



# MASTERCAM DYNAMIC MILLING TUTORIAL

June 2018



# MASTERCAM DYNAMIC MILLING TUTORIAL

June 2018

© 2018 CNC Software, Inc. – All rights reserved.

Software: Mastercam 2019

## Terms of Use

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

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

## Be sure you have the latest information!

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

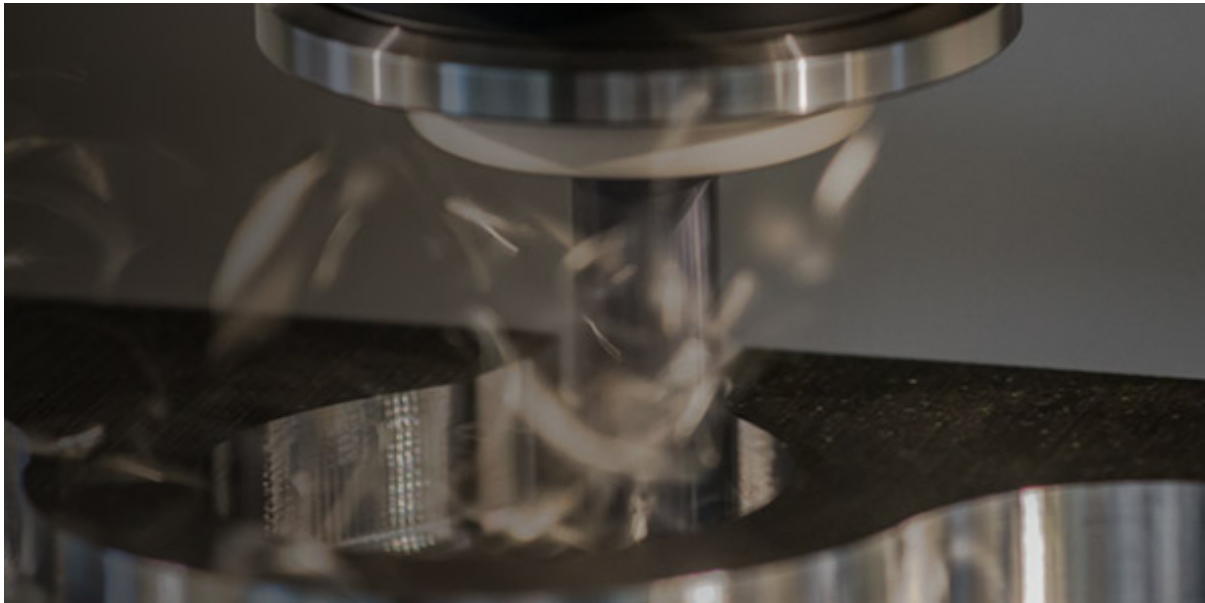
# TABLE OF CONTENTS

<b>Introduction</b>	<b>5</b>
General Tutorial Requirements	6
<b>Dynamic Toolpaths Overview</b>	<b>7</b>
Dynamic Toolpath Types	7
Dynamic Contour	7
Face	8
Dynamic Mill	8
Peel Mill	10
Dynamic OptiRough	11
Dynamic Toolpath Selection	12
Dynamic Milling Parameters	14
<b>The Dynamic Mill Toolpath</b>	<b>17</b>
Exercise 1: Opening Mastercam	17
Exercise 2: Setting up the stock	18
Exercise 3: Creating the first Dynamic Mill toolpath	22
Exercise 4: Creating the second Dynamic Mill toolpath	32
Exercise 5: Verifying the toolpaths	40
<b>The Face Toolpath</b>	<b>43</b>
Exercise 1: Creating the Face toolpath	43
Exercise 2: Verifying the toolpaths	48
<b>The Dynamic Contour Toolpath</b>	<b>51</b>
Exercise 1: Setting up the stock	51
Exercise 2: Creating the first Dynamic Contour toolpath	53
Exercise 3: Creating the second Dynamic Contour toolpath	60
Exercise 4: Analyzing the toolpaths	66
Exercise 5: Verifying the toolpaths	67
<b>The Peel Mill Toolpath</b>	<b>71</b>
Exercise 1: Setting up the stock	71
Exercise 2: Creating the first Peel Mill toolpath	72

Exercise 3: Copying the Peel Mill toolpath .....	78
Exercise 4: Re-chaining the copied Peel Mill operations .....	79
Exercise 5: Verifying the toolpaths .....	82
<b>The Dynamic OptiRough Toolpath .....</b>	<b>85</b>
Exercise 1: Setting up the stock .....	85
Exercise 2: Creating the first Dynamic OptiRough toolpath .....	89
Exercise 3: Creating the Stock Model operation .....	96
Exercise 4: Creating the second Dynamic OptiRough toolpath .....	98
Exercise 5: Verifying the stock model and toolpaths .....	105
<b>Conclusion .....</b>	<b>109</b>
Mastercam Resources .....	109
Contact Us .....	109

# INTRODUCTION

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



These 2D and 3D high speed dynamic milling toolpaths utilize the entire flute length of their cutting tools to achieve great 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.

**WARNING:** Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 5 hours

## General Tutorial Requirements

All Mastercam 2019 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 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel:  
[www.youtube.com/user/MastercamTechDocs](http://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.

# DYNAMIC TOOLPATHS OVERVIEW

The following toolpaths use Dynamic Motion technology:

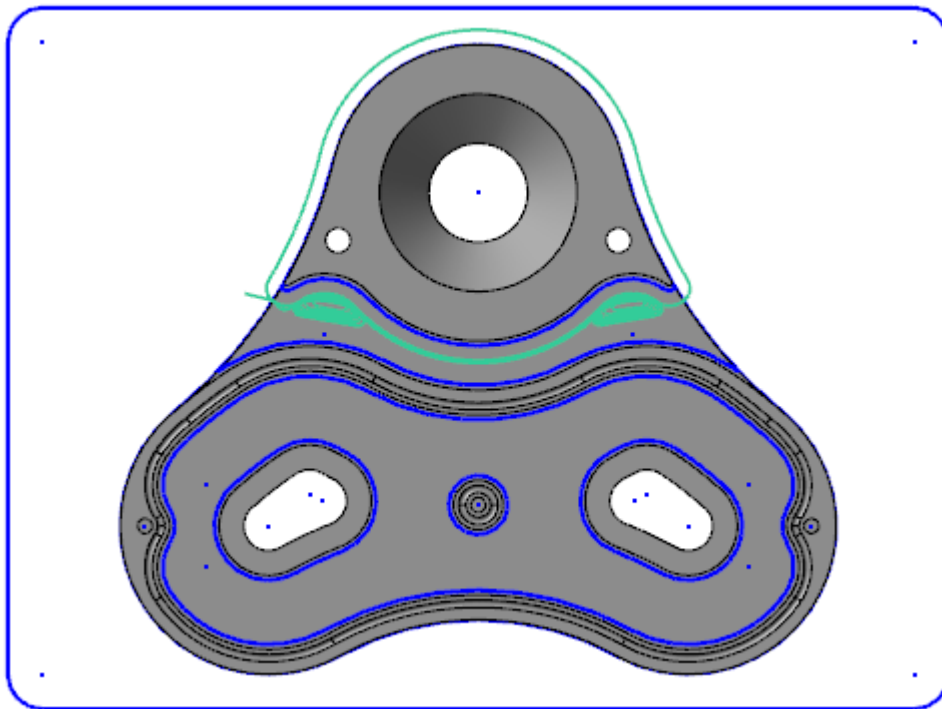
- Dynamic Contour
- Face
- Dynamic Mill
- Peel Mill
- Dynamic OptiRough

Each toolpath has various benefits that make them unique from each other and their standard counterparts.

## Dynamic Toolpath Types

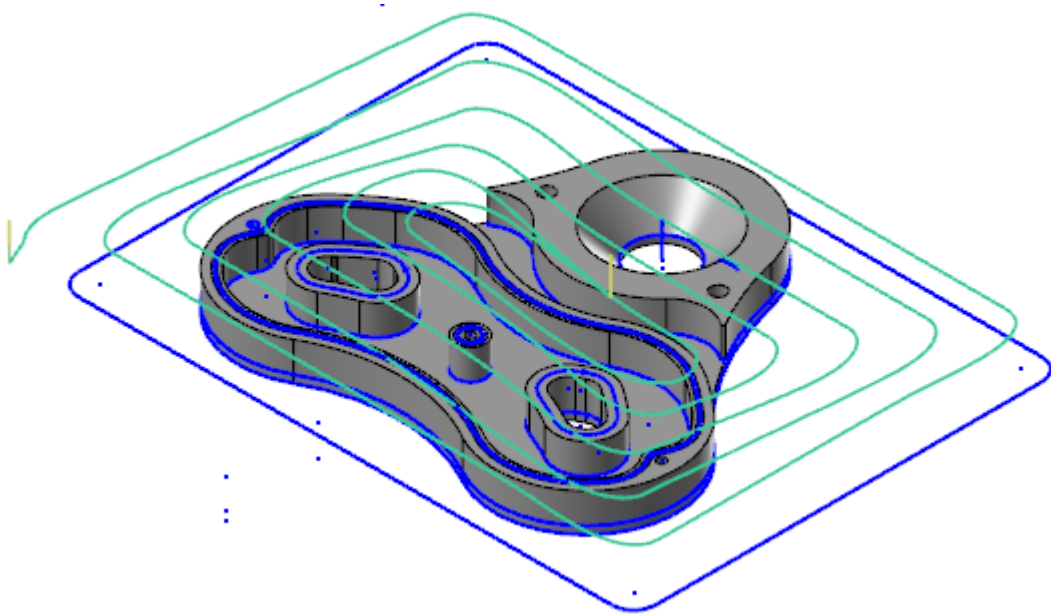
### Dynamic Contour

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 any material left on the walls.



## Face

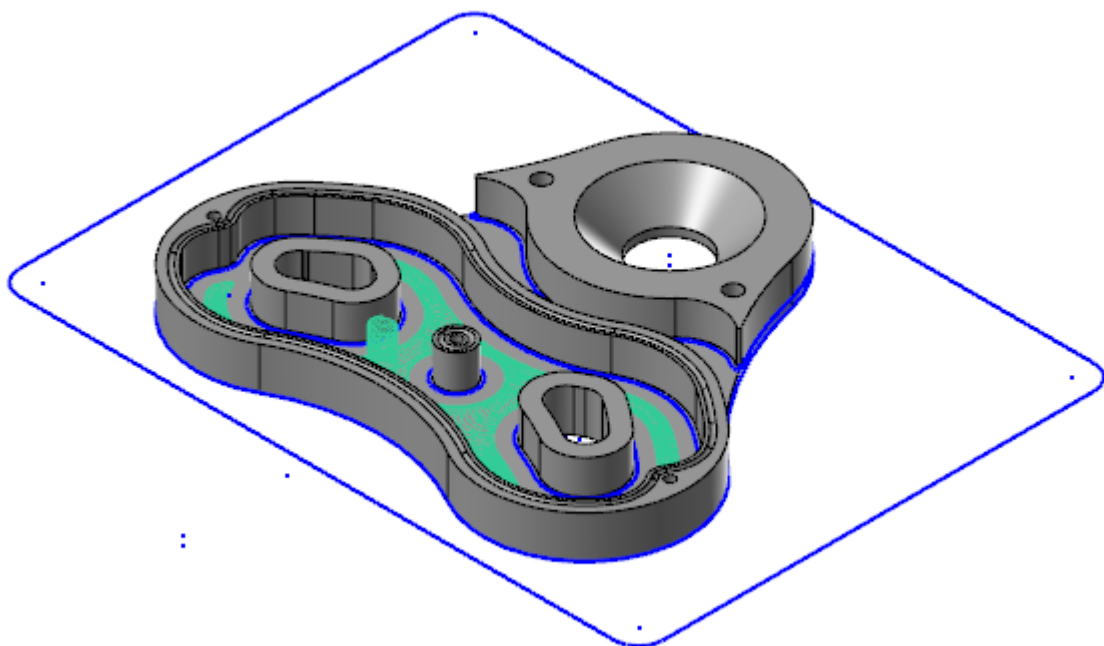
The Face toolpath, when the **Style** is set to **Dynamic**, quickly cleans stock from the top of a part and creates an even surface for future operations. You can base the toolpath on either chained geometry or on the current stock model.



## Dynamic Mill

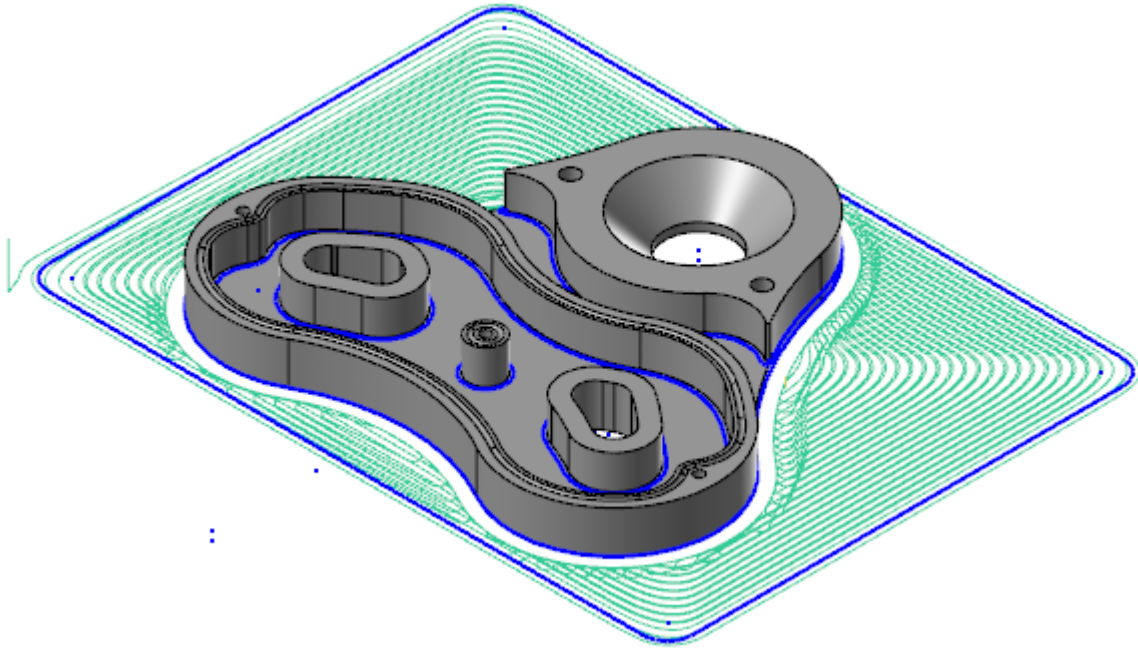
The Dynamic Mill toolpath machines pockets, leftover material, standing bosses, or cores. By setting the **Machining region strategy** to **Stay inside** or **From outside**, you can create a pocket toolpath that utilizes dynamic motion or a facing operation that has island avoidance capabilities.

By selecting the **Stay inside** strategy, Dynamic Mill becomes a pocket toolpath that utilizes 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 from the inside to the outside in a highly efficient manner.

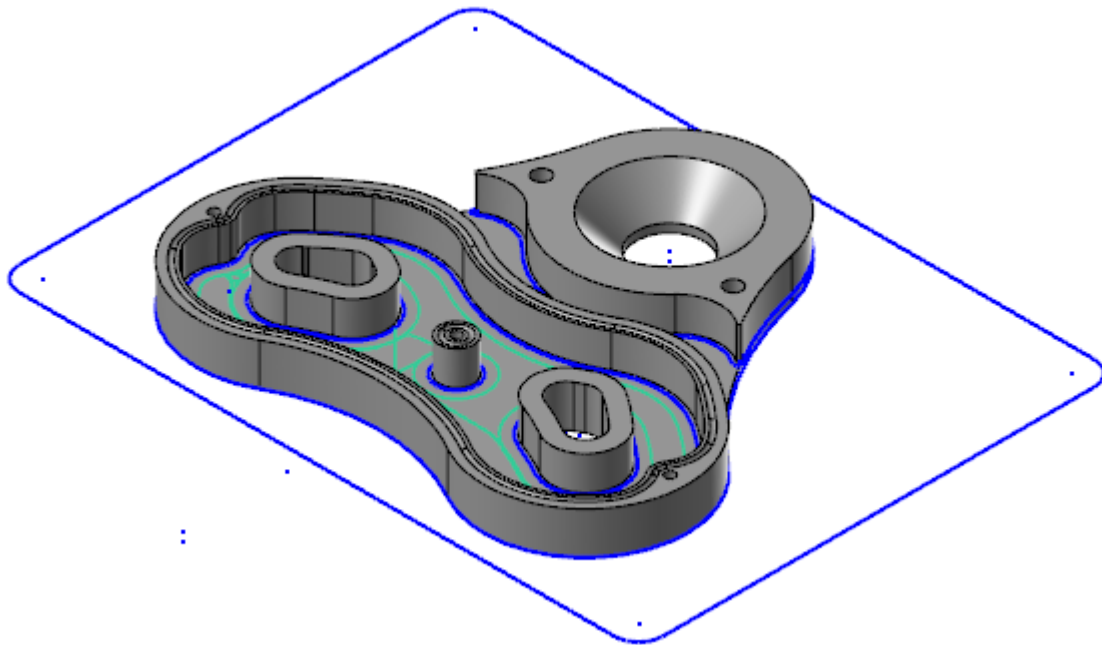




By selecting the **From outside** strategy, Dynamic Mill becomes a facing operation that has island avoidance capabilities. This movement creates a toolpath that is allowed to move outside the selected material and removes from the outside to the inside.

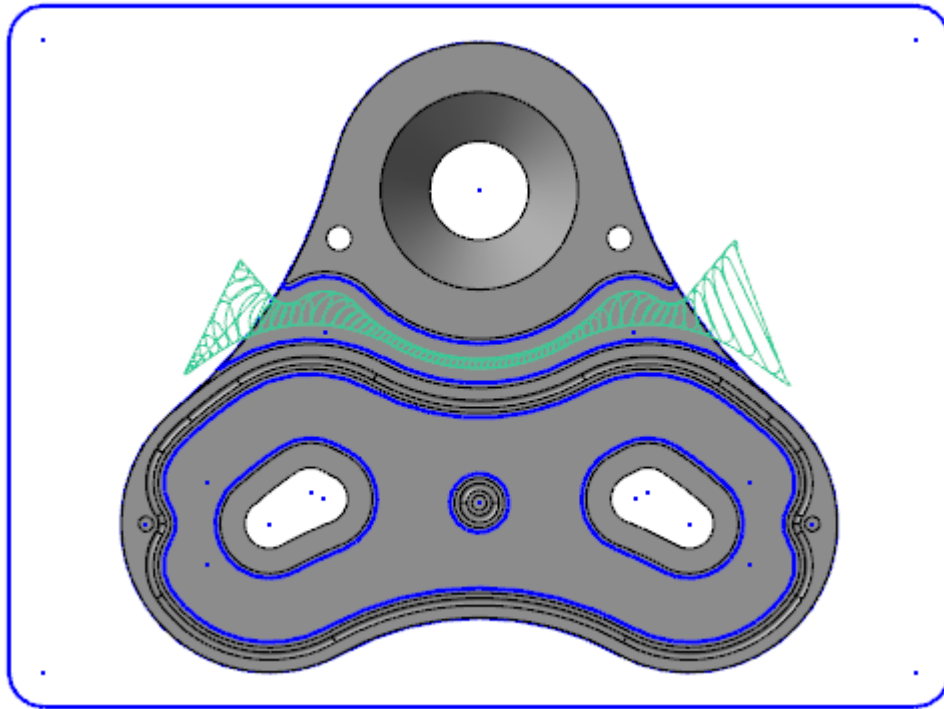


By enabling **Rest Material** on the Stock page, the Dynamic Mill toolpath removes material left by previous operations using dynamic motion. The Dynamic Mill operation calculates how much material to remove based on previous operations or previous roughing tools. Only unmachined areas are processed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



## Peel Mill

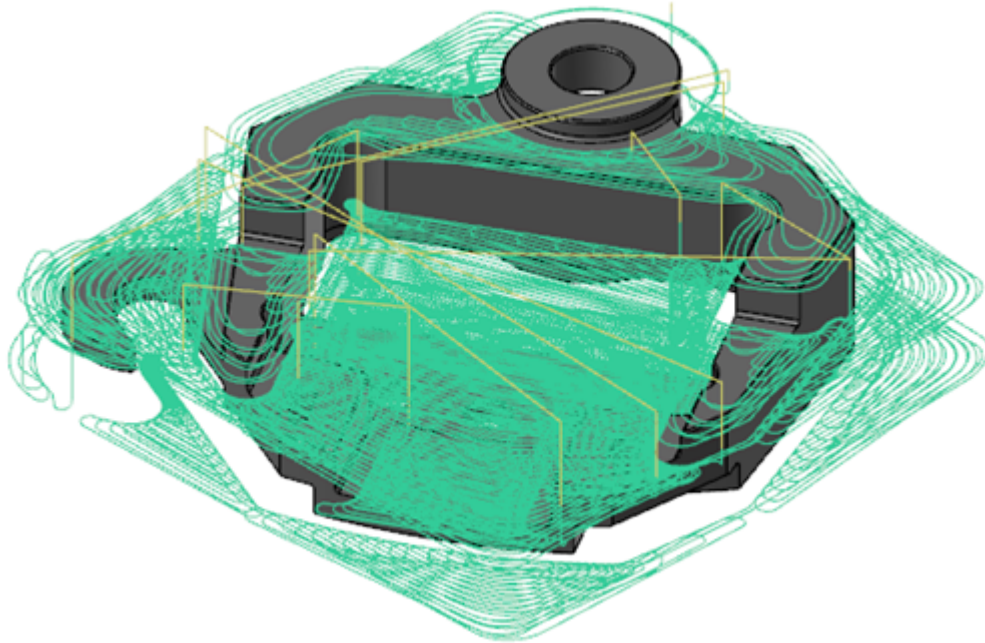
Peel Mill, with the **Cutting strategy** set to **Dynamic Peel**, allows for efficient milling between selected chains. It uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material. For a single chain, you define the width of the cut. Otherwise, the width 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 will be cut by the tool motion. Peel Mill requires at least one open chain and no closed chains.

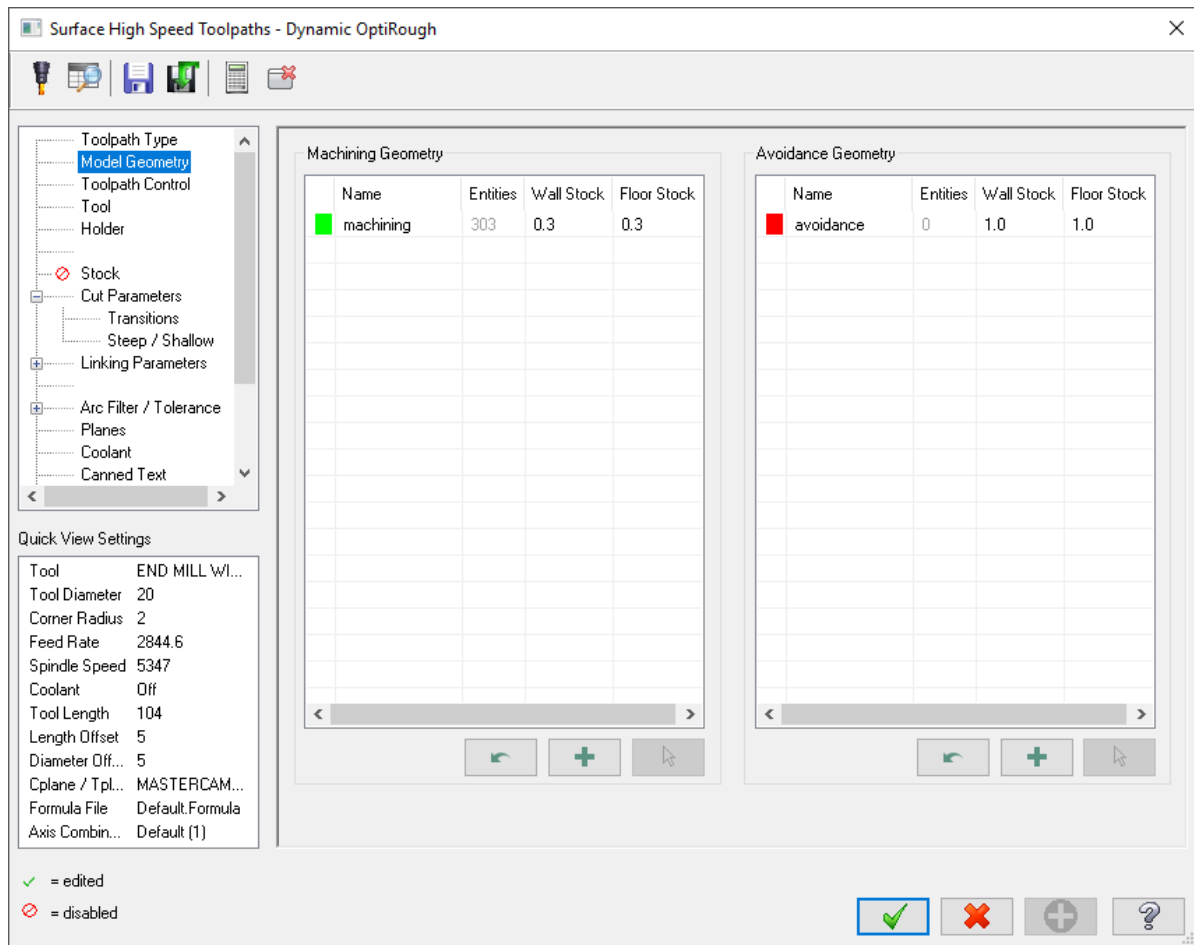
Dynamic Mill allows you to have even more control over your chaining by allowing you to select multiple kinds of regions with one toolpath. The types of regions are listed below:

- **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 will be automatically closed with a straight line.

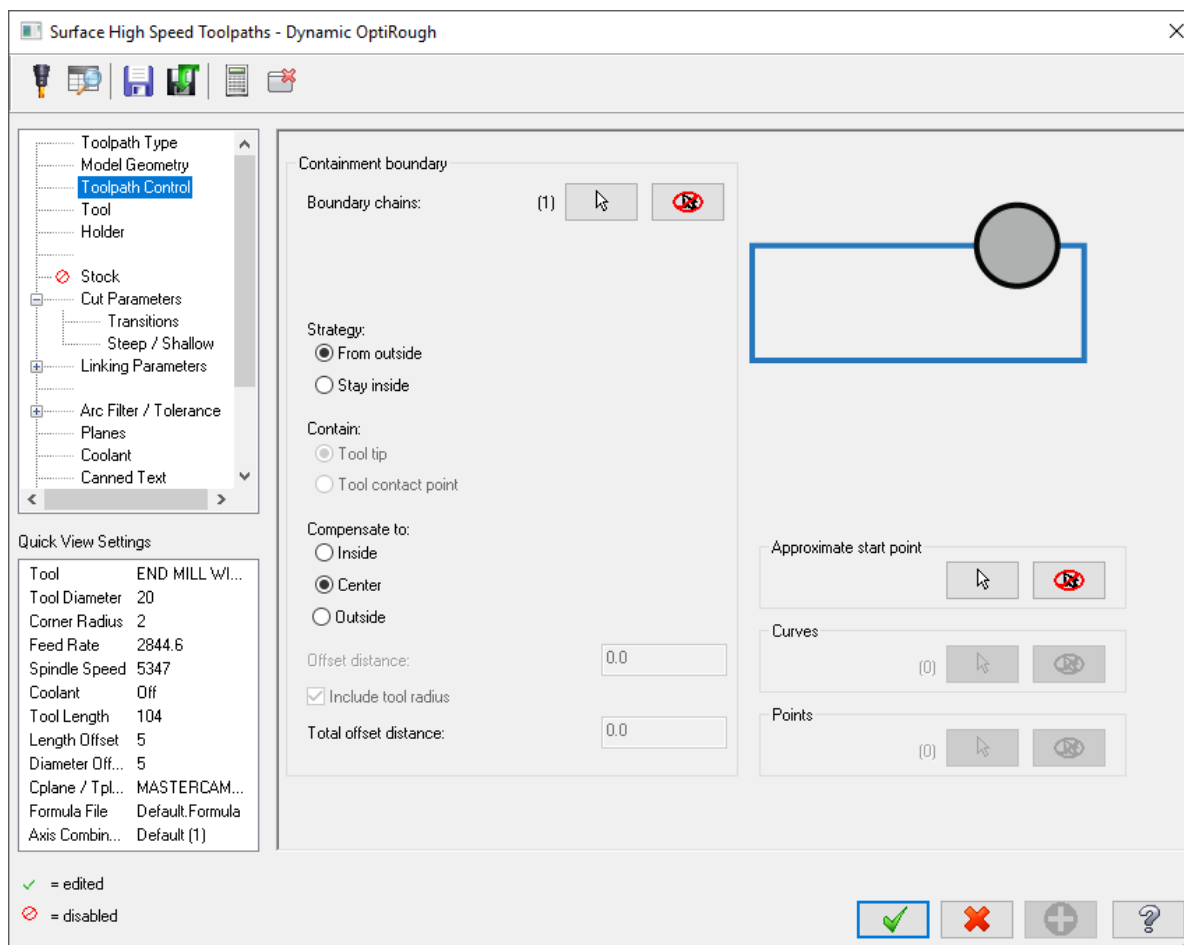
For more information, view Mastercam's **Help**.

Dynamic OptiRough, as it is the only 3D Dynamic toolpath, uses a different type of selection method compared to 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 for the Dynamic OptiRough toolpath, which controls the tool's position around the boundary of your part.



## Dynamic Milling Parameters

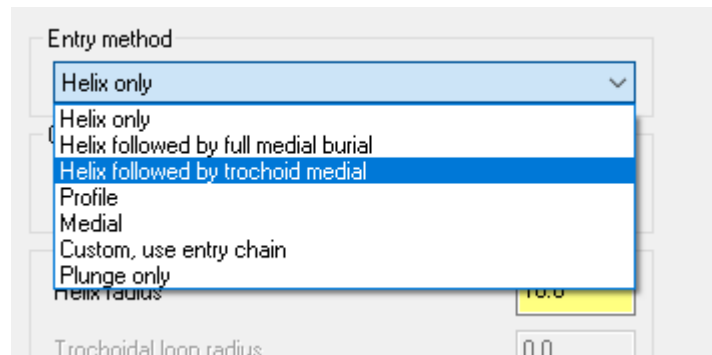
Several parameter options separate the Dynamic operations from their standard counterparts. These parameters are on the **Cut Parameters**, **Entry Motion**, and **Contour Wall** pages.

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

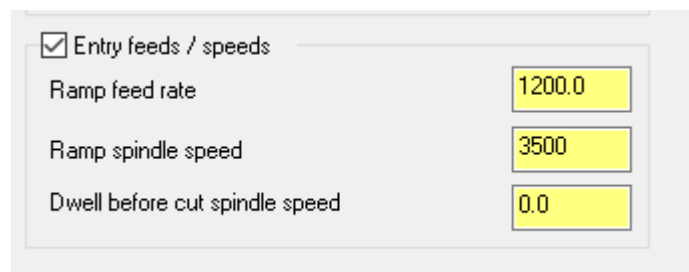
Micro lift options allow the tool to rise above the floor of the part for the portion of the toolpath that is not removing material. Benefits of the micro lift include chip clearing and avoiding excess tool heat. The feed rate is controllable for the back move as well.

Motion < Gap size, micro lift	
Micro lift distance	0.25
Back feedrate	2500.0

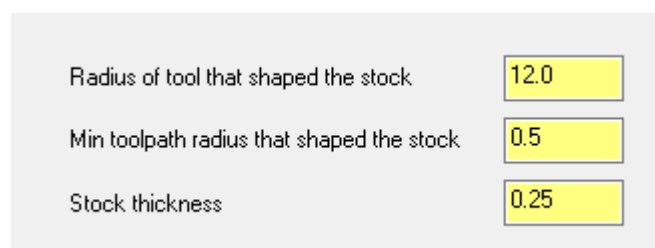
Entry methods provide several ways to enter the material for a Dynamic Mill toolpath. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's **Help** contains detailed information on each of these methods.



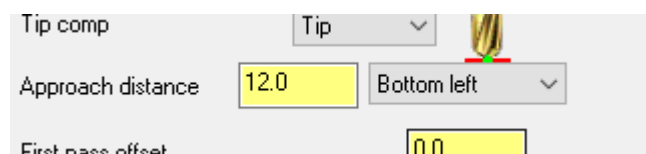
Use Entry feeds and speeds to avoid placing too much load on the tool upon entry. Set a short dwell interval after entry to allow the spindle to reach the necessary speed before beginning material removal.



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

Creating and reviewing the motion of dynamic toolpaths 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 lesson, you will create two Dynamic Mill toolpaths. The first one will clear 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.

### Lesson Goals

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

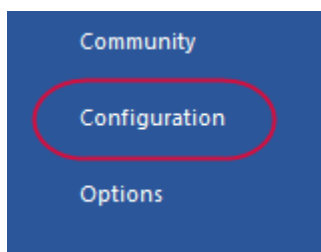
### Exercise 1: Opening Mastercam

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.



Repair File  
Perform routine mainten

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

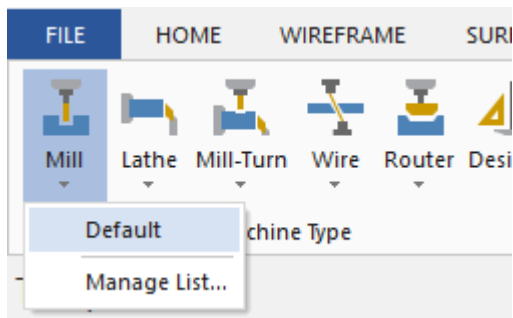


d. Click **OK**.

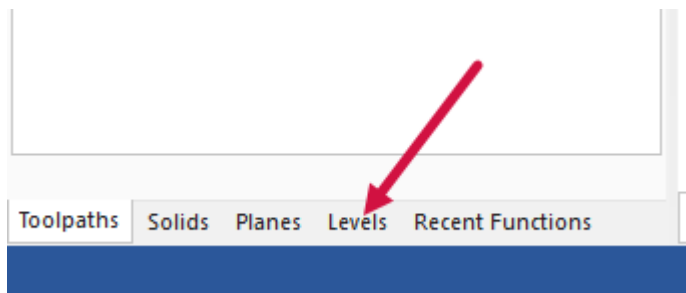
## Exercise 2: Setting up the stock

In this exercise, you will set up stock that will be 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**.

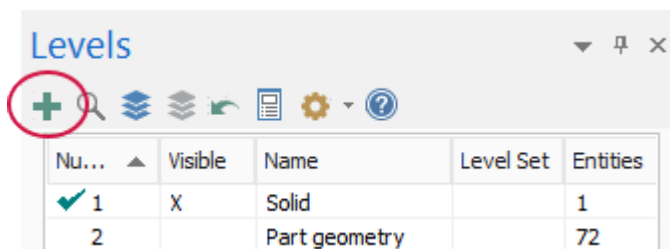


4. Select the Levels Manager, located in the same area as the Toolpaths Manager. If it is not displayed, select **Levels** on the **View** tab to toggle the display, or press **[Alt+Z]**.



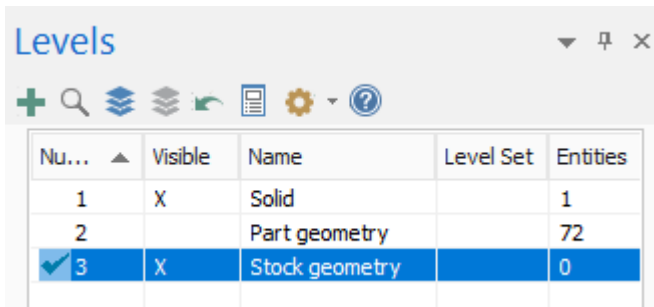
Before creating the stock shape, you will first create a level on which to place the stock geometry. This is a good practice for when you need to organize part files that have thousands of lines, arcs, solids, or surfaces.

