Lathe Custom Tools

August 2015

Mastercam X9

Mastercam® X9 Lathe Custom Tools

Date: August 2015
Copyright © 2015 CNC Software, Inc.— All rights reserved.
Software: Mastercam X9

TERMS OF USE

Use of this document is subject to the Mastercam End User License Agreement. A copy of the Mastercam End User License Agreement is included with the Mastercam product package of which this document is part. The Mastercam End User License Agreement can also be found at:
http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx

Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this 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.

Contents

Introduction 1

- Tutorial Goals 1

General Tutorial Requirements 1

1. Creating a Vertically Oriented Tool 3

- Drawing the Insert 3

Drawing the Insert's Geometry 3

Completing the Insert's Tip 6

Drawing the Holder 10

- Setting the Tool Parameters 12

Setting the Orientation and Position 13

Setting Tool Type and Compensation 16

- Setting Up the Tool 16

Setting Up the Tool 17

Testing the Tool in a Toolpath 18

2. Graphical Orientation and Tool Settings 21

Using Tool Orientation 21

Tool Position in the Graphics Window 25

Restore the Tool to its Original Settings 28

3. Creating a Horizontally Oriented Tool 31

- Repositioning the Tool 31

Rotating the Tool 31

- Setting the Tool Parameters 34

Setting the Orientation and Position 34

Setting Tool Type and Compensation 37

▶ Setting Up the Tool 37
Setting Up the Tool 37
Testing the Tool in a Toolpath 39

Introduction

In this tutorial, you learn to create lathe custom tools, as well as explore the parameters that you must understand to create valid tools. Specifically, you learn about the following topics.

Tutorial Goals

- Drawing a lathe custom tool
- Configuring a lathe custom tool
- Using a lathe custom tool

IMPORTANT: 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 the exercise results.

General Tutorial Requirements

All Mastercam 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 (HLE). The Demo/HLE file format (EMCX-9) is different from Mastercam (MCX-9), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam workstation, either with the tutorial or in any location that you prefer.
- 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.

- All Mastercam tutorials require you to configure Mastercam to work in a default metric or English configuration. The tutorial provides instructions for loading the appropriate configuration file.

LESSON 1
Creating a Vertically Oriented Tool

In this lesson, you create a vertically oriented tool that will be used in the machine's upper turret to cut material on the left spindle. To do this, you must complete the following procedures:

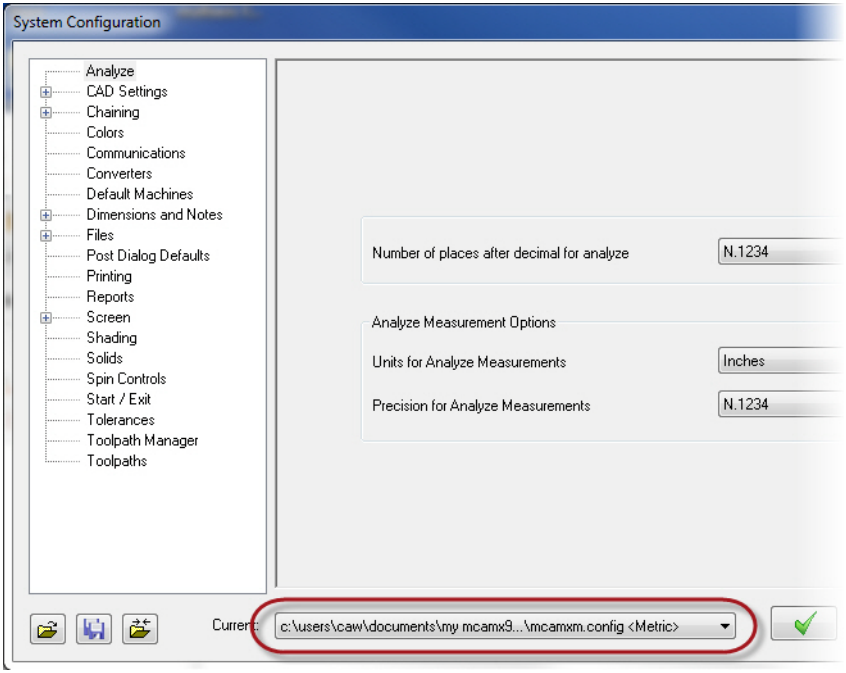
- Draw the insert.
- Draw the holder.
- Set the tool's properties.
- Test the tool with a toolpath.

Drawing the Insert

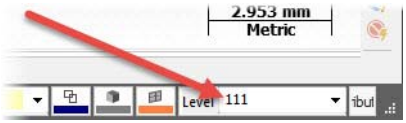
The first step in creating a custom tool is drawing its tool profile geometry. This geometry comprises two parts: the insert and the holder. In this exercise, you draw the tool insert, shown below.

▶ Drawing the Insert's Geometry

1. Start Mastercam.
2. If necessary, set Mastercam's configuration to **Metric**.



3 In Mastercam's Status Bar, change the level number to **111**.

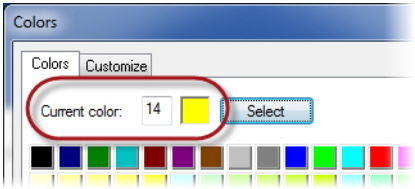


Because tool geometry must be on its own level, with no other geometry, you should pick a level number that you are unlikely to use in your part drawings.

4 In the Status Bar, click **Wireframe color**.

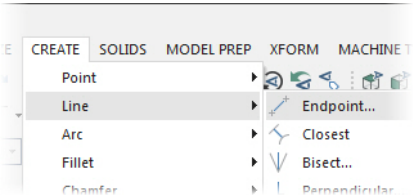
The **Colors** dialog box opens.

5 Change the **Current color** to **14**, and click **OK**.



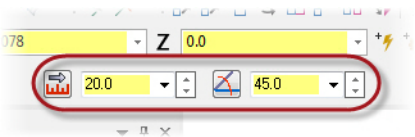
You must use different colors to draw the tool's insert and holder. The recommended color for insert up is 14. For insert down, the recommended color is 138. For the holder, use color 116.

6 From Mastercam's menu, choose **Create, Line, Endpoint**.



7 Click in the graphics window, and draw a short line anywhere.

8 In the ribbon bar, set **Length** to **20**, **Angle** to **45**, and press [**Enter**].



9 Draw another line from the first line's upper endpoint.

10 Set the line's length to **20**, angle to **90**, and press [**Enter**].

11 Draw a line from the previous line's endpoint, giving it a length of **35** and an angle of **0**.

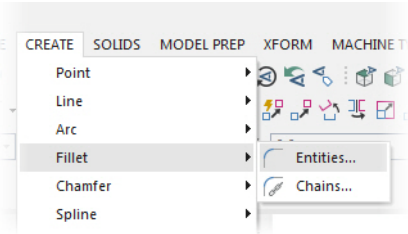
12 Draw a line from the previous line's endpoint, giving it a length of **25** and an angle of **270**.

