



# MASTERCAM LATHE TUTORIAL

January 2019



# MASTERCAM LATHE TUTORIAL

January 2019

© 2019 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2019

## Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

<http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx>

## Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

# TABLE OF CONTENTS

<b>Introduction</b>	<b>7</b>
General Tutorial Requirements	8
<b>General Setup</b>	<b>9</b>
Exercise 1: Getting Ready to Work	9
Select configuration file	9
Exercise 2: Loading a Machine Definition	10
Rename the Machine Group	11
Exercise 3: Orienting the Part	11
Exercise 4: Using a Lathe Coordinate System	14
Exercise 5: Creating 2D Geometry	16
Create a level for the new profile geometry	16
Create the profile	17
Exercise 6: Setting Up the Stock in the Left Spindle	18
Exercise 7: Defining the Chuck Jaws	22
<b>Facing, Roughing, and Finishing the Outer Diameter</b>	<b>25</b>
Exercise 1: Facing the Part	25
Enter the cutting values	26
Exercise 2: Roughing the Outer Diameter	27
Chain the geometry	27
Enter the toolpath parameters	28
Exercise 3: Finishing	34
Enter the toolpath parameters	34
Exercise 4: Backplotting the Toolpaths	35
<b>Adding Grooves and Threads</b>	<b>41</b>
Exercise 1: Grooving on the Outer Diameter: Multiple Chains	41
Exercise 2: Grooving on the Outer Diameter: Rough Pass Only	44
Exercise 3: Finishing with Plunge Cuts	46
Exercise 4: Adding a Thread Toolpath	51
Select the tool	51

Enter thread shape parameters .....	52
Enter thread cut parameters .....	54
Exercise 5: Verifying the Toolpaths .....	55
<b>C-Axis Drilling Operations .....</b>	<b>59</b>
Making Drill Geometry .....	59
Exercise 1: Adding a New Toolpath Group .....	62
Exercise 2: Creating a C-Axis Drill Operation .....	63
Enter the Toolpath Parameters .....	65
Exercise 3: Copying the Drilling Operation .....	68
Exercise 4: Modifying the Drilling Parameters .....	69
<b>Cutoff and Stock Flip .....</b>	<b>73</b>
Exercise 1: Cutting Off the Part .....	73
Exercise 2: Programming a Stock Flip .....	75
Re-align Part Geometry .....	76
Enter new stock and chuck jaw positions .....	77
<b>Machining the Inner Diameter .....</b>	<b>81</b>
Exercise 1: Creating New Tools in the Lathe Tool Manager .....	81
Create the 9.6 mm drill .....	81
Create the 11.4 mm drill .....	84
Create the ID roughing tool .....	85
Create the ID finishing tool .....	86
Create the ID threading tool .....	87
Exercise 2: Facing the Back of the Part .....	88
Exercise 3: Drilling the First Inner Diameter .....	88
Exercise 4: Drilling the Second Inner Diameter .....	91
Exercise 5: Roughing and Finishing the Third Inner Diameter .....	93
Exercise 6: Adding an ID Thread .....	96
Exercise 7: Refining Your Verification Results .....	99
<b>Post Output .....</b>	<b>103</b>
Exercise 1: Renumbering Tools .....	103

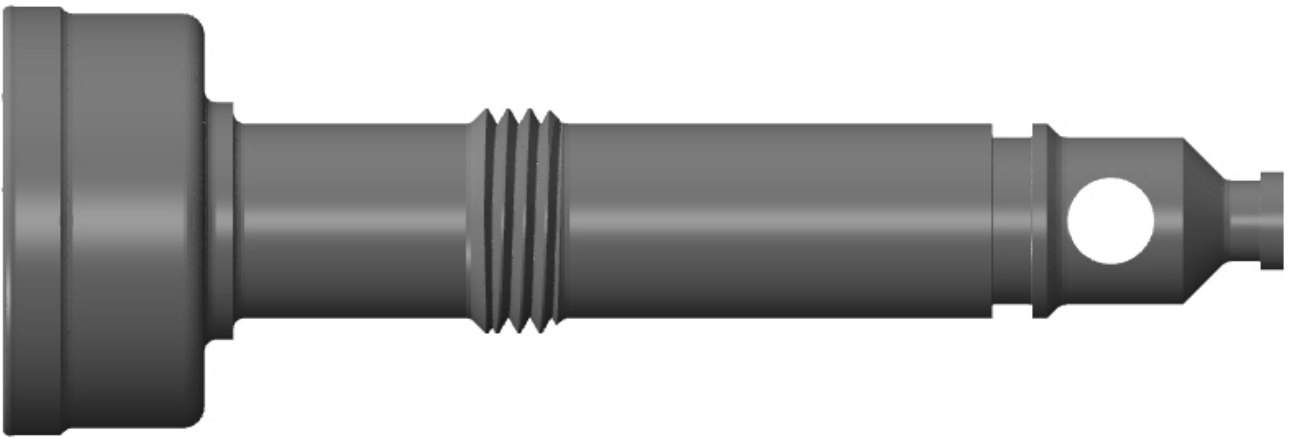
Exercise 2: Posting .....	104
<b>Conclusion .....</b>	<b>106</b>
Mastercam Resources .....	106
Contact Us .....	106



# INTRODUCTION

Mastercam Lathe delivers turning software with powerful toolpaths and techniques. In this tutorial, you create general turning, milling (c-axis), and miscellaneous operations to program the interior core of a hose nozzle in Mastercam 2019. The part requires basic lathe operations such as facing, roughing, and finishing as well as grooving and threading toolpaths.

Starting with [Chapter 2 on page 25](#), a blueprint at the beginning of each lesson provides the necessary dimensions you need to create the toolpaths. Within the parts folder that is delivered with this tutorial, you will find the original SOLIDWORKS™ part used in the tutorial.



- Nozzle - 2.SLDPRT

For your reference, the folder also provides a sample of the part after each lesson:

- Hose Nozzle - Inner Core - L1.mcam
- Hose Nozzle - Inner Core - L2.mcam
- Hose Nozzle - Inner Core - L3.mcam
- Hose Nozzle - Inner Core - L4.mcam
- Hose Nozzle - Inner Core - L5.mcam
- Hose Nozzle - Inner Core - L6.mcam
- Hose Nozzle - Inner Core - L7.mcam

Place these files (extracted from Mastercam Lathe Tutorial.zip) anywhere on your system, but be sure to also keep unmodified copies.

Mastercam is a CAD/CAM software program with solutions for a wide array of machining applications. While this tutorial requires only a basic familiarity with Mastercam, its intention is to provide you with an introduction to the Lathe product. Use the resources listed at the end of this tutorial to explore and learn more about other features and functions in Lathe and in Mastercam.

## Goals

- Set up the job by creating a stock and fixtures.
- Move the solid model to its machine orientation and create any necessary 2D geometry.
- Create and edit tools as required by the part operations and dimensions.
- Program operations to work on a part's inner and outer diameter.
- Use Mastercam's verification tools and posting to check your work.

**WARNING:** Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 5 hours

## General Tutorial Requirements

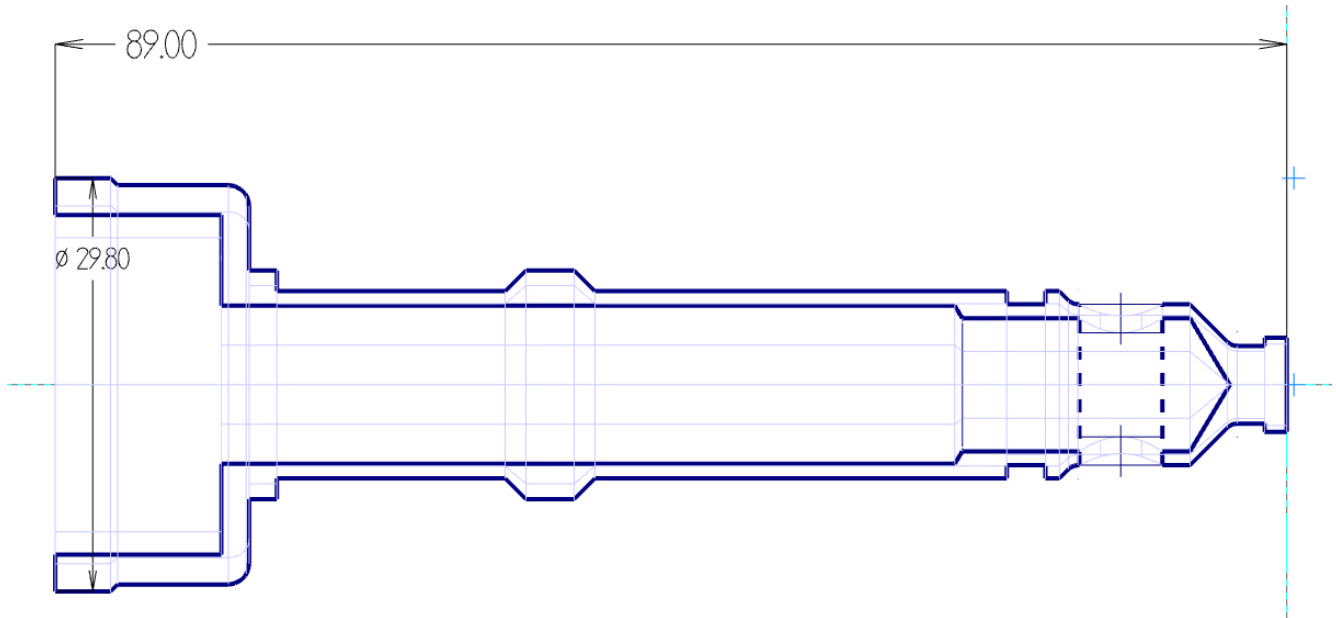
All Mastercam 2019 tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- 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 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel:  
[www.youtube.com/user/MastercamTechDocs](http://www.youtube.com/user/MastercamTechDocs)
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

# CHAPTER 1

## GENERAL SETUP

Before generating toolpaths for the part, you must prepare Mastercam and the part file. This preparation includes such tasks as selecting a machine definition and defining the stock.



### Goals

- Open and orient the part.
- Select a machine.
- Create 2D CAD part geometry.
- Define stock boundaries.
- Add chuck jaws.

### Exercise 1: Getting Ready to Work

This tutorial includes the files you need to complete the exercises. You can find these files in the tutorial's Parts folder. Place these files on your system wherever convenient, but be sure to keep an unmodified set. In preparation for this tutorial, set up Mastercam as described here.

#### Select configuration file

1. Start Mastercam using your preferred method:
  - a. Double-click Mastercam's desktop icon.



OR

b. Launch Mastercam from the Windows Start menu.

2. Select the default metric configuration file:

a. Click the **File** tab.

b. Choose **Configuration** from Mastercam's Backstage View to open the **System Configuration** dialog box.



c. Choose ...\\mcamxm.config <Metric> from the **Current** drop-down list.



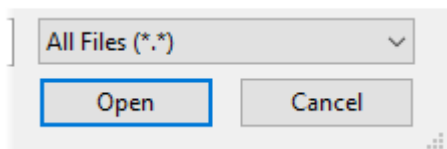
d. Click **OK**.

## Exercise 2: Loading a Machine Definition

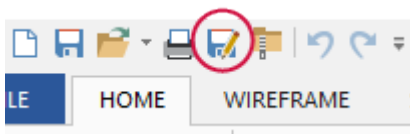
Your part file needs at least one machine group before you can create toolpaths. Mastercam creates a machine group whenever you select a machine. To see how machine selection works, complete the following exercise.

1. Open the part file, *Nozzle-2.SLDPRT*, which is provided with this tutorial.

**Note:** If SOLIDWORKS files are not displayed, select **All Files (\*.\*)** from the file type drop-down.

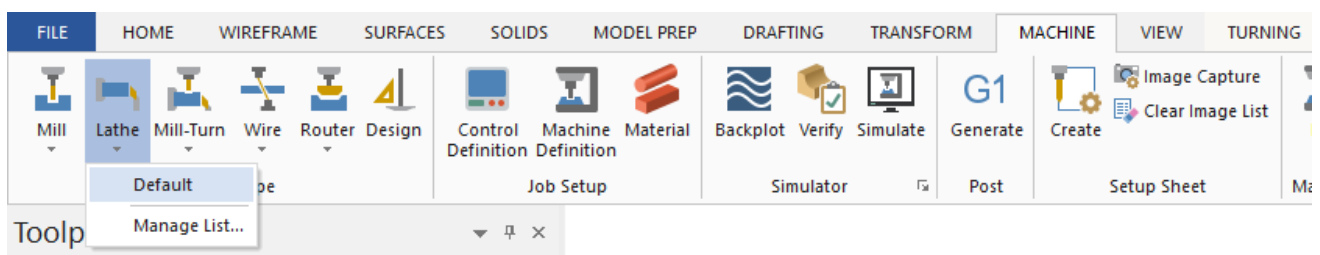


2. Save the part as *Hose Nozzle - Inner Core.mcam*.



Saving the part under a new name helps prevent you from accidentally modifying the original.

3. From the **Machine** tab, choose the **Lathe** machine type, and select **Default**.



The menu lists all available machine definitions. This tutorial uses Mastercam's default Lathe machine. Normally, you would select the machine on which you plan to cut the part from the list displayed here.

#### Notes:

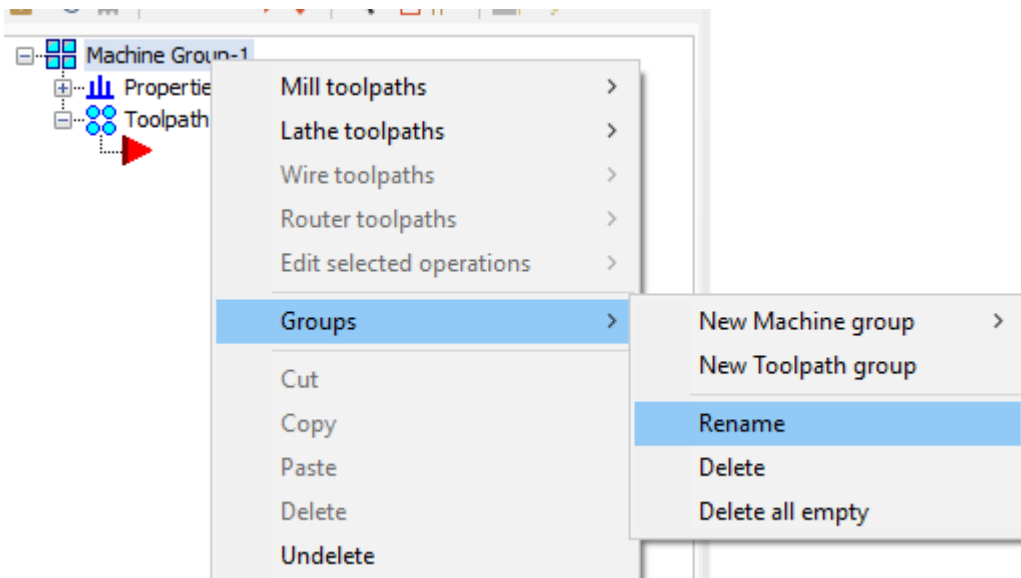
- To customize the Machine Selection lists, choose a machine type from the menu and **Manage List**. For more information on the dialog box that displays, click the dialog box's Help button.

### Rename the Machine Group

Machine groups store complete jobs for a specific machine. For example, if some toolpaths will be cut on a lathe, and other toolpaths on a mill, you can simply create a second machine group. Each machine group can store its own job setup information and tools, and use a different set of toolpath defaults. The toolpaths from each group will post to separate NC files.

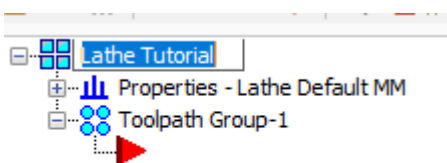
Mastercam lets you create as many machine groups as you need to organize your work.

- Right-click the machine group, and select **Groups, Rename** from the pop-up menu.



Mastercam highlights the current group name.

- Type a new machine group name.



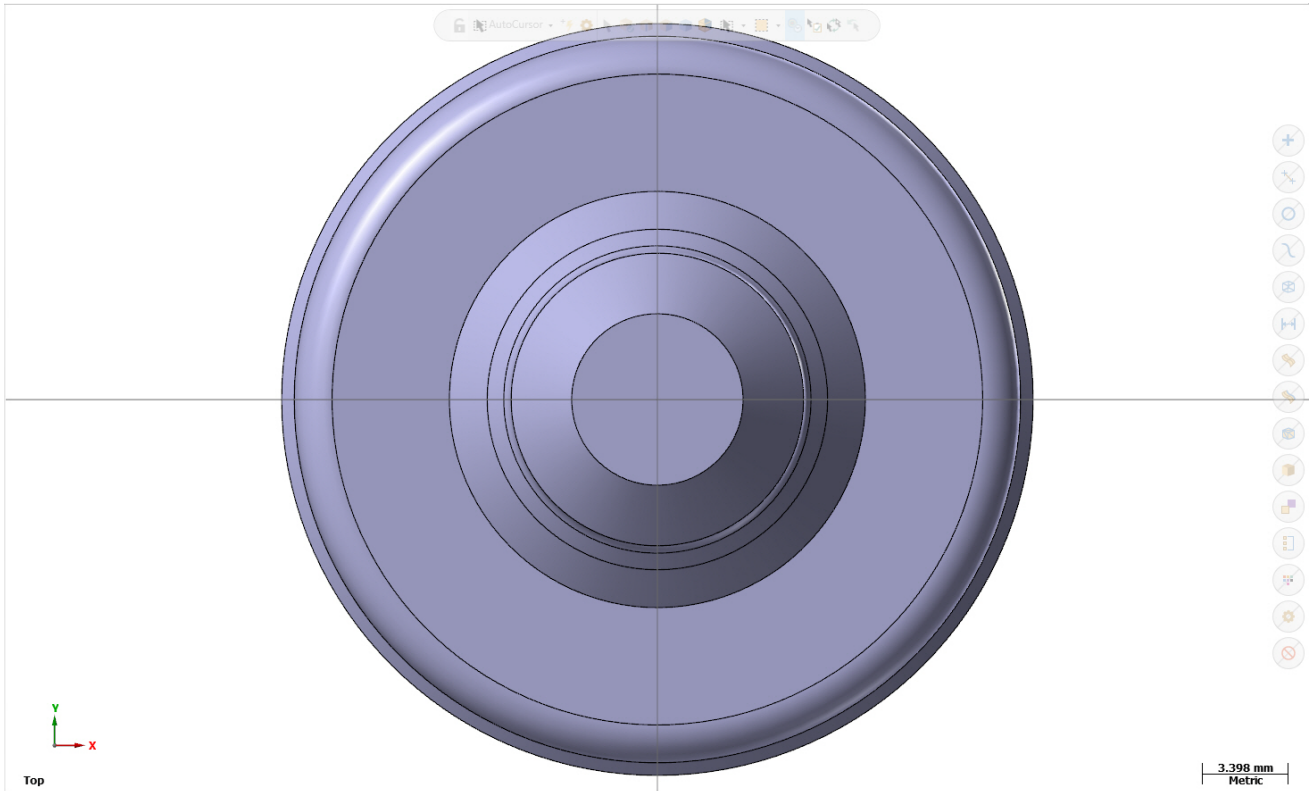
The machine group name can be anything you want, but it is best to choose a name that describes the machine and its operations.

- Select **File, Save** to save the file.

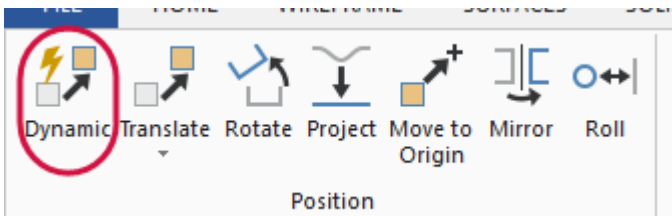
### Exercise 3: Orienting the Part

Before you can machine the part, you need to move the geometry to its machine orientation. In this exercise, you use Mastercam's dynamic gnomon to re-align the part geometry to a typical horizontal lathe.

1. If necessary, press [F9] to display axes.

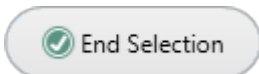


2. Right-click and change the GView to **Isometric** to view the entire part from another angle. You may wish to unzoom to get a clearer view of the origin.
3. Select **Dynamic** from the **Transform** tab.

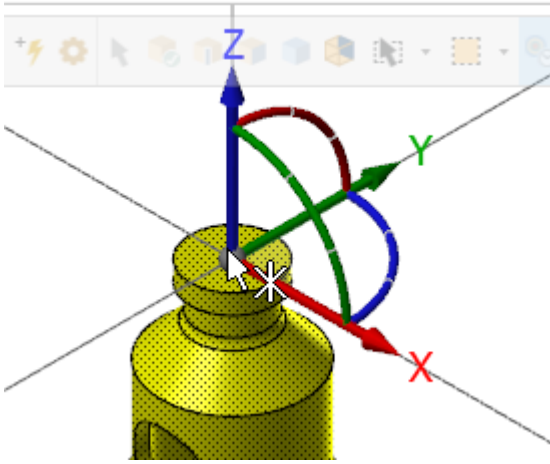


The **Dynamic** function panel displays.

4. Select the part and press [Enter], or click the **End Selection** button.



5. Use the AutoCursor to place the dynamic gnomon on the origin and click.

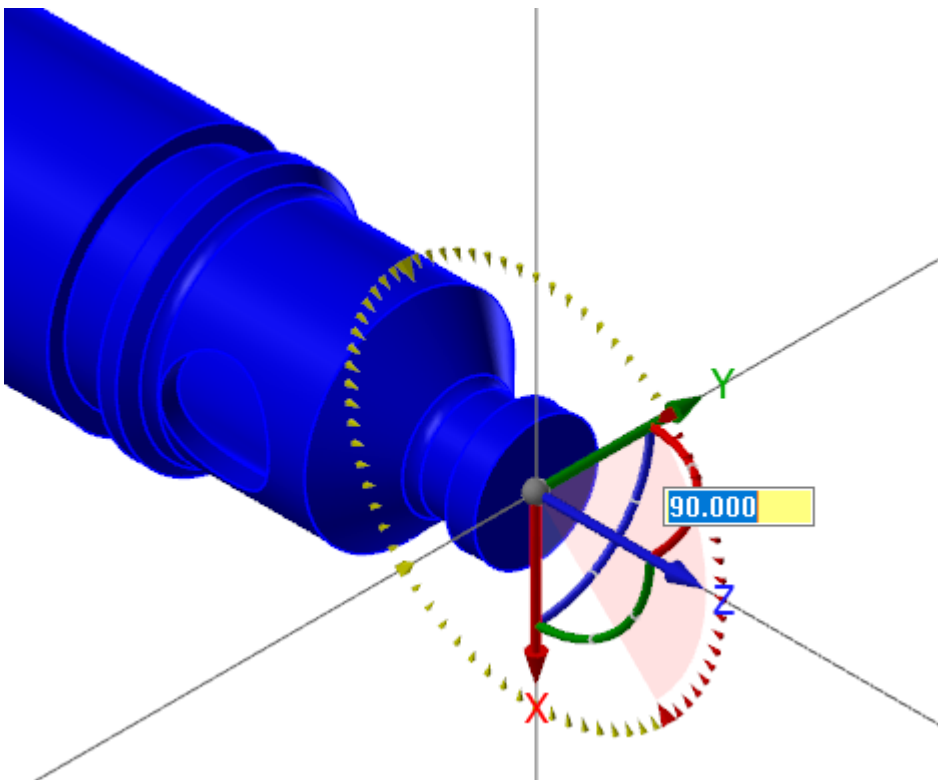


See Mastercam Help for more information on the AutoCursor.

6. Confirm that **Move** is selected in the function panel.



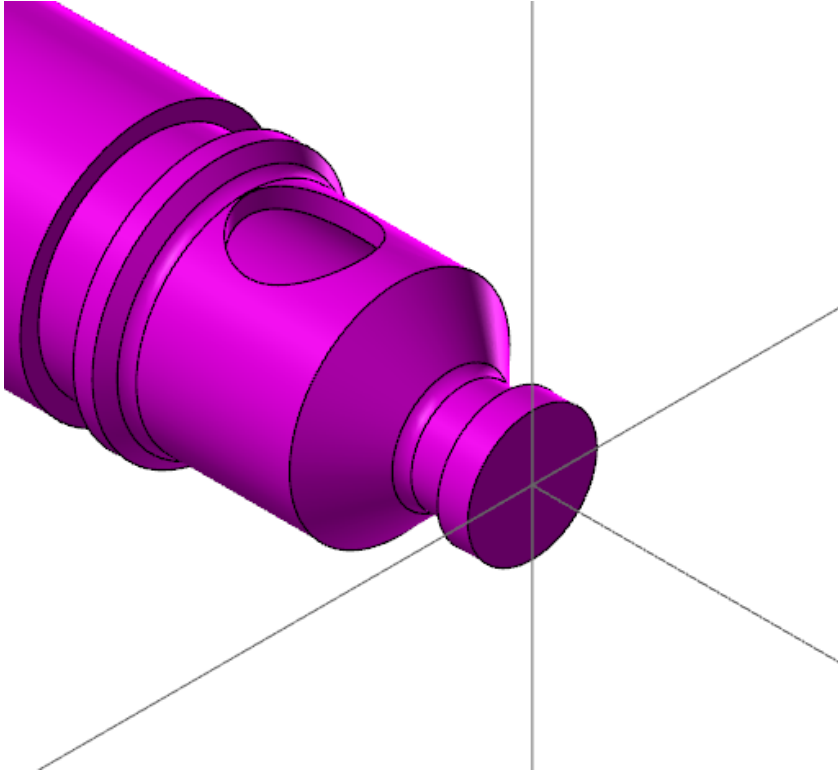
7. Select the center segment of the green arc to rotate the part 90 degrees so that its length lies on the X axis. Use your mouse to rotate the gnomon, or enter **90** into the on-screen input box.



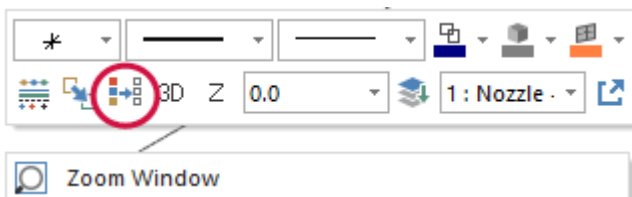
8. On the function panel, click the blue button (**OK and Create New Operation**).



9. Select the part again, and rotate it 90 degrees so that the hole aligns with the Z axis.  
Use the same method you used previously to rotate the part from its original orientation.



10. Click **OK** to accept the transformation.
11. Right-click and choose the **Clear Colors** command from the mini-toolbar to remove the result colors from the translated geometry.



12. Save the file.

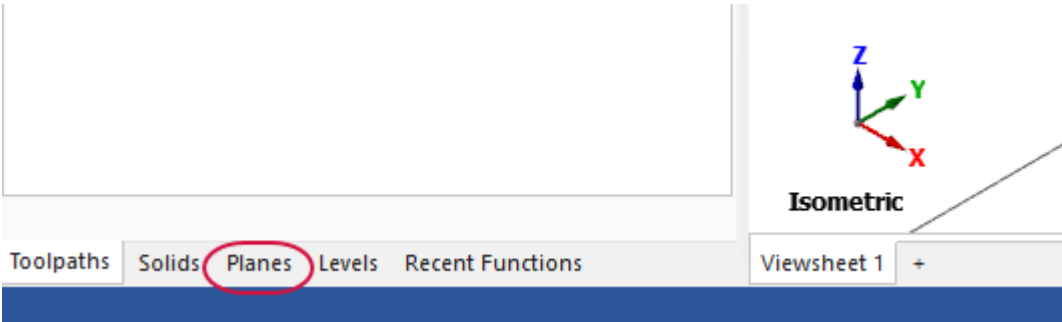
## Exercise 4: Using a Lathe Coordinate System

The coordinate system for a traditional lathe turning application differs from the coordinate system for milling applications. Instead of a 3D/XYZ space, a lathe coordinate system for typical turning applications is 2D, with the tool axis perpendicular to the spindle (Z axis) instead of parallel to it. In other words,

- The D (diameter) axis in Lathe is equal to the Y axis in Mill.
- The Z (length) axis in Lathe is equal to the X axis in Mill.

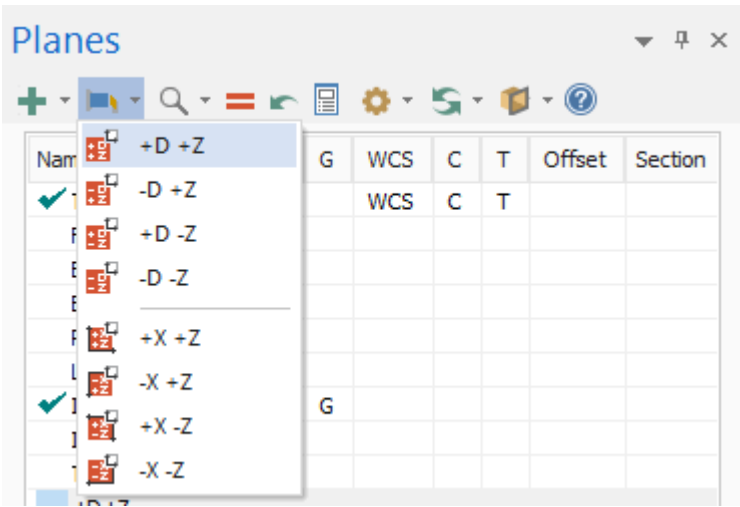
Mastercam includes special lathe construction planes that let you work in radius or diameter coordinates. For conventional 2D turning applications, use the options in the Planes Manager to select the desired coordinate system.

- 1. Select the **Planes** tab to open the Planes Manager.



**Note:** If a **Managers** tab is not available, select it from the **Managers** group on the **View** tab.

- 2. Click **Select lathe plane** from the Planes Manager toolbar, and select **+D +Z** from the drop-down menu.



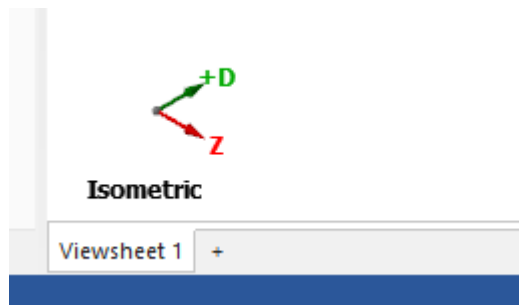
When working with a lathe, you can orient the construction plane to work in radius (X/Z) or diameter (D/Z) coordinates.

- 3. Click in the **+D +Z** row of the C column to set +D +Z as your construction plane.

Name	G	WCS	C	T	Offset	Section
✓ Top		WCS	C	T		
Front						
Back						
Bottom						
Right						
Left						
✓ Isometric	G					
Isometric reverse						
Trimetric						
+D+Z						

The **Status bar** indicator displays the new settings. The gnomon in the lower-left corner of the graphics win-

ow displays the current coordinate system.



By default the Tplane is the same as the Cplane. For 2D turning, you can leave the WCS as the Top plane.

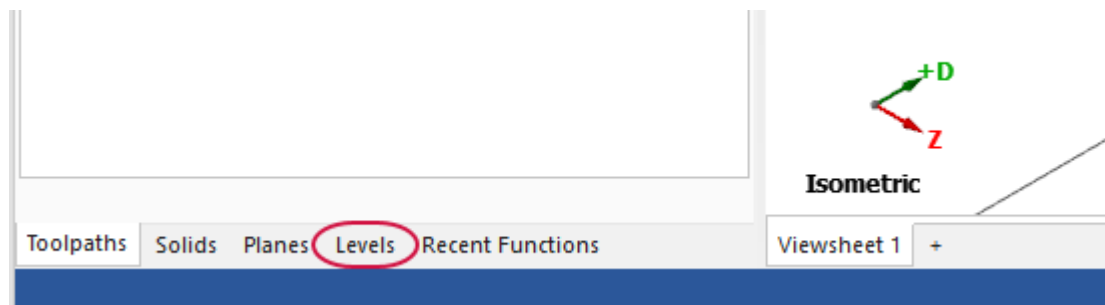
## Exercise 5: Creating 2D Geometry

Hose Nozzle - Inner Core.mcam is an imported solid with no wireframe geometry. To create Lathe toolpaths, you need two-dimensional geometry. This exercise teaches you how to create a two-dimensional profile of a solid part.

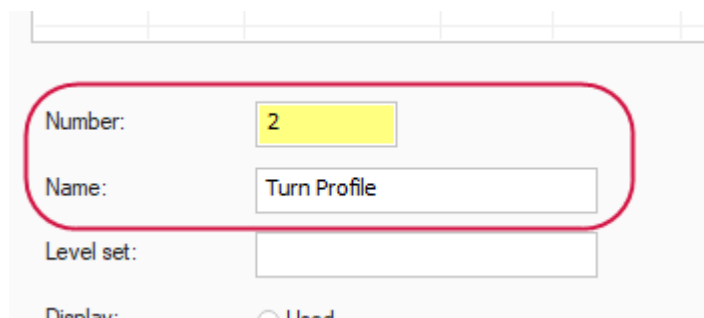
### Create a level for the new profile geometry

Use levels to organize part geometry. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and what is selectable. In this section, you create a level specifically for the 2D profile geometry that you will create in the next section.

1. Click the **Levels** tab to open the Levels Manager.



2. Create a new level by typing **2** and **Turn Profile** in the **Number** and **Name** fields.

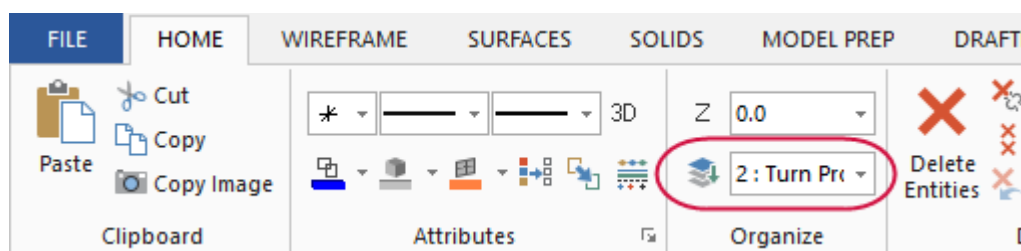


The new level displays in the table with zero entities. A blue check indicates that it is the current level.

Levels

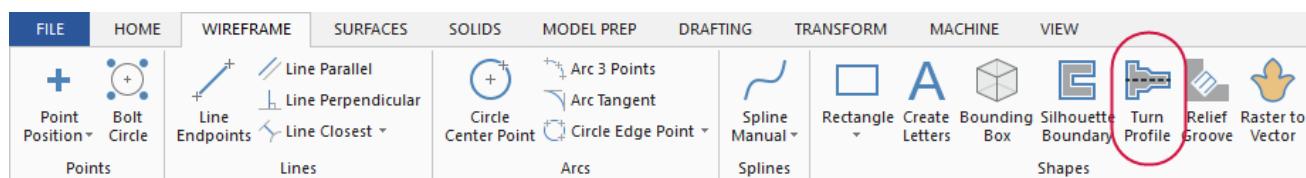
Num...	Visible	Name	Level Set	Entities
1	X	Nozzle-2		1
✓ 2	X	Turn Profile		0

The current level displays on the **Home** tab.

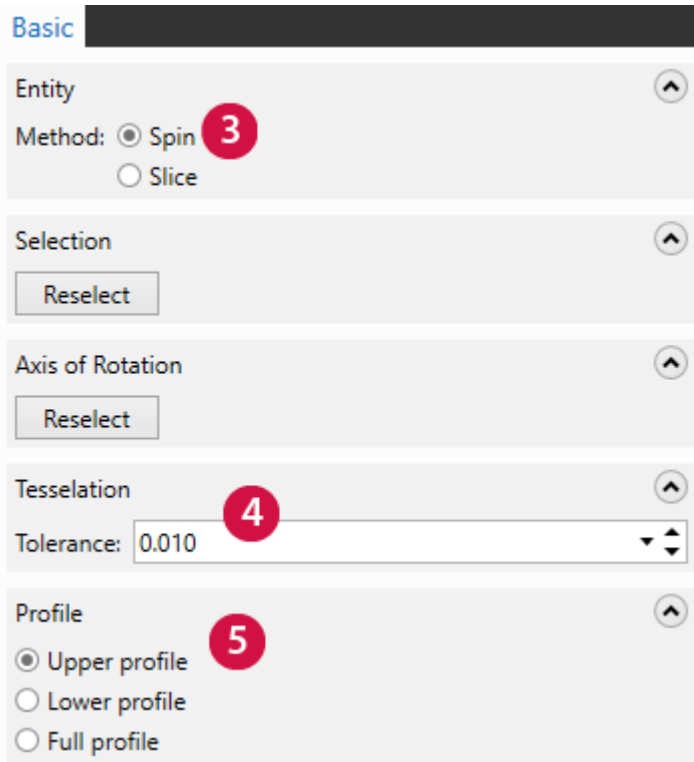


## Create the profile

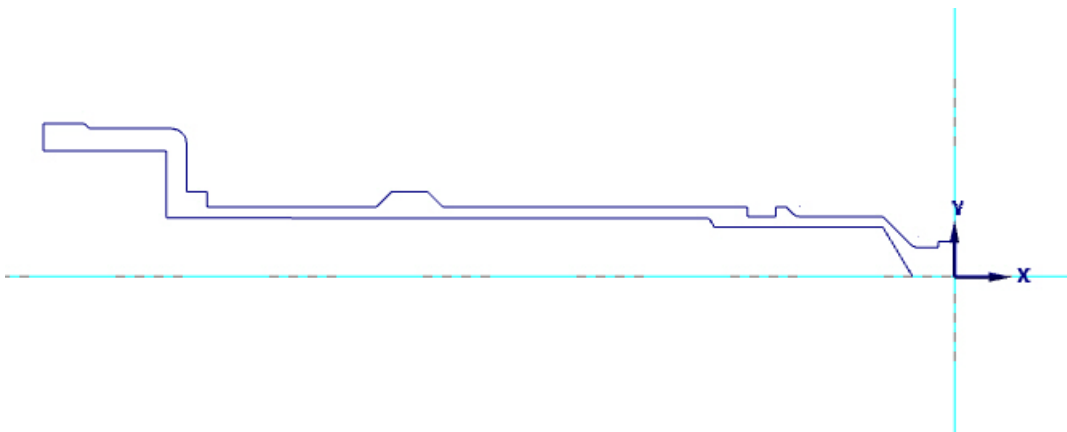
1. From the **Wireframe** tab, select **Turn Profile**.



2. Select the part as prompted. Press **[Enter]**, or click the **End Selection** button.
3. Use the **Spin** method.
  - The Spin method creates a profile by spinning the geometry about an axis.
  - The Slice method produces a profile by creating a cross-section through the geometry on the XY plane.
4. Reduce the **Tessellation tolerance** to **0.01**.
5. If necessary, choose the **Upper profile** option.



6. Click **OK**.
7. Return to the Levels Manager, and hide **Level 1** (click the **X** in the Visible column) to view the profile.



