# MASTERCAM FOR SOLIDWORKS DYNAMIC MILLING

April 2018

POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam

# MASTERCAM FOR SOLIDWORKS DYNAMIC MILLING

April 2018
© 2018 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2019 for SOLIDWORKS

## Terms of Use

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

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

## Be sure you have the latest information!

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

# TABLE OF CONTENTS


**Introduction** ..... **5**

General Tutorial Requirements ..... 6

**Dynamic Toolpaths Overview** ..... **7**

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

**The Dynamic Mill Toolpath** ..... **17**

Exercise 1: Opening Mastercam for SOLIDWORKS ..... 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 ..... 35

Exercise 5: Verifying the toolpaths ..... 45

**The Face Toolpath** ..... **49**

Exercise 1: Creating the Face toolpath ..... 49

Exercise 2: Verifying the toolpaths ..... 54

**The Dynamic Contour Toolpath** ..... **57**

Exercise 1: Setting up the stock ..... 57

Exercise 2: Creating the first Dynamic Contour toolpath ..... 59

Exercise 3: Creating the second Dynamic Contour toolpath ..... 66

Exercise 4: Analyzing the toolpaths ..... 72

Exercise 5: Verifying the toolpaths ..... 73

**The Peel Mill Toolpath** ..... **77**

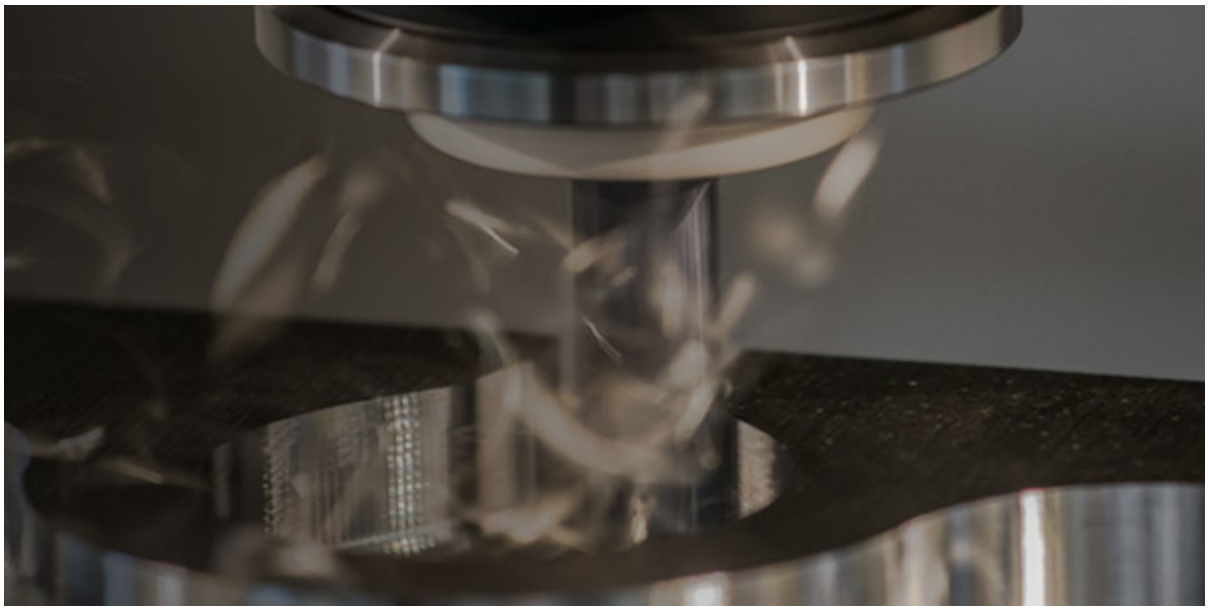
Exercise 1: Setting up the stock ..... 77

Exercise 2: Creating the first Peel Mill toolpath ..... 78

Exercise 3: Copying the Peel Mill toolpath ..... 84
Exercise 4: Re-chaining the copied Peel Mill operations ..... 85
Exercise 5: Verifying the toolpaths ..... 89
**The Dynamic OptiRough Toolpath ..... 91**
Exercise 1: Setting up the stock ..... 91
Exercise 2: Creating the first Dynamic OptiRough toolpath ..... 94
Exercise 3: Creating the Stock Model operation ..... 102
Exercise 4: Creating the second Dynamic OptiRough toolpath ..... 104
Exercise 5: Verifying the stock model and toolpaths ..... 111
**Conclusion ..... 113**

# 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 for SOLIDWORKS 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 for SOLIDWORKS tutorials have the following general requirements:

- You must be comfortable using the Windows® operation system.
- You must have a seat of SOLIDWORKS® 2015 or higher to complete this tutorial.
- 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 for SOLIDWORKS 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

# DYNAMIC TOOLPATHS OVERVIEW

The following toolpaths use Dynamic Motion technology:

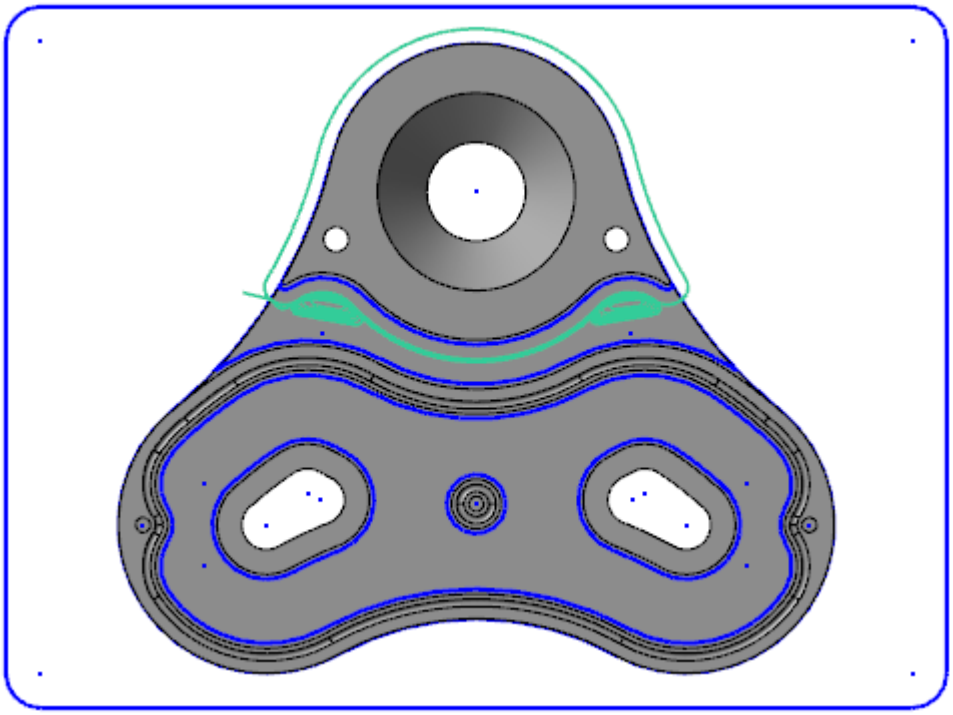
- Dynamic Contour
- Face
- Dynamic Mill
- Peel Mill
- Dynamic OptiRough

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

## Dynamic Toolpath Types

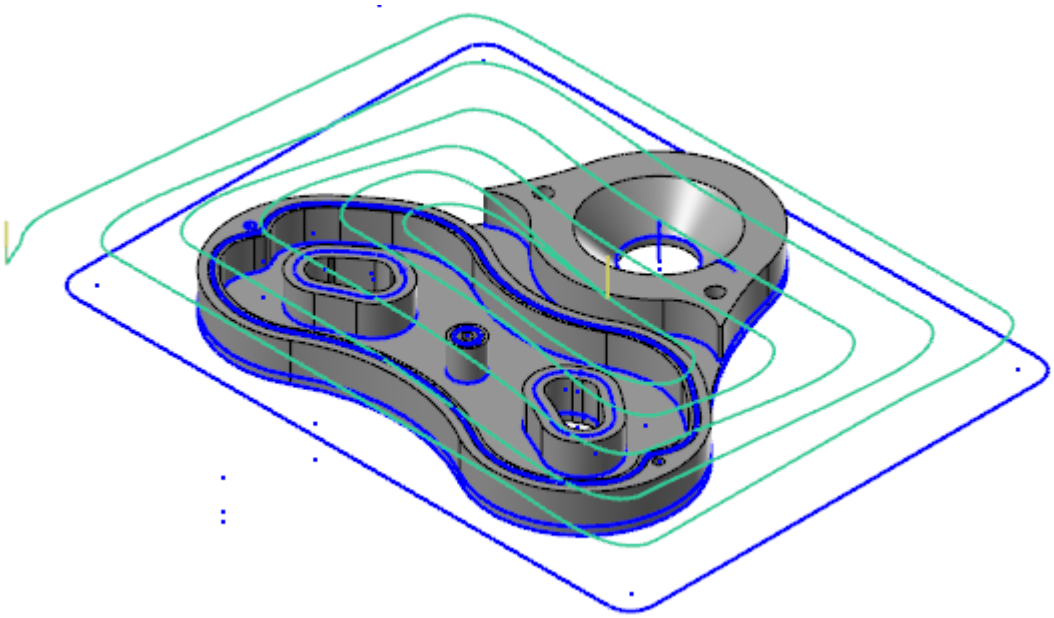
### Dynamic Contour

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

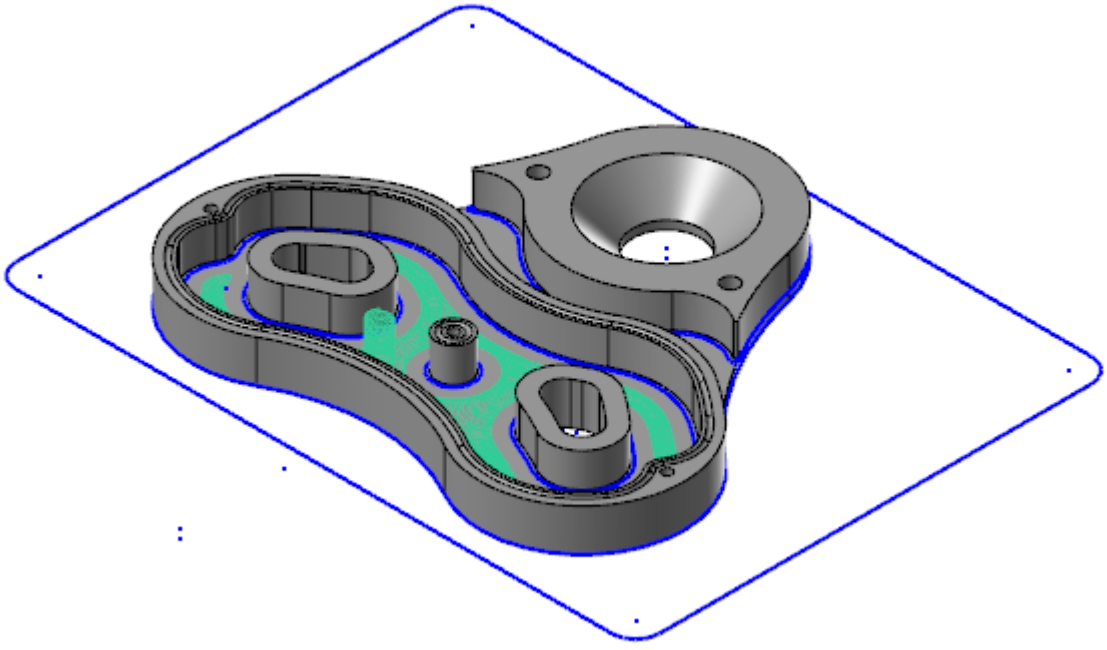
The Face toolpath, when the **Style** is set to **Dynamic**, quickly cleans stock from the top of a part and creates an even surface for future operations. You can base the toolpath on either chained geometry or on the current stock model.



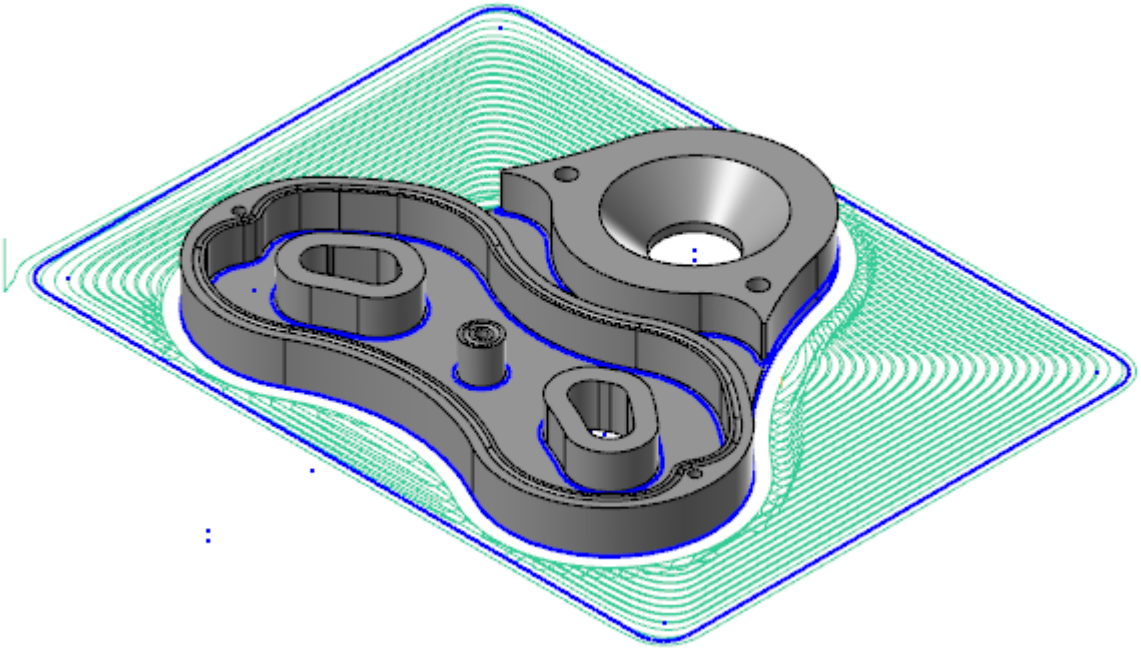
## Dynamic Mill

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.

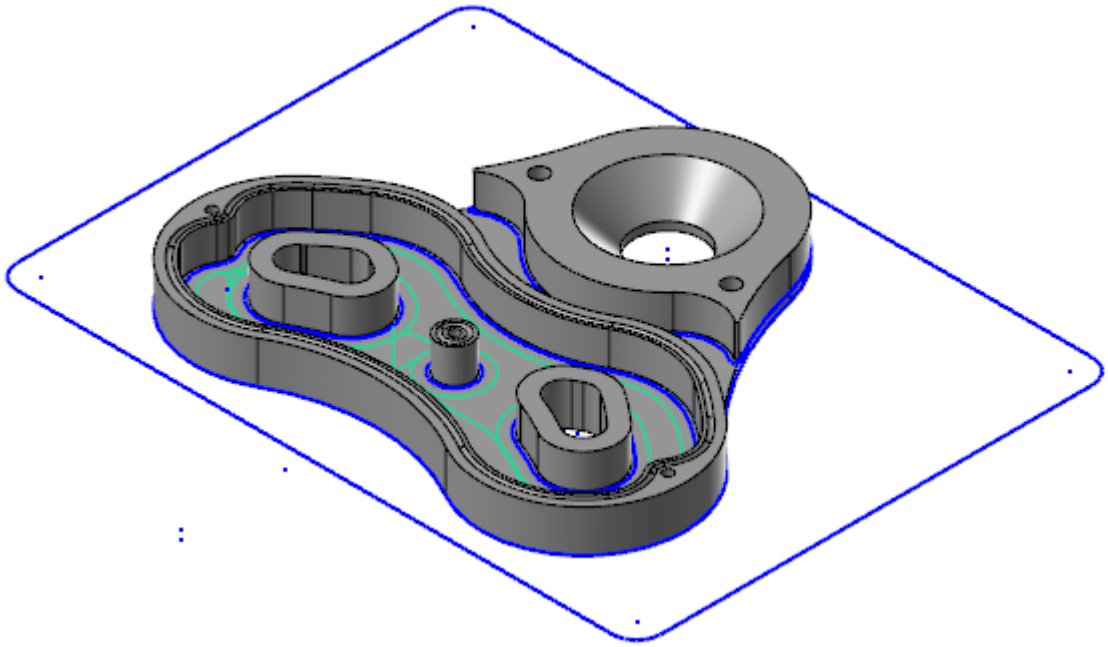
By selecting the **Stay inside** strategy, Dynamic Mill becomes a pocket toolpath that utilizes dynamic motion. You select a boundary, areas to avoid, and set tooling, parameters, entry method, and linking values to generate the toolpath. Material is removed from the inside to the outside in a highly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

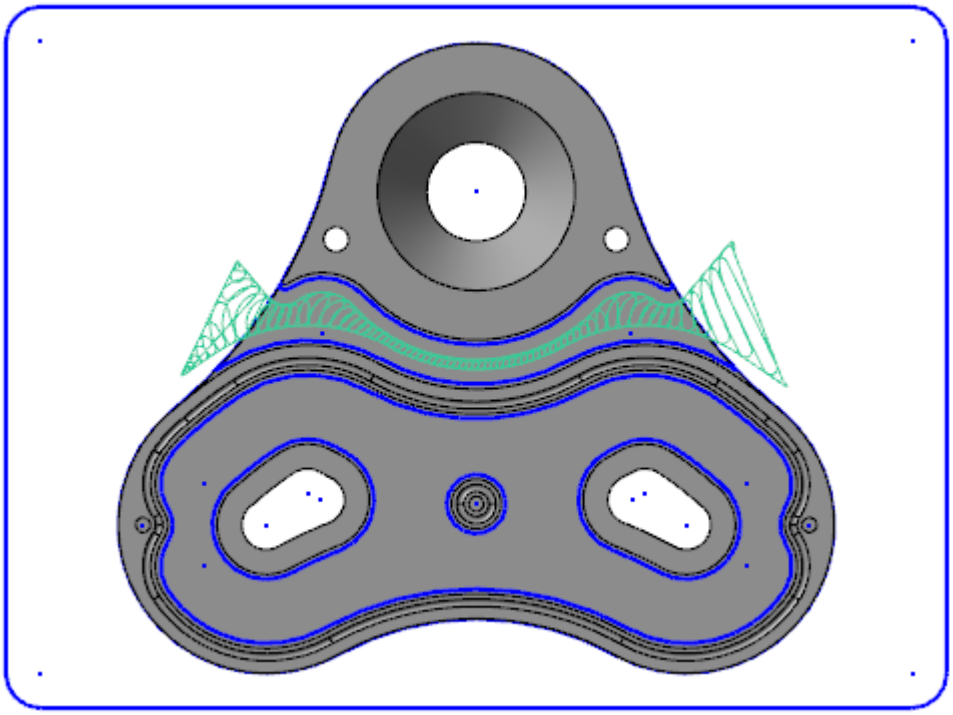


By enabling **Rest Material** on the Stock page, the Dynamic Mill toolpath removes material left by previous operations using dynamic motion. The Dynamic Mill operation calculates how much material to remove based on previous operations or previous roughing tools. Only unmachined areas are processed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



## Peel Mill

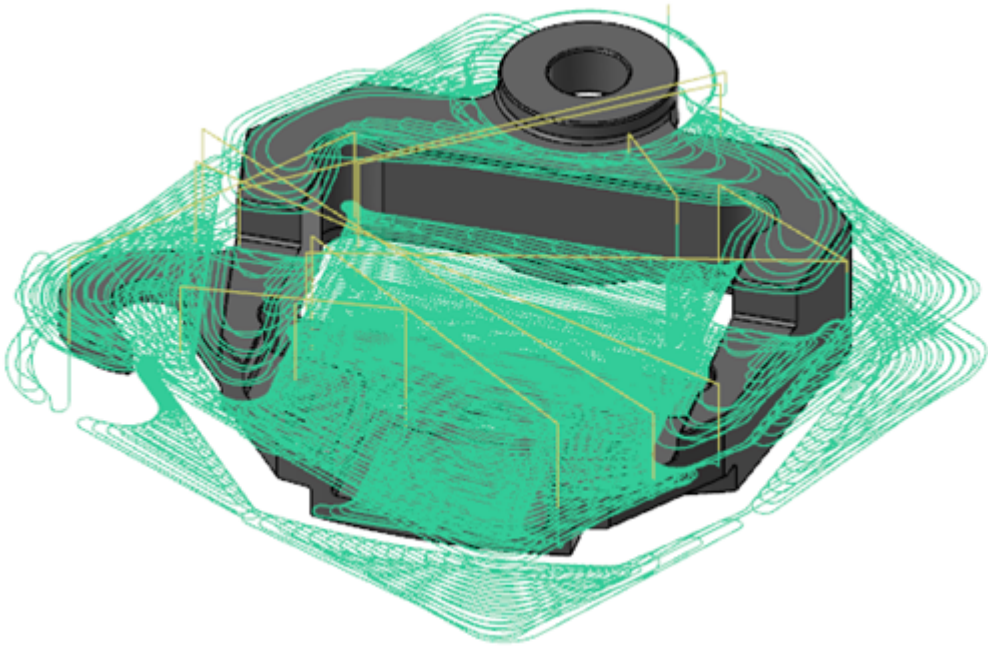
Peel Mill, with the **Cutting strategy** set to **Dynamic Peel**, allows for efficient milling between selected chains. It uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material. For a single chain, you define the width of the cut. Otherwise, the width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

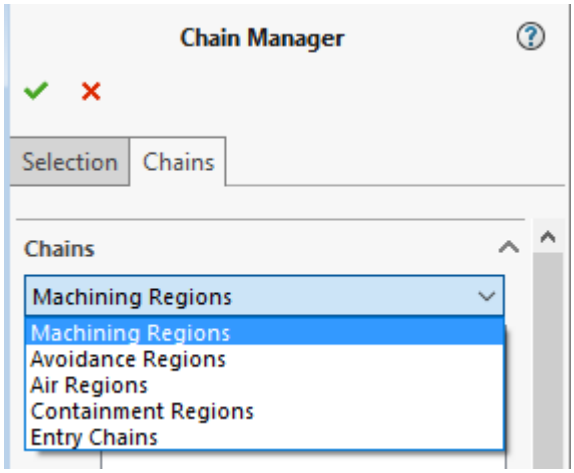


## Dynamic Toolpath Selection

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:

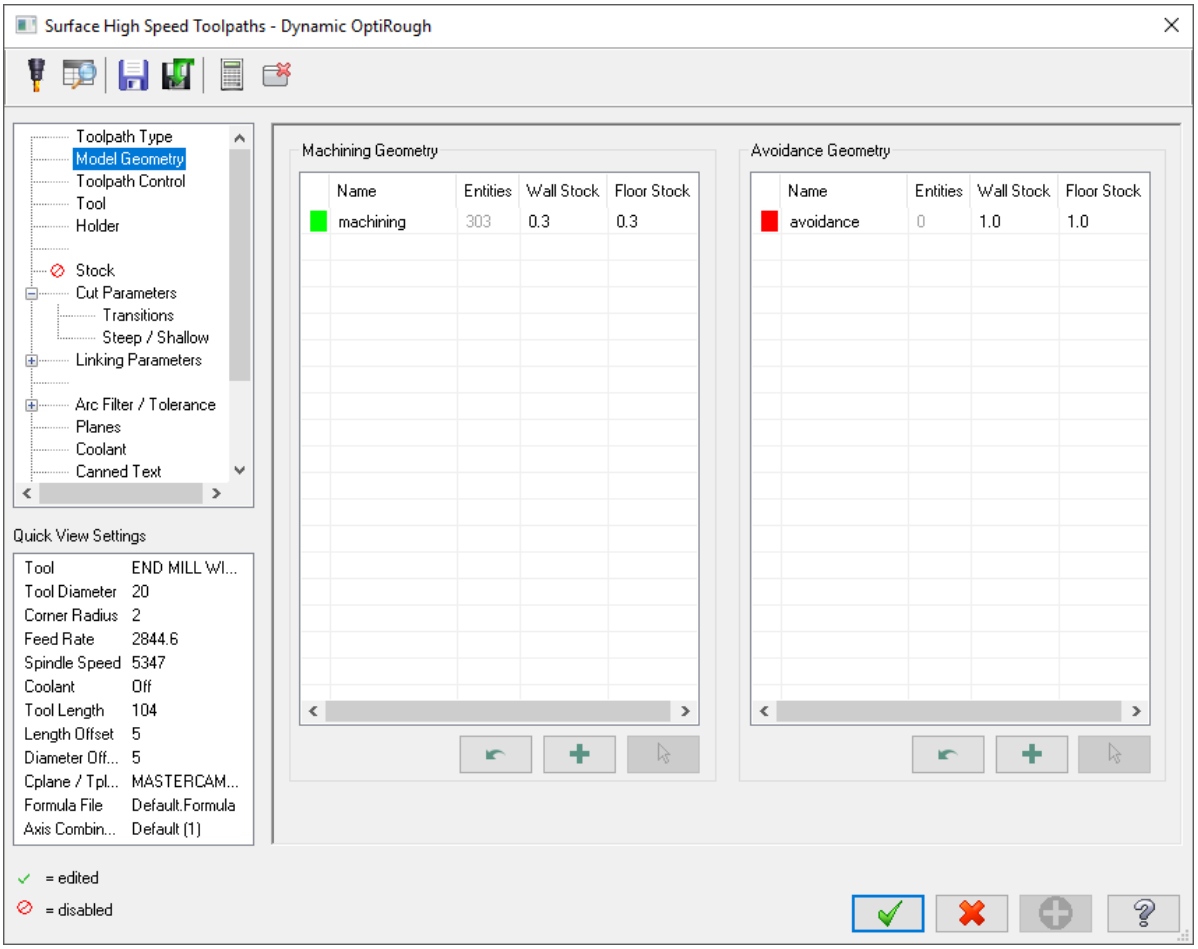



- **Machining Regions**: Areas to be machined.
- **Avoidance Regions**: Areas to be avoided during machining. Multiple avoidance regions can be selected.
- **Air Regions**: Areas that contain no material and allow the tool to travel through it when machining.
- **Containment Regions**: Areas that limit tool motion.
- **Entry Chains**: The point where the tool enters the part.

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.

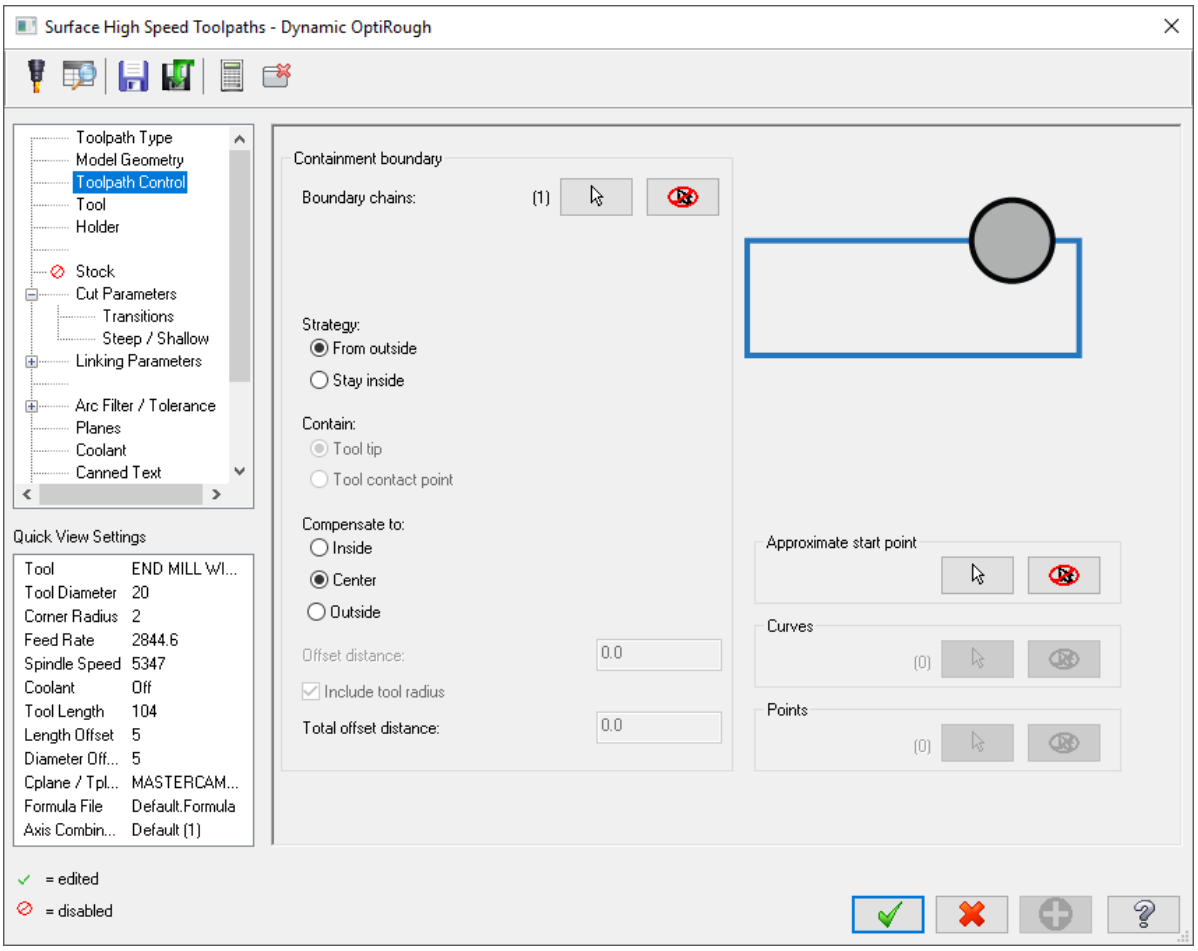
For more information, view Mastercam's **Help**.

Dynamic OptiRough, as it is the only 3D Dynamic toolpath, uses a different type of selection method compared to the other Dynamic toolpaths. After selecting your geometry in the Chain Manager, the **Model Geometry** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays, allowing you to make further changes.




- **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.

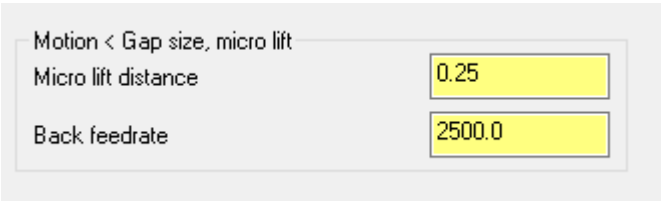



## Dynamic Milling Parameters

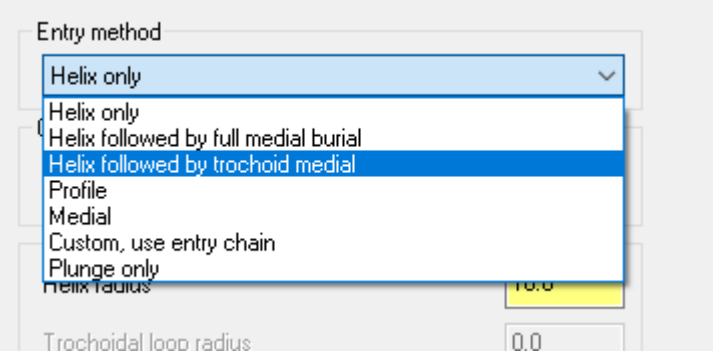
Several parameter options separate the Dynamic operations from their standard counterparts. These parameters are on the **Cut Parameters**, **Entry Motion**, and **Contour Wall** pages.

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

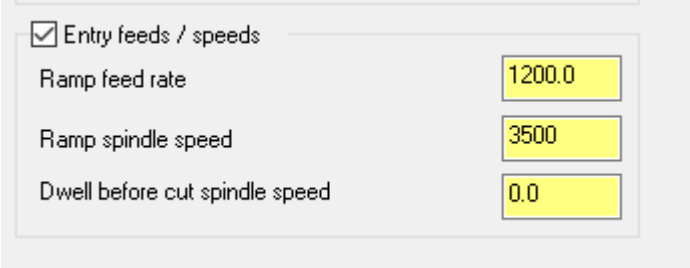
Micro lift options allow the tool to rise above the floor of the part for the portion of the toolpath that is not removing material. Benefits of the micro lift include chip clearing and avoiding excess tool heat. The feed rate is controllable for the back move as well.



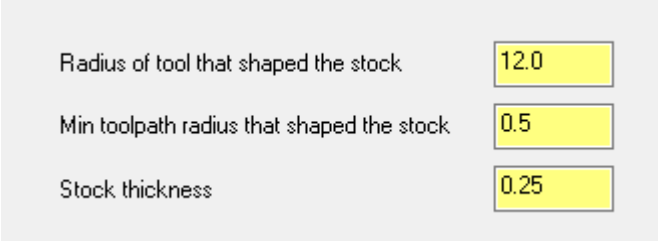
Entry methods provide several ways to enter the material for a Dynamic Mill toolpath. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's **Help** contains detailed information on each of these methods.



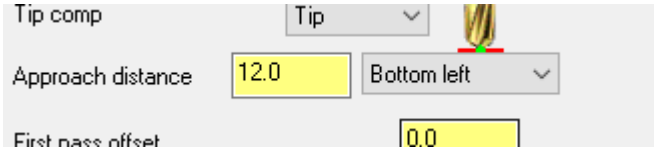

Use Entry feeds and speeds to avoid placing too much load on the tool upon entry. Set a short dwell interval after entry to allow the spindle to reach the necessary speed before beginning material removal.




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.

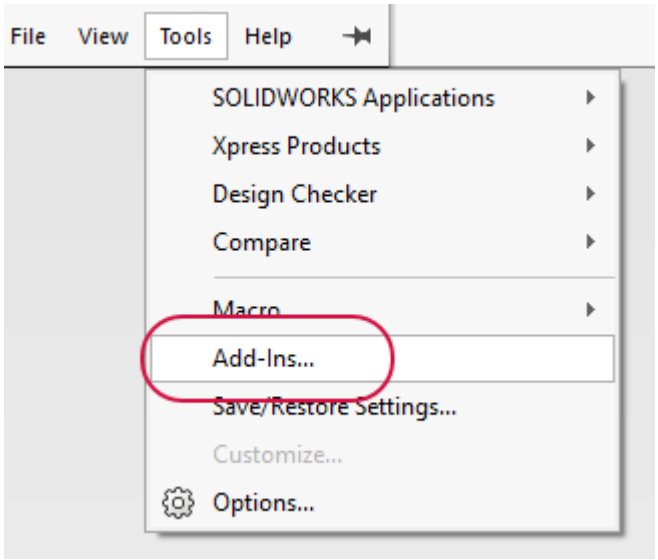
In this lesson, you will create two Dynamic Mill toolpaths. The first one will clear material around the outside of the part. The second Dynamic Mill toolpath uses the previous one as a source operation for removing the remaining material on the top of the part.

## Lesson Goals

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

## Exercise 1: Opening Mastercam for SOLIDWORKS

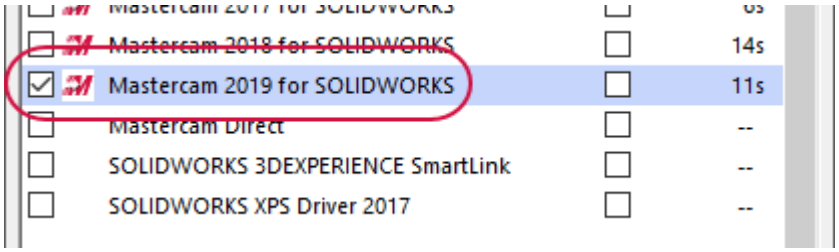
1. Start SOLIDWORKS.
2. Select **Tools**, **Add-Ins**.



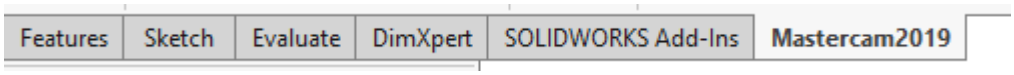

The **Add-Ins** dialog box displays.

3. Find **Mastercam 2019 for SOLIDWORKS** in the Active Add-Ins list, and then do the following:
   a. To load the Mastercam 2019 for SOLIDWORKS add-in for the current session, select the checkbox to the left of the Mastercam 2019 for SOLIDWORKS entry.
   b. To load the Mastercam 2019 for SOLIDWORKS add-in at every SOLIDWORKS start-up, select the check-box to the right of the Mastercam 2019 for SOLIDWORKS entry.

c. Click **OK** to close the dialog box.



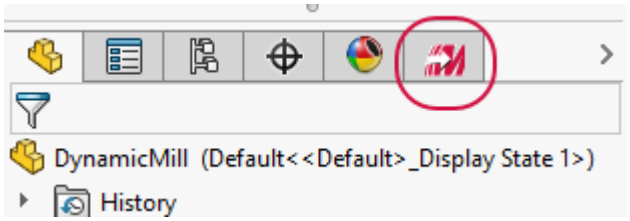

4. After a few moments the add-in loads. When you open a part, the **Mastercam2019** tab displays in SOLIDWORKS.



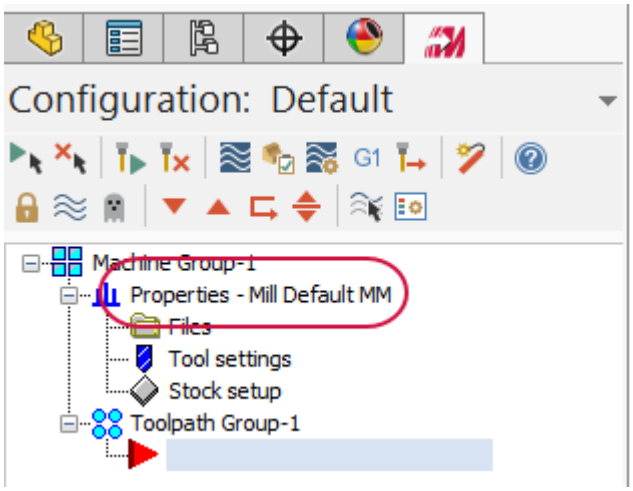

## Exercise 2: Setting up the stock

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

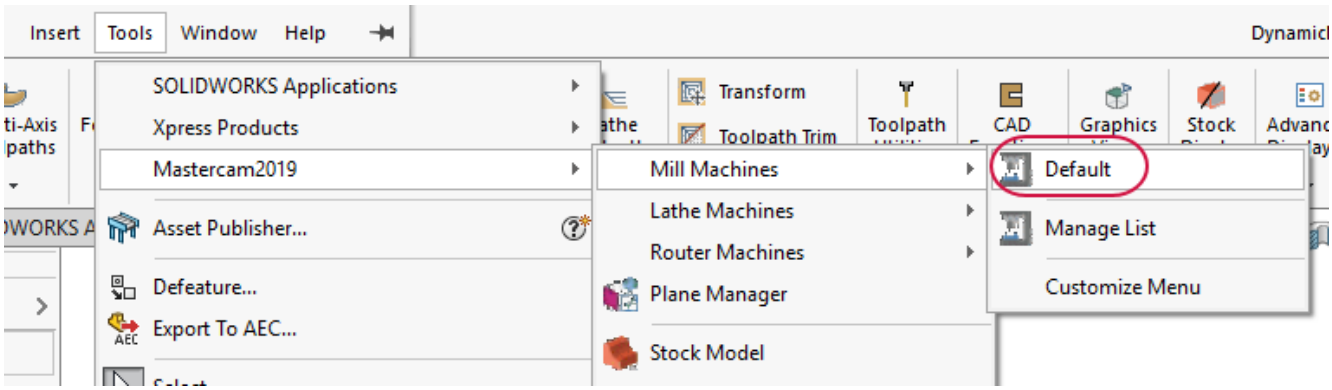
1. Open the part file, `DynamicMill`, provided with this tutorial.
2. Save the part as `DynamicMill-xxx`, replacing `xxx` with your initials. You may be asked to save the part file to the current version.
3. Select the **Mastercam Toolpath Manager** tab to display the Toolpaths Manager.



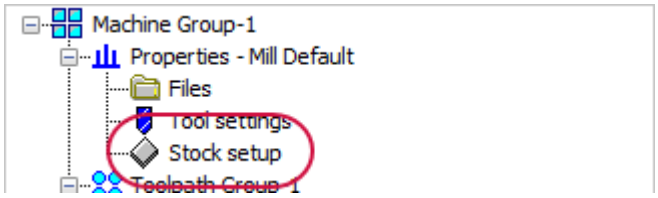

Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.



4. If you do not have the correct default machine, select **Mastercam2019**, **Mill Machines**, **Default** from the **Tools** menu.

