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

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 seems 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

8 - TOOL LENGTH OFFSET CHANGE 123

9 - BLOCK SKIP APPLICATIONS 131

10 - STANDARD AND RIGID TAPPING 143

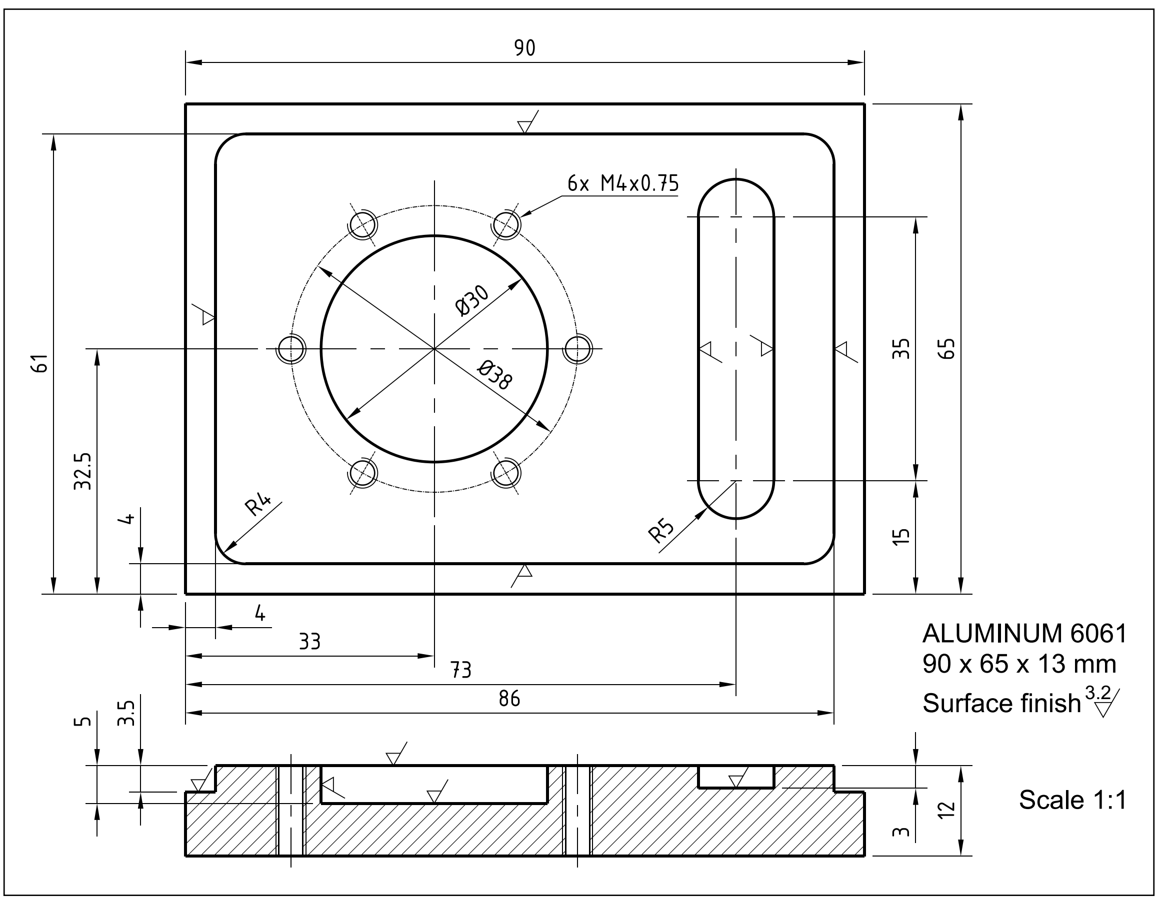
1 PART PROGRAM DEVELOPMENT

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   m (millionth of a meter = 0.000 001 m = 10^{-6}). In practical terms, the required finish of 3.2   m 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, draftsperson, 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

