



*Getting Started Series Tutorials*

# *Basic 3D Machining*



mastercam x getting started tutorials  
.....

# Basic 3D Machining

October 2009

**Be sure you have the latest information!**

Information might have been changed or added since this document was published. Contact your local Reseller for the latest information.



**Mastercam® X4 Basic 3D Machining**

Date: October 2009

Copyright © 2009 CNC Software, Inc.— All rights reserved.

First Printing: October 2009

Software: Mastercam X4

**TERMS OF USE**

Use of this document is subject to the Mastercam End User License Agreement. A copy of the Mastercam End User License Agreement is included with the Mastercam product package of which this document is part. The Mastercam End User License Agreement can also be found at:

[www.mastercam.com/legal/licenseagreement/](http://www.mastercam.com/legal/licenseagreement/)



# Contents

.....

<b>Introduction .....</b>	<b>1</b>
▶ Tutorial Goals.....	1
▶ Before You Begin.....	1
▶ If You Need More Help.....	3
▶ Additional Documentation.....	4
 <b>Machining the Connector .....</b>	 <b>5</b>
<b>1. Toolpath Setup.....</b>	<b>7</b>
▶ Lesson Goals.....	7
▶ Exercise 1: Assigning a Machine Definition .....	7
▶ Exercise 2: Setting Stock for Machining .....	8
▶ Exercise 3: Creating Toolpath Groups .....	10
<b>2. Roughing the Interior .....</b>	<b>13</b>
▶ Lesson Goals.....	13
▶ Exercise 1: Drilling the Holes .....	13
▶ Exercise 2: Roughing the Pocket .....	19
▶ Exercise 3: Roughing the Slot .....	23
▶ Exercise 4: Rest Roughing the Interior .....	28
<b>3. Finishing the Interior.....</b>	<b>33</b>
▶ Lesson Goals.....	33
▶ Exercise 1: Finishing the Pocket Floor .....	33
▶ Exercise 2: Finishing the Slot.....	36
<b>4. Machining the Exterior .....</b>	<b>41</b>
▶ Lesson Goals.....	41
▶ Exercise 1: Roughing the Exterior .....	41
▶ Exercise 2: Finishing the Exterior .....	46
▶ Exercise 3: Drilling a hole from the Front Plane.....	49

<b>5. Adding Details .....</b>	<b>53</b>
▶ Lesson Goals.....	53
▶ Exercise 1: Lettering with a Project Toolpath.....	53
▶ Exercise 2: Lettering with a Contour Toolpath.....	57
<b>6. Backplot/Verify Your Toolpaths .....</b>	<b>63</b>
▶ Lesson Goals.....	63
▶ Exercise 1: Backplotting All Toolpaths.....	63
▶ Exercise 2: Verifying All Toolpaths.....	64
<b>7. Posting Toolpaths .....</b>	<b>67</b>
▶ Lesson Goals.....	67
▶ Exercise 1: Posting All Toolpath Operations.....	67
▶ Sending NC Files to the Machine .....	69
<b>Machining the Tool Holder .....</b>	<b>71</b>
<b>1. Toolpath Setup.....</b>	<b>73</b>
▶ Lesson Goals.....	73
▶ Exercise 1: Assigning a Machine Definition .....	73
▶ Exercise 2: Setting Stock for Machining .....	74
<b>2. Roughing the Part .....</b>	<b>77</b>
▶ Lesson Goals.....	77
▶ Exercise 1: Roughing the Part .....	77
▶ Exercise 2: Creating a Leftover Toolpath .....	82
<b>3. Semi-Finishing the Part .....</b>	<b>85</b>
▶ Lesson Goals.....	85
▶ Exercise 1: Creating a Surface Finish Blend .....	85
▶ Exercise 2: Creating a Surface Finish Parallel Steep .....	89
▶ Exercise 3: Copy and Modify an Operation .....	91
<b>4. Finishing the Part.....</b>	<b>95</b>
▶ Lesson Goals.....	95
▶ Exercise 1: Creating a Swept 3D.....	95
▶ Exercise 2: Creating a second Swept 3D .....	98
▶ Exercise 3: Creating a Flowline .....	101

<b>5. Backplot/Verify Your Toolpaths.....</b>	<b>105</b>
▶ Lesson Goals.....	<b>105</b>
▶ Exercise 1: Backplotting all Toolpaths .....	<b>105</b>
▶ Exercise 2: Verifying All Toolpaths.....	<b>106</b>
<b>6. Posting Toolpaths .....</b>	<b>109</b>
▶ Lesson Goals.....	<b>109</b>
▶ Exercise 1: Posting All Toolpath Operations .....	<b>109</b>
▶ Sending NC Files to the Machine .....	<b>111</b>
<b>Conclusion .....</b>	<b>112</b>



# INTRODUCTION

This tutorial focuses on applying several milling toolpaths to parts previously designed in Mastercam. The objective is to provide a general overview of a handful of the toolpath options available to you. The options and parameters selected are intended to get you started on the path toward learning and exploring all of the features Mastercam has to offer.

The toolpaths created illustrate possible solutions for machining parts using 3D toolpaths - mainly surfacing operations. There are many solutions for machining these and similar parts. The goal of this tutorial is to provide the basic concepts for you to explore and expand upon. Alter and adjust the outline to fit your learning and programming environment after completing the lessons.

Toolpath creation is a repetitive, step-by-step process as demonstrated by this tutorial. Comprehensive and conceptual information can be found in the Help and other documentation installed with Mastercam.

## *Tutorial Goals*

- Open multiple part files, assign default machine definitions, and create stock boundaries for the geometry
- Create and rename toolpath groups
- Create roughing operations to efficiently remove material from the part
- Create semi-finishing operations to safely transition between roughing and finishing the part
- Create finishing operations to create a complete part with final dimensions
- Copy operations, change parameters, and regenerate the operation
- Assign toolpath parameters to meet the roughing, semi-finishing, and finishing requirements
- Backplot toolpath operations to review tool motion
- Verify the machining of the part from a stock model display
- Post all toolpath operations to an NC file, review/edit the code as necessary, and save the NC file

## *Before You Begin*

This is a module of the *Mastercam Getting Started Tutorial Series*. The series introduces general Mastercam functions and teaches basic skills for getting started with Mastercam. Other tutorial series include:

- *Focus Series*: Focuses on a specific Mastercam feature—for example, Setup Sheets or FBM Drill, and teaches basic and advanced skills.
- *Exploring Series*: Explores a single Mastercam product—for example, Mill, Solids, or Wire, and teaches in-depth skills for working with the product.

The Mastercam tutorial series is in continual development, and we will add modules as we complete them. For information and availability, please contact your local Mastercam Reseller.

---

*Note: Screen colors in the tutorial pictures enhance image quality; they may not match your Mastercam settings.*

---

## General Tutorial Requirements

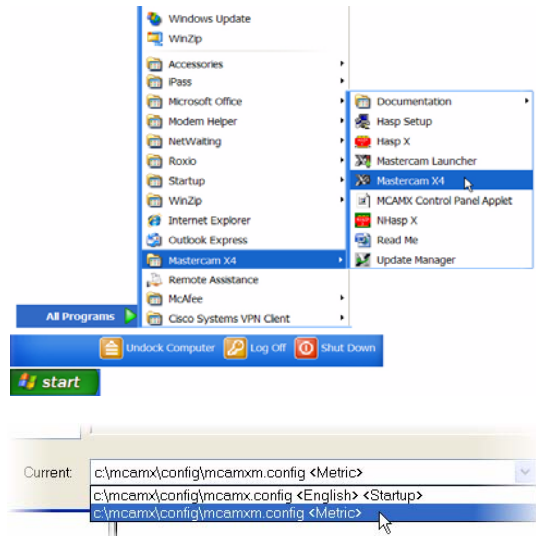
All Mastercam tutorials have the following general requirements:

- You must be comfortable using the Windows<sup>®</sup> operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition (HLE). The Demo/HLE file format (EMCX) is different from Mastercam (MCX), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- *Focus Series* and *Exploring Series* tutorials require, at minimum, a mastery of the basic Mastercam skills taught in the *Getting Started Series* modules. A general knowledge of machining principals and practices is also required.
- You must have a seat of Mastercam X4 Design or higher to complete most of the tutorials in the *Getting Started Series*.
- The *Basic 2D Machining* module in the *Getting Started Series* requires, at minimum, a seat of Mill Entry or Router Entry.
- The *Basic 3D Machining* module in the *Getting Started Series* requires Mill Level 3 or Router Pro.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam workstation, either with the tutorial or in any location that you prefer.
- The *Getting Started Series* tutorials are available in an Adobe<sup>®</sup> Flash<sup>®</sup> compatible video format. Additional tutorial videos may also be available. Contact your local Mastercam Reseller for more information.
- You must install Adobe Flash Player to display tutorial videos. You can download Adobe Flash Player from [www.adobe.com](http://www.adobe.com).
- You must configure Mastercam to work in metric units. Complete the instructions in the following section **Preparing for a Tutorial** to set Mastercam to metric.

## Preparing for a Tutorial


Before you start a tutorial, be sure you have completed the following tasks:

- 1 Start Mastercam using your preferred method:
  - ♦ Double-click Mastercam's desktop icon.
  - Or
  - ♦ Launch Mastercam from the Windows Start menu.
- 2 Select the metric configuration file:
  - a Select **Settings, Configuration** from Mastercam's menu.
  - b Choose **..\config\mcamxm.config** <**Metric**> from the **Current** drop-down list.
  - c Click **OK**.



## If You Need More Help

There are many ways to get help with Mastercam, including the following:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's menu bar or by pressing [**Alt+H**] on your keyboard. Also, most dialog boxes and ribbon bars feature a Help button  that opens Mastercam Help directly to related information.
- *Online help*—You can search for information or ask questions on the Mastercam Web forum, located at [www.emastercam.com](http://www.emastercam.com). You can also find a wealth of information, including many videos, at [www.mastercam.com](http://www.mastercam.com) and [www.mastercamedu.com](http://www.mastercamedu.com).
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or [support@mastercam.com](mailto:support@mastercam.com)) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Documentation feedback*—For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at [techdocs@mastercam.com](mailto:techdocs@mastercam.com).
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit [www.mastercamu.com](http://www.mastercamu.com), or email [training@mastercam.com](mailto:training@mastercam.com).

## *Additional Documentation*

You can find more information on using Mastercam in the following materials, located in the \Documentation folder of your Mastercam installation:

- *Mastercam X4 Installation Guide*
- *Mastercam X4 Administrator Guide*
- *Mastercam X4 Quick Start*
- *Mastercam X4 Reference Guide*
- *Mastercam X4 Transition Guide*
- *Mastercam X4 Quick Reference Card*
- *Mastercam X4 Wire Getting Started Guide*
- *Version 9 to X Function Map*



---

# **Section 1**

## **Machining the Connector**



## LESSON 1

# Toolpath Setup

Before generating toolpaths for a part, you must select a machine definition. Defining stock creates a visual representation of the stock placed in your machine. Creating and naming toolpath groups will organize operations and maintain a logical structure for the part. This lesson covers these topics.

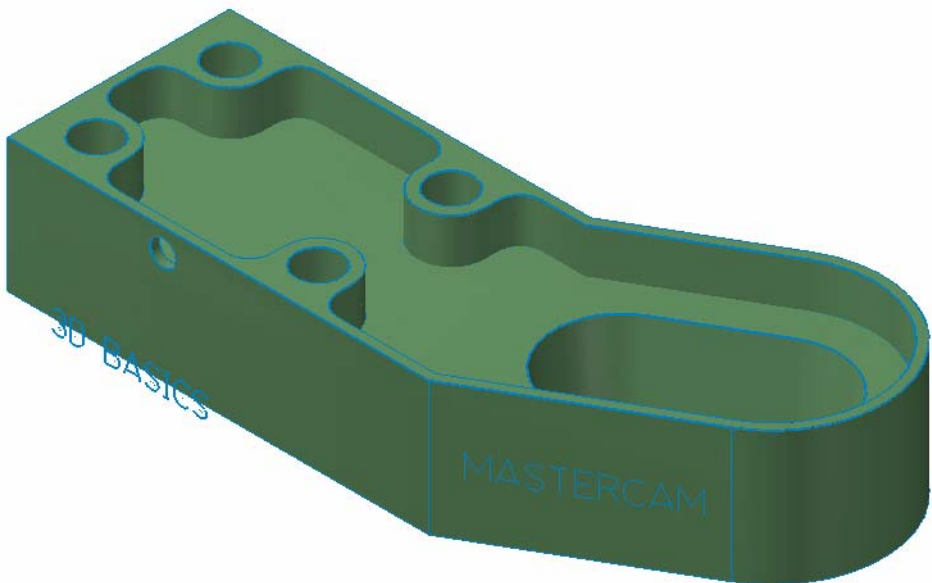
### *Lesson Goals*

- Open a part file and assign a machine definition.
- Define stock to be used in machining the part.
- Rename toolpath groups.

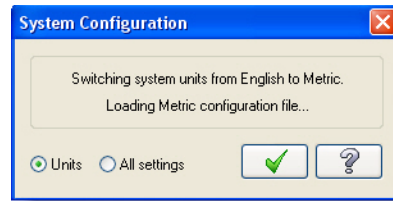
### *Exercise 1: Assigning a Machine Definition*

Toolpaths are organized in machine groups and toolpath groups. A machine group is created when a machine definition is assigned. This exercise guides you through selecting a machine definition.

- 1 Open the tutorial part file `Basic_3D_Machining_Part1_Start.MCX`, which was provided with the tutorial.

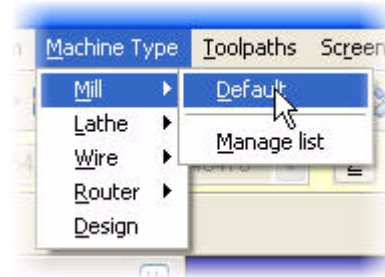


- 2 Click **OK** if prompted to switch to a metric configuration.



- 3 Choose **Machine Type, Mill, Default** to open the default Mill machine definition.

In Mastercam, you select a machine definition before creating any toolpaths. The machine definition is a model of your machine tool's capabilities and features. It acts like a template for setting up machining jobs.




---

*Note: Parts that have previously been saved with a machine definition automatically load the associated machine definition.*

---

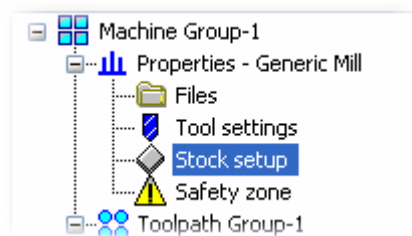
- 4 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

## Exercise 2: Setting Stock for Machining

Define stock to help you more clearly visualize toolpaths. The stock can help to generate toolpaths, or be used when backplotting or verifying toolpaths. This exercise guides you through creating a stock boundary for your part.

- 1 In the Toolpath Manager, select **Stock setup**. If necessary, click the [+] next to Properties to expand the list.

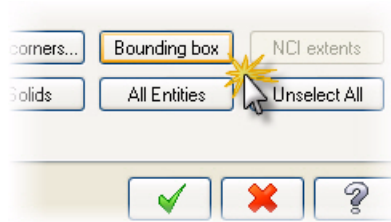
The Machine Group Properties dialog box opens to the Stock Setup tab.



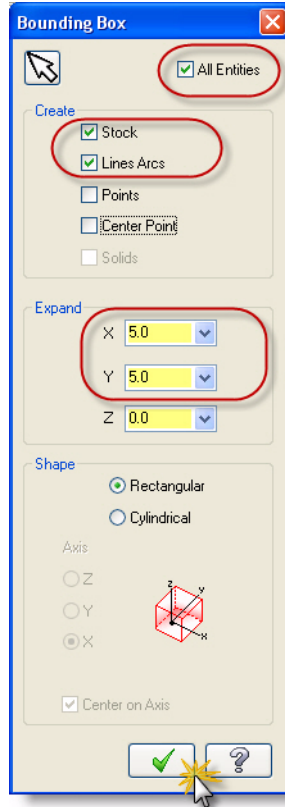
**2 Click Bounding box.**

The Bounding Box dialog box opens.

Bounding box is a quick and convenient method for creating stock around the outer boundary of your geometry.



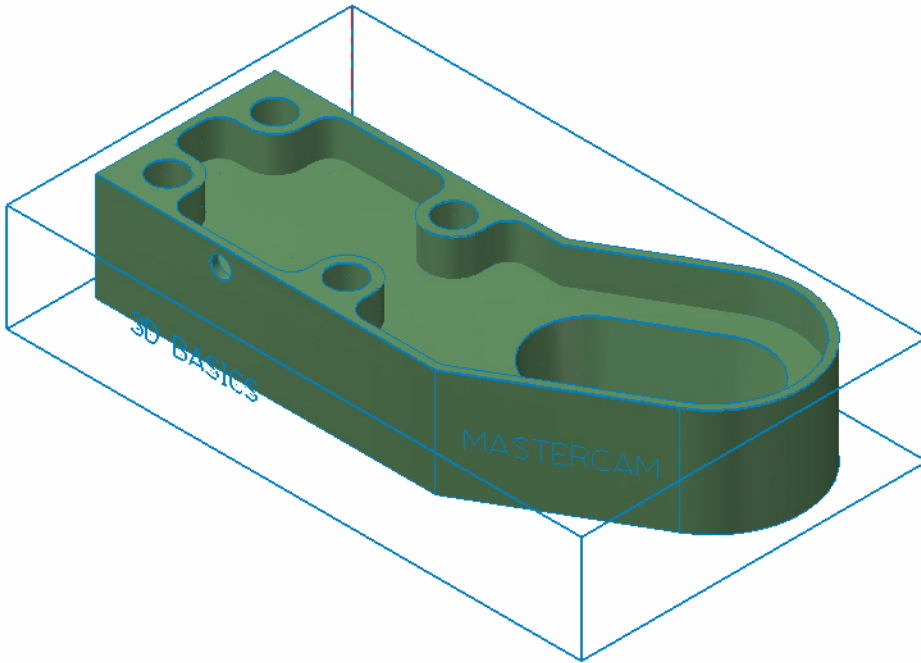
**3 Set the options and parameters as shown, then click OK.**



**4 Click OK in the Machine Group Properties dialog box.**



- 5 Press **[Alt+F1]**, or right-click and select **Fit**, to fit the geometry to the screen.



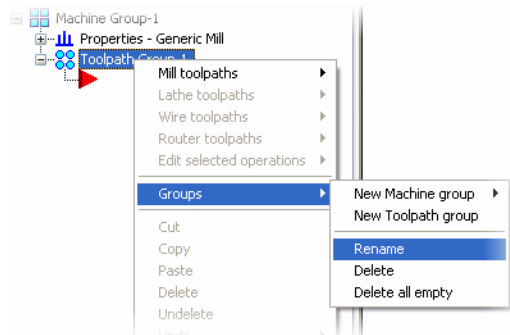
- 6 Choose **File, Save** or click the **Save** button to save the part with the machine definition and defined stock.



## Exercise 3: Creating Toolpath Groups

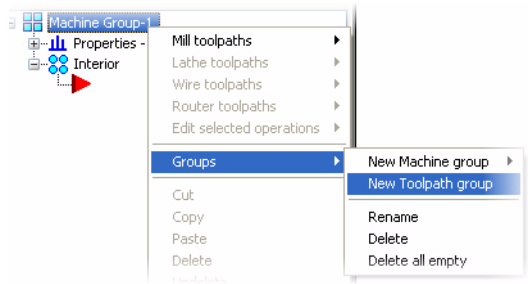
Toolpath groups aid in organizing toolpaths logically. A well organized part file makes it easy for any user to understand the process involved in making the part. This exercise guides you through creating and renaming toolpath groups.

- 1 In the Toolpath Manager, right-click **Toolpath Group-1**, select **Groups**, **Rename**.



- 2 Type **Interior** and press **[Enter]**.
- 3 In the Toolpath Manager, right-click on **Machine Group-1**, select **Groups, New Toolpath Group**.

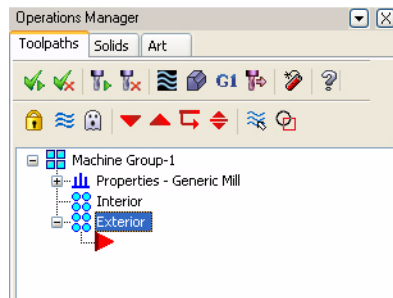
The new toolpath group is named Toolpath Group-1. If you create the second toolpath group before renaming the first, the new group is named Toolpath Group-2.



**TIP:** Right-clicking the machine group name creates the new toolpath group at the same level in the tree as the Interior toolpath group. You can right-click **Interior** to create a new toolpath group one level below Interior.

- 4 Type **Exterior** and press **[Enter]**.

The Toolpath Manager should look as shown.



- 5 Choose **File, Save** or click the **Save** button to save the part.







## LESSON 2

# Roughing the Interior

The first toolpaths for a part typically involve removing large amounts of material. This is referred to as roughing the part. This lesson guides you through several roughing toolpaths. These include drilling the holes, roughing the inner pocket, roughing the inner slot, and rest roughing the interior portion of the part.

### *Lesson Goals*

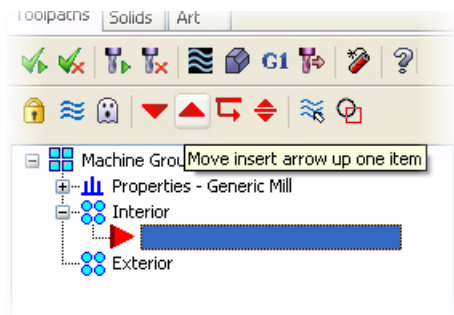
- Create a drill toolpath (including drill point selection, choosing tooling, and setting machine values).
- Create High Speed and surface roughing operations.
- Create a High Speed rest roughing operation (including toolpath refinement).

### *Exercise 1: Drilling the Holes*

- 1 In the Toolpath Manager, click the **Move insert arrow up one item** button.

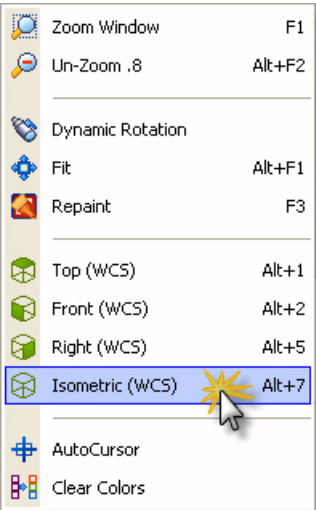
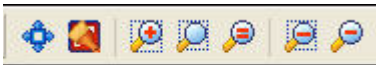
The Toolpath Manager should look as shown after performing this step.

The insert arrow controls the location where new toolpaths will be added to the Toolpath Manager.

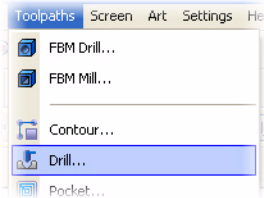


- 2 Right-click in the graphics window and choose **Isometric (WCS)** from the menu to view the part and toolpath in the isometric view.

You may need to center the part in the graphics window to see it. The easiest way to do this is to use the graphics window right-click menu and select **Fit** to fit the part to the screen size [Alt+F1]. Then unzoom using [Alt+F2]. You can also use the fit/zoom/unzoom buttons in the View Manipulation toolbar.

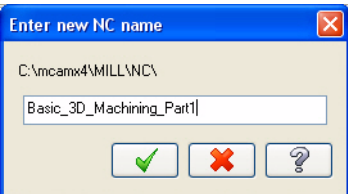


- 3 Choose **Toolpaths, Drill**.

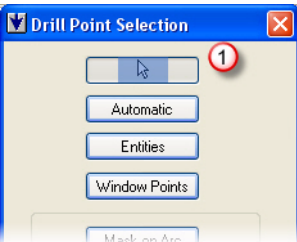


- 4 Click **OK** when prompted to enter new NC name.

The name displayed will be the name you chose to save the file under. You may change the NC file name now if necessary.

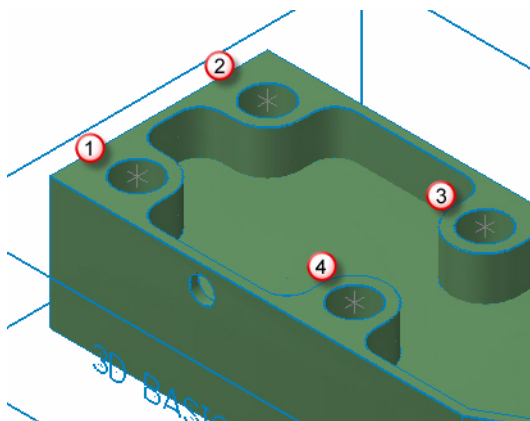


- 5 Use the default option for **Select drill point position in the graphics screen** in the Drill Point Selection dialog box.



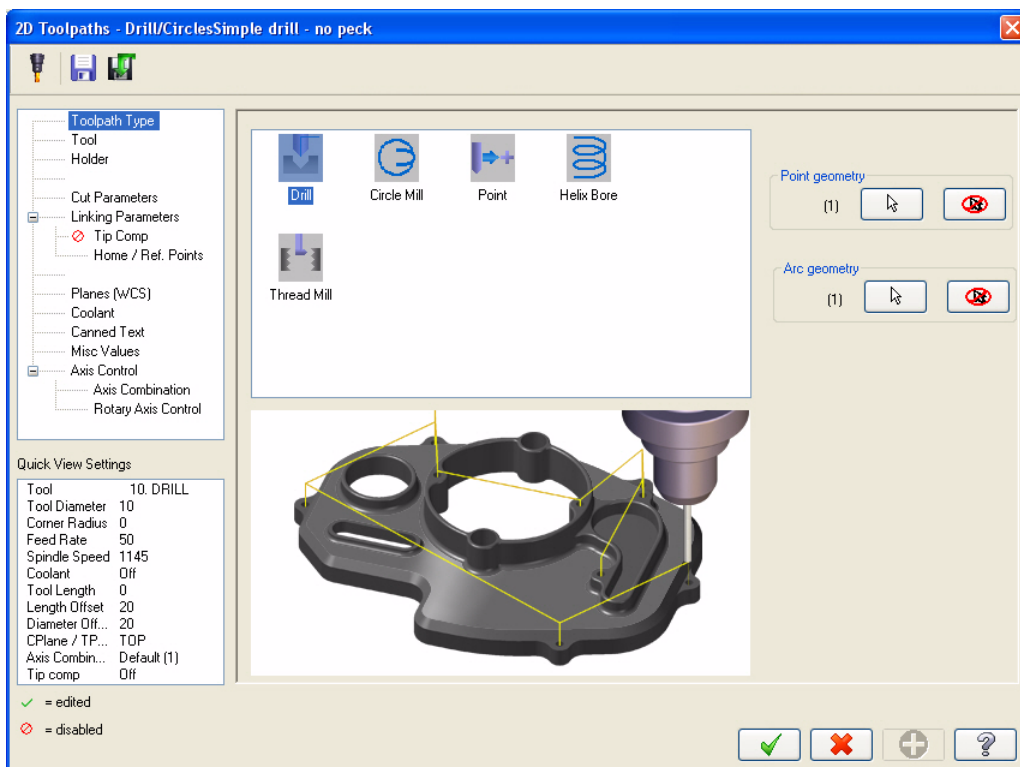
- 6 Select the arc centers of the four holes as indicated.

