

CNC Programming Techniques

An Insider's Guide to Effective Methods and Applications

First Edition

CNC Programming Techniques

An Insider's Guide to Effective Methods and Applications

Peter Smid

Industrial Press, Inc.
200 Madison Avenue
New York, New York 10016-4078, USA
<http://www.industrialpress.com>

< <PAGE TO BE REPLACED> > Library of Congress Cataloging-in-Publication Data

Smid, Peter
CNC Programming Techniques: xxx/
Peter Smid.
p. cm.
ISBN 0-8311-3158-6
1. Machine-tools--Numerical Control--Programming--Handbooks, manuals, etc.,...I.
Title.

TJ1189 .S592 2000
621.9'023--dc21
00-023974

First Edition

CNC Programming Techniques

Industrial Press Inc.
200 Madison Avenue
New York, NY 10016-4078

Editor: John Carleo
Cover Design: Janet Romano

Copyright 2005. 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.

1 2 3 4 5 6 7 8 9 10

Acknowledgments

To

**John Carleo
Janet Romano
and Patrick Hansard**

... without you, this book would not happen ...

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 bench marking, 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, monthly magazine columns, 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

CNC Programming Handbook, A Comprehensive Guide to CNC Programming

Fanuc CNC Custom Macros: Practical Resources for Fanuc Custom Macro B Users

Both hardcover books have been published by Industrial Press, Inc.

The author always welcomes comments, suggestions and other input from educators, students and industrial users.

You can send e-mail to the author from the *CNC Programming Techniques* page at

www.industrialpress.com

Preface

I would like to express my most sincere thanks to all programmers, machinists, operators, engineers, students and many other readers and users who made my two previous books - also CNC oriented - such a great success. Both were published by *Industrial Press, Inc.* (New York, NY, USA):

📖 CNC Programming Handbook, Second Edition with CD-ROM
A Comprehensive Guide to CNC Programming
ISBN: (0-8311-) 3158-6

and

📖 Fanuc CNC Custom Macros, with CD-ROM
Practical Resources for Fanuc Custom Macro B Users
ISBN: (0-8311-) 3157-8

This third handbook also relates to the subject of CNC programming, this time from a somewhat different angle. First, there several programming subjects that are virtually impossible to find anywhere else, for example, how to program cams or tapered end mills. Other, more common, subjects are covered in a great depth, such as the coverage of cutter radius offset or thread milling.

As in my previous publications, I have included many overall and detailed drawings, to help visualize the subject or procedures covered. Where applicable, a complete programming example is provided, or - at least the most significant part is shown.

In view of the recent, and rather significant, emergence of metric system in many North American industries, particularly in the USA, I have focused on more examples presented in metric units than those in imperial units. Working on the premise that a professional CNC programmer should have no problem working with either units selection (after all, number are numbers), many examples in this handbook emphasize the metric system. For balance, a significant number of examples using imperial units are also included. Speaking of *imperial* units - in my previous books, I had used the term *English* units instead. It may seem frivolous, but the fact is that modern Great Britain is now a metric country and the so called *English units* are the thing of the past - of the *imperial* era in British history.

I also feel that I should mention the relationship of this book to the *CNC Programming Handbook*. In terms of focus, these are very different publications. *CNC Programming Techniques* is a book that does not replace my previous books, but complements them in a special way. In terms of subjects covered, there are minor similarities in some chapters, but the coverage of each subject is fresh, and with much more detail provided. At the end of the book, I had included references to subjects covered in the *CNC Programming Handbook*. My feeling was that those readers who may need some additional background will benefit from these references. On the other hand, those, who do not need the background can safely ignore those few pages and explore the subjects covered in this book only.

I sincerely hope that this book will help you become even a better CNC programmer (or even a better CNC Operator) by understanding not just the 'hows' but also the 'whys' of many programming techniques. Thanks you for your continuing interest.

Peter Smid
November 2005

TABLE OF CONTENTS

1 - PART PROGRAM DEVELOPMENT	1
Program Development Drawing	1
Drawing Evaluation	2
Material and Stock	3
Part Setup	3
Part Reference Point	3
Part Orientation	3
Selecting Part Zero	4
Tooling Selection	4
Identifying Machining Operations	4
Face Milling	5
Contour Milling	6
Circular Pocket Milling	6
Slot Milling	7
Spot Drilling	8
Drilling	9
Tapping	9
Summary of Tools Used	9
Machining Data	10
Spindle Speed	10
Cutting Feedrate	11
Tooling Data	11
Details of Operations	11
Tool 1 - Face Milling	12
Tool 2 - Outside Contour	13
Tool 2 - Circular Pocket	15
Tool 3 - Slot Milling	16
Tool 4 - Spot Drilling	17
Tool 5 - Drilling	18
Tool 6 - Tapping	19
Complete Program	20
2 - CALCULATING CONTOUR POINTS	23
Tools and Knowledge	23
Mathematical Knowledge	23
Organized Approach	25
Process of Calculating XY Coordinates	25
Step 1 - Establish the Main Contour Points	26
Step 2 - Fill-in the Coordinate Sheet	26
Step 3 - Identify Calculation Zones	27
Step 4 - Helpful Ideas for Calculations	27
Step 5 - Calculations for Zone 1	29
Step 6 - Calculations for Zone 2	31
Updating Coordinate Sheet	32
Writing the CNC Program	32

3 - FORMULAS FOR CONTOURING	33
Contour Point Between Two Lines (Lathe)	33
Contour Point Between Line and Arc	34
Intersecting Contour Point	34
Tangent Contour Point	35
Calculating the Sharp Point	39
Contour Point Between Two Arcs	40
Intersecting Arcs	40
Tangent Arcs	41
4 - USING CUTTER RADIUS OFFSET	43
General Concepts	43
Benefits Of Cutter Radius Offset	44
Controlling Cutter Radius	44
Radius Offset Commands	45
Commands G40-G41-G42	45
Using the D-offset Number	45
Basic Programming Techniques	46
Cutter Radius Activation	46
Cutter Radius Application	47
Cutter Radius Cancellation	47
D-offset Stored Amount	47
Equidistant Centerline - G40 Mode	48
Equidistant Centerline - G41/G42 Mode	48
Drawing Dimensions - G40 Mode	48
Drawing Dimensions - G41/G42 Mode	48
Radius vs Diameter	48
Contour Lead-In and Lead-Out	49
Methods For Lead-In - Linear Motion	49
Methods For Lead-In - Arc Motion	52
Methods For Lead-Out - Linear Motion	53
Methods For Lead-Out - Arc Motion	53
Program Example	53
Internal Contours	54
Linear Slot Machining	54
Circular Slot Machining	55
Finishing Internal Contour	56
Maintaining Dimensional Sizes	58
Basic Rule	58
Handling Dimensional Tolerances	58
Handling Cutter Radius Offset Errors	60
Common Errors	60
Offset Programmed Too Late or Too Early	61
Offset Start or End on an Arc	62
Tool Nose Radius Offset	62
Command Point and Radius Center	62
Tool Tip Orientation	63
Common Tool Nose Radius Errors	64

5 - PART REVERSAL IN MILLING	67
Project Description	67
Material and Setup Conditions	67
Cutting Tools	68
Material Removal	68
Machining Process	69
Clamping 1	69
Clamping 2	70
Program Zero Selection	70
First Clamping	70
Second Clamping	71
Programming Methods	72
Tool Length Settings	72
First Clamping	73
Second Clamping	74
Using the WORK OFFSET Method - G54-G55	74
Common Toolpath	76
Program Listing - WORK OFFSETS G54-G55	77
Program Listing - WORK OFFSETS G54-G55 - with Subprograms	79
Using the LOCAL COORDINATES Method - G52	81
Program Listing - LOCAL COORDINATE SYSTEM G52	83
Using the DATUM SHIFT Method - G10	85
Program Listing - DATUM SHIFT G10	86
Summary	88
6 - USING TAPERED END MILLS	89
Types of Tapered End Mills	89
Tool Material	90
Range of Taper Angles	90
Flat Tip Tapered End Mills	90
Ball Tip Tapered End Mills	91
Effective Diameter Calculation	91
Flat Tip	91
Stock Removal	93
Ball Tip with Specified Radius	94
Flat Tip with Added Blend Radius	94
Tapered Holes	96
7 - SPECIAL PURPOSE G-CODES	97
Single Direction Positioning - G60	97
Special Cutting Modes	98
Exact Stop Check G09 - G61	99
Automatic Corner Override - G62	100
Tapping Mode - G63	100
Normal Cutting Mode - G64	101
Stored Stroke Limits Definitions - G22 - G23	101
Spindle Fluctuation G25 - G26	103

Machine Zero Commands - G27- G28 - G29 - G30	104
Primary Machine Zero Return - G28.	104
Return from Machine Zero - G29	106
Machine Zero Return Position Check - G27.	106
Secondary Machine Zero Return - G30	108
Position Register - G92/G50	108
G92 Position Register for Milling	109
G50 Position Register for Turning	111
Tool Change Position	113
Conversion of G50 to Geometry Offset	117
Skip Command - G31	118
Other Seldom Used G-codes	119
Tool Length Offset Negative - G44	119
Tool Length Offset Cancel - G49	119
Conclusion.	122
8 - TOOL LENGTH OFFSET CHANGE	123
Tool Length Offset	123
Offset Adjustment	124
Practical Application	124
Programming Method 1 - No Offset Adjustment.	125
Programming Method 2 - With Offset Adjustment	125
Programming Method 3 - Advanced Macro Method	126
Offset Adjustment - Setup for Two Parts	128
Method 1 - One Work Offset + One Length Offset	128
Method 2 - Two Work Offsets + One Length Offset	129
Method 3 - Two Work Offsets + Two Length Offsets	130
9 - BLOCK SKIP APPLICATIONS	131
General Applications	131
Similar Parts Applications	132
Programming a Trial Cut	134
Trial Cut for Milling	134
Trial Cut for Turning	136
Irregular Stock Removal	137
Variable Stock in Milling	137
Variable Stock in Turning	139
Summary of Rules	139
Block Skip Within A Block	140
Conflicting Words	140
One Program - Two Materials	140
Numbered Block Skip Functions	142
10 - STANDARD AND RIGID TAPPING	143
Standard Tapping Method	143
Basic Principles	143
Why Underfeed?	144
Feed-In Slower - Feed-Out Faster.	144

Rigid Tapping Method	146
Basic Principles	146
Benefits	146
Setup	147
Possible Problems	147
Programming Approach	147
11 - POLAR COORDINATES	149
Definition and G-codes	149
Polar Coordinates and Planes	150
G15 - G16 Polar Coordinates	151
Programming Format	151
Toolpath Direction	153
Applications in Planes	154
12 - SUBPROGRAM DEVELOPMENT	155
Definition and Usage	155
Drawing Evaluation	156
Subprogram Planning	156
Depth Control	157
Width of Cut Control	157
Cutting Tool Selection	157
Developing the Subprogram	158
Method 1 - Full Width and Full Depth	158
Method 2 - Full Width and Divided Depth	159
Method 3 - Smaller Width and Full Depth	160
Round Pocket Subprogram	162
Single Depth Pocket with Stepovers	162
Multidepth Pocket with Stepovers	164
Rough and Finish Cuts with a Subprogram	165
One Toolpath for Two Cuts	165
Lead-In and Lead-Out	167
Common Contour Toolpath	167
Main Program	168
13 - TURNING AND BORING IN DEPTH	169
Program Zero Selection	169
Corner Radius and Back Angle Selection	170
Cutter Radius Offset	171
Imaginary Tool Point.	172
Stock Allowance	172
Contour Shape	173
Cutting Tool Used.	173
Stock Allowance in X and Z axes.	173
Grinding Allowance	174