The illustration above shows the part from the TOP view.

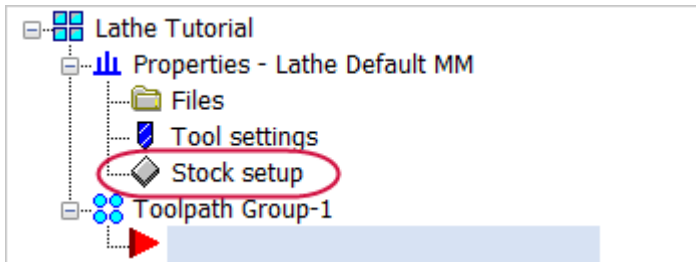
8. Save the file.

## Exercise 6: Setting Up the Stock in the Left Spindle

This exercise teaches you to create the stock for the part, set the stock margins and grip length, and place the stock at the correct Z position.

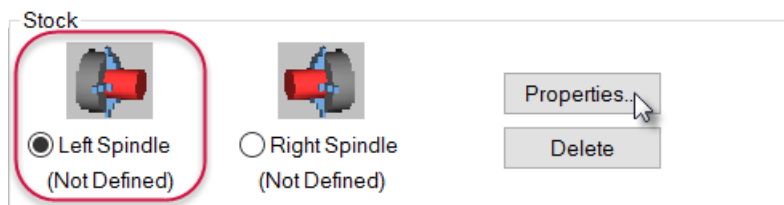
Creating the stock before creating the chuck jaws makes it easier to locate the stock relative to your part. Then, when you create the jaws, you can choose to automatically position them relative to the stock.

1. Open the Toolpaths Manager. Expand the **Properties** group and click **Stock setup** to open the **Stock Setup** tab of the **Machine Group Properties** dialog box.

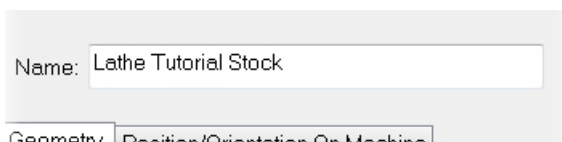


2. In the **Stock** section, select **Left Spindle** and click **Properties**.

The **Machine Component Manager - Stock** dialog box opens.

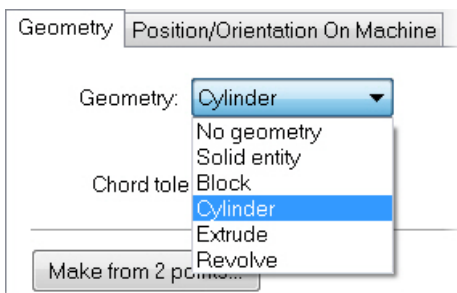


3. In the **Name** field, type **Lathe Tutorial Stock** to name the stock setup for the left spindle.

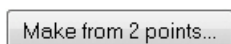


4. Choose **Cylinder** from the **Geometry** drop-down selections.

Cylinder lets you create 3D bar stock. This is suitable for most turning operations.

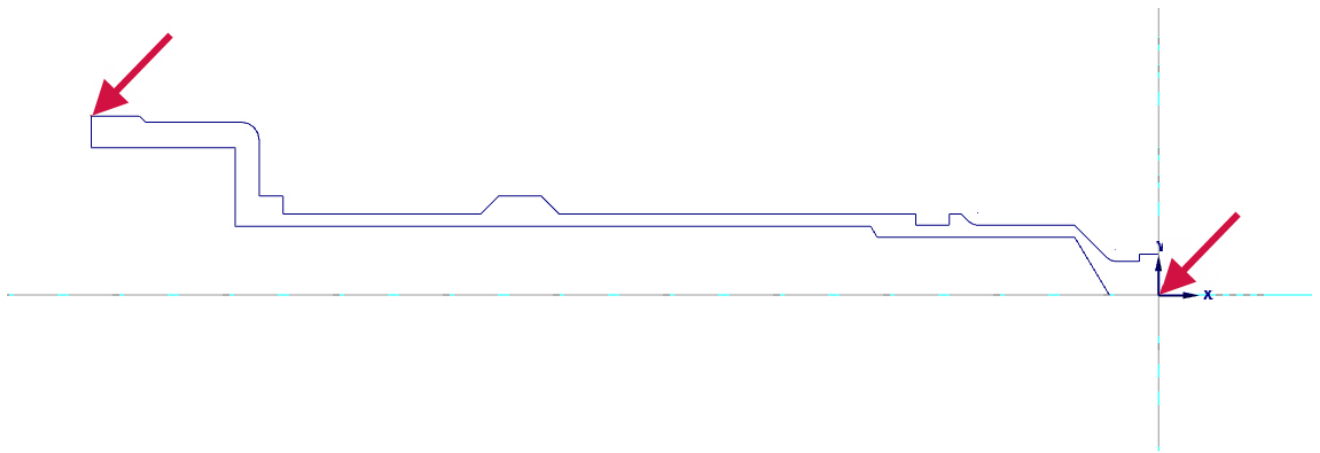


5. Click the **Make from 2 points** button.



6. In the graphics window, select the origin and the top of the part's back face.

If you have not already done so, change the GView to **TOP**.



Mastercam calculates the distance between the points and enters the values of the part's outer diameter (OD) and length directly into the fields.

move from the part

OD:

ID:

Length:

7. Add an additional **30** mm to the **Length** field.

The additional stock (about 30 percent) provides enough material for the left spindle chuck to hold the stock in place during machining.

Length:

**Note:** Fields that take number values have a built-in calculator that let you enter simple formulas directly into the field.

Length:

8. Confirm that the **Position Along Axis** is set to **0.0**, and the **Axis** field has **-Z** for the stock direction.

These two values determine the location and orientation of the cylinder. The center of the part's face lies on the origin, and the part is positioned along -Z.

Position Along Axis

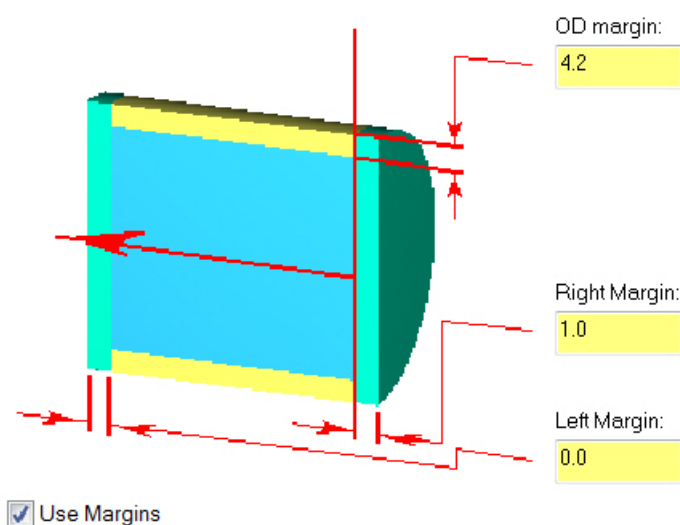
Z:

Axis:

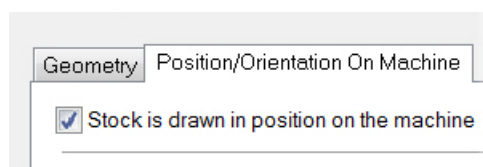
9. Select the **Use Margins** option to activate the stock margin fields.

- Enter **4.2 mm** in the OD margin field and **1.0 mm** in the Right Margin field.

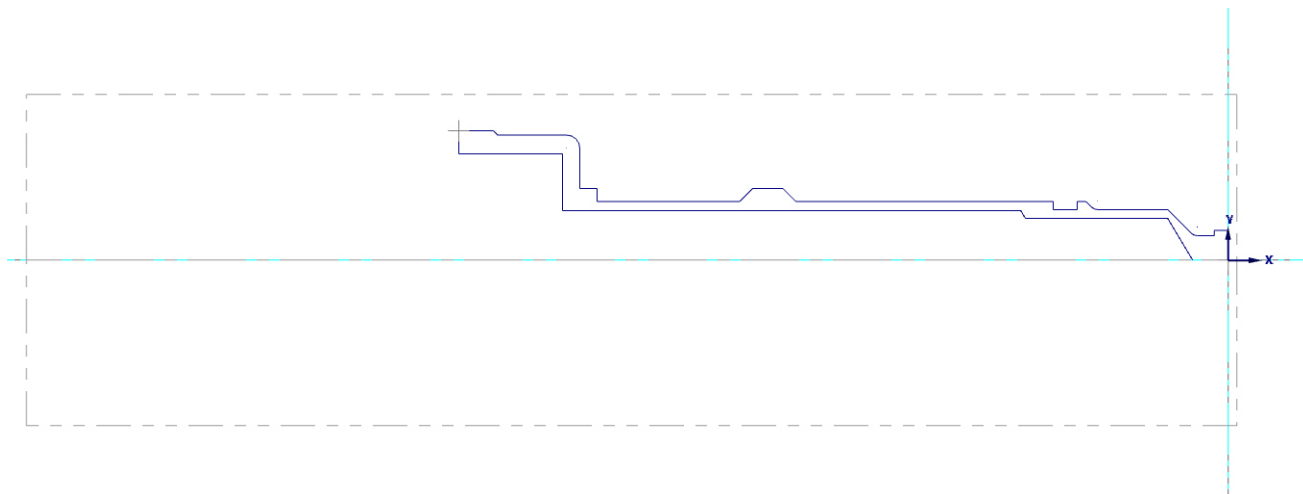
Mastercam adds the margin to the stock boundary.



10. Click the **Position/Orientation On Machine** tab, and make sure the option, **Stock is drawn in position on the machine** is selected.



11. Return to the **Geometry** tab and click the **Preview Lathe Boundaries** button to view your results.



12. Press **[Enter]** to return to the **Machine Component Manager - Stock** dialog box.
13. Click **OK** to accept your lathe stock setup settings.

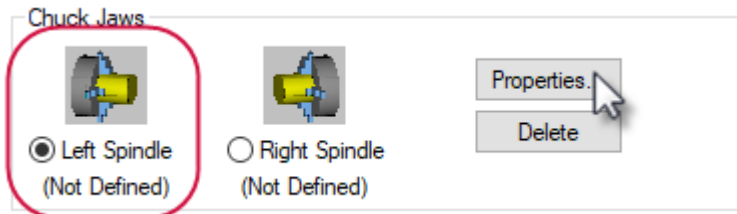
The left spindle stock is defined.



## Exercise 7: Defining the Chuck Jaws

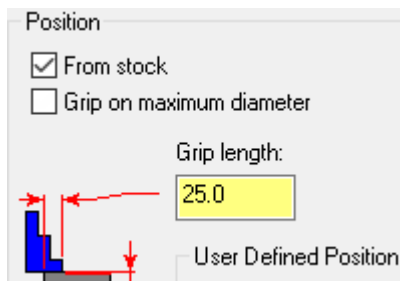
In this exercise you will define the position and grip length for the chuck jaws. You can only use the method described below after the stock has been set up.

1. In the **Chuck Jaws** section, select **Left Spindle** and click **Properties**.

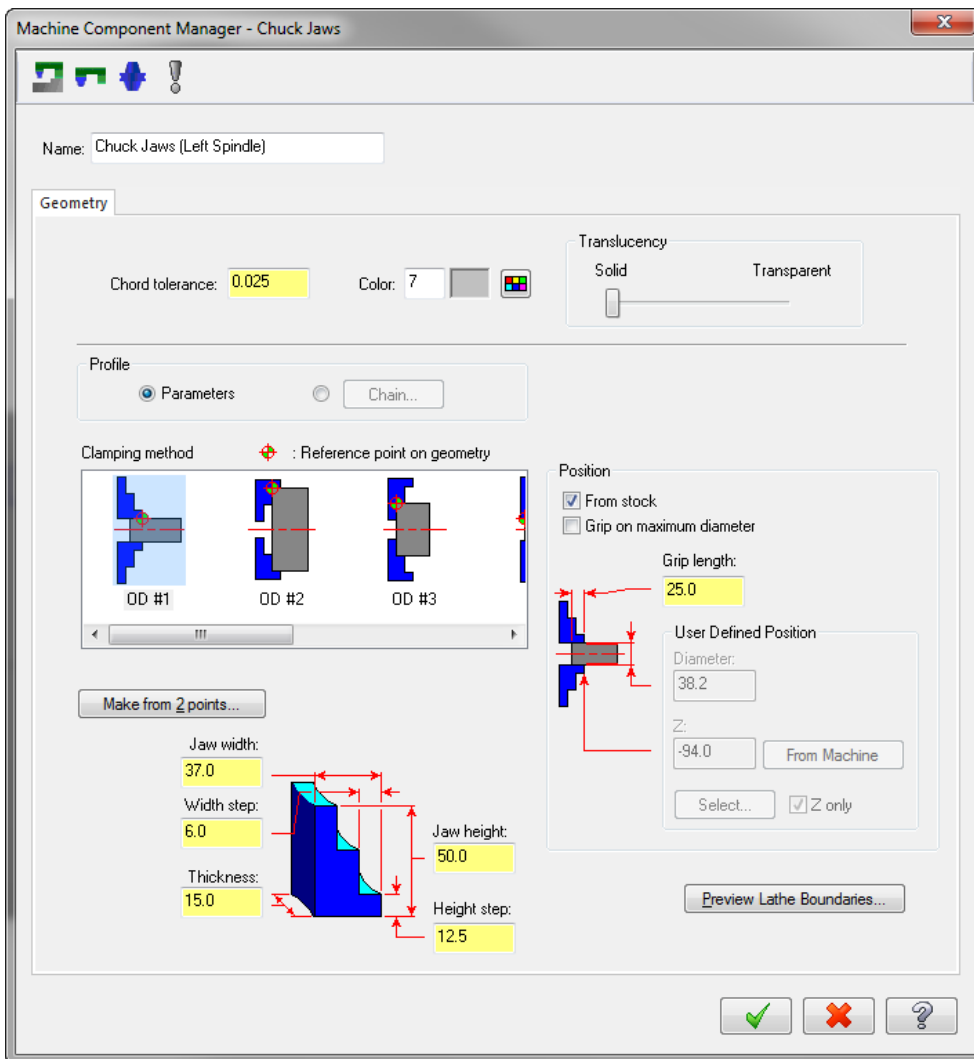


2. In the **Position** section:
  - Select **From stock**.
  - Enter **25** mm in the **Grip length** field.

Mastercam uses these settings to calculate the position of the chuck jaws relative to the stock, and to determine how much stock is being held by the chuck jaws. Since you added 30 mm of extra stock in [Exercise 6](#), a grip length of 25 mm gives enough clearance to cutoff the part.



3. Confirm that the other values in the dialog box match those in the graphic below.

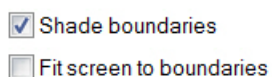


4. Click **OK** to accept these settings.

The **Stock Setup** tab displays with the left spindle's chuck jaws defined.

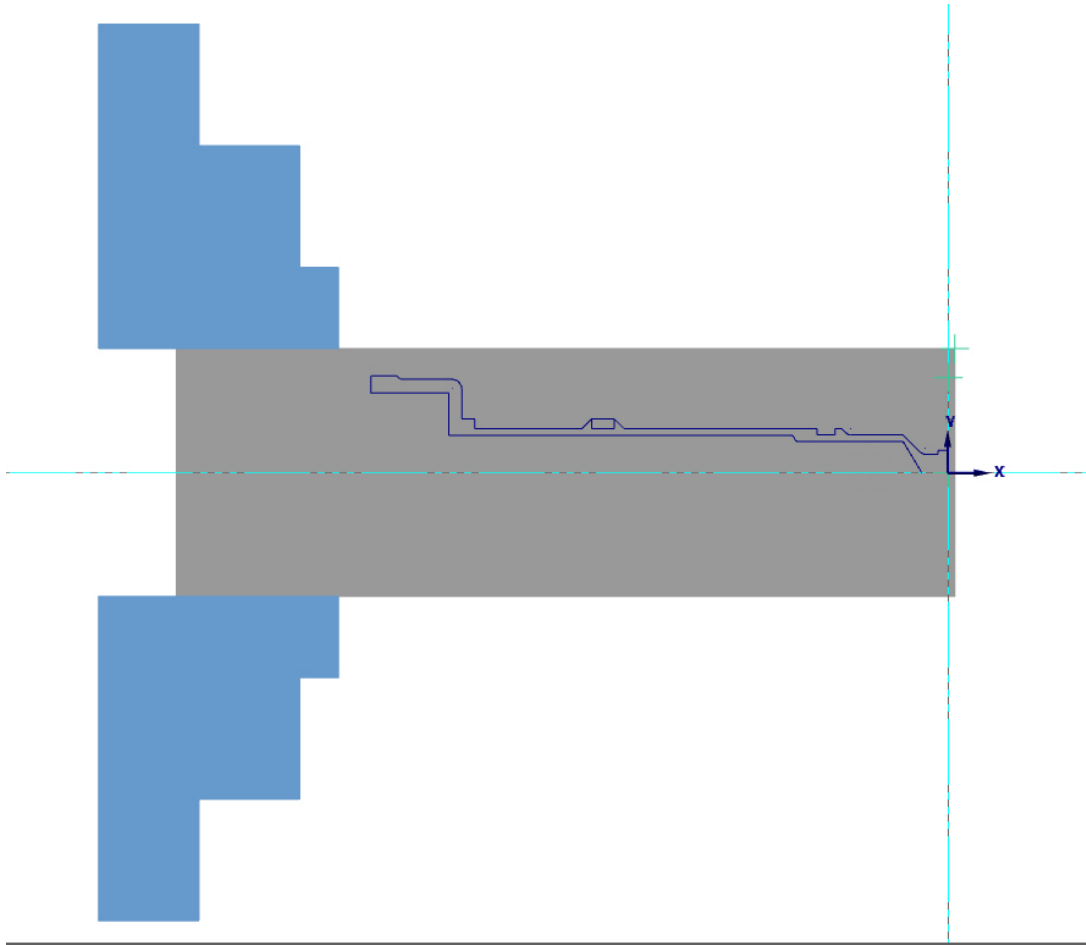


5. Select the **Shade boundaries** option to more easily see the stock boundaries and chuck jaws you have created.



6. Click **OK** to close the **Machine Group Properties** dialog box.

7. Save the file.

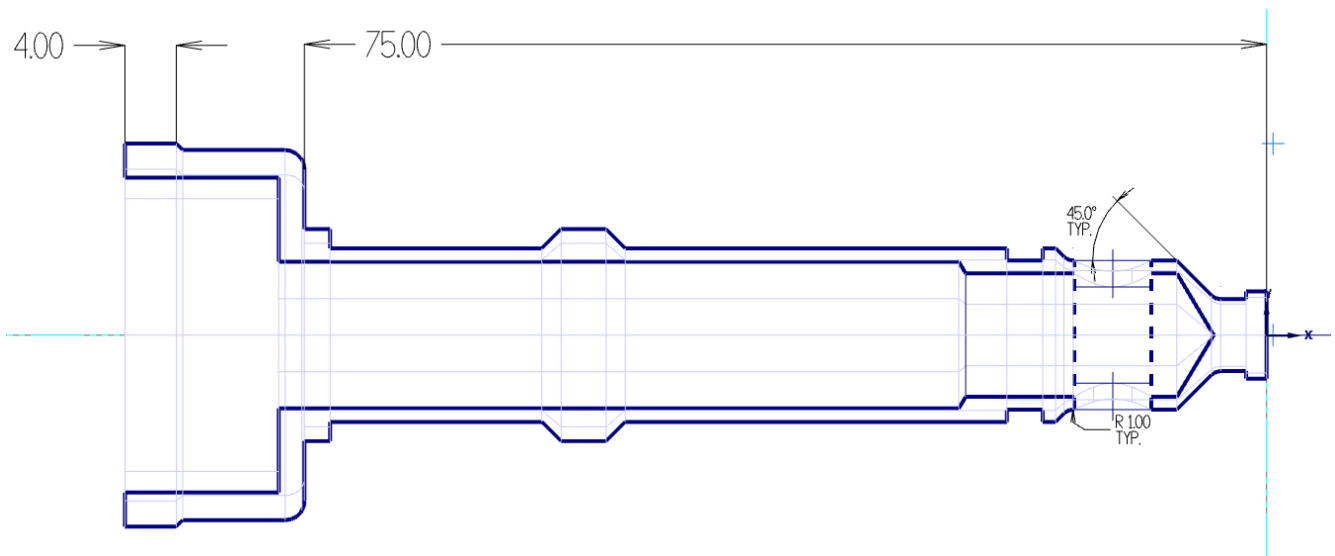


You have prepared your part. Now you can create toolpaths.

## CHAPTER 2

# FACING, ROUGHING, AND FINISHING THE OUTER DIAMETER

Once you have set up your job, you can begin creating toolpaths. Several toolpaths are normally involved in machining a lathe part. In this lesson you create the toolpaths necessary to shape the outer diameter (OD) of the part. Then, you backplot the operations you created to check your work.



### Goals

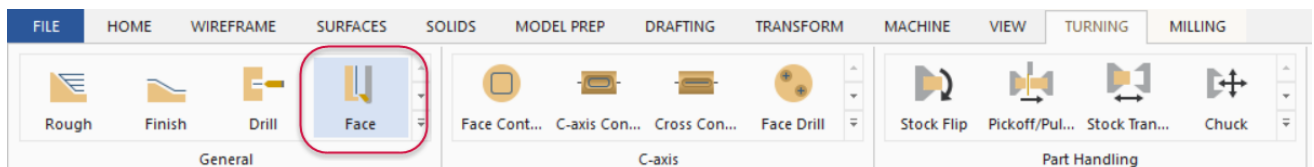
- Apply lathe toolpaths to the outer diameter of the part.
- Select tools and enter cutting values for each toolpath.
- Use Backplot to check your work.

### Exercise 1: Facing the Part

Face toolpaths prepare the face of the part for further machining. Once the face of the part is clean, you can use it to set tools or determine tool offsets.

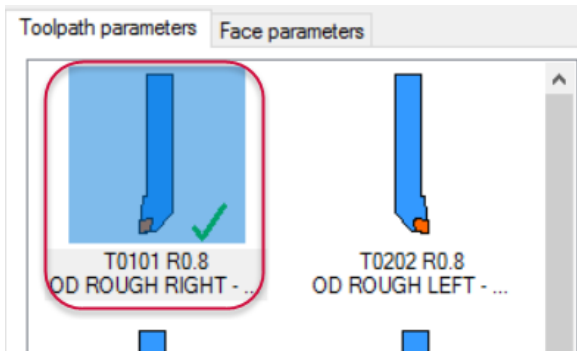
You do not need to chain geometry to create a face toolpath. Mastercam can create the toolpath entirely from parameters you enter in the **Lathe Face** dialog box.

1. Choose **Face** from the Lathe **Turning** tab.



2. From the **Toolpath parameters** tab, select the default OD roughing tool: **T0101 R0.8 OD ROUGH RIGHT - 80 DEG.**

The tool numbers called out in this tutorial are the defaults listed in the default library: `Lathe_mm.Tooldb`. Your tools may have different tool numbers.



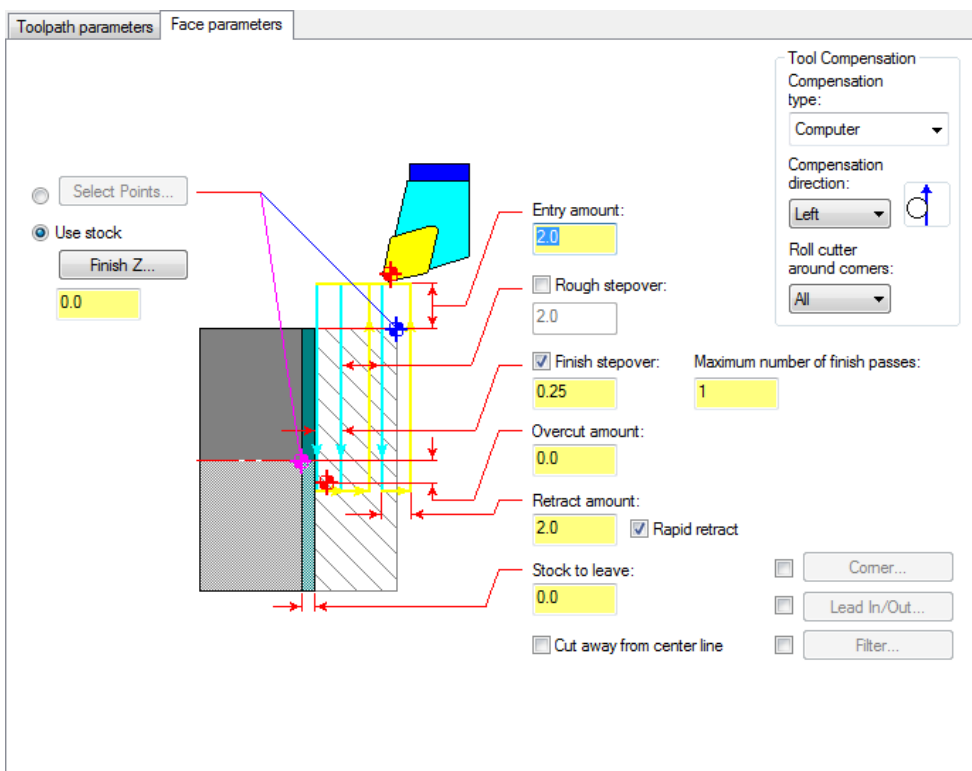
Mastercam uses different colors to represent the orientation of lathe tool inserts.

- The tool has an orange insert when the insert faces away from you (insert down).
- The tool has a yellow insert when the insert faces towards you (insert up).

3. Keep all other parameters on this page at their default values.

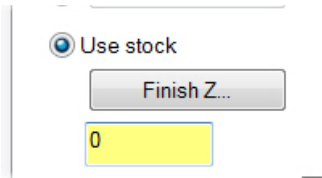
## Enter the cutting values

1. Click the **Face parameters** tab.



2. Confirm that **Use stock** is selected.

**Use stock** is available only if you have defined the stock boundary in **Stock Setup**. (See ["Setting Up the Stock in the Left Spindle"](#) on page 18.)



3. If necessary, enter **0** in the text box, or click **Finish Z** and select the origin from the graphics window to place the finished face at the origin.
4. Keep all other parameters on this page at their default values.
5. Click **OK** to create the toolpath.

**Note:** If you use stock for the start and end positions of each pass and the stock changes, the start and end positions of each pass are automatically updated when you regenerate the toolpath.

## Exercise 2: Roughing the Outer Diameter

Use rough toolpaths to quickly remove large amounts of stock in preparation for a finish pass. Roughing passes are typically straight cuts parallel to the Z-axis.

Mastercam includes several types of roughing toolpaths:

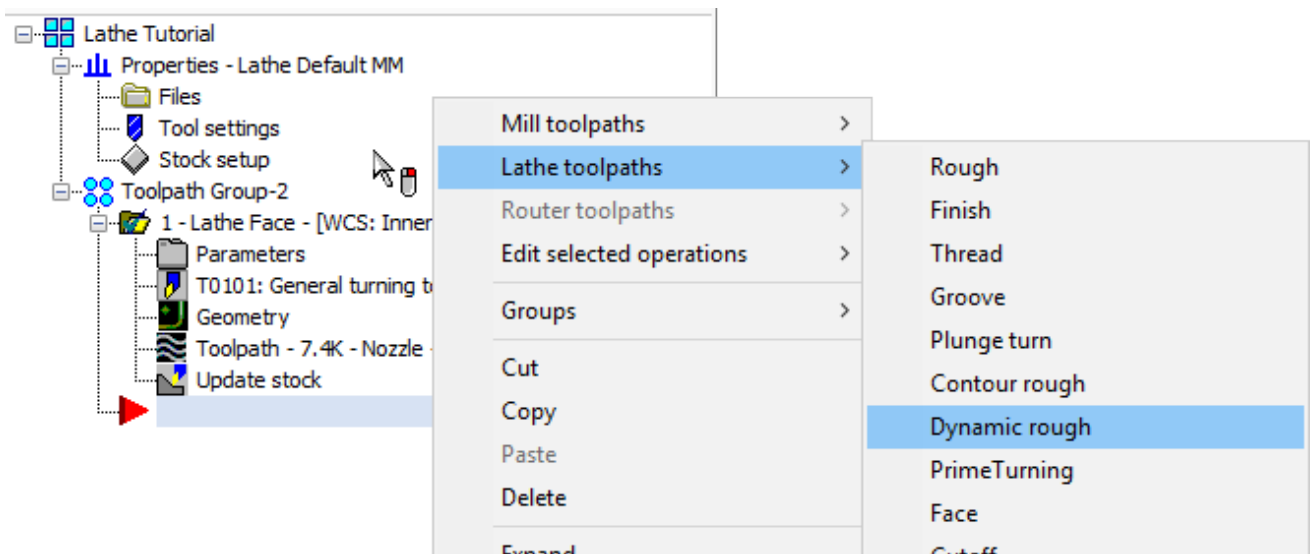
- Standard rough toolpaths, which let you access all of Mastercam's roughing options.
- Canned rough toolpaths, which use your machine tool's canned cycles to create the most efficient code (however, these do not offer as many options as the standard rough toolpaths).
- Canned pattern repeat toolpaths, which create roughing passes in the shape of the part contour, rather than cutting parallel to the Z-axis.
- Dynamic rough toolpaths, which remain engaged in the material more effectively, and use more of the surface of your insert, extending the tool life and increasing the cutting speed.
- Contour rough toolpaths, which are useful for parts where the initial stock shape is similar to the final part shape, such as using a casting for stock.

In this exercise, you create a dynamic rough toolpath with a semi-finish pass.

Click [here](#) to see the difference between a dynamic rough toolpath and a standard rough toolpath.

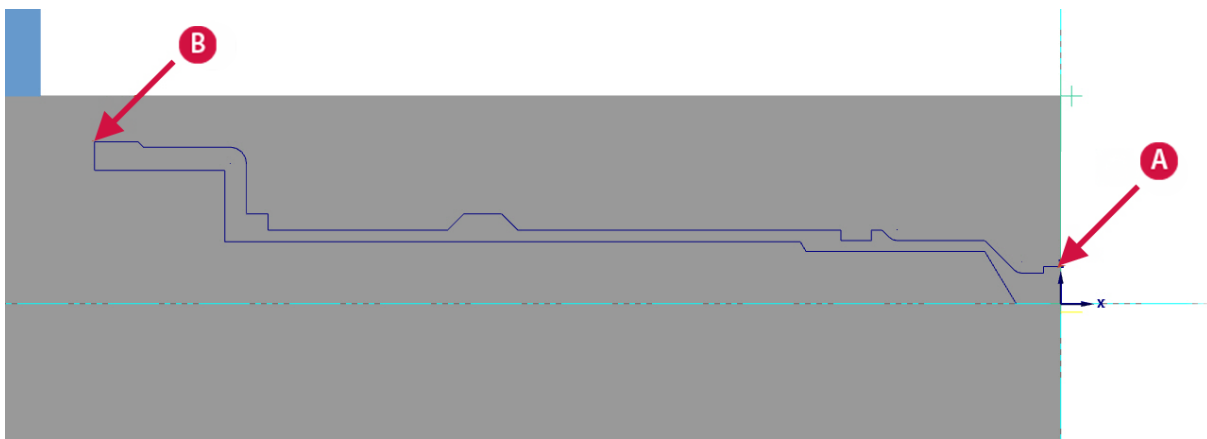
### Chain the geometry

1. Right-click in the Toolpaths Manager. Select **Lathe toolpaths, Dynamic rough**.  
The **Chaining** dialog box displays, and you are prompted to select an entry point.  
Mastercam Lathe defaults to partial chaining.



**Note:** Access an extensive list of commands from the Toolpaths Manager's right-click menu.

- Click the geometry at the first entity (a) and then, at the last entity (b) to create a partial chain.



- Click **OK** in the **Chaining** dialog box to accept the chain.
- The **Lathe Dynamic Rough** dialog box opens.

## Enter the toolpath parameters

Like many other toolpaths in Mastercam, you create the Dynamic Rough toolpath by entering tool and cutting values.

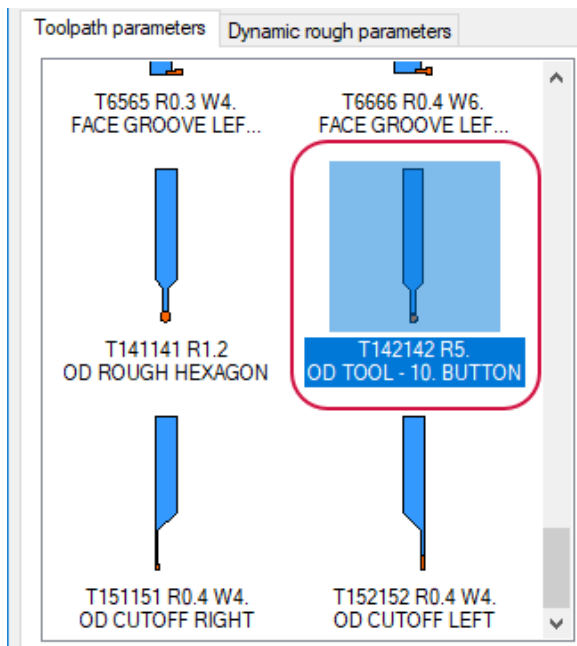
### Select the tool

Use the **Toolpath parameters** tab to select a tool, set feeds and speeds, and modify other general toolpath parameters. This tab is similar for most Lathe toolpaths.

The tool numbers called out in this tutorial are the defaults listed in the default library: `Lathe_mm.Tooldb`. Your tools may have different tool numbers.

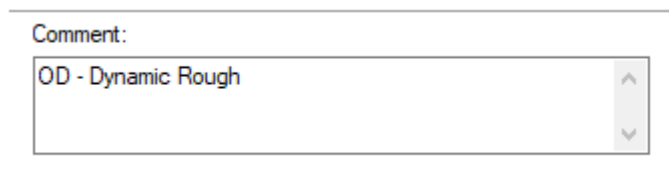
1. From the **Toolpath parameters** tab, select the OD roughing tool: **T142142 R5 OD TOOL - 10. BUTTON**.

**Note:** Lathe dynamic rough only supports button tools.



2. Type **OD - Dynamic Rough** in the **Comment** field.

Comments help identify the operation in the Toolpaths Manager. They can be output to the NC file when you post.

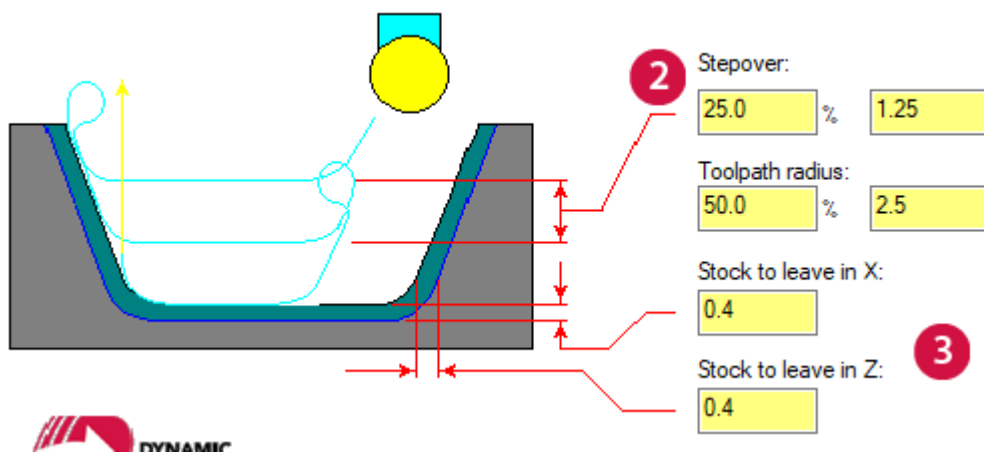


**Note:** Comments are optional. In this tutorial, not all operations will have comments.

3. Keep all other parameters on this page at their default values.

Enter the cutting values

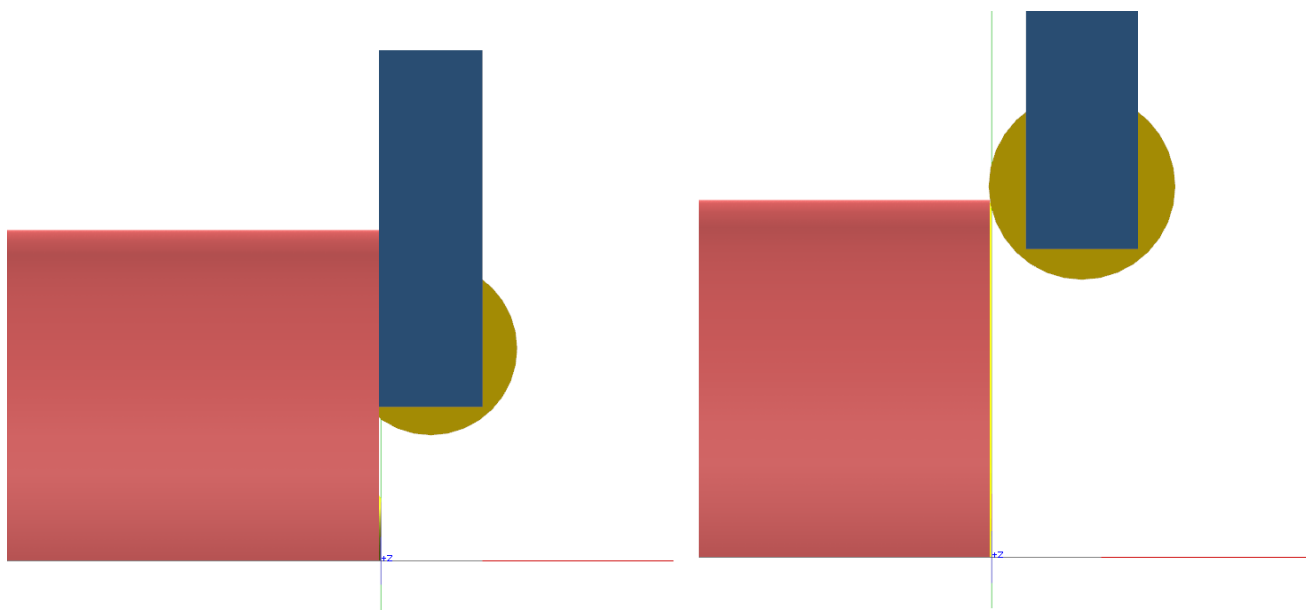
1. Click the **Dynamic rough parameters** tab.
2. Decrease the **Stepover** value to **25%**.
3. Increase the **Stock to leave** in **X** and **Z** to **0.4**.