The autocursor changes to indicate the arc center as you approach.

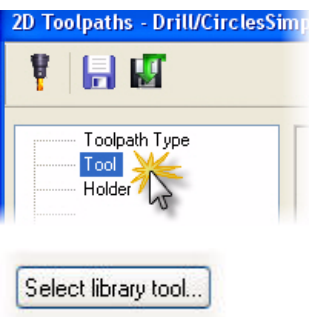


- 7 Click **OK** in the Drill Point Selection dialog box.

The 2D Toolpaths dialog box opens to the Toolpath Type page. Drill is selected as the toolpath type. (Do not click **OK** on the 2D Toolpaths dialog box until all pages are complete.)



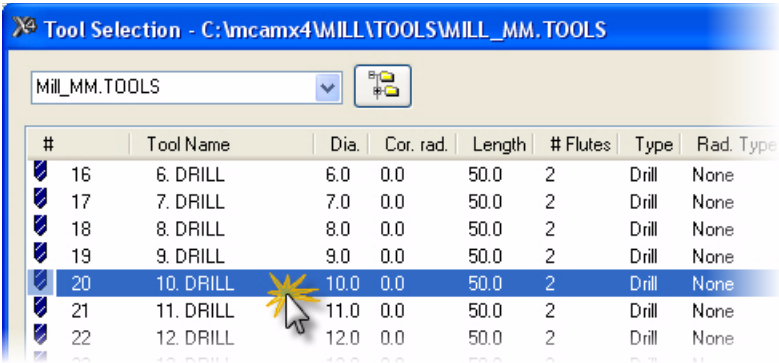
- 8 Select **Tool** from the Tree View pane on the left.



- 9 Click the **Select library tool** button.

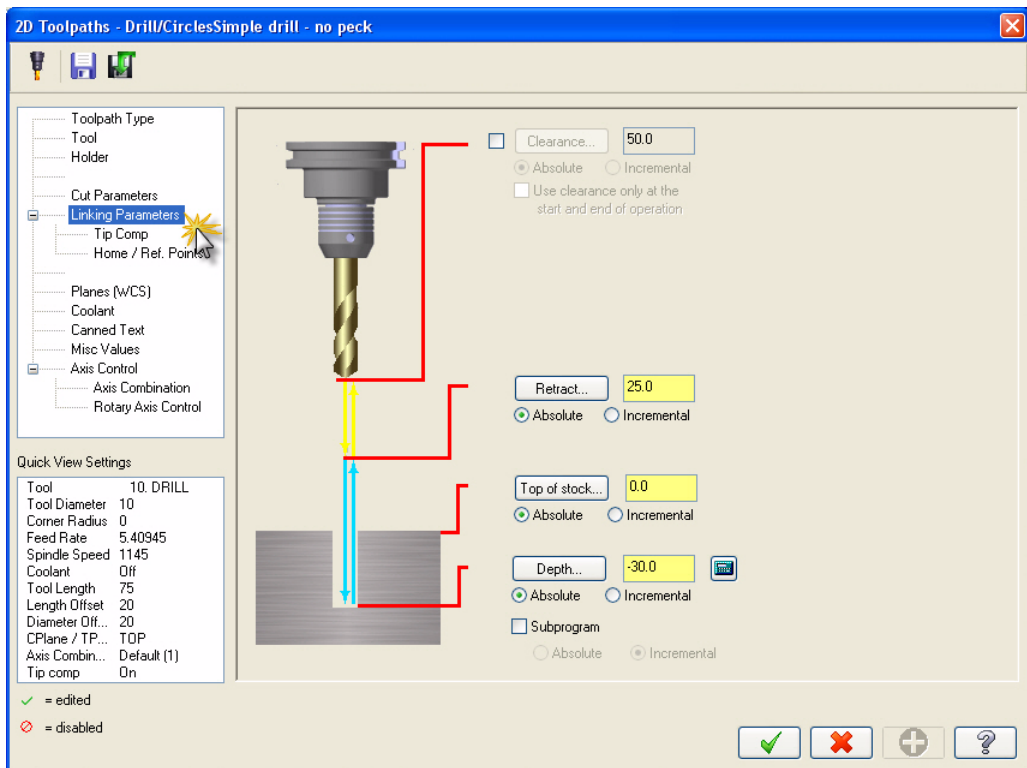
The default metric tool library opens.

- 10 Select the 10mm drill and click **OK**.



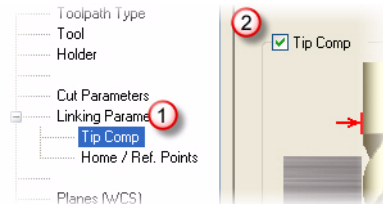
**TIP:** Use the default tool settings for the purposes of the tutorial. Tool speeds, feeds, number, and other parameters should be changed to fit your machine and tooling before cutting the part.

**11** Select **Linking Parameters** from the Tree View pane and enter values as shown.

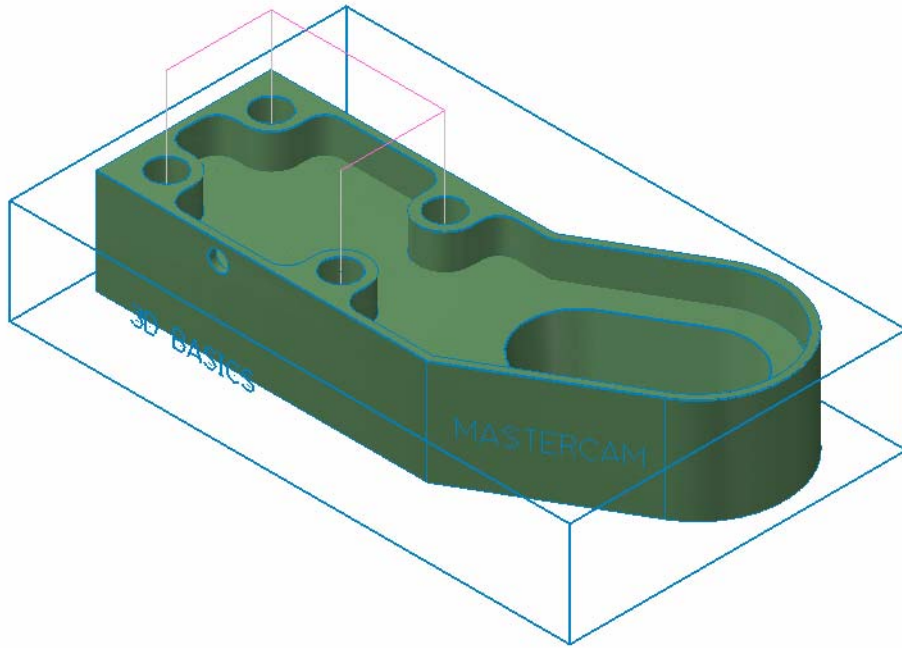


These values will control the depth the tool will move to, where the top of stock is located, as well as the height to retract the tool to.

**12** Select **Tip Comp** from the Tree View pane. Click the **Tip Comp** checkbox to activate this feature. Use default values for the tip comp parameters.



- 13 Click **OK** in the 2D Toolpath dialog box to generate the drill toolpath.



- 14 Click the **Toggle toolpath display on selected operations** button. If necessary, select the Drill/Counterbore operation.

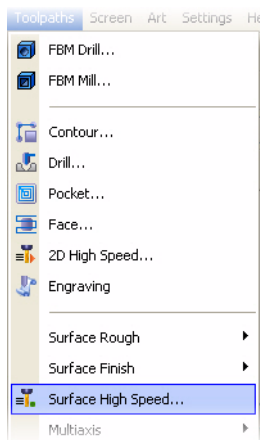
The toolpath display for the Drill toolpath is turned off. Perform this step after the creation of each operation for clarity in selecting geometry for subsequent steps.



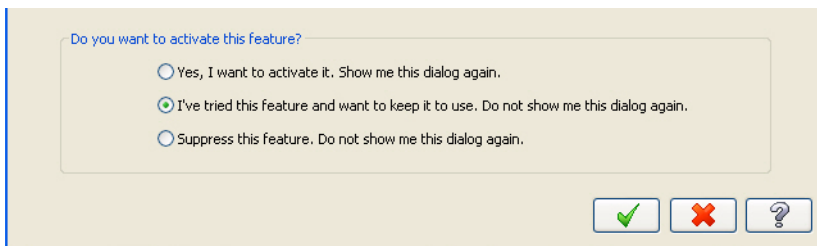
- 15 Save your part file.

## Exercise 2: Roughing the Pocket

- 1 Choose **Toolpaths, Surface High Speed**.



- 2 If the **New 3D Advanced Toolpath Refinement Feature!** dialog box opens, select the option to eliminate this dialog box and make refinement available for use. Click **OK** to close the dialog box.

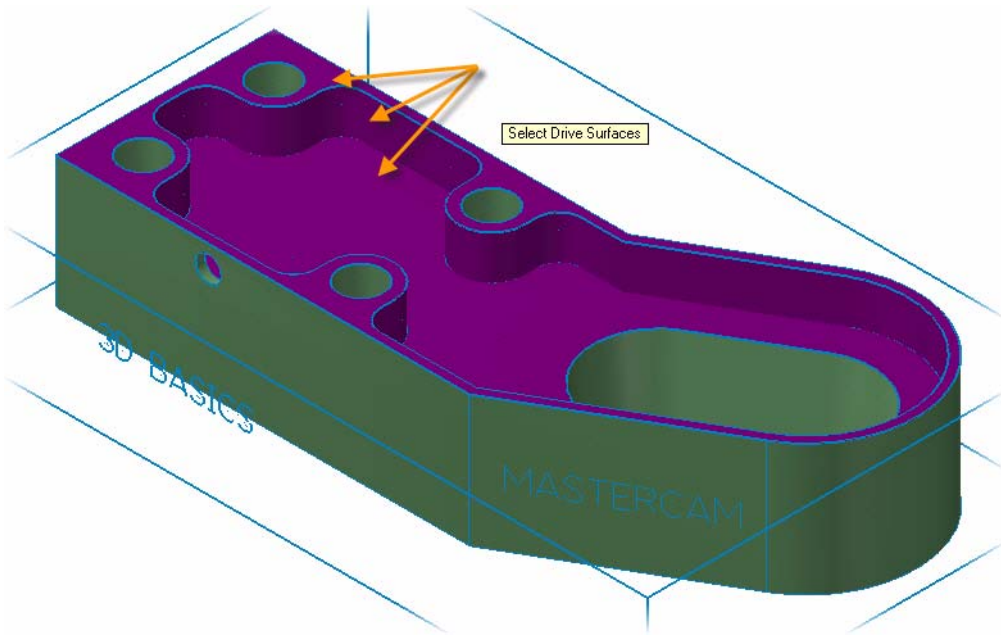


---

*Note: This dialog box introduces the 3D Advanced Toolpath Refinement feature. Use this feature to fine-tune your toolpath motion.*

---

- 3 Follow the prompt to select drive surfaces as shown.

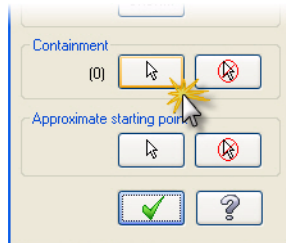


- 4 Press **[Enter]** or click the **End Selection** button to accept the selection.



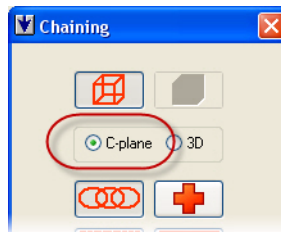
- 5 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

The Chaining dialog box opens.



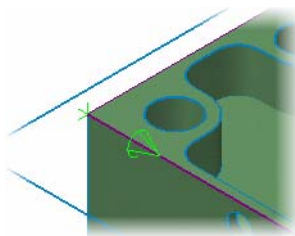
- 6 Click the **C-plane** radio button.

C-plane selection limits chaining to entities that are parallel to the current construction plane.





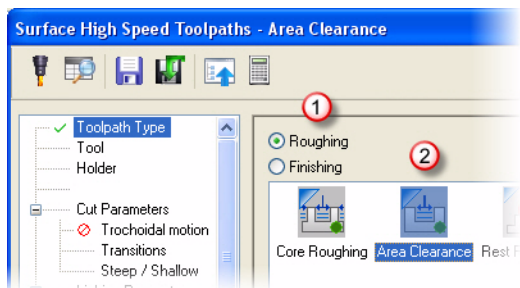
- 7 Select the chain on the top of the part. The selection can be in either direction for a closed containment boundary.



- 8 Click **OK** on the Chaining dialog box to return to the Toolpath/Surface selection dialog box.  
9 Click **OK** on the Toolpath/surface selection dialog box.

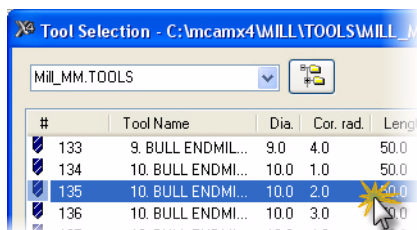
The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.

- 10 Select **Roughing** and **Area Clearance** on the Toolpath Type page.



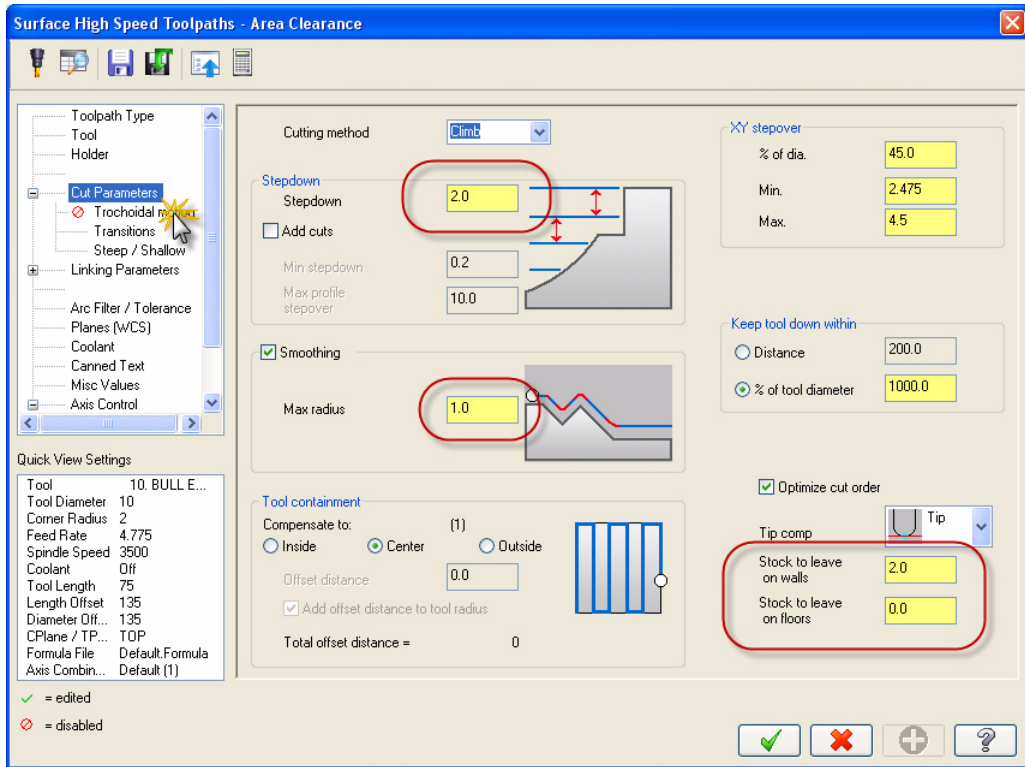
- 11 Select **Tool** from the Tree View pane.  
12 Click the **Select library tool** button. The default metric tool library opens.  
13 Select the 10mm bull endmill with 2mm corner radius and click **OK**.

Select library tool...

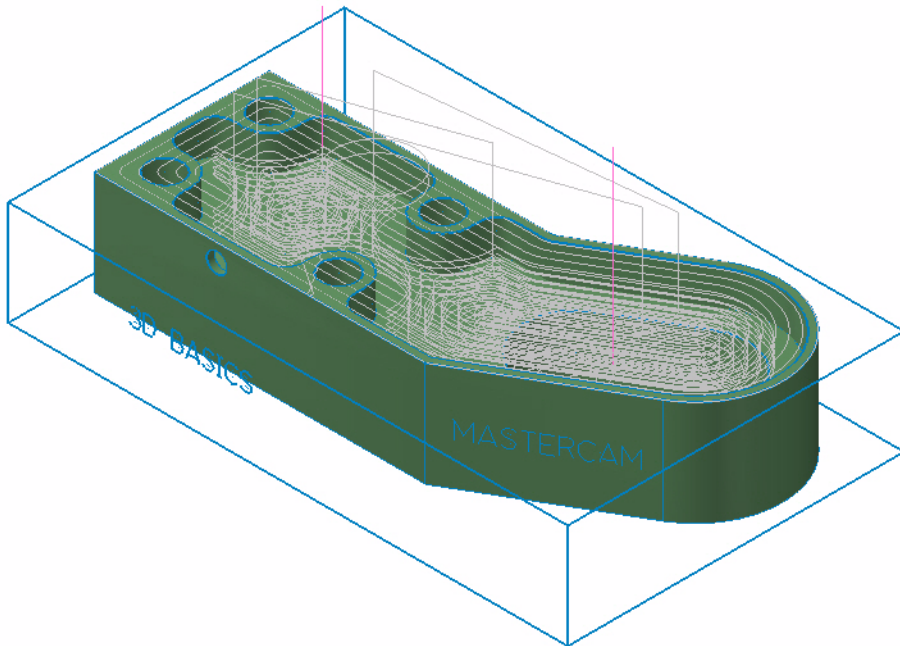


**TIP:** Adjust the Filter options on the right of the dialog box to limit the types of tools displayed.

14 Select **Cut Parameters** from the Tree View pane. Set the parameters as indicated.



- 15 Click OK to generate the area clearance toolpath.

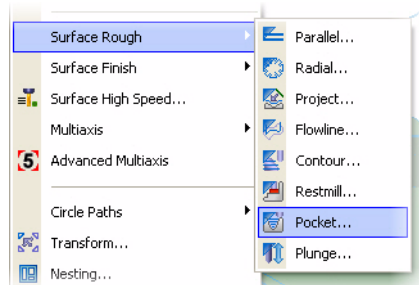


- 16 Turn off the toolpath display for the operation.

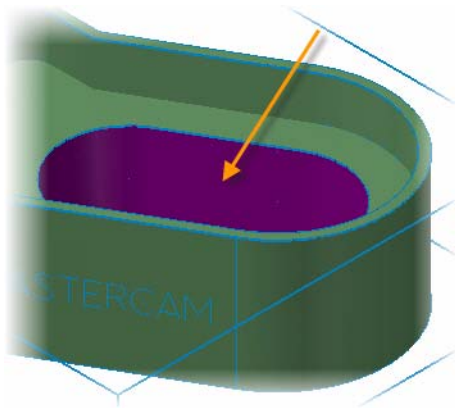
- 17 Save your part file.

### *Exercise 3: Roughing the Slot*

- 1 Choose **Toolpaths, Surface Rough, Pocket**.



- 2 Follow the prompt to select drive surfaces as shown.



- 3 Press **[Enter]** or click the **End Selection** button.
- 4 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

The Chaining dialog box opens.

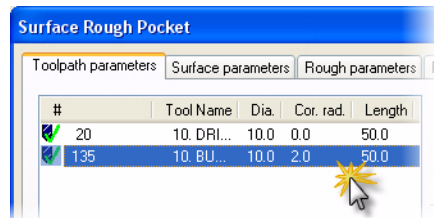
- 5 Click the **C-plane** radio button in the Chaining dialog box.
- 6 Select the chain at the top of the slot as shown.



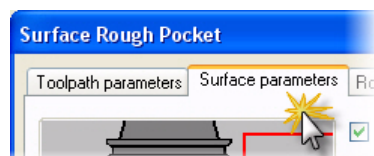
- 7 Click **OK** on the Chaining dialog box.
- 8 Click **OK** on the Toolpaths/surface selection dialog box.

The Surface Rough Pocket dialog box opens to the Toolpath parameters tab.

- 9 Select the 10mm bull endmill with the 2mm corner radius showing in the tool list.

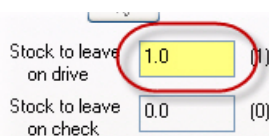


- 10 Click the **Surface parameters** tab.

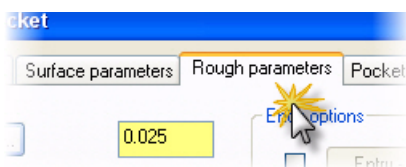


- 11** Set **Stock to leave on drive** to a value of 1mm.

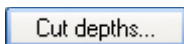
Remaining parameters on this tab will be left at defaults.



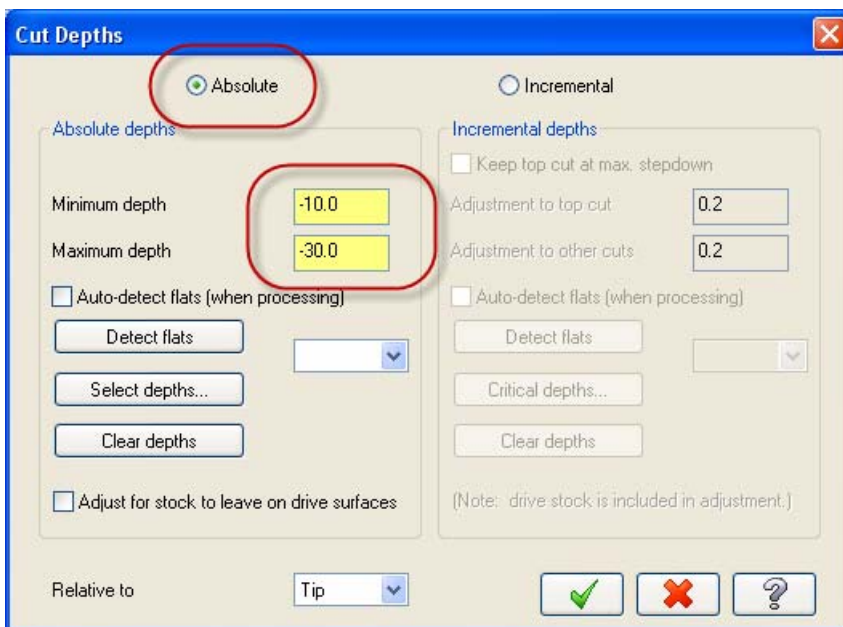
- 12** Click the **Rough parameters** tab.



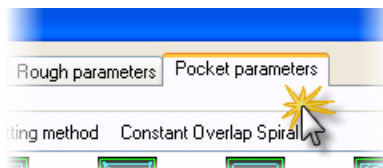
- 13** Click the **Cut Depths** button.



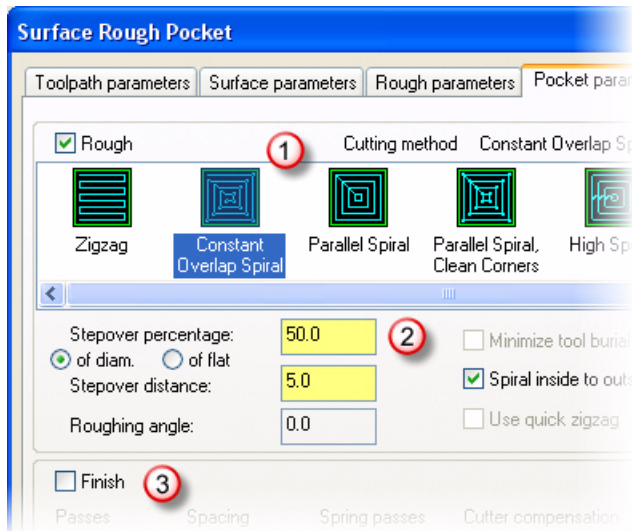
- 14** Set the parameters as shown and click **OK**.



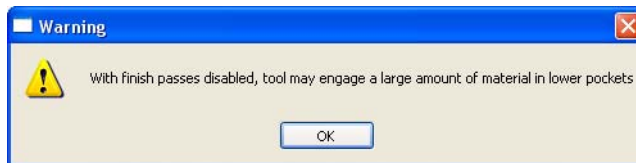
- 15** Click the **Pocket parameters** tab.



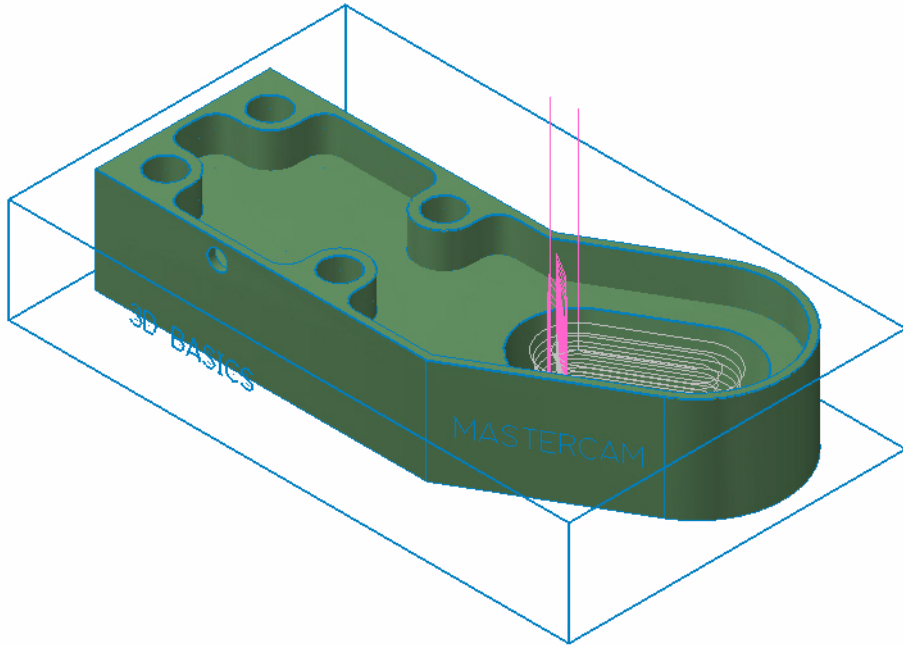
- 16 Click **Constant Overlap Spiral**, set the parameters as shown, and click **OK**.



