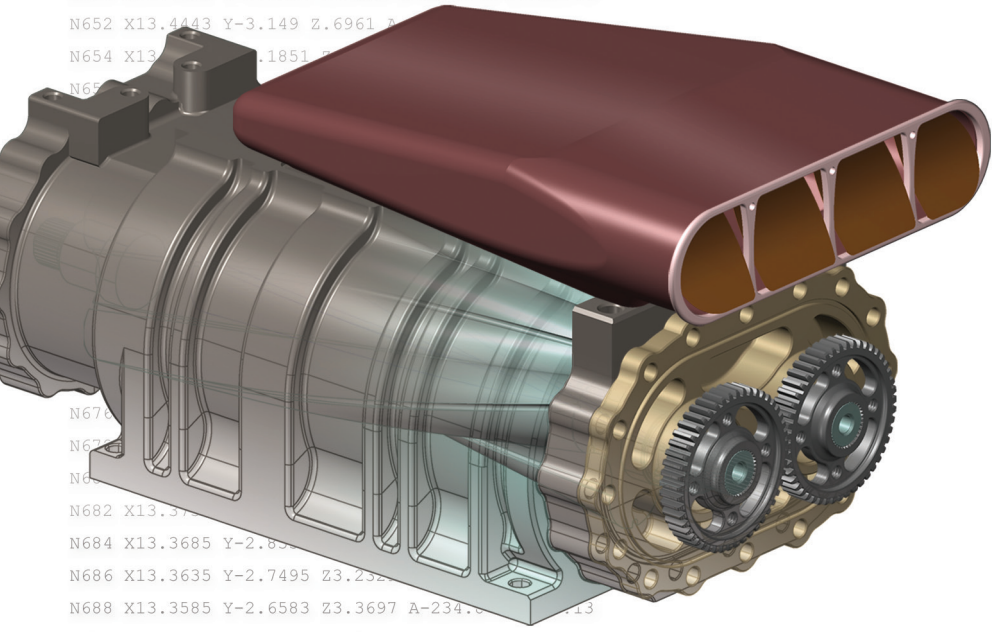
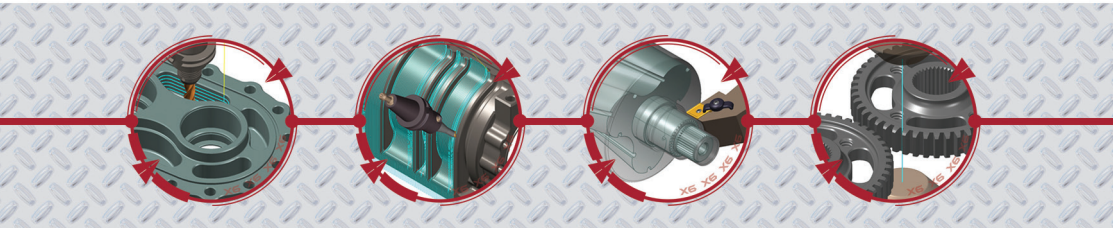
# Mastercam X6

## Dynamic Milling



When Second Best Won't Cut It.



mastercam x getting started tutorials

# Dynamic Milling

December 2011

**Be sure you have the latest information!**

Information might have been changed or added since this document was published. Contact your local Reseller for the latest information.

Mastercam X

**Mastercam® X6 Dynamic Milling**

Date: December 2011
Copyright © 2011 CNC Software, Inc.— All rights reserved.
First Printing: December 2011
Software: Mastercam X6

**TERMS OF USE**

Use of this document is subject to the Mastercam End User License Agreement. A copy of the Mastercam End User License Agreement is included with the Mastercam product package of which this document is part. The Mastercam End User License Agreement can also be found at:
www.mastercam.com/legal/licenseagreement/

# Contents


**Introduction** ..... 1

- Tutorial Goals ..... 2

General Tutorial Requirements ..... 3

**1. Dynamic Milling Overview** ..... 5

- Chain Selection for Dynamic Toolpaths ..... 5
- Dynamic Area Mill ..... 6
- Dynamic Rest Mill ..... 7
- Dynamic Core Mill ..... 7
- Dynamic Contour ..... 8
- Dynamic Milling Parameters ..... 9

**2. Chain and Entry Selection** ..... 11

- Lesson Goals ..... 11
- Exercise 1: Preparing Mastercam for Toolpath Creation ..... 11
- Exercise 2: Creating a Dynamic Toolpath ..... 13
- Exercise 3: Adding Chains to an Operation ..... 16
- Exercise 4: Selecting an Entry Method ..... 18

**3. Dynamic Area Mill Toolpath** ..... 21

- Lesson Goals ..... 21
- Exercise 1: Preparing Mastercam for Toolpath Creation ..... 21
- Exercise 2: Creating a Dynamic Area Mill Toolpath ..... 22
- Exercise 3: Creating a Pocket Toolpath ..... 27
- Exercise 4: Comparing the Operations ..... 30

**4. Dynamic Rest Mill Toolpath ..... 33**
- Lesson Goals ..... 33
- Exercise 1: Preparing Mastercam for Toolpath Creation ..... 33
- Exercise 2: Creating a Dynamic Rest Mill Toolpath ..... 34
- Exercise 3: Creating a Rest Mill Toolpath ..... 37
- Exercise 4: Comparing the Operations ..... 40

**5. Dynamic Core Mill Toolpath ..... 43**
- Lesson Goals ..... 43
- Exercise 1: Preparing Mastercam for Toolpath Creation ..... 43
- Exercise 2: Creating a Dynamic Core Mill Toolpath ..... 44
- Exercise 3: Creating a Pocket Toolpath ..... 47
- Exercise 4: Comparing the Operations ..... 50

**6. Dynamic Contour Toolpath ..... 53**
- Lesson Goals ..... 53
- Exercise 1: Preparing Mastercam for Toolpath Creation ..... 53
- Exercise 2: Creating a Dynamic Contour Toolpath ..... 54
- Exercise 3: Creating a Contour Toolpath ..... 57
- Exercise 4: Comparing the Operations ..... 59

**Conclusion ..... 60**
- Mastercam Resources ..... 61
- Mastercam Documentation ..... 62

Contact Us ..... 62

# INTRODUCTION

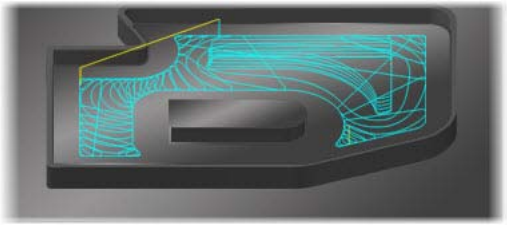
Mastercam's 2D high speed dynamic milling toolpaths utilize the entire flute length of their cutting tools to achieve great efficiency in milling. They are designed to maximize material removal while minimizing tool wear.

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

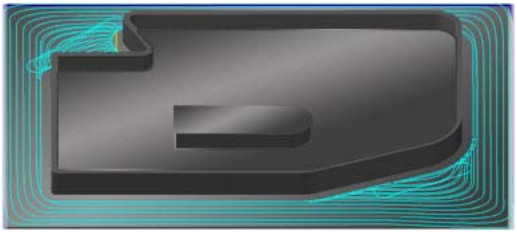
- Tool burial avoidance
- Minimum heat buildup
- Better chip evacuation

This tutorial introduces you to all of the following intelligent application-specific 2D high speed dynamic milling toolpaths available in Mastercam.

**Dynamic Area Mill—**Machines pockets using one or more chains to drive the toolpath. The chain that encompasses the largest 2D area contains the toolpath; all subsequent chains are considered islands. Use Dynamic Area Mill for parts you normally would machine with a Pocket toolpath.

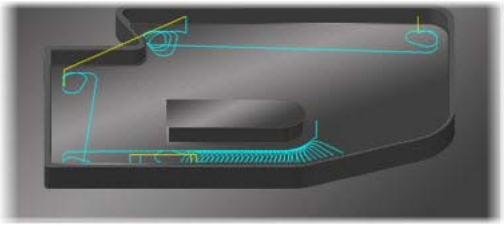


**Dynamic Core Mill—**Machines open pocket shapes or standing core shapes using the outmost chain as the stock boundary. The tool moves freely outside of this area. Use Dynamic Core Mill on parts where you need to face an area, generally on the outside of a part.



**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.

**Dynamic Rest Mill—**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 Dynamic Rest Mill when the roughing tools used leave material in corners or narrow part areas.



# 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-6`) is different from Mastercam (`MCX-6`), 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.
- *Focus Series* and *Exploring Series* tutorials require, at minimum, a mastery of the basic Mastercam skills taught in the *Getting Started Series* modules. A general knowledge of machining principals and practices is also required.
- You must have a seat of Mastercam X6 Design or higher to complete most of the tutorials in the *Getting Started Series*.
- The *Basic 2D Machining* module in the *Getting Started Series* requires, at minimum, a seat of Mill Entry or Router Entry.
- The *Basic 3D Machining* module in the *Getting Started Series* requires Mill Level 3 or Router Pro.
- 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.
- The *Getting Started Series* tutorials are available in an Adobe® Flash® compatible video format. Additional tutorial videos may also be available. Contact your local Mastercam Reseller for more information.
- You must install Adobe Flash Player to display tutorial videos. You can download Adobe Flash Player from www.adobe.com.

- 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

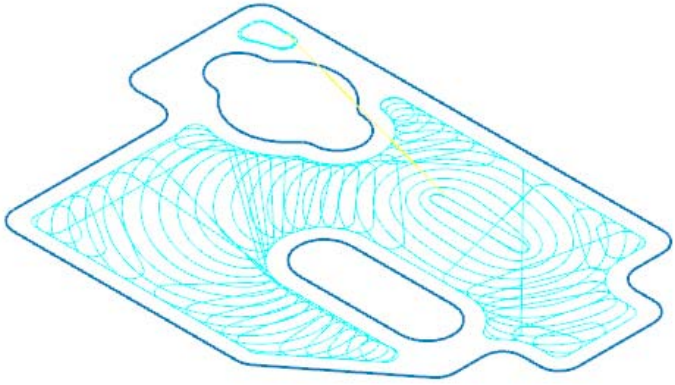
Mastercam's 2D high speed toolpaths (HST) contain several types that utilize dynamic motion:

- Dynamic Area Mill
- Dynamic Rest Mill
- Dynamic Core Mill
- Dynamic Contour

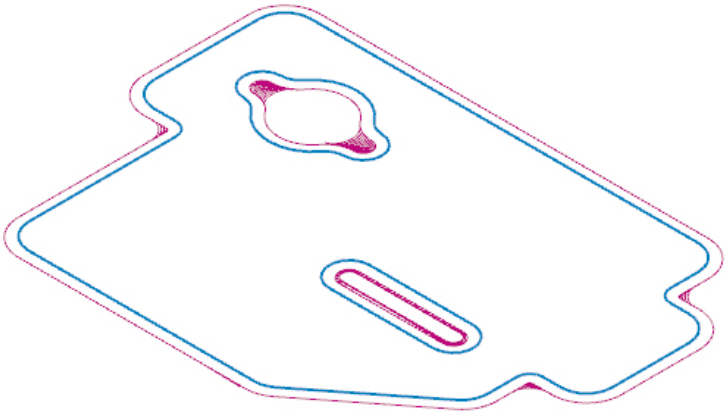
Dynamic toolpaths utilize as much of the tool's flute length as possible, while minimizing tool burial. This method enables the toolpath to rapidly and efficiently remove the maximum amount of material and extend tool life at the same time.

