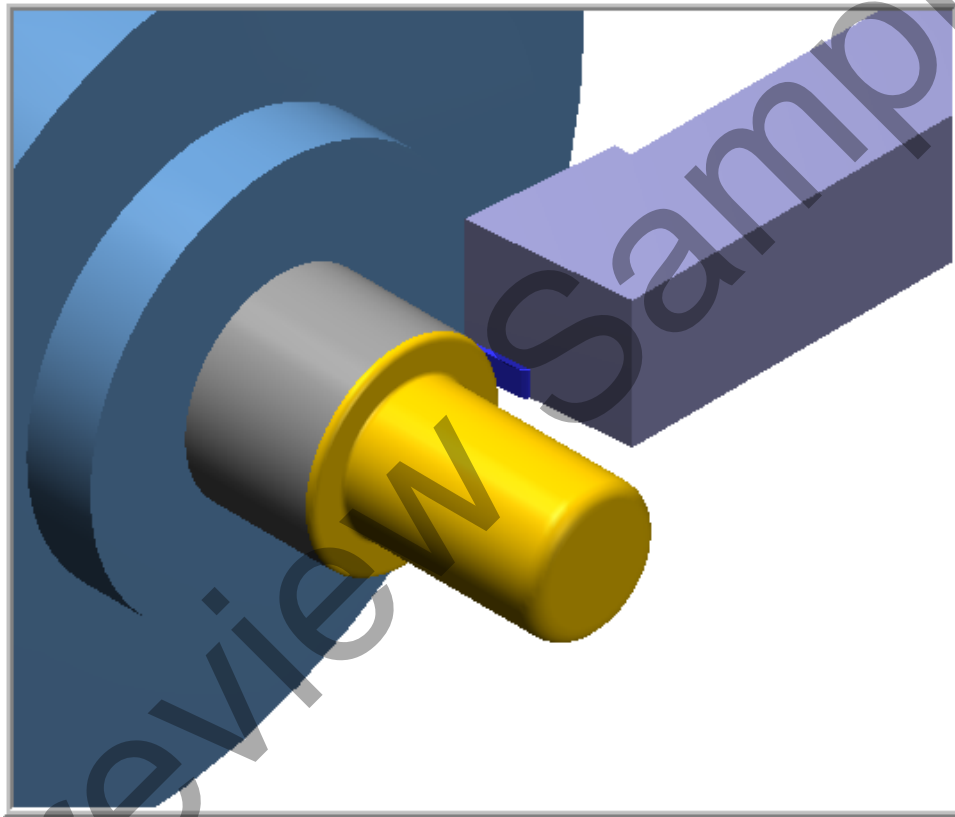


Mastercam X⁵

TRAINING

GUIDE



LATHE-LESSON-1

FACE, ROUGH, FINISH AND CUTOFF

camInstructor

Objectives

You will create the geometry for Lathe-Lesson-1, and then generate a toolpath to machine the part on a CNC lathe. This lesson covers the following topics:

➤ **Create a 2-dimensional drawing by:**

Creating lines.
Creating fillets.

➤ **Establish Stock and Chuck settings:**

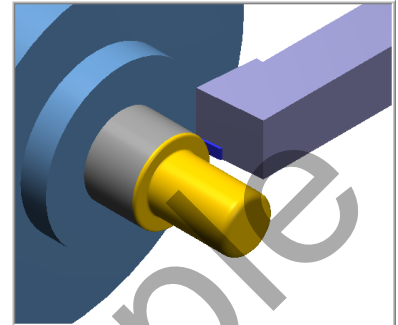
Stock size.
Chuck Configuration.
Material for the part.
Feed calculation.

➤ **Generate a 2-dimensional lathe toolpath consisting of:**

Lathe Face.
Lathe Rough.
Lathe Finish.
Lathe Cutoff.

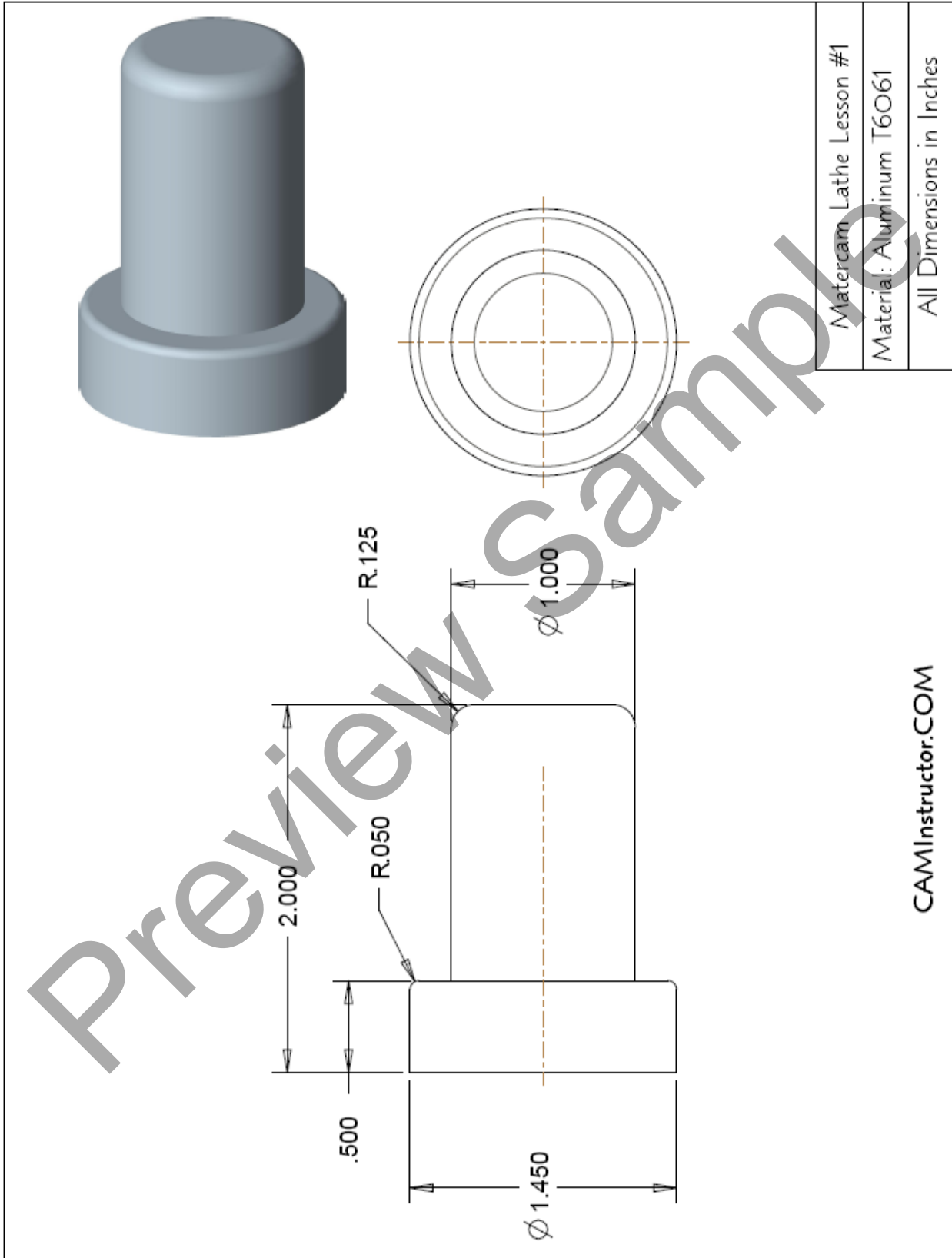
➤ **Inspect the toolpath using Mastercam's Verify and Backplot by:**

Launching the Verify function to machine the part on the screen.
Using Backplot to identify the correctness of the toolpaths.
Generating the NC- code.



Preview Sample

LATHE-LESSON-1 DRAWING





Mastercam Training Guide

TOOL LIST

Two tools will be used to create this part.

- ☞ Tool #1 Face, Rough and Finish the outside diameters
- ☞ Tool #2 Cutoff the part

Tool List of LATHE-LESSON-1.MCX-5						
Proj./Part No.:	0	Date	: 06/16/10			
Drawing No.:	1	Customer	: -			
Prog. No.:	1	Programmer	: 1			
<hr/>						
	Tool type	: T0101: specific tool type - OD ROUGH RIGHT - 80 DEG.				
	Manufact.code	:				
	Holder/Insert	: DCGNR-164D / CNMG-432				
	Setup length	:				
	Spindle RPM	: 200	Feedrate UPR	: 0.01	Corner radius	: 0.0313
	Length offset	: 1	Tool chan. D,Z:	14 , 8		
	Tool type	: T0202: specific tool type - OD CUTOFF RIGHT				
	Manufact.code	:				
	Holder/Insert	: /				
	Setup length	:				
	Spindle RPM	: 200	Feedrate UPR	: 0.01	Corner radius	: 0.016
	Length offset	: 2	Tool chan. D,Z:	14 , 8		

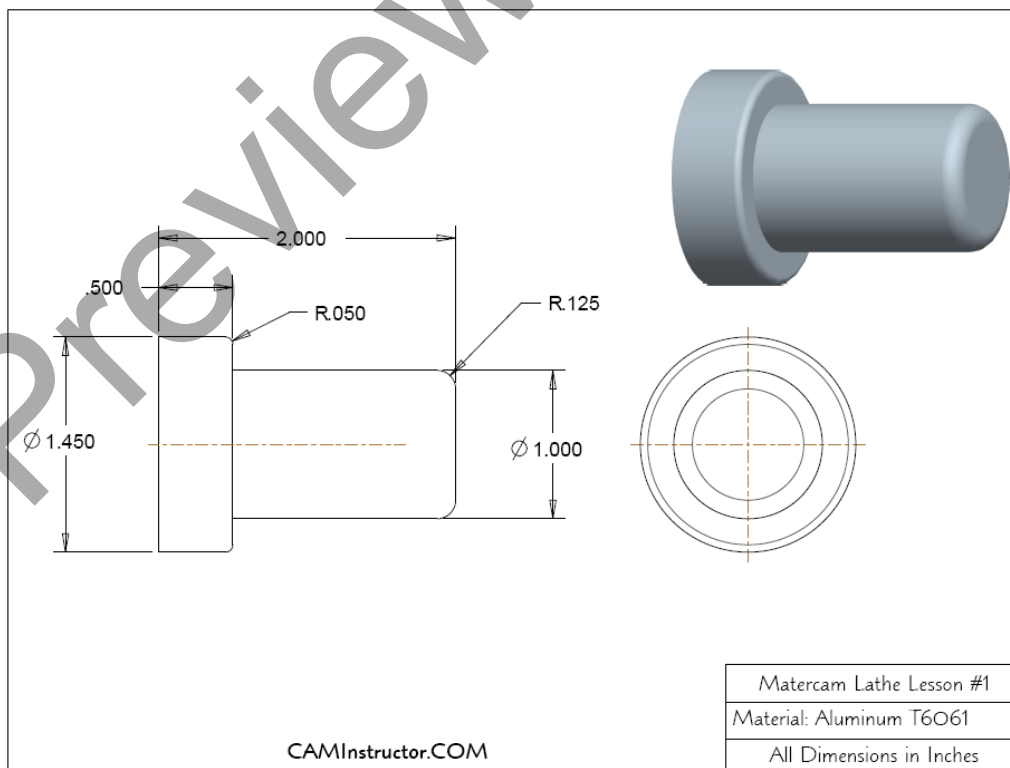
LATHE - LESSON - 1 - THE PROCESS

Geometry Creation

- TASK 1:** Setting the Environment
- TASK 2:** Setting the Construction Planes
- TASK 3:** Create the Geometry
- TASK 4:** Create the Fillets (Radius)
- TASK 5:** Save the Drawing

