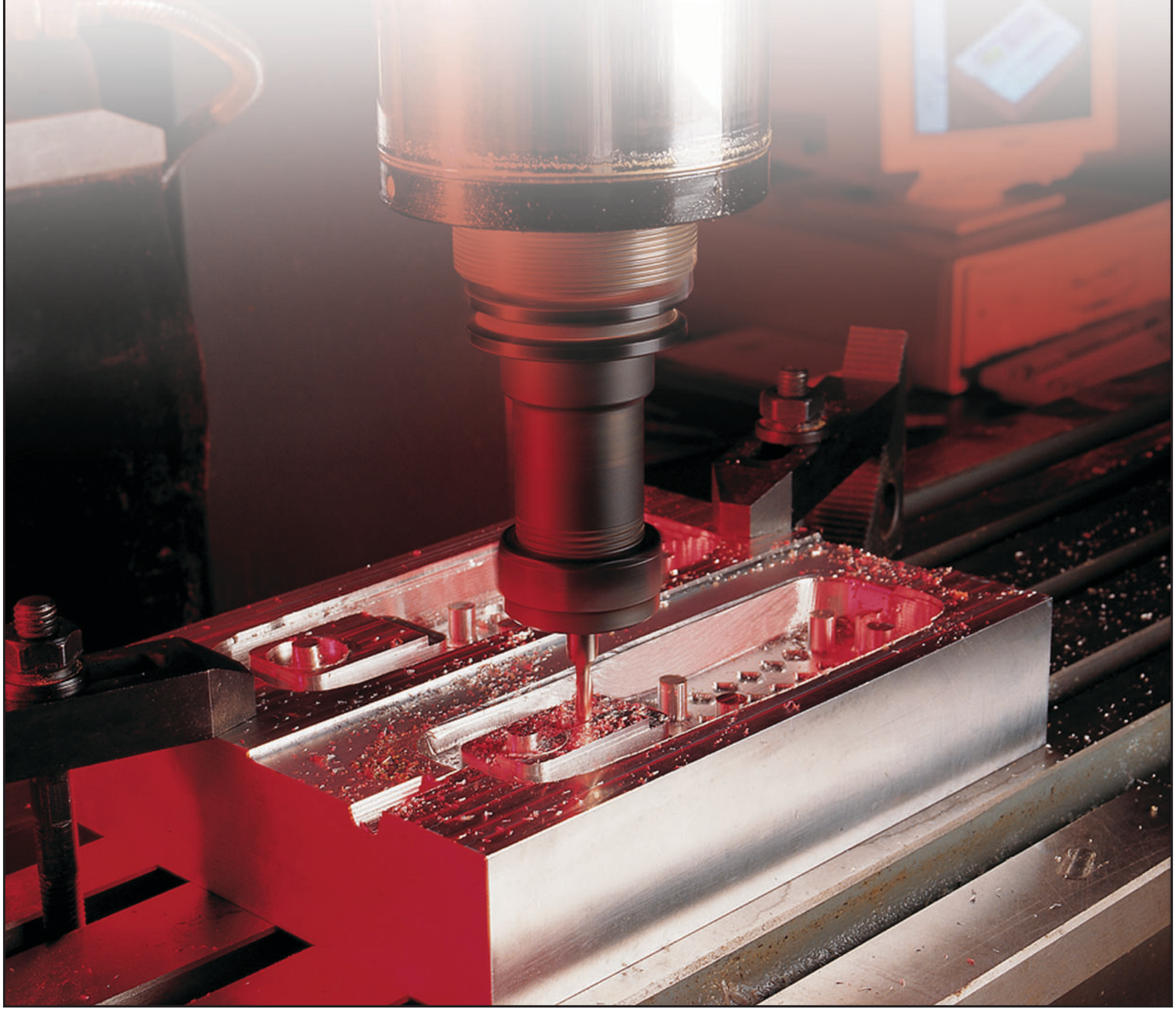
Version 9

Mastercam®

Mill/Design

Tutorial



Mastercam®

Version 9 Mill/Design Tutorial

(Metric version)

Mastercam Version 9 Mill
Mastercam Version 9 Design

Mastercam® Version 9 Mill/Design Tutorial (Metric version)

Date: January 21, 2002

First Printing: January 21, 2002
Software: Mastercam Mill Version 9, Mastercam Design Version 9

Intellectual Property Rights

The Software is and includes intellectual property of CNC. All associated intellectual property rights, including, without limitation, worldwide patent, trademark, copyright and trade secret rights, are reserved. CNC retains all right, title and interest in and copyrights to the Software, regardless of the form or media in or on which the original or other copies may subsequently exist. This Software License Agreement shall not constitute a sale of the Software and no title or proprietary rights to the Software are transferred to you hereby. You acknowledge that the Software is a unique, confidential and valuable asset of CNC, and CNC shall have the right to seek all equitable and legal redress, which may be available to it for the breach or threatened breach of this Software License Agreement including, without limitation, injunctive relief. Unauthorized copying of the Software or failure to comply with the above restrictions shall result in automatic termination of this License and this Software License Agreement without prejudice to all other legal rights and remedies of CNC.

Confidentiality

You acknowledge that the Software contains proprietary trade secrets of CNC and you hereby agree to maintain the confidentiality of the Software using at least as great a degree of care as you use to maintain the confidentiality of your own most confidential information. You agree to reasonably communicate the terms and conditions of this Software License Agreement to those persons employed by you who come into contact with the Software, and to use reasonable best efforts to ensure their compliance with such terms and conditions, including, without limitation, not knowingly permitting such persons to use any portion of the Program for the purpose of deriving the source code of the Program or defeating the Security Mechanism.

Enforcement Obligations

In the event you become aware that any person or entity in your employ or under your control in a manner not authorized by this Software License Agreement is using the Software, you shall immediately use reasonable best efforts to have such unauthorized use of the Software immediately cease. You shall promptly notify CNC in writing of any unauthorized use of the Software of which you become aware.

Limited Warranties

CNC WARRANTS THAT THE MEDIA ON WHICH THE PROGRAM IS DISTRIBUTED WILL BE FREE OF DEFECTS IN MATERIAL OR WORKMANSHIP FOR A PERIOD OF THIRTY (30) DAYS AFTER PURCHASE. THE FOREGOING LIMITED WARRANTY EXCLUDES DEFECTS ARISING OUT OF ACCIDENT, NEGLECT, MISUSE, FAILURE OF ELECTRIC POWER AND CAUSES OTHER THAN ORDINARY AND AUTHORIZED USE. EXCEPT FOR THE FOREGOING LIMITED WARRANTY, THE SOFTWARE IS PROVIDED “AS IS.” YOUR SOLE REMEDY AND CNC’S SOLE OBLIGATION HEREUNDER SHALL BE, AT CNC’S SOLE OPTION, REPLACEMENT OF THE DEFECTIVE MEDIA OR REFUND OF THE PURCHASE PRICE OF THE SOFTWARE. ANY USE BY YOU OF THE SOFTWARE IS AT YOUR OWN RISK. THIS LIMITED WARRANTY IS THE ONLY WARRANTY PROVIDED BY CNC REGARDING THE SOFTWARE. TO THE MAXIMUM EXTENT PERMITTED BY LAW, CNC DISCLAIMS ALL OTHER WARRANTIES OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, WITHOUT LIMITATION, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. CNC IS NOT OBLIGATED TO PROVIDE ANY UPDATES TO THE SOFTWARE. SHOULD THE SOFTWARE PROVE DEFECTIVE FOLLOWING ITS PURCHASE, YOU (AND NOT CNC, ITS DISTRIBUTOR, OR RETAILER) ASSUME THE ENTIRE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION AND ANY INCIDENTAL OR CONSEQUENTIAL DAMAGES.

Limitation of Liability

IN NO EVENT WILL CNC, OR ITS EMPLOYEES, SHAREHOLDERS OR SUPPLIERS BE LIABLE TO YOU FOR ANY INDIRECT, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING WITHOUT LIMITATION, SPECIAL, PUNITIVE, OR EXEMPLARY DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, OR LOSS OF BUSINESS INFORMATION) ARISING OUT OF OR IN CONNECTION WITH THIS SOFTWARE LICENSE AGREEMENT OR THE SUBJECT MATTER HEREOF EVEN IF CNC HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. CNC’S ENTIRE LIABILITY WITH RESPECT TO ITS OBLIGATIONS UNDER THIS SOFTWARE LICENSE AGREEMENT OR OTHERWISE SHALL NOT EXCEED THE AMOUNT OF THE LICENSE FEE PAID BY YOU FOR THE SOFTWARE. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF IMPLIED WARRANTIES OR LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THE ABOVE LIMITATIONS OR EXCLUSIONS MAY NOT APPLY TO YOU.

Indemnification

You shall indemnify and hold harmless CNC, its officers, directors, employees, suppliers and agents from and against all losses, settlements, claims, actions, suits, proceedings, judgments, awards, damages, liabilities, costs and expenses including, without limitation, reasonable attorneys’ fees (collectively “Losses”) which arise out of or as a result of any breach of this Software License Agreement by you or your employees, agents, resellers, dealers or sub-dealers and shall reimburse CNC for any and all legal, accounting and other fees, costs and expenses reasonably incurred by any of them in connection with investigating, mitigating or defending any such Losses.

Educational Pricing

If you received this Software under or in accordance with a CNC "Educational Pricing" plan, option, schedule or program you shall not use this Software to conduct any computer aided design, computer aided drafting or computer aided machining activities that intentionally, incidentally, directly or indirectly result in the receipt, derivation or generation of profit to or by you.

Termination

This Software License Agreement is effective until terminated. You may terminate this Software License Agreement at any time by returning to CNC all copies of the Software under your control and by returning the Security Mechanism to CNC. CNC may terminate this Software License Agreement if CNC finds in its sole discretion that you have violated the terms of this Software License Agreement. Upon termination of this Software License Agreement, you agree to immediately return to CNC all copies of the Software and return the Security Mechanism to CNC, and to certify to CNC in writing that all known copies, including backup copies, have been returned. All provisions relating to confidentiality, proprietary rights, indemnification and non-disclosure shall survive the termination of this Software License Agreement.

General

This Software License Agreement shall be construed, interpreted and governed by the laws of the state of Connecticut, without regard to conflicts of law provisions. The sole jurisdiction and venue for any litigation arising from or related to this Software License Agreement or the subject matter hereof shall be in an appropriate state or federal court located in Hartford, Connecticut, and you hereby submit to the jurisdiction of such courts. This Software License Agreement shall constitute the entire agreement between you and CNC with respect to the subject matter hereof. Any waiver or modification of this Software License Agreement shall be valid only if it is in writing and signed by both parties hereto. If any part of this Agreement is found invalid or unenforceable by a court of competent jurisdiction, the remainder of this Agreement shall be interpreted so as to reasonably effect the intention of the parties.

U.S. Government Restricted Rights

The Software provided hereunder is a "commercial item," as that term is defined in 48 C.F.R. 2.101, consisting of "commercial computer software" and "commercial computer software documentation," as such terms are used in 48 C.F.R. 12.212. Consistent with 48 C.F.R. 12.212 and 48 C.F.R. 227.7202-1 through 227.7202-4, the Software made available to the United States of America, its agencies and/or instrumentalities, is provided with only those rights set forth in this Agreement. Use, duplication or disclosure of the Software by the government is subject to the restrictions as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software-Restricted Rights clause at 48 C.F.R. 52.227-19, as amended, or any successor regulations thereto.

Export Restrictions

You represent and warrant that you will not, without obtaining prior written authorization from CNC and, if required, of the Bureau of Export Administration of the United States Department of Commerce or other relevant agency of the United States Government, export or reexport, directly or indirectly, the Software from the United States to (i) any country destination to which export is restricted by the Export Administration Regulations of the United States Department of Commerce; (ii) any country subject to sanctions administered by the Office of Foreign Assets Control, United States Department of the Treasury; or (iii) such other countries to which export is restricted by any other United States government agency. You further agree that you are solely responsible for compliance with any import laws and regulations of the country of destination of a permitted export or reexport, and any other import requirement related to a permitted export or reexport.

Exceptions To Security Mechanism Requirements

CNC software programs MASTERCAM DRAFT and MASTERCAM DEMO do not require the use of Security Mechanisms, and the provisions in this Software License Agreement relating to Security Mechanisms do not apply to your use of such programs, provided, however, that such provisions shall apply to your use of all other Software provided hereunder.

Survival

All provisions of this Software License Agreement relating to confidentiality, non-disclosure, CNC's proprietary rights, disclaimers, and limits of liability, or indemnification by Customer shall survive termination of this License for any reason.

Printed in the United States of America.

This book was printed on recycled paper.

Table of Contents

1 Introduction to Mastercam

Welcome to Mastercam Version 9. Mastercam Design is a full-featured modeling application that combines 2D and 3D wireframe geometry and surfacing abilities with powerful editing and transformation tools. Mastercam Mill builds on Mastercam Design by letting you create and manage a wide variety of machining operations. Mastercam uses a feature called associativity to link machining operations to the geometry so that toolpaths can be automatically regenerated when part geometry changes.

To help you learn Mastercam, this tutorial and extensive online help accompany the product.

- Use this tutorial as a self-training aid to orient yourself to the Mastercam program and interface. After completing the tutorial, you will have a good introduction to accomplishing common drafting and milling operations with Mastercam. However, the tutorial does not try to cover every Mastercam feature.
- Use the online help as a reference for specific "How to…" or "What's this…" questions, like "How do I machine an open pocket," "What's a Coons surface," or "How do I create a new tool definition?" This tutorial shows you how to use online help.

Using the sample parts

The sample parts for all the exercises in this tutorial are located in the **C:\Mcam9\Tutorials\Mill Tutorial\Metric** folder. The sample parts are read-only, so you do not accidentally write over them. You should create a separate working folder where you can save your own parts as you complete the tutorial.

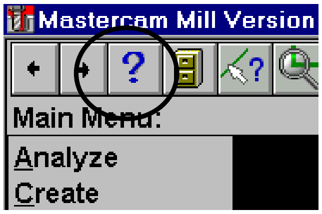
Note: The parts for the exercises in this tutorial were created using metric units of measurement. When you open one of the tutorial parts, if you are using a configuration file based on different units of measure, Mastercam will automatically switch configuration files to match the units in the current file. For example, if you are working with the metric configuration file for Mastercam Mill (Mill9m.cfg) and you open an inch part, the system switches to the inch configuration file (Mill9.cfg).

If you need more help

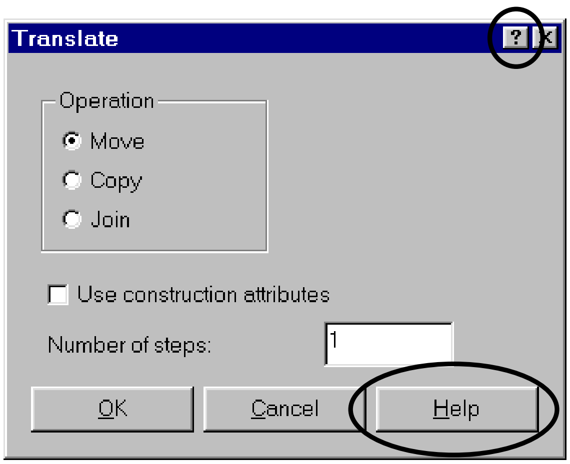
Online help

Online help contains the latest and most up-to-date information about Mastercam. The following pictures show how to use the online help.

Choose the Help toolbar button to get information on the current menu.



Tip: You can also press [Alt + H] anywhere within Mastercam to get additional help.



Click on the question mark then click on any field for more information.

Press the Help button to get information about the dialog box.

Dealers

If you have a question about Mastercam and have not been able to locate the answer in this tutorial or the online help, contact your local Mastercam dealer.

Technical support

If your dealer is unavailable, you can call CNC Software Support Services Monday through Friday, 8:00 a.m.–6:00 p.m., USA Eastern Standard Time.

When calling CNC Software, Inc. for technical support, please follow these guidelines:

- Be sure you have already tried to contact your Mastercam dealer.
- Be ready to describe the problem in detail. Write down what happened, particularly if you cannot call immediately after the problem occurs.
- Be in front of your computer when you call.
- If possible, try to duplicate the problem before calling. Our Support Services technician may require you to duplicate the problem while you are on the phone.
- When you call, have ready a complete description of your hardware, including your operating system (OS), central processing unit (CPU), mouse, and memory.

You can also leave a message for CNC Support Services twenty-four hours a day, seven days a week via our e-mail or web site addresses or the BBS. A member of our technical support staff will return your e-mail or call you on the next business day.

Keep the following information on hand in case you need to reach us:

Important Information	
Address	CNC Software, Inc. 671 Old Post Road Tolland, Connecticut, 06084-9970 USA
Phone	(860) 875-5006
Fax	(860) 872-1565
BBS	(860) 871-8050
TELNET and ftp://	ftp.mastercam.com or 172.16.100.100
Internet Address	http://www.mastercam.com
E-mail	support@mastercam.com

Additional resources

- For information on training, contact your Mastercam dealer.
- For an ongoing discussion of Mastercam-related topics, visit the Mastercam online forum at **http://www.emastercam.com**.

2 Getting Started

This chapter introduces you to the Mastercam interface. You will learn about some major Mastercam features, create and delete some simple geometry, learn how to access the online reference material, and save files.

Exercise 1 – Learning the Mastercam interface

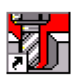
This exercise shows you how to navigate through Mastercam. You will learn the following skills:

- **Starting Mastercam**
- **Learning about the different areas on the screen**
- **Creating a point**
- **Displaying the construction origin and coordinate axes**

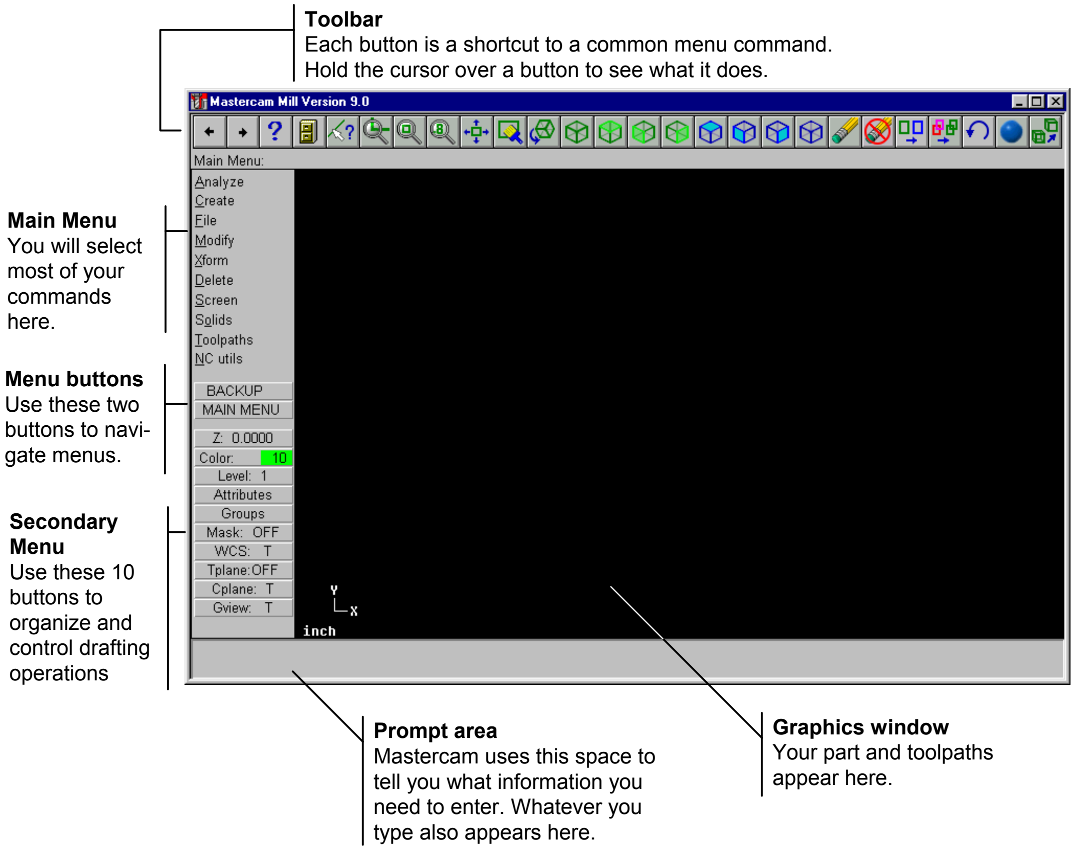
▶ *Starting Mastercam*

1. Double-click the appropriate Mastercam icon on your Windows® desktop:

    for Mastercam Design

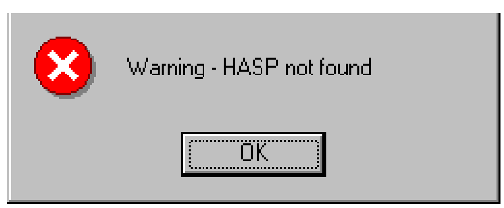
    for Mastercam Mill

   The following picture shows you some of the main features of the Mastercam workspace.



▶ *Learning about the HASP and NetHASP*

Mastercam uses two types of licensing: single-user licensing and network licensing. If you are using single-user licensing, you need to have a special piece of hardware called a HASP (sometimes called a dongle or SIM) attached to your parallel or USB port. If you get an error message like the following:



this component is either missing or not configured properly. Refer to your installation instructions (included in a separate document) or contact your dealer for assistance.

If you are using network licensing, then a NetHASP must be installed on a computer on your network. If you see any of the following messages, see your network administrator:

- **`Error checking out a Mill license. No licenses have been purchased for this product.`**
- **`Active NetHASP server not found.`**
- **`All available licenses are in use.`**

For more information on NetHASP installation, see **Mastercam Network Licensing.doc** in your main Mastercam folder.

▶ *Creating a point*

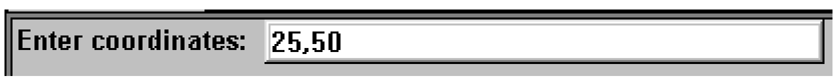
The Main Menu is where you will start most tasks in Mastercam.

1. Choose **Main Menu, Create, Point, Position**.

   Notice that the prompt area at the bottom left of the screen displays the message *Create point: specify a point*.

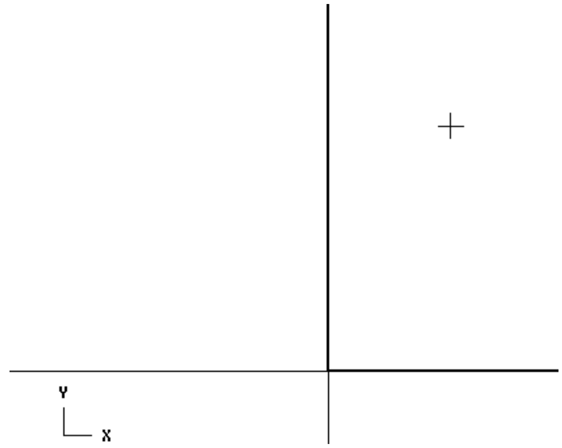
2. Without moving your mouse, type **25,50**.

   Notice that the values you type (the coordinates) appear in the prompt area as you type.

   Tip: You can also enter coordinates as XY-pairs, for example, X25,Y50

3. Press [Enter] to display the point at position 25,50.
4. Press [F9] to display the construction origin. The graphics window should look like the following picture.



Tip: Press [F9] again to clear the axes from the screen.

Exercise 2 – Designing a rectangle

This exercise shows you how to design a simple rectangle. You will learn the following skills:

- **Changing the current color**
- **Creating a rectangle**
- **Resizing the screen**
- **Zooming and panning**
- **Saving the part for later use**

Before you begin, you should create a working folder where you will store your parts as you are working on them. Choose a different folder than **C:\Mcam9\Tutorials\Mill Tutorial\Metric** so you will not mistakenly overwrite the original parts.

▶ *Changing the current color*

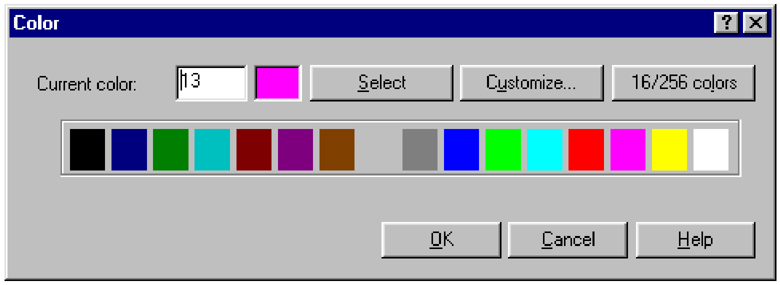
You should make the rectangle a different color so you can distinguish its corner from the point you created in the last exercise.

1. Choose the **Color** button: 

. It tells you that the current color is bright green.

   Tip: Every color has a corresponding number (green is number 10).

2. Change the current color to magenta by clicking on the magenta button and choosing **OK**.

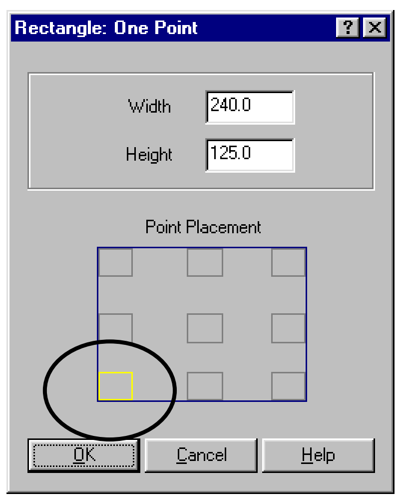


The Color button changes to show the new color. From now on, everything you create will be in the new color.

▶ *Creating a rectangle*

In this task, you will create a rectangle with its bottom-left corner at X25,Y50. The rectangle will be 240 mm wide by 125 mm high.

1. Choose **Main Menu, Create, Rectangle, 1 point**.
2. Enter **240** for the **Width**.
3. Enter **125** for the **Height**.
4. Choose the lower-left **Point Placement** box as shown in the following picture.



5. Choose **OK**.

6. Drag the cursor near the point you created earlier until you see a small white square appear around the point. Notice that the cursor "snaps" into position when you come near the point; this is Mastercam's AutoCursor feature.
7. Click the mouse button. This positions the rectangle so that its lower left corner is at the coordinates 25,50. Notice that the rectangle function is still active; every time you click the mouse, you will create another rectangle.
8. Press [Esc] to exit the rectangle function.

   *Note: If you accidentally create more than one rectangle, choose the **Undo** button from the toolbar. It will undo the most recent rectangle.*

9. Choose the **Screen–Fit** button on the toolbar so you can see the entire rectangle.
10. You can pan the screen by pressing the arrow keys (Left, Right, Up, and Down). Press the arrow keys to try it.
11. Press the [Page Up] and [Page Down] keys to zoom in or out. Each key press zooms in or out by 5%.

    Your part should look like the following picture.

▶ *Saving the file*

You will save the part in the working folder you created earlier. If you have not created a working folder yet, do so now.

1. Choose **Main Menu, File, Save**.
2. Save the file as **firstpart.mc9** in your working folder.

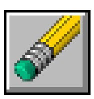
   *Note: It is a good idea to save your file frequently as you work. This way, if you make an error, you can choose **File, Get** to open a recent version of the file. Make a practice of saving your file every time you successfully complete an exercise.*

Exercise 3 – Deleting the rectangle and using help

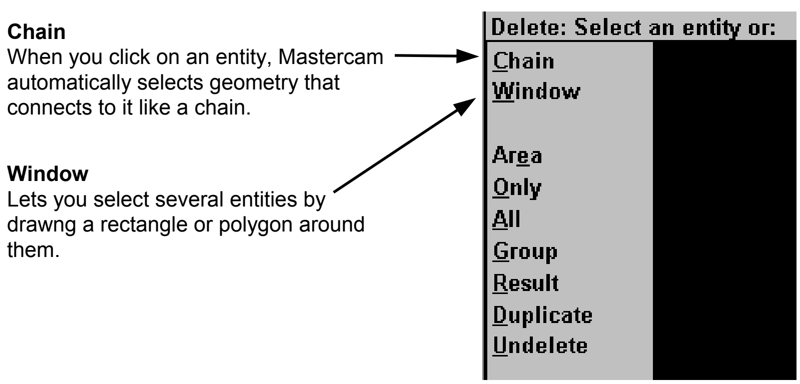
The Create Rectangle function has several useful options. You can learn about them in the online help. In this exercise, you will:

- **Delete the rectangle**
- **Use Help to learn about rectangle options**
- **Set rectangle options and create a new rectangle**
- **Use AutoSave to quickly save your work**

▶ *Delete the rectangle*

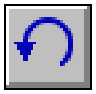


1. Choose the **Delete** button on the toolbar. You will see the menu shown in the following picture. The different options give you different ways of selecting what geometry you want to delete.



Tip: To delete a single entity, click on it; you don' t need to choose anything from the menu.

2. Choose **Chain** from the menu.
3. Click anywhere on the rectangle. The whole rectangle highlights, telling you that it is selected.
4. Choose **Done**. The rectangle is deleted.



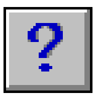
5. Choose the **Undo** button on the toolbar. The rectangle reappears.

Tip: Choose the **Undelete–Single** button  to make the rectangle reappear one segment at a time.

6. Choose the **Delete** button again. This time, instead of choosing **Chain** from the menu, click on the rectangle. Without chaining, Mastercam only selects one line at a time.
7. Delete the remaining lines.

▶ *Using Help to learn about rectangle options*

1. Choose **Main Menu, Create, Rectangle**.

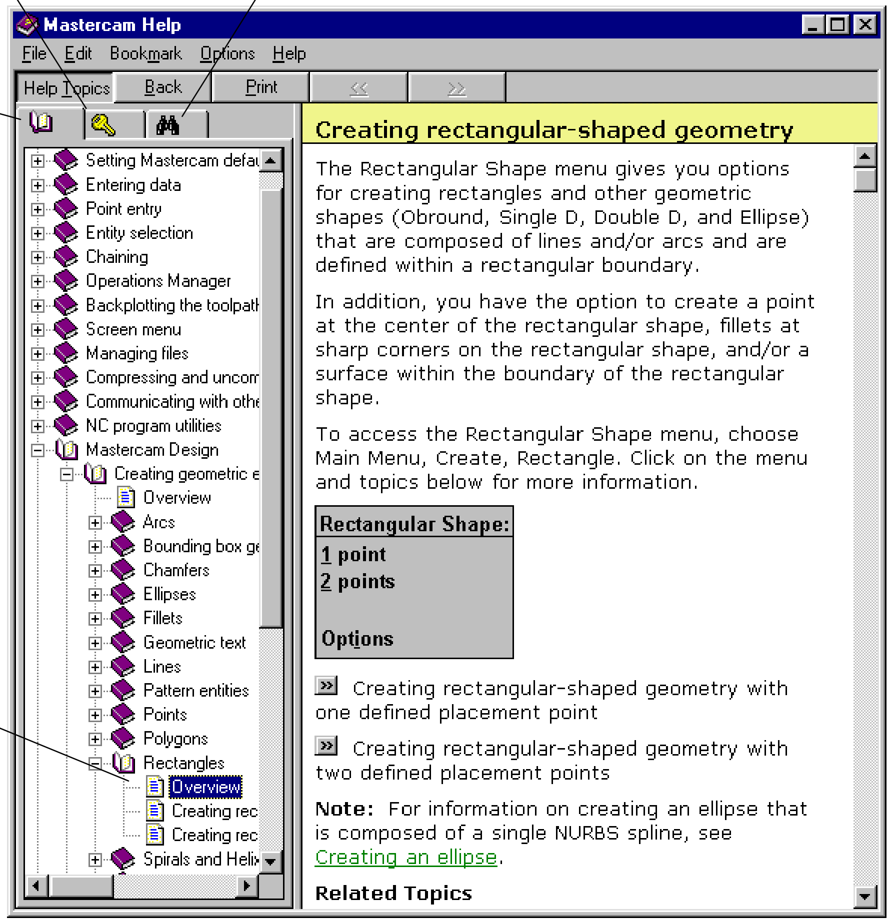


2. You can see the three options on the menu: **1 point**, **2 point**, and **Options**. Choose the **Help** button on the toolbar to learn more about them.
3. The following help screen displays. Every time you choose the Help button while a menu is active, you get a topic about that menu.

Choose this tab to search the index.

Choose this tab to search the help system.

Choose this tab to see the Table of Contents for the entire help system.



The currently displayed topic is highlighted. Click on any other topic in the Table of Contents to read about it.

Note: You can also press [Alt + H] at any time to get help.

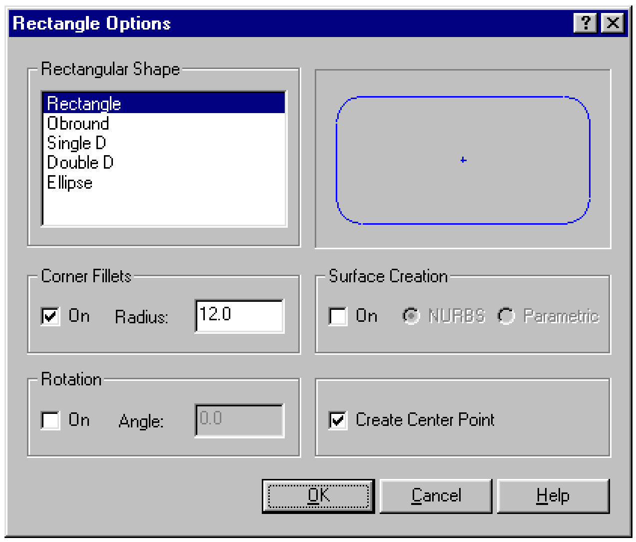
4. Read through the topic to learn about the different kinds of rectangle-related shapes you can create. The help topic also gives you links to instructions for specific procedures and related topics.

5. Close the Help window.

▶ *Setting rectangle options and creating a new rectangle*

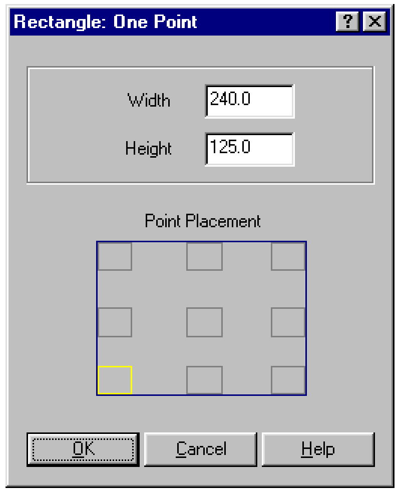
According to the help topic, you can use the Options command to automatically create rounded corners and a center point on the rectangle. You will create a new rectangle with these options.

1. Choose **Options** from the menu.
2. Select the **Corner Fillets–On** check box and enter a **Radius** of **12**.
3. Select the **Create Center Point** check box. Your dialog box should match the following picture.



4. Choose the  button on the title bar of the dialog box and notice how the cursor changes shape. Click on one of the fields in the dialog box and you will see a short description of what it does. This "What's This" help is available on almost every dialog box in Mastercam. Use it whenever you need a quick explanation of what a dialog box option does or means.
5. Choose **OK** to close the dialog box.
6. Choose **1 point** from the menu.

7. Enter the values shown in the following picture and choose **OK**.



8. Move the cursor over the point you created earlier until it is highlighted. Click to place the rectangle.
9. Press [Esc]. You can see the rounded corners and center point that were automatically added.

▶ *Using AutoSave to quickly save your work*

1. Press [Alt + A] to open the AutoSave dialog box.
2. Choose **OK** to quickly save the file with the same name and in the same location. This will overwrite whatever version is already saved to disk.

   Tip: You can choose the **Active** checkbox and type in a time interval. Mastercam will then automatically save your work at the set times.

3. Choose **Yes** to confirm the overwrite.

Now that you've seen how to get around in Mastercam, you're ready to create your first part. The next chapter will show you how to use Mastercam's design features to draft a part from a blueprint. Then, you'll create a complete operation to machine it.

3 Creating a 2D Part and Contour Toolpath

This chapter introduces you to the major steps that typically go into making a part:

- Drawing the part
- Creating a toolpath
- Previewing the toolpath
- Editing the toolpath
- Posting the finished toolpath

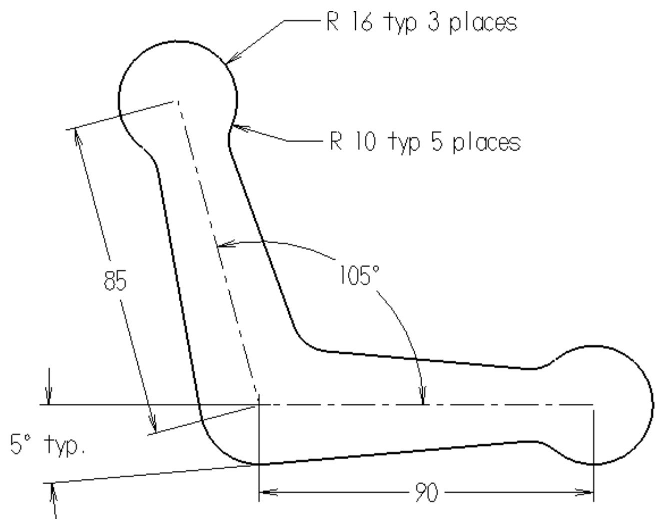
Along the way, you will see how Mastercam's associativity feature links the different parts of your operation. You will create a basic 2D part and cut it with a contour toolpath.

Exercise 1 – Designing the part

This exercise introduces you to some basic drawing functions. You will learn the following skills:

- **Creating a new file**
- **Creating points, lines, arcs, and fillets**
- **Mirroring and rotating lines**
- **Trimming lines and arcs**

The following blueprint shows the part you will create.



Creating a new file

If necessary, follow these steps to create a new, blank Mastercam drawing.

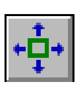
1. Choose **Main Menu, File, New.**
2. Choose **Yes** when prompted to create a new drawing.
3. If the current file has had any changes made to it since the last time it was saved, you will be asked whether or not you wish to save it. Choose **Yes** again if you wish to save it.

Creating construction guides

The first step is to create some construction guides to properly locate and orient the drawing.

1. Create the center point of the elbow. Choose **Main Menu, Create, Point, Position**.
2. Enter the coordinates **75,75**. As soon as you start typing, the numbers will appear in the prompt area.

   Tip: Press [Enter] after entering the number(s) in the prompt area.

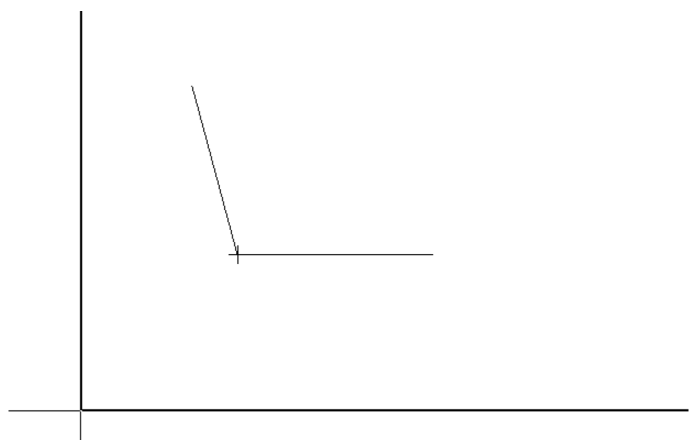


3. Choose the **Screen–Fit** button on the toolbar to center the point in the graphics window.
4. Next, draw the center lines for the two arms. Choose **Main Menu, Create, Line, Polar**.

5. Click on the point to select it as an endpoint.

   Tip: Pass the cursor over the point. When a square displays, click the mouse button.

6. In the prompt area, enter an angle of **0**.
7. Enter a line length of **90**. The guide for the horizontal arm appears.
8. Mastercam automatically prompts you to select an endpoint for another polar line. Click on the same point as in step 5.
9. Enter an angle of **105**.
10. Enter a line length of **85**.
11. Press [F9] to show the construction origin and XY axes. The part should look like the following picture.



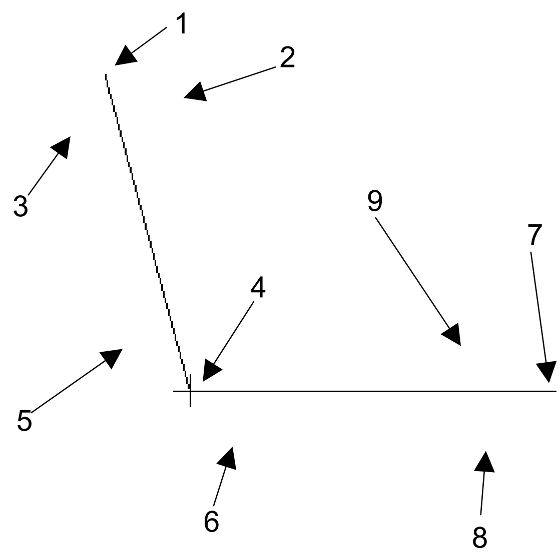
12. Press [F9] again to clear the axes from the screen.

▶ *Drawing the arcs*

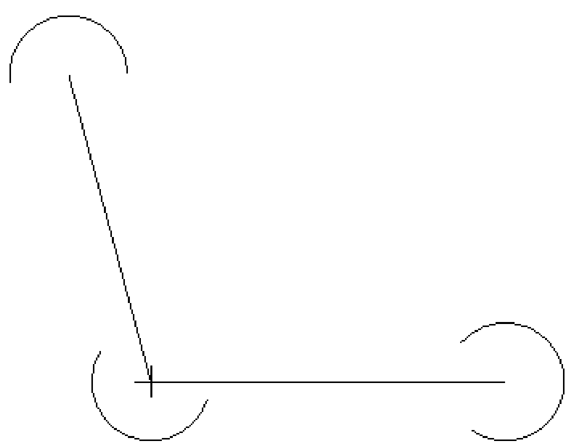
Use arcs to create the curved ends and outside bend of the part.

1. Choose **Main Menu, Create, Arc, Polar, Sketch**.

2. Select the center point for the first arc. Click on the line endpoint at position 1, as shown in the picture to the right.
3. Type the radius of the arc: **16**
4. Specify the approximate ending positions of the arc. Click at position 2 then at position 3.



5. Repeat steps 2 through 4 to create the second and third arcs. Click on points 4, 5, 6 and 7, 8, 9 to create the other arcs.
6. If necessary, choose the **Screen–Fit** button again to fit the part completely in the screen. It should look like the following picture:

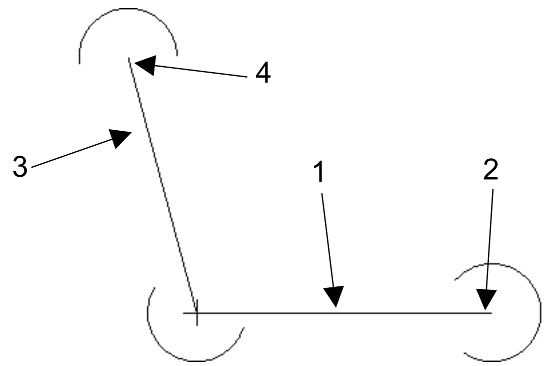


▶ *Rotating lines to create the arms*

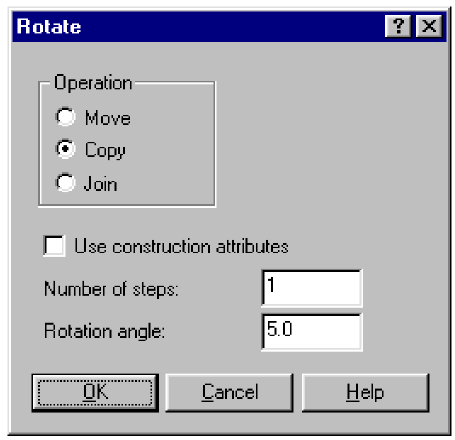
To create the outside edges of the arms, you will rotate the center line guides you created earlier.

1. Choose **Main Menu, Xform, Rotate**.

4. Select the line at position 1.
5. Choose **Done.**
6. Select the endpoint at position 2.



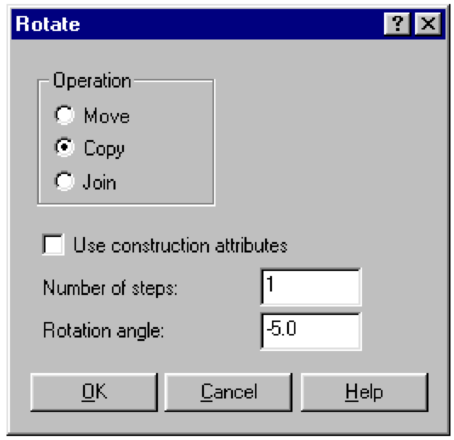
7. Enter the values shown on the following dialog box and choose **OK**.

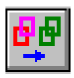


8. Select the line at position 3.
9. Choose **Done**.

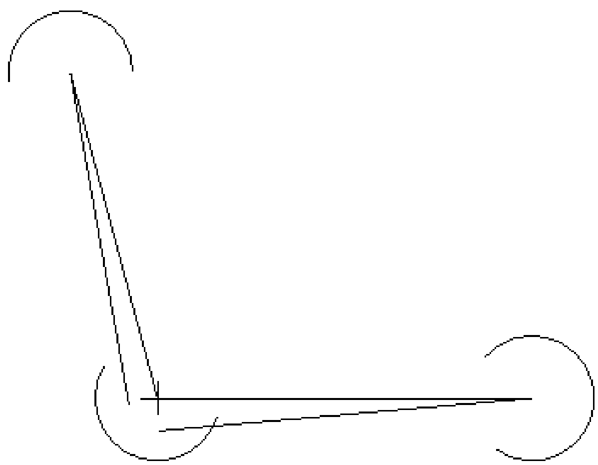
Tip: Instead of choosing menu items with the mouse, you can type the shortcut letter. For each item on the menu, the shortcut letter is underlined.

10. Select the line endpoint at position 4.
11. Enter the values shown on the following dialog box and choose **OK**.





12. Whenever you do a Xform operation, Mastercam changes the colors of the original geometry and the new geometry so you can clearly see the results of the operation. Choose the **Screen-Clear colors** button on the toolbar to return the lines to their original color. The part should look like the following picture.

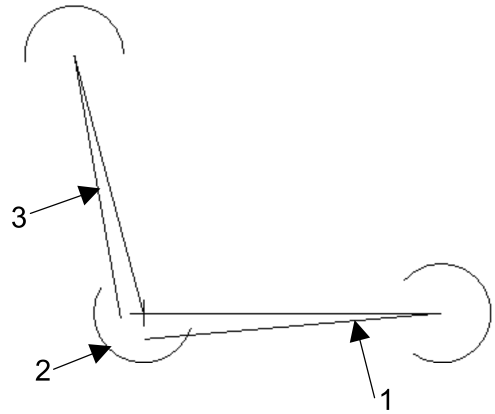


▶ *Moving the lines to the proper position*

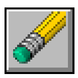
You've created the lines at the proper angle and orientation. Now, you need to move them to the proper position tangent to the arcs.

1. Choose **Main Menu, Create, Line, Parallel, Arc**.

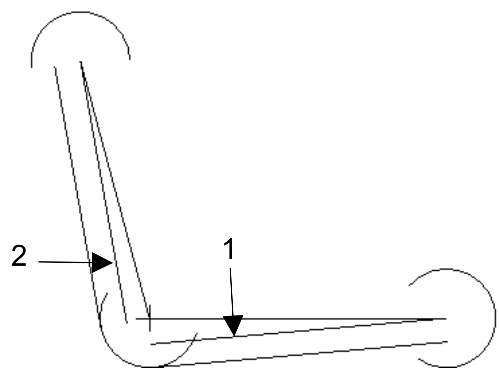
2. Select the line at position 1.
3. Select the arc at position 2.
4. Mastercam shows you two possible lines. Click on the bottom line to keep it.
5. To create the second line, select the line at position 3.



6. Select the arc at position 2 again.
7. Click on the left line to keep it.

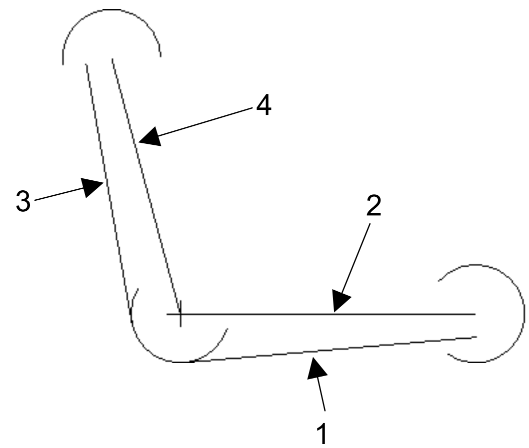


8. Choose the **Delete** button on the toolbar.
9. Click on lines 1 and 2 as shown in the following picture to delete them.

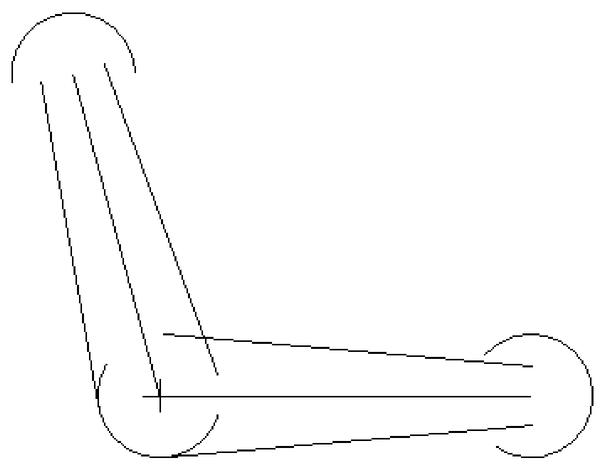


10. Finally, create the other side of the arms by mirroring the lines around the construction guides. Choose **Main Menu, Xform, Mirror**.

11. Select the line at position 1.
12. Choose **Done**.
13. Select the line at position 2.
14. Choose **Copy** and **OK** from the Mirror dialog box.
15. Repeat steps 11 through 14 for lines 3 and 4.



16. Clear the screen colors. Your part should look like the following picture.

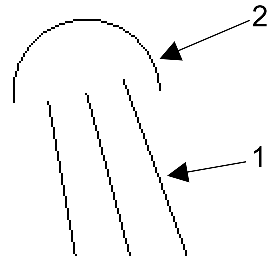


▶ *Creating the fillets*

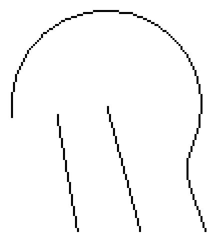
Create fillets to join the lines and arcs. You will also see how Mastercam can automatically trim lines to the base of the fillets. Complete the part by deleting the remaining construction guides.

1. Choose **Main Menu, Create, Fillet, Radius**.
2. Enter the fillet radius: **10**

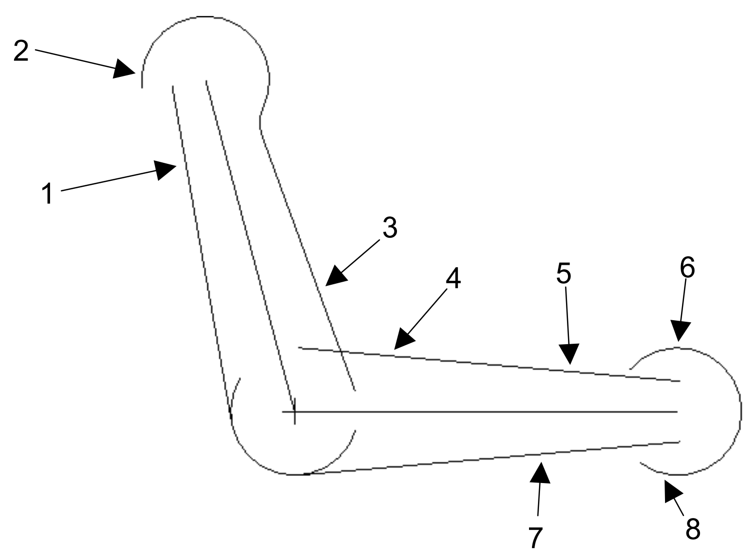
3. Select the line at position 1 and the arc at position 2.



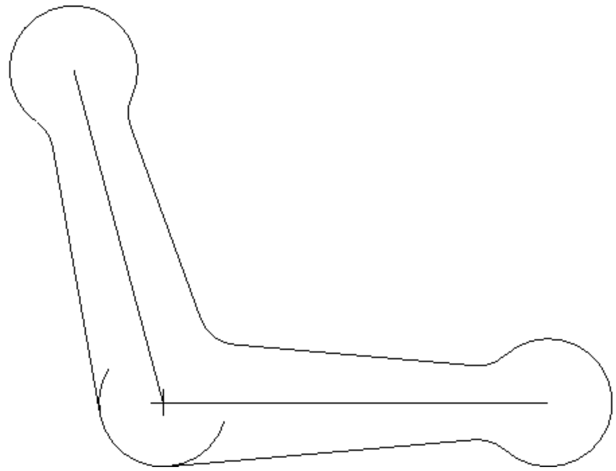
The fillet should look like the following picture.



4. Select the lines and arcs in the order shown in the following picture to create the remaining fillets.

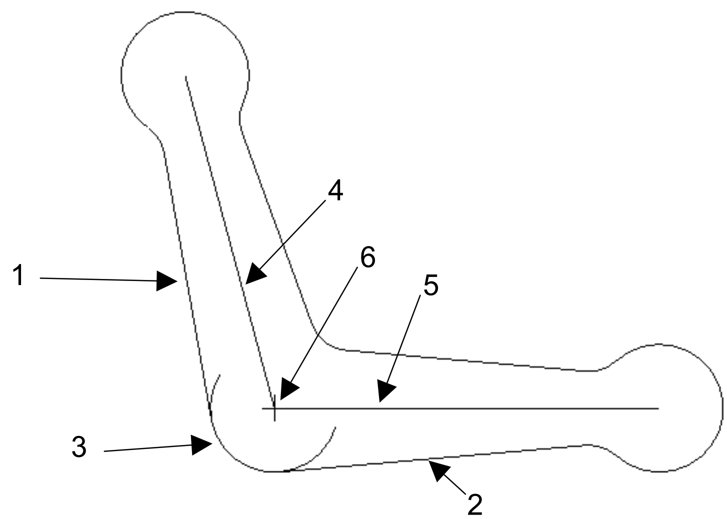


The part should look like the following picture when you are done.



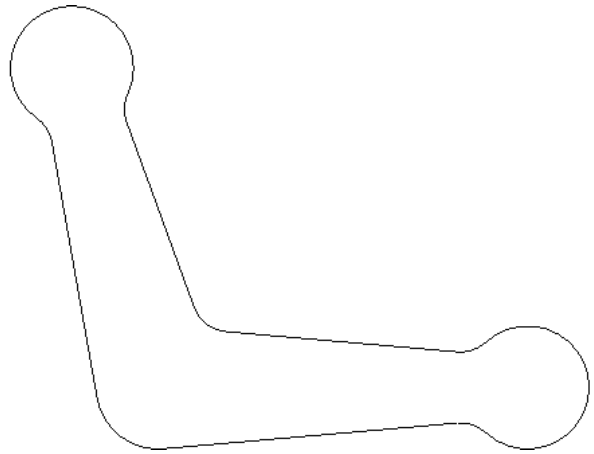
Tip: To turn off the automatic trim feature, choose **Trim** from the Fillet menu so that it is set to **N**.

5. Trim the last arc to the adjoining lines. Choose **Main Menu, Modify, Trim, 3 entities**.
6. Select the lines at positions 1 and 2, then the arc at position 3.



7. Delete the lines at positions 4 and 5.
8. Delete the point at position 6.

   The completed part should look like the following picture.



▶ *Saving the file*

1. Choose **Main Menu, File, Save**.
2. Save the file as **elbow1.mc9** in your working folder.

Exercise 2 – Creating the contour toolpath

This exercise shows you the basic steps for creating a toolpath and posting it to an NC file that can be read by your machine tool. In this exercise, you will create a contour toolpath. In a contour toolpath, the tool follows the shape of a curve or chain of curves. You will use the part you created in Exercise 1.

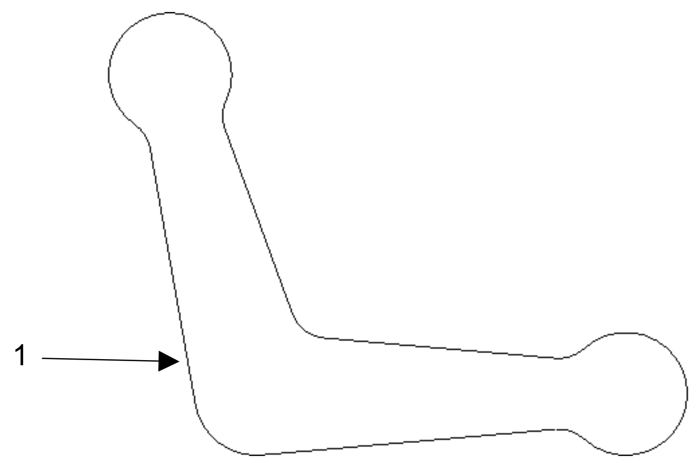
In this exercise, you will learn the following skills:

- **Creating a contour toolpath**
- **Chaining geometry**
- **Choosing a tool and setting toolpath parameters**
- **Selecting a tool library**
- **Using the backplot function to preview a toolpath**
- **Posting the toolpath to an NC file**

▶ *Choosing the toolpath and chaining the geometry*

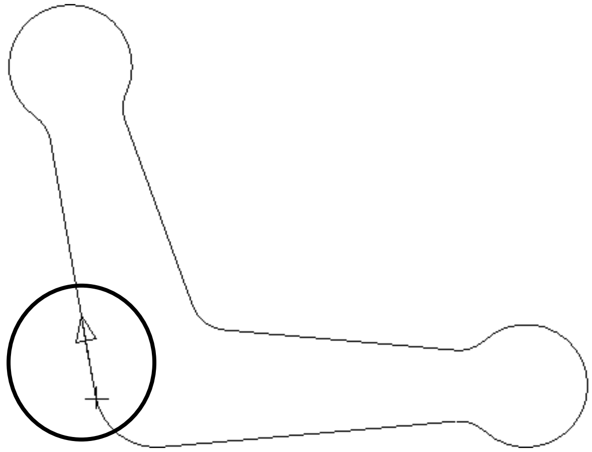
Chaining is the process of selecting geometry for a toolpath or other Mastercam function. A chain is a set of curves (lines, arcs and/or splines) that have adjoining endpoints (points can also be chained). A toolpath can have more than one chain.

1. If necessary, open the file from the previous exercise, **elbow1.mc9**.
2. Choose **Main Menu, Toolpaths, Contour**.
3. Select the line at position 1 to start the chain. You should see the whole part highlight.



To help you select the right geometry, Mastercam highlights the line when the cursor is close to it. After selecting the line properly, you will see an arrow display as shown in the following picture.

Tip: The arrow indicates the chaining direction. This is the direction the tool will move when you create the toolpath.

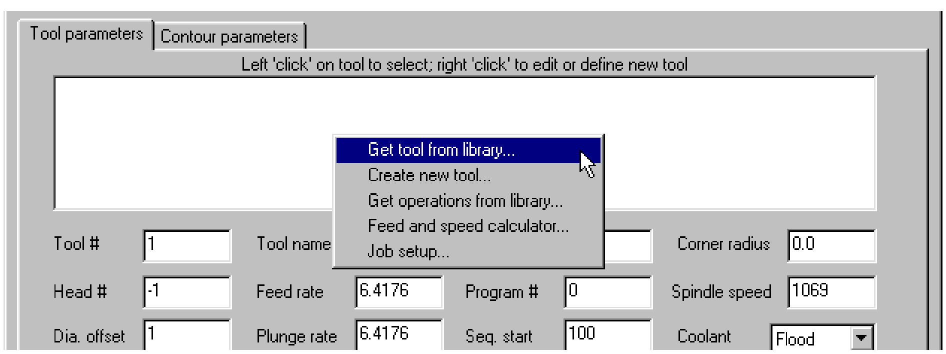


4. Choose **Done**.

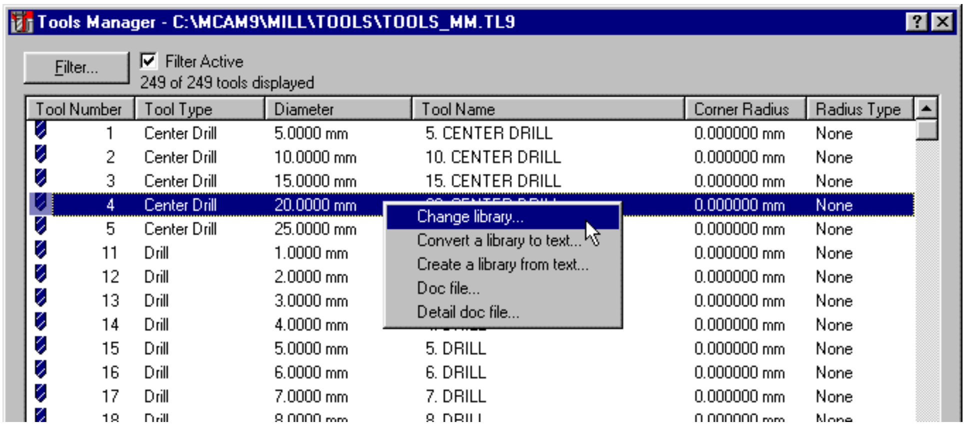
▶ *Selecting a tool*

As soon as you finish selecting geometry for the toolpath, Mastercam automatically prompts you to select a tool and enter parameters. Each toolpath can use only one tool. Mastercam organizes tool definitions into libraries. You can have as many libraries as you wish. In this procedure, you will also learn how to select a different tool library.

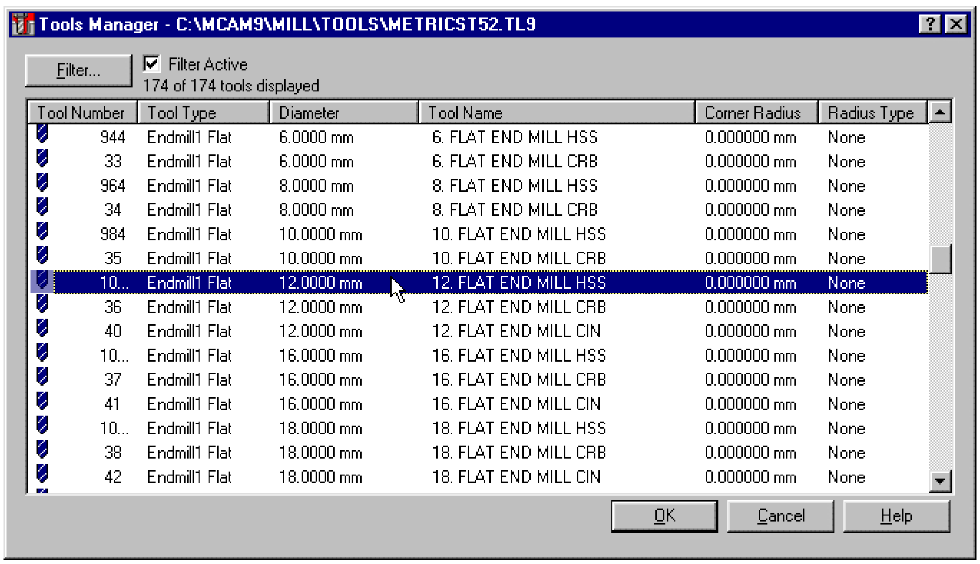
1. Mastercam automatically shows you the Tool parameters tab. Right-click in the large white area and choose **Get tool from library** as shown in the following picture.



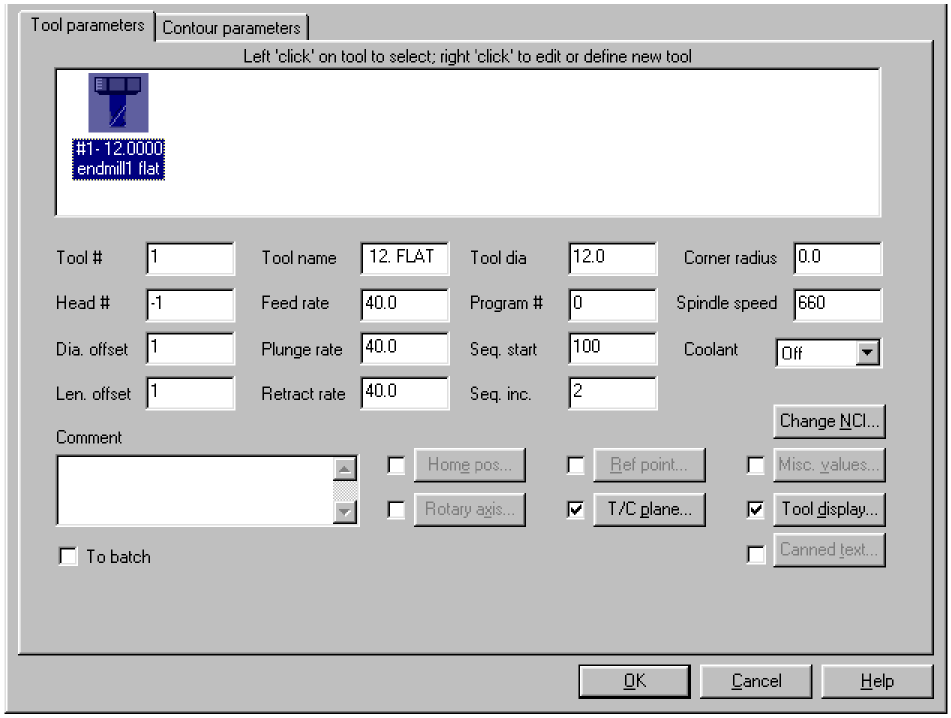
2. Right-click in the tool list and choose **Change library**.



3. Select the file **MetricST52.tl9** and choose **Save**.
4. Select the 12 mm HSS (high-speed steel) flat endmill as shown in the following picture.



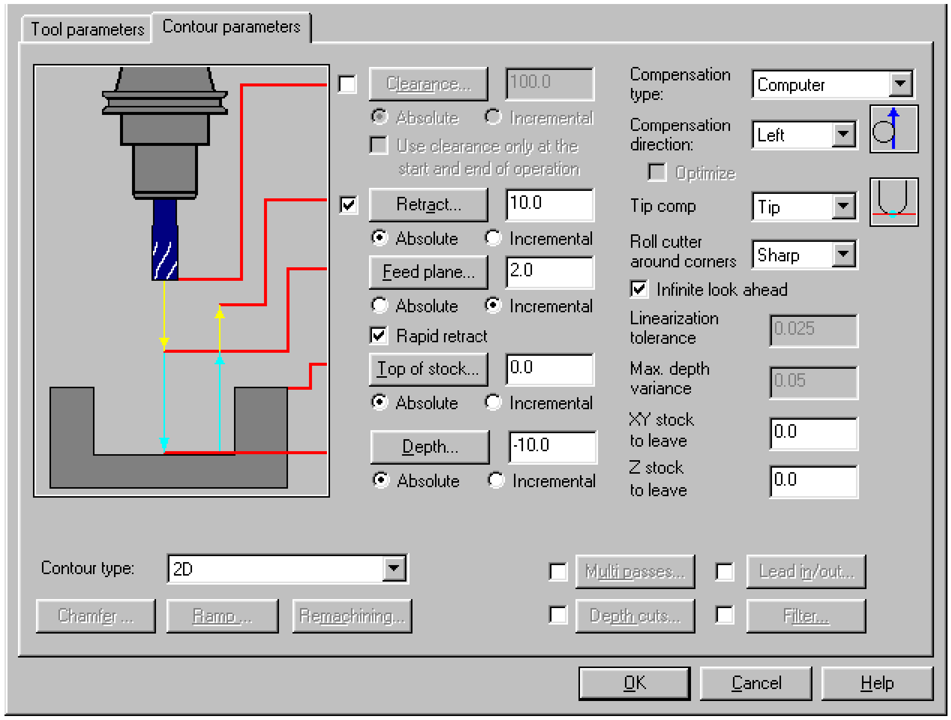
5. Choose **OK**. The tool appears in the tool display area as shown in the following picture.



▶ *Entering toolpath parameters*

Mastercam automatically fills in many of the fields with default values. For this toolpath, you will use the default values for all the Tool parameters, and edit the Contour parameters.

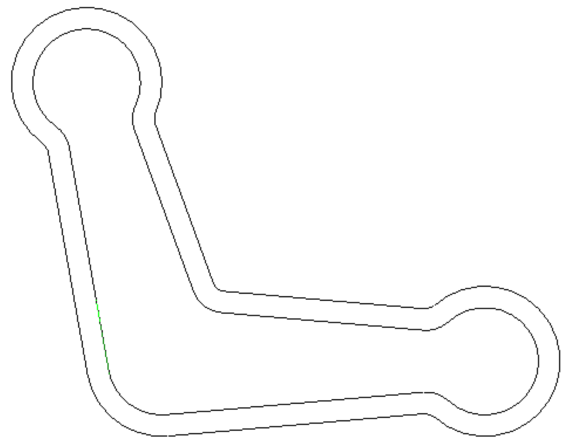
1. Choose the **Contour parameters** tab.
2. Enter a **Retract** height of **10**.
3. Enter a **Feed plane** of **2**.
4. Enter a **Depth** of **–10**. Make sure your other parameters match the following picture.



The parameters as shown in the preceding picture instruct the tool to make the following motions:

- Rapid from the home position to 10 mm above the starting position of the toolpath (the Retract height).
- Rapid straight down to 2 mm above the stock (the Feed plane).
- Plunge into the stock to a depth of –10 mm at the Plunge rate, 40 mm/min., as set on the Tool parameters tab. Because the feed plane is set to 2 mm above the part, the total plunge distance is 12 mm.
- Feed around the part at the feed rate of 40 mm/min., as set on the Tool parameters tab. Because the compensation is set to Left, the tool will be offset from the part geometry by its radius, 6 mm.
- When the tool returns to the original starting point, it will rapid to the retract height, since the **Rapid retract** check box is selected.

5. Choose **OK** to generate the toolpath. It should look like the following picture.



Note: Remember that the online help has complete descriptions of all the fields, buttons, and options on each dialog box.

▶ *Backplotting to view the toolpath*

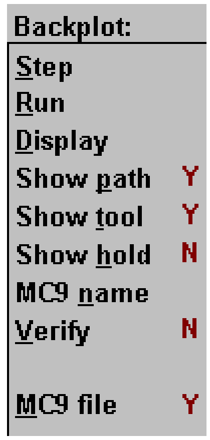
Mastercam has two functions that you can use to preview toolpaths and operations and catch errors before you create the NC program:

- Backplot, which gives you a precise view of specific tool movements.
- Verify, which gives you a better view of stock removal.

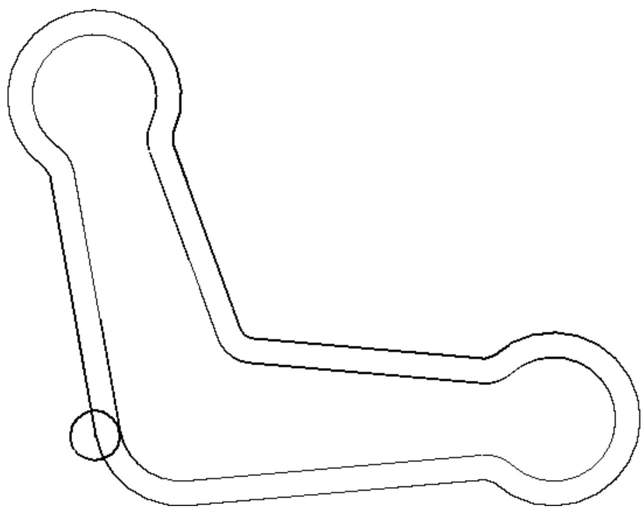
This exercise will show you how to backplot your toolpath (you will use the Verify function in the next chapter).

1. Choose **Operations** to open the Operations Manager.
2. Choose **Backplot**.
3. Make sure the settings on the Backplot menu match the following picture.

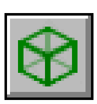
Tip: To change a setting from Y to N, click on the menu option or type the underlined letter.



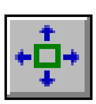
13. Choose **Step** from the Backplot menu or press [S] repeatedly. Mastercam will step through the toolpath. It should look like the following picture.



14. You will see a confirmation message when the backplot has finished. Choose **OK**.

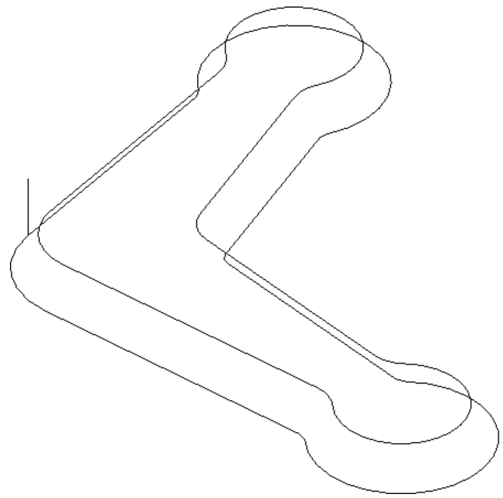


15. You can also preview the tool motion in 3D. Choose the green **Gview–Isometric** button from the toolbar to look at the part in isometric view.



16. If necessary, choose the **Screen–Fit** button to fit the part in the screen.

17. Press [S] again to backplot through the toolpath. Now you can see the plunge and retract moves clearly. Notice that the rapid moves are in yellow and the feed moves are in light blue.



18. When the backplot is complete, choose **OK**.

Exercise 3 – Making changes to the toolpath

This exercise shows you how to make changes to your part or toolpath and automatically regenerate your operation. In this exercise, you will make the following changes:

- Edit the toolpath parameters to add entry and exit moves
- Change the part geometry
- Switch to a different tool

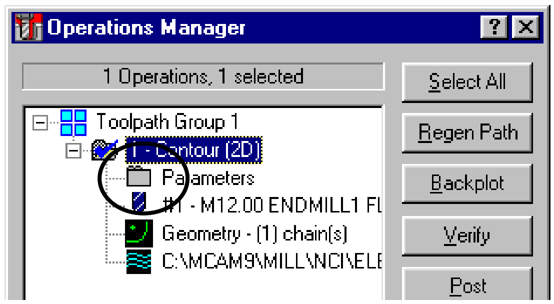
After you've made all the changes, you will post the toolpath to an NC file. You will learn the following skills:

- **Using the Operations Manager to edit toolpaths**
- **Changing toolpath parameters**
- **Regenerating operations**
- **Adding entry and exit (lead in/out) moves**
- **Using a selection mask**
- **Posting a toolpath to an NC file**

▶ *Adding lead in/out moves*

For this part, you need to change how the tool enters the material. Plunging directly into the part is not desirable because of the dwell marks left behind at the tool entry spot. In this exercise, you add entry and exit moves to the toolpath to eliminate the dwell marks.

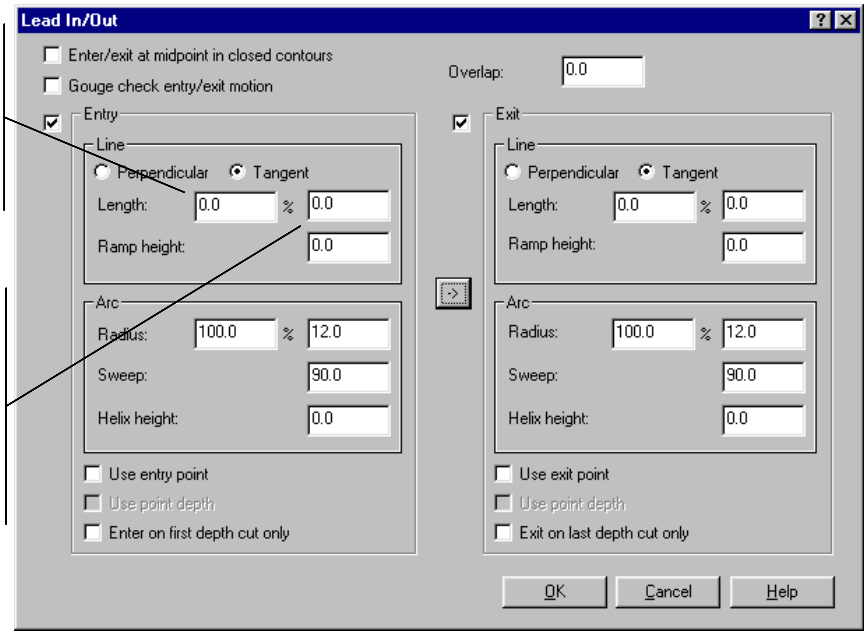
1. Press [Esc] to return to the Operations Manager.
2. Choose the **Parameters** icon.



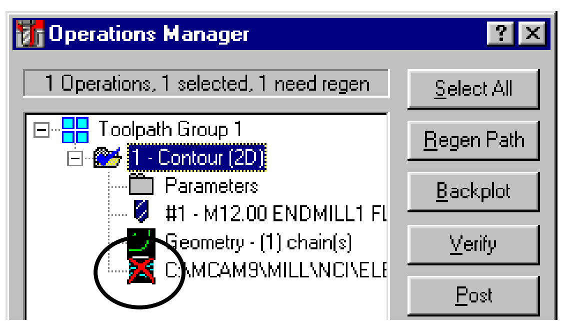
3. Choose the **Lead in/out** check box and button.
4. The Lead in/out dialog box lets you specify entry and exit moves: either lines, arcs, or a combination of both. For this part, you want to use just arcs, so enter **0** in the **Line–Length** field in the Entry section to disable line moves. (You will use the default arc dimensions.)
5. Choose the  button to copy the Entry arc dimensions to the Exit section. Make sure your settings match the following picture.

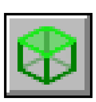
You can enter the line length in either of two ways. You can type a percentage of the tool diameter here…

…or type the absolute length in here. When you type a number in one field, the other automatically updates. (The **Arc Radius** works the same way.)

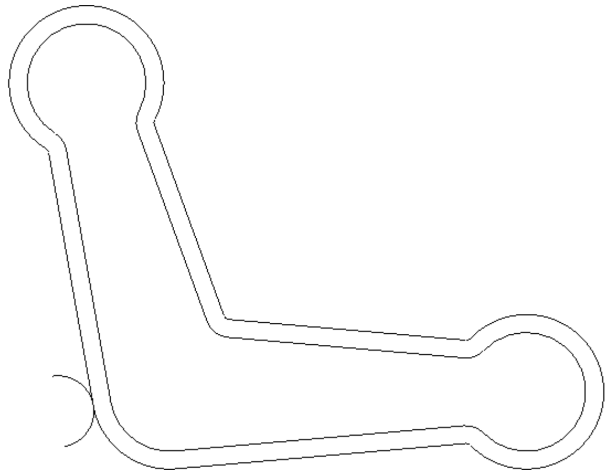


6. Choose **OK** twice.
7. When you return to the Operations Manager, you will see a red X as shown in the following picture. This means that some part of the toolpath has changed (in this case, you've added the lead in/out moves) and the operation needs to be regenerated. Choose the **Regen Path** button.



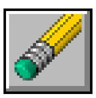


8. Choose **OK** to close the Operations Manager.
9. Choose the green **Gview–Top** button from the toolbar. The new toolpath should look like the following picture.

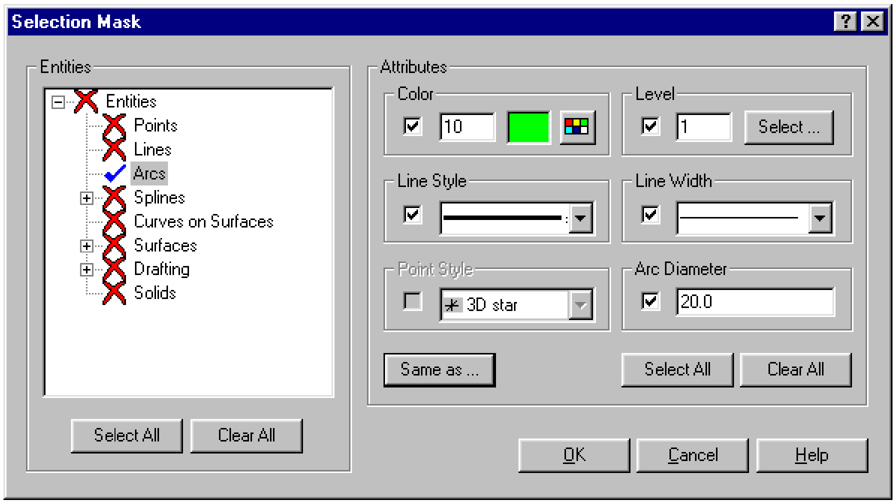


▶ *Changing the part geometry*

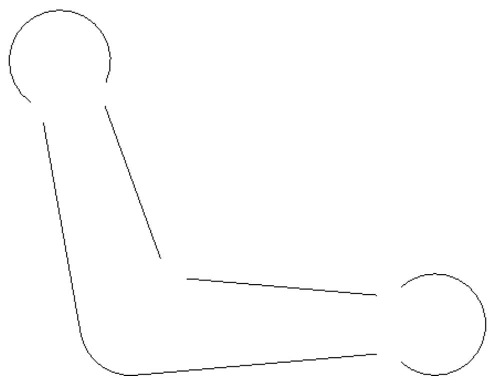
In this procedure, you will make a design change to the part, changing the 10 mm radius fillets to 6 mm fillets.



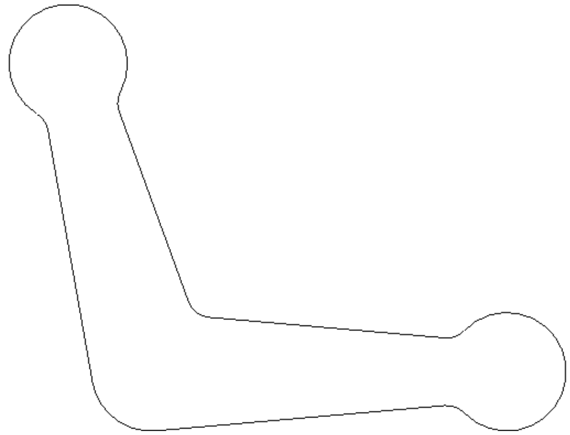
1. Choose **Delete** from the toolbar.
2. Choose **All, Mask**.
3. The Selection Mask dialog box lets you describe which types entities to delete. In the **Entities** list, choose **Arcs**.
4. Choose **Same as**.
5. Select any of the 10 mm fillets. When you return to the Selection Mask dialog box, you see that all of the fields are filled in with the attributes of the 10 mm fillet. Mastercam will use this mask to select all of the fillets and delete them.



6. Choose **OK**.
7. Choose **Yes** at the confirmation prompt. Your part should look like the following picture.



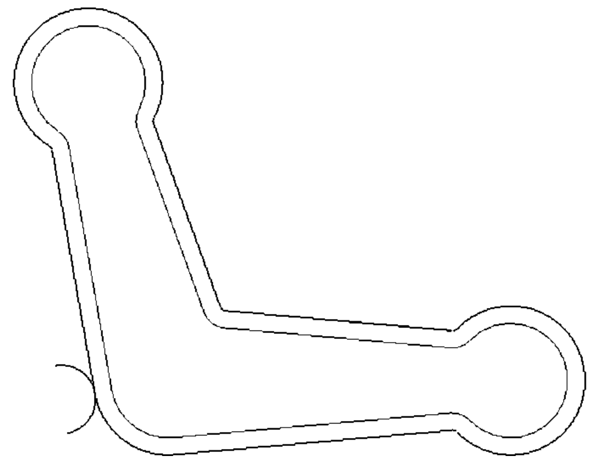
8. Create 6 mm fillets in all of the gaps. (See page 24 if you don't remember how to create fillets.) Your part should look like the following picture.



▶ *Changing the tool*

When you created the toolpath for this part, you used a 12 mm endmill. Since the fillets are now smaller and the same radius as the tool, you will switch to a smaller tool so you can get smoother tool motion around the fillets.

1. Press [Alt + O] to open the Operations Manager.
2. Choose the **Parameters** icon.
3. Choose the **Tool parameters** tab.
4. Right-click in the tool display area and choose **Get tool from library**.
5. Select the 10 mm HSS flat endmill and choose **OK**.
6. Choose **OK** again to return to the Operations Manager.
7. Choose **Regen Path** to regenerate the toolpath with the new tool and new geometry. The new toolpath should look like the following picture.

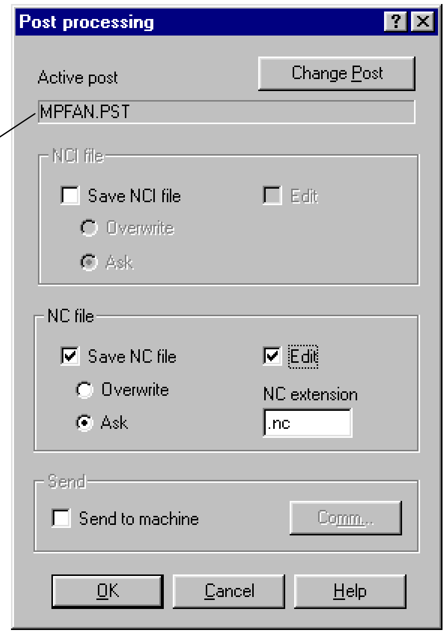


Creating an NC program

In order to cut a part on a CNC machine tool, you need to give it a program in a format that your control can read. The act of making this file (called an NC program) is called post processing, or posting. When you post a file, Mastercam runs a special program called a post processor that reads your Mastercam file and creates an NC program from it. Your original Mastercam file isn't changed.

1. Choose **Post**. (The Operations Manager window should still be open.)
2. Select the **Save NC file** check box, and choose the **Edit** option.
3. Choose the **Ask** option (this means that it will prompt you for a file name). Your dialog box should match the following picture.

Post processors are machine- and control-specific. When you installed Mastercam, you selected a default post processor. The current post processor is listed here. If you need to, you can select a different one by choosing **Change Post**.

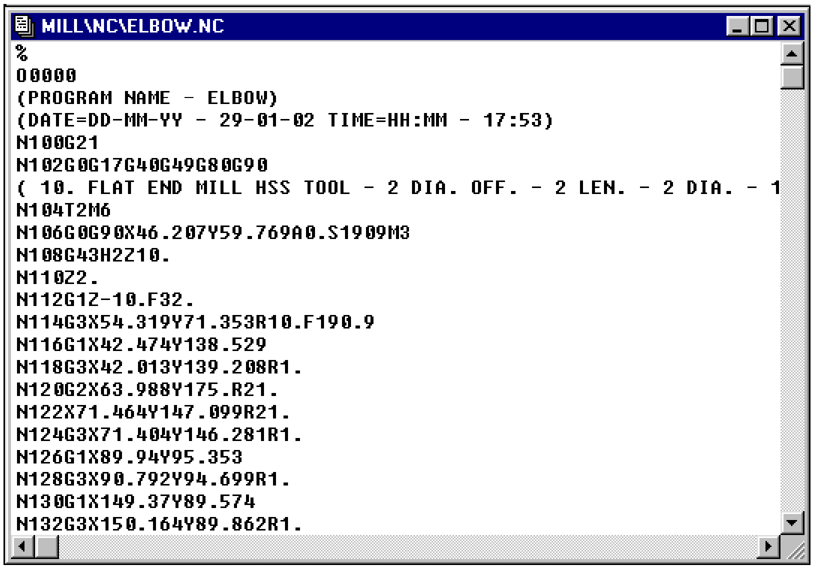


WARNING: Before running an NC program on your machine tool, you MUST ensure that it was created with the proper post processor. If the correct post processor was not used, you could crash your machine tool and cause serious injury or damage. Do NOT assume that the post processor shown in these examples is compatible with your own machine tool.

4. Choose **OK**.
5. Type in a file name when prompted. If you wish, you can navigate to a different folder; the default is **Mcam9\Mill\Nc**. Choose **Save** when you are done.

   Tip: Check your machine tool or control documentation to see what file names are allowed. For example, you might be limited to 8 characters or less.

6. After you save the file, it will appear in a text-editing window so you can review it or make changes, as shown in the following picture.

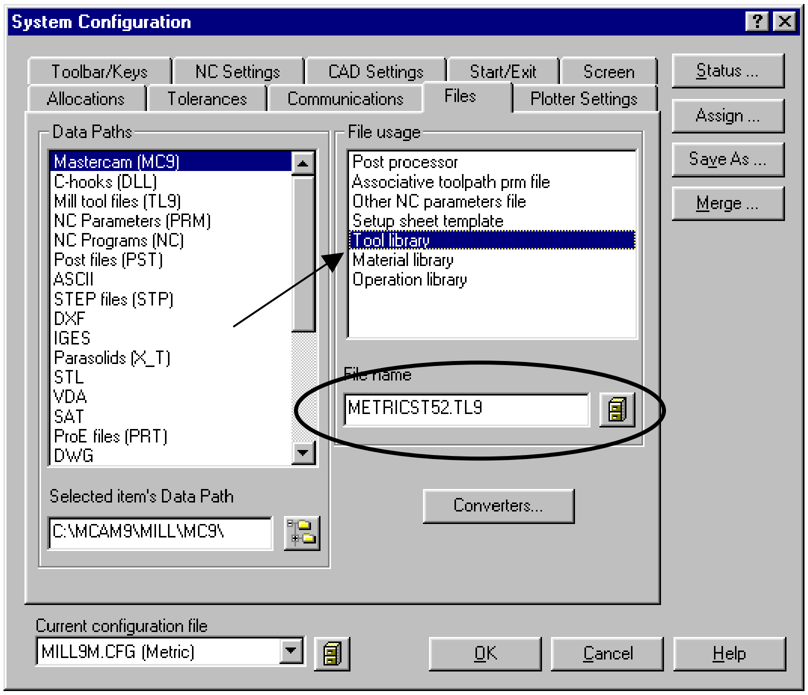


7. Close the NC program window to return to Mastercam.
8. Close the Operations Manager and press [Alt + A] to save the file.

▶ *Setting the default tool library*

The remaining exercises in this tutorial will use tools from the MetricST52.tl9 tool library that you selected earlier. In this procedure, you will make this the default tool library, so that you do not have to keep selecting it.

1. Choose **Main Menu, Screen, Configure**.
2. Choose the **Files** tab.
3. Choose **Tool library** in the **File usage** list.
4. Make sure that **METRICST52.TL9** appears in the **File name** field as shown in the following picture. If it doesn't, choose the **File**  button and select it.



5. Choose **Save As** to save the setting to the configuration file.
6. The file in which it is saved is shown in the **Current configuration file** field, **MILL9M.CFG**. Choose **Save**.
7. Choose **Yes** when asked to overwrite the current file.
8. Choose **OK**.

You've now seen all the major stages of creating a part and an operation to machine it. In the next chapter, you will use the simple operation you created in this chapter as a building block for more sophisticated operations.

4 Copying and Transforming Operations

This chapter shows you how to use the simple toolpath you created in the previous chapter as a building block for more sophisticated operations. You will create the following new operations:

- Finishing and multi-pass roughing operations
- A chamfering operation
- A mirrored copy of the operation

The part used in this chapter is the same one that you saved at the end of Chapter 3. If you did not complete Chapter 3, use the file **new elbow-mm.mc9**, in the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.

Tip: When opening this file, select the **Restore entire NCI on file get** option.

Exercise 1 – Creating roughing and finishing passes

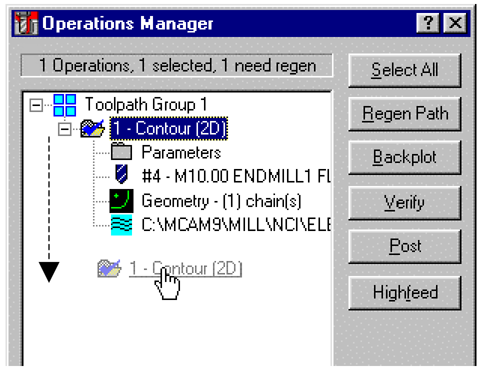
The 2D contour toolpath you created in the previous chapter only has a single cutting pass. You decide that it takes off too much stock for a single pass, so you decide to rough out the part in multiple passes with a larger tool. You will complete the part with a separate finishing operation. In this exercise, you will use the following skills:

- **Copying operations**
- **Creating multiple passes**
- **Creating finishing operations**
- **Changing tools and feed rates**

▶ *Copying operations*

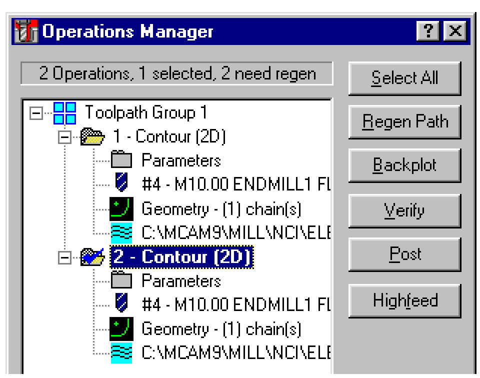
To create the separate operations for roughing and finishing with the minimum number of steps, you will copy the current 2D contour operation and then edit the parameters for each copy.

1. If necessary, open the part file you saved at the end of Chapter 3. If you did not complete Chapter 3, choose **File, Get** from the menu, and open the file **new elbow-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Press [Alt + O] to open the Operations Manager.
3. Right-click on the Contour folder icon and drag it below the NCI icon.

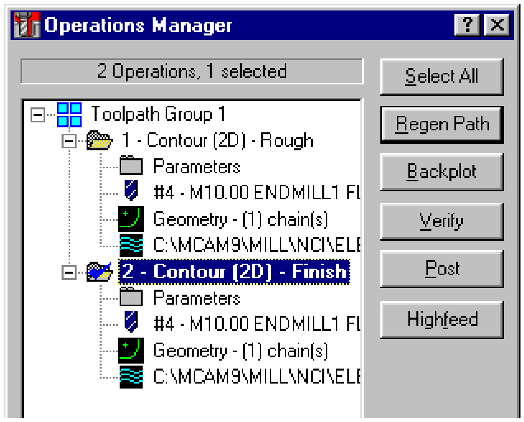


4. Release the mouse button and choose **Copy after**.

A copy of the operation appears as shown in the following picture.



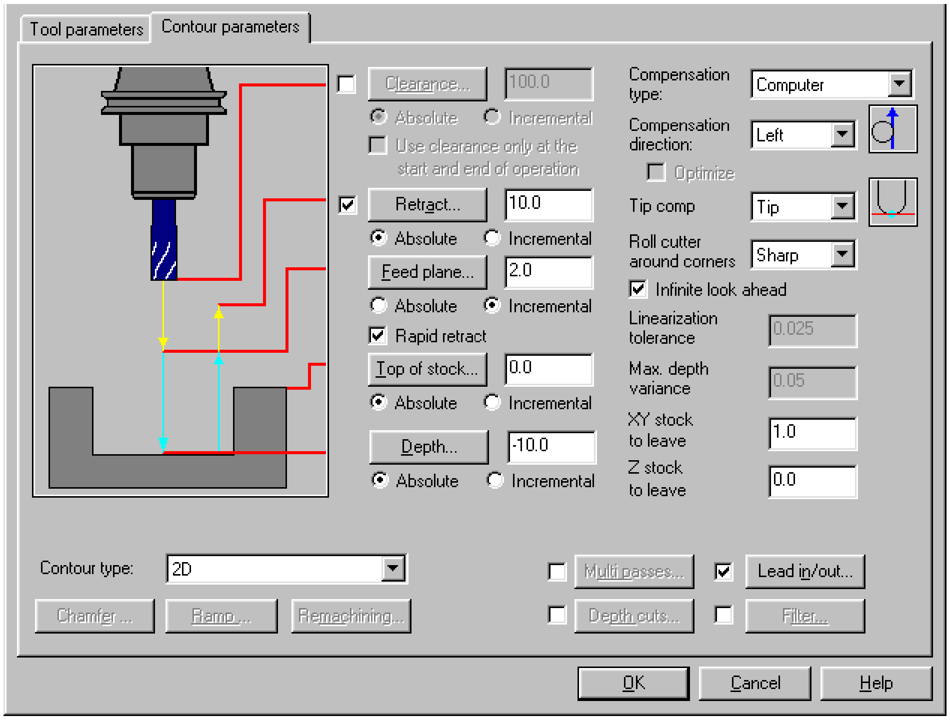
5. Click on the name of the first operation until it highlights for editing, and type in a new name: **Rough**
6. Repeat for the second operation and type the new name: **Finish**. The operations should look like the following picture.



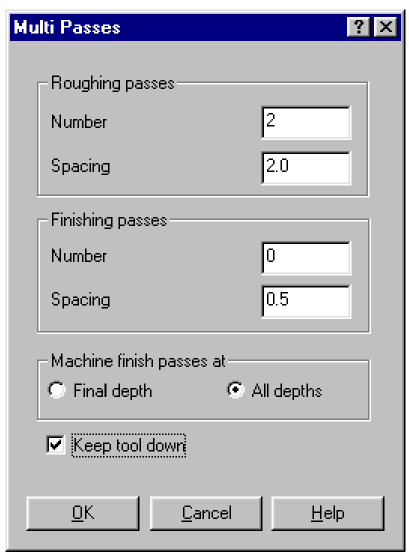
▶ *Setting the roughing parameters*

To make the first operation a true roughing operation, you will select a bigger tool for it and specify multiple passes.

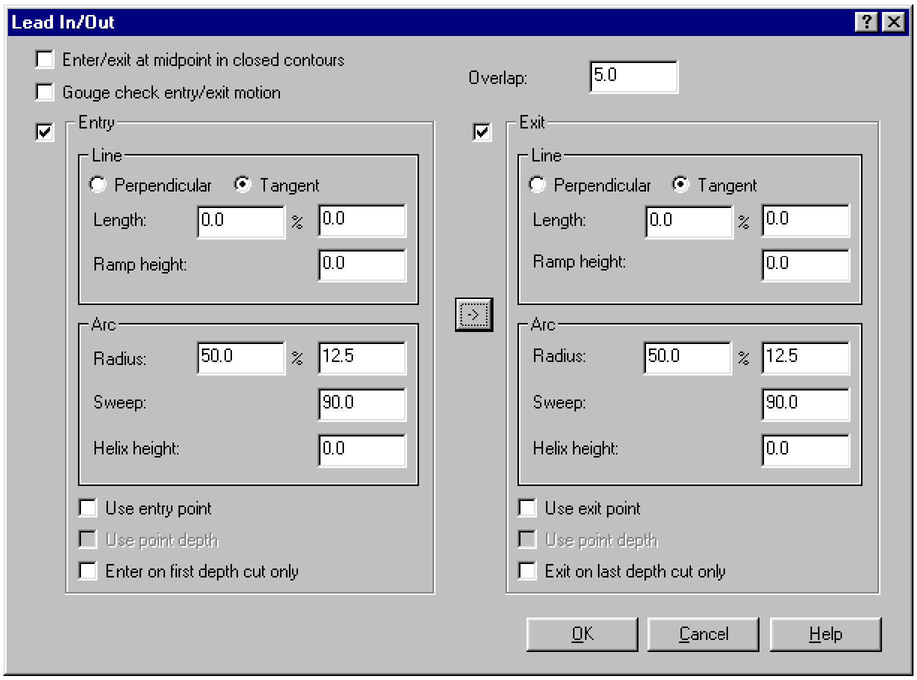
1. Choose the **Parameters** icon for the Rough operation.
2. Choose the **Tool parameters** tab.
3. Right-click in the tool display area and choose **Get tool from library**.
4. Select the 25 mm HSS flat endmill and choose **OK**.
5. Choose the **Contour parameters** tab.
6. Since this is a roughing pass, you should leave some stock for the finish operation. Enter **1** in the **XY stock to leave** field. Your contour parameters should match the following picture.



7. Select the **Multi passes** check box and button.
8. Enter **2** for the **Number** of **Roughing passes** and select **Keep tool down**. The rest of the values should match the following picture.



9. Choose **OK**.
10. Choose **Lead in/out**.
11. Enter an **Overlap** of **5**. This means that the entry and exit arcs will overlap by this distance.
12. Change the **Entry Arc–Radius %** to **50**.
13. Choose the  button to copy the settings to the Exit section. Your values should match the following picture.



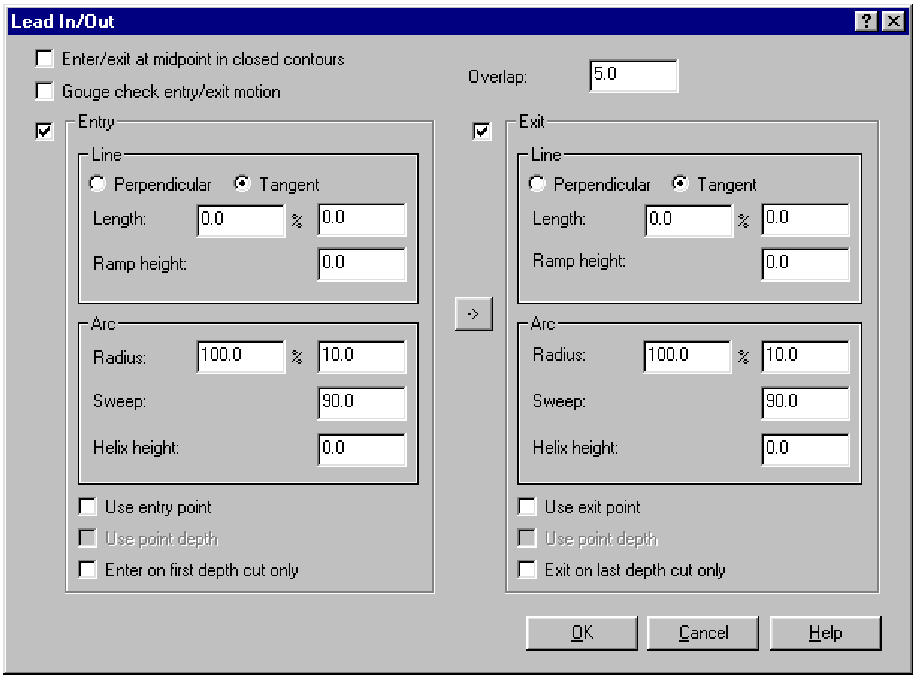
14. Choose **OK** twice.

▶ *Setting the finish parameters*

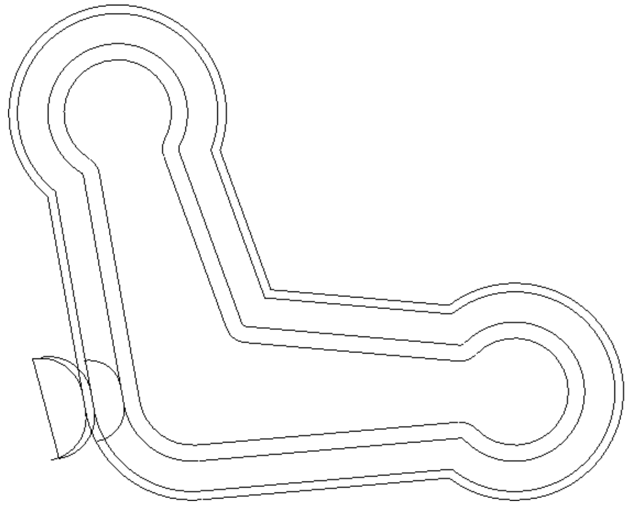
Since the second operation will be a finish operation, you will keep the original tool, but will use a slower feed rate. Also, you will edit the lead in/out moves so they have the same overlap as the roughing cuts.

1. Choose the **Parameters** icon for the Finish operation.
2. Choose the **Tool parameters** tab.

3. Enter a **Feed rate** of **20**.
4. Choose the **Contour parameters** tab.
5. Choose **Lead in/out**.
6. Enter an **Overlap** value of **5**. Make sure the other values match the following picture.



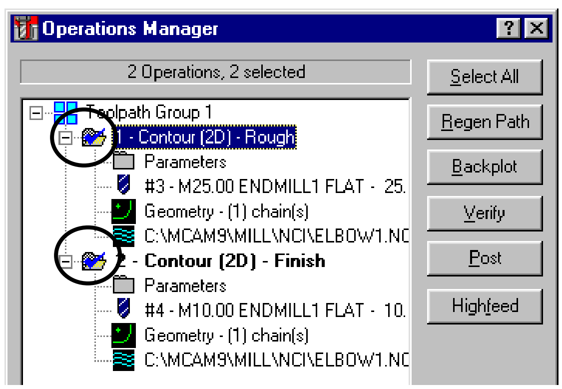
7. Choose **OK** twice to return to the Operations Manager.
8. Choose **Select All**.
9. Choose **Regen Path** to regenerate both operations with the new parameters. The new toolpaths should look like the following picture.



▶ *Backplotting the new toolpaths*

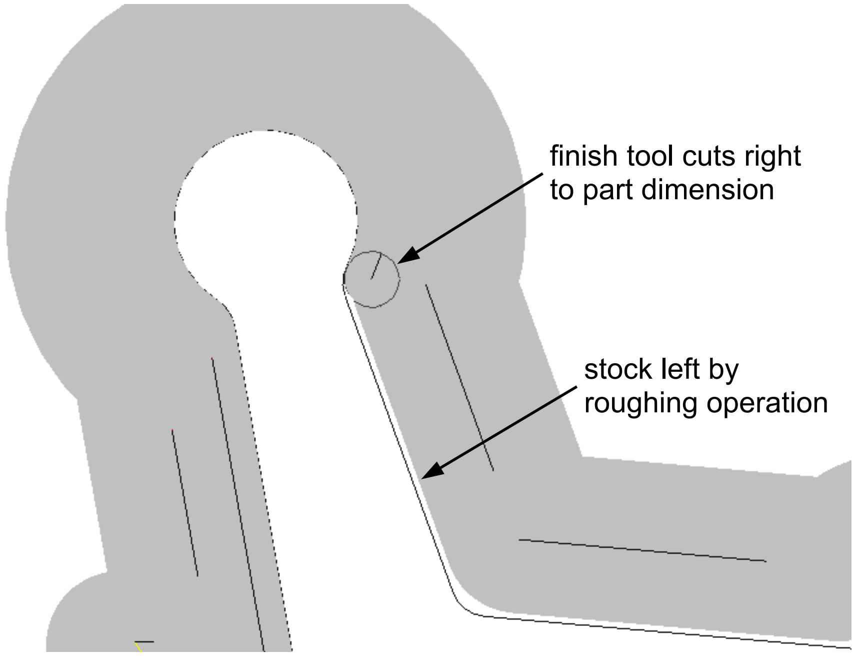
1. Choose **Backplot**. Make sure both operations are still selected as shown in the following picture.

Tip: The blue check marks indicate which operations are selected.



2. Set the **Verify** option to **Y**.
3. Press [S] to step through the toolpath. Notice how the stock is removed after each pass, and how the finish tool cleans out the areas that the roughing tool cannot reach.

   The following picture shows you a snapshot midway through the final finish operation. You can see the stock left by the roughing operation and how the finish operation is cutting right to the blueprint dimension.



4. Press **OK** when the backplot is finished.
5. Choose **Backup** to return to the Operations Manager, and choose **OK** to close it.
6. Choose **Main Menu, File, Save** and save the file in your working folder as **elbow2.mc9**.

Exercise 2 – Creating a contour chamfer

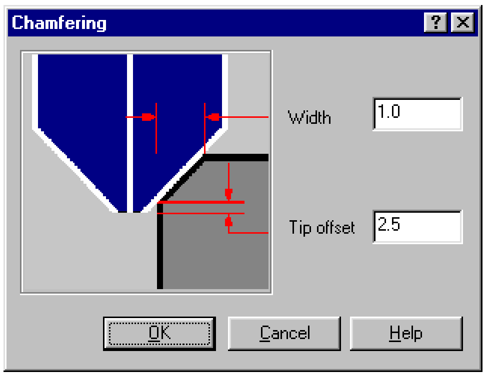
Next, you would like to add a chamfer to the contour. Mastercam has contour toolpath options that let you easily create a chamfer by specifying some simple dimensions. You will create a separate operation for the chamfer that uses a chamfer tool. You will learn the following skills:

- **Creating 2D chamfer operations**
- **Using the Verify feature to preview stock removal**

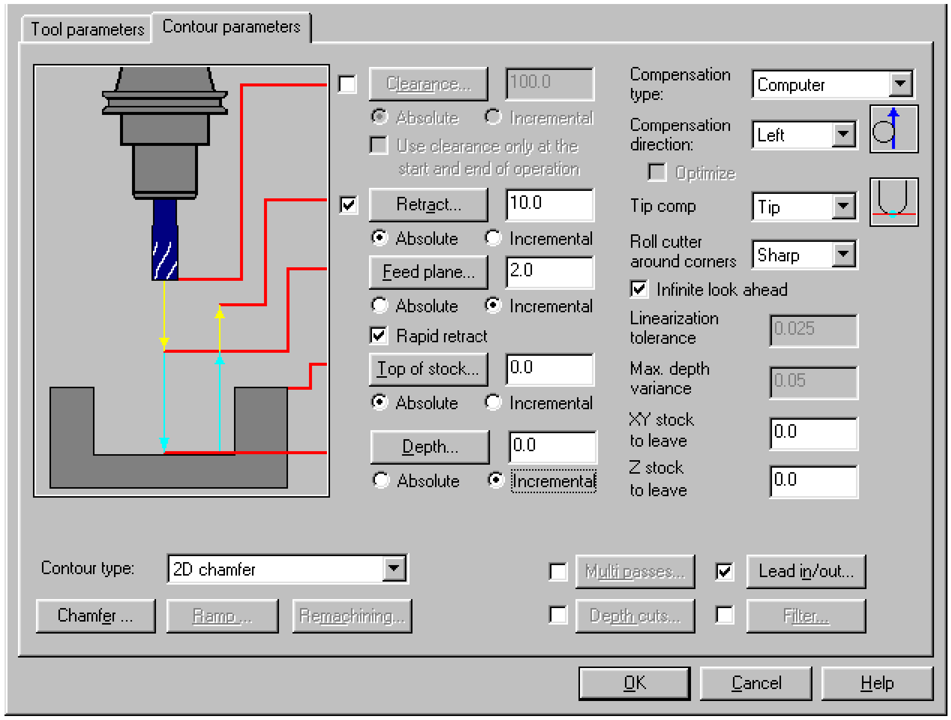
▶ *Creating the chamfer operation*

You will create the chamfer operation in the same way that you created the finishing operation in the previous exercise, by copying an existing operation and editing its parameters.

1. Press [Alt + O] to open the Operations Manager.
2. Make a copy of the Finish operation and name the copy **Chamfer**. (See page 45 if you don't remember how to do this.)
3. Choose the **Parameters** icon for the new Chamfer operation.
4. Choose the **Tool parameters** tab.
5. Select the 10 mm HSS chamfer mill from the tool library.
6. Choose the **Contour parameters** tab.
7. In the **Contour type** drop-down list, select **2D chamfer**.
8. Choose the **Chamfer** button.
9. Enter **1** for the **Width**, and **2.5** for the **Tip offset**. Your selections should match the following picture.



10. Choose **OK** to return to the Contour parameters dialog box.
11. Enter **0.0** for the **Depth**, and choose **Incremental**. The actual cutting depth achieved by the chamfer mill is determined by the width and tip offset you entered in step 9. Your contour settings should match the following picture.

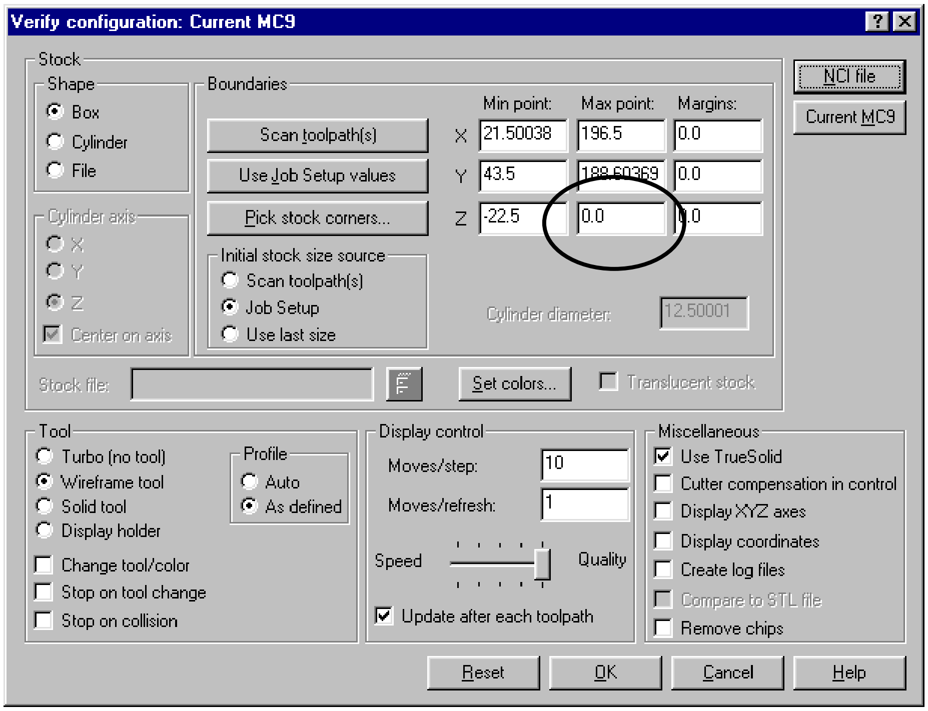


12. Choose **OK** to return to the Operations Manager.
13. Choose **Regen Path** to create the toolpath.

▶ *Using the Verify feature to preview stock removal*

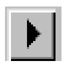
In previous exercises, you used the Backplot function to preview toolpath motion. In this exercise, you will use the Verify function in the Operations Manager instead. This function gives you a better picture of the 3D part.