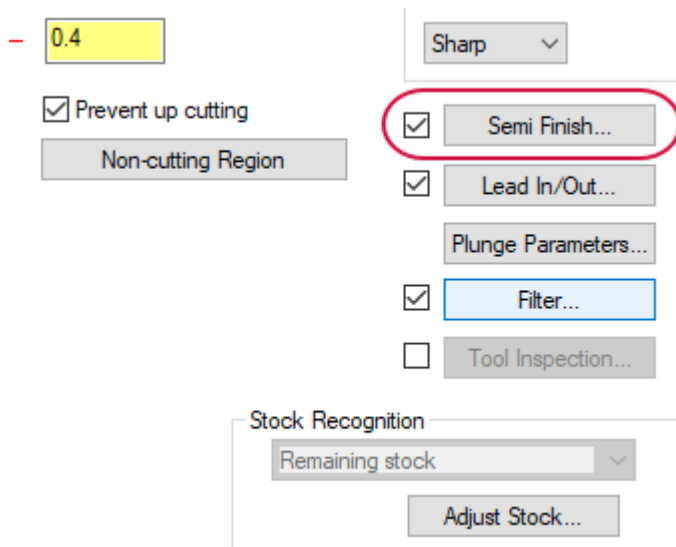
4. Select the **Prevent up cutting** option, and click the **Non-cutting Region** button to view these parameters.

**Prevent up cutting** lets you specify the non-cutting portion of a round insert. When **Prevent up cutting** is selected, and **Angle** is set to **0** degrees, Mastercam cuts only with the lower portion of the tool insert.

The image on the left shows the part with **Prevent up cutting** turned off. The tool shank crashes into the material as it tries to cut the toolpath. The image on the right shows the part with **Prevent up cutting** turned on. The tool cuts only with the specified portion.



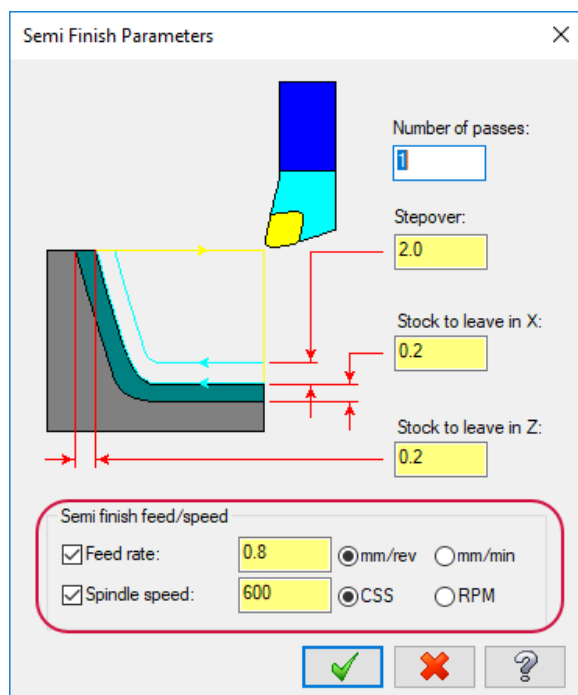
5. Close the **Non-cutting Region** dialog box after you have viewed it. If the **Angle** is set to **0**, you do not need to modify any parameters.
6. Select the **Semi Finish** option and click the button to open the **Semi Finish Parameters** dialog box.



The arc motion of the dynamic toolpath can leave an uneven surface. After the rough passes, the semi-finish pass follows the contour of the part with the roughing tool, smoothing and preparing it for the finish operation.

- a. Increase the semi-finish **Feed rate** to **0.8 mm/rev**.
- b. Increase the semi-finish **Spindle speed** to **600 CSS**.

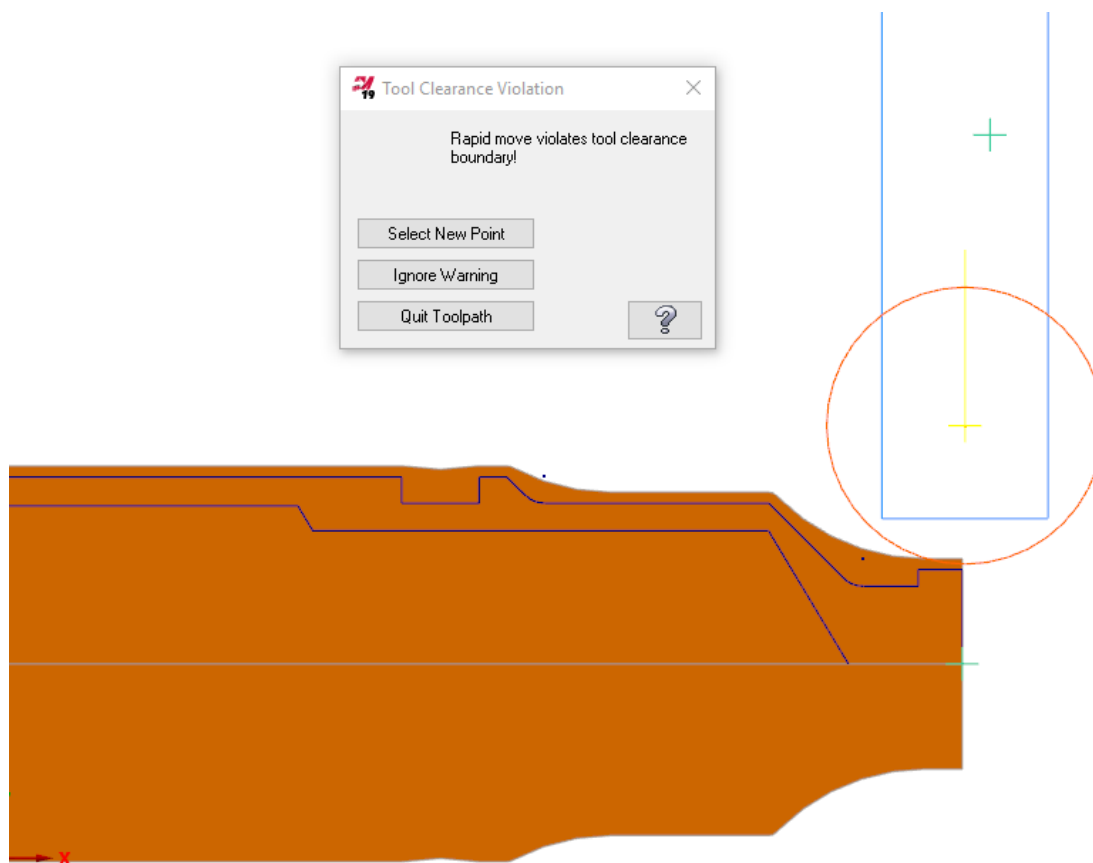
**Note:** Click **OK** if the **Tool Settings Modified** message displays.



- c. Click **OK** to return to the **Lathe Dynamic Rough** dialog box.

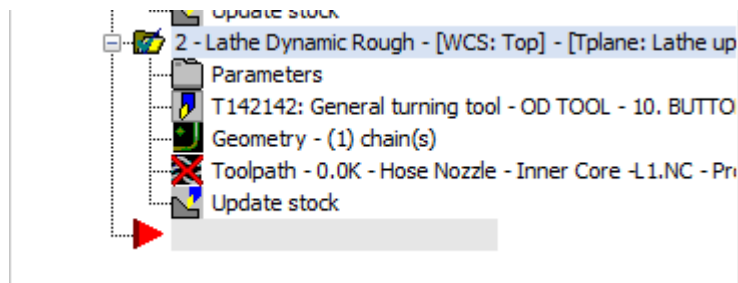
7. Click **OK** to create the toolpath.

Mastercam detects that the tool is approaching the stock closer than the specified clearance distance, stops the operation, and displays the **Tool Clearance Violation** warning. Additionally, the graphics window displays where the tool collides with the stock.



8. Select the option to quit the toolpath, and choose to keep the operation when you are prompted.

Mastercam marks the attempted Dynamic Rough operation as dirty.



9. Click the **Parameters** folder to reopen the **Lathe Dynamic Rough** dialog box and modify the operation.
10. Click the **Lead In/Out** button.

The **Lead In/Out** dialog box opens.

The parameters in this dialog box control how the tool approaches and/or retracts from the part for each pass in the toolpath. This eliminates the need to create extra geometry for this purpose.

- a. Click the **Lead in** tab.

The **Lead in** and **Lead out** tabs have identical options for creating entry and exit moves. This allows you to set different values for each move and to combine different types of moves.

Lead in    Lead out

**Adjust Contour**

☐ Extend / shorten start of contour

Amount:     ☒ Extend    ☐ Shorten

☐ Add Line...

☐ Entry Arc...

**Feed rate**

   ☒ mm/rev    ☐ mm/min

☐ Use rapid feed rate for vector moves

☒ Same as toolpath

**Entry Vector**

☒ Use entry vector

**Fixed Direction**

☒ None    ☐ Tangent    ☐ Perpendicular


Angle:     Length:     Resolution (deg.):

IntelliSet

**Auto-calculate vector**

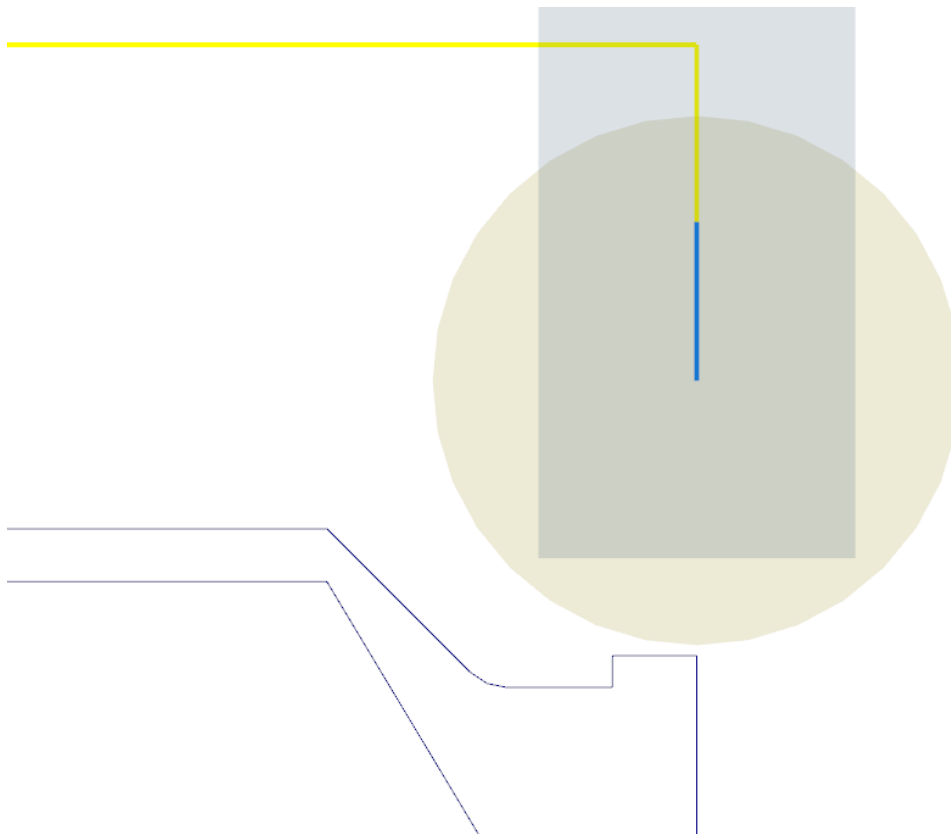
☐ Automatically calculate entry vector

Minimum vector length:



- b. Enter **-90** in the **Angle** field, and increase the **Length** of the vector to **3.0**.

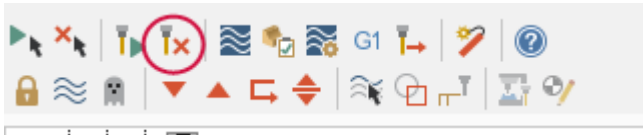
This is the angle the tool follows as it approaches the part and the distance it travels as it moves towards the part. Modifying the entry vector will keep the rapid motion of the tool from violating the clearance as it begins the semi-finish.



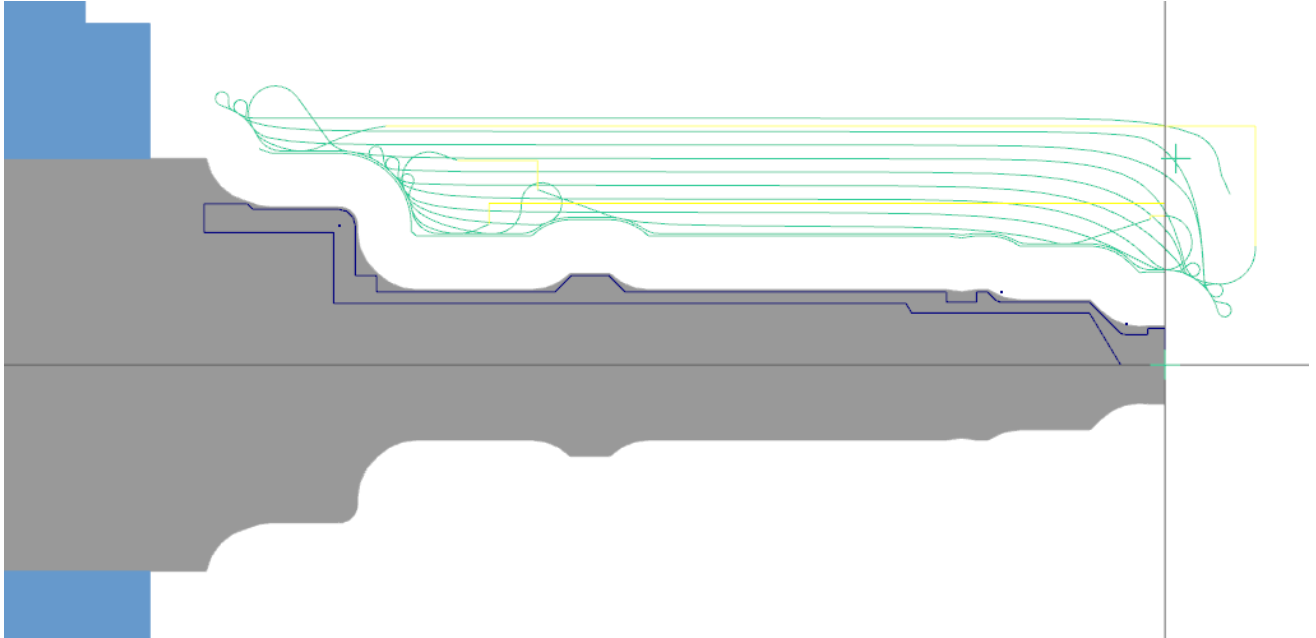
- c. Click **OK** to return to the **Lathe Dynamic Rough** dialog box.

11. Keep all other parameters on this page at their default values, and click **OK**.

12. Regenerate the toolpath, and save your part.



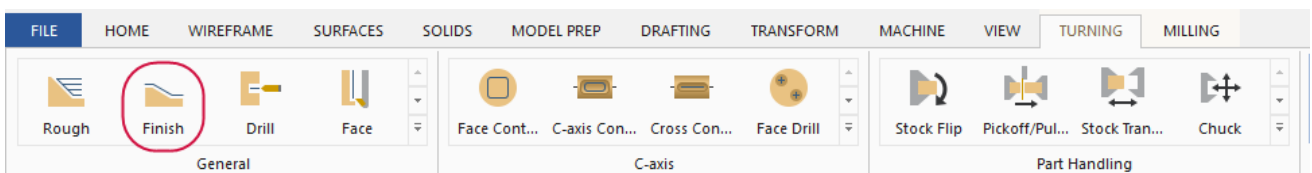
The graphics window displays the Dynamic Rough toolpath.



## Exercise 3: Finishing

Use finish toolpaths to have the tool follow the contour of chained geometry. Typically, a finish toolpath follows a roughing toolpath.

1. Choose **Finish** from the **Lathe Turning** tab.



The **Chaining** dialog box displays, and you are prompted to select a point or chain a contour.

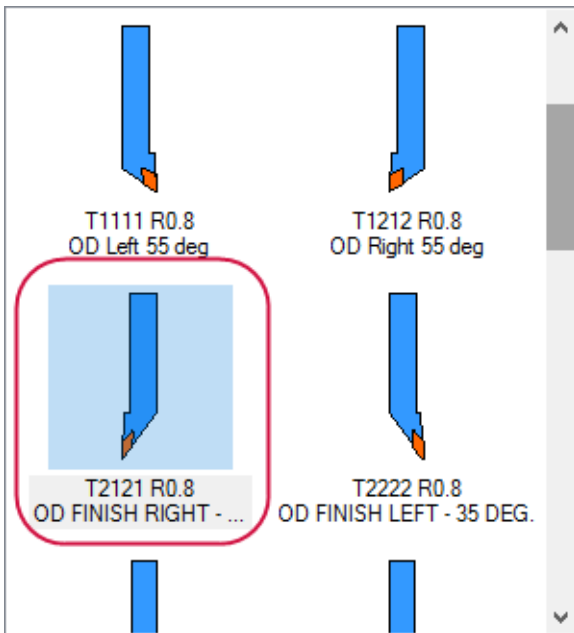
2. Select the same geometry that you selected for the Dynamic Rough toolpath. (See ["Chain the geometry" on page 27.](#))
3. Click **OK** in the **Chaining** dialog box to accept the chain.

The **Lathe Finish** dialog box opens.

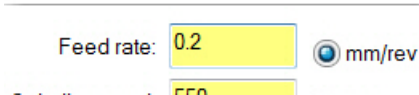
## Enter the toolpath parameters

Entering the toolpath parameters for any finish toolpath uses the same workflow as other lathe toolpaths: first you select the tool and tool options, and then you enter the cutting values.

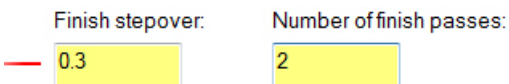
1. Select the finishing tool: **T2121 R0.8 OD FINISH RIGHT - 35 DEG.**



2. Decrease the feed rate to **0.2**.



3. Keep all other parameters on this page at their default values.
4. Click the **Finish parameters** tab.
5. Change the **Finish stepover** to **0.3** and the **Number of finish passes** to **2**.



The operation makes two, finer finish passes and avoids excessive cut depths.

6. Leave all other parameters on this page at their default values.
7. Click **OK**.

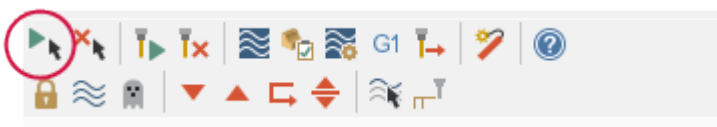
Mastercam creates a finish operation on top of the previous roughing operation.

8. Save the part.

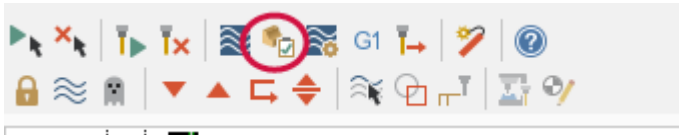
## Exercise 4: Backplotting the Toolpaths

In this exercise, you use Backplot to check the work that you have done. Backplotting shows the path the tools take to cut your part and lets you spot errors in the program before you send it to the machine.

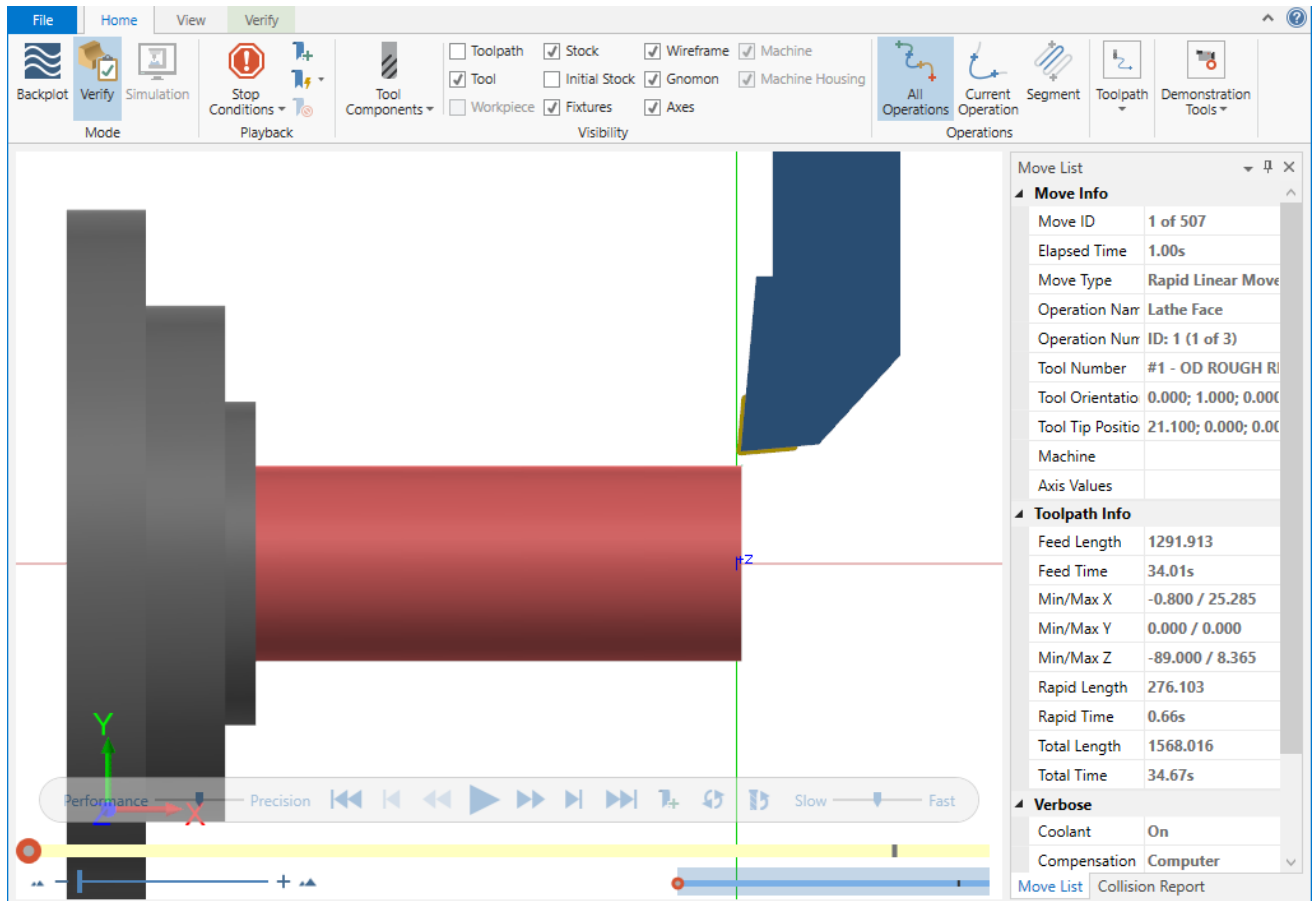
1. Click the **Select all operations** button in the Toolpaths Manager.



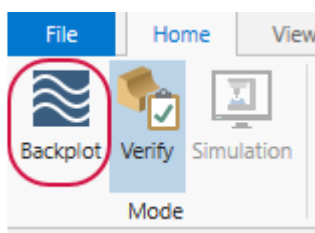
- Click the **Verify selected operations** button.



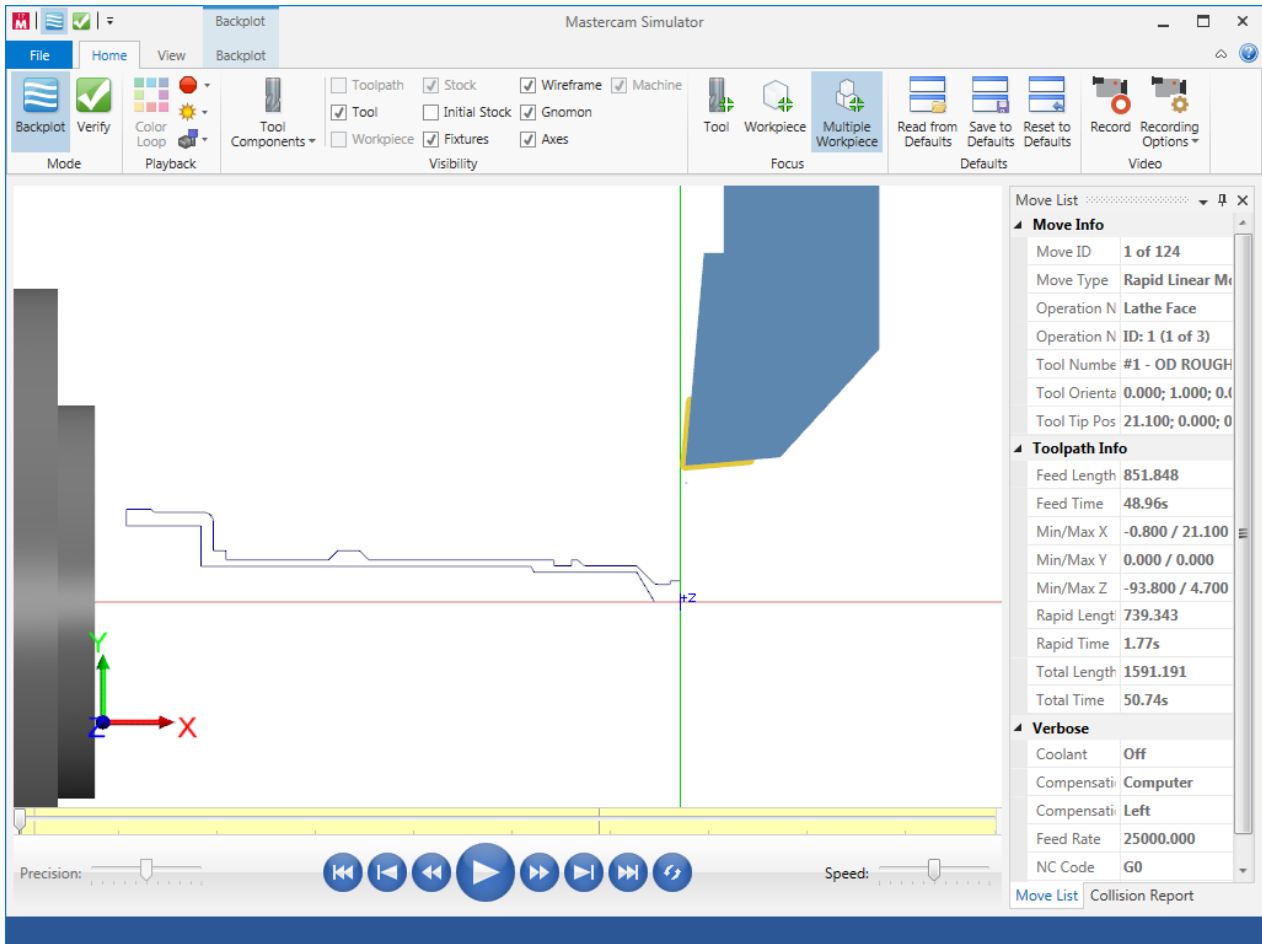
Mastercam Simulator opens a view of the part in Verify mode in a separate window.



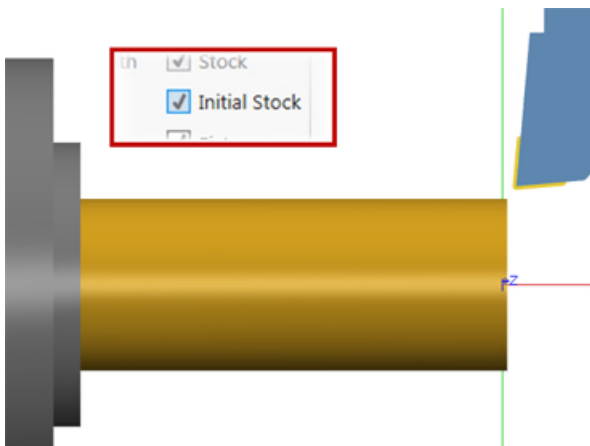
- Click **Backplot** on the **Home** tab.



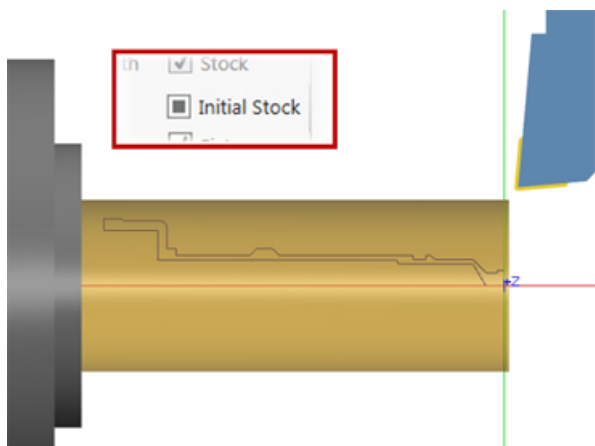
The Simulator window changes to Backplot mode.



4. Select the **Initial Stock** option in the ribbon bar's **Visibility** group to see the stock before machining.



5. Select the **Initial Stock** option again to view the part profile against a translucent display of the initial stock.



6. For a better view of the operations in the Mastercam Simulator, use the **Page Up** key to zoom into the part and **[Ctrl + Arrows]** to pan the view.

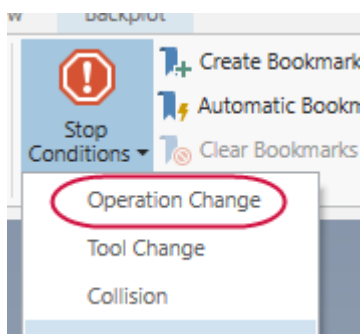
The views in Mastercam and Mastercam Simulator windows can be synchronized. Turn this option on or off from the **View** tab of Mastercam Simulator.

7. Click the **Play** button at the bottom of the Simulator screen.



Simulator backplots all three toolpaths with information about the current toolpath motion displayed in the **Move List** on the right-side of the screen.

8. To pause the simulator at the end of each operation, select **Operation Change** from the **Stop Conditions** drop-down.



9. Use the Play **[R]**, Step Forward **[S]**, and Step Backward **[B]** keys to view the operations again at your own pace.  
There are several shortcut keys in Mastercam Simulator that can make your workflow more efficient. See Help for a complete list.
10. Minimize the Mastercam Simulator, or move it to another monitor.
  - You can dock the Mastercam Simulator on a second monitor, and refresh your toolpaths as you make changes.
  - When you close the Mastercam Simulator window, the current layout is saved and used at the application's startup.

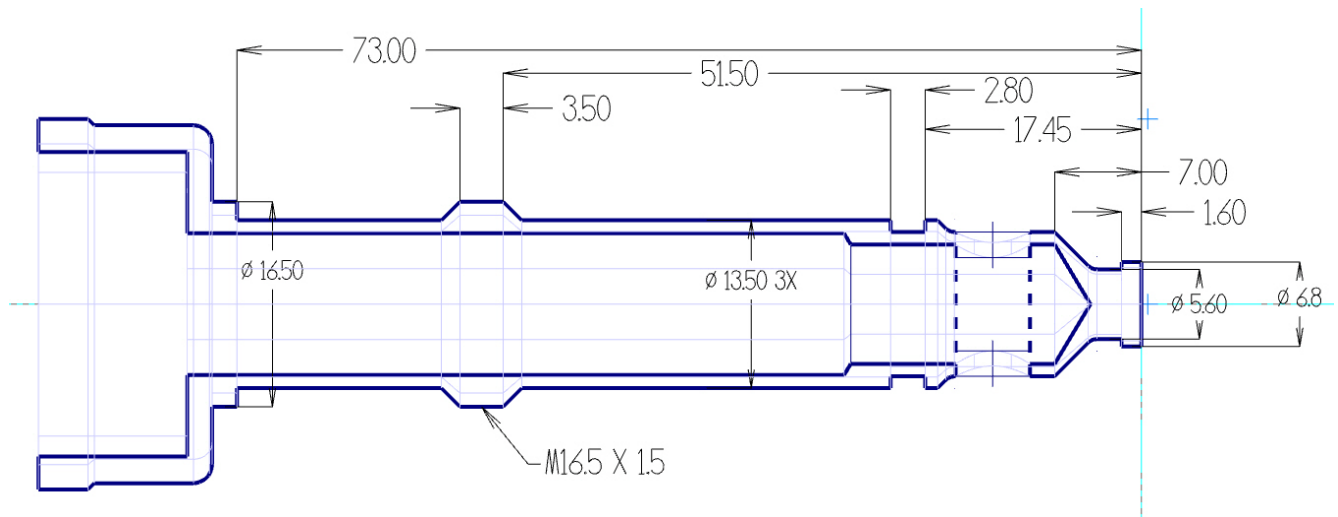
The Face, Dynamic Rough, and Finish operations reveal the basic shape of the outer diameter of the hose nozzle. In the next lesson, you further refine the outer diameter with grooves and threads. You also check your work against Mastercam Simulator's Verify option.



## CHAPTER 3

# ADDING GROOVES AND THREADS

Grooves and threads are common features on a lathe part. A groove is an indented or recessed area in a work-piece. A thread is the helical ridge of a screw. Both groove and thread toolpaths require that you define the shape and orientation of the operation as well as their cutting values. In this lesson, you create groove and thread toolpaths on the outer diameter of the part. You also see how you can employ plunge parameters to efficiently remove material in a finish operation. To analyze the viability of these operations, you use advanced verification features in the Mastercam Simulator.



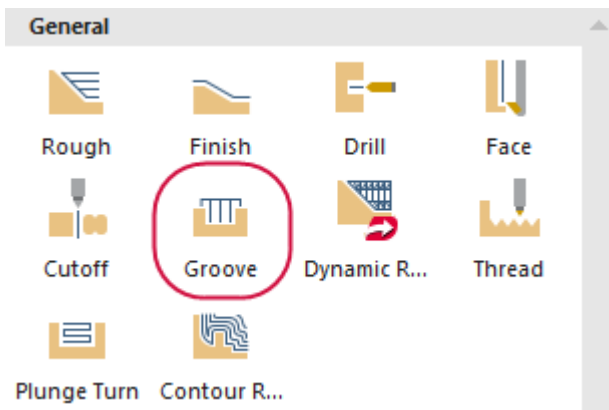
### Goals

- Create groove operations on a single chain and on non-adjacent geometry.
- Use plunge cuts in a finish operation to efficiently remove material.
- Manually enter thread parameters to create a thread on the outer diameter.
- Use Verify and its functions to check your work.

### Exercise 1: Grooving on the Outer Diameter: Multiple Chains

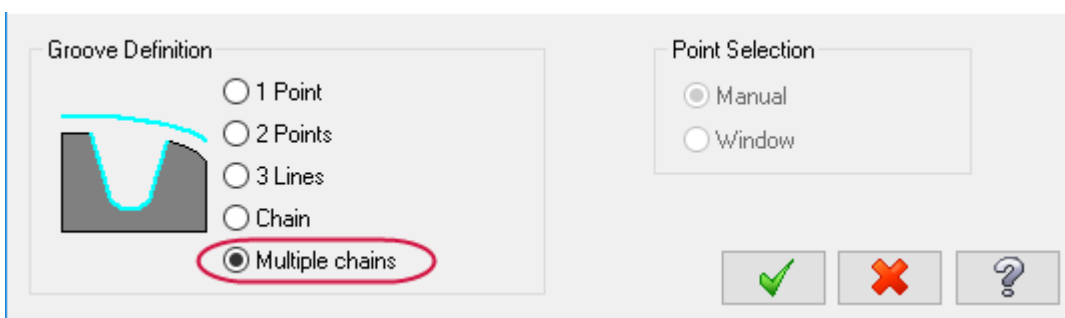
Groove toolpaths are useful for machining indented or recessed areas that are not otherwise machinable by roughing toolpaths or tools. You can machine several grooves in a single operation, even if their geometry never connects. In this exercise, you create one groove operation from two distinct chains.

1. Expand the **General** gallery, and select **Groove** from the **Lathe Turning** tab.



The **Grooving Options** dialog box opens.

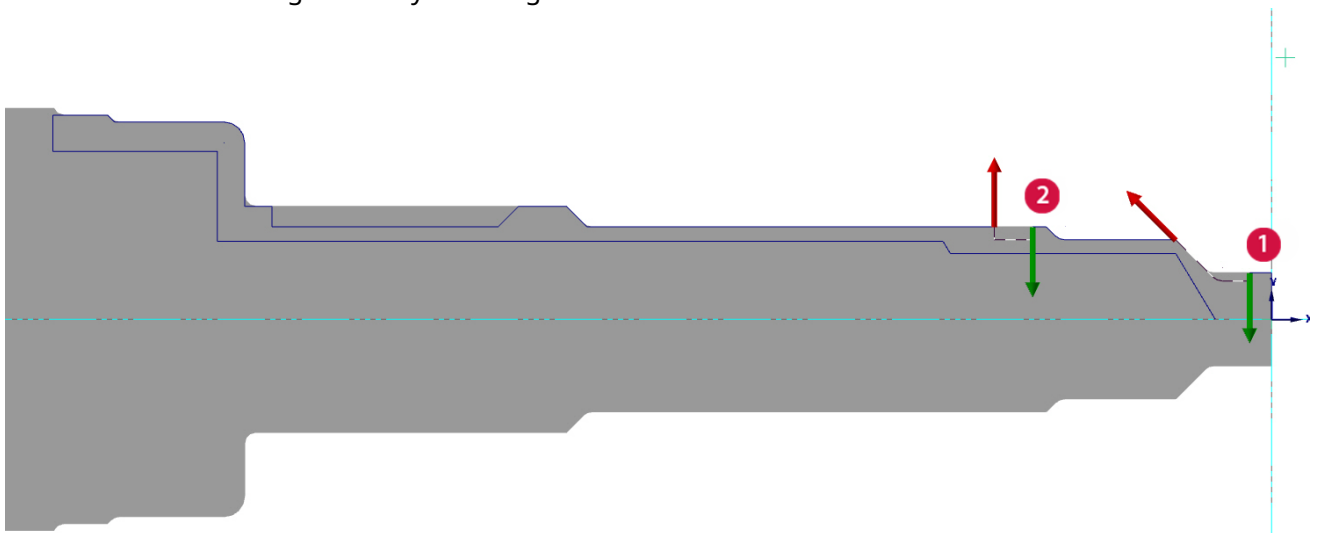
2. Select the **Multiple chains** option.



3. Click **OK**.

The **Chaining** dialog box displays and you are prompted to select an entry point.

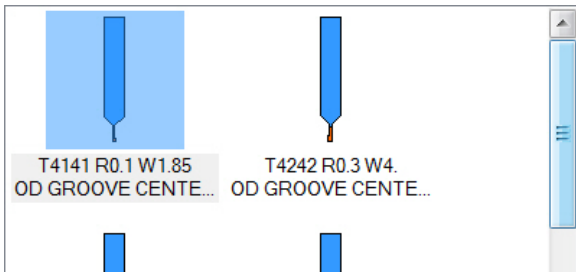
4. Chain two of the three grooves by selecting the first and last entities of each chain as shown in the illustration.