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.

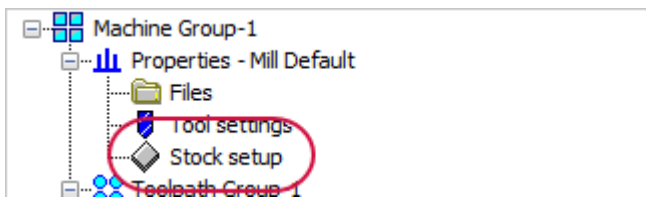
6. Double-click in the **Name** column of the level you have just created and rename it to **Stock geometry**.



Nu...	Visible	Name	Level Set	Entities
1	X	Solid		1
2		Part geometry		72
3	X	Stock geometry		0

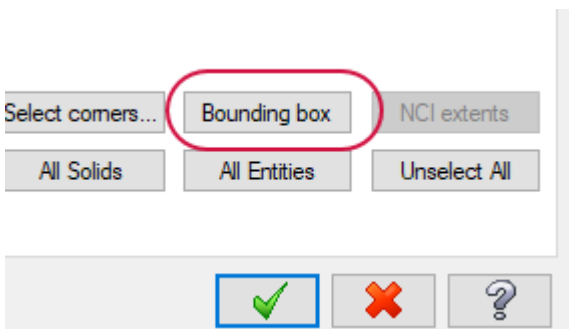
This level will now be used to place the geometry created by the stock setup.

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



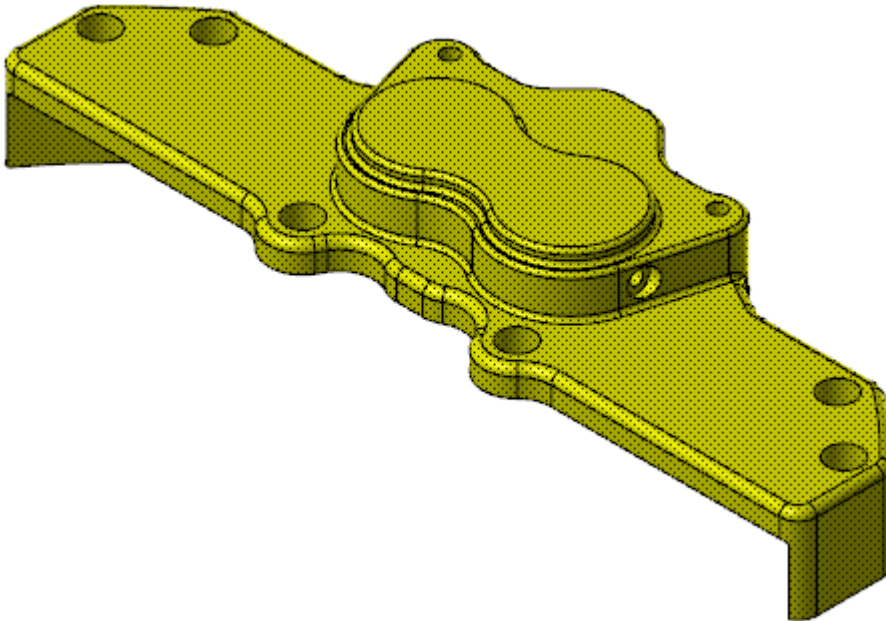
The **Machine Group Properties** dialog box displays.

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:

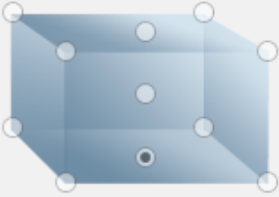
Shape

☒ Rectangular

☐ Cylindrical

Rectangular Settings

Origin:



Size:

X: 152.4

Y: 49.53

Z: 28.6

Cylindrical Settings

Push Pull

Create Geometry

☒ Lines and arcs

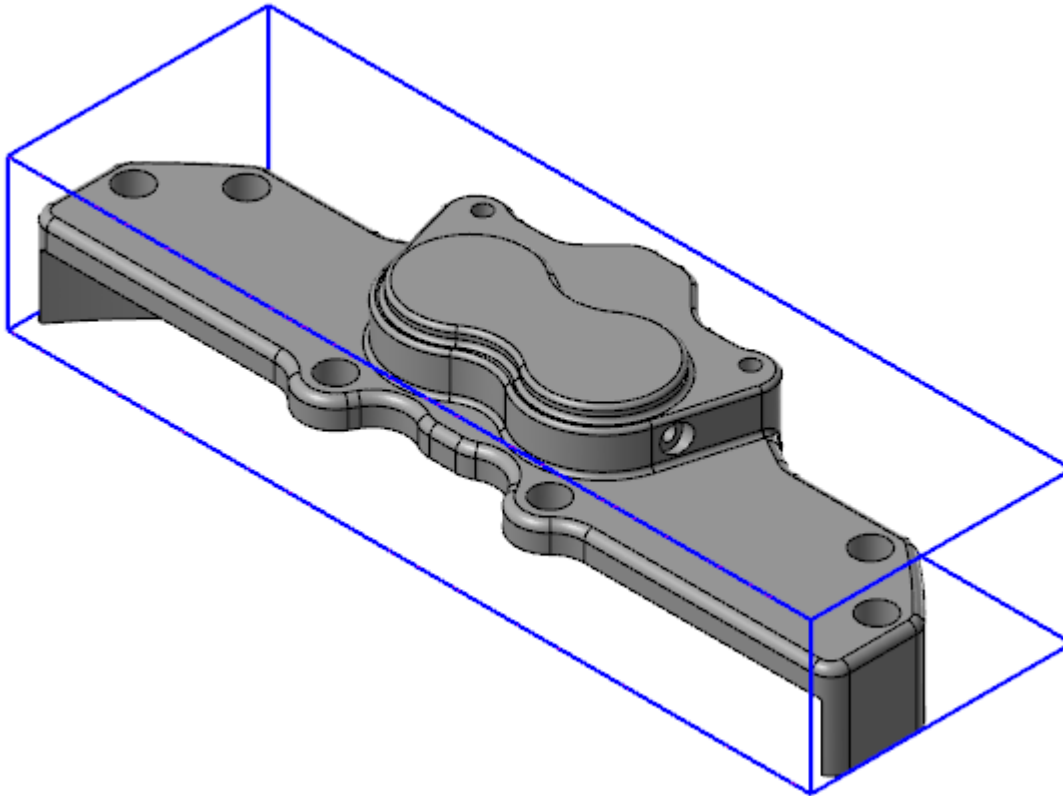
☐ Corner points

☐ Center point

☐ Face center points

☐ Solids

- **Shape** to **Rectangular**. This shape is better suited for this part as opposed to using 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 lines and arcs based on the stock shape 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.



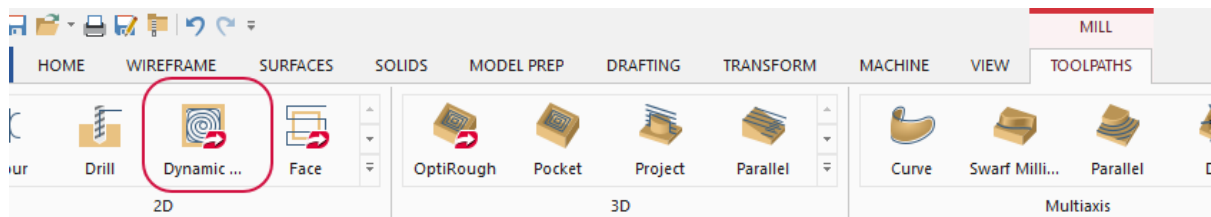
You have now created the necessary stock and geometry to create the Dynamic Mill toolpaths.

14. Save your part.

## Exercise 3: Creating the first Dynamic Mill toolpath

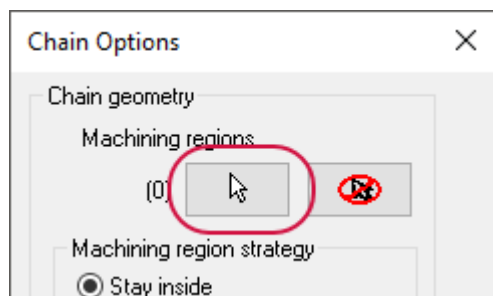
You now create the first Dynamic Mill toolpath to machine the outside of the part.

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



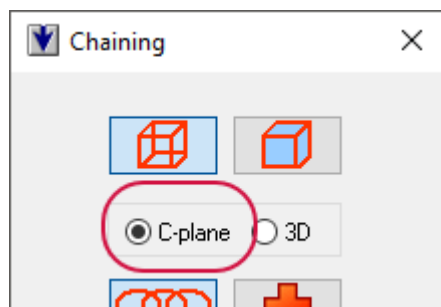
The **Chain Options** dialog box displays.

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



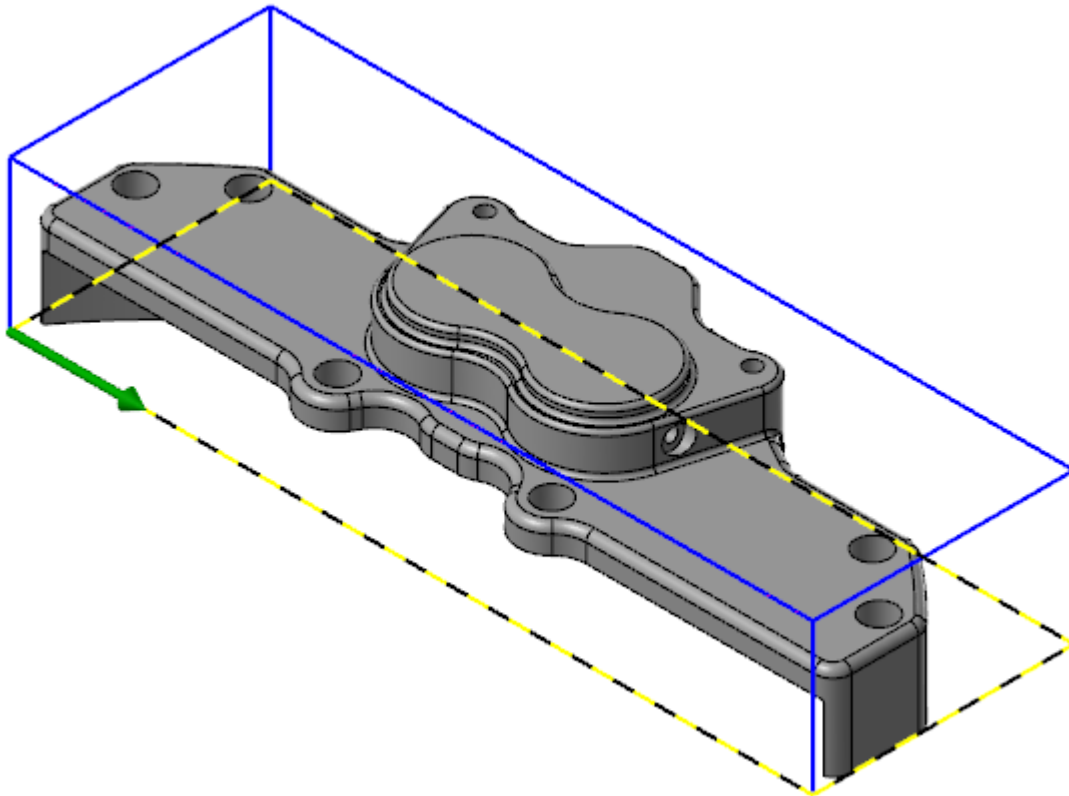
The **Chaining** dialog box displays.

3. Select **C-plane** in the **Chaining** dialog box.



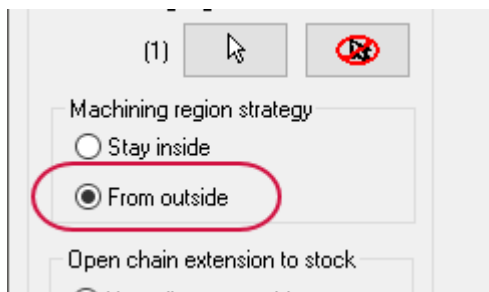
This option allows you to chain only entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain.

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



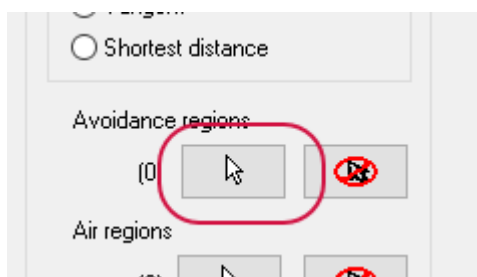
By changing the chaining selection to Cplane, you avoid selecting each line individually.

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



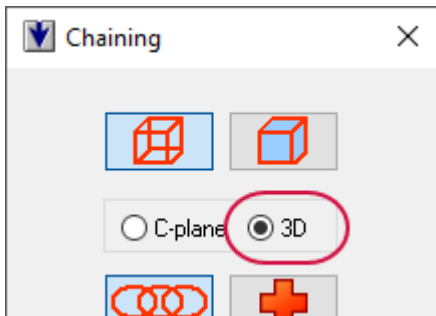
**From outside** uses free-flowing motion to machine features such as standing bosses or cores. This is the strategy you want to clear the outside of the part.

7. 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.



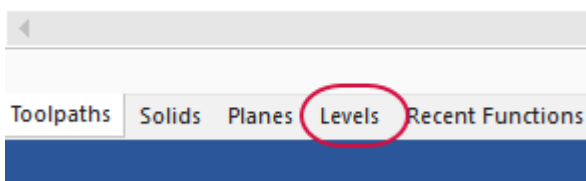
The **Chaining** dialog box displays.

8. Select **3D** in the **Chaining** dialog box.

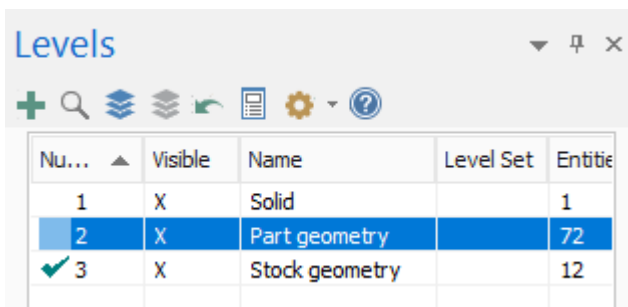


This mode chains entities defined in the X, Y, and Z axes simultaneously.

9. Select the Levels Manager. Move the **Chaining** dialog box out of the way if necessary.



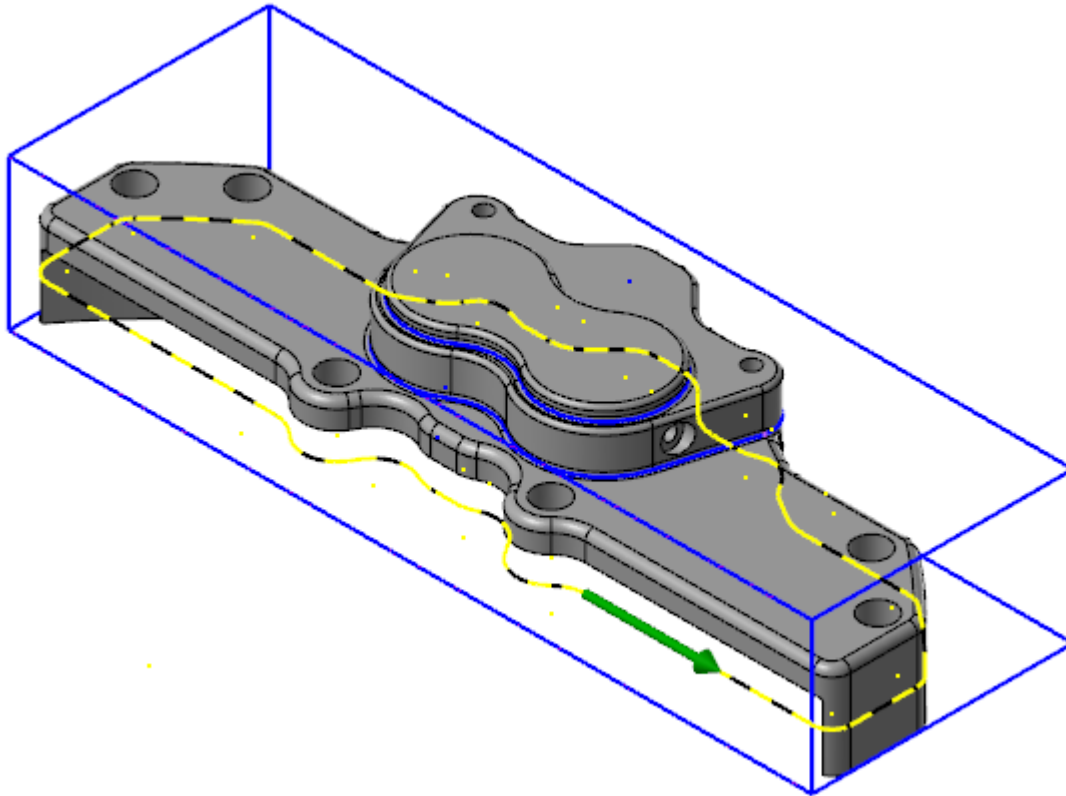
10. Click in the **Visible** column for the Part geometry level.



This will display the geometry we need to select. The geometry is a silhouette of the part. This allows you to machine around the part and clear out the stock, but without cutting into the part.



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



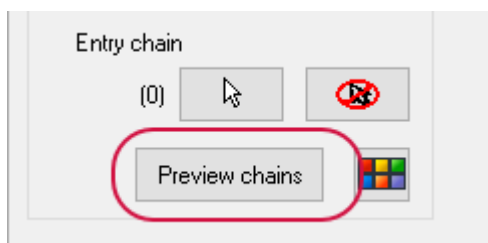
12. Click **OK** in the **Chaining** dialog box to accept the chain and return to the **Chain Options** dialog box.

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

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



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



Your part displays as shown in the following image:



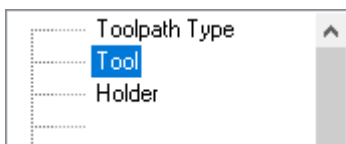
Mastercam displays your material in the red and black crosshatch, the motion region as blue, and the tool containment as yellow. By using **Preview chains**, you can determine if that is what you want from the toolpath before entering any parameters.

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

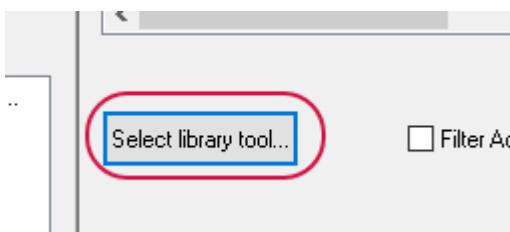
15. Once you are satisfied, right-click in the graphics window and select **Isometric (WCS)**.
16. Click **OK** in the **Chain Options** dialog box to accept these chains.

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

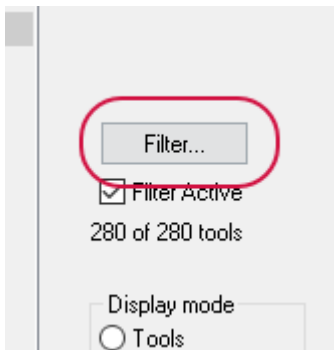
17. Select the **Tool** page.



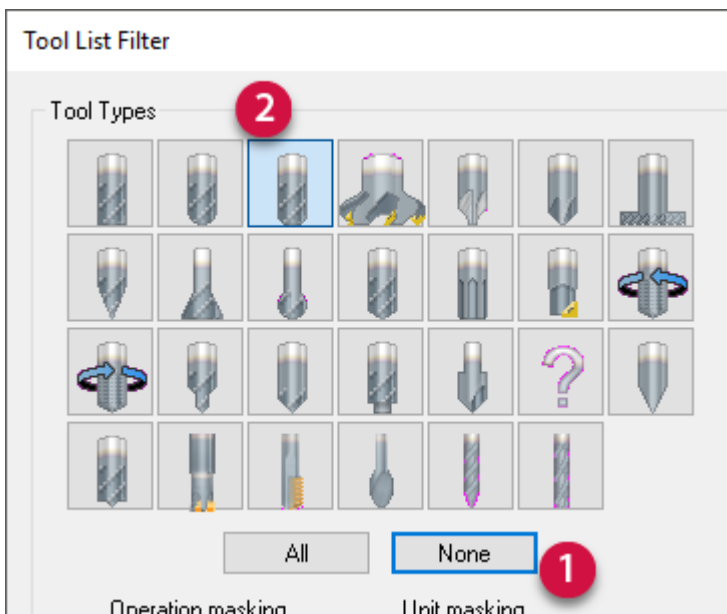
18. Click **Select library tool**.



The **Tool Selection** dialog box displays.

19. Select **Filter**.

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

20. Select **None** to remove all filters, and then select **EndMill3 Bull**. This ensures that you are only showing Bull end mills tools.

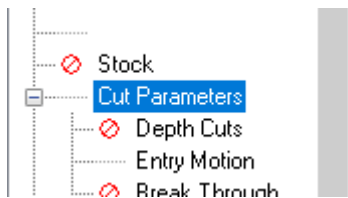
**Note:** You can also double-click on a tool image to set the filter.

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

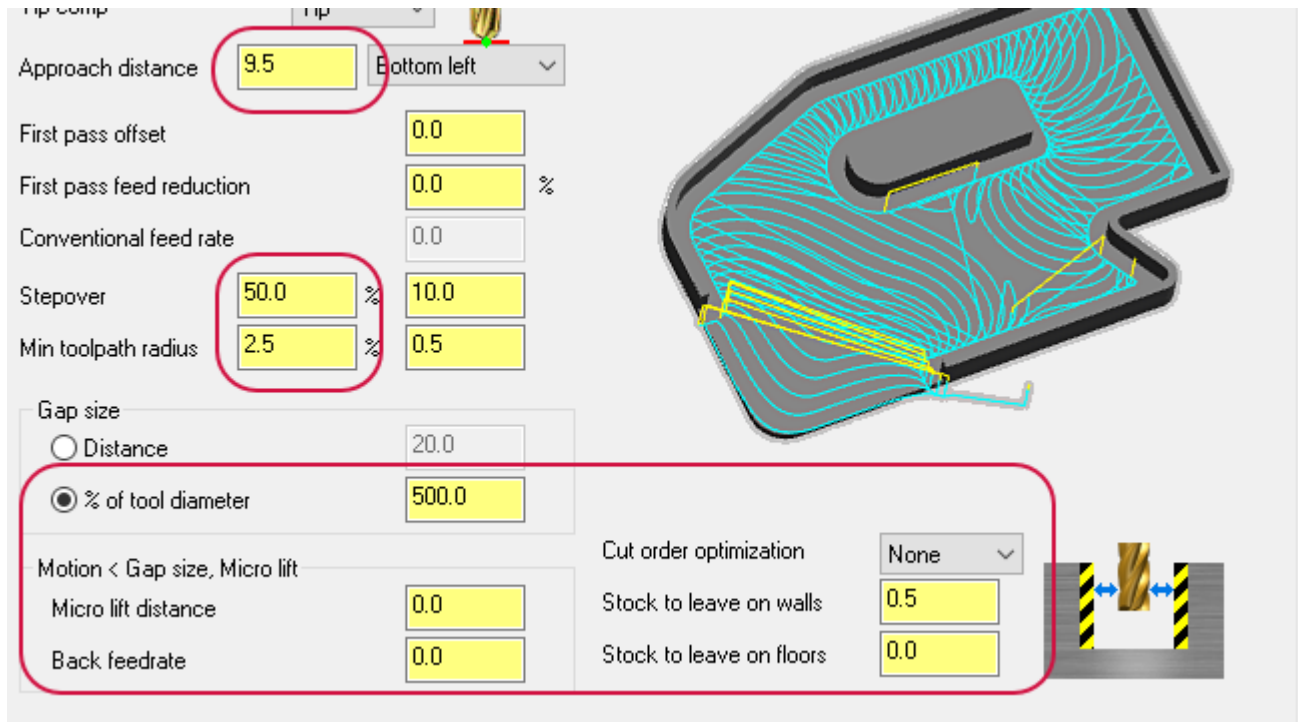
#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. .
5	END MILL WITH RADIUS - 10 / R2.0	--	10.0	2.0	16.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 8 / R0.5	--	8.0	0.5	13.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 12 / R0.5	--	12.0	0.5	19.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 10 / R1.0	--	10.0	1.0	16.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 20 / R4.0	--	20.0	4.0	32.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 5 / R0.5	--	5.0	0.5	8.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 20 / R1.0	--	20.0	1.0	32.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 6 / R1.0	--	6.0	1.0	10.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 12 / R1.0	--	12.0	1.0	19.0	4	Bull e...	Corne
5	END MILL WITH RADIUS - 6 / R0.5	--	6.0	0.5	10.0	4	Bull e...	Corne

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

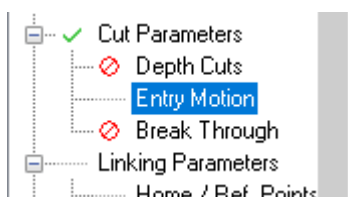
**Note:** You can also double-click the tool to add it to the toolpath.

24. Select the **Cut Parameters** page.

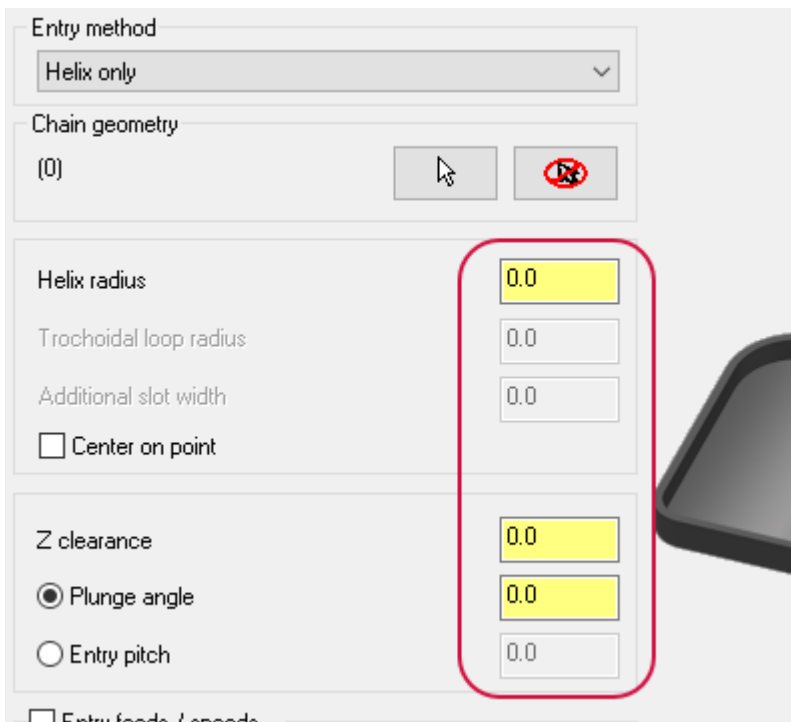
## 25. Set the following parameters:



- **Approach distance** to 9.5. This 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 feed rate** 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.

26. Select the **Entry Motion** page.

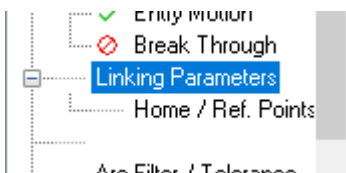
27. Set the following parameters:



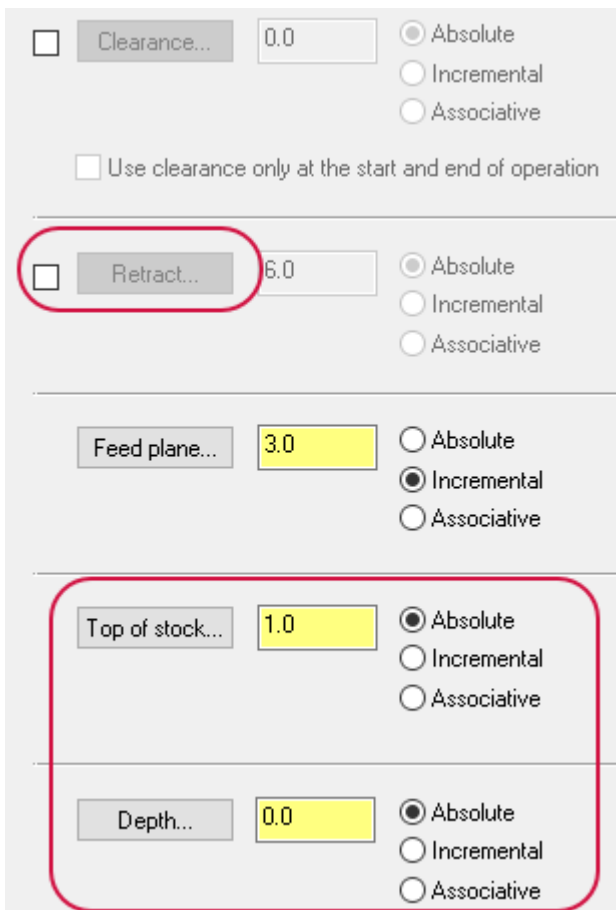
- **Helix radius** to **0.0**. Sets the radius of the entry helix.
- **Z clearance** to **0.0**. Sets an extra height used in the ramping motion down from a top profile.
- **Plunge angle** to **0.0**. Sets the angle of descent for the entry move and determines the pitch.

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

28. Select the **Linking Parameters** page.



## 29. Set the following parameters:



☐ Clearance... 0.0 ☒ Absolute  
☐ Incremental  
☐ Associative  
☐ Use clearance only at the start and end of operation

---

☐ Retract... 6.0 ☒ Absolute  
☐ Incremental  
☐ Associative

---

Feed plane... 3.0 ☐ Absolute  
☒ Incremental  
☐ Associative

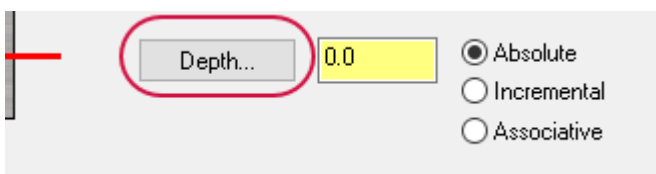
---

Top of stock... 1.0 ☒ Absolute  
☐ Incremental  
☐ Associative

---

Depth... 0.0 ☒ Absolute  
☐ Incremental  
☐ Associative

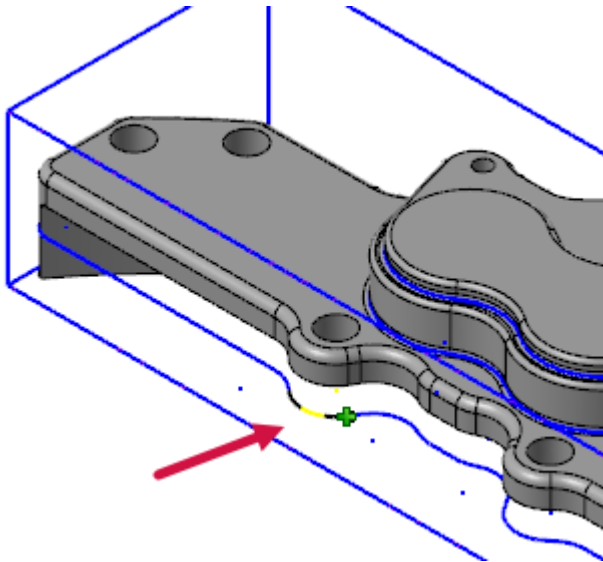
- Deselect **Retract**. The retract distance is the height that the tool moves up to before the next tool pass.
- Change **Top of stock** and **Depth** to **Absolute**. Mastercam measures absolute values from the origin 0,0,0.

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


☐ Depth... 0.0 ☒ Absolute  
☐ Incremental  
☐ Associative

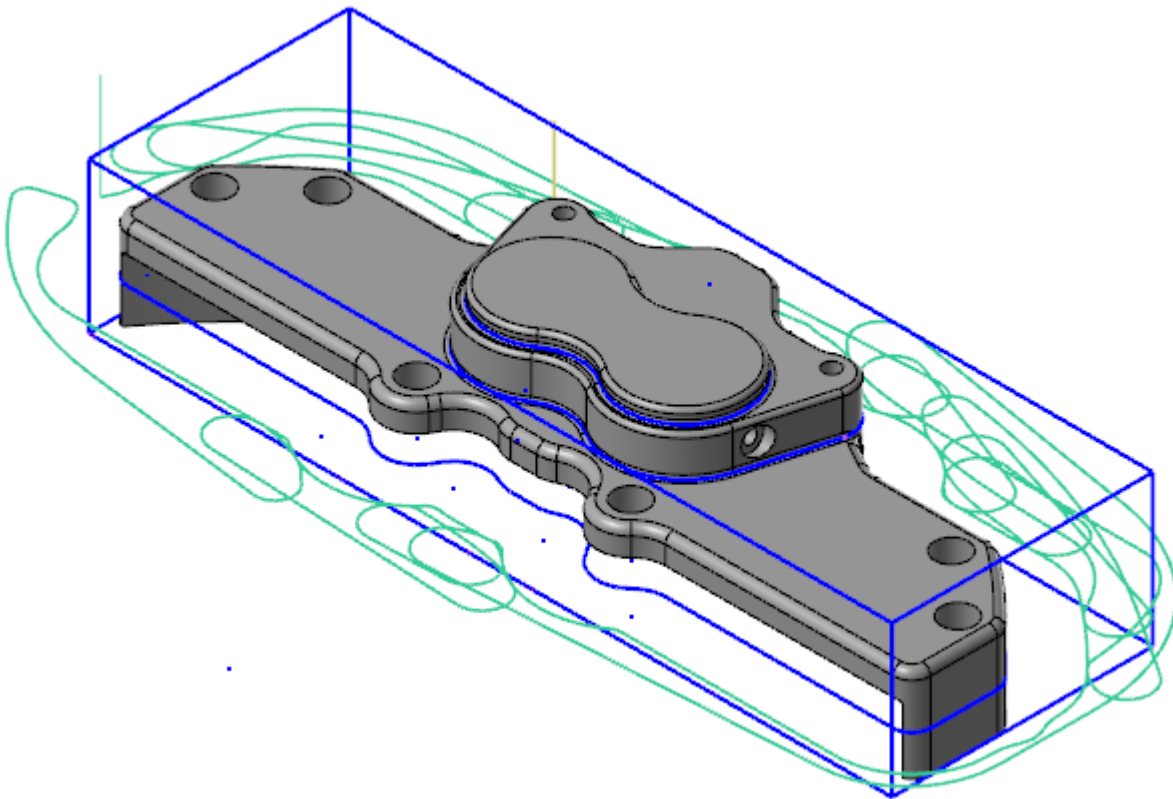
This determines the final machining depth and the lowest depth that the tool descends into the stock. In this case, the machining depth will be in the middle of the part because Dynamic Mill uses the entire tool to cut.

31. Select the chain shown below:



You will automatically return to the **2D High Speed Toolpath - Dynamic Mill** dialog box. Your **Depth** should now be set at **-20.32**.

