Terms of Use

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

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

Be sure you have the latest information!

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

**Introduction** ........................................................................................................................................... 5  
General Tutorial Requirements .................................................................................................................. 6

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

Dynamic Toolpath Types .............................................................................................................................. 7
  Dynamic Contour ....................................................................................................................................... 7
  Face ......................................................................................................................................................... 8
  Dynamic Mill ........................................................................................................................................... 8
  Peel Mill ................................................................................................................................................. 10
  Dynamic OptiRough ................................................................................................................................. 11
  Dynamic Toolpath Selection ..................................................................................................................... 12
  Dynamic Milling Parameters .................................................................................................................... 14

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

Exercise 1: Opening Mastercam ................................................................................................................ 17
Exercise 2: Setting up the stock .................................................................................................................. 18
Exercise 3: Creating the first Dynamic Mill toolpath .............................................................................. 22
Exercise 4: Creating the second Dynamic Mill toolpath ......................................................................... 32
Exercise 5: Verifying the toolpaths ......................................................................................................... 40

**The Face Toolpath** .................................................................................................................................... 43

Exercise 1: Creating the Face toolpath ..................................................................................................... 43
Exercise 2: Verifying the toolpaths .......................................................................................................... 48

**The Dynamic Contour Toolpath** ......................................................................................................... 51

Exercise 1: Setting up the stock ................................................................................................................. 51
Exercise 2: Creating the first Dynamic Contour toolpath ....................................................................... 53
Exercise 3: Creating the second Dynamic Contour toolpath .................................................................. 60
Exercise 4: Analyzing the toolpaths ......................................................................................................... 66
Exercise 5: Verifying the toolpaths ......................................................................................................... 67

**The Peel Mill Toolpath** ......................................................................................................................... 71

Exercise 1: Setting up the stock ................................................................................................................. 71
Exercise 2: Creating the first Peel Mill toolpath ...................................................................................... 72
Exercise 3: Copying the Peel Mill toolpath ................................................................. 78
Exercise 4: Re-chaining the copied Peel Mill operations .............................................. 79
Exercise 5: Verifying the toolpaths .............................................................................. 82

**The Dynamic OptiRough Toolpath** ....................................................................... 85

Exercise 1: Setting up the stock .................................................................................. 85
Exercise 2: Creating the first Dynamic OptiRough toolpath ......................................... 89
Exercise 3: Creating the Stock Model operation ......................................................... 96
Exercise 4: Creating the second Dynamic OptiRough toolpath .................................... 98
Exercise 5: Verifying the stock model and toolpaths .................................................... 105

**Conclusion** ........................................................................................................... 109

Mastercam Resources ................................................................................................. 109
Contact Us ................................................................................................................. 109
INTRODUCTION

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

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

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

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

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

Tutorial Goals

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

Note: You must have a 3D license to complete the Dynamic OptiRough chapter.

WARNING: Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.
Estimated time to complete this tutorial: 5 hours

General Tutorial Requirements

All Mastercam 2019 tutorials have the following general requirements:

- You must be comfortable using the Windows® operation system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (emcam) is different from Mastercam (mcam), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.
DYNAMIC TOOLPATHS OVERVIEW

The following toolpaths use Dynamic Motion technology:

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

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

**Dynamic Toolpath Types**

**Dynamic Contour**

The Dynamic Contour toolpath creates a familiar contour operation using dynamic motion and specialized options found only in dynamic toolpaths. The dynamic motion prevents tool burial and binding in small radii corners. Additionally, you can specify how Mastercam machines any material left on the walls.
Face
The Face toolpath, when the Style is set to Dynamic, quickly cleans stock from the top of a part and creates an even surface for future operations. You can base the toolpath on either chained geometry or on the current stock model.

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

By selecting the Stay inside strategy, Dynamic Mill becomes a pocket toolpath that utilizes dynamic motion. You select a boundary, areas to avoid, and set tooling, parameters, entry method, and linking values to generate the toolpath. Material is removed from the inside to the outside in a highly efficient manner.
By selecting the **From outside** strategy, Dynamic Mill becomes a facing operation that has island avoidance capabilities. This movement creates a toolpath that is allowed to move outside the selected material and removes from the outside to the inside.