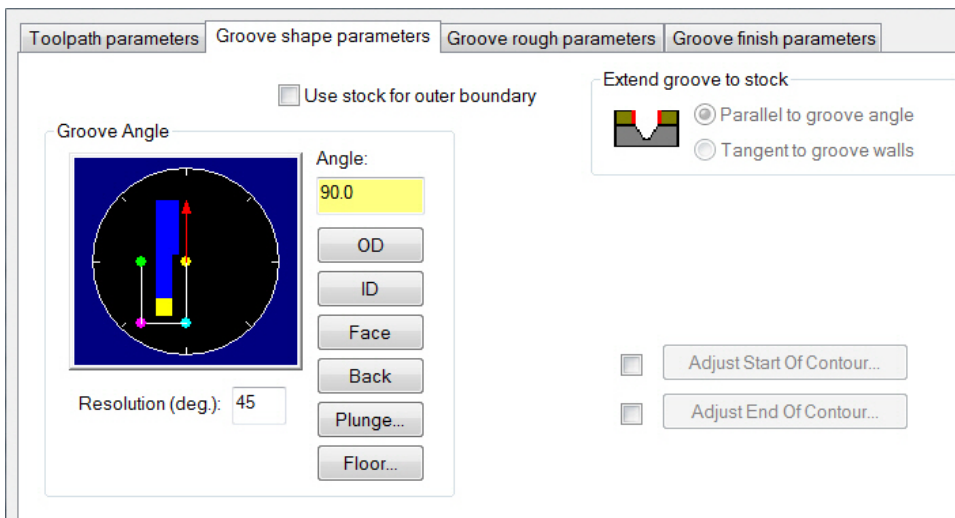
- Right-click in the graphics window, and use **Zoom Window** to more easily select the groove geometry.
- The shortcut **[Alt + T]** toggles the toolpath geometry display.

5. Click **OK** in the **Chaining** dialog box to accept the chains.

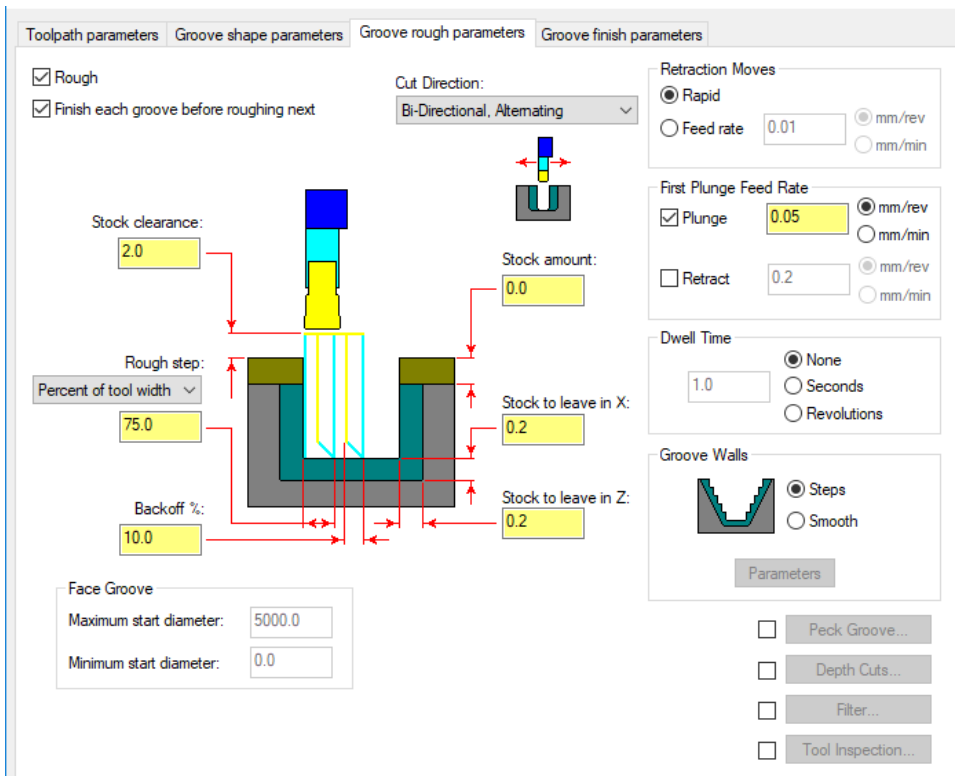
6. From the **Toolpath parameters** tab, select the grooving tool: **T4141 R0.1 W1.85 OD GROOVE CENTER - NARROW.**



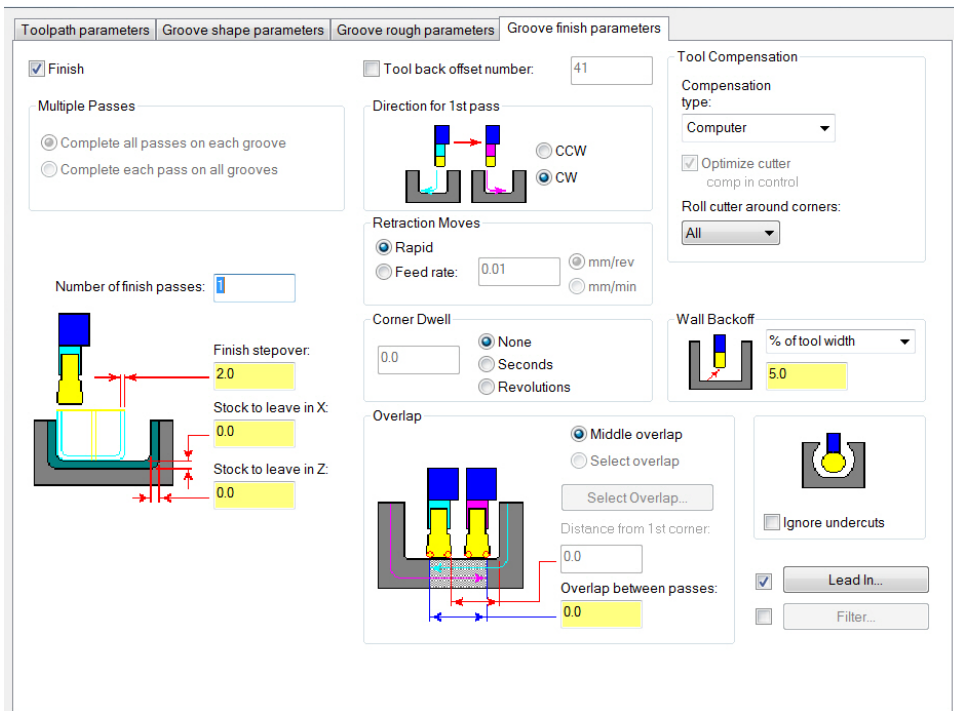
7. Keep all other parameters on this page at their default values.
8. Click the **Groove shape parameters** tab, and make sure that all values are as shown.



9. Click the **Groove rough parameters** tab. Confirm that all values are as shown.

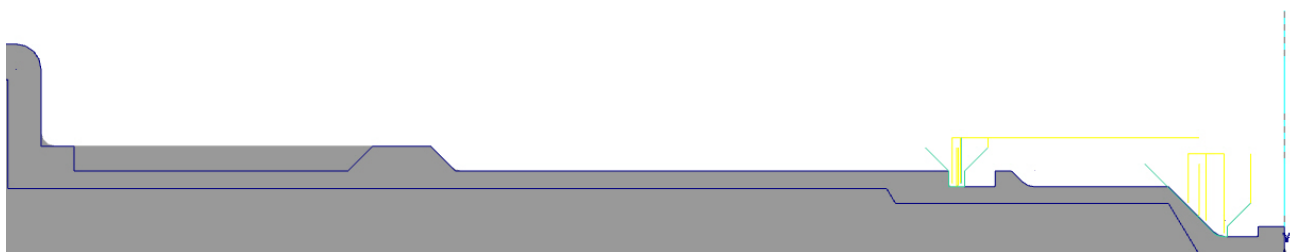
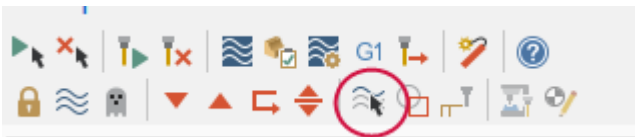


10. Click the **Groove finish parameters** tab. Confirm that all values are as shown.



11. Click **OK** to create the toolpath.

12. Click **Only display selected toolpaths** to view the toolpath geometry.



## Exercise 2: Grooving on the Outer Diameter: Rough Pass Only

In this exercise, you restrict a second groove toolpath to machine just a rough pass on a single, recessed area.

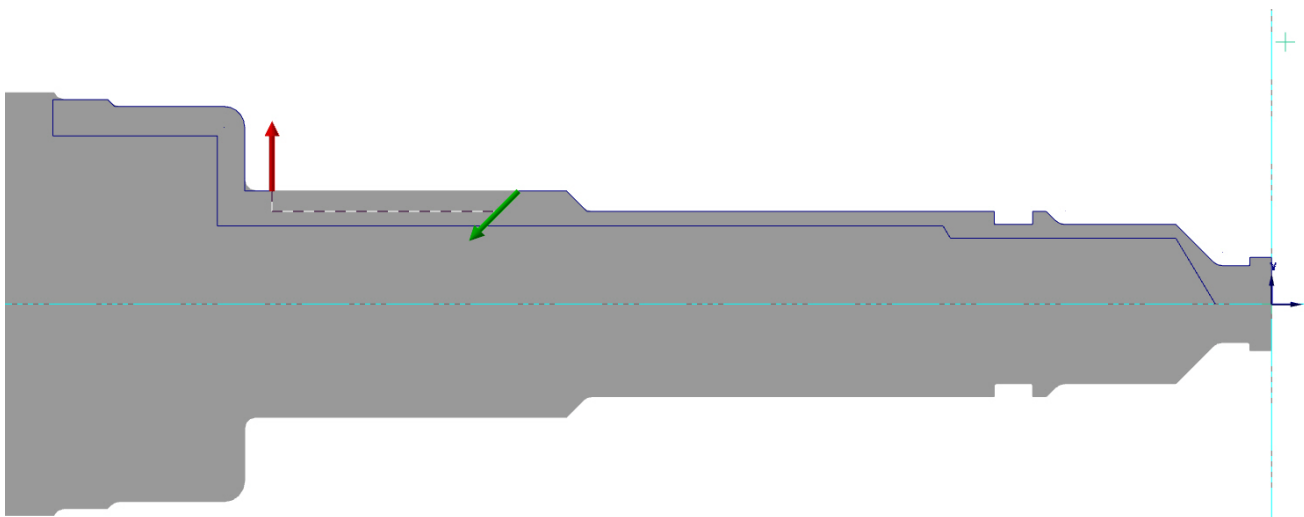
1. Select **Groove** from the **Lathe Turning** tab.
2. Select the **Chain** option from the **Grooving Options** dialog box.



3. Click **OK**.

The **Chaining** dialog box displays and you are prompted to select an entry point.

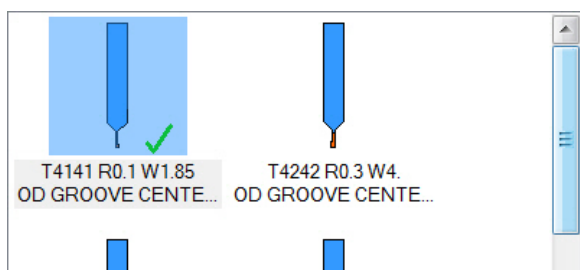
4. Chain the third groove by selecting the first and last entity of the chain as shown in the illustration, and click **OK**.



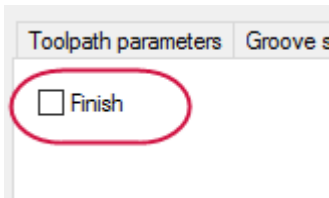
5. Click **OK** to accept the chain.

The **Lathe Groove (Chain)** dialog box opens.

6. Keep **T4141 R0.1 W1.85 OD GROOVE CENTER - NARROW** selected.

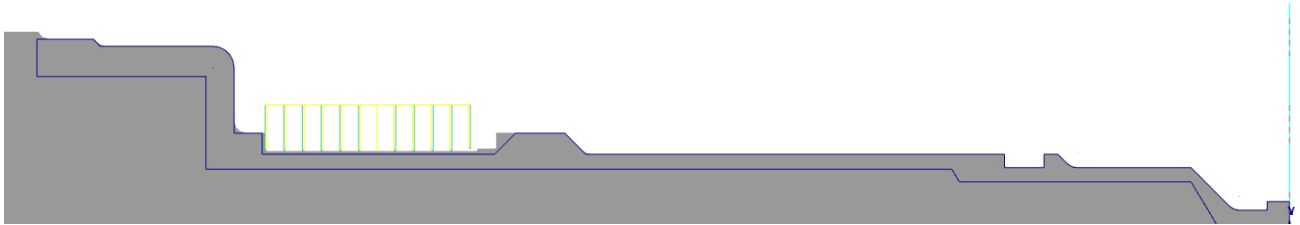


7. Open the **Groove finish parameters** page, and deselect the **Finish** option.



In the next exercise, you use a standard finish operation to remove remaining stock from this groove.

8. Click **OK** to create the toolpath.

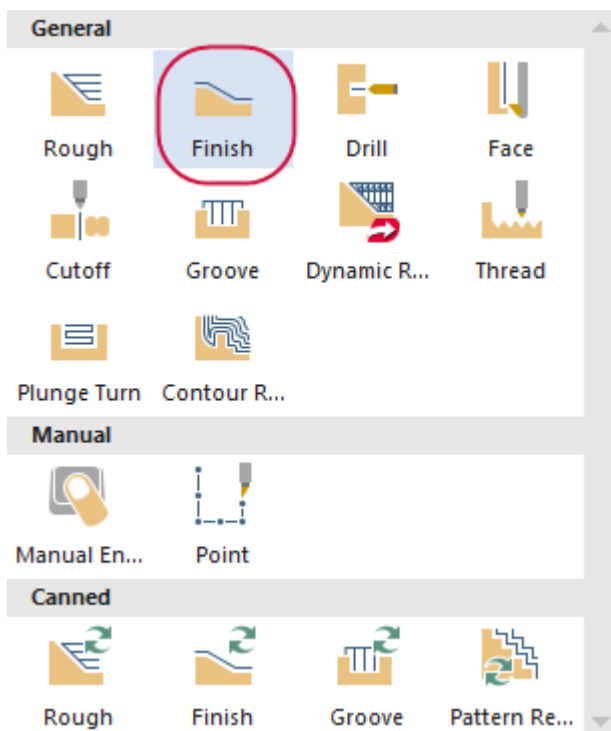


### Exercise 3: Finishing with Plunge Cuts

In this exercise, you finish the groove you created. To more efficiently remove the material, activate plunging in both directions.

1. Expand the **General** gallery and select **Finish**.

The **Chaining** dialog box displays, and you are prompted to select a point or chain a contour.

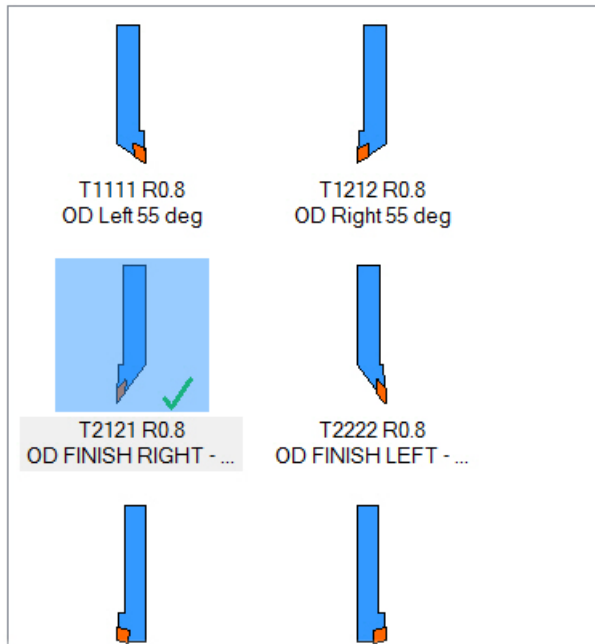


2. Select the **Last** button to reselect the geometry chained for the previous grooving operation.

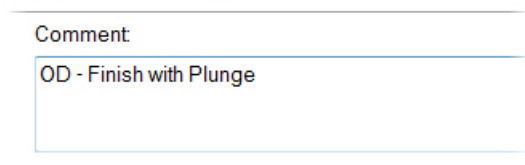


3. Click **OK** in the **Chaining** dialog box to accept the chain.

The **Lathe Finish** dialog box opens with the finish tool used in Operation 3 already selected.

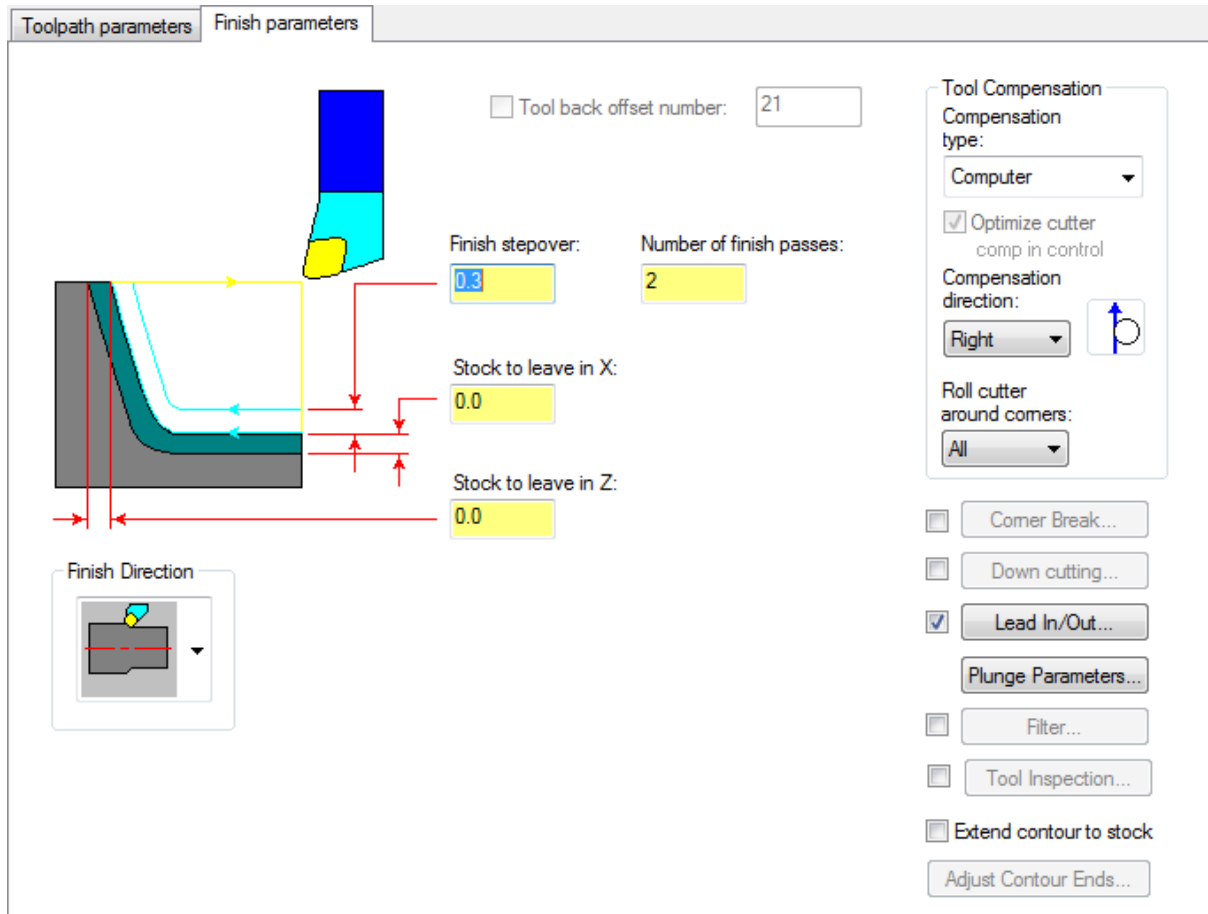


4. Type **OD - Finish with Plunge** in the **Comment** field.



5. Open the **Finish parameters** page.

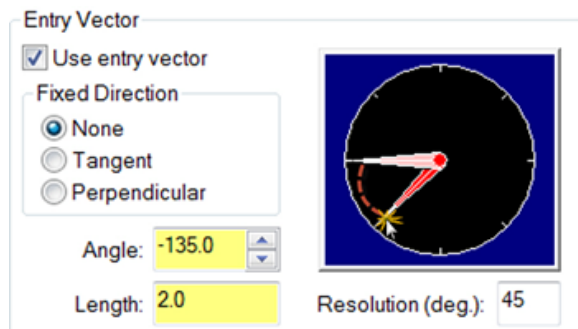
You do not need to edit any fields on this page.



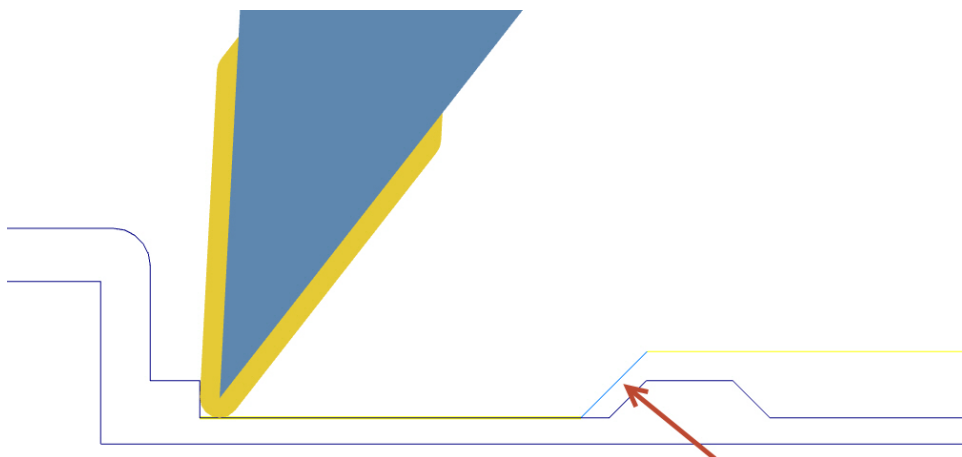
6. Click the **Lead In/Out** button.

The **Lead In/Out** dialog box opens.

- a. Enter **-135** in the **Angle** field, or drag the angle dial to define the entry vector.



The tool will follow this angle as it enters the cut.



b. Click **OK** to return to the **Lathe Finish** dialog box.

7. Click the **Plunge Parameters** button.

Plunge Parameters...

The **Plunge Cut Parameters** dialog box opens. Use this dialog box to define how you want the tool to handle plunges along the toolpath. You can choose to plunge in either or both axes. (See image [on the next page.](#))

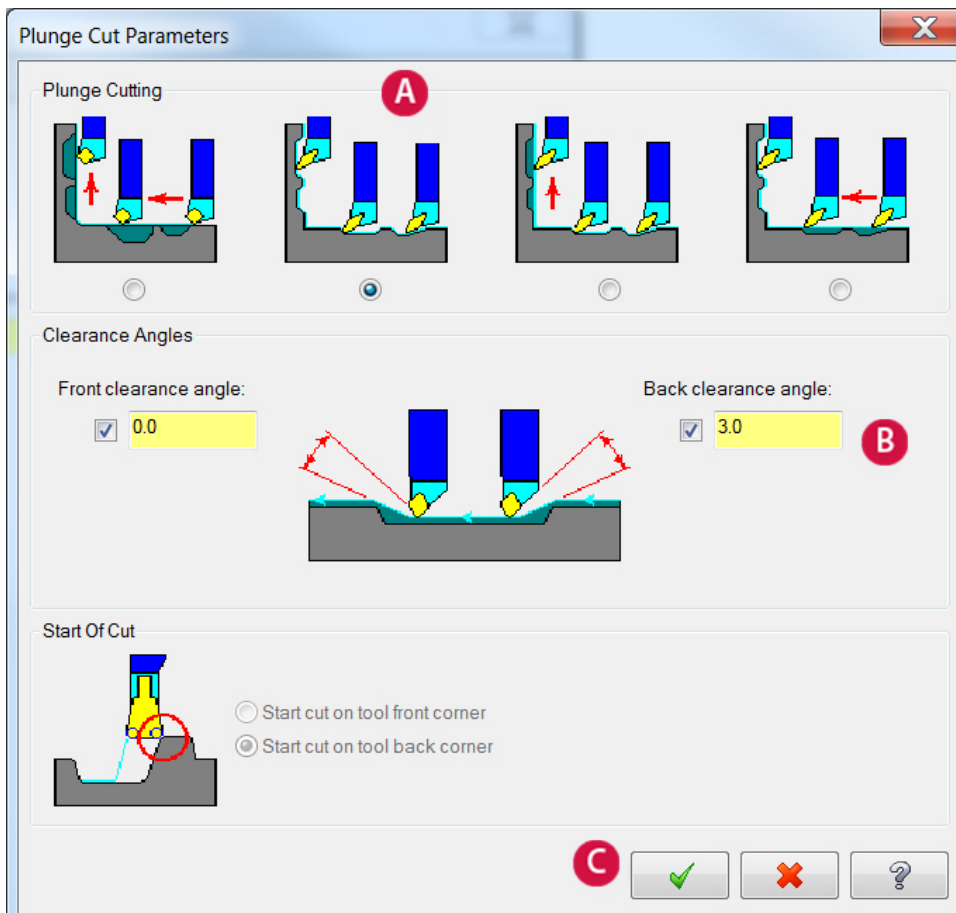
- a. Select the option to allow plunging in both directions.
  - The tool plunges into all sections on the chained path.
  - Choosing to plunge in both directions activates the **Front clearance angle** and **Back clearance angle** fields.
- b. Keep the default values for both clearance angle fields.

Clearance angles control how the tool plunges and gives additional clearance at the undercut wall.

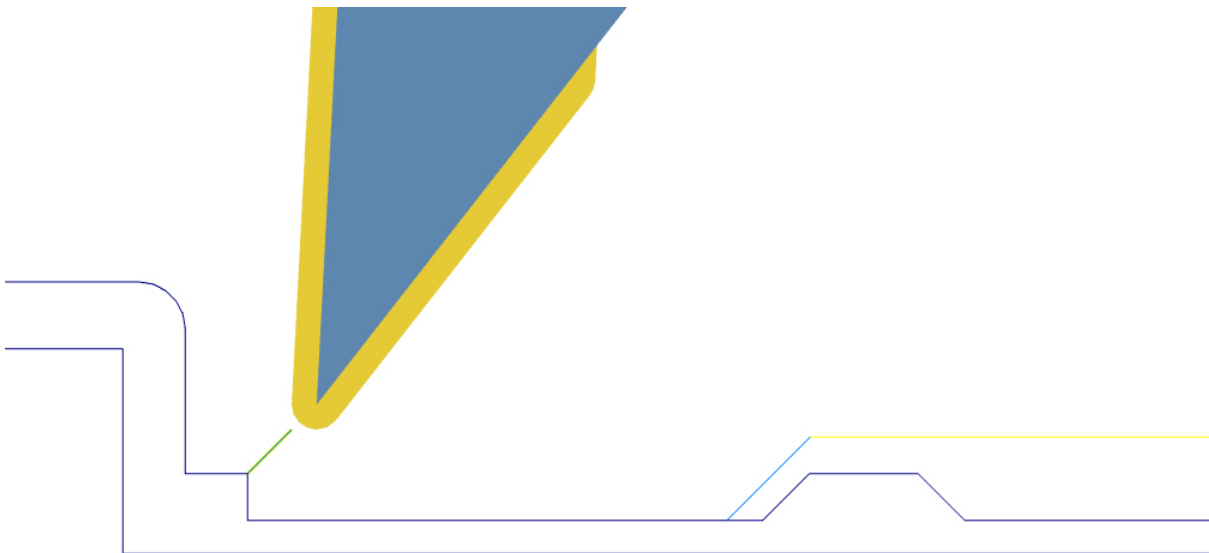
- The front clearance angle prevents the tool front from cutting with the entire length of the insert's front as the tool cuts up to the wall.
- The back clearance angle prevents the tool back from cutting with the entire length of the insert's back as the tool plunges to the next cut depth.



- c. Click **OK** to return to the Lathe Finish dialog box.



8. Click **OK** to create the toolpath.



9. Save the part.
10. (Optional) Backplot the groove and finish toolpaths that you have created in this lesson.

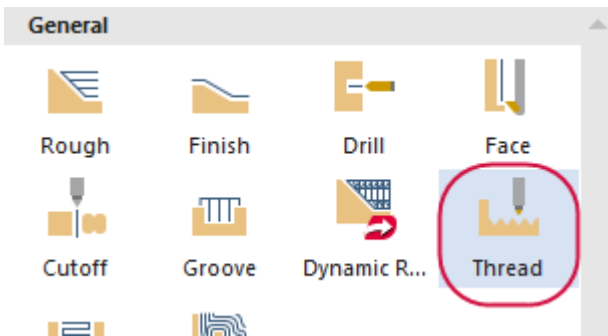
## Exercise 4: Adding a Thread Toolpath

A threading toolpath is typically the last toolpath performed on a lathe part because of the need for accuracy. A threaded part has to fit precisely into another part. You can program threads on the OD or ID to secure parts to each other.

With Mastercam, you can directly enter toolpath parameters or select geometry to create a thread toolpath.

In this exercise, you use part geometry to create an OD thread toolpath.

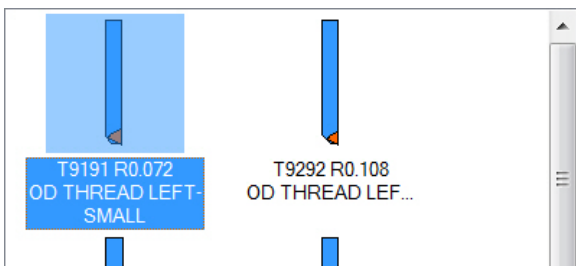
1. Expand the **General** gallery, and choose **Thread**.



The **Lathe Thread** dialog box opens.

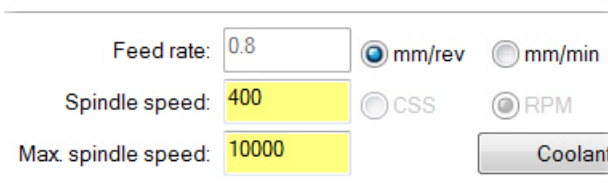
### Select the tool

1. From the **Toolpath parameters** tab, select the thread tool: **T9191 R0.072 OD THREAD LEFT - SMALL**.



2. Enter **400** into the **Spindle speed** field, and select the **mm/rev** option for the **Feed rate**.

The feed and speed rates you enter here overwrite the default settings for this operation.

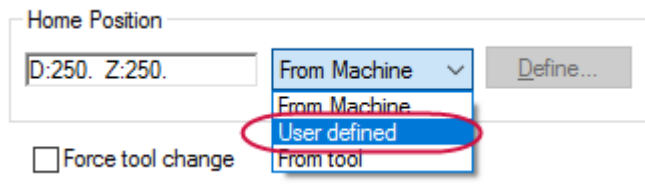


3. Change the tool's **Home Position**.

The home position is where the spindle or turret typically travels to for tool changes. Mastercam lets you control the home position from several different sources.

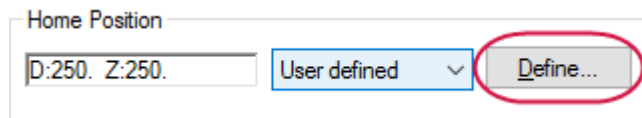
In this case, the operation defaults to using the D (250) and Z (250) coordinates defined in the machine definition. The following procedure shows you how to override the default home position.

- a. Choose **User defined** from the drop-down menu.

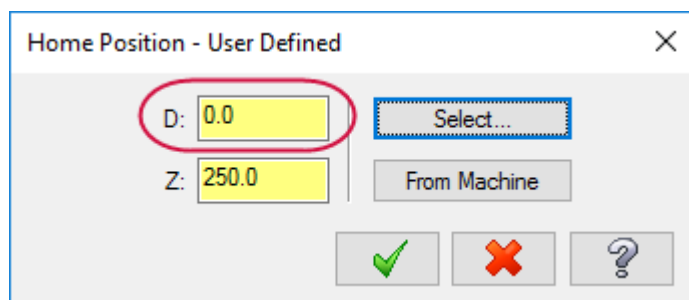


- b. Click the **Define** button.

The **Home Position - User Defined** dialog box displays.



- c. Change the **D** coordinate to **0**.



- d. Click **OK** to return to the **Lathe Thread** dialog box.

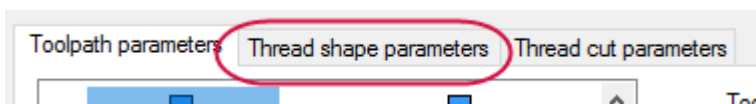
4. Keep all other parameters on this page at their default values.

## Enter thread shape parameters

The values you enter into this tab determine the following:

- Thread shape
- Cut location
- Amount of variation

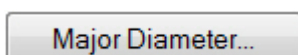
1. Click the **Thread shape parameters** tab.



2. In the **Lead** field, enter **1.5 mm**.

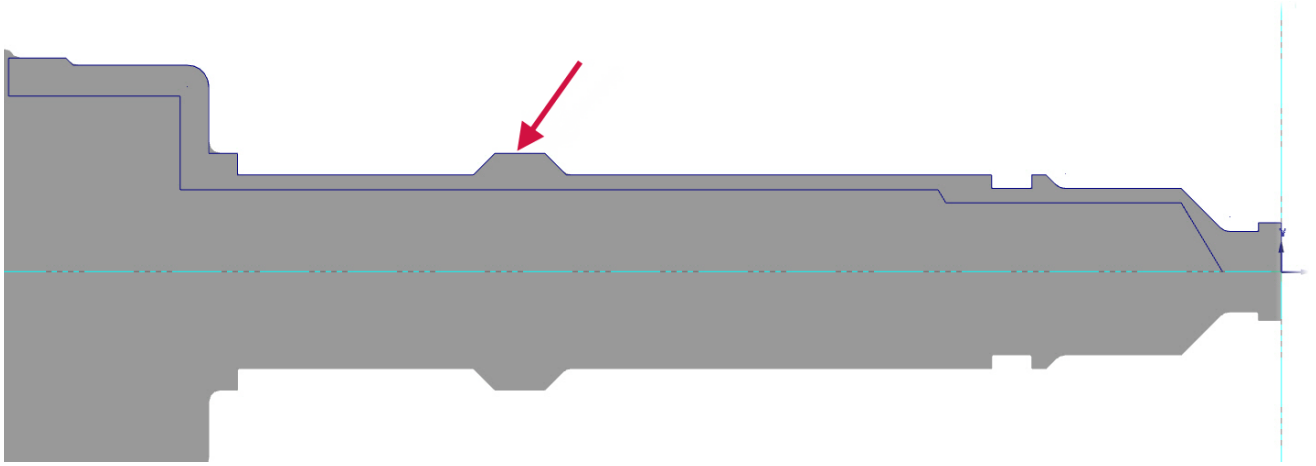
The blueprint in [on page 41](#) shows the lead value.

3. Click the **Major Diameter** button.

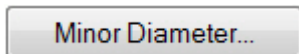


You are prompted to select a point in the graphics window.

4. Click the thread's largest diameter.

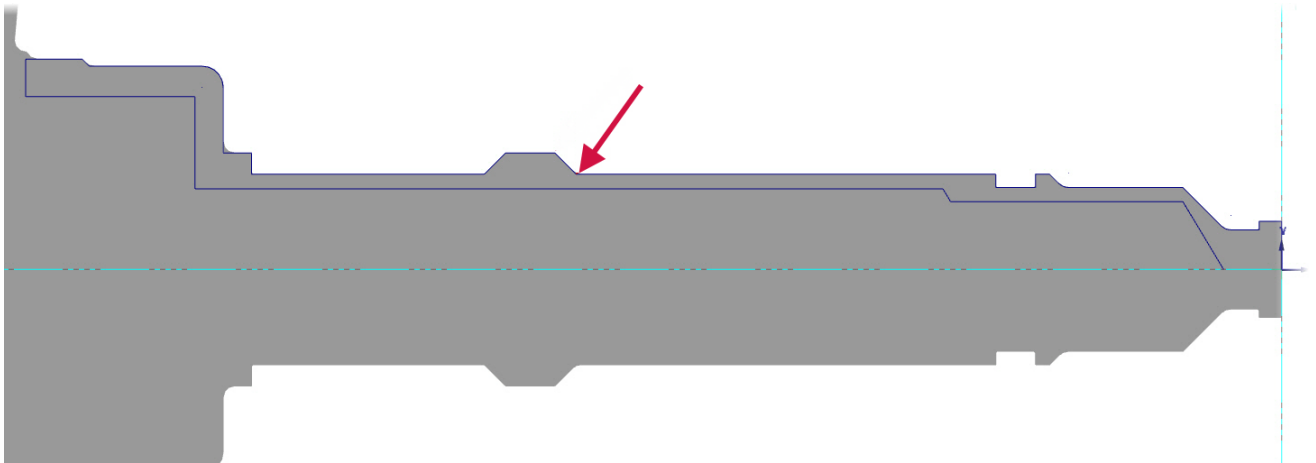


5. Click the **Minor Diameter** button.



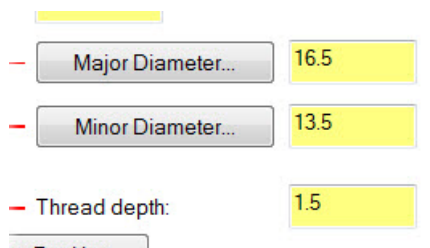
You are prompted to select a point in the graphics window.

6. Click the thread's smallest diameter.

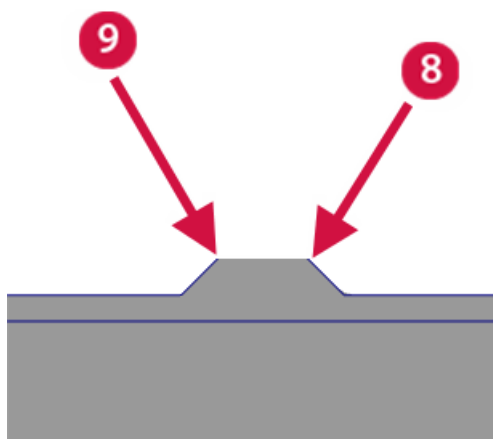


7. Make sure that your major and minor values are as shown.

Mastercam automatically calculates the **Thread depth** from the major and minor diameter values.



8. Click the **Start Position** button and select the beginning of the thread from graphics window. **OK** your selection.



9. Click the **End Position** button and select the end of the thread from graphics window. **OK** your selection.  
The fields on the tab show the location of the start and end of the thread on the Z axis.

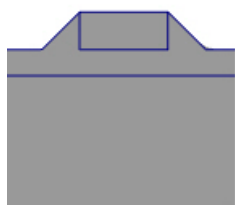


10. Confirm that the **Thread orientation** is **OD**.

Thread orientation determines whether the thread is created on the inner or outer diameter of the part.

11. Click the **Draw Thread** button to preview the thread geometry.

Mastercam creates an outline of the thread.

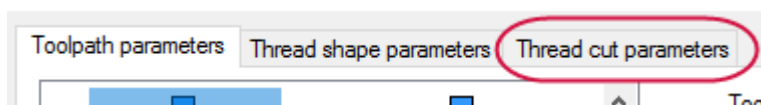


12. Choose **Yes** to save it with your part.

## Enter thread cut parameters

The values entered in this tab define how Mastercam will cut the material.

