Dynamic Milling

March 2015

Mastercam X9

Mastercam® X9 Dynamic Milling

Date: March 2015

Software: Mastercam X9

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 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.

Contents

Introduction

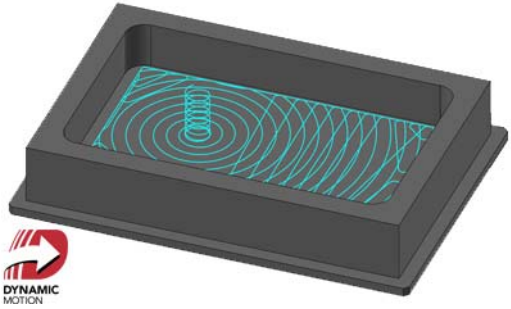
Mastercam's 2D high speed dynamic milling toolpaths utilize the entire flute length of their cutting tools to achieve great efficiency in milling. They are designed to maximize material removal while minimizing tool wear.

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

- Tool burial avoidance
- Minimum heat buildup
- Better chip evacuation

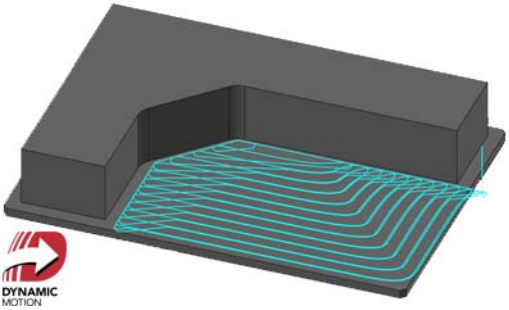
This tutorial introduces you to all of the following intelligent application-specific 2D high speed dynamic milling toolpaths available in Mastercam and compares them against their standard counterparts.

Dynamic Mill (Stay Inside Strategy)—Machines pockets using one or more chains to drive the toolpath. This strategy is enabled by default.

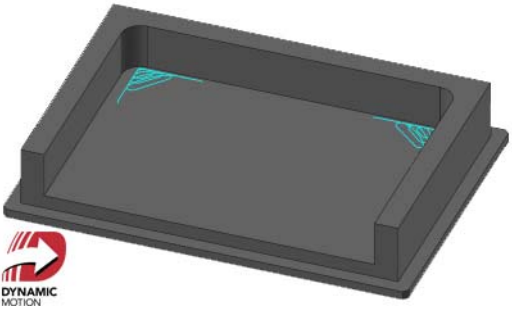


Dynamic Mill (From Outside Strategy)—Machines open pocket shapes or standing core shapes using the outmost chain as the stock boundary. The tool moves freely

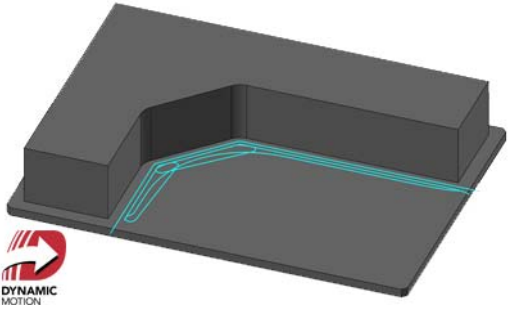
outside of this area.The toolpath starts from the outside and works its way towards the inner boundary.



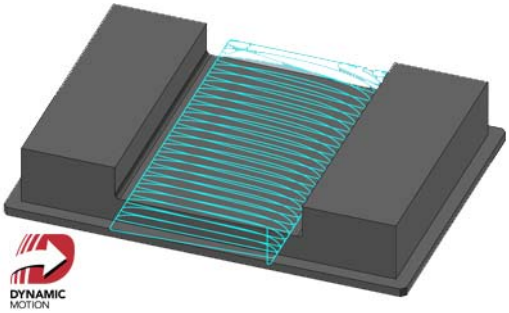
Dynamic Mill (Rest Mill Strategy) —Targets material left from previous operations. Dynamic motion is used to rapidly remove material left by previous operations or by a specified tool size. Use this strategy when the roughing tools leave material in corners or narrow part areas.



Dynamic Contour— Mills material off walls supporting closed or open chains. The tool motion converts to dynamic in areas where the tool would bind or encounter too much material. Use Dynamic Contour on walls or contours that contain small radii.



Peel Mill (Dynamic Peel Cutting style) —Allows for efficient milling between two selected contours or along a single contour.



Tutorial Goals

- Learn the benefits and uses of the Dynamic toolpath types
- Understand chaining and entry options specific to Dynamic toolpaths
- Create basic Dynamic toolpaths
- Compare and contrast the Dynamic toolpaths with their non-Dynamic counterparts.

IMPORTANT: 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 the exercise results.

General Tutorial Requirements

All Mastercam tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition (HLE). The Demo/HLE file format (`EMCX-9`) is different from Mastercam (`MCX-9`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- 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 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 English configuration. The tutorial provides instructions for loading the appropriate configuration file.

LESSON 1
Dynamic Milling Overview

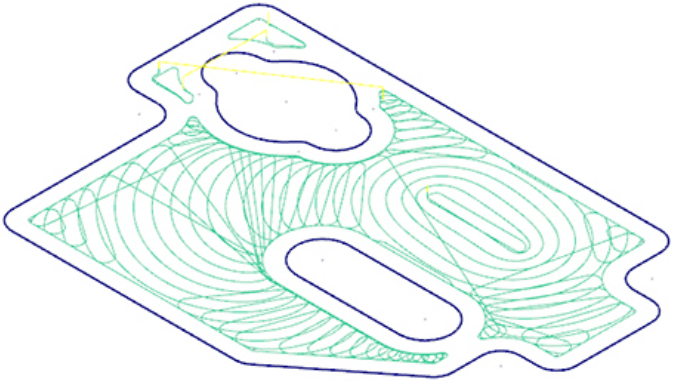
Mastercam's 2D high speed toolpaths (HST) contain three types that utilize dynamic motion:

- Dynamic Mill
- Dynamic Contour
- Peel Mill (Dynamic Peel Cutting style)

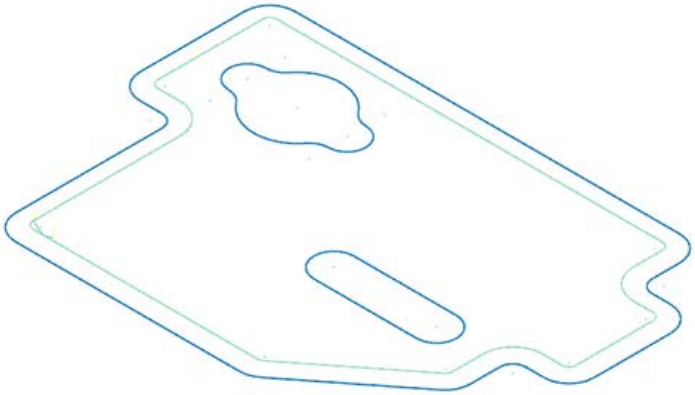
Dynamic toolpaths utilize as much of the tool's flute length as possible, while minimizing tool burial. This method enables the toolpath to rapidly and efficiently remove the maximum amount of material and extend tool life at the same time.

Chain Selection for Dynamic Toolpaths

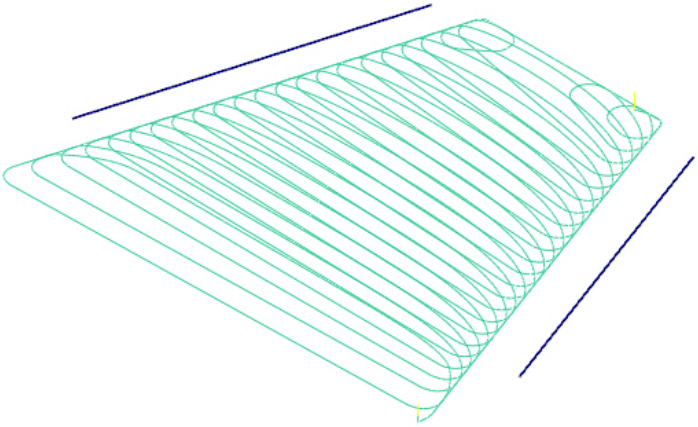
The Dynamic toolpaths require a unique way of understanding chain selection. Dynamic Mill requires at least one Machining region defined, which can be either a closed chain or open chain. Any remaining geometry can be defined as an Avoidance region, Containment region, or Air region.



Dynamic Contour does not require a closed chain to calculate a toolpath. All geometry selected will be cut by the tool motion.



Peel Mill, when using the Dynamic Peel cutting style requires at least one open chain, and no closed chains.

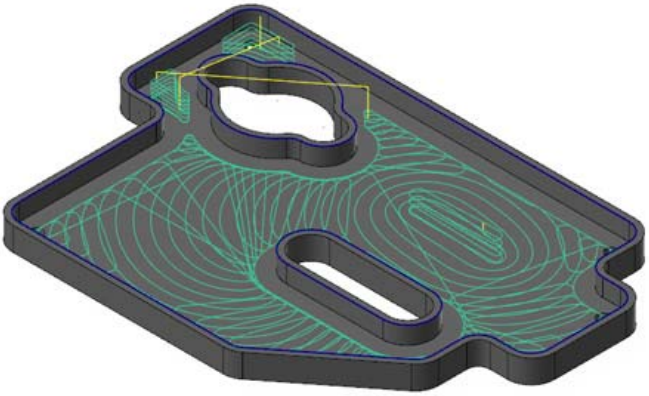


Chaining options are explored further throughout subsequent lessons.

Dynamic Mill (Stay Inside strategy)

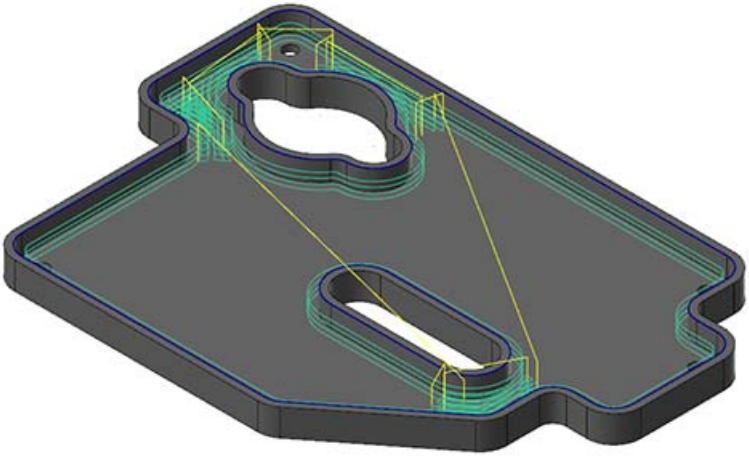
Dynamic Mill with the **Machining region strategy** set to **Stay Inside** is a pocket toolpath that utilizes dynamic motion. A boundary is selected; islands, bosses, and geometry to avoid are added; tooling, parameters, entry method, and linking values are

entered; and the toolpath is generated. Material is removed from the inside to the outside in a highly efficient manner.



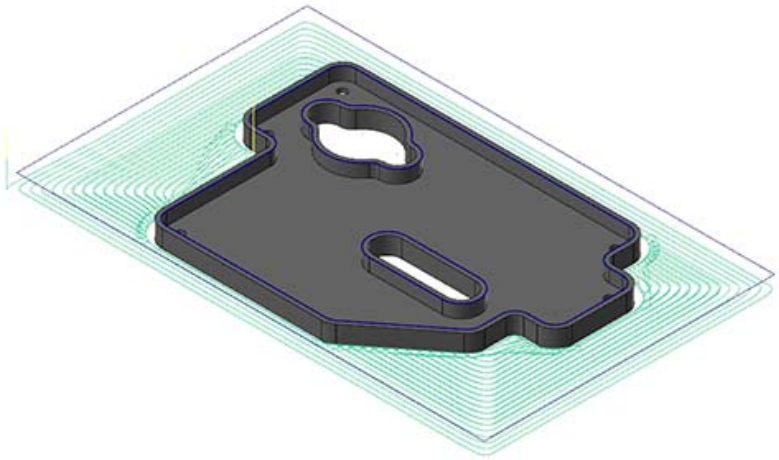
Dynamic Mill (Rest Material strategy)

Dynamic Mill with the Rest Mill strategy enabled specializes in removing material left by previous operations using dynamic motion. The material to remove is calculated by previous operations or previous roughing tools. Only unmachined areas are processed for a Dynamic Mill operation.



Dynamic Mill (From Outside Strategy)

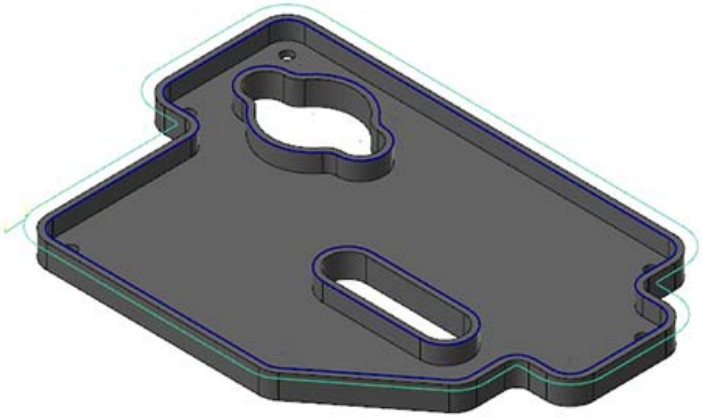
Dynamic Mill with the **Machining region strategy** set to **From Outside** is a facing operation that has island avoidance capabilities. The toolpath is created using dynamic motion for maximum efficiency. The tool removes material from the outside to the inside. This movement creates a toolpath that is allowed to move outside the selected boundary.



Dynamic Contour

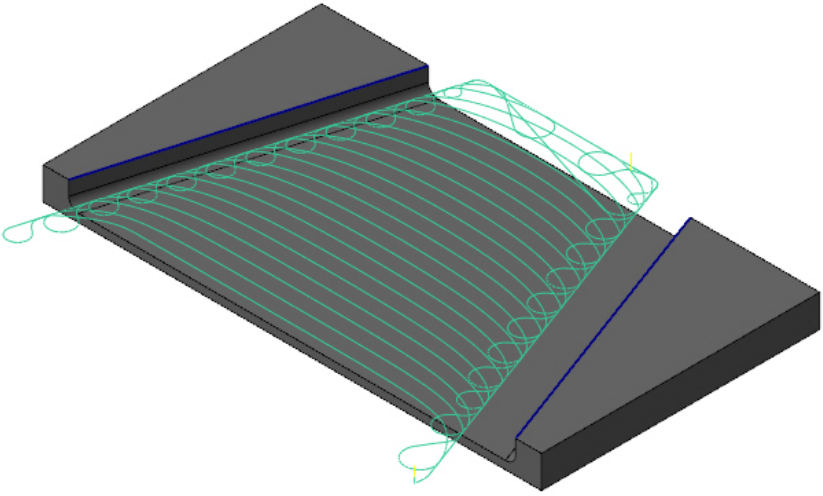
Dynamic Contour creates a familiar contour operation using dynamic motion and specialized options found in the dynamic toolpath parameters. The dynamic motion prevents tool burial and binding in small radii corners. In addition, parameters are

provided for you to instruct Mastercam on the material left on the walls to be machined.



