Mastercam 2017 for SOLIDWORKS Tutorial (Mill)

May 2016

Mastercam 2017

Mastercam® 2017 Mastercam 2017 for SOLIDWORKS Tutorial (Mill)

Date: May 2016

Software: Mastercam 2017 for SOLIDWORKS

TERMS OF USE

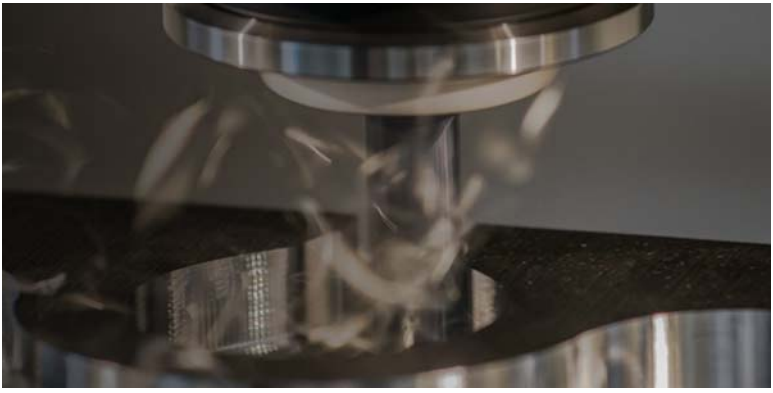
Be sure you have the latest information!

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

Contents

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

Estimated time to complete this tutorial: 5 hours

General Tutorial Requirements

All Mastercam for SOLIDWORKS tutorials have the following general requirements:

- You must be comfortable using the Windows® operating 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 preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam 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

The following toolpaths use Dynamic Motion technology:

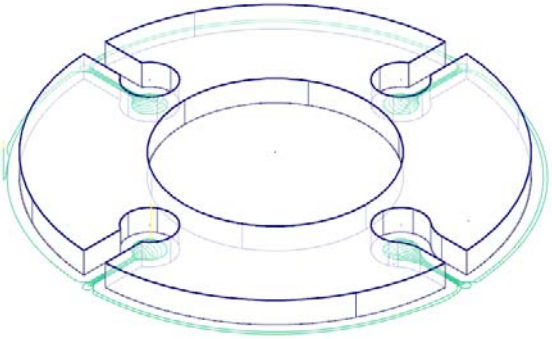
- Dynamic Contour
- Face
- Dynamic Mill
- Peel Mill
- Dynamic OptiRough

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

Dynamic Contour

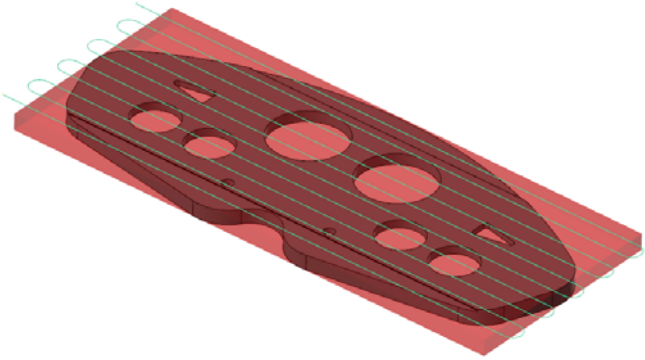
The Dynamic Contour toolpath creates a familiar contour operation using dynamic motion and specialized options found only in dynamic toolpaths. The dynamic motion

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.

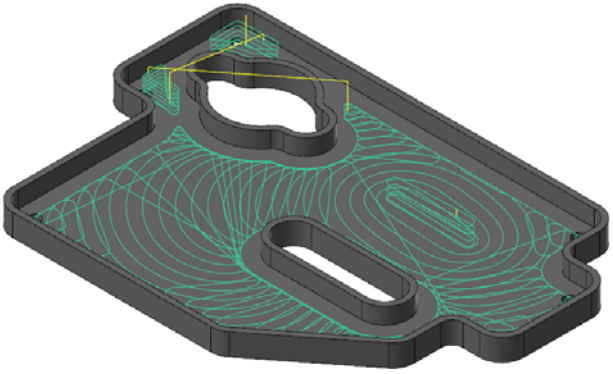


Dynamic Mill

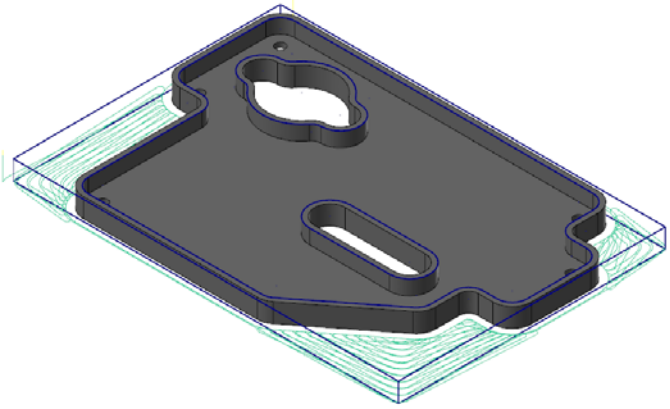
The Dynamic Mill toolpath machines pockets, leftover material, standing bosses, or cores. By setting the Machining region strategy to **Stay inside** or **From outside**, you can create a pocket toolpath that utilizes dynamic motion or a facing operation that has island avoidance capabilities.

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, param-

eters, entry method, and linking values to generate the toolpath. Material is removed from the inside to the outside in a highly efficient manner.

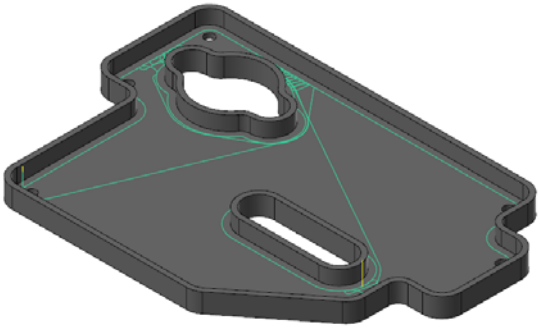


By selecting the From outside strategy, Dynamic Mill becomes a facing operation that has island avoidance capabilities. This movement creates a toolpath that is allowed to move outside the selected material and removes from the outside to the inside.



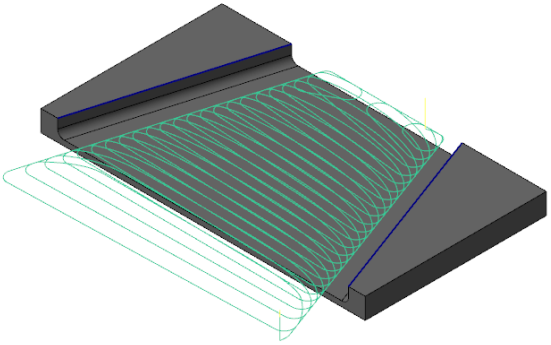
By enabling **Rest Material** on the Stock page, the Dynamic Mill toolpath removes material left by previous operations using dynamic motion. The Dynamic Mill operation calculates how much material to remove based on previous operations or

previous roughing tools. Only unmachined areas are processed for a Dynamic Mill operation.



Peel Mill

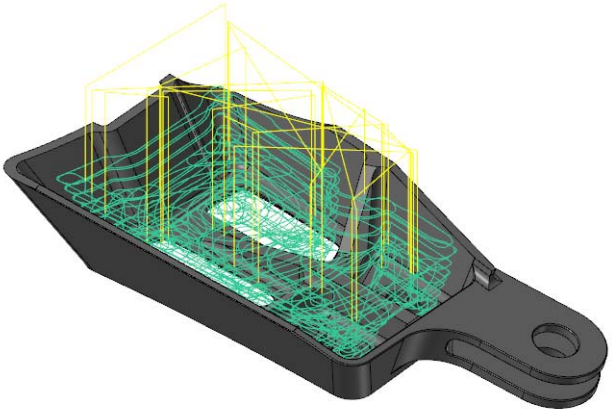
Peel Mill, with the Cutting strategy set to **Dynamic Peel**, allows for efficient milling between selected chains. It uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material. For a single chain, you define the width of the cut. Otherwise, the width is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

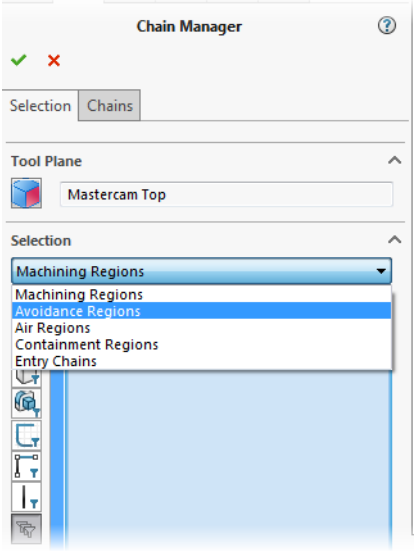


Dynamic Toolpath Selection

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:

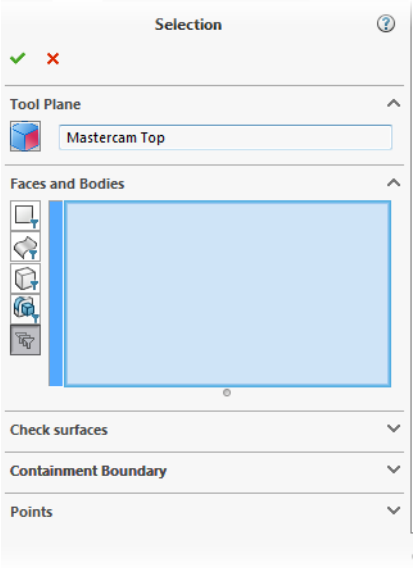


- **Machining Regions**: Areas to be machined.
- **Avoidance Regions**: Areas to be avoided during machining. Multiple avoidance regions can be selected.
- **Air Regions**: Areas that contain no material and allows the tool to travel through it when machining.
- **Containment Regions**: Areas where the tool cannot travel outside of.
- **Entry Chain**: Select 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.

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

Dynamic OptiRough uses a different type of selection method compared to the other Dynamic toolpaths, as it is the only 3D Dynamic toolpath. When creating a Dynamic OptiRough toolpath, you have the following selection options:



- **Faces and Bodies**: Surfaces, solid faces, or solid bodies to be machined.
- **Check surfaces**: Surfaces, solid faces, or solid bodies that you want to avoid.
- **Containment Boundary**: One or more closed chains that limit tool motion.
- **Points:** Select the point near where the toolpath will start.

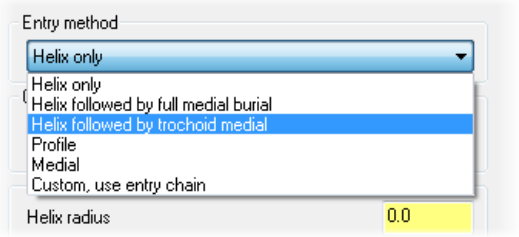
Dynamic Milling Parameters

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

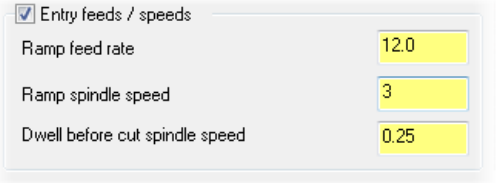
- Micro lifts
- Entry methods
- Entry feeds/speeds
- Contour wall
- Approach distance

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 feedrate 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 the methods available.

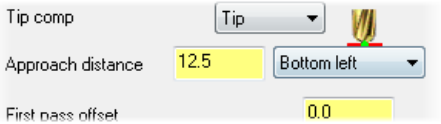


Use Entry feeds and speeds to avoid placing too much load on the tool upon entry. Set a short dwell interval after entry to allow the spindle to reach the necessary speed before beginning material removal.



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 absolute 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 2D and 3D HST family of toolpaths with many similarities, but with the stated enhancements. The high speed toolpaths themselves are a leap ahead of the standard pocket, contour, and area roughing toolpaths.

Creating and reviewing the motion of dynamic toolpaths is the best way to learn about the benefits and uses. Continue on to Lesson 1 to learn about the Dynamic Mill toolpath.

LESSON 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 clearing out remaining material on the top of the part.

Lesson Goals

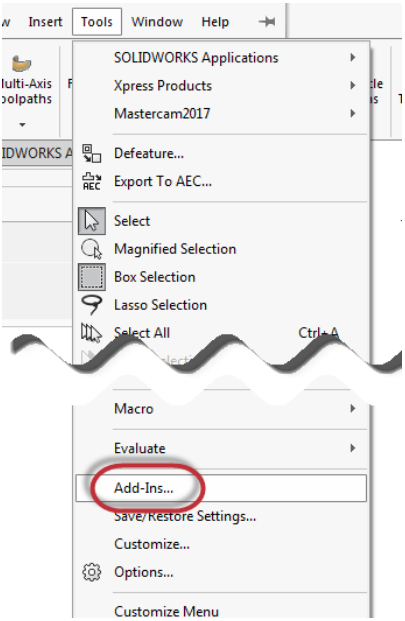
- Set up stock
- Create two Dynamic Mill toolpaths
- Preview region chains
- Verify the created toolpaths

Exercise 1: Opening Mastercam for SOLIDWORKS

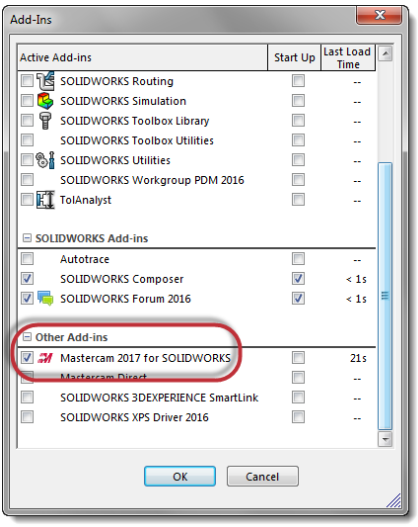
1 Start SOLIDWORKS.

2 Select **Add-Ins** from the Tools menu.

The Add-Ins dialog box displays.



3 Find **Mastercam 2017 for SolidWorks** in the Active Add-Ins list, and then do the following:



   a To load the Mastercam 2017 for SOLIDWORKS add-in for the current session, select the checkbox to the left of the Mastercam 2017 for SolidWorks entry.

   b To load the Mastercam 2017 for SOLIDWORKS add-in at every SOLIDWORKS start-up, select the checkbox to the right of the Mastercam 2017 for SolidWorks entry.

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

   After a few moments the add-in loads.

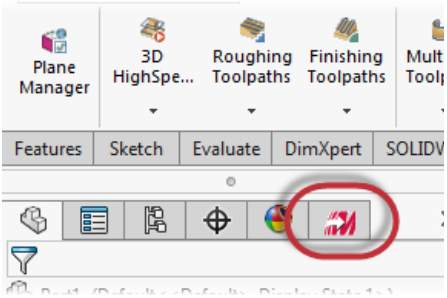
Exercise 2: Set up the stock

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

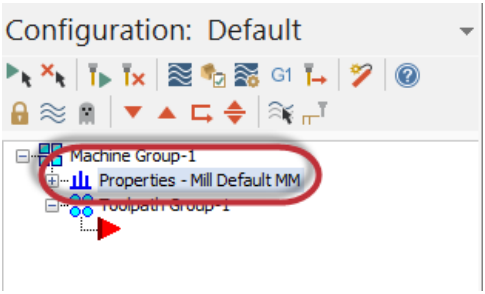
1 Open the part file, `DynamicMill`, provided with this tutorial.

2 Save the part as `DynamicMill-xxx`, replacing `xxx` with your initials.

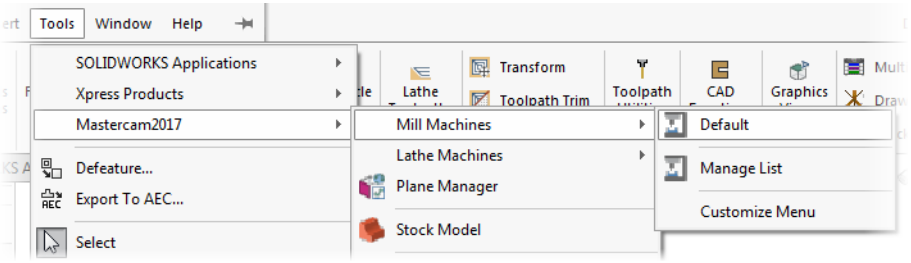
3 Select the **Mastercam Toolpaths Manager** tab to display the Toolpaths Manager.



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

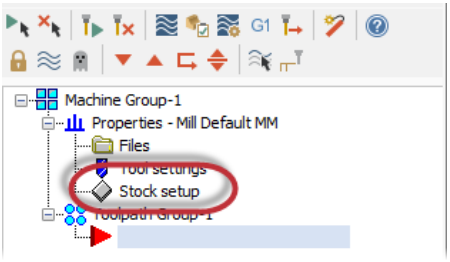


4 If you do not have the correct default machine, select **Mastercam2017**, **Mill Machines**, **Default** from the **Tools** menu.



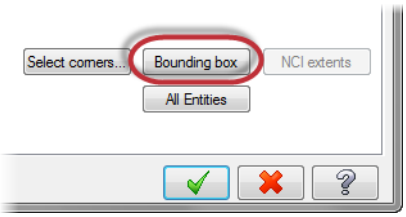
5 Select **Stock setup** in the Toolpaths Manager.

The Machine Group Properties dialog box displays.



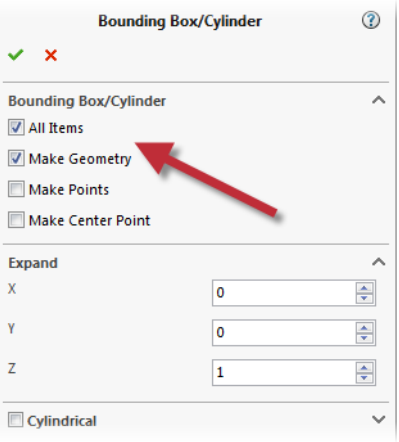
6 On the Stock Setup page, select **Bounding box**.

The Bounding Box function panel displays.



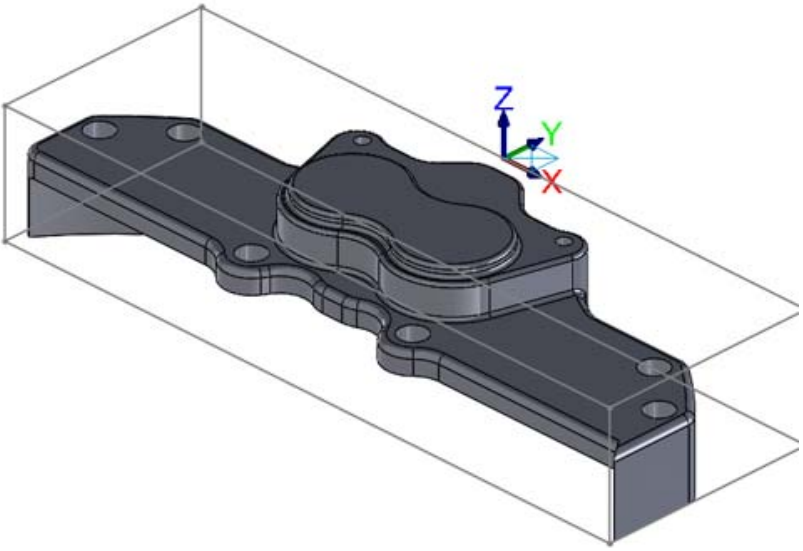
7 Set the following parameters:

- Ensure that **All Items** is selected. This will automatically create 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.

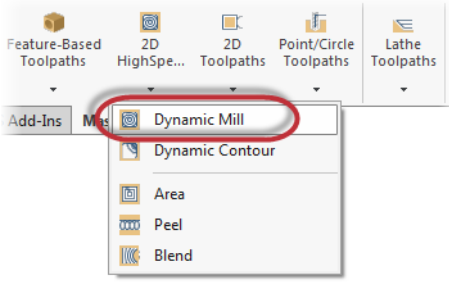
10 Save your part.

Exercise 3: Create the first Dynamic Mill toolpath

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

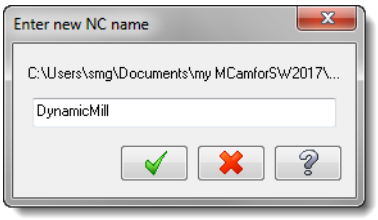
1 Select **Dynamic Mill** from the **2D HighSpeed** menu on the **Mastercam2017** tab.

The Enter new NC name dialog box displays.



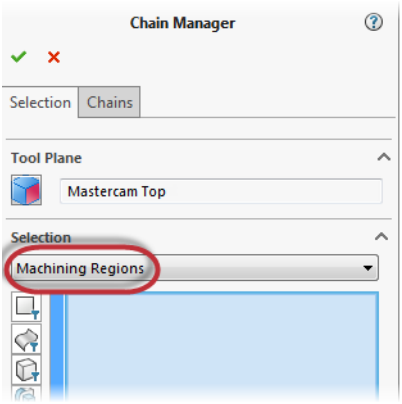
2 Click **OK**.

The Chain Manager displays.

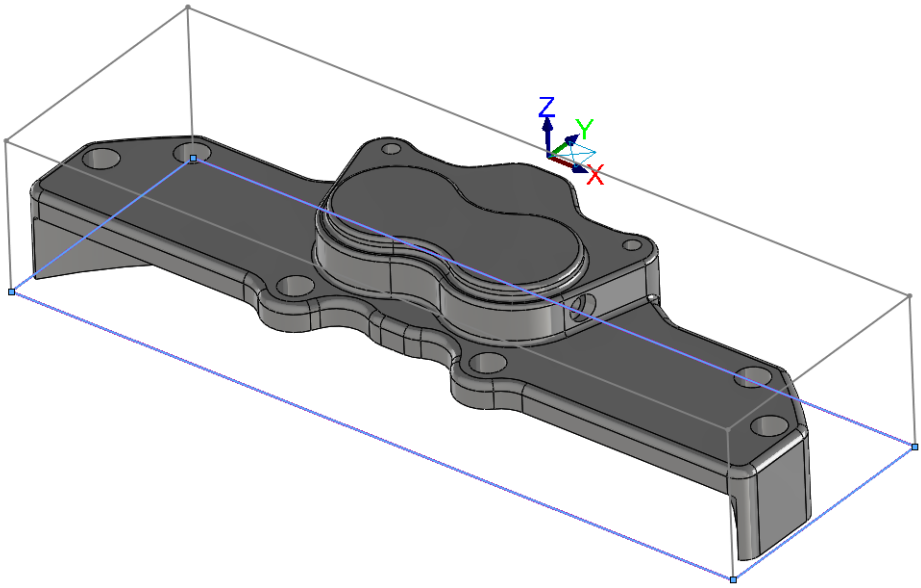


3 Ensure that you are on the Selection tab and **Selection** is set to **Machining Regions**.

Machining regions are areas to be cut.

