

Mastercam for SOLIDWORKS X9 Tutorial (Mill)

March 2015

Mastercam X⁹

Mastercam® X9 MCfSW X9 Tutorial (Mill)

Date: March 2015

Copyright © 2015 CNC Software, Inc.— All rights reserved.

Software: Mastercam for SOLIDWORKS X9

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:

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

Be sure you have the latest information!

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

Contents

Introduction	1
▶ Tutorial Goals.....	2
1. General Setup	5
▶ Lesson Goals	5
▶ Exercise 1: Loading Mastercam for SOLIDWORKS	6
▶ Exercise 2: Selecting Your Machine	7
▶ Exercise 3: Defining the Stock	9
2. The Facing Toolpath	13
▶ Lesson Goals	13
▶ Exercise 1: Defining the Tool Plane	13
▶ Exercise 2: Creating the Facing Toolpath	15
▶ Exercise 3: Backplot and Verify the Facing Toolpath.....	19
3. The Dynamic Contour Toolpath	23
▶ Lesson Goals	23
▶ Exercise 1: Creating the Dynamic Contour Toolpath	23
▶ Exercise 2: Backplot and Verify the Dynamic Contour Toolpath.....	29
4. The Dynamic Mill Toolpath	31
▶ Lesson Goals	31
▶ Exercise 1: Creating the Dynamic Mill Toolpath (Stay Inside).....	31
▶ Exercise 2: Creating the Dynamic Mill Toolpath (From Outside)	34
▶ Exercise 3: Backplot and Verify the Dynamic Mill Toolpaths.....	39
5. The Drill Toolpaths	41
▶ Lesson Goals	41

- ▶ Exercise 1: Creating the Drill Toolpaths 41
- ▶ Exercise 2: Backplot and Verify the Drill Toolpath 43

6. Final Verification and Posting 45

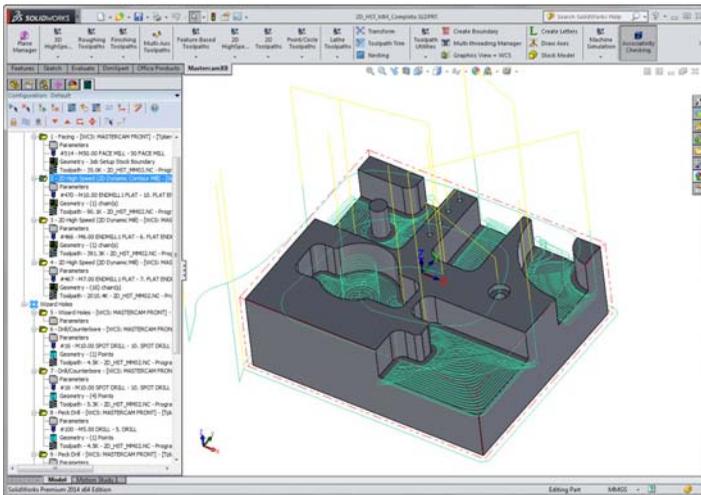
- ▶ Lesson Goals 45
- ▶ Exercise 1: Backplot the Part's Toolpaths 45
- ▶ Exercise 2: Verify the Part's Toolpaths 46
- ▶ Exercise 3: Machine Simulation 47
- ▶ Exercise 4: Post the Part's Toolpaths 50

Conclusion 51

- ▶ Mastercam Resources 51
- ▶ Mastercam for SOLIDWORKS Documentation 52

Introduction

Mastercam for SOLIDWORKS[®] X9 provides Mastercam X9 CAM functionality within SOLIDWORKS, where you can access many toolpath strategies, including the following:



- 3D high speed toolpaths (HST)
- 2D feature based machining (FBM) mill toolpaths
- 2D feature based machining drill toolpaths
- Non-FBM high speed 2D toolpaths
- Multiaxis toolpaths
- Lathe toolpaths

Besides these powerful toolpaths, Mastercam for SOLIDWORKS features many supporting functions, including:

- Stock model
- Boundary creation
- Part setup (including tool and material managers)
- Associativity Checking
- Toolpath transformations
- Backplotting and verification

- Post processing and communications
- Thread management
- Machine Simulation

In this tutorial, you learn many strategies for applying mill toolpaths to parts in Mastercam for SOLIDWORKS.