## *Chain Selection for Dynamic Toolpaths*

Dynamic Area, Dynamic Core, and Dynamic Rest milling require a unique way of understanding chain selection. These toolpaths must have at least one closed chain in order for a toolpath to be generated. The chain that encompasses the largest 2D area is considered to be the toolpath boundary. All remaining chains are treated as geometry to avoid. The exception to that is when an open chain is selected for use as an entry method.



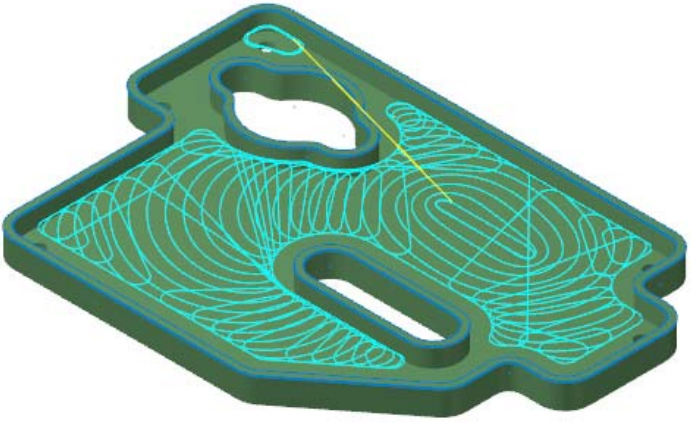
Dynamic Contour does not require a closed chain to calculate a toolpath. All geometry selected will be cut by the tool motion.



Chaining options are explored further throughout subsequent lessons.

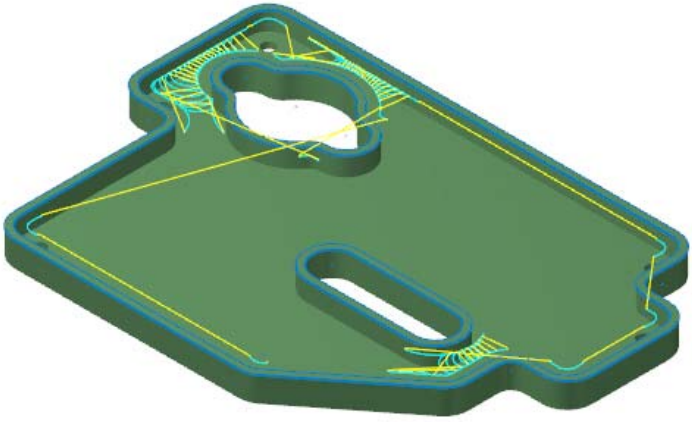
## Dynamic Area Mill

Dynamic area mill 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.



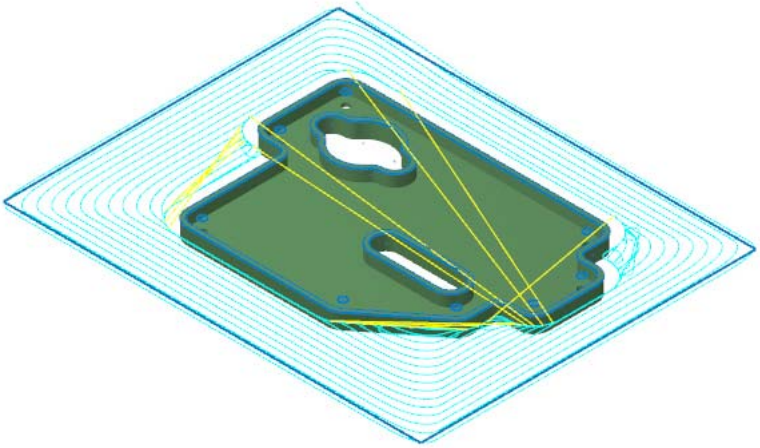
# Dynamic Rest Mill

Dynamic rest mill 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 Rest Mill operation.



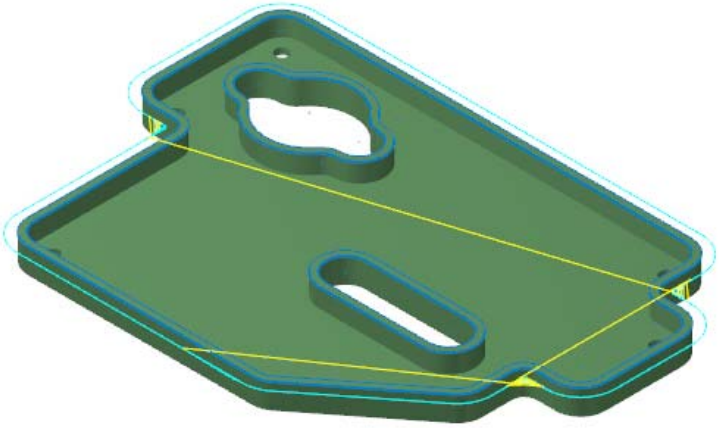
# Dynamic Core Mill

Dynamic core mill 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.



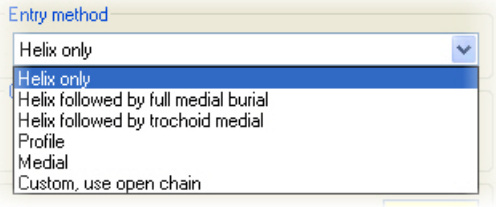
# Dynamic Milling Parameters

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

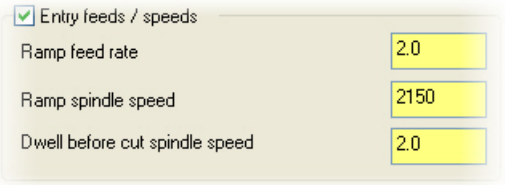
- Micro lift
- Entry methods
- Entry feeds/speeds
- Contour wall

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.

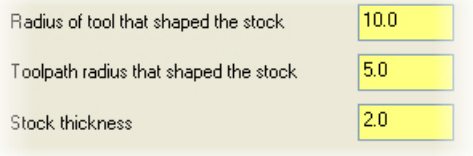
Entry methods provide several ways to enter the material for a Dynamic Area Mill operation. Options range from a simple helix to a medial path with trochoidal motion to a selected open chain. Mastercam's Help contains detailed information on each of the methods available.




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. Please review the *Basic 3D Design* and *Basic 3D Machining* tutorials before continuing if you are not familiar with these concepts.

## Lesson Goals

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

## Exercise 1: Preparing Mastercam for Toolpath Creation

1. Start Mastercam using your preferred method:

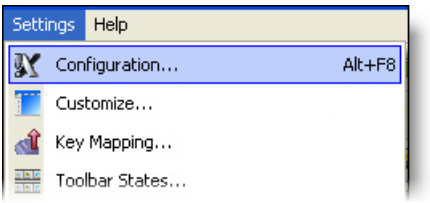
   - Double-click Mastercam's desktop icon.

     Or

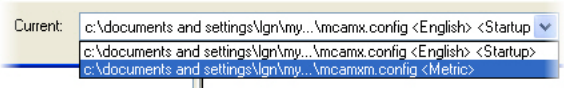
   - Launch Mastercam from the Windows Start menu.

2. Select the default metric configuration file:

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

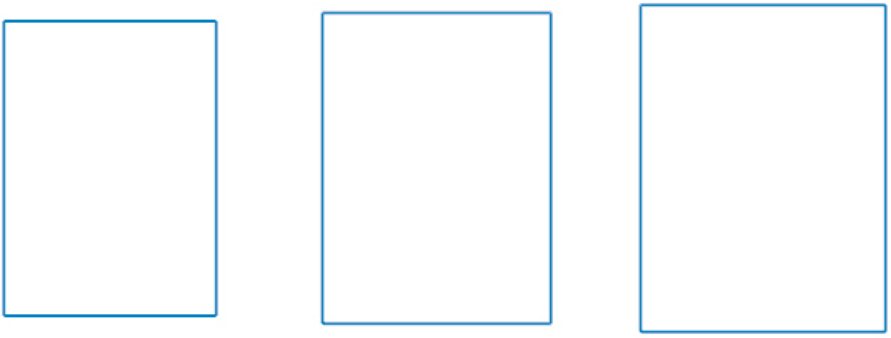



**b** Choose **...\mcamxm.config <Metric>** from the **Current** drop-down list.




**c** Click **OK**.

**3** Open the part file `Chaining_Examples.MCX-6`, which was provided with the tutorial.



**4** Select the default Mill metric machine definition.

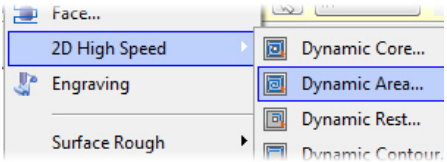
**5** Fit the geometry to the screen using [**Alt+F1**] or the **Fit** button.

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

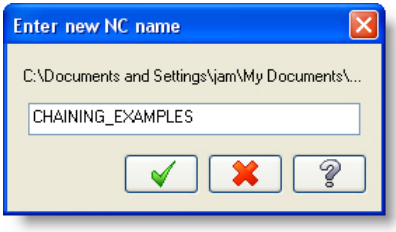
# Exercise 2: Creating a Dynamic Toolpath

Dynamic Area Mill is used to demonstrate chaining and entry options. The same principles apply to the remaining dynamic toolpaths.

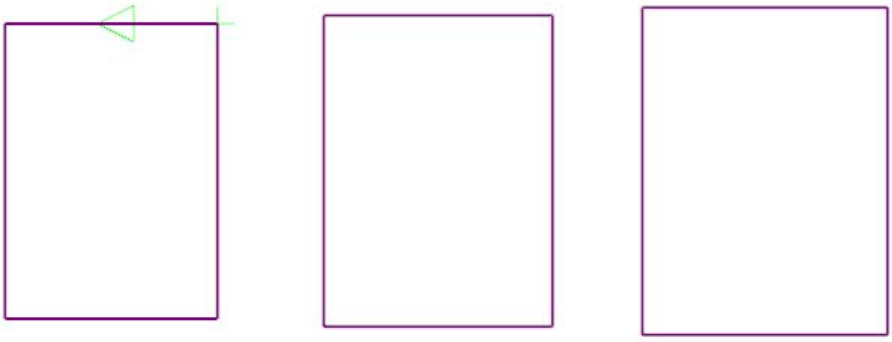
**1** From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Area**.




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

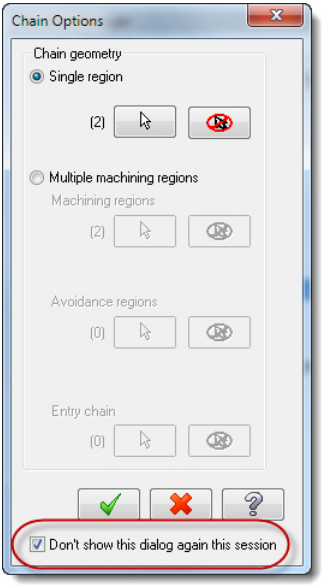



**3** Chain the three rectangles in a counterclockwise direction.



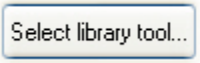
**4** Click **OK** on the Chaining dialog box to go to the Toolpath Type page.

