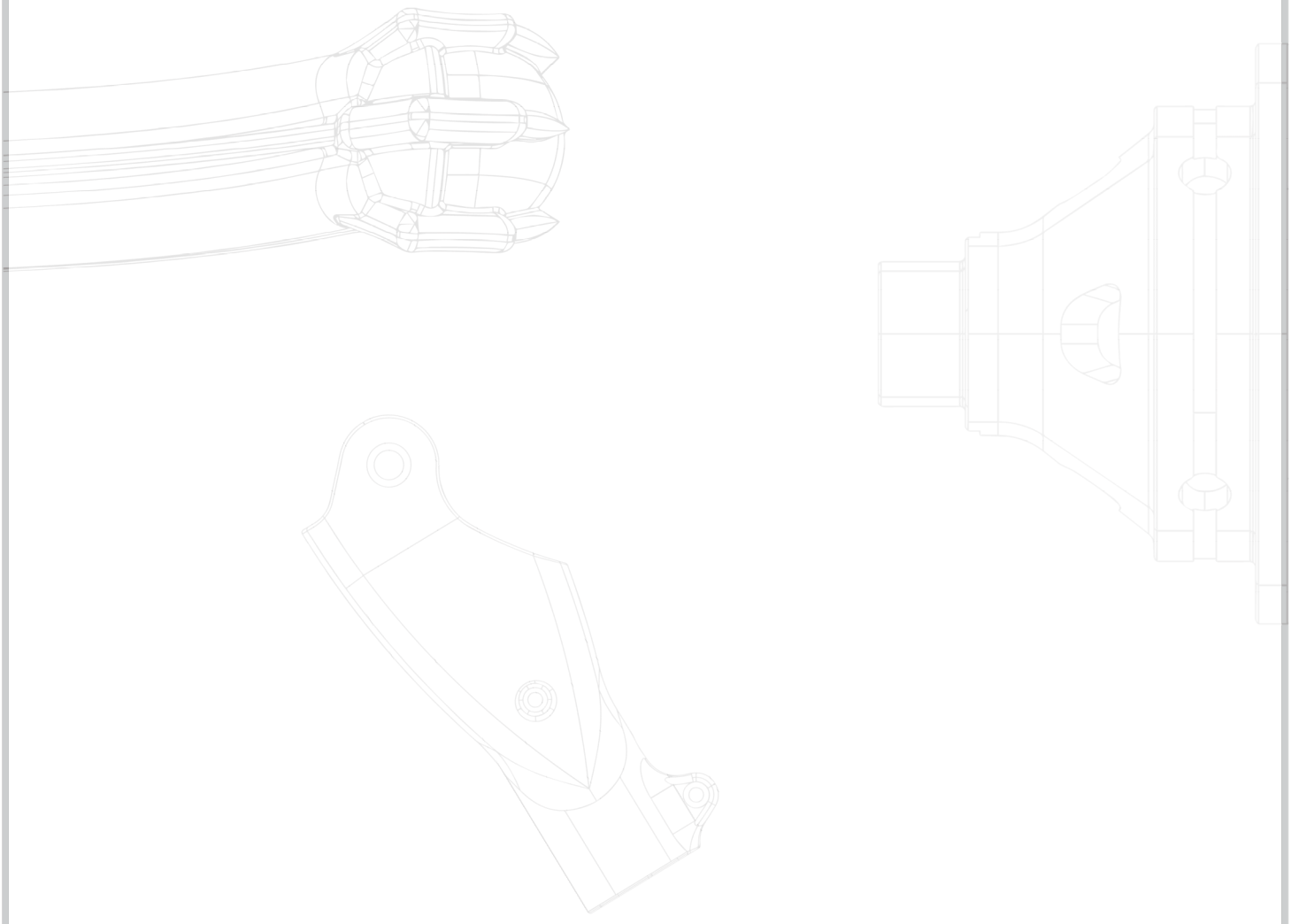
MASTERCAM DYNAMIC MILLING TUTORIAL

March 2020



POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

MASTERCAM DYNAMIC MILLING TUTORIAL

March 2020

Software: Mastercam 2021

Terms of Use

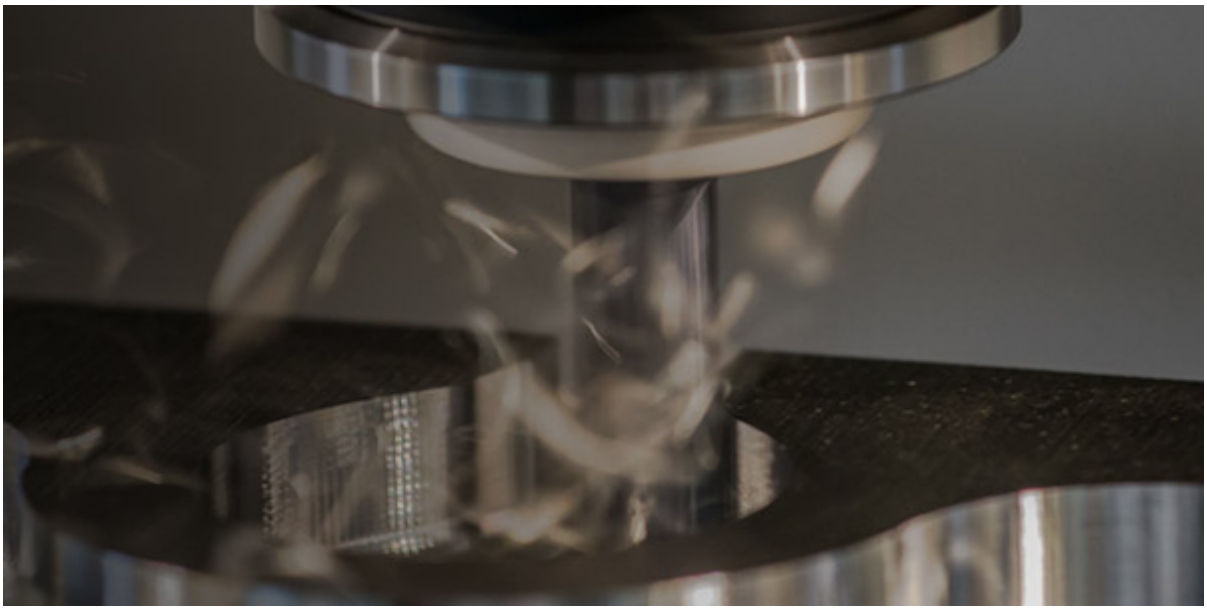
Be sure you have the latest information!

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

TABLE OF CONTENTS

INTRODUCTION

Mastercam's Dynamic Motion toolpaths deliver powerful benefits that help you get the most from any machine.



These 2D and 3D high-speed dynamic milling toolpaths utilize the entire flute length of their cutting tools to achieve greater efficiency in milling. They are designed to maximize material removal while minimizing tool wear.

Additional benefits you gain by using high-speed dynamic milling toolpaths include:

- Tool burial avoidance
- Minimum heat buildup
- Better chip evacuation
- Extended tool life

This tutorial introduces you to Mastercam's intelligent, application-specific, 2D and 3D high-speed dynamic milling toolpaths.

Tutorial Goals

- Learn the benefits and uses of the dynamic toolpath types
- Create basic dynamic toolpaths
- Create stock models
- Learn to use other Mastercam toolpath utilities

> **NOTE**
> You must have a 3D license to complete the Dynamic OptiRough chapter.

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: 5 hours

General Tutorial Requirements

All Mastercam 2021 tutorials have the following general requirements:

- You must be comfortable using the Windows® operation system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be access from the Mastercam 2021 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.

The Chaining Dialog Box

Starting with Mastercam 2020, the **Chaining** dialog box displays as one of two versions: the **Wireframe Chaining** dialog box or the **Solid Chaining** dialog box. When this tutorial refers to these dialog boxes in general, it will use the term "chaining dialog box," with no uppercase or bolding.

DYNAMIC TOOLPATHS OVERVIEW

The following toolpaths use Dynamic Motion technology:

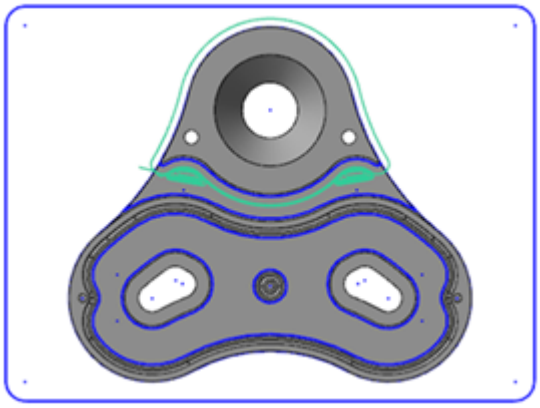
- Dynamic Contour
- Face
- Dynamic Mill
- Peel Mill
- Dynamic OptiRough

Each toolpath has benefits that make it unique.

Dynamic Toolpath Types

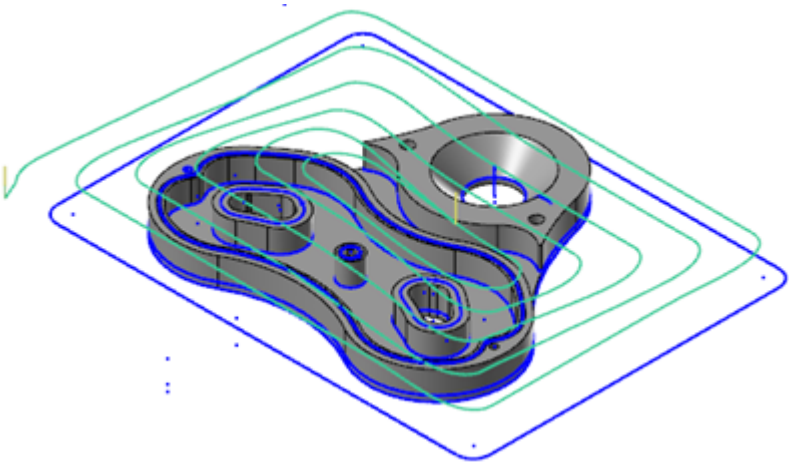
Dynamic Contour

The Dynamic Contour toolpath creates a familiar contour operation using dynamic motion and specialized options found only in dynamic toolpaths. The dynamic motion prevents tool burial and binding in small radii corners. Additionally, you can specify how Mastercam machines material left on the walls.



Face

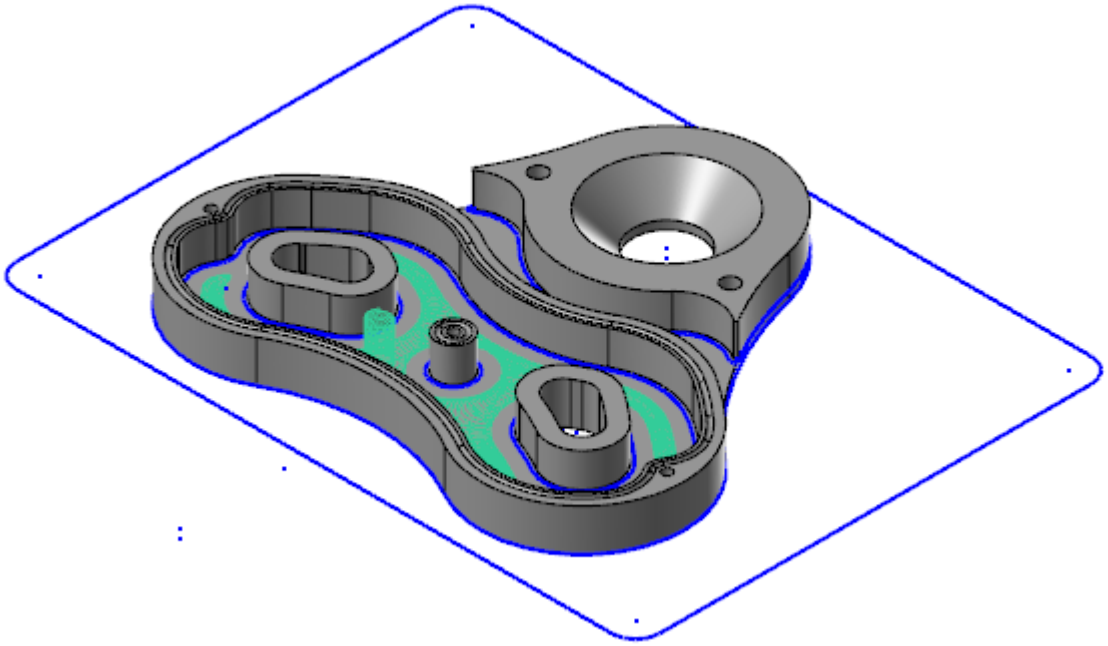
The Face toolpath, when the **Style** is set to **Dynamic**, quickly cleans stock from the top of a part and creates an even surface for future operations. You can base the toolpath on either chained geometry or on the current stock setup.



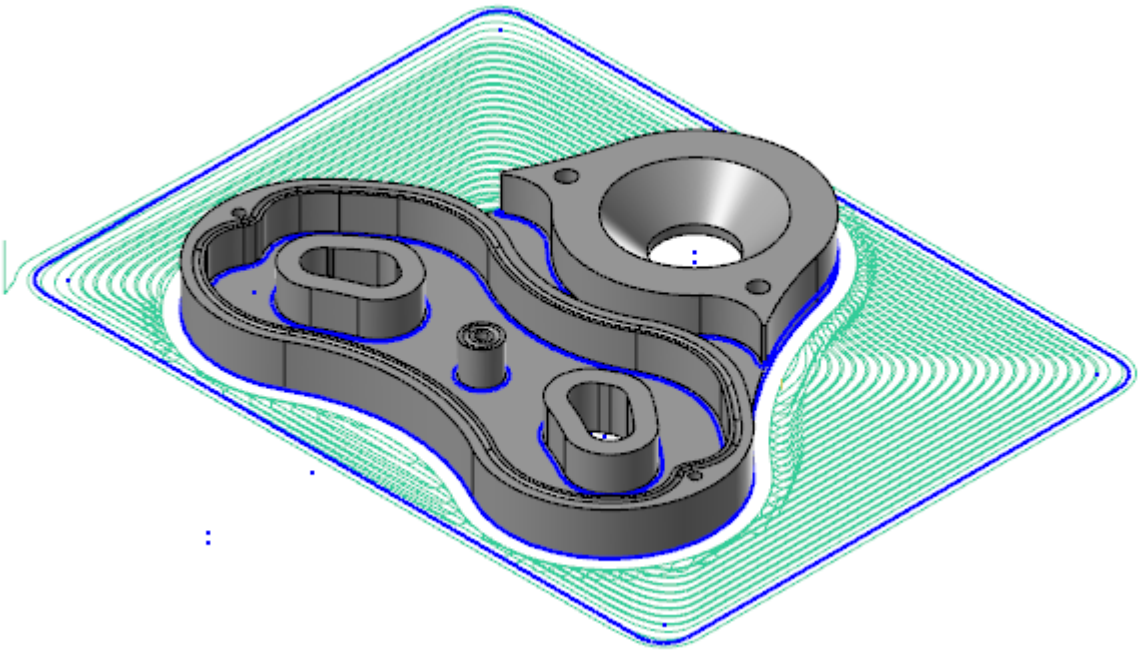
Dynamic Mill

The Dynamic Mill toolpath machines pockets, leftover material, standing bosses, or cores. By setting the **Machining region strategy** to **Stay inside** or **From outside**, you can create a pocket toolpath that uses dynamic motion or a facing operation that has island avoidance capabilities.

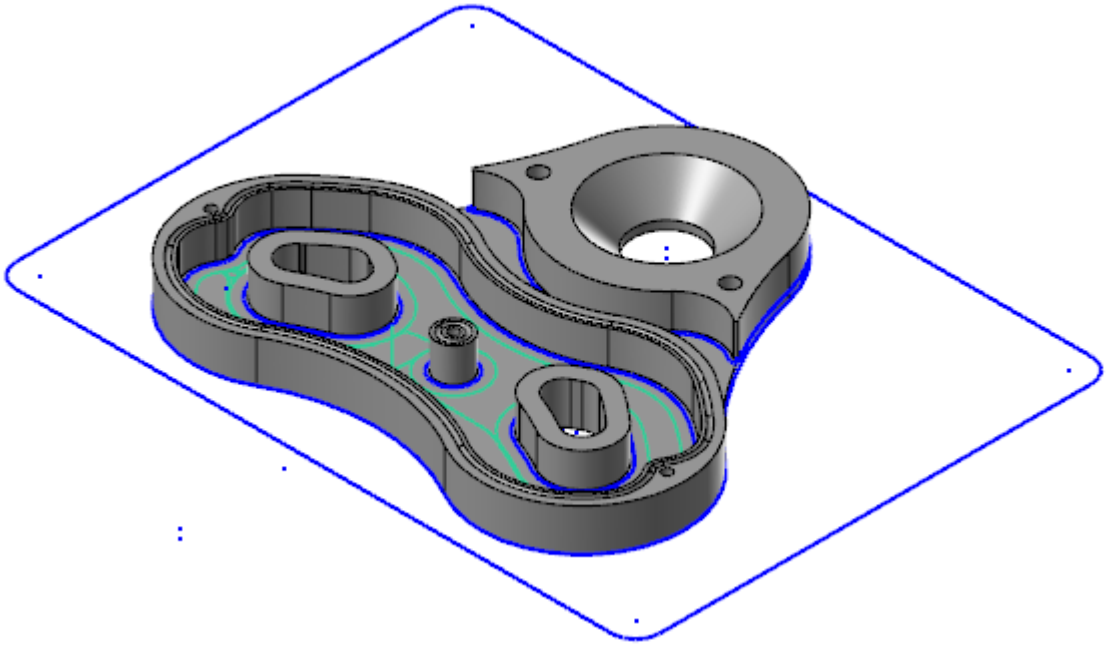
By selecting the **Stay inside** strategy, Dynamic Mill becomes a pocket toolpath that uses dynamic motion. You select a boundary, areas to avoid, and set tooling, parameters, entry method, and linking values to generate the toolpath. Material is removed inside-to-outside in a highly efficient manner.



By selecting the **From outside** strategy, Dynamic Mill becomes a facing operation that has island avoidance capabilities. This movement creates a toolpath that is allowed to move outside the selected material and removes material outside-to-inside.

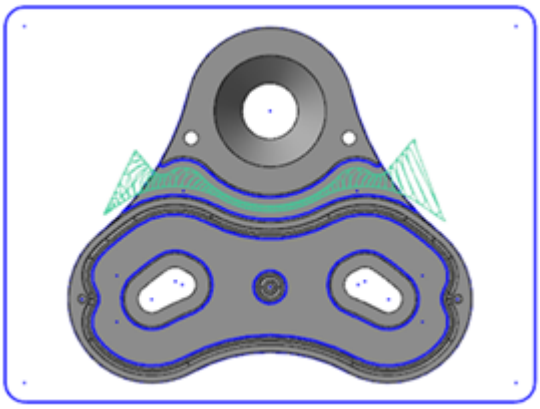


By enabling **Rest Material** on the **Stock** page, the Dynamic Mill toolpath uses dynamic motion to remove material left by previous operations. The Dynamic Mill operation calculates how much material to remove based on previous operations or previous roughing tools. Only unmachined areas are processed for a Dynamic Mill operation.



Peel Mill

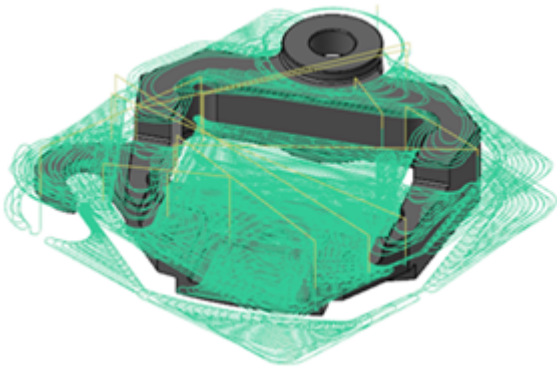
Peel Mill, with the **Cutting strategy** set to **Dynamic Peel**, allows for efficient milling between selected chains. It uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material. For a single chain, you define the width of the cut. Otherwise, the width is defined by the area between two contours.



Dynamic OptiRough

The Dynamic OptiRough toolpath is currently the only 3D toolpath that uses Mastercam's Dynamic Motion technology. It supports cutters capable of machining very large depths of cut. A single Dynamic OptiRough toolpath can cut material in two directions: on stepdowns (-Z) and stepups (+Z). This bi-directional cutting strategy removes the maximum amount of material with the minimum amount of stepdowns, significantly reducing cycle times.

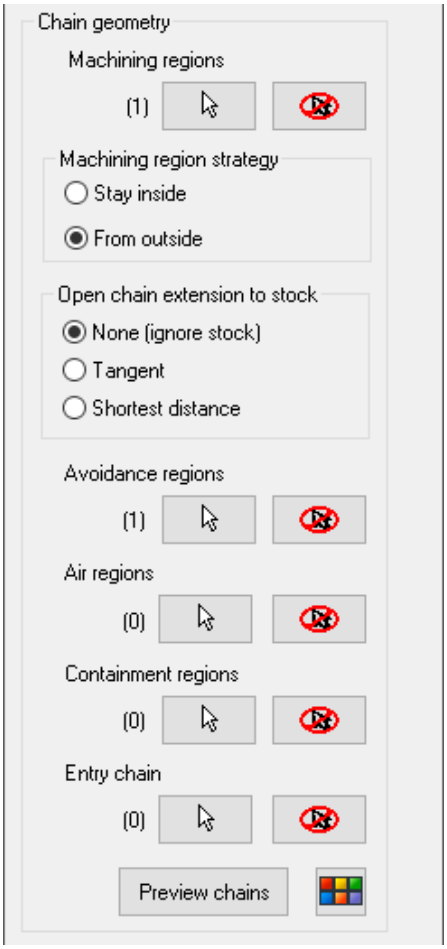
Similar to Dynamic Mill, you can select **From outside** or **Stay inside** when machining. However, instead of selecting regions, you select machining geometry, avoidance geometry, and containment boundaries.



Dynamic Toolpath Selection

The Dynamic toolpaths, especially the Dynamic Mill toolpath, require a unique way of understanding chain selection. Dynamic Contour does not require a closed chain to calculate a toolpath. All geometry selected is cut by the tool motion. Peel Mill requires at least one open chain and no closed chains.

Dynamic Mill allows you to have even more control over your chaining by allowing you to select multiple kinds of regions with one toolpath. The types of regions are listed below:



- **Machining regions**: Areas to be machined.
- **Avoidance regions**: Areas to be avoided during machining. Multiple avoidance regions can be selected.
- **Air regions**: Areas that contain no material and allow the tool to travel through it when machining.

- **Containment regions**: Areas that limit tool motion.
- **Entry chain**: The point where the tool enters the part or a custom chain for the entry motion to follow.

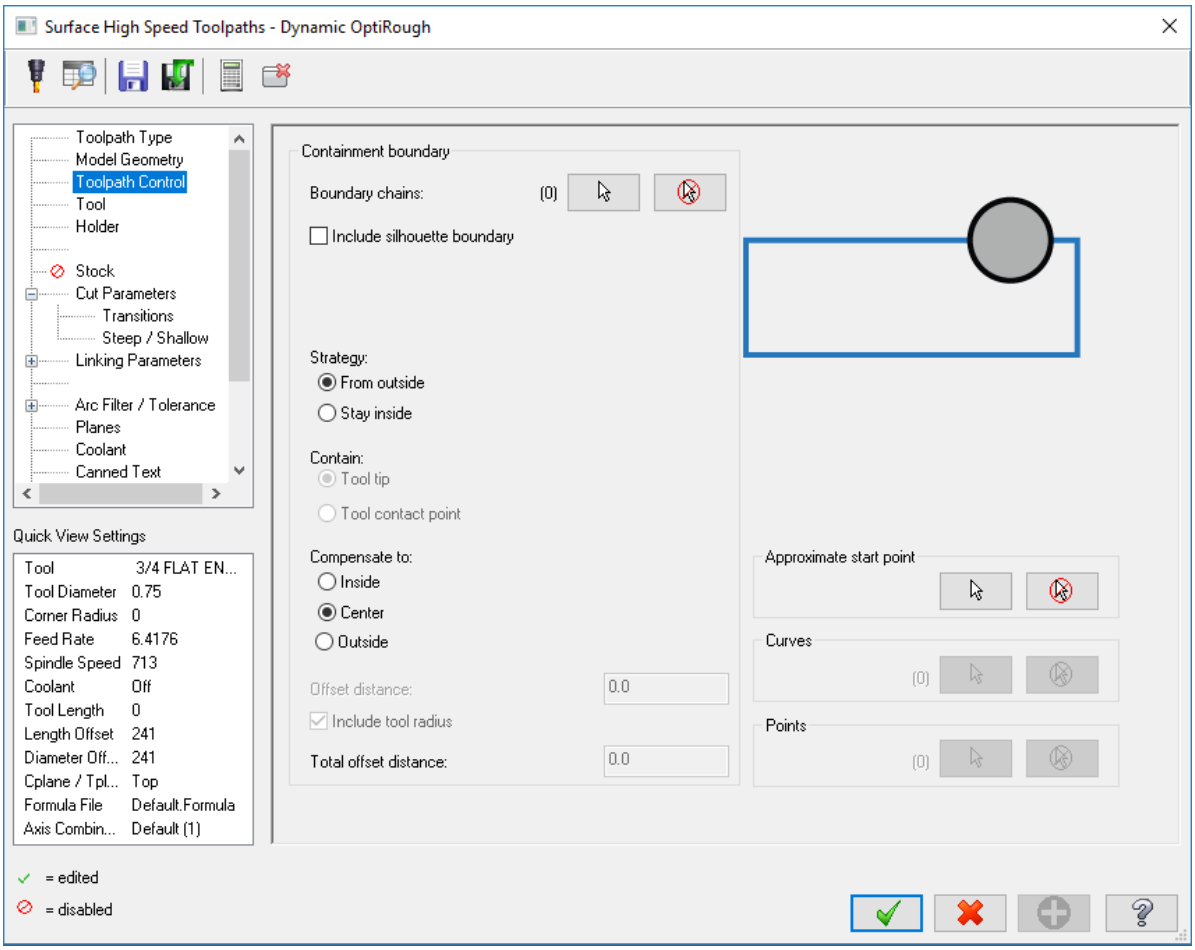
If the machining regions are open chains, you can extend the open chain to stock with the **Open chain extension to stock** options. You can also close the open chains with open **Air regions**, connecting the start and end of the open **Machining regions**. If no open chains are selected, and no **Open chain extension to stock** options are selected, they are automatically closed with a straight line.

For more information, view Mastercam's **Help**.

As the only 3D Dynamic toolpath, Dynamic OptiRough uses a different selection method than the other Dynamic toolpaths. When creating a Dynamic OptiRough toolpath, you are brought to the **Model Geometry** page in the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box. You then have the following options:

- **Machining Geometry**: Geometry to be machined. You can have multiple machining groups.
- **Avoidance Geometry**: Geometry to be avoided. You can have multiple avoidance groups.
- For each geometry group, you can assign variable stock-to-leave values on its walls and floors.

You can then use the **Toolpath Control** page to set containment parameters, which control the tool's position around the boundary of your part.

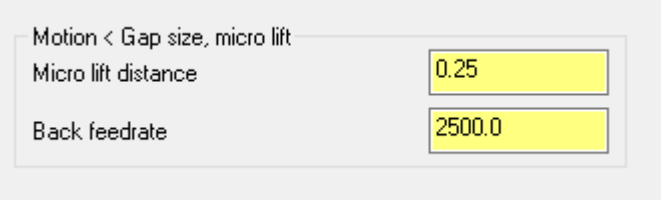


Dynamic Milling Parameters

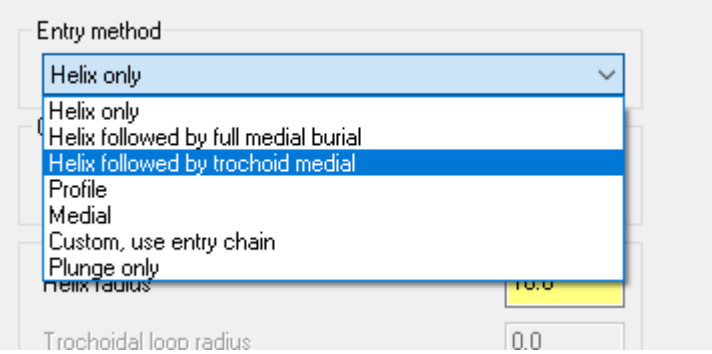
Several parameter options separate the Dynamic operations from their standard counterparts. These parameters are on the **Cut Parameters**, **Entry Motion**, and **Contour Wall** pages.

- Micro lifts
- Entry methods
- Entry feeds/speeds
- Contour wall
- Approach distance

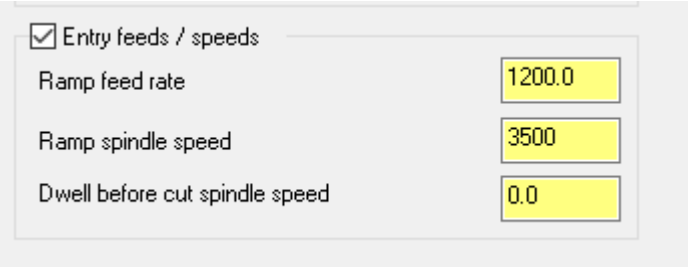
Micro lift options allow the tool to rise above the floor of the part for the portion of the toolpath that is not removing material. Benefits of the micro lift include chip clearing and avoiding excess tool heat. The feed rate is controllable for the back move as well.



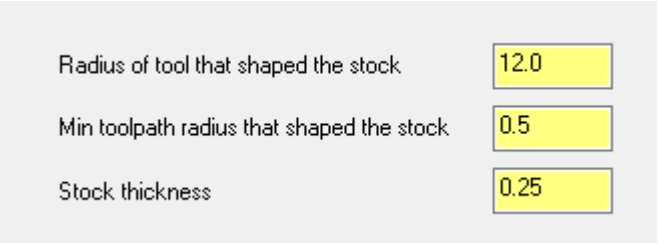
Entry methods provide several ways to enter the material for a Dynamic Mill toolpath. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's Help contains detailed information on each method.



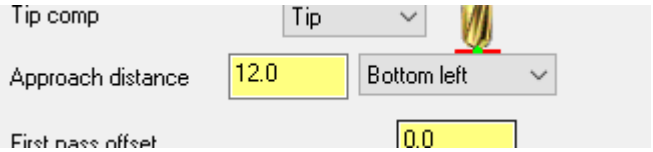
Use Entry feeds and speeds to avoid placing excessive load on the tool upon entry. Set a short dwell interval after entry to allow the spindle to reach the necessary speed before beginning material removal.



Contour wall parameters provide input for remaining stock and previous tool information. The values entered help Mastercam to calculate the most efficient material removal strategy when machining part walls.



The **Approach distance** parameter allows you to add a specified incremental distance to the beginning of the toolpath's first cut.



Dynamic toolpaths, which are part of the group of 2D and 3D High Speed Toolpaths (HST), create highly efficient motion based on selected chains and surfaces. The dynamic toolpaths are similar to standard Pocket, Contour, and Area Roughing toolpaths, but with the stated enhancements.

Creating and reviewing the motion of dynamic toolpaths, which you do in the rest of this tutorial, is the best way to learn about the benefits and uses.

CHAPTER 1

THE DYNAMIC MILL TOOLPATH

The Dynamic Mill toolpath machines pockets, open pocket shapes and standing cores, or material left from previous operations.

In this chapter, you create two Dynamic Mill toolpaths. The first one clears material around the outside of the part. The second Dynamic Mill toolpath uses the previous one as a source operation for removing the remaining material on the top of the part.

Goals

- Set up stock
- Create two Dynamic Mill toolpaths
- Preview region chains
- Verify the created toolpaths

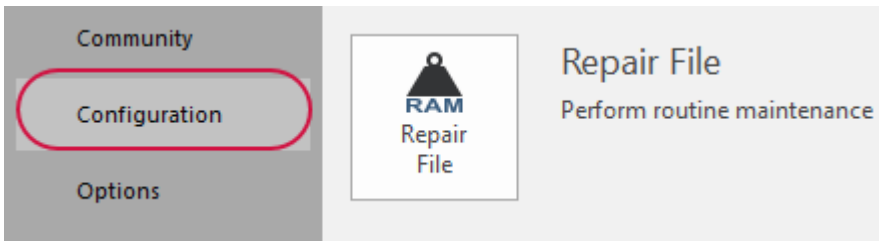
Exercise 1: Opening Mastercam

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

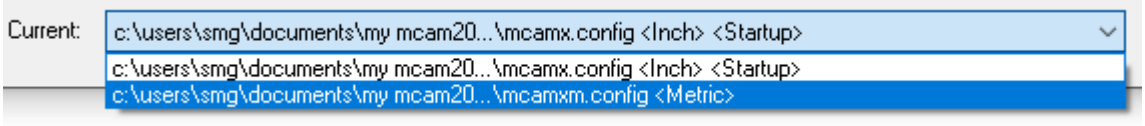
    

    OR

    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.

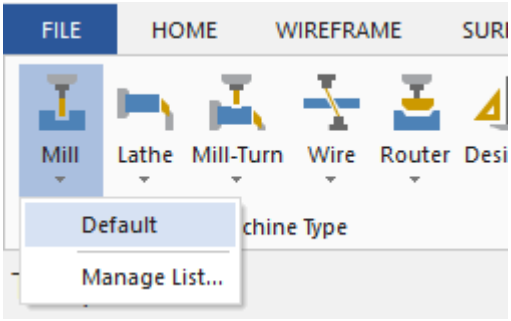


d. Click **OK**.

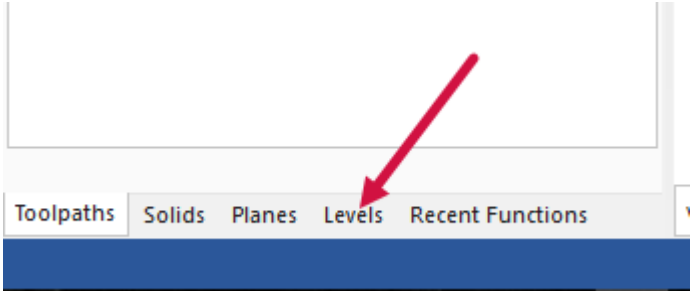
Exercise 2: Setting up the stock

In this exercise, you set up stock that is used for the first toolpath.

1. Open the part file, DynamicMill, provided with this tutorial.
2. Save the part as DynamicMill-xxx, replacing xxx with your initials.
3. On the **Machine** tab, select **Mill**, **Default.**

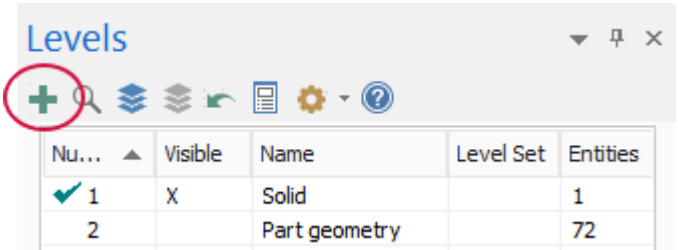


4. Select the Levels Manager, located in the same area as the Toolpaths Manager. If it is not displayed, select **Levels** on the **View** tab to toggle the display, or press [**Alt+Z**].



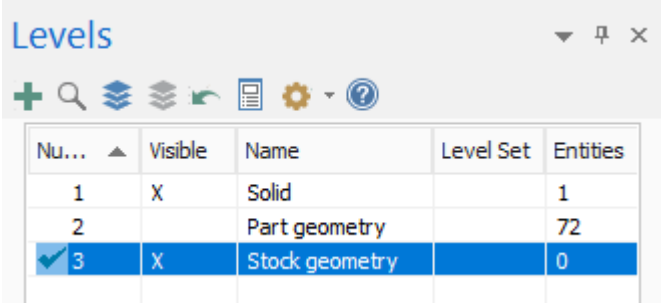
Before creating the stock shape, you first create a level on which to place the stock geometry. This is a good practice for organizing part files that have thousands of lines, arcs, solids, or surfaces.

5. On the Levels Manager, select **Add a new level**.



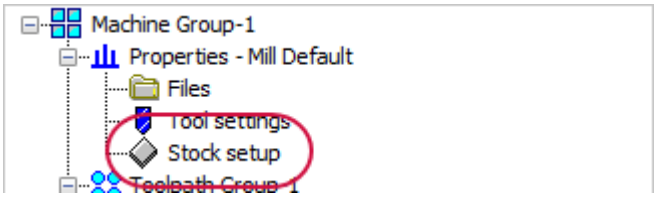
Mastercam adds a new level to the list, and sets it as the main level.