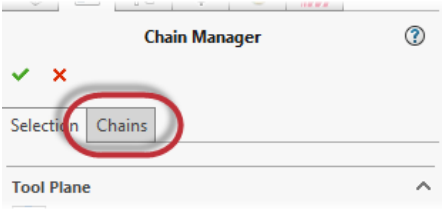


4 Select the bottom stock geometry shown below as the machining region:



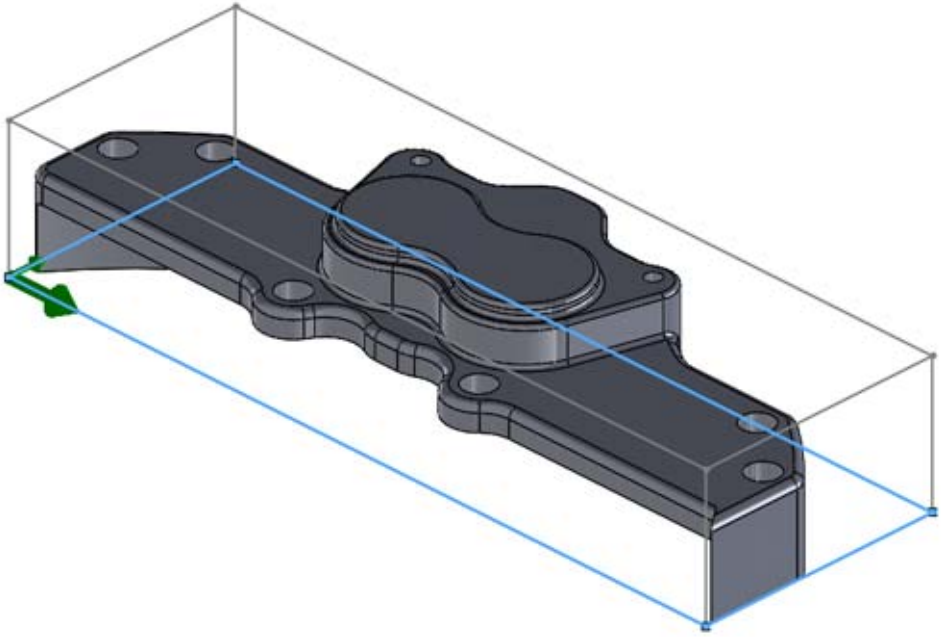
To select the bottom stock geometry, select each line separately.

5 Select the **Chains** tab in the Chain Manager.

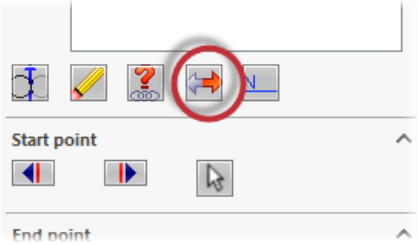


6 Select **Chain #1** in the chains list.

7 Ensure that the chain direction matches the direction shown below:



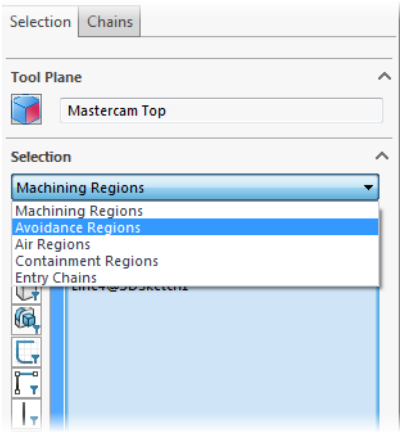
Use the **Reverse** button to reverse the chain if necessary.



8 Select the **Selection** tab.

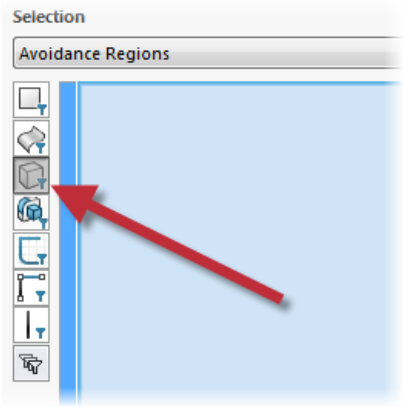
9 Select **Avoidance Regions** from the Selection drop-down.

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

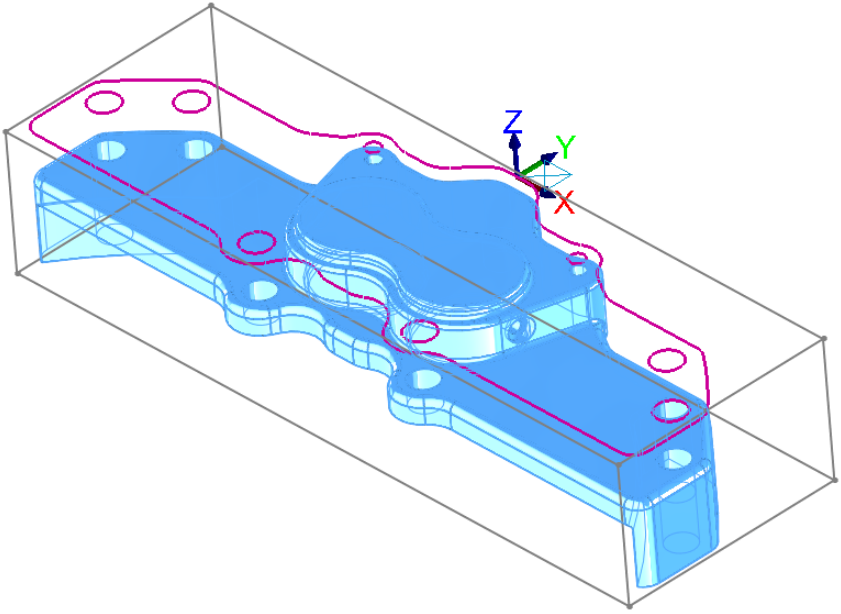


10 Select the **Bodies** filter.

This filters your selections to only bodies.

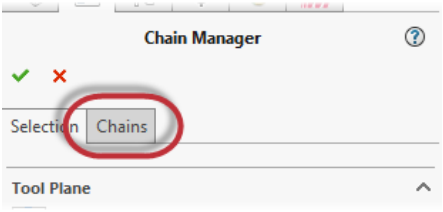


11 Select the entire part body as shown below:



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.

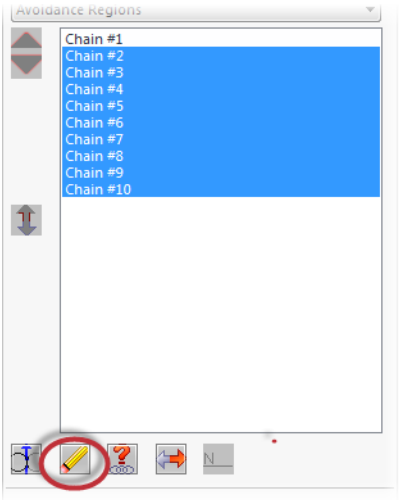
12 Select the **Chains** tab.



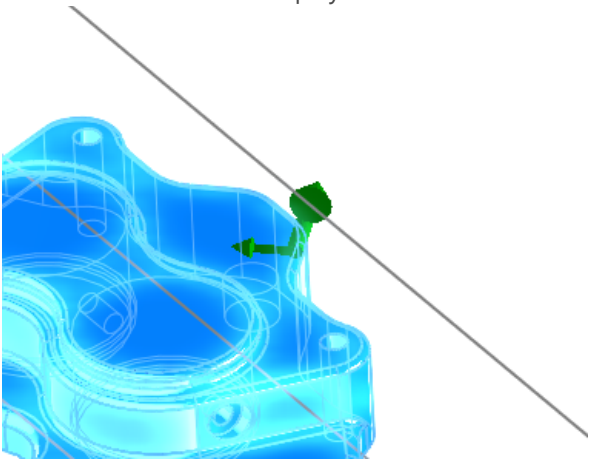
13 In the chains list, delete Chain #2 - Chain #10 using the **Delete** button.

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:

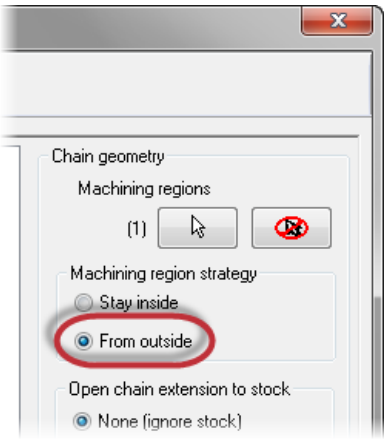


15 Click **OK** in the Chain Manager to accept the chains and begin creating the toolpath.

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

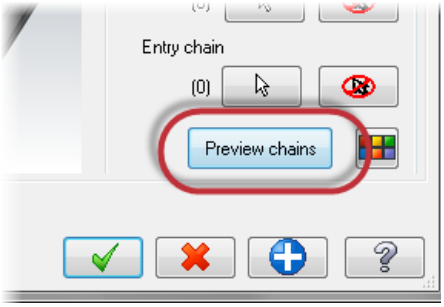
16 On the Toolpath Type page, set the Machining region strategy to **From outside**.

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.

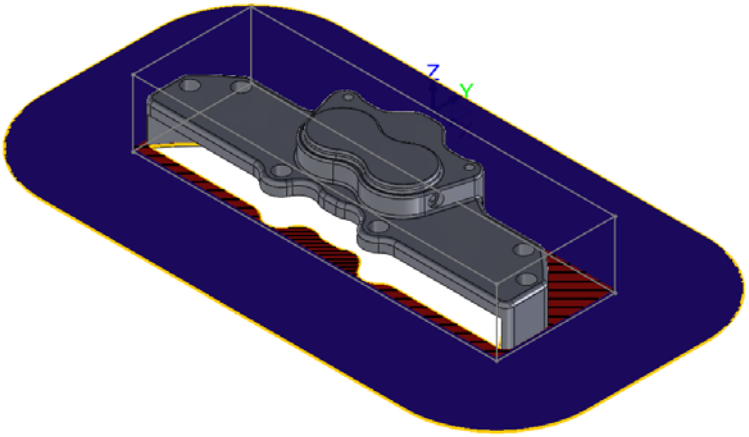


Before continuing with the toolpath creation, you will preview your toolpath chains to ensure that you have selected the geometry that you want.

17 Click **Preview chains**.



Your part then displays as shown below:

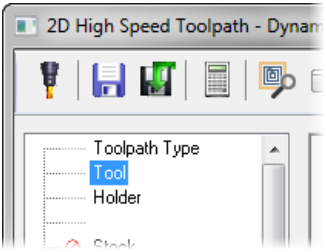


Mastercam displays your material in the red and black crosshatch, the motion region as blue, and the tool containment as yellow. By using Preview chains, you can determine if that is what you want from the toolpath before even entering any parameters.

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

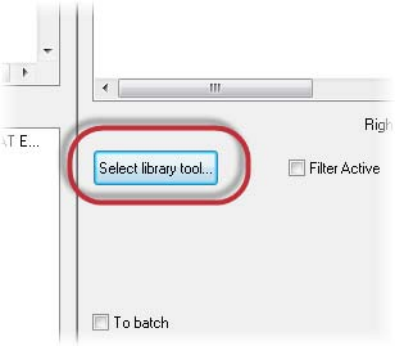
18 Click **Preview chains** again to turn off the display.

19 Select the **Tool** page.



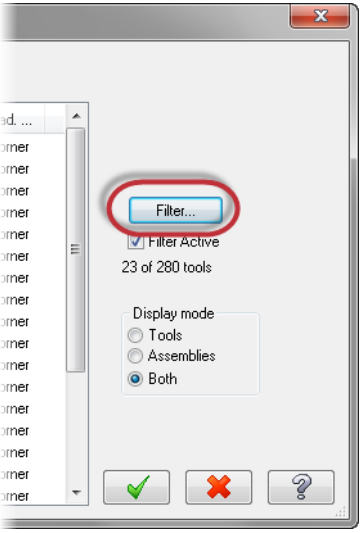
20 Click the **Select library tool** button.

The Tool Selection dialog box displays.



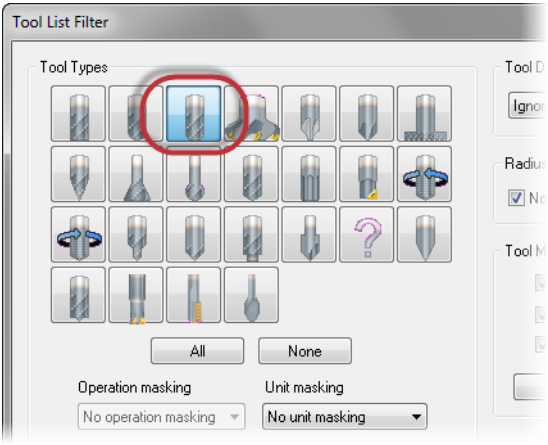
21 Select the **Filter** button.

The Tool List Filter dialog box displays.



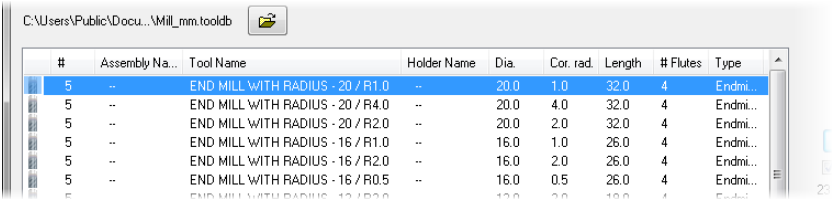
22 Select **None** to remove all filters.

23 Select the **Endmill3 Bull** filter. This ensures that you are only selecting Bull Endmill tools.



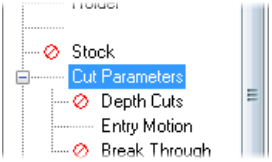
24 Click **OK** in the Tool List Filter dialog box.

25 In the Tool Selection dialog box, select the **END MILL WITH RADIUS - 20/ R1.0** tool.

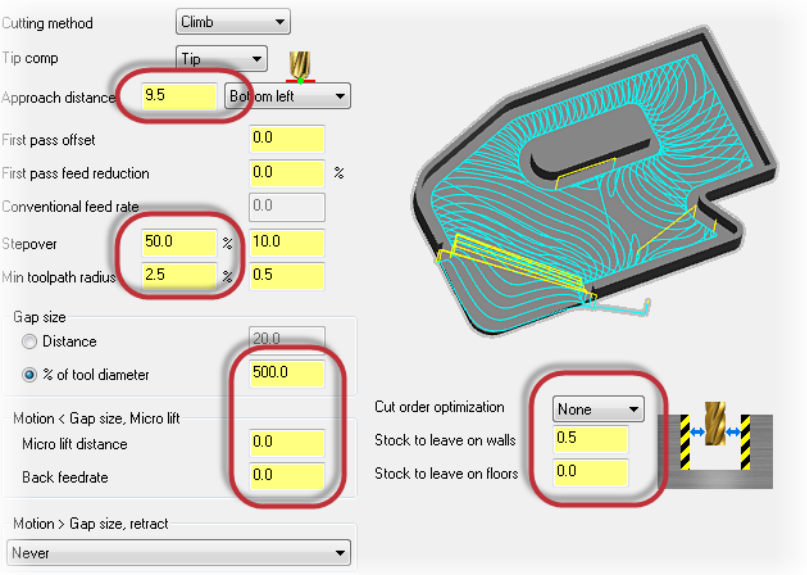


26 Click **OK** in the Tool Selection dialog box to add the tool to the toolpath.

27 Select the **Cut Parameters** page.



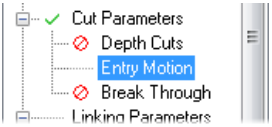
28 Set the following parameters:



- **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.
- **Min toolpath radius** to **2.5**%. Mastercam uses this radius in combination with the Micro lift distance and Back feedrate parameters to calculate 3D arc moves between cutting passes.
- **% of tool diameter** to **500.0**. Mastercam will not create a retract move if the distance from the end of one pass to the start of the next pass is less than this distance. The tool to stay down and move directly between the passes at the feed rate.
- **Micro lift distance** to **0.0**. The distance the tool lifts off the part on back moves.
- **Back feedrate** to **0.0**. Controls the speed of the backfeed movement of the tool.
- **Cut order optimization** to **None**. 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.

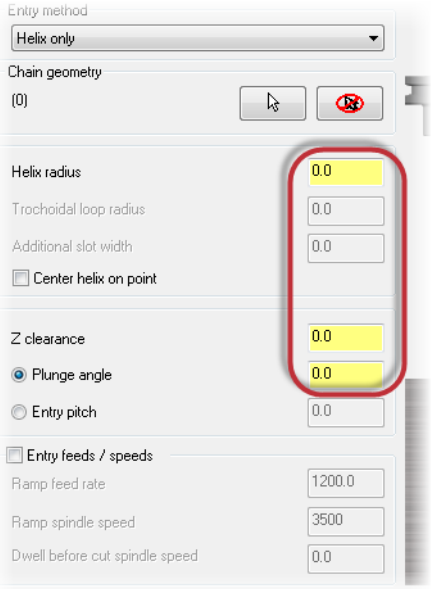
29 Select the **Entry Motion** page.



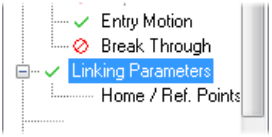
30 Set the following parameters:

- **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.

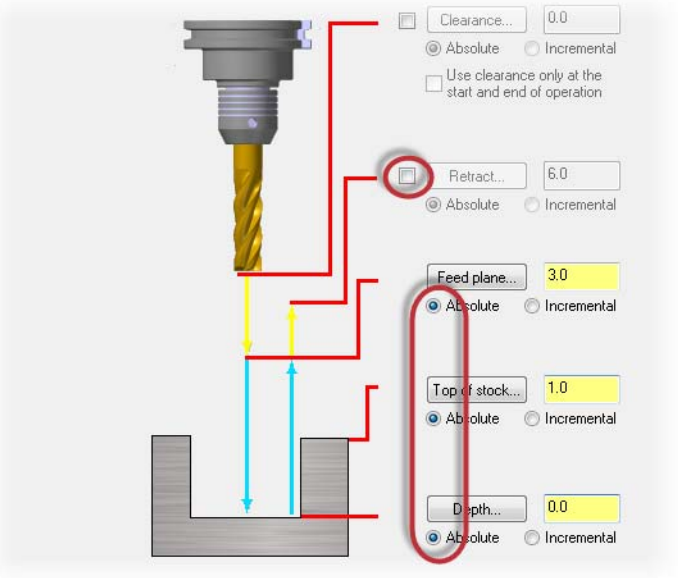
This toolpath does not enter down into the stock, so you do not need to set any entry parameters.



31 Select the **Linking Parameters** page.

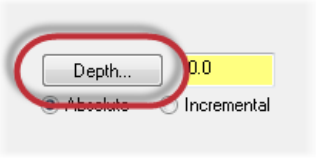


32 Set the following parameters:



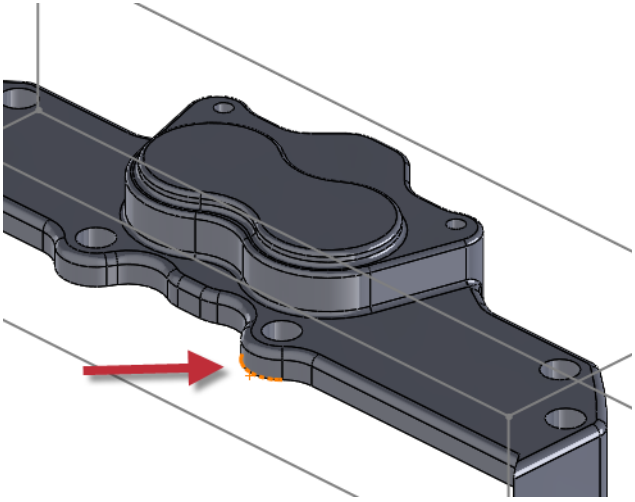
- Deselect **Retract**. Sets the height that the tool moves up to before the next tool pass.
- Change **Top of stock** and **Depth** to **Absolute**. Mastercam measures absolute values from the origin 0,0,0.

33 Select the **Depth** button 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.

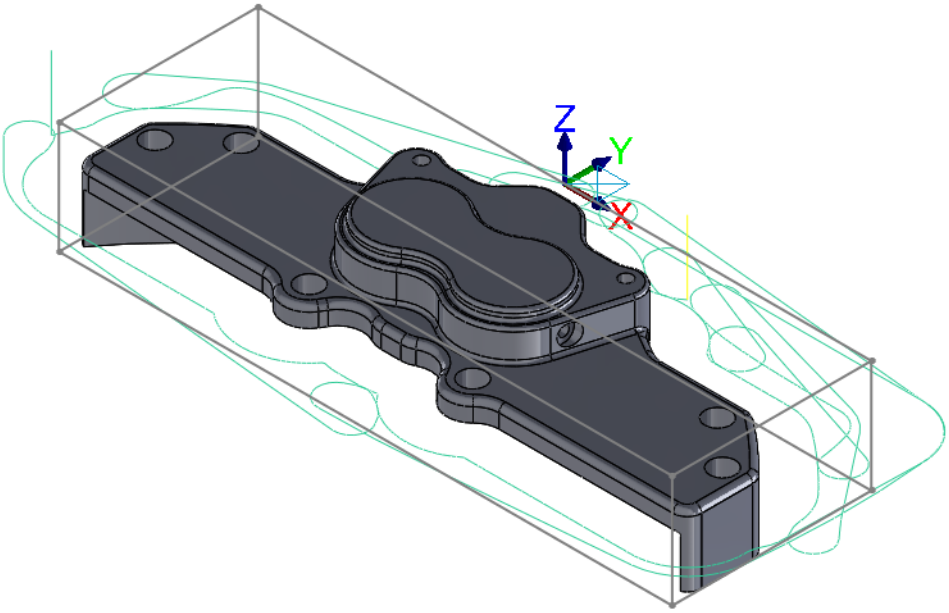
34 Select the edge shown below:



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:



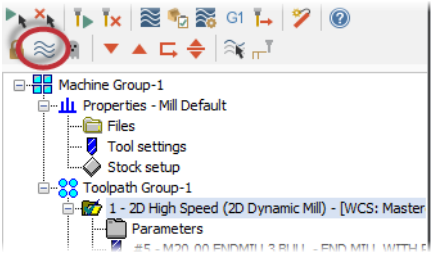
37 Save your part.

Exercise 4: Create the second Dynamic Mill toolpath

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

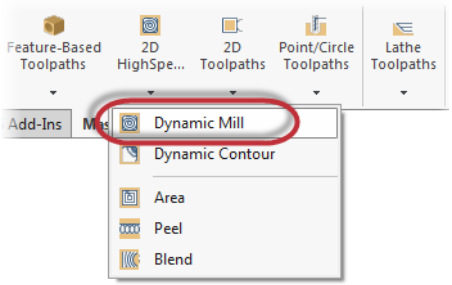
1 In the Toolpaths Manager with the Dynamic Mill toolpath selected, select **Toggle display on selected operations**.