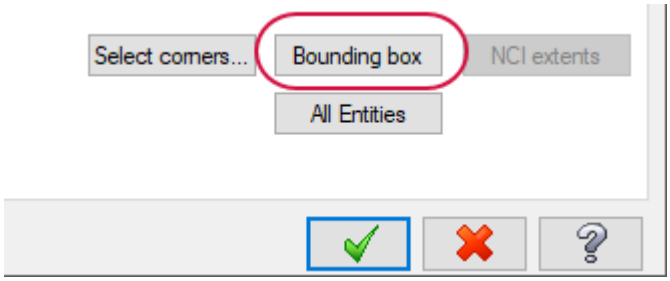



5. Select **Stock setup** in the Toolpaths Manager.



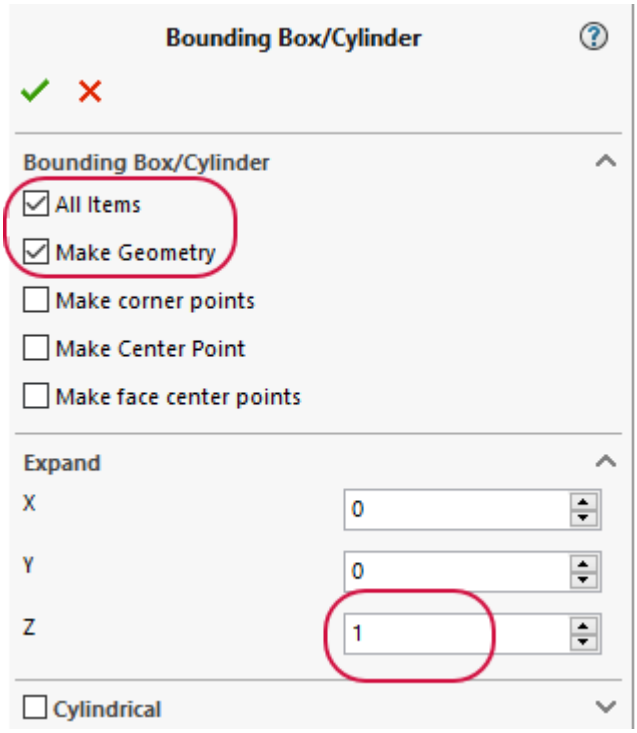

The **Machine Group Properties** dialog box displays.

6. On the **Stock Setup** tab, click **Bounding box**.



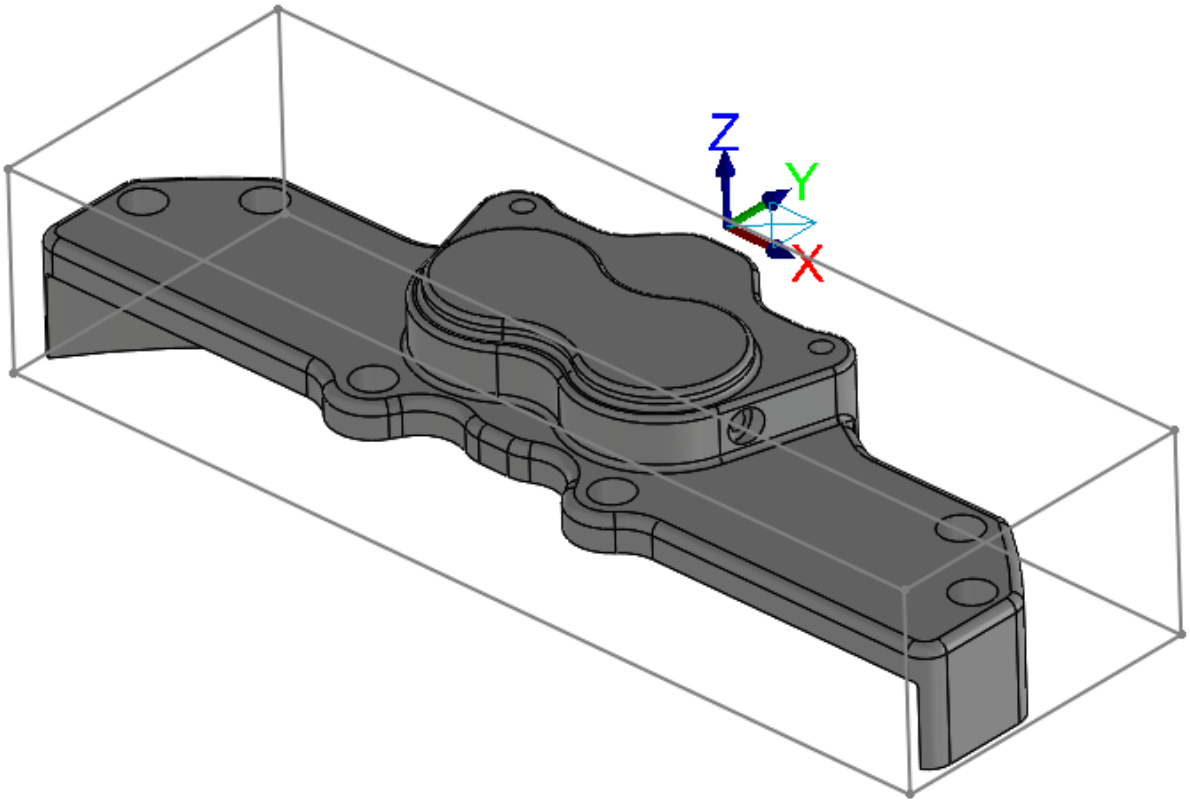

The **Bounding Box/Cylinder** function panel displays.

7. Set the following parameters:




   - Ensure that **All Items** is selected. This creates a bounding box around all entities.
   - Select **Make Geometry**. This creates a closed boundary of lines that will be used for the Dynamic Mill toolpath.
   - Expand **Z** by **1.0**. This enlarges the bounding box in the Z axis.
8. Click **OK** to accept and create the boundary, and return to the **Machine Group Properties** dialog box.

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



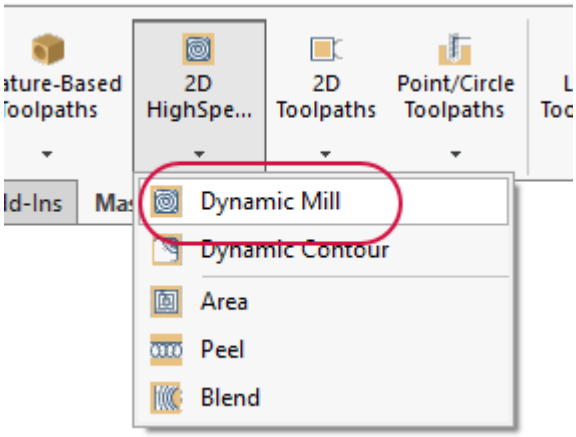

You have now created the necessary stock and geometry to create the Dynamic Mill toolpaths.

10. Save your part.

## Exercise 3: Creating the first Dynamic Mill toolpath

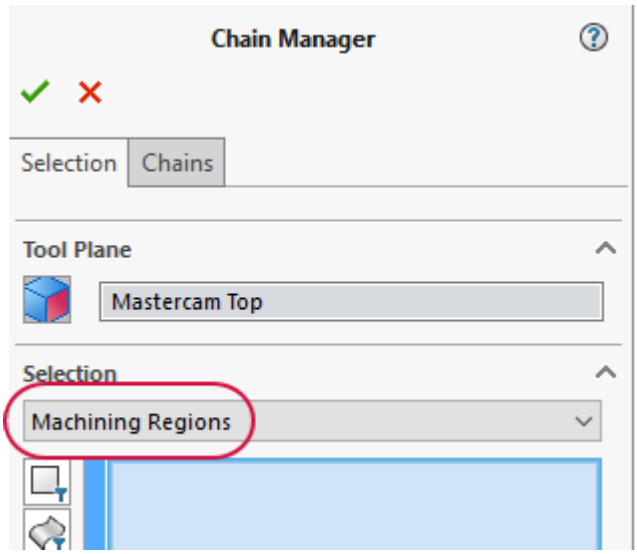
You now create the first Dynamic Mill toolpath to machine the outside of the part.

1. Select **Dynamic Mill** from the **2D HighSpeed** menu on the **Mastercam2019** tab.



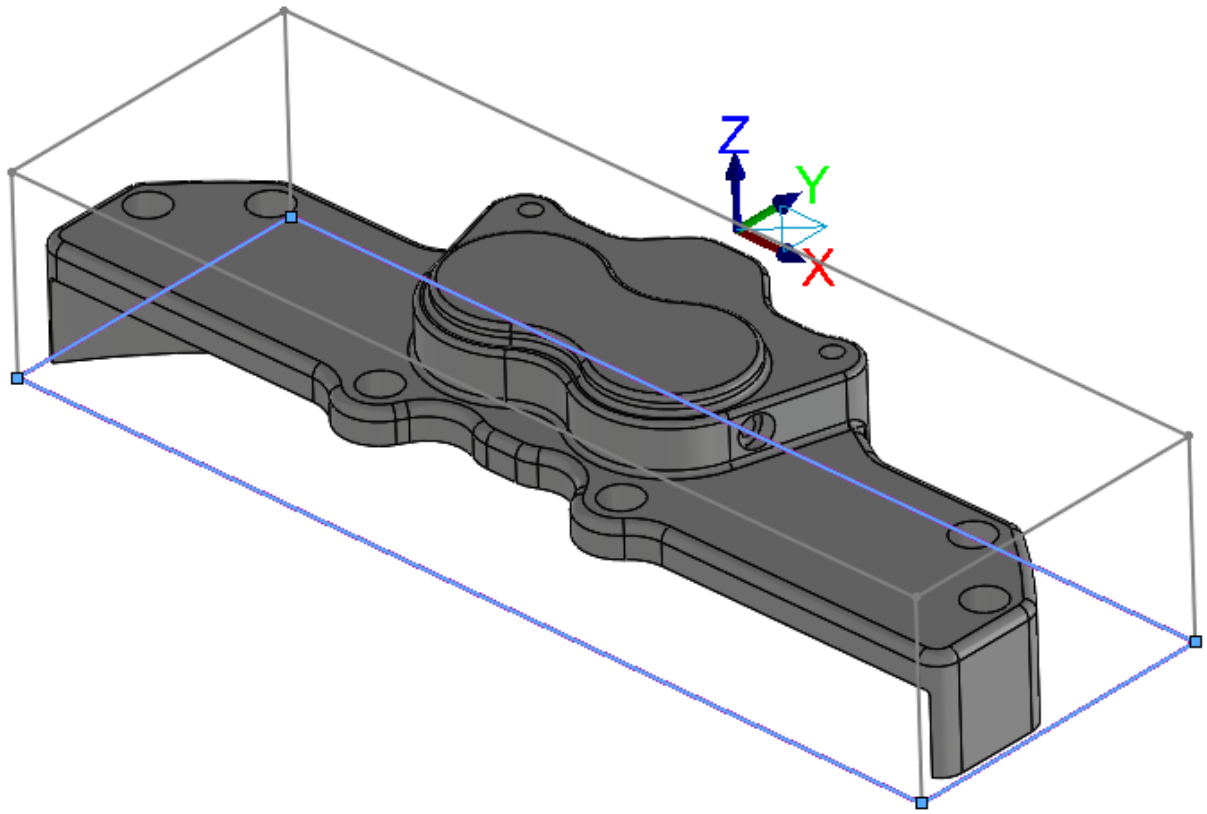

   The Chain Manager displays.

2. Ensure that you are on the **Selection** tab and **Selection** is set to **Machining Regions**.

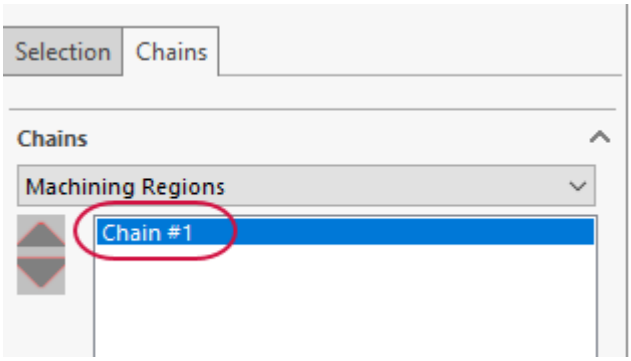



   Machining regions are areas to be cut.

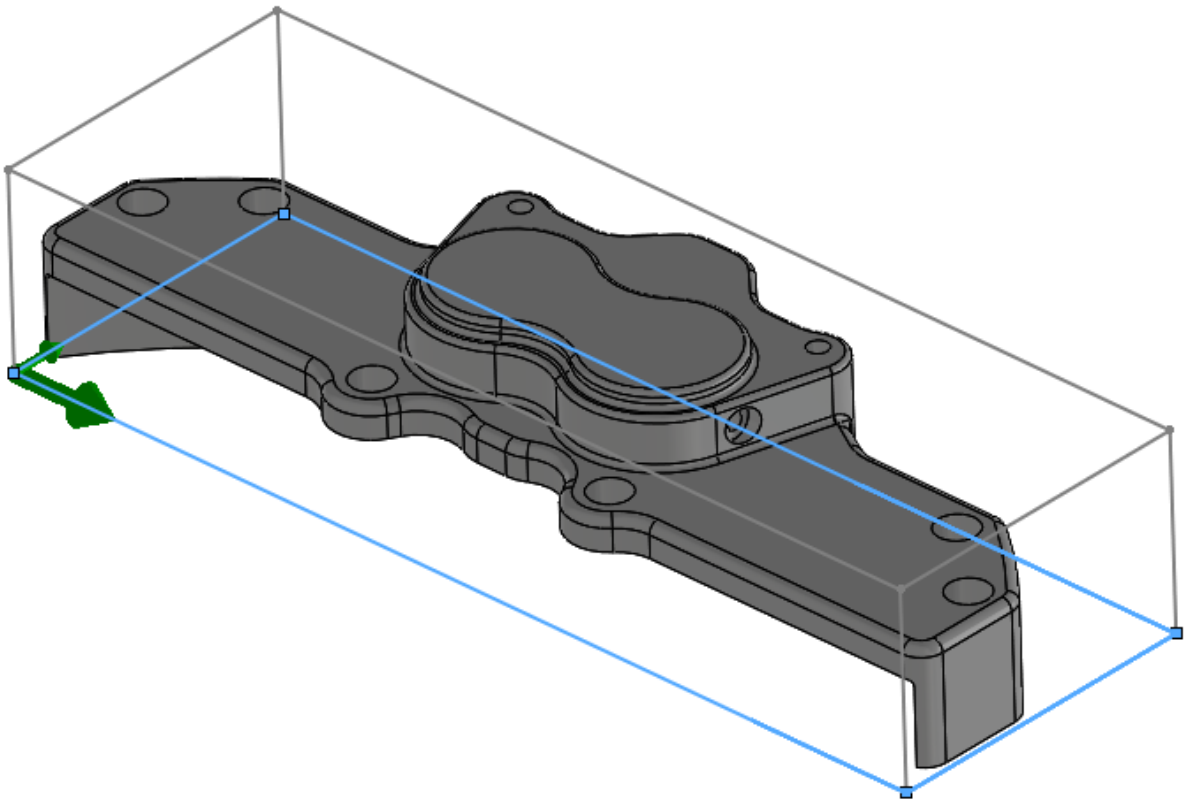
3. Select the bottom stock geometry shown below as the machining region:



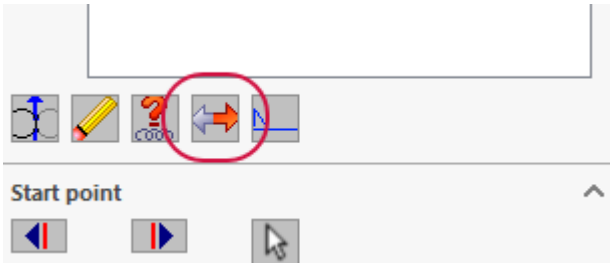

4. Select the **Chains** tab in the Chain Manager.
5. Select **Chain #1** in the chains list.



6. Ensure that the chain direction matches the direction shown below:

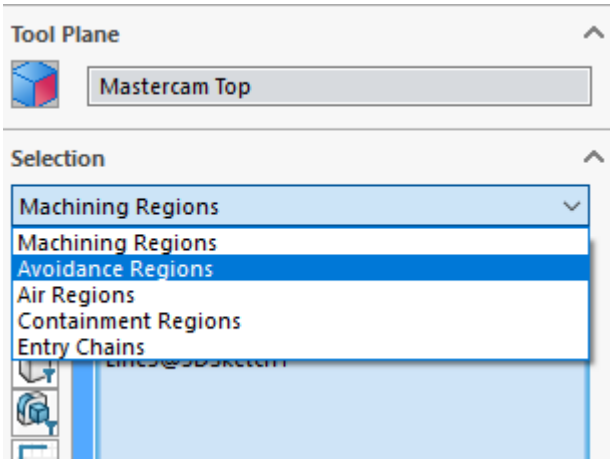


7. Use **Reverse** to reverse the chain if necessary.



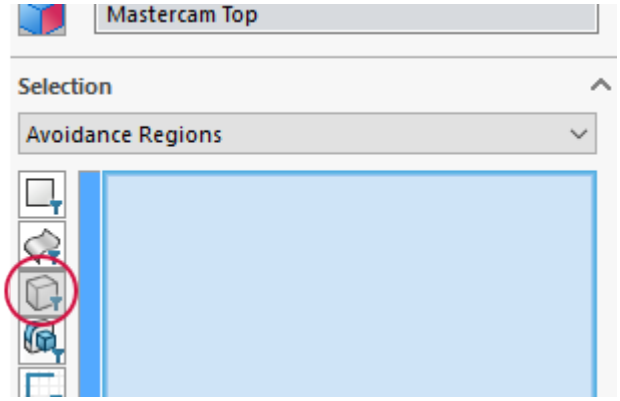

8. Select the **Selection** tab.

9. Select **Avoidance Regions** from the **Selection** drop-down.



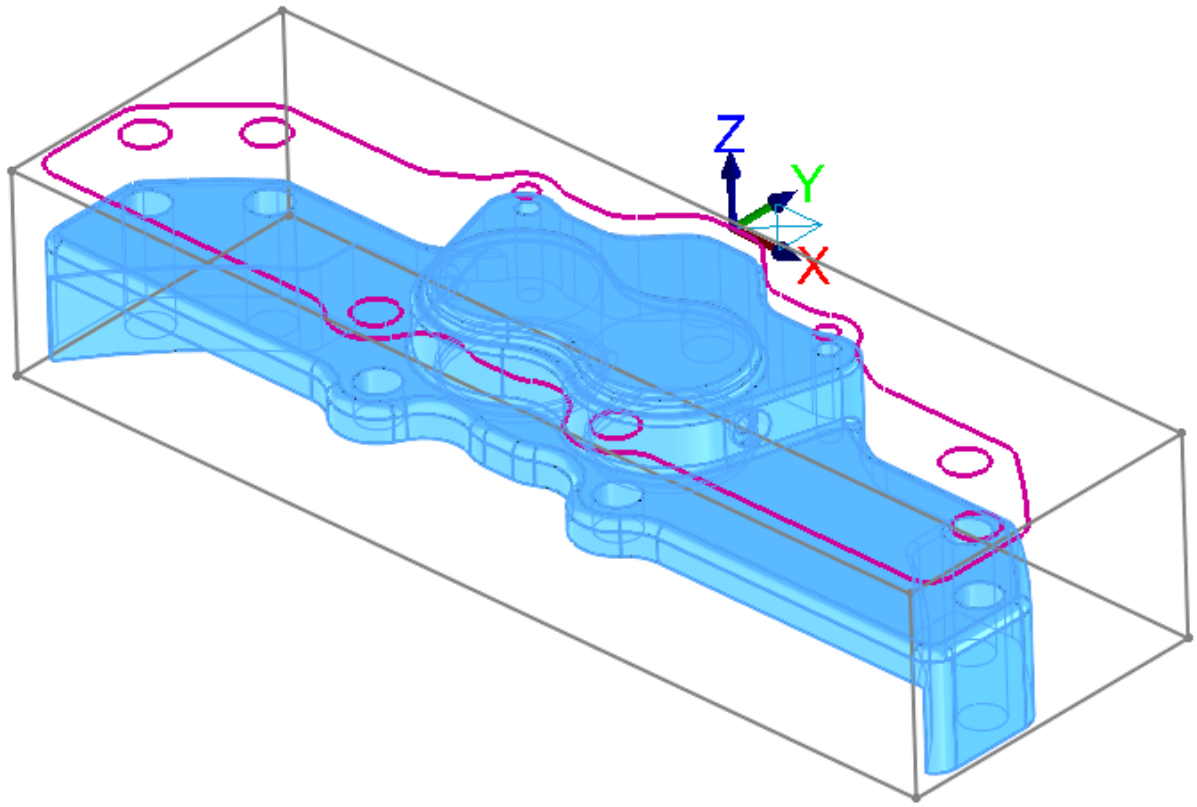

Avoidance regions are areas to avoid during machining. In this case, you want to avoid the part and clear out the area between the stock boundary and the part.

10. Select the **Bodies** filter.



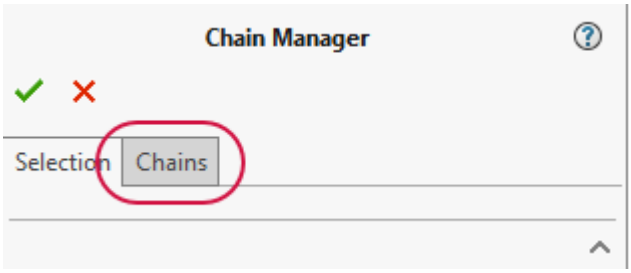

This filters your selections to only bodies.

11. Select the entire part body as shown below:

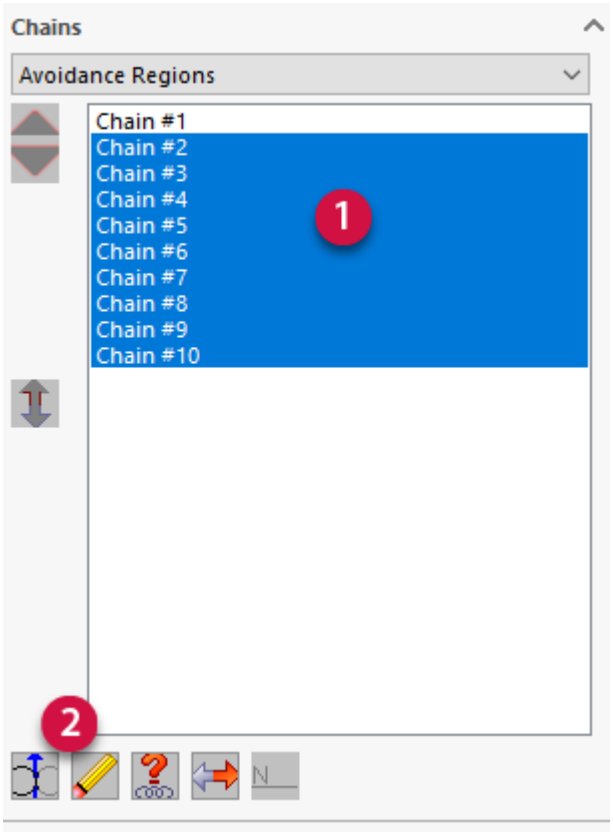



The **Project to Tool Plane** option is automatically turned on. This option projects the selected geometry to the tool plane, so that you can see what geometry you have selected.

12. Select the **Chains** tab.

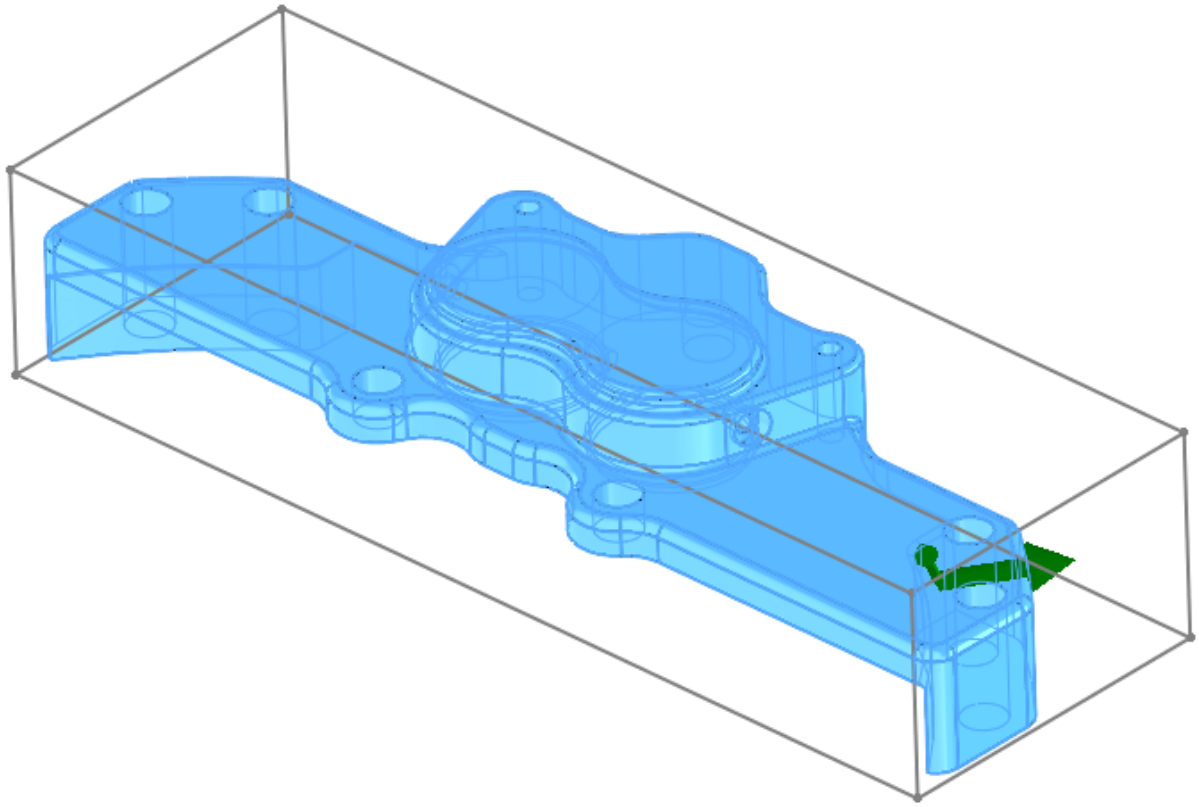


13. In the chains list, delete **Chain #2** - **Chain #10** using **Delete**.




**Chain #1** will be the only chain left. By doing this, you remove all of the internal chains, leaving only the exterior chain.

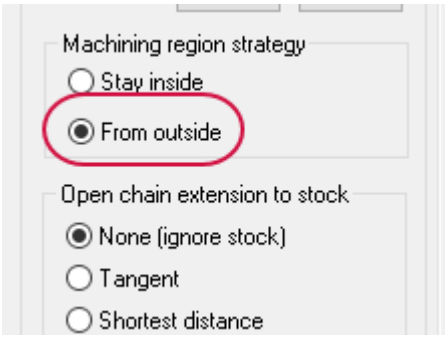
14. Select **Chain #1** in the list. It will display as shown below:



15. Click **OK** in the Chain Manager to accept the chains and begin creating the toolpath.

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

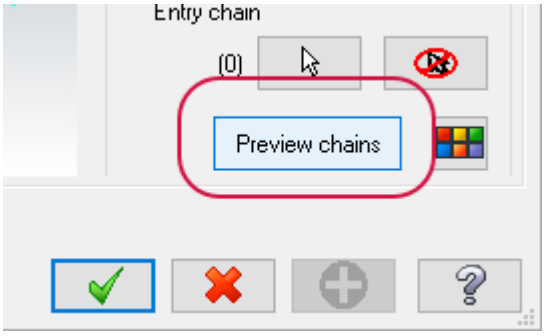
16. On the Toolpath Type page, set the Machining region strategy to **From outside**.



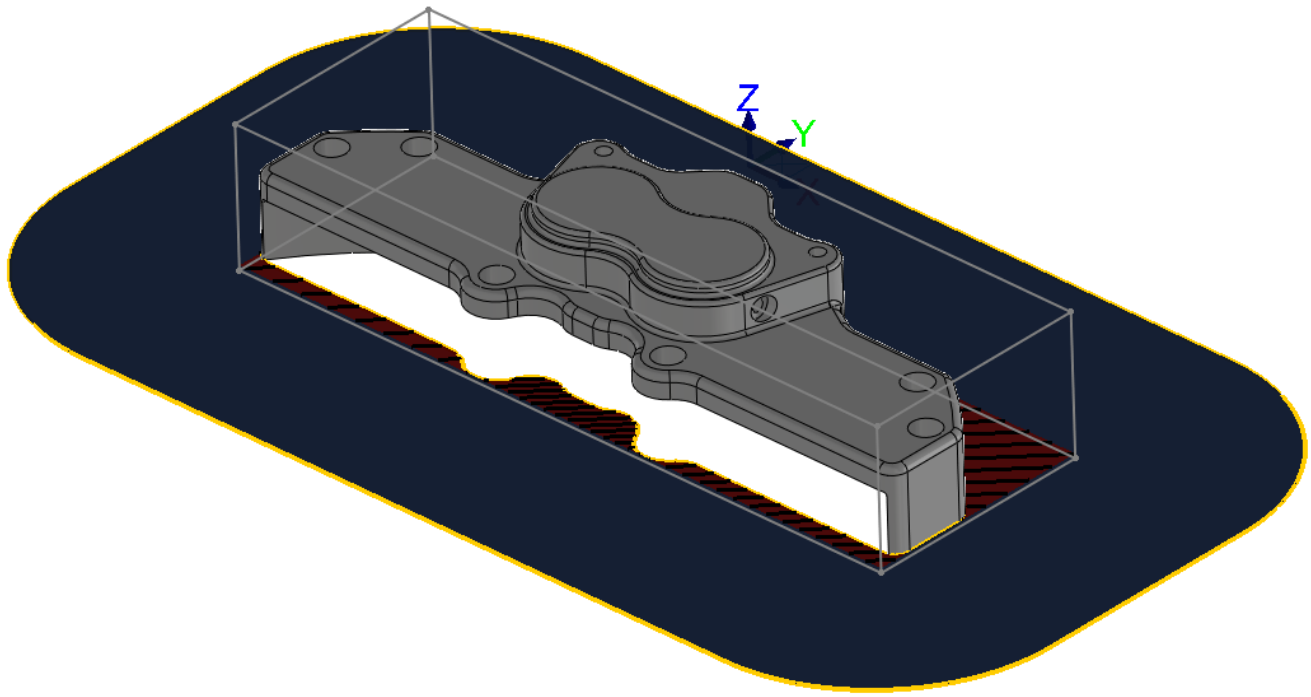

    **From outside** uses free-flowing motion to machine features such as standing bosses or cores. This is the strategy you want to clear the outside of the part.

    Before continuing with the toolpath creation, you will preview your toolpath chains to ensure that you have selected the geometry that you want.

17. Click **Preview chains**.



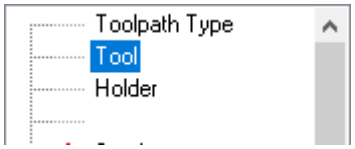

    Your part then displays as shown below:



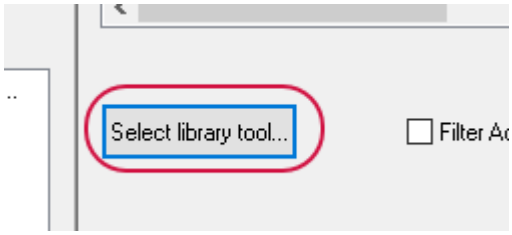
Mastercam for SOLIDWORKS displays your material in the red and black crosshatch, the motion region as blue, and the tool containment as yellow. By using Preview chains, you can determine if that is what you want from the toolpath before entering any parameters.

You can change these colors by selecting the **Color** button or in the **Colors** page of the **System Configuration** dialog box.

18. Click **Preview chains** again to turn off the display.
19. Select the **Tool** page.

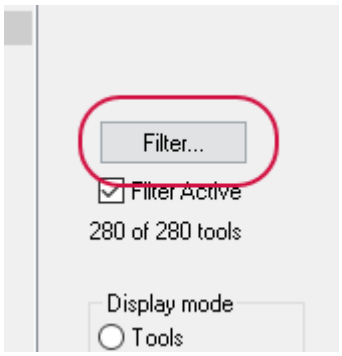



20. Click **Select library tool**.



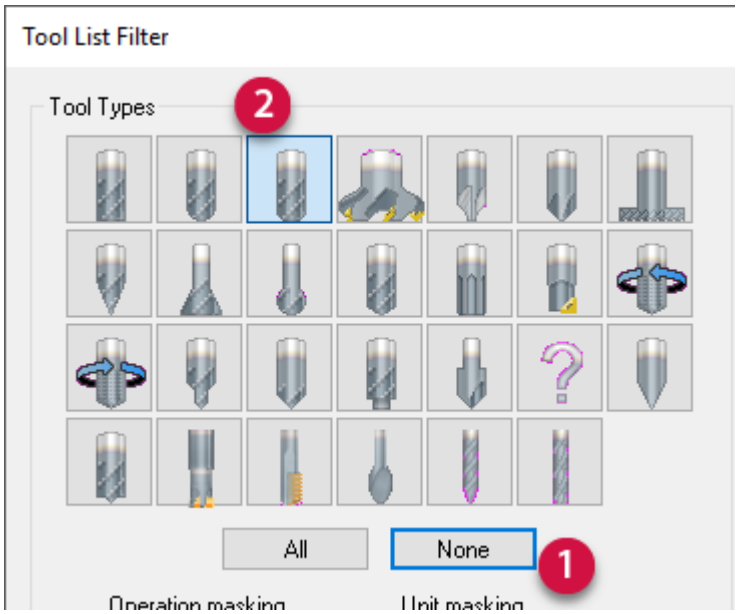

The **Tool Selection** dialog box displays.

21. Select **Filter**.




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

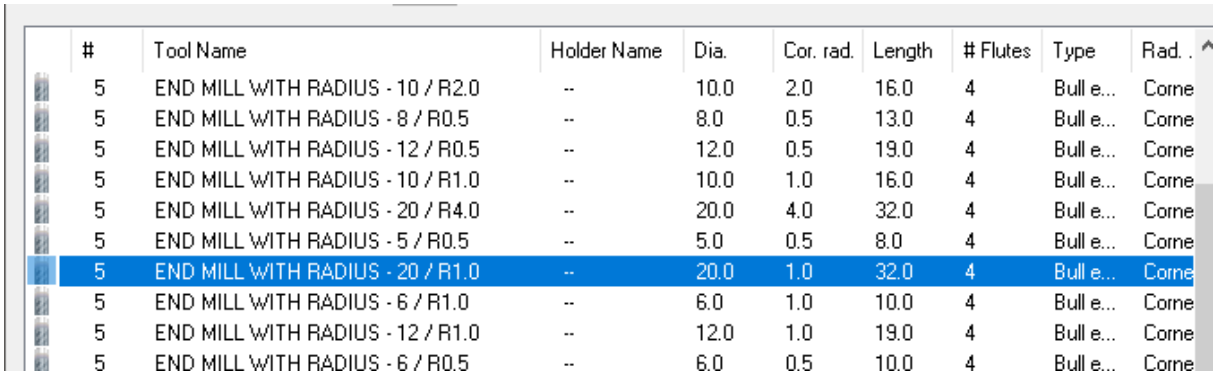
22. Select **None** to remove all filters, and then select **EndMill3 Bull**. This ensures that you are only showing Bull end mills tools.




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

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

24. 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.

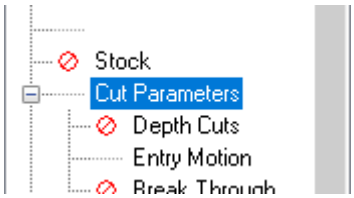


| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Type | Rad. . |
|---|---|---|---|---|---|---|---|---|
| 5 | END MILL WITH RADIUS - 10 / R2.0 | -- | 10.0 | 2.0 | 16.0 | 4 | Bull e... | Corne |
| 5 | END MILL WITH RADIUS - 8 / R0.5 | -- | 8.0 | 0.5 | 13.0 | 4 | Bull e... | Corne |
| 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 |

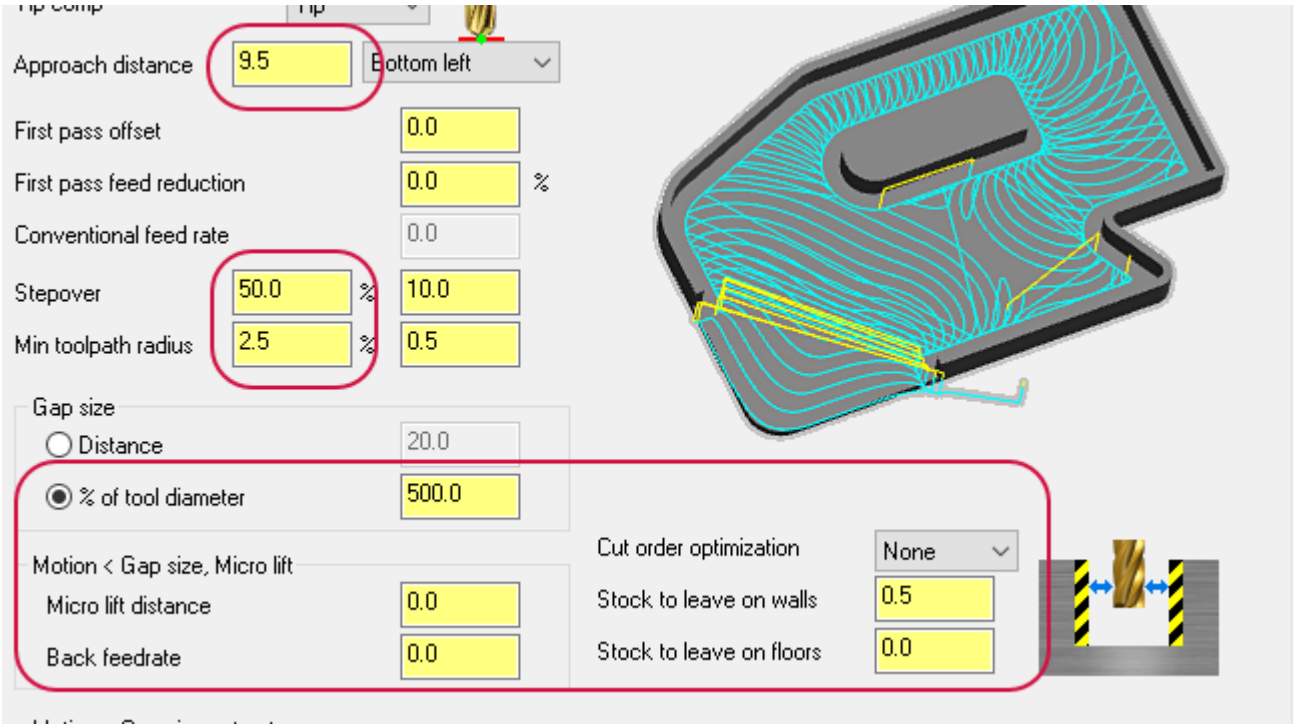
25. Click **OK** in the **Tool Selection** dialog box to add the tool to the toolpath.

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

26. Select the **Cut Parameters** page.

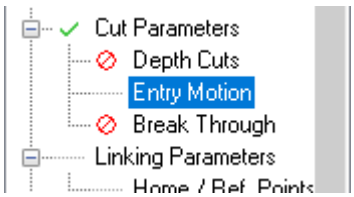


27. Set the following parameters:

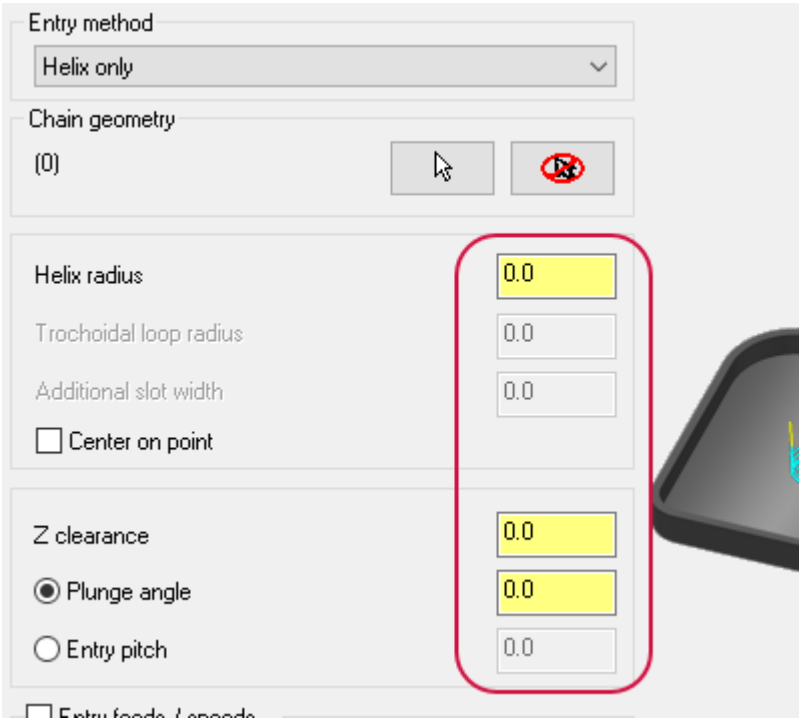



- **Approach distance** to **9.5**. This adds a specified distance to the beginning of the toolpath's first cut.
- **Stepover** to **50.0**. Sets the distance between cutting passes in the X and Y axes.
- **Minimum toolpath radius** to **2.5**. Mastercam uses this radius in combination with the **Micro 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.