Toolpath Creation

- TASK 6:** Define the Stock and Chuck Parameters
- TASK 7:** Face the Front of the Part
- TASK 8:** Rough the Outside Diameters
- TASK 9:** Finish the Outside Diameters
- TASK 10:** Cut off the Part
- TASK 11:** Backplot the Toolpath
- TASK 12:** Verify the Toolpath
- TASK 13:** Save the Updated Mastercam File
- TASK 14:** Post and Create the CNC Code File



Geometry Creation

TASK 1: SETTING THE ENVIRONMENT

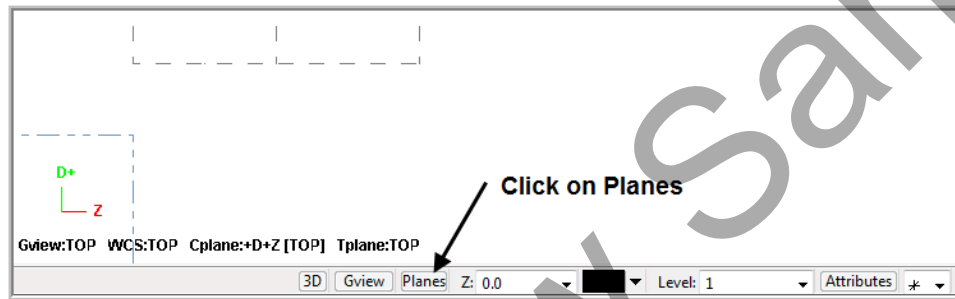
Before starting the geometry creation you should set up the grid and toolbars as outlined in the **Setting the Environment** section at the beginning of this text:

1. Set up the Grid. This will help identify the location of the origin.
2. Customize the toolbars to machine a part on the Lathe.
3. Set the machine type to the Lathe Default.

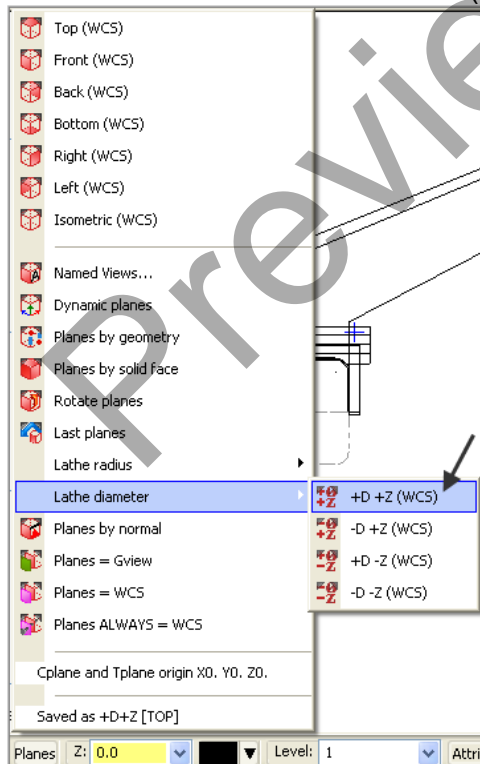
TASK 2: SETTING THE CONSTRUCTION PLANES:

➤ **Set the Construction Plane to Lathe diameter +D +Z (WCS)**

1. Click on Planes at the bottom of the screen as shown below:



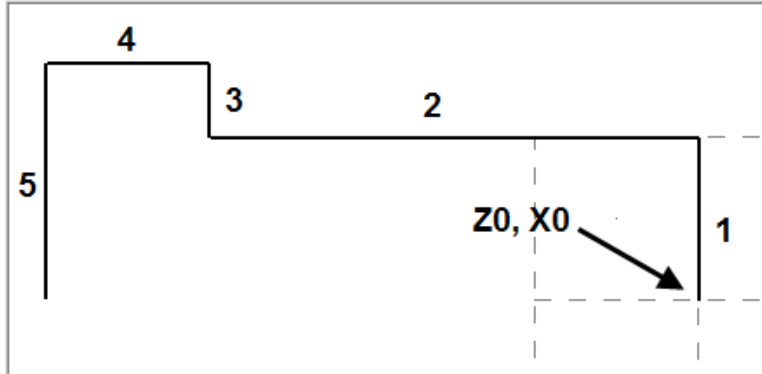
2. Click on Lathe diameter>+D +Z (WCS) as shown below:



TASK 3:

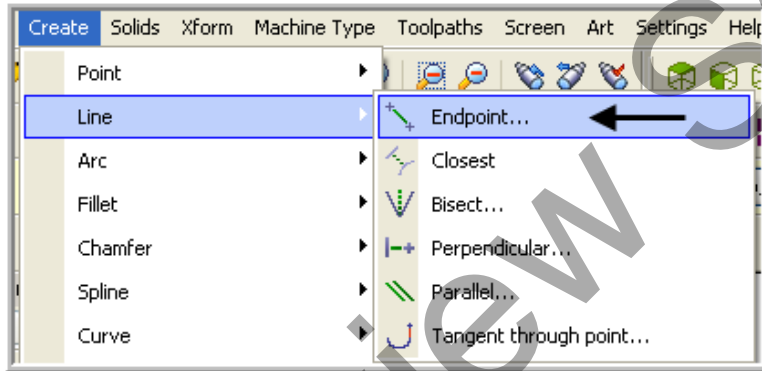
CREATE THE GEOMETRY – THE RIGHT HAND FACE IS AT Z0

- ☞ This task explains how to create the geometry of this part. In this lathe part you only need to create **half of the geometry**, the geometry above the center line.
- ☞ Lines 1 through 5 will be created first and then the fillet and chamfer will be created.

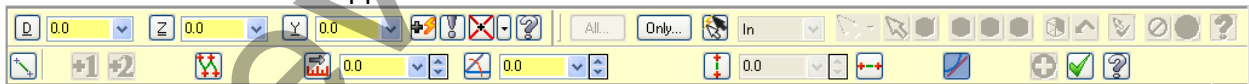



☞ **Create Line #1**

1. Watch the videos **Lesson-1 - Introduction** and **Lesson-1 - Task 1 to 3**.
2. Select from the pull down menu **Create>Line>Endpoint...**

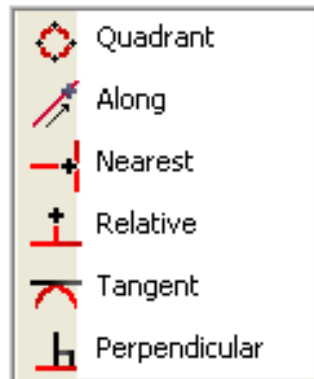
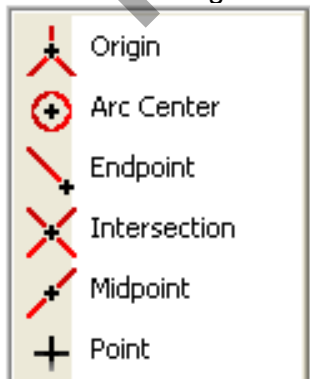


3. The **Line ribbon bar** appears.



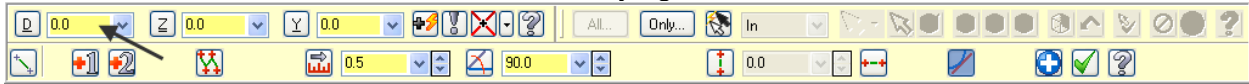
4. Move the cursor over the **center of the grid** and as you get close to the origin a visual cue appears.  With this visual cue highlighted click the **left mouse** button.


☞ The following are Mastercam Visual Cues:

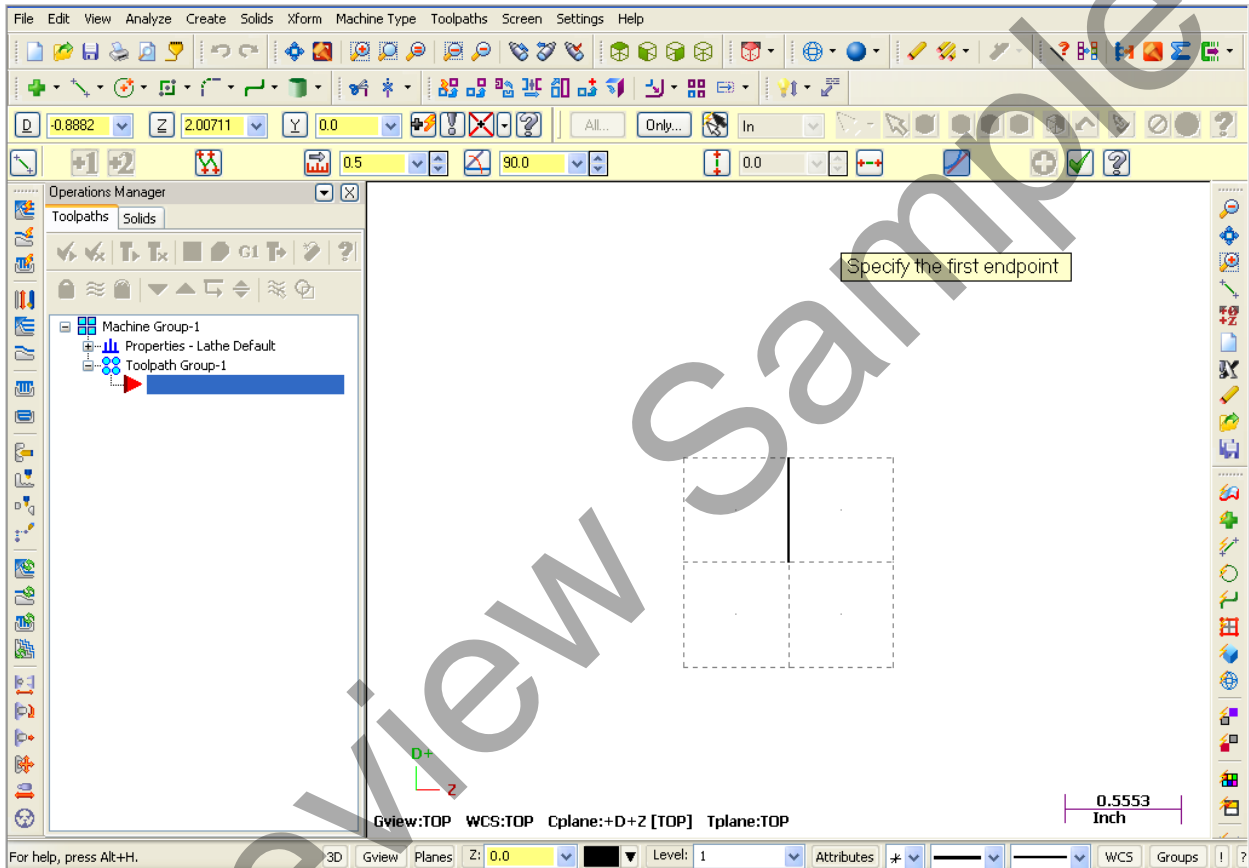




Mastercam Training Guide

5. You are prompted to “Specify the second endpoint”. Click in the **D** value space (Diameter) (as shown by the arrow below) and enter a value of **1.0**. Hit the Enter key and enter a value of **0** for the **Z**, hit the Enter key again and enter a value of **0** for the **Y** enter.