- 17 The following warning message displays because the finish passes are turned off. Click **OK** to continue.



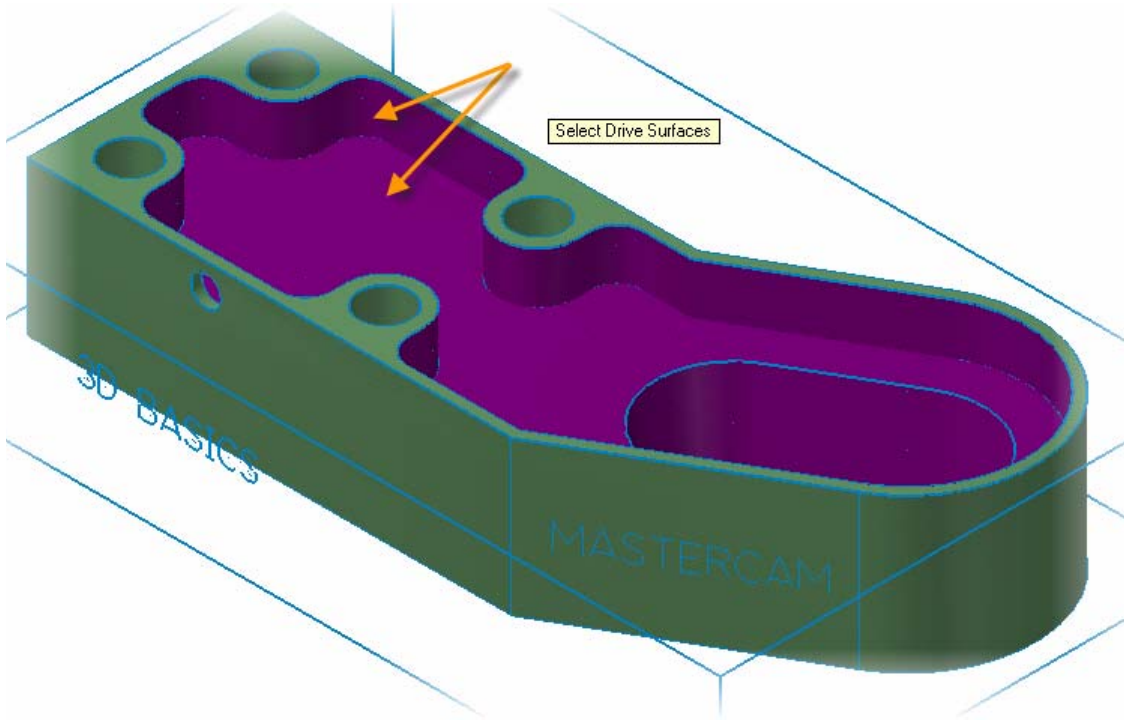
- 18** The surface rough pocket toolpath is generated on the selected geometry.



- 19** Turn off the toolpath display for the operation.
- 20** Save your part file.

## *Exercise 4: Rest Roughing the Interior*

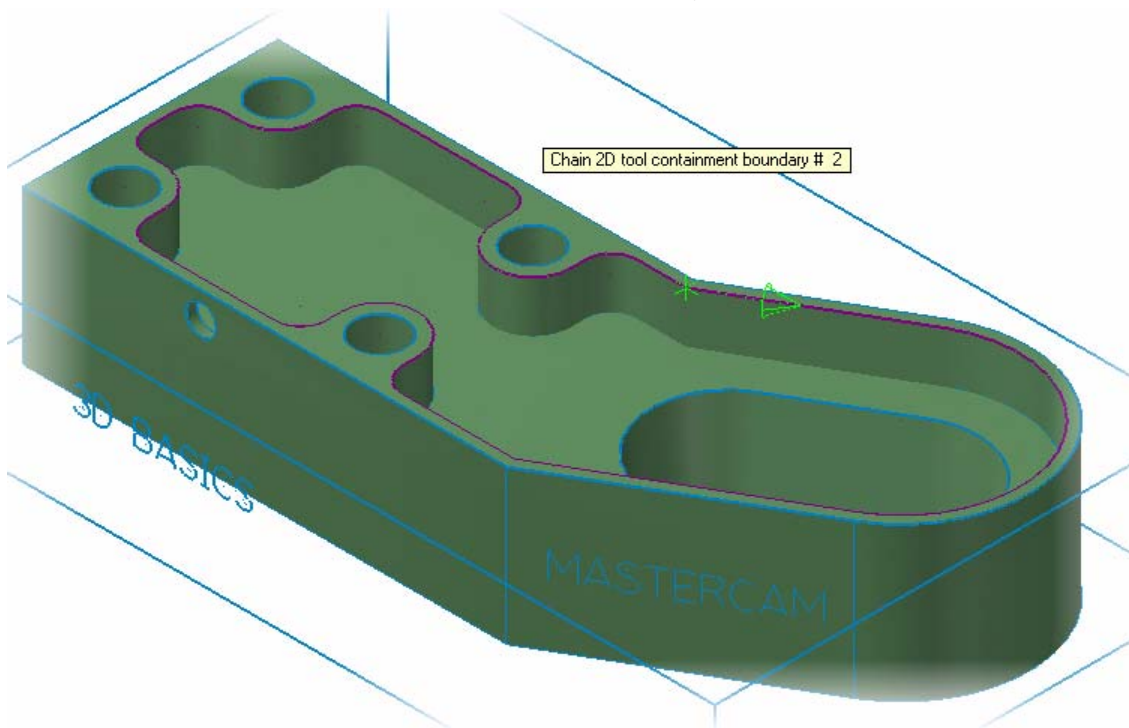
- 1 Choose **Toolpaths, Surface High Speed**.
- 2 Follow the prompt to select drive surfaces as shown.



- 3 Press **[Enter]** or click **End Selection** to accept the selection.
- 4 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.  
The Chaining dialog box opens.
- 5 Click the **C-plane** radio button.



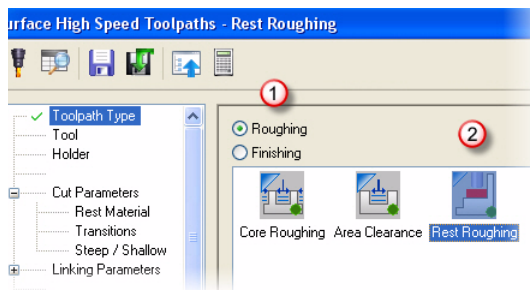
- 6 Follow the prompt to chain a containment boundary as shown.



- 7 Click **OK** in the Chaining and the Toolpath/surface selection dialog boxes.

The Surface High Speed toolpaths dialog box opens.

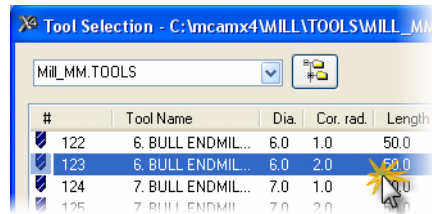
- 8 Select **Toolpath Type** from the Tree View pane, and then select **Roughing** and **Rest Roughing** on the Toolpath Type page.



- 9 Select **Tool** from the Tree View pane.
- 10 Click the **Select library tool...** button. The default metric tool library opens.

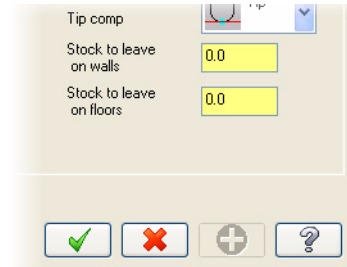
Select library tool...

- 11** Select the 6mm bull endmill with the 2mm corner radius and click **OK**.



- 12** Select **Cut Parameters** from the Tree View pane.

Set the stock to leave values as indicated.

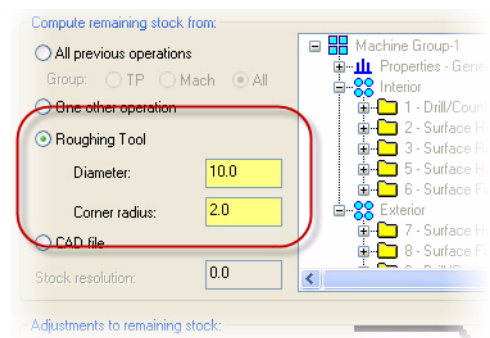


- 13** Select **Rest Material** from the Tree View pane.

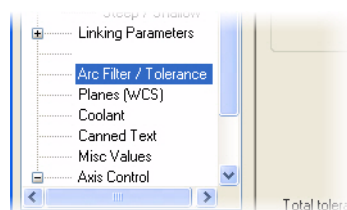
Click the plus sign in front of Cut Parameters to expand the tree if necessary.



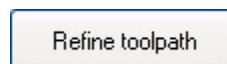
- 14** Select **Roughing Tool** and set the values as shown.



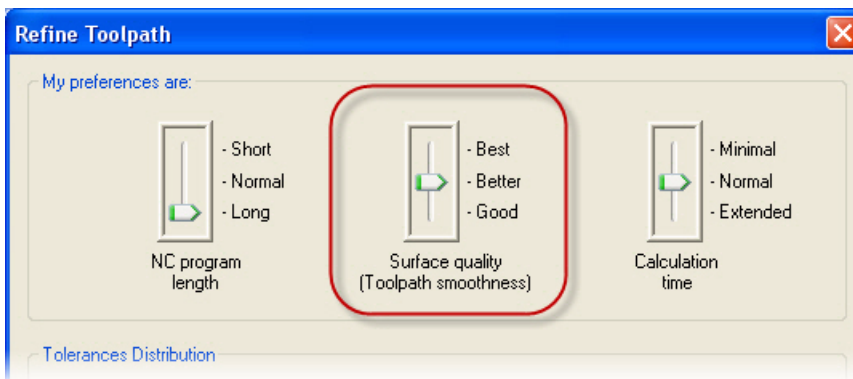
- 15** Select **Arc Filter/Tolerance** from the Tree View pane.



- 16** Click the **Refine toolpath** button.  
The Refine Toolpath dialog box opens.



- 17 Move the slider for Surface quality to the **Better** setting.

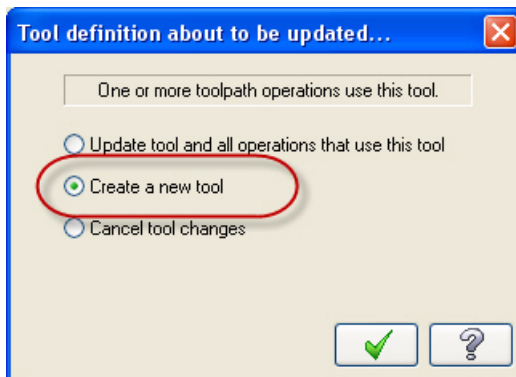


---

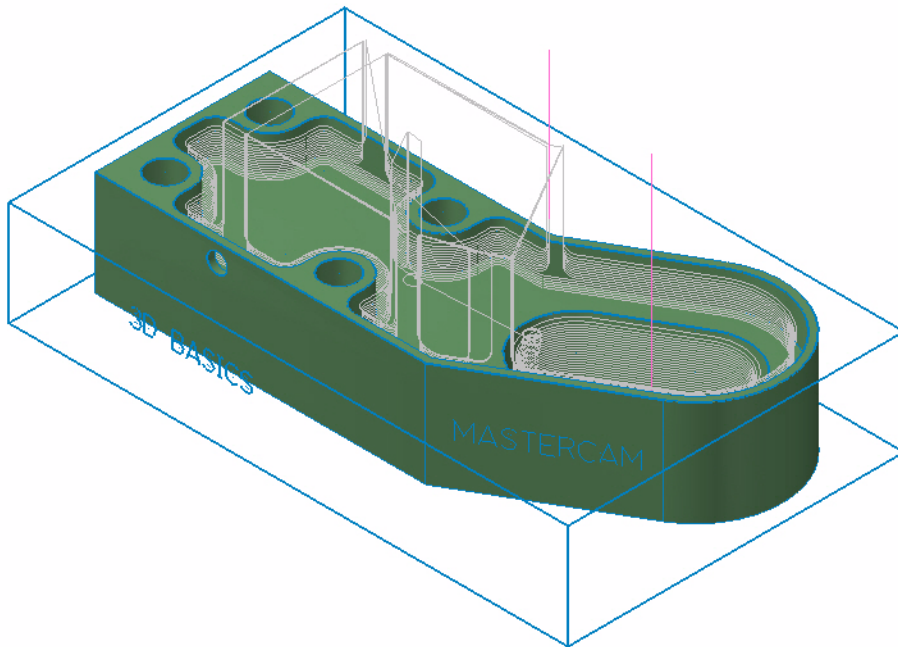
*Note: Move the slider(s) to quickly adjust parameters in the dialog box. Consult Help for more detailed information.*

---

- 18 Click **OK** to accept the settings on the Refine Toolpath dialog.
- 19 Click **OK** in the Surface High Speed Toolpath dialog box.
- 20 If the following dialog box displays, select **Create a new tool**, then click **OK**.



- 21** The rest roughing toolpath is generated on the selected geometry.



- 22** Turn off the toolpath display for the operation.
- 23** Save your part file.

## LESSON 3

# Finishing the Interior

After roughing a part, or section of a part, finishing operations will generally need to be performed. Finishing operations remove remaining stock while improving surface finish quality. Tighter tolerances, smaller stepover values, smaller tools, as well as toolpath refinement will be used in this lesson to achieve the desired finish quality.

### *Lesson Goals*

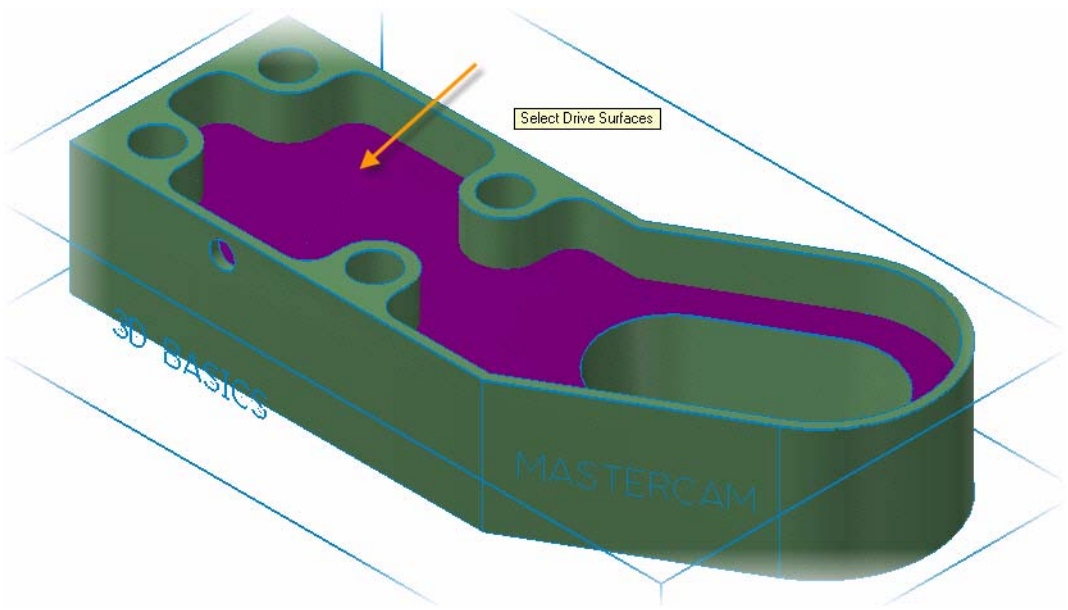
- Create a finishing operation for the pocket floor.
- Create a finishing operation for the slot.

### *Exercise 1: Finishing the Pocket Floor*

- 1 Choose **Toolpaths, Surface High Speed**.

The prompt to select Drive Surfaces appears.

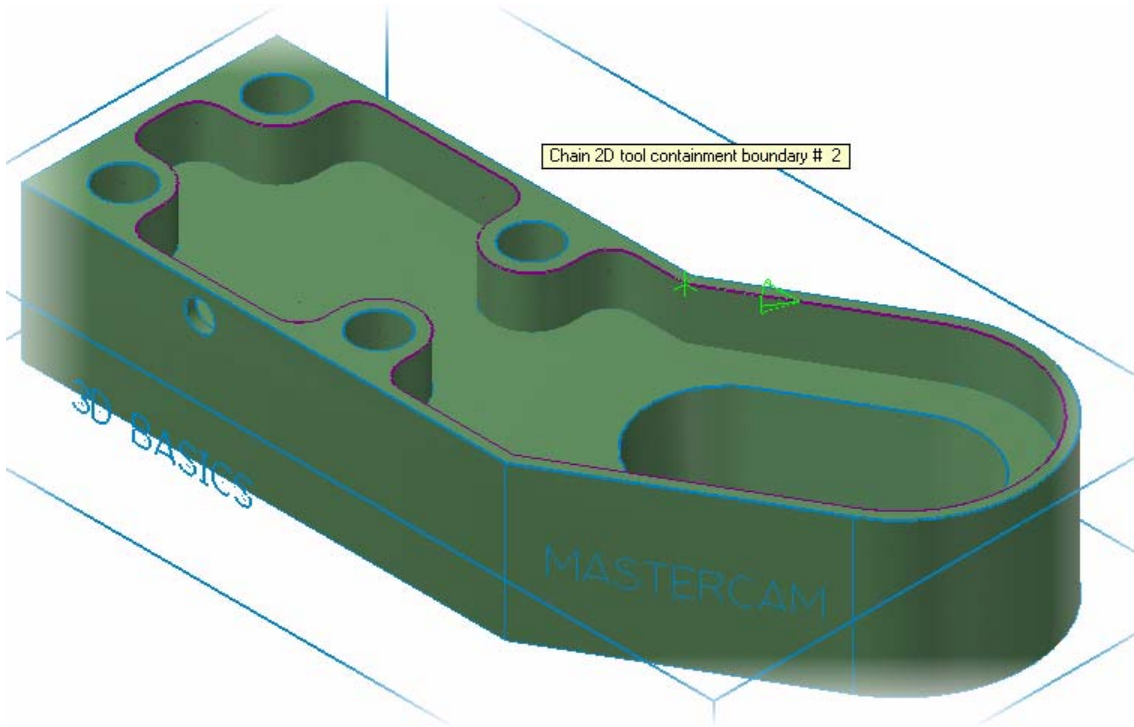
- 2 Follow the prompt to select drive surfaces as shown.



- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 4 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

The Chaining dialog box opens.

- 5 Click the **C-plane** radio button.
- 6 Follow the prompt to chain a containment boundary as shown.

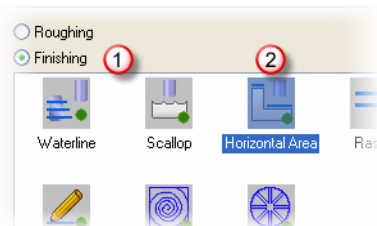


- 7 Click **OK** in the Chaining and the Toolpath/surface selection dialog boxes.

The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.

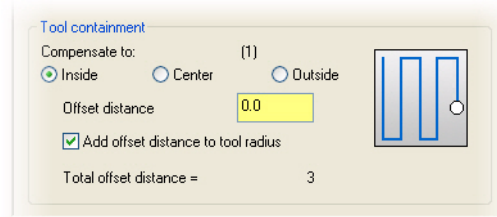
- 8 Select the Toolpath Type page, and then **Finishing** and **Horizontal Area**.

Horizontal Area is designed to quickly and efficiently machine flat areas of a part.

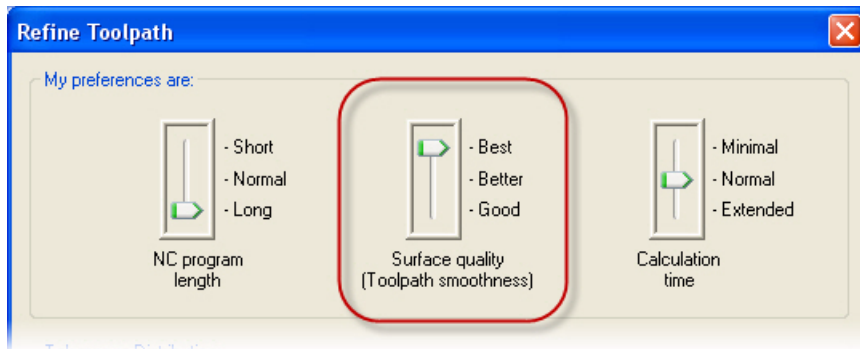


- 9 Select **Tool** from the Tree View pane.
- 10 Select the 6mm bull endmill with the 2mm corner radius showing in the tool list.

- 11 Select **Cut Parameters** from the Tree View pane. Set the tool containment as shown.

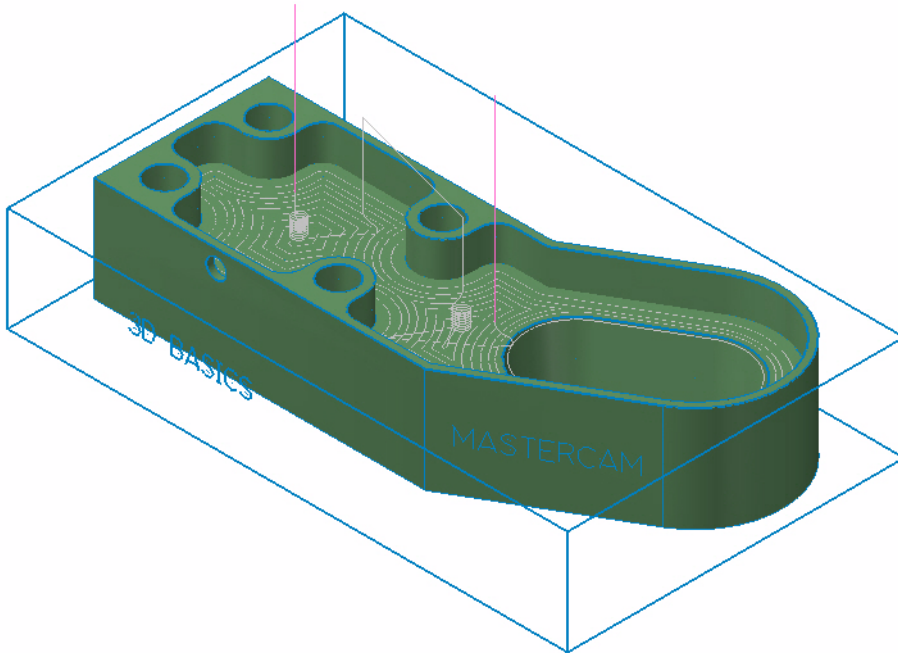


- 12 Select **Arc Filter/Tolerance** from the Tree View pane.
- 13 Click the **Refine Toolpath** button.
- 14 Move the slider for Surface quality to the **Best** setting.



- 15 Click **OK** to accept the settings on the Refine Toolpath dialog.

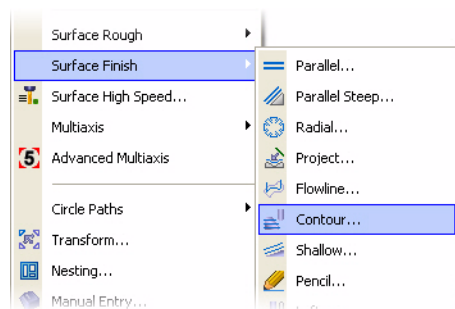
- 16 Click **OK** in the Surface High Speed Toolpath dialog box to generate the horizontal area toolpath.



- 17 Turn off the toolpath display for the operation.
- 18 Save your part file.

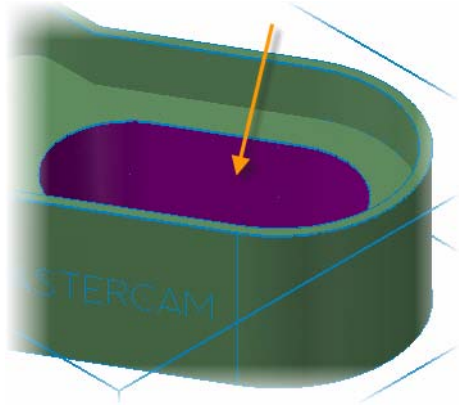
## *Exercise 2: Finishing the Slot*

- 1 Choose **Toolpaths, Surface Finish, Contour**.





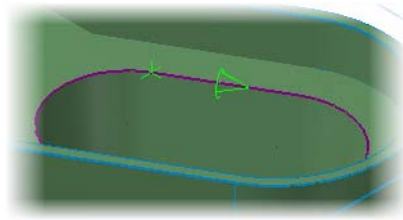
- 2 Follow the prompt to select drive surfaces as shown.



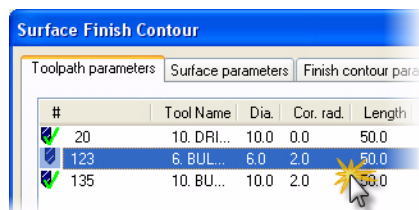
- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 4 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

The Chaining dialog box opens.

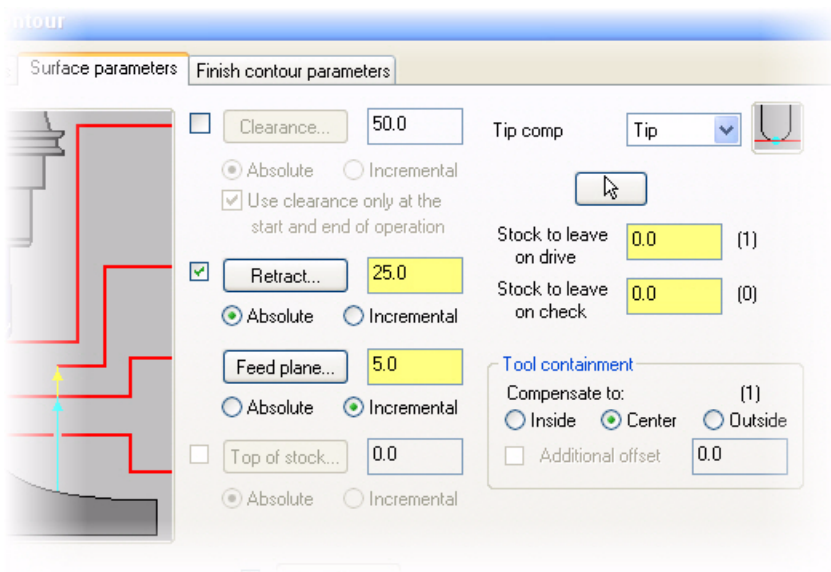
- 5 Click the **C-plane** radio button.
- 6 Follow the prompt to chain a containment boundary as shown.



