LATHE CUSTOM TOOLS

June 2017

POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam®
Be Dynamic.

LATHE CUSTOM TOOLS

June 2017

Software: Mastercam 2018

Terms of Use

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

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

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: 2 hours

General Tutorial Requirements

All Mastercam 2018 tutorials have the following general requirements:

- You must be comfortable using the Windows® operation 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 access from the Mastercam 2018 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.

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.

Exercise 1: Drawing the Insert's Geometry

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.

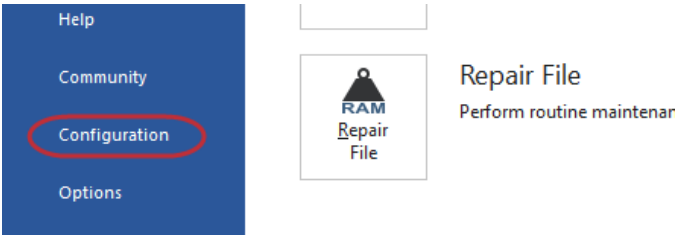
1. Start Mastercam using your preferred method:
    a. Double-click Mastercam's desktop icon.

    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

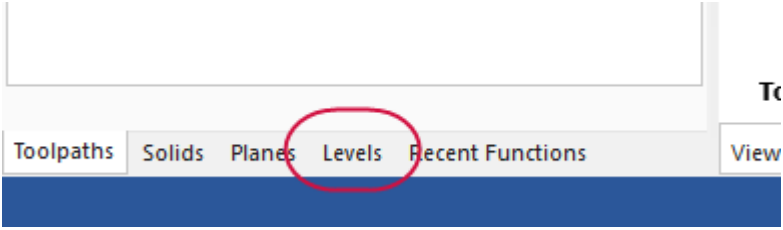
System Configuration dialog box.



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

d. Click **OK**.

3. Click the **Levels** tab in the lower-left of Mastercam's window.

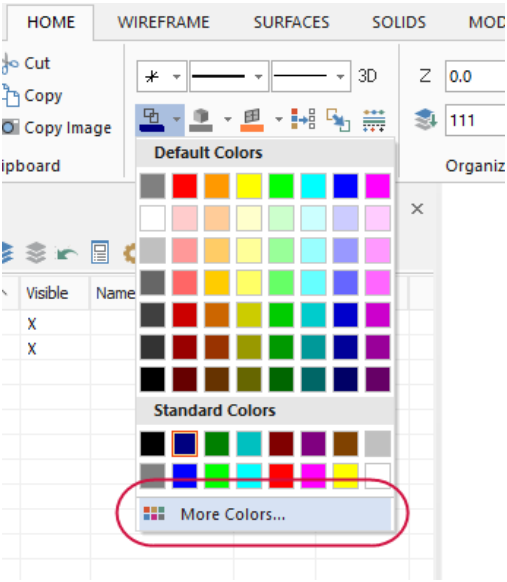


The Levels Manager displays.

4. Change the **Number** field to 111.

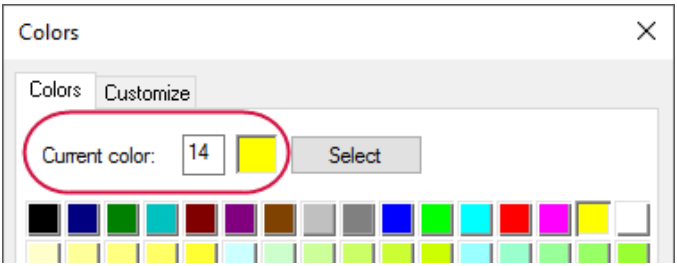
Because tool geometry must be on its own level, with no other geometry, you should pick a level number that you are unlikely to use in your part drawings.

5. Select **Wireframe Color**, **More Colors** from the **Home** tab.



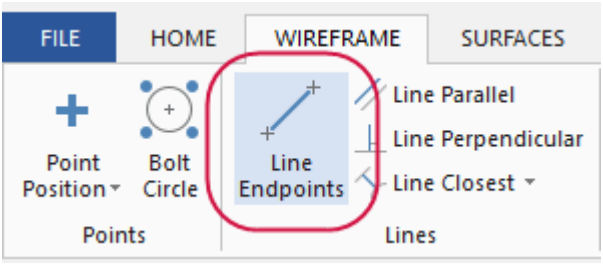
The Colors dialog box displays.

6. Change **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.

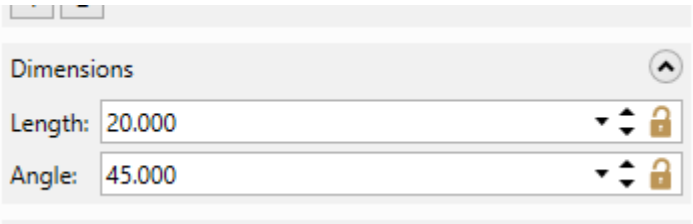
7. Choose **Line Endpoints** from the **Wireframe** tab.



The Line Endpoints function panel displays.

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

9. In the function panel, set **Length** to **20**, **Angle** to **45**, and press [**Enter**].



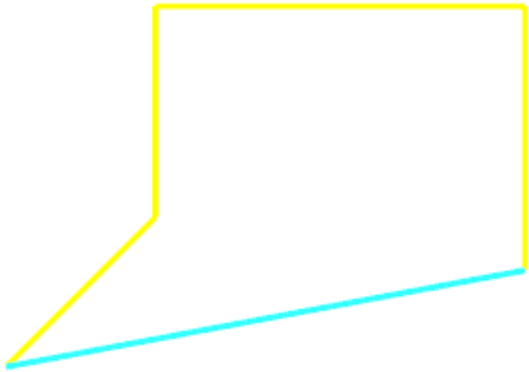
10. Draw another line from the first line's upper endpoint.

11. Set the line's **Length** to **20**, **Angle** to **90**, and press [**Enter**].

12. Draw a line from the previous line's endpoint, giving it a **Length** of **35** and an **Angle** of **0**.

13. Draw a line from the previous line's endpoint, giving it a **Length** of **25** and an **Angle** of **270**.

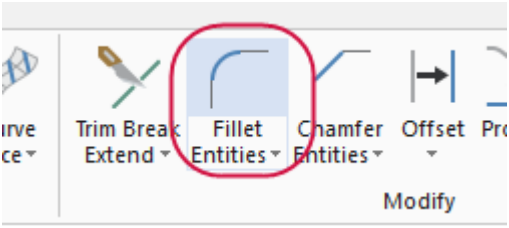
14. Draw a final line that closes the insert's shape, and click **OK** to exit the function panel.



Exercise 2: Completing the Insert's Tip

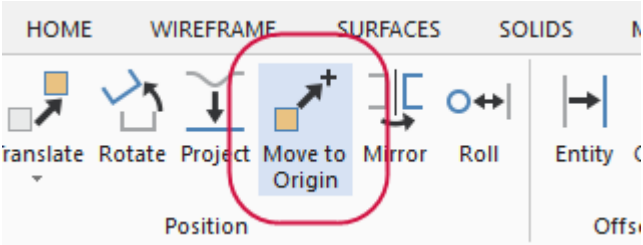
1. Zoom in on the insert's tip.

2. Choose **Fillet Entities** from the **Wireframe** tab.



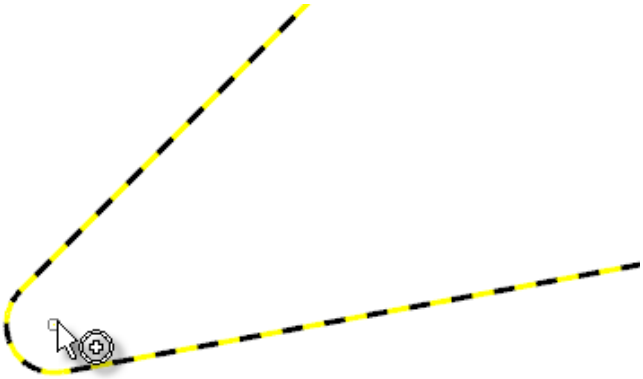
3. Select the tip's two lines, set **Radius** to **0.8**, and click the function panel's **OK** button.

4. Select **Move to Origin** from the **Transform** tab.



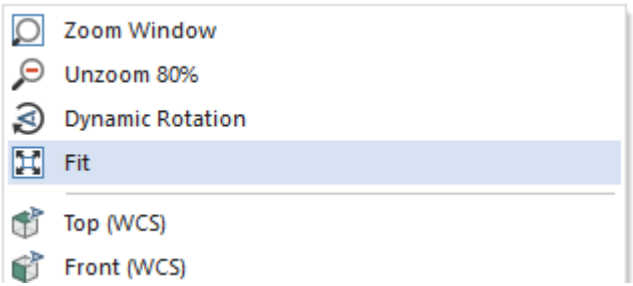
All of the geometry is selected automatically.

5. Select the fillet's centerpoint.

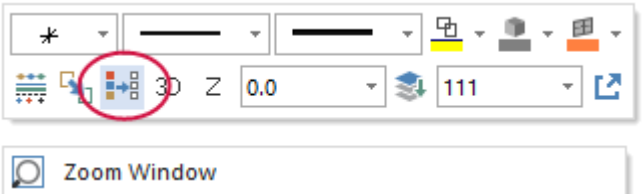


The insert moves to the origin, with the tip arc's centerpoint centered at 0,0.