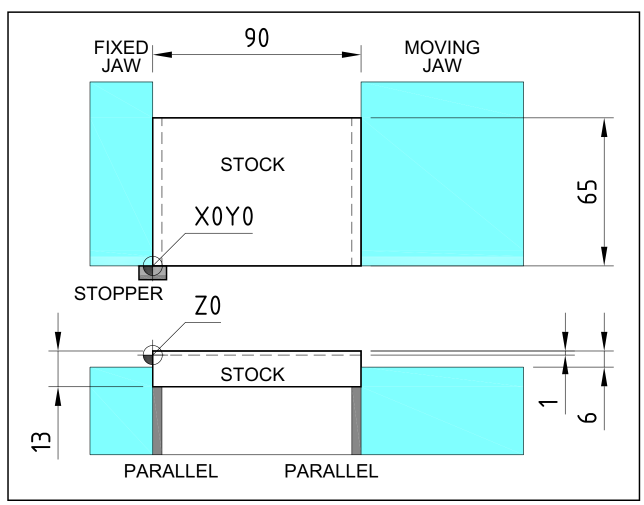
Although the enclosed sample drawing does include material the part is made of, there are many issues related to the material before machining (and programming). Apart of size of the material, its ***shape and condition*** are equally important. Shape could be a simple block or cylinder, it could be 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 devise) 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

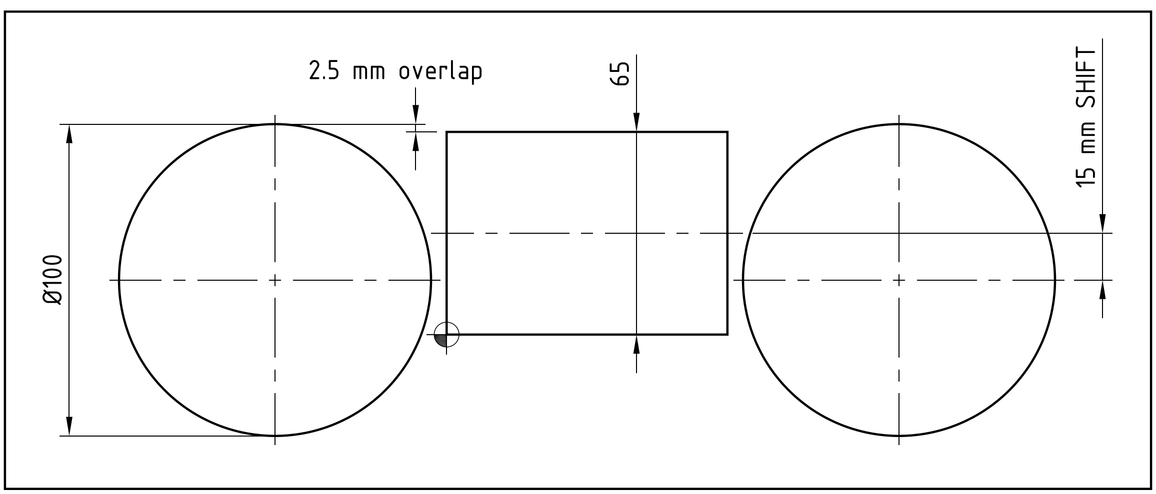
As the part is 1 mm thicker than the final thickness, a face milling tool has to be used to machine the top face to provide final thickness of 12 mm, as per drawing. The amount of one millimeter represents a small depth for facing, so only one depth cut will be necessary to face the part. In order to 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):

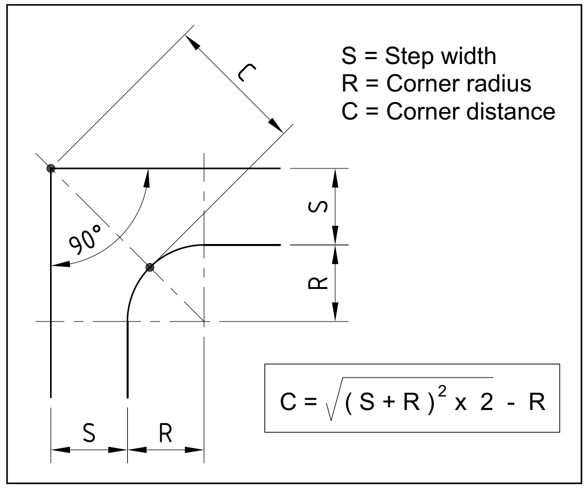
$$\boxed{\text{Maximum shift from part middle} = (100 - 65) / 2 = 17.5 \text{ mm}}$$

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.

