Introduction to Multiaxis Toolpaths

April 2016

Mastercam® 2017

Mastercam® 2017 Multiaxis Toolpaths

Date: April 2016

Software: Mastercam 2017

TERMS OF USE

Be sure you have the latest information!

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

Contents

Introduction

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.

Tutorial 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

Introduction to Multiaxis Toolpath Requirements

- Mastercam 2017 with Mill or Mastercam Router
- Curve 5-axis / Drill 5-axis add-on (included in the Multiaxis add-on)

Estimated time to complete this tutorial: 1.5 hours

General Tutorial Requirements

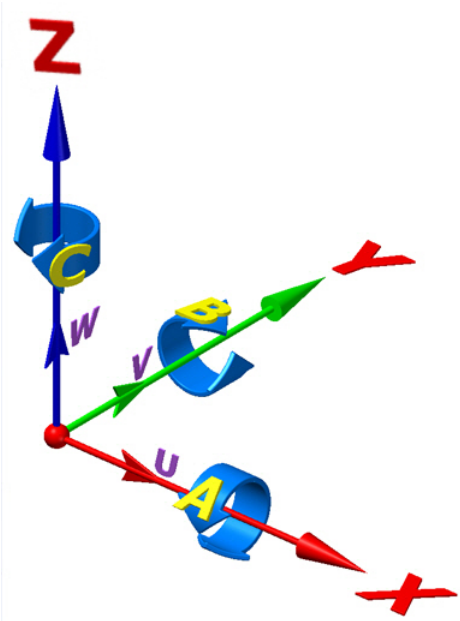
All Mastercam tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.

- The tutorials cannot be used with Mastercam Demo/Home Learning Edition (HLE). The Demo/HLE file format (`emcam`) is different from the Mastercam file format (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs.
- All Mastercam tutorials require you to configure Mastercam to work in a default metric or English configuration. The tutorial provides instructions for loading the appropriate configuration file.

LESSON 1
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. The point is that 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. However, due to the enhanced power of the machine, there is no "standard 5-axis 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

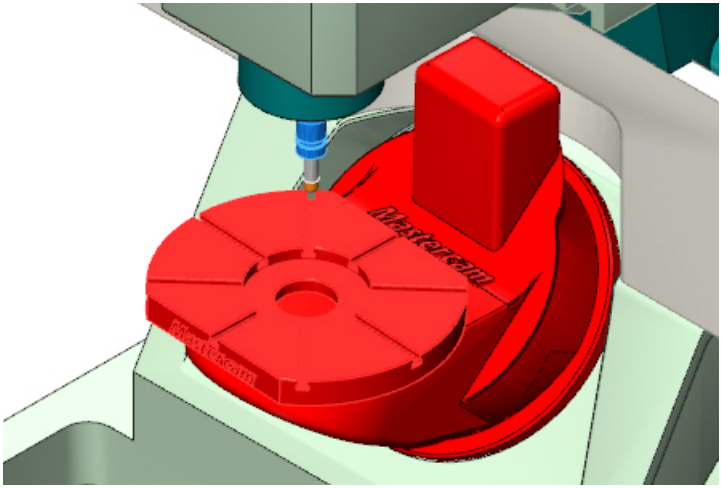
need to locate the part within Mastercam. The part must be located in Mastercam where it will be sitting on the machine. The only exception is for a head/head arrangement.

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

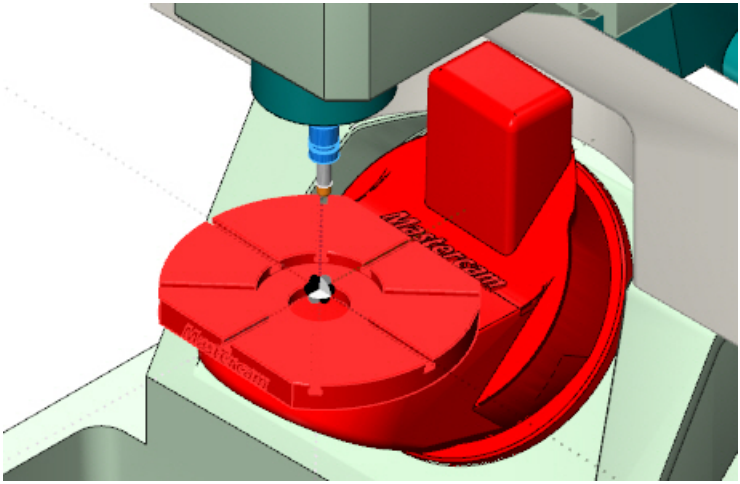
Table/Table Machine

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 shown in red in the following images.

Table/Table configuration



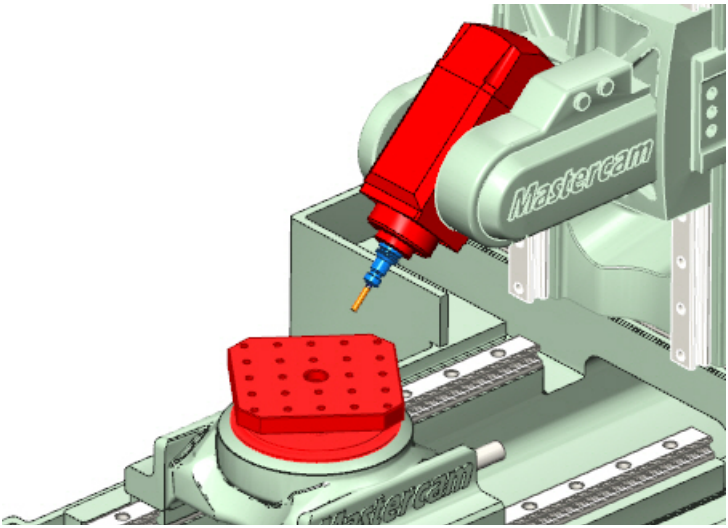
Table/Table machine zero



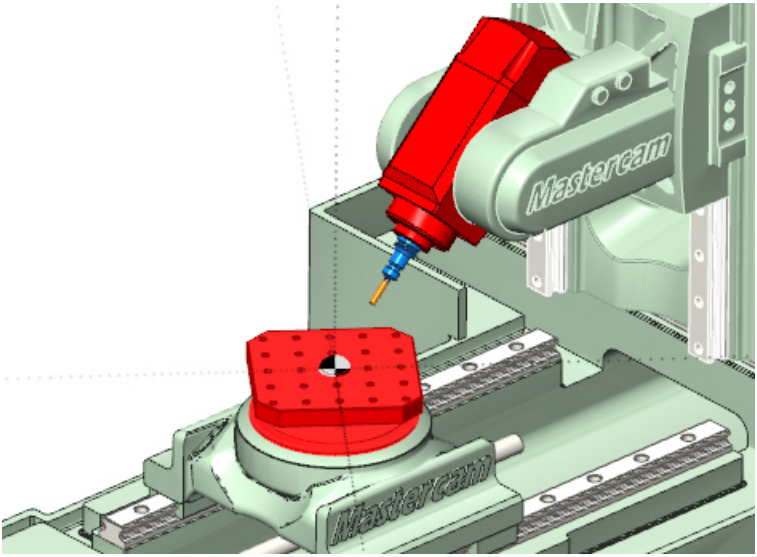
Head/Table Machine

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 shown in red in the following images.