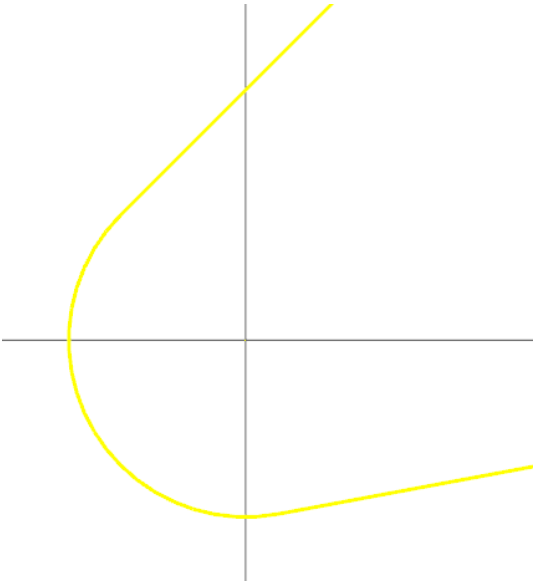
6. Right-click in the graphics window, and select **Fit**.



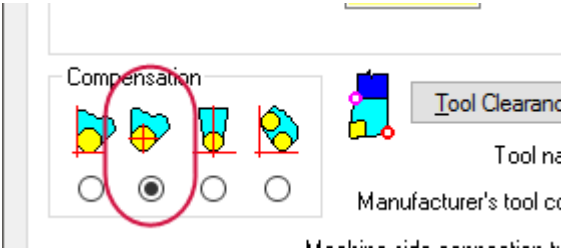
7. Right-click again, and select **Clear Colors** and then press [**F9**] to turn on the axes in the graphics window.



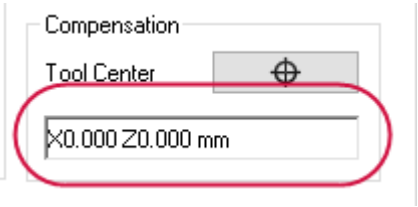
8. Zoom in on the origin, and notice how the insert's tip is now positioned.



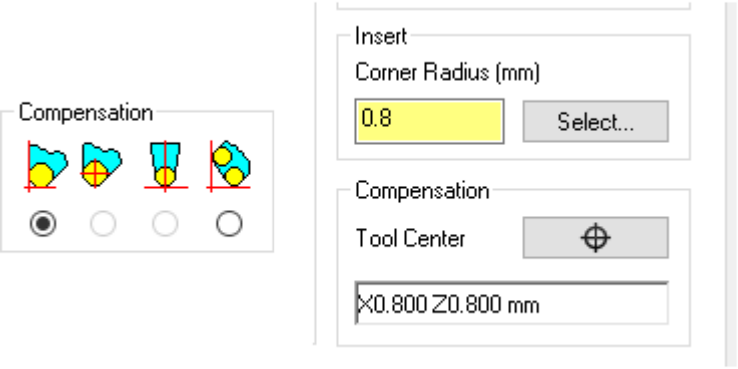
The insert's position in the graphics window determines how you define the tool's parameters in Mastercam. For example, if you were to leave the insert's position as it is in Step 8, when you set the tool's parameters, you would choose the compensation point shown below.



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

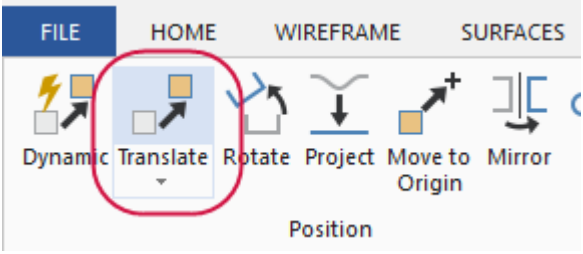


The more common position for a vertically oriented tool is with the insert's driving point fully in the upper-right quadrant of the graphics window. In this case, you would set the compensation point and tool center as shown in the following image.



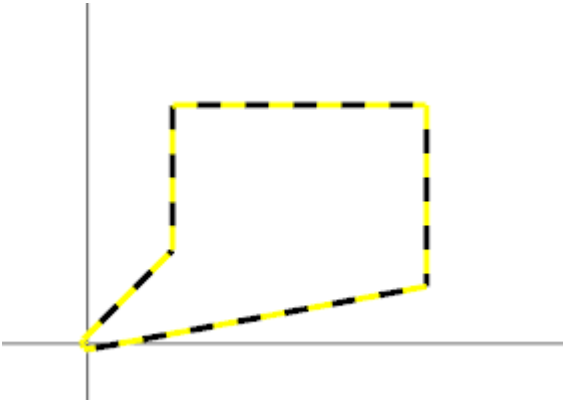
In the remaining steps of this exercise, you move the insert into the graphics window's upper-right quadrant.

9. Select **Translate** from the **Transform** tab.

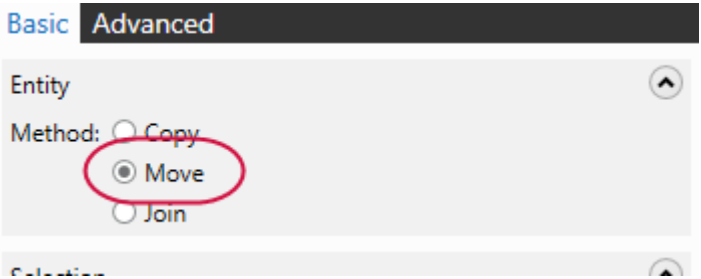


The Translate function panel displays.

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



11. Set **Method** to **Move**.

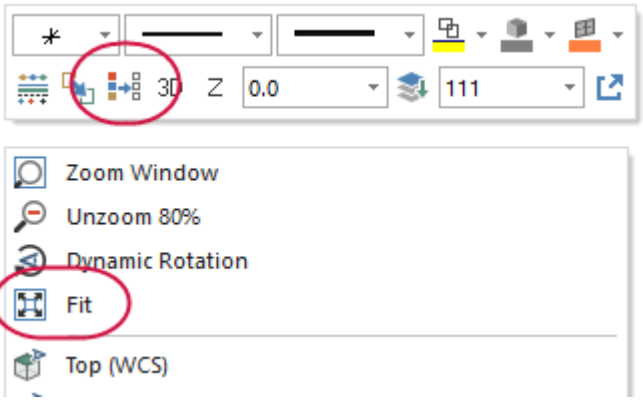


12. Set **X** and **Y** to **0.8**, and click **OK**.

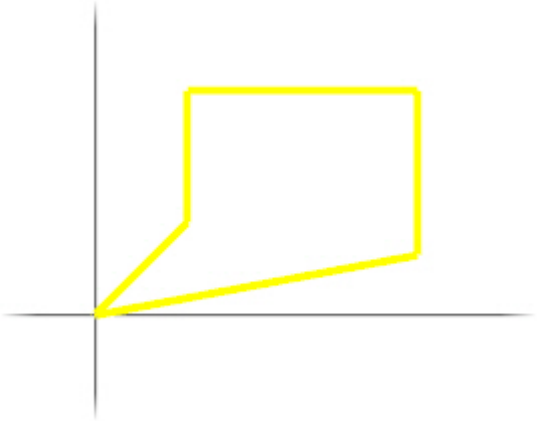
The geometry moves to the selected position.

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

14. Fit the insert to the graphics window, and choose **Clear Colors** to restore the insert's color to yellow.



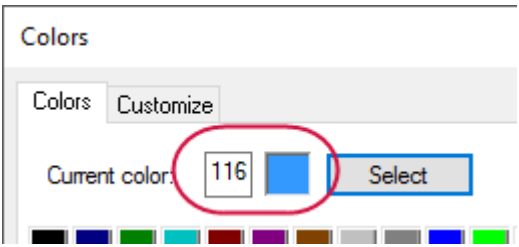
15. Your completed lathe tool insert should look like the image below.



16. Save the file as `CustomTool01_xxx`, where `xxx` is your initials.

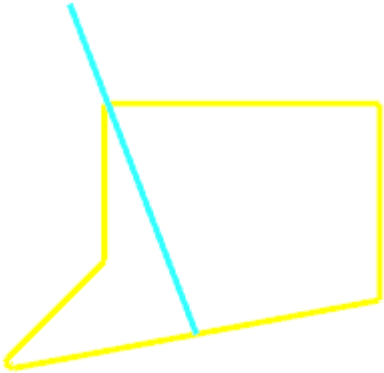
Exercise 3: Drawing the Holder

1. Press [**F9**] to hide the axes.
2. Change the **Wireframe Color** to **116**.



This is the recommended color for tool holders.

3. Draw a line starting at the midpoint of the insert's lower line, setting its **Length** to **45** and its **Angle** to **111**.



4. Draw a line from the top end of the previous line, setting its **Length** to **210** and its **Angle** to **90**.

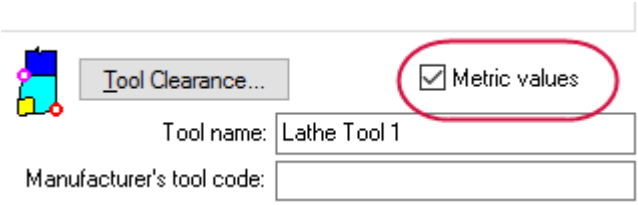
5. Draw a line from the top end of the previous line, setting its **Length** to **50** and its **Angle** to **0**.

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

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

8. 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.



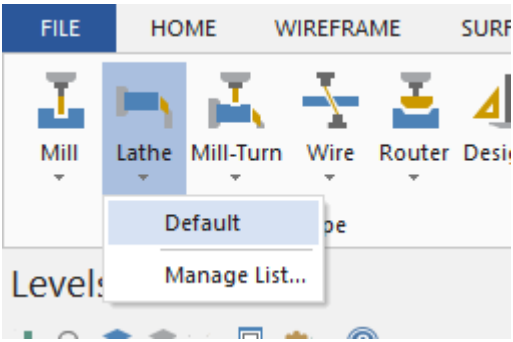
Exercise 4: Setting the Orientation and Position

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.