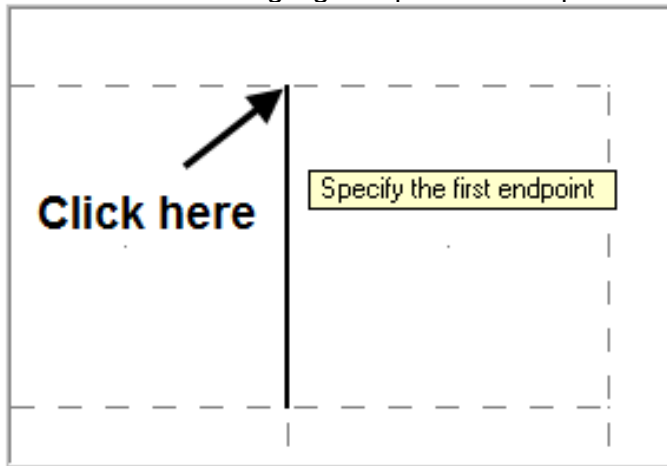
6. Click on Apply .
7. A **vertical line** should be visible as shown below:



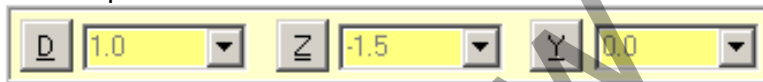
NOTE: If you make a mistake creating lines, click on the **Accept** icon  and click on the **Undo** icon . Then redo the Line.


➤ **Create Line #2.**

8. You are next prompted to **“Specify the first endpoint”**. Click on the **end of the line** that was just created as shown below and as you get close to the end point a visual cue appears.  This is the cue that will allow you to snap to the endpoint of this line. With this visual cue highlighted pick the end point of the line.



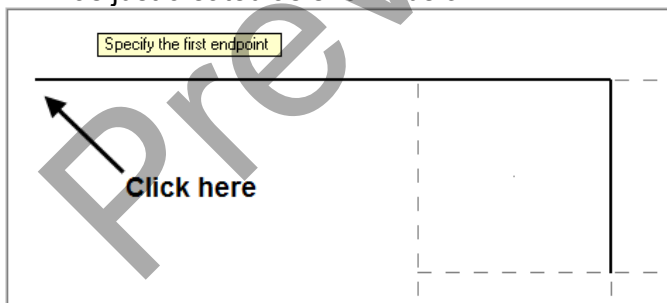
9. You are next prompted to **“Specify the second endpoint”**. On the Line ribbon bar click in the **D** value space and enter **1.0**. Hit the Enter key and enter a value of **-1.5 for the Z**, hit the Enter key again and enter a value of **0 for the Y**. Hit the Enter key once again to complete this line.



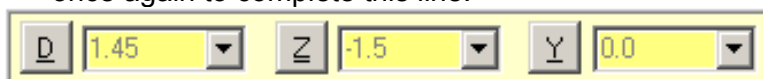
10. Click on Accept .

➤ **Create Line #3**

11. You are next prompted to **“Specify the first endpoint”**. Click on the **end of the line** that was just created as shown below:



12. You are next prompted to **“Specify the second endpoint”**. On the Line ribbon bar click in the **D** value space (Diameter) and enter a value of **1.45**. Hit the Enter key and enter a value of **-1.5 for the Z**, hit the Enter key again and enter a value of **0 for the Y**. Hit the Enter key once again to complete this line.

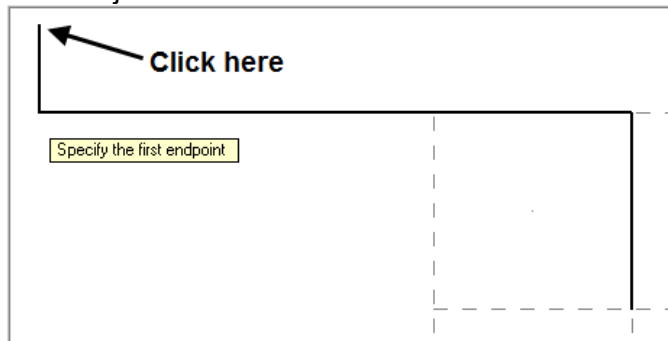


13. Click on Accept .

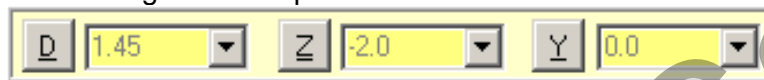
Mastercam Training Guide

➤ Create Line #4

14. You are next prompted to “Specify the first endpoint”. Click on the **end of the line** that was just created as shown below:



15. You are next prompted to “Specify the second endpoint”. On the Line ribbon bar click in the **D** value space (Diameter) and enter a value of **1.45**. Hit the Enter key and enter a value of **-2.0** for the **Z**, hit the Enter key again and enter a value of **0** for the **Y**. Hit the Enter key once again to complete this line.



16. Click on **Accept** .

17. Fit the image to the screen by clicking on the **Fit to Screen** icon as shown below:

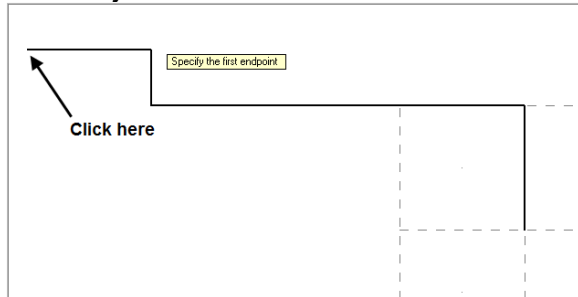


18. Then by clicking on the **Un-Zoom .8** icon as shown below:

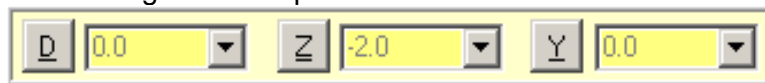


➤ **Create Line #5**


19. You are next prompted to **“Specify the first endpoint”**. Click on the **end of the line** that was just created as shown below:



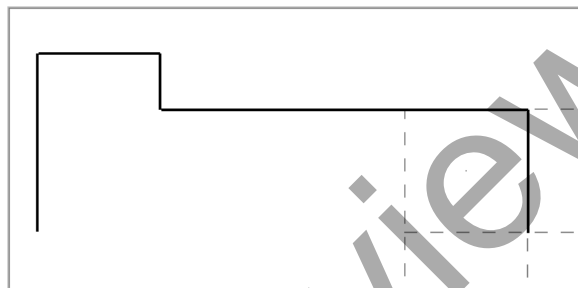
20. You are next prompted to **“Specify the second endpoint”**. On the Line ribbon bar click in the **D** value space (Diameter) and enter a value of **0**. Hit the Enter key and enter a value of **-2.0 for the Z**, hit the Enter key again and enter a value of **0 for the Y**. Hit the Enter key once again to complete this line.



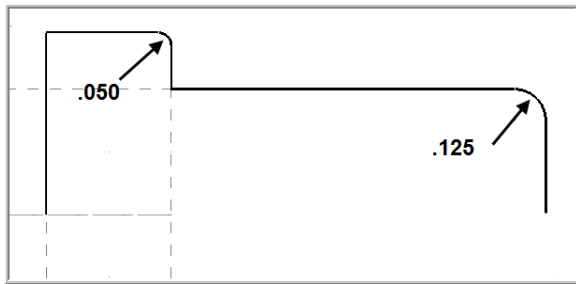
21. Click on the **OK** icon  to complete this feature.

22. Select the **Screen Fit** icon  to fit the part to the screen.

23. Your geometry should look like the figure below:

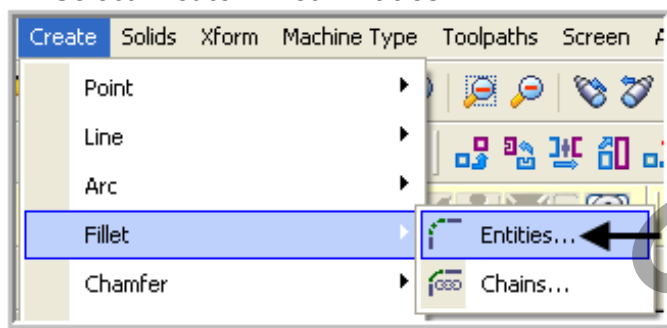


TASK 4: CREATE THE FILLETS



➤ Create the .050 fillet radius.

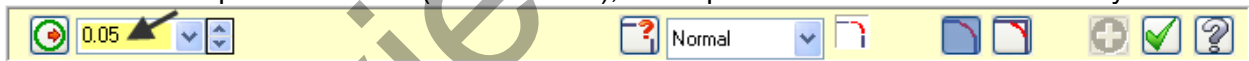
1. Select **Create>Fillet>Entities...**



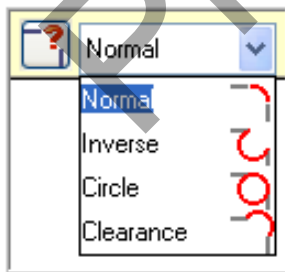
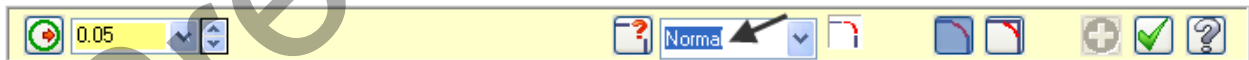
2. The **Fillet Entities** ribbon bar appears and you are prompted to “**Fillet: Select an entity**”.



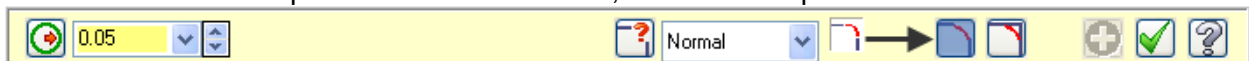
3. Click in the space for radius (circled below), and input **.050** and then hit the tab key.



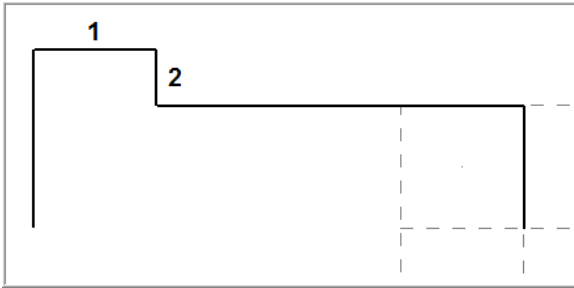
4. You are now transported over to the **Fillet Style field** (circled below). Click on the drop down arrow to review the various fillet radius styles and then ensure **Normal** is selected before moving on.