6. Double-click in the **Name** column of the level you have just created and rename it to **Stock geometry**.



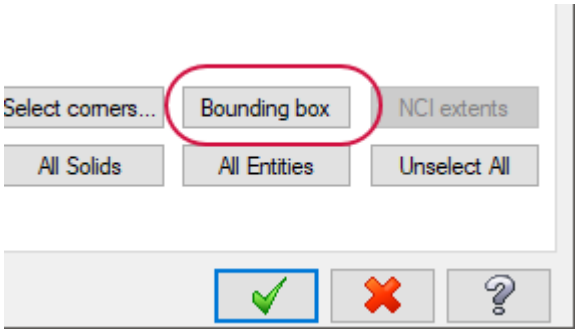
You now use this level for the geometry created by the stock setup.

7. Select **Stock setup** in the Toolpaths Manager.



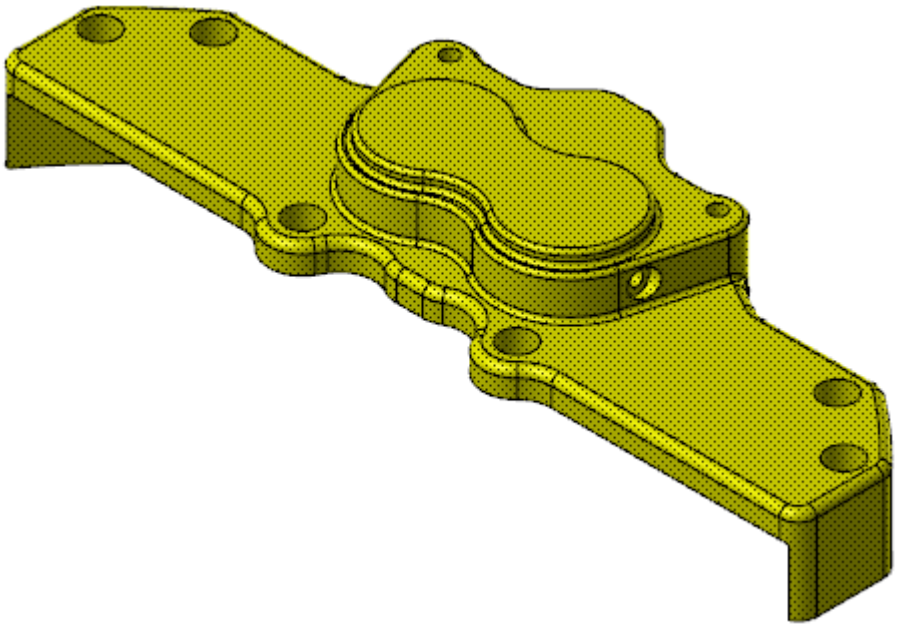
The **Machine Group Properties** dialog box displays.

8. On the **Stock setup** tab, click **Bounding box**.

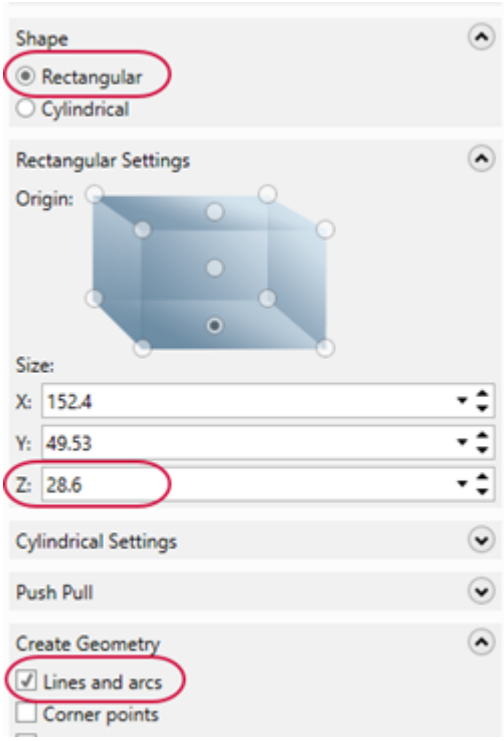


The **Bounding Box** function panel displays.

9. Select the entire part, as shown below:



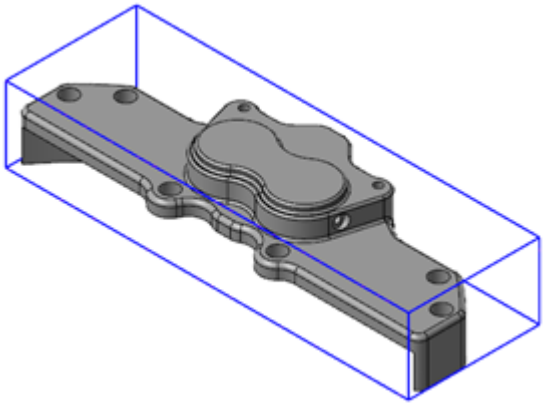
10. Press [**Enter**] or click **End Selection** to accept the selection.
11. Set the following parameters in the **Bounding Box** function panel:



- **Shape** to **Rectangular**. This shape is better suited for this part than **Cylindrical**.
- **Z** to **28.6**. Sets the size of the box in the Z axis.
- Under **Create Geometry**, select **Lines and arcs**. This will create, based on the stock shape, lines and arcs that can be used for many functions.

12. Click **OK** to accept and create the boundary and return to the **Machine Group Properties** dialog box.

13. Click **OK** in the **Machine Group Properties** dialog box. The part now displays with the stock boundary and geometry.



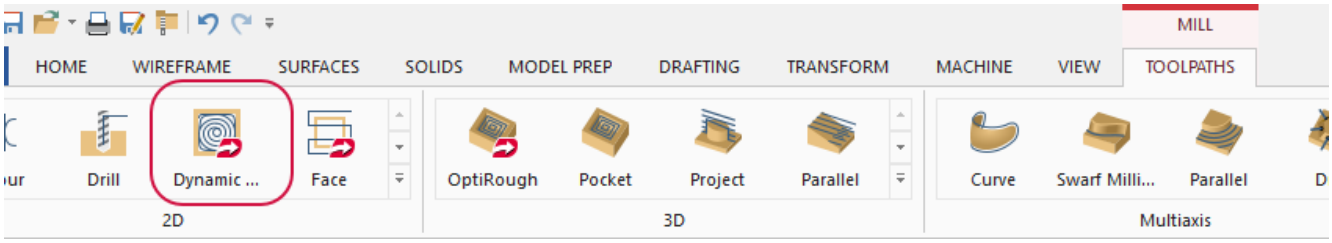
You have now created the necessary stock and geometry to create the Dynamic Mill toolpaths.

14. Click in the graphics window to ensure that all selected geometry is deselected.
15. Save your part.

Exercise 3: Creating the first Dynamic Mill toolpath

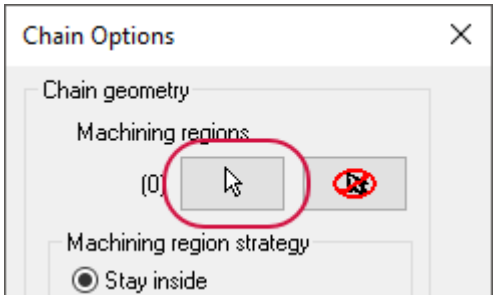
You now create the first Dynamic Mill toolpath to machine the outside of the part.

1. Select **Dynamic Mill** from the **2D** gallery on the **Mill Toolpaths** contextual tab.

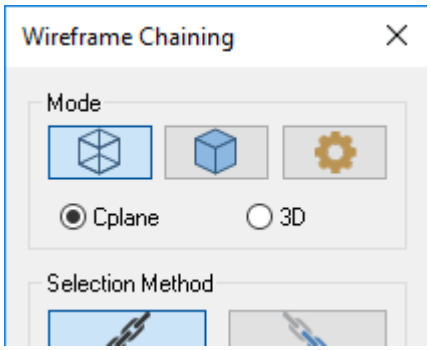


   The **Chain Options** dialog box displays.

2. Click **Select** under **Machining regions**. Machining regions are the regions that are going to be machined.

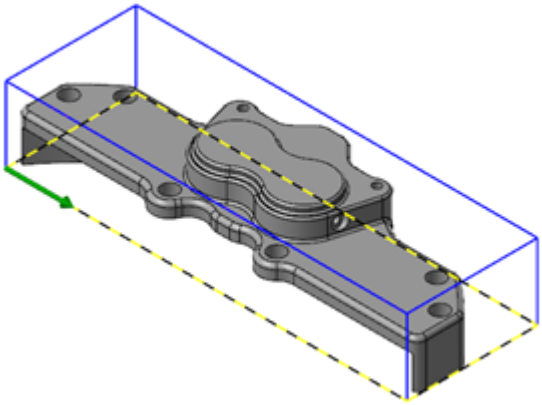


3. If the **Solid Chaining** dialog box displays, click the **Wireframe** button.
4. Select **Cplane** in the **Wireframe Chaining** dialog box.



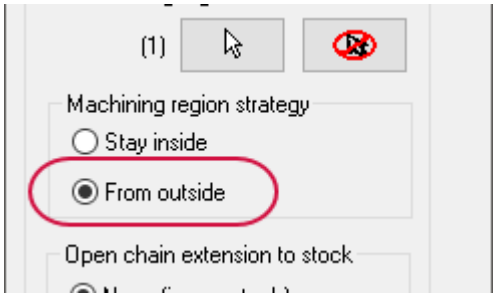
   This option allows you to chain only entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain.

5. Select the chain shown below as the machining region:



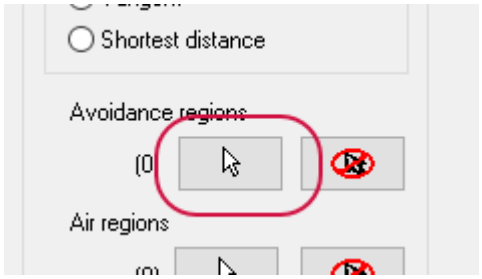
By changing the chaining selection to **Cplane**, you avoid selecting each line individually.

6. Click **OK** in the chaining dialog box to return to the **Chain Options** dialog box.
7. Set the **Machining region strategy** to **From outside**.



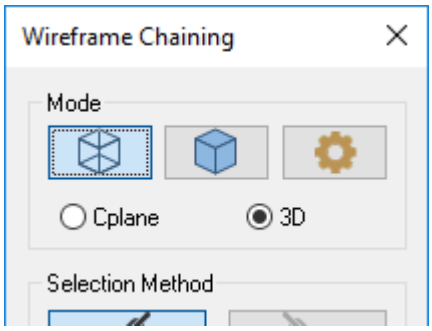
From outside uses free-flowing motion to machine features such as standing bosses or cores. To clear the outside of the part, this is the strategy you want.

8. Click **Select** under **Avoidance regions**. Avoidance regions are regions that you want to avoid during machining. In this case, you want to avoid the part itself and clear out the area between the stock boundary and the part. To do this, you will need to activate another level.



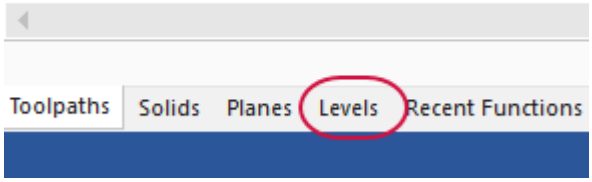
The chaining dialog box displays.

9. Select **3D** in the chaining dialog box.

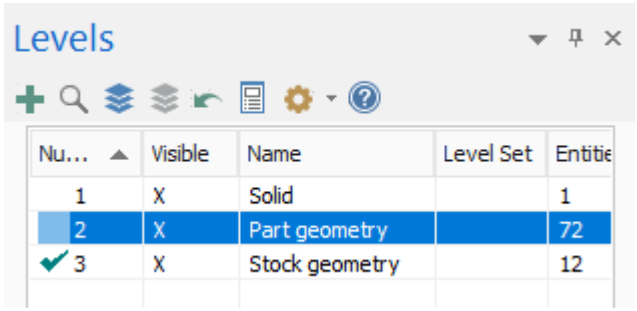


This mode chains entities defined in the X, Y, and Z axes simultaneously.

10. Select the Levels Manager. Move the chaining dialog box out of the way if necessary.

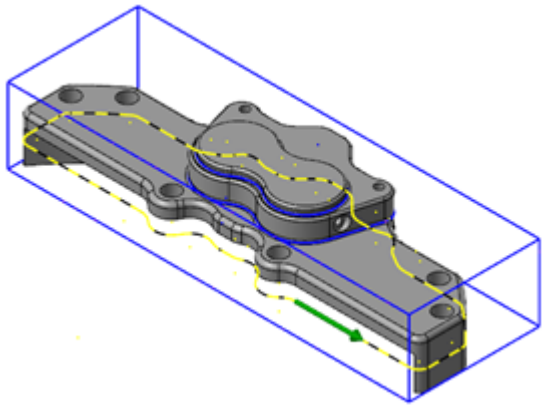


11. Click in the **Visible** column for the Part geometry level.



The graphics window now displays the geometry you need to select. A silhouette of the part, this geometry allows you to machine around the part and clear out the stock, but without cutting into the part.

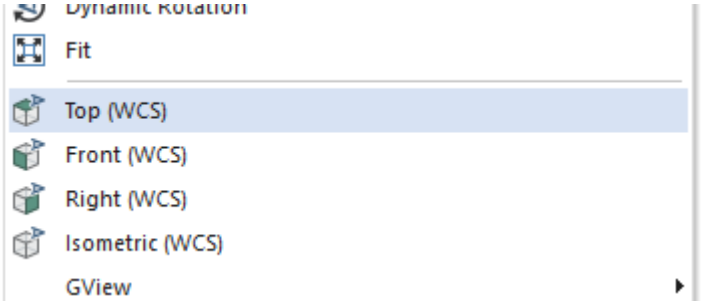
12. Select the chain shown below as an avoidance region:



13. Click **OK** in the **Wireframe Chaining** dialog box to accept the chain and return to the **Chain Options** dialog box.

Before you accept these chains in the **Chain Options** dialog box, you first preview them to ensure that these are the chains you want.

14. Right-click in the graphics window, and set the view to **Top (WCS)**.



15. In the **Chain Options** dialog box, click **Preview chains**.

Your part displays as shown in the following image:

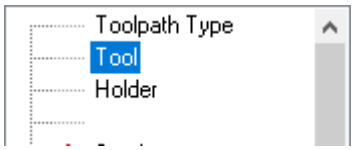
Mastercam displays the material in the red and black crosshatch, the motion region as blue, and the tool containment as yellow. By using **Preview chains**, you can determine if the results are correct before entering parameters.

You can change these colors by selecting the **Color** button. These options are also available on the **Toolpath Type** page.

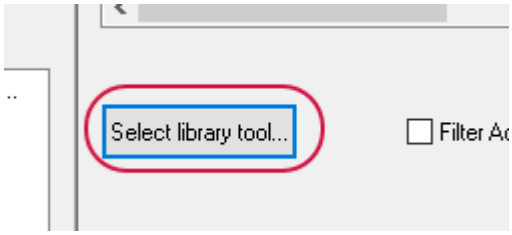
16. Once you are satisfied, right-click in the graphics window and select **Isometric (WCS)**.
17. Click **OK** in the **Chain Options** dialog box to accept these chains.

The **2D High Speed Toolpath - Dynamic Mill** dialog box displays.

18. Select the **Tool** page.

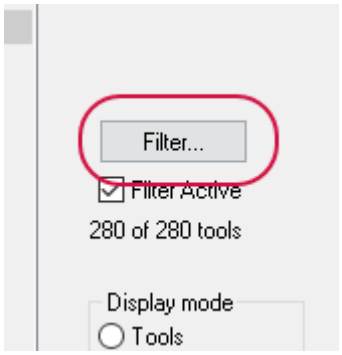


19. Click **Select library tool**.



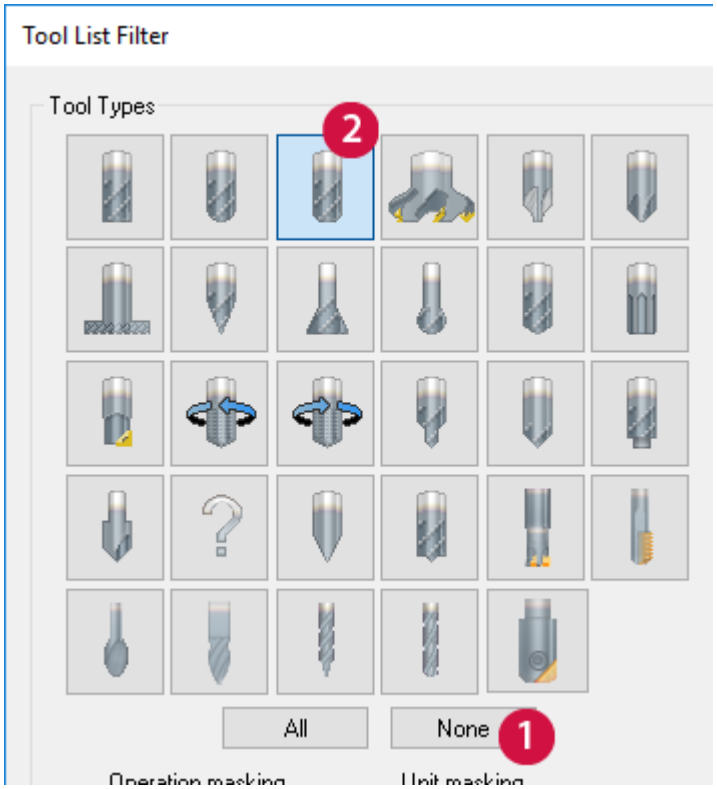
The **Tool Selection** dialog box displays.

20. Select **Filter**.



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

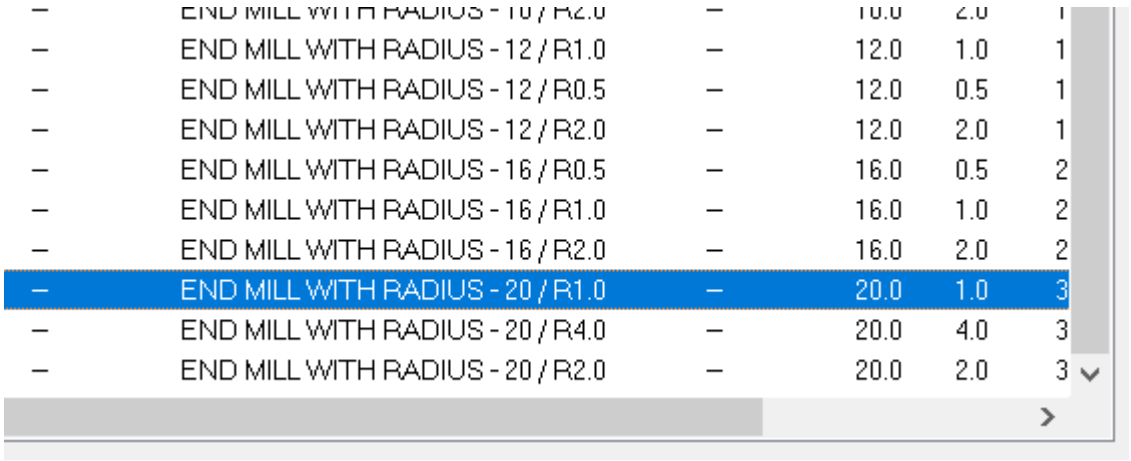
21. Select **None** to remove all filters, and then select **EndMill3 Bull**. This ensures that you show only Bull end mill tools.



You can also double-click a tool image to set the filter.

22. Click **OK** in the **Tool List Filter** dialog box.

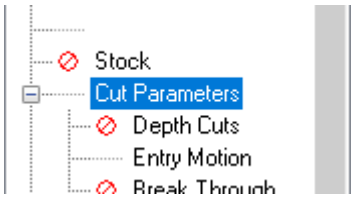
23. In the **Tool Selection** dialog box, select the **END MILL WITH RADIUS - 20/R1.0** tool. If necessary, expand the **Tool Name** column to view the entire tool name.



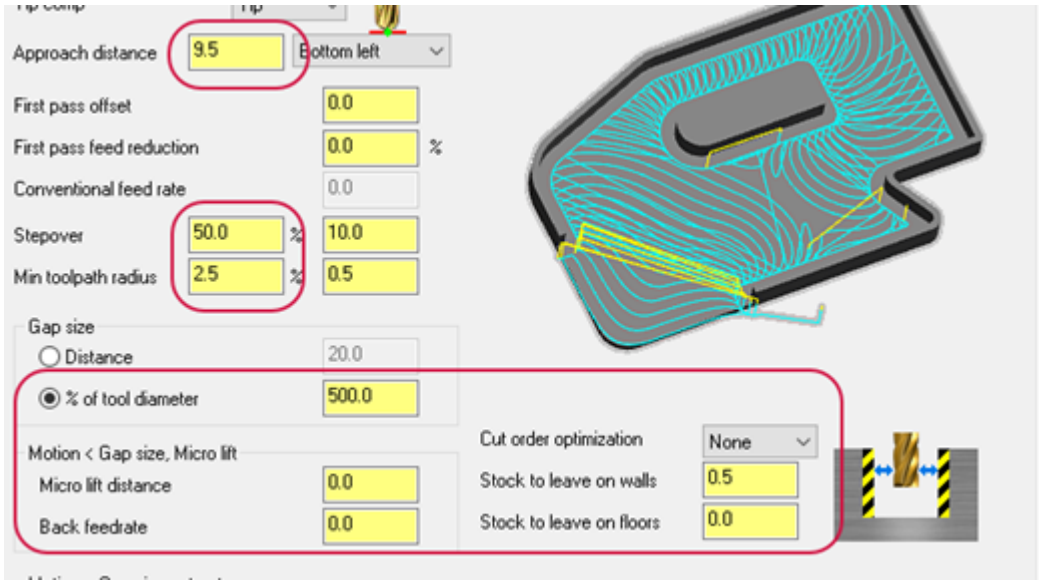
24. Click **OK** in the **Tool Selection** dialog box to add the tool to the toolpath.

You can also double-click the tool to add it to the toolpath.

25. Select the **Cut Parameters** page.

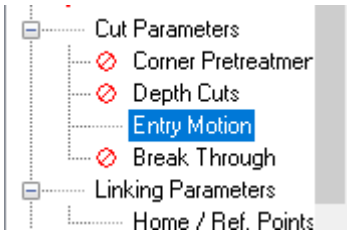


26. Set the following parameters:

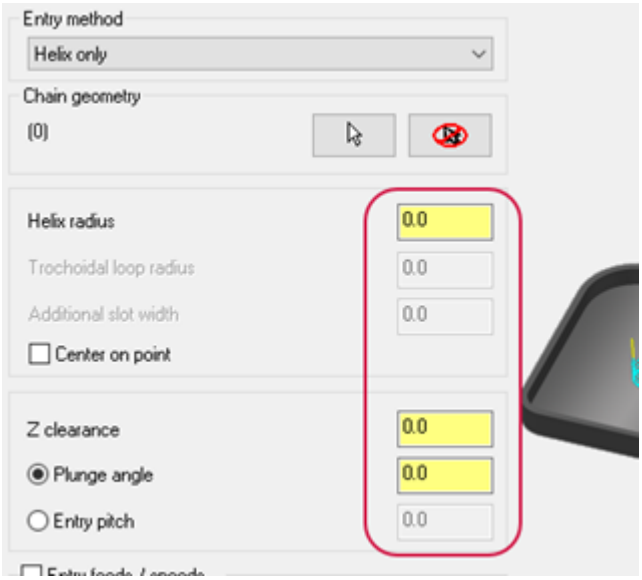


- **Approach distance** to **9.5**. Adds a specified distance to the beginning of the toolpath's first cut.
- **Stepover** to **50.0**. Sets the distance between cutting passes in the X and Y axes.
- **Minimum toolpath radius** to **2.5**. Mastercam uses this radius in combination with the **Micro lift distance** and **Back feed rate** parameters to calculate 3D arc moves between cutting passes.
- **Micro lift distance** to **0.0**. Sets the distance the tool lifts off the part on back moves.
- **Back feedrate** to **0.0**. Controls the speed of the back feed movement of the tool.
- **Cut order optimization** to **None**. Directs the toolpath to start at the most recently machined material when cutting.
- **Stock to leave on walls** to **0.5**. Leaves a defined amount of stock on the vertical drive geometry.
- **Stock to leave on floors** to **0.0**. Leaves a defined amount of stock on horizontal drive geometry.

27. Select the **Entry Motion** page.



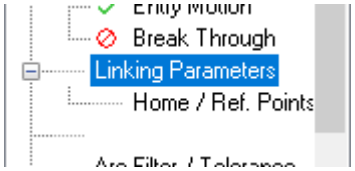
28. Set the following parameters:



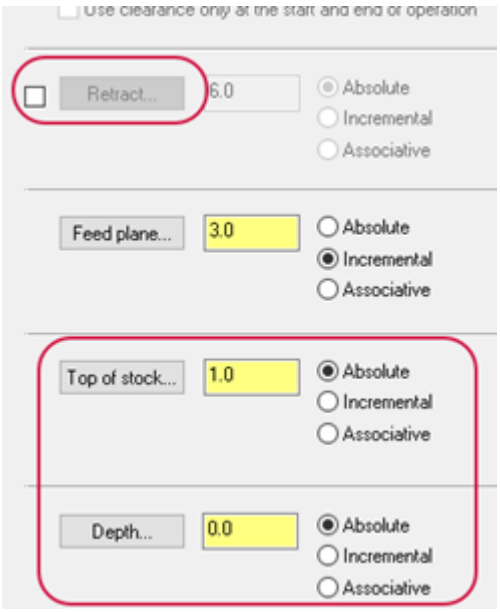
- **Helix radius** to **0.0**. Sets the radius of the entry helix.
- **Z clearance** to **0.0**. Sets an extra height used in the ramping motion down from a top profile.
- **Plunge angle** to **0.0**. Sets the angle of descent for the entry move and determines the pitch.

This toolpath does not enter down into the stock, so you do not need to set entry parameters.

29. Select the **Linking Parameters** page.

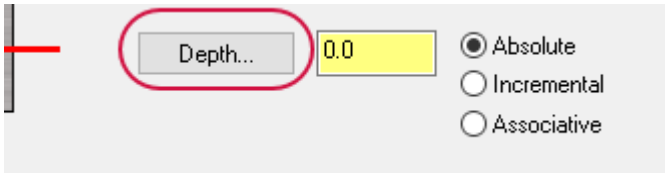


30. Set the following parameters:



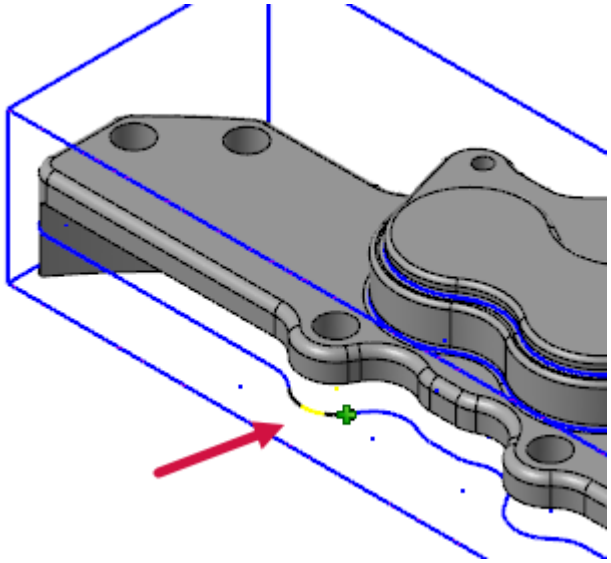
- Deselect **Retract**. The retract distance is the height that the tool moves up to before the next tool pass.
- Set **Top of stock** to **1.0**.
- Change **Top of stock** and **Depth** to **Absolute**. Mastercam measures absolute values from the origin 0,0,0.

31. Select **Depth** to return to the graphics window.



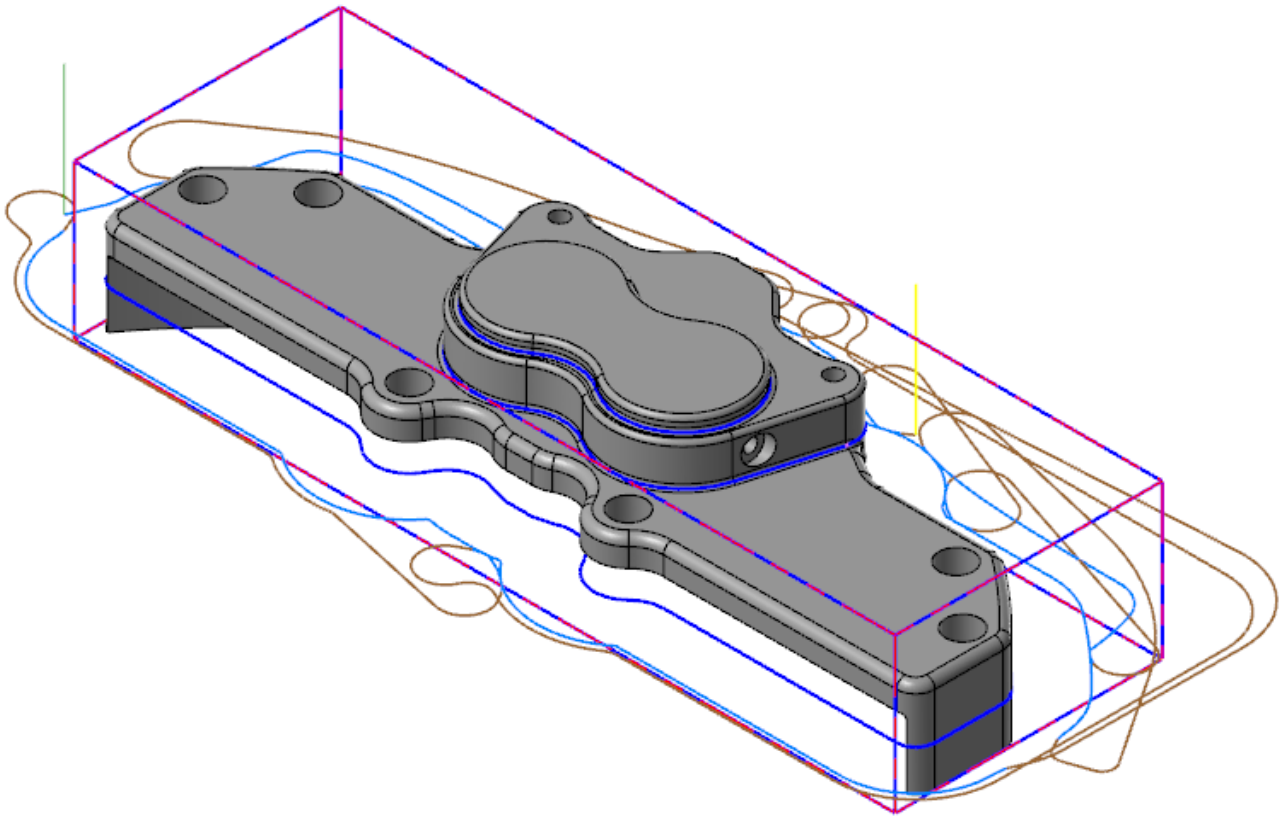
Depth determines the final machining depth and the lowest depth that the tool descends into the stock. In this case, the machining depth is in the middle of the part because Dynamic Mill uses the entire tool to cut.

32. Select the chain shown below:



Mastercam returns to the **2D High Speed Toolpath - Dynamic Mill** dialog box. Your **Depth** should now be set at **-20.32**.

33. Click **OK** in the **2D High Speed Toolpath - Dynamic Mill** dialog box to generate your toolpath. The toolpath displays as shown below:

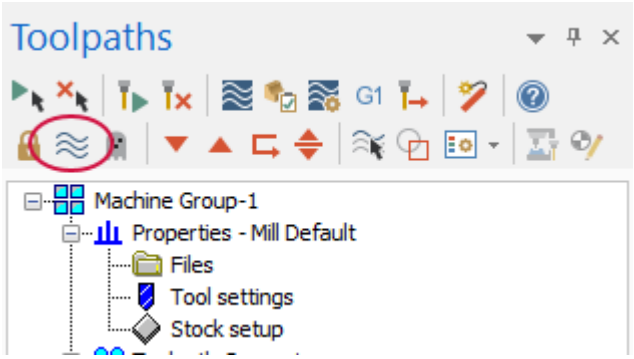


34. Save your part.

Exercise 4: Creating the second Dynamic Mill toolpath

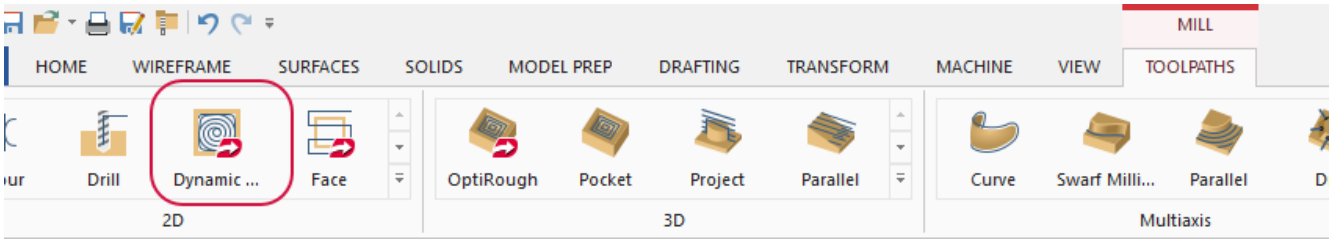
You now create a second Dynamic Mill toolpath, using the **Rest material** option.

1. In the Toolpaths Manager with the Dynamic Mill toolpath selected, select **Toggle display on selected operations**.



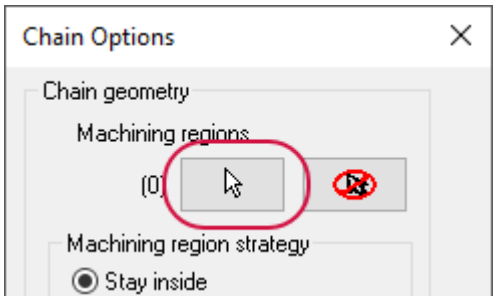
   This option hides the display of the first toolpath and keeps your graphics view uncluttered while you create another toolpath.

2. Select **Dynamic Mill** from the **2D** gallery on the **Mill, Toolpaths** contextual tab.



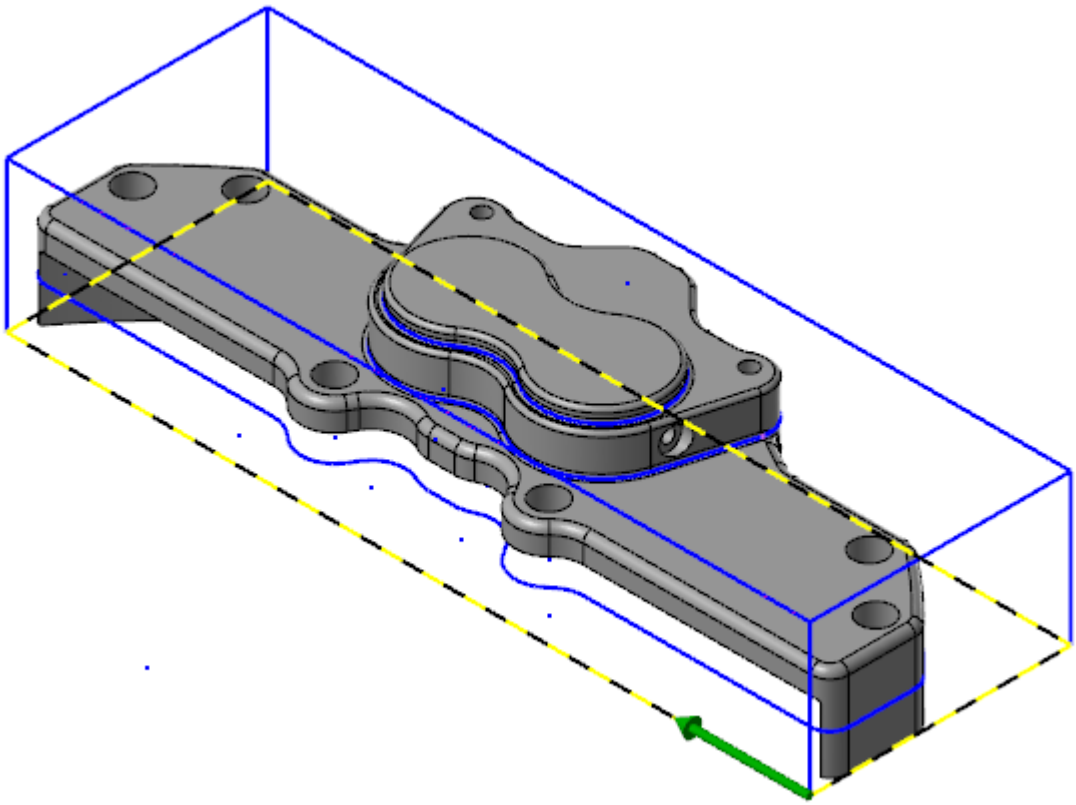
   The **Chain Options** dialog box displays.