28. Select the **Entry Motion** page.



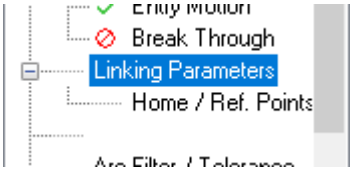
29. Set the following parameters:



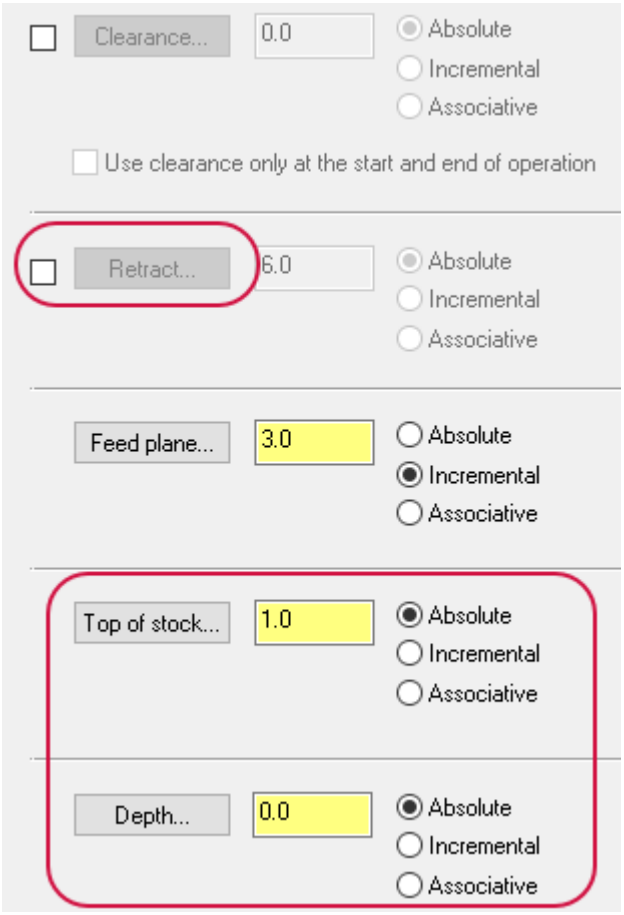

- **Helix radius** to **0.0**. Sets the radius of the entry helix.
- **Z clearance** to **0.0**. Sets an extra height used in the ramping motion down from a top profile.
- **Plunge angle** to **0.0**. Sets the angle of descent for the entry move and determines the pitch.

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

30. Select the **Linking Parameters** page.

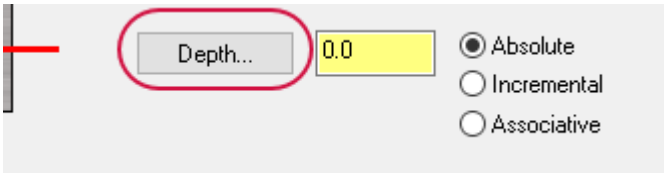


31. Set the following parameters:



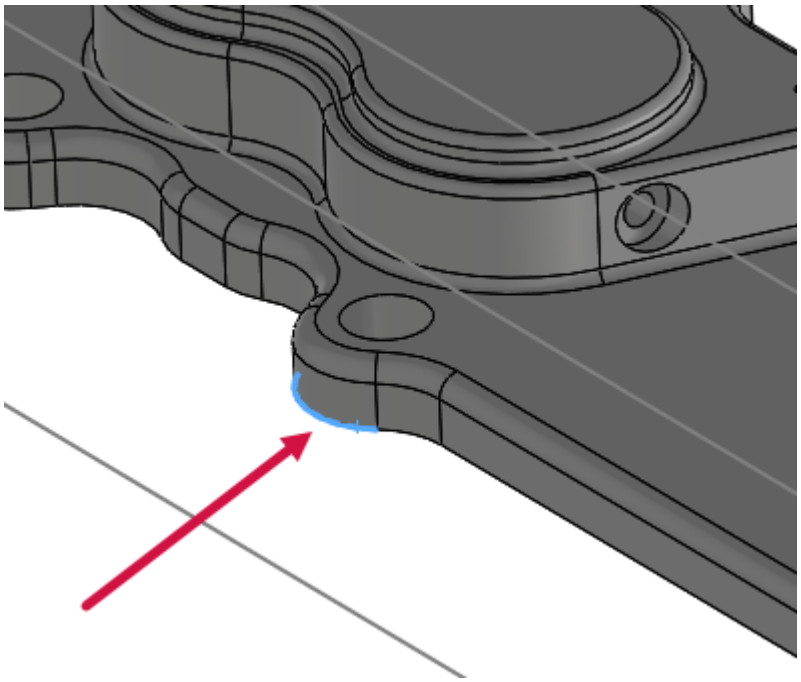

- Deselect **Retract**. The retract distance is the height that the tool moves up to before the next tool pass.
- Change **Top of stock** and **Depth** to **Absolute**. Mastercam measures absolute values from the origin 0,0,0.

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

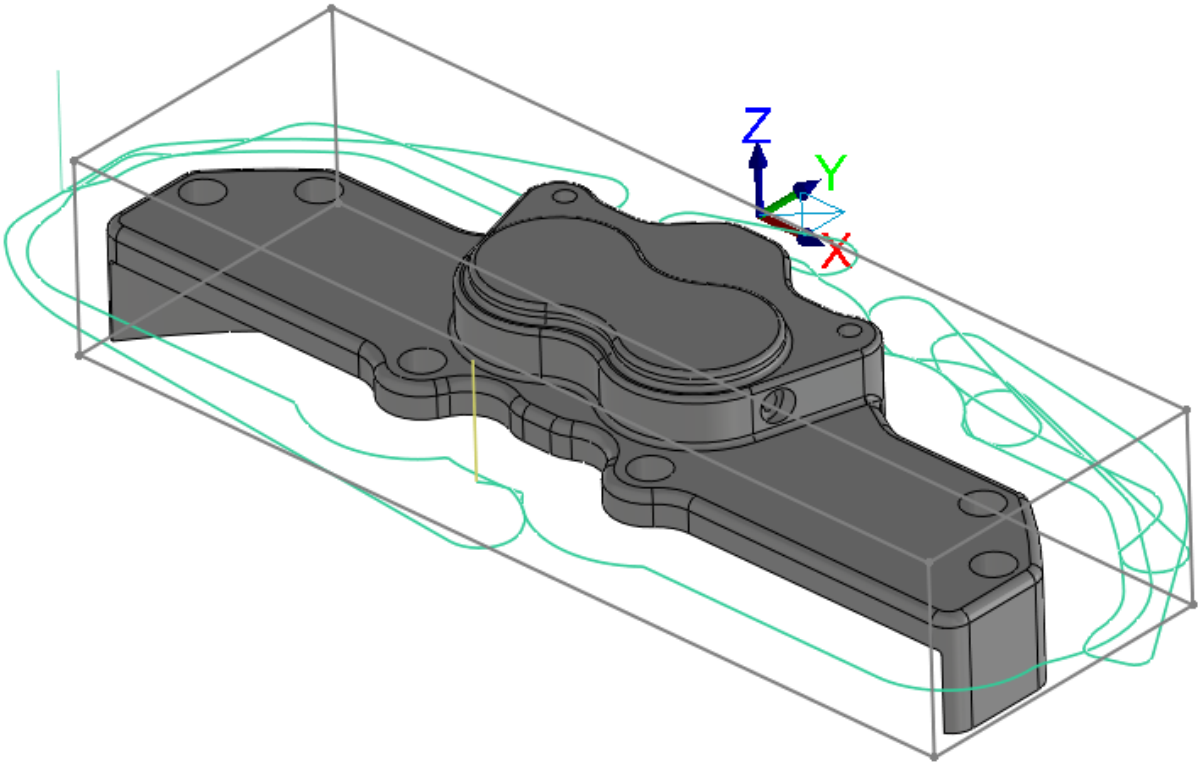



This determines the final machining depth and the lowest depth that the tool descends into the stock. In this case, the machining depth will be in the middle of the part because Dynamic Mill uses the entire tool to cut.

33. Select the edge shown below:



34. Click **OK** to accept the edge as the depth. Your **Depth** should be **-15.24**.
35. Click **OK** in the **2D High Speed Toolpath - Dynamic Mill** dialog box to generate your toolpath.
36. The toolpath will display as shown below:

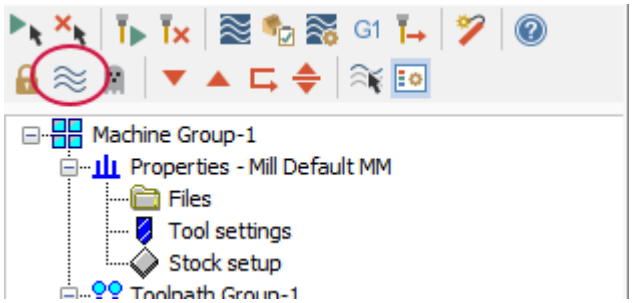



37. Save your part.

## Exercise 4: Creating the second Dynamic Mill toolpath

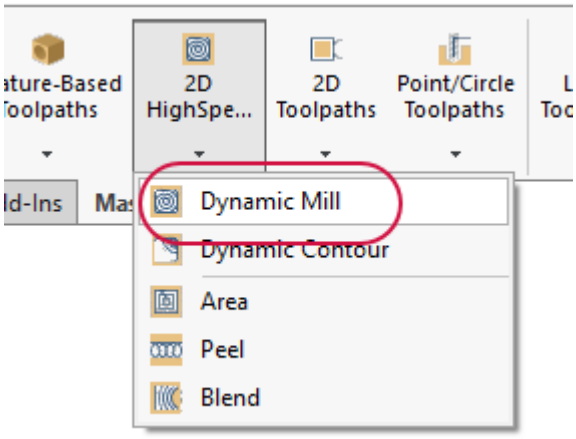
You will now create a second Dynamic Mill toolpath using the **Rest material** option.

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



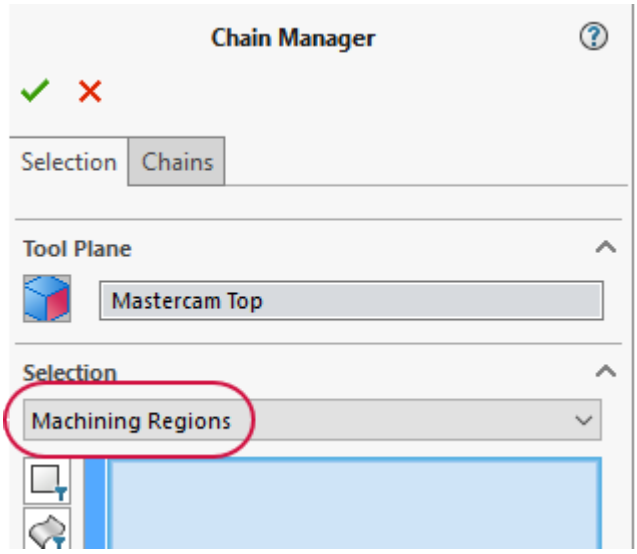

   This will hide the display of the first toolpath and keep your graphics view clean while you create another toolpath.

2. Select **Dynamic Mill** from the **2D HighSpeed** menu on the **Mastercam2019** tab.

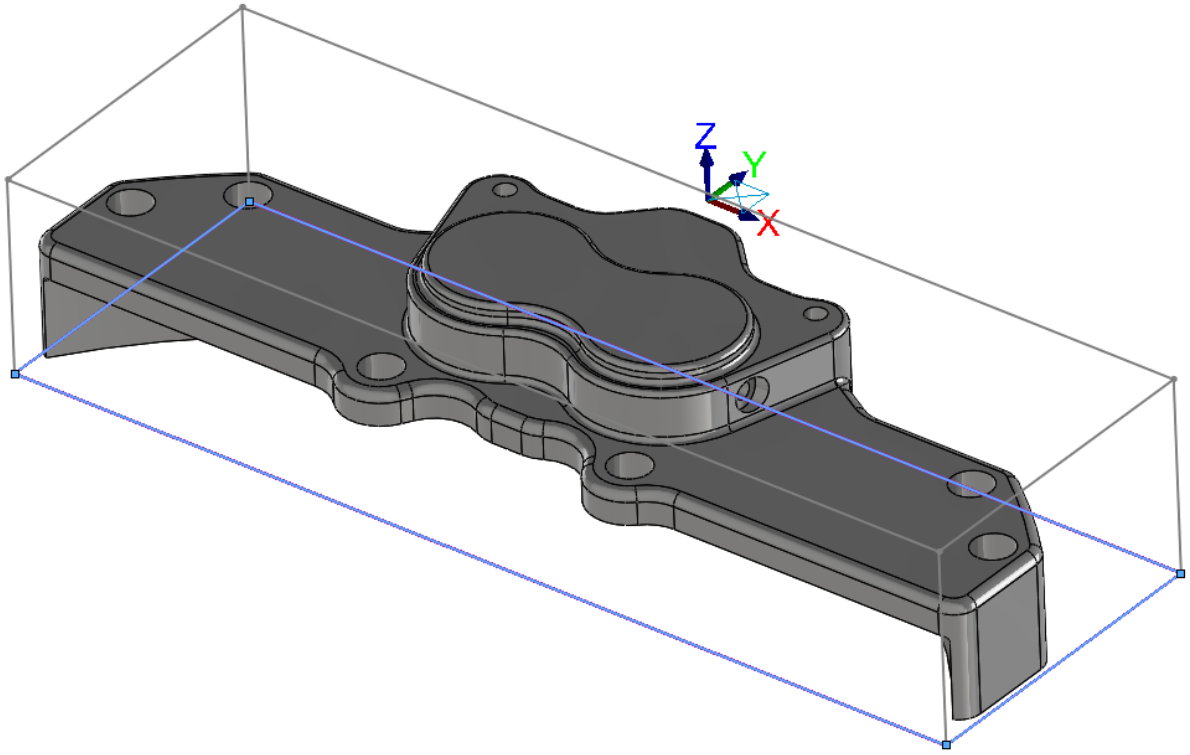



   The Chain Manager displays.

3. Ensure that you are on the **Selection** tab and **Selection** is set to **Machining Regions**.



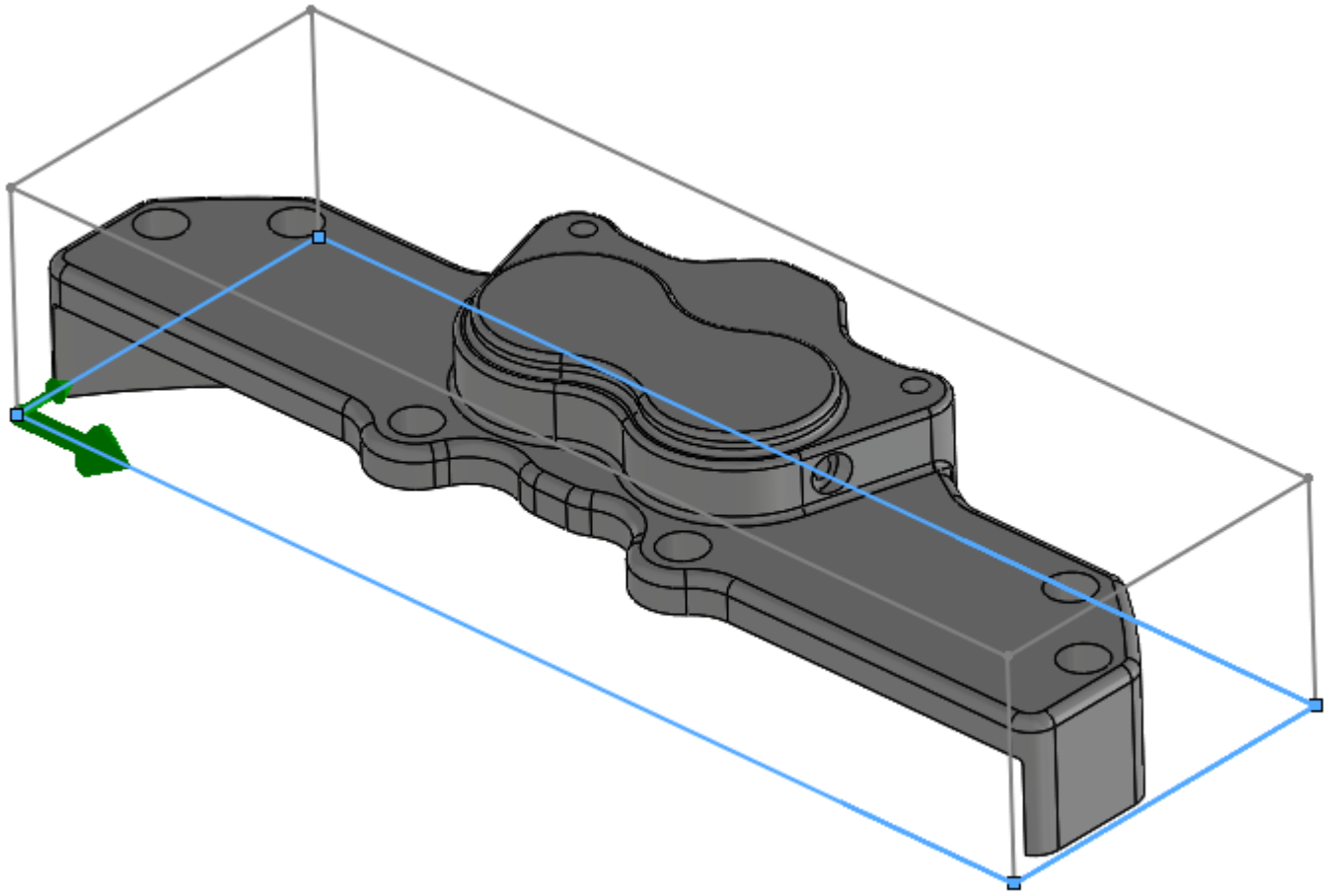
4. Select the geometry shown below as the machining region.



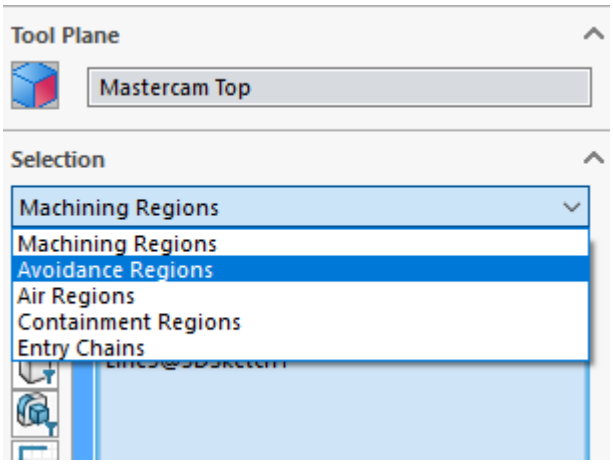

The selection is the same geometry used in the first Dynamic Mill toolpath.

5. Select the **Chains** tab and select **Chain #1**.

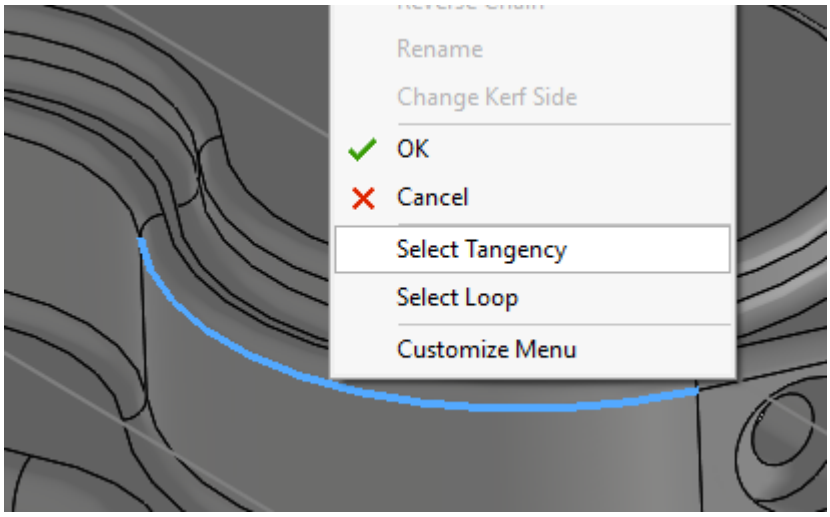
6. Ensure that the chain is going in the direction shown below. If necessary, use the **Reverse** button.



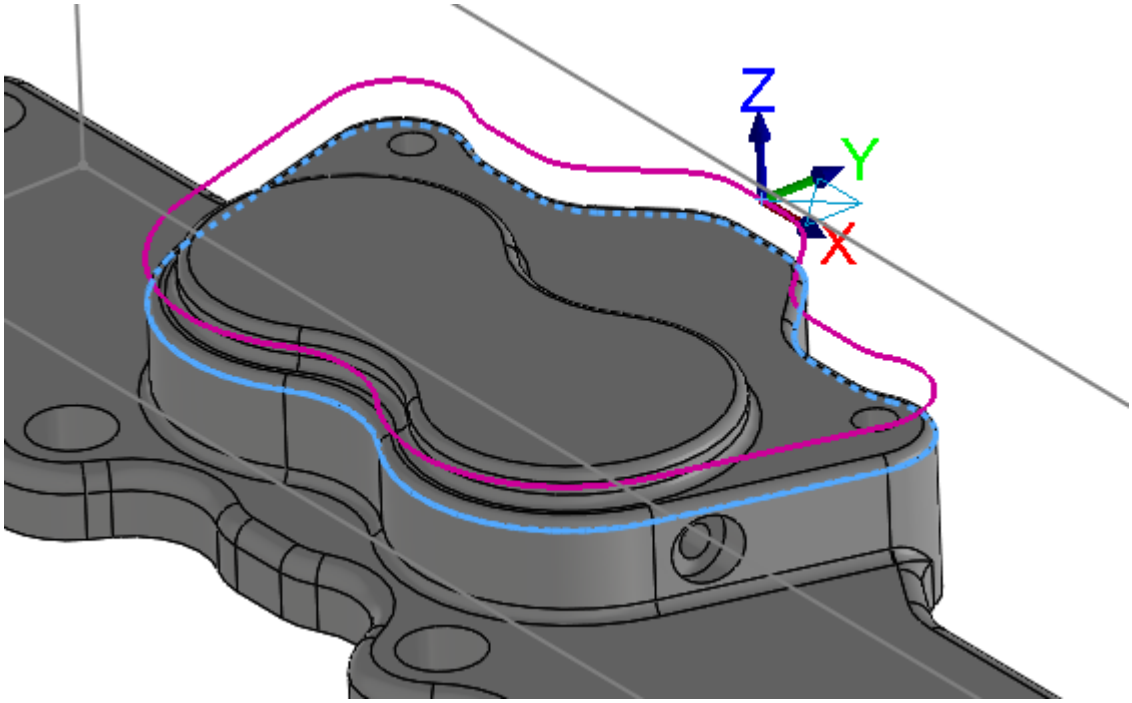
7. Select the **Selection** tab.
8. Select **Avoidance Regions** from the **Selection** drop-down.



9. Right-click on the edge shown below and choose **Select Tangency**:



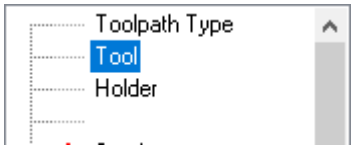

**Select Tangency** automatically selects the same type of entities that are tangent to your initial selection of an edge, curve, or face. The selected edges display as shown below:



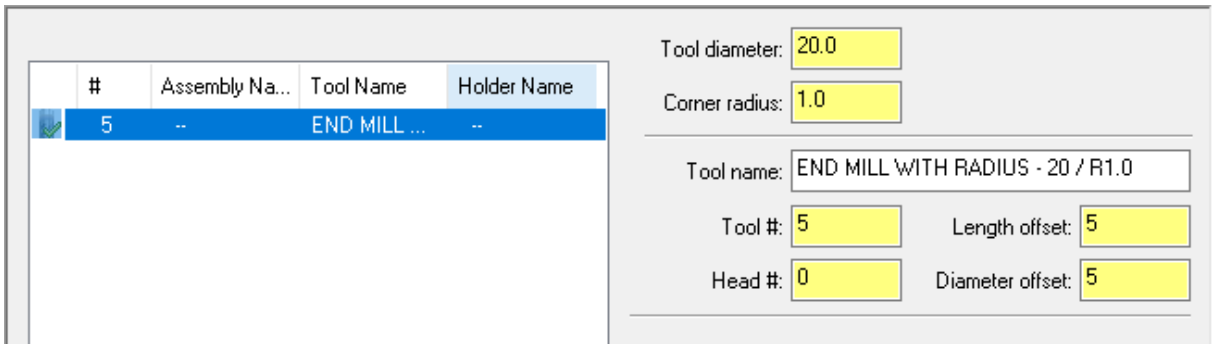

10. Click **OK** in the Chain Manager to accept these chains.

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

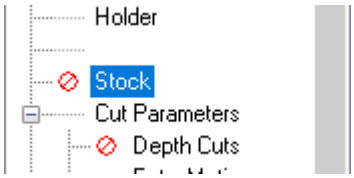
11. Select the **Tool** page.



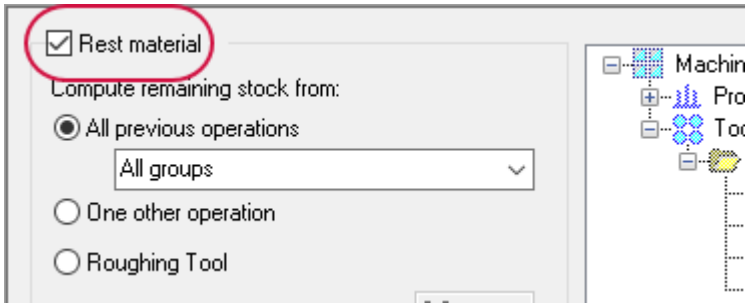
12. In the Tool List, select the tool that was used for the previous Dynamic Mill toolpath, which is **END MILL WITH RADIUS - 20/R1.0.**




13. Select the **Stock** page.

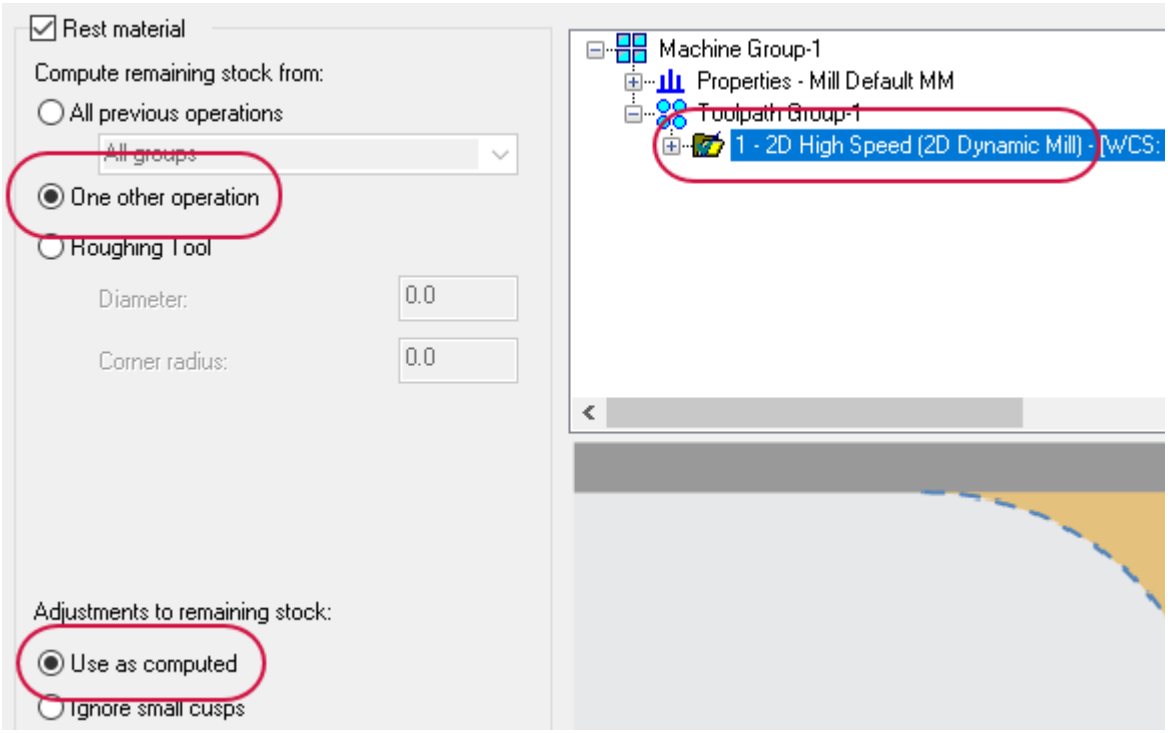



14. Select **Rest material**.



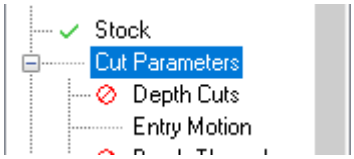

This enables the page and tells Mastercam to calculate the cutting passes based on the remaining stock.

15. Set the following parameters:

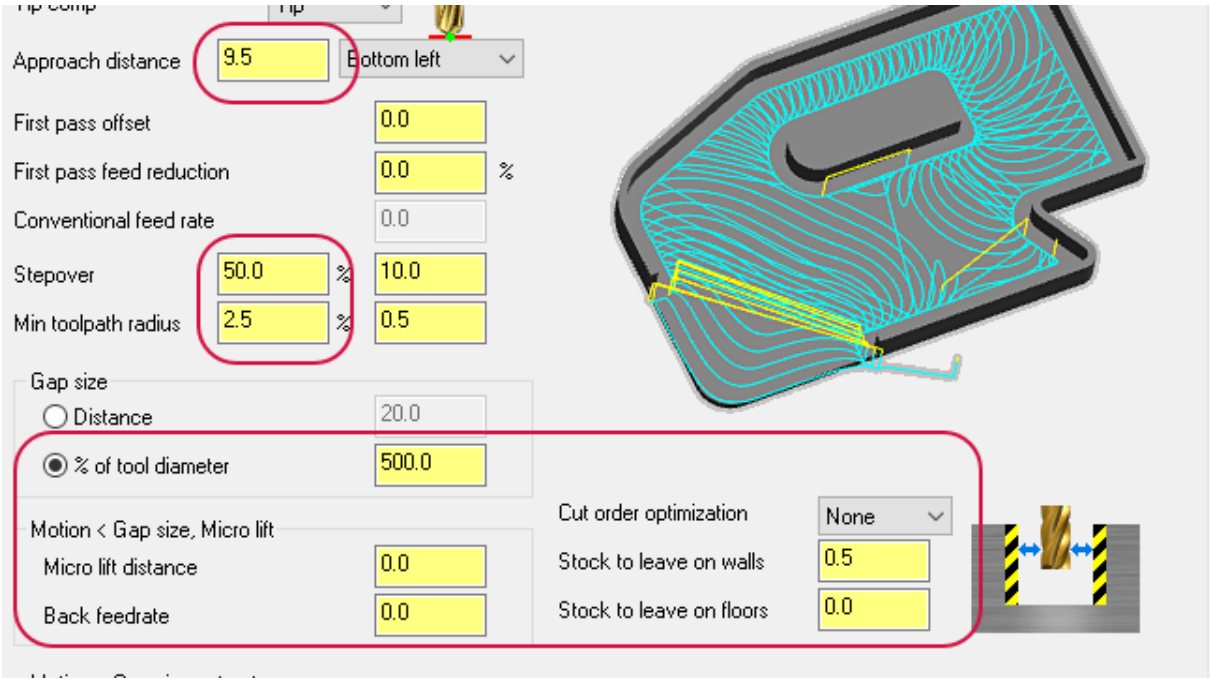



- Set **Compute remaining stock from** to **One other operation**. This calculates the remaining stock from a single operation.
- In the **Operation List**, select the **2D High Speed (2D Dynamic Mill)** toolpath.
- Set **Adjustments to remaining stock** to **Use as computed**. This option makes no adjustments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

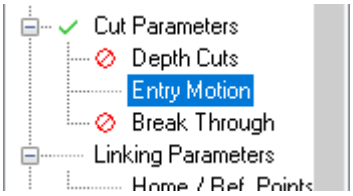
16. Select the **Cut Parameters** page.



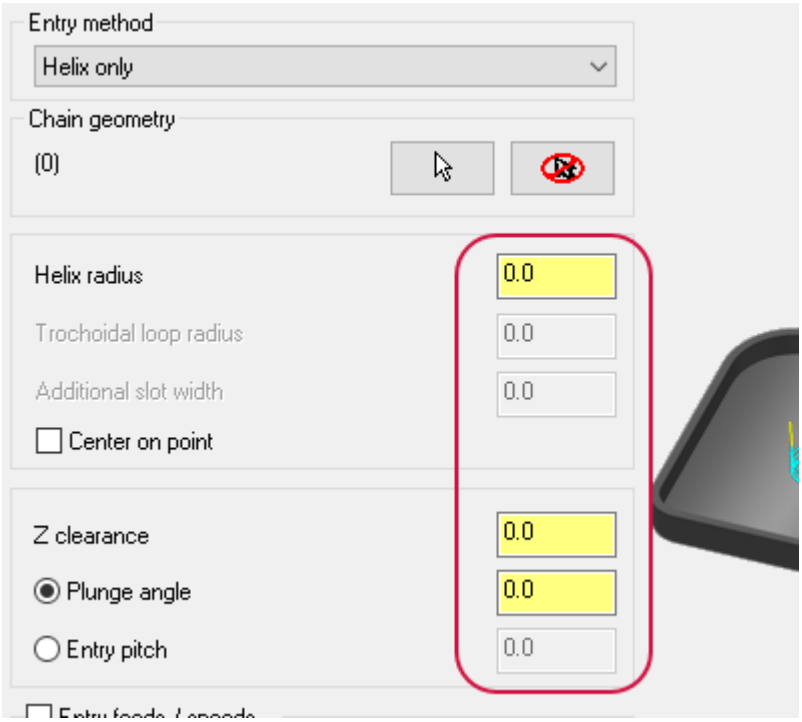
17. The parameters are the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



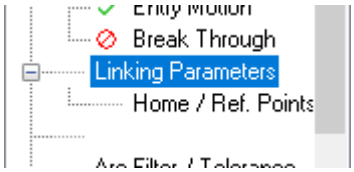

18. Select the **Entry Motion** page.



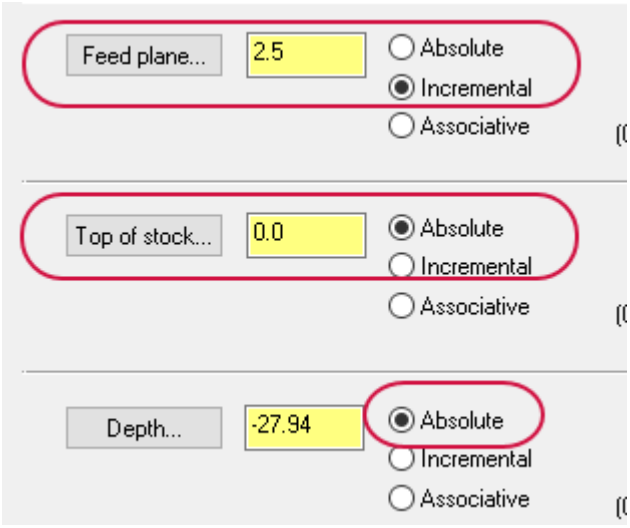

19. The parameters are also the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



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

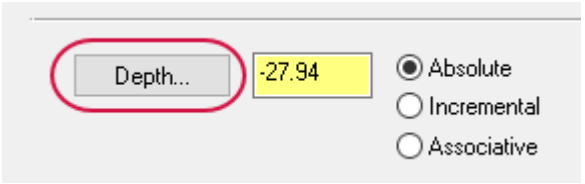



21. Set the following parameters:

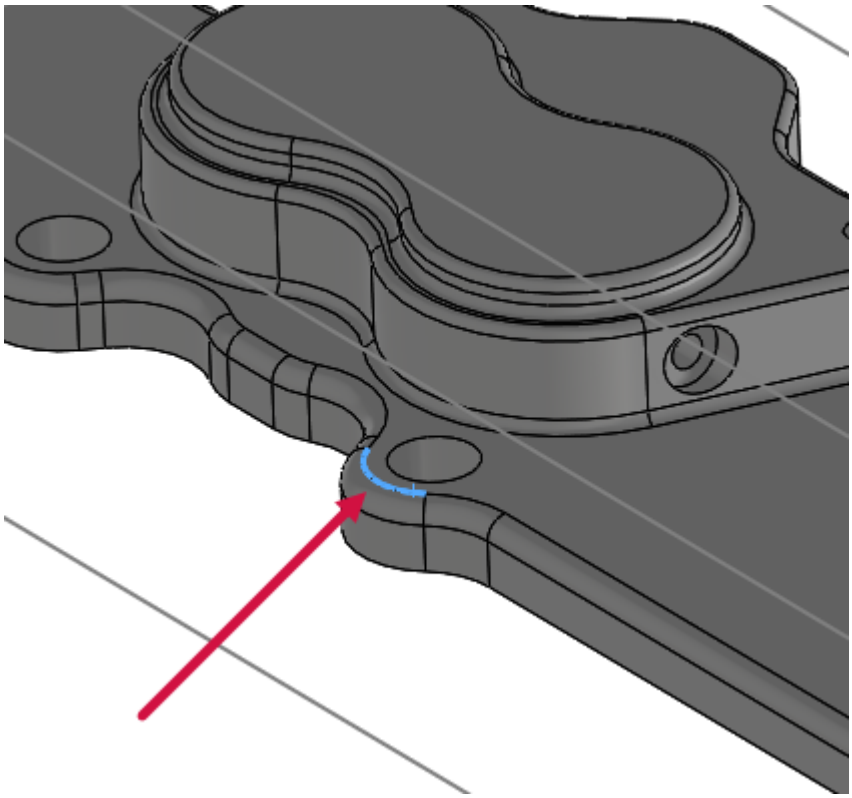



- **Feed plane** to **2.5**. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- Ensure that **Feed plane** is set to **Incremental**.
- **Top of Stock** to **0.0**. Sets the top of the stock.
- Ensure that **Top of stock** and **Depth** are set to **Absolute**.

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

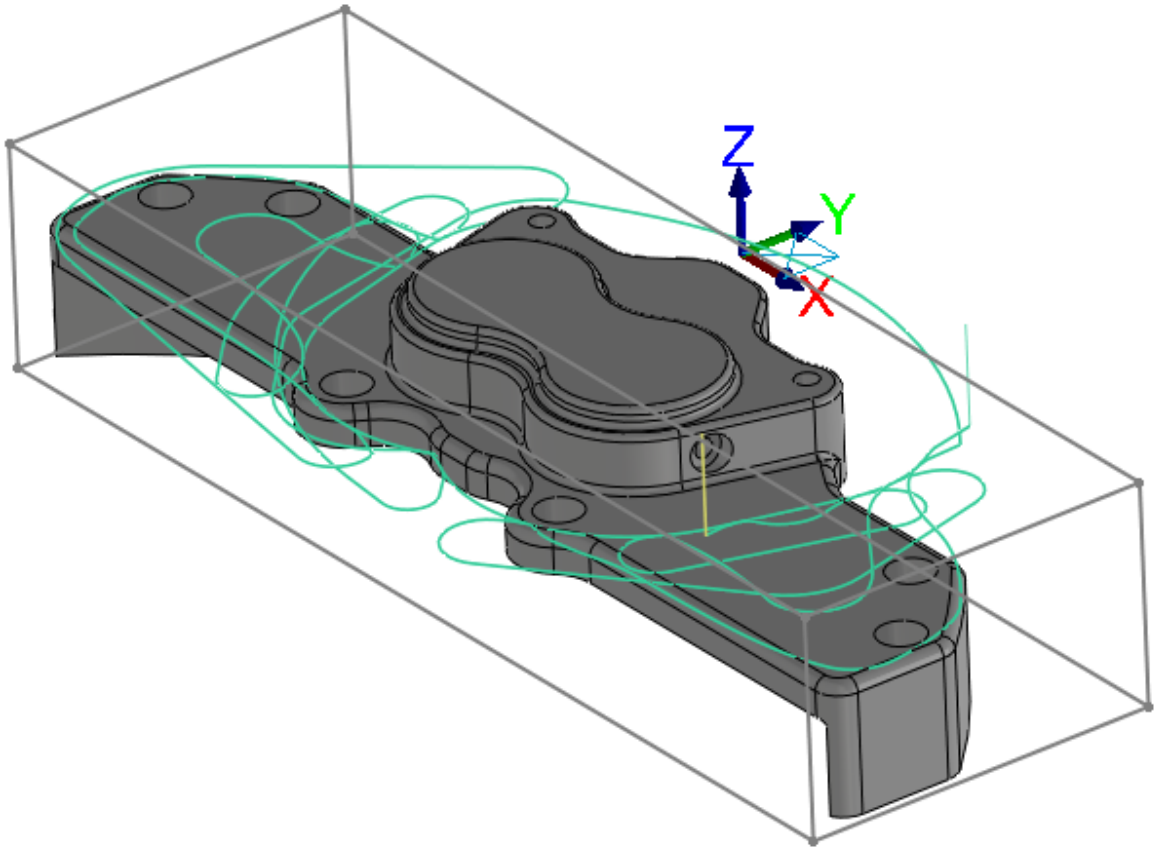


23. Select the edge shown below:



24. Click **OK** to 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:



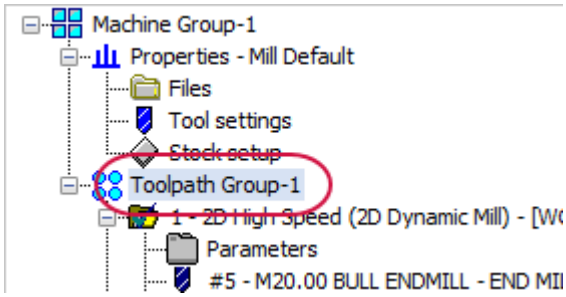

You will notice that the toolpath clears out the top of the part while avoiding the large feature that was selected as the avoidance region.