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

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

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

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

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

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

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

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

For more information, view Mastercam's Help.
Dynamic OptiRough, as it is the only 3D Dynamic toolpath, uses a different type of selection method compared to the other Dynamic toolpaths. When creating a Dynamic OptiRough toolpath, you are brought to the Model Geometry page in the Surface High Speed Toolpaths - Dynamic OptiRough dialog box. You then have the following options:

- **Machining Geometry**: Geometry to be machined. You can have multiple machining groups.
- **Avoidance Geometry**: Geometry to be avoided. You can have multiple avoidance groups.
- For each geometry group, you can assign variable stock to leave values on its walls and floors.
You can then use the Toolpath Control page to set containment parameters for the Dynamic OptiRough toolpath, which controls the tool's position around the boundary of your part.

![Surface High Speed Toolpaths - Dynamic OptiRough](image)

**Dynamic Milling Parameters**

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

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

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

![Motion: Gap size, micro lift](image)
Entry methods provide several ways to enter the material for a Dynamic Mill toolpath. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's Help contains detailed information on each of these methods.

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

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

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

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

Creating and reviewing the motion of dynamic toolpaths is the best way to learn about the benefits and uses.
CHAPTER 1
THE DYNAMIC MILL TOOLPATH

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

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

Lesson Goals

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

Exercise 1: Opening Mastercam

1. Start Mastercam using your preferred method:
   a. Double-click Mastercam’s desktop icon.
   OR
   b. Launch Mastercam from the Windows Start menu.

2. Select the default metric configuration file:
   a. Click the File tab.
   b. Choose Configuration from Mastercam’s Backstage View to open the System Configuration dialog box.
   c. Choose ...\mcaxm.config <Metric> from the Current drop-down list.
d. Click OK.

**Exercise 2: Setting up the stock**

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

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

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

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

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

A new level is added to the list and is then set as the main level.
6. Double-click in the Name column of the level you have just created and rename it to **Stock geometry**.

   ![Levels](image)

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

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

   ![Machine Group Properties](image)

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

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

   ![Bounding box](image)

   The **Bounding Box** function panel displays.
9. Select the entire part, as shown below:

10. Press [Enter] or click End Selection to accept the selection.

11. Set the following parameters in the Bounding Box function panel:

   - Shape: Rectangular
   - Rectangular Settings
     - Origin:
     - Size:
       - X: 152.4
       - Y: 49.53
       - Z: 28.6
   - Cylindrical Settings
   - Push Pull
   - Create Geometry
     - Lines and arcs

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

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

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

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

14. **Save your part.**
Exercise 3: Creating the first Dynamic Mill toolpath

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

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

   ![Image of Dynamic Mill toolpath selection](image)

   The Chain Options dialog box displays.

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

   ![Image of Chain Options dialog box](image)

   The Chaining dialog box displays.

3. Select Cplane in the Chaining dialog box.

   ![Image of Chaining dialog box with Cplane selected](image)

   This option allows you to chain only entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain.
4. Select the chain shown below as the machining region:

![Diagram of part with highlighted machining region]

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

5. Click OK in the Chaining dialog box to return to the Chain Options dialog box.

6. Set the Machining region strategy to From outside.

![Screenshot of Chain Options dialog box with 'From outside' selected]

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

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

![Screenshot of Avoidance regions selection]

The Chaining dialog box displays.

8. Select 3D in the Chaining dialog box.

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

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

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

This will display the geometry we need to select. The geometry is a silhouette of the part. This allows you to machine around the part and clear out the stock, but without cutting into the part.
11. Select the chain shown below as an avoidance region:

![Chain Diagram]

12. Click OK in the Chaining dialog box to accept the chain and return to the Chain Options dialog box.
   Before you accept these chains in the Chain Options dialog box, you will first preview them to see if these are the chains you want.