13 Draw a final line that closes the insert's shape, and click **OK** to exit the line function.

Completing the Insert's Tip

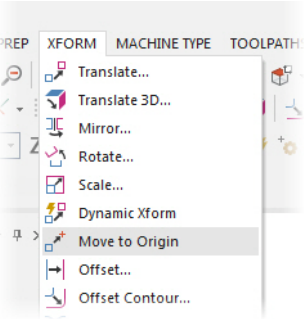
1 Zoom in on the insert's tip.

2 In Mastercam's menu, choose **Create, Fillet, Entities**.



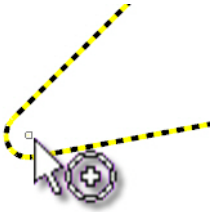
3 Select the tip's two lines, set **Radius** to **0.8**, and click **OK**.

4 Select the entire insert, and choose **Xform, Move to Origin** from Mastercam's menu.



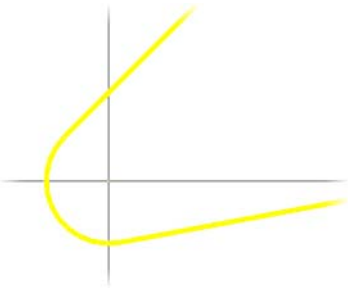
5 Select the fillet's centerpoint.

The insert moves to the origin, with the tip arc's centerpoint centered at 0,0.

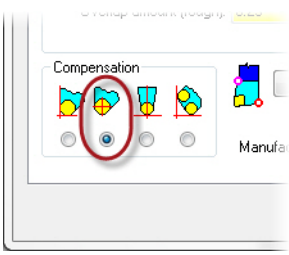


6 Select **Fit**, select **Clear Colors**, and then press [**F9**] to turn on the axes in the graphics window.

7 Zoom in on the origin, and notice how the insert's tip is now positioned.

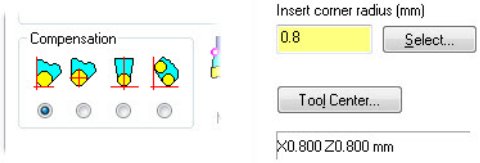


The insert's position in the graphics window determines how you define the tool's parameters in Mastercam. For example, if you were to leave the insert's position as it is in Step 7, when you set the tool's parameters, you would choose the compensation point shown to the right.



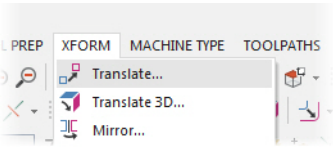
Also, the tool's center would be at 0,0.

The more common position for a vertically oriented tool is with the insert's driving point fully in the upper-right quadrant of the graphics window. In this case, you would set the compensation point and tool center as shown below.



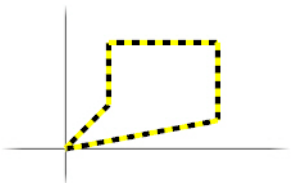
In the remaining steps of this exercise, you move the insert into the graphics window's upper-right quadrant.

8 Choose **Xform, Translate**.



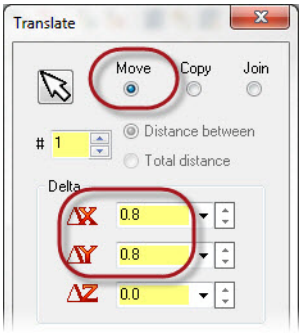
9 Select the entire insert, and press [**Enter**].

The **Translate** dialog box opens.

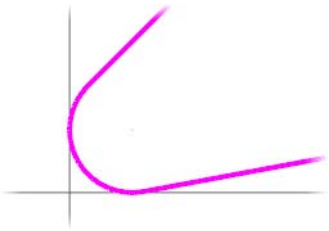


10 Select **Move**, set Delta **X** and **Y** to **0.8**, and click **OK**.

The geometry moves to the selected position.

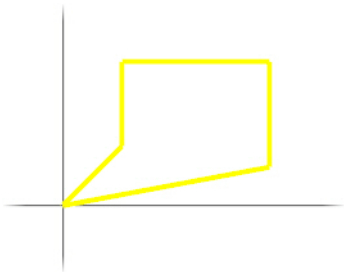


11 Zoom in on the origin to see that the insert's tip is now oriented correctly.



12 Click **Clear Colors** to restore the insert's color to yellow, and then click **Fit**.

Your completed lathe tool insert should look like the picture to the right.

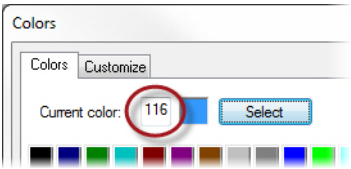


13 Save the file as `CustomTool01_`*xxx*, where *xxx* is your initials.

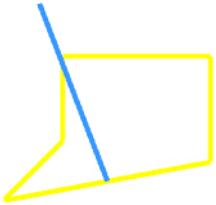
Drawing the Holder

1 Change the wireframe color to **116**.

This is the recommended color for tool holders.



2 Draw a line starting at the midpoint of the insert's lower line, setting its length to **45** and its angle to **111**.



3 Draw a line from the top end of the previous line, setting its length to **210** and its angle to **90**.

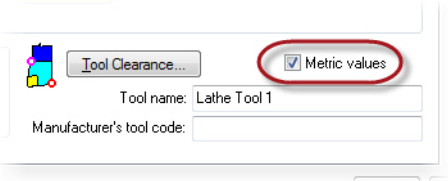
4 Draw a line from the top end of the previous line, setting its length to **50** and its angle to **0**.

5 Draw a line from the right end of the previous line, setting its length to **247** and its angle to **270**.

6 Draw a final line that closes the holder's shape, and click **OK** to exit the line function.

7 Save the file as `CustomTool02_xxx`, where *xxx* is your initials.

NOTE: The geometry used for custom tools is scaled to the current system units (inch or metric). Mastercam assumes that the file with the tool geometry uses the same units selected on the Parameters tab of the Define Tool dialog box.



Setting the Tool Parameters

After you draw the tool's geometry, you must set the tool parameters that tell Mastercam how to interpret and use the tool geometry. These parameters include the following:

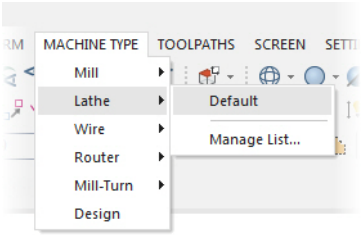
- **Geometric Tool Orientation** - Tells Mastercam how to interpret the tool profile geometry, including cut and plunge direction.
- **Tool Geometry** - Tells Mastercam whether to read the geometry from a file or from a level in the current part file.
- **Tool Orientation in Turret** - Defines how the tool is mounted in the turret, including cut and plunge direction.

