.30
0.2900		L	
0.2913			7.40
0.2950		M	
0.2953			7.50
0.2969	19/64		
0.2992			7.60
0.3020		N	
0.3031			7.70
0.3051			7.75
0.3071			7.80
0.3110			7.90
0.3125	5/16		
0.3150			8.00
0.3160		O	
0.3189			8.10
0.3228			8.20
0.3230		P	
0.3248			8.25
0.3268			8.30
0.3281	21/64		
0.3307			8.40
0.3320		Q	
0.3346			8.50
0.3386			8.60
0.3390		R	
0.3425			8.70
0.3438	11/32		
0.3445			8.75
0.3465			8.80
0.3480		S	
0.3504			8.90
0.3543			9.00
0.3580		T	
0.3583			9.10
0.3594	23/64		
0.3622			9.20
0.3642			9.25
0.3661			9.30
0.3680		U	
0.3701			9.40
0.3740			9.50
0.3750	3/8		
0.3770		V	
0.3780			9.60
0.3819			9.70
0.3839			9.75
0.3858			9.80
0.3860		W	
0.3898			9.90
0.3906	25/64		
0.3937			10.00
0.3970		X	
0.4040		Y	
0.4062	13/32		
0.4130		Z	
0.4134			10.50
0.4219	27/64		
0.4331			11.00
0.4375	7/16		
0.4528			11.50
0.4531	29/64		
0.4688	15/32		

Decimal inch	Fraction	Number / Letter	Metric (mm)
0.4724			12.00
0.4844	31/64		
0.4921			12.50
0.5000	1/2		12.70
0.5118			13.00
0.5156	33/64		
0.5312	17/32		
0.5315			13.50
0.5469	35/64		
0.5512			14.00
0.5625	9/16		
0.5709			14.50
0.5781	37/64		
0.5906			15.00
0.5938	19/32		
0.6094	39/64		
0.6102			15.50
0.6250	5/8		
0.6299			16.00
0.6406	41/64		
0.6496			16.50
0.6562	21/32		
0.6693			17.00
0.6719	43/64		
0.6875	11/16		
0.6890			17.50
0.7031	45/64		
0.7087			18.00
0.7188	23/32		
0.7283			18.50
0.7344	47/64		
0.7480			19.00
0.7500	3/4		
0.7656	49/64		
0.7677			19.50
0.7812	25/32		
0.7874			20.00
0.7969	51/64		
0.8071			20.50
0.8125	13/16		
0.8268			21.00
0.8281	53/64		
0.8438	27/32		
0.8465			21.50
0.8594	55/64		
0.8661			22.00
0.8750	7/8		
0.8858			22.50
0.8906	57/64		
0.9055			23.00
0.9062	29/32		
0.9219	59/64		
0.9252			23.50
0.9375	15/16		
0.9449			24.00
0.9531	61/64		
0.9646			24.50
0.9688	31/32		
0.9843			25.00
0.9844	63/64		
1.0000	1		25.40

All tap drill sizes in the following tables are based on the approximate full thread depth of 72-77% of nominal.