1. Click the **Thread cut parameters** tab.



2. Select the **Compute** option on the right side of the dialog box.

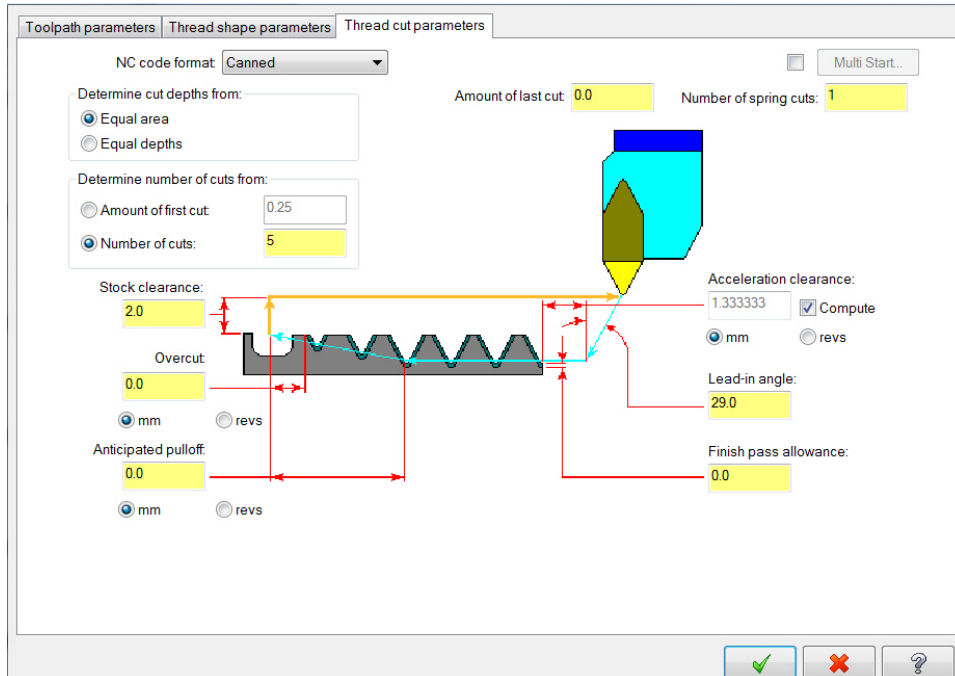
Mastercam automatically calculates the required acceleration clearance from the lead and left spindle speed.

Acceleration clearance:

1.333333 ☒ Compute

☒ mm ☐ revs

- Keep all other parameters on this page at their default values.



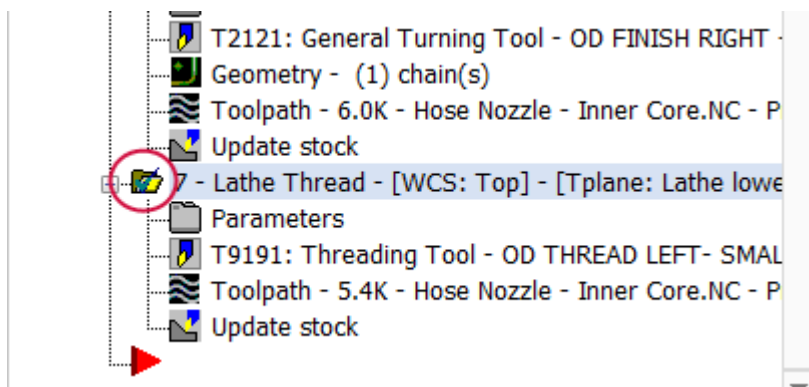
- Click **OK** to create the toolpath.
- Save the file.

## Exercise 5: Verifying the Toolpaths

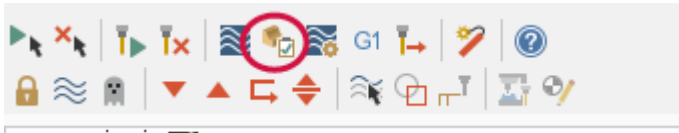
Verifying your toolpaths allows you to use solid models to simulate part machining against a selected stock definition. The result Verify creates represents the surface finish, and shows collisions, if any exist. You can identify and correct program errors before they reach the shop floor.

In this exercise, you use some of Verify's features to check the thread operation you created in this lesson.

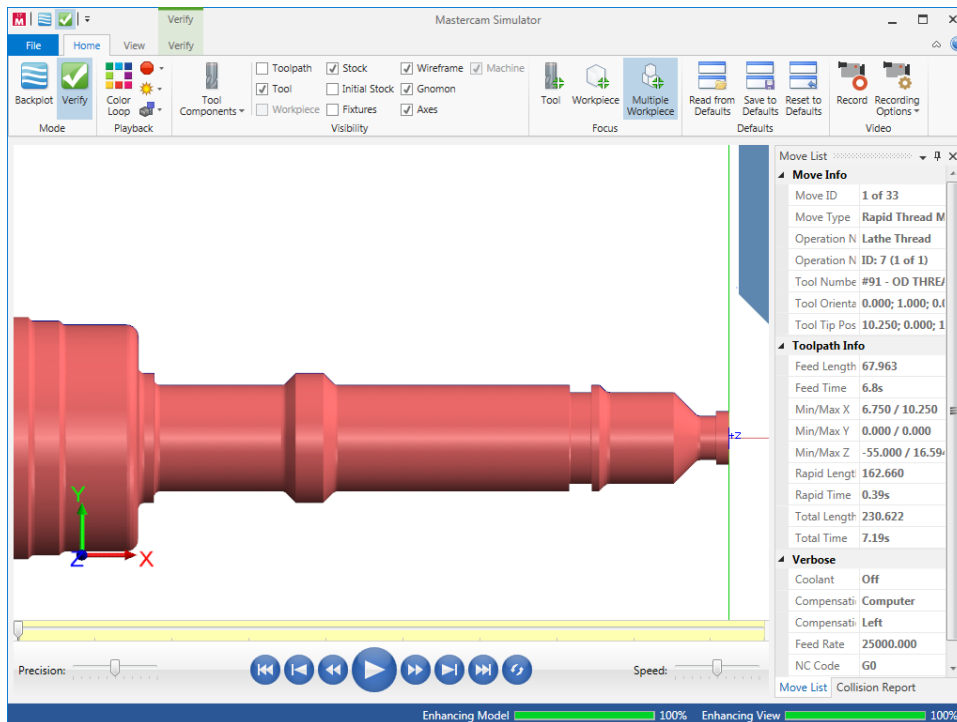
- Make sure that the **Thread** operation is selected in the Toolpaths Manager.



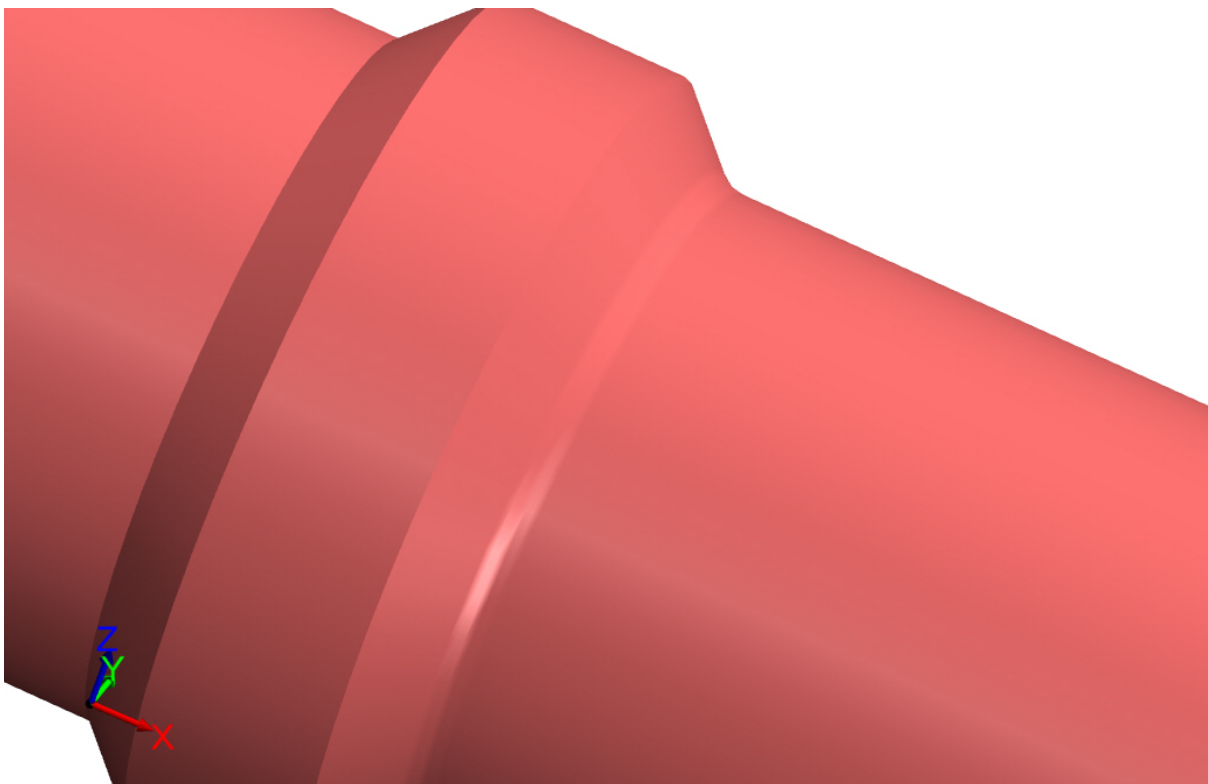
- Click the **Verify selected operations** button.



Verify displays in the Mastercam Simulator window.



3. Right-click in the graphics window and change the view to **Isometric**. Pan and zoom into the area that will be threaded.

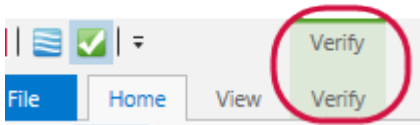


**Note:** The illustrations here show the part displayed with **Initial Stock** turned off.

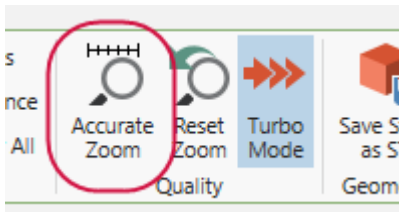
4. Click the **Play** button or press [R] to verify the toolpath.

Mastercam Simulator verifies the toolpath.

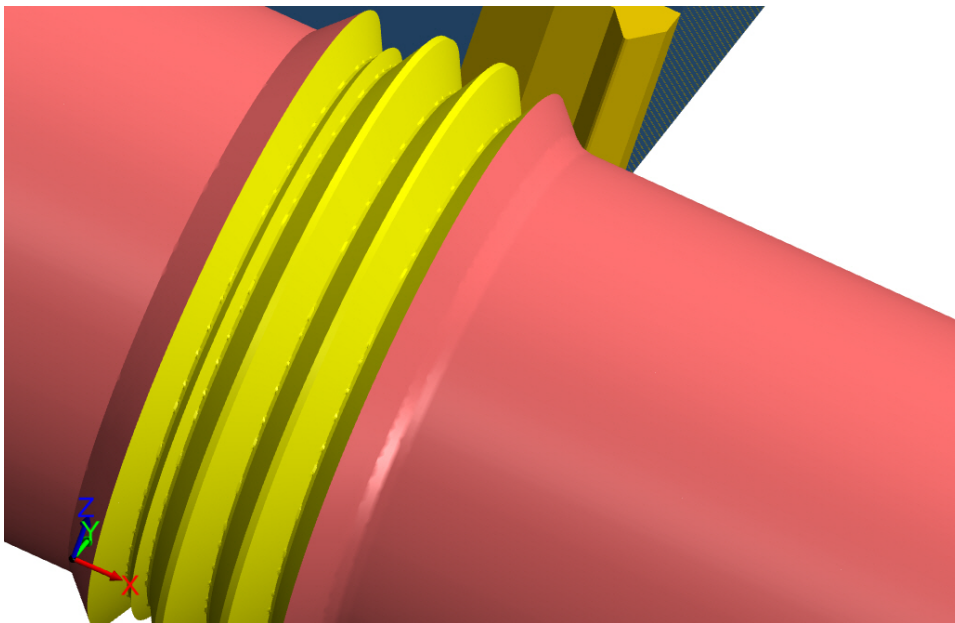
5. Select the **Verify** tab.



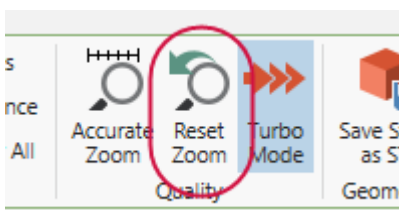
6. Click **Accurate Zoom**.



**Accurate Zoom** smooths surfaces when you view the part in close-up.



Click **Reset Zoom** to restore the original magnification result.



7. Minimize the Mastercam Simulator, or move it to another monitor.

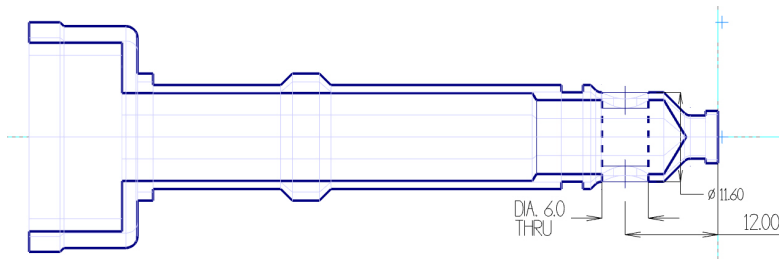
You have completed machining the outer diameter. In the next lesson, you will create drilling operations using the C-axis.



## CHAPTER 4

# C-AXIS DRILLING OPERATIONS

Lathe C-axis toolpaths are included with Mastercam for most common contour and drill applications. When you select one of these toolpaths, Mastercam automatically sets the Tplane and Cplane to the appropriate settings for the desired application. These toolpaths also support Y-axis rotation and axis substitution. To use C-axis toolpaths, the active machine definition must support the appropriate axes. In this lesson, you use two operations to drill a hole through the part on the C-axis.



### Goals

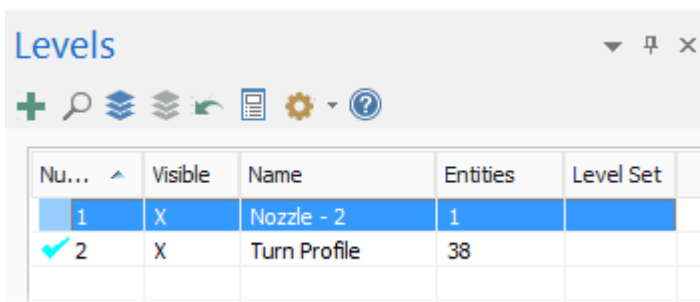
- Create 2D geometry from circular holes in the solid body.
- Add a toolpath group.
- Create a C-axis drill operation.
- Copy and modify an existing operation to create a new one.

### Making Drill Geometry

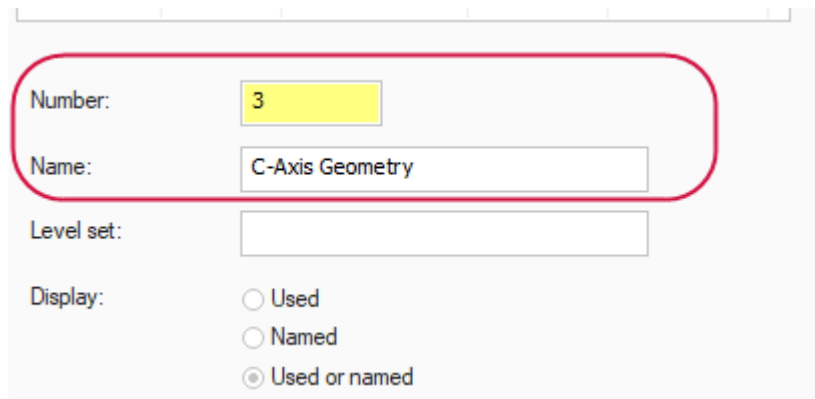
The drill toolpath requires the selection of two-dimensional geometry. Because the method you used to create 2D geometry in ["Creating 2D Geometry" on page 16](#) did not create a cross section of the part, the geometry for the drilled hole was not created. In this exercise, you use Mastercam's **Hole Axis** function to create the necessary geometry from circular holes in the solid body.

1. Open the **Levels Manager**, and make **Level 1** visible.

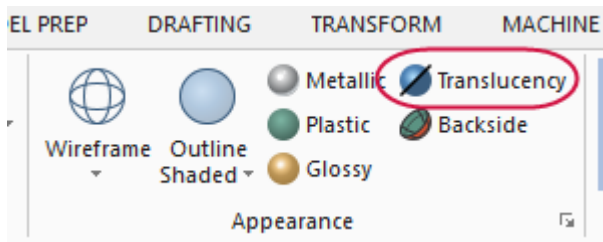
The solid model displays in the graphics window.



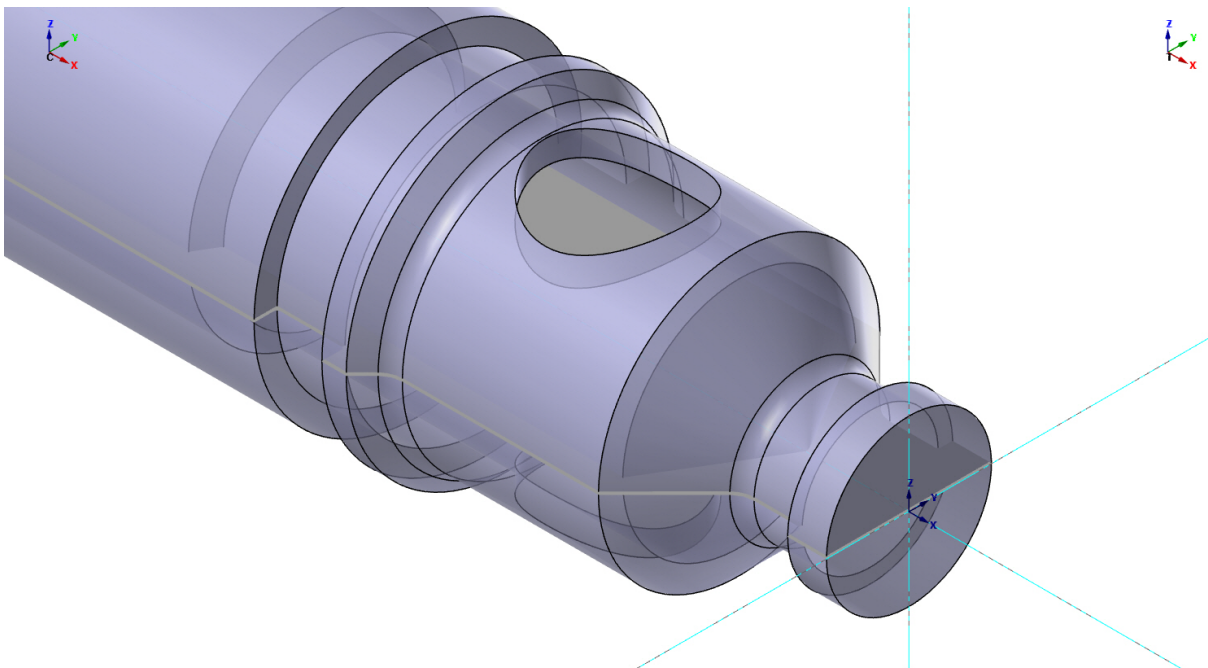
2. Create a Level 3. Name this level, **C-Axis Geometry**.



3. From the **View** tab, turn on translucent shading to view the interior of the solid.

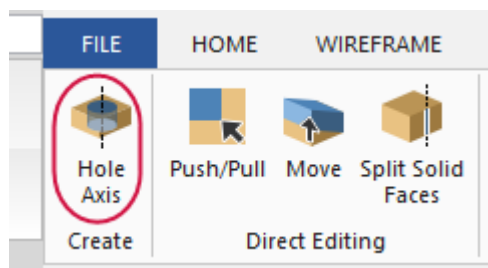


4. Use the **Isometric Gview** and **Zoom** to more easily see the model's drilled hole.



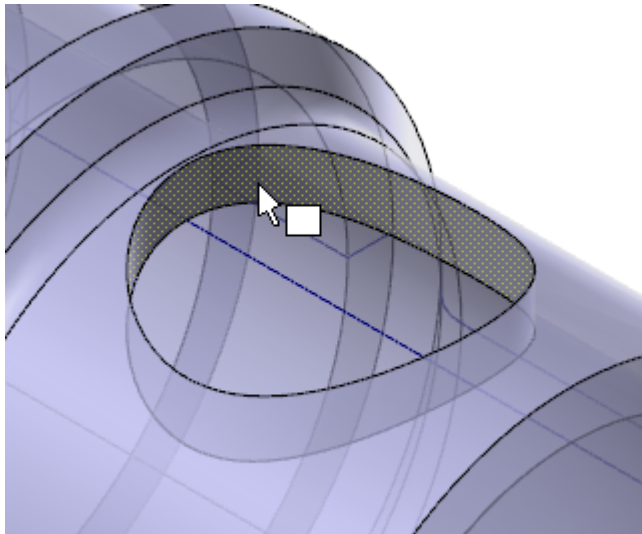
5. Create the C-Axis geometry.

- a. Select **Model Prep, Hole Axis**.

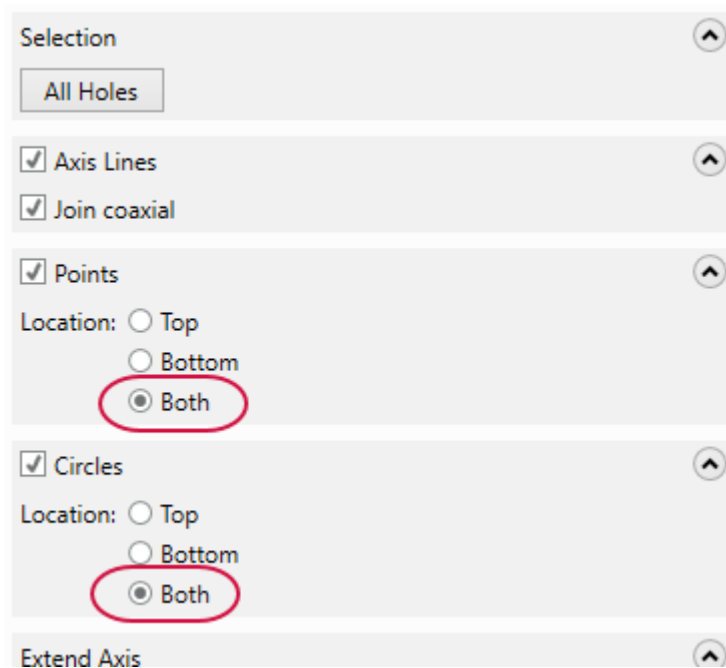


- b. [Ctrl + click] the internal face of the upper hole to select both openings of the through hole.

Use [Ctrl + click] to select faces with matching diameters.

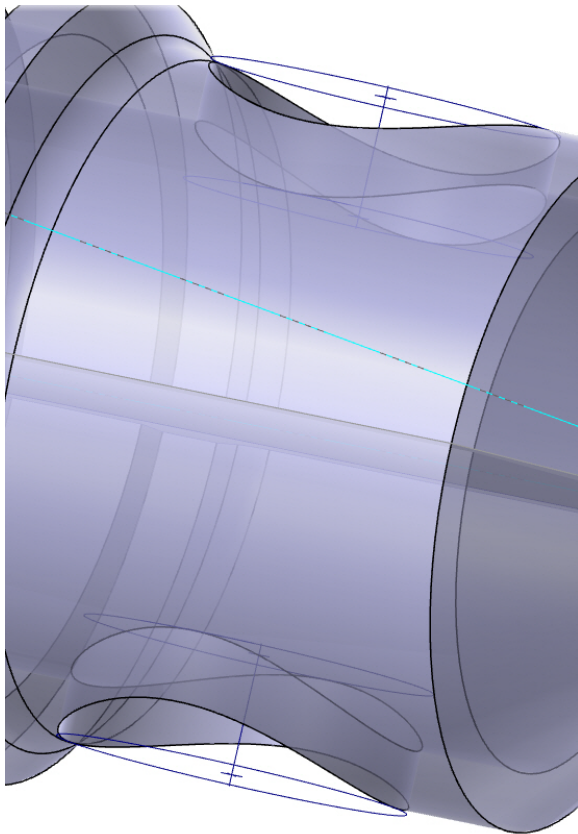


- c. Place points and circles at both the top and bottom of the holes.



- d. Click **OK** to accept your selection.

Mastercam creates geometry for both ends of the through hole.

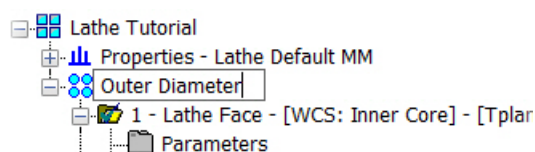


## Exercise 1: Adding a New Toolpath Group

Toolpath groups are subgroups that are “children” of machine groups. They inherit machining properties and other important properties from their parent groups. Toolpath groups house toolpath operations as well as other subgroups. They are useful for creating sets of toolpaths on the same machine that you will want to post separately. Mastercam supports unlimited subgroups.

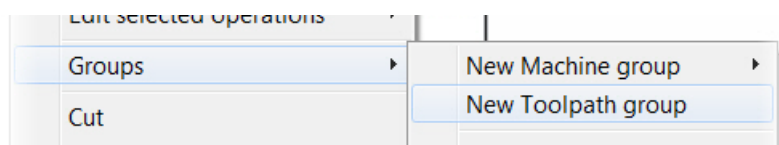
In this exercise you organize your operations into two different toolpath groups: Outer Diameter and C-Axis Drill.

1. In the Toolpaths Manager, right-click **Toolpath Group-1**, and select **Groups, Rename** from the menu.

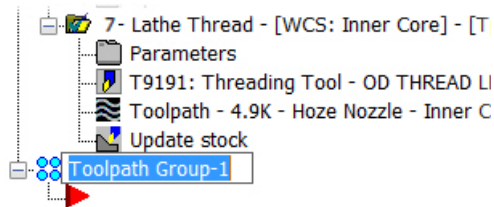


2. Change the name of **Toolpath Group-1** to **Outer Diameter**, and press [Enter].
3. Create the **C-Axis Drill** toolpath group.
  - a. Right-click the machine group and select **Groups, New Toolpath group** from the menu.

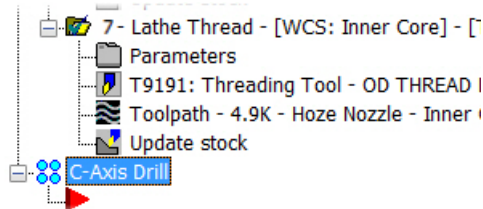
A new toolpath group is created.



- b. If necessary, move the insert arrow past the new toolpath group.



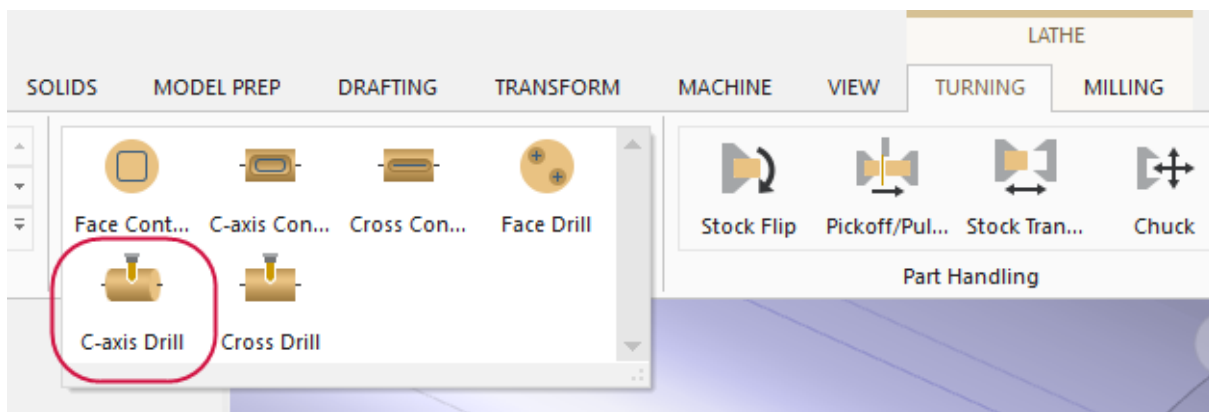
- c. Rename the new toolpath group: **C-Axis Drill**, and press [Enter].



## Exercise 2: Creating a C-Axis Drill Operation

In this exercise, you create a center drill operation with a C-axis drill toolpath.

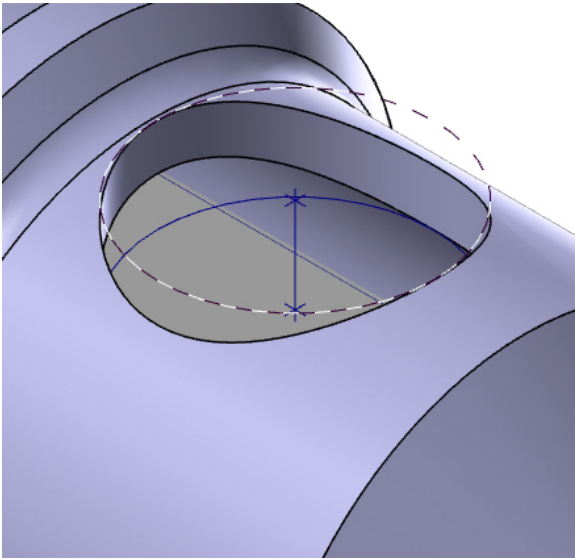
1. Expand the **C-axis** gallery, and select **C-axis Drill** from the **Lathe Turning** tab.



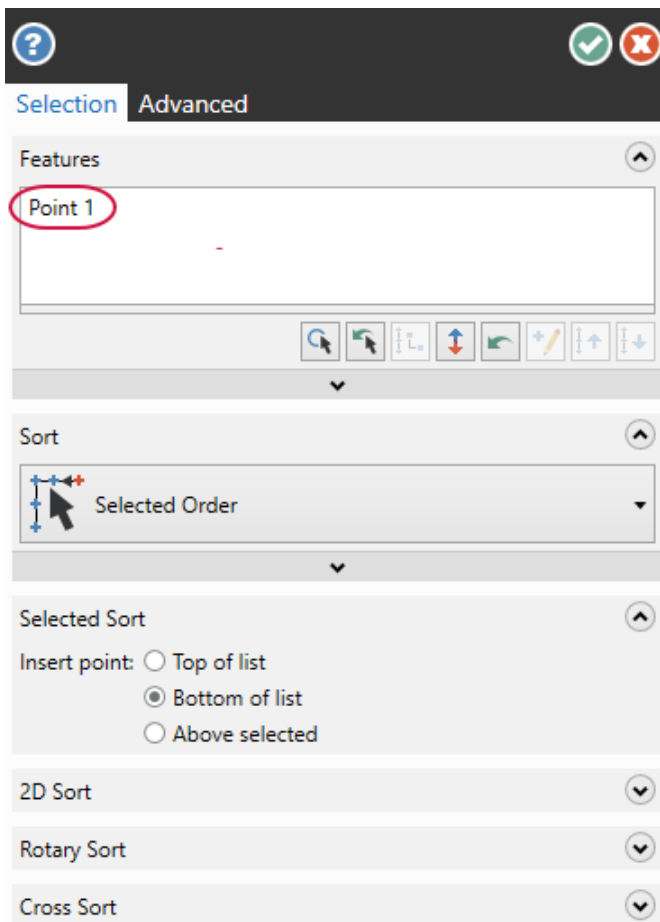
The **Toolpath Hole Definition** function panel opens. Although this operation only requires a simple selection, the **Toolpath Hole Definition** function panel provides many techniques for selecting points when your hole-making toolpath requires something more complex. See Mastercam Help for more information about this function panel.

2. Select the top point or the top arc.

**Note:** The illustration [below](#) shows the top arc selected. When you select open or closed arcs, the drill points are placed in the center of the arcs.



The **Features** list displays one defined point.



3. Click **OK**.

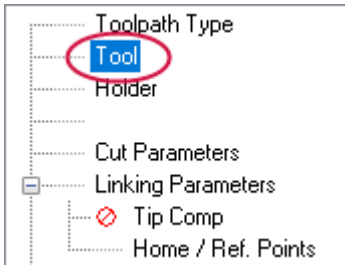
The **C-Axis Toolpath - C-Axis Drill** dialog box opens.

## Enter the Toolpath Parameters

In addition to selecting tools and entering cutting values, the C-axis drill operation requires that you define the rotary axis motion.

### Select the tool

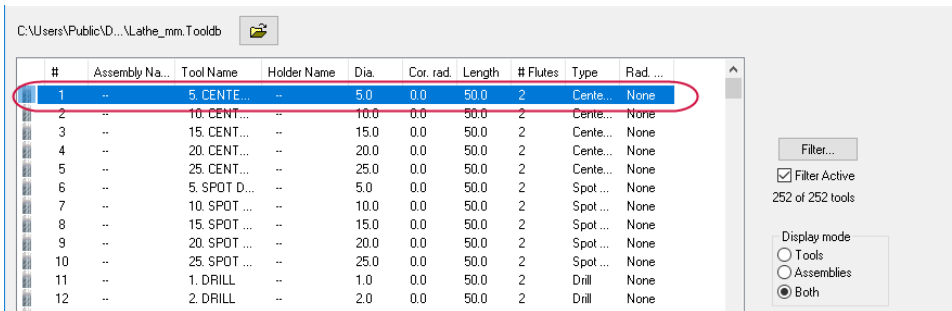
1. From the Tree View, open the **Tool** page.



2. Click the **Select library tool** button.

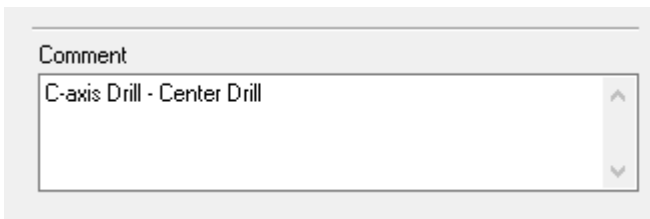
The **Tool Selection** dialog box opens.

3. Choose **5.CENTER DRILL** from the list, and click **OK** to return to the **Tool** page.



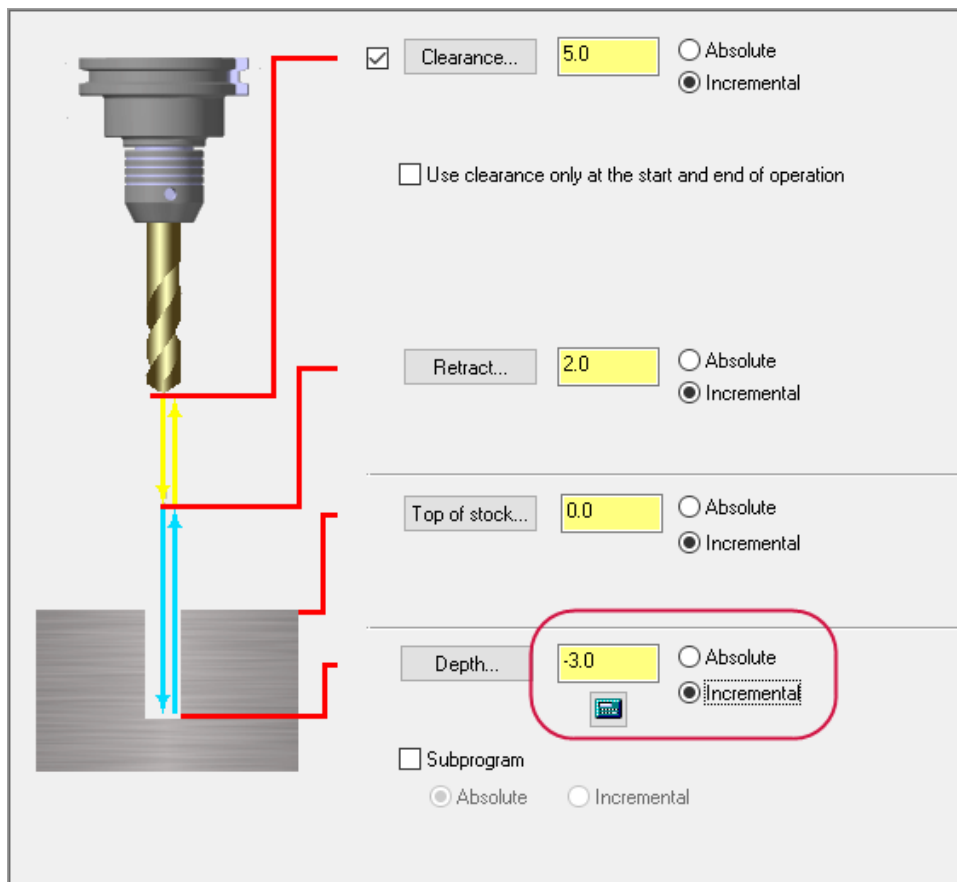
**Note:** Use the **Filter** to narrow the tools displayed in the list.

4. In the **Comment** window, type **C-axis Drill - Center Drill**.



### Define the Hole Depth

1. Open the **Linking Parameters** page.
2. Enter an incremental cutting depth of **-3.0**.



3. Keep all other options on the **Linking Parameters** page at their default values.

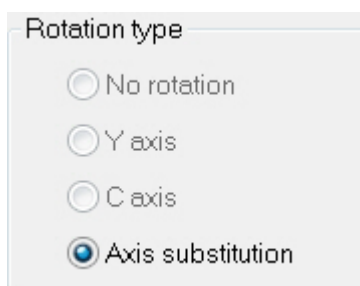
### Configure Rotary Axis Motion

C-axis toolpaths automatically set the rotation type to **Axis substitution** around the Z axis. This gives you the choice of chaining either flat geometry which will be rolled around the cylinder, or geometry which is already properly positioned in 3D space.

1. Open the **Rotary Axis Control** page.

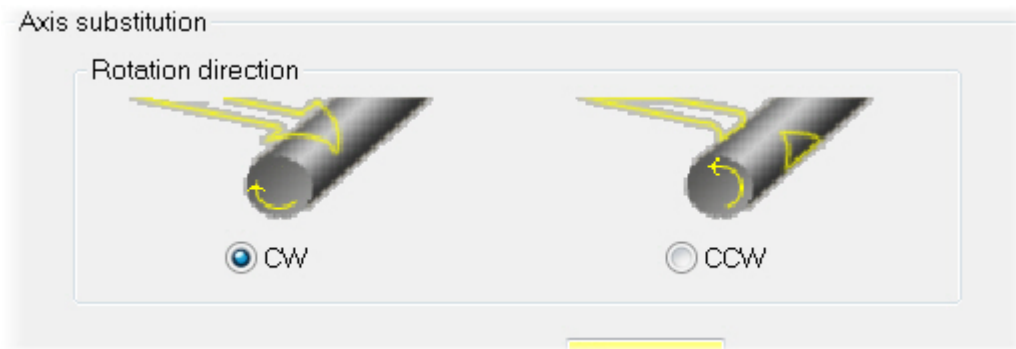


The **Axis substitution** option is the only rotation type available for C-axis drill operations.