- **Insert corner radius** - Specifies the radius of the tool's tip.
- **Tool Center** - Specifies the location of the tool tip's centerpoint.
- **Compensation** - Determines the tool compensation to use when Mastercam calculates a toolpath.

Setting the Orientation and Position

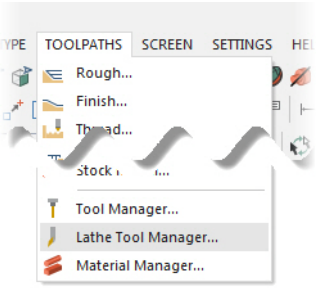
1 From Mastercam's **Machine Type** menu, select the default lathe machine.

Mastercam starts a new machine group in Toolpaths Manager.



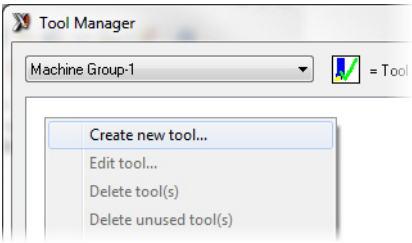
2 From Mastercam's menu, choose **Toolpaths, Lathe Tool Manager**.

The **Tool Manager** dialog box opens.



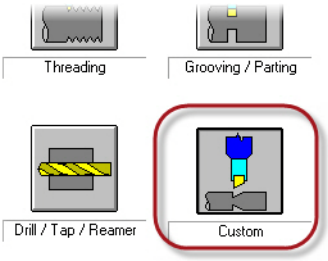
3 Right-click in the upper box, and choose **Create new tool** from the pop-up menu.

The **Define Tool** dialog box opens.



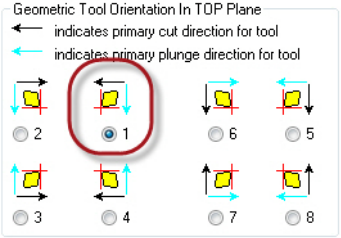
4 Select the **Custom** tool type.

The dialog box automatically switches to the **Geometry** tab.



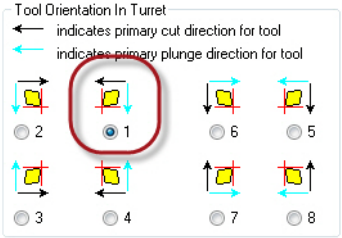
5 In the **Geometric Tool Orientation In TOP Plane** box, ensure that orientation **1** is selected.

This is the orientation of the tool's geometry as it is viewed in the Top plane. Note that choices **1** and **5** have the same geometric orientation, but the cut and plunge directions are different. Orientation **5** is typically used for horizontal tools.

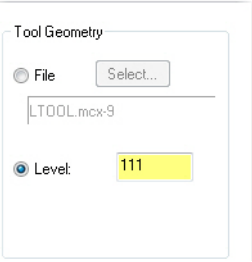


6 In the **Tool Orientation In Turret** box, ensure that orientation **1** is selected.

This is the orientation of the tool as it will be mounted in the turret. Note that, again, choices **1** and **5** have the same geometric orientation, but the cut and plunge directions are different.



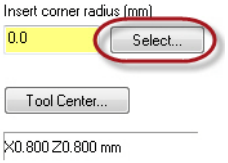
7 In the **Tool Geometry** box, select **Level**, and ensure that the level shown is **111**. This is the level on which you drew the tool geometry.



NOTE: If you have your tool geometry in a separate file, you would choose the **File** option, and then use the **Select** button to browse to the file containing the custom tool.

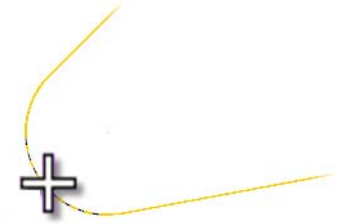
8 Click the **Select** button next to the **Insert corner radius** box.

The dialog box minimizes so that you can access the graphics window.



9 Zoom in on the insert's tip, and select the insert's fillet.

Mastercam restores the **Define Tool** dialog box, with **0.8** entered into the **Insert corner radius** box.

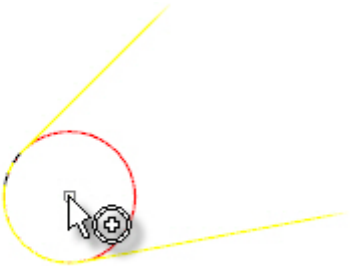


10 Click the **Tool Center** button.

The dialog box minimizes so that you can access the graphics window.

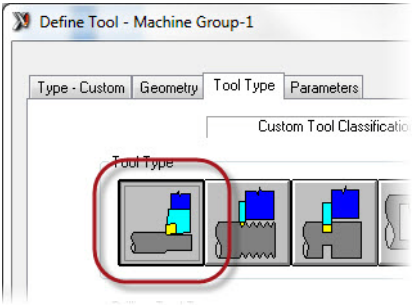
11 Zoom in on the insert's tip, and select the fillet arc's centerpoint.

Mastercam restores the **Define Tool** dialog box, with **X0.800 Z0.800 mm** entered as the tool's center.



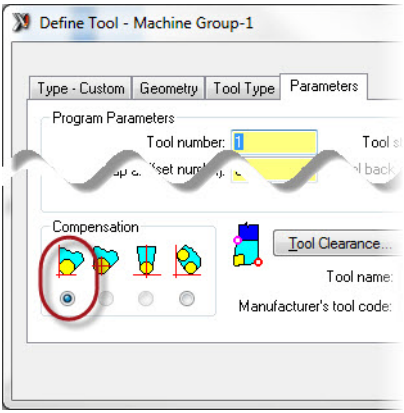
Setting Tool Type and Compensation

1 Click the **Tool Type** tab, and ensure that **General Turning** is selected.



2 Click the **Parameters** tab, and ensure that the first **Compensation** option is selected.

This selection specifies that compensation for a toolpath is based on the sharp corner of the tip corner radius. You were first introduced to this setting on page 7.



Setting Up the Tool

When you have finished specifying the tool's general parameters, you must perform the tool setup. These settings define the physical orientation of a tool and include, among others, the following:

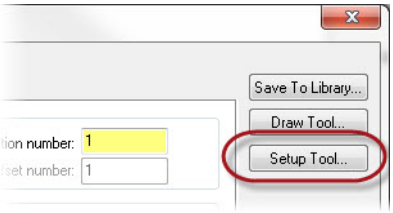
- **Mounting Position** - Specifies whether the tool is mounted vertically or horizontally in the turret.
- **Reverse tool** - Specifies if you want the tool rotated in the turret 180 degrees from the default orientation. For example, you would use this option when you want to use a general turning tool for ID turning or a boring bar for OD machining.
- **Turret** - Tells Mastercam in which turret the tool is mounted.

