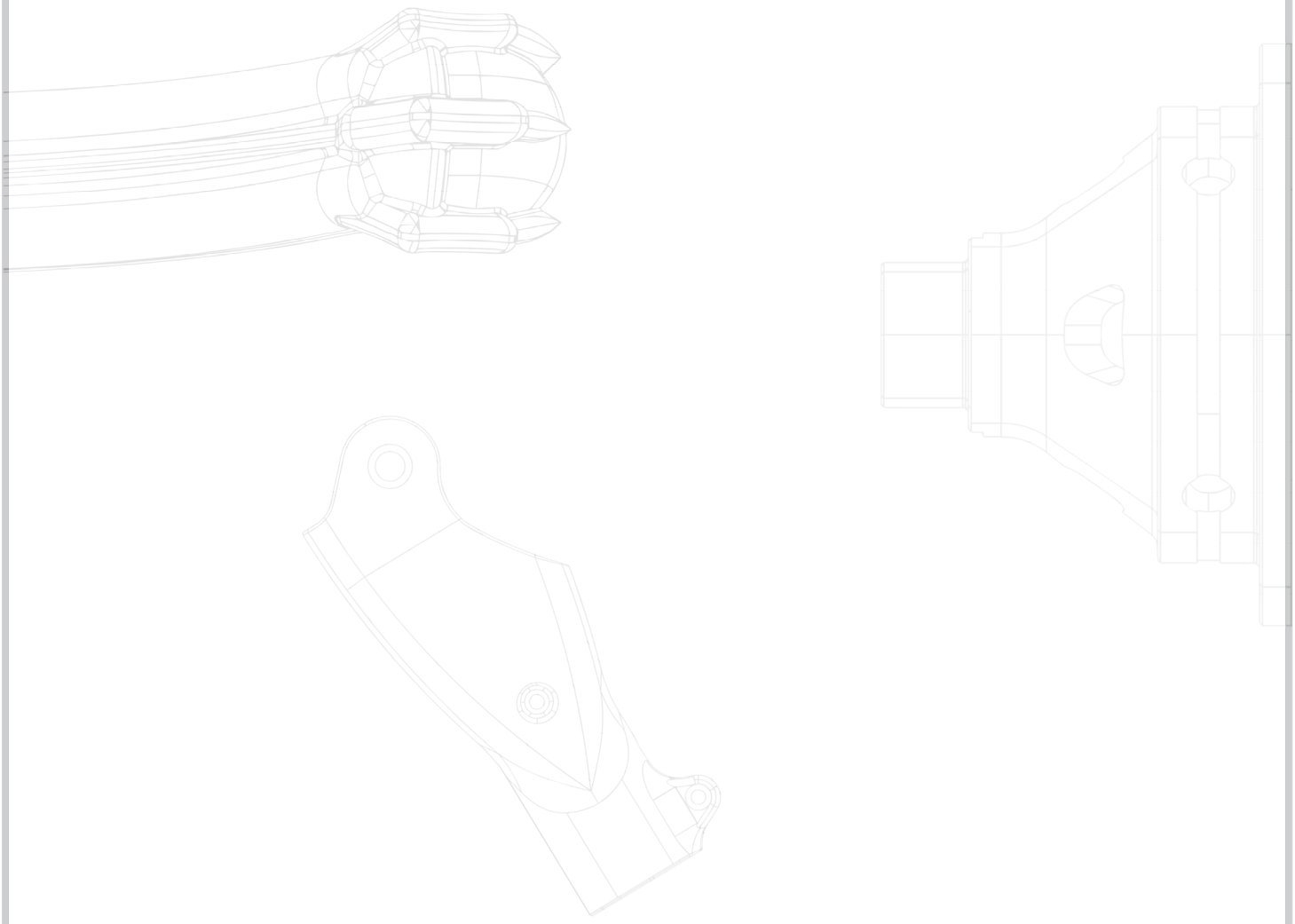
MASTERCAM MULTIAXIS TUTORIAL

June 2018



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

MASTERCAM MULTIAXIS TUTORIAL

June 2018
© 2018 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2019

Terms of Use

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

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

Be sure you have the latest information!

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

TABLE OF CONTENTS

Introduction **5**

Mastercam Multiaxis Tutorial Requirements 5

General Tutorial Requirements 5

Basic Machine Overview **7**

Table/Table Machine 8

Head/Table Machine 9

Head/Head Machine 10

Multiaxis Toolpath Controls Overview **11**

Cut Pattern 11

Tool Axis Control 12

Tool Tip Control 13

Mastercam Interface and Workflow **17**

Cut Pattern Page 19

Tool Axis Control Page 19

Collision Control Page 20

Multiaxis Curve Toolpath **21**

Exercise 1: Getting Started with Toolpath Creation 21

Exercise 2: Creating a Multiaxis Curve Toolpath 22

Exercise 3: Backplotting the Operation 27

Exercise 4: Running Machine Simulation 29

Tool Axis Control Options **31**

Exercise 1: Copying and Editing an Operation 31

Exercise 2: Copying and Editing the Second Operation 35

Multiaxis Drill Toolpath **39**

Exercise 1: Creating a Multiaxis Drill Toolpath 39

Exercise 2: Adding a Second Multiaxis Drill Operation 45

Exercise 3: Backplotting the operations 48

Conclusion **51**

Mastercam Resources 51

Contact Us 51

INTRODUCTION

This tutorial introduces the concepts of multiaxis machining, beginning with the machine architecture and ending with multiaxis toolpath creation. Multiaxis toolpaths are basically the familiar contour, pocket, and surface toolpaths in X, Y, and Z, with rotational motion added in A, B, and C. The available axes vary based on your particular machine setup.

The workflow is consistent regardless of the multiaxis toolpath selected. Mastercam's Multiaxis interface follows a uniform structure through the toolpathing process. Select the toolpath family, select a toolpath type, progress from top to bottom through the tree-style interface, enter parameters on the necessary pages, and generate the toolpath. Additional tools such as Mastercam Simulator and Machine Simulation allow you to review your toolpath before cutting begins on the machine.

By completing this tutorial, you gain a general understanding of the multiaxis process. The information contained in these pages gives you the knowledge and confidence to work with Mastercam's multiaxis toolpaths.

Goals

- Understand the basic architecture of a multiaxis machine
- Review the controls of a multiaxis toolpath: cut pattern, tool axis control, and tool tip control
- Follow the workflow of Mastercam's Multiaxis toolpath interface
- Create and modify a Multiaxis Curve toolpath
- Create and modify a Multiaxis Drill toolpath

Mastercam Multiaxis Tutorial Requirements

- Mastercam 2019 with Mill or Mastercam Router
- Curve 5-axis / Drill 5-axis Add-In (included in the Multiaxis Add-In)

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: 1.5 hours

General Tutorial Requirements

All Mastercam 2019 tutorials have the following general requirements:

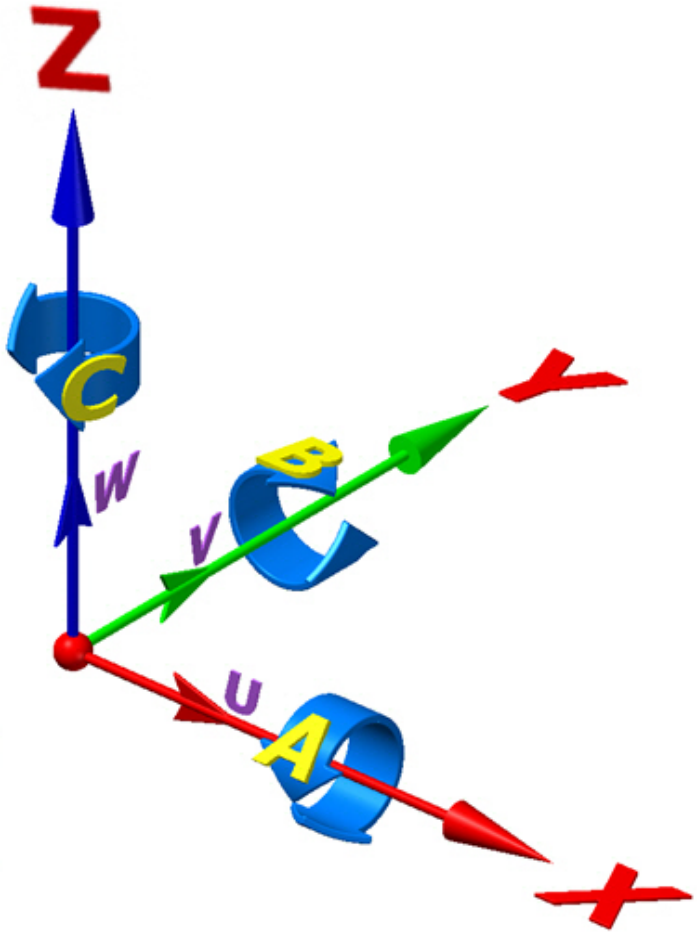
- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.

- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be access from the Mastercam 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

CHAPTER 1

BASIC MACHINE OVERVIEW

With a multiaxis machine, instead of a straightforward machine with XYZ axes capabilities, you have a machine that can handle five or more axes of motion. Typically, the axes are defined as XYZ/ABC/UVW as shown below.



The axes may be in a different orientation or have different relationships. Your machine now has a much greater range of motion than a standard 3-axis vertical or horizontal machine.

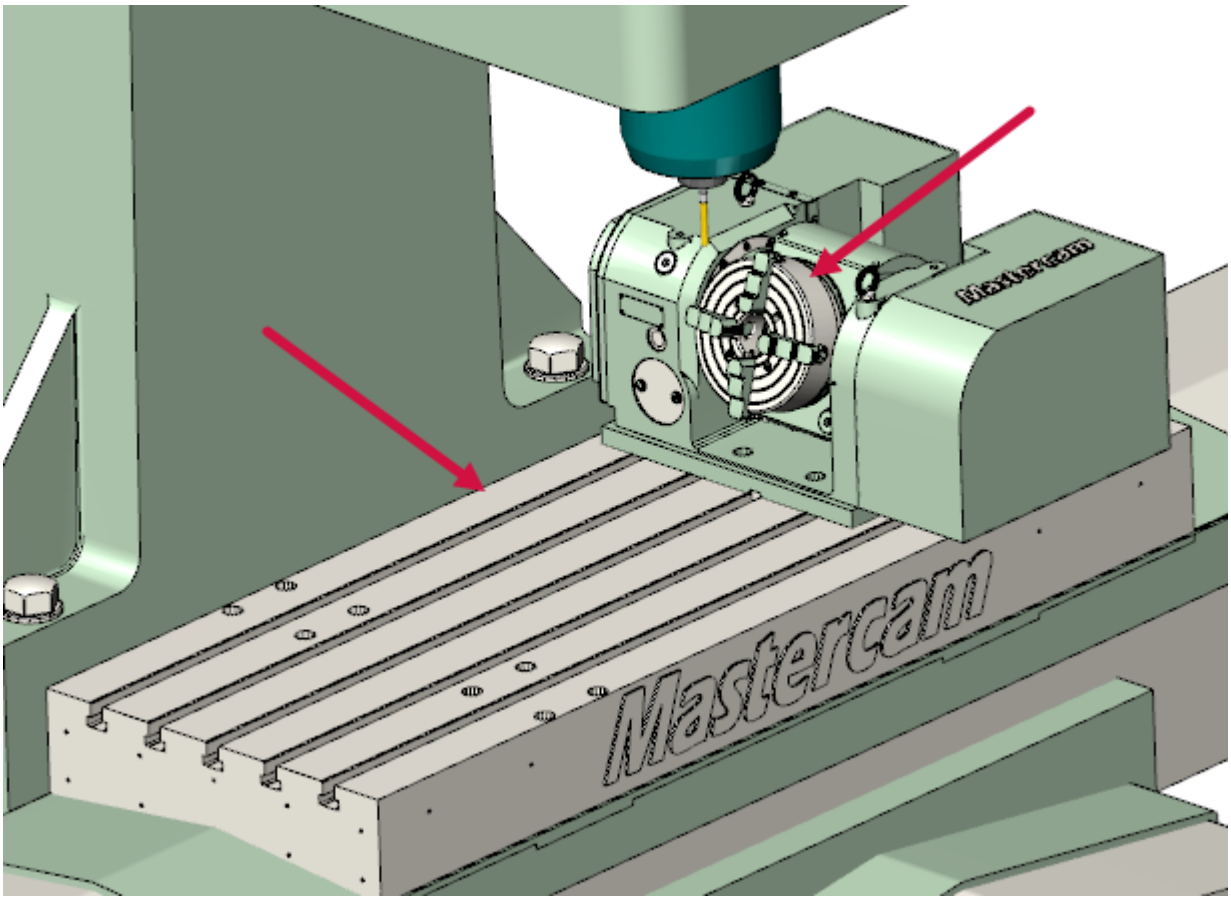
The added motion of a multiaxis machine enhances the capabilities of your shop, as well as expands the variety of parts that you can machine. Nearly every machine is unique in its axes combination, orientation, travel and rotation limits, and controller.

Common terms for multiaxis machines are **Table/Table**, **Head/Table**, and **Head/Head**. These terms describe the relationship of the rotary components on the machine. The configuration of the rotaries determines the zero location of the machine and how you need to locate the part within Mastercam. The part must be located in Mastercam where it will be sitting on the machine. The only exception is for a Head/Head arrangement.

CAUTION: Improper location of your geometry will cause the part to be cut incorrectly.

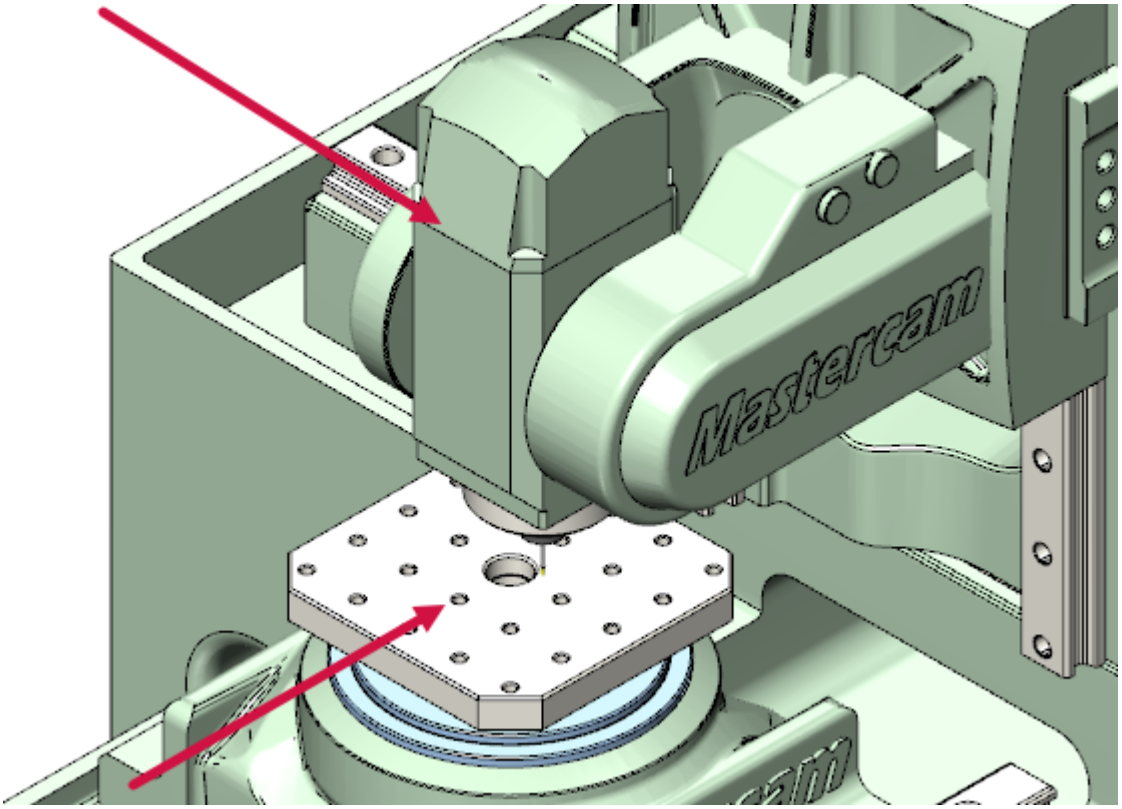
Table/Table Machine

A **Table/Table** machine has both rotary axes connected to the table. Typically this is a trunnion carrying a rotary table. The machine axes zero position is located at the intersection of rotary axes. Your part is located in Mastercam relative to the machine zero position. The rotary components are pointed out in the following image, which is a **Table/Table** configuration with machine zero.