For this tutorial, Mastercam Simulator is used to Backplot and Verify toolpaths. To enable Mastercam Simulator, select **Backplot** from the Mastercam Advanced Configuration utility and disable **Classic Backplot**.

Tutorial Goals

- Set up a part in Mastercam for SOLIDWORKS.
- Define a tool plane and stock.
- Create a classic 2D toolpath.
- Create 2D High Speed Dynamic toolpaths.
- Create a Wizard Holes toolpath.
- Backplot and verify toolpaths.



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 for SOLIDWORKS tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- You must have a seat of SOLIDWORKS® 2013 or higher to complete this tutorial.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam for SOLIDWORKS workstation, either with the tutorial or in any location that you prefer.

- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel:
www.youtube.com/user/MastercamTechDocs.

LESSON 1

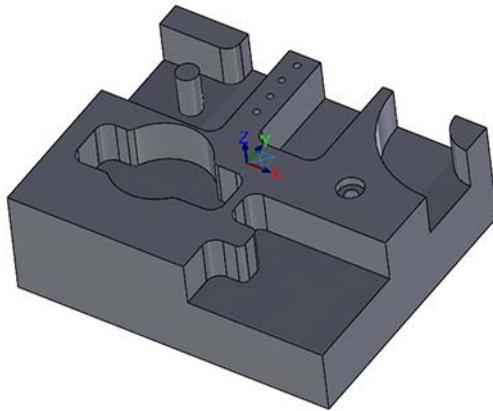
General Setup

Before generating toolpaths for a part, you must prepare Mastercam for SOLIDWORKS and the part file. This preparation includes such tasks as selecting a machine definition and defining the stock.

Lesson Goals

- Explore the part files.
- Select a machine.
- Rename a machine group.
- Set up a stock model.

The lessons in this tutorial are based on the `2D_HST_MM.SLDPRT` file, which is included with the tutorial. The following picture shows the part file as it displays in SOLIDWORKS.



The part files included with the tutorial are:

- `2D_HST_MM.SLDPRT` — The tutorial file for starting Lesson 1
- `2D_HST_MM02.SLDPRT` — The tutorial file for starting Lesson 2
- `2D_HST_MM03.SLDPRT` — The tutorial file for starting Lesson 3
- `2D_HST_MM04.SLDPRT` — The tutorial file for starting Lesson 4
- `2D_HST_MM05.SLDPRT` — The tutorial file for starting Lesson 5
- `2D_HST_MM_Complete.SLDPRT` — The complete tutorial file

Exercise 1: Loading Mastercam for SOLIDWORKS

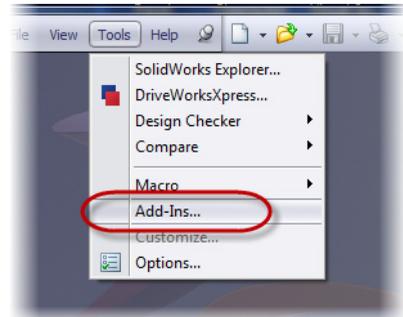
Before you can follow this tutorial, you must verify that the Mastercam for SOLIDWORKS add-in is loaded properly into SOLIDWORKS. The following procedure leads you through this process.

- 1 Start SOLIDWORKS.



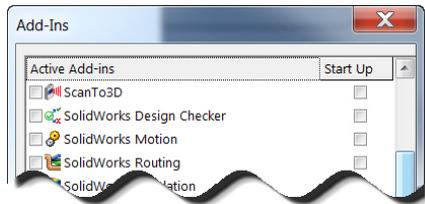
- 2 From the SOLIDWORKS menu bar, choose **Tools, Add-Ins**.