Head/Table configuration



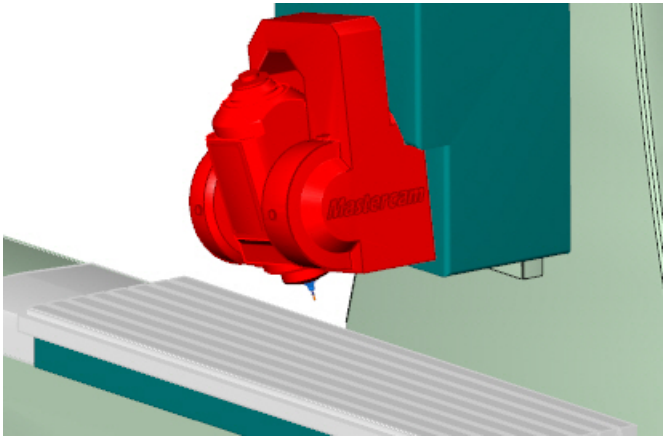
Head/Table machine zero



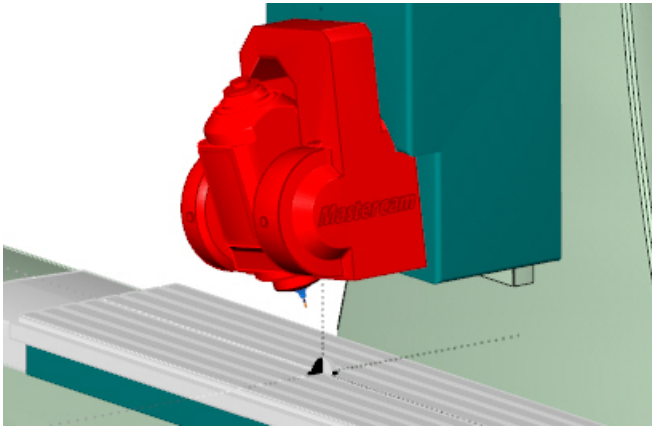
Head/Head Machine

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 in the second image as this is where the spindle face lies when all axes are set to zero. The rotary components are shown in red in the following images.

Head/Head configuration



Head/Head 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 Lesson 2 to learn about the necessary controls for creating an efficient multiaxis toolpath.

LESSON 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

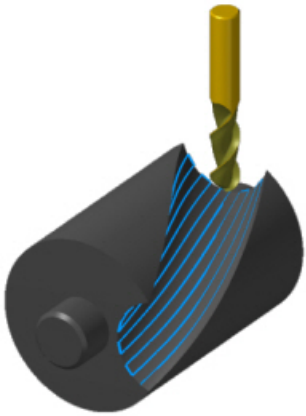
Effective use of these controls is the deciding factor in your ability to create efficient toolpaths for your particular machine. This tutorial starts you on the path to becoming a more effective programmer. However, there is no substitute for experience and knowing your machine's capabilities.

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.

Curve

Surface



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

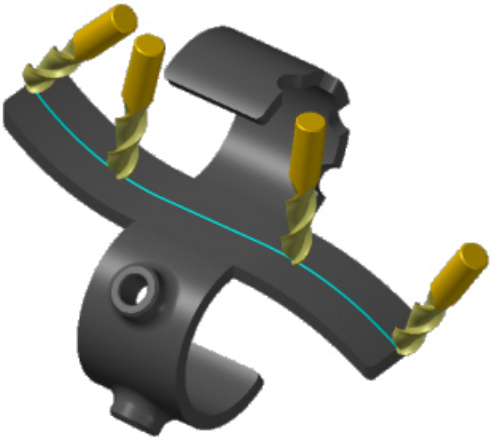
- Cutting method: zigzag, one way, spiral
- Compensation: type and direction
- Stock to leave: drive surfaces, walls
- Stepover: across, along, increment

Subsequent lessons and tutorials cover cut pattern options in more detail.

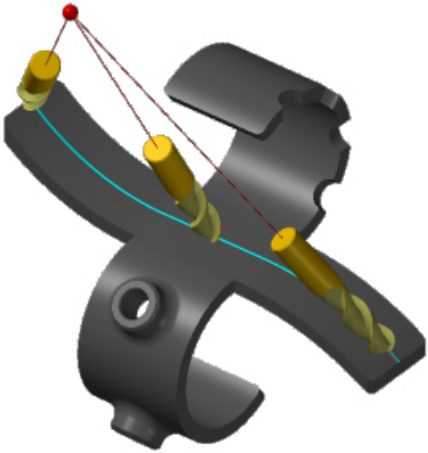
Tool Axis Control

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:

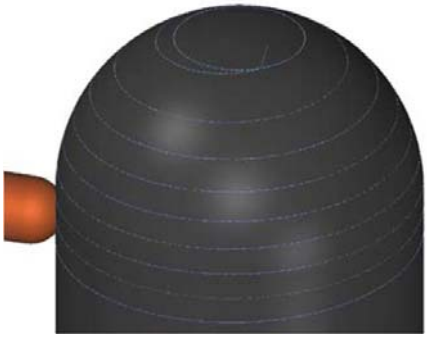
- Lines, surface, from point, chain
- Fixed angle to axis, be tilted relative to contact point
- Rotated around axis, from point
- Parallel to line, surface, plane

Click the Help button in Mastercam to find further details on the available options and their function.

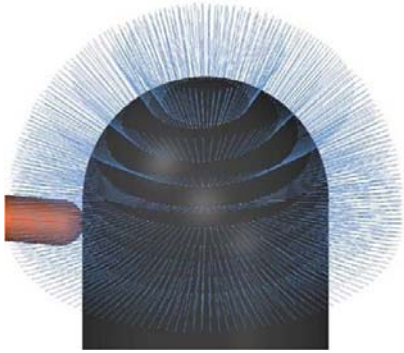
Tool Tip Control

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:

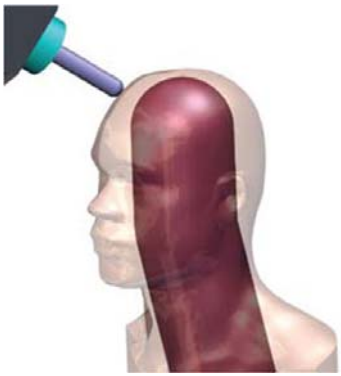
1 Tool positions are generated along the selected cut pattern.



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.



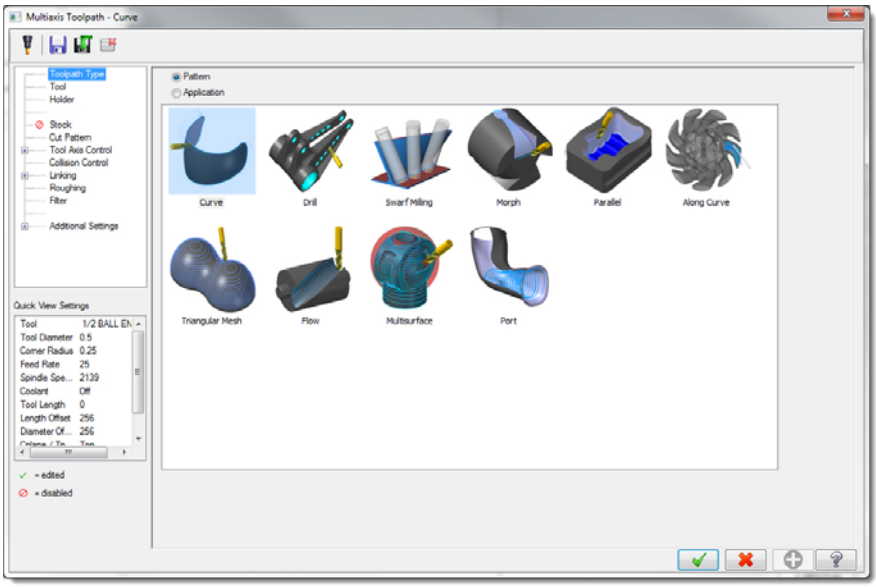
TIP: Use the method depicted above when you have very complicated, or less than perfect, part geometry. The clean core geometry is used to generate the cut pattern and tool axis control vectors. Tip control is then used to cut the outer surfaces with much cleaner motion.

The three controls described above 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.