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

![View Options]

14. In the Chain Options dialog box, click Preview chains.
Your part displays as shown in the following image:

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

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

15. Once you are satisfied, right-click in the graphics window and select Isometric (WCS).

16. Click OK in the Chain Options dialog box to accept these chains.

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

17. Select the Tool page.

18. Click Select library tool.

The Tool Selection dialog box displays.

The Tool List Filter dialog box displays.

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

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

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

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

Note: You can also double-click the tool to add it to the toolpath.
24. Select the Cut Parameters page.

- Approach distance to 9.5. This adds a specified distance to the beginning of the toolpath's first cut.
- Stepover to 50.0. Sets the distance between cutting passes in the X and Y axes.
- Minimum toolpath radius to 2.5. Mastercam uses this radius in combination with the Micro lift distance and Back feed rate parameters to calculate 3D arc moves between cutting passes.
- Micro lift distance to 0.0. Sets the distance the tool lifts off the part on back moves.
- Back feed rate to 0.0. Controls the speed of the back feed movement of the tool.
- Cut order optimization to None. Directs the toolpath to start at the most recently machined material when cutting.

25. Set the following parameters:

- Stock to leave on walls to 0.5. Leaves a defined amount of stock on the vertical drive geometry.
- Stock to leave on floors to 0.0. Leaves a defined amount of stock on horizontal drive geometry.

26. Select the Entry Motion page.
27. Set the following parameters:

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

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

28. Select the Linking Parameters page.
29. Set the following parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clearance</td>
<td>0.0</td>
<td>Absolute</td>
</tr>
<tr>
<td>Retract distance</td>
<td>5.0</td>
<td>Absolute</td>
</tr>
<tr>
<td>Feed plane distance</td>
<td>3.0</td>
<td>Absolute</td>
</tr>
<tr>
<td>Top of stock</td>
<td>1.0</td>
<td>Absolute</td>
</tr>
<tr>
<td>Depth</td>
<td>0.0</td>
<td>Absolute</td>
</tr>
</tbody>
</table>

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

30. Select Depth to return to the graphics window.

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

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

32. Click OK in the 2D High Speed Toolpath - Dynamic Mill dialog box to generate your toolpath.

33. The toolpath will display as shown below:

34. Save your part.
Exercise 4: Creating the second Dynamic Mill toolpath

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

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

![Toolpaths Manager screenshot](image)

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

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

![2D gallery screenshot](image)

The Chain Options dialog box displays.

3. Click Select under Machining regions.

![Chain Options dialog box](image)

The Chaining dialog box displays.
4. Select the geometry shown below as the machining region. Use the Cplane button to select the entire chain.

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

5. Click OK in the Chaining dialog box to return to the Chain Options dialog box.

6. Ensure that the Machining region strategy is set to From outside.

7. Click Select under Avoidance regions.

The Chaining dialog box displays.
8. Select the chain shown below as an avoidance region. Set the chaining to 3D.

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

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

10. Click OK in the Chain Options dialog box to accept these chains.

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

11. Select the Tool page.

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

14. Select Rest material.

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

15. Set the following parameters:

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

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

18. Select the Entry Motion page.
19. The parameters are also the same as the previous operation’s, so ensure that your toolpath matches the parameters shown below:

![Parameter Settings](image)

20. Select the Linking Parameters page.

![Linking Parameters](image)

21. Set the following parameters:

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

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

![Depth Input](image)

23. Select the arc shown below:

![Selected Arc](image)

You will need to rotate the part to select it.

24. You will automatically return to the **2D High Speed Toolpath - Dynamic Mill** dialog box. Your **Depth** should be -10.16.

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

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

27. Save your part.
Exercise 5: Verifying the toolpaths

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

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

   ![Image of Toolpaths Manager]

   This will select both Dynamic Mill toolpaths.