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

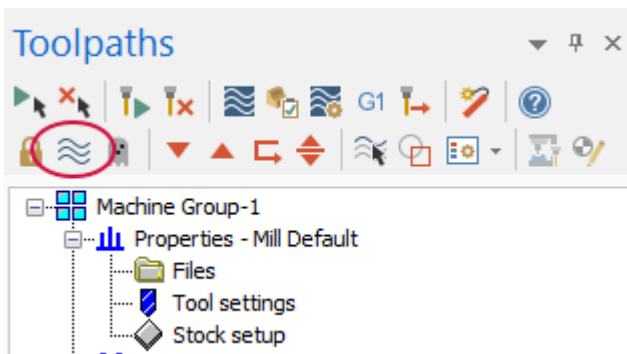


34. Save your part.

## Exercise 4: Creating the second Dynamic Mill toolpath

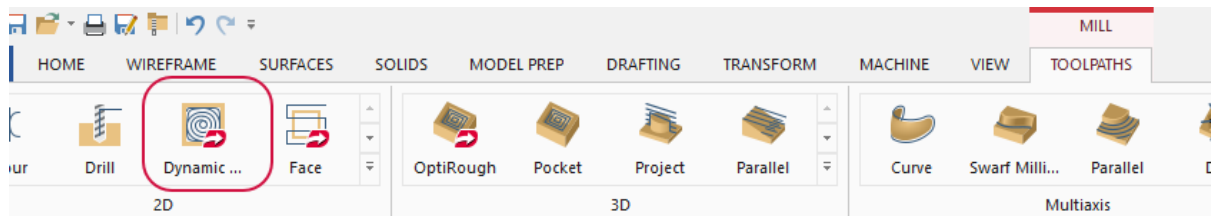
You will now create a second Dynamic Mill toolpath using the **Rest material** option.

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



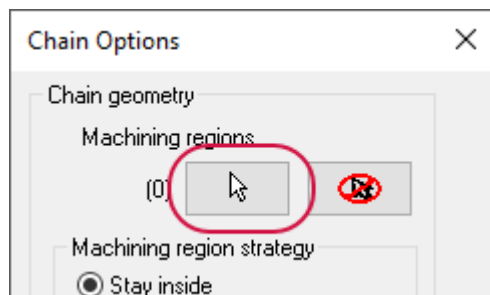
This will hide the display of the first toolpath and keep your graphics view clean while you create another toolpath.

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



The **Chain Options** dialog box displays.

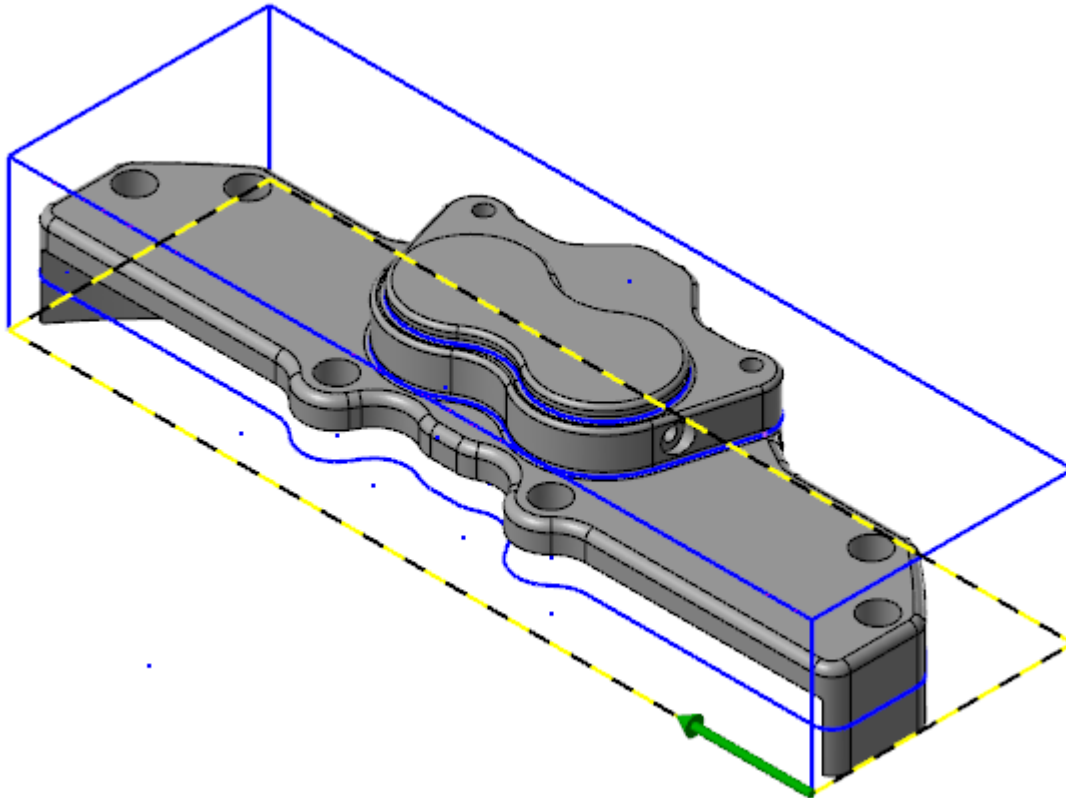
3. Click **Select** under Machining regions.



The **Chaining** dialog box displays.

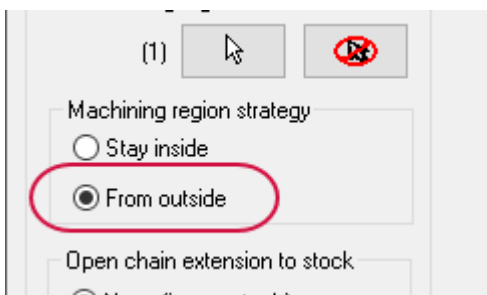


4. Select the geometry shown below as the machining region. Use the **Cplane** button to select the entire chain.

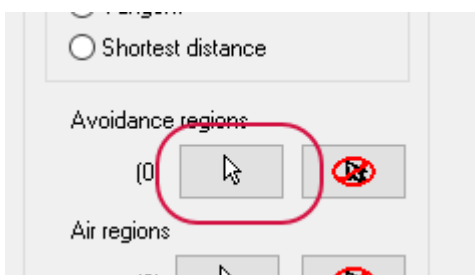


The selection is the same geometry used in the first Dynamic Mill toolpath.

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

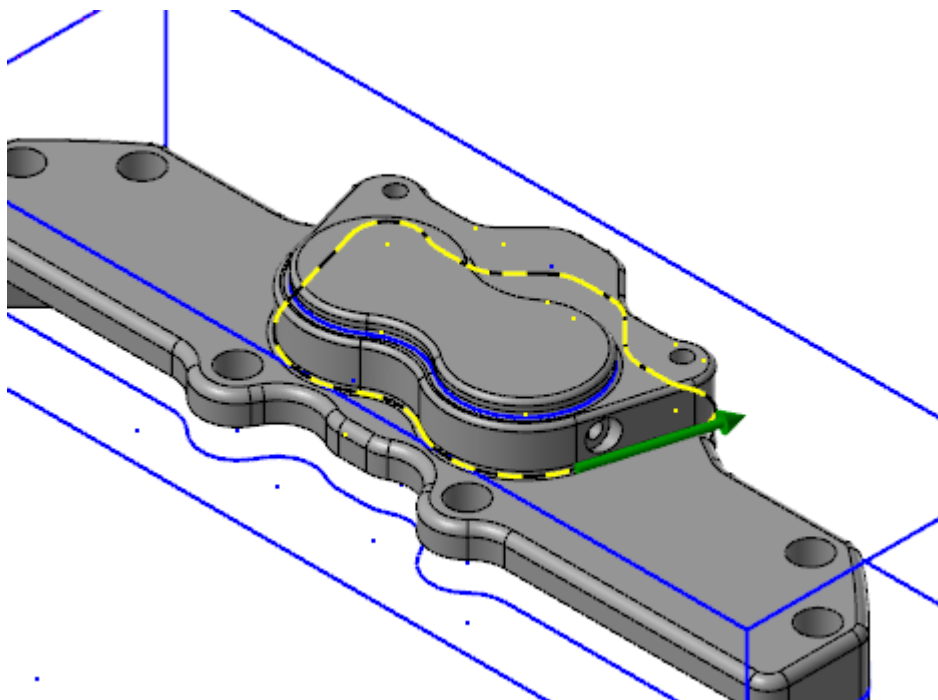


7. Click **Select** under **Avoidance regions**.



The **Chaining** dialog box displays.

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

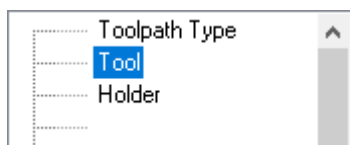


You may need to rotate the part or turn off the **Solid** level to select the chain.

9. Click **OK** in the **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**.

#	Assembly Na...	Tool Name	Holder Name
5	--	END MILL ...	--

Tool diameter:

Corner radius:

Tool name:

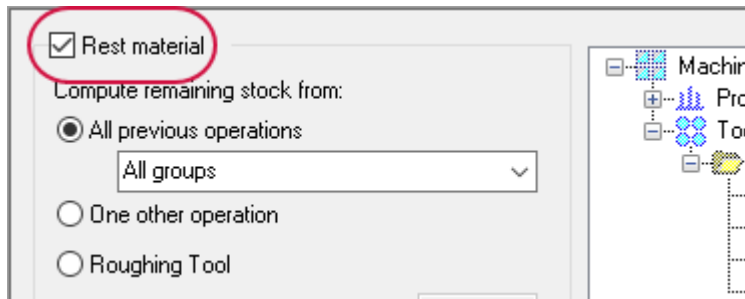
Tool #:  Length offset:

Head #:  Diameter offset:

13. Select the **Stock** page.

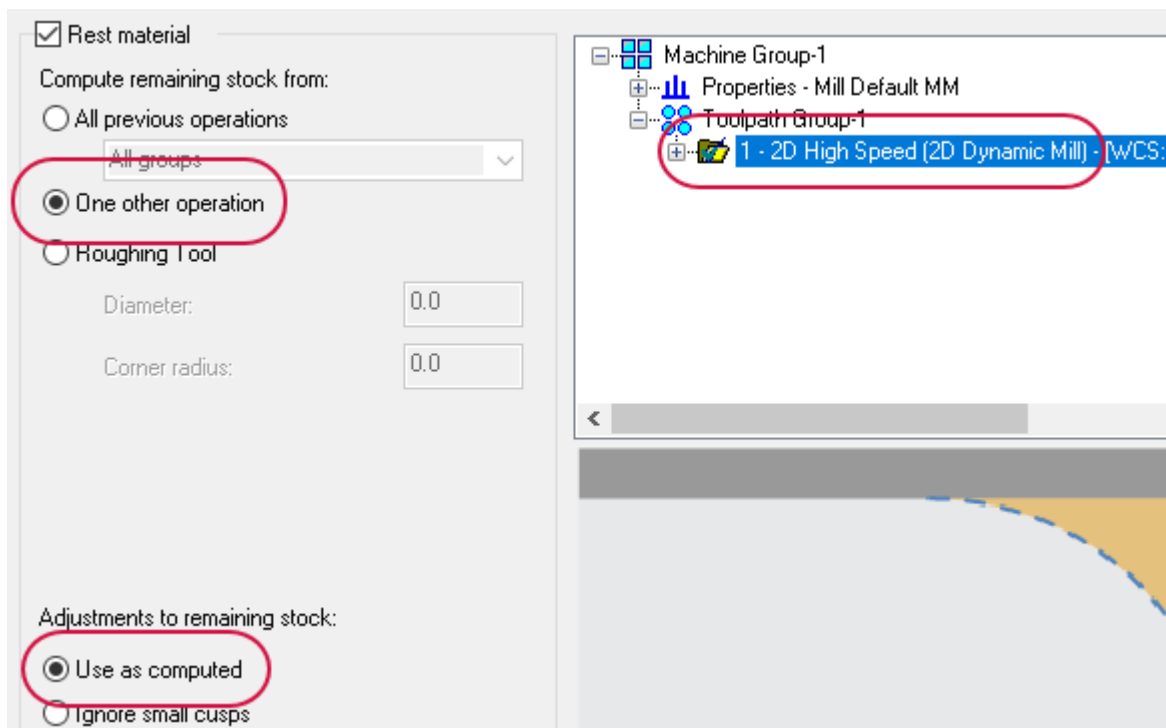


14. Select **Rest material**.



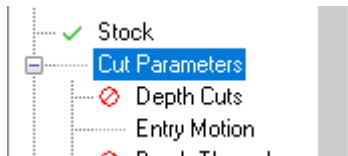
This enables the page and tells Mastercam to calculate the cutting passes based on the remaining stock.

15. Set the following parameters:

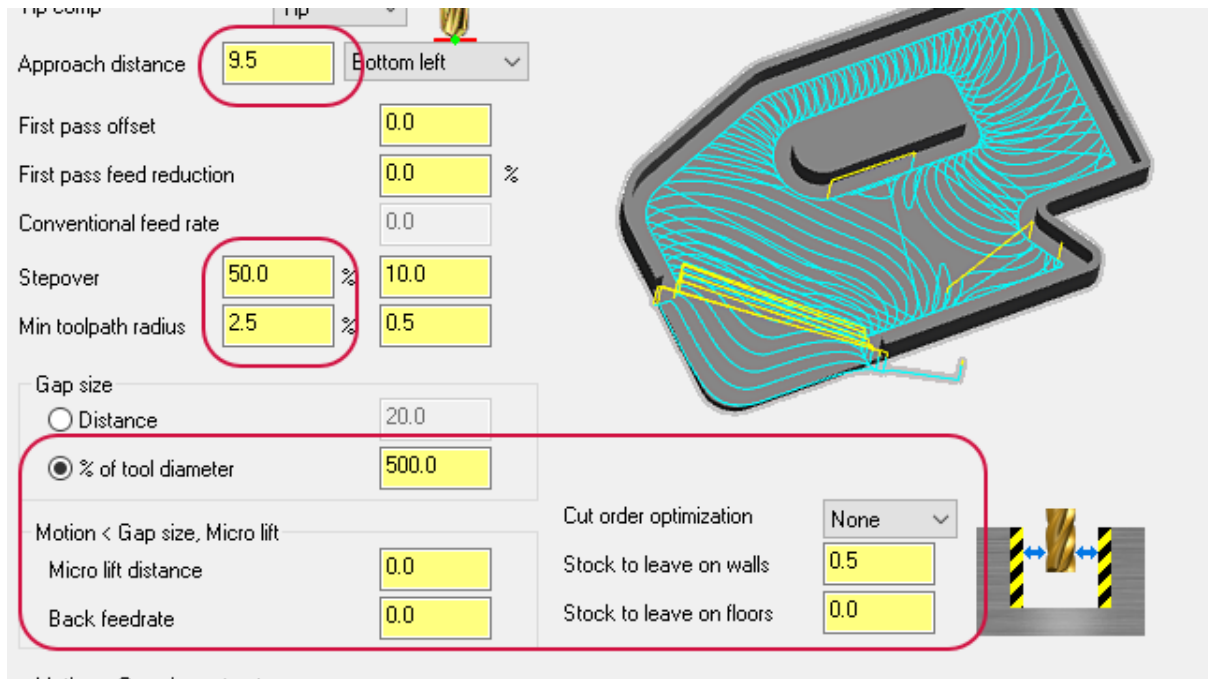


- Set **Compute remaining stock from** to **One other operation**. This calculates the remaining stock from a single operation.
- In the **Operation List**, select the **2D High Speed (2D Dynamic Mill)** toolpath.
- Set **Adjustments to remaining stock** to **Use as computed**. This option makes no adjustments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

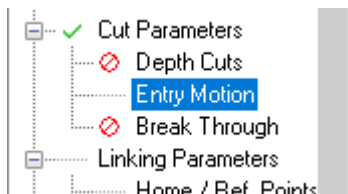
16. Select the **Cut Parameters** page.



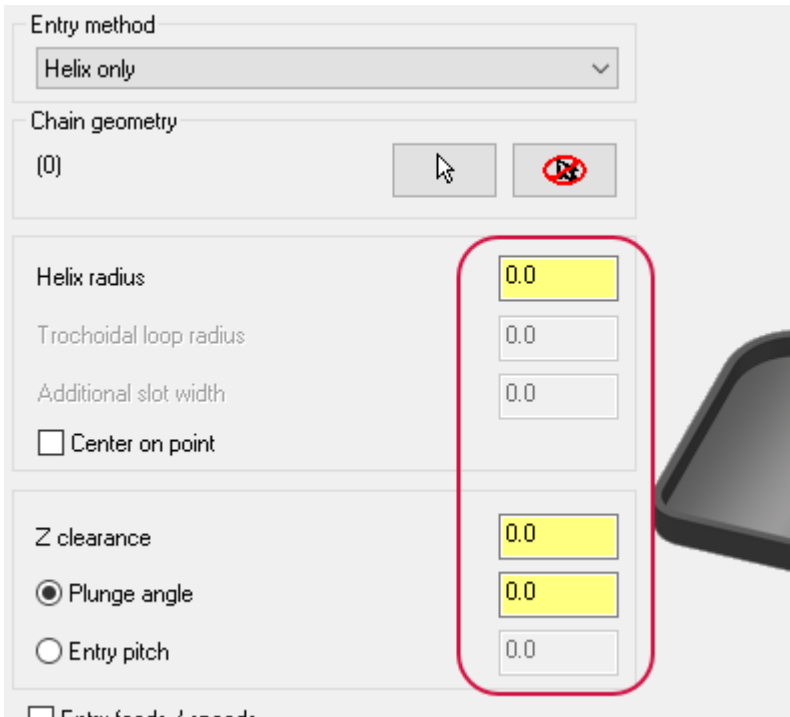
17. The parameters are the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



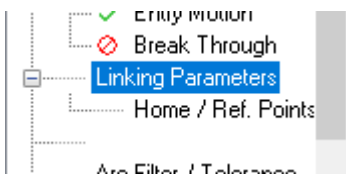
18. Select the **Entry Motion** page.



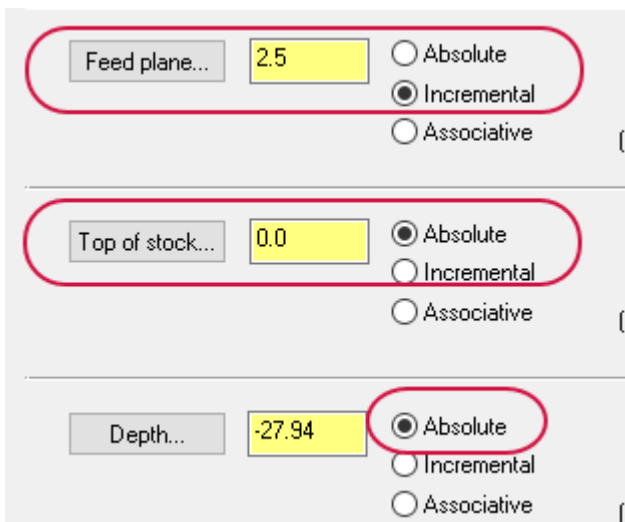
19. The parameters are also the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



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



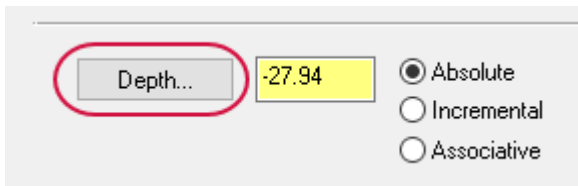
21. Set the following parameters:



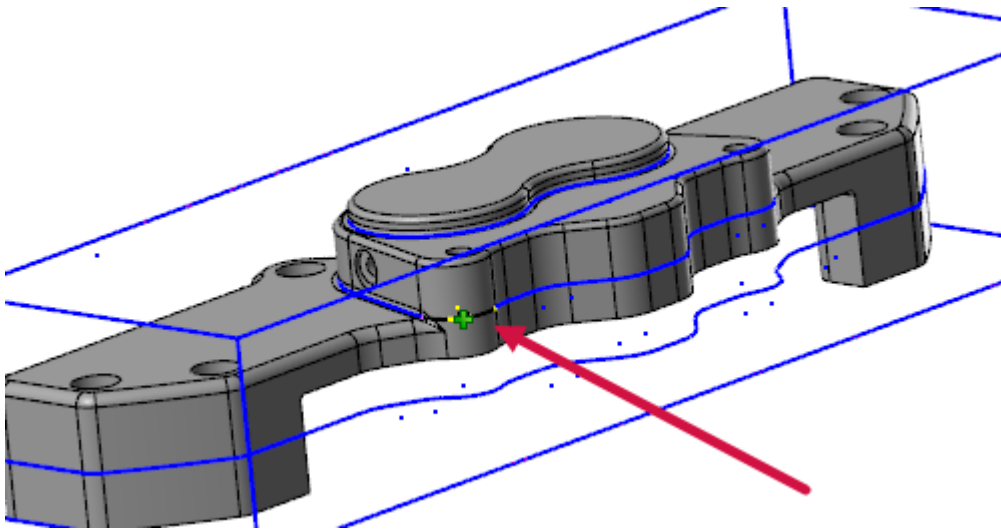
- **Feed plane** to 2.5. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- Ensure that **Feed plane** is set to **Incremental**.

- **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.



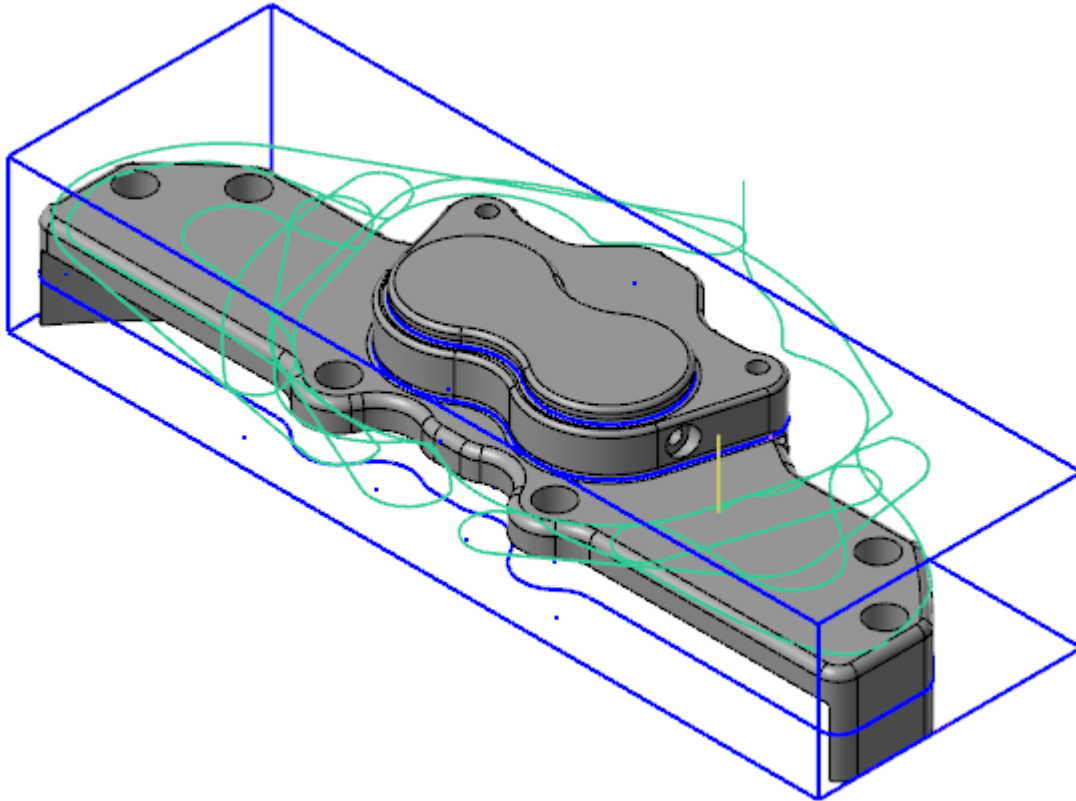
23. Select the arc shown below:



You will need to rotate the part to select it.

24. You will automatically return 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:



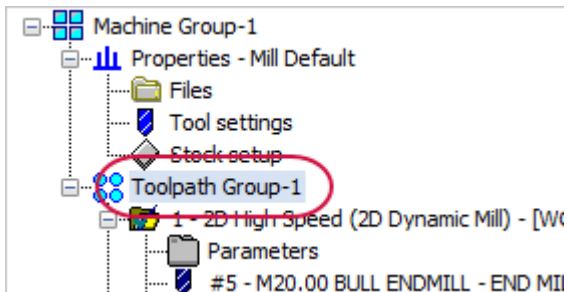
You will notice that the toolpath clears out the top of the part while avoiding the large feature that was selected as the avoidance region.

27. Save your part.

## Exercise 5: Verifying the toolpaths

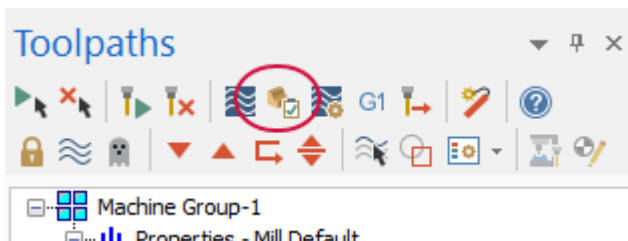
You will now verify both Dynamic Mill toolpaths with Mastercam Simulator. Mastercam Simulator allows you to verify your toolpaths by using solid models to simulate part machining against a selected stock definition. Verify creates a representation of the surface finish and shows collisions, if any exist.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

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



This will select both Dynamic Mill toolpaths.

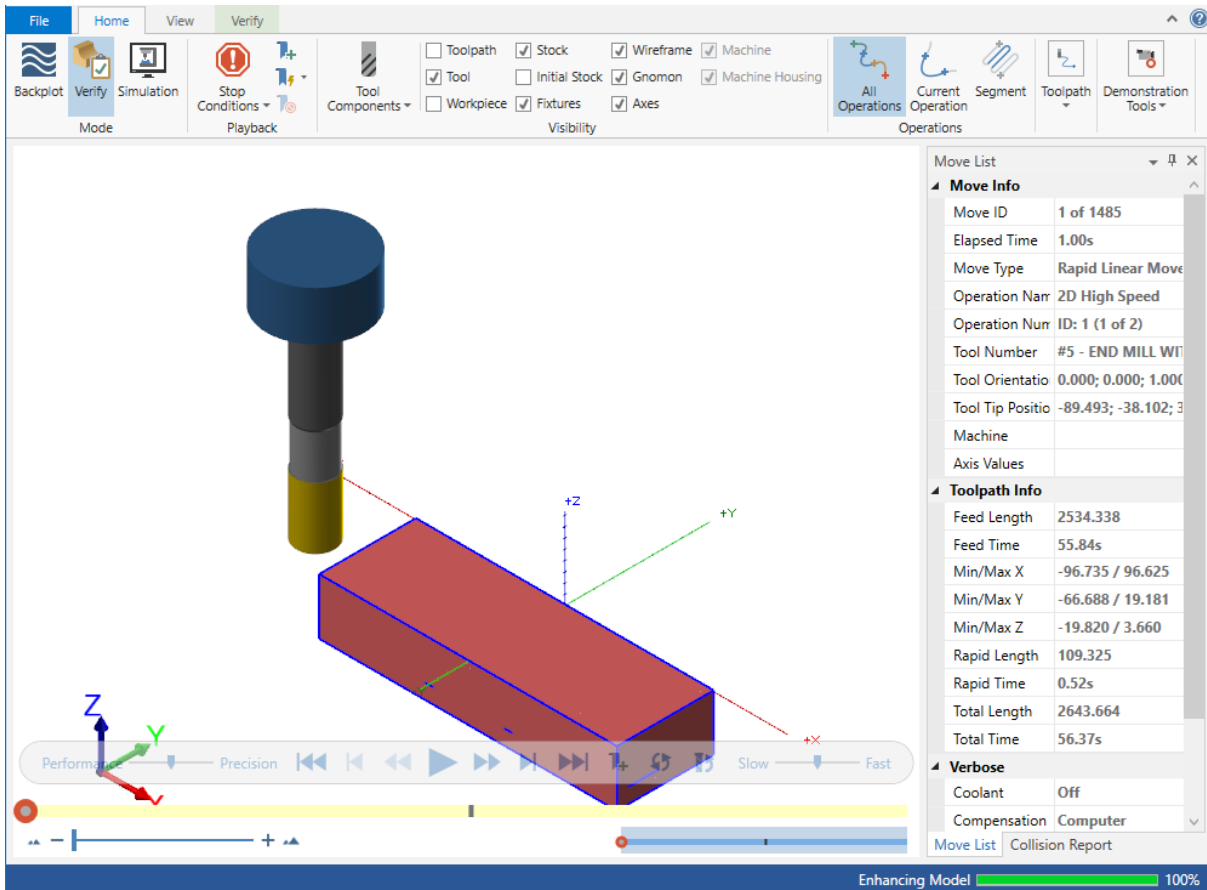
2. Select **Verify selected operations**.



Mastercam Simulator displays.



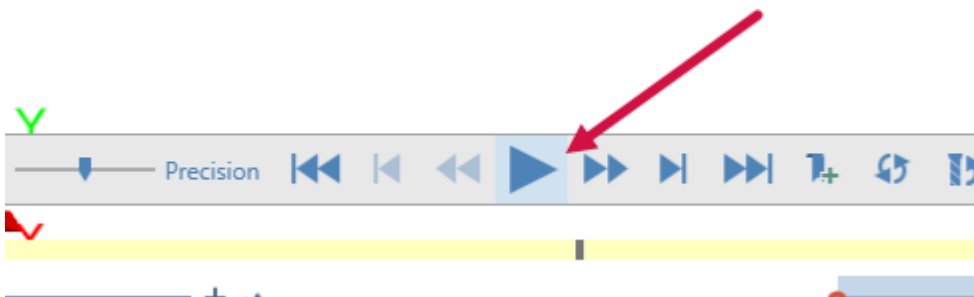
3. Right-click in the graphics view and select **Fit** and **Isometric** from the menu. Mastercam Simulator should match the image shown below.



4. View the Move List, which displays all pertinent information about the toolpaths being verified. The two Dynamic Mill toolpaths are able to machine the part in the time shown with the given parameters.

Rapid Length	109.325
Rapid Time	0.52s
Total Length	2643.664
Total Time	56.37s
<b>Verbose</b>	
Coolant	Off
Compensation	Computer

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 easily and quickly remove all of the stock on the outside and prepares the part for any other necessary operations. In the next lesson, you will create a Face toolpath to further machine this part.

## CHAPTER 2

# THE FACE TOOLPATH

Face toolpaths quickly 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.

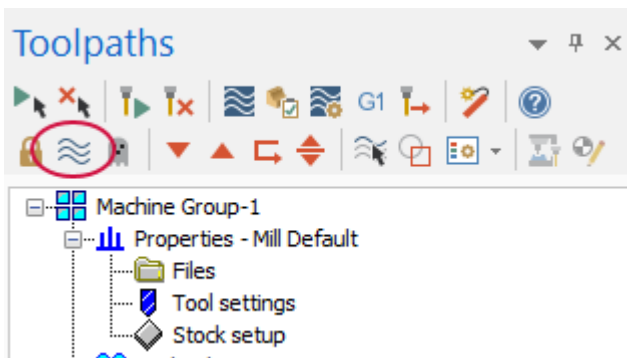
For this lesson, you will create a Face toolpath on the top of the part from the Dynamic Mill lesson.

### Lesson Goals

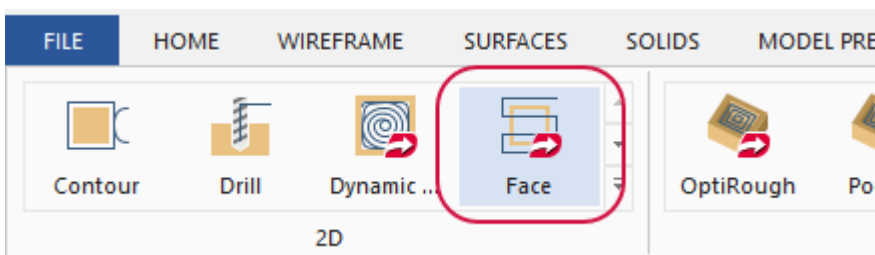
- Create a Face toolpath
- Preview the toolpath before it has been created
- Verify the created toolpath

### Exercise 1: Creating the Face toolpath

1. Open the part file, *FaceMill*, provided with this tutorial or use your part file created from the Dynamic Mill lesson.
2. If you are using your part from the previous lesson, select **Toggle display on selected operations** on the Toolpaths Manager. This will hide the two Dynamic Mill toolpaths and keep the graphics view clean.

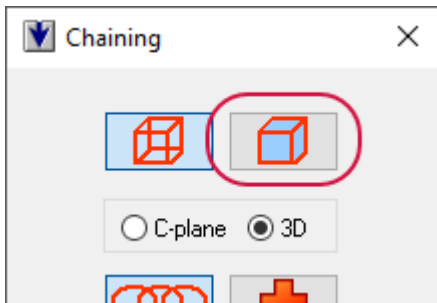


3. Save the part as *FaceMill-xxx*, replacing *xxx* with your initials.
4. Select **Face** from the 2D gallery on the **Mill Toolpaths** contextual tab.



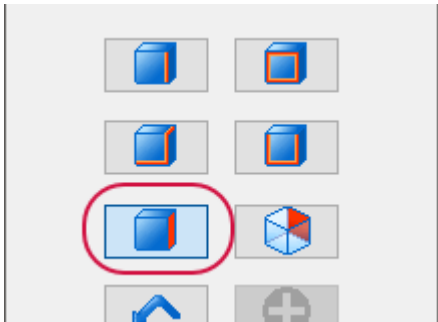
The **Chaining** dialog box displays.

5. Select **Solids** to activate solid selection.

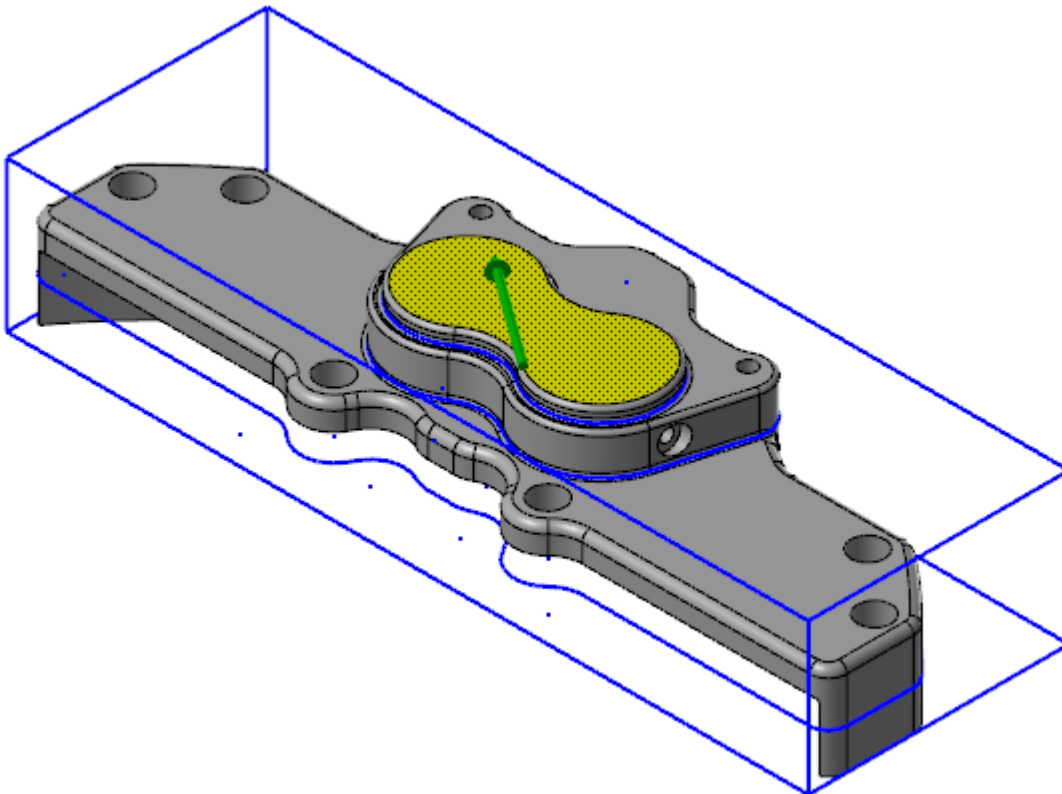


6. Activate **Face** selection.

This allows you to select the face of a solid.