LESSON 3
Mastercam Interface and Workflow

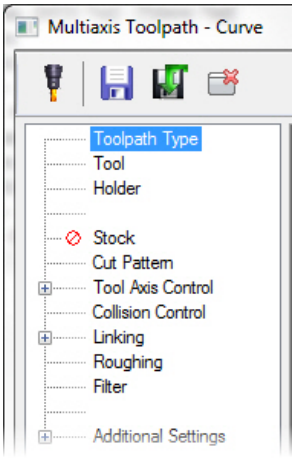
Mastercam's Multiaxis toolpath interface follows the familiar tree-style layout. You progress through the pages of the tree, make the necessary selections, enter appropriate parameters, and generate your toolpath. Viewing the process in simplified terms helps to lessen the complexity of creating a multiaxis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

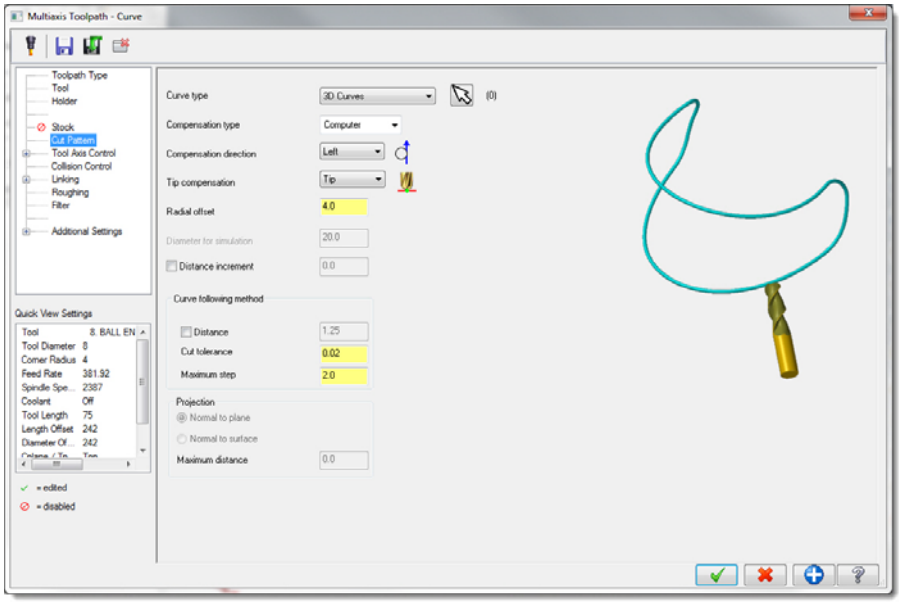


The pages within the tree structure vary with the toolpath type. The procedure remains the same regardless of what the tree looks like. If you can answer the following questions, you are well on your way to generating a multiaxis toolpath:

- 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?

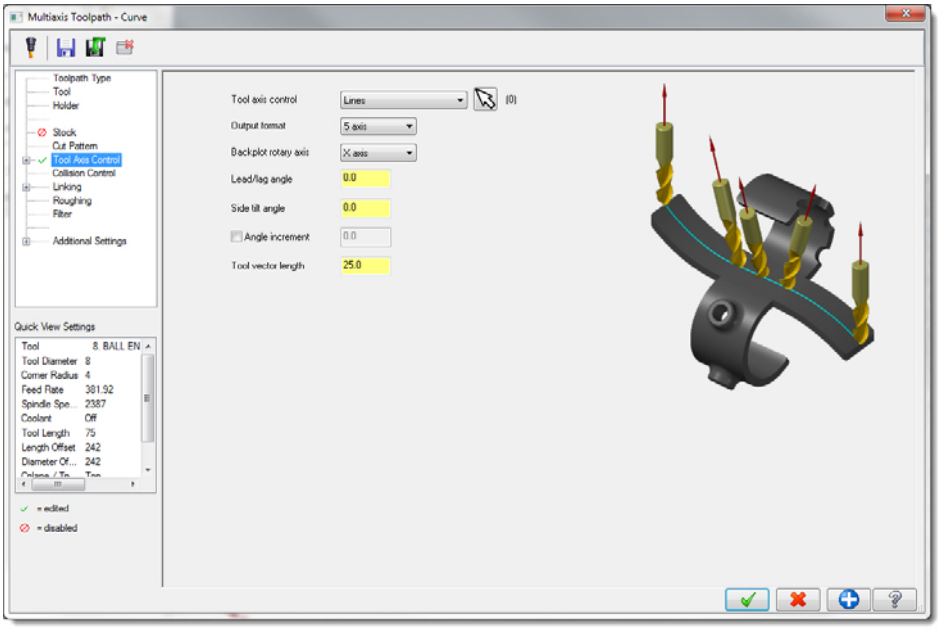
Cut Pattern Page

The controls discussed in the previous lesson 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 a sample of the cut pattern page for a multiaxis Curve toolpath.



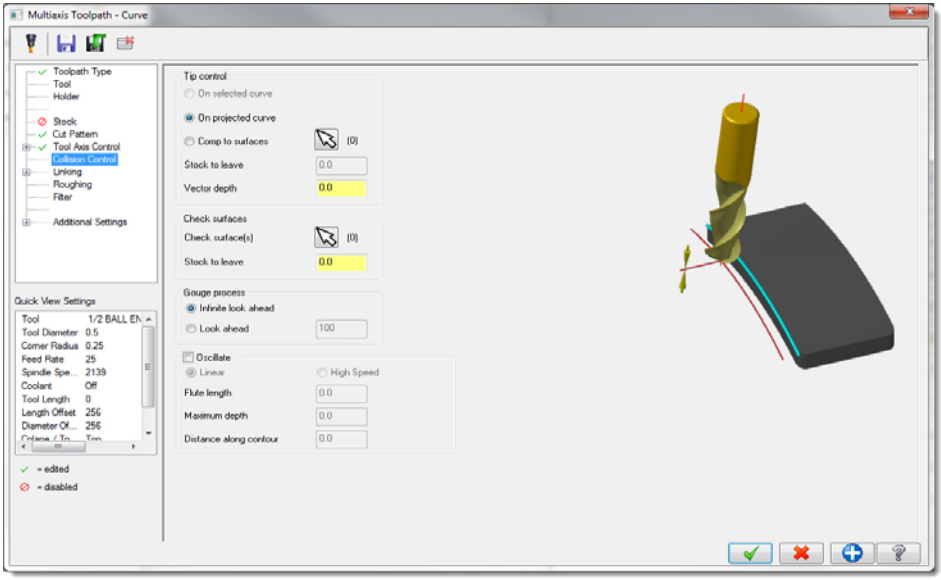
Tool Axis Control Page

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 a sample of the Tool Axis Control page for a multiaxis Curve toolpath.



Collision Control Page

Tool tip control does not have its own page like the other two controls. It can be found on the Collision Control page. Use the parameters to instruct Mastercam how the tip of the tool should be placed in relation to the cut pattern. Below is a sample of the Collision Control page for a multiaxis 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. Review them, and click Help if you want to know details about these pages. Continue on to Lesson 4 to begin creating a multiaxis Curve toolpath.

LESSON 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 Introduction to Mastercam tutorial before continuing if you are not familiar with these concepts.

Lesson 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:

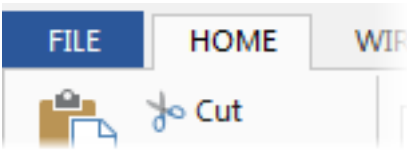
- Double-click Mastercam's desktop icon.

  Or

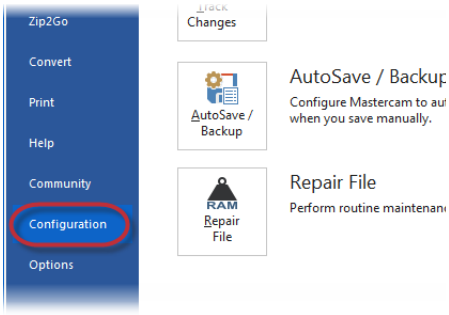
- Launch Mastercam from the Windows Start menu.

2 Select the default metric configuration file:

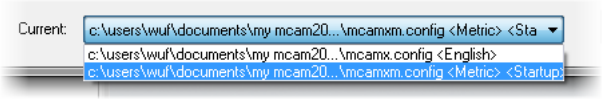
a Click the **File** tab.