Peel Mill (Dynamic Peel Cutting Strategy)

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 single chains, you define the width of the cut. Otherwise, the width is defined by the area between two contours.



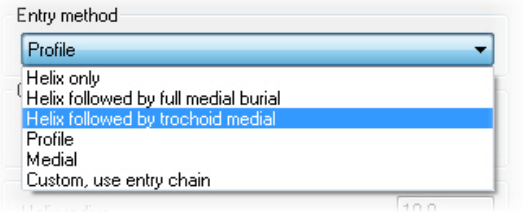
Dynamic Milling Parameters

Several parameter options separate the Dynamic operations from their standard 2D HST counterparts.

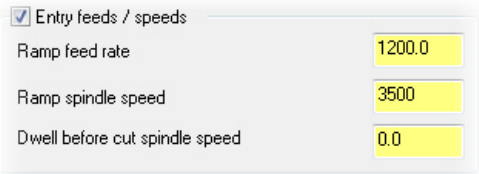
- Micro lift
- Entry methods
- Entry feeds/speeds
- Contour wall

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 micro lift include chip clearing and avoiding excess tool heat. The feedrate is controllable for the back move as well.

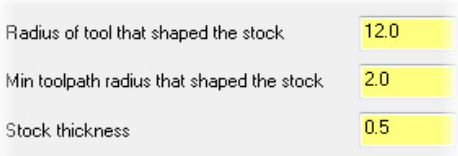
Entry methods provide several ways to enter the material for a Dynamic Mill operation. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's **Help** contains detailed information on each of the methods available.



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.



Dynamic Milling toolpaths create highly efficient motion based on selected chains. They are part of the 2D HST family of toolpaths with many similarities, but with the stated enhancements. They are an incremental step in the evolution of high speed machining options provided by Mastercam. The high speed toolpaths themselves are a leap ahead of the standard pocket, face, and contour operations.

Creating and reviewing the motion of dynamic toolpaths is the best way to learn about the benefits and uses. Continue on to Lesson 2 to learn about chain and entry selection and the effect it has on your toolpath.

LESSON 2
Chain and Entry Selection

Lesson 1 provided a brief overview of Mastercam's Dynamic toolpath options. This lesson focuses on chain and entry selection using simplified examples. The techniques and behavior demonstrated are easily translated to the more complicated and intricate parts you deal with on a daily basis. The geometry, selections, and parameters are simplified to ensure the methods and results are the focal points.

Detailed steps on such actions as selecting a machine definition, changing the graphics view or construction plane, making levels visible, and backplotting will not be provided.

Lesson Goals

- Configure Mastercam for metric operation
- Create a Dynamic Mill toolpath
- Add geometry, modify parameters, and review the tool motion

Exercise 1: Preparing Mastercam for Toolpath Creation

1 Start Mastercam using your preferred method:

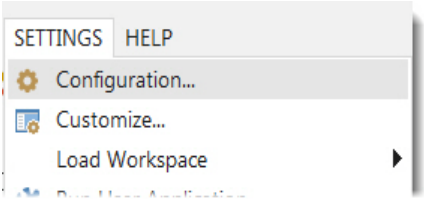
- Double-click Mastercam's desktop icon.

  Or

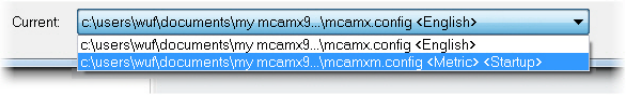
- Launch Mastercam from the Windows Start menu.

2 Select the default metric configuration file:

a Select **Settings, Configuration** from Mastercam's menu.

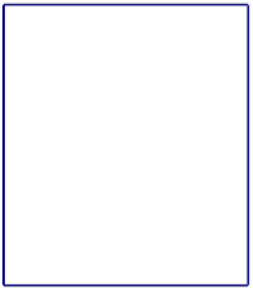


b Choose **...\mcamxm.config <Metric>** from the **Current** drop-down list.



c Click **OK**.

3 Open the part file `Chaining_Examples.MCX-9`, which was provided with the tutorial.



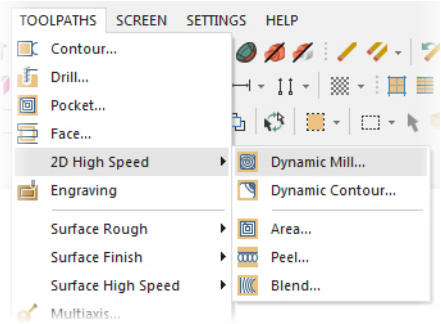
4 If necessary, fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

5 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Creating a Dynamic Toolpath

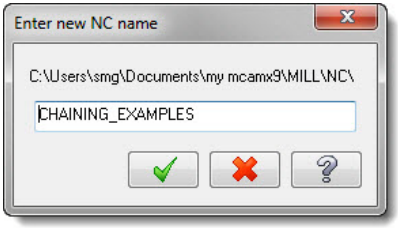
In this exercise, the Dynamic Mill toolpath is used to demonstrate chaining and entry options. Some of these options also apply to Dynamic Contour and Peel Mill.

1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Mill**.



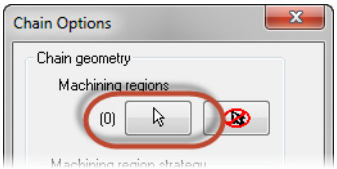
2 Click **OK** if prompted to enter a new NC file name.

The Chain Options dialog box displays.

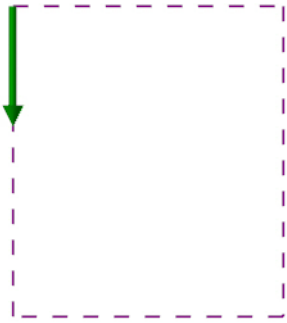


3 Click **Select** under Machining regions. The Chaining dialog box displays.

Machining regions are areas to be machined.



4 Chain the rectangle in a counterclockwise direction.

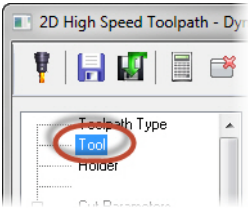


NOTE: To change the direction of a chain, select the Flip direction button on the Chaining dialog box.

5 Click **OK** on the Chaining dialog box. Click **OK** in the Chain Options dialog box.

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

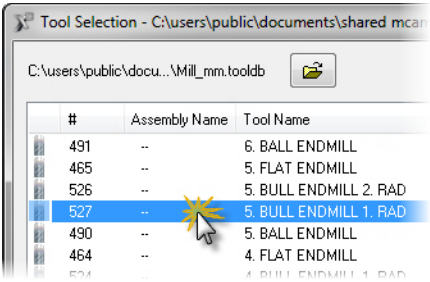
6 Select the **Tool** page.



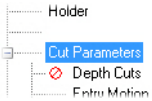
7 Click the **Select library tool** button.

The Tool Selection dialog box displays.

8 Select the **5mm bull endmill** with a 1.0mm radius and click **OK**.

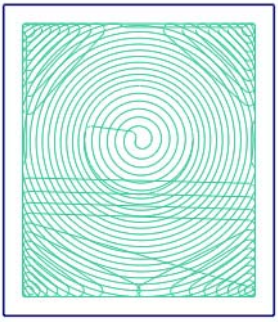


9 Select the **Cut Parameters** page.



These parameters will be used in the next few lessons to help create a more efficient toolpath. For now, make no changes.

10 Click **OK** to generate the Dynamic Mill toolpath on the selected geometry. Observe the toolpath pattern. Backplot the operation if necessary.



11 Save your part file.

The toolpath is created inside of the selected Machining regions. With the **Machining region strategy** set to **Stay Inside**, the toolpath does not venture outside of the selected chain.

TIP: Use **Analyze, Volume/Area, 2D Area** on your selected chains if the toolpath is not being created as expected. The cause will invariably be an incorrect assumption about the enclosed 2D Area of your geometry.

12 From the Toolpaths Manager, press [**T**] to toggle off the toolpath display in the graphics window.

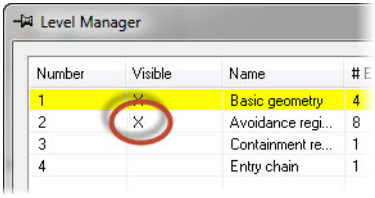
Hiding the operation's toolpath makes it easier to select geometry in the next exercise.

Exercise 3: Adding Avoidance Regions to an Operation

1 Select the **Level Manager** from the Status Bar. The Level Manager dialog box displays.

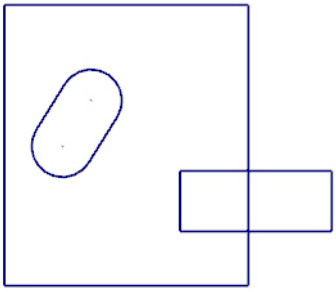
2 In the Visible column for level **2**, click once to turn on the display of the **Avoidance regions** geometry.

The X in the **Visible** column indicates that the level is displayed in the graphics window.

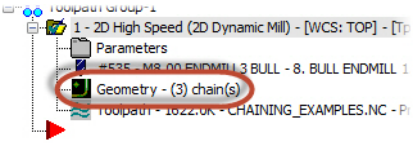


3 Click **OK** on the Level Manager.

Level **2:Avoidance wireframe** is now visible.



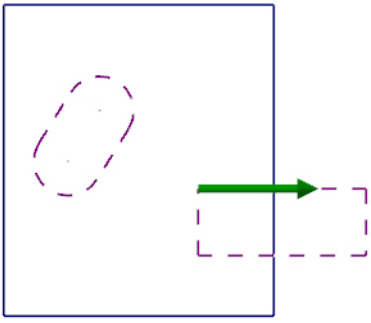
4 Click **Geometry** under operation 1. The Chain Options dialog box opens.



5 Click **Select** under Avoidance regions.

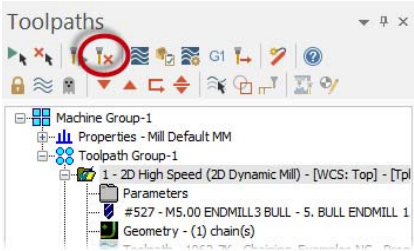
The Chaining dialog box opens.

6 Select the oval and small rectangle as indicated. Chain direction does not matter.

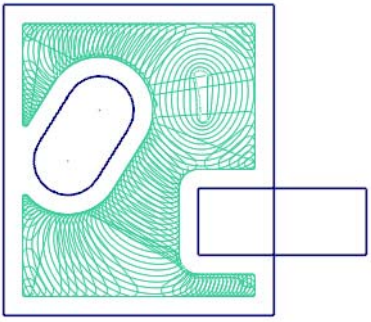


7 Click **OK** in the Chaining and Chain Options dialog boxes.

8 Click **Regenerate all dirty operations** in the Toolpaths Manager.



9 Press [**T**] to toggle on the toolpath display in the graphics window.



NOTE: Based on your defaults, your toolpath may differ slightly from the one in the image.

10 Backplot the operation if necessary.

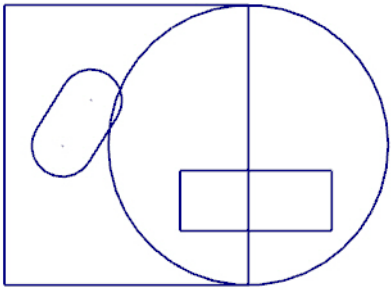
11 Save your part file.

The toolpath avoids the added chains since the encompassed 2D area of each is less than the outer chain. Of particular interest is the small rectangle that overlaps the larger one. A standard pocket toolpath would require you to trim geometry and rechain the boundary. Dynamic Mill gives you the ability to add and avoid geometry simply by adding a chain.

For instance, you draw a simple rectangle to represent a clamp that must be used to hold your part. Sketch a rectangle, add it to the operation's chains, regenerate the operation, and you have the toolpath needed. There is no need to alter the original part geometry.

Exercise 4: Adding Containment Regions to an Operation

1 Use the Level Manager to make level **3:Containment regions** visible, as shown in the previous exercise.

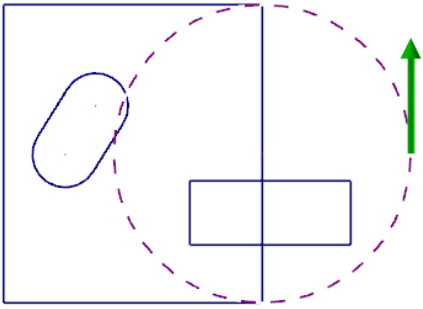


2 Press [**T**] to toggle the toolpath display off.

3 Click **Geometry** under operation 1. The Chain Options dialog box displays.

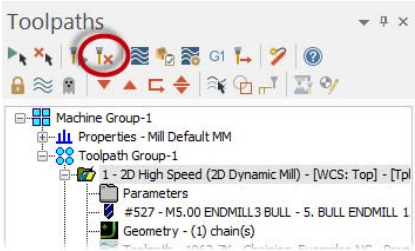
4 Click **Select** under Containment regions. The Chaining dialog box displays.

5 Select the circle as indicated. Chain direction does not matter.

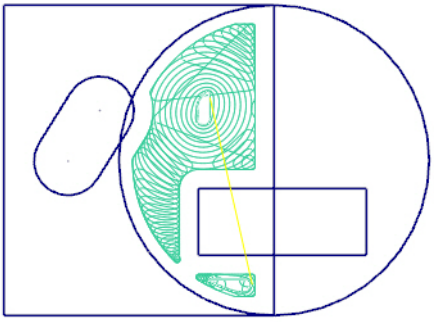


6 Click **OK** in the Chaining and Chain Options dialog boxes.

7 Click **Regenerate all dirty operations** in the Toolpaths Manager.



8 Press [**T**] to toggle on the toolpath display in the graphics window.



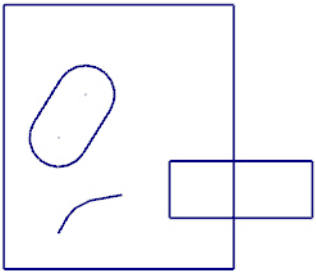
9 Backplot the operation if necessary.