- 7 Click **OK** on the Chaining and Toolpaths/surface selection dialog box.  
The Surface Finish Contour dialog box opens to the Toolpath parameters tab.
- 8 Select the 6mm bull endmill with the 2mm corner radius showing in the tool list.

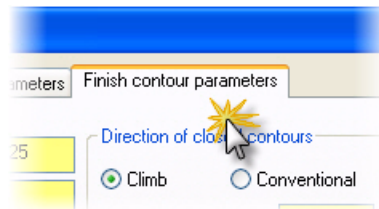


9 Click the **Surface parameters** tab. Set the parameters as shown.

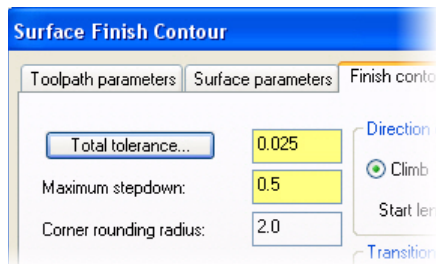


*Note: Retract is set to absolute to ensure proper clearance when the tool is pulling out of the deep slot.*

10 Click the **Finish contour parameters** tab.



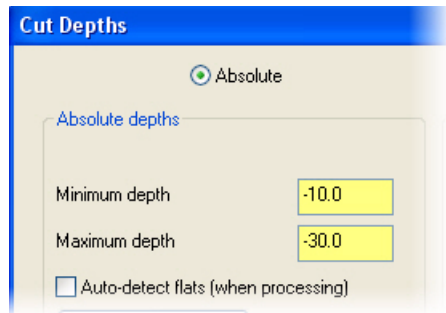
11 Set the parameters as shown.



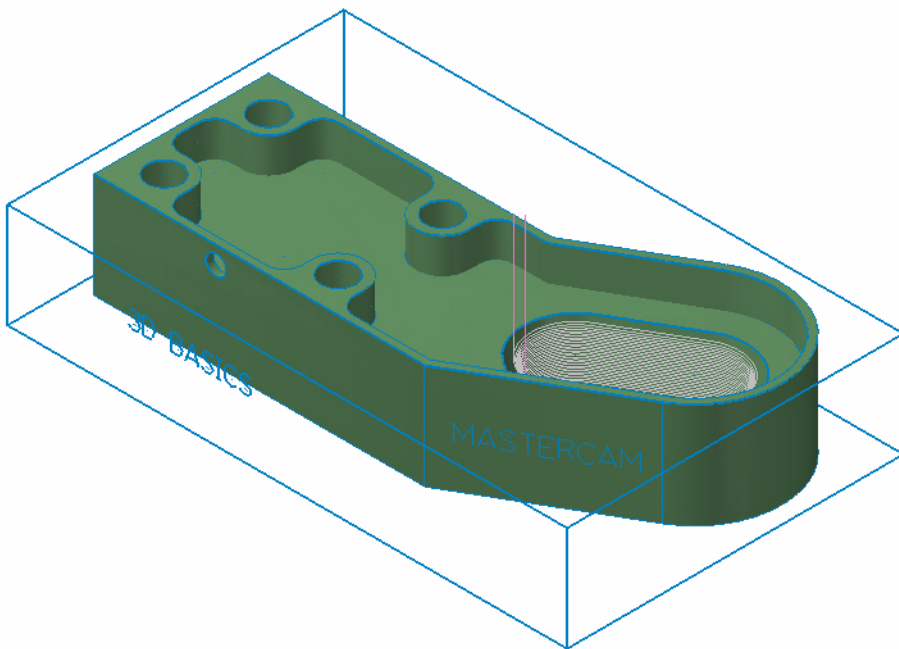
12 Click the **Total tolerance** button.

13 Move the slider for Surface quality to the Best setting and click **OK**.

- 14 Click the **Cut Depths** button.
- 15 Set the parameters as shown and click **OK**.



- 16 Click **OK** from the Surface Finish Contour dialog box to generate the finish contour toolpath.



- 17 Turn off the toolpath display for the operation.
- 18 Save your part file.



## LESSON 4

# Machining the Exterior

Roughing and finishing are required on the exterior of the part provided. The simplicity of the exterior surfaces allows for a single roughing and a single finishing operation. This lesson will cover each of those as well as using a different tool plane to drill a hole.

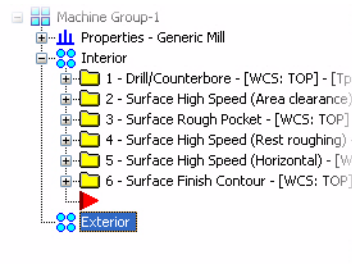
### *Lesson Goals*

- Create a Core Roughing toolpath.
- Create a Surface Finish Contour toolpath.
- Create a Drill toolpath in the Front plane.

### *Exercise 1: Roughing the Exterior*

- 1 Select the toolpath group **Exterior**.

Click the minus (-) in front of the Interior toolpath group to collapse the tree.



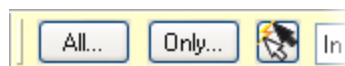
- 2 Click the **Position insert arrow after selected operation or after selected group** button.

The insertion arrow is placed in the Exterior group.

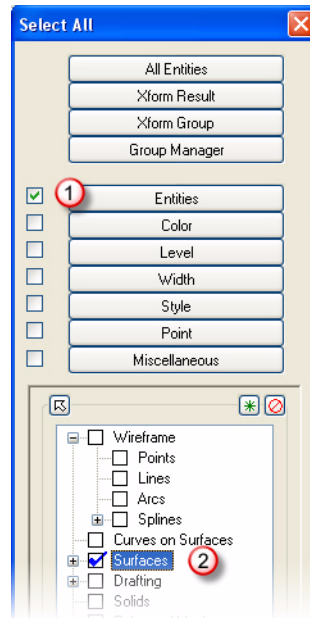


Moving the insertion arrow ensures new toolpaths are created in the Exterior toolpath group. Toolpath organization is an important ability as the number of operations grows.

- 3 Choose **Toolpaths, Surface High Speed**.
- 4 Click the **All** button on the General Selection ribbon bar when prompted to select drive surfaces.



- 5** Select the **Entities** and **Surfaces** checkboxes in the Select All dialog box as shown.

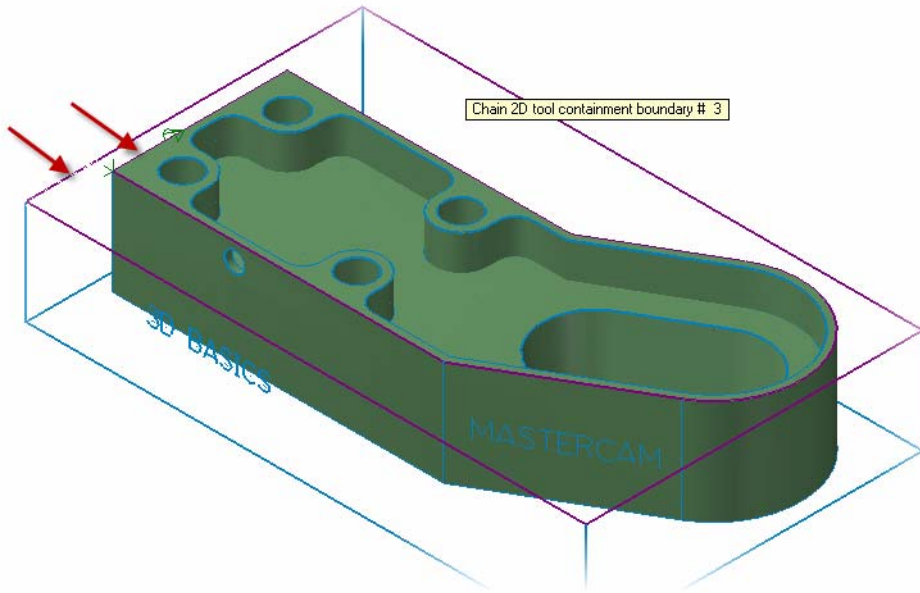


- 6** Click **OK** in the Select All dialog box.
- 7** Press **[Enter]** or click the **End Selection** button.
- 8** Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

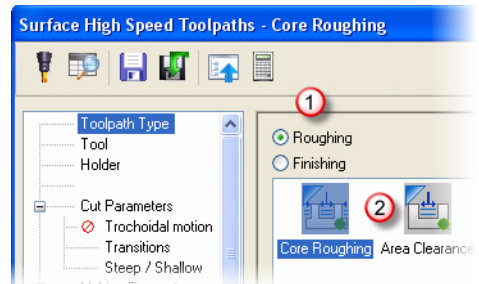
The Chaining dialog box opens.

- 9** Click the **C-plane** radio button.

- 10 Select two chains as shown to be used as containment boundaries. The chains lie on the top of the part and the top of the stock boundary created in the Toolpath Setup lesson.

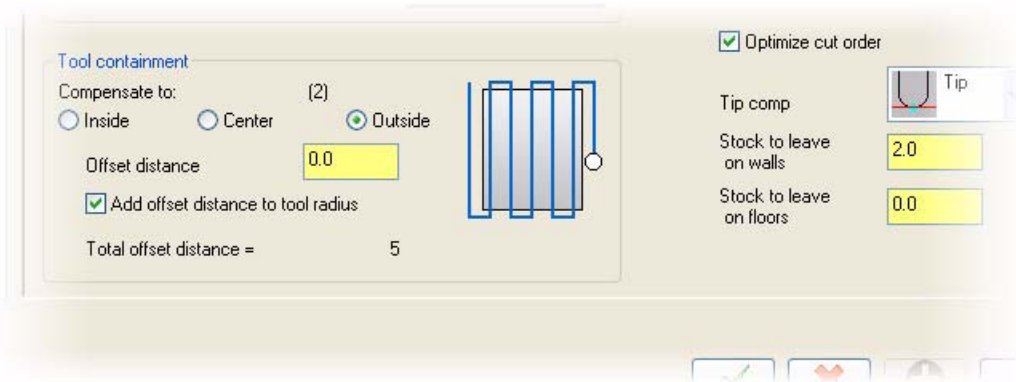


- 11 Click **OK** in the Chaining and the Toolpath/surface selection dialog boxes.  
The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.
- 12 Select the Toolpath Type page, and then **Roughing** and **Core Roughing**.



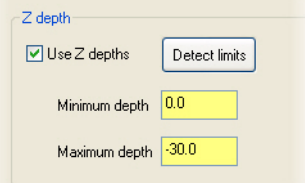
- 13 Select **Tool** from the Tree View pane.
- 14 Select the 10mm bull endmill with the 2mm corner radius showing in the tool list.

- 15** Select **Cut Parameters** from the Tree View pane. Set the tool containment and stock to leave values as indicated.



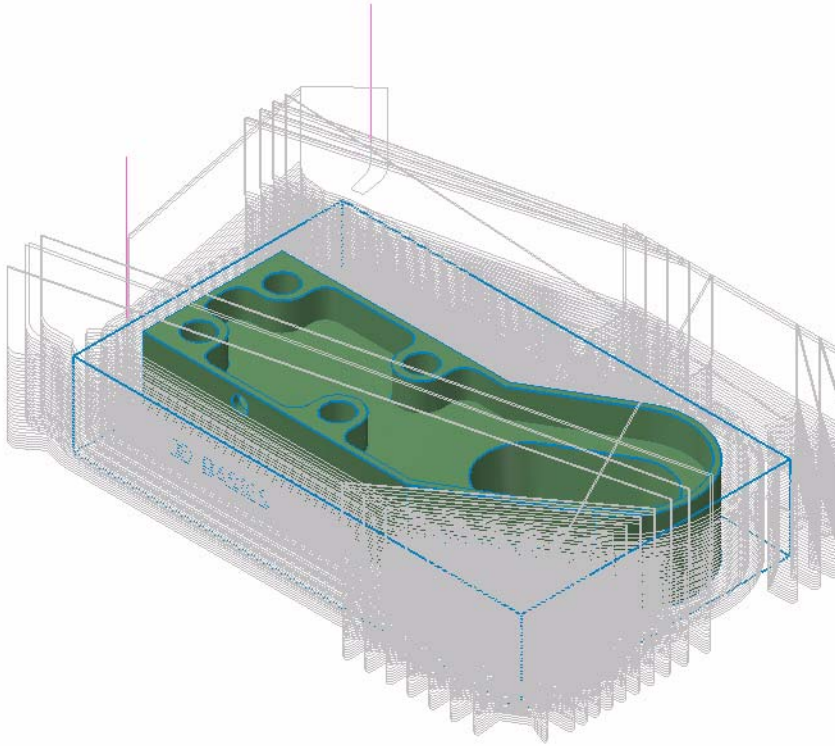
- 16** Select **Steep/Shallow** from the Tree View pane. Set the parameters as shown.

Click the plus [+] in front of **Cut Parameters** if necessary to expand the tree.





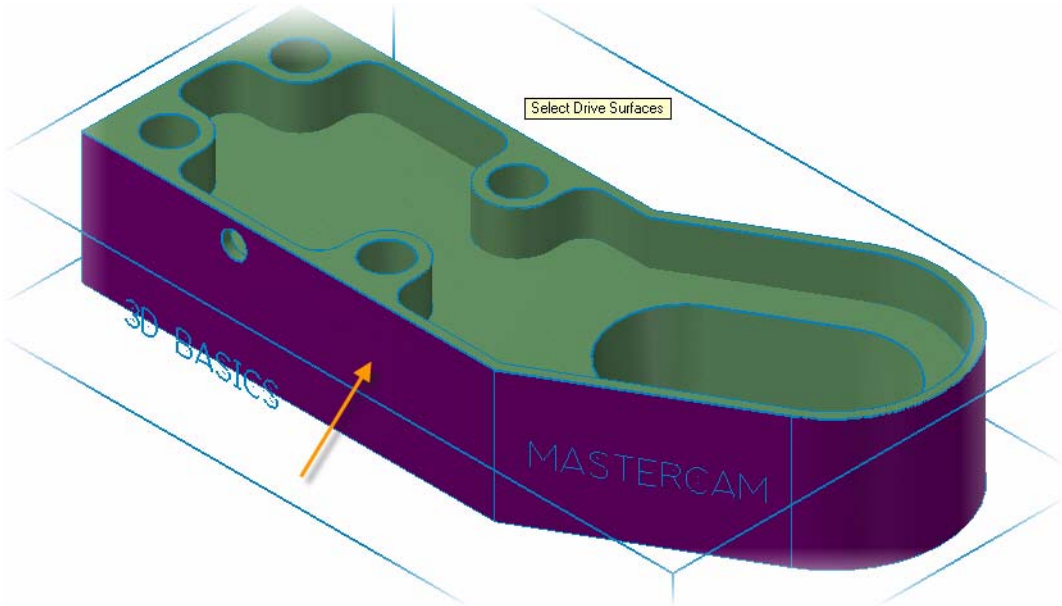
- 17** Click **OK** in the Surface High Speed Toolpath dialog box to generate the core roughing toolpath.



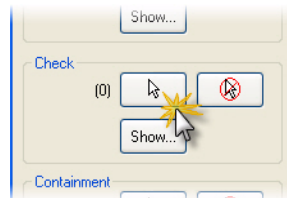
- 18** Turn off the toolpath display for the operation.
- 19** Save your part file.

## Exercise 2: Finishing the Exterior

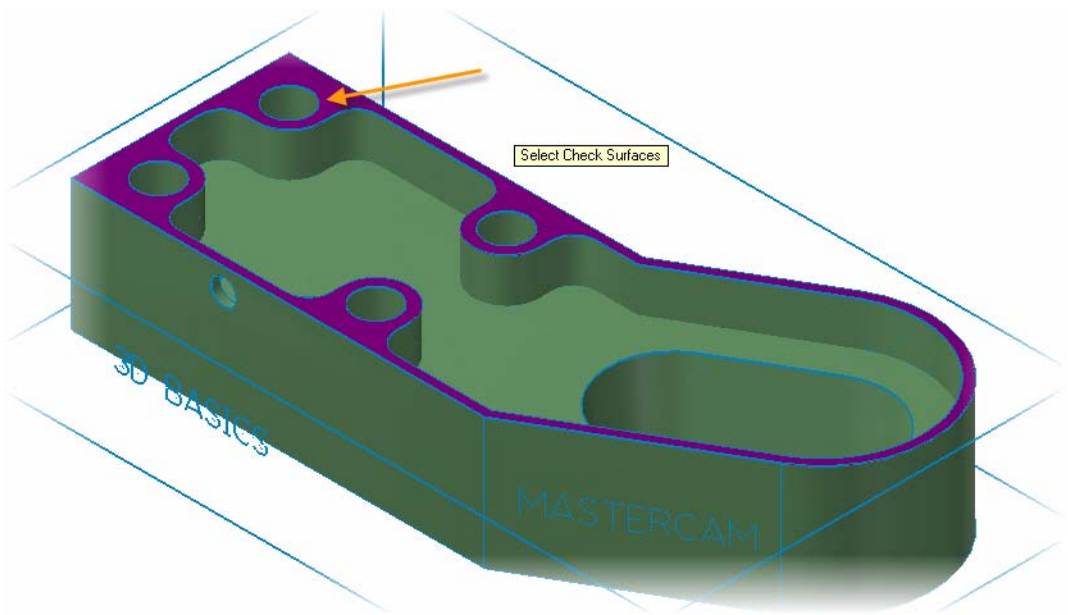
- 1 Choose **Toolpaths, Surface Finish, Contour**.
- 2 Follow the prompt to select drive surfaces as shown.



- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 4 Click the **Select** button in the Check Surfaces area of the Toolpath/Surface selection dialog box.



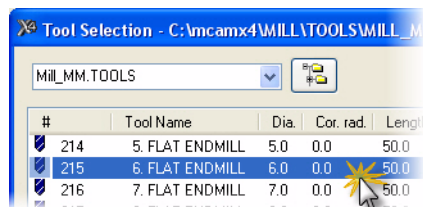
- 5 Follow the prompt to select check surfaces as shown.



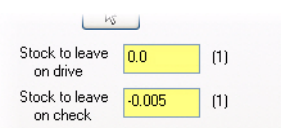
- 6 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 7 Click **OK** on the Toolpath/Surface selection dialog box.
- 8 Click the **Select library tool** button.

Select library tool...

- 9 Select the 6mm flat endmill and click **OK**.



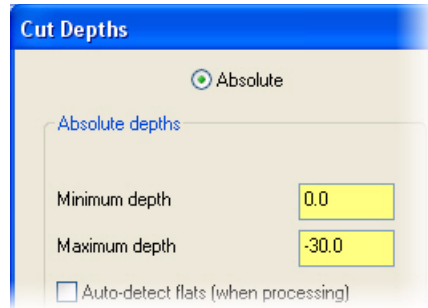
- 10 Click the **Surface parameters** tab and set the stock to leave values as shown.



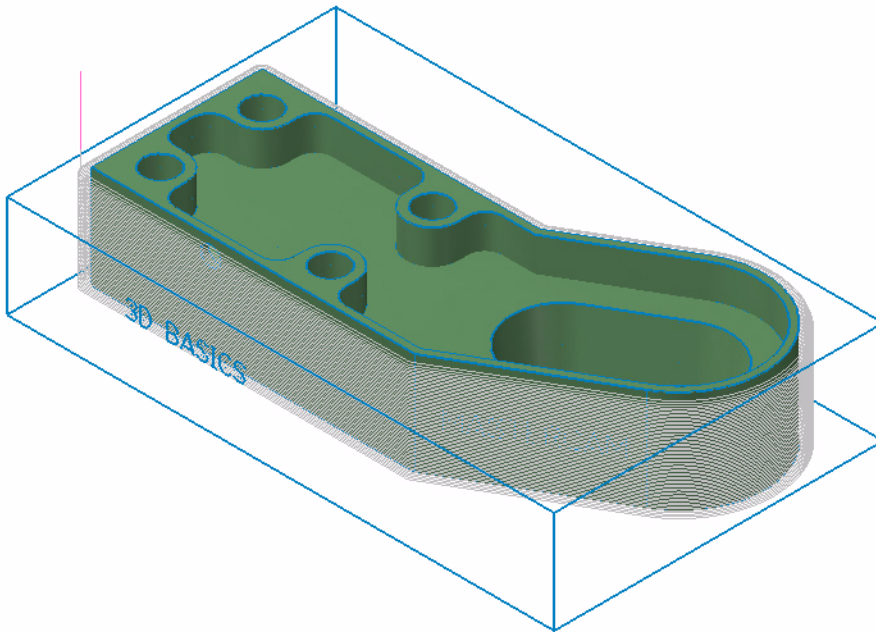
- 11 Click the **Finish contour parameters** tab.
- 12 Click the **Total tolerance** button.
- 13 Move the slider for Surface quality to the Best setting and click **OK**.

**14** Click the **Cut Depths** button.

**15** Set the parameters as shown and click **OK**.



**16** Click **OK** to generate the surface finish contour toolpath.



**17** Turn off the toolpath display for the operation.

**18** Save your part file.

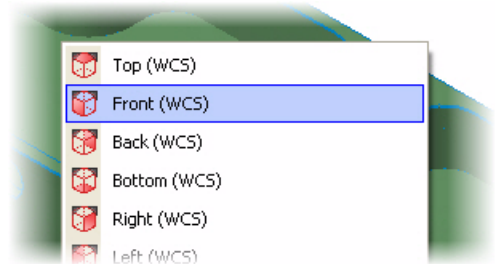
## Exercise 3: Drilling a hole from the Front Plane

- 1 Click the **Planes** button located on the Status bar at the bottom of the screen.

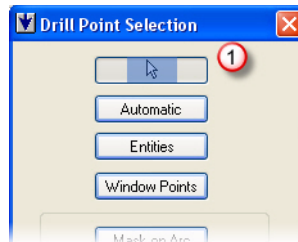
Planes

- 2 Select **Front (WCS)** from the list.

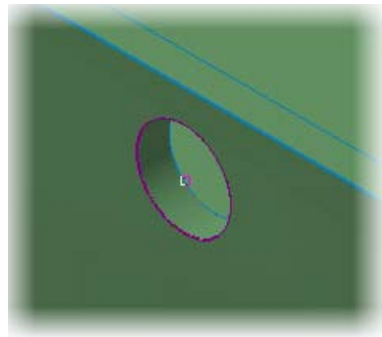
Planes selection sets your construction and tool plane.



- 3 Choose **Toolpaths, Drill**.
- 4 Use the default option for **Select drill point position in the graphics screen** in the Drill Point Selection dialog box.



- 5 Select the arc center as indicated.

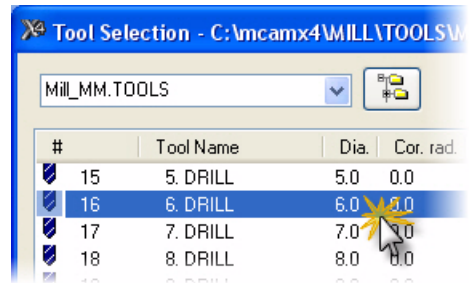


- 6 Click **OK** in the Drill Point Selection dialog box.

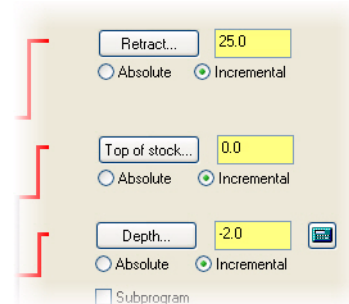
The 2D Toolpaths dialog box opens to the Toolpath Type page. Drill is selected as the toolpath type.

- 7 Select **Tool** from the Tree View pane.
- 8 Click the **Select library tool** button.

- 9 Select the 6mm drill and click **OK**.

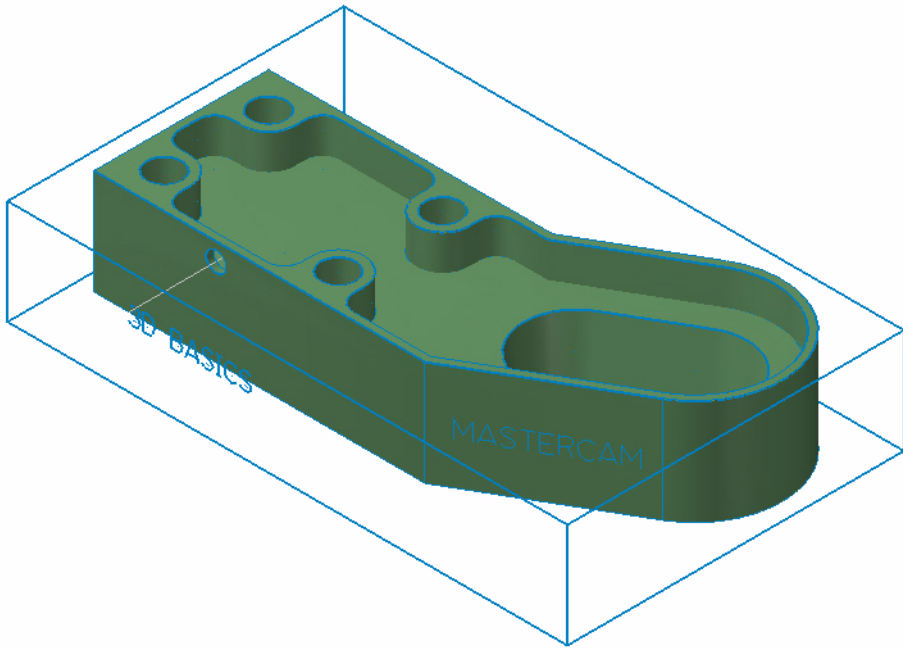


- 10 Select **Linking Parameters** from the Tree View pane and enter values as shown.



- 11 Select **Tip Comp** from the Tree View pane. Click the **Tip Comp** checkbox to activate this feature. Use default values for the tip comp parameters.

- 12** Click **OK** to generate the drill toolpath.