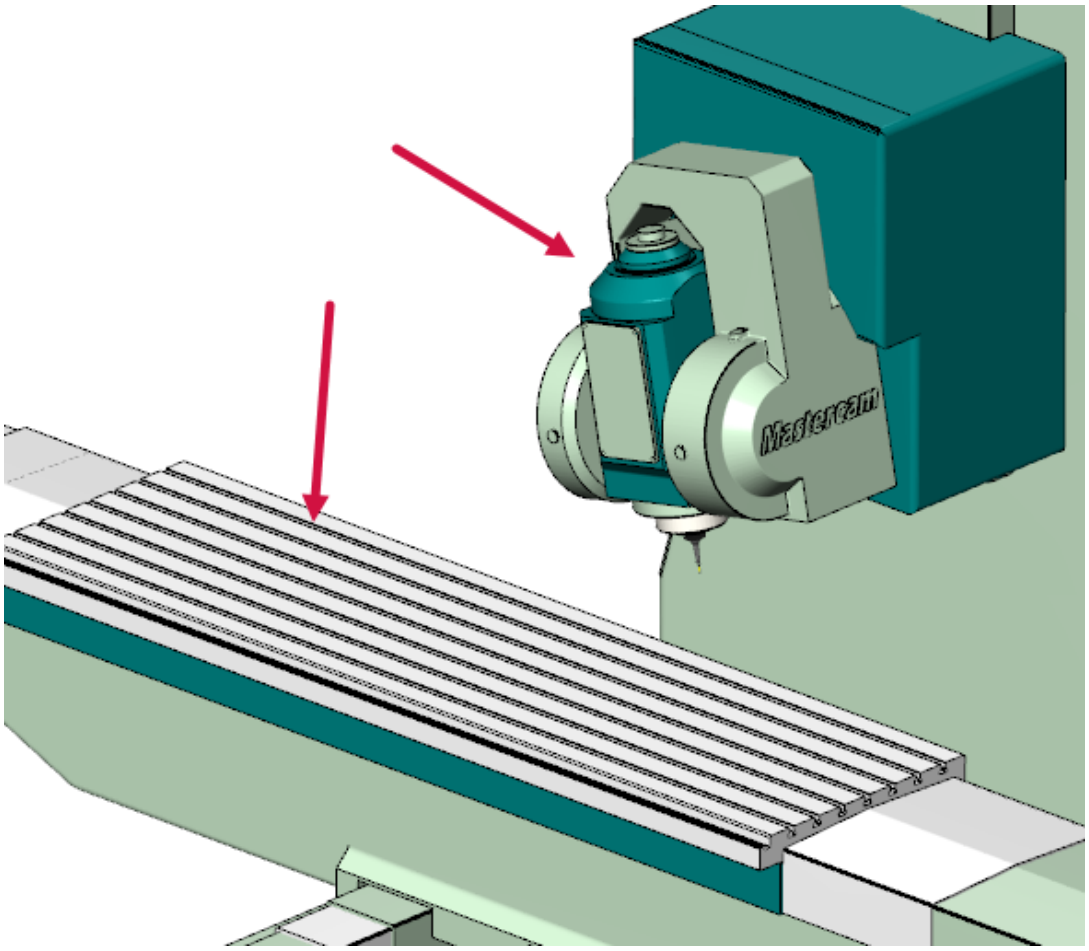
Head/Table Machine

A **Head/Table** machine has one rotary on the table and the other on the spindle. They operate independently of each other. The table could be comprised of a trunnion or simply a rotary attached to the table. The machine axis zero is located at the intersection of the rotary axes. Your part is located in Mastercam relative to the machine zero position, as in the **Table/Table** configuration. The rotary components are pointed out in the following image, a **Head/Table** configuration with machine zero.



Head/Head Machine

A **Head/Head** machine has both rotary axes connected and attached to the spindle. The machine axis zero is typically located on the face of the spindle. The origin is shown on the table as this is where the spindle face lies when all axes are set to zero. The rotary components are shown in the following image, a **Head/Head** configuration with machine zero.



These are the basic arrangements of multiaxis machines. Be aware that multiaxis machines may be as simple as a single rotary for 4-axis work, or as complex as having five axes and a nutating head. No matter how the machine is configured, the inputs for a good toolpath follow the same pattern from machine to machine. Continue on to the next chapter to learn about the necessary controls for creating an efficient multiaxis toolpath.

CHAPTER 2

MULTIAXIS TOOLPATH CONTROLS OVERVIEW

Three controls separate multiaxis toolpaths from the typical 2- and 3-axis toolpaths:

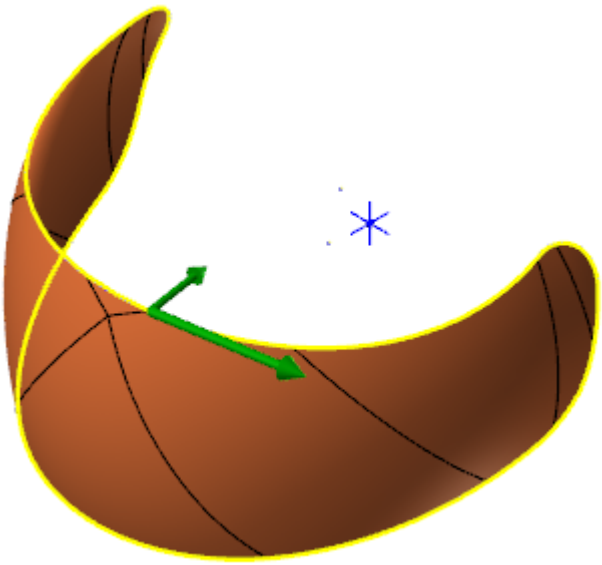
- Cut Pattern
- Tool Axis Control
- Tool Tip Control

Effective use of these controls is the deciding factor in your ability to create efficient toolpaths for your particular machine. This tutorial starts you on the path to becoming a more effective programmer. However, there is no substitute for experience and knowing your machine's capabilities.

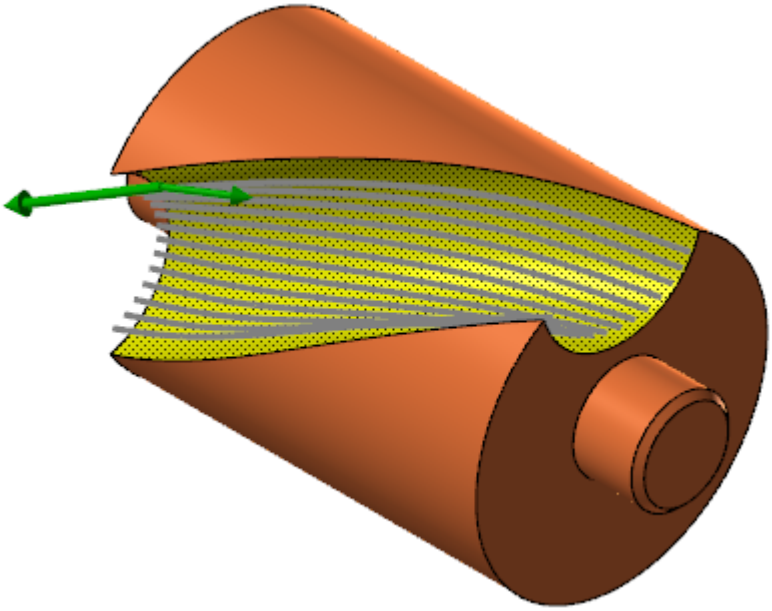
Cut Pattern

What do you want the tool to follow? The answer is the **Cut Pattern**. Selecting a toolpath family is the initial phase of establishing the cut pattern. Different toolpath families vary the type of geometry allowed for the cut pattern. Geometry selection can range from a contour or chain, to a surface edge or edges, and on through single or multiple surfaces or solids. Below are two examples of cut pattern selections.

3D Curves



Surface



In addition to the geometry selected for the toolpath, cut pattern includes such parameters as:

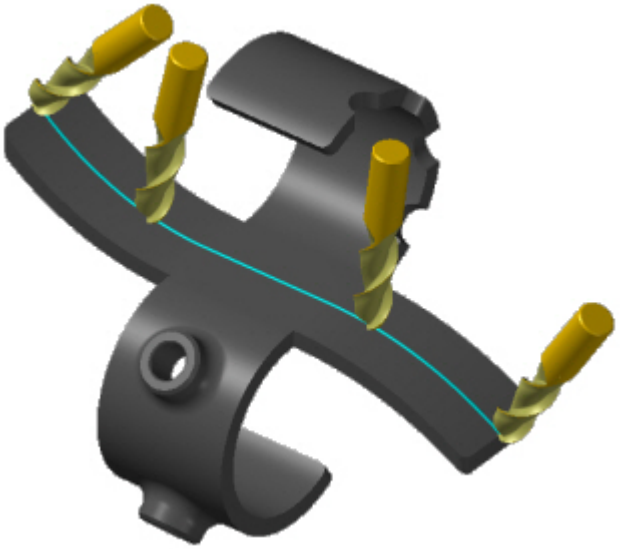
- **Cutting method**: Zigzag, One way, Spiral
- **Compensation**: Type and Direction
- **Stock to leave**: Drive surfaces, Walls
- **Stepover**: Across, Along, Increment

Subsequent chapters cover cut pattern options in more detail.

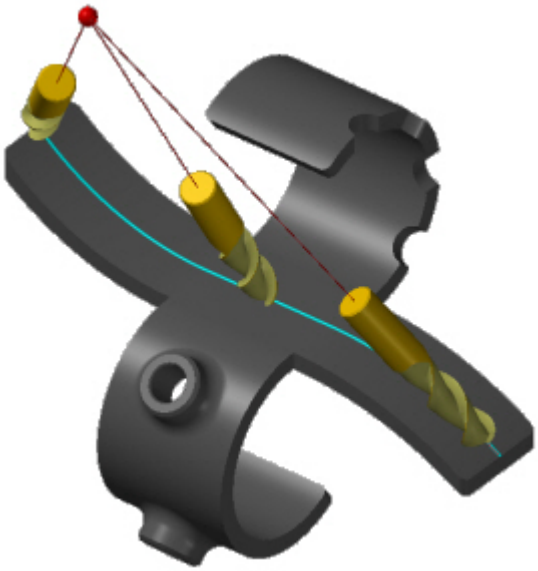
Tool Axis Control

How do you want the tool axis to behave as it follows the cut pattern? **Tool axis control** answers this question. Multiaxis toolpaths include numerous options on how to manipulate the tool axis. The options allow you to control the part of the tool in contact with the material, and the amount of tool movement the toolpath generates, as well as being able to set the number of axes of output for the posted toolpath. Below are two examples of tool axis control.

Surface



From point



Toolpath family and toolpath type determine which tool axis controls you can access. Some examples of the controls available include:

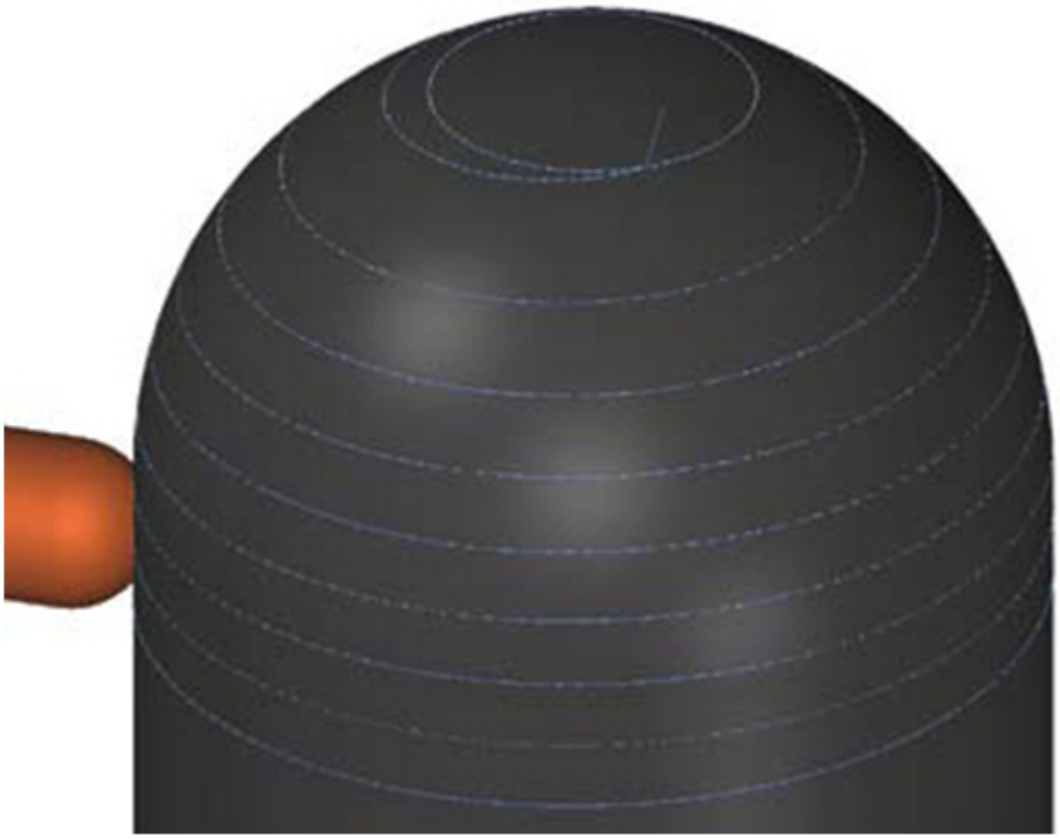
- Lines, surface, from point, chain
- Fixed angle to axis, be tilted relative to contact point

- Rotated around axis, from point
- Parallel to line, surface, plane

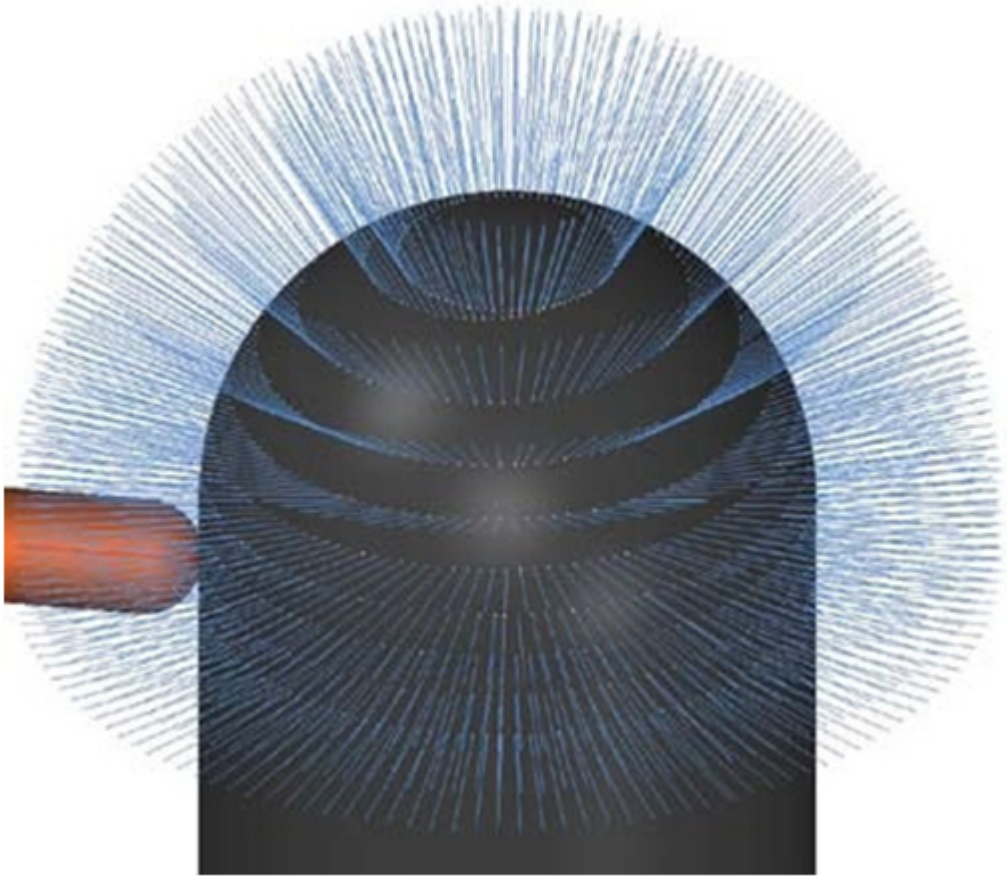
Tool Tip Control

What controls the depth of the tool along the tool axis? **Tool tip control** handles this function. Compensation surfaces are included in tool tip control. Applying tool tip control is a three step process:

1. Tool positions are generated along the selected cut pattern.



2. Tool axis vectors are created at each position based on the tool axis control settings.



3. Depth along the tool axis is applied based on the tip compensation method.

Note: Use the method depicted above when you have very complicated, or less than perfect, part geometry. The clean core geometry is used to generate the cut pattern and tool axis control vectors. Tip control is then used to cut the outer surfaces with much cleaner motion.