5. Ensure the **Trim** option for fillet is activated, the icon is depressed as shown below:

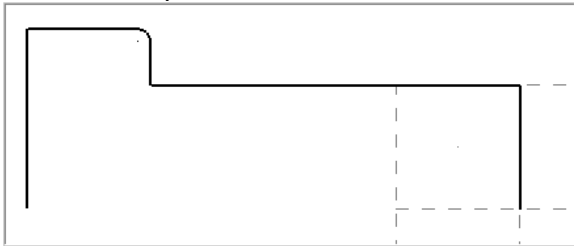


6. Click on **line 1** and then click on **Line 2** as shown below:



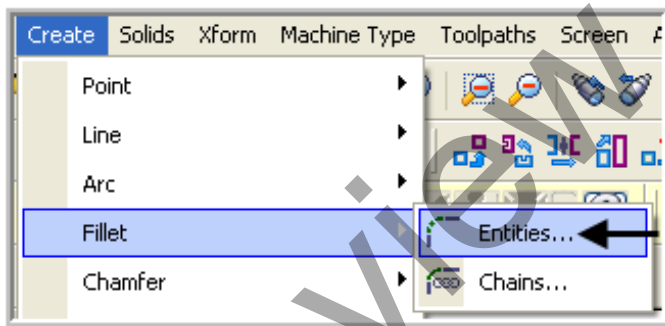
7. Click on the **OK** icon  to complete this feature.

8. The completed fillet is shown below:

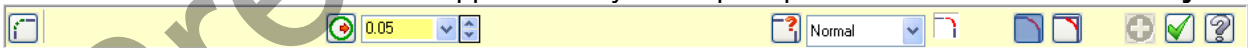


➔ **Create the .125 radius**

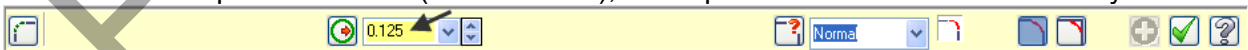
9. Select **Create>Fillet>Entities...**



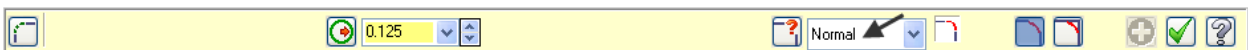
10. The **Fillet Entities** ribbon bar appears and you are prompted to “**Fillet: Select an entity**”.



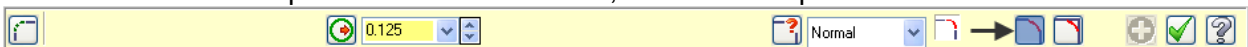
11. Click in the space for radius (circled below), and input **.125** and then hit the tab key.



12. You are now transported over to the **Fillet Style field** (circled below). Ensure **Normal** is selected before moving on.

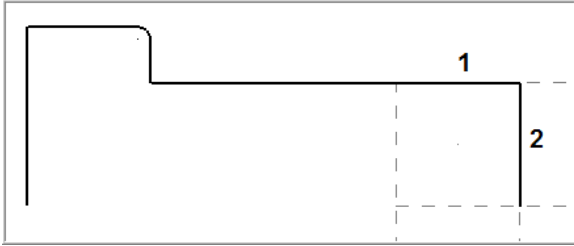


13. Ensure the **Trim** option for fillet is activated, the icon is depressed as shown below:



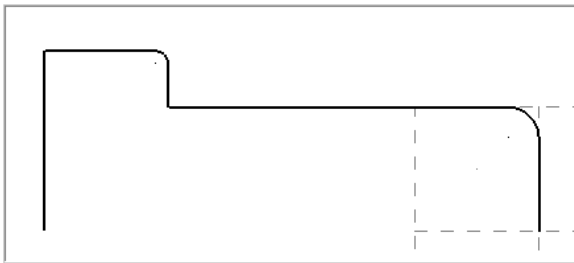
Mastercam Training Guide

14. Click on **Line 1** and then click on **Line 2** as shown below:



15. Click on the **OK** icon to complete this feature.

16. The completed fillet is shown below:



17. This completes the geometry for this part.



TASK 5: SAVE THE DRAWING

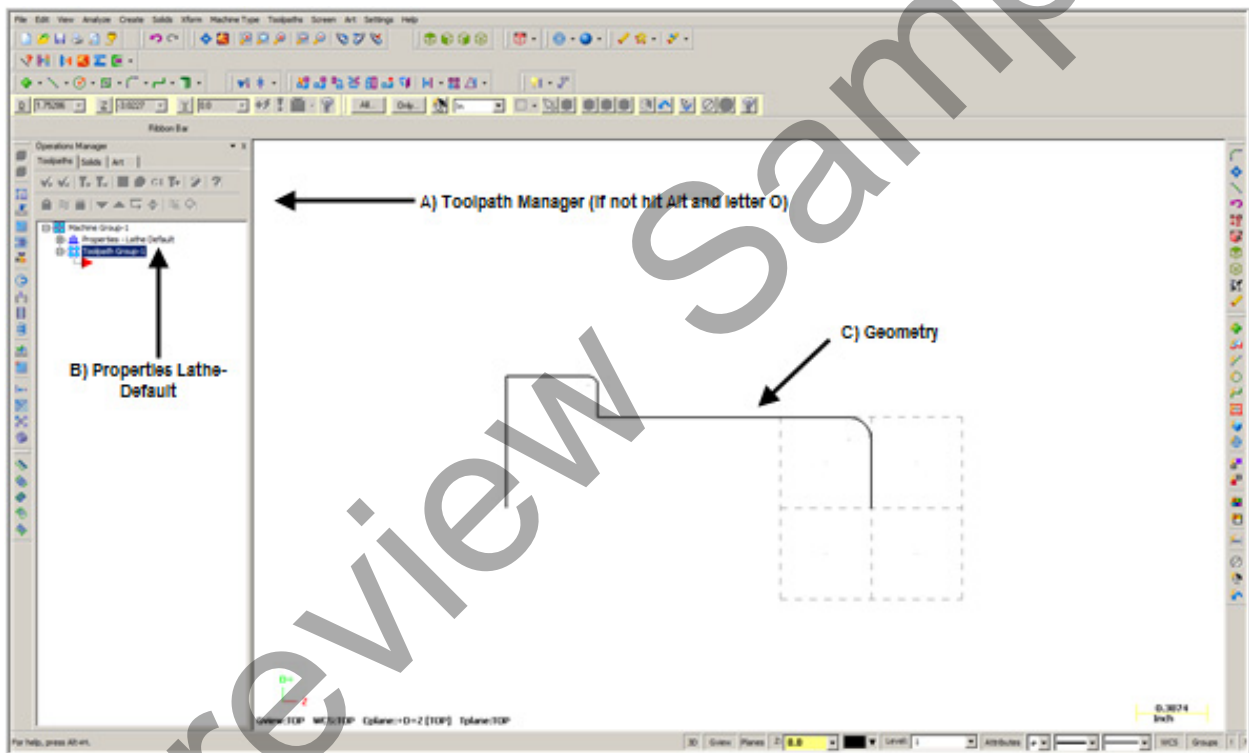
1. Select **File**.
2. Select **Save As...**
3. In the “**File name**” box, type **Lathe-Lesson-1**.
4. Save to an appropriate location.
5. Select the green check mark button to save the file and complete this function.

File name:	LATHE-LESSON-1.MCX-5	▼
Save as type:	Mastercam X5 Files (*.MCX-5)	▼

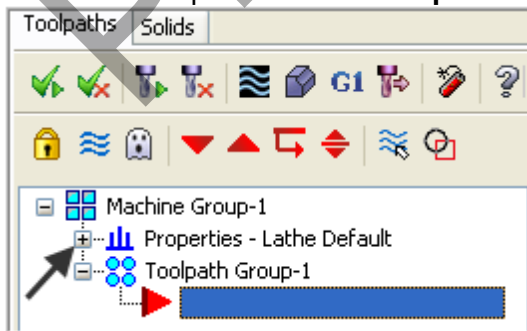
Toolpath Creation

TASK 6: DEFINING THE STOCK AND CHUCK PARAMETERS

1. Select the **screen fit** icon 
2. Select **Un Zoom previous / .5** 
3. Ensure your screen looks like the image below:
 - a. The Toolpath Manager is open, if it is not Select Alt and O on your keyboard to open it.
 - b. The properties icon displays Lathe Default. If it is not turn to the section titled **Setting the Environment** at the beginning of this book.
 - c. The **Lathe - Lesson-1** Geometry is showing.

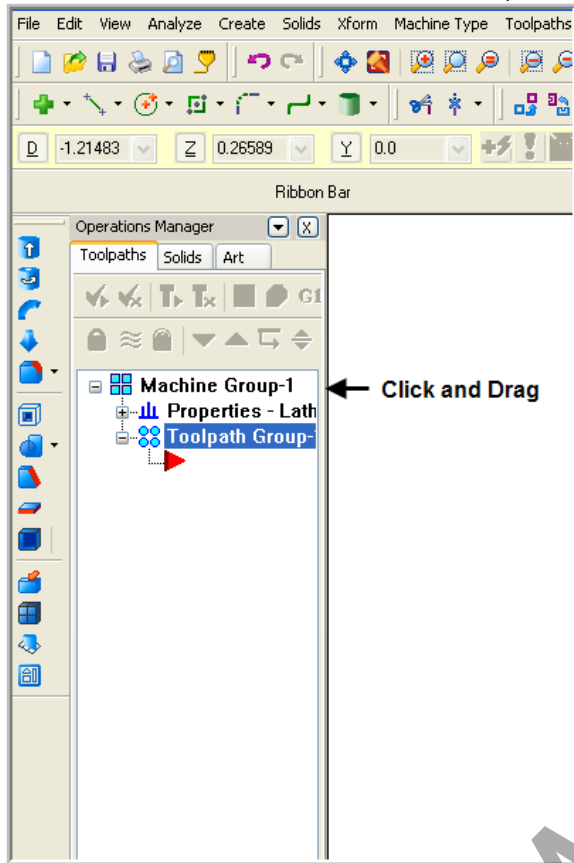


4. Select the plus in front of **Properties** to expand the Machine Group Properties.

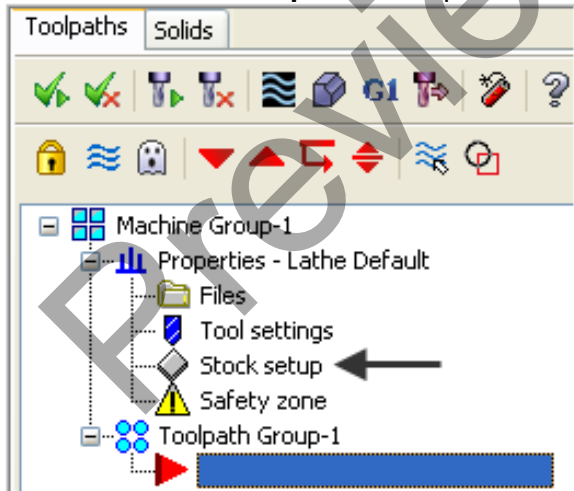


Mastercam Training Guide

5. **This is optional** - To expand the toolpaths manager window, click on the outside of the window with the left mouse button (hold the button down) and drag it to the right.