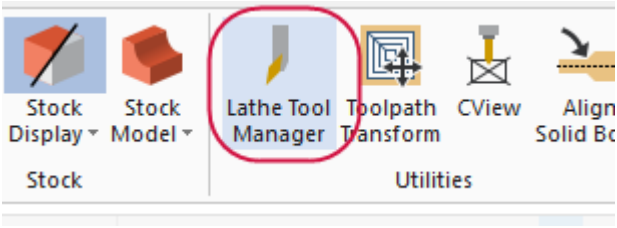
In this exercise, you will set the orientation and position of the tool.

1. Select **Lathe**, **Default** from the **Machine** tab.



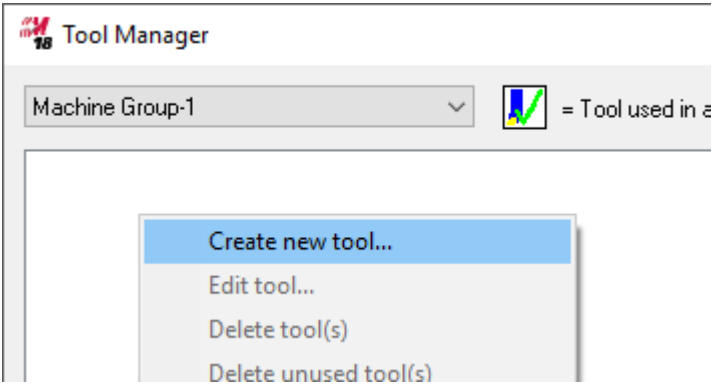
   Mastercam starts a new machine group in Toolpaths Manager.

2. Select **Lathe Tool Manager** from the **Lathe Turning** contextual tab.



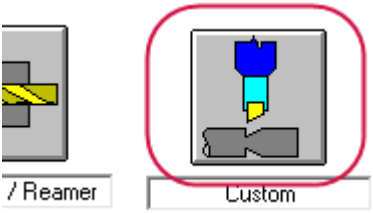
The Tool Manager dialog box displays.

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



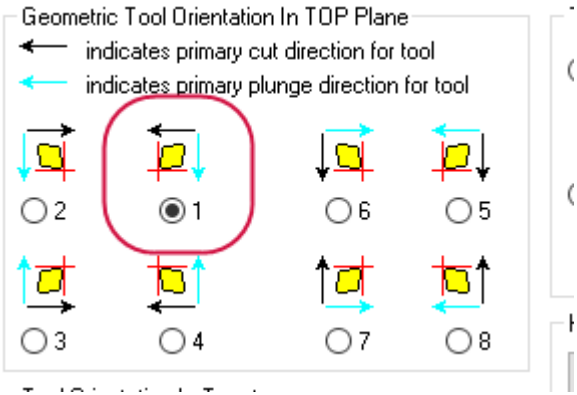
The Define Tool dialog box displays.

4. Select the **Custom** tool type.



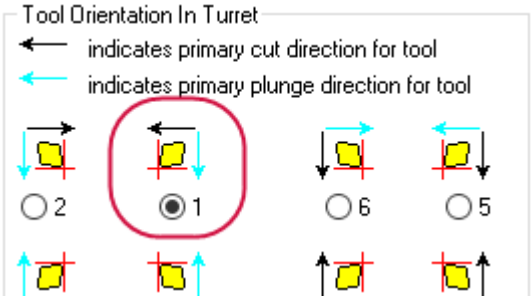
The dialog box automatically switches to the **Geometry** tab.

5. Ensure that for **Geometric Tool Orientation In TOP Plane**, 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.

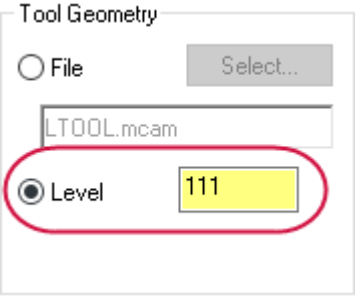
6. Ensure that for **Tool Orientation In Turret**, orientation **1** is selected.



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.

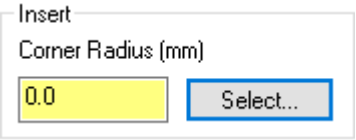
Note: If your machine has a B axis head, capable of rotating the tool for cutting at any angle, then the custom tool must be defined in the tool change orientation position.

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



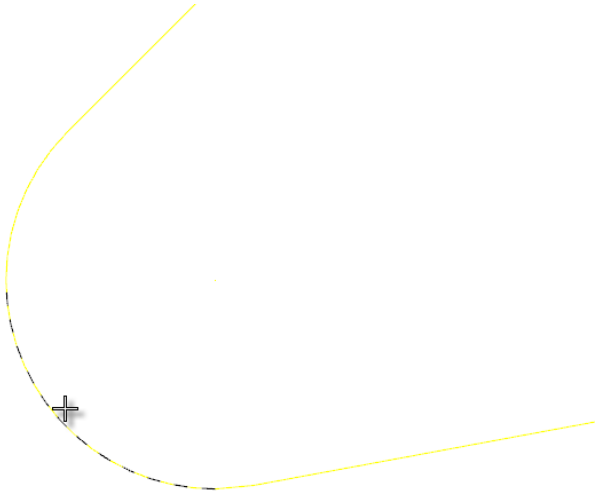
Note: If you have your tool geometry in a separate file, you would choose the **File** option, and then use **Select** to browse to the file containing the custom tool.

8. Click **Select** next to **Corner Radius (mm)**.



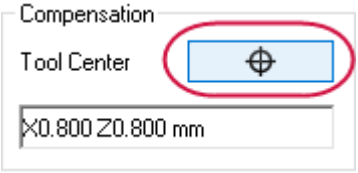
The dialog box minimizes so that you can access the graphics window.

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



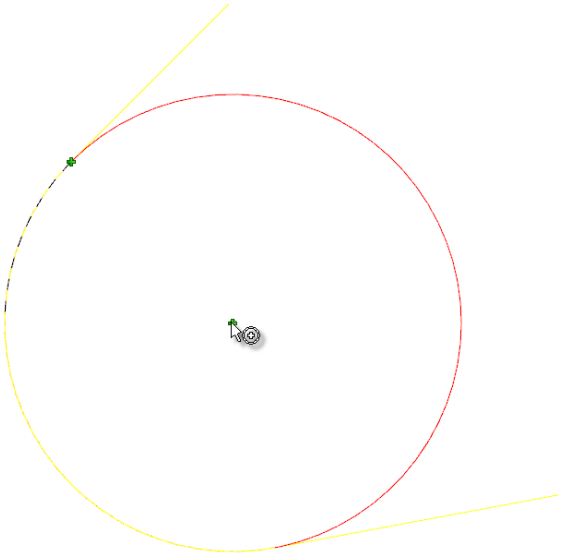
Mastercam returns you to the Define Tool dialog box, with **0.8** entered for **Corner Radius (mm)**.

10. Select **Tool Center**.



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

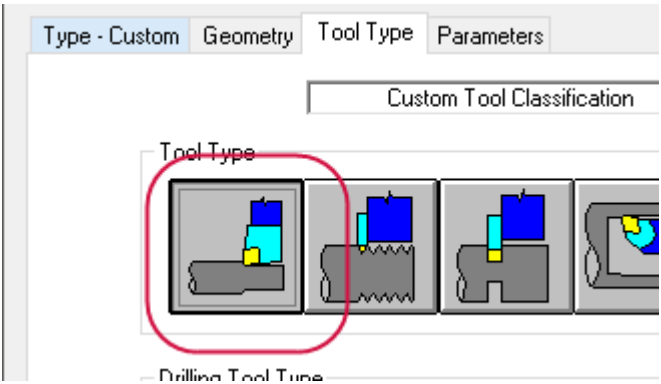
11. Zoom in on the insert's tip, and select the fillet arc's centerpoint.



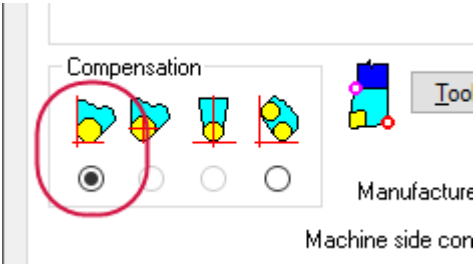
Mastercam returns you to the Define Tool dialog box, with **X0.800 Z0.800 mm** entered as the tool's center.

Exercise 5: 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.



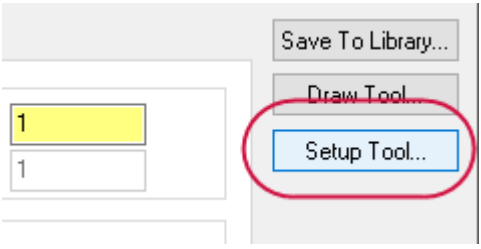
This selection specifies that compensation for a toolpath is based on the sharp corner of the tip corner radius.

Exercise 6: Setting Up the Tool

When you have specified 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.
- **Tool Angle**: Specifies the angle the tool is rotated for the toolpath.
- **Home Position**: Specifies the position to which the tool moves for tool changes.

1. Select **Setup Tool**.



   The Lathe Tool Setup dialog box displays.

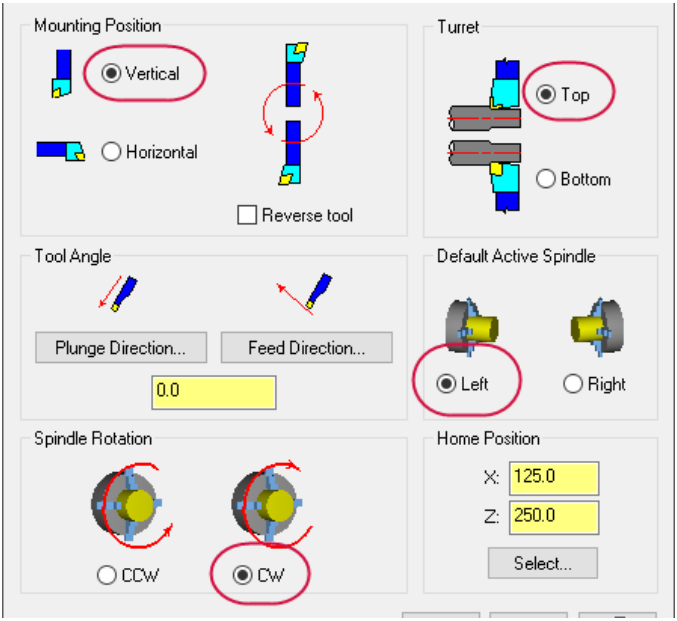
2. Select the following:

a. **Mounting Position** to **Vertical**.

b. **Turret** to **Top**.

c. **Default Active Spindle** to **Left**.