3. Click **Select** under **Machining regions**.



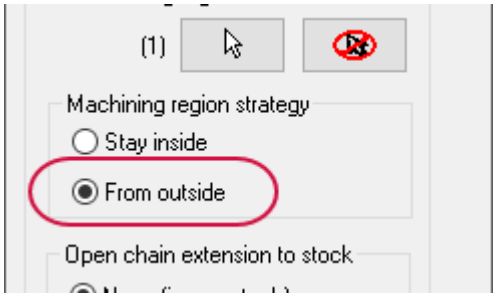
   The chaining dialog box displays.

4. Select the bottom stock geometry as the machining region. Use the **Cplane** button to select the entire chain.

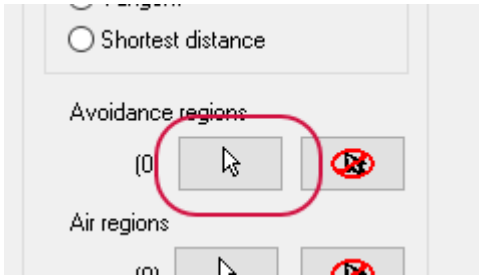


The selection is the same geometry used in the first Dynamic Mill toolpath.

5. Click **OK** in the **Wireframe Chaining** dialog box to return to the **Chain Options** dialog box.
6. Ensure that the **Machining region strategy** is set to **From outside**.

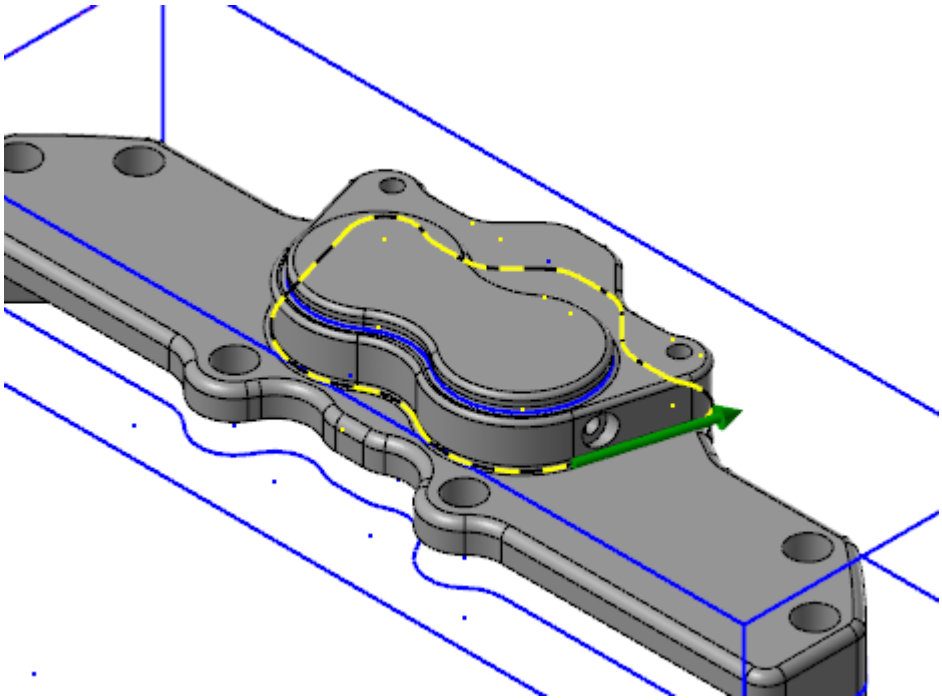


7. Click **Select** under **Avoidance regions**.



The chaining dialog box displays.

8. Select the chain shown below as an avoidance region. Set the chaining to **3D**.

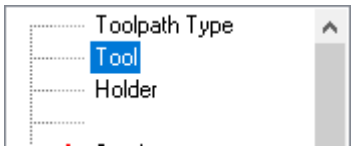


You may need to rotate the part or turn off the **Solid** level to select the chain.

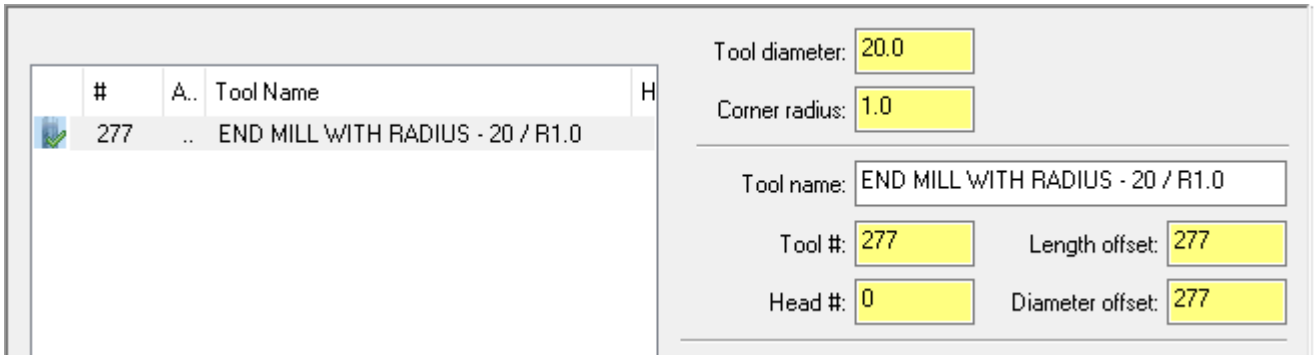
9. Click **OK** in the **Wireframe Chaining** dialog box to accept the chain and return to the **Chain Options** dialog box.
10. Click **OK** in the **Chain Options** dialog box to accept these chains.

The **2D High Speed Toolpath - Dynamic Mill** dialog box displays.

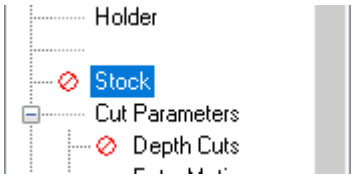
11. Select the **Tool** page.



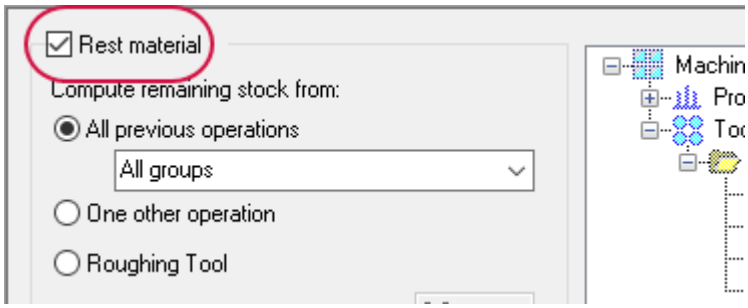
12. In the tool list, select the tool that was used for the previous Dynamic Mill toolpath, which is **END MILL WITH RADIUS - 20/R1.0.**



13. Select the **Stock** page.

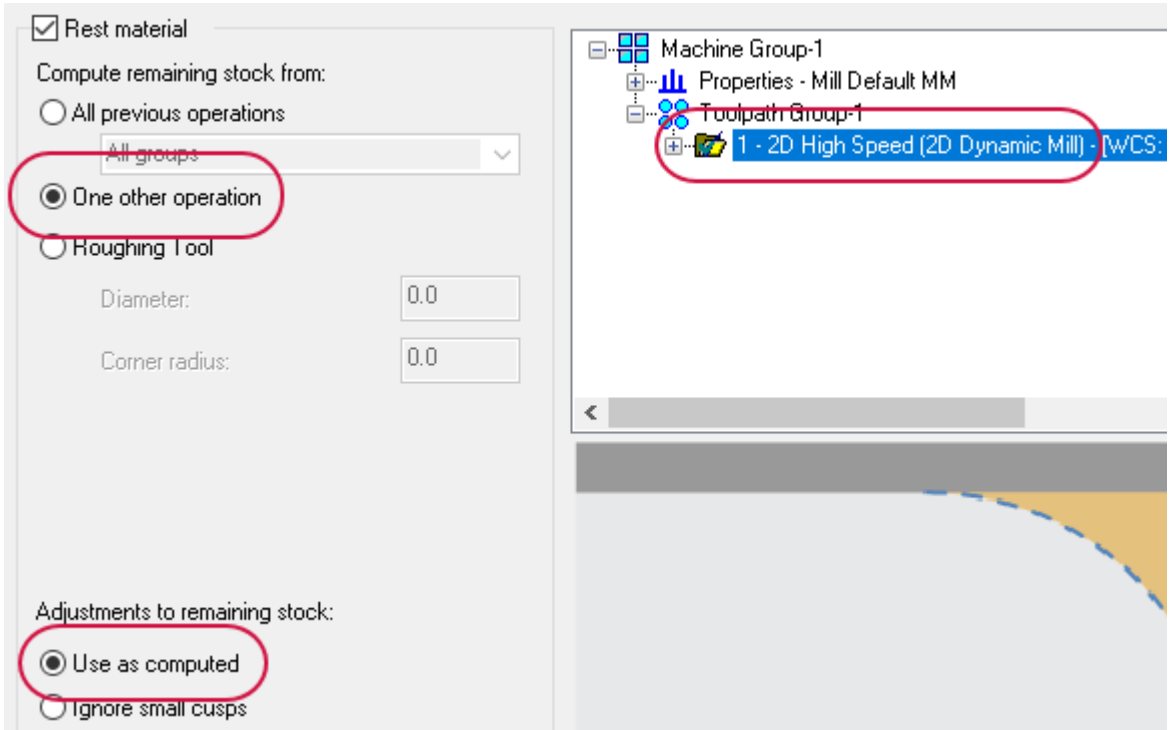


14. Select **Rest material**.



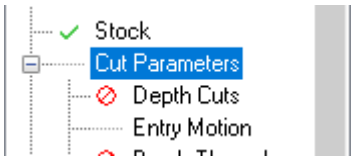
This enables the page and tells Mastercam to calculate the cutting passes based on the remaining stock.

15. Set the following parameters:

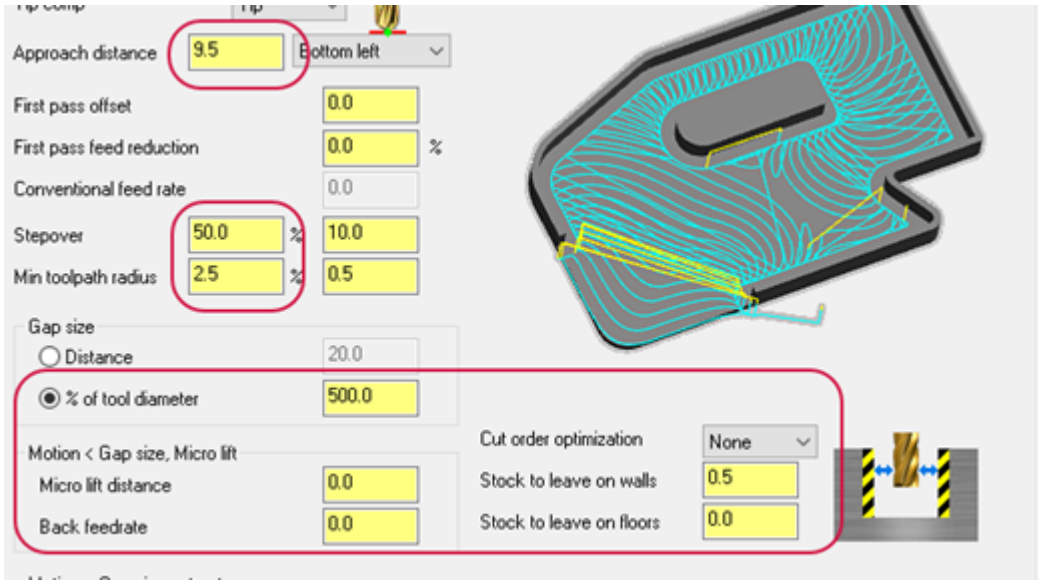


- Set **Compute remaining stock from** to **One other operation**. This calculates the remaining stock from a single operation.
- In the **Operation List**, select the **2D High Speed (2D Dynamic Mill)** toolpath.
- Set **Adjustments to remaining stock** to **Use as computed**. This option makes no adjustments to the stock model.

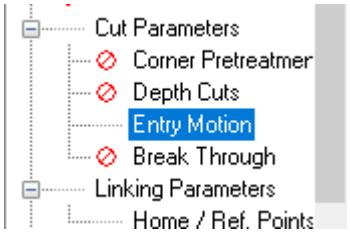
16. Select the **Cut Parameters** page.



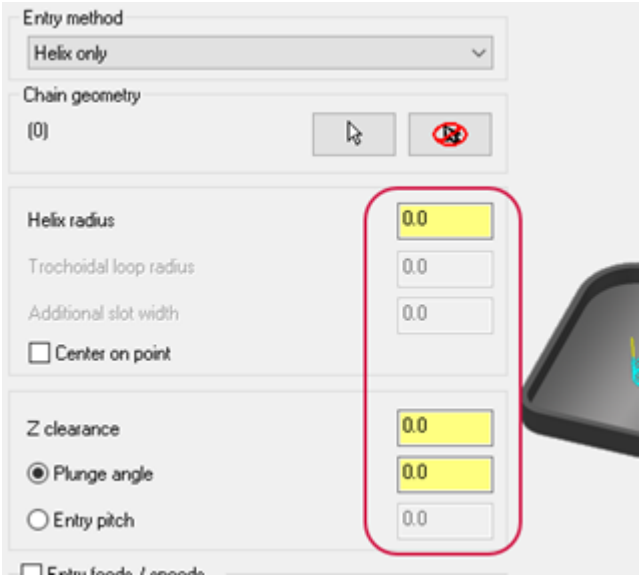
17. The parameters are the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



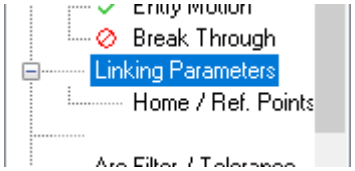
18. Select the **Entry Motion** page.



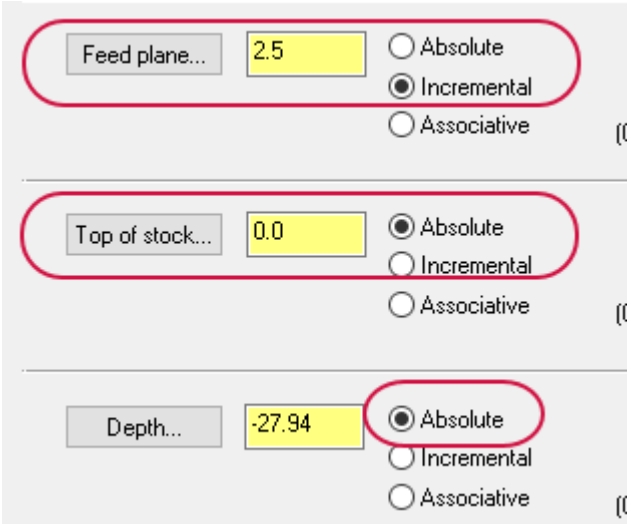
19. The parameters are also the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



20. Select the **Linking Parameters** page.

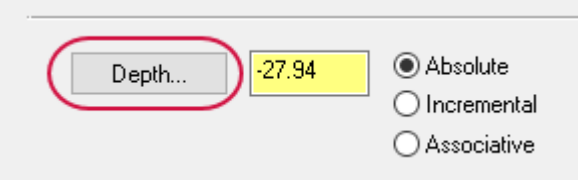


21. Set the following parameters:

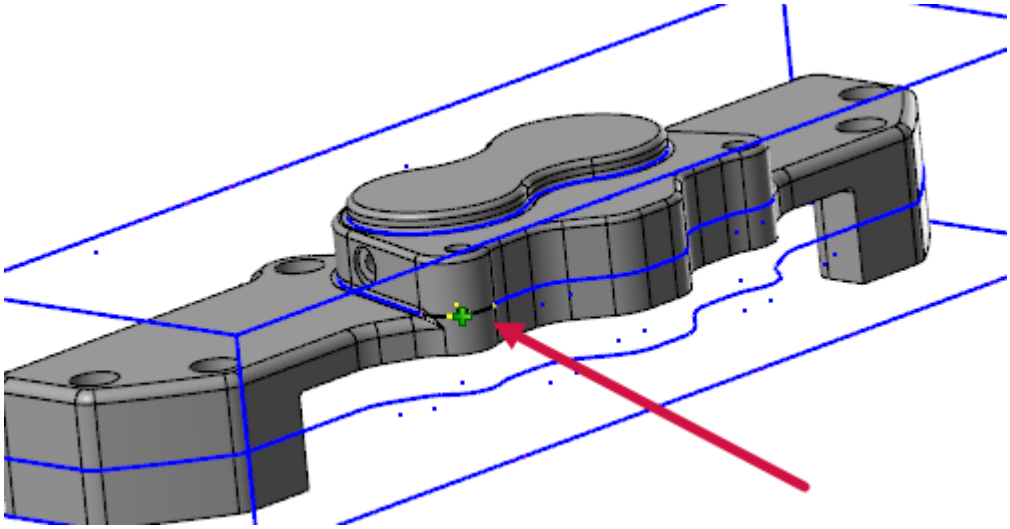


- **Feed plane** to **2.5**. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- Ensure that **Feed plane** is set to **Incremental**.
- **Top of Stock** to **0.0**. Sets the top of the stock.
- Ensure that **Top of stock** and **Depth** are set to **Absolute**.

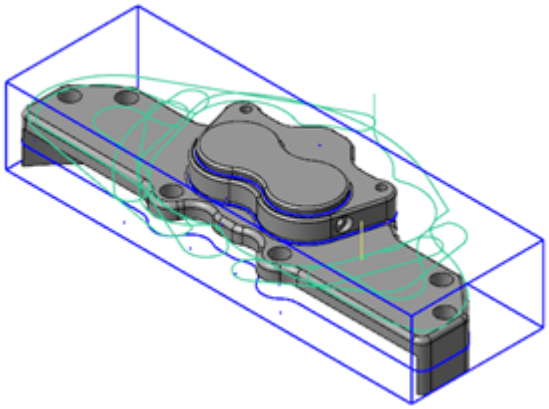
22. Click **Depth** to return to the graphics window.



23. Select the arc shown below (you will need to rotate the part):



24. Mastercam returns to the **2D High Speed Toolpath - Dynamic Mill** dialog box. Your **Depth** should be **-10.16**.
25. Click **OK** in the **2D High Speed Toolpath - Dynamic Mill** dialog box to generate your toolpath.
26. The toolpath will display as shown below:



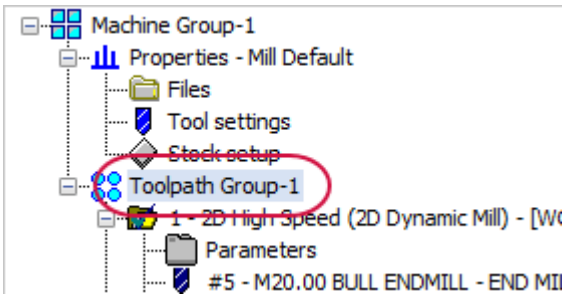
Notice that the toolpath clears out the top of the part while avoiding the large feature that was selected as the avoidance region.

27. Save your part.

Exercise 5: Verifying the toolpaths

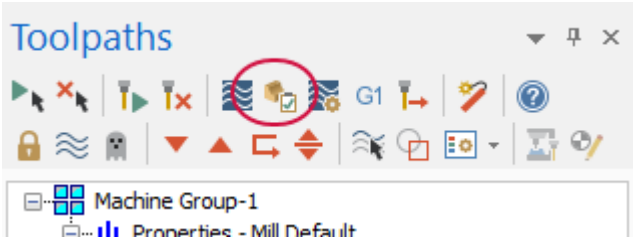
You will now verify both Dynamic Mill toolpaths with Mastercam Simulator. Mastercam Simulator allows you to verify your toolpaths by using solid models to simulate part machining against a selected stock definition. Verify creates a representation of the surface finish and shows collisions, if any exist. Use Verify to identify and correct program errors before they reach the shop floor.

1. Select **Toolpath Group-1** in the Toolpaths Manager.



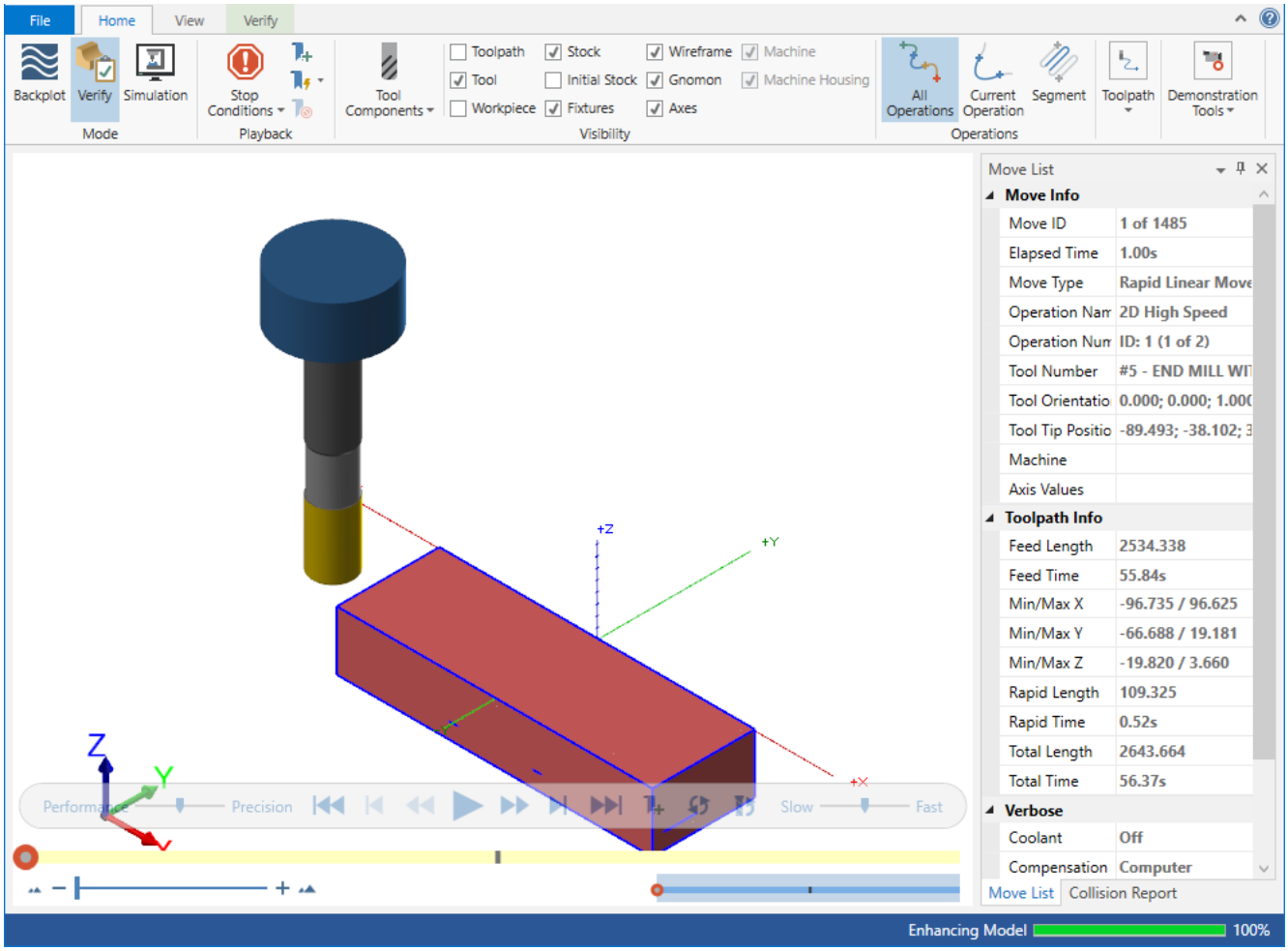
Both Dynamic Mill toolpaths are selected.

2. Select **Verify selected operations**.

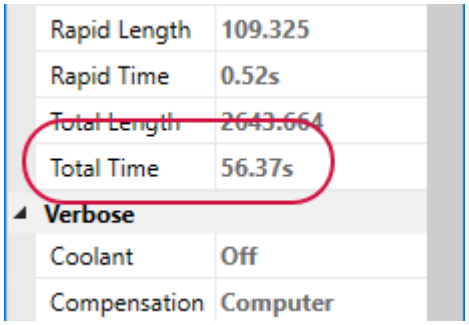


Mastercam Simulator displays.

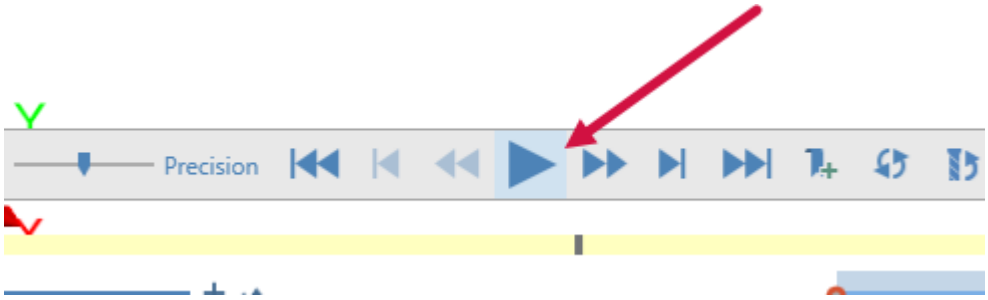
3. Right-click in the graphics view and select **Fit** and **Isometric** from the menu. Mastercam Simulator should match the image shown below.



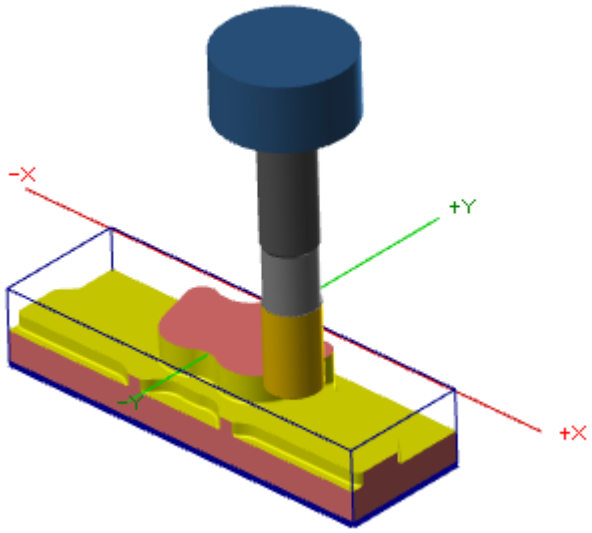
4. View the **Move Info**, which displays all pertinent information about the toolpaths being verified. With the given parameters, Dynamic Mill toolpaths can machine the part in the time shown. (Your time may be different from what is shown here.)



5. Press **Play** to preview the toolpath motion for both operations.



6. Once you are satisfied with your results, close Mastercam Simulator.



7. Save your part file.

The Dynamic Mill toolpaths quickly remove the stock on the outside and prepare the part for other necessary operations. In the next lesson, you create a Face toolpath to further machine this part.

CHAPTER 2

THE FACE TOOLPATH

Face toolpaths clean the stock from the top of a part and create an even surface for future operations. This toolpath can be based on chained geometry or on the current stock model.

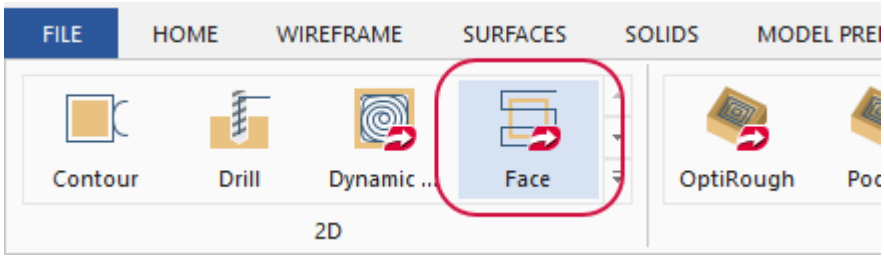
For this chapter, you create a Face toolpath on the top of the part from the Dynamic Mill lesson.

Goals

- Create a Face toolpath
- Preview the toolpath before it has been created
- Verify the created toolpath

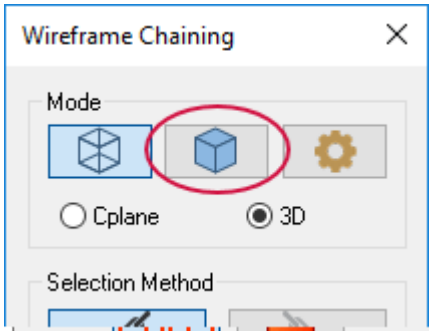
Exercise 1: Creating the Face toolpath

1. Open the part file, FaceMill, provided with this tutorial.
2. Save the part as FaceMill-xxx, replacing xxx with your initials.
3. Select **Face** from the **2D** gallery on the **Mill Toolpaths** contextual tab.

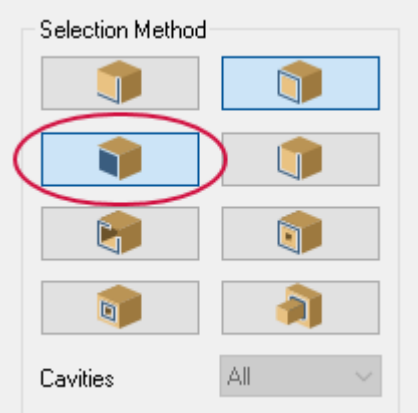


The chaining dialog box displays.

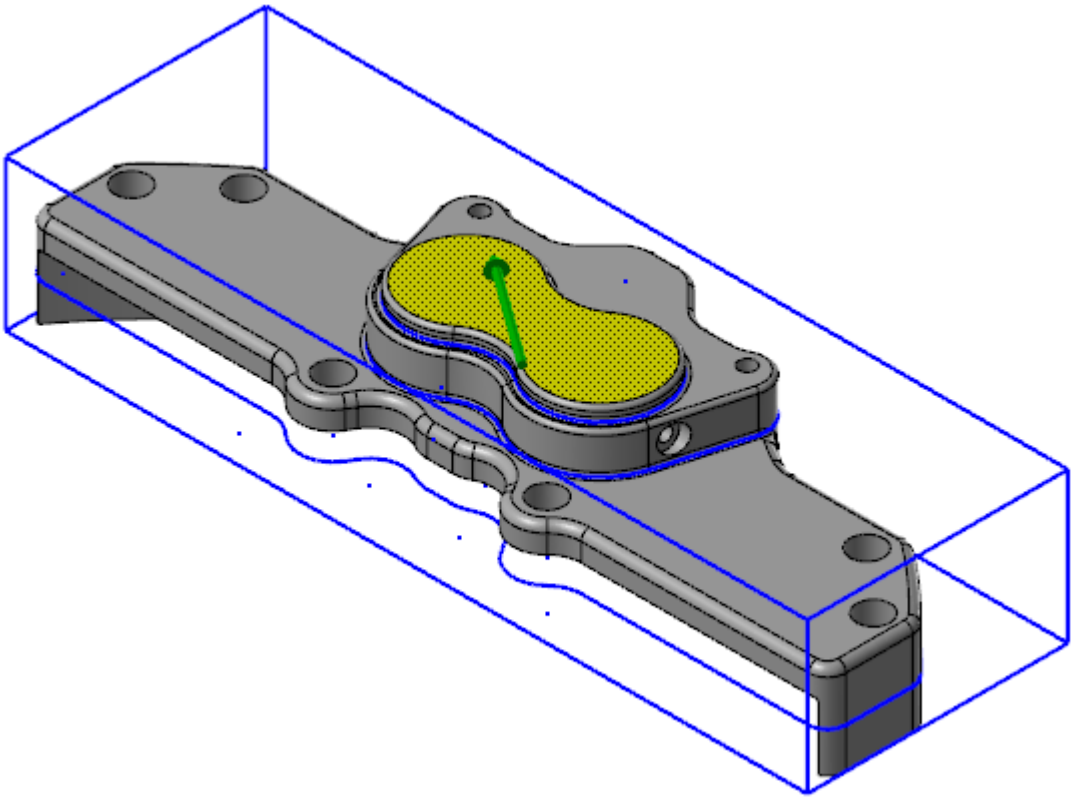
4. If necessary, select **Solids** to activate solid selection.



5. Activate **Face** selection. This allows you to select the face of a solid.



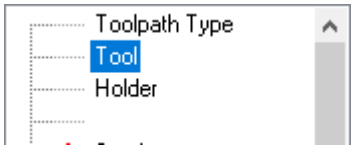
6. Select the face shown below:



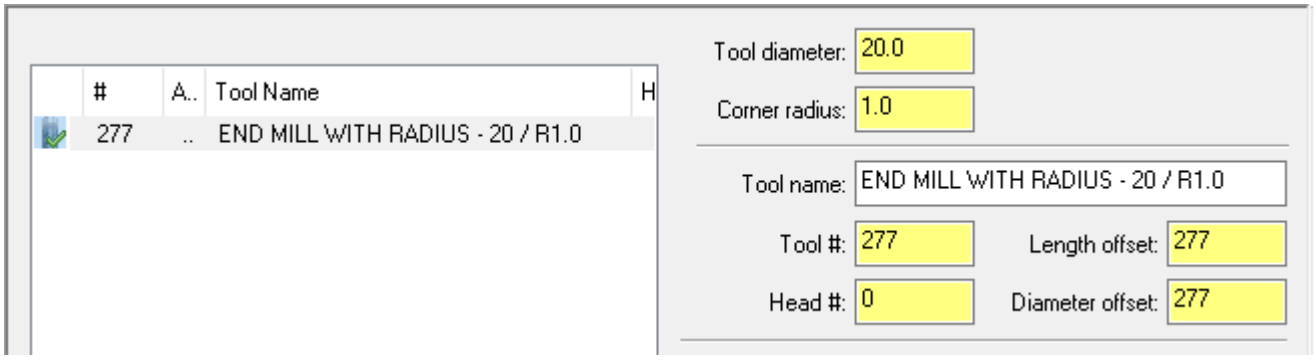
7. Select **OK** in the **Solid Chaining** dialog box.

   The **2D Toolpaths - Facing** dialog box displays.

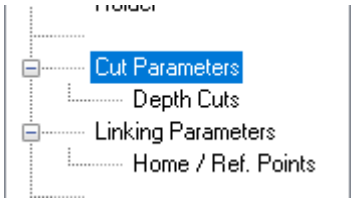
8. Select the **Tool** page.