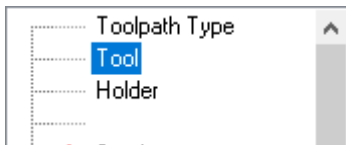
7. Select the face shown below:



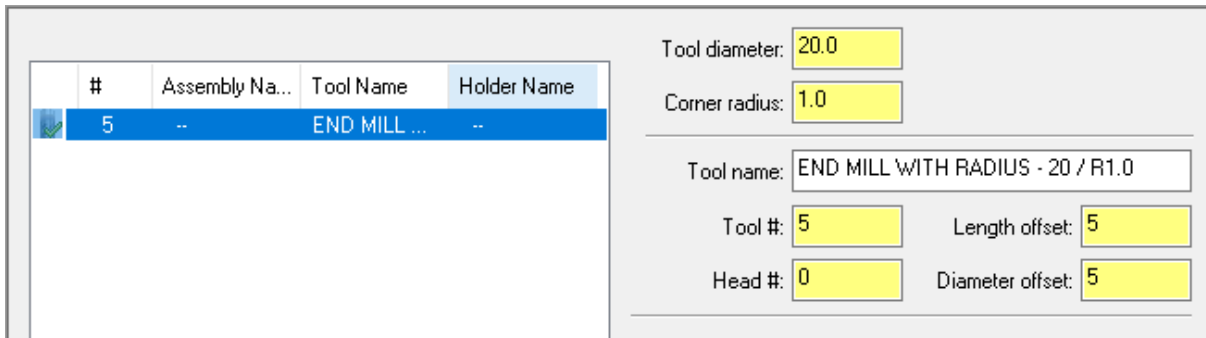
8. Select **OK** in the **Chaining** dialog box.

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

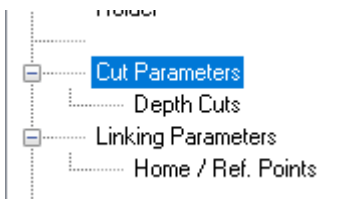
9. Select the **Tool** page.



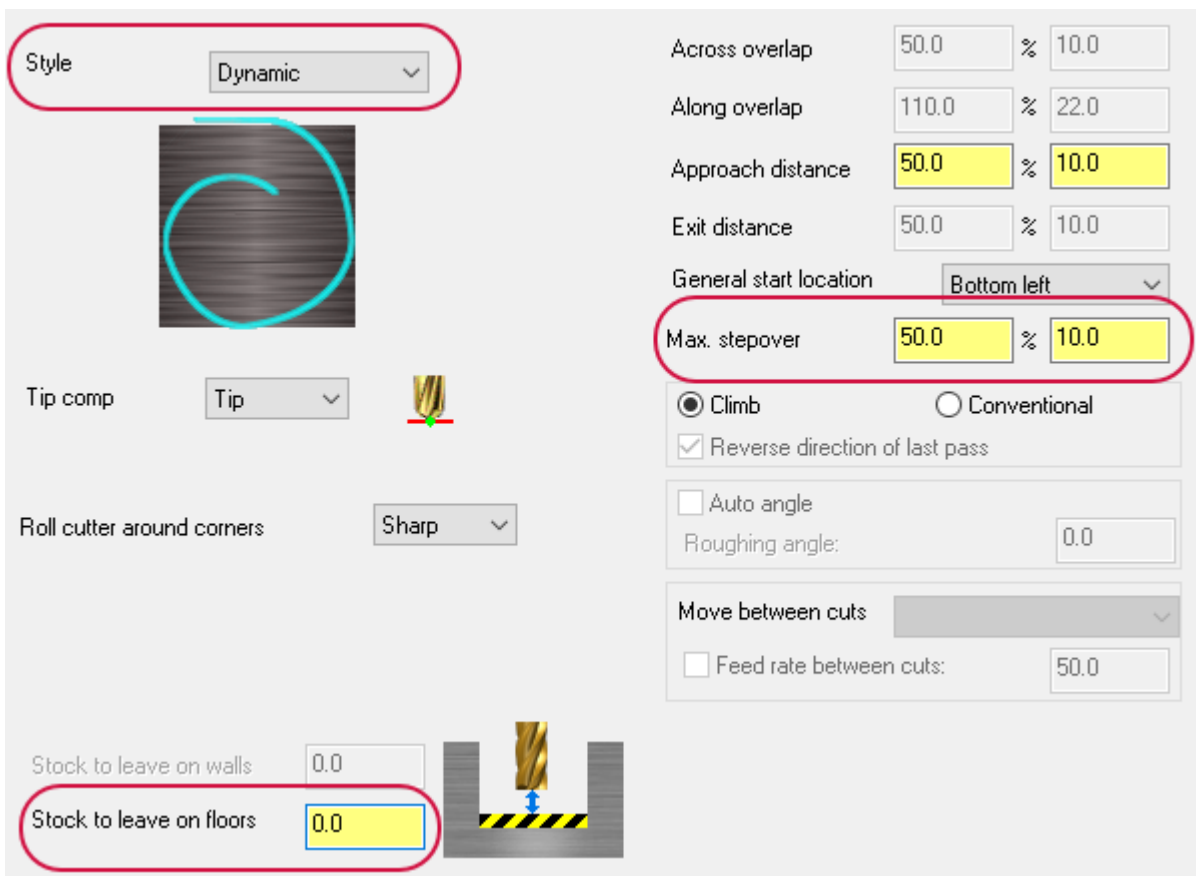
10. Select the **END MILL WITH RADIUS - 20 / R 1.0**, which was used in the previous two Dynamic Mill toolpaths. If necessary, expand the **Tool Name** column to see the entire tool name.



11. Select the **Cut Parameters** page.

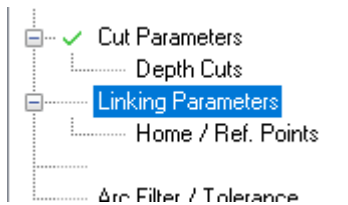


12. Set the following parameters:

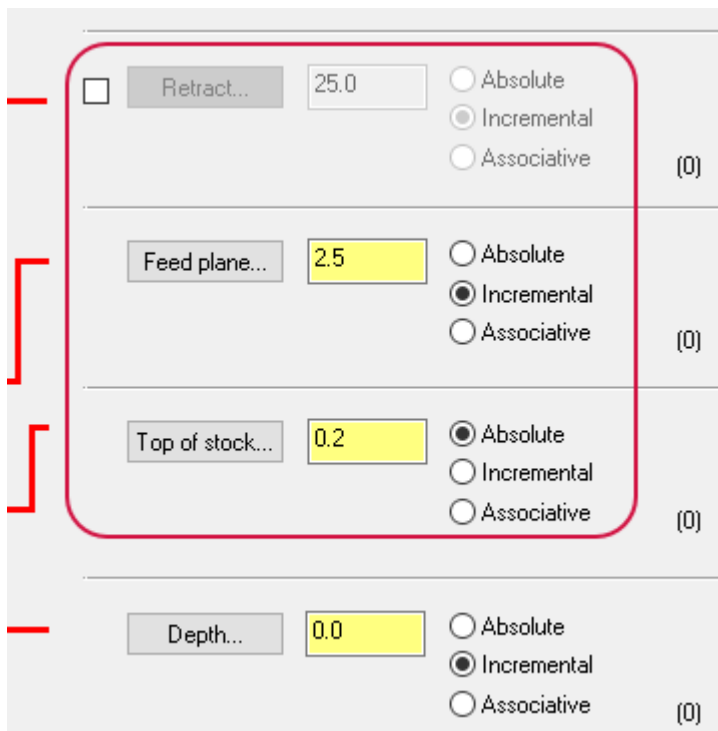


- **Style to Dynamic.** Creates a smooth controlled motion that cuts from the outside to the inside, maintaining a constant load on the tool with minimal entries and exits.
- **Stock to leave on floors to 0.0.** Sets the amount of stock to leave on the floors during machining.
- **Maximum stepover to 50.0.** Sets the distance between adjacent passes in the toolpath.

13. Select the **Linking Parameters** page.



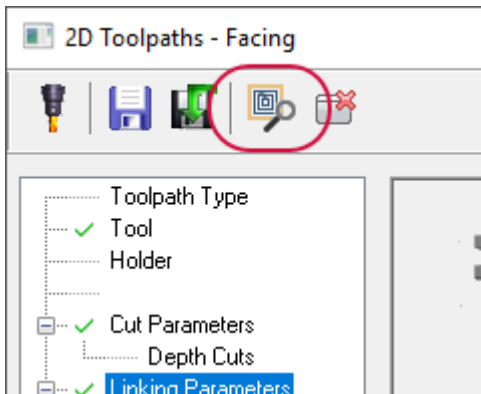
14. Set the following parameters:



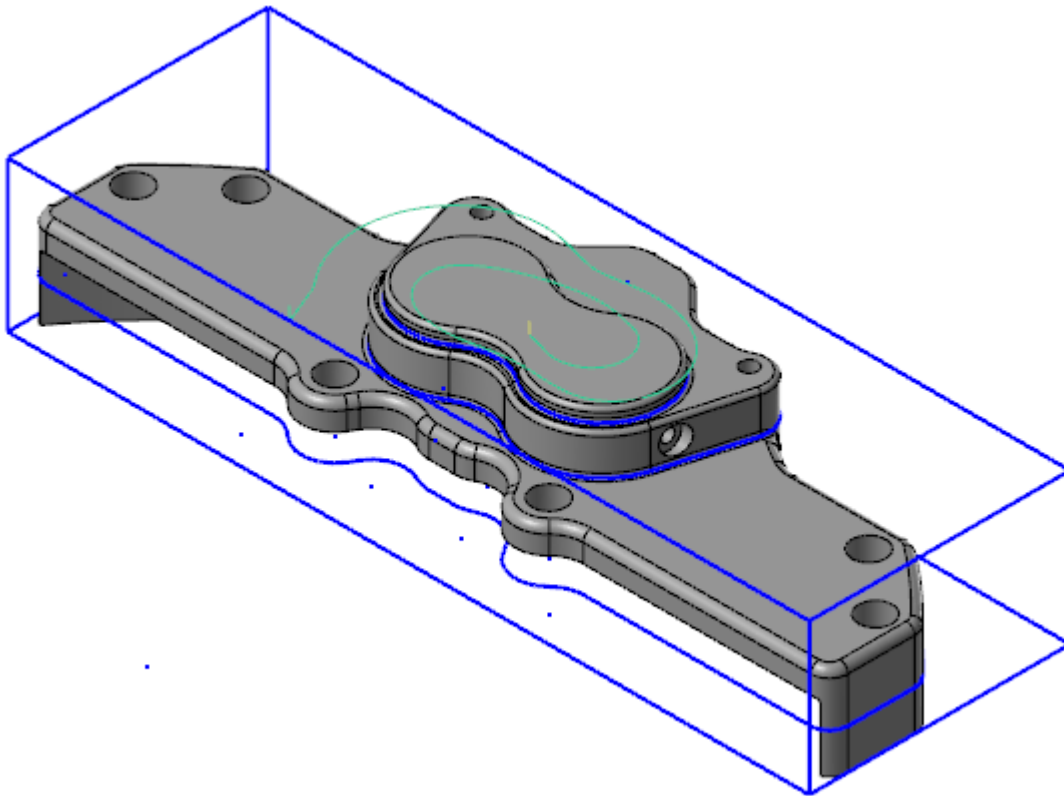
- Deselect **Retract**. Sets the height that the tool moves up to before the next tool pass.
- Set **Feed plane** to **2.5**. Sets the height that the tool rapids before changing to the plunge rate to enter the part.
- Set **Top of stock** to **0.2**. Sets the height of the material in the Z axis.

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

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



16. The toolpath will display on the part as shown below:



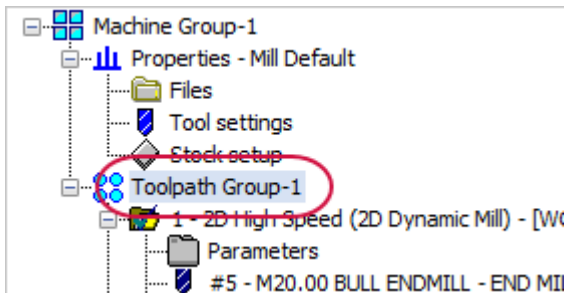
17. Click **OK** in the **2D Toolpaths - Facing** dialog box to close and generate the toolpath.
18. Save your part.

## Exercise 2: Verifying the toolpaths

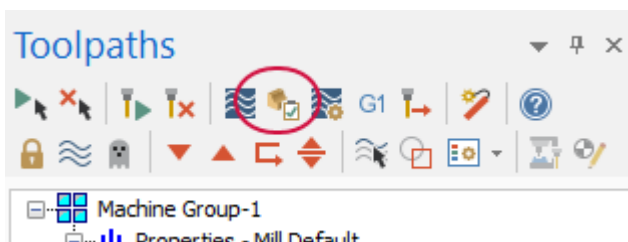
You now verify the two Dynamic Mill toolpaths created from the previous chapter and the Face toolpath together.

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

This selects both Dynamic Mill toolpaths and the Facing toolpath.



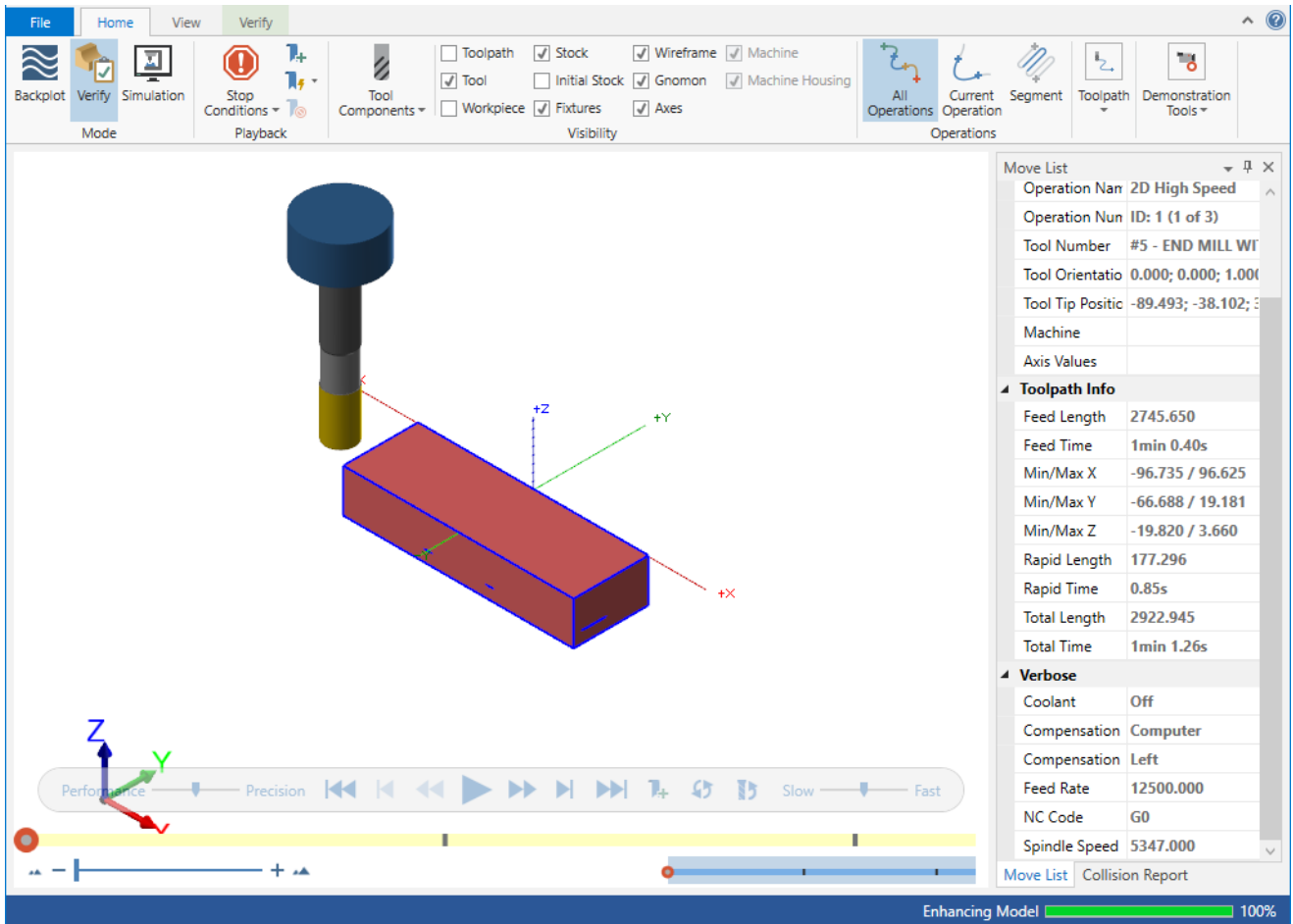
2. Select **Verify selected operations.**



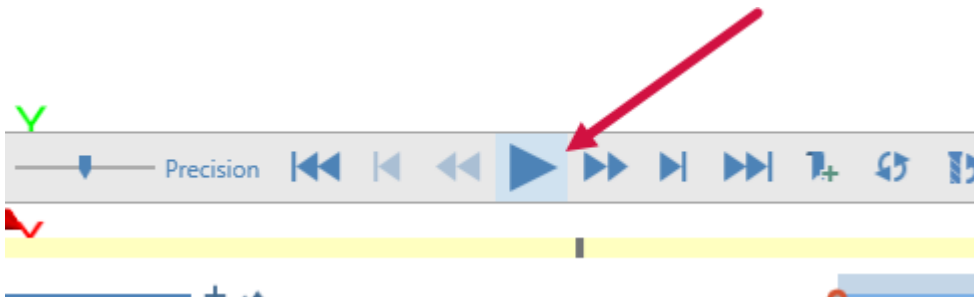
Mastercam Simulator displays.



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



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

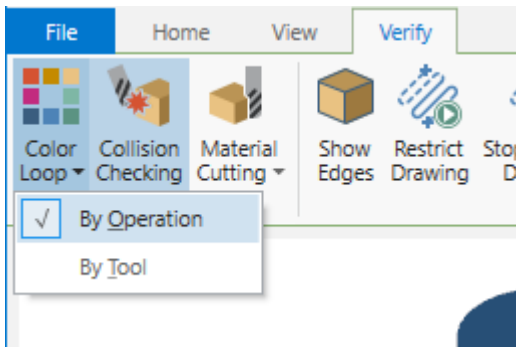


There will be material remaining on the top face of the part. That is to be expected.

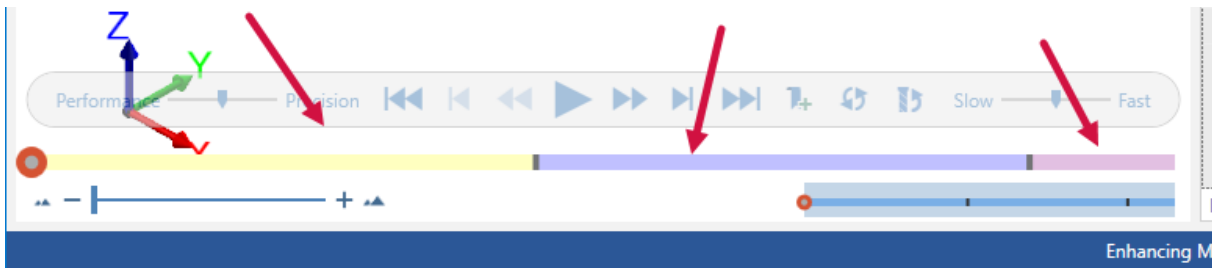
- View the Move List, which displays all pertinent information about the toolpaths being verified. The three Dynamic Mill toolpaths are able to machine the part in the time shown with the given parameters.

Rapid Length	177.296
Rapid Time	0.85s
Total Length	2922.945
Total Time	1min 1.26s
Verbose	
Coolant	Off
Compensation	Computer

6. Select **Color Loop, By Operation** on the **Verify** tab to help distinguish between each toolpath.



Mastercam Simulator color-codes each toolpath so that you can distinguish where each toolpath cuts the part. The colors on the part correspond to those on the time bar.



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 much more efficient cutting by using the entire flute of the tool.

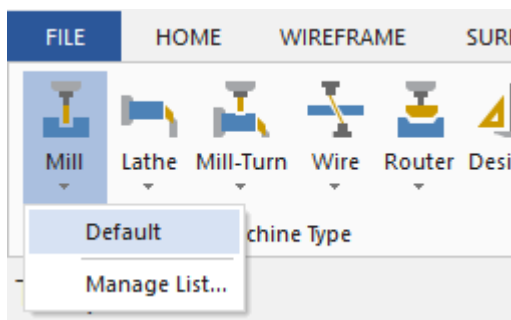
In this lesson, you create a Dynamic Contour toolpath to clear around the walls of the part.

### Lesson Goals

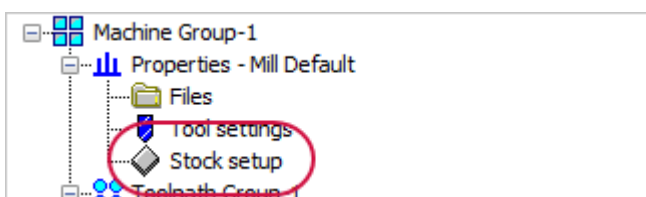
- Create two Dynamic Contour toolpaths
- Analyze the toolpaths
- Verify the created toolpaths

### 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**.

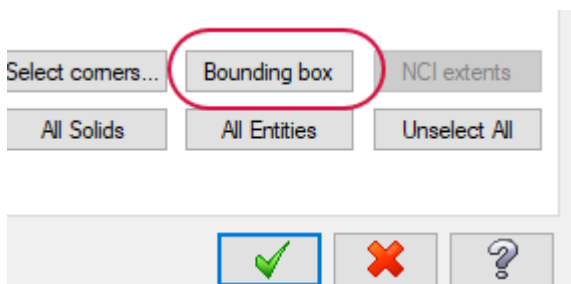


4. Select **Stock setup** from the Toolpaths Manager.



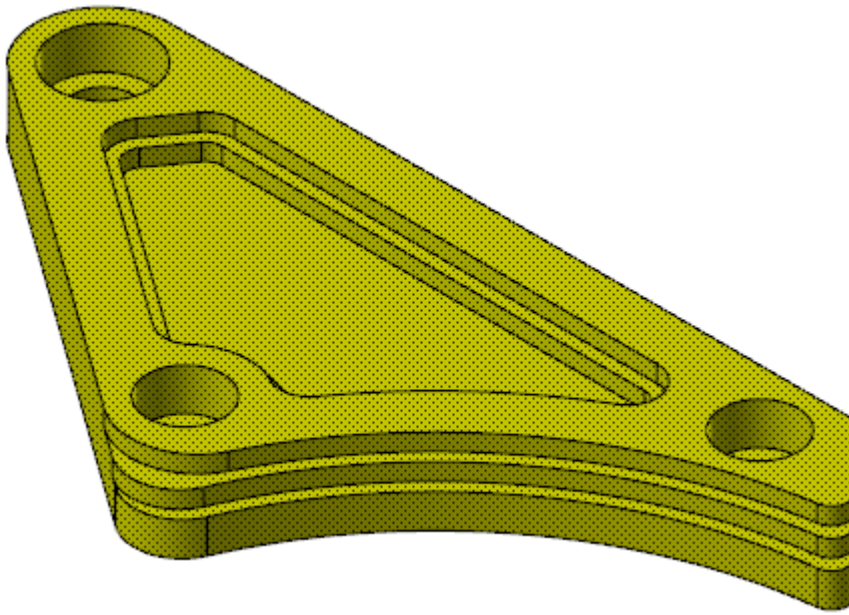
The **Machine Group Properties** dialog box displays.

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

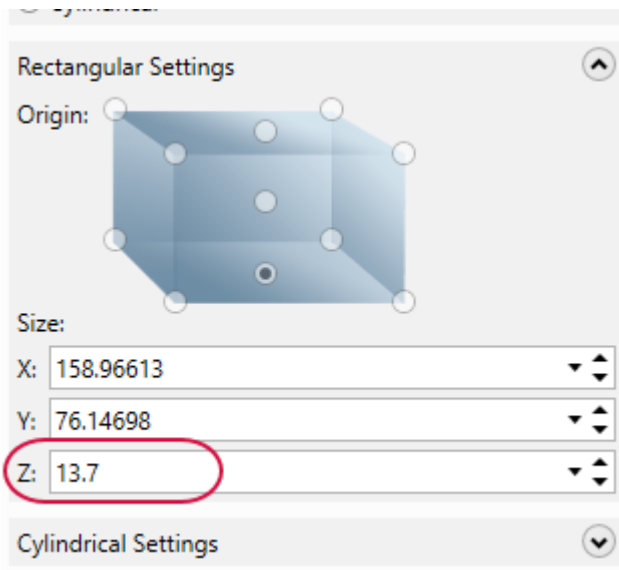


The **Bounding Box** function panel displays.

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



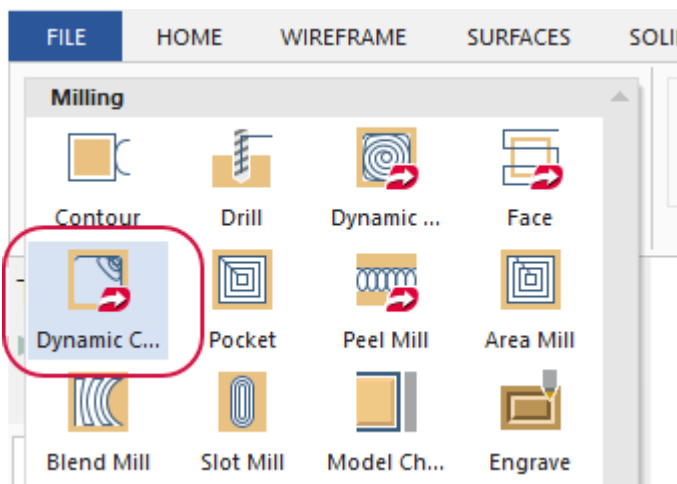
7. Set **Z** to **13.7**. This adds a small amount of stock to the Z height of the stock boundary.



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

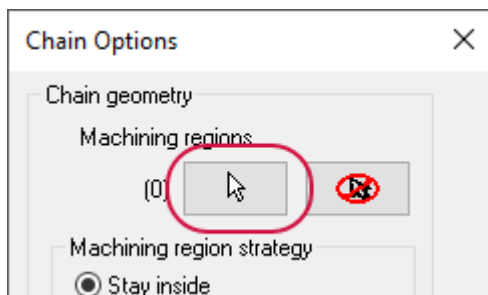
## Exercise 2: Creating the first Dynamic Contour toolpath

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



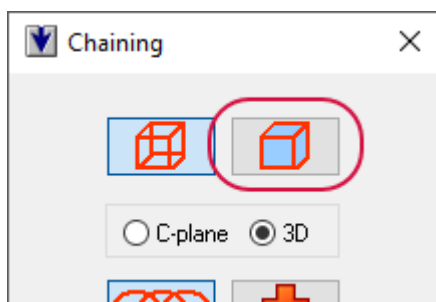
The **Chain Options** dialog box displays.

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



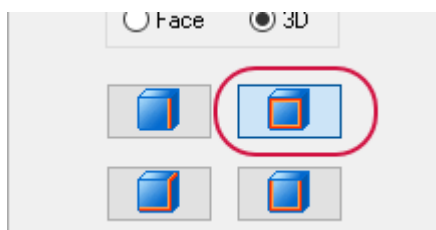
The Chaining dialog box displays.

3. Select **Solids** to activate solid chaining mode.



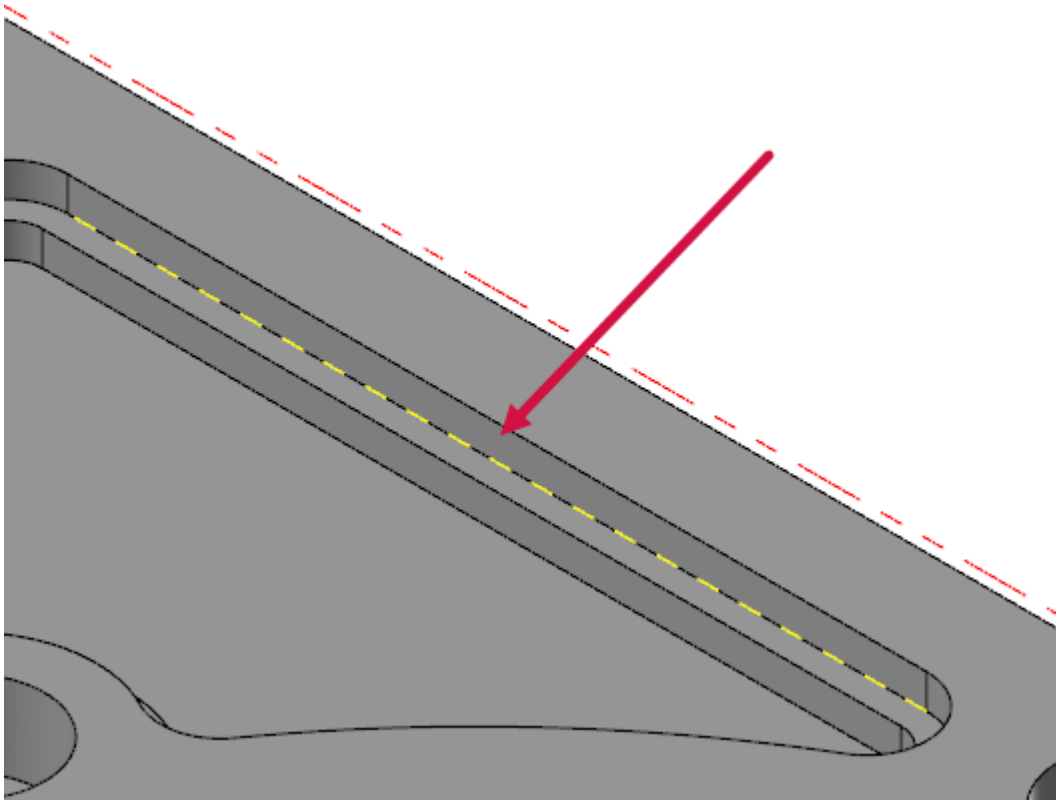
In Solid chaining mode, the **Chaining** dialog box gives you options to manage the chaining of solid entities.

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



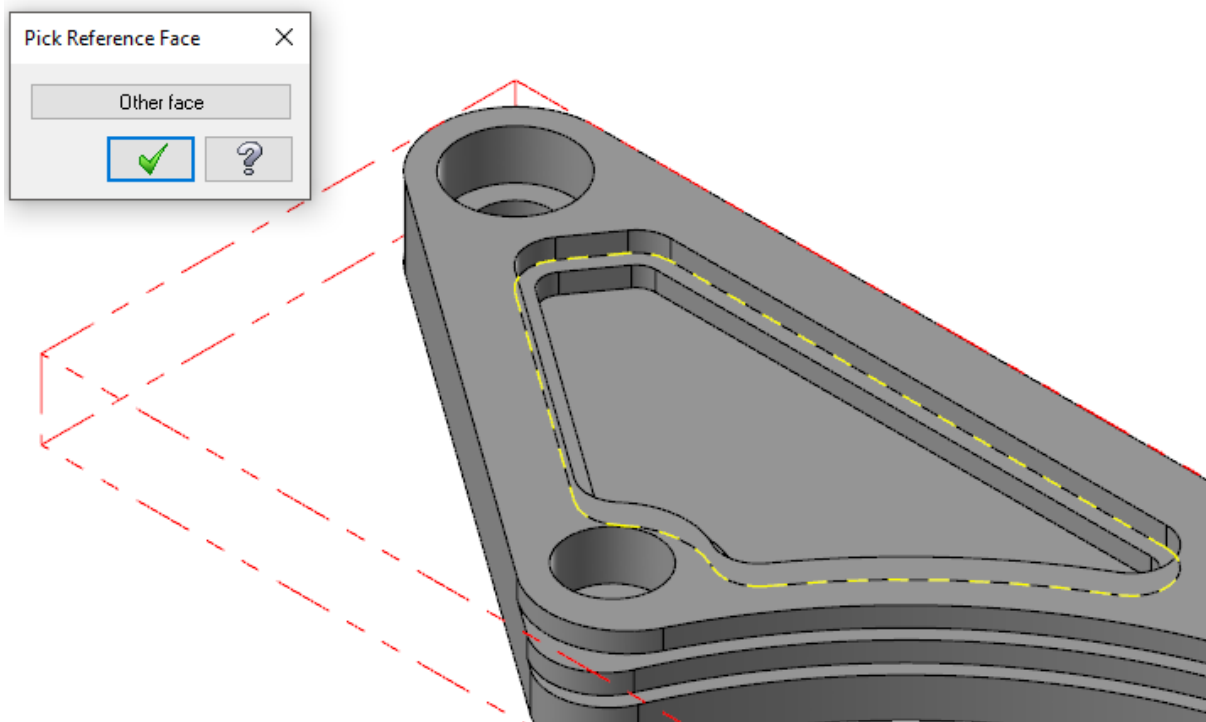
This allows you to select a closed chain on a reference face.

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



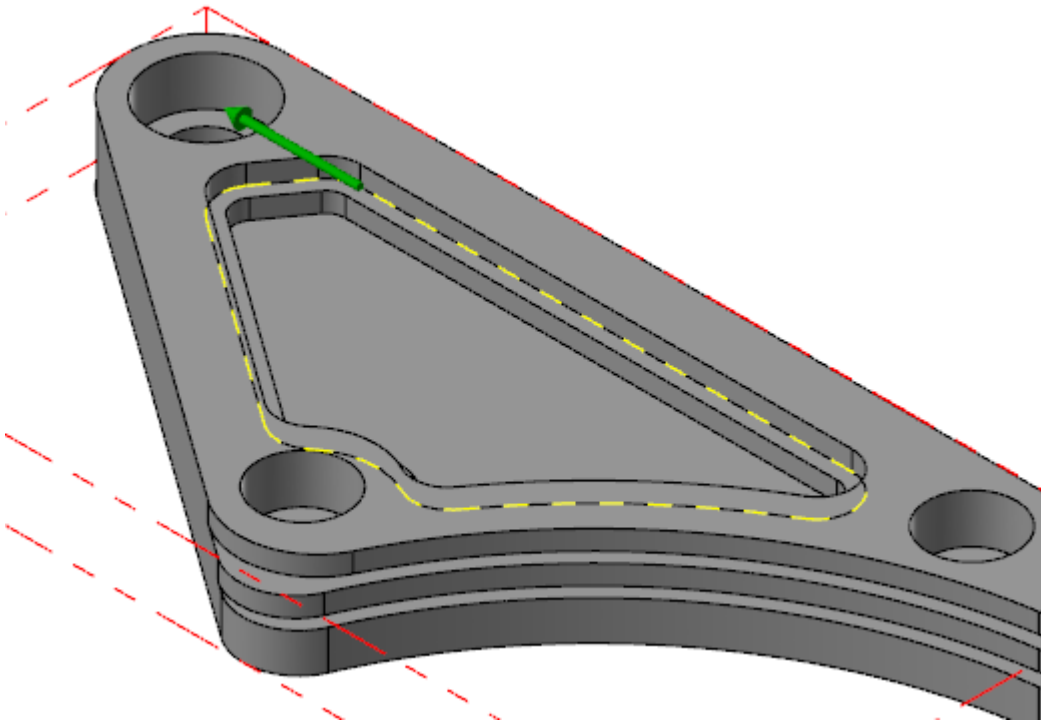
The **Pick Reference Face** dialog box displays. This dialog box allows you to cycle through possible reference faces on a solid until you reach the one you want.

6. Ensure that the face shown below is selected:



Select **Other face** if necessary to cycle through the possible reference faces.

7. Click **OK** to return to the **Chaining** dialog box. The chain will be displayed as shown below:

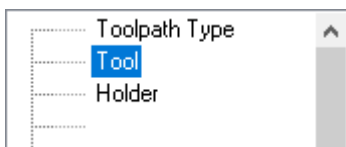


Reverse the chain if necessary.