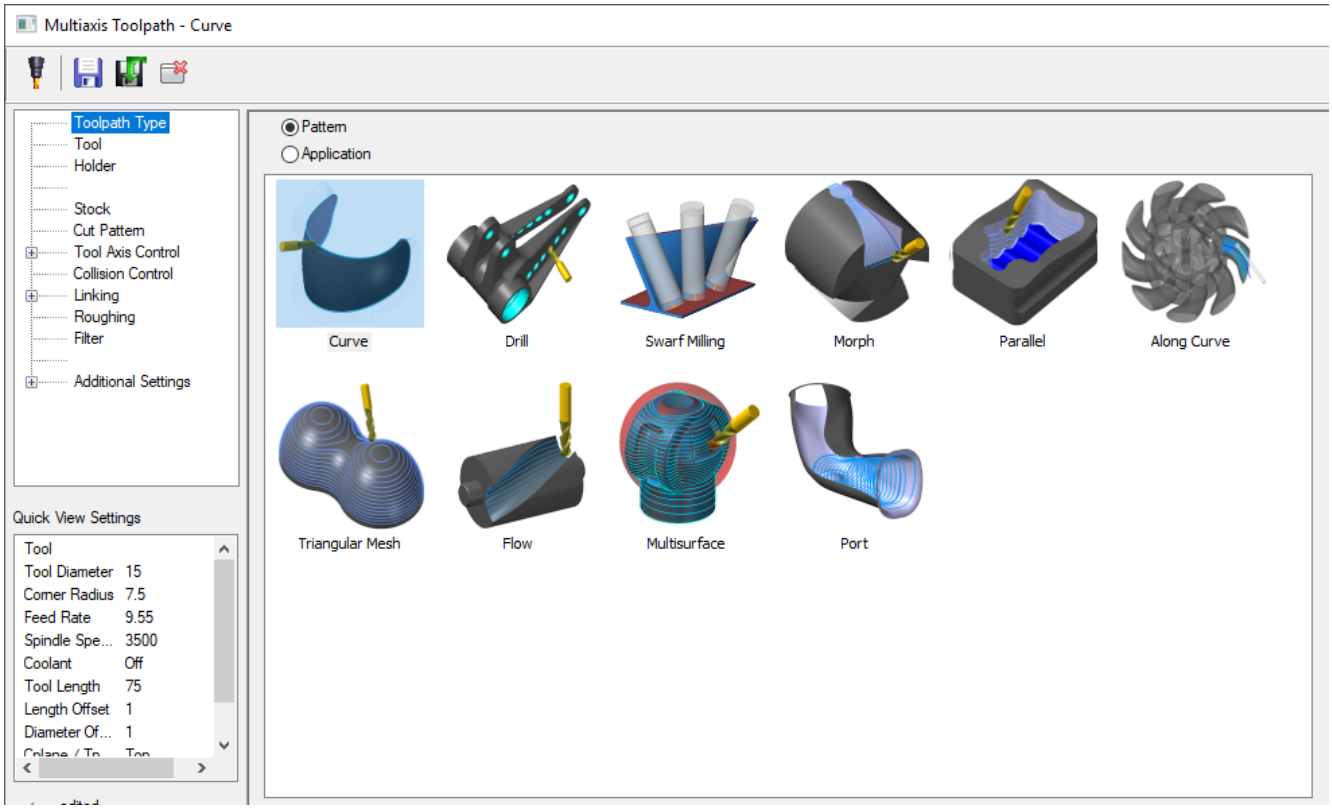
The three controls described previously form the core of all your multiaxis toolpaths. Additional refinement is added through collision control, linking, tool selection, and more. Building confidence with these controls will greatly ease and enhance your ability to generate efficient multiaxis toolpaths. Continue on to Lesson 3 to see how these controls are an integral part of the multiaxis workflow within Mastercam.

CHAPTER 3

MASTERCAM INTERFACE AND WORKFLOW

Mastercam's Multiaxis toolpath interface follows the familiar tree-style layout. You progress through the pages of the tree, make the necessary selections, enter appropriate parameters, and generate your toolpath. Viewing the process in simplified terms helps to lessen the complexity of creating a multiaxis toolpath.

Begin the toolpath creation process by selecting an appropriate machine definition. This applies to all toolpath types in Mastercam. After selecting a machine definition that supports multiaxis movement, choose a Multiaxis toolpath from the **Multiaxis** gallery on the **Toolpaths** contextual tab. The following image shows the starting point for a Curve toolpath.

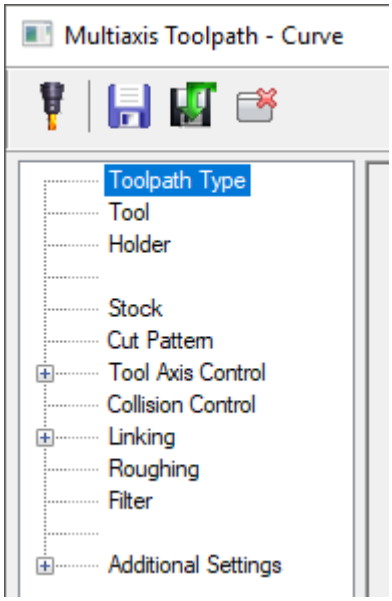


Multiaxis toolpaths are divided into two toolpath families, **Pattern** and **Application**. Each toolpath family contains different toolpath types.

Pattern toolpaths are general toolpaths that can be used in a variety of ways to manufacture your part. These toolpaths are very flexible and provide a wide range of machining options. **Application** toolpaths are designed to solve particular machining problems while automating some of the processes. These toolpaths include machining options and strategies specific to the application.

Selecting the most efficient type will come from experience and usage. For now, you focus on the general workflow rather than specific applications.

The tree structure provides the basic roadmap to creating a toolpath. While you can select the pages as needed, the most efficient route is from top to bottom. Click the plus sign before any page to expose additional pages for input.

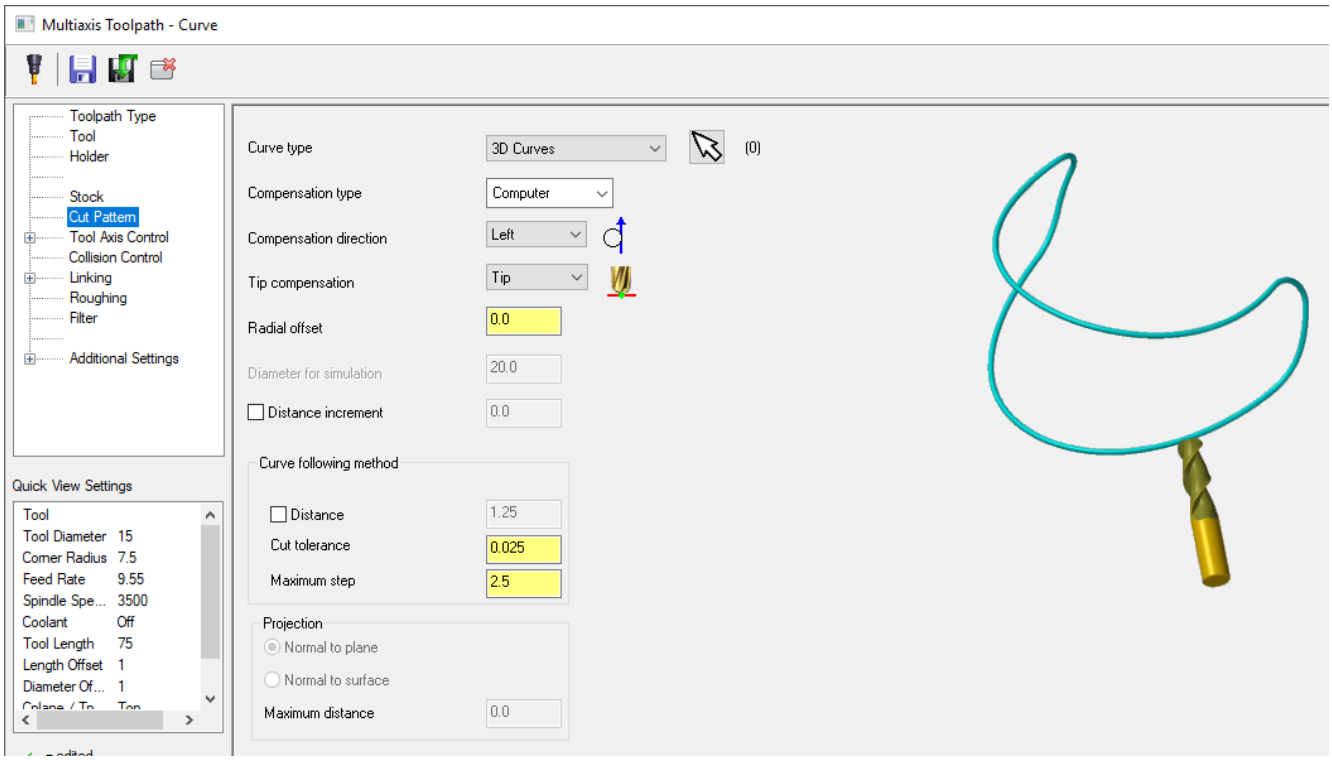


The pages within the tree structure vary with the toolpath type. The procedure remains the same regardless of what the tree looks like. If you can answer the following questions, you are well on your way to generating a multiaxis toolpath:

- What do I want the tool to follow?
- How do I want my tool axis to behave?
- What controls the depth along the tool axis?

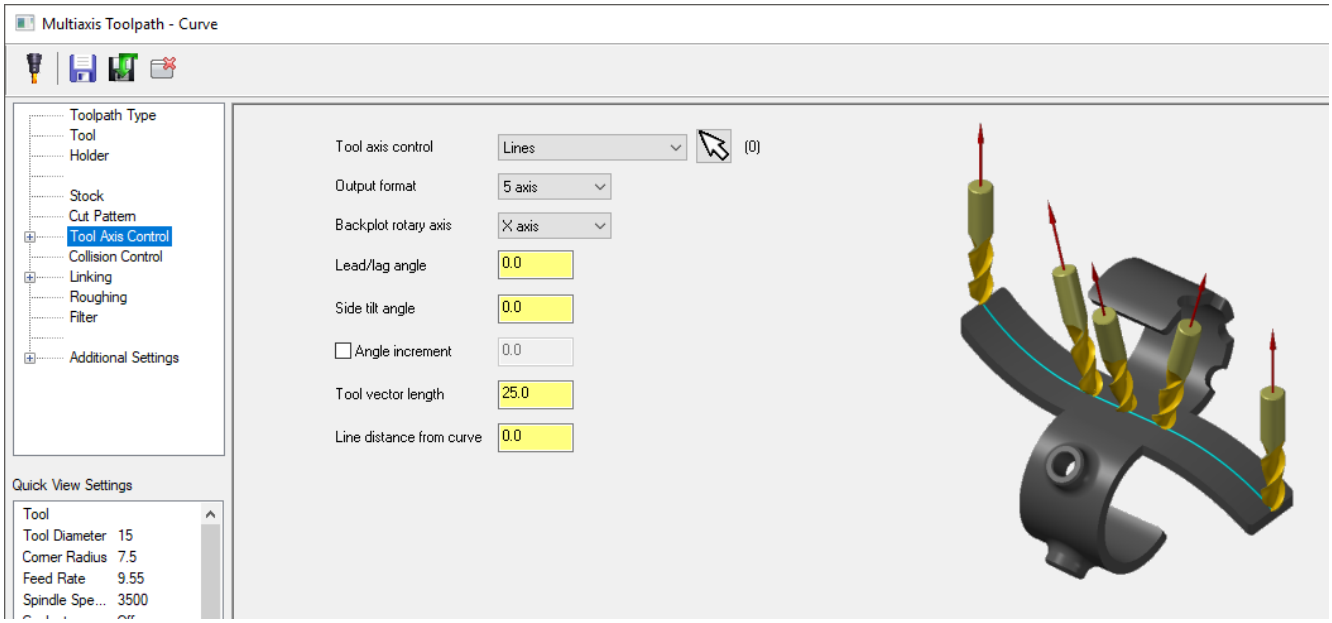
Cut Pattern Page

The controls discussed in the previous chapter are pivotal for multiaxis work. The **Cut Pattern** page is where you select your drive geometry, set compensation parameters, and set various other options. Below is the **Cut Pattern** page for a Curve toolpath.



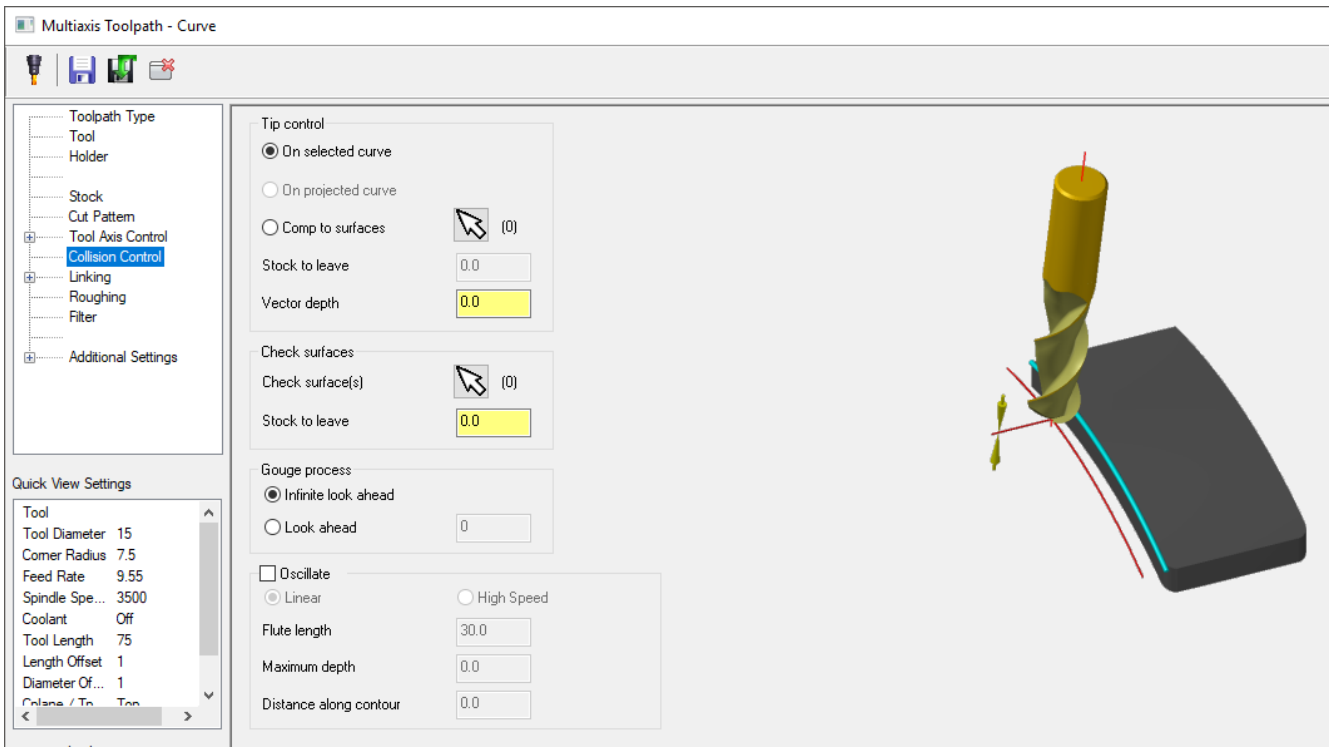
Tool Axis Control Page

The **Tool Axis Control** page defines the tilting motion of the tool axis as it moves along the cut pattern. The options available vary by toolpath type, just as the tree structure and other pages vary. Tool axis control is what sets multiaxis toolpaths apart from 2- and 3-axis toolpaths. The ability to manipulate the tool axis allows for complex and powerful control. Below is the **Tool Axis Control** page for a Curve toolpath.



Collision Control Page

Tool tip control is found on the **Collision Control** page. Use the parameters to instruct the toolpath how the tip of the tool should be placed in relation to the cut pattern. Below is the **Collision Control** page for a Curve toolpath.



Complete the remaining pages in the tree if necessary. Additional parameters include linking information (how the tool moves when not in contact with material) and roughing options. The **Additional Settings** branch provides pages that generally do not need to be touched for multiaxis programming. Continue on to the next chapter to begin creating a Curve toolpath.

CHAPTER 4

MULTIAXIS CURVE TOOLPATH

Experience with design and toolpath creation is assumed at this point of the tutorial. Detailed steps on such actions as selecting a machine definition, changing the graphics view or construction plane, or making levels visible, are not provided. Please review the *Mastercam Basics Tutorial* before continuing if you are not familiar with these concepts.

Goals

- Open a part file and assign a machine definition.
- Create a Multiaxis Curve toolpath.
- Backplot the toolpath.

Exercise 1: Getting Started with Toolpath Creation

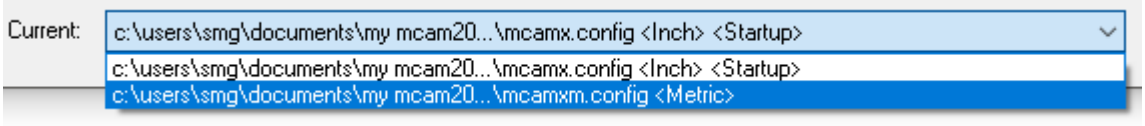
Assigning a machine definition is the first step in creating a toolpath. Setting the graphics view to allow the easiest geometry selection plays a part in visualizing your work. This exercise guides you through the initial steps involved with creating a toolpath.

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

   OR

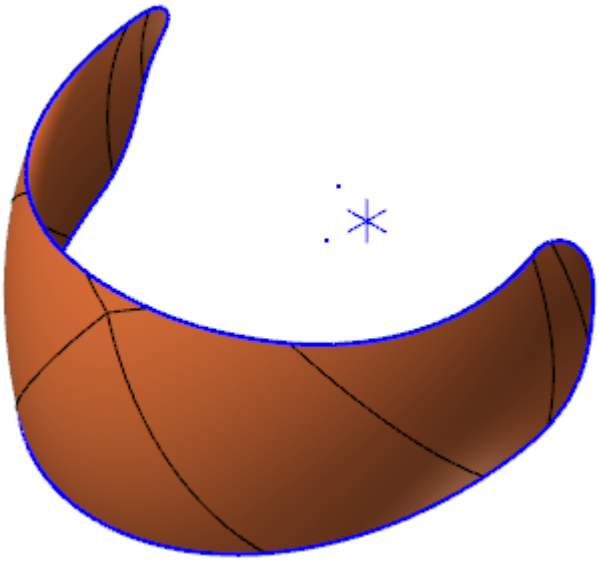
   b. Launch Mastercam from the Windows Start menu.
2. Select the default metric configuration file:
   a. Click the **File** tab.
   b. Choose **Configuration** from Mastercam's Backstage View to open the **System Configuration** dialog box.