Tool Approach Techniques - Lead-In	176
Approaching the Front Face	176
Approaching a Diameter	176
Approaching a Chamfer	177
Approaching a Radius	177
Approaches to Avoid	179
Tool Retract Techniques - Lead-Out	179
Retract from a Face	179
Retract from a Diameter	180
Retract from a Chamfer	180
Retract from a Radius	181
Retracts to Avoid	181
One Job - Two Operations	182
About Jaws	182
Single Setup - Two Chuckings	183
Two Setups - Two Operations	184
Multi Cut Facing	184
Width of Cut Distribution	184
Breaking Corners	185
Direction Specification	186
Using Tailstock	186
Types of Tailstock	187
Programming a Tailstock with a Bar Stopper	187
Using 45-degree Tool	189
Machining Thin Stock	192
Adjusting Chuck Pressure	192
Using an Inner Plug / Outer Ring	192
Using Special Split Jaws	192
G70/G71/G72 Cycle Methods	193
Programming Formats - G71	193
Programming Formats - G72	194
G70 - Finishing Cycle	196
G71 and G72 Compared	196
Programming Undercuts	198
Hard Turning	198
14 - PROGRAMMING TAPERS	199
What is a Taper?	199
Taper Definitions	199
Taper per Foot	200
Taper Ratios	203
Taper Defined as Percentage	205
Taper Angle Defined in D-M-S	206
Taper Length and Angle	206
Chamfers	206
45 Chamfer	207
Start Chamfer with a Clearance	209
End Chamfer with a Clearance	209
Other Chamfers	210

Tapers with Leads	211
Taper with a Lead Chamfer	212
Taper with a Lead Fillet	213
15 - TECHNIQUES FOR GROOVING	215
Tooling for Grooves	215
Cutting Width	215
Cutting Depth	215
Groove Location	216
Setting the Command Point	216
Plunge and Retract Method	217
G75 Cycle	217
Grooving for Precision	218
Machining Procedure	219
Programming Procedure	220
Deep Grooving	221
Grooves with Tapers - O-Ring Grooves	222
Grooves with Tapers - V-Pulley Grooves	225
Insert Selection	225
Depth Calculation	226
Tool Setup and Program	226
16 - TECHNIQUES FOR THREADING	227
Types of Thread Forms	227
UN - Unified National and Metric	227
Other Thread Forms	228
Thread Depth Calculation	229
Infeed Methods	230
Tool Motions	231
Cutting Conditions	231
Acceleration and Deceleration	231
Cutting Depth	232
Hand of Thread	232
External Threading - Right Hand Thread	232
External Threading - Left Hand Thread	233
Internal Threading - Right Hand Thread	233
Internal Threading - Left Hand Thread	233
Sample Thread Evaluation	234
Initial Data	234
Cutting Conditions	234
Number of Threading Passes	235
Distribution of Depth Cuts	236
G32 Threading Method	236
Radial Infeed Example	237
Flank Infeed Example	237
Tapping with G32	238
G92 Threading Method	240
G76 Threading Method	241
Programming Format	241

17 - RESTRICTIONS IN THREADING	243
Thread Programming Basics	243
Threading Feedrate	244
Standard Example	244
Final Threading Program - Single Start	245
Special Example	245
Final Threading Program - Multi Start	246
Slow Spindle Speed	247
Metric Applications	247
Long Thread Programming	248
Defining a Long Thread	248
Lead Error	249
Number of Decimal Places	250
18 - PRACTICAL THREAD MILLING	251
Thread Milling - General	251
Helical Interpolation	251
Is Helical Interpolation Available ?	252
Benefits of Thread Milling	252
Selecting Tools	253
Initial Factors	253
Types of Thread Milling Cutters	254
Accuracy Issues	254
Thread Mill Data	255
Cutting Direction	255
External and Internal Thread Milling	255
Climb Milling and Conventional Thread Milling	255
Right Hand and Left Hand Thread Milling	255
External Thread Milling Illustrated	256
Internal Thread Milling Illustrated	257
Helix - Helical Curve	258
Programming External Threads	260
Tooling Selection	260
Cutting Conditions	261
Lead-In and Lead-Out	261
Cutter Radius Offset	262
Program Development - External Thread	262
Programming Internal Threads	266
Tooling Selection	266
Cutting Conditions	266
Lead-In and Lead-Out	267
Cutter Radius Offset	267
Program Development - Internal Thread	268
Pipe Thread Milling	271
Thread Milling Software	272

19 - KNURLING ON CNC LATHES	273
Knurling Operations	273
Tooling Selection	273
Knurling Pitch	274
Programming and Machining Techniques	275
Tool Motions	275
Depth and Feedrate	275
Troubleshooting	276
20 - FOUR-AXIS LATHES	277
General Setup	277
Tool Tip Numbers	278
Programming Method	278
Spindle Speed and Feedrate	278
M-Functions	279
Synchronization Functions	279
Program Structure	279
21 - PALLET CHANGERS	283
Types of Automatic Pallets	283
Rotary Pallets	283
Shuttle Pallets	283
Setup and Work Areas	284
Programming Methods	285
M60 Function	285
General Format	285
Programming Example	286
Initial Conditions	286
Part Program	286
22 - WORKING WITH PLANES	289
Mathematical Planes	289
Machine Planes	290
G-codes for Plane Selection	290
Effect of Planes in Programming	291
Planes and Circular Motion	291
Planes and Cutter Radius Offset	293
Working With Planes In Detail	295
G17 with G41 and G02	295
G17 with G42 and G03	296
G18 with G42 and G03	297
G18 with G41 and G02	298
G19 with G41 and G02	299
G19 with G42 and G03	300
Using Right-Angle Attachment	301
Basic Concepts	301
Side Face Drilling	302
Side Face Milling	304

23 - PROGRAMMING CAMS	307
Overview of Cams	307
Cam Drawing Example	308
Cam RISE and FALL - Sections Evaluation	309
The RISE Section	310
Calculating Radius Length	311
Calculating XY Coordinates	311
The FALL Section	312
Calculating Radius Length	313
Calculating XY Coordinates	314
Summary	314
Writing the Program	315
24 - INTRODUCTION TO MACROS	317
Special Introduction	317
Skills Required	317
Macro is an Option	318
Common Features and Applications	318
Macro Structure	319
Macro Definition and Call	320
Variable Declarations and Expressions	322
Macro Functions	323
Branching and Looping	326
Macro Development - Bolt Circle	327
Evaluation of Drawings	328
Bolt Hole Macro Features	329
Assignment of Variables	330
Internal Calculations	330
Other Calculations	330
Final Considerations	331
Macro Call	332
25 - DID YOU KNOW THAT ... ?	333
26 - REFERENCES AND RESOURCES	335
Index	339

1

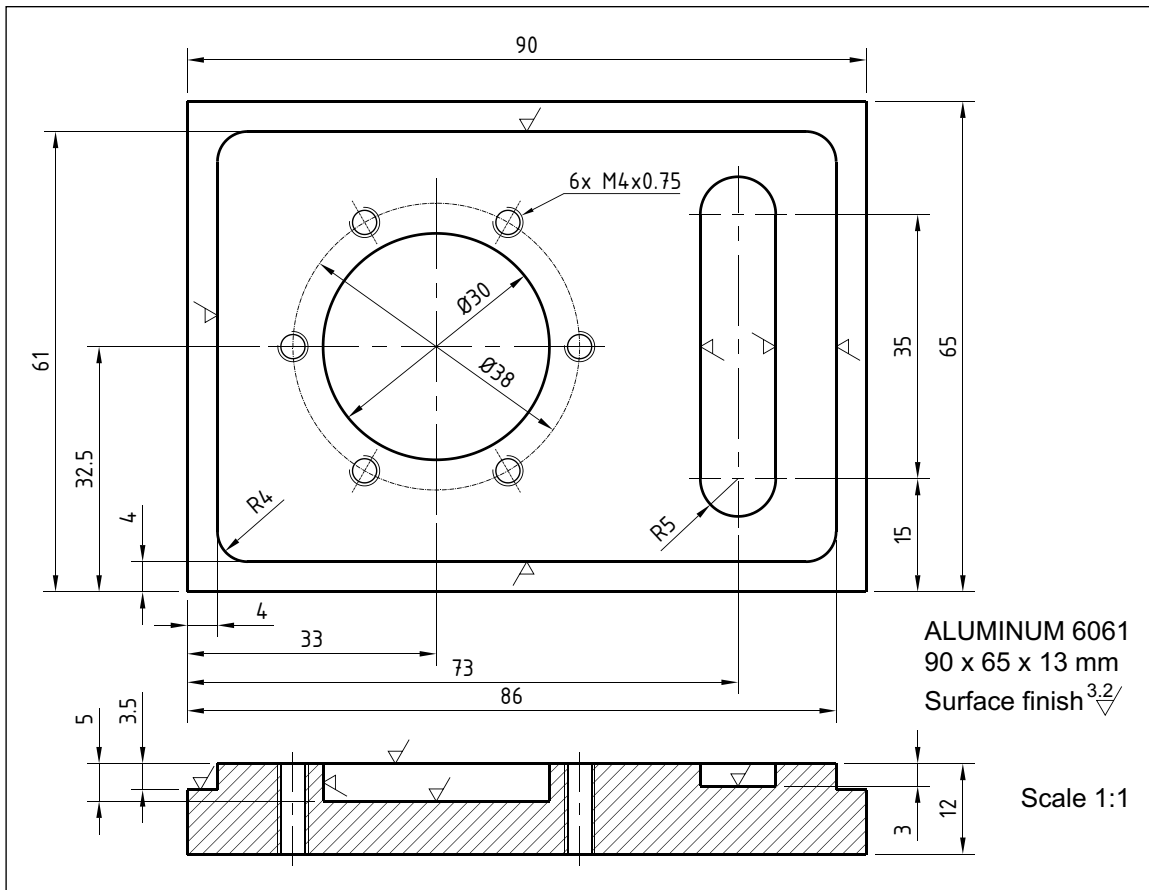
PART PROGRAM DEVELOPMENT

This introductory chapter to the *CNC Programming Techniques* presents the most basic technique of all - *how to develop a part program* in an organized way. Its purpose is to present an engineering drawing, evaluate it and develop all procedures required to write the final program.

In strict terms, program development using a step by step procedure is not the most appropriate approach, because it suggests '*finish step 1 before starting step 2*'. That is never the case in CNC programming, as many 'steps' interact with each other. For example, a change in setup may influence the tooling selection, width or depth of cut, etc. Keep this in mind when studying this chapter.

Program Development Drawing

The drawing below will be used throughout the chapter, including required details, calculations, all complete with explanations of individual steps required in the CNC program development. The drawing includes some of the most common machining operations - face cutting, machining holes, contouring, circular pocket, and a slot milling. The design has been kept intentionally simple.



Drawing Evaluation

The first thing a CNC programmer should always do before writing the program, is to evaluate the drawing in order to get a general idea about the part. Such an evaluation includes several observations that can be summed up:

- ◆ Drawing units and scale
- ◆ Dimensioning method
- ◆ Tolerances
- ◆ Material type, size, shape and condition
- ◆ Surface finish requirements
- ◆ Title block information
- ◆ Drawing revisions
- ◆ Bill of Materials (BOM) - if available
- ◆ Omissions and other errors

In the enclosed drawing the *drawing units* are not specified directly, but it is obvious from the drawing that it uses metric dimensions in millimeters. *Scale* is not always specified in the drawing, often because of various forms of copying may not match the original scale. The drawing for this chapter shows a full size drawing and is specified as 1:1.

Drawing *dimensions* are always important for the CNC programmer, at least for two reasons. One, they establish the important features of the part, two, they serve as the most important resource in determine the part zero (part origin). In the drawing, all dimensions originate from the lower left corner of the part - in this case, that will also be the most suitable location of the part zero. Keep in mind that this is not always the case.

Tolerances are closely related to dimensions. This particular drawing does not contain any tolerances, so the aim of the programmer (and operator) will be to adhere to general company standards.

Not all drawings describe every aspect of the *material* the part is made of, but for programming purposes, the type, size, shape and condition of the blank material are the most important. The sample drawing specifies the material type and its size. *Aluminum 6061* is easy to machine, and fairly high speeds and feeds can be used for efficient machining. The material size is specified in the drawing as 90 65 13 mm (L W D). Here comes the first item that will have a direct relationship with the tooling selection and machining operations. Although the length and width of the material are the same as the final length and width of the part, that is not the case with the material depth (thickness). There is 1 mm difference that has be accounted for during programming and even setup.

The drawing also specifies the overall *surface finish* of 3.2 for all marked surfaces. Not all drawings specify individual surfaces. The amount of 3.2 is a statistical deviation from the ideal profile and is measured in μm (millionth of a meter = $0.000\ 001\ \text{m} = 10^{-6}$). In practical terms, the required finish of 3.2 μm is attainable with standard milling cutters at relatively fast spindle speeds and moderate feedrates, assuming proper setup and quality tooling.

Small and simple drawings seldom have an elaborate *title block*. Title block is usually a rectangular area in a corner of the drawing that contains text data, such as drawing name, part number, drafterperson, dates, revisions, material, etc. *Revisions* are changes made to the drawing from its original version - they are very important to the CNC programmer. Always make sure you develop the program using the latest drawing version. Keeping a copy of the drawing is also a good idea.