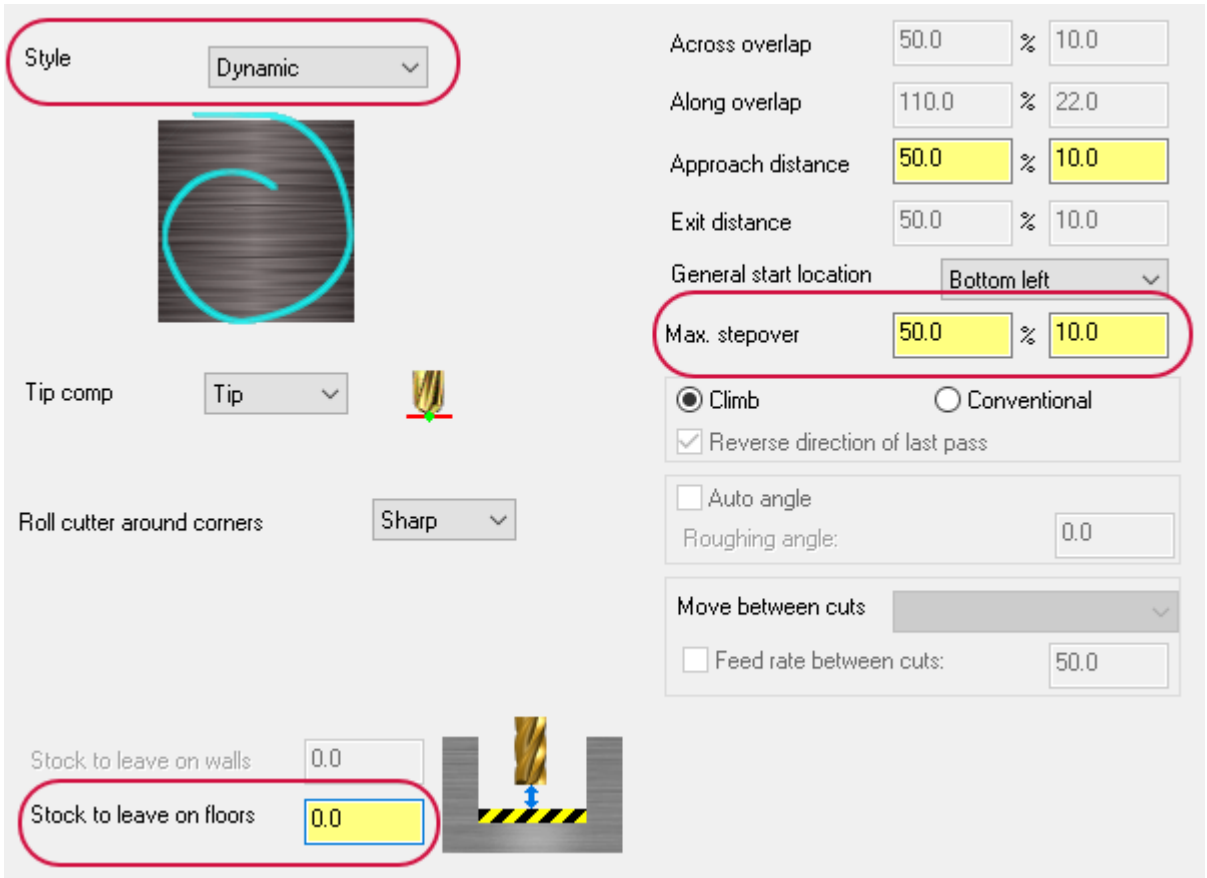
9. Select the **END MILL WITH RADIUS - 20 / R 1.0** that was used in the previous two Dynamic Mill toolpaths. If necessary, expand the **Tool Name** column to see the entire tool name.



10. Select the **Cut Parameters** page.



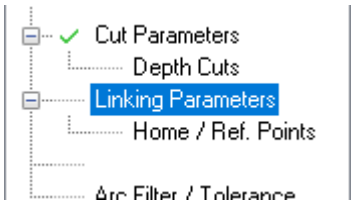
11. Set the following parameters:



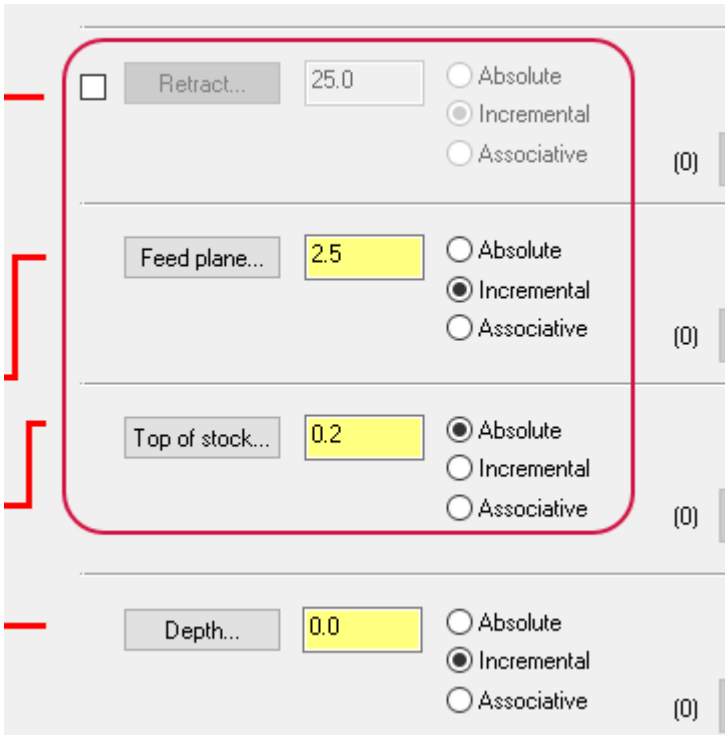
- **Style** to **Dynamic**. Creates a smooth controlled motion that cuts from the outside to the inside, maintaining a constant load on the tool with minimal entries and exits.

- **Stock to leave on floors** to **0.0**. Sets the amount of stock to leave on the floors during machining.
- **Maximum stepover** to **50.0**. Sets the distance between adjacent passes in the toolpath.

12. Select the **Linking Parameters** page.



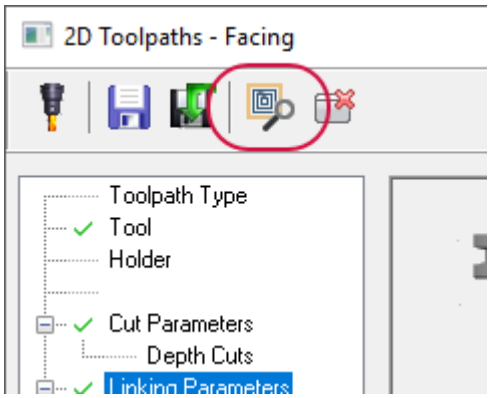
13. Set the following parameters:



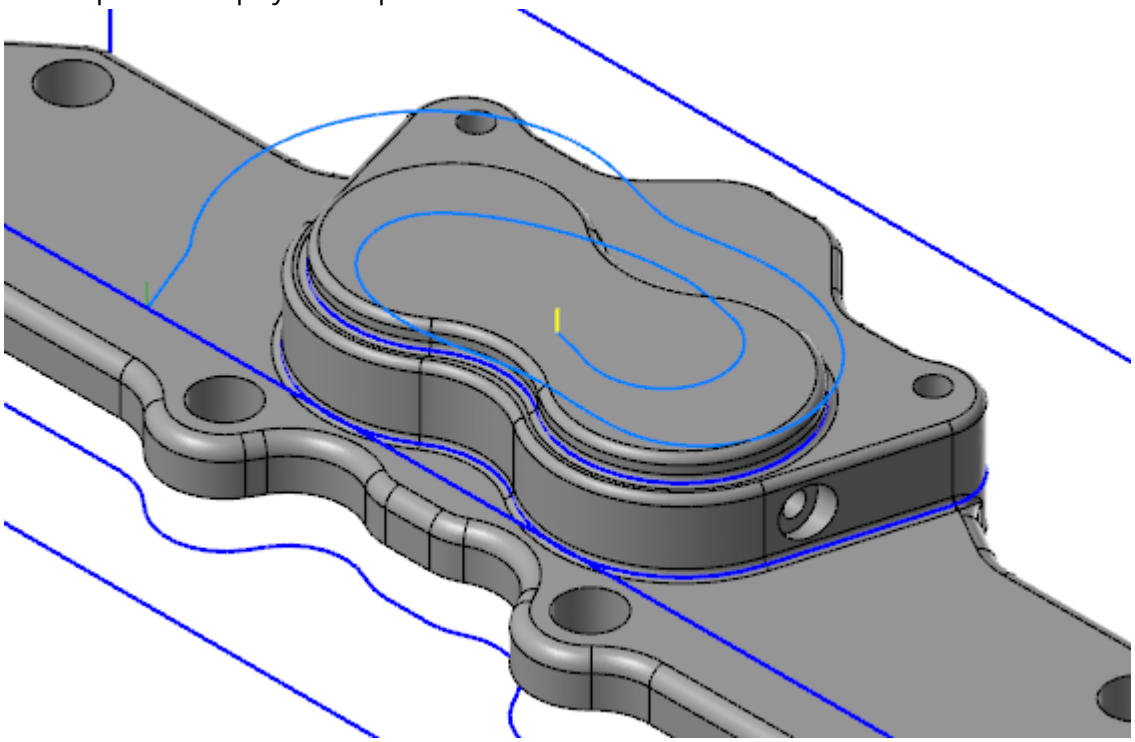
- Deselect **Retract**. Sets the height that the tool moves up to before the next tool pass.
- Set **Feed plane** to **2.5**. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- Set **Top of stock** to **0.2**. Sets the height of the material in the Z axis.

Before clicking **OK**, you first preview the toolpath. Previewing toolpaths allows you to view the toolpath before generating it so that you can make adjustments as necessary.

14. Select **Preview toolpath** on the **2D Toolpaths - Facing** dialog box.



The toolpath will display on the part as shown below:



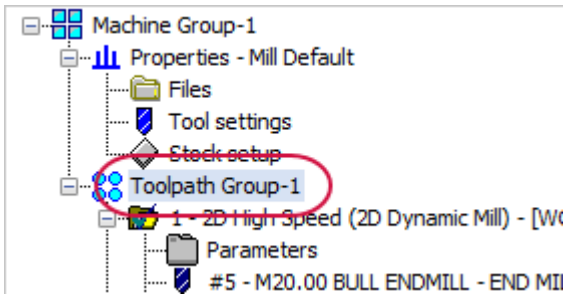
15. Click **OK** in the **2D Toolpaths - Facing** dialog box to close and generate the toolpath.
16. Save your part.

Exercise 2: Verifying the toolpaths

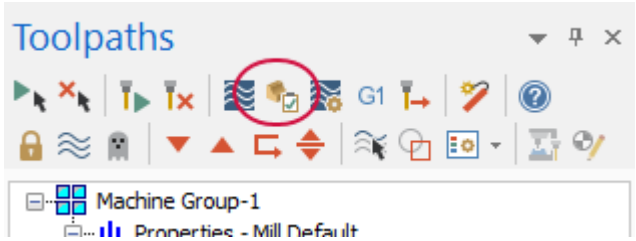
You now verify the two Dynamic Mill toolpaths created from the previous chapter and the Face toolpath together.

1. Select **Toolpath Group-1** in the Toolpaths Manager.

   This selects both Dynamic Mill toolpaths and the Facing toolpath.

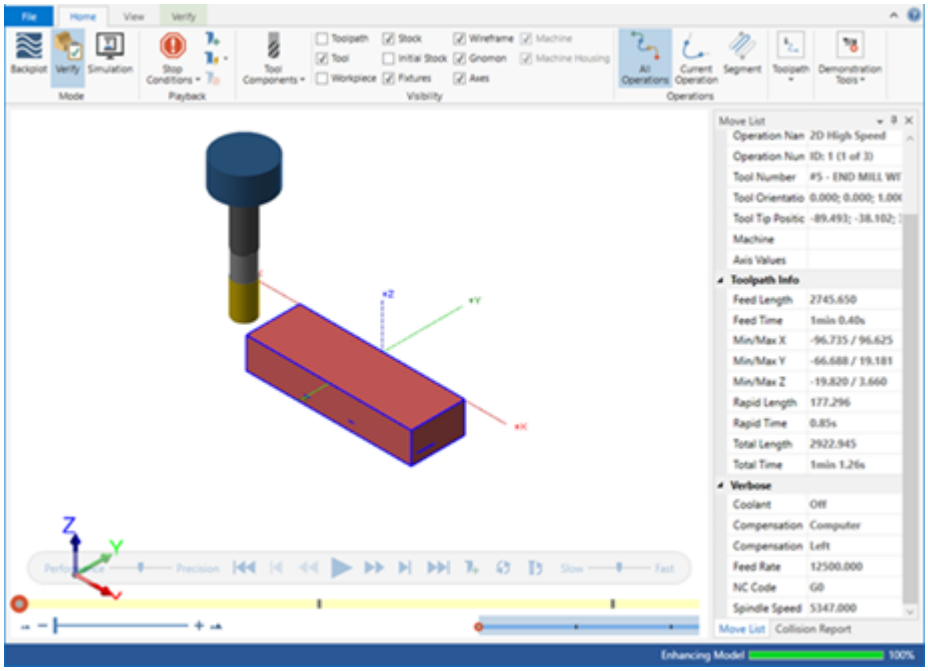


2. Select **Verify selected operations**.

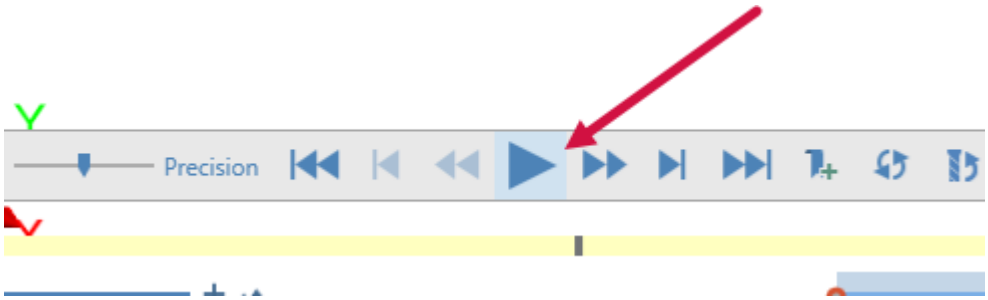


   Mastercam Simulator displays.

3. Right-click in the graphics view and select **Fit** and **Isometric** from the menu. Mastercam Simulator should match the image shown below.

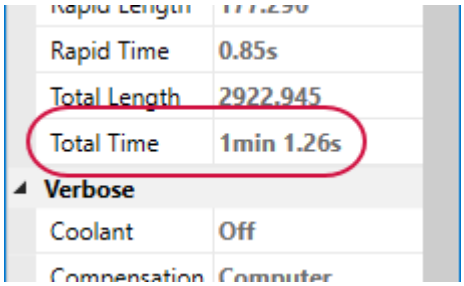


4. Press **Play** to preview the toolpath motion for the toolpaths.

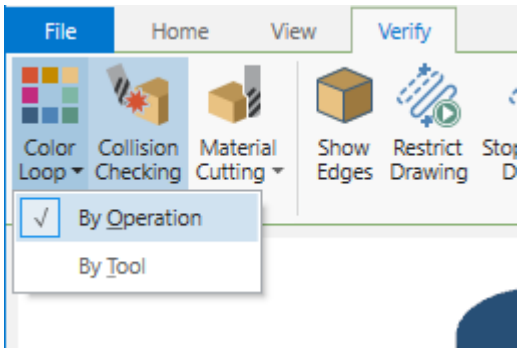


There will be material remaining on the top face of the part. That is to be expected.

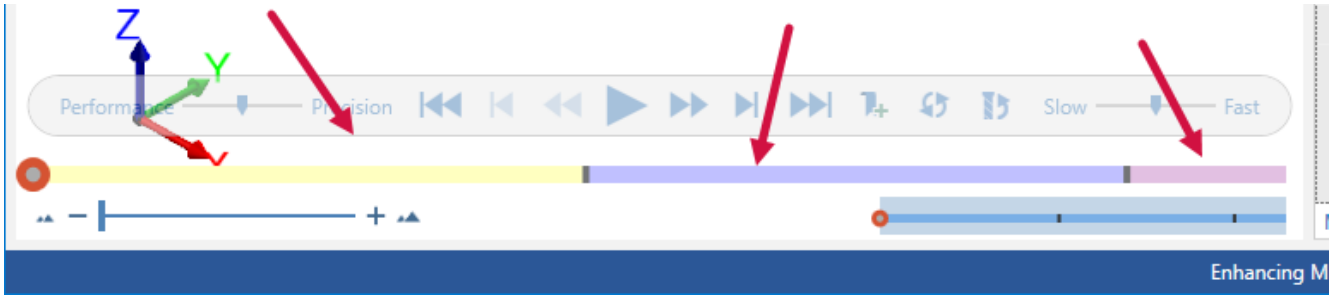
5. View the **Move Info**, which displays information about the toolpaths being verified. With the given parameters, the three toolpaths can machine the part in the time shown.

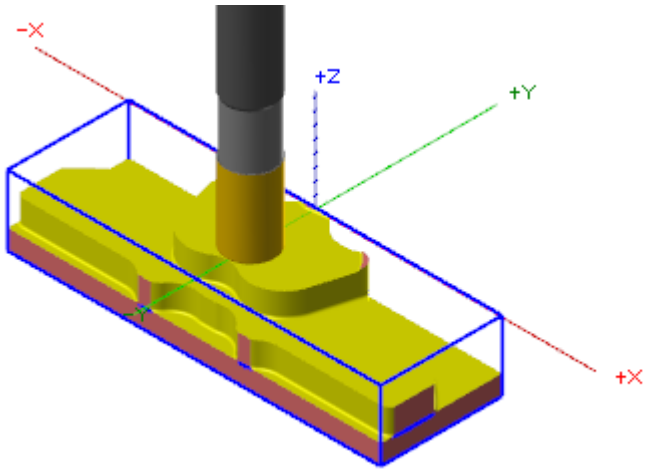


6. On the **Verify** tab, click the **Color Loop** button to activate the function. Then select **Color Loop**, **By Operation** to help distinguish between each toolpath.



Mastercam Simulator color-codes each toolpath so that you can distinguish where they cut the part. The colors on the part correspond to those on the time bar. The second image following shows the results.





7. Once you are satisfied with your results, close Mastercam Simulator.
8. Save your part file.

Using the Dynamic Mill toolpaths and the Face toolpath, you have quickly cleared the part to prepare it for finishing operations. In the next lesson, you create two Dynamic Contour toolpaths.

CHAPTER 3

THE DYNAMIC CONTOUR TOOLPATH

Mastercam's Dynamic Contour toolpath mills material off walls and supports closed or open chains. Compared to its standard counterpart, the Dynamic Contour toolpath provides more efficient cutting by using the entire flute of the tool.

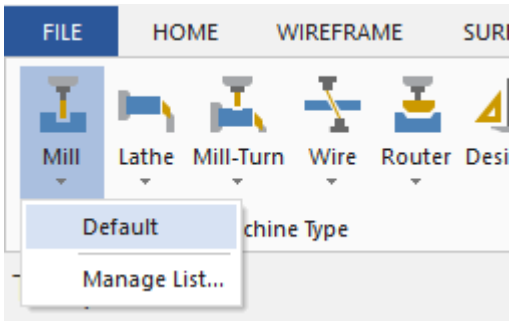
In this chapter, you create a Dynamic Contour toolpath to clear around the walls of the part.

Goals

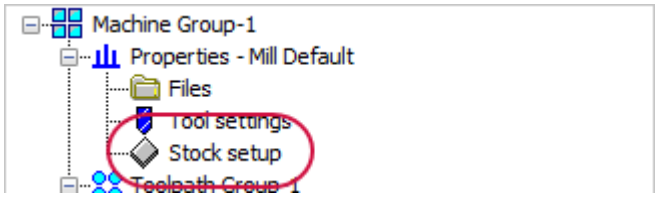
- Create two Dynamic Contour toolpaths
- Analyze the toolpaths
- Verify the created toolpaths

Exercise 1: Setting up the stock

1. Open the part file, DynamicContour, provided with this tutorial.
2. Save the part as DynamicContour-xxx, replacing xxx with your initials.
3. On the **Machine** tab, select **Mill**, **Default.**

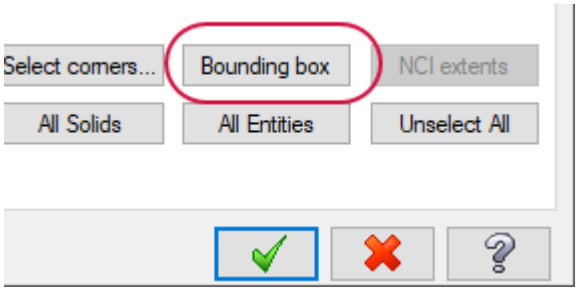


4. Select **Stock setup** from the Toolpaths Manager.



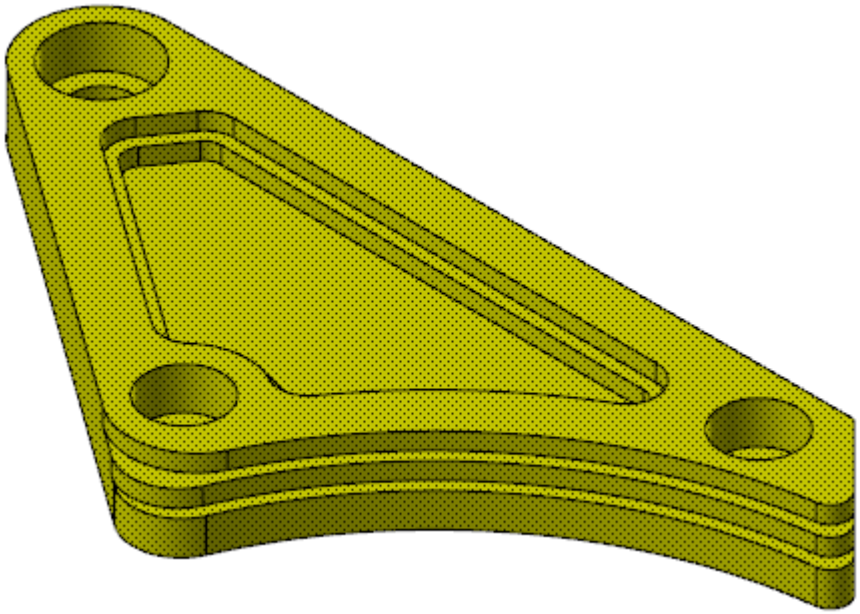
The **Machine Group Properties** dialog box displays.

5. Select **Bounding box** on the **Stock Setup** tab.

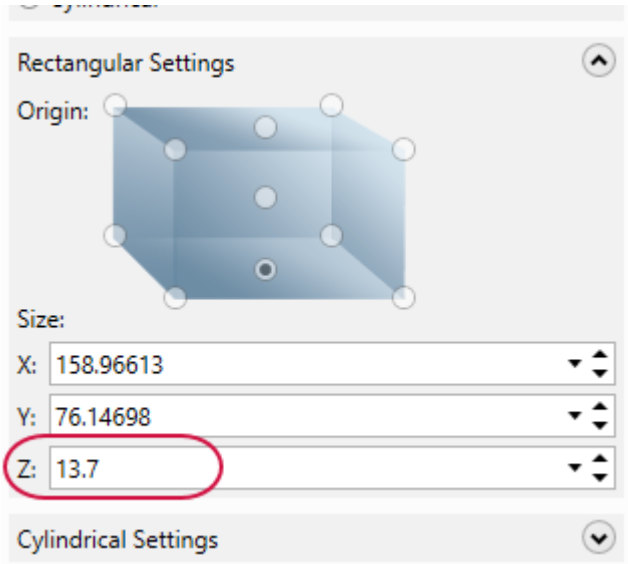


The **Bounding Box** function panel displays.

6. Select the part and press [**Enter**] or select **End Selection**.



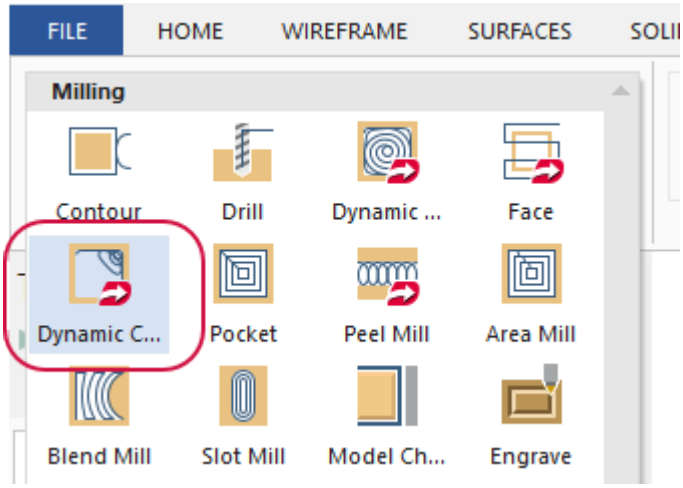
7. Set **Z** to **13.7**. This adds a small amount of stock to the Z height of the stock boundary.



8. Click **OK** to have Mastercam create the boundary.
9. Click **OK** to accept the stock setup.

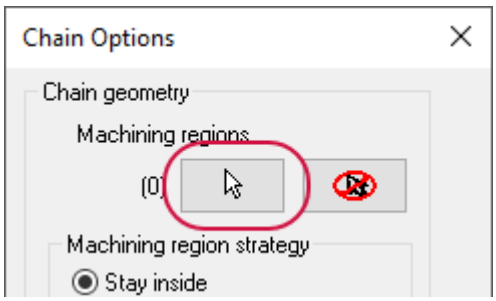
Exercise 2: Creating the first Dynamic Contour toolpath

1. Select **Dynamic Contour** from the **2D** gallery on the **Mill Toolpaths** contextual tab.



The **Chain Options** dialog box displays.

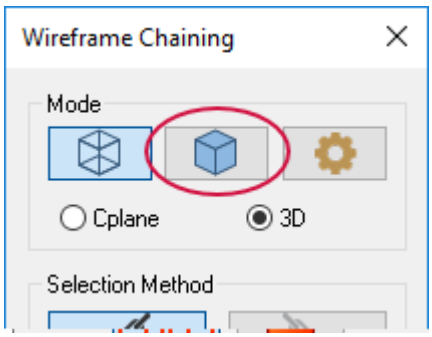
2. Click **Select** under **Machining regions**.



The chaining dialog box displays.

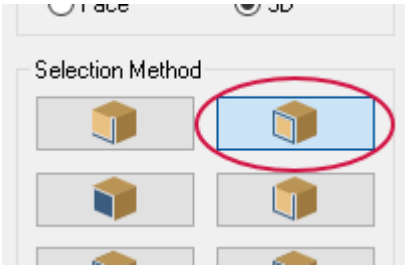
3. If the **Wireframe Chaining** dialog box displayed, select **Solids** to activate solid chaining mode.

From this point on, this tutorial assumes that you know to switch from one version of the chaining dialog box to the other, as needed.



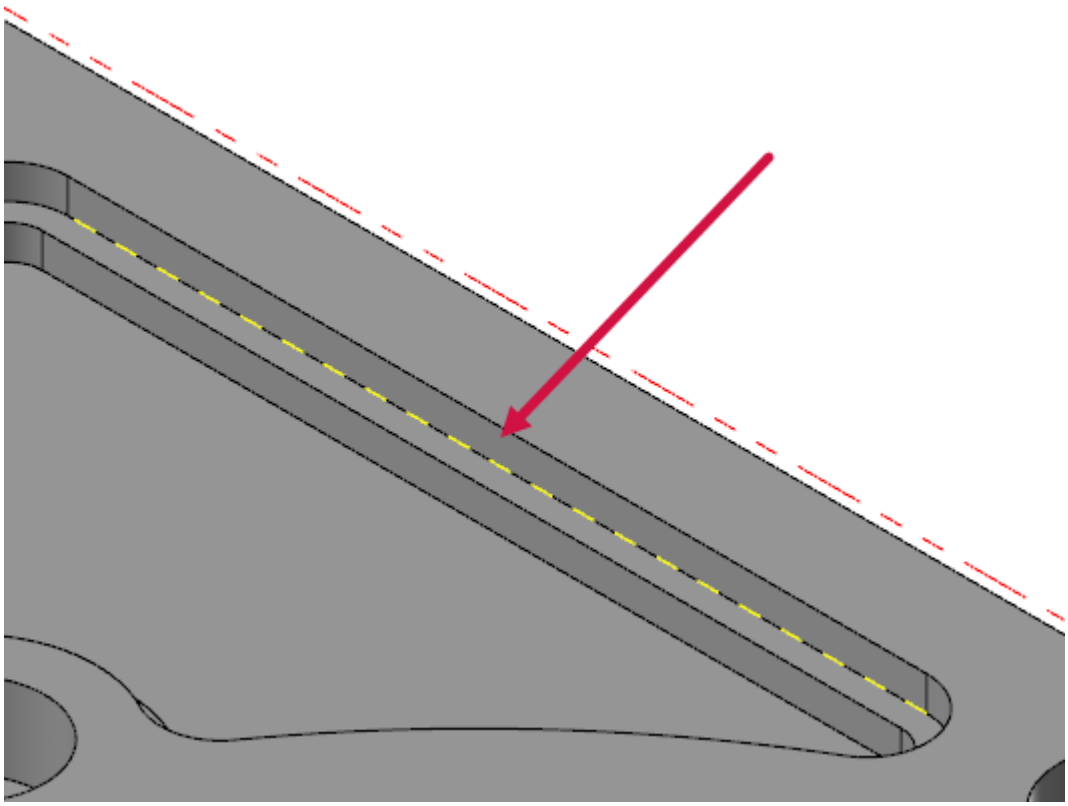
In solid chaining mode, the **Solid Chaining** dialog box gives you options to manage the chaining of solid entities.

4. Select **Loop** if it is not already selected. Deselect **Face** if it is selected.

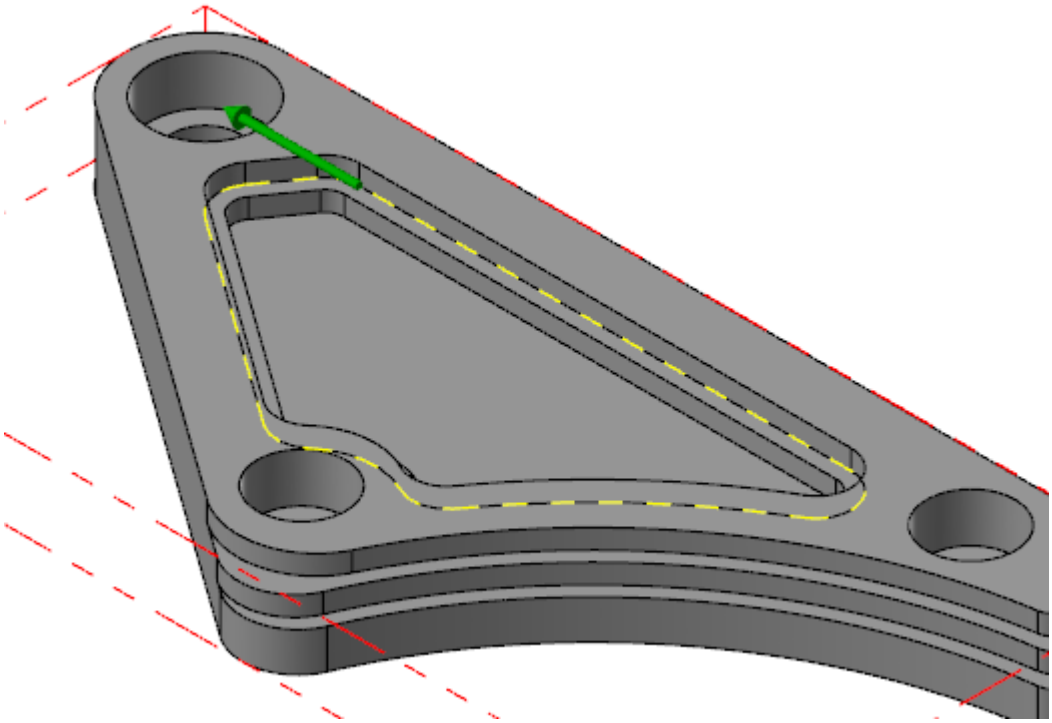


This allows you to select a closed chain on a reference face.

5. Select the edge shown below. You may need to zoom in.

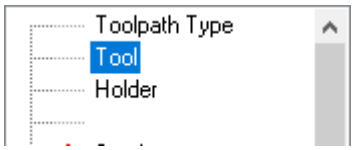


6. If necessary, click **Reverse** so that the chain looks like the following image.



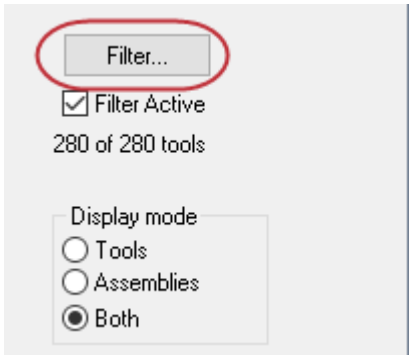
7. Click **OK** in the **Solid Chaining** dialog box to accept the chains and return to the **Chain Options** dialog box.
8. Click **OK** in the **Chain Options** dialog box.

   The **2D High Speed Toolpath - Dynamic Contour** dialog box displays.
9. Select the **Tool** page.



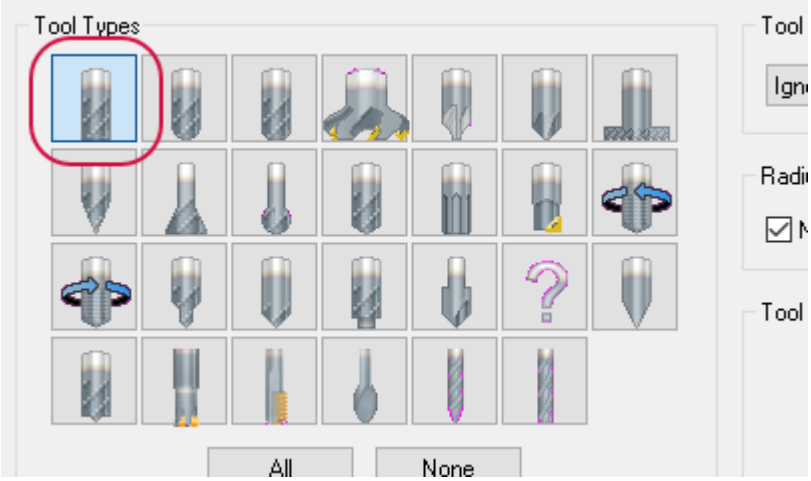
10. Click **Select library tool**.

    The **Tool Selection** dialog box displays.
11. Click **Filter**.



    The **Tool List Filter** dialog box displays.
12. Set the **Tool Type** filter to **None**.

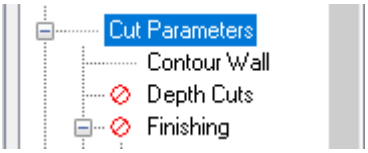
13. Select the **Endmill1 Flat** filter. This ensures that you are only selecting flat end mill tools.



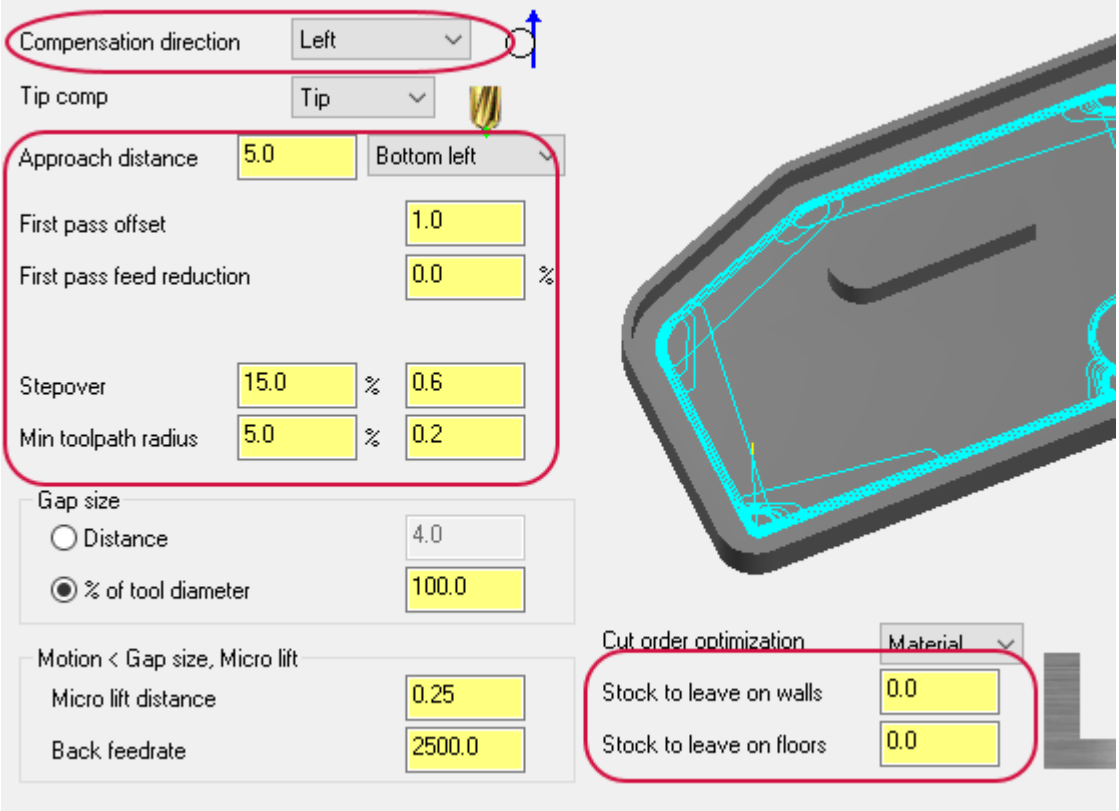
14. Click **OK** in the **Tool List Filter** dialog box.
15. In the **Tool Selection** dialog box, select the **FLAT END MILL - 4** tool.