## Imperial Threads - UNC/UNF

Thread-TPI	Tap Drill Size	Inch equivalent	Metric alternative
#0-80	3/64	0.0469	
1/16-64	3/64	0.0469	
#1-64	#53	0.0595	
#1-72	#53	0.0595	
#2-56	#50	0.0700	
#2-64		0.0709	1.80
3/32-48	#49	0.0730	
#3-48	#47	0.0785	
#3-56	#45	0.0820	
#4-32	#45	0.0820	
#4-36	#44	0.0860	
#4-40	#43	0.0890	
#4-48	#42	0.0935	
#5-40	#39	0.0995	
#5-44	#37	0.1040	
1/8-40	#38	0.1015	
#6-32	#36	0.1065	
#6-36	#34	0.1110	
#6-40	#33	0.1130	
5/32-32	1/8	0.1250	
5/32-36	#30	0.1285	
#8-32	#29	0.1360	
#8-36	#29	0.1360	
#8-40	#28	0.1405	
3/16-24	#26	0.1470	
3/16-32	#22	0.1570	
#10-24	#25	0.1495	
#10-28	#23	0.1540	
#10-30	#22	0.1570	
#10-32	#21	0.1590	
#12-24	#16	0.1770	
#12-28	#14	0.1820	
#12-32	#13	0.1850	4.70
7/32-24	#16	0.1770	
7/32-32	#12	0.1890	4.80
#14-20	#10	0.1935	
#14-24	#7	0.2010	5.10
1/4-20	#7	0.2010	5.10
1/4-28	#4	0.2090	
1/4-32	7/32	0.2188	5.50
5/16-18	F	0.2570	6.50
5/16-20	17/64	0.2656	
5/16-24	I	0.2720	6.90
5/16-32	9/32	0.2813	7.10
3/8-16	5/16	0.3125	8.00
3/8-20	21/64	0.3281	
3/8-24	Q	0.3320	8.50
3/8-32	11/32	0.3438	
7/16-14	U	0.3680	9.40
7/16-20	25/64	0.3906	9.90
7/16-24	X	0.3970	10.00
7/16-28	Y	0.4040	
½-13	27/64	0.4219	
½-20	29/64	0.4531	11.50
½-28	15/32	0.4688	
9/16-12	31/64	0.4844	
9/16-18	33/64	0.5156	13.00

Thread-TPI	Tap Drill Size	Inch equivalent	Metric alternative
9/16-24	33/64	0.5156	13.00
5/8-11	17/32	0.5313	13.50
5/8-12	35/64	0.5469	
5/8-18	37/64	0.5781	
5/8-24	37/64	0.5781	
11/16-12	39/64	0.6094	
11/16-16	5/8	0.6250	
11/16-24	41/64	0.6406	
3/4-10	21/32	0.6563	16.50
3/4-12	43/64	0.6719	17.00
3/4-16	11/16	0.6875	17.50
3/4-20	45/64	0.7031	17.50
3/4-28	23/32	0.7188	
13/16-12	47/64	0.7344	
13/16-16	3/4	0.7500	
7/8-9	49/64	0.7656	19.50
7/8-12	51/64	0.7969	20.00
7/8-14	13/16	0.8125	
7/8-16	13/16	0.8125	
7/8-20	53/64	0.8281	
15/16-12	55/64	0.8594	
15/16-16	7/8	0.8750	
15/16-20	57/64	0.8906	
1-8	7/8	0.8750	
1-12	59/64	0.9219	
1-14	15/16	0.9375	
1-20	61/64	0.9531	
1 1/16-12	63/64	0.9844	
1 1/16-16	1.0	1.0000	

## Straight Pipe Taps NPS

Tap Size	Tap Drill Ø (in.)	Metric alternative
1/8-27	S	8.80
1/4-18	29/64	11.50
3/8-18	19/32	15.00
½-14	47/64	18.50
3/4-14	15/16	23.75
1-11 ½	1 3/16	30.25
1 1/4-11 ½	1 33/64	38.50
1 ½-11 ½	1 3/4	44.50
2-11 ½	2 7/32	56.00

Pipe Size	TPI	Tap Drill	Decimal Size
1/16	27	1/4	0.2500
1/8	27	11/32	0.3438
1/4	18	7/16	0.4375
3/8	18	37/64	0.5781
½	14	23/32	0.7188
3/4	14	59/64	0.9219
1.0	11-1/2	1-5/32	1.1563
1-1/4	11-1/2	1-1/2	1.5000
1-1/2	11-1/2	1-3/4	1.7500
2.0	11-1/2	2-7/32	2.2188