- 13** Turn off the toolpath display for the operation.
- 14** Save your part file.





## LESSON 5

# Adding Details

The basic part has been machined to the desired quality. Remaining details may now be added to the part such as engraved numbers or letters, tapping of holes, or any other detail work not included in the previous operations. This lesson covers two methods for adding lettering to your part.

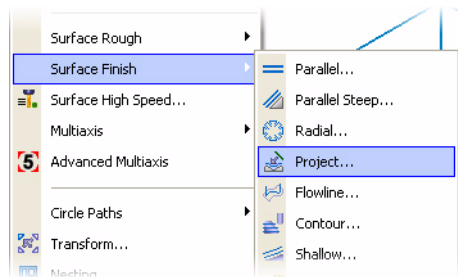
### *Lesson Goals*

- Create lettering using a Surface Finish Project toolpath in the front plane.
- Create lettering using a Contour toolpath in a custom plane.

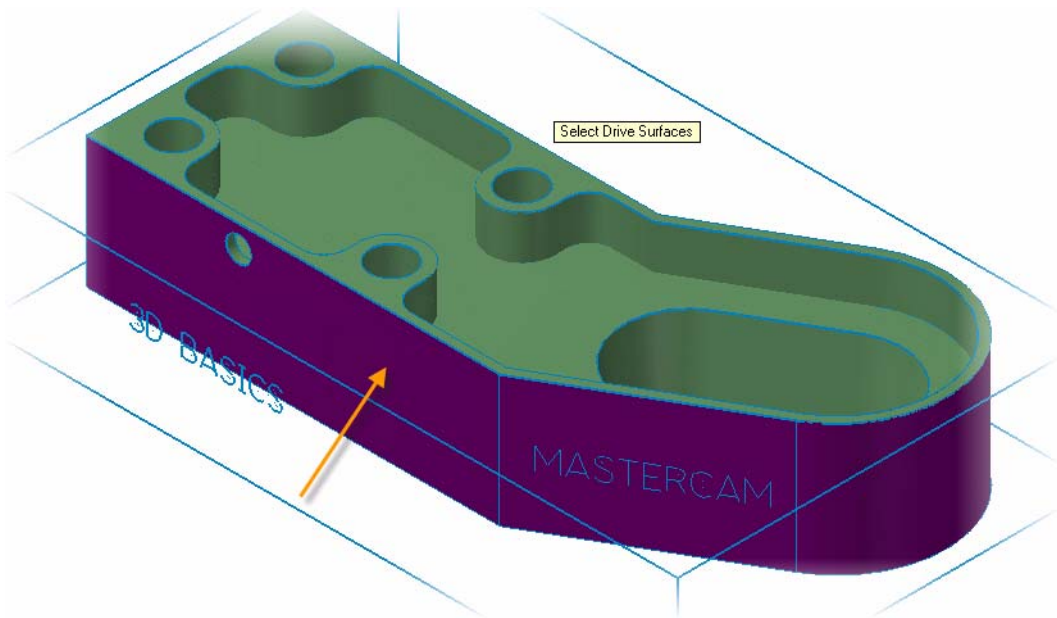
### *Exercise 1: Lettering with a Project Toolpath*

- 1 Choose **Toolpaths, Surface Finish, Project**.

The tool and construction planes should be set to Front from the previous exercise. If not, reset them to Front.



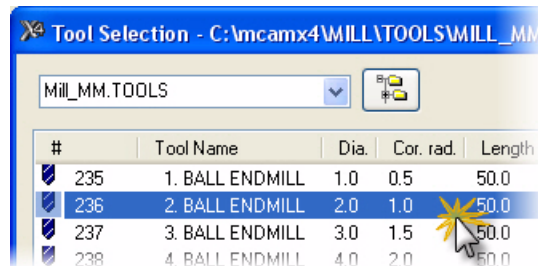
- 2 Follow the prompt to select drive surfaces as shown.



- 3 Press **[Enter]** or click the **End Selection** button.
- 4 Click **OK** from the Toolpath/Surface selection dialog box.
- The Surface Finish Project dialog box opens to the toolpath parameters tab.
- 5 Click the **Select library tool** button.

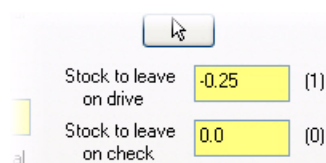
Select library tool...

- 6 Select the 2mm ball endmill and click **OK**.



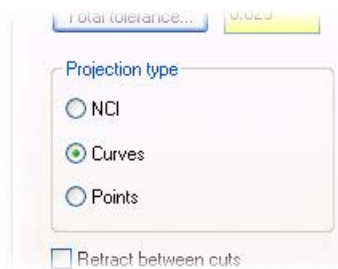
- 7 Click the **Surface parameters** tab and set the stock to leave values as shown.

A negative stock to leave value creates the toolpath below the surface. The letters become engraved into the part.

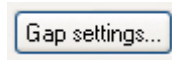


- 8 Click the **Finish project parameters** tab.

- 9 Set the Projection type to **Curves**.

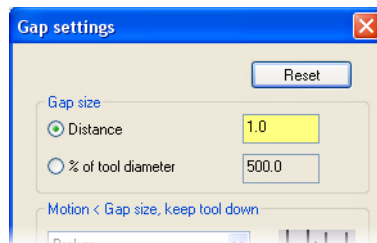


- 10 Click the **Total tolerance** button.
- 11 Move the slider for Surface quality to the Best setting and click **OK**.
- 12 Click the **Gap Settings** button.



- 13 Adjust the **Distance** value in Gap settings as shown.

Only the value preceded by the radio button selection will be used in the toolpath calculation.



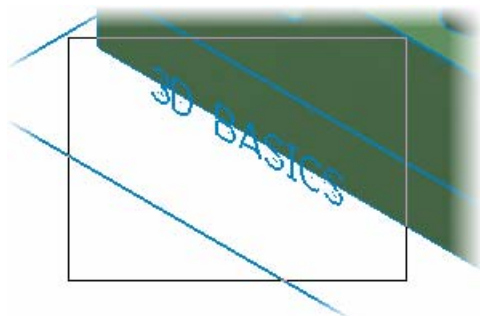
- 14 Click **OK** in the Gap settings dialog.
- 15 Click **OK** from the Surface Finish Project dialog.

The Chaining dialog box opens.

- 16 Set the chaining mode to window by clicking the **Window** button.

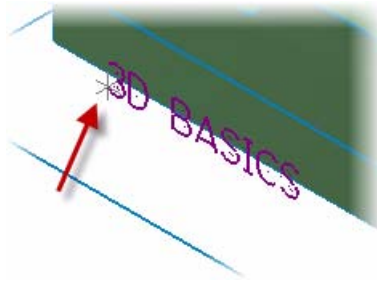


- 17 To select the curves to be used in the toolpath, click and hold the mouse button above and to the left of the text.
- 18 Drag your mouse down and to the right to draw a selection rectangle.

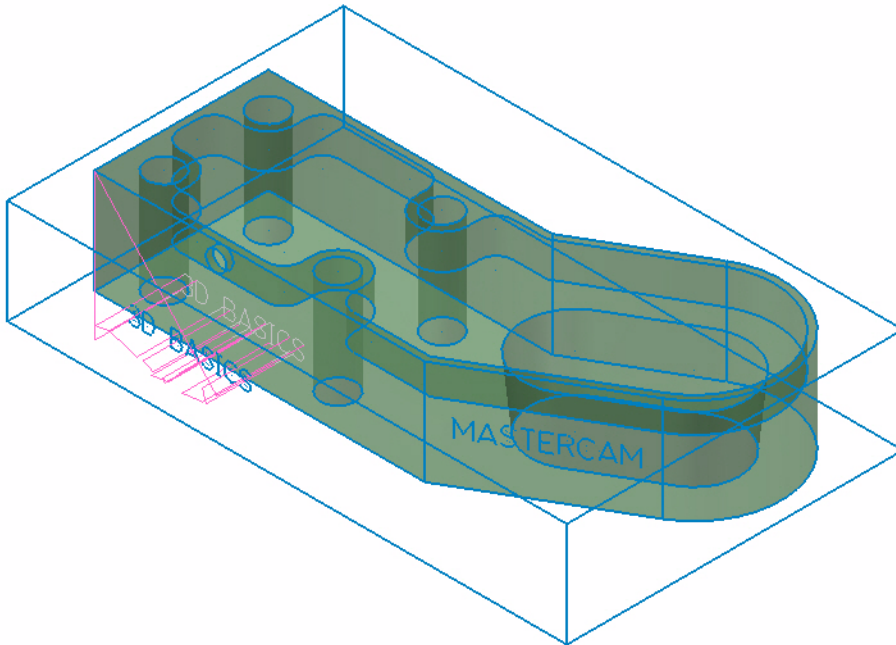


- 19 Click again to set the lower right corner of the rectangle and complete the window.

- 20** Click the lower left endpoint of the text when prompted to **Sketch approximate start point**.



- 21** Click **OK** to generate the surface finish project toolpath.



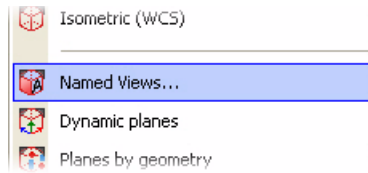
- 22** Turn off the toolpath display for the operation.
- 23** Save your part file.

## Exercise 2: Lettering with a Contour Toolpath

- 1 Click the **Planes** button on the Status bar.

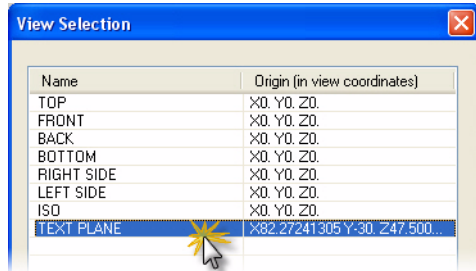
- 2 Select **Named Views** from the list.

The View Selection dialog opens.



- 3 Select **Text Plane** from the list of available planes. Click **OK** to set the new plane.

Review *Basic 3D Design* to see a procedure for creating named planes.



- 4 Choose **Toolpaths, Contour**.

- 5 Set the chaining mode to window by clicking the **Window** button.

- 6 Window-select the Mastercam letters that lie in the Text Plane.

- 7 Click the lower left endpoint of the text when prompted to **Sketch approximate start point**.



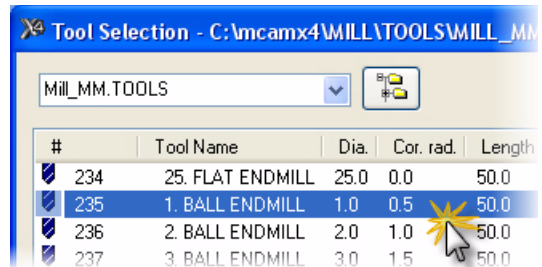
- 8 Click **OK** from the Chaining dialog box.

The 2D Toolpath dialog box opens to the Toolpath Type page.

- 9 Select **Tool** from the Tree View pane.

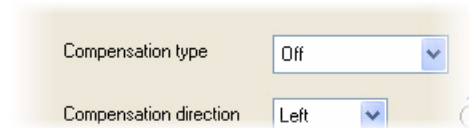
- 10 Click the **Select library tool** button.

- 11 Select the 1mm ball endmill and click **OK**.



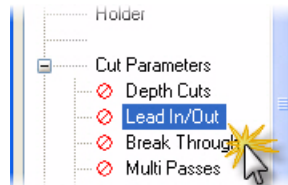
- 12 Select **Cut Parameters** from the Tree View pane.

- 13 Set Compensation type to **Off**.

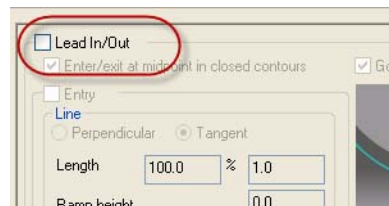


- 14 Select **Lead In/Out** from the Tree View pane.

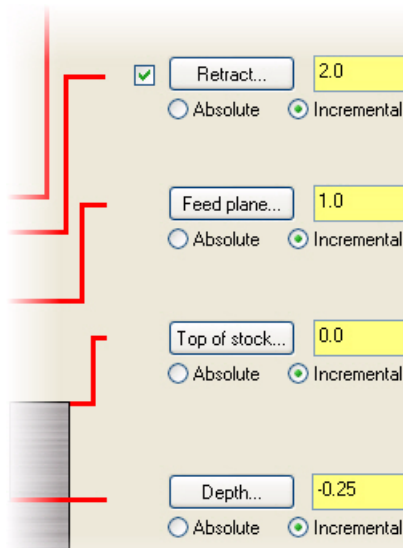
Expand Cut Parameters by clicking the [+], if necessary.



- 15 Uncheck lead in/out moves.



- 16** Select **Linking Parameters** from the Tree View pane and set the parameters as shown.

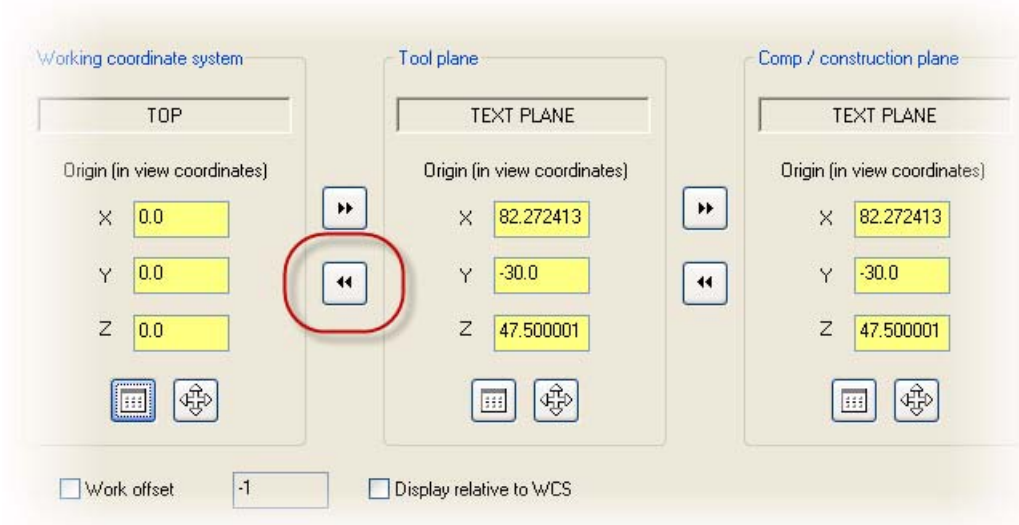


- 17** Select **Planes (WCS)** from the Tree View pane.

The Planes (WCS) page is displayed.

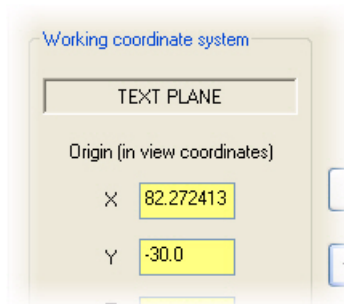


**18** Click the **Copy to WCS** button highlighted below.



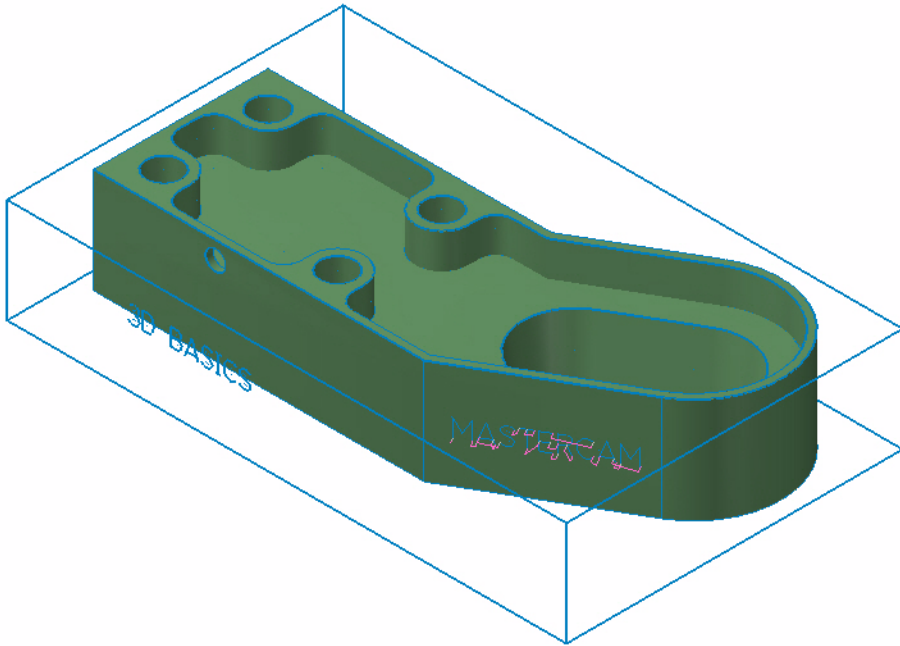
**19** The resulting working coordinate system is as shown.

Click the **Help** button for details on working with WCS.





- 20** Click **OK** from the 2D Toolpaths dialog box to generate the contour toolpath.



- 21** Turn off the toolpath display for the operation.
- 22** Save your part file.



## LESSON 6

# Backplot/Verify Your Toolpaths

Now that all toolpaths are complete, it is time to backplot and verify your operations. This process allows you to review the tool motion before any cutting takes place on the actual machine. This lesson guides you through the backplot and verify functions.

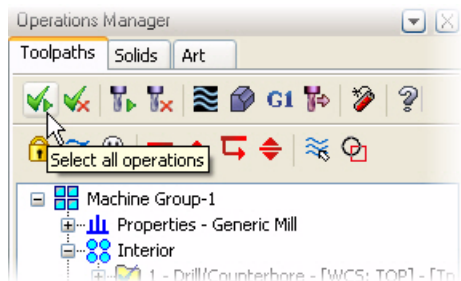
### *Lesson Goals*

- Backplot all toolpaths.
- Verify all toolpaths.

### *Exercise 1: Backplotting All Toolpaths*

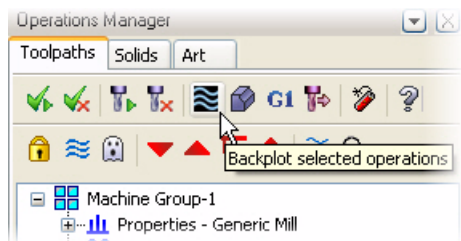
- 1 In the Toolpath Manager, click the **Select all operations** button.

All operations should now have a green check on them.



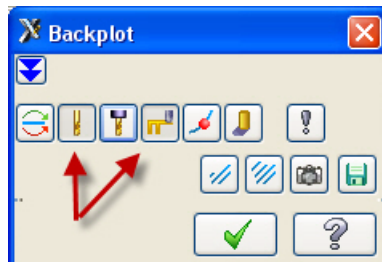
- 2 Click the **Backplot selected operations** button in the Toolpath Manager.

The Backplot dialog box and Backplot VCR bar open.



- 3 In the Backplot dialog box, select the **Display tool** and **Display rapid moves** buttons.

These options will display a simulation of the tool as well as the rapid motion.



**TIP:** To further customize your backplot display, choose other buttons on the Backplot dialog box. For example, choose the **Options** button to open the Backplot Options dialog box. This dialog box lets you set various backplot options such as tool display, holder display, and tool motion colors. Setting the option to Cleanup screen on Tool change is recommended with a large number of operations.

- 4 Use the buttons and sliders on the Backplot VCR bar to backplot the operations. The **Play** button (shown to the right) begins the backplot.



Click the **Help** button on the VCR bar for more information on each of the controls.



- 5 When finished, click **OK** on the backplot dialog box to exit the backplot function.

**TIP:** The backplot display is easily customizable. See the Mastercam Help for details on each of the buttons, fields, and display options in the Backplot and the Backplot Options dialog boxes.

## Exercise 2: Verifying All Toolpaths

- 1 In the Toolpath Manager, make sure all operations are selected and click the **Verify selected operations** button.

The Verify dialog box opens.



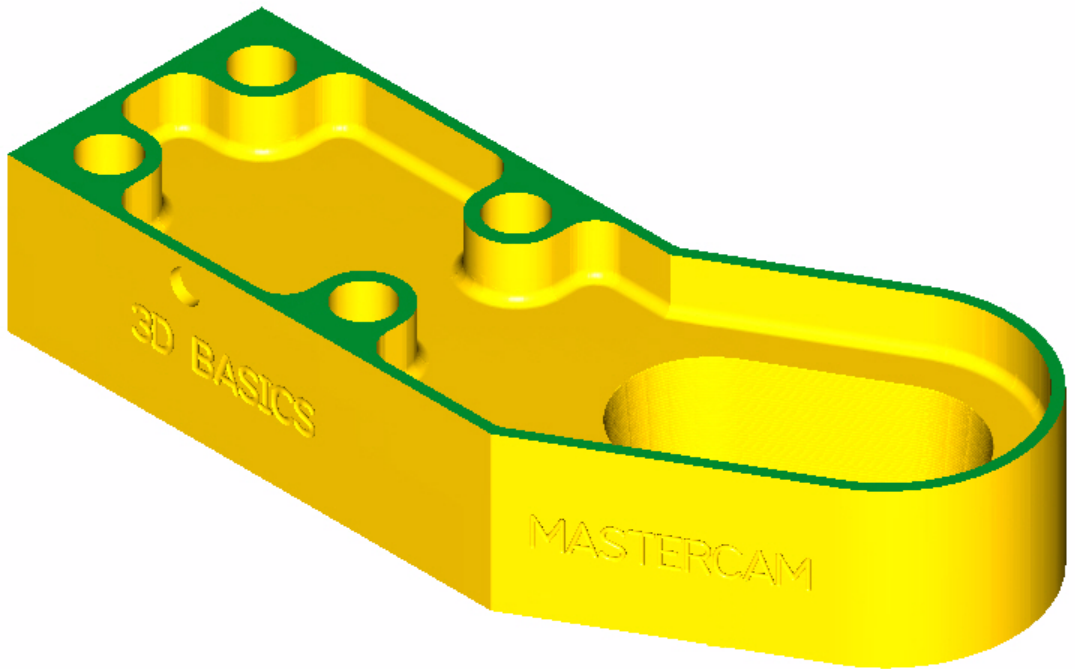
- 2 In the Verify dialog box, select the **Machine** button.



The part, stock, and toolpaths are simulated.

**TIP:** Use the buttons, fields, and controls in the Verify dialog box to customize and manage the toolpath verification process. Click the **Help** button in the dialog box for details.

When the verification process is complete, the part should look similar to the image shown here.





## LESSON 7

# Posting Toolpaths

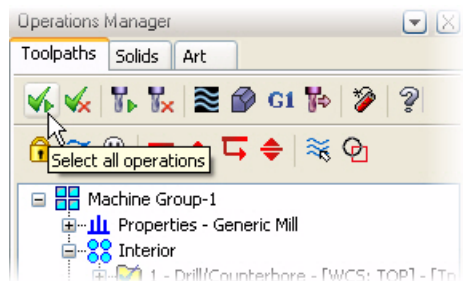
Operations must be run through a post processor in order to be usable by your machine. This process is commonly referred to as posting. This lesson will guide you through selecting and posting all of your operations. Communicating the posted code to your machine is the last step of the toolpathing process in Mastercam.

### *Lesson Goals*

- Post all toolpaths to create an NC file.

### *Exercise 1: Posting All Toolpath Operations*

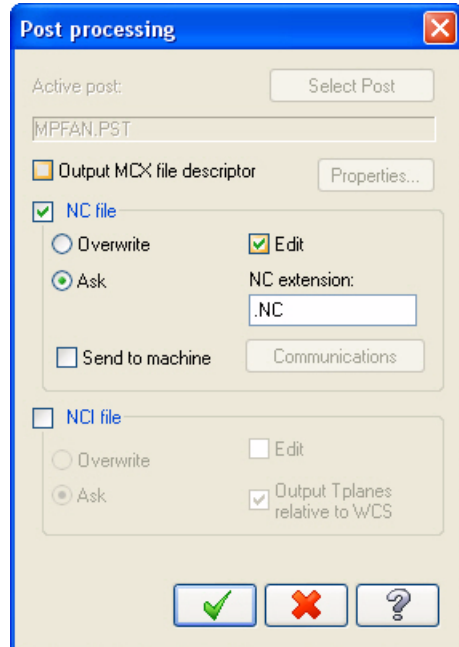
- 1 In the Toolpath Manager, click the **Select all operations** button.



- 2 Click the **Post selected operations** button.  
The Post processing dialog box opens.



3 Set the post processing options as shown.



4 Click **OK**.

The Save As dialog box opens.