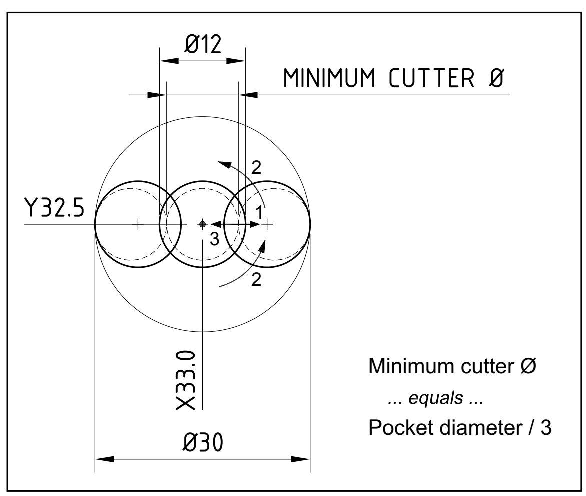


The step depth along the edges is 3.5 mm and should also be considered when the cutting tools are selected. For this job, the depth does not present any special difficulties, and can be machined in one pass.

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.

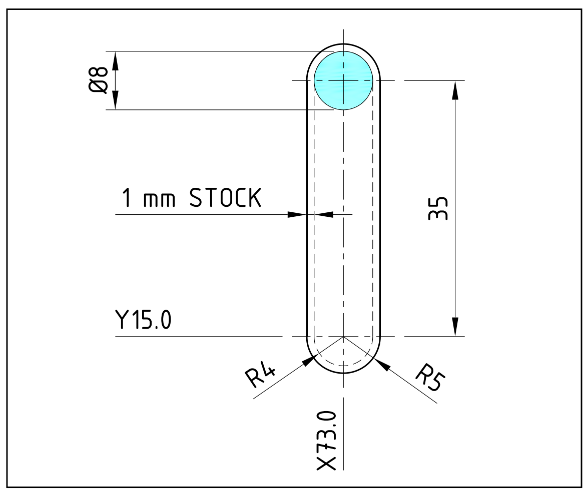
In the description of contouring (see above), the choice of tools was narrowed down to two end mills - 10 mm end mill or 12 mm end mill. Either tool will work well for the contouring, but only the 12 mm end mill is also suitable for machining the circular pocket. The selection of 12 mm end mill for *both* 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:

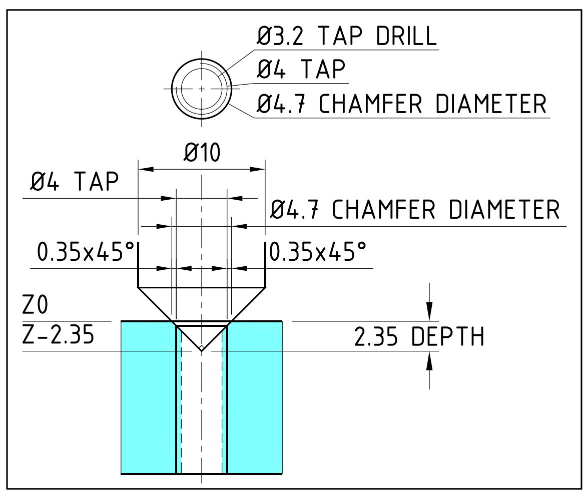
- **To start-up a hole with a small dimple at its exact location ... control of hole location**
- **To machine a chamfer on a hole, by controlling the depth of cut ... control of chamfer size**

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 .