27. Save your part.

## Exercise 5: Verifying the toolpaths

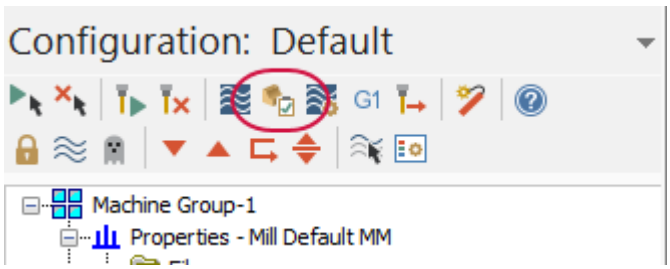
You will now verify both Dynamic Mill toolpaths with Mastercam Simulator. Mastercam Simulator allows you to verify your toolpaths by using solid models to simulate part machining against a selected stock definition. Verify creates a representation of the surface finish and shows collisions, if any exist.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

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



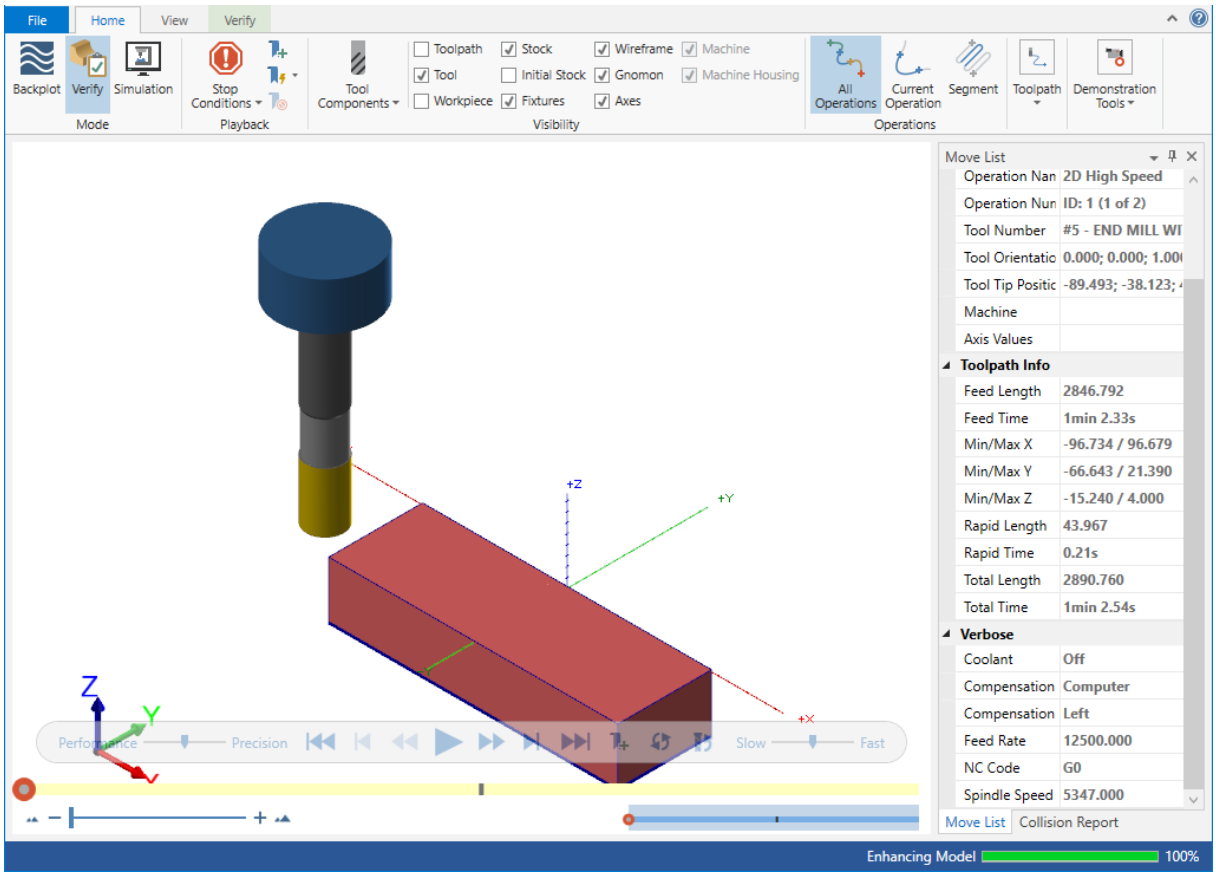

   This will select both Dynamic Mill toolpaths.

2. Select **Verify selected operations**.

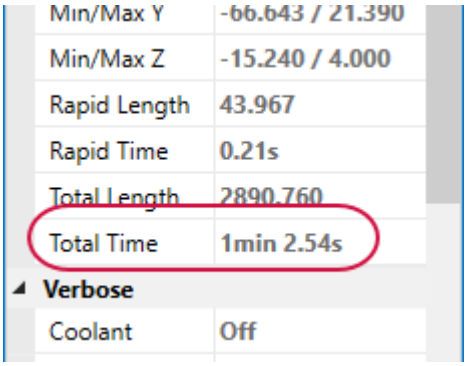



   Mastercam Simulator displays.

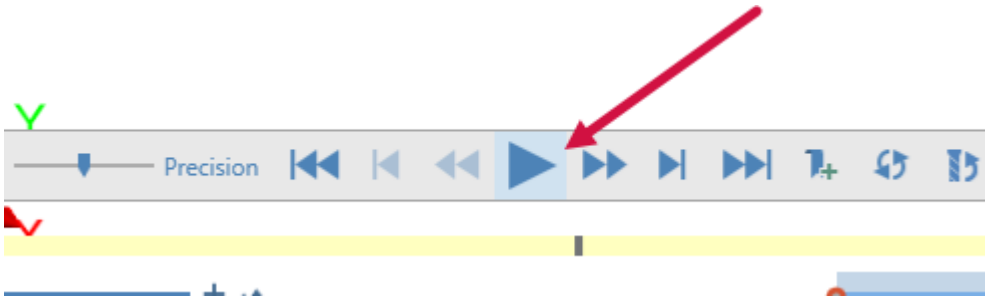
3. Right-click in the graphics view and select **Fit** and **Isometric** from the menu. Mastercam Simulator should match the image shown below.




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.




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.

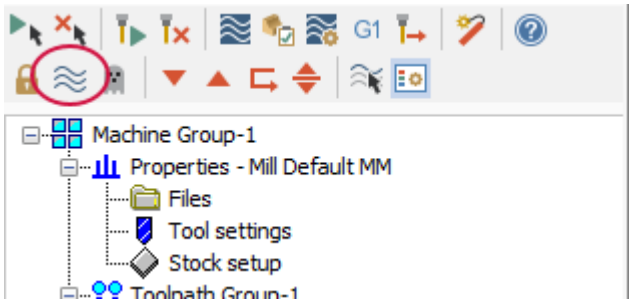
For this lesson, you will create a Face toolpath on the top of the part from the Dynamic Mill lesson.

## Lesson Goals

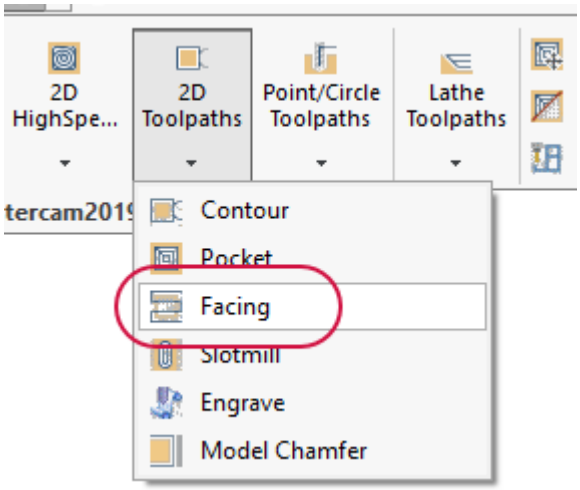
- Create a Face toolpath
- Preview the toolpath before it has been created
- Verify the created toolpath

## 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.

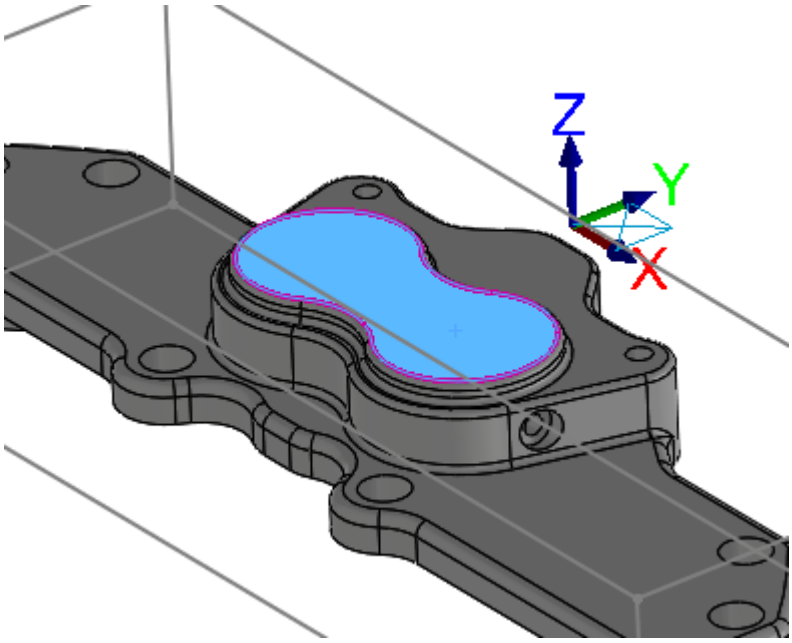



3. Save the part as `FaceMill-xxx`, replacing `xxx` with your initials.
4. Select **2D Toolpaths**, **Facing** from the **Mastercam2019** tab.




The Chain Manager displays.

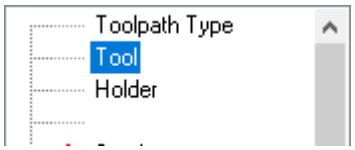
5. Select the face shown below:



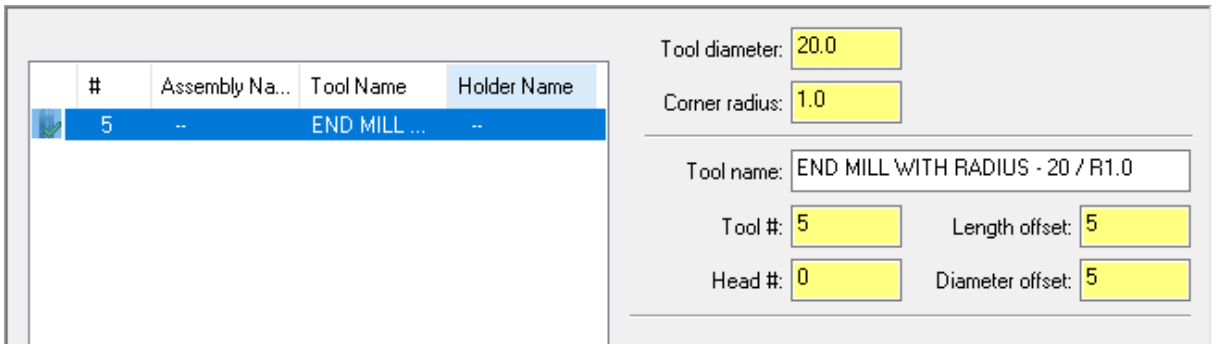

6. Click **OK** in the Chain Manager.

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

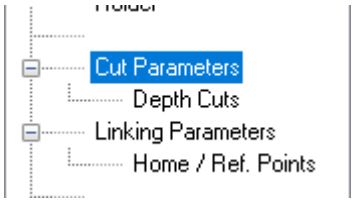
7. Select the **Tool** page.



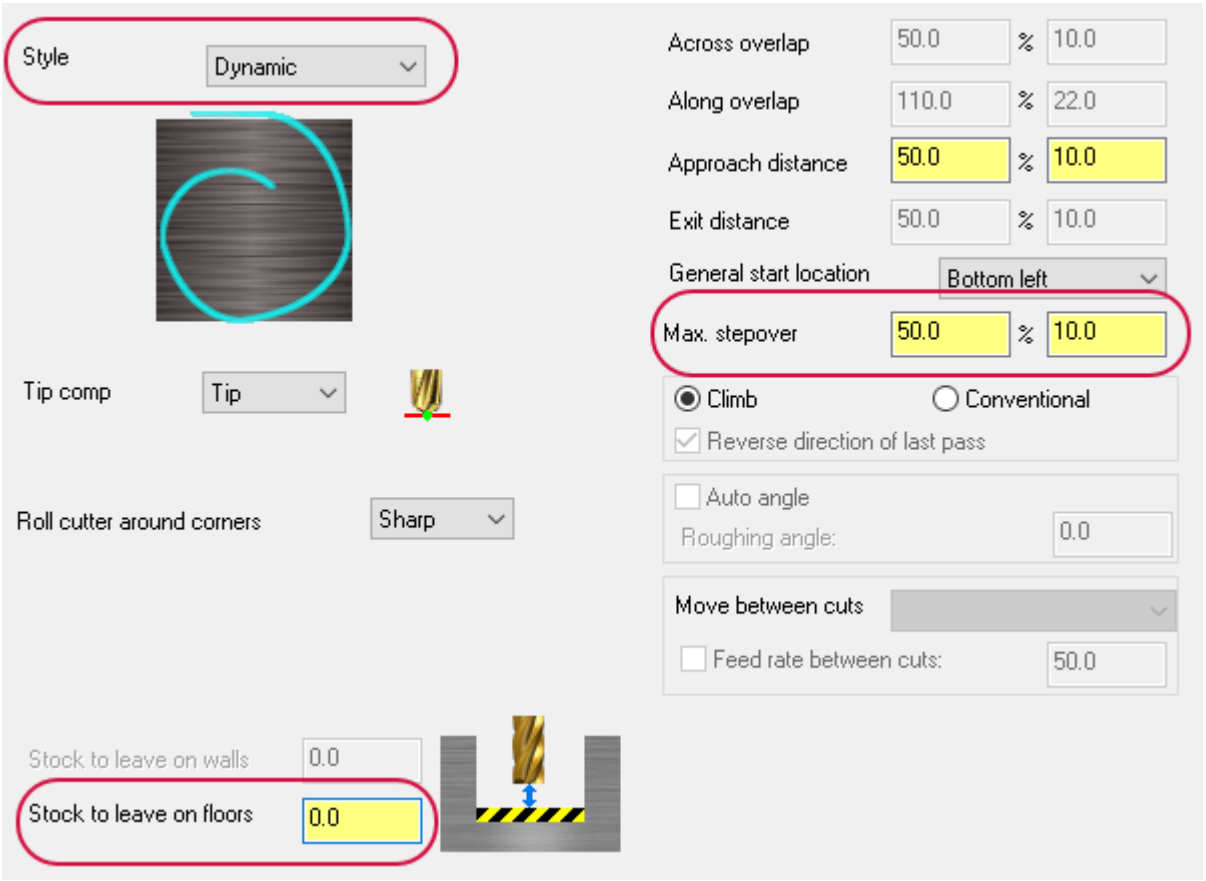

8. Select the **END MILL WITH RADIUS - 20 / R 1.0**, which was used in the previous two Dynamic Mill toolpaths. If necessary, expand the **Tool Name** column to see the entire tool name.




9. Select the **Cut Parameters** page.

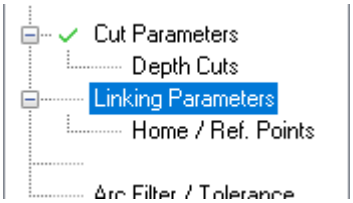


10. Set the following parameters:

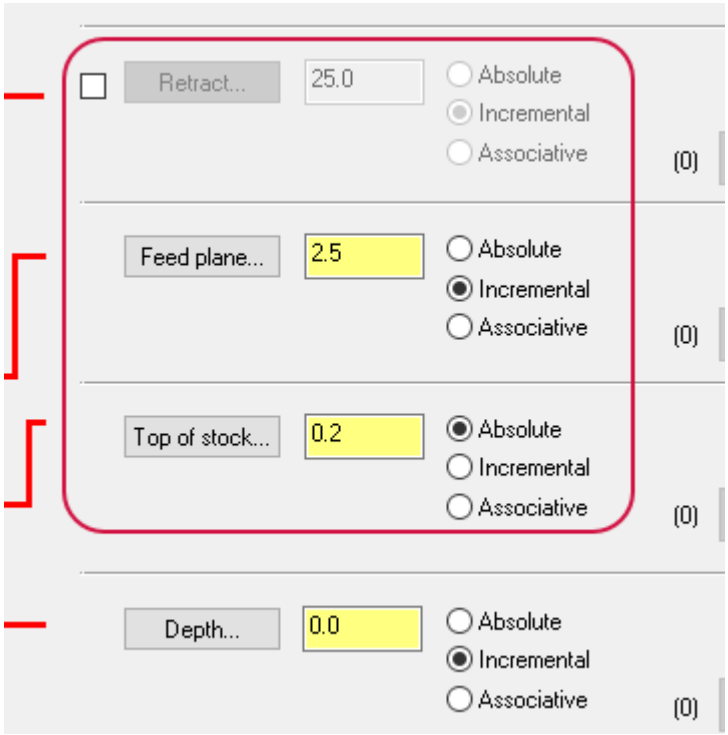



- **Style** to **Dynamic**. Creates a smooth controlled motion that cuts from the outside to the inside, maintaining a constant load on the tool with minimal entries and exits.
- **Stock to leave on floors** to **0.0**. Sets the amount of stock to leave on the floors during machining.
- **Maximum stepover**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

11. Select the **Linking Parameters** page.



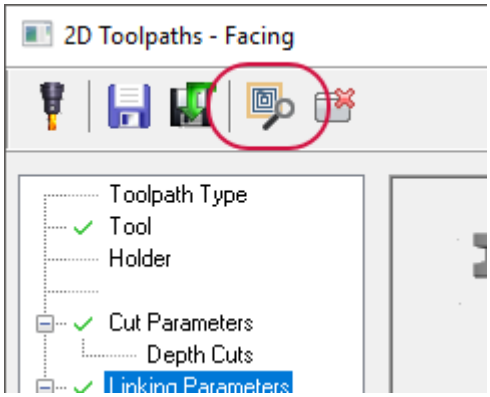
12. Set the following parameters:



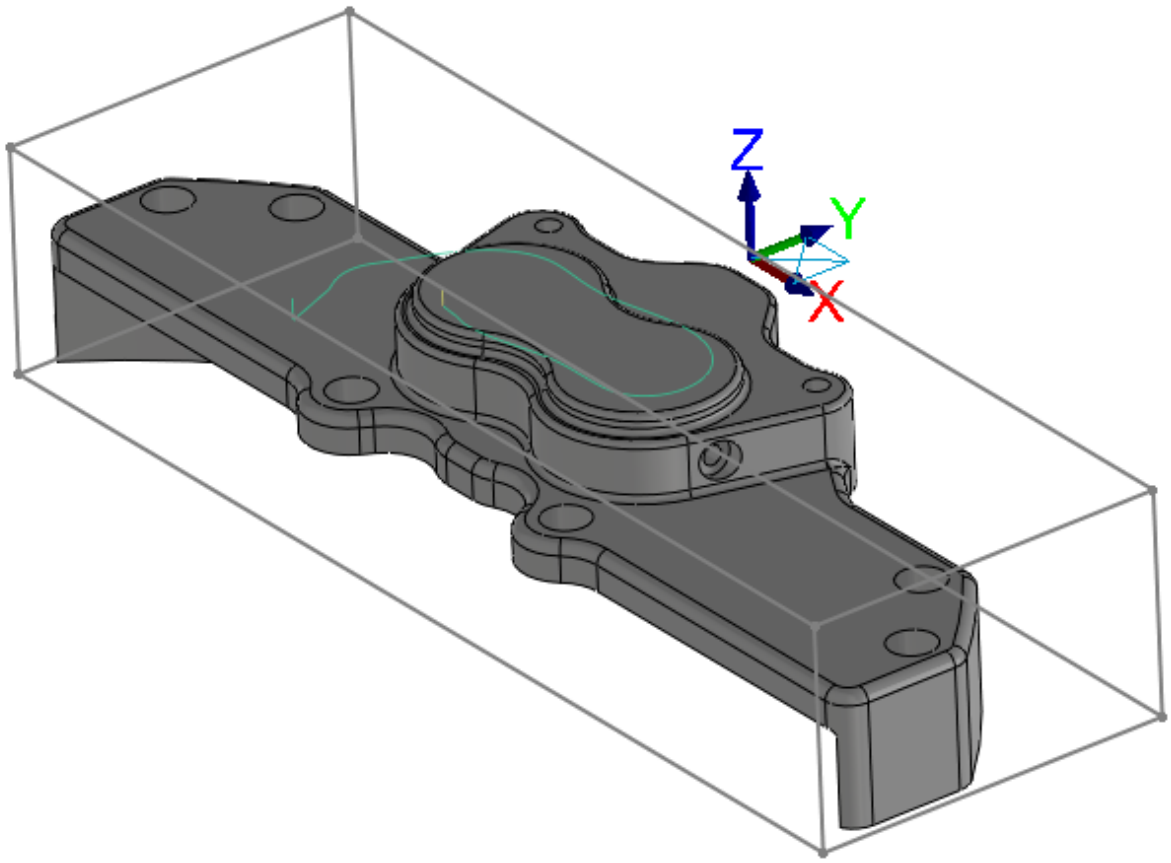

- Deselect **Retract**. Sets the height that the tool moves up to before the next tool pass.
- Set **Feed plane** to **2.5**. Sets the height that the tool rapids before changing to the plunge rate to enter the part.
- Set **Top of stock** to **0.2**. Sets the height of the material in the Z axis.

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.

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



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



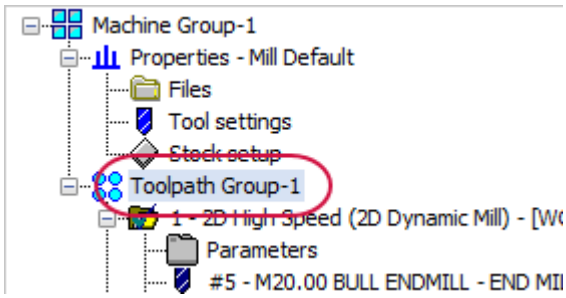

15. Click **OK** in the **2D Toolpaths - Facing** dialog box to close and generate the toolpath.
16. Save your part.

## Exercise 2: Verifying the toolpaths

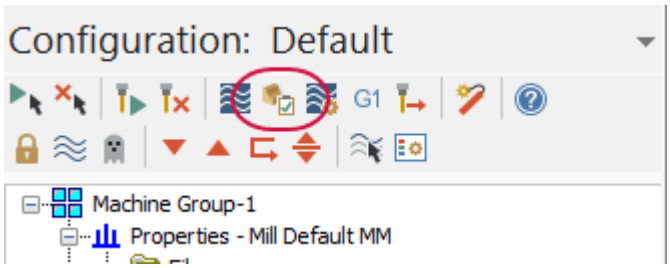
You now verify the two Dynamic Mill toolpaths created from the previous chapter and the Face toolpath together.

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

   This selects both Dynamic Mill toolpaths and the Facing toolpath.

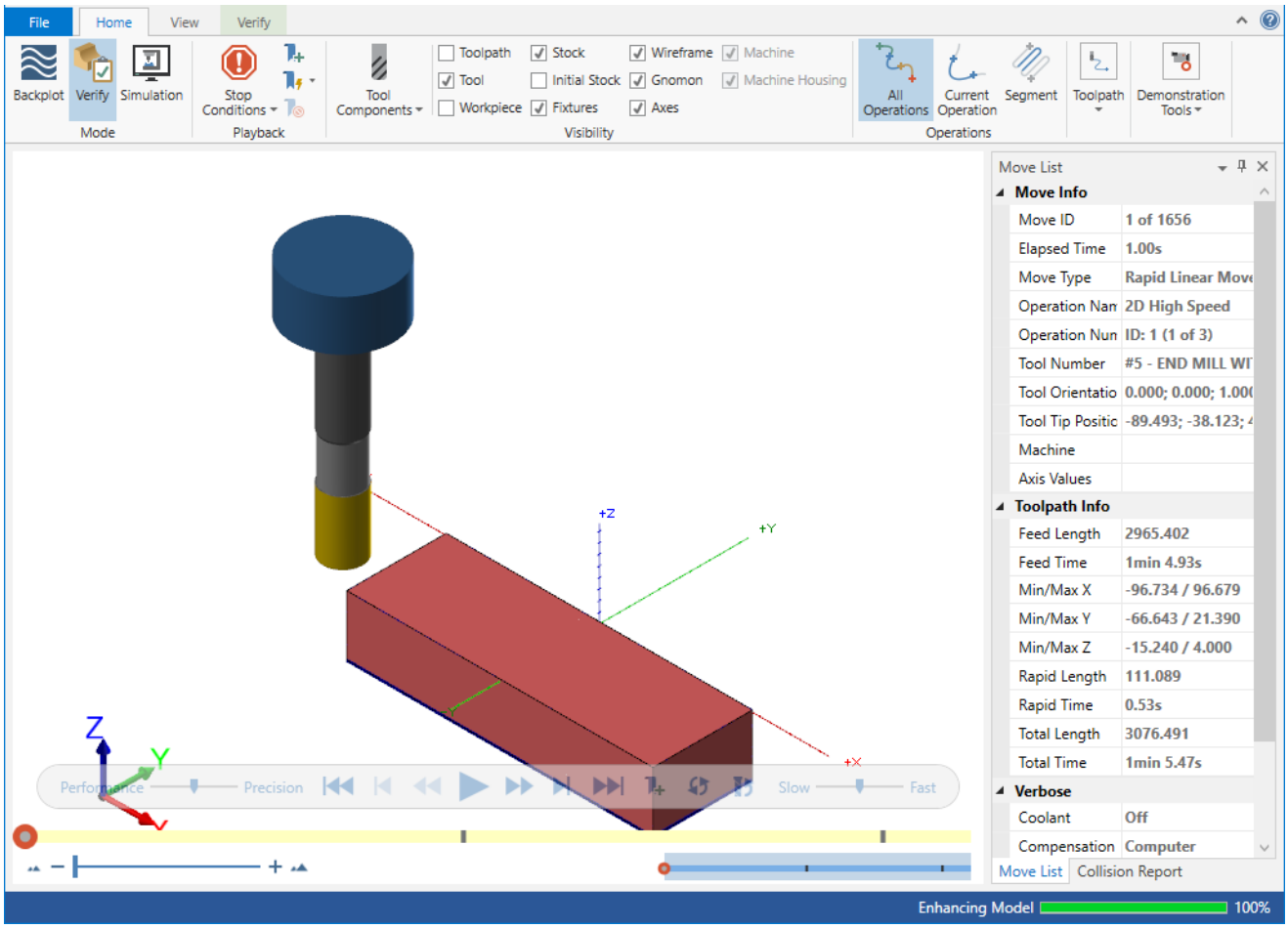



2. Select **Verify selected operations**.

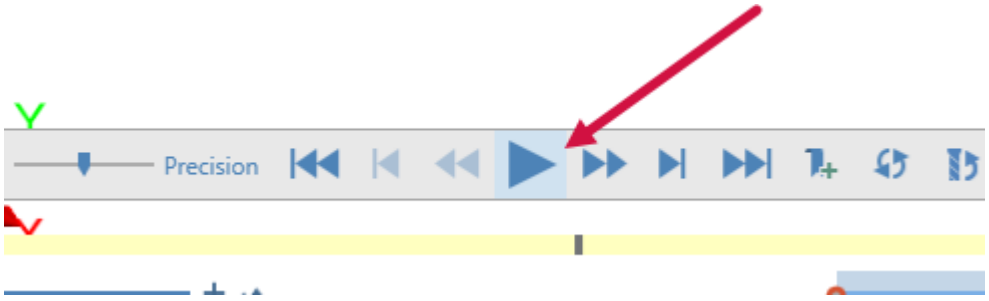



   Mastercam Simulator displays.

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

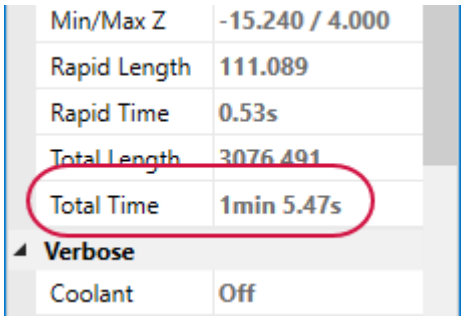



4. Press **Play** to preview the toolpath motion for the toolpaths.

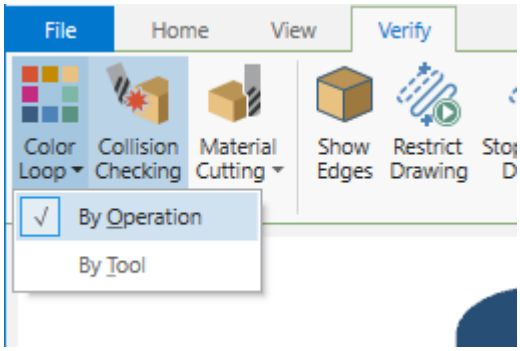



There will be material remaining on the top face of the part. That is to be expected.

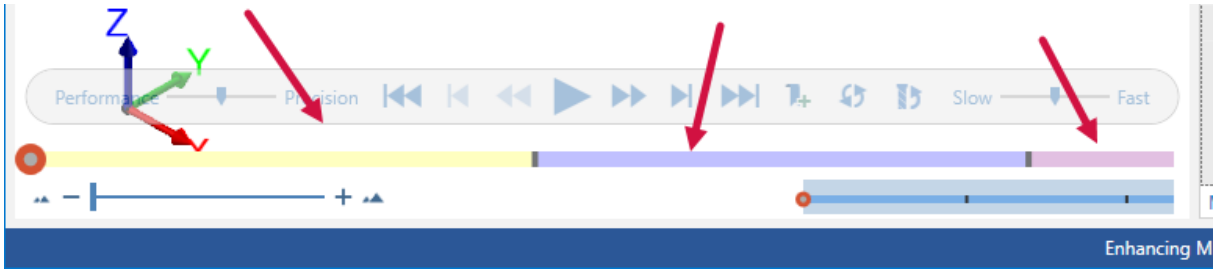
5. View the Move List, which displays all pertinent information about the toolpaths being verified. The three Dynamic Mill toolpaths are able to machine the part in the time shown with the given parameters.




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.

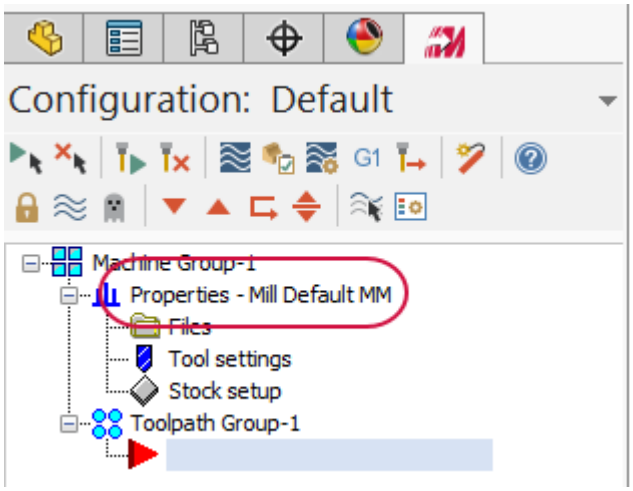
## Lesson Goals

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

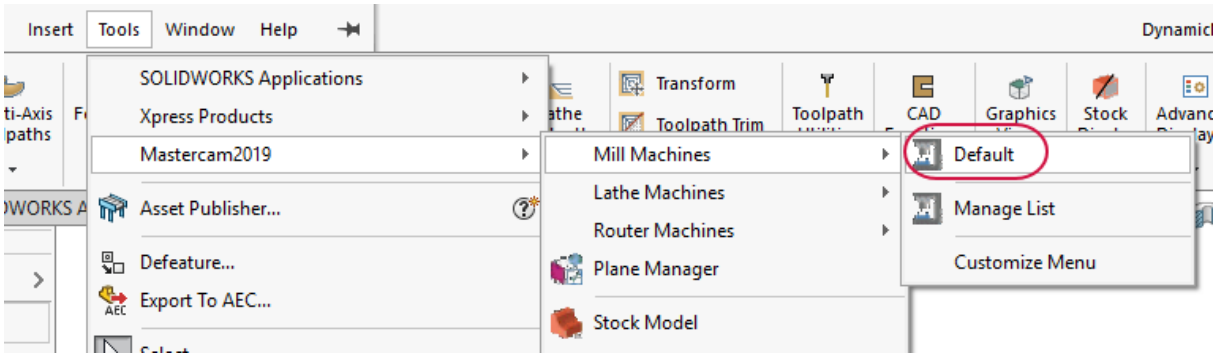
## Exercise 1: Setting up the stock

1. Open the part file, `DynamicContour`, provided with this tutorial.
2. Save the part as `DynamicContour-xxx`, replacing `xxx` with your initials.

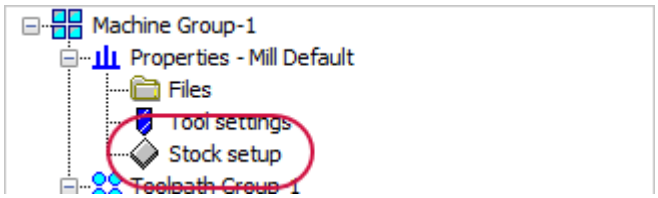
   Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.




3. If you do not have the correct default machine, select **Mastercam2019**, **Mill Machines**, **Default** from the **Tools** menu.

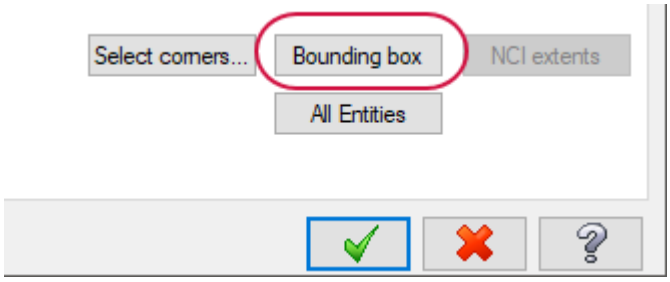


4. Select **Stock setup** from the Toolpaths Manager.



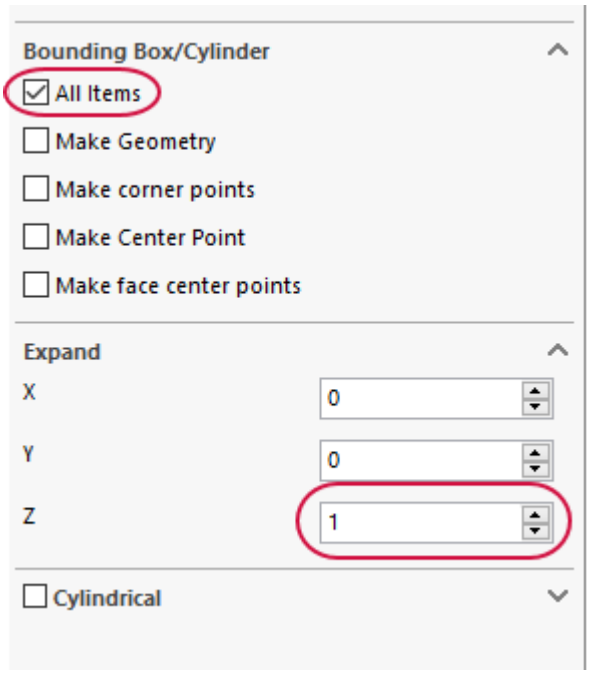

The **Machine Group Properties** dialog box displays.

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




The **Bounding Box/Cylinder** function panel displays.

6. Set the following parameters:



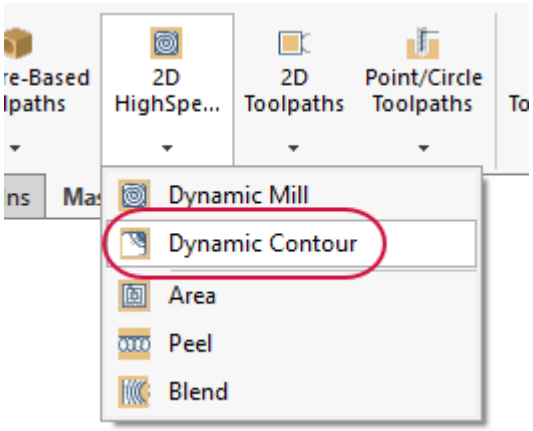

- Ensure that **All Items** is selected.
- Set **Z** to **1.0**. This adds a small amount stock to the Z height of the stock boundary.

This will create a boundary box around all of your geometry.

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

## Exercise 2: Creating the first Dynamic Contour toolpath

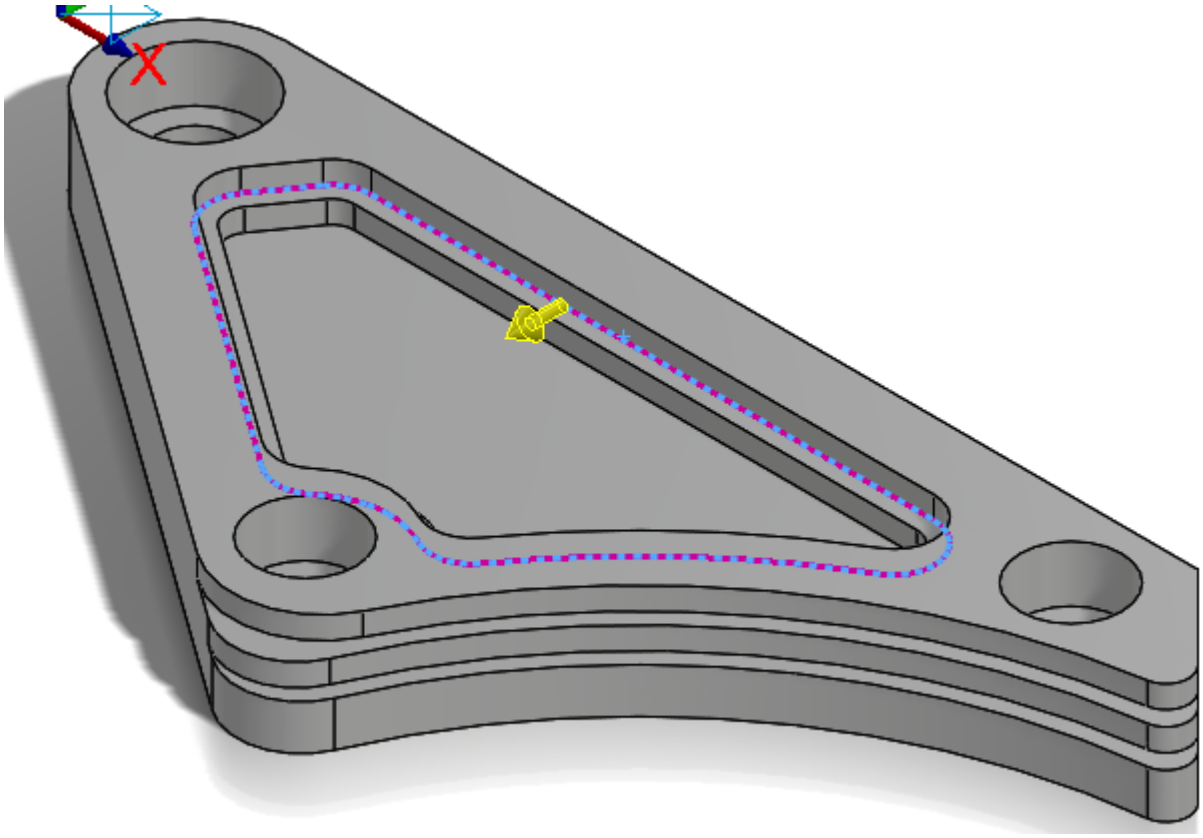
1. Select **2D HighSpeed**, **Dynamic Contour** on the **Mastercam2019** tab.




The Chain Manager displays.