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

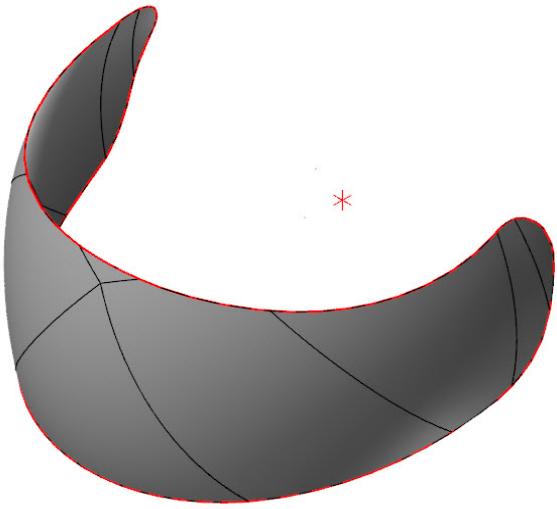


c Choose **...\mcamxm.config <Metric>** from the **Current** drop-down list.



d Click **OK**.

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



4 If not already loaded, select the default Mill metric machine definition.

NOTE: The default machine definition has rotary axes defined that allow multiaxis toolpaths. Load the machine definition of your choice as long as it supports 5-axis motion.

5 Set your graphics view to **Isometric**.

6 Activate shading if necessary.

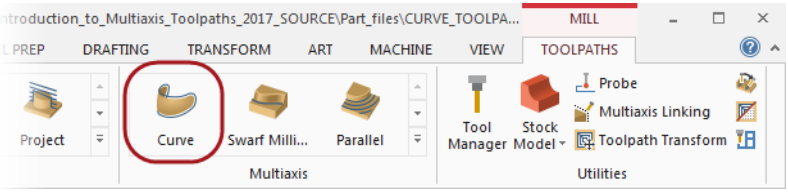
7 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button on the View tab.

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

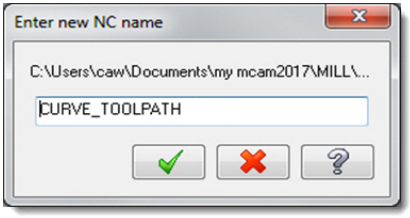
Exercise 2: Create a Multiaxis Curve Toolpath

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.

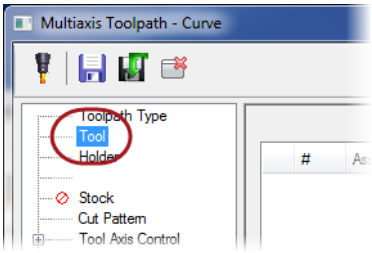
1 From the Toolpaths tab, choose **Multiaxis, Curve**.



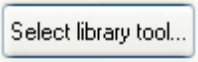
2 Click **OK** if prompted to enter a new NC file name.



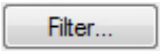
3 Select **Tool** from the tree view pane.



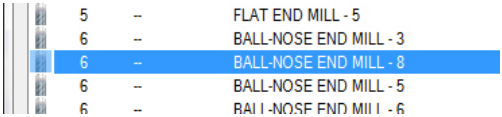
4 Click the **Select library tool** button. The tool library defined in the machine definition opens.



5 Click the **Filter** button, and set the filter to show only **Endmill2 Sphere** tools.



6 Select the 8mm ball endmill and click **OK**.

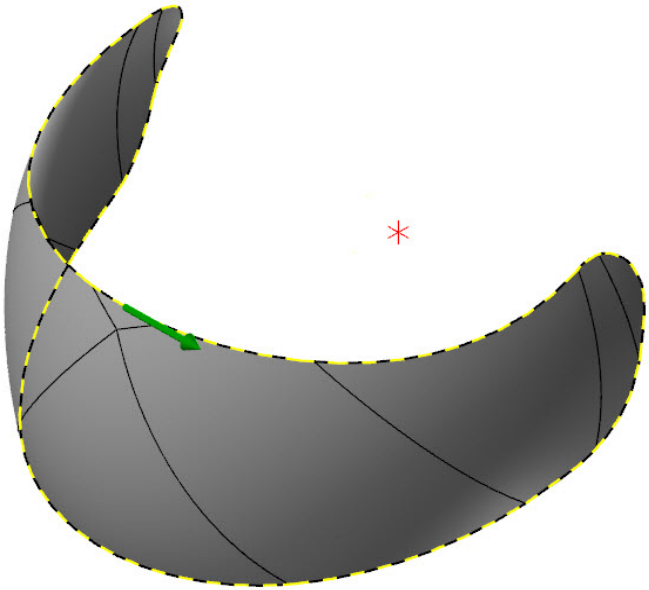


7 Select **Cut Pattern** from the tree view pane.

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

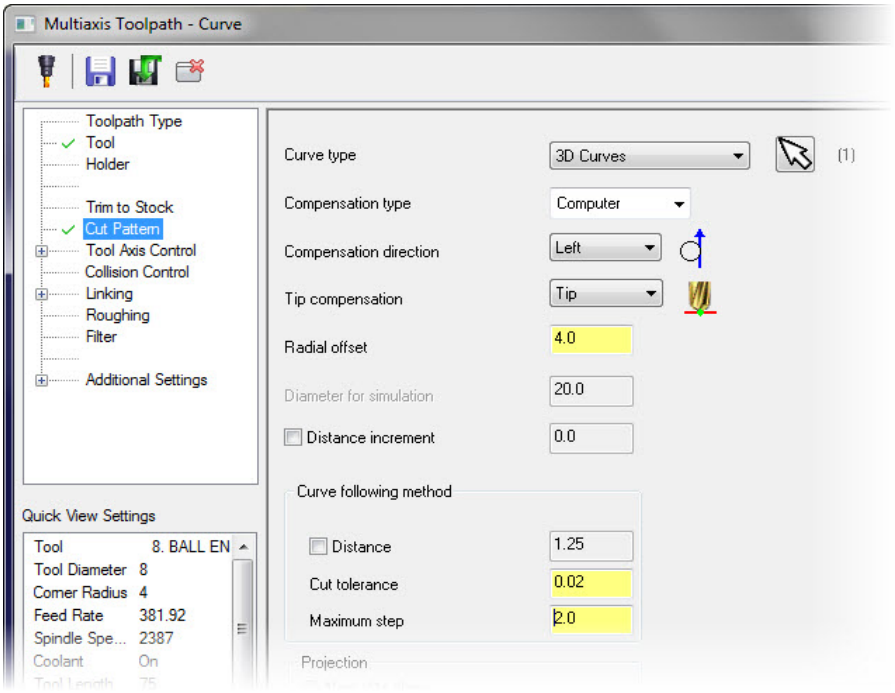
9 Click the **Select points** button next to 3D Curves to return to the graphics window for curve selection. The **Chaining** dialog box displays.

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.



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

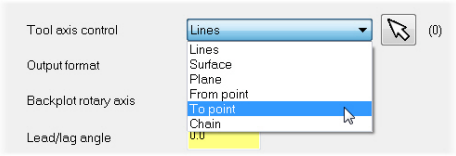
12 Set the remaining parameters as shown.



TIP: The radial offset value for this particular operation should equal the tool radius. Use this field in a similar manner as a stock to leave. A value equal to the tool radius is equivalent to zero stock to leave. Take advantage of this functionality for trimming operations.

13 Select **Tool Axis Control** from the tree view pane.

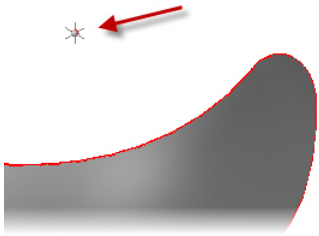
14 Choose **To point** from the Tool axis control drop-down selection.



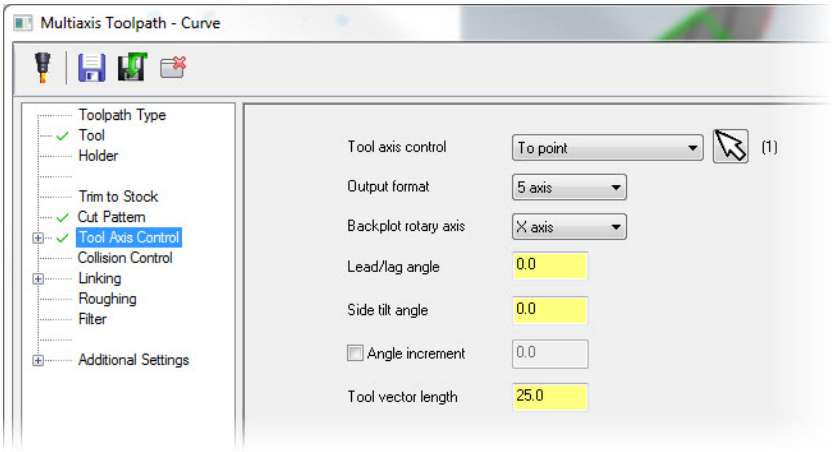
15 Click the **Select** button to return to the graphics window for point selection.