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

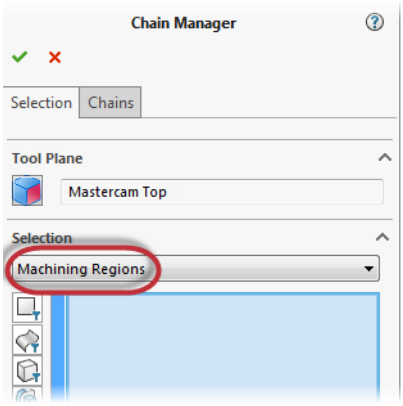


2 Select **Dynamic Mill** from the **2D HighSpeed** menu on the **Mastercam2017** tab.

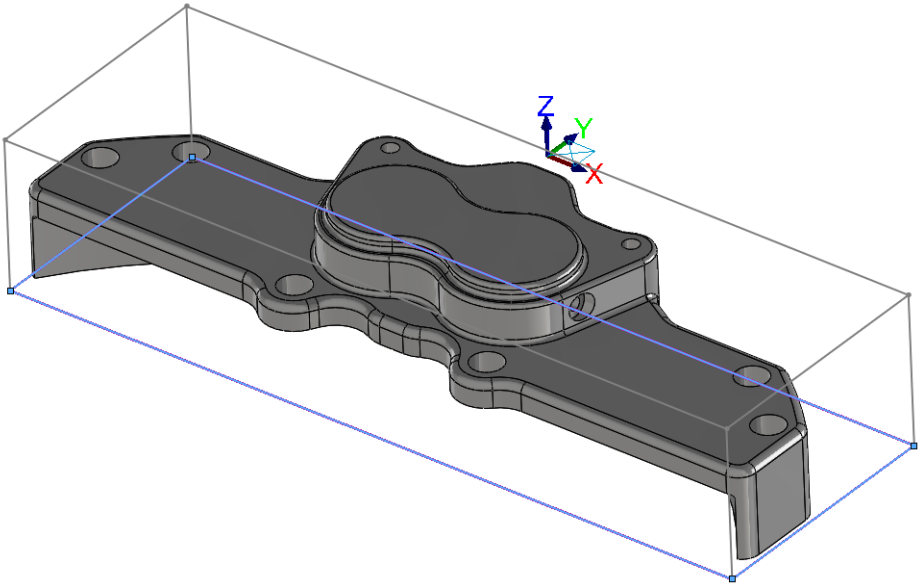
The Chain Manager displays.



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



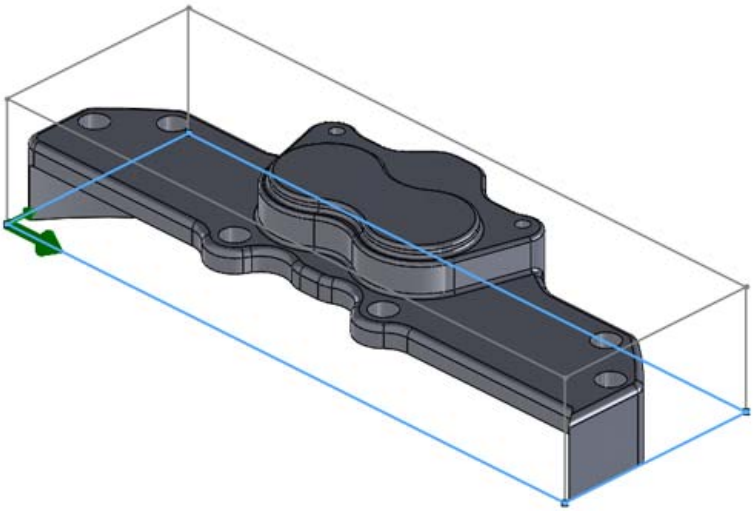
4 Select the geometry shown below as the machining region.



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

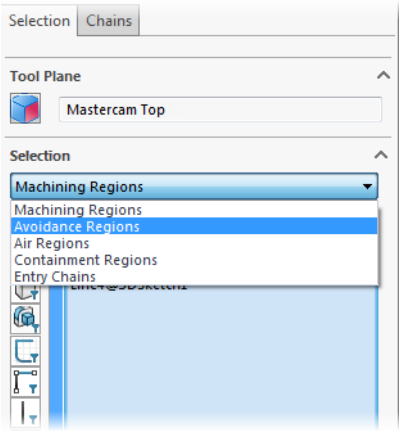
5 Select the **Chains** tab and select **Chain #1**.

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

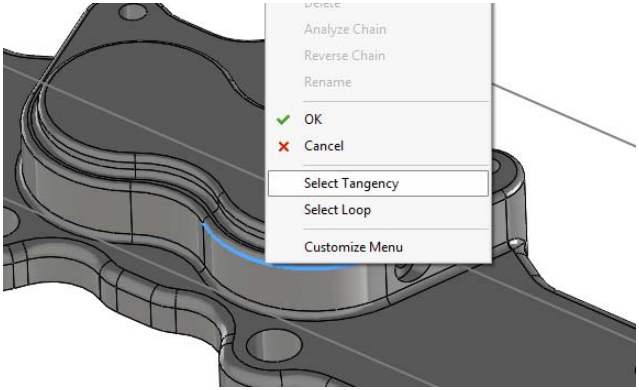


7 Select the **Selections** tab.

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

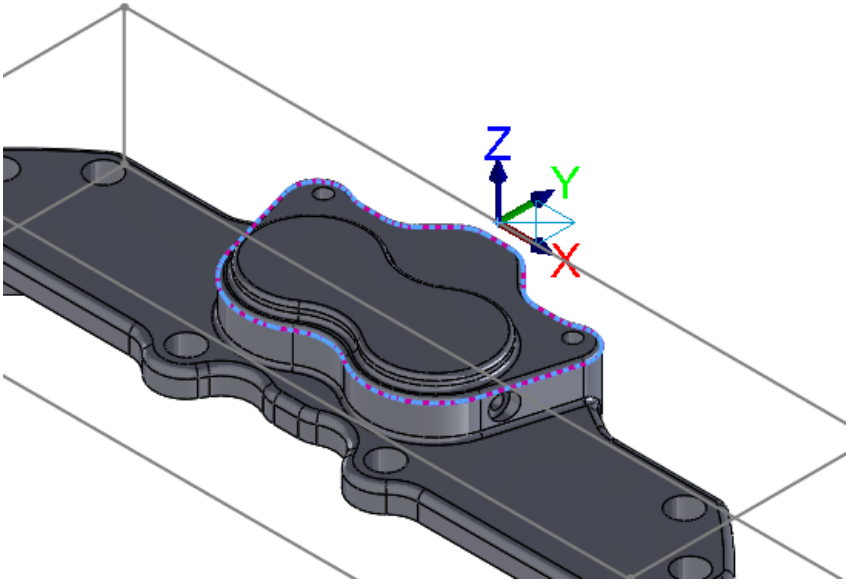


9 Right-click on the edge shown below and choose **Select Tangency**:



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

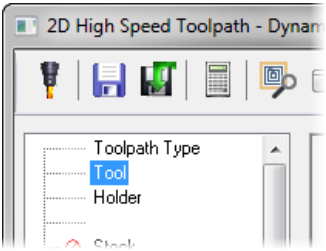
The selected edges display as shown below:



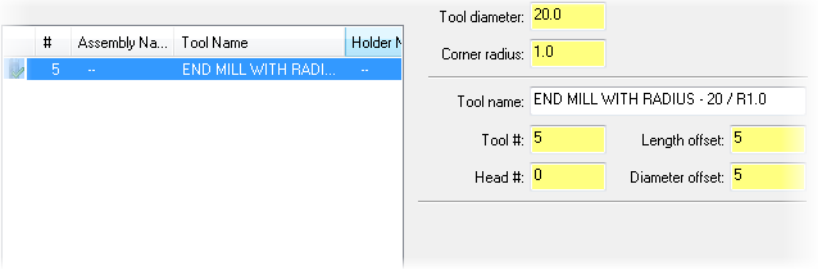
10 Click **OK** in the Chain Manager to accept these chains.

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

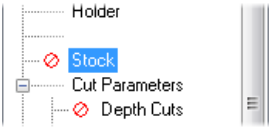
11 Select the **Tool** page.



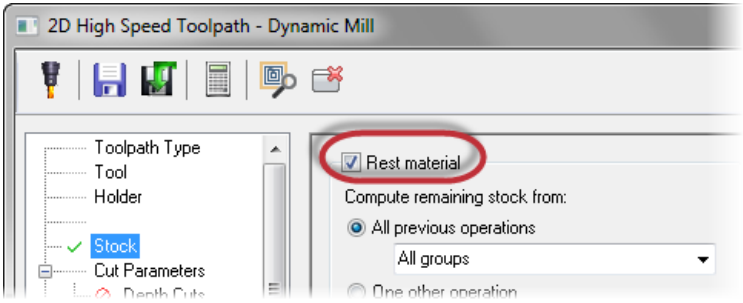
12 In the Tool List, select the tool that was used for the previous Dynamic Mill toolpath, which is **END MILL WITH RADIUS - 20/R1.0.**



13 Select the **Stock** page.

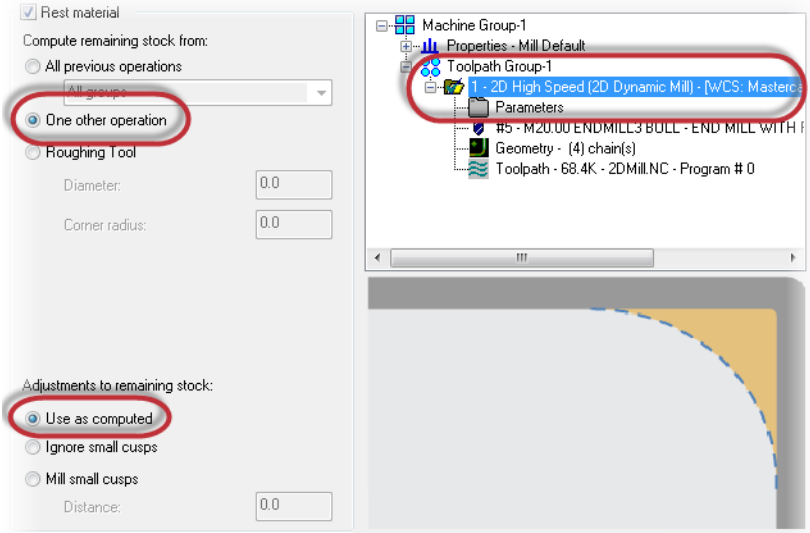


14 Select the **Rest material** checkbox.



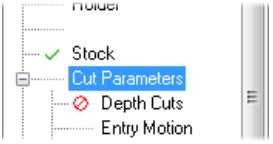
This enables the page and tells Mastercam to calculate the cutting passes based on the remaining stock.

15 Set the following parameters:

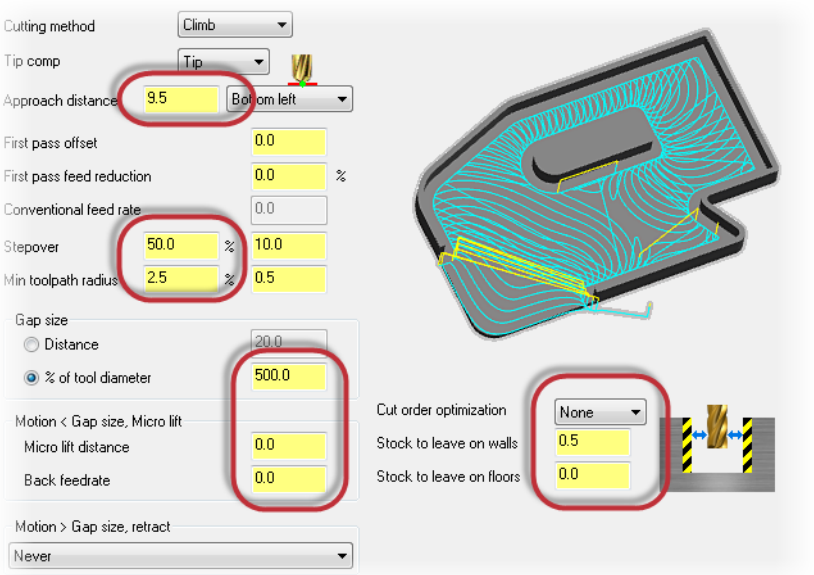


- Set **Compute remaining stock from** to **One other operation**. This calculates the remaining stock from a single operation.
- In the **Operation List**, select the **2D High Speed (2D Dynamic Mill)** toolpath.
- Set **Adjustments to remaining stock** to **Use as computed**. This option makes no adjustments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

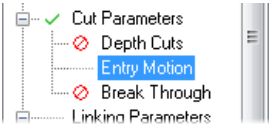
16 Select the **Cut Parameters** page.



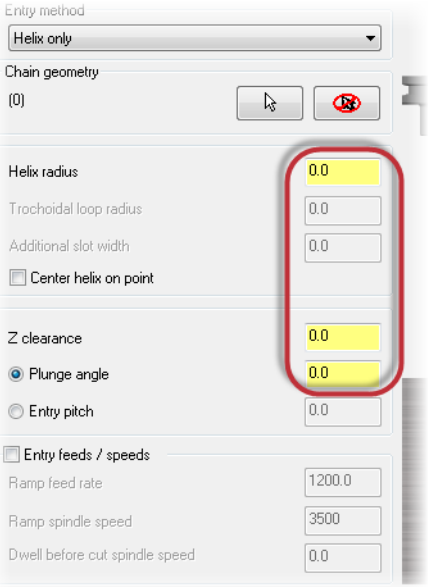
17 The parameters are the same as the previous operation's, so ensure that your toolpath matches the parameters shown below:



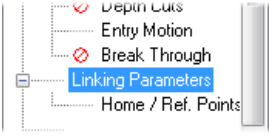
18 Select the **Entry Motion** page.



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

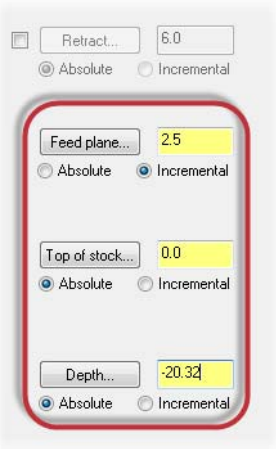


20 Select the **Linking Parameters** page.

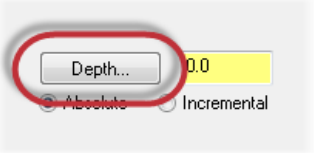


21 Set the following parameters:

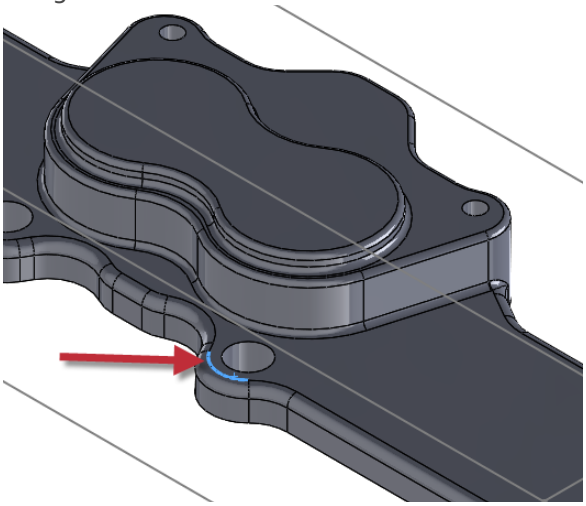
- **Feed plane** to **2.5**. Sets the height that the tool rapids to before changing to the plunge rate to enter the part.
- Ensure that **Feed plane** is set to **Incremental**.
- **Top of Stock** to **0.0**. Sets the top of the stock.
- Ensure that **Top of stock** and **Depth** are set to **Absolute**.



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



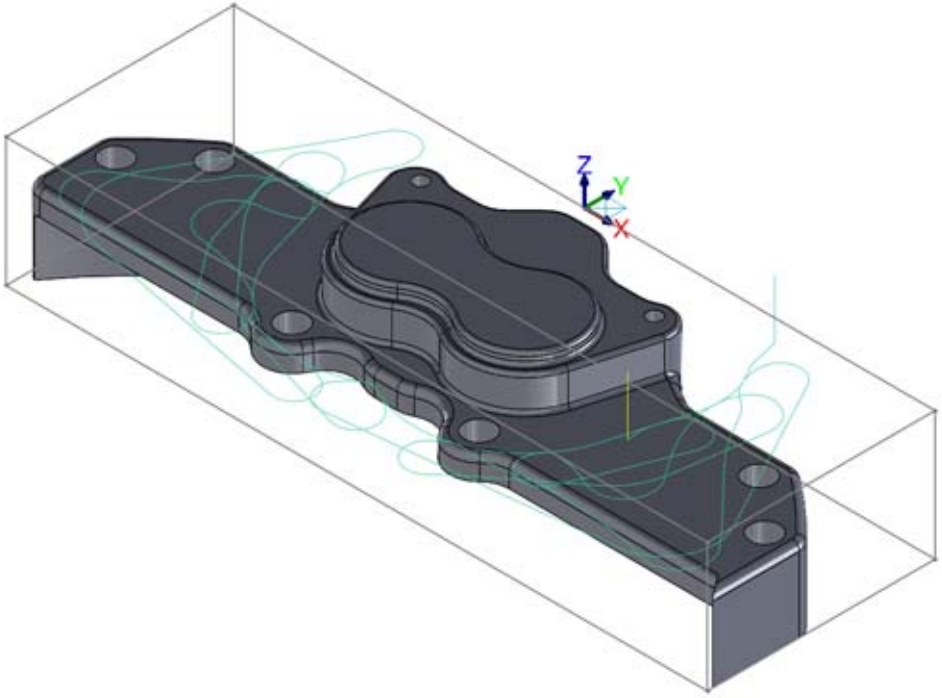
23 Select the edge shown below:



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.

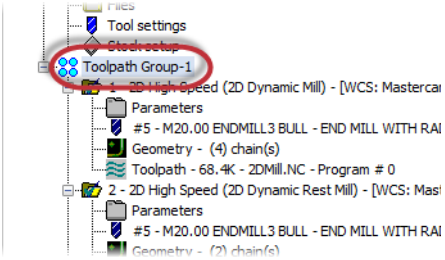
27 Save your part.

Exercise 5: Verify the toolpaths

You will now verify both Dynamic Mill toolpaths with Mastercam Simulator.

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

This will select both Dynamic Mill toolpaths.

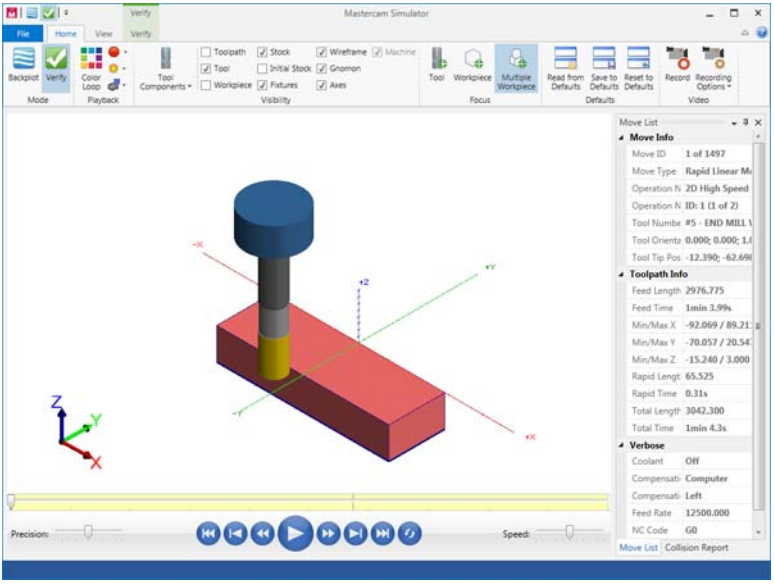


2 Select **Verify selected operations**.

Mastercam Simulator displays.

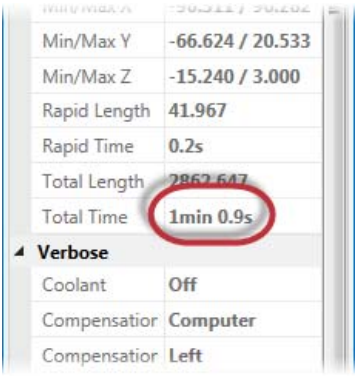
Mastercam Simulator allows you to verify your toolpaths by using solid models to simulate part machining against a selected stock definition. The result Verify creates represents the surface finish, and shows collisions, if any exist. Use it identify and correct program errors before they reach the shop floor.

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



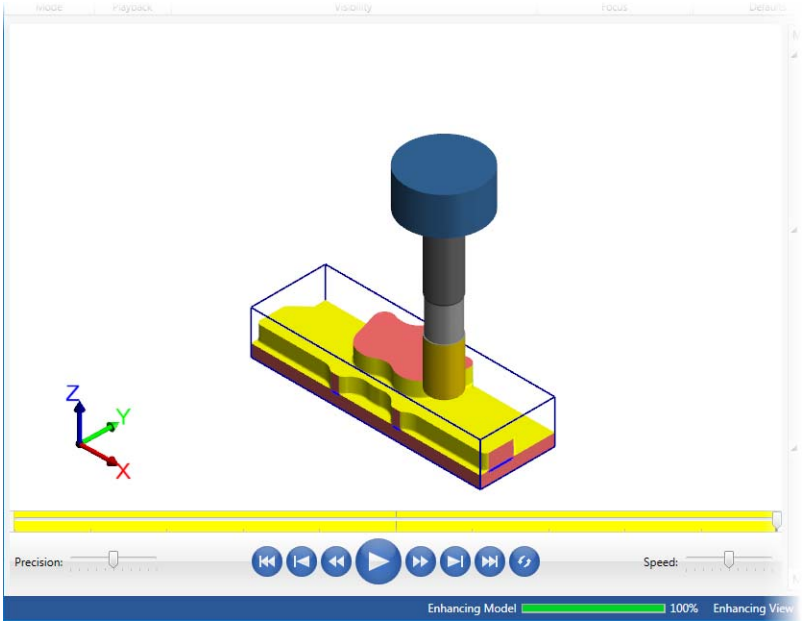
4 View the Move List, which displays all pertinent information about the toolpaths being verified.

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.

LESSON 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: Create 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.

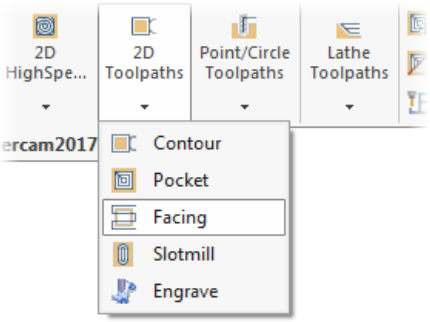
   

   This will hide the two Dynamic Mill toolpaths and keep the graphics view clean.

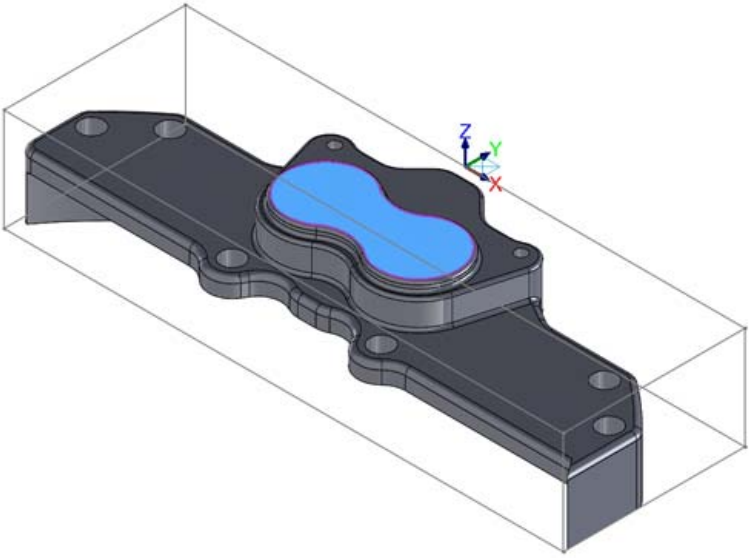
3. Save the part as `FaceMill-xxx`, replacing `xxx` with your initials.

4 Select **2D Toolpaths**, **Facing** from the **Mastercam2017** tab.

The Chain Manager displays.