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

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

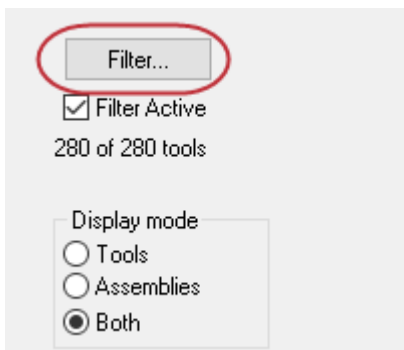
10. Select the **Tool** page.



11. Click **Select library tool**.

The **Tool Selection** dialog box displays.

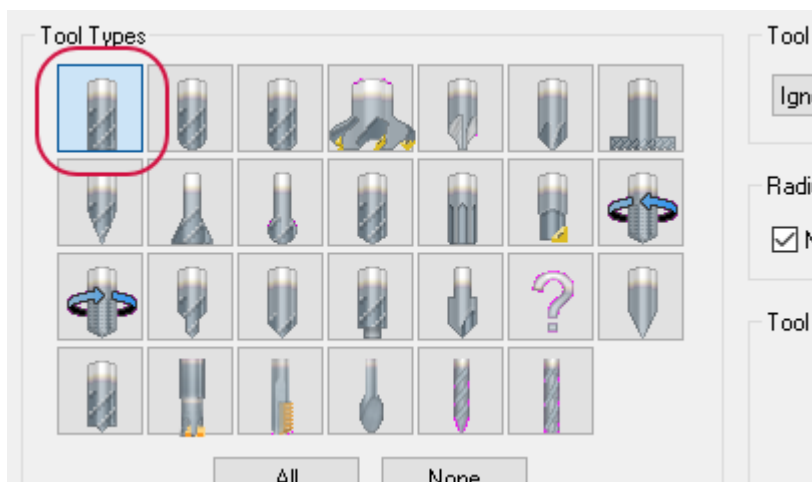
12. Click **Filter**.



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

13. Set the **Tool Type** filter to **None**.

14. Select the **Endmill1 Flat** filter, and deselect any other filter. This ensures that you are only selecting flat end mill tools.

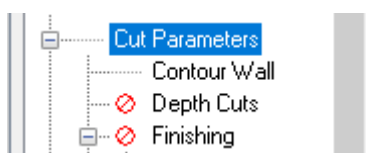


15. Click **OK** in the **Tool List Filter** dialog box.
16. In the **Tool Selection** dialog box, select the **FLAT END MILL - 4** tool.

C:\Users\Public\Docu... \Mill\_mm.tooldb

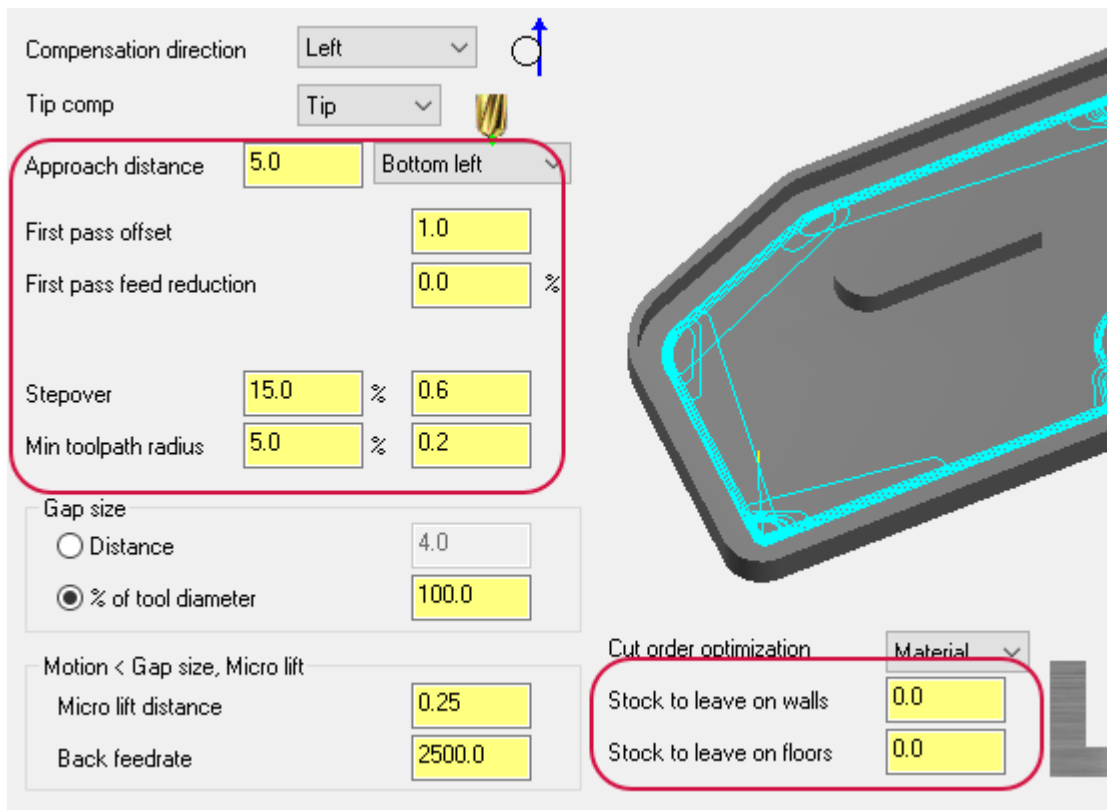
#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ty
5	FLAT END MILL - 18	--	18.0	0.0	29.0	4	Fla
5	FLAT END MILL - 20	--	20.0	0.0	32.0	4	Fla
5	FLAT END MILL - 3	--	3.0	0.0	5.0	4	Fla
5	FLAT END MILL - 4	--	4.0	0.0	7.0	4	Fla
5	FLAT END MILL - 5	--	5.0	0.0	8.0	4	Fla
5	FLAT END MILL - 6	--	6.0	0.0	10.0	4	Fla
5	FLAT END MILL - 8	--	8.0	0.0	12.0	4	Fla

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





19. Set the following parameters:



Compensation direction: Left

Tip comp: Tip

Approach distance: 5.0 Bottom left

First pass offset: 1.0

First pass feed reduction: 0.0 %

Stepover: 15.0 % 0.6

Min toolpath radius: 5.0 % 0.2

Gap size:

☐ Distance 4.0

☒ % of tool diameter 100.0

Motion < Gap size, Micro lift

Micro lift distance: 0.25

Back feedrate: 2500.0

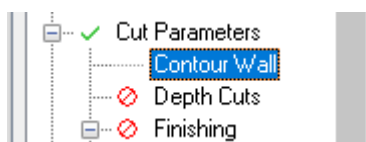
Cut order optimization: Material

Stock to leave on walls: 0.0

Stock to leave on floors: 0.0

- **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 any stock on the vertical and horizontal surfaces.

20. Select the **Contour Wall** page.



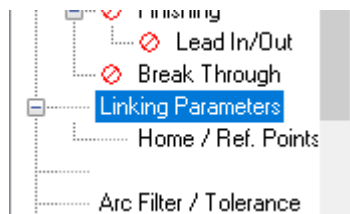
21. Set the following parameters:

Radius of tool that shaped the stock	8.0
Min toolpath radius that shaped the stock	1.0
Stock thickness	0.025

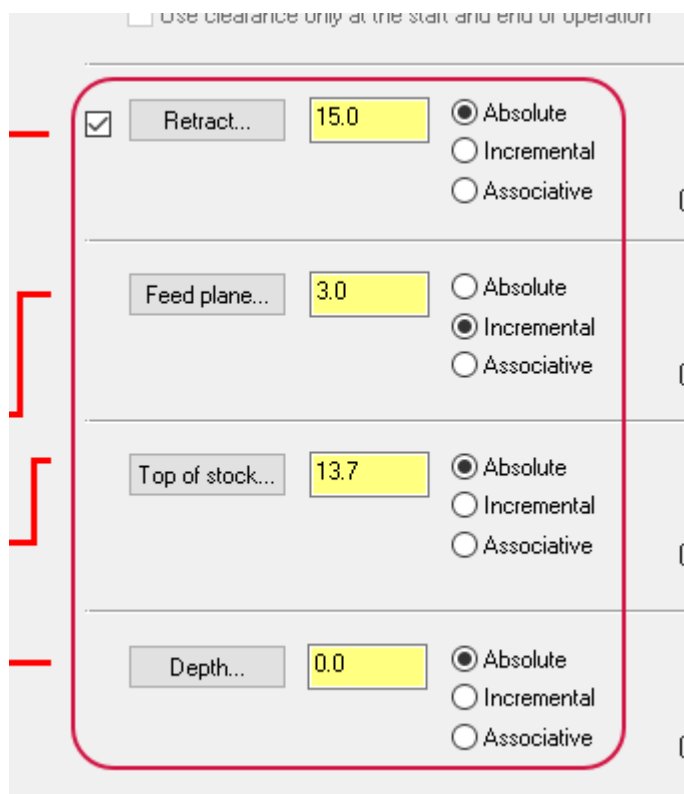
- **Radius of tool that shaped the stock to 8.0.**
- **Minimum toolpath radius that shaped the stock to 1.0.**
- **Stock thickness to 0.025.**

Both 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.

22. Select the **Linking Parameters** page.

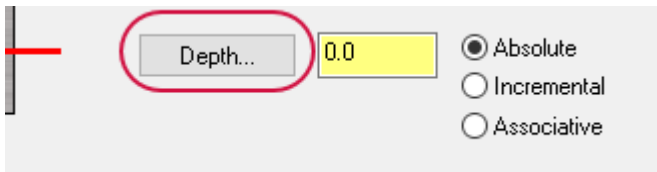


23. Set the following parameters:



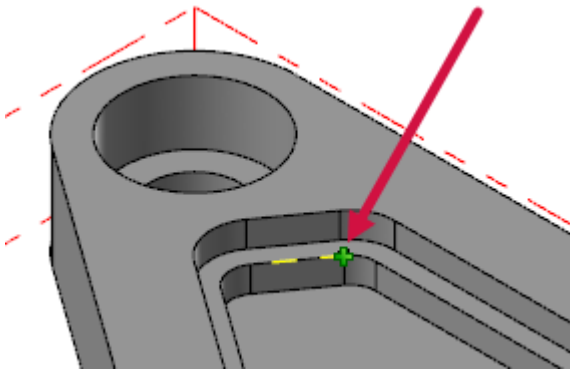
- **Retract, Top of stock, and Depth to Absolute.** This measures absolute values from the origin 0,0,0.
- **Retract to 15.0.** Sets the height that the tool moves up before the next tool pass.
- **Feed plane to 3.0.** Sets the height that the tool rapids to before changing to the plunge rate to enter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

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



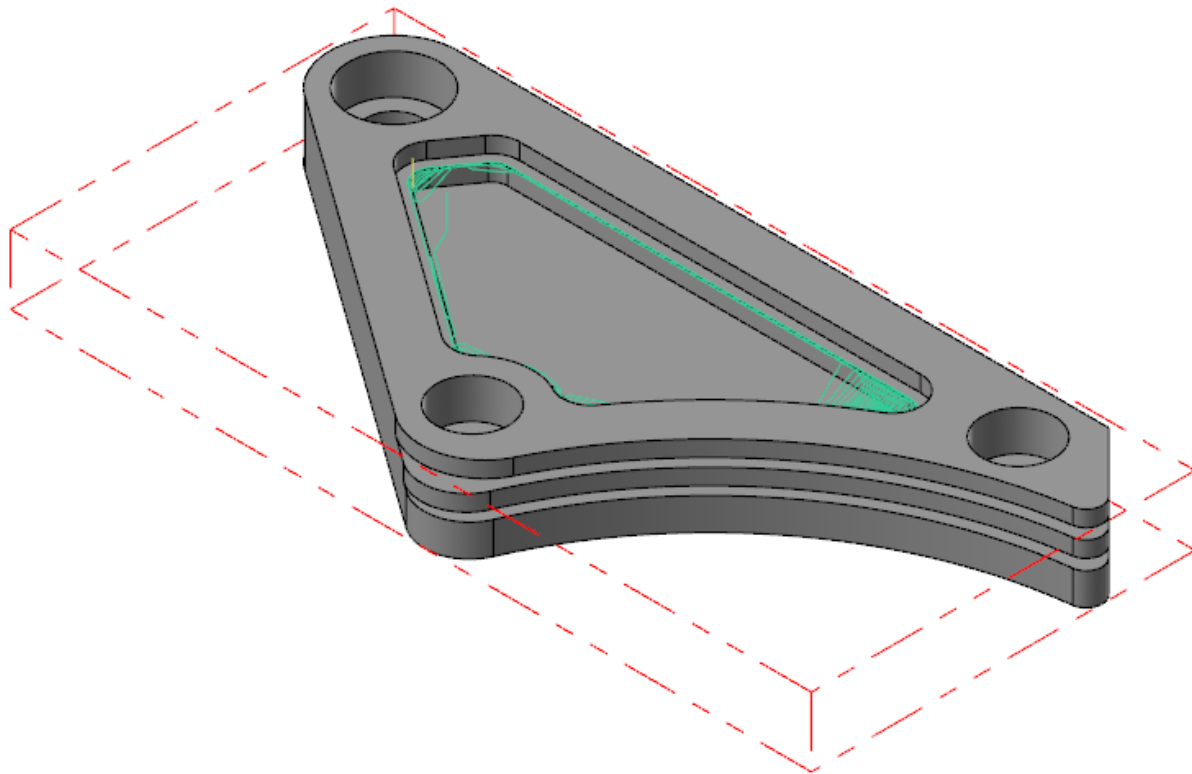
Depth determines the final machining depth and the lowest depth that the tool descends into stock. You will select an edge for this toolpath.

25. Select the edge shown below:



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

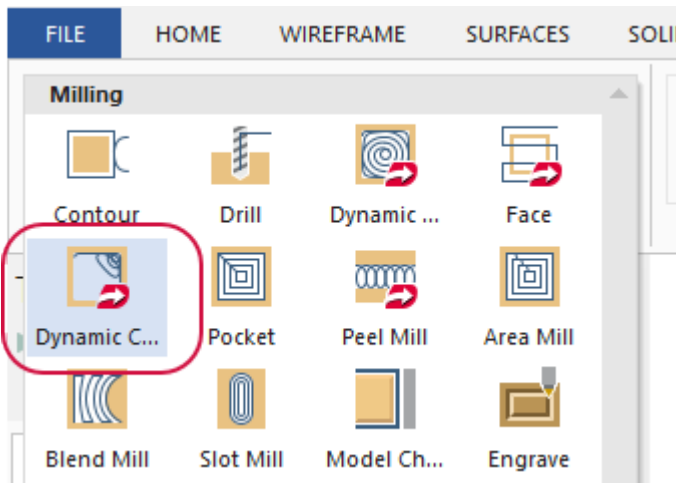
26. Click **OK** in the **2D High Speed Toolpath - Dynamic Contour** dialog box to accept and generate your toolpath.
27. Your toolpath displays as shown below:



28. Save your part.

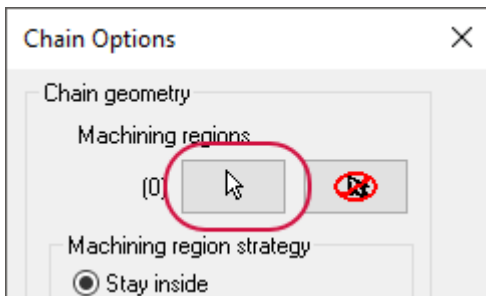
## Exercise 3: Creating the second Dynamic Contour toolpath

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



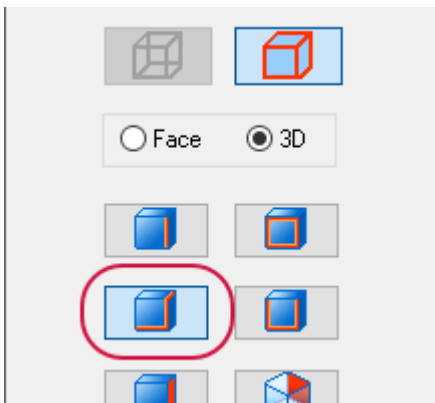
The **Chain Options** dialog box displays.

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



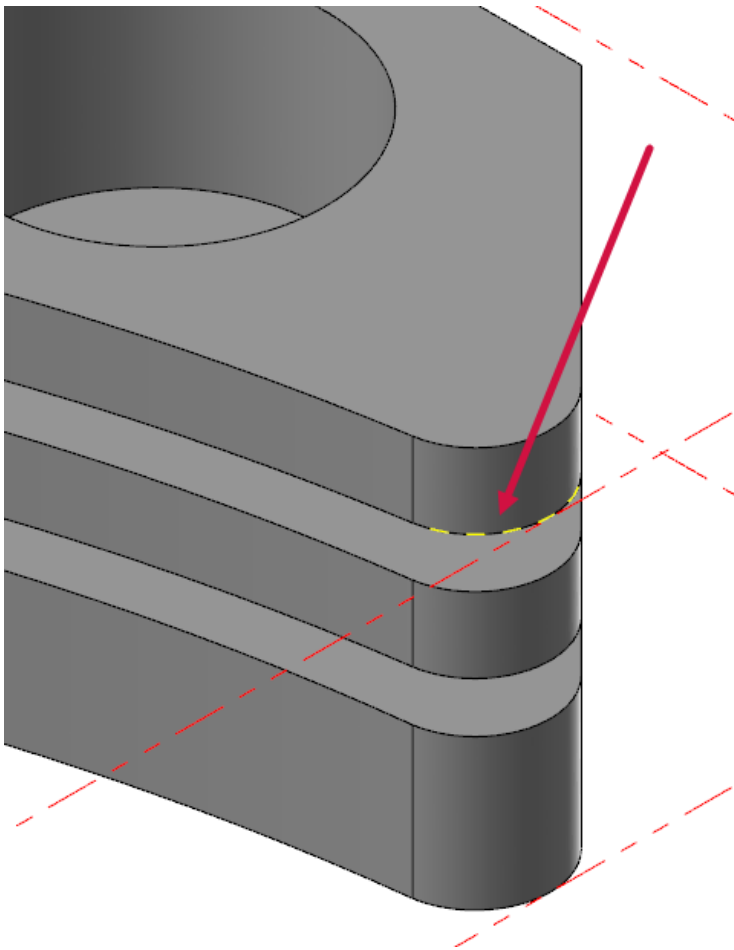
The **Chaining** dialog box displays.

3. Activate **Solids** selection if necessary.
4. Set the chaining to **Linked edges**.

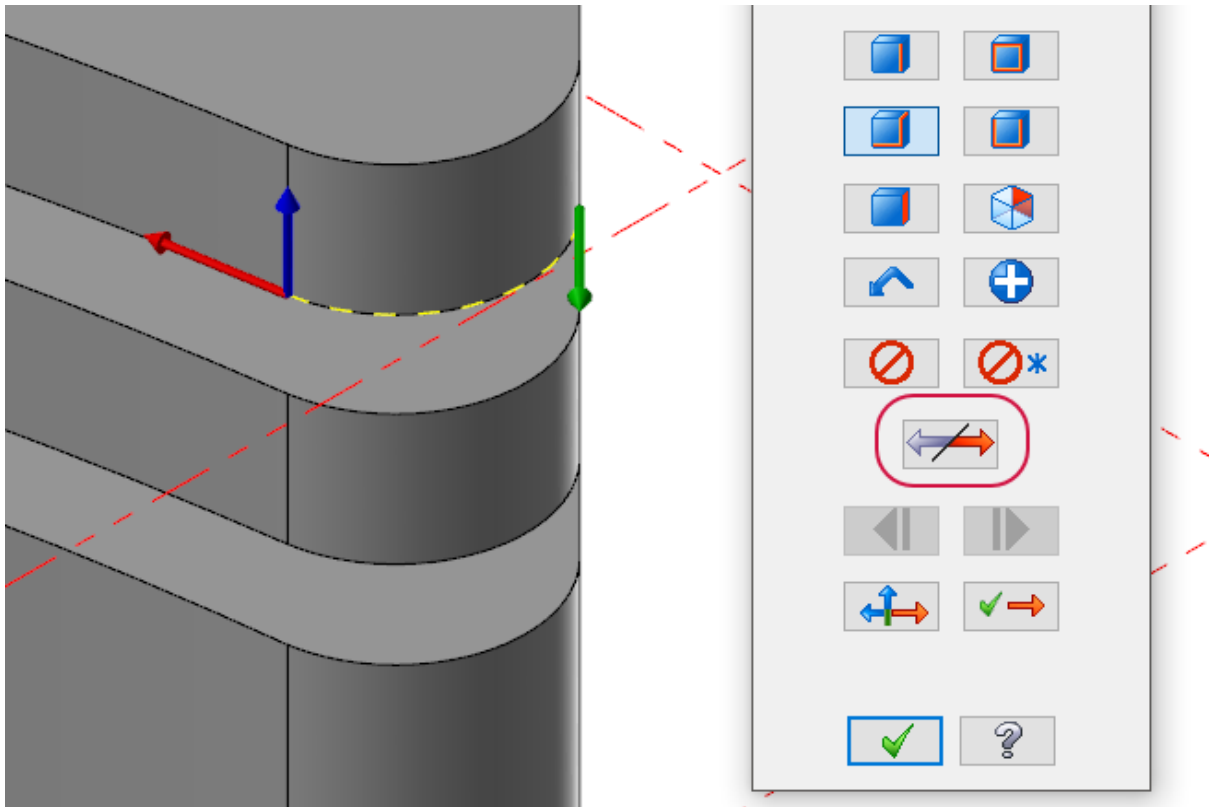


This allows you to select contiguous chaining of edges.

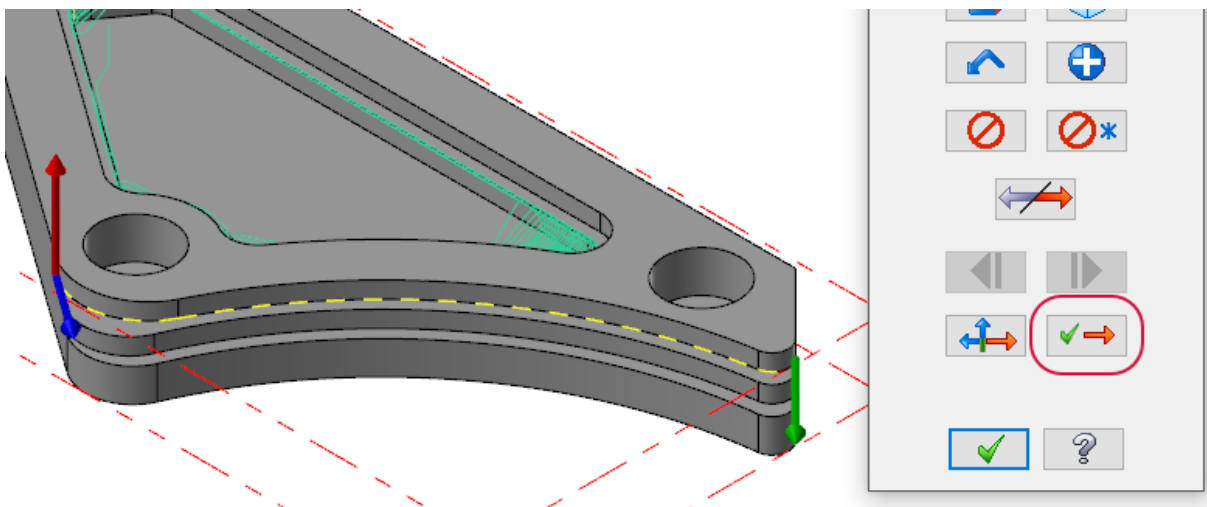
5. Zoom in using the mouse wheel and select the edge shown below.



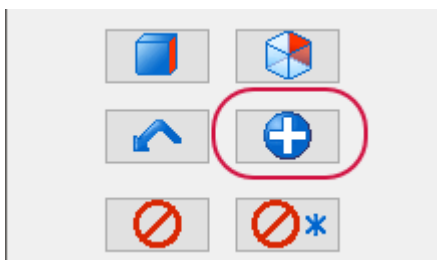
6. The chain will display as shown below. Reverse the direction if needed, using **Reverse** on the **Chaining** dialog box.



7. Use the **Next** control in the **Chaining** dialog box to select the chain shown below.

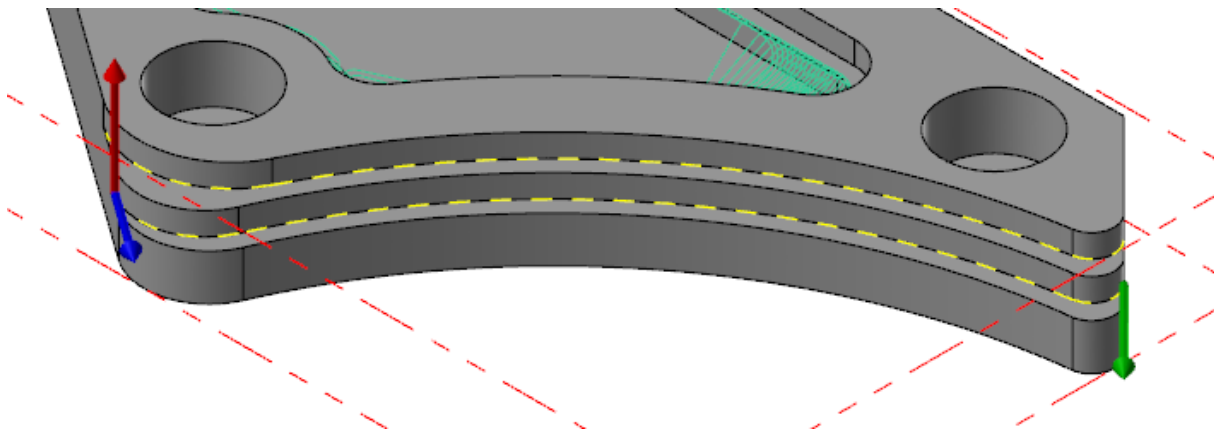


8. In the **Chaining** dialog box, click **End chain**.



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

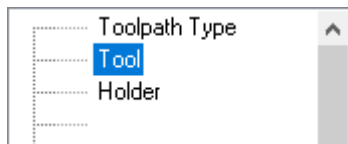
9. Using the controls in the **Chaining** dialog box, select the second partial chain shown below:



10. Click **OK** in the **Chaining** dialog box and return to the **Chain Options** dialog box.  
 11. Click **OK** in the **Chain Options** dialog box.

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

12. Select the **Tool** page.



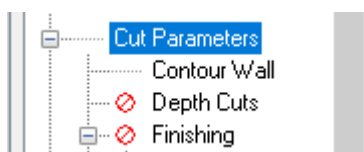
13. Click **Select library tool**.

The **Tool Selection** dialog box displays.

14. Select the **FLAT END MILL - 8** tool.

#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. ...
5	FLAT END MILL - 16	--	16.0	0.0	26.0	4	Flat e...	None
5	FLAT END MILL - 18	--	18.0	0.0	29.0	4	Flat e...	None
5	FLAT END MILL - 20	--	20.0	0.0	32.0	4	Flat e...	None
5	FLAT END MILL - 3	--	3.0	0.0	5.0	4	Flat e...	None
5	FLAT END MILL - 4	--	4.0	0.0	7.0	4	Flat e...	None
5	FLAT END MILL - 5	--	5.0	0.0	8.0	4	Flat e...	None
5	FLAT END MILL - 6	--	6.0	0.0	10.0	4	Flat e...	None
5	FLAT END MILL - 8	--	8.0	0.0	13.0	4	Flat e...	None
9	SHOULDER MILL - 100	--	100.0	0.0	15.0	8	Flat e...	None
9	SHOULDER MILL - 125	--	125.0	0.0	15.0	10	Flat e...	None
9	SHOULDER MILL - 160	--	160.0	0.0	15.0	12	Flat e...	None
9	SHOULDER MILL - 200	--	200.0	0.0	15.0	12	Flat e...	None

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



17. Set the following parameters:

Compensation direction: Left

Tip comp: Tip

Approach distance: 12.0

Bottom left

First pass offset: 3.0

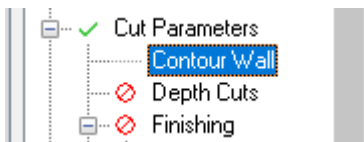
First pass feed reduction: 0.0 %

Stepover: 15.0 %

Min toolpath radius: 5.0 %

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

18. Select the **Contour Wall** page.



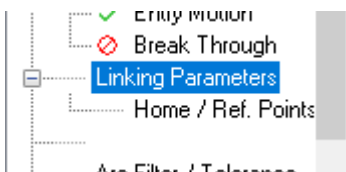
19. Set **Radius of tool that shaped the stock** to 10.0.

Radius of tool that shaped the stock: 12.0

Min toolpath radius that shaped the stock: 1.0

Stock thickness: 0.025

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





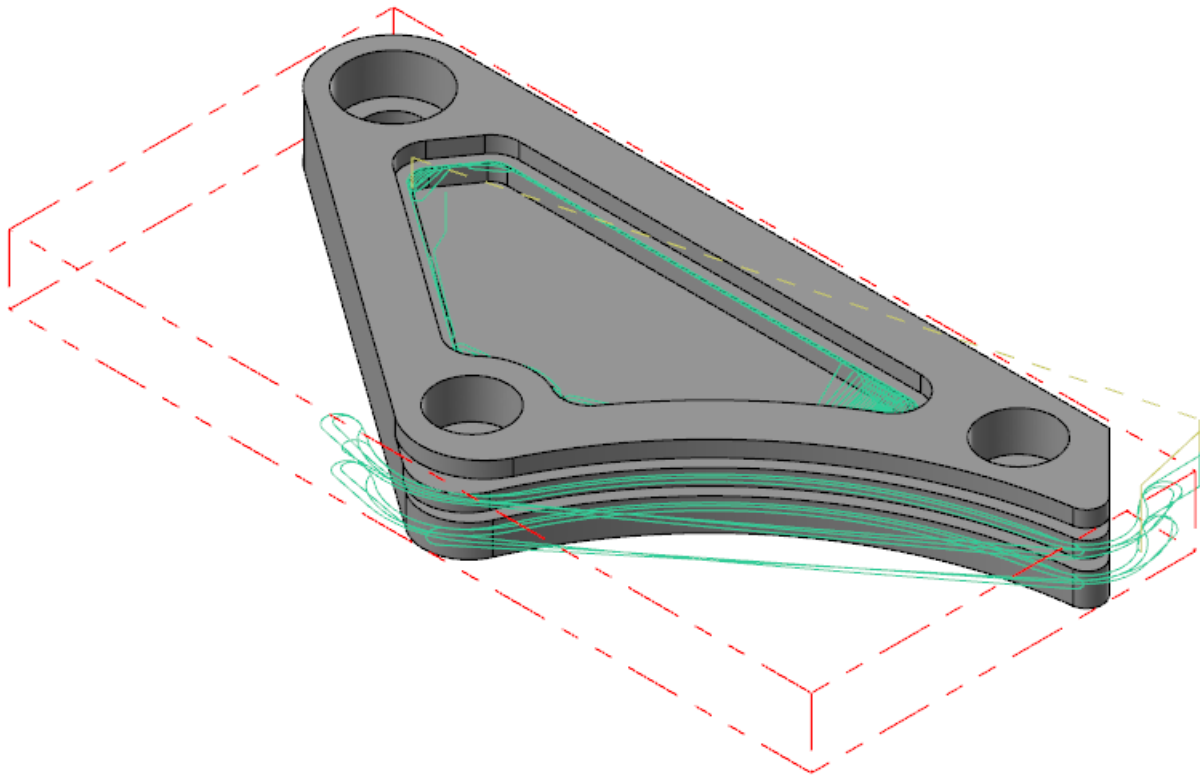
21. Set the following parameters:

<input checked="" type="checkbox"/>	Retract...	16.0	<input checked="" type="radio"/> Absolute <input type="radio"/> Incremental <input type="radio"/> Associative	(0)
	Feed plane...	3.0	<input type="radio"/> Absolute <input checked="" type="radio"/> Incremental <input type="radio"/> Associative	(0)
	Top of stock...	9.525	<input checked="" type="radio"/> Absolute <input type="radio"/> Incremental <input type="radio"/> Associative	(0)
	Depth...	0.0	<input type="radio"/> Absolute <input checked="" type="radio"/> Incremental <input type="radio"/> Associative	(0)

- **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.**

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

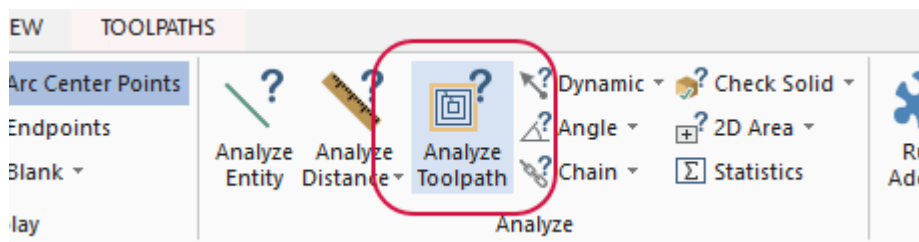
The toolpath will display as shown below:



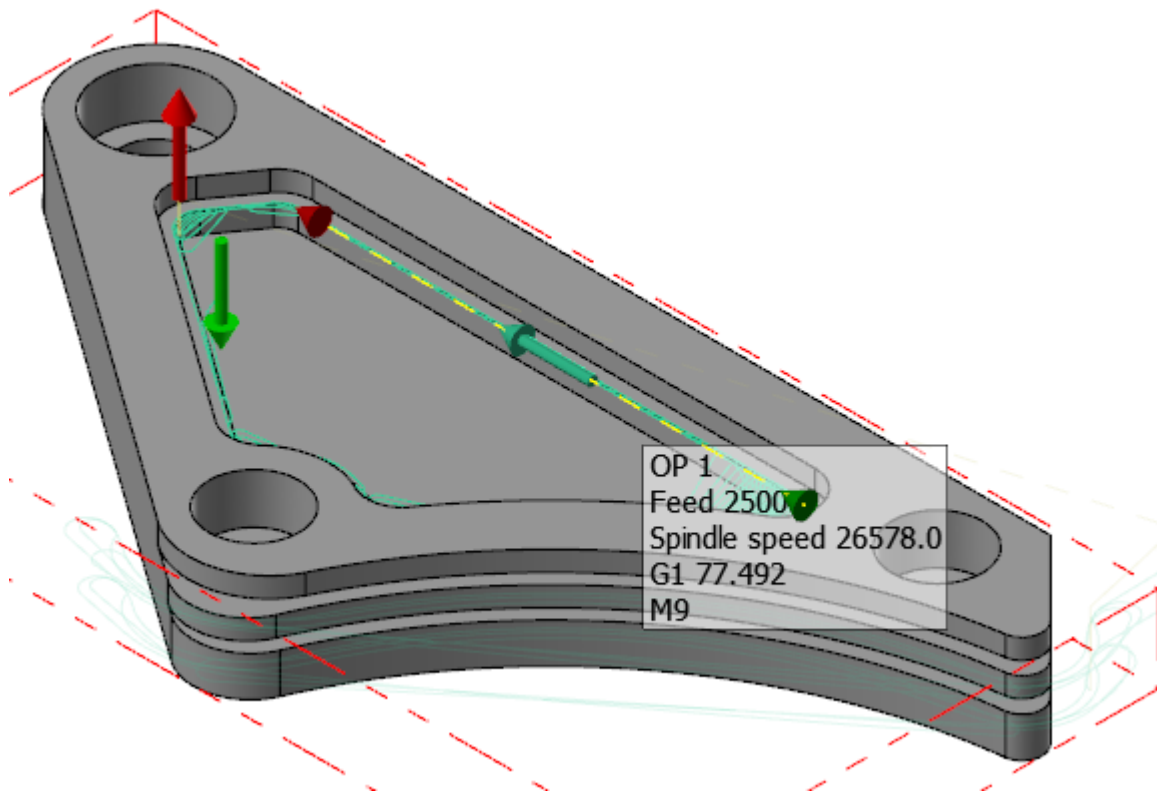
## Exercise 4: Analyzing the toolpaths

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:



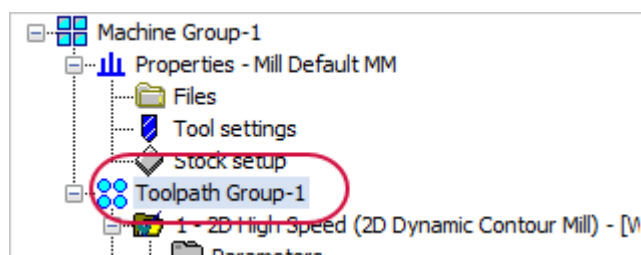
Analyze Toolpath informs you of the operation number, the feed move, spindle speed, G1 and the line length, and the coolant code. It also displays the start and end of the toolpath and of the section you are hovering over. If there is more than one toolpath displayed at the time, then toolpaths other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

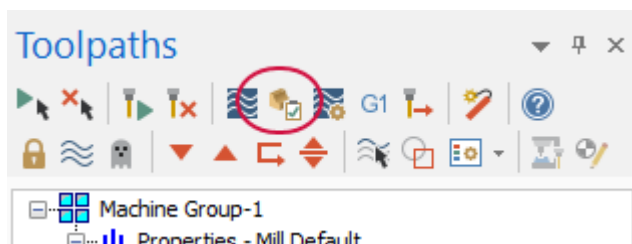
You will now verify both toolpaths.