2. Select **Verify selected operations**.

   ![Image of Mastercam Simulator]

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

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

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

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

The Dynamic Mill toolpaths easily and quickly remove all of the stock on the outside and prepares the part for any other necessary operations. In the next lesson, you will create a Face toolpath to further machine this part.
CHAPTER 2
THE FACE TOOLPATH

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

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

Lesson Goals

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

Exercise 1: Creating the Face toolpath

1. Open the part file, FaceMill, provided with this tutorial or use your part file created from the Dynamic Mill lesson.

2. If you are using your part from the previous lesson, select **Toggle display on selected operations** on the Toolpaths Manager. This will hide the two Dynamic Mill toolpaths and keep the graphics view clean.

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

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

The **Chaining** dialog box displays.
5. Select **Solids** to activate solid selection.

![Chaining Dialog Box](image)

6. Activate **Face selection**.

   This allows you to select the face of a solid.

![Face Selection Icons](image)

7. Select the face shown below:

![3D Model](image)

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

   The **2D Toolpaths - Facing** dialog box displays.
9. Select the Tool page.

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

11. Select the Cut Parameters page.

12. Set the following parameters:

   - **Style**: Dynamic
   - **Tip comp**: Tip
   - **Roll cutter around corners**: Sharp
   - **Max. stepover**: 50.0%
   - **Across overlap**: 50.0%
   - **Along overlap**: 110.0%
   - **Approach distance**: 50.0%
   - **Exit distance**: 50.0%
   - **General start location**: Bottom left
   - **Climb**: On
   - **Reverse direction of last pass**: On
   - **Auto angle**: On
   - **Roughing angle**: 0.0
   - **Move between cuts**: 50.0
- **Style to Dynamic.** Creates a smooth controlled motion that cuts from the outside to the inside, maintaining a constant load on the tool with minimal entries and exits.

- **Stock to leave on floors** to 0.0. Sets the amount of stock to leave on the floors during machining.

- **Maximum stepover** to 50.0. Sets the distance between adjacent passes in the toolpath.

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

14. Set the following parameters:

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

Before clicking OK, you will first preview the toolpath. Previewing toolpaths allows you to view the toolpath before generating it so that you can make adjustments as necessary.
15. Select Preview toolpath on the 2D Toolpaths - Facing dialog box.

![2D Toolpaths - Facing dialog box](image)

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

![Toolpath on part](image)

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

18. Save your part.
Exercise 2: Verifying the toolpaths

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

1. **Select Toolpath Group-1 in the Toolpaths Manager.**
   
   This selects both Dynamic Mill toolpaths and the Facing toolpath.

2. **Select Verify selected operations.**
   
   Mastercam Simulator displays.
3. Right-click in the graphics view and select Fit and Isometric from the menu. Mastercam Simulator should match the image shown below.

![Mastercam Simulator](image)

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

![Toolpath Preview](image)

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.

![Move List](image)
6. Select **Color Loop, By Operation** on the **Verify** tab to help distinguish between each toolpath.

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

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

8. Save your part file.

Using the Dynamic Mill toolpaths and the Face toolpath, you have quickly cleared the part to prepare it for finishing operations. In the next lesson, you create two Dynamic Contour toolpaths.
CHAPTER 3
THE DYNAMIC CONTOUR TOOLPATH

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

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

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

Exercise 1: Setting up the stock
1. Open the part file, DynamicContour, provided with this tutorial.
2. Save the part as DynamicContour-xxx, replacing xxx with your initials.
4. Select Stock setup from the Toolpaths Manager.
5. Select Bounding box on the Stock Setup tab.
The **Bounding Box** function panel displays.

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

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

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

9. Click **OK** to accept the stock setup.
Exercise 2: Creating the first Dynamic Contour toolpath

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

   ![Dynamic Contour toolpath selection](image)

   The Chain Options dialog box displays.

2. Click Select under Machining regions.

   ![Chain Options dialog box](image)

   The Chaining dialog box displays.

3. Select Solids to activate solid chaining mode.

   ![Chaining dialog box](image)

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

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

   ![Chaining options](image)

   This allows you to select a closed chain on a reference face.
5. Select the edge shown below. You may need to zoom in.