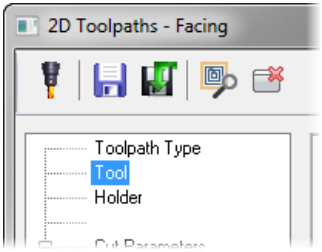
5 Select the solid face shown below:



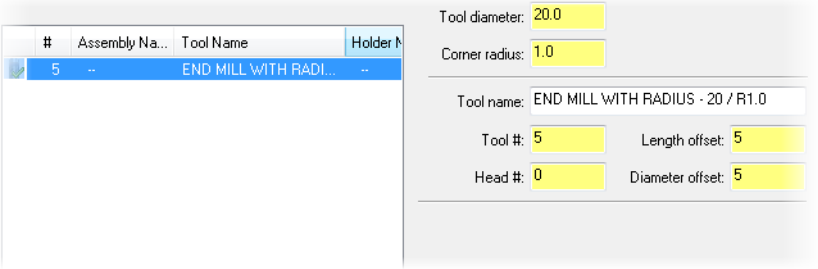
6 Click **OK** in the Chain Manager.

The 2D Toolpaths - Facing dialog box displays.

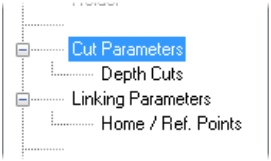
7 Select the **Tool** page.



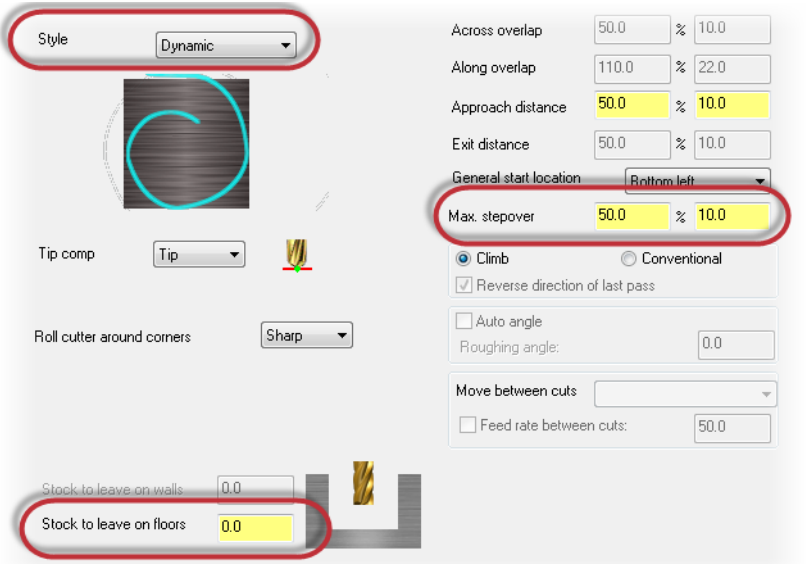
8 Select the **END MILL WITH RADIUS - 20 / R 1.0**, which was used in the previous two Dynamic Mill toolpaths.



9 Select the **Cut Parameters** page.

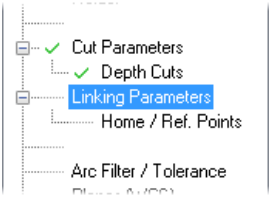


10 Set the following parameters:

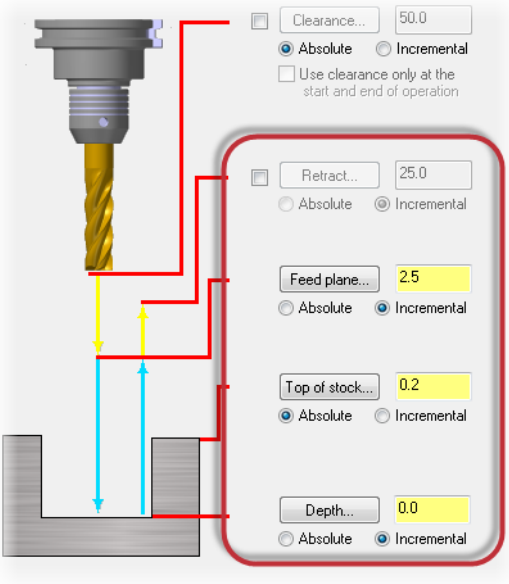


- **Style** to **Dynamic**. Creates a smooth controlled motion that cuts from the outside to the inside, maintaining a constant load on the tool with minimal entries and exits.
- **Stock to leave on floors** to **0.0**. Sets the amount of stock to leave on the floors during machining.
- **Max. stepover**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

11 Select the **Linking Parameters** page.



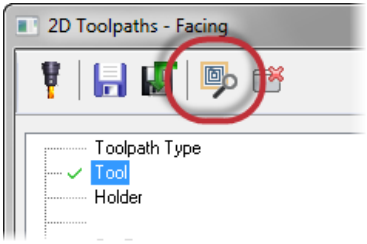
12 Set the following parameters:



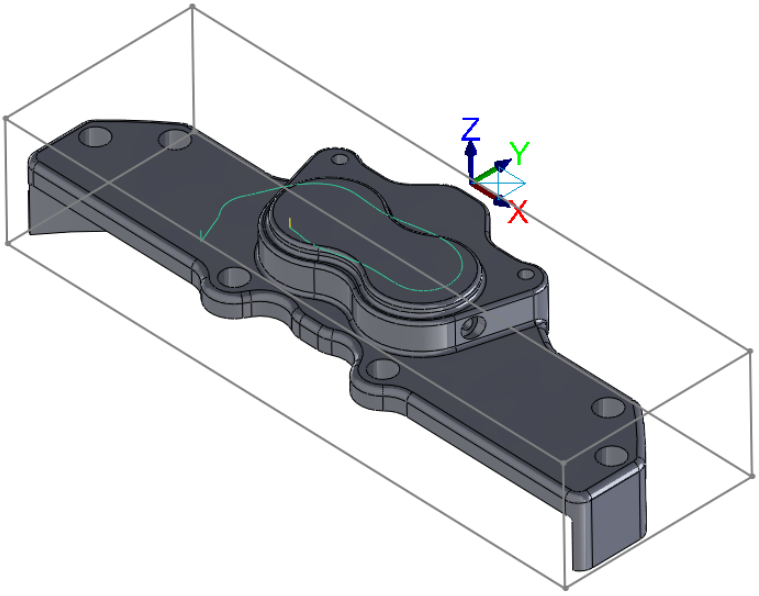
- Deselect **Retract**. Sets the height that the tool moves up to before the next tool pass.
- Set **Feed plane** to **2.5**. Sets the height that the tool rapids before changing to the plunge rate to enter the part.
- Set **Top of stock** to **0.2**. Sets the height of the material in the Z axis.

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.



The toolpath will display on the part as shown below:



14 Click **OK** in the 2D Toolpaths - Facing dialog box to close and generate the toolpath.

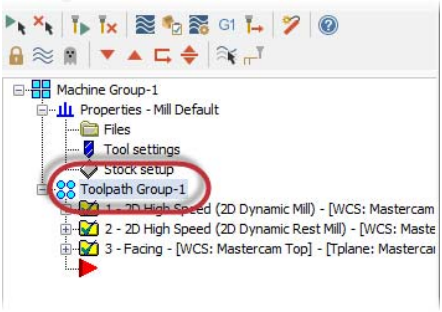
15 Save your part.

Exercise 2: Verify the toolpaths

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

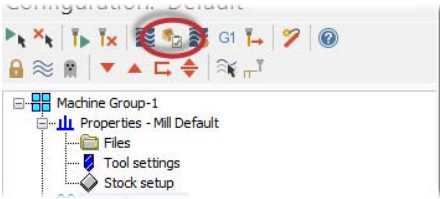
1 Select **Toolpath Group-1** in the Toolpaths Manager.

This selects both Dynamic Mill toolpaths and the Facing toolpath.

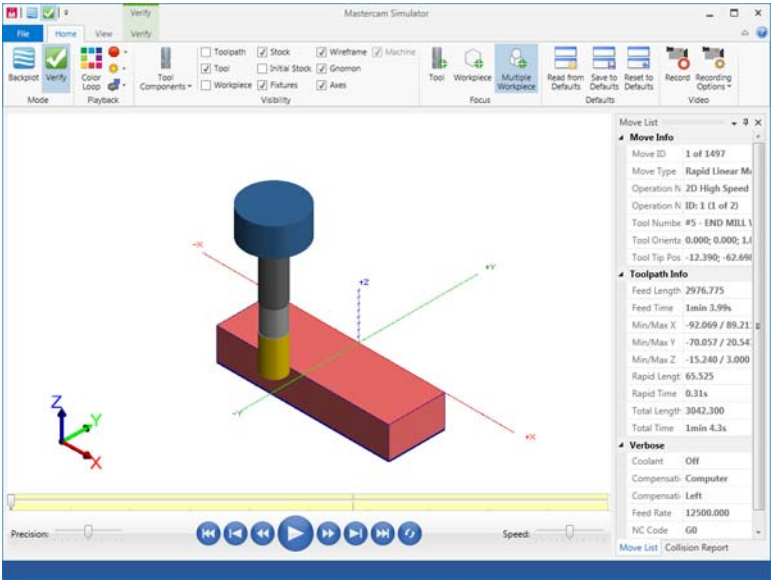


2 Select **Verify selected operations**.

Mastercam Simulator displays.



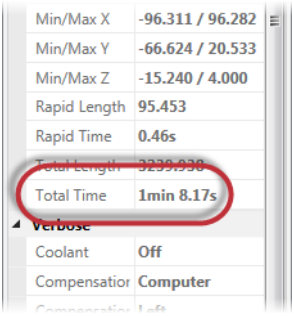
3 Right-click in the graphics view and select **Fit** and **Isometric** from the menu. Mastercam Simulator should match the image shown below.



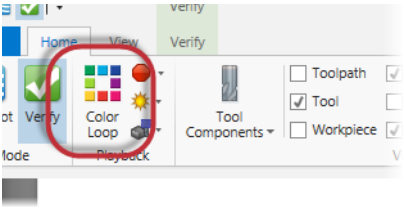
4 Press **Play** to preview the toolpath motion for the toolpaths.

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

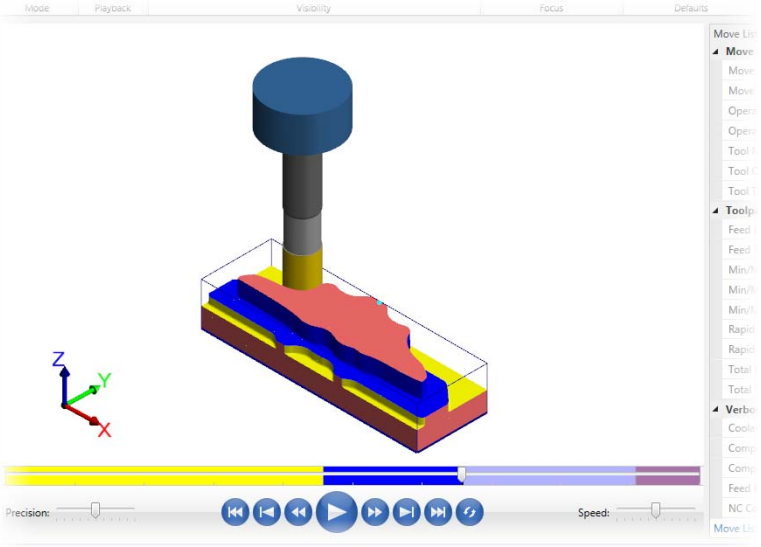
5 View the Move List, which displays all pertinent information about the toolpaths being verified. The three Dynamic Mill toolpaths are able to machine the part in the time shown with the given parameters.



6 Use Verify's **Color Loop** option 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.

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.

LESSON 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, Dynamic Contour is much more efficient for cutting by using the entire flute of the tool, as with all dynamic motion toolpaths.

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

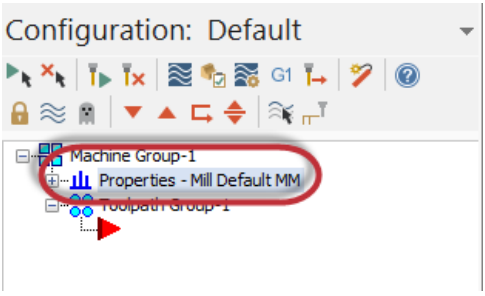
Lesson Goals

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

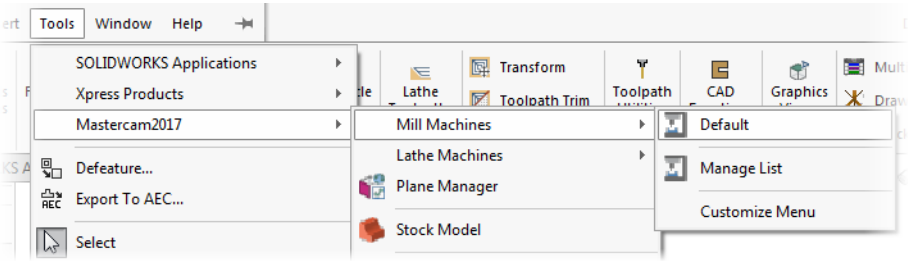
Exercise 1: Set up the stock

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.

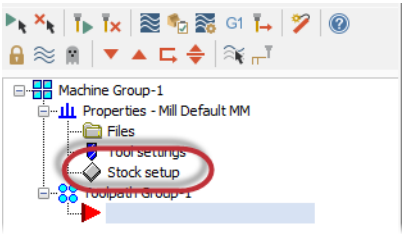


3 If you do not have the correct default machine, select **Mastercam2017**, **Mill Machines**, **Default** from the **Tools** menu.



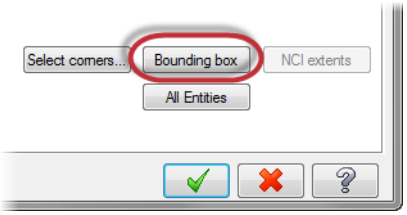
4 Select **Stock setup** from the Toolpaths Manager.

The Machine Group Properties dialog box displays.



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

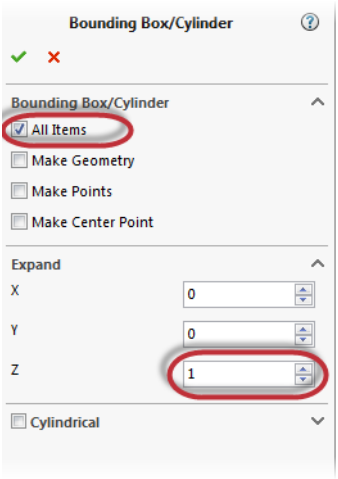
The Bounding box function panel displays.



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.



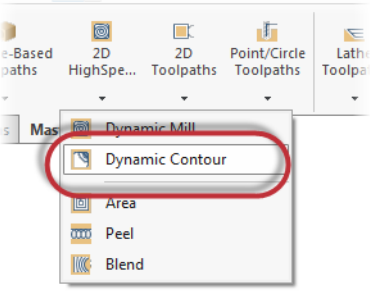
7 Click **OK** to create the boundary.

8 Click **OK** to accept the stock setup.

Exercise 2: Create the first Dynamic Contour toolpath

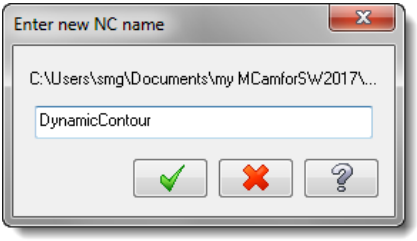
1 Select **2D HighSpeed**, **Dynamic Contour** on the **Mastercam2017** tab.

The Enter new NC name dialog box displays.

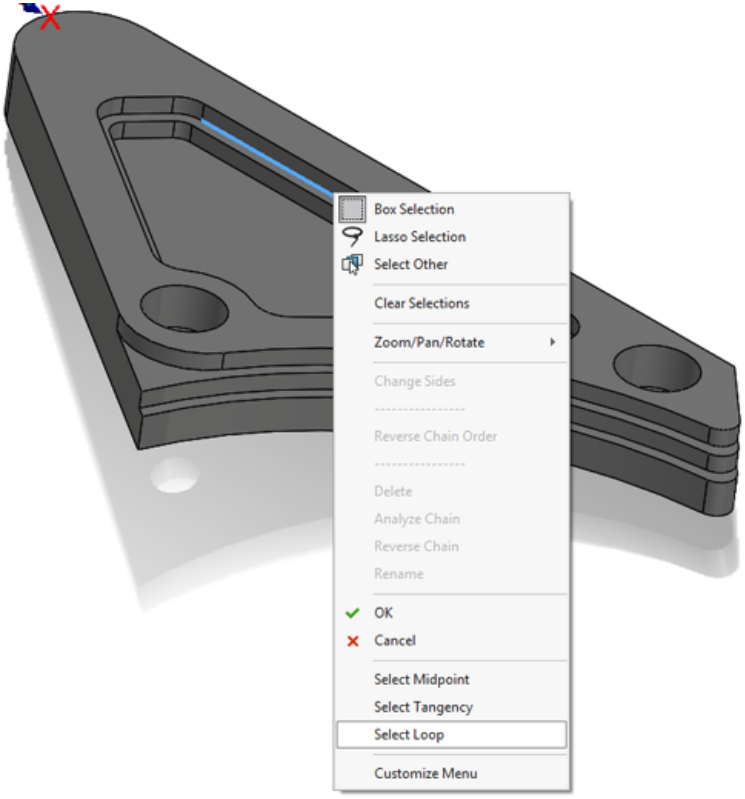


2 Click **OK** in the Enter new NC Name dialog box. You would use this dialog box to change the name of the resulting NC file from the default name.

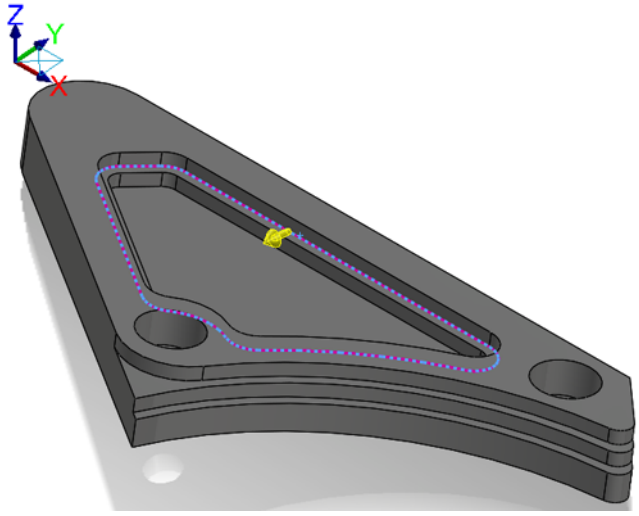
The Chain Manager displays.



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



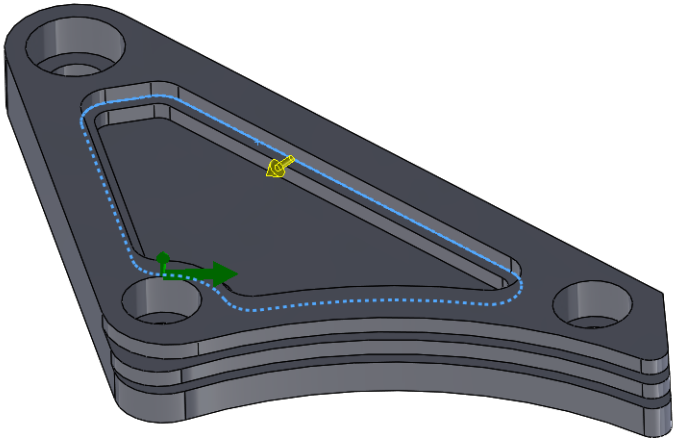
4 The loop should be selected as shown below:



Click the arrow if necessary to select the correct loop.

5 Click the **Chains** tab.

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

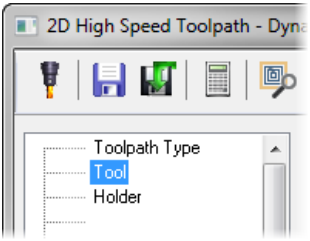


If necessary, use the **Reverse** button to reverse the chain.

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

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

8 Select the **Tool** page.

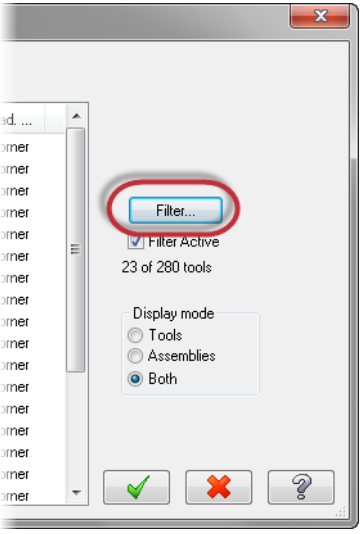


9 Click the **Select library tool** button.

The Tool Selection dialog box displays.

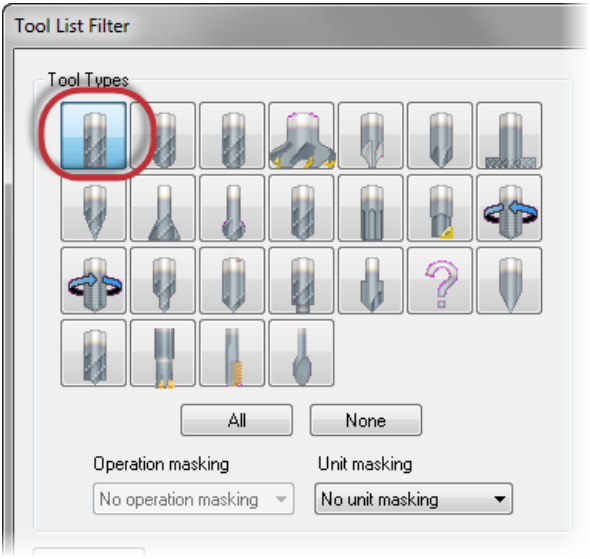
10 Click the **Filter** button.

The Tool List Filter dialog box displays.



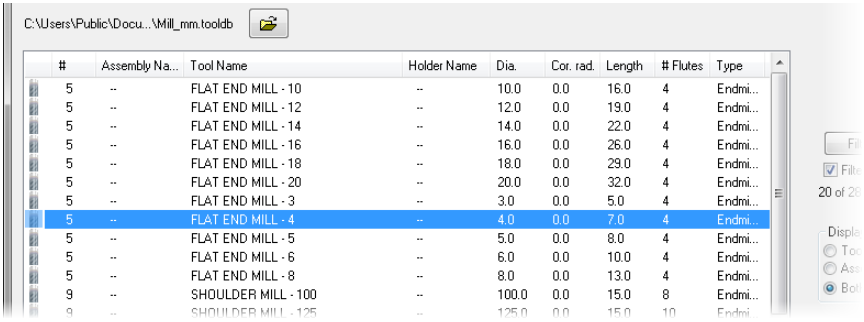
11 Set the Tool Type filter to **None**.