**5** Select **Don't show this dialog again this session** and click **OK**. The 2D High Speed Toolpath dialog box opens.



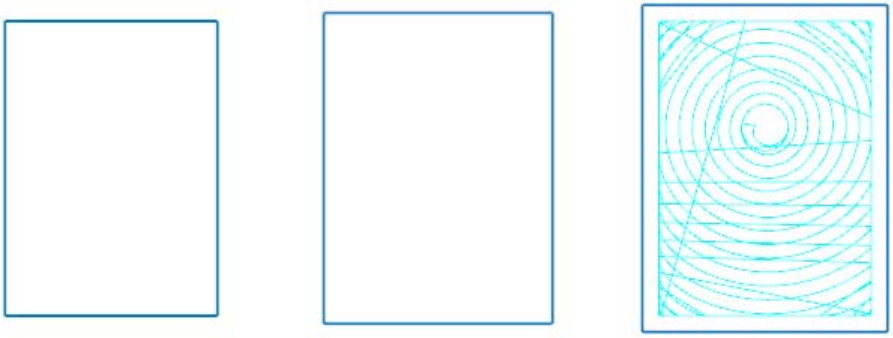

**6** Select **Tool** from the Tree View pane.

**7** Click the **Select library tool** button. The tool library defined in the machine definition opens.




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

**9** Click **OK** to generate the dynamic area mill toolpath on the selected geometry. Observe the toolpath pattern. Backplot the operation if necessary.



**10** Save your part file.

The toolpath is created inside the chain that encompasses the largest 2D area. The dynamic toolpaths treat the largest 2D area as the portion to be machined. All other areas are to be avoided, as stated in the first lesson. The area differences are visually apparent in the selected set of chains. However, the resulting toolpath may be unexpected if you have forgotten how the dynamic toolpaths handle chain selection.

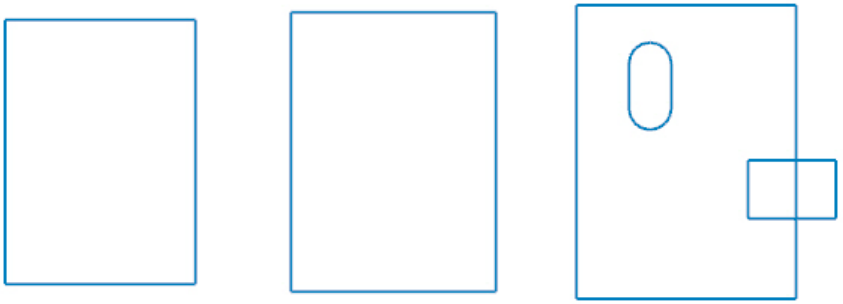
**TIP:** Use **Analyze, Volume/Area, 2D Area** on your selected chains if the toolpath is not being created as expected. The cause will invariably be an incorrect assumption about the enclosed 2D Area of your geometry.

**11** From the Operations 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.

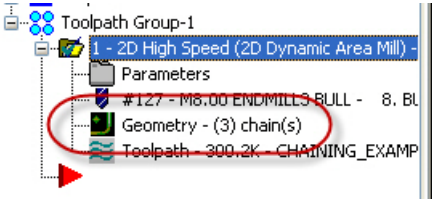
# Exercise 3: Adding Chains to an Operation

**1** Use the Level Manager to make level **2:Avoidance wireframe** visible.

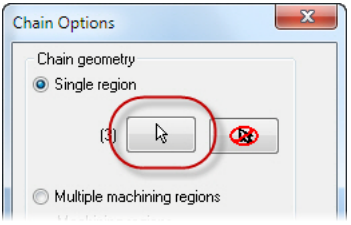


**2** Click **Geometry** under operation 1. The Chain Options dialog box opens.

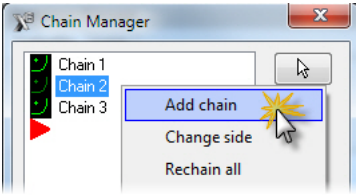
This dialog behaves the same as the first Chain Options dialog box, but must remain visible for rechaining purposes.



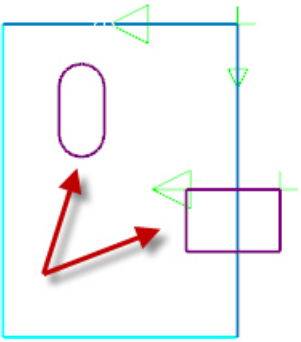

**3** Click **Select** under Single region. The Chain Manager dialog box opens.




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



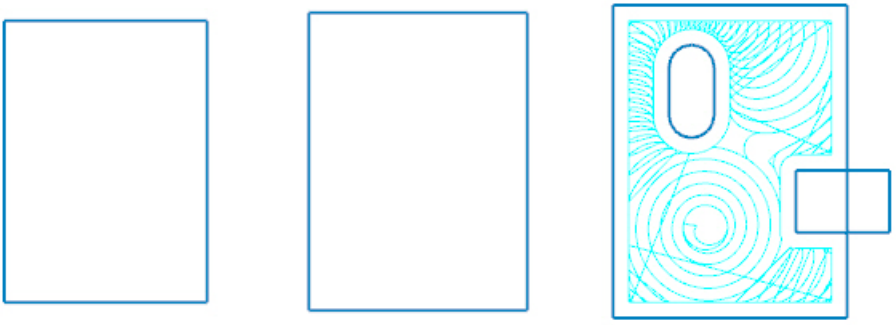
**5** Chain the oval and small rectangle as indicated. Chain direction does not matter.



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

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

**8** 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.*

8 Backplot the operation if necessary.

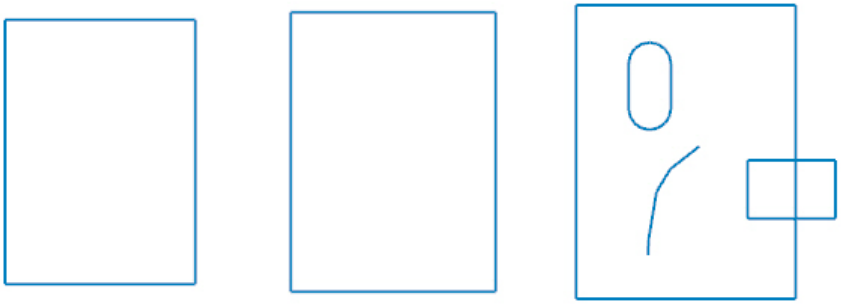
9 Save your part file.

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 toolpaths give 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.

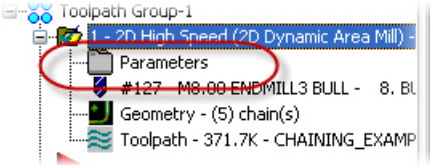
## Exercise 4: Selecting an Entry Method

1 Use the Level Manager to make level **4:Entry curve** visible.



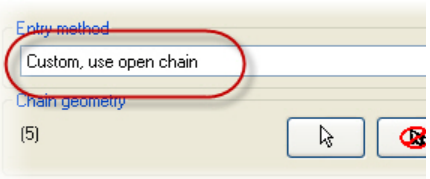
**TIP:** Press [**T**] to toggle the toolpath display in the graphics window.

2 Click **Parameters** under operation 1. The 2D High Speed Toolpath - Dynamic Area Mill dialog box opens.

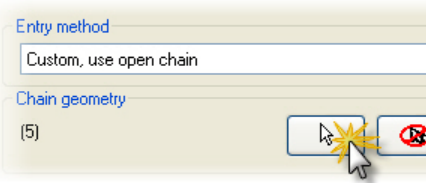


**3** Select **Entry Motion** from the Tree View pane.

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

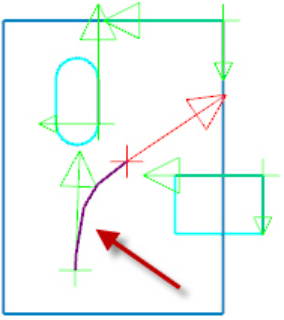



**5** Click **Select** in the Chain geometry group box. The Chain Manager dialog box opens.




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

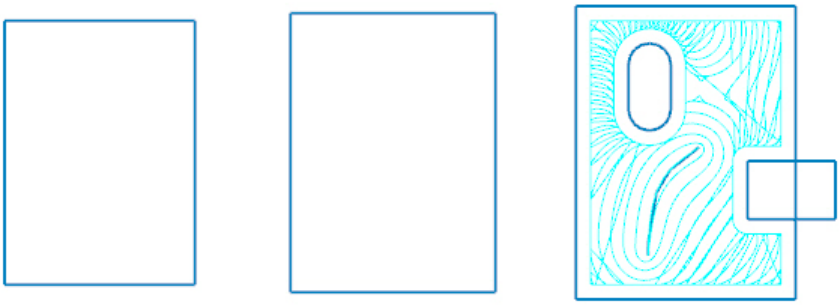
**7** Select the open chain as indicated.



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

**9** Click **Regenerate all dirty operations** in the Toolpath Manager.

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



**11** 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.

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 effect the backplot. When ready, continue on to Lesson 3 for a more in-depth look at a Dynamic Area Mill toolpath.

LESSON 3

# Dynamic Area Mill Toolpath

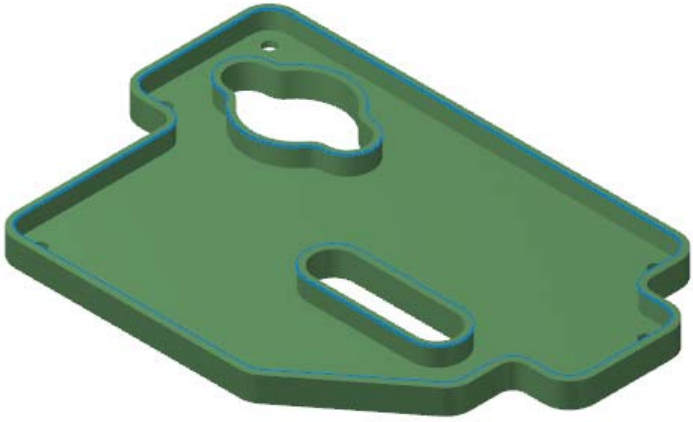
Lesson 2 exposed how chaining is handled for dynamic mill toolpaths, in addition to a brief glimpse of the entry methods. This lesson centers around creating a dynamic area mill toolpath. A pocket toolpath is created for comparison with the dynamic toolpath.

## Lesson Goals

- Create a dynamic area mill toolpath
- Create a pocket toolpath
- Compare the two operations

## Exercise 1: Preparing Mastercam for Toolpath Creation

**1** Open the part file `Dynamic_Area_Mill.MCX-6`, which was provided with the tutorial.



**2** Select the default Mill metric machine definition.

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

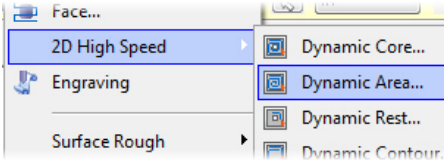
4 Shade the part, if necessary, using [**Alt+S**].