16 Select the point as indicated.

You return to the Tool Axis Control page.



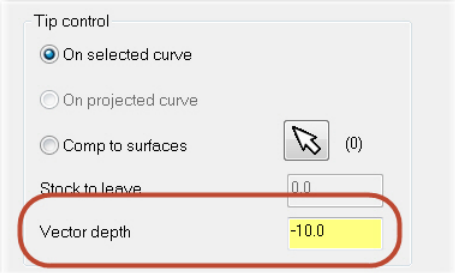
17 Set the remaining parameters as shown.



TIP: Click in a field to see a progressive image on the right side of the page. The image depicts what the field controls. Click **Help** for detailed information on each field.

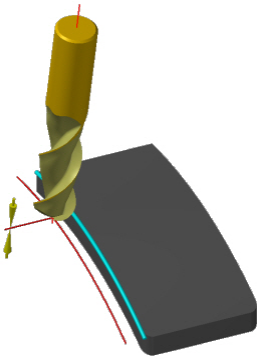
18 Select **Collision Control** from the tree view pane.

19 Set **Vector depth** to **-10.0**. The value allows the curve operation to trim the part with the side of the tool, avoiding the ball portion at the tip.

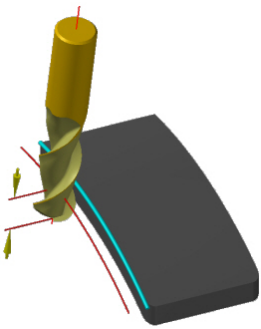


The images below show how the difference between vector depth values affects the toolpath.

Vector depth = 0.0



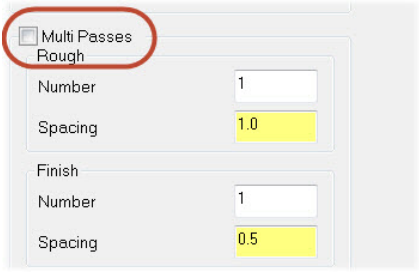
Vector depth = negative



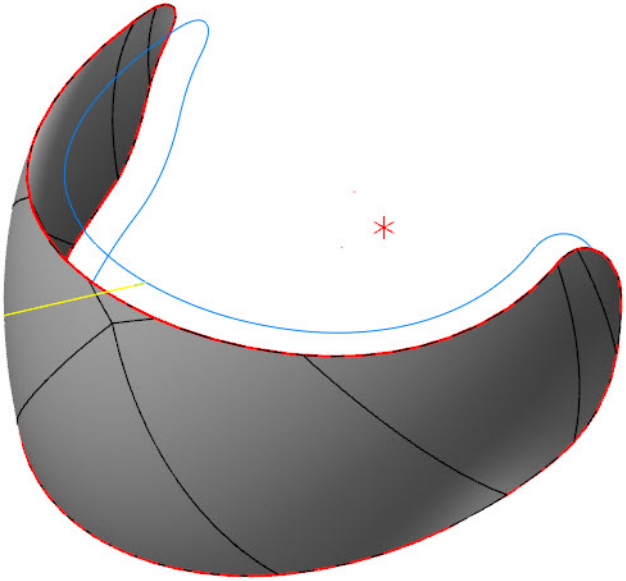
20 Select **Roughing** from the tree view pane.

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

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



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

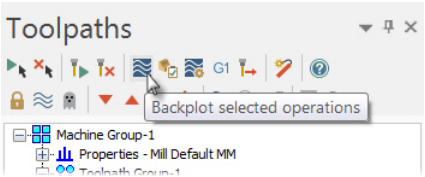


23 Save your part file.

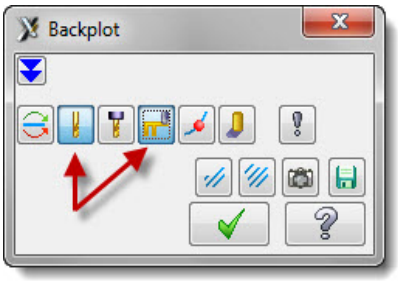
Exercise 3: Backplot the Operation

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

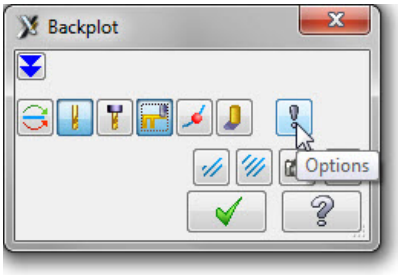
1 Click **Backplot selected operations** in the Toolpaths Manager.



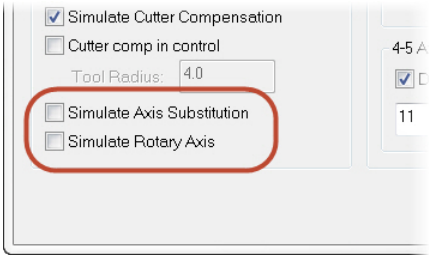
2 In the Backplot dialog box, select the **Display tool** and **Display rapid moves** buttons.



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



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



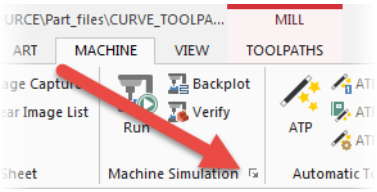
5 Click **Play** to begin the backplot of your toolpath.

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

Exercise 4: Machine Simulation

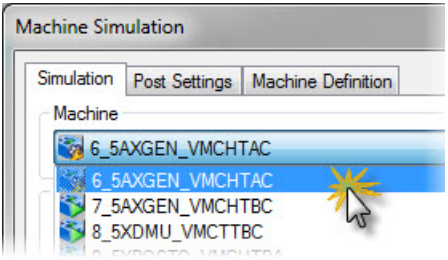
Machine simulation is the next step in validating your operations. The tool motion is shown on your part as it is positioned on your machine. Collisions between various machine components and the part are visually apparent, as well as reported in text on the screen. The following is a quick run-through of Machine Simulation. For a more detailed explanation, review **Help**, as well as the Machine Simulation tutorial.

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

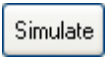


2 Select **6_5AXGEN_VMCHTAC** from the drop-down list of machines.

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 simulation playback buttons to review the tool motion contained in your operations. (If you do not see the part, click the **Workpiece** button, which is located in the **Simulation** tab's **Visibility** drop-down.).)

5 Exit machine simulation to return to the Mastercam interface.

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. Proceed to Lesson 5 to explore these strategies.

LESSON 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 lesson to gain exposure with two additional tool axis control options for a multiaxis Curve toolpath.

Lesson Goals

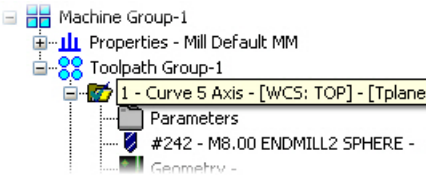
- Open a previously saved part.
- Edit the operation's parameters, specifically tool axis control.
- Generate the toolpath and observe the tool motion.

Exercise 1: Preparing the Part

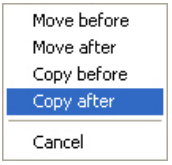
1. Use **File, Open** to open the part file `Curve_Toolpath_Lesson_5`, which was provided with the tutorial.
2. Set your graphics view to Isometric.
3. Activate shading if necessary.
4. Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.
5. Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Copy and Edit an Operation

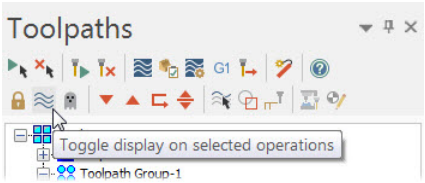
1. Right-click on the first operation and hold the mouse button down.
2. Drag the operation to a position farther down the Toolpaths Manager.