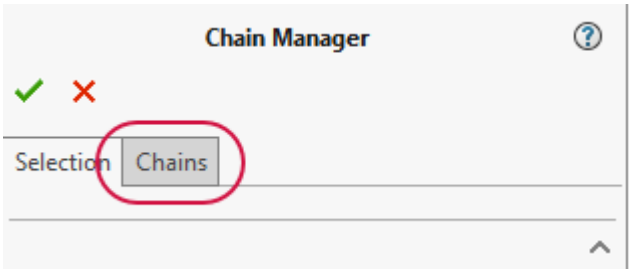
2. Right-click on the edge shown below and choose **Select Loop**.

3. The loop should be selected as shown below:

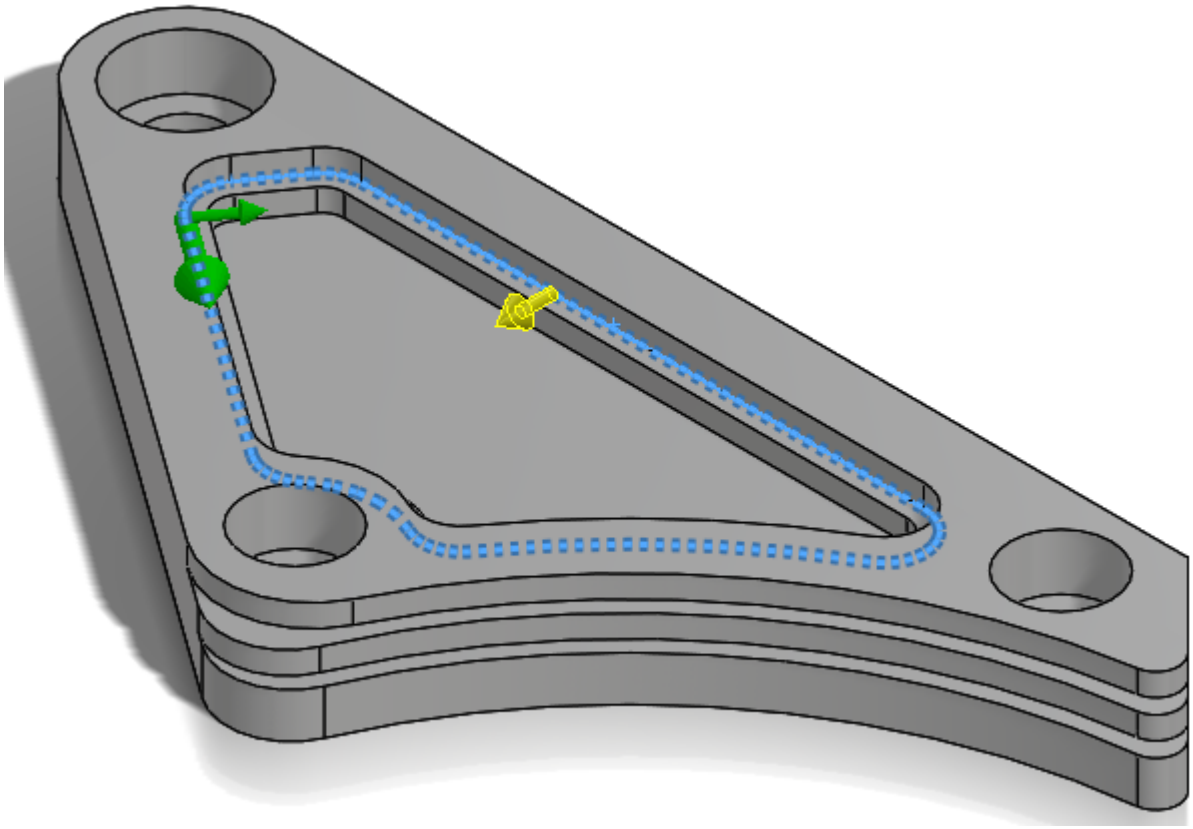


Click the arrow if necessary to select the correct loop.

4. Click the **Chains** tab.



5. Select **Chain #1** in the chain list. Ensure that the chain direction matches the image shown below.

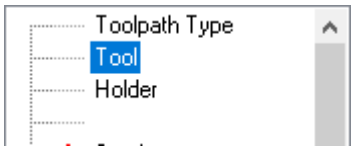


If necessary, use **Reverse** to reverse the chain.

6. Click **OK** in the Chain Manager.

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

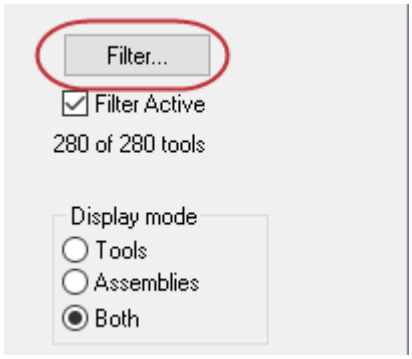
7. Select the **Tool** page.




8. Click **Select library tool**.

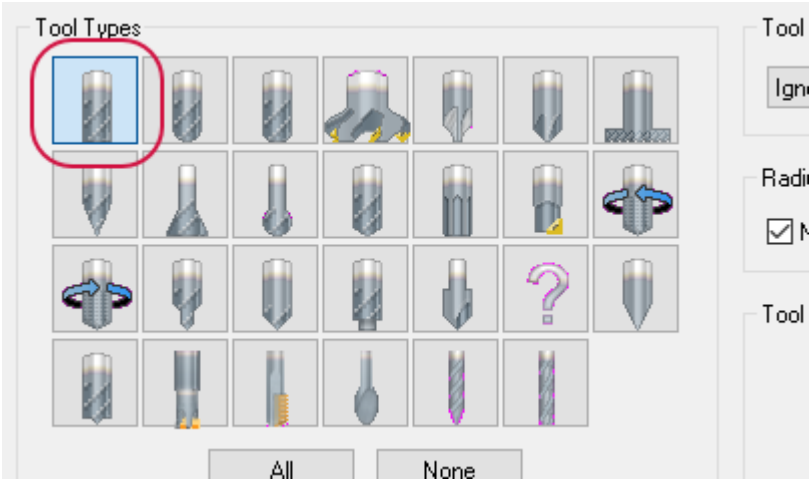
The **Tool Selection** dialog box displays.

9. Click **Filter**.

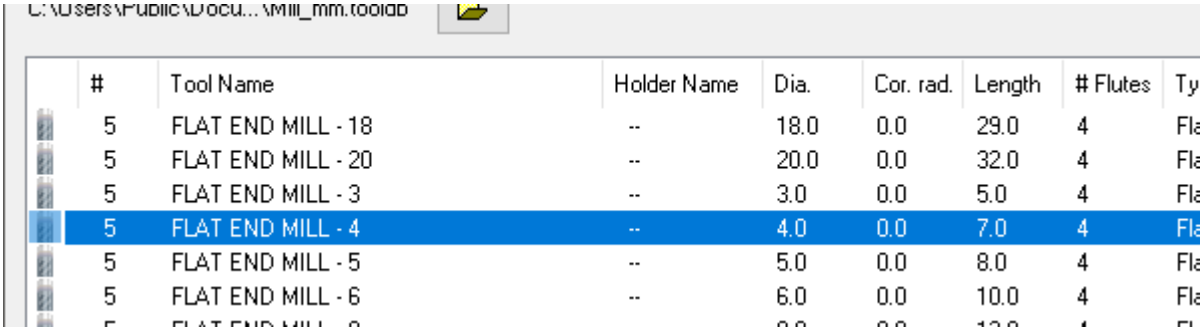



The **Tool List Filter** dialog box displays.

10. Set the **Tool Type** filter to **None**.
11. Select the **Endmill1 Flat** filter, and deselect any other filter. This ensures that you are only selecting flat end mill tools.

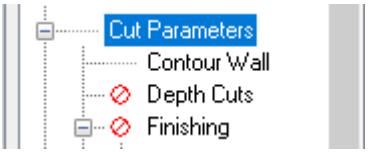



12. Click **OK** in the **Tool List Filter** dialog box.
13. In the **Tool Selection** dialog box, select the **FLAT END MILL - 4** tool.

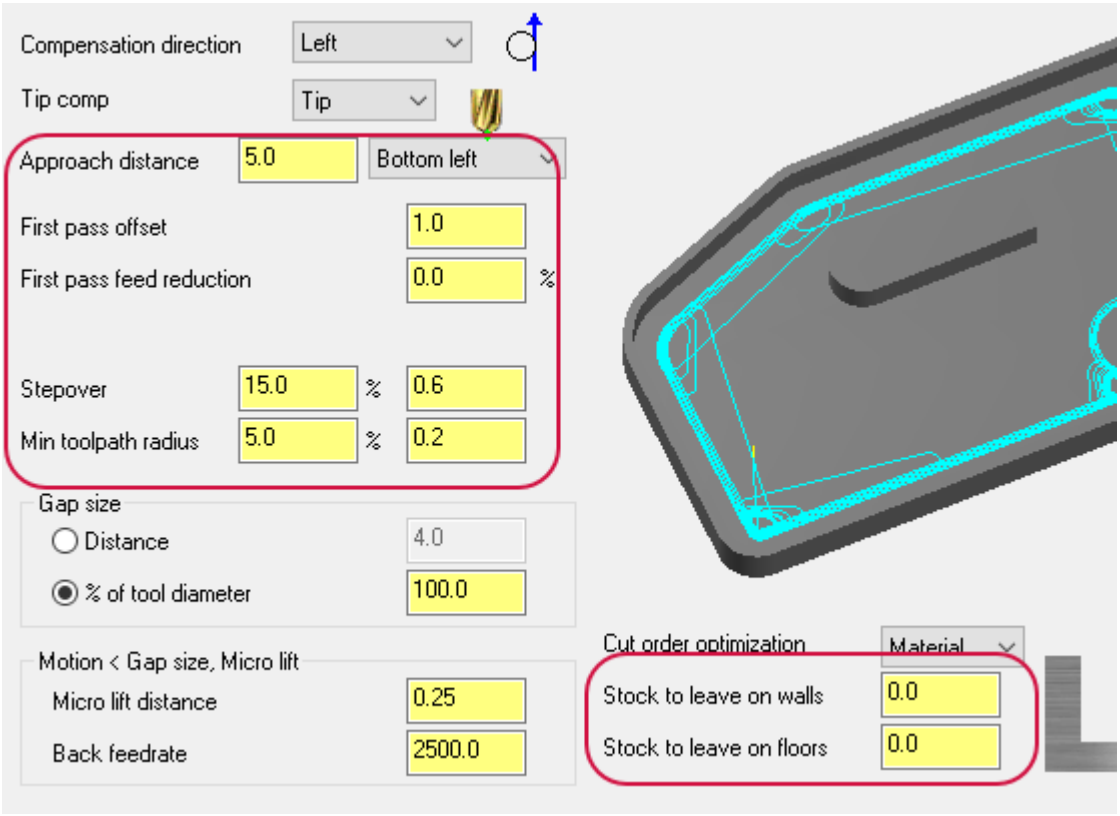


| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Ty |
|---|---|---|---|---|---|---|---|
| 5 | FLAT END MILL - 18 | -- | 18.0 | 0.0 | 29.0 | 4 | Fla |
| 5 | FLAT END MILL - 20 | -- | 20.0 | 0.0 | 32.0 | 4 | Fla |
| 5 | FLAT END MILL - 3 | -- | 3.0 | 0.0 | 5.0 | 4 | Fla |
| 5 | FLAT 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 |

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

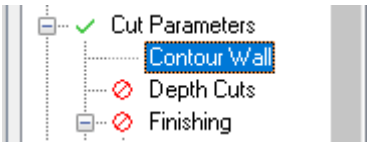


16. Set the following parameters:

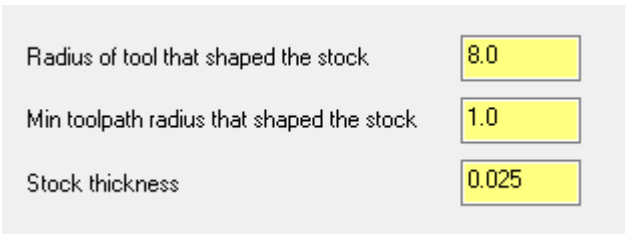



- **Compensation direction** to **Left**. Offsets the tool to the right or left of the toolpath.
- **Approach distance** to **5.0**. Adds a specified incremental distance to the beginning of the toolpath's first cut.
- **First pass offset** to **1.0**. Offsets the machining region to ensure the tool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

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



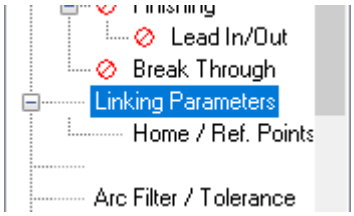

18. Set the following parameters:



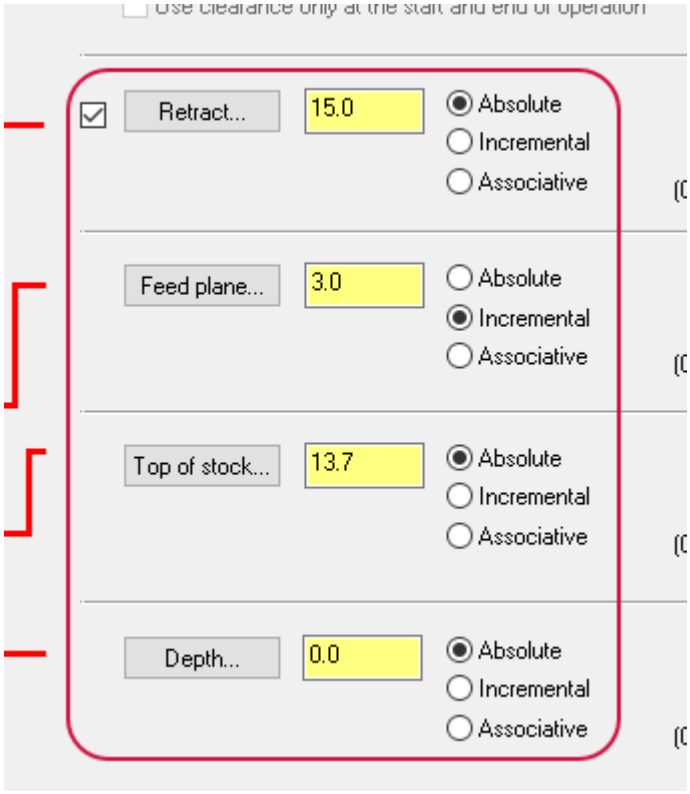
- **Radius of tool that shaped the stock** to **8.0**.
- **Minimum toolpath radius that shaped the stock** to **1.0**.
- **Stock thickness** to **0.025**.

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.

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

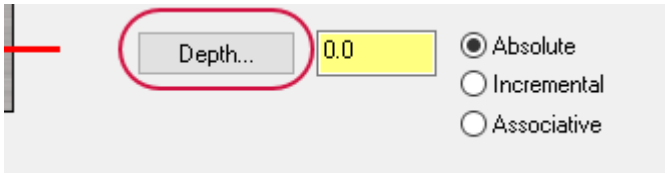



20. Set the following parameters:



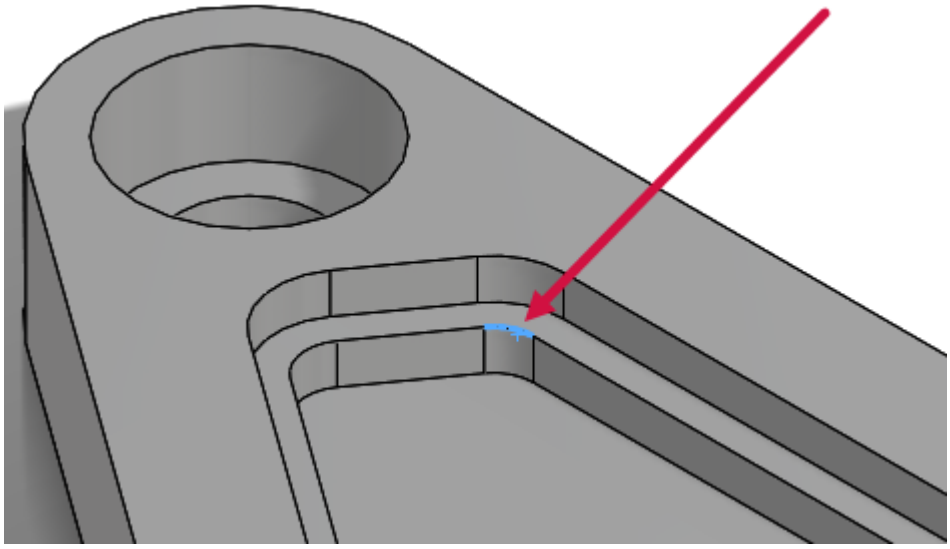

- **Retract**, **Top of stock**, and **Depth** to **Absolute**. This measures absolute values from the origin 0,0,0.
- **Retract** to **15.0**. Sets the height that the tool moves up before the next tool pass.
- **Feed plane** to **3.0**. Sets the height that the tool rapids to before changing to the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

21. 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.

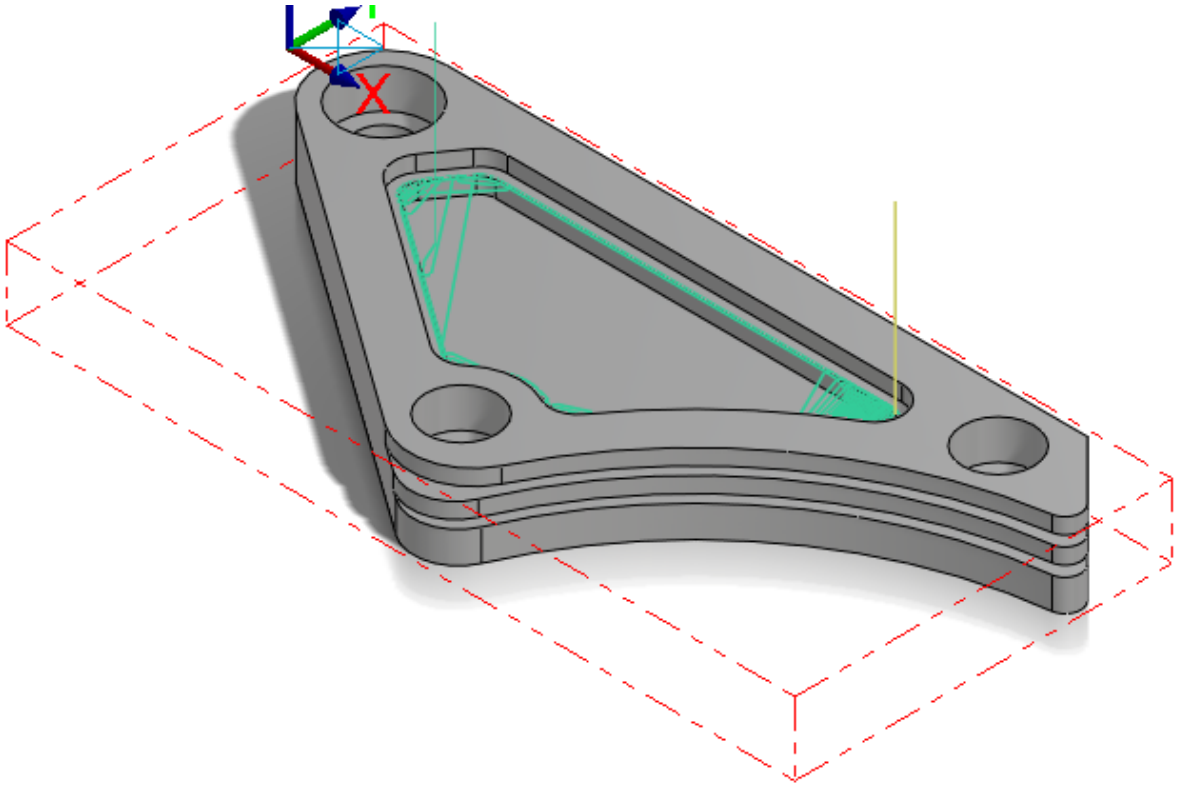
22. Select the edge shown below:



Click **OK** to accept the edge and return to the **Linking Parameters** page. Your **Depth** should now be **-22.1985**.

23. Click **OK** in the **2D High Speed Toolpath - Dynamic Contour** dialog box to accept and generate your toolpath.

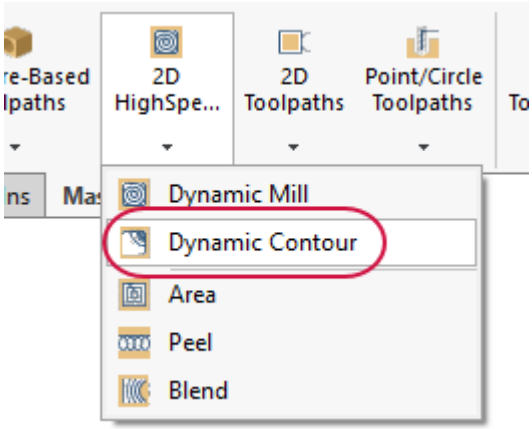
24. Your toolpath displays as shown below:



25. Save your part.

## Exercise 3: Creating the second Dynamic Contour toolpath

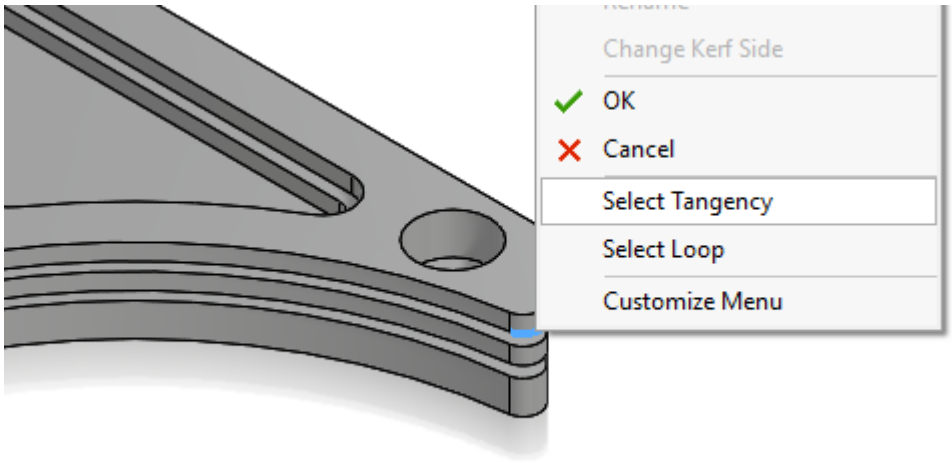
1. Select **Dynamic Contour** from the **2D HighSpeed** menu on the **Mastercam2019** tab.




The Chain Manager displays.

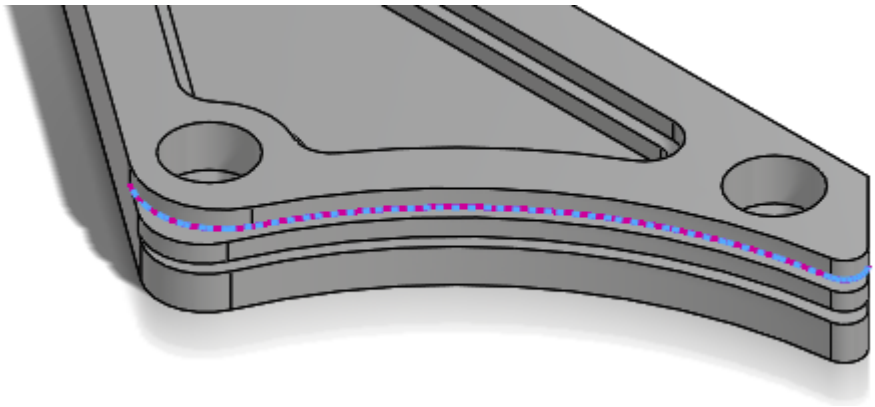
2. Ensure that you are on the **Selection** tab.

3. Right-click on the edge shown below and choose **Select Tangency**.

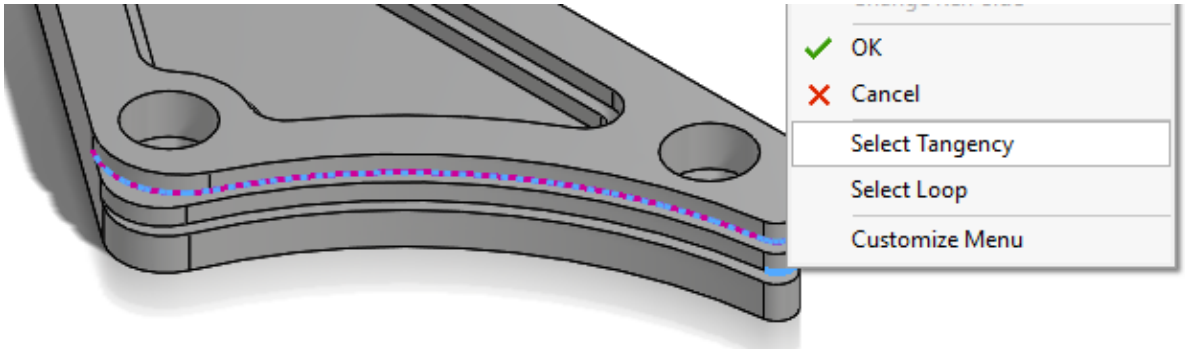



Select Tangency automatically selects curves, edges, or faces that are tangent to your initial selection.

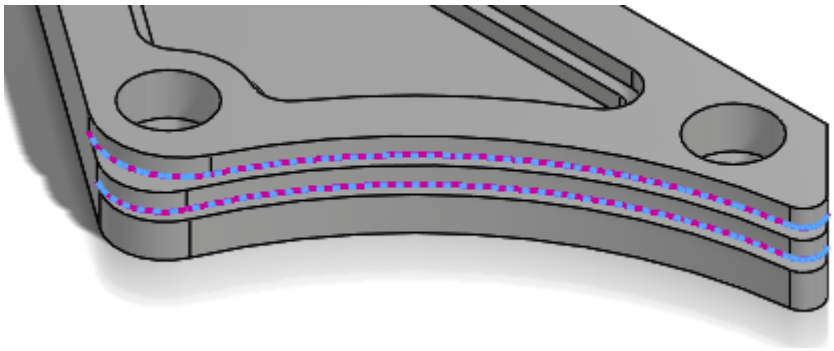
Your selection should match the image below:



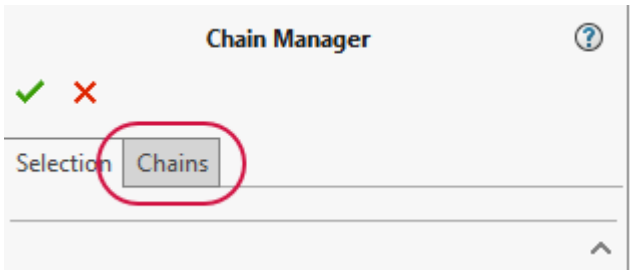
4. Right-click on the second edge shown below and choose **Select Tangency**.



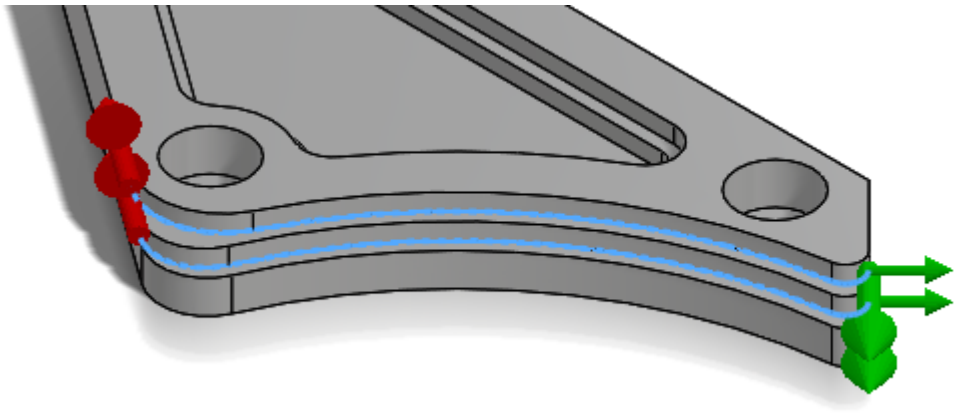
You should have the following edges selected.



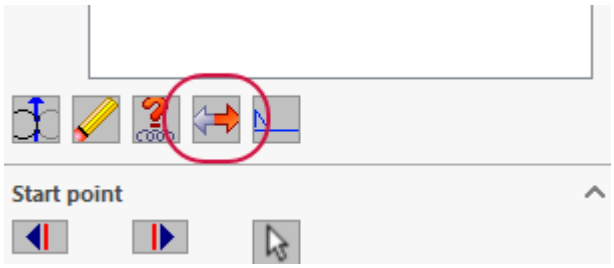
5. Select the **Chains** tab.




6. Ensure that both chains are going in the same direction by selecting the individual chains in the chain list.



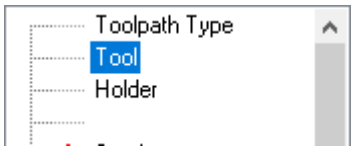
7. If the chains are going in a different direction, use **Reverse** to reverse the chain's direction.




8. Click **OK** in the Chain Manager.

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

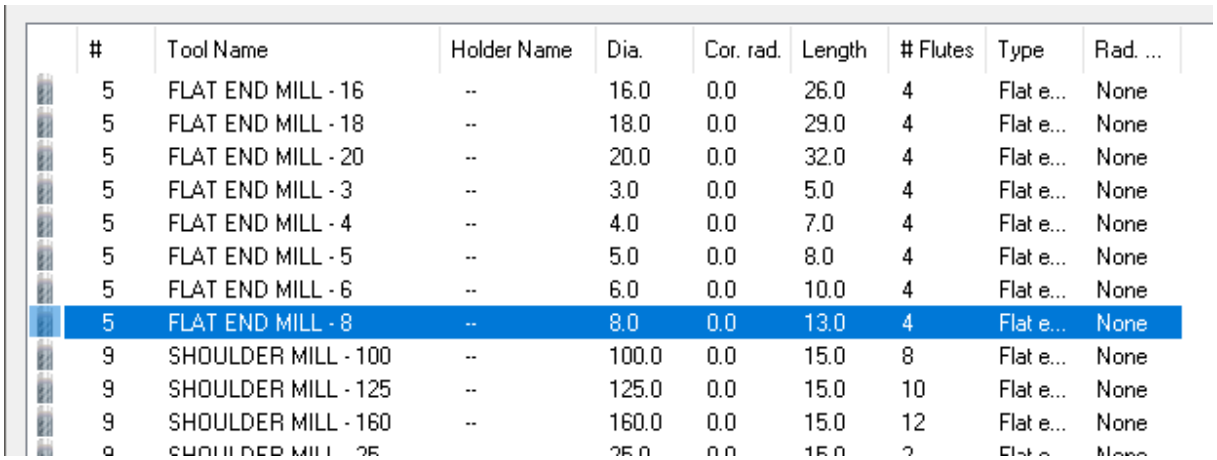
9. Select the **Tool** page.




10. Click **Select library tool**.

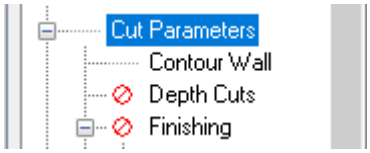
    The **Tool Selection** dialog box displays.

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

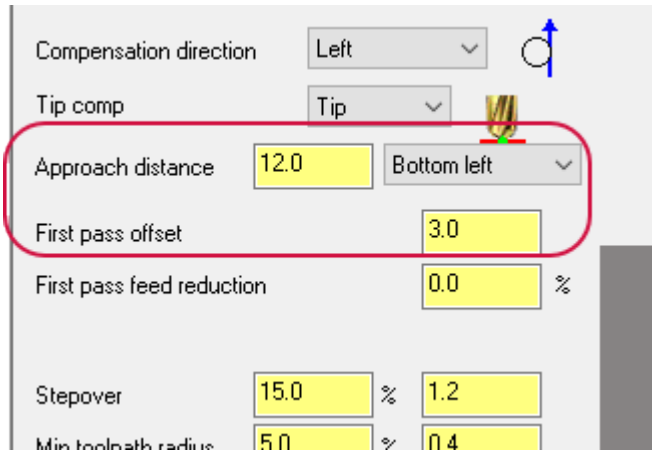


| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Type | Rad. ... |
|---|---|---|---|---|---|---|---|---|
| 5 | FLAT END MILL - 16 | -- | 16.0 | 0.0 | 26.0 | 4 | Flat e... | None |
| 5 | FLAT END MILL - 18 | -- | 18.0 | 0.0 | 29.0 | 4 | Flat e... | None |
| 5 | FLAT END MILL - 20 | -- | 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 |

12. Click **OK** in the **Tool Selection** dialog box.
13. Select the **Cut Parameters** page.

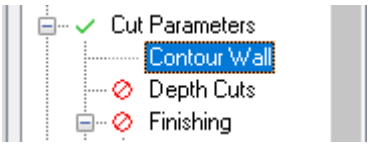



14. Set the following parameters:

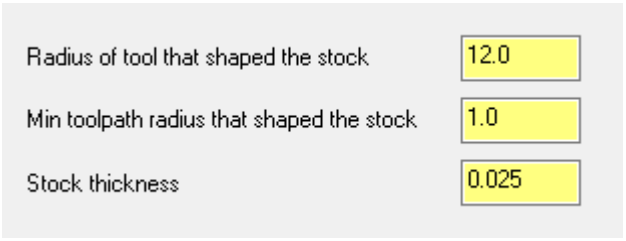



- **Approach distance** to **12.0**.
- **First pass offset** to **3.0**.

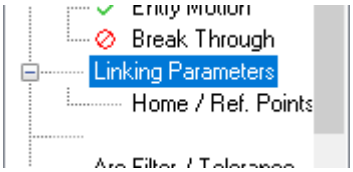
15. Select the **Contour Wall** page.



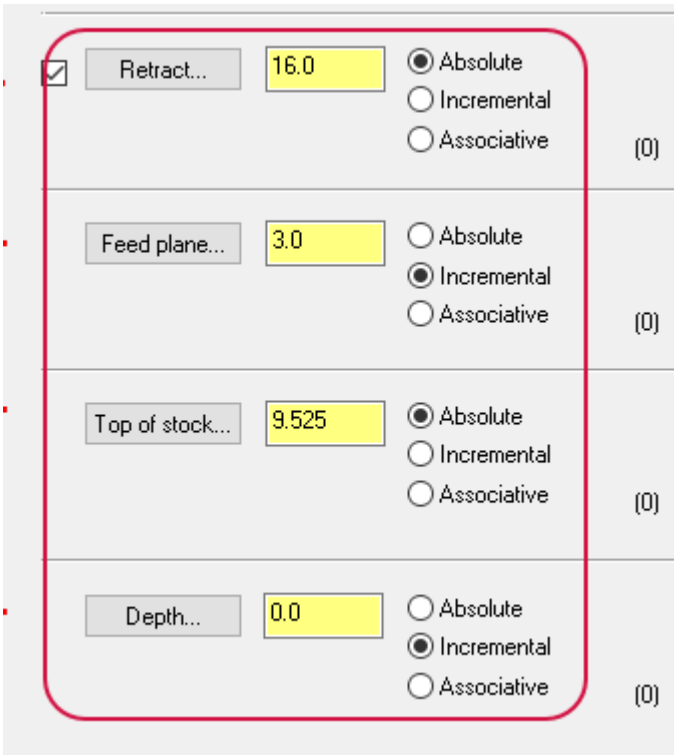

16. Set **Radius of tool that shaped the stock** to **10.0**.




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



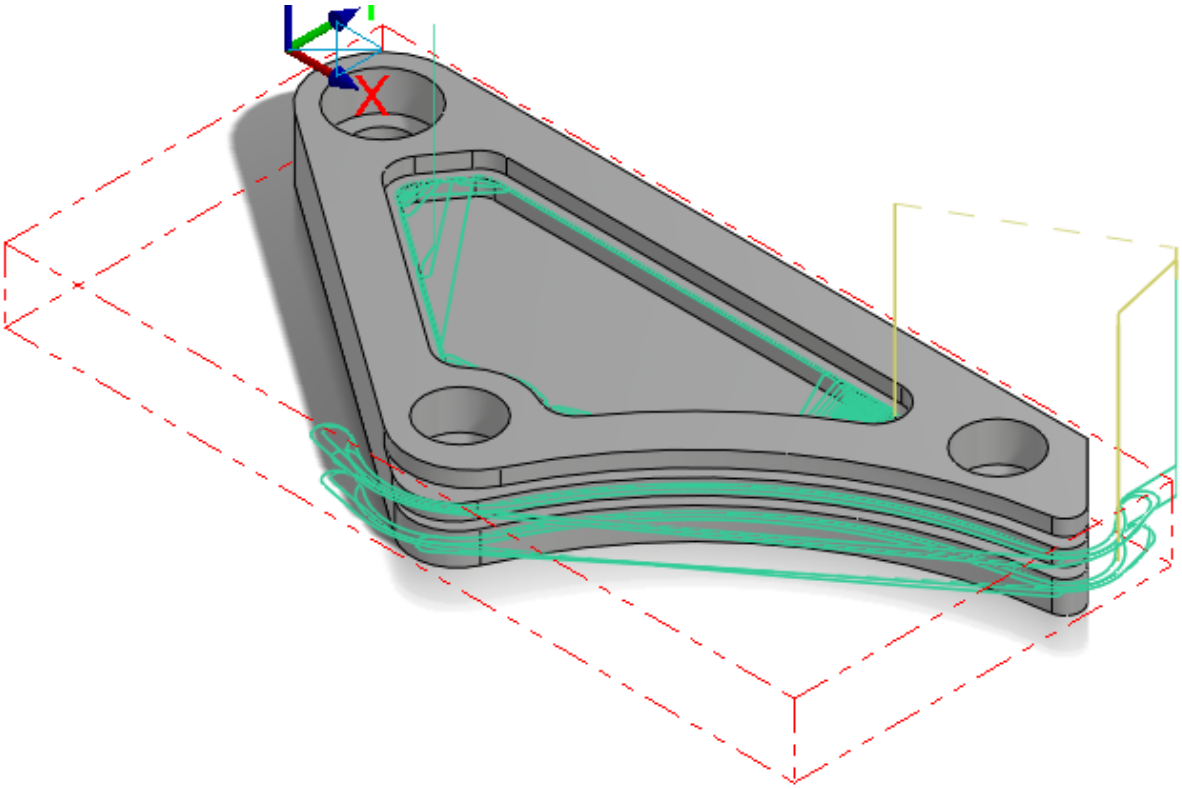

18. Set the following parameters:




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

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

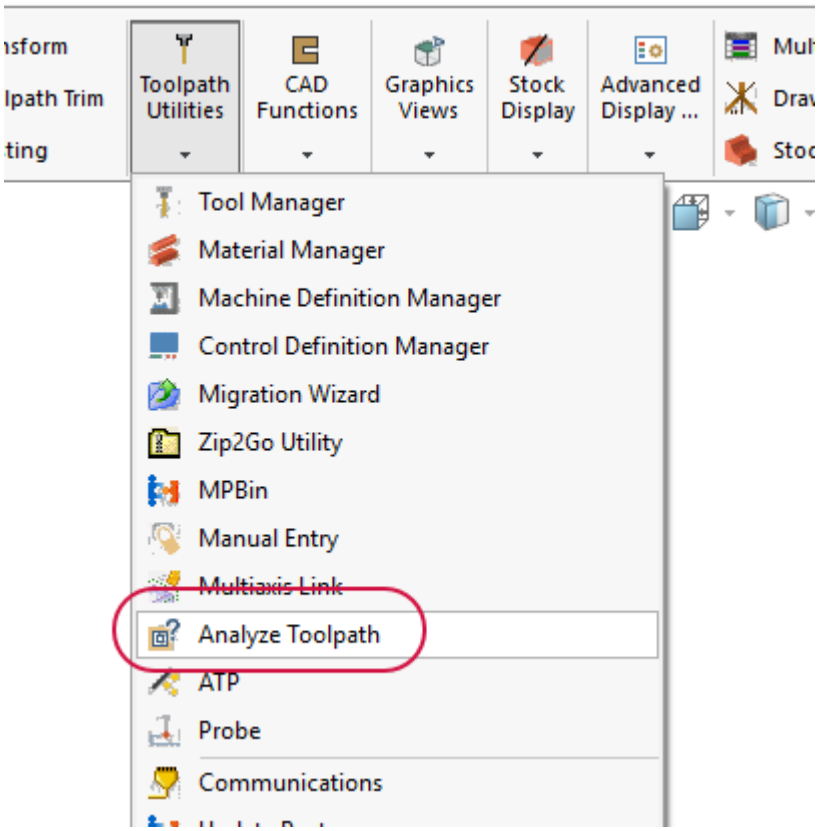
    The toolpath will display as shown below:



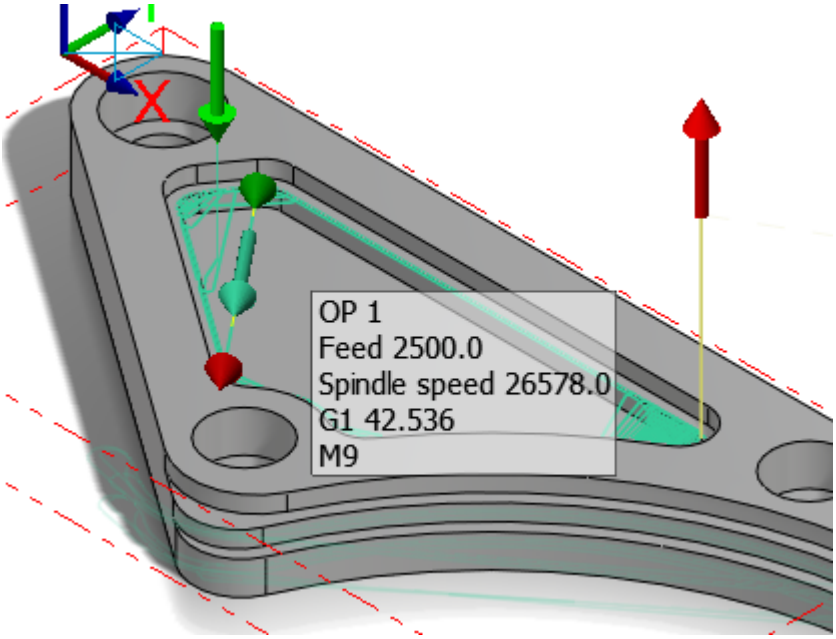
## Exercise 4: Analyzing the toolpaths

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 **Toolpath Utilities** menu on the **Mastercam2019** tab.