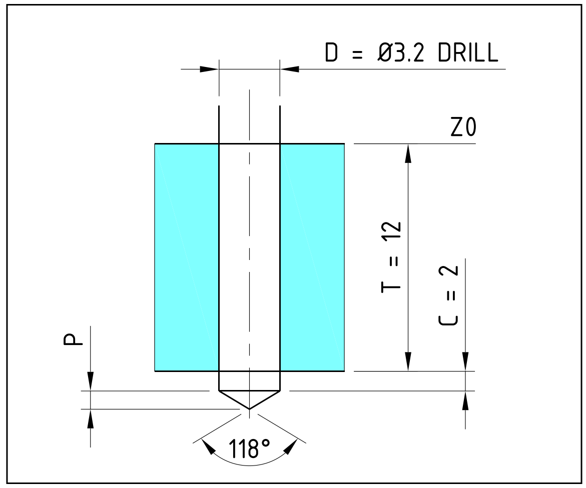
When using spot drills, always consider the fact that only a portion to the angular tool tip is used. That means the programmed depth controls the chamfer size - *or* - the chamfer size is controlled by the programmed depth. If we choose the chamfer size to be 0.35, that means the chamfer is applied 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:

DEPTH OF SPOT DRILL = (2 chamfer size hole diameter) / 2 = (2 0.35 4) / 2 = 2.35 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 that 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 aped (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 programmers 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
$r/min \quad \frac{m/min \quad 1000}{D}$	$r/min \quad \frac{ft/min \quad 12}{D}$

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:

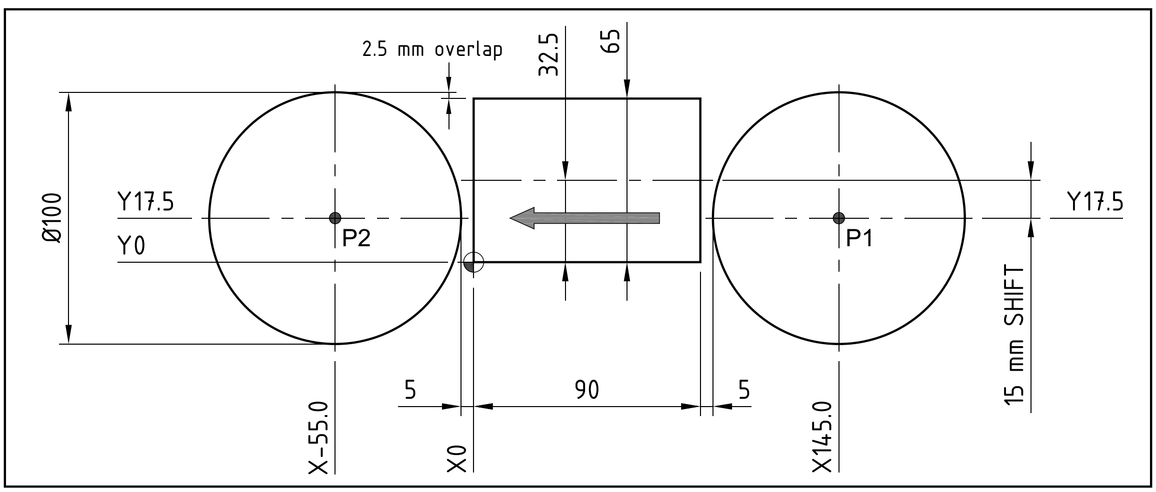
X-coordinate of P1 = 90 + 5 + 50 = X145.0 (part length + clearance + cutter radius)

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

X-coordinate of P2 = - 5 + 50 = 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:

Y-coordinate (P1 and P2) = 65/2 -15 = Y17.5

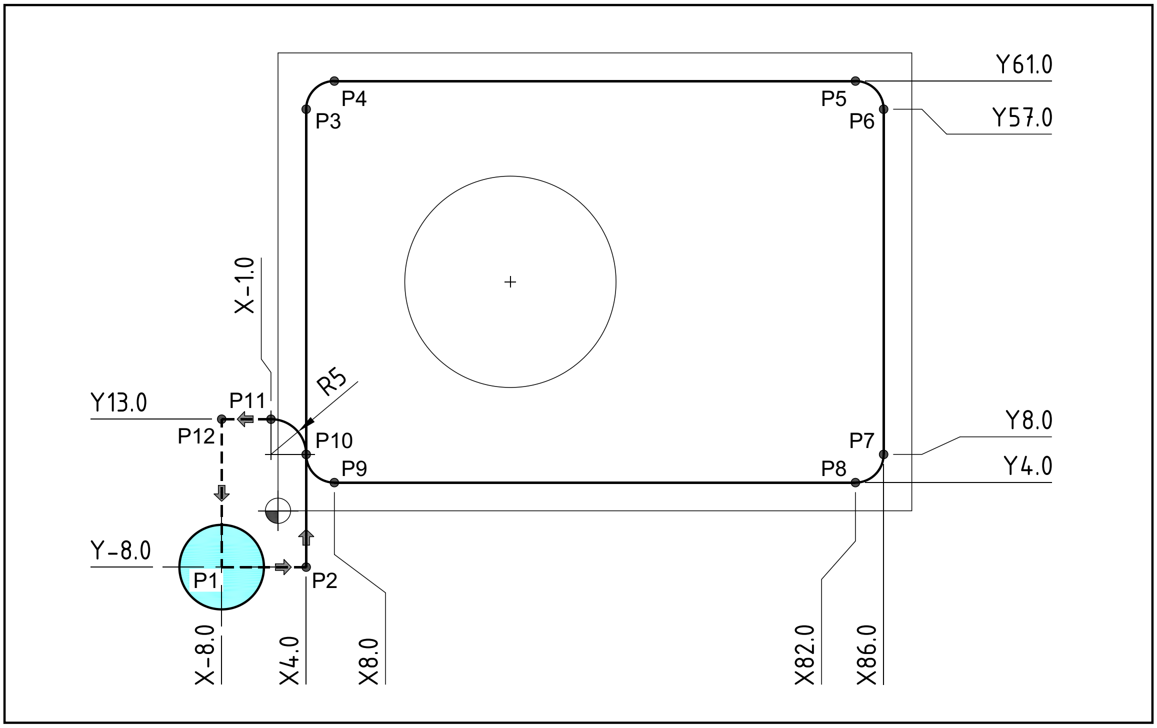


The program for this tool can now be written. This is the first tool of the program, and the format structure will reflect that:

```
(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 is 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.

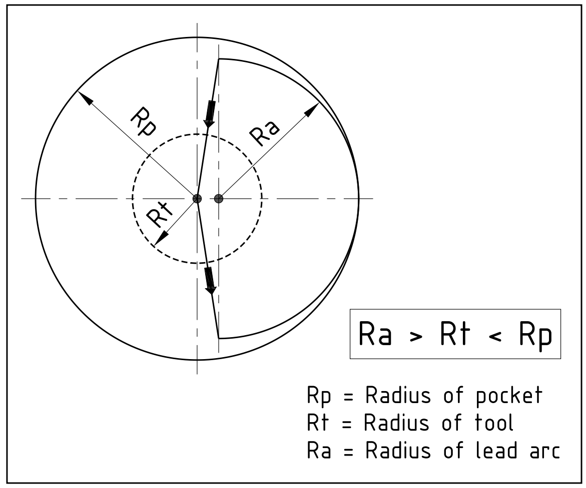
The program also includes a lead-in motion (block N16) and lead-out motions (N25-N27). G41 specifies that radius offset will be to the left of cutting direction, providing a climb milling action.

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.

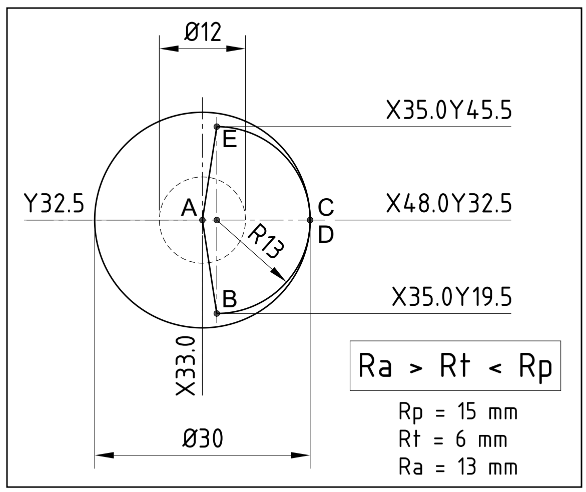


Also shown is the direction of the cut, starting from the pocket center. The toolpath is simple, starting at the center:

☞ **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    ©)
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.

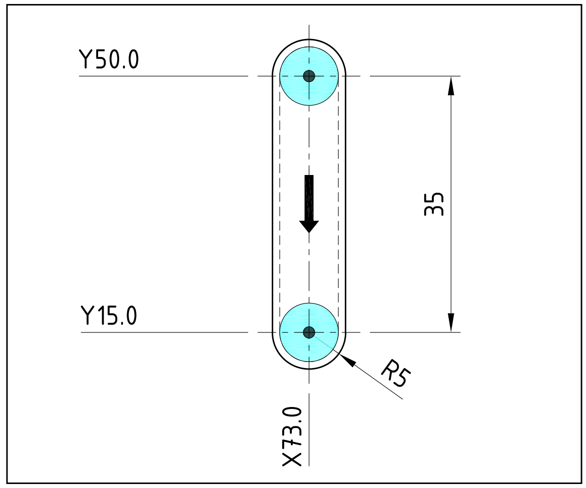