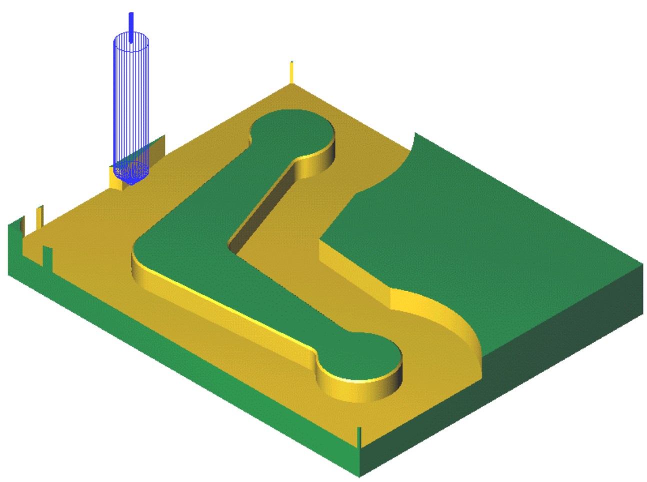
1. While inside the Operations Manager, choose **Select All** so you can verify all the operations.
2. Choose **Verify**.
3. Choose the  button to set the Verify configuration.
4. Enter **0** for the **Z–Max point.** Your values should match the following picture.



Tip: In later chapters you will see how to use Job Setup to build a stock model.

5. Choose **OK**.

6. Choose the  button. The part should look like the following picture.



7. Choose the  button on the Verify toolbar to end the Verify session and return to the Operations Manager.
8. Choose **OK** to close the Operations Manager.

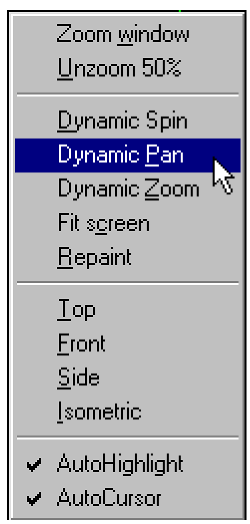
Exercise 3 – Mirroring the part and toolpath

You are required to manufacture both left-hand and right-hand versions of the part. You can do this by mirroring the part and toolpath. This lets you maintain the original toolpath parameters and machining direction for all the operations, ensuring that the duplicated part has the identical finish and size as the original. In this exercise, you will learn the following skills:

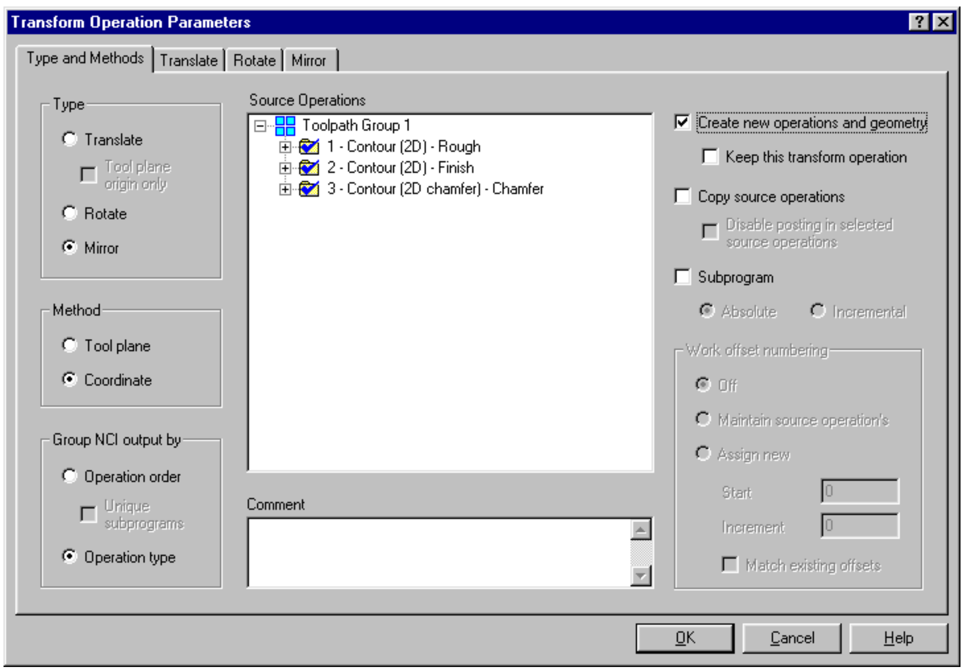
- **Creating mirror images of parts and toolpaths**
- **Re-ordering operations in the Operations Manager for greater machining efficiency**

▶ *Mirroring the part*

1. Press the [Page Down] key several times to zoom out from the part.
2. Right-click anywhere in the graphics window and choose **Dynamic Pan** from the menu.

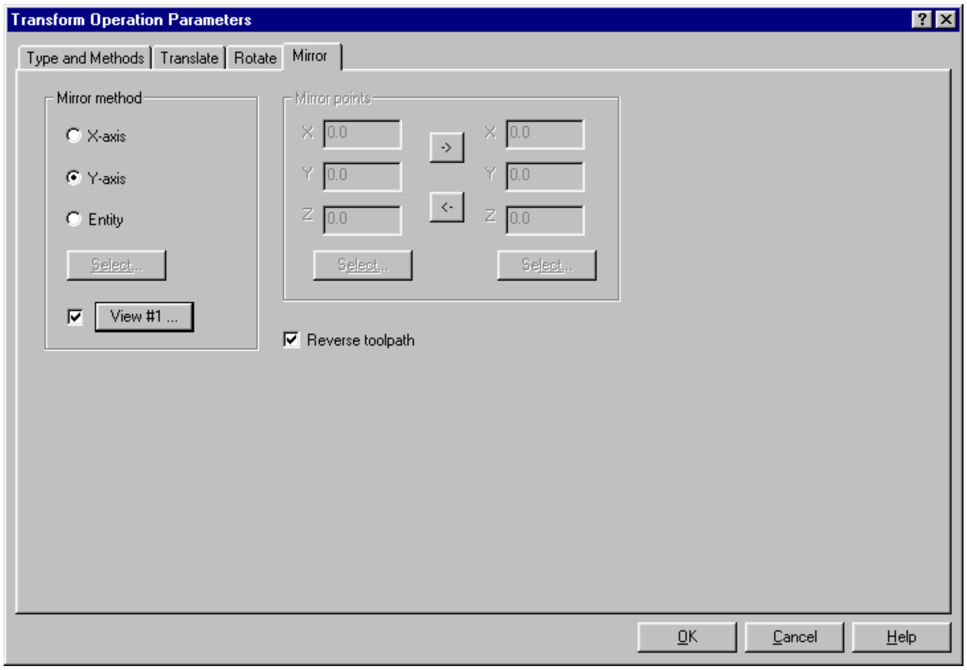


3. Click and drag to the right until the part is at the right edge of the screen.
4. Click again to exit dynamic panning.
5. Press [F9] to display the coordinate axes.
6. Choose **Main Menu, Toolpaths, Next menu, Transform**.
7. Choose **Toolpath Group 1**. This selects all the operations.
8. Choose **Type–Mirror**.
9. Choose **Create new operations and geometry**. Make sure your other selections should match the following picture.

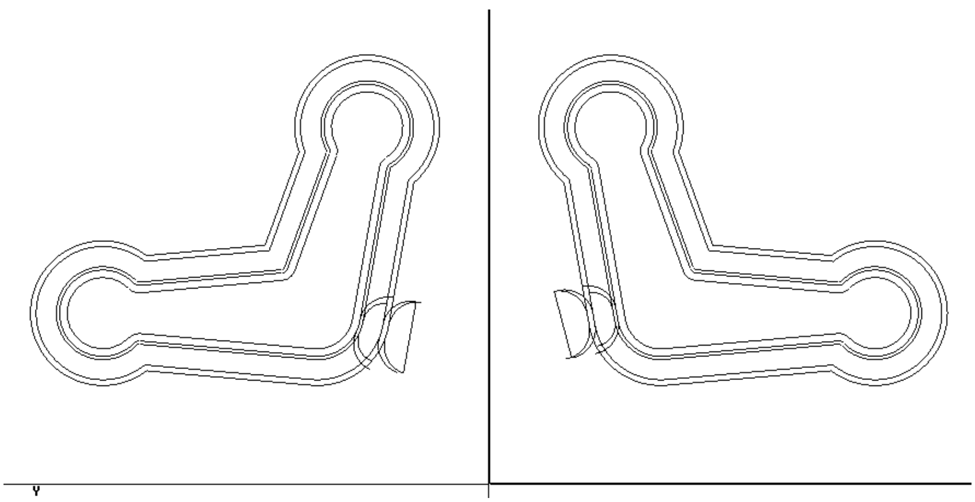


10. Choose the **Mirror** tab.
11. Choose **Reverse toolpath**. The original toolpath used climb milling; selecting **Reverse toolpath** means that the mirrored part will also use climb milling, so the finish on both parts will match.

    The Mirror tab should match the following picture.



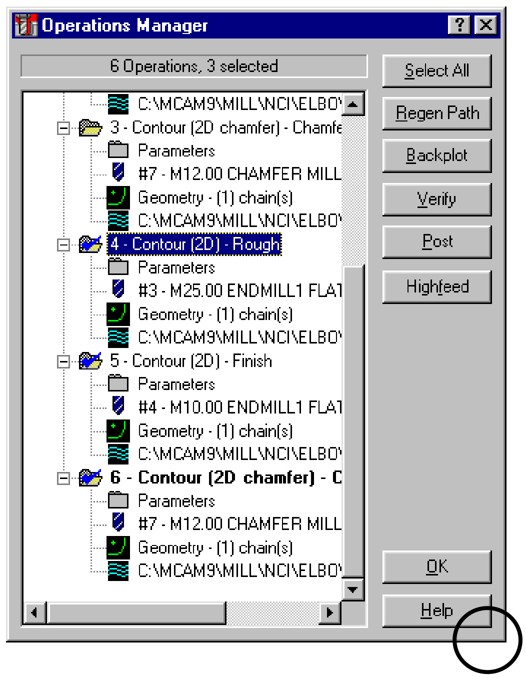
12. Choose **OK**. The part and toolpaths should look like the following picture.



▶ *Reordering operations to minimize tool changes*

Because you selected **Create new operations and geometry** when you mirrored the part, Mastercam added three new operations to the operations list: a separate roughing, finishing, and chamfering operation for the mirrored part. The way the operations are ordered now, Mastercam will rough the first part, then finish and chamfer it before roughing the second part, resulting in unnecessary tool changes. In this procedure, you will rearrange the operations so that the roughing, finishing, and chamfering operations are grouped to minimize tool changes.

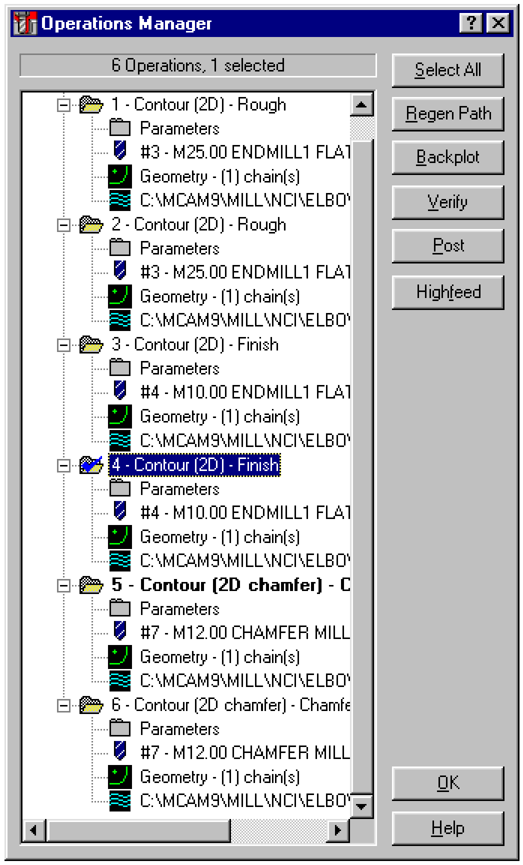
1. Press [Alt + O] to open the Operations Manager, and scroll down to the bottom of the list. You can see the new operations 4, 5, and 6.



2. Click and drag the corner of the Operations Manager window as shown in the preceding picture to make it larger, so you can see all the operations.

3. Click on the second rough operation (Operation 4) and drag it on top of the first rough operation.

4. Click on the second finish operation and drag it on top of the first finish operation. Your operations should be in the following order.



5. Choose **Select All**, **Backplot**.

6. Choose **Run**. You should see the operations machined in the proper order.

7. When the backplot is finished, close the Operations Manager and save the file.

In this chapter, you saw how to mirror parts and toolpaths. In the next chapter, you'll learn techniques for rotating geometry and operations around a center point so you can easily draw and machine circular parts.

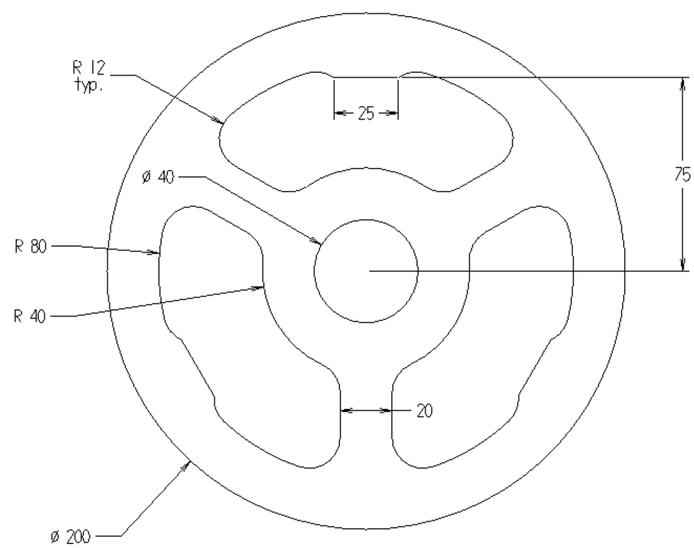
5 Rotating Geometry and Toolpaths

This chapter builds on the skills learned in previous chapters by focusing on a circular part, showing you techniques for orienting and rotating geometry around a center point. After you create the part, you will learn how to rotate toolpaths as well.

Exercise 1 – Creating the geometry

In this exercise, you will design a wheel with three symmetrical slots as shown in the following blueprint. You will use the following skills:

- **Creating arcs, tangent arcs, and lines**
- **Rotating geometry**
- **Trimming geometry**



▶ Creating the inner and outer circles

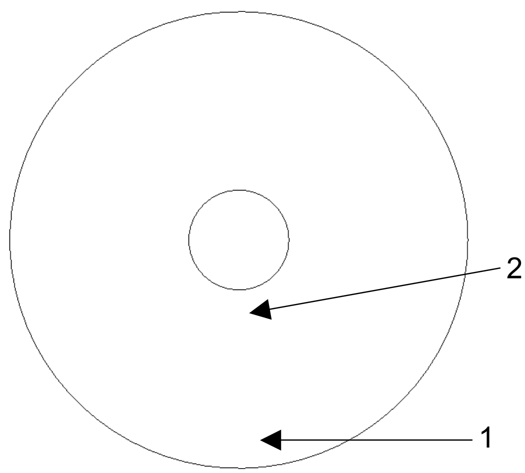
1. If necessary, create a new file. Choose **Main Menu, File, New**.
2. Choose **Main Menu, Create, Arc, Circ pt+dia**.
3. Enter **200** for the diameter.
4. Press [O] to select the origin for the center point.

   Note: Press the letter "O," not zero.
5. Press [Esc] and re-select **Circ pt+dia.**
6. Enter **40** for the diameter of the inner circle.
7. Press [O] again to select the origin for the center point.
8. Press [Esc] to exit the Create Arc function.
9. Right-click anywhere in the graphics window and choose **Fit screen** from the menu.

▶ Creating construction lines for the slot

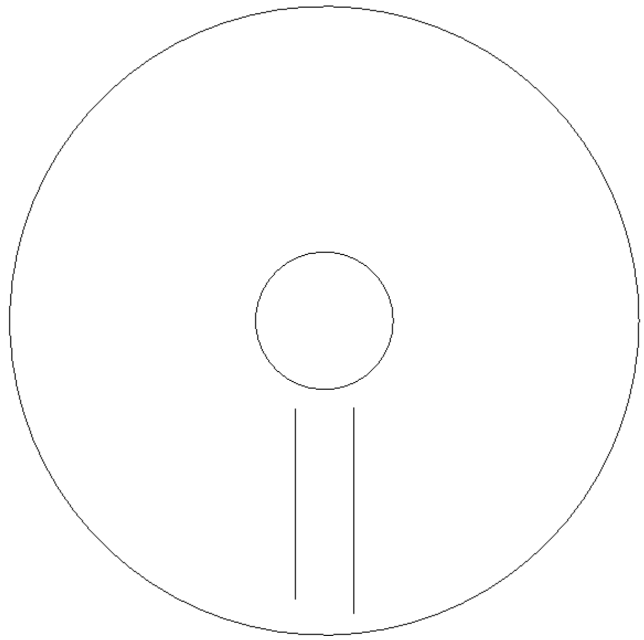
To rough out the slot, begin by defining the vertical edges, then rotating to the proper position.

1. Choose **Main Menu, Create, Line, Vertical**.
2. Select at position 1 then at position 2 (these are only approximate positions) to draw the construction guide.

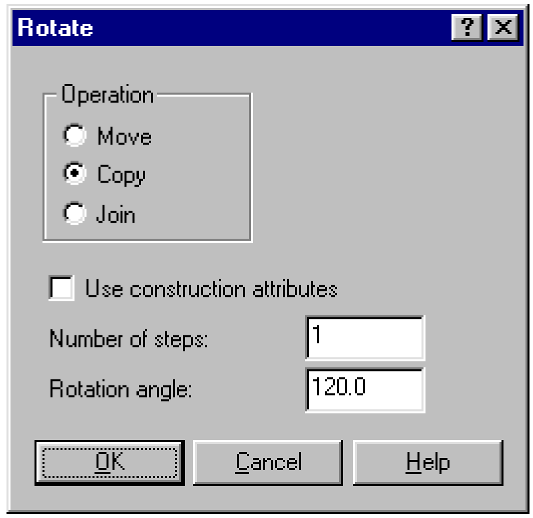


3. Enter **10** for the X coordinate of the line.

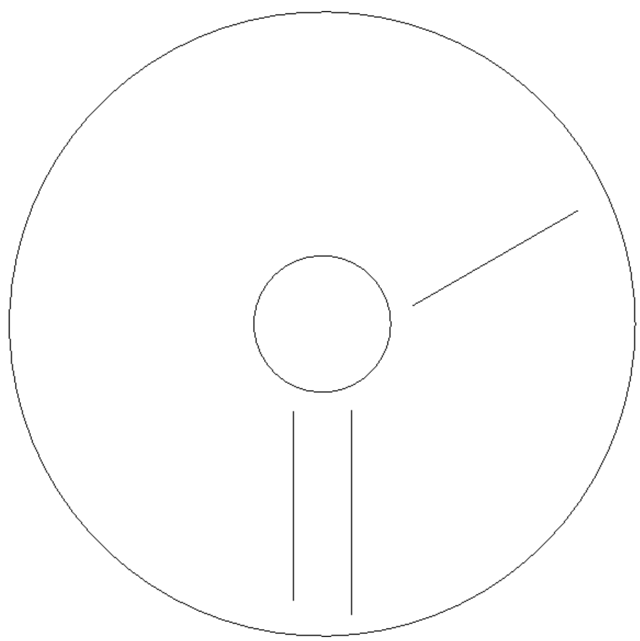
4. Create a second vertical line by selecting near positions 1 and 2 again.
5. Enter **–10** for the X coordinate of the line. The part should look like the following picture.



6. Next, you will rotate one of the lines to form the other edge of a slot. Choose **Main Menu, Xform, Rotate**.
7. Select the left line.
8. Choose **Done, Origin**.
9. Choose **Operation–Copy**.
10. Enter a **Rotation angle** of **120**. Your values should match the following picture.



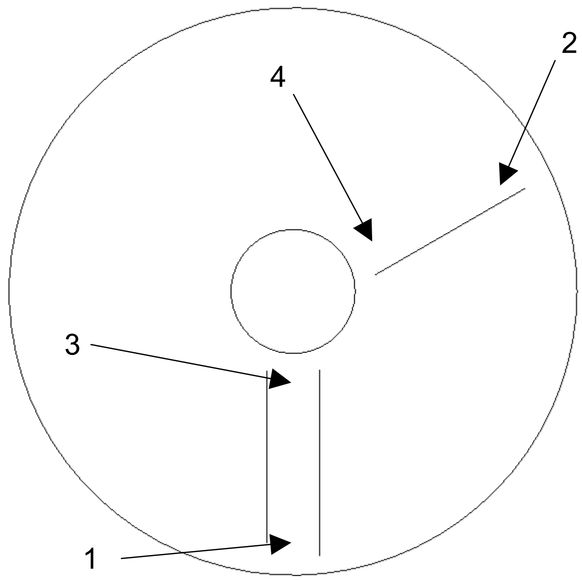
11. Choose **OK**. The line should rotate as shown in the following picture.



▶ *Creating the arcs for the slot outline*

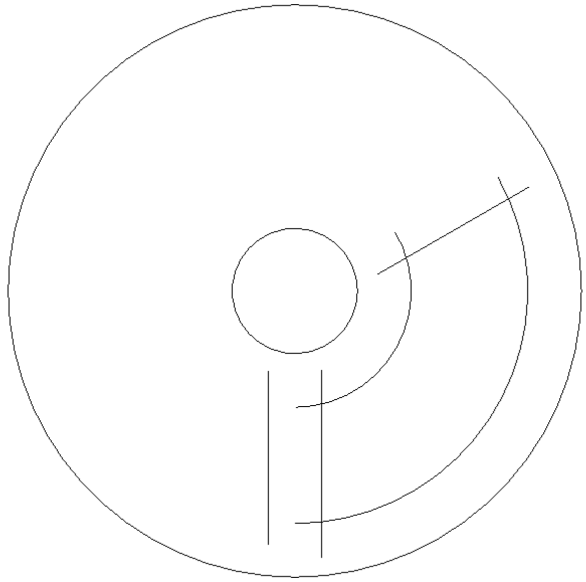
Create the inner and outer curves along with the fillet arcs for one of the slots.

1. Choose **Main Menu, Create, Arc, Polar, Sketch**.
2. Press [O] to select the origin as the center point.
3. Type in the radius of the outer arc: **80**
4. Click near positions 1 and 2 in the following picture to locate the approximate starting and ending angles of the outer arc.

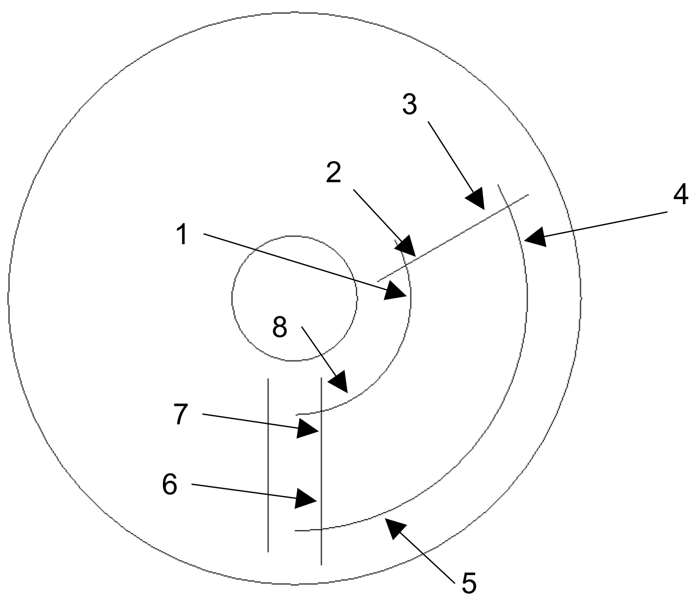


Note: Mastercam measures all arcs in a counterclockwise direction. The 3 o'clock position = 0º.

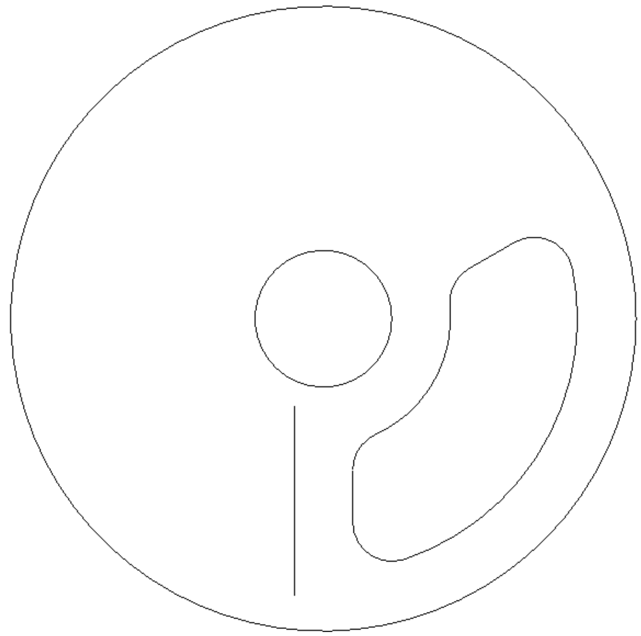
5. To create the inner arc, type [O] again to locate its center point at the origin.
6. Enter its radius: **40**
7. Click near positions 3 and 4 in the previous picture to locate the arc's endpoints. The part should look like the following picture.



8. Now create 12 mm fillets at the four corners of the slot. Choose **Main Menu, Create, Fillet, Radius**.
9. Enter **12** for the radius.
10. Click on the positions shown in the following picture in order.



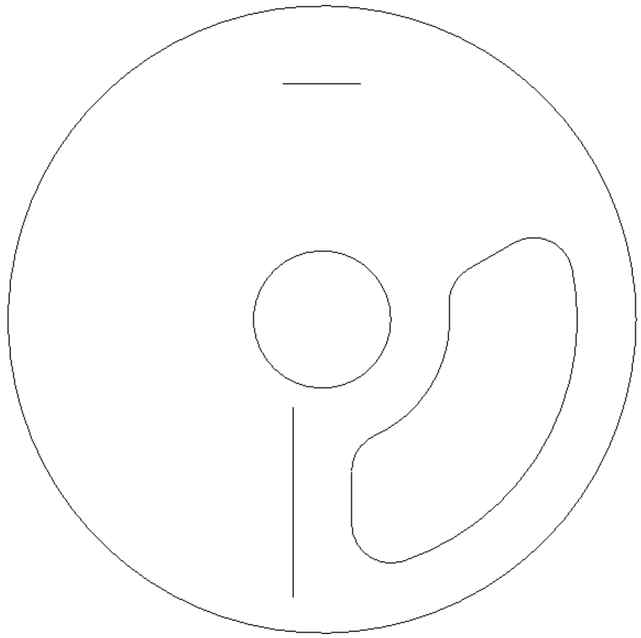
The part with the fillets should look like the following picture.



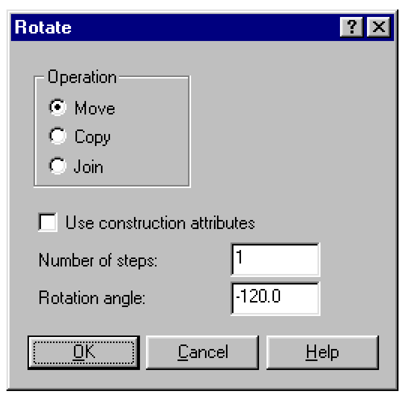
▶ *Completing the first slot*

To complete the first slot, you need to create the flat edge on the outside radius. First you will create the horizontal line. Then you will rotate it into position and create the arcs that connect it to the slot.

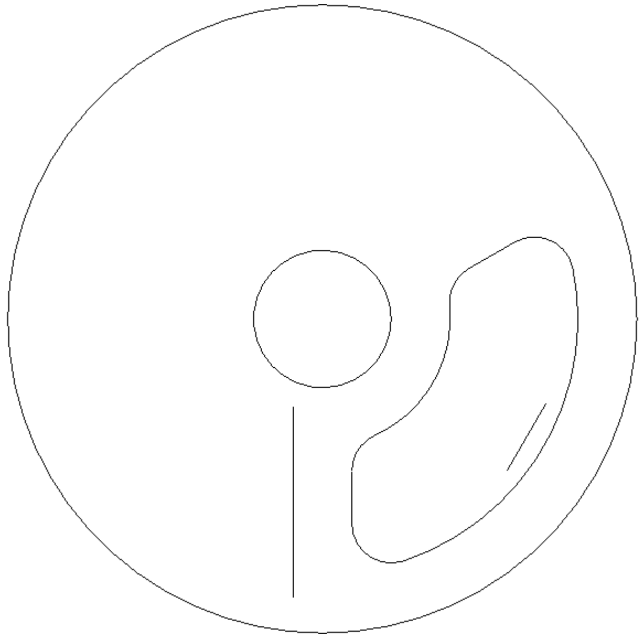
1. Choose **Main Menu, Create, Line, Endpoints**.
2. Enter the coordinates for the first endpoint: **–12.5, 75**
3. Enter the coordinates for the second endpoint: **12.5, 75**. The line should look like the following picture.



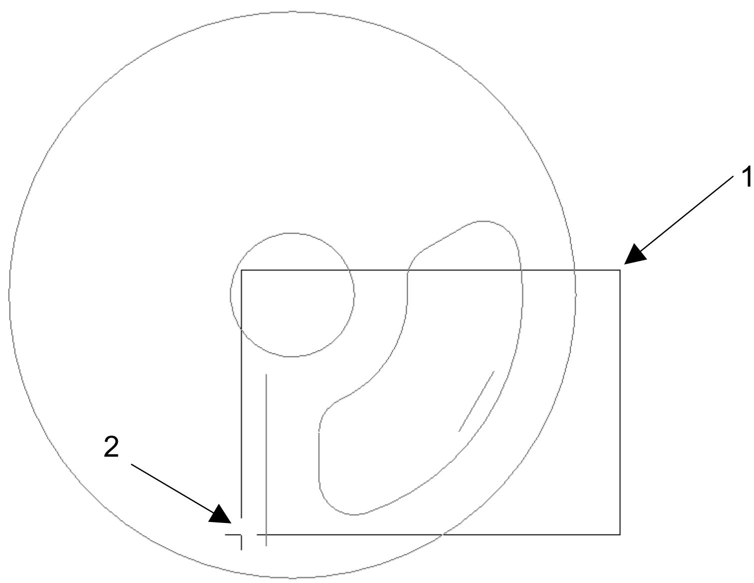
4. Choose **Main Menu, Xform, Rotate**.
5. Select the horizontal line you just created.
6. Choose **Done, Origin**.
7. Choose **Operation–Move**.
8. Enter a **Rotation angle** of **–120**. Your values should match the following picture.



9. Choose **OK**. The line should rotate into position as shown in the following picture.

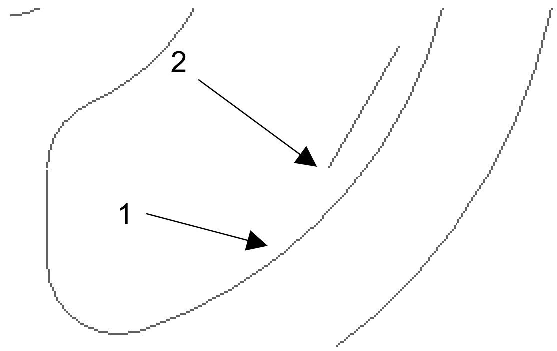


10. Zoom in on the new line. Right-click anywhere in the graphics window and choose **Zoom window**. Click once near position 1 and then near position 2.

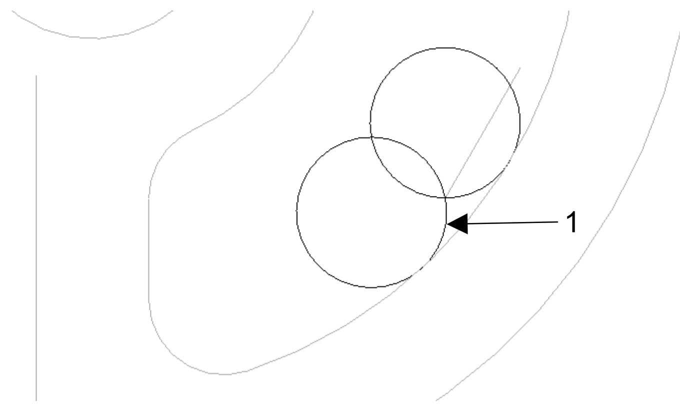


11. Choose **Main Menu, Create, Arc, Tangent, Point**.

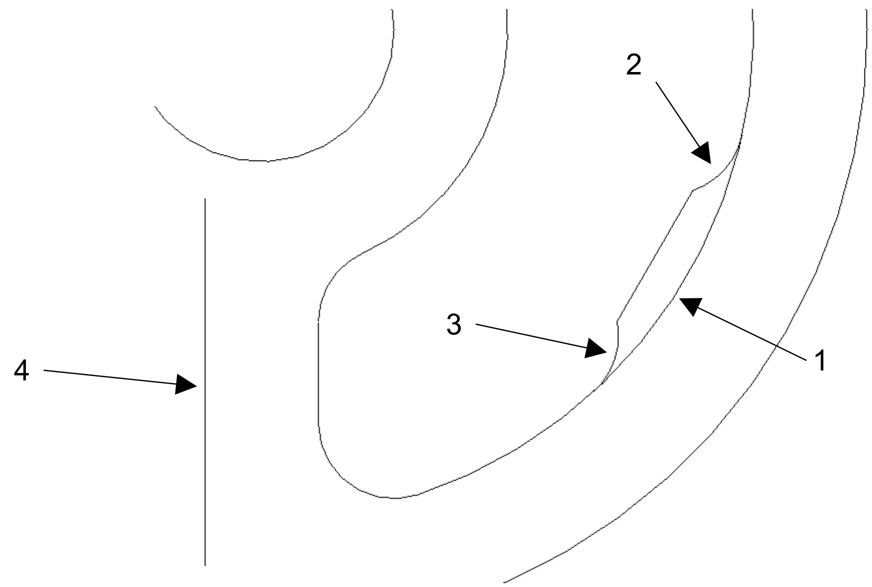
12. Select the arc at position 1.
13. Select the endpoint of the line at position 2.



14. Enter the radius of the arc: **12**
15. Since there are several possible arcs through the endpoint of the line and tangent to the arc, Mastercam asks you to select the one you want to keep. Select the arc at position 1.



16. Repeat steps 12–15 to create the arc at the other endpoint of the line.
17. Use Mastercam's Trim function to delete the segment of the outer radius between the two arcs. Choose **Main Menu, Modify, Trim, Divide**.
18. Select the arcs at positions 1, 2, and 3 as shown in the following picture.

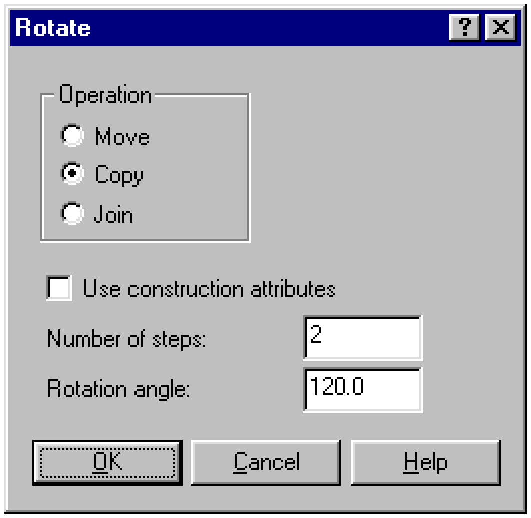


19. Delete the construction line shown at position 4. Press [Page Down] to unzoom, if necessary.

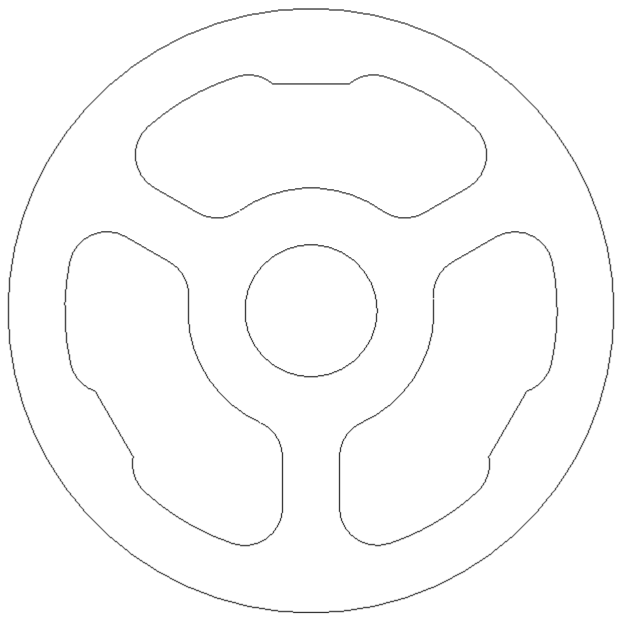
▶ *Creating the other slots by rotating the first slot*

Now that you've created the first slot, copy and rotate it about the center point to create the other two slots.

1. Fit the part to the screen.
2. Choose **Main Menu, Xform, Rotate, Chain**.
3. Click anywhere on the slot.
4. Choose **Done, Done, Origin**.
5. In the Rotate dialog box, choose **Operation–Copy**.
6. Enter **2** for **Number of steps**.
7. Enter a **Rotation angle** of **120**. Your values should match the following picture.



8. Choose **OK**. The part should look like the following picture.



9. Save the file in your working folder as **slotted wheel.mc9**.

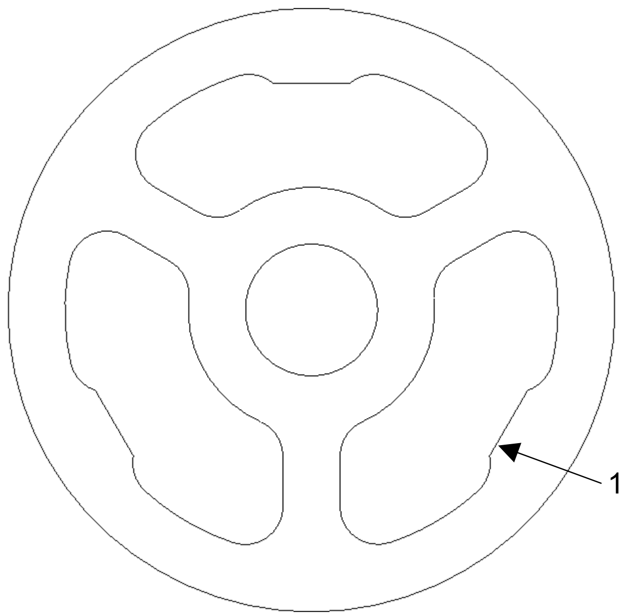
Exercise 2 – Cutting the slots

In this exercise, you create a toolpath to cut the slots. You will cut around the inside contour to cut the slot out completely, instead of cutting it as a pocket. You will create a toolpath for the first slot, and then add the other slots to it. You will learn the following skills:

- **Cutting an inside contour**
- **Adding more geometry to an existing toolpath**

▶ *Creating the toolpath*

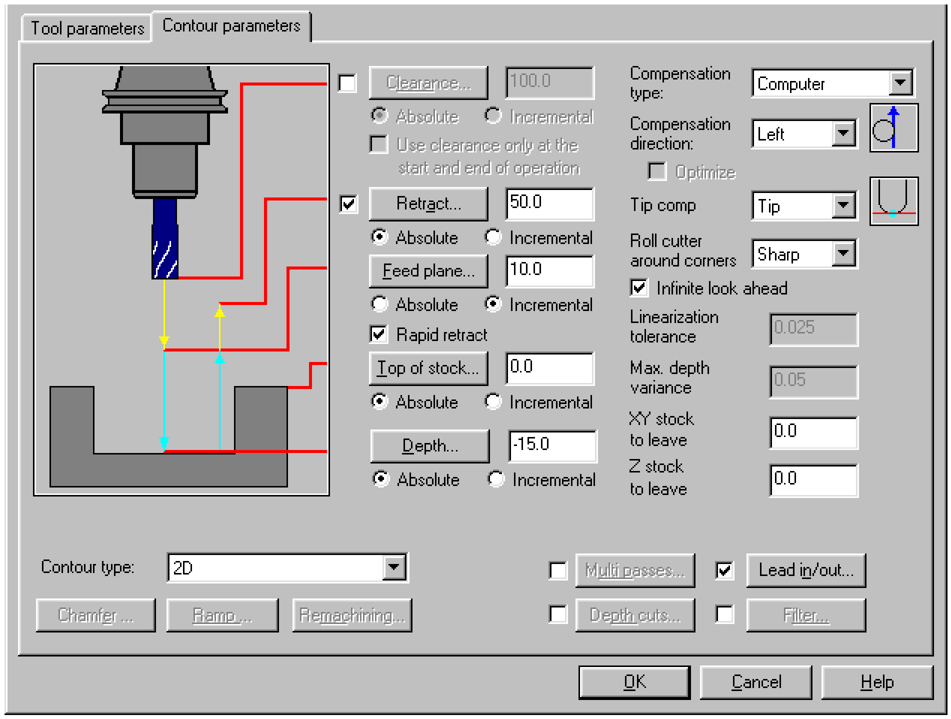
1. Choose **Main Menu, Toolpaths, Contour**.
2. Select the start point for the chain at position 1.



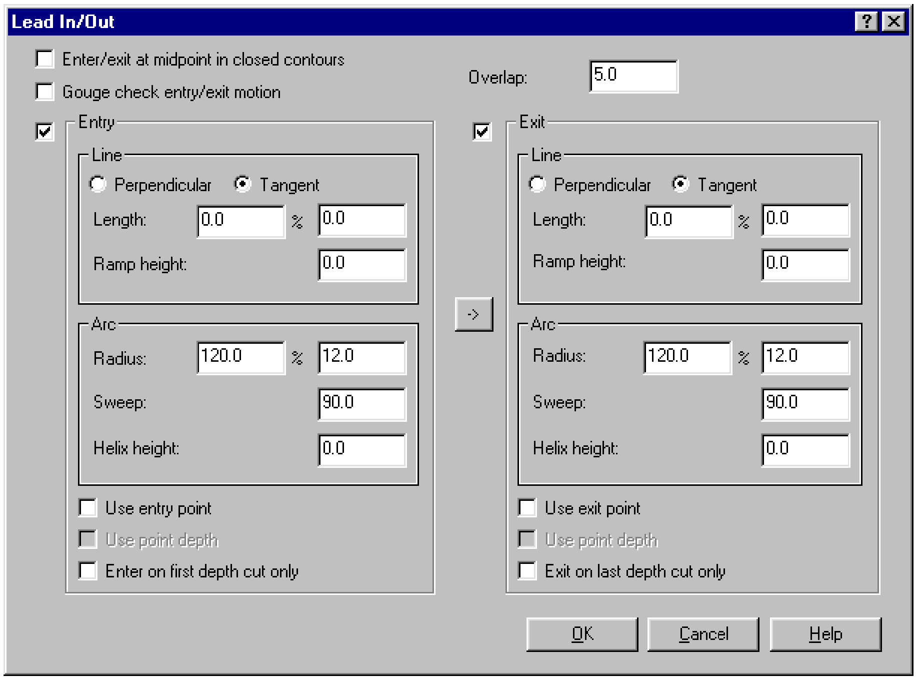
   The chaining direction arrow should point counterclockwise. If it doesn't, choose **Reverse** from the menu.

3. Choose **Done**.
4. Right-click in the tool display area and choose **Get tool from library**.
5. Select the 10 mm HSS flat endmill from the tool library.

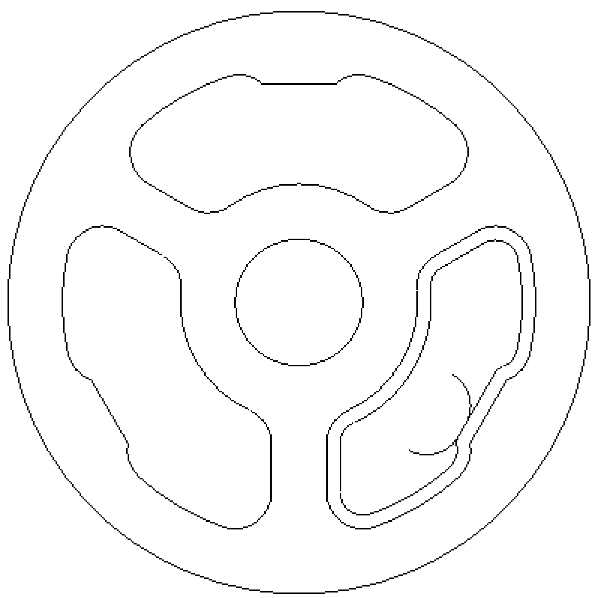
6. Select the **Contour parameters** tab.
7. Enter a **Depth** of **–15**. (The part is 12 mm thick, and you will cut through an additional 3 mm.) Your other parameters should match the following picture.



8. Select the **Lead in/out** check box and button.
9. Enter **0** for **Entry Line–Length**.
10. Enter **12** for **Entry Arc–Radius**.
11. Choose the -> button to copy the entry arc dimensions to the Exit section.
12. Enter an **Overlap** of **5**. Your values should match the following picture.

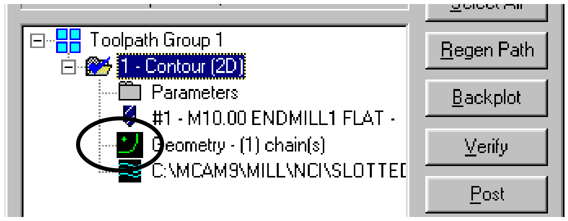


13. Choose **OK** twice. The toolpath should look like the following picture.

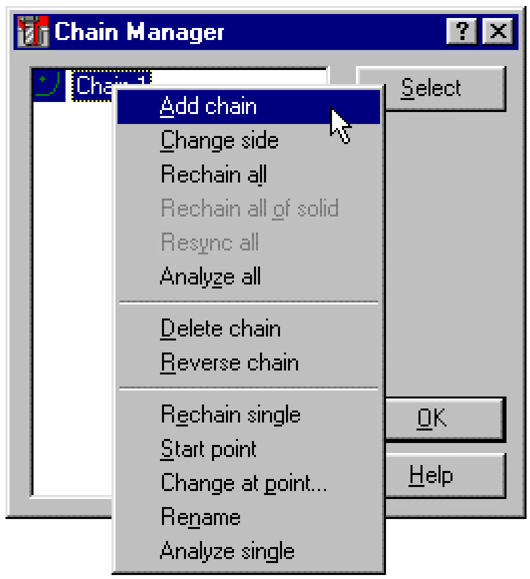


▶ *Adding the other slots to the toolpath*

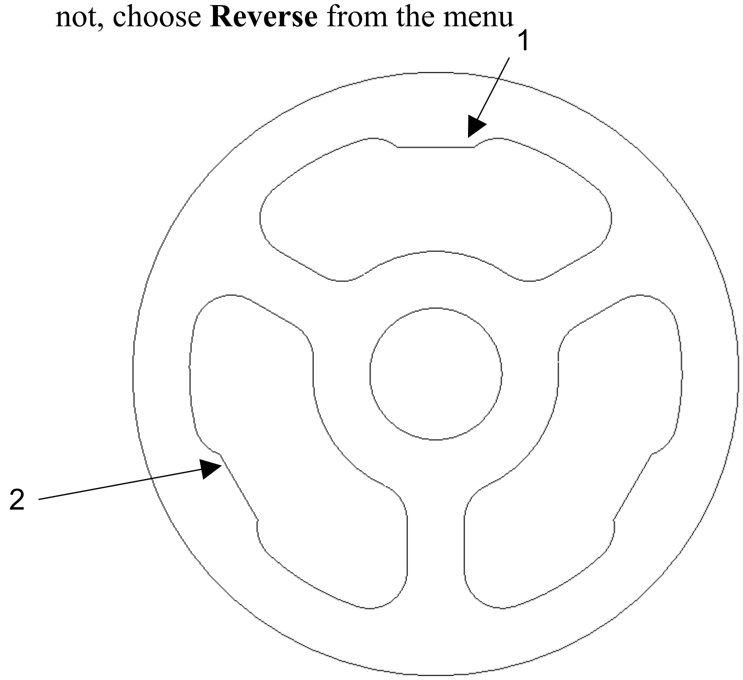
1. Choose **Operations** to open the Operations Manager.
2. Choose the **Geometry** icon for the toolpath.



3. Right-click on **Chain 1** and choose **Add chain** from the menu.



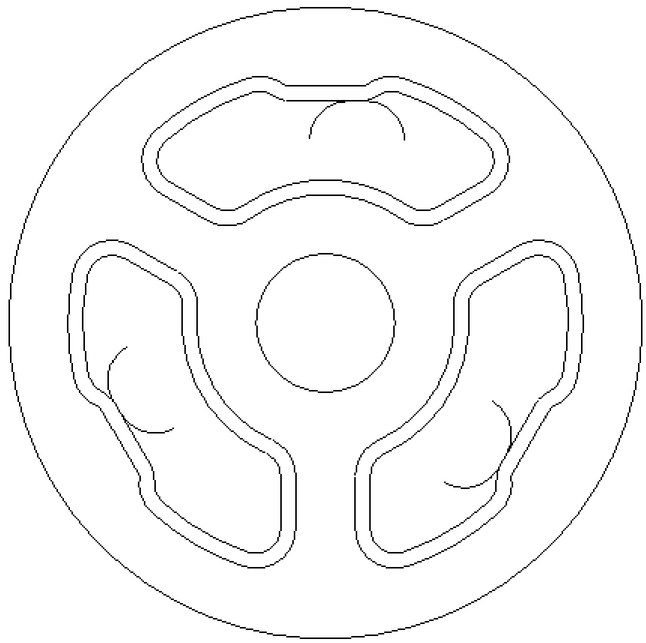
4. Click on locations 1 and 2 in the following picture to add the other two slots to the toolpath. After selecting each slot, the chaining arrow should be pointing counterclockwise. If it does not, choose **Reverse** from the menu



5. Choose **Done**. The Chain Manager displays the chains for all three slots.

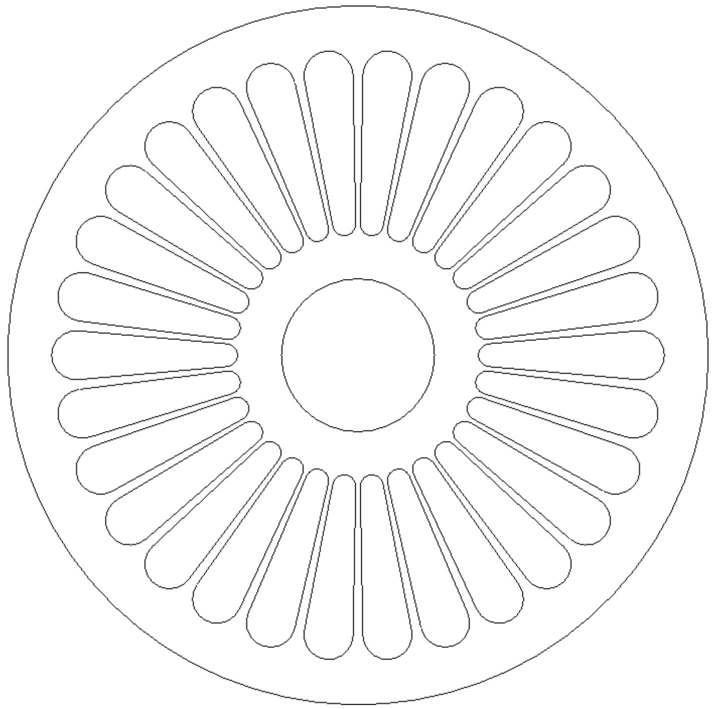
6. Choose **OK**.
7. Choose **Regen Path**.

8. Choose **OK** after the toolpath has been regenerated. It should look like the following picture. You can see how Mastercam has automatically replicated the lead in/out moves in each of the other slots.



Exercise 3 – Rotating a toolpath

The part shown in the following picture has 30 identical slots. In this exercise, you will create a toolpath for a single slot and, instead of rotating and copying the slot, you will rotate the toolpath. Using this approach to machine the other slots means that you don't have to create geometry for them.



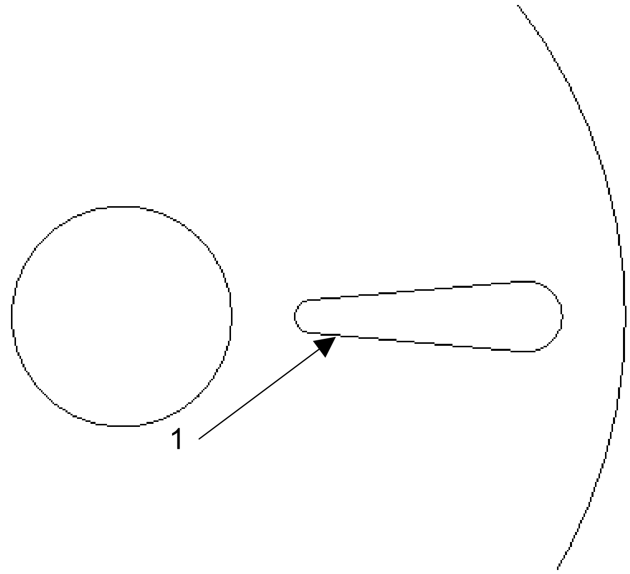
You will learn the following skill:

- **Transforming and rotating toolpaths**

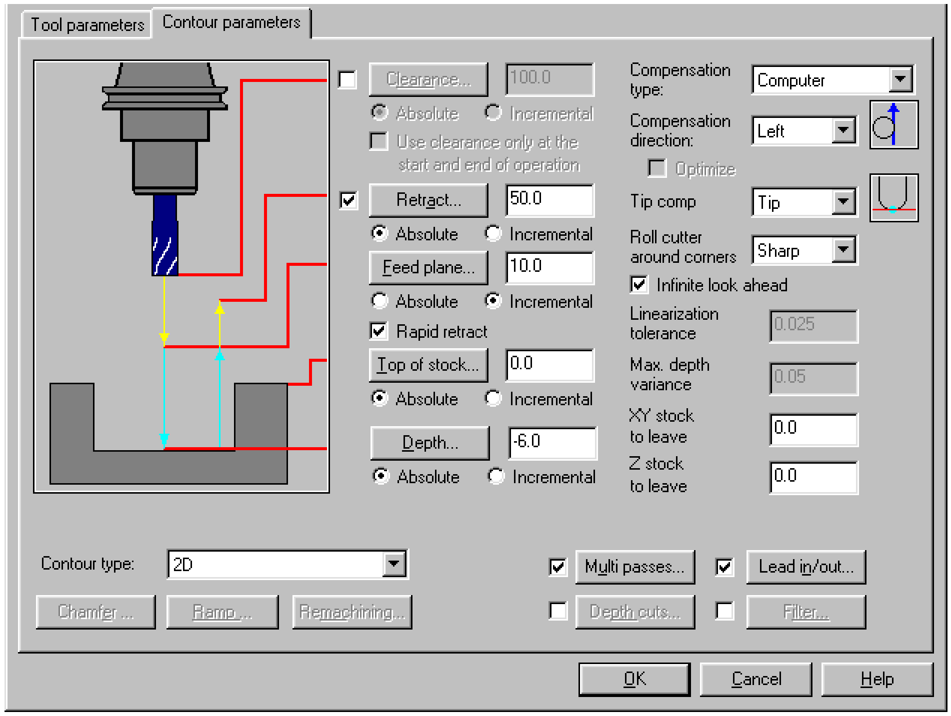
▶ *Creating the toolpath*

1. Open the file **rotation-mm.mc9**.
2. Choose **Main Menu, Toolpaths, Contour**.

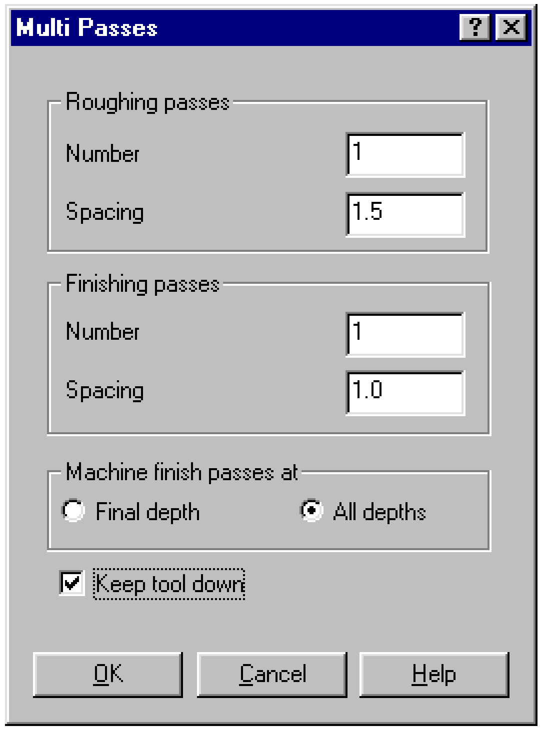
3. Select the start point of the chain at position 1. The chain direction should be counterclockwise.



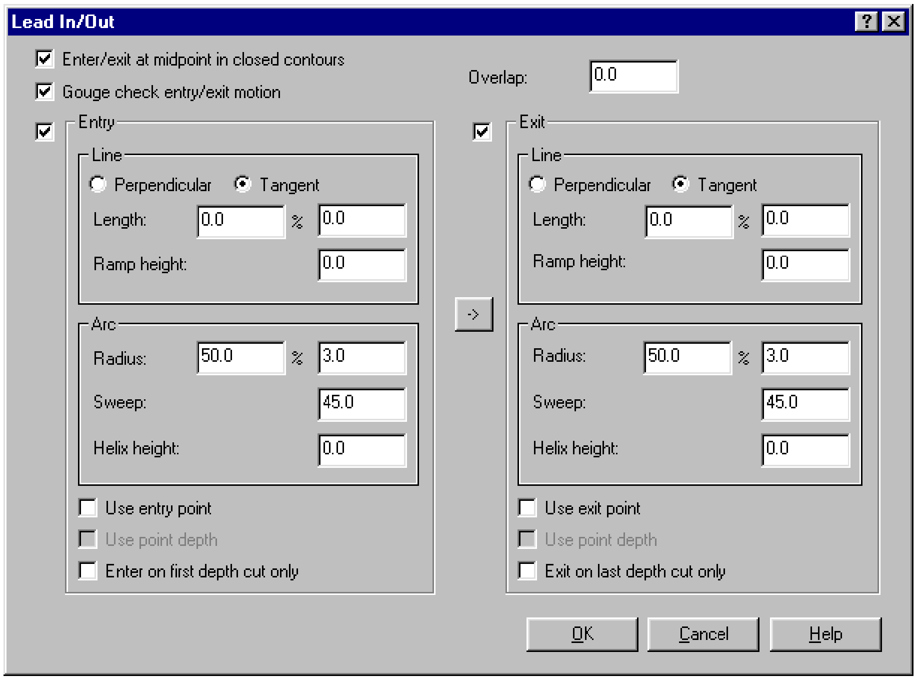
4. Choose **Done**.
5. Select a 6 mm HSS flat endmill from the tool library.
6. Choose the **Contour parameters** tab.
7. Enter a **Depth** of **–6**. The rest of the parameters should match the following picture.



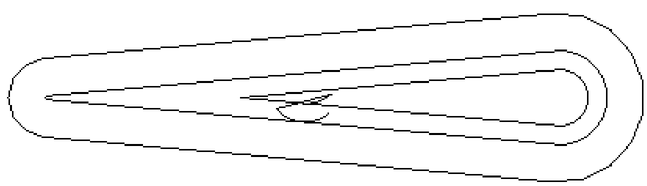
8. Choose the **Multi passes** check box and button.
9. You will create a single roughing pass and a single finishing pass. Enter **1** for the **Roughing passes–Number** and **1.5** for **Spacing**.
10. Enter **1** for the **Finishing passes–Number** and **1** for **Spacing**.
11. Select **Keep tool down**. Your values should match the following picture.



12. Choose **OK**.
13. Choose the **Lead in/out** check box and button.
14. Select the **Enter/exit at midpoint in closed contours** check box. This ensures that the entry and exit move will take place in the middle of the slot, instead of at the narrow end where there isn't enough room.
15. Enter **0** for the **Entry Line–Length**.
16. Enter **3** for the **Entry Arc–Radius**.
17. All you need for an entry arc is a partial arc, so enter **45** for the **Entry Arc–Sweep**.
18. Choose the  button to copy the entry arc dimensions to the Exit section. Your values should match the following picture.



19. Choose **OK** twice. Mastercam generates the toolpath shown in the following picture.

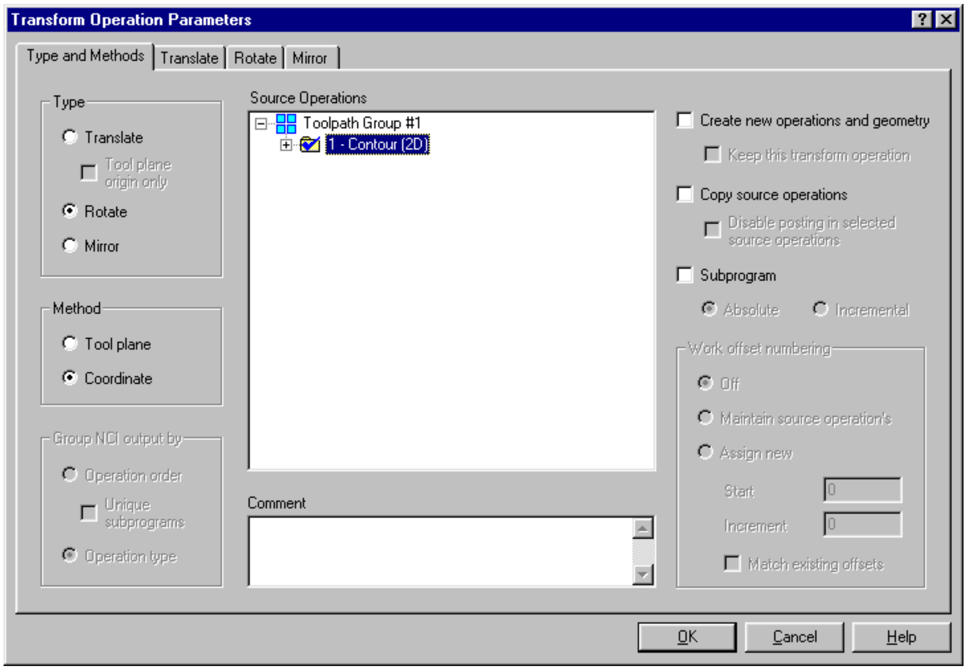


20. Choose **OK** to close the Operations Manager.

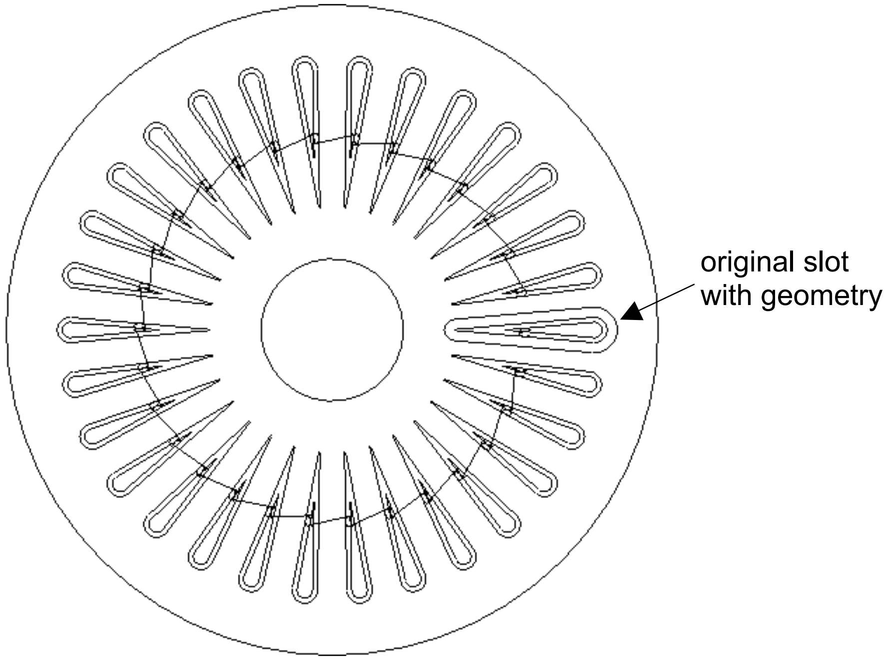
▶ *Rotating the toolpath*

1. Choose **Next menu, Transform**.
2. Choose **Type–Rotate**.
3. Choose **Method–Coordinate**. This means that each rotated toolpath will be generated by calculating the coordinates of each slot within the same plane, rather than by shifting the orientation of the tool and part for each successive slot.

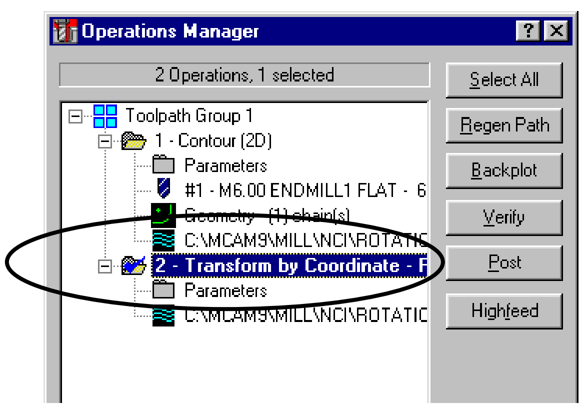
Your selections should match the following picture.



4. Choose the **Rotate** tab.
5. Choose **Origin** for the **Rotate point**.
6. Enter **29** for the **Number of steps**.
7. Enter a **Start angle** of **12** and a **Rotation angle** of **12** (because 360º ÷ 30 = 12).
8. Choose **OK**. The toolpath should look like the following picture.



You can see in the preceding picture that no new geometry has been created for the new slots; only the original slot has the geometry. All of the tool movements for the 29 other slots are contained in the single Transform operation as shown in the following picture.

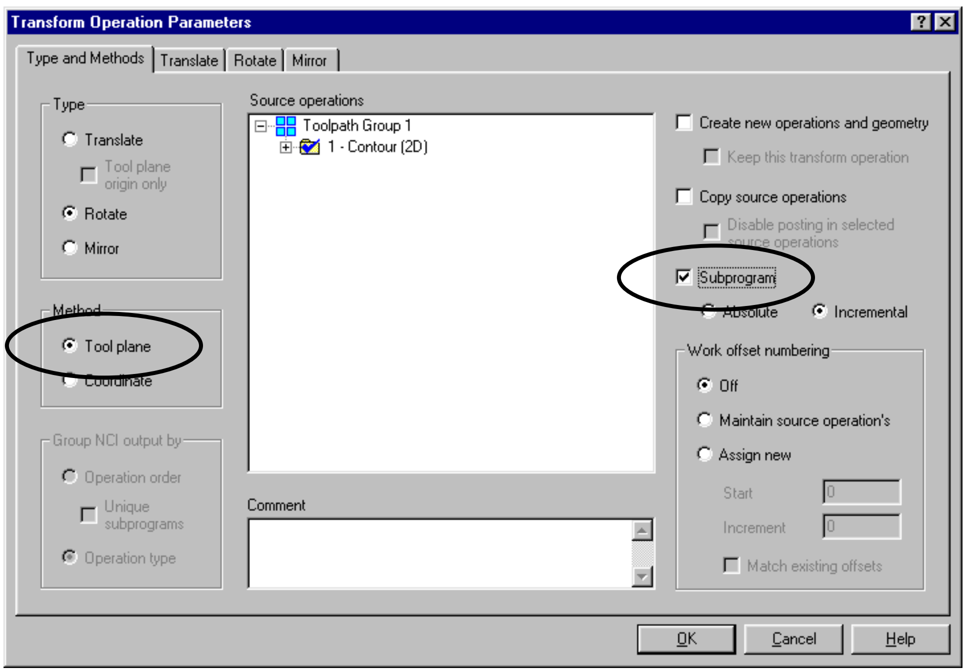


9. Save the file as **rotation1.mc9**.

▶ *Rotating the part with an indexer*

The previous procedure assumes that the part remains stationary on the table and the tool moves around it. This procedure shows you how to cut the multiple slots when you are using a rotary indexer to rotate the part.

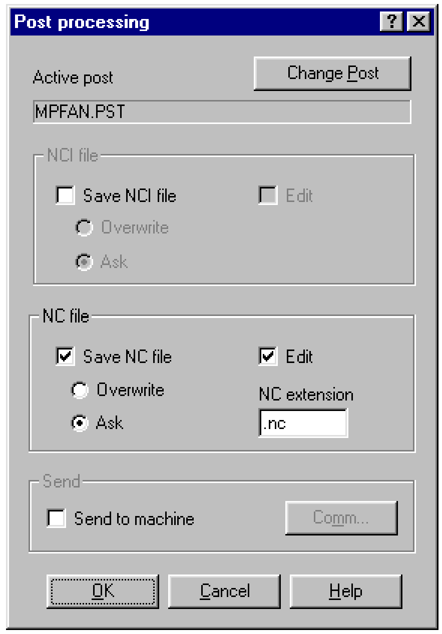
1. Press [Alt + O] to open the Operations Manager.
2. Choose the **Parameters** icon for the Transform operation.
3. Change the transform method to **Tool plane**.
4. Select the **Subprogram** option. Make sure that **Incremental** is selected. Your selections should match the following picture.



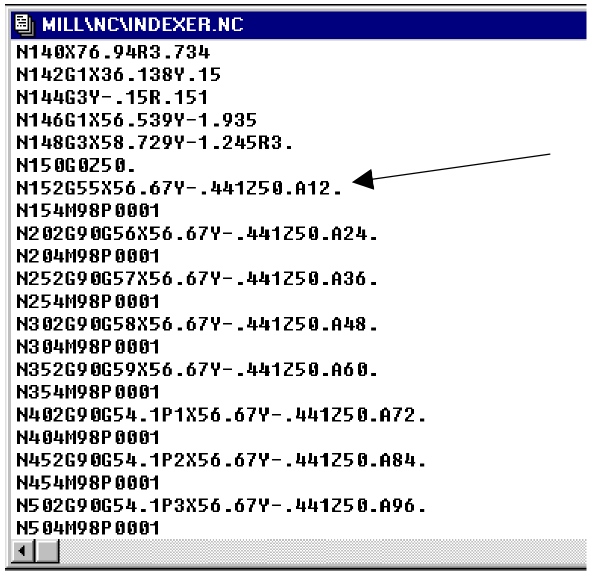
Note: Some posts may not support subprograms. If your post does not, leave the Subprogram option unchecked.

5. Choose **OK**.
6. Choose **Regen Path**.
7. Choose **Select All, Post**.

8. If necessary, choose **Change Post** and select the proper post for your machine.
9. Make sure your other values match the following dialog box and choose **OK**.



10. When prompted, save the NC file as **indexer.nc** in the default folder.
11. When the NC program appears in the editor window, scroll down until you see the lines shown in the following picture. You can see the A codes used to increment the indexer.



```
MILL\NC\INDEXER.NC
N140X76.94R3.734
N142G1X36.138Y.15
N144G3Y-.15R.151
N146G1X56.539Y-1.935
N148G3X58.729Y-1.245R3.
N150G0Z50.
N152G55X56.67Y-.441Z50.A12.
N154M98P0001
N202G90G56X56.67Y-.441Z50.A24.
N204M98P0001
N252G90G57X56.67Y-.441Z50.A36.
N254M98P0001
N302G90G58X56.67Y-.441Z50.A48.
N304M98P0001
N352G90G59X56.67Y-.441Z50.A60.
N354M98P0001
N402G90G54.1P1X56.67Y-.441Z50.A72.
N404M98P0001
N452G90G54.1P2X56.67Y-.441Z50.A84.
N454M98P0001
N502G90G54.1P3X56.67Y-.441Z50.A96.
N504M98P0001
```

12. Close the editor window.
13. Choose **OK** to close the Operations Manager.
14. Press [Alt + A] to save the file.

You've now seen a number of techniques for creating 2D geometry and toolpaths. The next chapter introduces you to creating 3D geometry and toolpaths.

6 Creating Drill Toolpaths

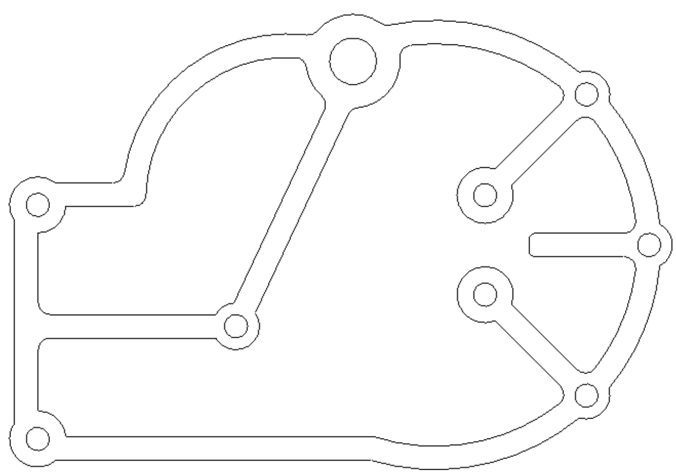
Mastercam includes many types of drill toolpaths and offers many different techniques to create them. This chapter introduces you to some basic techniques. The first exercise shows you how to construct a simple drill toolpath by selecting arcs. In the second exercise, you will change one of the arcs and regenerate the drilling operation. The third exercise shows you how to drill at different depths and combine multiple drilling operations on the same holes.

Exercise 1 – Creating a basic drill toolpath

This exercise introduces to some basic techniques for creating drill toolpaths. You will learn the following skills:

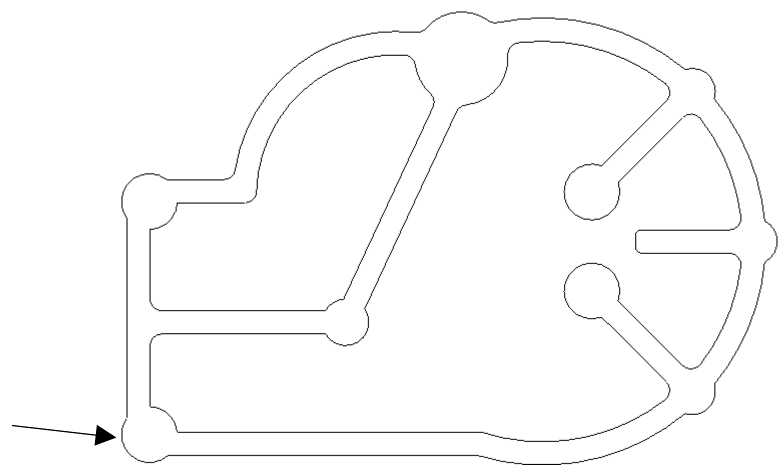
- **Using a mask to select arcs for the drill toolpath**
- **Sorting points to set the drilling order**
- **Filtering the tool library**
- **Creating the drill toolpath**

You will create the gasket shown in the following picture.



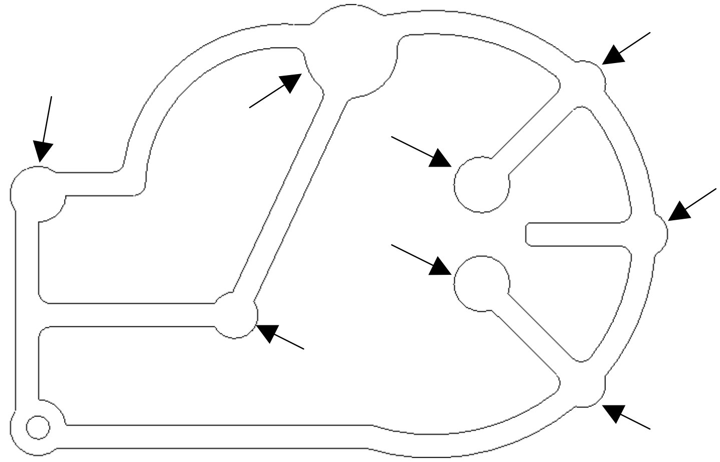
▶ *Creating the drill holes*

1. Choose **Main Menu, File, Get**.
2. Open the file **gasket-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
3. Choose **Main Menu, Create, Arc, Circ pt + dia**.
4. Enter a diameter of **6** in the prompt area.
5. Mastercam prompts you for the center point of the first 6 mm arc. Since you want to locate the new arcs at the center of the arcs that are already in the drawing, press [C], then select the arc as shown in the following picture. Mastercam automatically places the new arc at the center.

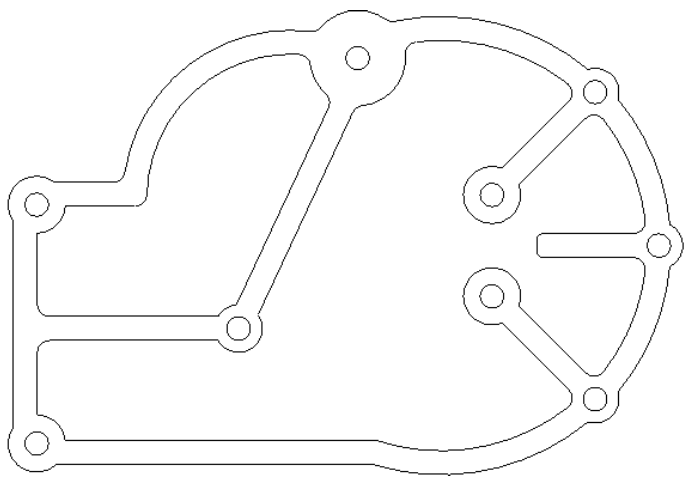


6. Repeat steps 4 and 5 for each of the remaining arcs shown in the following picture.

Tip: To avoid having to press [C] for every arc, right-click in the graphics window and turn off AutoCursor. Be sure to turn it on again when you are through.



Your part should now look like the following picture.



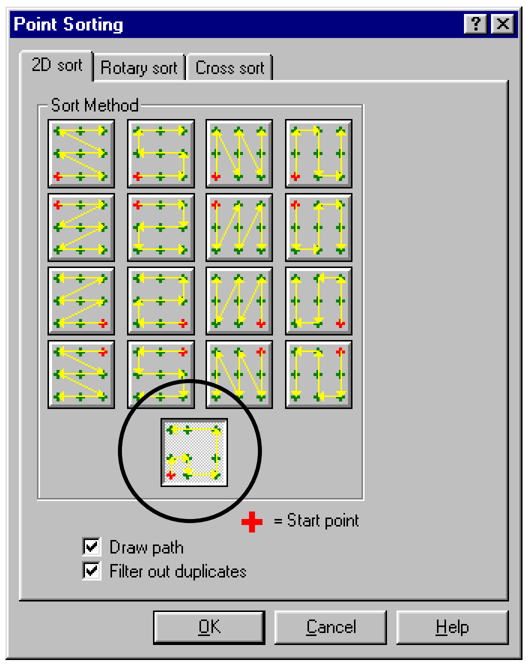
7. Press [Esc] to exit the Create Arc function.

▶ *Choosing holes for the drill toolpath*

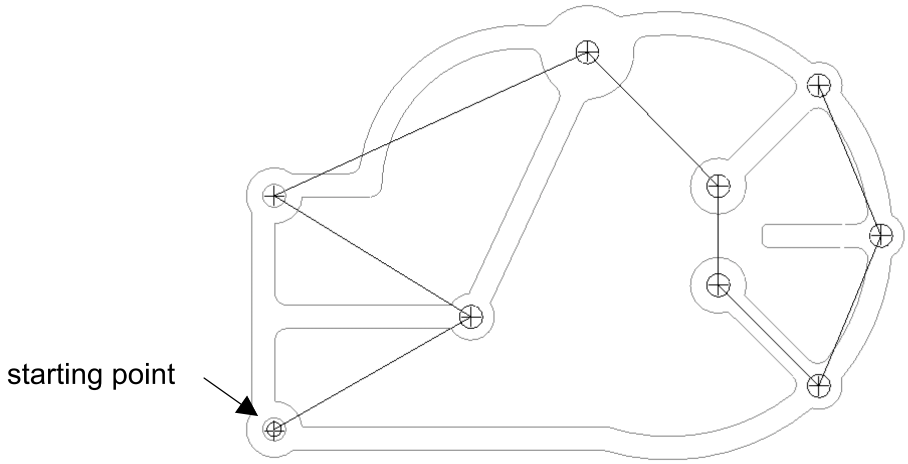
To specify which arcs will be the drill holes for the toolpath, you will use the **Mask on arc** feature. This lets you select an arc in your drawing and have Mastercam automatically choose all the arcs that match it.

1. Choose **Main Menu, Toolpaths, Drill**.

2. Choose **Mask on arc**.
3. Select any of the 6 mm holes.
4. Press [Enter] to accept the default tolerance value.
5. Choose **Window**.
6. Click above and to the left of the part, and drag a rectangle that encloses the whole part. Click at the lower-right corner.
7. Choose **Done**. Mastercam selects all the 6 mm holes.
8. The lines joining the holes show the order in which they will be drilled. To select a more efficient pattern, choose **Options**.
9. Choose the **Point to Point** sorting button as shown in the following picture.



10. Choose **OK**.
11. When Mastercam prompts you to select the starting point, select the arc in the lower-left corner. The holes should be sorted as shown in the following picture.

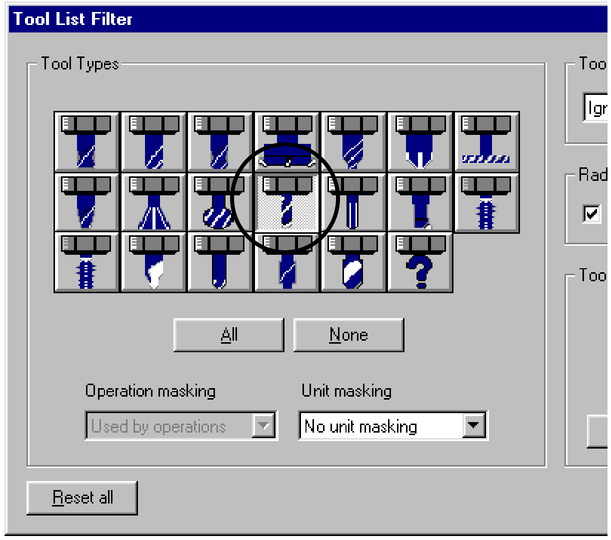


12. Choose **Done**.

▶ *Using the Tool Library filter to select a tool*

As soon as you choose **Done** in the previous procedure, Mastercam automatically displays the **Simple drill – no peck** dialog box where you can select a drill and set other drill parameters.

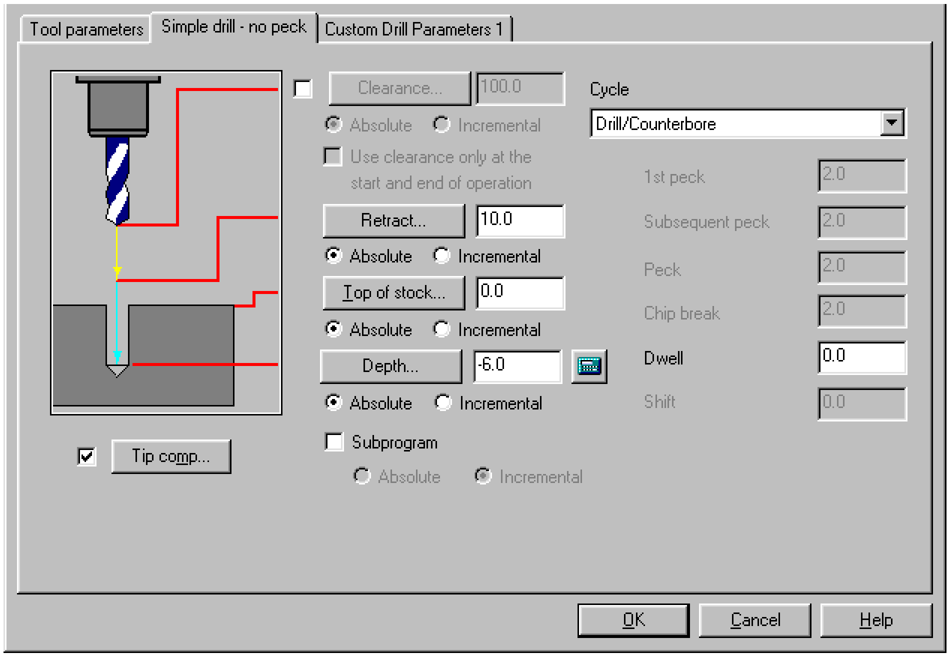
1. Right-click in the tool display area and choose **Get tool from library**.
2. Choose the **Filter** button.
3. Choose the **None** button to cancel the current filter setting.
4. Choose the **Drill** button as shown in the following picture. This means that when you return to the Tools Manager window, you will see only drills, making it easier to select the proper tool.



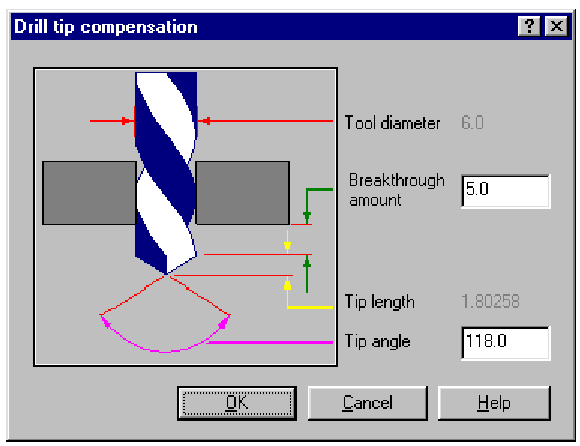
5. Choose **OK**.
6. Select the 6 mm HSS drill and choose **OK**.

▶ *Setting the drilling parameters*

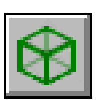
1. Choose the **Simple drill – no peck** tab.
2. Enter **–6** in the **Depth** field. Your other values should match the following picture.



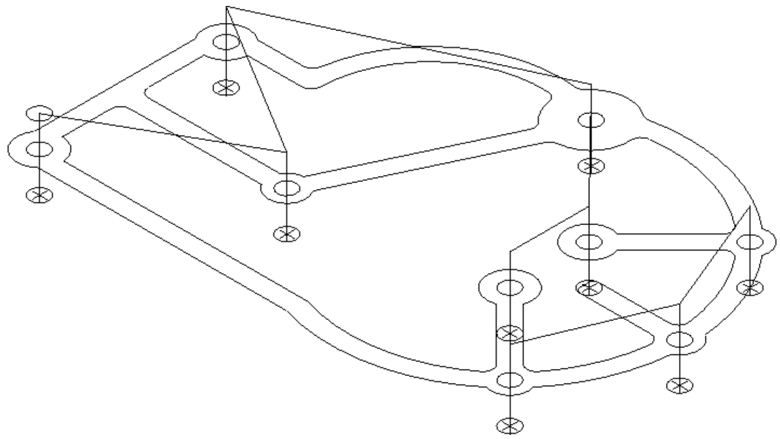
3. Choose the **Tip comp** check box and button.
4. Make sure your values match the following picture and choose **OK**.



5. Choose **OK** again. Mastercam generates the drill toolpath.



6. Choose the green **Gview–Isometric** button from the toolbar to see the toolpath more clearly. It should look like the following picture.



7. Choose **Main Menu, File, Save** and save the part in your working folder as **new gasket.mc9.**

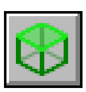
Exercise 2 – Changing the size of a drill hole

Your customer told you that one of the drill holes needs to have a 12 mm diameter instead of a 6 mm diameter. One way to do this is to delete the hole and create a new hole in its place. Then, create a new drill toolpath for the new hole with a larger drill. Because each operation in Mastercam can have only a single tool, the different drill size requires a new operation. In this exercise, you will learn the following skill:

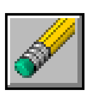
- **Creating a new drill toolpath for a different size hole**

▶ *Drawing the new hole*

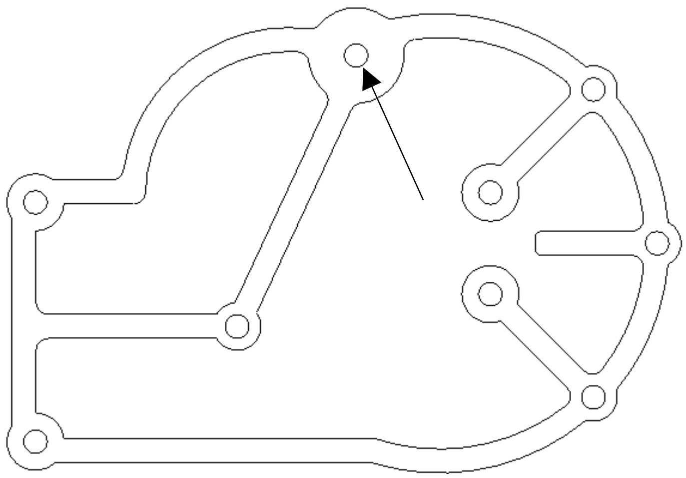
Delete the existing hole, then draw the new one in its place.



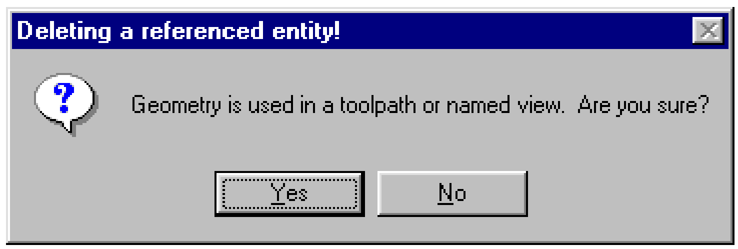
1. Choose the green **Gview–Top** button to switch out of isometric view.



2. Choose the **Delete** button from the toolbar.
3. Select the arc shown in the following picture.



4. Choose **Yes** when you see the following warning message.

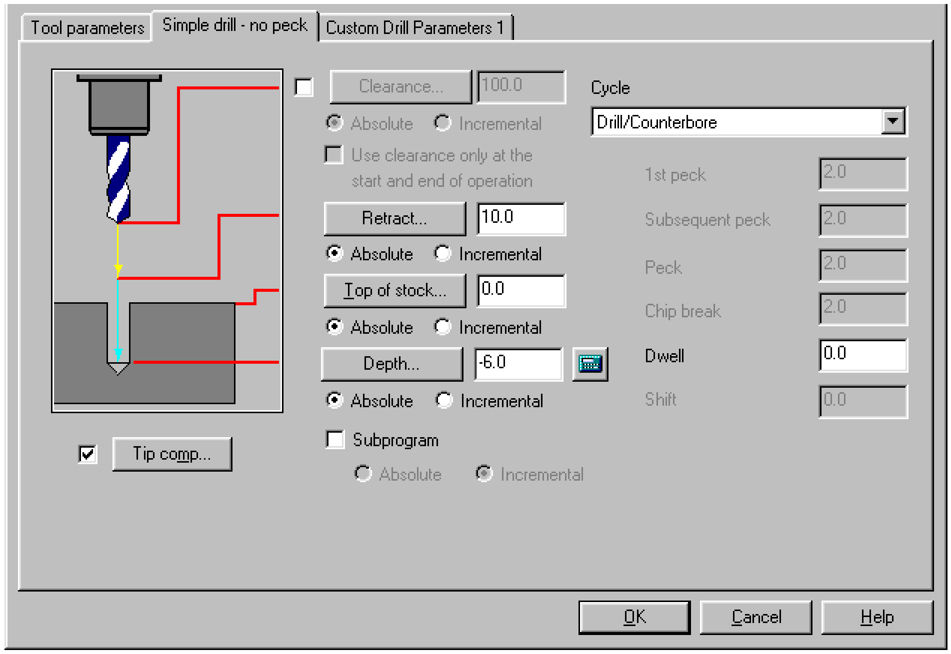


5. Choose **Main Menu, Create, Arc, Circ pt+dia**.
6. Enter a diameter of **12**.
7. Press [C] and select one of the arcs which surround the location of the deleted arc.

▶ *Creating the drill toolpath for the new hole*

1. Choose **Main Menu, Toolpaths, Drill, Manual**.
2. Press [C] and select the new 12 mm hole.
3. Press [Esc].
4. To select the start point, select the 12 mm hole again.
5. Choose **Done**.
6. Select the 12 mm HSS drill from the tool library.
7. Choose the **Simple drill – no peck** tab.

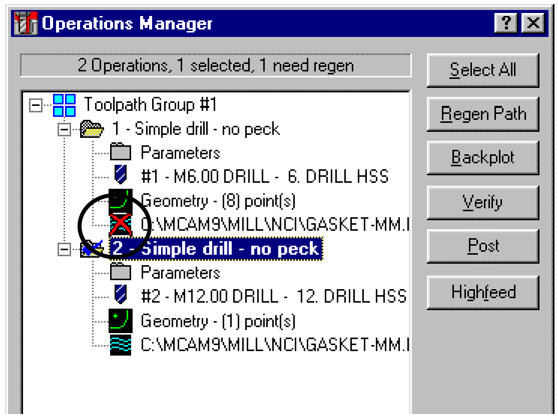
8. Make sure the drill parameters match the following picture. They should be the same values you entered for the first drilling operation.



Note: Mastercam automatically calculates a new tip comp for the new 12 mm drill, so you do not have to choose ***Tip comp*** *again.*

9. Choose **OK**. Mastercam calculates the drill toolpath for the new hole.

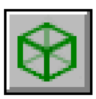
10. Choose **Operations**. You should see two toolpaths listed. The first toolpath is marked with a red X, as shown in the following picture. This means that it needs to be regenerated. Because you deleted a hole, the toolpath has to be updated.



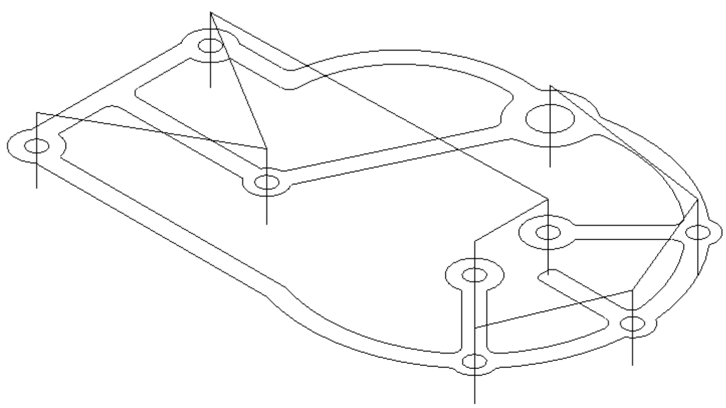
11. Click on the first operation to select it. The blue check mark on the folder tells you that it is selected.
12. Choose **Regen Path** to regenerate the toolpath.

▶ *Backplotting the toolpaths*

1. Choose **Select All**.
2. Choose **Backplot**.



3. Choose the green **Gview–Isometric** button from the toolbar.
4. Press [S] to step through the toolpaths. The completed toolpaths should look like the following picture.



5. Choose **OK** when the backplot is done.
6. Choose **Backup** to return to the Operations Manager.
7. Choose **OK** to close the Operations Manager.

8. Press [Alt + A] and choose **OK** to save the file.

Exercise 3 – Drilling at different Z depths

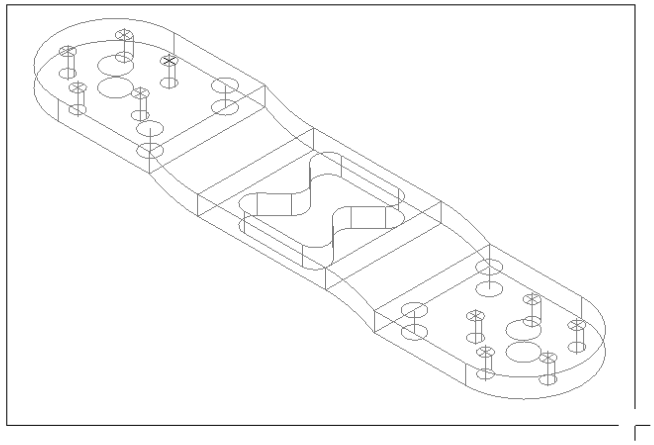
Some parts require you to drill holes starting at different Z depths. With Mastercam, you can select all the points at one time even if they lie at different Z depths, and you can include them in the same drilling operation. In this exercise, you will see how to use incremental values to set the drilling parameters so that you only need to create one set of parameters for all the holes.

Also, in this exercise, you will perform multiple drilling operations on each hole. First, you will predrill the holes with a center drill. Then, you will drill the holes to their proper dimension. You will learn the following skills:

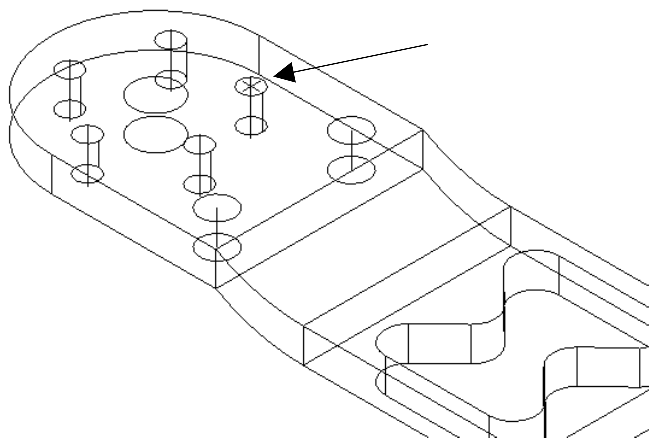
- **Creating a drill toolpath for multiple Z depths**
- **Using incremental values to set drill parameters**
- **Predrilling holes**
- **Using viewports to look at your part from several angles**

▶ *Predrilling with the centerdrill*

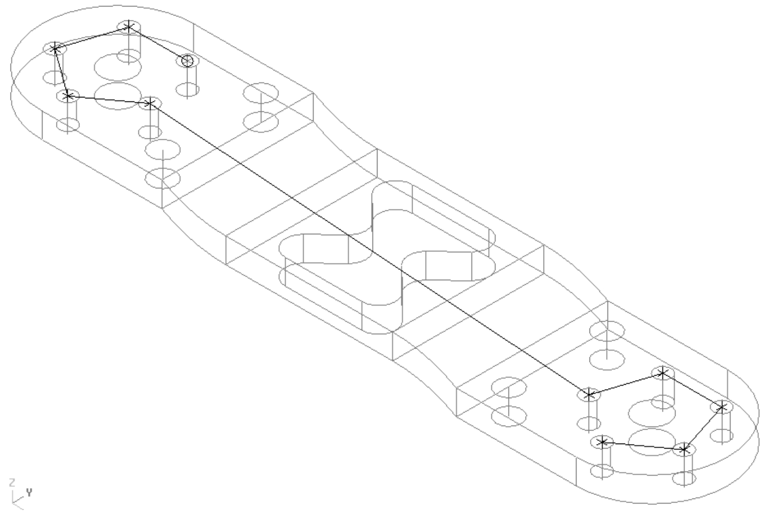
1. Open the file **tab-mm.mc9**.
2. Choose **Main Menu, Toolpaths, Drill, Window pts**.
3. Click above and to the left of the part and drag a window as shown in the following picture. Click the mouse when done.



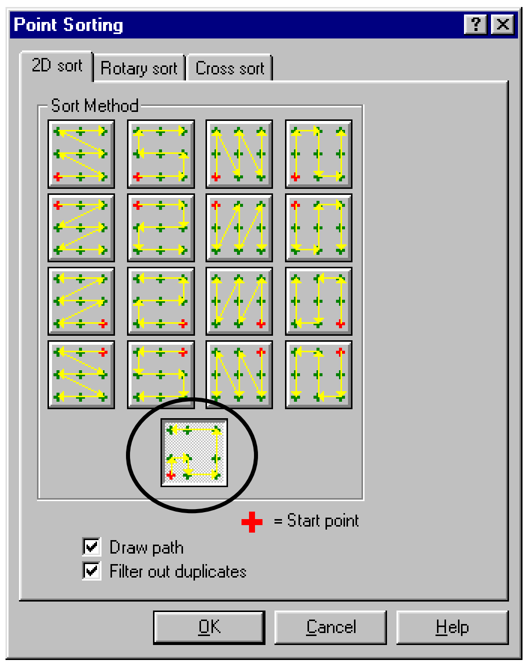
4. Mastercam prompts you to select a starting point. Select the point shown in the following picture.



The drill order should look like the following picture.



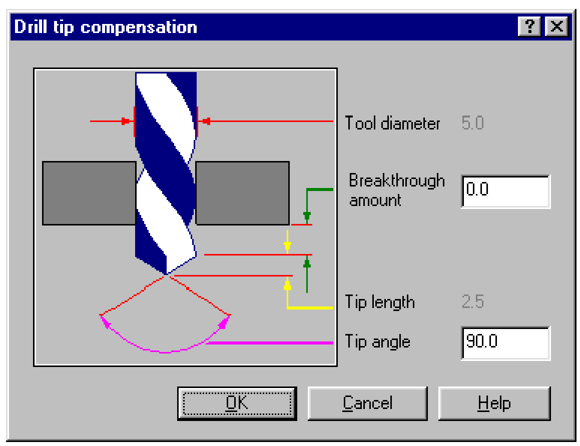
*Note: If your drill order doesn't match the preceding picture, choose **Options** from the menu, and choose the **Point-to-point** sorting method as shown in the following picture.*



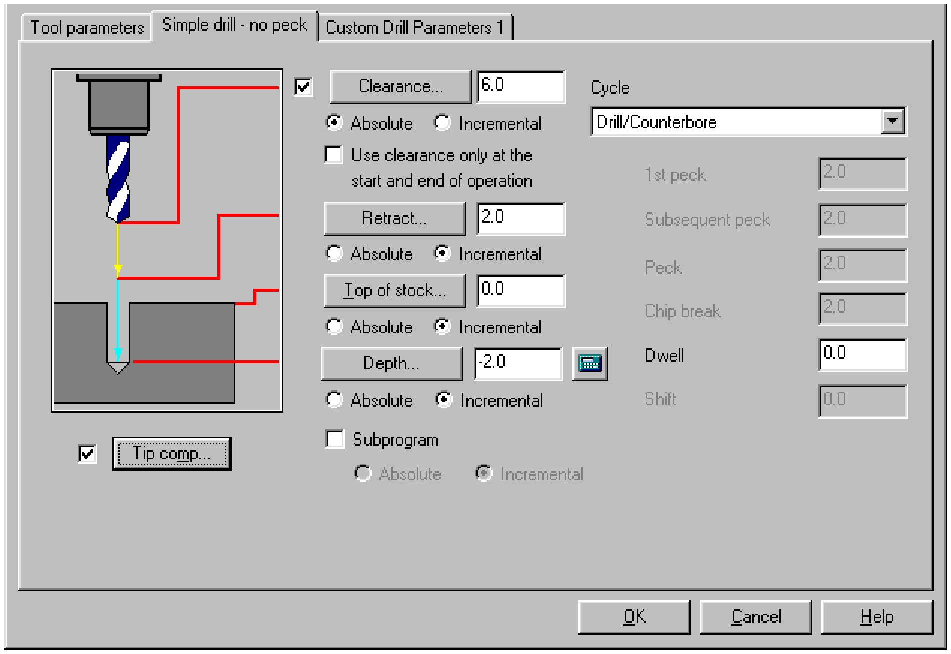
Then, select the starting point as shown above.

5. Choose **Done**.

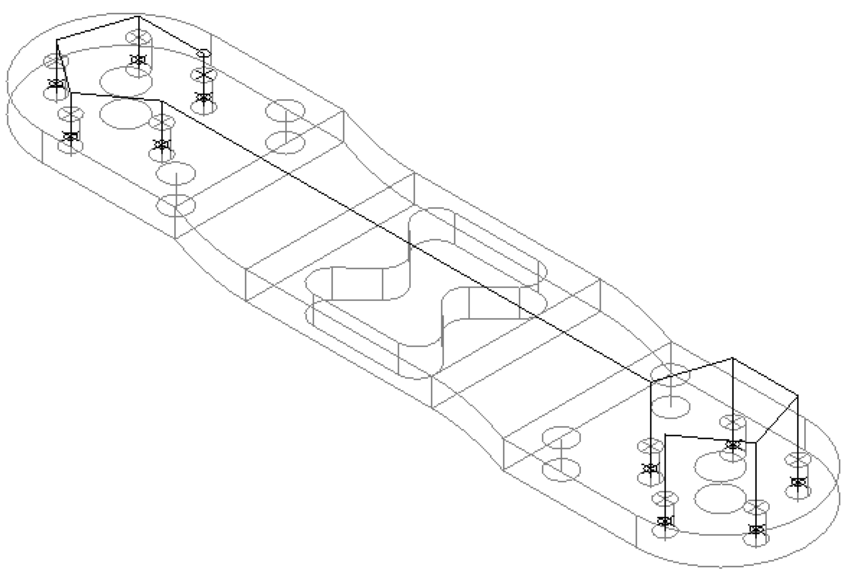
6. Choose the 5 mm HSS center drill from the tool library. If necessary, clear the **Filter active** check box to see the center drills.
7. Choose the **Simple drill – no peck** tab.
8. Select the **Clearance** check box and enter **6**. Make sure **Absolute** is selected. Because the holes are at different Z depths, it is important to have an absolute clearance plane distinct from the retract height to ensure that the drill will clear all areas of the part as it moves from hole to hole.
9. Change **Retract** to **Incremental** and enter a value of **2**.
10. Change **Top of stock** to **Incremental**. This means that the top of stock for each hole will change according to its Z depth. Since the depth and retract are also incremental, those values will be measured from the top of stock and will therefore also change with each hole's Z depth.
11. Enter a **Depth** of **–2** and choose **Incremental**.
12. Choose the **Tip comp** check box and button.
13. Enter a **Breakthrough amount** of **0**. Your values should match the following picture.



14. Choose **OK**.
15. Verify that your values match the following picture.



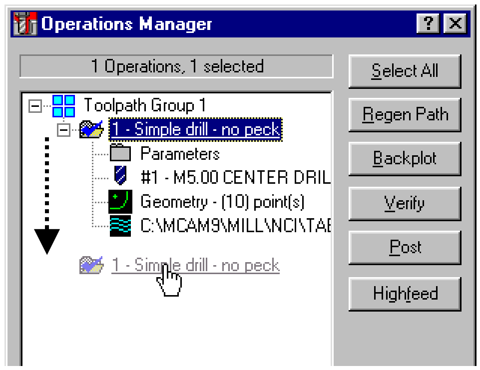
16. Choose **OK** to generate the drill toolpath. It should look like the following picture.



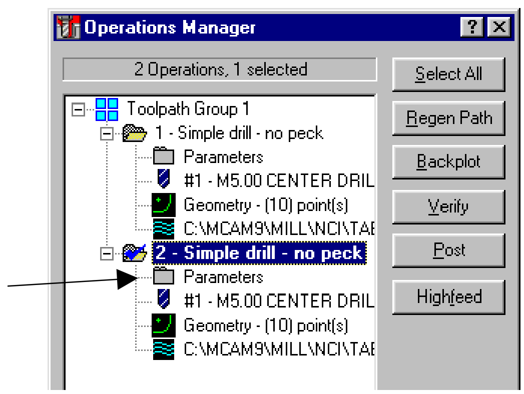
▶ *Drilling the holes*

To create the second drill toolpath, you will copy the first drilling operation and edit the parameters.

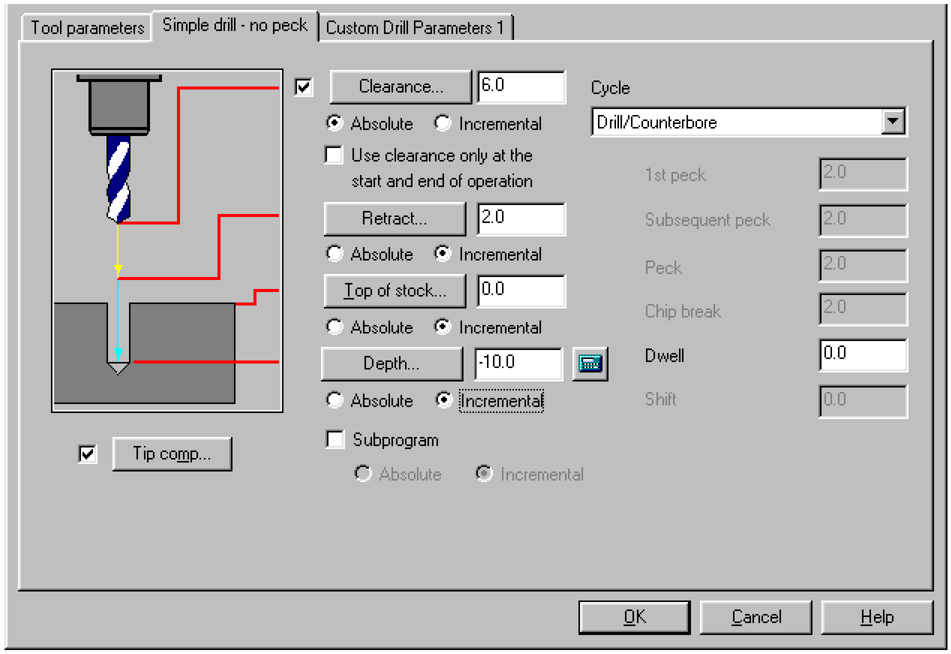
1. Choose **Operations**.
2. Right-click on the drilling operation, drag it down as shown in the following picture, and release the mouse.



3. Choose **Copy after** from the menu.
4. Choose the **Parameters** icon for the second toolpath.



5. Choose the **Tool parameters** tab.
6. Select the 6 mm HSS drill from the tool library.
7. Choose the **Simple drill – no peck** tab.
8. Change the **Depth** to **–10**. Your drill parameters should match the following picture.



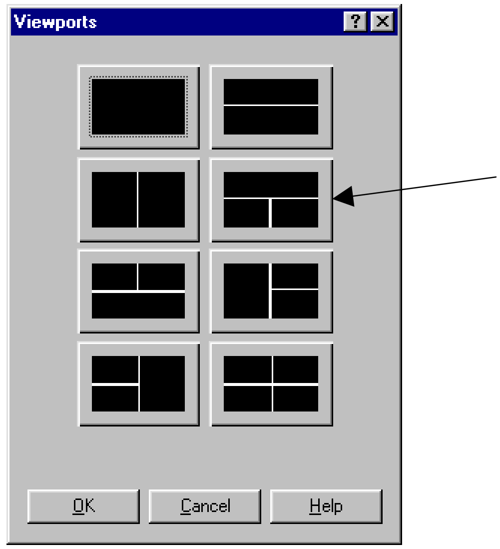
9. Choose **Tip comp**.
10. Enter a **Breakthrough amount** of **1**.
11. Choose **OK** twice.
12. Choose **Regen Path**. Mastercam regenerates the second toolpath.

▶ *Backplotting with viewports*

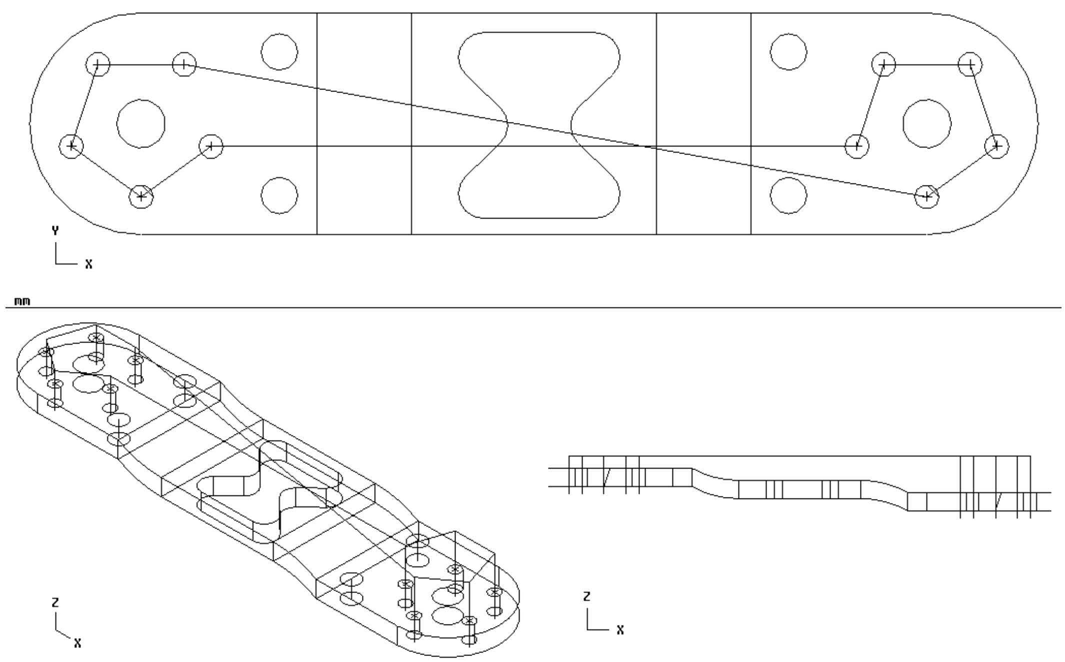
Now you can use the Viewports function to backplot the toolpaths. Viewports are display layouts that combine different part views on the same screen. The different views update simultaneously, making this feature ideal for backplotting toolpaths.