5 Click **Save** to save the NC file in the default location with the recommended file name.

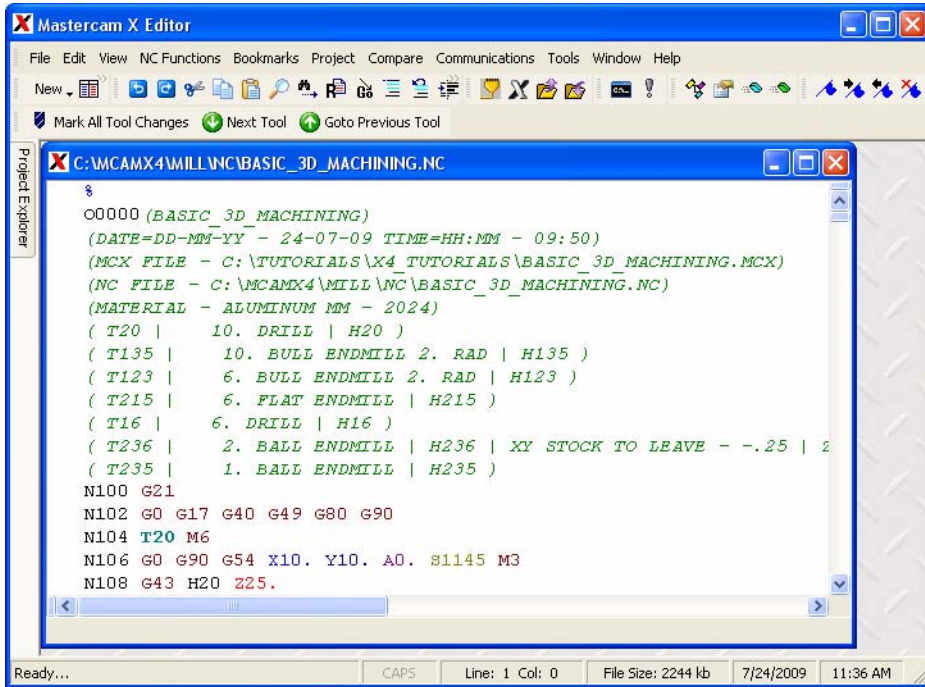
---

*Notes:*

- Posting may take several minutes. When it is complete, the NC code will display in your default editor/communicator. This tutorial uses Mastercam Editor to display the NC code.
  - Producing the correct NC code for your machine and application depends on properly configuring the machine definition, control definition, and .PST file. For detailed information on machine definitions, control, definitions, and posting, please see the following documentation supplied with Mastercam:
    - Mastercam Help
    - Mastercam X4 Reference Guide (choose Reference Guide from the Mastercam Help menu)
    - Mastercam NCI & Parameter Reference Guide (in the Documentation folder under your Mastercam installation folder)
-



- 6 Your chosen editor opens (in this case, Mastercam Editor), displaying the posted NC code as shown below.



```

Mastercam X Editor
File Edit View NC Functions Bookmarks Project Compare Communications Tools Window Help
New [Icons]
Mark All Tool Changes Next Tool Goto Previous Tool

Project Explorer
C:\MCAMX4\MILL\NC\BASIC_3D_MACHINING.NC

8
O0000 (BASIC_3D_MACHINING)
  (DATE=DD-MM-YY - 24-07-09 TIME=HH:MM - 09:50)
  (MCX FILE - C:\TUTORIALS\X4_TUTORIALS\BASIC_3D_MACHINING.MCX)
  (NC FILE - C:\MCAMX4\MILL\NC\BASIC_3D_MACHINING.NC)
  (MATERIAL - ALUMINUM MM - 2024)
  ( T20 | 10. DRILL | H20 )
  ( T135 | 10. BULL ENDMILL 2. RAD | H135 )
  ( T123 | 6. BULL ENDMILL 2. RAD | H123 )
  ( T215 | 6. FLAT ENDMILL | H215 )
  ( T16 | 6. DRILL | H16 )
  ( T236 | 2. BALL ENDMILL | H236 | XY STOCK TO LEAVE - -.25 | 2 )
  ( T235 | 1. BALL ENDMILL | H235 )
N100 G21
N102 G0 G17 G40 G49 G80 G90
N104 T20 M6
N106 G0 G90 G54 X10. Y10. A0. S1145 M3
N108 G43 H20 Z25.

Ready... CAPS Line: 1 Col: 0 File Size: 2244 kb 7/24/2009 11:36 AM

```

- 7 Scroll through the NC code to verify that each line of code meets your expectations. Edit and save as necessary.

## Sending NC Files to the Machine



**IMPORTANT:** This tutorial is based on the Mastercam Mill Default machine definition for training purposes only. It is not possible to provide a step-by-step procedure for sending the NC code to your machine control because machine setups are customizable and most likely different from the machine definition used here. Following is a general description of how the NC code is communicated to machines and their controls for machining.

After the NC file is reviewed, edited, and saved, you can set up your machine control to accept the NC file. This is done according to your machine and control manufacturer's procedures.

When the machine control is ready to receive the NC file, configure your preferred editor or communications program to communicate with your machine control. Refer to your communications program documentation for details.

Send the NC code to your machine control according to your machine and control manufacturer's documentation. Once you start the communication process, the send/receive data processing is mostly managed by your machine control.

Contact your local Mastercam Reseller for customized machine/control definitions, post (PST) files, and support.

---

## **Section 2**

# **Machining the Tool Holder**



## LESSON 1

# Toolpath Setup

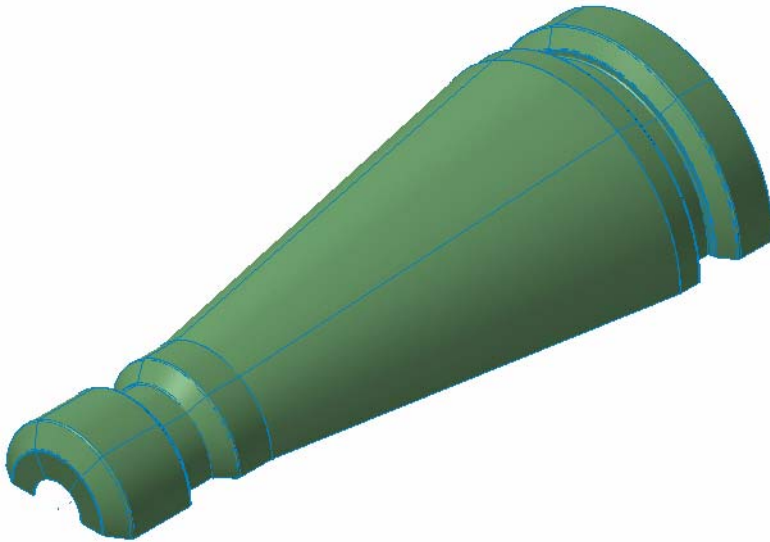
Before generating toolpaths for a part, you must select a machine definition. Defining stock creates a visual representation of the stock placed in your machine. This lesson covers these topics.

### *Lesson Goals*

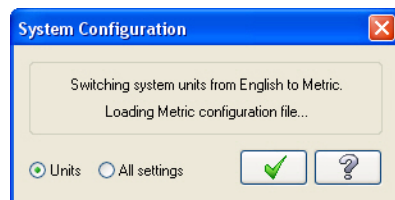
- Open a part file and assign a machine definition.
- Define stock to be used in machining the part.

### *Exercise 1: Assigning a Machine Definition*

- 1 Open the tutorial part file `Basic_3D_Machining_Part2_Start.MCX`, which was provided with the tutorial.

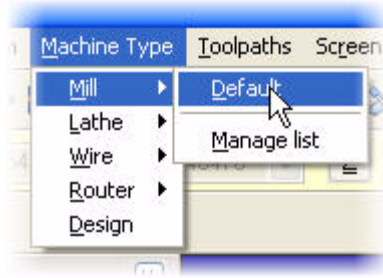


- 2 Click **OK** if prompted to switch to a metric configuration.



- 3** Choose **Machine Type, Mill, Default** to open the default Mill machine definition.

In Mastercam, you select a machine definition before creating any toolpaths. The machine definition is a model of your machine tool's capabilities and features. It acts like a template for setting up machining jobs.




---

*Note: Parts that have previously been saved with a machine definition automatically load the associated machine definition.*

---

- 4** Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

## Exercise 2: Setting Stock for Machining

- 1** In the Toolpath Manager, select **Stock setup**. If necessary, click the [+] next to Properties to expand the list.

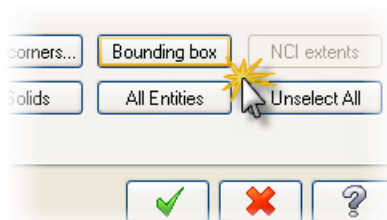
The Machine Group Properties dialog box opens to the Stock Setup tab.



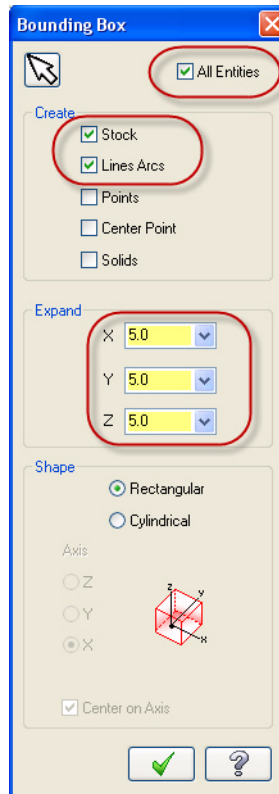
- 2** Click **Bounding box**.

The Bounding Box dialog box opens.

Bounding box is a quick and convenient method for creating stock around the outer boundary of your geometry.



- 3 Set the options and parameters as shown, and then click **OK**.

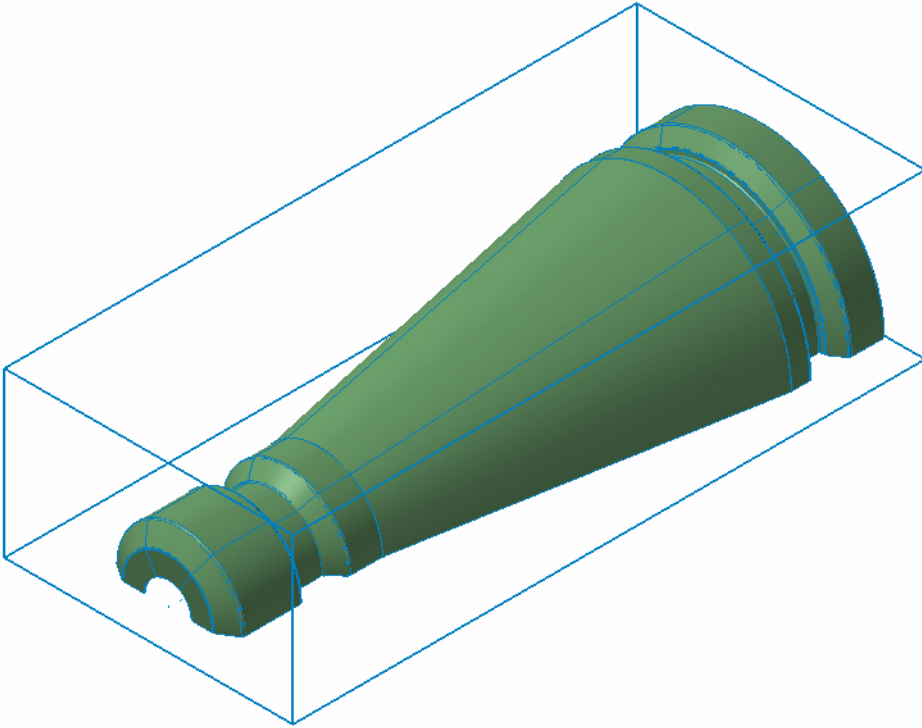


- 4 Click **OK** in the Machine Group Properties dialog box.



- 5 Press **[Alt+S]** to shade the geometry, if necessary.

- 6 Press **[Alt+F1]** to fit the geometry to the screen.



- 7 Choose **File, Save** or click the **Save** button to save the part with the machine definition and defined stock.





## LESSON 2

# Roughing the Part

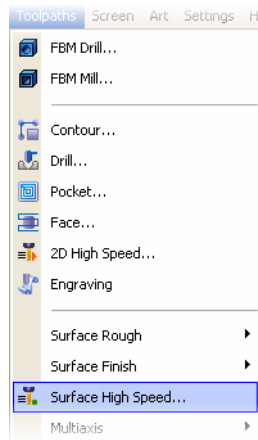
The first toolpaths for a part typically involve removing large amounts of material. This is referred to as roughing the part. This lesson guides you through several roughing toolpaths. These include using a core roughing operation and a leftover operation.

### *Lesson Goals*

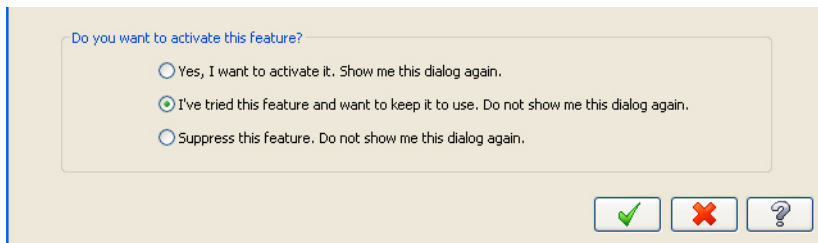
- Create a core roughing toolpath to efficiently remove material.
- Create a leftover toolpath.

### *Exercise 1: Roughing the Part*

- 1 Choose **Toolpaths, Surface High Speed**.



- 2 If the **New 3D Advanced Toolpath Refinement Feature!** dialog box opens, select the option to eliminate this dialog box and make refinement available for use. Click **OK** to close the dialog box.



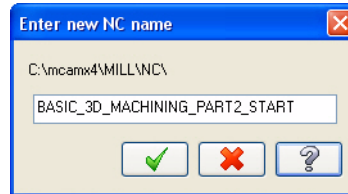
---

*Note: This dialog box introduces the 3D Advanced Toolpath Refinement feature. Use this feature to fine-tune your toolpath motion.*

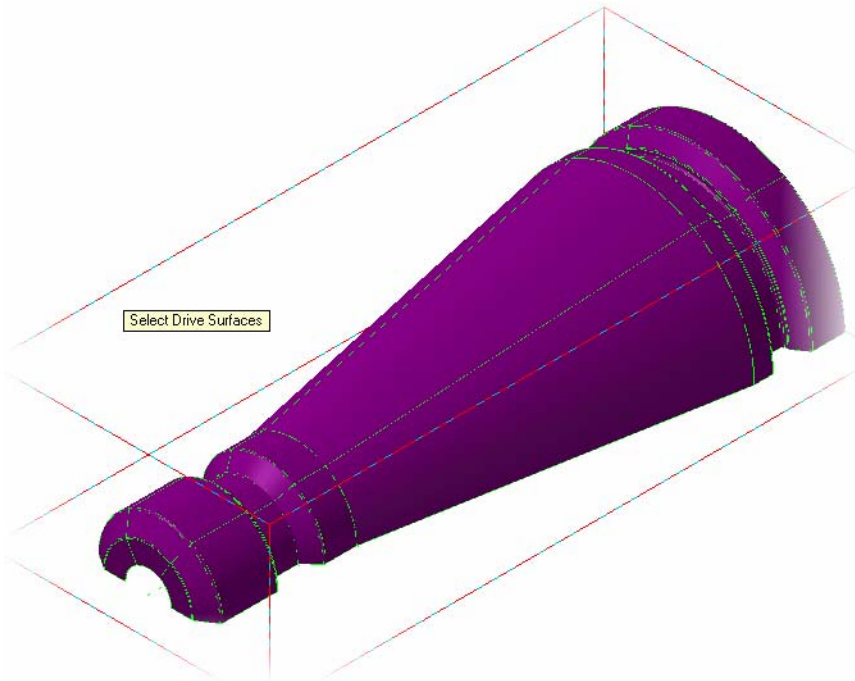
---

- 3** Click **OK** when prompted to enter new NC name.

The name displayed will be the name you chose to save the file under. You may change the NC file name now if necessary.



- 4** Press **[Alt+S]** to shade the geometry, if necessary.
- 5** Follow the prompt to select drive surfaces as shown.

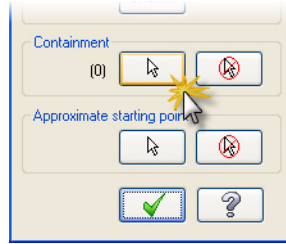


- 6** Press **[Enter]** or click the **End Selection** button to accept the selection.



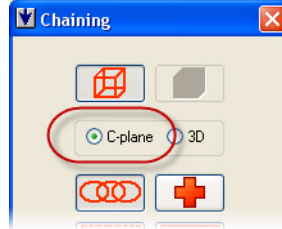
- 7 Click the **Select** button in the Containment area of the Toolpath/Surface selection dialog box.

The Chaining dialog box opens.

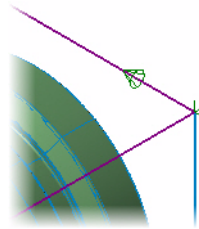


- 8 Click the **C-plane** radio button.

C-plane selection limits chaining to entities that are parallel to the current construction plane.



- 9 Select the chain on top of the stock boundary.

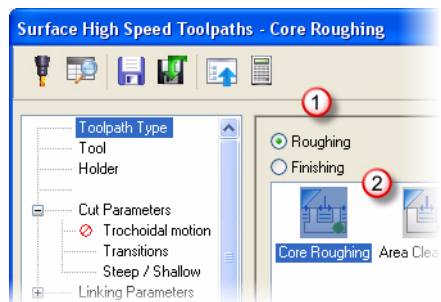


- 10 Click **OK** on the Chaining dialog box to return to the Toolpath/Surface selection dialog box.

- 11 Click **OK** on the Toolpath/surface selection dialog box.

The Surface High Speed toolpaths dialog box opens on the Toolpath Type page.

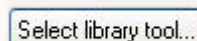
- 12 Select **Roughing** and **Core Roughing** on the Toolpath Type page.



- 13 Select **Tool** from the Tree View pane.

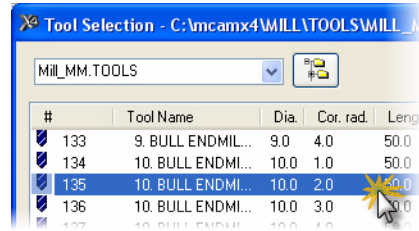
- 14 Click the **Select library tool** button.

The default metric tool library opens.

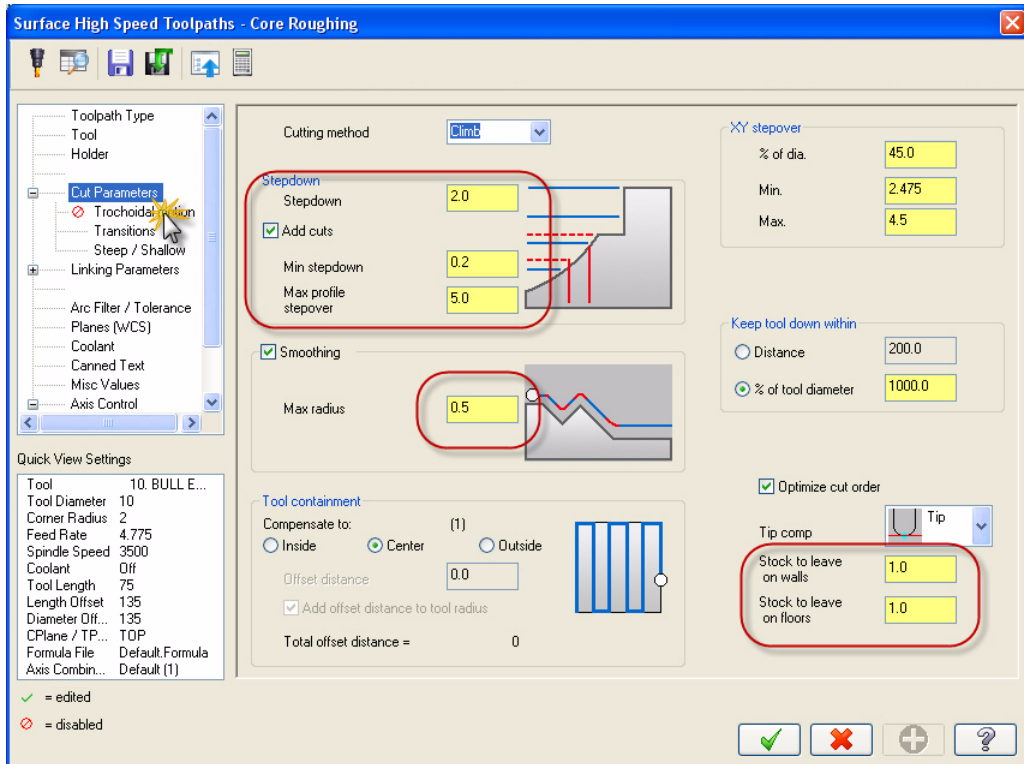


- 15 Select the 10mm bull endmill with 2mm corner radius and click **OK**.

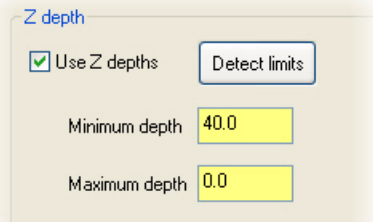
**TIP:** Adjust the Filter options on the right of the dialog box to limit the types of tools displayed.



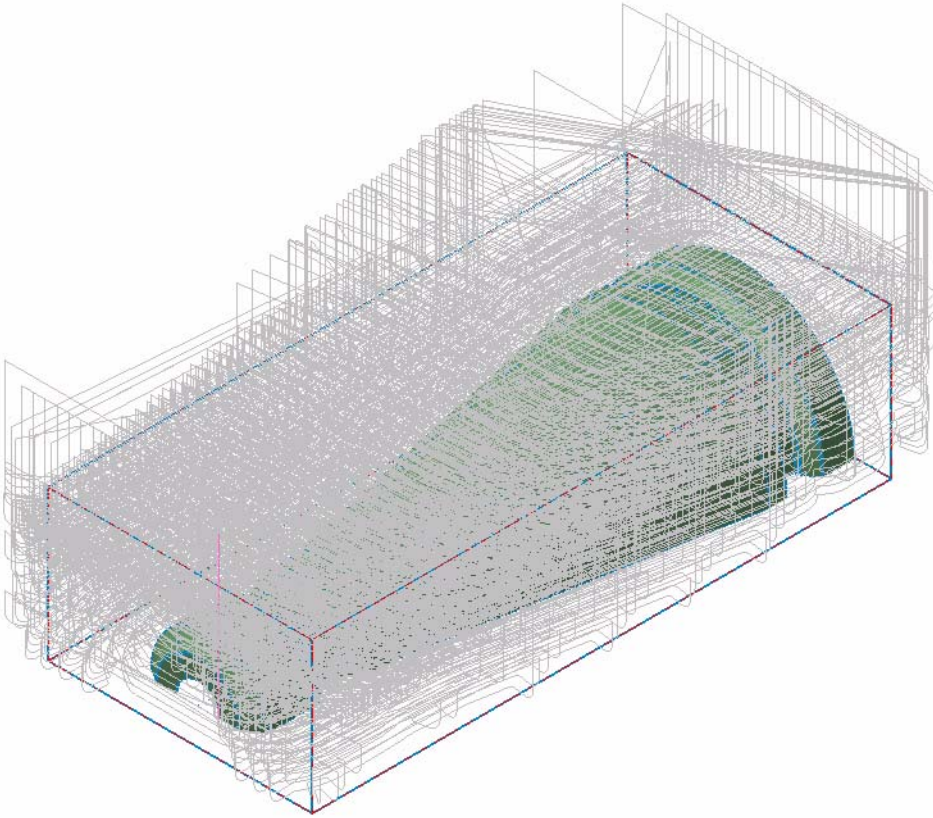
- 16 Select **Cut Parameters** from the Tree View pane. Set the parameters as indicated.



- 17 Select **Steep / Shallow** from the Tree View pane. Set the parameters as indicated.



- 18** Click **OK** to generate the core roughing toolpath. It might take a few moments to generate.



- 19** Click the **Toggle toolpath display on selected operations** button. If necessary, select the Core Roughing operation.

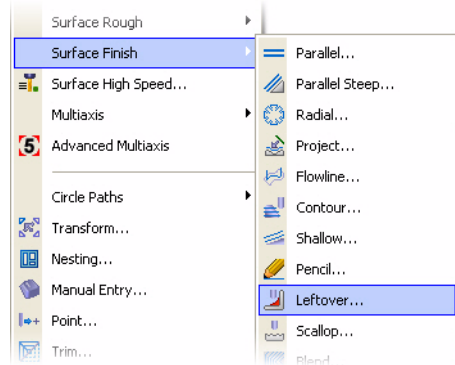
The toolpath display for the Core Roughing operation is turned off. Perform this step after the creation of each operation for clarity in selecting geometry for subsequent steps.



- 20** Save your part file.

## Exercise 2: Creating a Leftover Toolpath

- 1 Choose **Toolpaths, Surface Finish, Leftover**.



- 2 Click the **Select last** button on the General Selection ribbon bar when prompted for drive surfaces.



The surface(s) used in the previous operation are selected.

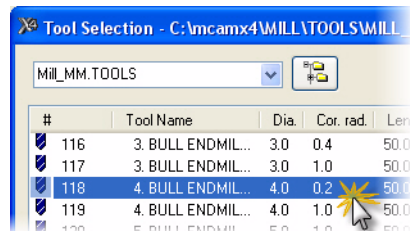
- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 4 Click **OK** on the Toolpath/surface selection dialog box.

The Surface Finish Leftover dialog box opens to the Toolpath parameters tab.

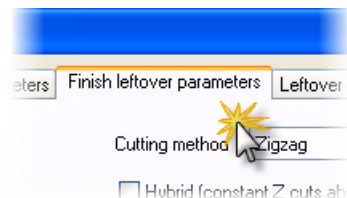
- 5 Click the **Select library tool** button.

The default metric tool library opens.

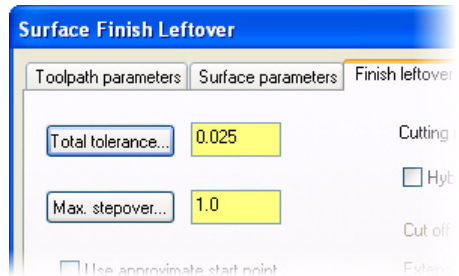
- 6 Select the 4mm bull endmill with 0.2mm corner radius and click **OK**.



- 7 Click the **Surface parameters** tab. Set the stock to leave on drive to 1.0.
- 8 Click the **Finish leftover parameters** tab.

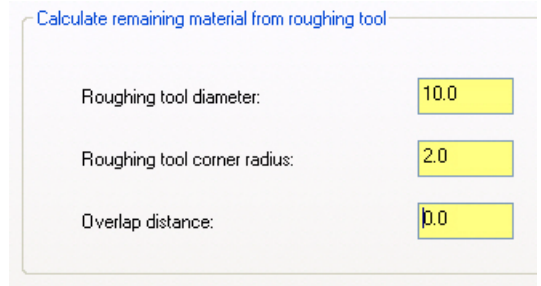


- 9 Set the parameters as shown.