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

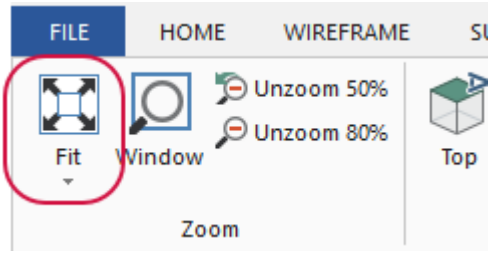


d. Click **OK**.

3. Open the `Curve_Toolpath` part file, which was provided with the tutorial.



4. Fit the geometry to the screen using [**Alt+F1**] or **Fit** on the **View** tab.

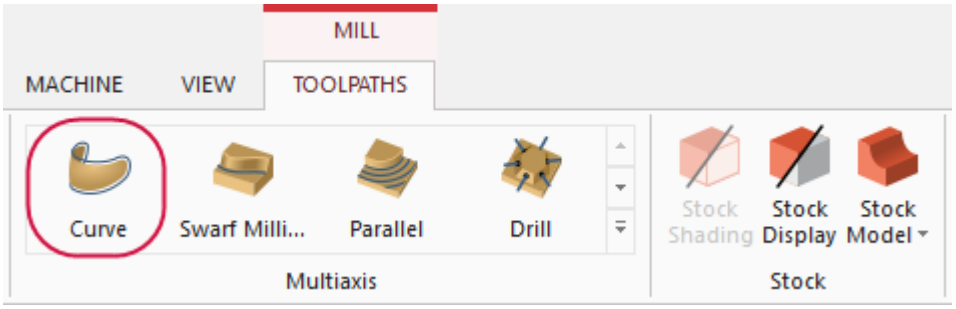


5. Choose **File**, **Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Creating a Multiaxis Curve Toolpath

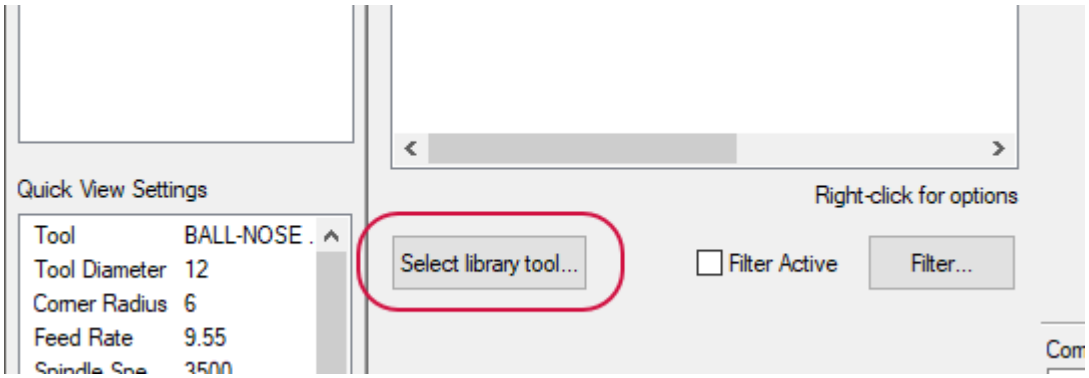
Creating a toolpath involves selecting the toolpath type, tool, and geometry, as well as completing the parameter pages. Completing this process yields a program capable of cutting your part on a machine.

1. From the **Mill Toolpaths** contextual tab, select **Curve** from the **Multiaxis** gallery.



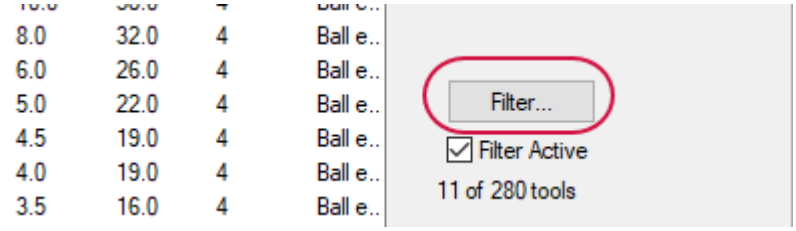
The **Multiaxis Toolpaths - Curve** dialog box opens.

2. On the **Tool** page, click **Select library tool**.



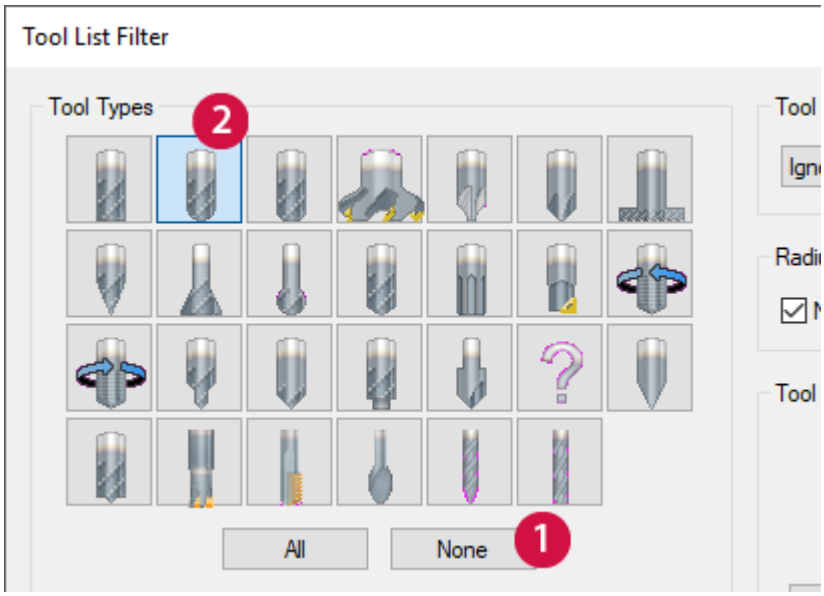
The **Tool Selection** dialog box opens.

3. Click **Filter**.

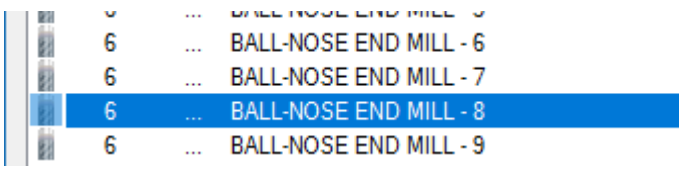


The **Tool List Filter** dialog box opens.

4. Select **None**, and then select **Endmill2 Sphere**.

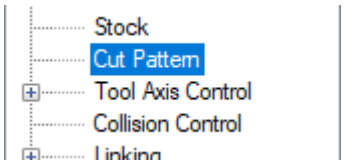


5. Click **OK** to close the **Tool List Filter** dialog box and return to the **Tool Selection** dialog box.
6. Select **BALL-NOSE END MILL - 8** and click **OK**.

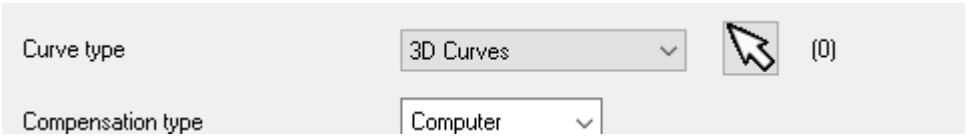


The tool has been added to the toolpath.

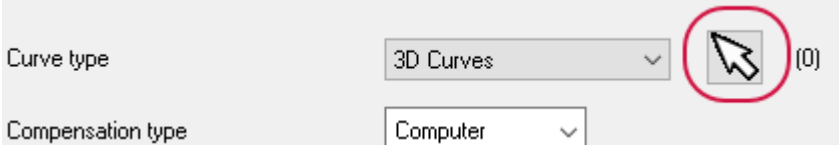
7. Select the **Cut Pattern** page.



8. Choose **3D Curves** from the **Curve type** drop-down selection.

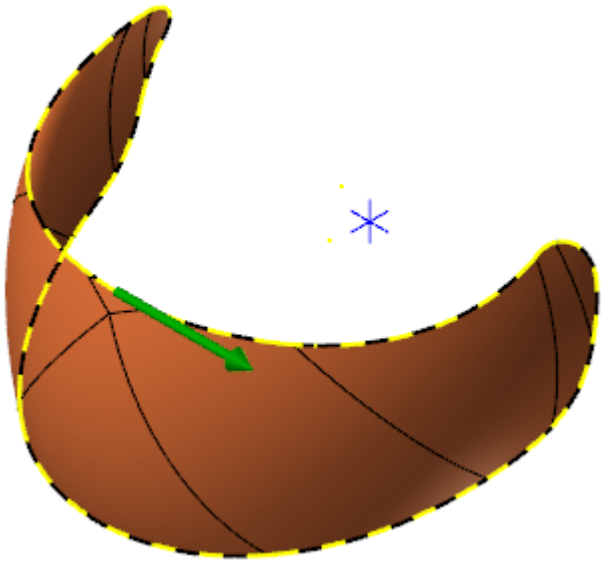


9. Click **Select points** next to 3D Curves to return to the graphics window for curve selection.

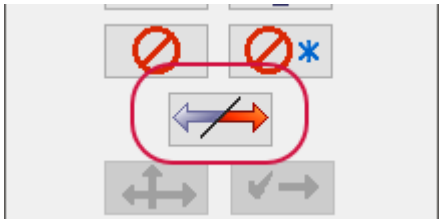


The **Chaining** dialog box displays.

10. Select the chain on the edge of the part as shown. The start position is not important for the purpose of the tutorial. The chain should go in a clockwise direction, regardless of the start point.

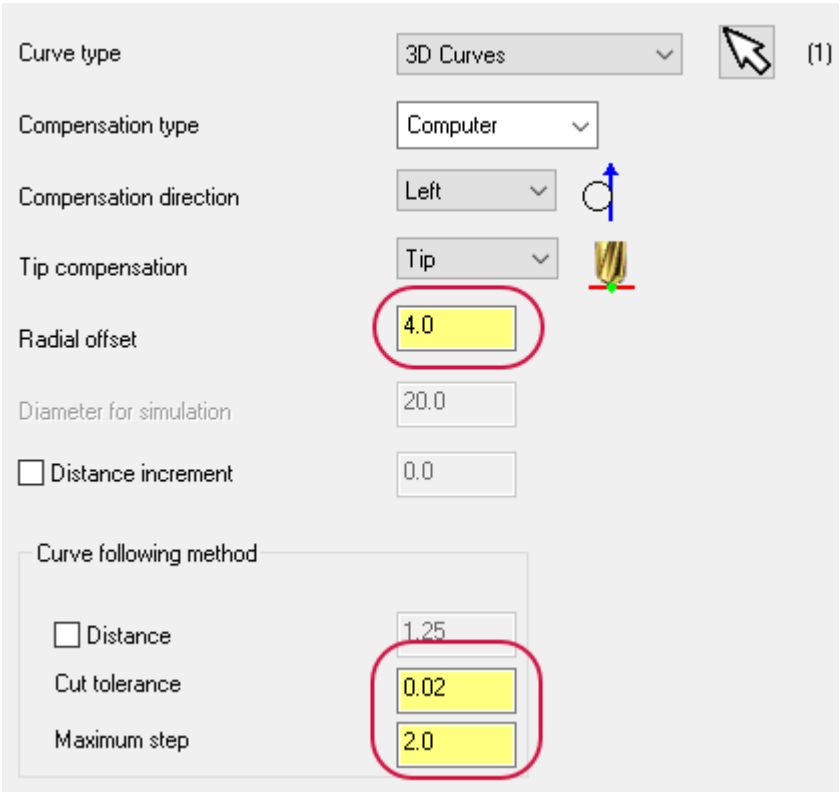


Click **Reverse** in the **Chaining** dialog box if necessary.



11. Click **OK** in the **Chaining** dialog box to return to the **Cut Pattern** page.

12. Set the following parameters:



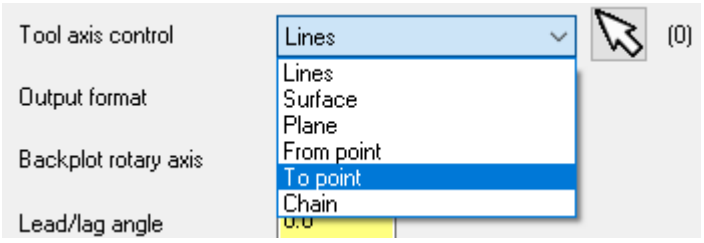
   a. Set **Radial offset** to **4.0**. For this particular operation, it should equal the tool radius. This parameter sets the distance that the center of the tool is offset (left or right) based on the Compensation direction.

   b. Set **Cut tolerance** to **0.02**. This parameter determines the accuracy of the toolpath. A smaller value creates a more accurate toolpath, but may take longer to generate.

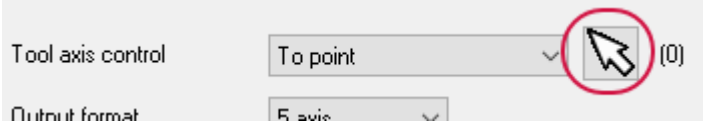
   c. Set **Maximum step** to **2.0**. This is the maximum space allowed between tool vectors.

13. Select the **Tool Axis Control** page.

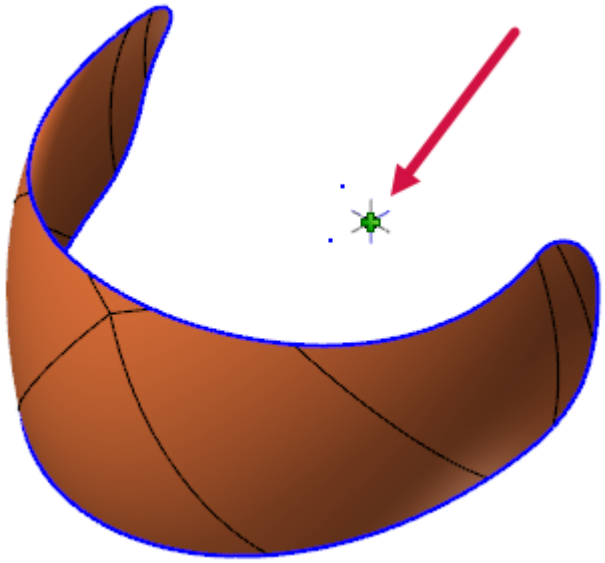
14. Choose **To point** from the **Tool axis control** drop-down selection.



15. Click **Select** to return to the graphics window for point selection.

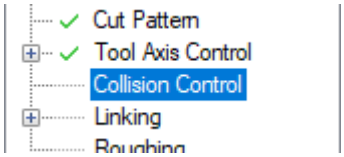


16. Select the point as indicated.

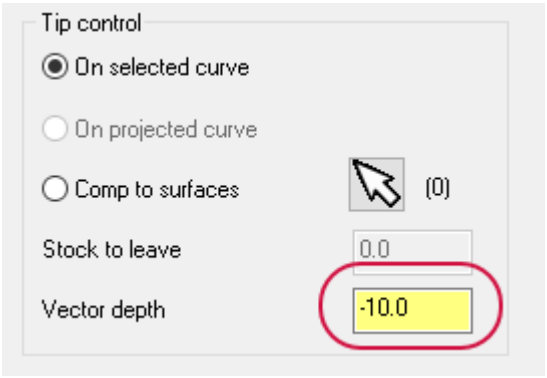


After selecting the point, you automatically return to the **Tool Axis Control** page.

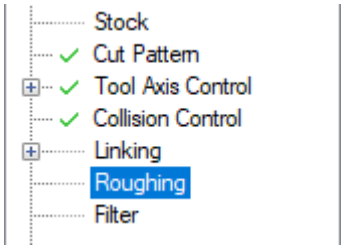
17. Select the **Collision Control** page.



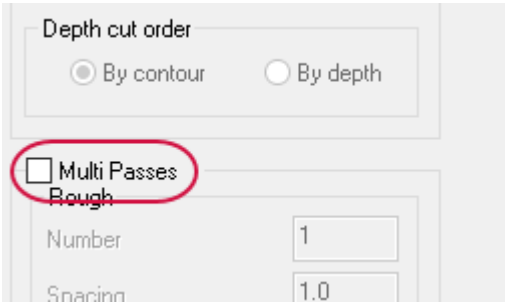
18. Set **Vector depth** to **-10.0**. This parameter offsets the tool tip along the tool axis vector.



19. Select the **Roughing** page.

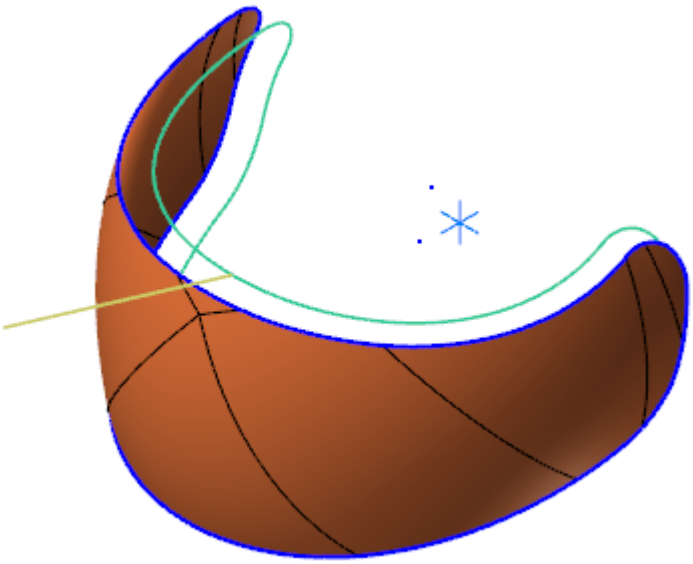


20. Ensure that **Multi Passes** has been deselected.



Trimming operations using a Curve toolpath generally utilize a single pass.