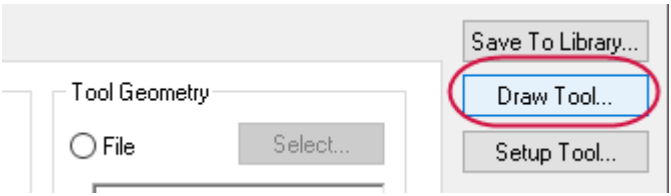
d. **Spindle Rotation** to **CW**.



Your settings (except possibly **Home Position**) should match the above image. Notice that, in Mounting Position, 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. Select **Draw Tool**.



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

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

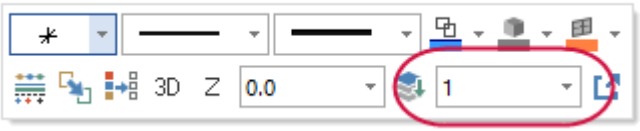
6. Press [**Enter**] to return to the dialog box.
7. Click **OK** in the Define Tool and Tool Manager dialog boxes.

   The tool is now ready to test.

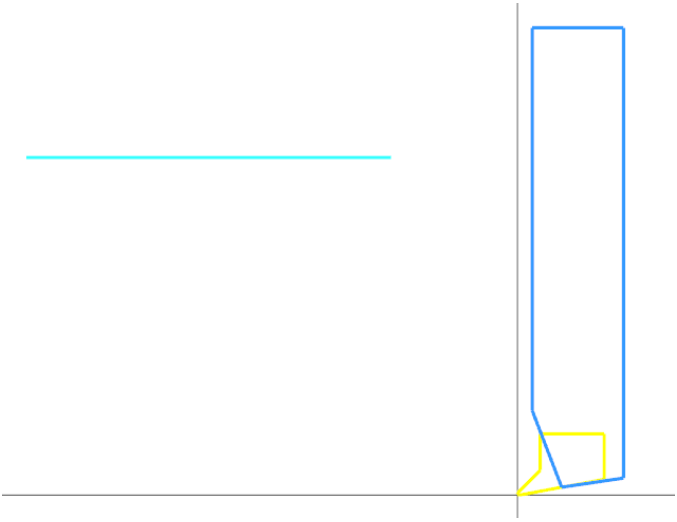
8. Save the file as `CustomTool03_xxx`, where `xxx` is your initials.

Exercise 7: Testing the Tool in a Toolpath

1. Press [**F9**] to display the axes.
2. Right-click in the graphics window, and set the level to **1**.



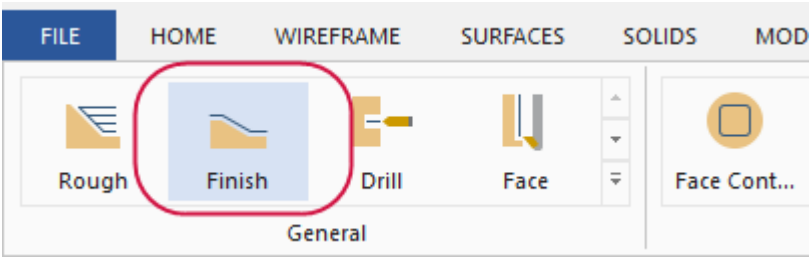
3. Use **Line Endpoints** to draw a line in the upper-left quadrant (minus X and plus Y position), setting its **Length** to **200** and its **Angle** to **0**.



   You will use this line to create a simple Lathe Finish toolpath.

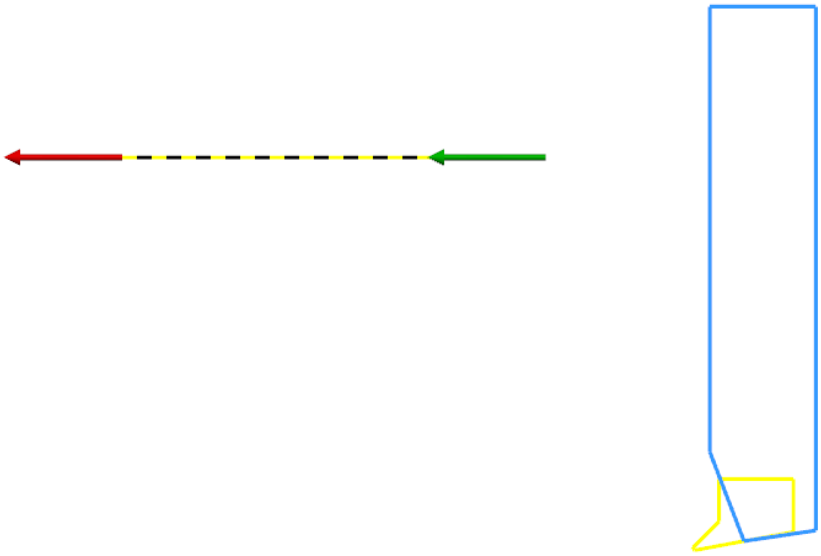
4. Click **OK**.
5. Press [**F9**] again to hide the axes.

6. Select **Finish** from the **General** gallery on the **Lathe Turning** contextual tab.



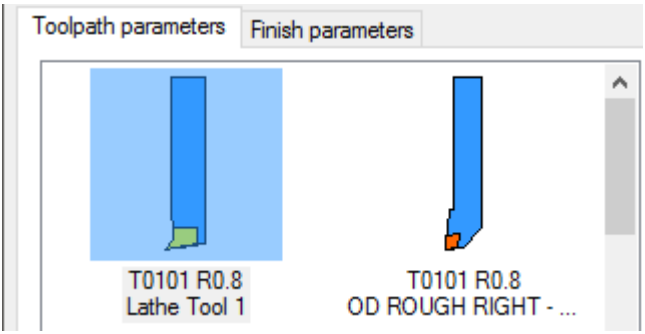
The Chaining dialog box displays.

7. Chain the line as shown below, and click **OK**.



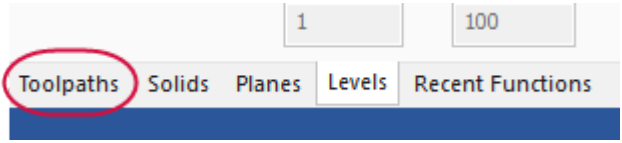
The Lathe Finish dialog box displays.

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



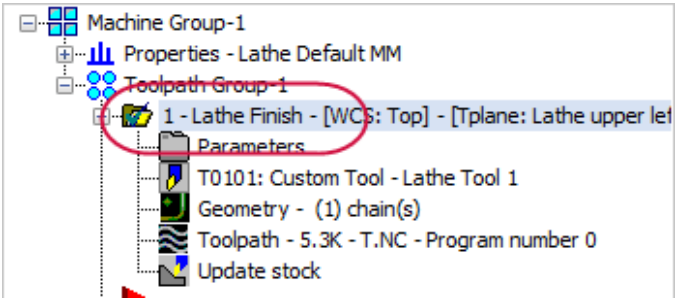
Mastercam creates the toolpath.

9. Select the **Toolpaths** tab.

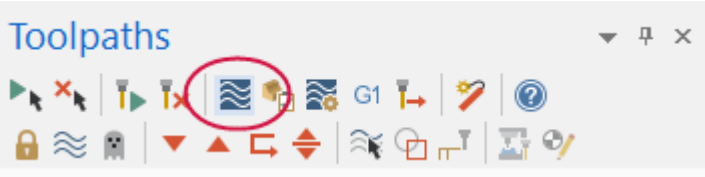


The Toolpaths Manager displays.

10. In the Toolpaths Manager, select the Lathe Finish toolpath.

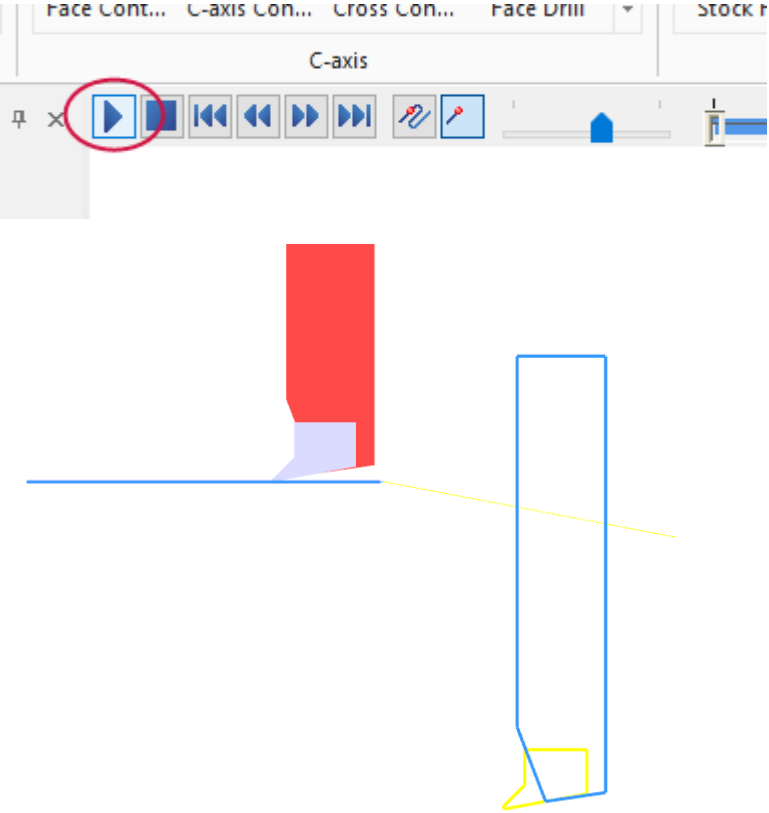


11. Select **Backplot selected operations**.

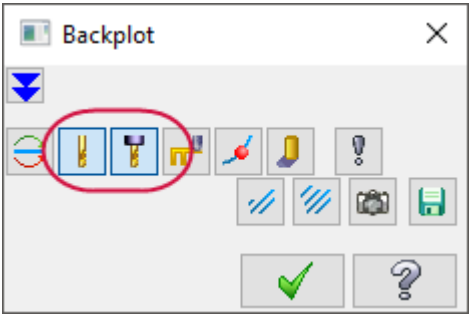


The Backplot dialog box displays.