Bill of Materials (abbreviated as BOM) is a special list that contains individual items required to manufacture the part. Typical items included in BOM are stock items, purchased items and various other parts required for assembly. Large complex drawing are more likely to have BOM than small simple drawings.

Even the best engineers and draftspersons make a mistake. One important part of drawing evaluation is to look for **errors**, **omissions** and other discrepancies. It is most frustrating to go through many calculations just to find out that a critical dimension was missing. The programmer should also look for double dimensioning, where one dimension is in conflict with another dimension. Finally, when evaluating a drawing, never try to find dimensions by scaling the lines or arcs, never assume a dimension or a feature. *If in doubt, always ask.*

Material and Stock

Although the enclosed sample drawing does include material the part is made of, there are many issues related to the material before machining (and programming). Apart of size of the material, its **shape and condition** are equally important. Shape could be a simple block or cylinder, it could be cored or solid, it could be a complex casting or forging, and so on. Shape of the material is most important in program development during setup and tooling decisions as well as toolpath. The condition of the material includes the overall quality - such as burrs, flakes, premachining, hardness, etc. It also includes the accuracy of the shape. For example, it is important that the material supplied for the sample drawing is exactly 90 65 mm in length and width, square at corners, as these sizes are already final and need no machining. There is a little bit more flexibility in the material thickness, because the top will have to be faced off to suit drawing dimension of the part height.

Part Setup

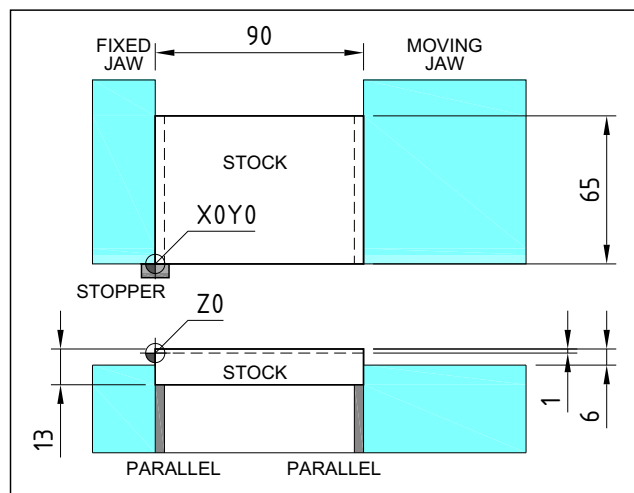
A high quality vise, specially designed for CNC work is the most common fixture (part holding device) for small to medium size parts on vertical machining centers.

Part Reference Point

Part reference point is another common name for *part zero* or *part origin*. Before any toolpath can be developed in the form of coordinates, the CNC programmer has to select *part zero*. As a general rule, the part zero for setups in a vise should always be located on a non moving jaw (fixed jaw), and a part stopper or similar device is also recommended for repeatability.

Part Orientation

How the part blank (stock) is oriented in the vise often influences the method of machining. Looking at the sample drawing, the part blank can be oriented either horizontally or vertically, looking



from the CNC operator's direction. Horizontal orientation has the advantage that the machined part will match the drawing. Another benefit is that the lower left corner of the part will be located at the intersection of the vise non moving jaw and a stopper. The only benefit of vertical orientation would be the width of grip being 65 mm rather than 90 mm in the horizontal orientation, minimizing any 'bending' effect caused by pressure of the jaws. For this job, we select the horizontal orientation, as the 25 mm difference between widths has no practical disadvantage.

Selecting Part Zero

Based on the previous considerations in this section, the selection of part zero for the X and Y axes presents no problem. The lower left corner of the part will be the part zero. This is also the coordinate location that will be used to set the work offset G54.

Setting the part zero for the Z-axis requires some thinking as well as evaluation of several possibilities. Unless the machine shop uses off-machine presetter to set the tool length, the most common method of tool length setting is the *touch-off* method. Selecting Z0 is important part of the setup. The most common method is the *top of the finished part*, but bottom of the part or some other fixture location may also be considered. In this example, the top of the *finished* part will be Z0, which brings up a question - what to do with the 1 mm extra height? The top face will be face milled, and the 1 mm extra thickness will be removed by that operation. All tool lengths will be set on the finished face. Later section of this chapter will provide more details. Look at the illustration shown on the previous page.

Tooling Selection

Selecting tool holders and cutting tools is another very critical part of part program development process. Tool holders are generally used with different cutting tools and remain unchanged for extended periods of time. Tools, such as drills, reamers, taps, end mills, carbide inserts, etc. are perishable tools. Some toolholders are used for a specific tool group, such as a collet holder is a better choice for end mills, Weldon type toolholders prevent tool spinning, Jacobs chucks are used for drills, etc. Some tools, such as taps, require special holders, designed for a single purpose.

Tooling selection is always closely related to the setup and cutting conditions. When selecting tools, always keep in mind these other two considerations. When the part setup method has been established, the tools can be selected on the basis of drawing dimensions and machining operations required. Tools are always selected on the basis of machining operations required.

Identifying Machining Operations

From the sample drawing, even a casual look will identify the types of operations required to machine the part - all are very common and can be adapted to other jobs:

- ◆ Face milling
- ◆ Contour milling
- ◆ Circular pocket milling
- ◆ Slot milling
- ◆ Spot drilling
- ◆ Drilling
- ◆ Tapping

One of the most important machining rules is that heavier operations are always done before lighter operations. That does not just mean roughing before finishing, which is a common sense, it also means milling before drilling, for example. Milling has the tendency to shift the material in the XY axes, whereby drilling pushes the material towards the fixture (Z-axis). The above operations list is suitable to be used as the order of operations.

Face Milling

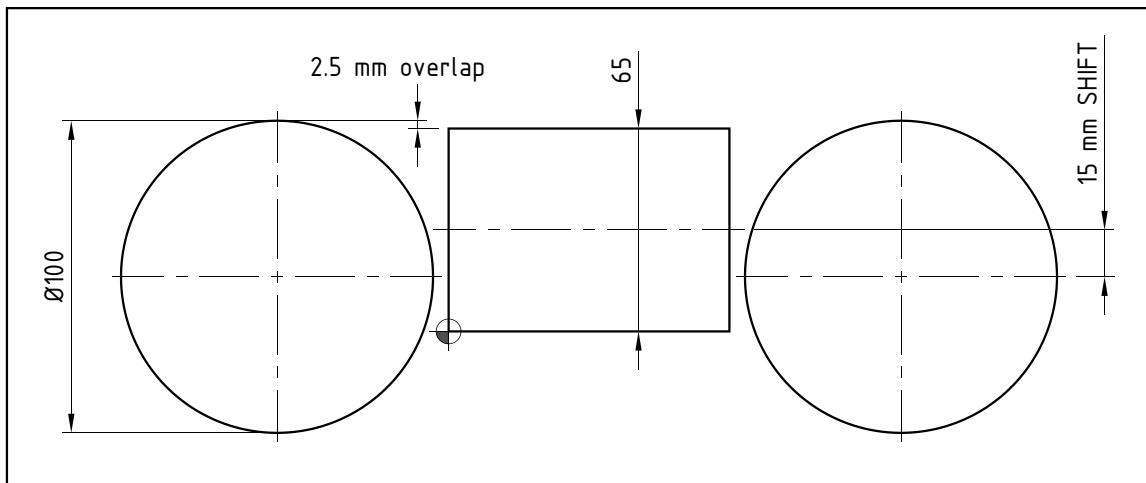
As the part is 1 mm thicker than the final thickness, a face milling tool has to be used to machine the top face to provide final thickness of 12 mm, as per drawing. The amount of one millimeter represents a small depth for facing, so only one depth cut will be necessary to face the part. In order to select the best face mill diameter, the part length and/or width have to be considered as well. In many cases, the length or width of the part also determines the *direction* of the cut. The length of the part is 90 mm, its width is 65 mm. If either direction is acceptable, a standard 100 mm face mill would be the best choice. A 75 mm face mill will only be able to face along the X-axis. If both face mills are available, which one is a better choice? Number of flutes (cutting inserts) in the tool should also be considered, to employ as many edges at the same time. If the 100 mm face mill has more cutting edges, it will be a better choice. It will also be the choice for this job.

Cutting direction is very important as well. Although the 100 mm face mill can cut in four directions along the two axes, choice of the X-direction, from the X+ to the X- (right to left) is recommended. The reason is that the cutter will be 'pushing' against the vise fixed jaw, creating favorable cutting conditions. Cutting along the Y-axis can cause the face cutter to pull the part up. Last, and equally important consideration is the location of the face mill center relative to the part. It is not recommended to locate the cutter center at the middle of the part, but slightly off the middle. Such tool position will better control of chips during the entry and exit, to and from the material. It also minimizes chatter.

The Y-axis location of the face mill center must still guarantee that the full part width of 65 mm can be machined. To calculate the maximum shift possible, take one half of the difference between the face mill diameter (100 mm) and the part width (65 mm):

$$\text{Maximum shift from part middle} = (100 - 65) / 2 = 17.5 \text{ mm}$$

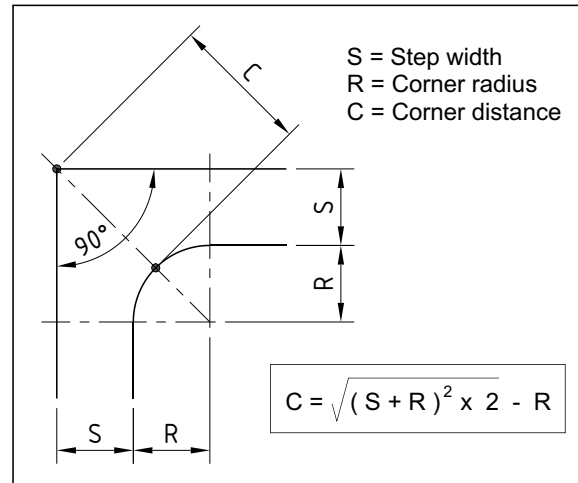
Select the Y-location a little less, to make the tool overhang the part. For this example, 15 mm is a reasonable shift, with 2.5 mm left for edge overlap - see illustration.



Contour Milling

The filleted rectangular shape is located four millimeters from all edges, except at the corners, where the distance is much greater. This corner distance is very important when selecting cutter diameter. If only a single contouring toolpath is required, the cutter diameter should be larger than the corner distance C - see illustration. If the corner distance is too large for any cutter size, two or more passes may be required, or some other method of machining needs to be selected.

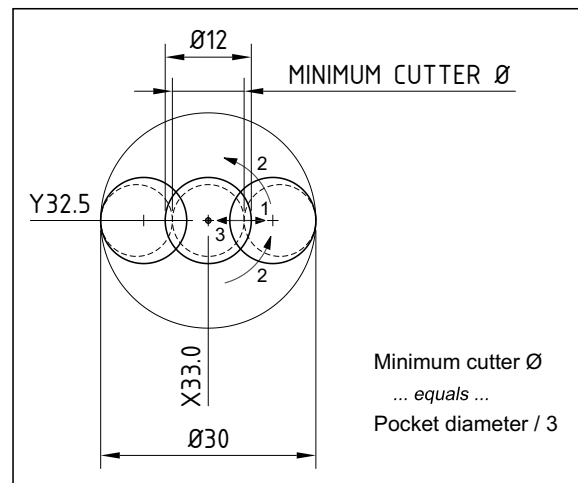
The step depth along the edges is 3.5 mm and should also be considered when the cutting tools are selected. For this job, the depth does not present any special difficulties, and can be machined in one pass.



In our case, the step width S is 4 mm, the corner radius R is also 4 mm. Using the above formula, the C dimension (maximum corner distance) is 7.31371 mm. In practice, any end mill 8 mm in diameter or larger can be used to clean the corners in a single pass. This conclusion is correct, but actual cutting conditions have to be considered as well. Selecting the 8 mm end mill will not be a good choice in this case, because the radius of the end mill is the same as the step width (4 mm). The center of the cutter will follow the exact edge of the part, which is not the best machining condition. In addition, a larger tool diameter will provide more tool rigidity and prevent deflection. A two-flute or a three-flute end mill, especially designed for cutting aluminum will be the best choice. As for the diameter, the selection should not always be related to the current operation only, but other operations as well. Either a 10 mm or 12 mm end mill will be a good choice - the final decision will depend on the tool requirements for the circular pocket. The slot width is 10 mm, so the tool diameter must be smaller than that for optimum cutting conditions.