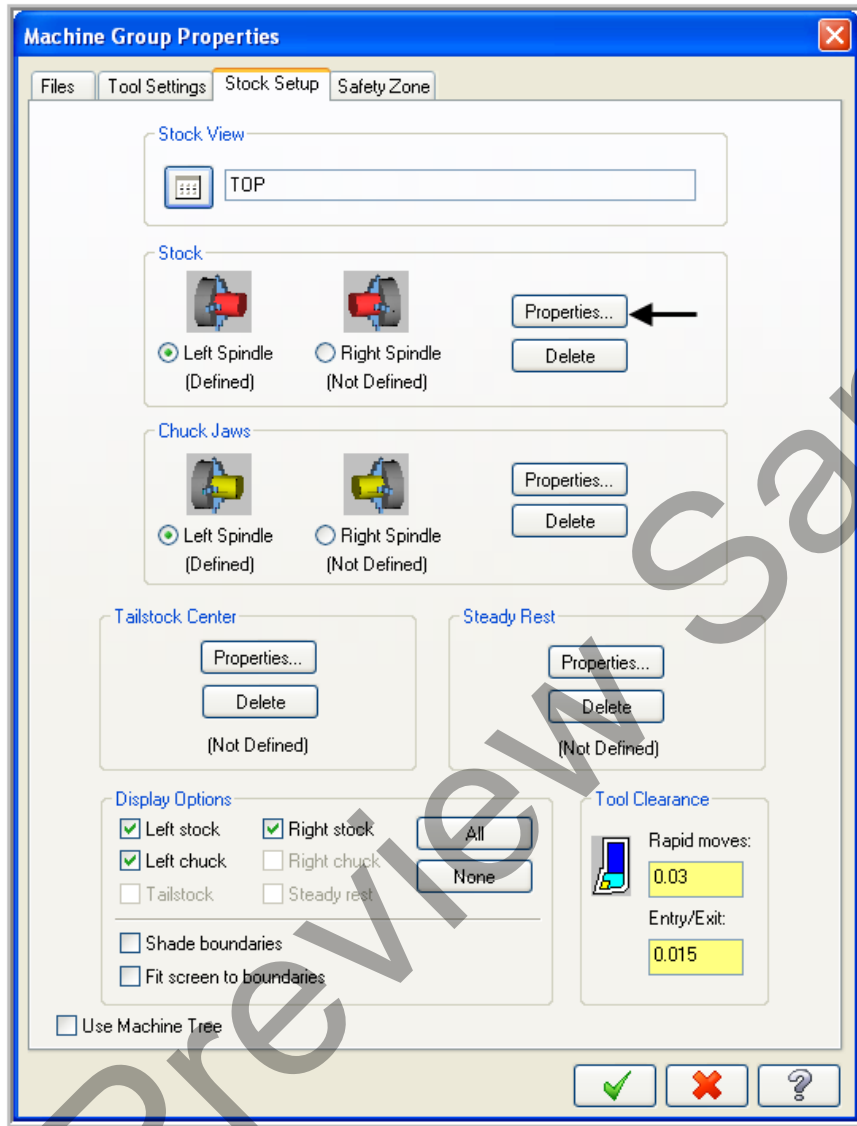


6. Select **Stock setup** in the toolpath manager window.

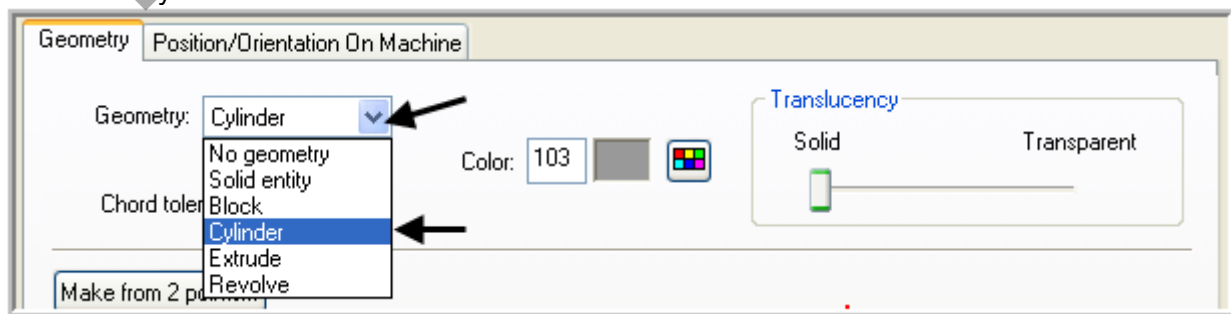


7. Select the **Stock Properties** button in the Stock Setup page as shown in the screenshot below:

➤ Note: To learn how to complete this section of the **Stock Setup** refer to the **Tips and Techniques** section on the Mastercam Training Guide – Lathe CD that accompanies this book.