The Add-Ins dialog box displays.

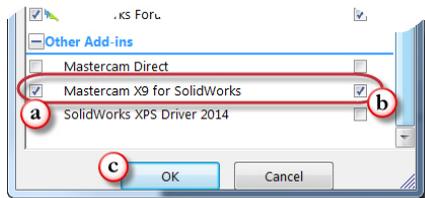


- 3 Find the **Mastercam X9 for SOLIDWORKS** item in the Active Add-ins list, and then do the following:

- a To load the Mastercam for SOLIDWORKS add-in for the current session, select the checkbox to the left of the **Mastercam X9 for SOLIDWORKS** entry.

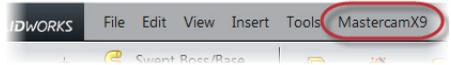


- b To load the Mastercam for SOLIDWORKS add-in at every SOLIDWORKS start-up, select the checkbox to the right of the **Mastercam X9 for SOLIDWORKS** entry.



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

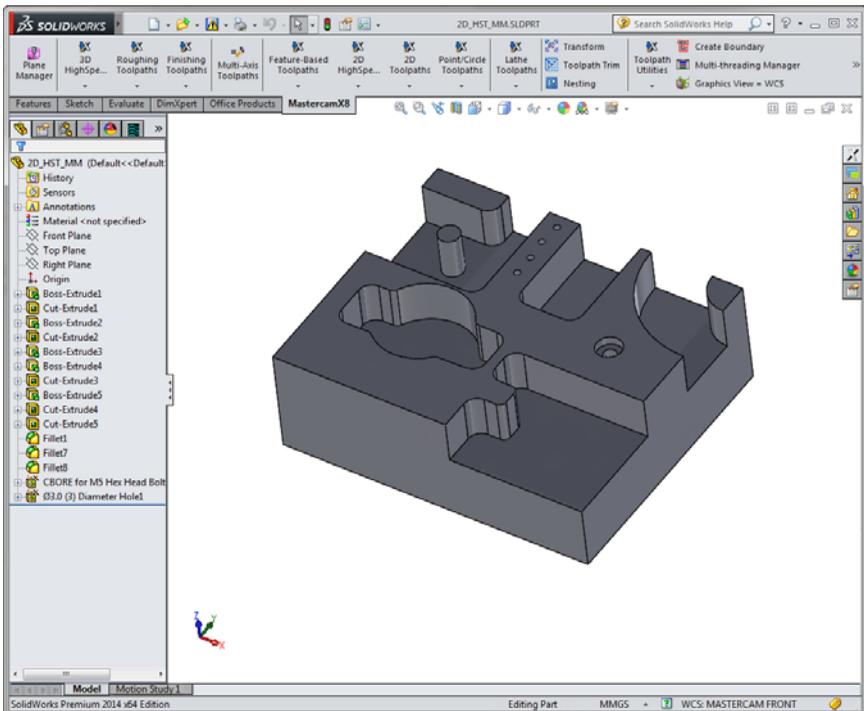
- After the add-in loads, ensure that you have a **MastercamX9** menu in the SOLIDWORKS menu bar.



Exercise 2: Selecting Your Machine

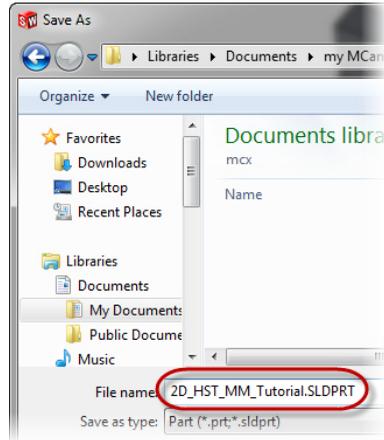
Your part file needs at least one machine group before you can toolpath the part. Mastercam for SOLIDWORKS automatically creates a machine group for each machine you select from the MastercamX9 menu item. To see how machine selection works, complete the following exercise.