## Taper Pipe Taps NPT

Tap Size	Tap Drill Ø (in.)	Metric alternative
1/16-27	D	6.30
1/8-27	R	8.70
1/4-18	7/16	11.10
3/8-18	37/64	14.50
1/2-14	45/64	18.00
3/4-14	59/64	23.25
1-11 1/2	1 5/32	29.00
1 1/4-11 1/2	1 1/2	38.00
1 1/2-11 1/2	1 47/64	44.00
2-11 1/2	2 7/32	56.00
2 1/2-8	2 5/8	67.00
3-8	3 1/4	82.50

Pipe Size	TPI	Drilled Only		Taper Reamed	
		Tap Drill	Dec. Size	Tap Drill	Dec. Size
1/16	27	D	0.2460	15/64	0.2344
1/8	27	Q	0.3320	21/64	0.3281
1/4	18	7/16	0.4375	27/64	0.4219
3/8	18	37/64	0.5781	9/16	0.5625
1/2	14	45/64	0.7031	11/16	0.6875
3/4	14	29/32	0.9062	57/64	0.8906
1.0	11-1/2	1-9/64	1.1406	1-1/8	1.1250
1-1/4	11-1/2	1-31/64	1.4844	1-15/32	1.4688
1-1/2	11-1/2	1-47/64	1.7344	1-23/32	1.7188
2.0	11-1/2	2-13/64	2.2031	2-3/16	2.1875

## Metric Coarse Threads

Nominal Ø x Pitch (mm)	Tap Drill Ø (mm)	Inch equivalent
M1 x 0.25	0.75	0.0295
M1.2 x 0.25	0.95	0.0374
M1.4 x 0.3	1.10	0.0433
M1.5 x 0.35	1.15	0.0453
M1.6 x 0.35	1.25	0.0492
M1.8 x 0.35	1.45	0.0571
M2 x 0.4	1.60	0.0630
M2.2 x 0.45	1.75	0.0689
M2.5 x 0.45	2.05	0.0807
M3 x 0.5	2.50	0.0984
M3.5 x 0.6	2.90	0.1142
M4 x 0.7	3.30	0.1299
M4.5 x 0.75	3.75	0.1476
M5 x 0.8	4.20	0.1654
M6 x 1	5.00	0.1969
M7 x 1	6.00	0.2362
M8 x 1.25	6.75	0.2657
M9 x 1.25	7.75	0.3051
M10 x 1.5	8.50	0.3346
M11 x 1.5	9.50	0.3740
M12 x 1.75	10.20	0.3937
M14 x 2	12.00	0.4724
M16 x 2	14.00	0.5512

Nominal Ø x Pitch (mm)	Tap Drill Ø (mm)	Inch equivalent
M18 x 2.5	15.50	0.6102
M20 x 2.5	17.50	0.6890
M22 x 2.5	19.50	0.7677
M24 x 3	21.00	0.8268
M27 x 3	24.00	0.9449
M30 x 3.5	26.50	1.0433

## Metric Fine Threads

Nominal Ø x Pitch (mm)	Tap Drill Ø (mm)	Inch equivalent
M3 x 0.35	2.65	0.1043
M3.5 x 0.35	3.15	0.2283
M4 x 0.5	3.50	0.1378
M4.5 x 0.5	4.00	0.1575
M5 x 0.5	4.50	0.1772
M5.5 x 0.5	5.00	0.1969
M6 x 0.75	5.25	0.2067
M7 x 0.75	6.25	0.2461
M8 x 1	7.00	0.2756
M9 x 1	8.00	0.3150
M10 x 0.75	9.25	0.3642
M10 x 1	9.00	0.3543
M10 x 1.25	8.75	0.3445
M11 x 1	10.00	0.3937
M12 x 1	11.00	0.4331
M12 x 1.25	10.75	0.4232
M12 x 1.5	10.50	0.4134
M13 x 1.5	11.50	0.4528
M13 x 1.75	11.25	0.4429
M14 x 1.25	12.75	0.5020
M14 x 1.5	12.50	0.4921
M15 x 1.5	13.50	0.5315
M16 x 1	15.00	0.5906
M16 x 1.5	14.50	0.5709
M17 x 1.5	15.50	0.6102
M18 x 1	17.00	0.6693
M18 x 1.5	16.50	0.6496
M18 x 2	16.00	0.6299
M20 x 1	19.00	0.7480
M20 x 1.5	18.50	0.7283
M20 x 2	18.00	0.7087
M22 x 1	21.00	0.8268
M22 x 1.5	20.50	0.8071
M22 x 2	20.00	0.7874
M24 x 1	23.00	0.9055
M24 x 1.5	22.50	0.8858
M24 x 2	22.00	0.8661
M25 x 1.5	23.50	0.9252
M27 x 2	25.00	0.9843
M28 x 2	26.00	1.0236
M30 x 2	28.00	1.1024
M30 x 3	27.00	1.0630