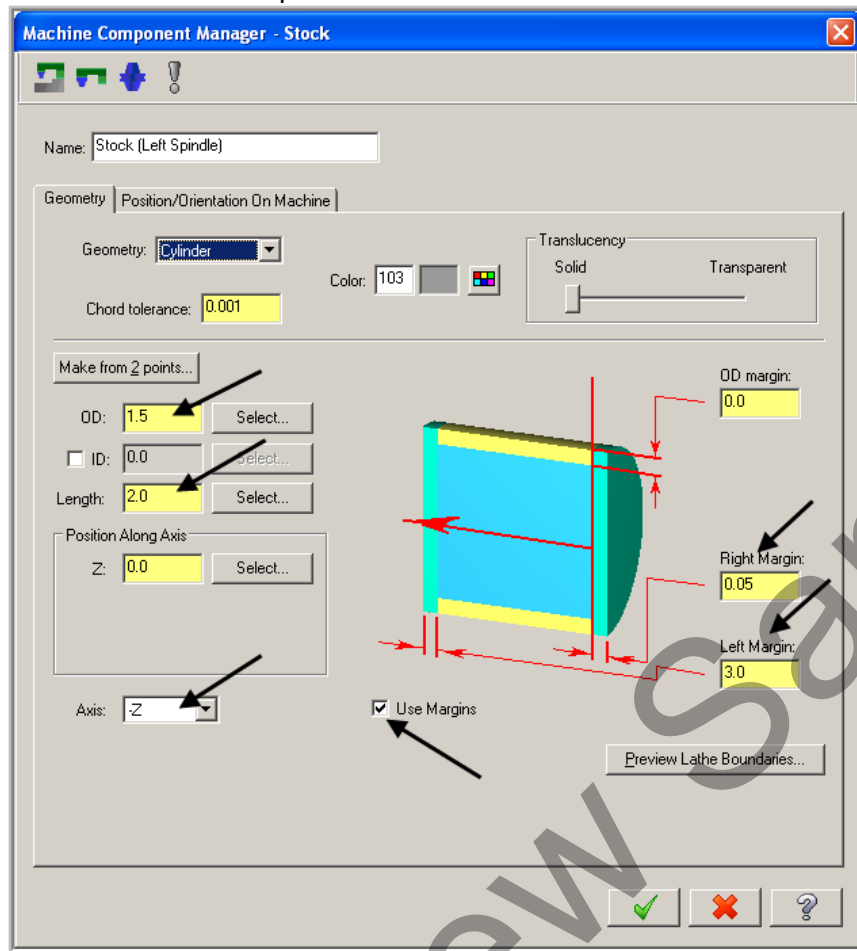


8. In the **Machine Component Manager-Stock** window click on the Geometry button and select Cylinder as shown below:



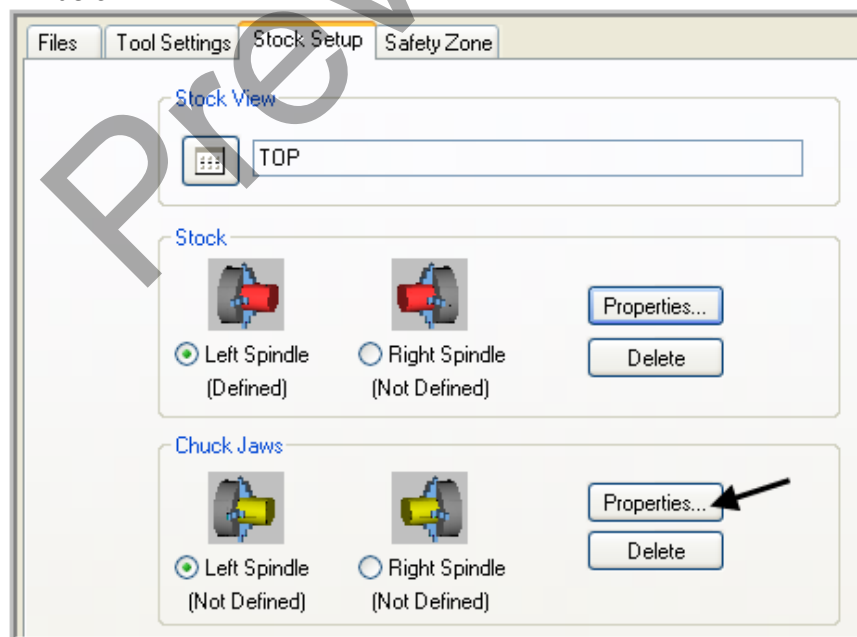
Mastercam Training Guide

9. In the **Stock** setup set the values as shown below:

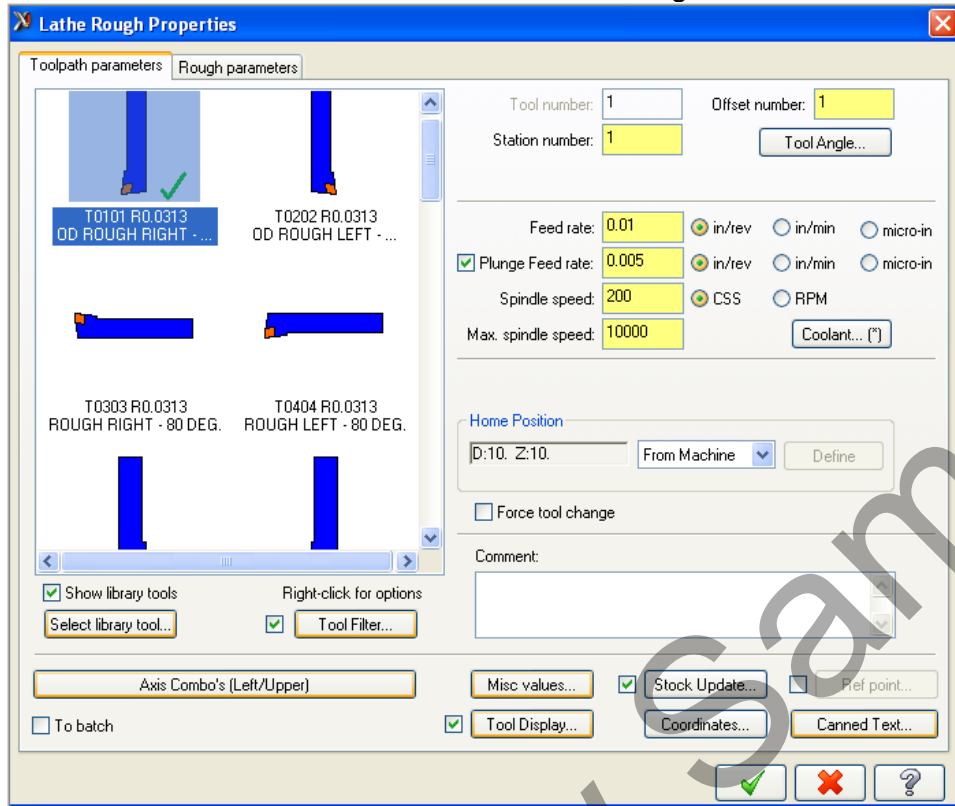


10. Click on the **OK** icon  to complete this feature.

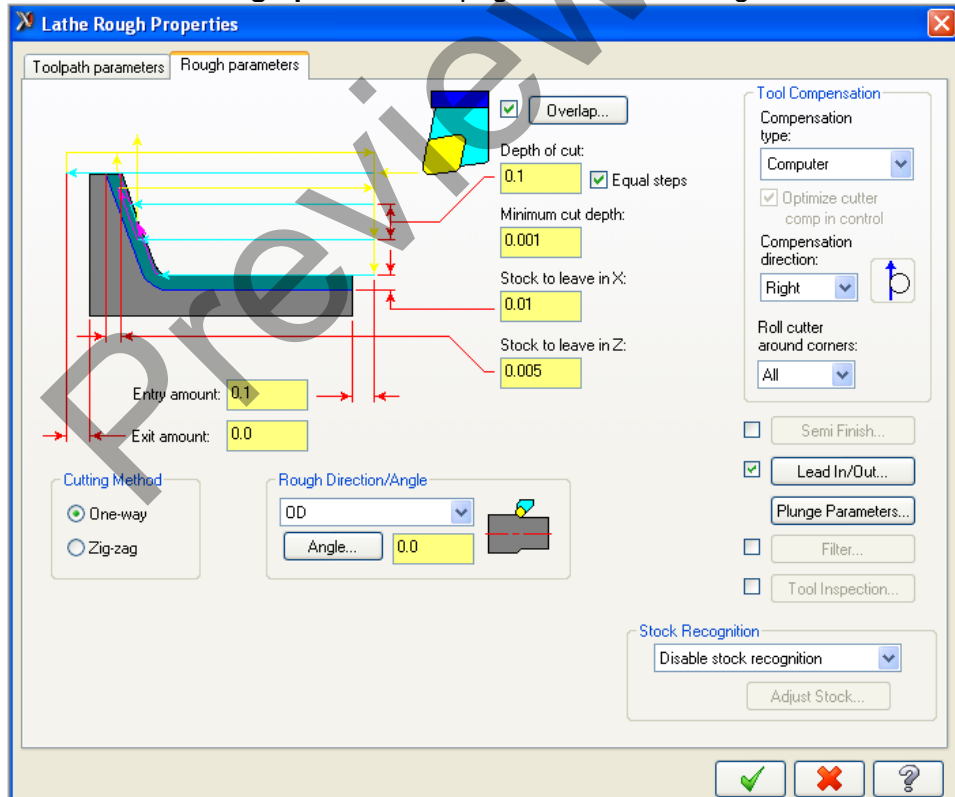
11. Select the **Chuck Properties** button in the Stock Setup page as shown in the screenshot below:



6. In the **Toolpath parameters** page select the same tool used to face the part **Tool #1 OD ROUGH RIGHT** and make make sure the settings are the same as shown below:

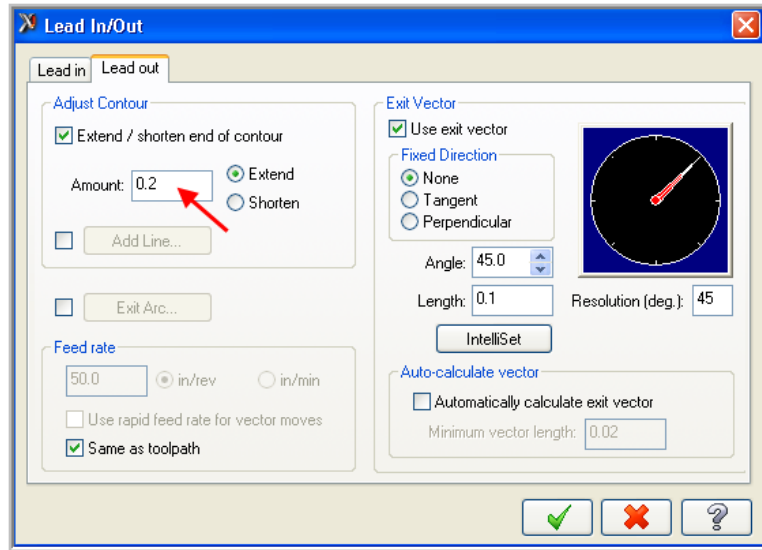


7. Select the **Rough parameters** page and make changes as shown below:



Mastercam Training Guide

8. Select the **Lead In/Out** button select the **Lead out** page and extend the contour by .2 as shown below:

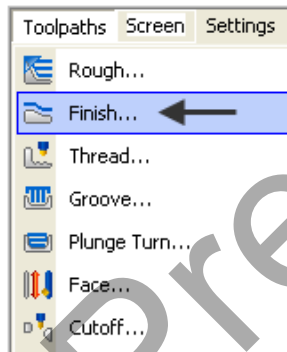


9. Select the **OK** button to exit this function.
10. Select the **OK** button to exit Rough Parameters.

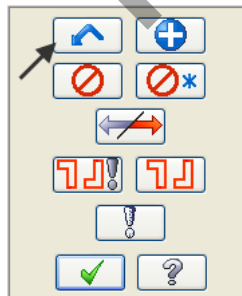
TASK 9: FINISH THE OUTSIDE DIAMETERS

- In this task you will finish the outside diameters in one cut using **Tool #1 an OD Rough-Right – 80 deg.**

1. From the menu bar select **Toolpaths>Finish...**

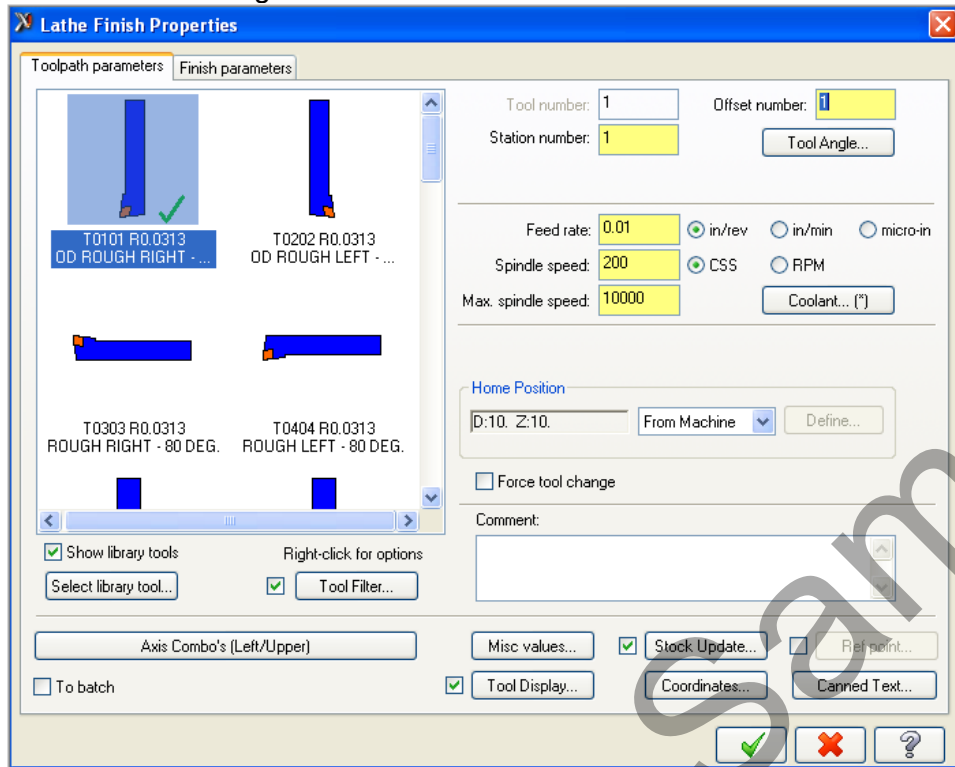


2. Select **Last** in the Chaining dialog box.

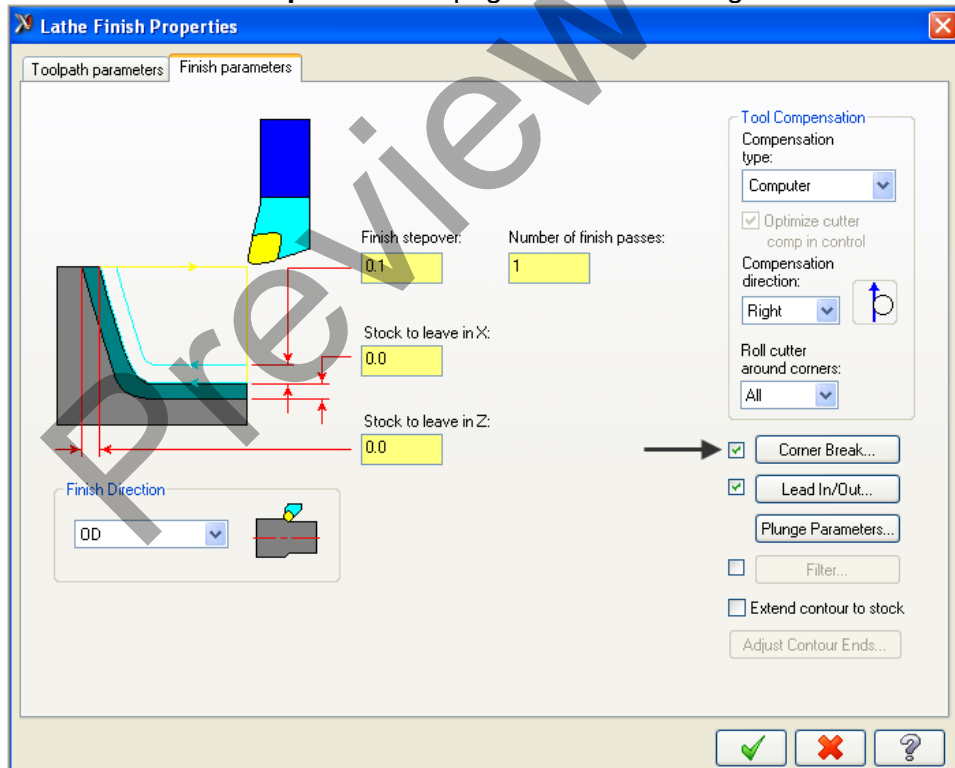


3. Select the **OK** button to complete the selection.

- Select the same tool used to rough the part; **Tool #1 OD Rough Right** tool from the tool list and make changes as shown below:



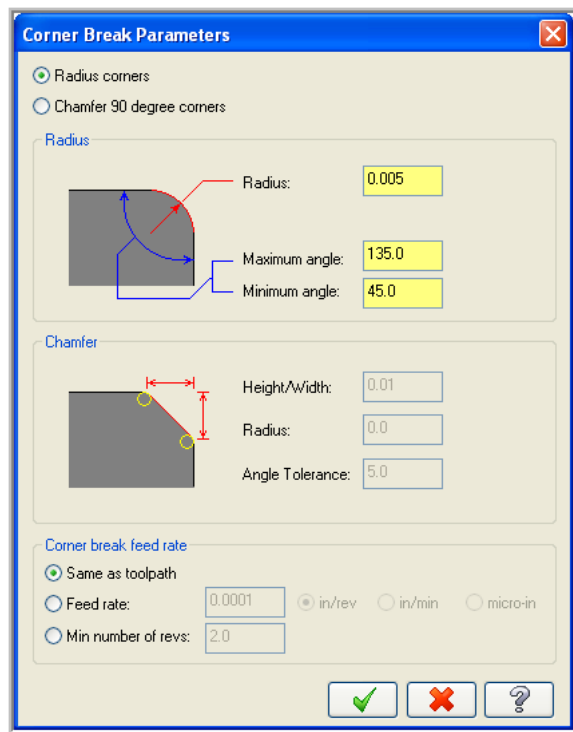
- Select the **Finish parameters** page and make changes as shown below:



Corner Break: Select the check box to automatically create radii or chamfers on all outer corners of the toolpath. Click the button to edit the corner break settings.