Circular Pocket Milling

The pocket in the drawing is 30 mm in diameter and 5 mm in depth. Unlike the end mill selected for contouring, the end mill for the pocket must be *center-cutting*, in order to plunge into solid material. Center-cutting end mills are also called slot drills, because they were originally designed for milling of standard slots. High helix, three-flute precision ground center-cutting end mills, are the best overall solution to cutting aluminum. Three flute end mills are often frowned upon - unfairly, because they cannot be measured with standard verniers or micrometers. Yet, they offer the strength, the chip flow and the surface finish that many aluminum parts require. Of course, the more common two-flute end mills are also a suitable choice.



Circular pockets can be machined many different ways, but the most economical method is the single pass method, particularly for small pockets. As shown in the illustration, the minimum diameter of the selected cutter must be one third of the pocket diameter. The reason for the one third is that such tool selection guarantees the complete clean-up of the pocket bottom. The circular pocket in the drawing is 30 mm, so the minimum cutter diameter required is $30/3 = 10$ mm. Although correct mathematically, in practice it is always better to use a slightly larger cutter diameter, to overlap all cuts at the pocket bottom.

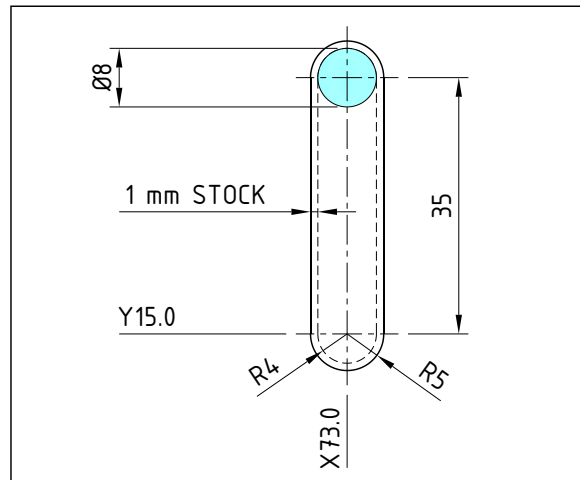
In the description of contouring (see above), the choice of tools was narrowed down to two end mills - 10 mm end mill or 12 mm end mill. Either tool will work well for the contouring, but only the 12 mm end mill is also suitable for machining the circular pocket. The selection of 12 mm end mill for *both* operations will also offer benefits in a shorter setup time, tool maintenance, and even inventory. The final selection is the 12 mm end mill, used for both - the contour and the pocket.

There is one final condition that has to be considered. In order to machine the pocket, a center-cutting end mill has to be used. That is not required to machine the contour. If only one tool is used for both operations, it has to be a center-cutting end mill for this job. It is perfectly suitable for the contour, and its center-cutting design is mandatory for cutting the circular pocket.

Slot Milling

Unless there are some very tight tolerances required, a standard slot can be machined with a single tool, usually a center-cutting end mill. There is an established - *and very specific* - procedure to cut standard slots (described later). It includes a single pass between centers, followed by the internal contour to make the slot according to the engineering drawing. The main consideration in tooling selection is the *slot width*.

In the drawing, the slot radius is 5 mm, so its width is the double the radius, 10 mm. Whatever your programming style may be, resist the temptation to use a 10 mm center cutting end mill between slot end centers only - the quality of the slot will be very poor and so will its dimensions. Much better choice is to select a center-cutting end mill that is a bit smaller than the slot width. Is the size of such an end mill important? If so, why? YES - the size of the selected end mill is very important, because it will control the *amount of stock left* on the slot walls for finishing. For example, if we choose a 7 mm end mill, it will leave 1.5 mm per side for finishing; 9 mm end mill will leave 0.5 mm per side for finishing, and 8 mm end mill will leave 1 mm per side for finishing.



All of the three basic selections are correct, but the choice has to be made for this particular part, as represented by the drawing. Leaving 1 mm per side for finishing is reasonable, so the cutting tool selected for the slot will be an 8 mm center-cutting end mill (two or three flutes) - *see illustration*.

If the slot is dimensioned with tight tolerances, it may be a better choice to select *two* suitable tools - one that would make the roughing cut, the other the finishing along the slot walls. Whether one or two tools are used, the finishing cut should be made with cutter radius offset in effect, so the final slot dimensions can be fine-tuned at the machine, via the control system.

Spot Drilling

When drilling holes, a small chamfer - or even a small corner break - is very desirable at the top of the hole. A chamfer will eliminate burrs or sharp edges that are the natural result of drilling, allowing a smoother entry of a tap, or just eliminating the burrs for easier handling. Spot drill is a tool that is used for this purpose; it resembles a regular drill and has two main purposes:

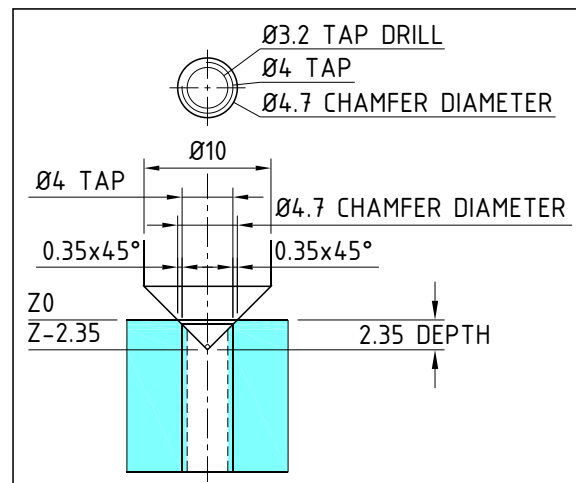
- ◆ To start-up a hole with a small dimple at its exact location ... control of hole location
- ◆ To machine a chamfer on a hole, by controlling the depth of cut ... control of chamfer size

In the example, no chamfering or corner breaking of the hole is specified. In CNC work, it is quite customary to break sharp corners, unless the drawing instructions specifically prohibits it.

Unlike any drill selection, spot drills are offered only in about three to six diameters, depending on the choice of individual tool manufacturers. One of the most common spot drills used is a very versatile diameter of 10 mm (or 12.7 mm = 0.5 inch in Imperial units). If you are not familiar with spot drills, bear in mind that only the *angular portion* of the drill is used, never its own full diameter. Also keep in mind that the majority of spot drills have a 90° included angle at the tool tip. This is a very practical angle, as it allows to make 45° chamfers on small and medium size holes.

One purpose of spot drilling is to make a small dimple at the XY hole location - the exact size of the dimple is not important, so 2-3 mm depth will usually be sufficient, depending on the hole diameter. On the other hand, many holes are not just spot drilled for the purpose of maintaining their XY location - they are also *chamfered* with same tool, to a particular chamfer specification, typically at 45°.

The six holes in the drawing have to be spot drilled, drilled and tapped. The tap diameter is 4 mm, which is the largest size of the hole. Any functional chamfer for a given hole diameter must be bigger than the largest hole. The drawing specifications may include the chamfer size - otherwise the decision is in the hands of the programmer. For the drawing in the given example, no chamfers are given, so an arbitrary decision has to be made - by the CNC part programmer. Chamfers are usually very small, typically within the range of 0.125 to 0.5 mm (0.005 to 0.02 inches) at 45 degrees. For small holes, the chamfer can also be quite small. As this is an arbitrary decision for the example provided, the six holes in the drawing will be 0.35 x 45°.



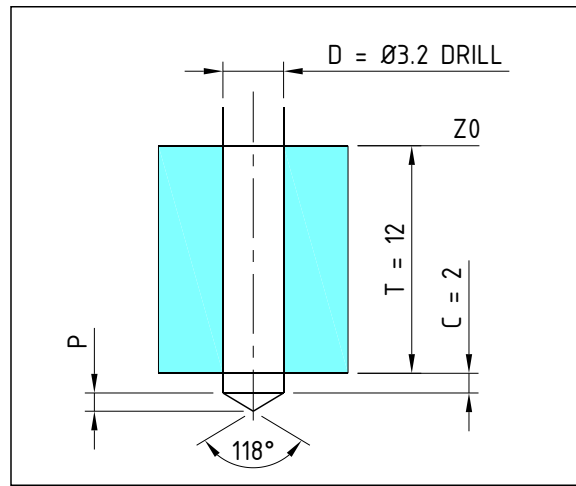
When using spot drills, always consider the fact that only a portion to the angular tool tip is used. That means the programmed depth controls the chamfer size - *or* - the chamfer size is controlled by the programmed depth. If we choose the chamfer size to be 0.35, that means the chamfer is applied on both sides of the hole centerline. The 4 mm hole with 0.35 mm per side chamfer will have a chamfer diameter of $0.35 + 4 + 0.35 = 4.7$ mm. The depth is calculated directly from the chamfer diameter and the tool point angle. Since the spot drill point angle is 90°, it means the programmed depth *must be* one half of the chamfer diameter:

$$\text{DEPTH OF SPOT DRILL} = (2 \text{ chamfer size} + \text{hole diameter}) / 2 = (2 \cdot 0.35 + 4) / 2 = 2.35 \text{ mm}$$

Drilling

To drill the six holes, the drill selection must be related to the tapping operation that follows. There is a correlation between the tap size and the pre-drill hole size. The drilled hole must be smaller than the nominal tap size, in order to provide material to be cut by the tapping operation. The best source of information about the tap drill selection can be found in various charts from tooling manufacturers, and also in the *Machinery's Handbook*, published by Industrial Press, Inc. In either source, the suggested drill selection - called the *tap drill* - can be found. For an M4x0.75 metric tap, the common selection of the tap drill size is 3.2 mm (or 3.25 mm).

The drawing calls for a through hole, drilled and tapped as per given drawing dimensions. The thickness of the part T at this stage is 12 mm (already faced), which means the drill has to penetrate the part thickness with its full body diameter. It is never a good idea to line the end of the full drill diameter D with the bottom of the part. In practice, the tap drill should penetrate not only the thickness of the part, but also provide additional 1-2 mm of breakthrough clearance. In addition to the part thickness T and the breakthrough clearance C , we also have to consider the drill point length (shown as P in the illustration). In the programming section, the drill data will be calculated, resulting in the Z-depth of the drill (its final position below the part).



As a conclusion to the tap drill selection, standard 3.2 metric drill will be used for the sample part.

Tapping

Once the drilling depth is established (through the part in this example), the tapping depth presents no problems, particularly for through holes. In fact, the final Z-depth for the drill may be used as the final tapping depth, eliminating additional calculations.

Tapping operations have other issues to consider, particularly the relationship between the spindle speed and the tap pitch. These two items will also influence the starting position before tapping begins, as well as the feedrate calculation. This subject will be discussed later in this chapter. In this example, a standard tapping head (tension/compression type) will be assumed. Another name for this type of tapping head is a *floating tap holder*. Its purpose is to prevent tap breakage, when the tap has reached the final depth, but the spindle still decelerates to a full stop. It also work in the opposite direction, when the return feedrate has already started, but the spindle has not completed its acceleration to the programmed spindle speed (r/min).

Summary of Tools Used

Tools used for this job have been carefully selected, based on several important considerations. Check again the details associated with each operation. There are other ways to machine this part - consider the presented method as only one of several possibilities. One of the programmer's responsibilities is to assign a tool number to each tool. Keep in mind, that ascending order from tool number 1 is not always practical, as many frequently used tools loaded in the machine tool magazine may use the same number from one job to another.

For this drawing example and the program development, the cutting tools will be numbered in their order of use, as shown in the following table, with particular descriptions:

Tool Number	Description	Size in mm	Type	Comments
T01	Face mill	100	5-6 cutting edges	Top of the face - one pass only
T02	Center-cutting end mill	12	3-flutes	Contour and circular pocket
T03	Center-cutting end mill	8	2 or 3 flutes	Slot - complete
T04	Spot drill	10	90 point angle	Chamfer 0.35 x 45
T05	Tap drill	3.2	118 point angle	Through the part thickness
T06	Plug tap	M4 0.75	High spiral flutes	Through the part thickness

Needless to say, a change in the drawing specifications may have a profound effect on the tooling selection. Changes in the setup will also have to be considered for the newly selected cutting tool, or adjusted as necessary. *RANDOM MEMORY TOOL CHANGE METHOD WILL USED.*

Machining Data

Machining data considerations cover a large and important area of program development. What is commonly referred to as selecting 'speeds and feeds' is much more than that. When the programmer reaches this stage, the following decisions have to be made - *for each tool* - if applicable:

- ◆ Spindle speed in r/min
- ◆ Cutting feedrate per minute
- ◆ Depth of cut
- ◆ Width of cut

There may be many other related decisions to be made, depending on the complexity of the part.