- Open the part file 2D_HST_MM.SLDPRJT, which was provided with this tutorial.



- 2 Save the part under the name
2D_HST_MM_Tutorial.SLDPRT.

Saving the part under a new name helps prevent your accidentally modifying the original.



- 3 From the **MastercamX9** menu, choose **Mill Machines**.

Mastercam for SOLIDWORKS displays the available machine definitions. Normally, you would select the machine on which you plan to cut the part. *For this tutorial, however, close the menu without making a choice.*

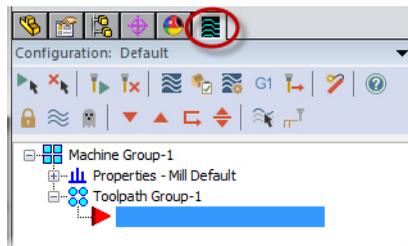


TIP: To customize the Machine Selection lists, choose **MastercamX9, Mill Machines, Manage List** or **MastercamX9, Lathe Machines, Manage List**. For more information on the dialog box that displays, click the dialog box's Help button.



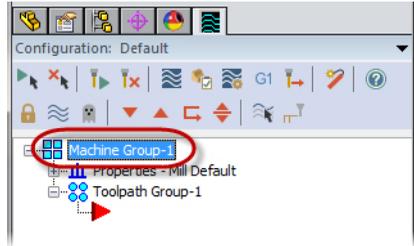
- 4 In the SOLIDWORKS FeatureManager, click the **Mastercam Toolpath Manager** tab to display the Toolpaths Manager.

Notice that this part already has a machine group for the default mill machine.



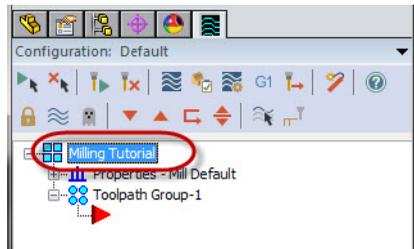
- Right-click the machine group, and select **Groups, Rename** from the pop-up menu.

Mastercam for SOLIDWORKS highlights the current group name.

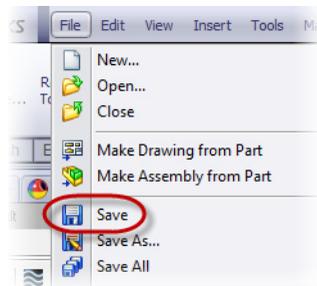


- Type a new machine group name.

The machine group name can be anything you want, but it's best to choose a name that describes the machine and its operations. For this tutorial, name the machine group **Milling Tutorial**.



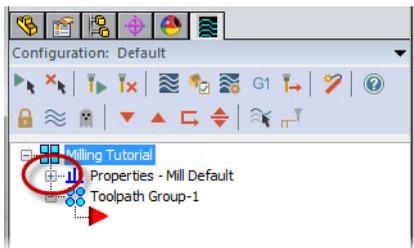
- Select **File, Save** to save the file.



Exercise 3: Defining the Stock

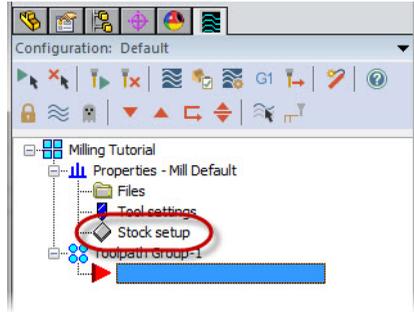
- In the Toolpaths Manager, click the plus sign next to **Properties**.

The Properties entry expands to show more information.



- 2 Click the **Stock setup** entry.

The Stock Setup page of the Machine Group Properties dialog box displays.



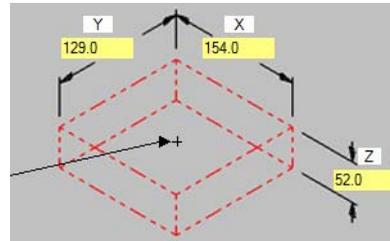
- 3 In the dialog box, click the **All Entities** button.