2. Hover over the first toolpath:



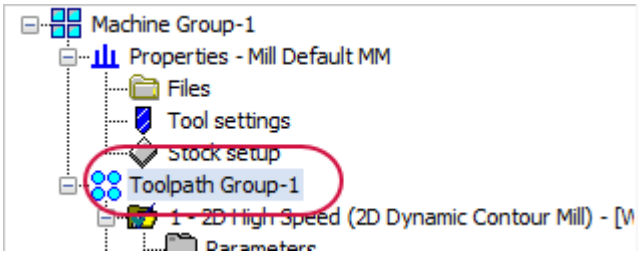
Analyze Toolpath informs you of the operation number, the feed move, spindle speed, G1 and the line length, and the coolant code. It also displays the start and end of the toolpath and of the section you are hovering over. If there is more than one toolpath displayed at the time, then toolpaths other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

3. Continue hovering over areas of both toolpaths. Once you are satisfied, click **OK** to exit the function.

## Exercise 5: Verifying the toolpaths

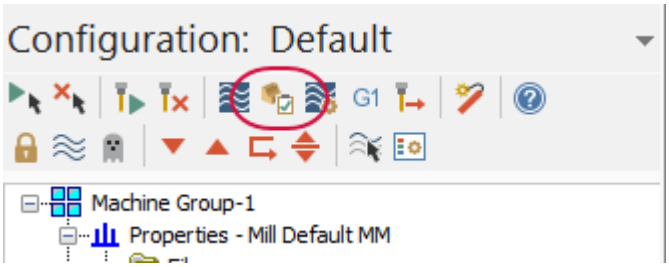
You will now verify both toolpaths.

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



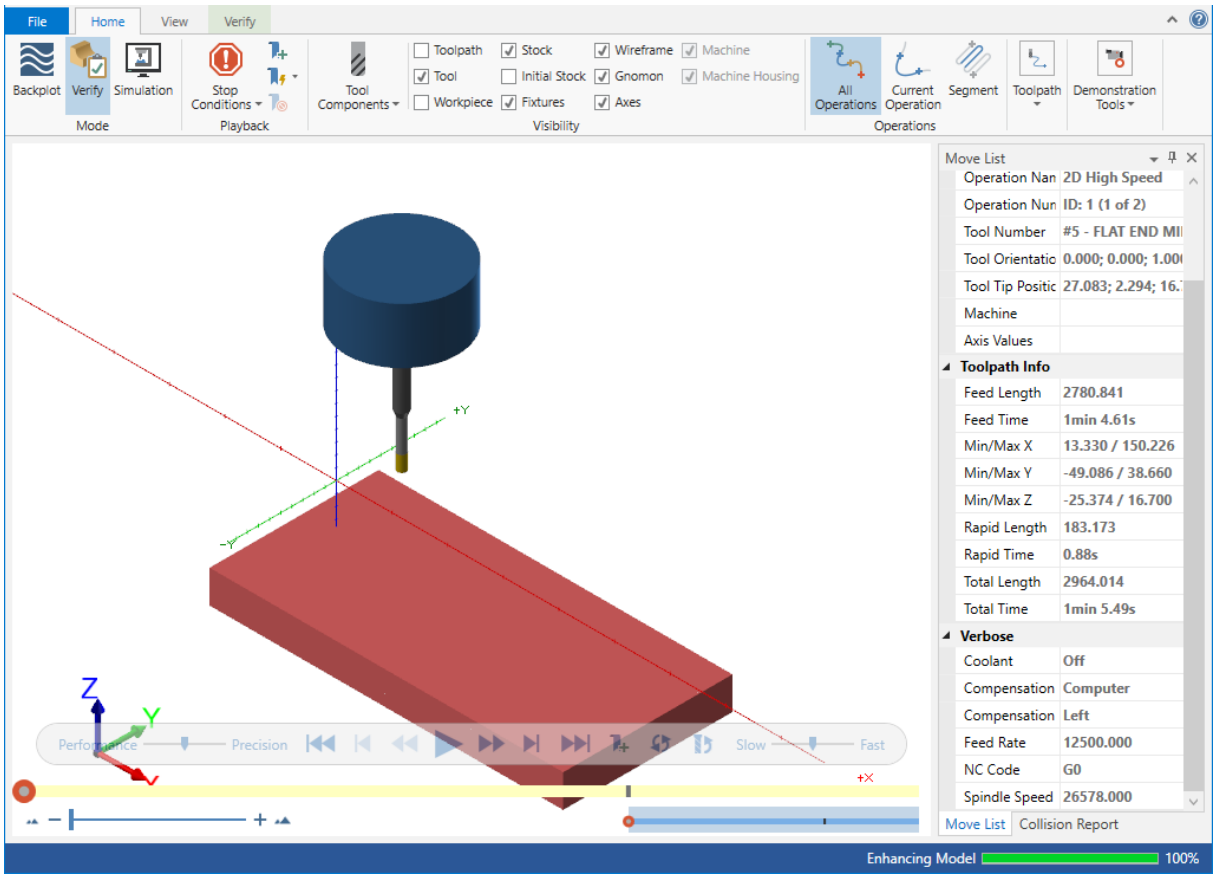

This selects both toolpaths so that they can be verified at the same time.

2. Select **Verify selected operations**.

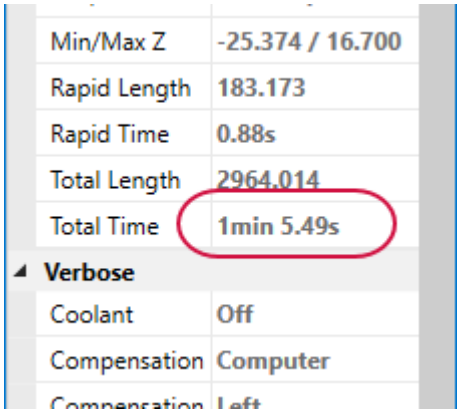



Mastercam Simulator displays.

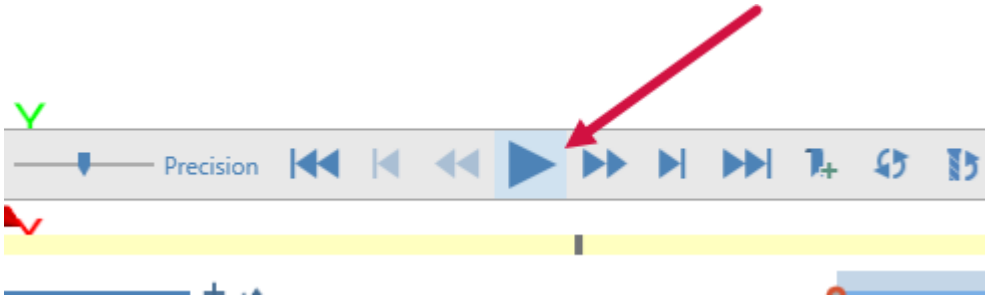
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:




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.




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.

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.

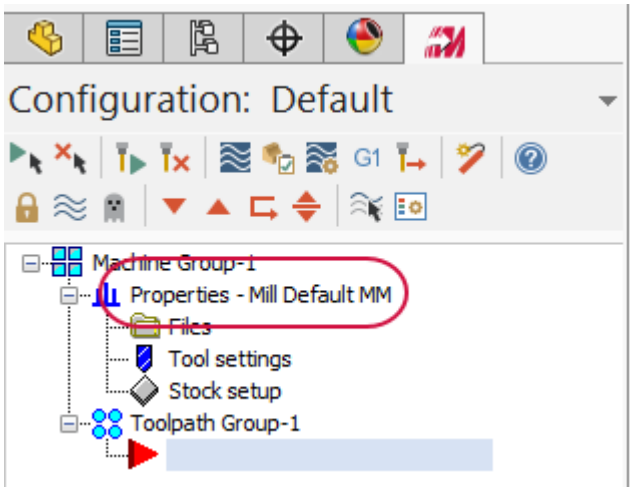
In this lesson you will create one Peel Mill toolpath, copy it twice, and make edits.

## Lesson Goals

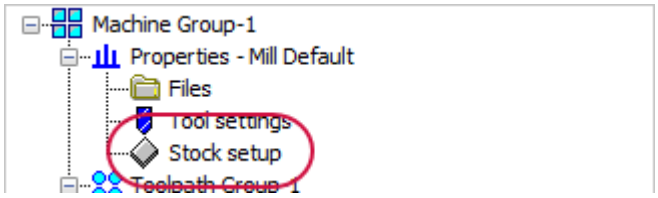
- Create a Peel Mill toolpath
- Copy and paste the toolpath
- Make edits to the copied toolpaths
- Verify the created toolpaths

## Exercise 1: Setting up the stock

1. Open the part file, `PeelMill`, provided with this tutorial.
2. Save the part as `PeelMill-xxx`, replacing `xxx` with your initials.
3. Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.

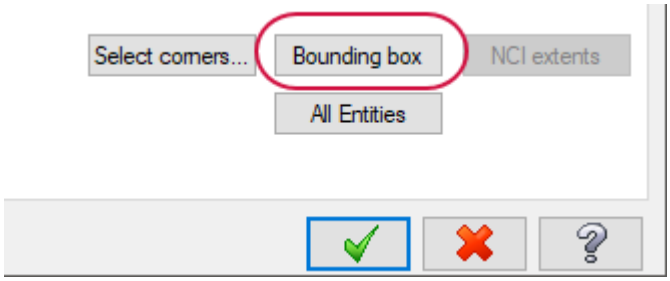



4. If you do not have the correct default machine, select **Mastercam2019**, **Mill Machines**, **Default** from the **Tools** menu.
5. Select **Stock setup** from the Toolpaths Manager.



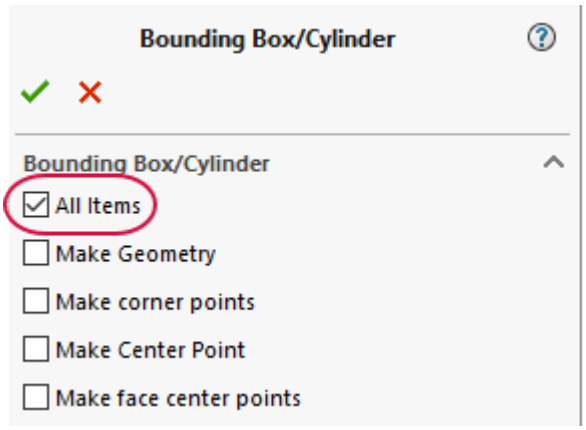

The **Machine Group Properties** dialog box displays.

6. Select **Bounding box**.




The **Bounding Box/Cylinder** function panel displays.

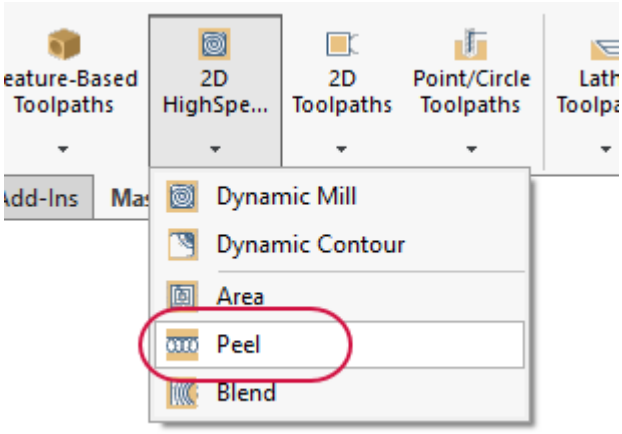
7. Ensure that **All Items** is selected.




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

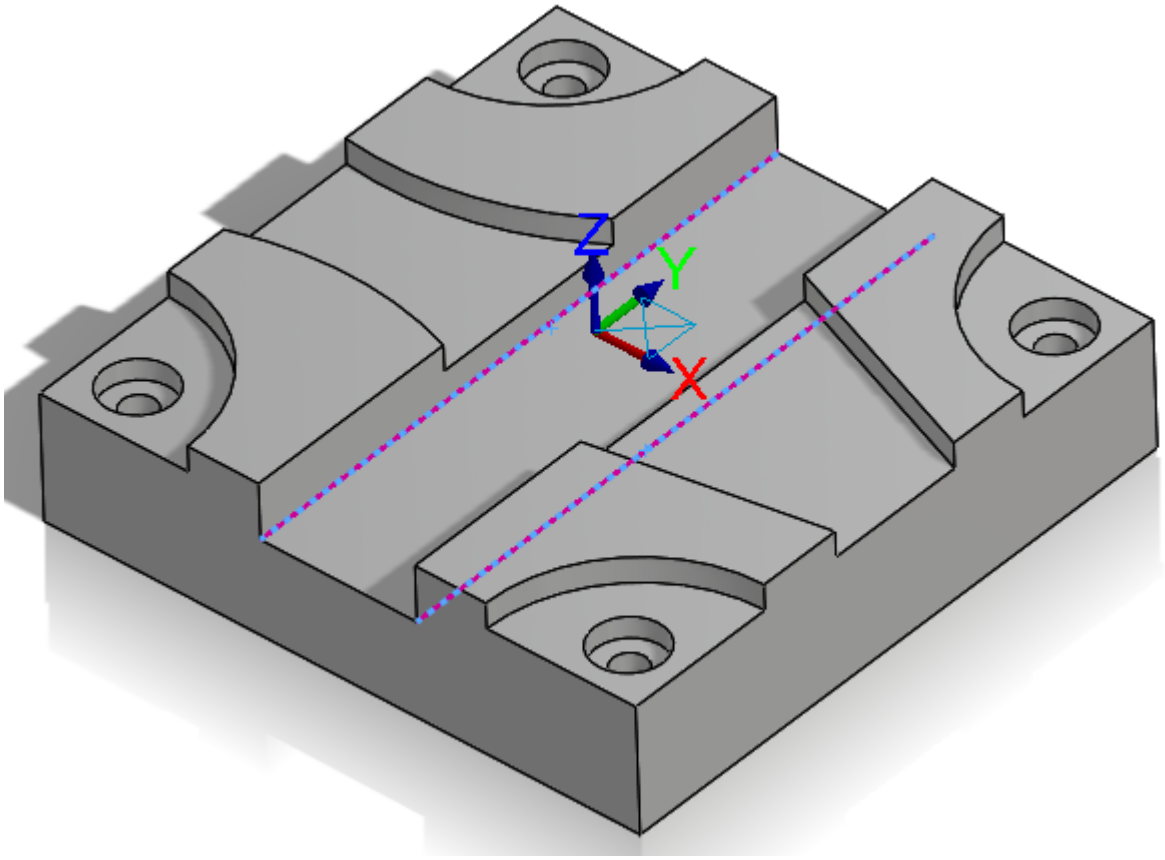
## Exercise 2: Creating the first Peel Mill toolpath

1. Select **Peel** from the **2D HighSpeed** menu on the **Mastercam2019** tab.

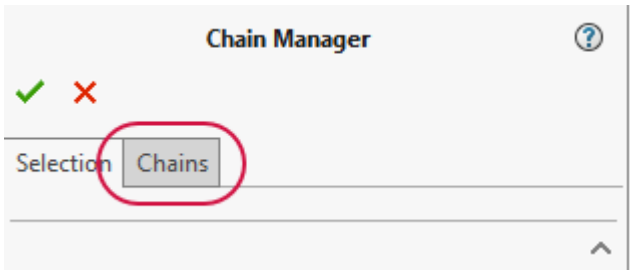



The Chain Manager displays.

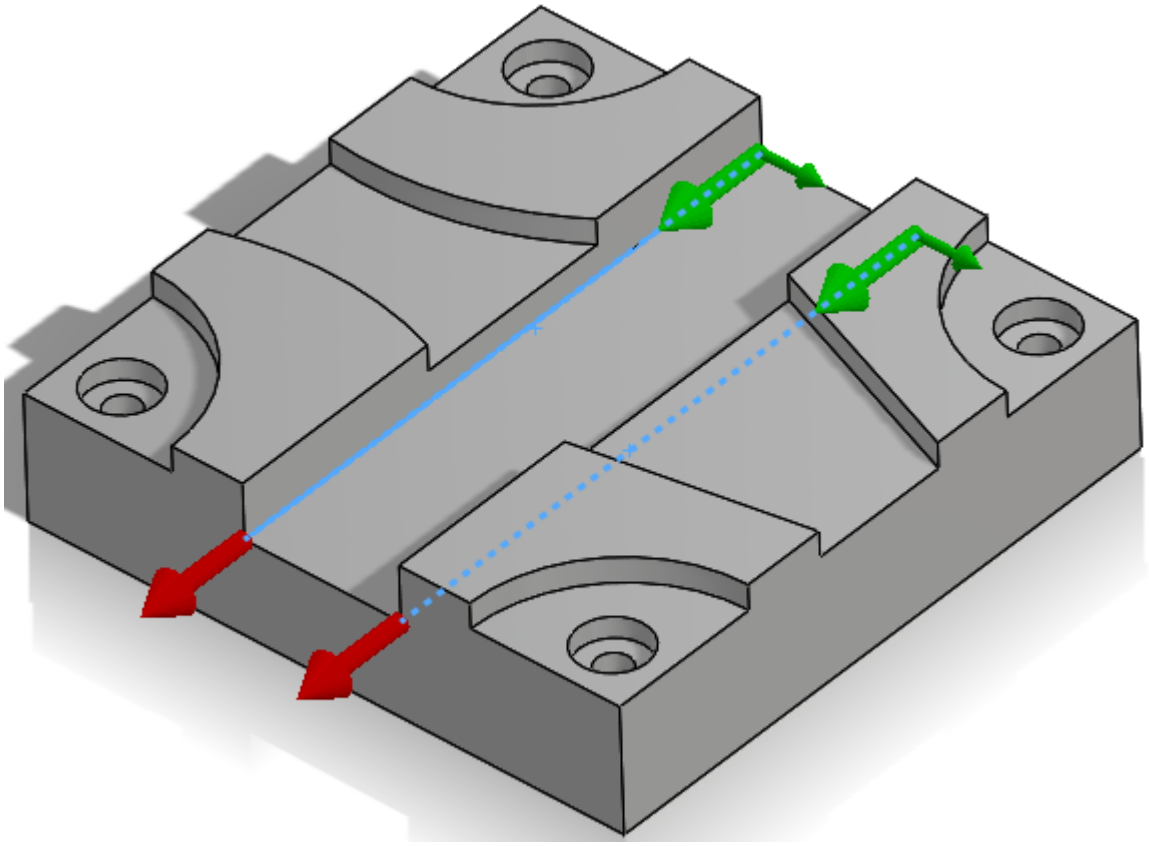
2. Select two edges shown below. You may need to rotate the part to select them.



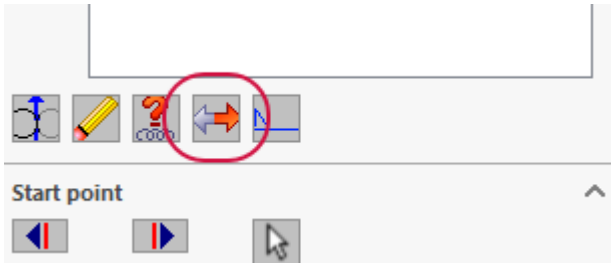

3. Select the **Chains** tab.



4. Select each chain from the chain list to ensure that they are going in the same direction.



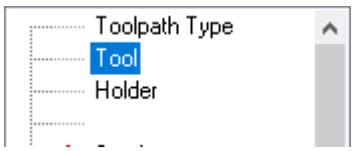
5. If one of your chains is going in the opposite direction, use **Reverse** to change its direction.



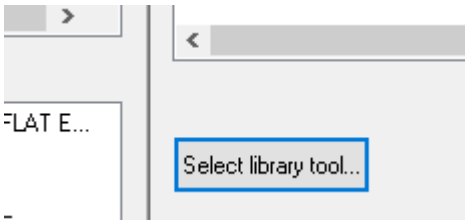

6. Click **OK** in the Chain Manager.

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

7. Select the **Tool** page.

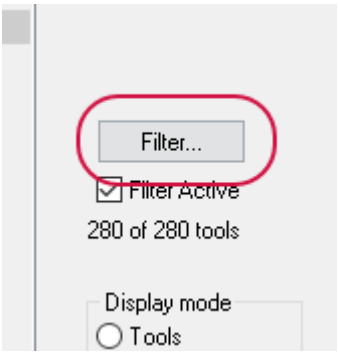



8. Click **Select library tool**.



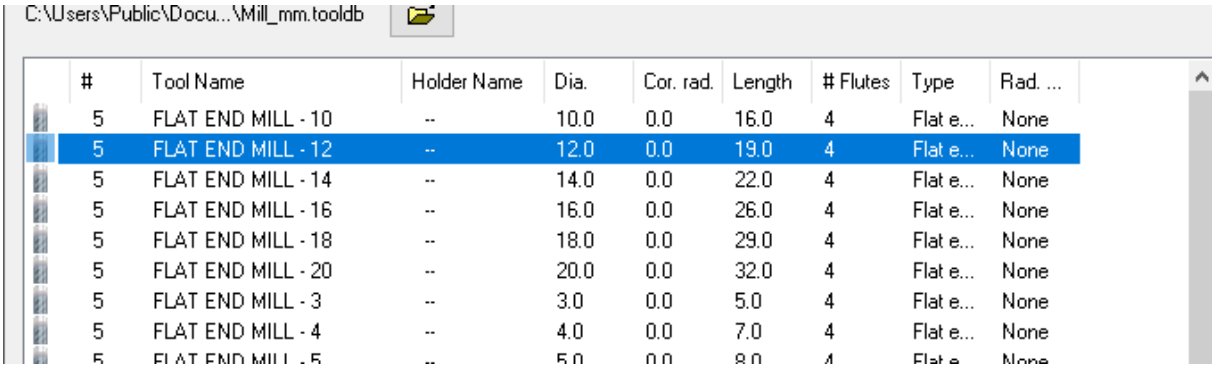
The **Tool Selection** dialog box displays.

9. Click **Filter**.



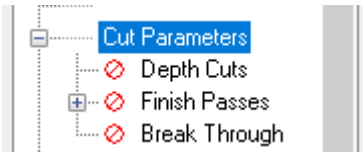

The **Tool List Filter** dialog box displays.

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

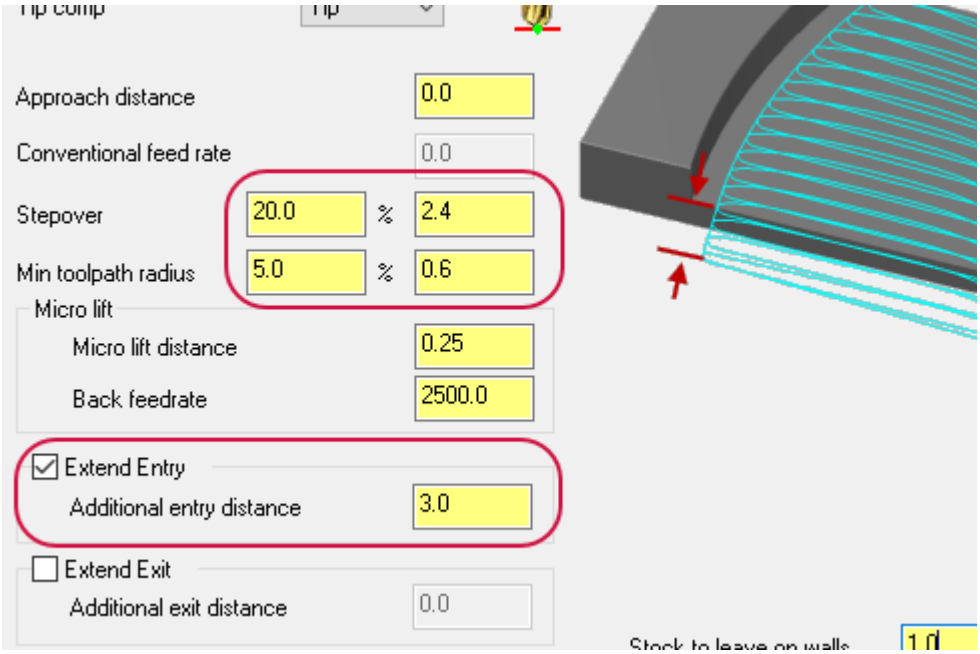



| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Type | Rad. ... |
|---|---|---|---|---|---|---|---|---|
| 5 | FLAT END MILL - 10 | -- | 10.0 | 0.0 | 16.0 | 4 | Flat e... | None |
| 5 | FLAT END MILL - 12 | -- | 12.0 | 0.0 | 19.0 | 4 | Flat e... | None |
| 5 | FLAT END MILL - 14 | -- | 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 |

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

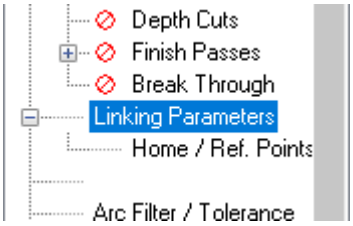


15. Set the following parameters:

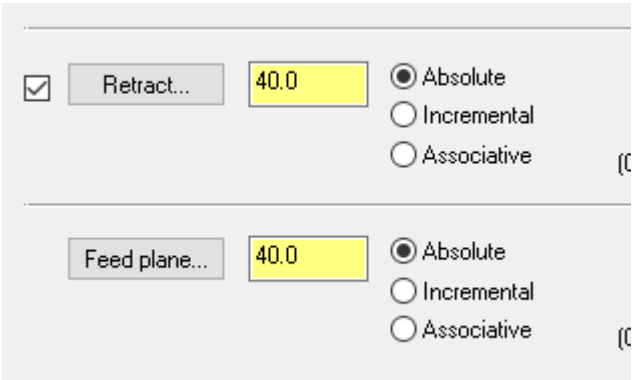



- **Stepover** to **20.0**%.
- **Minimum toolpath radius** to **5.0**%.
- Select **Extend Exit** and set **Additional exit distance** to **3.0**. This adjusts the final tool engagement with the material.

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



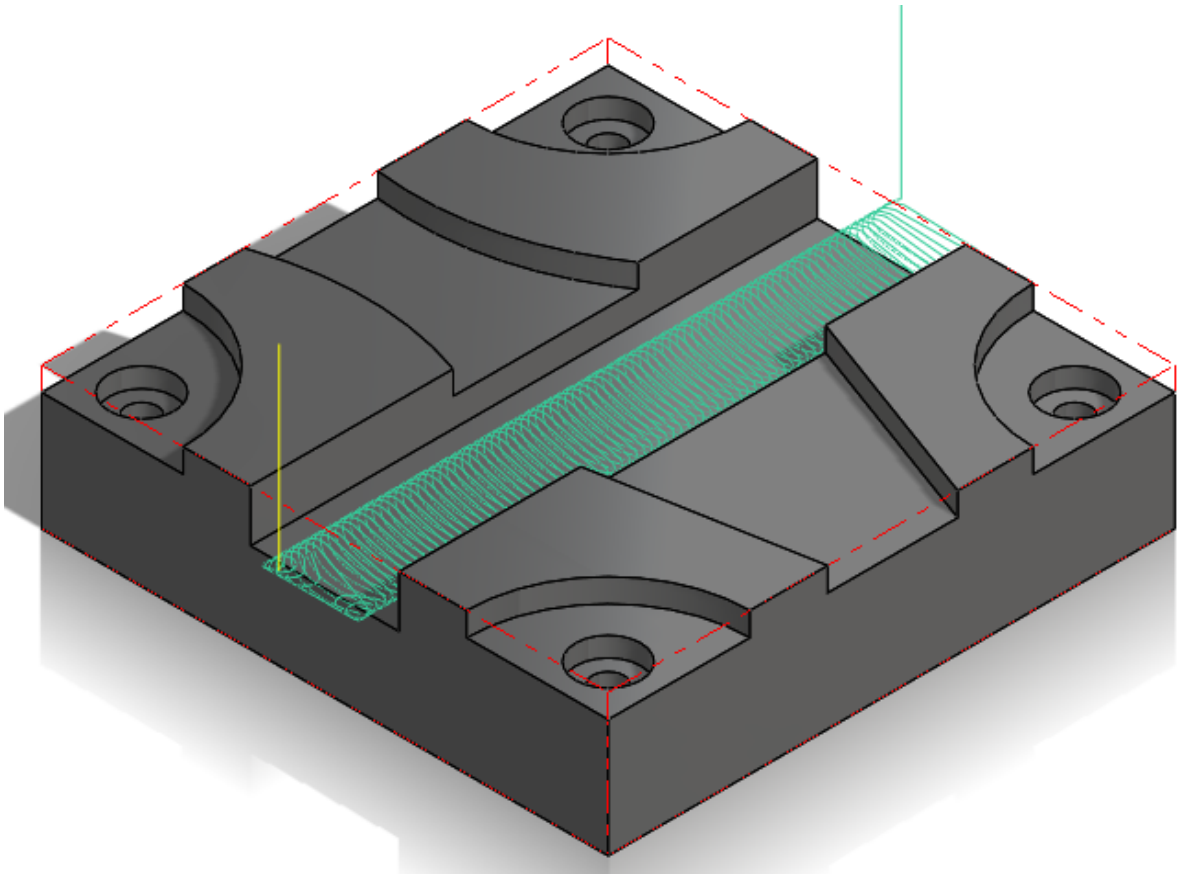

17. Set the following parameters:




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

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

The toolpath will display as shown below:

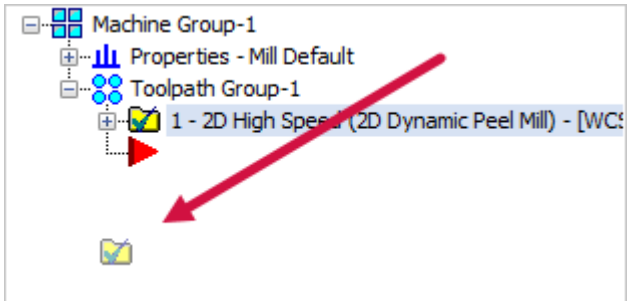


19. Save your file.

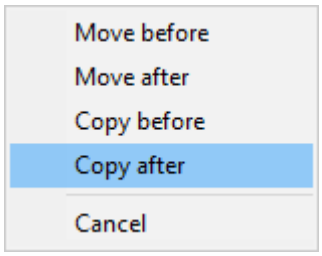
## Exercise 3: Copying the Peel Mill toolpath

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.

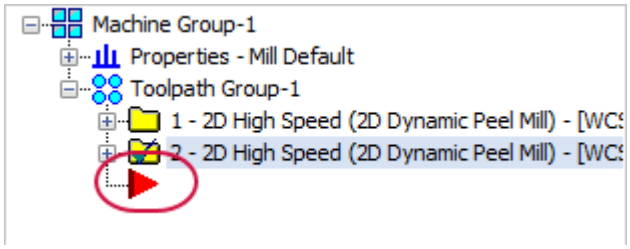



2. Select **Copy after** from the right-click menu that displays.




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

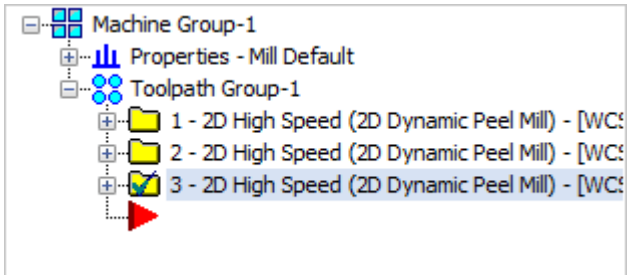
3. Click and drag the red arrow to position it below the copied operation.




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.

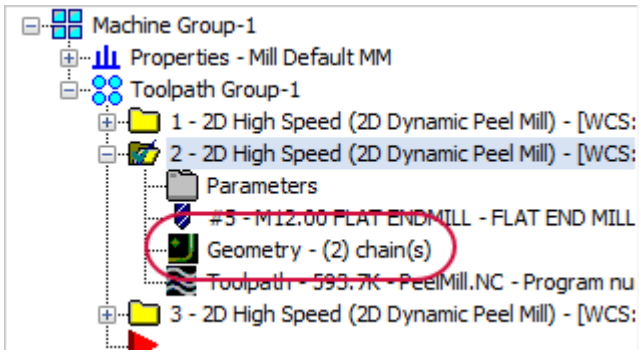



6. Save your file.

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

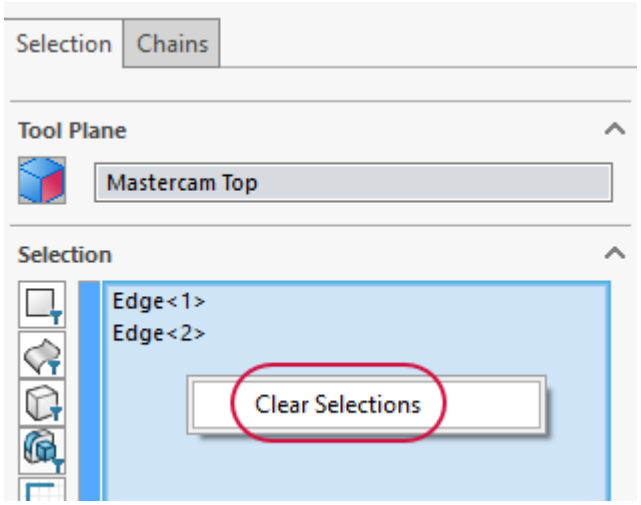
In this exercise, you will re-chain the geometry for the Peel Mill toolpaths that were copied.

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

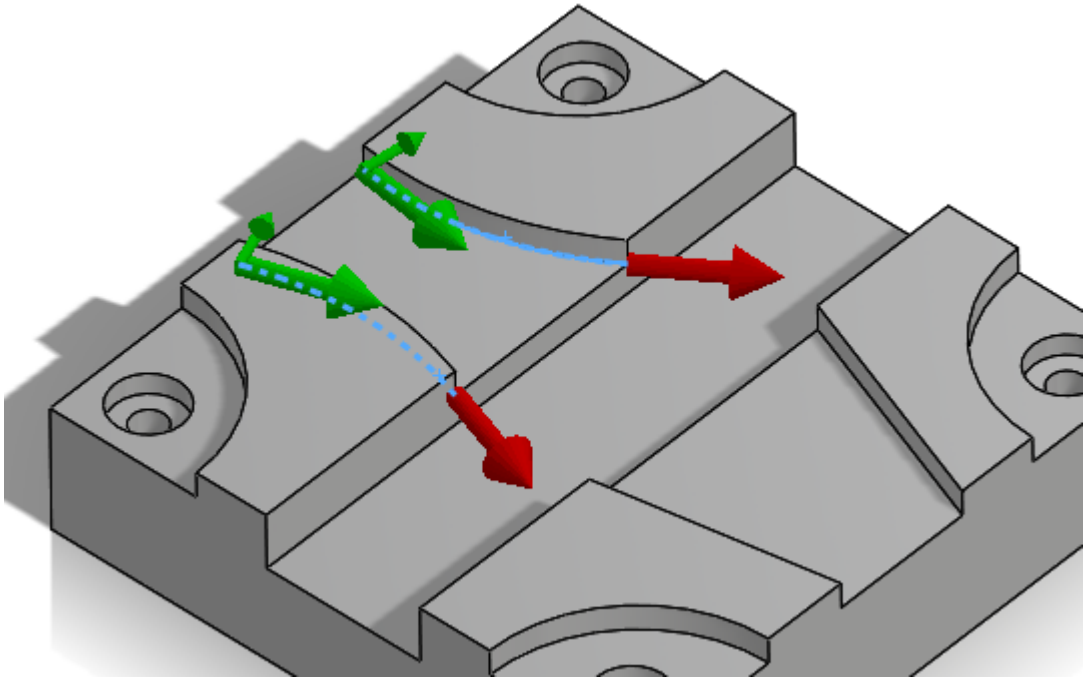



   The Chain Manager displays.

2. In the **Selection** tab, right-click in the Selection list and select **Clear Selections**.



3. Select the two edges shown below. You may need to rotate the part to select them.

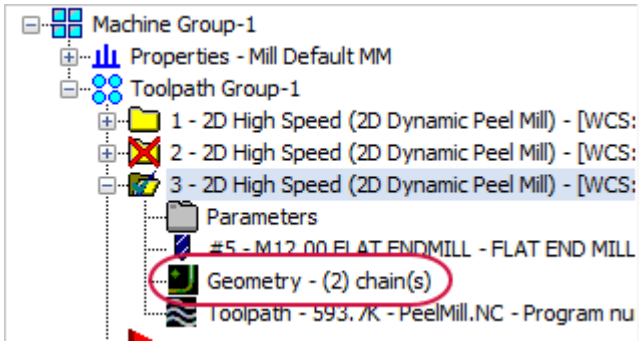


Click the **Chains** tab and ensure that both chains are going in the same direction. Use **Reverse** if necessary. It does not matter which direction they are going, but both chains must be going in the same direction.

4. Click **OK** in the Chain Manager.

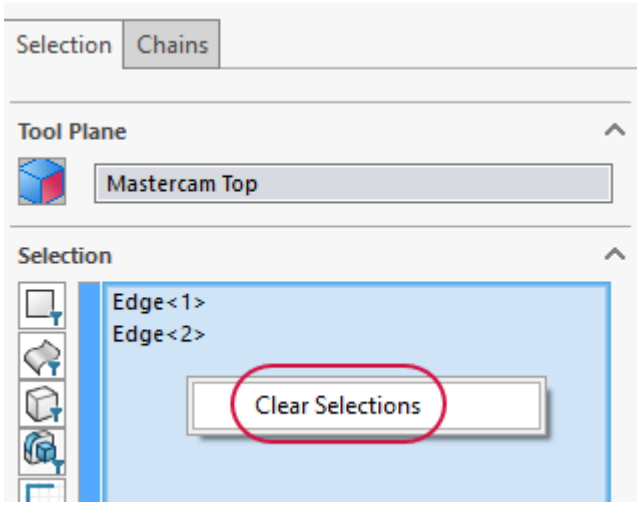
Before regenerating the toolpath, you will re-chain the third operation.

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

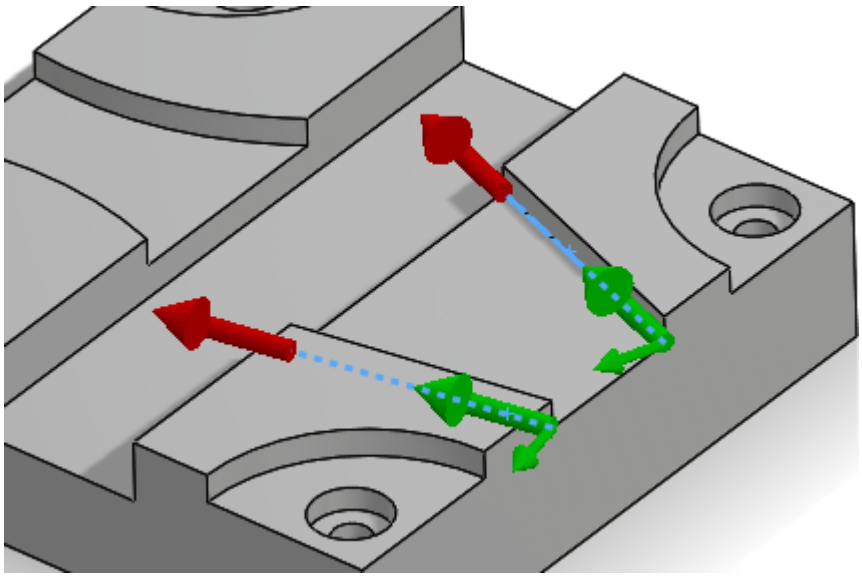



The Chain Manager displays.