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

6. Ensure that the face shown below is selected:

Select *Other face* if necessary to cycle through the possible reference faces.
7. Click OK to return to the Chaining dialog box. The chain will be displayed as shown below:

Reverse the chain if necessary.

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

9. Click OK in the Chain Options dialog box.

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

10. Select the Tool page.

11. Click Select library tool.

The Tool Selection dialog box displays.

12. Click Filter.

The Tool List Filter dialog box displays.

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

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

16. In the Tool Selection dialog box, select the **FLAT END MILL - 4** tool.

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

18. Select the **Cut Parameters** page.
19. Set the following parameters:

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

20. Select the Contour Wall page.

21. Set the following parameters:
- Radius of tool that shaped the stock to 8.0.
- Minimum toolpath radius that shaped the stock to 1.0.
- Stock thickness to 0.025.

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

22. Select the Linking Parameters page.

23. Set the following parameters:

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

![Depth selection](image)

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

25. Select the edge shown below:

![Selected edge](image)

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

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

27. Your toolpath displays as shown below:

![Toolpath display](image)

28. Save your part.
Exercise 3: Creating the second Dynamic Contour toolpath

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

   ![Dynamic Contour 2D Gallery](image)

   The Chain Options dialog box displays.

2. Click Select under Machining regions.

   ![Chain Options Dialog](image)

   The Chaining dialog box displays.

3. Activate Solids selection if necessary.

4. Set the chaining to Linked edges.

   ![Chaining Dialog](image)

   This allows you to select contiguous chaining of edges.
5. Zoom in using the mouse wheel and select the edge shown below.
6. The chain will display as shown below. Reverse the direction if needed, using **Reverse** on the **Chaining** dialog box.

![Image of chain display]

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

![Image of chain selection]

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

![Image of End chain button]

This ends the chain so that you can start a new one that is not linked to the previous chain.
9. Using the controls in the Chaining dialog box, select the second partial chain shown below.

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

11. Click OK in the Chain Options dialog box.

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

12. Select the Tool page.

13. Click Select library tool.

   The Tool Selection dialog box displays.

14. Select the FLAT END MILL - 8 tool.

15. Click OK in the Tool Selection dialog box.

16. Select the Cut Parameters page.
17. Set the following parameters:

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

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

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

20. Select the **Linking Parameters** page.
21. Set the following parameters:

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

The toolpath will display as shown below:

Exercise 4: Analyzing the toolpaths

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

1. Select Analyze Toolpath from the Home tab.
2. Hover over the first toolpath:

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

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

Exercise 5: Verifying the toolpaths

You will now verify both toolpaths.


This selects both toolpaths so that they can be verified at the same time.
2. Select Verify selected operations.

Mastercam Simulator displays.

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

4. View the Move List, which displays all pertinent information about the toolpaths being verified. By using two Dynamic Contour toolpaths, the total machining time is approximately one minute.
5. Press Play to preview the toolpath motion for both operations.

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

7. Save your part file.

In the next lesson you will create three Peel Mill toolpaths.
CHAPTER 4
THE PEEL MILL TOOLPATH

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

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

Lesson Goals

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

Exercise 1: Setting up the stock

1. Open the part file, PeelMill, provided with this tutorial.
2. Save the part as PeelMill-xxx, replacing xxx with your initials.
4. Select Stock setup from the Toolpaths Manager.
5. Select Bounding box.

The Machine Group Properties dialog box displays.
The **Bounding Box** function panel displays.

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

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

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

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

**Exercise 2: Creating the first Peel Mill toolpath**

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

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

![Chaining Dialog Box](image)

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

![Face and Edge Options](image)

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

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

![Translucency Option](image)

This toggles the translucency of the current model and will help with selection for this first toolpath.
5. Select the two edges shown below. Use Reverse if necessary to ensure both chains go in the same direction.

6. Toggle Translucency to off.

7. Click OK in the Chaining dialog box.

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

8. Select the Tool page.

9. Click Select library tool.

   The Tool Selection dialog box displays.
10. Click Filter.

![Filter dialog box]

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.