Spindle Speed

Most speeds are calculated from a standard machining formula, based on a peripheral surface speed per minute. For metric units, the surface speed is in *m/min*, for Imperial units it is in *ft/min*:

Metric		Imperial	
r/min	$\frac{\text{m/min}}{D} \cdot 1000$	r/min	$\frac{\text{ft/min}}{D} \cdot 12$

In both formulas, the parameter *D* refers to the diameter of the tool in milling, or the diameter of the part in turning, in millimeters or in inches. The surface speed for many materials can be found in various tooling catalogues and technical publications.

Cutting Feedrate

Feedrate can be calculated from the spindle speed, chipload per tooth and the number of cutting edges (teeth or flutes):

Feedrate calculation - Metric or Imperial
Feedrate = r/min C N

☞ ... where **C** is chipload per tooth in either inches or millimeters and **N** is the number of flutes (teeth)
or the number of cutting edges

Tooling Data

For the purpose of this example, the spindle speed and feedrate calculations will be made based on the following surface speeds and chipload for each machining group (higher rates are possible):

Tool Number	Description	Size in mm	Surface speed in m/min	Chipload per tooth in mm	Spindle r/min	Feedrate mm/min
T01	Face mill	100	150	0.35	477	501.0
T02	Center-cutting end mill	12	55	0.06	1459	175.0
T03	Center-cutting end mill	8	55	0.06	2188	263.0
T04	Spot drill 10 mm	4.7	25	0.08	1693	135.0
T05	Tap drill	3.2	25	0.07	2487	174.0
T06	Plug tap	M4 0.75	10	N/A	796	597.0

The feedrate for the face mill was calculated for three cutting edges in the material, and the feedrate for both end mills, for two flutes cutting. Tapping speed was calculated by multiplying the spindle speed by the thread pitch.

The cutting data presented here are only examples - always evaluate the actual work conditions

Details of Operations

In this section, each operation associated with a particular tool will be described in sufficient detail. Although all individual procedures are correct in principle, there are many other ways to produce the desired results. Hopefully, with the presentation of one method, you can adapt the acquired knowledge and develop another method. Actual program code will be generated for each tool.

The main goal of this chapter is not only to develop a working CNC program, but also describe and explain the steps required to complete this important task.

Tool 1 - Face Milling

The first part of this operation has been decided earlier, in the tooling selection. A 100 mm face mill with 5-6 cutting edges has been selected. In order to provide the best cutting conditions, the center of the face mill has been shifted by 15 mm, still leaving a 2.5 mm edge overlap.

In order to develop a part program for this tool, the start and end points of the facing cut must also be selected - or calculated. As a good machining practice, the top face surface finish will be of better quality, if the cutter starts and ends 'in the air' - *away from the part*. Selecting a 5 mm clearance from the two opposite edges is arbitrary, but reasonable. The X-coordinate at the start point (P1) will be:

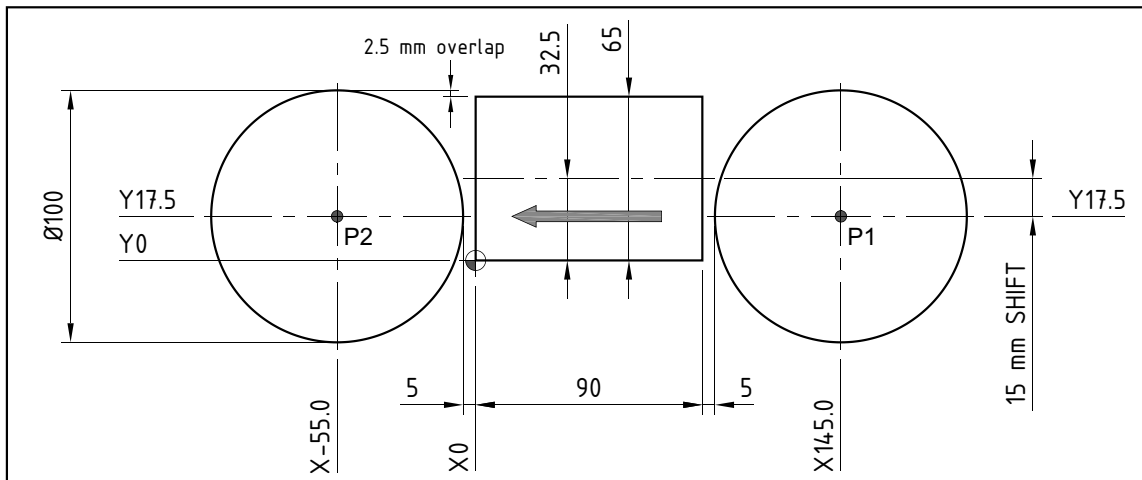
$$\text{X-coordinate of P1} = 90 + 5 + 50 = \text{X145.0 (part length + clearance + cutter radius)}$$

At the end point (P2), the calculation is similar, but does not include the part length:

$$\text{X-coordinate of P2} = -5 + 50 = \text{X-45.0 (clearance + cutter radius)}$$

The Y-coordinate is the same for both points. Based on the 15 mm shift and 32.5 mm half width of the part, the Y-coordinate is:

$$\text{Y-coordinate (P1 and P2)} = 65/2 - 15 = \text{Y17.5}$$

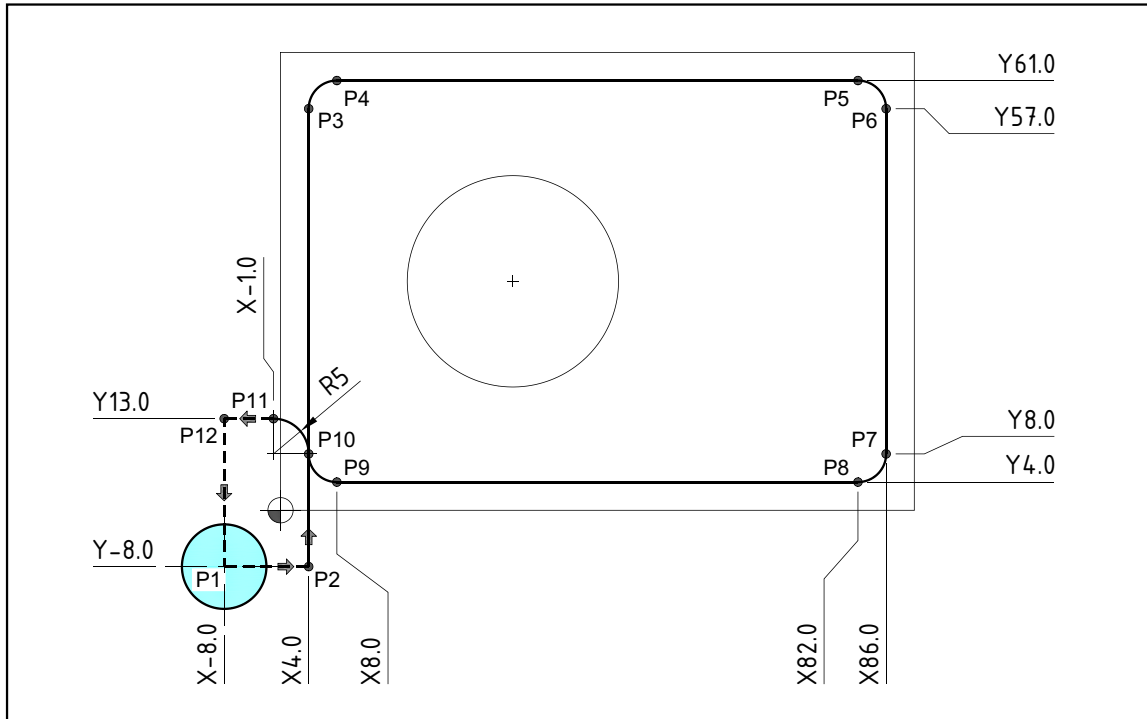


The program for this tool can now be written. This is the first tool of the program, and the format structure will reflect that:

```
(T01 - 100 MM FACE MILL - 1 MM OFF THE TOP FACE)
N1 G21
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X145.0 Y17.5 S477 M03 T02
N5 G43 Z10.0 H01 M08
N6 Z0
N7 G01 X-55.0 F501.0
N8 G00 Z10.0 M09
N9 G28 Z10.0 M05
N10 M01
```

Tool 2 - Outside Contour

Tool 2 is a 12 mm center-cutting end mill. It will be used for two operations - the contour and the pocket. The first activity of Tool 2 is to machine the outside contour.



The calculation of individual end points on the contour itself should present no problems - the 4 mm radius is constant around the contour. Study the lower left corner area of the illustration, and you will see that some heavy activity takes place right there. This is the area where the cutter is first positioned, where it approaches the part contour (a motion called *lead-in*) and also where it leaves the contour (a motion called *lead-out*). Because the contour is closed, the lead-in and lead-out motions are close together. In an open contour, they may be much further apart from each other, although the basic principles will not change.

When a large number of points exists in a single contour, making a chart or a table of points and their coordinates will make the program development much easier and better organized. It is always a good idea to define all points in the order of machining, so the coordinates can be easily transferred from the table to the program:

Pt	X	Y	Pt	X	Y	Pt	X	Y
P1	X-8.0	Y-8.0						
P2	X4.0	Y-8.0	P6	X86.0	Y57.0	P10	X4.0	Y8.0
P3	X4.0	Y57.0	P7	X86.0	Y8.0	P11	X-1.0	Y13.0
P4	X8.0	Y61.0	P8	X82.0	Y4.0	P12	X-8.0	Y13.0
P5	X82.0	Y61.0	P9	X8.0	Y4.0	P1	X-8.0	Y-8.0

Once the coordinate points and their machining order has been established, the program for the contouring operation can be written (*e/mill* is the same as *end mill*):

```
(T02 - 12 MM CENTER-CUTTING E/MILL)
(OUTSIDE CONTOUR CUTTING - D52 = 6.000)
N11 T02
N12 M06
N13 G90 G54 G00 X-8.0 Y-8.0 S1459 M03 (P1)
N14 G43 Z10.0 H02 M08
N15 Z-3.5
N16 G41 G01 X4.0 D52 F175.0 (P2)
N17 Y57.0 (P3)
N18 G02 X8.0 Y61.0 I4.0 J0 (P4)
N19 G01 X82.0 (P5)
N20 G02 X86.0 Y57.0 I0 J-4.0 (P6)
N21 G01 Y8.0 (P7)
N22 G02 X82.0 Y4.0 I-4.0 J0 (P8)
N23 G01 X8.0 (P9)
N24 G02 X4.0 Y8.0 I0 J4.0 (P10)
N25 G03 X-1.0 Y13.0 I-5.0 J0 (P11)
N26 G00 X-8.0 (P12)
N27 G40 Y-8.0 (P1)
N28 Z2.0
<machining will continue for circular pocket>
```

The important programming features have been used in the above program section:

- ◆ Cutter radius offset
- ◆ Numbering of offsets

The most important programming feature for this toolpath was the use of drawing dimensions for the programmed contour and combining it with the cutter radius offset (G41 ... D52). When a part program contains cutter radius offset, the CNC operator enters the tool radius amount into the control registry and lets the computer do all calculations. There are several do's and don'ts, but overall it is quite an easy way to develop a part program. The operator must know how the toolpath was generated - in typical manual programming, the drawing dimensions are used. In this case, the nominal amount for the offset is the cutter radius (D52 = 6.000). Typical to many CAD/CAM systems, the program output may be to the center of the cutter. In this case, the nominal amount of the offset will be zero (D52 = 0.000). It is always a good idea to include the *suggested* offset amount in the program itself - see the above example.

This part of the program uses tool 2 (T02). As each tool also requires tool length offset, it makes sense to assign the tool length offset the same number the tool number (H02). On the other hand, not all tools in the program use the cutter radius offset. If the tool does require cutter radius offset, the address *D* must be programmed, also with the offset number. That presents a difficulty if the control system has only a single memory bank for *both* types of offsets, so called *shared memory*. In this case, the programmer usually shifts the number by an arbitrary amount (such as 50 for D52). The shift amount can be different, but it should always be higher than the number of tools that can be stored in the magazine. The amount of 50 is also about half way between the range of offsets, usually available in the range of 01-99.