Assembly Na...	Tool Name	Holder Name	Dia.
--	FLAT END MILL - 3	--	3.0
--	FLAT END MILL - 4	--	4.0
--	FLAT END MILL - 5	--	5.0
--	FLAT END MILL - 6	--	6.0
--	FLAT END MILL - 8	--	8.0
--	FLAT END MILL - 10	--	10.0

16. Click **OK** in the **Tool Selection** dialog box.
17. Select the **Cut Parameters** page.

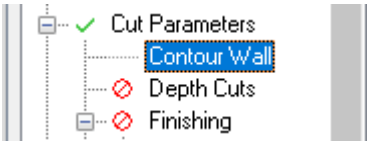


18. Set the following parameters:

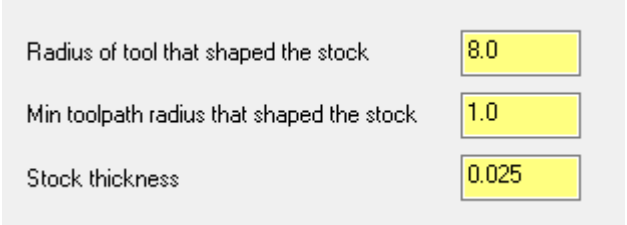


- **Compensation direction** to **Left**. Offsets the tool to the right or left of the toolpath.
- **Approach distance** to **5.0**. Adds a specified incremental distance to the beginning of the toolpath's first cut.
- **First pass offset** to **1.0**. Offsets the machining region to ensure the tool does not engage too much material during the first pass of the toolpath when entering stock from the outside.
- **Stepover** to **15.0**. Sets the distance between cutting passes in the X and Y axes.
- **Minimum toolpath radius** to **5.0**. Sets the minimum toolpath radius for the operation.
- Set **Stock to leave on walls** and **Stock to leave on floors** to **0.0**. You are not leaving stock on the vertical and horizontal surfaces.

19. Select the **Contour Wall** page.



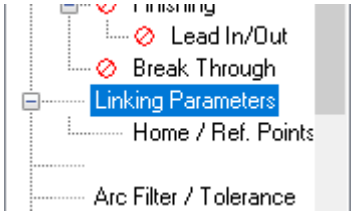
20. Set the following parameters:



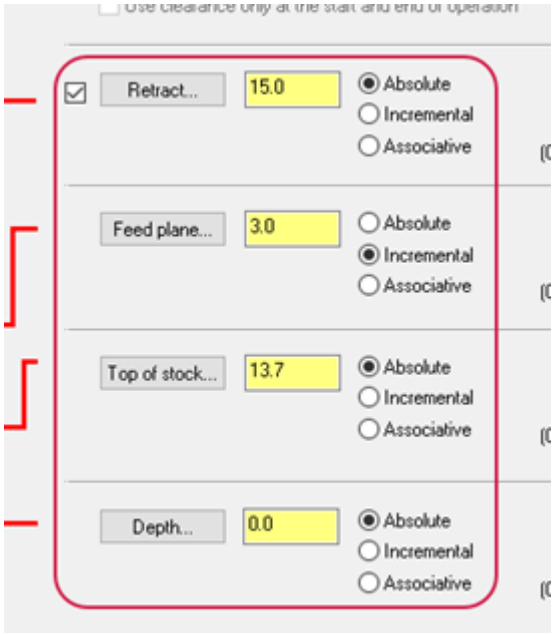
- **Radius of tool that shaped the stock** to **8.0**.
- **Minimum toolpath radius that shaped the stock** to **1.0**.
- **Stock thickness** to **0.025**.

The parameters help define the shape of the stock removed by the toolpath. Mastercam calculates the stock to remove along the contour wall by using these parameters.

21. Select the **Linking Parameters** page.

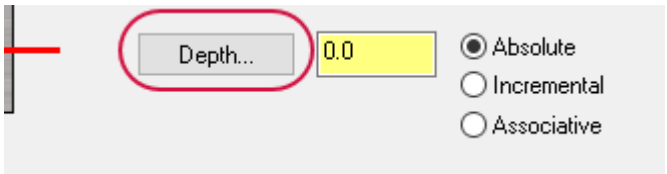


22. Set the following parameters:



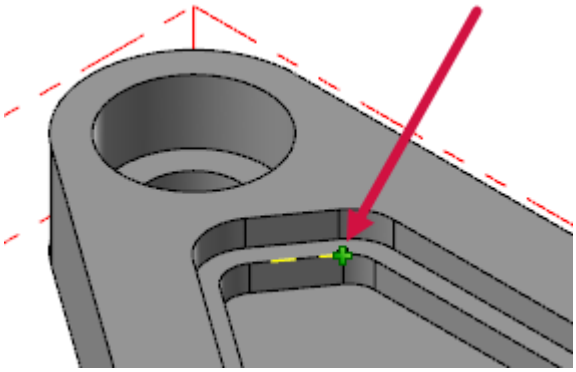
- **Retract**, **Top of stock**, and **Depth** to **Absolute**. Measures absolute values from the origin 0,0,0.
- **Retract** to **15.0**. Sets the height that the tool moves up before the next tool pass.
- **Feed plane** to **3.0**. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- **Top of stock** to **13.7**. Sets the height of the material in the Z axis.

23. Select **Depth**. You return to the graphics window.



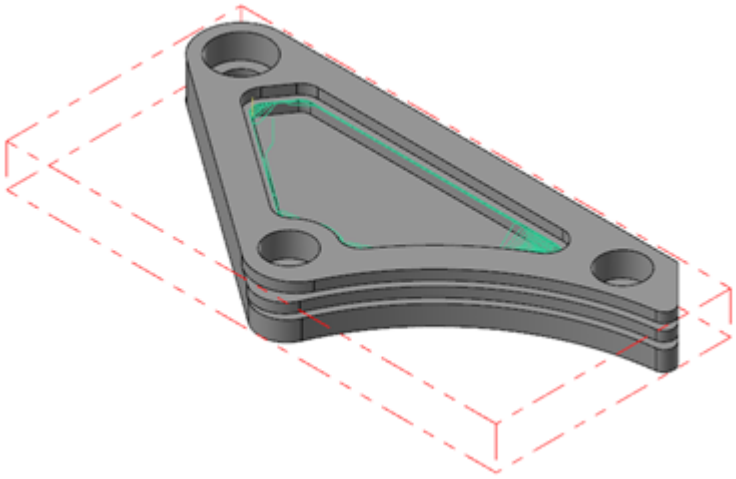
Depth determines the final machining depth and the lowest depth that the tool descends into stock. You will select an edge for this toolpath.

24. Select the edge shown in the following image:



You automatically return to the **Linking Parameters** page. Your **Depth** should now be **9.525**.

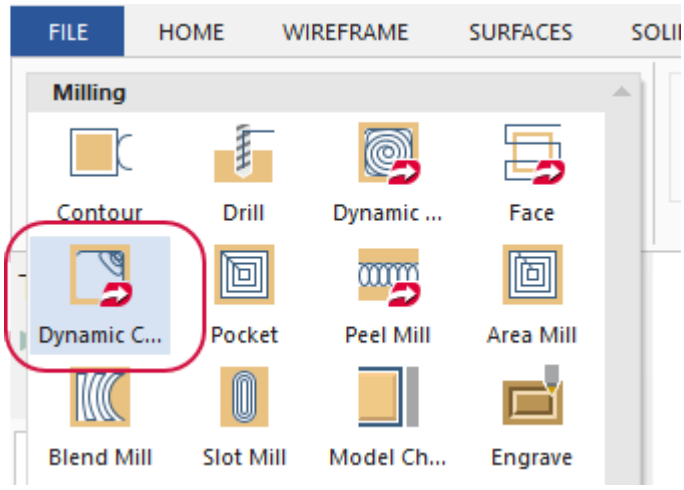
25. Click **OK** in the **2D High Speed Toolpath - Dynamic Contour** dialog box to accept and generate your toolpath. Your toolpath displays as shown below:



26. Save your part.

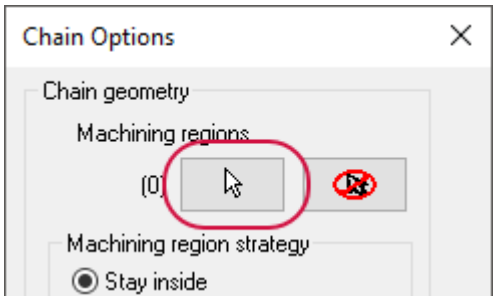
Exercise 3: Creating the second Dynamic Contour toolpath

1. Select **Dynamic Contour** from the **2D** gallery on the **Mill Toolpaths** contextual tab.



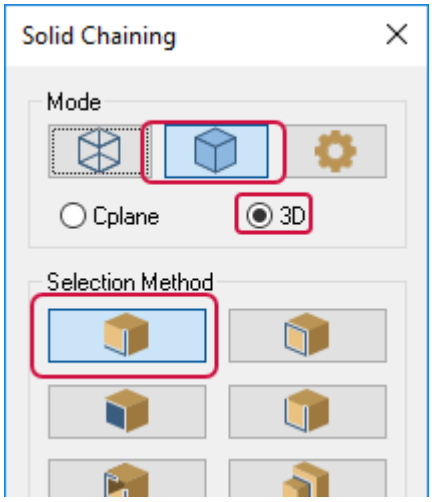
The **Chain Options** dialog box displays.

2. Click **Select** under **Machining regions**.



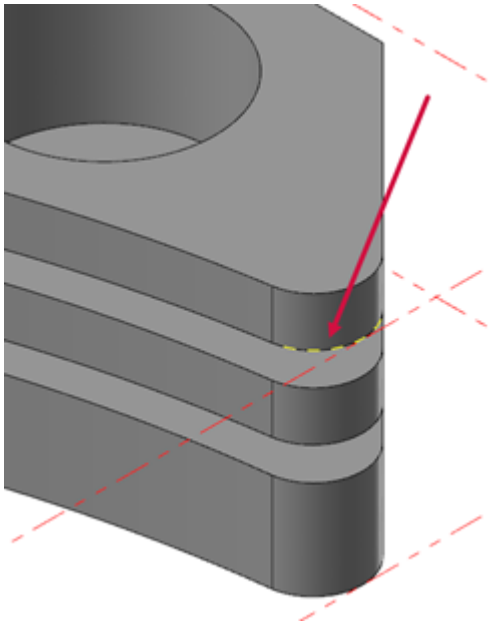
The chaining dialog box displays.

3. Set the chaining to **Edges**. Also ensure that **3D** is selected.

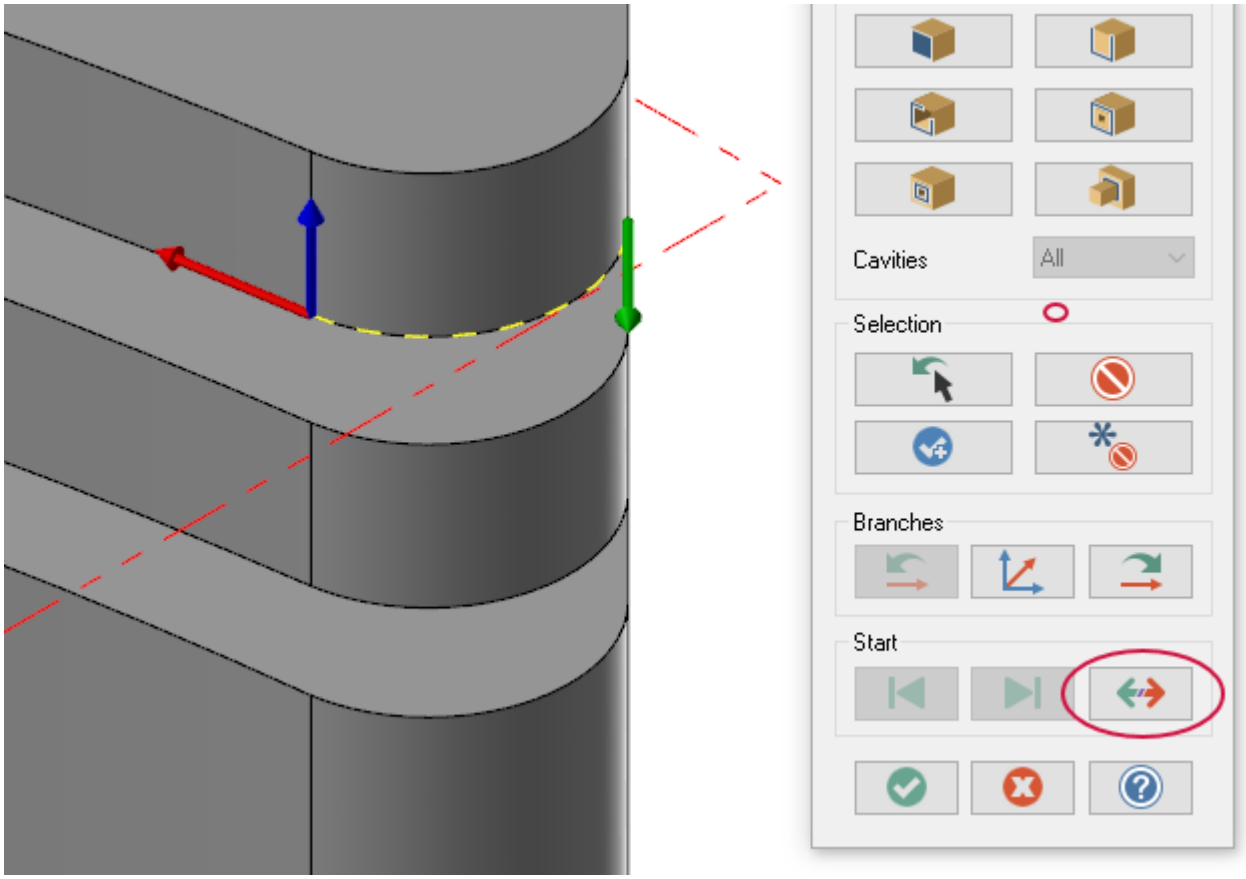


This allows you to select contiguous chaining of edges.

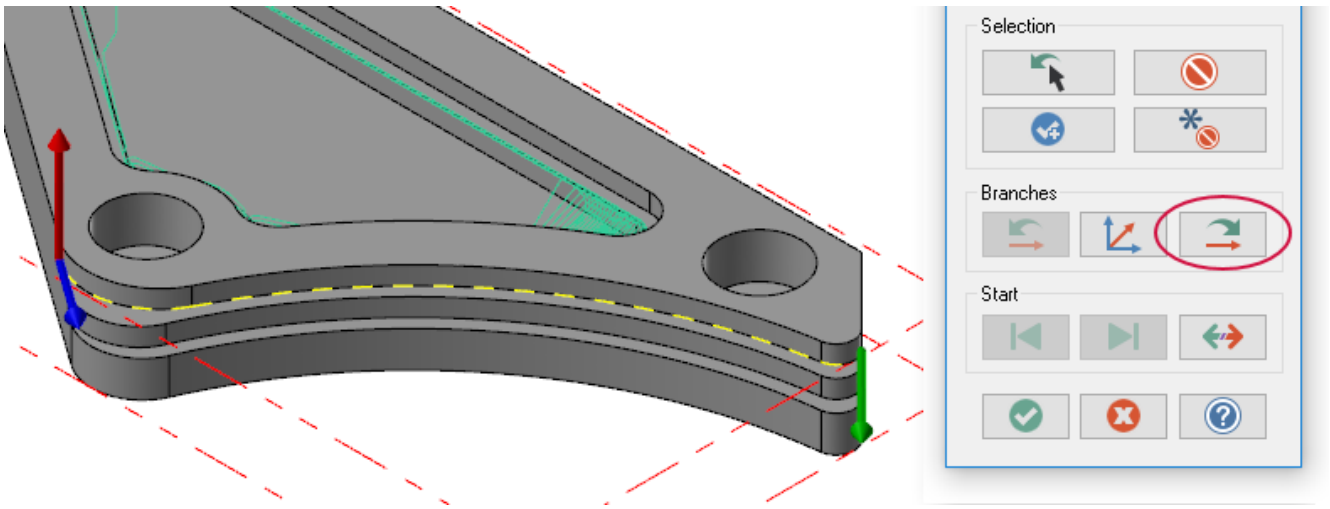
4. Zoom in using the mouse wheel, and select the edge shown below.



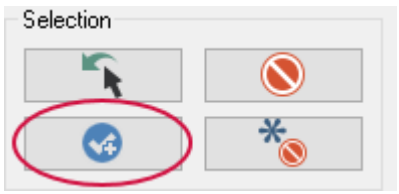
5. The chain will display as shown below. Reverse the direction if needed, using **Reverse** on the chaining dialog box.



6. Use the **Next** control in the **Solid Chaining** dialog box to extend the chain, as shown below.

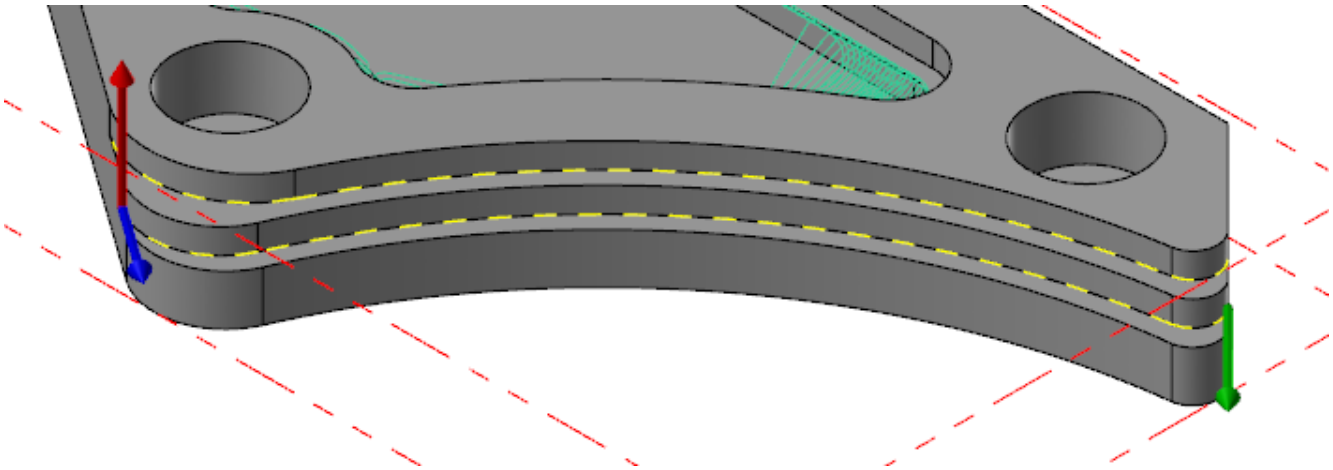


7. In the **Solid Chaining** dialog box, click **End chain**.



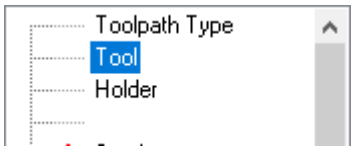
This ends the chain so that you can start a new one that is not linked to the previous chain.

8. Using the controls in the **Solid Chaining** dialog box, select the second partial chain shown below:



9. Click **OK** in the **Solid Chaining** dialog box to return to the **Chain Options** dialog box.
10. Click **OK** in the **Chain Options** dialog box.

    The **2D High Speed Toolpath - Dynamic Contour** dialog box displays.
11. Select the **Tool** page.

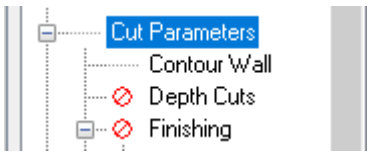


12. Click **Select library tool**.

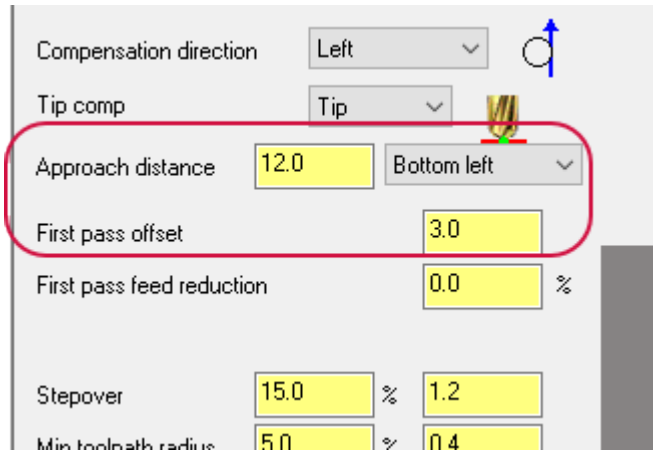
    The **Tool Selection** dialog box displays.
13. Select the **FLAT END MILL - 8** tool.

Assembly Na...	Tool Name	Holder Name	Dia.	Cor. ra
--	FLAT END MILL - 3	--	3.0	0.0
--	FLAT END MILL - 4	--	4.0	0.0
--	FLAT END MILL - 5	--	5.0	0.0
--	FLAT END MILL - 6	--	6.0	0.0
--	FLAT END MILL - 8	--	8.0	0.0
--	FLAT END MILL - 10	--	10.0	0.0
--	FLAT END MILL - 12	--	12.0	0.0
--	FLAT END MILL - 14	--	14.0	0.0
--	FLAT END MILL - 16	--	16.0	0.0

14. Click **OK** in the **Tool Selection** dialog box.
15. Select the **Cut Parameters** page.

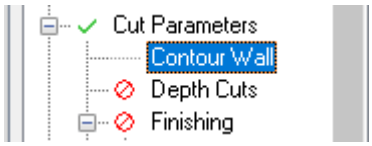


16. Set the following parameters:

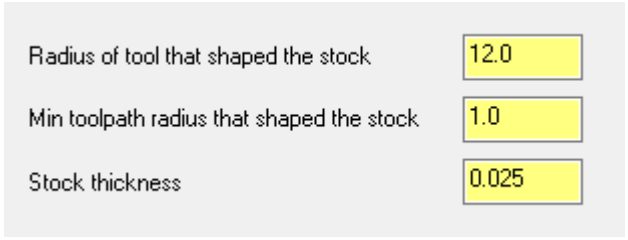


- **Approach distance** to **12.0**.
- **First pass offset** to **3.0**.

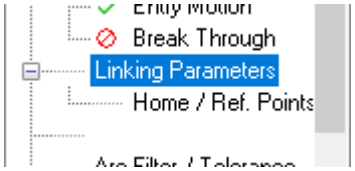
17. Select the **Contour Wall** page.



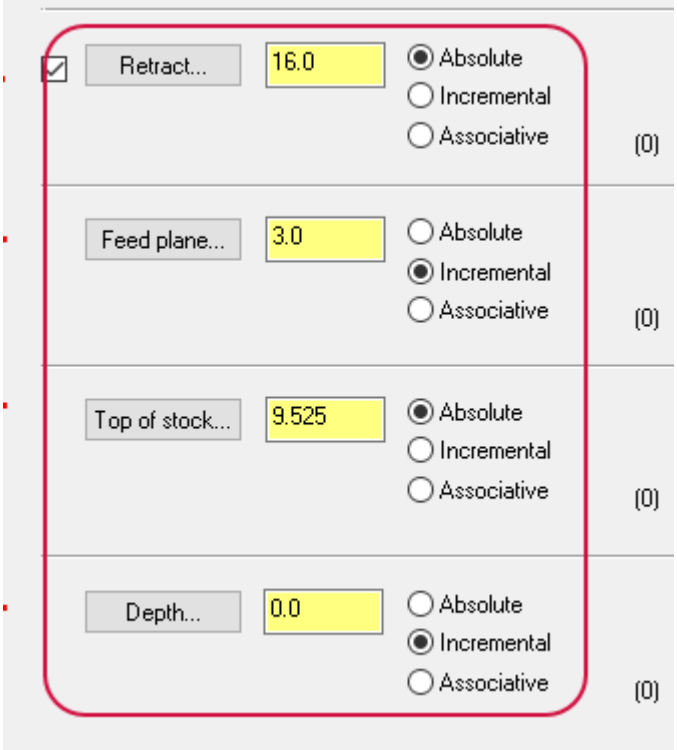
18. Set **Radius of tool that shaped the stock** to **12.0**.



19. Select the **Linking Parameters** page.



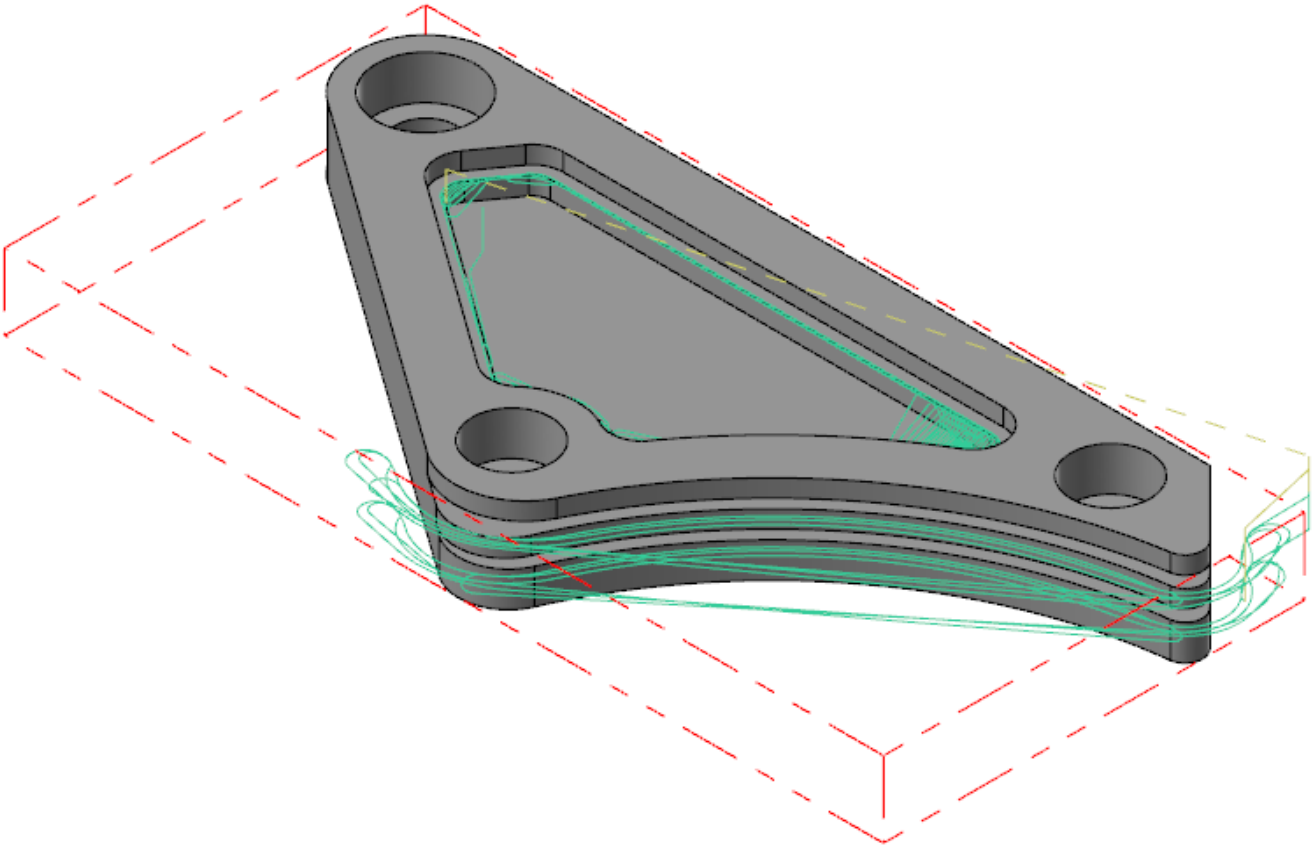
20. Set the following parameters:



- **Retract** to **16.0** and **Absolute**.
- **Feed plane** to **3.0** and **Incremental**. Incremental values are relative to other parameters or chained geometry.
- **Top of stock** to **9.525** and **Absolute**.
- **Depth** to **0.0** and **Incremental**.

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

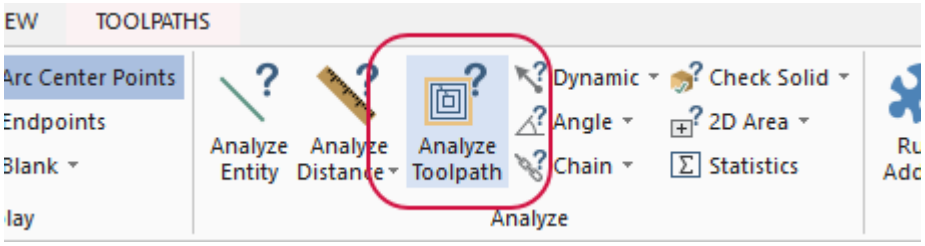
    The toolpath will display as shown below:



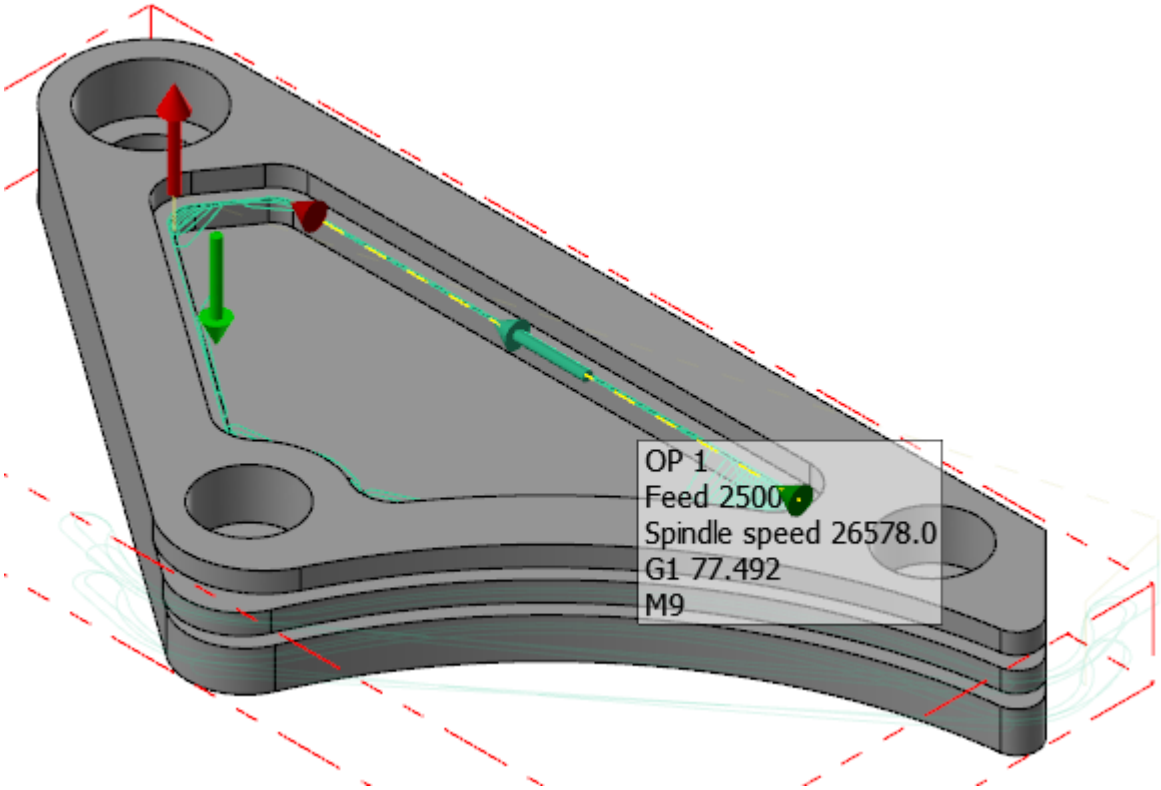
Exercise 4: Analyzing the toolpaths

You now analyze the toolpath. **Analyze Toolpath** allows you to view toolpath properties, such as coordinates, direction, operation number, and other information by hovering over the toolpath.

1. Select **Analyze Toolpath** from the **Home** tab.



2. Hover over the first toolpath:



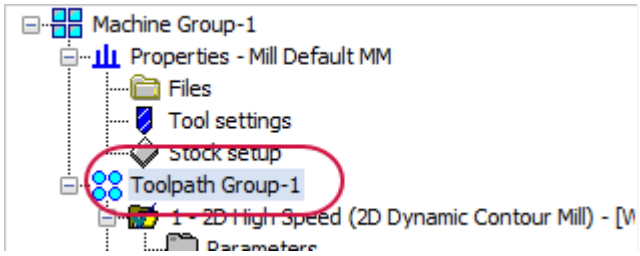
Analyze Toolpath reports the operation number, the feed move, spindle speed, G1 and the line length, and the coolant code. It also displays the start and end of the toolpath and of the section you are hovering over, and sometimes position information. If there is more than one toolpath displayed at the time, then toolpaths other than the one you are hovering over are dimmed.

3. Continue hovering over areas of both toolpaths. Once you are satisfied, press [**Esc**] or **Cancel** out of the function.

Exercise 5: Verifying the toolpaths

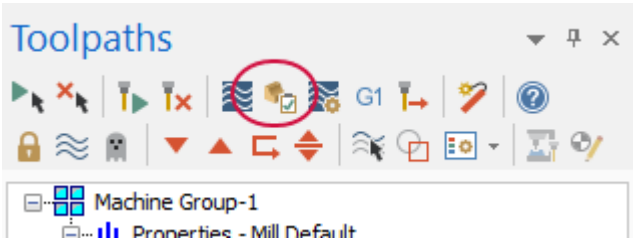
You will now verify both toolpaths.

1. Select **Toolpath Group-1** in the Toolpaths Manager.



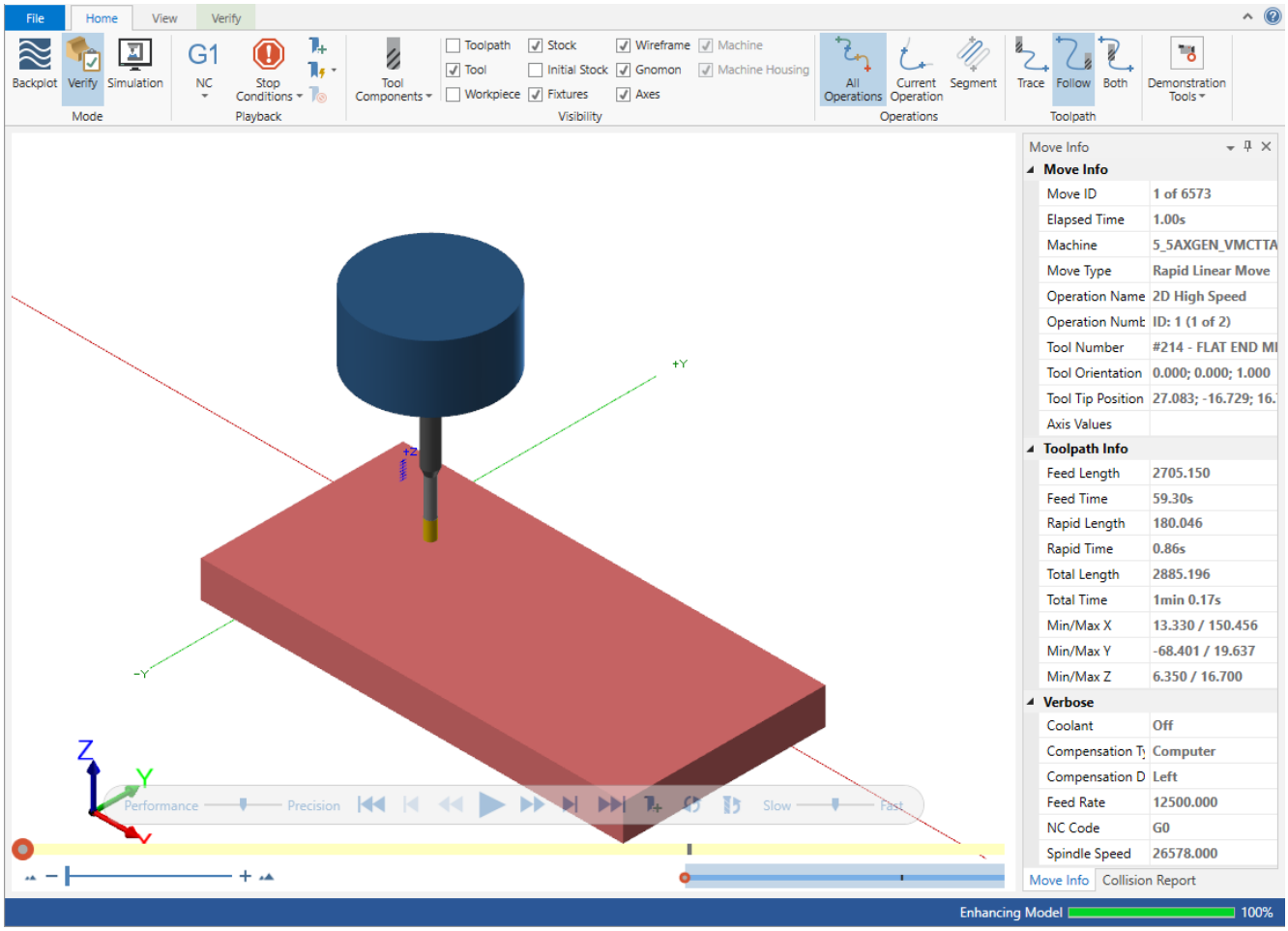
This selects both toolpaths so that they can be verified at the same time.