- **Default Active Spindle** - Defines the spindle that Mastercam selects by default when you use this tool in an operation.
- **Spindle Rotation** - Tells Mastercam the direction of spindle rotation.

Setting Up the Tool

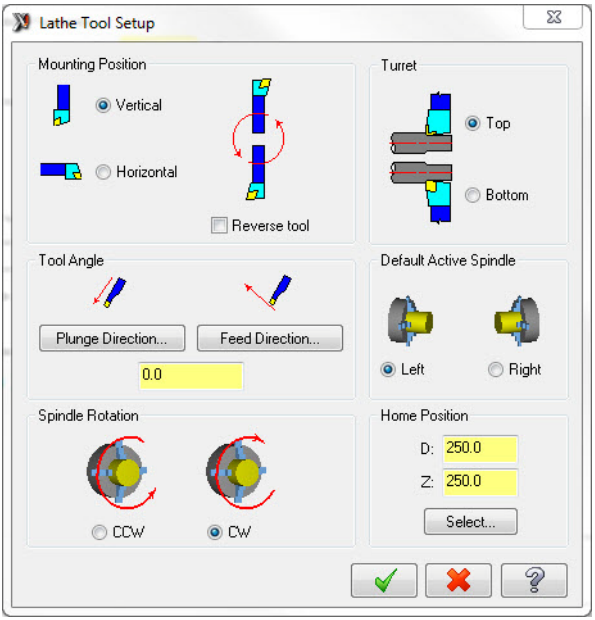
1 Click the **Setup Tool** button.

The **Lathe Tool Setup** dialog box opens.



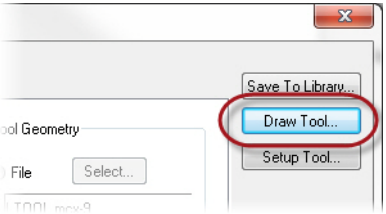
2 Select the **Vertical** mounting position, the **Top** turret, the **Left** default spindle, and **CW** spindle rotation.

Your settings (except possibly **Home Position**) should match the picture below. Notice that, in the **Mounting Position** box, the vertical tool picture is oriented as you expect for the upper turret, left spindle.



3 Click **OK** in the **Lathe Tool Setup** dialog box.

4 Click **Draw Tool**.



The dialog box minimizes so that you can see the tool's current state in the graphics window.

TIP: As you manipulate the tool's setup, use the **Draw Tool** button often to check the results of your changes and to ensure that the tool looks as you expect.

5 Press [**Enter**] to return to the dialog box.

6 Click **OK** in the **Define Tool** and **Tool Manager** dialog boxes.

The tool is now ready to test.

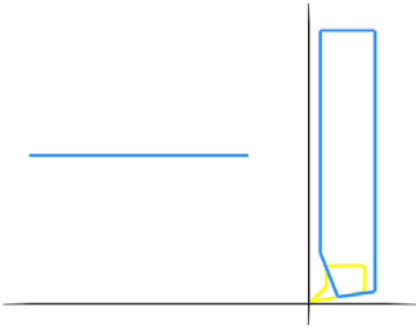
7 Save the file as `CustomTool03_xxx`, where *xxx* is your initials.

Testing the Tool in a Toolpath

1 In Mastercam's Status Bar, change the level number to **1**.

2 Draw a line in the upper-left quadrant (minus X and plus Y position), setting its length to **200** and its angle to **0**.

You will use this line to create a simple lathe finish toolpath.

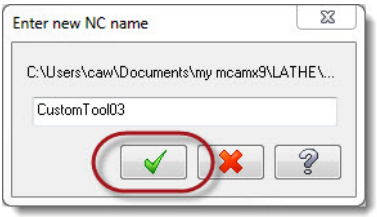


3 From Mastercam's menu, choose **Toolpaths, Finish**.

The **Enter new NC name** dialog box opens.

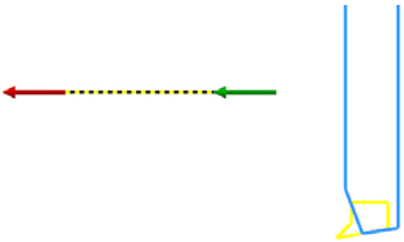
4 Click **OK** to accept the default file name.

The **Chaining** dialog box opens.



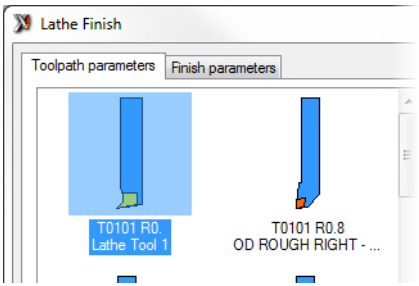
5 Chain the line as shown to the right, and click **OK**.

The **Lathe Finish** dialog box opens.

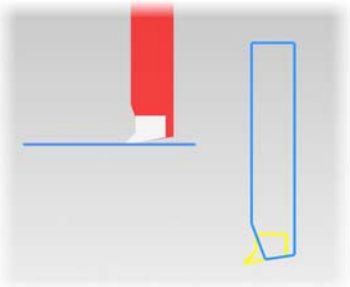


6 Ensure that your custom tool is selected, and click **OK**.

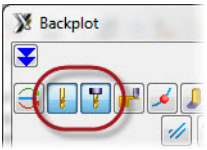
Mastercam creates the toolpath.



7 Backplot the toolpath to ensure that it works as expected.



NOTE: If you do not see the tool and holder, turn them on in the **Backplot** dialog box. These settings are off by default.



8 Save the file as `CustomTool04_xxx`, where *xxx* is your initials.

LESSON 2
Graphical Orientation and Tool Settings

In this lesson, you experiment with some of the tool settings to see how they relate to the tool's location in the graphics window. You will see that you can draw the tool wherever you like, but that the tool's position affects the settings you must use, as well as the relative location of the toolpath.

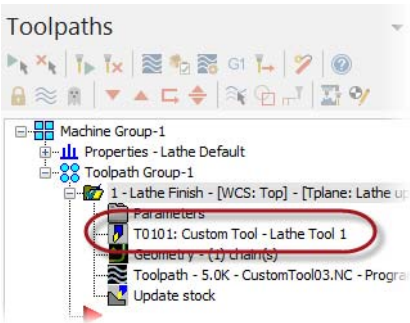
Continue with the part file you completed in the previous lesson, or load the file `CustomTools04` included with this tutorial.

▶ Using Tool Orientation

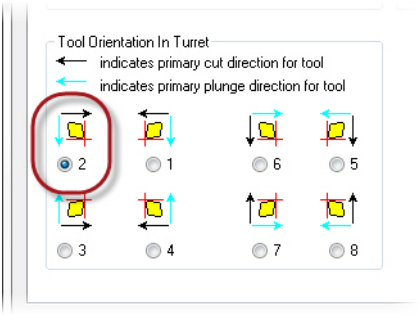
1 Backplot the toolpath, and notice how the tool is moving correctly along the geometry.