10 Save your part file.

Containment regions are areas that the tool cannot move outside of. The toolpath is now contained inside the circle and also avoids the geometry previously selected as Avoidance regions.

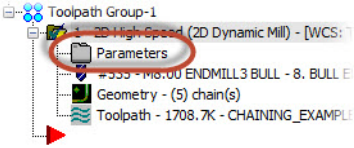
Exercise 5: Selecting an Entry Method

1 Use the Level Manager to make level **4:Entry chain** visible, level **3:Containment regions** invisible and toggle the toolpath display off.



2 Click **Parameters** under operation 1.

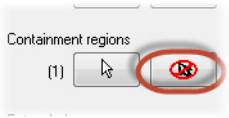
The 2D High Speed Toolpath - Dynamic Mill dialog box opens.



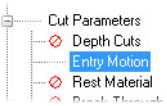
3 Select the **Toolpath Type** page.

4 Under Containment regions, click **Deselect**.

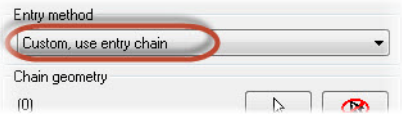
This will remove all Containment regions from the toolpath.



5 Select the **Entry Motion** page.

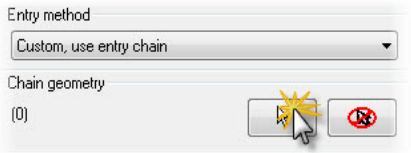


6 Select **Custom, use entry chain** from the drop-down list in the Entry method group box.



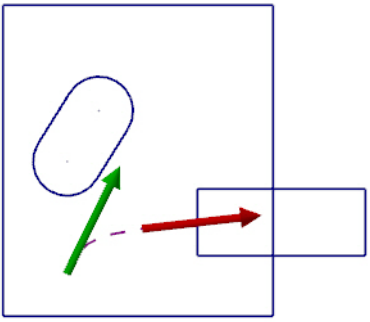
7 Click **Select** in the Chain geometry group box.

The Chain Manager dialog box displays.



8 Right-click in the Chain Manager dialog box and select **Add chain**.

9 Select the open chain as indicated.

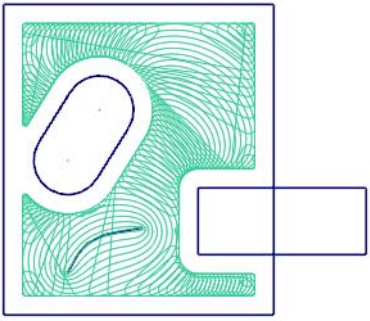


10 Click **OK** in both the Chaining and Chain Manager dialog boxes, as well as the 2D High Speed Toolpath - Dynamic Mill dialog box.

11 Click **Regenerate all dirty operations** in the Toolpaths Manager.

12 Press [**T**] to toggle the toolpath display in the graphics window.

13 Backplot the operation if necessary. Pay special attention to how the tool enters the material.

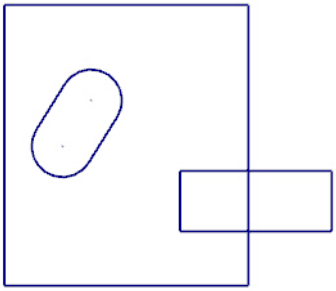


14 Save your part file.

Entry methods provide numerous ways to begin removing material from your part. These methods will be reviewed in subsequent lessons. Consult Mastercam's *Help* if you wish to learn more about the entry options for dynamic toolpaths at this time.

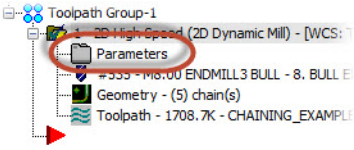
Exercise 6: Adding Air regions to an Operation

1 Use the Level Manager to make level **4:Entry chain** invisible and toggle off the toolpath display.



2 Click **Parameters** under operation 1.

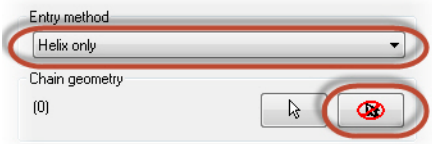
The 2D High Speed - Dynamic Mill Toolpath dialog box displays.



3 Select the **Entry Motion** page.

4 Set the Entry method to **Helix only** and deselect the Chain geometry.

This will remove the Entry chain from the toolpath.



5 Select the **Toolpath Type** page.

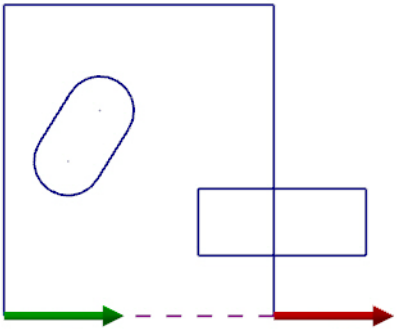
6 Click **Select** under Air regions.

The Chaining dialog box displays.

7 In the Chaining dialog box, select **Partial**.

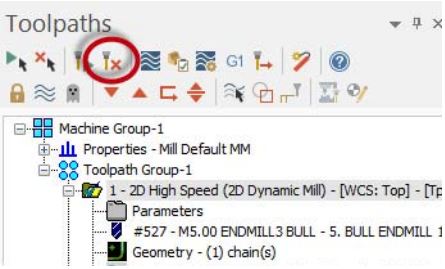
This allows you to create an open chain.

8 Select the line as indicated. Chain direction does not matter.

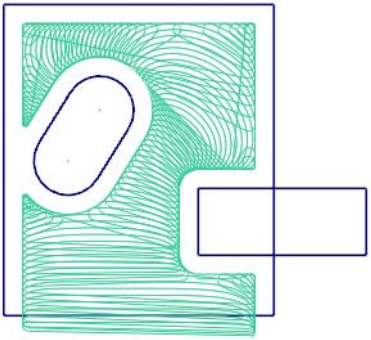


9 Click **OK** in the Chaining dialog box and in the 2D High Speed - Dynamic Mill Toolpath Dialog box.

10 Click **Regenerate all dirty operations** in the Toolpaths Manager.



11 Press [**T**] to toggle on the toolpath display in the graphics window.



12 Backplot the operation if necessary.

13 Save your part file.

Notice that the toolpath now enters from the selected Air region chain. By partially chaining that line segment, it tells Mastercam that there is no material there. This allows the tool to travel through it when machining.

At this point you can see how quickly and easily you make significant changes to your toolpath motion. Take some time to change entry methods, add and remove chains from your toolpath, and observe how the changes affect the backplot. When ready, continue on to the next lessons for a more in depth look at Dynamic Mill and its machining strategies.

LESSON 3
Dynamic Mill Toolpath (Stay Inside Strategy)

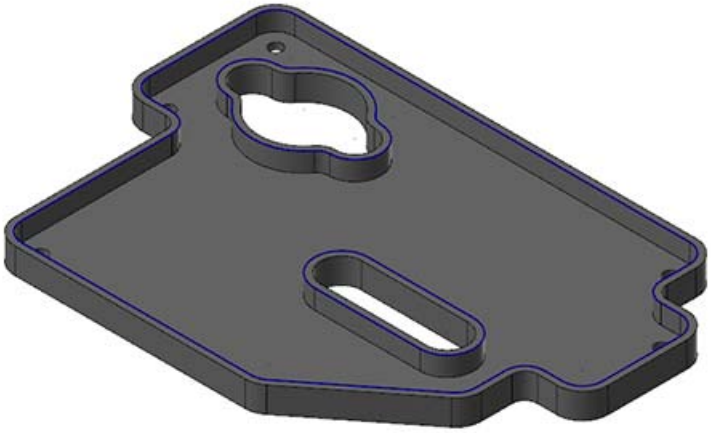
Lesson 2 exposed how chaining is handled for Dynamic Mill toolpaths, in addition to a brief glimpse of the entry methods. This lesson centers around creating a Dynamic Mill toolpath, with the **Machining region strategy** set to **Stay Inside**. A pocket toolpath is created for comparison with the dynamic toolpath.

Lesson Goals

- Create a Dynamic Mill toolpath with the Stay Inside strategy
- Create a Pocket toolpath
- Compare the two toolpaths

Exercise 1: Preparing Mastercam for Toolpath Creation

1 Open the part file `Dynamic_Mill_StayInside.MCX-9`, which was provided with the tutorial.



2 If necessary, select the default Mill metric machine definition.

3 Select **Isometric (WCS)** from the Gview menu.

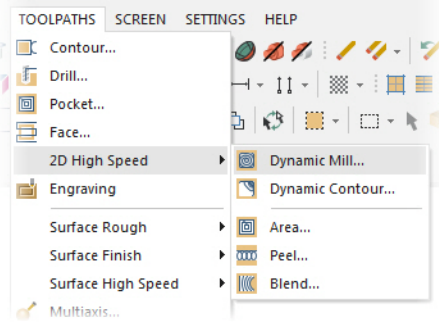
4 Shade the part, if necessary, using [**Alt+S**].

5 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

6 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

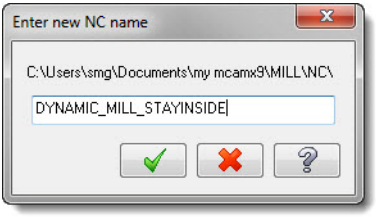
Exercise 2: Creating a Dynamic Mill Toolpath

1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Mill**.



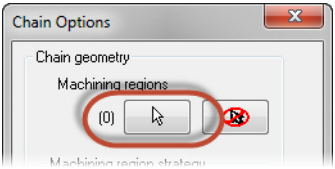
2 Click **OK** if prompted to enter a new NC file name.

The Chain Options dialog box displays.

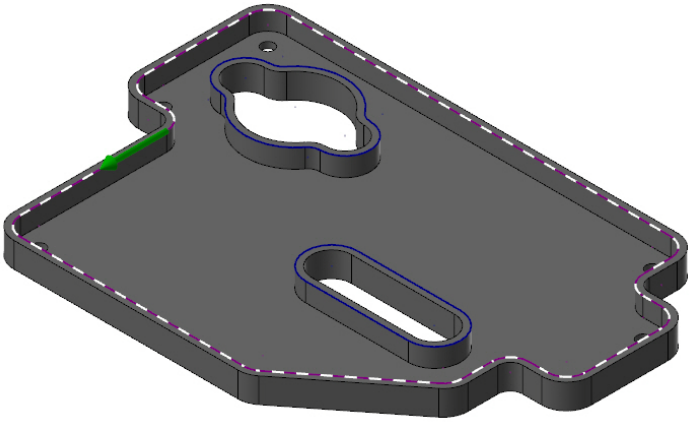


3 Click **Select** under Machining regions.

The Chaining dialog box displays.



4 Chain the wireframe geometry around the part as indicated.

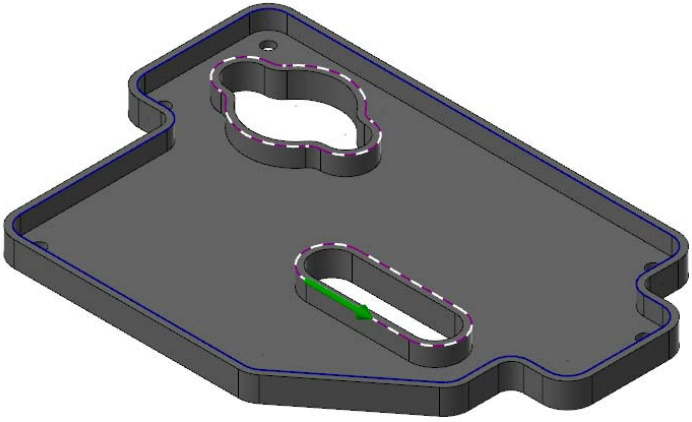


5 Click **OK** to close the Chaining dialog box and return to the Chain Options dialog box.

6 Click **Select** under Avoidance regions.

The Chaining dialog box displays.

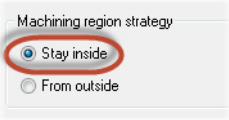
7 Chain the wireframe geometry as indicated.



8 Click OK to close the Chaining dialog box.

9 Ensure that the **Machining region strategy** is set to **Stay inside**.

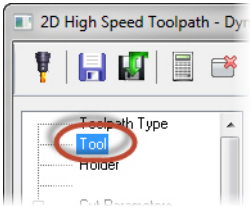
This strategy is enabled by default and keeps the toolpath inside the selected machining region.



10 Click **OK** to close the Chain Options dialog box.

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

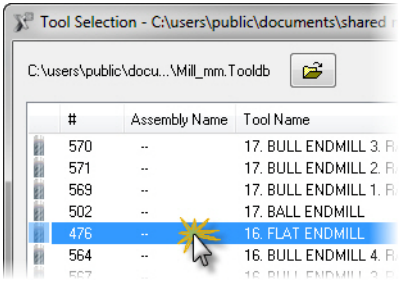
11 Select the **Tool** page.



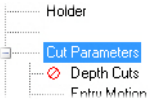
12 Click the **Select library tool** button.

The Tool Selection dialog box displays.

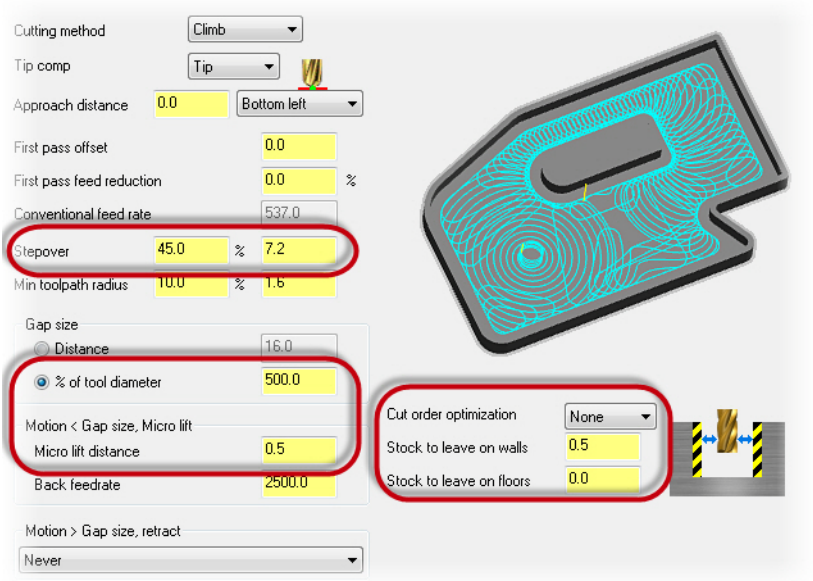
13 Select the **16mm flat endmill** and click **OK**.