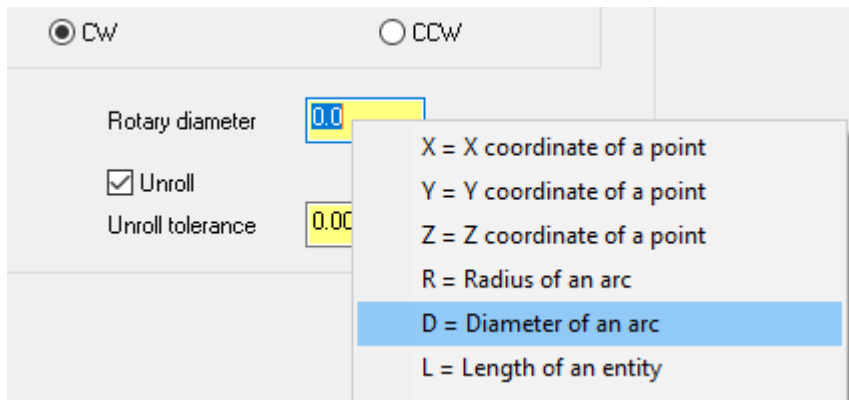
2. Set **Rotation direction** to **CW** (clock-wise), and confirm that **Unroll** is selected.

Mastercam unrolls the geometry on to a flat plane in a clockwise direction relative to the selected axis. When you post the toolpath, the geometry wraps back onto the cylinder using the axis substitution and rotary diameter parameters.

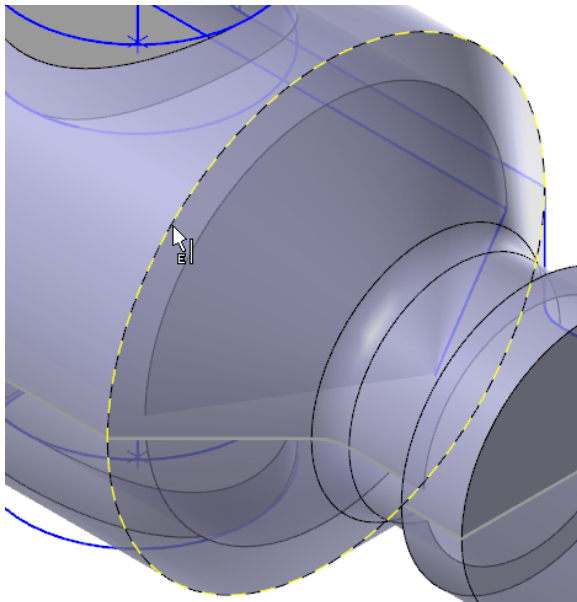


3. Enter the **Rotary diameter**. Use the dimension from the blueprint [on page 59](#), or follow the procedure below.

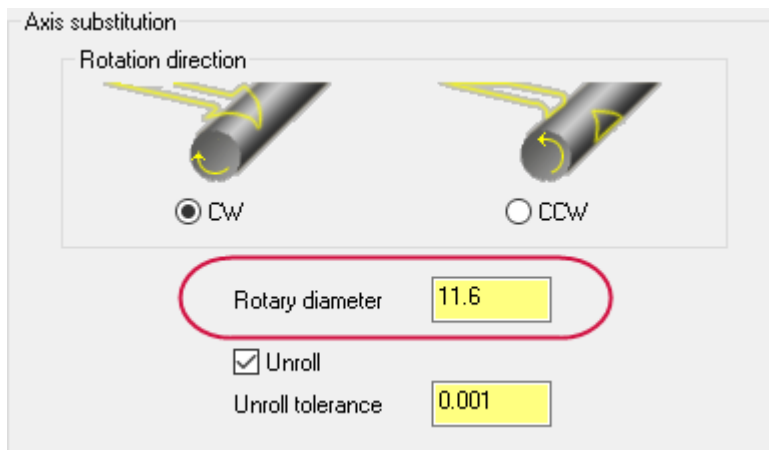
- a. Right-click in the **Rotary diameter** field, and select **D = Diameter of an arc**.



- b. Select the circular edge of the model.



A built-in calculator reads the dimensions of the selected arc directly into the **Rotary diameter** field.

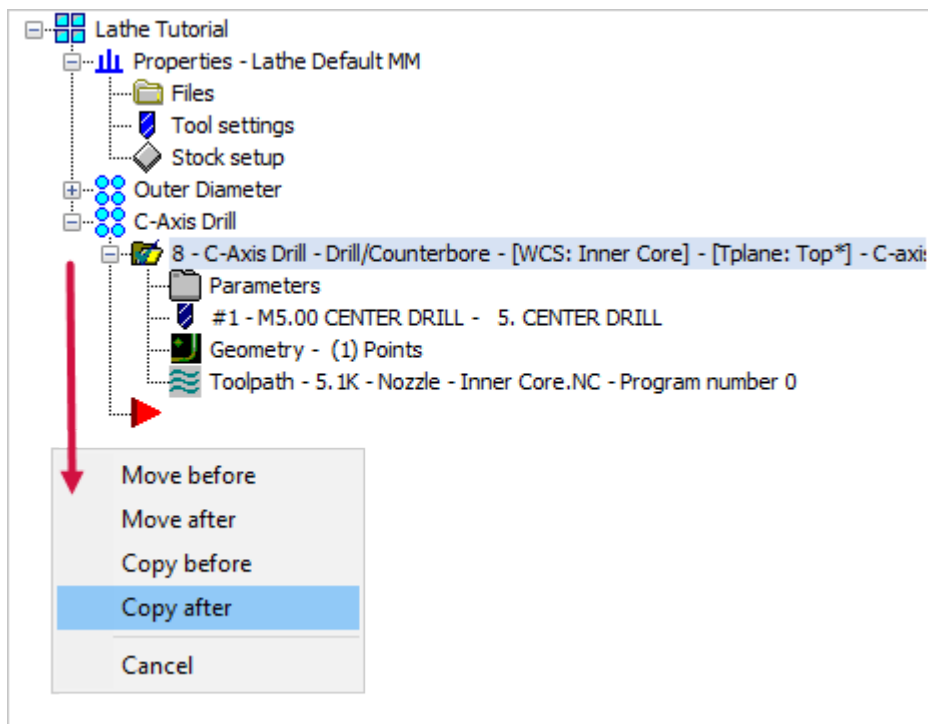


4. Click **OK** to create the toolpath.
5. Save the part.
6. Backplot and/or Verify this toolpath following the procedures you learned in the previous lessons.

### Exercise 3: Copying the Drilling Operation

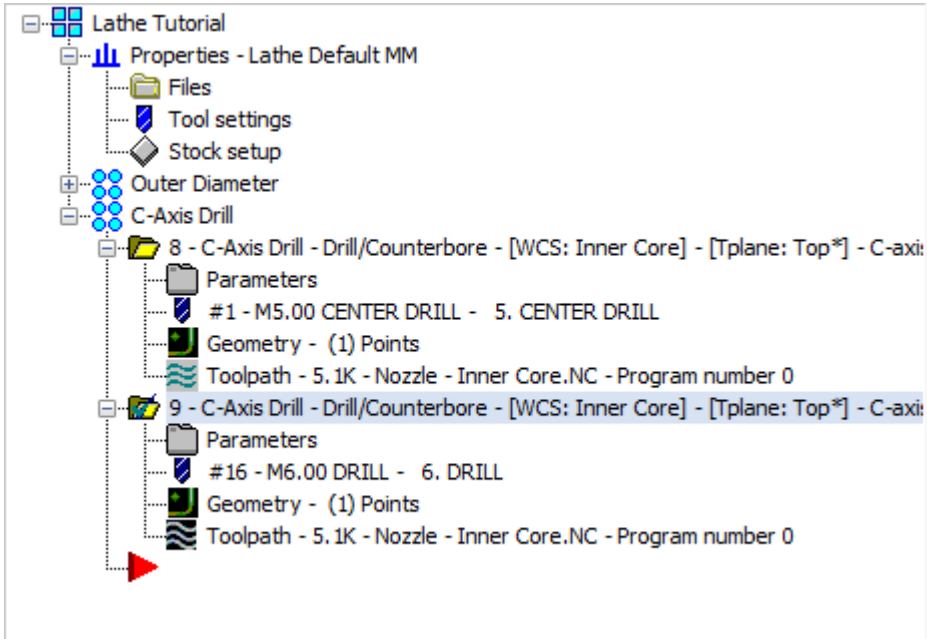
The final drill operation uses the same geometry and many of the same parameters as the center drill operation you created in [Exercise 2](#). In this exercise, you copy and paste one operation in the Toolpaths Manager in order to quickly create a new one.

1. Right-click and drag the drill toolpath you just created below the red arrow.
2. Release the mouse button and choose **Copy after** from the menu.



Mastercam creates a new operation.

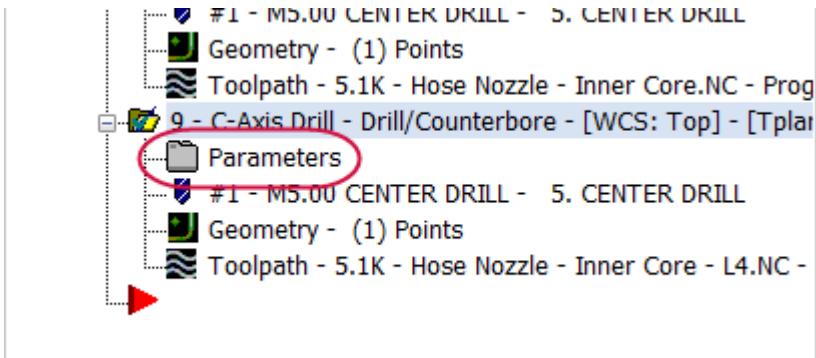
3. If necessary, move the insert arrow past the new toolpath.



## Exercise 4: Modifying the Drilling Parameters

In this exercise, you create the drill-through operation by modifying the parameters in the center drill operation that you copied.

1. Click the new toolpath's **Parameters** folder to open the dialog box.



2. Open the **Tool** page.
3. Click the **Select library tool** button, and select the 6 mm drill.

#	Assembly Na...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. ...
4	--	20. CENT...	--	20.0	0.0	50.0	2	Cente...	None
5	--	25. CENT...	--	25.0	0.0	50.0	2	Cente...	None
6	--	5. SPOT D...	--	5.0	0.0	50.0	2	Spot ...	None
7	--	10. SPOT ...	--	10.0	0.0	50.0	2	Spot ...	None
8	--	15. SPOT ...	--	15.0	0.0	50.0	2	Spot ...	None
9	--	20. SPOT ...	--	20.0	0.0	50.0	2	Spot ...	None
10	--	25. SPOT ...	--	25.0	0.0	50.0	2	Spot ...	None
11	--	1. DRILL	--	1.0	0.0	50.0	2	Drill	None
12	--	2. DRILL	--	2.0	0.0	50.0	2	Drill	None
13	--	3. DRILL	--	3.0	0.0	50.0	2	Drill	None
14	--	4. DRILL	--	4.0	0.0	50.0	2	Drill	None
15	--	5. DRILL	--	5.0	0.0	50.0	2	Drill	None
16	--	6. DRILL	--	6.0	0.0	50.0	2	Drill	None
17	--	7. DRILL	--	7.0	0.0	50.0	2	Drill	None
18	--	8. DRILL	--	8.0	0.0	50.0	2	Drill	None
19	--	9. DRILL	--	9.0	0.0	50.0	2	Drill	None
20	--	10. DRILL	--	10.0	0.0	50.0	2	Drill	None

- Click **OK** to return to the **Tool** page.
- Type **C-axis Drill** in the **Comment** field.

Comment

C-axis Drill

- Open the **Linking Parameters** page.
- Using the dimension [on page 59](#), enter the cut depth.

Top of stock... 0.0 ☐ Absolute ☒ Incremental

Depth... -11.6 ☐ Absolute ☒ Incremental

☐ Subprogram ☒ Absolute ☐ Incremental

- Open the **Tip Compensation** page.

Use this page to configure how the tool drills all the way through the stock.

Tool

Holder

Cut Parameters

✓ Linking Parameters

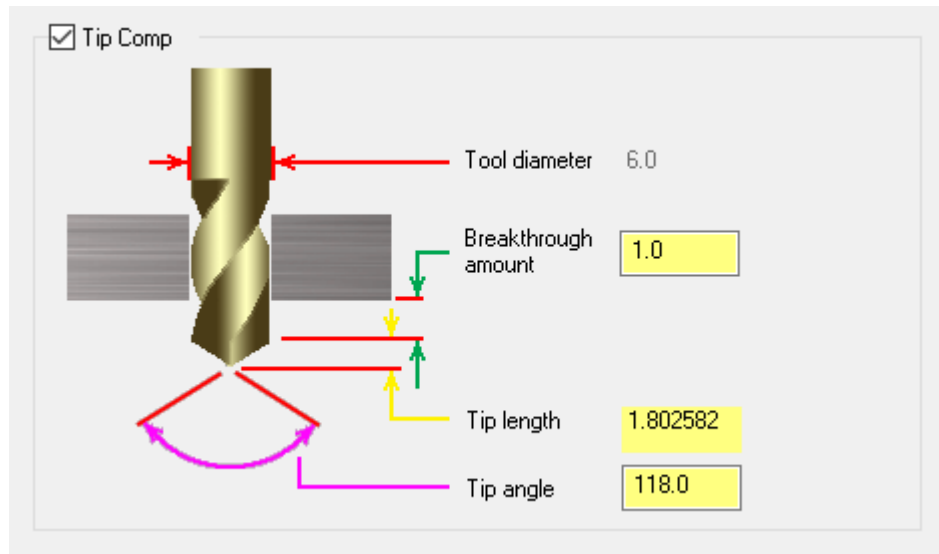
Tip Comp

Home / Ref. Points

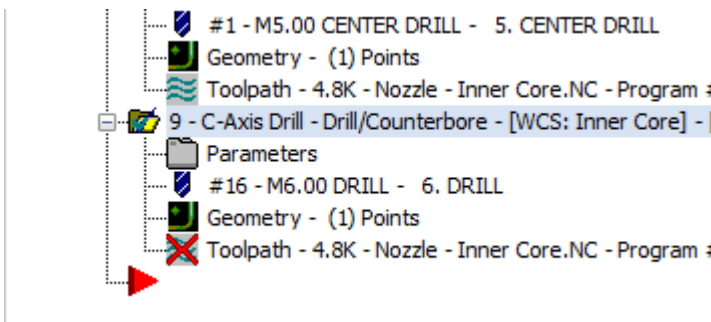
Planes (WCS)

- a. Select the **Tip Compensation** checkbox to activate the parameters on the page.
- b. Enter a **1.0 mm** breakthrough amount.

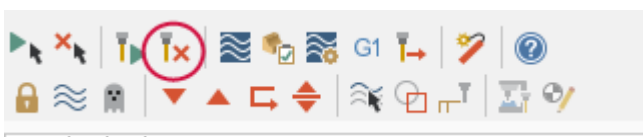
This insures that the full diameter of the tool breaks through the bottom of the stock.



9. Click **OK**.
10. The new toolpath is marked dirty in the Toolpaths Manager.



11. Regenerate the toolpath, and save your part.



You have completed machining the outer diameter of the hose nozzle. In the next lesson, you prepare for machining the inner diameter with Mastercam's cutoff and stock flip operations.



## CHAPTER 5

# CUTOFF AND STOCK FLIP

Lathe's part handling operations manipulate the stock and reposition chucks, tailstocks, and steady rests. Operations can output a comment and a program stop in the NC code to let the operator manually reposition the stock or a peripheral, or they can output code to automatically control peripherals. Mastercam Lathe currently supports the following part handling operations:

- Stock flip (reposition stock on the same spindle)
- Stock transfer (reposition stock to a different spindle)
- Stock advance (program a bar feeder)
- Chuck (clamp/unclamp/reposition)
- Tailstock (advance/retract)
- Steady rest reposition

In this lesson, you prepare for machining the inner diameter with Mastercam's cutoff and stock flip operations.

### Goals

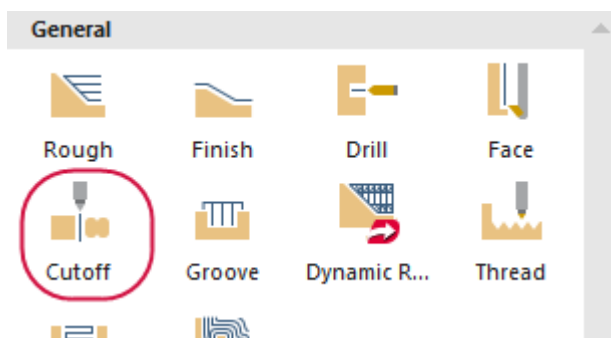
- Create a cutoff operation.
- Program a stock flip.

### Exercise 1: Cutting Off the Part

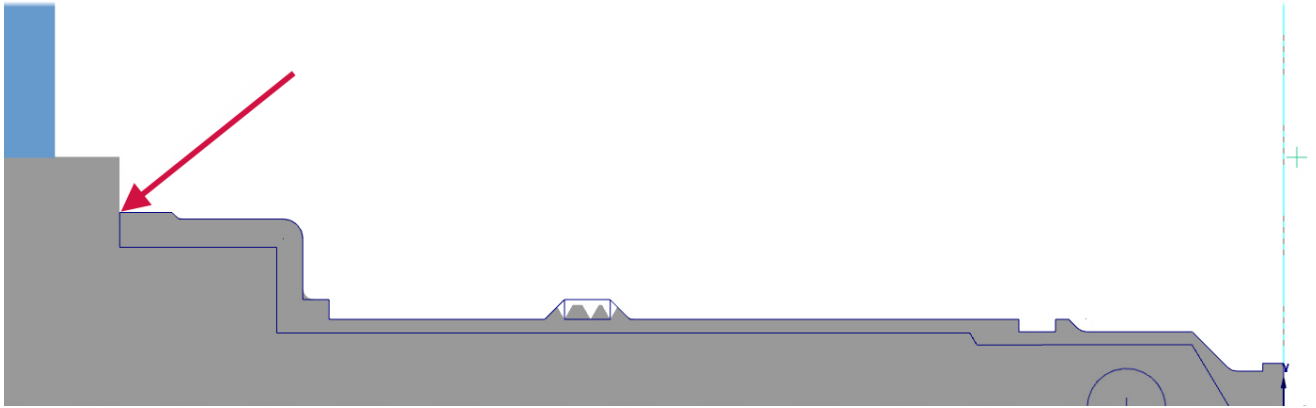
Cutoff toolpaths vertically cut off pieces of the part, such as sections of bar stock. You do not need to chain geometry for a cutoff toolpath. Instead, you select the point where the part is cut off.

In this exercise, you use the options in the cutoff toolpath to separate the part from the bar stock.

1. Expand the **General** gallery, and select **Cutoff** from the **Lathe Turning** tab.



2. Select the cutoff boundary point at the point indicated.

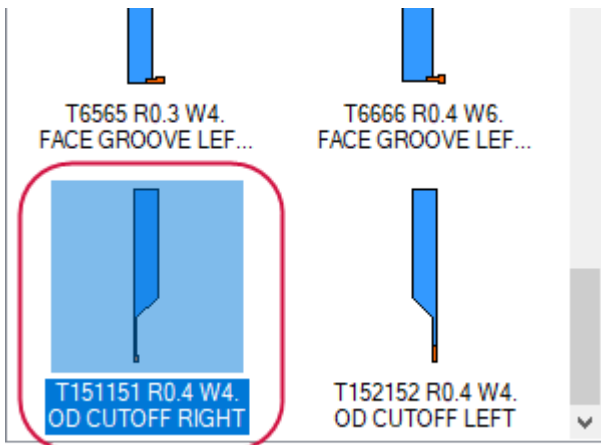


**Note:** To more easily select the cutoff boundary point:

- Change the GView to **Top**
- Hide **Level 1**
- Turn off the toolpath display [**Alt + T**]

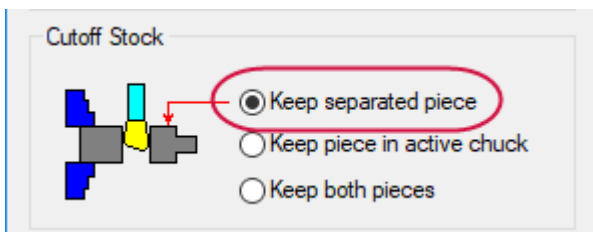
The **Lathe Cutoff** dialog box opens.

3. Select the cutoff tool: **T151151 R0.4 W4. OD CUTOFF RIGHT.**



4. Click the **Stock Update** button to open the **Stock Update Parameters** dialog box.

Use this dialog to define whether the stock will be based on the finished part or leftover stock after the cutoff operation.

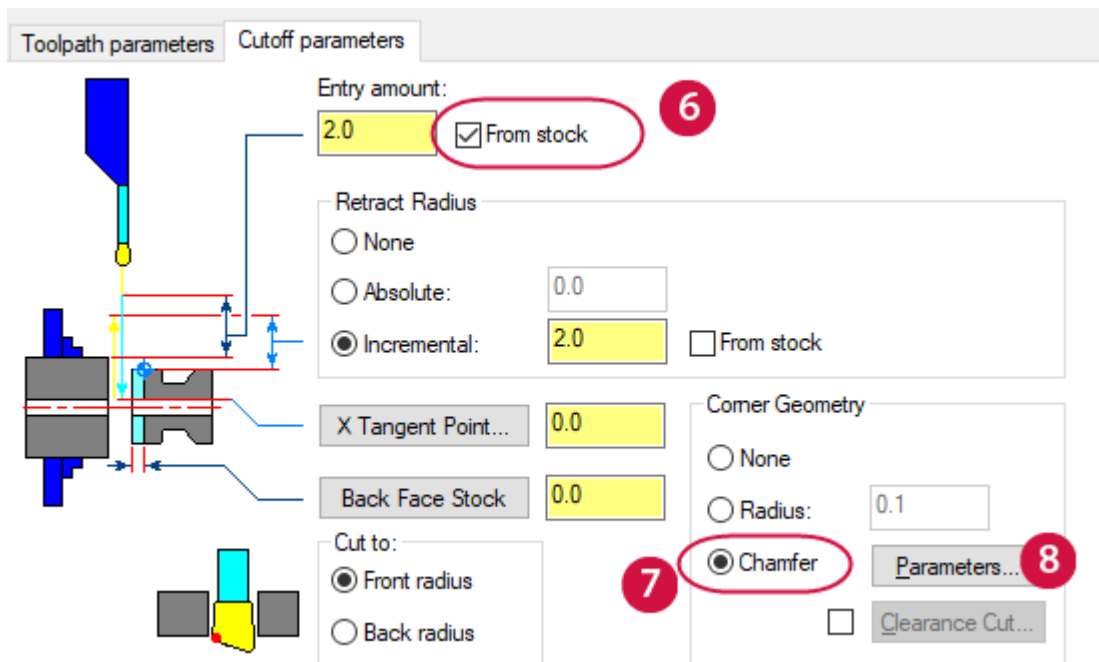


a. Select the **Keep separated piece** option.

Mastercam will display the part boundary after the part is cut from the stock.

b. Click **OK**.

5. Click the **Cutoff parameters** tab.



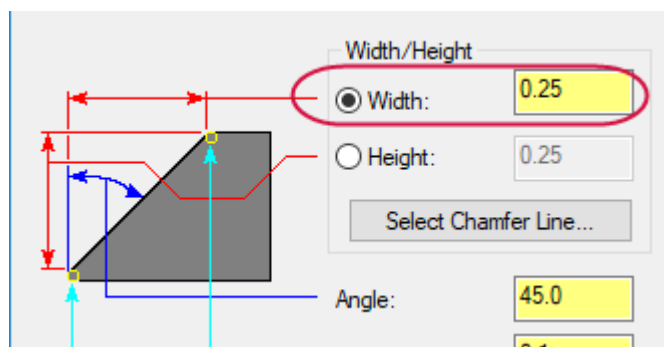
6. Select the **From stock** option.

Mastercam computes the **Entry amount** from the maximum stock diameter at the Z coordinate of the cutoff boundary point.

7. Select **Chamfer** to cut a chamfer on the corner of the part.
8. Click **Parameters** to enter chamfer dimensions in the **Cutoff Chamfer** dialog box.

- a. Enter **0.25** in the **Width** field.

The chamfer is defined by its width. Mastercam automatically calculates the height from this dimension and the angle.



- b. Click **OK** to return to the **Cutoff parameters** tab.

9. Click **OK** to create the operation.

Mastercam cuts the part from the stock.

10. Save the part.

## Exercise 2: Programming a Stock Flip

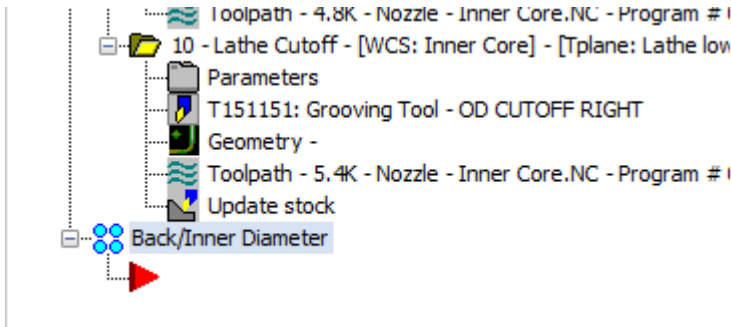
A stock flip lets you program operations on the opposite side or back of a lathe part in the same Mastercam file. Stock flip operations output a comment and program stop in the NC code, which lets the operator manually remove the stock and reposition (flip) it in the chuck.

In this exercise, you program the new stock and chuck position.

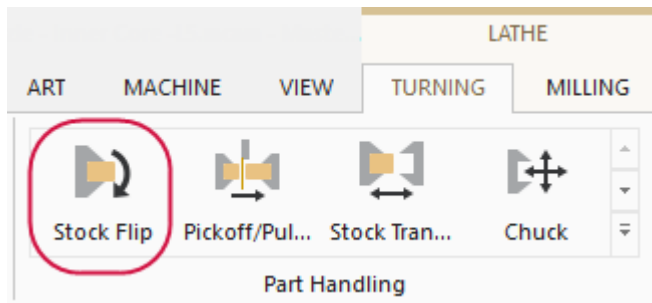
**Note:** You can only program part handling operations such as a stock flip when they are supported by the active machine definition.

1. Use the procedure described [on page 62](#) to create a new toolpath group. Rename the group **Back/Inner Diameter**.

The new toolpath group will include the stock flip and the operations machining the back and inner diameter of the part.



2. Select **Stock Flip** from the **Part Handling** group on the **Lathe Turning** tab.



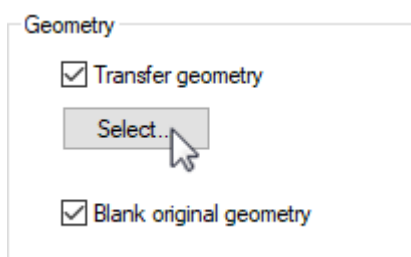
The **Lathe Stock Flip** dialog box opens.

## Re-align Part Geometry

Use the options in the **Lathe Stock Flip** dialog box to create a copy of the part geometry that is flipped about the Lathe Z axis and to hide the original geometry.

**Note:** Mastercam blanks the original entities. They are not deleted. (See Help for more information on blanked entities.)

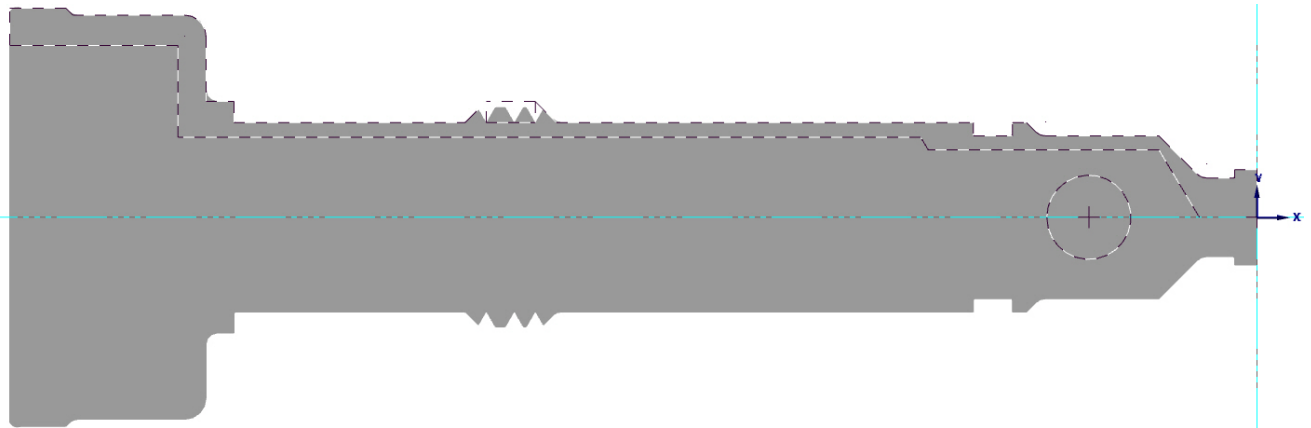
1. If necessary, select the **Transfer geometry** and **Blank original geometry** options.
2. Click the **Select...** button under the **Transfer geometry** option.



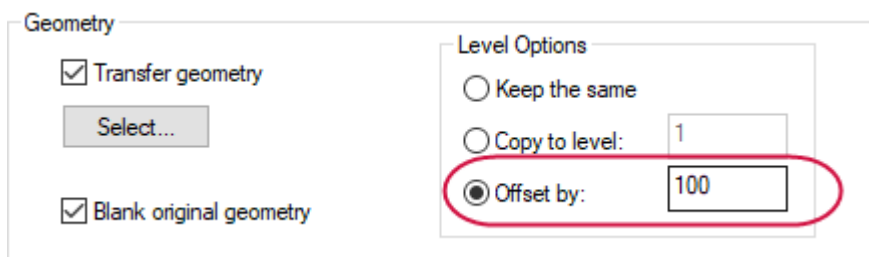
Mastercam returns you to the graphics window.

3. Window-select the entire part, and press **[Enter]** or the **End Selection** button.

All entities are selected.



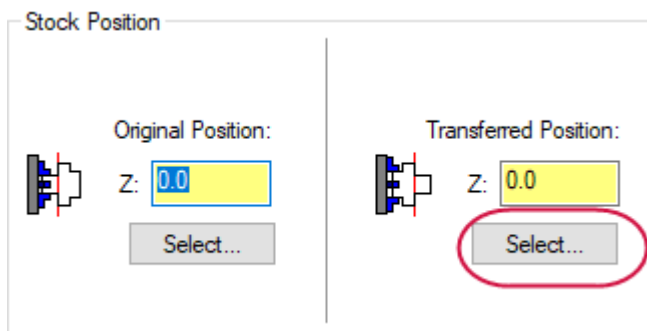
4. In the **Level Options** section, offset the level of the transfer geometry by **100**.

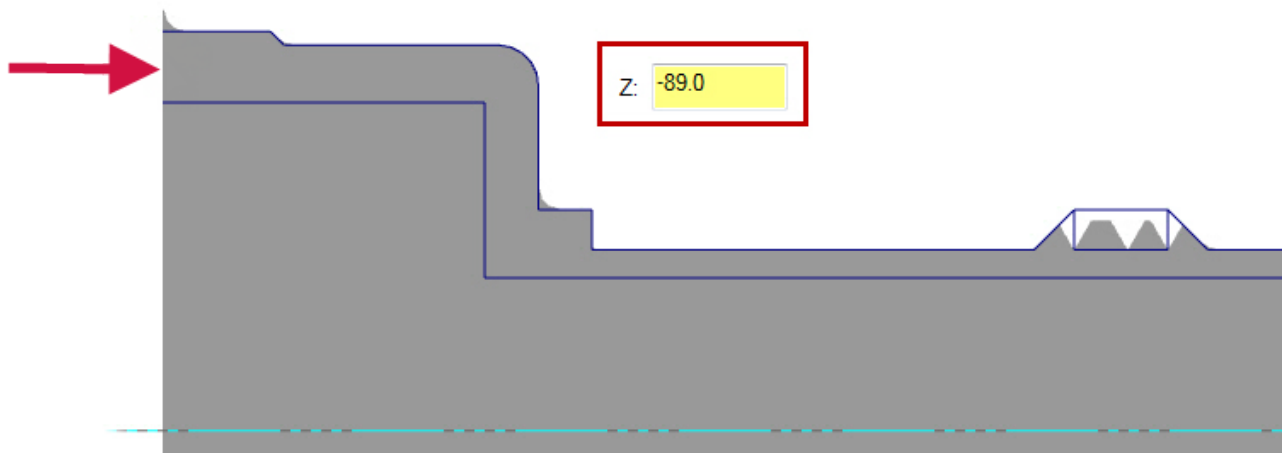


## Enter new stock and chuck jaw positions

Use this procedure to define the new stock position and the final position of the chuck jaws after the stock flip.

1. To position the transferred stock, click the **Select** button under **Transferred Position**, and choose any point on the back edge of the part.





As indicated by the image in the dialog box, the point you select (Z -89.0) will be transferred to Z0 after the stock flip operation





**Note:** The point you choose does not have to be on the face of the stock. You can choose any convenient reference point.

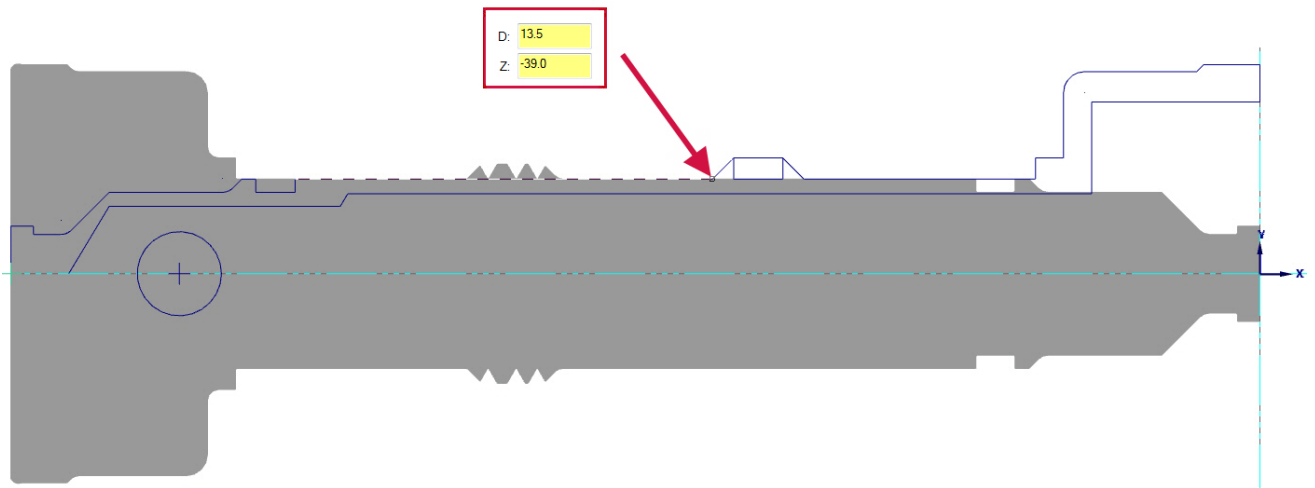
2. To set the final position of the chuck jaws, click the **Select** button under **Final Position**, and chose the point from the graphics window, as indicated in the graphic [on the facing page](#).

This is where the chuck jaws will grip the stock after the stock flip operation.

Chuck Position

☐ Diameter only

Original Position:	Final Position:
 <p>D: 38.2</p> <p>Z: -94.0</p> <p>Select...</p>	 <p>D: 38.2</p> <p>Z: -94.0</p> <p>Select...</p>



3. Click **OK** to create the operation.

The stock and chuck display in the graphics window in their transferred positions.



4. Save the part.

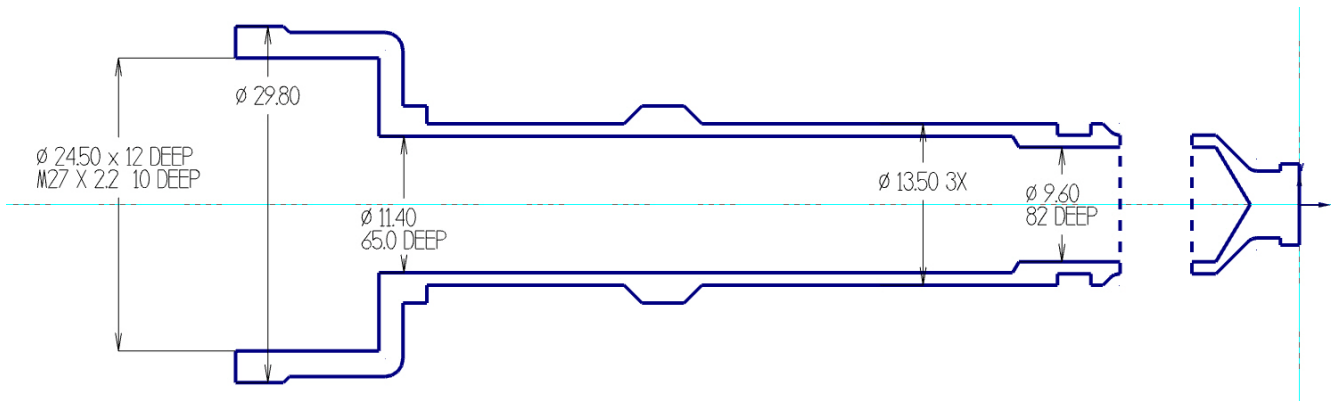
The part is now in the correct position for you to machine the features of the back end and inner diameter.



## CHAPTER 6

# MACHINING THE INNER DIAMETER

In this lesson you create the toolpaths necessary to machine the inner diameter (ID) of the hose nozzle. You may notice that many of the toolpaths need only slight adjustments applied to work on the inner diameter. After you have created the toolpaths, use Verify to inspect your work.



### Goals

- Create new tools based on existing tools.
- Create toolpaths on the inner diameter of the part.
- Use Verify's display tools to inspect your work.

### Exercise 1: Creating New Tools in the Lathe Tool Manager

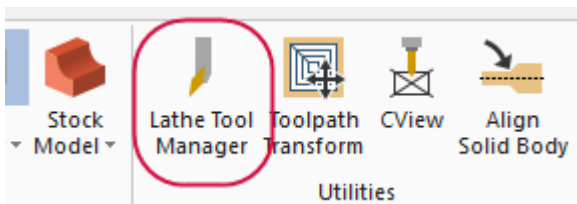
The interior of the hose nozzle part is hollow. You use two drill operations to cut the longest two inner diameters. Both operations require creating new tools that match the diameter of each drill hole (9.6 mm and 11.4 mm). The third and widest inner diameter requires roughing and finishing tools that fit within dimensions of the bore (24.5 mm), as well as an ID threading tool that can cut to the thread standard for garden hoses (GHT).