When selecting the lead-in/out arc, first watch the rule of relationship as shown in the illustrations. For the example in this chapter, the *Ra* (lead arc radius) must be larger than *Rt* (tool radius). Based on the pocket radius *Rp*, the range should be between 6 mm and 15 mm. In order to achieve 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.

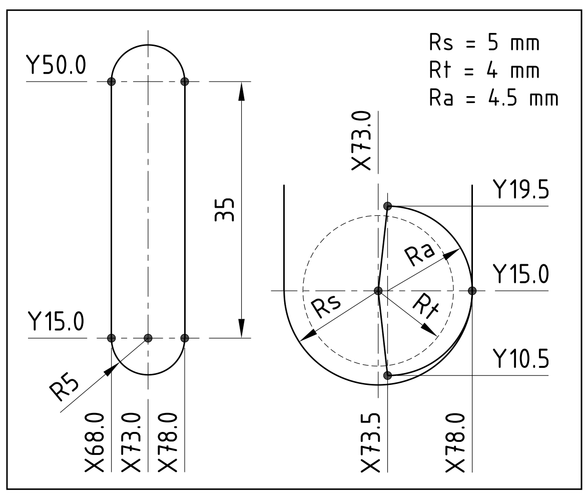
The question *'which slot end should I start from?'* is unnecessary - start from either end, it makes no difference in machining at all. In the example, the tool will rough the slot from its upper position to its lower position, as shown at right:




```
(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 *Rt* is 4 mm, radius of the slot *Rs* is 5 mm. As the lead arc *Ra* must be somewhere between these two values, there is not much flexibility here. For the example in this chapter, the lead arc radius *Ra* 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:



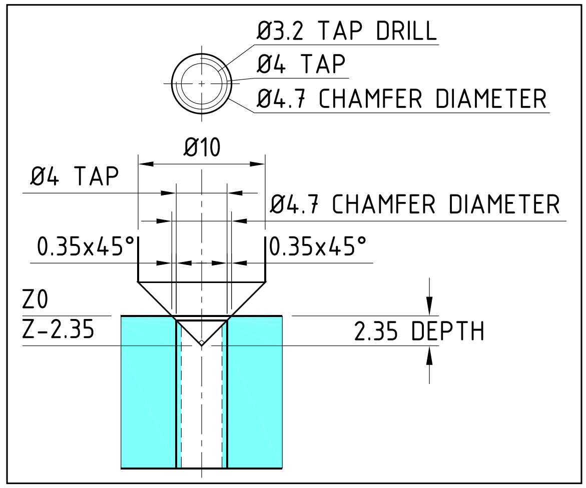

```
(SLOT FINISHING)
N46 G41 X73.5 Y10.5 D53
N47 G03 X78.0 Y15.0 I0 J4.5
N48 G01 Y50.0
N49 G03 X68.0 I-5.0 J0
N50 G01 Y15.0
N51 G03 X78.0 I5.0 J0
N52 X73.5 Y19.5 I-4.5 J0
N53 G40 G01 X73.0 Y15.0
N54 G00 Z10.0 M09
N55 G28 Z10.0 M05
N56 M01
```