6. Right-click in the Selection list and select **Clear Selections**.

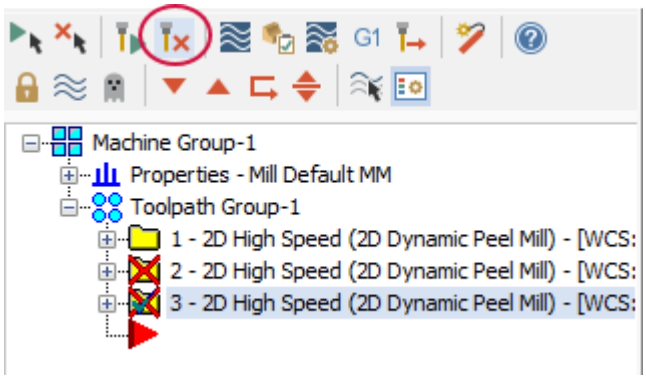



7. Select the two edges shown below. You may need to rotate the part to select them.

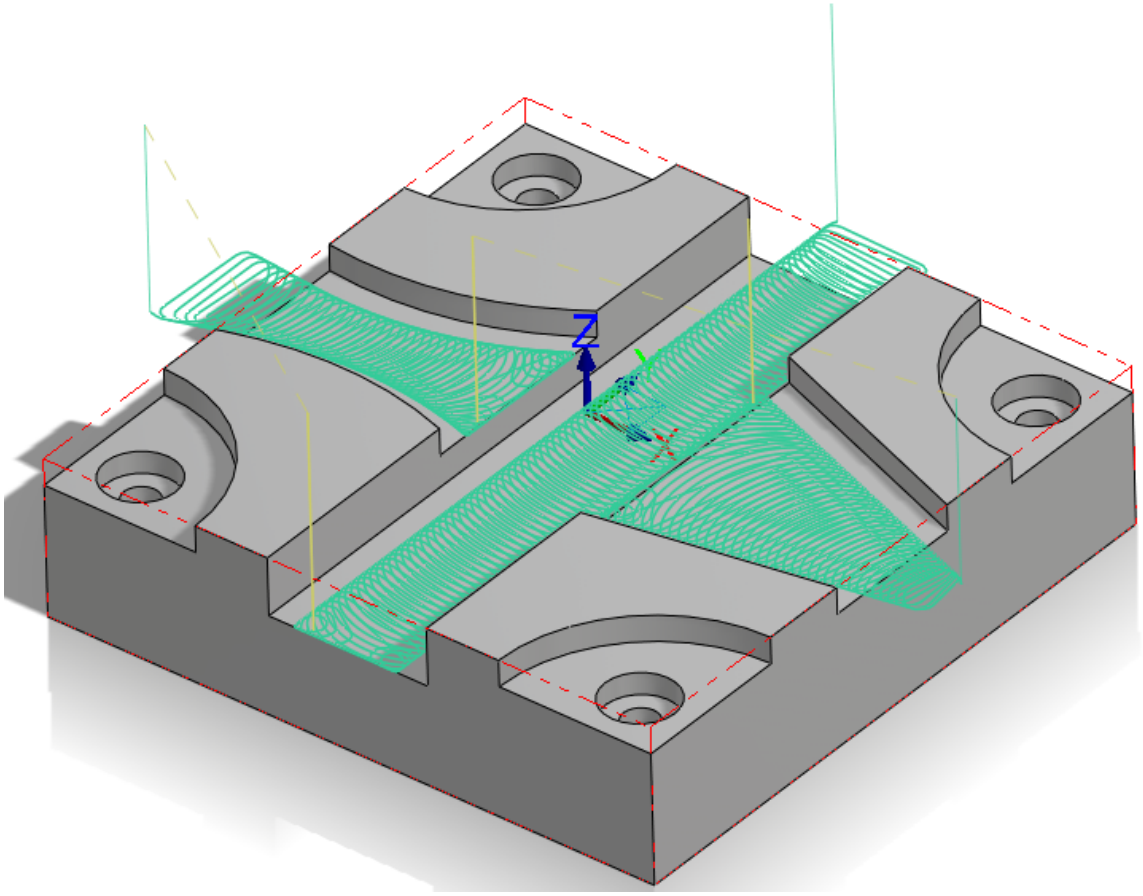


Use the Chains tab to ensure that both chains are going in the same direction.

8. Click **OK** in the Chain Manager.
9. 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:

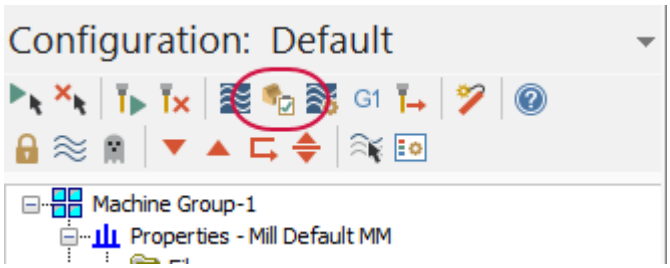



10. Save your file.

## Exercise 5: Verifying the toolpaths

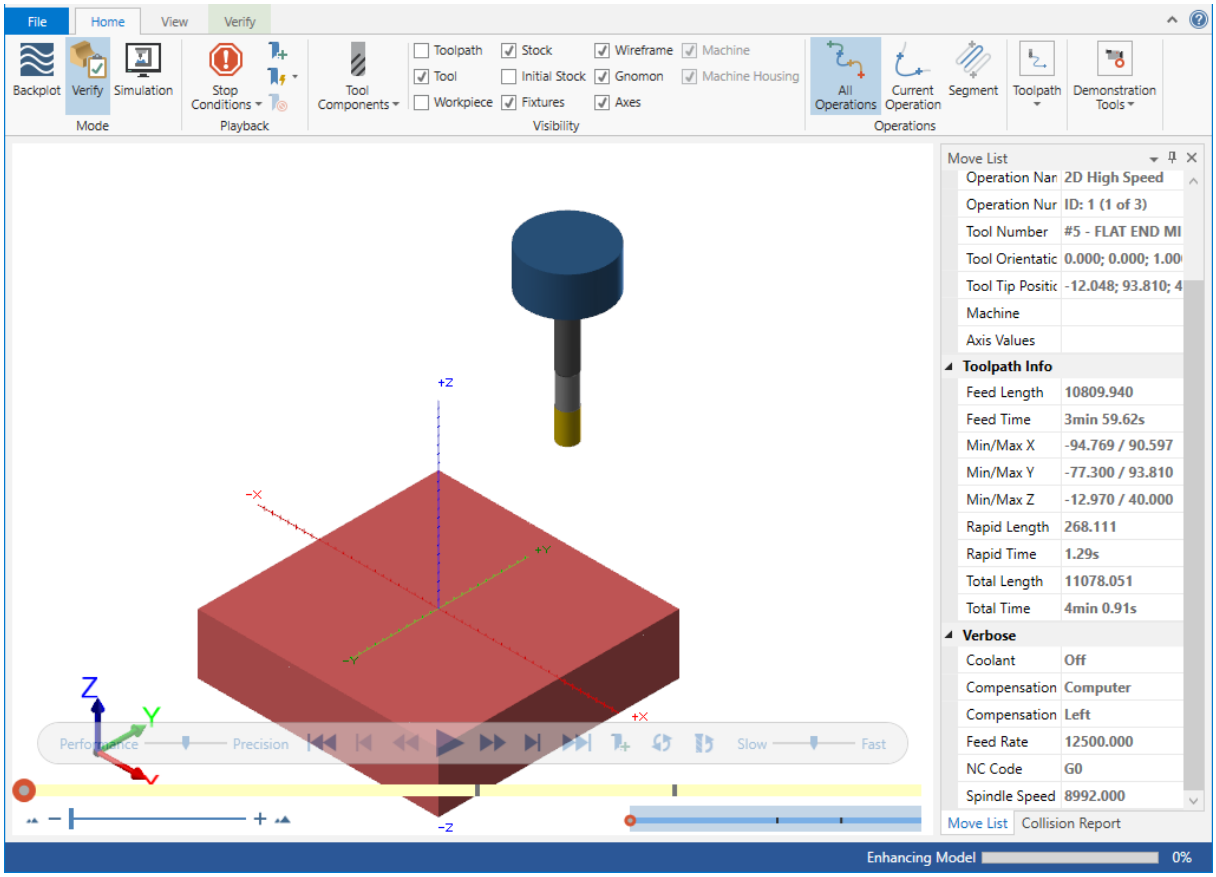
In this exercise, you will verify all three Peel Mill operations together.

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

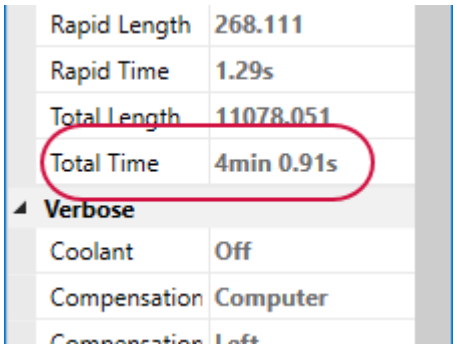



Mastercam Simulator displays.

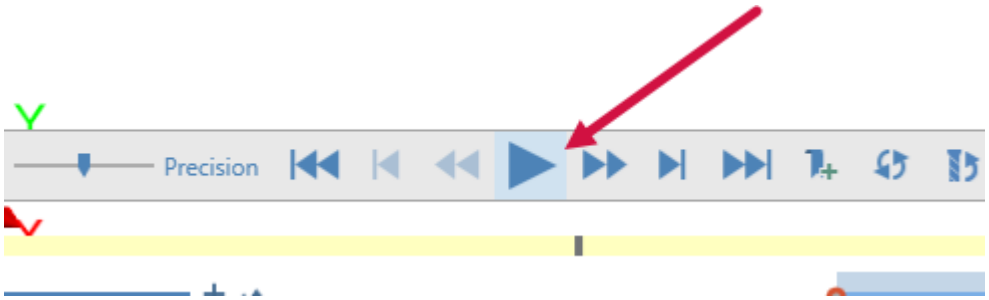
3. Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:



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




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.

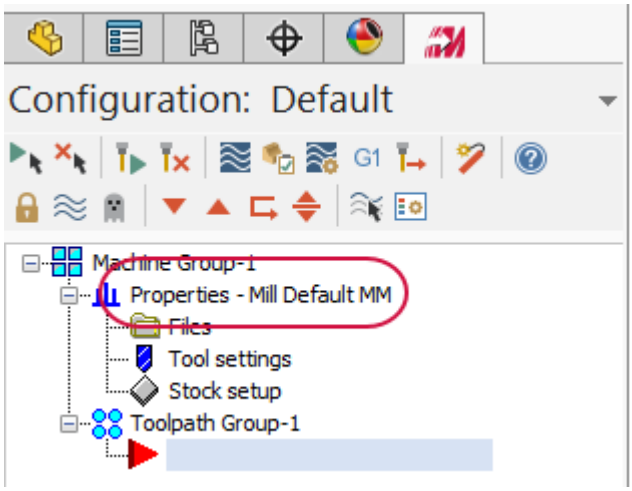
## Lesson Goals

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

## Exercise 1: Setting up the stock

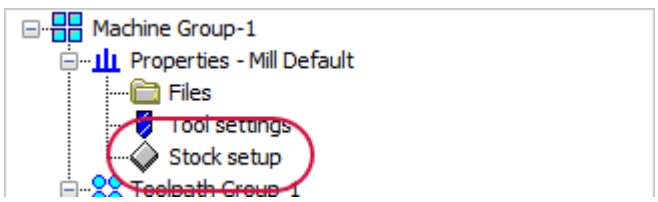
In this exercise, you will create the stock model that will be used for the first Dynamic OptiRough toolpath.

1. Open the part file, `DynamicOptiRough`, provided with this tutorial.
2. Save the part as `DynamicOptiRough-xxx`, replacing `xxx` with your initials.



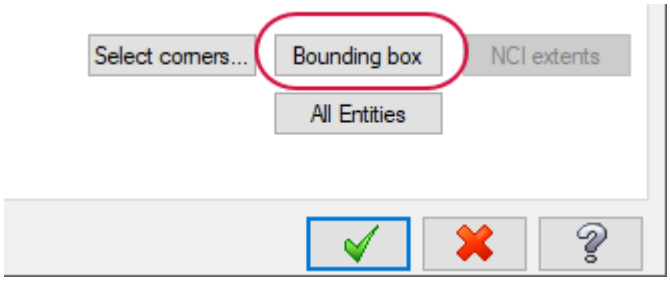

Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.

3. If you do not have the correct default machine, select **Mastercam2019**, **Mill Machines**, **Default** from the **Tools** menu.
4. Select **Stock setup** from the Toolpaths Manager.



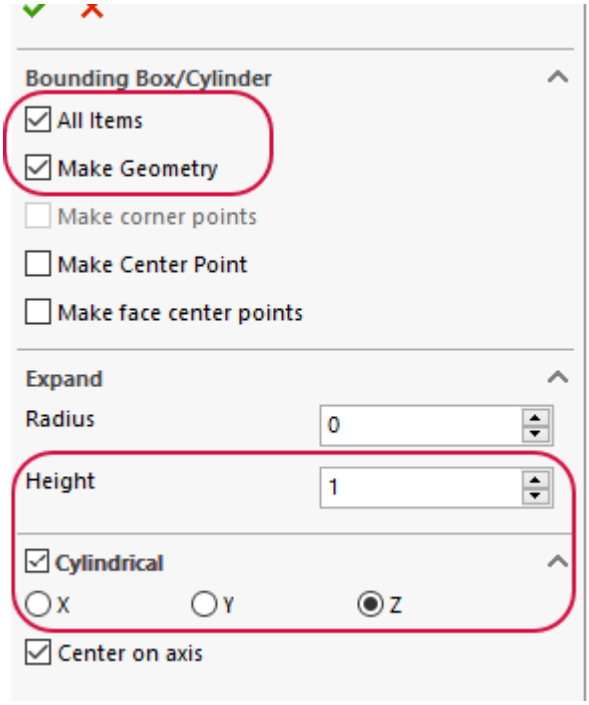
The **Machine Group Properties** dialog box displays.

5. Select **Bounding box**.




The **Bounding Box/Cylinder** function panel displays.

6. Set the following parameters:

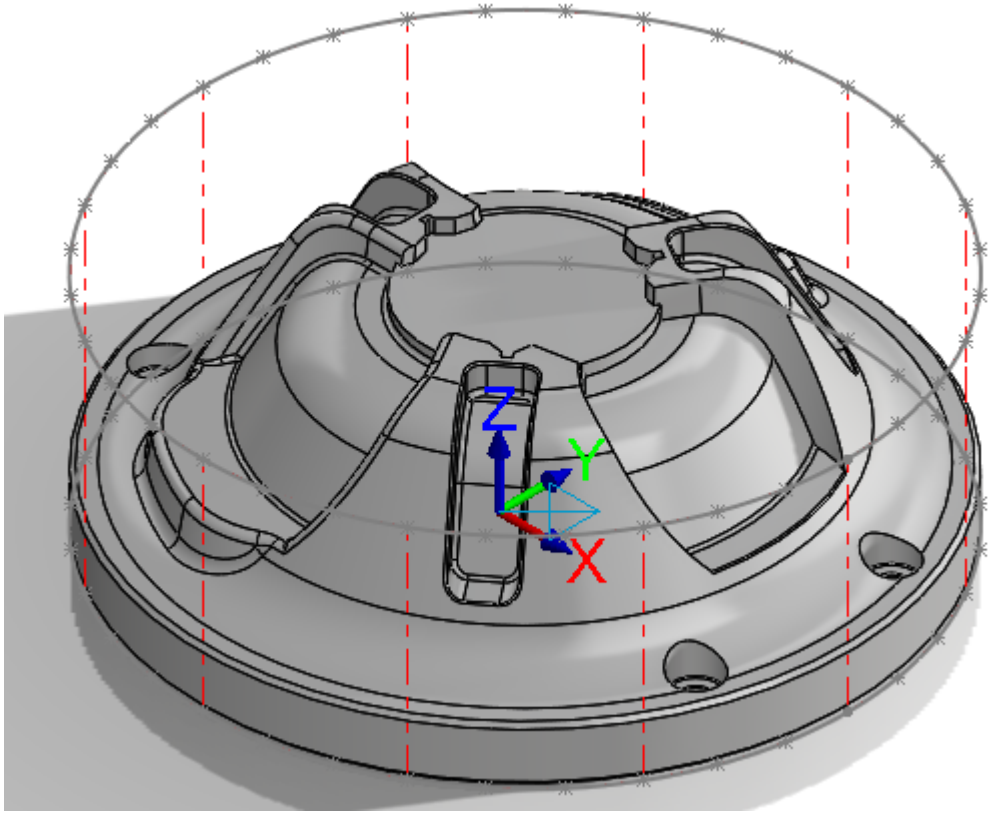



- Ensure that **All Items** is selected.
- Select **Make Geometry**.
- Set **Height** to **1.0**.
- Select **Cylindrical** and the **Z** axis.

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

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

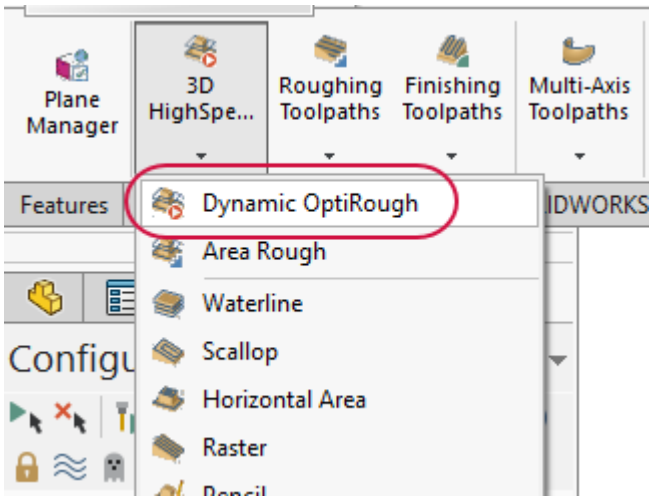
The stock model displays as a dashed outline in red. The geometry that was created with the Bounding Box also displays.



## Exercise 2: Creating the first Dynamic OptiRough toolpath

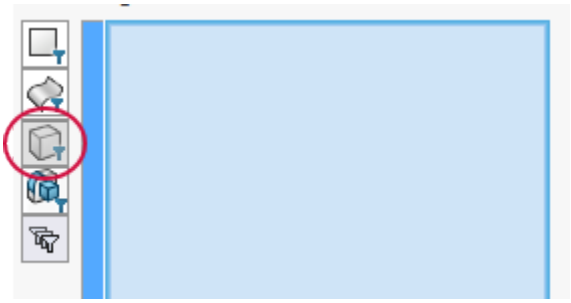
In this exercise, you create the first Dynamic OptiRough toolpath that uses this stock boundary.

1. Select **Dynamic OptiRough** from the **3D HighSpeed** menu on the **Mastercam2019** tab.



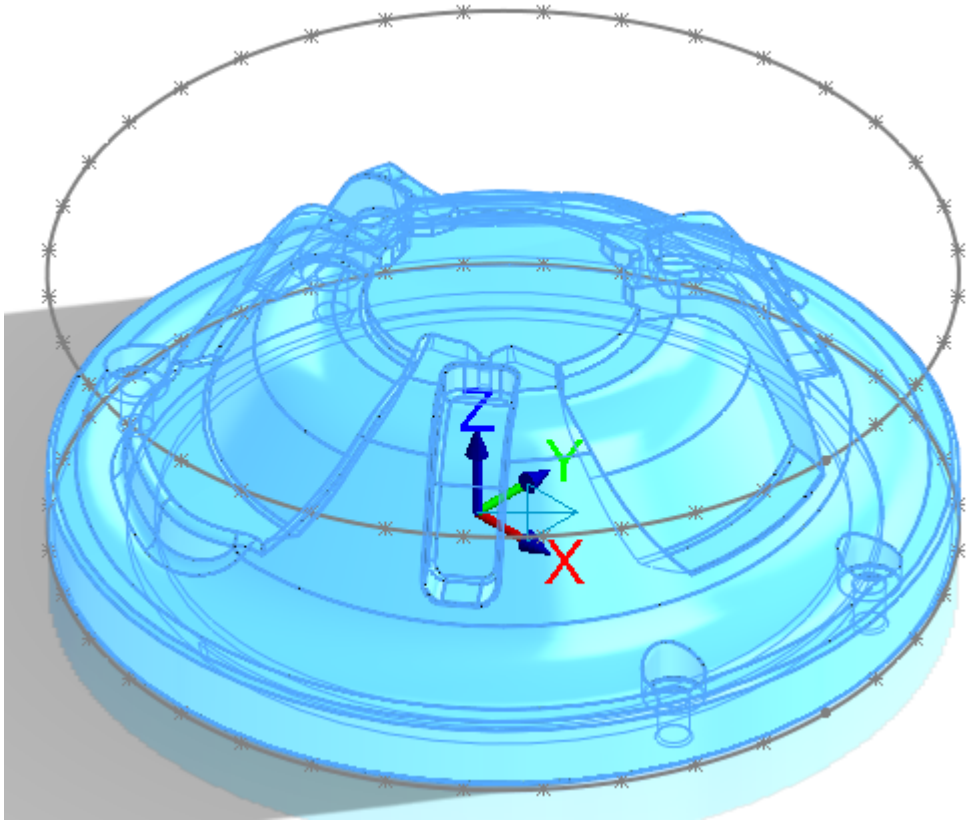

   The **Selection** panel displays.

2. In the Selection panel, activate the **Bodies** filter.



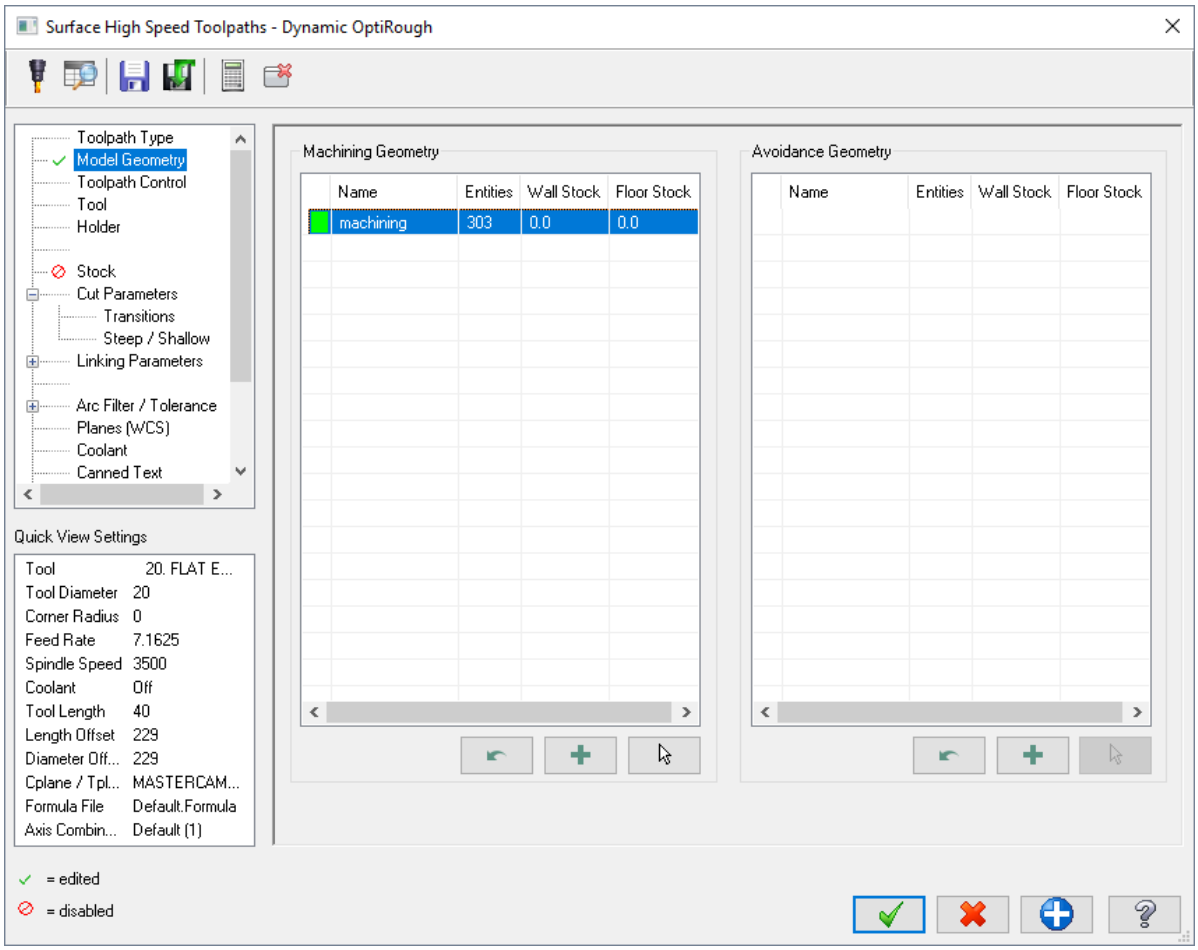
   This limits your geometry selection to only bodies.

3. Select the part:

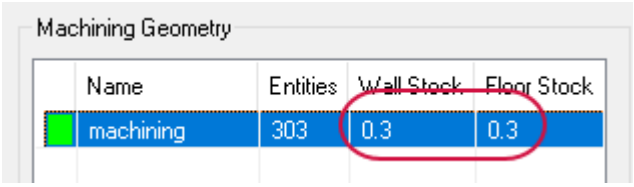



4. Click **OK** in the **Selection** panel.

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

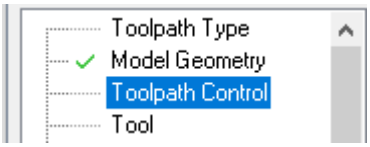



6. Since you are creating a roughing toolpath, enter the following stock to leave amounts on the wall and floor. Double-click to activate the field.

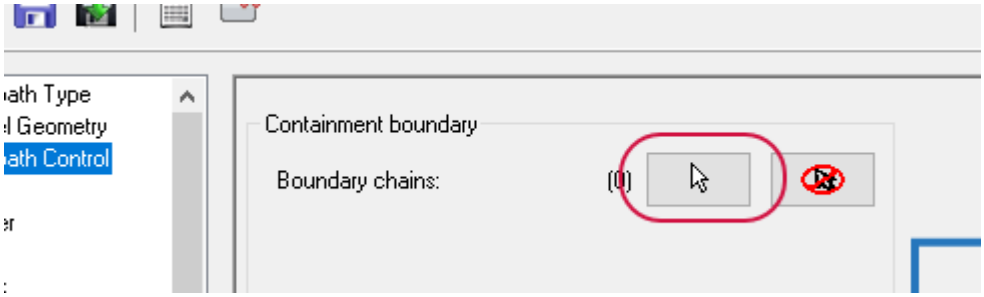



- Enter **0.3** in **Wall Stock**
- Enter **0.3** in **Floor Stock**

7. Select the **Toolpath Control** page.

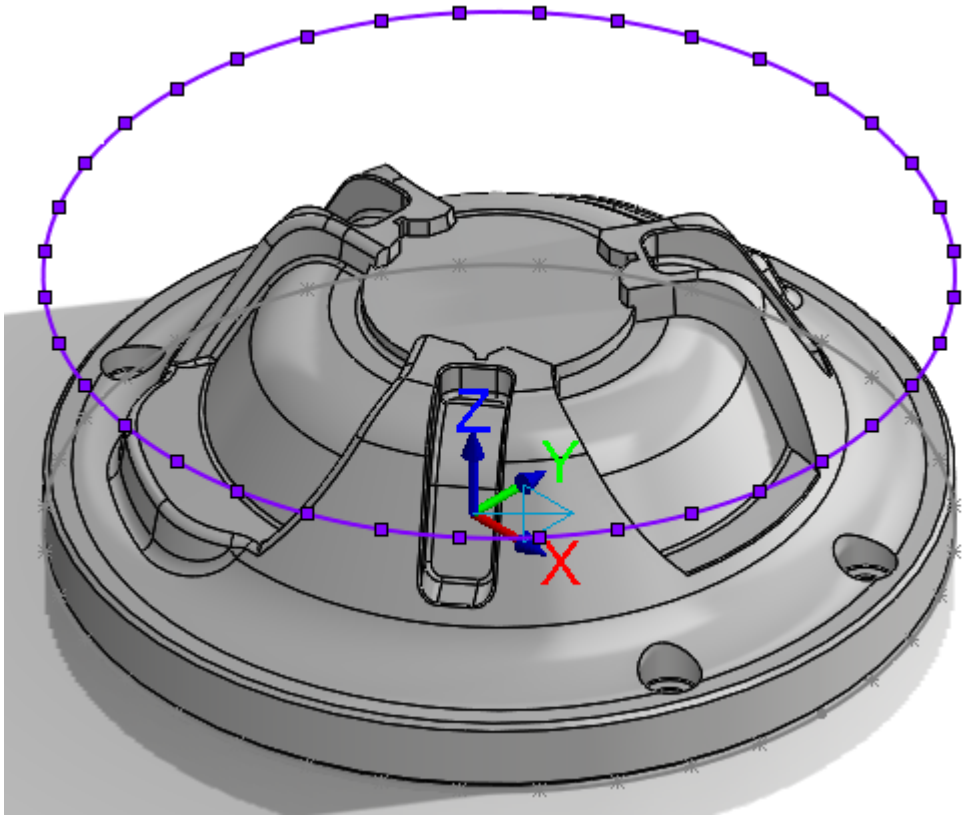


8. Click the **Boundary chains** selection arrow.




This opens the **Selection** panel to select one or more closed chains of curves to limit the tool motion.

9. Select the spline shown below to be the containment boundary.

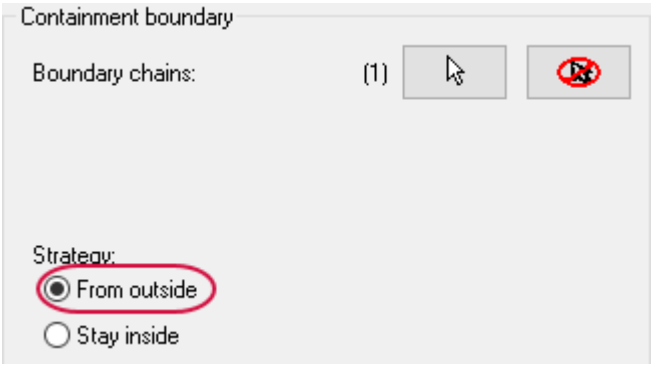



The containment boundary limits tool motion to a selected area. In this case, you want to limit the tool motion to around the part.

10. Click **OK** in the **Selection** panel.

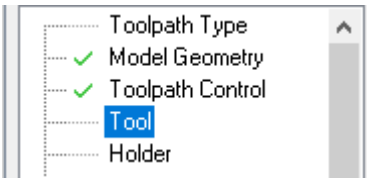
You return to the **Toolpath Control** page of the **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box.

11. 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.

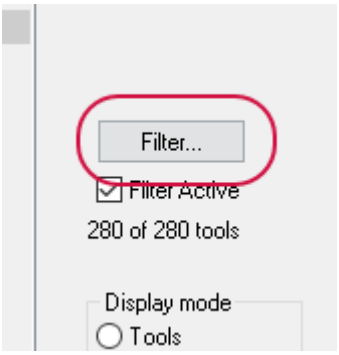
12. Select the **Tool** page.




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

    The **Tool Selection** dialog box displays.

14. Select **Filter**.

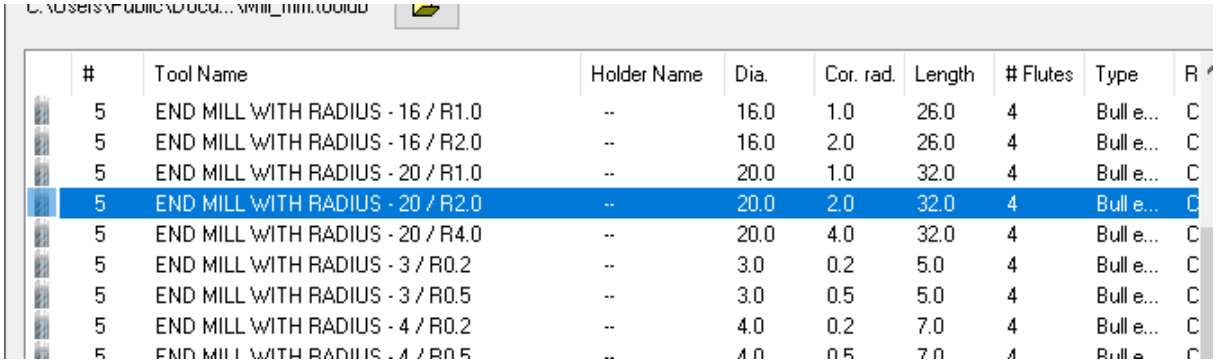



    The **Tool List Filter** dialog box displays.

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

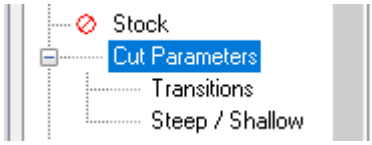
    You return to the **Tool Selection** dialog box.

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

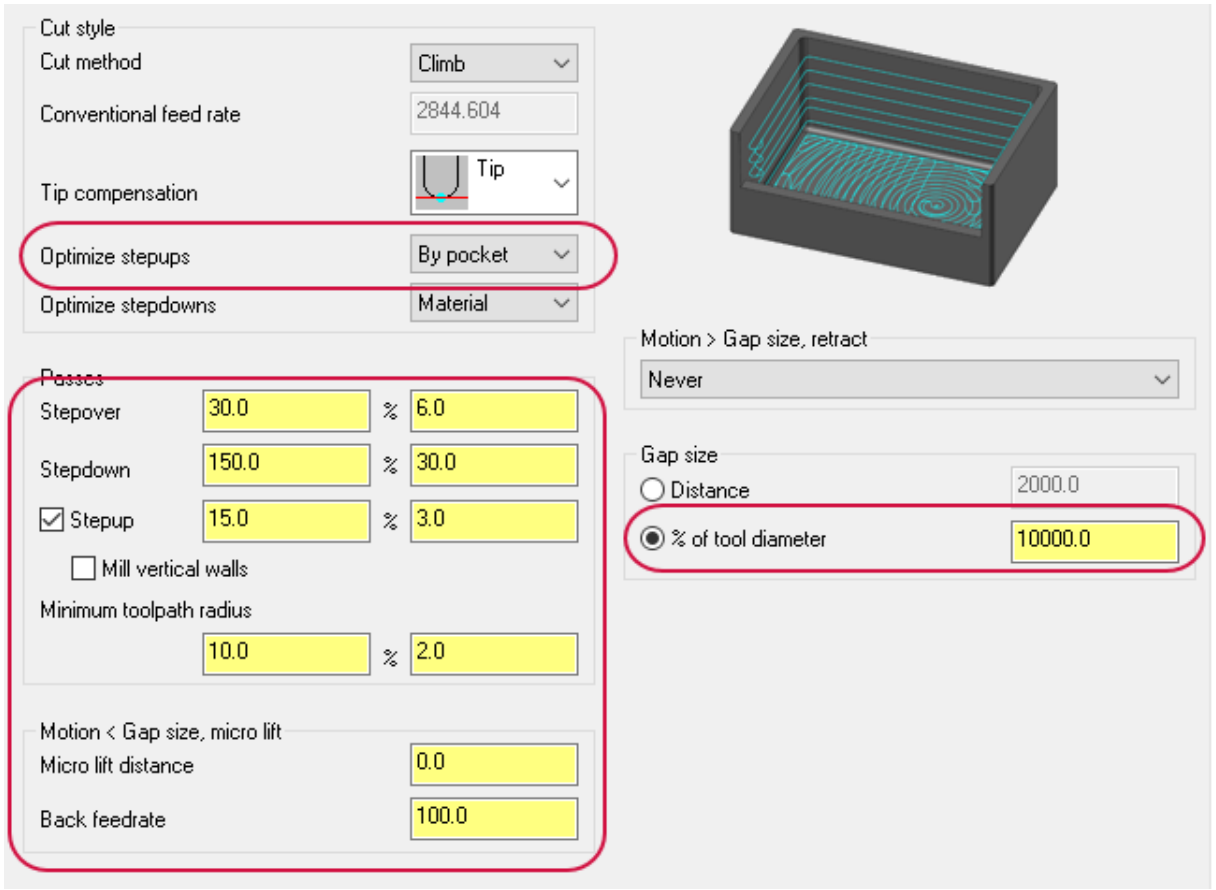


| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Type | R |
|---|---|---|---|---|---|---|---|---|
| 5 | END MILL WITH RADIUS - 16 / R1.0 | -- | 16.0 | 1.0 | 26.0 | 4 | Bull e... | C |
| 5 | END MILL WITH RADIUS - 16 / R2.0 | -- | 16.0 | 2.0 | 26.0 | 4 | Bull e... | C |
| 5 | END 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 |

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



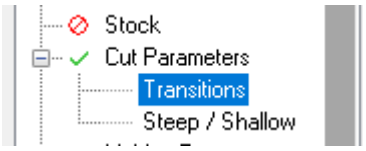

21. Set the following parameters:



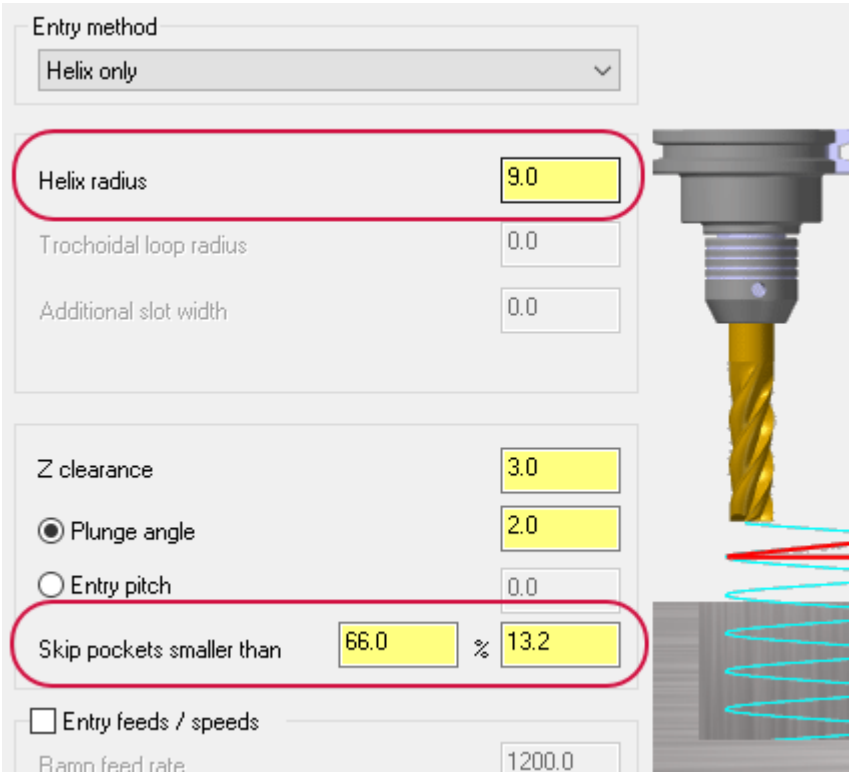

- **Optimize stepups** to **By pocket**. Mastercam for SOLIDWORKS first machines all stepdowns, moving from pocket to pocket. After all stepdowns on a Z-level are machined to completion, Mastercam machines the stepups by next closest, in the safest cut order.
- **Stepover** to **30.0%**. Sets the distance 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 for SOLIDWORKS 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**.

22. Select the **Transitions** page.

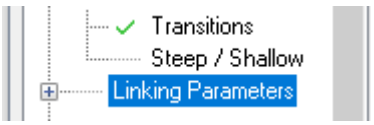



23. Set the following parameters:

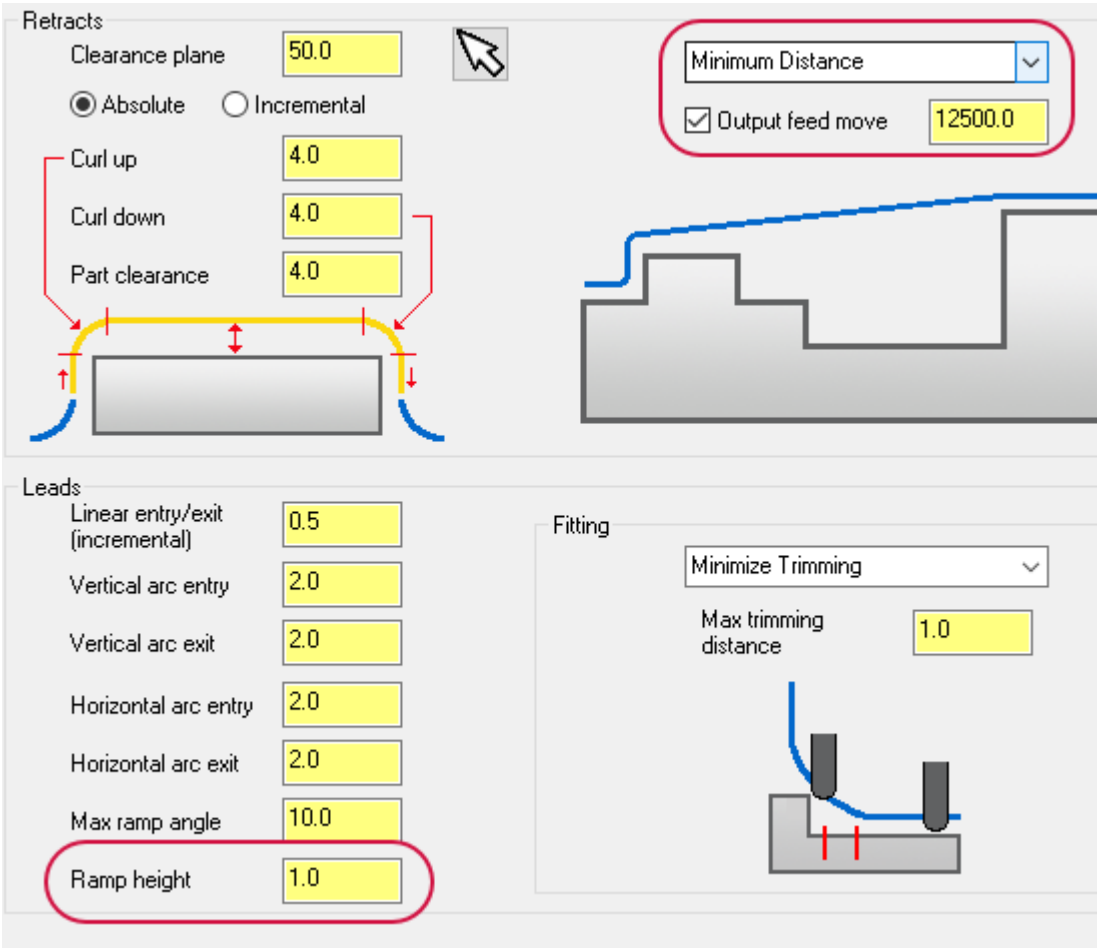



- **Helix radius** to **9.0**. Sets the radius of the entry helix.
- **Skip pockets smaller than** to **66.0**. Specifies a minimum pocket size that Mastercam will create a cutting pass for. This helps with problems where Mastercam thinks that a pocket is large enough to accommodate the tool, but the entry move is so compressed the tool is effectively plunging into the part.