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

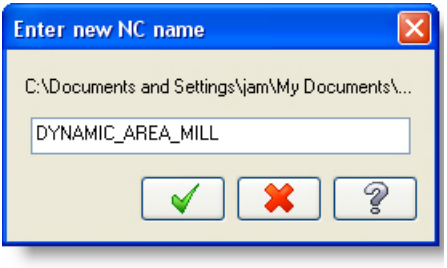
6 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

# *Exercise 2: Creating a Dynamic Area Mill Toolpath*

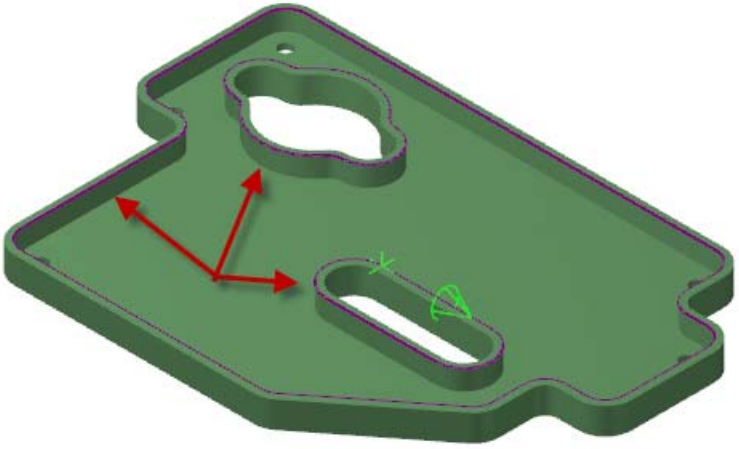
1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Area**.




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



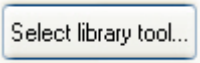
**3** Chain the wireframe geometry at the top of the three walls as indicated.



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

**5** Select **Tool** from the Tree View pane.

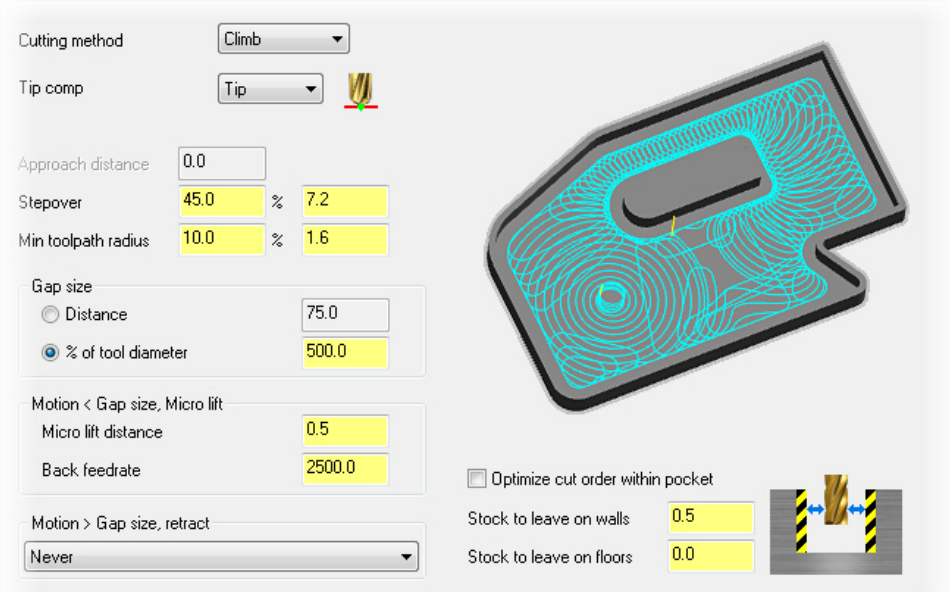
**6** Click the **Select library tool** button. The tool library defined in the machine definition opens.




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

**8** Select **Cut Parameters** from the Tree View pane.

**9** Set the parameters as shown.

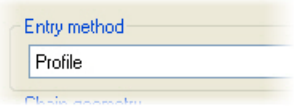



Use micro lift distance to raise the tool above the floor when making a connecting move. The lift creates a period of time where heat is not being added to the tool or the part, extending tool life.

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

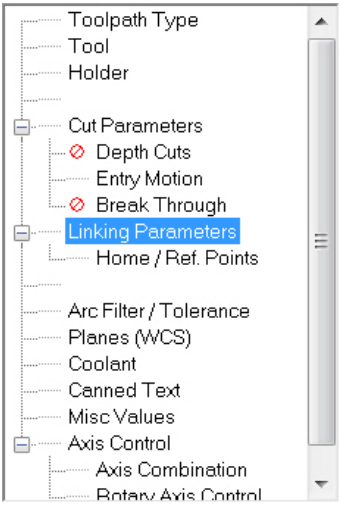
**10** Select **Entry Motion** from the Tree View pane.

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

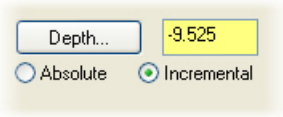


**12** Enter **1.0** mm for Z clearance and **2.0** degrees for the Plunge angle.

**13** Select **Linking Parameters** from the Tree View pane.



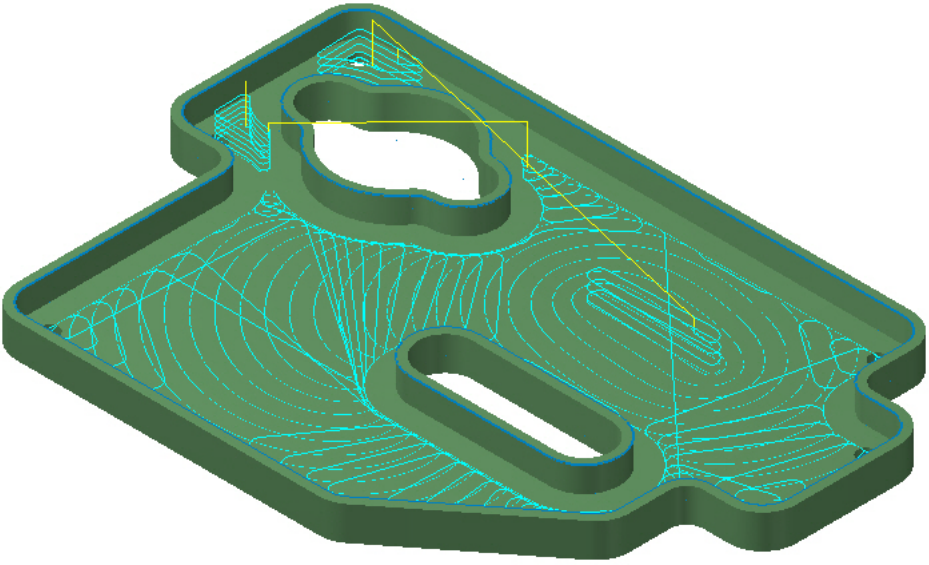

**14** Enter **-9.525** mm for the Depth.



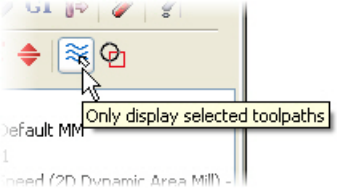

**15** Select **Arc Filter / Tolerance** from the Tree View pane.

**16** Select **2:1** from the Filter ratio drop-down list.

**17** Click **OK** to generate the dynamic area mill toolpath.



**18** Click **Only display selected toolpaths** in the Toolpath Manager.



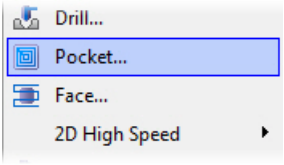

Clicking the button does not affect the current display. The graphics view will be less cluttered when additional operations are added later in this lesson.

**19** Save your part file.

Dynamic area 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**.




2 Click **Last** in the Chaining dialog box.

The chains used in the previous operation are selected. Last is a convenient option when creating multiple operations on the same geometry.

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

4 Select **Cut Parameters** from the Tree View pane.

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.

6 Select **Roughing** from the Tree View pane.

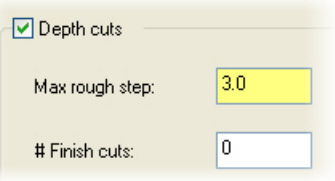
The defaults will be used instead of changing any parameters. The page is selected simply to show the stepover percentage is set to 75%, equalling a stepover distance of 12.0 mm.

7 Select **Entry Motion** from the Tree View pane.

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

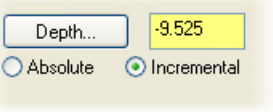
**9** Select **Depth Cuts** from the Tree View pane.

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



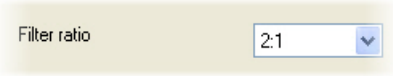

**11** Select **Linking Parameters** from the Tree View pane.

**12** Enter **-9.525** mm for the Depth.

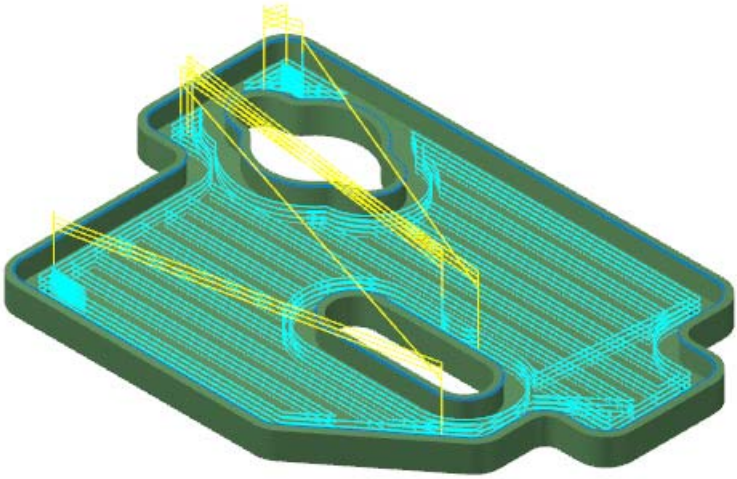



**13** Select **Arc Filter / Tolerance** from the Tree View pane.

**14** Select **2:1** from the Filter ratio drop-down list.




**15** Click **OK** to generate the pocket toolpath.

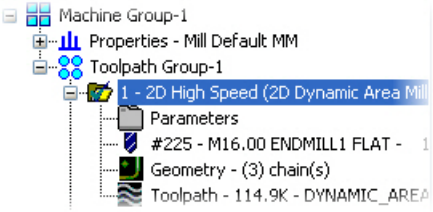


**16** Save your part file.

The same geometry is used to generate the toolpath for dynamic area 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 Toolpath Manager to quickly view the differences.

# Exercise 4: Comparing the Operations

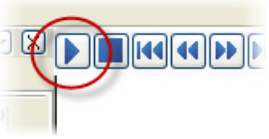
**1** Select operation 1 in the Toolpath Manager.