Notes :
---------

## B

## SPEEDS AND FEEDS

In machines shops, the words '*speeds and feeds*' are often associated with descriptions of *any* cutting conditions required to machine a part. This is not a correct approach to use a single description for a large number of possibilities, but there is a certain reason for it. In CNC work, speeds and feeds play such a dominant role in program development, that they often overshadow other - *equally important* - machining decisions. In addition to speeds and feeds, CNC programmer has to consider cutting tool and its capabilities, such as depth and width of cut, method of entry into the part, machining power and forces, etc. Machine manuals, tooling catalogues, text books, charts, internet, and many other resources, offer extensive range of speeds and feeds for various tools and materials, and how they work together. We can benefit from all of them.

The first thing to keep in mind is that there is no recipe, no one method or a single formula, that would point towards the one and only possible spindle speed or cutting feedrate for a particular job. *It is just not possible*. Speeds and feeds are a small, but important, part of a much larger subject - the *total setup*. Speeds and feeds are provided as a *range* of selections, from which the programmer selects the

one most suitable. Large ranges do not offer much of a solution that novice programmers are seeking. Experienced programmers also use their own experience, based on successfully completed programs.

## SURFACE SPEED AND SPINDLE SPEED

Depending on the units of measurement used for a job, programmed spindle speed is typically based on *surface speed* for a given tool material as well as material being machined. Metric units use *meters per minute (m/min)* - imperial units use *surface feet per minute (ft/min)*. These base values are used to calculate spindle speed in *revolutions per minute (r/min)*. Surface speed is an industry rating that is the result of extensive research and can be found in tooling catalogues, for example. Surface speed does *not* consider any active tool diameter. For roughing and finishing, CNC lathes use surface speed directly, for CNC machining centers, spindle revolutions have to be calculated. Two simple tables are presented here to show basic differences between machinability specifications of different materials - one is in metric, the other in imperial units:

Metric - m/min	Drilling		Tapping	Milling - Rough		Milling - Finish		Turning	
Material	HSS	Carbide	HSS	HSS	Carbide	HSS	Carbide	Carbide Rough	Carbide Finish
Aluminum	55 - 60	90 - 115	25 - 35	60 - 90	135 - 180	75 - 105	120 - 240	120 - 240	150 - 270
Brass - Soft	40 - 45	75 - 100	20 - 30	45 - 60	60 - 75	60 - 90	90 - 210	105 - 210	135 - 260
Cast Iron - Medium	12 - 18	35 - 45	8 - 15	15 - 25	55 - 75	25 - 30	60 - 90	60 - 90	75 - 135
Steel - Mild	10 - 15	18 - 27	10 - 20	18 - 30	45 - 60	23 - 40	75 - 105	75 - 105	100 - 145
Stainless Steel (Free machining)	8 - 18	25 - 37	5 - 10	10 - 15	55 - 67	12 - 25	60 - 100	75 - 120	85 - 150