1. Choose **Select All**.
2. Choose **Backplot**.
3. Press [Alt + W].

4. Choose the viewport indicated by the following picture.



5. Press [S] to step through the toolpaths. Your screen should look like the following picture.

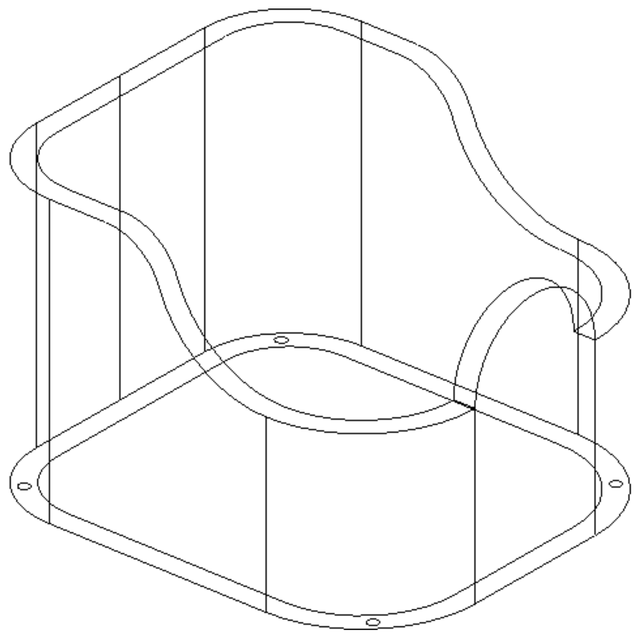


6. Press **OK** when Mastercam tells you the backplot is finished.
7. Choose **Backup** to return to the Operations Manager.
8. Choose **OK** to close the Operations Manager.
9. Save the file in your working folder as **drilled tab.mc9**.

You've now seen a number of techniques for creating 2D geometry and toolpaths. The next chapter introduces you to creating 3D geometry and toolpaths.

7 Working in 3D

This chapter guides you through the design of a 3D wireframe part. You will use Cplanes and the View Manager to work on the part in different 3D orientations. In the final exercise, you will create a drill toolpath to drill upwards into the part from the bottom. The part you'll be creating is the housing shown in the following picture. The top edge is defined by a 3D contour, and there are four small mounting holes on the bottom edge.

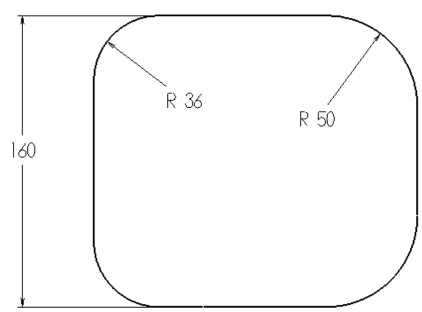


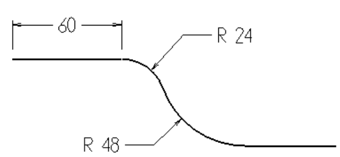
Exercise 1 – Creating 3D geometry

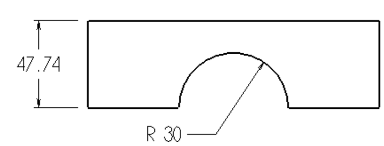
In this exercise, you will use many of the same drawing techniques you used in previous chapters and exercises, but this time in 3D. You will learn the following skills.

- **Orienting geometry by choosing Cplanes**
- **Orienting geometry by setting the Z depth**

The following pictures show the dimensions of the 3D contour. For clarity, only the outer boundary of the top edge of the part is shown here.







▶ *Orienting geometry in 3D*

Mastercam uses construction planes (Cplanes) to orient two-dimensional entities, such as lines and arcs, in 3D space. Cplanes 1–6 align with a face of a cube. All of your work so far has been done in the Top Cplane, as if you were looking down on the top of the part. Whenever you switch to a different Cplane, it has the effect of rotating the XY coordinate plane to align with the selected face. The Z axis always corresponds to moving towards or away from the part.

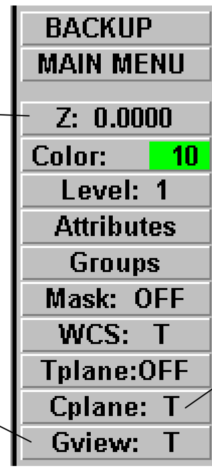
Once you select the Cplane, set the Z depth to tell Mastercam how deep to create the geometry. All of your work so far has been at a Z depth of 0. Choose a different Z depth to create new geometry above or below other geometry.

Complete the steps in this procedure to see the effects of selecting Cplanes and Z depth before drawing the actual part.

1. If necessary, choose **Main Menu, File, New** to create a blank Mastercam drawing.

   The Secondary Menu tells you how new geometry will be oriented:

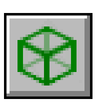
Z depth
This button tells you the current Z depth (0.0000). Choose the button to set a different value. Once you choose the button, you can either type in a new value, or select geometry on the screen to set the Z depth to match the selected entity.



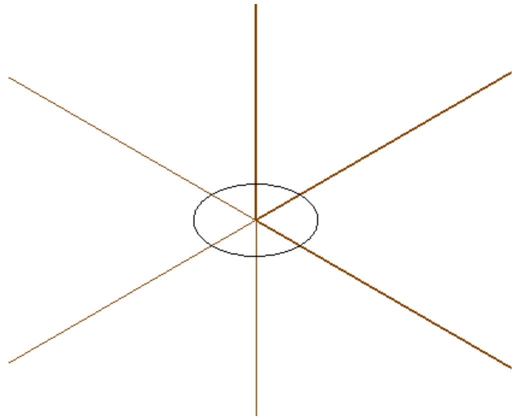
Cplane
This button tells you the current Cplane (T = Top). Choose the button to select a different plane from the menu, or use the toolbar buttons as shortcuts.

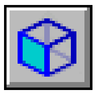
Gview
This button tells you the current Gview. Often while creating geometry, you want this to be the same as the Cplane, but you can change the Gview to examine the geometry from different perspectives.

2. Choose **Main Menu, Create, Arc, Circ pt + dia**.
3. Enter a diameter of **25**.
4. Press [O] to locate the circle's center point at the origin. Press the letter "O," not zero.
5. Press [F9] to display the coordinate axes.

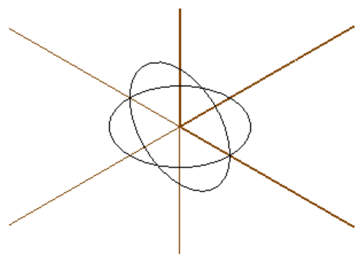


6. Choose the green **Gview–Isometric** button from the toolbar to display the circle in 3 dimensions. Your screen should look like the following picture.



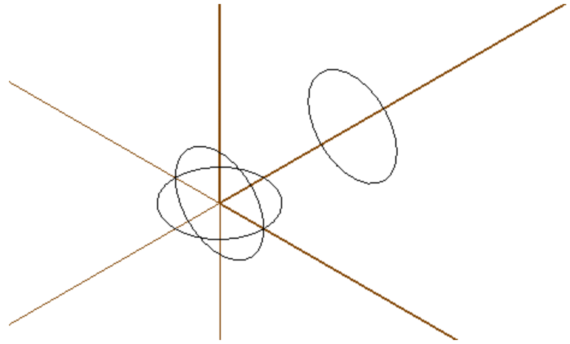


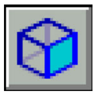
7. Choose the blue **Cplane–Front** button from the toolbar to change the Cplane.
8. Press [O] again to create a new circle with its center point at the origin. You can see that the center point is the same, but the second circle is now aligned with the front of the part.



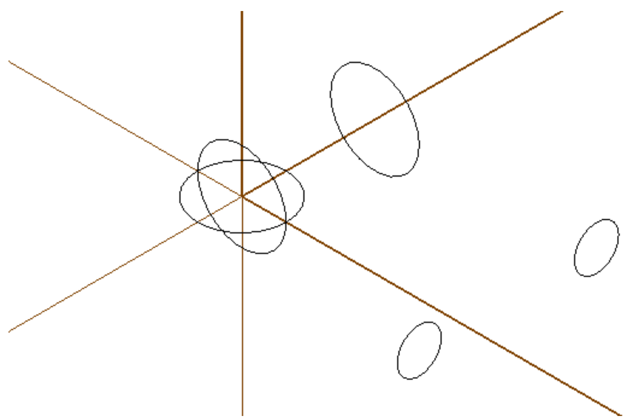
9. Choose the **Z** button from the Secondary Menu.
10. Type **–40** and press [Enter]. The new Z depth appears on the button.

11. Press [O] again. Another circle appears with its center point at the origin. It is aligned with the same plane as the previous circle, but shifted 40 mm into the part because of the new Z depth, as shown in the following picture.





12. Choose the blue **Cplane–Side** button from the toolbar to align new geometry with the left side of the part.
13. Choose **Z** and enter **75**.
14. Choose **Backup, Circ pt + dia**.
15. Press [Enter] to accept the diameter of **25**.
16. Enter the coordinates **X–25,Y10** to create the first circle.
17. Enter the coordinate **X25** to create a second circle. The new circles are aligned with the left face of the part, 75 mm in front of the system origin, as shown in the following picture.



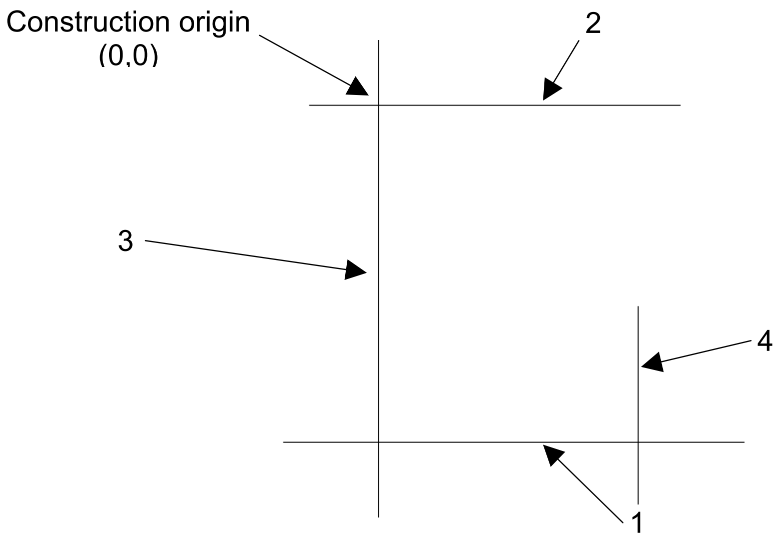
18. Choose **Main Menu, File, New** to clear the geometry from the screen. Choose **No** when prompted to save the file.

    *Note: After choosing **File, New**, the Cplane and Gview are reset to Top and the Z depth is reset to 0.*

Creating the first construction lines

Now that you've gotten a feel for creating geometry in 3D, you can begin drawing the part. You will begin by drawing the top-front corner.

1. Press [F9] to display the coordinate axes.
2. Choose **Main Menu, Create, Line, Horizontal**.
3. Draw the line at position 1.
4. Enter the Y coordinate: **–80**



5. Draw the line at position 2.
6. Enter the Y coordinate: **0**
7. Choose **Backup, Vertical**.
8. Draw the line at position 3.
9. Enter the X coordinate: **0**
10. Draw the line at position 4.

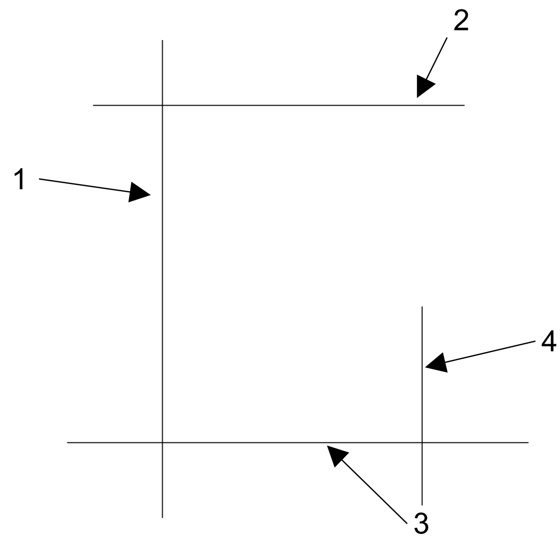
    Tip: If you need more space to finish drawing the line, press the [Down arrow] key a few times.

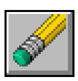
11. Enter the X coordinate: **60**

▶ *Trimming the lines*

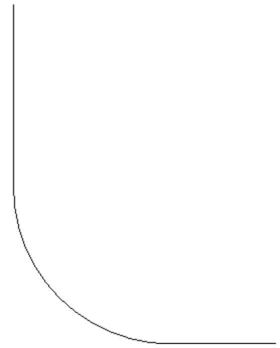
Complete the upper corner of the part by trimming the lines and adding the fillet.

1. Choose **Main Menu, Modify, Trim, 1 entity**.
2. Select the lines at positions 1, 2, 3, and 4 in that order.





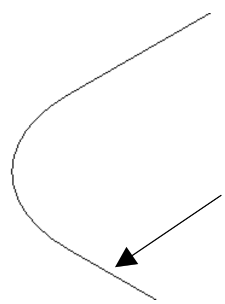
3. Choose the **Delete** button on the toolbar.
4. Click on lines 2 and 4 as shown in the preceding picture to delete them.
5. Choose **Main Menu, Create, Fillet, Radius**.
6. Enter the fillet radius: **36**
7. Click on each of the two remaining lines. When you are done, the part should look like the following picture.



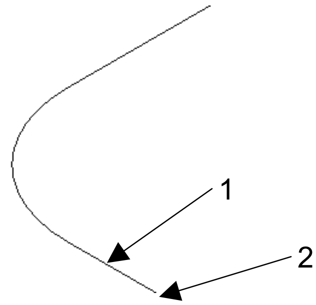
Drawing the front face

In this procedure, you will draw the curve on the front face of the part.

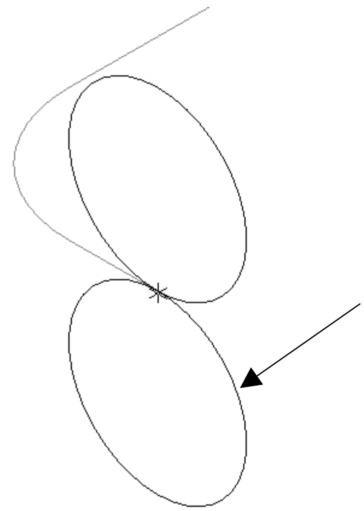
1. Set the **Gview** to **Isometric**.
2. Set the **Cplane** to **Front**.
3. Choose the **Z** button.
4. Select the line shown in the following picture to set the Z depth to 80.



5. Choose **Main Menu, Create, Arc, Tangent, 1 entity**.
6. Select the line again at position 1.

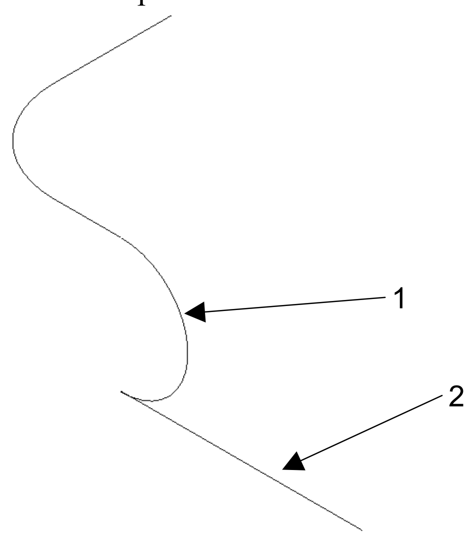


7. Choose the tangent point by selecting the endpoint at position 2.
8. Enter the radius of **24**.
9. Select the arc shown in the picture at right.

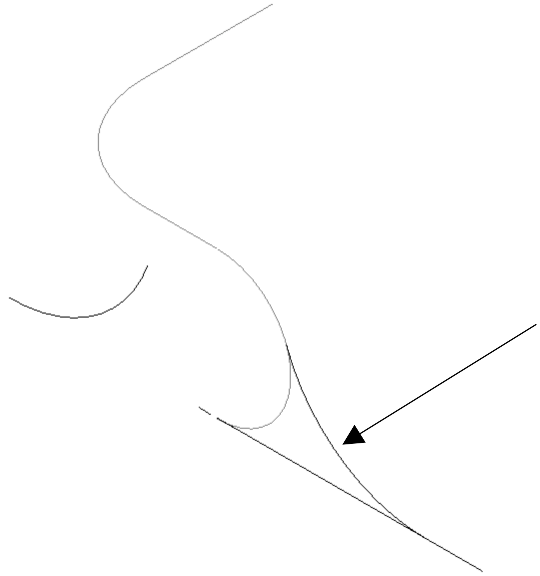


10. Choose **Main Menu, Create, Line, Horizontal**.
11. Draw the line as shown in the following picture.

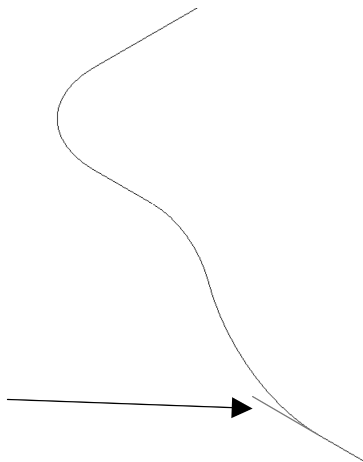
12. Enter a Y coordinate of **–47.74**.
13. Choose **Main Menu, Create, Fillet, Radius**.
14. Enter **48** for the fillet radius.
15. Select the arc and line at positions 1 and 2.



16. Select the fillet shown in the following picture.



The part should look like the following picture.

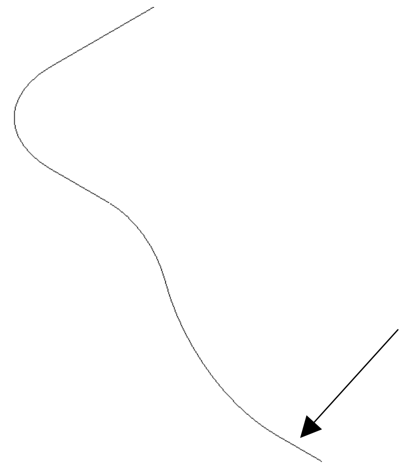


▶ *Drawing the bottom edge*

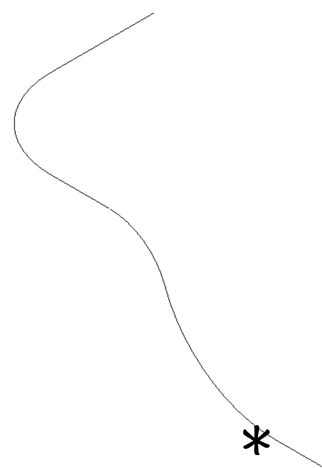
To draw the bottom edge, you need to use two Cplanes. You need the Side Cplane to draw the small arc in the center, and the Top Cplane to draw the arc which joins it to the front face.

1. Set the Cplane to **Top**.
2. Choose the **Z** button.

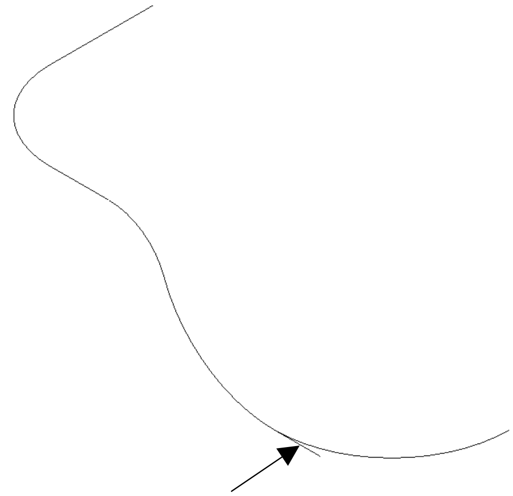
3. Select the lower line to set the Z depth to **–47.74**.



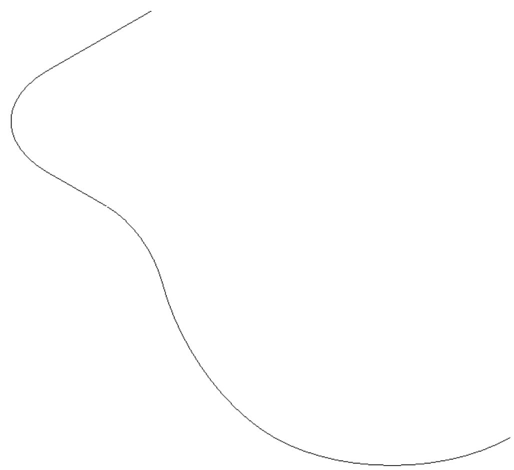
4. Choose **Main Menu, Create, Arc, Polar, Start pt**.
5. Choose **Endpoint**. Now when you select geometry, the AutoCursor will only highlight on endpoints.
6. Select the endpoint of the fillet as shown in the following picture.



7. Enter **50** for the radius.
8. Enter an initial angle of **270**.
9. Enter a final angle of **0**. The part should look like the following picture.

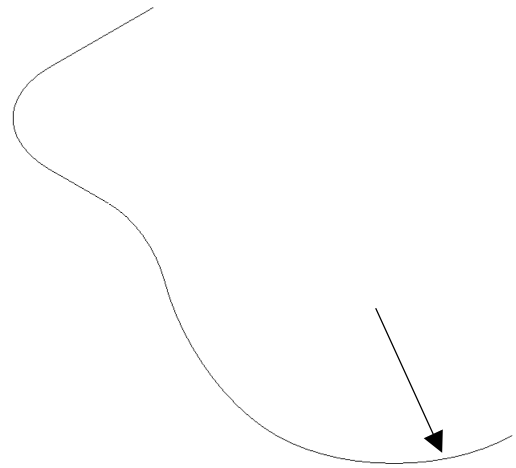


10. Delete the line as shown in the preceding picture. Your part should now look like the following picture.



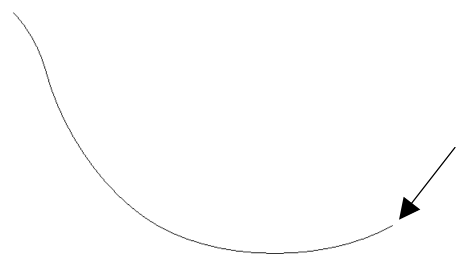
▶ *Drawing the arc on the side of the part*

1. Set the Cplane to **Side**.
2. Choose the **Z** button.
3. Press [E] to lock the selection on endpoints and select the arc as shown in the following picture.

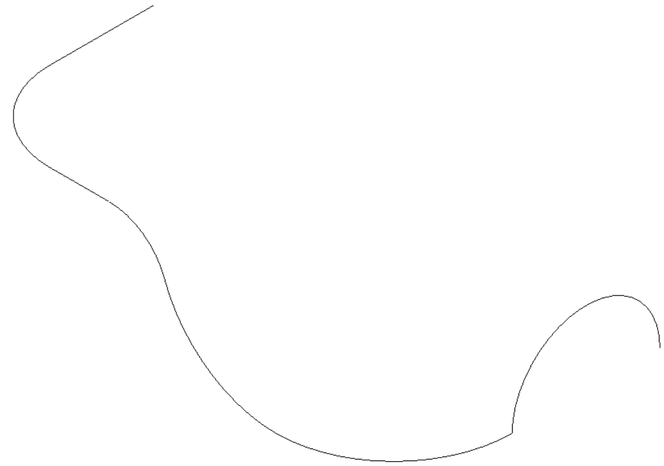


The Z depth should be 177.79.

4. Choose **Main Menu, Create, Arc, Polar, End pt**.
5. Select the endpoint of the arc as shown in the following picture.



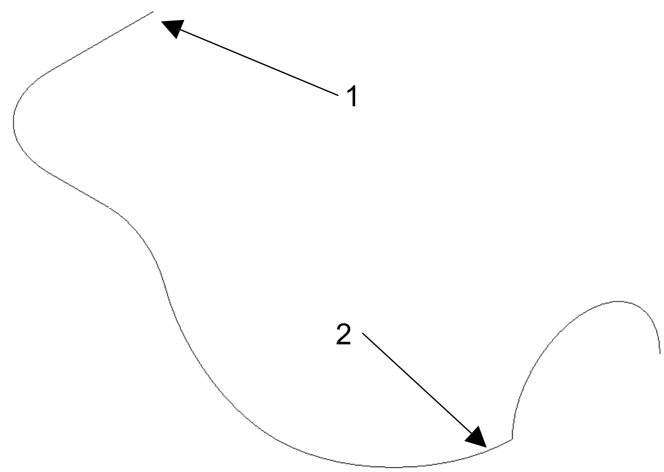
6. Enter **30** for the radius of the new arc you're creating.
7. Enter **0** for the initial angle.
8. Enter **180** for the final angle. The part should look like the following picture.



Finishing the contour by mirroring the geometry

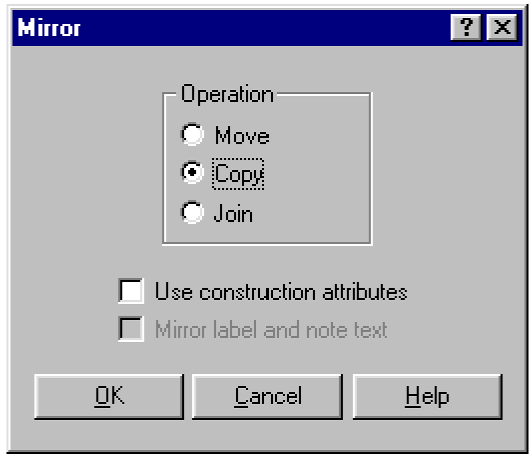
Now that you've completed half of the contour, you can create the other half by mirroring what you've already created.

1. Set the Cplane to **Top**.
2. Choose **Main Menu, Xform, Mirror, Chain, Partial**.
3. Select the starting point for the chain at position 1.

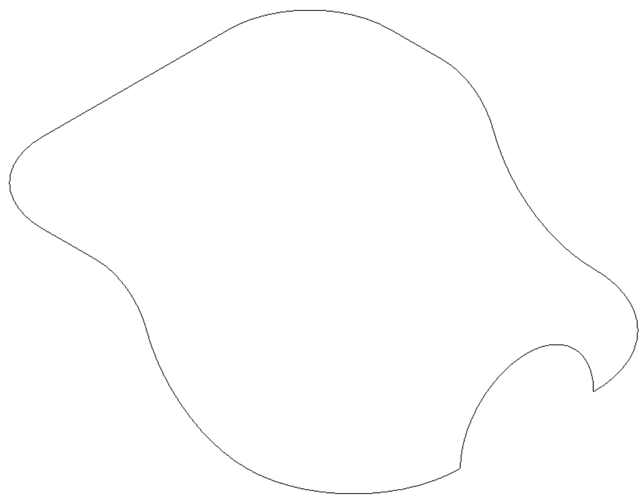


4. Select the end of the chain at position 2. Notice that the final arc was not chosen; the selected geometry will fit around it when you mirror it.
5. Choose **Done, Done, X axis**.

6. Choose **Operation–Copy**.



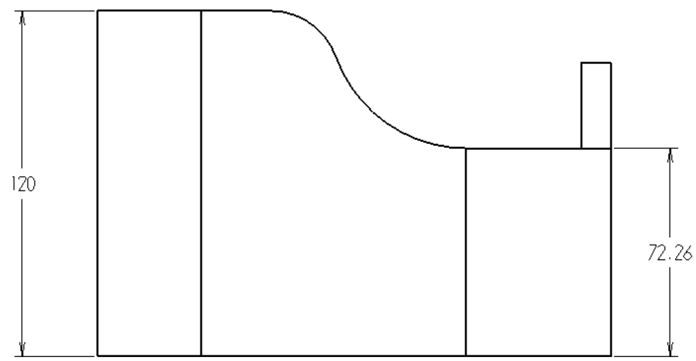
7. Choose **OK**. The completed contour should look like the following picture.

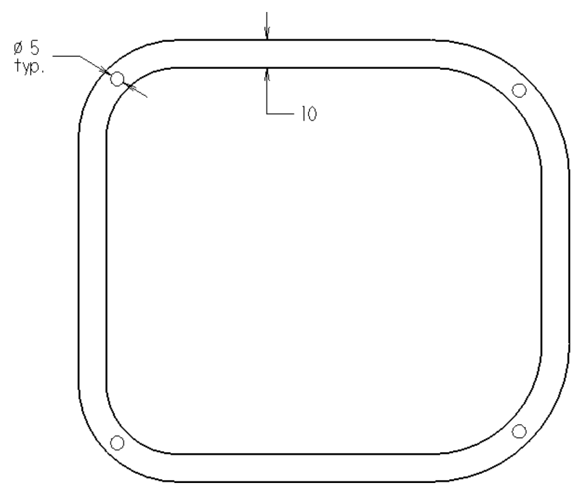


8. Choose **Main Menu, File, Save** and save the part in your working folder as **3D cover.mc9**.

Exercise 2 – Drawing the bottom of the part

In this exercise, you will draw the bottom edge of the part and add the four mounting holes. The following pictures show you the necessary dimensions.





You will use the following skills:

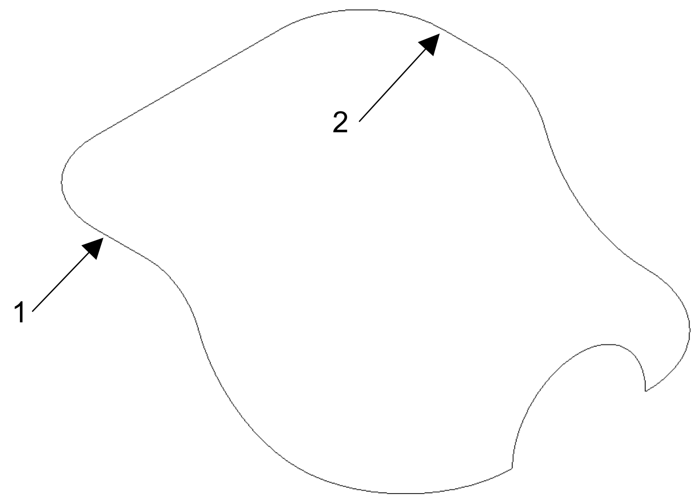
- **Copying geometry with the Translate function**
- **Offsetting a contour**
- **Using dynamic Gviews**
- **Using the View Manager to change the orientation of the coordinate system.**

◆ **Using the new system view to work on the bottom of the part**

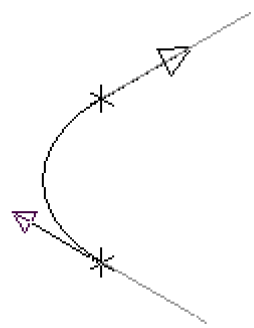
Drawing the bottom contour

First, you will project the four corners to the Z depth of the bottom of the part. Then you will join them with new lines.

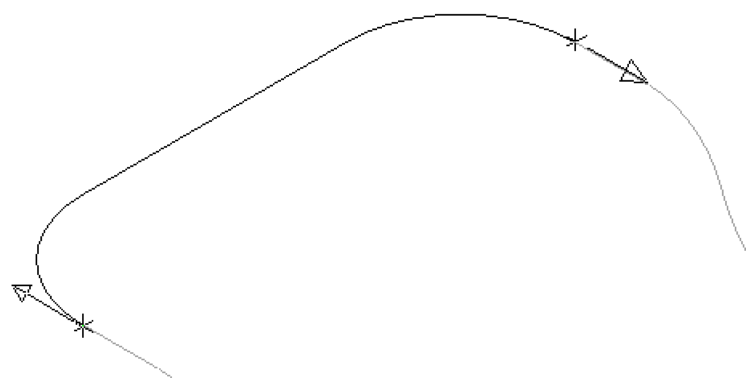
1. Choose **Main Menu, Xform, Translate, Chain, Partial**.
2. Select the endpoint of the corner arc at position 1.



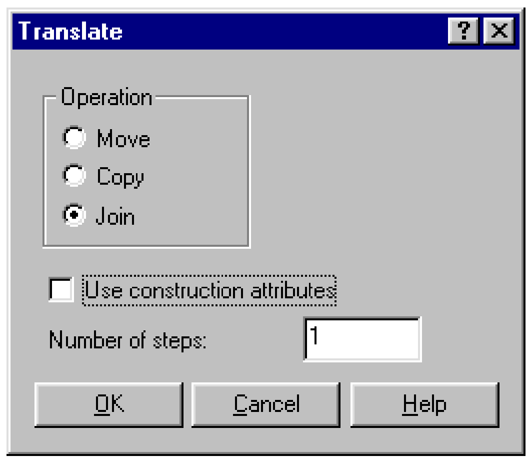
The corner of the arc should be highlighted:



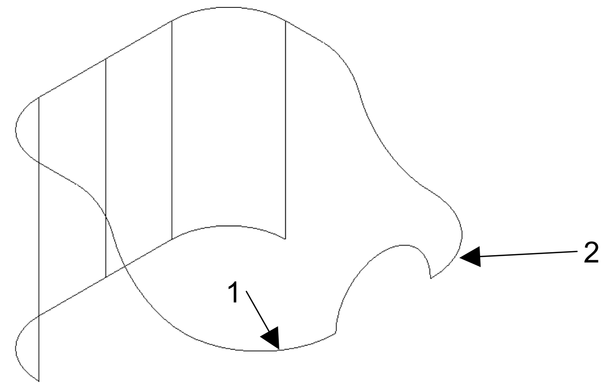
3. Select the endpoint of the other corner arc as shown in position 2 in the preceding picture. The selected geometry should look like the following picture.



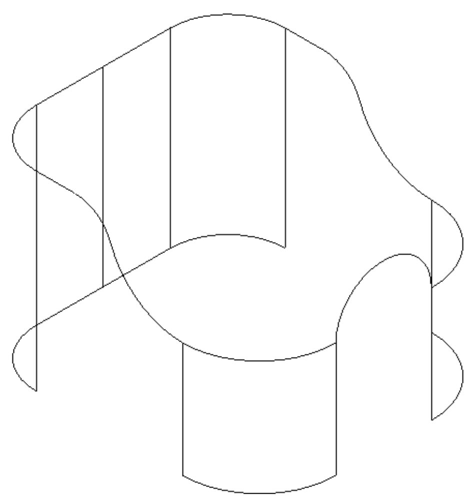
4. Choose **Done, Done, Rectang**.
5. Enter a translation vector of **Z–120**.
6. Choose **Operation–Join**. This tells Mastercam to create guidelines connecting the copy to the original geometry.



7. Choose **OK**. The part should look like the following picture.

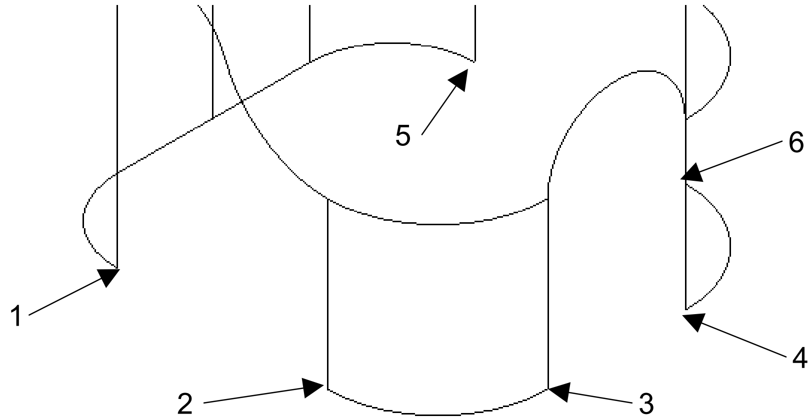


8. To translate the other corners, choose the arcs at position 1 and 2.
9. Choose **Done, Rectang**.
10. Enter **Z–72.26**.
11. Choose **OK** at the Translate dialog box.
12. Fit the part on the screen. It should look like the following picture.

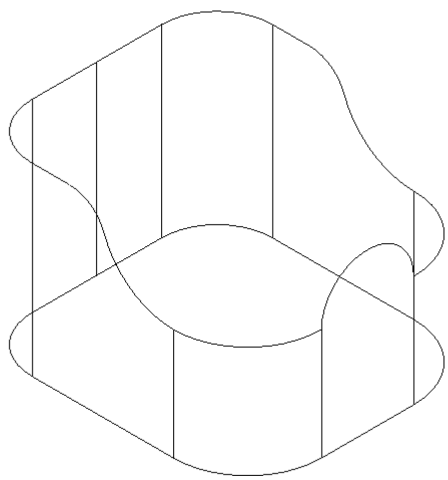


13. Choose the **Z** button and click on any of the new arcs on the bottom of the part. The Z depth should be set to **–120.000**.

14. Choose **Main Menu, Create, Line, Endpoints**. Choose points 1 and 2 in the following picture to create a line joining them.



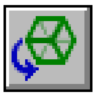
15. Choose points 3, 4, 5, and 6 to complete the other lines.
16. Press [Esc] when you are done. Your part should look like the following picture.



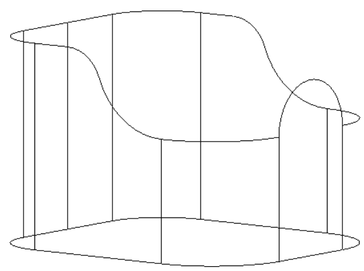
▶ *Drawing the inner boundary*

To draw the inner boundaries of the top and bottom edges, you will create an offset copy of the outer boundaries that you just drew. The difference between translating and offsetting is that the translate function just copies the geometry to the new position, while the offset function adjusts the size and scale of the copy as needed to maintain the offset distance.

1. Choose **Main Menu, Xform, Ofs ctour**.

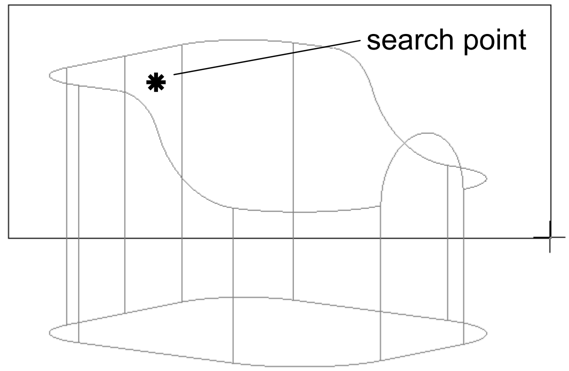


2. You will shift your viewing angle slightly so that you can easily select just the entities that comprise the top edge. Choose the green **Gview–Dynamic** button from the toolbar.
3. Click anywhere in the bottom half of the graphic window and drag the mouse upward until the part looks like the following picture. Notice that the viewing angle changes as you move the mouse.

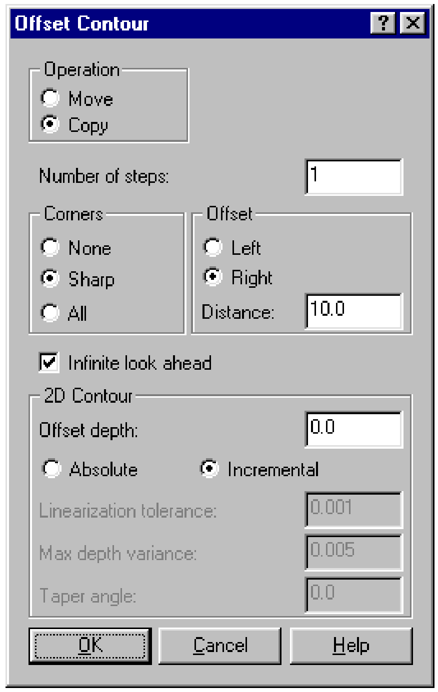


Click the mouse again to freeze the view in this position.

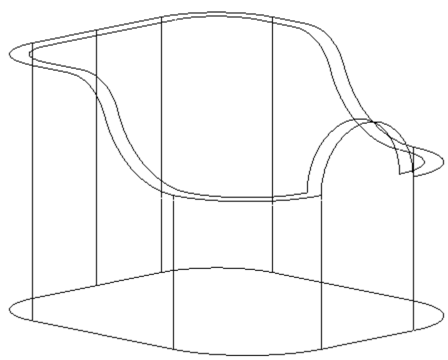
4. Choose **Window**.
5. Click above and to the left of the part and drag a window as shown in the following picture. Include the entire top edge, but none of the bottom edge. Click the mouse when done.



6. When prompted to enter a search point, click near the position indicated in the preceding picture. The entire top edge should highlight.
7. Choose **Done**.
8. Choose **Operation–Copy**.
9. Choose **Right** for **Offset** direction.
10. Enter a **Distance** of **10**. Your selections should match the following picture.

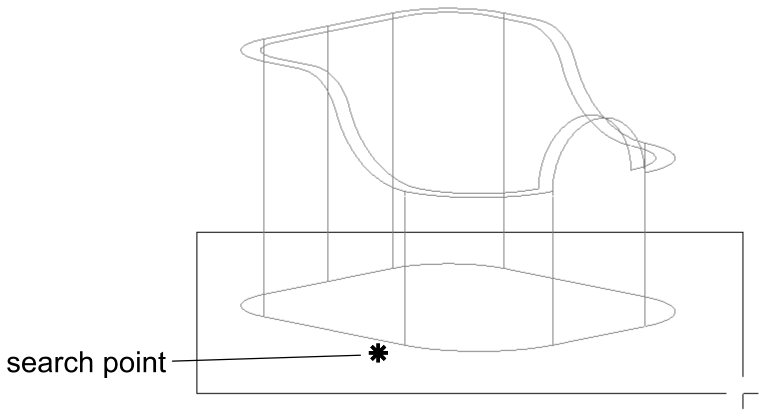


11. Choose **OK**. The inner boundary should appear as shown in the following picture.

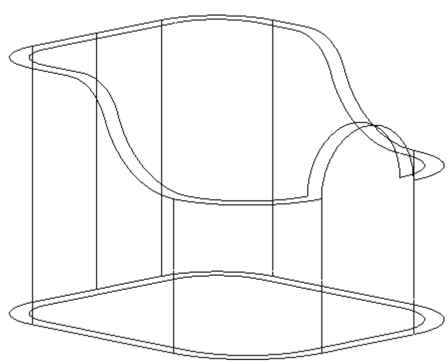


12. Choose **Window** to begin offsetting the bottom edge.

13. Draw a window around the bottom edge.



14. When prompted for a search point, click near the position indicated in the preceding picture.
15. Choose **Done**.
16. Choose **OK** when the Offset Contour dialog box appears, since you will use all the same values as before. Your part should look like the following picture.

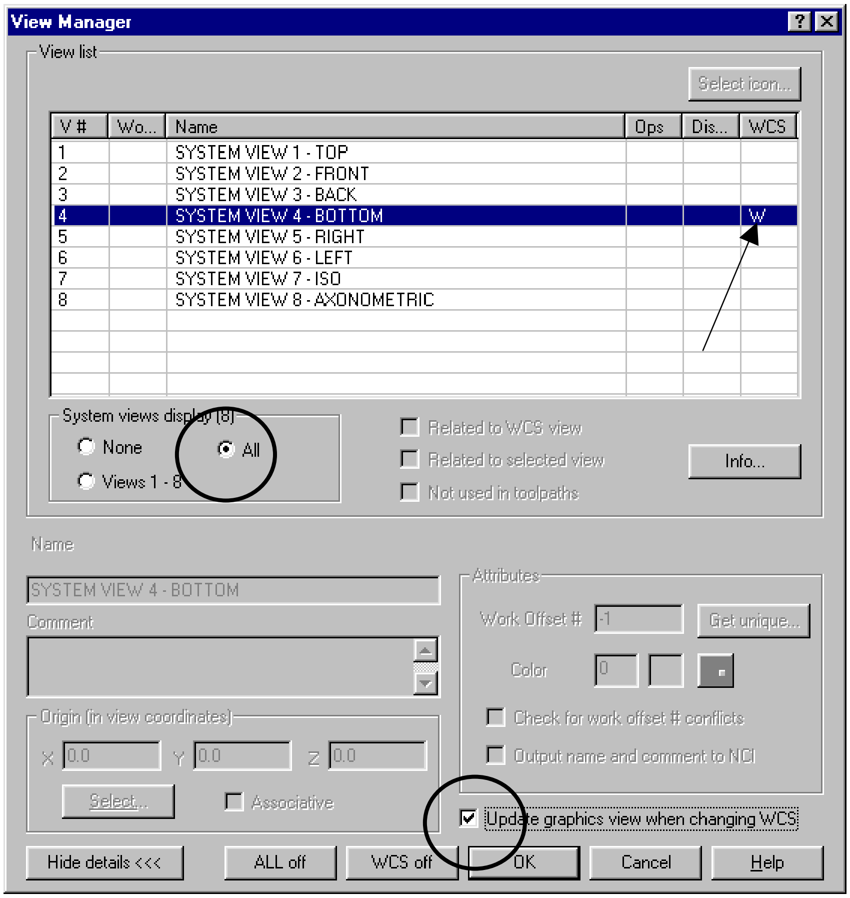


Switching the system view

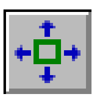
Now you need to create the holes on the bottom of the part. In this procedure, you will switch the system view from Top to Bottom so that you can work more easily on the bottom of the part. Switching the system view means that the entire coordinate system shifts; even the orientation of the Cplanes and Gviews changes so that they are relative to the new system view.

1. Switch to **Gview–Isometric** to cancel the dynamic Gview you've been working in.

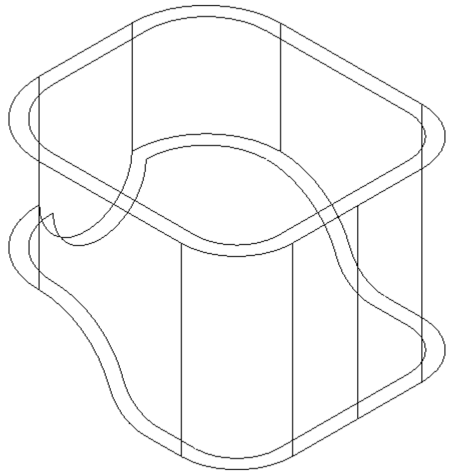
2. Choose the **WCS** button from the Secondary Menu. The View Manager dialog box displays.
3. Under **System Views Display**, choose **All**.
4. Activate **System View–Bottom** by selecting the **WCS** column for **V# 4** as shown in the following picture.
5. Select the **Update graphics view when changing WCS** check box. Your selections should match the following picture.



6. Choose **OK**.

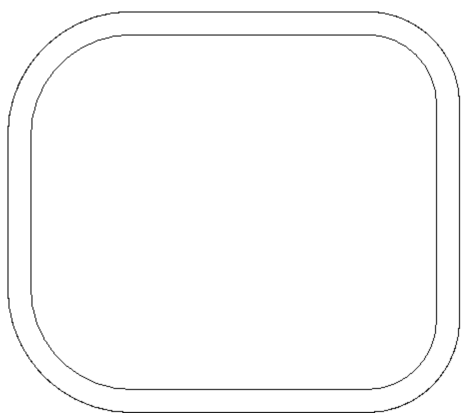


7. Choose the **Screen–Fit** button from the toolbar. You should now be looking at the part in an isometric view from the bottom.



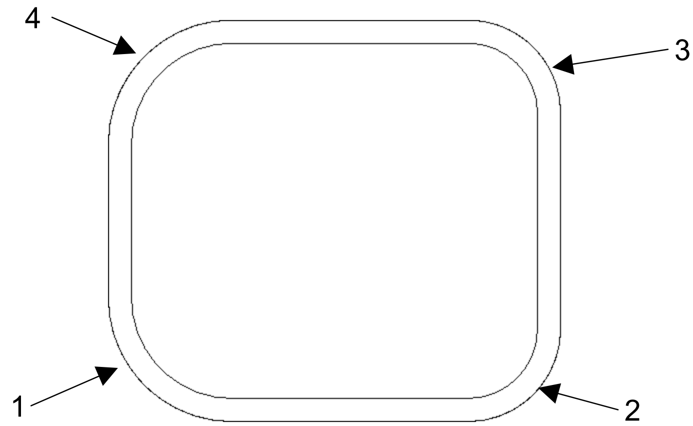
▶ *Drawing the holes on the bottom*

1. Choose the **Z** button and click anywhere on the bottom edge of the part. The Z depth should be **120.000**.
2. Switch the Gview to Top and fit the part in the screen. Since you switched to System View–Bottom, all of the Gviews have also flipped over, like the isometric view in the preceding picture. So now when you select Gview–Top, you are actually looking at the bottom of the part. It should look like the following picture.

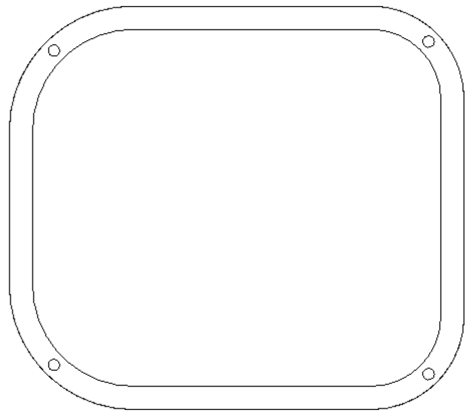


3. Now create the holes. Choose **Main Menu, Create, Arc, Circ pt + dia**.
4. Enter **5** for the diameter.

5. Choose **Relative, Midpoint**.
6. Select arc 1 as shown in the following picture.



7. Choose **Polar**.
8. Enter **5** for the relative distance.
9. Enter **45** for the relative angle.
10. Repeat steps 5 through 8 for arc 2.
11. Enter **135** for the relative angle.
12. Repeat steps 5 through 8 for arc 3.
13. Enter **225** for the relative angle.
14. Repeat steps 5 through 8 for arc 4.
15. Enter **315** for the angle. The part should look like the following picture.

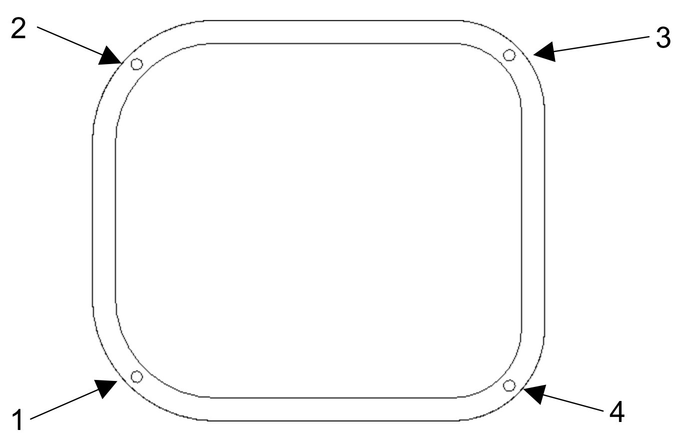


Exercise 3 – Creating a drill toolpath in the new system view

In this exercise, you will create a drill toolpath to drill the holes from the bottom. By using System View–Bottom, you can use standard values for the drilling parameters; Mastercam uses the system view selection to automatically orient the geometry of the drilling motions. For example, you will still use a minus–Z value for the depth since you're drilling into the part, even though the drilling direction is upward.

▶ *Creating the drillpath*

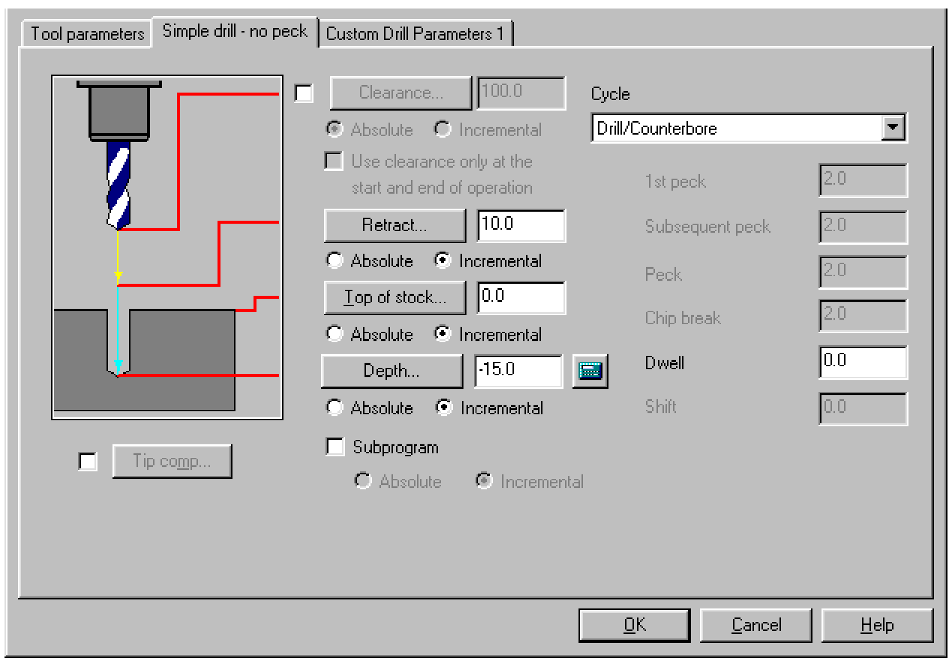
1. Choose **Main Menu, Toolpaths, Drill, Manual, Center**.
2. Select hole number 1 in the following picture.



3. Choose **Center** and hole # 2.
4. Repeat step 3 for holes 3 and 4.
5. Press [Esc].

   Note: If you are prompted for a starting point, choose the first hole.

6. Choose **Done** to accept the default drilling order. The Drill parameters dialog box displays.
7. Right-click in the tool display area and choose **Get tool from library**.
8. Choose the 5 mm HSS drill and choose **OK**.
9. Choose the **Simple drill – no peck** tab.
10. Select **Incremental** for **Retract**.
11. Select **Incremental** for **Top of Stock** and enter a value of **0**.
12. Select **Incremental** for **Depth** and enter a value of **–15**. Make sure your other values match the following picture.

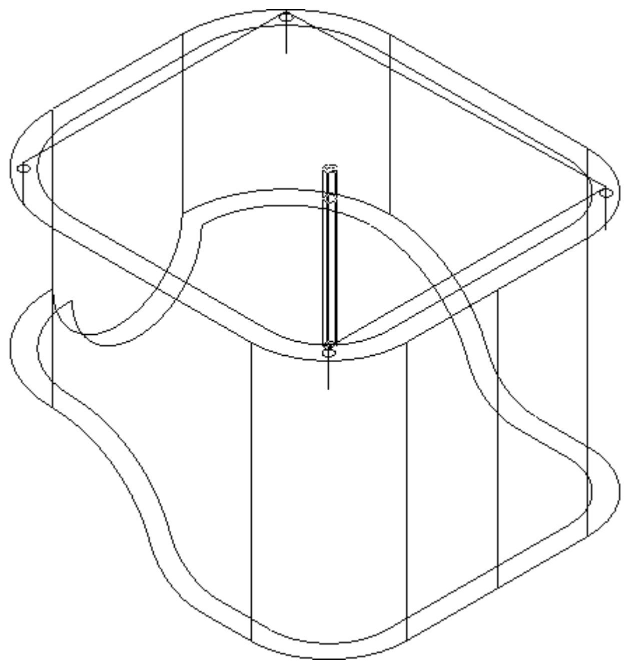


13. Choose **OK**. Mastercam generates the drill toolpath.

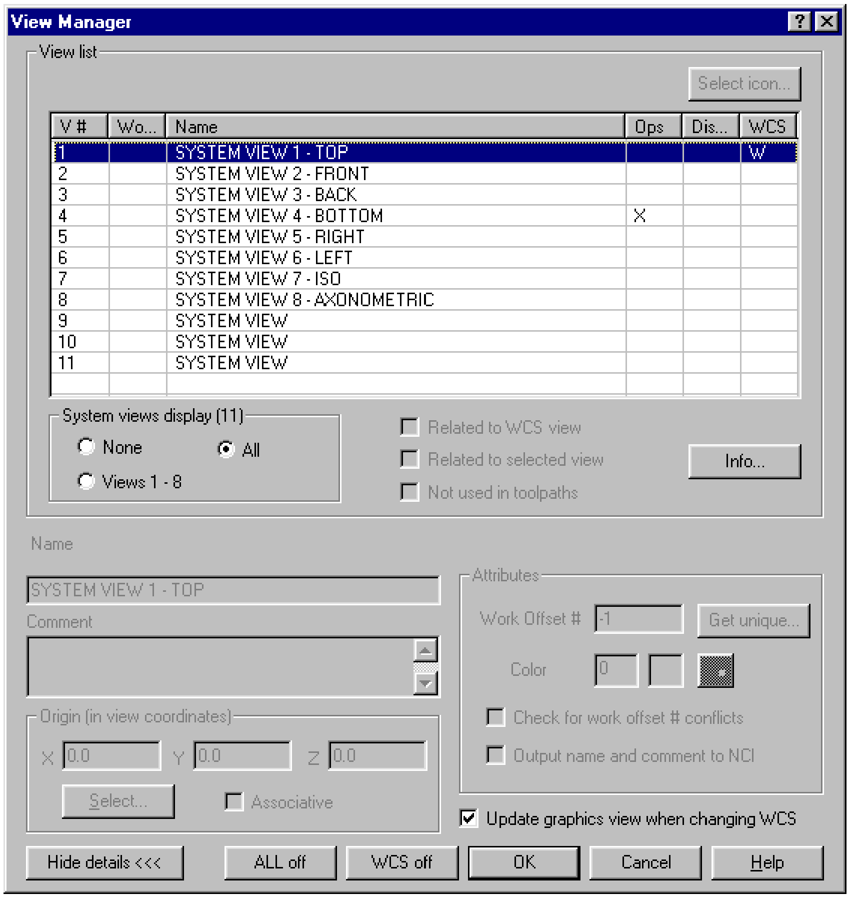
▶ *Backplotting the toolpath*

Backplot the toolpath in the current WCS and then in the original WCS to see the difference.

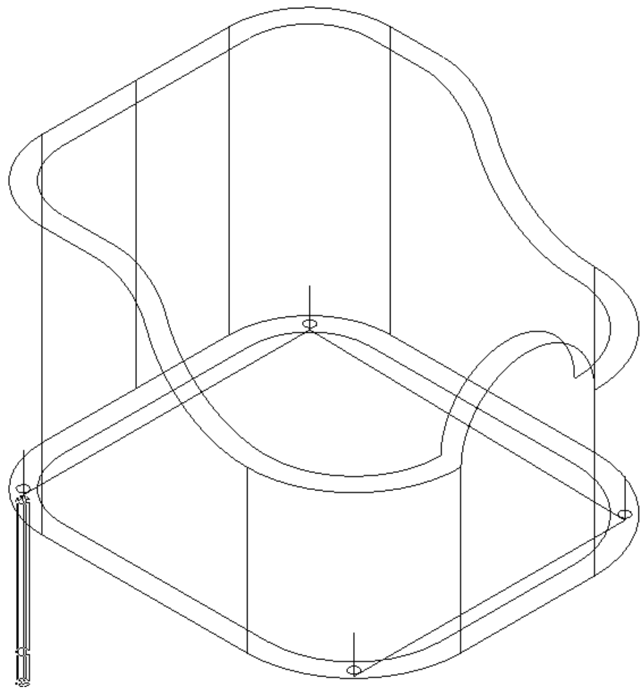
1. Choose **Operations, Backplot**.
2. Set the Gview to **Isometric** so you can see the drilling action more clearly.
3. Press [S] repeatedly to backplot through the drill path. It should look like the following picture.



4. Choose **OK** when the backplot is complete.
5. Choose the **WCS** button from the Secondary Menu.
6. Under **System Views Display**, choose **All**.
7. Activate **System View–Top** by selecting the **WCS** column for **V# 1**.
8. Make sure that **Update graphics view when changing WCS** is selected. Your selections should match the following picture.



9. Choose **OK**.
10. If necessary, fit the part in the screen.
11. Press [S] repeatedly to backplot through the drill path. You should see the part in its normal orientation, with the tool drilling up into the part from the bottom. It should look like the following picture.



12. Choose **OK** when the backplot ends.
13. Choose **OK** to close the Operations Manager.
14. Press [Alt + A] and choose **OK** to save the file.

Now that you've seen a variety of drilling techniques, you're ready to progress to Mastercam's circle milling functions, including thread milling and slot milling.

8 Using Circle Toolpaths

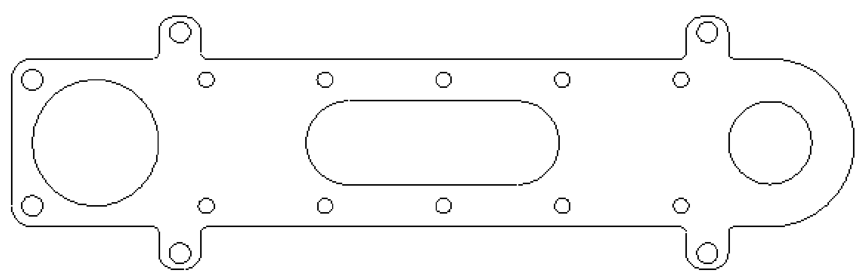
This chapter shows you how to use Mastercam's circle toolpaths to perform a variety of different milling operations:

- circle milling
- helical boring
- slot milling

In addition, you will see several other Mastercam features:

- Auto drilling, which lets you automatically create many different drilling operations at the same time, instead of creating each operation individually.
- Remachining, which lets you efficiently remove the stock left over from a previous operation without cutting the entire contour again.

The part that you will create is the mounting plate shown in the following picture. For this exercise, you will assume that it is a subassembly of a larger part. Because of this, the part doesn't lie flat, but is oriented at an angle in 3D space. You will use the View Manager to create a custom view aligned to the part geometry. This will let you look at the part and create toolpaths as if the part were lying flat. Mastercam then automatically translates all the toolpath geometry for you.



Exercise 1 – Creating a custom view

In this exercise, you will examine the part and create a custom view using the View Manager. This will let you:

- shift the standard system views so that they are aligned with the part geometry.
- move the construction origin to a location convenient to the part geometry.

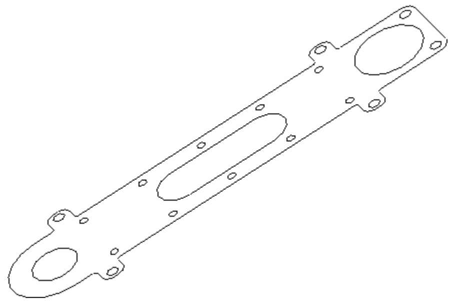
You can then use the new view to create toolpaths while the part appears flat. If you wish, you can even use the View Manager to associate a specific work offset with a view. In this exercise, you will learn the following skills:

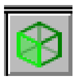
- **Creating and naming a custom view**
- **Orienting views to part geometry**

▶ *Examining the part*

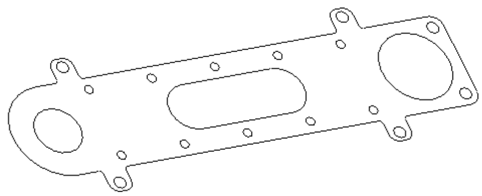
Before creating the new view, review the part and its orientation.

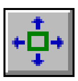
1. Open the file **mounting plate-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**
2. Press [F9] to display the coordinate axes.
3. Press [Page Down] several times to zoom out until you can see the part and axes at the same time. Even though the Gview: T button on the Secondary Menu says that you are looking at the part from the top, you can see that it appears skewed.



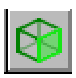


4. Choose the green **Gview–Front** button. Your part should look like the following picture.

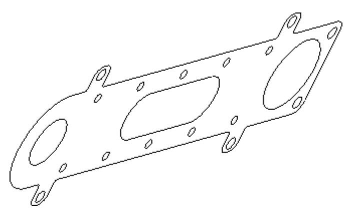




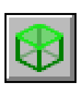
If necessary, choose the **Screen–Fit** button to see the whole part.



5. Choose the green **Gview–Side** button. Your part should look like the following picture.



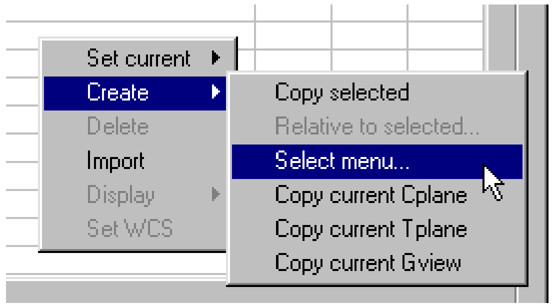
If necessary, choose the **Screen–Fit** button to see the whole part.



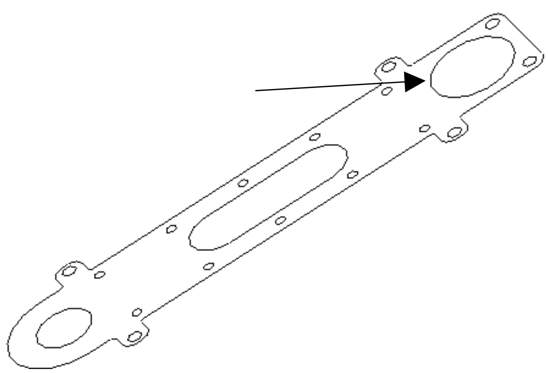
6. You can see that the part isn't aligned with any of the standard views. Choose the green **Gview–Top** button to return to the Top view.
7. Press [F9] to turn off the axes display.

► *Creating the custom view*

1. Choose the **WCS** button on the Secondary Menu to display the View Manager.
2. Right-click in the Views list and choose **Create, Select menu** from the menu. This lets you create a new view based on your part geometry, and displays the Select menu so you can choose the geometry from the graphics window.

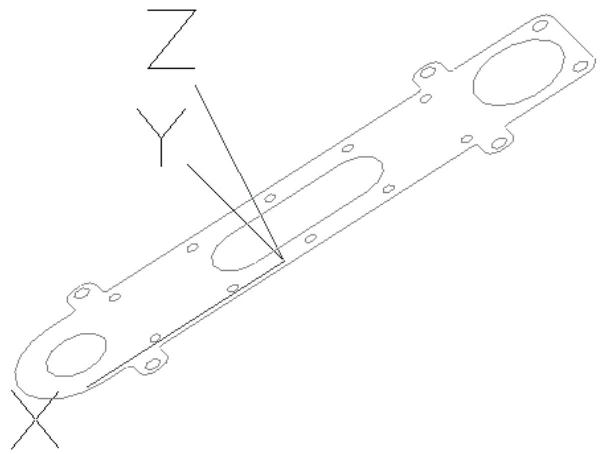


3. Choose **Entity**.
4. Select the arc as shown in the following picture.

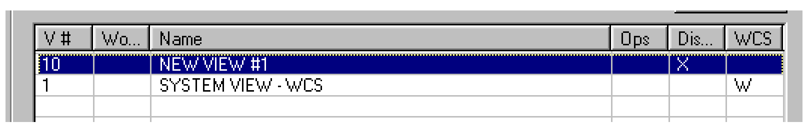


A new set of coordinate axes appears.

5. You want the arc to align with the XY plane. Choose **Next** to rotate the new axes to the desired orientation shown in the following picture.



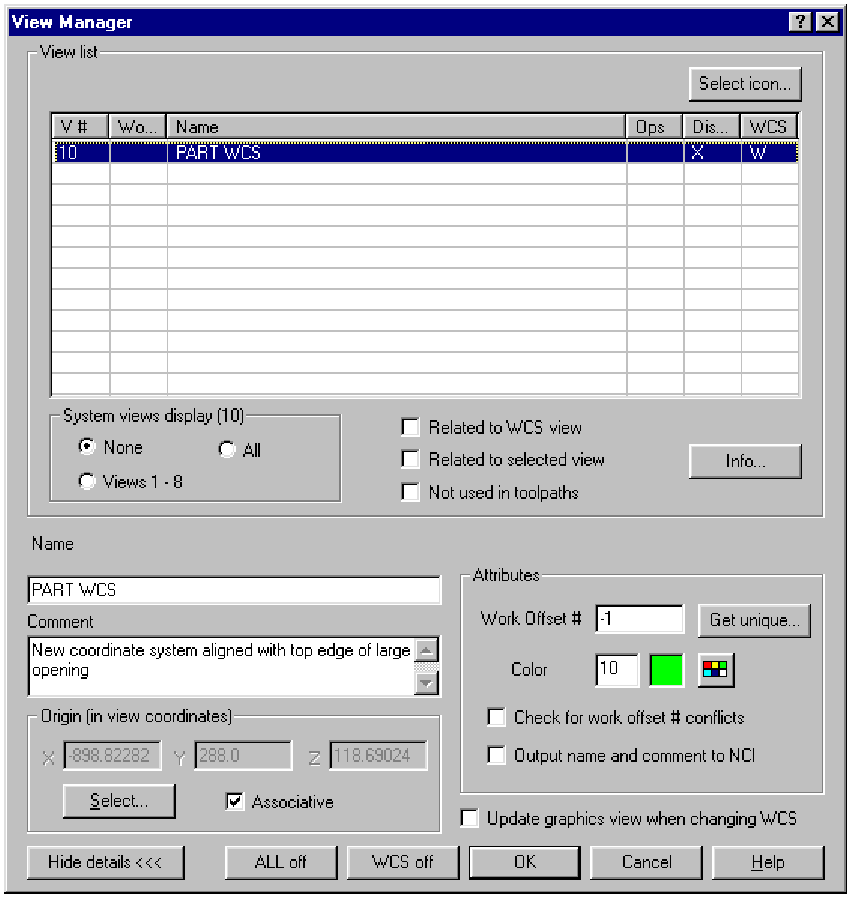
6. Choose **Save**. You will return to the View Manager, where you can see the new view listed.
7. Choose **None** under **System views display**. The new view will be listed as shown in the following picture.



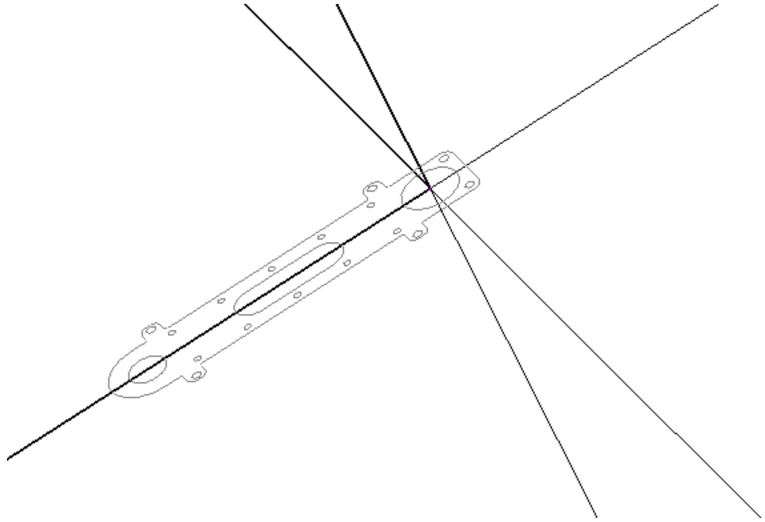
V #	Wo...	Name	Ops	Dis...	WCS
10		NEW VIEW #1		X	
1		SYSTEM VIEW - WCS			W

8. In the **Name** field, type in the following name for the view: **Part WCS**
9. In the **Comment** field, type the following comment: **New coordinate system aligned with top edge of large opening**

   When you post the file, the comment will appear in the NC code created by the post processor.
10. Choose the **WCS** column in the new **Part WCS** line. The "W" should move there. Your View Manager selections should match the following picture.

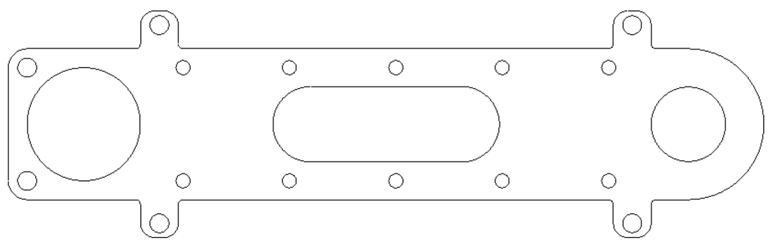


11. Choose **OK** to return to the graphics window.
12. Press [F9] to see the new coordinate axes (in blue) and the original system coordinate axes (in brown). The new axes should look like the following picture.



Notice that the origin is automatically located at the center of the arc.

13. Press [F9] again to clear the axes from the screen.
14. Choose the **Gview–Top** button from the toolbar. If necessary, choose the **Screen–Fit** button. The part appears as if you are looking straight down on it.



15. Choose **Main Menu, File, Save** and save the file to your working folder with the name **new wcs.mc9**.

Exercise 2 – Machining the outside contour

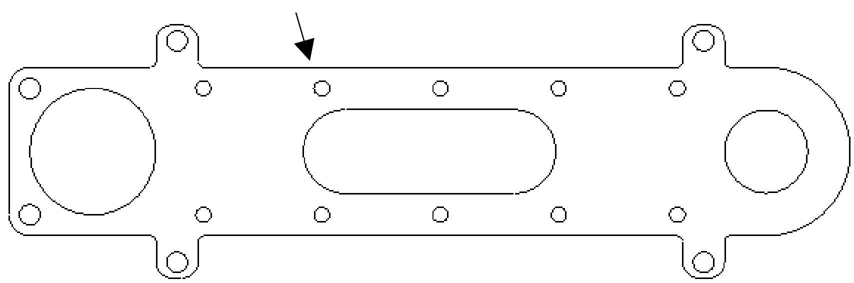
In this exercise, you will machine the outside contour of the part. You will use two different operations. First, you will use a 2D contour toolpath to rough out the contour. Then, you will use a remachining toolpath to remove the stock from the corners that the larger roughing tool couldn't reach. You will see how Mastercam automatically figures out how much stock was left by the larger tool, to create an efficient cutting path for the remachining operation.

In this exercise, you will learn the following skills:

- **Using depth cuts and multiple passes in a contour toolpath**
- **Remachining a part to remove leftover stock**

▶ *Creating the contour toolpath*

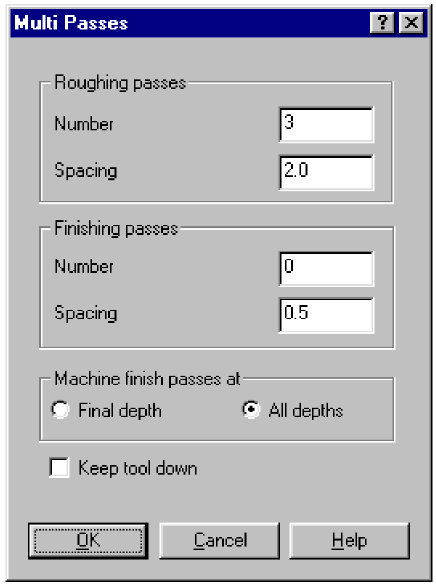
1. Choose **Main Menu, Toolpaths, Contour**.
2. Select a location on the left half of the flat edge as shown in the following picture.



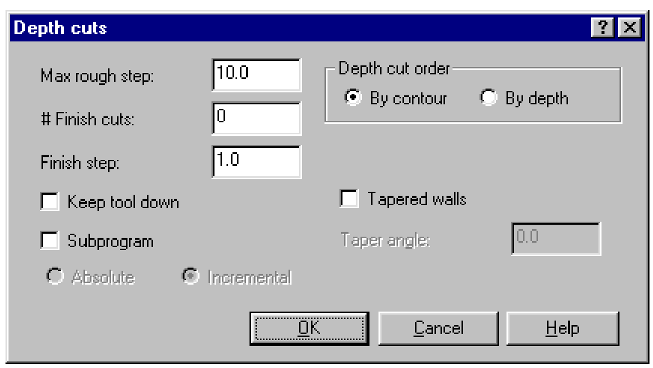
The chaining arrow should point in the clockwise direction when you are done.

3. Choose **Done**.
4. Right-click in the tool display area and choose **Get tool from library**.
5. Choose the 20 mm HSS flat endmill and choose **OK**.
6. Choose the **Contour parameters** tab.
7. Select the **Clearance** check box.

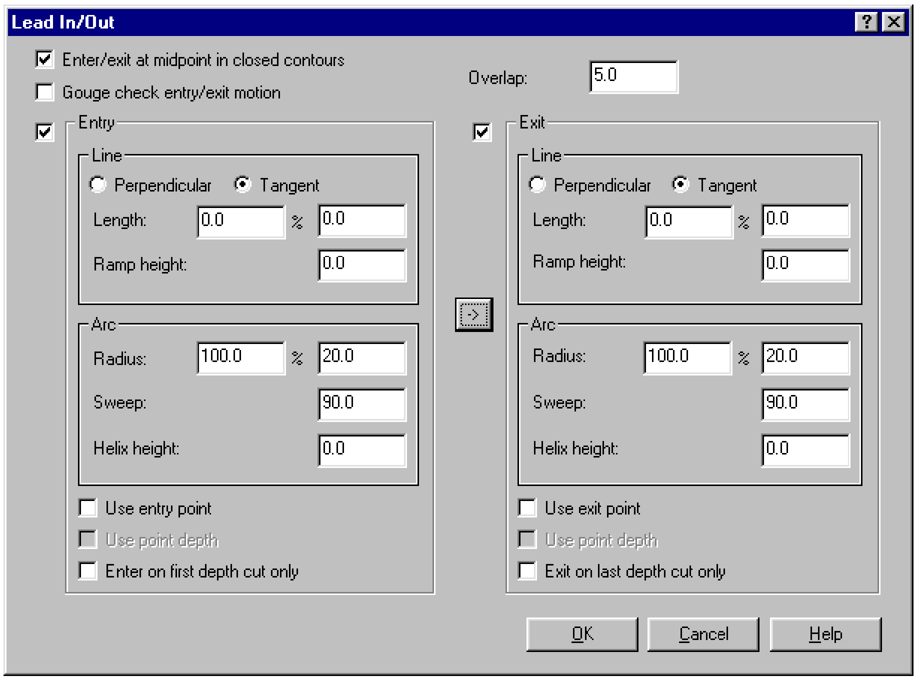
8. Choose **Use clearance only at the start and end of operation**.
9. Enter a **Depth** of **–25**. Make sure the **Absolute** option is selected.
10. Choose the **Multi passes** check box and button.
11. Enter **3** for **Number** of **Roughing passes** and **2** for **Spacing**. Your other values should match the following picture.



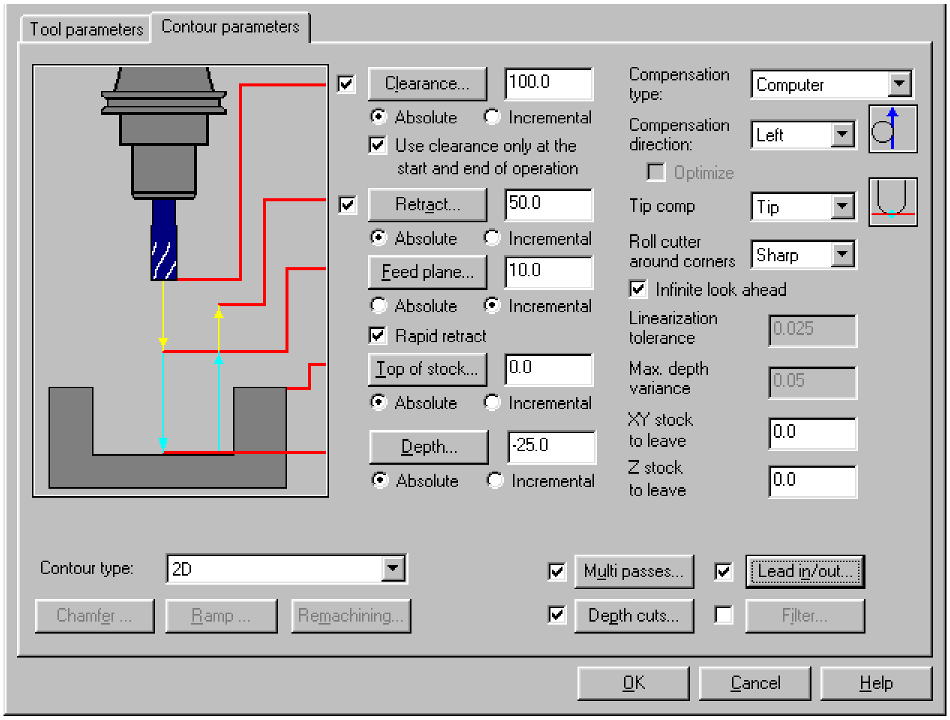
12. Choose **OK**.
13. Choose the **Depth cuts** check box and button.
14. Enter **10** for the **Max rough step**. Your other values should match the following picture.



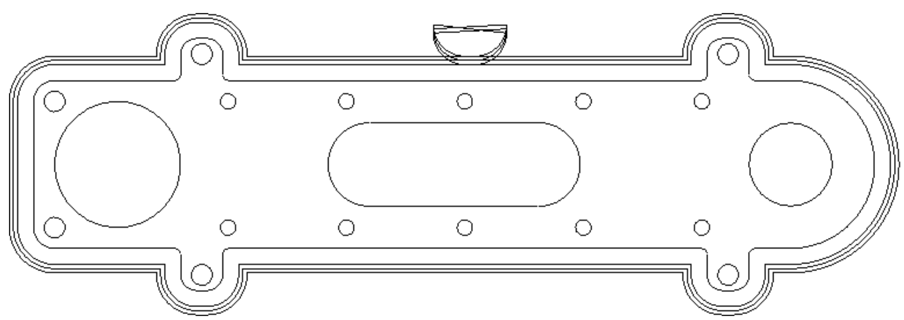
15. Choose **OK**.
16. Choose the **Lead in/out** check box and button.
17. Choose **Enter/exit at midpoint in closed contours**.
18. Enter an **Overlap** value of **5**.
19. Enter **0** for **Entry–Line–Length**.
20. Choose the  button to copy the entry arc dimensions to the Exit section. Your values should match the following picture.



21. Choose **OK**.
22. Verify that your other contour parameters match the following picture, and choose **OK**.



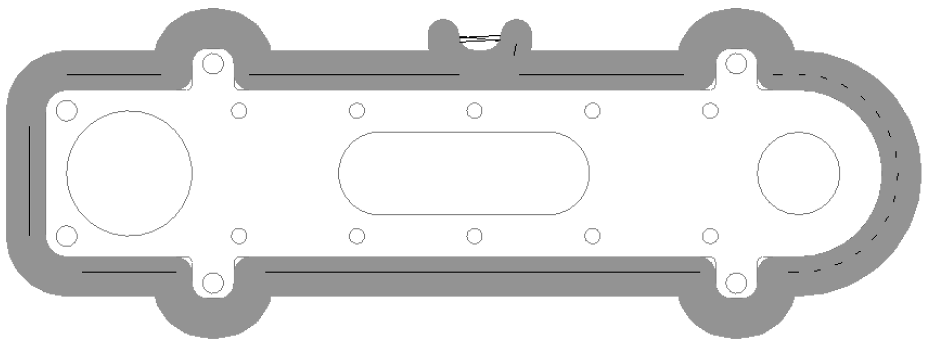
Mastercam generates the toolpath. It should look like the following picture.



▶ *Backplotting the toolpath*

1. Press [Alt + O] to open the Operations Manager.
2. Choose **Backplot**.
3. Toggle Verify to **Y**.

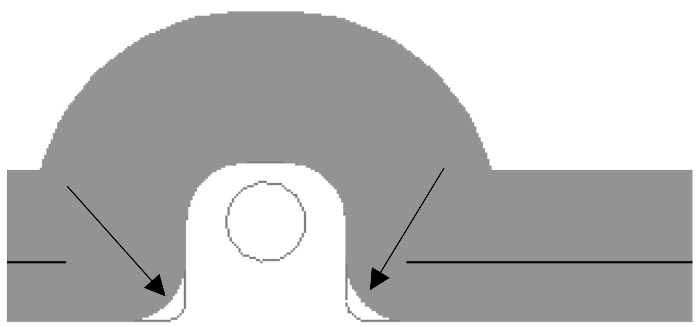
4. Press [S] repeatedly to step through the toolpath. You can see the multiple passes approach the part boundary. When you are done, your toolpath should look like the following picture.



Tip: To step through the toolpath quickly, hold down both mouse buttons while pressing [S]. At any time, press [R] to finish the backplot in one step.

▶ *Cleaning out the corners with remachining*

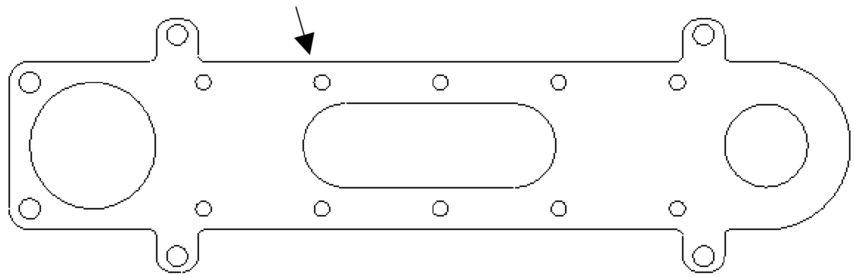
After backplotting the contour toolpath, you can see that the 20 mm endmill is too large to clean out the corners.



You will use a remachining toolpath with a smaller tool to remove the leftover stock. Remachining toolpaths automatically calculate the stock left over from an earlier operation and clean it out.

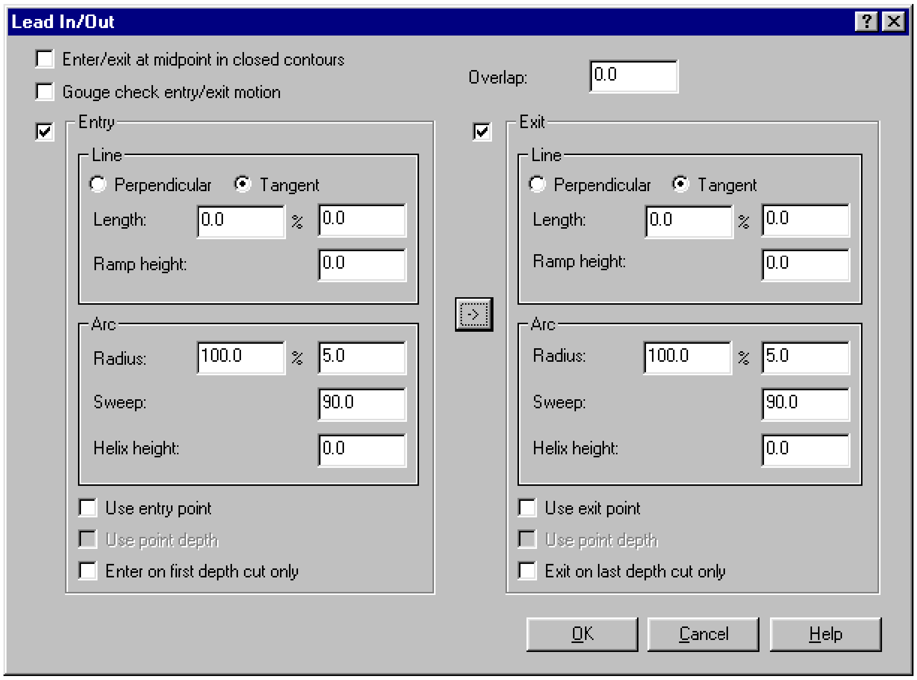
1. When the backplot is complete, choose **Backup** to return to the Operations Manager.
2. Right-click in the operations list window and choose **Toolpaths, Contour** from the menu.

3. Press [Alt + T] to clear the earlier toolpath from the screen.
4. Select a location on the left half of the top edge, just like you did for the previous toolpath.

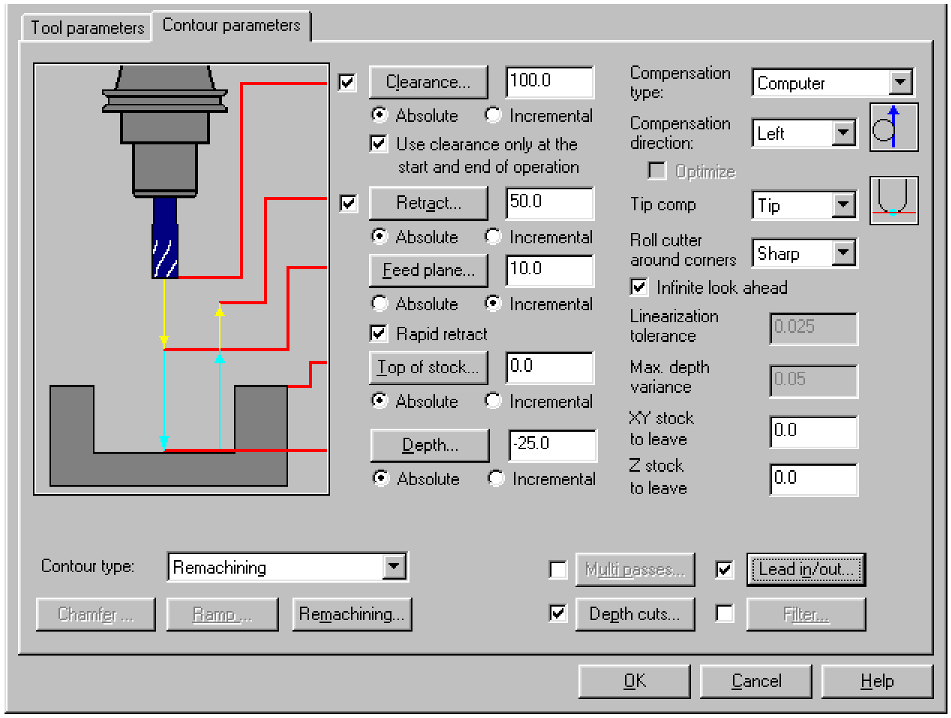


   The chaining arrow should point in the clockwise direction when you are done.

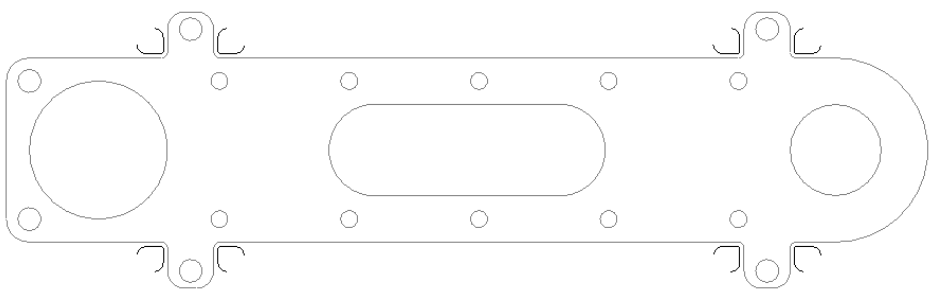
5. Choose **Done**.
6. Select a 5 mm HSS flat endmill from the tool library.
7. Choose the **Contour parameters** tab.
8. Choose **Remachining** from the **Contour type** drop list.
9. Enter a **Depth** of **–25** and make sure the **Absolute** option is selected.
10. Clear the **Multi passes** check box. Since there is only a little stock to remove, you only need a single pass.
11. Choose the **Lead in/out** button.
12. Clear the **Enter/exit at midpoint of closed contour** check box.
13. Enter an **Overlap** value of **0**. Your other values should match the following picture.



14. Choose **OK**.
15. Make sure your other contour parameters match the following picture and choose **OK**.

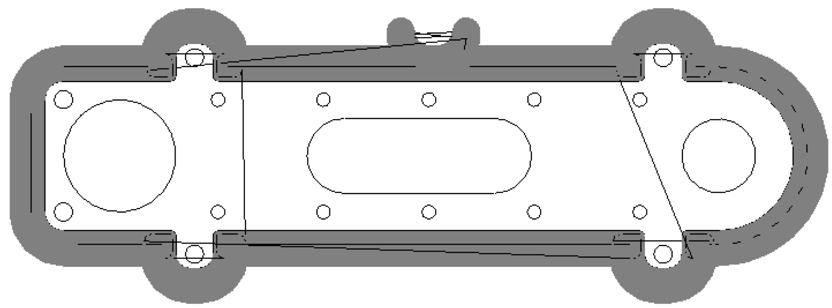


Mastercam generates the following toolpath. You can see that it has created toolpaths for just the small areas of stock left over from the previous operation.



16. Choose **Select All, Backplot, Run**.

    The backplotted toolpaths should look like the following picture. You can see the corners are completely cleaned out.



17. Return to the Operations Manager and close it.
18. Press [Alt + A] and choose **OK** to save the file.

Exercise 3 – Machining the holes and slot

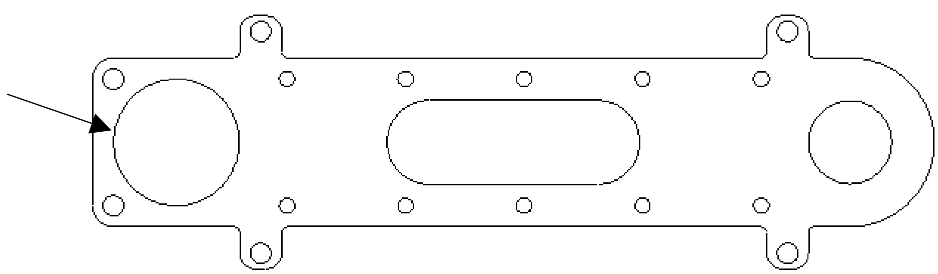
In this exercise, you will use some of Mastercam's circle milling toolpaths to machine the two large holes and the slot. You will learn the following techniques:

- **circle milling**
- **helical boring**
- **slot milling**

Creating a circle milling toolpath

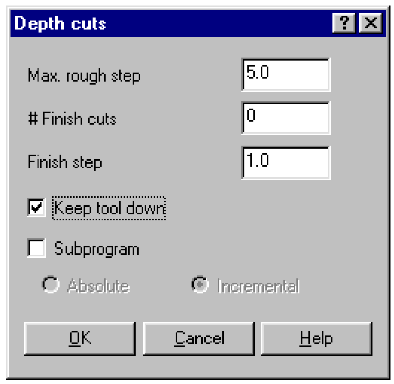
You will machine the largest hole with a circle milling toolpath.

1. Choose **Main Menu, Toolpaths, Next menu, Circ tlpths, Circle mill**.
2. Select the large arc on the left.

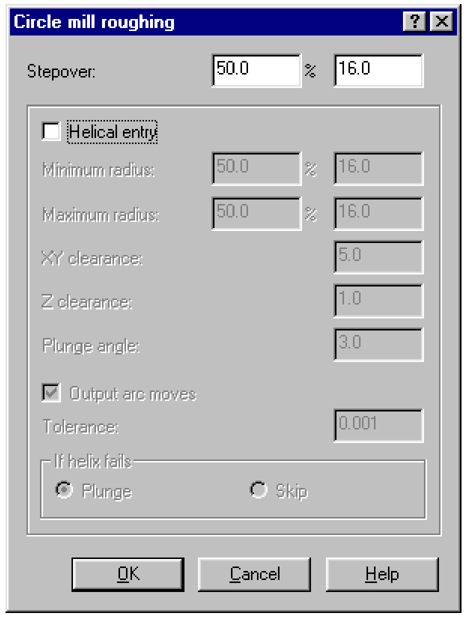


3. Choose **Done** twice.

4. Choose a 32 mm HSS flat endmill from the tool library.
5. Choose the **Circmill parameters** tab.
6. Enter **–25** for the **Depth**.
7. Enter **3** for the **Overlap**.
8. Choose the **Depth cuts** check box and button.
9. Enter **5** for the **Max. rough step**.
10. Select **Keep tool down**. Your values should match the following picture.

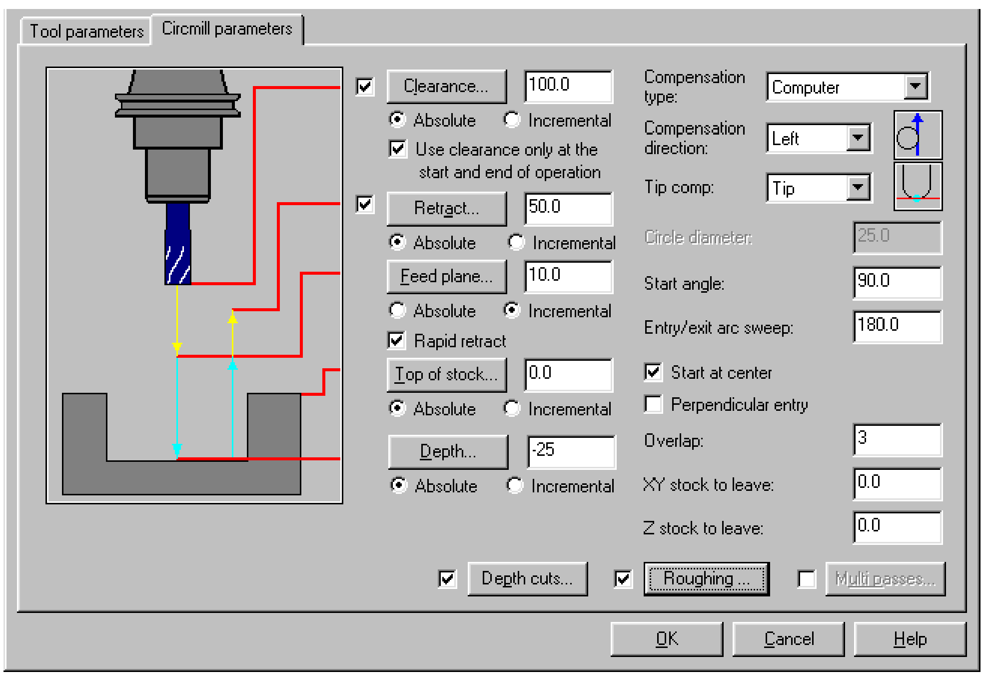


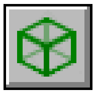
11. Choose **OK**.
12. Choose the **Roughing** check box and button.
13. Clear the **Helical entry** check box. Your roughing parameters should match the following picture.



14. Choose **OK**.

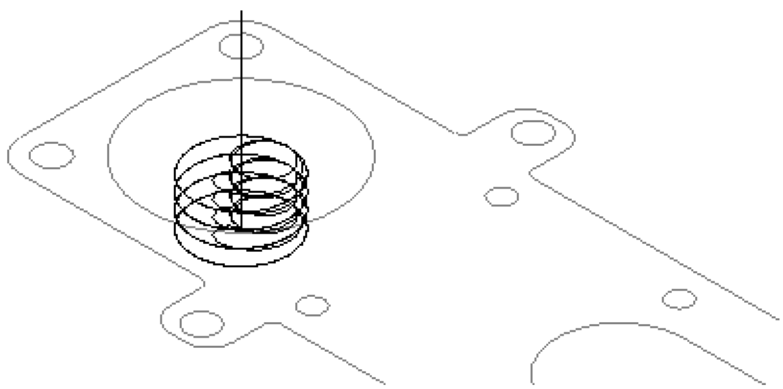
15. Make sure your other circle milling parameters match the following picture and choose **OK**.





16. Choose the green **Gview–Isometric** button on the toolbar. If necessary, choose the **Screen–Fit** button to fit the part on the screen.
17. Press [Alt + O] to display the Operations Manager.
18. Choose the **Circle Mill** operation so the blue check mark appears.

19. Choose **Backplot**.
20. Toggle Verify to **N**.
21. Press [S] to step through the toolpath. You should see the circle milled as shown in the following picture.



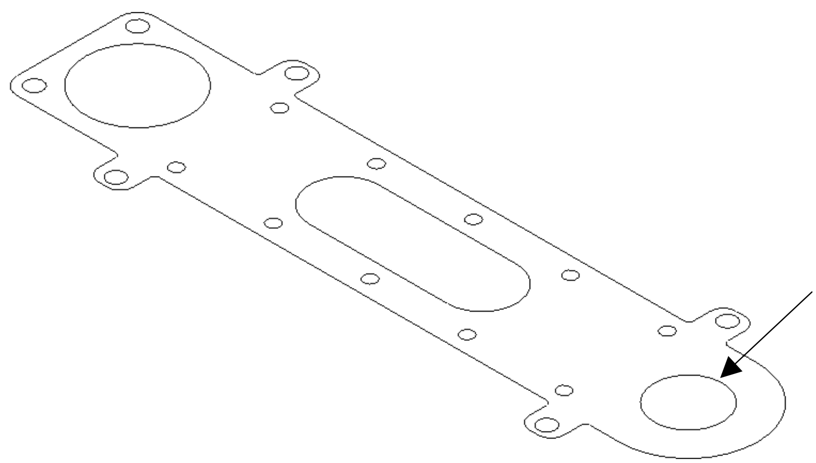
22. Close the Operations Manager when the backplot is done.

▶ *Creating a helical boring operation*

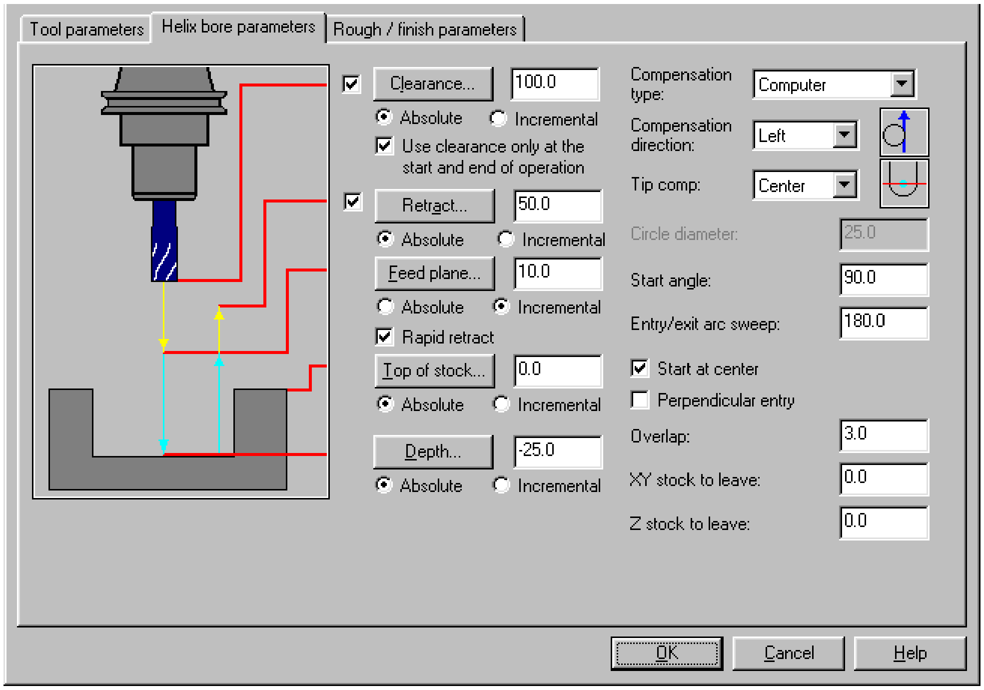
Mastercam's helix boring function takes advantage of the special features of Felix® tools. You will use it to bore the other large hole. In this exercise, you will use a 32 mm flat endmill to approximate the Felix® tool.

1. Choose **Main Menu, Toolpaths, Next menu, Circ tlpths, Helix bore**.

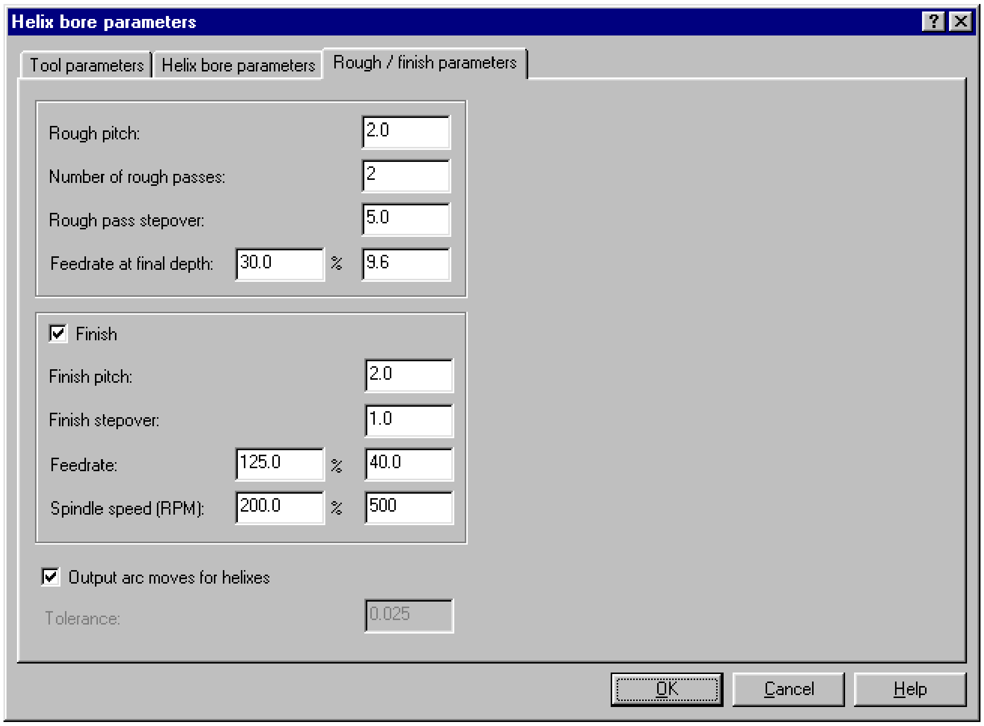
2. Select the large arc on the right.



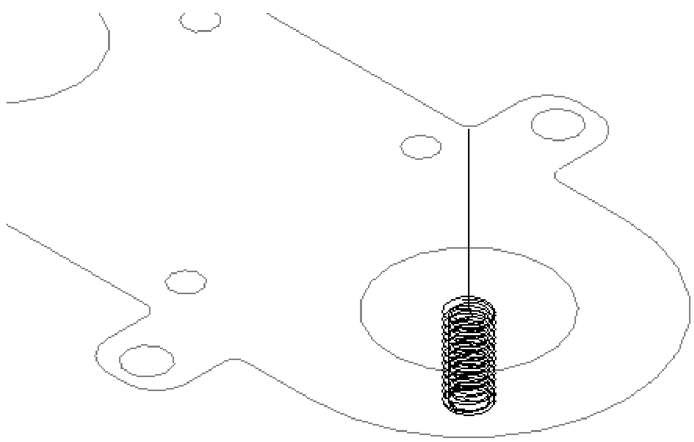
3. Choose **Done** twice.
4. Choose the 32 mm flat endmill.
5. Choose the **Helix bore parameters** tab.
6. Enter **–25** for the **Depth**.
7. Choose **Start at center**.
8. Enter **3** for the **Overlap**. Your values should match the following picture.



9. Choose the **Rough/finish parameters** tab.
10. Enter **2** for **Rough Pitch**.
11. Enter **2** for **Number of Rough passes** and **5** for the **Rough pass stepover** amount.
12. Select the **Finish** check box.
13. Enter **1** for the **Finish stepover**. Your values should match the following picture.



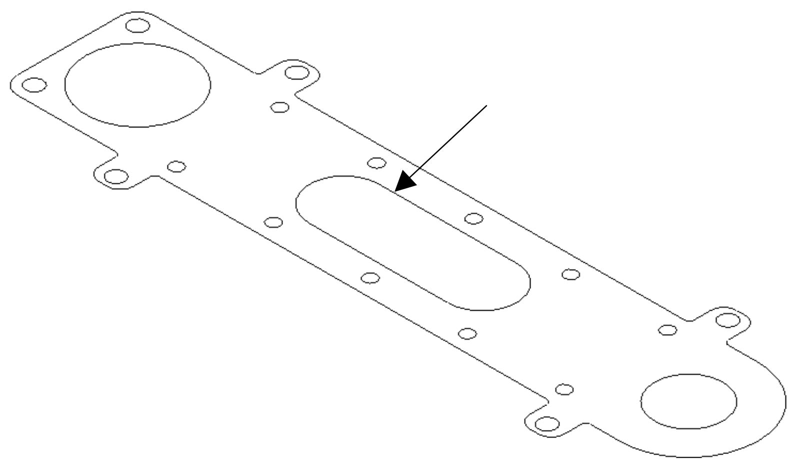
14. Choose **OK**. Mastercam generates the toolpath. It should look like the following picture.



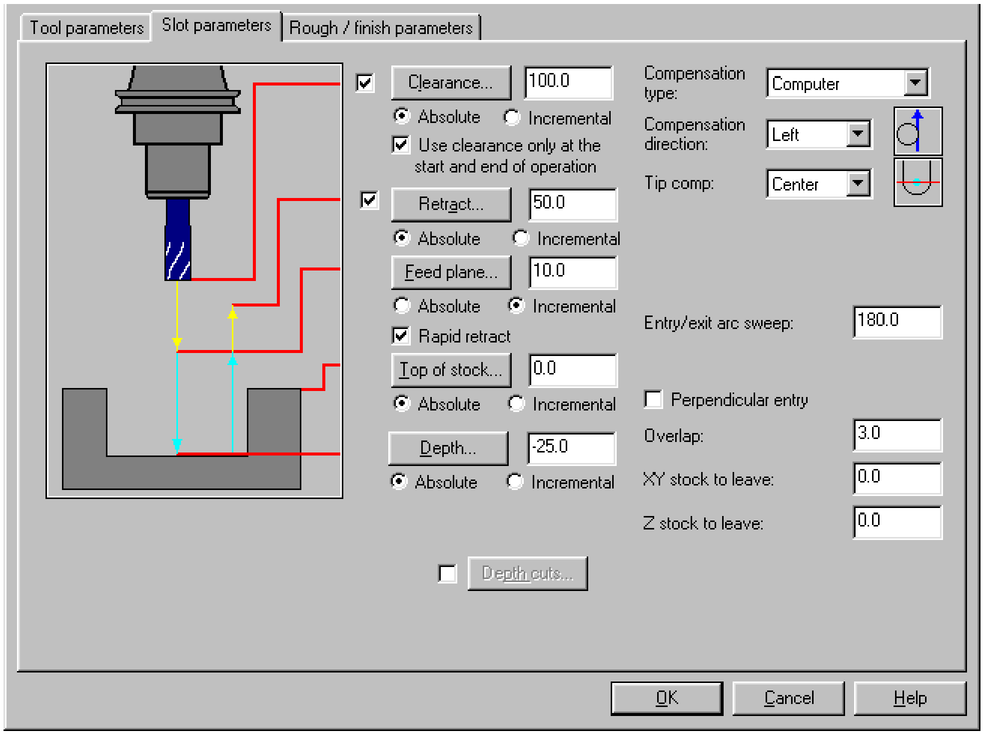
▶ *Creating a slot milling operation*

Mastercam also includes a slot milling toolpath that is a variation of the circle milling toolpath. Use it to machine the slot in the middle of the part.

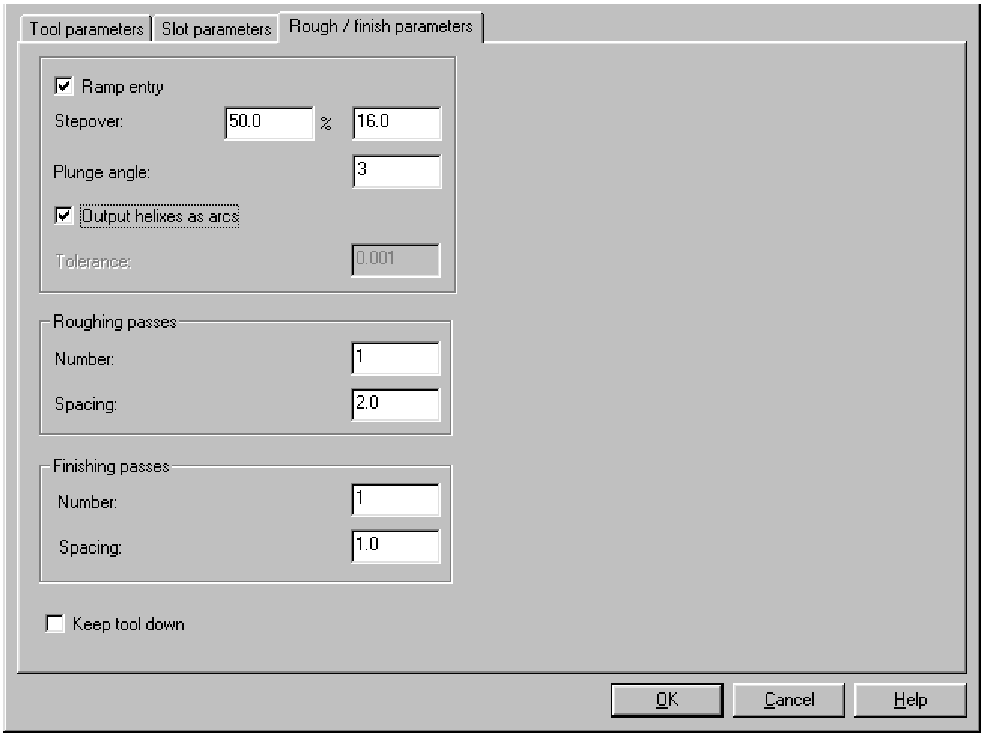
1. Choose **Circ tlpths, Slot mill**.
2. Select the slot as shown in the following picture.



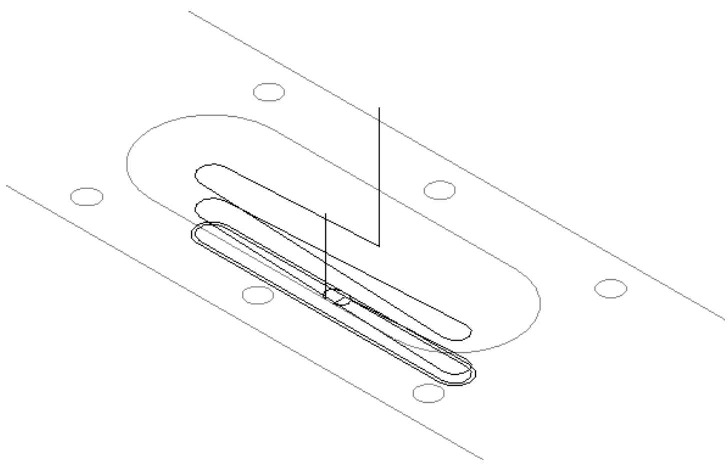
3. Choose **Done**.
4. Choose the 32 mm flat endmill.
5. Choose the **Slot parameters** tab.
6. Enter **–25** for the **Depth**.
7. Enter **3** for the **Overlap**. Your values should match the following picture.



8. Choose the **Rough/Finish parameters** tab.
9. Enter a **Plunge angle** of **3**.
10. Enter **1** for **Finishing passes–Number** and **1** for **Spacing**. Make sure your other parameters match the following picture.