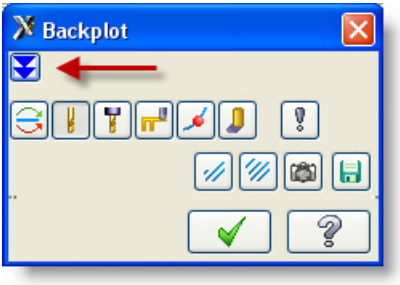

**2** Press **F3** to refresh the graphics display if necessary.

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

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

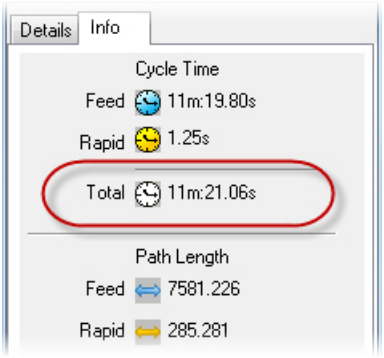


**5** Click **Expand** in the upper left corner of the Backplot dialog box.



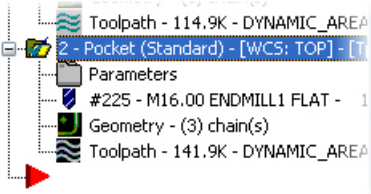
6 Select the **Info** tab in the lower half of the expanded Backplot dialog box.

Valuable information is available on this tab. Of most interest in this exercise is the total runtime. Dynamic Area 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 Toolpath Manager.



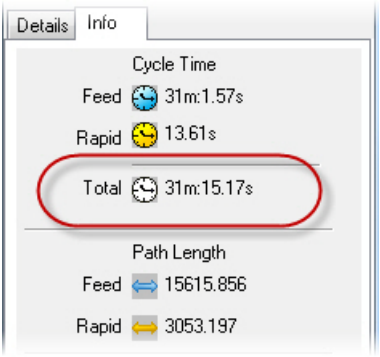

9 Press **F3** to refresh the graphics display if necessary.

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 Area Mill 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 Rest Mill toolpath.

LESSON 4

# Dynamic Rest Mill Toolpath

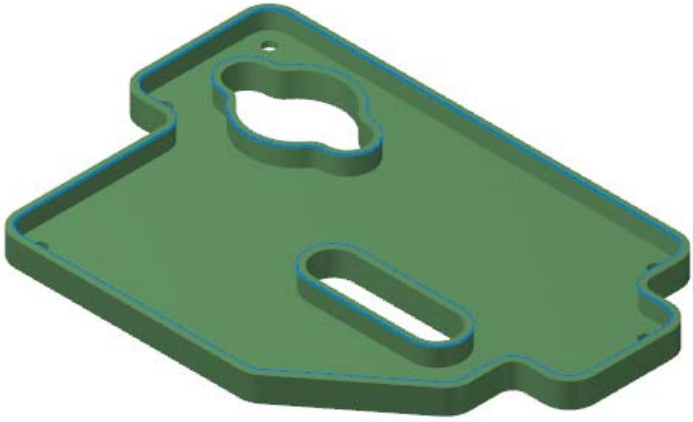
Lesson 3 demonstrated some of the advantages a dynamic area mill toolpath has over a pocket toolpath. This lesson shows how a dynamic rest mill operation improves upon the previous generation of rest mill operation. Each toolpath has uses, however, in areas where you want to avoid tool burial while taking a full depth of cut, the dynamic option is the better choice.

## Lesson Goals

- Create a dynamic rest mill operation
- Create a rest mill operation
- Compare the two operations

## Exercise 1: Preparing Mastercam for Toolpath Creation

1 Open the part file `Dynamic_Rest_Mill.MCX-6`, which was provided with the tutorial. Optionally, you may continue using the file from the end of the previous lesson.



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

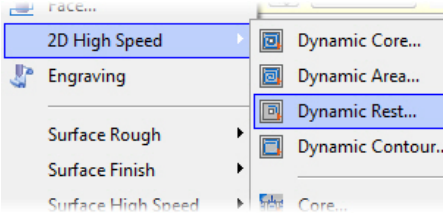
3 Save your part file if you are using the file from the previous lesson.

OR

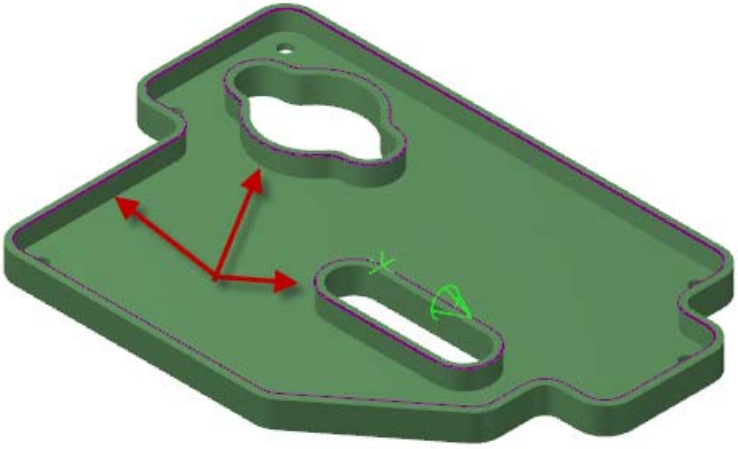
Choose **File, Save As**, and save the part under a different file name, if you opened `Dynamic_Rest_Mill.MCX-6`. (See Step 1)

# Exercise 2: Creating a Dynamic Rest Mill Toolpath

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



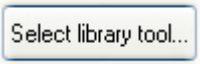

2 Chain the wireframe geometry at the top of the three walls as indicated.



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

4 Select **Tool** from the Tree View pane.

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

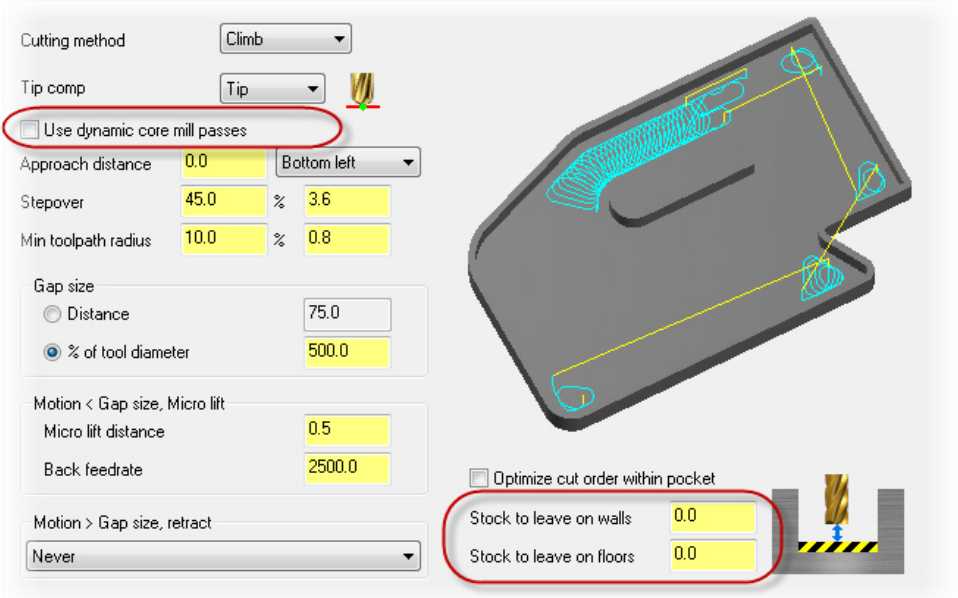



**6** Select the 8mm flat endmill and click **OK**.

**7** Select **Cut Parameters** from the Tree View pane.

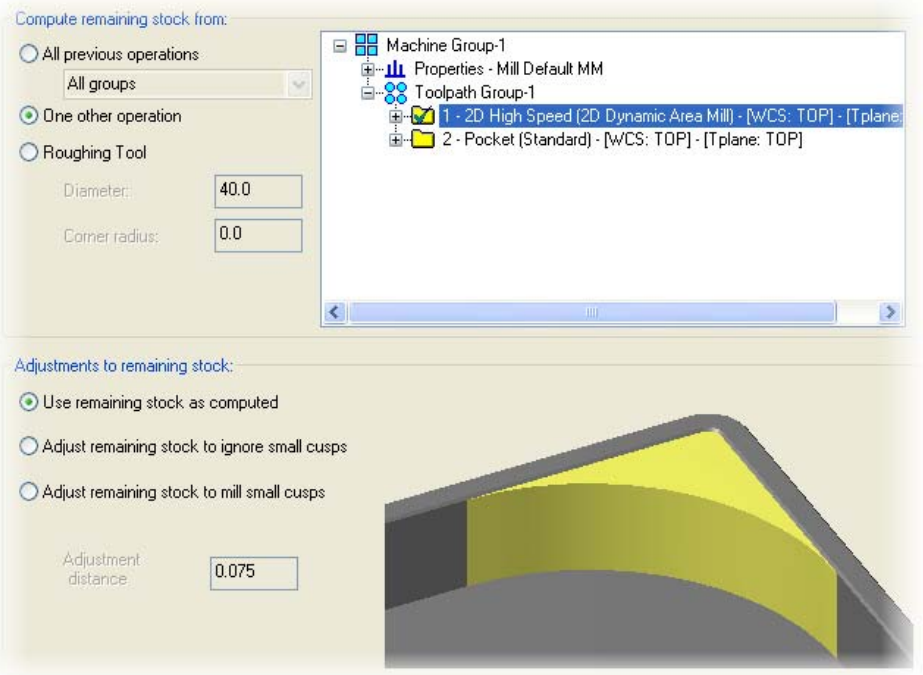
**8** Set the parameters as shown. Be sure to:

- deselect the option to **Use dynamic core mill passes**
- set the **Stock to leave on walls** and **Stock to leave on floors** parameters to **0.0**.

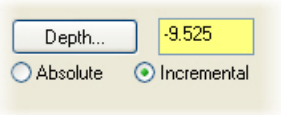



**9** Select **Rest Material** from the Tree View pane.

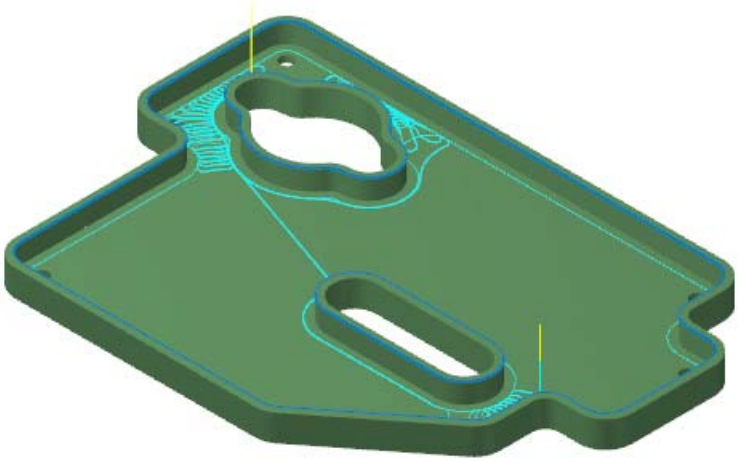
**10** Select **One other operation**, operation **1**, and **Use remaining stock as computed** as shown.