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



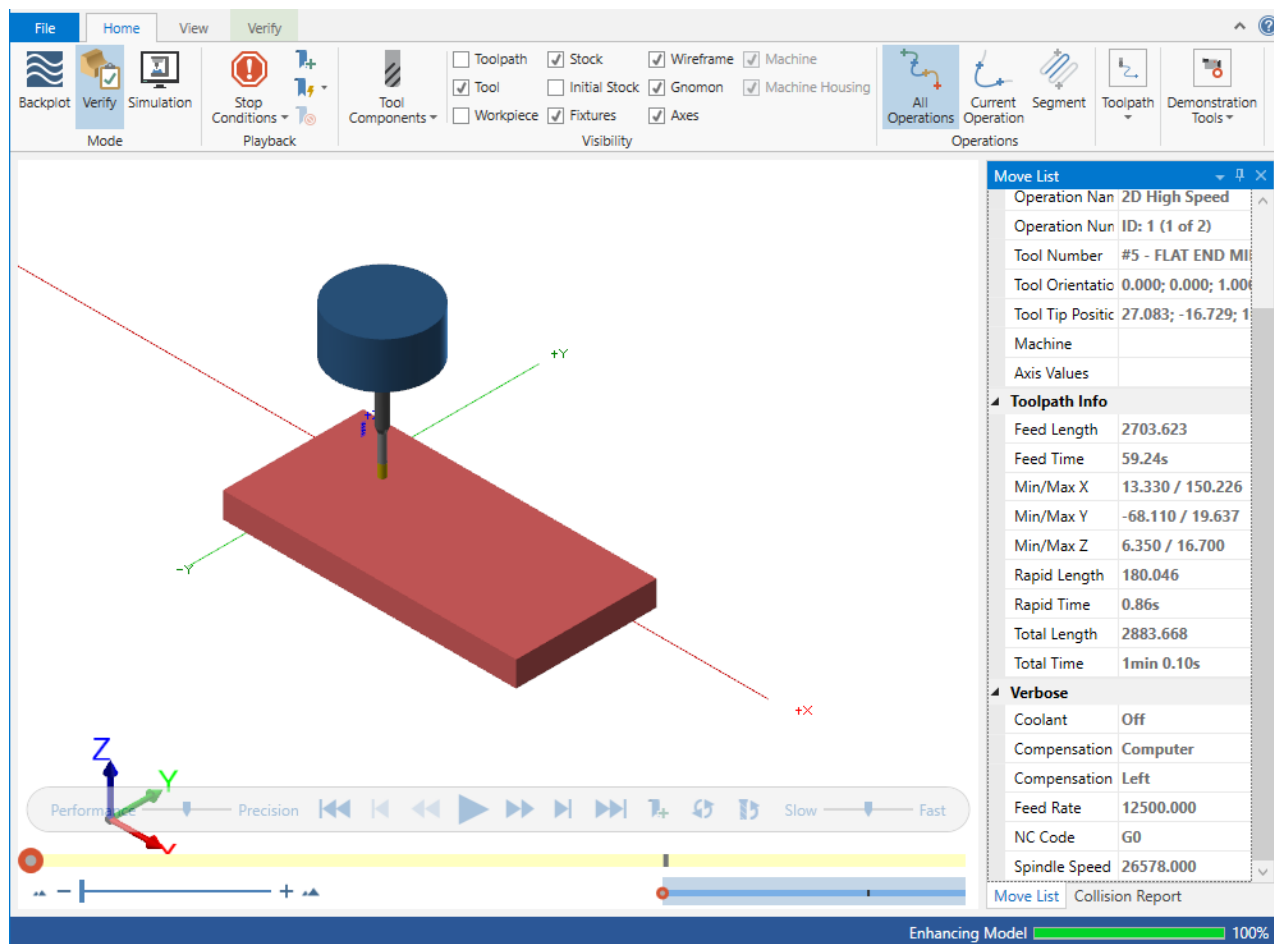
This selects both toolpaths so that they can be verified at the same time.

2. Select **Verify selected operations**.



Mastercam Simulator displays.

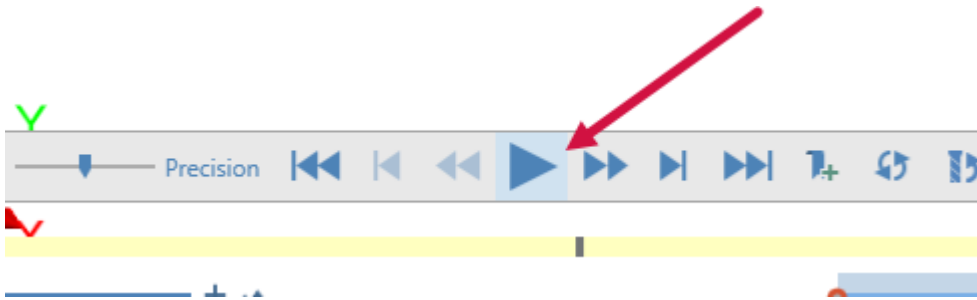
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:



4. View the Move List, which displays all pertinent information about the toolpaths being verified. By using two Dynamic Contour toolpaths, the total machining time is approximately one minute.

Min/Max Z	6.350 / 16.700
Rapid Length	180.046
Rapid Time	0.86s
Total Length	2883.668
Total Time	1min 0.10s
<b>Verbose</b>	
Coolant	Off
Compensation	Computer

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



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

In the next lesson you will 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.

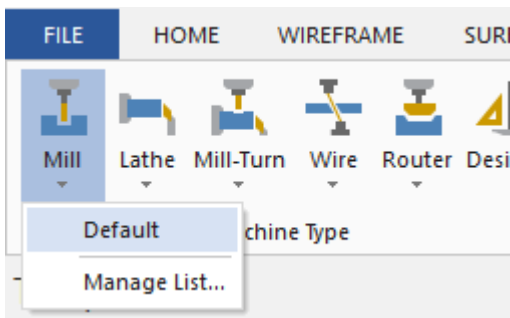
In this lesson you will create one Peel Mill toolpath, copy it twice, and make edits.

### Lesson Goals

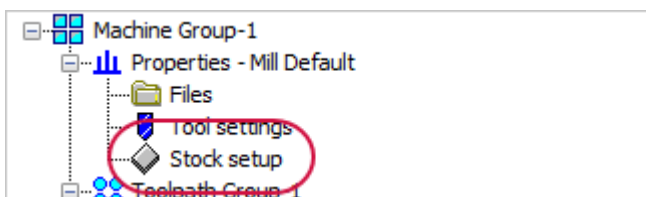
- Create a Peel Mill toolpath
- Copy and paste the toolpath
- Make edits to the copied toolpaths
- Verify the created toolpaths

### Exercise 1: Setting up the stock

1. Open the part file, *PeelMill1*, provided with this tutorial.
2. Save the part as *PeelMill1-xxx*, replacing *xxx* with your initials.
3. On the **Machine** tab, select **Mill, Default**.

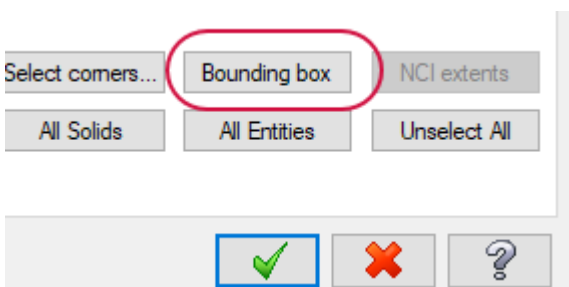


4. Select **Stock setup** from the Toolpaths Manager.



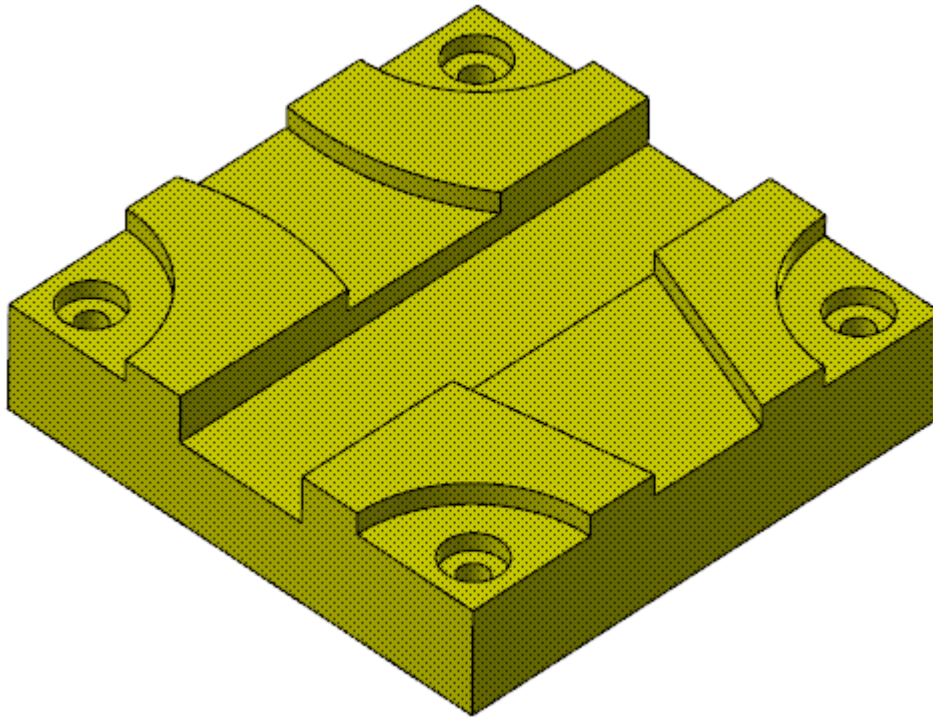
The **Machine Group Properties** dialog box displays.

5. Select **Bounding box**.



The **Bounding Box** function panel displays.

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

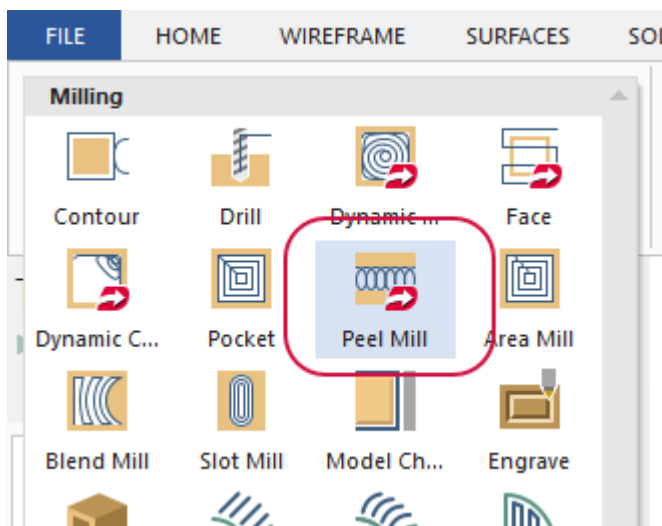


For this part, no geometry needs to be created. The bounding box will be accepted as it is.

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

## Exercise 2: Creating the first Peel Mill toolpath

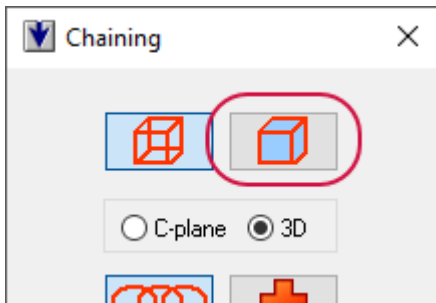
1. Select **Peel Mill** from the **2D** gallery on the **Mill Toolpaths** contextual tab.



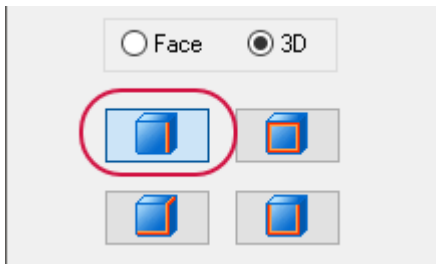
The **Chaining** dialog box displays.



2. Activate **Solids** selection from the Chaining dialog box.

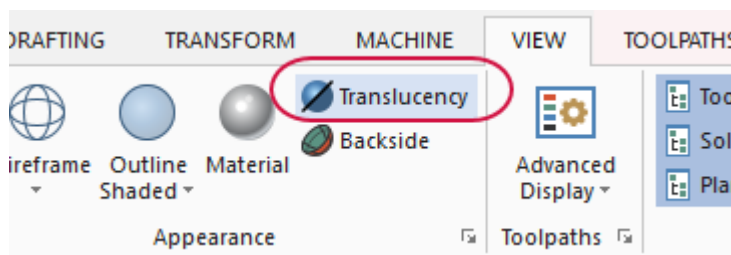


3. Select **Edge** if it has not already been selected. Deselect **Face** if it is selected.



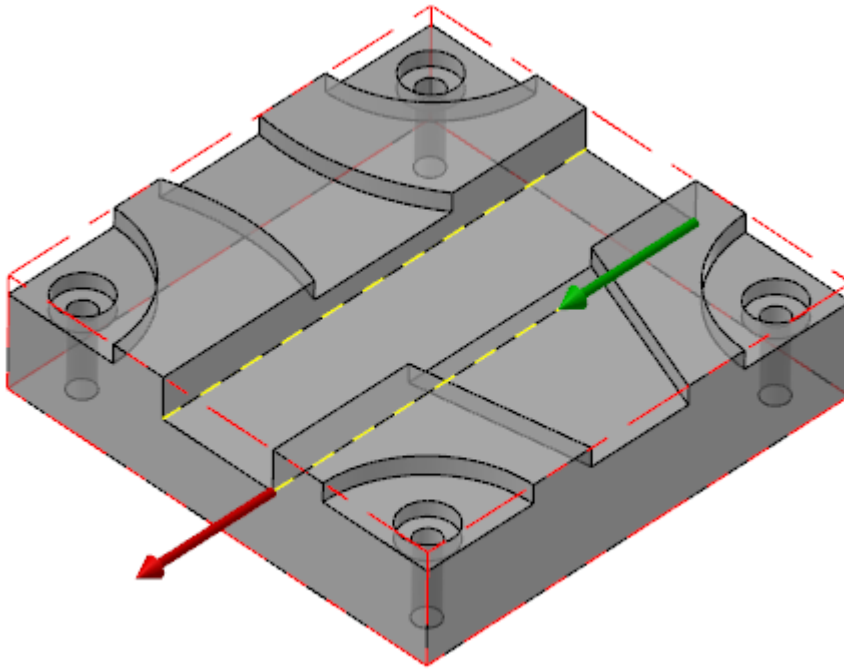
This allows you to select the solid edges of a model.

4. Select **Translucency** on the **View** tab.



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

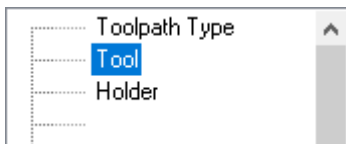
5. Select the two edges shown below. Use **Reverse** if necessary to ensure both chains go in the same direction.



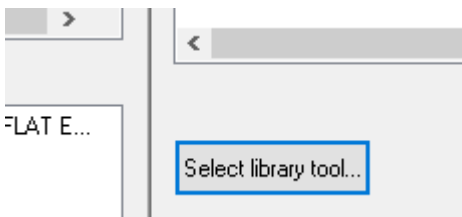
6. Toggle **Translucency** to off.
7. Click **OK** in the **Chaining** dialog box.

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

8. Select the **Tool** page.

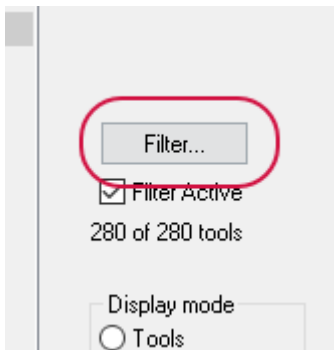


9. Click **Select library tool**.



The **Tool Selection** dialog box displays.

10. Click **Filter**.



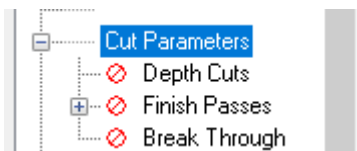
The **Tool List Filter** dialog box displays.

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

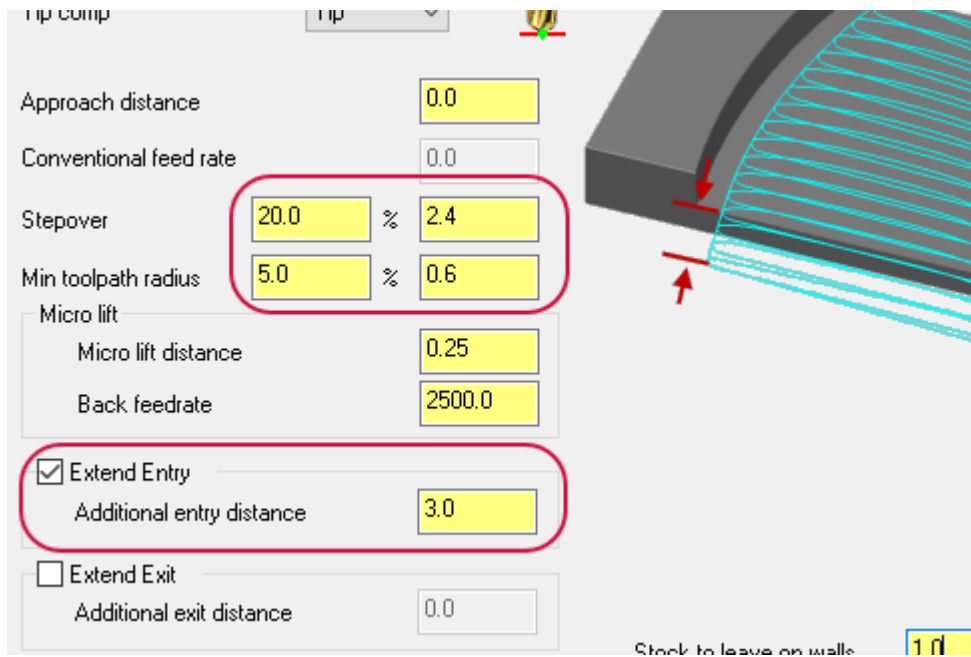
C:\Users\Public\Docu... \Mill\_mm.tooldb

#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. ...
5	FLAT END MILL - 10	--	10.0	0.0	16.0	4	Flat e...	None
5	FLAT END MILL - 12	--	12.0	0.0	19.0	4	Flat e...	None
5	FLAT END MILL - 14	--	14.0	0.0	22.0	4	Flat e...	None
5	FLAT END MILL - 16	--	16.0	0.0	26.0	4	Flat e...	None
5	FLAT END MILL - 18	--	18.0	0.0	29.0	4	Flat e...	None
5	FLAT END MILL - 20	--	20.0	0.0	32.0	4	Flat e...	None
5	FLAT END MILL - 3	--	3.0	0.0	5.0	4	Flat e...	None
5	FLAT END MILL - 4	--	4.0	0.0	7.0	4	Flat e...	None
5	FLAT END MILL - 5	--	5.0	0.0	8.0	4	Flat e...	None

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

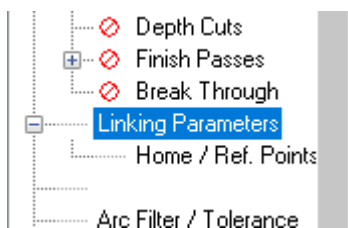


16. Set the following parameters:

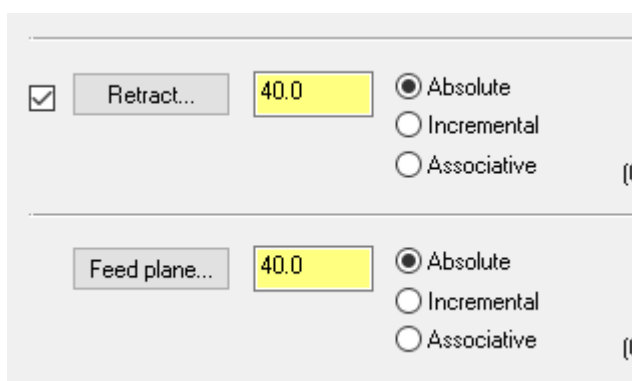


- **Stepover** to 20.0%.
- **Minimum toolpath radius** to 5.0%.
- Select **Extend Entry** and set **Additional entry distance** to 3.0. This adjusts the final tool engagement with the material.

17. Select the **Linking Parameters** page.

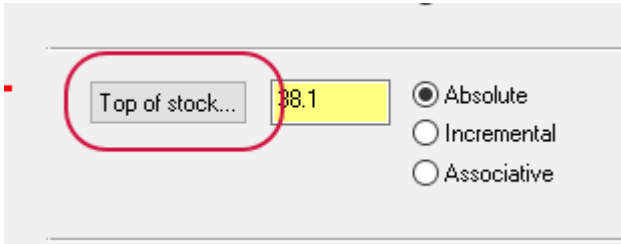


18. Set the following parameters:

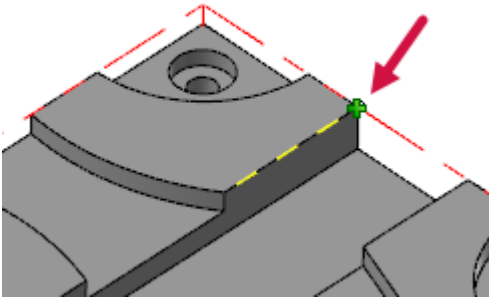


- Set **Retract** to 40.0.
- Set **Feed plane** to 40.0.

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



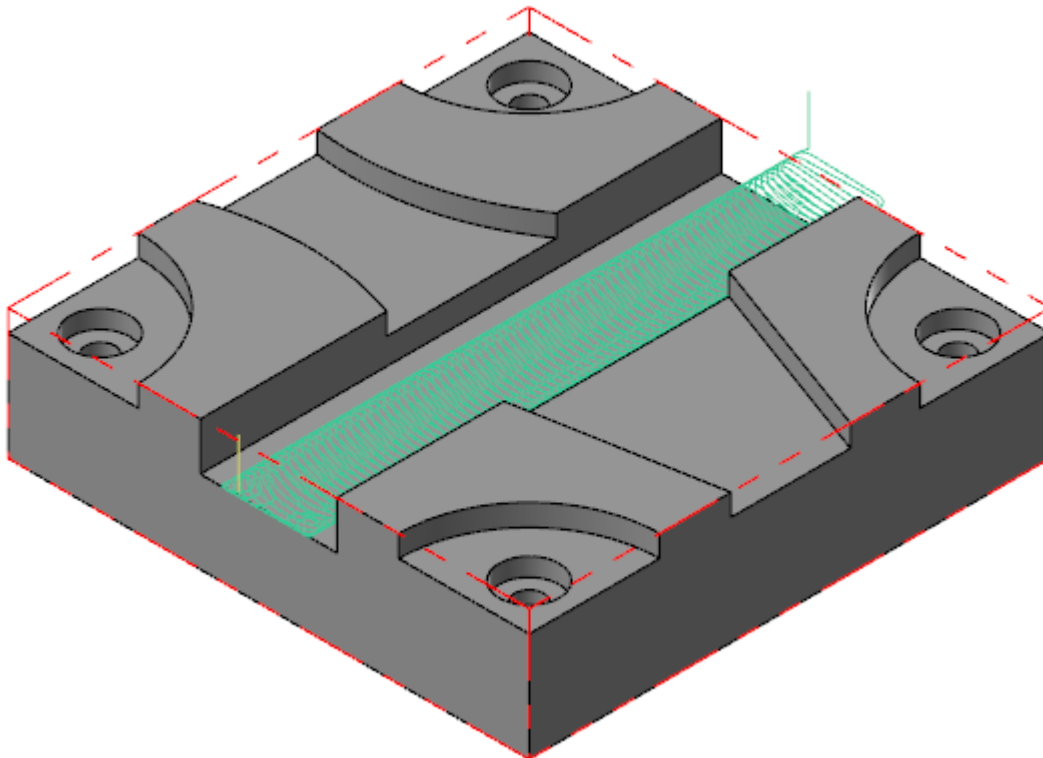
20. Select the point shown below:



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

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

The toolpath will display as shown below:

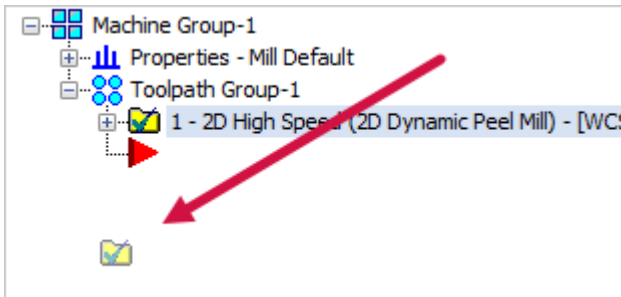


22. Save your file.

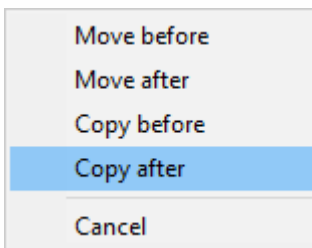
## Exercise 3: Copying the Peel Mill toolpath

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.

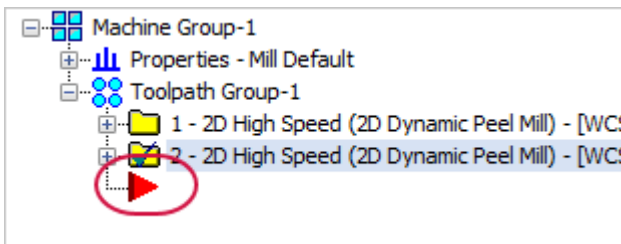


2. Select **Copy after** from the right-click menu that displays.



The toolpath is copied below the original. There are now two Peel Mill toolpaths in the Toolpaths Manager.

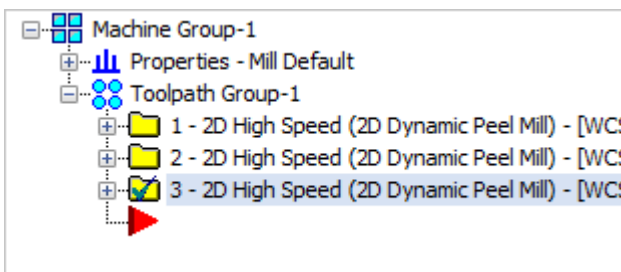
3. Click and drag the red arrow to position it below the copied operation.



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.

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

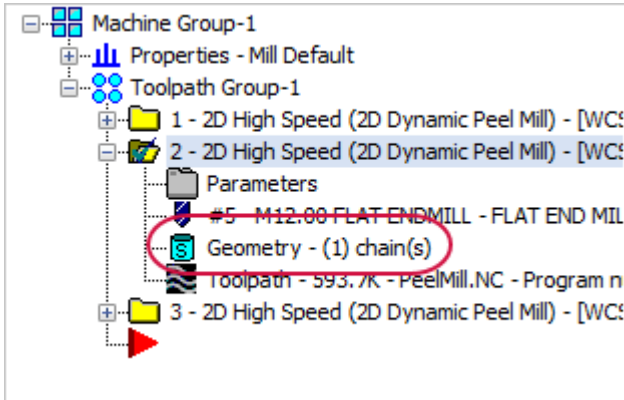


6. Save your file.

## Exercise 4: Re-chaining the copied Peel Mill operations

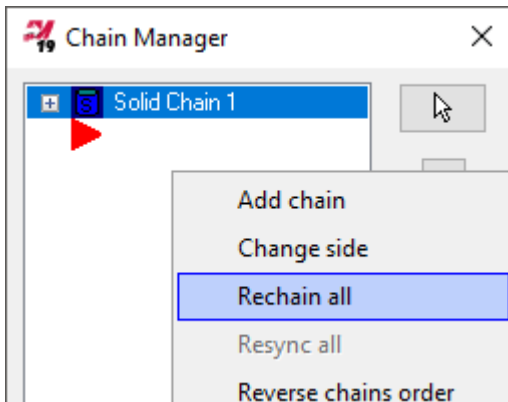
In this exercise, you will re-chain the geometry for the Peel Mill toolpaths that were copied.

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



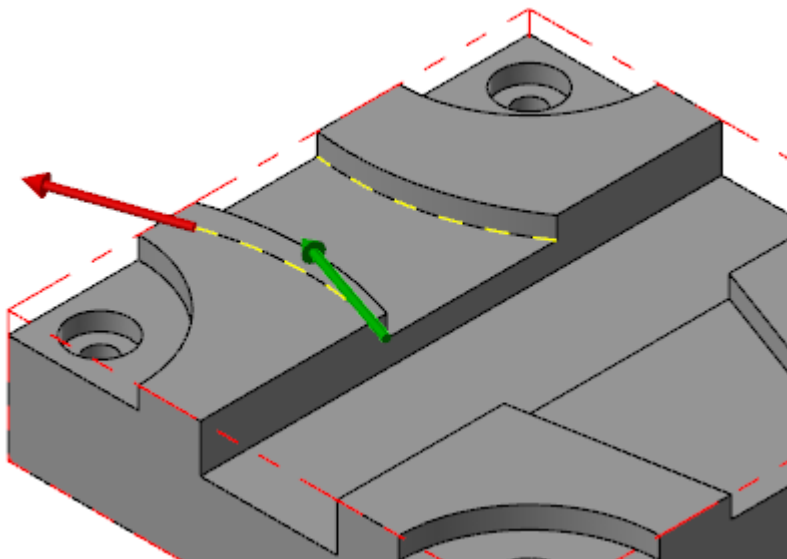
The **Chain Manager** dialog box displays.

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.

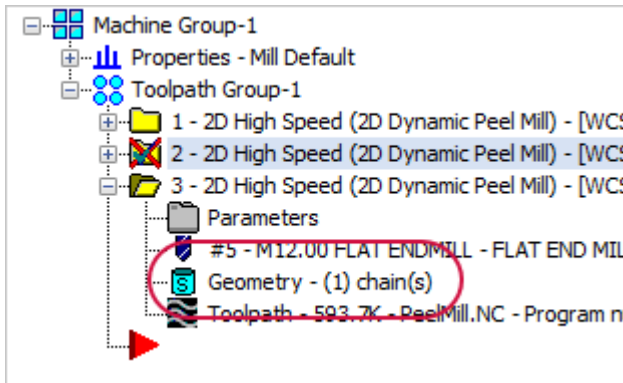


By doing this, you only need to select the chains for the toolpath without having to go through creating the entire toolpath again.

4. Click **OK** in the **Chaining** dialog box and 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.

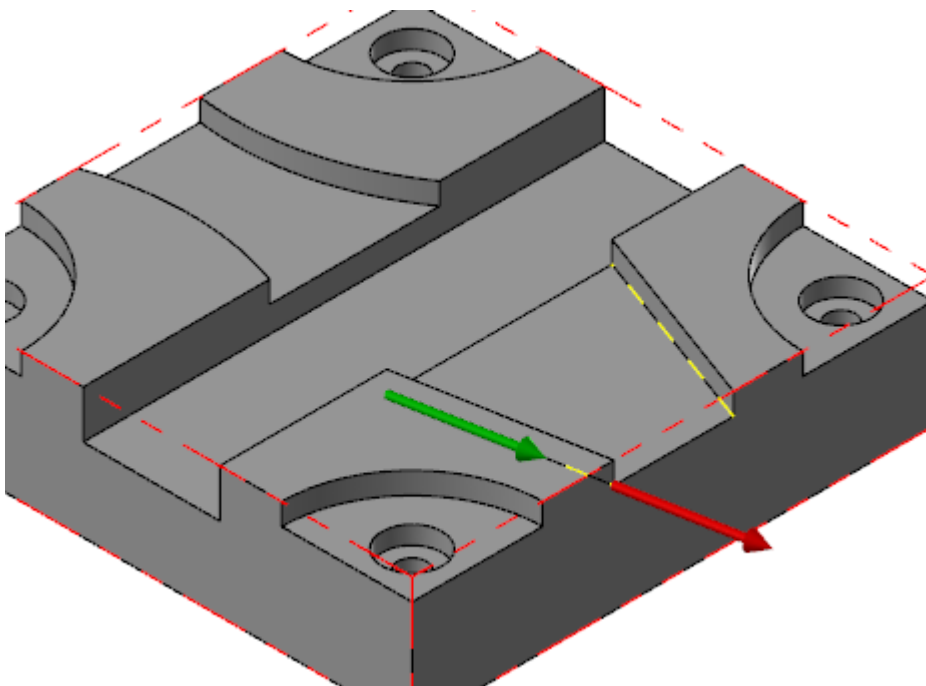


The **Chain Manager** dialog box displays.

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

The **Chaining** dialog box displays.

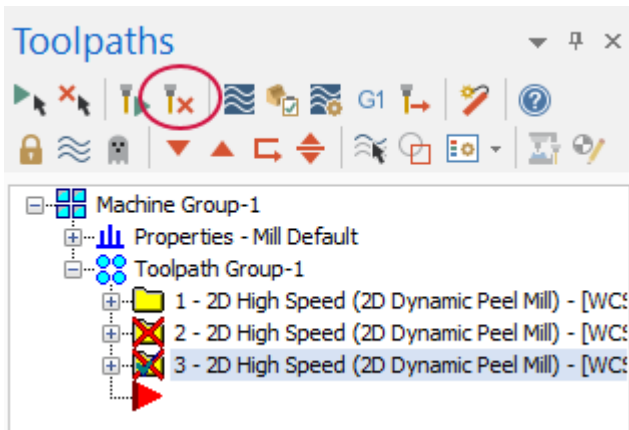
8. Select the chains shown below. Ensure that the chains are both going in the same direction.