11. Choose **OK**. Mastercam generates the toolpath shown in the following picture. You can see that the 3-degree plunge angle causes the tool to plunge into the part gradually and continuously during the roughing pass.

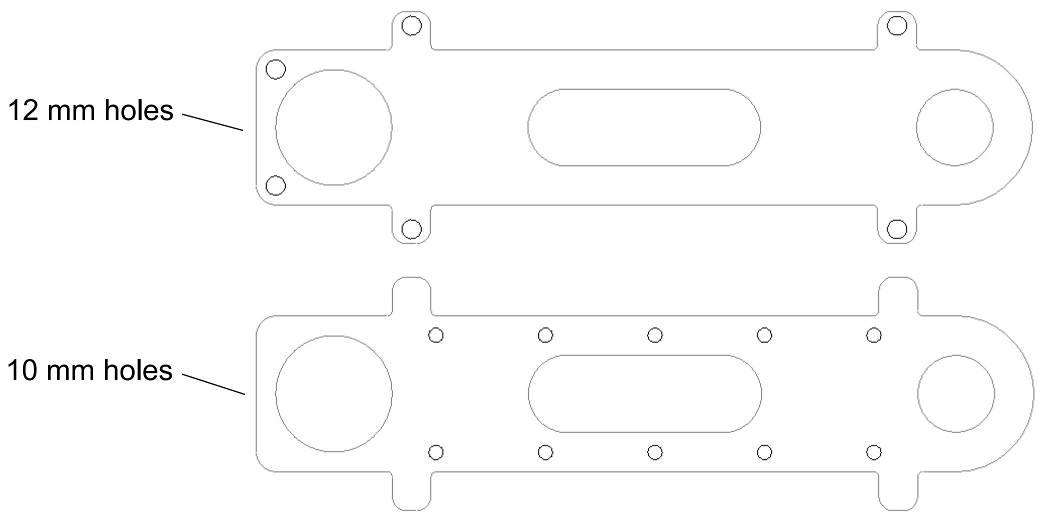


12. Press [Alt + A] to save the part.

Exercise 4 – Using Auto drill to create multiple drilling operations

Frequently you need to combine several different drilling operations on the same set of holes; for example, spot drilling, pre-drilling, drilling, and tapping. In previous chapters, you created multiple operations on the same geometry by copying one operation and editing its parameters. For drilling, Mastercam includes an even more powerful feature called Auto drilling. It lets you create a complete series of drilling operations from within a single dialog box.

In this exercise, you will use Auto drilling twice to drill two sets of holes. First, you will drill the 12 mm through holes on the part corners and tabs. Then, you will drill and tap the two rows of 10 mm holes. The holes are identified in the following picture.



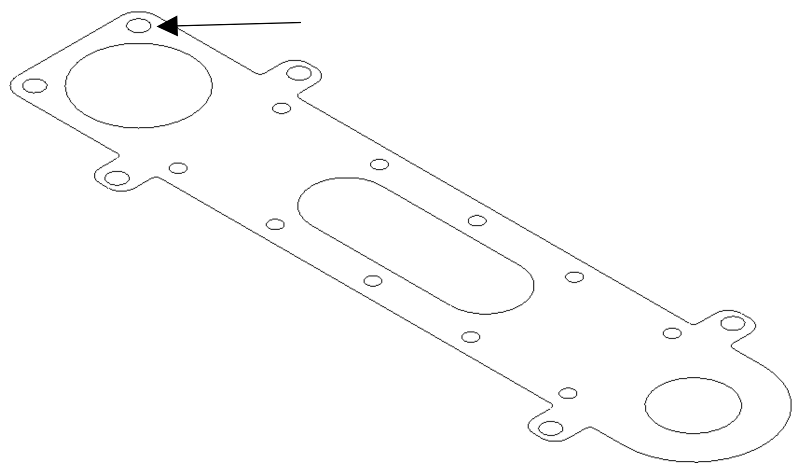
You will learn the following skill:

- **Using Auto drill to create spot drilling, pre-drilling, drilling, tapping, and chamfering operations**

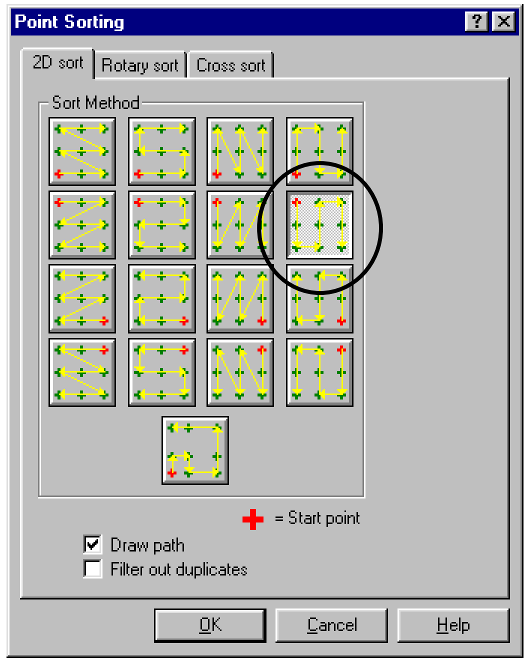
Selecting the holes for the first operations

Your first drilling operations will be on the 12 mm holes.

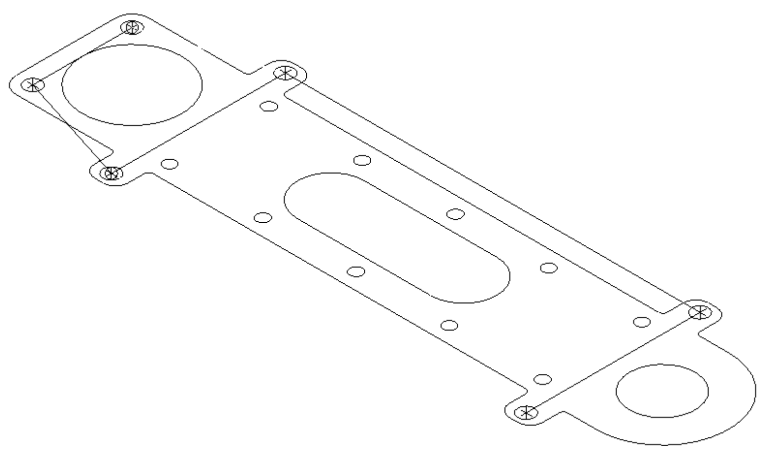
1. Choose **Main Menu, Toolpaths, Next menu, Circ tlpths, Auto drill**.
2. Choose **Mask on arc**. This lets you select an arc and have Mastercam automatically choose arcs which match it.
3. Select the 12 mm arc shown in the following picture.



4. Press [Enter] to confirm the default tolerance value of **0.001**. Mastercam uses this tolerance value to decide which arcs are the same size as the one you selected.
5. Choose **Window**.
6. Toggle **Use mask = Y**.
7. Click and drag a window around the whole part to select all the holes on the part and click again.
8. Choose **Done**. Mastercam highlights the six 12 mm arcs and shows you a default drill path.
9. Choose **Options** so you can select a more efficient drilling pattern.
10. Choose the drilling pattern shown in the following picture.



11. Choose **OK**. Mastercam should sort the holes as shown in the following picture.

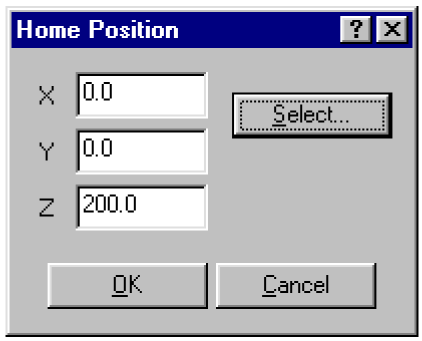


12. Choose **Done**. The Automatic Arc Drilling dialog box displays.

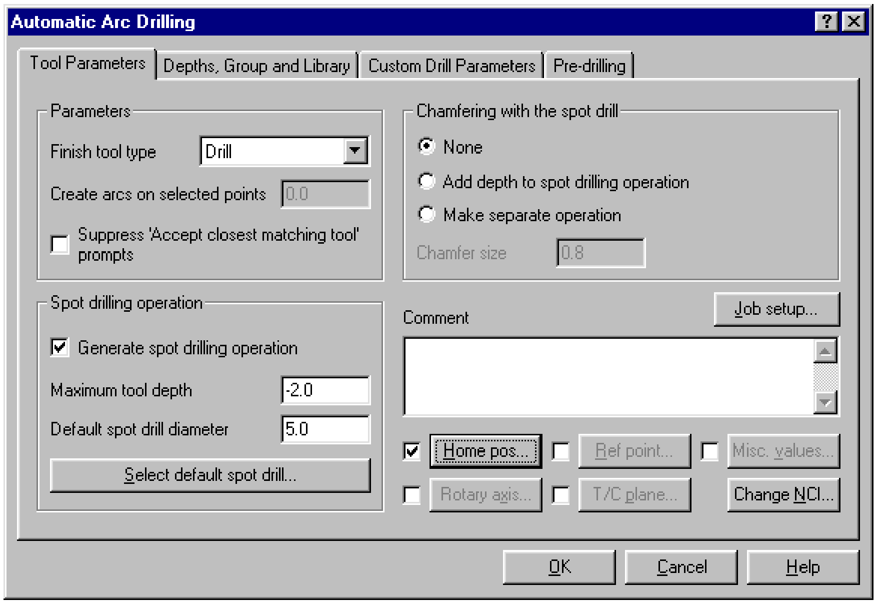
▶ *Setting the Auto drill parameters*

For these holes, you want to include a spot drill operation, then pre-drill in 3 mm increments. No chamfering or threading is required for these holes.

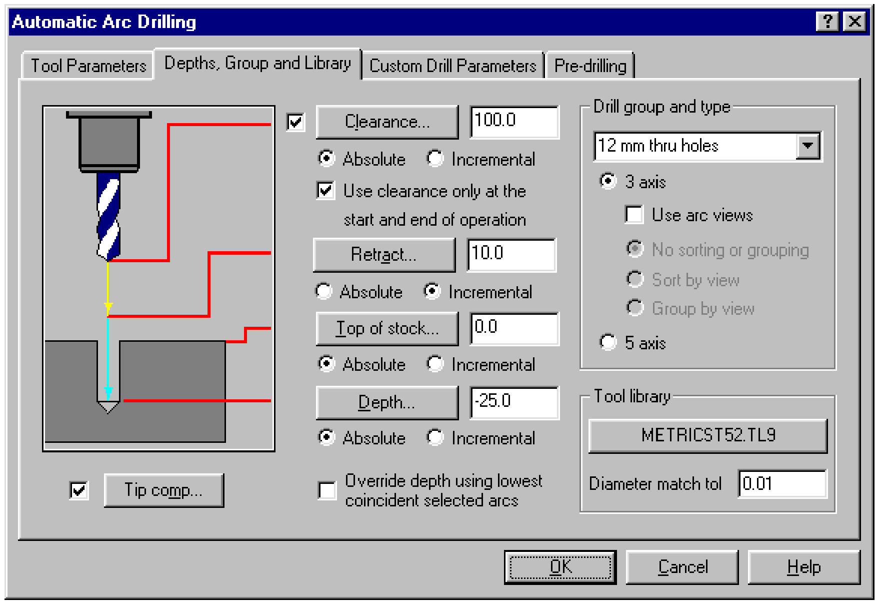
1. Select a **Finish tool type** of **Drill**.
2. Select **Generate spot drilling operation**.
3. In the **Spot drilling operation** section, enter a **Maximum tool depth** of **–2**.
4. Enter **5** for the **Default spot drill diameter**.
5. Select the **Home pos** check box and button.
6. Enter a **Z** coordinate of **200** for the home position as shown in the following picture.



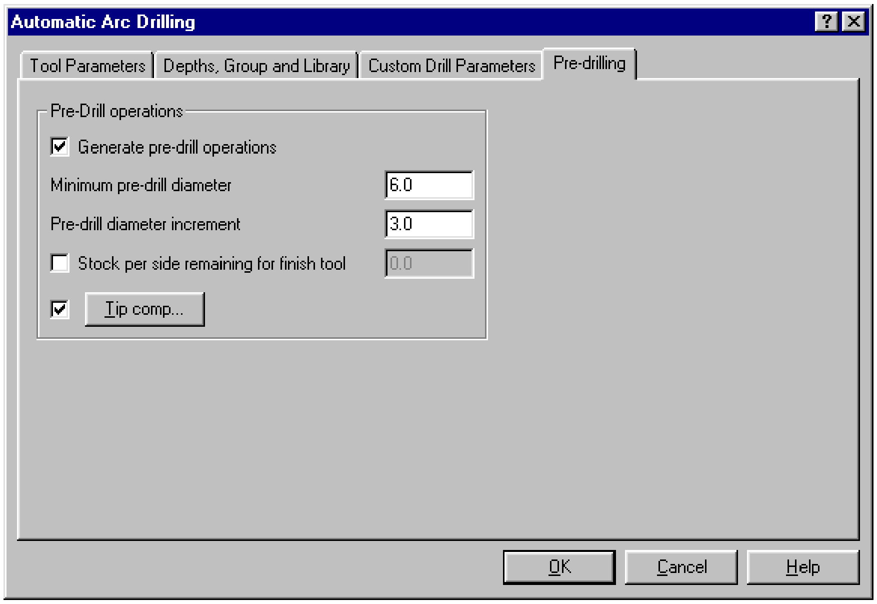
7. Choose **OK**.
8. Verify that your tool parameters match the following picture.



9. Choose the **Depths, Group, and Library** tab.
10. In the **Drill group and type** field, enter the following name for the drilling operations: **12 mm thru holes**
11. Clear the **Use arc views** check box.
12. Enter a **Depth** of **–25**.
13. Clear the **Override depth using lowest coincident selected arc** check box.
14. Choose the **Tip comp** check box and button.
15. Enter a **Breakthrough amount** of **2**.
16. Choose **OK**. Make sure your parameters match the following picture.



17. Choose the **Pre-drilling** tab.
18. Choose **Generate pre-drill operations**.
19. Enter **6** for the **Minimum pre-drill diameter**.
20. Enter **3** for the **Pre-drill diameter increment**. Your parameters should match the following picture.



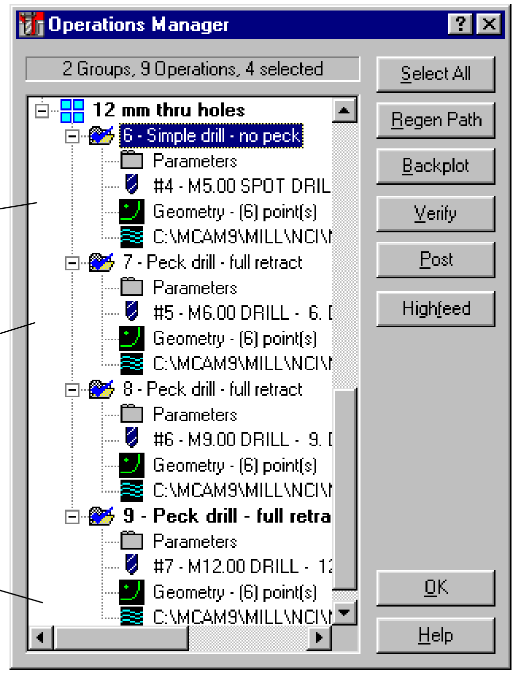
21. Choose **OK** to generate the drill operations.
22. Press [Alt + O] to return to the Operations Manager.
23. Click and drag the lower corner of the window to make it bigger, if necessary. You can see the list of new operations that were created. Mastercam automatically creates a new toolpath group to contain them all.

Toolpath Group name
You entered this name on the **Depths, Group and Library** tab.

Spot drill operation

Pre-drill operations
Mastercam creates a separate operation (operations 7–8) for each pre-drill cycle.

Finish drill operation
This is determined by the **Finish tool type** on the Tool Parameters tab.



Tip: You can edit any individual operation to fine-tune or customize it. Choose the **Parameters** icon for an individual operation to change the drilling parameters or select a different drill, or choose the **Geometry** icon to add or delete holes.

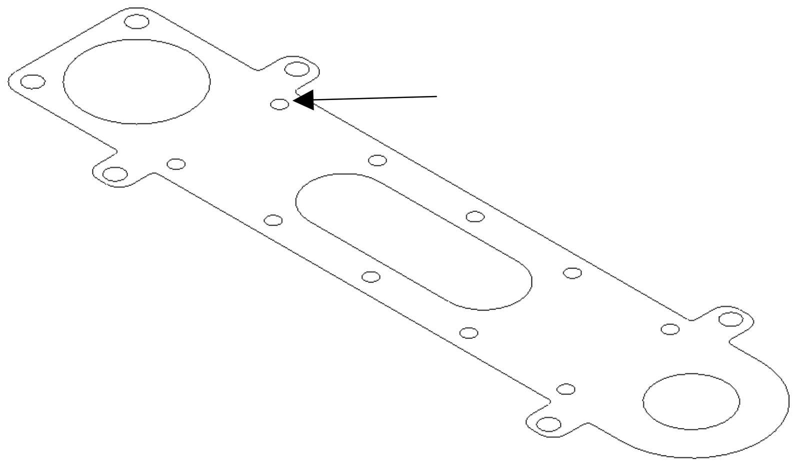
24. Choose **OK** to close the Operations Manager.

▶ *Selecting the holes for the second set of operations*

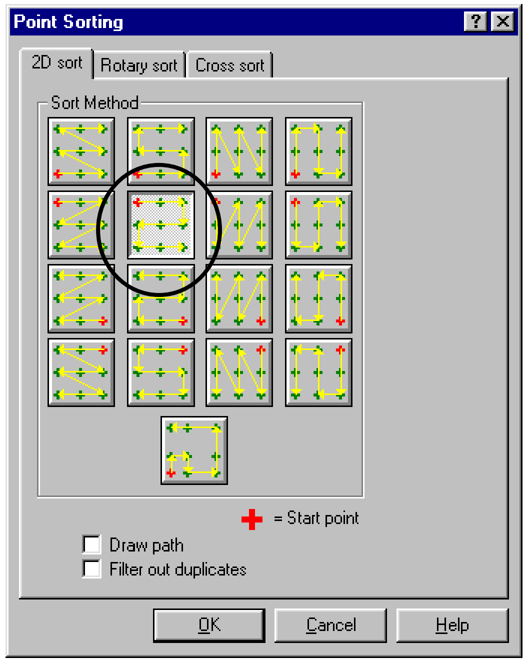
Your second set of drilling operations will be on the 10 mm holes.

1. Choose **Main Menu, Toolpaths, Next menu, Circ tlpths, Auto drill, Mask on arc**.

2. Select the 10 mm arc shown in the following picture.



3. Press [Enter] to confirm the default tolerance value.
4. Choose **Window**.
5. Click and drag a window around the whole part and click again.
6. Choose **Done**.
7. Choose **Options**.
8. Choose the drilling pattern shown in the following picture.



9. Choose **OK**.
10. Choose **Done**. The Automatic Arc Drilling dialog box displays.

▶ *Setting the parameters for the second Auto drill operations*

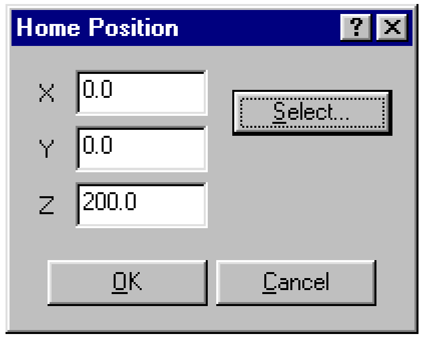
For these holes, you want to include the following drilling cycles:

- spot drill
- pre-drill in 2 mm increments
- thread with a right-hand tap
- chamfer

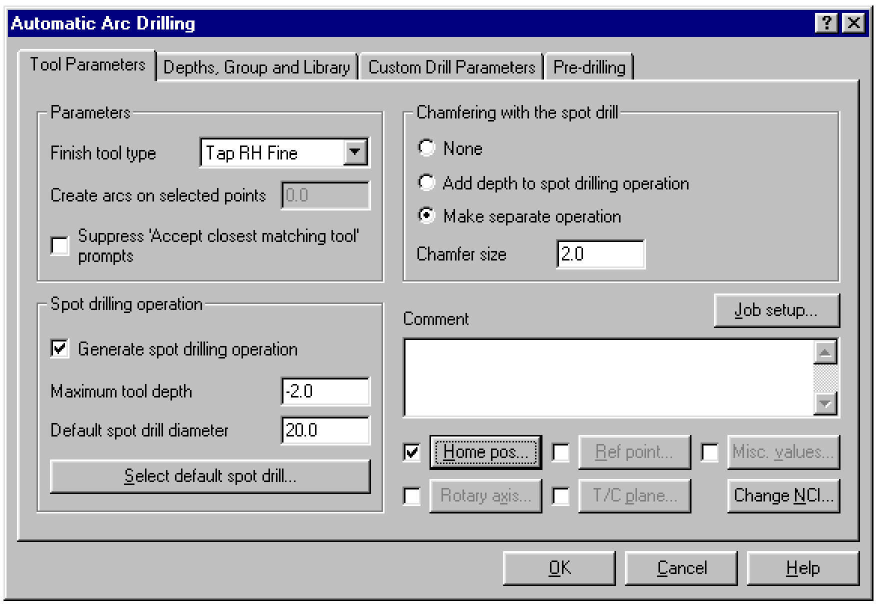
The different parameters are pre-set with the values from your previous Auto drilling session, so you only need to enter the changes.

1. From the **Finish tool type** drop list, choose **Tap RH Fine**.
2. Enter a **Default spot drill diameter** of **20**.
3. In the **Chamfering with the spot drill** section, choose **Make separate operation**.
4. Enter a **Chamfer size** of **2**.
5. Select the **Home pos** check box and button.

6. Enter a **Z** coordinate of **200** for the home position as shown in the following picture.

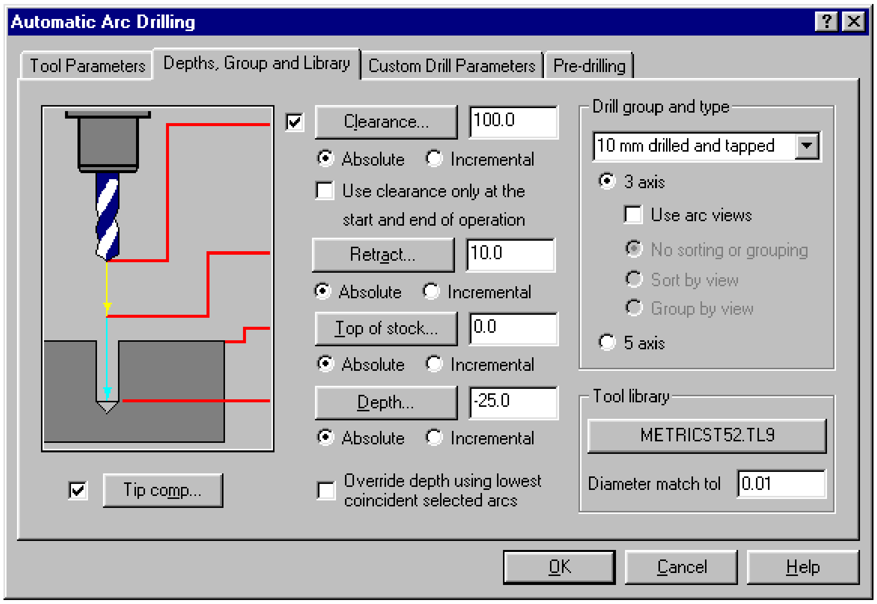


7. Choose **OK**. Your settings should match the following picture.

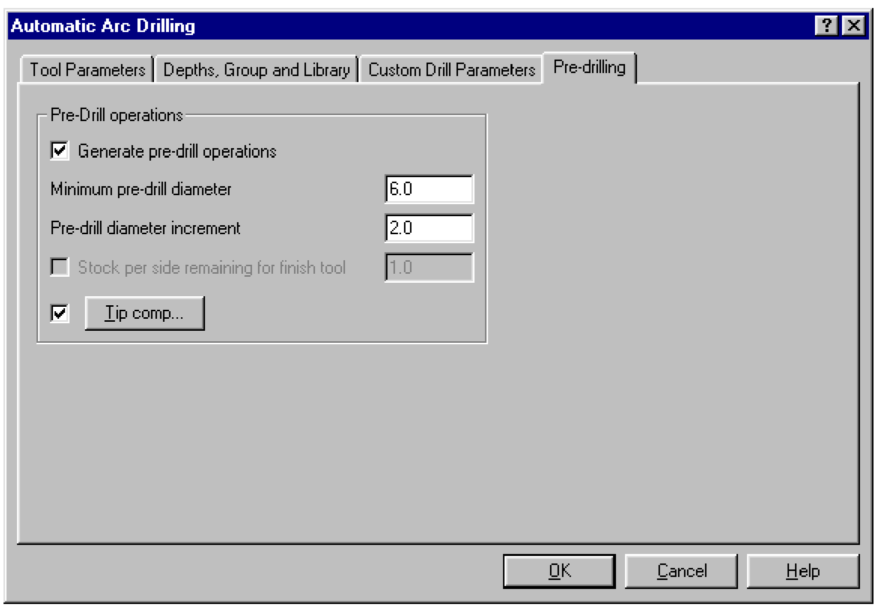


8. Choose the **Depths, Group, and Library** tab.
9. In the **Drill group and type** field, enter the following name for the drilling operations: **10 mm drilled and tapped**

   Make sure your parameters match the following picture.

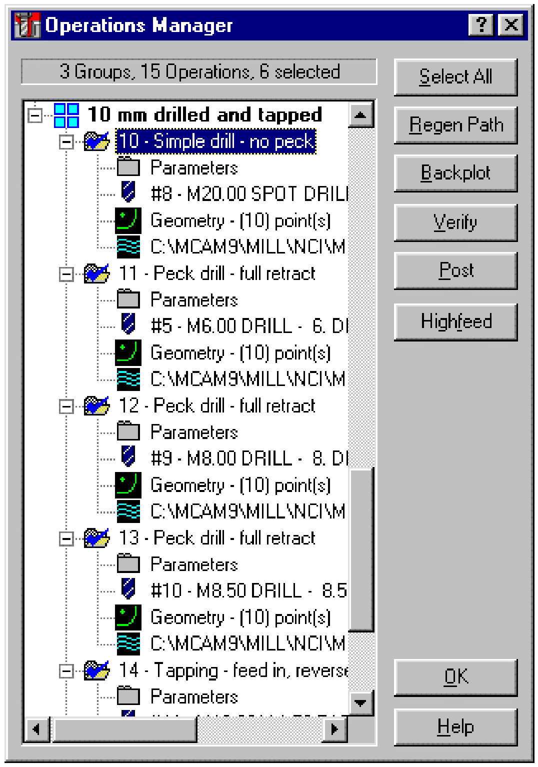


10. Choose the **Pre-drilling** tab.
11. Enter **2** for the **Pre-drill diameter increment**. Verify that your parameters match the following picture.



12. Choose **OK** to generate the drill operations.

13. Press [Alt + O] to return to the Operations Manager.
14. Review the list of new operations that were created.



15. Choose **OK** to close the Operations Manager.
16. Press [Alt + A] to save the part.

You've now seen a variety of applications for Mastercam's circle toolpaths. The next chapter will introduce you to more general purpose pocketing toolpaths and related functions.

9 Facing and Pocketing Toolpaths

This chapter shows you how to create a toolpath for facing your stock and introduces some basic concepts for creating pocket toolpaths:

- Comparing different pocket cutting methods
- Using an entry point to start a pocket toolpath
- Using a contour ramp toolpath

Exercise 1 – Facing the stock with high-speed loops

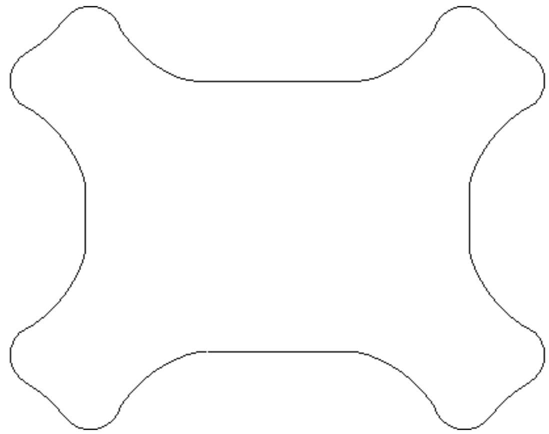
A facing toolpath quickly cleans the stock from the top of a part and creates an even surface for future operations. When facing the stock, it is important to have the tool overlap the edges of the part by at least 50% of its diameter to prevent leaving little scallops of material at the edges of the stock. In this exercise, you will learn the following skills:

- **Using Job Setup to define a stock boundary**
- **Creating a facing toolpath**

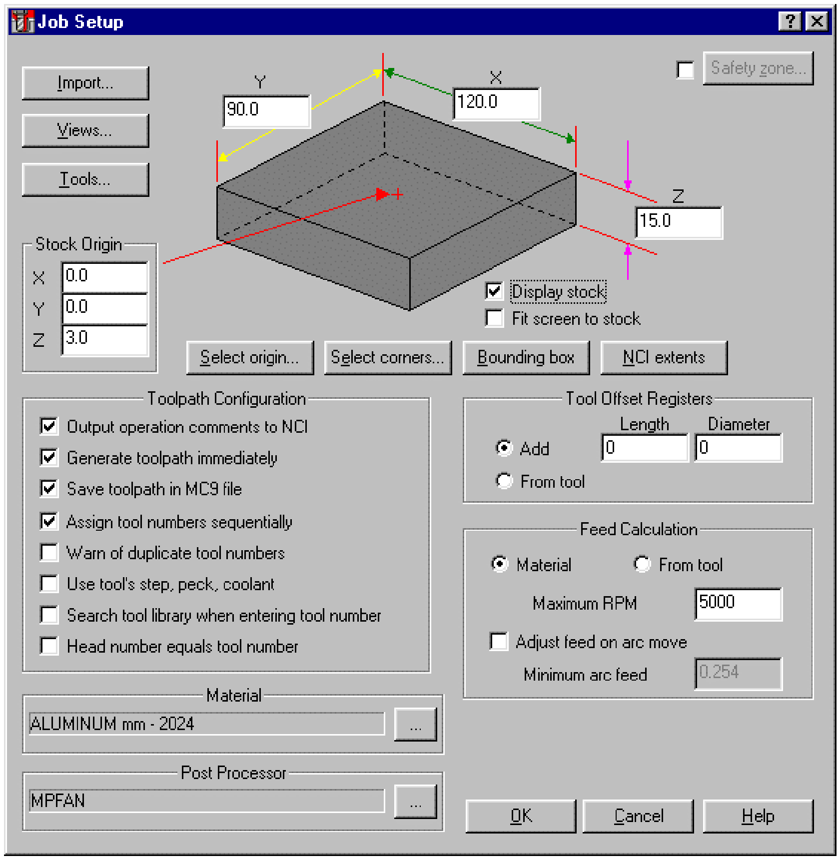
Entering stock boundaries

Assume that your stock is a 90 mm x 120 mm block. You want to face it so that the final thickness will be 12 mm.

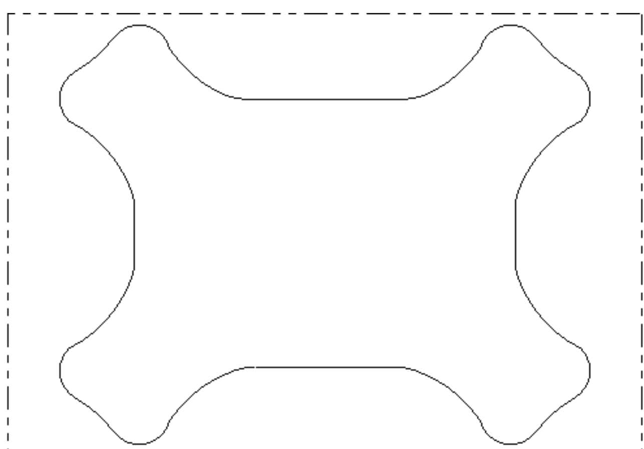
1. Open the file **facing-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.** It should look like the following picture.



2. Choose **Main Menu, Toolpaths, Job Setup**.
3. Enter **90** for the **Y** dimension of the stock model.
4. Enter **120** for the **X** dimension.
5. Enter **15** for the **Z** dimension. Since the goal is to finish with a 12 mm block, you will use 15 mm as a starting dimension and face off 3 mm of stock.
6. Enter **3** for **Stock Origin–Z**. By setting the top of the rough stock at Z = 3, you ensure that when the facing toolpath is finished, the top of the qualified stock will be Z = 0.
7. Select **Display stock**. Your other values should match the following picture.



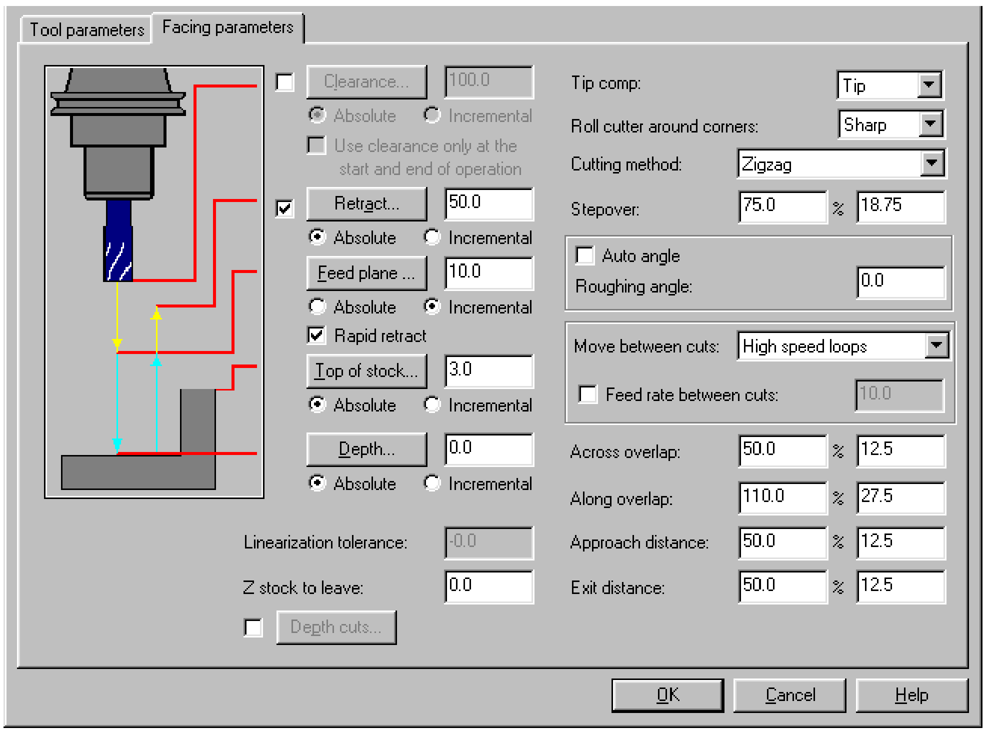
8. Choose **OK**. You will see a dotted red line around your part, representing the stock boundary.



▶ *Creating the facing toolpath*

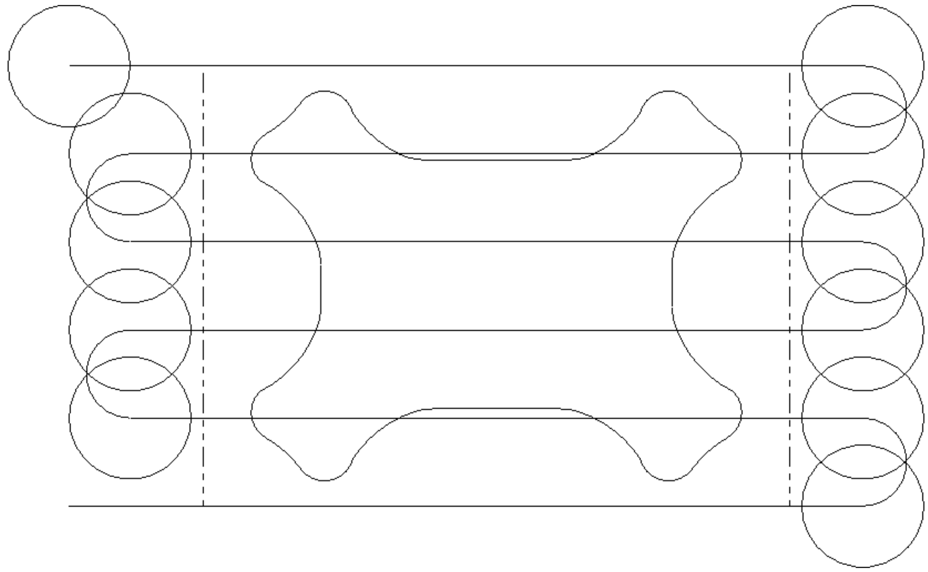
1. Choose **Face, Done**.
2. Right-click in the tool display area and choose **Get tool from library**.
3. Select a 25 mm HSS flat endmill and choose **OK**.
4. Choose the **Facing parameters** tab.

   For this exercise, you will use the default values shown in the following picture. Mastercam has already read the Top of stock value of 3 from Job Setup.



*Note: The **Move between cuts** field is set to High speed loops to provide smooth movement between passes. This type of motion reduces wear on the tool.*

5. Choose **OK**. The toolpath should look like the following picture.



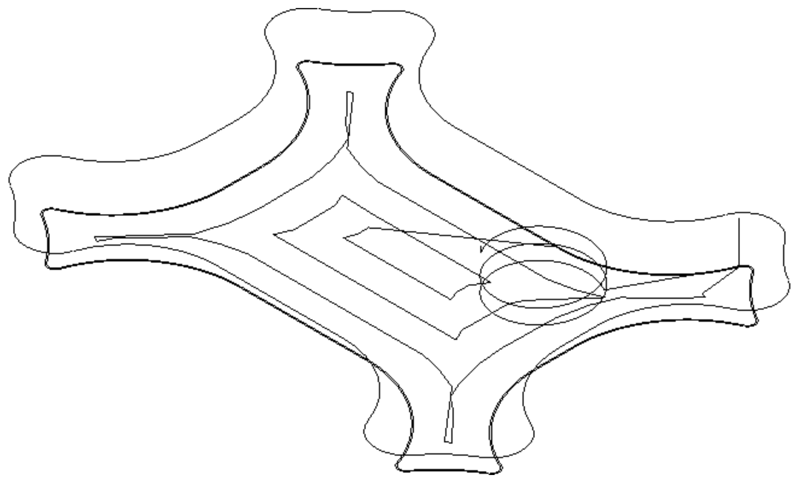
6. Choose **Main Menu, File, Save** and save the part in your working folder as **facing toolpath.mc9**.

Exercise 2 – Comparing different pocket cutting methods

Mastercam offers you seven different pocketing methods. In this exercise, you will compare two of these: the *parallel spiral* and *constant overlap spiral* cutting methods. Each method has advantages:

- The parallel spiral cutting method creates a relatively short NC program but does not guarantee complete cleanout depending on the shape of the pocket and the stepover of each pass.
- The constant overlap spiral cutting method cleans out more stock than parallel spiral because it analyzes the stock after each pass.

The following picture shows the final pocket toolpath for the part used in this exercise.

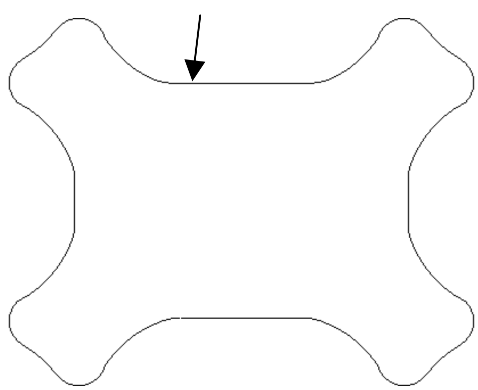


In this exercise, you will learn the following skills:

- **Creating pocket toolpaths**
- **Using the Parallel Spiral cutting method**
- **Using the Constant Overlap Spiral cutting method**
- **Using helical entry**

▶ *Chaining the pocket toolpath and selecting the tool*

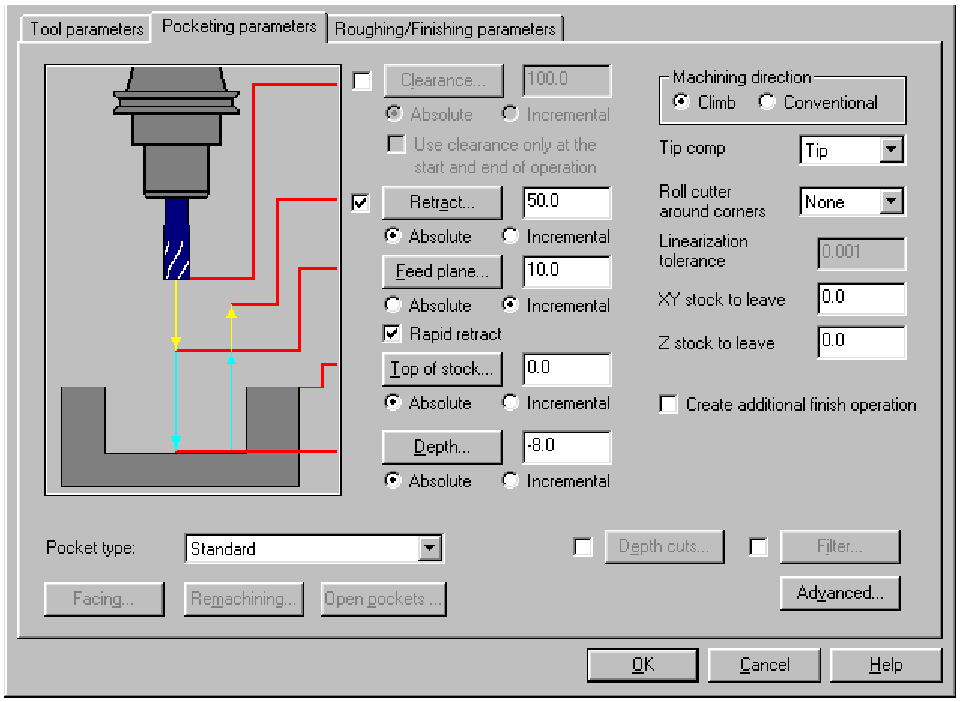
1. Open **pocket-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**
2. Choose **Main Menu, Toolpaths, Pocket**.
3. Select the part at the location shown in the following picture.



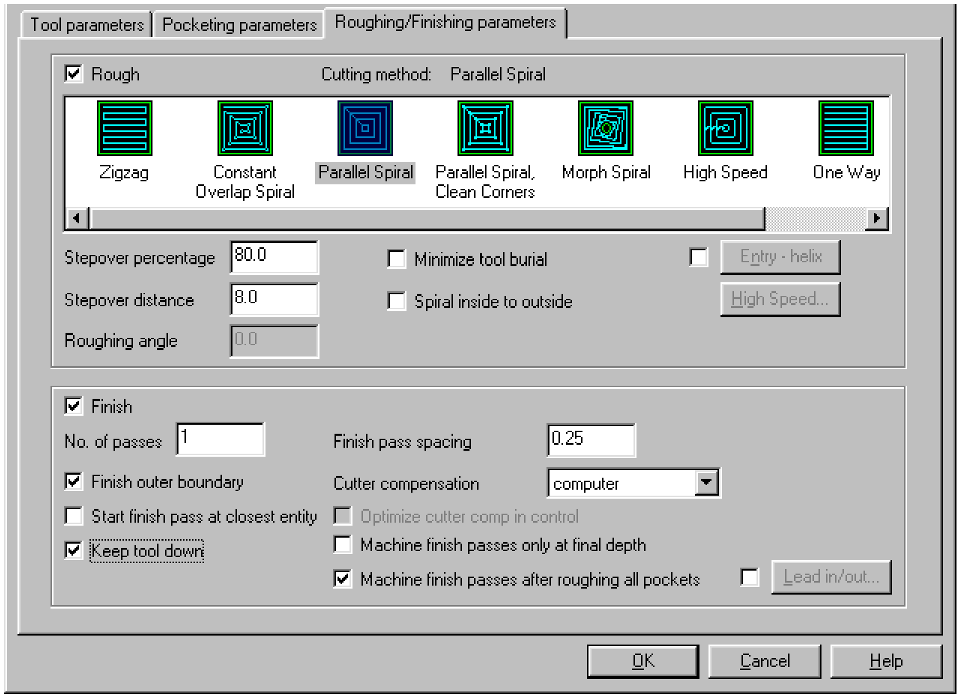
4. Choose **Done**.
5. Right-click in the tool display area.
6. Choose **Get tool from library**.
7. Select the 10mm HSS flat endmill and choose **OK**.

▶ *Entering the toolpath parameters*

1. Select the **Pocketing parameters** tab.
2. Enter the values shown on the following dialog box.



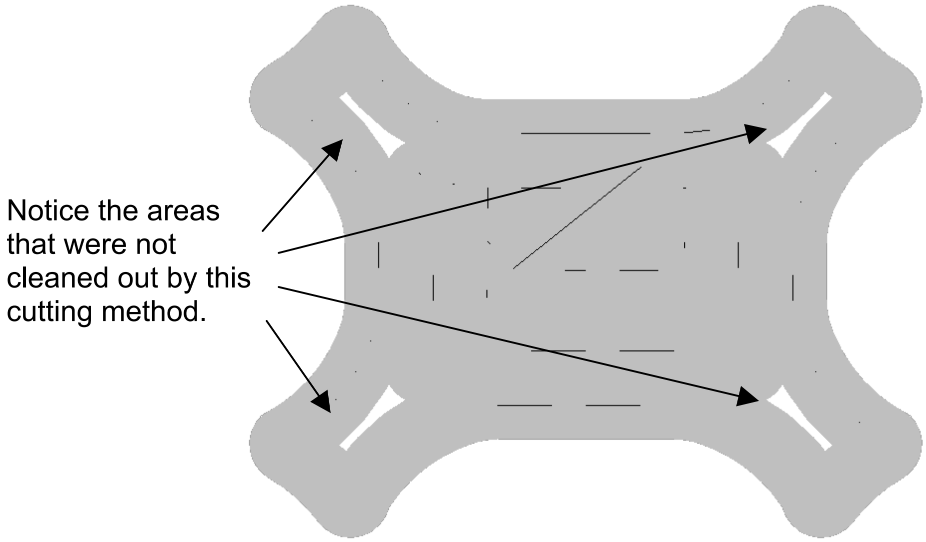
3. Select the **Roughing/Finishing parameters** tab.
4. Enter the values shown on the following dialog box.



5. Choose **OK**.

▶ *Backplotting and verifying the toolpath*

1. Choose **Operations**. The Operations Manager displays.
2. Choose **Backplot**.
3. On the Backplot menu, set the **Verify** option to **Y** (Yes). This will let you see the stock that was cleaned out by the tool.
4. Choose **Run**. The entire backplot is completed at once. The toolpath should look like the following picture.

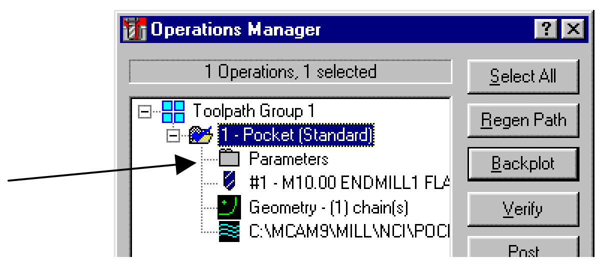


5. Choose **Backup** to return to the Operations Manager.

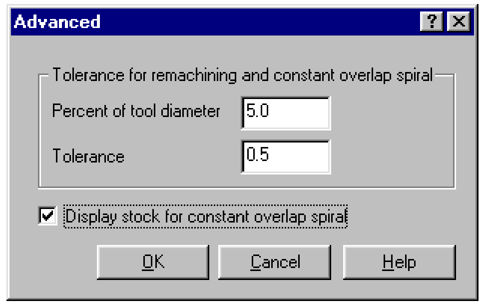
▶ *Changing the cutting method*

Follow these steps to select a different pocket cutting method that will do a better job of cleaning out the pocket.

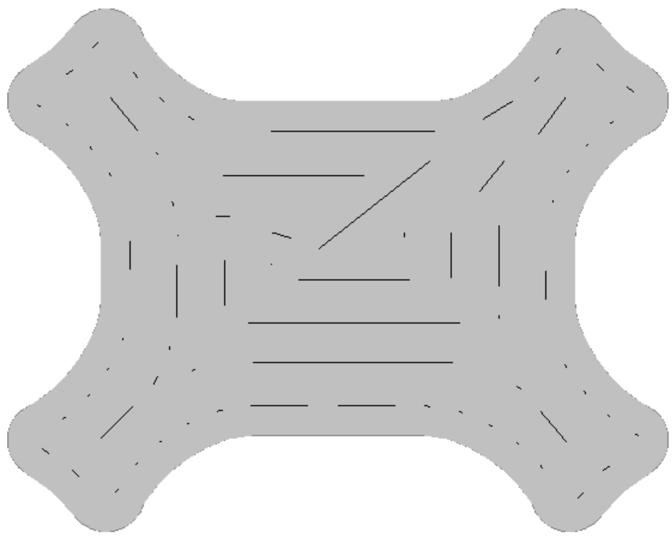
1. Choose the **Parameters** icon for the toolpath.



2. Choose the **Roughing/Finishing parameters** tab.
3. Select the **Constant Overlap Spiral** cutting method.
4. Choose the **Pocketing parameters** tab.
5. Choose **Advanced**.
6. Enter the values shown on the following dialog box.



7. Choose **OK** twice.
8. Choose **Regen Path** to regenerate the modified toolpath.
9. Choose **Backplot**.
10. Choose **Run**. The toolpath should look like the following picture.



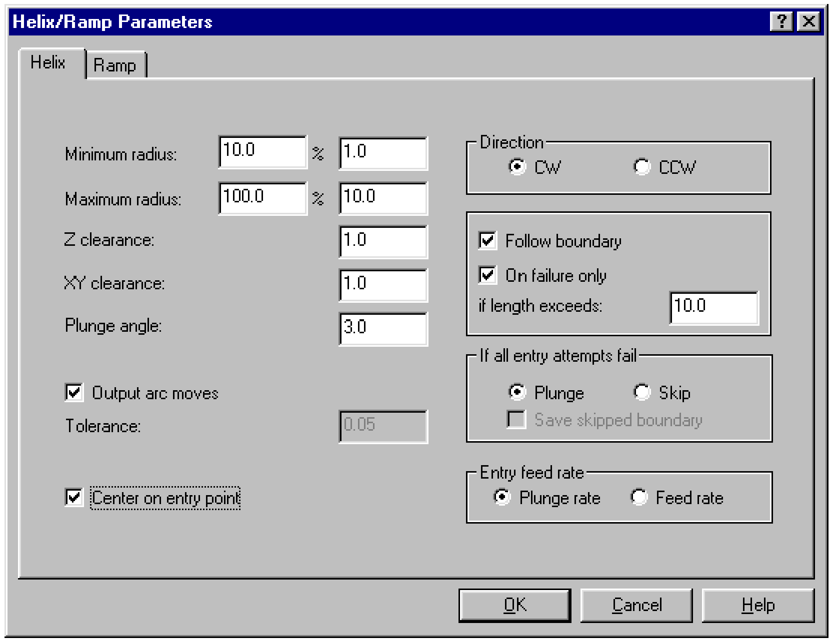
You can see that the new toolpath has completely cleaned out the pocket.

▶ *Adding a helical entry*

Finally, add a helical entry move so the tool enters the pocket more smoothly.

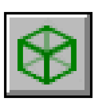
1. Choose **Backup** to return to the Operations Manager.
2. Choose the **Parameters** icon.

3. Choose the **Roughing/Finishing parameters** tab.
4. Select the **Entry - helix** (or **Entry - ramp**) check box and button.
5. Choose the **Helix** tab.
6. Enter the values shown on the following dialog box.

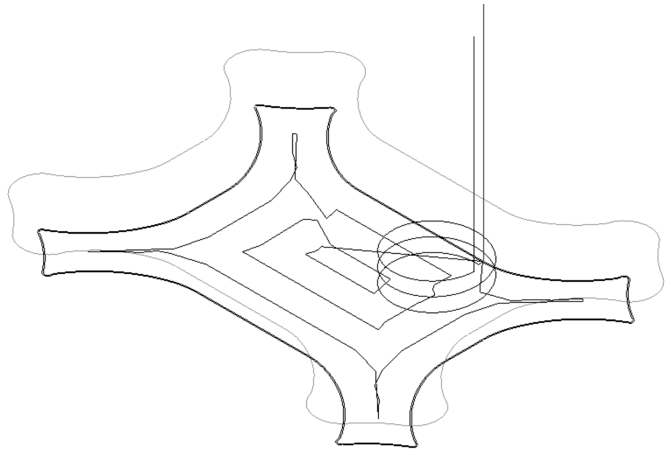


Note: The Z clearance is the distance above the top of stock where the helix starts.

7. Choose **OK** twice.
8. Choose **Regen Path**.
9. Choose **OK** to close the Operations Manager.



10. Choose the green **Gview–Isometric** button from the toolbar to switch into isometric view so you can see the new toolpath more clearly. It should look like the following picture.



11. Save the part in your working folder as **spiral pocket.mc9**.

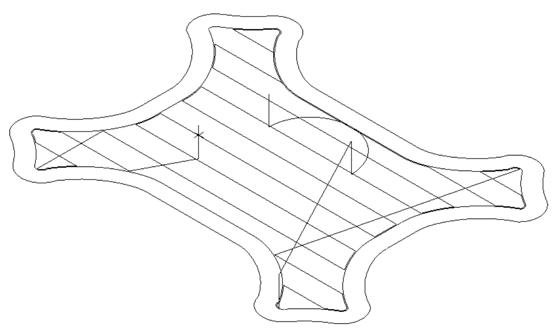
Exercise 3 – Specifying an entry point

Specifying an entry point for a pocket toolpath can be necessary for tools that cannot plunge directly into the material. Instead of drilling a hole in the material at the desired entry point, which would require creating a separate drilling operation, you can add an extra point when chaining the pocket toolpath. Mastercam will automatically use this point as the entry point for the toolpath.

In this exercise, you will learn the following skills:

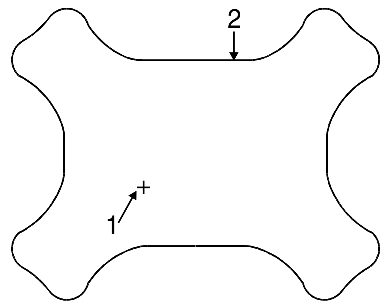
- **Using an entry point in a pocket toolpath**
- **Creating lead in/out moves for a finish pass**

You will create a toolpath that looks like the following picture:



▶ *Chaining the pocket toolpath and selecting the tool*

1. Open **entry point-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**
2. Choose **Main Menu, Toolpaths, Pocket**.
3. Select the point at position 1.
4. Select the part boundary at position 2.
5. Choose **Done**.

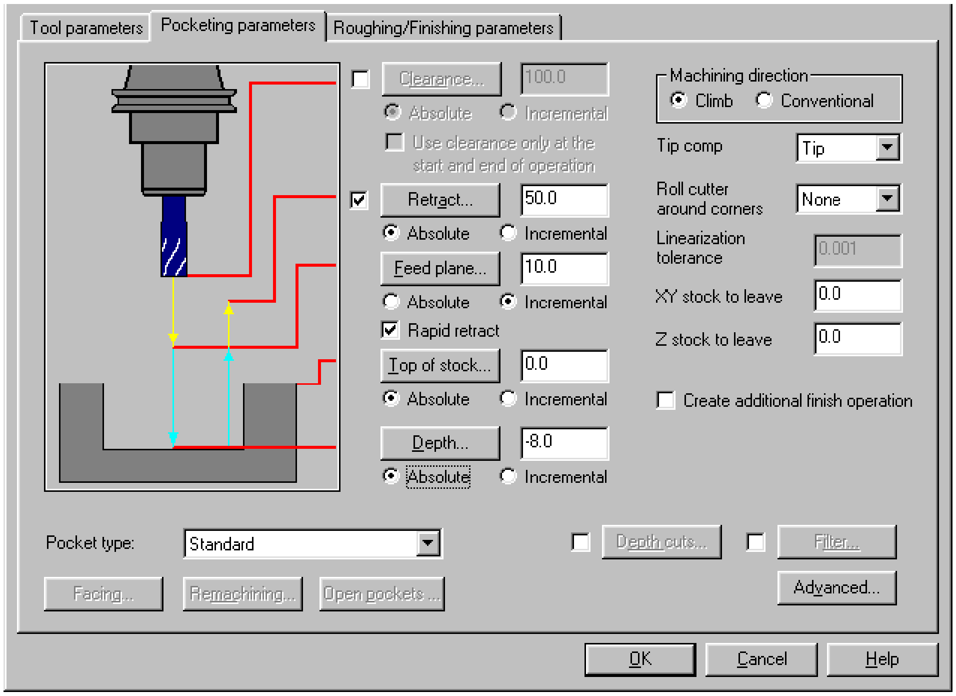


   Note: You could have selected either the point or the boundary first. Mastercam automatically knows that the point should be used as the entry point for the pocket toolpath.

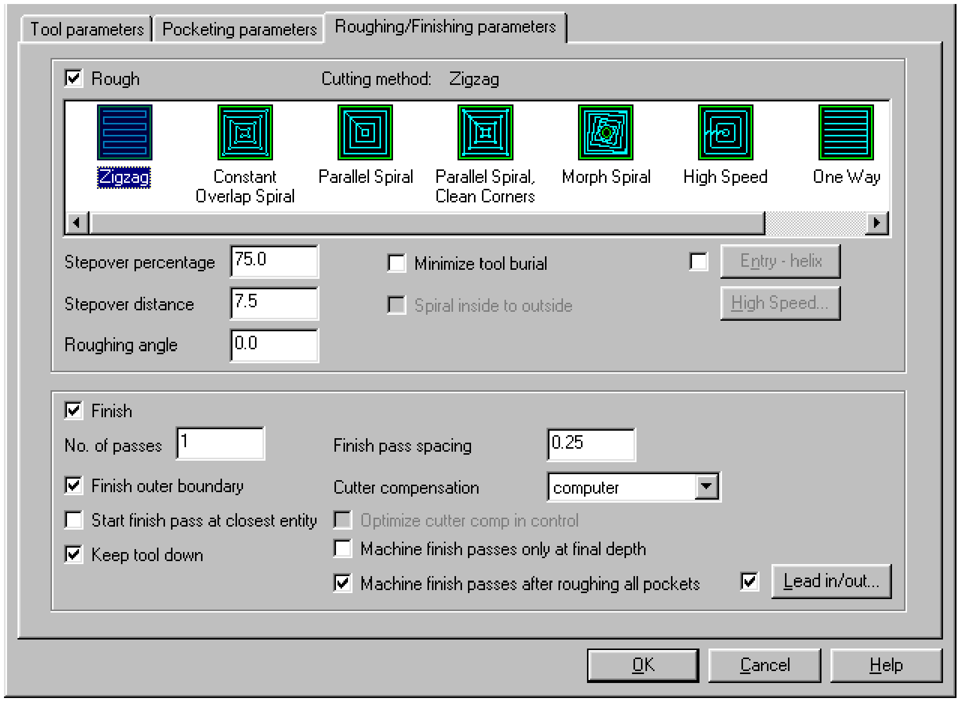
6. Right-click in the tool display area and select the 10 mm HSS flat endmill.

▶ *Entering the toolpath parameters*

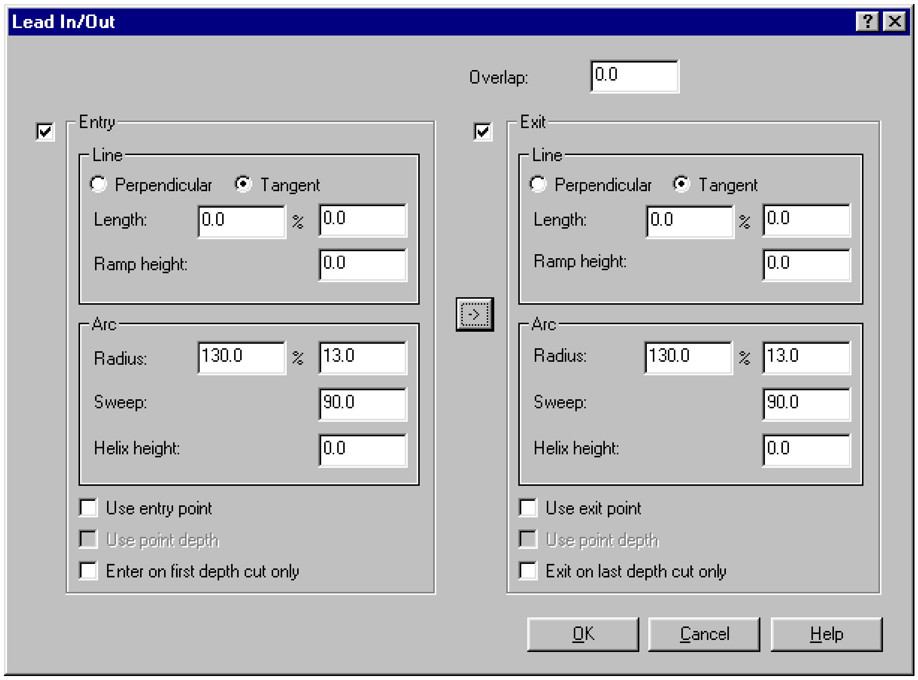
1. Select the **Pocketing parameters** tab.
2. Enter the values shown on the following dialog box.



3. Select the **Roughing/Finishing** parameters tab.
4. Enter the values shown on the following dialog box.



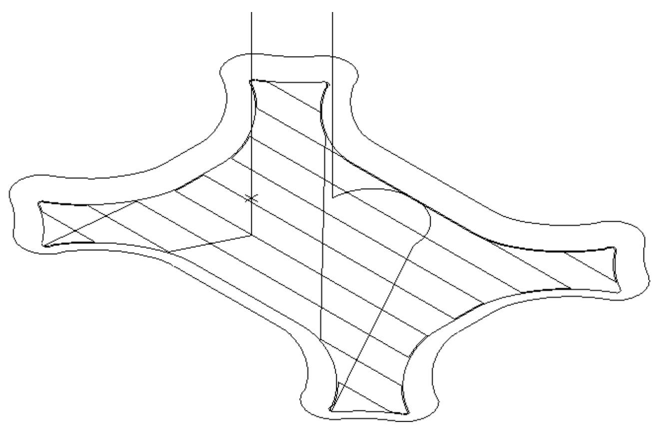
5. Choose the **Lead in/out** check box and button.
6. Enter the values shown on the following dialog box.



Note: Adding a lead in/out to the finish pass prevents a dwell mark on the part at the end of the toolpath.

7. Choose **OK** twice.

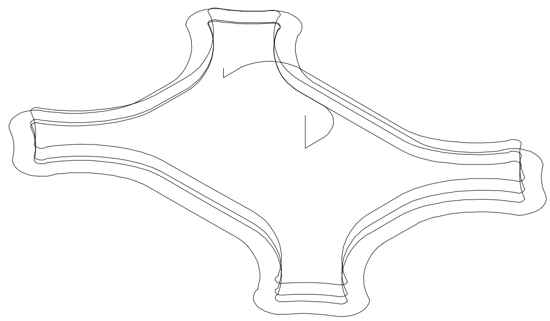
8. Switch to **Gview–Isometric**. The completed toolpath should look like the following picture.



9. Save the part in your working folder as **entry point pocket.mc9**.

Exercise 4 – Using contour ramp

In this exercise, the contour toolpath includes a ramping motion to move between depth cuts. The ramp uses constant stepdown in the Z axis which can be effective for high-speed machining. The following picture shows the completed toolpath. This toolpath is appropriate if the pocket has already been roughed out in a previous operation.

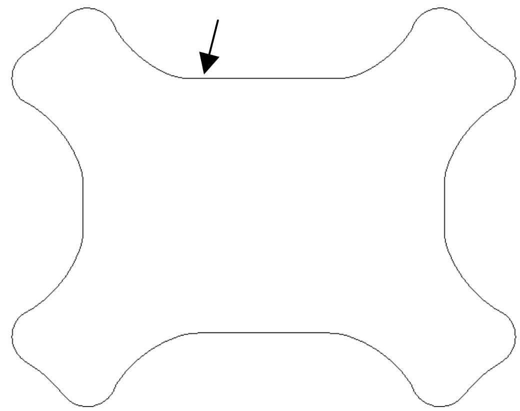


In this exercise, you will learn the following skill:

- **Using ramp contours**

▶ *Chaining the geometry and select the tool*

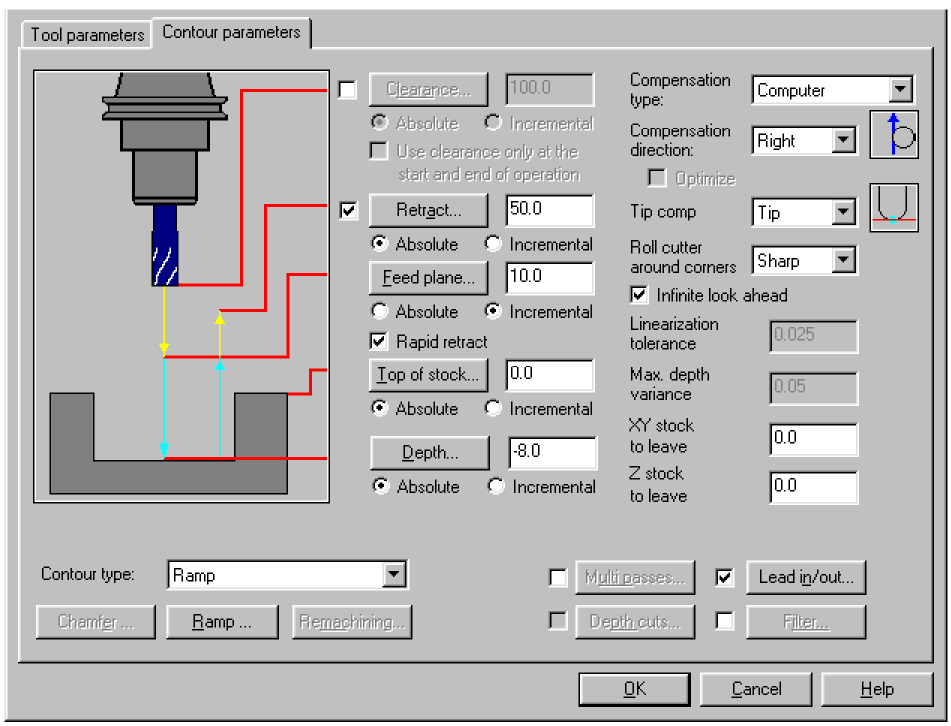
1. Open **contour ramp-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**
2. Choose **Main Menu, Toolpaths, Contour**.
3. Select the part at the location shown in the following picture. The chaining arrow should point clockwise when you are done.



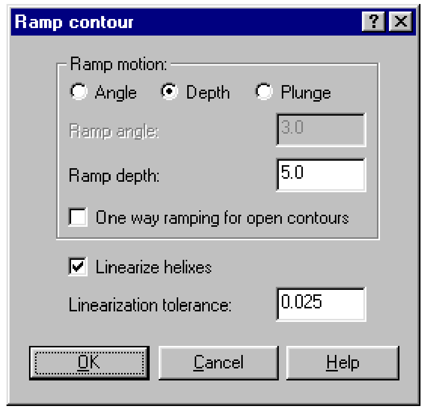
4. Choose **Done**.
5. Select the10 mm HSS flat endmill from the tool library.

▶ *Entering the toolpath parameters*

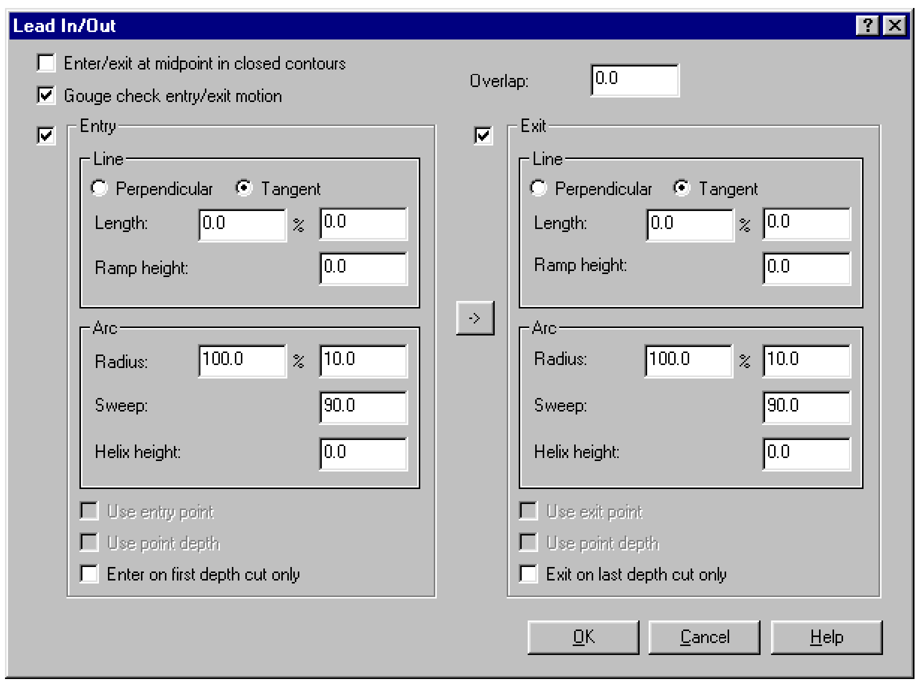
1. Select the **Contour parameters** tab.
2. Enter the values shown on the following dialog box.



3. Choose the **Ramp** button and enter the values shown on the following dialog box.

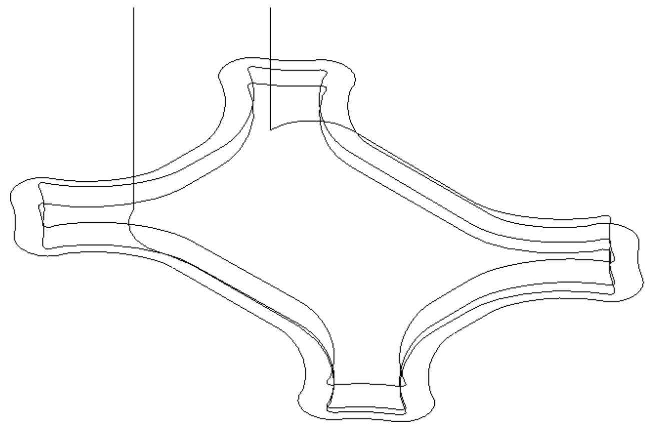


4. Choose **OK**.
5. Choose the **Lead in/out** button and enter the values shown on the following dialog box.



6. Choose **OK** twice. Mastercam generates the toolpath.
7. Choose **Operations** to open the Operations Manager.

8. Choose **Backplot.**
9. Switch to **Gview–Isometric** and set **Verify=N**. Step through the backplot to see the results of the contour ramp. The toolpath should look like the following picture. You can see that there are no direct steps between successive depths. The tool moves continuously in the negative Z direction.



10. Save the part in your working folder as **ramp pocket.mc9**.

You've already seen in this chapter how different pocketing methods can improve efficiency. The next chapter will show you even more techniques for efficient machining and optimal clean out, including remachining and high speed machining.

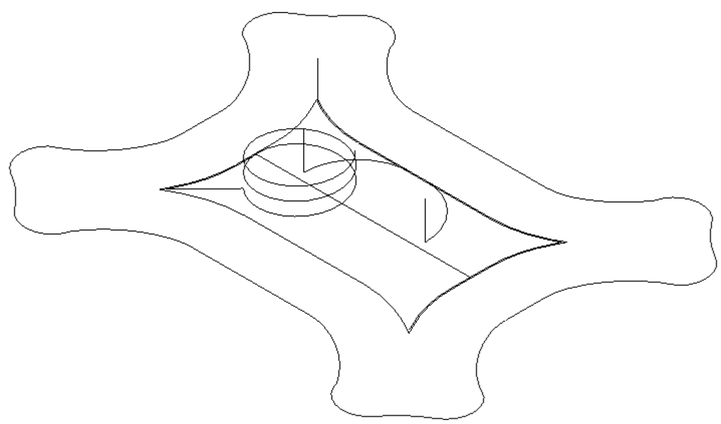
10 Pocket and Contour Toolpath Techniques

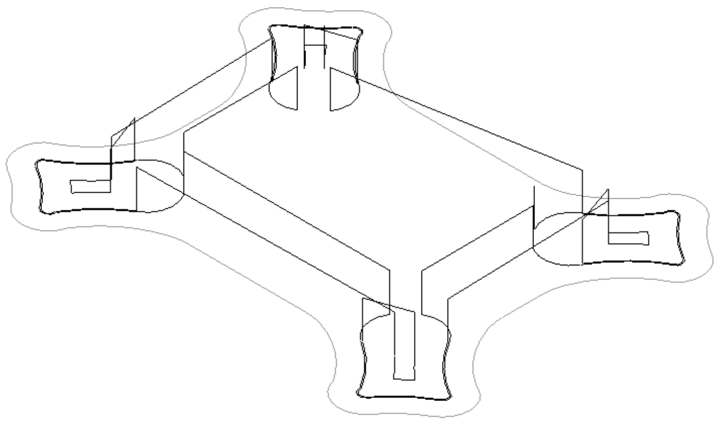
This chapter covers some additional concepts for creating pocket and contour toolpaths.

- Using pocket remachining for more efficient clean out.
- Combining pocket depth cuts, tapered walls, and island facing to remove material from a more complicated part with multiple islands.
- Using the Toolpath Editor to edit a toolpath.

Exercise 1 – Remachining pockets

Using remachining in pocket toolpaths can save time, because you can use a large tool on your roughing toolpath to quickly clean out as much material as possible. You can then create a remachining toolpath to go back into the part with a smaller tool and clean out the areas that the large tool was not able to reach. The following pictures show the rough and remachining toolpaths for this exercise.



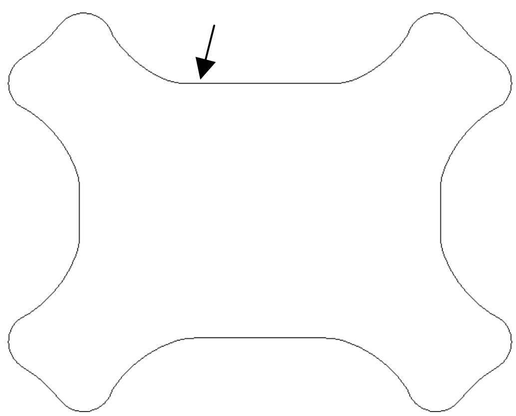


In this exercise, you will learn the following skills:

- **Remachining pockets**
- **Using the helical entry and lead in/out features**

▶ *Chaining the pocket geometry and selecting the tool*

1. Open **remachine-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**
2. Choose **Main Menu, Toolpaths, Pocket**.
3. Select the part at the location shown in the following picture. The chaining arrow should point clockwise when you are done.

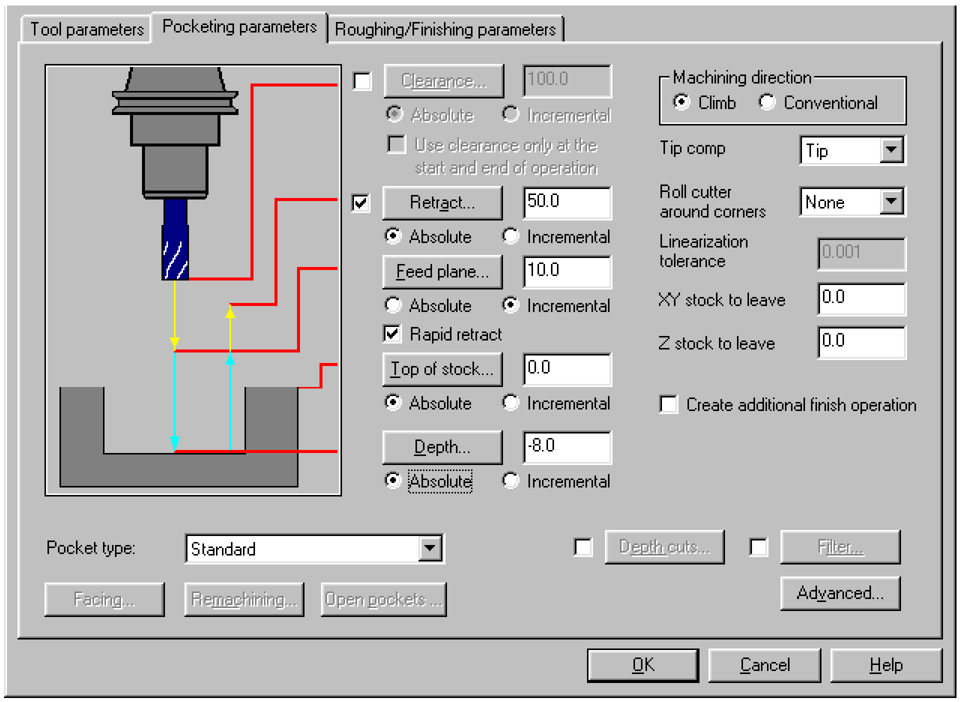


4. Choose **Done**.

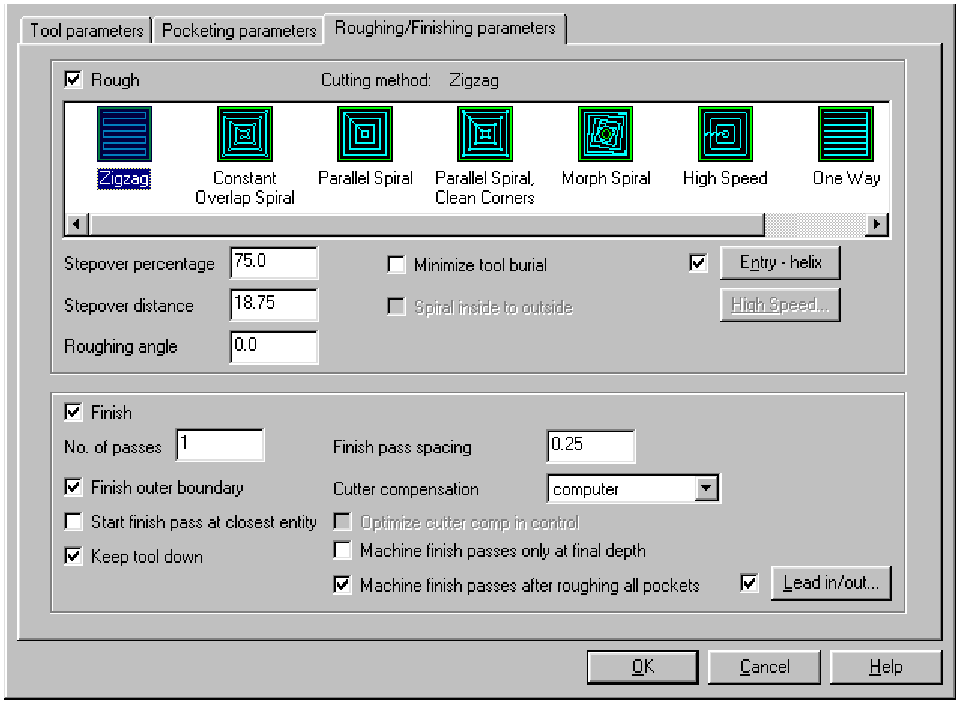
5. Right-click in the tool display area and choose **Get tool from library**.
6. Select the 25 mm HSS flat endmill and choose **OK**.

▶ *Entering the toolpath parameters*

1. Select the **Pocketing parameters** tab.
2. Enter **–8** for the **Depth**. Your other values should match the following picture.

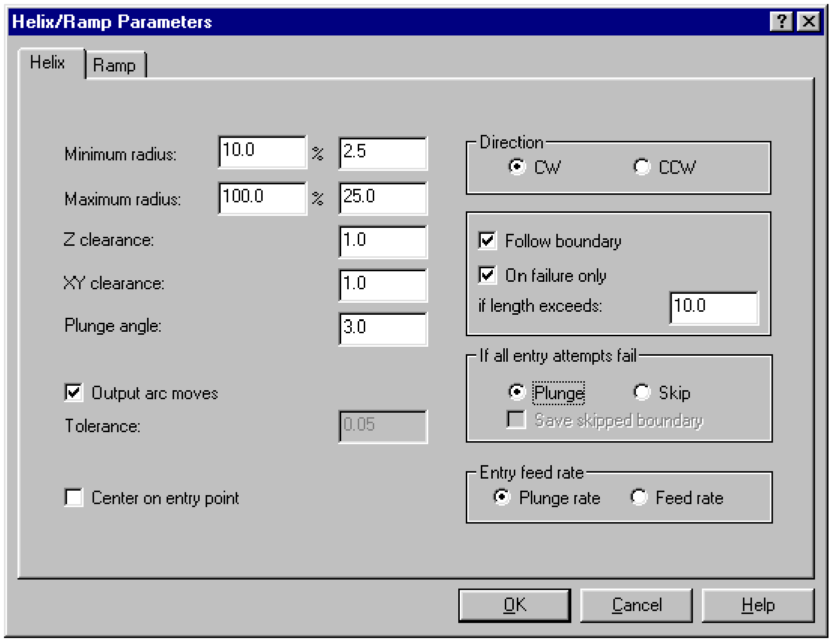


3. Select the **Roughing/Finishing parameters** tab.
4. Make the selections shown on the following dialog box.



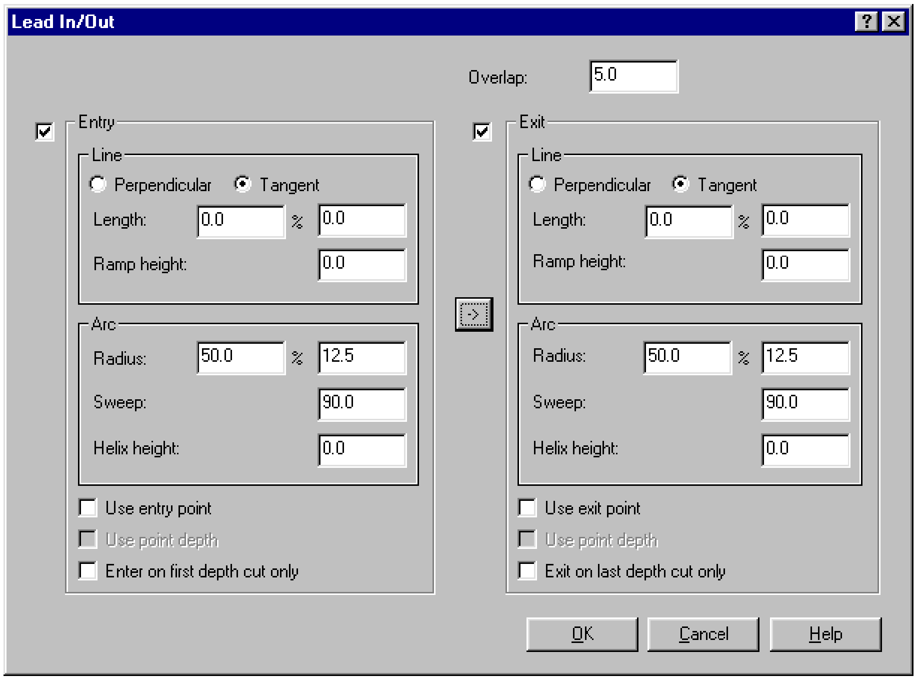
5. Choose the **Entry – ramp** (or **Entry – helix**) button.

6. Choose the **Helix** tab.
7. Enter the values shown on the following dialog box.

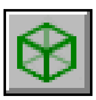


Tip: These options prevent the tool from plunging directly into the material by adding a helix to the start of the pocket toolpath.

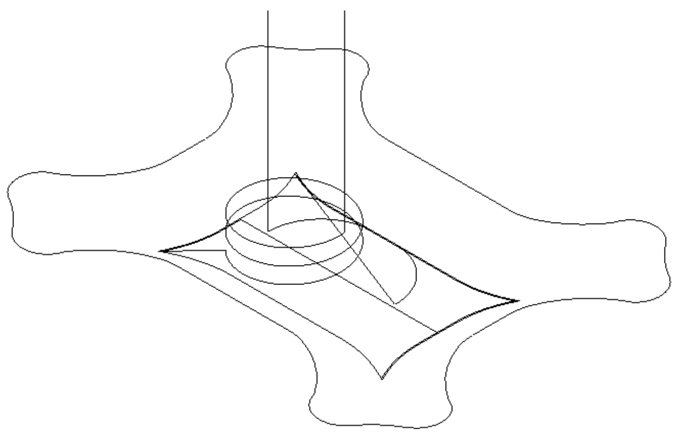
8. Choose **OK**.
9. Choose the **Lead in/out** button.
10. Enter the values shown on the following dialog box.



11. Choose **OK** twice. Mastercam generates the toolpath.

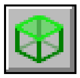


12. Choose the green **Gview–Isometric** button from the toolbar so you can see the new toolpath more clearly. It should look like the following picture.

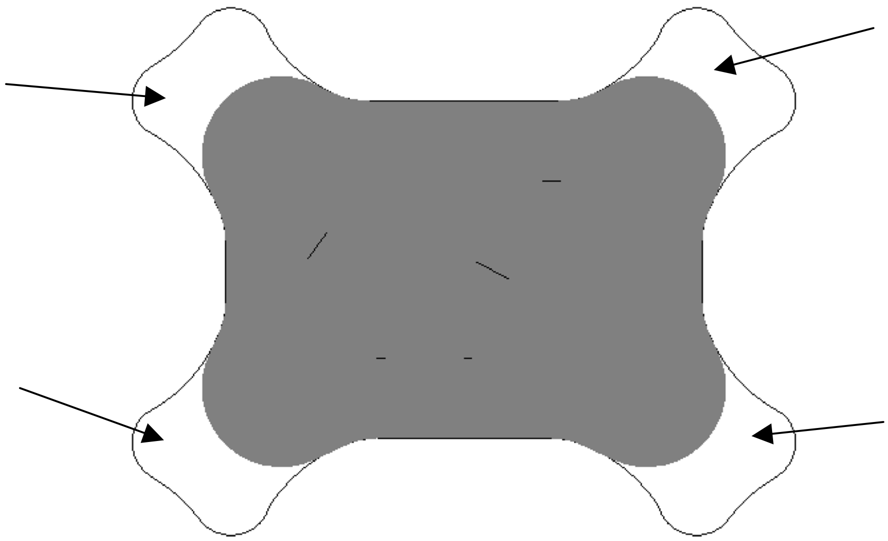


▶ *Backplotting the toolpath*

1. Choose **Operations**.
2. Choose **Backplot**.
3. Toggle the **Verify** option to **Y**.



4. Choose the green **Gview–Top** button to return to the Top view.
5. Press [S] repeatedly to step through the toolpath.
6. Choose **OK** when the backplot is complete. It should look like the following picture.

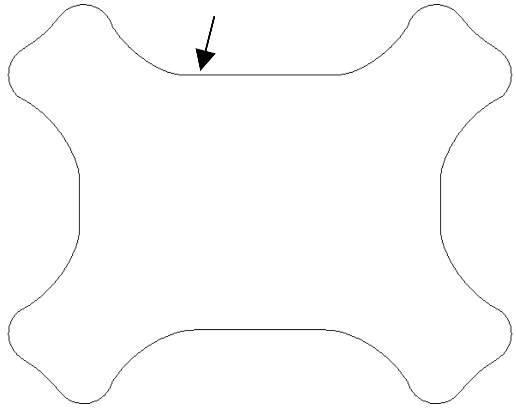


The arrows show you where the tool could not reach into the corners of the pocket. The next procedure shows you how to create a remachining toolpath to clean out the remaining stock.

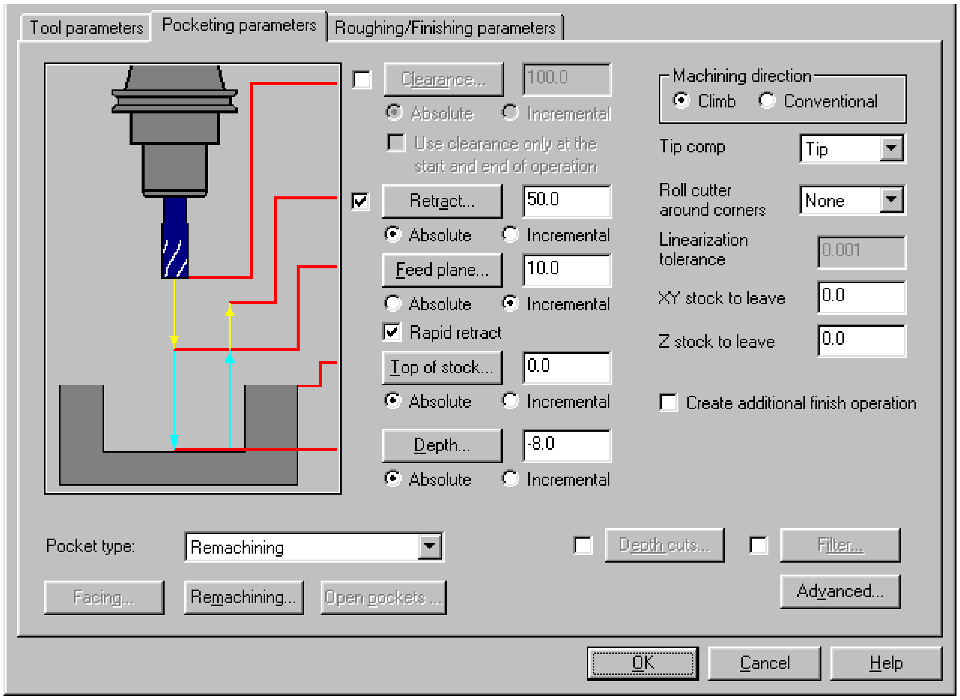
7. Choose **Backup** to return to the Operations Manager.
8. Choose **OK** to close the Operations Manager.
9. Save the file in your working folder as **remachined pocket.mc9**.

▶ *Creating the remachining toolpath*

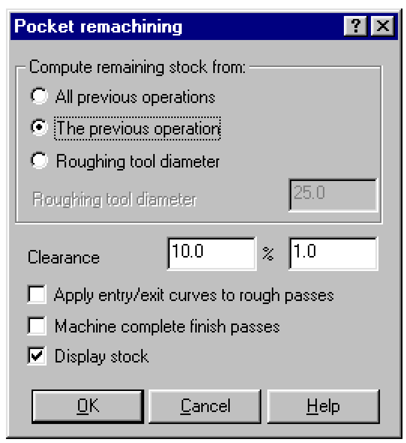
1. Choose **Main Menu, Toolpaths, Pocket**.
2. Press [Alt + T] to clear the toolpath from the screen.
3. Select the pocket boundary as shown in the following picture.



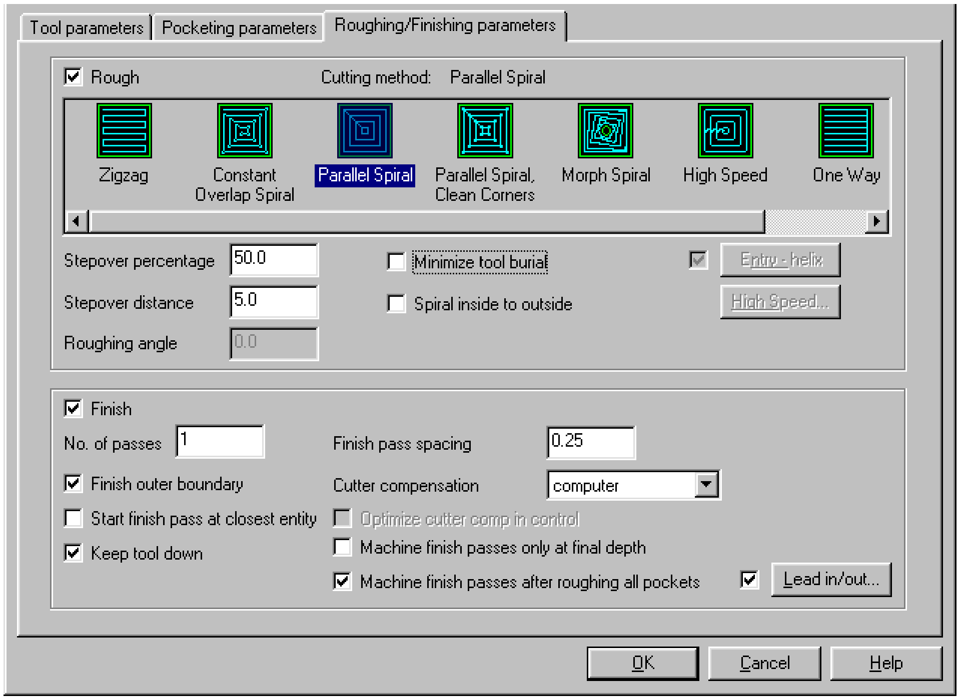
4. Choose **Done**.
5. Select the 10 mm HSS flat endmill from the tool library.
6. Select the **Pocketing parameters** tab.
7. Enter a **Depth** of **–8**.
8. Choose **Remachining** from the **Pocket type** drop-down list. The other values should match the following picture.



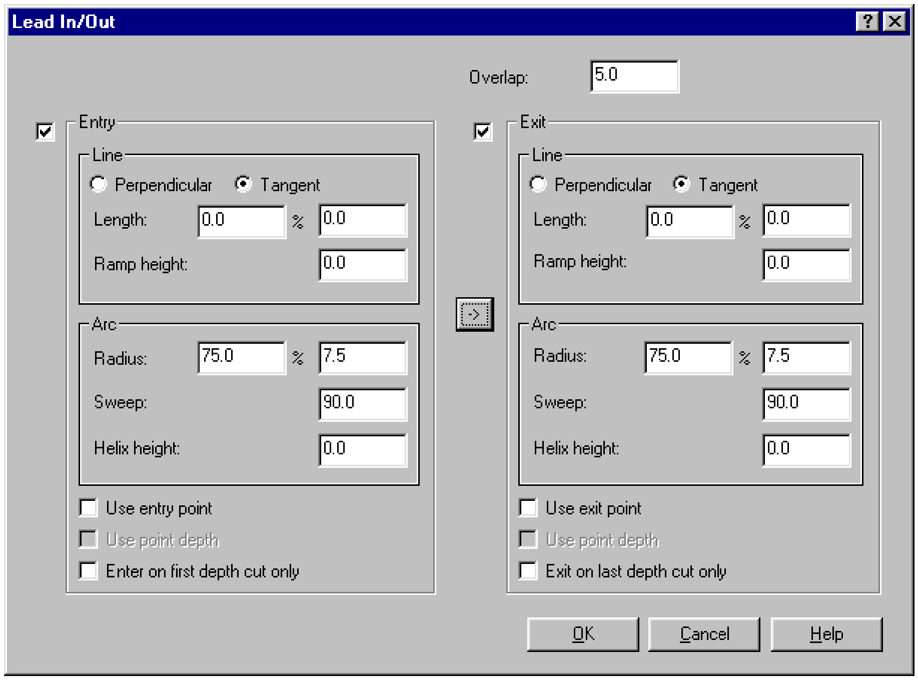
9. Choose the **Remachining** button.
10. Enter the values shown on the following dialog box.



11. Choose **OK**.
12. Select the **Roughing/Finishing** parameters tab.
13. Enter the values shown on the following dialog box.

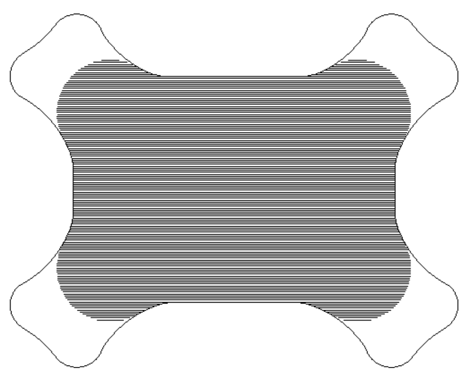


14. Choose the **Lead in/out** button.
15. Change the **Entry** and **Exit Arc–Radius** to **75%**. Your values should match the following dialog box.

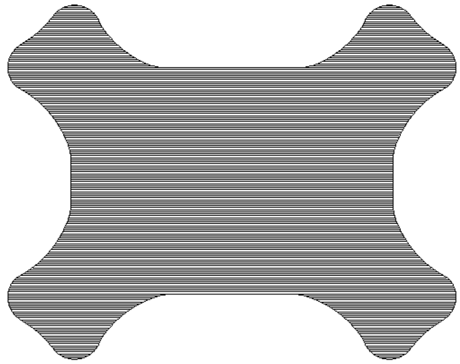


16. Choose **OK** twice.

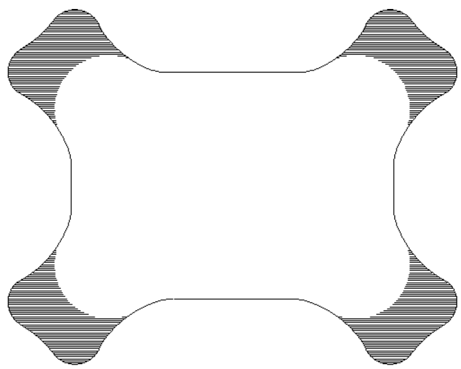
17. Mastercam will show you previews of the remachining operation. First, it shows you the areas where the rough tool could reach, with the message, "Machinable area for rough tool" in the prompt area. Press [Enter] to continue.



18. Next, it shows you the areas that the finish tool you selected should be able to reach. Press [Enter] to continue.

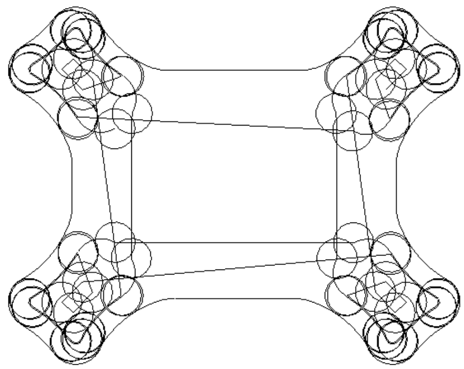


19. Next, it shows the stock that remains for the finish tool to remove, as shown in the following picture. Press [Enter] to continue.



20. Finally, Mastercam tells you how much material will remain after the remachining operation. In this case, you should see the message, "Remaining stock after remachining. Area = 0.0000." Press [Enter] to continue.

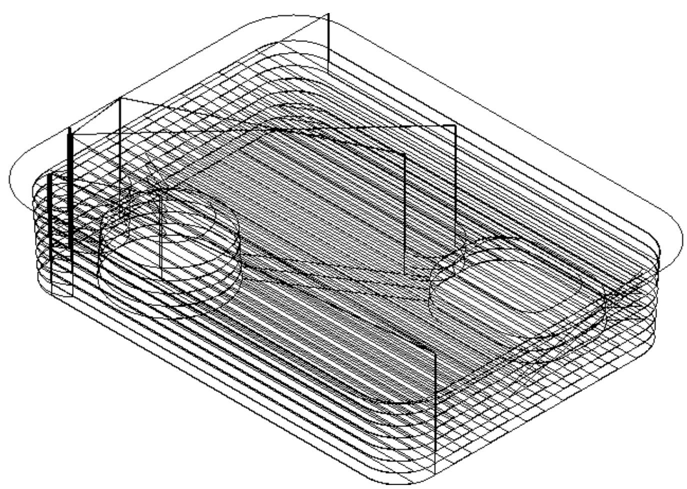
The remachining toolpath should look like the following picture.



21. Press [Alt + A] to save the file in your working folder.

Exercise 2 – Using depth cuts, island facing, and tapered walls

Depth cuts, island facing, and tapered walls are often used together on more complex pocket toolpaths. For example, tapered walls require multiple depth cuts so that the offset for the walls can be computed at each depth. Also, when you face the islands, Mastercam can automatically use the island depth as one of the depth cut values. The completed toolpath should look like the following picture.

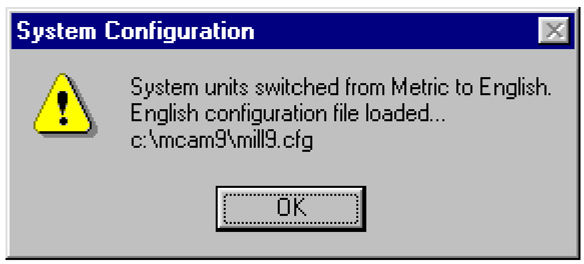


In this exercise, you will use an inch part. You will see how Mastercam automatically adjusts when you open the part and prompts you with the proper tool library and default values. In this exercise, you will learn the following skills:

- **Using an inch part**
- **Using depth cuts**
- **Using island facing**
- **Machining tapered walls**

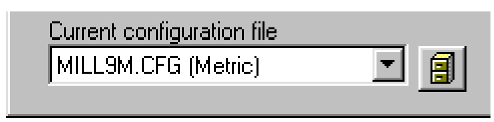
▶ *Chaining the pocket geometry*

1. Open **islands.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Inch**.
2. Choose **OK** when you see the following message.

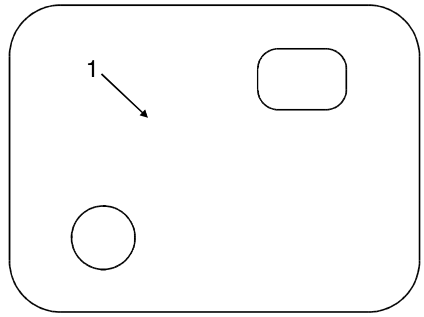


*Note: The configuration file determines which units Mastercam is using. The default files are **mill9.cfg** for working in inches and **mill9m.cfg** for metric operations. Mastercam knows which units were used to create a part file and prompts you with the previous message if it needs to switch.*

Tip: To change the current part drawing from inch to metric, choose **Main Menu, Screen, Config**. The configuration file is listed in the lower left corner:



3. Choose **Main Menu, Toolpaths, Pocket, Area**.
4. Click near location 1 to select all the boundaries.

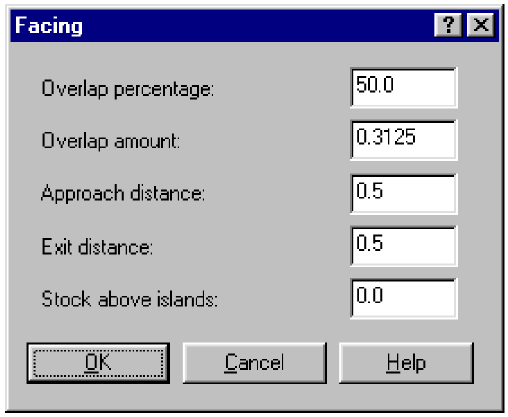


The Area selection method lets you select an area that is surrounded by a completely closed boundary by clicking inside it. Mastercam automatically knows about the islands inside the pocket when you use this method.

5. Choose **Done**.
6. Select a 5/8" flat endmill from the tool library and choose **OK**. Mastercam automatically shows you the library of inch tools.

▶ *Entering the pocketing parameters*

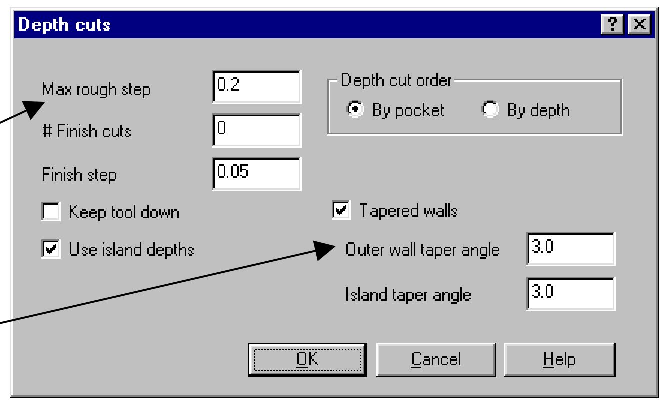
1. Select the **Pocketing parameters** tab.
2. Enter **0.25** for the **Retract** height.
3. Enter **0.1** for **Feed plane**.
4. Enter a **Depth** of **–1.75**.
5. Choose **Island facing** from the **Pocket type** drop list.
6. Choose the **Facing** button. When you choose an **Island facing** pocket type, the settings in this dialog box determine the cutting action Mastercam uses to trim the tops of the islands. Your values should match the following picture.



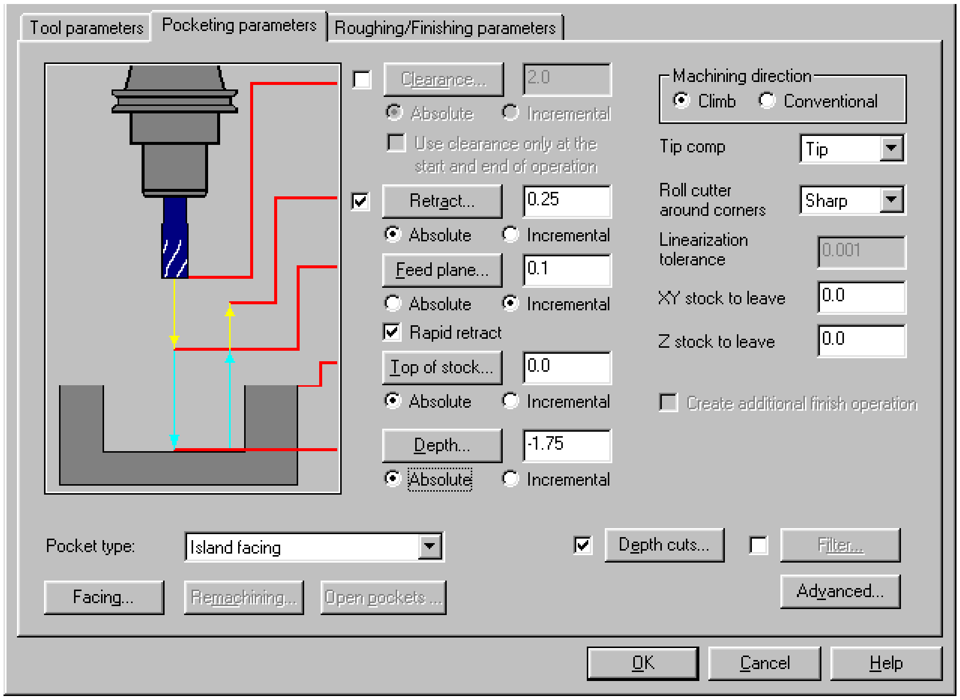
7. Choose **OK**.
8. Choose the **Depth cuts** check box and button.
9. Select **Use island depths**.
10. Select **Tapered walls**. Make sure your other values match the following dialog box.

Tip: The actual rough step used by the toolpath might be smaller than the Max rough step entered here. This is because Mastercam calculates equal step sizes (less than or equal to the Max rough step) between the top of stock and the final depth.

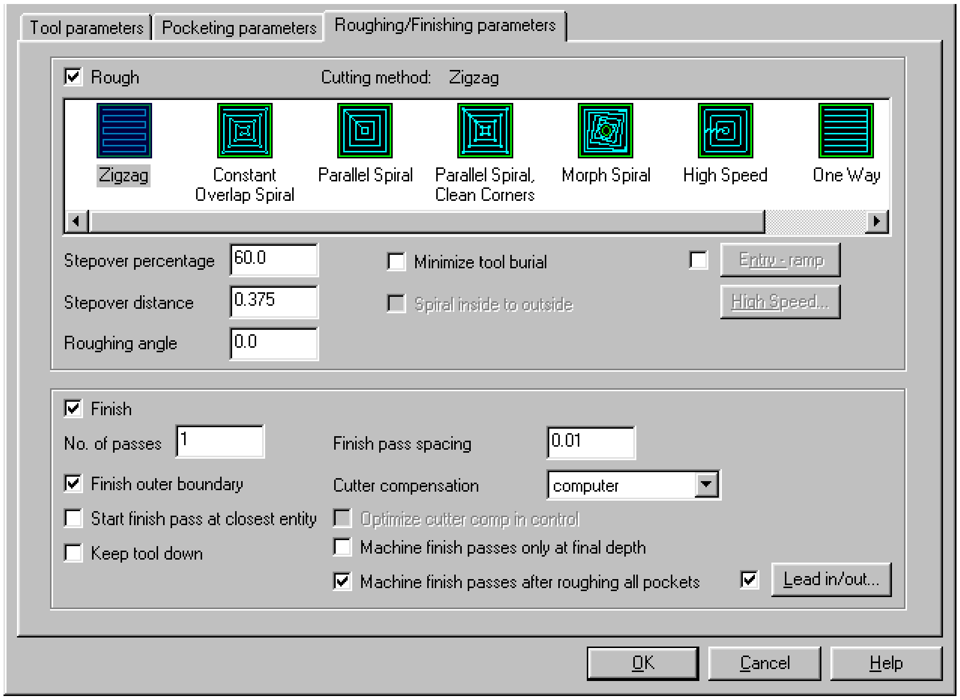
Tip: This value sets the taper for the outer boundary.



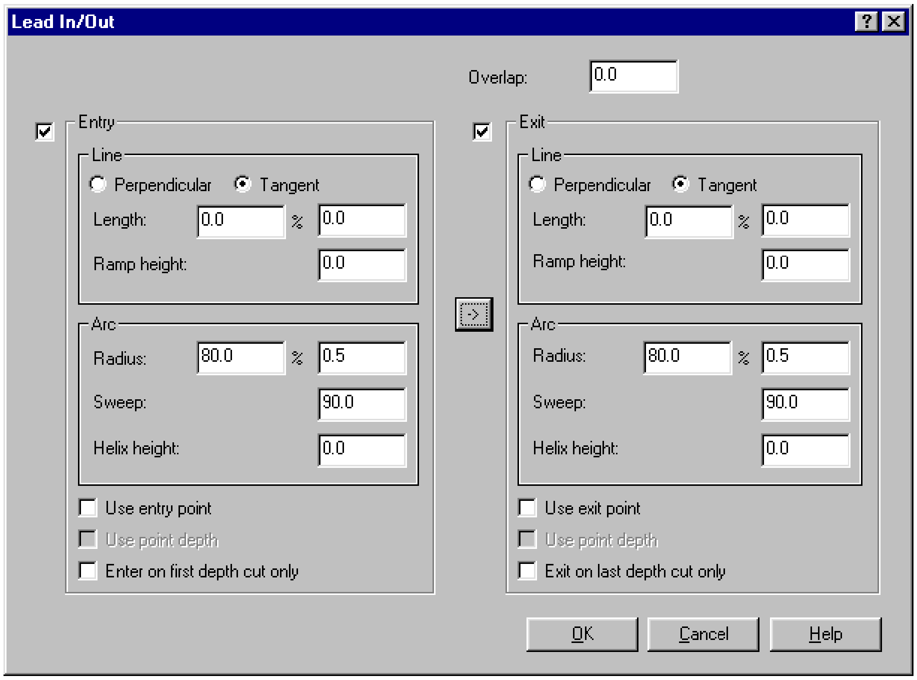
11. Choose **OK**. Your other pocketing parameters should match the following dialog box.



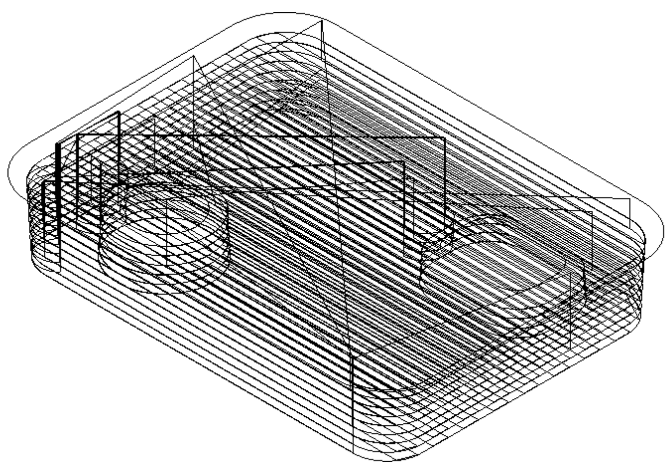
12. Select the **Roughing/Finishing parameters** tab.
13. Enter the values shown on the following dialog box.



14. Choose the **Lead in/out** check box and button.
15. Enter the values shown on the following dialog box.



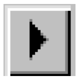
16. Choose **OK** twice. Mastercam generates the toolpath.
17. Switch to isometric view to see the toolpath more clearly. It should look like the following picture.



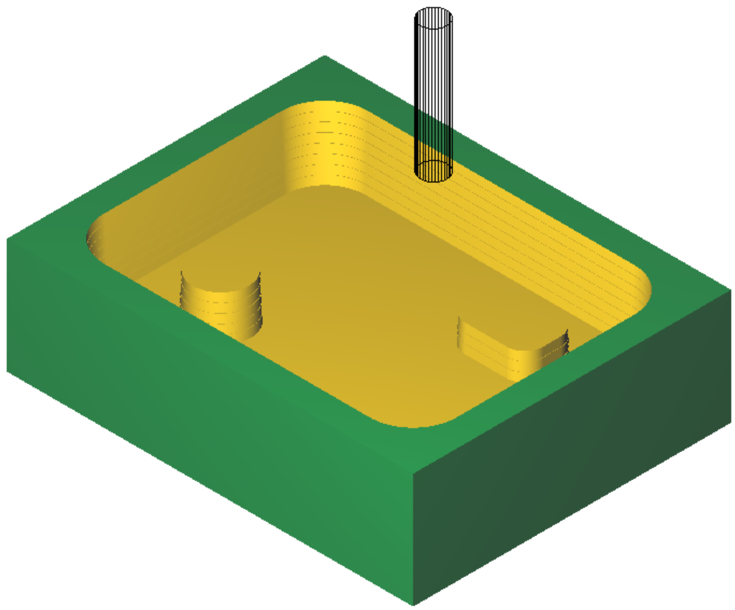
Verifying the toolpath

To get a better picture of the toolpath results, you will use the Verify function. Because it previews the actual stock removal, it will give you a clearer picture of how the islands were machined.