The program also includes a lead-in motion (block N16) and lead-out motions (N25-N27). G41 specifies that radius offset will be to the left of cutting direction, providing a climb milling action.

Cutter radius offset cannot be started or canceled while an arc motion is in effect

Tool 2 - Circular Pocket

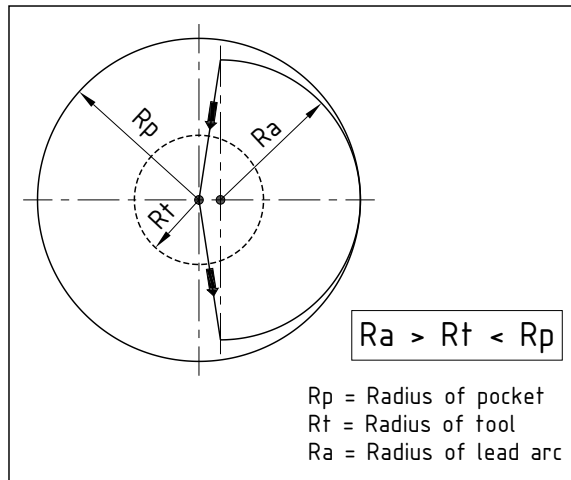
The second activity of Tool 2 is to machine the circular pocket. From its last position in block N28, the tool will move towards the pocket center. Circular and other symmetrical pockets are much easier to program when the first cut starts at the pocket center. The cutter diameter of 12 mm has been selected for one important reason - it can cut the pocket in just a single cut around and still maintain a clean bottom. This cut will be preceded by a lead-in, and followed by a lead-out. Cutter radius offset will be applied and canceled during a linear motion.

When machining a circular pocket, the relationship between the pocket size, tool size, and lead-in/out arc is very important. In the illustration at right, the relationship is shown.

Also shown is the direction of the cut, starting from the pocket center. The toolpath is simple, starting at the center:

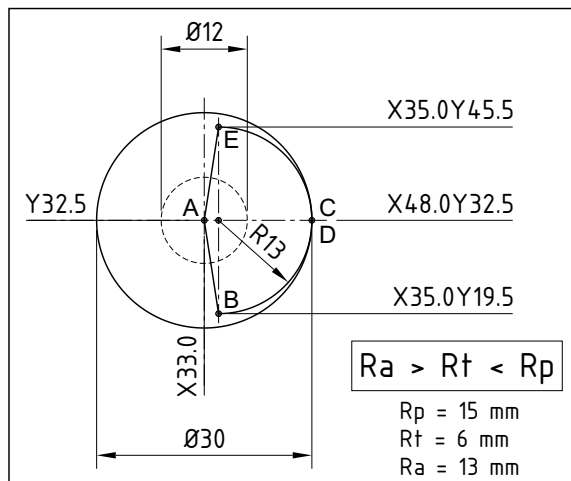
- 1 - Lead-in line - G41 G01 in effect
- 2 - Lead-in arc - G03 with 90° sweep
- 3 - Full circle to cut the pocket - G03
- 4 - Lead-out arc - G03 with 90° sweep
- 5 - Lead-out line - G40 G01 in effect

This procedure will be applied to the actual program (continued):



```
(POCKET CUTTING - D62 = 6.000)
N29 X33.0 Y32.5 (A)
N30 G01 Z-5.0 Z100.0
N31 G41 X35.0 Y19.5 D62 F175.0 (B)
N32 G03 X48.0 Y32.5 I0 J13.0 (C)
N33 I-15.0 (D)
N34 X35.0 Y45.5 I-13.0 J0 (E)
N35 G40 G01 X33.0 Y32.5 (A)
N36 G00 Z10.0 M09
N37 G28 Z10.0 M05
N38 M01
```

Note that the cutter radius offset has been changed - *for the same tool!* Instead of D52 that controls the outside contour, D62 controls the final size of the pocket. The radius amount stored in the control could be the same for both applications, but the operator can fine tune one without affecting the other.



When selecting the lead-in/out arc, first watch the rule of relationship as shown in the illustrations. For the example in this chapter, the *Ra* (lead arc radius) must be larger than *Rt* (tool radius). Based on the pocket radius *Rp*, the range should be between 6 mm and 15 mm. In order to achieve better machining results, selecting a larger radius provides better results, as the tangential entry into the pocket diameter is much smoother. If a tolerance is given on the pocket diameter and/or depth, two cuts would be necessary, perhaps even two tools. However, the programming procedure described here will remain exactly the same.

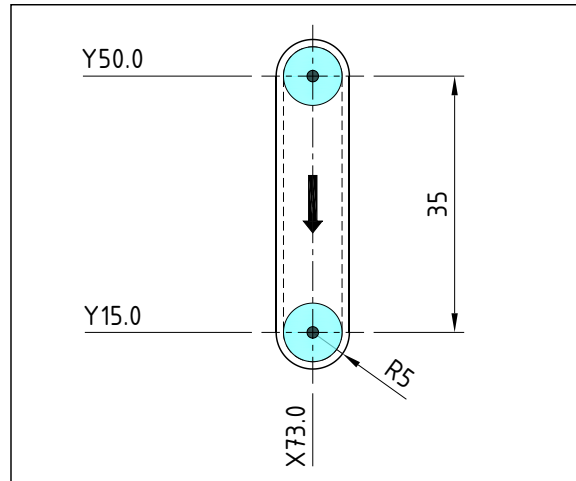
Tool 3 - Slot Milling

Tool 3 is an 8 mm center-cutting end mill. It will be used for roughing and finishing of the vertical slot. The approach towards the finish pass is very much similar to the approach described for the circular pocket, also in climb milling mode. The major difference is that the radius of the tool and the lead arc radius will be much closer together, details are described for the finishing cut.

First, the roughing toolpath - this one cannot be any simpler. The end mill will make a rapid move to the center of one slot radius in XY axes, feeds to the full depth of 3 mm and makes a straight cut to the center of the opposite radius.

The question 'which slot end should I start from?' is unnecessary - start from either end, it makes no difference in machining at all. In the example, the tool will rough the slot from its upper position to its lower position, as shown at right:

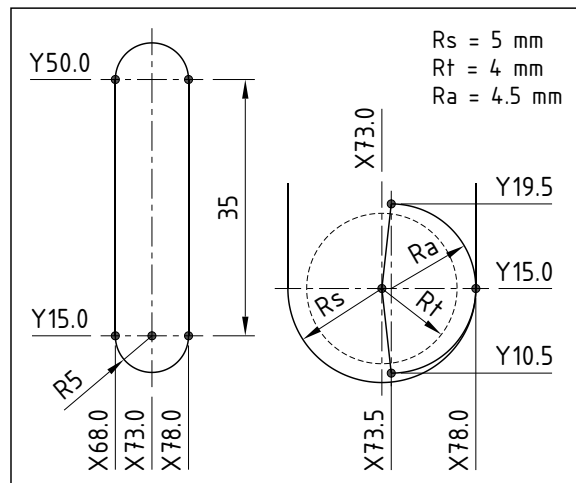
```
(T03 - 8 MM CENTER-CUTTING E/MILL)
(D53 = 4.000)
N39 T03
N40 M06
N41 G90 G54 G00 X73.0 Y50.0 S2188 M03 T04
N42 G43 Z10.0 H03 M08
N43 Z2.0
N44 G01 Z-3.0 F100.0
N45 Y15.0 F263.0
```



... will continue for slot finishing

Radius of the tool R_t is 4 mm, radius of the slot R_s is 5 mm. As the lead arc R_a must be somewhere between these two values, there is not much flexibility here. For the example in this chapter, the lead arc radius R_a will be 4.5 mm. The programmed toolpath will follow the same process already established for the circular pocket toolpath, but instead of the pocket being machined, it will be the slot:

```
(SLOT FINISHING)
N46 G41 X73.5 Y10.5 D53
N47 G03 X78.0 Y15.0 I0 J4.5
N48 G01 Y50.0
N49 G03 X68.0 I-5.0 J0
N50 G01 Y15.0
N51 G03 X78.0 I5.0 J0
N52 X73.5 Y19.5 I-4.5 J0
N53 G40 G01 X73.0 Y15.0
N54 G00 Z10.0 M09
N55 G28 Z10.0 M05
N56 M01
```

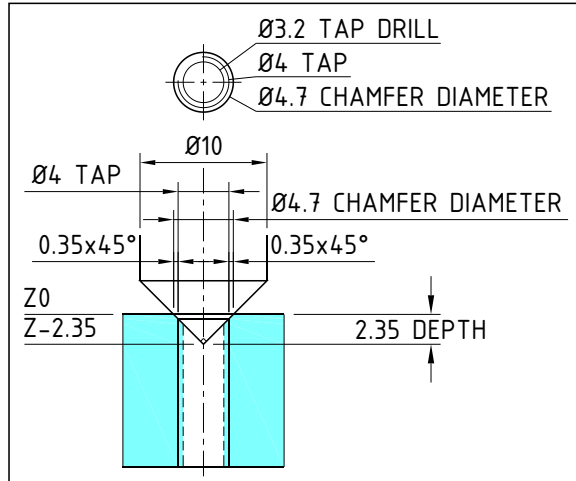


As for the pocket cutting, two tools may be used for controlling the precision of the slot, particularly if tight tolerances are required. The programming technique will not change.

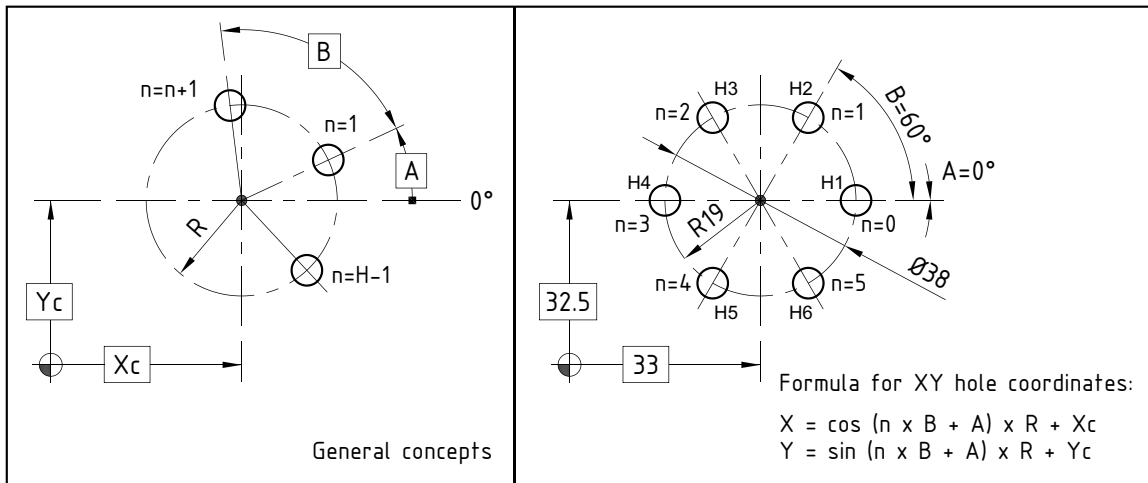
Tool 4 - Spot Drilling

There are two major differences between a standard drill and a spot drill. One is in the tool design, the other in the way the tool is used. The design between the two drill types affects the flutes, the web thickness, the overall length, and the tool point angle. The way how the two types of drill are used is a major consideration when programming. In the earlier section covering the tool selection, the details for using the spot drill for this example has already been covered, and the illustration is presented as a reference.

In a summary, the spot drill depth of each hole spotted will be Z-2.35, at the calculated XY location of each hole.



Apart from the depth of cut, another critical part of programming a spot drill is to calculate the XY coordinates of the six holes, which means a good calculator will be needed. Once these coordinates are established, they will also be used for the drilling and tapping of this bolt hole pattern.