12. Press **Play** to backplot the toolpath to ensure that it works as expected.



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



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

2

Graphical Orientation and Tool Settings

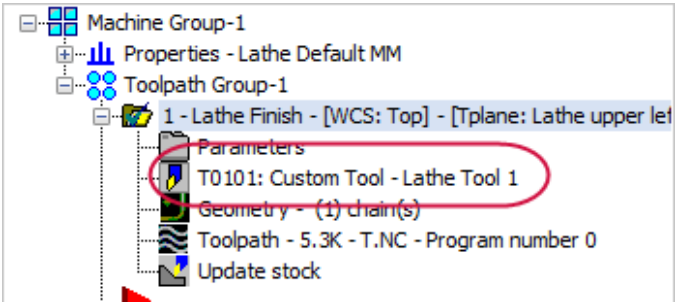
In this lesson, you experiment with some of the tool settings to see how they relate to the tool's location in the graphics window. You will see that you can draw the tool wherever you like, but that the tool's position affects the settings you must use, as well as the relative location of the toolpath.

Continue with the part file you completed in the previous lesson, or load the `CustomTool04` file included with this tutorial.

Exercise 1: Using Tool Orientation

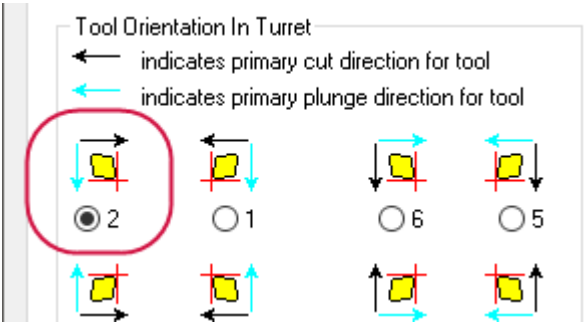
In the previous exercise, you used Backplot to test the tool to ensure that it moved correctly across the geometry. In this exercise, you change the orientation of the tool and then view it in Backplot.

1. In the Toolpaths Manager, select **Custom Tool**.



   The Define Tool dialog displays.

2. Change **Tool Orientation In Turret** to **2**.



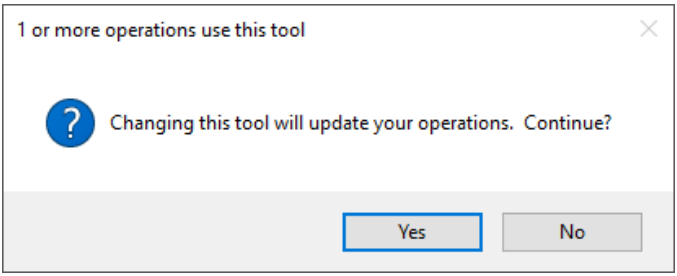
   Note: Make sure that you change the **Tool Orientation In Turret** setting and not the **Geometric Tool Orientation In TOP Plane** setting. Because the geometric orientation is based on how you drew the tool in the graphics window, you do not need to change it once it is set.

3. Select **Draw Tool**, and notice how the tool orientation has changed to match the selected option.

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

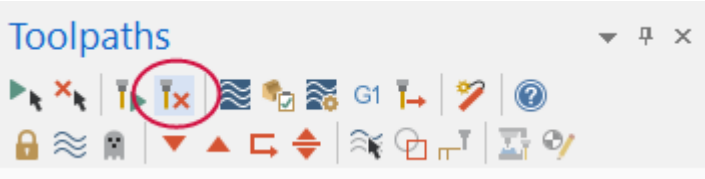
   The 1 or more operations use this tool dialog box displays.

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

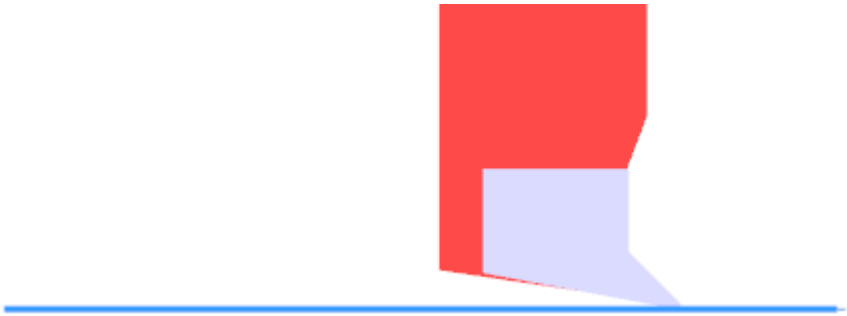


   Mastercam marks the toolpath as dirty.

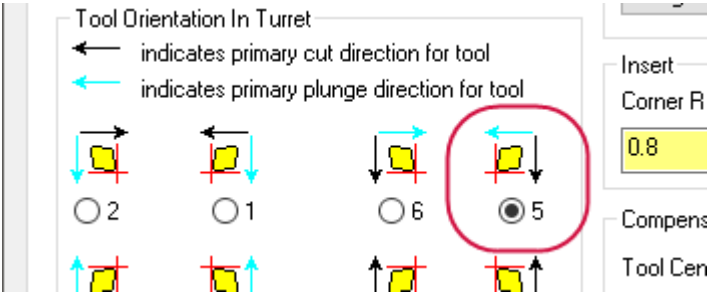
6. Select **Regenerate all dirty operations** in the Toolpaths Manager.



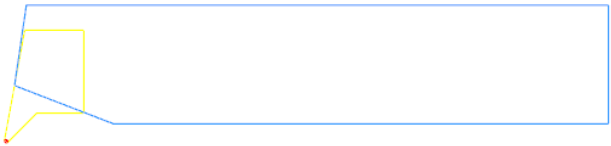
7. Select **Backplot selected operations**.
8. Press **Play** to backplot the operation, and notice how the tool is now oriented in the opposite direction.



9. Click **OK** in the Backplot dialog box.
10. Select the custom tool again in Toolpaths Manager, and change **Tool Orientation In Turret** to **5**.



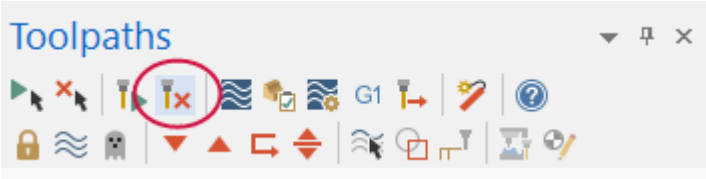
11. Click **Draw Tool**, and notice how the tool orientation has changed to match the selected option.



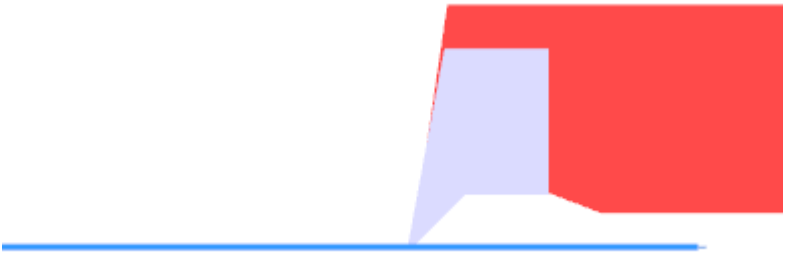
12. Press [**Enter**] to return to the dialog box, and click **OK**.

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

13. Click **Yes** to accept that the changes will update the operation.
14. Regenerate the operation.

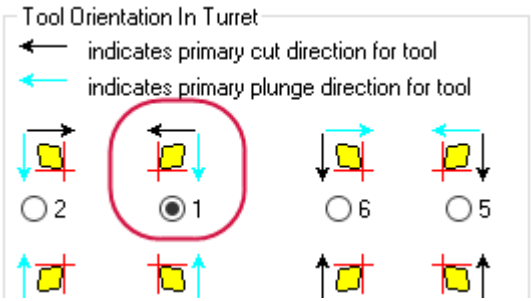


15. Backplot the operation, and notice how the tool's orientation has changed.



16. Click **OK** in Backplot dialog box.

17. Select the custom tool and change **Tool Orientation In Turret** back to **1**, and click **OK**.

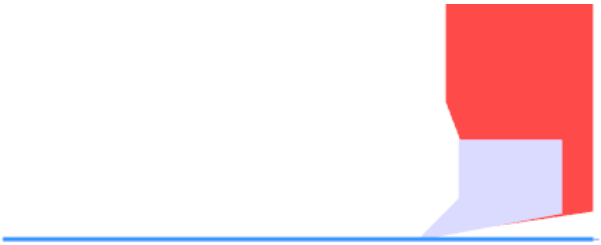


18. Click **Yes** to accept that the changes will update the operation.

    Mastercam marks the toolpath as dirty.