12 Select the **Endmill1 Flat** filter. This ensures that you are only selecting Flat Endmill tools.



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

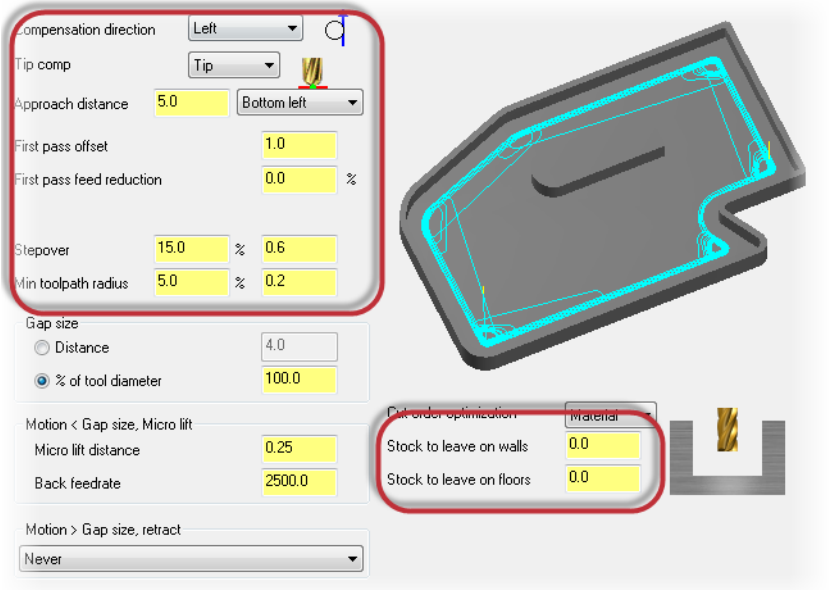
14 Select the **FLAT END MILL - 4** tool.



15 Click **OK** in the Tool Selection dialog box.

16 Select the **Cut Parameters** page.

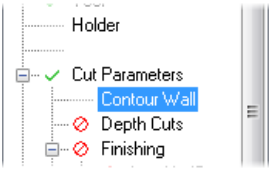
17 Set the following parameters:



- **Compensation direction** to **Left**. Offsets the tool to the right or left of the toolpath.
- **Approach distance** to **5.0**. Adds a specified absolute 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 entering stock from the outside.
- **Stepover** to **15.0**%. Sets the distance between cutting passes in the X and Y axes.
- **Min toolpath radius** to **5.0**%. Sets the minimum toolpath radius for the operation.

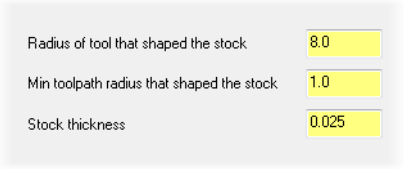
- Set **Stock to leave on walls** and **Stock to leave on floors** to **0.0**. You are not leaving any stock on the vertical and horizontal surfaces.

18 Select the **Contour Wall** page.



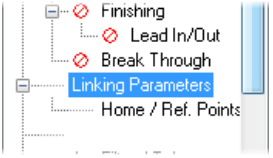
19 Set the following parameters:

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

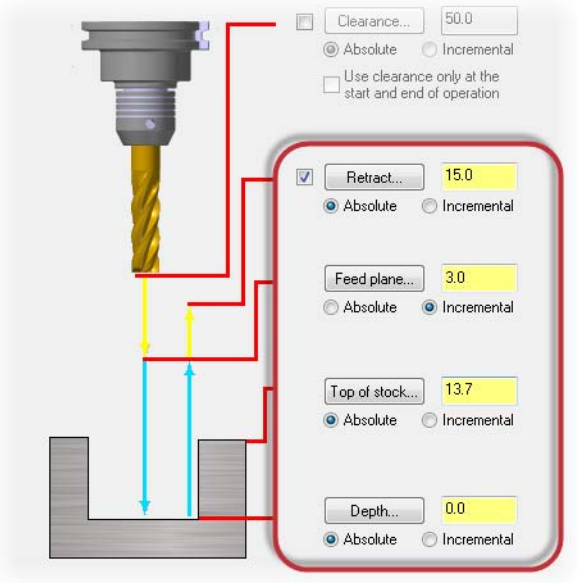


Both parameters help define the shape of the stock removed by the toolpath. Mastercam calculates the stock to remove along the contour wall by using these parameters.

20 Select the **Linking Parameters** page.

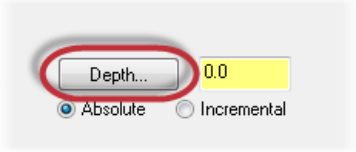


21 Set the following parameters:



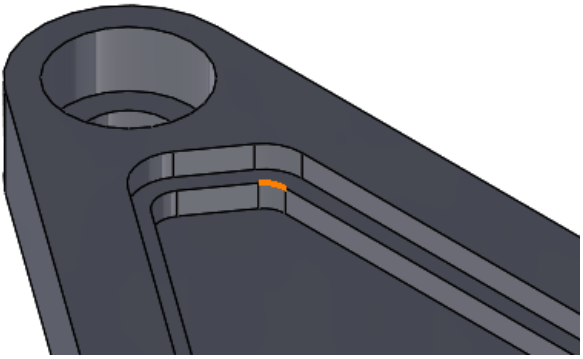
- **Retract**, **Top of stock**, and **Depth** to **Absolute**. This measures absolute values from the origin 0,0,0.
- **Retract** to **15.0**. Sets the height that the tool moves up before the next tool pass.
- **Feed plane** to **3.0**. Sets the height that the tool rapids to before changing to the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

22 Click the **Depth** button. 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.

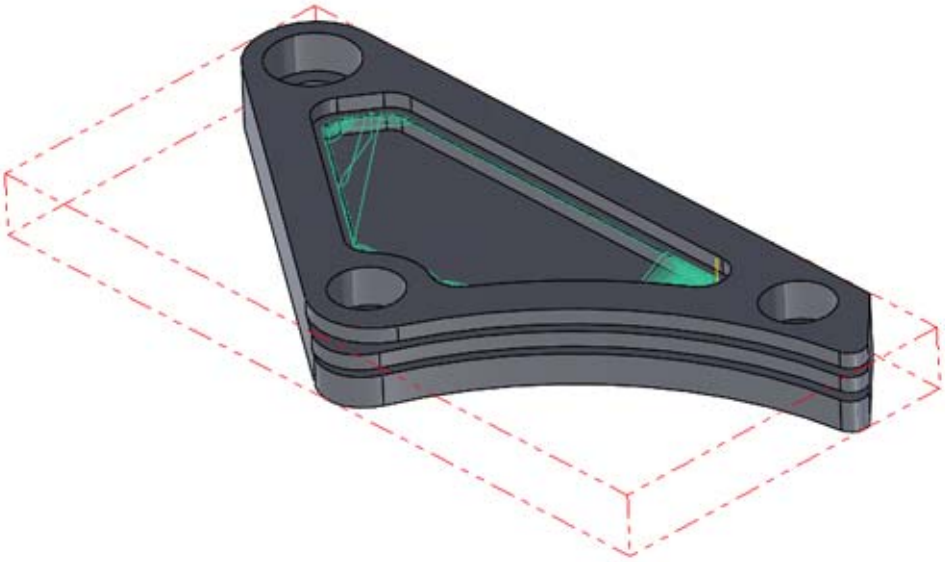
23 Select the edge shown below:



Click **OK** to accept the edge and return to the Linking Parameters page. Your **Depth** should now be **9.525**.

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

25 Your toolpath display as shown below:

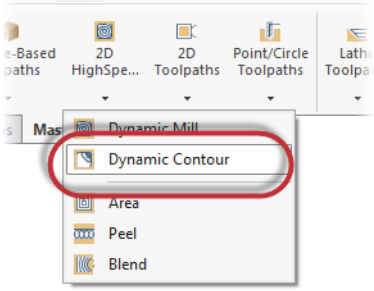


26 Save your part.

Exercise 3: Create the second Dynamic Contour toolpath

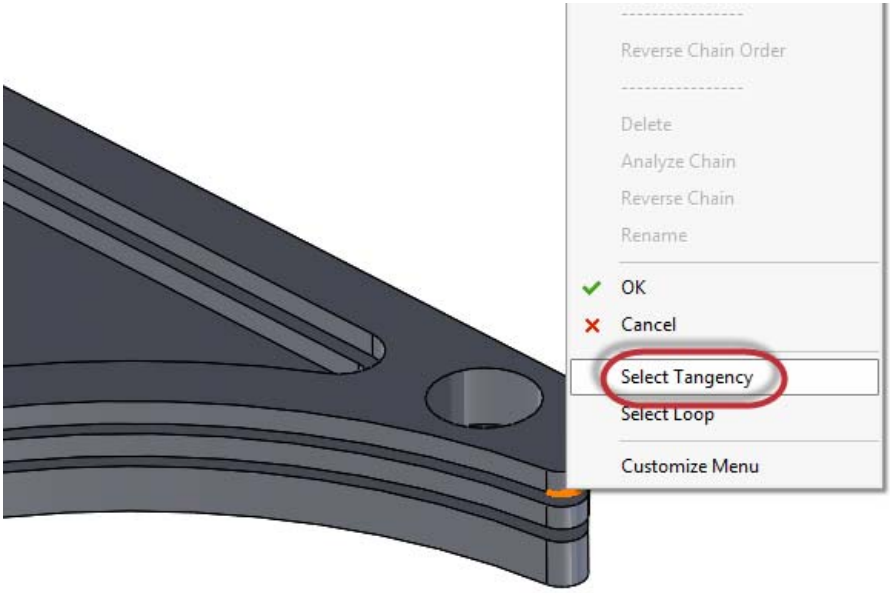
1 Select **Dynamic Contour** from the **2D HighSpeed** menu on the **Mastercam2017** tab.

The Chain Manager displays.



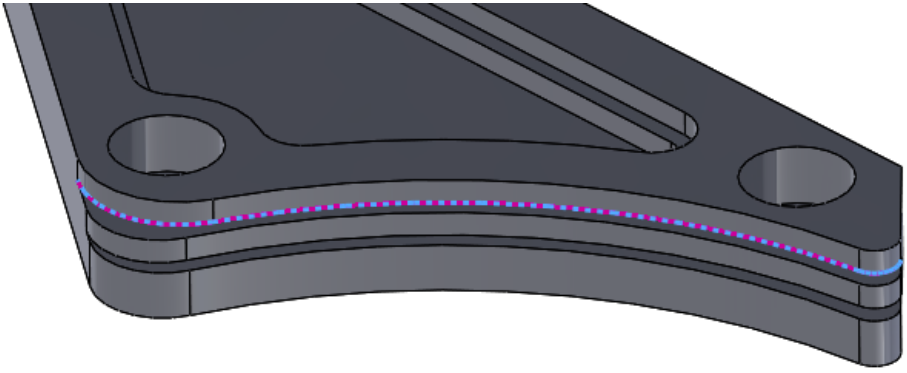
2 Ensure that you are on the **Selection** tab.

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

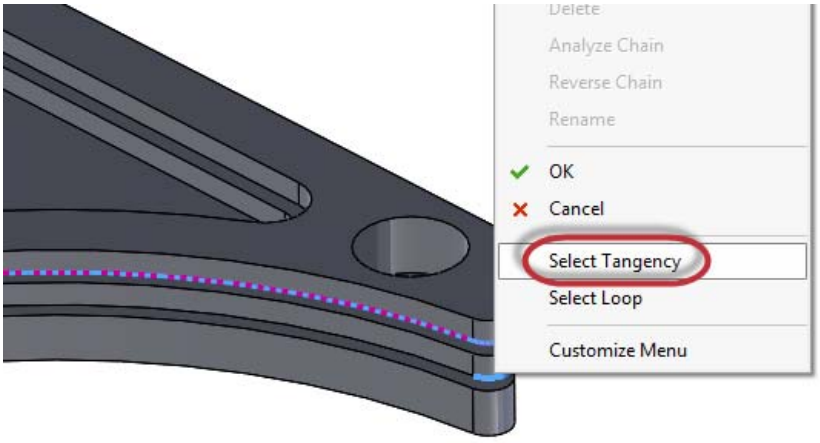


Select Tangency automatically selects curves, edges, or faces that are tangent to your initial selection.

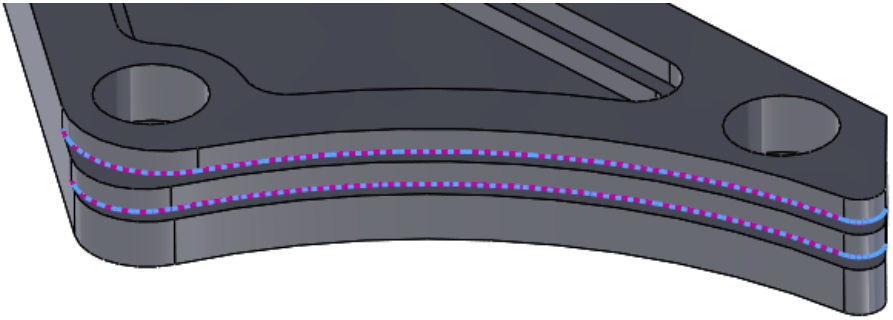
Your selection should match the image below:



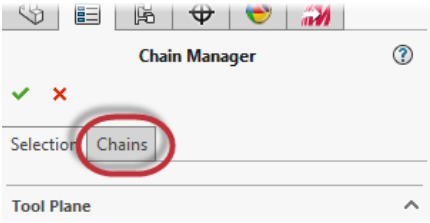
4 Right-click on the second edge shown below and choose **Select Tangency**.



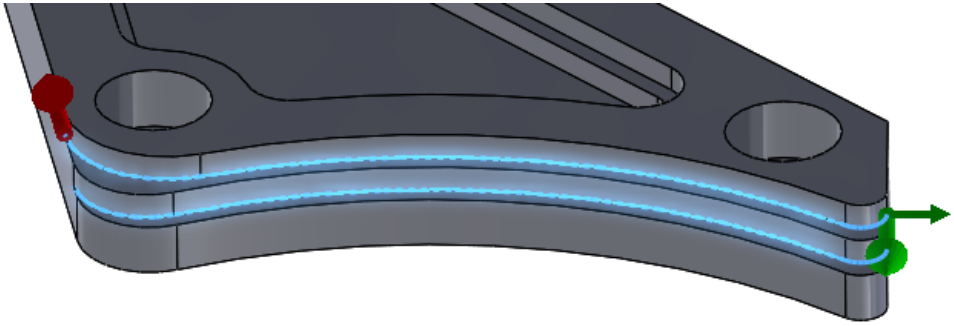
You should have the following edges selected.



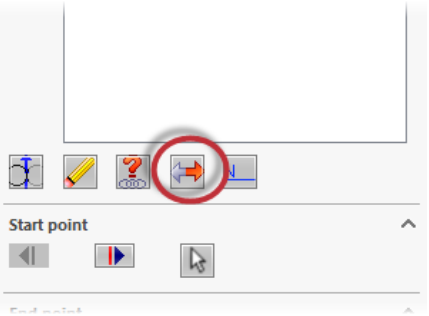
5 Select the **Chains** tab.



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



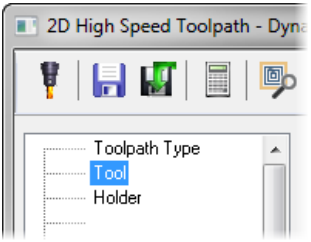
7 If the chains are going in a different direction, use the **Reverse** button to reverse the chain's direction.



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

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

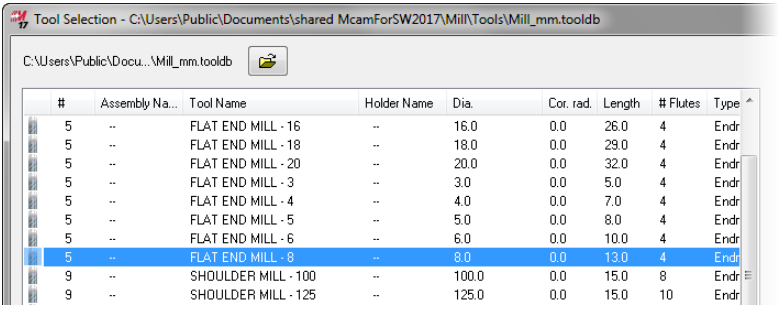
9 Select the **Tool** page.



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

The Tool Selection dialog box displays.

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

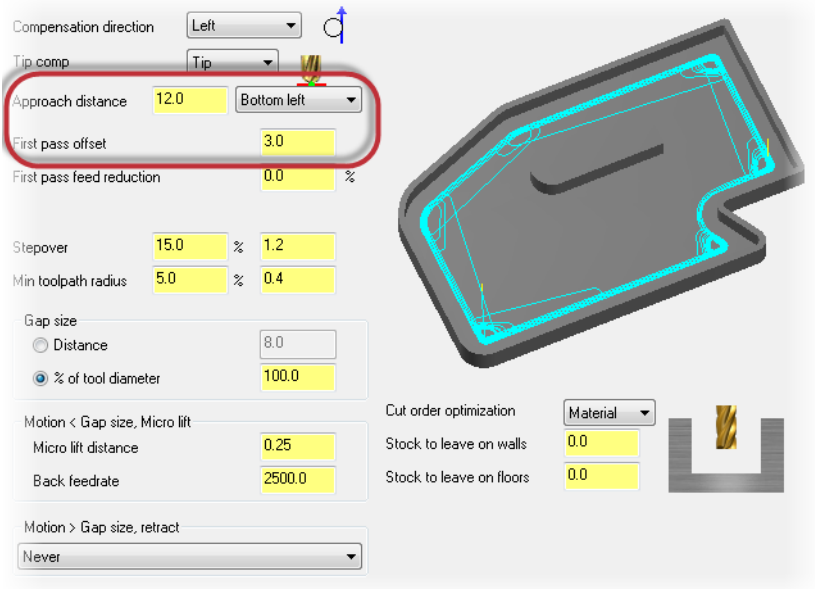


#	Assembly Na...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type
5	--	FLAT END MILL - 16	--	16.0	0.0	26.0	4	Endr
5	--	FLAT END MILL - 18	--	18.0	0.0	29.0	4	Endr
5	--	FLAT END MILL - 20	--	20.0	0.0	32.0	4	Endr
5	--	FLAT END MILL - 3	--	3.0	0.0	5.0	4	Endr
5	--	FLAT END MILL - 4	--	4.0	0.0	7.0	4	Endr
5	--	FLAT END MILL - 5	--	5.0	0.0	8.0	4	Endr
5	--	FLAT END MILL - 6	--	6.0	0.0	10.0	4	Endr
5	--	FLAT END MILL - 8	--	8.0	0.0	13.0	4	Endr
9	--	SHOULDER MILL - 100	--	100.0	0.0	15.0	8	Endr
9	--	SHOULDER MILL - 125	--	125.0	0.0	15.0	10	Endr

12 Click **OK** in the Tool Selection dialog box.

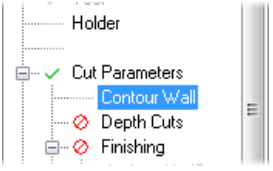
13 Select the **Cut Parameters** page.

14 Set the following parameters:

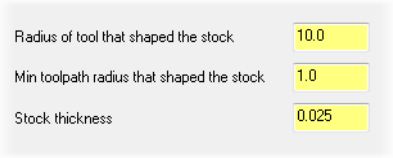


- **Approach distance** to **12.0**.
- **First pass offset** to **3.0**.

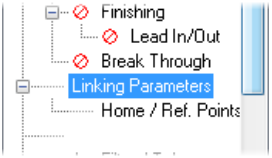
15 Select the **Contour Wall** page.



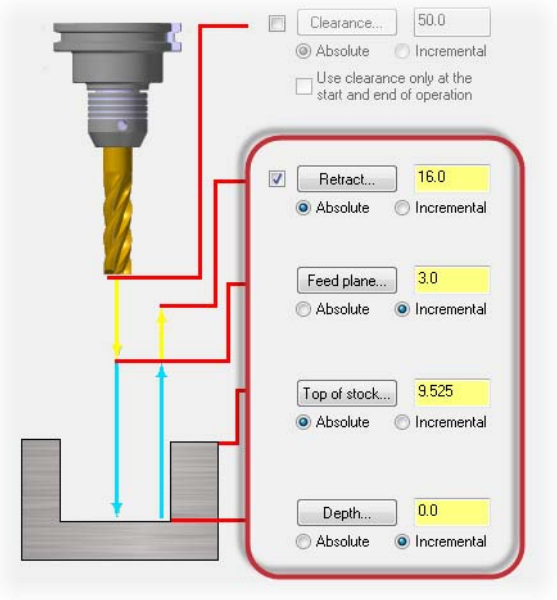
16 Set **Radius of tool that shaped the stock** to **10.0**.



17 Select the **Linking Parameters** page.



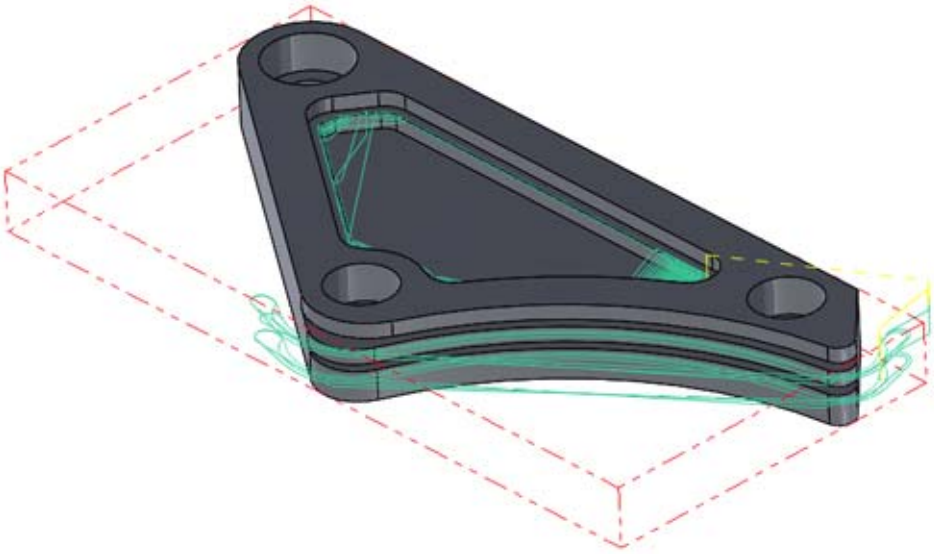
18 Set the following parameters:



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

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

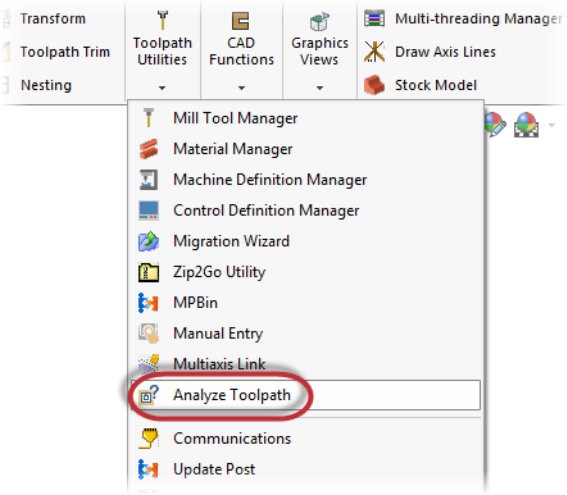
The toolpath will display as shown below:



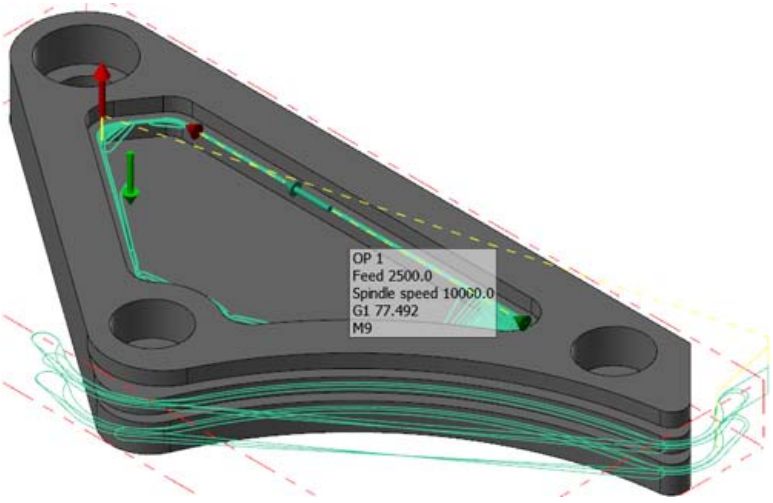
Exercise 4: Analyze the toolpaths

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 **Mastercam2017** 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.