Although your shop may choose to use standard tools to cut these toolpaths, in this exercise, you create all of these tools in the Lathe Tool Manager.

#### Create the 9.6 mm drill

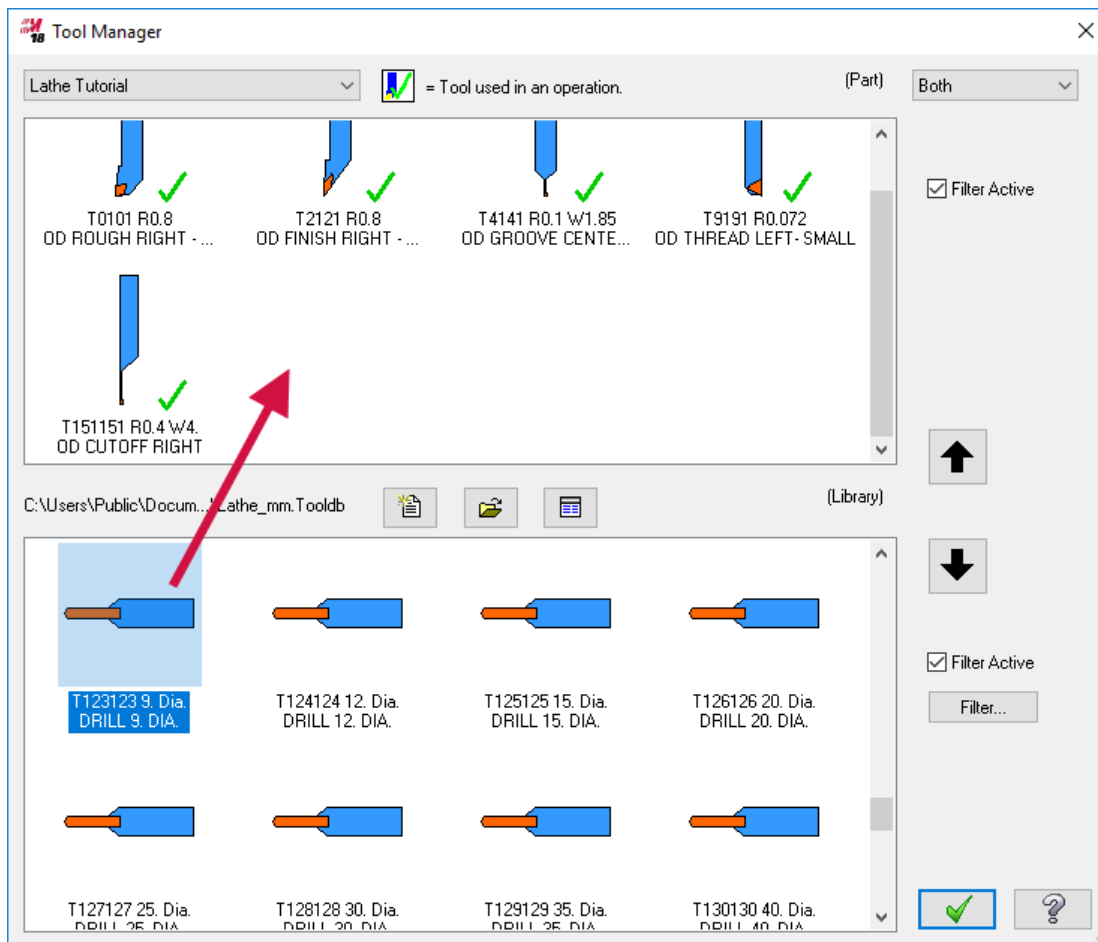
The 9.6 mm drill is based upon a 9 mm drill that is already in the tool library.

1. Select **Lathe Tool Manager** from the **Utilities** group on the **Lathe Turning** tab.



The Tool Manager opens. The upper window displays the tools used in the current Mastercam file.

2. Click and drag **T123123 9. Dia DRILL 9. DIA** from the library window to the part window.

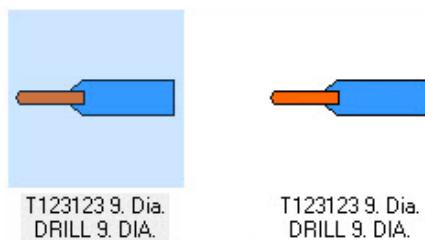


When you create or work on a tool in the part window, you are working on a tool definition that is stored in the machine group. That means if you edit a tool from the part window, you are only changing the definition in the machine group and not the tool library. If you import the tool from another Mastercam file, the changes you make here will not appear in the original file.

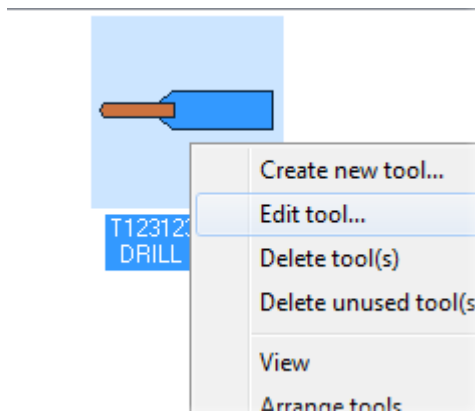
**Note:** Select the **Filter Active** option to filter the list of tools. Click the **Filter** button to change the filter criteria with the **Lathe Tool Filter** dialog box.

3. Using the right-click menu, copy and paste the 9 mm drill in the part window.
4. When prompted, click **Yes** to add a similar tool.

A copy of the 9 mm drill is added to the part window.

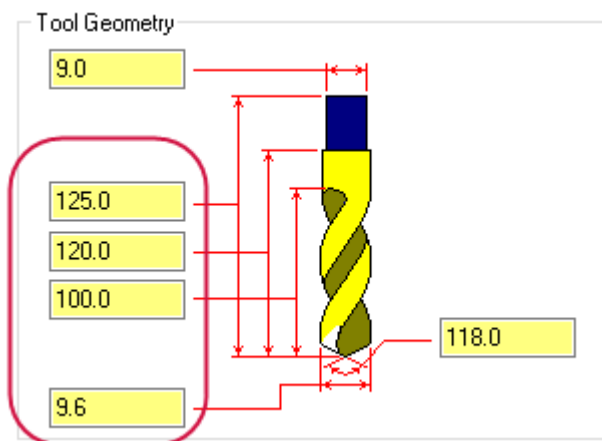


5. Right-click the copied tool, and select **Edit tool** from the menu.
- The **Define Tool** dialog box opens.

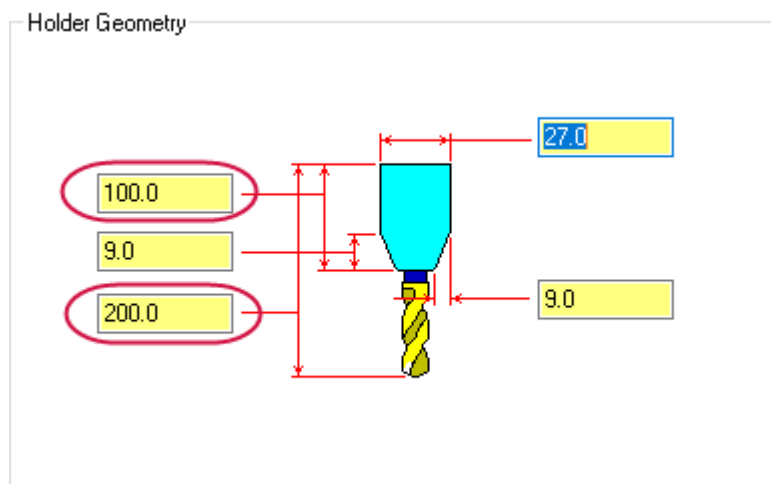


6. Edit the tool geometry as shown.

These values define the dimensions of the drill body.



7. Click the **Holders** tab, and edit the dimensions of the holder.



8. Click the **Parameters** tab and rename the tool.


Type - Drill/Tap/Reamer Tools Holders Parameters

Save To Library...  
Draw Tool...  
Setup Tool...

Program Parameters  
Tool number: 123 Tool station number: 123  
Tool offset number: 123 Tool back offset number: 123

Default Cutting Parameters  
Feed rate: 0.05 ☒ mm/rev ☐ mm/min  
Plunge rate: 0.0 ☐ mm/rev ☒ mm/min  
% of Material Feed/Rev: 50.0 Coolant...  
Spindle speed: 200 ☐ CSS ☒ RPM  
% of Material CS: 50.0 Compute From Material...

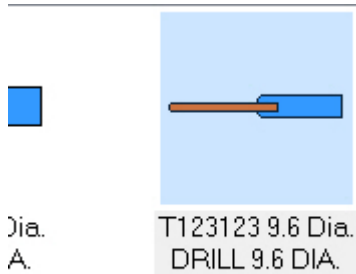
Drilling Parameters  
First peck: 4.5 Retract amount: 0.9  
Subsequent peck: 0.0 Dwell: 0.0  
Peck clearance: 1.8 Cycle: Peck Drill

Compensation  
 Tool Clearance... ☒ Metric values  
Tool name: DRILL 9.6 DIA.  
Manufacturer's tool code:  
Machine side connection type: Generic  
☐ Requires tool locator

✓ ✗ ?

9. Click **OK** to finalize your changes.

The new tool profile displays in the part window.

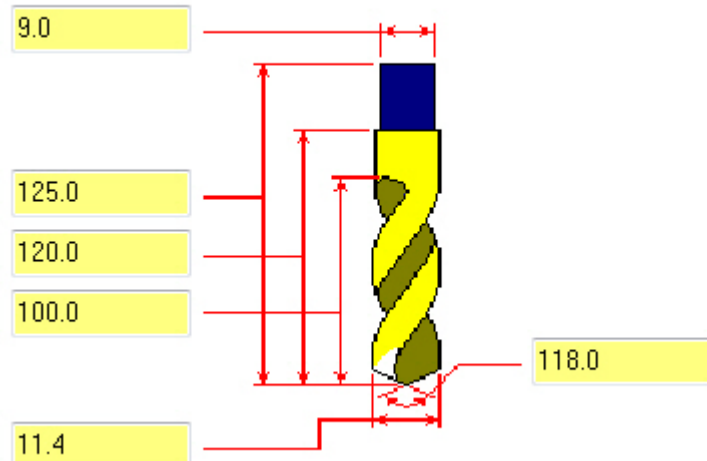


## Create the 11.4 mm drill

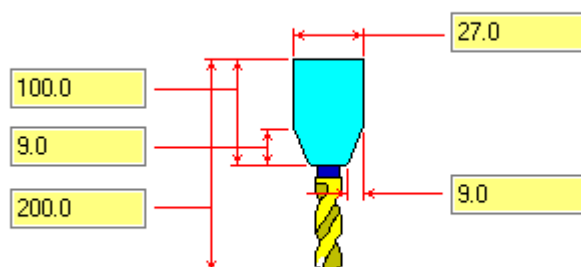
Copy and paste the 9 mm drill in the part window again, and use the same procedure you used to create the 9.6 mm drill to create a 11.4 mm drill.

- Enter the following values to create the tool and holder dimensions.
- Rename the tool: **DRILL 11.4 DIA.**

## Tool Geometry



## Holder Geometry



The original tool and the two new tools display in the part window.



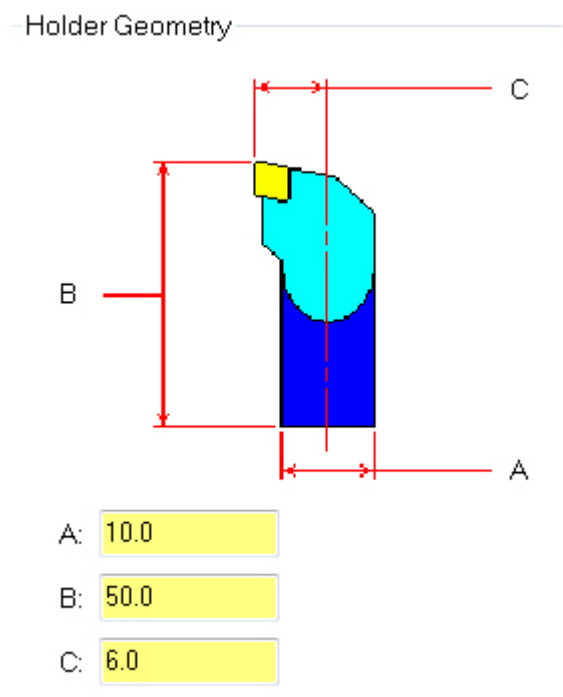
## Create the ID roughing tool

1. Click and drag **T7171 R0.4 ID ROUGH MIN 16. DIA - 80 DEG.** from the library window to the part window.
2. Right-click the tool, and select **Edit tool** from the menu.  
The **Define Tool** dialog box opens.
3. Choose **05** from the **IC Dia./Length** drop-down.

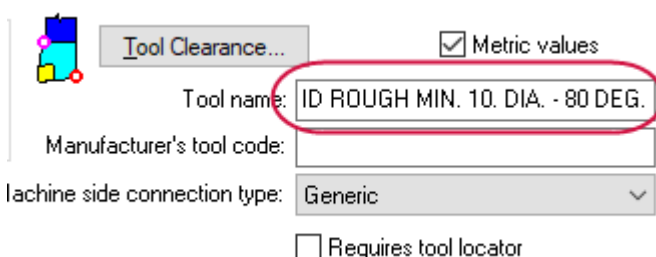


This value defines the insert's IC diameter or length (depending on the selected shape). The IC diameter (IC = inscribed circle) is determined by placing a circle in the insert shape and measuring the circle's diameter.

4. Leave the other parameters on this tab at their default values, and click the **Boring Bars** tab.
5. Change the holder geometry as shown to fit the newly edited insert.



6. Click the **Parameters** tab, and rename the tool: **ID ROUGH MIN. 10. DIA. - 80 DEG.**



7. Click **OK** to finalize your changes.

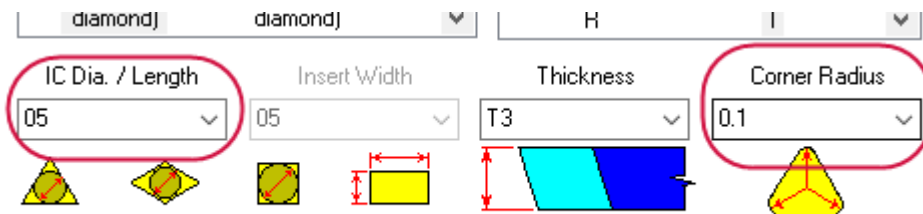
## Create the ID finishing tool

Create a finishing tool for the bore you will turn.

1. In the Lathe Tool Manager's part window, use the right-click menu to copy and paste the tool you just edited (ID ROUGH MIN. 10. DIA. - 80 DEG).
2. When prompted, click **Yes** to add a similar tool.
3. Right-click the copied tool, and select **Edit tool** from the menu.

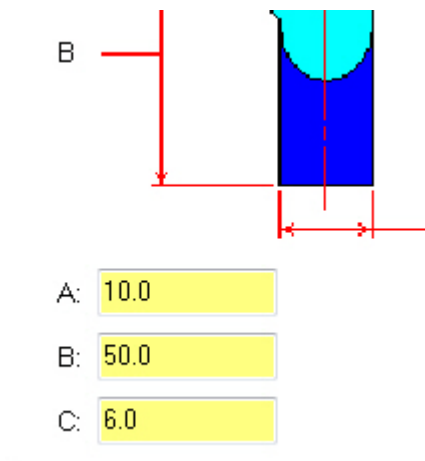
The **Define Tool** dialog box opens.

4. Enter the following values to create the insert's inscribed circle diameter and a smaller corner radius.



**Note:** You can either enter dimensions directly or select them from the drop-down menu.

5. Keep the holder geometry as shown to fit the newly edited insert.



6. Name the new tool: **ID FINISH MIN. 10. DIA. - 80 DEG.**
7. Click **OK** to finalize your changes.

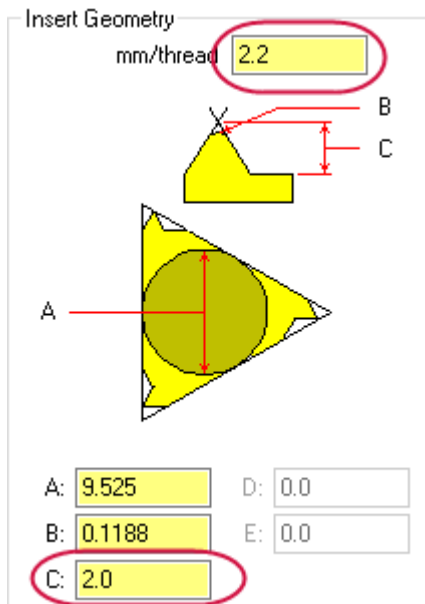
## Create the ID threading tool

1. Click and drag **T101101 R0.036 ID THREAD MIN. 20. DIA.** from the library window to the part window.
2. Right-click the tool, and select **Edit tool** from the menu.

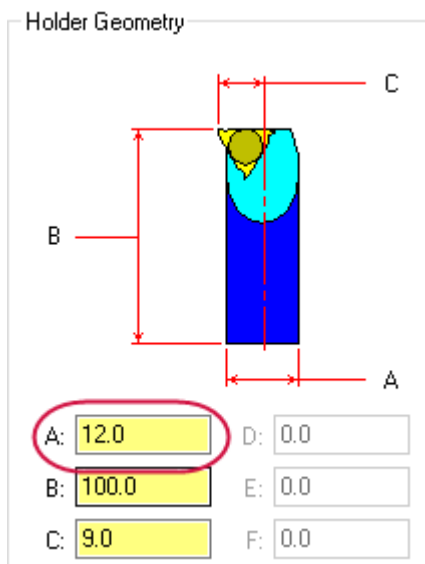
The **Define Tool** dialog box opens.

3. Edit the insert geometry by increasing the lead (**mm/thread**) and cutting depth (**C**) of the insert.

Mastercam uses these values to calculate the default feed rate based on the spindle speed value entered on the **Parameters** page.



- On the **Holders** tab, decrease the diameter of the holder (**A**) to **12.0**.



- Click the **Parameters** tab, and rename the tool: **ID THREAD MIN. 12. DIA.**
- Click **OK** to finalize your changes.

You have now finished creating the tools that you need to machine the inner diameter of the part.

- Click **OK** to exit the Lathe Tool Manager.

## Exercise 2: Facing the Back of the Part

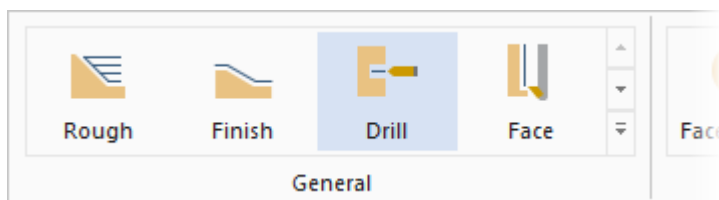
It is a best practice to prepare the back face of the part for further machining. Create a face toolpath using the default roughing tool (T0101 R0.8 OD ROUGH RIGHT - 80 DEG) to finish the back face of the part to Z0.

Use the same procedure to face the back of the part as you used to face the front. If you feel you still need instruction to create this operation, refer back to ["Facing the Part" on page 25](#).

## Exercise 3: Drilling the First Inner Diameter

Drill operations in Lathe are on the center line of the part and are placed by default at the part origin.

1. Select **Drill** from the **Lathe Turning** tab.

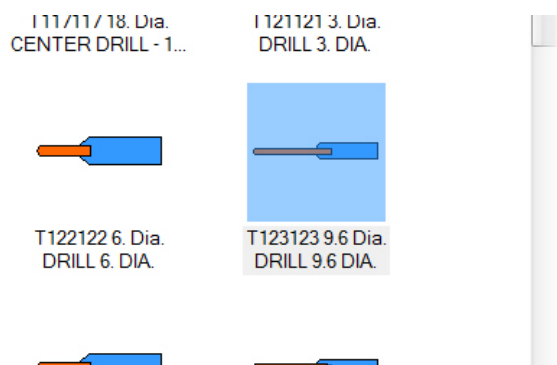


The **Lathe Drill** dialog box opens.

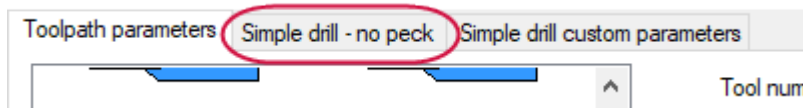
Unlike most toolpaths, lathe drill toolpaths do not require you to select geometry before entering the toolpath parameters.

2. Select the 9.6 mm drill from the tool window.

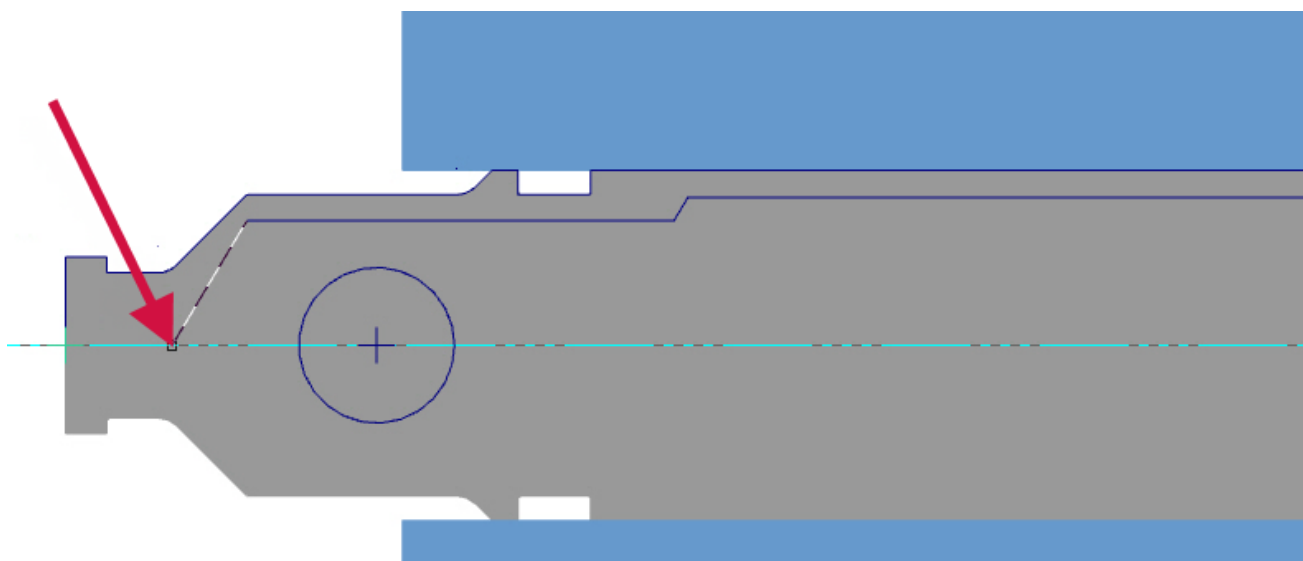
If desired, enter comments for the toolpath.



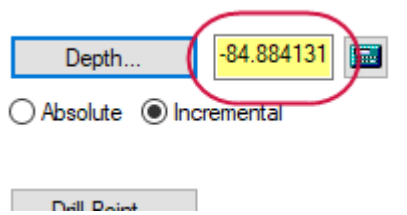
3. Click the **Simple drill - no peck** tab.



4. Click the **Depth** button to define the bottom of the drill hole from a point in the graphics window.
5. Select the point indicated.



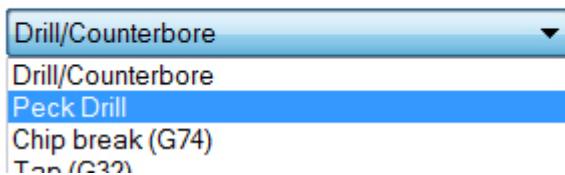
Mastercam returns you to the **Lathe Drill** dialog box and automatically enters the correct value in the **Depth** field.



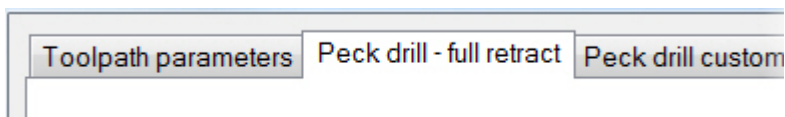
6. Choose **Peck Drill** from the **Cycle** drop-down list.

#### Drill Cycle Parameters

Cycle:

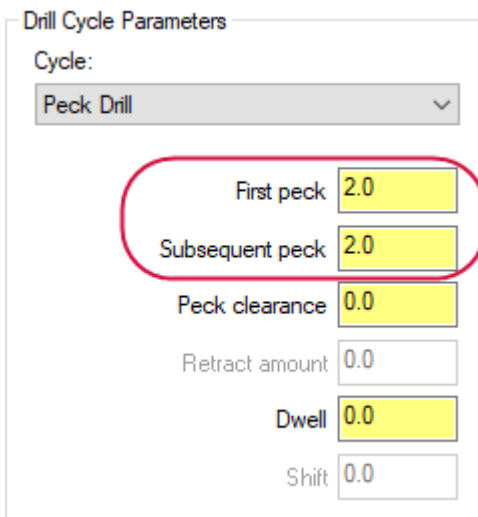


The name that appears on this tab depends on the cycle you select in the **Drill Cycle Parameters** drop-down list. The cycles which are available to you depend on the active control definition.



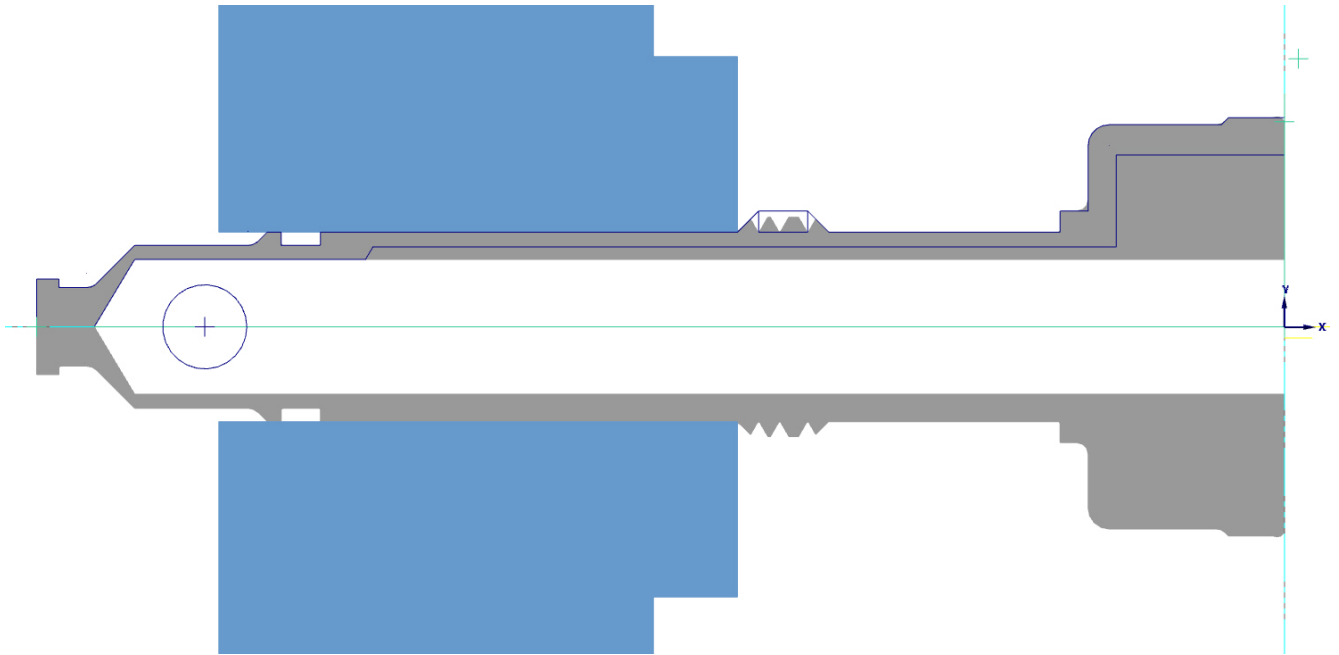
The parameters tab changes from **Simple drill - no peck** to **Peck drill - full retract**.

7. Set the depth for the first and all subsequent peck moves to **2.0** mm.



8. Click **OK** to complete the operation.

The graphics window displays how the first inner diameter is drilled.

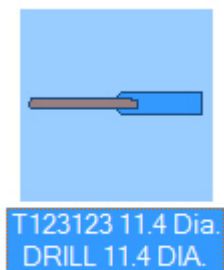


## Exercise 4: Drilling the Second Inner Diameter

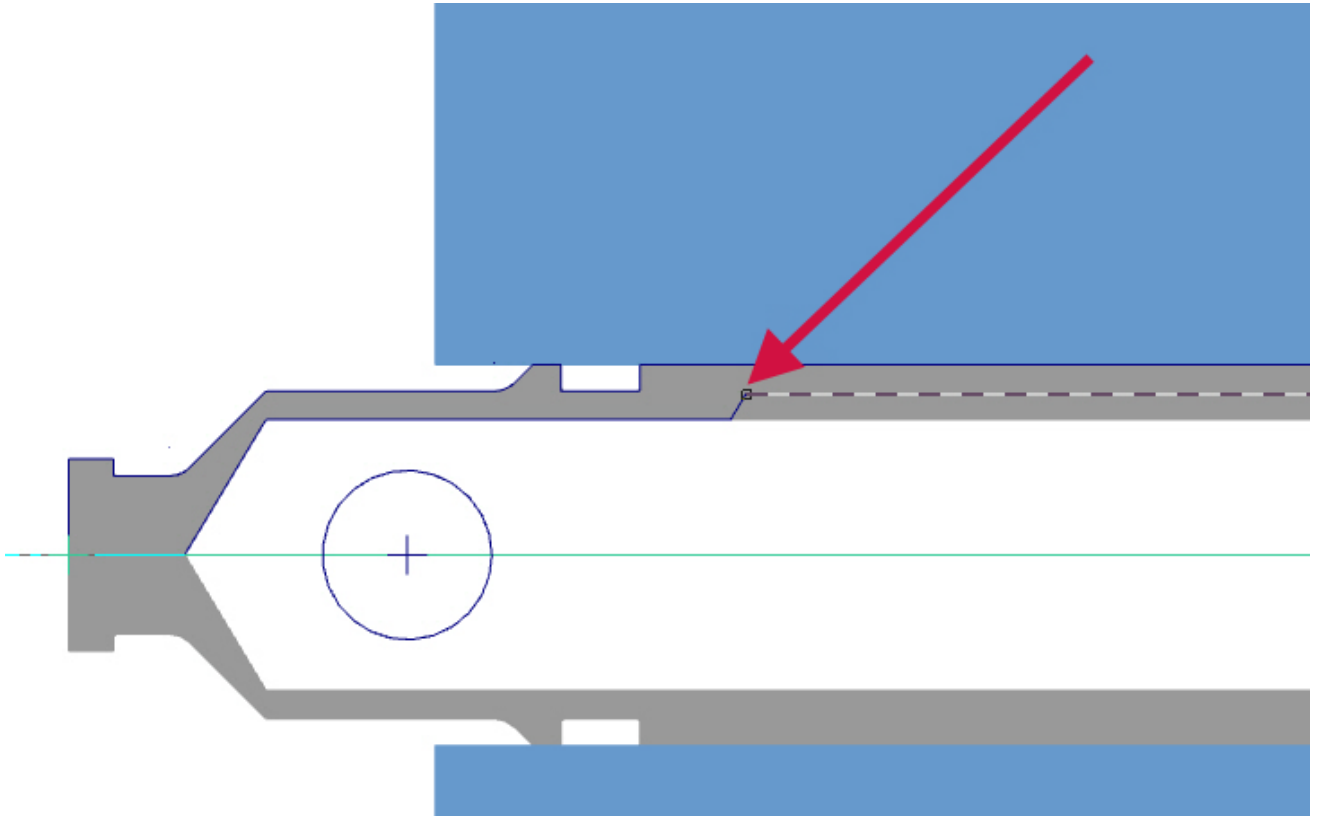
This operation uses the depth calculator to adjust the drilling depth of the tool.

1. Select **Drill** from the **Lathe Turning** tab.
2. Select the 11.4 mm drill from the tool window.

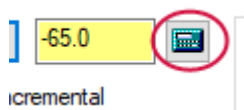
If desired, enter comments for the toolpath.



3. Click the **Peck drill - full retract** tab.  
Mastercam remembers the last drill cycle you selected.
4. Click the **Depth** button to define the bottom of the new toolpath's drill hole.
5. Select the point indicated.

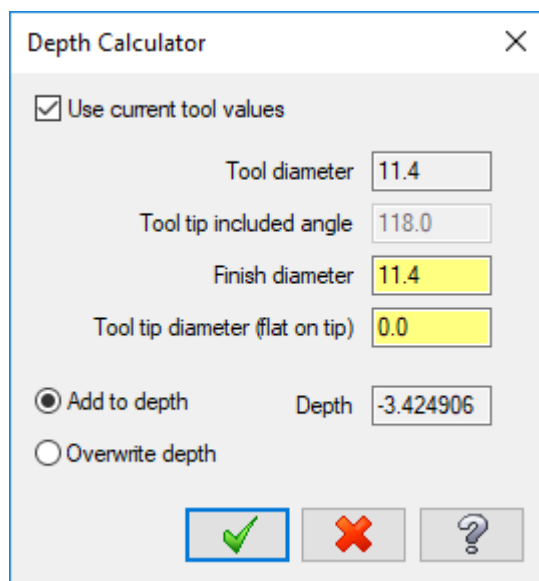


6. Click the **Depth Calculator** button.



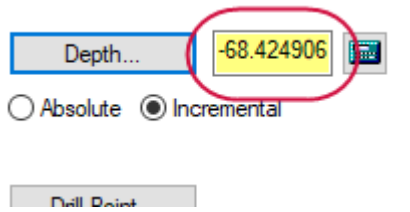
The **Depth Calculator** dialog box opens.

With the **Use current tool values** option selected, Mastercam displays the tool diameter, tool tip included angle, and tool tip diameter for the tool you choose on the **Toolpath** page. It calculates the proper drilling depth and adds it to the existing depth.



7. Click **OK** to return to the **Lathe Drill** dialog box.

The depth of the bottom of the drill point is entered into the field.



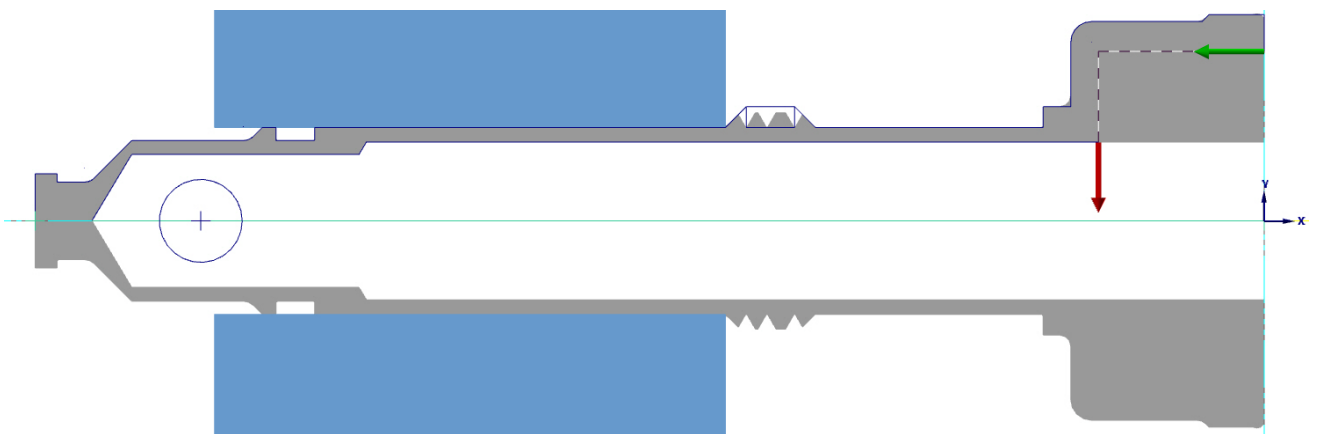
8. Leave all other parameters in the dialog box as is, and click **OK** to create the operation.



## Exercise 5: Roughing and Finishing the Third Inner Diameter

In this exercise, you bore the third inner diameter with roughing and finishing operations. With the exception of a standard finish toolpath, creating roughing and finishing operations on the inner diameter is very similar to creating them on the outer diameter as you did in [Chapter 2](#).

1. Select the **Rough** toolpath from the **Lathe Turning** tab.  
The **Chaining** dialog box displays and you are prompted to select an entry point.
2. Click the geometry to create a partial chain as shown.



3. Click **OK** in the **Chaining** dialog box to accept the chain.

The **Lathe Rough** dialog box opens.

4. Select the ID roughing tool you created [on page 85](#) **ID ROUGH MIN. 10. DIA - 80 DEG.**



5. Type **ID - Rough** in the **Comment** field.
6. Change the following cutting values on the **Rough parameters** tab.

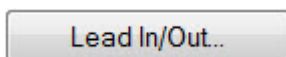
Incremental  
Depth of cut: 1.0  
Increments of: 0.1  
Min cut depth: 0.01  
Stock to leave in X: 0.5  
Stock to leave in Z: 0.05  
Variable depth: 0.0 % of depth

Comp in control  
Compensation direction: Left  
Roll cutter around corners: All

Semi Finish...  
Lead In/Out...  
Plunge Parameters...  
Filter...  
Tool Inspection...  
Chip Break...  
Section turning...

Stock Recognition  
Use stock for outer boundary  
Adjust Stock

- Change the **Depth of cut** to **1.0**.
  - Decrease the **Stock to leave in Z** to **0.05**.
  - Select **Use stock for outer boundary** from the **Stock Recognition** drop-down.
7. Click the **Lead In/Out** button.



The **Lead In/Out** dialog box opens.

- a. Click the **Lead out** tab.