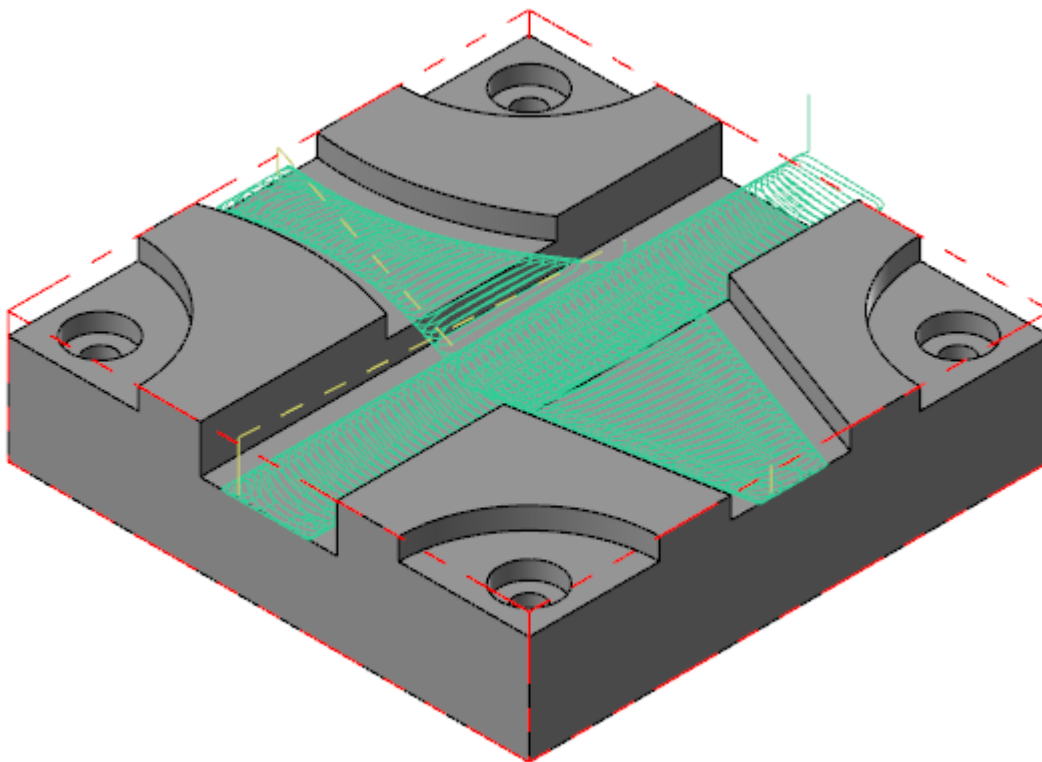
9. Click **OK** in the **Chaining** dialog box and return to the **Chain Manager** dialog box.
10. Click **OK** in the **Chain Manager** dialog box.



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:

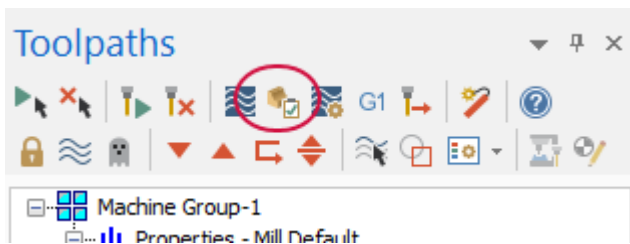


12. Save your file.

## Exercise 5: Verifying the toolpaths

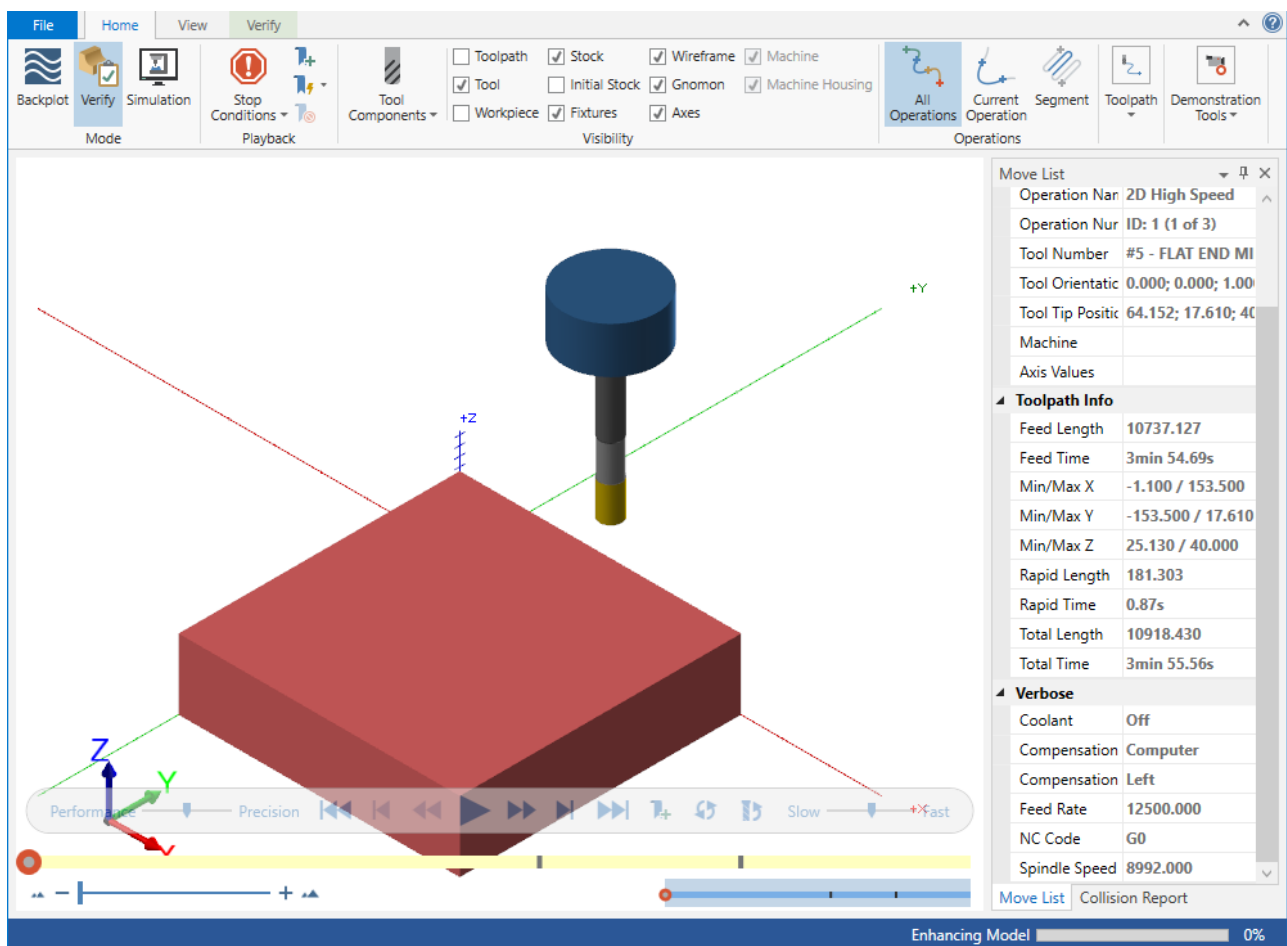
In this exercise, you will verify all three Peel Mill operations together.

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



Mastercam Simulator displays.

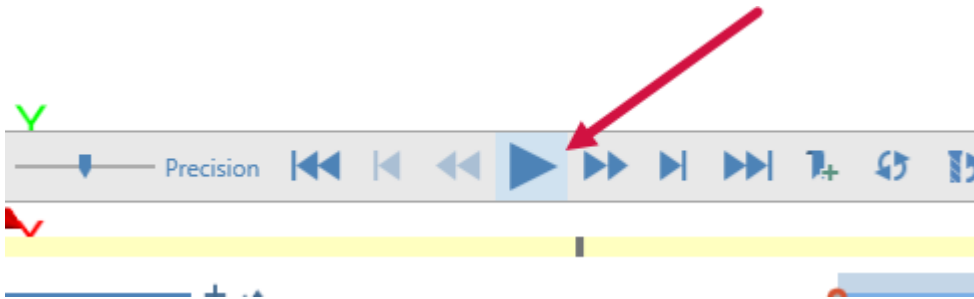
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:



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

Rapid Length	181.303
Rapid Time	0.87s
Total Length	10918.430
Total Time	3min 55.56s
Verbose	
Coolant	Off
Compensation	Computer

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

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

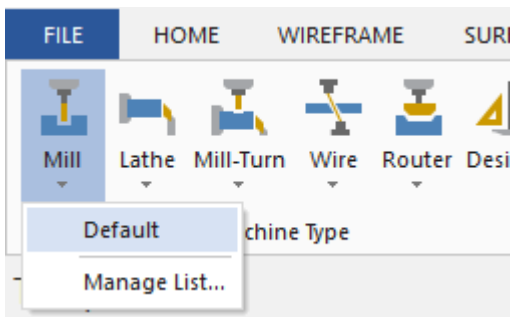
### Lesson Goals

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

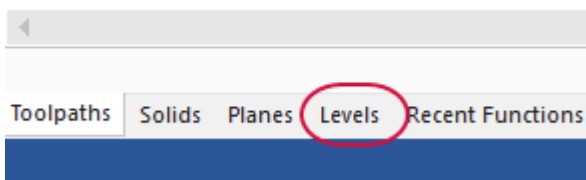
### Exercise 1: Setting up the stock

In this exercise, you will 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**.

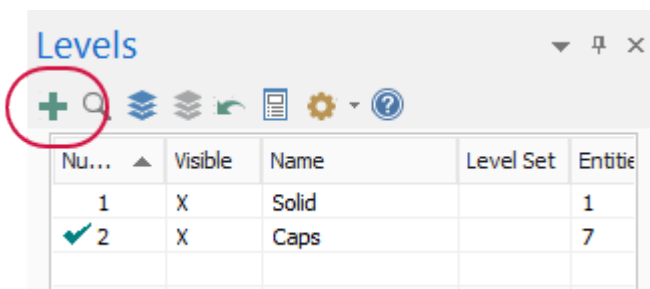


4. Select the Levels Manager, located in the same area as the Toolpaths Manager. If it is not displayed, select **Levels** from the **View** tab to toggle the display.



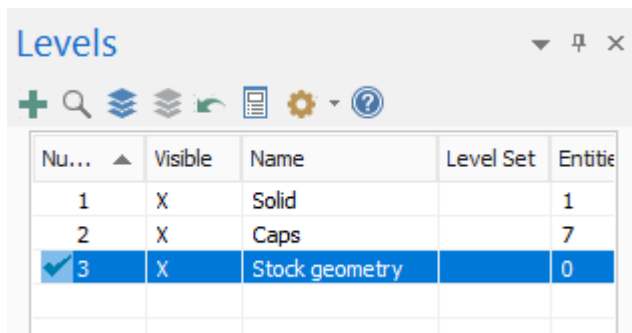
Before creating the stock shape, you will first create a level on which to place the stock geometry. This is a good practice for when you need to organize part files that have thousands of lines, arcs, solids, or surfaces.

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



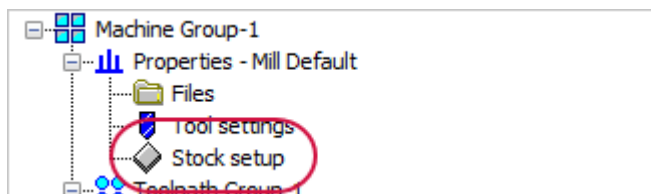
A new level is added to the list and is then set as the main level.

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



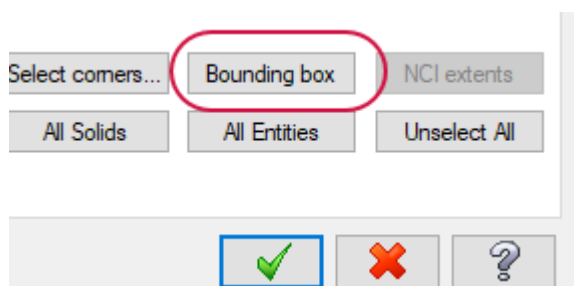
This level will now be used to place the geometry created by the stock setup.

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



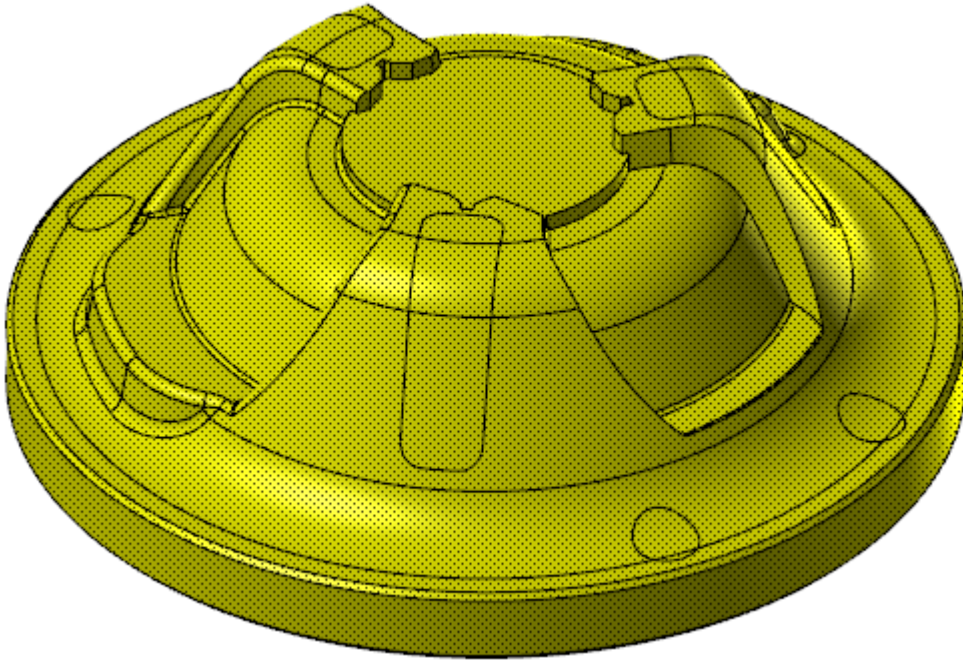
The **Machine Group Properties** dialog box displays.

8. Select **Bounding box**.

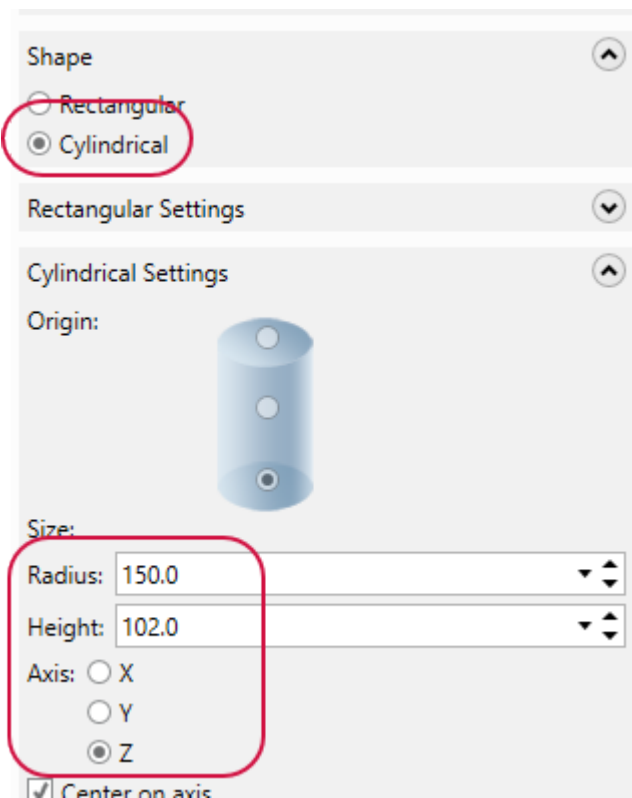


The **Bounding Box** function panel displays.

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



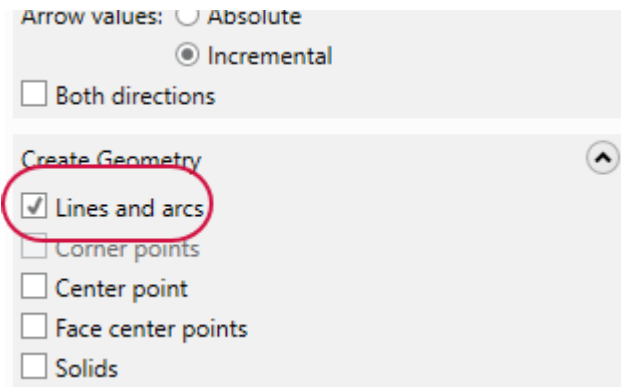
10. Press [Enter] to accept your selection.
11. Set the following parameters in the **Bounding Box** function panel:



- Set **Shape** to **Cylindrical**.
- Set **Radius** to 150.

- Set **Axis** to **Z**.
- Set **Height** to **102.00**. This will add a small amount of stock at the top of the part.

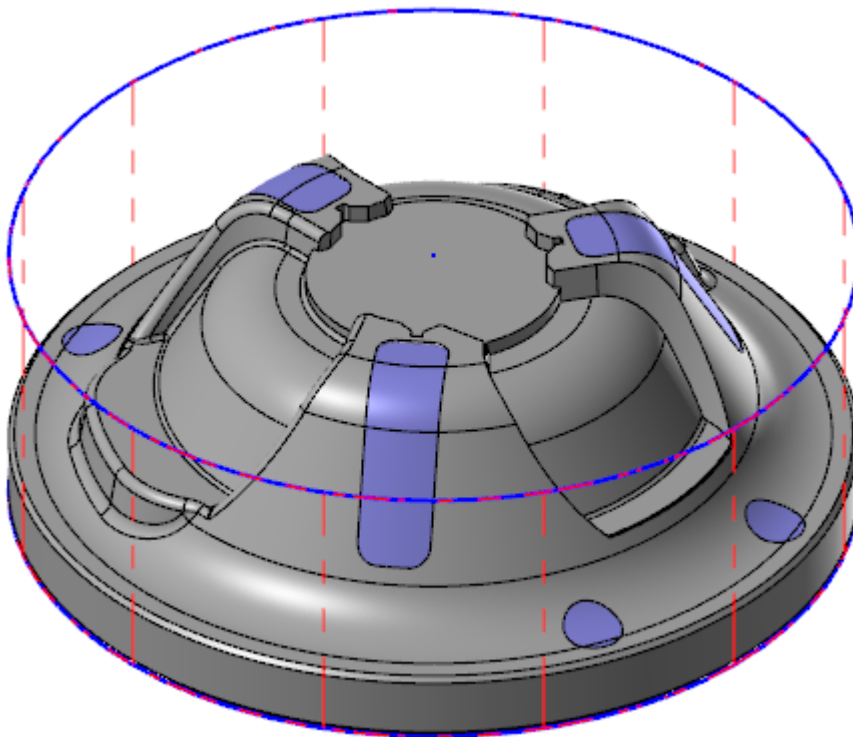
12. In the **Create Geometry** group, select **Lines and arcs**.



This will create lines and arcs based on the stock boundary. You will then use this geometry to contain the first Dynamic OptiRough toolpath.

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

The stock model displays as a dashed outline in red. The geometry that was created with the Bounding Box also displays.

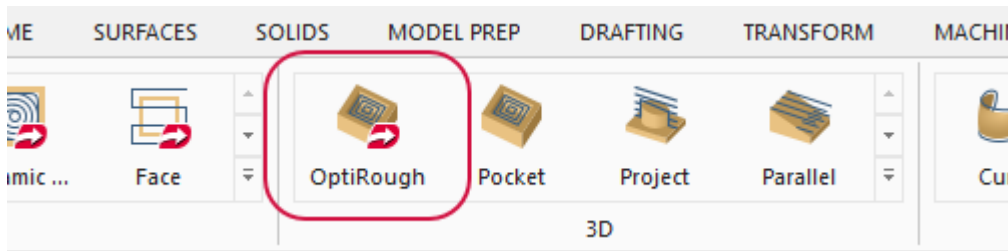




## Exercise 2: Creating the first Dynamic OptiRough toolpath

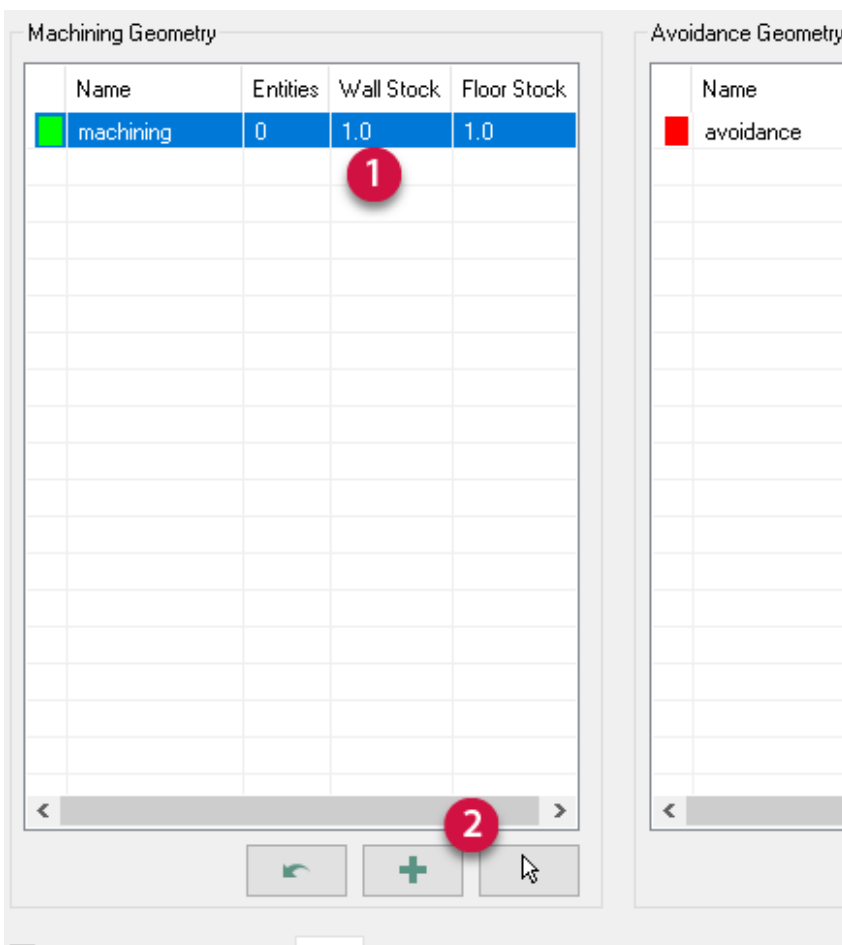
In this exercise, you create the first Dynamic OptiRough toolpath that uses this stock boundary.

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

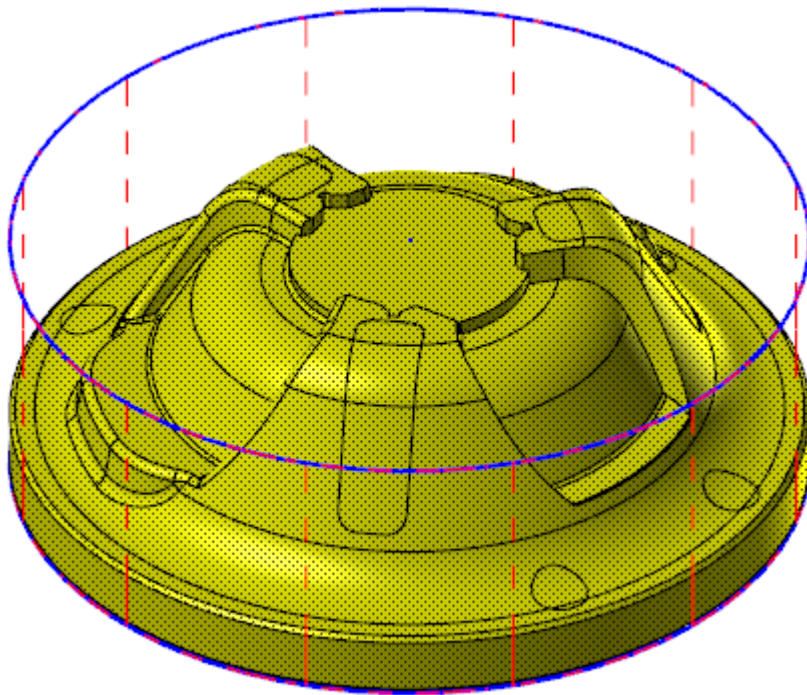


The **Model Geometry** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.


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



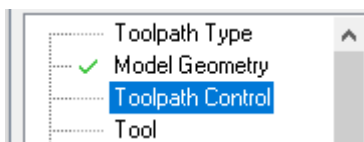
3. Use window select or any applicable keyboard shortcuts to select the part body as the drive surfaces. This includes the blue caps, as shown below.



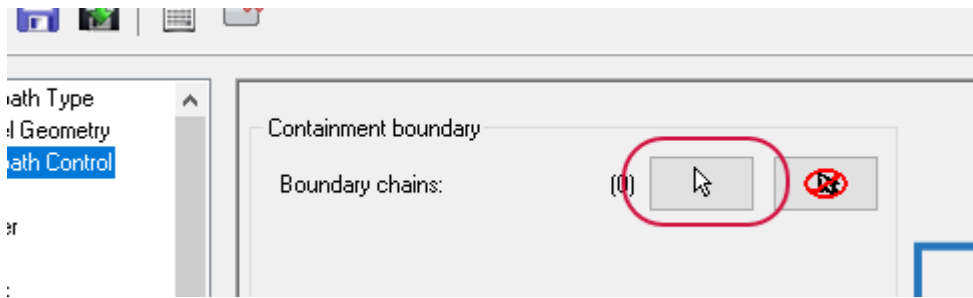
4. Press **[Enter]** or click **End Selection** to accept these surfaces.  
You return to the **Model Geometry** page.
5. Since you are creating a roughing toolpath, enter the following stock to leave amounts on the wall and floor. Double-click to activate the field.

Machining Geometry			
Name	Entities	Wall Stock	Floor Stock
 machining	449	0.3	0.3

- Enter **0.3** in **Wall Stock**
  - Enter **0.3** in **Floor Stock**
6. Select the **Toolpath Control** page.

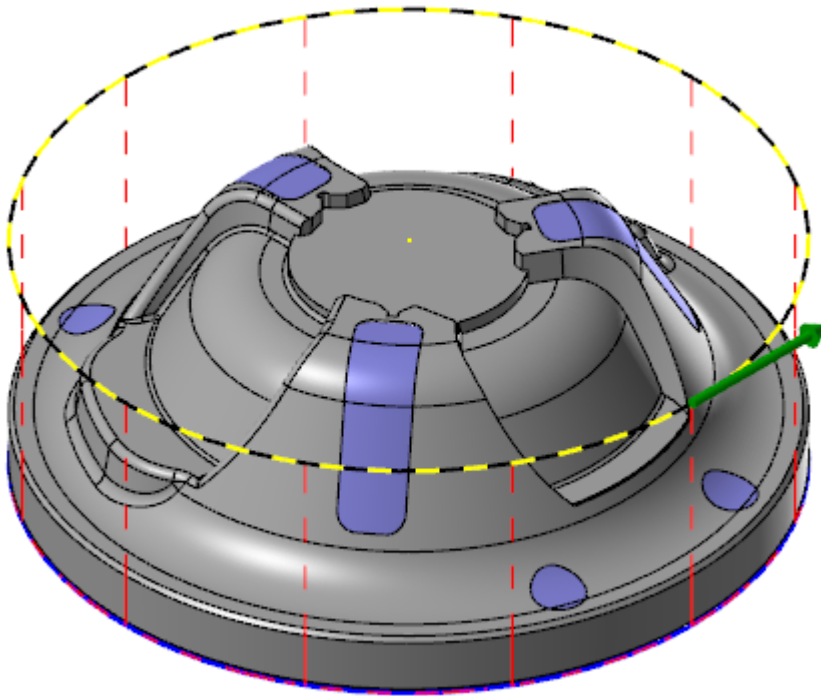


7. Click the **Boundary chains** selection arrow.

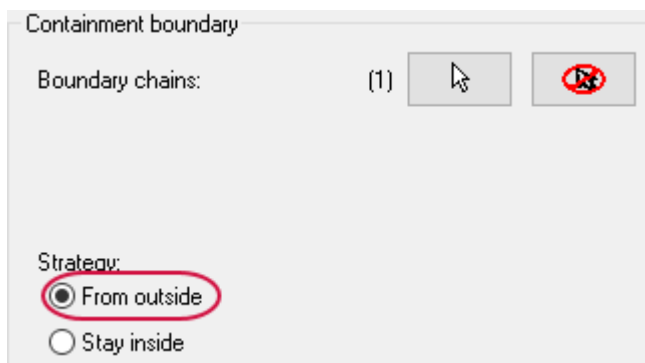


This returns you to the graphics window to select one or more closed chains of curves to limit the tool motion.

8. In the graphics window, select the geometry at the top of the stock boundary. You may need to change to **Wireframe** mode in the **Chaining** dialog box.

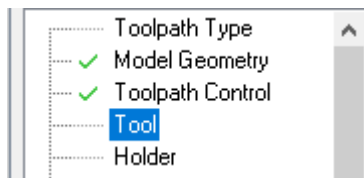


9. Click **OK** to return to the **Toolpath Control** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box.
10. Set the containment boundary **Strategy** to **From outside**.



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

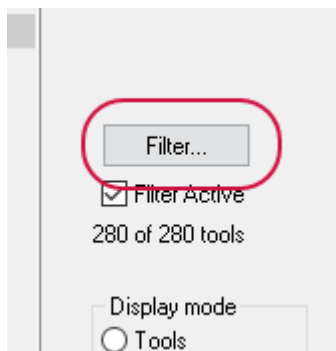
11. Select the **Tool** page.



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, and deselect any other tool filter. This ensures that you are only selecting bull end mill tools.

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

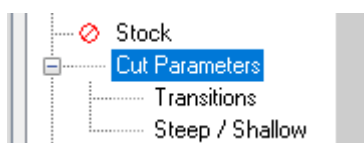
You return to the **Tool Selection** dialog box.

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

#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	R /
5	END MILL WITH RADIUS - 16 / R1.0	--	16.0	1.0	26.0	4	Bull e...	C
5	END MILL WITH RADIUS - 16 / R2.0	--	16.0	2.0	26.0	4	Bull e...	C
5	END MILL WITH RADIUS - 20 / R1.0	--	20.0	1.0	32.0	4	Bull e...	C
5	END MILL WITH RADIUS - 20 / R2.0	--	20.0	2.0	32.0	4	Bull e...	C
5	END MILL WITH RADIUS - 20 / R4.0	--	20.0	4.0	32.0	4	Bull e...	C
5	END MILL WITH RADIUS - 3 / R0.2	--	3.0	0.2	5.0	4	Bull e...	C
5	END MILL WITH RADIUS - 3 / R0.5	--	3.0	0.5	5.0	4	Bull e...	C
5	END MILL WITH RADIUS - 4 / R0.2	--	4.0	0.2	7.0	4	Bull e...	C
5	END MILL WITH RADIUS - 4 / R0.5	--	4.0	0.5	7.0	4	Bull e...	C

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:

Cut style  
Cut method: Climb  
Conventional feed rate: 2844.604  
Tip compensation: Tip  
Optimize stepups: By pocket  
Optimize stepdowns: Material

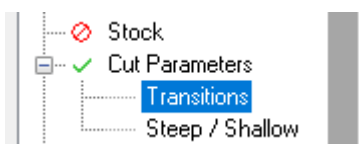
Passes  
Stepover: 30.0 % 6.0  
Stepdown: 150.0 % 30.0  
☒ Stepup: 15.0 % 3.0  
☐ Mill vertical walls  
Minimum toolpath radius: 10.0 % 2.0

Motion > Gap size, retract: Never  
Gap size:  
☐ Distance: 2000.0  
☒ % of tool diameter: 10000.0

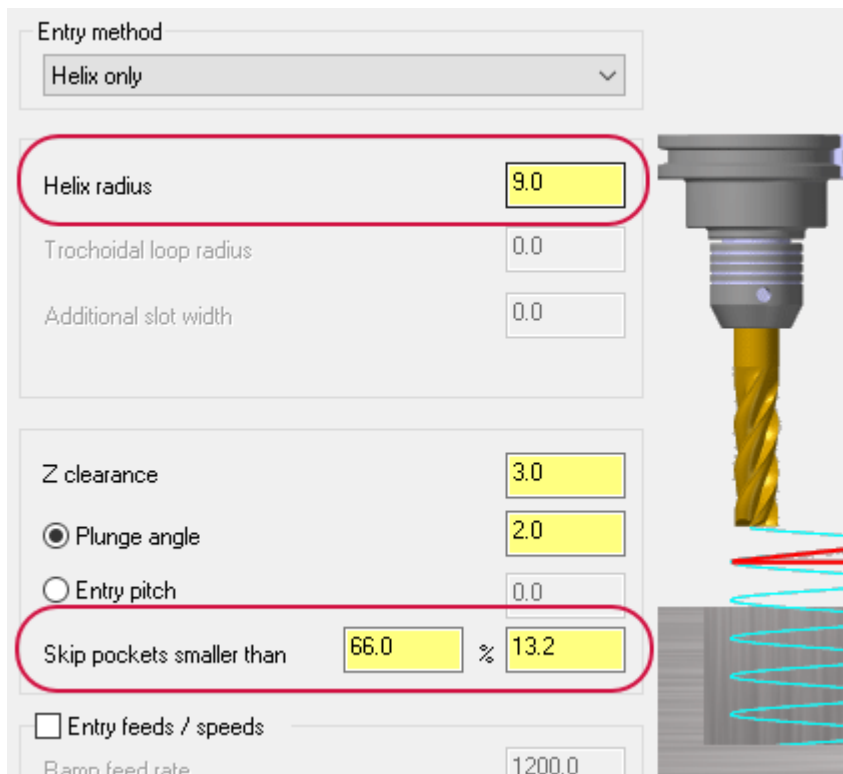
Motion < Gap size, micro lift  
Micro lift distance: 0.0  
Back feedrate: 100.0

- **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 feed rate** to **100.0**. Controls the speed of the backfeed movement of the tool.
- **% of tool diameter** to **10000.0**.

21. Select the **Transitions** page.

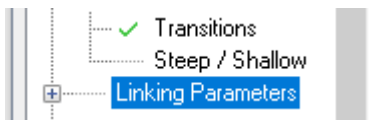


22. Set the following parameters:



- **Helix radius to 9.0.** Sets the radius of the entry helix.
- **Skip pockets smaller than to 66.0.** 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 the tool is effectively plunging into the part.

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



24. Set the following parameters:

**Retracts**

Clearance plane: 50.0

☒ Absolute ☐ Incremental

Curl up: 4.0

Curl down: 4.0

Part clearance: 4.0

Minimum Distance: [v]

☒ Output feed move: 12500.0

**Leads**

Linear entry/exit (incremental): 0.5

Vertical arc entry: 2.0

Vertical arc exit: 2.0

Horizontal arc entry: 2.0

Horizontal arc exit: 2.0

Max ramp angle: 10.0

Ramp height: 1.0

**Fitting**

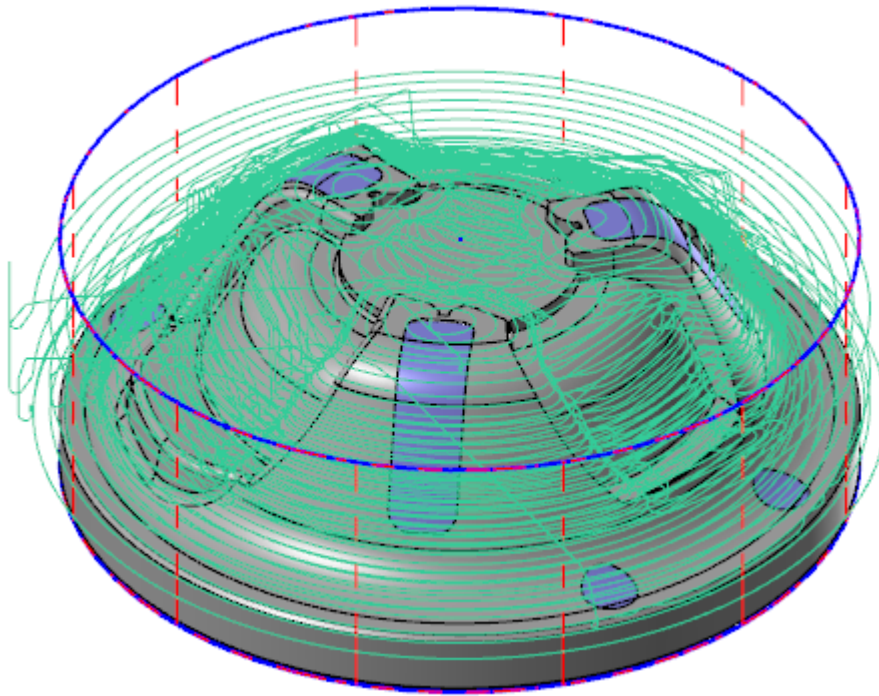
Minimize Trimming: [v]

Max trimming distance: 1.0

- 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.
- Set **Ramp height** to **1.0**. Sets the height of the helix when moving into a new region.