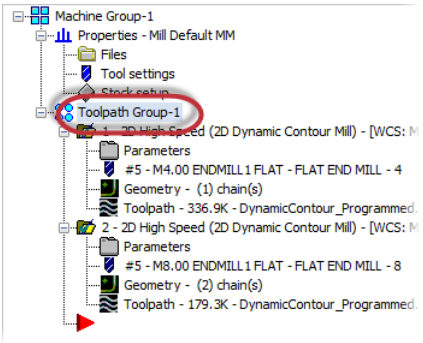
3 Continue hovering over areas of both toolpaths. Once you are satisfied, click **OK** to exit the function.

Exercise 5: Verify the toolpaths

You will now verify both toolpaths.

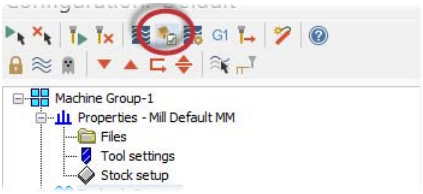
1 Select **Toolpath Group-1** in the Toolpaths Manager.

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

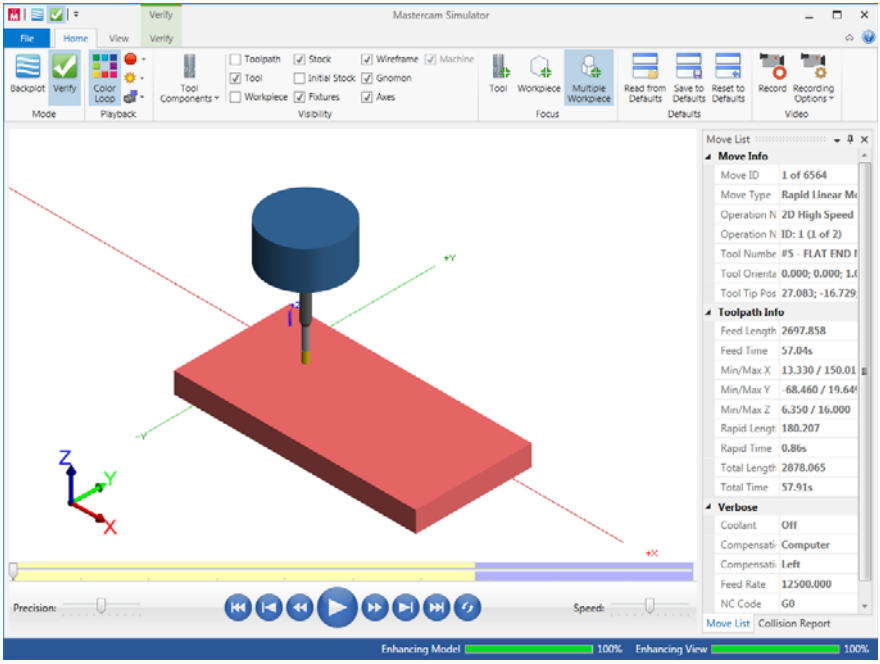


2 Select **Verify selected operations**.

Mastercam Simulator displays.

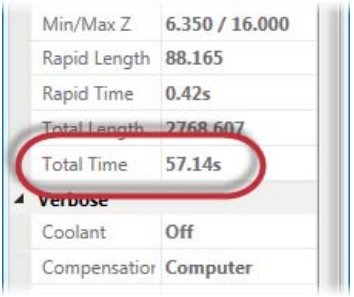


3 Right-click in the graphics view and select **Fit** and **Isometric**, if necessary, so that the display matches the image below:



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 around 1 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.

LESSON 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.

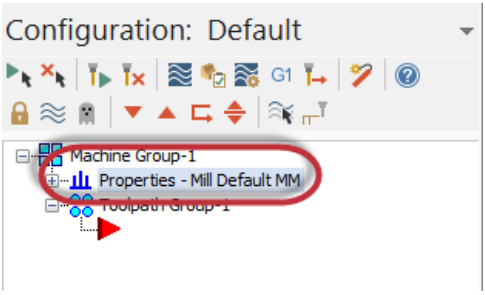
Lesson Goals

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

Exercise 1: Create the stock boundary

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

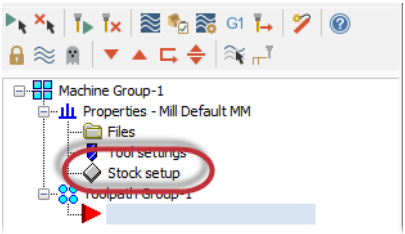
   Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.



3. If you do not have the correct default machine, select **Mastercam2017**, **Mill Machines**, **Default** from the **Tools** menu.

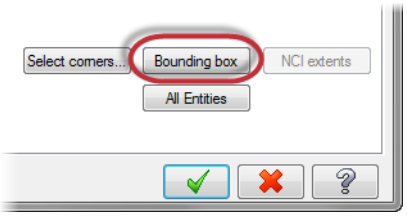
4 Select **Stock setup** from the Toolpaths Manager.

The Machine Group Properties dialog box displays.

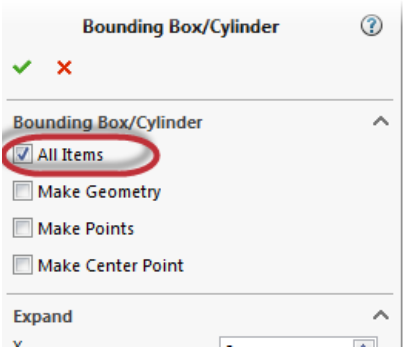


5 Select **Bounding box**.

The Bounding box function panel displays.



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



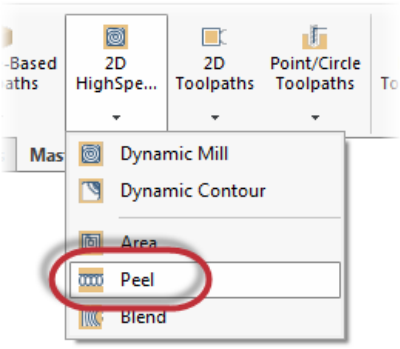
7 Click **OK** to create the boundary.

8 Click **OK** to accept the stock setup.

Exercise 2: Create the first Peel Mill toolpath

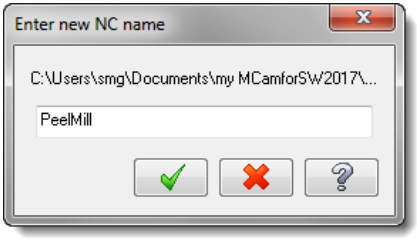
1 Select **Peel** from the **2D HighSpeed** menu on the **Mastercam2017** tab.

The Enter new NC name dialog box displays.

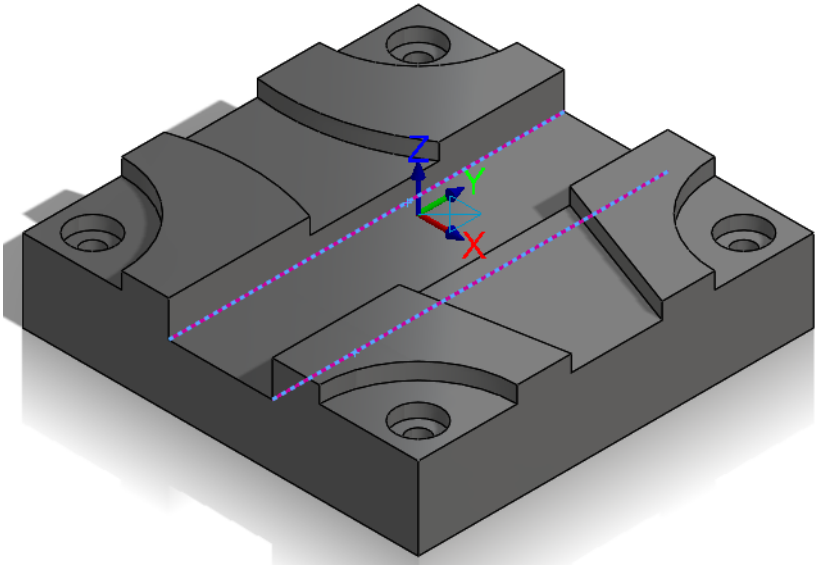


2 Click **OK** in the Enter new NC name dialog box.

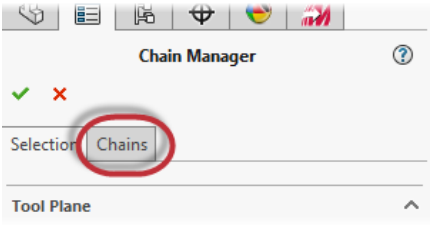
The Chain Manager displays.



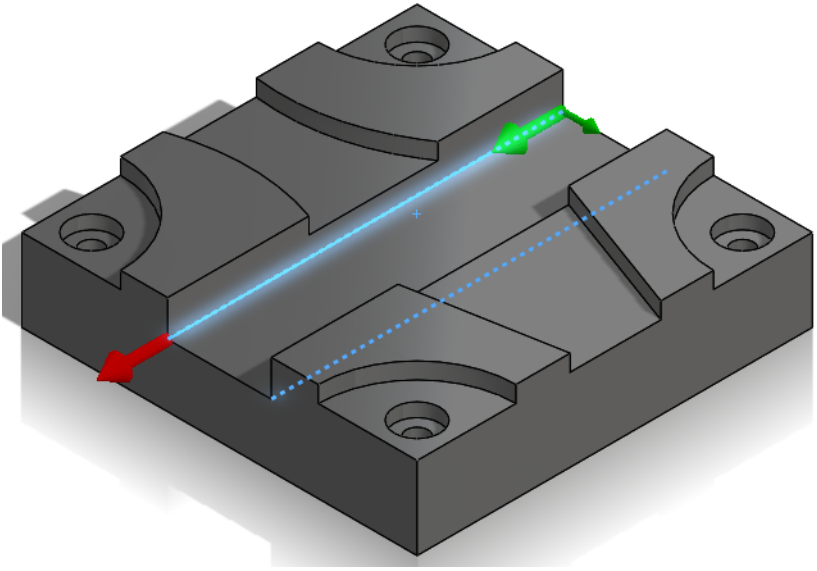
3 Select two edges shown below. You will need to rotate the part to select them.



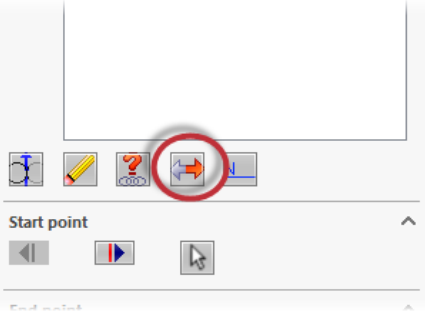
4 Select the **Chains** tab.



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



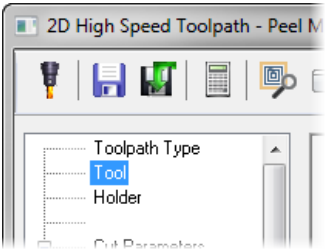
6 If one of your chains is going in the opposite direction, use the **Reverse** button to change its direction.



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

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

8 Select the **Tool** page.

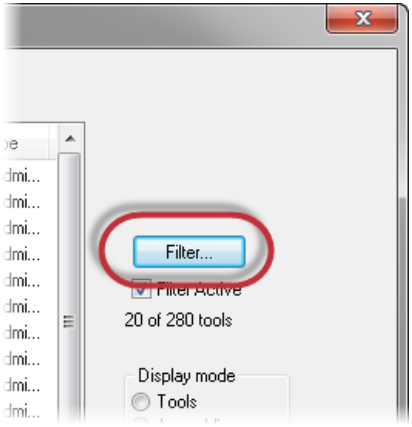


9 Click **Select library tool**.

The Tool Selection dialog box displays.

10 Click **Filter**.

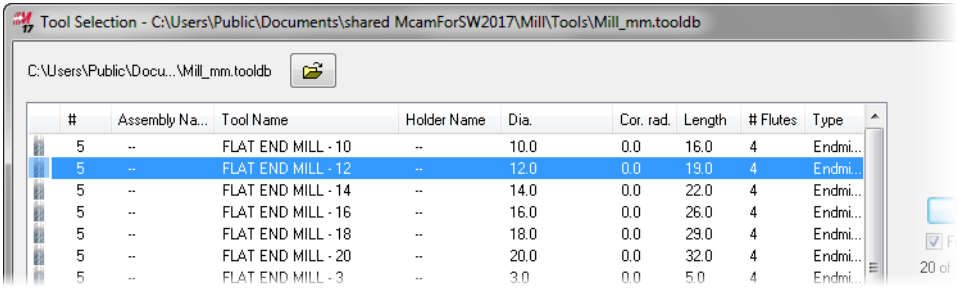
The Tool List Filter dialog box displays.



11 Ensure that tools are filtered to **Endmill1 Flat**.

12 Click **OK** in the Tool List Filter dialog box to accept the filter and return to the Tool Selection dialog box.

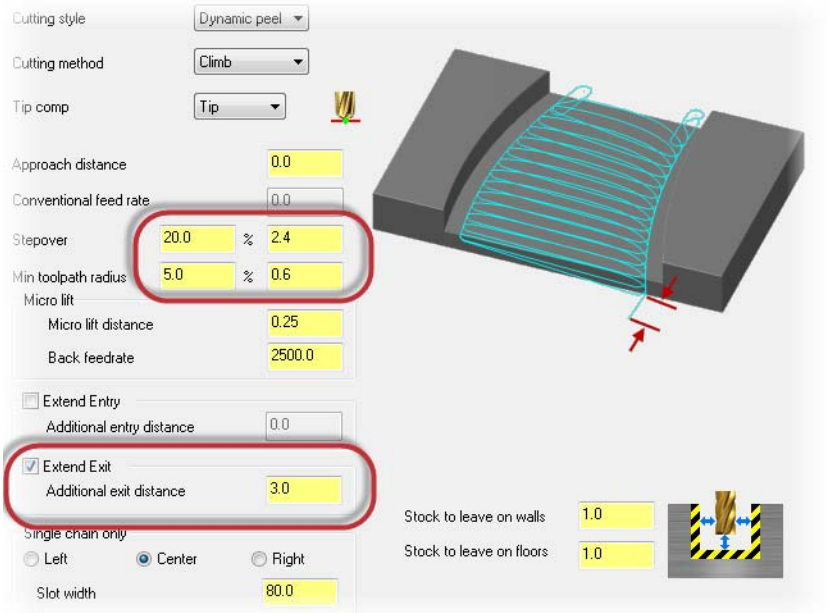
13 Select the **FLAT END MILL - 12** tool.



14 Click **OK** to add that tool and exit the Tool Selection dialog box.

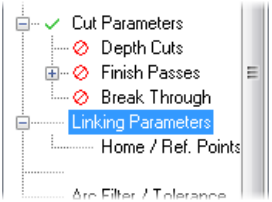
15 Select the **Cut Parameters** page.

16 Set the following parameters:



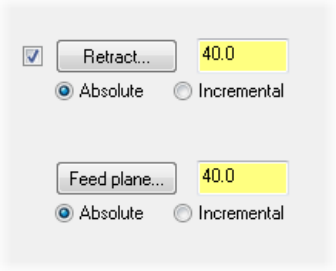
- **Stepover** to **20.0**%.
- **Min toolpath radius** to **5.0**%.
- Select **Extend Exit** and set **Additional exit distance** to **3.0**. This adjusts the final tool engagement with the material.

17 Select the **Linking Parameters** page.



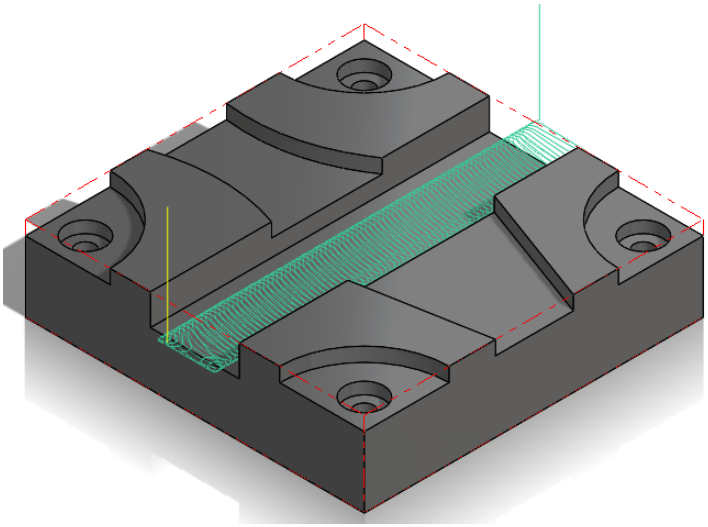
18 Set the following parameters:

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



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

The toolpath will display as shown below:

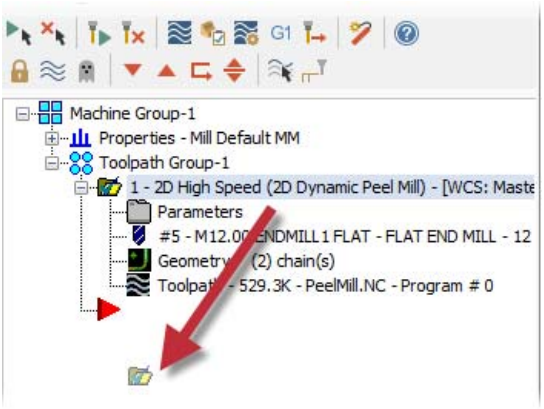


20 Save your file.

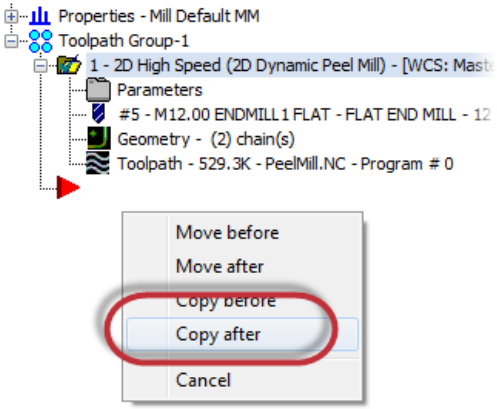
Exercise 3: Copy the Peel Mill toolpath

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.



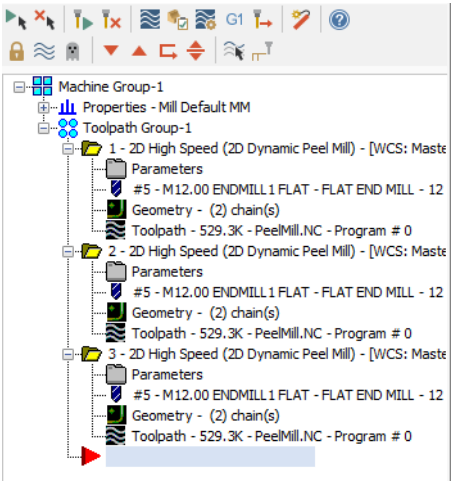
2 Select **Copy after** from the right-click menu that displays.



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

3 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.



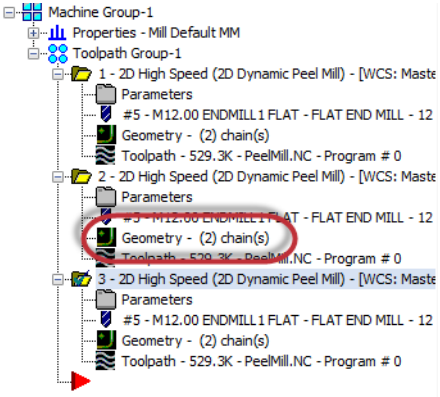
4 Save your file.

Exercise 4: Re-chain the copied Peel Mill operations

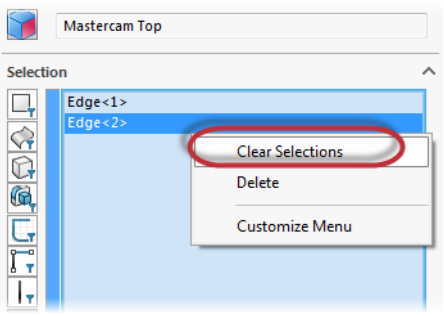
In this exercise, you will re-chain the geometry for the second Peel Mill toolpath.

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

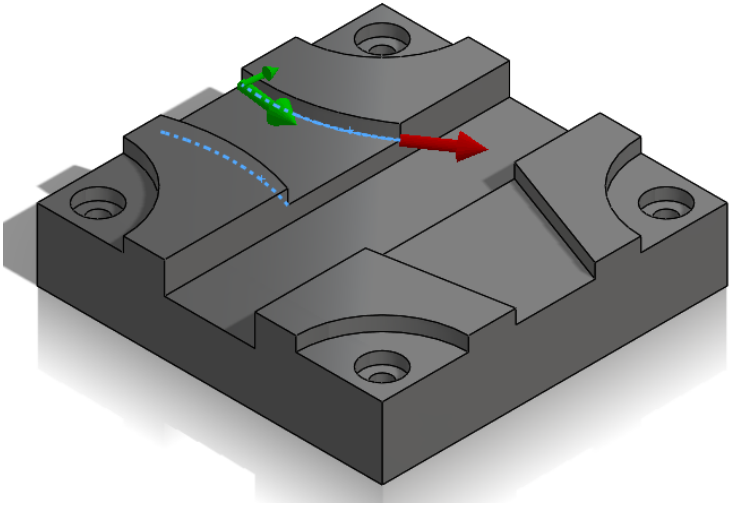
The Chain Manager displays.



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.



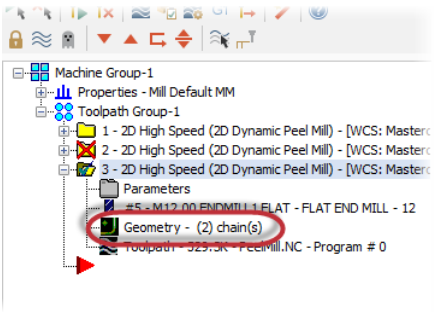
Ensure that both chains are going in the same direction. Use the **Reverse** button if necessary. It does not matter which direction they are going, except that both chains must be going in the same direction.

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

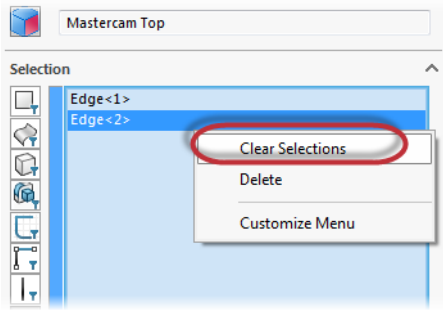
Before regenerating the toolpath, you will re-chain the third operation.