21. Click **OK** to generate the multiaxis curve toolpath on the selected geometry.

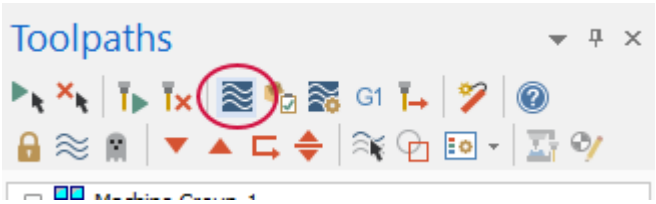


22. Save your part file.

Exercise 3: Backplotting the Operation

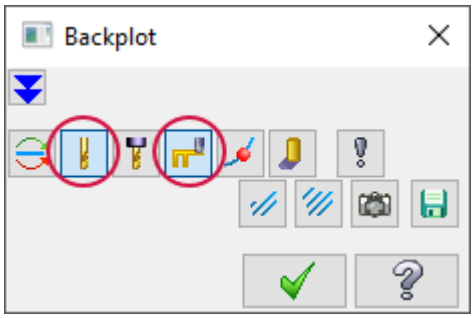
Backplotting an operation allows you to review the tool motion before any cutting takes place on the actual machine. Backplotting should be the first step in validating your toolpath.

1. Select **Backplot selected operations** in the Toolpaths Manager.

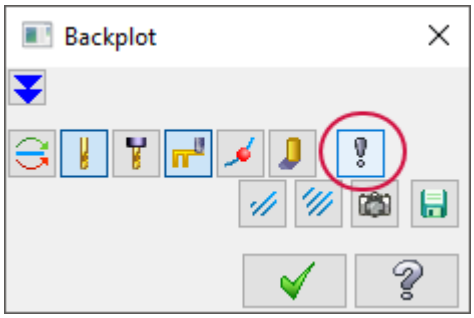


The **Backplot** dialog box opens.

2. Select **Display tool** and **Display rapid moves**.

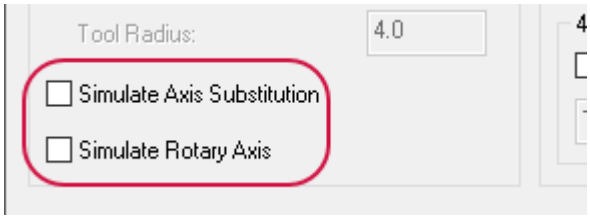


3. Select **Options**.

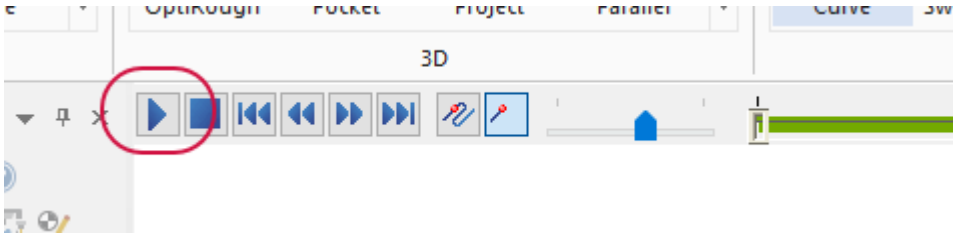


The **Backplot Options** dialog box opens.

4. Deselect **Simulate Axis Substitution** and **Simulate Rotary Axis**, and click **OK**. This prevents your part from spinning around and moving out of view.



5. Click **Play** to begin the backplot of your toolpath.

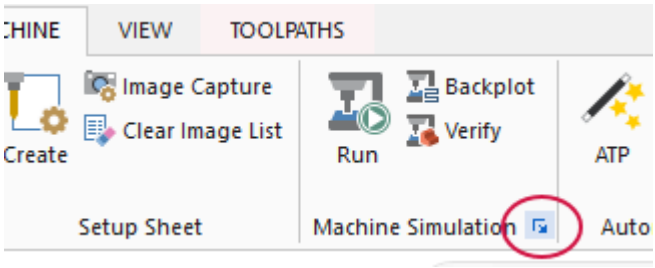


6. Click **OK** in the **Backplot** dialog box when you have finished reviewing the tool motion.

Exercise 4: Running Machine Simulation

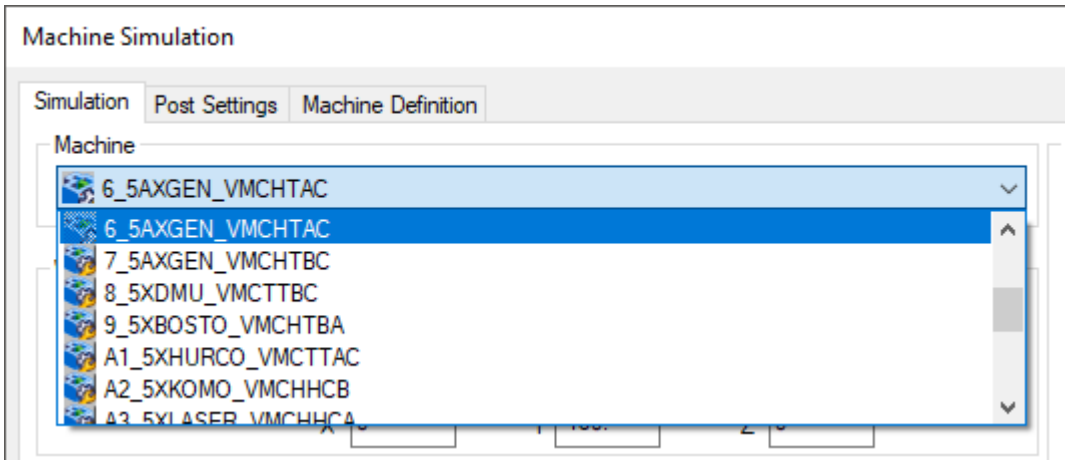
Machine simulation is the next step in validating your operations. The tool motion is shown on your part as it is positioned on your machine and collisions between various machine components and the part are visually apparent, as well as reported in text on the screen.

1. On the **Machine** tab, select the dialog box launcher in the **Machine Simulation** group.



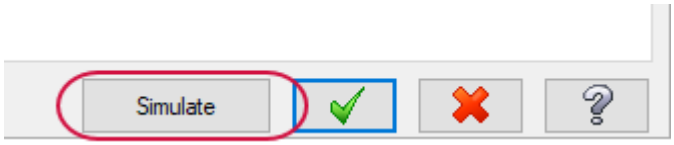
   The **Machine Simulation** dialog box opens.

2. Select **6_5AXGEN_VMCHTAC** from the **Machine** drop-down.

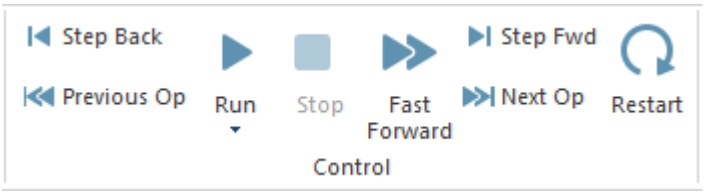


   The machine selection is saved with the part file. Running simulation on the same part file will reload the selected machine.

3. Click **Simulate** to begin the machine simulation of the selected operation.



4. Use the playback buttons to review the tool motion contained in your operations.



   If you do not see the part, click **Workpiece**, which is located on the **Simulation** tab.

5. Exit Machine Simulation to return to Mastercam.
6. Save your part file. The same file will be used in a subsequent lesson.

You have now completed a basic multiaxis Curve toolpath. Curve is generally used for trimming operations. The motion is typically straightforward, making it an ideal toolpath for demonstrating some different tool axis control strategies and the effects they can produce on the tool.

CHAPTER 5

TOOL AXIS CONTROL OPTIONS

Tool axis control is what separates multiaxis toolpaths from normal surface, chain, and point toolpaths. The ability to manipulate the angle of the tool, relative to the cut pattern, puts a great amount of control in your hands. The cut speed, finish quality, and tool life can all be affected with tool axis control. Complete this chapter to learn more about two additional tool axis control options for a Curve toolpath.

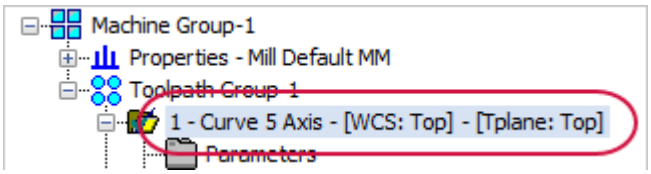
Goals

- Open the previously saved part.
- Edit the operation's parameters, specifically tool axis control.
- Generate the toolpath and observe the tool motion.

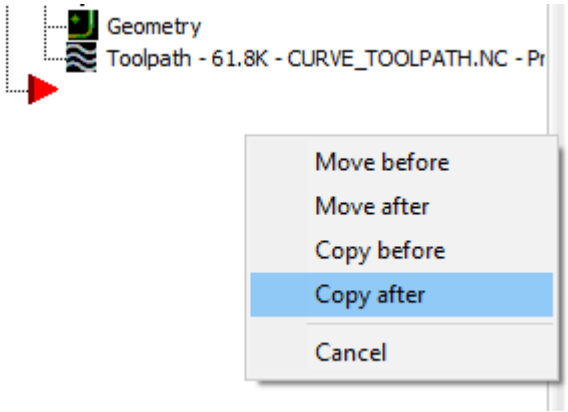
Exercise 1: Copying and Editing an Operation

For this exercise, continue with your `CURVE_TOOLPATH` part from the previous chapter, or open the `CURVE_TOOLPATH_05` part file provided with this tutorial.

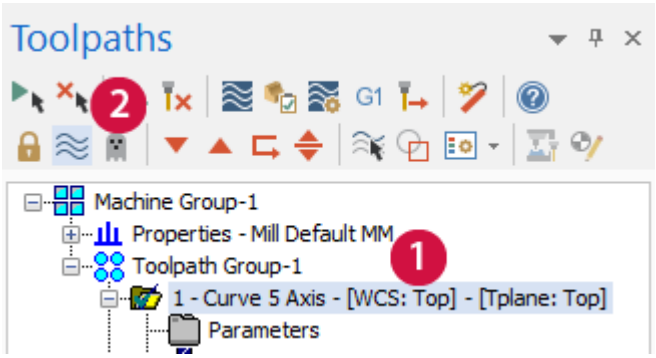
1. If you opened the `CURVE_TOOLPATH_05` part file, choose **File**, **Save As**, and save the part under a different file name.
2. Right-click on the **Curve 5 Axis operation** and hold the mouse button down.



3. Drag the operation to a position farther down the Toolpaths Manager.
4. Release the right mouse button and select **Copy after**.

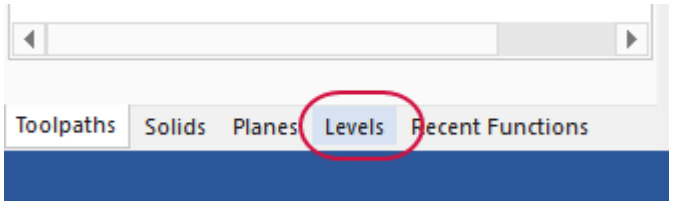


5. Select **1 - Curve 5 Axis** in the Toolpaths Manager, and select **Toggle display on selected operations**.

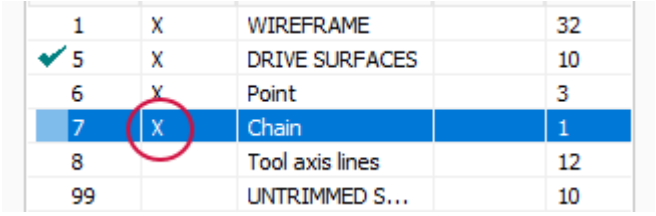


This hides the toolpath to clean up the graphics window.

6. Select the **Levels** tab.

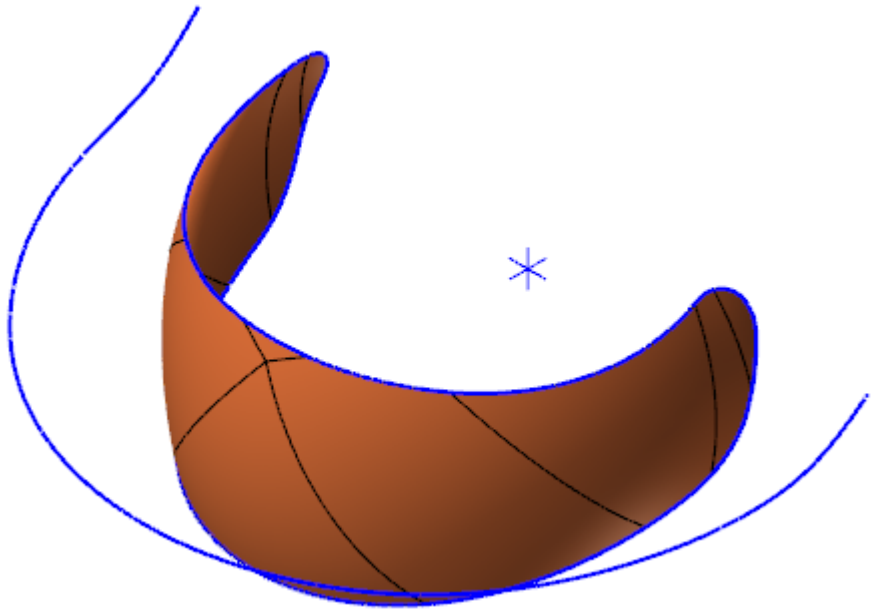


7. Select the **Visible** column for level 7.

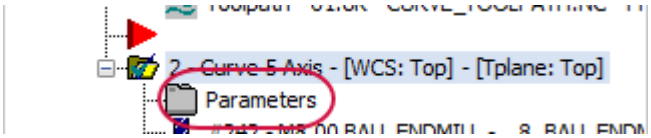


1	X	WIREFRAME		32
✔ 5	X	DRIVE SURFACES		10
6	X	Point		3
7	X	Chain		1
8		Tool axis lines		12
99		UNTRIMMED S...		10

8. The chain geometry that will be used for this exercise is displayed in the graphics window.

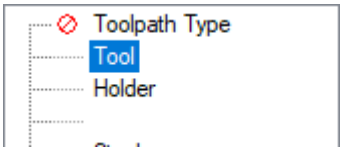


9. In Toolpaths Manager, click **Parameters** under **2 - Curve 5 Axis**.

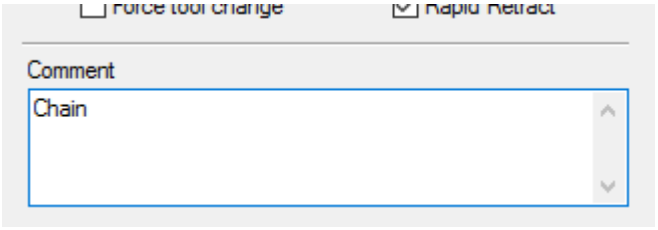


The **Multiaxis Toolpath - Curve** dialog box opens.

10. Select the **Tool** page.

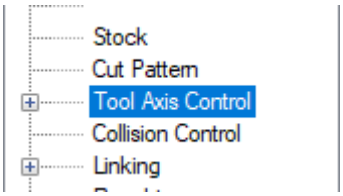


11. Enter **Chain** in the **Comment** box.

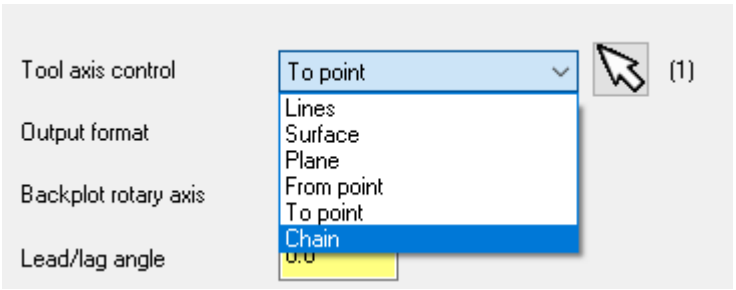


A comment helps to distinguish operations of the same toolpath type.