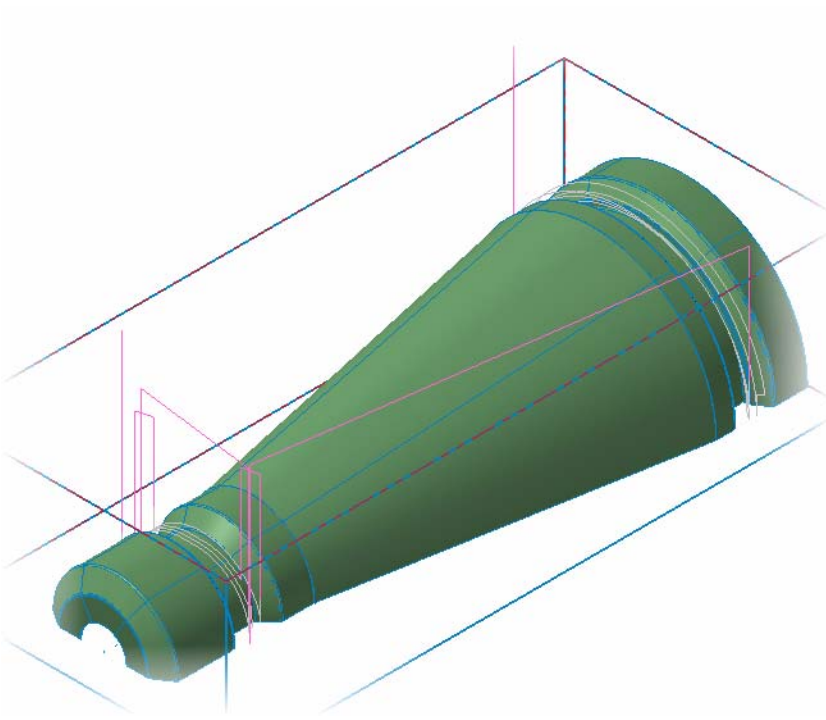
The screenshot shows the 'Surface Finish Leftover' dialog box with three tabs: 'Toolpath parameters', 'Surface parameters', and 'Finish leftover'. The 'Surface parameters' tab is selected. It contains two input fields: 'Total tolerance...' with a value of 0.025 and 'Max. stepover...' with a value of 1.0. There are also checkboxes for 'Use approximate start point' and 'Hybrid', and a 'Cut off' button.

- 10 Click the **Leftover material parameters** tab. Enter the roughing tool information as shown.



The screenshot shows the 'Surface Finish Leftover' dialog box with the 'Finish leftover' tab selected. It contains a section titled 'Calculate remaining material from roughing tool' with three input fields: 'Roughing tool diameter:' with a value of 10.0, 'Roughing tool corner radius:' with a value of 2.0, and 'Overlap distance:' with a value of 0.0.

- 11 Click **OK** to generate the surface finish leftover toolpath. It might take a few moments to generate.



---

*Note: Additional toolpath calculation lines may be displayed after generating the toolpath. Your screen may look slightly different from the image above.*

---

- 12 Turn off the toolpath display for the operation.
- 13 Save your part file.



## LESSON 3

# Semi-Finishing the Part

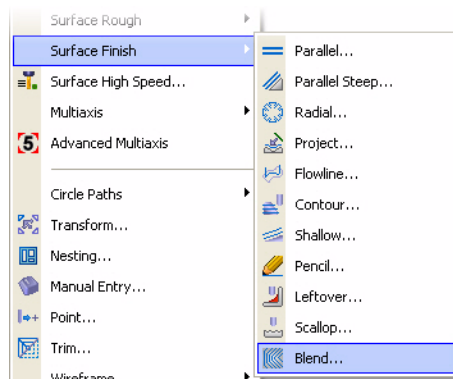
After roughing a part, it may be necessary to perform semi-finishing operations. These operations remove material that remains from roughing while leaving material for finishing operations. They are an intermediate step in the process to a finished part. In general, the tolerance value, the stepover, and the stock to leave separate the finishing operations from the semi-finishing operations. This lesson guides you through surface finish blend, surface finish parallel steep, and copying and modifying an operation.

### *Lesson Goals*

- Create several finishing operations, including parallel steep and blend.
- Set parameters to create semi-finishing operations.
- Copy an operation and change parameters.

### *Exercise 1: Creating a Surface Finish Blend*

- 1 Choose **Toolpaths, Surface Finish, Blend**.



- 2 Click the **Select last** button when prompted for drive surfaces.
- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.

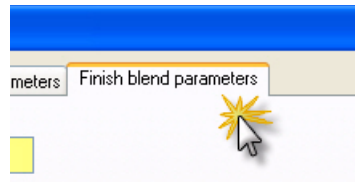


- 4 Click **OK** on the Toolpath/surface selection dialog box.

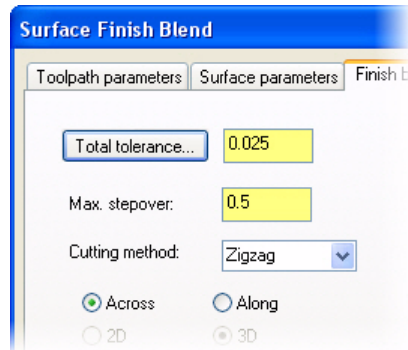
The Surface Finish Blend dialog box opens on the Toolpath parameters tab.

- 5 Select the 4mm bull endmill with the 0.2mm corner radius showing in the tool list.
- 6 Click the **Surface parameters** tab. Set the stock to leave on drive to 0.25.

- 7 Click the **Finish blend parameters** tab.

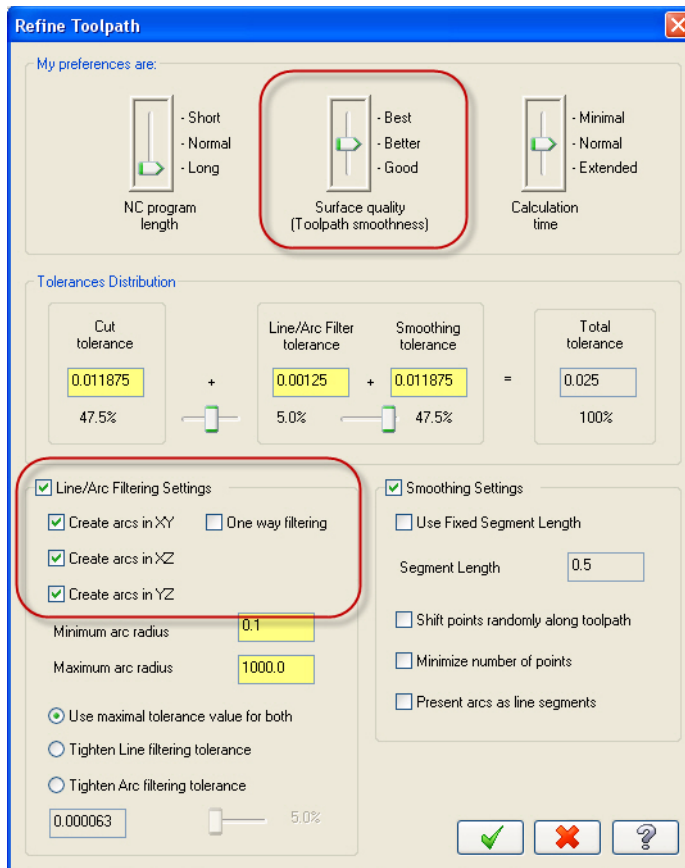


- 8 Set the parameters as shown.



- 9 Click the **Total tolerance** button to open the Refine Toolpath dialog box.

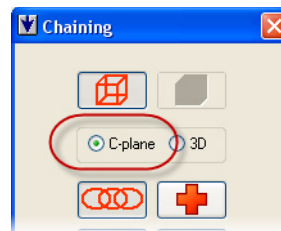
- 10 Set the parameters as shown and click **OK**.



- 11 Click **OK** from the Surface Finish Blend dialog box.

The Chaining dialog box opens.

- 12 Click the **C-plane** radio button.

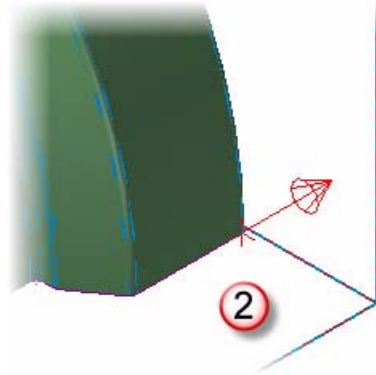
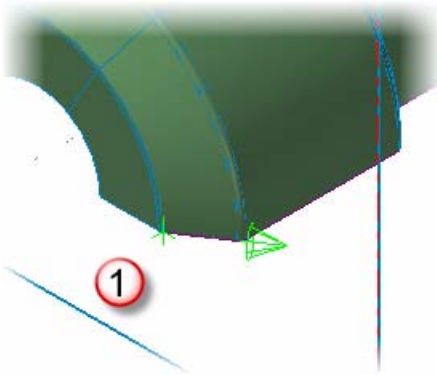


- 13 Click the **Partial** button.

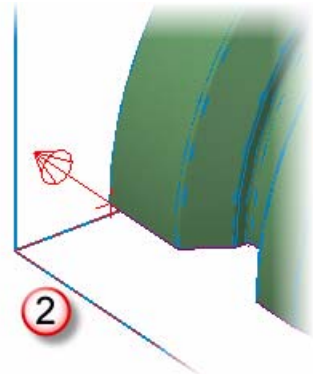
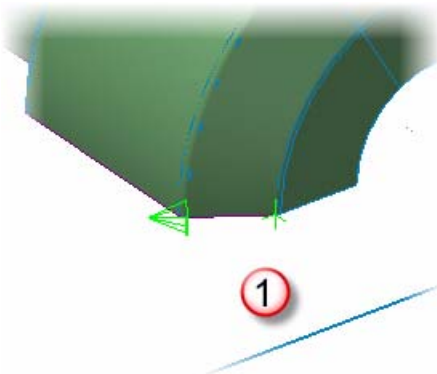
Partial is used to select a section of a full chain.



- 14** Select chain 1 as shown. The start of the chain is at 1 (narrow end of the part) with the end of the chain at 2 (wide end of the part). Zoom in and out as necessary.

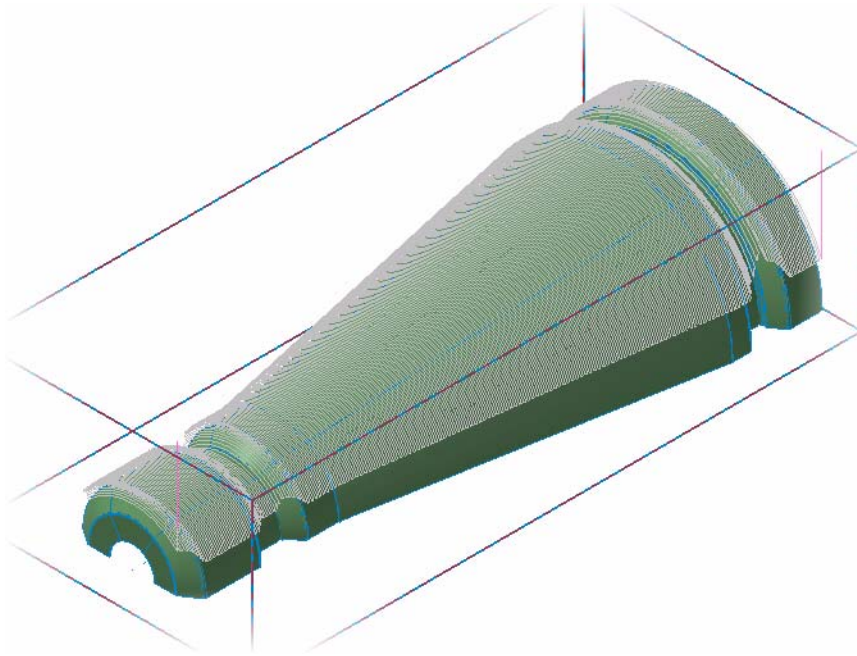


- 15** Select chain 2 on the opposite side of the part as shown. The start of the chain is at 1 (narrow end of the part) with the end of the chain at 2 (wide end of the part). It will be necessary to rotate the part to view the left side as shown by the images below. Zoom and rotate as necessary.



- 16** Right-click in the graphics area and select **Isometric (WCS)**.

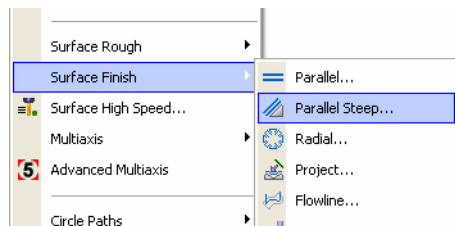
- 17 Click **OK** from the chaining dialog box to generate the surface finish blend toolpath.



- 18 Turn off the toolpath display for the operation.  
19 Save your part file.

## *Exercise 2: Creating a Surface Finish Parallel Steep*

- 1 Choose **Toolpaths, Surface Finish, Parallel Steep**.



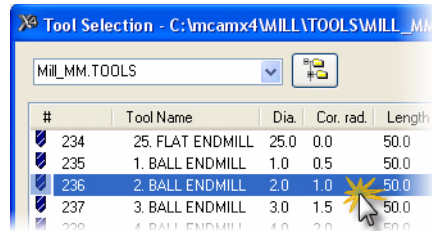
- 2 Click the **Select last** button when prompted for drive surfaces.  
3 Press **[Enter]** or click the **End Selection** button to accept the selection.  
4 Click **OK** on the Toolpath/surface selection dialog box.

The Surface Finish Parallel Steep dialog box opens to the Toolpath parameters tab.

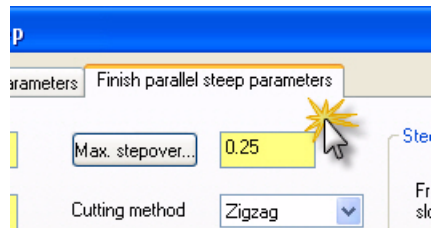
- 5 Click the **Select library tool** button.

The default metric tool library opens.

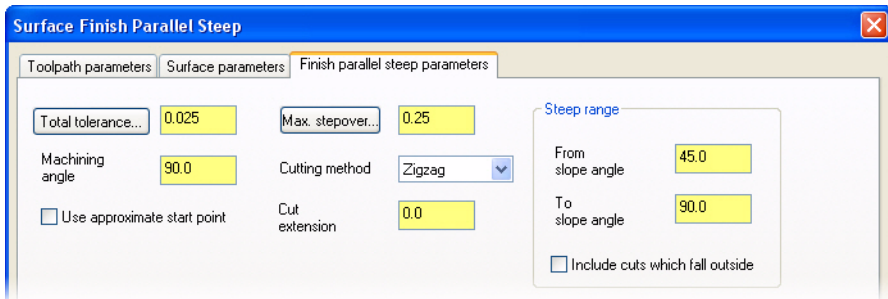
- 6 Select the 2mm ball endmill and click **OK**.



- 7 Click the **Surface parameters** tab. Set the stock to leave on drive to 0.25.
- 8 Click the **Finish parallel steep parameters** tab.

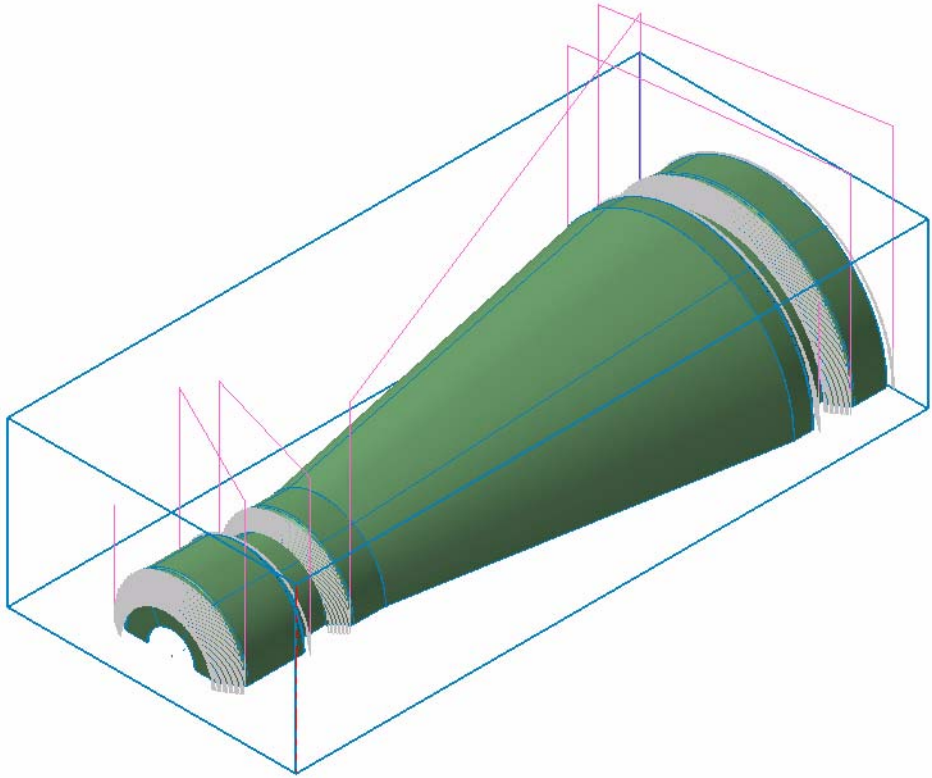


- 9 Set the parameters as shown.



- 10 Click the **Total tolerance** button to open the Refine Toolpath dialog box.
- 11 Set the surface quality to **Better**, and then click **OK**.

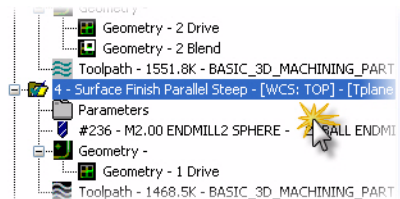
- 12 Click **OK** to generate the parallel steep toolpath.



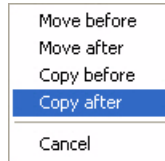
- 13 Turn off the toolpath display for the operation.
- 14 Save your part file.

### *Exercise 3: Copy and Modify an Operation*

- 1 Right-click and hold the button down on the Surface Finish Parallel Steep operation.
- 2 Drag the operation below the Surface Finish Parallel Steep operation and release the mouse button.

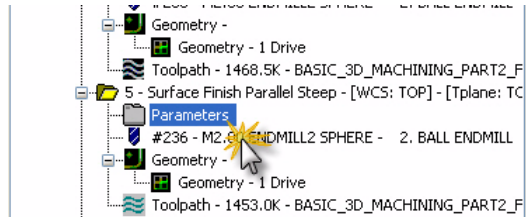


- 3 Select **Copy after** from the list of choices.



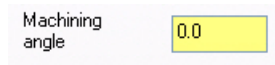
- 4 Click **Parameters** in the new operation.

Click the plus [+] to expand the operation if necessary.



- 5 Click the **Finish parallel steep parameters** tab.

- 6 Set the Machining angle to **0**.



- 7 Click **OK**.

- 8 Turn on the toolpath display for the copied operation.

---

*Note: The copied operation inherits display settings from the original operation.*

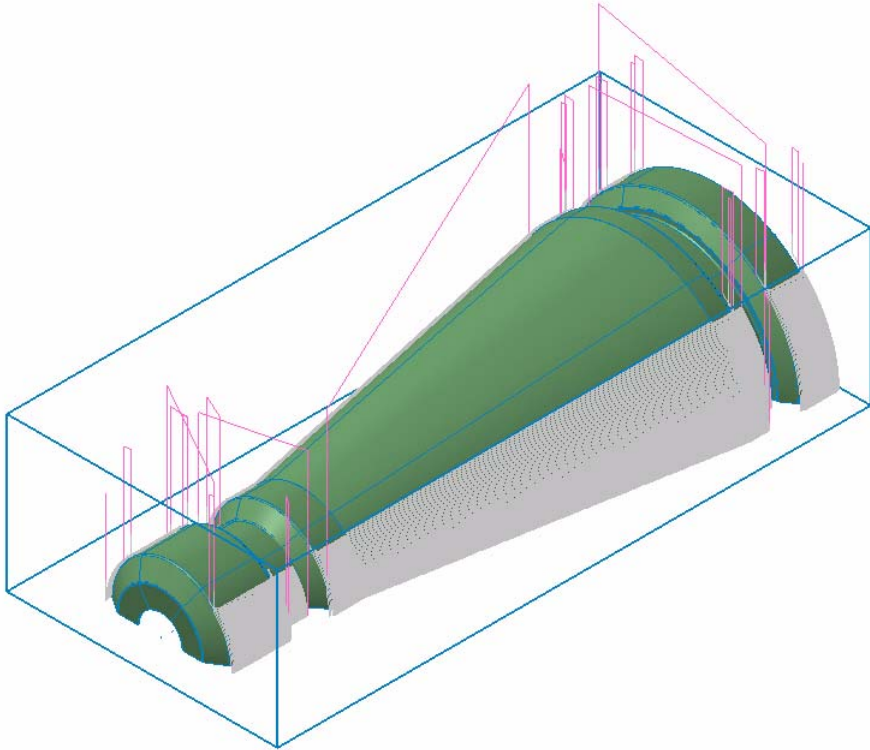
---

- 9 Click the **Regenerate all dirty operations** button in the Toolpath Manager.





- 10** The parallel steep toolpath with the new parameters will be generated as shown.



- 11** In the Toolpath Manager, click the **Move insert arrow down one item** button.



- 12** Turn off the toolpath display for the operation.
- 13** Save your part file.



## LESSON 4

# Finishing the Part

After roughing and semi-finishing the part, finishing operations will generally need to be performed. Finishing operations remove remaining stock while improving surface finish quality. Tighter tolerances, smaller stepover values, smaller tools, as well as toolpath refinement will be used in this lesson to achieve the desired finish quality.

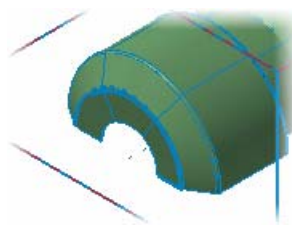
### *Lesson Goals*

- Create Swept 3D toolpaths to finish the front and back of the part.
- Create a Flowline toolpath to finish the entire part.

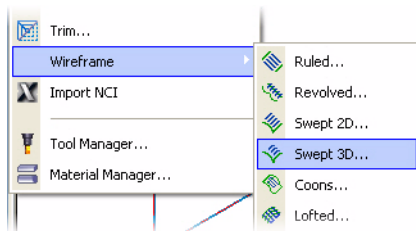
### *Exercise 1: Creating a Swept 3D*

- 1 Zoom in on the front of the part.

Zooming allows for easier chain selection.

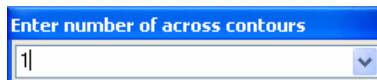


- 2 Choose **Toolpaths**, **Wireframe**, **Swept 3D**.



- 3 Type **1** for the number of across contours and press **[Enter]**.

The chaining dialog box opens.



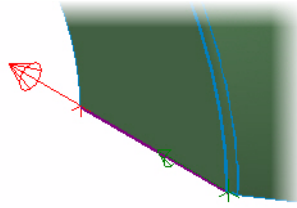
- 4 Click the **Single** button in the chaining dialog box.



- 5 Select the line segment as shown when prompted to **define the across contour**.

The prompt changes to define along contour 1.

Chain direction directly affects the toolpath outcome. Be sure to select the chain in the indicated direction.

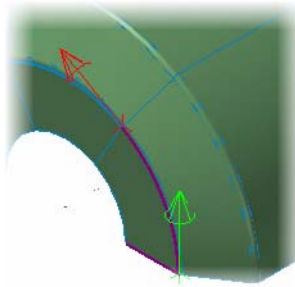


- 6 Click the **Partial** button and the **Wait** checkbox in the chaining dialog.



- 7 Select the arc as shown when prompted to **define along contour 1**.

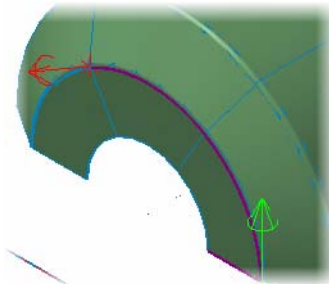
The prompt changes to select the last entity. The wait option allows you to select multiple entities to add to the chain. Review Mastercam's **Help** for more information.



- 8 Select the next arc as indicated.

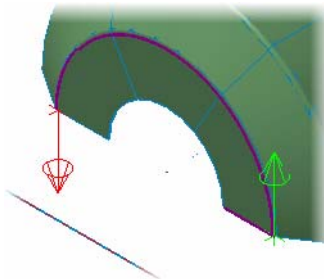
The resulting chain is shown.

Subsequent segments can be selected at any location along the entity. Chain direction has been set by the first entity in the chain.



- 9 Select the last arc as indicated.

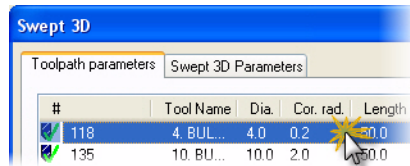
The resulting chain is shown.



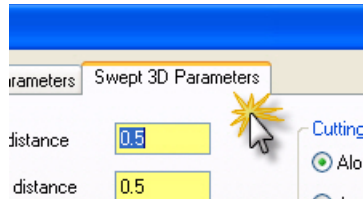
- 10 Click **OK** on the Chaining dialog box.

The Swept 3D dialog box opens to the Toolpath parameters tab.

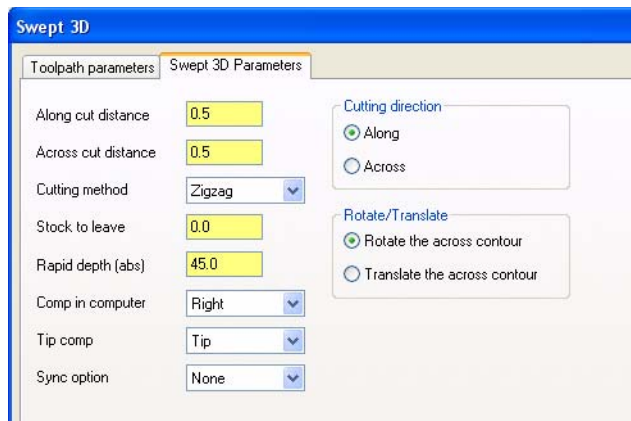
- 11 Select the 4mm bull endmill with the 0.2mm corner radius showing in the tool list.



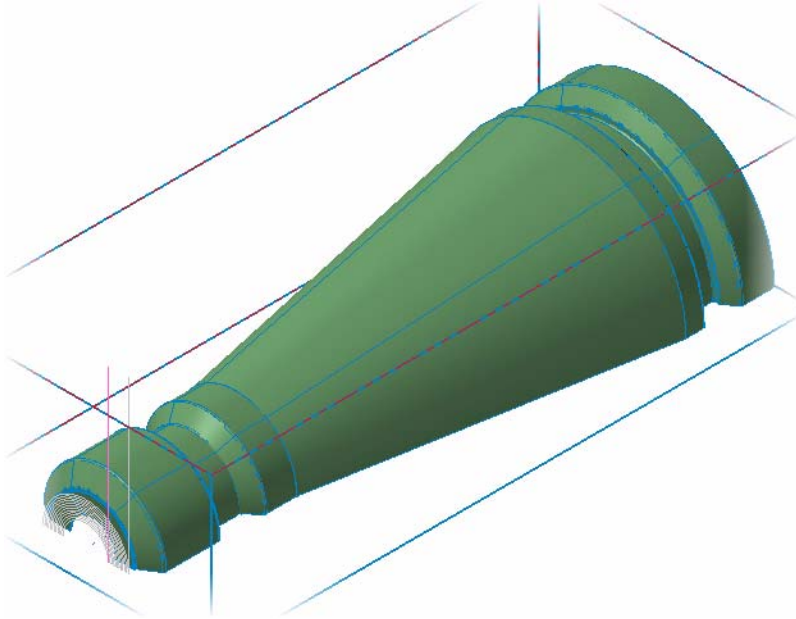
- 12 Click the **Swept 3D Parameters** tab.