2 In the **Toolpaths Manager**, click the custom tool.

The **Define Tool** dialog box opens.



3 Change **Tool Orientation in Turret** to **2**.



NOTE: Make sure that you change the **Tool Orientation in Turret** setting and not the **Geometric Tool Orientation in TOP Plane** setting. Because the geometric orientation is based on how you drew the tool in the graphics window, you do not need to change it once it is set.

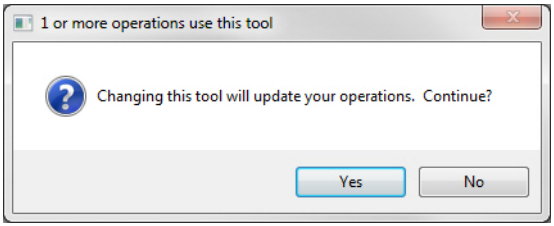
4 Click the **Draw Tool** button, and notice how the tool orientation has changed to match the selected option.

5 Press [**Enter**] to return to the dialog box, and click **OK**.

The **1 or more operations use this tool** dialog box opens.

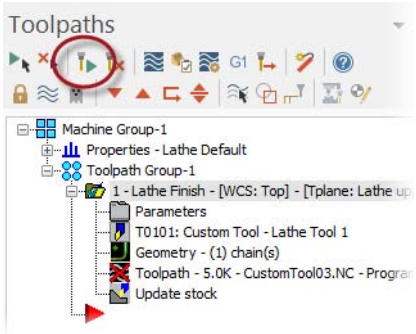
6 Click **Yes** to accept that the changes will update the operation.

Mastercam marks the toolpath dirty.



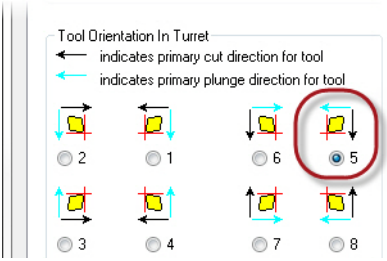
7 Click **Regenerate all selected operations**.

Mastercam regenerates the operation.



8 Backplot the operation, and notice how the tool is now oriented the opposite direction.

9 Click on the tool in **Toolpaths Manager**, and change **Tool Orientation in Turret** to **5**.



10 Click the **Draw Tool** button, and notice how the tool orientation has changed to match the selected option.

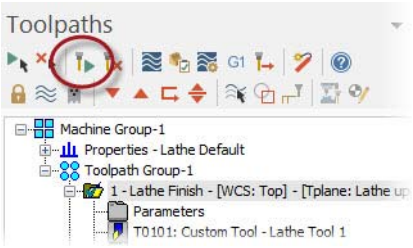
11 Press [**Enter**] to return to the dialog box, and click **OK**.

The **1 or more operations use this tool** dialog box opens.

12 Click **Yes** to accept that the changes will update the operation.

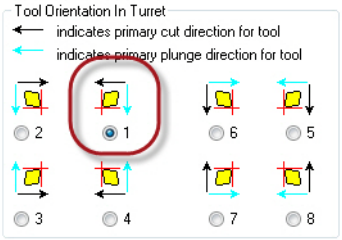
Mastercam marks the toolpath dirty.

13 Regenerate the operation.



14 Backplot the operation, and notice how the tool's orientation has changed.

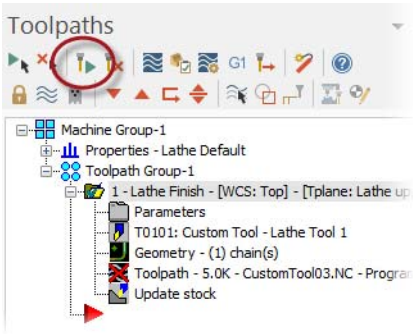
15 Click the tool in **Toolpaths Manager**, change **Tool Orientation in Turret** back to **1**, and click **OK**.



16 Click **Yes** to accept that the changes will update the operation.

Mastercam marks the toolpath as dirty.

17 Regenerate the operation, and backplot.

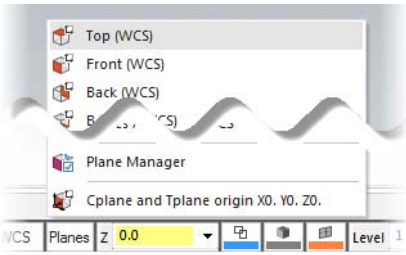


The operation is now back to its original state, as shown below.

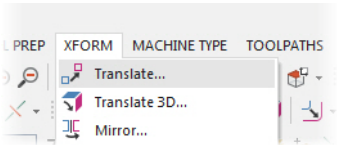
Tool Position in the Graphics Window

1 If your gnomon in the graphics window shows +D and Z (rather than X and Y), in Mastercam's Status Bar, click **Planes**, and select **Top (WCS)**.

The graphics window changes to the Top view.



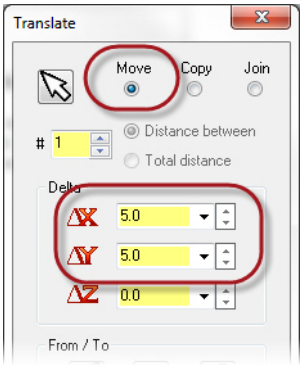
2 From Mastercam's menu, choose **Xform, Translate**.



3 In the graphics window, select all of the tool geometry, and press [**Enter**].

The **Translate** dialog box opens.

4 Translate the entire tool by **5** in both X and Y, and click **OK**.

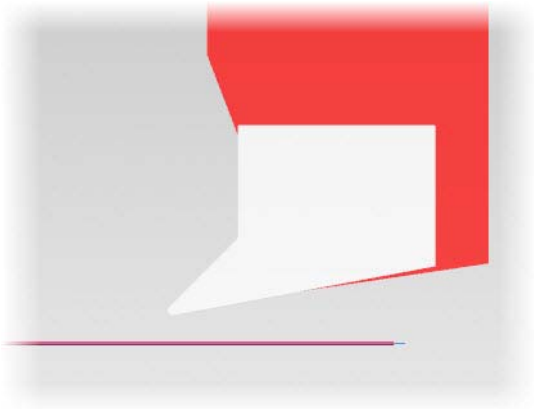


5 Click **Clear Colors** to reset the tool's color values.

6 Zoom in on the origin to see how the insert's tip is now oriented.

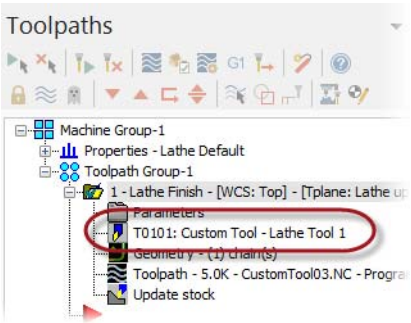
7 Backplot, and notice that the image of the tool is now above the toolpath.