12. Select the **Tool Axis Control** page.

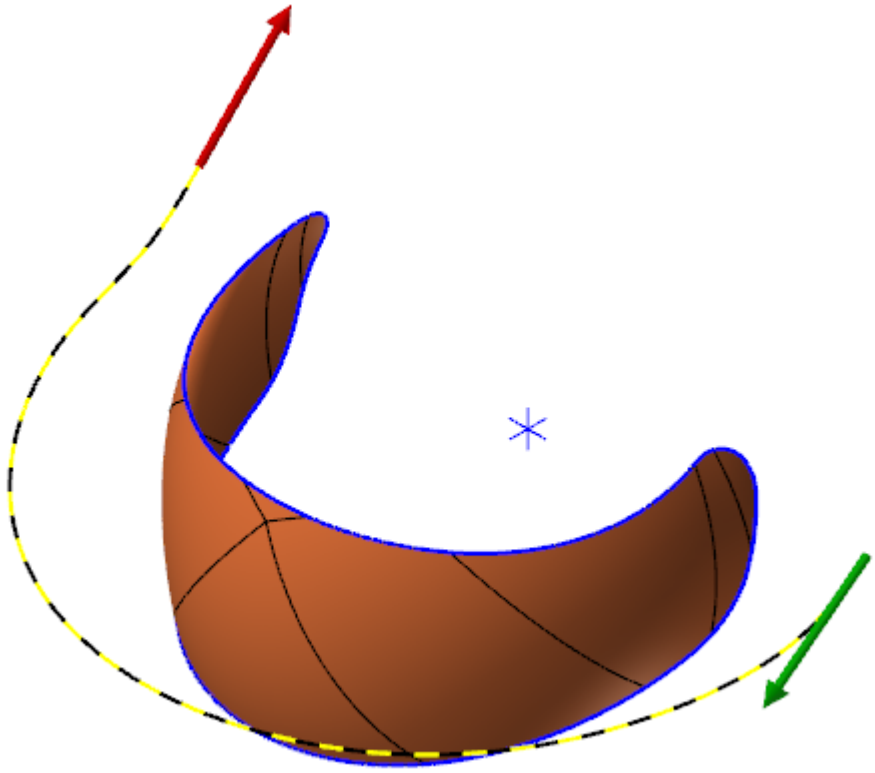


13. Choose **Chain** from the **Tool axis control** drop-down.



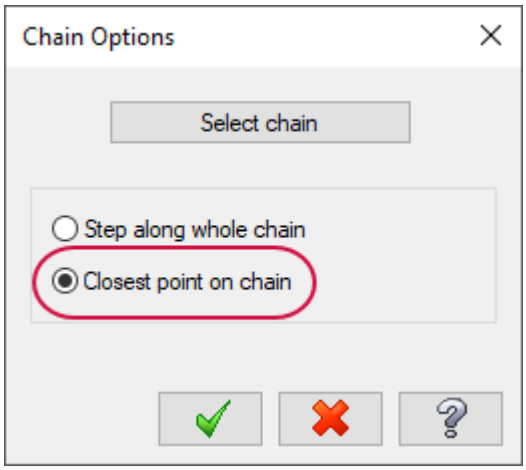
14. Click **Select** to return to the graphics window and open the **Chaining** dialog box.

15. Select the chain as shown, and click **OK** in the **Chaining** dialog box.

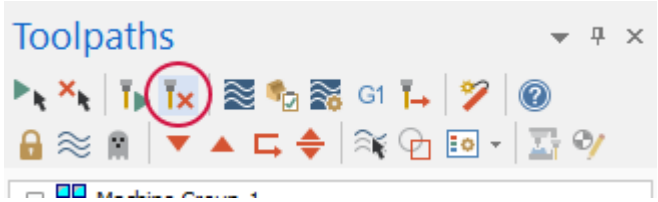


The **Chain Options** dialog box opens.

16. Select **Closest point on chain**, and click **OK** to return to the **Tool Axis Control** page.



17. Click **OK** in the **Multiaxis Toolpath - Curve** dialog box to accept the parameter changes.
18. Select **Regenerate all dirty operations** in the Toolpaths Manager.



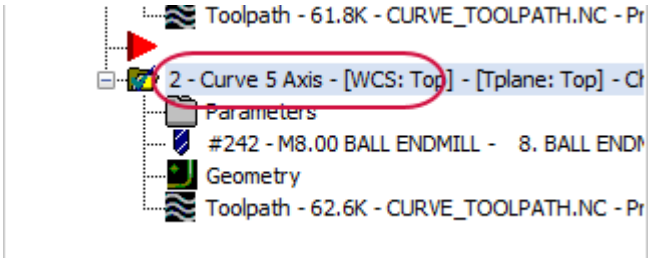
19. Backplot the operation to observe the tool motion.
20. Save your part file.

At this point in the chapter, take a moment to review the two operations created. Operation 1 uses **To point** for tool axis control. The tool is always pointing at the selected point. Operation 2 uses **Chain** for tool axis control. The tool is always pointing from a point on the chain, in this case, the closest point of the chain. The different options create drastically different tool and machine motion. The edge of the part will also have a different taper depending on the tool axis.

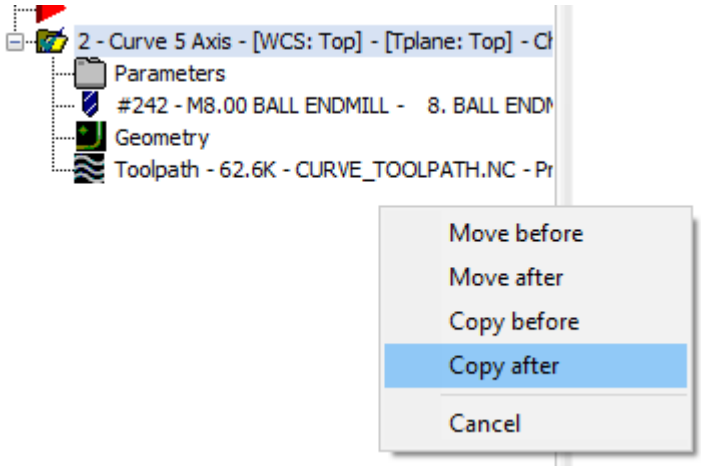
Toggle the toolpath display for the two operations so that they are both visible. Notice the different tool angle. Backplot the operations to see the different tool motion. Select both operations and run Machine Simulation as in the previous lesson. A collision is reported for operation 2 before you even run the simulation. This indicates that using the existing chain for tool axis control is not going to work on this particular machine. This does not indicate a bad toolpath; only the fact that this toolpath will not run on the selected machine.

Exercise 2: Copying and Editing the Second Operation

1. In the Toolpaths Manager, right-click on **2 - Curve 5 Axis** and hold the mouse button down.

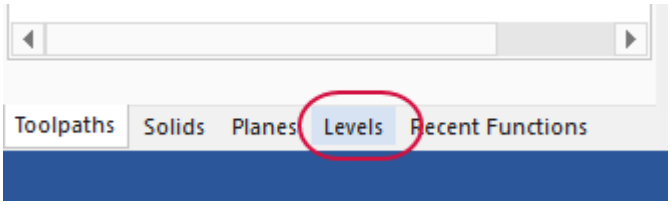


2. Drag the operation to a position farther down the Toolpaths Manager.
3. Release the right mouse button and select **Copy after**.

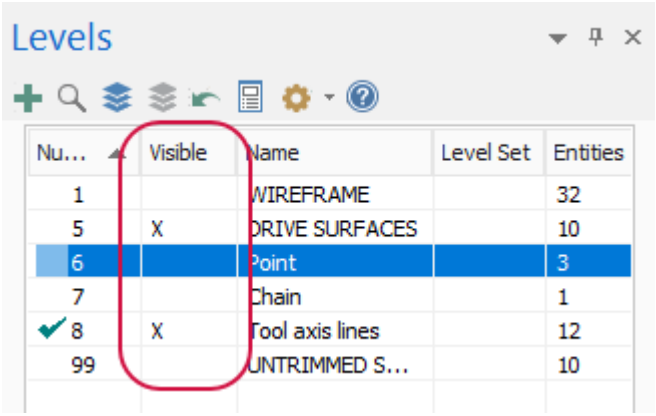


4. Select **2 - Curve 5 Axis** and turn off the toolpath display, by pressing **T**.

   The display for operations 1 and 2 are now both off.
5. Select the **Levels** tab to open the Levels Manager.

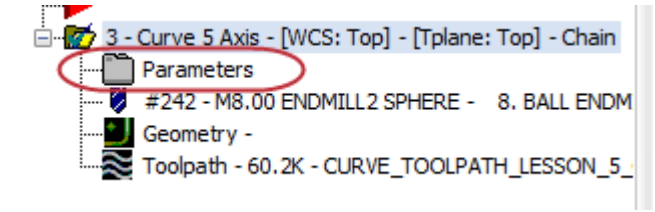


6. Select in the **Visible** column for level 8 to display the tool axis geometry. Select in the **Visible** column for levels 1, 6, and 7 to hide them.



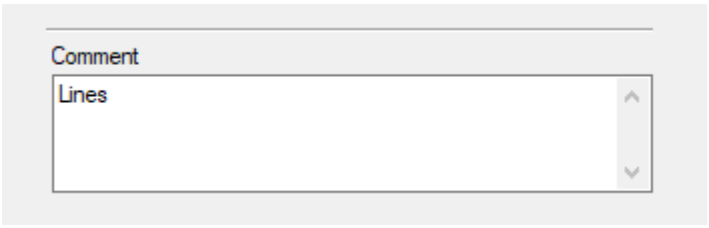
Note: You will need to change the main level in order to hide level 1. Select the **Number** column for level 8 to move the main level.

7. In Toolpaths Manager, click **Parameters** under **3 - Curve 5 Axis**.

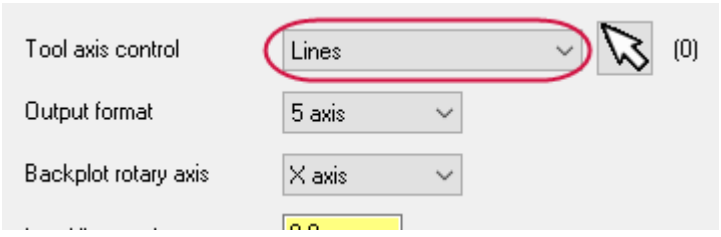


The **Multiaxis Toolpath - Curve** dialog box opens.

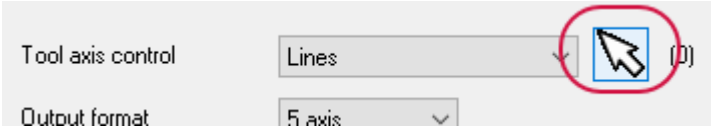
8. Select the **Tool** page.
9. Enter **Lines** in the **Comment** box.



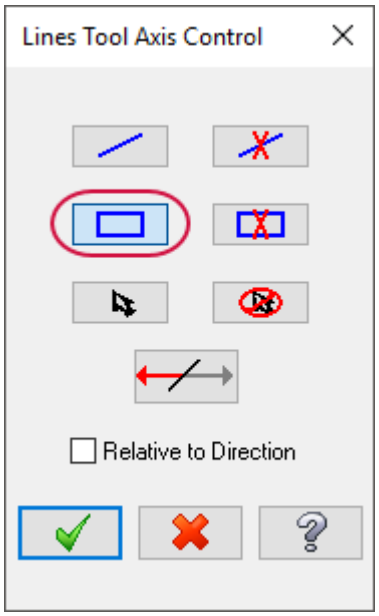
10. Select the **Tool Axis Control** page.
11. Choose **Lines** from the **Tool axis control** drop-down selection.



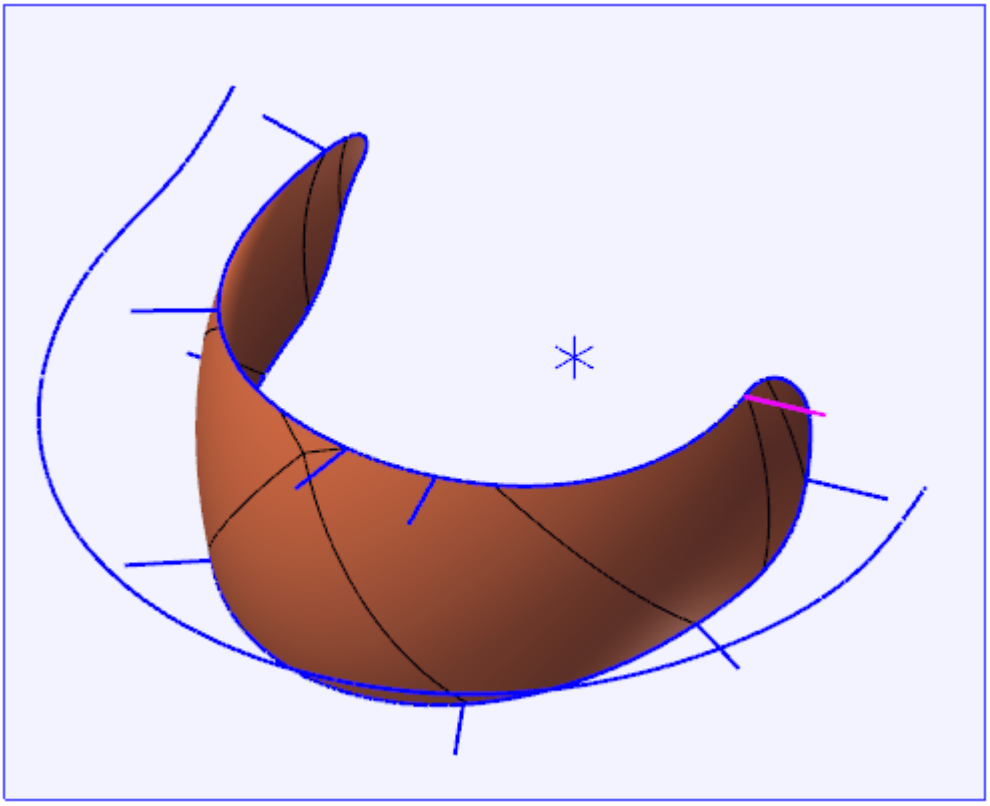
12. Click **Select** to return to the graphics window and open the **Lines Tool Axis Control** dialog box.



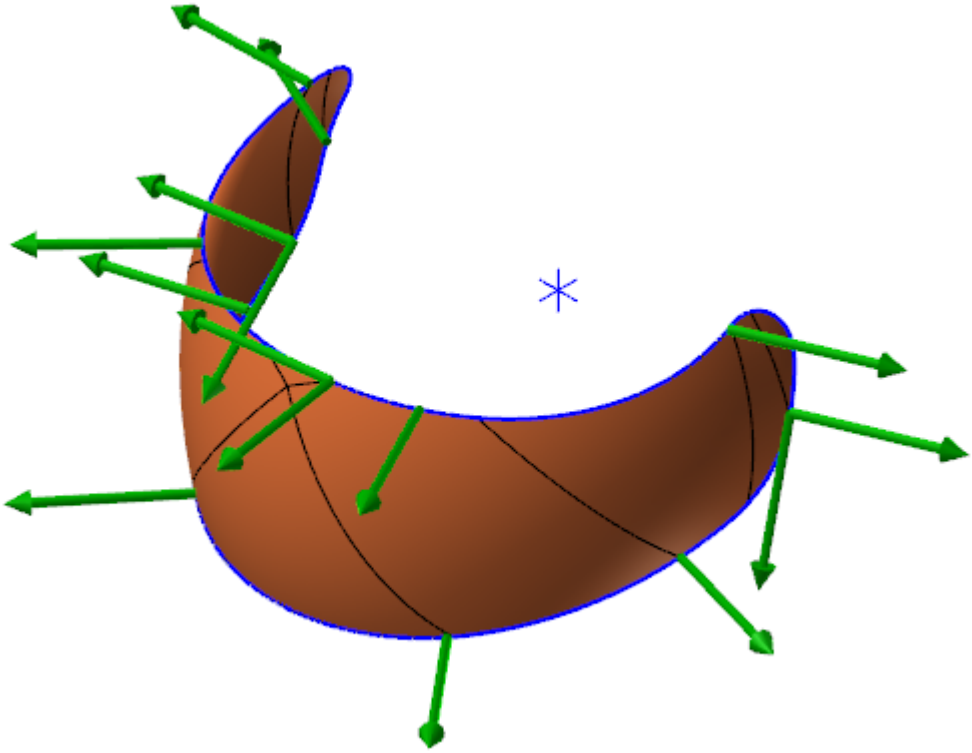
13. Select **Window select lines** to allow you to select lines.



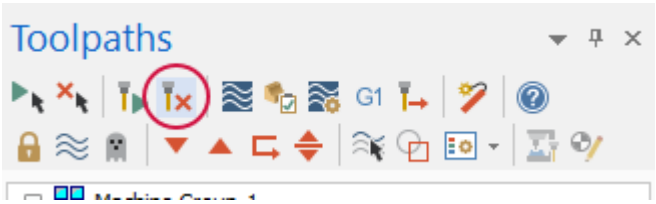
14. Left-click in the graphics window to create a selection window around the part, and then release the mouse button.



15. The part will then match the picture below:



16. Click **OK** in the **Lines Tool Axis Control** dialog box to return to the **Tool Axis Control** page.
17. Click **OK** in the **Multiaxis Toolpath - Curve** dialog box to accept the parameter changes.
18. Select **Regenerate all dirty operations** in the Toolpaths Manager.



19. Backplot the operation to observe the tool motion.
20. Save your part file.

Review the tool motion for operation 3. Observe how the tool axis lines up with each line as it approaches that point in the toolpath. The axis transitions from alignment with one line as it moves to the next. Use as many lines as necessary to achieve the desired motion.

Use Backplot and Machine Simulation to review the tool and machine motion for all three operations. The subtle differences in tool axis control put a great deal of power in your hands for generating complex toolpaths. Tool axis control is a key element to all multiaxis toolpaths, no matter which options are available on the page.