14 Select the **Cut Parameters** page.



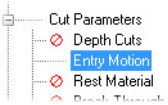
15 Set the parameters as shown.



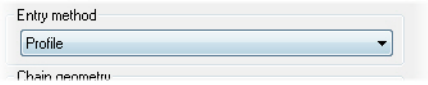
Use micro lift distance to raise the tool above the floor when making a connecting move. The lift creates a period of time where heat is not being added to the tool or the part, extending tool life.

TIP: Click **Help** to learn more about the individual fields. Information is provided to explain how the available options affect your toolpath.

16 Select the **Entry Motion** page.

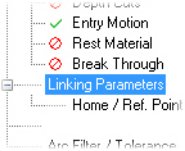


17 Select **Profile** from the Entry method drop-down list.

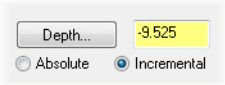


18 Enter **1.0** mm for **Z clearance** and **2.0** degrees for the **Plunge angle**.

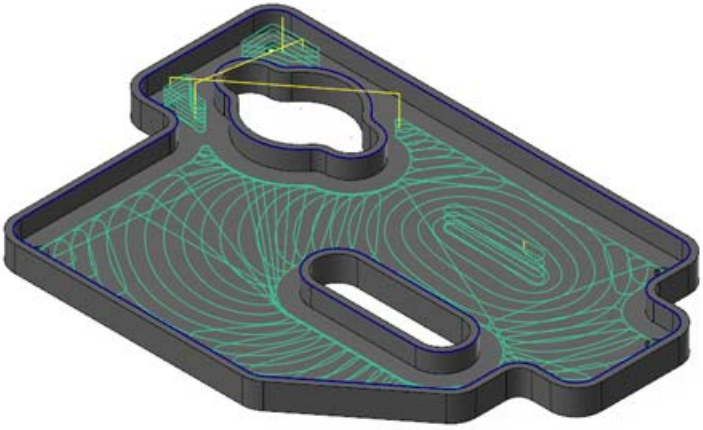
19 Select the **Linking Parameters** page.



20 Enter **-9.525** mm for the **Depth**. Ensure that it is set to **Incremental**.



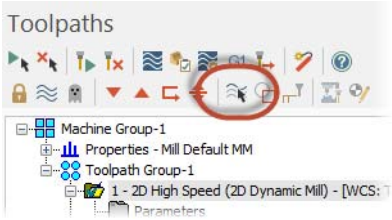
21 Click **OK** to generate the Dynamic Mill toolpath.



With the **Machining region strategy** set to **Stay inside**, the toolpath does not leave the selected Machining region. The toolpath also avoids the two selected chains.

22 Click **Only display selected toolpaths** in the Toolpaths Manager.

Clicking the button does not affect the current display. The graphics view will be less cluttered when additional operations are added later in this lesson.



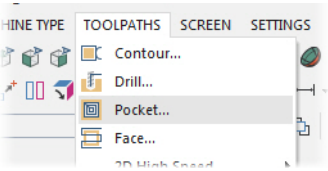
23 Save your part file.

Dynamic Mill is able to cut the material at full depth using as much of the tool's flute length as necessary. Depth cuts are available for pocket depths and materials that prevent machining at full depth with one pass.

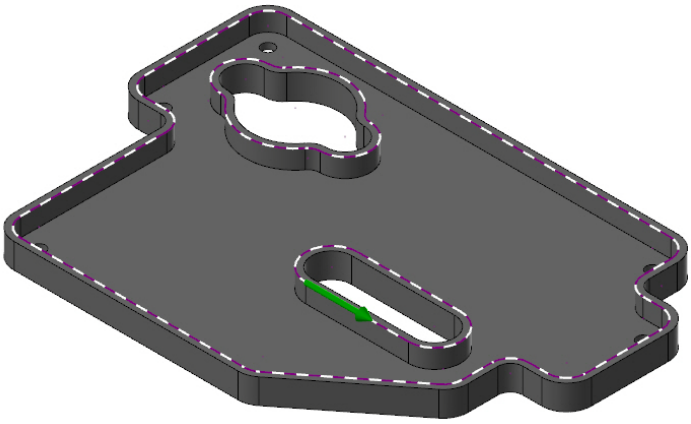
Exercise 3: Creating a Pocket Toolpath

1 From the Mastercam menu, choose **Toolpaths, Pocket**.

The Chaining dialog box displays.



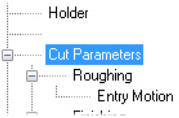
2 Chain the wireframe geometry around the part and features as indicated:



3 Click **OK** in the Chaining dialog box.

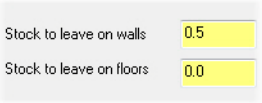
The 2D Toolpaths - Pocket dialog box displays.

4 Select the **Cut Parameters** page.



Pocket is automatically selected on the **Toolpath Type** page as well as the **16mm flat endmill** on the **Tool** page. Select those pages if you wish to verify the selections.

5 Enter **0.5** mm for the **Stock to leave on walls**, and **0.0** mm for **Stock to leave on floors**.

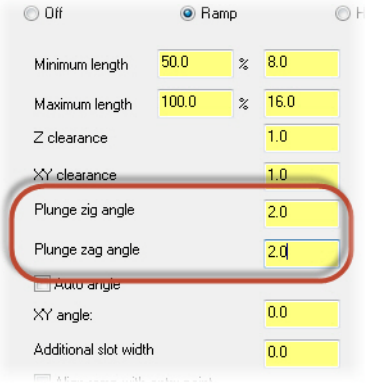


6 Select the **Roughing** page.

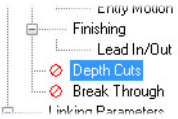
The defaults will be used instead of changing any parameters. The page is selected simply to show the **Stepover percentage** is set to **50.0**, equalling a **Stepover distance** of **8.0** mm.

7 Select the **Entry Motion** page.

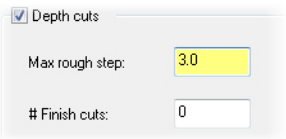
8 Activate **Ramp** and set the parameters as shown. The only options that should require changing are circled.



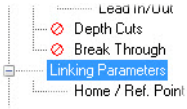
9 Select the **Depth Cuts** page.



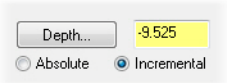
10 Click the **Depth cuts** checkbox and set the Max rough step to **3.0** mm.



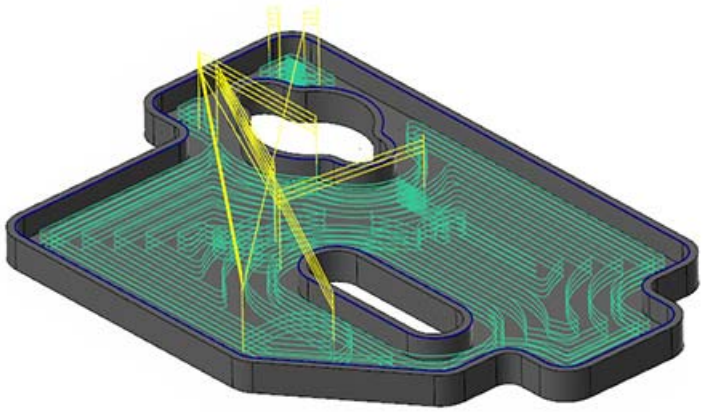
11 Select the **Linking Parameters** page.



12 Enter **-9.525** mm for the **Depth**. Ensure that it is set to **Incremental**.



13 Click **OK** to generate the Pocket toolpath.

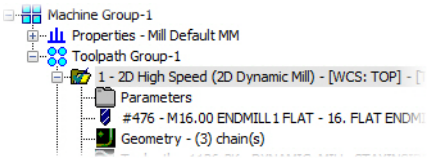


14 Save your part file.

The same geometry is used to generate the toolpath for Dynamic Mill and the Pocket toolpath. The tool motion is drastically different. Both operations will machine your part for you with the given parameters. Switch back and forth between selected operations in the Toolpaths Manager to quickly view the differences.

Exercise 4: Comparing the Operations

1 Select operation 1 in the Toolpaths Manager.

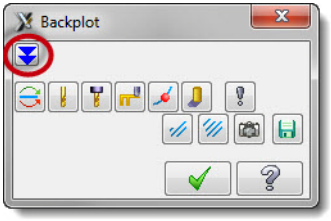


2 Press [**F3**] to refresh the graphics display if necessary.

3 Click **Backplot selected operations** in the Toolpaths Manager.

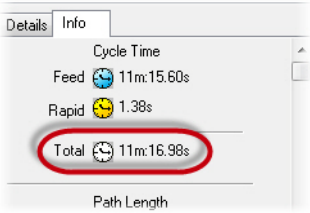
4 Click **Play** to review the tool motion for the Dynamic Mill operation.

5 Click **Expand** in the upper left corner of the Backplot dialog box.



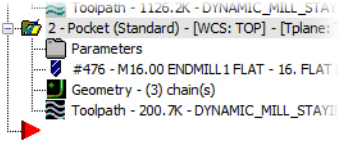
6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.

Valuable information is available on this tab. Of most interest in this exercise is the total runtime. Dynamic Mill is able to machine the part in the time shown with the given parameters.



7 Click **OK** to close the Backplot dialog box.

8 Select operation 2 in the Toolpaths Manager.



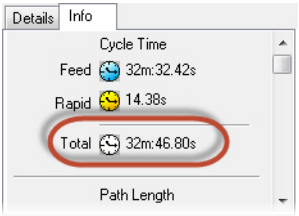
9 Press [**F3**] to refresh the graphics display if necessary.

10 Click **Backplot selected operations**.

11 Click **Play** to review the tool motion for the Pocket operation.

12 Select the **Info** tab in the lower half of the expanded Backplot dialog box.

Pocket is able to machine the part in the time shown with the given parameters.



Dynamic Mill with the **Machining strategy** set to **Stay inside**, is able to remove the material in the pocket in less than half the time it takes for the standard Pocket operation. Each operation is set to leave 0.5 mm of stock on the pocket walls. The Pocket operation has a single finish pass enabled to achieve a finish on the walls similar to what you get from the dynamic toolpath. Disable the finish pass in the pocket operation and the runtime drops to just under 22 minutes, which is still significantly longer than the dynamic operation.

Substantial improvements in productivity and efficiency can be achieved using the dynamic toolpaths. Neither of the two operations in this lesson are fully optimized for your part, material, tooling, or machine. Even so, you will see significant benefits in using dynamic motion and the dynamic toolpaths in Mastercam. When ready, continue on to Lesson 4 to begin working with a Dynamic Mill toolpath with a Rest Mill strategy.

LESSON 4
Dynamic Mill Toolpath (Rest Mill Strategy)

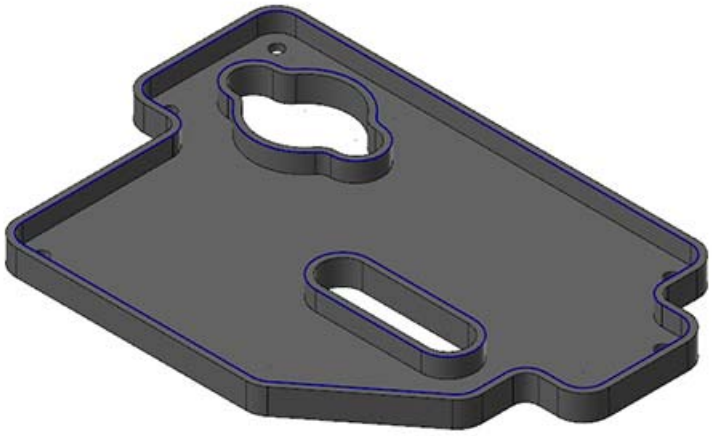
Lesson 3 demonstrated some of the advantages a Dynamic Mill toolpath with the Stay Inside strategy has over a Pocket toolpath. This lesson shows how a Dynamic Mill toolpath with the Rest Mill strategy improves upon the previous generation of the Area Mill toolpath with the Rest Mill strategy. Each toolpath has uses, however, in areas where you want to avoid tool burial while taking a full depth of cut, the dynamic option is the better choice.

Lesson Goals

- Create a Dynamic Mill toolpath with the Rest Mill strategy
- Create an Area Mill toolpath with the Rest Mill strategy
- Compare the two toolpaths

Exercise 1: Preparing Mastercam for Toolpath Creation

1 Open the part file `Dynamic_Mill_Rest.MCX-9`, which was provided with the tutorial.



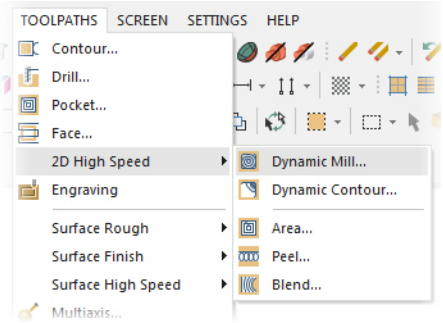
2 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

3 Choose **File, Save As**, and save the part under a different file name.

Exercise 2: Creating a Dynamic Mill Toolpath

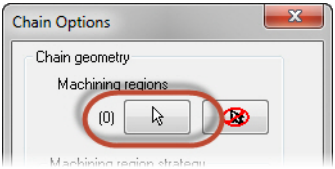
1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Mill**.

The Chain Options dialog box displays.

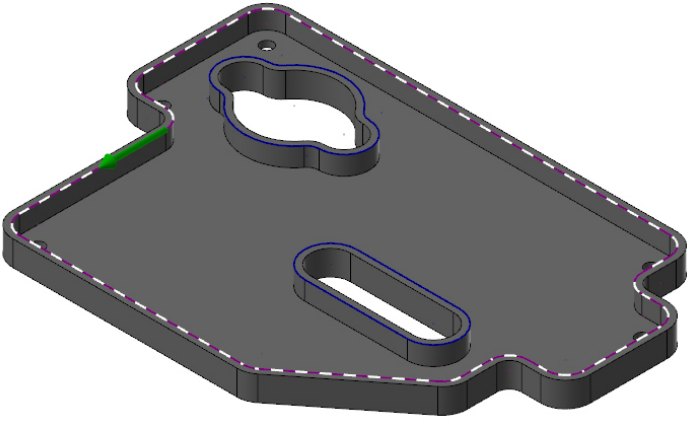


2 Click **Select** under Machining regions.

The Chaining dialog box displays.