Mastercam for SOLIDWORKS fills in the **X**, **Y**, and **Z** extents of the required stock.



- 4 Add **4 mm** to the **X** and **Y**, and **2 mm** to the **Z**, giving **154**, **129**, and **52**, respectively.

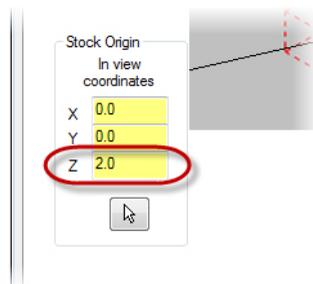
These values add extra stock around the part.



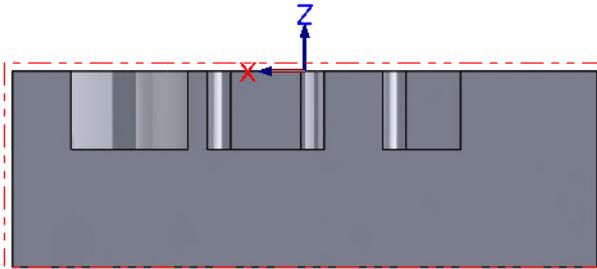
- 5 Change the Stock Origin's **Z** to **2.0**.

This value moves the stock up so that all of the extra Z stock is above the tool plane.

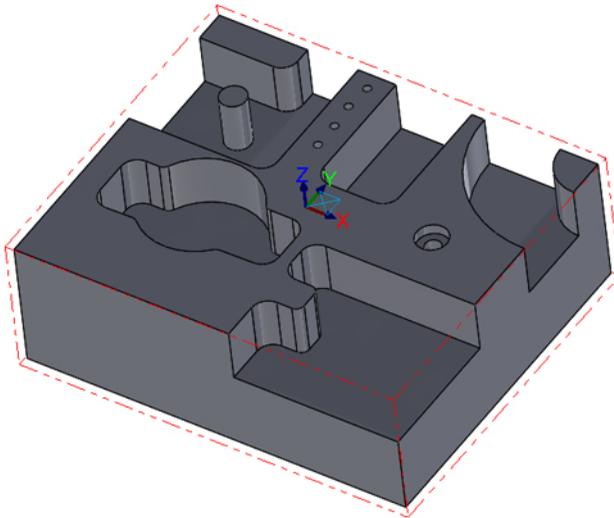
- 6 Click **OK** to exit the dialog box and create the stock.



You now have 2 mm of extra stock all around the part, except for the bottom.



The stock displays around your part as a dashed red line.



7 Save your work.

LESSON 2

The Facing Toolpath

Your part file, `2D_HST_MM_Tutorial.SLDPRT`, now has a machine assigned (in this case, just the default mill) and stock specified. In this lesson, you will add a facing operation to the part file.

Lesson Goals

- Define the part's tool plane.
- Add the 2D Facing toolpath.
- Backplot the toolpath.
- Verify the toolpath.

Exercise 1: Defining the Tool Plane

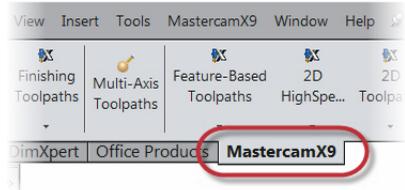
- 1 If necessary, open the `2D_HST_MM_Tutorial.SLDPRT` file that you created in the previous lesson.



TIP: You can also begin this lesson by loading the file `2D_HST_MM02.SLDPRT`, which is provided with this tutorial.

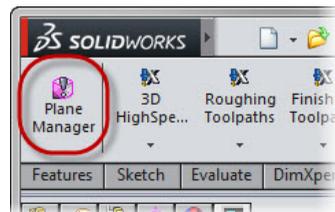
- 2 In the CommandManager, click the **MastercamX9** tab.

The CommandManager displays the Mastercam for SOLIDWORKS commands.