On the left of the above illustration is a general concept of calculating bolt circle hole locations. On the right, the concept is applied for this particular example. Using the formula shown, the XY coordinates of all six holes can be defined:

H1 (X)=cos (0 60 0) 19 33 = X52.0	H1 (Y)=sin(0 60 0) 19 32.5 = X32.5
H2 (X)=cos (1 60 0) 19 33 = X42.5	H2 (Y)=sin(1 60 0) 19 32.5 = X48.954
H3 (X)=cos (2 60 0) 19 33 = X23.5	H3 (Y)=sin(2 60 0) 19 32.5 = X48.954
H4 (X)=cos (3 60 0) 19 33 = X14.0	H4 (Y)=sin(3 60 0) 19 32.5 = X32.5
H5 (X)=cos (4 60 0) 19 33 = X23.5	H5 (Y)=sin(4 60 0) 19 32.5 = X16.046
H6 (X)=cos (5 60 0) 19 33 = X42.5	H6 (Y)=sin(5 60 0) 19 32.5 = X16.046

Once the coordinates are known (depth is known already), the spot drilling program can be written:

```

(T04 - 10 MM SPOT DRILL - CHAMFER DIAMETER = 4.7)
N57 T04
N58 M06
N59 G90 G54 G00 X52.0 Y32.5 S1693 M03 T05
N60 G43 Z10.0 H04 M08
N61 G99 G82 R2.0 Z-2.35 P200 F135.0      (H1)
N62 X42.5 Y48.954                        (H2)
N63 X23.5                                  (H3)
N64 X14.0 Y32.5                            (H4)
N65 X23.5 Y16.046                          (H5)
N66 X42.4                                  (H6)
N67 G80 G00 Z10.0 M09
N68 G28 Z10.0 M05
N69 M01

```

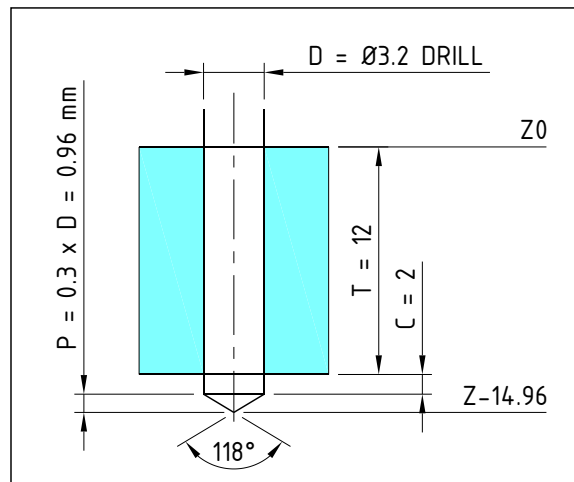
G82 fixed cycle has been used for the spot drill operation. This cycle is very similar to the G81 drilling cycle (see next operation), but it requires a dwell. The purpose of dwell is to pause at the bottom of the hole, before retracting to the clear position - *the reason?* In order to make sure the surface of the spot drilled hole is smooth, the tool has to rotate at least one spindle revolution, to make it clean. To achieve this goal, the following formula calculates the minimum dwell for fixed cycles - units are in milliseconds and there are no decimal places when milliseconds are used (1 sec = 1000 ms):

$$\text{Minimum dwell (ms)} = (60 \quad 1000) / r/\text{min}$$

In the program, the spindle speed is 1693 r/min, so the minimum dwell in milliseconds is 35.44 ms. Although that is the minimum dwell mathematically, practically we have to look at another possible situation, and that is the status of the spindle override switch, located at the control panel. On the majority of CNC machines, this switch has the range of 50-120%. For the dwell calculation, the concern should be with the lowest setting. In order to guarantee at least one full revolution of the spot drill, even at 50% setting, the minimum dwell has to be doubled. In the example, it would be 35.44 ms \times 2, which is 70.88 ms. Doubling or even tripling this time will result in more revolutions at the bottom of the cut. The 200 ms in the program will guarantee almost three revolutions at 50% spindle override.

Tool 5 - Drilling

Most of the drilling parameters have been established earlier, in the tool selection section of this chapter. Additional programming decisions focus on the method of drilling the hole - whether a single cut is sufficient for the depth required, or several interrupted cuts are needed. Interrupted drilling - also called *peck drilling* - is not needed in this example. For holes that penetrate the material - so called through holes - the program must provide a breakthrough clearance C (2 mm in the example). In addition, the tool point length P (also known as the drill point length), has to be calculated. Constant of 0.3 is used for standard twist drills, with 118° point angle. Adding the breakthrough clearance C and the drill point length P to the thickness of the part T , the final drilling depth can be calculated:



Z-depth = $T \quad C \quad P = 12 \quad 2 \quad 0.96 = 14.96 = Z-14.96$ in the program

The drilling program can be written, using the calculated values and the previous XY coordinates:

```
(T05 - 3.2 MM TAP DRILL - THROUGH)
N70 T05
N71 M06
N72 G90 G54 G00 X52.0 Y32.5 S2487 M03 T06
N73 G43 Z10.0 H05 M08
N74 G99 G81 R2.0 Z-14.96 F174.0           (H1)
N75 X42.5 Y48.954                       (H2)
N76 X23.5                                 (H3)
N77 X14.0 Y32.5                           (H4)
N78 X23.5 Y16.046                         (H5)
N79 X42.4                                 (H6)
N80 G80 G00 Z10.0 M09
N81 G28 Z10.0 M05
N82 M01
```

At the machine, the CNC operator can make some changes to the program, as needed - for example, the G81 cycle can be changed to G83 or G73 peck drilling cycle, just by changing the cycle number and adding the Q-amount of each peck. Here is a variation on the above program (cycle call only):

```
N74 G99 G83 R2.0 Z-14.96 Q5.0 F174.0     (H1)
```

The depth of each peck will be 5 mm. Working with fixed cycles for machining holes offers a generous amount of flexibility - in programming as well as at the machine.

Tool 6 - Tapping

Once the spot drill and drill calculations are out of the way, the tapping is quite simple. There is no need to calculate the XY hole locations (already done for the spot drill). There is also no need to calculate the final tapping depth - the depth already calculated for the drill will be suitable for tapping as well. So - what considerations are unique to tapping? *Feed level clearance and the feedrate!*

```
(T06 - M4x0.75 TAP DRILL - THROUGH)
N83 T06
N84 M06
N85 G90 G54 G00 X52.0 Y32.5 S796 M03 T01
N86 G43 Z10.0 H06 M08
N87 G99 G84 R5.0 Z-14.96 F597.0         (H1)
N88 X42.5 Y48.954                       (H2)
N89 X23.5                                 (H3)
N90 X14.0 Y32.5                           (H4)
N91 X23.5 Y16.046                         (H5)
N92 X42.4                                 (H6)
N93 G80 G00 Z10.0 M09
N94 G28 Z10.0 M05
N95 G28 X42.4 Y16.046
N96 M30
%
```

Note the increased feed level clearance (R-level) - the increase is an adjustment needed to absorb feed acceleration before the tool touches the part, due to heavy feedrate. The feedrate itself is always a combination of two related items - spindle speed and tap pitch. The tapping feedrate is:

Tapping feedrate = r/min tap pitch = 796 0.75 = 597.0 = F597.0 (underfeeding may be applicable)

That concludes the chapter on part program development. Complete program is listed next.

Complete Program

```

(T01 - 100 MM FACE MILL - 1 MM OFF THE TOP FACE)
N1 G21
N2 G17 G40 G80 T01
N3 M06
N4 G90 G54 G00 X145.0 Y17.5 S477 M03 T02
N5 G43 Z10.0 H01 M08
N6 Z0
N7 G01 X-55.0 F501.0
N8 G00 Z10.0 M09
N9 G28 Z10.0 M05
N10 M01

(T02 - 12 MM CENTER-CUTTING E/MILL)
(OUTSIDE CONTOUR CUTTING - D52 = 6.000)
N11 T02
N12 M06
N13 G90 G54 G00 X-8.0 Y-8.0 S1459 M03 (P1)
N14 G43 Z10.0 H02 M08
N15 Z-3.5
N16 G41 G01 X4.0 D52 F175.0 (P2)
N17 Y57.0 (P3)
N18 G02 X8.0 Y61.0 I4.0 J0 (P4)
N19 G01 X82.0 (P5)
N20 G02 X86.0 Y57.0 I0 J-4.0 (P6)
N21 G01 Y8.0 (P7)
N22 G02 X82.0 Y4.0 I-4.0 J0 (P8)
N23 G01 X8.0 (P9)
N24 G02 X4.0 Y8.0 I0 J4.0 (P10)
N25 G03 X-1.0 Y13.0 I-5.0 J0 (P11)
N26 G00 X-8.0 (P12)
N27 G40 Y-8.0 (P1)
N28 Z2.0
(POCKET CUTTING - D62 = 6.000)
N29 X33.0 Y32.5 ( A )
N30 G01 Z-5.0 Z100.0
N31 G41 X35.0 Y19.5 D62 F175.0 ( B )
N32 G03 X48.0 Y32.5 I0 J13.0 ( C )
N33 I-15.0 ( D )
N34 X35.0 Y45.5 I-13.0 J0 ( E )
N35 G40 G01 X33.0 Y32.5 ( A )
N36 G00 Z10.0 M09
N37 G28 Z10.0 M05
N38 M01

(T03 - 8 MM CENTER-CUTTING E/MILL)
(D53 = 4.000)
N39 T03
N40 M06
N41 G90 G54 G00 X73.0 Y50.0 S2188 M03 T04
N42 G43 Z10.0 H03 M08
N43 Z2.0
N44 G01 Z-3.0 F100.0
N45 Y15.0 F263.0
(SLOT FINISHING)
N46 G41 X73.5 Y10.5 D53
N47 G03 X78.0 Y15.0 I0 J4.5
N48 G01 Y50.0
N49 G03 X68.0 I-5.0 J0
N50 G01 Y15.0
N51 G03 X78.0 I5.0 J0
N52 X73.5 Y19.5 I-4.5 J0
N53 G40 G01 X73.0 Y15.0

```

```

N54 G00 Z10.0 M09
N55 G28 Z10.0 M05
N56 M01

(T04 - 10 MM SPOT DRILL - CHAMFER DIAMETER = 4.7)
N57 T04
N58 M06
N59 G90 G54 G00 X52.0 Y32.5 S1693 M03 T05
N60 G43 Z10.0 H04 M08
N61 G99 G82 R2.0 Z-2.35 P200 F135.0 (H1)
N62 X42.5 Y48.954 (H2)
N63 X23.5 (H3)
N64 X14.0 Y32.5 (H4)
N65 X23.5 Y16.046 (H5)
N66 X42.4 (H6)
N67 G80 G00 Z10.0 M09
N68 G28 Z10.0 M05
N69 M01

(T05 - 3.2 MM TAP DRILL - THROUGH)
N70 T05
N71 M06
N72 G90 G54 G00 X52.0 Y32.5 S2487 M03 T06
N73 G43 Z10.0 H05 M08
N74 G99 G81 R2.0 Z-14.96 F174.0 (H1)
N75 X42.5 Y48.954 (H2)
N76 X23.5 (H3)
N77 X14.0 Y32.5 (H4)
N78 X23.5 Y16.046 (H5)
N79 X42.4 (H6)
N80 G80 G00 Z10.0 M09
N81 G28 Z10.0 M05
N82 M01

(T06 - M4x0.75 TAP DRILL - THROUGH)
N83 T06
N84 M06
N85 G90 G54 G00 X52.0 Y32.5 S796 M03 T01
N86 G43 Z10.0 H06 M08
N87 G99 G84 R5.0 Z-14.96 F597.0 (H1)
N88 X42.5 Y48.954 (H2)
N89 X23.5 (H3)
N90 X14.0 Y32.5 (H4)
N91 X23.5 Y16.046 (H5)
N92 X42.4 (H6)
N93 G80 G00 Z10.0 M09
N94 G28 Z10.0 M05
N95 G28 X42.4 Y16.046
N96 M30
%
```

In this introductory chapter, you have learned many CNC programming techniques. They all can be easily adapted to a large number of programs. It is always difficult to put something in print, when so many variations exist. For example, the speeds and feeds used in this chapter may prove to be too low for a production of a large volume of parts. You may also find that your way of machining may be better than the one shown here. That is all to be expected. After all, CNC programming is almost like following a recipe - the ingredients are there, even the process - but it still needs the skilled hand of the cook - *the CNC programmer* - to make all the elements work well together.