Imperial - ft/min	Drilling		Tapping	Milling - Rough		Milling - Finish		Turning	
Material	HSS	Carbide	HSS	HSS	Carbide	HSS	Carbide	Carbide Rough	Carbide Finish
Aluminum	175 - 200	300 - 375	80 - 110	200 - 300	450 - 600	250 - 350	400 - 800	400 - 800	500 - 900
Brass - Soft	125 - 150	250 - 325	70 - 100	150 - 200	200 - 250	200 - 300	300 - 700	350 - 700	450 - 850
Cast Iron - Medium	40 - 60	115 - 150	25 - 50	50 - 80	175 - 250	80 - 100	200 - 300	200 - 300	250 - 450
Steel - Mild	30 - 50	60 - 90	30 - 70	60 - 100	150 - 200	75 - 125	250 - 350	250 - 350	325 - 475
Stainless Steel (Free machining)	25 - 60	80 - 120	15 - 30	30 - 50	180 - 220	40 - 80	200 - 325	250 - 400	275 - 500

Both tables show only **suggested** surface speeds  
Always consider complete setup conditions

CNC lathes and turning centers use *surface speed* directly (G96) and the control system calculates r/min based on the current part diameter. CNC machining centers require programmed values in direct *revolutions per minute*, so the part programmer is responsible for their calculation.

## CHIPLOAD

Programmed feedrate also depends on the units of measurement. When metric units are selected for CNC lathe applications, feedrate is programmed in *millimeters per revolution (mm/rev)*. For imperial units, feedrate is measured in *inches per revolution (in/rev)*. In CNC milling, feedrate is independent of the spindle speed, and is programmed in *mm/min* or *in/min*.

Chipload	Chip load - per tooth or per flute (fl)	
	Metric (mm) mm/fl	Imperial (inches) in/fl
Aluminum	0.200	0.008
Brass - Soft	0.125	0.005
Cast Iron - Medium	0.080	0.003
Steel - Mild	0.100	0.004
Stainless Steel (Free machining)	0.050	0.002

Feedrate specifies the tool velocity when being fed into the material. Feedrates affect the speed of metal removal, wear of the cutting edge, and chip formation. Speeds and feeds also have a major effect on productivity of the machine tool. Formulas to calculate feedrates are also available, usually from the same sources as various speeds. For both spindle speeds and feedrates, formulas should always serve as *starting point* only. Current setup conditions will always play the most important role.

## BASIC FORMULAS

Two common machine shop formulas can be used for metric units and imperial units respectively, where the *D* is the cutter diameter (for milling) or the part diameter (for turning) in millimeters or inches:

$$r/min = \frac{m/min \times 1000}{\pi \times D} \quad r/min = \frac{ft/min \times 12}{\pi \times D}$$

To calculate feedrate in *mm/min* or *in/min*:

➡ Metric feedrate - in mm/min :

$$F = r/min \times mm/fl \times N$$

🔍 where ...

F = Programmed cutting feedrate  
r/min = Programmed spindle speed (S)  
mm/fl = Chipload per flute in millimeters  
N = Number of cutting flutes (teeth)

➡ Imperial feedrate - in in/min :

$$F = r/min \times in/fl \times N$$

🔍 where ...

F = Programmed cutting feedrate  
r/min = Programmed spindle speed (S)  
in/fl = Chipload per flute in inches  
N = Number of cutting flutes (teeth)

## RELATED FORMULAS

There are other basic formulas related to machining that can be used for various calculations. They have all been covered in various chapters, and present only a summary here:

- Tapping feedrate
- Z-depth for spot drilling
- Z-depth for drilling
- Dwell time calculation
- Cutting time

### ◆ Tapping Feedrate

Tapping feedrate must always synchronize the tap *pitch* and the programmed spindle *speed*:

$$F = r/min \times pitch$$

### ◆ Z-depth for Spot Drilling

The main purpose of spot drilling is to make a small indentation on the part surface, so the angular drill point can enter the part at a precise location. Because of its end design (90° point angle), spot drill is also used for chamfering holes, which works very well, usually for up to size 25 mm or 1 inch hole diameters.