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

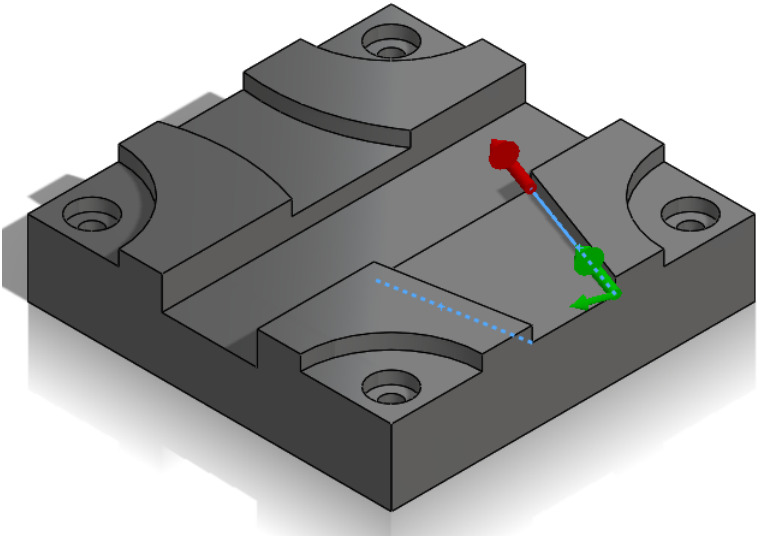
The Chain Manager displays.



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.

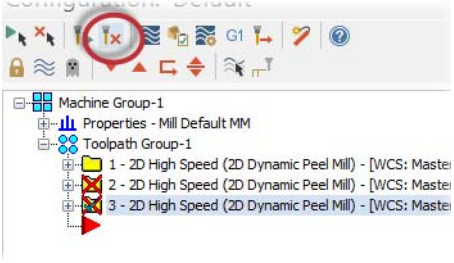


Ensure that both chains are going in the same direction.

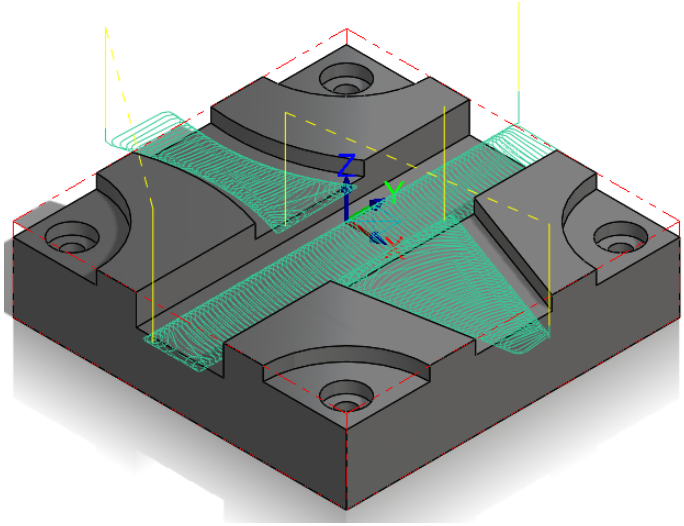
8 Click **OK** in the Chain Manager.

9 Select **Regenerate all dirty operations** in the Toolpaths Manager.

The second and third Peel Mill toolpaths regenerate using the newly selected chains.



10 All three Peel Mill toolpaths will display as shown below:



11 Save your file.

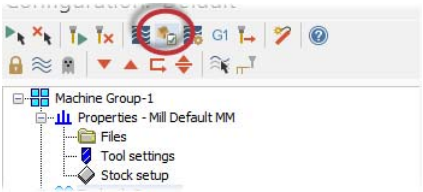
Exercise 5: Verify the toolpaths

Now you verify all three Peel Mill operations together.

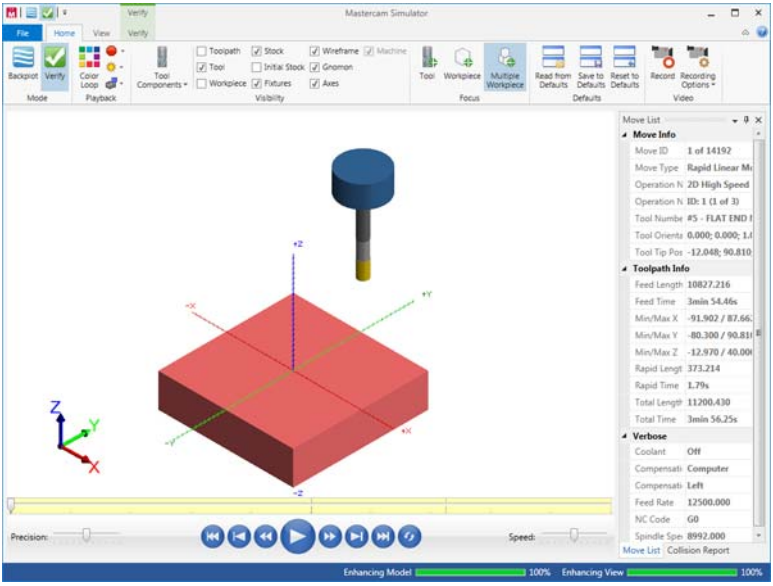
1 Select **Toolpath Group-1** in the Toolpaths Manager to select all of the toolpaths.

2 Select **Verify selected operations**.

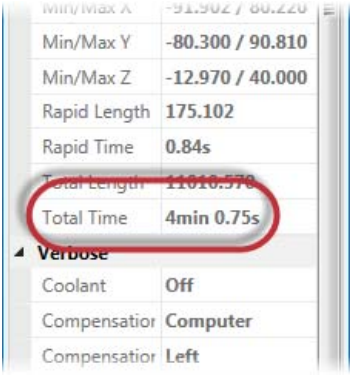
Mastercam Simulator displays.



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 runtime 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 save time. In the next lesson you create two Dynamic OptiRough operations and a stock model operation.

LESSON 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.

Lesson Goals

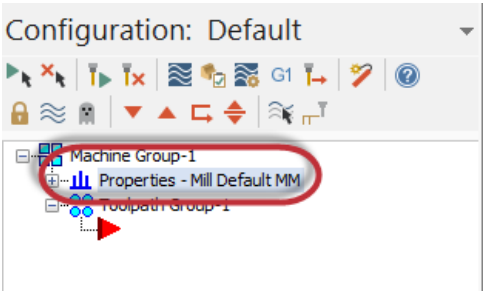
- Set up the stock
- Create two Dynamic OptiRough toolpaths
- Create a stock model operation

Exercise 1: Set 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.

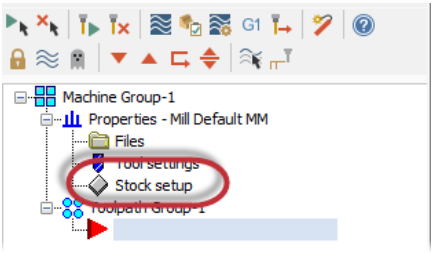
   Mastercam for SOLIDWORKS automatically adds a default machine to the Toolpaths Manager upon opening a file.



3. If you do not have the correct default machine, select **Mastercam2017**, **Mill Machines**, **Default** from the **Tools** menu.

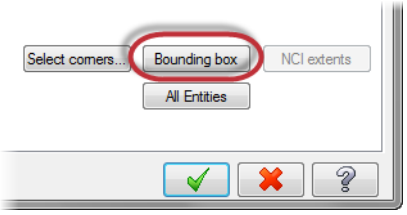
4 Select **Stock setup** from the Toolpaths Manager.

The Machine Group Properties dialog box displays.



5 Select **Bounding box**.

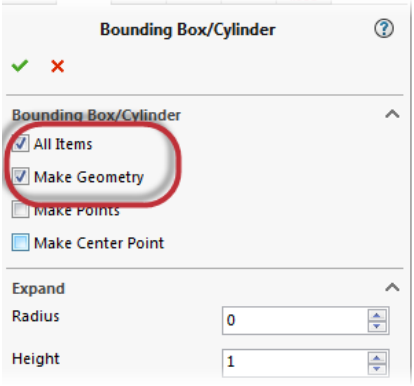
The Bounding box function panel displays.



6 Set the following parameters:

- Ensure that **All Items** is selected.
- Select **Make Geometry**.
- **Height** to **1.0.**

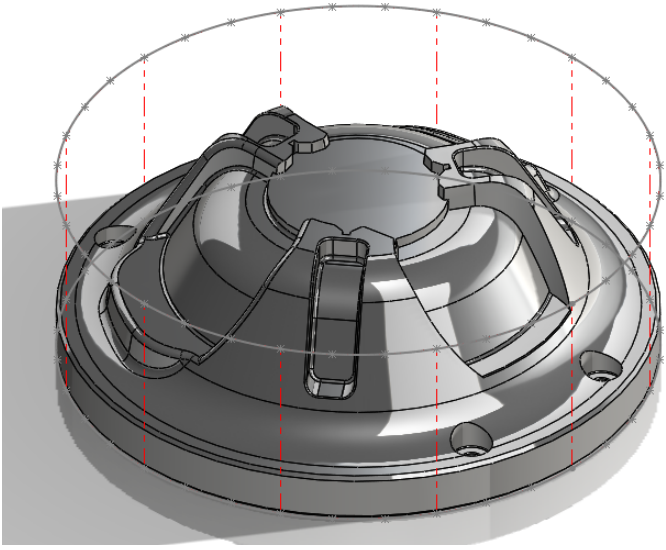
This will create geometry based on the stock boundary you created. You will then use this geometry to contain the first Dynamic OptiRough toolpath.



7 Click **OK** to create the boundary.

8 Click **OK** to 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.

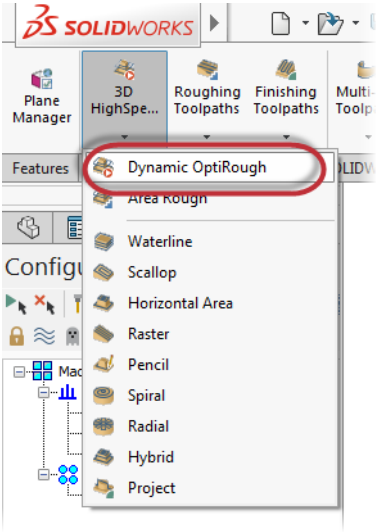


Exercise 2: Create the first toolpath

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

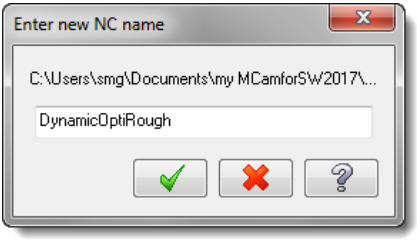
1 Select **Dynamic OptiRough** from the **3D HighSpeed** menu on the **Mastercam2017** tab.

The Enter new NC name dialog box displays.

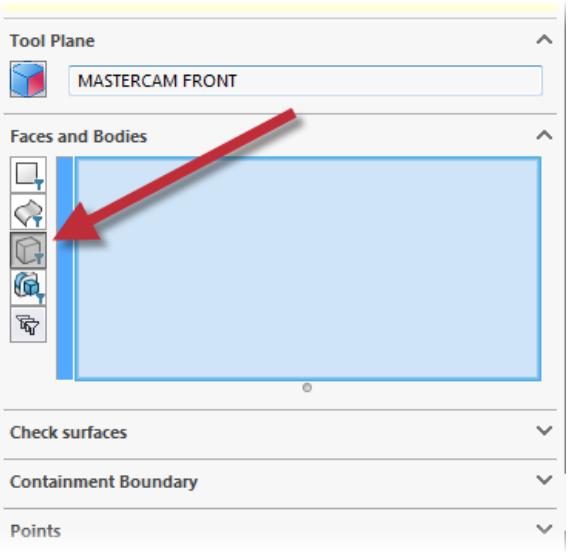


2 Click **OK** in the Enter new NC name dialog box.

The Selection panel displays.

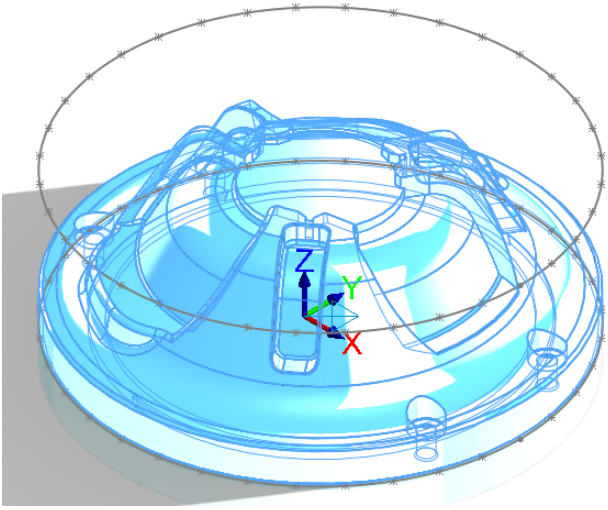


3 In the Selection panel, activate the **Bodies** filter.

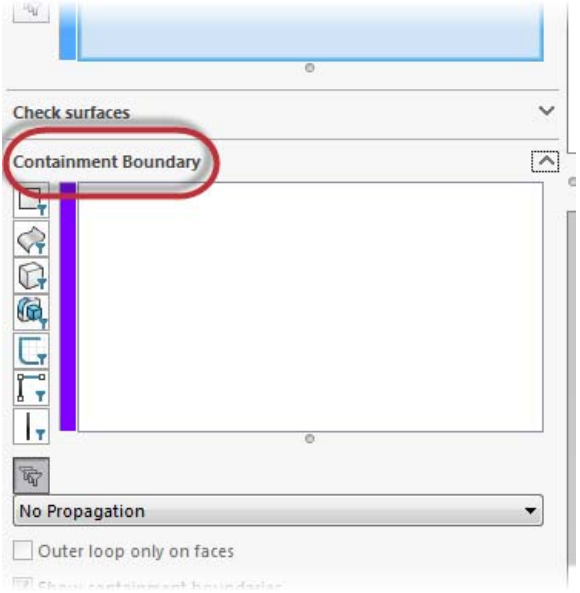


This limits your geometry selection to only bodies.

4 Select the part:

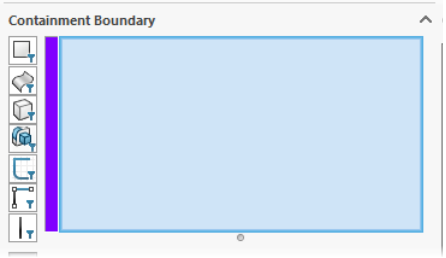


5 Expand the **Containment Boundary** section..

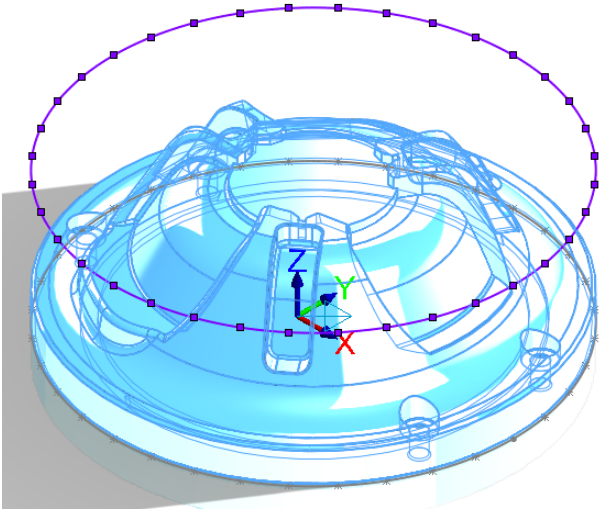


6 Click in the list to activate Containment Boundary selection.

The list window will turn blue to acknowledge that you are now selecting a containment boundary.



7 Select the spline as shown below to be the containment boundary:

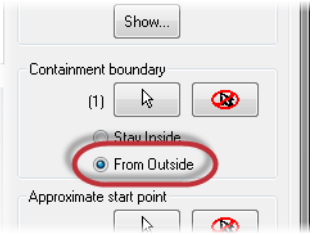


The containment boundary limits tool motion to a selected area. In this case, you want to limit tool motion to around the part.

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

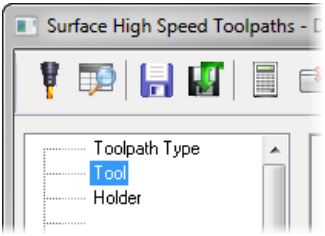
The Surface High Speed Toolpaths - Dynamic OptiRough dialog box displays.

9 On the Toolpath Type page, ensure that the Containment boundary is set 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.

10 Select the **Tool** page.

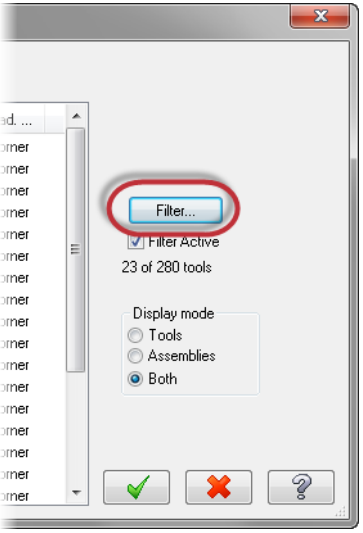


11 Click the **Select library tool** button.

The Tool Selection dialog box displays.

12 Select the **Filter** button.

The Tool List Filter dialog box displays.



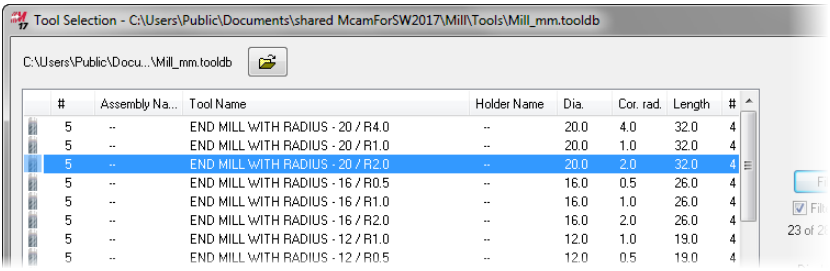
13 Set the Tool Type filter to **None**.

14 Select the **Endmill3 Bull** filter. This ensures that you are only selecting Bull Endmill tools.

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

You return to the Tool Selection dialog box.

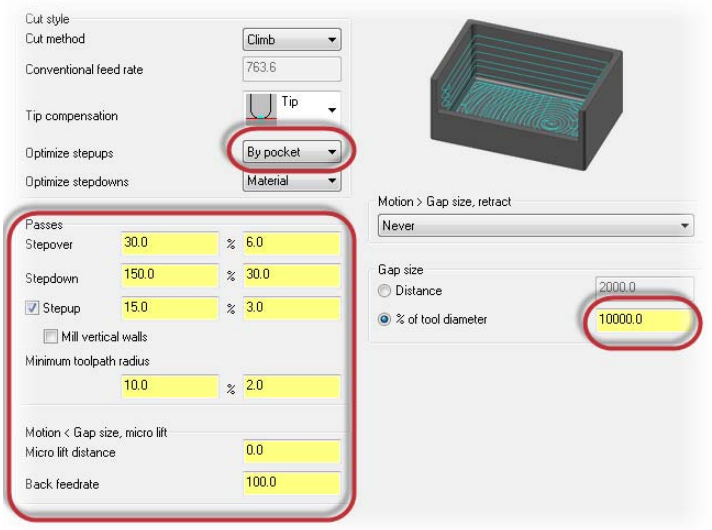
16 In the Tool Selection dialog box, select the **END MILL WITH RADIUS - 20/ R2.0** tool.



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

18 Select the **Cut Parameters** page.

19 Set the following parameters:

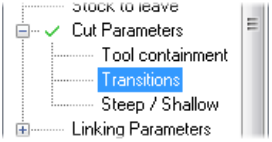


- **Optimize stepups** to **By pocket**. Mastercam first machines all stepdowns, moving from pocket to pocket. After all stepdowns on a z-

level are machined to completion, Mastercam machines the stepups by next closest, in the safest cut order.

- **Stepover** to **30.0**%. Sets the distance between cutting passes in the X and Y axes.
- **Stepdown** to **150.0**%. Determines the Z spacing between adjacent cutting passes.
- Select the **Stepup** checkbox and enter **15.0**. Mastercam calculates and adds step up 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**.

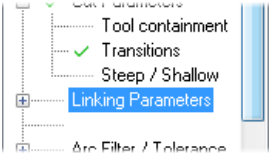
20 Select the **Transitions** page.



21 Set the following parameters:

- Set **Helix radius** to **9.0**. Sets the radius of the entry helix.
- Set **Skip pockets smaller than** to **13.2**. 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.

22 Select the **Linking Parameters** page.



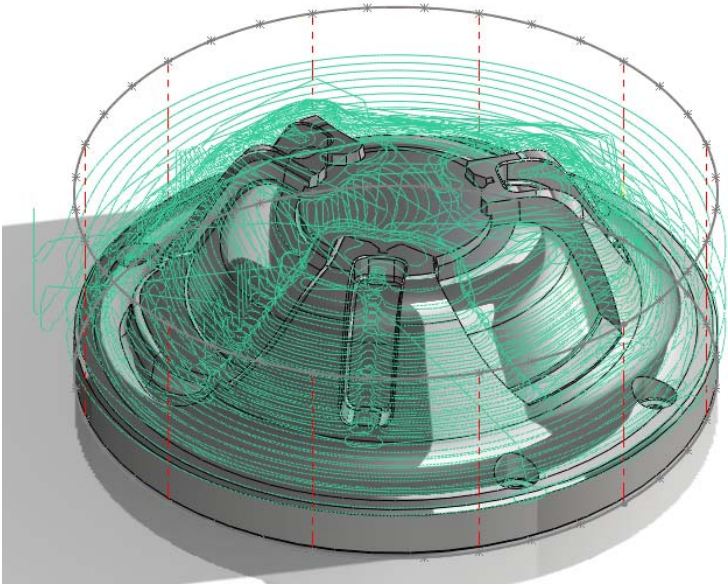
23 Set the following parameters:

- Set the **Retract Method** drop-down to **Minimum Distance**. Creates high speed loops into and out of each retract move.
- Check **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.

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

25 Your toolpath should display as follows:

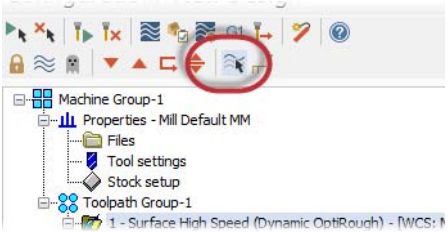


26 Save your part file.

Exercise 3: Create the stock model operation

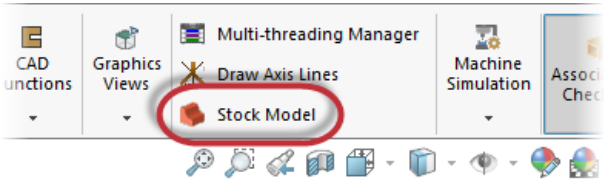
You now create a stock model operation that uses the previous operation as its source. This stock model will be used in the second Dynamic OptiRough toolpath to clear out any remaining stock.

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



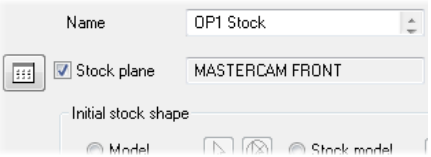
This will only display the toolpath you have selected. It will also clean up the graphics view when making modifications to the part or creating another toolpath.