CHAPTER 6

MULTIAXIS DRILL TOOLPATH

Multiaxis drill toolpaths differ from other multiaxis toolpaths in several ways:

- Limited cut pattern selection: points, points/lines, lines, or features
- Restricted tool axis control: parallel to line, surface, or plane
- Fewer tip control options: point, projected point, or compensation surface

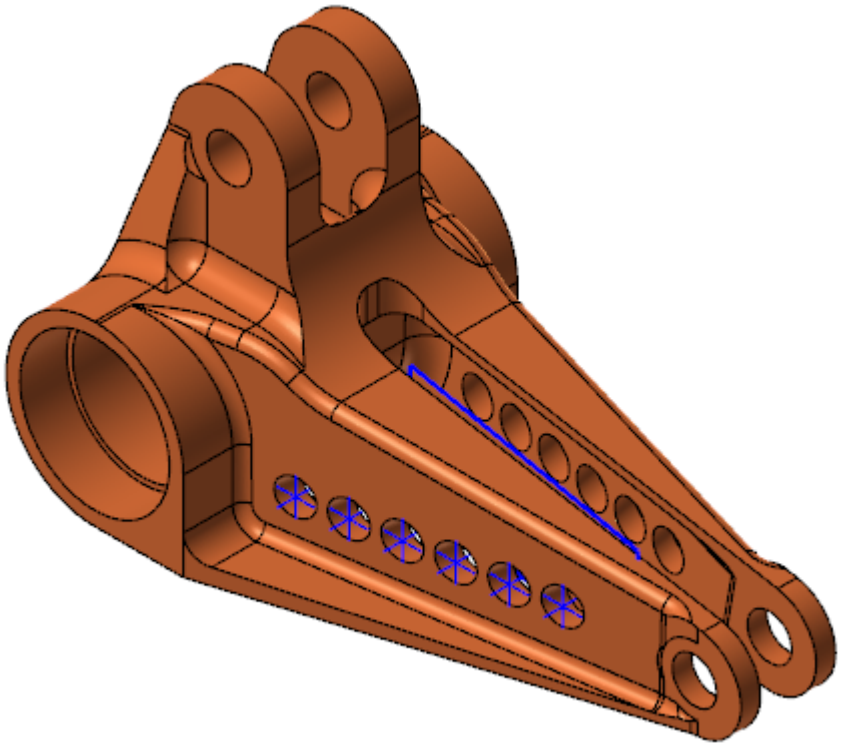
The power of multiaxis drill becomes apparent on parts that contain multiple holes with center lines pointing in different directions. Your multiaxis machine is capable of drilling many different holes with a single fixture setup, saving time and increasing productivity.

Goals

- Create two Multiaxis Drill operations.
- Backplot the operations.

Exercise 1: Creating a Multiaxis Drill Toolpath

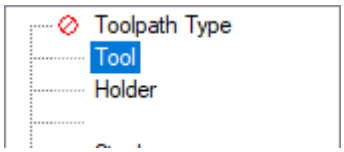
1. Open the part file `Drill_Toolpath`, which was provided with the tutorial.



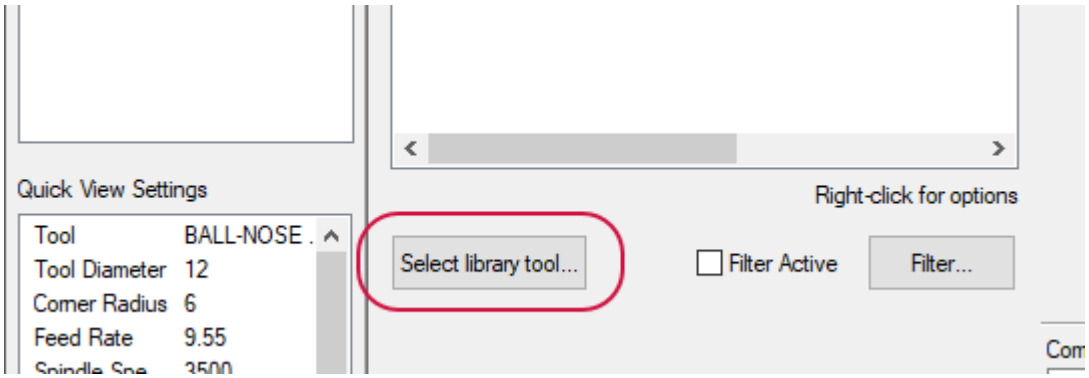
2. Save the part file under a different name to protect the original.

3. Select **Drill** from the **Multiaxis** gallery on the **Mill Toolpaths** contextual tab.

4. Select the **Tool** page.

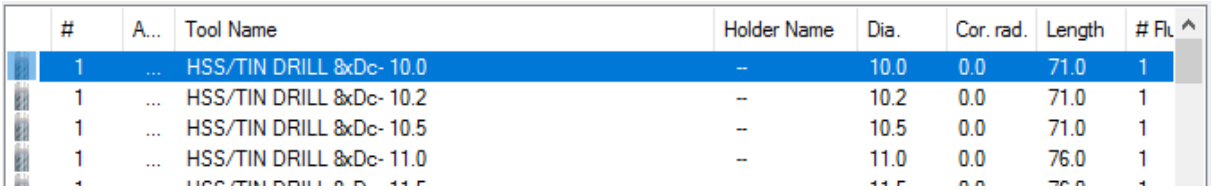


5. Click **Select library tool**.



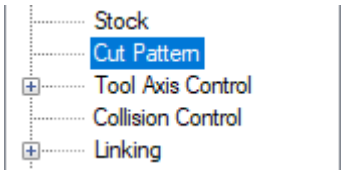
The **Tool Selection** dialog box opens.

6. Select the **HSS/TIN DRILL 8xDc-10.0** tool, and click **OK**.

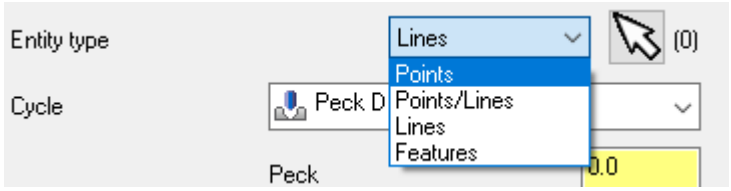


Adjust the tool filter settings as necessary to make tool selection easier.

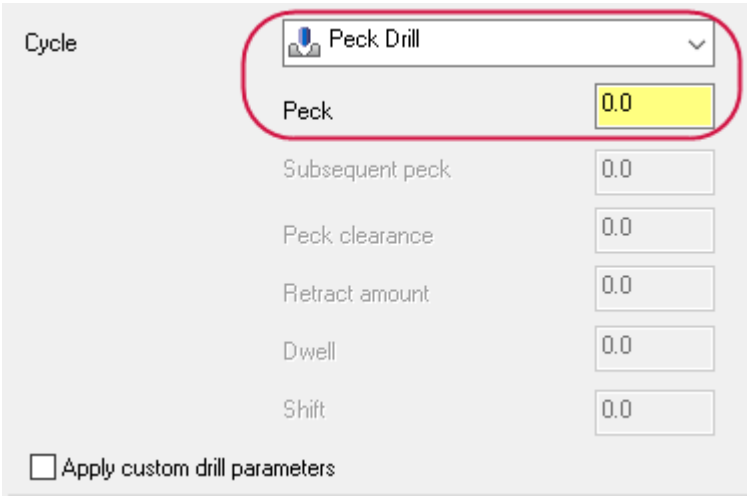
7. Select the **Cut Pattern** page.



8. Set **Entity type** to **Points**.

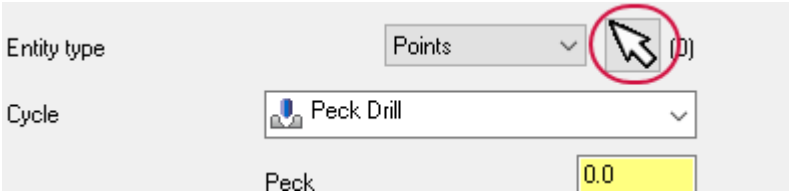


9. Ensure that **Peck Drill** is selected for **Cycle** and that **Peck** is set to **0.0**.



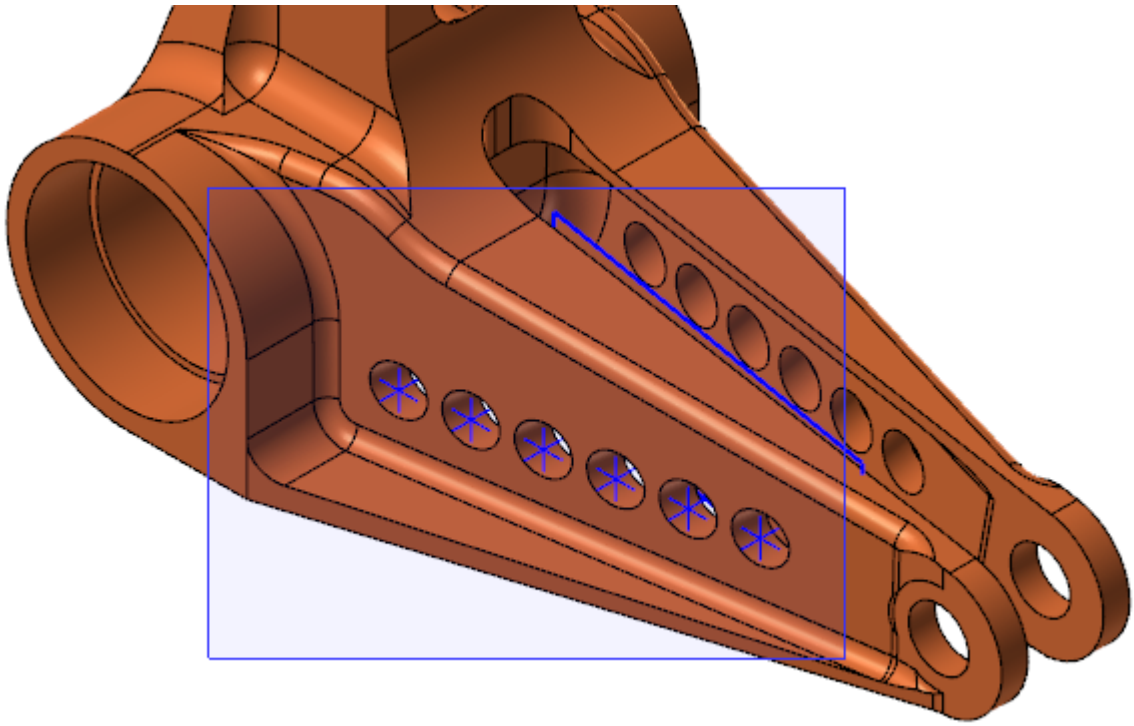
Cycle selection determines the available parameters.

10. Click **Select points** to return to the graphics window for point selection.



The **Toolpath Hole Definition** function panel opens.

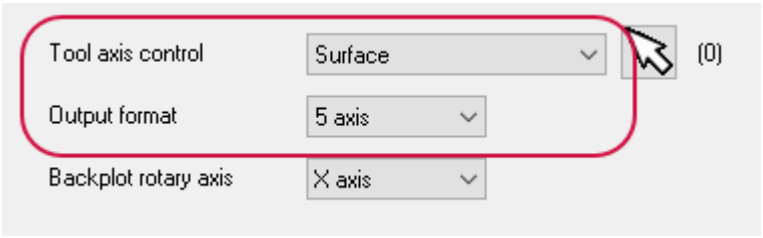
11. Create a window around the visible points, as shown below.



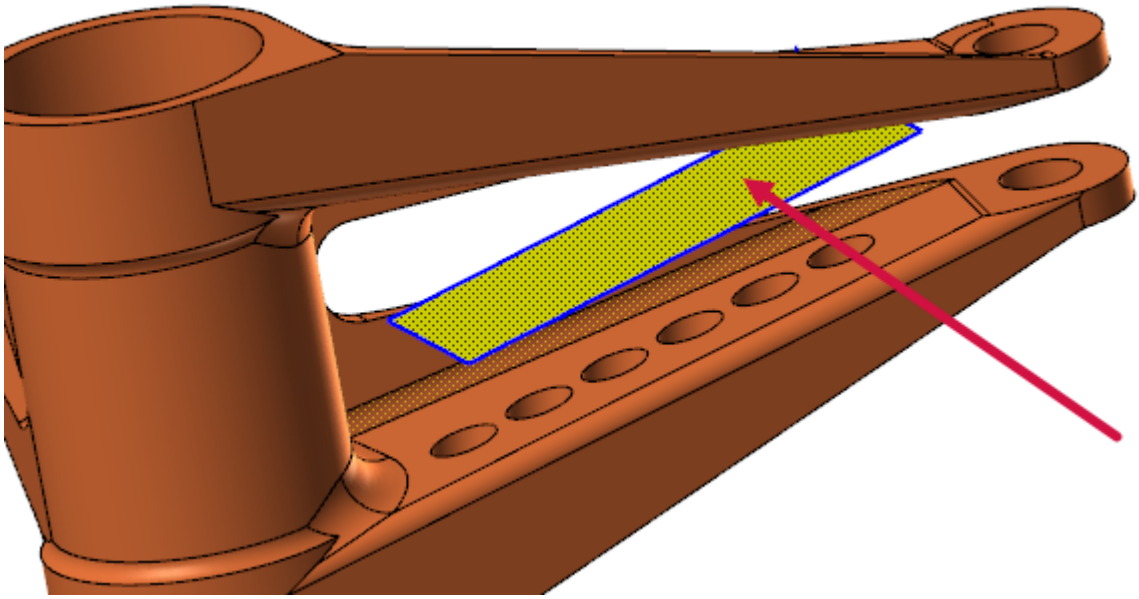
12. Click **OK** in the **Toolpath Hole Definition** function panel to return to the **Multiaxis Toolpath - Drill** dialog box.

13. Select the **Tool Axis Control** page.

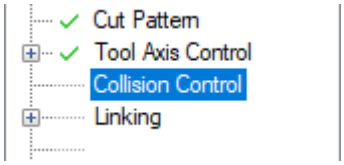
14. Set **Tool axis control** to **Surface** and **Output format** to **5 axis**.



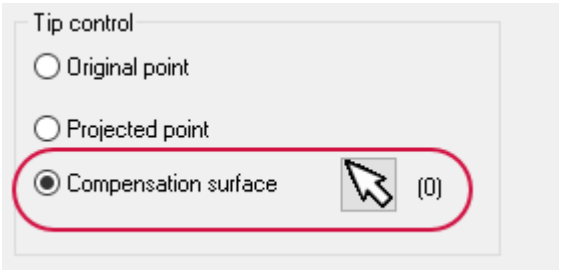
15. Click **Select** to return to the graphics window for surface selection.
16. Select the surface as shown. Rotate the geometry as needed to allow selection.



17. Press **[Enter]** or click **End Selection** when done to return to the toolpath dialog box.
18. Select the **Collision Control** page.

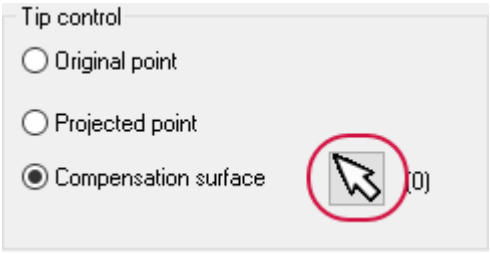


19. Set **Tip control** to **Compensation surface**.



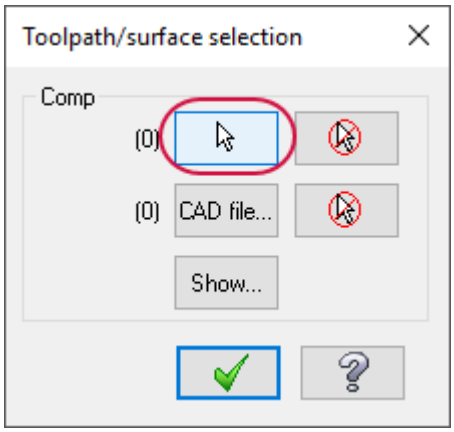
By selecting Compensation surface, the toolpath places the tool tip on the selected surfaces.

20. Click **Select compensation surfaces**.



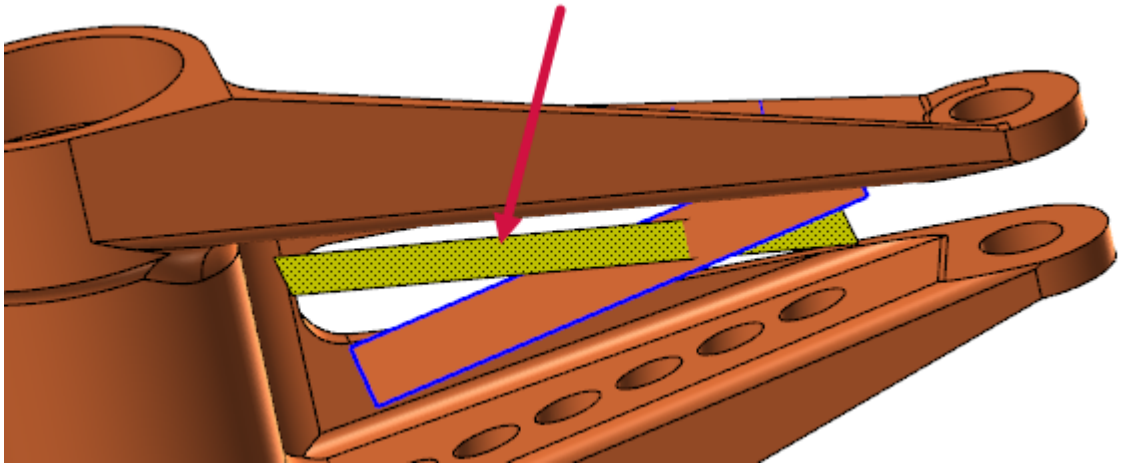
The **Toolpath/surface selection** dialog box opens.