This page is intentionally blank

INDEX

!	
# symbol	318
A	
Acceleration	143,231
ACOS function	324
Angular head setting	303
APC	283
ASIN function	324
ATAN function	324
Automatic corner breaking	185
Direction specification	186
Automatic corner override	100
Automatic Pallet Changer - APC	283
Axis substitution	292
B	
Back angle clearance	170
Backlash	97
Backlash compensation	97
Ball nose end mills	297
B-axis	285
Bill of Materials	3
Billet stock	139
Block delete function	131
Block skip function	131-142
Block skip within a block	140
Irregular stock removal	137
Numbering block skips	142
ON and OFF modes	131
Similar parts application	132
Slash symbol	131
Switch setting	133
Trial cut application	134
Used within a block	140
Bolt hole pattern	152
Boring	
Offset errors	65
Box threading cycle	240
Breakthrough clearance	18
Bull nose end mills	297
C	
Calculation zones	27
Cams	307
Angle orientation	309
Applications	307
Best curve	309
Cam zero	309
Contour spline	309
Events of a cam cycle	307
Motion transfer	307
Origin	309
Overview	307
Spline approximation	309
Center-cutting end mills	6
Chamfers	206
Chuck pressure	192
Circular pocket	15
CNC vise	3
Command point	172,216
Comment	183
Common variables	322
Conflicting words in a block	140
Conical thread	271
Constant spindle speed	278
Constant surface speed	278
Constants	319-320
Contour lead-in and lead-out	49,176-181
Contour Point Between Line and Arc	34
Contour Point Between Two Arcs	40
Contour Point Between Two Lines	33
Contour points	23-32
Formulas for calculations	33-42
Intersecting arcs	40
Intersecting point	34
Sharp point calculation	39
Tangent arcs	41
Tangent point	35
Control registry (offsets)	45
Corner breaking	185
Corner radius	170
Corner radius selection	170
COS function	324
CRC interference	60
Cut-Off	221
Cutter radius offset	14,43-66
Benefits	44
Commands G40-G41-G42	45
D-address	45-47
Drawing dimensions	48
Equidistant toolpath	48
Error handling	60
Excessive cutting	61,172
General concepts	43
Insufficient cutting	64,172
Lead-in and lead-out	49
Line-Arc lead-in and lead-out	52
Line-Line lead-in and lead-out	49
Maintaining tolerances	58
Missing axis motion	52
Offset activation	46
Offset application	47
Offset cancellation	47
Programming techniques	46
Radius vs diameter	48
Tool nose radius	62
Cycle start	318
D	
D-address	45-48
Datum shift	85

Deceleration	143,231
Dimensions	2
Distance-to-go	60
Drawing units	2
Drill point length	18
Drilling	9,18
Peck drilling	18
Dwell	18
Minimum dwell	18

E

Effective cutting diameter	91
Effects of plane selection	291
Equidistant toolpath	44-45
Exact stop check	98-99
Exact stop check mode	98-99
External cutting	197

F

Face cut	134
Multicut facing	184
Offset errors	64
Face milling	
Cutting direction	5
FALSE values	326
Fanuc Macro B	126
Fanuc User Macros B	146,251,317
Feedrate	11
Fixed cycles	
R-level	134
Fixed cycles in planes	301
Depth	303
Initial level	303
R-level	303
Using G81 drilling cycle	303
Floating tap holder	143
Formulas in calculations	33
Four-axis lathe	
General setup	277
Program structure	279
Programming method	278
Special M-functions	279
Spindle speed and feedrate	278
Tool tip orientation numbers	278
Waiting codes	279
Functions	319-320

G

G00 command	52
G01 command	52
G02 command	295-300
G03 command	295-296,298-300
G09 command	97-99
G10 command	85,126-127
G15 command	151
G16 command	151
G17 command	290-292,294-296,301-302

G18 command	290,292-293,295-298,301,304
G19 command	290,294-296,299-301,303,305-306
G22 command	97,101-102
G23 command	97,101-102
G25 command	97,103-104
G26 command	97,103-104
G27 command	97,104,106-108,114
G28 command	97,104
G29 command	97,104,106
G30 command	97,104,108
G31 command	97,118
G32 command	236-238
G40 command	45-48
G41 command	45-48,52,295-300
G42 command	45-48,52,295-300
G43 command	119,122,126
G44 command	119
G49 command	119-122
G50 command	97,108-109,111-118
G52 command	81,129
G60 command	97
G61 command	97-99,101
G62 command	97-99
G63 command	97-100,145
G64 command	97-101
G65 command	320
G70 command	169,193,196
G71 command	169,173,193
G72 command	169,173,193-194
G73 command	197
G75 command	217
G76 command	241
G81 command	302,304
G81command	303
G92 command	97,108-110,240
G96 command	278
G97 command	231,278
G98 command	133-134
G99 command	133-134
G-codes	97
Geometry offset	62-63,126
Grinding allowance	174
Grooving operations	
Command point	216
Cutting depth	215
Cutting width	215
Deep groove	221
Face grooving	226
General topics	215
Groove location	216
Grooves with tapers	222
Grooving for precision	218
Offset errors	66
Part-Off	221
Plunge and retract	217
Pulley grooves	225

H

Hard turning	175,198
------------------------	---------

-
- Headstock 186
 Helical interpolation 251
 Availability 252
 Helix 258
 Hole chamfering 8
-
- I**
-
- Imaginary tool point 171
 Indexing axis 285
 Inner plug for tubular stock 192
 Inscribed circle 189-190
 Insert back angle 198
 Insert lead angle 198
 Insufficient clearance 64
 Internal cutting 197
 Intersecting arc 178
-
- K**
-
- Knurling 273-276
 Depth and feedrate 275
 Knurling pitch 274
 Programming and machining 275
 Tool motions 275
 Troubleshooting 276
 Types of knurl 273
-
- L**
-
- Lathe cycles
 P and Q blocks 194-195
 Lathe jaws 182
 Hard jaws 182
 Soft jaws 182
 LE function 326
 Lead 244
 Lead error 249
 Lead-in motion 13,46,49,176-179,259
 Lead-out motion 13,46,179-181,259
 Left hand threads 232
 Local coordinate offset 129
 Local coordinate system 81
 Local variables 322
 Logical functions 319
 AND 326
 EQ 326
 GE 326
 GT 326
 LE 326
 LT 326
 NE 326
 OR 326
 XOR 326
-
- M**
-
- M00 function 183
 Machine zero 114
 Machine zero commands 104
 Machine zero position 116
 Machining corners 185
 Machining operations 4
 Machining thin stock
 Adjusting chuck pressure 192
 Using an inner plug 192
 Using special split jaws 192
 Macro in main program 319
 Macro programming 317
 Macros 251
 Arguments 320
 Bolt hole circle pattern 327
 Bolt hole example 329
 Bolt hole pattern example 327
 Branching and looping 320,326
 Evaluation of drawings 328
 Features and applications 318
 Functions and constants 320
 Introduction to macros 317-332
 Local variables 321
 Logical functions 320,326
 Macro call 320
 Macro functions 323
 Rounding functions 324
 Skills required 317
 Variable declarations 322
 Variables 320
 Macros for machining 329
 Main program 319
 Manual Data Input (MDI) 318
 Maximum feedrate 244
 Memory registers 45
 Metric thread form 227
 Metric threads 229
 M-Functions 279
 Milling threads 251
 Modal commands 133
 Multicut facing 184
 Multiple repetitive cycles 193,227
 External cutting 197
 G70 cycle 196
 G71 and G72 compared 196
 G72 cycle 194
 Internal cutting 197
 Pattern repeating cycle 197
 Programming formats 193
-
- N**
-
- Nominal dimensions 58
 Normal cutting mode 99
 Null variables 322
-
- O**
-
- Optional block skip 142
 O-ring grooves 222
 Overcutting error 60
 Overheat alarm 103

P	
Palletization	284
Pallets	283
Transfer methods	283
Types of pallets	283
Parametric programming	317,327-330
Part orientation	3
Part reversal - lathe	182-183
Part reversal - mill	67-88
Machining process	69
Program zero selection	70
Programming methods	72
Tool length settings	72
Using work offset	74
Part zero	3-4
Partial arc	35,178
Part-Off	221
Pitch	229,244
Plane selection	251
Planes	
Angular head setting	303
Circular motion	291-292
Circular motions	296
Cutter radius offset	293
Effect of planes in programming	291
Machine planes	290
Mathematical definition	289
Preparatory commands	290
Side face drilling	302
Side face milling	304
Planes and fixed cycles	301
Pocketing	
Finishing rectangular pocket	56
Polar coordinate system	149-154
G-codes	149
Planes	150,154
Programming format	151
Toolpath direction	153
Program development	1
Drawing evaluation	2
Machining data	10
Material and stock	3
Part setup	3
Selecting part zero	4
Tooling selection	4
Program stop	183
Program zero selection	169
Pulley grooves	
Depth calculation	226
Insert selection	225
Tool setup	226
Pythagorean Theorem	34,41-42,178
R	
Rear type CNC lathes	62
Recess programming	198
Rectangular coordinate system	151
References and resources	335-338
Revisions	2
Right hand threads	232
Right-angle head	301
Rigid tapping	146
Benefits	146
Possible problems	147
Programming approach	147
Special functions	148
Spindle speed	147
R-level	134
ROUND function	324
S	
SIN function	324
Single direction positioning	97
Skip command	118
Slot machining	
Circular slot	55
Linear slot	54
Slot milling	7,16
Slot width	7
Solving triangles	35
Special cutting modes	98
Special purpose G-codes	97-122
Spherical radius	178
Spindle fluctuation	103
Spindle override switch	18
Spindle speed	10
Split soft jaws	192
Spot drilling	8,17
Stepover	162
Stock allowance	172
Allowance for grinding	174
Compound stock	174
Depth of cut calculation	174
Lead angle	173
U and W programming	173
Stored stroke limits	101
Subprogram	
Cutting Tool Selection	157
Definition and usage	155
Depth control	157,164
Development	158
Drawing evaluation	156
Planning	156
Roughing and finishing	165
Round pocket	162
Stepovers	162
Width of cut control	157
Surface finish	2
Synchronized tapping	146
System variables	322
T	
Table clamp and unclamp	285
Tailstock	186
Using tailstock	187
TAN function	324
Tap drill	9
Tapered end mills	89-95
Ball end	91,94
Descriptions	89
Effective cutting diameter	91

- Flat end 90-91
 Tapered holes 96
 Tapered holes 96
 Tapered thread 271
 Tapered walls 89
 Tapers 199-214
 Lead-in - Lead-out 211
 Taper angle 206
 Taper as a percentage 205
 Taper per foot 199-200
 Taper ratio 199,203
 Tapping 9,19
 Feedrate reduction (underfeeding) 144
 Rigid tapping method 146
 Standard tapping method 143
 Tap holders 143
 Thread stripping 144
 Tapping feedrate 19
 Tapping mode (G63) 145
 Tapping operation 143
 Tapping with G32 238
 Thread depth 229
 Thread depth calculation 229
 Thread hob 260
 Thread milling 251
 Benefits 252
 Cutter radius offset 262
 Cutting direction 255
 External and internal 255
 External threads 260
 Helical interpolation 251
 Helix 258
 Internal threads 266
 Lead-in and lead-out motions 261
 Selection of tools 253
 Simulated toolpath 251
 Thread milling in planes 251
 Thread milling software 272
 Z-depth calculations 263
 Threading
 Depth calculation 229
 Depth calculation constants 229
 Depth of thread calculation 229
 Distribution of depth cuts 236
 Hand of thread 231-232
 Imperfect thread 231
 Infeed angle 237-238
 Infeed methods 230
 Lead error 249
 Lead vs. pitch 244
 Long thread programming 248
 Restrictions in threading 243
 Special threads 228
 Thread chamfering 231
 Thread forms 227
 Threading feedrate 244
 Threading passes 235
 TIR 147
 Title block 2
 Tolerances 2
 Tolerances in programming 58
 Tool change position 111-114
 Tool length offset 45,119
 Tool length offset cancel 119
 Tool length setting
 Offset registry 45
 Touch-off method 4
 Tool life 58
 Tool nose radius offset 62
 Tool point length 18
 Tool tip orientation 44,62-63
 Tooling for grooves 215
 Total indicator reading 147
 Touch-off method of tool length 123
 TPI - Threads Per Inch 229
 Trial cut 134
 Milling application 134
 Turning application 136
 TRUE values 326
-
- U**
- U and W stock in G71/G72 cycle 173
 UN thread form 227
 UN threads 229
 Undercuts 198
 Using jaws 182
 Using tailstock 186
-
- V**
- Vacant variables 322
 Variable stock
 Milling applications 137
 Turning applications 139
 Variables 319-320
 Common 322
 Local 322
 Null variable 322
 System 322
 Variables in CNC program 318
 Virtual tool point 171
 V-thread 228
-
- W**
- Wear offset 62-63,126
 Width of cut 162
 Working with planes 289
 Working with tolerances 58