Mastercam Training Guide

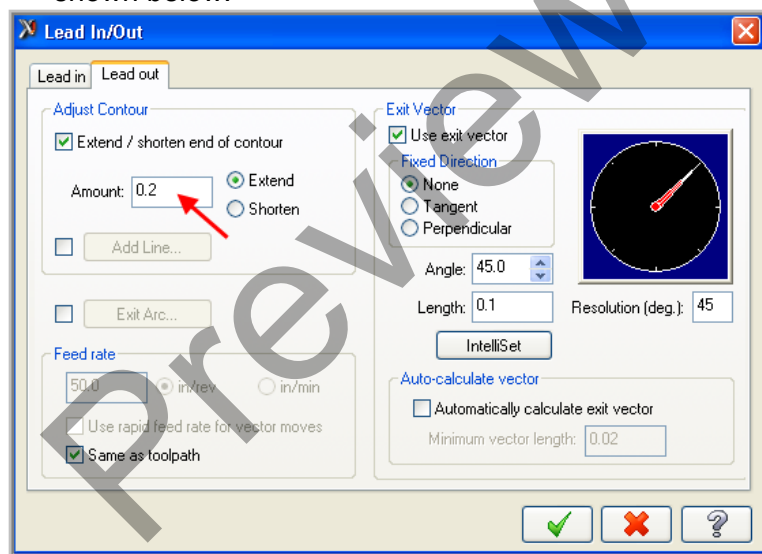
6. Select the **Corner Break** button and make changes as shown below:



Corner Break

Use this dialog box to automatically create radii or chamfers on all outer corners of lathe finish toolpaths. You can also set the feed rate when the tool creates the radii or chamfers.

7. Select the OK button  to complete this feature.
8. Select the **Lead In/Out** button select the **Lead out** page and extend the contour by .2 as shown below:

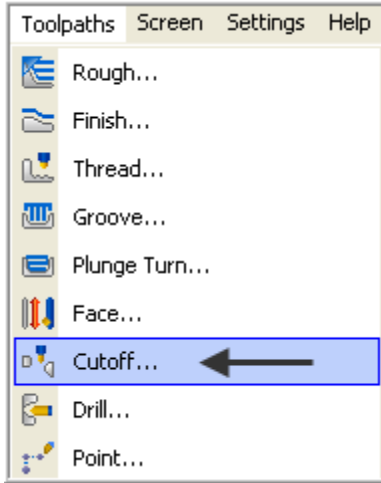


9. Select the OK button  to exit this function.
10. Select the OK button  to exit **Finish** parameters.

**TASK 10:
CUTOFF THE PART**

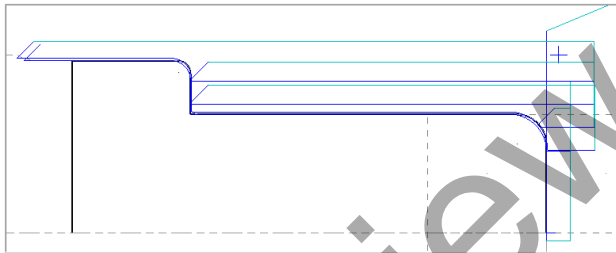
☞ In this task you will cutoff the part using a .125 wide cutoff tool.

1. From the menu bar select **Toolpaths>Cutoff...**

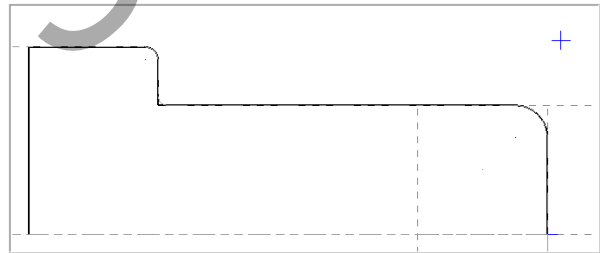


2. Select the Alt key and the T key on the keyboard to hide the toolpath lines.

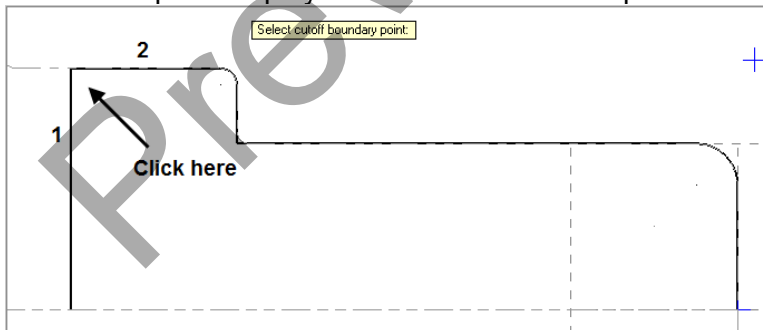
Toolpath Lines visible:



Press Alt T to hide toolpath Lines:

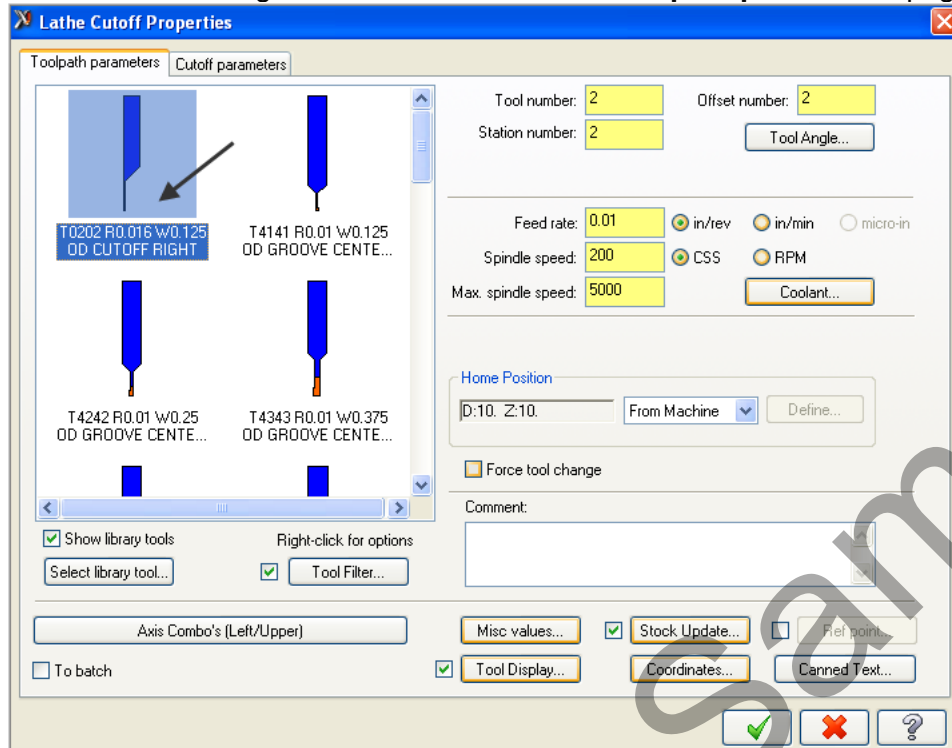


3. Move the cursor over the corner (where **Line 1** and **Line 2** meet) until the visual cue for End point displays and then click on this point as shown below:

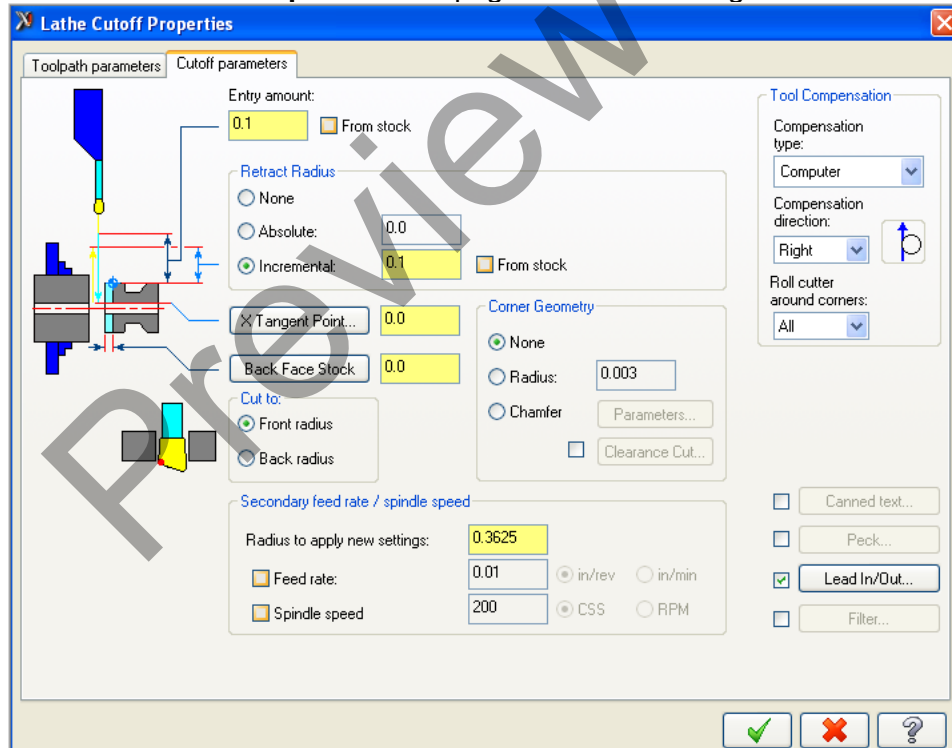


Mastercam Training Guide

4. Scroll down if required in the tool window and select the **OD Cutoff Right Width .125** tool and make changes as shown below in the **Toolpath parameters** page:






5. Select the **Cutoff parameters** page and make changes as shown below:



6. Select the **OK** button  to exit **Cutoff parameters**.

**TASK 11:
BACKPLOT THE TOOLPATH**

- ☞ In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.
- ☞ Backplot will enable us to review the cutting motions and identify any problem areas when cutting the part.