3. Release the right mouse button and select **Copy after**.



4 Select operation 1 and turn off the toolpath display. Press [**T**] or click **Toggle display on selected operations**.

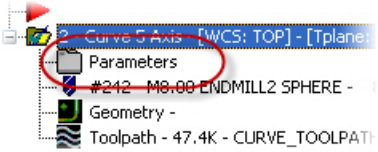


The visible toolpath belongs to operation 2.

5 Use the Levels Manager to make level **7:Chain** visible.

The geometry on level 7 is used for tool axis control in this exercise.

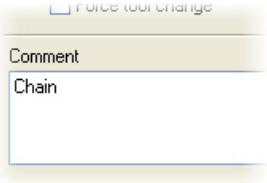
6 In Toolpaths Manager, click **Parameters** under operation 2. The Multiaxis Toolpath - Curve dialog box opens.



7 Select **Tool** from the tree view pane.

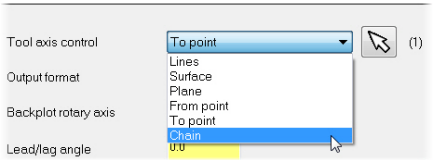
8 Enter **Chain** in the **Comment** box.

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



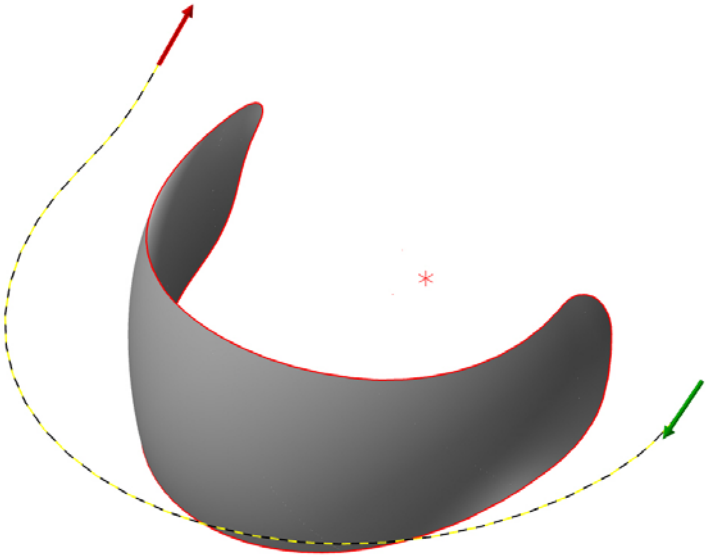
9 Select **Tool Axis Control** from the tree view pane.

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

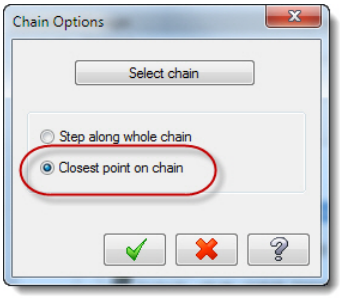


11 Click **Select** to open the Chaining dialog box.

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

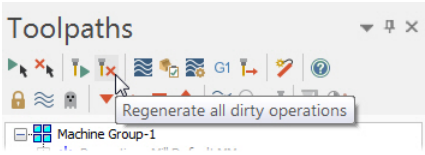


13 In the Chain Options dialog box, select **Closest point on chain**, and click **OK** to return to the Tool Axis Control page.



14 Click **OK** to accept the parameter changes.

15 Click **Regenerate all dirty operations** in the Toolpaths Manager.

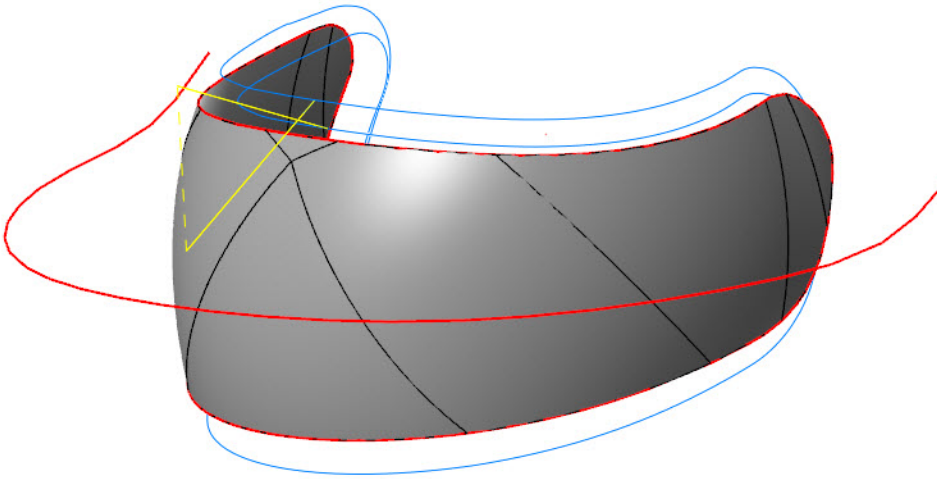


16 Backplot the operation to observe the tool motion.

17 Save your part file.

At this point in the lesson, 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 around the entire part shown below. 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.



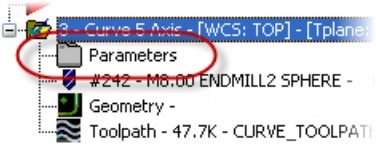
Exercise 3: Copy and Edit a Second Operation

1. Right-click on the second operation 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 operation 2 and turn off the toolpath display, by pressing [**T**] or clicking the **Toggle display on selected operations** button.

   The display for operations 1 and 2 are now both off.

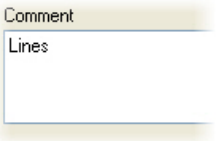
5 Use the Levels Manager to make level **8:Tool axis lines** visible. Turn off the display of levels 1, 6, and 7.

6 In Toolpaths Manager, click **Parameters** under operation 3. The Multiaxis Toolpath - Curve dialog box opens.



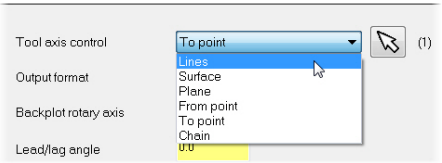
7 Select **Tool** from the tree view pane.

8 Enter **Lines** in the **Comment** box.



9 Select **Tool Axis Control** from the tree view pane.

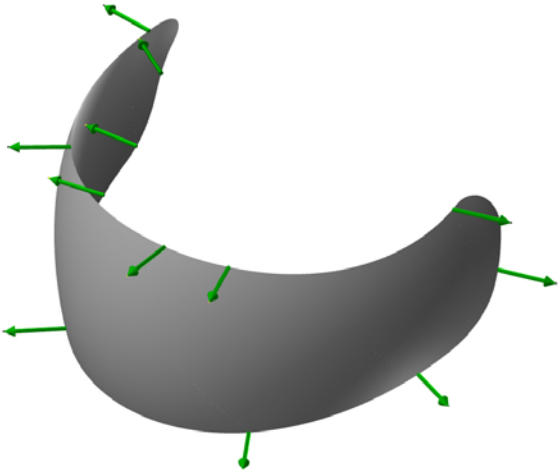
10 Choose **Lines** from the **Tool axis control** drop-down selection.



11 Click **Select** to open the Lines Tool Axis Control dialog box.

12 Click the **Window select lines** button to select lines for tool axis control.

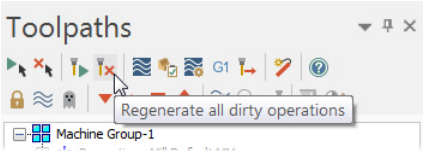
13 Draw a selection window around the entire part. Only lines are selected.