2. Select **Verify selected operations**.

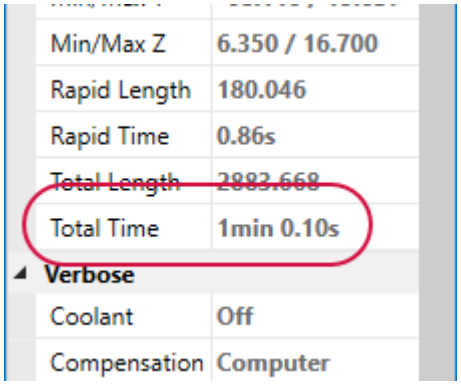


Mastercam Simulator displays.

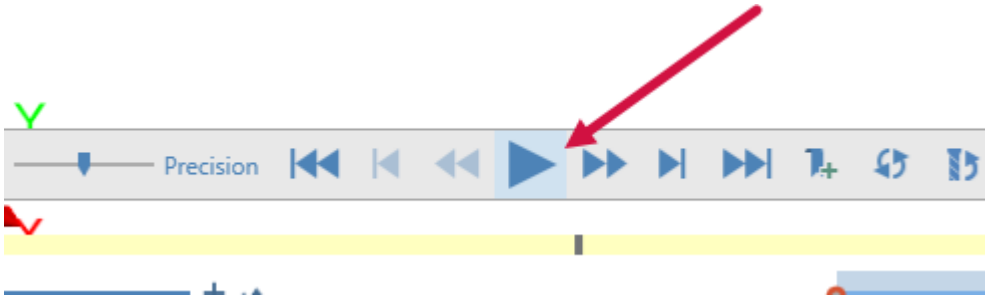
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:

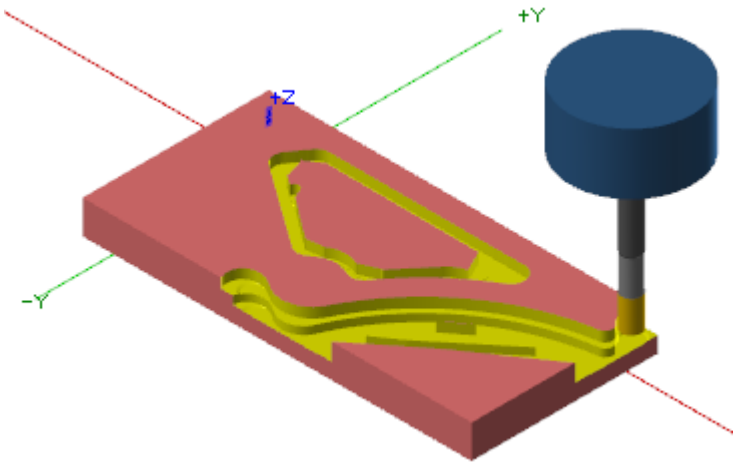


4. View the **Move Info**, which displays all pertinent information about the toolpaths being verified. By using two Dynamic Contour toolpaths, the total machining time is approximately one minute. (You time result may vary.)



5. Press **Play** to preview the toolpath motion for both operations. The second image following shows the results.





6. Once you are satisfied with your results, close Mastercam Simulator.
7. Save your part file.

In the next chapter, you create three Peel Mill toolpaths.

CHAPTER 4

THE PEEL MILL TOOLPATH

The Peel Mill toolpath, with the **Cutting strategy** set to **Dynamic peel**, allows for efficient milling between selected chains. It uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material.

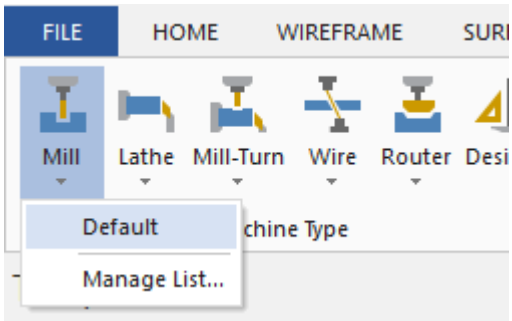
In this chapter, you create one Peel Mill toolpath, copy it twice, and make edits.

Goals

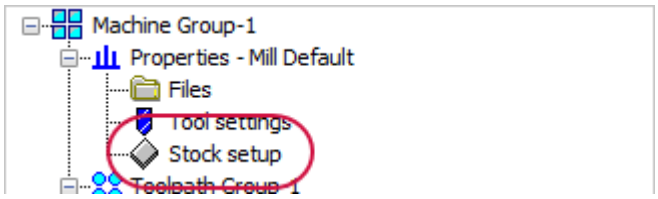
- Create a Peel Mill toolpath
- Copy and paste the toolpath
- Make edits to the copied toolpaths
- Verify the created toolpaths

Exercise 1: Setting up the stock

1. Open the part file, PeelMill, provided with this tutorial.
2. Save the part as PeelMill-xxx, replacing xxx with your initials.
3. On the **Machine** tab, select **Mill**, **Default.**

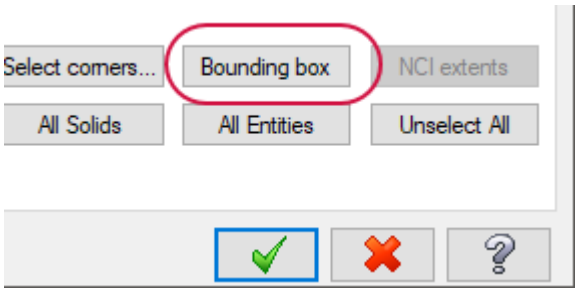


4. Select **Stock setup** from the Toolpaths Manager.



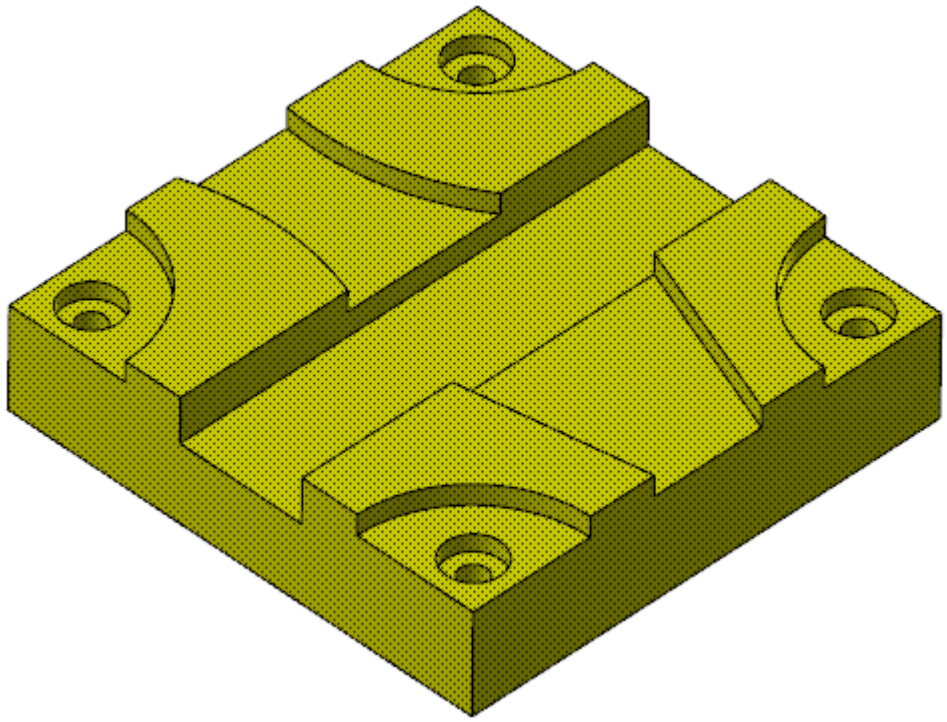
The **Machine Group Properties** dialog box displays.

5. Select **Bounding box**.



The **Bounding Box** function panel displays.

6. Select the part, and press [**Enter**] or click **End Selection**.

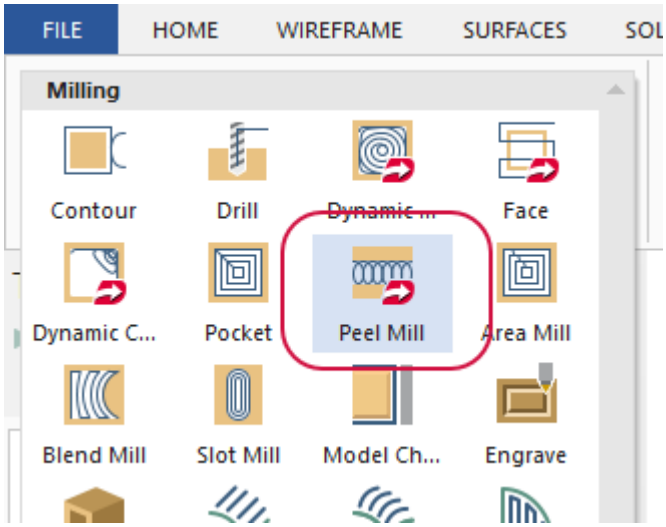


This part needs no additional geometry. The bounding box will be accepted as it is.

7. Click **OK** to create the boundary.
8. Click **OK** to accept the stock setup.

Exercise 2: Creating the first Peel Mill toolpath

1. Select **Peel Mill** from the **2D** gallery on the **Mill Toolpaths** contextual tab.

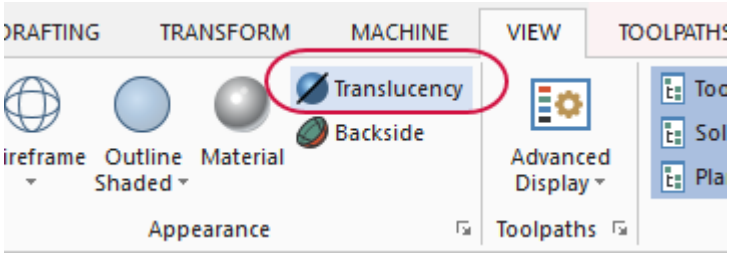


The chaining dialog box displays.

2. Select **Edges** if it has not already been selected. Deselect **Face** if it is selected.

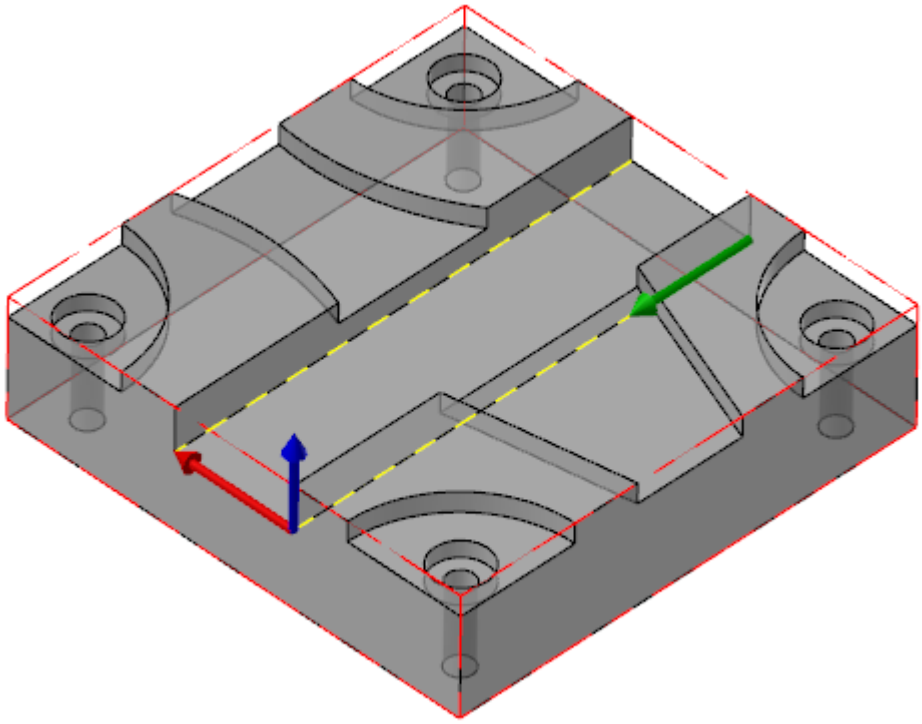
This allows you to select the solid edges of a model.

3. Select **Translucency** on the **View** tab. You can also turn on translucency with the Translucency button in the status bar or by pressing [**Ctrl+T**].



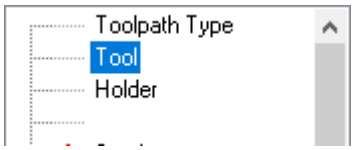
This toggles the translucency of the current model and will help with selection for this first toolpath.

4. Select the two edges shown below. Use **Reverse** if necessary to ensure both chains go in the same direction.

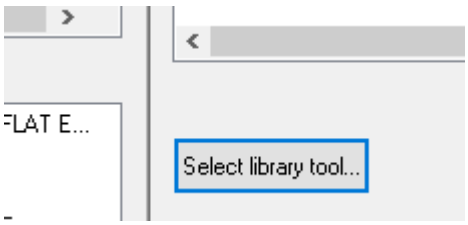


5. Toggle **Translucency** off.
6. Click **OK** in the **Solid Chaining** dialog box.

   The **2D High Speed Toolpath - Peel Mill** dialog box displays.
7. Select the **Tool** page.

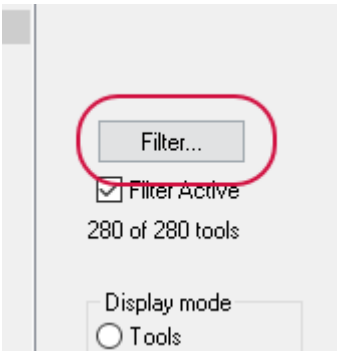


8. Click **Select library tool**.



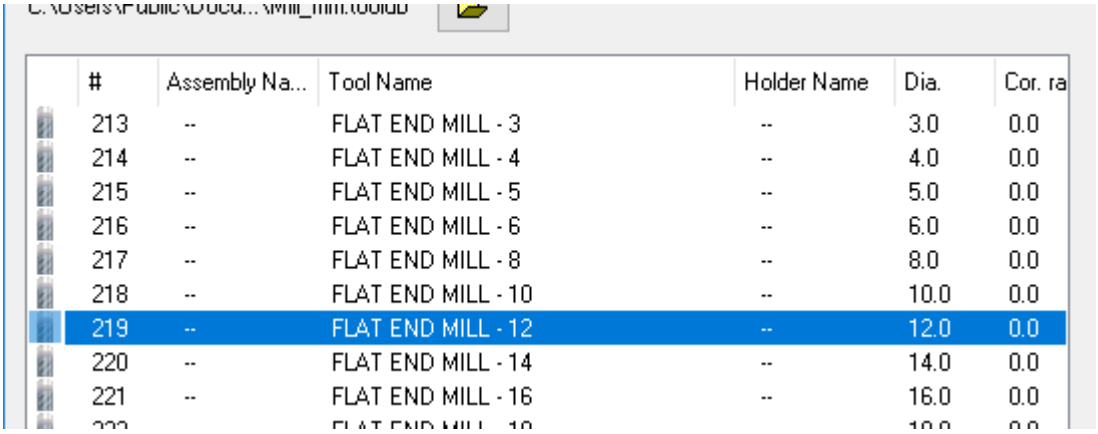
   The **Tool Selection** dialog box displays.

9. Click **Filter**.



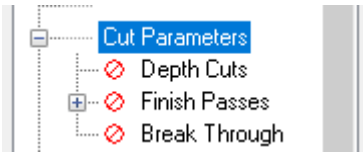
The **Tool List Filter** dialog box displays.

10. Verify that tools are filtered to **Endmill1 Flat**.
11. Click **OK** in the **Tool List Filter** dialog box to accept the filter and return to the **Tool Selection** dialog box.
12. Select the **FLAT END MILL - 12** tool.

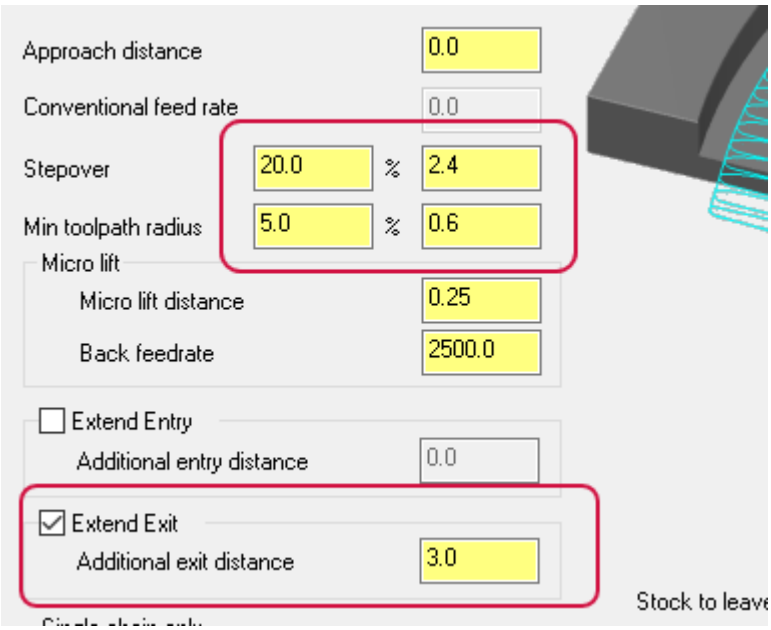


#	Assembly Na...	Tool Name	Holder Name	Dia.	Cor. ra
213	--	FLAT END MILL - 3	--	3.0	0.0
214	--	FLAT END MILL - 4	--	4.0	0.0
215	--	FLAT END MILL - 5	--	5.0	0.0
216	--	FLAT END MILL - 6	--	6.0	0.0
217	--	FLAT END MILL - 8	--	8.0	0.0
218	--	FLAT END MILL - 10	--	10.0	0.0
219	--	FLAT END MILL - 12	--	12.0	0.0
220	--	FLAT END MILL - 14	--	14.0	0.0
221	--	FLAT END MILL - 16	--	16.0	0.0

13. Click **OK** to add that tool and exit the **Tool Selection** dialog box.
14. Select the **Cut Parameters** page.

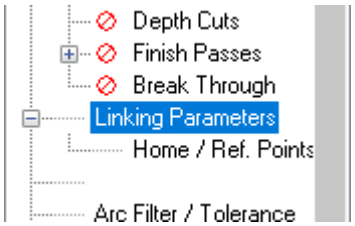


15. Set the following parameters:

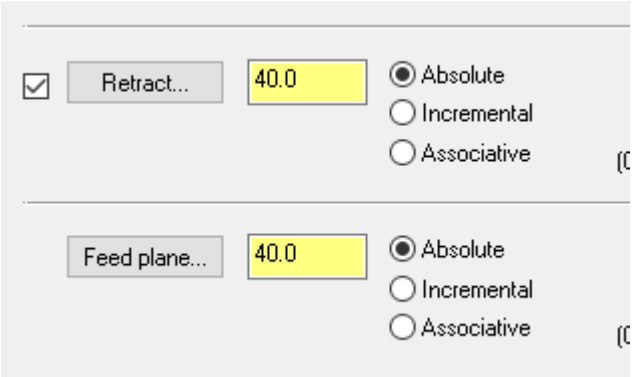


- **Stepover** to **20.0%**.
- **Minimum toolpath radius** to **5.0%**.
- Select **Extend Exit** and set **Additional exit distance** to **3.0**. This adjusts the final tool engagement with the material.

16. Select the **Linking Parameters** page.

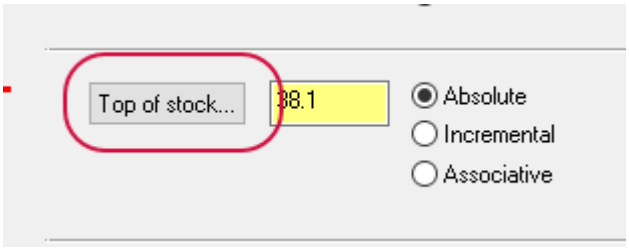


17. Set the following parameters:

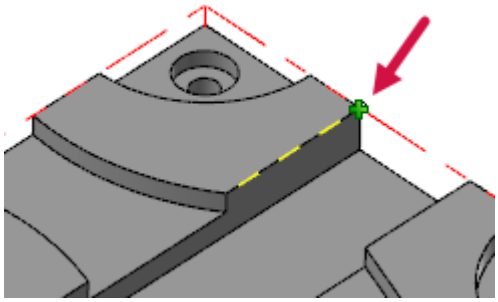


- Set **Retract** to **40.0**.
- Set **Feed plane** to **40.0**.

18. Click **Top of stock** to return to the graphics window to select the top of the stock of the part.



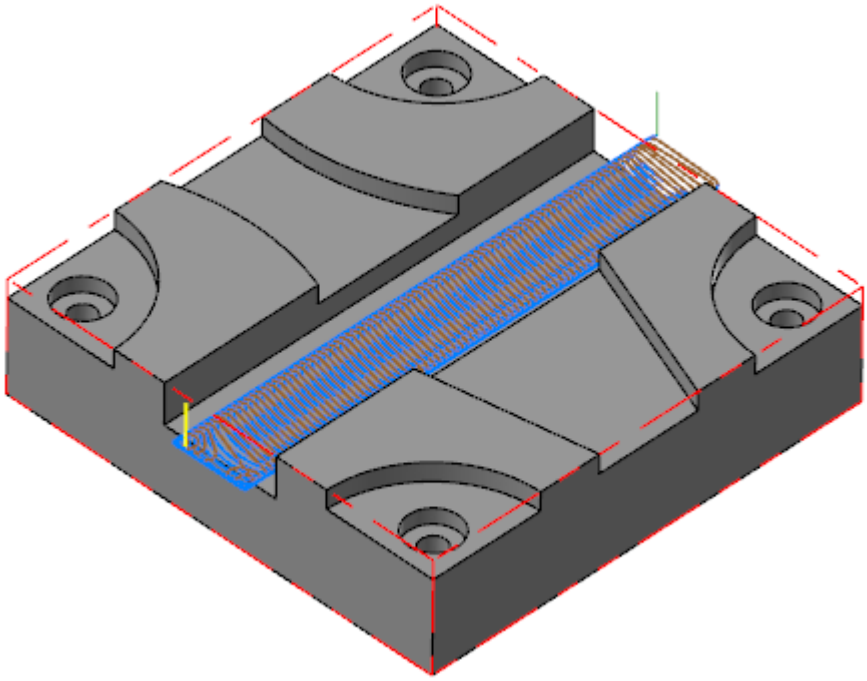
19. Select the point shown below:



You automatically return to the **Linking Parameters** page. **Top of stock** will be set at **38.1**.

20. Click **OK** in the **2D High Speed Toolpaths - Peel Mill** dialog box to generate the toolpath.

The toolpath will display as shown below:

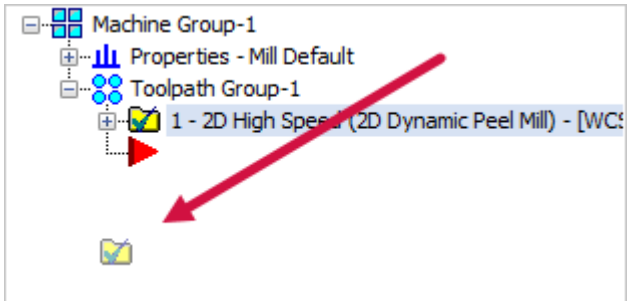


21. Save your file.

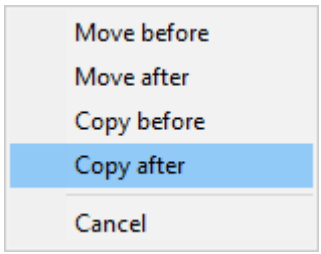
Exercise 3: Copying the Peel Mill toolpath

In this exercise, you copy the Peel Mill operation twice in the Toolpaths Manager.

1. Right-click on the **2D High Speed (2D Dynamic Peel Mill)** toolpath, and drag and drop it below the red insert arrow.

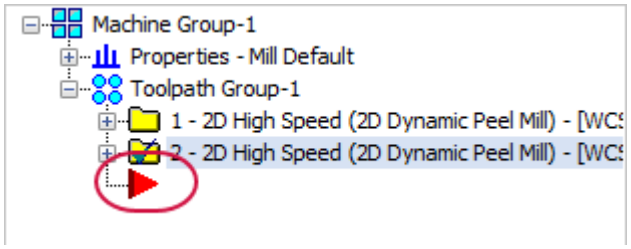


2. Select **Copy after** from the right-click menu that displays.



Mastercam copies the toolpath below the original. There are now two Peel Mill toolpaths in the Toolpaths Manager.

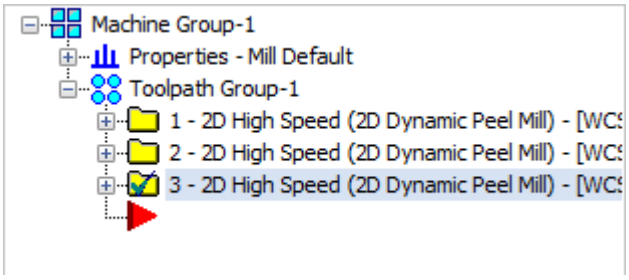
3. Click and drag the red arrow to position it below the copied operation.



4. Copy the toolpath once more. You will now have three Peel Mill toolpaths in the Toolpaths Manager.

   Copying and pasting toolpaths is helpful when you have to do similar cuts on the same part. In the case of these toolpaths, you will have to re-chain the geometry.

   a. Click and drag the red arrow below the third toolpath.

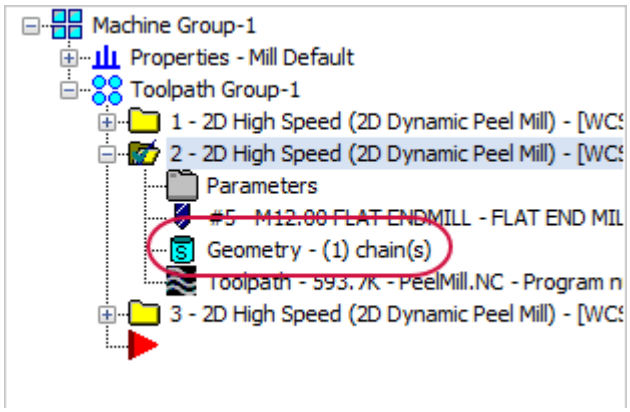


5. Save your file.

Exercise 4: Re-chaining the copied Peel Mill operations

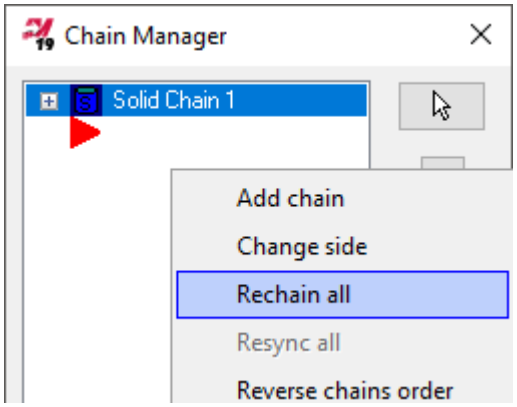
In this exercise, you will re-chain the geometry for the Peel Mill toolpaths that were copied.

1. Select **Geometry** under the second Peel Mill toolpath.



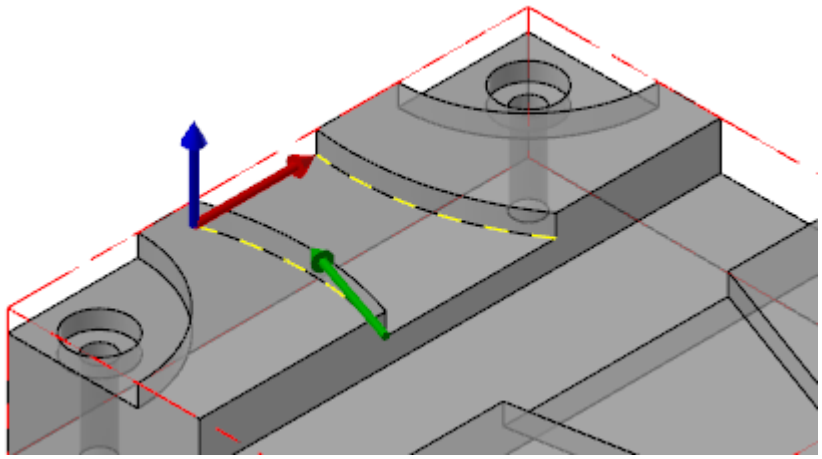
   The **Chain Manager** dialog box displays.

2. Right-click in the **Chain Manager** dialog box, and select **Rechain all**.



   The chaining dialog box displays.

3. Select the chains shown below. Ensure that the chains are both going in the same direction. You may need to rotate the part or select **Translucency** on the **View** tab to select them.

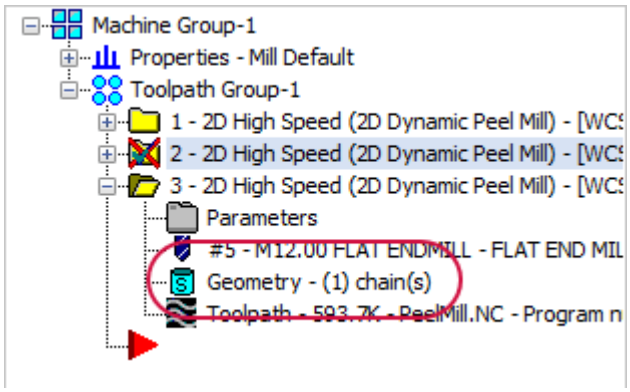


4. Click **OK** in the **Solid Chaining** dialog box to return to the **Chain Manager** dialog box.

5. Click **OK** in the **Chain Manager** dialog box.

   Before regenerating the toolpath, you will re-chain the third operation.

6. Select **Geometry** under the third Peel Mill toolpath.

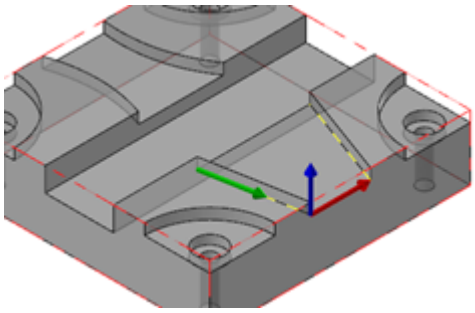


   The **Chain Manager** dialog box displays.

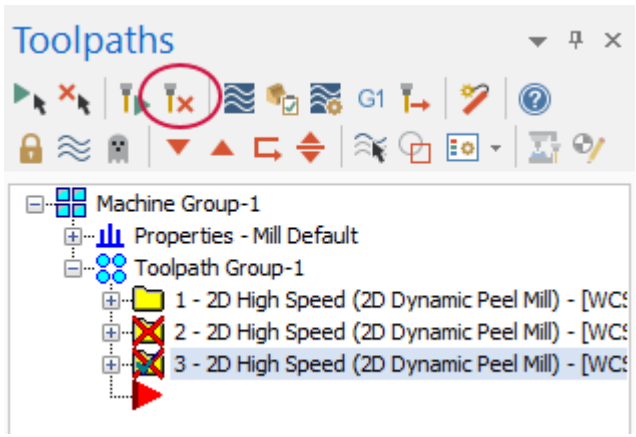
7. Right-click in the **Chain Manager** dialog box and select **Rechain all**.

   The **Solid Chaining** dialog box displays.

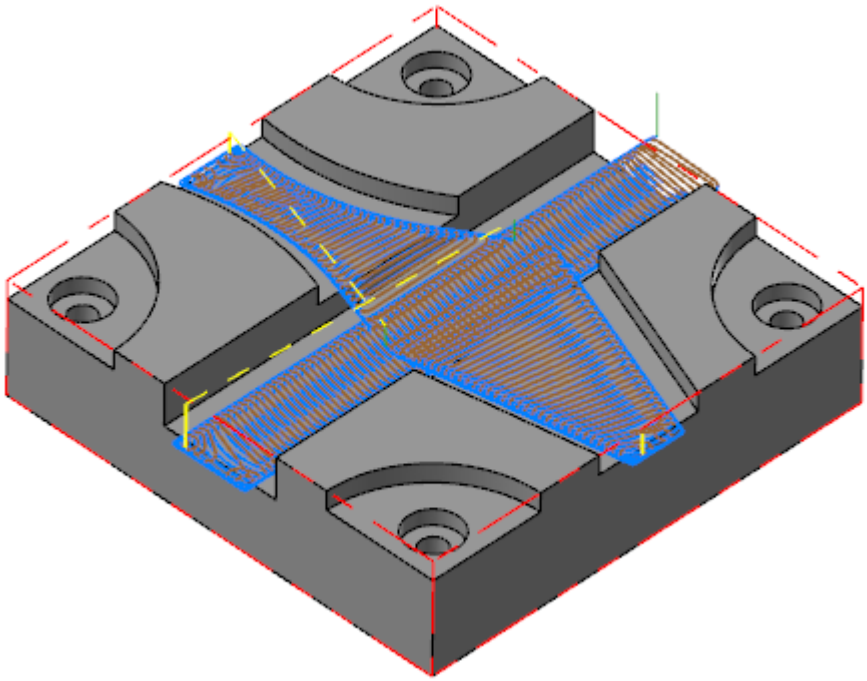
8. Select the chains shown below. Ensure that the chains are both going in the same direction.



9. Click **OK** in the chaining dialog box to return to the **Chain Manager** dialog box.
10. Click **OK** in the **Chain Manager** dialog box.
11. Select **Regenerate all dirty operations** in the Toolpaths Manager.



   The second and third Peel Mill toolpaths regenerate using the newly selected chains. All three Peel Mill toolpaths will display as shown below:

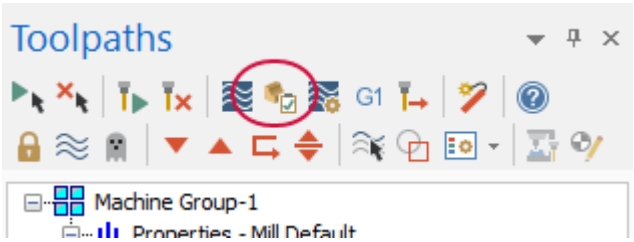


12. Save your file.

Exercise 5: Verifying the toolpaths

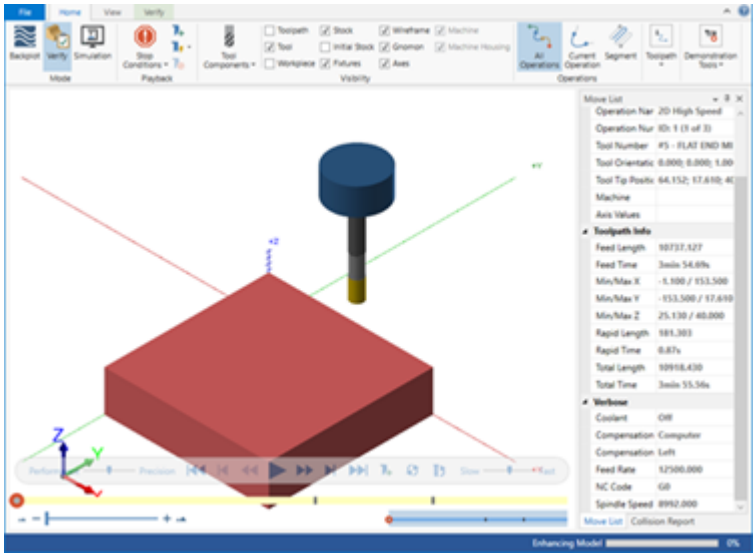
In this exercise, you will verify all three Peel Mill operations together.