This is because you have moved the tool's centerpoint to X5.8, Y5.8, but Mastercam still thinks it's at X0.8, Y0.8.



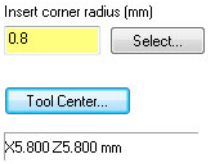
8 Click the tool in **Toolpaths Manager**.

The **Define Tool** dialog box opens.



9 Click the **Tool Center** button, reset the tool center to its new location.

The tool center should be set to **X5.800 Z5.800 mm**.

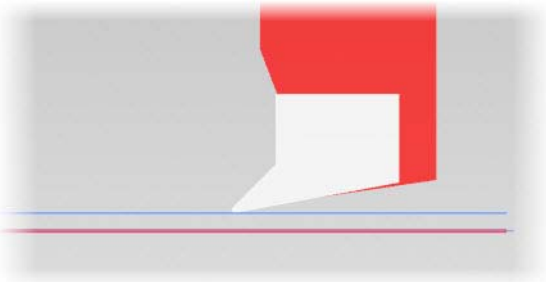


10 Click **OK** in the **Define Tool** dialog, click **Yes** in the message that displays, and regenerate the toolpath.

11 Backplot the toolpath.

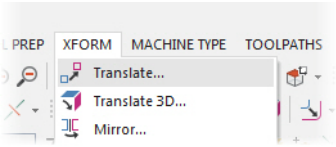
Notice that the tool is now back in the correct position, but the toolpath has been moved down by 5 mm.

This is because the relative position of the tool to the toolpathed geometry has changed by 5 mm.



Restore the Tool to its Original Settings

1 From Mastercam's menu, choose **Xform, Translate**.

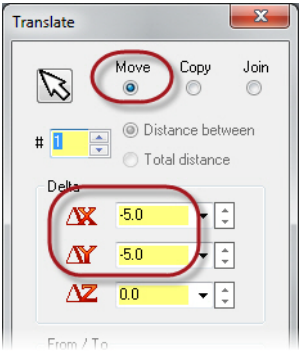


2 Select all of the tool geometry, and press [**Enter**].

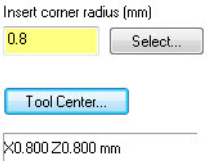
The **Translate** dialog box opens.

3 Translate the entire tool by -**5** in both X and Y, and click **OK**.

This translation returns the tool geometry to its original position.



4 Reset the tool's centerpoint, so that it is set to **X0.800 Z0.800 mm**.



5 Click **OK** in the dialog box, and then click **Yes** in the message box that displays.

6 Click **Clear Colors**, regenerate the operation, and backplot.

The operation is now back to its original state, as shown below.

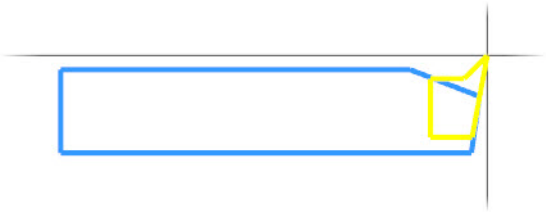
LESSON 3
Creating a Horizontally Oriented Tool

In this lesson, you create a horizontally oriented tool that will be used in the machine's upper turret to cut material on the right spindle. To do this, you complete the following procedures:

- Reposition the vertical tool you created in Lesson 1.
- Set the tool's properties.
- Test the tool with a toolpath.

Repositioning the Tool

In this exercise, rather than draw a new tool from scratch, you reorient the vertical tool you created so that it lies horizontally in the lower-left quadrant of the graphics window, as shown below. This puts the tool in the expected position for a horizontal tool that will cut on the right spindle.

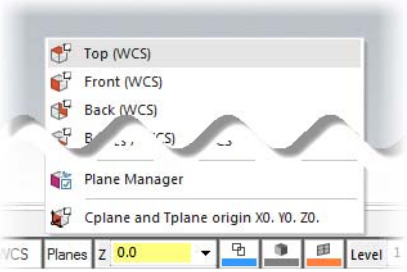


▶ Rotating the Tool

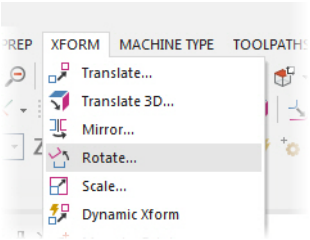
1. Start Mastercam.
2. Load the `CustomTool02` file, either the one you created in Lesson 1 or the one included with this tutorial.
3. If necessary, press [**F9**] to display the axes in the graphics window.

4 If your gnomon in the graphics window shows +D and Z (rather than X and Y), in Mastercam's Status Bar, click **Planes**, and select **Top (WCS)**.

The graphics window changes to the Top view.

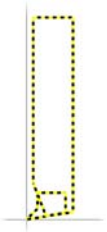


5 From Mastercam's menu, choose **Xform, Rotate**.

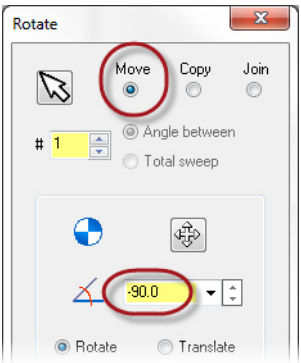


6 Select all of the tool geometry, and press [**Enter**].

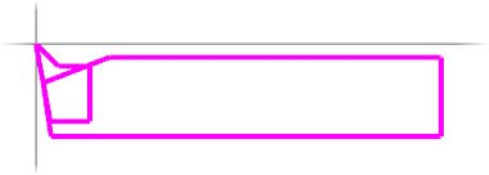
The **Rotate** dialog box opens.



7 Select **Move**, set the rotation angle to **-90** degrees, and click **OK**.

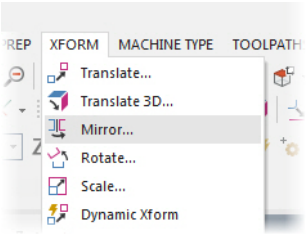


The tool geometry rotates into the position shown below.

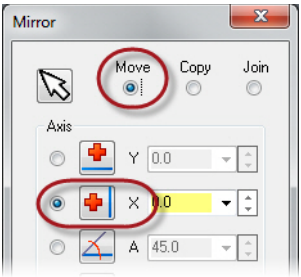


8 Choose **Xform, Mirror**, select all of the tool geometry, and press [**Enter**].

The **Mirror** dialog box opens.