21. Click **Select** to select surfaces in the graphics window.

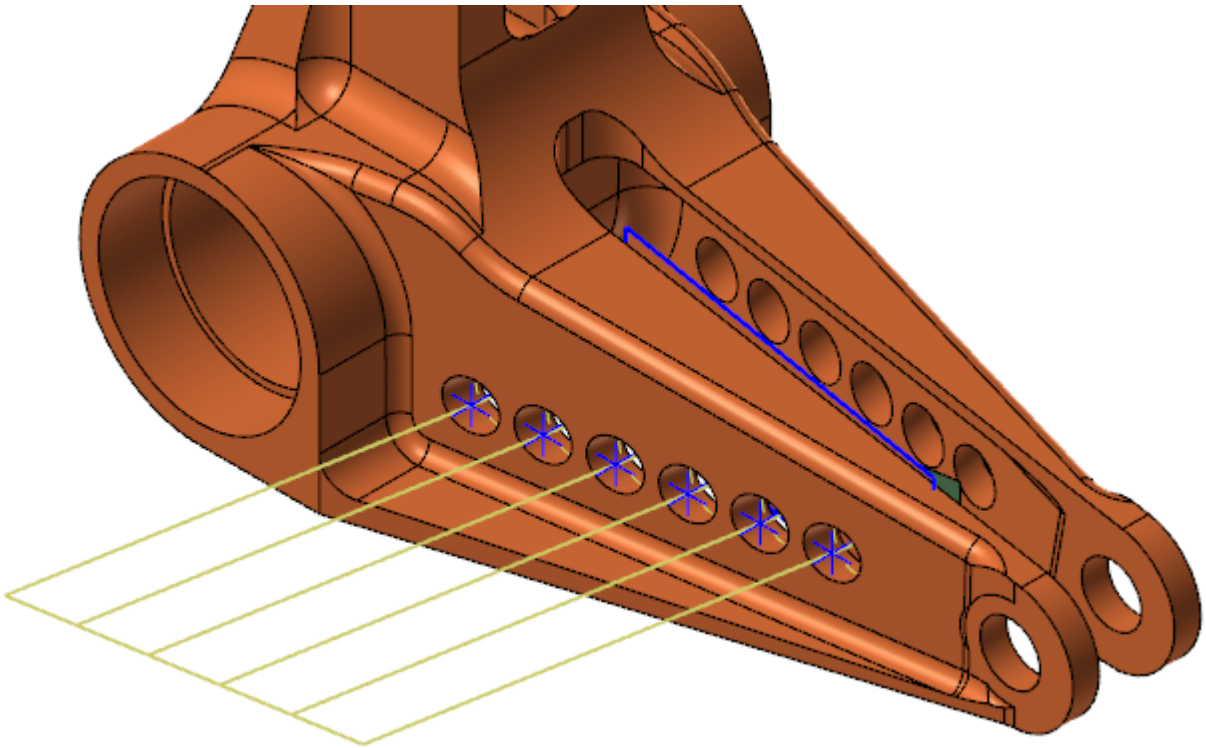


22. Select the **Levels** tab to open the Levels Manager.
23. Select in the **Visible** column for level 4 to display it.

24. Select the surface as shown. Rotate the geometry as needed to allow selection.



25. Press [**Enter**] or click **End Selection** when done.
26. Click **OK** in the **Toolpath/surface selection** dialog box.
27. Click **OK** to generate the multiaxis drill toolpath on the selected points.

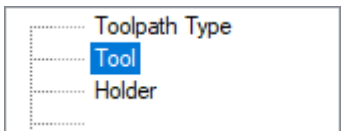


28. Save your part file.

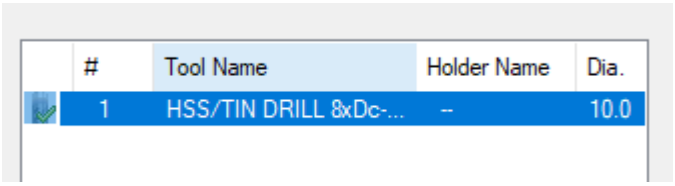
Exercise 2: Adding a Second Multiaxis Drill Operation

1. In Levels Manager, turn off level 1, and make level 3 visible.
2. Select **Drill** from the **Multiaxis** gallery on the **Toolpaths** tab.

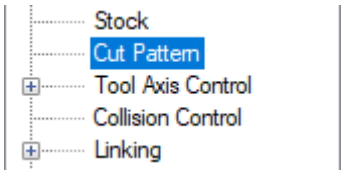
3. Select the **Tool** page.



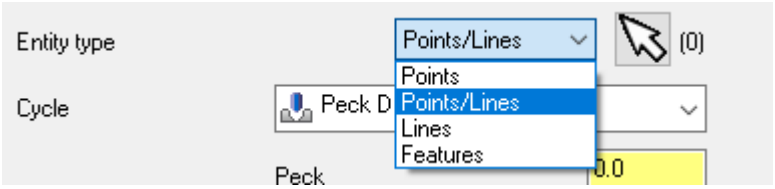
4. Select the 10mm drill that was used in the previous Drill toolpath.



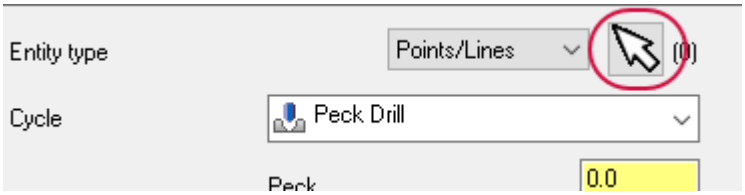
5. Select the **Cut Pattern** page.



6. Set **Entity type** to **Points/Lines**.

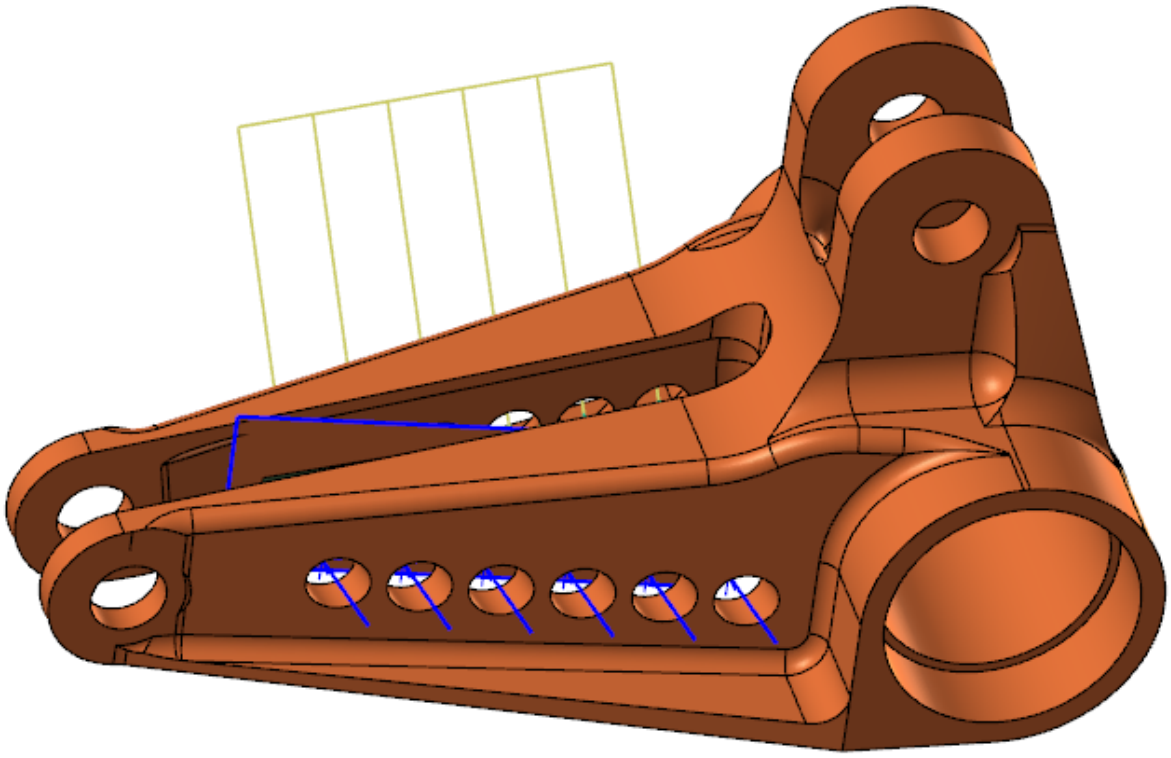


7. The **Cycle** should still be **Peck Drill**. Set it if necessary.
8. Click **Select points** to return to the graphics window for point selection.

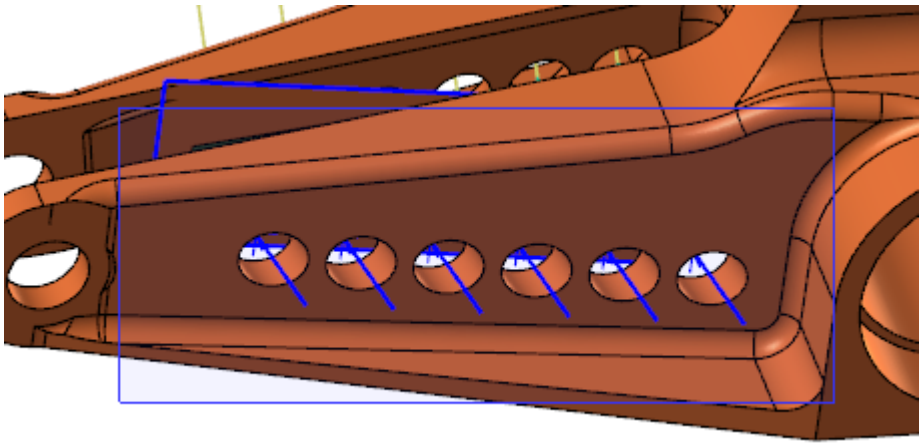


The **Toolpath Hole Definition** function panel opens.

9. Rotate the part so that you can see the points and lines that were displayed when level 3 was turned on.



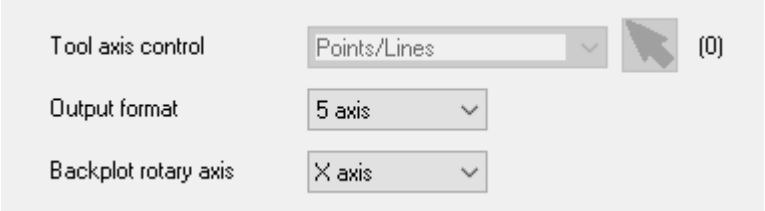
10. Window select the points and lines.



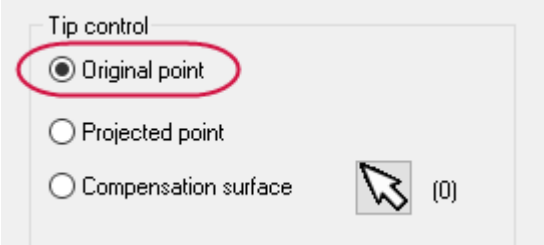
Note: It is not necessary to select the lines, even though the entity type is set to Points/Lines. The lines are automatically selected as long as they end on the point.

11. Click **OK** in the **Toolpath Hole Definition** function panel to return to the toolpath dialog box.
12. Select the **Tool Axis Control** page.

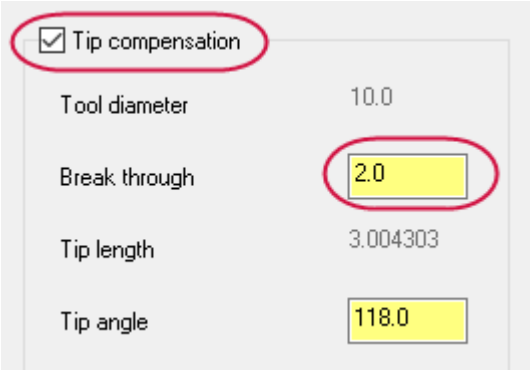
13. Check **Output format** to be sure it is set to 5 axis. Notice how **Tool axis control** is set to Points/Lines. The lines attached to the points are automatically set as the tool axis control.



14. Select **Collision Control** from the tree view pane.
15. Set **Tip control** to **Original point**.

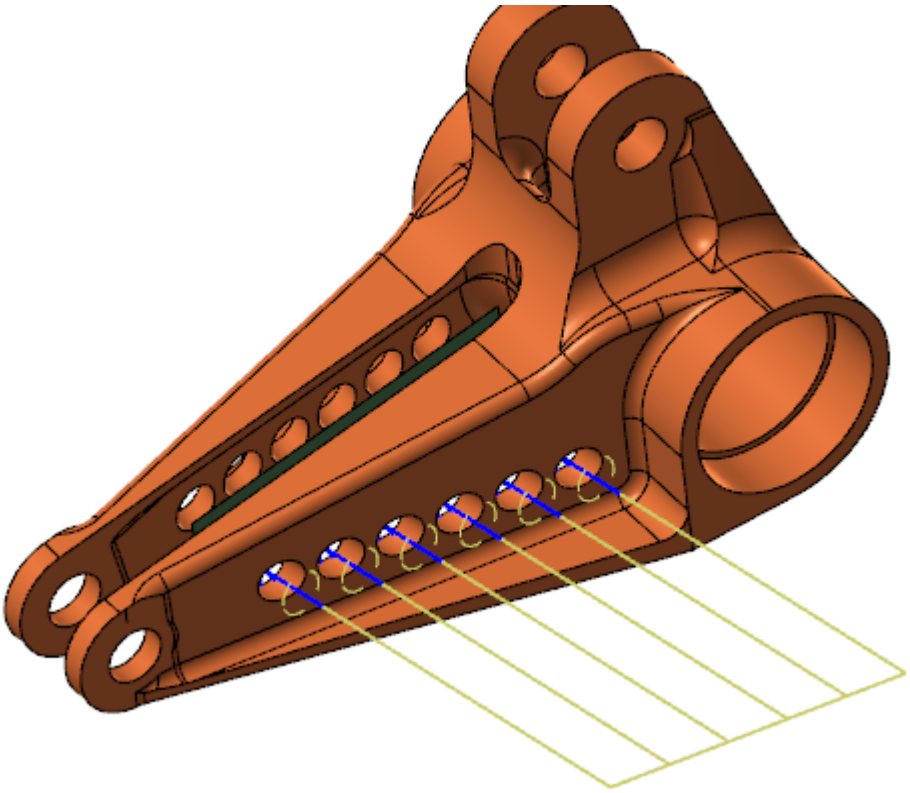


16. Activate **Tip compensation** by selecting the checkbox. Set **Break through** to 2.0.



The **Break through** parameter determines how far the full diameter of the drill penetrates beyond the selected tip control geometry.

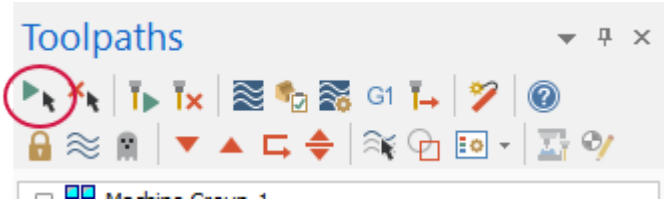
17. Click **OK** to generate the toolpath.



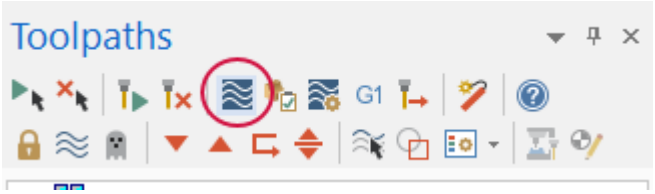
18. Save your part file.

Exercise 3: Backplotting the operations

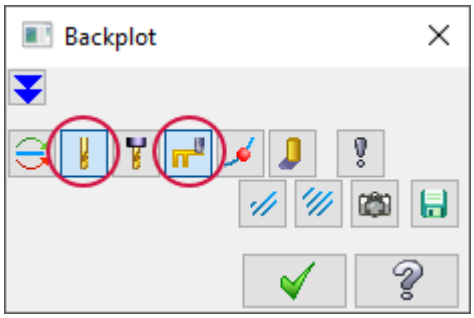
1. Set your graphics view to **Isometric**.
2. In the Toolpaths Manager, click **Select all operations**.



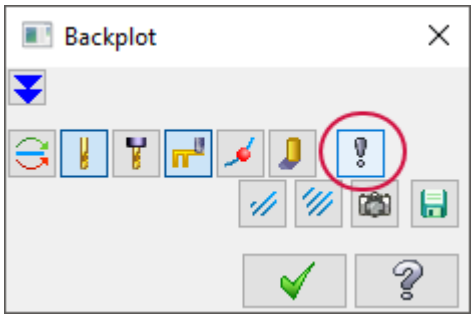
3. Click **Backplot selected operations** in the Toolpaths Manager.



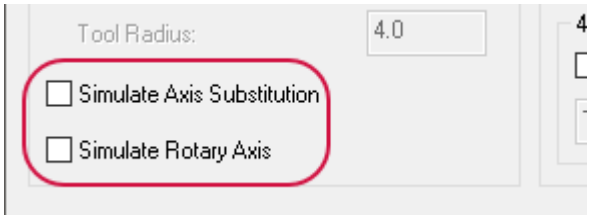
4. In the Backplot dialog box, select **Display tool** and **Display rapid moves**.



5. Click **Options** in the Backplot dialog box.



6. Deselect the options shown and click **OK**. This prevents your part from spinning around and moving out of view.



7. Click **Play** to begin the backplot of your toolpath.
8. Click **OK** in the Backplot dialog box when you have finished reviewing the tool motion.

You have now completed a Multiaxis Drill toolpath. While this is the simplest of the Multiaxis toolpaths, care must be taken when selecting geometry, tool axis control, and tip control methods. The differences between the control methods are subtle but can produce drastically different tool motion.

CONCLUSION

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

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

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing **[Alt+H]** on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) 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

www.mastercam.com

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