$$Z = H/2 + C$$

☞ **where ...**

- Z = Programmed depth for spot drill
- H = Hole diameter
- C = Chamfer size

*Note - the spot drill must be greater than double Z-depth*

#### ◆ Z-depth for Drilling (Through Hole)

Drilling hole through a part requires a small amount of breakthrough clearance below the part, so the drill diameter goes all the way through the part - standard 118° drill formula is shown:

$$Z = T + C + H \times 0.3$$

☞ **where ...**

- Z = Programmed depth for drill
- T = Part thickness
- C = Breakthrough clearance
- H = Hole diameter

For a blind hole, just add  $H \times 0.3$  to the full drill depth.

*Note - the 0.3 is a constant for 118° drill point angle only*

#### ◆ Dwell Time Calculation

To pause at the bottom of a cut, a dwell  $P$  is necessary. The minimum dwell calculation is:

$$P = 60 / r/\text{min}$$

*Note - for practical purposes the minimum dwell is usually programmed three times this amount*

#### ◆ Cutting Time

Cutting time in minutes can be calculated from the current feedrate and the length of cut:

$$T = L / F$$

☞ **where ...**

- T = Cutting time in minutes
- L = Length of cut in current units (mm or inches)
- F = Feedrate in current units (mm/min or in/min)

ALWAYS USE CUTTING FORMULAS AS A GUIDELINE !

Notes :





# NCPlot FEATURES

**NCPlot™** is an editor and backplotter for 4-axis mill and 2-axis lathe G-code programs. This software combines editing, formatting and translation tools useful for CNC programmers with a backplotter for instant G-code verification.

## Full Featured Text Editor

- Unlimited undo/redo
- Cut, Copy and Paste
- Find and replace
- Font/Color settings may be applied to any part of program
- Address coloring improves program readability

## Program Formatting Tools

- Remove block numbers, blank lines, comments or spaces
- Program renumbering with automatic updating of:
- Macro GOTO references
- M98 H_ references (for controls that support this)
- M97 P_ references (for HAAS style subprograms)
- Lathe G70 - G73 'Q' and 'P' addresses
- Insert spaces between letter addresses and macro keywords
- Fix invalid end of line characters
- Convert programs to all caps
- Add or remove comment or block skip characters

## Program Translation Tools

- Mirror
- Rotate
- Shift
- Scale
- Address Adjustment ...  
*Apply math operations to any program addresses*
- Address Replace ...  
*Good for changing axis names, A to B, etc.*
- Address Remove ...  
*Unnecessary addresses and their values can be removed*

- Address Swap ...  
*Exchange any two program addresses*
- Convert absolute I / J / K arc center coordinates to or from incremental I / J / K
- Convert R specified arcs to or from I / J / K specified arcs

## Advanced G-code Backplotter

*** Includes support for Custom Macro B ***

- Backplotting for Mill and Lathe programs
- Backplot macro programs offline !
- Can be customized for different control requirements
- Built in macro expression calculator lets you test macro expressions and set and view variables
- Step by step execution of macro programs allows you to watch program variables as the program is executed
- Breakpoint expression allows the program to run until the expression becomes True
- Subprogram display allows you to follow program execution even into subprograms that are not loaded

*Backplotter supports:*

- G65 custom macro calls
- M98 subprogram call in multiple command formats
- Plane selection G17, G18 and G19
- Helical motions
- Automatic corner rounding and chamfering in G01 motions
- Work offsets G54 - G59
- Extended work offsets G54.1 P1 - P300
- G52 Local work shift
- G92 Coordinate system setting
- G16 Polar coordinates
- G51 Coordinate scaling
- G51.1 Mirror image
- G68 Coordinate system rotation
- Local, common and some system variables
- Macro keywords IF, THEN, GOTO, WHILE, DO and END