![Tool list]

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

15. Select the Cut Parameters page.

![Cut Parameters]

- Depth Cuts
- Finish Passes
- Break Through
16. Set the following parameters:

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

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

18. Set the following parameters:

- **Set Retract** to 40.0.
- **Set Feed plane** to 40.0.
19. Click **Top of stock** to return to the graphics window to select the top of the stock of the part.

   ![Top of stock setting]

20. Select the point shown below:

   ![Point selection]

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

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

   The toolpath will display as shown below:

   ![Toolpath display]

22. Save your file.
Exercise 3: Copying the Peel Mill toolpath

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

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

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

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

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

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

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

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

Exercise 4: Re-chaining the copied Peel Mill operations

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

1. Select Geometry under the second Peel Mill toolpath.

The Chain Manager dialog box displays.

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

The Chaining dialog box displays.

3. Select the chains shown below. Ensure that the chains are both going in the same direction. You may need to rotate the part or select Translucency on the View tab to select them.
By doing this, you only need to select the chains for the toolpath without having to go through creating the entire toolpath again.

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

5. Click OK in the Chain Manager dialog box.

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

6. Select Geometry under the third Peel Mill toolpath.

The Chain Manager dialog box displays.

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

The Chaining dialog box displays.

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

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

10. Click OK in the Chain Manager dialog box.
11. Select Regenerate all dirty operations in the Toolpaths Manager.

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

12. Save your file.
Exercise 5: Verifying the toolpaths

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

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

Mastercam Simulator displays.

3. Right-click in the graphics view and select Fit and Isometric, if necessary, so that the display matches the image below:
4. View the Move List to view the total run time of the toolpaths.

![Image of Move List showing total time]

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

![Image of toolpath preview]

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

7. Save your part file.

When creating similar toolpaths, copying and pasting saves time. In the next lesson you create two Dynamic OptiRough operations and a stock model operation.
CHAPTER 5
THE DYNAMIC OPTIROUGH TOOLPATH

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

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

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

Lesson Goals

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

Exercise 1: Setting up the stock

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

1. Open the part file, DynamicOptiRough, provided with this tutorial.
2. Save the part as DynamicOptiRough-xxx, replacing xxx with your initials.
4. Select the Levels Manager, located in the same area as the Toolpaths Manager. If it is not displayed, select Levels from the View tab to toggle the display.

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

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

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

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

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

The Machine Group Properties dialog box displays.

8. Select *Bounding box*.

The Bounding Box function panel displays.
9. Window select all entities to create a bounding box around the entire part.


11. Set the following parameters in the Bounding Box function panel:

- Set Shape to Cylindrical.
- Set Radius to 150.
- Set Axis to Z.
- Set Height to 102.00. This will add a small amount of stock at the top of the part.

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

![Image showing Create Geometry options](image)

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

13. Click OK to create the boundary.

14. Click OK in the Machine Group Properties dialog box to accept the stock setup.

The stock model displays as a dashed outline in red. The geometry that was created with the Bounding Box also displays.
Exercise 2: Creating the first Dynamic OptiRough toolpath

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

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

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

2. Select the Machining group, and then click Select entities under Machining Geometry.
3. Use window select or any applicable keyboard shortcuts to select the part body as the drive surfaces. This includes the blue caps, as shown below.

4. Press [Enter] or click End Selection to accept these surfaces. You return to the Model Geometry page.

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

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

6. Select the Toolpath Control page.
7. Click the **Boundary chains** selection arrow.

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

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

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

10. Set the containment boundary **Strategy** to **From outside**.

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

![Tool path type selection](image)

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

The Tool Selection dialog box displays.

13. **Select Filter.**

![Filter selection](image)

The Tool List Filter dialog box displays.

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

15. **Select the Endmill3 Bull filter, and deselect any other tool filter. This ensures that you are only selecting bull end mill tools.**

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

You return to the Tool Selection dialog box.

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

![Tool list](image)

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

19. **Select the Cut Parameters page.**

![Cut parameters](image)
20. Set the following parameters:

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

- **Stepover** to 30.0%. Sets the distance between cutting passes in the X and Y axes.