1. Select **Toolpath Group-1** in the Toolpaths Manager to select all of the toolpaths.
2. Select **Verify selected operations**.

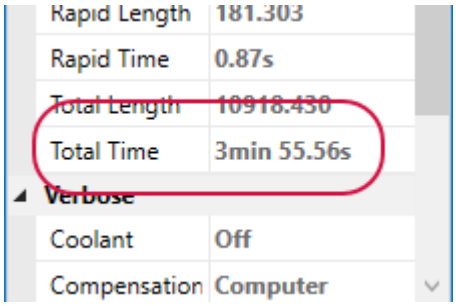


Mastercam Simulator displays.

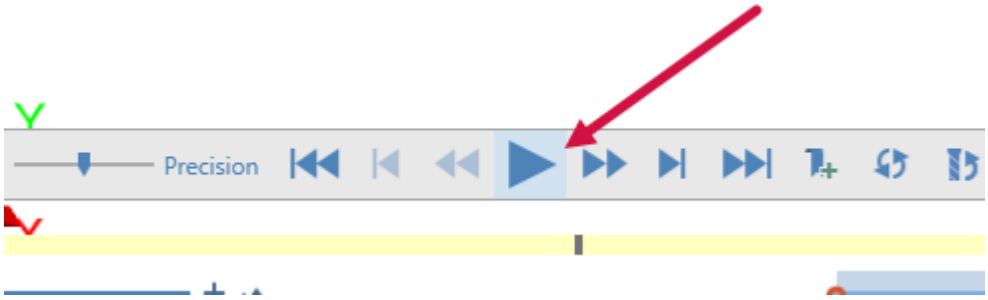
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:



4. View the **Move Info** to view the total run time of the toolpaths.



5. Press **Play** to preview the toolpath motion for the operations.



6. Once you are satisfied with your results, close Mastercam Simulator.
7. Save your part file.

When creating similar toolpaths, copying and pasting saves time. In the next chapter, you create two Dynamic OptiRough operations and a stock model operation.

CHAPTER 5

THE DYNAMIC OPTIROUGH TOOLPATH

The 3D surface high-speed Dynamic OptiRough toolpath uses an aggressive, fast, intelligent roughing algorithm based on Mastercam's 2D high-speed dynamic milling motion.

In this chapter, you create two Dynamic OptiRough toolpaths using different cutting strategies and a stock model operation in order to rough out the part.

You must have a 3D license to complete this Dynamic OptiRough lesson.

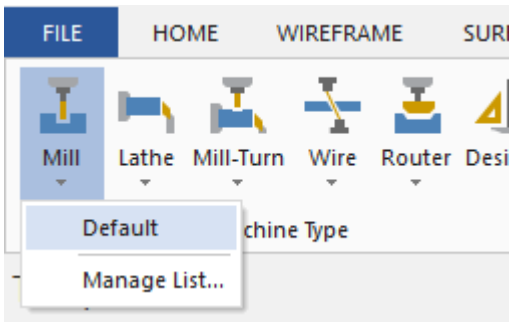
Goals

- Set up the stock
- Create two Dynamic OptiRough toolpaths
- Create a stock model operation

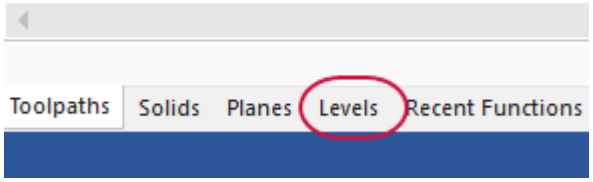
Exercise 1: Setting up the stock

In this exercise, you create the stock model that will be used for the first Dynamic OptiRough toolpath.

1. Open the part file, DynamicOptiRough, provided with this tutorial.
2. Save the part as DynamicOptiRough-xxx, replacing xxx with your initials.
3. On the **Machine** tab, select **Mill**, **Default.**

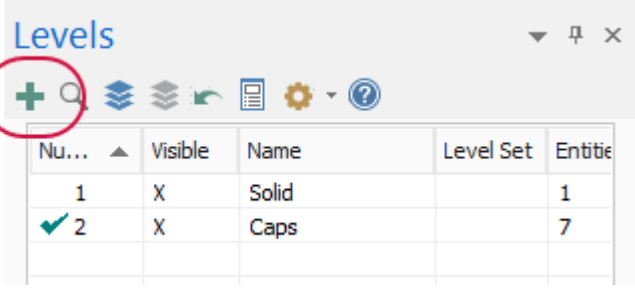


4. Select the Levels Manager, located in the same area as the Toolpaths Manager. If it is not available, select **Levels** from the **View** tab to toggle the display.



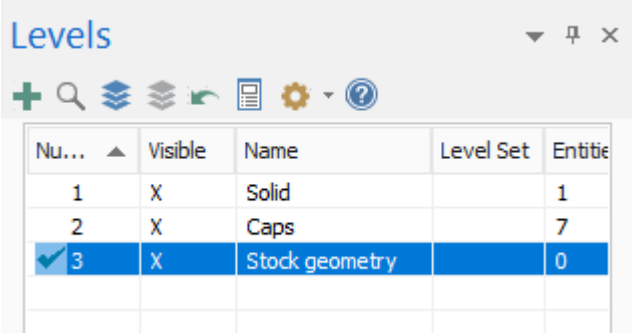
Before creating the stock shape, you will first create a level on which to place the stock geometry. This is a good practice for when you need to organize part files that have thousands of lines, arcs, solids, or surfaces.

5. On the Levels Manager, select **Add a new level**.



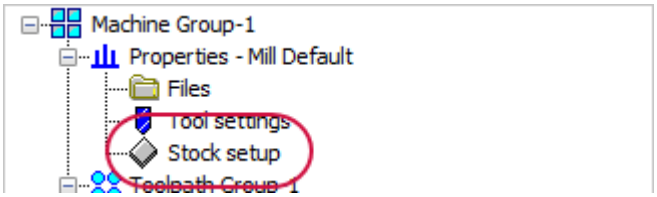
A new level is added to the list and is then set as the main level.

6. Double-click in the **Name** column of the level you have just created and rename it to **Stock geometry**.



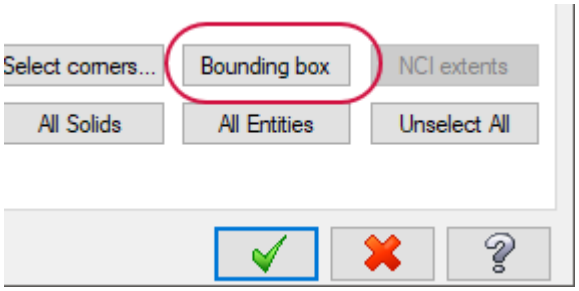
This level will now be used to place the geometry created by the stock setup.

7. Select **Stock setup** from the Toolpaths Manager.



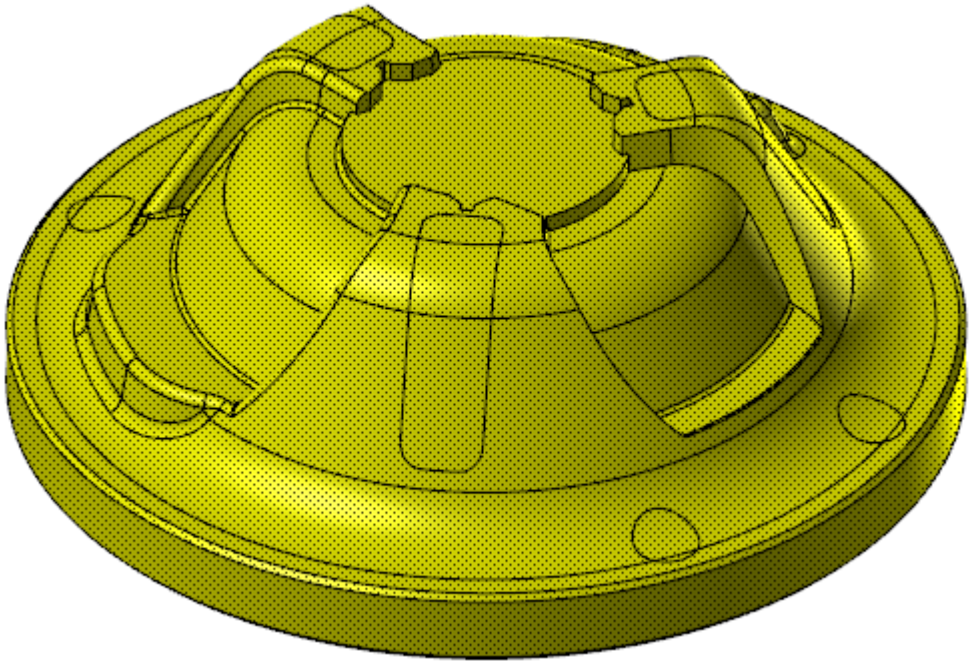
The **Machine Group Properties** dialog box displays.

8. Select **Bounding box**.

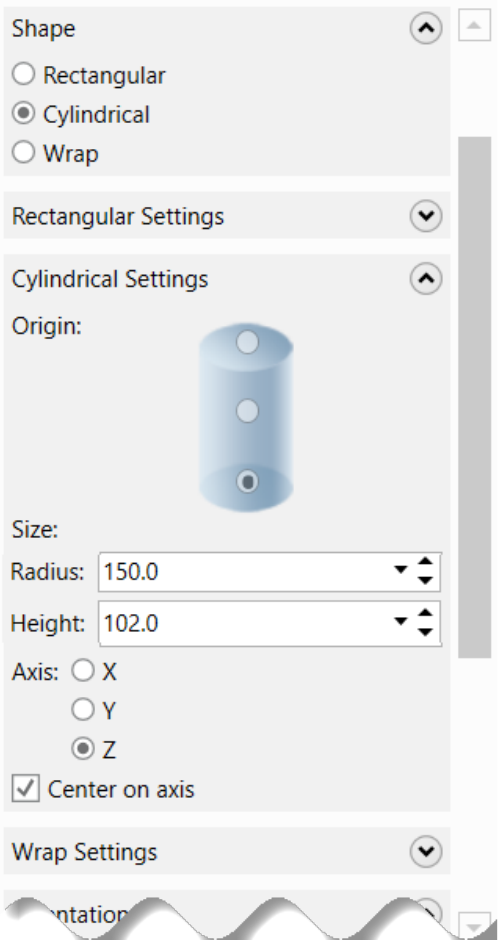


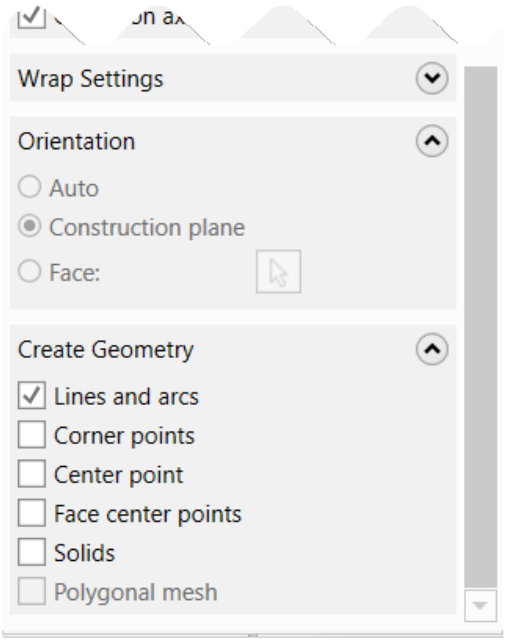
The **Bounding Box** function panel displays.

9. Window select all entities to create a bounding box around the entire part.



10. Press [**Enter**] to accept your selection.
11. Set the following parameters in the **Bounding Box** function panel:

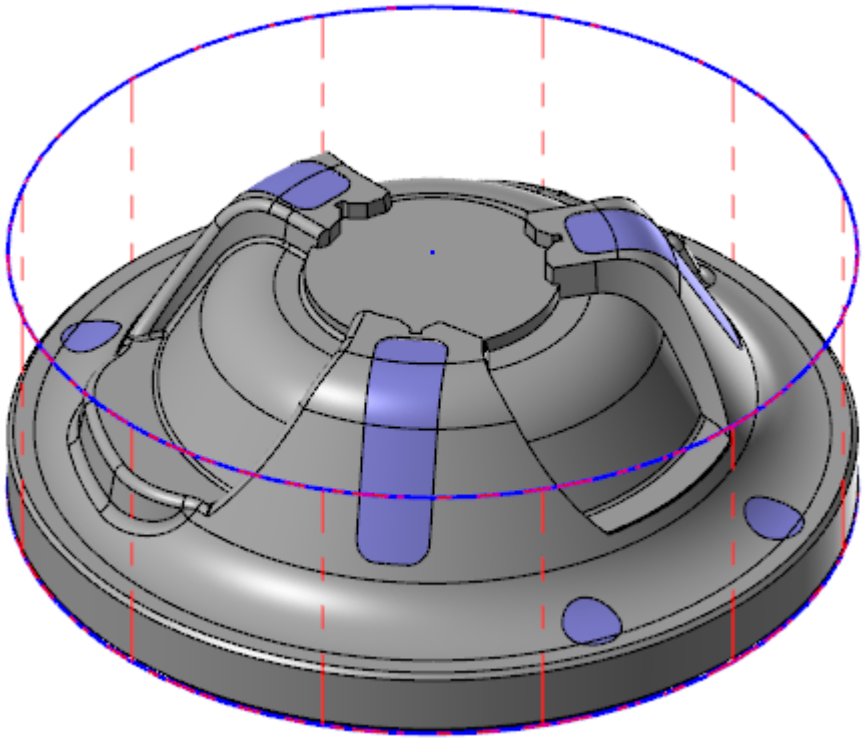




- Set **Shape** to **Cylindrical**.
- Set **Radius** to **150**.
- Set **Height** to **102.00**. This will add a small amount of stock at the top of the part.
- Set **Axis** to **Z**.
- **Center on axis** should be on.
- **Lines and arcs** should be on.

12. Click **OK** to create the boundary.
13. Click **OK** in the **Machine Group Properties** dialog box to accept the stock setup.

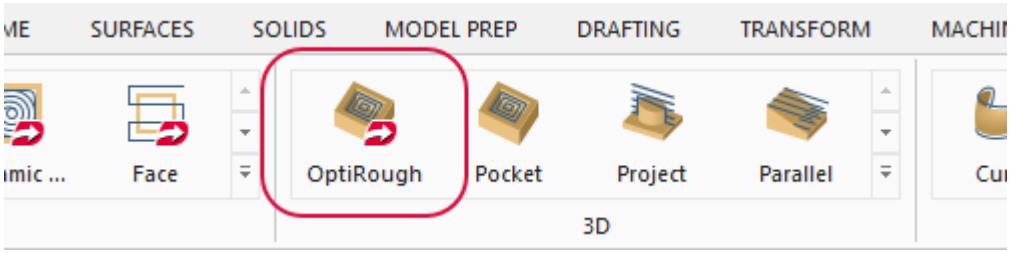
    The stock displays as a dashed outline in red. The geometry that was created with the bounding box also displays.



Exercise 2: Creating the first Dynamic OptiRough toolpath

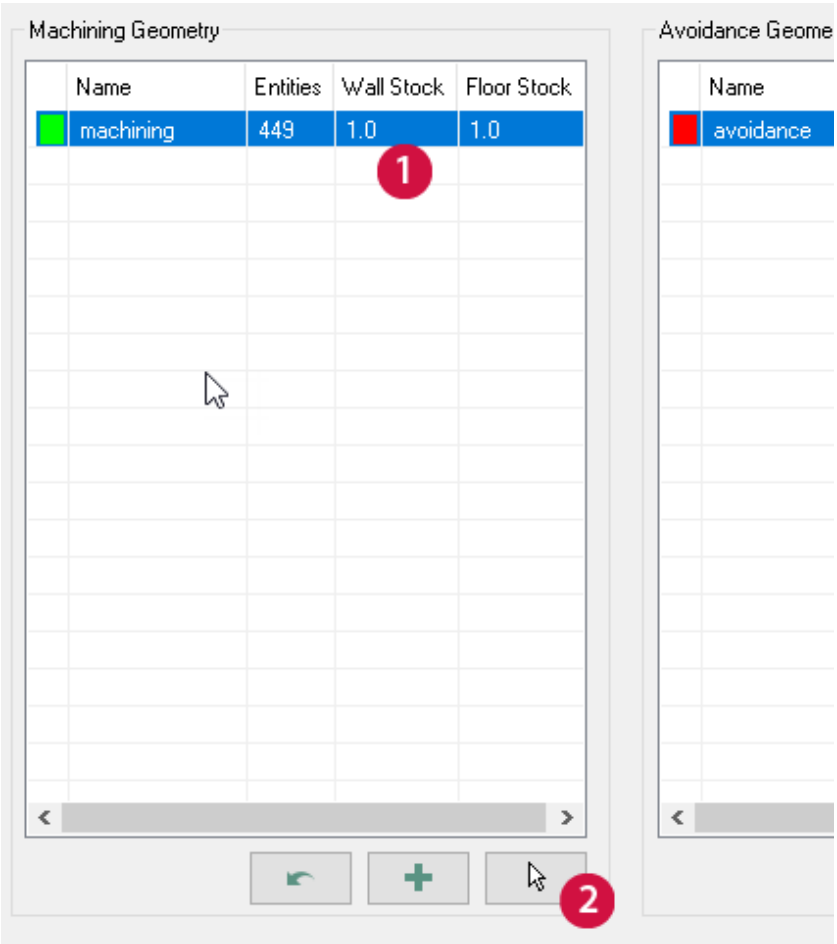
In this exercise, you create the first Dynamic OptiRough toolpath that uses this stock boundary.

1. Select **OptiRough** from the **3D** gallery on the **Mill Toolpaths** contextual tab.

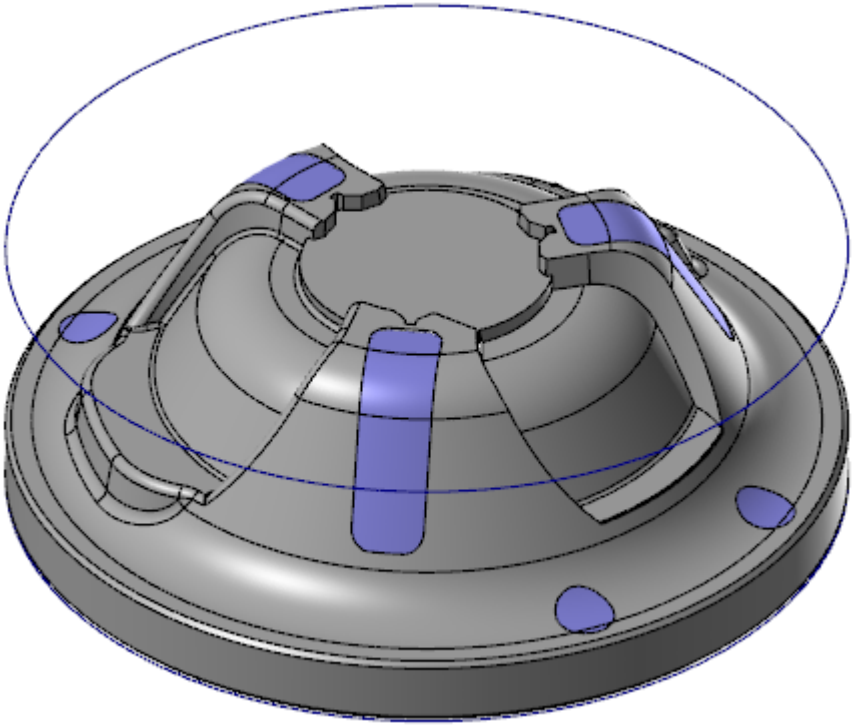


   The **Model Geometry** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.

2. Select the **Machining** group, and then click **Select entities** under Machining Geometry.



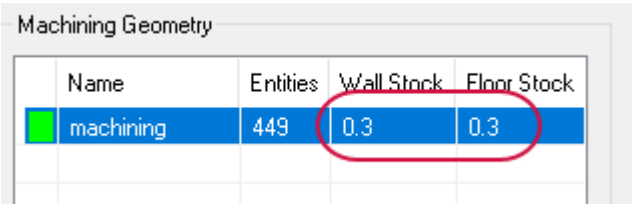
3. Use window select or any applicable keyboard shortcuts to select the part body as the machining geometry. This includes the blue caps, as shown below.



4. Press [**Enter**] or click **End Selection** to accept these surfaces.

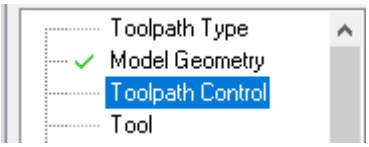
   You return to the **Model Geometry** page.

5. Because you are creating a roughing toolpath, enter the following stock-to-leave amounts on the wall and floor. Double-click to activate the field.

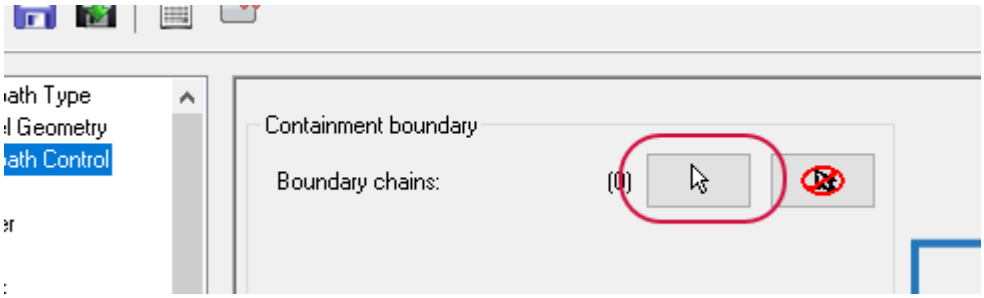


   - Enter **0.3** in **Wall Stock**
   - Enter **0.3** in **Floor Stock**

6. Select the **Toolpath Control** page.

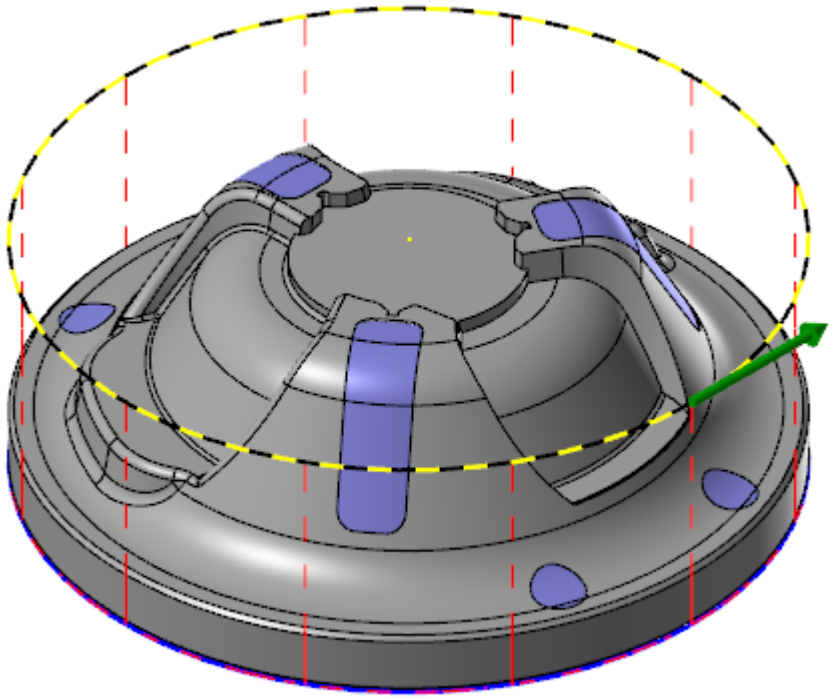


7. Click the **Boundary chains** selection arrow.



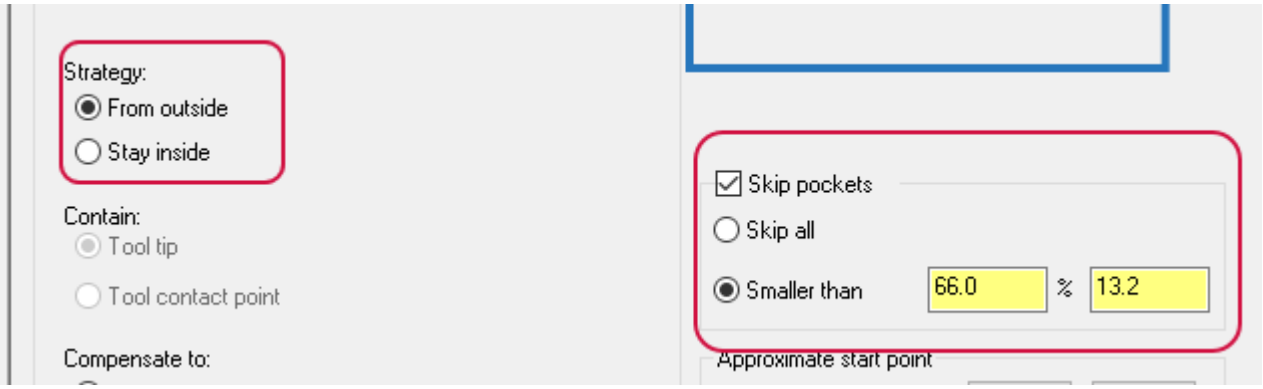
This button returns you to the graphics window to select one or more closed chains of curves to limit the tool motion.

8. Ensure that the chaining dialog box is set to **Wireframe** and, in the graphics window, select the circular geometry at the top of the stock boundary.



9. Click **OK** to return to the **Toolpath Control** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box.

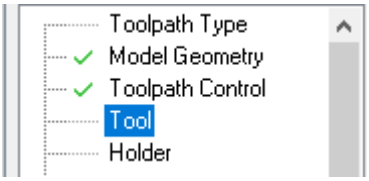
10. Set the containment boundary **Strategy** to **From outside** and, in the **Skip pockets** section, set **Smaller than** to **66.0**.



From outside sets the toolpath to machine from outside the containment boundary moving in. This strategy is best used for machining cores.

Skip pockets specifies a minimum pocket size that Mastercam will create a cutting pass for. This helps with problems where Mastercam thinks that a pocket is large enough to accommodate the tool, but the entry move is so compressed that the tool is effectively plunging into the part.

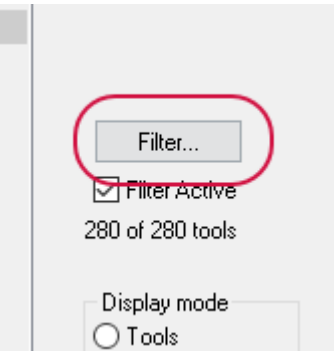
11. Select the **Tool** page.



12. Click the **Select library tool** button.

    The **Tool Selection** dialog box displays.

13. Select **Filter**.

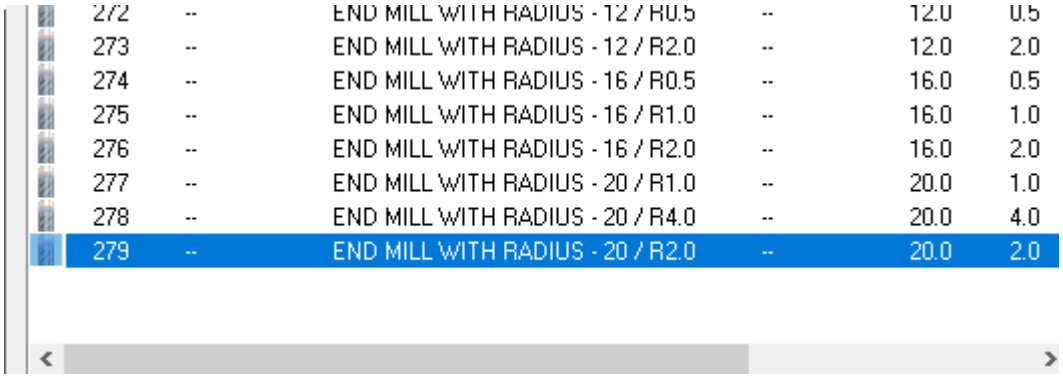


    The **Tool List Filter** dialog box displays.

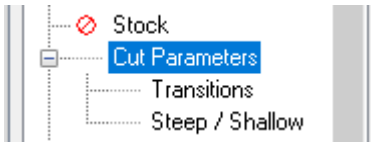
14. Set the **Tool Type** filter to **None**.
15. Select the **Endmill3 Bull** filter. This ensures that you are only selecting bull end mill tools.
16. Click **OK** in the **Tool List Filter** dialog box.

    You return to the **Tool Selection** dialog box.

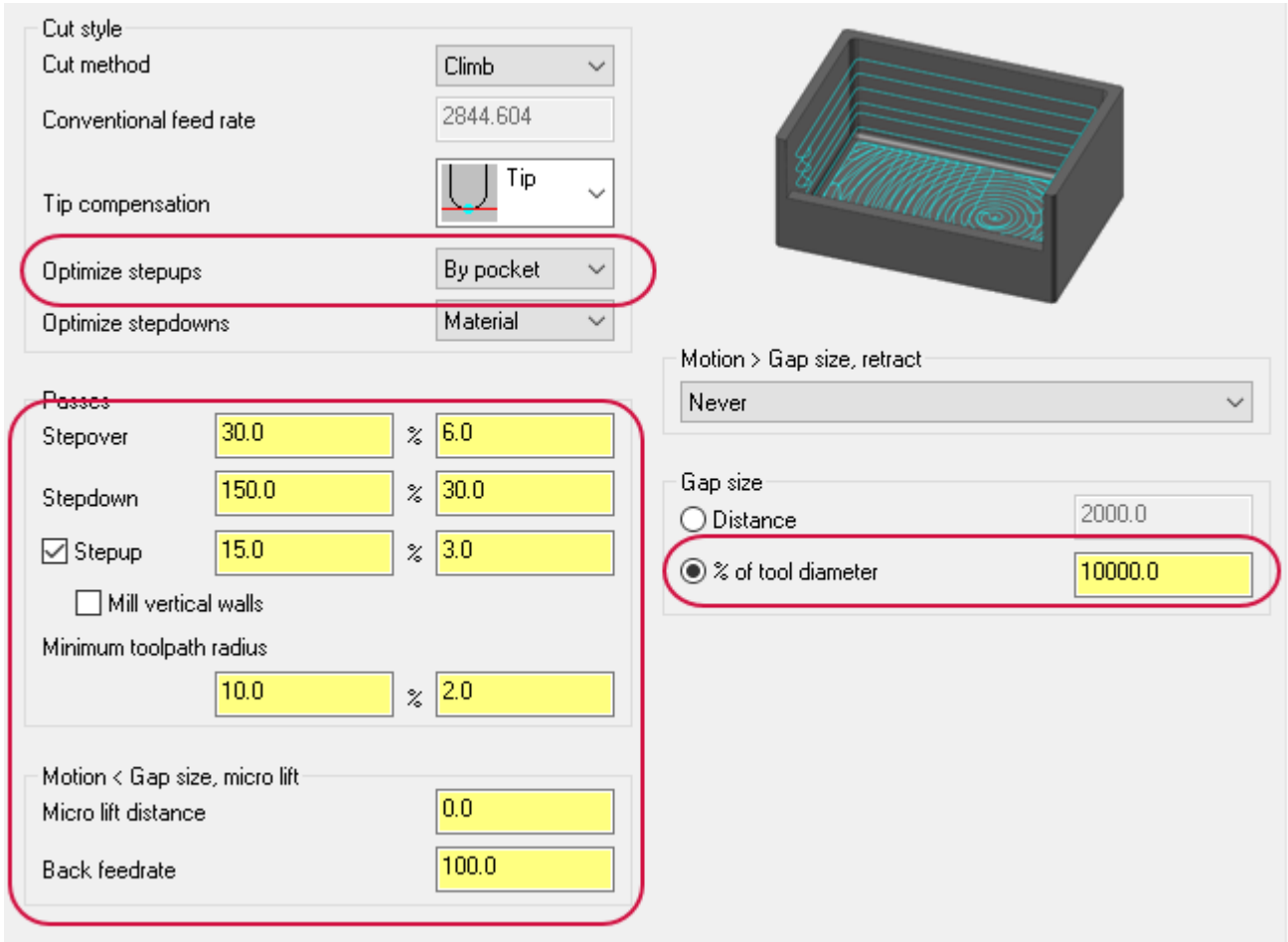
17. In the **Tool Selection** dialog box, select the **END MILL WITH RADIUS - 20/R2.0** tool.



18. Click **OK** in the **Tool Selection** dialog box to add the tool to the toolpath.
19. Select the **Cut Parameters** page.



20. Set the following parameters:

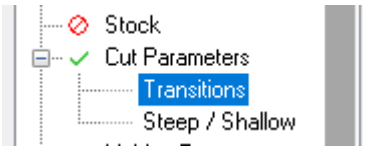


- **Optimize stepups** to **By pocket**. Mastercam first machines all stepdowns, moving from pocket to pocket. After all stepdowns on a Z-level are machined to completion, Mastercam machines the stepups

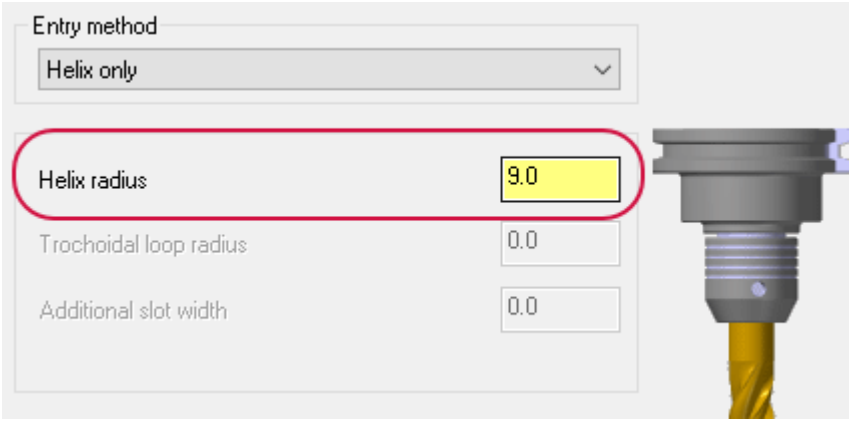
by next closest, in the safest cut order.

- **Stepover** to **30.0**%. Sets the distance between cutting passes in the X and Y axes.
- **Stepdown** to **150.0**%. Determines the Z spacing between adjacent cutting passes.
- Select the **Stepup** checkbox and enter **15.0**. Mastercam calculates and adds stepup cutting passes (+Z) to the toolpath.
- **Minimum toolpath radius** to **10.0**. Sets the minimum toolpath radius to create in the operation.
- **Micro lift distance** to **0.0**.
- **Back feedrate** to **100.0**. Controls the speed of the backfeed movement of the tool.
- **% of tool diameter** to **10000.0**.

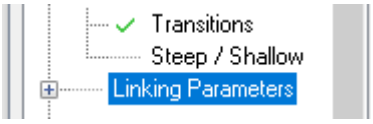
21. Select the **Transitions** page.



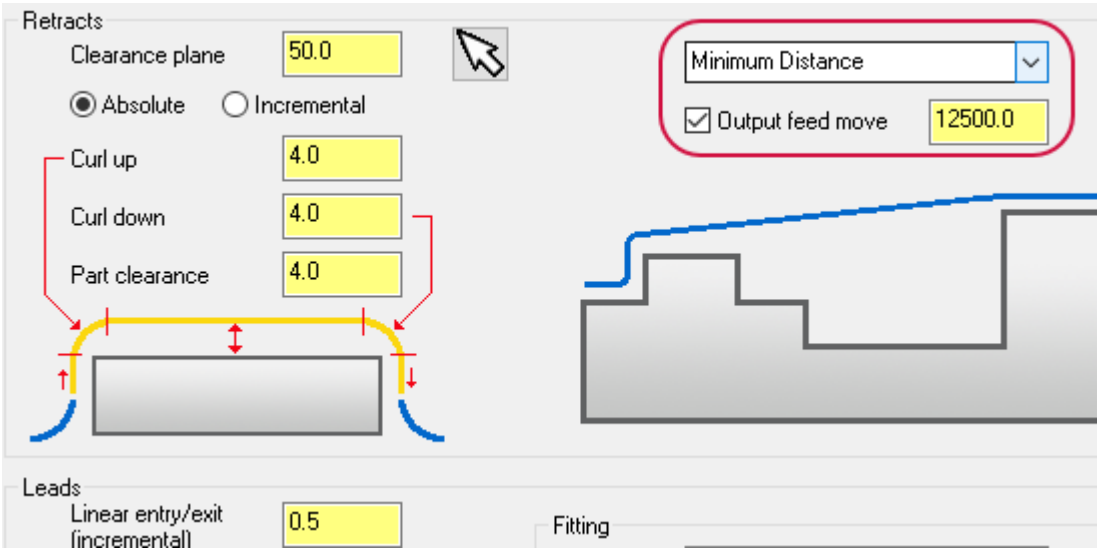
22. Set **Helix radius** to **9.0**.