## Flexible Viewport Controls

- A wide assortment of plot controls lets you quickly find any mistakes in your program
- 7 view modes for Mill and 4 for Lathe
- Dynamic zoom, rotate and pan
- Animate, step forward or step backward
- Draw to the cursor, from the cursor or just the selected portion of the program
- Clicking an entity on the viewport will display the entity properties and highlight the associated block in the program

## View Controls

- Many easy to use view controls to quickly zoom, pan or rotate the display

## Calc Tools

- Select entities on the viewport to quickly calculate offsets, blend arcs and find intersection points

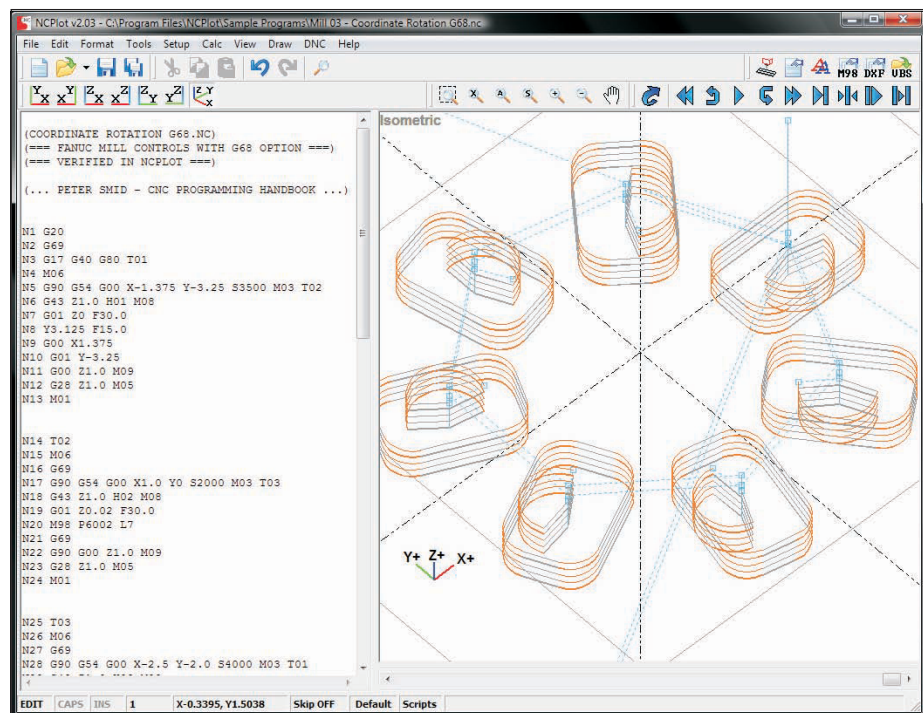
## DXF Importing / Exporting

- Import DXF drawings to quickly create G-code for both Mill and Lathe
- Any backplot can be saved as a DXF drawing file for loading back into a CAD or CAM system

Courtesy of NCPlot Software, LLC  
www.ncplot.com

Screenshot of NCPlot showing program example from Chapter 42:

Program 04202



## Other Tools

### Macro Translator

- Execute variable macros offline!

All variable commands are evaluated and the resulting values are used in their place. This creates a standard G-code program that can be run on any machine

NCPlot comes standard with macros for bolt circles, grid pattern and others

### Support for VBScripting

- Many of NCPlot's internal functions are available for scripting

This is a powerful tool that makes it possible to automate many common tasks such as converting files from one machine to run on another

### Text to G-Code converter

- Instantly create G-Code from font outlines

Great for etching, creates lettering of any size, on an angle, or even on an arc

### Macro Expression Calculator

- This built-in calculator accepts expressions written in Custom Macro B format - it can also be used to view or set variable values