2 Select **Stock Model** on the **Mastercam2017** 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.

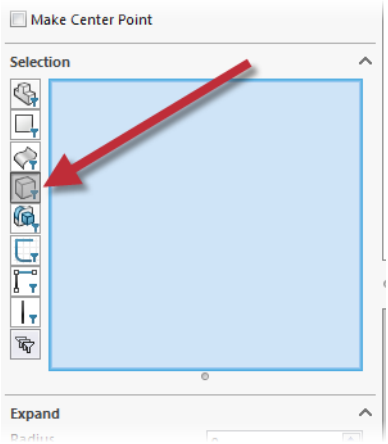
5 Select the **Bounding box** button.

The Bounding box function panel displays.

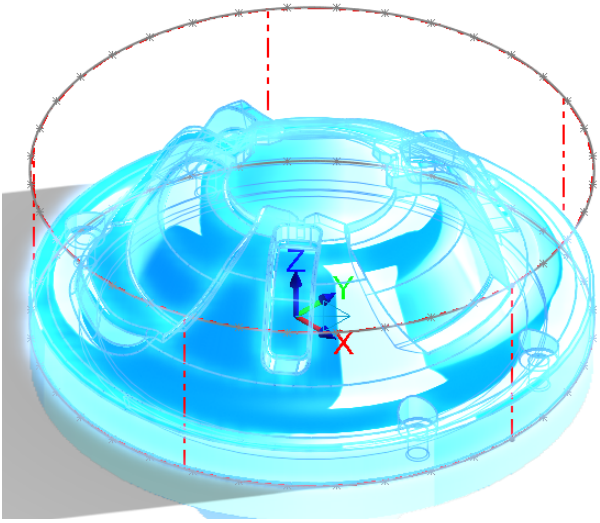
6 Deselect **All Items** and **Make Geometry**.

In this case, you do not need to create a stock boundary for all geometry, as it would then include the stock geometry created in the previous stock boundary.

7 Select the **Bodies** filter to limit selection to bodies.



8 Select the part as shown below:

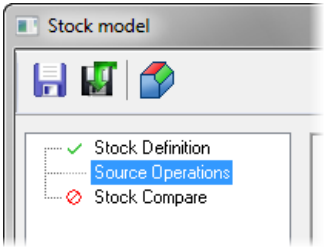


9 Set the **Height** to **1.0**.

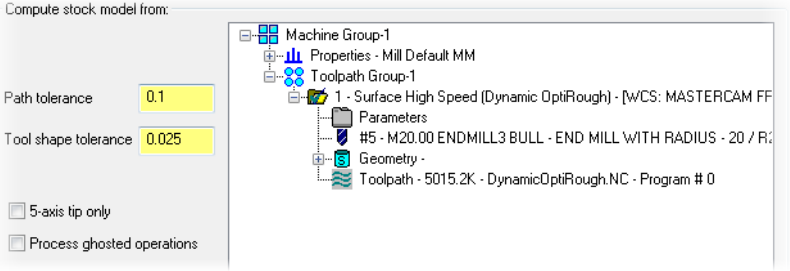
10 Select **Cylindrical** and set it to the **Z** axis.

11 Click **OK** to accept the stock model and return to the Stock model dialog box.

12 Select the **Source Operations** page.



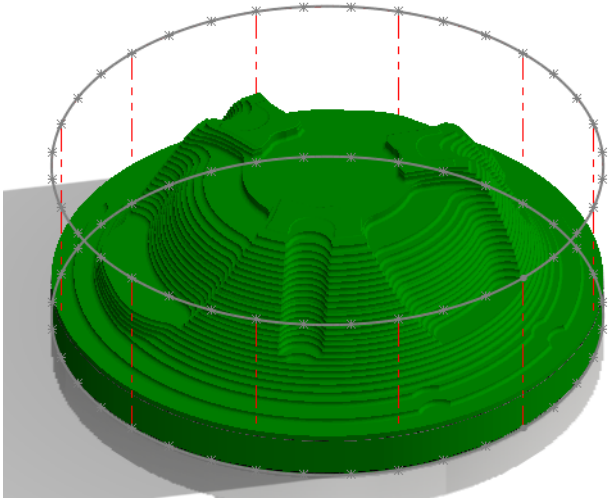
13 Select the **Surface High Speed (Dynamic OptiRough)** toolpath as the source operation.



The stock model is generated by running the selected operation against the parameters on the Stock Definition page.

14 Click **OK** to create the stock model operation.

15 Your stock model displays as shown below:



This may take some time to calculate.

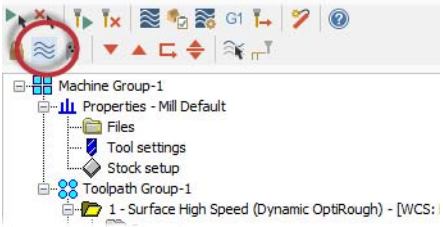
16 Save your part.

Exercise 4: Create the second toolpath

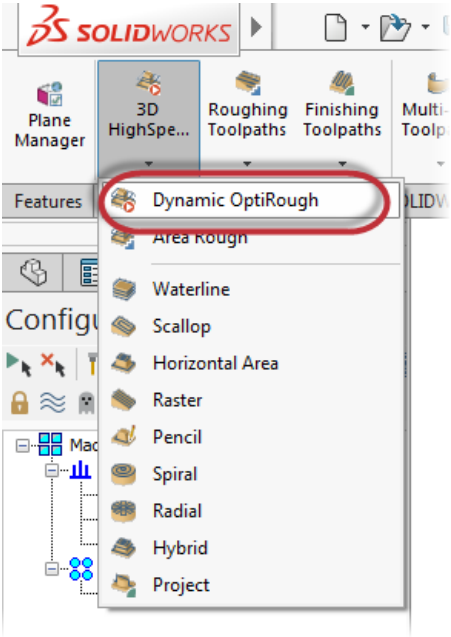
You now create the second Dynamic OptiRough toolpath that uses the stock model you created in the previous exercise.

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.

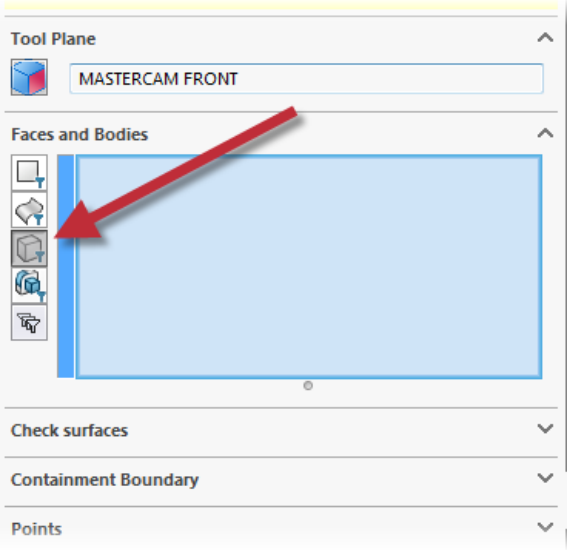


3 Select **Dynamic OptiRough** from the **3D HighSpeed** menu on the **Mastercam2017** tab.

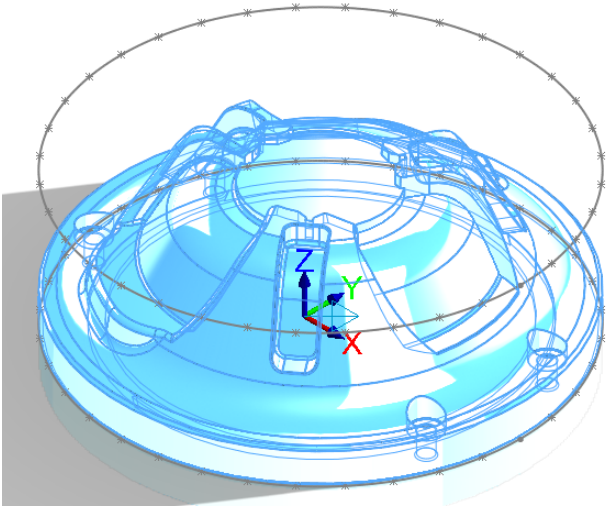


The Selection panel displays.

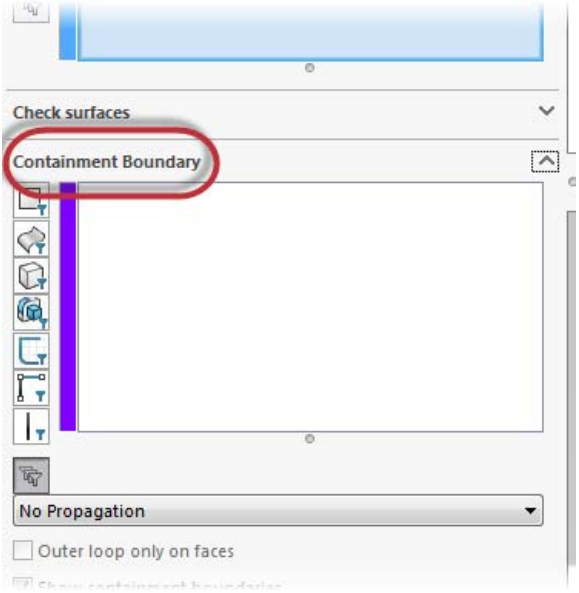
4 In the Selection panel, ensure the **Bodies** filter is activated.



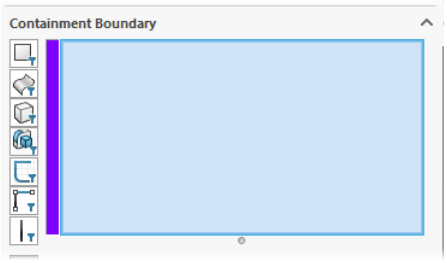
5 Select the part as shown below:



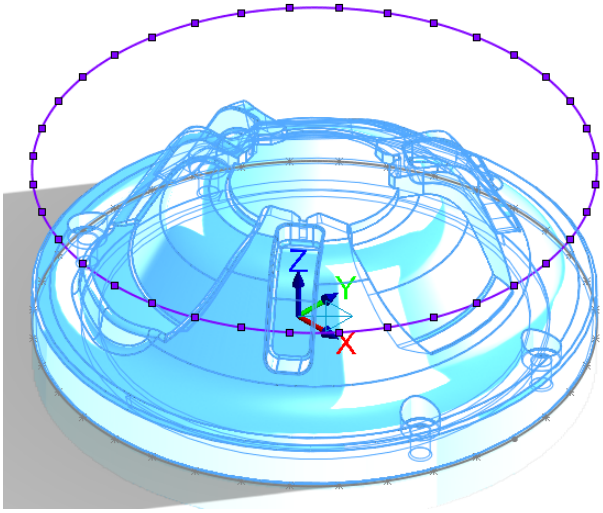
6 Expand the **Containment Boundary** section.



7 Click in the list to activate Containment Boundary selection.



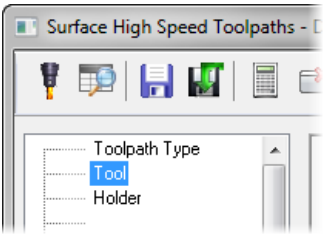
8 Select the spline shown below as the containment boundary:



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

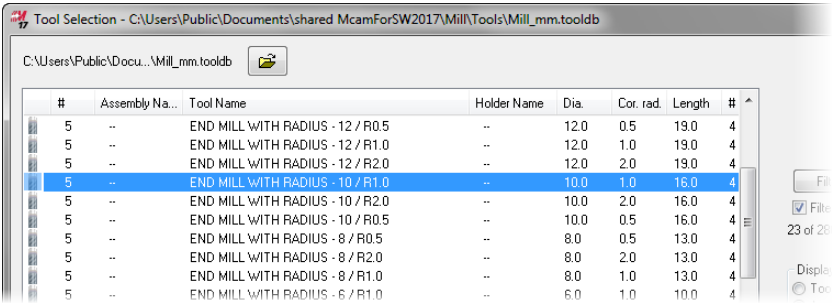
The Surface High Speed Toolpaths - Dynamic OptiRough dialog box displays.

10 Select the **Tool** page.



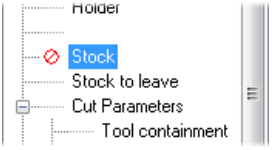
11 Click the **Select library tool** button.

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

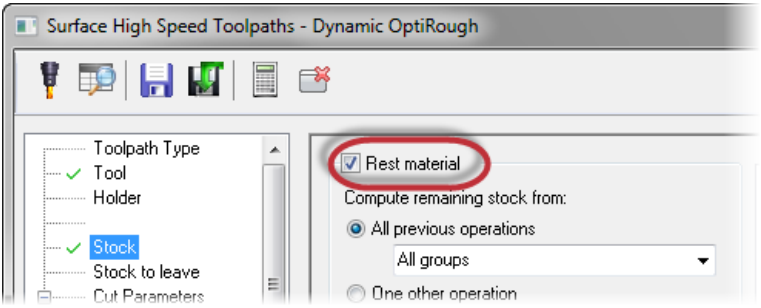


13 Click **OK** to add the tool to the toolpath.

14 Select the **Stock** page.

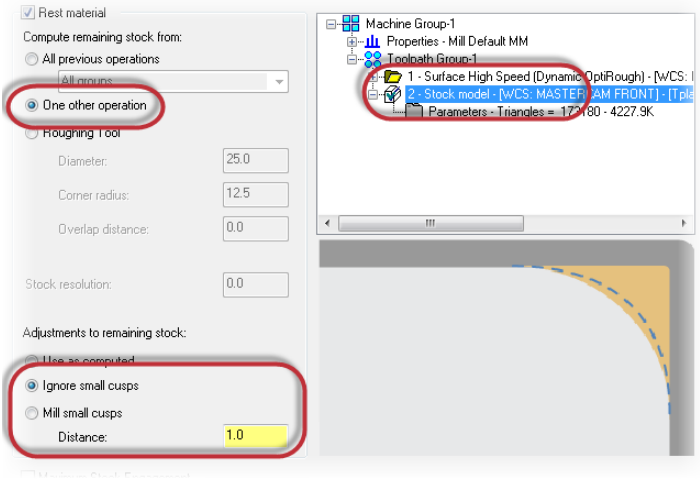


15 Select the **Rest material** checkbox.



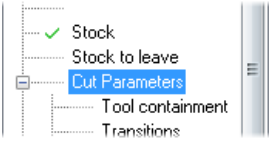
When you enable this page, Mastercam calculates the cutting passes based on the remaining stock.

16 Set the following parameters:

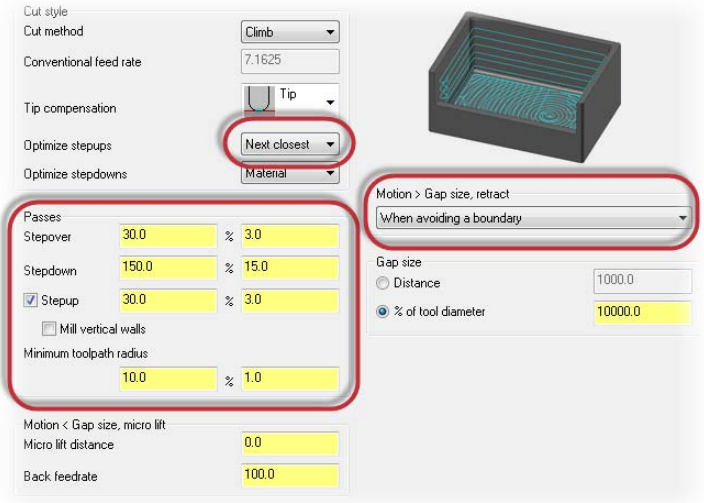


- Set **Compute remaining stock from** to **One other operation**.
- In the **Operation List**, select **Stock Model - OP1 Stock**.
- Set **Adjustments to remaining stock** to **Ignore small cusps**.
- Set **Distance** to **1.0**. This option is used in conjunction with **Ignore small cusps**. Mastercam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

17 Select the **Cut Parameters** page.



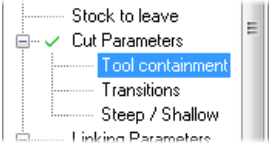
18 Set the following parameters:



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

19 Select the **Tool containment** page.

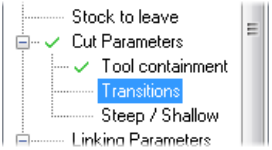
This page controls the tool's position around the boundary of your part.



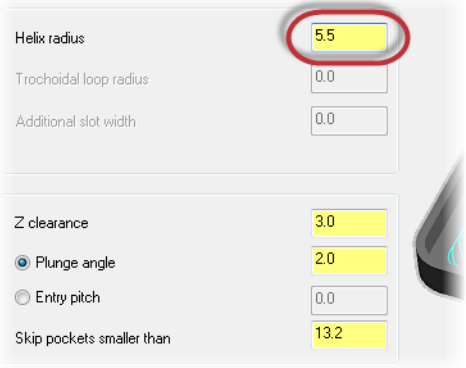
20 Set the following parameters:

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

21 Select the **Transitions** page.

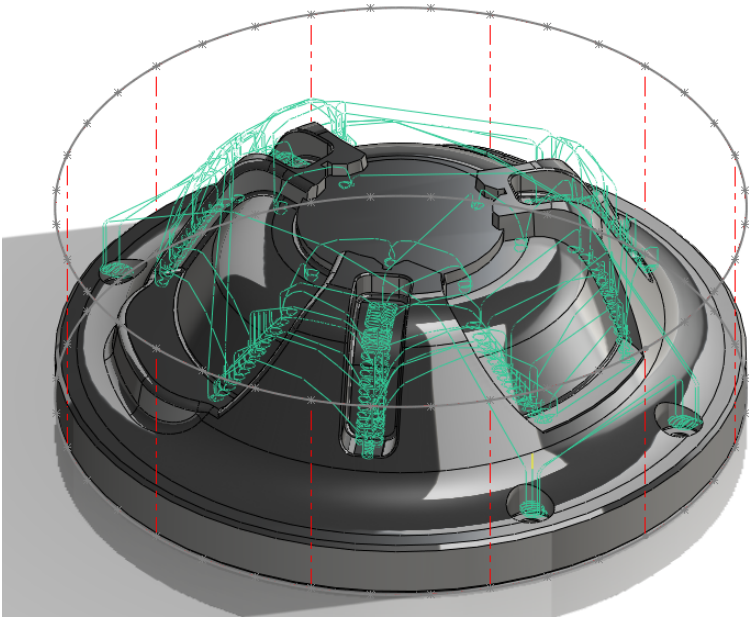


22 Set the **Helix radius** to **5.5**.



23 Click **OK** to create and generate the Dynamic OptiRough toolpath.

24 Your toolpath will display as shown below:

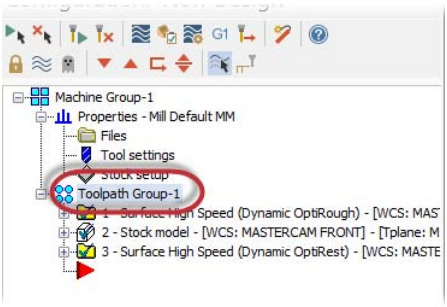


25 Save your part file.

Exercise 5: Verify the stock model and toolpaths

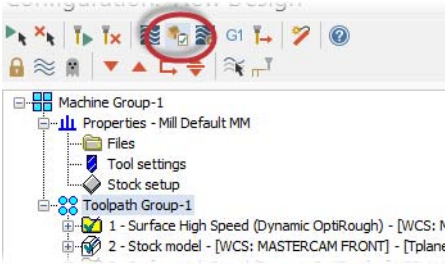
In this exercise, you use Mastercam Simulator to verify your toolpaths and stock model. Verifying your toolpaths allows you to use solid models to simulate part machining against the selected stock definition.

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

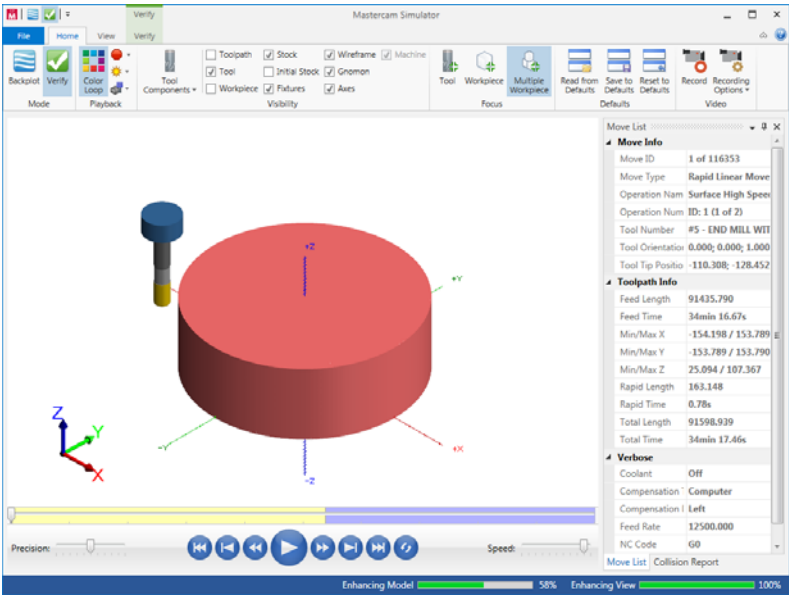


2 Select **Verify selected operations** in the Toolpaths Manager.

Mastercam Simulator displays.

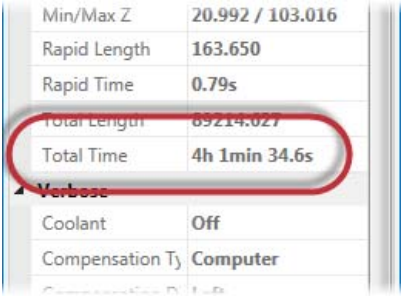


3 If necessary, right-click in the graphics window and change the view to **Isometric** and **Fit** the part to the screen so that it matches the image shown below.



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

5 View the Move List to view the total runtime 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 2017 for SOLIDWORKS Tutorial (Mill)* tutorial. Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

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

Mastercam for SOLIDWORKS Resources

Enhance your Mastercam for SOLIDWORKS experience by using the following resources:

- *Mastercam for SOLIDWORKS Help*—Access Mastercam for SOLIDWORKS Help by selecting **Mastercam2017, Contents** from the SOLIDWORKS Help menu, or by clicking the **Help Topics** link in the Mastercam Info Center on the SOLIDWORKS Task Pane. Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam for SOLIDWORKS Help directly to related information.
- *Mastercam for SOLIDWORKS Reseller*—Your local Mastercam for SOLIDWORKS Reseller can help with most questions about Mastercam for SOLIDWORKS.
- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—CNC offers a series of tutorials to help registered users become familiar with basic Mastercam features and functions. The Mastercam tutorial series is in continual development, with new modules added as we complete them. Visit our website to see the latest publications.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.

- *Online communities*— You can find a wealth of information, including many videos, at www.mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

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

  To register, select **Help**, **Mastercam2017**, **Link on Mastercam.com** from the Mastercam menu and follow the instructions.

Mastercam for SOLIDWORKS Documentation

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

- *What's New in Mastercam 2017 for SOLIDWORKS*
- *Mastercam 2017 for SOLIDWORKS Installation Guide*
- *Mastercam 2017 for SOLIDWORKS Administrator Guide*
- *Mastercam 2017 for SOLIDWORKS Transition Guide*
- *Mastercam 2017 for SOLIDWORKS Quick Reference Card*
- *Mastercam 2017 for SOLIDWORKS Tutorial (Mill)*
- *Mastercam 2017 for SOLIDWORKS ReadMe*

Contact Us

For questions about this or other Mastercam for SOLIDWORKS documentation, contact the Technical Documentation department by email at tech-docs@mastercam.com.

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

CNC software, inc.

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