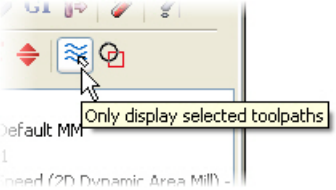

**11** Enter **-9.525** mm for the Depth on the Linking Parameters page.



**12** Click **OK** to generate the dynamic rest mill toolpath.



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




**14** Save your part file.

Dynamic rest mill removes material with a full depth of cut just like dynamic area mill. The same efficient motion is used to safely and quickly remove all remaining material.

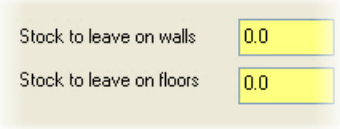
## *Exercise 3: Creating a Rest Mill Toolpath*

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

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

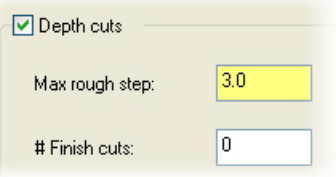
3 Select **Cut Parameters** from the Tree View pane.

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



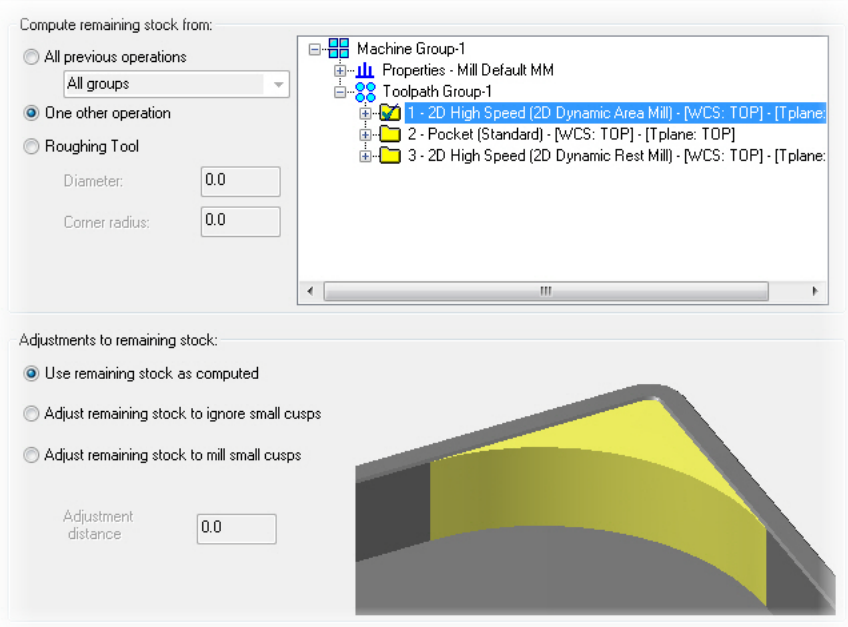

5 Select **Depth Cuts** from the Tree View pane.

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

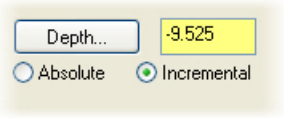



7 Select **Rest Material** from the Tree View pane.

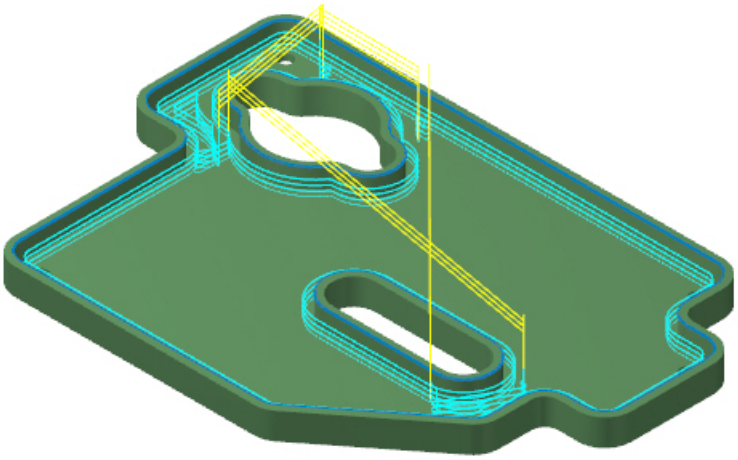
**8** Select **One other operation**, operation **1**, and **Use remaining stock as computed** as shown.




**9** Enter **-9.525** mm for the Depth on the Linking Parameters page.



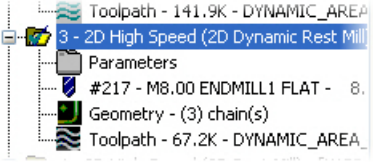
**10** Click **OK** to generate the rest mill toolpath.



**11** Save your part file.

# Exercise 4: Comparing the Operations

**1** Select operation 3 in the Toolpath Manager.




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

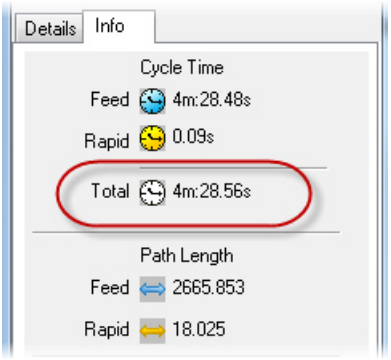
**3** Click **Backplot selected operations** in the Toolpath Manager.

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

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

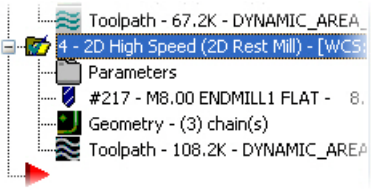
**6** Select the **Info** tab in the lower half of the expanded Backplot dialog box.

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 Toolpath Manager.

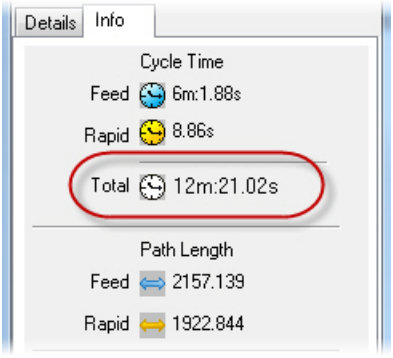



**9** Press **F3** to refresh the graphics display if necessary.

**10** Click **Backplot selected operations**.

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

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



Edit the rest mill and dynamic rest 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 rest mill and dynamic rest mill are offered in Mastercam. When ready, continue on to Lesson 5 to examine a Dynamic Core Mill toolpath.

LESSON 5

# Dynamic Core Mill Toolpath

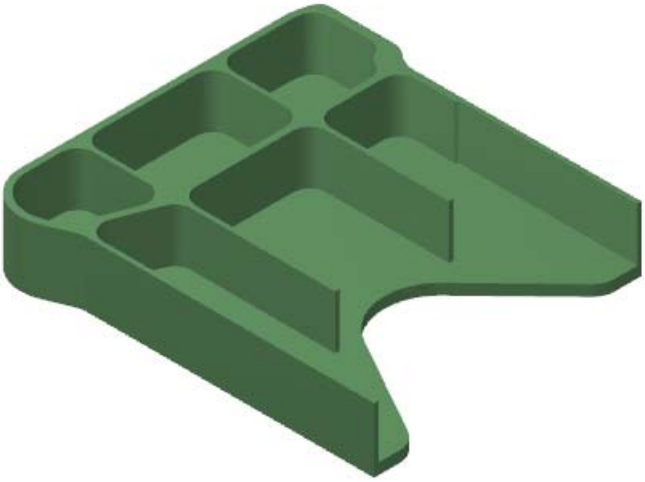
Lesson 4 showed how the dynamic rest mill toolpath is an improvement over the standard 2D HST rest mill toolpath. This lesson compares a dynamic core mill toolpath with a standard pocket toolpath.

## Lesson Goals

- Create a dynamic core mill operation
- Create a pocket operation
- Compare the two operations

## Exercise 1: Preparing Mastercam for Toolpath Creation

**1** Open the part file `Dynamic_Core_Mill.MCX-6`, which was provided with the tutorial.



2 Select the default Mill metric machine definition.

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

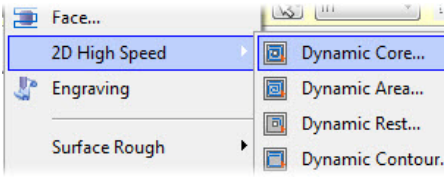
4 Shade the part, if necessary, using [**Alt+S**].

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

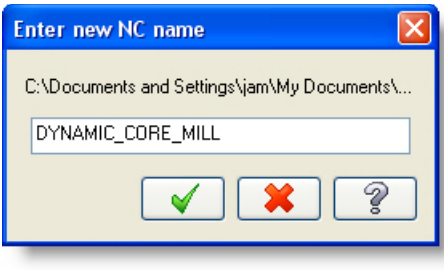
6 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

# Exercise 2: Creating a Dynamic Core Mill Toolpath

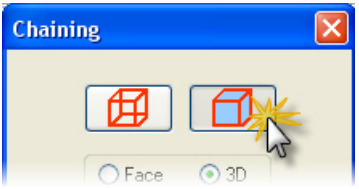
1 From the Mastercam menu, choose **Toolpaths, 2D High Speed, Dynamic Core**.




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

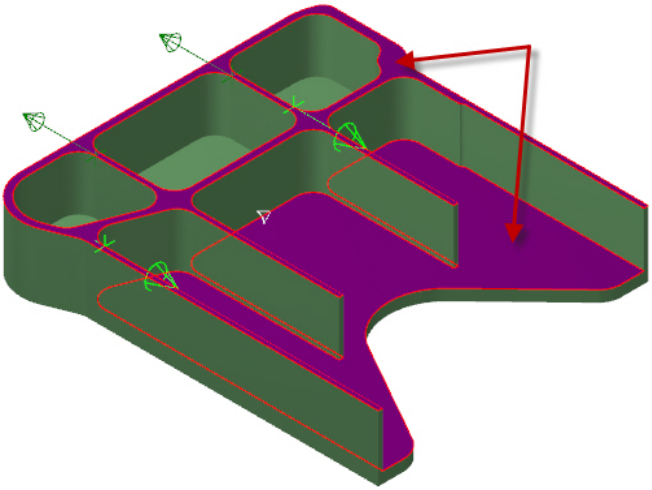



3 Select **Solids** in the Chaining dialog box.



**4** Select the solid faces as shown.

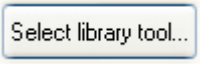
The lower face is used as the area to be machined. The face on top of the walls is used as geometry to be avoided because it has a smaller 2D area.