24. Select the **Linking Parameters** page.



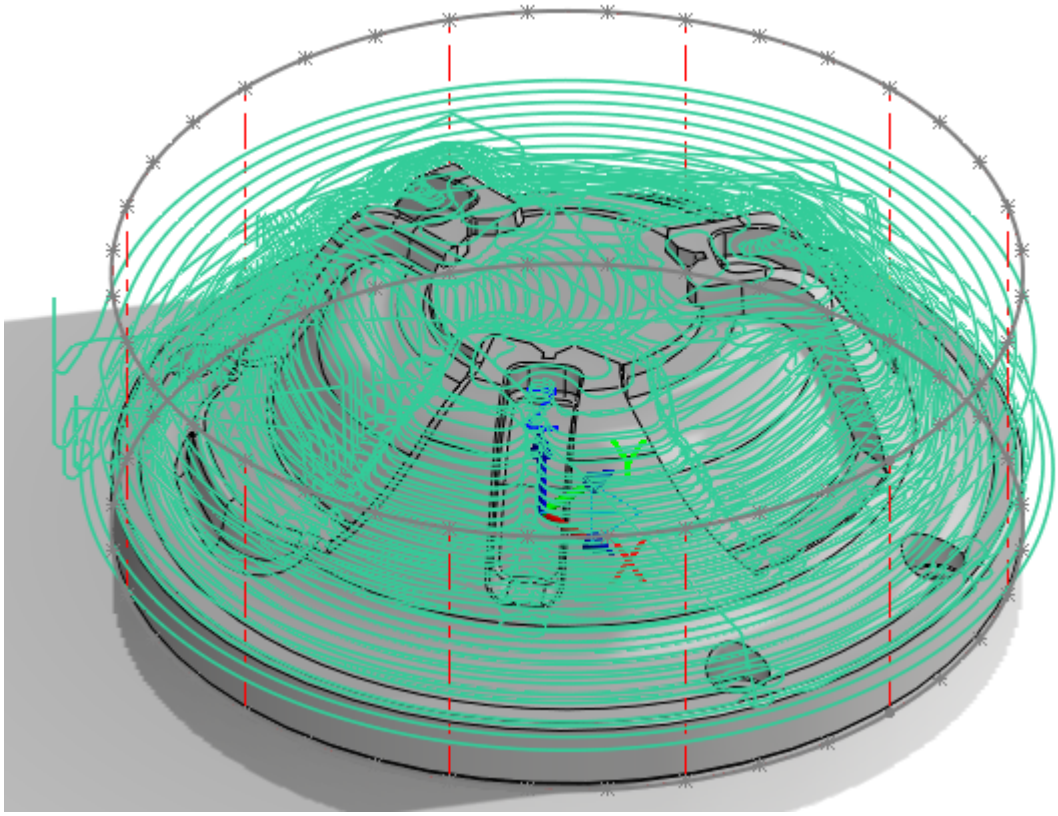
25. Set the following parameters:




- Set the **Retract Method** drop-down to **Minimum Distance**. Creates high-speed loops into and out of each retract move.
- Select **Output feed move** and enter **12500.0**. Outputs the rapid move between passes as a feed rate move instead of a rapid move. This helps when the tool needs to make many irregular moves per pass to jump between different areas of the part.
- Set **Ramp height** to **1.0**. Sets the height of the helix when moving into a new region.

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

27. Your toolpath should display as follows:



28. Save your part file.

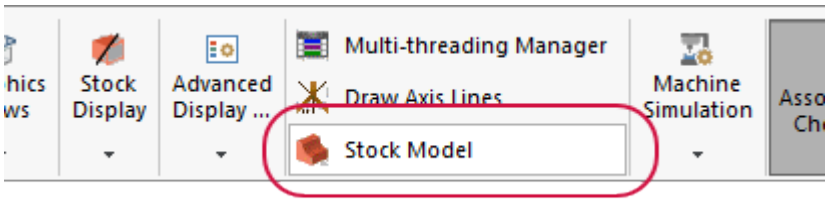
## Exercise 3: Creating the Stock Model operation

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 **Mastercam2019** 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.

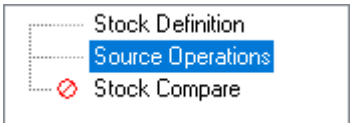
4. Set **Color** to **128**.

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

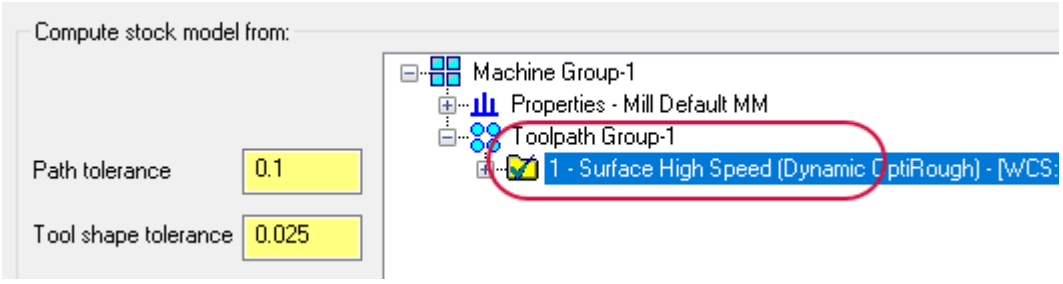
5. Select **Stock Setup**.

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.



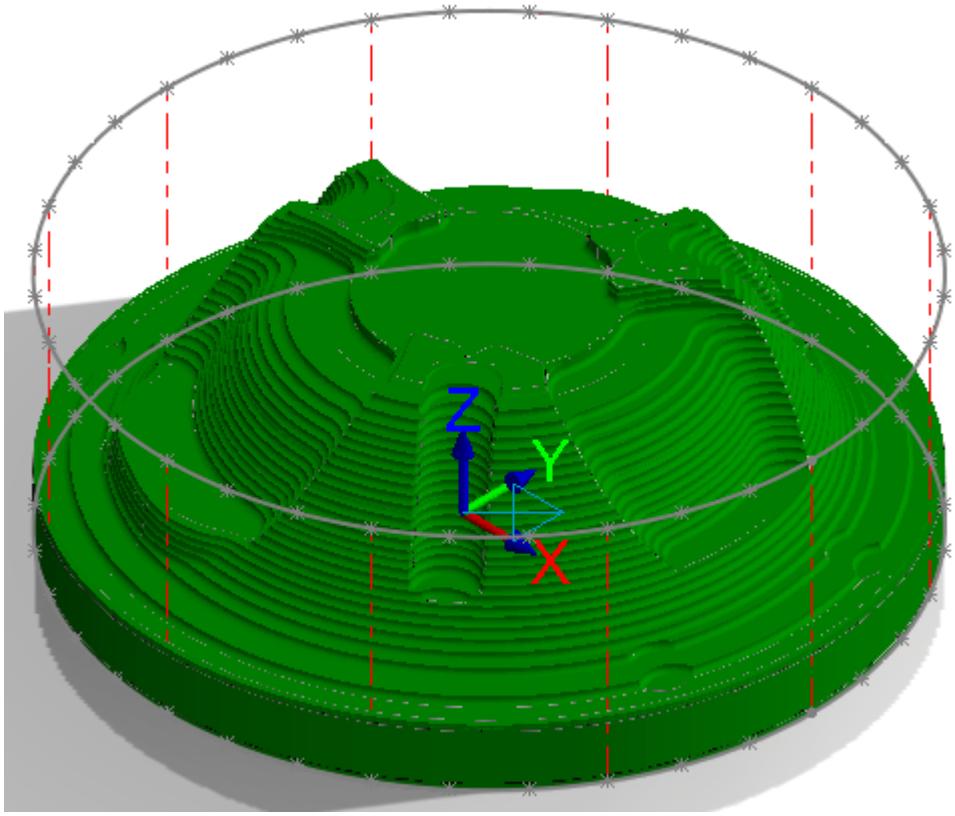

7. Select the **Surface High Speed (Dynamic OptiRough)** toolpath as the source operation.




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:

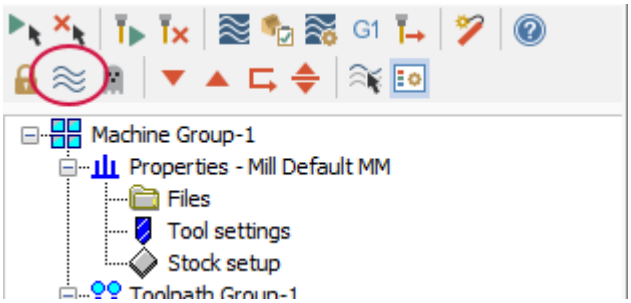



10. Turn off **Only display selected toolpaths** from the Toolpaths Manager.
11. Save your part.

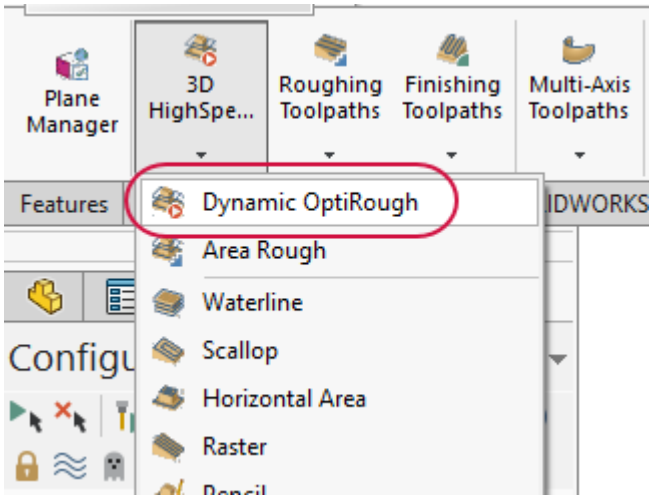
## Exercise 4: Creating the second Dynamic OptiRough toolpath

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.

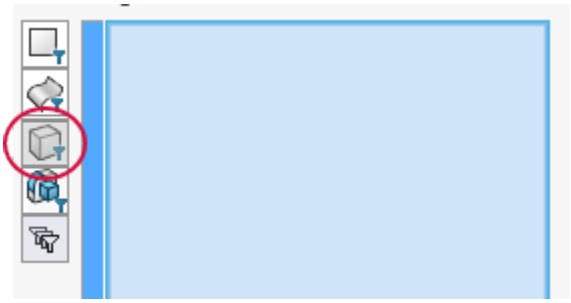


3. Select **Dynamic OptiRough** from the **3D HighSpeed** menu on the **Mastercam2019** tab.

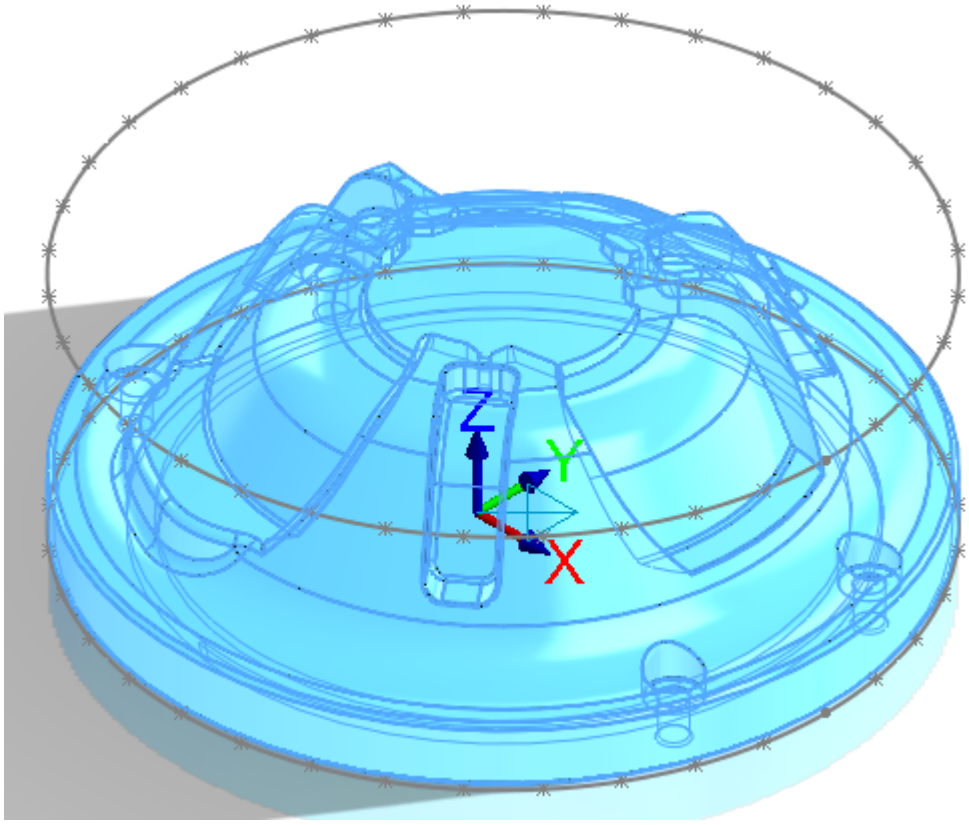



The Selection panel displays.

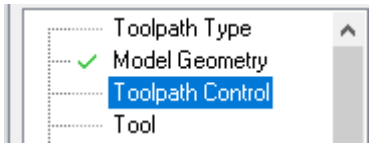
4. In the Selection panel, ensure the **Bodies** filter is activated.



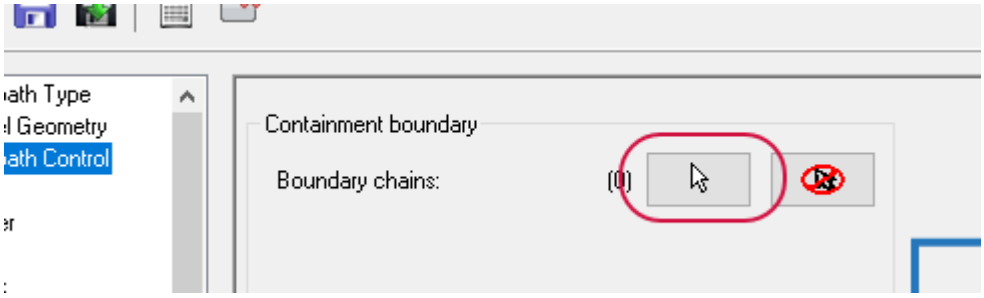
5. Select the part as shown below:



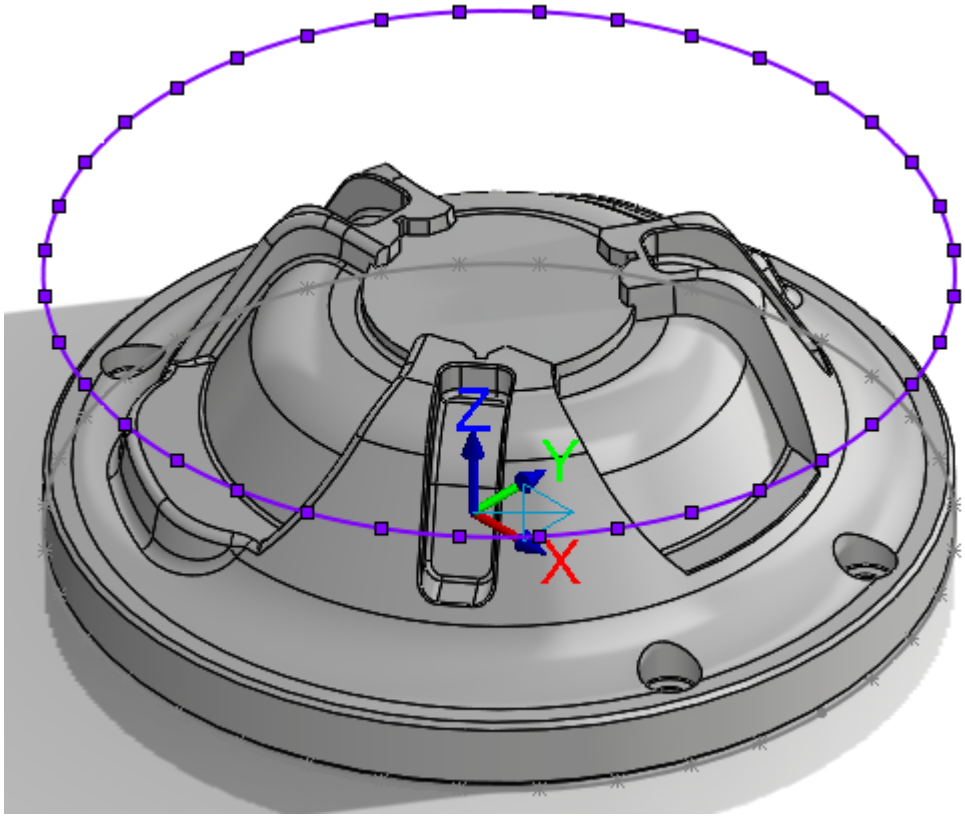

6. Click **OK** in the Selection panel. The **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.
7. Select the **Toolpath Control** page.



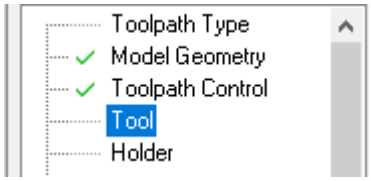

8. Click the selection arrow for **Boundary chains** to return to the graphics window.



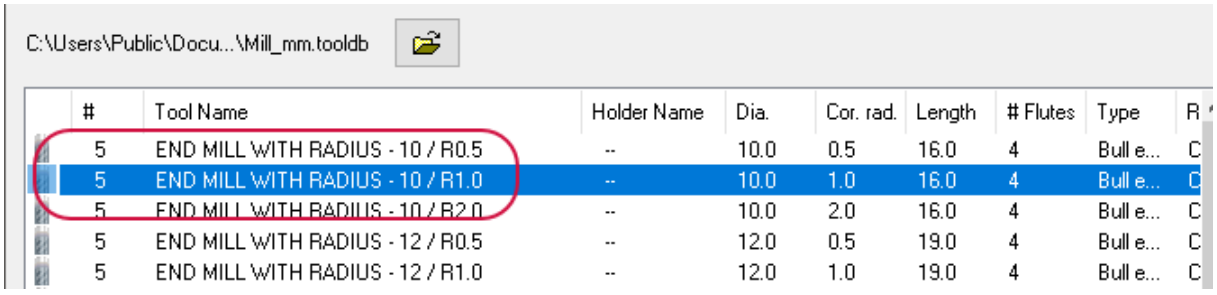
9. Select the spline shown below as the containment boundary:




10. Click **OK** in the Selection panel to return to the toolpath dialog box.
11. Select the **Tool** page.




12. Click **Select library tool**.
13. In the **Tool Selection** dialog box, select the **END MILL WITH RADIUS - 10/R1.0** tool.

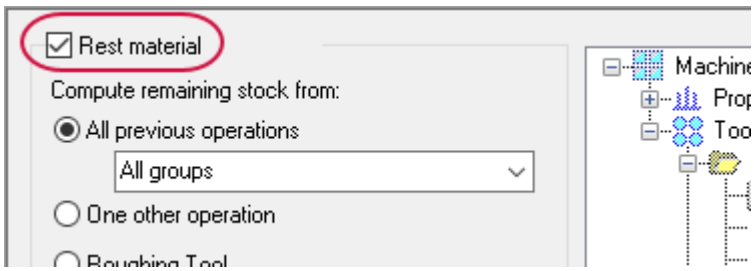



| # | Tool Name | Holder Name | Dia. | Cor. rad. | Length | # Flutes | Type | R |
|---|---|---|---|---|---|---|---|---|
| 5 | END MILL WITH RADIUS - 10 / R0.5 | -- | 10.0 | 0.5 | 16.0 | 4 | Bull e... | C |
| 5 | END MILL WITH RADIUS - 10 / R1.0 | -- | 10.0 | 1.0 | 16.0 | 4 | Bull e... | C |
| 5 | END MILL WITH RADIUS - 10 / R2.0 | -- | 10.0 | 2.0 | 16.0 | 4 | Bull e... | C |
| 5 | END MILL WITH RADIUS - 12 / R0.5 | -- | 12.0 | 0.5 | 19.0 | 4 | Bull e... | C |
| 5 | END MILL WITH RADIUS - 12 / R1.0 | -- | 12.0 | 1.0 | 19.0 | 4 | Bull e... | C |

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

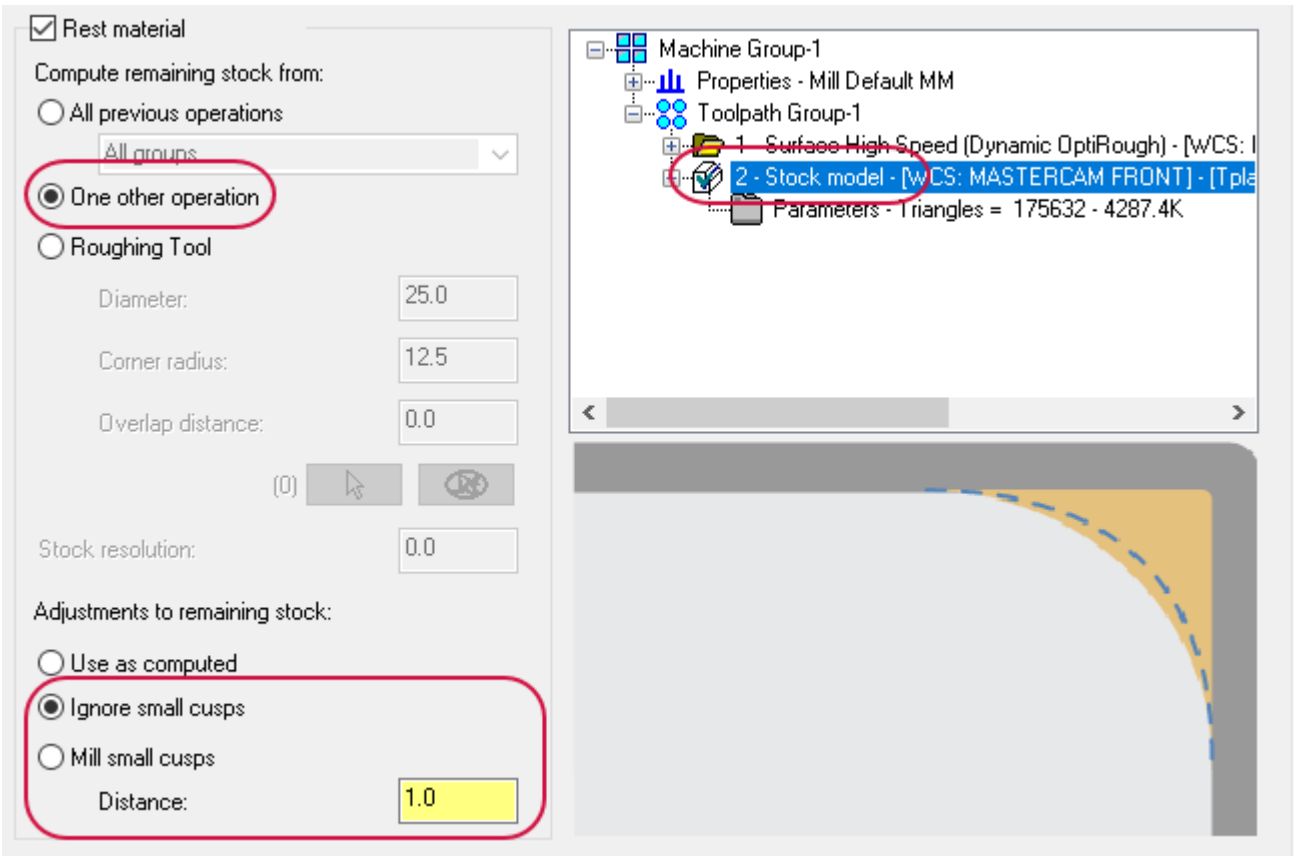
15. Select the **Stock** page.

16. Select **Rest material**.



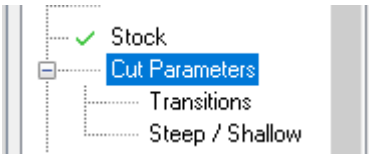

When you enable this page, Mastercam calculates the cutting passes based on the remaining stock.

17. Set the following parameters:

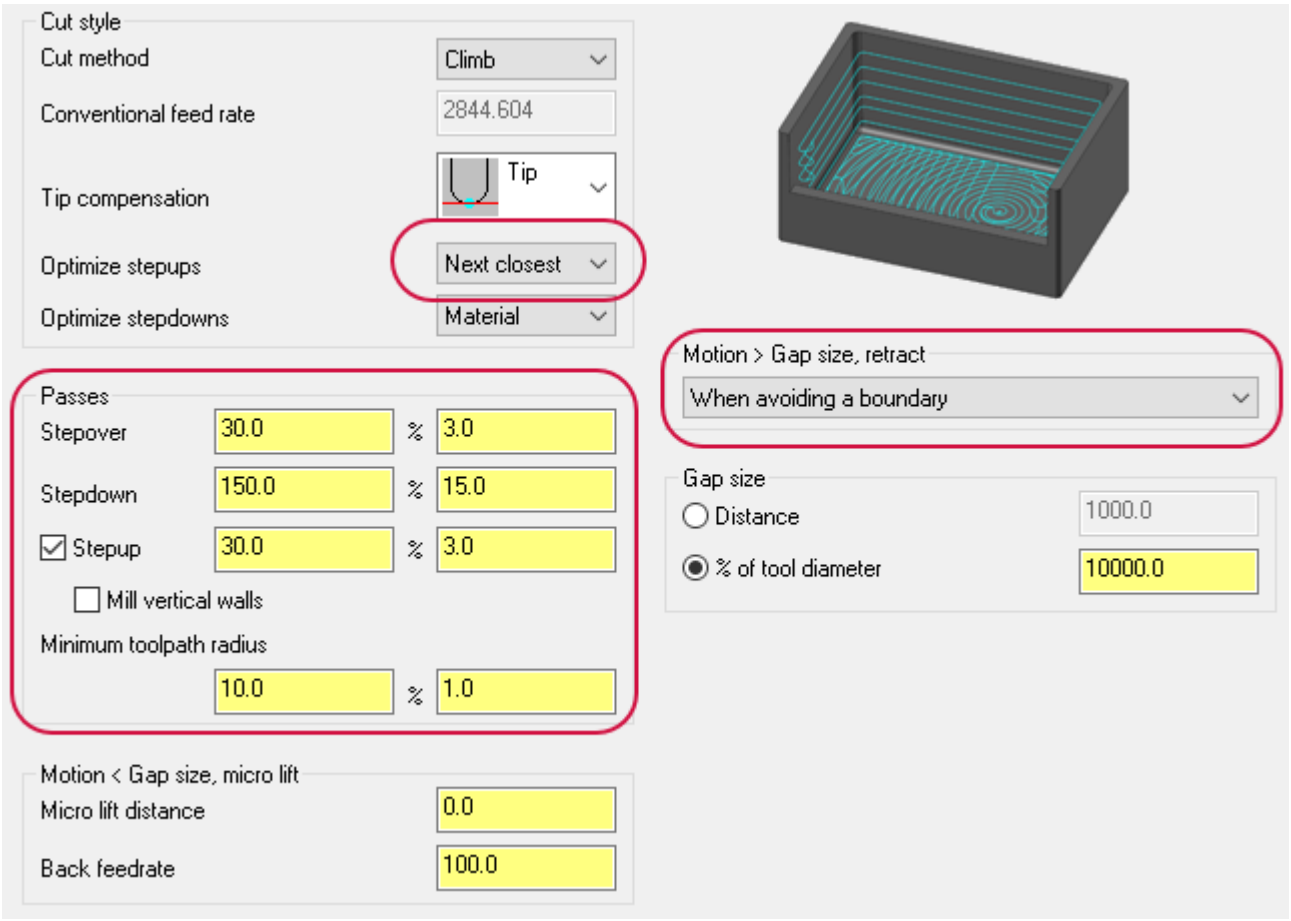



- Set **Compute remaining stock from** to **One other operation**.
- In the **Operation List**, select **Stock Model - OP1 Stock**.
- Set **Adjustments to remaining stock** to **Ignore small cusps**.
- Set **Distance** to **1.0**. This option is used in conjunction with **Ignore small cusps**. Mastercam will output cuts that engage large amounts of material.

18. Select the **Cut Parameters** page.

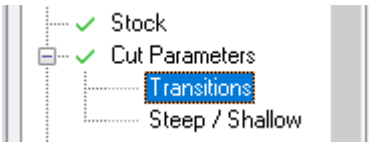



19. Set the following parameters:

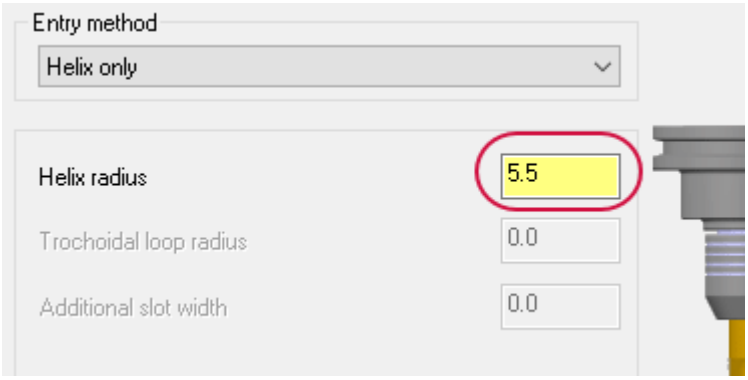



- **Optimize stepups** to **Next closest**. Mastercam for SOLIDWORKS moves to the closest cut from its last position on the previous cut.
- **Stepover** to **30.0%**.
- **Stepdown** to **150.0%**.
- **Stepup** to **30.0%**.
- **Minimum toolpath radius** to **10.0%**.
- **Motion > Gap size, retract**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

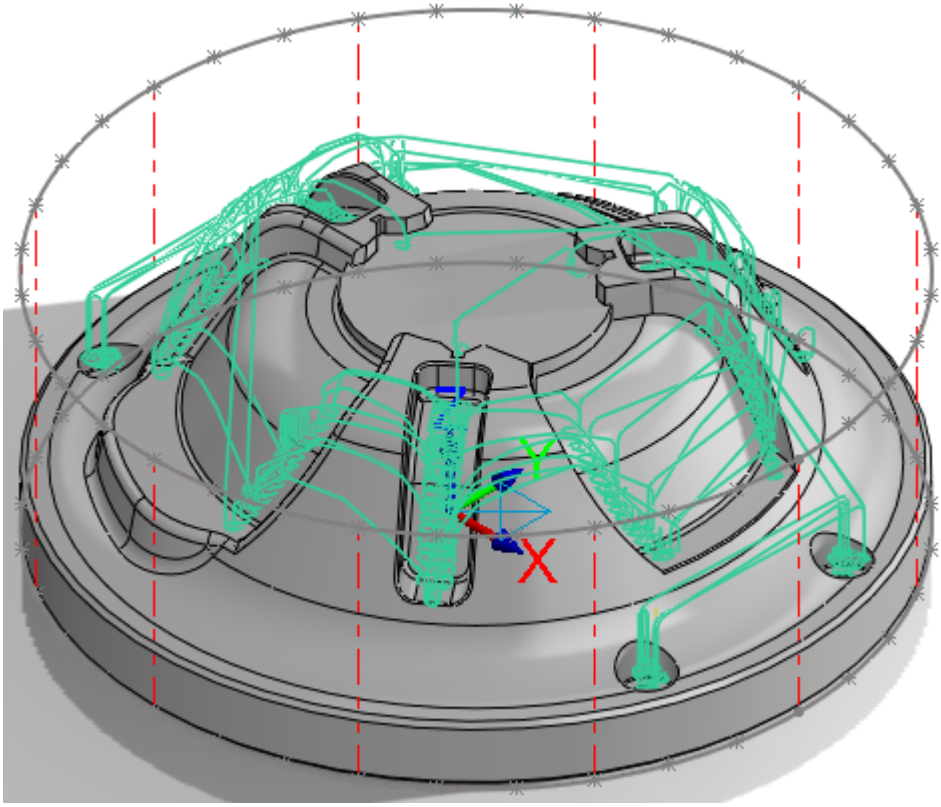
20. Select the **Transitions** page.



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




22. Click **OK** to create and generate the Dynamic OptiRough toolpath.
23. Select **Toggle display on selected operations** to display toolpaths again, and select the second Dynamic OptiRough toolpath to view the newly created toolpath.

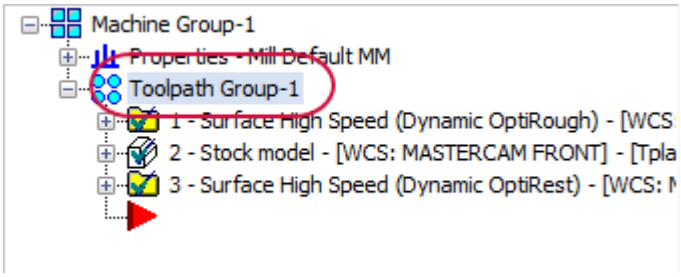



24. Save your part file.

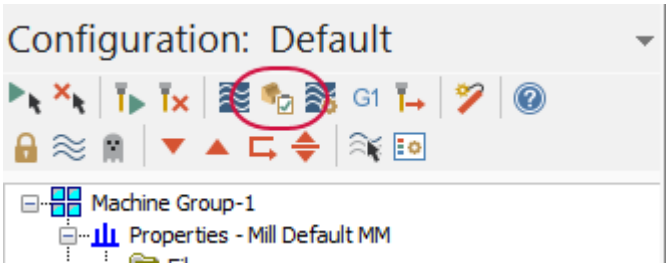
## Exercise 5: Verifying the stock model and toolpaths

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.

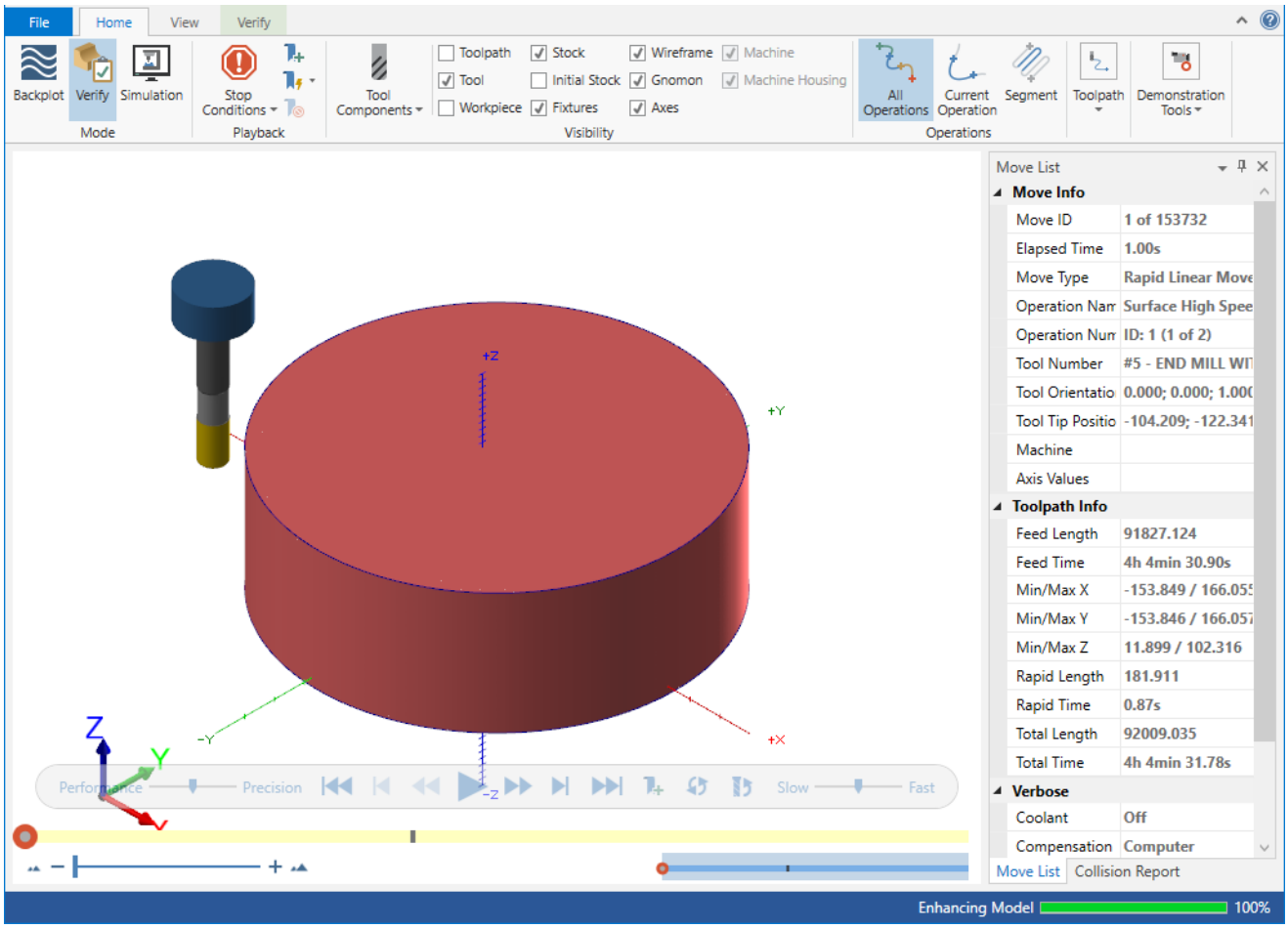



2. Select **Verify selected operations** in the Toolpaths Manager.

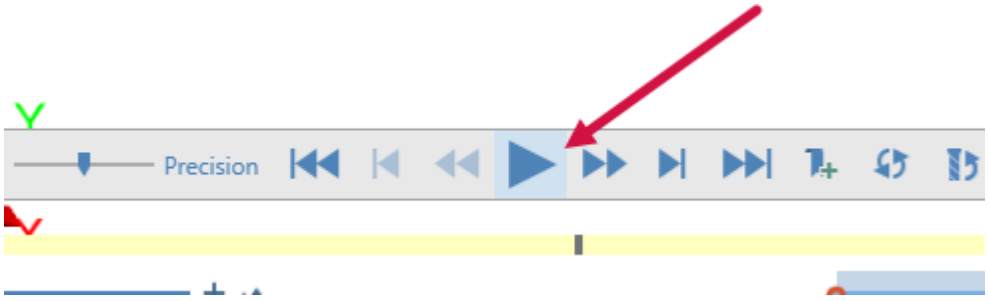



Mastercam Simulator displays.

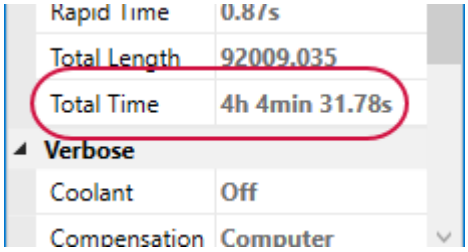
3. If necessary, right-click in the graphics window and change the view to **Isometric** and **Fit** the part to the screen so that it matches the image shown below.




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.




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

# CONCLUSION

Congratulations! You have completed the *Mastercam for SOLIDWORKS Dynamic Milling*! 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 to see the latest publications.

**ATTENTION! UPDATES MAY BE AVAILABLE. PLEASE REFER TO MASTERCAM.COM/SUPPORT FOR THE LATEST DOWNLOADS.**

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

www.mastercam.com

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