14 Click **OK** in the Lines Tool Axis Control dialog box. You return to the Tool Axis Control page.

15 Click **OK** to accept the parameter changes.

16 Click the **Regenerate all dirty operations** button in the Toolpaths Manager.



17 Backplot the operation to observe the tool motion.

18 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. Proceed to Lesson 6 where you will create two Multiaxis Drill operations using two of the tool axis control methods available for that toolpath type.

LESSON 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.

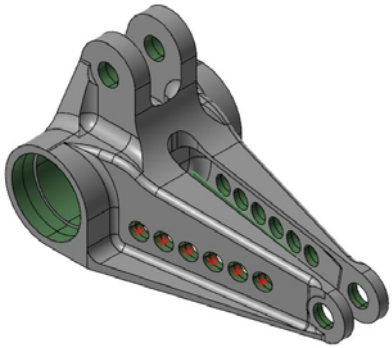
Lesson Goals

- Open a part file and assign a machine definition.
- Create some Multiaxis Drill operations.
- Backplot the operations.

Exercise 1: Getting Started with Toolpath Creation

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

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



2 If not loaded, select the default Mill metric machine definition.

NOTE: The default machine definition has rotary axes defined that allow multiaxis toolpaths. Load the machine definition of your choice as long as it supports 5-axis motion.

3 Set your graphics view to **Isometric**.

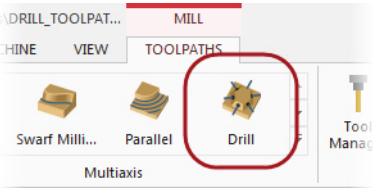
4 Activate shading if necessary.

5 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

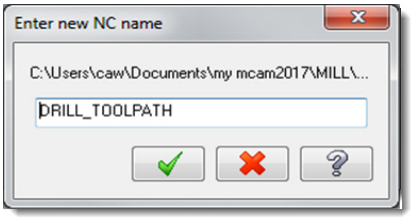
6 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Create a Multiaxis Drill Toolpath

1 From the Toolpaths tab, choose **Multiaxis, Drill**.

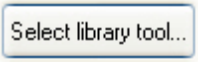


2 Click **OK** if prompted to enter new NC file name.



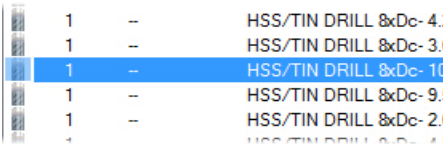
3 Select **Tool** from the tree view pane.

4 Click the **Select library tool** button. The library defined in the machine definition opens.



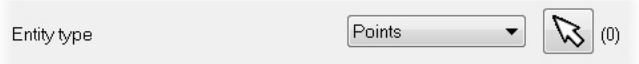
5 Select the 10mm drill, and click **OK**.

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

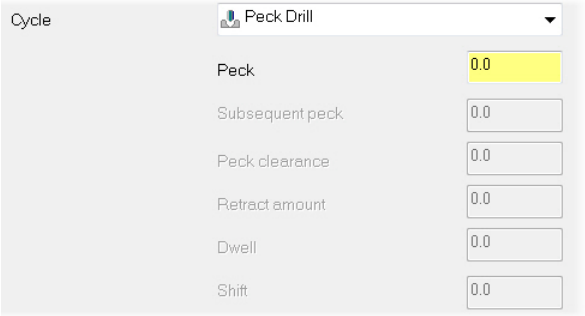


6 Select **Cut Pattern** from the tree view pane.

7 Choose **Points** for the entity type.



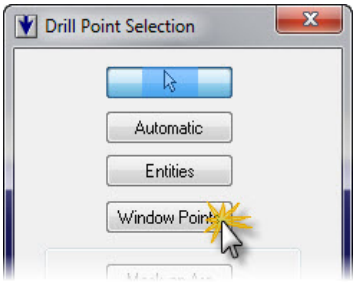
8 Select **Peck Drill** for the cycle and set the subsequent parameters.



Cycle selection determines the available parameters. Choose different cycles to review the parameters.

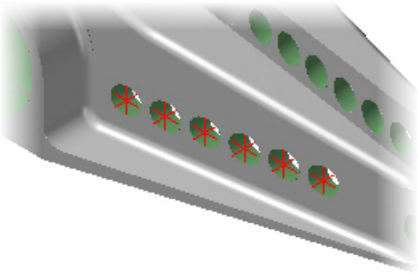
9 Click **Select points** to return to the graphics window for point selection.

10 Select **Window Points** in the Drill Point Selection dialog box.



11 Create a window around the visible points.

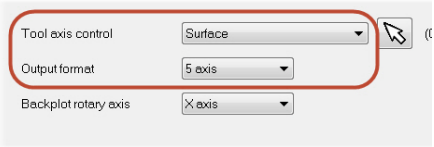
The point geometry is on level 1.



12 Click **OK** in the Drill Point Selection dialog box to return to the Cut Pattern page.

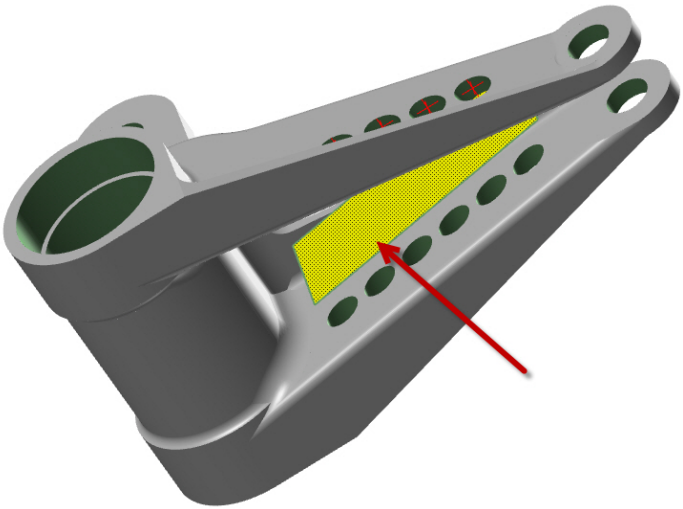
13 Select **Tool Axis Control** from the tree view pane.

14 Choose **Surface** for **Tool axis control** and **5 axis** for **Output format**.



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.



The surface is on level 5. Use the Levels Manager to make it visible if necessary.

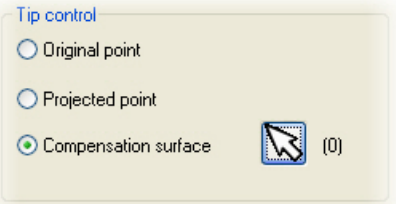
TIP: Manipulating the visible levels makes the selection process easier and less susceptible to error.

17 Press [**Enter**] or click **End Selection** when done.

18 Select **Collision Control** from the tree view pane.

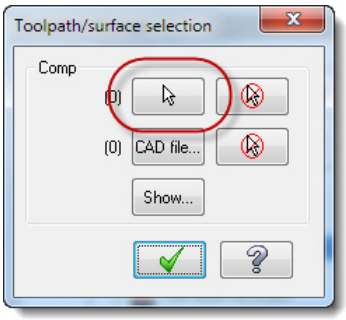
19 Set **Tip control** to **Compensation surface**.

Click **Help** for an explanation of the different tip control options.



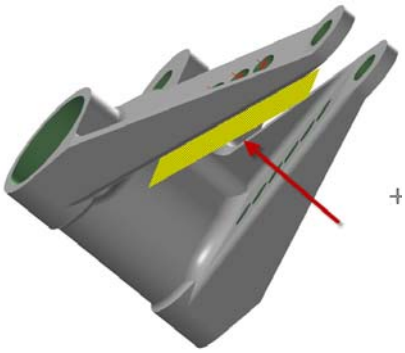
20 Click the **Select comp surfaces** button next to **Compensation surface**. The Toolpath/surface selection dialog box opens.