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.

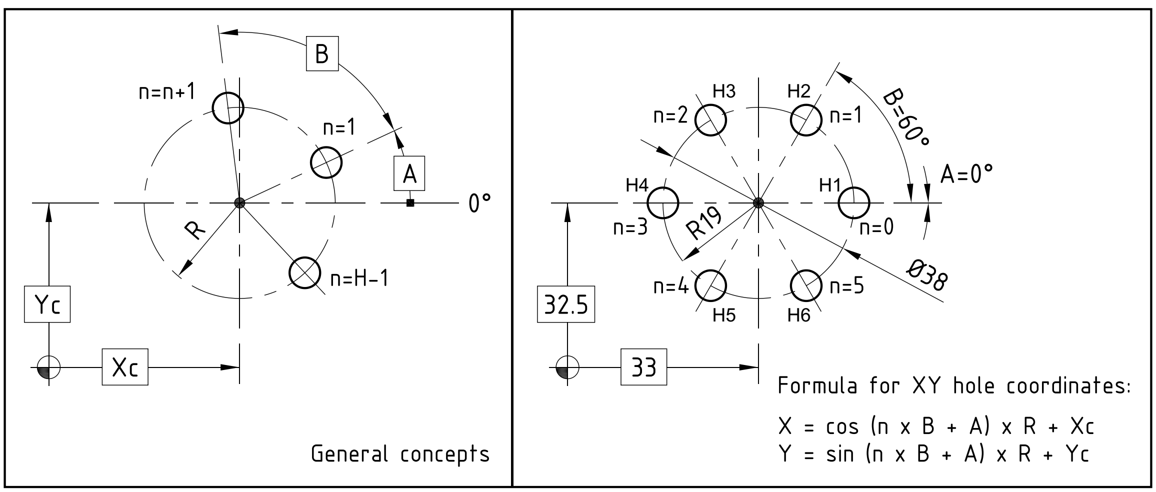
Tool 4 - Spot Drilling

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
```

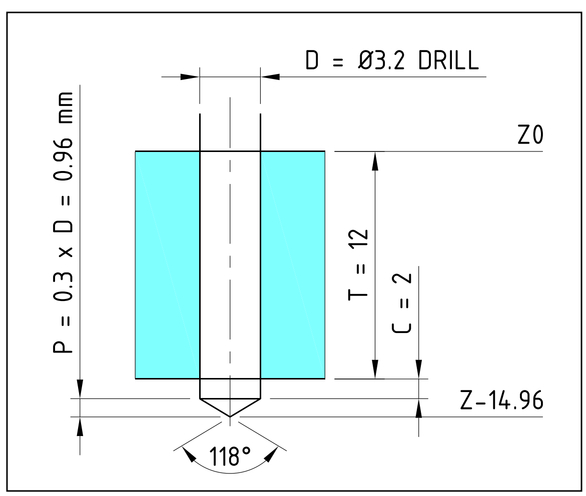
G82 fixed cycle has been used for the spot drill operation. This cycle is very similar to the G81 drilling cycle (see next operation), but it requires a dwell. The purpose of dwell is to pause at the bottom of the hole, before retracting to the clear position - *the reason?* In order to make sure the 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):

Minimum dwell (ms) = (60 1000) / r/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 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 establish 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 pont 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 C P = 12 2 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

!

A

B

C

D

P

R

S

T

U

V

W