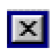
1. Choose **Operations**.
2. Choose **Verify**.



3. Choose the **Machine** button on the Verify toolbar. You should see the part machined as shown in the following picture.



Note: The stock boundary that you see in the Verify picture was already defined for you in the file.

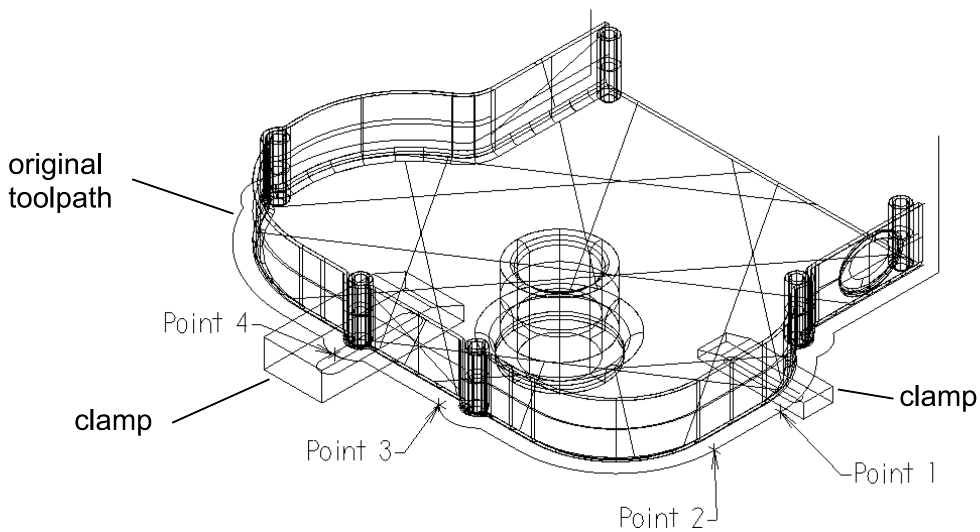
4. Choose the **Close** button  on the Verify toolbar to return to the Operations Manager.
5. Choose **OK** to close the Operations Manager.
6. Save the file in your working folder as **island pocket.mc9**.

Exercise 3 – Modifying a toolpath using the Toolpath Editor

This exercise shows you how to make corrections to a toolpath using Mastercam's Toolpath Editor. The Toolpath Editor gives you a fine level of control over the motion in your toolpath. You can make modifications to the tool motion created by Mastercam and change the areas of the part that are machined.

In this exercise, the part drawing shows the clamps that you will use to hold the part while it is being machined. You will backplot the toolpath to see if the clamps interfere with it, and then modify it so that the tool doesn't make contact with the clamps.

It is important to note that when you change a toolpath with the Toolpath Editor, the changes are not associative. This means that if you make other changes to the operation—for example, changing any of the tool or contour parameters, or changing any geometry—you will not be able to update the toolpath by choosing the Regen Path button like you have done in other exercises. The NCI file for the operation is locked so you don't overwrite the changes you have made with the Toolpath Editor. The following picture shows the part, clamps, and original toolpath.



Note: To make other changes to the toolpath after using the Toolpath Editor, unlock the NCI file and regenerate the toolpath. Make your changes, and then re-edit with the Toolpath Editor. See the online help for more information about unlocking NCI files.

In this exercise, you will learn the following skill:

- **Using the Toolpath Editor to modify selected points on a toolpath**

▶ *Inspecting the toolpath*

1. Open **tp_editor-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.**

   Note: If you have just completed the previous exercise, the units will switch to millimeters.

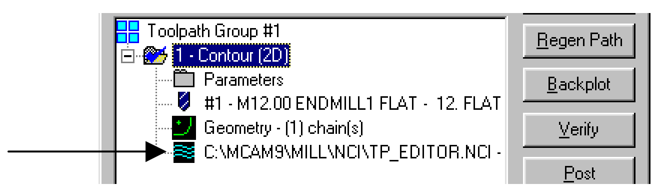
2. Choose **Main Menu, Toolpaths, Operations, Backplot**.
3. Press [S] repeatedly to step through the toolpath. You can see that the tool passes right through the clamps.
4. Choose **OK** when the backplot is complete.
5. Choose **Backup**.

   The next two procedures show you how to use the Toolpath Editor to fix the toolpath.

▶ *Adding points with the Toolpath Editor*

Your first change will be to add some points to the toolpath so it can move around the first clamp.

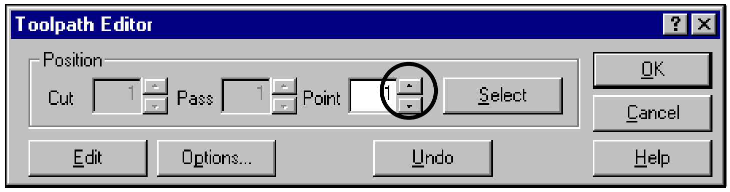
1. In the Operations Manager, right-click on the **NCI** icon for the operation:

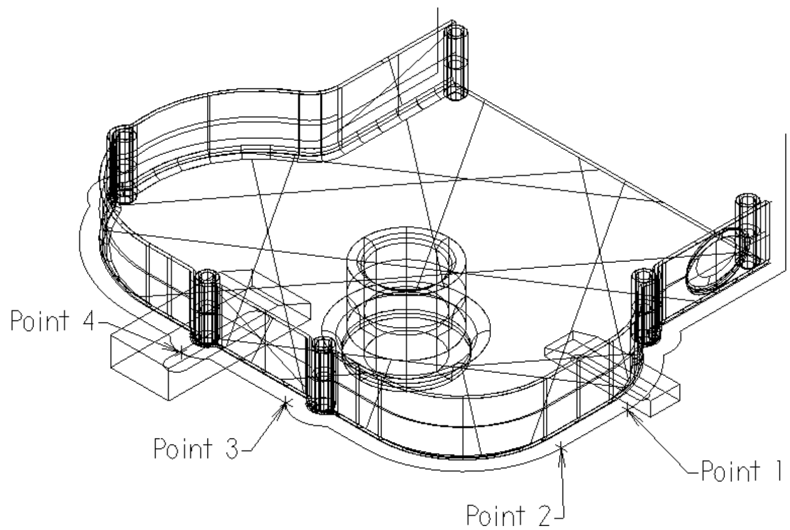
   The Toolpath Editor opens.

   Note: You can move the dialog box to the top of the graphics window to view the toolpath while you make edits.

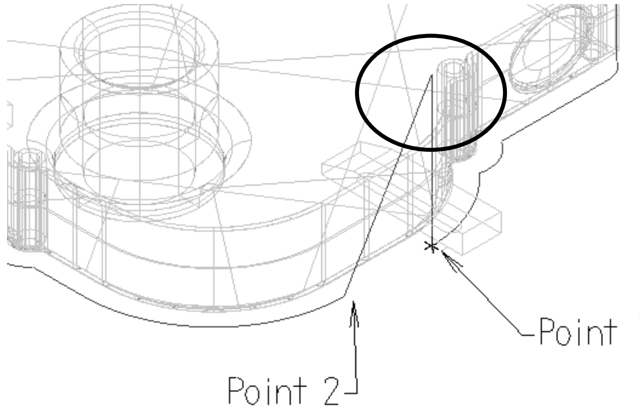
2. Use the up arrow scroll key (circled in the following picture) to move to point 8.



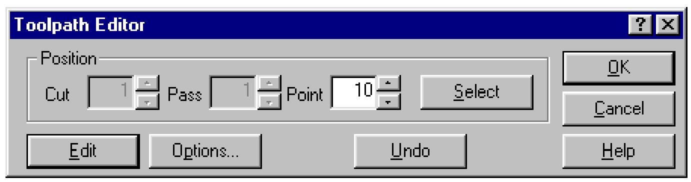
Point 1 in the following picture should be highlighted.



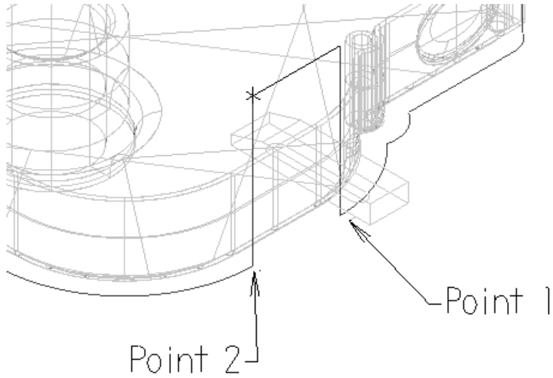
3. Choose the **Options** button.
4. Under **Point Insertion Mode**, choose **After** and choose **OK**.
5. Choose the **Edit** button.
6. Choose **Add Point, Relative, Last, Rectang**.
7. Type **z50** in the prompt area and press [Enter]. The point should be added as shown:



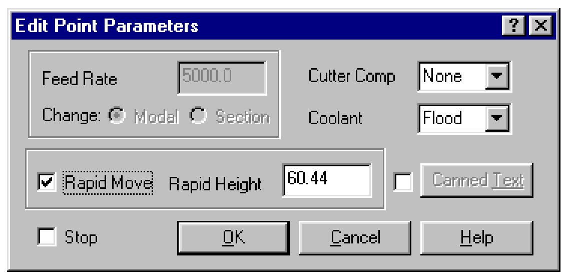
8. Click the up arrow scroll button twice. You should see **Point 10** in the Toolpath Editor dialog box as shown in the following picture, and Point 2 highlighted on the geometry.



9. Choose the **Options** button. Under **Point Insertion Mode**, choose **Before** and choose **OK**.
10. Choose the **Edit** button.
11. Choose **Add Point, Relative, Last, Rectang**.
12. Type **z50** in the prompt area and press [Enter]. The second point should be added as shown in the following picture. You can see that the new toolpath will clear the clamp.



13. Now edit the new moves so that they are rapid moves instead of feed moves. Choose **Edit, Edit Point**.
14. When the Edit Point Parameters dialog box opens, choose **Rapid Move** and choose **OK**.

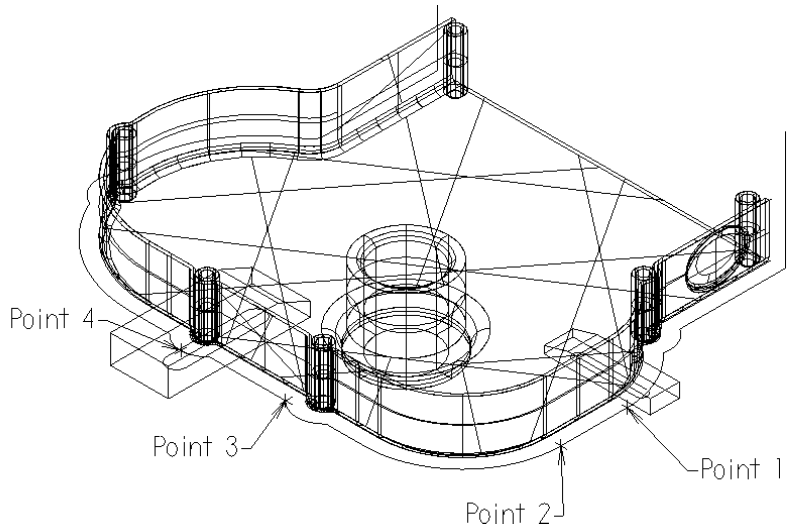


15. Use the down arrow scroll button in the Toolpath Editor to move to point 9. (This is the first point that you added.)
16. Choose **Edit, Edit Point**.
17. Choose **Rapid Move** again and choose **OK**.

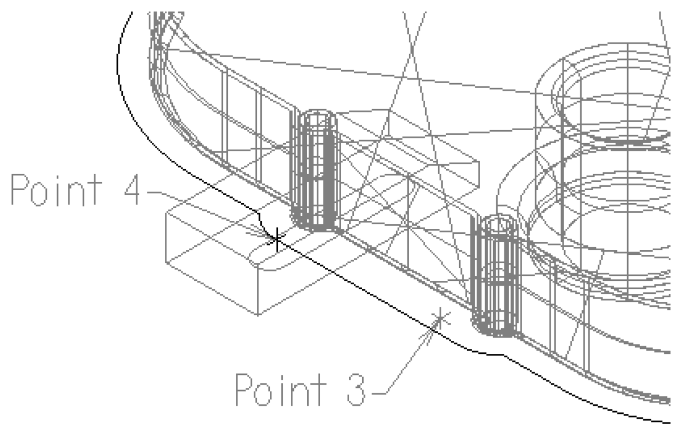
▶ *Deleting points with the Toolpath Editor*

To clear the second clamp, you will delete the section of the toolpath that crashes into it.

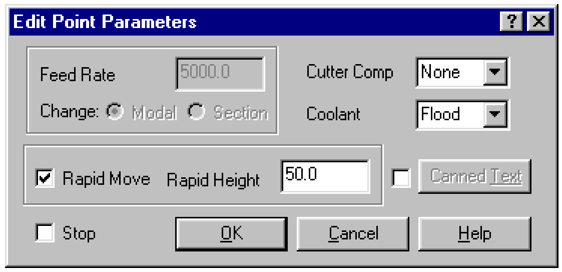
1. Choose the **Select** button and select point 3 on the geometry as shown in the following picture.



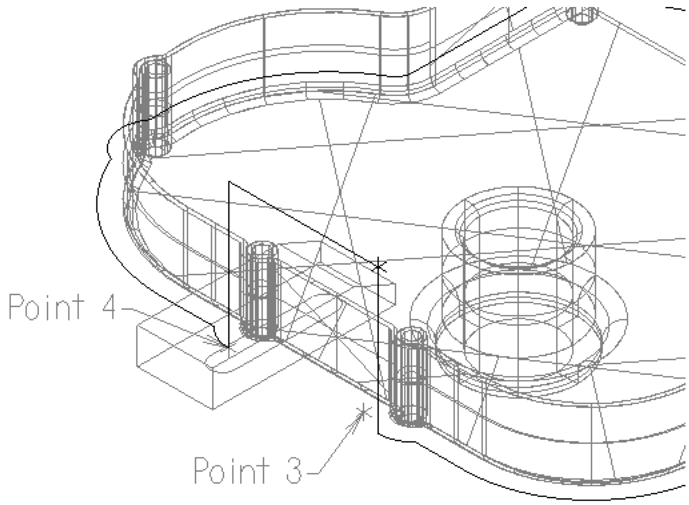
2. Choose the **Edit** button and choose **Del Section, Forward Step, Done**.
3. Choose **OK** at the confirmation message. The toolpath should look like the following picture.



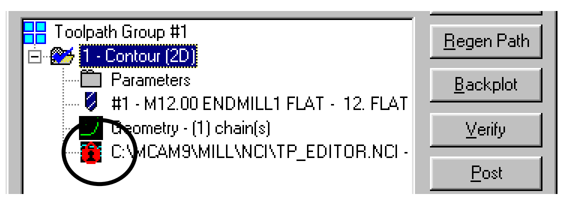
4. Choose the **Select** button and select point 4 on the geometry as shown in the previous picture.
5. Choose **Edit, Edit Point**.
6. Select **Rapid Move**.
7. Enter **50** for the **Rapid Height**.



8. Choose **OK**. The toolpath should look like the following picture.



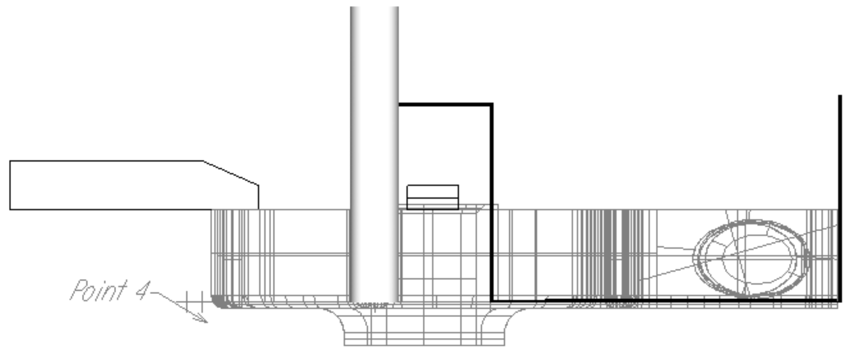
9. Choose **OK** to close the Toolpath Editor. The Operations Manager opens.



The Operations Manager shows a red lock over the NCI icon. Because you used the Toolpath Editor, the toolpath is automatically locked.

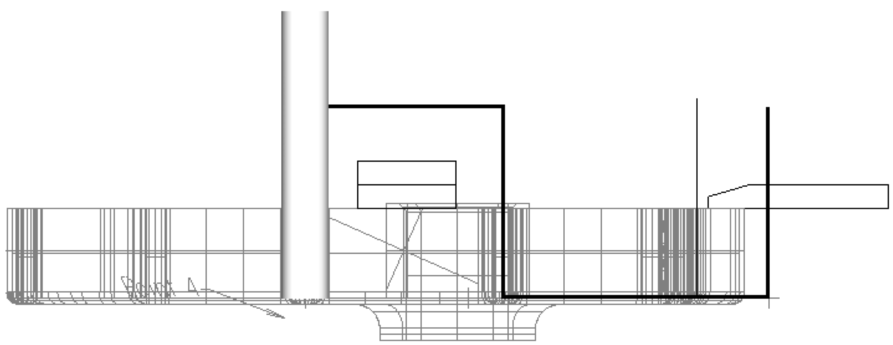
10. Choose **Backplot** to verify the new toolpath and make sure it clears the clamps.

11. To more clearly verify the first clamp, choose the **Gview–Side** button from the toolbar.
12. Press [S] until you see the tool move around the clamp.



Tip: In this picture, the tool is shown shaded. To do this, choose **Display** from the **Backplot** menu, then **Appearance** and **Shaded**.

13. To verify the second clamp, choose the **Gview–Front** button from the toolbar.
14. Press [S] until you see the tool move around the clamp.



In this chapter and several previous ones, you've seen how you can create new toolpaths efficiently by copying and editing toolpaths. The next chapter will show you how to expand on this by directly saving and importing operations and making part programs more efficient by using subprograms.

11 Reusing Operations

In several previous chapters, you've seen how to create new operations efficiently by copying and editing existing operations. In this chapter, you will expand on this concept by using the following techniques:

- grouping geometry
- saving operations to a library and importing them into a new file
- importing an operation from another part file
- using subprograms to make your part program more efficient

Exercise 1 – Creating an operations library

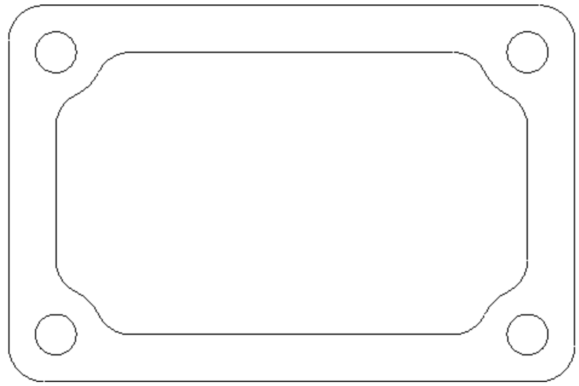
One of the best ways to reuse operations is to save them to a library. An operations library is a file that contains a collection of toolpaths. These toolpaths can then be imported into a part file and applied to geometry in it. In this exercise, you will create some drilling operations and save them to a library.

Another way to reuse operations is to import one from a part file and apply it to geometry in your current file. In this exercise, you will also create a pocketing operation and save it with the file so you can import it into another file later. You will learn the following skills:

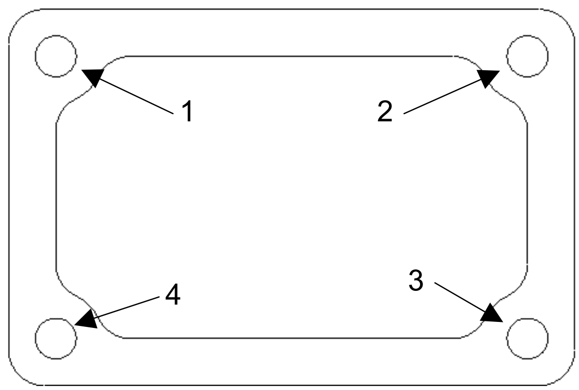
- **Creating an operations library**
- **Saving operations to a library**
- **Creating a pocketing operation that can be imported into another file**

▶ *Creating a center drill operation*

1. Open the file **cover-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.** It should look like the following picture.



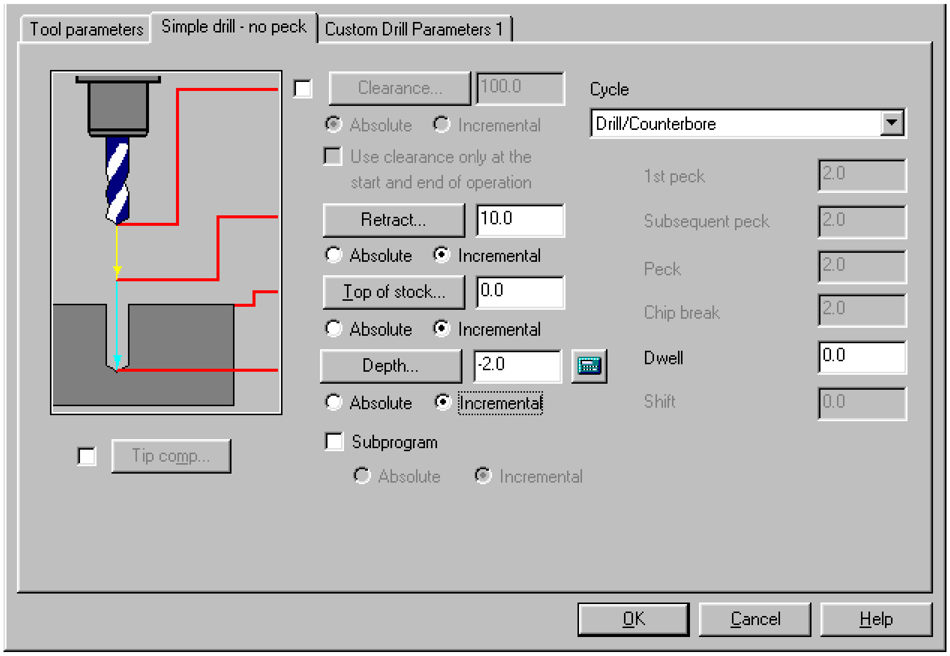
2. Choose **Main Menu, Toolpaths, Drill**.
3. Select each of the four holes as shown in the following picture.



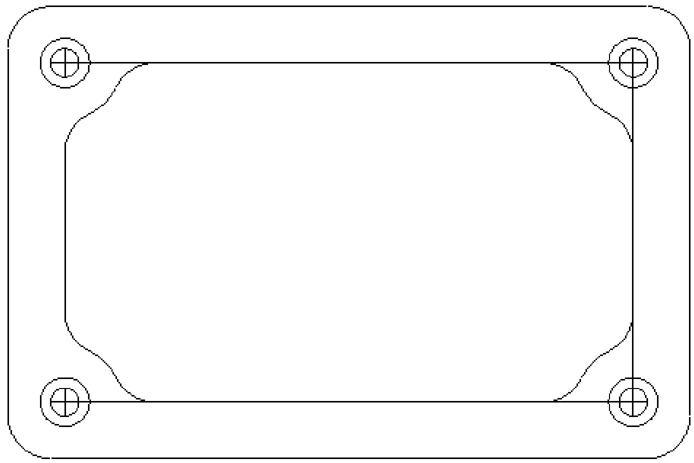
4. Choose **Done**. Mastercam shows you the drilling order of the holes.
5. Choose **Done**.
6. Right-click in the tool display area and choose **Get tool from library**.
7. Clear the **Filter Active** check box.
8. Select a 10 mm center drill from the tool library.
9. Choose the **Simple drill – no peck** tab.
10. Enter a **Depth** of **–2**.
11. Select the **Incremental** option for **Retract**, **Top of Stock**, and **Depth**.

Note: Selecting the Incremental option means that these values are relative to the chained geometry. This means that when you apply this operation to a different part, Mastercam will still drill to a depth of 2 mm even if the hole starts at a different Z-depth.

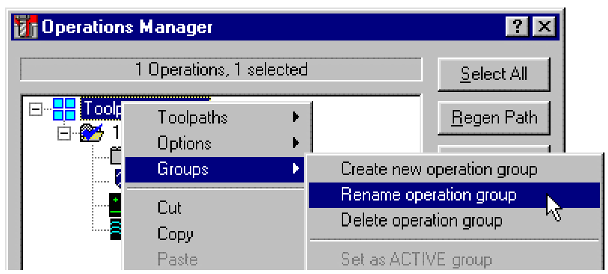
Your other parameters should match the following picture.



12. Choose **OK**. The drill path displays as shown in the following picture.



13. Choose **Operations**.
14. Right-click on **Toolpath Group 1** and choose **Groups, Rename operation group**.



15. Type the new name **Drill Operations** and press [Enter].

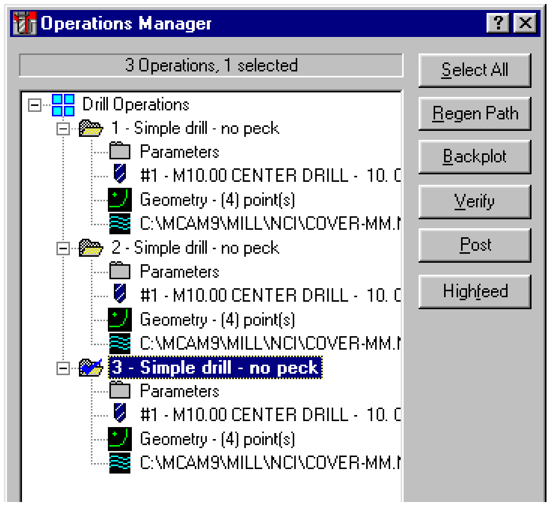
Adding the drill operation

You will copy the center drill operation and edit its parameters to create both the drill and tap operations. In this procedure, you will create the drill operation.

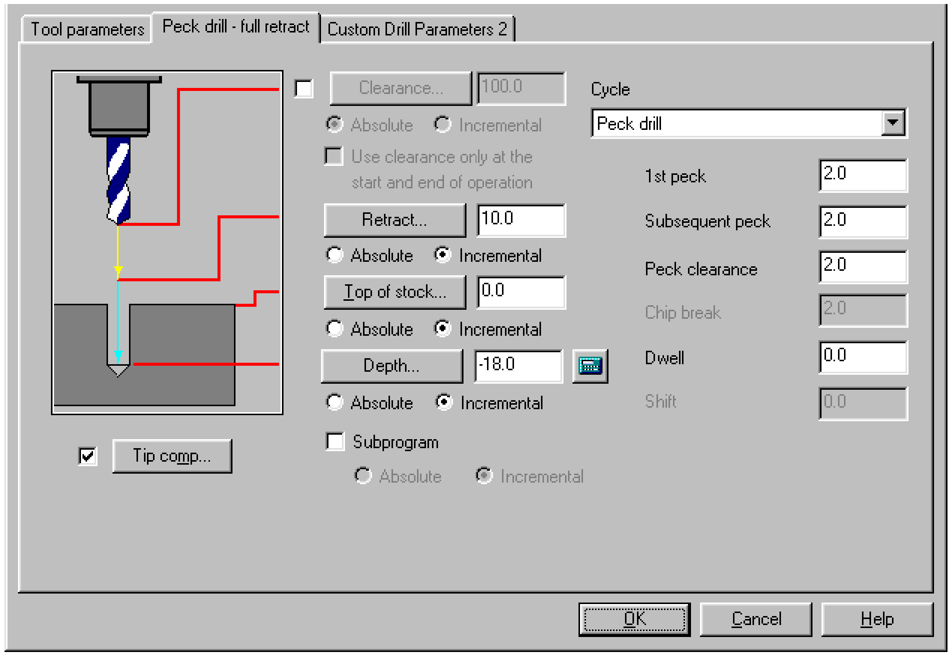
1. While the **Simple drill–no peck** operation is selected, right-click in the white area below it and choose **Copy**.
2. Right-click again and choose **Paste**.

   Tip: You can also press [Ctrl + C] and [Ctrl + V] to copy and paste operations.

3. Repeat steps 1 and 2 to create a second copy. You should see three operations as shown in the following picture.



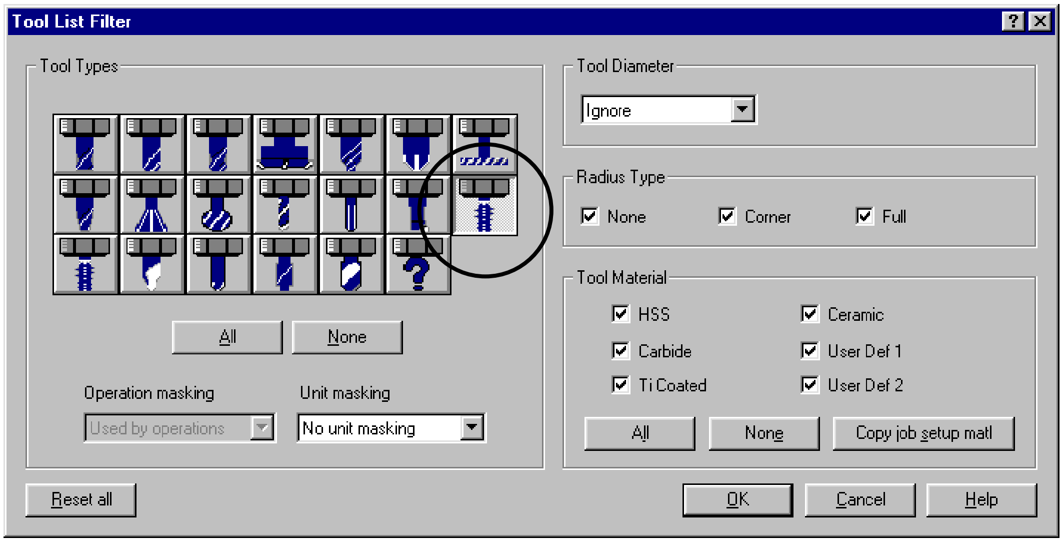
4. Choose the **Parameters** icon for operation 2.
5. Choose the **Tool parameters** tab.
6. Right-click in the tool display area and choose **Get tool from library**.
7. Select a 8.5 mm HSS drill from the tool library.
8. Choose the **Simple drill–no peck** tab.
9. Select the **Tip comp** check box. This turns on tip compensation and adds a breakthrough amount.
10. Change the **Depth** to **–18**.
11. Change the **Cycle** to **Peck drill** and enter the peck dimensions shown in the following dialog box.



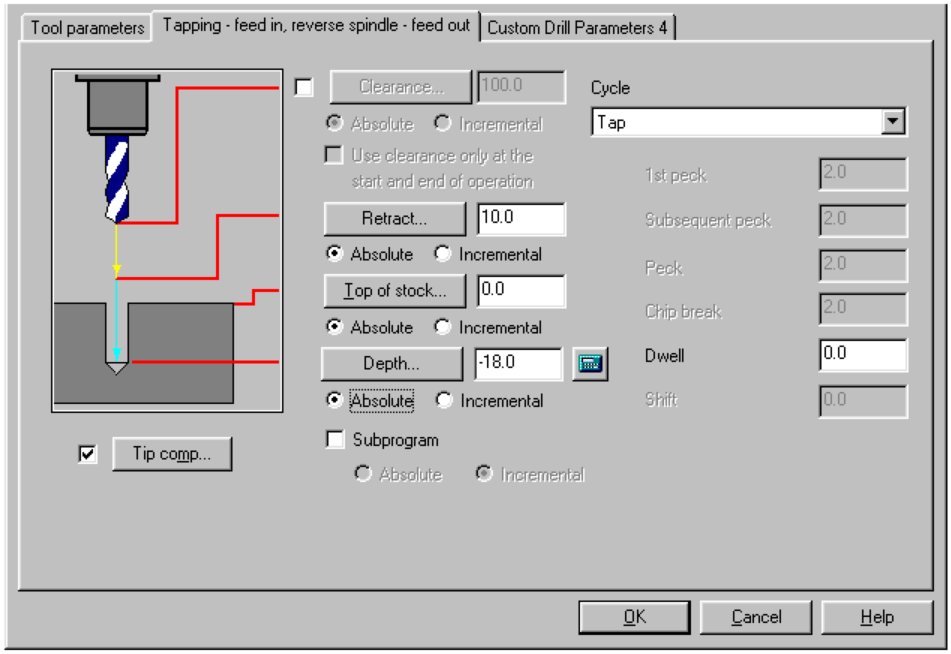
12. Choose **OK**.

Adding the tap operation

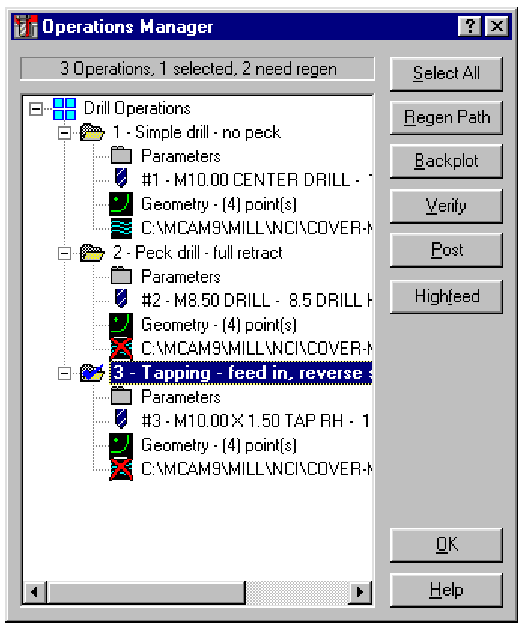
1. Choose the **Parameters** icon for operation 3.
2. Choose the **Tool parameters** tab.
3. Right-click in the tool display area and choose **Get tool from library**.
4. You need to select a 10-1.5 right-hand tap from the tool library. Choose **Filter** to use the Tool List Filter to help find the right tool.
5. Choose **None**.
6. Choose the **Tap-RH** icon as shown in the following picture.



7. Choose **OK**.
8. Select the 10-1.5 right-hand tap and choose **OK**.
9. Choose the **Simple drill–no peck** tab.
10. Select the **Tip comp** check box.
11. Change the **Depth** to **–18**.
12. Change the **Cycle** to **Tap**. Your other values should match the following dialog box.



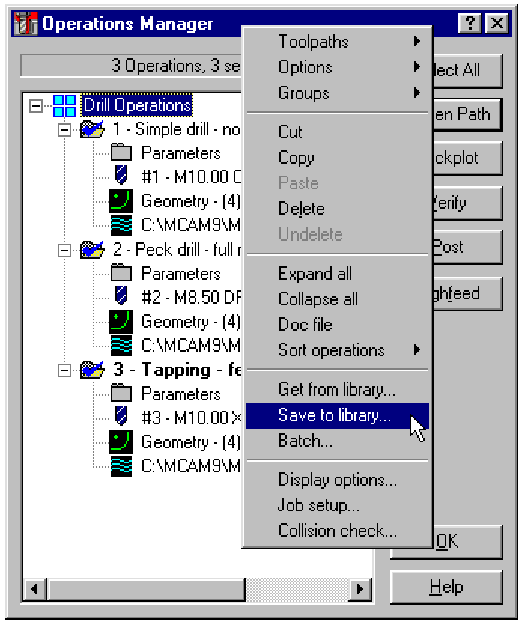
13. Choose **OK**. Your operations list should now show the following three operations.



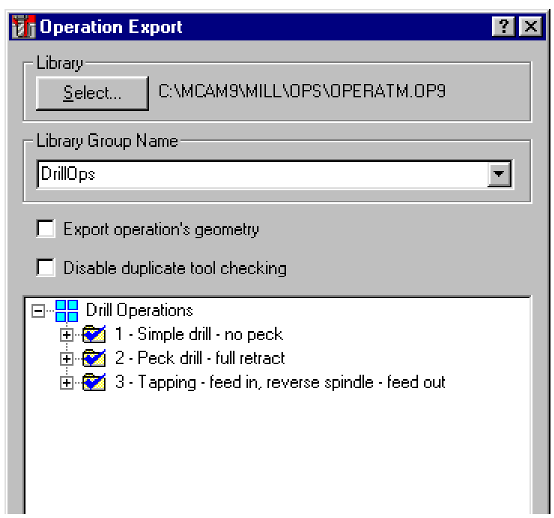
14. Choose **Select All, Regen Path** to regenerate the new operations.

▶ *Saving the drill operations to a library*

1. Right-click in the operations list window and choose **Save to library**. Make sure all the operations are selected.



2. Enter the name **DrillOps** in the **Library Group Name** field, as shown in the following picture.

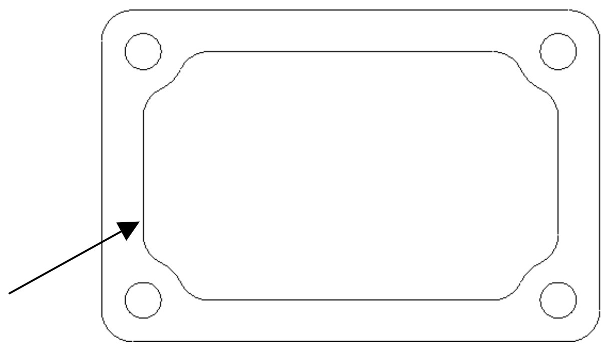


3. Choose **OK**. The operations are now saved the library.

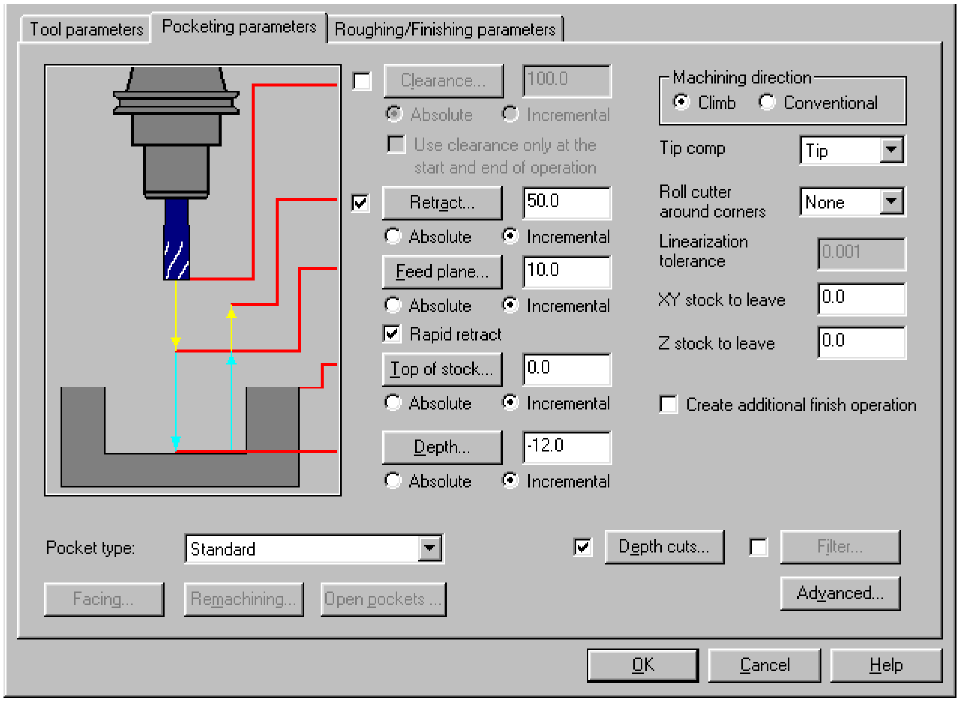
▶ *Creating the pocket toolpath*

You will now create a toolpath to machine the pocket. Just like with the drilling operations, you will specify incremental values for the depths, so that you can more easily apply it to a different part.

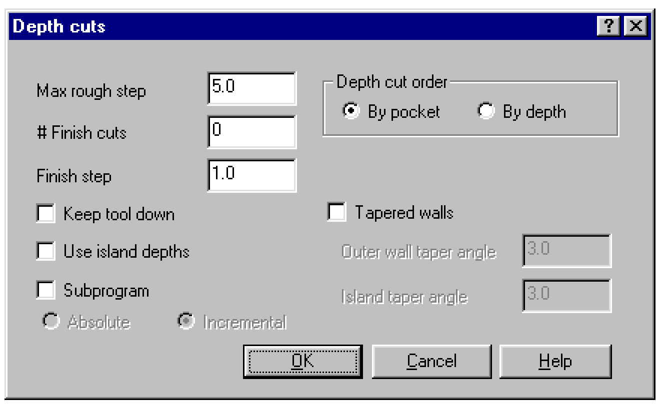
1. Right-click in the operations list window and choose **Toolpaths, Pocket**.
2. Select the pocket boundary at the location shown in the following picture.



3. Choose **Done**.
4. Select a 10 mm HSS flat endmill from the tool library.
5. Choose the **Pocketing parameters** tab.
6. Enter the values shown in the following dialog box.



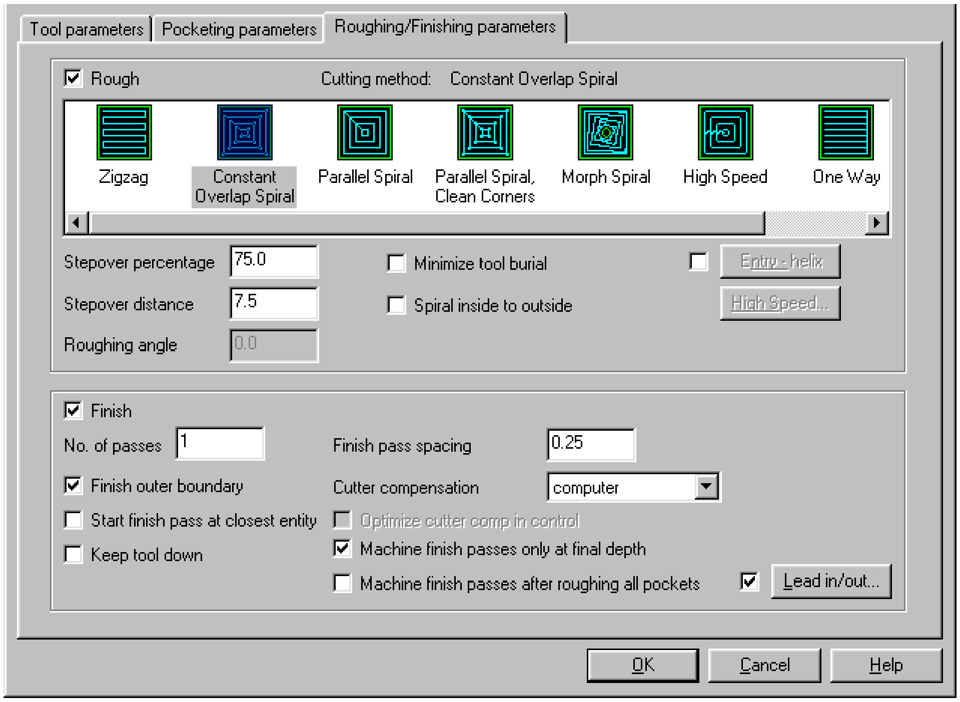
7. Choose the **Depth cuts** check box and button.
8. Enter the values shown in the following dialog box.



*Note: The **Depth cut order – By pocket** setting means that if you chain multiple pockets to this operation, each pocket will be cleaned out completely before the tool moves to the next pocket.*

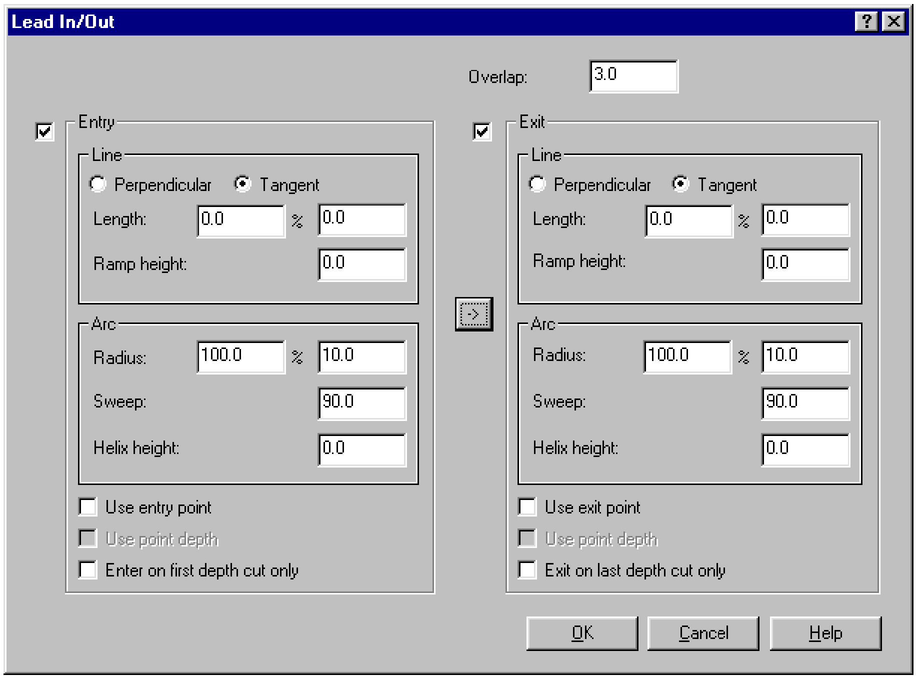
9. Choose **OK**.
10. Choose the **Roughing/Finishing parameters** tab.

11. Enter the values shown in the following dialog box.

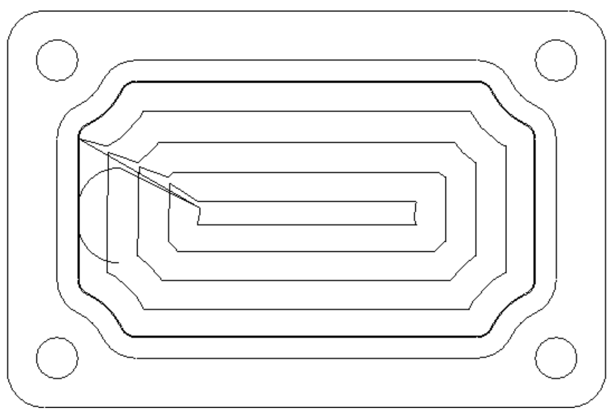


12. Choose **Lead in/out**.

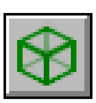
13. Enter the values shown in the following dialog box.



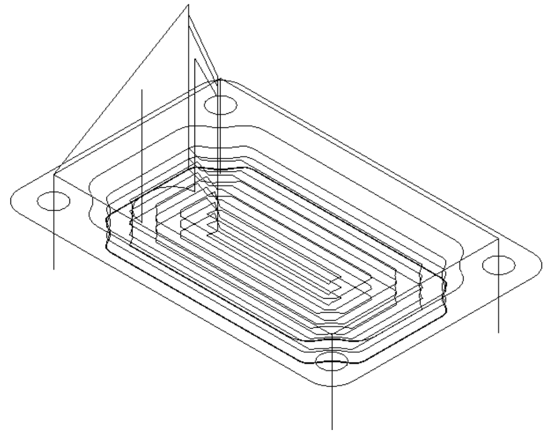
14. Choose **OK** twice. Your toolpath should look like the following picture.



15. Choose **Select All, Backplot**.



16. Choose the green **Gview–Isometric** button from the toolbar and toggle **Verify=N** so you can see the toolpaths more clearly.
17. Press [S] repeatedly to step through the toolpaths. They should look like the following picture.



18. When the backplot is done, choose **OK** and **Backup** to return to the Operations Manager.
19. Choose **OK** to close the Operations Manager.
20. Choose **Main Menu, File, Save** and save the file as **pocket cover.mc9** in your working folder.

Exercise 2 – Importing operations

The part featured in this exercise requires that you machine holes and pockets with toolpath parameters similar to the last exercise. You will import the drill operations from the operations library and the pocket operation from the **pocket cover.mc9** file.

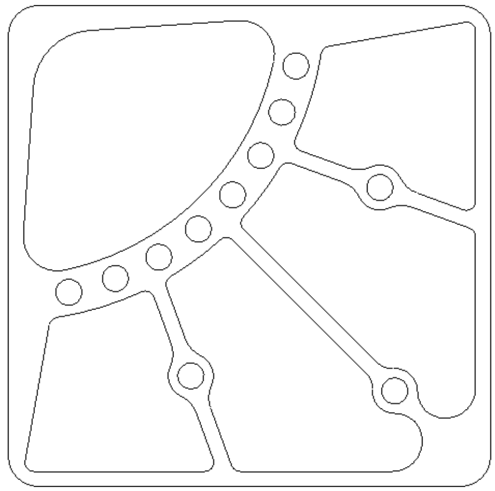
In this exercise, you will learn the following skills:

- **Grouping geometry**
- **Importing operations from a library**
- **Importing an operation from another MC9 file**

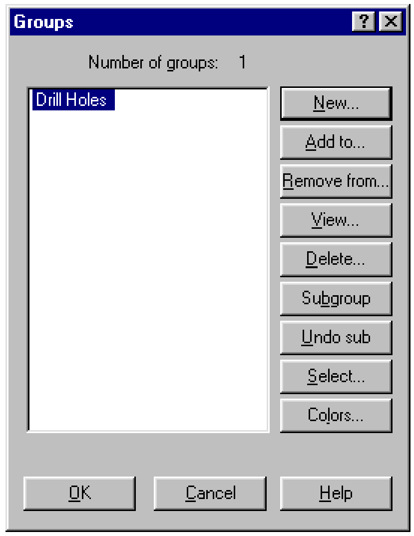
▶ *Grouping the geometry*

Use groups when you will want to use the same set of entities multiple times. Instead of selecting the different entities each time, you can use the group name as a shortcut. In this procedure, you will assign a group name to a set of holes that will be used for multiple drilling operations.

1. Open the file **cover2-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.** It should look like the following picture.



2. Choose **Groups** from the Secondary Menu.
3. Choose **New**.
4. Mastercam prompts you to enter a group name. Type the name **Drill Holes** and press [Enter].
5. Select each of the drill holes.
6. Choose **Done**. Your Groups dialog box should look like the following picture.

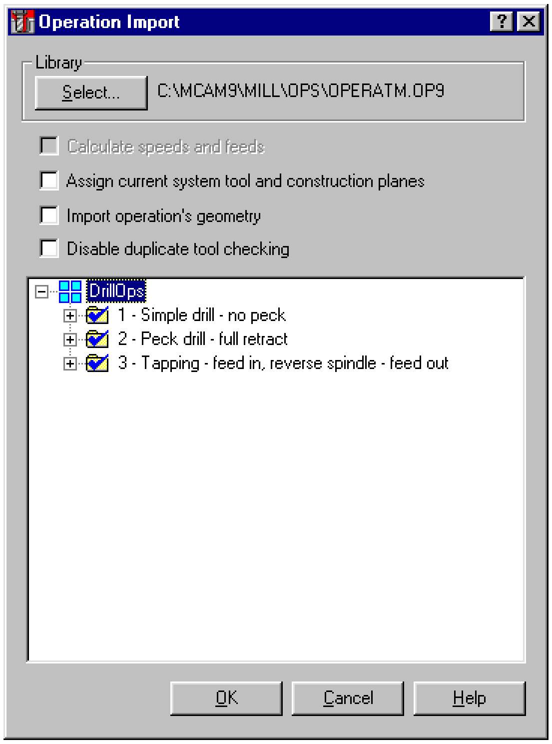


7. Choose **OK**.

▶ *Importing the drill operations*

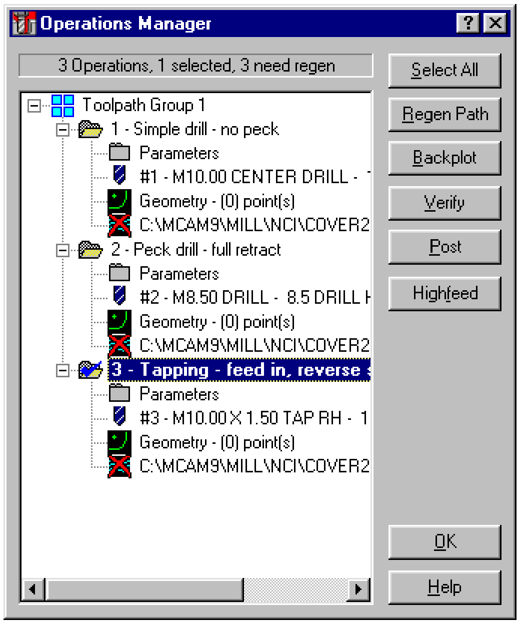
You will import the drill operations from the library that you created in the previous exercise, then apply each operation to the group of holes.

1. Choose **Main Menu, Toolpaths, Operations**.
2. Right-click in the empty operations list and choose **Get from library**.
3. Choose the **DrillOps** group so that all the operations have a blue check mark in front of them. Your dialog box should look like the following picture.

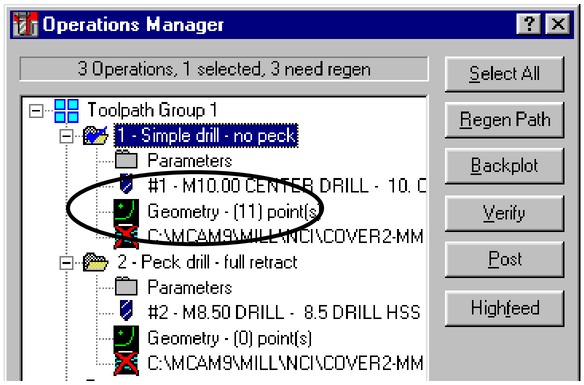


4. Choose **OK**.
5. Choose **No** at the following message.

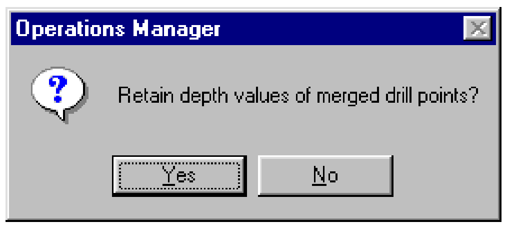
6. Press [E] to expand all the operations. Your operations list should match the following picture.



7. For each operation, you can see "Geometry - (0) points." This means that there are no points associated with any of the drilling operations. To add drill points to the first drilling operation, choose the **Geometry** icon for operation 1.
8. Choose **Add pts, Entities, Group**.
9. Choose **Drill Holes** from the Groups dialog box.
10. Choose **OK**. Mastercam adds all the holes from the group that you created ealier.
11. Choose **Done** three times. You can see the new holes listed in the operations list.



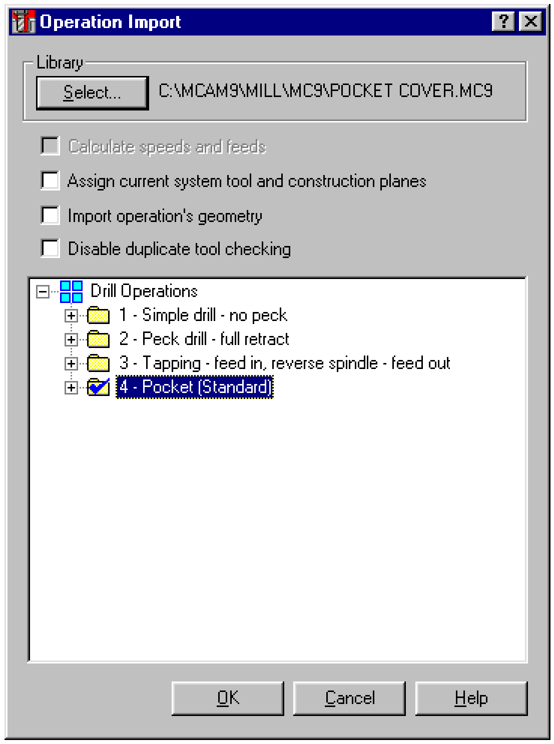
12. To quickly add the same holes to the other operations, click on the **Geometry** icon for operation 1 and drag it on top of the Geometry icon for operation 2.
13. Choose **Replace** from the menu.
14. Choose **Yes** at the following message.



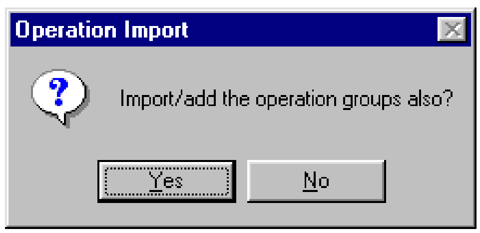
15. Repeat steps 12 through 14 for operation 3.
16. Choose **Select All, Regen Path**. Mastercam regenerates the drill toolpaths with the new holes added.

▶ *Importing the pocketing operation*

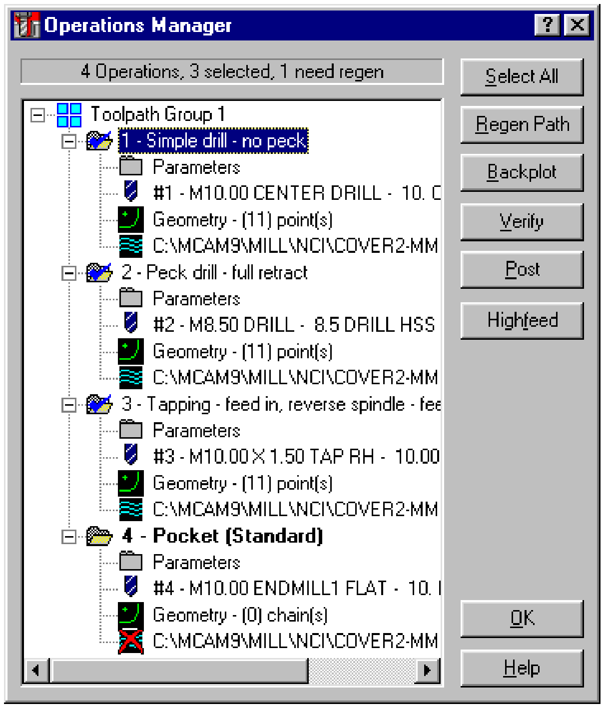
1. Right-click in the operations list and choose **Get from library**.
2. Choose the **Select** button.
3. From the **Save as type** drop-list, choose **All Files (*.*)**.
4. Locate and select the **pocket cover.mc9** file that you created in the previous exercise.
5. Choose the **Pocket** operation. The other settings should match the following picture.



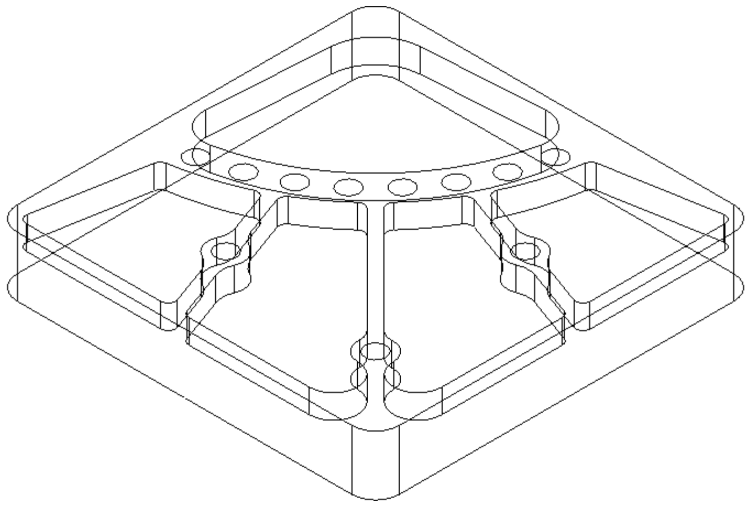
6. Choose **OK**.
7. Choose **No** at the following message.



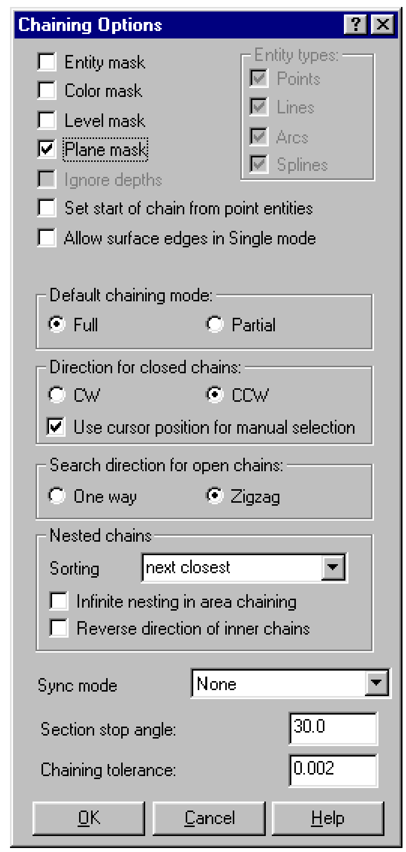
You should see the Pocket operation added to the operations list.



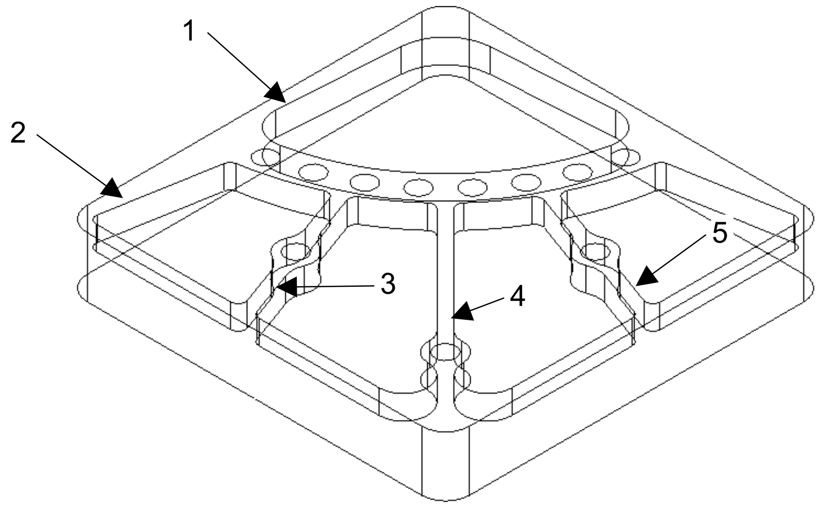
8. Select the **Geometry** icon for the Pocket operation.
9. Right-click in the Chain Manager dialog box and choose **Add chain**.
10. Switch to Isometric view so you can see the top and bottom of the pockets more clearly.



11. Choose **Chain, Options**.
12. Choose **Plane Mask** as shown in the picture at right. Setting a plane mask means that when you chain one of the pocket boundaries, Mastercam won't try to chain the construction lines connecting the top and bottom boundaries of the pocket.
13. Choose **OK**.



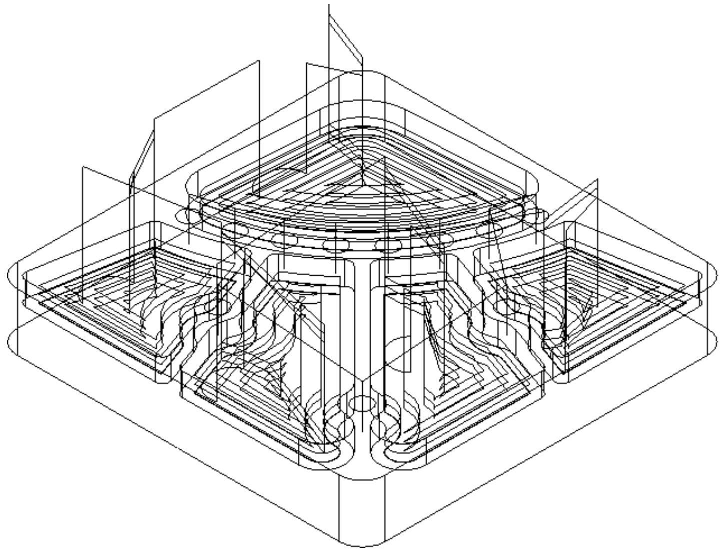
14. Select the pockets at the positions and in the order shown in the following picture.



Note: Make sure you select the top boundary of each pocket, not the bottom.

15. Choose **Done**. The Chain Manager now shows you the five chains you just added.

16. Choose **OK, Regen Path**. The completed toolpath should look like the following picture.



17. Choose **OK** to close the Operations Manager.
18. Save the file in your working folder as **imported pockets.mc9**.

Exercise 3 – Using subprograms

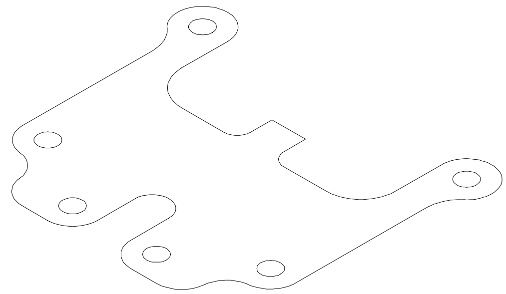
A subprogram is a section of an NCI file that is reused at different locations, thereby reducing the size of the file and of the NC program that results when you post it. The part in this exercise uses contour depth cuts; because the tool motion at each depth is the same, you will create a subprogram for the cutting pass that Mastercam will use for each depth cut. Subprograms can also be used on pocket depth cuts, circle mill toolpaths, drill toolpaths, and transform toolpaths.

In this exercise, you will learn the following skills:

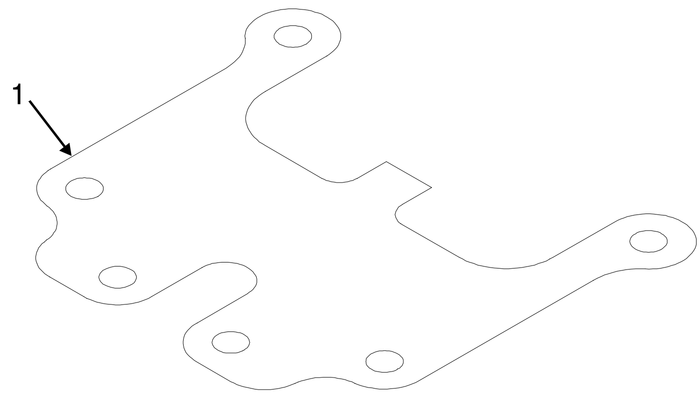
- **Creating a subprogram**
- **Viewing the resulting NC file**

▶ *Open the file and generate the toolpath*

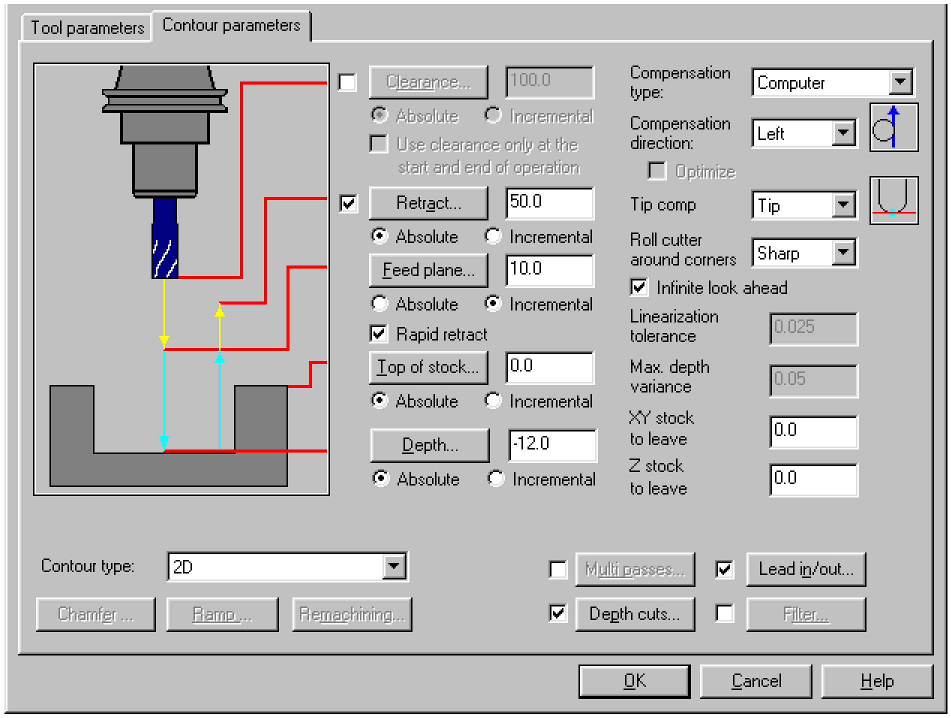
1. Open **subprogram-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric.** It should look like the following picture.



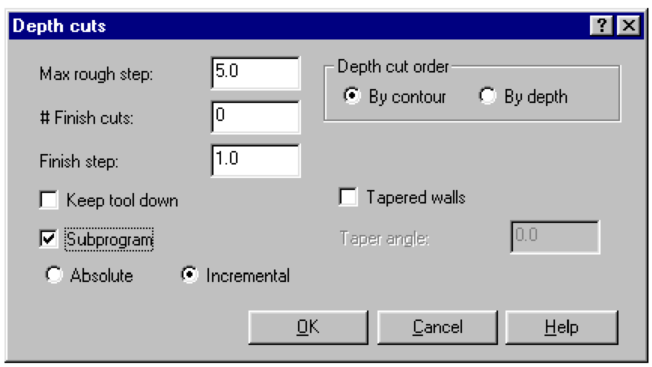
2. Choose **Main Menu, Toolpaths, Contour**.
3. Chain the contour at position 1 as shown in the following picture. Make sure the arrow points in the clockwise direction. If it does not, choose **Reverse**.



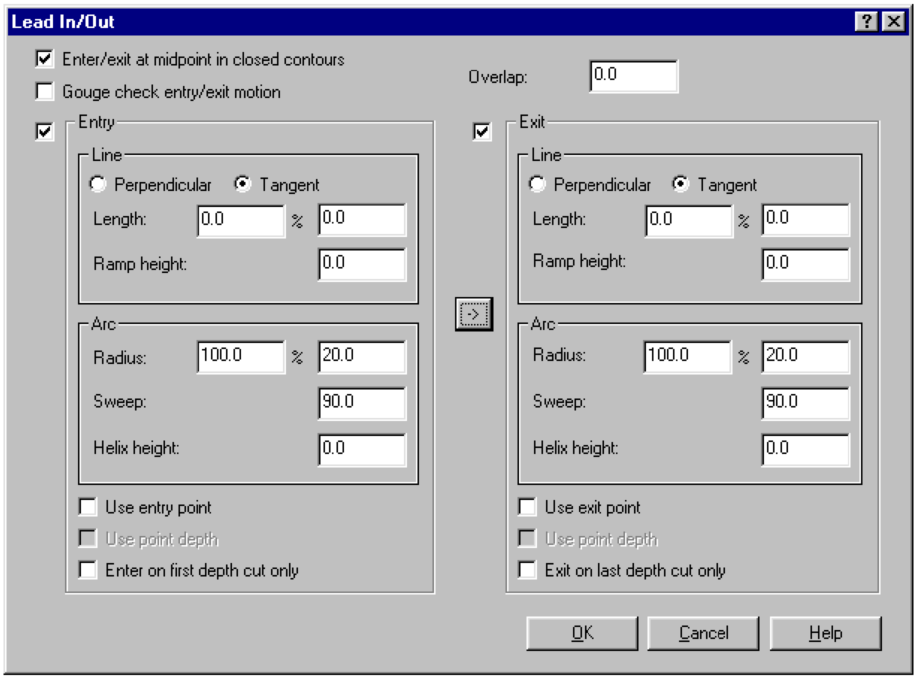
4. Choose **Done**.
5. Select the 20 mm HSS flat endmill from the tool library.
6. Select the **Contour parameters** tab.
7. Enter the values shown on the following dialog box.



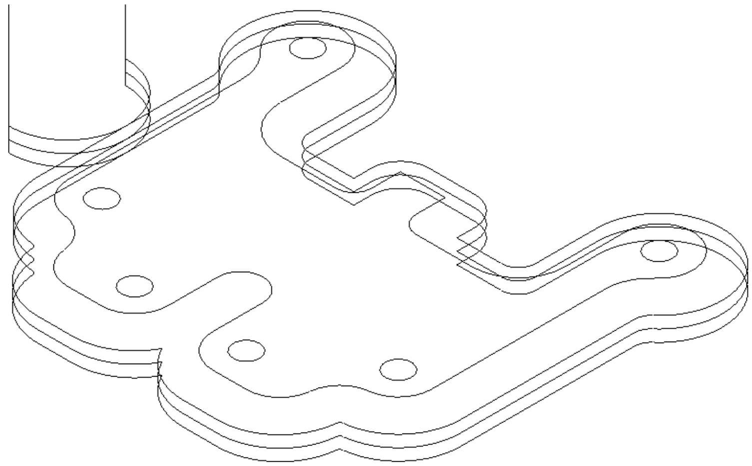
8. Choose the **Depth cuts** button.
9. Enter the values shown on the following dialog box.



10. Choose **OK**.
11. Choose the **Lead in/out** button.
12. Enter the values shown on the following dialog box.

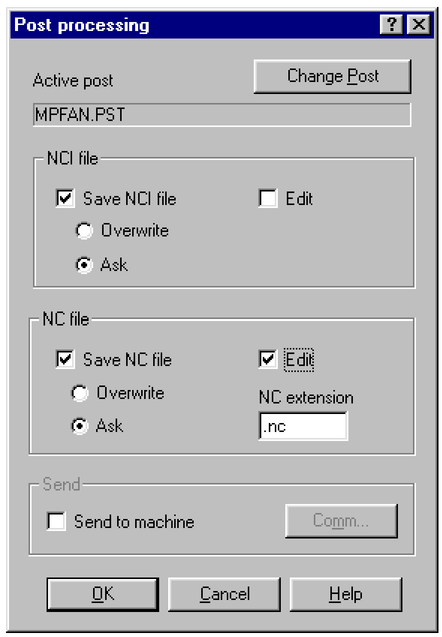


13. Choose **OK** twice. Mastercam generates the toolpath as shown in the following picture.



Post the operation and view the NC file

1. Choose **Operations**.
2. Choose **Post**.
3. Enter the values shown on the following dialog box.



4. Choose **OK**.
5. Choose **Save** to save the NCI file as **subprogram.nci**.

6. Choose **Save** to save the NC file as **subprogram.nc**. Mastercam displays the NC file in the default text editor. The NC file should look like the following picture.

```
%
O0000
(PROGRAM NAME - SUBPROGRAM)
(DATE=DD-MM-YY - 11-02-02 TIME=HH:MM - 14:51)
N100G21
N102G0G17G40G49G80G90
/N104G91G28Z0.
/N106G28X0.Y0.
/N108G92X0.Y0.Z0.
( 20. FLAT END MILL HSS TOOL - 1 DIA. OFF. - 41 LEN.
N110T1M6
N112G0G90X-92.349Y-49.532A0.S400M3
N114G43H1Z50.
N116Z10.
N118G1Z-4.F36.
N120M98P1001
N186G0G90Z50.
N188Y-49.532
N190Z6.
N192G1Z-8.
N194M98P1001
N260G0G90Z50.
N262Y-49.532
N264Z2.
N266G1Z-12.
N268M98P1001
N334G0G90Z50.
N336M5
N338G91G28Z0.
N340G28X0.Y0.A0.
N342M30
%
```

Tip: The M98 or M99 code is a subprogram indicator.

The subprogram code replaces many blocks that would otherwise be repeated, reducing the size of the NC file.

7. Close the text editor.
8. Choose **OK** to close the Operations Manager.
9. Save the file in your working folder as **cut-depth subprogram.mc9**.

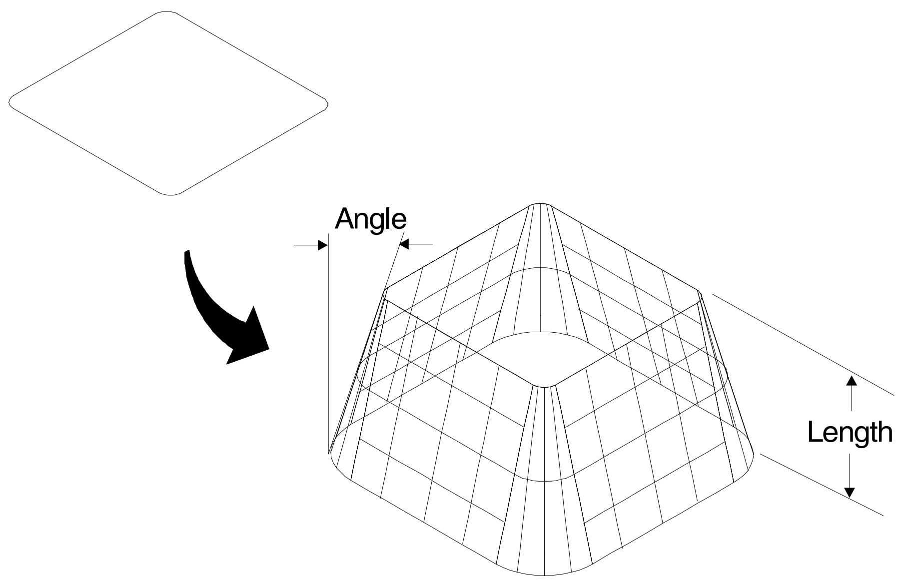
This chapter completes the section on wireframe geometry. The rest of this tutorial introduces you to machining more sophisticated geometry such as surfaces and solids.

12 Choosing a Surface Type

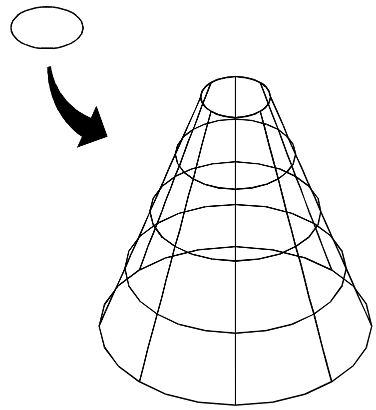
The next several chapters introduce you to surface machining. A surface is a 2D or 3D bounded shape that has no thickness. This chapter explains the types of surfaces you can create with Mastercam and shows examples of each type. In the next chapter, you will create and machine some surfaces.

Draft

The Draft surface function creates a surface that has angled (or tapered) walls defined by a given length and angle. The following picture shows the draft surface that is created from a single chain of curves. You can create this surface type by choosing **Main Menu, Create, Surface, Draft**.

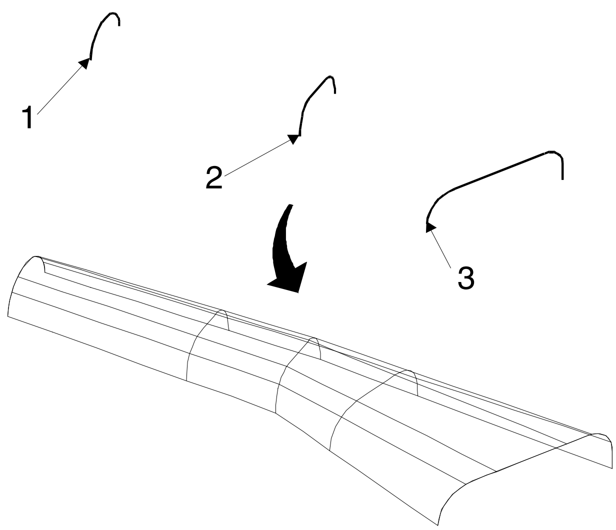


You can also use the Draft surface function to create a chain of curves that contains a single entity.

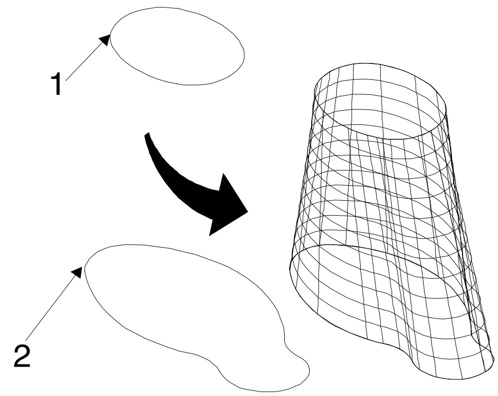


Ruled

The Ruled surface function creates a surface by transitioning between two or more chains of curves in the order that you select them and by using linear blending between each section of the surface. It is important to select each chain of curves at the same relative position to each other. The following picture shows the surface created when you select at positions 1, 2, and 3. You can create this surface type by choosing **Main Menu, Create, Surface, Ruled**.

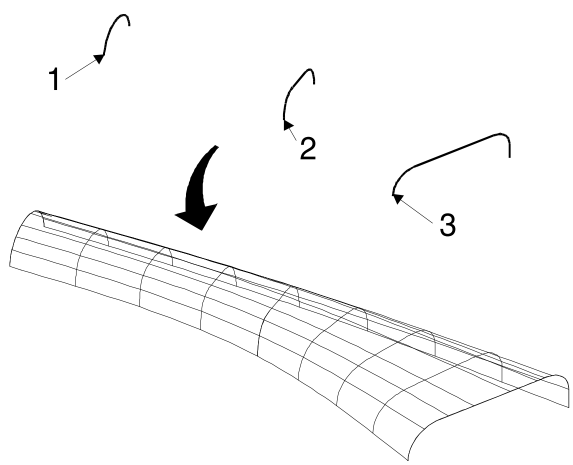


The following picture shows the ruled surface that is created when you select at positions 1 and 2.

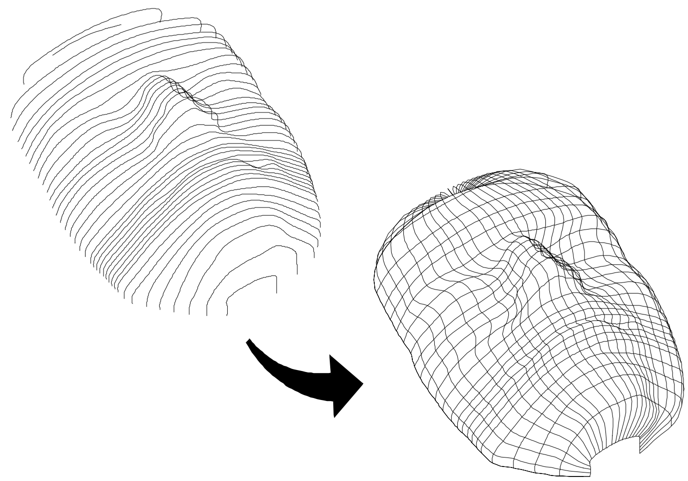


Loft

The Loft surface function creates a surface by transitioning between two or more chains of curves in the order that you select them and calculating a smooth blend by considering all the section chains at once. It is important to select each chain of curves at the same relative position to each other. The following picture shows the surface that is created when you select at positions 1, 2, and 3. Notice the difference between the Loft surface and the Ruled surface on the previous page using the same wireframe geometry. You can create this surface type by choosing **Main Menu, Create, Surface, Loft**.

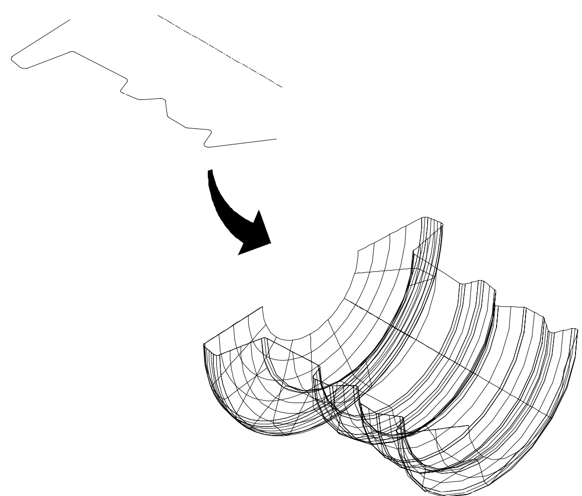


The loft surface shown in the following picture uses 40 cross-sections.



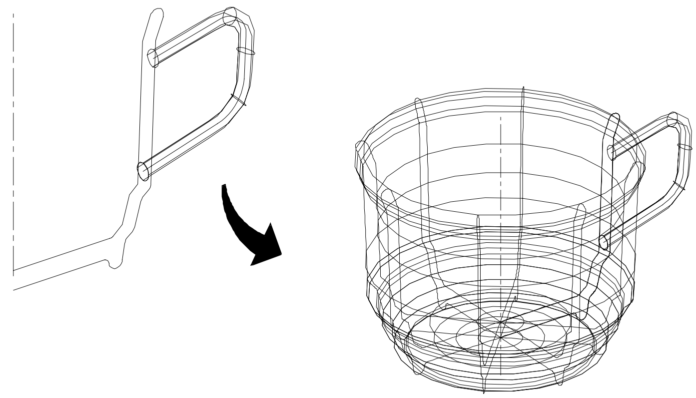
Revolved

The Revolved surface function creates a circular surface by driving the shape of a selected chain of curves about an axis using given start and end angles. Use Revolved when a cross-section and an axis can describe a surface, as shown in the following example. You can create this surface type by choosing **Main Menu, Create, Surface, Revolve**.



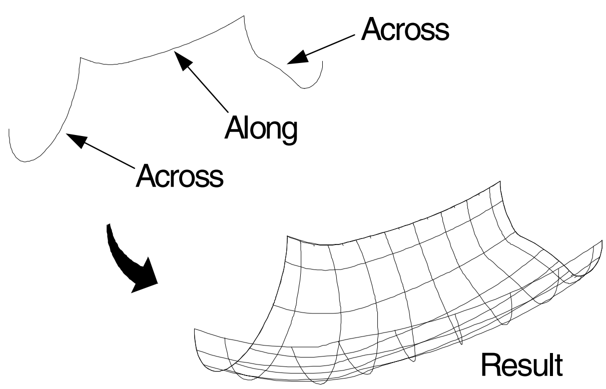
The coffee cup shown in the following picture is another example of a revolved surface.

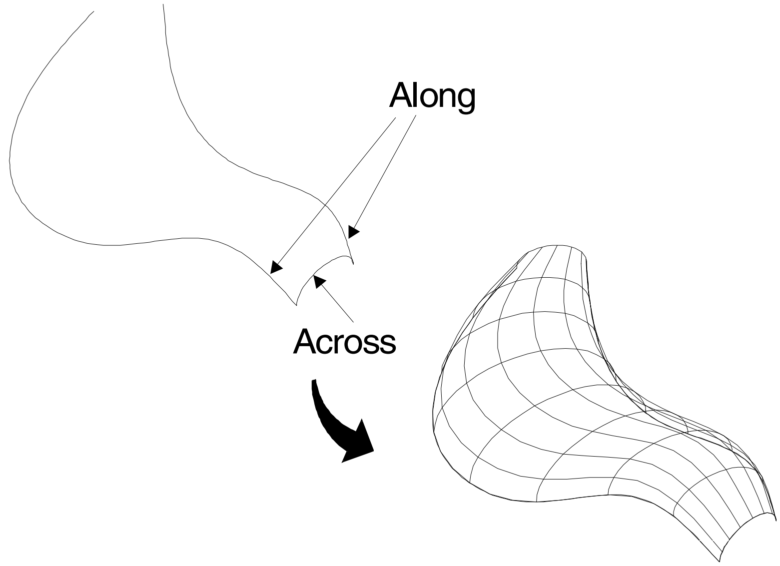
Note: The handle of the coffee cup is created separately using a swept surface function. It is not part of the revolved surface.

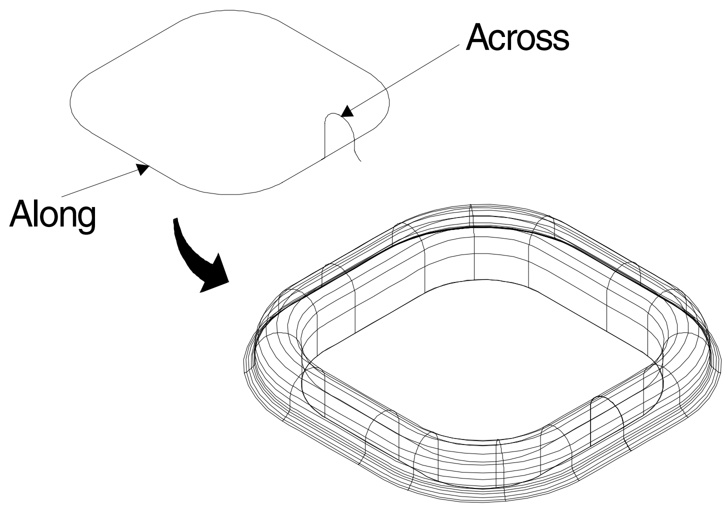


Swept

The Swept surface function creates many different surface configurations depending on the curves that you select. The system sweeps chains of curves called "across contours" over other chains of curves called "along contours." You can select any number of across curves if you are using one along curve. This surface type is shown in the following three pictures. You can create this surface type by choosing **Main Menu, Create, Surface, Sweep**.

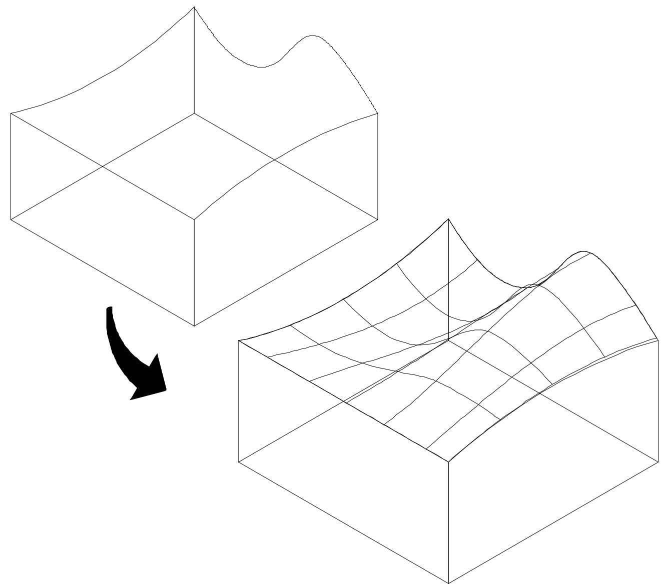




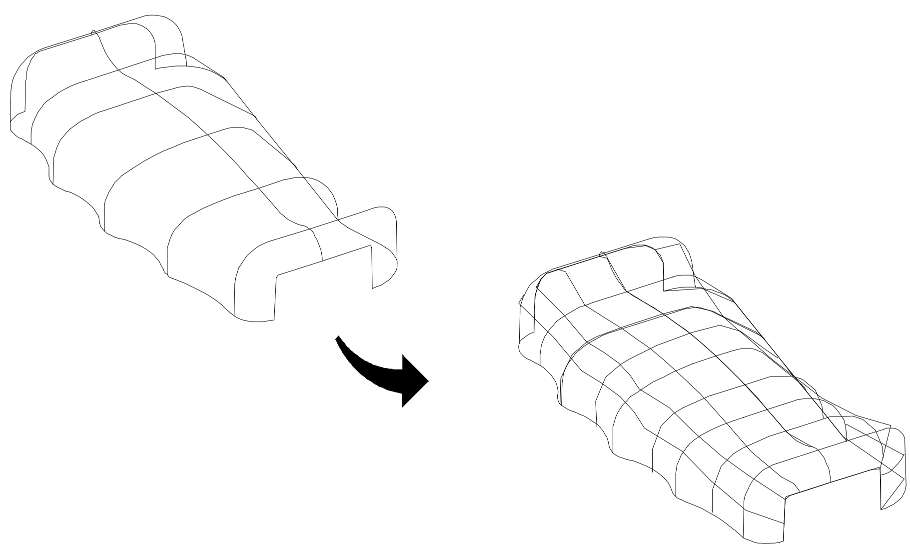


Coons

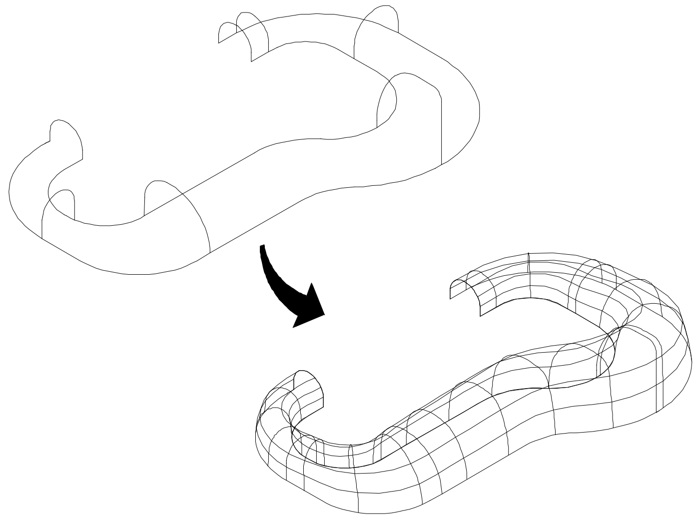
The Coons surface function creates a surface from a grid of curves. You can create this surface type by choosing **Main Menu, Create, Surface, Coons**.



Single Patch Coons



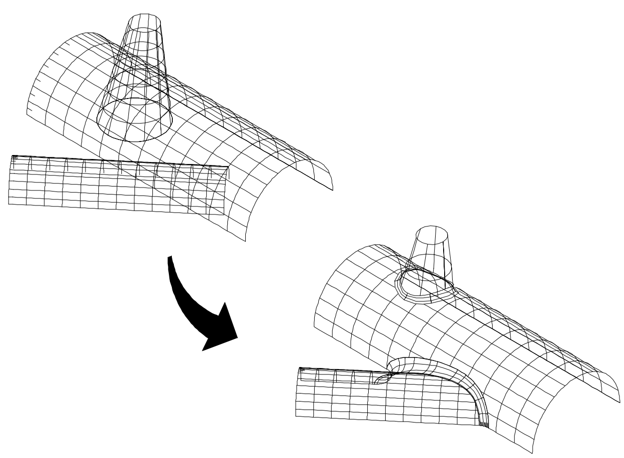
Multiple Patch Coons



Multiple Patch Coons

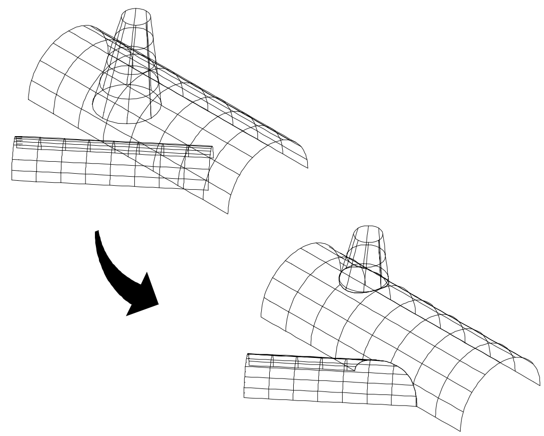
Fillet

The Fillet surface function creates a fillet surface, which is mathematically equivalent to a series of arcs and is tangent to one or two surfaces based on the construction method you choose. Fillet surfaces can be created between a plane and a surface, between a curve and a surface, or between two surfaces. You can create this surface type by choosing **Main Menu, Create, Surface, Fillet**. The following picture shows a surface to surface fillet example.



Trim, To surfaces

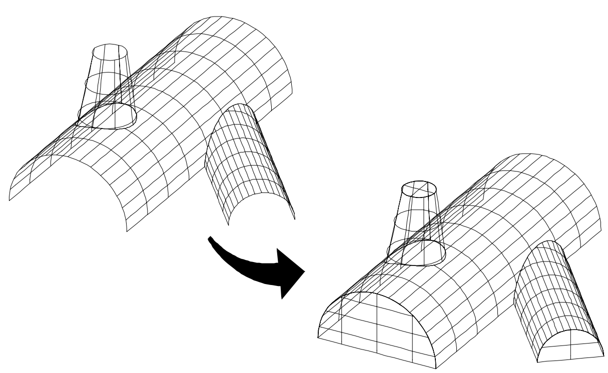
The Trim, To surfaces surface function trims surfaces to each other. You can create this surface type by choosing **Main Menu, Create, Surface, Trim/extend, To surfaces**.



Trim, Flat boundary

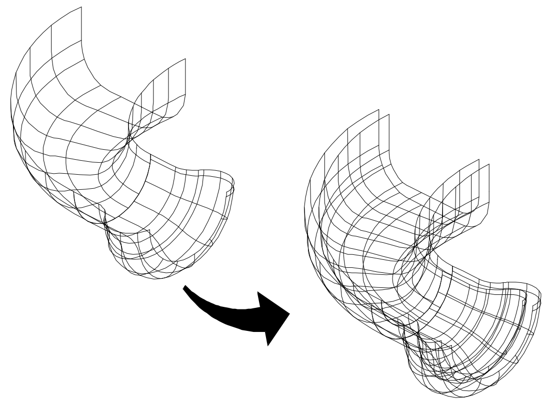
The Trim, Flat boundary surface function creates a flat, trimmed surface from one or more planar sets of curves. You can use this surface function to cap the ends of existing surfaces if the wireframe geometry that defines the surface edge exists. You can create this surface type by choosing **Main Menu, Create, Surface, Trim/extend, Flat bndy**.

Note: You can select open or closed chains of curves. If you select open chains of curves, the system prompts you to close them.



Offset

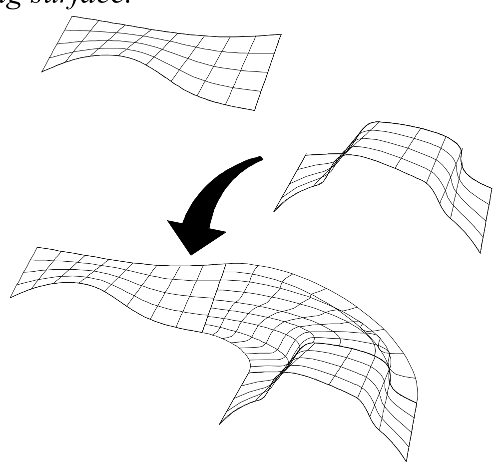
The Offset surface function creates a surface in which the offset surface is a fixed distance from an existing surface. You can create this surface type by choosing **Main Menu, Create, Surface, Offset**.



2 Surface Blend

The 2 Surface Blend surface function creates a blended surface between two existing surfaces. This surface type is shown in the following example. You can create this surface type by choosing **Main Menu, Create, Surface, Next menu, 2 surf blnd**.

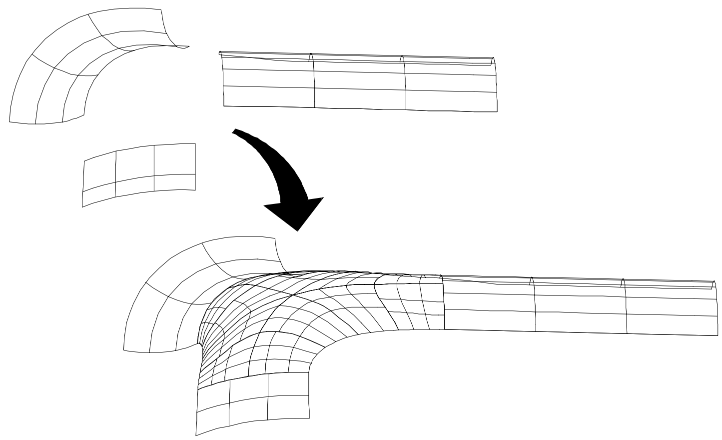
Note: The blend direction and position you set for each selected surface affects the resulting surface.



3 Surface Blend

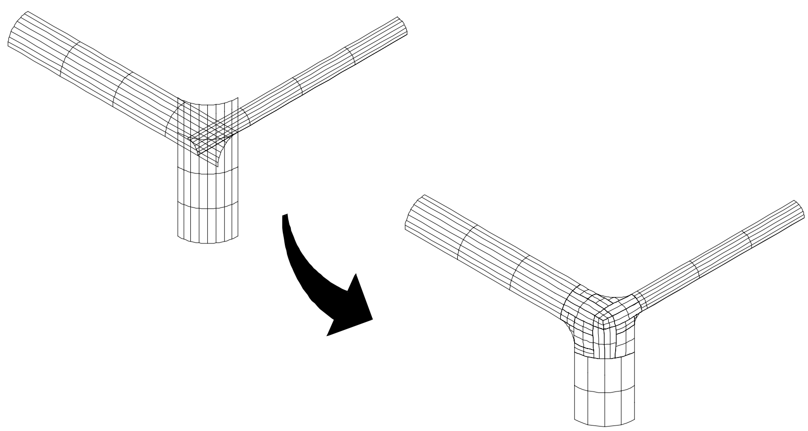
The 3 Surface Blend surface function creates a blended surface between three existing surfaces. You can create this surface type by choosing **Main Menu, Create, Surface, Next menu, 3 surf blnd**.

Note: The blend direction and position you set for each selected surface affects the resulting surface.



Fillet Blend

The Fillet Blend surface function blends three intersecting fillet surfaces to create one or more blend surfaces. You can create this surface type by choosing **Main Menu, Create, Surface, Next menu, Fillet blnd**.



Now that you've been introduced to different surface types and their applications, you're ready to create a part with surfaces and machine it with surface toolpaths. The next chapter will show you how to do both.

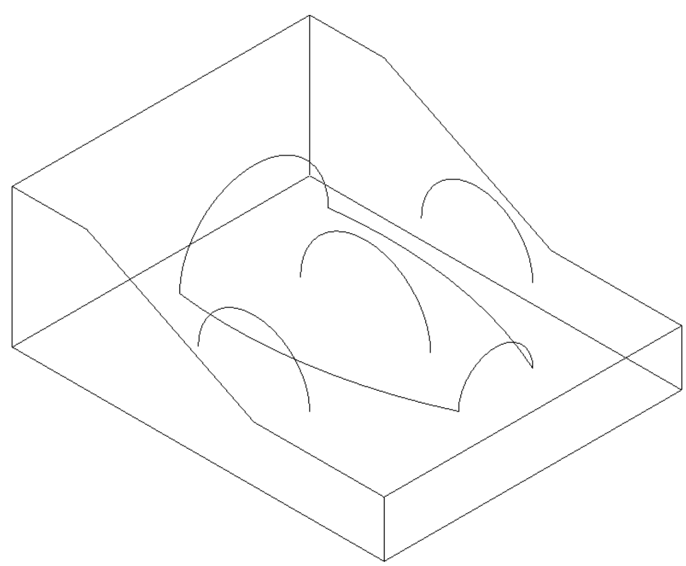
13 Creating and Machining Surfaces

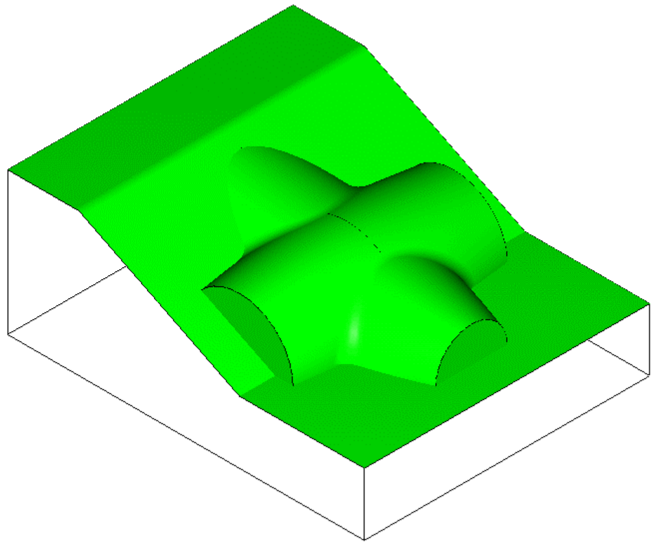
This chapter introduces you to Mastercam's surface machining capabilities. First, you will create several different kinds of surfaces. Then, you will create a number of different roughing and finishing surface toolpaths. In this chapter, you will work with the following types of surfaces:

- Ruled surfaces are created by a linear blend between several chains.
- Loft surfaces are created by a curved blend between several chains.
- Coons surfaces are created from grids of chains or curves.

Exercise 1 – Creating surfaces

In this exercise, you will open a part file that already has some wireframe geometry and add surfaces to it. The following pictures show the wireframe geometry and completed surfaces.





In this exercise, you will learn the following skills:

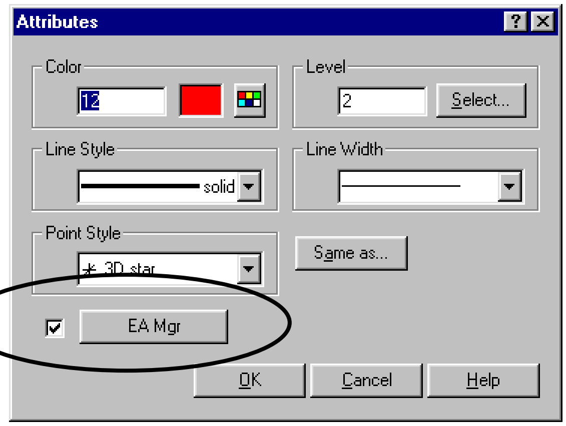
- **Defining surface attributes with the Entity Attributes Manager**
- **Creating a ruled surface**
- **Creating a loft surface**
- **Creating a Coons surface**
- **Creating surface fillets**

▶ *Setting the level and color for the new surfaces*

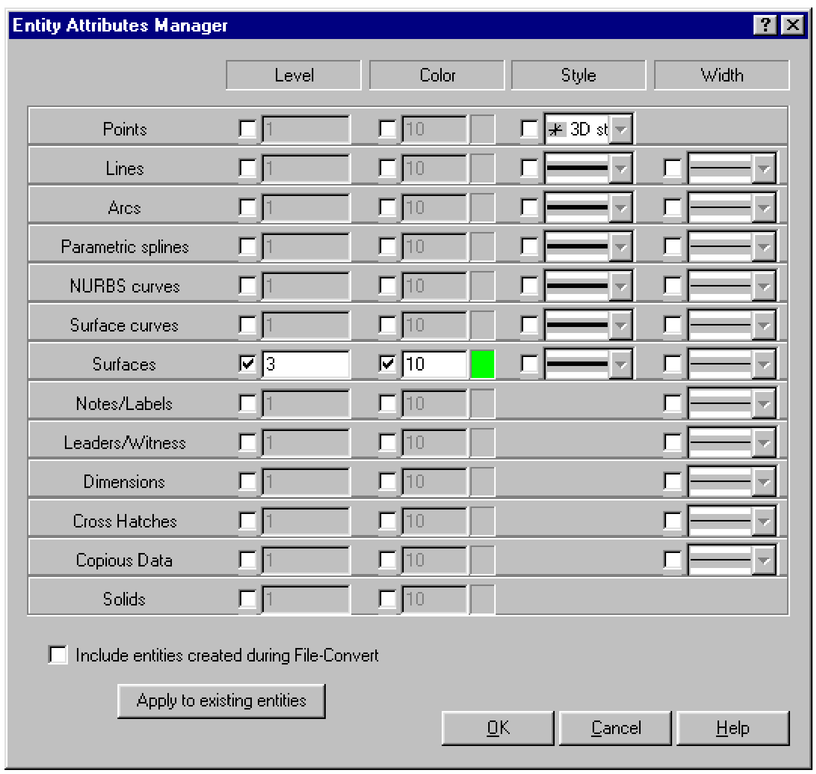
Organizing your work with levels can make working on complicated parts much easier. In this exercise, the wireframe geometry is on level 2, and you will create the surface geometry on level 3.

Use the Entity Attributes Manager to set default properties for surfaces so that when you create surfaces, they are automatically created in the proper color and placed on the proper level.

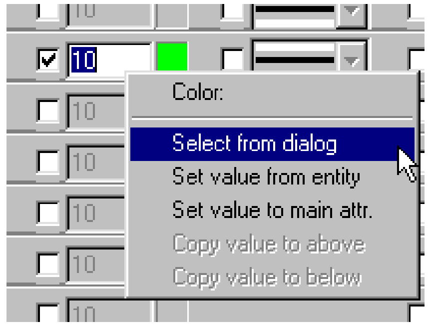
1. Open **surfaces-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Attributes** from the Secondary Menu.
3. Choose the **EA Mgr** check box and button.



4. Find the line for **Surfaces**. Select the **Level** check box and enter **3**. This means that every time you create a surface, Mastercam will place it on level 3, regardless of what the current level is.
5. Select the **Color** check box. You want the surfaces to be green (color 10), which is the default color, so you do not need to change it. Your values should match the following picture.



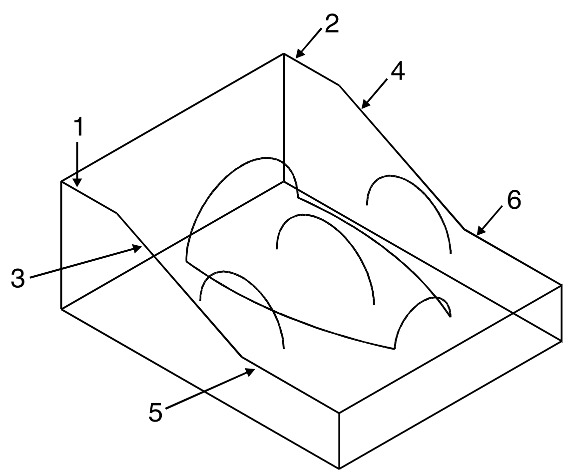
Tip: To change the default color when you don't know the number, right-click in the number field and choose **Select from dialog**. This will show you the same dialog box as when you choose the **Color** button from the Secondary Menu.



6. Choose **OK** twice.

▶ *Creating the ruled surfaces*

1. Choose **Main Menu, Create, Surface, Ruled, Single**.
2. Select the lines at position 1 and position 2.

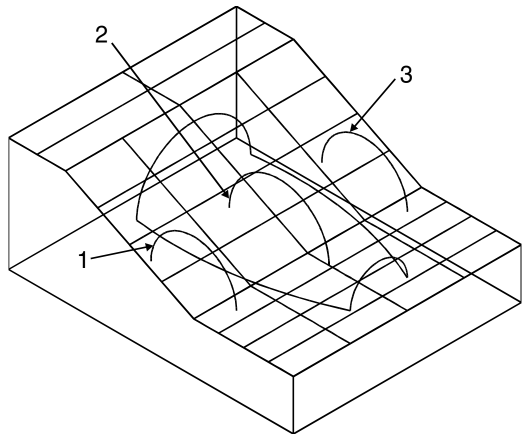


Tip: If you used the Chain option instead of Single, only one ruled surface with rounded corners would be created. This would not follow the shape of the part.

3. Choose **Done, Do it**.
4. Repeat steps 2 and 3 for positions 3 and 4. (Before selecting the lines, choose **Single** to make sure you are using Single chaining.)
5. Repeat for positions 5 and 6. This creates a total of three ruled surfaces.

▶ *Creating the loft surface*

1. Choose **Backup, Loft**.
2. Select the arcs at positions 1, 2, and 3 in that order.

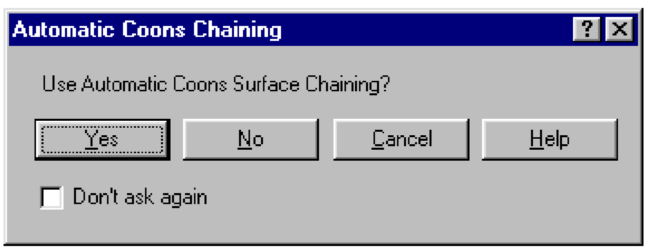


Tip: A ruled surface would not work for this geometry because it would create sharp corners in the middle of the surface. A Coons surface would not work because the sections are not connected.

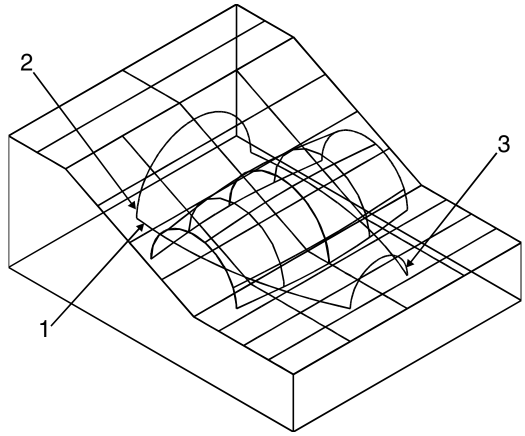
3. Choose **Done, Do it**.

▶ *Creating the Coons surface*

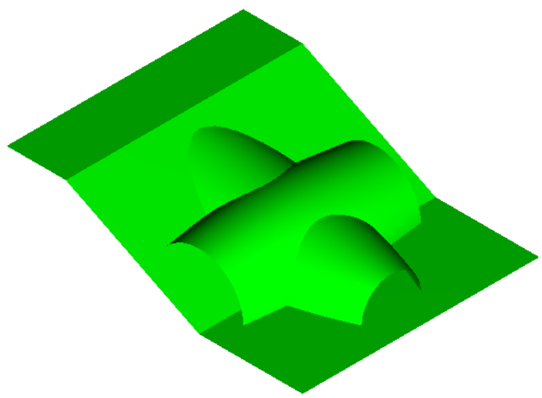
1. Choose **Backup, Coons**.
2. Choose **Yes** when you see the following message:



3. Select at positions 1 and 2.



4. Select at position 3.
5. Choose **Do it.**
6. Choose **Backup**.
7. Press [Alt + S] to see a shaded view of the surfaces. The surfaces should look like the following picture.

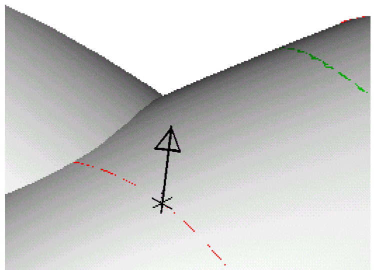


Note: Shading the surfaces makes selection easier when creating surface fillets.