3 Chain the wireframe geometry around the part as indicated.

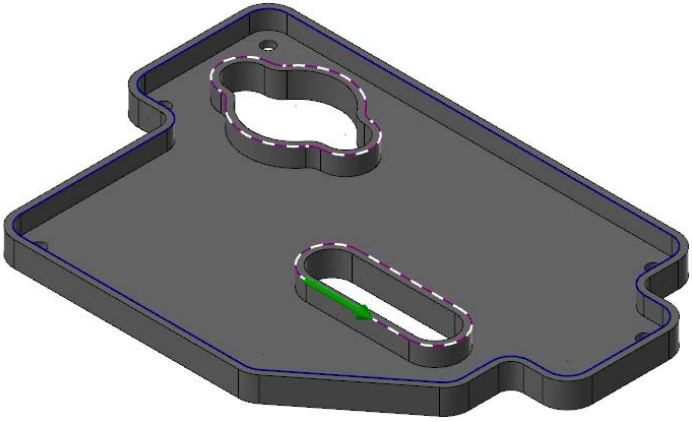


4 Click **OK** to close the Chaining dialog box and return to the Chain Options dialog box.

5 Click **Select** under Avoidance regions.

The Chaining dialog box displays.

6 Chain the wireframe geometry around the two features as indicated.



7 Click **OK** to close the Chaining dialog box. Click **OK** to close the Chain Options dialog box.

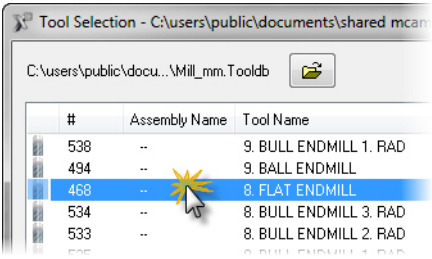
The 2D High Speed - Dynamic Mill Toolpath dialog box displays.

8 Select the **Tool** page.

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

The Tool Selection dialog box displays.

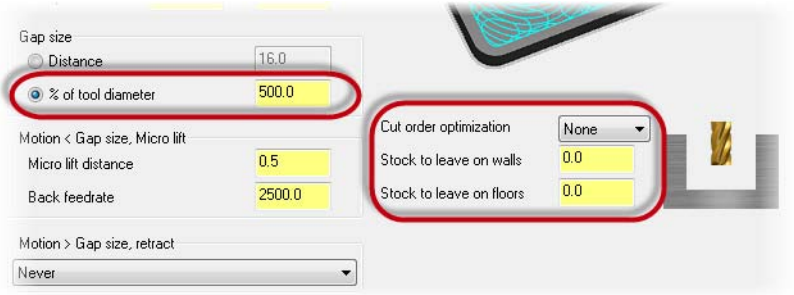
10 Select the **8mm flat endmill** and click **OK**.



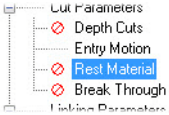
11 Select the **Cut Parameters** page.

12 Set the following parameters:

- Set **Cut order optimization** to **None**.
- Set **% of tool diameter** to **500.0**.
- Set **Stock to leave on walls** and **Stock to leave on floors** to **0.0**.

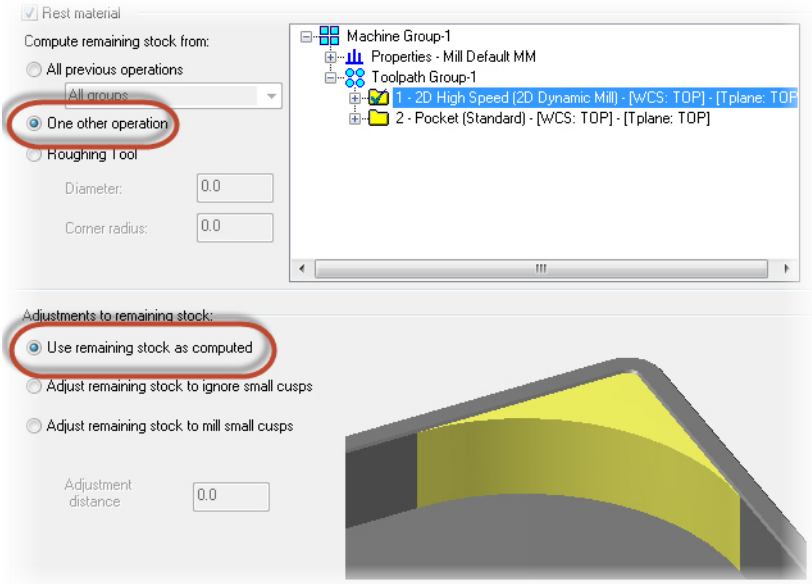


13 Select the **Rest Material** page.

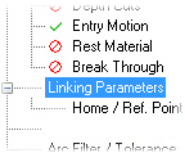


14 Select the **Rest material** checkbox to enable the page.

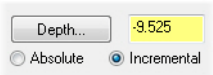
15 Select **One other operation**, operation **1**, and **Use remaining stock as computed** as shown.



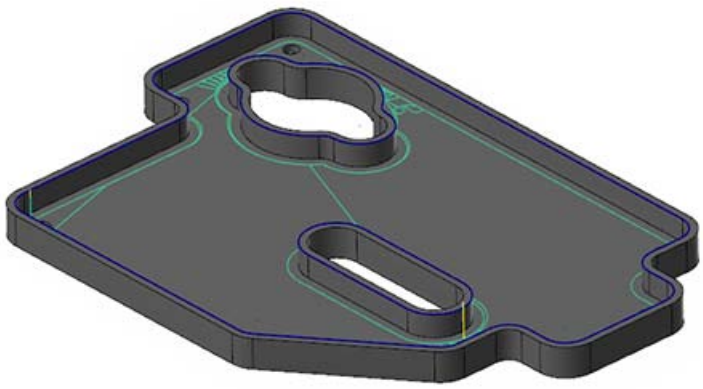
16 Select the **Linking Parameters** page.



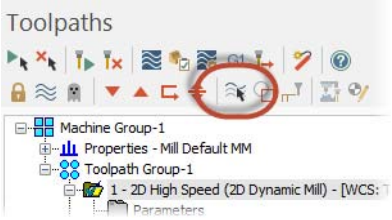
17 Enter **-9.525** mm for the **Depth**. Ensure that it is set to **Incremental**.



18 Click **OK** to generate the Dynamic Mill toolpath.



19 Activate **Only display selected toolpaths** in the Toolpaths Manager, if it is not already selected.



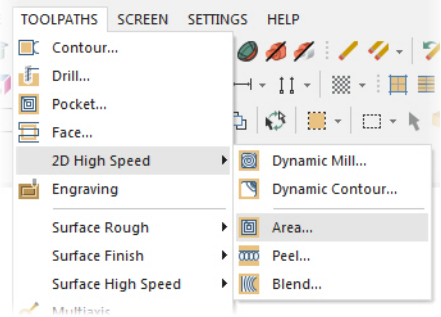
20 Save your part file.

Dynamic Mill removes material with a full depth of cut just like the previous lesson's Dynamic Mill toolpath. The same efficient motion is used to safely and quickly remove all remaining material.

Exercise 3: Creating An Area Mill Toolpath

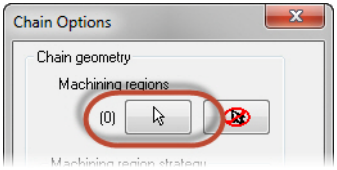
1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Area**.

The Chain Options dialog box displays.

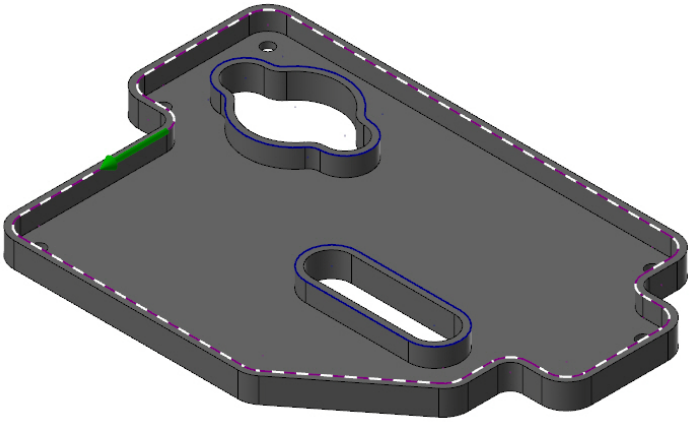


2 Click **Select** under Machining regions.

The Chaining dialog box displays.



3 Chain the wireframe geometry around the part as indicated.

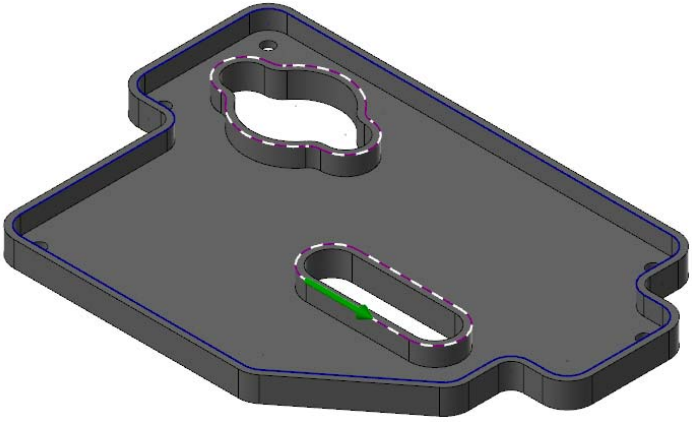


4 Click **OK** to close the Chaining dialog box and return to the Chain Options dialog box.

5 Click **Select** under Avoidance regions.

The Chaining dialog box displays.

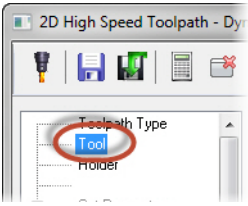
6 Chain the wireframe geometry around the two features as indicated.



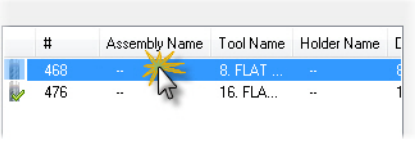
7 Click **OK** to close the Chaining dialog box. Click **OK** to close the Chain Options dialog box.

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

8 Select the **Tool** page.

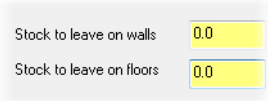


9 Select the **8mm flat endmill** from the tool list.



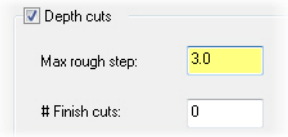
10 Select the **Cut Parameters** page.

11 Enter **0.0** mm in the **Stock to leave on walls** and **Stock to leave on floors** parameters.



12 Select the **Depth Cuts** page.

13 Click the **Depth cuts** checkbox and set the **Max rough step** to **3.0** mm.



14 Select the **Rest Material** page.

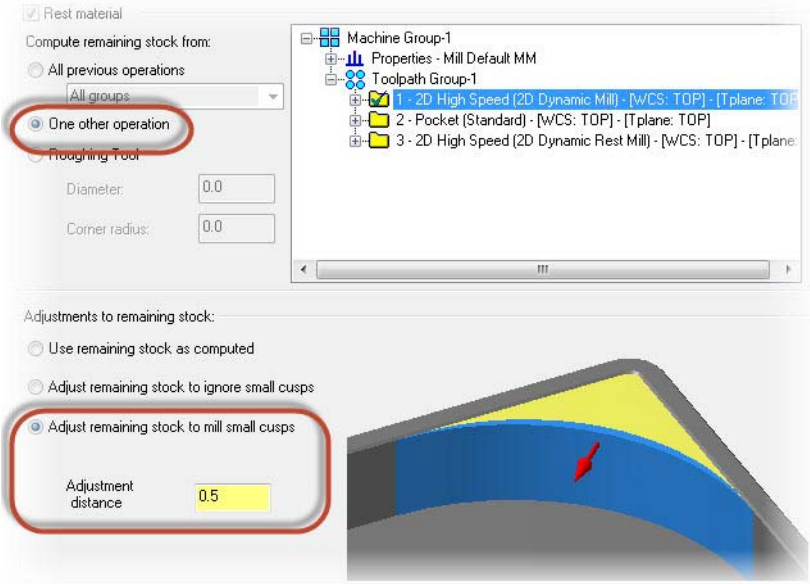
15 Select the **Rest material** checkbox to enable the page.

16 Set the following parameters:

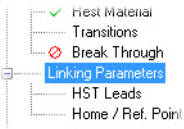
- Set Compute remaining stock from to **One other operation.** Select operation 1.
- Set Adjustments to remaining stock to **Adjust remaining stock to mill small cusps**.

  This option sets the adjustment distance to expand the stock model and Mastercam outputs cuts that engage small amounts of material.

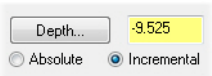
- Enter **0.5** for the **Adjustment distance**.



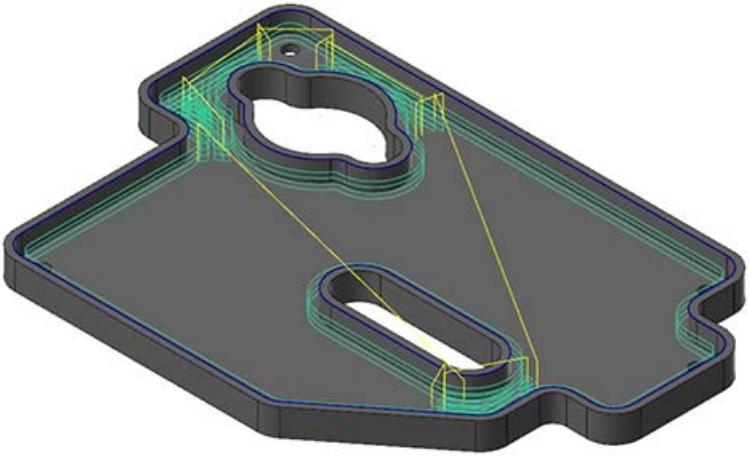
17 Select the **Linking Parameters** page.