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

**6** Select **Tool** from the Tree View pane.

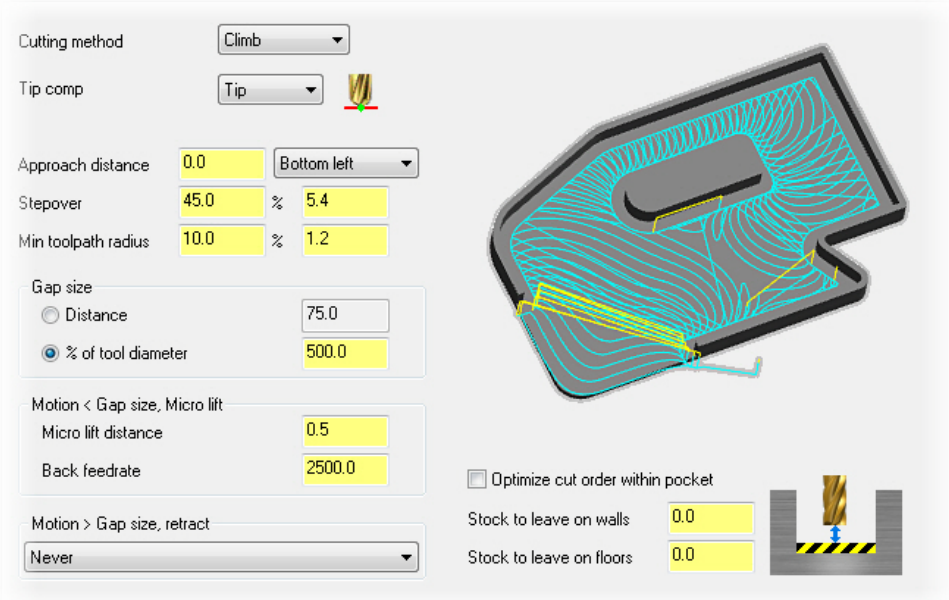
**7** Click the **Select library tool** button.



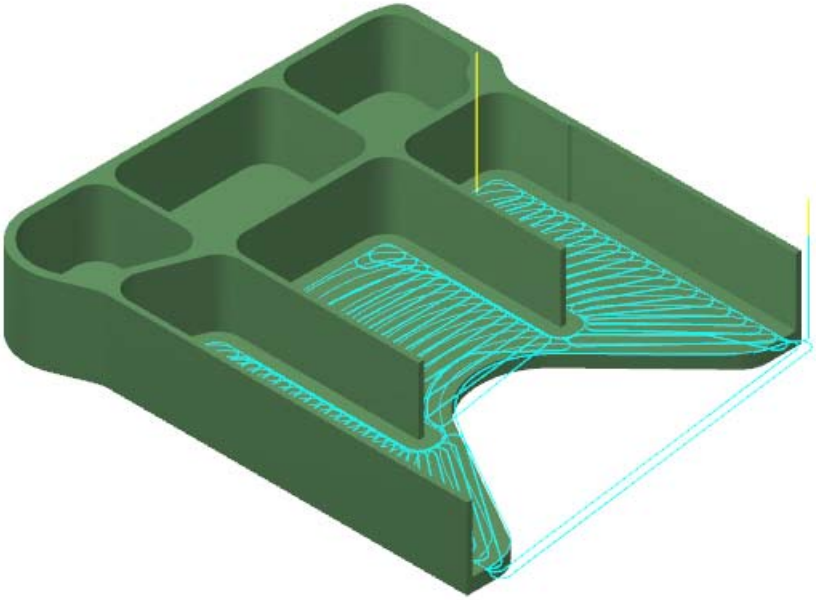

**8** Select the 12mm flat endmill and click **OK**.

**9** Select **Cut Parameters** from the Tree View pane.

**10** Set the parameters as shown.



**11** Click **OK** to generate the dynamic core mill toolpath.



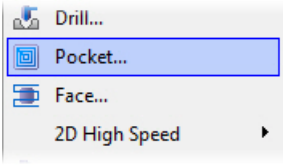
**12** Click **Only display selected toolpaths** in the Toolpath Manager.

**13** Save your part file.

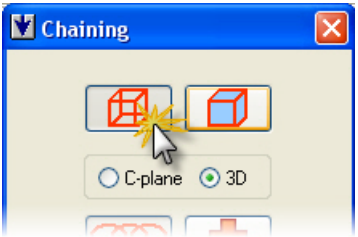
## *Exercise 3: Creating a Pocket Toolpath*

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

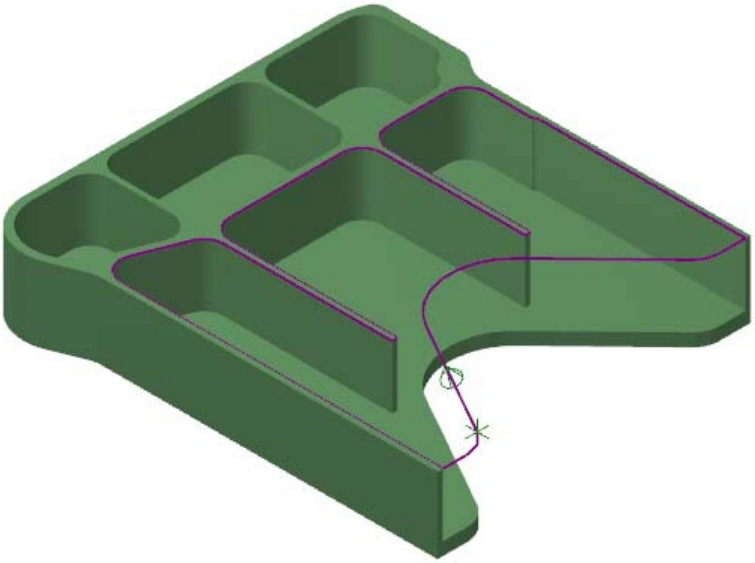
**2** From the Mastercam menu, choose **Toolpaths, Pocket**.



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

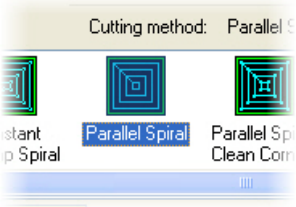



**4** Select the chain as shown and click **OK**.



**5** Select **Roughing** from the Tree View pane.

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

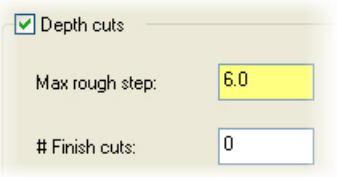



**7** Select **Entry Motion** from the Tree View pane.

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

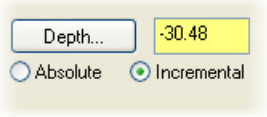
**9** Select **Depth Cuts** from the Tree View pane.

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

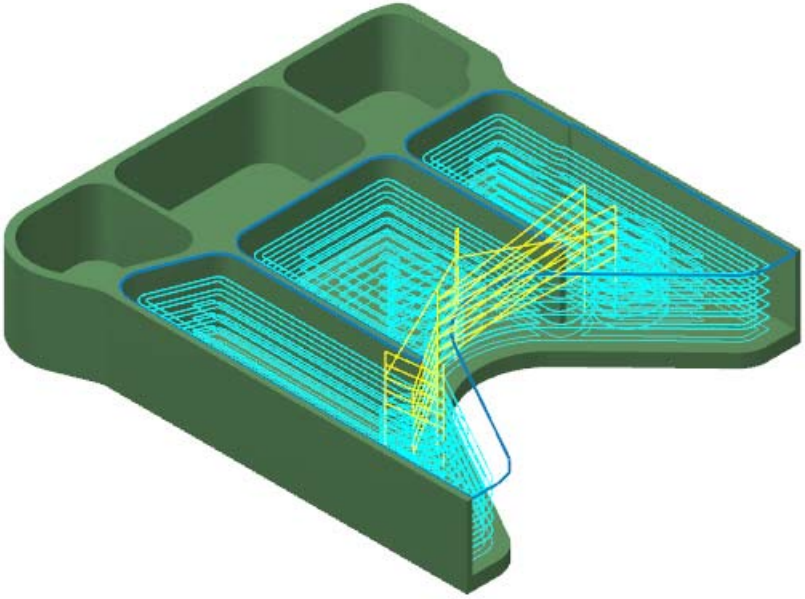



**11** Select **Linking Parameters** from the Tree View pane.

**12** Enter **-30.48** mm for the Depth.



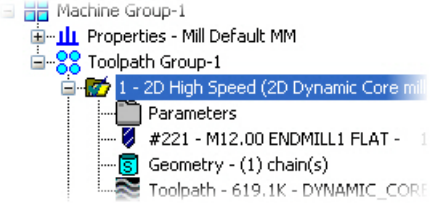
**13** Click **OK** to generate the pocket toolpath.



**14** Save your part file.

## *Exercise 4: Comparing the Operations*

**1** Select operation 1 in the Toolpath Manager.



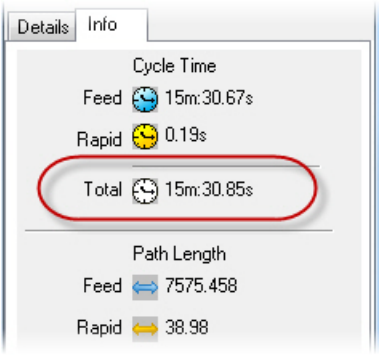

**2** Press **F3** to refresh the graphics display if necessary.

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

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

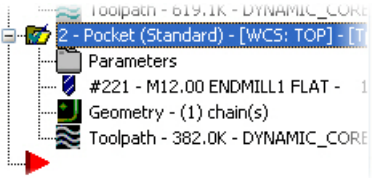
5 Click **Expand** in the upper left corner of the Backplot dialog box.

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




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

8 Select operation 2 in the Toolpath Manager.

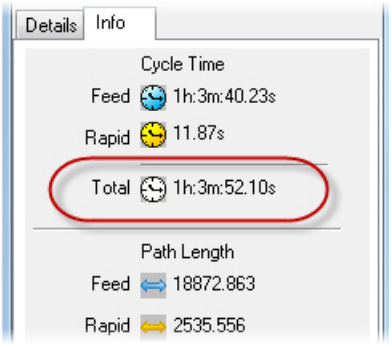



9 Press **F3** to refresh the graphics display if necessary.

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 core 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

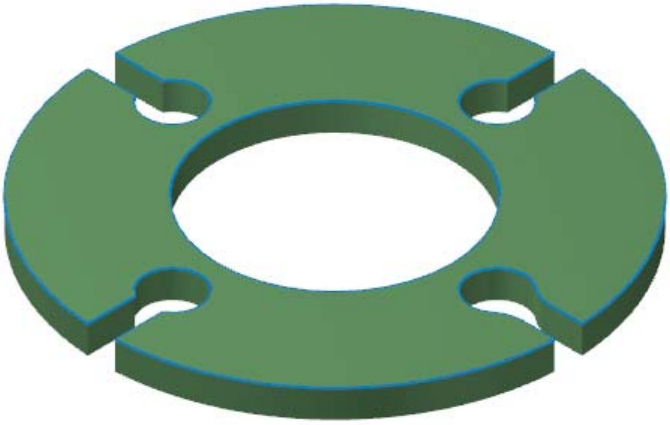
Lesson 5 presented dynamic core mill and pocket as options for milling a sample part. This lesson creates a dynamic contour toolpath in contrast to a standard contour toolpath. The part presents challenging areas where a standard contour toolpath will bury the tool if cutting anything other than a finish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

## Lesson Goals

- Create a dynamic contour operation
- Create a contour operation
- Compare the two operations

## Exercise 1: Preparing Mastercam for Toolpath Creation

**1** Open the part file `Dynamic_Contour.MCX-6`, which was provided with the tutorial.



**2** Select the default Mill metric machine definition.

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

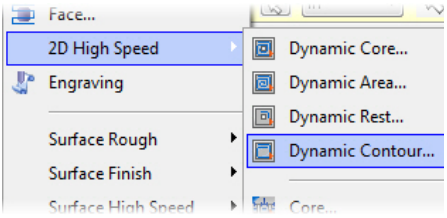
4 Shade the part, if necessary, using [**Alt+S**].