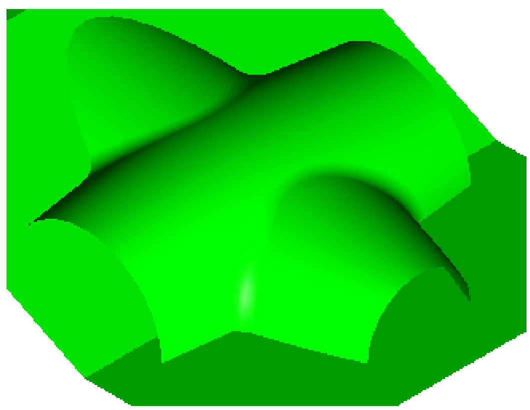
▶ *Creating surface fillets between the loft and Coons surfaces*

1. Choose **Main Menu, Create, Surface, Fillet, Surf/surf**.
2. Select the loft surface.
3. Choose **Done**.

4. Select the Coons surface.
5. Choose **Done**.
6. Enter a radius of **6**
7. Choose **Check norms, Cycle**.
8. The surface normal (represented by the arrow) should point out as shown in the following picture. If it does not, choose **Flip** from the menu. When it is correct, choose **OK**.

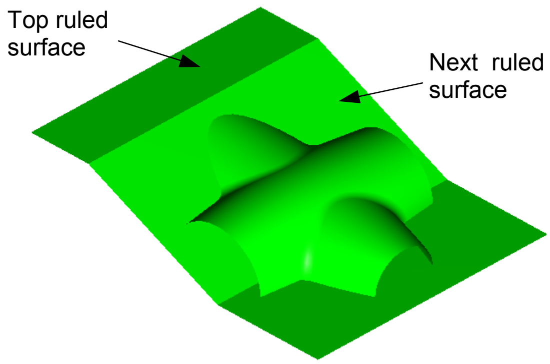


9. Repeat step 8 for the next normal.
10. Choose **Do it**. The fillets should look like the following picture.



▶ *Creating surface fillets on two of the ruled surfaces*

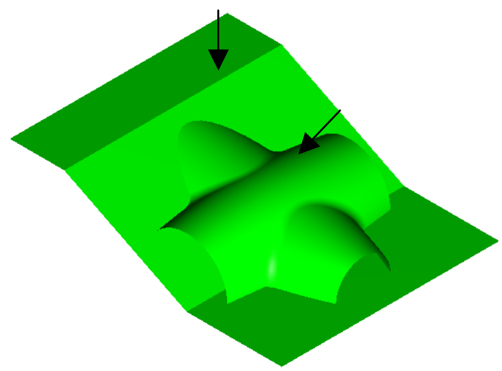
1. Choose **Surfaces**.
2. Select the top ruled surface.
3. Choose **Done**.
4. Select the next ruled surface.
5. Choose **Done**.



6. Enter a radius of **6.**
7. Set the **Trim** option to **Y**.

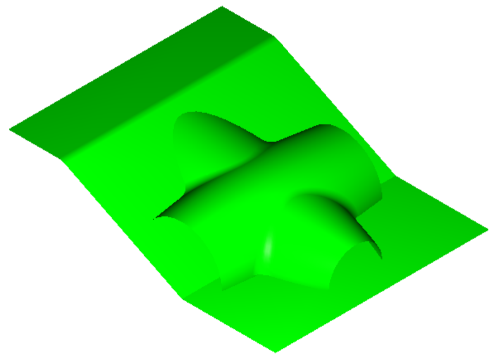
Tip: The surface normals must point inside the part. The arrows for the normals should match the following picture.

8. Choose
   - **Check norms**
   - **Cycle**
   - **Flip**
   - **OK**
   - **Flip**
   - **OK**
   - **Do it**



9. Choose **Main Menu, File, Save** and save the file in your working folder as **surfaces1.mc9**.

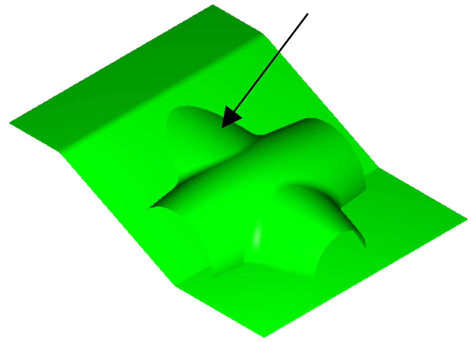
The part should look like the following picture.



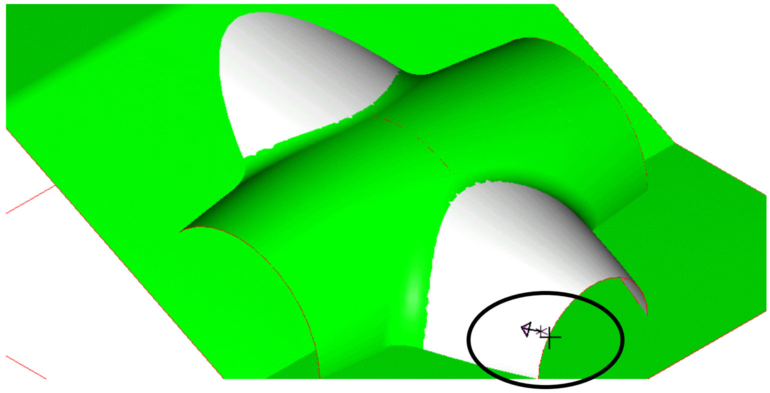
Note: The next procedure is optional—capping the surfaces only makes the part look better. It does not change the toolpath.

▶ *Capping the ends of the surfaces*

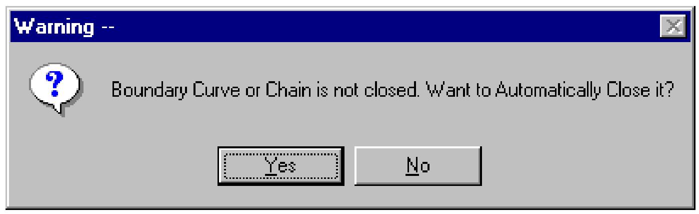
1. Choose
   - **Main Menu**
   - **Create**
   - **Surface**
   - **Trim/extend**
   - **Flat bndy**
   - **Manual**
2. Select the Coons surface.



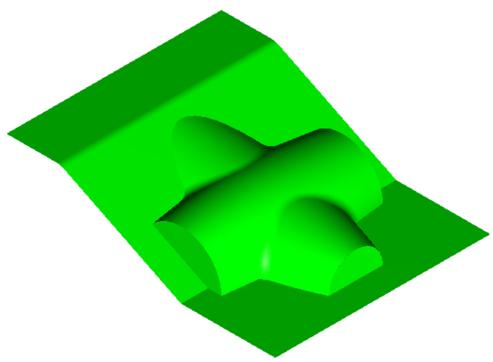
3. Drag the arrow cursor to the edge of the surface and click once as shown in the following picture.



4. Choose **End here, Do it**.
5. When you see the following message, choose **Yes**.



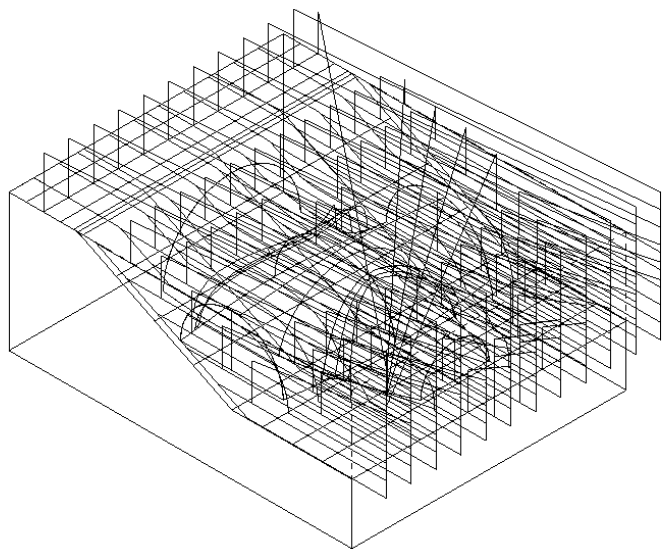
6. Choose **Manual**, and repeat steps 2 through 5 for the two ends of the loft surface. The part should look like the following picture.



7. Press [Alt + A] to save the file.

Exercise 2 – Creating a rough parallel toolpath

The rough parallel toolpath removes the bulk of the material quickly. Using a flat endmill instead of a ball endmill also speeds up the material removal. This cutting method does not work well on parts with multiple bosses because the toolpath involves too much plunging. Parallel roughing is the most efficient roughing toolpath for this particular part. The completed toolpath should look like the following picture.



Note: The surfaces do not have to be trimmed in order to be machined. Mastercam automatically cuts only the highest surfaces.

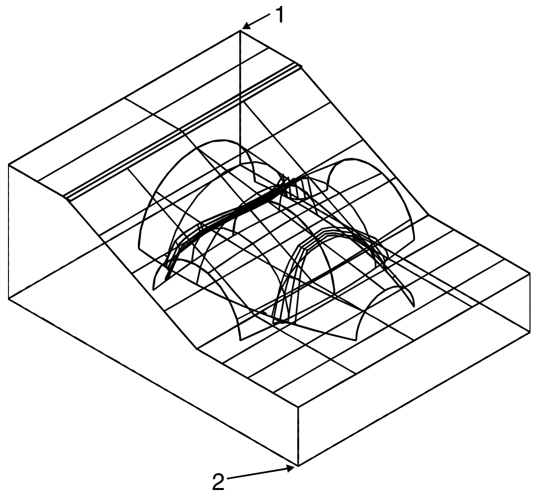
This exercise shows you the following skills:

- **Creating a rough parallel toolpath**
- **Using cutting direction**
- **Using cutting depths**

▶ *Defining the stock boundaries*

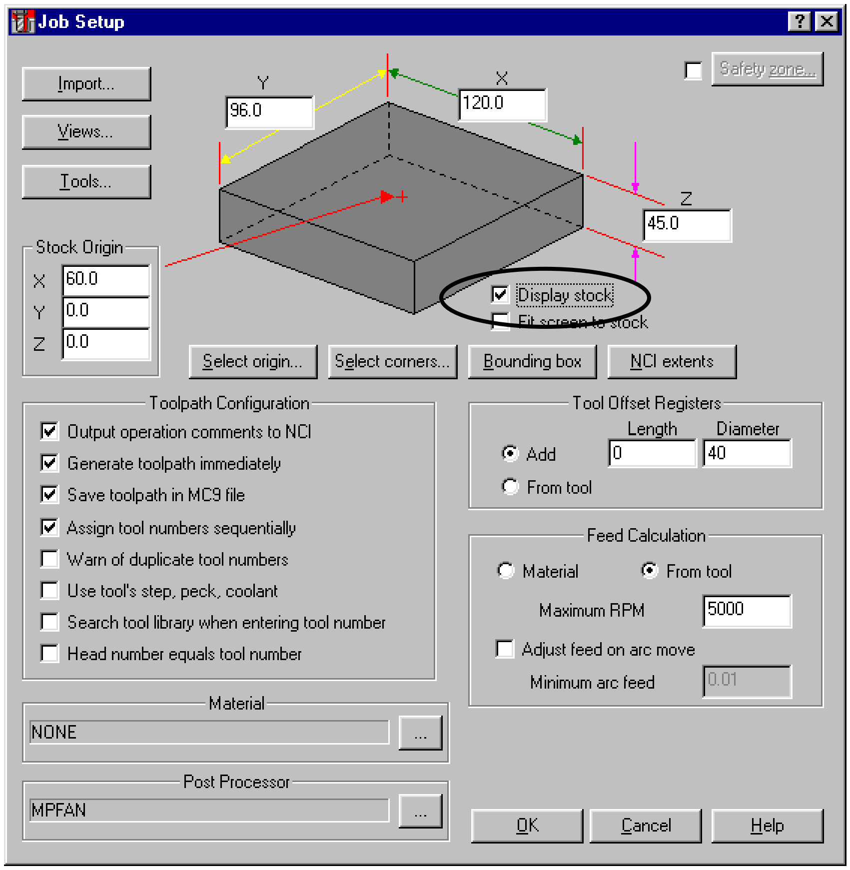
1. Press [Alt + S] to turn off the shading on the part.
2. Choose **Main Menu, Toolpaths, Job Setup**.
3. Choose the **Select corners** button.

4. Select the geometry at position 1 and position 2.



Tip: Setting the stock limits is not necessary, but allows for more accurate toolpath verification.

5. Select the **Display stock** check box.

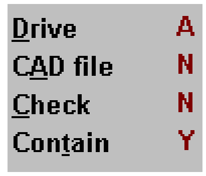


6. Choose **OK**.

▶ *Selecting the surfaces and surface parameters*

1. Choose **Surface.**
2. Toggle the **Drive** setting to **A**. This tells Mastercam you want to machine all the surfaces.
3. Toggle **Contain** to **Y**. This tells Mastercam that you want to use a tool containment boundary to limit the tool's motion.

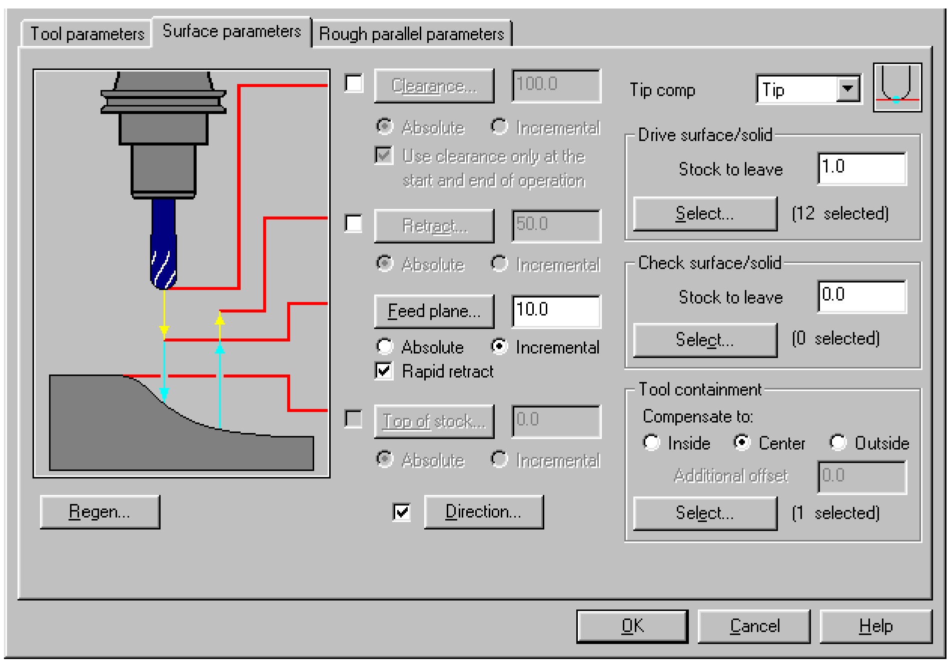
   Your surface selection menu options should match the following picture.



Tip: **Drive** surfaces are surfaces that will be machined.
Check surfaces are surfaces that Mastercam will avoid.
Tool containment is geometry that serves as a "fence," setting limits for the tool motion.
Choose **CAD file** to create a toolpath based on an external CAD file, instead of geometry in the current Mastercam file.

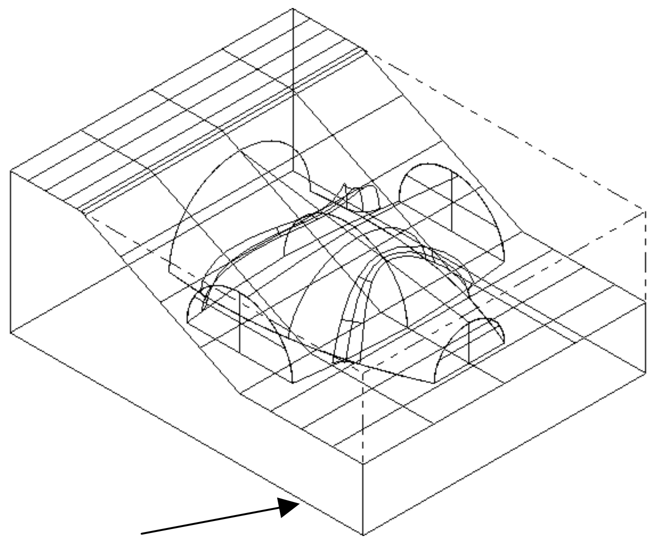
4. Choose **Rough, Parallel, Boss**.
5. Select the 12 mm flat endmill.

   Note: All the tools you will need in this chapter have been saved with the part. You do not need to get them from the tool library.
6. Select the **Surface parameters** tab.
7. Enter the values as shown on the following dialog box.

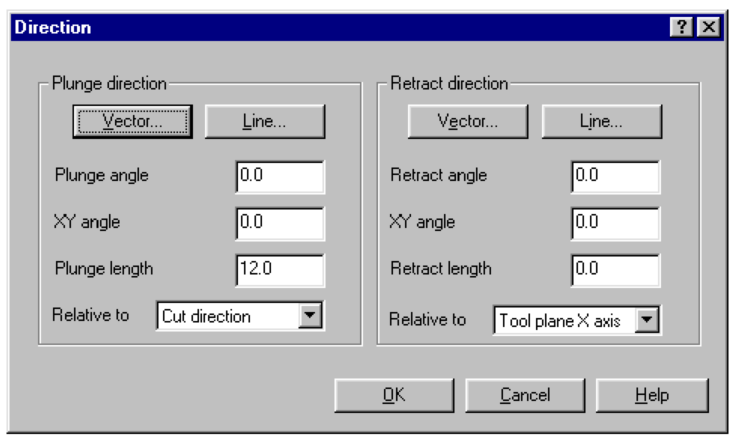


8. To determine the tool containment boundary, choose the **Select** button in the **Tool containment** section.

9. Choose **Chain, Options**.
10. Select the **Plane mask** option and choose **OK**.
11. Select the bottom of the part as shown in the following picture.



12. Choose **Done** to return to the Surface parameters dialog box.
13. Choose the **Direction** check box and button.
14. Enter the values shown on the following dialog box.

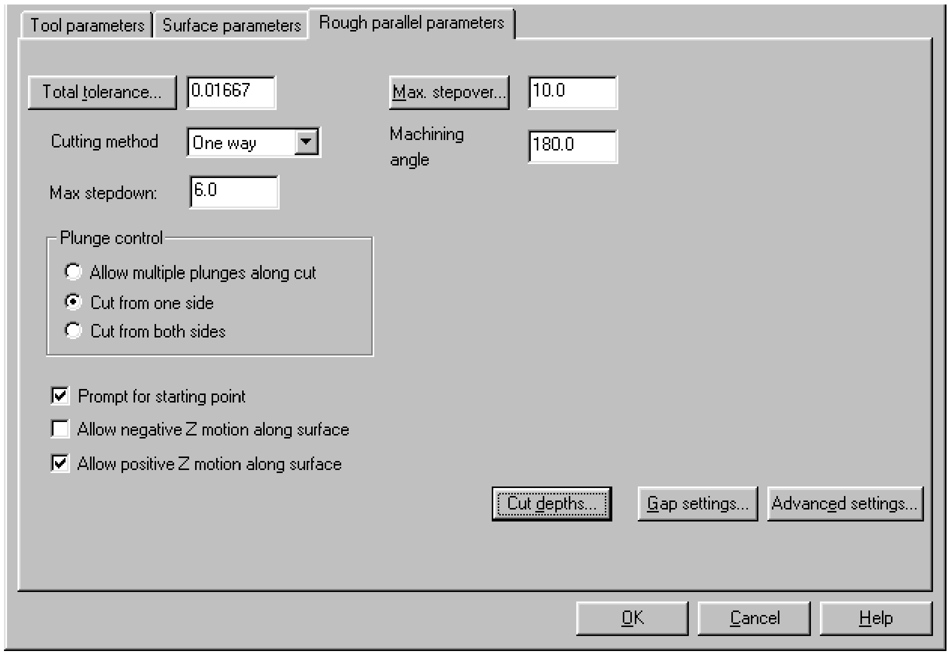


Tip: Setting a plunge length in the Direction dialog box allows the tool to plunge off the part.

15. Choose **OK**.

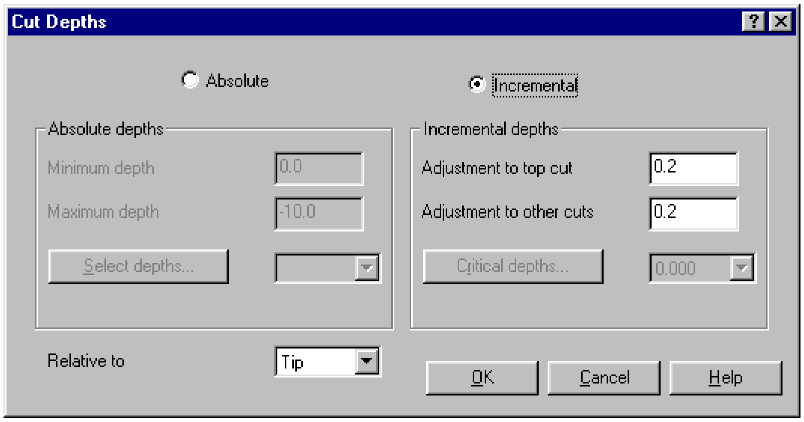
▶ *Entering the roughing parameters*

1. Select the **Rough parallel parameters** tab.
2. Enter the values shown on the following dialog box.



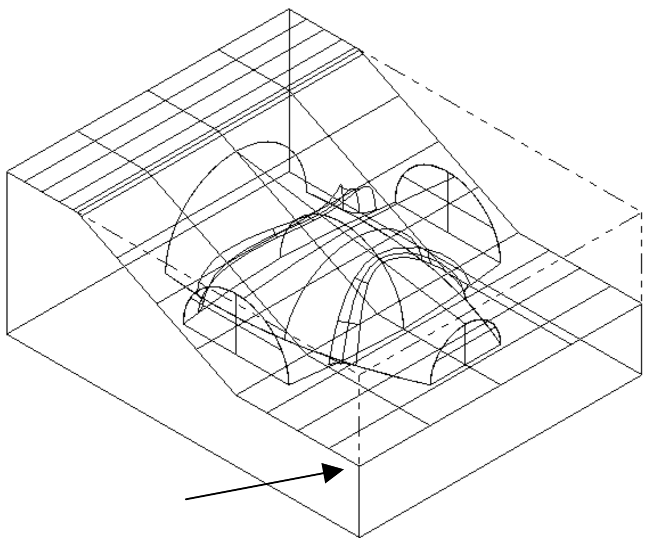
Note: Selecting only the Allow positive Z motion along surface option limits the tool motion and prevents the tool from plunging into the material.

3. Choose the **Cut depths** button.
4. Enter the values shown on the following dialog box.

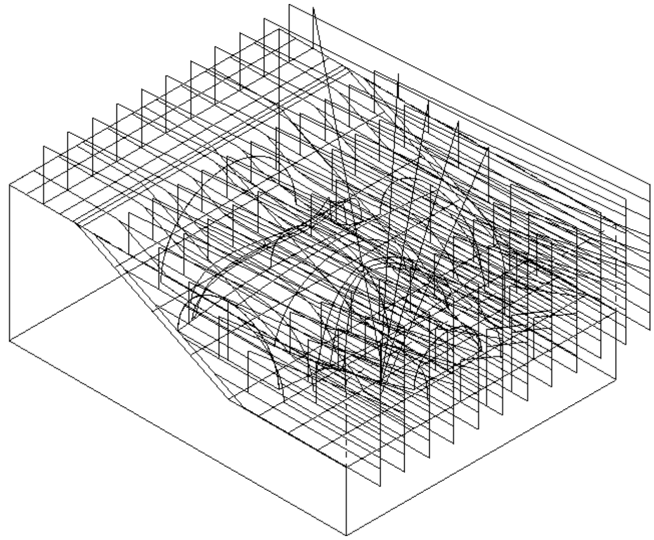


Tip: The adjustment to top cut option sets how far below the top of the surface the first cut lies. The adjustment to other cuts option sets how far above the bottom the last cut lies.

5. Choose **OK** twice.
6. Mastercam prompts you to select the starting point. Select near the front corner of the part as shown in the following picture.



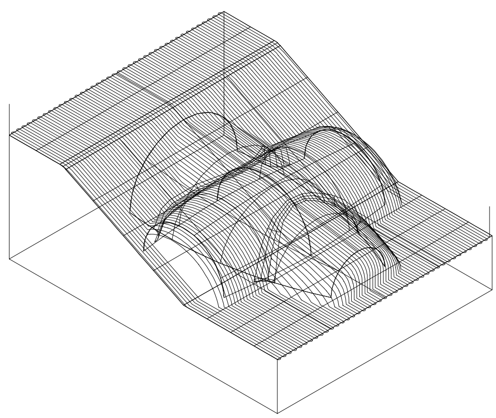
Mastercam generates the toolpath, which should look like the following picture.



7. Press [Alt + T] to clear the toolpath display from the screen.

Exercise 3 – Creating a finish parallel toolpath

Using a finish parallel toolpath allows Mastercam to machine over all the surfaces of this part. Parallel finishing is the most efficient choice for this part. The completed toolpath should look like the following picture.

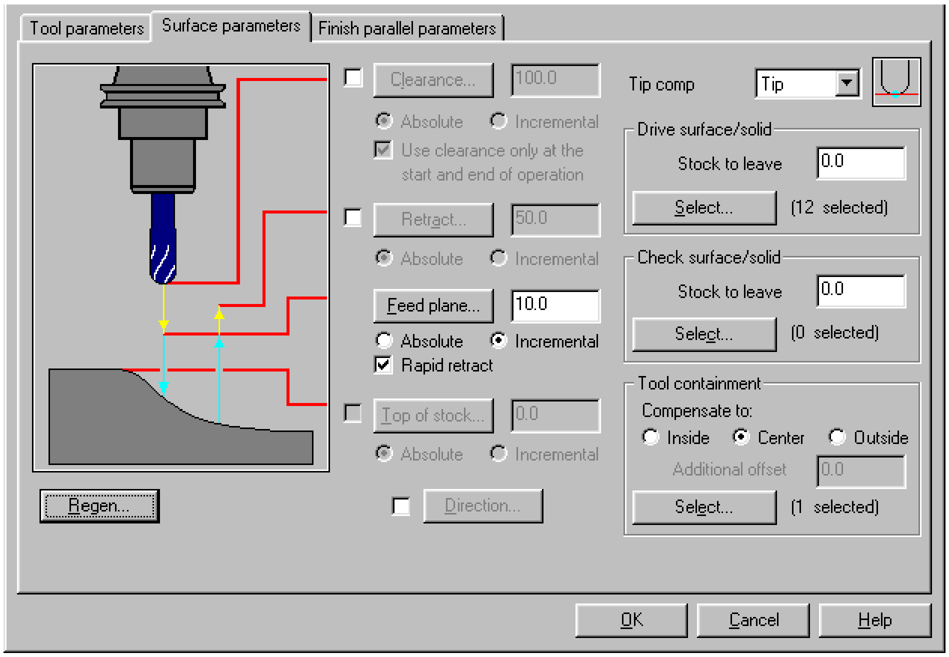


This exercise shows you the following skills:

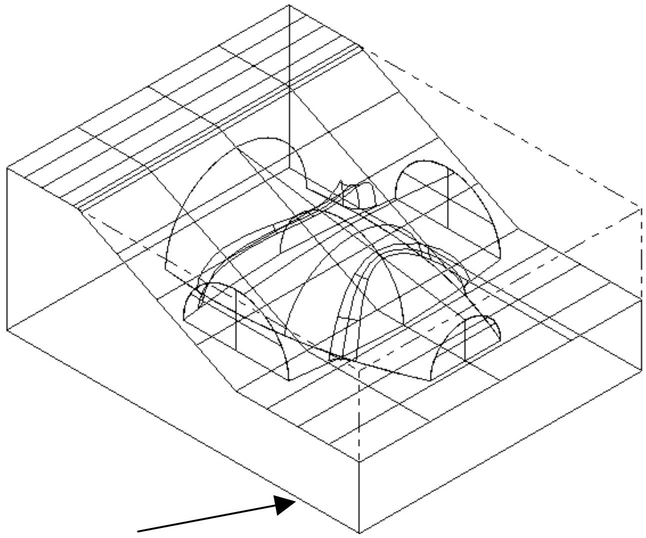
- **Creating a finish parallel toolpath**
- **Setting filter and tolerance values**
- **Using gap settings to reduce processing time**

▶ *Selecting the surface parameters*

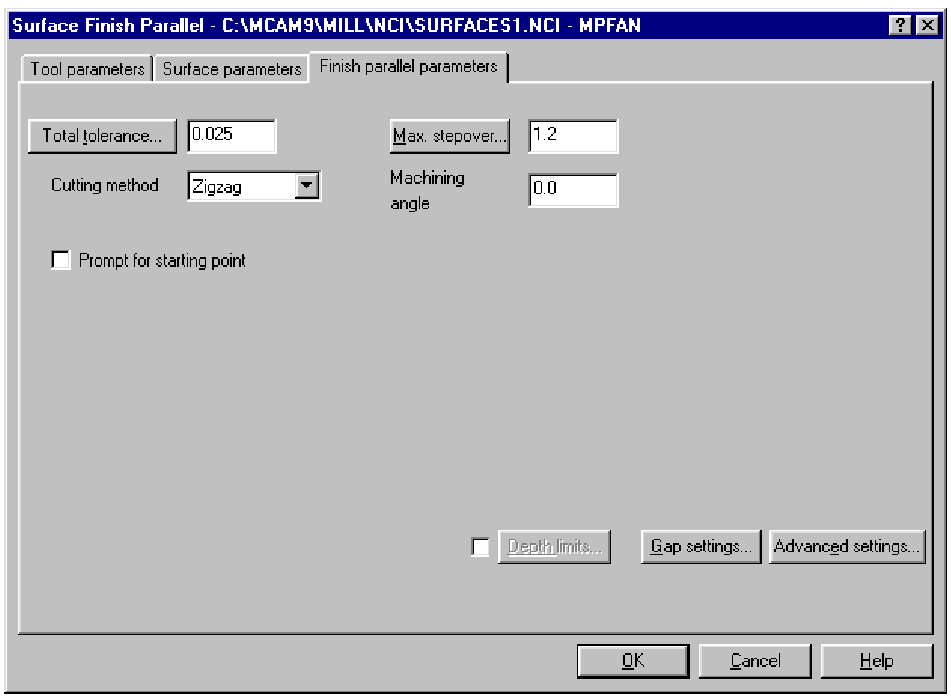
1. Choose **Main Menu, Toolpaths, Surface, Finish, Parallel**.
2. Select the 12 mm ball endmill.
3. Select the **Surface parameters** tab.
4. Enter the values as shown on the following dialog box.



5. Choose the **Select** button in the **Tool containment** section.
6. Select the bottom of the part as shown in the following picture.



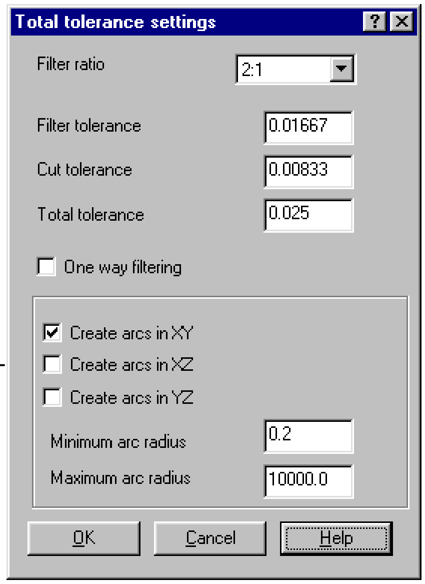
7. Choose **Done**.
8. Select the **Finish parallel parameters** tab.
9. Enter the values shown on the following dialog box.



10. Choose the **Total tolerance** button.
11. Choose a **Filter ratio** of **2:1**.
12. Set the **Total tolerance** to **0.025**. Your other values should match the following dialog box.

Z-plane arcs
Some controls aren't capable of machining arcs in the XZ or YZ planes. Check the documentation for your control before selecting these options.

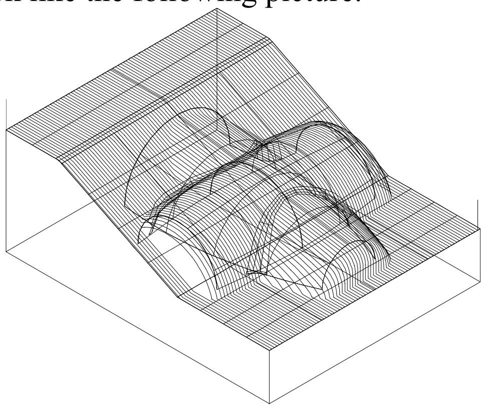
Also, verify that your post-processor is configured to handle XZ and YZ arcs.



Tip: The filter settings can reduce the size of the NC program. Collinear and nearly collinear moves (within the specified tolerance) are removed and arcs are inserted when possible to reduce the toolpath size.

Tip: The filter tolerance should be set to at least twice the cut tolerance. The filter ratio does this automatically.

13. Choose **OK** twice. Mastercam generates the toolpath, which should look like the following picture.

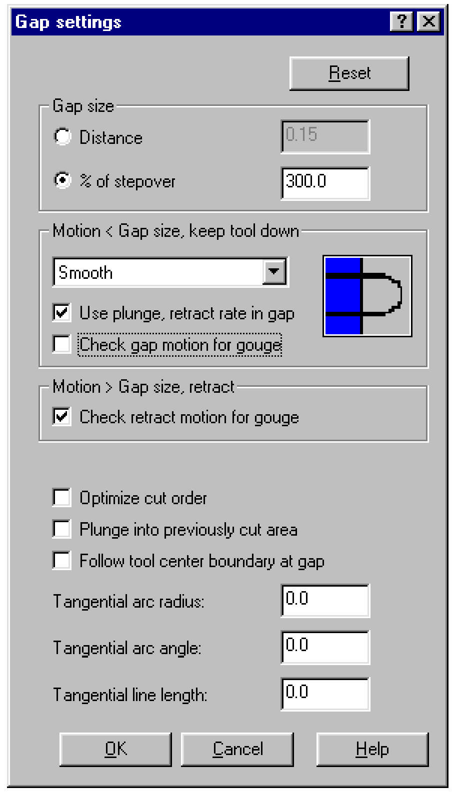


Notice how long this toolpath takes to process. The next procedure shows you how to reduce the processing time by adjusting the gap settings.

▶ *Changing the gap settings*

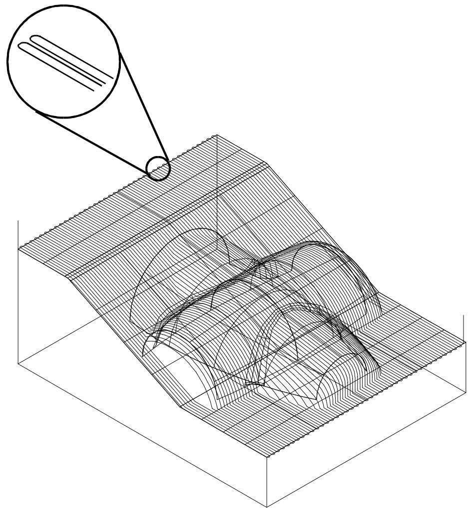
1. Press [Alt + O] to open the Operations Manager.
2. Choose the **Parameters** icon for the Surface Finish Parallel toolpath.
3. Choose the **Finish parallel parameters** tab.
4. Choose the **Gap settings** button.
5. Change the **Motion** to **Smooth**.
6. Clear the **Check gap motion for gouge** check box.

Tip: Setting the gap motion to Smooth creates smooth tool motion between passes. And since the tool motion between passes is on a flat plane, there is no need to check the gap motion for gouges. This setting reduces the time needed to process the toolpath.



7. Choose **OK** twice.
8. Choose **Regen Path**.

Mastercam regenerates the toolpath, which should look like the following picture. You should notice a reduction in the processing speed and smooth motion between the passes of the toolpath.



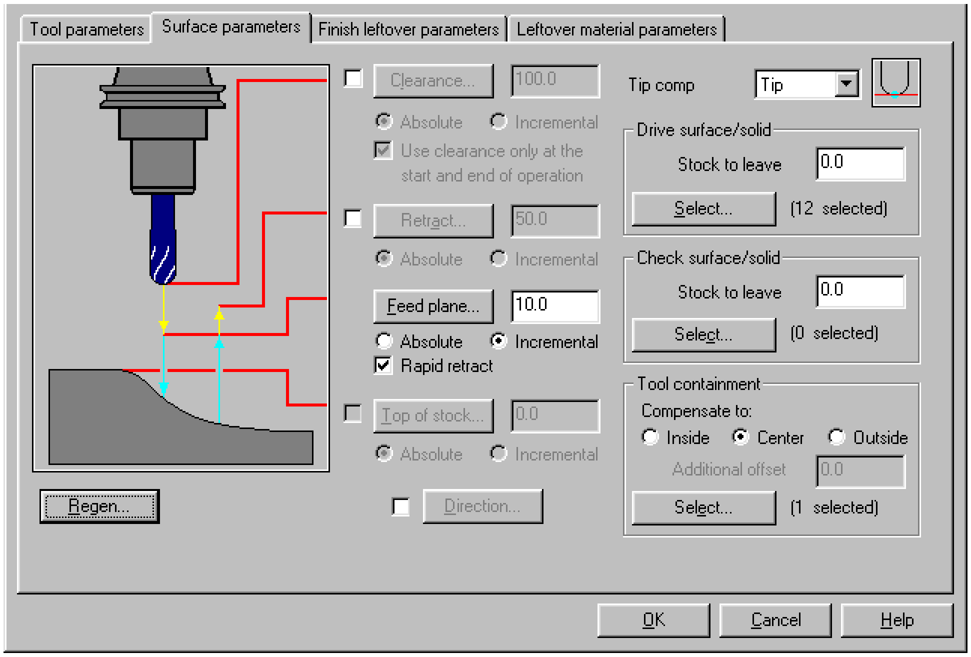
Exercise 4 – Creating a finish leftover toolpath

The finish leftover toolpath removes material left behind by the larger tool of the finish parallel toolpath. It also adjusts to different Z depths, unlike a restmill toolpath, which makes planar cuts at constant Z depths and is more appropriate for roughing operations. In this exercise, you will learn the following skill:

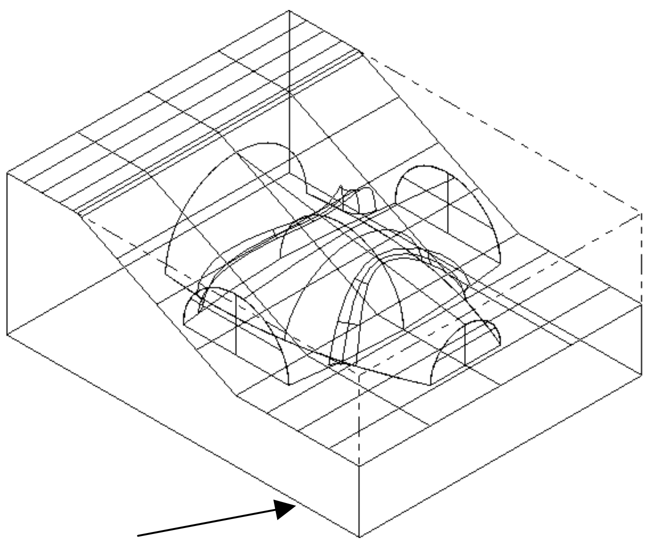
- **Creating a finish leftover toolpath**

▶ *Creating the finish leftover toolpath*

1. Right-click in the Operations Manager and choose **Toolpaths, Surface finish, Leftover**.
2. Select the 5 mm ball endmill.
3. Select the **Surface parameters** tab.
4. Enter the values shown on the following dialog box.

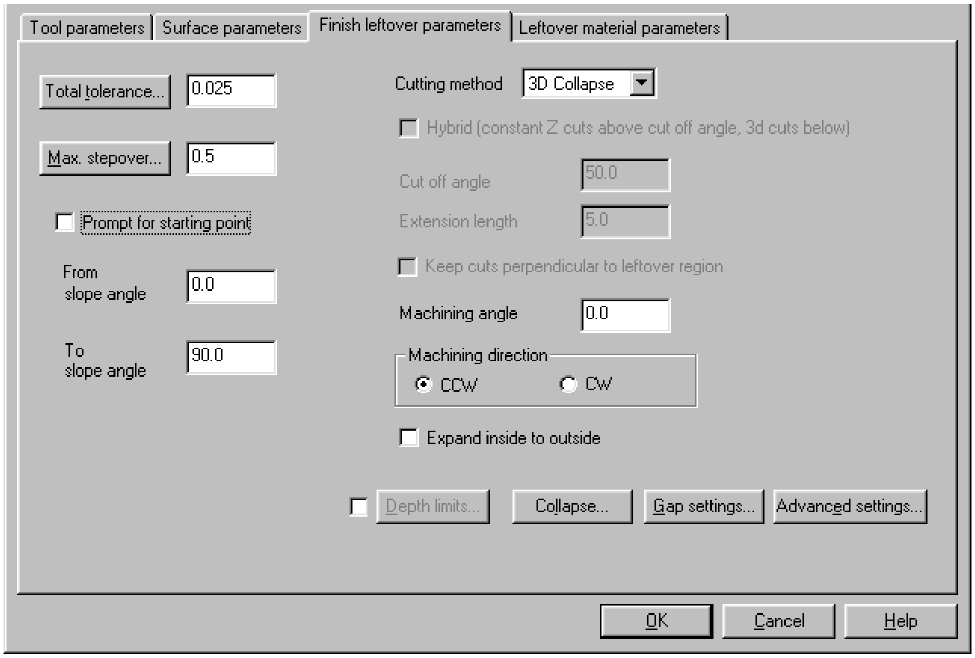


5. Choose the **Select** button in the **Tool containment** section.
6. Select the bottom of the part as shown in the following picture.

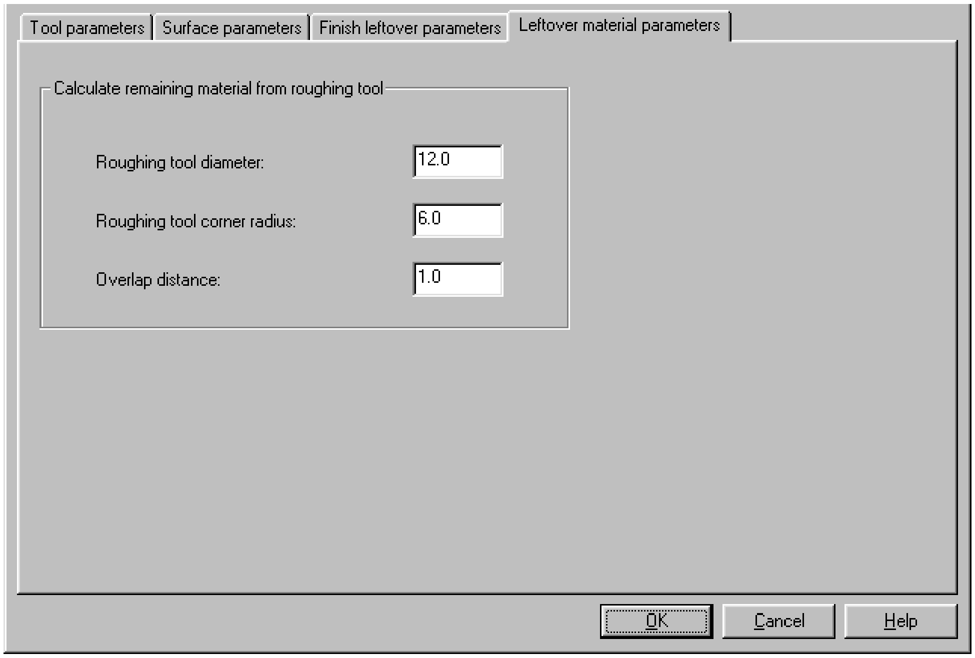


7. Choose **Done**.
8. Select the **Finish leftover parameters** tab.

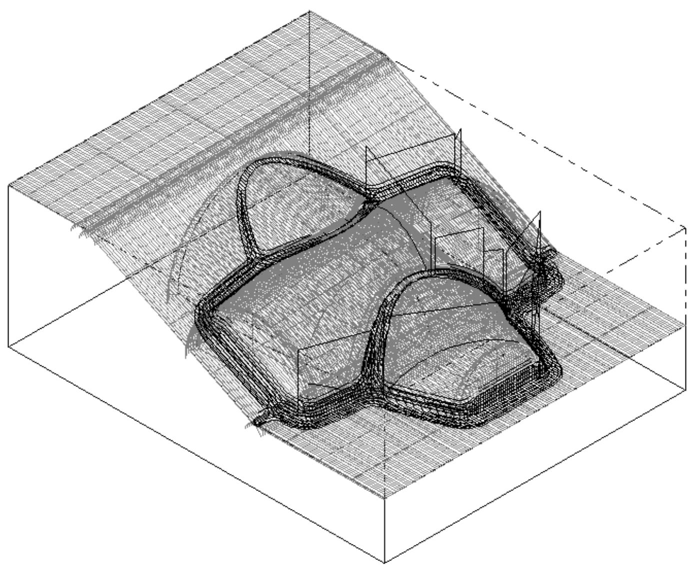
9. Enter the values shown on the following dialog box.



10. Select the **Leftover material parameters** tab.
11. Enter the values shown on the following dialog box.



12. Choose **OK**. Mastercam generates the toolpath, which should look like the following picture. It can take several minutes to generate the toolpath.



*Note: If you receive an error message stating that the toolpath allocation is too low, choose **Main Menu, Screen, Configure** and select the **Allocations** tab. Increase the value for the **Toolpath allocation in Kbytes** option and choose **OK**. Finally, regenerate the operation. This increases the amount of RAM designated for toolpath functions.*

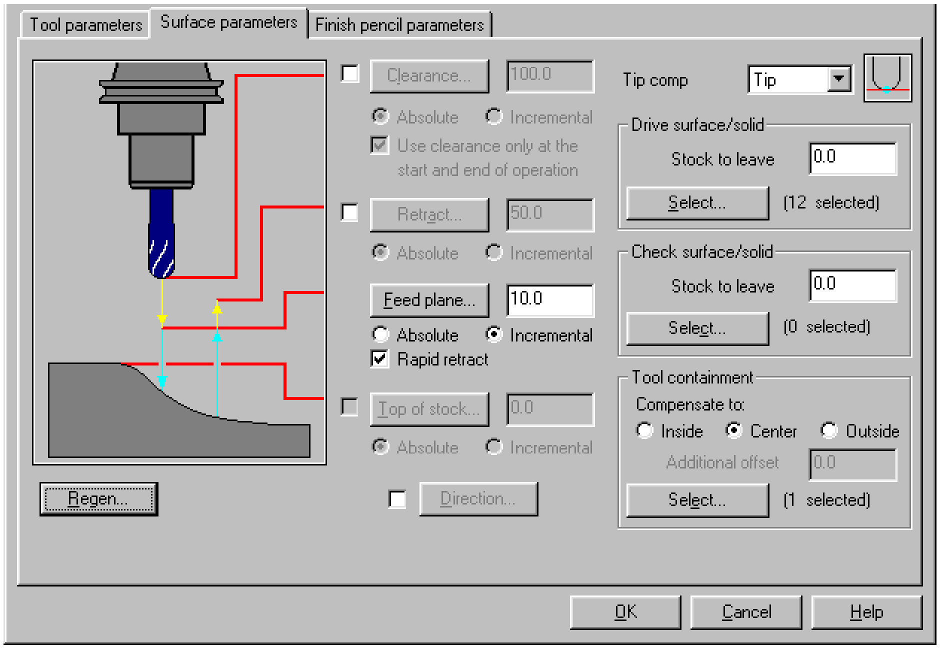
Exercise 5 – Creating a finish pencil toolpath

On this geometry, the finish pencil toolpath cleans up more of the material by driving the cutter tangent to two surfaces at a time. This exercise shows you the following skills:

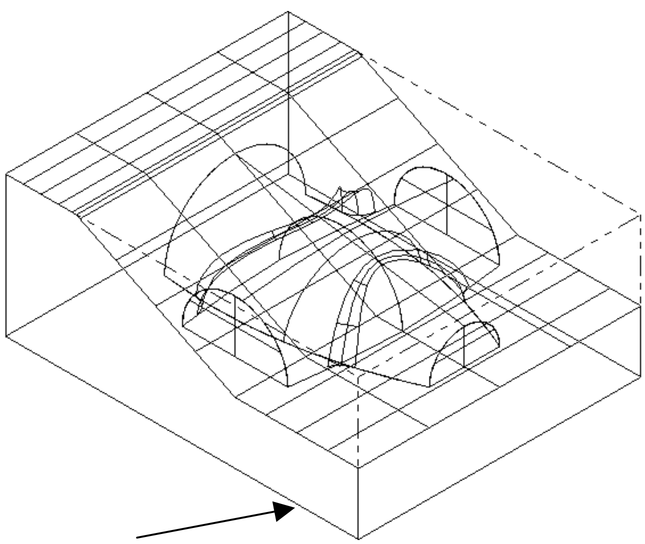
- **Creating a finish pencil toolpath**
- **Verifying the toolpath**

▶ *Creating the finish pencil toolpath*

1. Right-click in the Operations Manager and choose **Toolpaths, Surface finish, Pencil**.
2. Select the 2 mm ball endmill.
3. Select the **Surface parameters** tab.
4. Enter the values shown on the following dialog box.

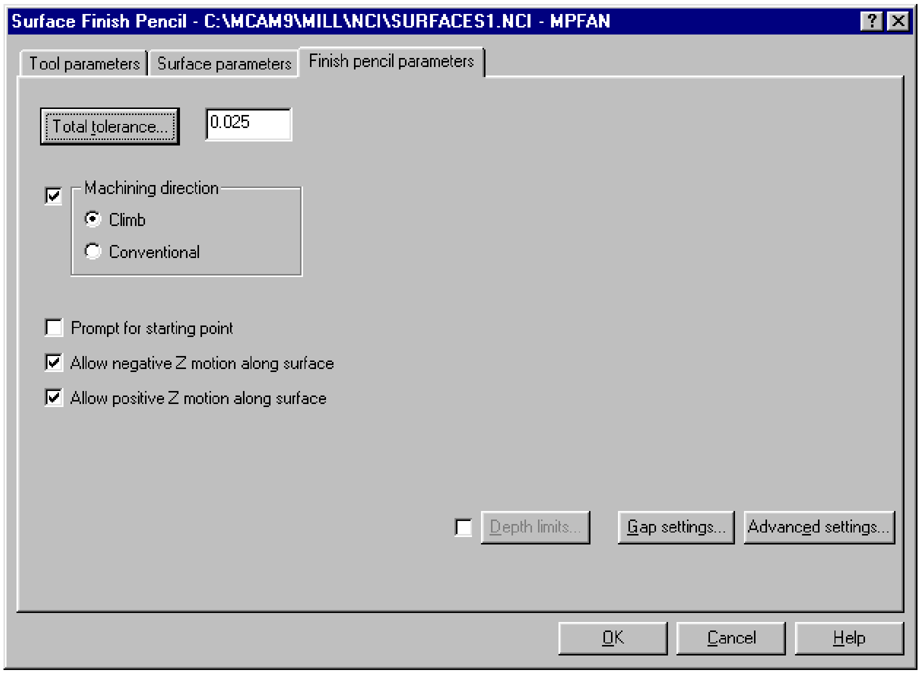


5. Choose the **Select** button in the **Tool containment** section.
6. Select the bottom of the part as shown in the following picture.

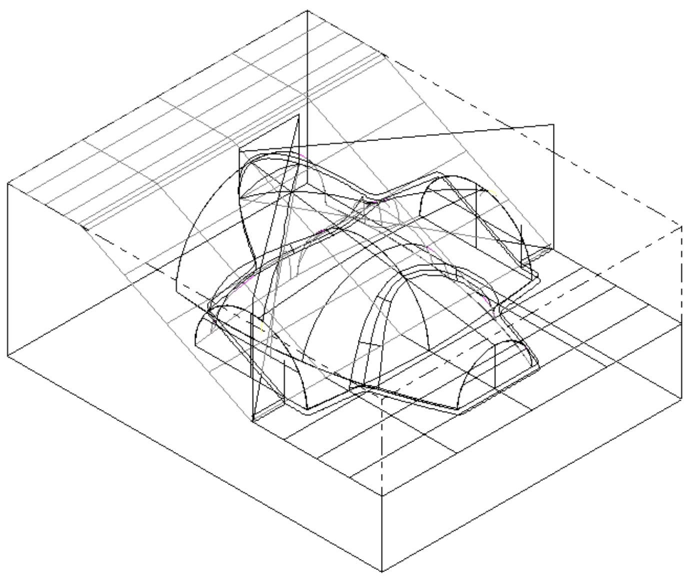


7. Choose **Done**.
8. Select the **Finish pencil parameters** tab.

9. Enter the values shown on the following dialog box.



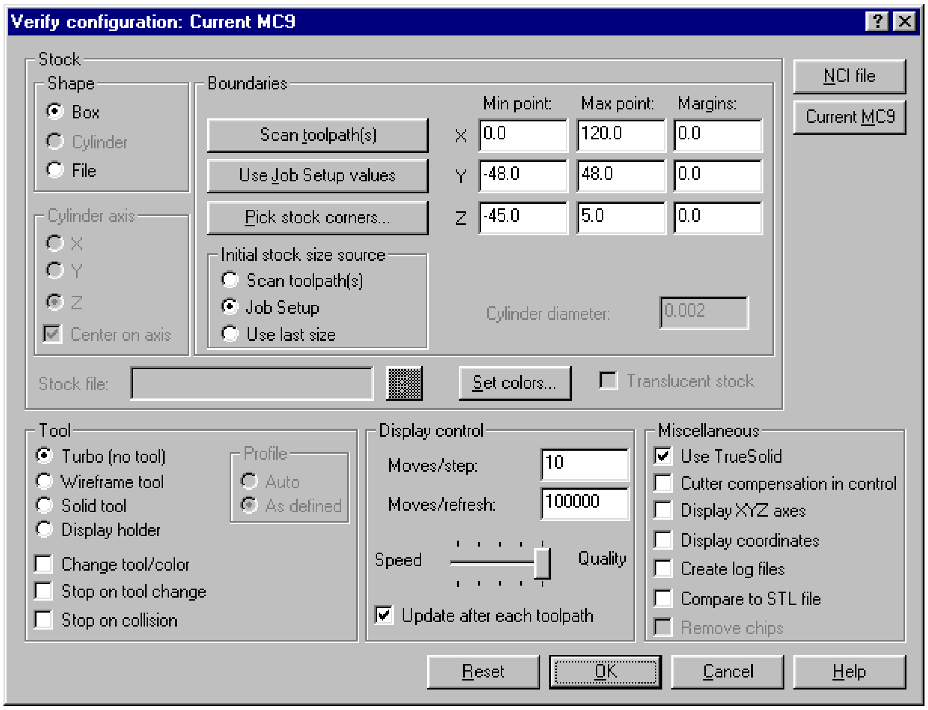
10. Choose **OK**. Mastercam generates the toolpath, which should look like the following picture.



▶ *Verifying all the surface toolpaths*

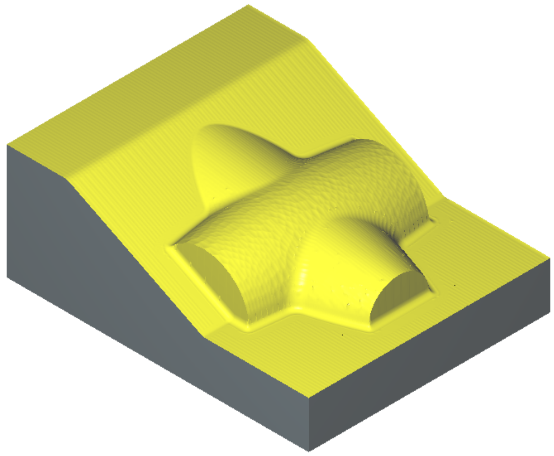
You can verify the toolpaths to see the stock removal. Verifying all the toolpaths can take a number of minutes, depending on the speed of your computer.

1. If necessary, press [Alt + O] to return to the Operations Manager.
2. Choose **Select All, Verify**.
3. Choose the **Configure** button on the Verify toolbar.
4. Enter the values shown on the following dialog box and choose **OK.**



5. Choose the **Machine** button on the Verify toolbar.

   Mastercam runs through the toolpaths and displays the verification results, which should look like the following picture.



6. Choose the **Close** button on the Verify toolbar to return to the Operations Manager.
7. Choose **OK** to close the Operations Manager.
8. Press [Alt + A] to save the file.

Now that you have experience with creating surface toolpaths, the next two chapters will show you more types of surface toolpaths and their applications.

14 Surface Roughing

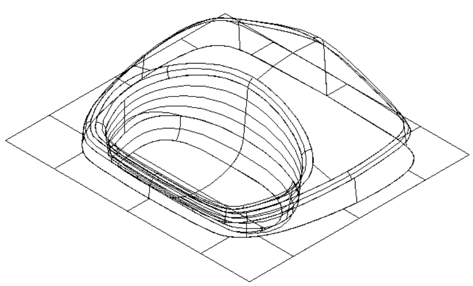
In the previous chapter, you roughed a part with a rough parallel toolpath. This chapter focuses on more roughing toolpaths that you can use for surface machining, including:

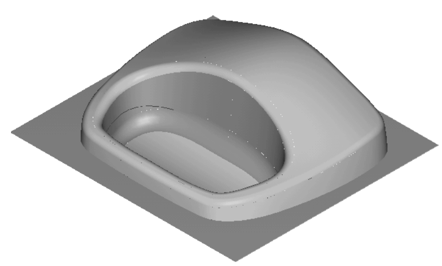
- rough pocket
- rough plunge
- restmill
- high-speed rough pocket

Mastercam also includes rough flowline, contour, and radial toolpaths. These are the same as finish flowline, contour, and radial toolpaths except that the roughing toolpaths allow multiple Z cuts. Finish flowline, contour, and radial toolpaths are discussed in the next chapter.

Exercise 1 – Creating a rough pocket toolpath

Rough pocket toolpaths remove a lot of stock quickly and prepare the part for the finish toolpath. Another benefit of using a rough pocket toolpath on a part is that you can start the toolpath at a point off the part and prevent the tool from plunging into the material. A rough pocket toolpath also creates a series of planar cuts (or constant Z), which is the preferred cutting method for most roughing tools. You will use the following part.





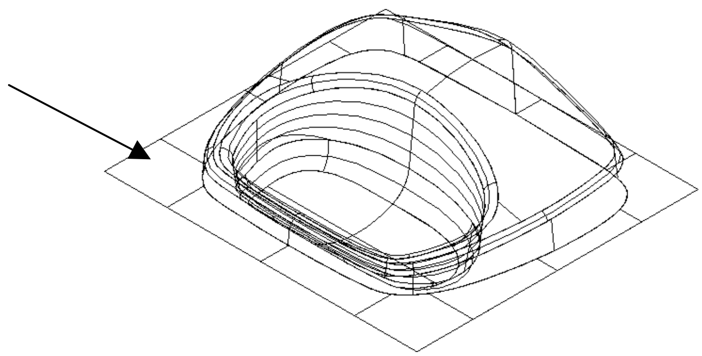
This exercise shows you the following skills:

- **Creating a rough pocket toolpath**
- **Creating a tool containment boundary**

Creating the tool containment boundary

In the previous chapter, you selected geometry that Mastercam used as a tool containment boundary. Since the tool containment boundary has to be wireframe geometry, and this part has only surfaces, you will create a wireframe boundary that you can use for the tool containment.

1. Open the file **rough pocket-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Create, Curve, All edges**.
3. Select the flat surface as shown in the following picture.



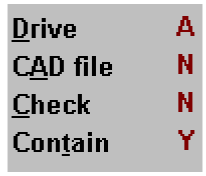
4. Choose **Done, Do it**. Mastercam creates lines around the outer border and curves around the inner border.

   Note: You might need to repaint the screen to see the new geometry. Choose the  *button on the toolbar.*

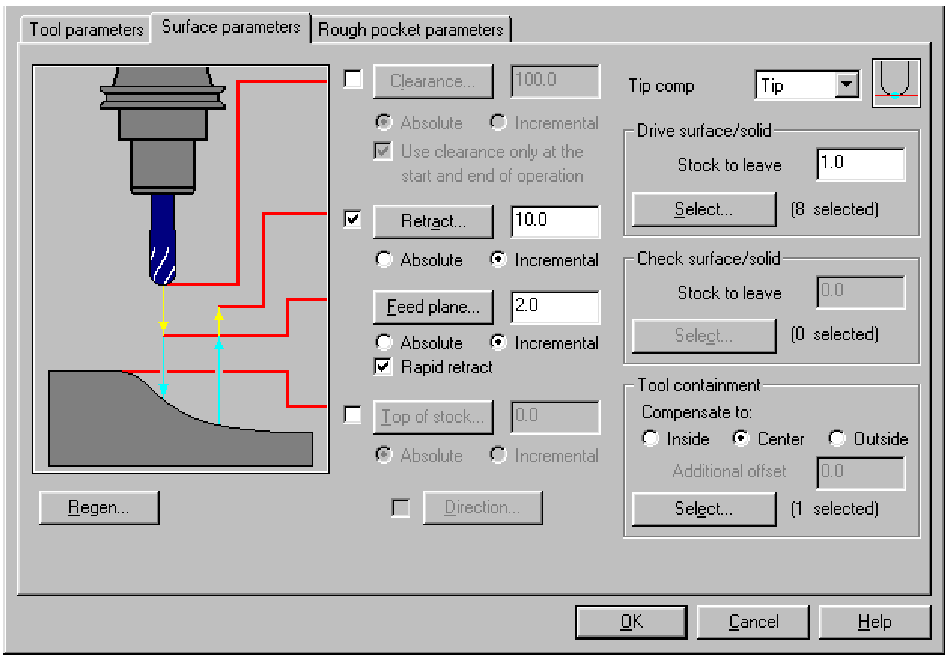
Selecting the surface parameters

1. Choose **Main Menu, Toolpaths, Surface**.
2. Toggle the **Drive** setting to **A**. This tells Mastercam that you want to machine all the surfaces.
3. Toggle **Contain** to **Y**.

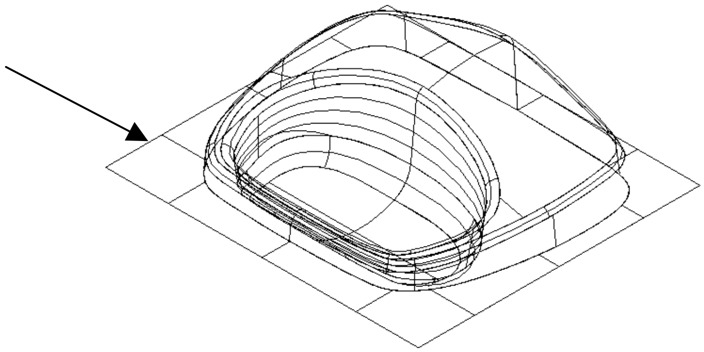
Your surface selection menu options should match the following picture.



4. Choose **Rough, Pocket**.
5. Right-click in the tool display area and select the 10 mm HSS flat endmill from the tool library.
6. Select the **Surface parameters** tab.
7. Enter the values as shown on the following dialog box.



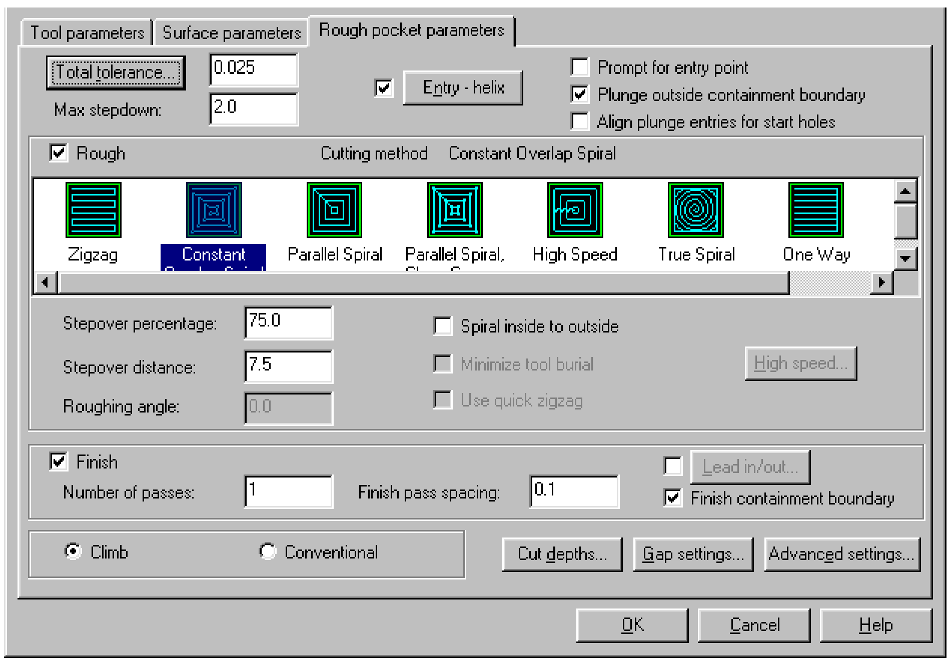
8. Choose the **Select** button in the **Tool containment** section.
9. Select the outer boundary of the part as shown in the following picture.



10. Choose **Done** to return to the Surface parameters dialog box.

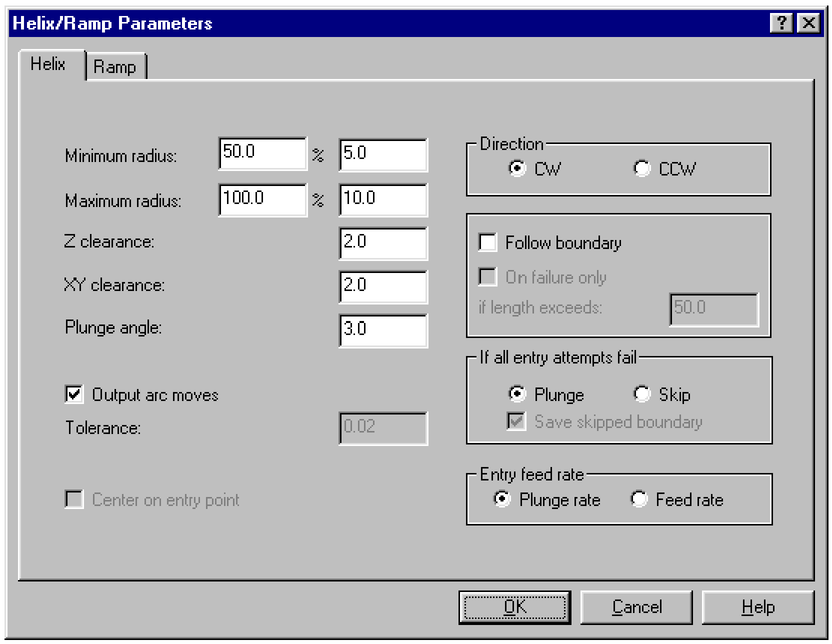
▶ *Entering the roughing parameters*

1. Select the **Rough pocket parameters** tab.
2. Enter the values shown on the following dialog box.



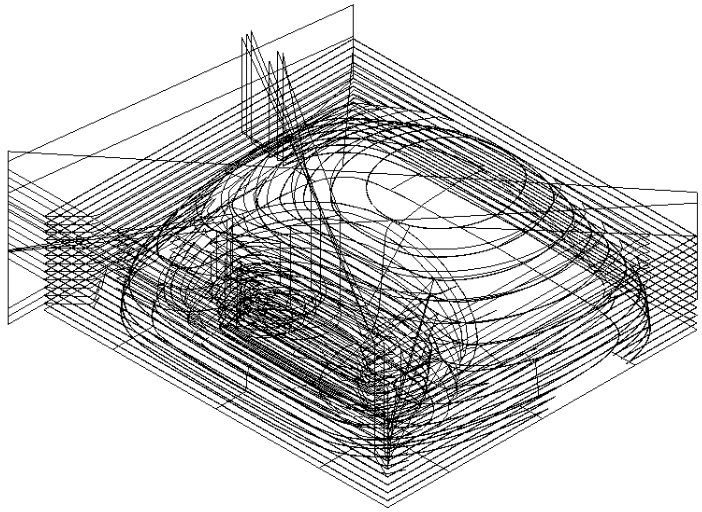
Note: The selections in the previous picture instruct Mastercam to both ***Plunge outside the tool center boundary*** *and to create an* ***Entry – helix*** *move. Mastercam will use a helical entry only where it is not possible to plunge outside the part—in the case of this part, when it machines the hollow on the front of the part.*

3. Choose the **Entry – helix** (or **Entry – Ramp**) check box and button.
4. Choose the **Helix** tab.
5. Enter the values shown in the following dialog box.



6. Choose **OK** twice.

   Mastercam generates the toolpath, which should look like the following picture.



Even though this was a pocket toolpath, it also roughs the outside of the part in addition to the pocket on the front of the part. Instead of using straight linear cuts like you did in the previous chapter, this toolpath uses the Constant Overlap Spiral cutting method which more closely approximates the part contour for a more effective roughing operation.

7. Save the file in your working folder as **rough spiral.mc9**.

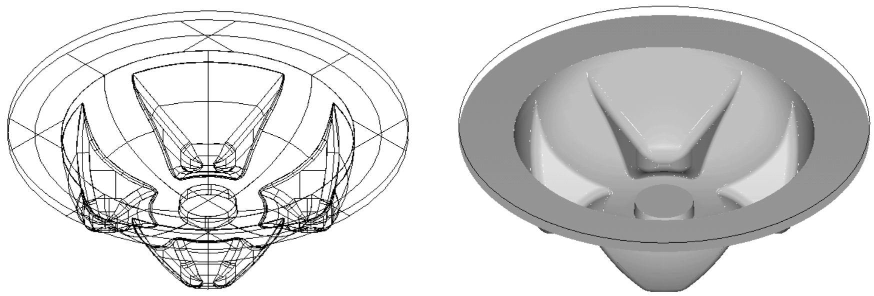
Exercise 2 – Creating a rough plunge toolpath

Rough plunge toolpaths rough a part quickly using a drilling-type motion. Shops that use these toolpaths often invest in special end cutting tools that have a flat bottom to remove stock quickly but can move coolant through the center of the tool to remove chips. Plunge roughing is an appropriate toolpath for deep cavities.

Mastercam gives you two techniques for creating rough plunge toolpaths:

- The zigzag method defines a rectangular grid and the tool plunges at intervals along it.
- The NCI method lets the tool plunge at intervals along a previously created toolpath.

The NCI option gives you much more control over the plunging actions, and will be shown in this exercise. The wireframe and surface geometry for the part is shown in the following picture.

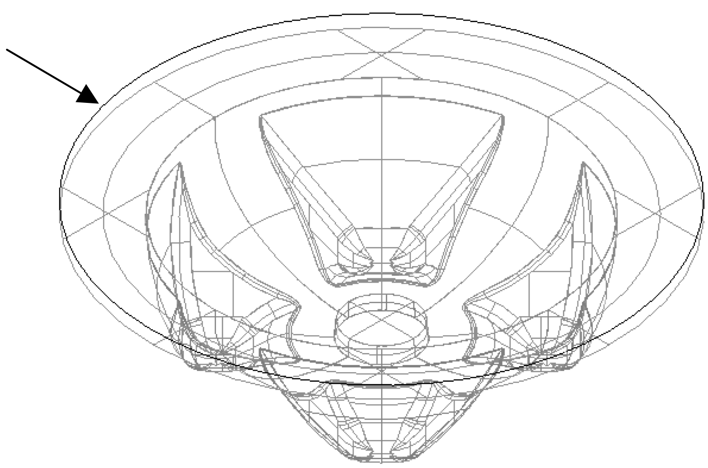


This exercise shows you the following skills:

- **Creating a rough plunge toolpath using an NCI file**
- **Using absolute cut depths**
- **Using check surfaces to restrict the toolpath**

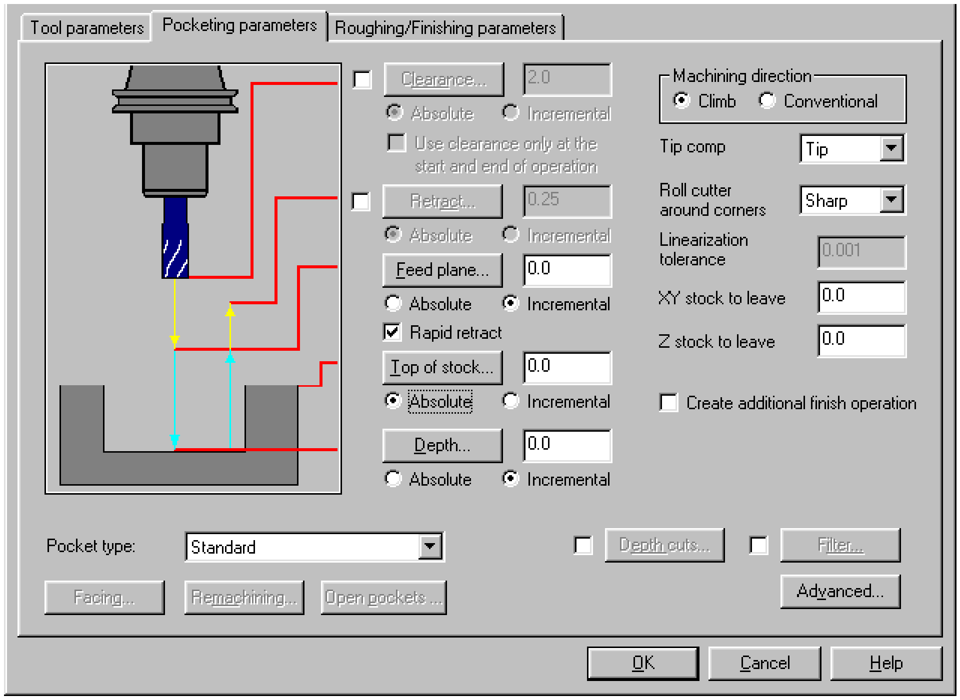
▶ *Creating the pocket toolpath*

1. Open the file **rough plunge-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Pocket**.
3. Select the green circle above the part.



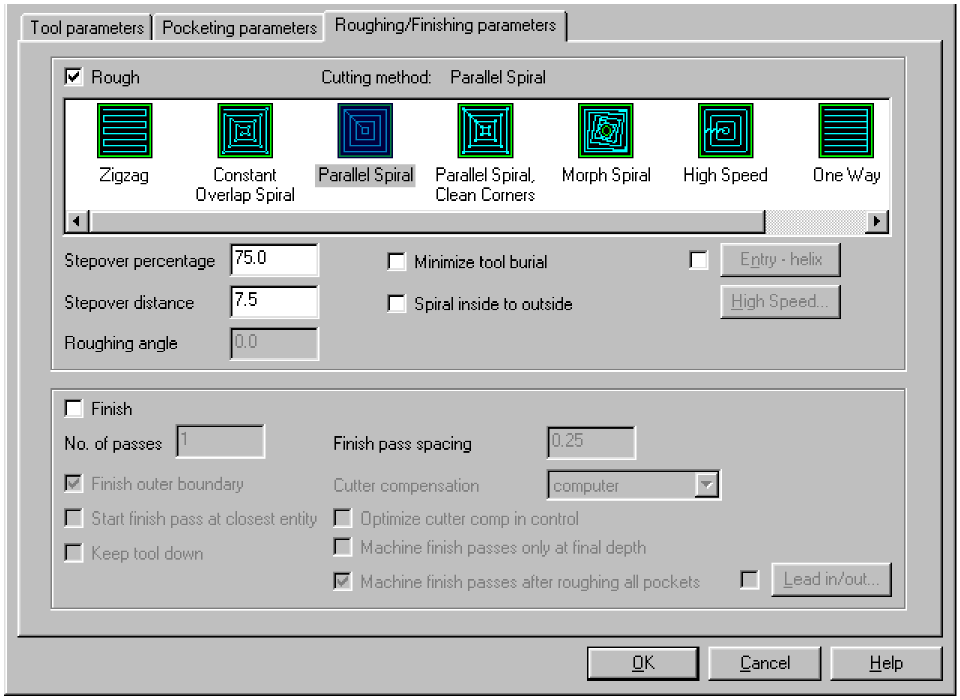
4. Choose **Done**.

5. Select the 10 mm HSS flat endmill from the tool library.
6. Choose the **Pocketing parameters** tab.
7. Enter the values shown in the following picture.

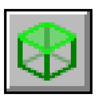


Note: The depth values are not important, since you won't actually be cutting the part at these depths. Instead, Mastercam will project this toolpath onto the part when you create the surface toolpath.

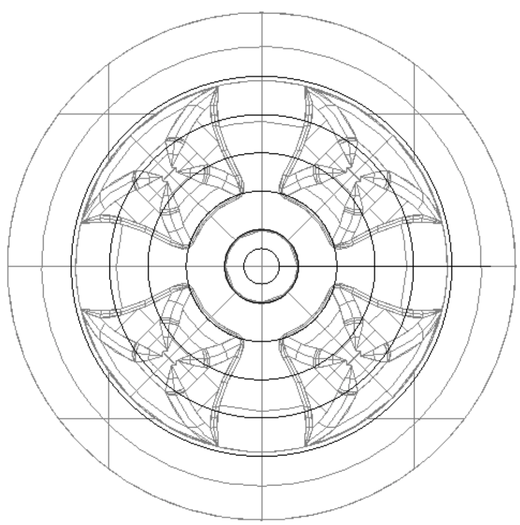
8. Choose the **Roughing/Finishing parameters** tab.
9. Enter the values shown in the following picture.



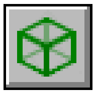
10. Choose **OK**. Mastercam creates the toolpath.



11. Choose the green **Gview–Top** button from the toolbar see the toolpath more clearly. Your toolpath should look like the following picture.



12. Press [Alt + T] to clear the toolpath from the screen.



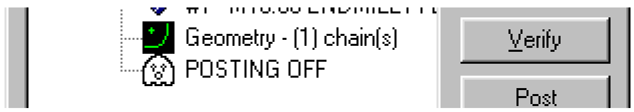
13. Choose the green **Gview–Isometric** button from the toolbar to return the screen to isometric view.

▶ *Disabling posting for the pocket toolpath*

Mastercam has an option that lets you disable posting for an operation. Since you will not actually be cutting the pocket toolpath, just the surface toolpath that you will create in the next procedure, you should disable posting for it so you do not accidentally include it in your NC program.

1. Press [Alt + O] to open the Operations Manager.
2. Right-click in the window and choose **Options, Posting, Off**.

   The following icon appears in the Operations Manager.

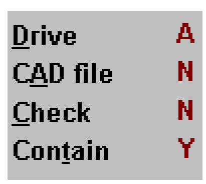


3. Choose **OK** to close the Operations Manager.

▶ *Creating the surface toolpath*

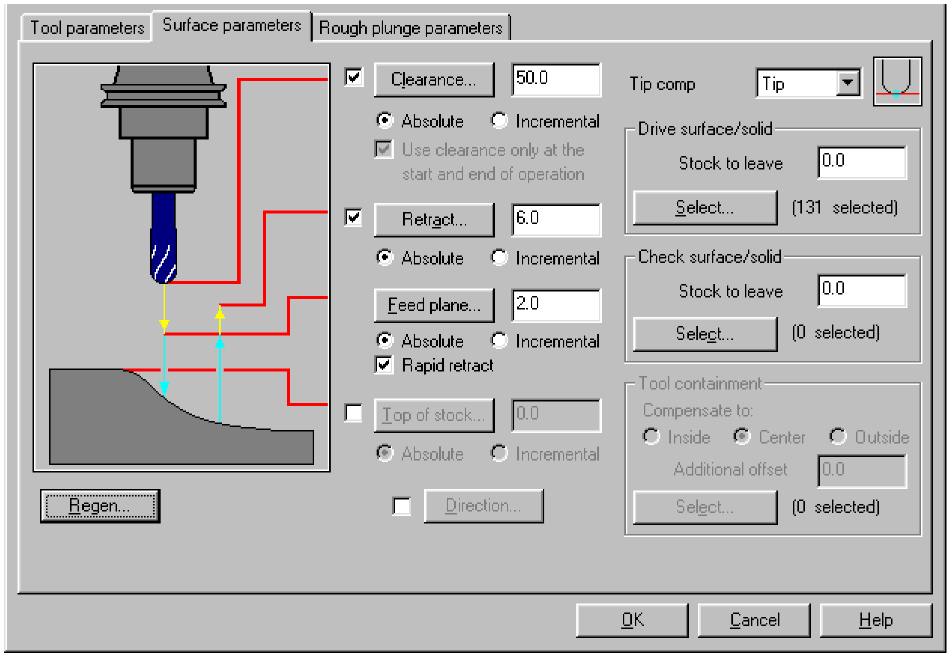
1. Choose **Surface**.
2. Toggle the **Drive** setting to **A**.

   Your options should match the following picture.



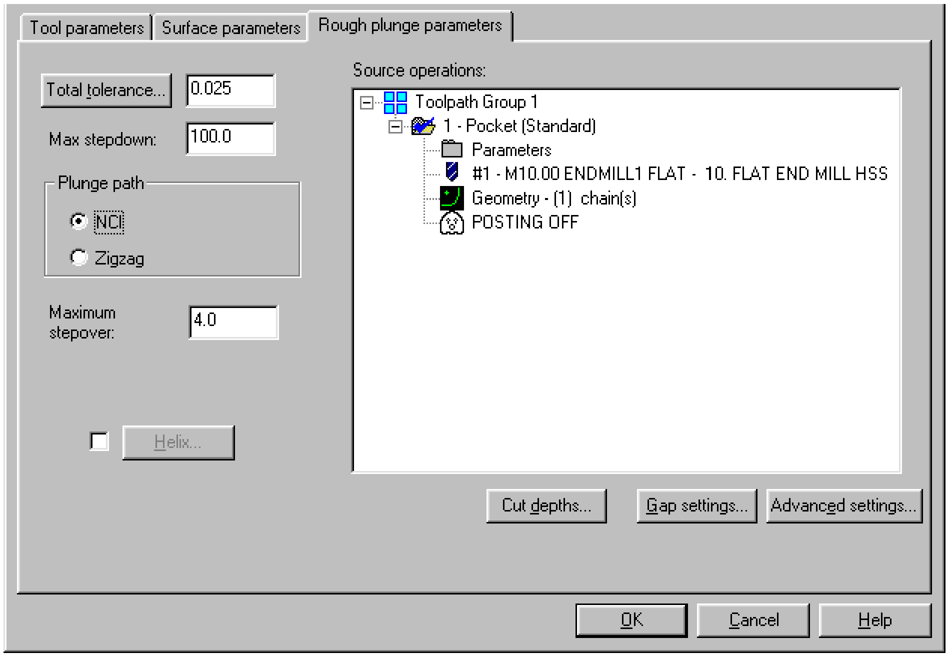
   Note: Rough Plunge toolpaths don't use tool containment boundaries, so the Contain setting doesn't matter.

3. Choose **Rough, Plunge**.
4. Select the 10 mm HSS flat endmill.
5. Select the **Surface parameters** tab.
6. Enter the values as shown on the following dialog box.

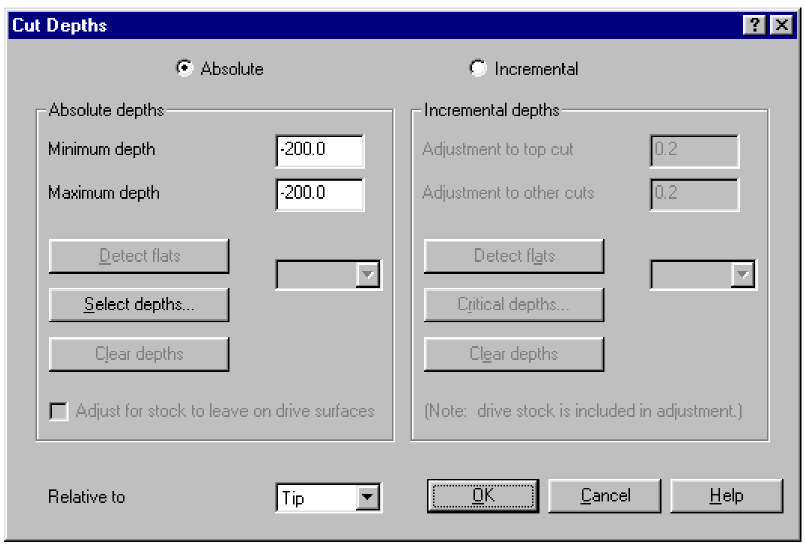


▶ *Entering the rough plunge parameters*

1. Select the **Rough plunge parameters** tab.
2. Choose the **NCI** option for **Plunge path**.
3. Select the pocket toolpath from the operations list in the **Source** window. Mastercam will use the NCI file from this operation to project the toolpath onto the drive surfaces you've selected.
4. Enter the other values as shown on the following dialog box.



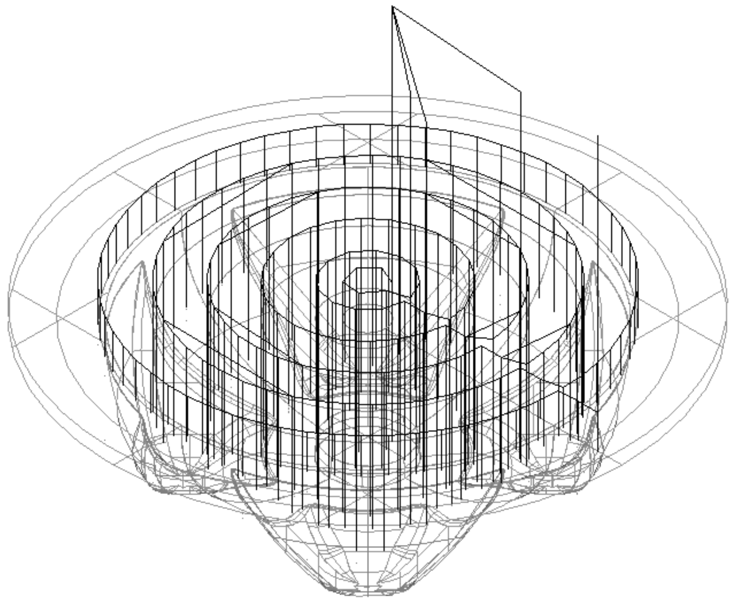
5. Choose the **Cut depths** button.
6. Enter the values shown on the following dialog box.



Tip: Setting minimum and maximum cut depths that are well below the bottom of the part as shown here ensures that the tool goes straight to the bottom of the part without using a pecking motion.

7. Choose **OK** twice.

   Mastercam generates the toolpath, which should look like the following picture.

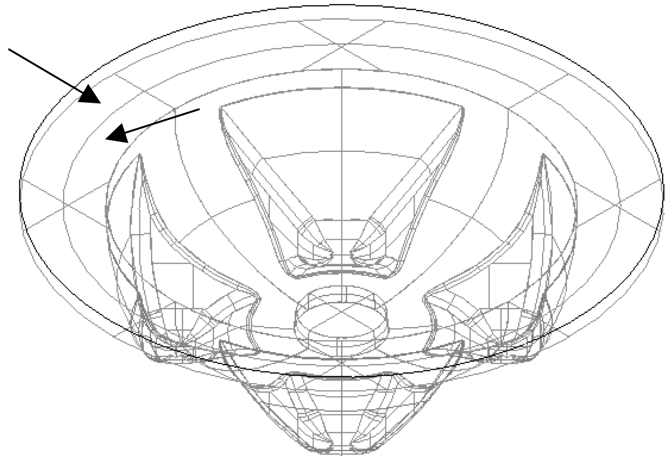


8. Save the file in your working folder as **NCI plunge.mc9**.

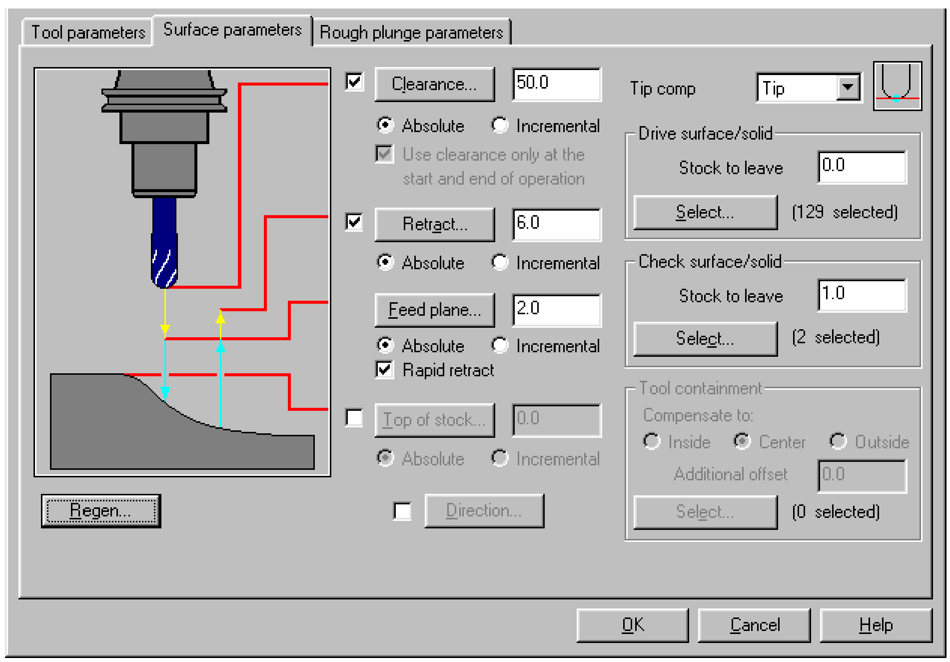
Using check surfaces to restrict the toolpath

After inspecting the toolpath, you realize that the tool is plunging on the shallow outer rim of the part. For a plunge toolpath, this is wasted tool activity. In this procedure, you will edit the surface selection so that the toolpath avoids these surfaces.

1. Press [Alt + O] to open the Operations Manager.
2. Choose the **Parameters** icon for the Surface Rough Plunge operation.
3. On the **Surface parameters** tab, choose the **Select** button in the **Check surface/solid** section.
4. Choose **Add**.
5. Choose the two outer surfaces as shown in the following picture.

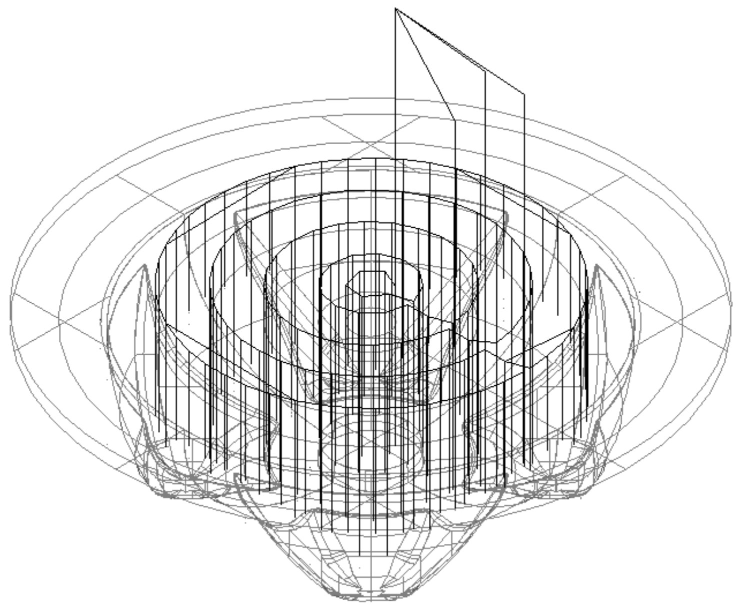


6. Choose **Done** twice to return to the Surface parameters dialog box. Mastercam will now avoid those two surfaces when creating the toolpath.
7. Enter **1.0** for the **Stock to leave** in the **Check surface/solid** section. Your new settings should match the following picture.



Note: Notice that Mastercam automatically updated the number of drive surfaces.

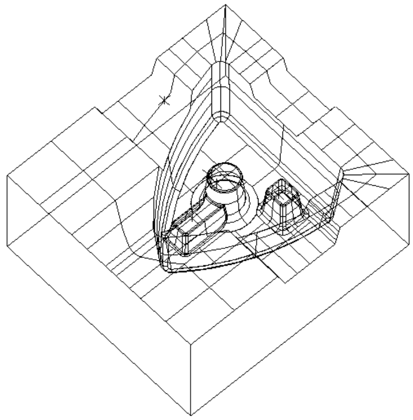
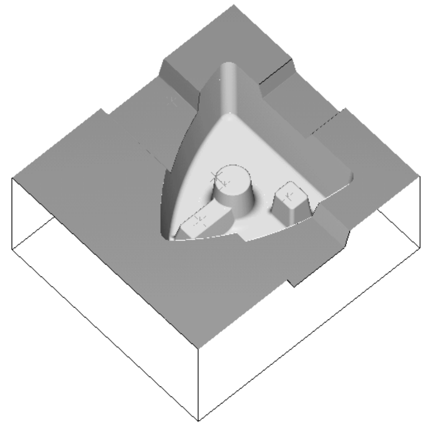
8. Choose **OK**.
9. Choose **Regen Path**. The new toolpath should look like the following picture. You can see that all the new plunge points are inside the well of the part.



10. Press [Alt + A] to save the file.

Exercise 3 – Creating a restmill toolpath

Restmilling is the only roughing toolpath that cleans up remaining stock with a roughing, planar (constant Z) cut motion. Because restmilling uses multiple Z cuts to remove the remaining stock, it is much more effective than a finish leftover toolpath for operations where the roughing operation has left a lot of stock to remove. (A finish leftover toolpath goes directly to the bottom of the remaining stock, so it is appropriate for finishing operations where a smaller amount of stock remains.) The following pictures show the wireframe and surface geometry for the part.

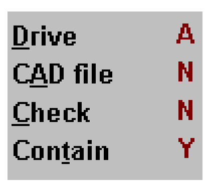
In this exercise, you will learn the following skills:

- **Using a restmill toolpath to clean up leftover stock**
- **Automatically detecting critical depths**

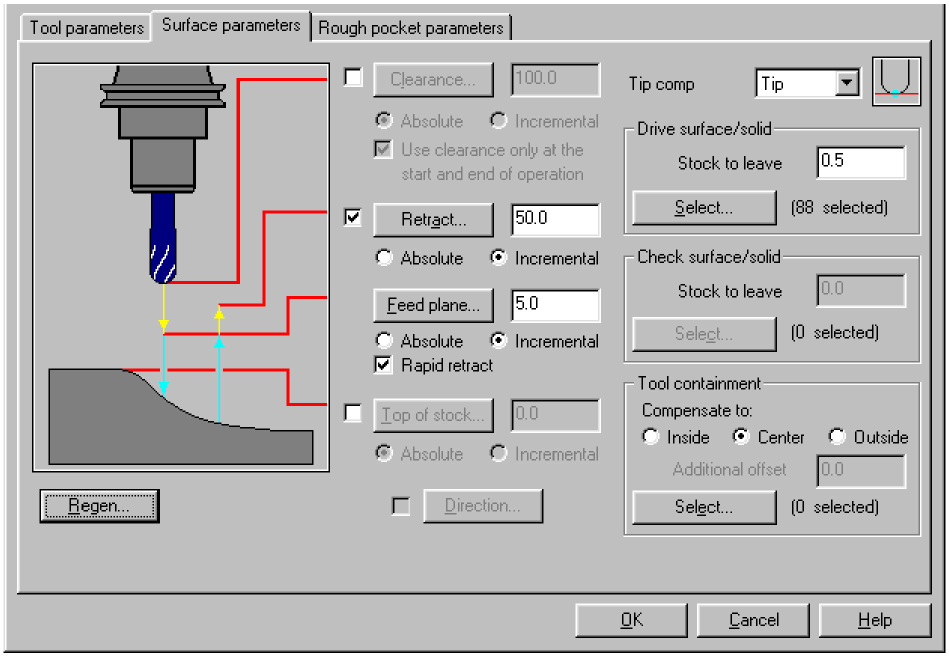
Choosing the surfaces and surface parameters

1. Open the file **restmill-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface.**
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **Y**.

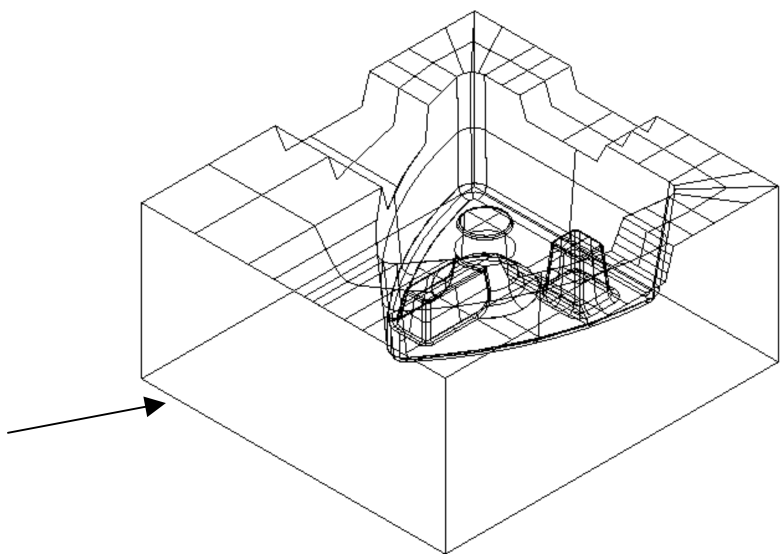
   Your surface selection menu options should match the following picture.



5. Choose **Rough, Pocket**.
6. Select the 18 mm HSS flat endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.



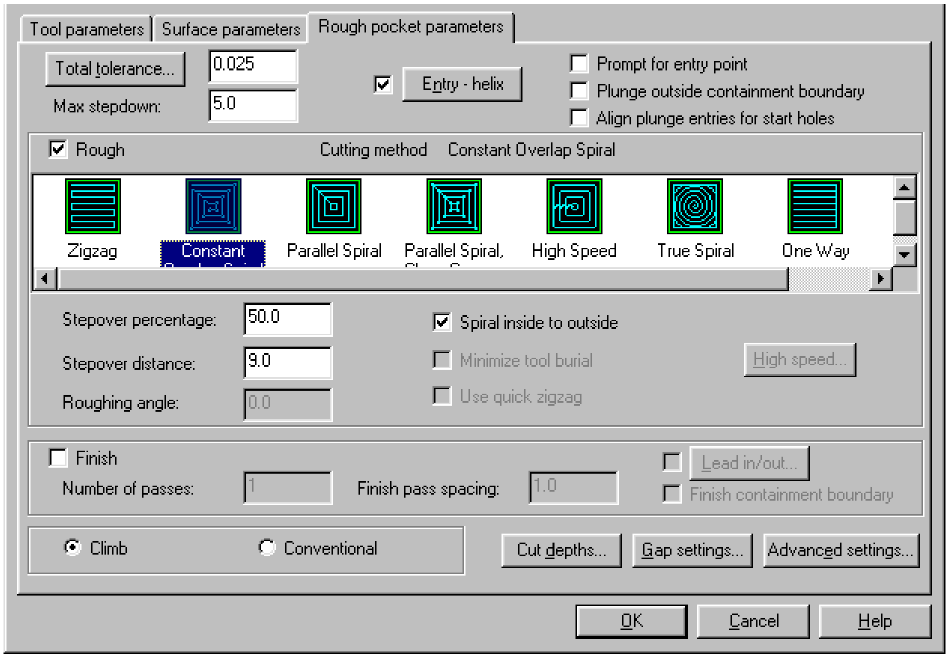
9. Choose the **Select** button in the **Tool containment** section.
10. Choose **Chain, Options**.
11. Select the **Plane mask** option and choose **OK**.
12. Select the bottom of the part as shown in the following picture.



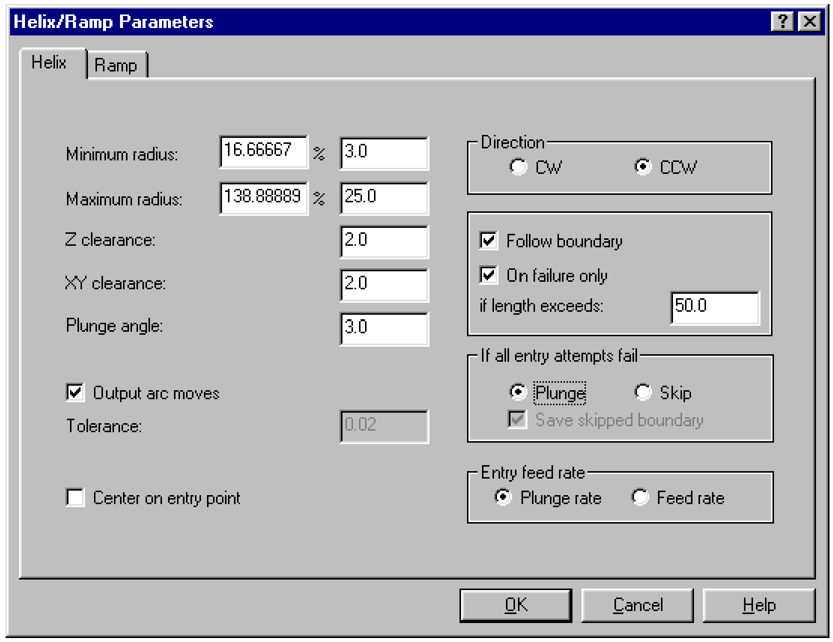
13. Choose **Done** to return to the Surface parameters dialog box.

▶ *Entering the pocket parameters*

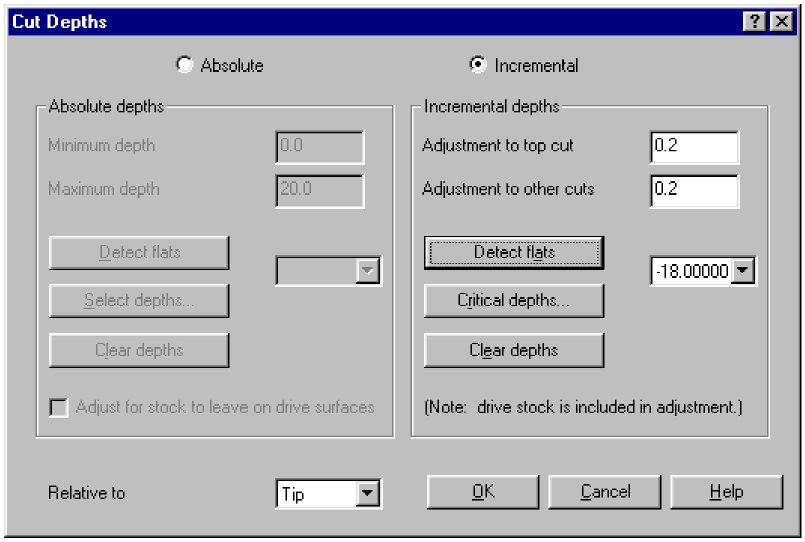
1. Select the **Rough pocket parameters** tab.
2. Enter the values shown on the following dialog box.



3. Choose the **Entry – helix** (or **Entry – ramp**) check box and button.
4. Choose the **Helix** tab.
5. Enter **3** for the **Minimum radius**.
6. Enter **25** for the **Maximum radius**. Your values should match the following dialog box.

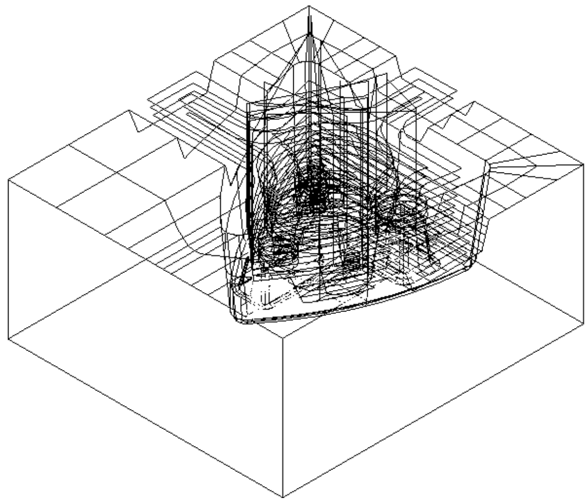


7. Choose **OK**.
8. Choose **Cut depths**.
9. Choose **Detect flats** in the **Incremental depths** section. Mastercam automatically identifies the tops of the islands and creates cutting passes at those heights. Your other settings should match the following picture.



10. Choose **OK** twice.
11. Choose **OK** at the following message.

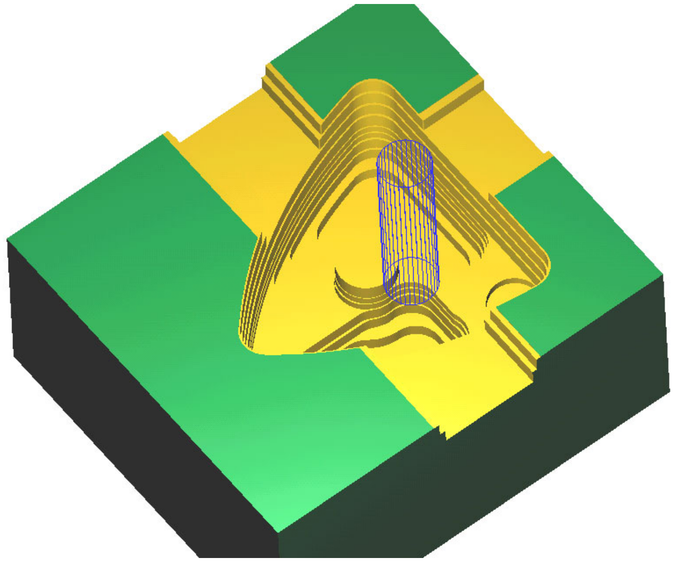
Mastercam generates the toolpath as shown in the following picture.



▶ *Verifying the toolpath*

Use the Verify function to see the toolpath more clearly.

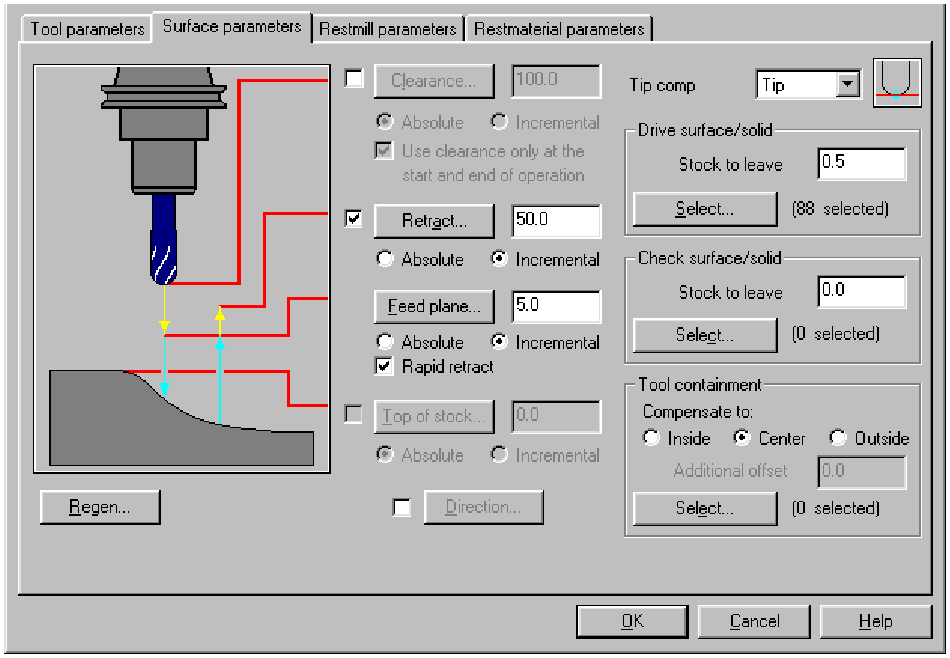
1. Press [Alt + O] to open the Operations Manager.
2. Choose **Verify**.
3. Choose the **Machine** button on the Verify toolbar.
4. When the verification is done, right-click in the graphics window and choose **Dynamic spin**.
5. Move the mouse to rotate the part as shown in the following picture, so you can see the stock removal inside the pocket. Click the mouse to anchor the view. You can see the areas where the tool couldn't reach. The next procedure will show you how to clean these out with a restmill toolpath.



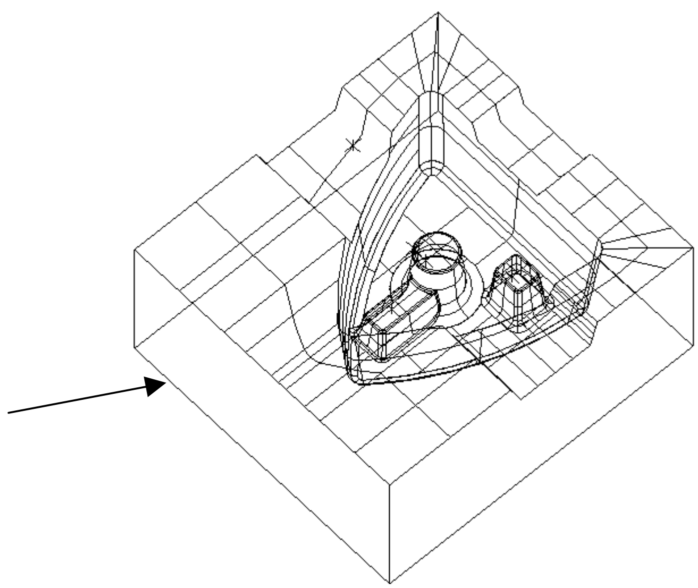
6. Close the Verify toolbar to return to the Operations Manager.

▶ *Creating the restmill toolpath*

1. Right-click in the Operations Manager window and choose **Toolpaths, Surface rough, Restmill**.
2. Select the 6 mm HSS flat endmill from the tool library.
3. Choose the **Surface Parameters** tab.
4. Enter the values as shown on the following dialog box.

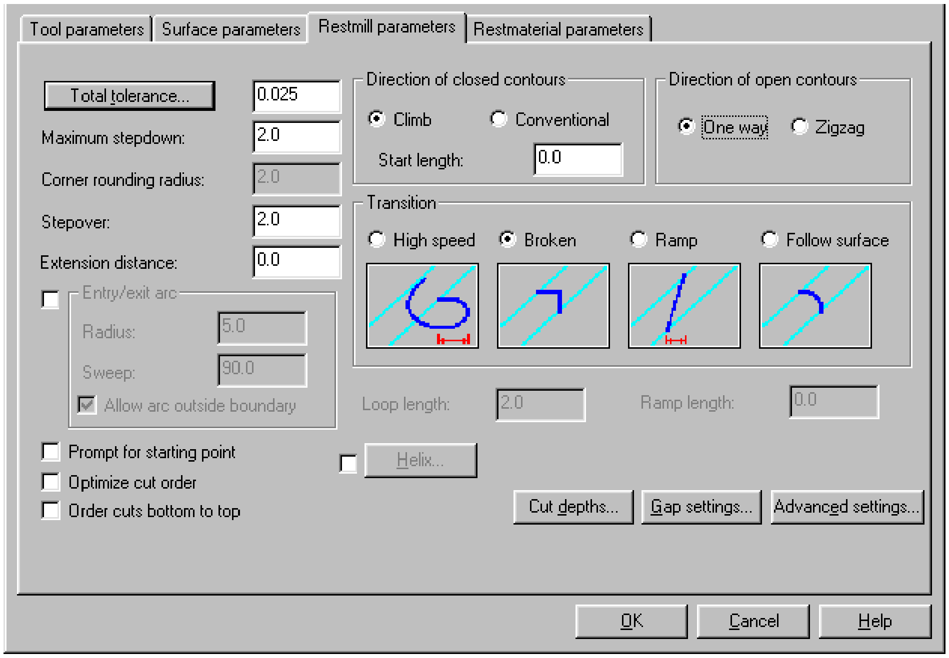


5. Choose the **Select** button in the **Tool containment** section.
6. Select the bottom of the part as shown in the following picture.

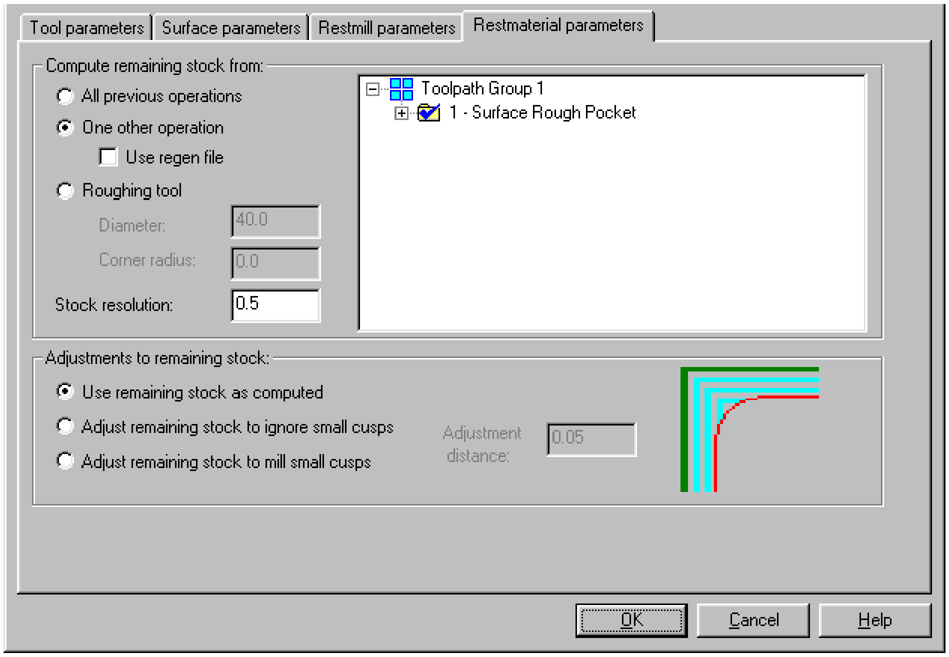


Note: The plane mask is still in effect from the previous procedure.