- 13 Set the parameters as shown.



- 14 Click **OK** to generate the swept 3D toolpath.

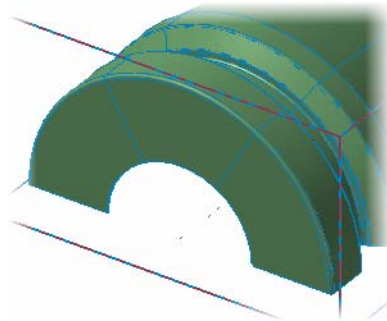


- 15 Turn off the toolpath display for the operation.  
16 Save your part file.

## *Exercise 2: Creating a second Swept 3D*

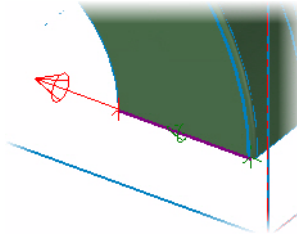
- 1 Rotate your part so you can clearly see the back face. Zoom in to make chain selection easier.

Chaining is possible through the part in an isometric view. Manipulating the part simply makes chaining easier.

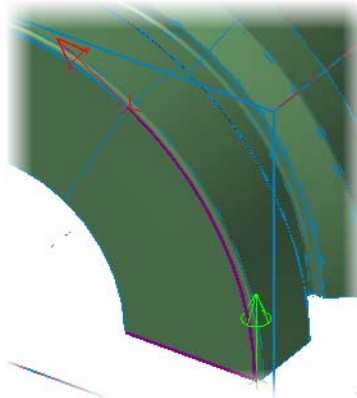


- 2 Choose **Toolpaths, Wireframe, Swept 3D**.  
3 Type **1** for the number of across contours and press **[Enter]**.  
The chaining dialog box opens.  
4 Click the **Single** button in the chaining dialog box.

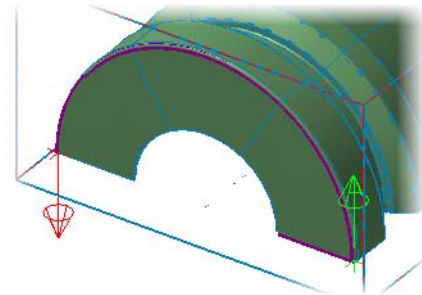
- 5 Select the line segment as shown when prompted to **define the across contour**.



- 6 Click the **Partial** button and the **Wait** checkbox in the chaining dialog.
- 7 Select the arc as shown when prompted to **define along contour 1**.

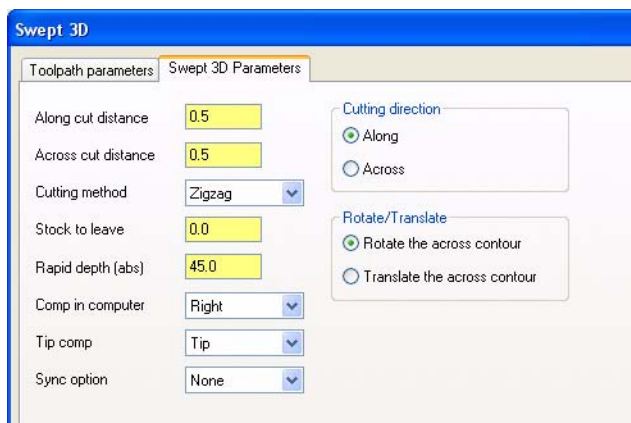


- 8 Select the remaining arcs using the same method as on the front of the part.  
The resulting chain should be as shown.

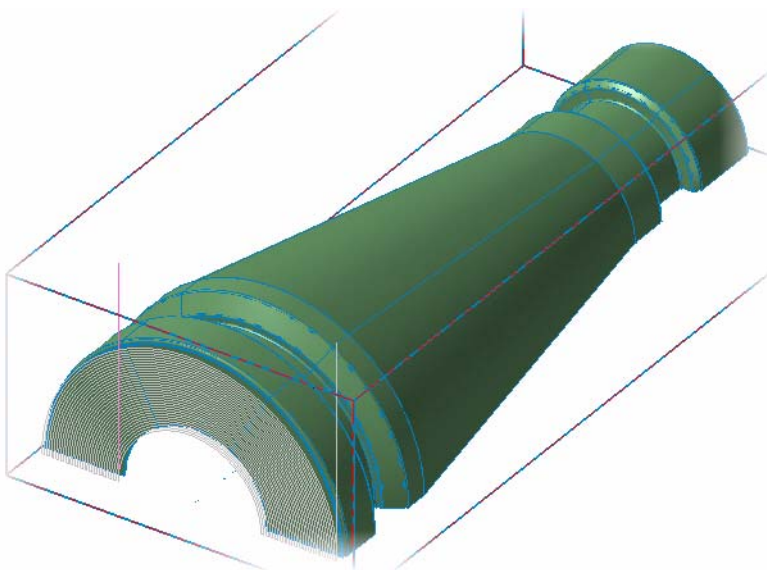


- 9 Click **OK** on the Chaining dialog box.  
The Swept 3D dialog box opens to the Toolpath parameters tab.
- 10 Select the 4mm bull endmill with the 0.2mm corner radius showing in the tool list.
- 11 Click the **Swept 3D Parameters** tab.

**12** Set the parameters as shown.



**13** Click **OK** to generate the swept 3D toolpath on the rear of the part.



**14** Right-click in the graphics area and select **Isometric (WCS)**.

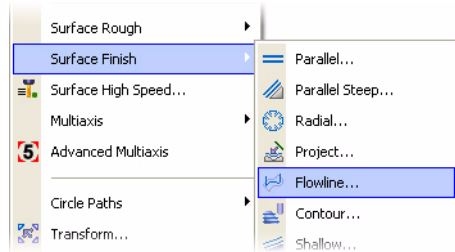
**15** Turn off the toolpath display for the operation.

**16** Save your part file.

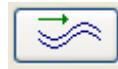


## Exercise 3: Creating a Flowline

- 1 Choose **Toolpaths, Surface Finish, Flowline**.

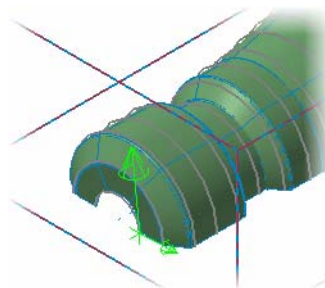


- 2 Click the **Select last** button when prompted for drive surfaces.
- 3 Press **[Enter]** or click the **End Selection** button to accept the selection.
- 4 Click the **Flow** button in the Toolpath/Surface selection dialog box.



- 5 Use the buttons in the **Flowline data** dialog box to manipulate the flow pattern.

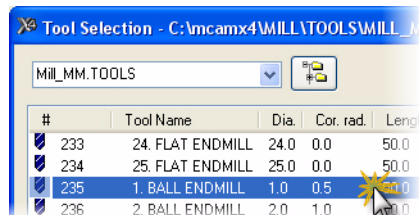
Click the buttons to toggle the related direction as needed. The pattern should be be set as shown. Click **Help** for detailed information about these buttons.



- 6 Click **OK** on the Flowline data dialog and the Toolpath/Surface selection dialog box. The Surface Finish Flowline dialog box opens to the Toolpath parameters page.

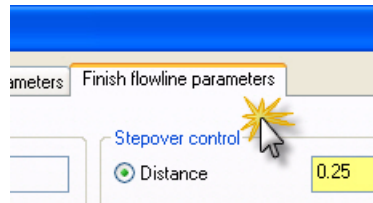
- 7 Click the **Select library tool** button. The default metric tool library opens.

- 8 Select the 1mm ball endmill and click **OK**.

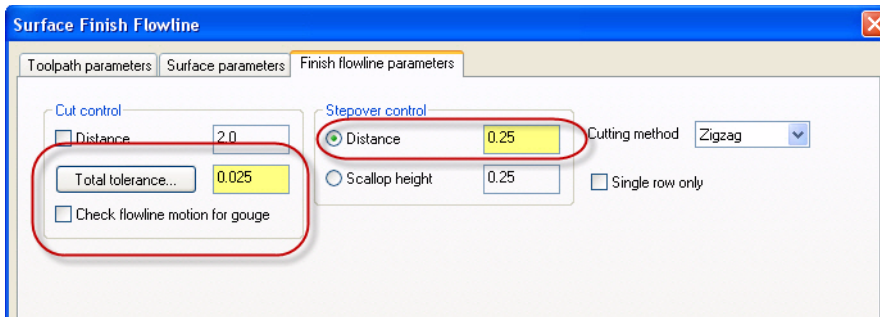


- 9 Click the **Surface parameters** tab. Set the stock to leave on drive to **0**.

10 Click the **Finish flowline parameters** tab.

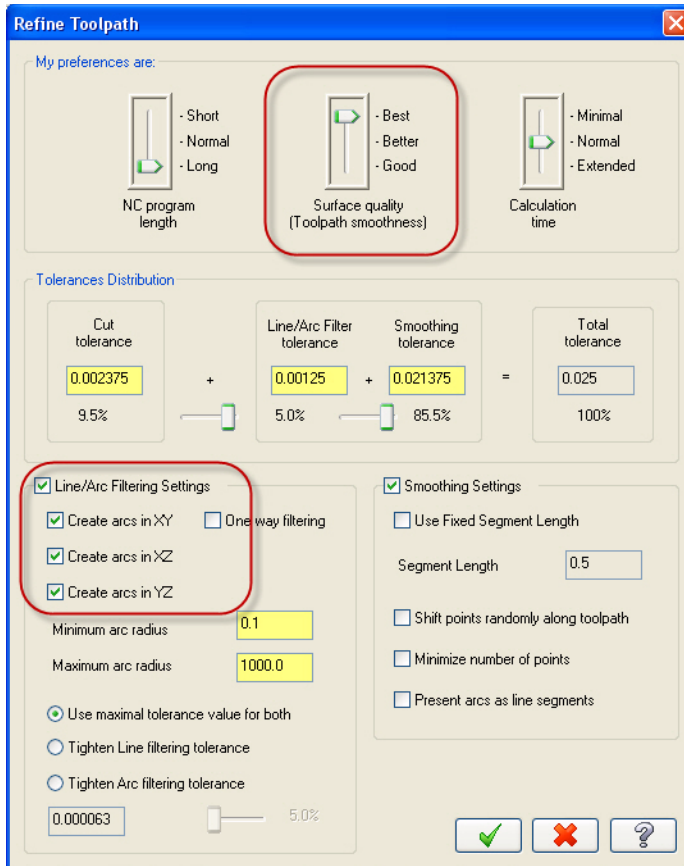


11 Set the parameters as shown.






12 Click the **Total tolerance** button to open the Refine Toolpath dialog box.

- 13 Set the parameters as shown and click **OK**.



The **Refine Toolpath** dialog box is shown with the following settings:

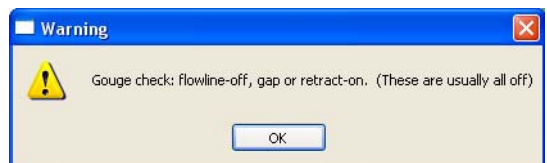
- My preferences are:**
  - NC program length: Short, Normal, Long
  - Surface quality (Toolpath smoothness): Best, Better, Good** (highlighted with a red box)
  - Calculation time: Minimal, Normal, Extended
- Tolerances Distribution:**
  - Cut tolerance: 0.002375 (9.5%)
  - Line/Arc Filter tolerance: 0.00125 (5.0%)
  - Smoothing tolerance: 0.021375 (85.5%)
  - Total tolerance: 0.025 (100%)
- Line/Arc Filtering Settings** (highlighted with a red box):
  - ☒ Create arcs in XY
  - ☐ One way filtering
  - ☒ Create arcs in XZ
  - ☒ Create arcs in YZ
  - Minimum arc radius: 0.1
  - Maximum arc radius: 1000.0
  - ☒ Use maximal tolerance value for both
  - ☐ Tighten Line filtering tolerance
  - ☐ Tighten Arc filtering tolerance
  - 0.000063 (5.0%)
- Smoothing Settings:**
  - ☒ Smoothing Settings
  - ☐ Use Fixed Segment Length
  - Segment Length: 0.5
  - ☐ Shift points randomly along toolpath
  - ☐ Minimize number of points
  - ☐ Present arcs as line segments

Buttons at the bottom:   

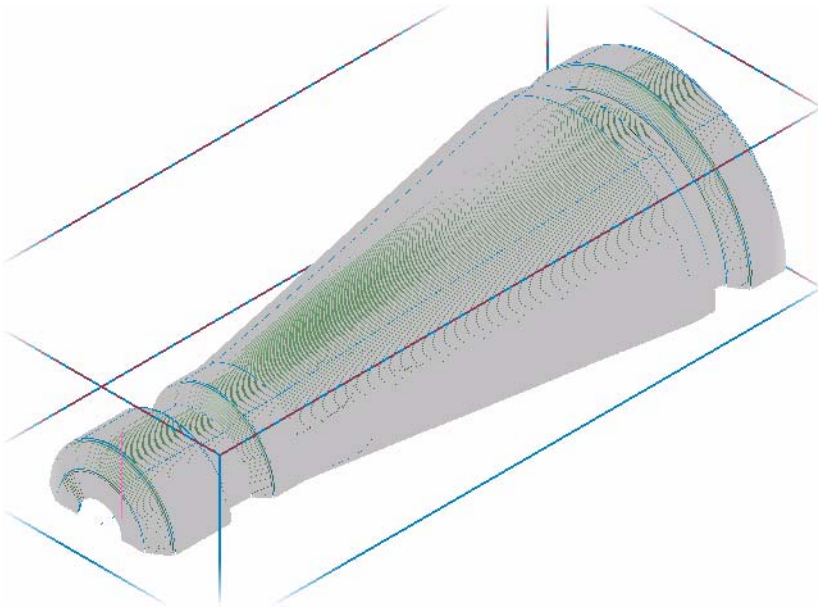
- 14 Click **OK**.

A Warning dialog box pops up.

The warning is a result of unchecking the **Check flowline motion for gouge** option on the Finish flowline parameters tab (see step 11).



- 15** Click **OK** to generate the flowline toolpath.



- 16** Turn off the toolpath display for the operation.

- 17** Save your part file.

## LESSON 5

# Backplot/Verify Your Toolpaths

Now that all toolpaths are complete, it is time to backplot and verify your operations. This process allows you to review the tool motion before any cutting takes place on the actual machine. This lesson guides you through the backplot and verify functions.

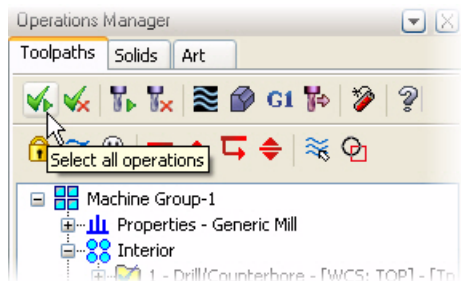
### *Lesson Goals*

- Backplot all toolpaths.
- Verify all toolpaths.

### *Exercise 1: Backplotting all Toolpaths*

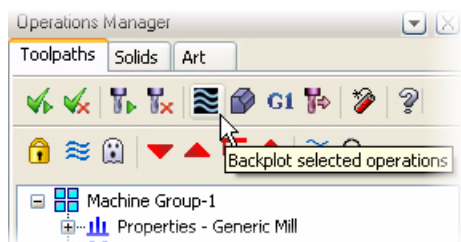
- 1 In the Toolpath Manager, click the **Select all operations** button.

All operations should now have a green check on them.



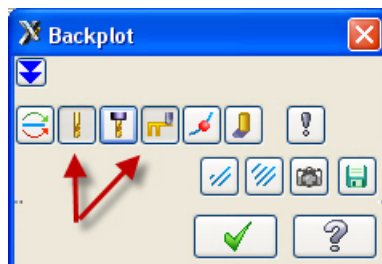
- 2 Click the **Backplot selected operations** button in the Toolpath Manager.

The Backplot dialog box and Backplot VCR bar open.



- 3 In the Backplot dialog box, select the **Display tool** and **Display rapid moves** buttons.

These options will display a simulation of the tool as well as the rapid motion.



**TIP:** To further customize your backplot display, choose other buttons on the Backplot dialog box. For example, choose the **Options** button to open the Backplot Options dialog box. This dialog box lets you set various backplot options such as tool display, holder display, and tool motion colors.

- 4 Use the buttons and sliders on the Backplot VCR bar to backplot the operations. The **Play** button (shown to the right) begins the backplot.



Click the **Help** button on the VCR bar for more information on each of the controls.



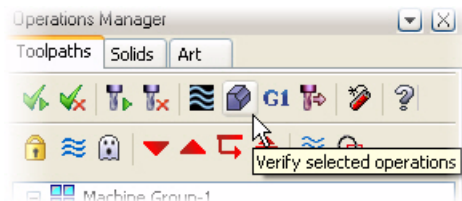
- 5 When finished, click **OK** on the Backplot dialog box to exit the backplot function.

**TIP:** The backplot display is easily customizable. See the Mastercam Help for details on each of the buttons, fields, and display options in the Backplot and the Backplot Options dialog boxes.

## Exercise 2: Verifying All Toolpaths

- 1 In the Toolpath Manager, make sure all operations are selected and click the **Verify selected operations** button.

The Verify dialog box opens.



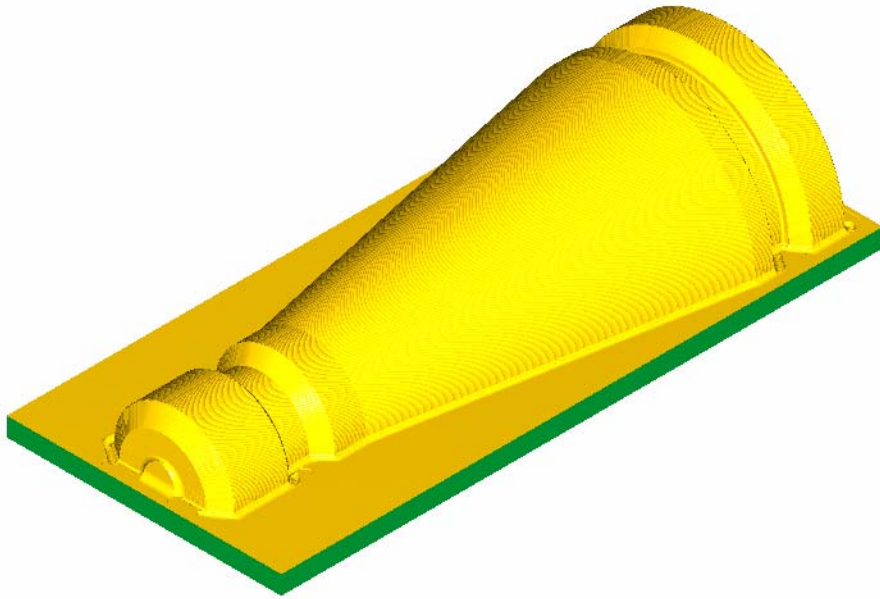
- 2 In the Verify dialog box, select the **Machine** button.



The part, stock, and toolpaths are simulated.

**TIP:** Use the buttons, fields, and controls in the Verify dialog box to customize and manage the toolpath verification process. Click the **Help** button in the dialog box for details.

When the verification process is complete, the part should look similar to the image shown here.







## LESSON 6

# Posting Toolpaths

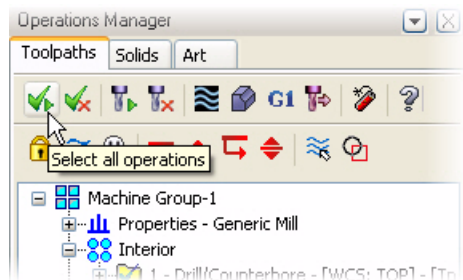
Operations must be run through a post processor in order to be usable by your machine. This process is commonly referred to as posting. This lesson will guide you through selecting and posting all of your operations. Communicating the posted code to your machine is the last step of the toolpathing process in Mastercam.

### *Lesson Goals*

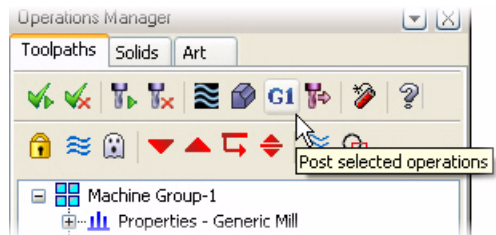
- Post all toolpaths to create an NC file.

### *Exercise 1: Posting All Toolpath Operations*

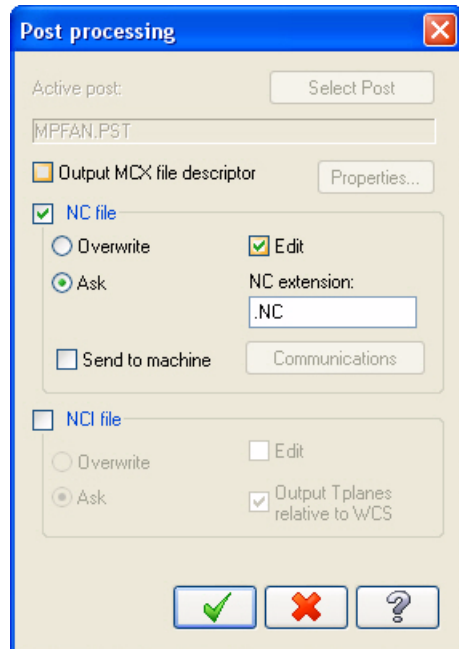
- 1 In the Toolpath Manager, click the **Select all operations** button.



- 2 Click the **Post selected operations** button.  
The Post processing dialog box opens.



3 Set the post processing options as shown.



4 Click **OK**.

The Save As dialog box opens.

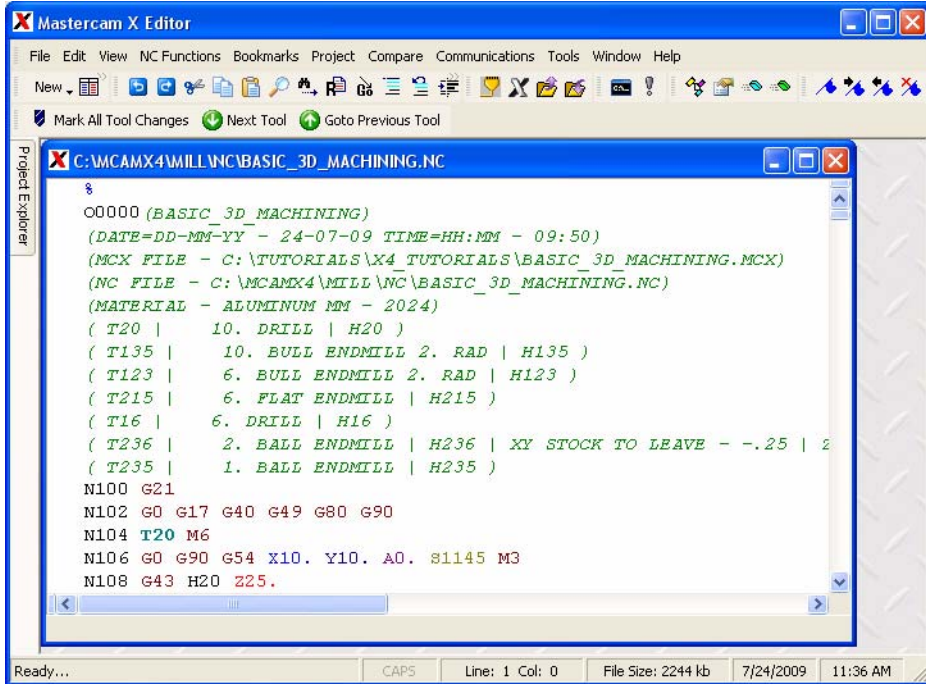
5 Click **Save** to save the NC file in the default location with the recommended file name.

---

*Notes:*

- Posting may take several minutes. When it is complete, the NC code will display in your default editor/communicator. This tutorial uses Mastercam Editor to display the NC code.
  - Producing the correct NC code for your machine and application depends on properly configuring the machine definition, control definition, and .PST file. For detailed information on machine definitions, control, definitions, and posting, please see the following documentation supplied with Mastercam:
    - Mastercam Help
    - Mastercam X4 Reference Guide (choose Reference Guide from the Mastercam Help menu)
    - Mastercam NCI & Parameter Reference Guide (in the Documentation folder under your Mastercam installation folder)
-

- 6 Your chosen editor opens (in this case, Mastercam Editor), displaying the posted NC code similar to the image below.



- 7 Scroll through the NC code to verify that each line of code meets your expectations. Edit and save as necessary.

## *Sending NC Files to the Machine*



**IMPORTANT:** This tutorial is based on the Mastercam Mill Default machine definition for training purposes only. It is not possible to provide a step-by-step procedure for sending the NC code to your machine control because machine setups are customizable and most likely different from the machine definition used here. Following is a general description of how the NC code is communicated to machines and their controls for machining.

After the NC file is reviewed, edited, and saved, you can set up your machine control to accept the NC file. This is done according to your machine and control manufacturer's procedures.

When the machine control is ready to receive the NC file, configure your preferred editor or communications program to communicate with your machine control. Refer to your communications program documentation for details.

Send the NC code to your machine control according to your machine and control manufacturer's documentation. Once you start the communication process, the send/receive data processing is mostly managed by your machine control.

Contact your local Mastercam Reseller for customized machine/control definitions, post (PST) files, and support.

## Conclusion

Congratulations! You have completed the *Basic 3D Machining* tutorial. Now that you have mastered the skills in this tutorial, we encourage you to explore Mastercam's other features and functions. Additional tutorials may be available in this or other series. Please contact your authorized Mastercam Reseller for further training.





***cnc software, inc.***

671 Old Post Road  
Tolland, CT 06084 USA  
[www.mastercam.com](http://www.mastercam.com)

*Printed in the USA*

***Mastercam X4 Getting Started Series - Basic 3D Machining: Part X4-PDF-TUT-3M***