18 Enter **-9.525** mm for the **Depth**. Ensure that it is set to **Incremental**.



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



20 Save your part file.

Exercise 4: Comparing the Operations

1 Select operation 3 in the Toolpaths Manager.

2 Press [**F3**] to refresh the graphics display if necessary.

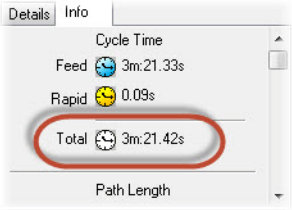
3 Click **Backplot selected operations** in the Toolpaths Manager.

4 Click **Play** to review the tool motion for the Dynamic Mill operation.

5 If necessary, click **Expand** in the upper left corner of the Backplot dialog box.

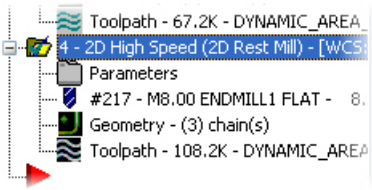
6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.

Once again, the total runtime is of most interest for this exercise.



7 Click **OK** to close the Backplot dialog box.

8 Select operation 4 in the Toolpaths Manager.

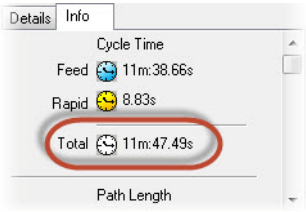


9 Press [**F3**] to refresh the graphics display if necessary.

10 Click **Backplot selected operations**.

11 Click **Play** to review the tool motion for the Area Mill operation.

12 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



Edit the Area Mill and Dynamic Mill parameters to match how you operate your machinery. Regenerate the toolpaths and compare the runtimes again. In most cases the dynamic options will be more efficient. There are exceptions and that is why both Area Mill and Dynamic Mill are offered in Mastercam. When ready, continue on to Lesson 5 to examine another Dynamic Mill toolpath.

LESSON 5
Dynamic Mill Toolpath (From Outside Strategy)

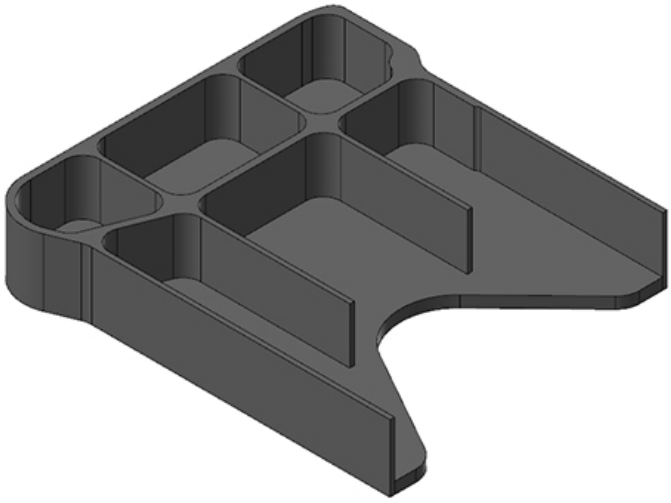
Lesson 4 showed how the Dynamic Mill toolpath is an improvement over the standard Area Mill toolpath. This lesson compares a Dynamic Mill toolpath with the **Machining region strategy** set to **From Outside** with a standard Pocket toolpath.

Lesson Goals

- Create a Dynamic Mill toolpath with the From Outside strategy
- Create a Pocket toolpath
- Compare the two operations

Exercise 1: Preparing Mastercam for Toolpath Creation

1 Open the part file `Dynamic_Mill_FromOutside.MCX-9`, which was provided with the tutorial.



2 Select **Isometric (WCS)** from the Gview menu.

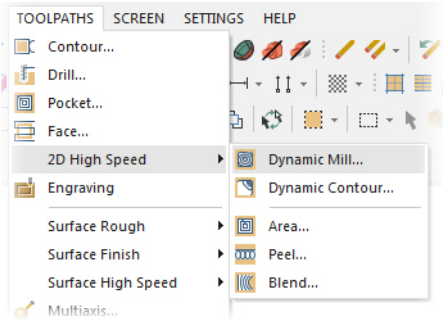
3 Shade the part, if necessary, using [**Alt+S**].

4 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

5 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

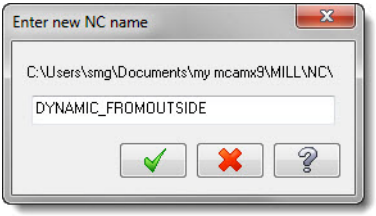
Exercise 2: Creating a Dynamic Mill Toolpath

1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Mill**.



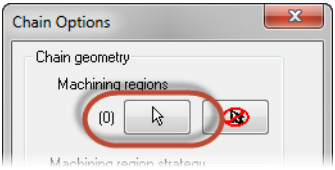
2 Click **OK** if prompted to enter a new NC file name.

The Chain Options dialog box displays.

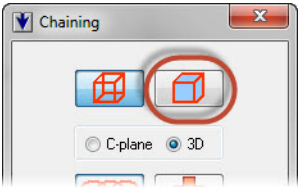


3 Click **Select** under Machining regions.

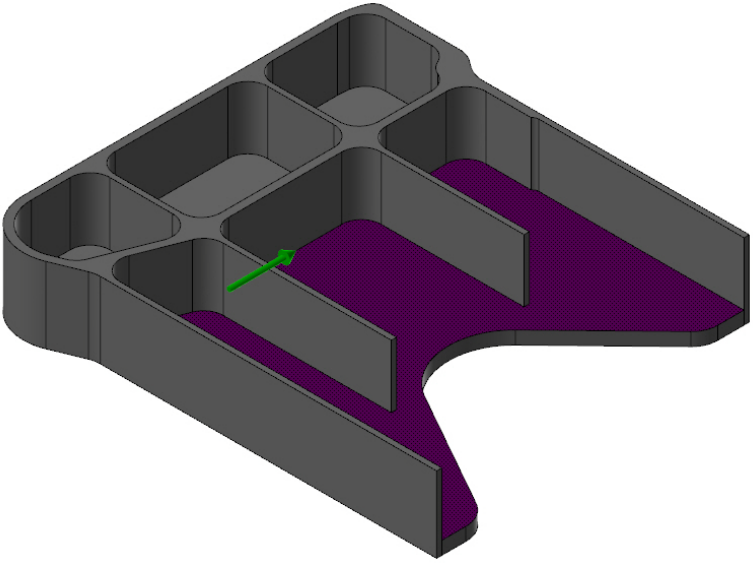
The Chaining dialog box displays.



4 Select **Solids** in the Chaining dialog box.



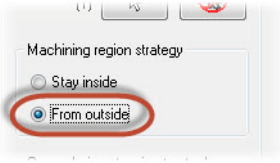
5 Select the solid face as shown.



NOTE: Be sure that the Face filter is selected, or else you will not be able to select the solid face for the toolpath.

6 Click **OK** to close the Chaining dialog box and return to the Chain Options dialog box.

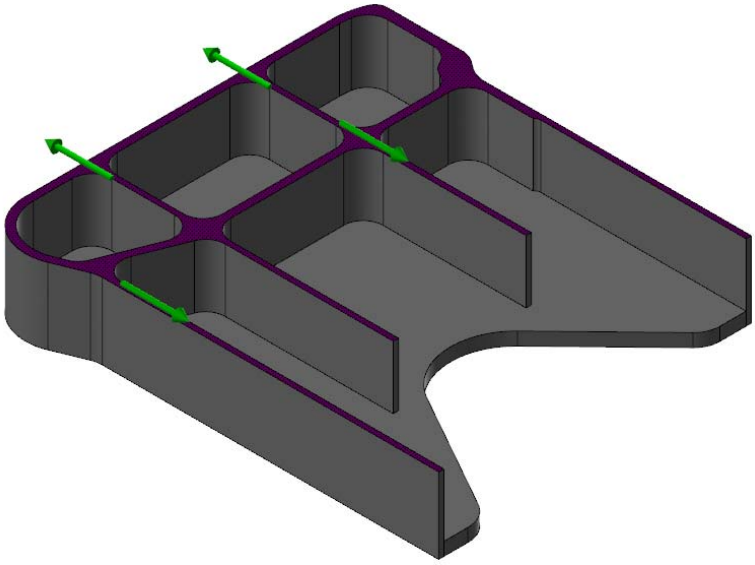
7 Set the **Machining region strategy** to **From outside**.



8 Click **Select** under Avoidance regions.

The Chaining dialog box displays.

9 Select the solid face as shown.



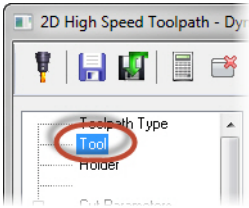
NOTE: Be sure that the Face filter is selected, or else you will not be able to select the solid face for the toolpath.

10 Click **OK** to close the Chaining dialog box, and again on the Chain Options dialog box.

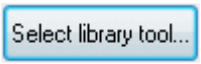
The 2D High Speed Toolpath - Dynamic Mill dialog box displays.

The lower face that was selected will be machined. The face on top of the walls is used as geometry to be avoided because it has a smaller 2D area.

11 Select the **Tool** page.

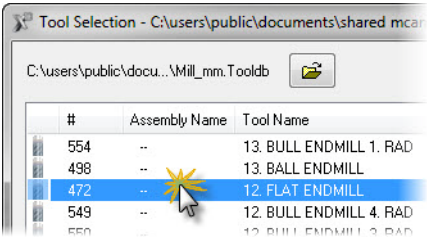


12 Click the **Select library tool** button.

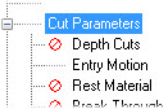


The Tool Selection dialog box displays.

13 Select the **12mm flat endmill** and click **OK**.

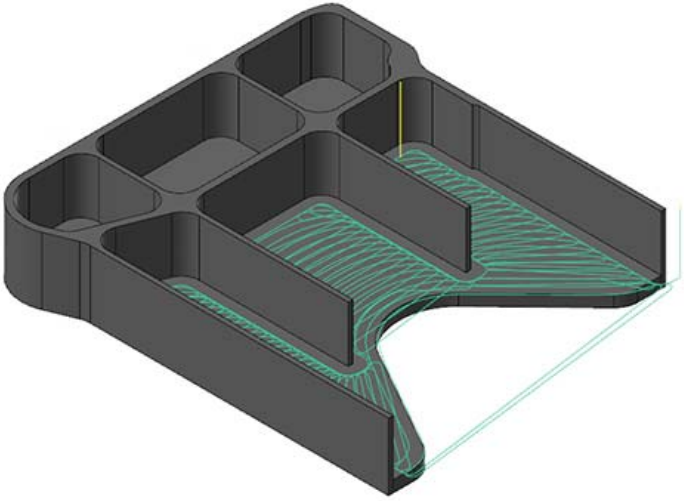


14 Select the **Cut Parameters** page.

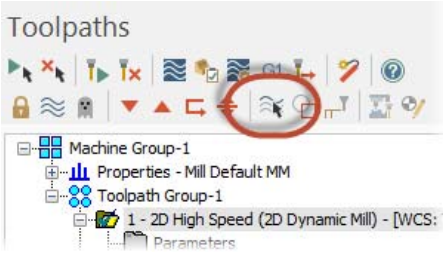


15 Set the parameters as shown.

16 Click **OK** to generate the Dynamic Mill toolpath.



17 Activate **Only display selected toolpaths** in the Toolpaths Manager, if it is not already selected.

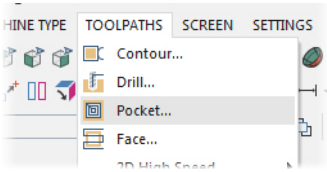


18 Save your part file.

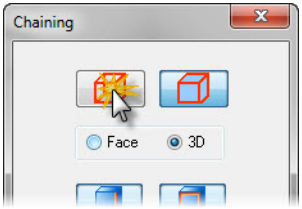
Exercise 3: Creating a Pocket Toolpath

1 Use the Level Manager to make level **5:Pocket boundary** visible.

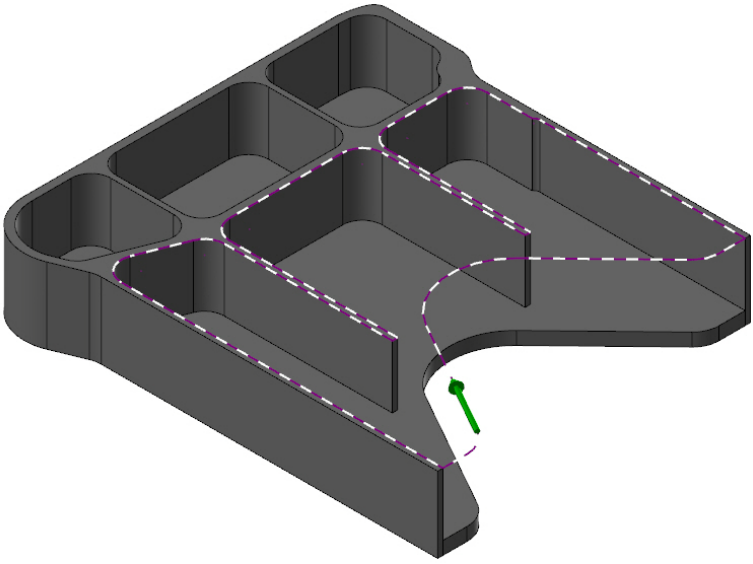
2 From the Mastercam menu, choose **Toolpaths, Pocket**.



3 Select **Wireframe** in the Chaining dialog box.

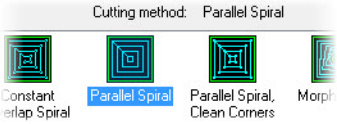


4 Select the chain as shown and click **OK**.

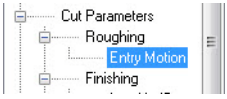


5 Select the **Roughing** page.

6 Select **Parallel Spiral** for the cutting method.

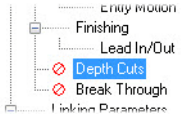


7 Select the **Entry Motion** page.



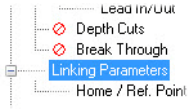
8 Activate **Helix**. Use the default parameters for the helix entry method.