7. Choose **Done** to return to the Surface parameters dialog box.
8. Choose the **Restmill parameters** tab.
9. Make sure your selections match the following dialog box.

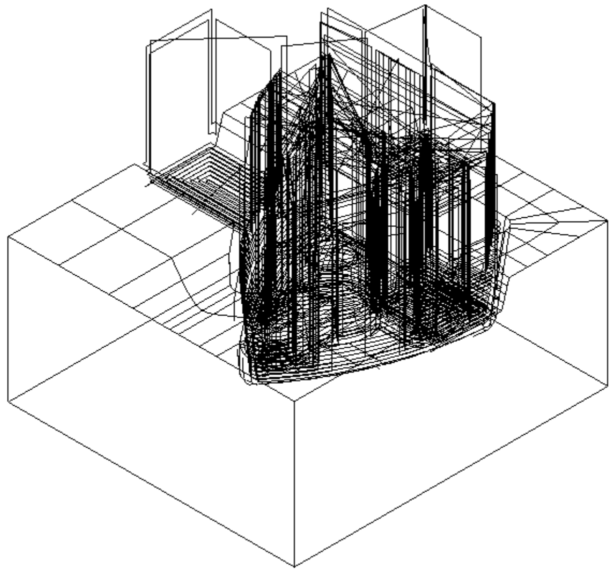


10. Choose the **Restmaterial parameters** tab.
11. Make sure your selections match the following dialog box.



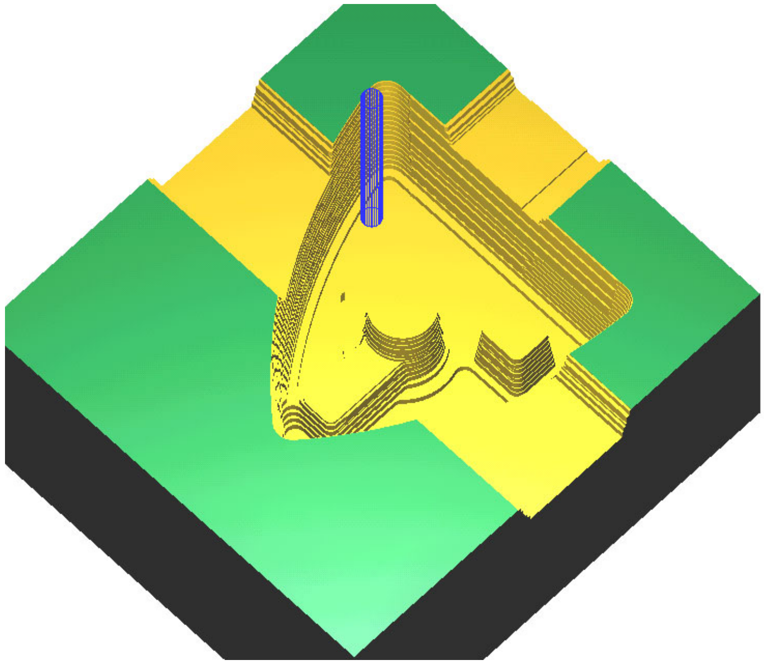
Tip: You can restmill stock left over from one or more earlier operations in the current file, or you can let Mastercam simulate a previous roughing operation by choosing the **Roughing tool** option and entering the tool dimensions.

12. Choose **OK**. Mastercam generates the toolpath as shown in the following picture.



Verifying the restmill toolpath

1. Press [Alt + O] to open the Operations Manager.
2. Choose **Select All, Verify**.
3. Choose the **Machine** button ▶ on the Verify toolbar.
4. When the verification is done, right-click in the graphics window and choose **Dynamic spin**.
5. Move the mouse to rotate the part as shown in the following picture, so you can see the results achieved by the restmill toolpath. Click the mouse to anchor the view.

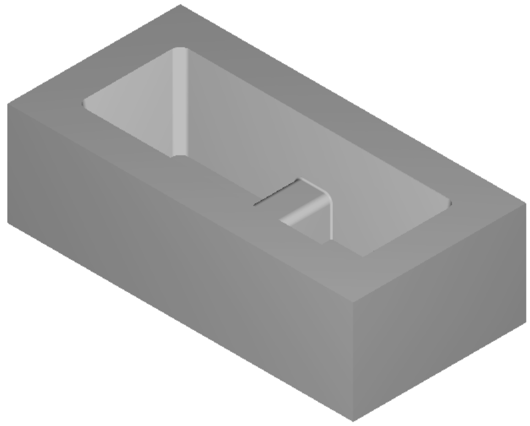


6. Close the Verify toolbar to return to the Operations Manager.
7. Choose **OK** to close the Operations Manager.
8. Save the file in your working folder as **restmill rough.mc9**.

Exercise 4 – Creating a high speed pocket toolpath

High speed surface machining is often performed using smooth tool motion, which means that throughout the entire toolpath, arcs or small line segments are smoothly connected without sharp corners. Mastercam provides several options for creating the smoothest possible tool motion between gaps in the toolpath and between depth cuts in the Z axis. In this exercise, you will create a trochoidal toolpath, in which the tool moves in small loops when it is in full contact with the material to minimize tool burial and optimize chip load.

In addition, you will create a separate operation that automatically calculates all the plunge points required by the toolpath and pre-drills them. To make the toolpath even more efficient, Mastercam can automatically align the plunge points at each cutting depth. The part you will machine is shown in the following picture.



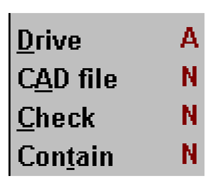
This exercise shows you the following skills:

- **Creating a high speed rough pocket toolpath**
- **Aligning plunge points**
- **Pre-drilling start holes**

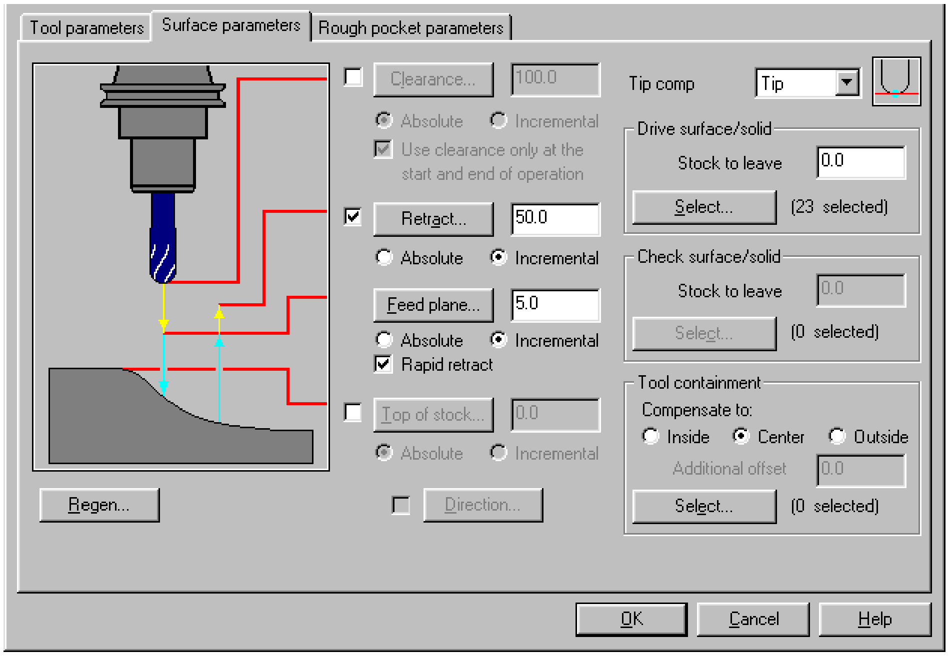
▶ *Choosing the surfaces and surface parameters*

1. Open the file **highspeed rough-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. If necessary, press [Alt + S] to turn on surface shading.
3. Choose **Main Menu, Toolpaths, Surface.**
4. Toggle the **Drive** setting to **A**.
5. Toggle **Contain** to **N**.

   Your surface selection menu options should match the following picture.

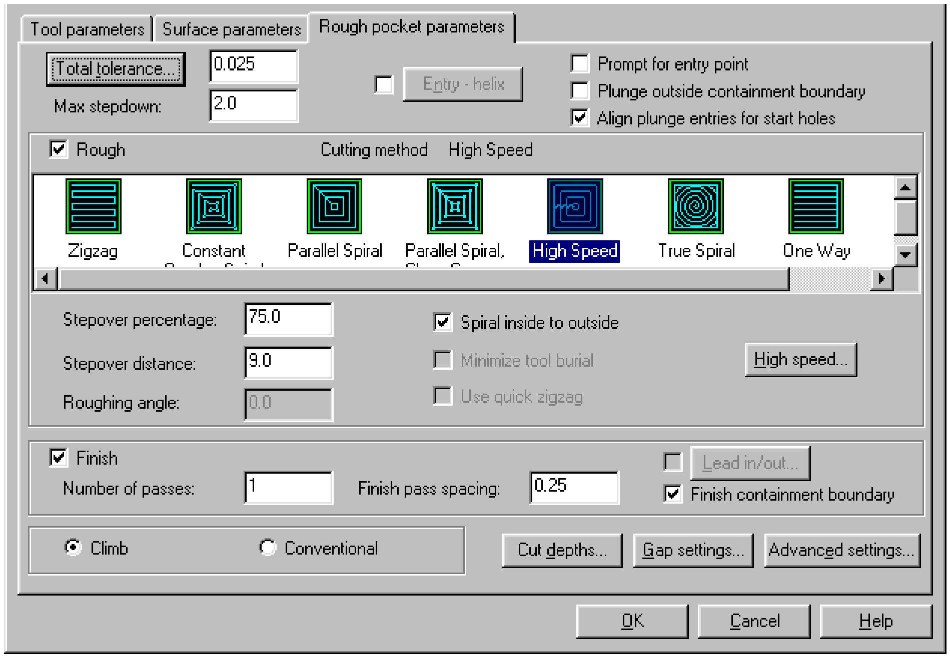


6. Choose **Rough, Pocket**.
7. Select the 12 mm HSS flat endmill from the tool library.
8. Select the **Surface parameters** tab.
9. Enter the values as shown on the following dialog box.



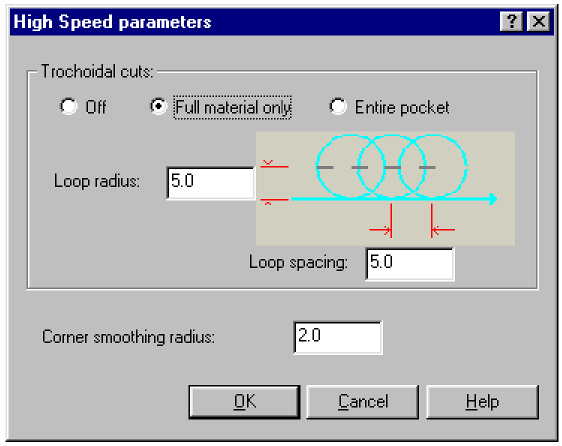
▶ *Entering the pocket parameters*

1. Select the **Rough pocket parameters** tab.
2. Enter the values shown on the following dialog box.

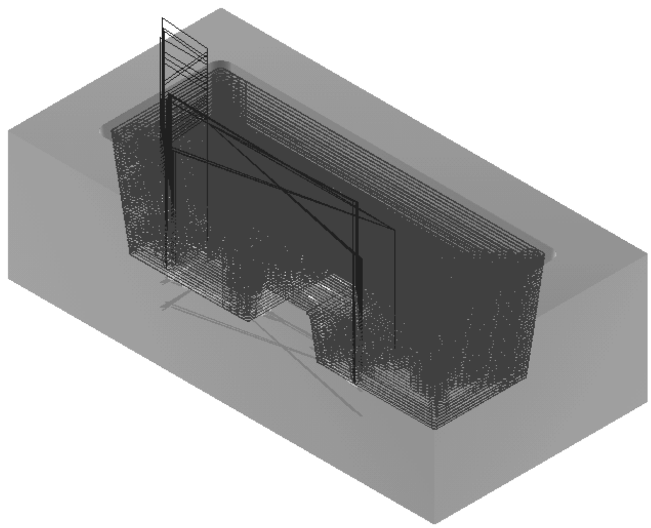


*Note: Because the walls of the pocket are tapered, the plunge points for each cutting pass would normally be shifted slightly at each new depth. Selecting **Align plunge entries for start holes** means that Mastercam keeps them lined up, so you can pre-drill just one hole.*

3. Choose the **High speed** button.
4. Select **Full material only** for the **Trochoidal cuts** option. This means that the tool will loop only when it is full contact with the material; when it is partial contact, it will follow the toolpath normally.
5. Enter the loop dimensions shown in the following dialog box.



6. Choose **OK** twice. Mastercam generates the toolpath as shown in the following picture.

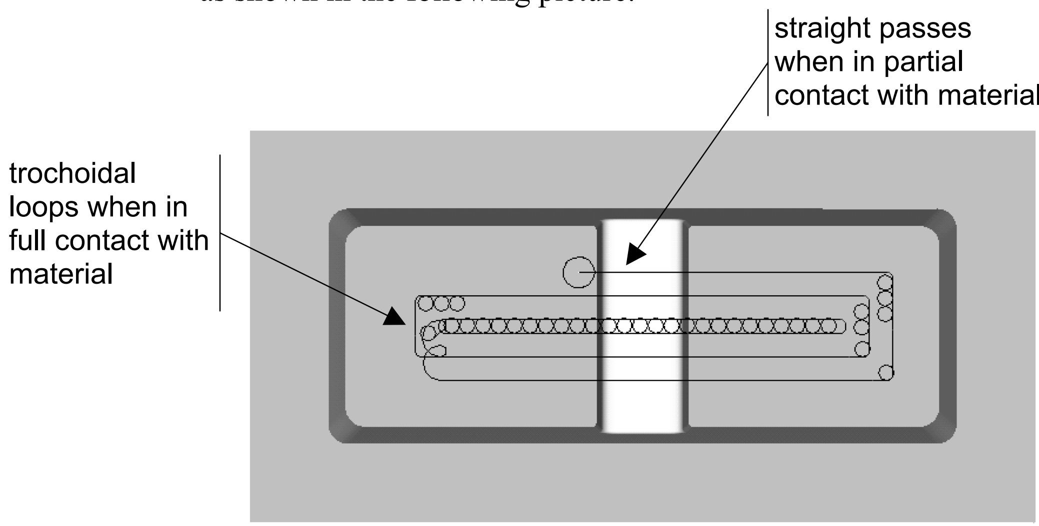


Because of the island in the middle of the part, a single plunge point for the cutting passes at the lower depths would result in an unmachinable toolpath. Mastercam recognized the island and automatically added a second plunge point. However, it still aligned the plunge points in each half of the pocket, minimizing the start hole drilling.

Backplotting the toolpath

Backplot the toolpath to see how Mastercam creates the loops.

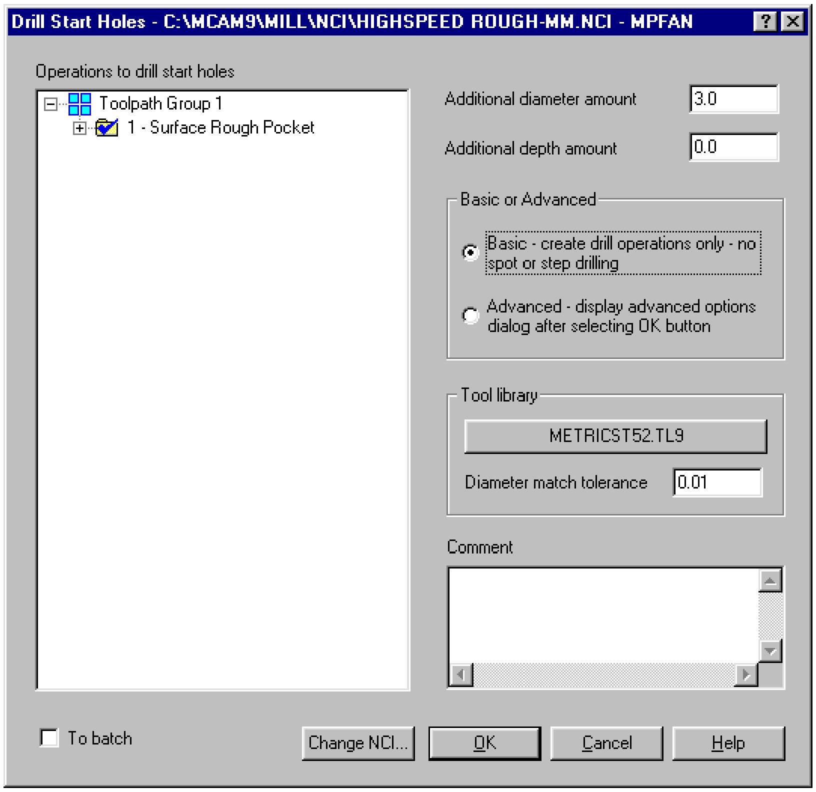
1. Press [Alt + O] to return to the Operations Manager.
2. Choose **Backplot**.
3. Switch to **Gview–Top**.
4. Press [S] several times. You will see the toolpath start to develop as shown in the following picture.



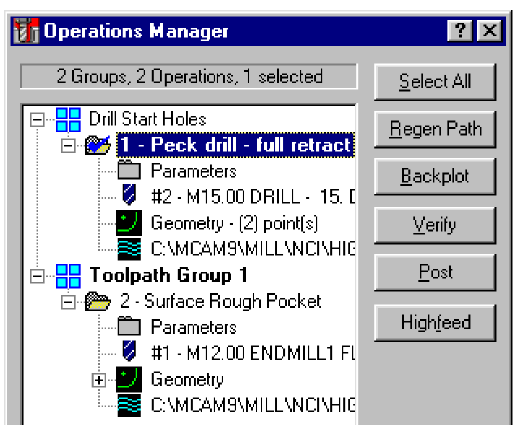
5. Press [Esc] and choose **OK** to end the backplot.
6. Switch back to **Gview–Isometric**.

Creating the start hole operation

1. Choose **Backup** to return to the Operations Manager.
2. Right-click and choose **Toolpaths, Circle paths, Drill start holes**.
3. Enter an **Additional diameter amount** of **3**, to provide some clearance for the tool.
4. Choose the **Basic** option. Your selections should match the following dialog box.

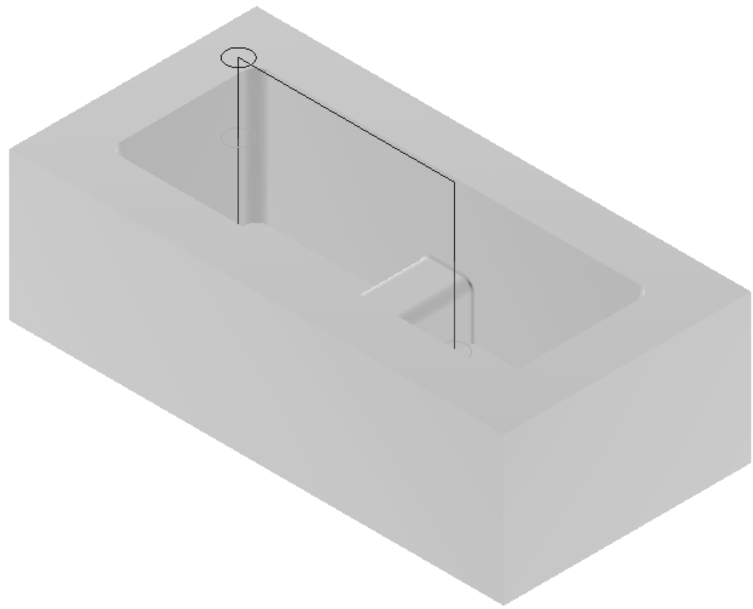


5. Choose **OK**. Mastercam creates the new drilling operation as shown in the following picture. Even though you created the drilling operation after the pocketing operation, Mastercam automatically places it first, so the operations are in their proper machining sequence.



Mastercam automatically calculates peck amounts and other parameters. You can edit any of the drilling parameters by choosing the Parameters icon for the drill operation.

The drill toolpath should look like the following picture.



*Note: The start hole operation and the pocket operation are **not** associative with each other. This means that if you change the pocket toolpath, you need to delete the start hole operation and recreate it.*

6. Choose **OK** to close the Operations Manager.
7. Save the file in your working folder as **highspeed-align.mc9**.

Now that you've see a sampling of Mastercam's surface roughing capabilities, you're ready to use some surface finishing techniques. The next chapter will show you several examples.

15 Surface Finishing

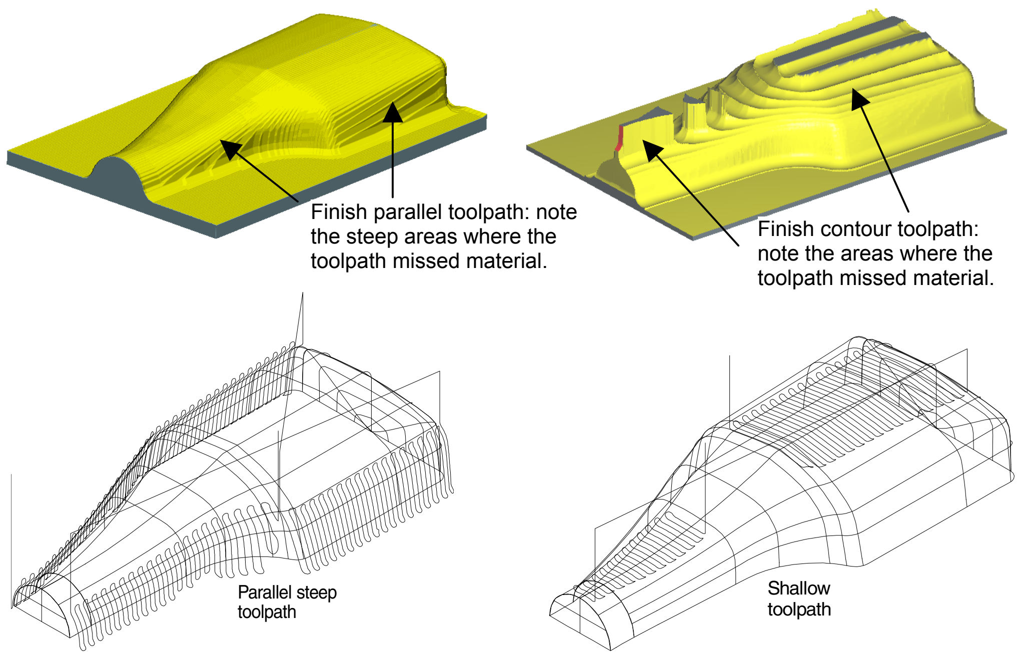
This chapter introduces some of the finishing toolpaths you can use in surface machining. Finish toolpaths remove material left behind by previous roughing toolpaths. This chapter shows you examples of the following finishing toolpaths:

- parallel steep
- parallel shallow
- radial
- project
- contour
- shallow contour
- scallop
- flowline

Other surface finishing toolpaths include finish parallel, leftover, and pencil toolpaths. These were shown in Chapter 13.

Exercise 1 – Using finish steep and shallow toolpaths

Using finish steep and shallow toolpaths on the following part makes sense because a finish parallel toolpath would miss material in the steep areas of the part and a finish contour toolpath would miss material in the shallow areas of the part (see the following pictures). A finish parallel steep toolpath is usually used after a finish parallel toolpath.



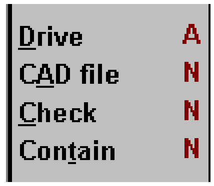
In this exercise, you will learn the following skills:

- **Creating a finish parallel steep toolpath**
- **Creating a finish shallow toolpath**
- **Using tangential arcs for gap settings**

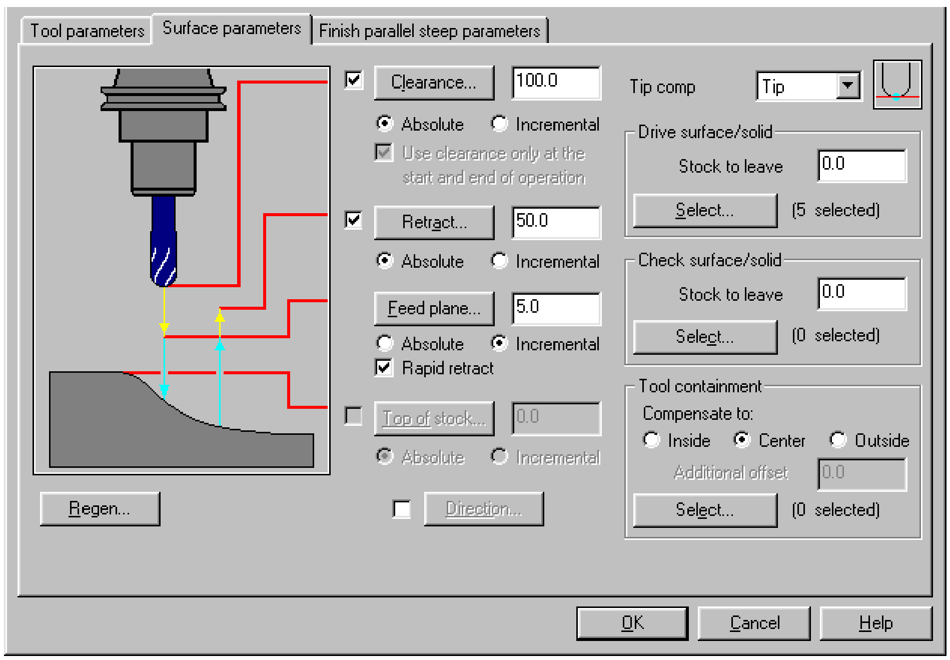
▶ *Creating the parallel steep toolpath*

1. Open the file **steep-shallow-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**. This tells Mastercam that you want to machine all the surfaces.
4. Toggle **Contain** to **N**. This tells Mastercam that you will not create a tool containment boundary.

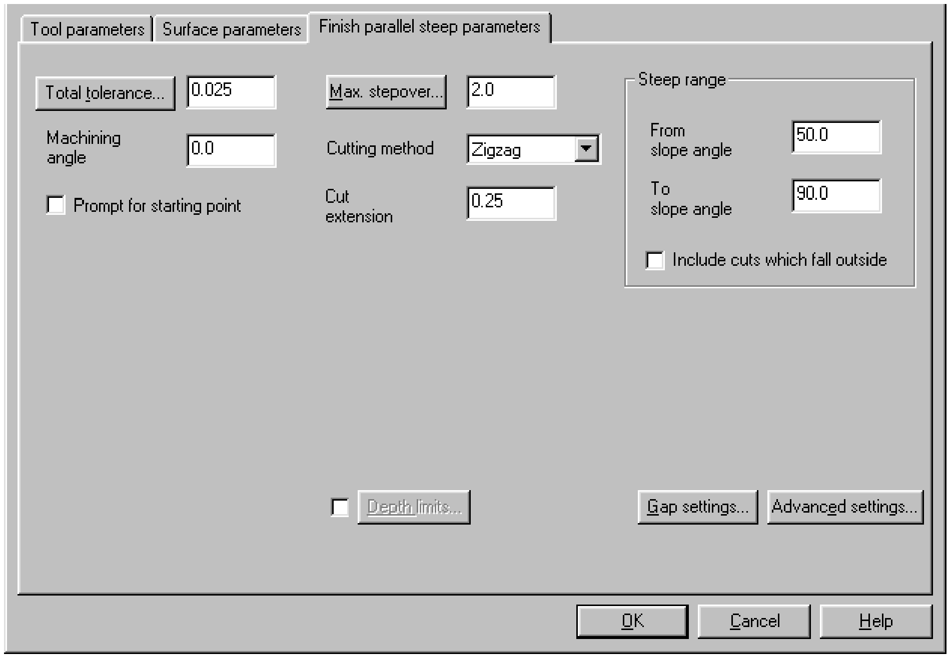
   Your surface selection menu options should match the following picture.



5. Choose **Finish, Par. Steep**.
6. Right-click in the tool display area and select the 6 mm HSS ball endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.

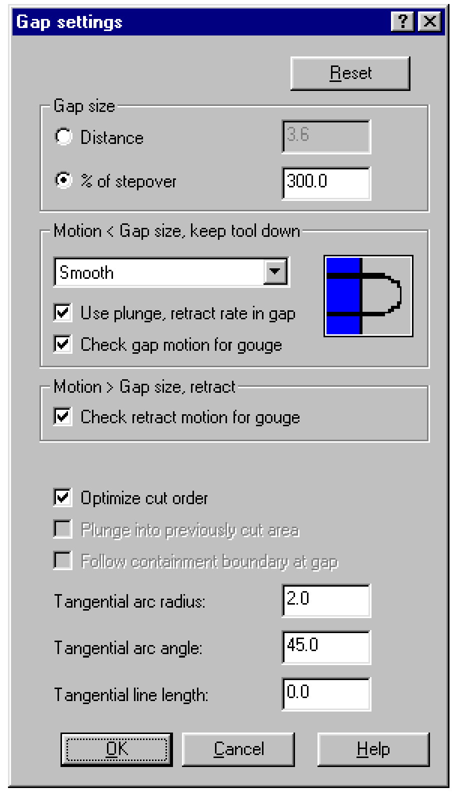


9. Select the **Finish parallel steep parameters** tab.
10. Enter the values shown on the following dialog box.



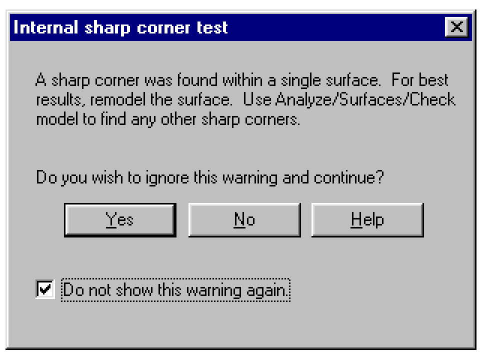
11. Choose **Gap settings**.

12. Enter the values shown in the dialog box at right.
13. Choose **OK** twice.



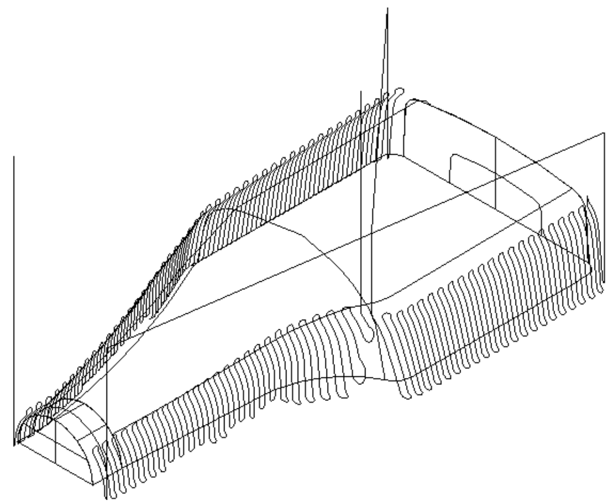
Tip: Tangential arcs are useful in steep and shallow toolpaths where you cut a previously finished surface. They allow the system to blend the entry and exit moves for each cut

14. Mastercam detects some corners in the part and displays the following message. (The sharp corner it detects is the ridge along the top of the part.) Since this doesn't affect the toolpath, choose the **Do not show this warning again** check box and choose **Yes**.



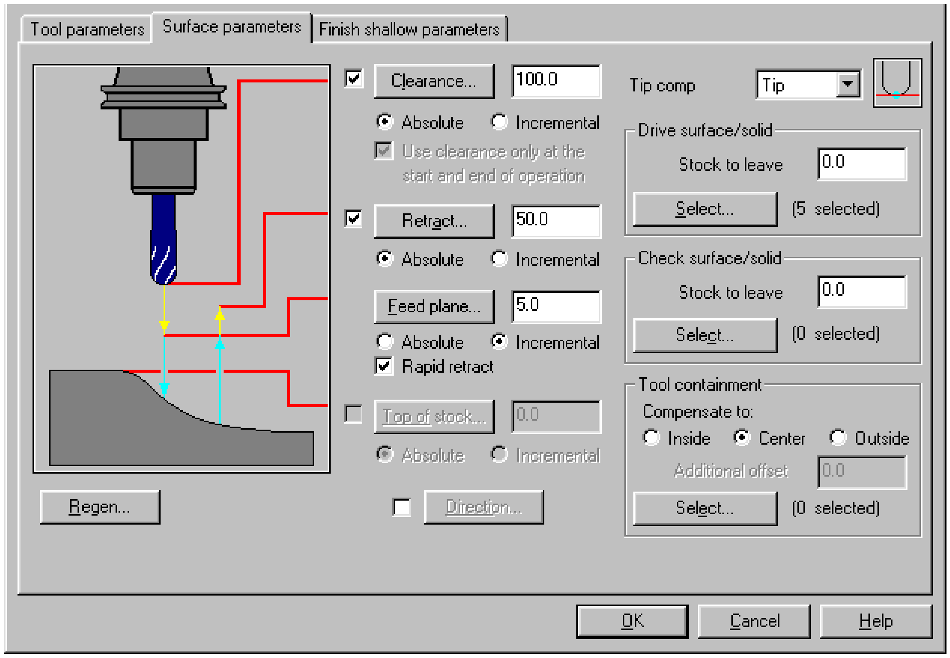
Tip: You can use the **Advanced settings** button on the **Finish parallel steep parameters** tab to tell Mastercam to ignore sharp corners.

Mastercam generates the toolpath on the steep areas of the part, between 50 and 90 degrees. It should look like the following picture.

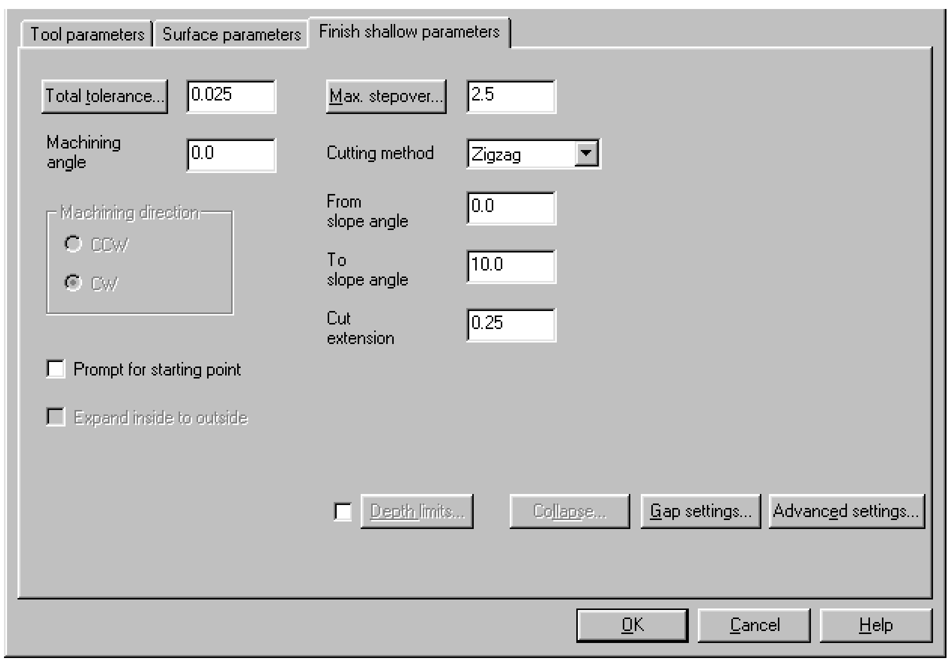


▶ *Creating the finish shallow toolpath*

1. Press [Alt + T] to clear the parallel steep toolpath from the screen.
2. Choose **Finish, Shallow**.
3. Select the 6 mm HSS ball endmill again.
4. Select the **Surface parameters** tab.
5. Enter the values as shown on the following dialog box.



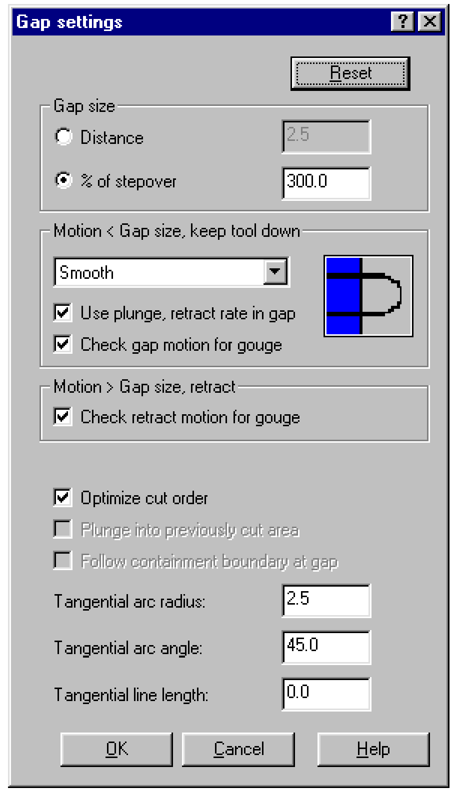
6. Select the **Finish shallow parameters** tab.
7. Enter the values shown on the following dialog box.



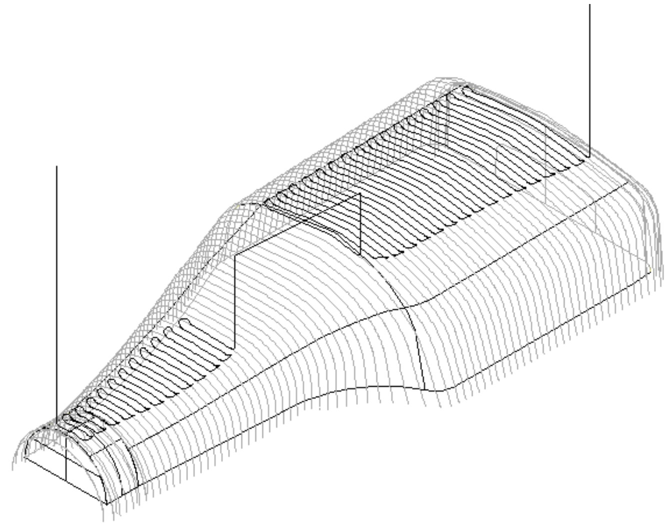
8. Choose **Gap settings**.

9. Enter the values shown in the dialog box at right.

10. Choose **OK** twice.



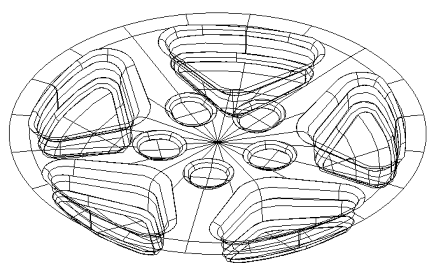
Mastercam generates the toolpath on the shallow areas of the part, between 0 and 10 degrees. It should look like the following picture.

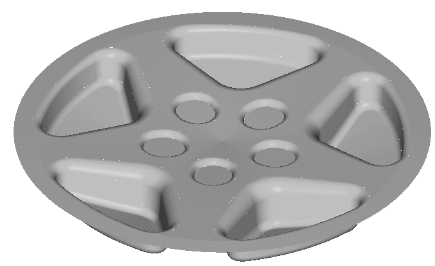


11. Save the file in your working folder as **steepandshallow.mc9**.

Exercise 2 – Creating a finish radial toolpath

Finish radial toolpaths can be the most efficient toolpaths for round parts. In this example, the tool zigzags from the center point to the outer edge of the part. The wireframe and surface geometry for the part is shown in the following pictures.





This exercise shows you the following skill:

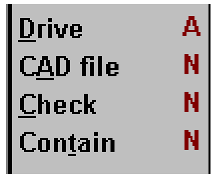
- **Creating a finish radial toolpath**

Creating the finish radial toolpath

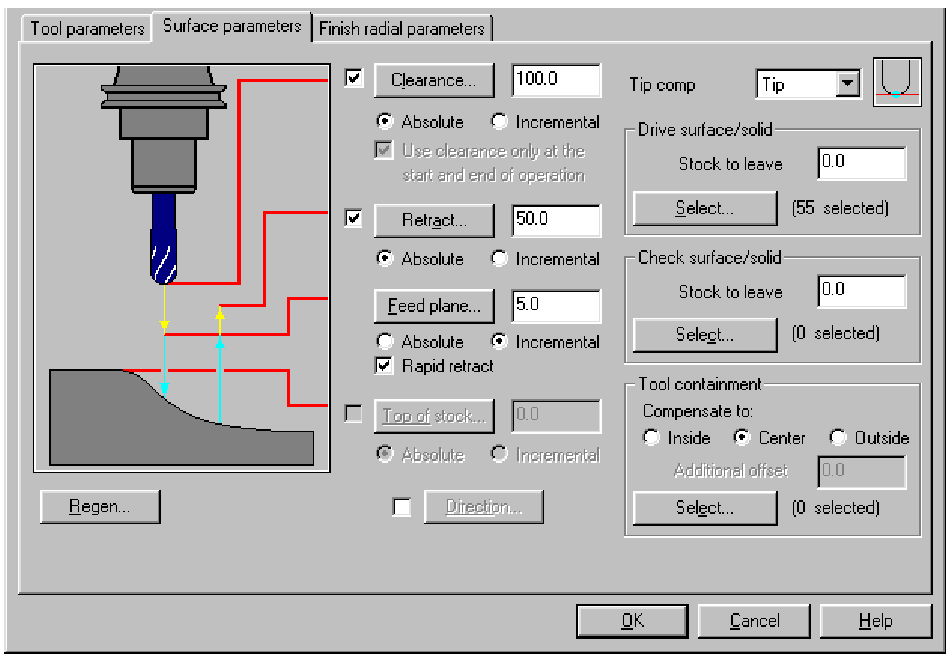
1. Open the file **radial-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.

2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle **Drive** to **A**.
4. Toggle **Contain** to **N**.

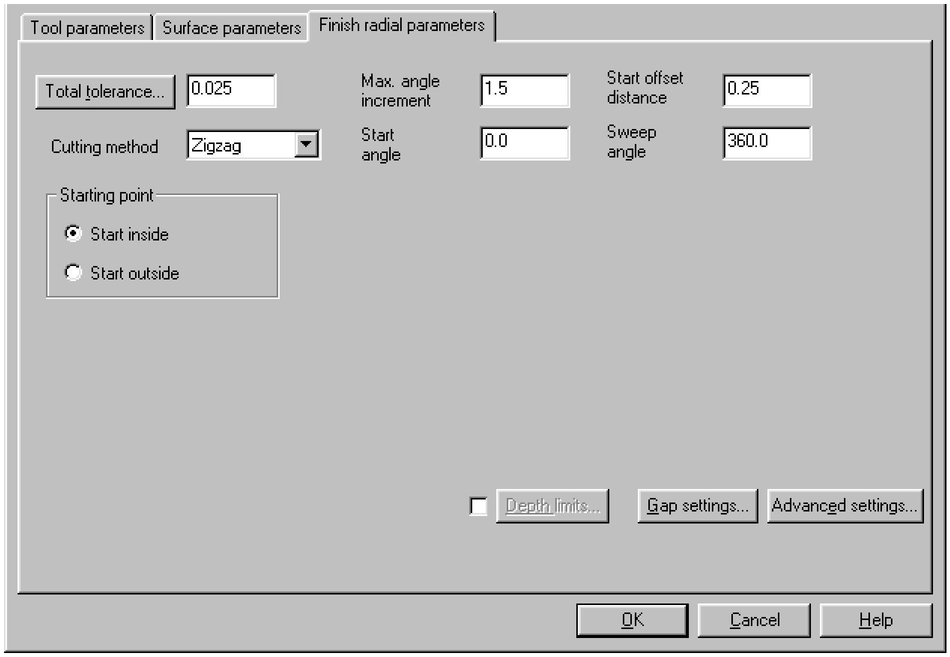
   Your surface selection menu options should match the following picture.



5. Choose **Finish, Radial**.
6. Right-click in the tool display area and select the 6 mm HSS ball endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.

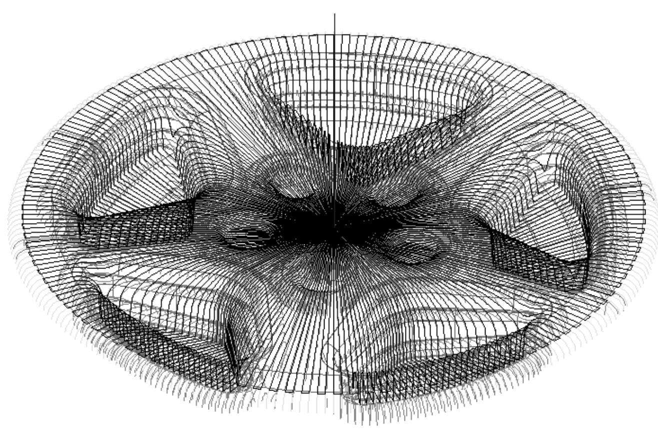


9. Select the **Finish radial parameters** tab.
10. Enter the values shown on the following dialog box.



11. Choose **OK**.
12. You are prompted to select a rotation point. This is the pivot point for the toolpath slices. Since the center of this part is X0Y0, press [O] to select the origin.

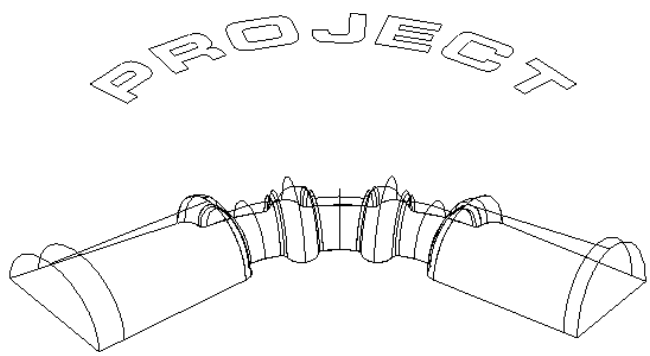
    Mastercam generates the toolpath as shown in the following picture.



13. Save the file in your working folder as **radial finish.mc9**.

Exercise 3 – Creating a finish project toolpath

Finish project toolpaths project either geometry or a toolpath from an earlier operation onto surfaces that you select. This finish toolpath provides free-form motion with the ability to match the cutting motions closely to the shape of the part. It also provides the most tool control. Engraving machining often uses project toolpaths. The part for this exercise is shown in the followmwing picture. The toolpath will project the word "PROJECT" onto the part below it.



Note: For more sophisticated engraving applications, you can purchase Mastercam Engraving, which works with Mastercam Mill.

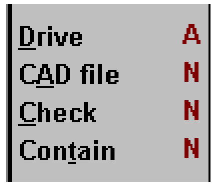
In this exercise, you will learn the following skill:

- **Creating a finish project toolpath**

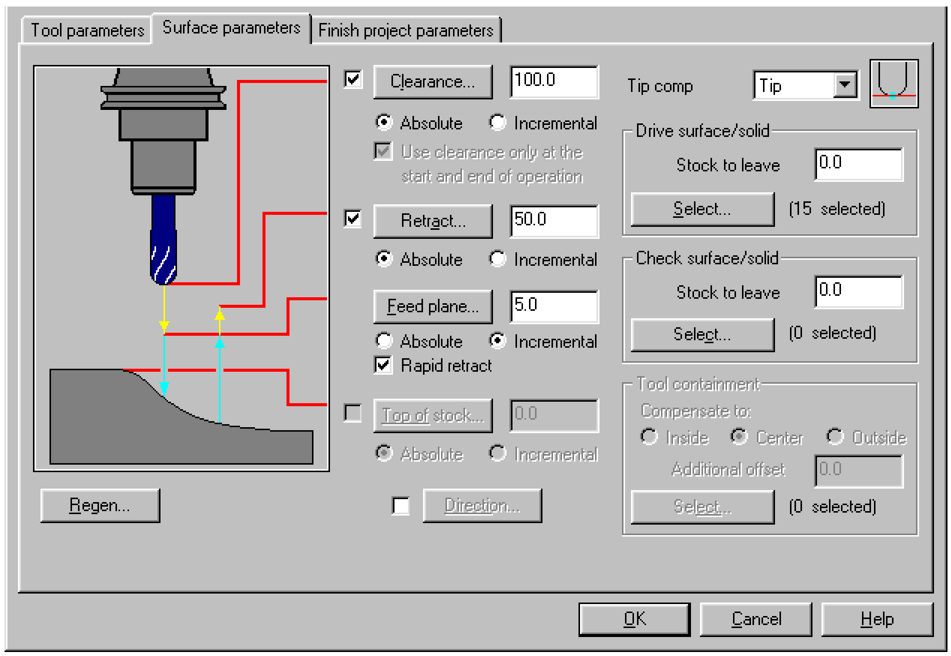
Creating the project toolpath

1. Open the file **project-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **N**.

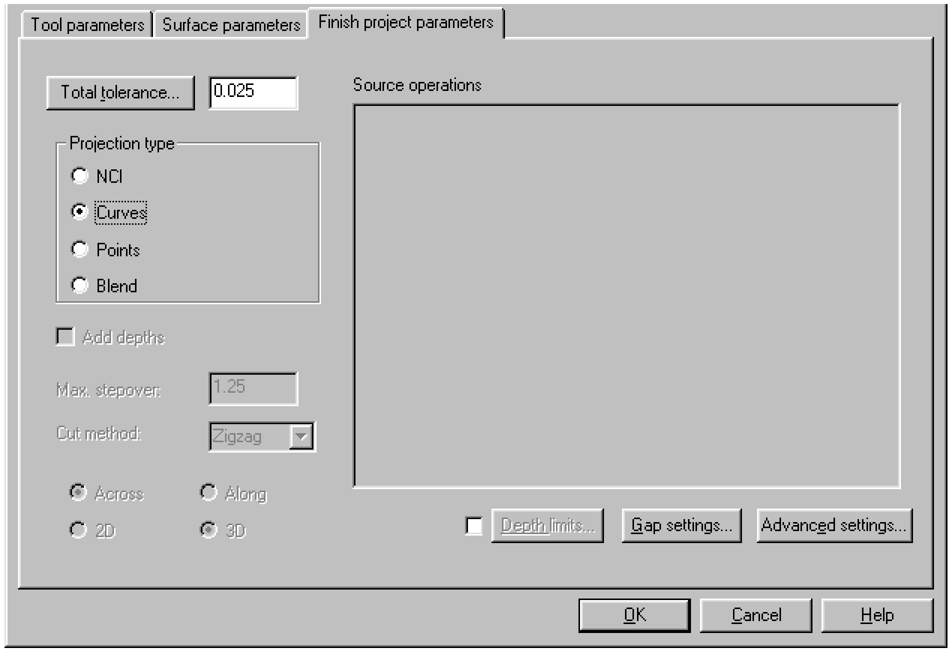
   Your surface selection menu options should match the following picture.



5. Choose **Finish, Project**.
6. Select the 1 mm ball endmill from the tool display window.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.

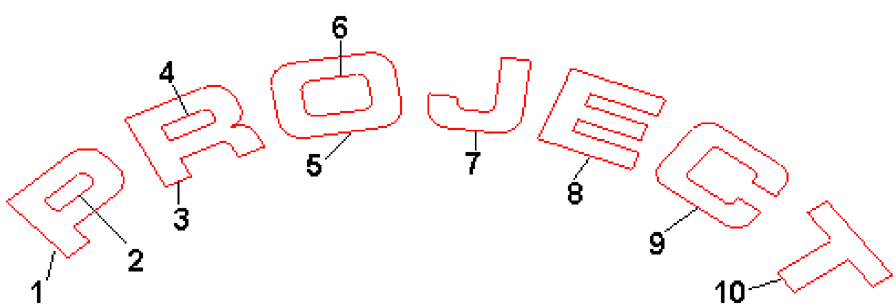


9. Select the **Finish project parameters** tab.
10. Enter the values shown on the following dialog box.

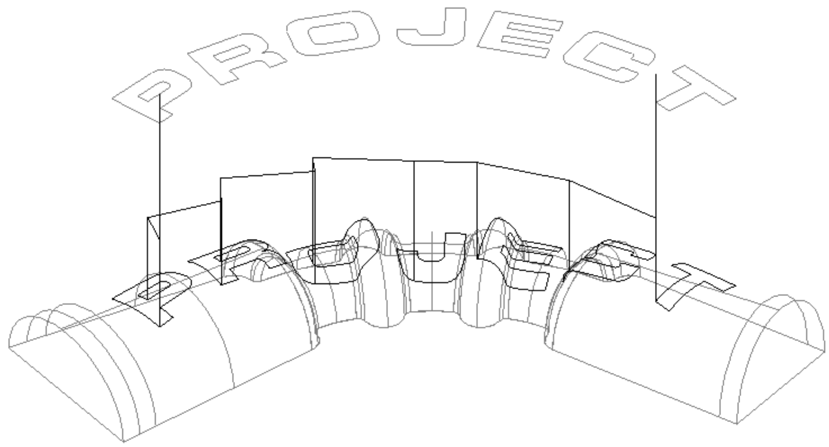


Note: If you had previously created other toolpaths, you could select a toolpath to project in the Source operations list area.

11. Choose **OK**.
12. Chain each section of the word "PROJECT" beginning with the outside boundary of the "P." Continue selecting the chains in the order shown in the following picture.



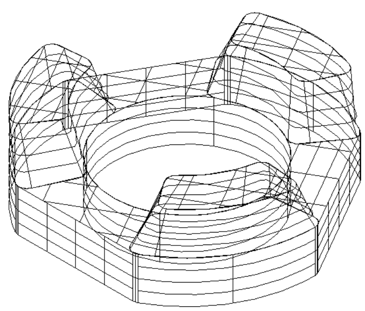
13. Choose **Done**. Mastercam generates the toolpath as shown in the following picture.

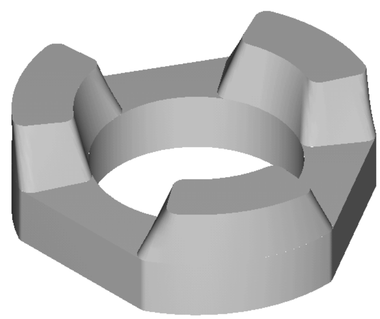


14. Save the file in your working folder as **finish project.mc9**.

Exercise 4 – Creating a finish contour toolpath

Finish contour works well for the following part because the part includes several steep walls. The finish contour toolpath allows the tool to step down gradually in the Z axis instead of stepping over in the X and Y axes. The following pictures show the wireframe and surface geometry for the part you will use.





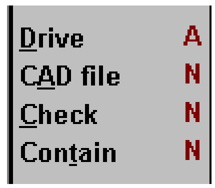
In this exercise, you will learn the following skills:

- **Creating a finish contour toolpath**
- **Automatically detecting flats**

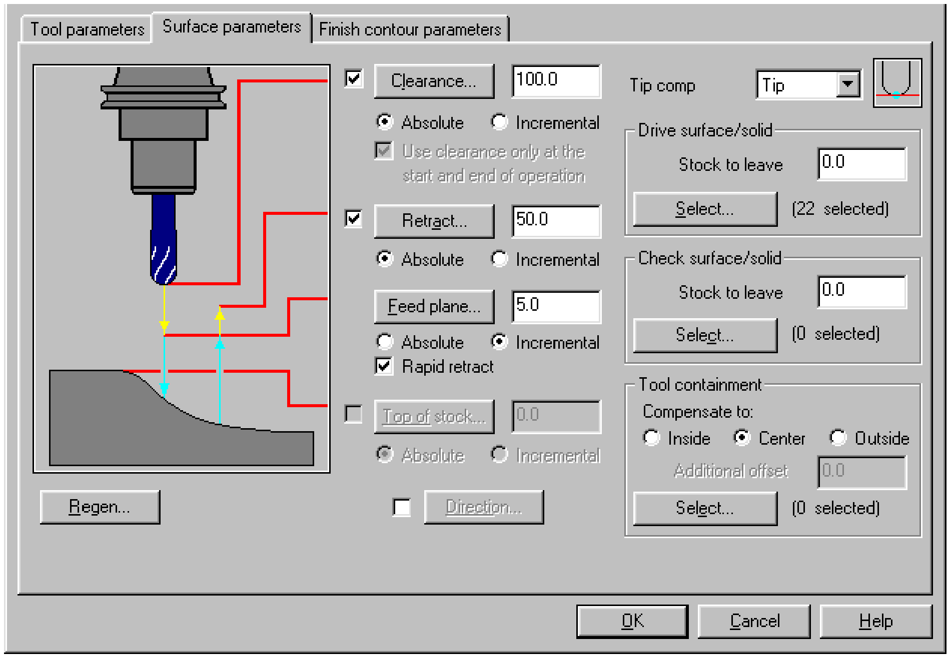
▶ *Creating the finish contour toolpath*

1. Open the file **finish contour-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **N**.

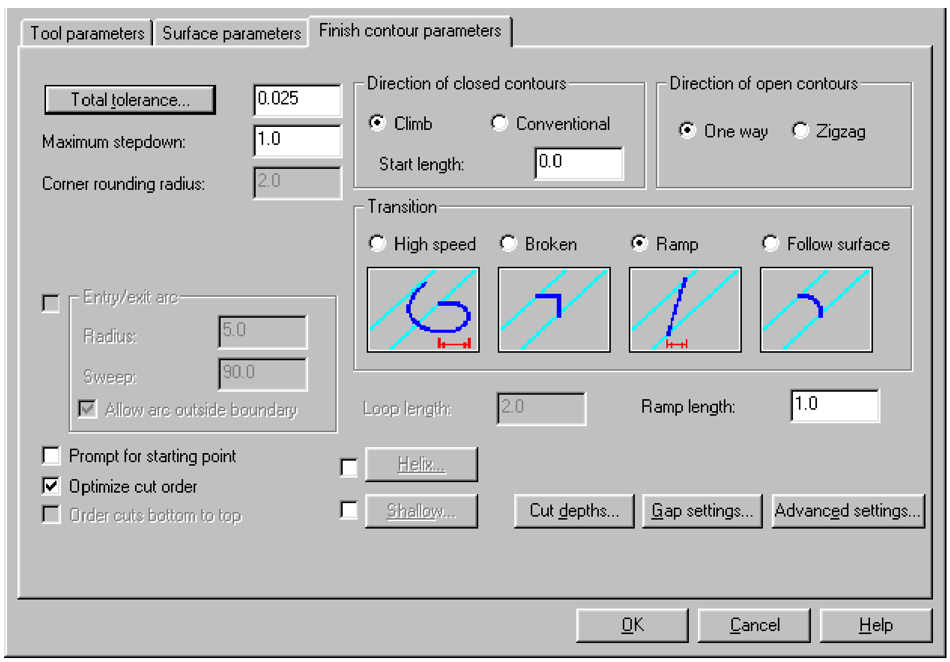
   Your surface selection menu options should match the following picture.



5. Choose **Finish, Contour**.
6. Right-click in the tool display area and select the 16 mm HSS bullnose endmill with 3 mm radius from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.

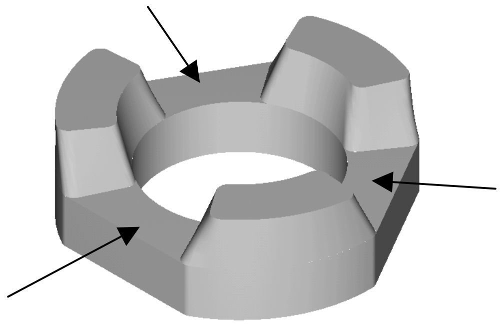


9. Select the **Finish contour parameters** tab.
10. Enter the values shown on the following dialog box.

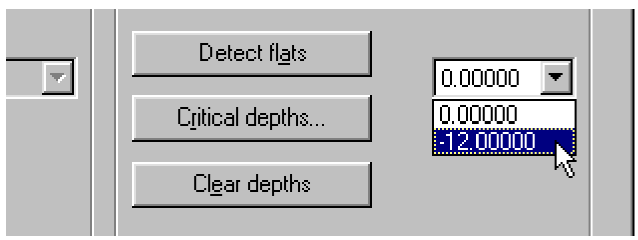


Note: The ramp length determines the size of the ramp between the constant Z depth cuts. This provides smooth motion between depth cuts, allowing for higher feed rates.

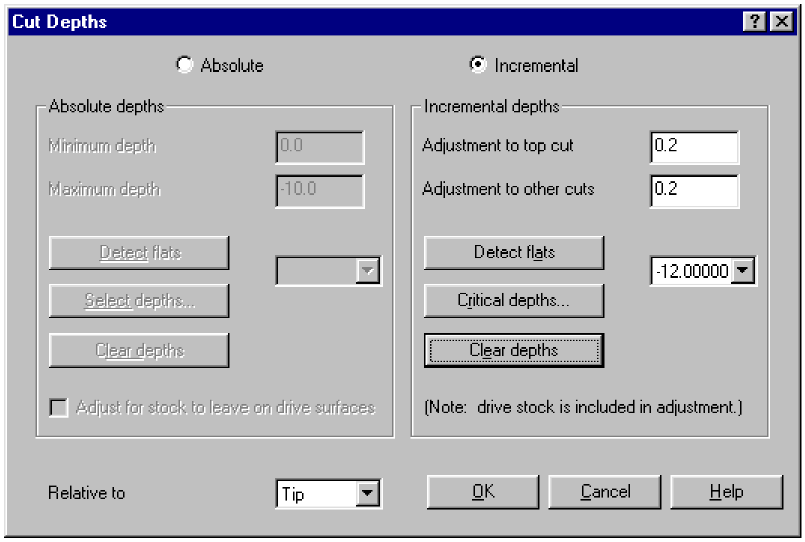
11. Choose **Cut depths**.
12. Choose **Detect flats** in the Incremental section. Mastercam automatically detects the flats shown in the following picture. When it creates the toolpath, it will create a cutting pass at this depth.



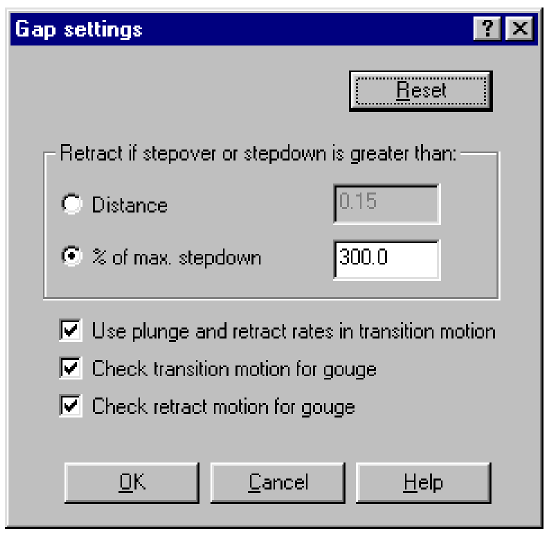
If you click on the list of depths, you can see the depth of the flats that it detected.



13. Make sure your other selections match the following dialog box.

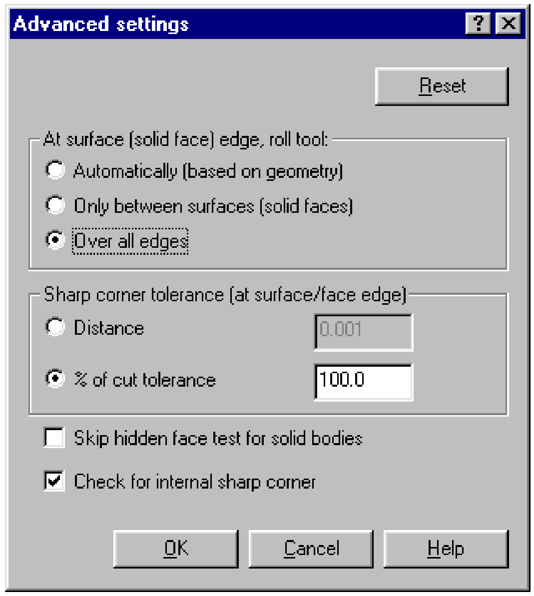


14. Choose **OK**.
15. Choose **Gap settings**.
16. Enter the values shown on the following dialog box.

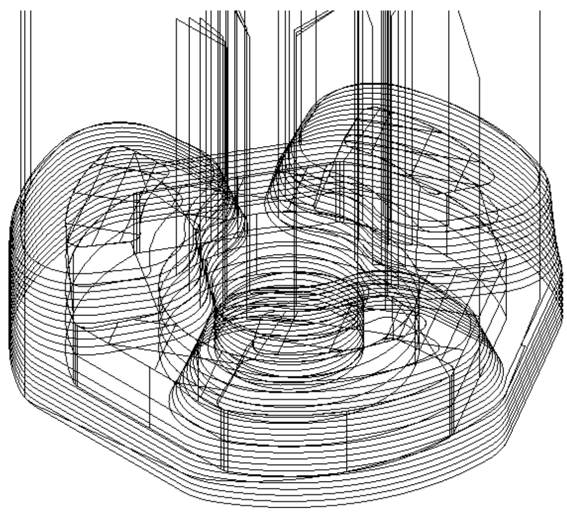


17. Choose **OK**.
18. Choose **Advanced settings**.
19. Enter the values shown on the following dialog box.

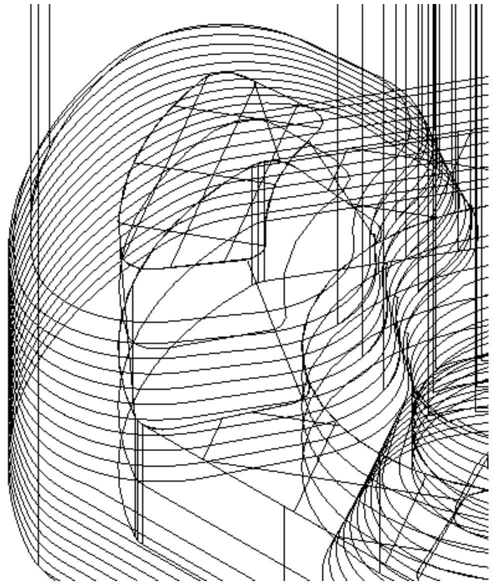
Tip: Setting the tool to roll over all surface edges results in better vertical wall recognition. Only the edge of a vertical wall is seen as an edge from the top, so this setting may be necessary in order for the vertical wall to be cut.



20. Choose **OK** twice. Mastercam generates the toolpath as shown in the following picture.



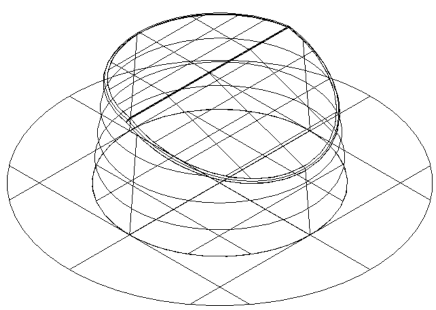
The following closeup shows the constant Z moves in greater detail.

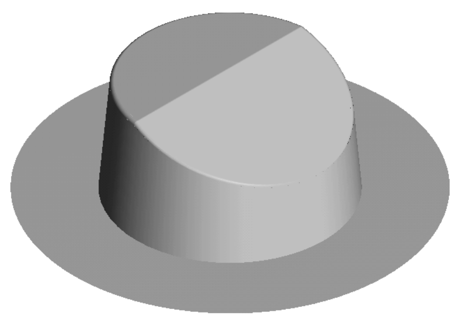


21. Save the part in your working folder as **finish contour 1.mc9**.

Exercise 5 – Creating a contour shallow toolpath

The finish contour toolpath that you created in the previous exercise has an option that gives you more control of the tool motion in shallow areas of a part. You can use this toolpath to reduce or increase the number of cuts in these areas. The cuts added to the shallow area can be partial or complete. A partial cut would cause tool motion to be added in only the shallow areas while a complete cut would cause tool motion to be added in shallow areas and possibly some steeper areas. The following pictures show the wireframe and surface geometry for the part you will use.





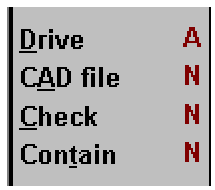
In this exercise, you will learn the following skill:

- **Optimizing a finish contour toolpath for a shallow part**

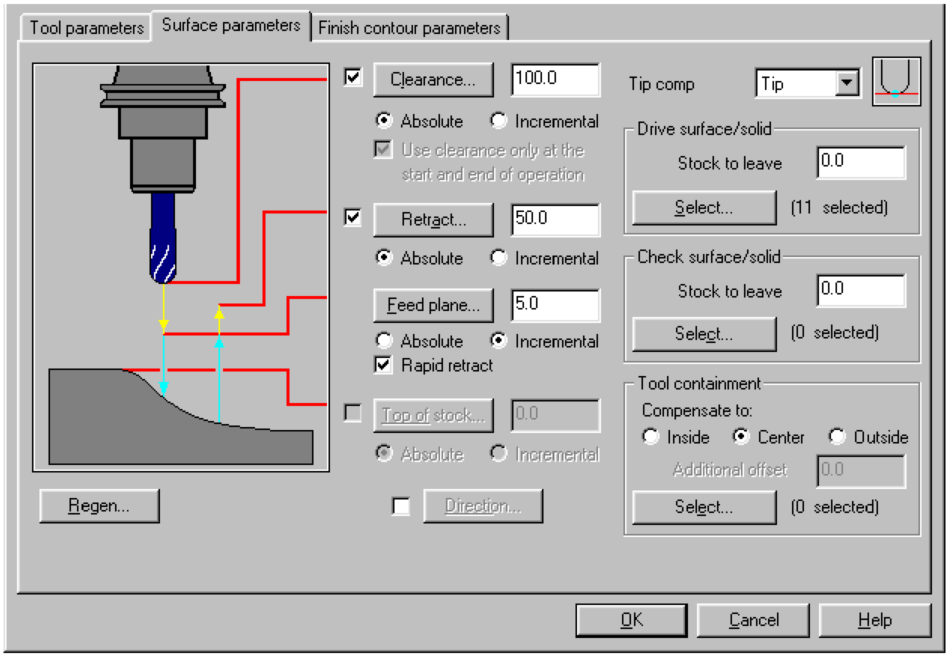
▶ *Creating the contour shallow toolpath*

1. Open the file **add cuts-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **N**.

   Your surface selection menu options should match the following picture.

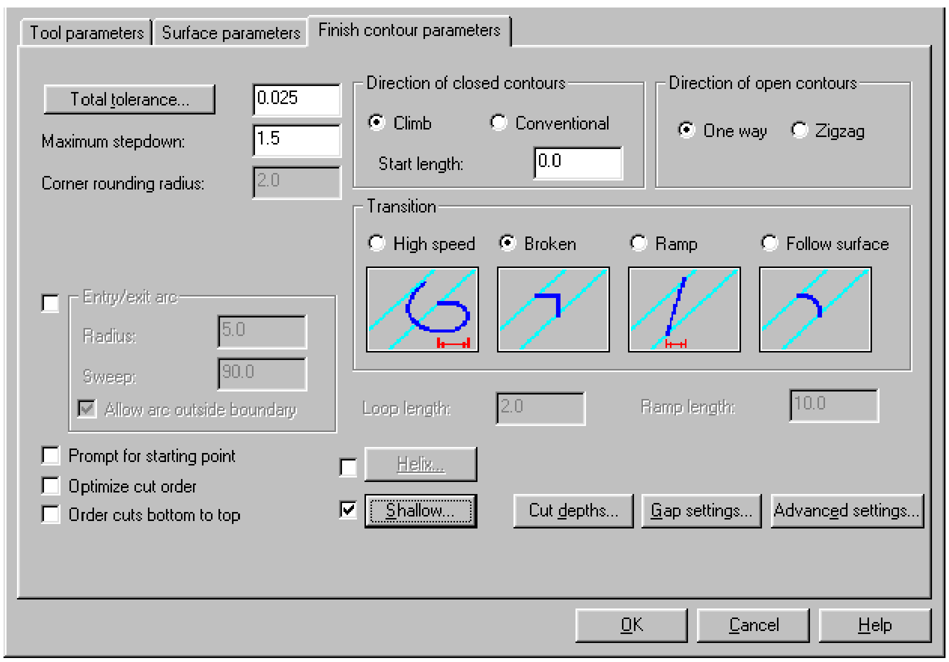


5. Choose **Finish, Contour**.
6. Right-click in the tool display area and select the 18 mm HSS flat endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.



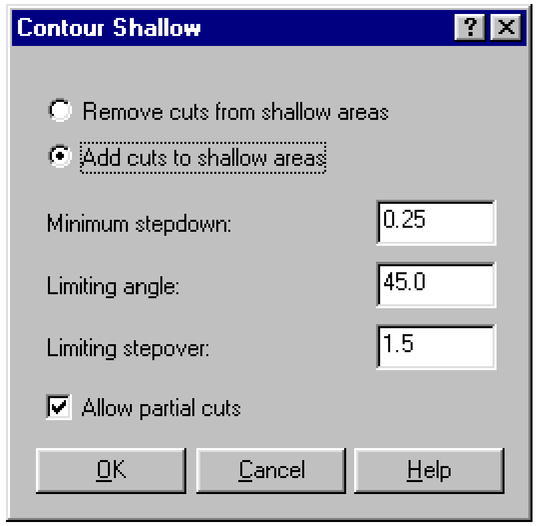
9. Select the **Finish contour parameters** tab.

10. Enter the values shown on the following dialog box.

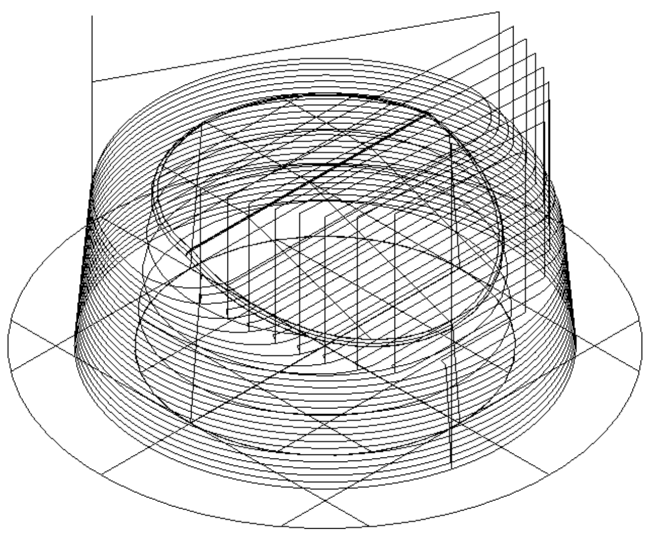


11. Choose the **Shallow** check box and button.
12. Enter the values shown in the following dialog box.

Tip: The Minimum stepdown value adds more cuts in shallow areas by stepping down the minimum amount.



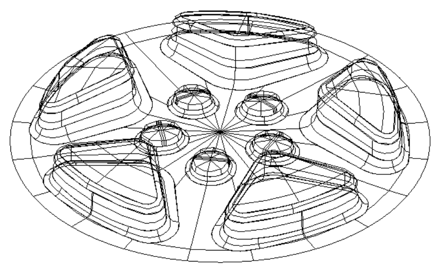
13. Choose **OK** twice. Mastercam generates the toolpath as shown in the following picture.

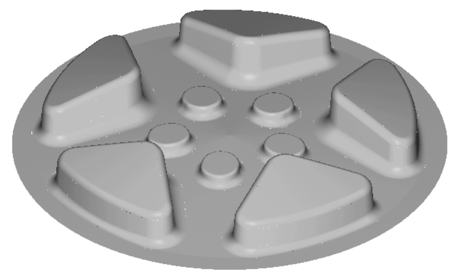


14. Save the file in your working folder as **finish contour shallow.mc9**.

Exercise 6 – Creating a finish scallop toolpath

For the part in this exercise, finish scallop creates a consistent scallop height over the whole part regardless of whether the surface becomes steep or shallow. Mastercam creates the consistent scallop height without having to double the step size in the steep areas. The following pictures show the wireframe and surface geometry for the part you will use.





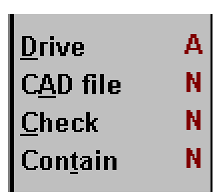
For this exercise, you will learn the following skills:

- **Creating a finish scallop toolpath**
- **Using the collapse resolution to fine-tune the toolpath**

▶ *Creating the finish scallop toolpath*

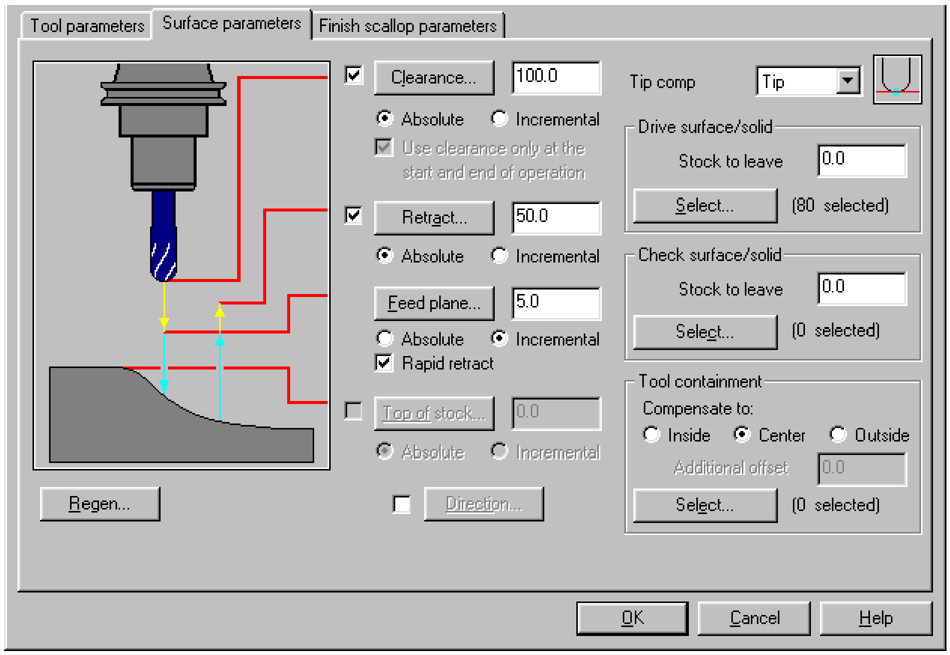
1. Open the file **scallop-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **N**.

   Your surface selection menu options should match the following picture.

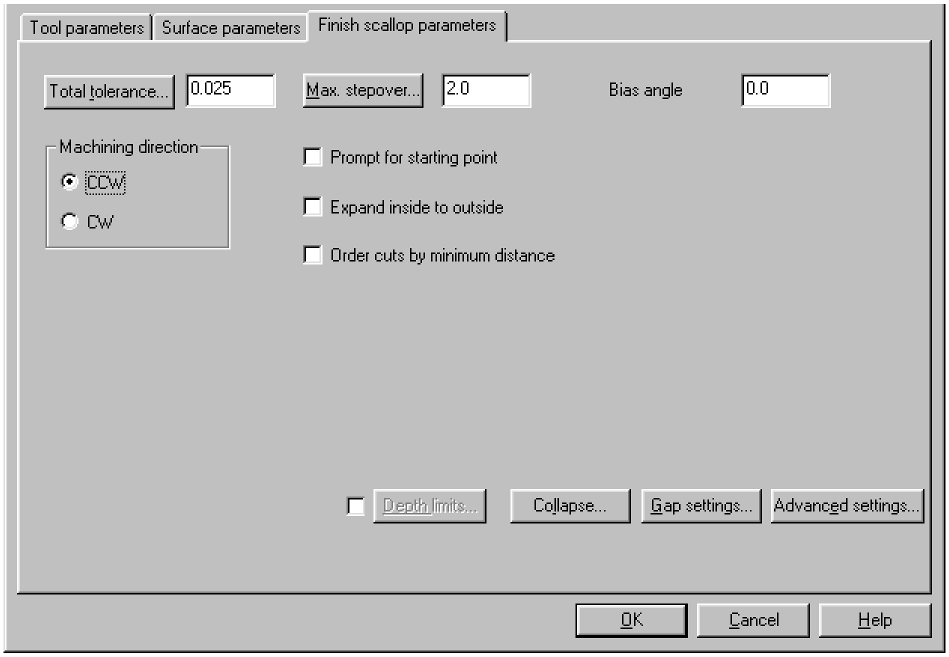


5. Choose **Finish, Scallop**.

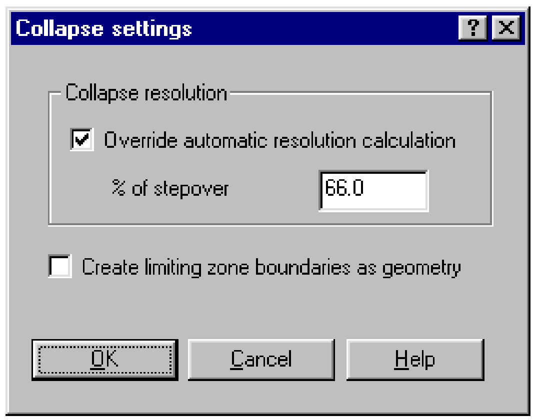
6. Right-click in the tool display area and select the 6 mm HSS ball endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.



9. Select the **Finish scallop parameters** tab.
10. Enter the values shown on the following dialog box.



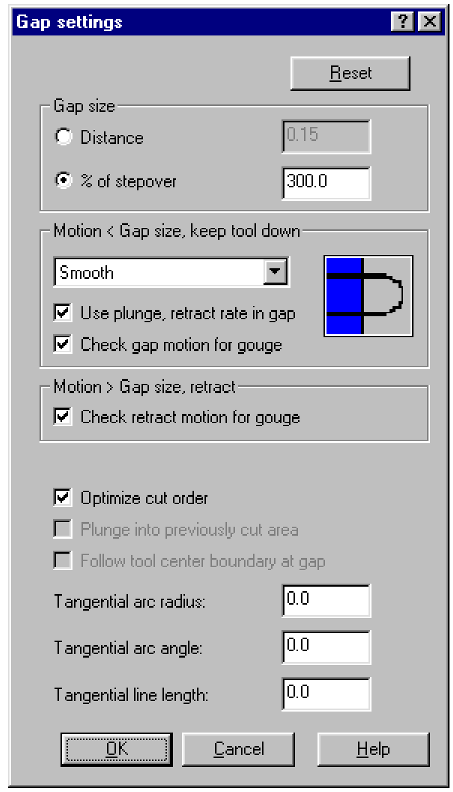
11. Choose **Collapse**.
12. Enter the values shown in the following dialog box.



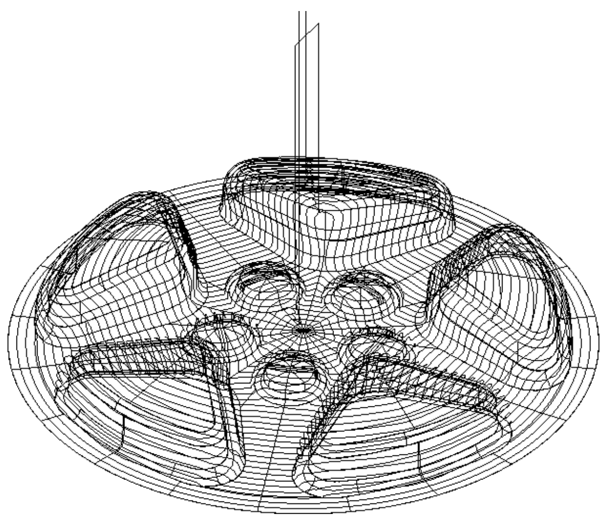
Tip: The collapse resolution defines how smoothly the collapse zones of the 3D collapse toolpath are created. The system uses the stepover percentage to create a "mesh" over the surfaces that determines where the toolpath is placed. A smaller collapse resolution creates a tighter mesh and a more accurate toolpath, but it also takes longer to generate and makes a longer NC program.

13. Choose **OK.**

14. Choose **Gap settings**.
15. Enter the values shown in the dialog box at right.



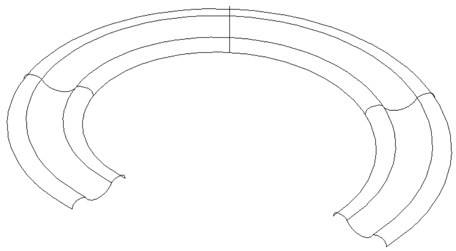
16. Choose **OK** twice. Mastercam generates the toolpath as shown in the following picture.

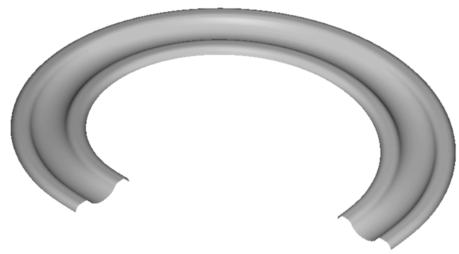


17. Save the part in your working folder as **finish scallop.mc9**.

Exercise 7 – Creating a finish flowline toolpath

Finish flowline toolpaths follow the shape and direction of the surfaces and create a smooth and flowing toolpath motion. A finish parallel toolpath machines the part at a set angle and does not flow with the surfaces, resulting in a lot of air cutting. The following pictures show the wireframe and surface geometry for the part you will use.





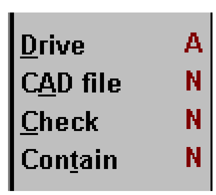
In this exercise, you will learn the following skill:

- **Creating a finish flowline toolpath**

▶ *Creating the flowline toolpath*

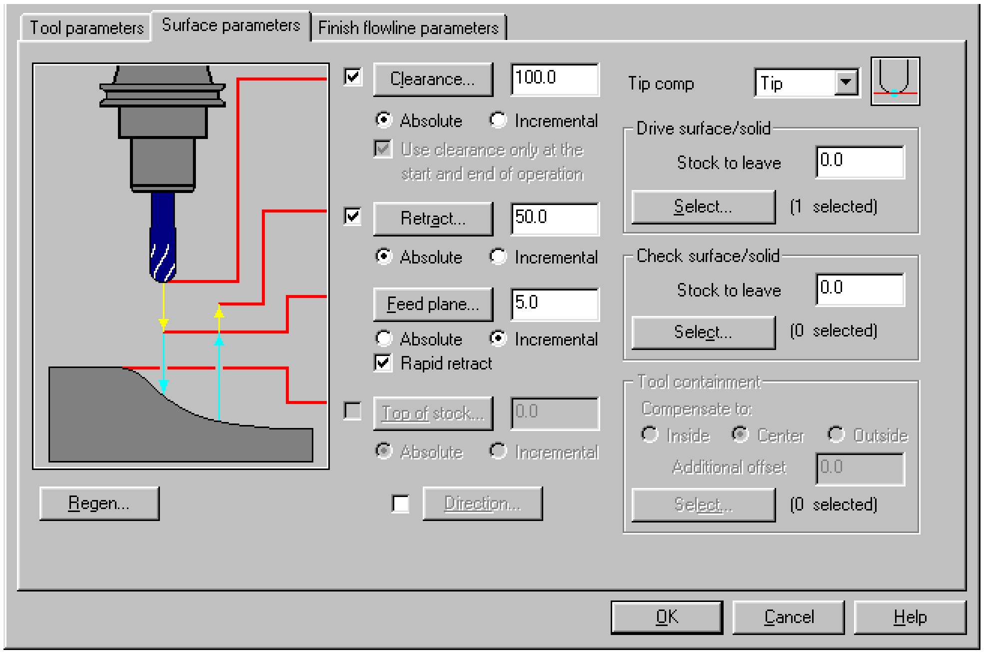
1. Open the file **flowline-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Surface**.
3. Toggle the **Drive** setting to **A**.
4. Toggle **Contain** to **N**.

   Your surface selection menu options should match the following picture.

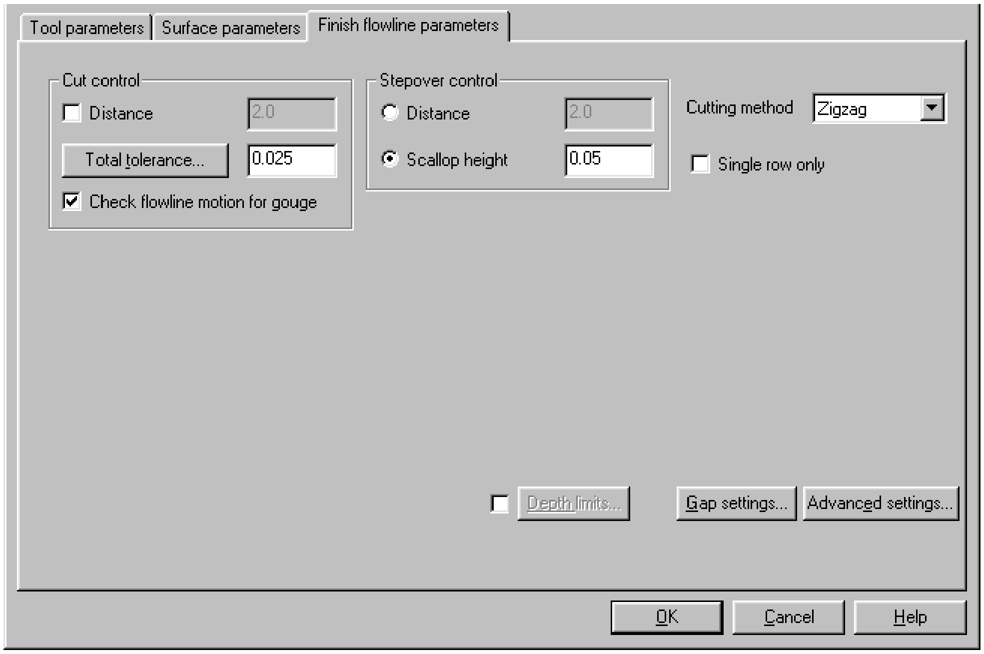


5. Choose **Finish, Flowline**.

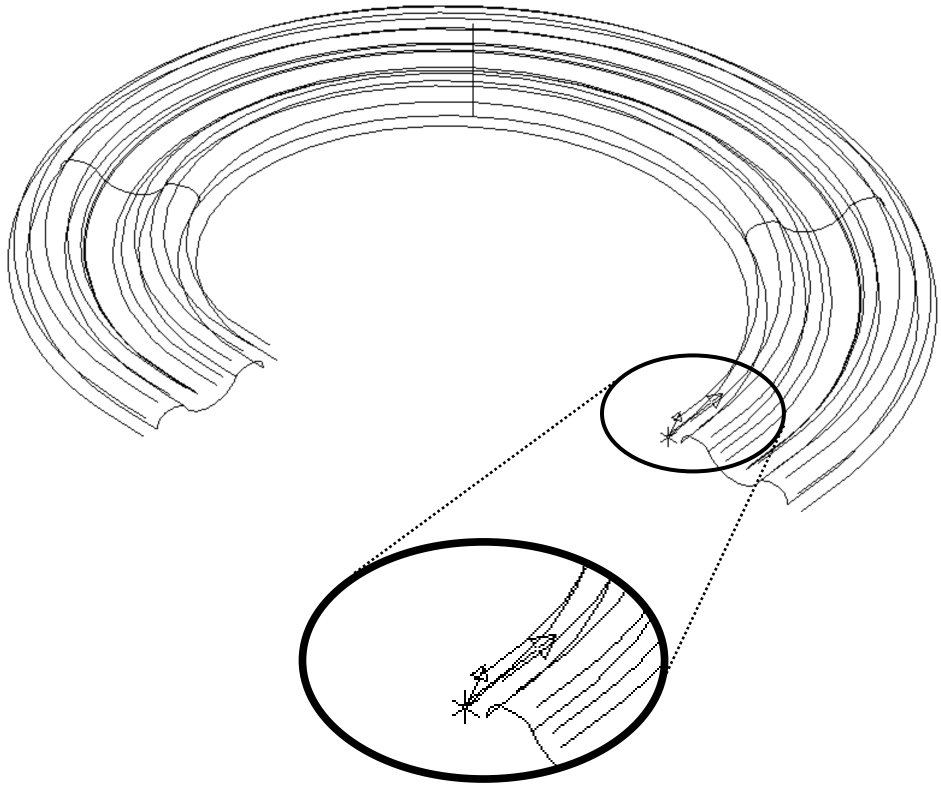
6. Right-click in the tool display area and select the 5 mm HSS ball endmill from the tool library.
7. Select the **Surface parameters** tab.
8. Enter the values as shown on the following dialog box.



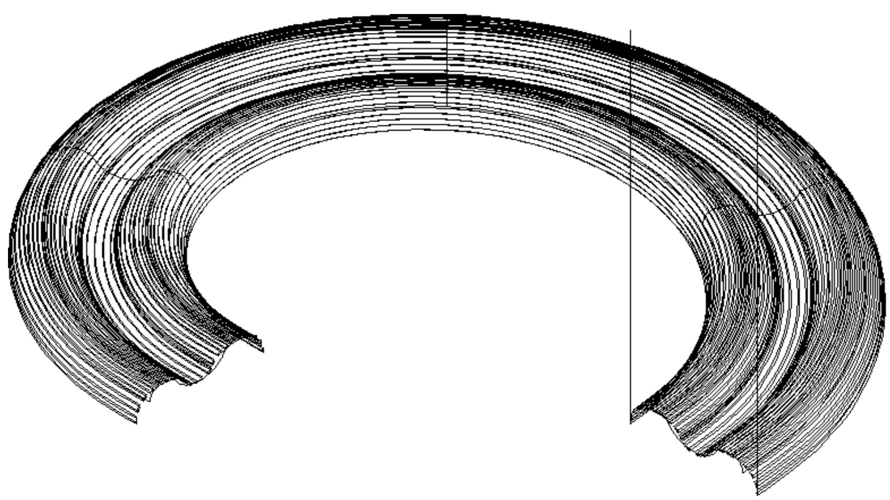
9. Select the **Finish flowline parameters** tab.
10. Enter the values shown on the following dialog box.



11. Choose **OK**.
12. You will return to the graphics window, with the Flowline menu displayed. Use the options on the menu to orient the cutting motion. Toggle the **Cut dir** setting so that the flow lines and starting arrows look like the following picture.



13. Choose **Done**. Mastercam generates the toolpath as shown in the following picture.



14. Save the file in your working folder as **finish flowline.mc9**.

In this chapter and the previous chapters, you have seen how Mastercam's surface machining toolpaths give you many options for tailoring toolpaths to your part geometry. The next chapter will show you how Mastercam can use the multiaxis capabilities of your machine tool to produce even more sophisticated surfacing toolpaths.

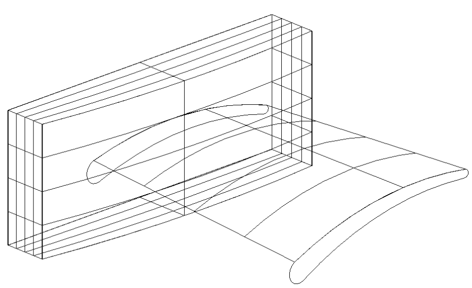
16 Creating Multiaxis Toolpaths

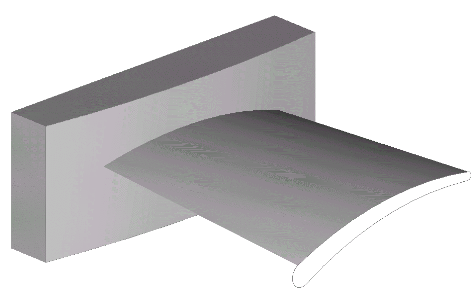
A multiaxis toolpath can be a 3-, 4-, or 5-axis toolpath. With a 3-axis toolpath, you can machine a part on one side, turn it over, and machine the other side. With a 5-axis toolpath, you can machine a part without having to turn the part over manually. This chapter introduces you to two different types of multiaxis toolpaths: a curve 5-axis toolpath and a swarf 5-axis toolpath.

Exercise 1 – Creating a curve 5-axis toolpath

This exercise shows you how to use 3D curves as an option in a 5-axis toolpath. This toolpath is useful for a part with curved surfaces because it allows for precise tool tip control. The part in this example, a blade and root part, has curved surfaces. Unlike the 3-axis contour toolpath, the curve 5-axis toolpath allows for more precise contact between the tool and the surface material.

The following pictures show wireframe and shaded views of the part.



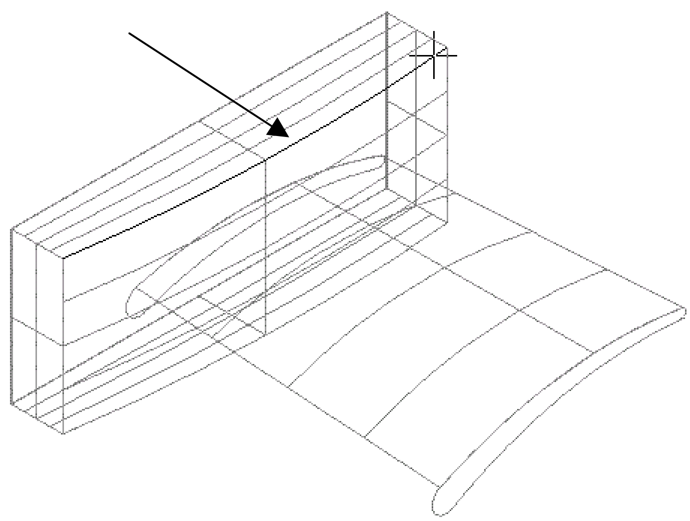


In this exercise, you will learn the following skills:

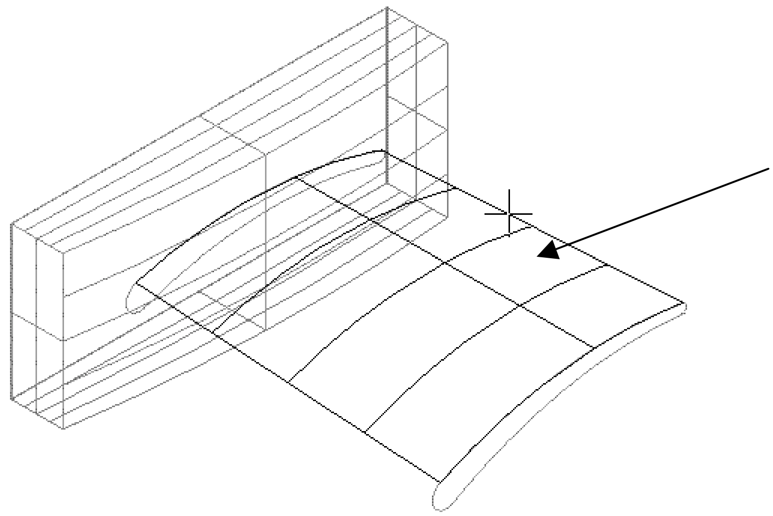
- **Creating a curve 5-axis toolpath with 3-axis output**
- **Creating a curve 5-axis toolpath with 5-axis output**

▶ *Creating the toolpath with 3-axis output*

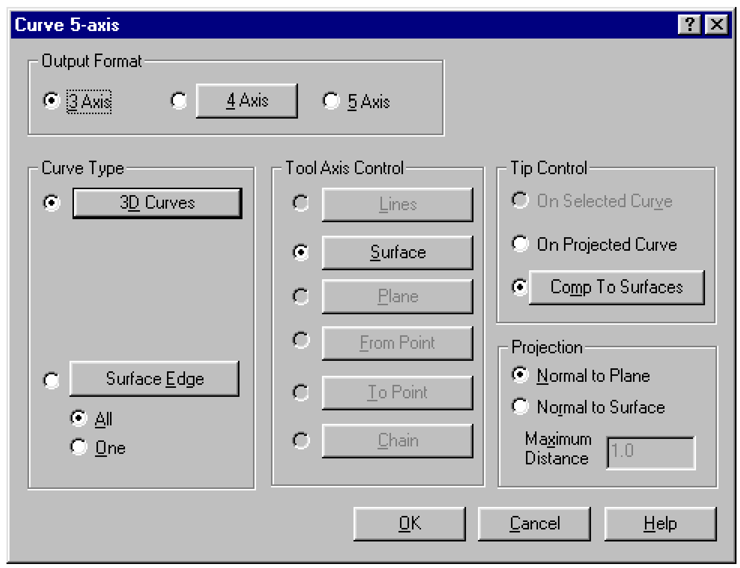
1. Open **curved blade-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Multiaxis, Curve5ax**.
3. Choose **Output Format – 3 Axis**.
4. Choose the **3D Curves** button.
5. Choose **Single** from the menu.
6. Select the top edge of the rib as shown in the following picture.



7. Choose **Done** to return to the Curve 5-axis dialog box.
8. Choose the **Comp to Surfaces** button.
9. Select the blade surface as shown in the following picture and choose **Done**.

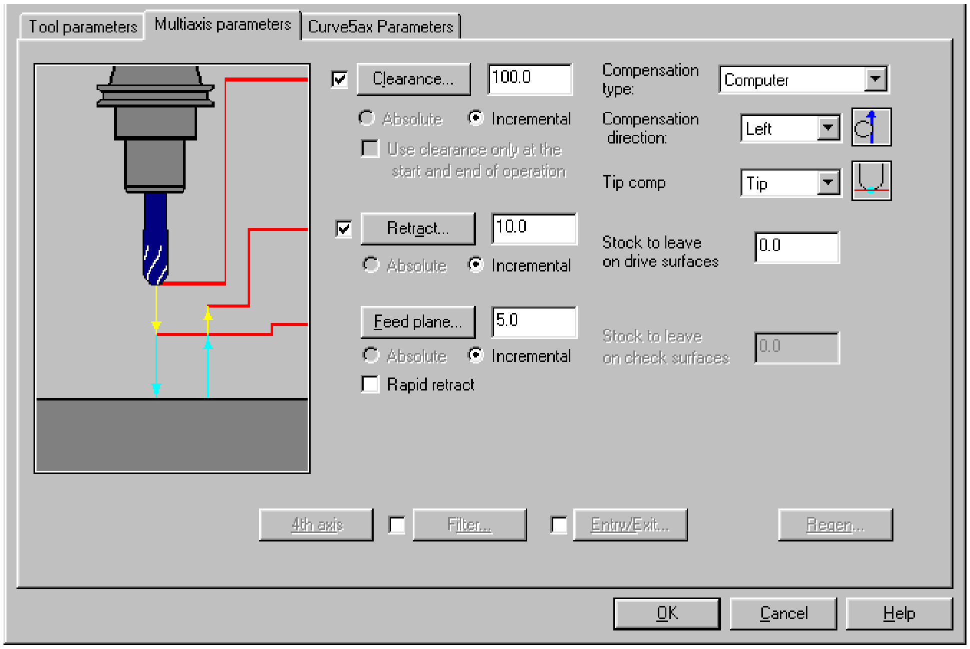


10. Make sure your other values match the following dialog box and choose **OK**.



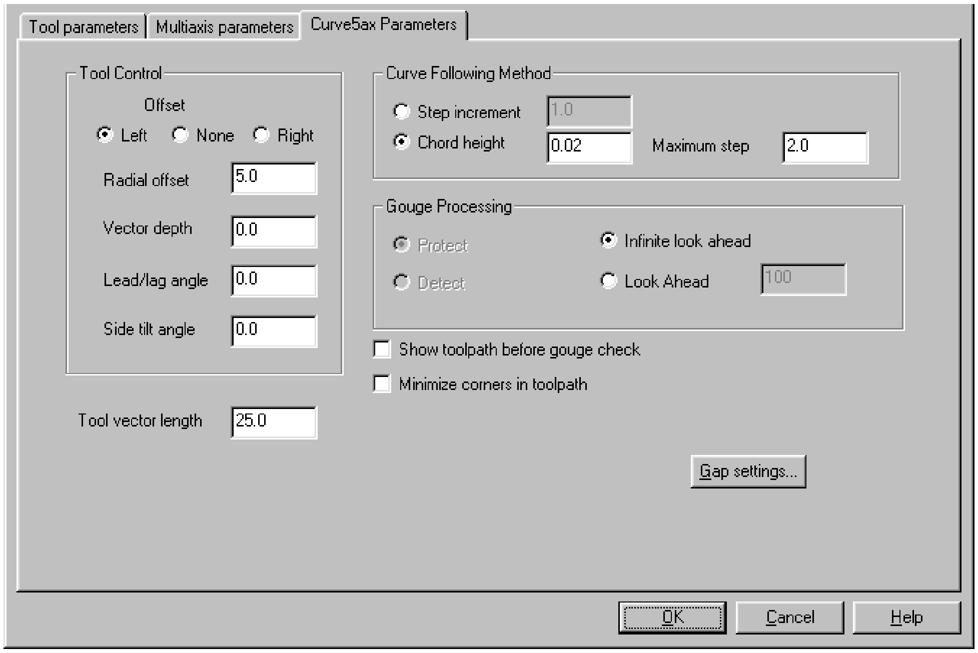
11. You are prompted to select a tool axis surface. Select the blade surface again as shown in step 9 and choose **Done**.
12. Right-click in the tool display area and choose the 10 mm HSS bullnose endmill with 3 mm radius from the tool library.
13. Select the **Multiaxis parameters** tab.

14. Enter the values shown on the following dialog box.

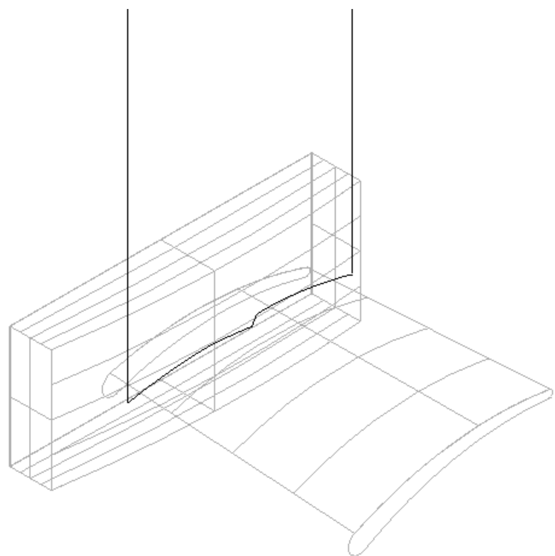


15. Select the **Curv5ax Parameters** tab.

16. Enter the values shown on the following dialog box.



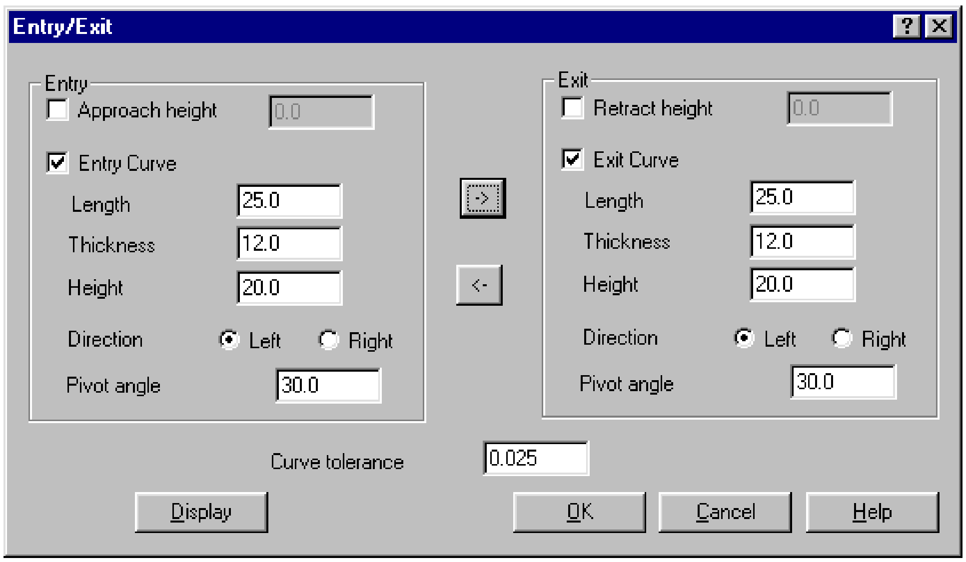
17. Choose **OK**. Mastercam generates the toolpath, which should look like the following picture.



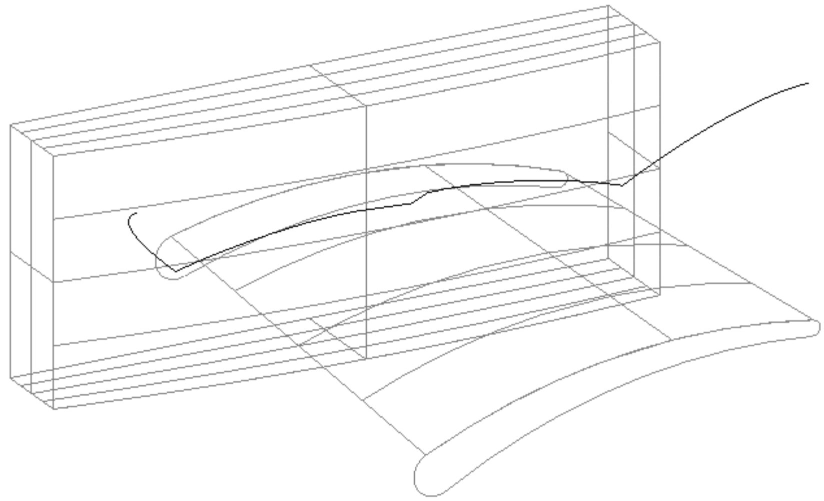
Adding entry/exit moves

The toolpath for the blade and root part is gouging the material upon entry. You can fix this by adding parameters for a smooth entry and exit.

1. Press [Alt + O] to open the Operations Manager.
2. Choose the **Parameters** icon for the toolpath.
3. Choose the **Multiaxis parameters** tab.
4. Choose the **Entry/Exit** check box and button.
5. Enter the values shown on the following dialog box.



6. Choose **OK** twice.
7. Choose **Regen Path**. The updated toolpath should look like the following picture.

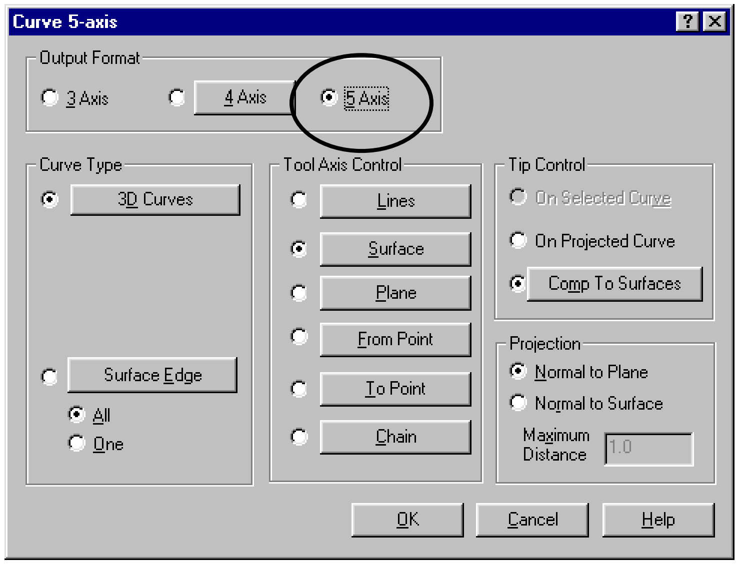


Note: This picture has been rotated with the Gview–Dynamic function to show the entry/exit moves more clearly.

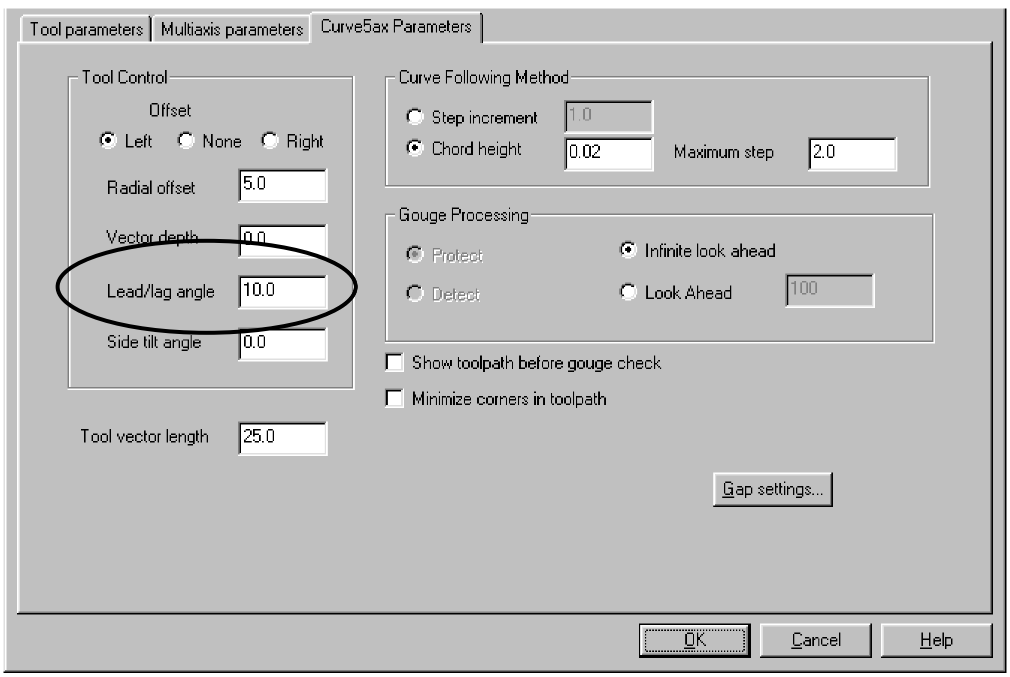
▶ *Changing the toolpath to 5-axis output*

Unlike the 3-axis toolpath you just created, the 5-axis toolpath lets the entire surface of the part control the orientation of the tool. Also, by setting a lead/lag angle for the tool, the tool can lean forward or backward for more effective cleanout.