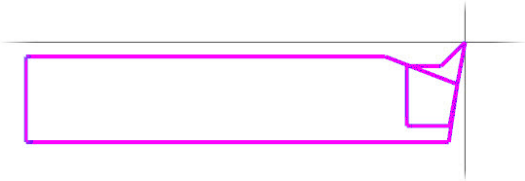


9 Choose **Move** and the radio button to the left of **X**, and then click **OK**.



NOTE: If you have D and Z axes instead of X and Y, choose **Z**.

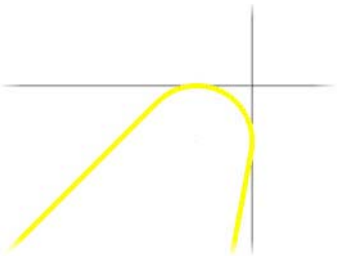
The tool geometry should now be in the position shown below.



10 In Mastercam's menu, click **Clear Colors**.

Mastercam restores the geometry to its original colors.

11 Zoom in on the origin, and notice how the insert's tip is now positioned.



12 Save the file as `CustomTool05_xxx`, where *xxx* is your initials.

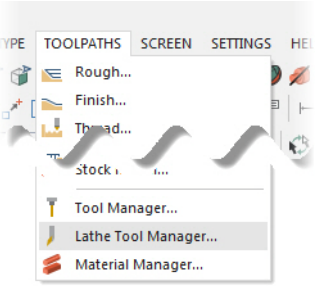
Setting the Tool Parameters

After you draw the tool's geometry, you must set the tool parameters that tell Mastercam how to interpret and use the tool geometry. These settings are different for this horizontal tool than for the vertically oriented tool you created earlier in the tutorial.

▶ Setting the Orientation and Position

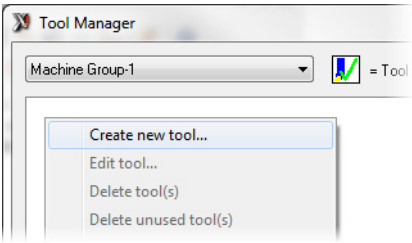
1 From Mastercam's menu, choose **Toolpaths, Lathe Tool Manager**.

The **Tool Manager** dialog box opens.



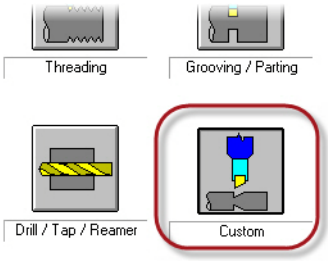
2 Right-click in the upper box, and choose **Create new tool** from the pop-up menu.

The **Define Tool** dialog box opens.



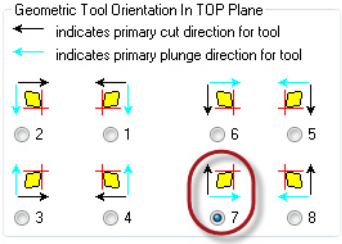
3 Select the **Custom** tool type.

The dialog box automatically switches to the **Geometry** tab.



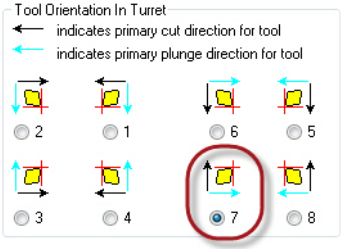
4 In the **Geometric Tool Orientation In TOP Plane** box, ensure that orientation **7** is selected.

This is the orientation of the tool's geometry as it is viewed in the Top plane. Note that choices **3** and **7** have the same geometric orientation, but the cut and plunge directions are different.



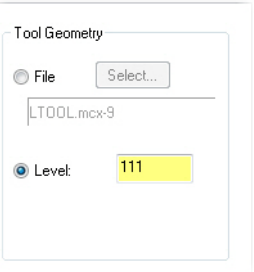
5 In the **Tool Orientation In Turret** box, ensure that orientation **7** is selected.

This is the orientation of the tool as it will be mounted in the turret. Note that, again, choices **3** and **7** have the same geometric orientation, but the cut and plunge directions are different.



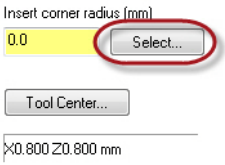
6 In the **Tool Geometry** box, ensure that **Level** is selected and that the level shown is **111**.

This is the level on which you drew the tool geometry.



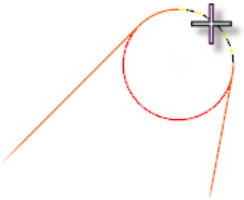
7 Click the **Select** button next to the **Insert corner radius** box.

The dialog box minimizes so that you can access the graphics window.



8 Zoom in on the insert's tip, and select the tip's arc.

Mastercam restores the **Define Tool** dialog box, with **0.8** entered into the **Insert corner radius** box.

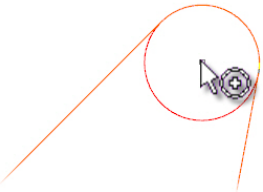


9 Click the **Tool Center** button.

The dialog box minimizes so that you can access the graphics window.

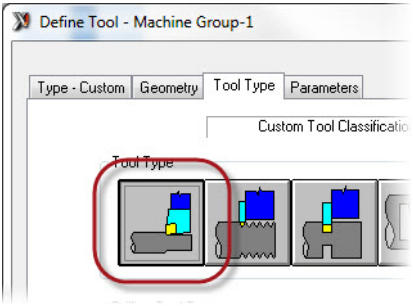
10 Zoom in on the insert's tip, and select the tip's centerpoint.

Mastercam restores the **Define Tool** dialog box, with **X-0.800 Z-0.800** entered as the tool's center.



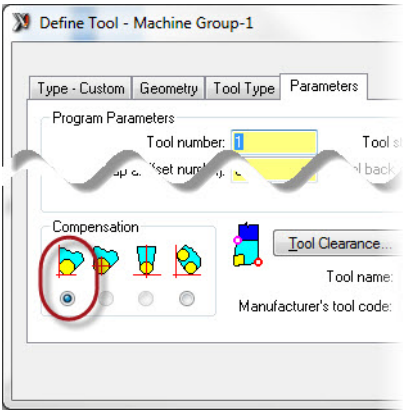
Setting Tool Type and Compensation

1 Click the **Tool Type** tab, and ensure that **General Turning** is selected.



2 Click the **Parameters** tab, and ensure that the first **Compensation** option is selected.

This selection specifies that compensation for a toolpath is based on the sharp corner of the tip corner radius.



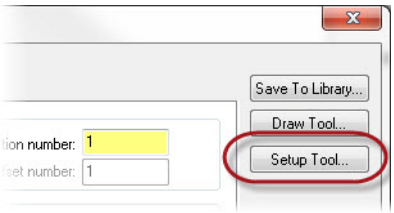
Setting Up the Tool

When you have finished setting the tool's general parameters, you can set the options in the Lathe Tool Setup dialog box.