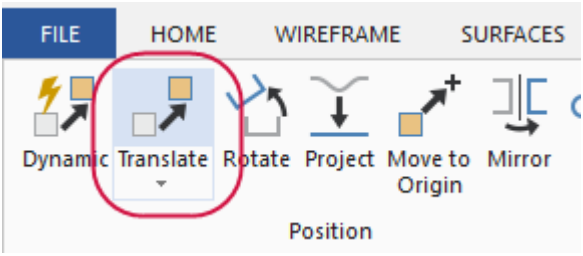
19. Regenerate the operation, and backplot.

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



Exercise 2: Tool Position in the Graphics Window

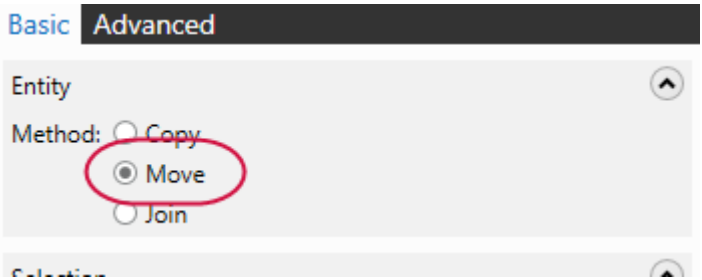
1. Select **Translate** from the **Transform** tab.



   The Translate function panel displays.

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

3. Set **Method** to **Move**.

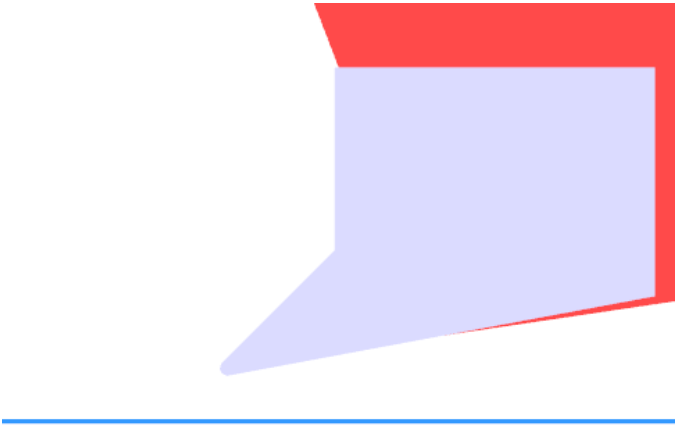


4. Set **Z** and **X** to **5.0**, and click **OK**.

5. Right-click in the graphics window and select **Clear Colors** to reset the tool's color values.
6. Press [**F9**] to display the axes.

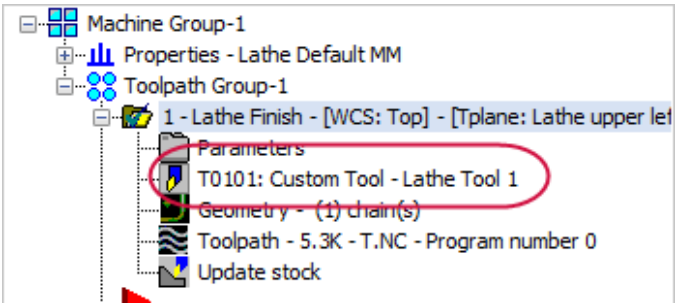
7. Zoom in on the origin to see how the insert's tip is now oriented.

8. Press [**F9**] again to hide the axes.
9. Backplot, and notice that the image of the tool is now above the toolpath.



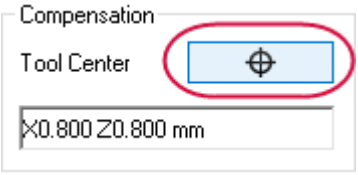
This is because you have moved the tool's centerpoint to X5.8, Y5.8, but Mastercam still thinks it's at X0.8, Y0.8.

10. Select **Custom Tool** in Toolpaths Manager.

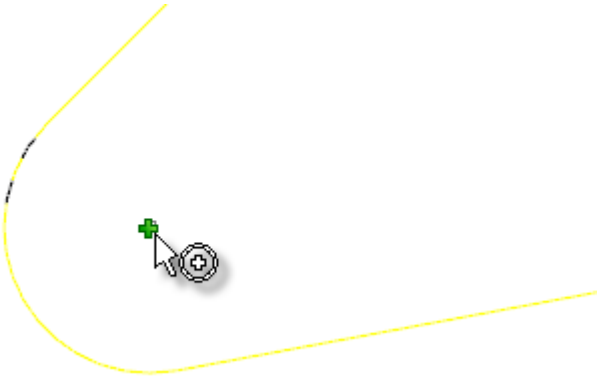


The Define Tool dialog box displays.

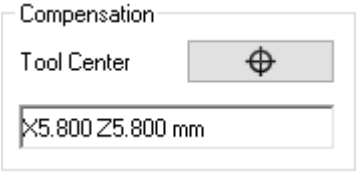
11. Click **Tool Center** to return to the graphics window.



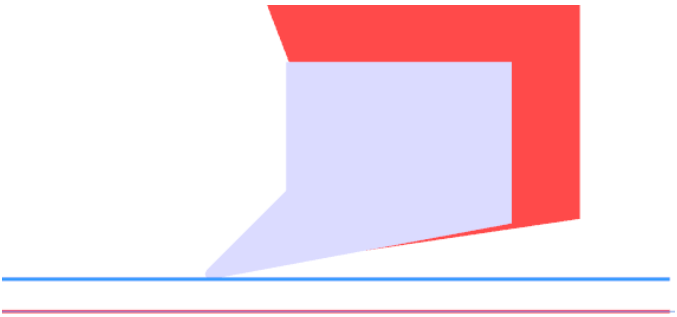
12. Select the center of the arc to reset the tool center to its new location.



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



14. Click **OK** in the Define Tool dialog, click **Yes** in the message that displays, and regenerate the toolpath.
15. Backplot the toolpath.
16. 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.

17. Click **OK** in the Backplot dialog box.

Exercise 3: Restore the Tool to its Original Settings

1. Select **Translate** on the **Transform** tab.

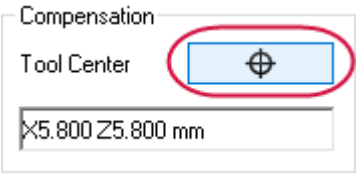
   The Translate function panel displays.

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

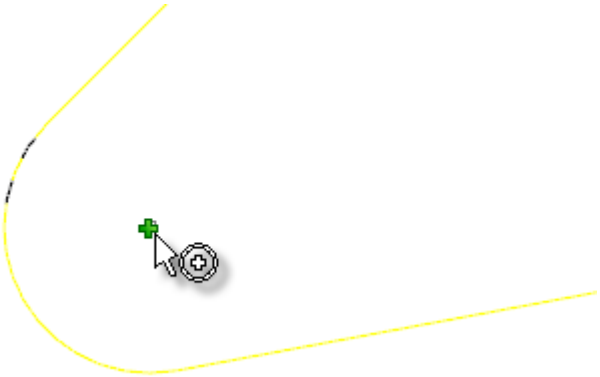
3. Set **Method** to **Move**.
4. Set **Z** and **X** to **-5.0**, and click **OK**.

5. Select the custom tool in the Toolpaths Manager.

6. Select **Tool Center**.



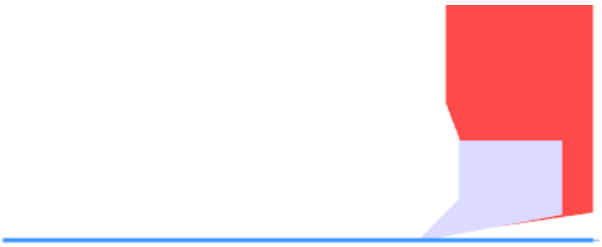
7. Select the center of the arc shown below to reset the tool's centerpoint.



The Tool Center should be reset back to **X0.800 Z0.800 mm**.

8. Click **OK** in the Define Tool dialog box, and then click **Yes** in the message box that displays.
9. Select **Clear Colors**, regenerate the operation, and backplot.

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



3

Creating a Horizontally Oriented Tool

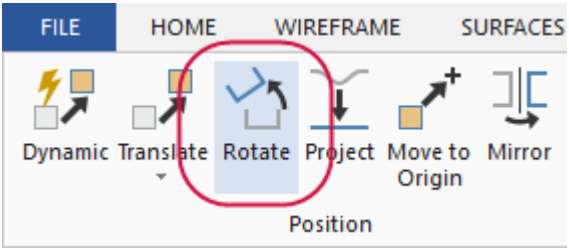
In this lesson, you create a horizontally oriented tool that will be used in the machine's upper turret to cut material on the right spindle. To do this, you complete the following procedures:

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

Exercise 1: Rotating 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.

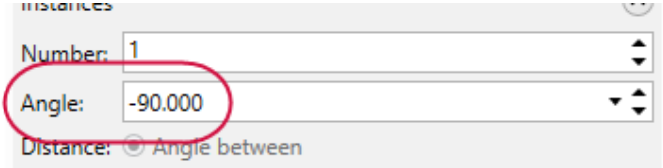
1. Open `CustomTool02`, either the one you created or the one included with this tutorial.
2. If necessary, press [**F9**] to display the axes in the graphics window.
3. Select **Rotate** from the **Transform** tab.



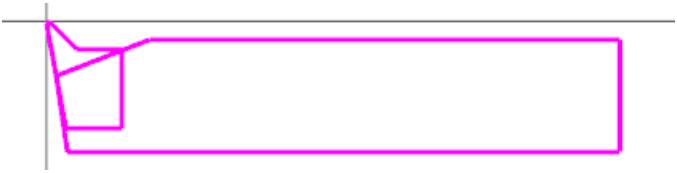
The Rotate function panel displays.