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



22 Use the Levels Manager to make level 4 visible.

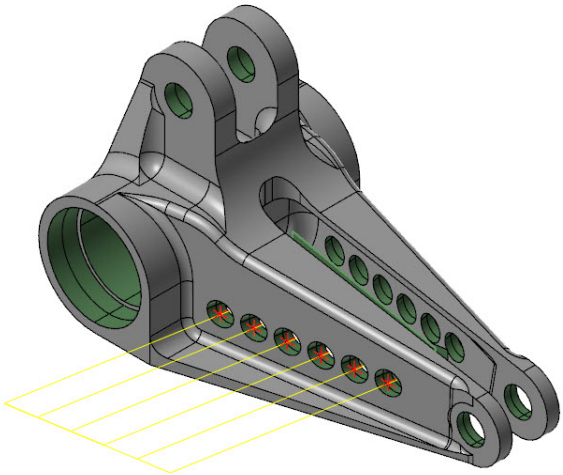
23 Select the surface as shown. Rotate the geometry as needed to allow selection.



24 Press **[Enter]** or click **End Selection** when done.

25 Click **OK** in the Toolpath/surface selection dialog box.

26 Click **OK** to generate the multiaxis drill toolpath on the selected points.



27 Save your part file.

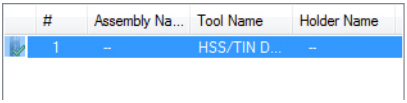
Exercise 3: Add a Second Multiaxis Drill Operation

1 In Levels Manager, turn off level 1, and make level 3 visible.

2 From the Toolpaths tab, choose **Multiaxis, Drill**.

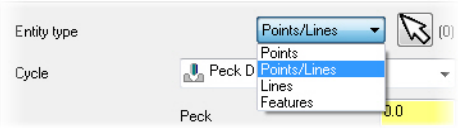
3 Select **Tool** from the tree view pane.

4 Select the 10mm drill shown in the tool list.



5 Select **Cut Pattern** from the Tree View pane.

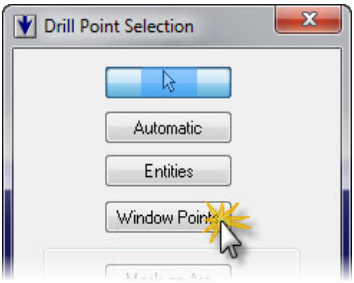
6 Choose **Points/Lines** for the entity type.



7 The **Cycle** should still be **Peck Drill**. Set it if necessary.

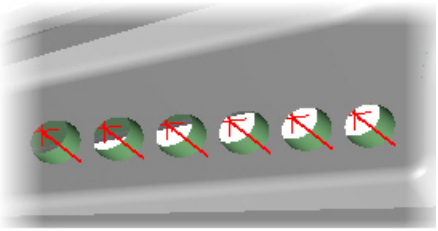
8 Click **Select points** to return to the graphics window for point selection.

9 Select **Window Points** in the Drill Point Selection dialog box.



10 Draw a window around the visible points.

Change the view to Back and rotate the geometry as needed to bring the points/lines into view. The points for this operation are on the back side of the part.

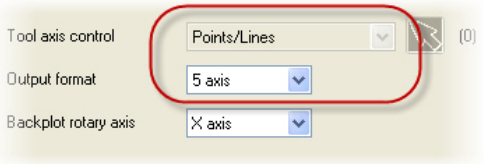


NOTE: It is not necessary to select the lines, even though the entity type is set to Points/Lines. The lines are automatically selected as long as they end on the point.

11 Click **OK** in the Drill Point Selection dialog box to return to the Cut Pattern page.

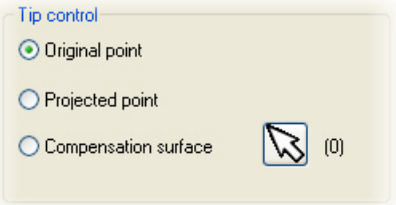
12 Select **Tool Axis Control** from the tree view pane.

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.



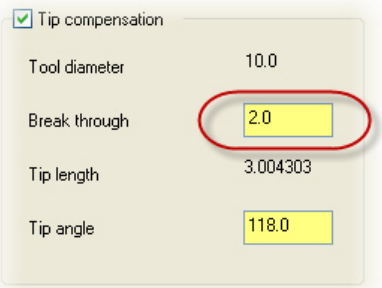
14 Select **Collision Control** from the tree view pane.

15 Set **Tip control** to **Original point**.

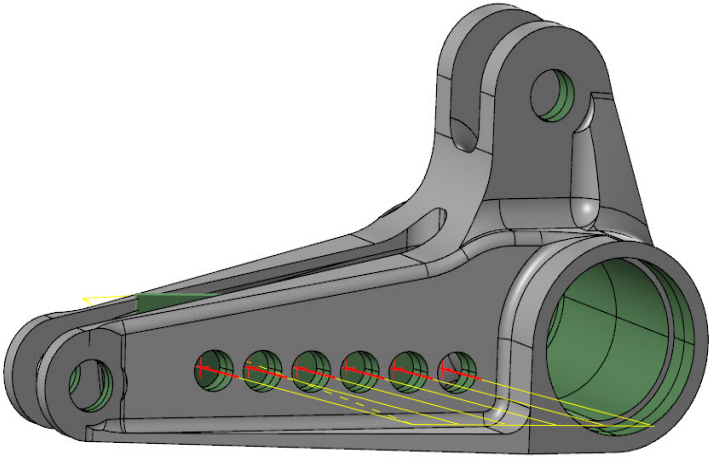


16 Activate **Tip compensation** by selecting the checkbox. Set **Break through** to 2.0.

The break-through value determines how far the full diameter of the drill penetrates beyond the selected tip control geometry.



17 Click **OK** to generate the multiaxis drill toolpath on the selected points.

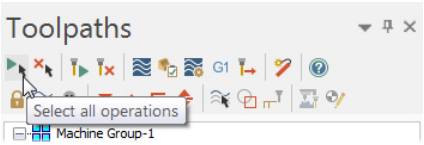


18 Save your part file.

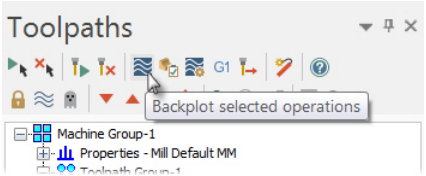
Exercise 4: Backplot the operations

1 Set your graphics view to Isometric.

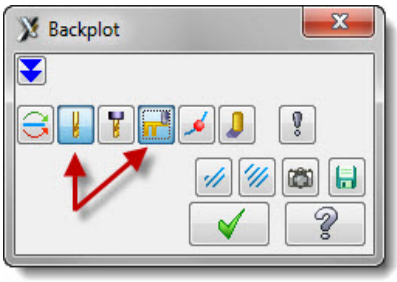
2 In the Toolpaths Manager, click **Select all operations**.



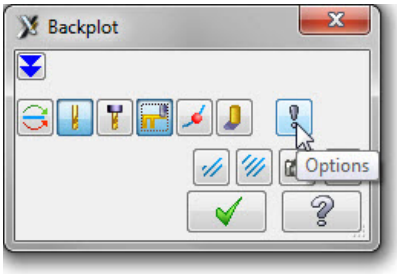
3 Click **Backplot selected operations** in the Toolpaths Manager.



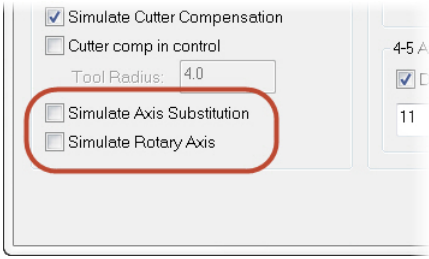
4 In the Backplot dialog box, select the **Display tool** and **Display rapid moves** buttons.



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 Introduction to Multiaxis Toolpaths tutorial. Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. The Mastercam tutorial series is in continual development, and we will add modules as we complete them. Visit our website, or select **Help, Tutorials** from the **File** tab to see the latest publications.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

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

Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledge base at kb.mastercam.com.

To register, select **Community**, **Link Account** from the **File** tab, and follow the instructions.

Mastercam Documentation

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

- What's New in Mastercam 2017
- Mastercam 2017 Installation Guide
- Mastercam 2017 Administrator Guide
- Mastercam 2017 Transition Guide
- Mastercam 2017 Quick Reference Card
- Mastercam 2017 ReadMe

Contact Us

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

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

CNC software, inc.

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