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

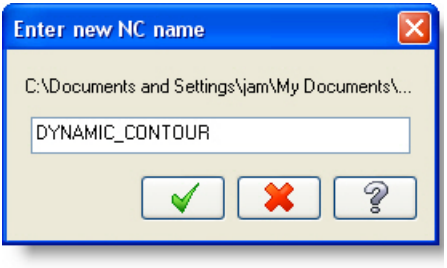
6 Choose **File, Save As**, and save the part under a different file name. This protects the original tutorial file from being overwritten.

# Exercise 2: Creating a Dynamic Contour Toolpath

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

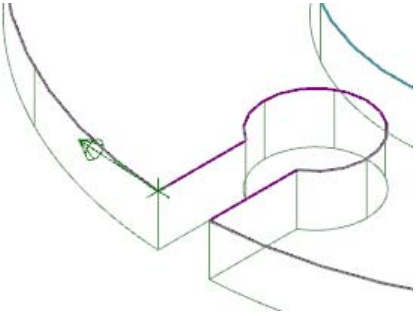



2 Click **OK** if prompted to enter a new NC file name.




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

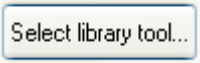
Chain the contour in a clockwise direction. Disable shading as needed to make selection and direction easier.



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

**5** Select **Tool** from the Tree View pane.

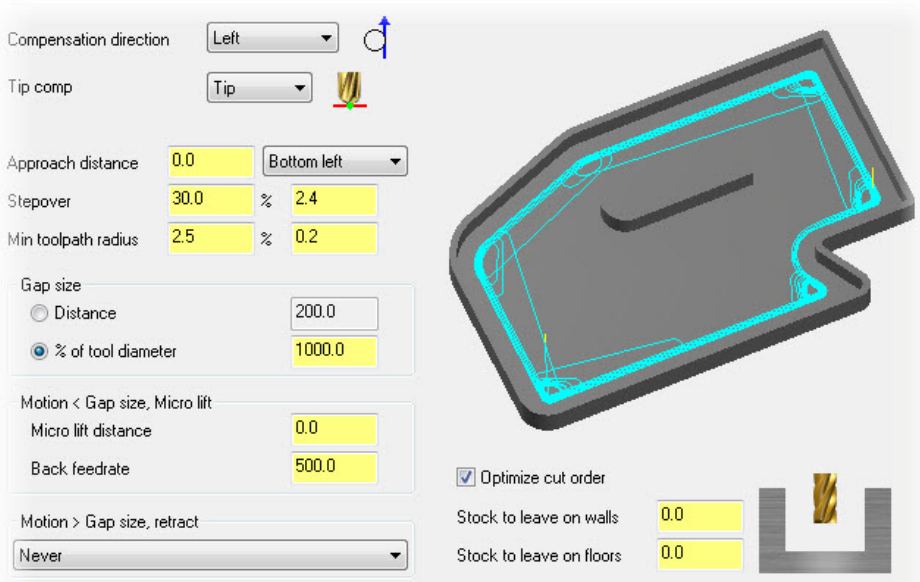
**6** Click the **Select library tool** button.




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

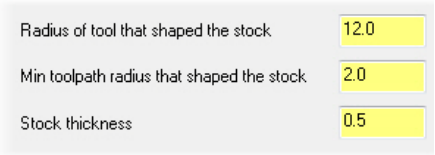
**8** Select **Cut Parameters** from the Tree View pane.

**9** Set the parameters as shown.



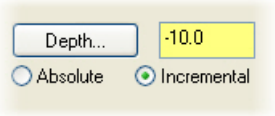

**10** Select **Contour Wall** from the Tree View pane.

**11** Enter the values as shown.

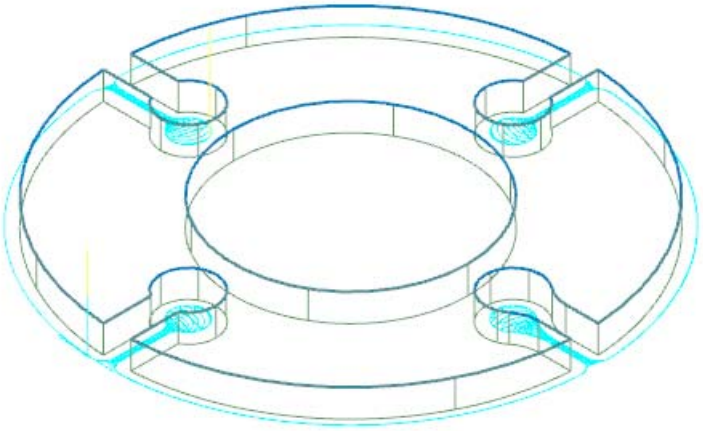



**12** Select **Linking Parameters** from the Tree View pane.

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




**14** Click **OK** to generate the dynamic contour toolpath. Shading has been disabled to make the tool motion more apparent.

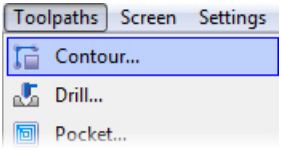


**15** Click **Only display selected toolpaths** in the Toolpath Manager.

**16** Save your part file.

# Exercise 3: Creating a Contour Toolpath

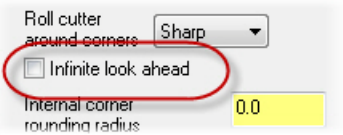
**1** From the Mastercam menu, choose **Toolpaths, Contour**.




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

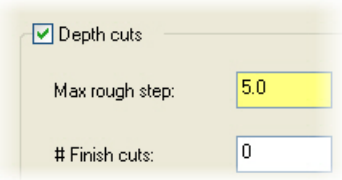
**3** Select **Cut Parameters** from the Tree View pane.

**4** Deselect **Infinite look ahead**.




**5** Select **Depth Cuts** from the Tree View pane.

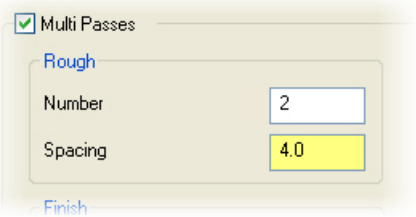
**6** Click the **Depth cuts** checkbox and set the Max rough step to **5.0** mm.




**7** Select **Multi Passes** from the Tree View pane.

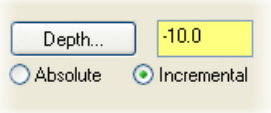
**8** Click the **Multi Passes** checkbox and enter the values shown.

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

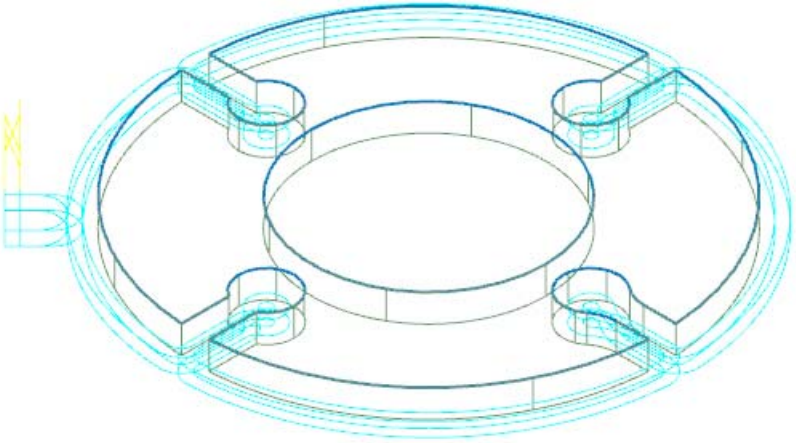



**9** Select **Linking Parameters** from the Tree View pane.

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




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



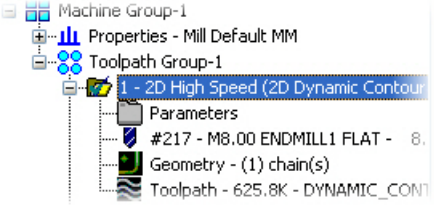
Immediately you can see unnecessary tool motion around the outer edge of the part. The multi passes that are needed to avoid tool burial in the concave sections generate extra passes on the external edges. You can get around this by creating operations to remove material in the concave sections before

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 Toolpath Manager.



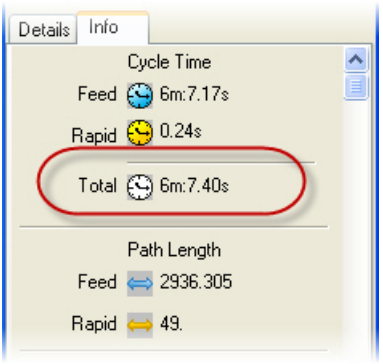

**2** Press **F3** to refresh the graphics display if necessary.

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

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

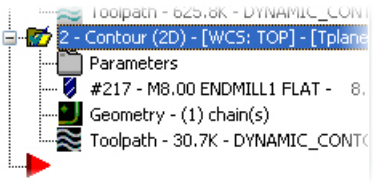
**5** Click **Expand** in the upper left corner of the Backplot dialog box.

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




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

8 Select operation 2 in the Toolpath Manager.

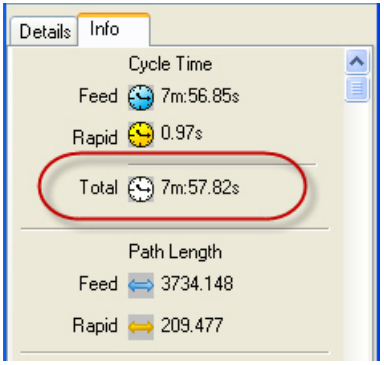



9 Press **F3** to refresh the graphics display if necessary.

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.

# 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. Additional tutorials may be available in this or other series.

This is a module of the *Mastercam Getting Started* Tutorial Series. The series introduces general Mastercam functions and teaches basic skills for getting started with Mastercam. Other tutorial series include:

- *Focus Series*: Focuses on a specific Mastercam feature—for example, Setup Sheets or FBM Drill, and teaches basic and advanced skills.
- *Exploring Series:* Explores a single Mastercam product—for example Mastercam® for SolidWorks® or Mastercam Swiss Expert, and teaches in-depth skills for working with the product.

The Mastercam tutorial series is in continual development, and we will add modules as we complete them. For information and availability or for further training, please contact your local Mastercam Reseller.

# 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 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 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 search for information or ask questions on the Mastercam Web forum, located at www.emastercam.com. You can also find a wealth of information, including many videos, at www.mastercam.com and www.mastercamedu.com.

  For tech tips and the latest Mastercam news, you can join us on Facebook (www.facebook.com/mastercam), follow us on Twitter (www.twitter.com/mastercam), and subscribe to our blog, *Mastercam Xtras* (http://

blog.mastercam.com). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

# Mastercam Documentation

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

- *Mastercam X6 Quick Start*
- *Mastercam X6 Administrator Guide*
- *Mastercam X6 Transition Guide*
- *Mastercam X6 Quick Reference Card*
- *Mastercam X6 File Location Card*
- *Mastercam X6 Post Debugger User's Guide*

## 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

*Printed in the USA on 100% Post Consumer Waste, FSC Stocks*