9 Select the **Depth Cuts** page.

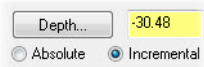


10 Click the **Depth cuts** checkbox and set the **Max rough step** to **6.0** mm.

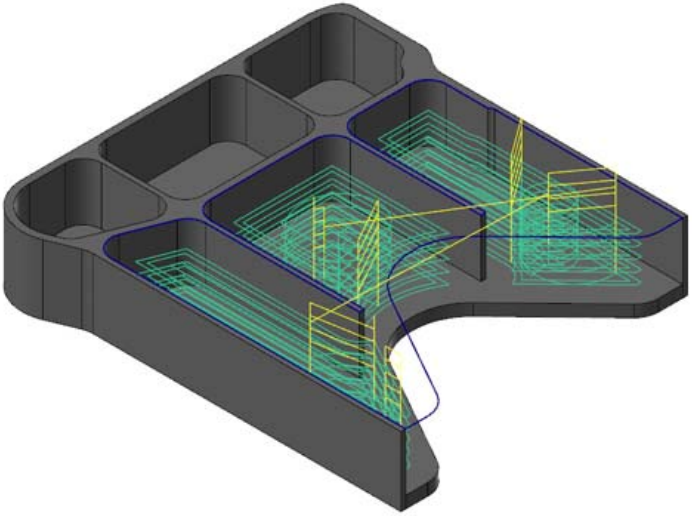
11 Select the **Linking Parameters** page.



12 Enter **-30.48** mm for the **Depth**. Ensure that it is set to **Incremental**.



13 Click **OK** to generate the Pocket toolpath.



14 Save your part file.

Exercise 4: Comparing the Operations

1 Select operation 1 in the Toolpaths Manager.

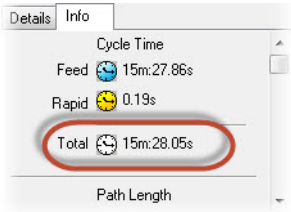
2 Press [**F3**] to refresh the graphics display if necessary.

3 Click **Backplot selected operations** in the Toolpaths Manager.

4 Click **Play** to review the tool motion for the Dynamic Mill operation.

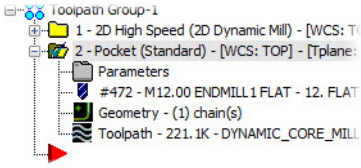
5 If necessary, click **Expand** in the upper left corner of the Backplot dialog box.

6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



7 Click **OK** to close the Backplot dialog box.

8 Select operation 2 in the Toolpaths Manager.

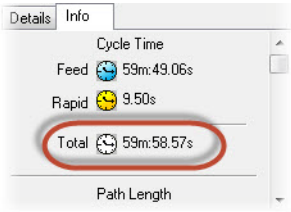


9 Press [**F3**] to refresh the graphics display if necessary.

10 Click **Backplot selected operations**.

11 Click **Play** to review the tool motion for the Pocket operation.

12 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



Edit the Pocket operation to test the runtime with parameters suitable for your machine. Experiment with reasonable values to see what parameters are necessary to better the runtime for the Dynamic Mill operation. Drastic changes, such as increasing the max rough step to 16.0 mm, result in a runtime that is still four minutes longer than the dynamic operation. When ready, continue on to Lesson 6 to review a Dynamic Contour toolpath.

LESSON 6
Dynamic Contour Toolpath

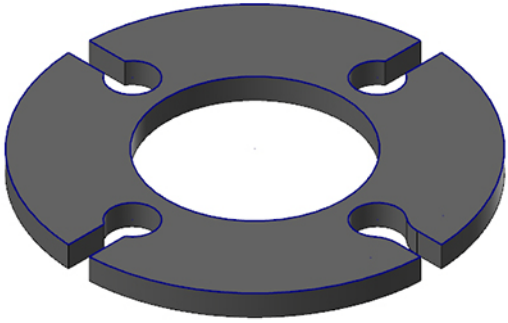
Lesson 5 presented Dynamic Mill and Pocket as options for milling a sample part. This lesson creates a Dynamic Contour toolpath in contrast to a standard Contour toolpath. The part presents challenging areas where a standard Contour toolpath will bury the tool if cutting anything other than a finish pass.

Lesson Goals

- Create a Dynamic Contour toolpath
- Create a Contour toolpath
- Compare the two operations

Exercise 1: Preparing Mastercam for Toolpath Creation

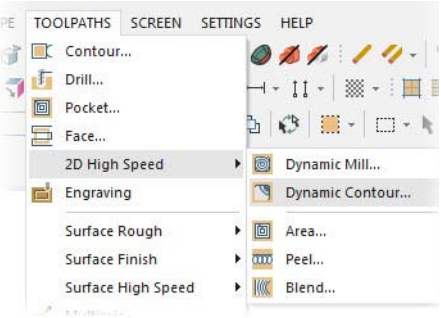
1. Open the part file `Dynamic_Contour.MCX-9`, which was provided with the tutorial.



2. Select **Isometric (WCS)** from the Gview menu.
3. Shade the part, if necessary, using [**Alt+S**].
4. Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.
5. Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

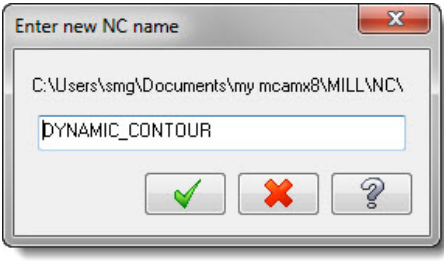
Exercise 2: Creating a Dynamic Contour Toolpath

1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Contour**.



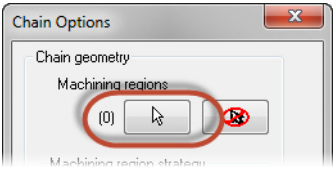
2 Click **OK** if prompted to enter a new NC file name.

The Chain Options dialog box displays.



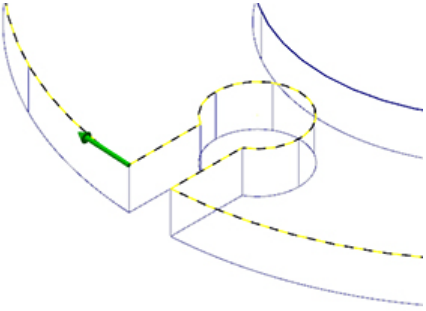
3 Click **Select** under Machining regions.

The Chaining dialog box displays.



4 Select the contour around the upper edge of the part as shown.

Chain the contour in a clockwise direction. Disable shading as needed to make selection and direction easier.



5 Click **OK** to close the Chaining dialog box, and again for the Chain Options dialog box.

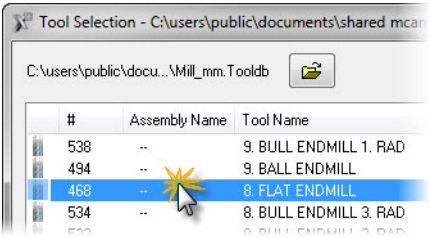
The 2D High Speed Toolpaths - Dynamic Contour dialog box displays.

6 Select the **Tool** page.

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

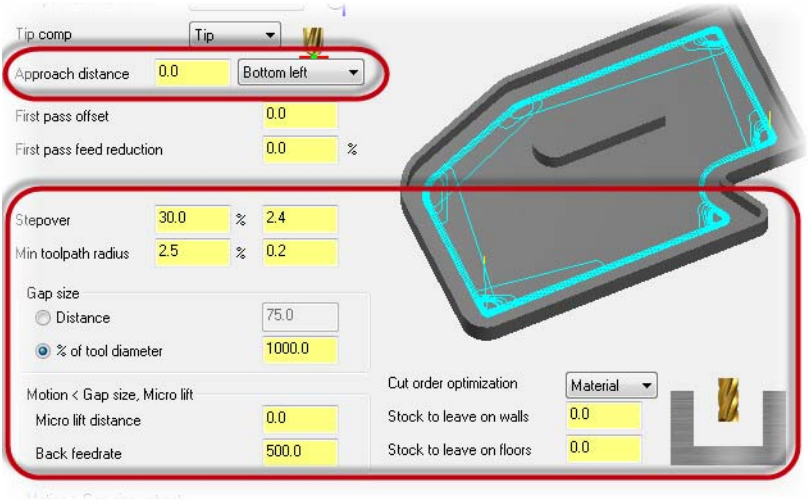
The Tool Selection dialog box displays.

8 Select the **8mm flat endmill** and click **OK**.



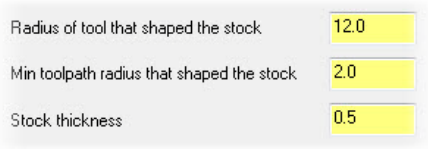
9 Select the **Cut Parameters** page.

10 Set the parameters as shown.

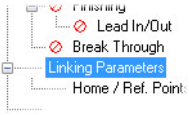


11 Select the **Contour Wall** page.

12 Enter the parameters as shown.

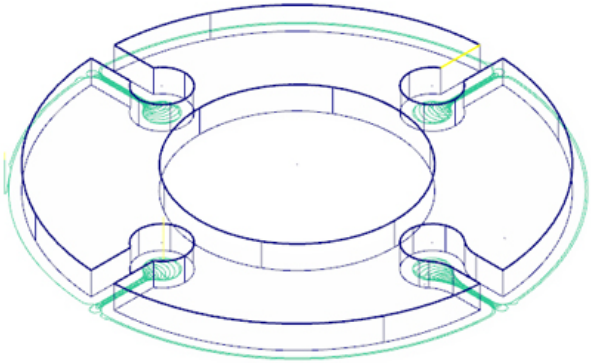


13 Select the **Linking Parameters** page.



14 Enter **-10.0** mm for the **Depth**.
Ensure that it is set to **Incremental**.

15 Click **OK** to generate the Dynamic Contour toolpath. Shading has been disabled to make the tool motion more apparent.



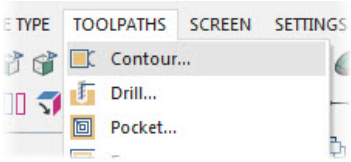
16 Click **Only display selected toolpaths** in the Toolpaths Manager, if it is not already selected.

17 Save your part file.

Exercise 3: Creating a Contour Toolpath

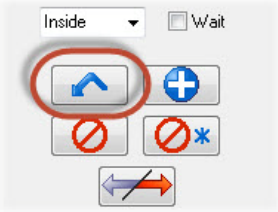
1 From the Mastercam menu, choose **Toolpaths, Contour**.

The Chaining dialog box displays.



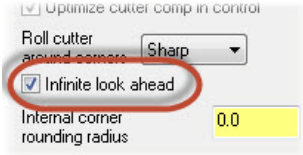
2 Click **Last** in the Chaining dialog box, then click **OK**.

The 2D Toolpaths - Contour dialog box displays.



3 Select the **Cut Parameters** page.

4 Deselect **Infinite look ahead**.



5 Select the **Depth Cuts** page.

6 Click the **Depth cuts** checkbox and set the Max rough step to **5.0** mm.

7 Select the **Multi Passes** page.

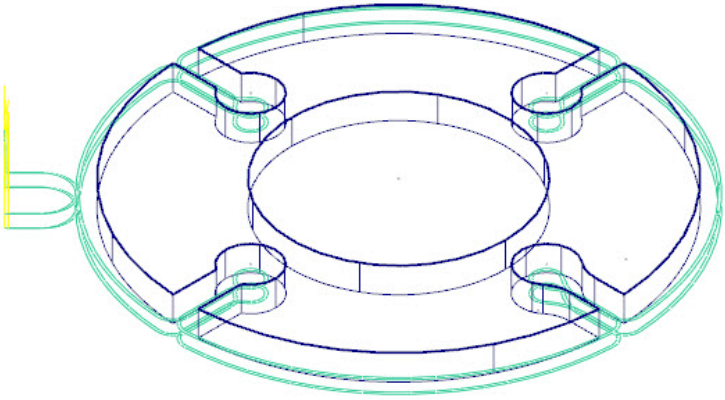
8 Click the **Multi Passes** checkbox and enter the values shown.

Multi Passes are used for this part to avoid burying the tool in the four concave sections of the contour.

9 Select the **Linking Parameters** page.

10 Enter **-10.0** mm for the **Depth**.

11 Click **OK** to generate the Contour toolpath.

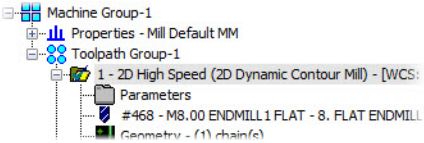


Immediately you can see unnecessary tool motion around the outer edge of the part. The multi passes that are needed to avoid tool burial in the concave sections generate extra passes on the external edges. You can get around this by creating operations to remove material in the concave sections before generating a simpler contour operation. However, the overall machining time, as well as programming time, exceeds that of the Dynamic Contour operation.

12 Save your part file.

Exercise 4: Comparing the Operations

1 Select operation 1 in the Toolpaths Manager.



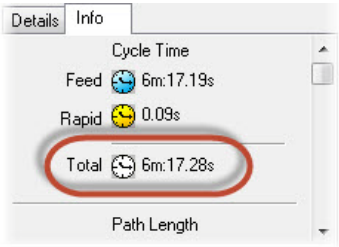
2 Press [**F3**] to refresh the graphics display if necessary.

3 Click **Backplot selected operations** in the Toolpaths Manager.

4 Click **Play** to review the tool motion for the Dynamic Contour operation.

5 If necessary, click **Expand** in the upper left corner of the Backplot dialog box.

6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



7 Click **OK** to close the Backplot dialog box.

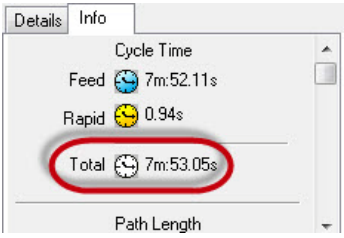
8 Select operation 2 in the Toolpaths Manager.

9 Press [**F3**] to refresh the graphics display if necessary.

10 Click **Backplot selected operations**.

11 Click **Play** to review the tool motion for the Pocket operation.

12 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



Once again you should edit the parameters for the contour operation to match your machine's capabilities. The standard contour operation has a shorter runtime without multi passes. If your machine, part material, and tooling allow for full tool burial then that may be a better option for your toolpath. Dynamic contour is the more efficient selection for parts containing pinched areas and small radii.

Continue onto the next lesson to review the Peel Mill toolpath.