Setting Up the Tool

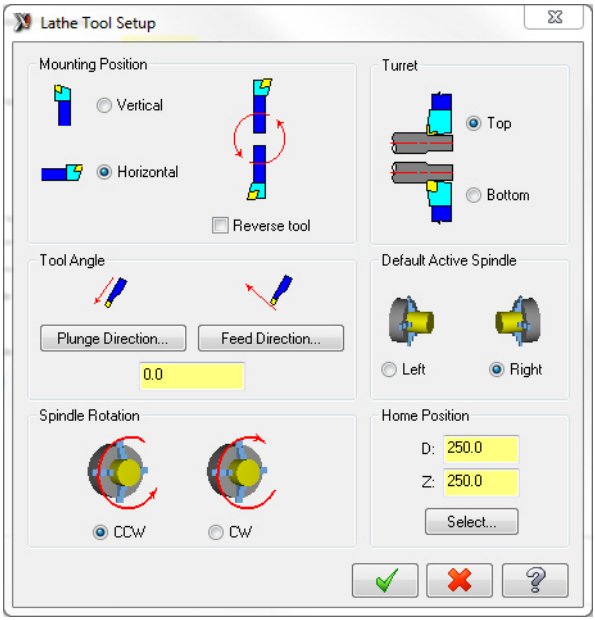
1 Click the **Setup Tool** button.

The **Lathe Tool Setup** dialog box opens.



2 Select the **Horizontal** mounting position, the **Top** turret, the **Right** spindle, and **CCW** spindle rotation.

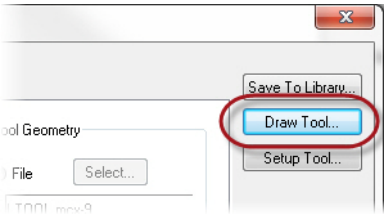
Your settings, except possibly **Home Position**, should match the picture below.



3 Click **OK** in the **Lathe Tool Setup** dialog box.

4 Click **Draw Tool**.

The dialog box minimizes so that you can see the tool's current state in the graphics window.



5 Press [**Enter**] to return to the dialog box.

6 Click **OK** in the **Define Tool** and **Tool Manager** dialog boxes.

The tool is now ready to test.

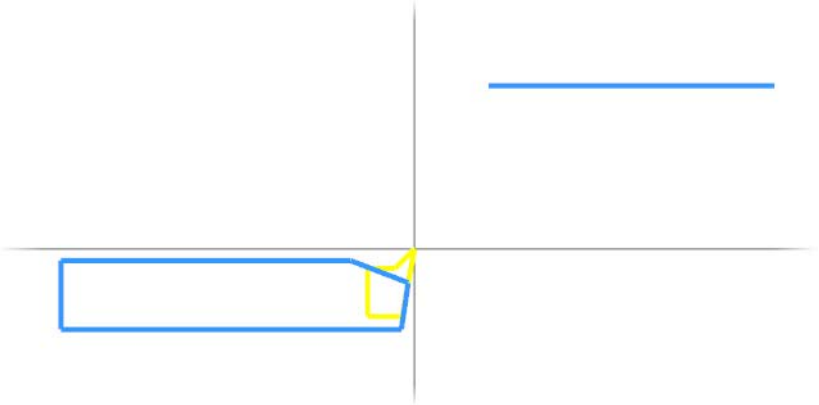
7 Save the file as `CustomTool06_xxx`, where *xxx* is your initials.

Testing the Tool in a Toolpath

1 In Mastercam's Status Bar, change the level number to **1**.

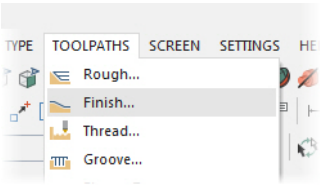
2 Draw a horizontal line on the screen in the upper-right quadrant of the graphics window.

You will use this line to create a simple lathe finish toolpath.



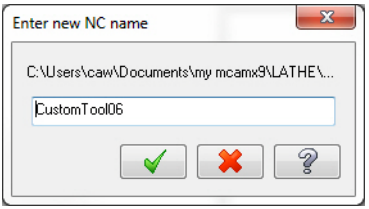
3 From Mastercam's menu, choose **Toolpaths, Finish**.

The **Enter new NC name** dialog box opens.



4 Click **OK** to accept the default file name.

The **Chaining** dialog box opens.

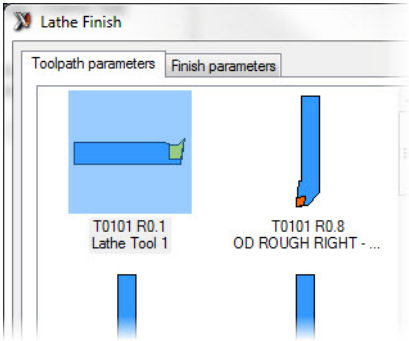


5 Chain the line as shown to the right, and click **OK**.

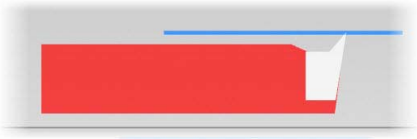
The **Lathe Finish** dialog box opens.

6 Ensure that your custom tool is selected, and click **OK**.

Mastercam creates the toolpath.



7 Backplot the toolpath to ensure that it works as expected.

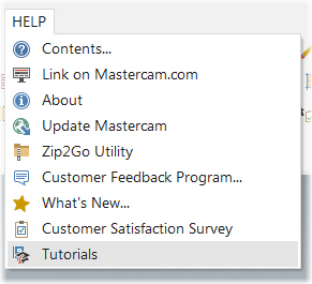


8 Save the file as `CustomTool07_xxx`, where *xxx* is your initials.

Conclusion

Congratulations! You have completed the *Lathe Custom Tools* 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. The Mastercam tutorial series is in continual development, and we will add modules as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.



Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's menu bar or by pressing [**Alt+H**] on your keyboard. Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—CNC offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. The Mastercam tutorial series is in continual development, with new modules added as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.

- *Online communities*— You can find a wealth of information, including many videos, at www.mastercam.com.

  For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

  Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledge base at kb.mastercam.com. To register, select **Help**, **Link on Mastercam.com** from the Mastercam menu and follow the instructions.

Mastercam Documentation

Mastercam installs the following documents in the `\Documentation` folder of your Mastercam installation:

- *What's New in Mastercam X9*
- *Mastercam X9 Installation Guide*
- *Mastercam X9 Administrator Guide*
- *Mastercam X9 Transition Guide*
- *Mastercam X9 Quick Reference Card*
- *Mastercam X9 ReadMe*

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

Attention! Updates may be available.
Go to Mastercam.com/Support for the latest downloads.

cnc software, inc.

671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com