Lead in Lead out

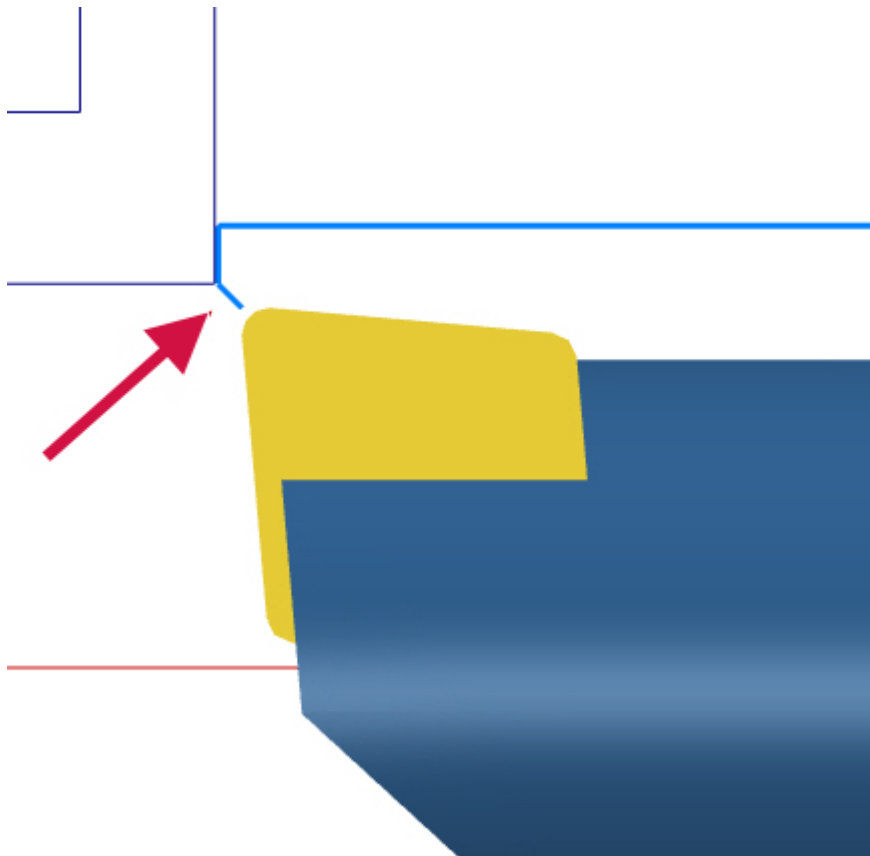
Adjust Contour  
☐ Extend / shorten start of contour

Entry V  
☒ Use  
☐ Fixed

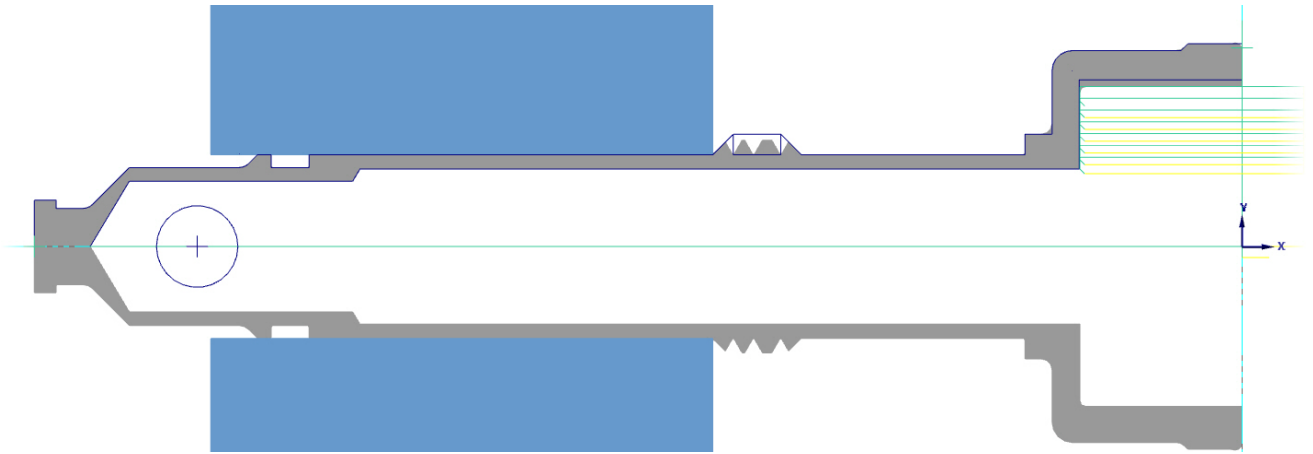
- b. Enter **0.5** in the **Length** field.

The screenshot shows the 'Lead in / Lead out' dialog box in Mastercam. The 'Lead out' tab is active. Under 'Adjust Contour', the 'Extend / shorten end of contour' checkbox is unchecked, with 'Amount' set to 0.0 and 'Extend' selected. Under 'Exit Vector', the 'Use exit vector' checkbox is checked. 'Fixed Direction' is set to 'None'. The 'Angle' is -45.0. The 'Length' field is highlighted with a red circle and contains the value 0.5. The 'Resolution (deg.)' is 45. Under 'Auto-calculate vector', the 'Automatically calculate exit vector' checkbox is unchecked, and the 'Minimum vector length' is 0.0. The 'Feed rate' is 0.1 mm/rev.

This is the length of the vector the tool follows as it retracts from the part.



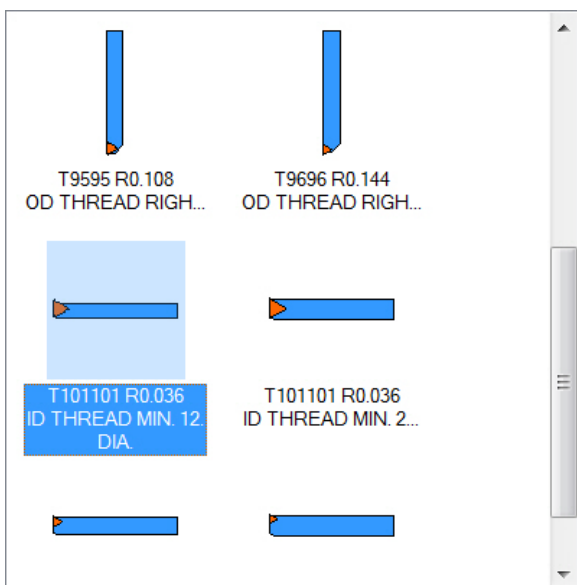
- c. Click **OK** to return to the **Lathe Rough** dialog box.
8. Keep all other parameters in the dialog box at their default values.
  9. Click **OK** to create the rough toolpath.



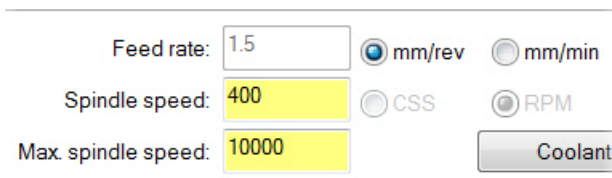
10. Create a finishing toolpath. Use **ID FINISH MIN. 10. DIA. - 80 DEG** to finish the bore you just roughed. (This is the finishing tool you created [on page 86](#).) Leave all settings at their default values.
11. Save the part.

## Exercise 6: Adding an ID Thread

1. Expand the **General** gallery, and choose **Thread** from the **Lathe Turning** tab.
2. From the **Toolpath parameters** tab, select the thread tool you created [on page 87](#): **ID THREAD MIN 12 DIA.**



3. Enter **400** into the **Spindle speed** field, and select the **mm/rev** option for the **Feed rate**.

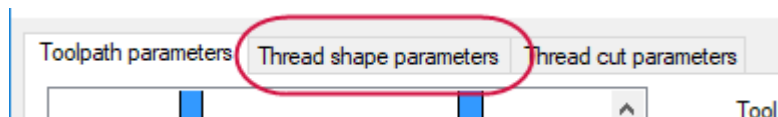


4. Type **ID - Thread** in the **Comment** field.

Comment

ID - Thread

5. Keep all other parameters on this page at their default values.
6. Click the **Thread shape parameters** tab.



7. Refer to the illustration below to enter the following thread dimensions:

Lead:  ☐ threads/mm ☒ mm/thread

Included angle:

Thread angle:

Major Diameter...  27.

Minor Diameter...  24.5

Thread depth:

Adjusted Diameters

Major/Minor Diameters

☒ Large end of taper ☐ Small end of taper

Thread Form

End Position...  Start Position...

Thread orientation: ID  ☐ Negative X

Taper angle:

Allowances

☒

Major allowance:

Minor allowance:

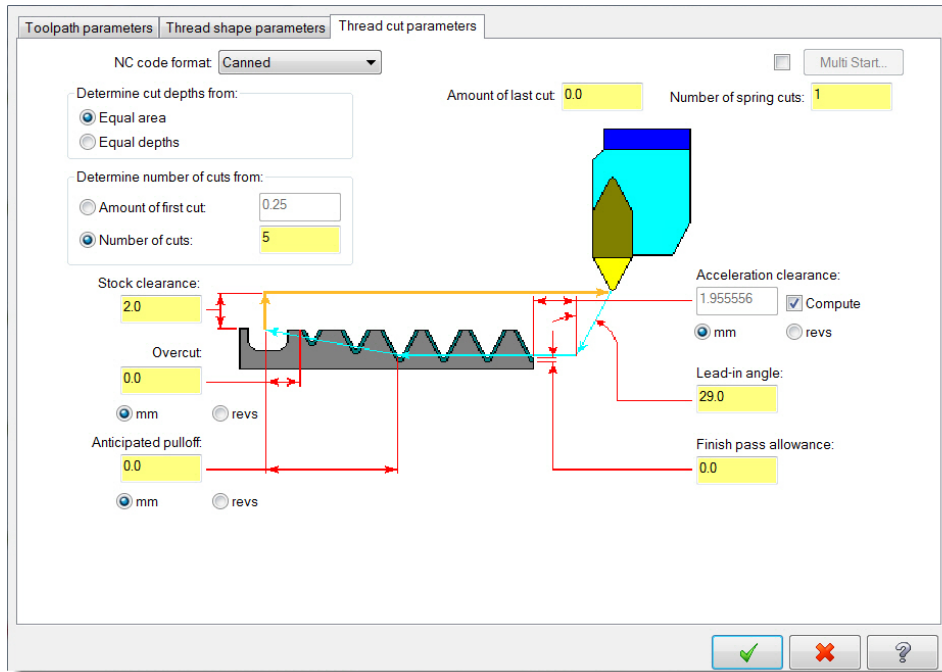
Allowance tolerance:

- Lead
- Major and Minor Diameters
- Start and End Positions

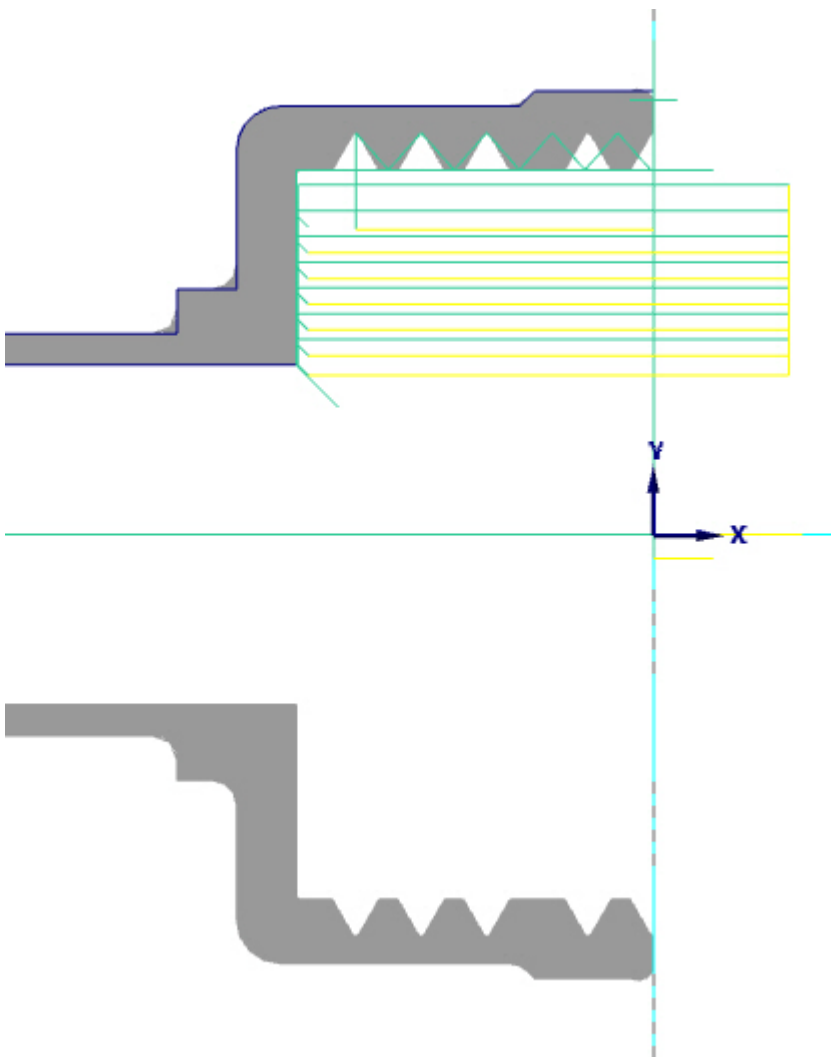
The blueprint [on page 81](#) also shows these values.

**Note:** If you are cutting a standard thread, click **Select** from table to display a list of standard thread tables. Mastercam will automatically fill in the fields for the thread dimensions with values from these tables. See Mastercam Help for more information.

8. Confirm that the **Thread orientation** is **ID**.
9. Click the **Thread cut parameters** tab.
10. If necessary, select the **Compute** option to calculate the required acceleration clearance.



11. Keep all other parameters on this page at their default values.
12. Click **OK** to create the toolpath.



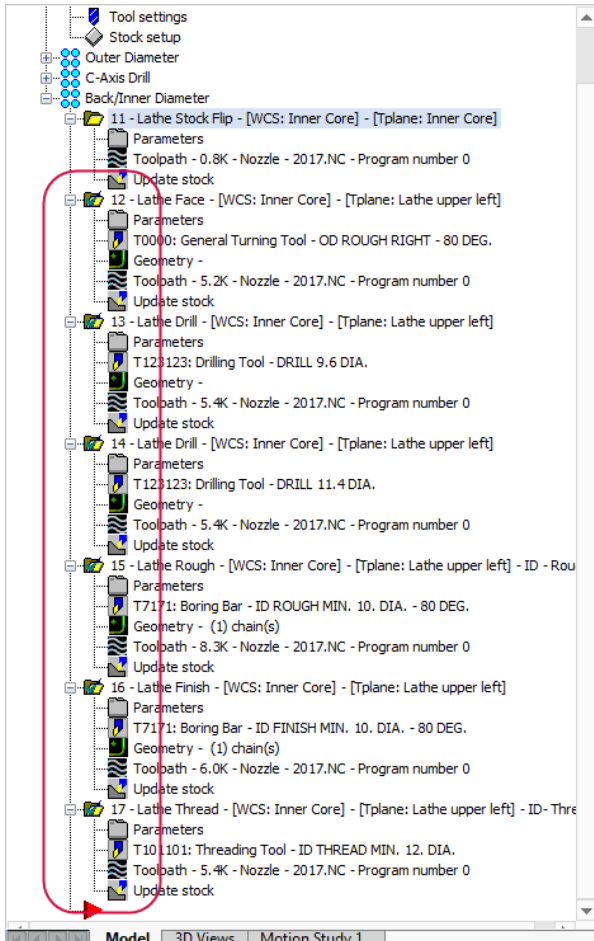
13. Save the part.

## Exercise 7: Refining Your Verification Results

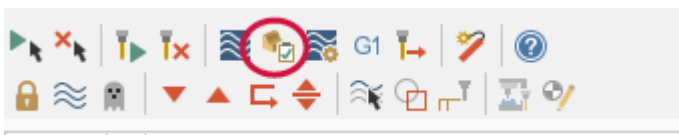
In [Adding Grooves and Threads](#), you used **Accurate Zoom** to more clearly see your work. In this exercise, you use Verify's Part Sectioning tools to view the cuts you made along the inner diameter.

1. Press **[Shift+ Click]** to select operations 12 through 17 in the Toolpaths Manager.

These are all the operations you created on the inner diameter.

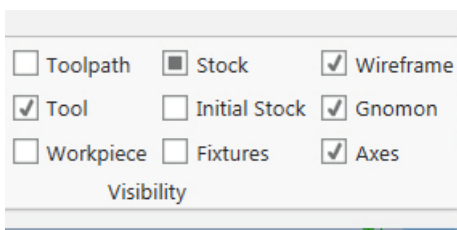


2. Click the **Verify selected operations** button.

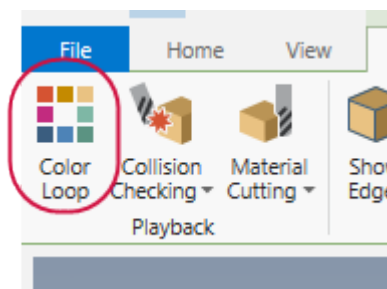


Verify displays in the Mastercam Simulator window.

3. Hide the fixture, and change the stock opacity to translucent.



- Click the **Verify** tab and turn the **Color Loop** on to view each operation's change to the stock as a different color.

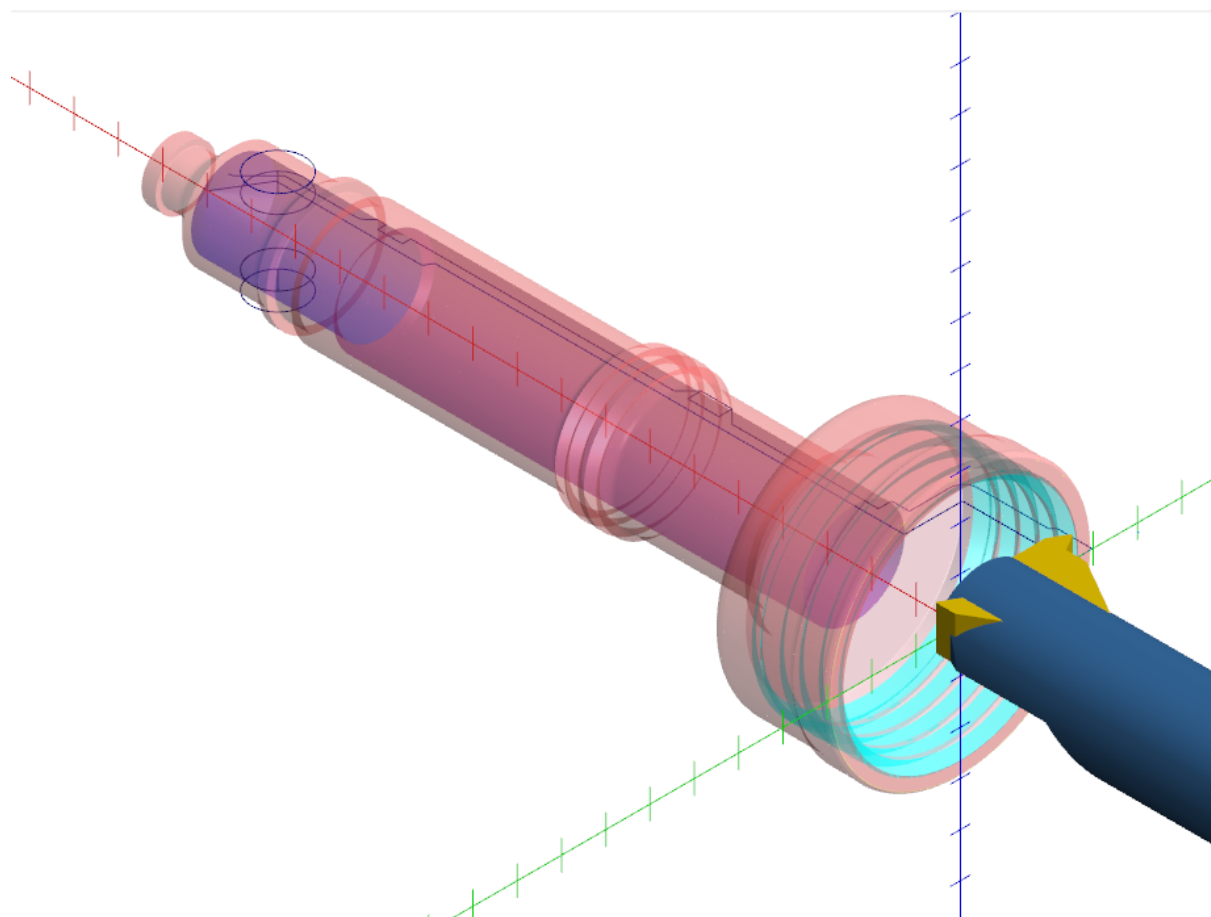


The color of each operation is also represented as a different color on the playback bar.



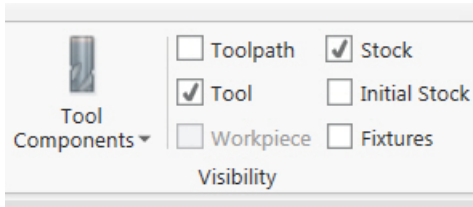
**Note:** Improve your view by panning, zooming, and rotating.

- Click the **Play** button or press [R] to verify the toolpath.

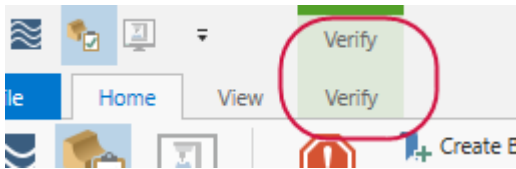


Although you are able to view the operations through the translucent stock, Mastercam Simulator's clipping plane and part sectioning tools offer an even better view of the part's interior.

- Change the stock back to opaque.

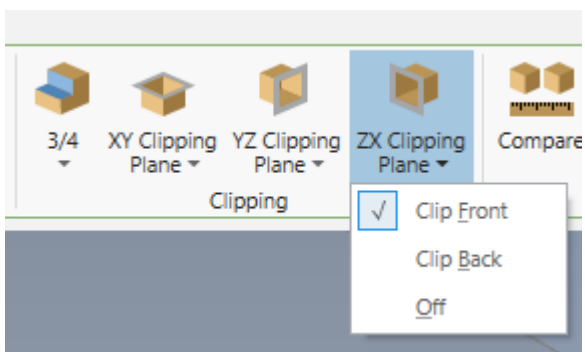


7. Select the **Verify** tab.

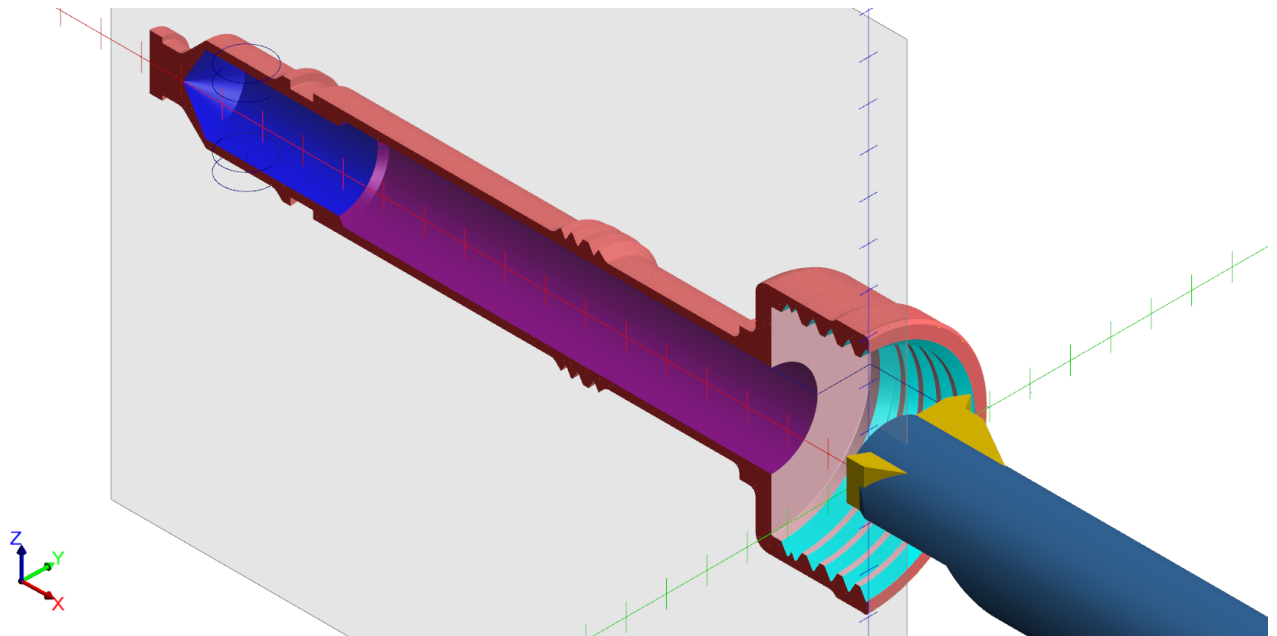


8. Select the **ZX Clipping Plane** button, and **Clip Front** from the drop-down menu.

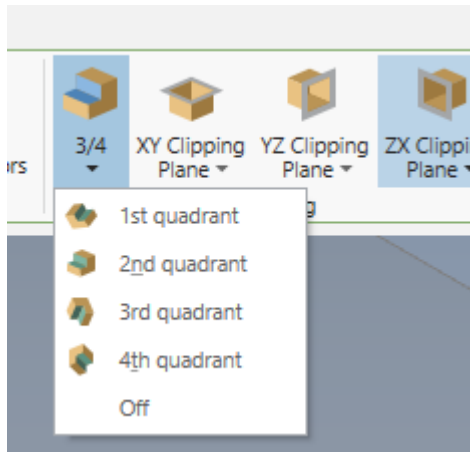
A cross-section is drawn through the stock's ZX plane.



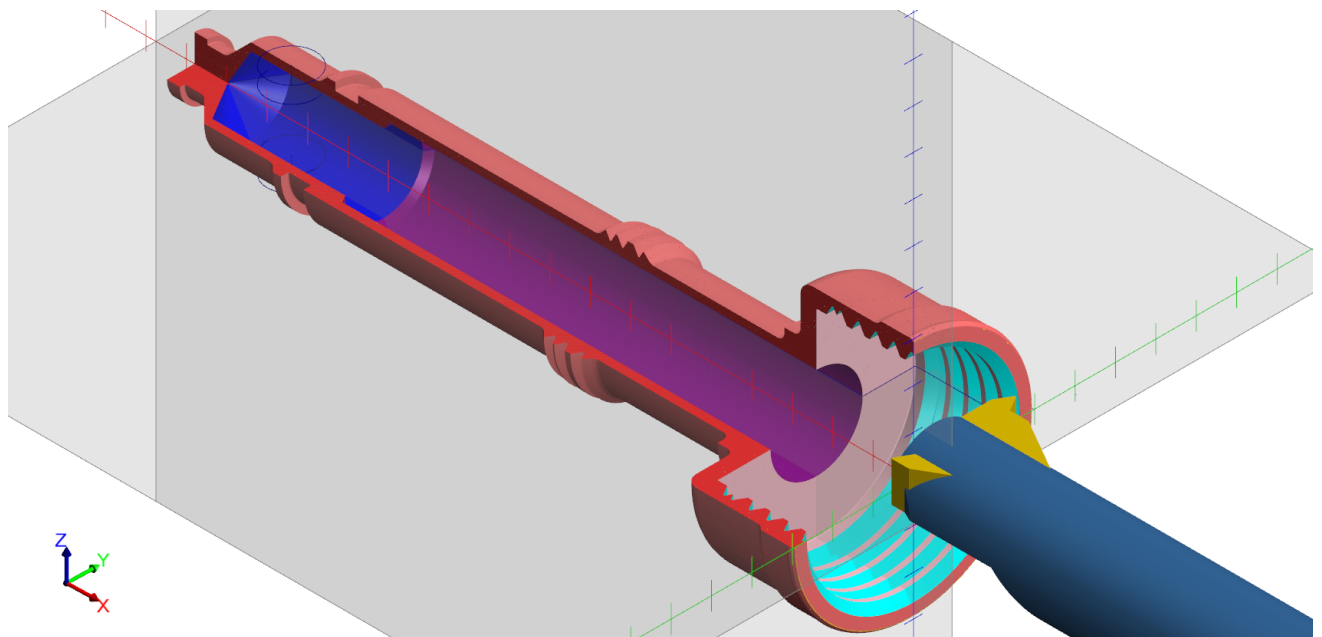
**Note:** Click and drag the plane to position the cross-section.



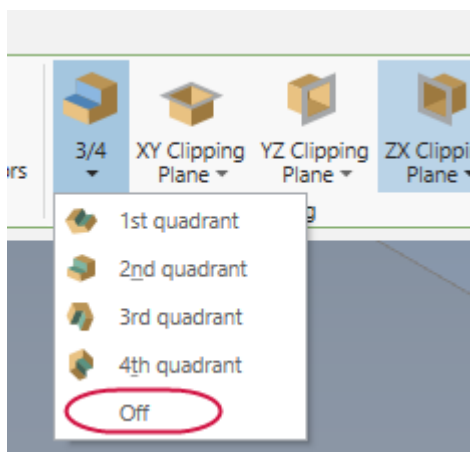
9. Select **Off** from the drop-down menu to stop displaying the clipping plane.
10. Select a quadrant from the **3/4** sectioning menu.



Part sectioning displays your stock in 3/4 sections using a combination of two clipping planes. Once you select a section, you can click and drag either plane to adjust the view.



11. Click **3/4**, **Off** to turn off part sectioning.



12. Close the Machine Simulator.

You have finished cutting the part. In the next lesson, you organize your tools and post your work to the machine.

## CHAPTER 7

# POST OUTPUT

Posting uses a specialized processing program to convert your toolpaths to NC code.

Tool numbers that are output to your NC file come from the **Toolpath parameters** tab or **Tool** page for each operation. When you select a tool for an operation, Mastercam generates a default tool number and default tool offsets. If your machine only accepts a limited range of tool numbers, Mastercam lets you reassign tool numbers for operations in a machine group.

In the final lesson of this tutorial, you renumber the tools used in the operations you created, and then post the entire set to the NC file.

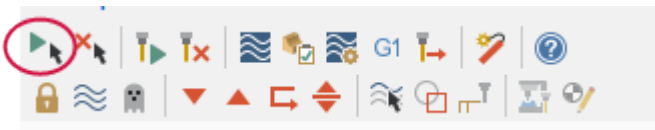
### Goals

- Renumber all tools used to cut the part.
- Post your work for inspection.

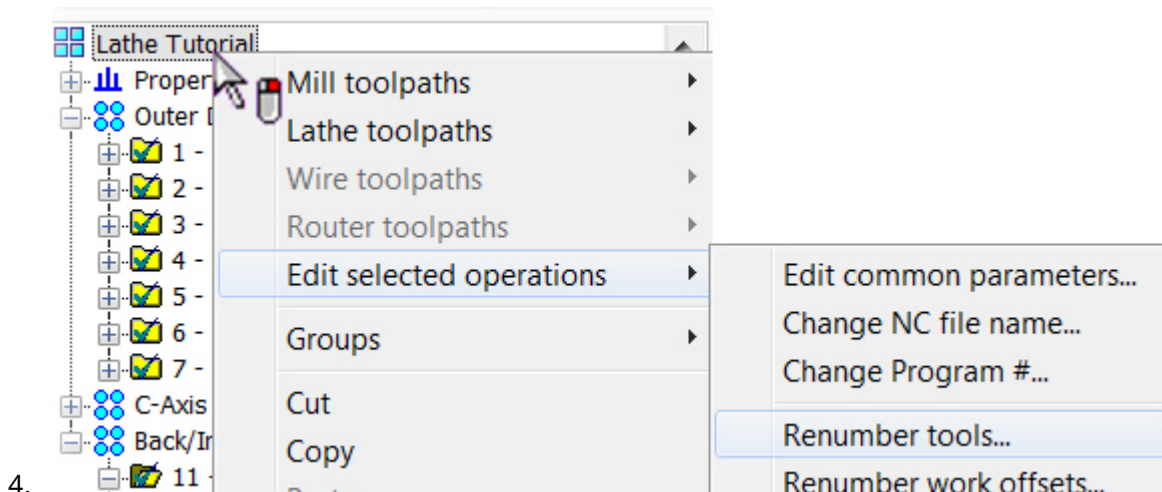
### Exercise 1: Renumbering Tools

In this exercise, you renumber the tools for each operation.

1. Select all operations in the Toolpaths Manager.

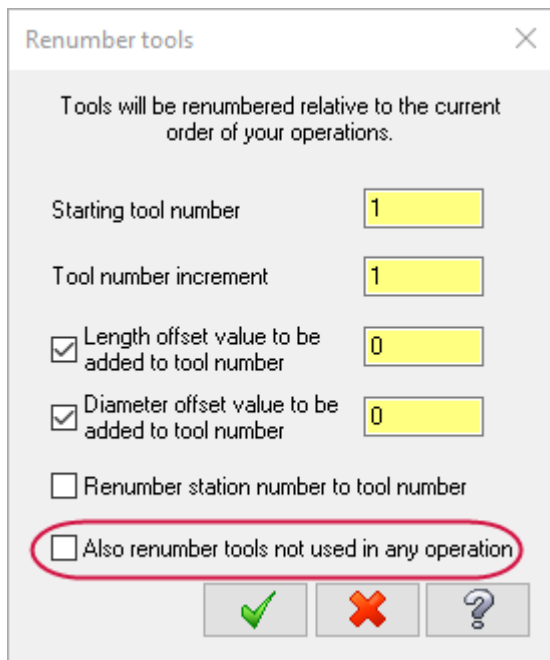


2. Right-click the machine group.
3. Select **Edit selected operations, Renumber tools** from the menu.



The **Renumber tools** dialog box displays.

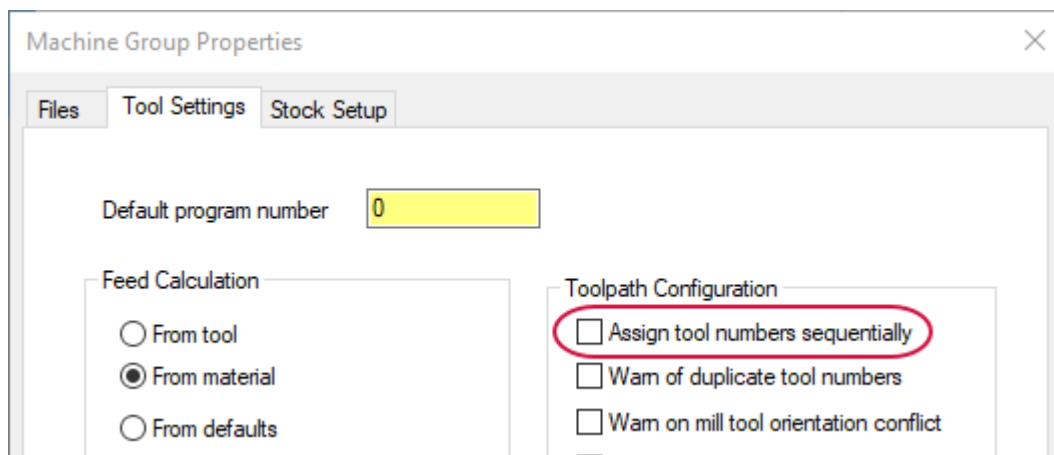
5. If necessary, clear the last option to renumber tools that are not used in any operation.



6. Click **OK**.

The tools in the Toolpaths Manager are renumbered relative to the current order of the operations.

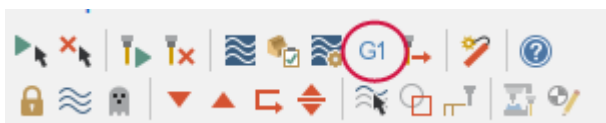
Use the procedure above to reassign tool numbers after you have already created toolpaths. To automatically assign sequential tool numbers as you create operations, select the option on the **Tool Settings** tab in the **Machine Group Properties** dialog box before you create toolpaths.



## Exercise 2: Posting

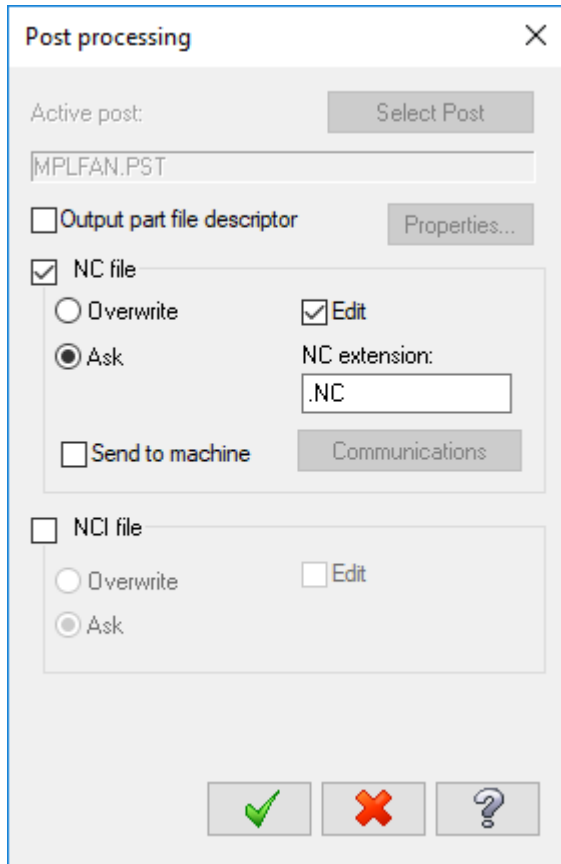
In this exercise, you post the operations in the machine group.

1. If necessary, select all operations in the Toolpaths Manager.
2. Click the **Post selected operations** button in the Toolpaths Manager.



If you have not selected all the operations in a machine group, Mastercam will ask if you want to post all the operations.

The **Post processing** dialog box displays. Mastercam uses these settings to handle the files that are generated when posting.



3. Click **OK**.

The **Save As** dialog box displays.

4. Rename the file, or click **Save** to accept the default NC file name.

Mastercam posts the file, and it is opened in your default file editor. Examine the NC file to before sending it to the machine.

# CONCLUSION

Congratulations! You have completed the *Mastercam Lathe Tutorial*! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help, Tutorials** from the **File** tab.

## Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's **File** tab or by pressing **[Alt+H]** on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 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.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help, Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare 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).
- *Online Communities*—You can find a wealth of information at [www.mastercam.com](http://www.mastercam.com). For tech tips and the latest Mastercam news, follow us on Facebook ([www.facebook.com/mastercam](http://www.facebook.com/mastercam)), Twitter ([www.twitter.com/mastercam](http://www.twitter.com/mastercam)), or Google+ ([plus.google.com/+mastercam](https://plus.google.com/+mastercam)). Visit our YouTube channel to see Mastercam in action ([www.youtube.com/user/MastercamCadCam](http://www.youtube.com/user/MastercamCadCam))! Registered users can search for information or ask questions on the Mastercam Web forum, [forum.mastercam.com](http://forum.mastercam.com), or use the knowledgebase at [kb.mastercam.com](http://kb.mastercam.com).

## Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at [techdocs@mastercam.com](mailto:techdocs@mastercam.com).





**ATTENTION! UPDATES MAY BE AVAILABLE.  
PLEASE REFER TO [MASTERCAM.COM/SUPPORT](http://mastercam.com/support)  
FOR THE LATEST DOWNLOADS.**

**CNC Software, Inc.**  
671 Old Post Road  
Tolland, CT 06084



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

***Mastercam***<sup>®</sup>

Mastercam<sup>®</sup> is a registered trademark of CNC Software, Inc. ©1983-2018. All rights reserved.  
All other trademarks are property of their respective owners.