LESSON 7
Peel Mill Toolpath

This lesson compares the Peel cutting style with the Dynamic peel cutting style when creating a Peel Mill toolpath. Unlike the previous toolpaths, the difference between the dynamic motion and standard motion is set in the Cut Parameters page and is not a separate toolpath. There are a few differences between these cutting styles, described below:

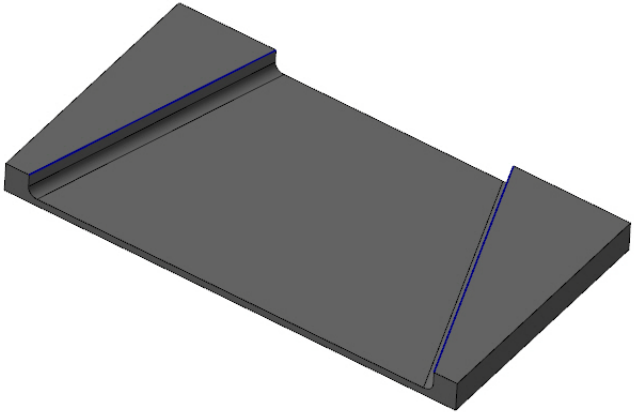
- The **Peel** cutting style machines areas by using an open or closed chain. This cutting style uses a trochoidal style of motion with accelerated back feed moves when the tool is not engaged in material.
- The **Dynamic peel** cutting style machines areas by using an open chain. This cutting style uses a dynamic style of motion with accelerated back feed moves when the tool is not engaged in material.

Lesson Goals

- Create a Peel Mill toolpath with the **Cutting style** set to **Dynamic peel**
- Create a Peel Mill toolpath with the **Cutting style** set to **Peel**
- Compare the two toolpaths

Exercise 1: Preparing Mastercam for Toolpath Creation

1 Open the part file `PeelMill.MCX-9`, which was provided with the tutorial.



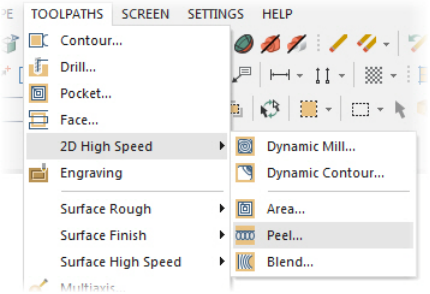
2 Select **Isometric (WCS)** from the Gview menu.

3 Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

4 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

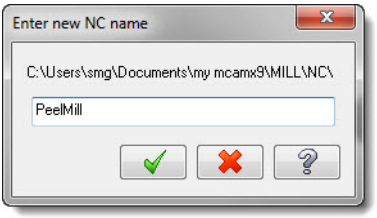
Exercise 2: Creating a Peel Mill Toolpath (Dynamic Peel)

1 From the Mastercam menu, select **Toolpaths**, **2D High Speed**, **Peel**.

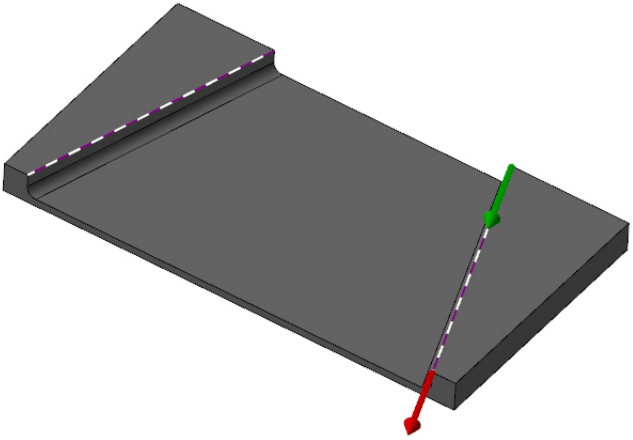


2 Click **OK** if prompted to enter a new NC file name.

The Chaining dialog box displays.



3 Select the wireframe geometry shown below. The chains must be going in the same direction.



4 Click **OK** to close the Chaining dialog box.

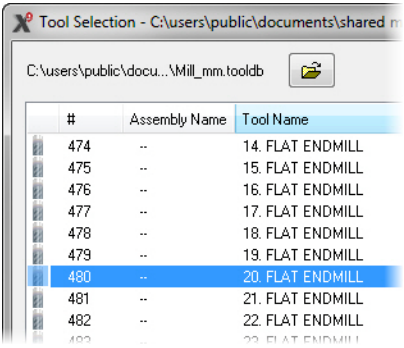
The 2D High Speed Toolpaths - Peel Mill dialog box displays.

5 Select the **Tool** page.

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

The Tool Selection dialog box displays.

7 Select the **20mm flat endmill** and click **OK**.

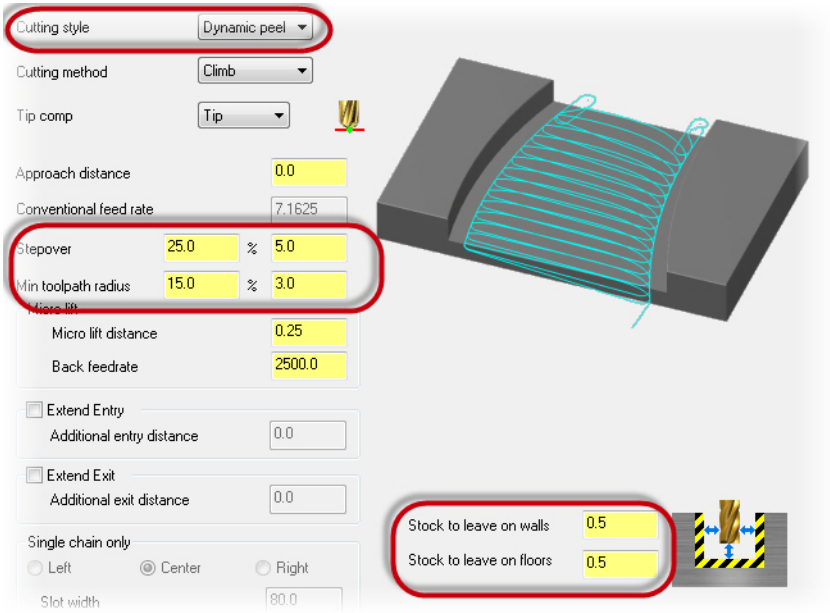


8 Select the **Cut Parameters** page.

9 Set the following parameters:

- Set the **Cutting style** to **Dynamic peel**.
- Set the **Stepover** to **25.0%**.
- Set the **Min toolpath radius** to **15.0**%.

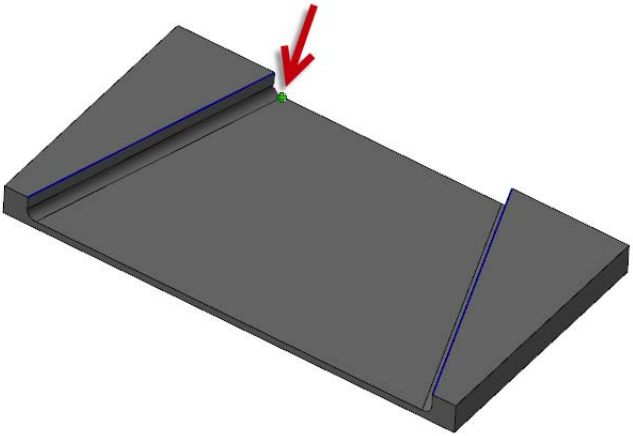
- Set the Stock to leave on walls and floors to **0.5**.



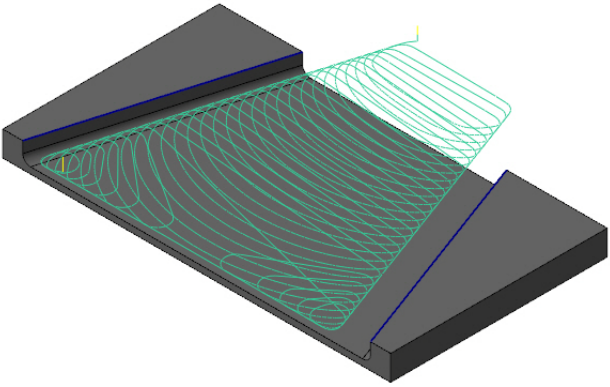
10 Select the **Linking Parameters** page.

11 Click the **Depth** button to return to the graphics window to set the depth of the toolpath.

12 Select the point shown below:



13 Click **OK** to generate the Peel Mill toolpath.



14 Click **Only display selected toolpaths** in the Toolpaths Manager, if it is not already selected.

15 Save your part file.

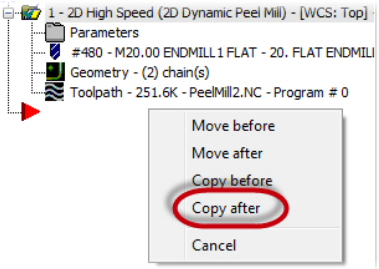
Exercise 3: Creating a Peel Mill Toolpath (Peel)

1 In the Toolpaths Manager, right-click the 2D High Speed (2D Dynamic Peel Mill) operation and drag it onto the Toolpaths Manager.

A pop-up will display.

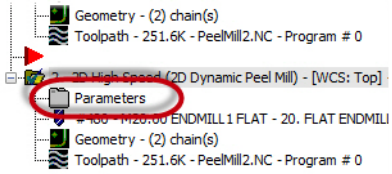
2 In the pop-up menu, select **Copy after**.

This copies the previous operation. Instead of creating a new toolpath, a copy of the first operation will be edited instead.



3 In the Toolpaths Manager, select **Parameters** under the second listed operation.

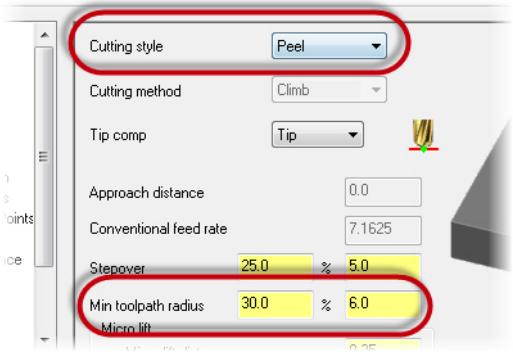
The 2D High Speed Toolpath - Peel Mill dialog box displays.



4 Select the **Cut Parameters** page if necessary.

5 Set the following parameters:

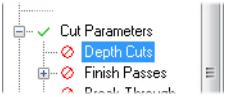
- Set the **Cutting style** to **Peel**.
- Set the **Min toolpath radius** to **30.0**%.



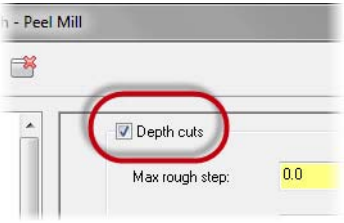
By changing the Cutting style, you will notice the Cutting method and Approach distance field become inactive. These fields are not available when using the standard cutting style.

The standard Peel Cutting style requires that the Min toolpath radius must be larger than the Stepover.

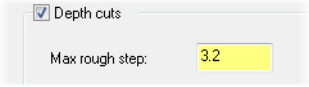
6 Select the **Depth Cuts** page.



7 Select the **Depth cuts** checkbox to activate the depth cut options.



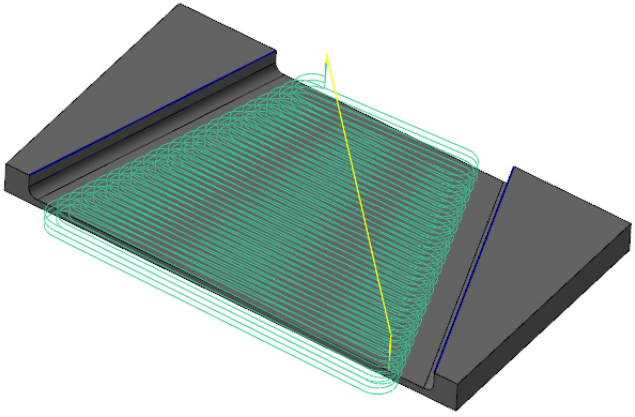
8 Set the **Max rough step** to **3.2** mm.



9 Click **OK** to accept the changes and close the 2D High Speed Toolpath - Peel Mill dialog box.

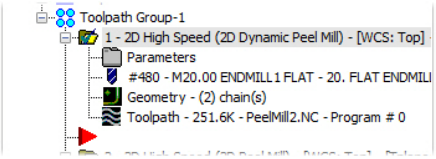
10 Click the **Regenerate all dirty operations** button in the Toolpaths Manager.

11 The Peel Mill toolpath displays, as shown below:.



Exercise 4: Comparing the Operations

1 Select the 2D Dynamic Peel Mill operation in the Toolpaths Manager.



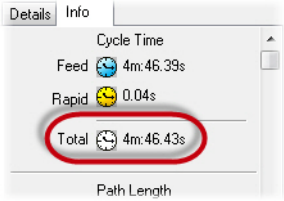
2 Press [**F3**] to refresh the graphics display if necessary.

3 Click **Backplot selected operations** in the Toolpaths Manager.

4 Click **Play** to review the tool motion for the Peel Mill operation.

5 If necessary, click **Expand** in the upper left corner of the Backplot dialog box.

6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.



7 Click **OK** to close the Backplot dialog box.

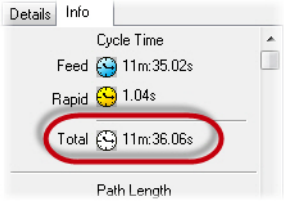
8 Select operation 2 in the Toolpaths Manager.

9 Press [**F3**] to refresh the graphics display if necessary.

10 Click **Backplot selected operations**.

11 Click **Play** to review the tool motion for the Peel Mill operation.

12 Select the **Info** tab in the lower half of the expanded Backplot dialog box.

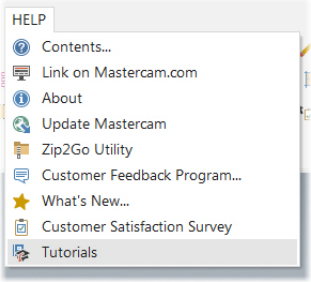


Edit both Peel Mill parameters to match how you operate your machinery. Regenerate the toolpaths and compare the runtimes again. In certain situations, the Peel cutting strategy may be more beneficial, especially when using closed chains. In most cases, however, the Dynamic peel strategy will be more efficient.

Conclusion

Congratulations! You have completed the *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. The Mastercam tutorial series is in continual development, and we will add modules as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.



Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's menu bar or by pressing [**Alt+H**] on your keyboard. Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *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 offer 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, or select **Tutorials** from the **Help** menu 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**, **Link on Mastercam.com** from the Mastercam menu and follow the instructions.

Mastercam Documentation

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

- *What's New in Mastercam X9*
- *Mastercam X9 Installation Guide*
- *Mastercam X9 Administrator Guide*
- *Mastercam X9 Transition Guide*
- *Mastercam X9 Quick Reference Card*
- *Mastercam X9 ReadMe*

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.
Go to Mastercam.com/Support for the latest downloads.

cnc software, inc.

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