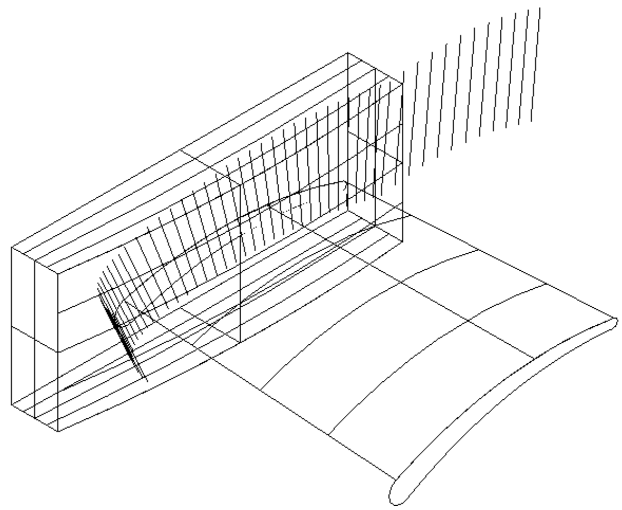
1. If necessary, press [Alt + O] to return to the Operations Manager.
2. Choose the **Geometry** icon for the toolpath.
3. Change the **Output Format** to **5 Axis** and choose **OK**.



4. Choose the **Parameters** icon for the toolpath.
5. Choose the **Curve5ax Parameters** tab.
6. Enter a **Lead/lag angle** of **10**.



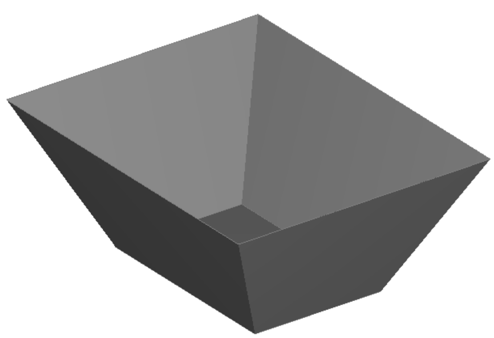
7. Choose **OK.**
8. Choose **Regen Path** to regenerate the toolpath. It should look like the following picture.

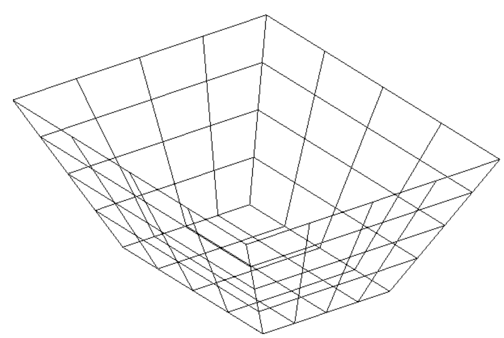


9. Choose **OK** to close the Operations Manager.
10. Save the file in your working folder as **5ax blade.mc9**.

Exercise 2 – Creating a swarf 5-axis toolpath

The swarf 5-axis toolpath uses the side of a tool to remove material from a pocket with tilted walls. The following pictures show the part in shaded and wireframe views.



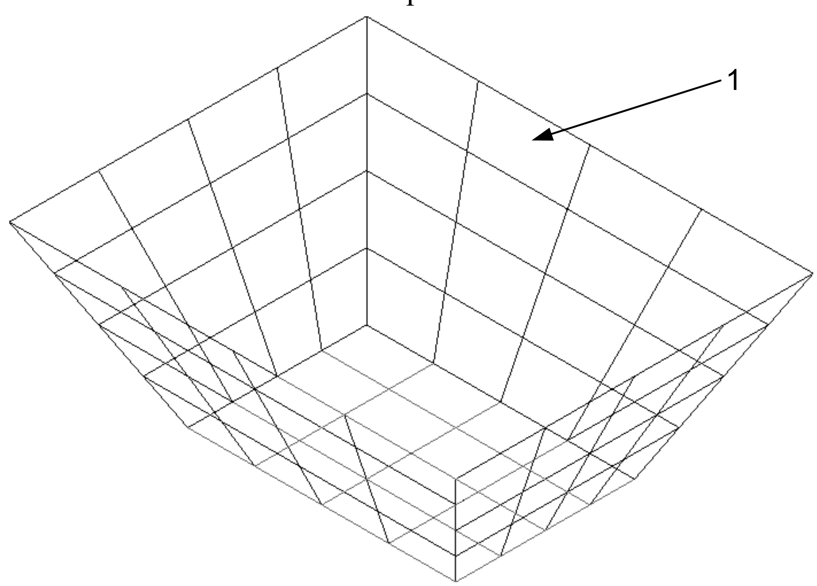


In this exercise, you will learn the following skills:

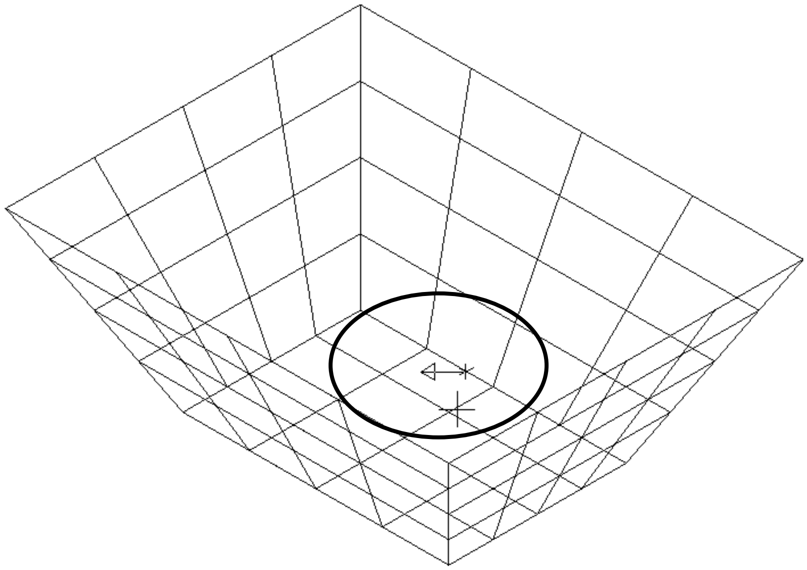
- **Creating a swarf 5-axis toolpath.**
- **Selecting geometry for tool tip control.**
- **Using the fan distance to control tool movement around the corners.**

▶ *Selecting the surfaces for the toolpath*

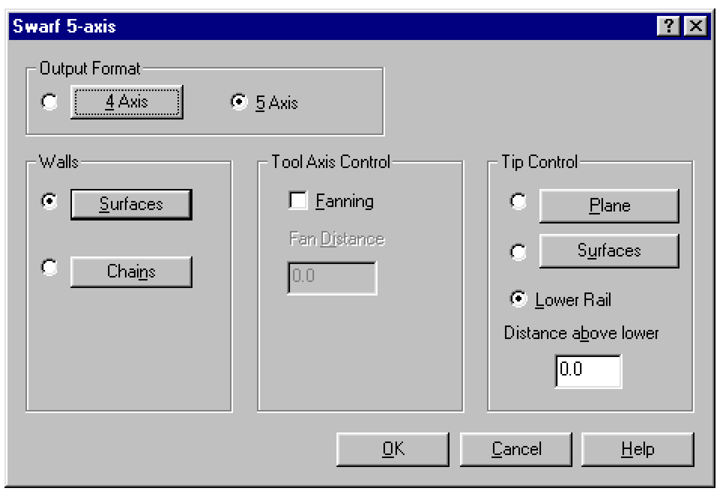
1. Open the file **swarf pocket-mm.mc9** from the the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Choose **Main Menu, Toolpaths, Multiaxis, Swarf5ax**.
3. Choose the **Surfaces** button in the **Walls** section. The dialog box closes so you can select the surfaces.
4. Select each of the four walls. Make sure you do not choose the floor.
5. Choose **Done**.
6. Mastercam prompts you to choose which surface will be machined first. Select at position 1.



7. Mastercam prompts you to identify the lower rail of the surface. Drag the mouse down until the arrow rests on the lower edge of the surface as shown in the following picture and click.



8. Choose **Flip** from the menu. The arrow should point left.
9. Choose **OK**. The **Swarf 5-axis** dialog box reopens. It should look like the following picture.

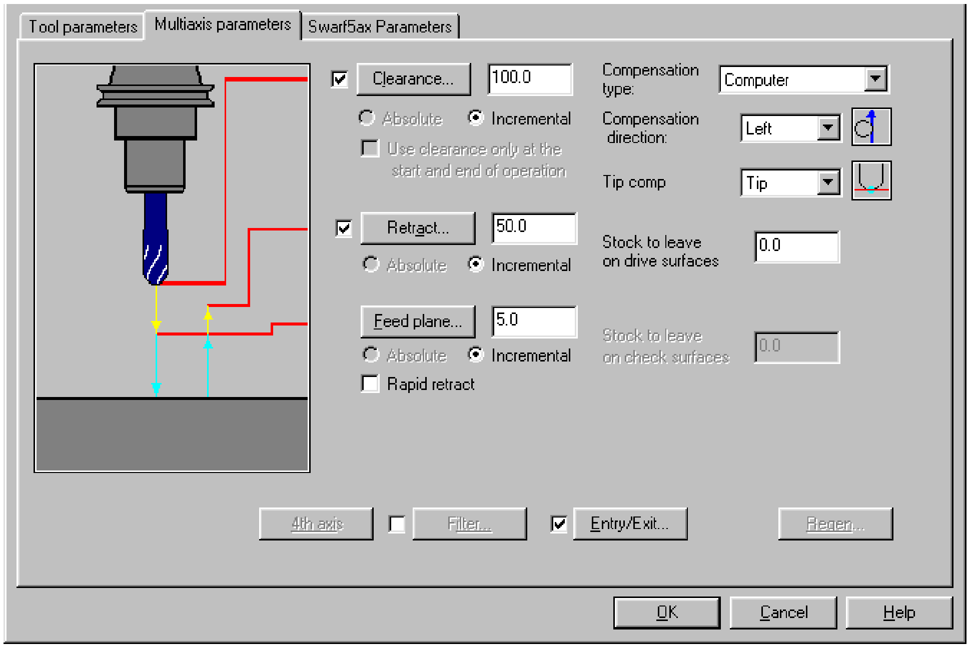


10. Choose **OK**.

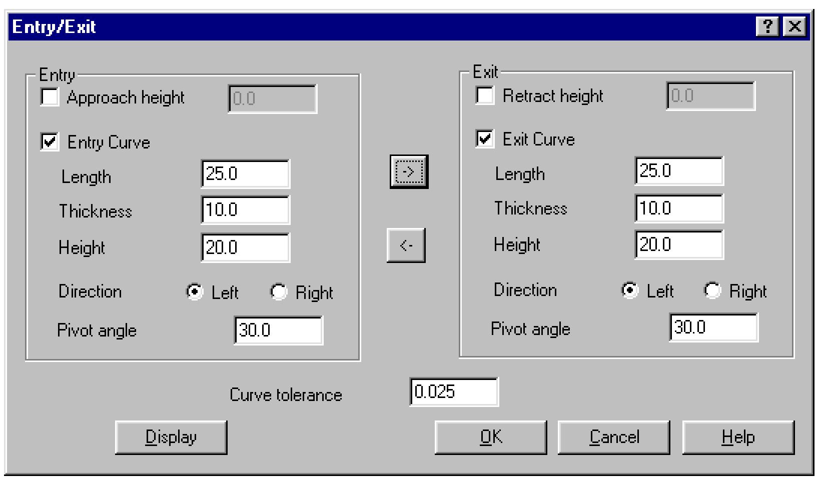
▶ *Selecting the tool and toolpath parameters*

1. Right-click in the tool display area and select the 12 mm bullnose endmill with the 1 mm corner radius from the tool library.

2. Select the **Multiaxis parameters** tab.
3. Enter the values shown on the following dialog box.

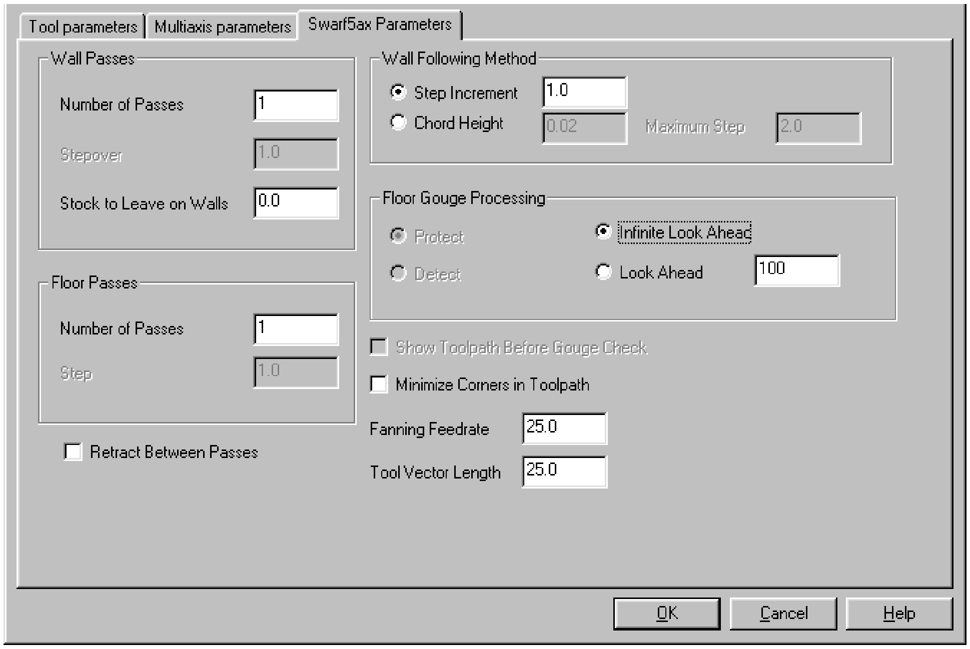


4. Choose the **Entry/Exit** check box and button.
5. Enter the values shown on the following dialog box.

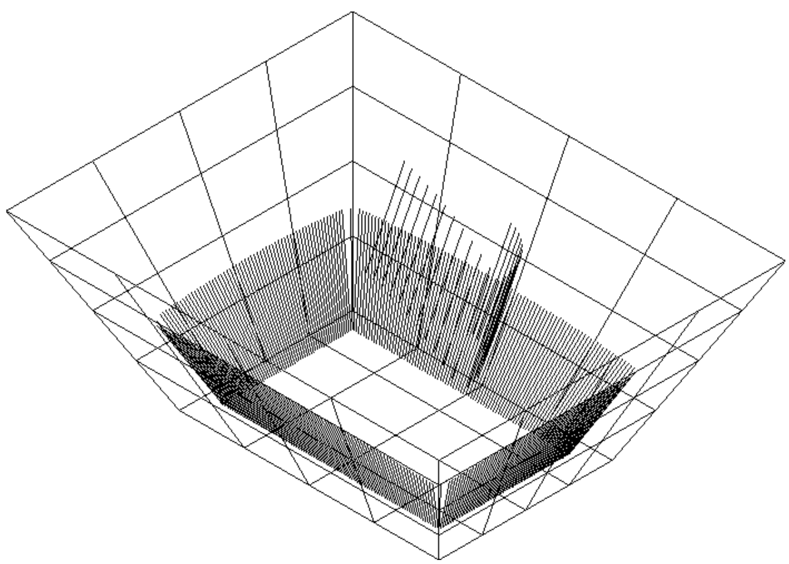


6. Choose **OK**.

7. Choose the **Swarf5ax Parameters** tab.
8. Enter the values shown on the following dialog box.



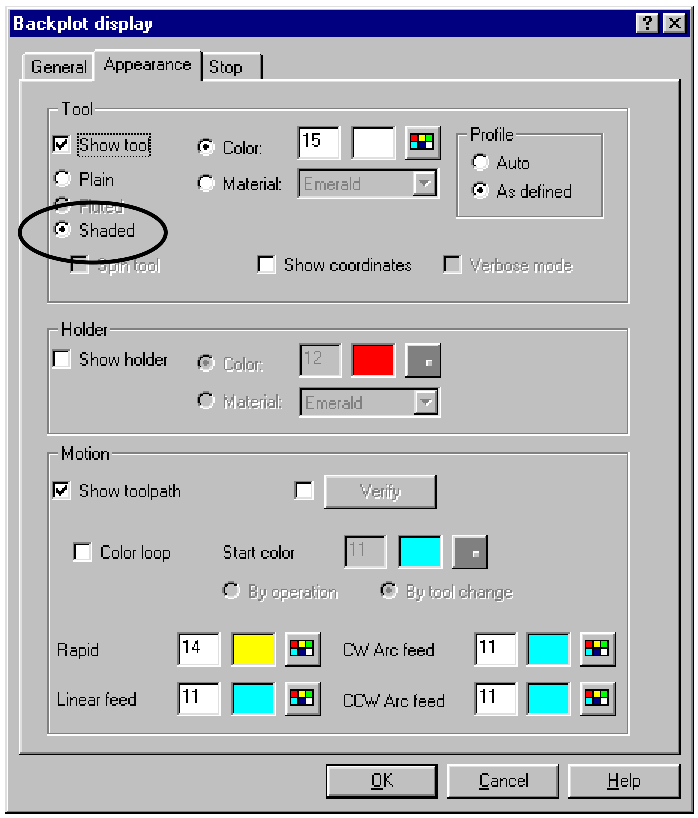
9. Choose **OK**. Mastercam generates the toolpath, which should look like the following picture.



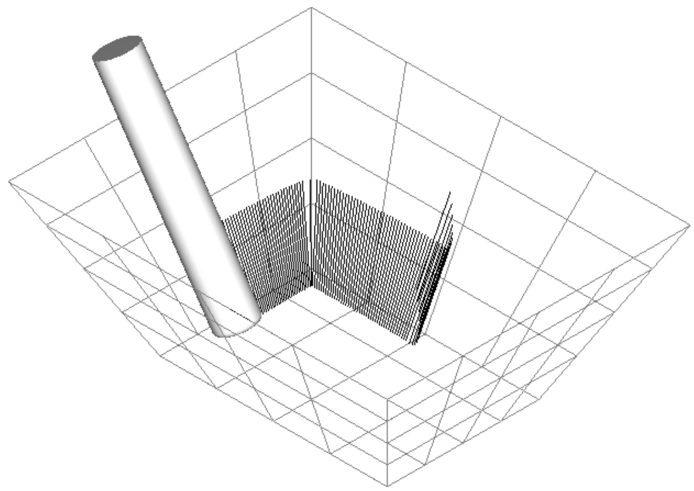
▶ *Backplotting the toolpath*

To get a better idea of how the side of the tool is cutting the sides of the pocket, backplot the toolpath.

1. Press [Alt + O] to return to the Operations Manager.
2. Choose **Backplot**.
3. Choose **Display**.
4. Choose the **Appearance** tab.
5. Choose the **Tool – Shaded** option as shown in the following picture.

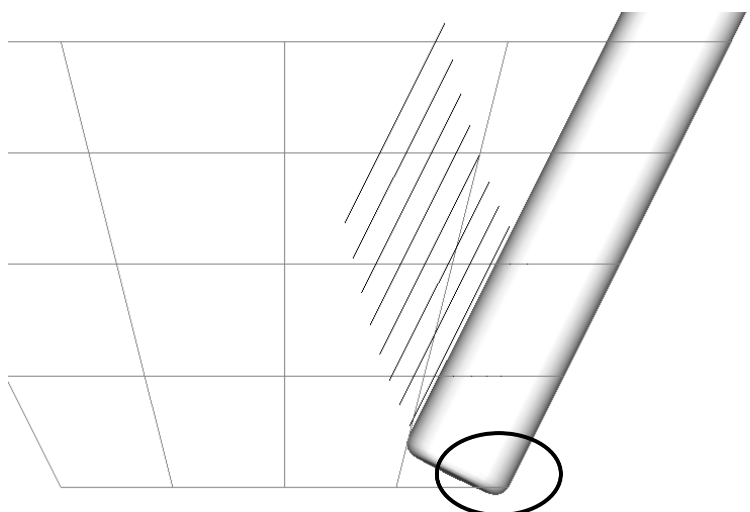


6. Choose **OK**.
7. Press [R] to preview the toolpath. You should see the tool angle around the pocket like in the following picture.



▶ *Adjusting the tool tip control*

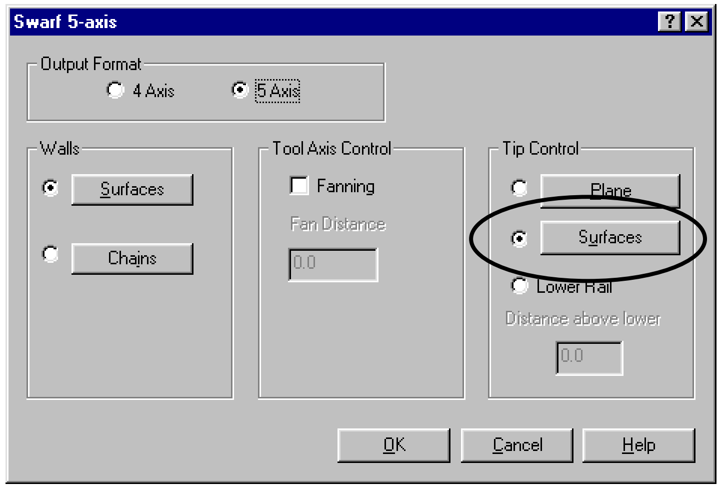
In the previous example, the tool tip control parameter was set to the lower rail, which is the line where the wall meets the floor. However, because the tool is at an angle, the tool will wind up gouging the floor, as shown in the following picture.



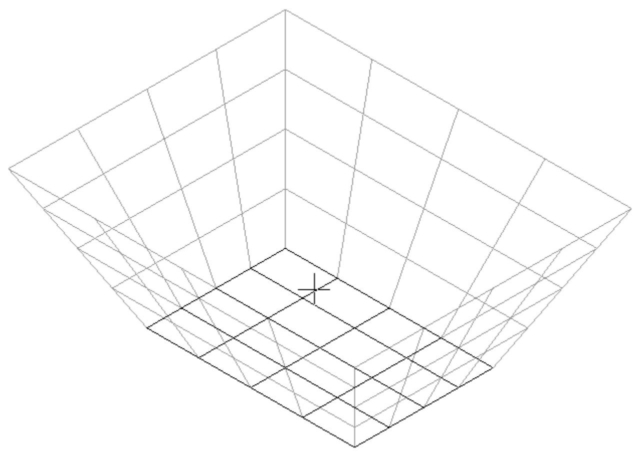
By setting the tool tip control to be the entire floor, Mastercam can prevent this.

1. If necessary, press [Esc] to return to the Operations Manager.
2. Choose the **Geometry** icon for the toolpath.

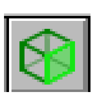
3. Choose the **Surfaces** button under Tip Control. The dialog box closes temporarily so you can choose a surface.



4. Choose the floor of the pocket as shown in the following picture.

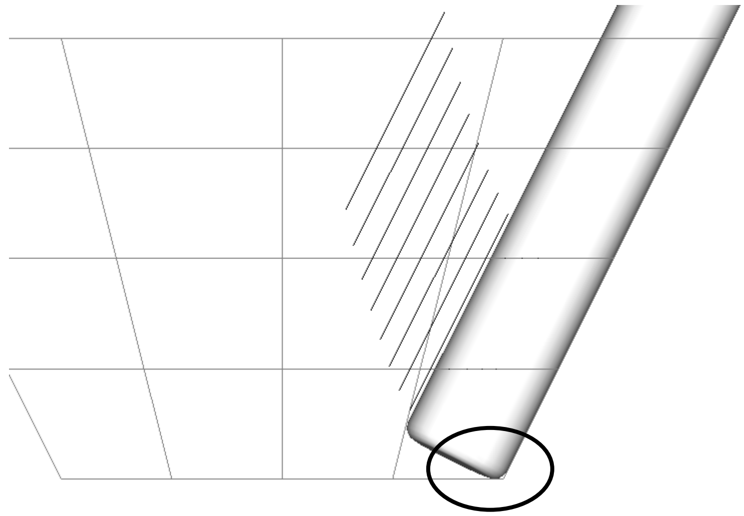


5. Choose **Done.**
6. Choose **OK** when the dialog box displays.
7. Choose **Regen Path**.
8. Choose **Backplot**.



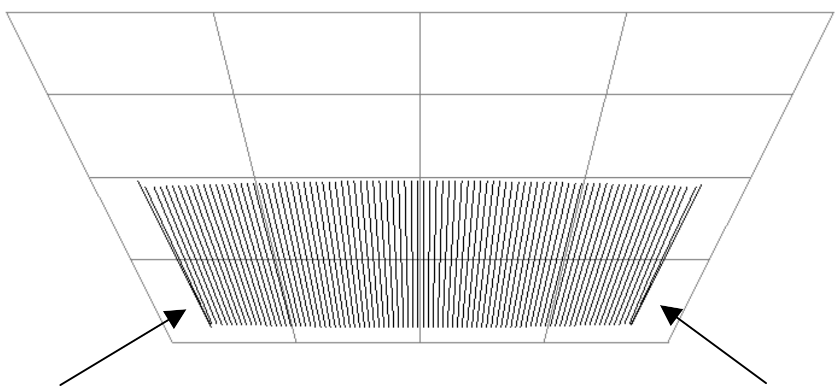
9. Choose the **Gview–Side** button from the toolbar and press [Page Up] several times to zoom in on the part, so you can see the tool approach the bottom.

10. Press [S] repeatedly to step through the toolpath. It should look like the following picture. Because the tool tip control is set to the entire bottom surface, not just the line, the tool tip never gouges the part.



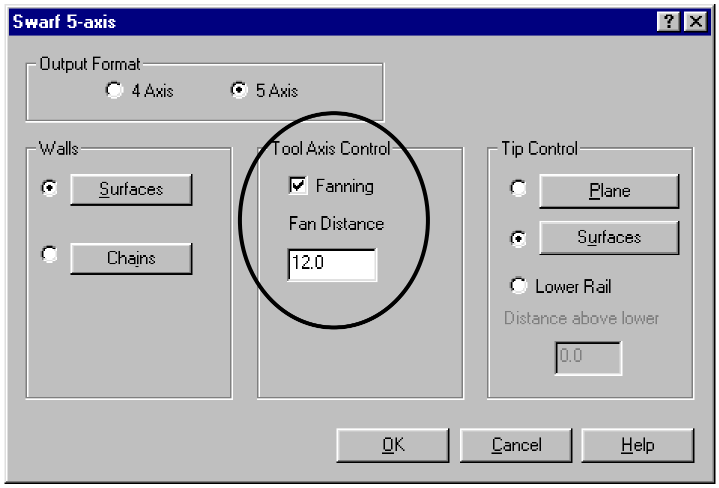
Optimizing the corner transition

In the current toolpath, as the the tool is moving along the walls, it is continually transitioning between the corner angles:

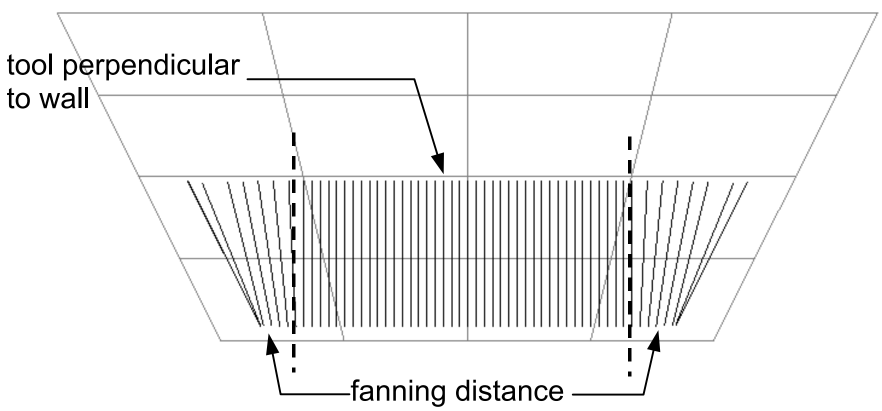


Mastercam has a fanning capability that you can use to specify a transition area. This way, the tool will be perpendicular to the floor while machining the walls. The tool only transitions to the corner angle when it reaches the fanning area.

1. Choose **Backup** to return to the Operations Manager.
2. Choose the **Geometry** icon for the toolpath.
3. Select **Fanning.**
4. Enter a **Fan Distance** of **12** as shown in the following picture.



5. Choose **OK**.
6. Choose **Regen Path**. As the toolpath machines the walls, it should look like the following picture.



7. Save the file in your working folder as **swarf.mc9**

This exercise completes your introduction to creating multiaxis toolpaths in Mastercam. You can learn about Mastercam's other types of multiaxis toolpaths—drill, multisurface, flowline, and rotary—in the online help. The next chapter, which is the final exercise in this tutorial, shows you an example of machining a solid and introduces Mastercam's feature-based solid drilling function.

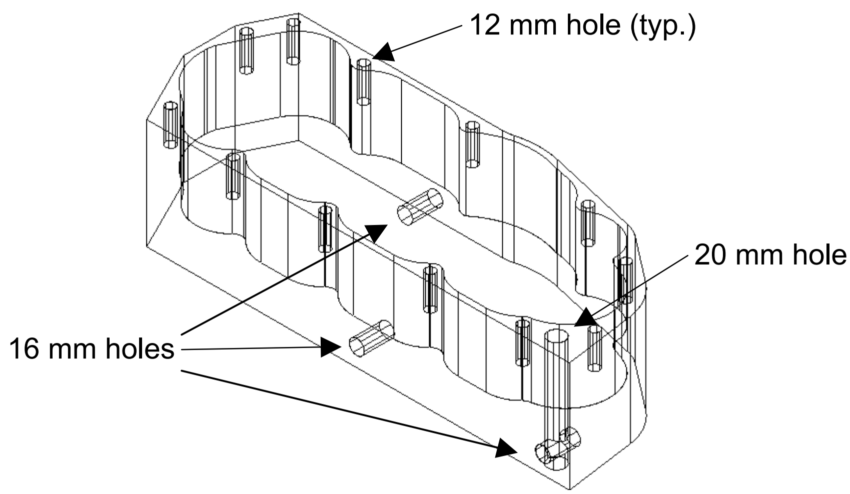
Note: You need a Mastercam Solids license to complete the next chapter .

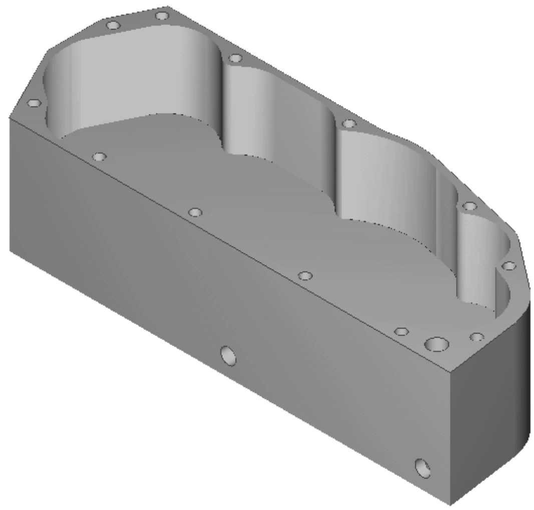
17 Machining Solids

This chapter introduces you to machining solid geometry. Solids are another type of Mastercam entity, like surfaces, lines, or arcs. You can either create them in Mastercam or import them from another program. This chapter does not show you how to create solids; see the *Mastercam Solids Tutorial* which came with your purchase of Mastercam Solids or the online help for more information about creating solid geometry.

The part you will machine in this chapter is a gearbox. The following pictures show wireframe and shaded views of the geometry. You will perform the following operations:

- pocket out the central cavity
- drill and tap the twelve 12 mm blind holes around the top edge
- drill the other holes





You will create a pocket toolpath to machine the cavity and use the Solid Drilling function to drill the holes.

Note: You need a Mastercam Solids license to complete the exercises in this chapter.

Exercise 1 – Machining the pocket

In this exercise, you will create roughing and finishing operations to machine the central cavity of the part. You will use the same type of pocket toolpath as you would for wireframe geometry, but apply it to the solid model.

In this exercise, you will learn the following skills:

- **Selecting solid geometry**
- **Creating a pocket toolpath using solids**
- **Creating separate roughing and finishing operations**

▶ *Selecting the solids and the pocketing parameters*

1. Open the file **gearbox-mm.mc9** from the folder **C:\Mcam9\Tutorials\Mill Tutorial\Metric**.
2. Press [Alt + S] to view the part as a shaded solid.
3. Choose **Main Menu, Toolpaths, Pocket, Solids**.

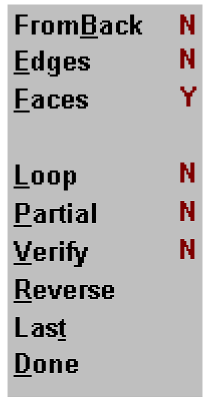
4. Toggle the menu options so that they match the following picture.

Tip: A chain for a solids toolpath is built from the edges of the solid faces. You can select a face to include all the edges around it, or you can select edges one at a time. Since solid models can be very complicated, use the menu options shown in the following picture to help you choose the proper geometry for your toolpath.

FromBack
Toggle this to **Y** to select faces or edges that are hidden or on the back of the model, so you don't have to rotate the part.

Edges
When set to **Y**, allows you to select edges. Set this to **N** if you only want to select faces.

Faces
When set to **Y**, allows you to select faces. Set this to **N** if you only want to select edges.

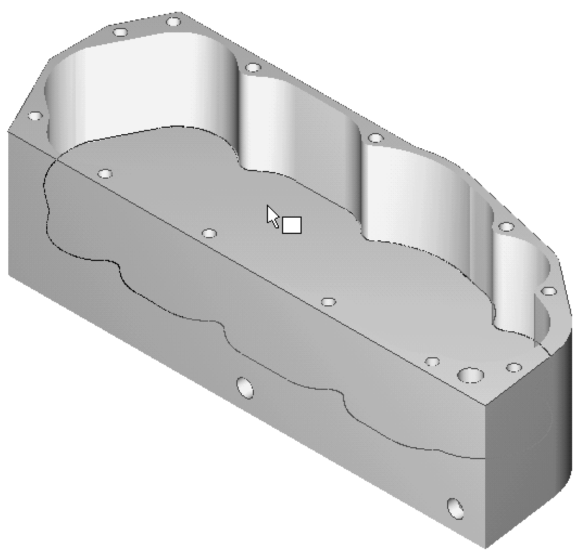


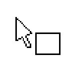
Loop
When set to **Y**, lets you select all the edges around a face.

Partial
Set this to Y to manually select the endpoint of the chain.

Verify
When this is set to Y, Mastercam will prompt you to select the proper face if there are several choices.

5. Select the floor of the pocket as shown in the following picture.

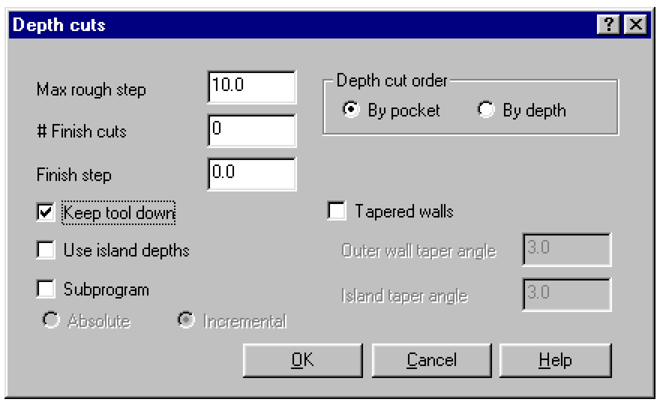


Tip: When a face is the active selection, the cursor looks like this: 

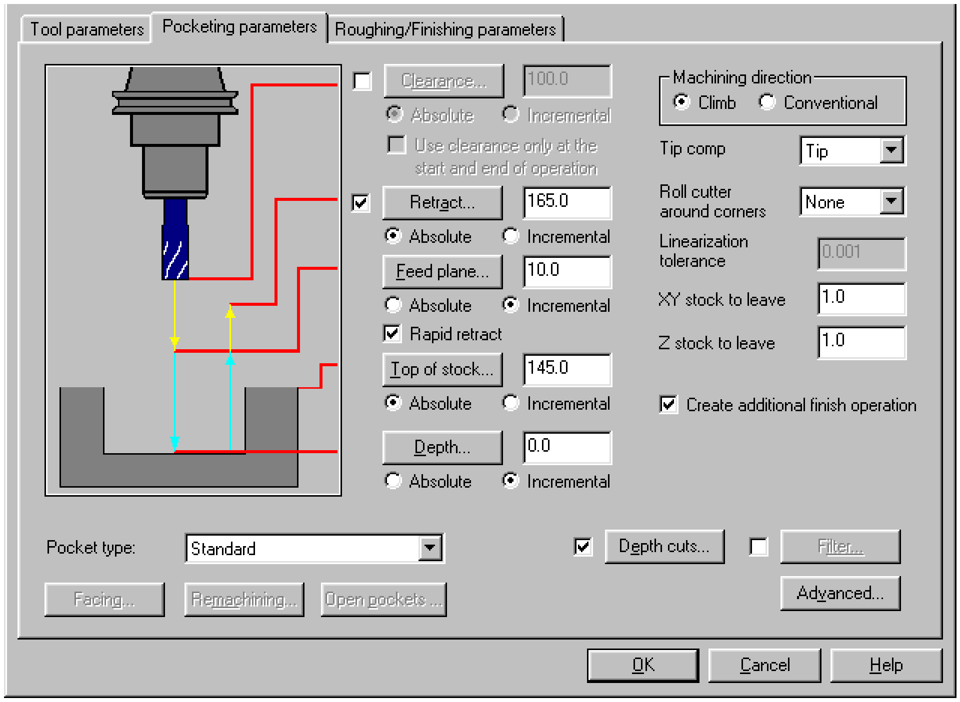
When an edge is the active selection, the cursor looks like this: 

6. Choose **Done** twice.

7. Right-click in the tool display area and select the 32 mm HSS flat endmill from the tool library.
8. Choose the **Pocketing parameters** tab.
9. Enter **165** for **Retract**.
10. Choose the **Incremental** option for **Depth**. The Depth should change to 0.0.
11. Enter **145** for **Top of stock**.
12. Enter **1.0** for both **XY stock to leave** and **Z stock to leave**.
13. Choose the **Create additional finish operation** check box.
14. Choose the **Depth cuts** check box and button.
15. Select **Keep tool down**.
16. Make sure your other values match the following picture and choose **OK**.

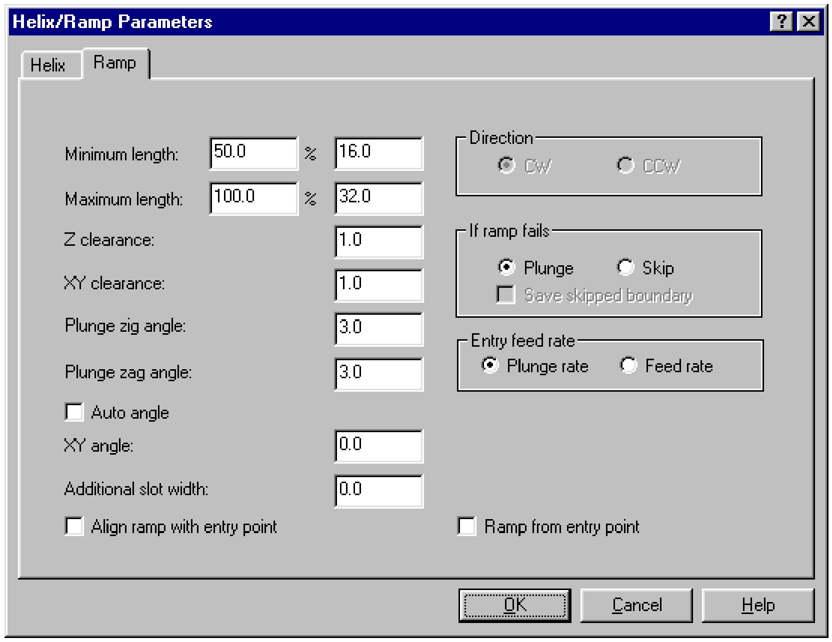


17. Verify that your other pocketing parameters match the following picture.

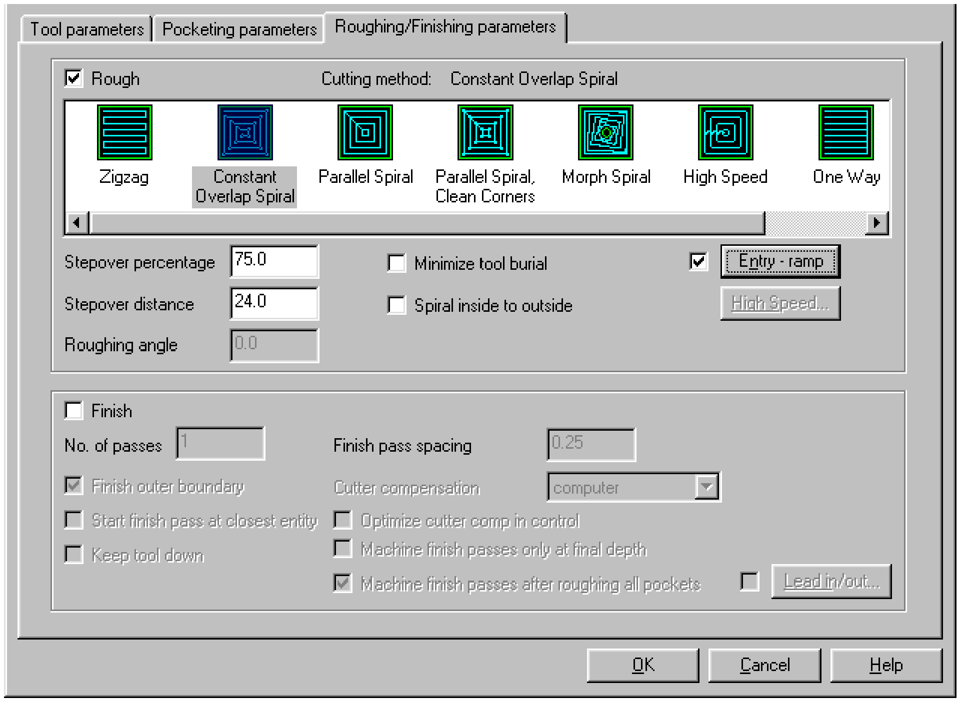


▶ *Selecting roughing and finishing parameters*

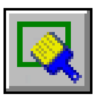
1. Choose the **Roughing/Finishing parameters** tab.
2. Choose the **Constant Overlap Spiral** cutting method.
3. Clear the **Finish** check box.
4. Choose the **Entry – ramp** (or **Entry – helix**) check box and button.
5. Choose the **Ramp** tab.
6. Enter the values shown on the following dialog box.



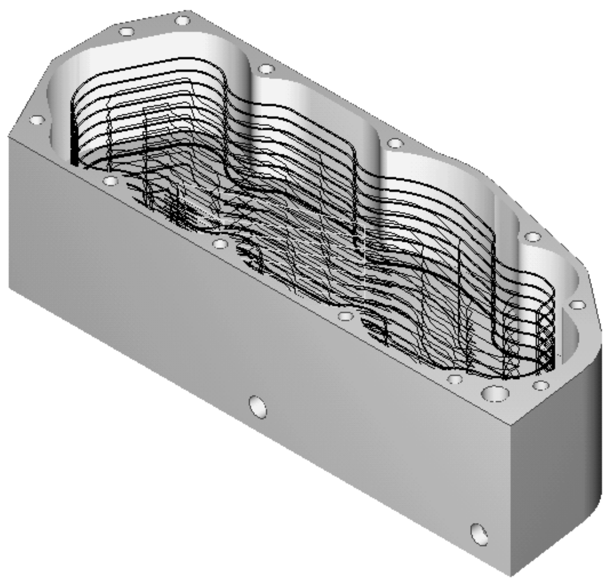
7. Choose **OK**.
8. Verify that your Roughing/Finishing parameters match the following picture and choose **OK**.



Mastercam generates the toolpath.



9. Choose the **Repaint** button from the toolbar to see the toolpath more clearly. It should look like the following picture.

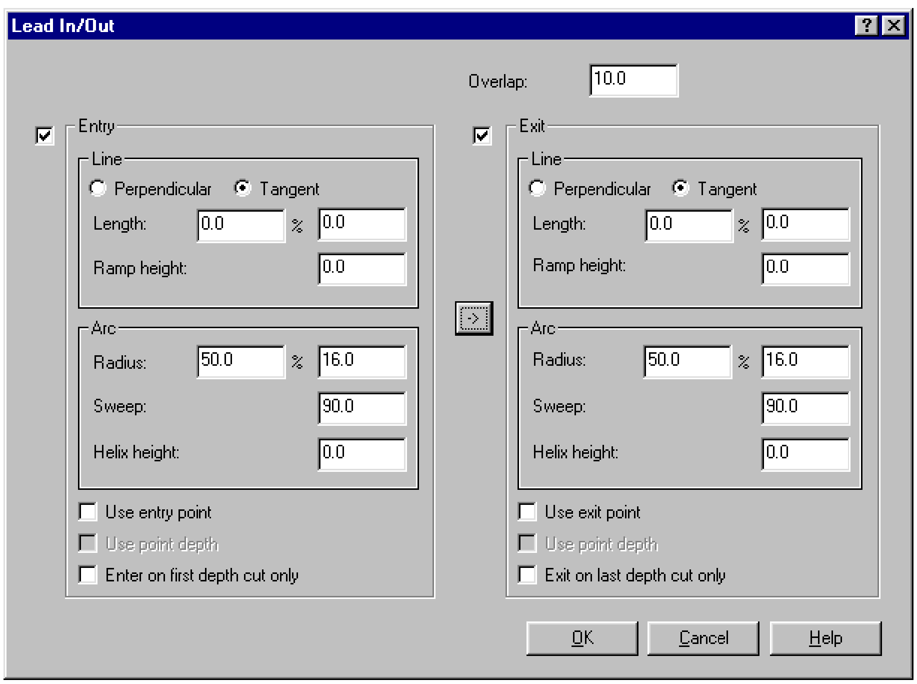


10. Press [Alt + T] to clear the toolpath display.

▶ *Editing the parameters for the finish operation*

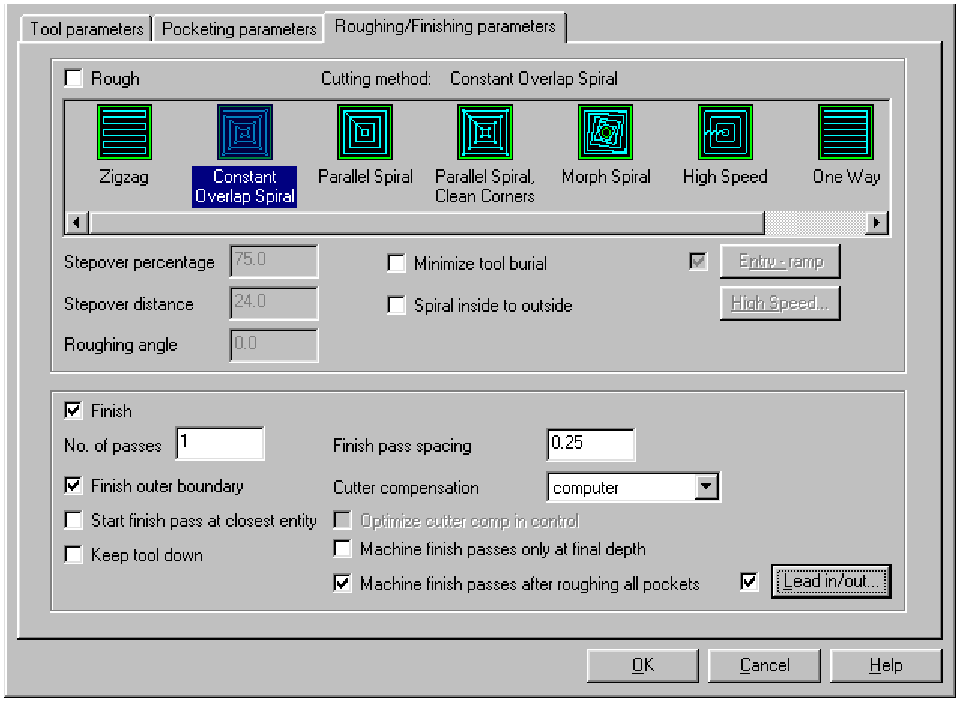
Since you told Mastercam to create a separate operation for the finish pass, you can edit its parameters. In this procedure, you will define the lead in/out moves.

1. Press [Alt + O] to open the Operations Manager. You can see two pocket operations.
2. Choose the **Parameters** icon for the second operation.
3. Choose the **Lead in/out** check box and button.
4. Enter **10** for **Overlap**.
5. Enter an **Entry–Line Length** of **0**.
6. Enter an **Entry–Arc Radius** of **50%**.
7. Copy the values to the **Exit** section. Your values should match the following picture.



8. Choose **OK**.

9. Verify that your Roughing/Finishing parameters match the following picture.



10. Choose the **Pocketing parameters** tab.

11. Enter **0** for the **Z stock to leave**.

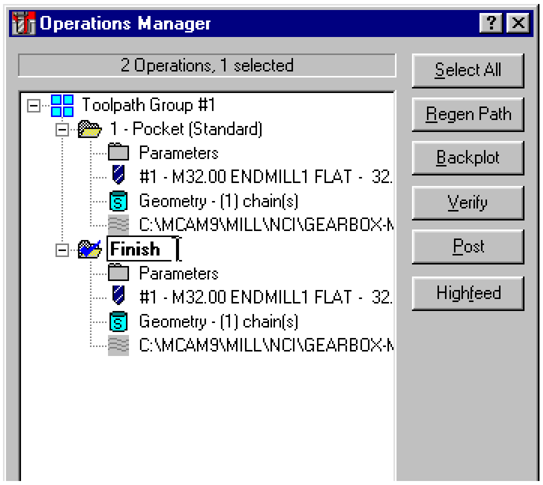
12. Choose **OK**.

13. Choose **Regen Path**. Mastercam regenerates the toolpath with the new lead in/out moves for the finish pass.

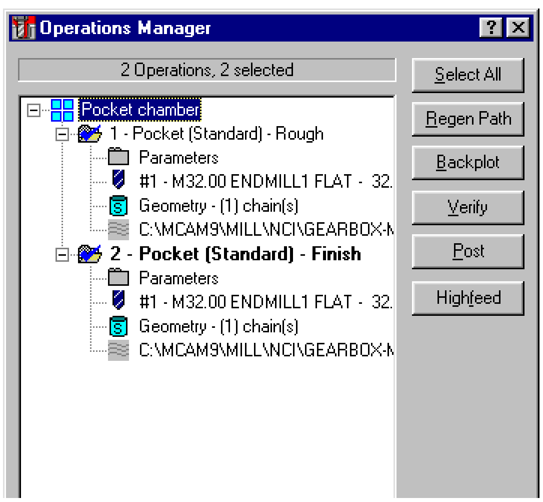
▶ *Renaming the operations*

Since you will be creating many operations to machine this part, you should start naming the operations and groups to keep them organized.

1. Click on Operation 2 and type in the name **Finish** as shown in the following picture.



2. Press [Enter] to set the name.
3. Click twice on operation 1, type the name **Rough**, and press [Enter].
4. Right-click on **Toolpath Group 1** and choose **Groups, Rename operation group**.
5. Enter the name **Pocket chamber**. Your operations list should look like the following picture.



6. Choose **OK** to close the Operations Manager.
7. Choose **Main Menu, File, Save** and save the file in your working folder as **gearbox1.mc9**.

Exercise 2 – Drilling the holes

Mastercam's Solid Drilling function incorporates feature-recognition technology to automatically find all the holes in your solids and create complete drilling operations for them. It is similar to the Auto Drilling function that you used in Chapter 8. First, you will drill and tap the 12 mm holes on the top edge of the part. Then, you will drill the remaining holes which are located on different faces of the part.

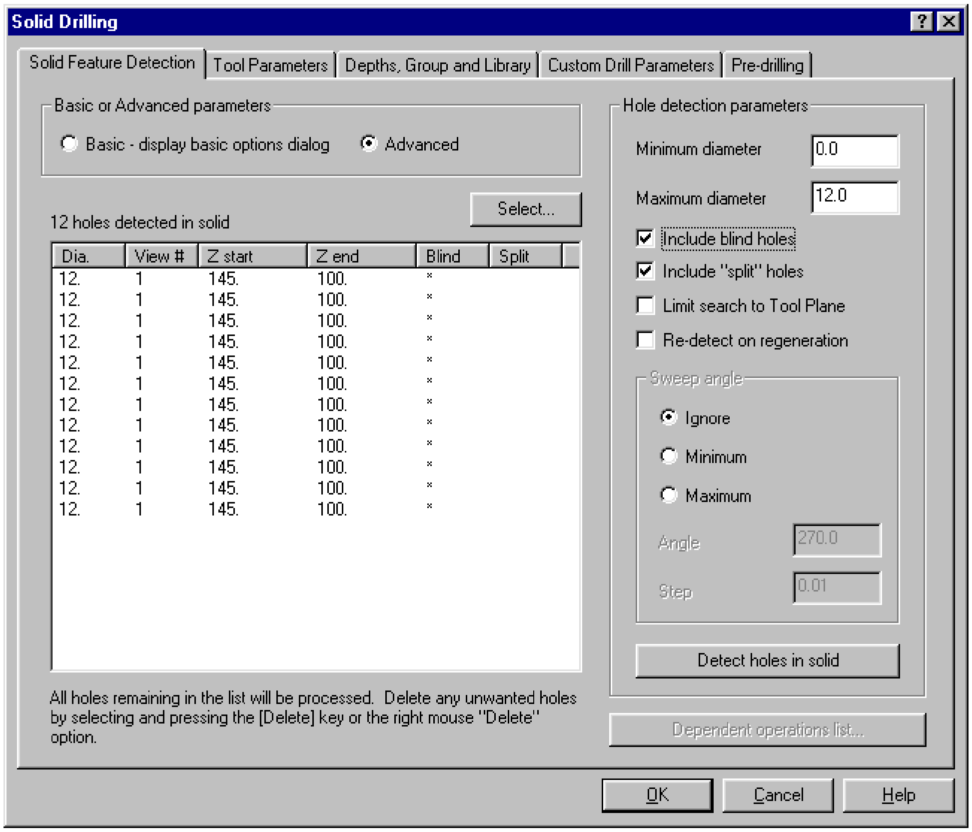
In this exercise, you will learn the following skills:

- **Using Solid Drilling to create drilling, tapping, chamfering, pre-drilling, and spot drilling operations**
- **Drilling holes in different views**

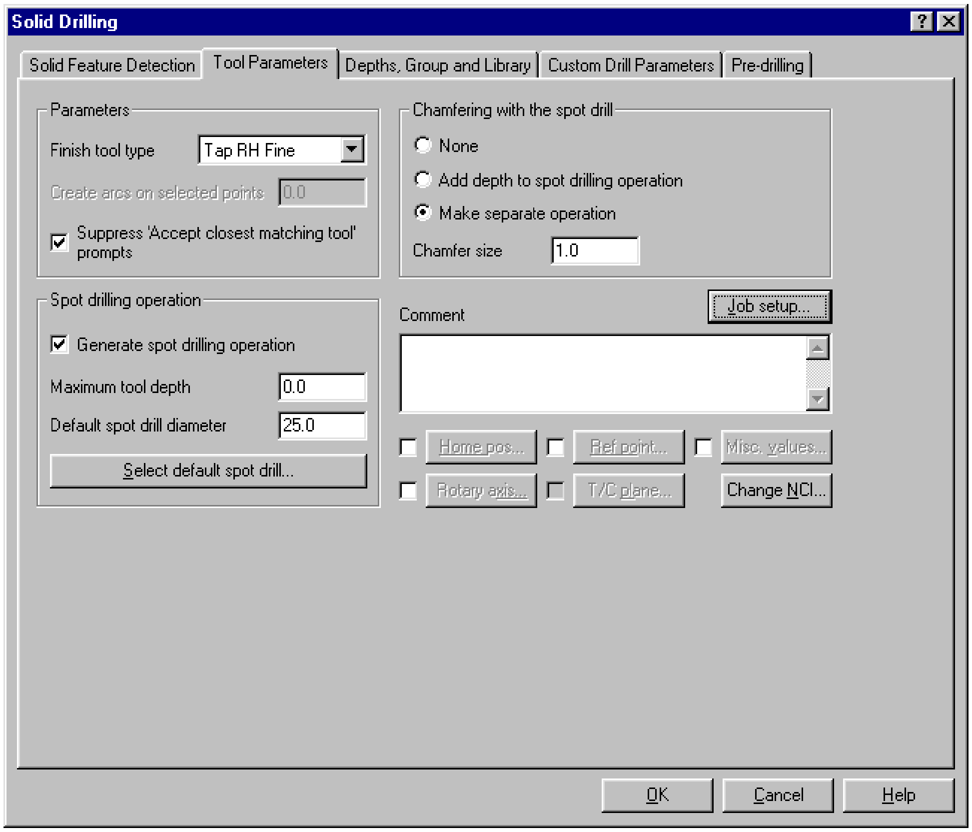
▶ *Drilling and tapping the top holes*

The first set of drilling operations will be on the twelve 12 mm holes on the top edge of the part. These need to be drilled and tapped. You will also create separate operations for spot drilling and chamfering.

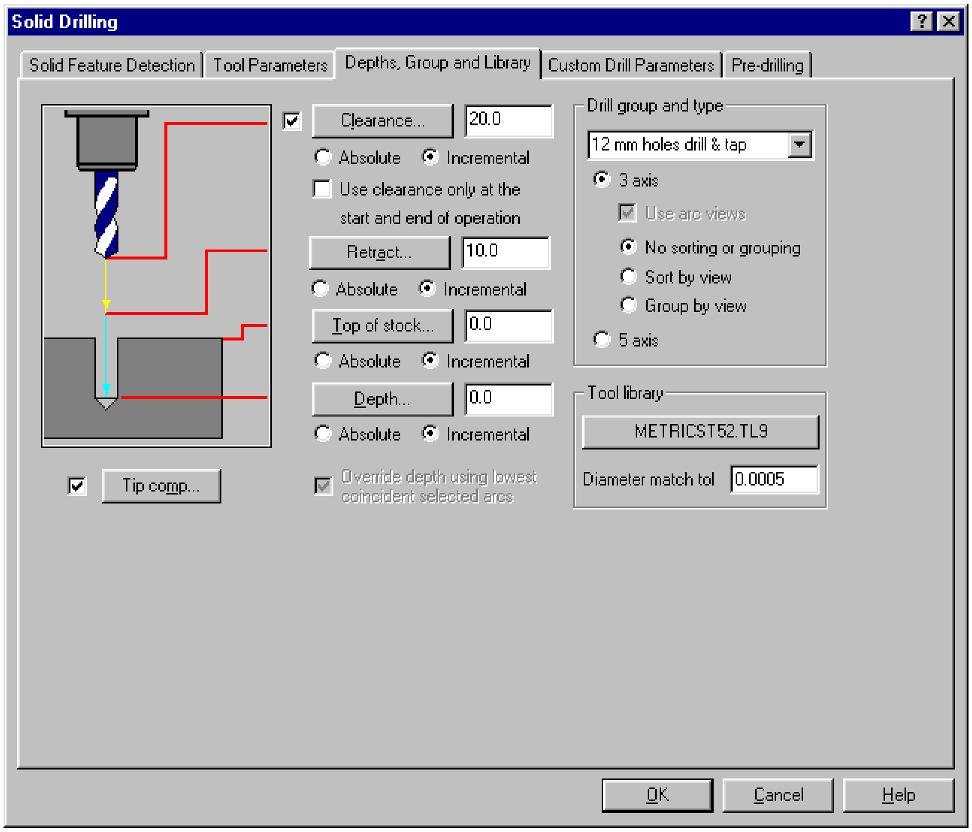
1. Choose **Main Menu, Toolpaths, Next menu, Solid drill**.
2. Choose the **Advanced** radio button. Mastercam automatically finds all the arcs in the solid.
3. You can see that the 12 mm holes are the smallest ones in the list. To quickly screen out the other holes, enter **12** in the **Maximum diameter** field and press [Enter]. Your dialog box should look like the following picture.



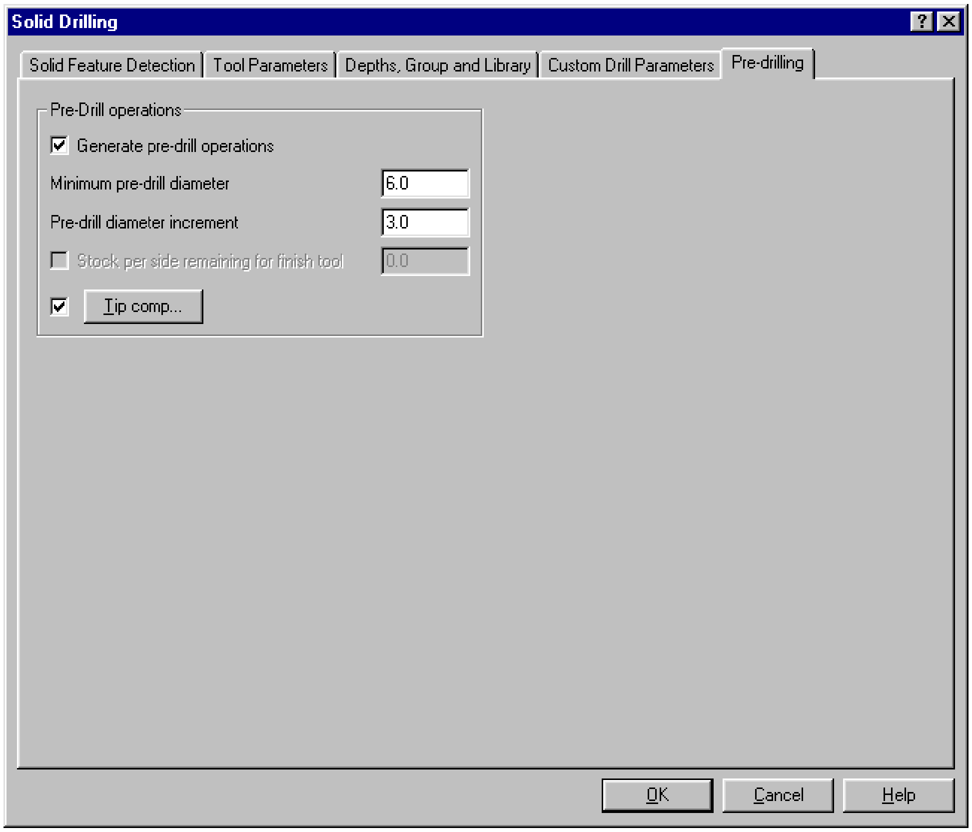
4. Choose the **Tool Parameters** tab.
5. Select a **Finish tool type** of **Tap RH Fine**.
6. Select **Generate spot drilling operation**.
7. Enter a **Default spot drill diameter** of **25**.
8. Select **Make separate operation** in the **Chamfering with spot drill** section.
9. Enter a **Chamfer size** of **1**. Your other values should match the following picture.



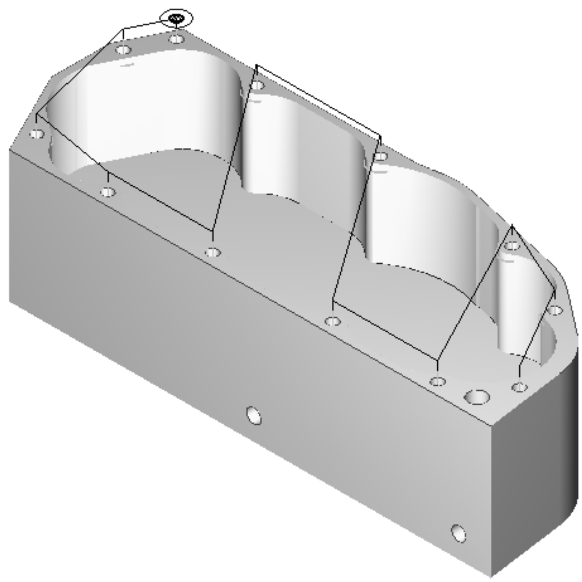
10. Choose the **Depths, Group and Library** tab.
11. Mastercam will create a separate toolpath group for the drilling operations for this group of holes. Enter the name **12 mm holes drill & tap** in the **Drill group and type** field.
12. Choose **No sorting or grouping**.
13. Select the **Tip comp** check box to turn on tip compensation for the drills. Your values should match the following picture.



14. Choose the **Pre-drilling** tab.
15. Choose **Generate pre-drill operations**.
16. Enter **6** for **Minimum pre-drill diameter**.
17. Enter **3** for **Pre-drill diameter increment**. Your values should match the following dialog box.



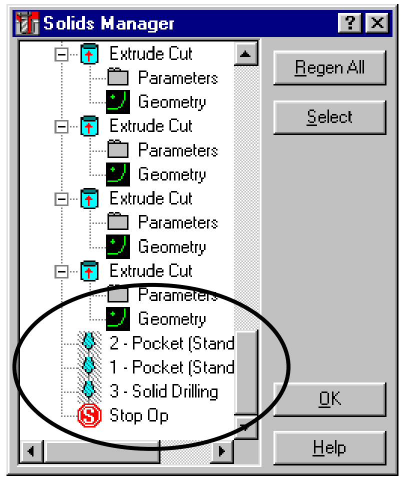
18. Choose **OK**. Mastercam generates the drill toolpaths as shown in the following picture.



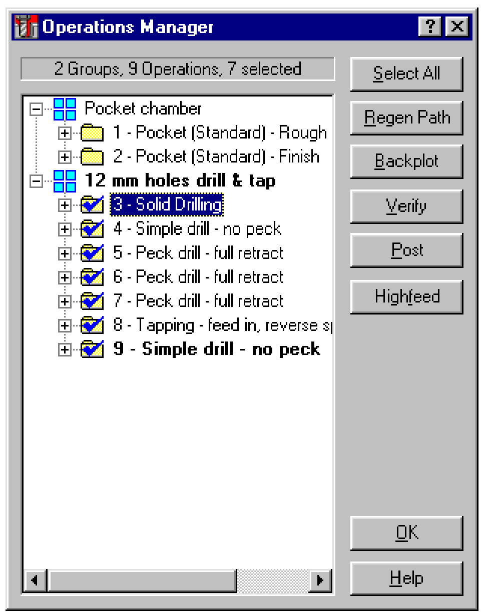
▶ *Using the Solids Manager to look at operations*

You can use the Solids Manager to help organize your solid machining operations. Machining operations involving solids are fully associative just like wireframe operations, so changes that you make in the Solids Manager are automatically recognized by the machining operations.

1. Choose **Main Menu, Solids, Solids mgr**.
2. Scroll down to the bottom of the solid operations list. You will see the toolpaths that you have created listed there.



3. Right-click in the window and choose **Operations Manager** from the menu.
4. Press [E] to collapse the list of operations so you can see them more clearly. It should look like the following picture.



Tip: Press [E] several more times to collapse and expand the operations list one level at a time.

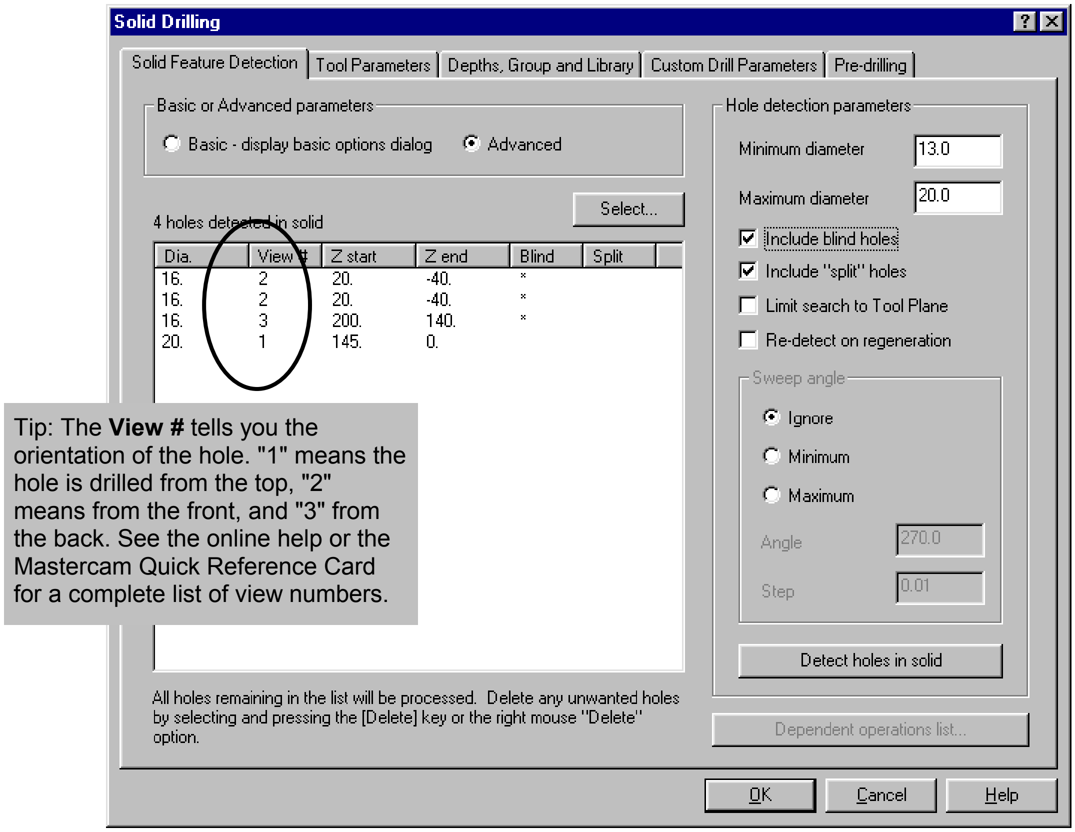
▶ *Drilling the remaining holes*

You will use the Solid Drilling function again to create drilling operations for the remaining holes. Solid Drilling automatically recognizes that the remaining holes are different sizes and are located on different faces of the part, and so need to be drilled from different angles.

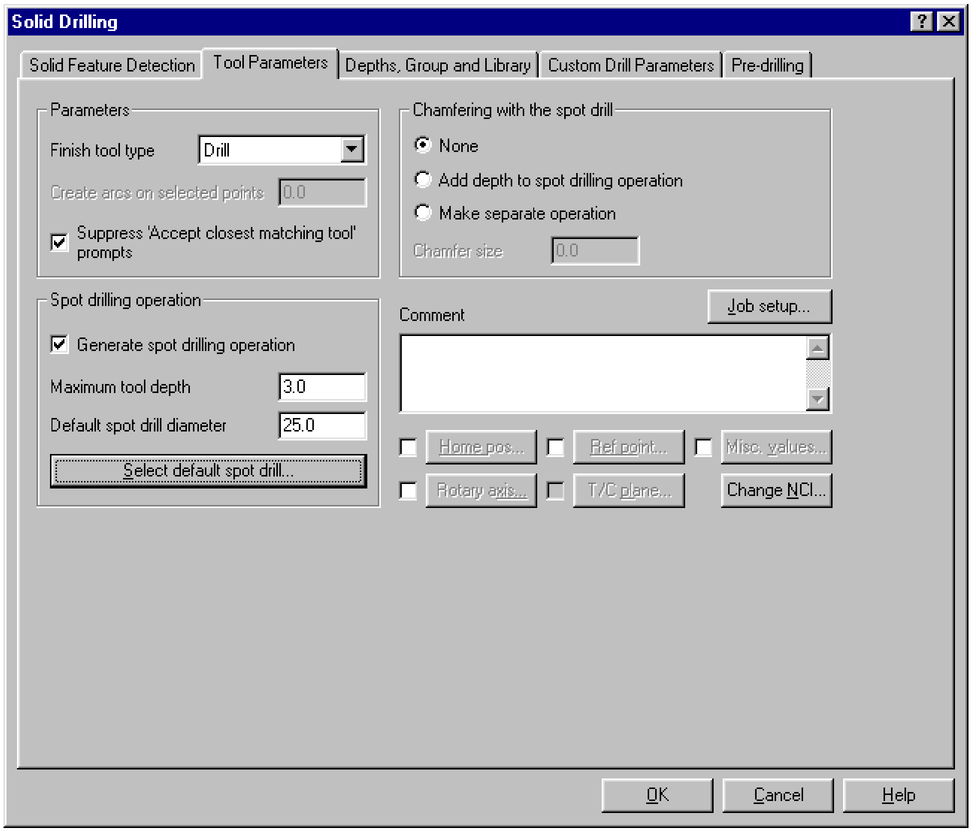
1. Right-click in the Operations Manager and choose **Toolpaths, Solids, Drilling**.
2. Choose **Advanced**. Mastercam reads all the arcs in the solid.

   Mastercam lists a number of arcs that you do not want to include in this operation, so you will filter them out by their diameter.
3. Type in a **Minimum diameter** of **13**. The holes smaller than this were drilled and tapped in the previous procedure, so you don't want to include them here.
4. Type in a **Maximum diameter** of **20** and press [Enter]. The arcs larger than this are in the contour of the pocket, not drilling holes, so you don't want to include them either.

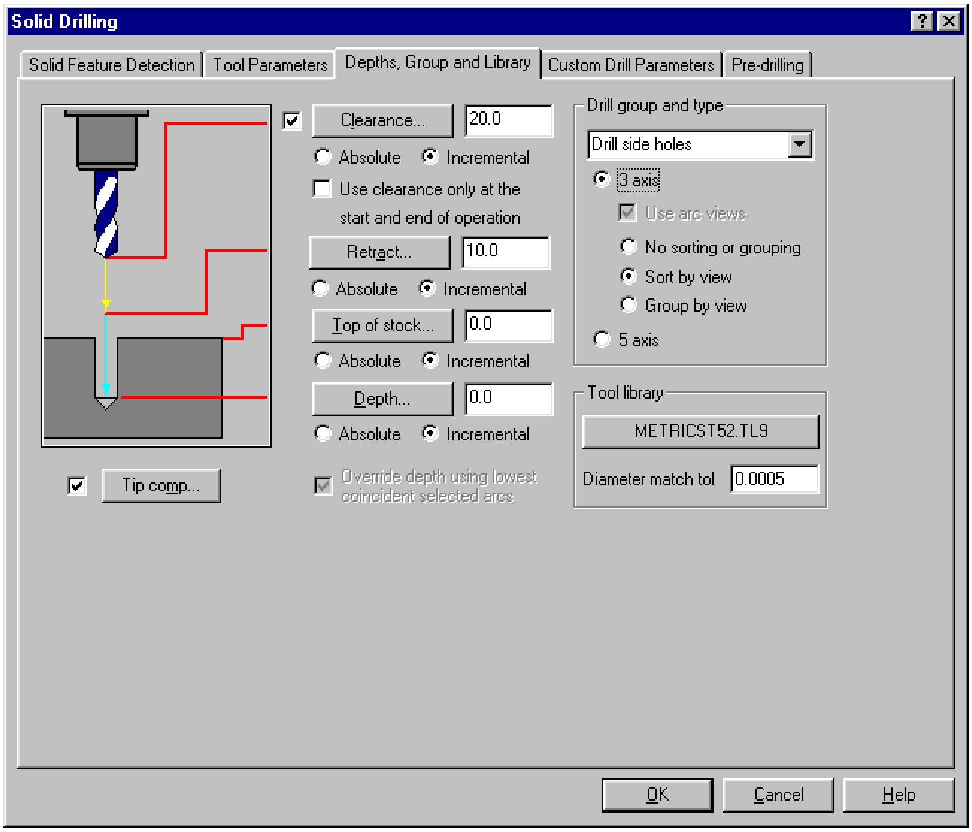
   Your selections should match the following picture.



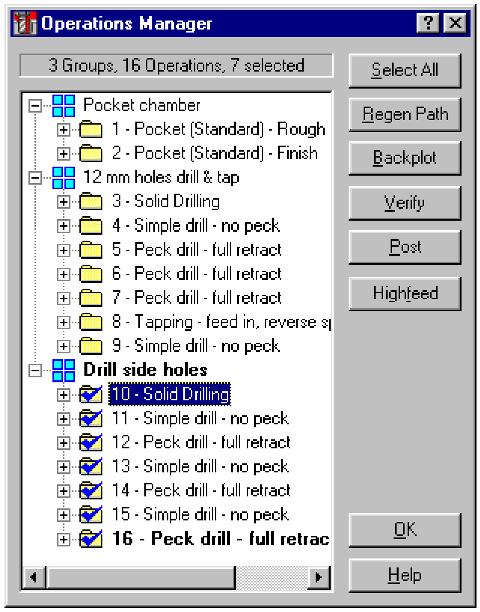
5. Choose the **Tool Parameters** tab.
6. Select **Drill** for **Finish tool type**.
7. Select **Generate spot drill operation**. Your other values should match the following dialog box.



8. Choose the **Depths, Group and Library** tab.
9. Enter the name **Drill side holes** in the **Drill group and type** field.
10. Choose **Sort by view**.
11. Select the **Tip comp** check box. Your values should match the following picture.



12. Choose **OK**.
13. Choose **No** if you are prompted to sort the other operations. Mastercam generates the drill toolpaths and shows you the new operations as shown in the following picture.



14. Choose **OK** to close the Operations Manager.
15. Press [Alt + A] to save the file.

Congratulations! You have completed the Version 9 Mill/Design Tutorial. The final two chapters provide you with a glossary of terms and a list of the Mastercam shortcut keys. To learn more about Mastercam:

- See the online help to read more information for features you learned about while completing this tutorial.
- Contact your local Mastercam reseller.
- Visit CNC Software on the Web to keep up with the latest Mastercam developments and learn about other Mastercam products. Visit **http://www.mastercam.com**, or visit the Mastercam forum at **http://www.emastercam.com**.
- If you have any comments about this tutorial, please send them to **techdocs@mastercam.com**.

18 Glossary

2 ½D (contour)	A toolpath consisting of multiple sections in which the depth can vary between sections but is constant within a section.
2D (contour or plane)	A toolpath or geometry that lies in a single plane.
3D (contour, plane or space)	A toolpath or geometry defined in X, Y, and Z axes simultaneously; consists of lines, arc, parametric splines, and NURBS splines.
4-axis	Toolpaths defined by X, Y, and Z locations, but with a tool axis with an additional degree of freedom, permitting the tool to be oriented parallel to an axis other than X, Y, or Z.
5-axis	Toolpaths defined by X, Y, and Z locations, but with a tool axis with two additional degrees of freedom, permitting the tool to be oriented parallel to an axis other than X, Y, or Z.

A

A axis	Axis of circular motion about the X axis; expressed in degrees.
absolute (coordinates, dimensioning, positioning)	Measured from a fixed reference point, usually 0,0,0.
across chain	When creating swept toolpaths, this is the profile to be swept along the curve.
AI	Extension for Adobe Illustrator files. This is a common vector graphics format.

along chain	When creating swept toolpaths, this is the curve along which the profile is swept.
along entity	A series of evenly spaced points along a line, arc, or spline.
arc	An open or closed planar curve in which all positions are at a fixed distance (radius) from the center of the curve. A circle is a 360-degree arc.
associativity (toolpath, dimensioning, and solids)	A relationship that links geometry with toolpath, tool, material, and parameter information to create a complete toolpath operation. Permits modifications to geometry or machining parameters to easily regenerate accurate, updated solid topology, dimensions and toolpaths. Also the relationship between dimensioning and its geometry.
attribute data	Attributes of entities: level, color, style, width.
AutoCursor	A feature that snaps the cursor to endpoints, midpoints, intersections, center points, quadrants or an arc, and the origin points in the vicinity of the cursor; automates and speeds point detection.
AutoHighlight	A feature that speeds and simplifies entity selection by dynamically highlighting the entity under the cursor before then entity is actually selected.
AutoSave	Feature that automatically saves current geometry and operations at a regular time interval.

B

B axis	Axis of circular motion about the Y axis; expressed in degrees.
backplot	Preview a toolpath, either step-by-step or continuously.
batch processing	Method for generating nesting results for several nesting sessions at the same time.
bitmap	A graphic composed of small dots, or pixels. Bitmap files use the BMP or PCX extension, and are often produced by scanning. See also **raster, vector**.

blank	To reduce the complexity of the graphics window by temporarily making one or more entities invisible. They remain blanked until the user unblanks them. The blanked entities remain in the database and are saved with the file. See also **hide**.
blend	Smooth connection of surfaces.
bolt circle	Circular array of evenly spaced points defined by the center, radius, and a number of points on the circle.
boss	In general, a plateau of material from a surrounding cavity.
boundary	An edge, border, or limit; a curve or chain that indicates an edge.
bounding box	A feature used to approximate the limits of the stock required to machine a part.
branch (point)	Point in a chain where the endpoints of three or more entities meet.
browse	In Mastercam, to preview actual images of Mastercam files in a selected directory. Also to look at the file names in a directory.

C

C axis	Axis of circular motion about the Z axis; expressed in degrees.
CAD	Acronym for computer-aided design.
CAD/CAM	Acronym for a combined CAD and CAM system.
CAM	Acronym for computer-aided manufacturing.
canned text	Post processor variables that can be associated with special commands, for example, an auto stop to check on a part during machining.
Cartesian	Coordinate system using X, Y, and Z values to locate a point in space.
CFG	Extension for Mastercam configuration files.

chain	Selection of one or more curves (lines, arcs, and/or splines) that have adjoining endpoints and often form boundaries; may be open or closed. Point entities can be chained using the point method for tool rapid moves; curves and points can be chained.
chain direction	The order of curve selection in a chain from start point to endpoint in an open chain; in a closed chain, may be clockwise or counterclockwise.
chain synchronization (Sync mode)	To break a chain into separate sections, each beginning and ending at a specified point, then match it with one or more other chains with the same number of synchronization points.
chaining tolerance	Maximum distance between two endpoints that can still be chained.
chamfer	Beveled or sloping edge that consists of one line that trims two intersecting lines. Each endpoint of the chamfer is positioned at a defined distance from the intersection of the two selected lines. In contour toolpaths, a chamfer is used to break sharp edges.
check surface	A surface or solid face that the system protects during toolpath generation on another surface.
C-Hook	Custom-made utility programs or add-ons (written in the C or C++ language) that run within Mastercam. C-Hooks that are automatically installed with Mastercam appear on menus with an asterisk (*) after the name. Other C-Hooks can be accessed by pressing [Alt + C].
chord height (tolerance)	In general, the amount of play allowed between a surface edge and the original geometry; determines the degree of precision with which edges of trimmed surfaces are created. See also **edge tolerance**. Also the tolerance with which Mastercam calculates surface shading independent of current display scale.
circle	A closed planar curve in which all positions are at a fixed distance (radius) from the center of the curve.

circle mill	A function that generates a toolpath to automatically machine full circles.
clearance plane or height	Height at which the tool moves between two separate machining operations.
climb milling	Cutting in which the tool rotates in a direction opposite the direction of travel along the side being cut. Generally produces a smoother surface finish than conventional milling. When the spindle is rotating clockwise, climb milling may be achieved by setting cutter compensation to the left. See also **conventional milling**.
closed chain	A chain whose start and end points are identical.
cluster	A set of parts that need to remain in a certain orientation to each other, such as a set of letters or numbers. The parts form a unit in which the position of the parts, relative to each other, stays the same.
CNC	Acronym for computer numerical control, which is a computer used to control machine tools.
CNCEDIT	File editor supplied with Mastercam that also provides some CNC and DNC capabilities.
collinear	Having the property of lying on the same line.
combine view	Combines all parallel views into a single view and moves arcs from separate parallel views to a single view.
communications (serial)	Transmission of information, one bit at a time over a single line, between a PC and any devices attached to it. See also **communications parameters**.
communications parameters	Parameters that control the transfer of information between a PC and devices attached to it. Parameters include format, port, baud rate, parity, data bits, stop bits, echo terminal emulation, strip carriage returns, strip line feeds, EOL (end of line) delay, and DOS communications mode. Communications is a File menu option (Communic).
compensation	See **cutter compensation.**

compensation direction	The orientation of the part with respect to the tool when cutter compensation is used. For example, the **left** direction means that if you are facing forward in the direction that the tool is moving, the tool will be to the left of the part.
compensation in computer	A setting which means that Mastercam computes the compensated positions and inserts them in the NC program. See also **cutter compensation**.
compensation in control	A setting which means that Mastercam does not calculate compensated positions, but instead inserts codes in the NC program (for example, G40/G41/G42) which signal the control to calculate them. See also **cutter compensation**.
composite curve	A chain of curves that meet endpoint to endpoint.
construction origin	Reference point (X0, Y0, Z0) for geometry creation; the same as the system origin unless reassigned by the user.
construction plane (Cplane)	Plane where geometry is created; may be different from the graphics view (Gview). Mastercam provides several standard construction planes: 3D, top, front, back, bottom, left and right side, isometric, and axonometric. Additional planes can be created.
context-sensitive help	Helpful information displayed on the screen that is relevant to the operation being performed.
contour	Path described by two or more axes. Also a method of analyzing selected boundaries or the boundary offset, thus simulating toolpath creation.
control points	Points that define a NURBS spline; usually do not lie on the spline.
conventional milling	Cutting in which the tool rotates in the same direction as the direction of travel along the side being cut. Selecting clockwise spindle rotation and cutter compensation to the right results in conventional milling. See also **climb milling**.

converter	A function that imports or exports files in different formats. Formats that can be translated include ASCII, CADL, DWG, DXF, IGES, NFL, Parasolid, SAT, STEP, STL, VDA, GEO, old GE3, as well as different versions of Mastercam.
Coons patch	A surface constructed by blending a grid of along curves and across curves. Named after Steven A. Coons. See also **Coons surface**.
Coons surface	A surface composed of one or more Coons patches.
coordinate	The combination of an axis and a number which represents a position along the axis, for example, X1. A pair of coordinates can represent a position in a plane, for example X1,Y2. Three coordinates can represent a position in 3-dimensional space, for example, X1,Y2,Z5.
copious data	An entity type that represents a collection of geometric forms (points and lines). Copious data originates in an IGES file. Mastercam can convert it to points and lines during translation. The Modify, Break, Cdata/line function can also be used to convert copious data to points and lines.
Cplane	See **construction plane**.
critical depths	Toolpath cut depths that must be machined even if depth increments must be adjusted to cut them.
cross-section	A section made by a plane cutting traversely through solids or surfaces. Also used in project toolpaths.
curvature (surface)	Measure of curving of a curve or surface.
curve	Line, arc, spline, or surface curve.
cut (toolpaths and solids)	When used with respect to toolpaths, refers to tool movement in the Z axis; do not confuse with pass. When used with respect to solids, a type of solid operation in which chains of curves are extruded, revolved, swept, or lofted as material is removed from an existing solid (target body).

cutter compensation	The process of offsetting a toolpath from the part geometry by an amount typically equal to the radius of the tool. See also **compensation in computer** and **compensation in control**.
cutter offset	Distance from the part surface to the axial tool center; tool radius.
cutter path	The path the center or tip of the tool follows over the part; the toolpath.

D

data bits	A communications parameter that defines the number of bits used to represent a character; must be the same for both the PC and the CNC controller or peripheral device.
depth cuts	Z-axis cuts that the tool makes in a contour, pocket, face, circle mill, or surface toolpath to get to the final depth in set increments.
DF9	Mastercam default parameter file format for Version 9 (*.DF9); contains default values for all toolpath types.
dirty operation	A solid or toolpath operation that has been modified in some way; for example, its parameters or geometry input. The system marks dirty solids and operations with a red 'X' in the Solids Manager dialog box or Operations Manager. When an operation is dirty, it must be regenerated for the toolpath or geometry to match the parameters.
display cues	Features that clarify how geometry is oriented in the graphics window: XYZ axes marker, dynamic arrow, surface backside display.
display list	An internal feature that saves the display data for each entity; used by Mastercam to determine what entities are visible on the screen and to speed redraws, view changes, and other screen functions.

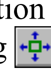
DNC	Acronym for direct numerical control or distributive numerical control. Direct numerical control uses a single computer to simultaneously control operation of a group of NC machines. Distributive numerical control uses a network of computers to coordinate operation of a group of CNC machine tools. Mastercam can be used in either situation.
dongle	Another name for a SIM, which is required to run Mastercam. See also **HASP**.
DOS (shell)	Acronym for Disk Operating System. A DOS shell can be used execute MS-DOS commands while Mastercam is running.
double D	A shape composed of two line entities and two arc entities.
dpi	Dots per inch, a measure of graphic resolution.
drafting entity	An entity used in dimensioning: witness lines, leader lines, dimensions, cross hatches, labels, notes, copious data. See also **geometric entity**.
drive surface	A surface and/or solid body that undergoes a surface or multiaxis machining operation. See also **check surface**.
dynamic arrow	Cursor display that permits dynamic movement along geometry to indicate a position; changes size to indicate orientation of arrow relative to viewer. When large, the arrow points toward viewer. When small, the arrow points away.

E

edge	A topological element of a solid model, which has an underlying curve.
edge profile	Defines the shape of the surface outer boundaries.
edge tolerance	The degree of precision with which edges of trimmed surfaces are created.
editor	An application used to modify files of certain types. See also **MCEDIT, PFEDIT32, CNCEDIT**.

ellipse	An oval-shaped curve, represented by a NURBS spline or collection of connected lines.
entity	A design building block. There are geometric entities (points, lines, arcs, splines, surface curves, surfaces, solids, copious data) and drafting entities (witness, lines, leader lines, dimensions, crosshatches, labels, notes).
entity association	The dependent relationship between one entity and a second entity or group of entities from which the first entity is generated.
EPS	Acronym for Encapsulated PostScript. This is a common vector graphics format.

F

feed plane	Height that the tool moves to before changing from the rapid rate to the plunge rate to enter the part.
feed rate	Cutting tool speed of movement in the cutting direction; usually expressed in inches per minute.
file information	Displayed when an operator presses [F9]: file name and path, date and time of last file save, file size in bytes, current display scale, relative positions of construction, tool, and system origins and axes.
fillet	An arc tangent to two non-tangent curves; a rounded interior or exterior corner.
filter (Filter)	The process of eliminating unnecessary tool movements from a toolpath. Do not confuse with mask. When capitalized, an utility that performs this function.
finish	Precision surface machining.
fit screen	To display the visible geometry so as to fill as much of the graphics window as possible; a Mastercam function that is available from the right-click menu, by choosing  from the toolbar, or by pressing [Alt + F1].
flat boundary	Used to create a flat, trimmed surface from one or more closed sets of curves.

flowline	Multiple curves along an entire surface in one constant parameter direction, that is, one of the directions in which the system creates the surface.
font	Text style. Mastercam fonts include Stick, Roman, European, Swiss, Hartford, Old English, Palatino, and Dayville. Windows® TrueType® fonts are also supported.
FPT	Feed per tooth.
free-form surface	A surface generated from arbitrarily shaped lines and curves; includes ruled, lofted, 2D swept, 3D swept, and Coons surfaces.
function	A single operation, for example Analyze, Set Norms.
function keys	Keyboard keys numbered [F1] through [F10]; may be assigned to functions, C-Hooks, and macros.

G

Gcode	In general, an NC part program or the language used to create it; specifically, a code that, among other things, defines part program coordinates.
GE3	Mastercam file format for geometry files prior to version 7 (*.GE3); does not contain toolpath information.
geometric entity, geometry	Geometric entities include points, lines, arcs, splines, surface curves, surfaces, and solids. A part model consists of these entities. See also **drafting entity**.
geometric surface	Surface composed of constant geometric shapes: sphere, cones, cylinders, draft surfaces, and surfaces of revolution.
global parameters	Dimension attributes that are applied to all drafting entities; includes dimension symbols, coordinate formats, tolerances, text properties, witness and leader line attributes.
gouge	The result or act of a tool machining away material that should not have been removed.
grain direction	The noticeable surface direction on materials such as wood. In Nesting, you can set the grain direction for parts and sheets to be either horizontal or vertical.

graphics view (Gview)	The point of view of the displayed geometry; may be top, front, side, isometric, as well as defined dynamically by the operator.
graphics window	Workspace area in Mastercam where the geometry displays.
group	A collection of entities or operations that can be manipulated as a single entity. See also **result**.
GUI	Acronym for graphic user interface.

H

hardcopy	Paper copy of the geometry visible in the graphics window.
HASP	Acronym for Hardware Against Software Piracy; refers to the type of SIM used by Mastercam 7.0 or later.
helix	A curve that is circular in the XY dimension and linear in the Z dimension. Mastercam lets you create helical entry and exit moves for many types of toolpaths.
hide	To make all entities except those selected temporarily invisible so as to simplify the graphics window. They remain invisible until unhidden as a group. Hidden entities are not saved with the file. See also **blank**.
highlight	To select with the cursor, with the result that the selected object changes color or reverses to white type on a dark background. See also **AutoHighlight**.
home position	Position where the tool returns for tool changes and at the end of the NC program.
HSS	High speed steel.

I

icon	Small symbol used to simplify access to a program or function; sometimes also called a button.
IGES	Acronym for Initial Graphics Exchange Standard, an international neutral format; used to transfer geometry from one brand of CAD system to another.

incremental (coordinates, dimensioning, positioning)	Measured from the immediately preceding point.
infinite look ahead	In contour analysis, to search the entire boundary to find self-intersections based on the current offset distance and cutter compensation.
integer	A whole number such as 3, 50, or 764; used as a data type for counting or numbering.

J

job	Contains a set of operations.
Job Setup	Machining job parameters, including stock setup, NCI configuration, material selection, and tool offsets.
jump height command	Allows a tool to be moved to a height above the clearance plane between points in a toolpath.

L

level	A grouping used to organize geometry in Mastercam.
level report	A report of what entities exist on each level of a geometry file.
line	Straight entity between two endpoints.
line style	The appearance of a line; may be solid, hidden, center, phantom, or Zbreak.
linear array	A repeating toolpath along the X or Y axis of the construction plane at a specific distance.
linearization tolerance	Used when converting 3D arcs and 2D or 3D splines in the chained geometry from curves to lines; represents the maximum distance between an arc or spline and its linear approximation.
loft surface	A surface composed of smoothly blended curves created by fitting through a set of cross-sectional curves.

M

macro	Group of commands and instructions that can be stored, recalled, and executed to perform a task; may be used to automate common or repetitive tasks.
Main Menu	Presents primary Mastercam functions: Analyze, Create, File, Modify, Xform, Delete, Screen, Exit, and in Mill and Lathe, Toolpaths, and NC Utilities.
mask	Restricts entity selection to certain types or levels. Do not confuse with filter.
Mastercam®	An integrated CAD/CAM software package created by CNC Software, Inc.
material library	Contains information on materials for machining that is used to set a base percentage for feed rates and spindle speeds; uses the MT9 file extension.
MC7	Format for a Mastercam file in Version 7 (*.MC7).
MC8	Format for a Mastercam file in Version 8 (*.MC8).
MC9	Format for a Mastercam file in Version 9 (*.MC9); contains geometry, toolpath parameters, material definition, NCI data, and tool information. See also **job** and **operation**.
MCEDIT	A Mastercam text editor; provides NC capabilities, file editing, and file manipulation capabilities. See also **PFE32** and **CNCEDIT**.
merge	To combine another Mastercam file with the current geometry file. Some or all of one or more configuration files can also be combined.
MT9	Mastercam material library file format for Version 9.
MTL	Mastercam tool library file format for versions prior to Version 7 (*.MTL).
multiaxis	Using more than one axis; often refers to 4- or 5-axis toolpaths.

N

NBT	File extension for a nesting batch file.
NC	Acronym for numerical control, a technique for controlling machine tools or processes by coded command instructions; also the file format output from Mastercam post processors.
NCI	Acronym for numerical control intermediate, the Mastercam intermediate toolpath file format. The post processor reads the NCI file to produce the NC file.
nesting	The process of fitting multiple copies of a part within a defined boundary (sheet).
nesting session	A file containing all of the sheet, part, group, and parameter settings from the Nesting dialog box. A nesting session is saved as an NST file.
node (spline)	Points in a parametric spline.
nonlinear	Not located on a single line.
normal (arrow)	Perpendicular to. There are two normal vectors for each planar chain of curves, which point in opposite directions. A normal arrow indicates the side of the selected surface on which the system creates the surface.
NPL	File extension for a nesting part library.
NSL	File extension for a nesting sheet or scrap library file.
NST	File extension for a nesting session file.
NURBS (spline)	Acronym for non-uniform rational b-spline; a two- or three-dimensional curve defined by knots and control points.
NURBS surface	A surface that is defined analogously to NURBS splines with the string of control points expanded in another direction resulting in a grid.

O

obround	A shape composed of two straight line entities and two 180-degree arc entities.

offset	To displace an entity or chain by a distance in a perpendicular direction relative to the current construction plane. In a curve, displacement is perpendicular to the direction vector at every location on the curve.
offset surface	A surface created by offsetting an existing surface by a distance.
OP9	Mastercam operation library file format for Version 9.
open chain	A chain whose first and last endpoints are not identical, such as a line.
OpenGL®	An operating system-independent standard for displaying graphics.
operation (toolpaths and solids)	When used with respect to toolpaths, consists of geometry, toolpath (NCI file), tool definition, and parameters. A set of operations makes up a job or MC9 file. Each operation includes only one toolpath. See also **job** and **MC9**. When used with respect to solids, the action or actions performed to create or modify a solid. Each operation, such as fillet or extrude, is listed separately in the history tree under the solid that it defines or modifies.
operation library	Contains default parameters for a specific toolpath; can be applied to current geometry; uses the OP9 file extension.
Operations Manager	Lists all operations in the current Mastercam file, including both associative and non-associative toolpaths, and offers options for managing them.
origin	Intersection point of coordinate axes: typically, the point X0Y0, or X0Y0Z0. See also **system origin**, **construction origin**, and **tool origin**.

P

pan	To move geometry within the graphics window. You can press the arrow keys, or right-click in the graphics window and choose **Dynamic pan** from the menu.
parallel views	Construction planes that exist in the same 2D plane but differ by rotation or position.

parametric spline	A 2D or 3D curve defined by a set of coefficients or nodes.
parametric surface	A surface composed of parametric splines in which each curve segment is expanded in another direction resulting in a patch.
part	The item to be machined or nested.
part drawing	Describes the shape and size of a part; usually includes part features, dimensions, tolerances, and surface roughness.
part feature	The distinctive shape and size to be produced in a part; can be 2D (flat surfaces, internal and external profiles, pockets, holes, etc.) or 3D (surfaces).
part library	In Nesting, a file containing part definitions so that they can be reused. A part library is saved as an NPL file.
pass	A tool movement in the X and Y axes. Do not confuse with cut.
patch	Area of a surface bounded by four segments of the generating curves.
peck	A tool move that occurs at the programmed feed rate as it feeds into and retracts out of the stock during a drill toolpath.
peck clearance	Depth that the tool rapids down to between peck movements during a drill toolpath.
PFE32	A Mastercam text editor; provides file editing and manipulation capabilities.
planar	Flat, lying within a single geometric plane.
plot	To output current graphics window to a plotter or file.
point (entity)	Entity that marks a position in 2D or 3D space but that has no dimension.
point (using the mouse)	To move the mouse until the mouse pointer on the screen rests on the item you want.
point data	Data consisting only of points.

polar (coordinates and dimension)	Coordinate system that uses a known point, length (radius), and angle to locate a point in space. The angle is calculated in a counterclockwise direction from the positive horizontal axis that runs through the known point in the current construction plane.
polygon	Irregular, closed shape with three or more straight sides. In Mastercam, can be created as a single NURBS spline or as a collection of individual lines.
port	A physical connection on a PC. Serial ports are used to connect to the CNC controller and are identified as COM1, COM2, etc.
post	Post processor. Also a post processor (PST) file.
post processor	A program that translates NCI data to a format usable by a machine, that is, to an NC part program or Gcode.
primitive	A surface or solid created using a predefined shape, such as a block or sphere. The parameters can be changed interactively, but it maintains its original shape. A primitive surface or solid is not defined by curve geometry. Mastercam primitives include cylinder, cone, block, extrusion (surfaces only), sphere, and torus.
PRM	Mastercam default parameter file format and file extension for versions prior to Version 7.
prompt area	A two- or four-line area at the bottom of the Mastercam interface used to display data or enter values with the keyboard.
PST	File format for a post processor customization file.

Q

quadrant	A section of a plane in which quadrant 1 lies between 0 and 90 degrees, quadrant 2 lies between 90 and 180 degrees, quadrant 3 lies between 180 and 270 degrees, and quadrant 4 lies between 270 and 360 degrees.

R

RAM	Acronym for random-access memory.

RAM-saver	An option that compacts the system database and frees up available RAM; can also perform an efficiency and integrity check on the database.
real number	A number that can be represented by digits in a numbering system with a fixed base, such as 0.5 or 25.4; used for storing measurements and other values to some limit of precision.
raster	A method of generating graphics in which images are stored as many small, independently controlled dots (pixels).
rectangle	A shape which consists of four straight lines and four right angles.
Rectangular nesting	A nesting method which treats each sheet and part as if it were contained in a rectangular bounding box.
redraw	To erase then redisplay visible geometry in the graphics window to clean up display remnants.
reference point	Point to which the tool moves before reentering a toolpath.
regenerate	In general, to recompute solids, drafting entities, or toolpaths when associated geometry or parameters have been changed. To rebuild the graphics window display list so as to improve the speed and results. The Regen path option in the Operations Manager recomputes a toolpath when the associated geometry or parameters have been modified.
relative (coordinates, dimensioning, and positioning)	Distance measured from specific point, not necessarily the zero or preceding point.
remachine	A machining operation used to clean up stock leftover from one or more previous operations. Remachining operations calculate the amount of leftover stock and use that as the starting point for the toolpath motions. See also **restmill**.
repaint	To erase then redisplay the visible geometry in the graphics window to clean up display remnants.
required pilot diameter	Minimum diameter necessary for the tool to enter the toolpath.

rest material	The leftover stock that forms the basis for a restmill toolpath. Mastercam computes this by looking at one or more operations that you select, or from the dimensions of a roughing tool that you supply. See also **restmill**.
restmill	A type of remachining toolpath used for roughing, in which the tool uses multiple depth cuts to remove stock leftover from one or previous operations. See also **remachine, rest material**.
result	The appearance of an entity group that has been transformed; may be selected for further transformation or translation. The default color of a result is purple.
retract amount	Distance that the drill retracts every time it makes a peck move during a drill toolpath.
retract height	The height to which the tool moves before the next tool pass.
revolved surface	A surface created by rotating a sectional shape around an axis or line.
right-click	To click on something using the right mouse button; displays alternate (right-click) menus.
right-click menu	A menu that opens when you right-click the mouse; gives quick access to many common features.
roll	To wrap a line, arc, or spline around a cylinder.
rough	To remove large amounts of material as rapidly as possible.
RPM	Revolutions per minute; a measure of spindle speed.
rubber-band	Temporary display of entities that will be created or modified; the display updates dynamically based on the cursor location to indicate the result with the cursor at that location.
ruled surface	A surface composed of linearly blended curves created by connecting straight lines between two or more lines or curves.

S

save some	To save selected entities to an MC9 file. Toolpaths cannot be saved using this method.

scale	To increase or decrease the size of an entity by a factor relative to the construction origin or some other point. Also see scaleXYZ.
scaleXYZ	To increase or decrease the size of an entity independently in X, Y, and Z dimensions. Also see scale.
scrap	Material remaining from a sheet used in a previous nesting session.
scrap library	A file containing scrap definitions so that they can be reused. A scrap library is saved as an NSL file.
Screen, Configure	A menu that sets Mastercam's default values. Default configuration files are MILL9.CFG (English units) and MILL9M.CFG (metric units).
segment	A section of a spline between two nodes
selection cues	In Mastercam, functions such as AutoHighlight, AutoCursor, etc., which help you determine what entities you can select.
selection grid	A grid of reference points that the cursor can snap to during sketching.
setup sheet	A file created by Mastercam that contains NCI file information including operation, tool reference, total programming time, and text entered manually during programming; uses the SET extension.
SFM	Acronym for surface feet per minute.
shading	Representation of light striking a colored surface or solid object using gradated fill.
sheet	The closed boundary that defines the material on which nested parts are placed.
sheet library	A file containing sheet definitions so that they can be reused. A sheet library is saved as an NSL file.

shortcuts	In Mastercam, a way of gathering data from the graphics window. Allows you to modify data collected from the graphics window by entering values in the prompt area. Shortcuts appear in the prompt area as X, Y, Z, R(adius), D(iameter), L(ine length), S(distance between two points), and A(ngle).
SIM	Acronym for Software Interface Module; sometimes called a dongle; required to run Mastercam.
single D	A shape composed of one line entity and one arc entity.
sketch	To create geometry or select entities by identifying points in the graphics window using the cursor and mouse.
slice	The process of creating points at the intersection of lines, arcs, and splines with a plane and creating points where they intersect. Also the process of creating curves at the intersection of surfaces and solids with a plane and creating curves where they intersect.
solid	A geometric representation of a closed three-dimensional object. In Mastercam, a solid is a geometric entity that differs from other types of geometric entities such as lines, arcs, splines in that each solid is also a topological entity that occupies a region of space and that consists of one or more faces, which define the closed boundary of the solid.
spindle speed	Tool rotation speed (RPM)
spline	Smooth, free-form curve controlled by points including the condition of its endpoints; may be parametric or NURBS spline.
startup file	Configuration file, which contains Mastercam default values.
statistics (screen)	Tally of visible entities by type.
step angle	Controls the degree of rotation Nesting can use when fitting parts on a sheet. For example, a 90 degree step angle allows Nesting to use 90, 180, and 270 degree rotations. A smaller step angle increases the amount of time needed to generate the nesting results because Nesting makes more attempts at fitting the part more precisely.

stepdown	The distance that separates adjacent cuts in the Z axis on a surface toolpath.
stepover	The distance that separates adjacent cuts in the XY plane on a surface toolpath.
stretch	To place around geometry a window that intersects other geometry, then to translate the entities that are completely inside the window and also lengthen or shorten any lines that cross the window (by translating the endpoint that is inside the window).
style/width	Line style and width used to display lines, arcs, and splines.
subprogram	A section of the NCI file that repeats at different locations.
supplementary angle	An angle that when added to another angle produces an angle of 180 degrees.
surface	A representation of a part's skin by mathematical equations; a boundary defining an exterior face of a solid model.
surface curve	A curve entity type created directly on a surface through the Create Curve function.
surface memory allocation	The amount of RAM allocated for surface generation.
surface model	Defines a surface, including the edges of each surface.
surface normal	Vector perpendicular to tangent plane of surface.
surface projection	Creates points (or curves) by projecting selected points (or curves) onto selected surfaces.
surface shading	Color fill added to surfaces and solids to make them more easily visible; may be full-time or studio.
surface types	Mastercam supports three surface types based on mathematical generation methods: parametric, NURBS, and curve-generated. Surfaces may also be typed by components and application into loft, ruled, Coons, revolved, swept, draft, fillet, offset, trim/extend, and blend surfaces.

swept surface	Created by sweeping one or two curves or chains of curves (across curves) through a trajectory of one or two other curves or chains of curves (along curves); may be 2D or 3D. Also called a drag surface.
swept toolpath	Created by sweeping one chains of curves (the across chain) along a second chain (the along chain).
Sync	A function that breaks a chain into separate sections, each beginning and ending at a specified point, then matches it with one or more other chains with the same number of synchronization points.
system origin	Fixed reference point for all geometry creation (X0, Y0, Z0).
system tolerance	Maximum distance between two points that can still be considered coincident.

T

tangent	Two curves whose slope is continuous in direction across their common endpoint.
tip comp	Cutter compensation calculated to the tool center or tip.
TL9	Mastercam tool library file format for Version 9.
tolerance	The precision with which an entity must fit another entity or process, or the maximum permissible deviation from a value; includes system, chaining, minimum arc length, curve minimum step size, curve maximum step size, curve chordal deviation, and maximum surface deviation tolerances. Tolerance dimension format is one of the global drafting parameters.
tool	The cutting or machining part, usually removable, of a lathe, planer, drill, or similar machine.

tool body	The body or bodies that are added to, removed from, or used to keep a common region with a selected target body during a Boolean operation. Once a solid is designated a tool body, it becomes part of the target body. In the Solids Manager dialog box, a tool body is listed under the solid and Boolean operation that it helps to define, and its icon is marked with the letter 'T'. Note: When you delete a Boolean operation, the system restores the operation's tool bodies as distinct, active solids. You can also duplicate a tool body to obtain an active copy of the solid.
tool center boundary	A closed set of curves that limits tool movement for a surface toolpath. The tool's center stays within the selected boundary.
tool library	Contains information on multiple mill and lathe tools, such as spindle speeds, plunge rates, and tool diameters; uses the TL9 file extension.
Tools Manager	A Mastercam function that provides a list of tools stored in the current job or in the current tool library; also allows management of tool libraries.
tool origin	The reference point (X0, Y0, Z0) in the tool plane (Tplane); the same as the system origin unless reassigned by the user.
tool plane (Tplane)	A 2D plane that represents the CNC machine's XY axis and origin; also called Tplane.
toolbar	Area on the screen that contains icons (buttons). The buttons are arranged in pages to which the user can scroll; may be moved and reassigned.
toolpath	Shows where a tool removes material from a part.
Tplane	See **tool plane**.
transform	To translate, mirror, rotate, scale, offset stretch, or roll geometry or toolpaths.
translate	To move or copy geometry or toolpaths to a new location without changing orientation. Also see transform.
trim	To act as a boundary for a entity or surface.

trim/extend surface	A surface created by trimming or extending existing surfaces.
trimmed surface	Surface bounded at a set of edges; can be created by applying any or a number of processes to untrimmed or trimmed surfaces, for example, projection of curves, intersection, or filleting with other surfaces.
trochoidal	A type of loop used in highspeed toolpaths in which a continuous loop is overlayed on top of a toolpath, and the tool moves along the path of the loop.
TrueShape nesting	A nesting method that fits parts on a sheet based on each part's actual shape and other parameters. It also allows access to sheet and part libraries and extended functionality beyond Rectangular nesting. TrueShape nesting increases the yield of parts from a sheet of material.

U

undo	To reverse the last action performed.
unwrap	To unroll a rolled entity.
unzoom	To return to the previous display scale or to the original display size.
unzoom by 0.8	To return to the previous display scale or reduce the size of the displayed geometry to 80% of its original size.

V

vector (1)	A graphics file format in which graphics are stored as lines and curves, instead of dots or pixels. Common file extensions are EPS and AI. See also bitmap, raster.
vector (2)	A directed line segment.
vertex	An endpoint of an edge.
view	Angle of observing the geometry – top, front, back, bottom, right side, left side, Cplane, isometric, or axonometric.
viewport	Area within the graphics window that displays the geometry.

W

WCS	See **work coordinate system**.

window (selection)	A polygon sketched around entities to select them.
wireframe model	Three-dimensional object composed of separate lines joined to create a model; a complete set of edge and skin profiles that create a surface.
witness (dimension) lines	Thin solid lines that project from a dimensioned object to indicate the extent of the leader lines.
work coordinate system (WCS)	A coordinate system in which the orientation and origin are shifted. Provides a way to orient geometry in the best way to work on it.
work offset	A value that shifts the origin and coordinate system of the tool plane when creating toolpaths at different locations (for example, tombstone work).

X

X axis	Horizontal axis relative to the construction origin; right of origin is positive; left of origin is negative. See also **Cplane**.
Xform	Abbreviation for transform, a function that can translate, mirror, rotate, scale, offset, stretch, and roll geometry.
XYZ axes marker	Indicates the axis orientation according to 3D space; displayed in the bottom left corner of the graphics window; updates to reflect the current graphics view (Gview).

Y

Y axis	Vertical axis relative to the construction origin; above origin is positive; below origin is negative. See also **Cplane**.

Z

Z axis	Perpendicular to the X and Y axis relative to the construction origin. See also **Cplane**.
Z depth	Current construction depth, which is the depth of the currently defined construction plane (Cplane) relative to the system origin.
zoom	To magnify a rectangular portion of the graphics window.

19 Mastercam Shortcut Keys

Alt + 0	Set Z depth for Cplane
Alt + 1	Set main color
Alt + 4	Choose Tplane
Alt + 5	Choose Cplane
Alt + 6	Choose Gview
Alt + A	AutoSave
Alt + B	Toolbar on/off
Alt + C	Run C-Hooks
Alt + D	Drafting global parameters
Alt + E	Hide/unhide geometry
Alt + F	Menu font
Alt + G	Selection grid parameters
Alt + H	Online help
Alt + J	Job setup
Alt + L	Set line style and width
Alt + N	View Manager
Alt + O	Operations Manager
Alt + P	Prompt area on/off
Alt + Q	Undo last operation
Alt + R	Edit last operation
Alt + S	Full-time shading on/off
Alt + T	In Toolpath menu, turn toolpath display on/off
Alt + U	Undo last action
Alt + V	Mastercam version number and SIM serial number
Alt + W	Viewport configuration

Alt + X	Set main color, level, line style and width from selected entity
Alt + Z	Level Manager
Alt + ‘	Create two-point circle
Alt + Tab	Switch between applications
Alt + -	With hidden entities, select additional entities to hide
Alt + =	Select entities to unhide
Alt + F1	Fit geometry to screen
Alt + F2	Unzoom by 0.8
Alt + F3	Cursor tracking on/off
Alt + F4	Exit Mastercam
Alt + F5	Delete using window selection
Alt + F7	Blank geometry
Alt + F8	System configuration
Alt + F9	Display all axes
F1	Zoom
F2	Unzoom
F3	Repaint
F4	Display the Analyze menu
F5	Display the Delete menu
F6	Display the File menu
F7	Display the Modify menu
F8	Display the Create menu
F9	Coordinate axes on/off

F10	List all functions and choose one to execute
Tab / **Shift + Tab**	Navigate between controls in dialog boxes
Esc	System interrupt or menu backup
Page up	Zoom in by 5%
Page down	Zoom out by 5%
Arrow keys	Pan
Alt + Arrow keys	Rotate

In the Operations Manager:

Ctrl + A	Select all operations
Ctrl + C	Copy selected operations
Ctrl + V	Paste selected operations
Ctrl + X	Cut selected operations
E	Expand or collapse all operations
G	Set parent group of selected operation as the active group
L	Toggle NCI locking on selected operations
P	Toggle posting on selected operations
T	Toggle toolpath display for selected operations