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

5. Set **Method** to **Move**.
6. Set **Angle** to **-90** degrees.

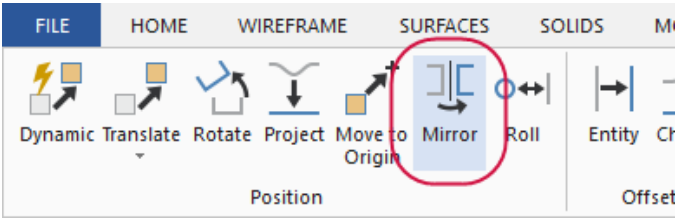


7. Click **OK**. The tool geometry rotates into the position shown below.



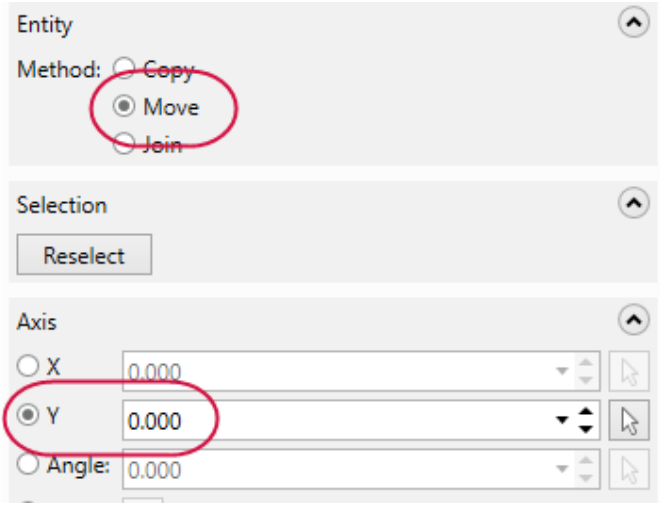
Note: If you prefer, you can use Dynamic transformation's on-screen controls to rotate the part. For more information, please refer to the Help.

8. Choose **Mirror** from the **Transform** tab.

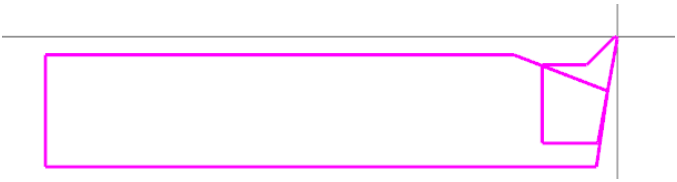


The Mirror function panel displays.

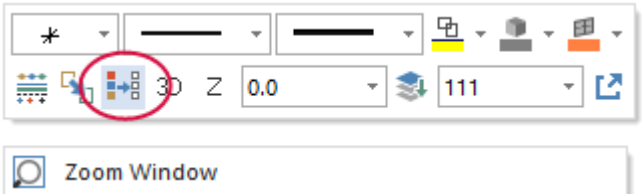
9. Select all of the tool geometry, and press [**Enter**].
10. Set **Method** to **Move** and select the **Y Axis**.



11. Click **OK**. The tool geometry should now be in the position shown below.

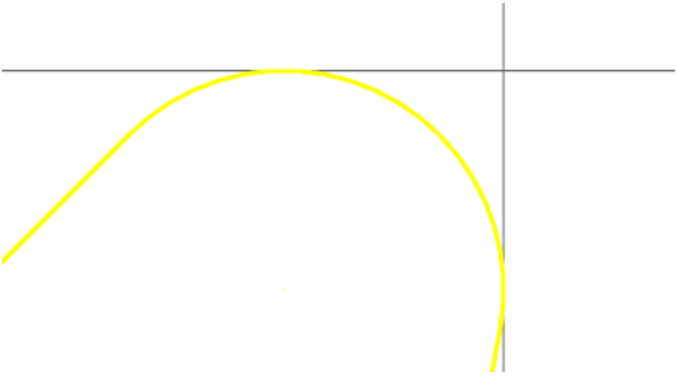


12. Right-click in the graphics window and select **Clear Colors**.



Mastercam restores the geometry to its original colors.

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

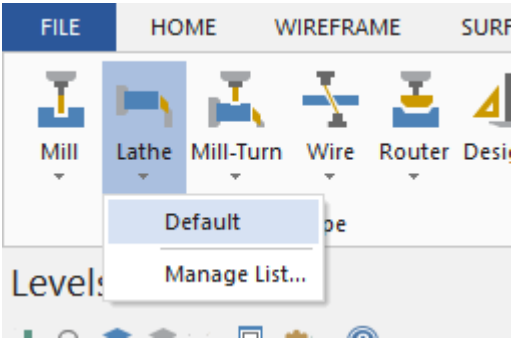


14. Press **[Alt + F1]** to fit the tool to the graphics window.
15. Save the file as `CustomTool05_xxx`, where `xxx` is your initials.

Exercise 2: Setting the Orientation and Position

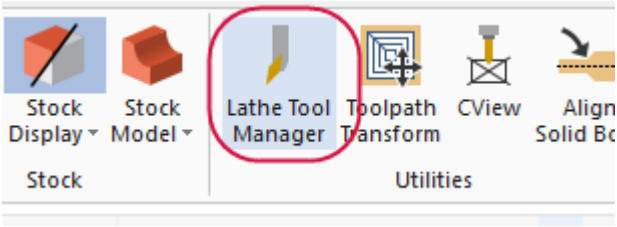
After you draw the tool's geometry, you must set the tool parameters that tell Mastercam how to interpret and use the tool geometry. These settings are different for this horizontal tool than for the vertically oriented tool you created earlier in the tutorial.

1. Select **Lathe**, **Default** from the **Machine** tab.



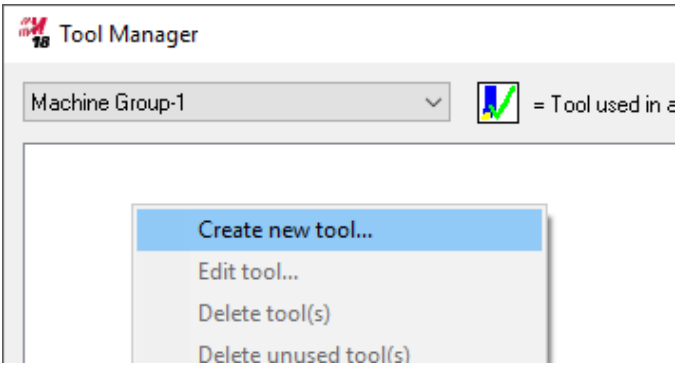
   Mastercam starts a new machine group in Toolpaths Manager.

2. Select **Lathe Tool Manager** from the **Lathe Turning** contextual tab.



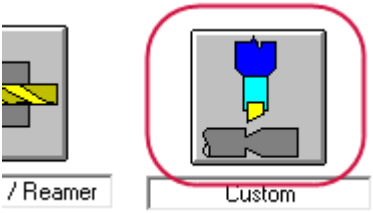
   The Tool Manager dialog box displays.

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



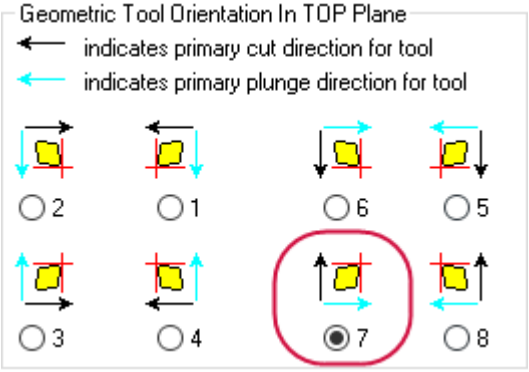
The Define Tool dialog box opens.

4. Select the **Custom** tool type.



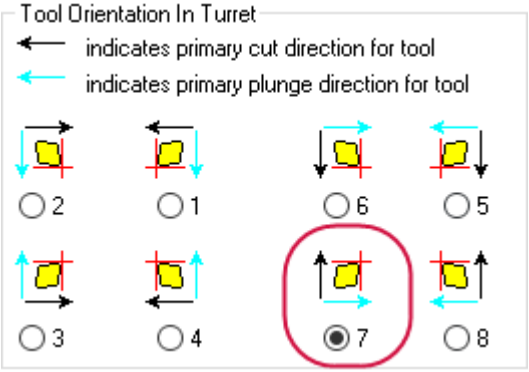
The dialog box automatically switches to the **Geometry** tab.

5. Ensure that for **Geometric Tool Orientation In TOP Plane**, orientation **7** is selected.



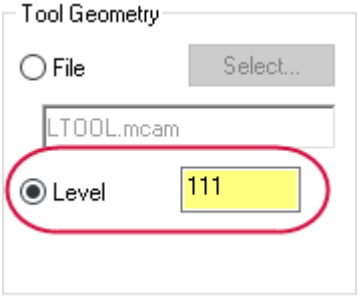
This is the orientation of the tool's geometry as it is viewed in the Top plane. Note that choices **3** and **7** have the same geometric orientation, but the cut and plunge directions are different.

6. Ensure that for **Tool Orientation In Turret**, orientation **7** is selected.



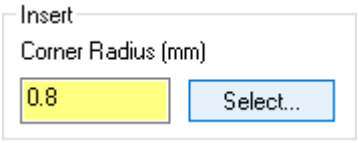
This is the orientation of the tool as it will be mounted in the turret. Note that, again, choices **3** and **7** have the same geometric orientation, but the cut and plunge directions are different.

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



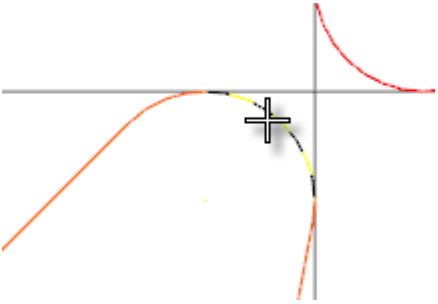
This is the level on which you drew the tool geometry.