- **Stepdown** to 150.0%. Determines the Z spacing between adjacent cutting passes.

- Select the **Stepup** checkbox and enter 15.0. Mastercam calculates and adds stepup cutting passes (+Z) to the toolpath.

- **Minimum toolpath radius** to 10.0. Sets the minimum toolpath radius to create in the operation.

- **Micro lift distance** to 0.0.

- **Back feed rate** to 100.0. Controls the speed of the backfeed movement of the tool.

- **% of tool diameter** to 10000.0.

21. Select the Transitions page.
22. Set the following parameters:

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

23. Select the **Linking Parameters** page.
24. Set the following parameters:

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

25. Click OK to close the Surface High Speed Toolpaths - Dynamic OptiRough dialog box and generate your toolpath.
26. Your toolpath should display as follows:

![Toolpath Display](image)

27. Save your part file.

**Exercise 3: Creating the Stock Model operation**

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

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

![Toolpaths Manager](image)

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

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

![Toolpaths Contextual Tab](image)
The **Stock model** dialog box displays.

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

   ![Stock model dialog box](image)

   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**.

   ![Color](image)

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

5. **Select Stock Setup**.

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

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

   ![Stock Definition page](image)

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

   ![Surface High Speed](image)

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

8. **Click OK to create the stock model operation. This may take some time to calculate.**
9. Your stock model displays as shown below:

![Stock Model Diagram]

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

**Exercise 4: Creating the second Dynamic OptiRough toolpath**

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

2. Select Toggle display on selected operations. This hides all selected toolpaths to keep the graphics window clean.
3. Select **OptiRough** from the 3D gallery on the **Mill Toolpaths** contextual tab.

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

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

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

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

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

The **Surface High Speed Toolpaths - Dynamic OptiRough** dialog box displays.
8. Select the Toolpath Control page.

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

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

11. Click OK in the Chaining dialog box to return to the Toolpath Control page.

12. Set the following parameters to control the tool's position around the boundary of your part.
- Set **Compensate to** to **Outside**. This sets the outer edge of the tool to be bound to the containment boundary.
- Set **Offset distance** to 10.0. This option adjusts the inside or outside tool containment boundary.

13. **Select the Tool page.**

14. **Click Select library tool.**

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

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

17. **Select the Stock page.**

18. **Select Rest material.**

When you enable this page, Mastercam calculates the cutting passes based on the remaining stock.
19. Set the following parameters:

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

20. Select the **Cut Parameters** page.
21. Set the following parameters:

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

22. Select the Transitions page.
23. **Set the Helix radius to 5.5.**

   ![Helix radius setting](image)

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

25. **Select Toggle display on selected operations to display toolpaths again, and select the second Dynamic OptiRough toolpath to view the newly created toolpath.**

   ![Dynamic OptiRough toolpath](image)

26. **Save your part file.**
Exercise 5: Verifying the stock model and toolpaths

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

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

![Toolpaths Manager](image)

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

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

4. Click the Play button or press [R] to verify the toolpaths.
5. View the Move List to view the total run time of the toolpaths.

6. Once you are satisfied with the verification, close Mastercam Simulator.

7. Save your part.
CONCLUSION

Congratulations! You have completed the Mastercam Dynamic Milling Tutorial! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select Help, Tutorials from the File tab.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- **Mastercam Documentation**—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- **Mastercam Help**—Access Mastercam Help by selecting Help, Contents from Mastercam's File tab or by pressing [Alt+H] on your keyboard.
- **Mastercam Reseller**—Your local Mastercam Reseller can help with most questions about Mastercam.
- **Technical Support**—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- **Mastercam Tutorials**—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select Help, Tutorials from Mastercam's File tab to see the latest publications.
- **Mastercam University**—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.
- **Online Communities**—You can find a wealth of information 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 knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.
ATTENTION! UPDATES MAY BE AVAILABLE. PLEASE REFER TO MASTERCAM.COM/SUPPORT FOR THE LATEST DOWNLOADS.