23. Select the **Linking Parameters** page.

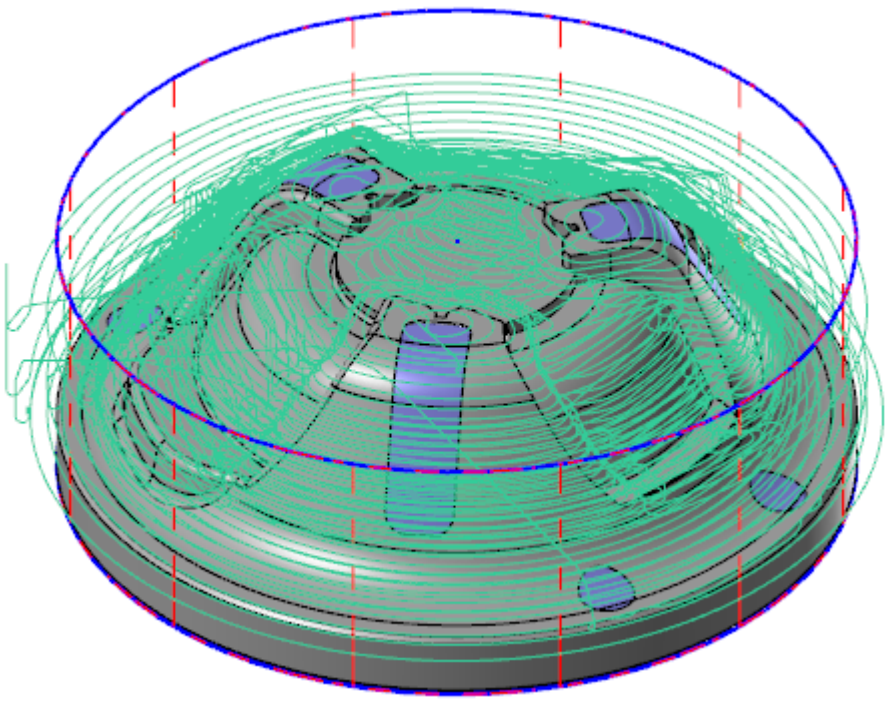


24. Set the following parameters:



- Set the **Retract Method** drop-down to **Minimum Distance**. Creates high-speed loops into and out of each retract move.
- Select **Output feed move** and enter **12500.0**. Outputs the rapid move between passes as a feed rate move instead of a rapid move. This helps when the tool needs to make many irregular moves per pass to jump between different areas of the part.

25. Click **OK** to close the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box and generate your toolpath. (It may take a few moments.)
26. Your toolpath should display as follows:



27. Save your part file.

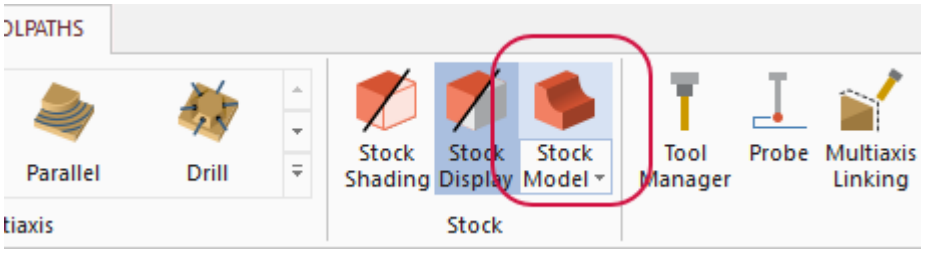
Exercise 3: Creating the Stock Model operation

You now create a Stock Model operation that uses the previous operation as its source. This stock model will be used in the second Dynamic OptiRough toolpath to remove any remaining stock.

1. On the Toolpaths Manager, select **Only display selected toolpaths**.

   Mastercam will now only display the toolpath you have selected. It will also clean up the graphics view when making modifications to the part or creating another toolpath.

2. Select **Stock Model** on the **Mill Toolpaths** contextual tab.



   The **Stock model** dialog box displays.

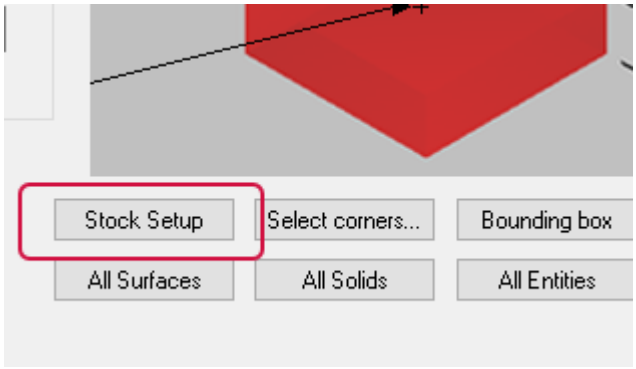
3. Name the stock operation **OP1 Stock**.

   Naming your stock model based on which operation it is using is helpful, especially when you have a part with multiple stock model operations.

4. Set **Color** to **128**.

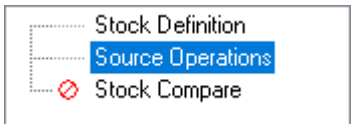
   This color will help differentiate the stock model from the solid or surface model.

5. Select **Stock Setup**.

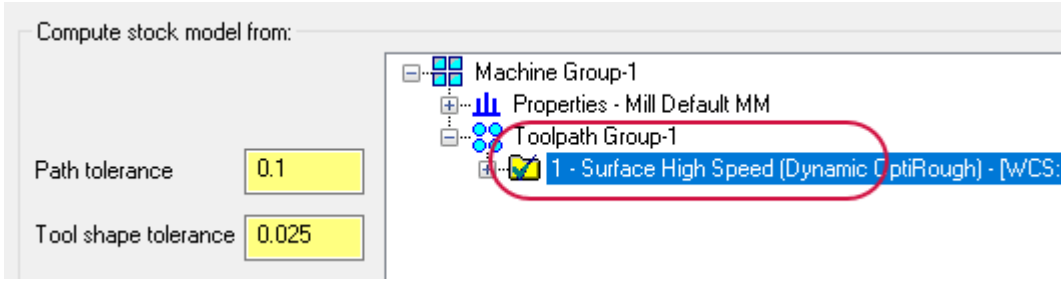


   Stock Setup copies the values defined in the **Machine Group Properties** dialog box to corresponding fields in the **Stock Definition** page. The values are not associative.

6. Select the **Source Operations** page.



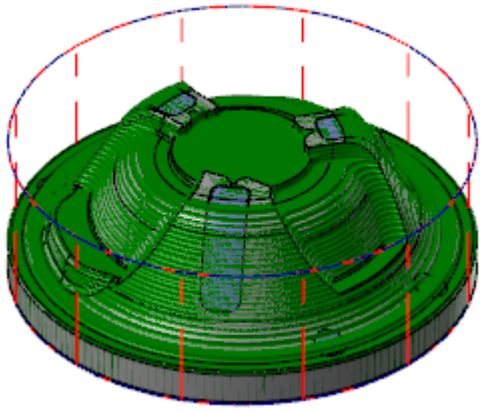
7. Select the **Surface High Speed (Dynamic OptiRough)** toolpath as the source operation.



Mastercam will generate the stock model by running the selected operation against the parameters on the **Stock Definition** page.

8. Click **OK** to create the stock model operation. This may take some time to calculate.

Your stock model displays as shown below:

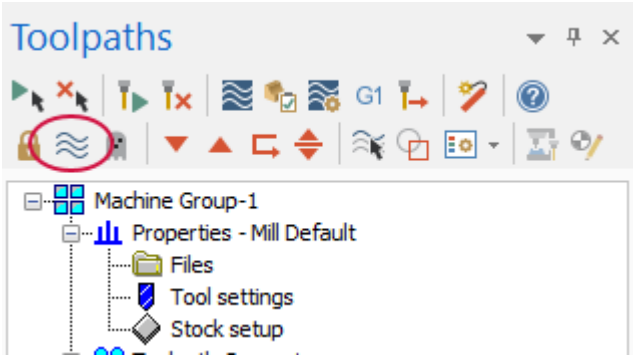


9. Turn off **Only display selected toolpaths** from the Toolpaths Manager.
10. Save your part.

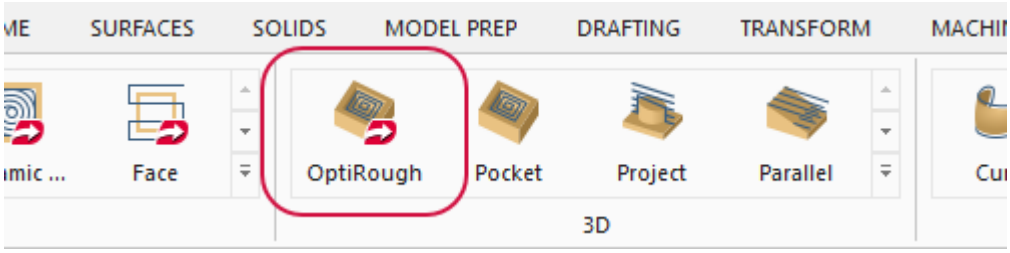
Exercise 4: Creating the second Dynamic OptiRough toolpath

You now create a Dynamic OptiRough toolpath that uses the stock model you created in the previous exercise.

1. Select **Toolpath Group-1** in the Toolpaths Manager.
2. Select **Toggle display on selected operations** twice. Mastercam hides all selected toolpaths to keep the graphics window clean.

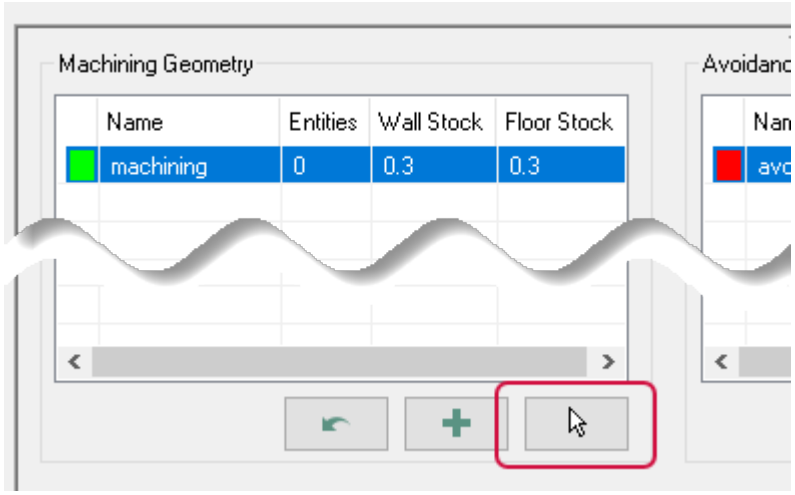


3. Select **OptiRough** from the **3D** gallery on the **Mill Toolpaths** contextual tab.

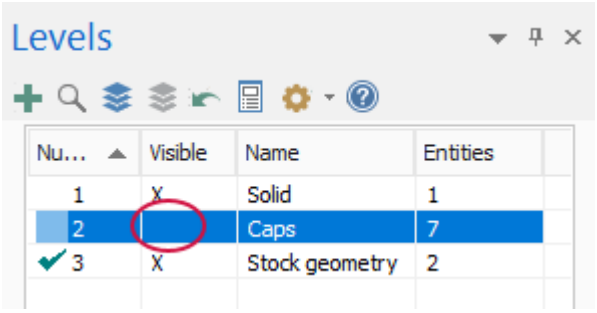


The **Model Geometry** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.

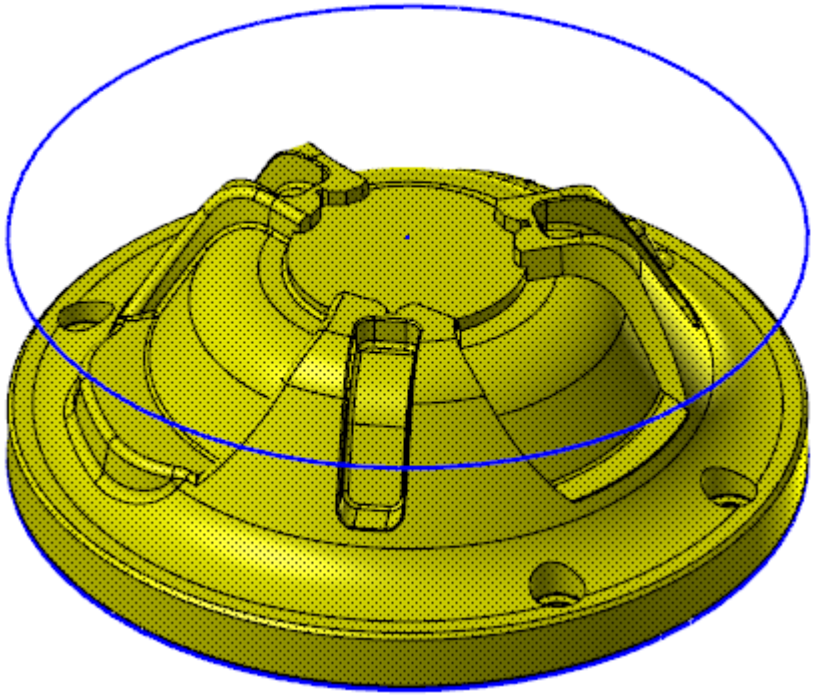
4. Select the **Machining** group, and then click **Select entities** under Machining Geometry.



5. In the Levels Manager, turn off the display of the **Caps** level and turn on the **Stock geometry** level to make it easier to select the drive surfaces.



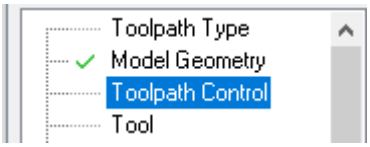
6. Select all but the blue caps as the drive surfaces shown below.



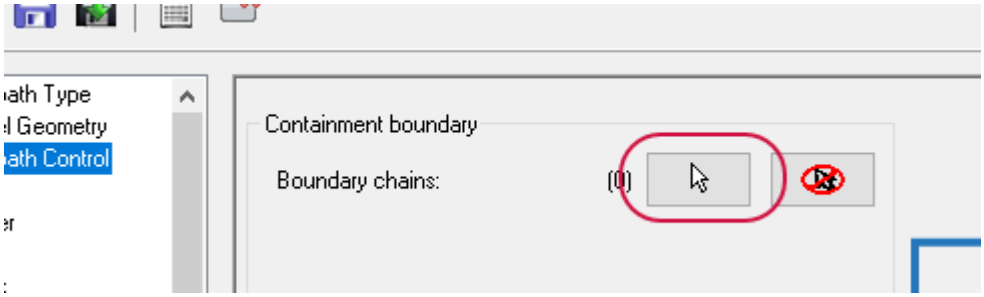
7. Press [**Enter**] or click **End Selection** to accept these surfaces.

   The **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.

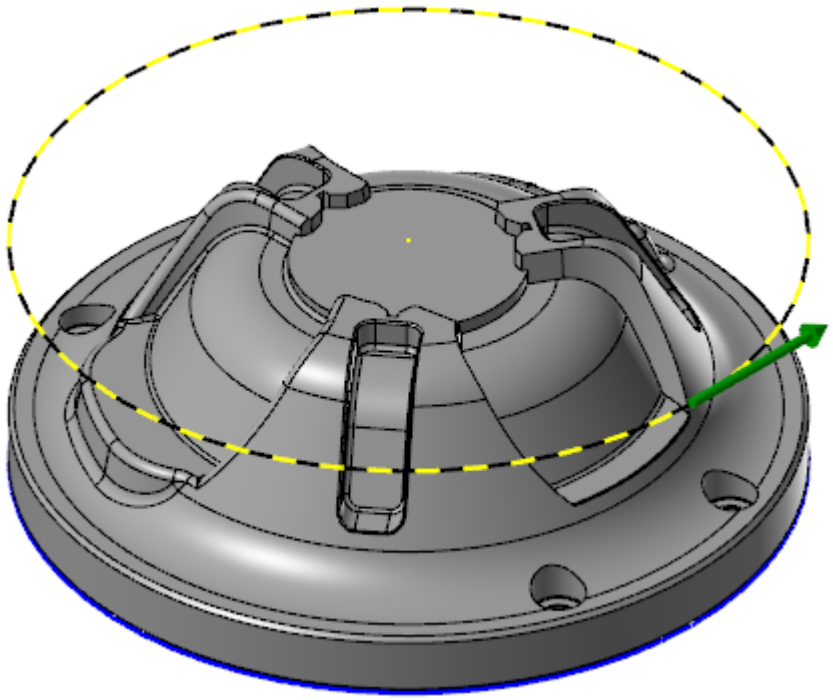
8. Select the **Toolpath Control** page.



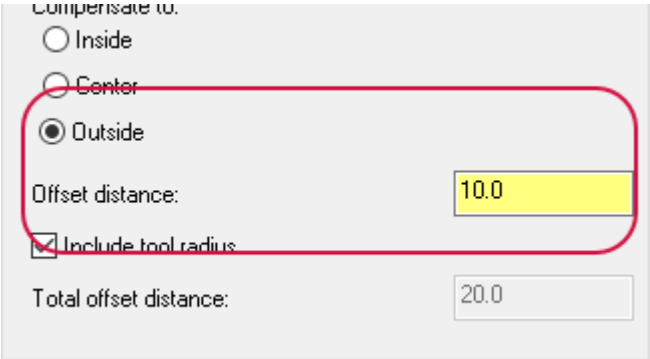
9. Click the selection arrow for **Boundary chains** to return to the graphics window.



10. In the graphics window, select the circular geometry at the top of the stock boundary.

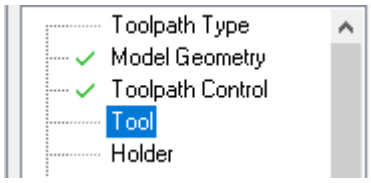


11. Click **OK** in the **Wireframe Chaining** dialog box to return to the **Toolpath Control** page.
12. Set the following parameters to control the tool's position around the boundary of your part.



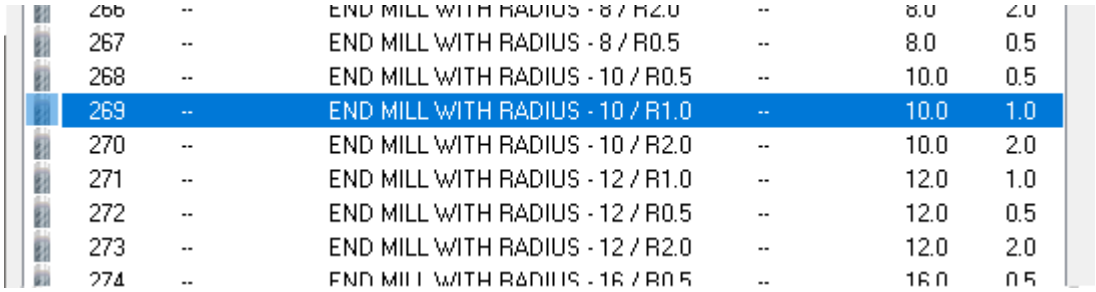
- Set **Compensate to** to **Outside**. This sets the outer edge of the tool to be bound to the containment boundary.
- Set **Offset distance** to **10.0**. This option adjusts the inside or outside tool containment boundary.

13. Select the **Tool** page.



14. Click **Select library tool**.

15. In the **Tool Selection** dialog box, select the **END MILL WITH RADIUS - 10/R1.0** tool.



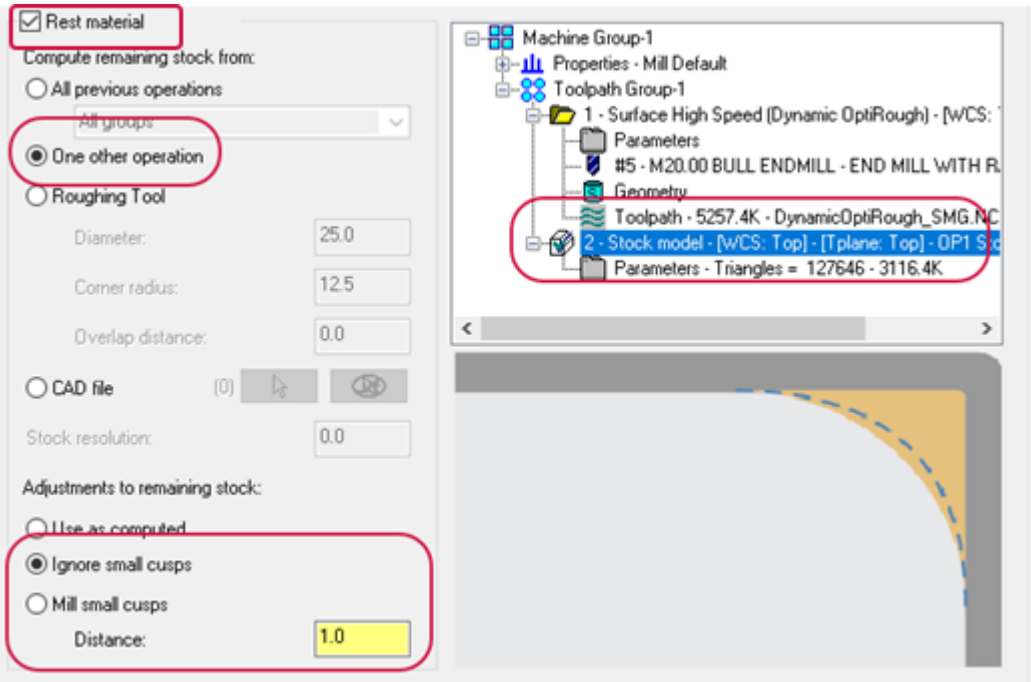
16. Click **OK** to add the tool to the toolpath.

17. Select the **Stock** page.

18. Select **Rest material**.

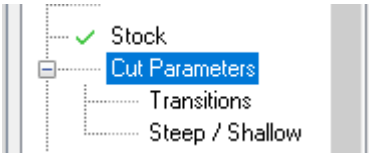
When you enable this page, Mastercam calculates the cutting passes based on the remaining stock.

19. Set the following parameters:

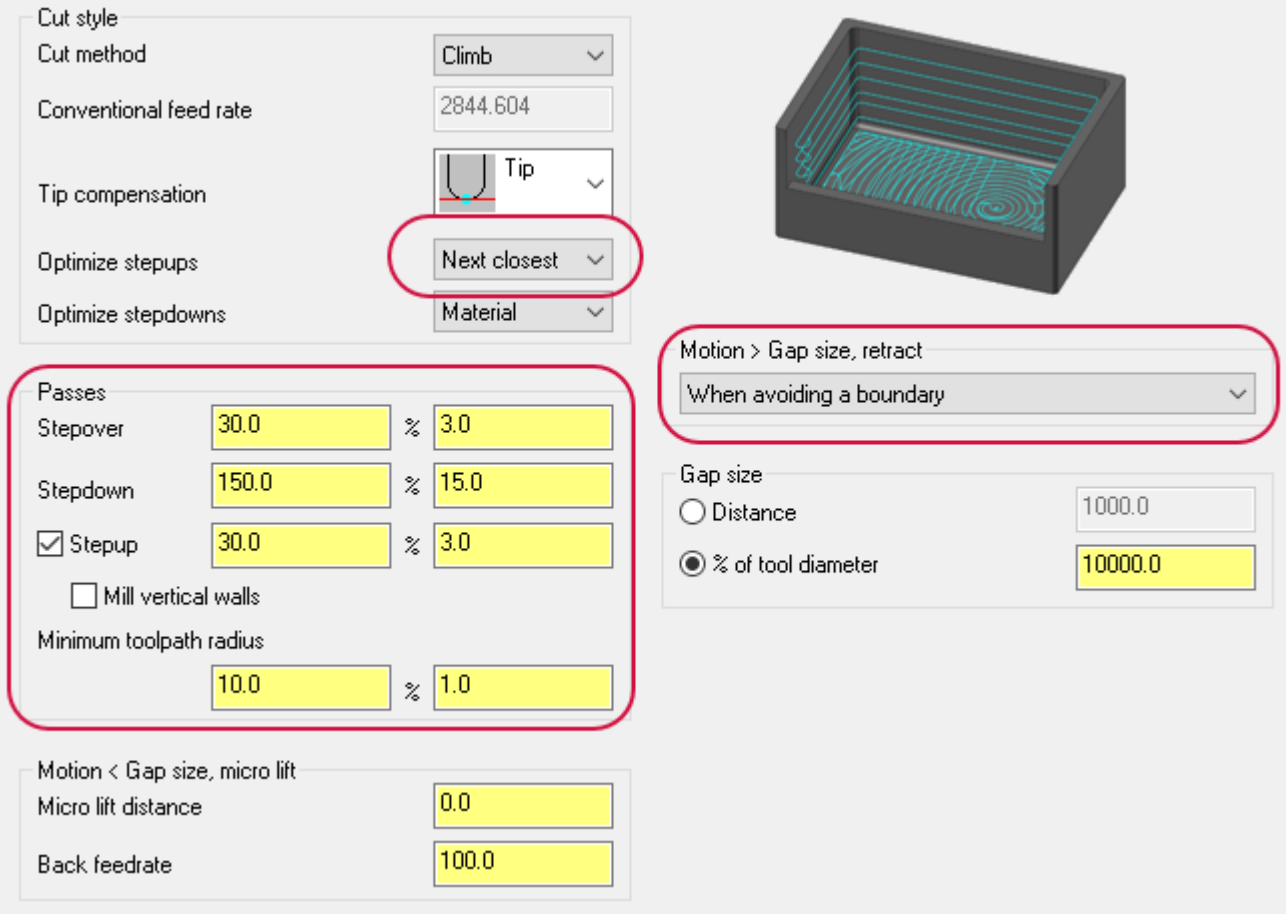


- Set **Compute remaining stock from** to **One other operation**.
- In the **Operation List**, select **Stock Model - WCS: Top**.
- Set **Adjustments to remaining stock** to **Ignore small cusps**.
- Set **Distance** to **1.0**. This option is used in conjunction with **Ignore small cusps**. Mastercam will output cuts that engage large amounts of material.

20. Select the **Cut Parameters** page.

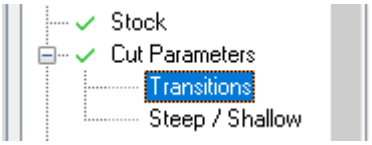


21. Set the following parameters:



- **Optimize stepups** to **Next closest**. Mastercam moves to the closest cut from its last position on the previous cut.
- **Stepover** to **30.0%**.
- **Stepdown** to **150.0%**.
- **Stepup** to **30.0%**.
- **Minimum toolpath radius** to **10.0%**.
- **Motion > Gap size, retract** to **When avoiding a boundary**. Adds retracts to avoid intersecting boundaries (a gouge boundary, or a boundary of material yet to be milled).

22. Select the **Transitions** page.



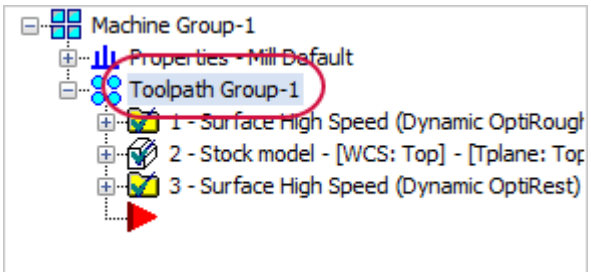
23. Set the **Helix radius** to **5.5**.

24. Click **OK** to generate the Dynamic OptiRough toolpath.
25. Select **Toggle display on selected operations** to display toolpaths again, and select the second Dynamic OptiRough toolpath to view the newly created toolpath.
26. Save your part file.

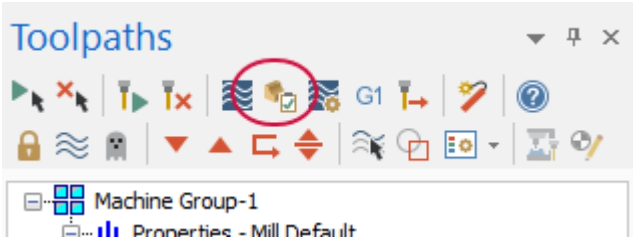
Exercise 5: Verifying the stock model and toolpaths

In this exercise, you use Mastercam Simulator to verify your toolpaths and stock model. Verifying your toolpaths allows you to use solid models to simulate part machining against the selected stock definition.

1. Select **Toolpath Group-1** in the Toolpaths Manager.

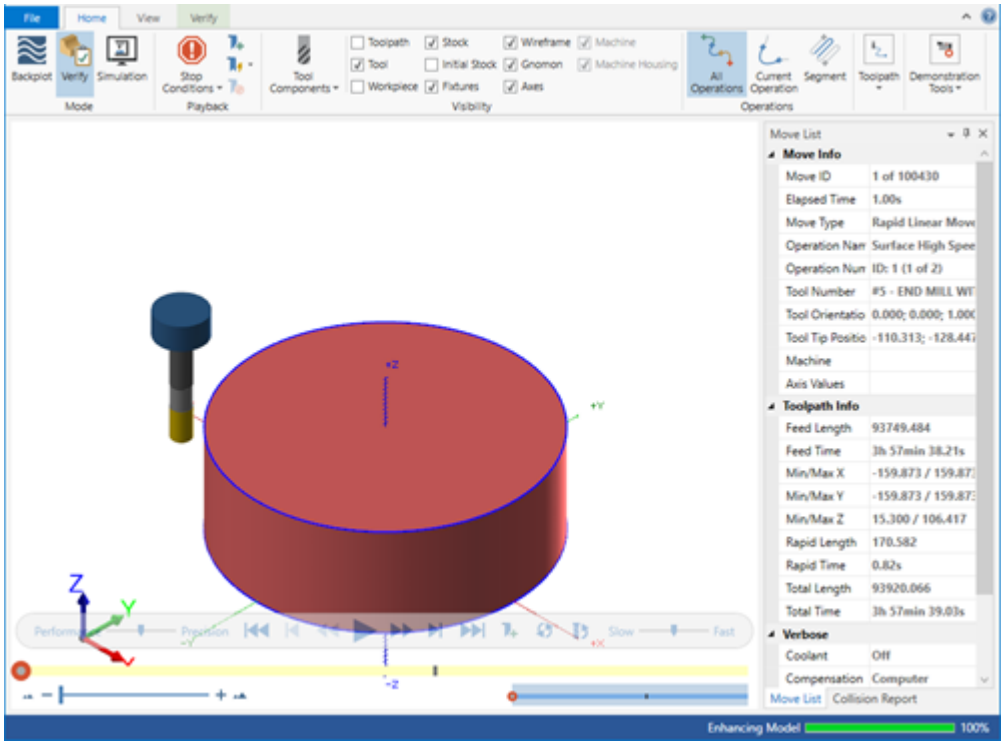


2. Select **Verify selected operations** in the Toolpaths Manager.

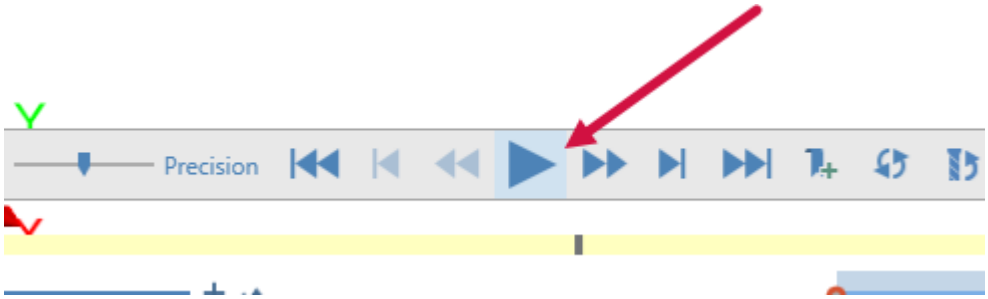


Mastercam Simulator displays.

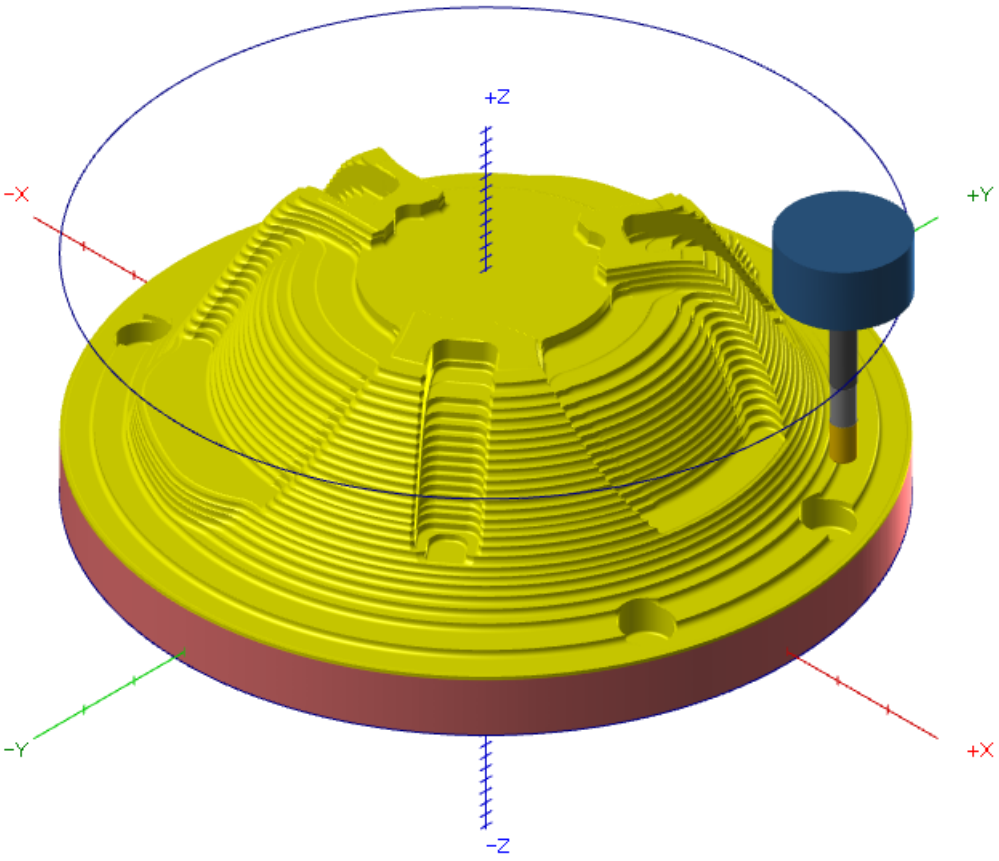
3. If necessary, right-click in the graphics window and change the view to **Isometric** and **Fit** the part to the screen so that it matches the image shown below.



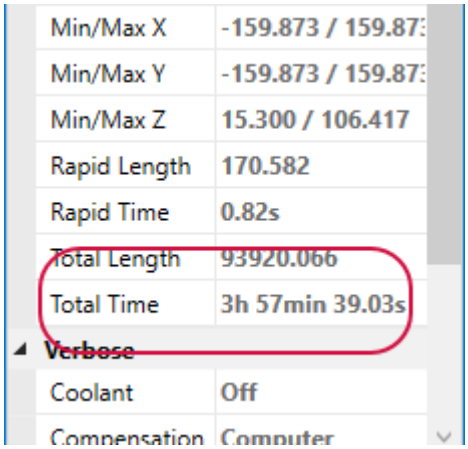
4. Click the **Play** button or press [**R**] to verify the toolpaths.



Your result should look like the following image:



5. View the Move Info to view the total run time of the toolpaths.



6. Once you are satisfied with the verification, close Mastercam Simulator.
7. Save your part.

CONCLUSION

Congratulations! You have completed the *Mastercam Dynamic Milling 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 2021 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 university.mastercam.com/, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com.
    - Follow us on Facebook (www.facebook.com/Mastercam), Twitter (twitter.com/Mastercam), and Instagram (www.instagram.com/mastercamcadcam/) for the latest tech tips and Mastercam news.
    - See Mastercam in action on YouTube (www.youtube.com/user/MastercamCadCam).
    - For more information on CNC Software, Inc., to find and apply to jobs, and connect with people using Mastercam, visit us on LinkedIn (www.linkedin.com/company/cnc-software/).
    - Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the Mastercam 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