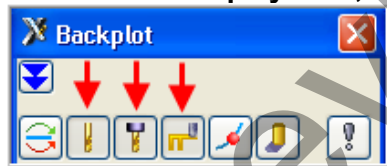
1. Select the **Fit** icon to fit the part to the screen .
2. Then select the **Un-Zoom Previous / .5** icon.  This function reduces the size of the displayed geometry to 50% of its current size.
3. To pick all the operations to backplot pick the **Select All** icon  circled below:



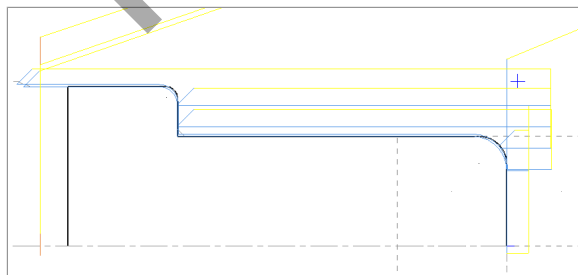
4. The next step is to select the **Backplot selected operations** icon shown below:




5. Before you **Backplot** the toolpath ensure the buttons shown below are activated. These buttons act like a toggle switch, when pressed in the function is turned on.
Turn on: Display Tool, Display holder, Display rapid moves as shown below:



6. Click on the **Play** button, (Note: if you need to slow down the speed move the speed toggle as shown below):



7. After reviewing the backplot of the toolpaths select the **OK** button  to exit Backplot.

Mastercam Training Guide

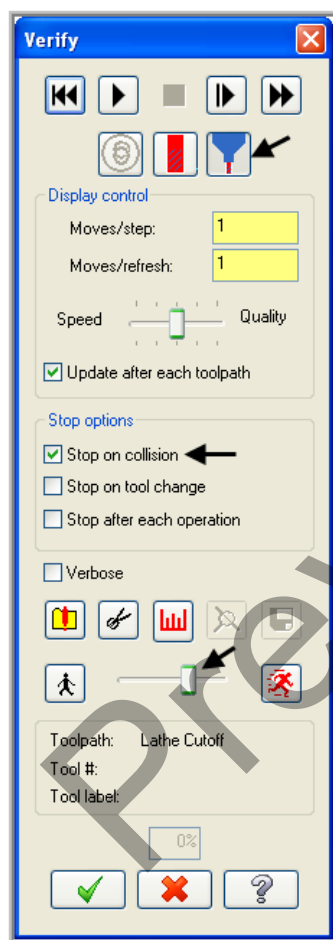
TASK 12: VERIFY THE TOOLPATH

☞ Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.

1. In the Toolpath Manager pick all the operations to backplot by picking the **Select All** icon.



2. Select the **Verify selected operations** button circled below:



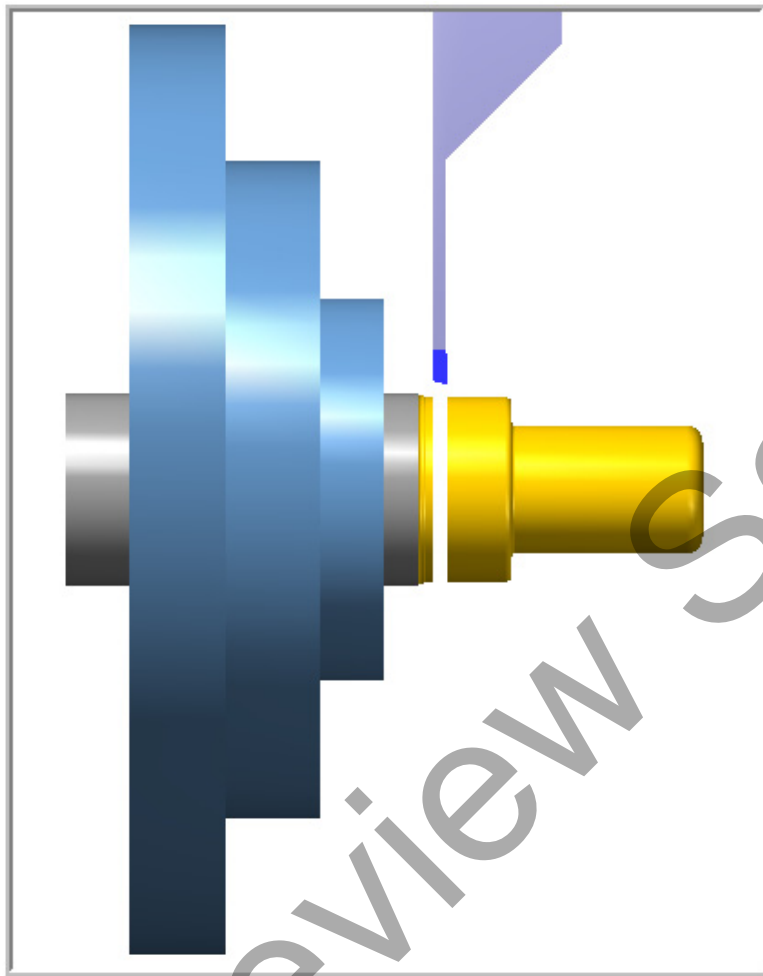
3. Adjust the **Verify speed** to midway along the speed control bar.



4. Select the **Play** button to verify the three toolpaths.



5. The verified toolpaths should appear as in the picture below:




6. Select the **OK** button to exit **Verify**.

**TASK 13:
SAVE THE UPDATED MASTERCAM FILE**

1. Select the **save** icon from the toolbar

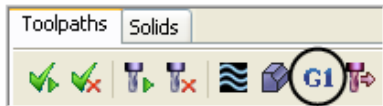


TASK 14: POST AND CREATE THE CNC CODE FILE

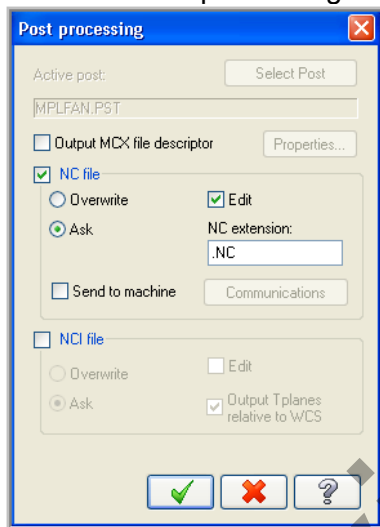
1. Ensure all the operations are selected by picking the **Select All** icon  from the **Toolpath manager**.



2. Select the **Post selected operations** button from the **Toolpath manager**.




3. In the Post processing window, make the necessary changes as shown below:

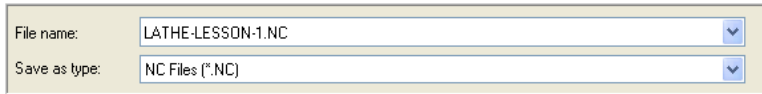


About Post Processing

NC file: Select this option to save the NC file. The file name and extension are stored in the machine group properties for the selected operation. If you are posting operations from different machine groups or Mastercam files, or batch processing, Mastercam will create several files according to the settings for each machine group.

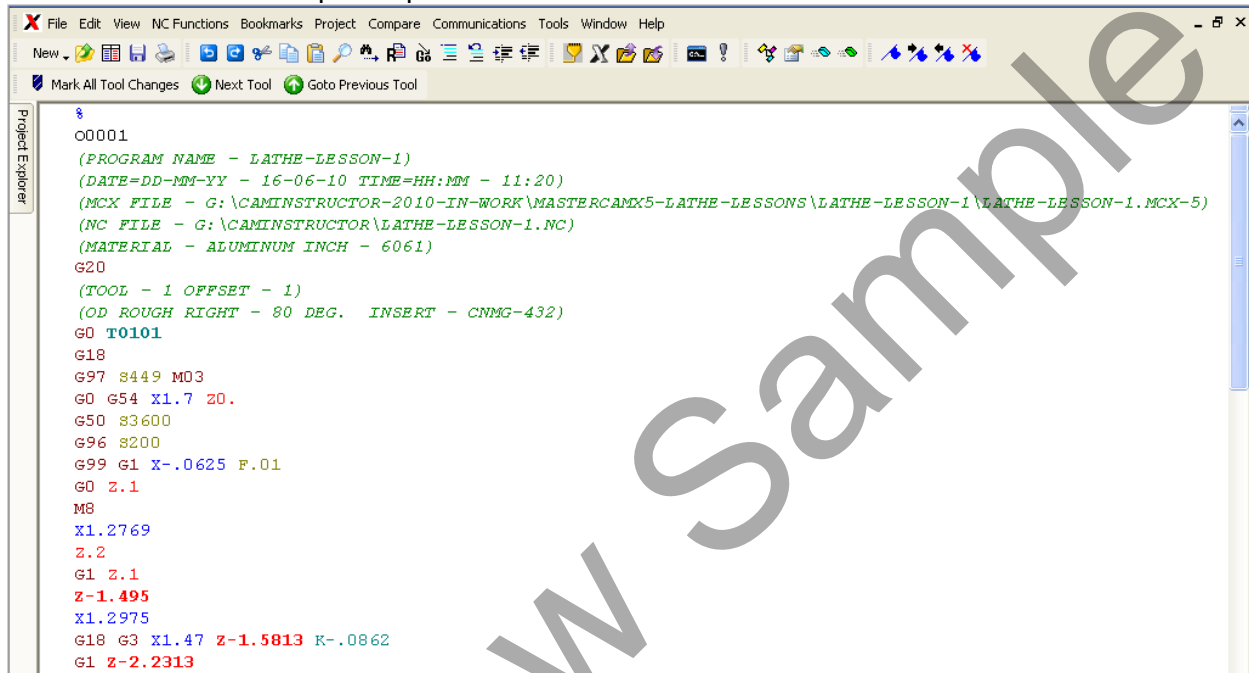
Edit: When checked, automatically launches the default text editor with the file displayed so that you can review or modify it.

4. Select the **OK** button  to continue.
5. Enter the same name as your Mastercam part file name in the **NC File name** field **LATHE-LESSON-1.**



File name: LATHE-LESSON-1.NC
Save as type: NC Files (*.NC)

6. Select the **Save** button.
7. The CNC code file opens up in the default editor.



```
O0001
(PROGRAM NAME - LATHE-LESSON-1)
( DATE=DD-MM-YY - 16-06-10 TIME=HH:MM - 11:20)
(MCX FILE - G:\CAMINSTRUCTOR-2010-IN-WORK\MASTERCAMX5-LATHE-LESSONS\LATHE-LESSON-1\LATHE-LESSON-1.MCX-5)
(NC FILE - G:\CAMINSTRUCTOR\LATHE-LESSON-1.NC)
(MATERIAL - ALUMINUM INCH - 6061)
G20
(TOOL - 1 OFFSET - 1)
(OD ROUGH RIGHT - 80 DEG. INSERT - CNMG-432)
G0 T0101
G18
G97 S449 M03
G0 G54 X1.7 Z0.
G50 S3600
G96 S200
G99 G1 X-.0625 F.01
G0 Z.1
M8
X1.2769
Z.2
G1 Z.1
Z-1.495
X1.2975
G18 G3 X1.47 Z-1.5813 K-.0862
G1 Z-2.2313
```

8. Select the  in the top right corner to exit the CNC editor.
9. This completes LATHE-LESSON-1.

LATHE-LESSON-1 EXERCISE

The image displays a 3D model of a lathe exercise part on the left. To its right is a technical drawing consisting of a top view and a side view. The top view shows a circular part with concentric circles representing different diameters. The side view shows a cylindrical part with a diameter of $\varnothing 1.375$ and a length of 1.625 . The part features a chamfered end with a radius of $R.125$ and a diameter of $\varnothing .625$. A second chamfered end with a radius of $R.125$ and a diameter of $\varnothing .875$ is located at the other end of the part. The distance between the centers of these two chamfers is $.375$. The distance from the center of the larger chamfer to the end of the part is $.500$. A watermark 'Preview Sample' is overlaid diagonally across the drawing.

Mastercam Lathe Exercise #1
Material: Aluminum T6O61
All Dimensions in Inches

CAMInstructor.COM