8. Click **Select** next to **Corner Radius (mm)**.



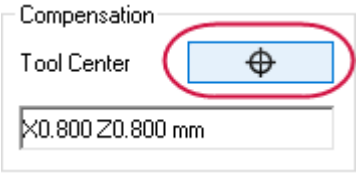
The dialog box minimizes so that you can access the graphics window.

9. Zoom in on the insert's tip, and select the tip's arc. You may see arcs that display the previous selected Corner Radius.



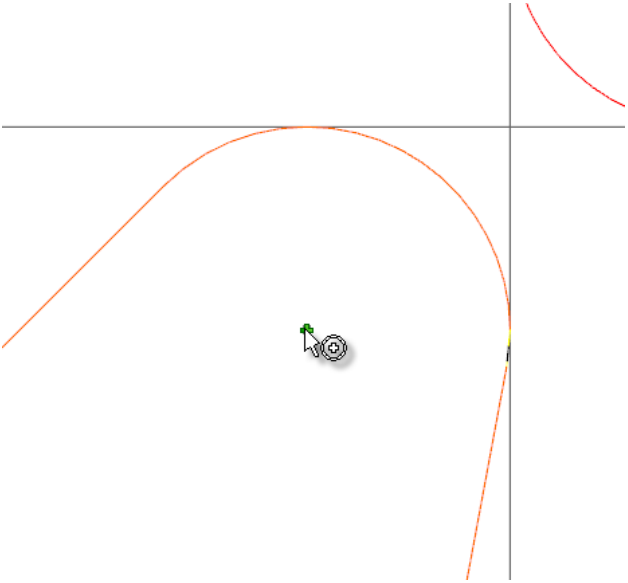
Mastercam returns you to the Define Tool dialog box.

10. Select **Tool Center**.



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

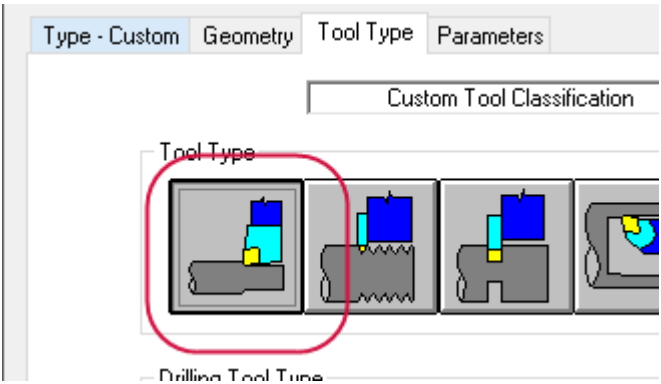
11. Zoom in on the insert's tip, and select the tip's centerpoint. You may see arcs that display the previous selected Tool Center.



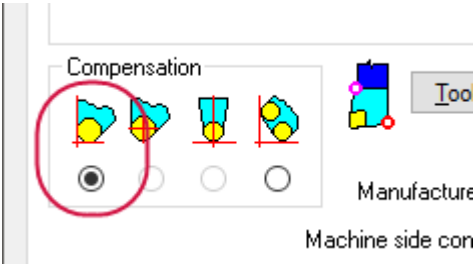
Mastercam returns you to the Define Tool dialog box, with **X-0.800 Z-0.800 mm** entered as the tool's center.

Exercise 3: Setting Tool Type and Compensation

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



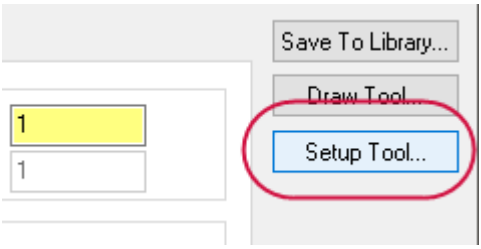
2. Click the **Parameters** tab, and ensure that the first **Compensation** option is selected.



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

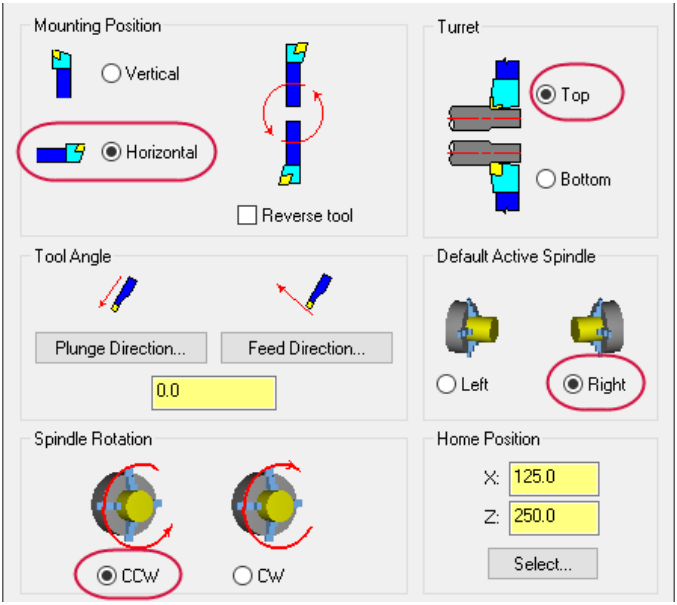
Exercise 4: Setting Up the Tool

1. Select **Setup Tool**.



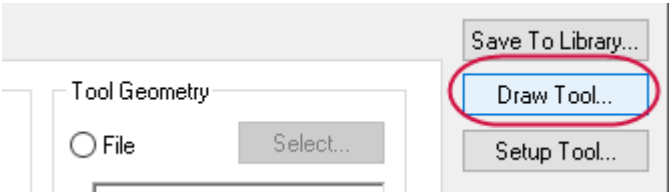
   The Lathe Tool Setup dialog box displays.

2. Select the following:
   a. **Mounting Position** to **Horizontal**.
   b. **Turret** to **Top**.
   c. **Default Active Spindle** to **Right**.
   d. **Spindle Rotation** to **CCW**.

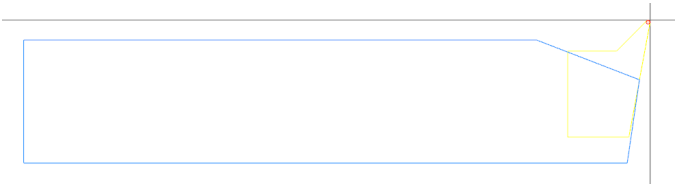


Your settings, except possibly **Home Position**, should match the above image.

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



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

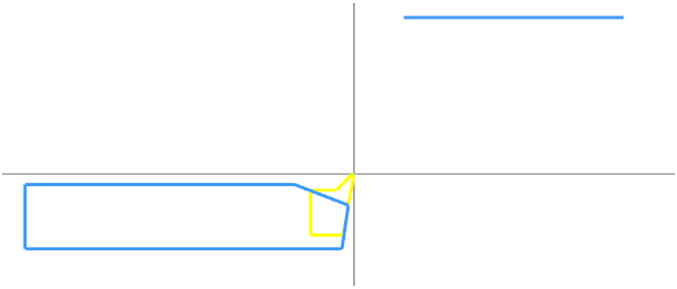


5. Press [**Enter**] to return to the dialog box.
6. Click **OK** in the Define Tool and Tool Manager dialog boxes.

   The tool is now ready to test.
7. Save the file as `CustomTool06_xxx`, where `xxx` is your initials.

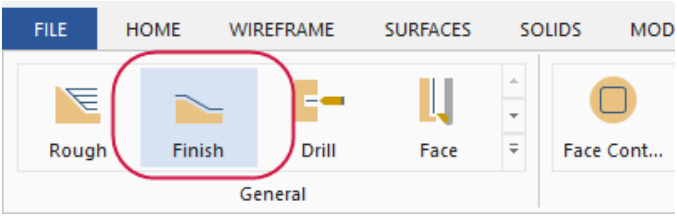
Exercise 5: Testing the Tool in a Toolpath

1. 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. Select **Finish** from the **General** gallery on the **Lathe Turning** contextual tab.

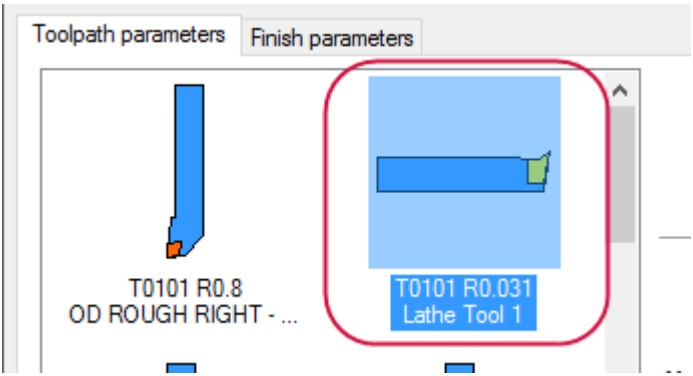


The Chaining dialog box displays.

4. Chain the line as shown below, and click **OK**.

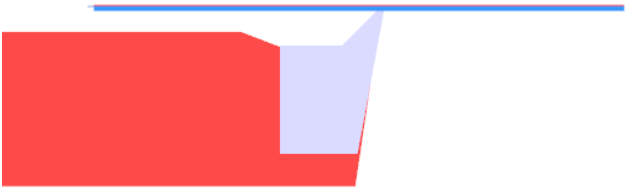
The Lathe Finish dialog box opens.

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



Mastercam creates the toolpath.

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



7. 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. 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 2018 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 (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*—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, or email training@mastercam.com.

- *Online Communities*—You can find a wealth of information 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 knowledgebase at kb.mastercam.com.

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.
PLEASE REFER TO MASTERCAM.COM/SUPPORT
FOR THE LATEST DOWNLOADS.

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

Mastercam
Be Dynamic.