25. Click **OK** to close the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box and generate your toolpath.

26. Your toolpath should display as follows:

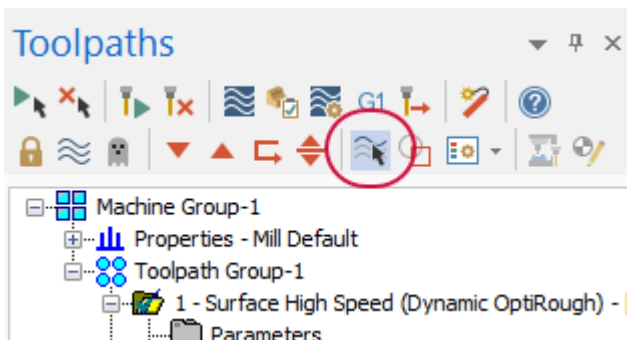


27. Save your part file.

### Exercise 3: Creating the Stock Model operation

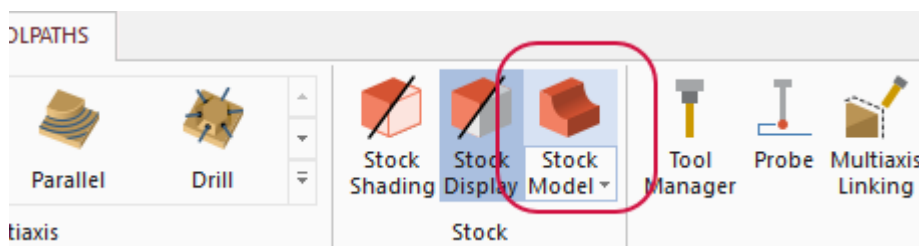
You now create a Stock Model operation that uses the previous operation as its source. This stock model will be used in the second Dynamic OptiRough toolpath to remove any remaining stock.

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



This will 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**.

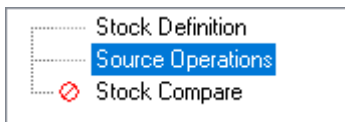


This will help differentiate the stock model from the solid or surface model.

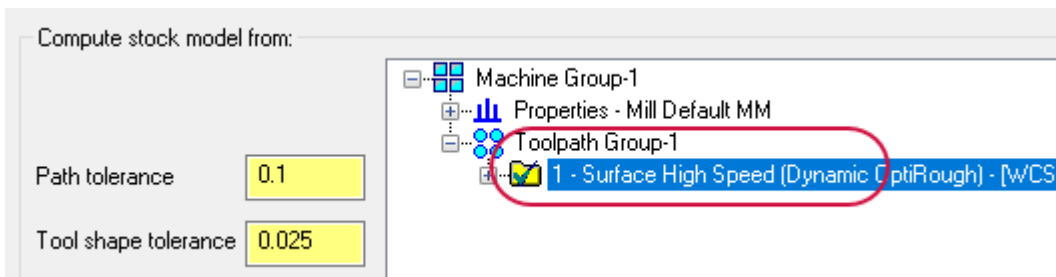
5. Select **Stock Setup**.

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.



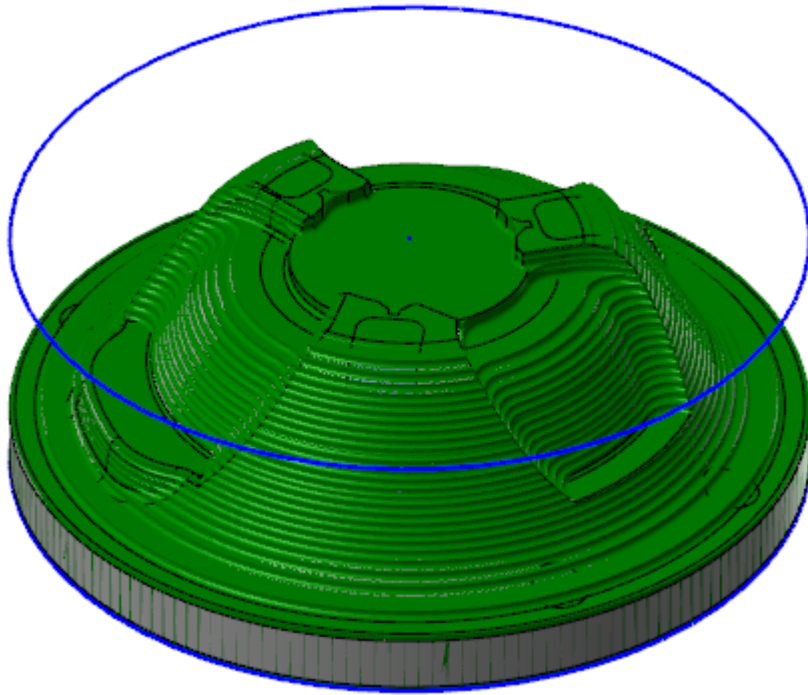
7. Select the **Surface High Speed (Dynamic OptiRough)** toolpath as the source operation.



The stock model is generated 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.

9. Your stock model displays as shown below:

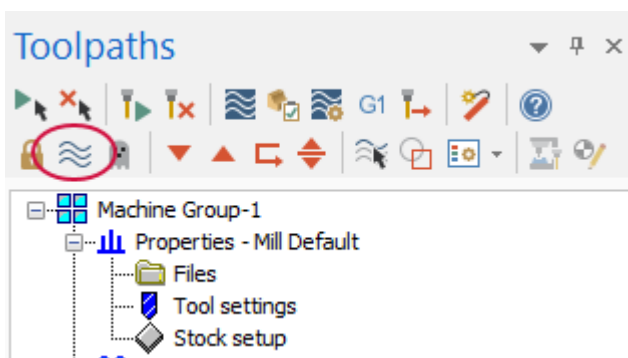


10. Turn off **Only display selected toolpaths** from the Toolpaths Manager.
11. Save your part.

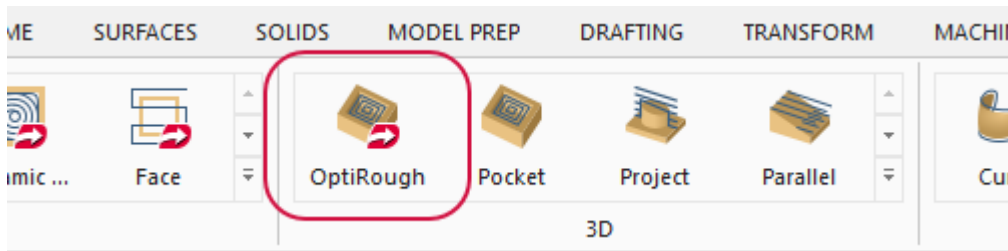
## Exercise 4: Creating the second Dynamic OptiRough toolpath

You now create the second 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**. This hides all selected toolpaths to keep the graphics window clean.

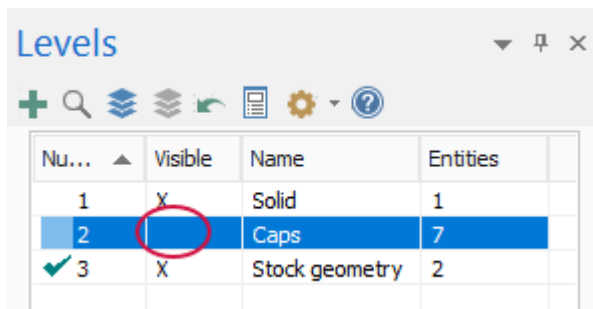


3. Select **OptiRough** from the **3D** gallery on the **Mill Toolpaths** contextual tab.

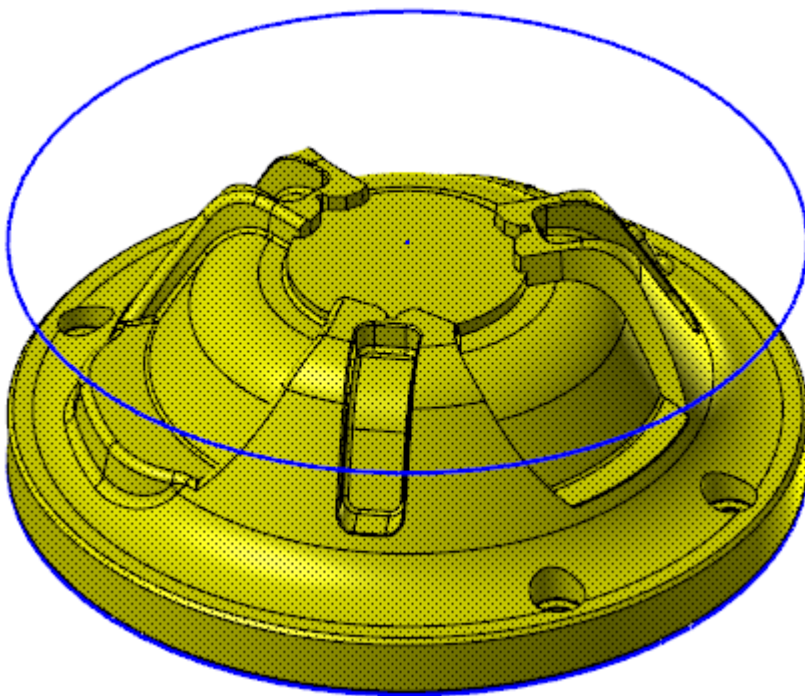


The **Model Geometry** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.

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.



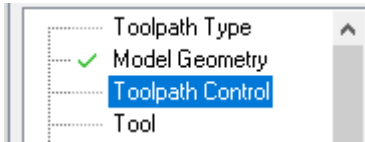
6. Select all but the blue caps as the drive surfaces shown below.



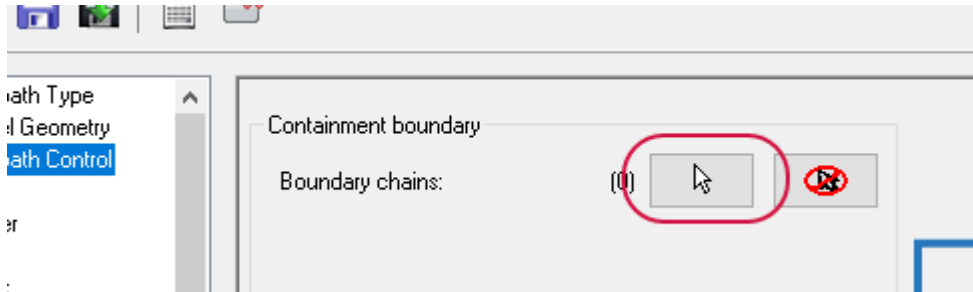
7. Press **[Enter]** or click **End Selection** to accept these surfaces.

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

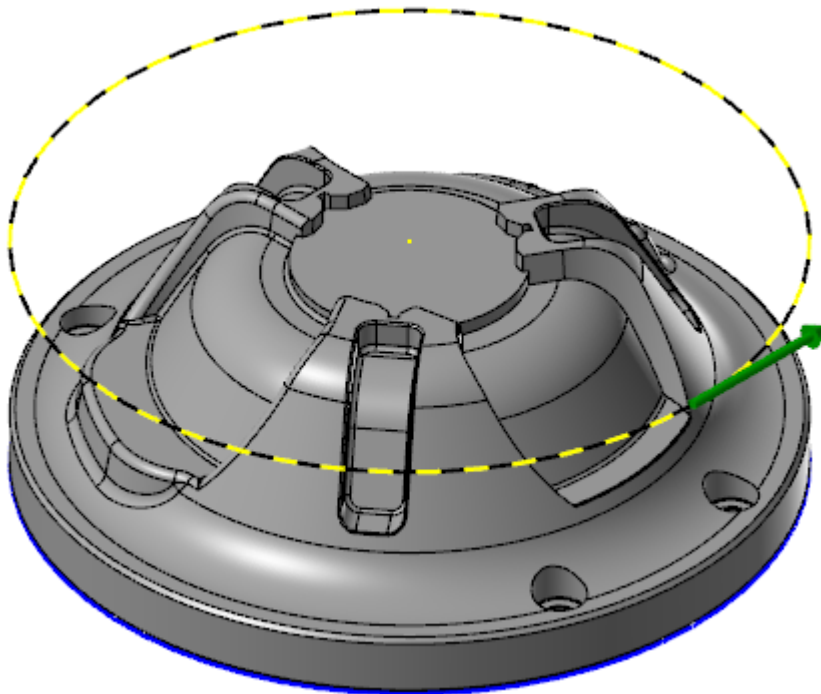
8. Select the **Toolpath Control** page.



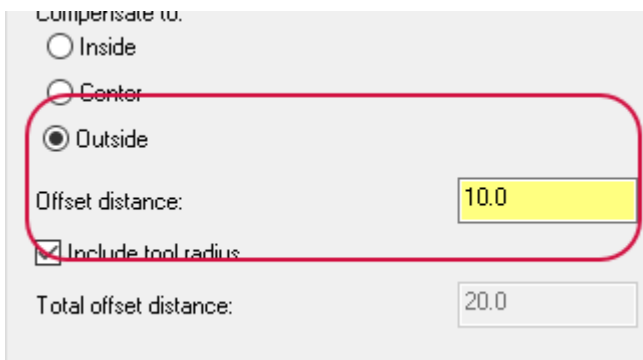
9. Click the selection arrow for **Boundary chains** to return to the graphics window.



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

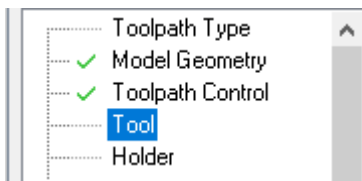


11. Click **OK** in the **Chaining** dialog box to return to the **Toolpath Control** page.
12. Set the following parameters to control the tool's position around the boundary of your part.



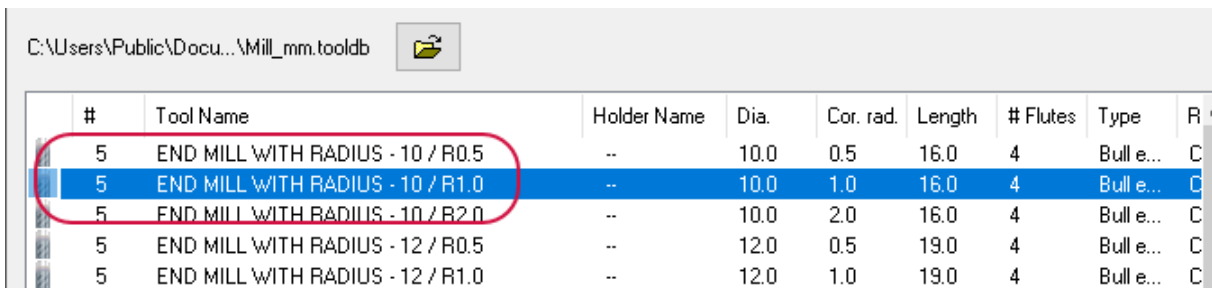
- Set **Compensate to** **Outside**. This sets the outer edge of the tool to be bound to the containment boundary.
- Set **Offset distance** to **10.0**. This option adjusts the inside or outside tool containment boundary.

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.

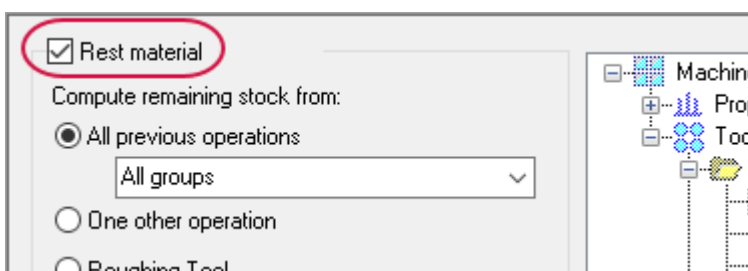


16. Click **OK** to add the tool to the toolpath.

17. Select the **Stock** page.

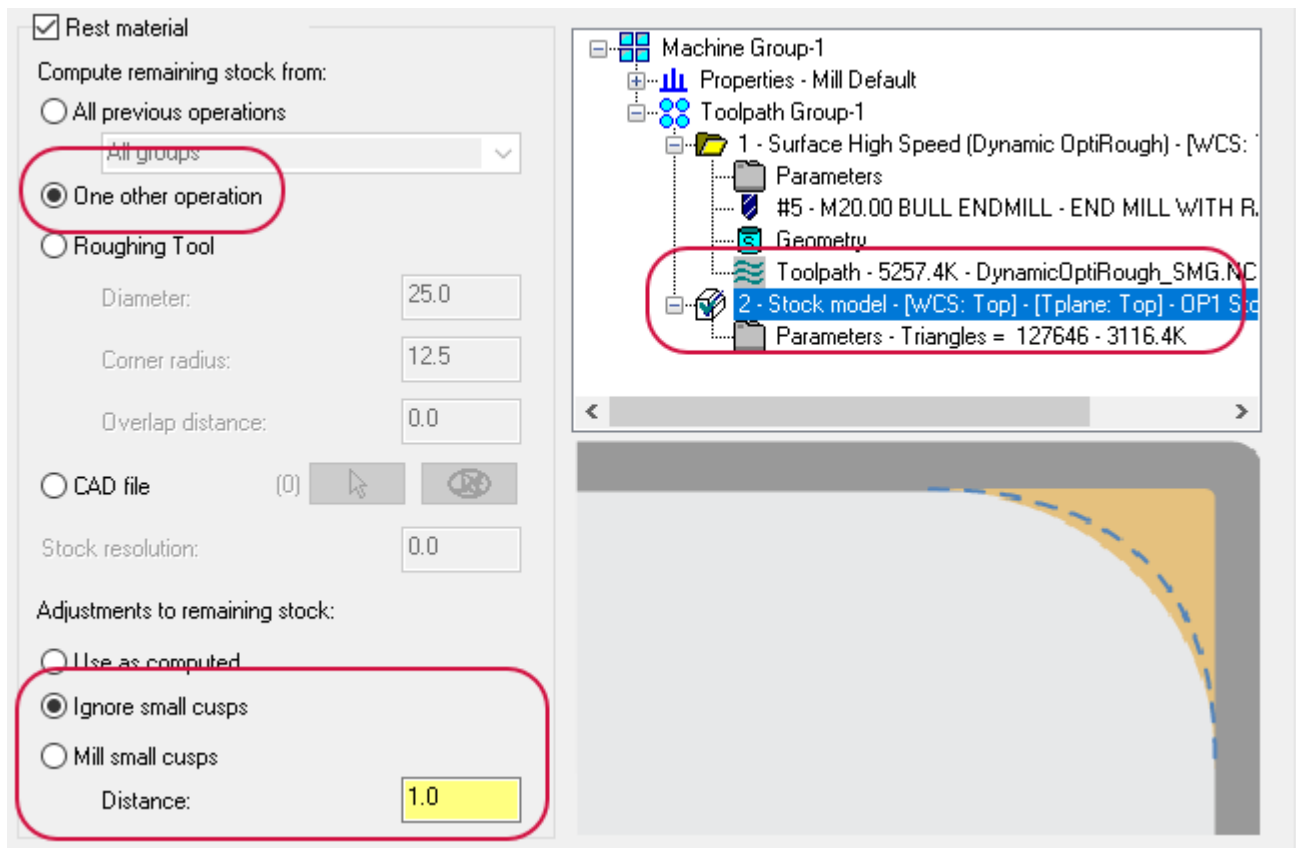


18. Select **Rest material**.

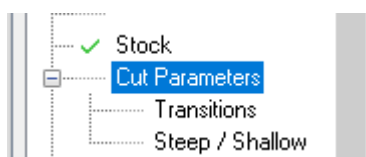


When you enable this page, Mastercam calculates the cutting passes based on the remaining stock.

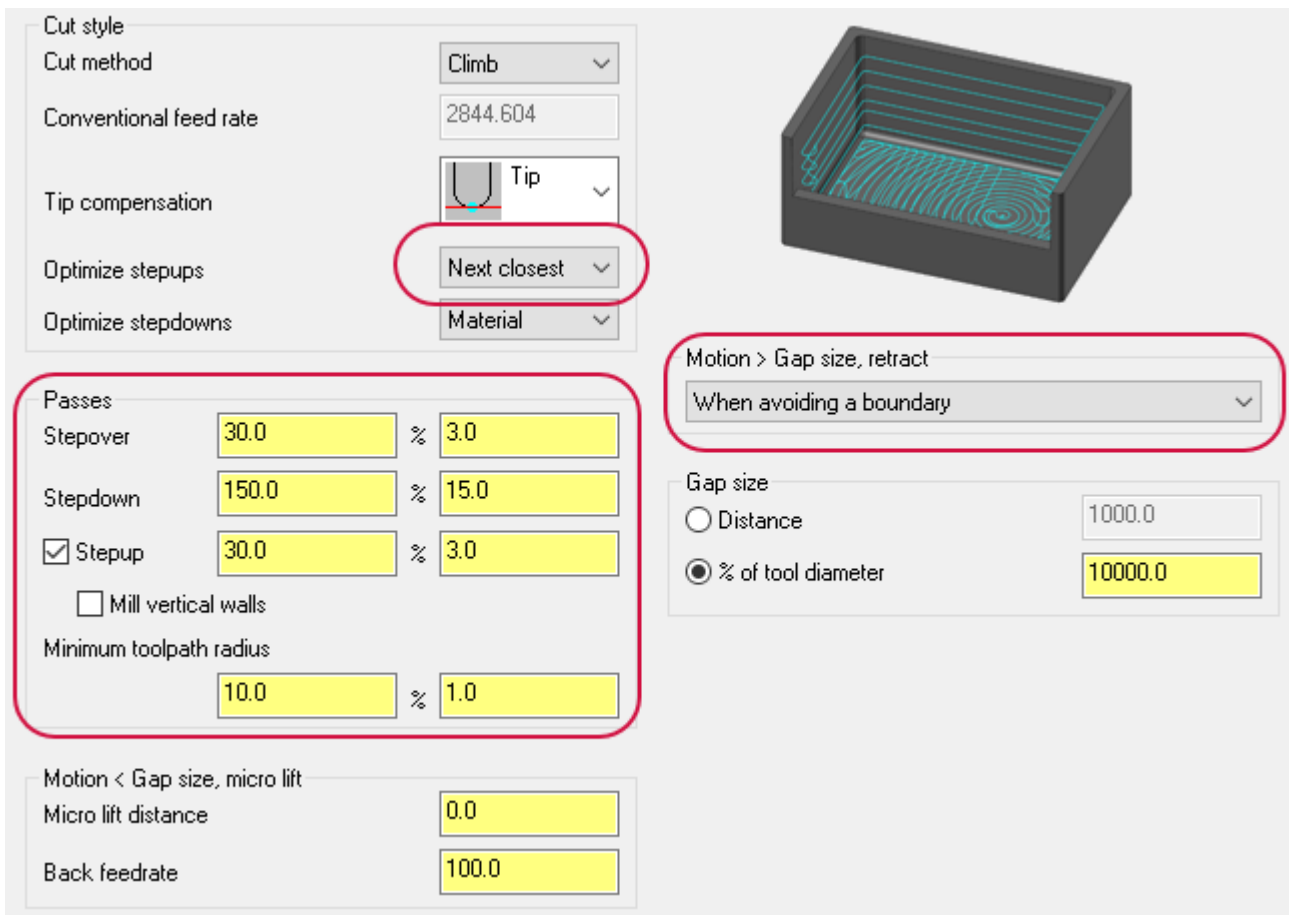
## 19. Set the following parameters:



- Set **Compute remaining stock from** to **One other operation**.
- In the **Operation List**, select **Stock Model - OP1 Stock**.
- 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:



Cut style

Cut method: Climb

Conventional feed rate: 2844.604

Tip compensation: Tip

Optimize stepups: **Next closest**

Optimize stepdowns: Material

Passes

Stepover: 30.0 % 3.0

Stepdown: 150.0 % 15.0

☒ Stepup: 30.0 % 3.0

☐ Mill vertical walls

Minimum toolpath radius: 10.0 % 1.0

Motion < Gap size, micro lift

Micro lift distance: 0.0

Back feedrate: 100.0

Motion > Gap size, retract: **When avoiding a boundary**

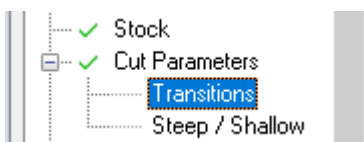
Gap size

☐ Distance: 1000.0

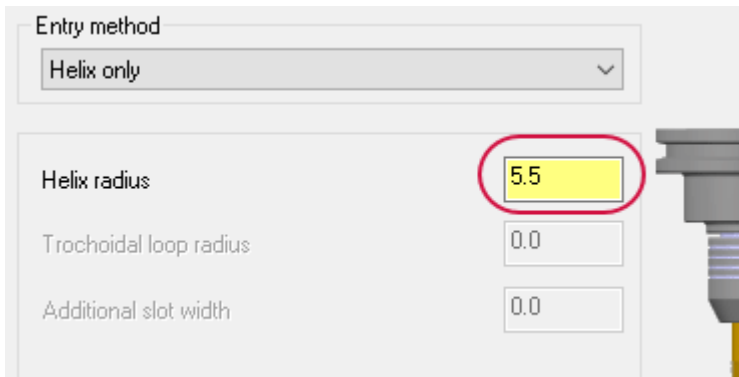
☒ % of tool diameter: **10000.0**

- **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).

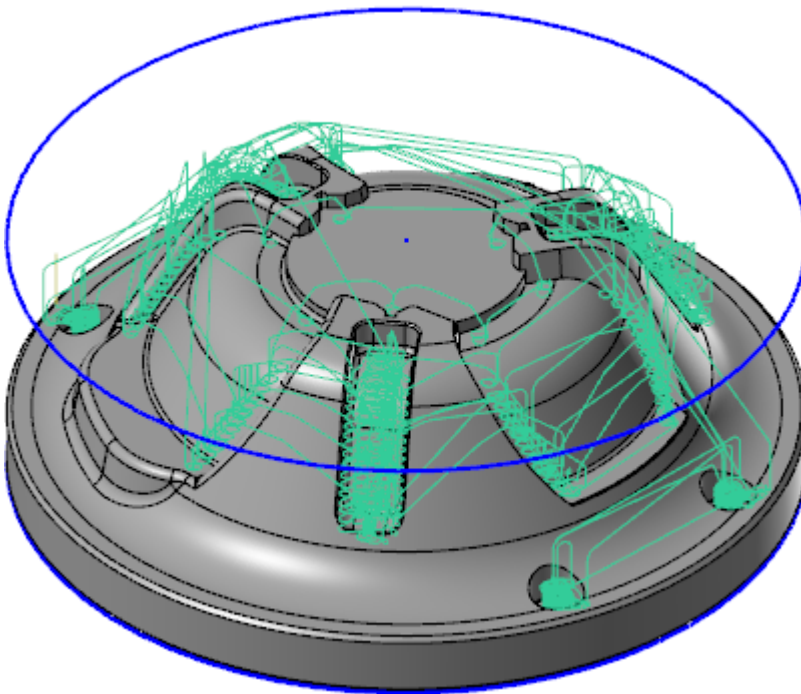
22. Select the **Transitions** page.



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



24. Click **OK** to create and 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.



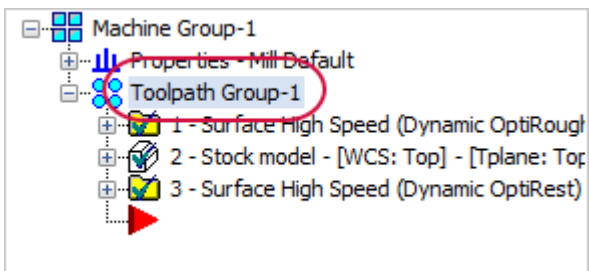
26. Save your part file.



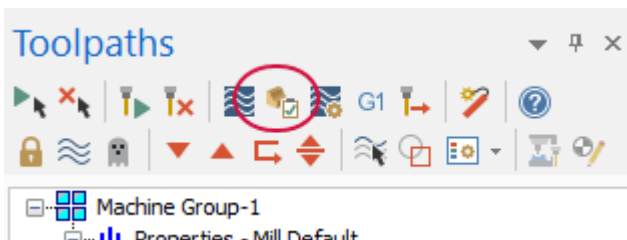
## Exercise 5: Verifying the stock model and toolpaths

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.

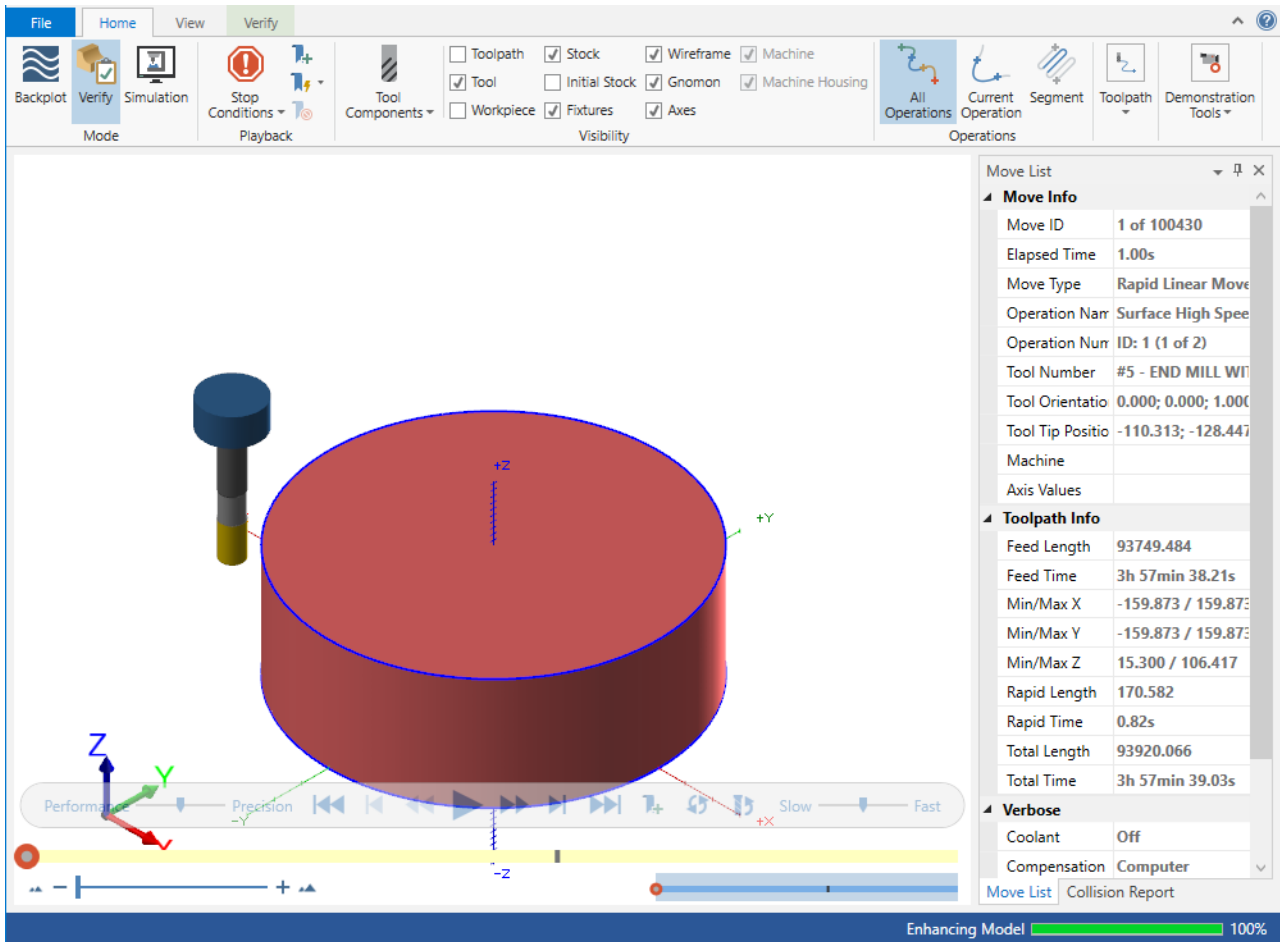


2. Select **Verify selected operations** in the Toolpaths Manager.

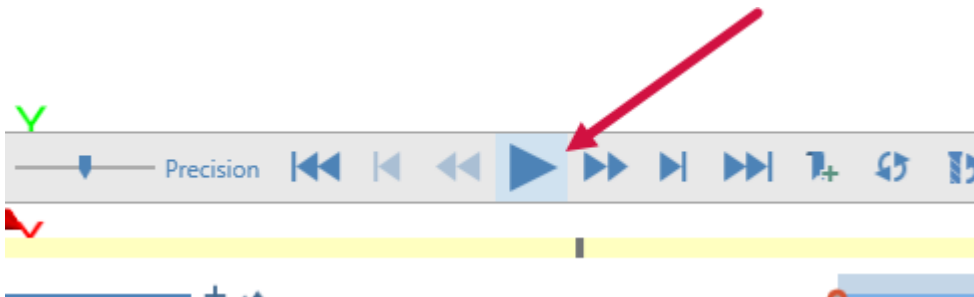


Mastercam Simulator displays.

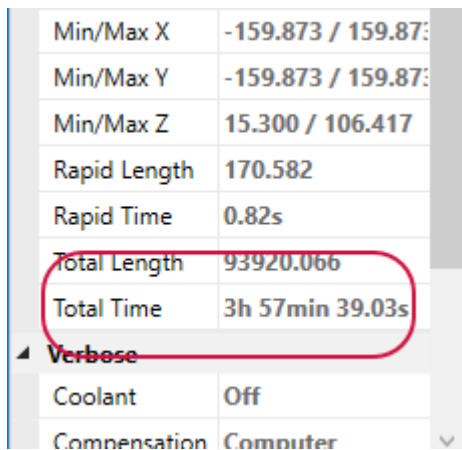
3. If necessary, right-click in the graphics window and change the view to **Isometric** and **Fit** the part to the screen so that it matches the image shown below.



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



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



Min/Max X	-159.873 / 159.873
Min/Max Y	-159.873 / 159.873
Min/Max Z	15.300 / 106.417
Rapid Length	170.582
Rapid Time	0.82s
Total Length	93920.066
Total Time	3h 57min 39.03s
▲ <b>Verbose</b>	
Coolant	Off
Compensation	Computer

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 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's **File** tab or by pressing **[Alt+H]** on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or [support@mastercam.com](mailto:support@mastercam.com)) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help, Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit [www.mastercamu.com](http://www.mastercamu.com), or email [training@mastercam.com](mailto:training@mastercam.com).
- *Online Communities*—You can find a wealth of information at [www.mastercam.com](http://www.mastercam.com). For tech tips and the latest Mastercam news, follow us on Facebook ([www.facebook.com/mastercam](http://www.facebook.com/mastercam)), Twitter ([www.twitter.com/mastercam](http://www.twitter.com/mastercam)), or Google+ ([plus.google.com/+mastercam](https://plus.google.com/+mastercam)). Visit our YouTube channel to see Mastercam in action ([www.youtube.com/user/MastercamCadCam](http://www.youtube.com/user/MastercamCadCam))! Registered users can search for information or ask questions on the Mastercam Web forum, [forum.mastercam.com](http://forum.mastercam.com), or use the knowledgebase at [kb.mastercam.com](http://kb.mastercam.com).

## Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at [techdocs@mastercam.com](mailto:techdocs@mastercam.com).



**ATTENTION! UPDATES MAY BE AVAILABLE.  
PLEASE REFER TO [MASTERCAM.COM/SUPPORT](http://mastercam.com/support)  
FOR THE LATEST DOWNLOADS.**

**CNC Software, Inc.**  
671 Old Post Road  
Tolland, CT 06084



[www.mastercam.com](http://www.mastercam.com)

***Mastercam***<sup>®</sup>

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