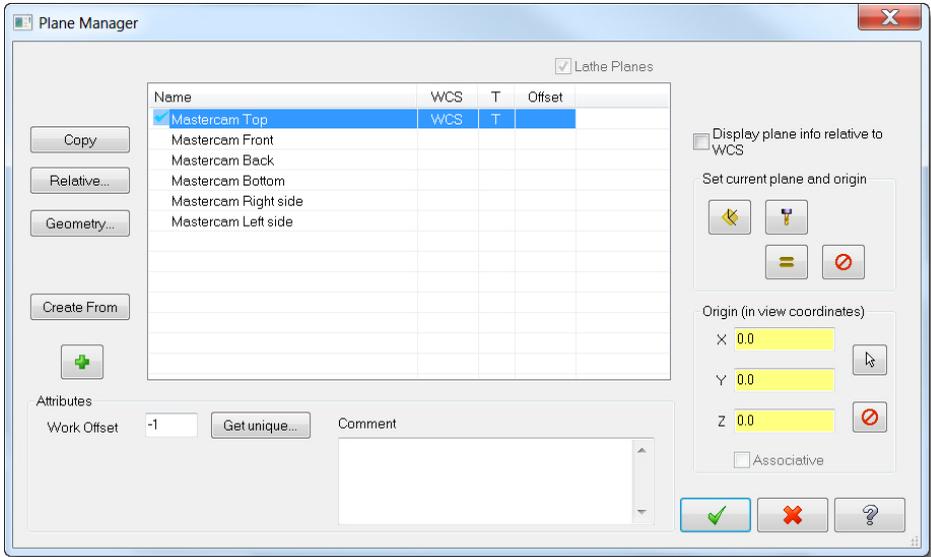


- 3 Click the **Plane Manager** button.

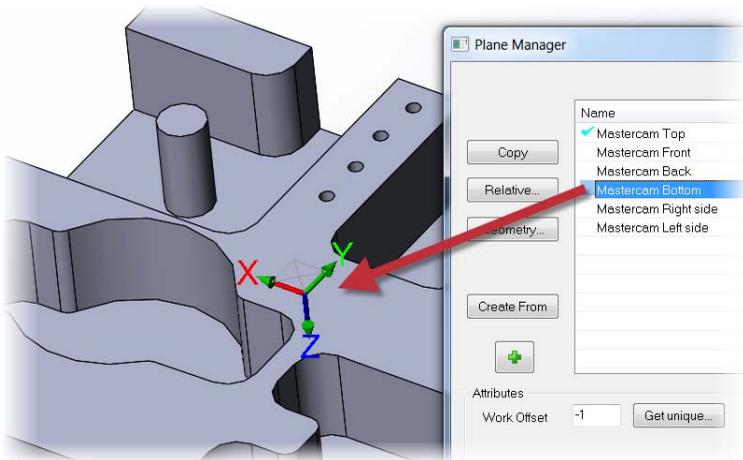
The Plane Manager dialog box displays.



Notice that Mastercam for SOLIDWORKS automatically creates six standard views.



- 4 Click each view and watch the gnomon in the graphics area change to show the selected view's orientation relative to the part.



- 5 Notice the **T** to the right of the **Mastercam Top** plane. The T indicates the currently set tool plane. In this example, this is the tool plane you will use.

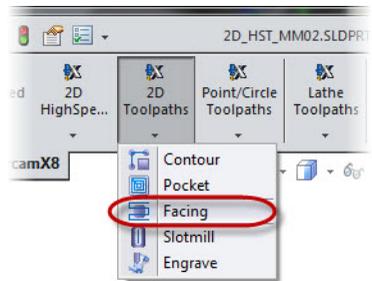
Name	WCS	T	C
Mastercam Top	WCS	T	
Mastercam Front			
Mastercam Back			
Mastercam Bottom			
Mastercam Right side			
Mastercam Left side			

- 6 Click **Cancel** to close the Plane Manager *without making changes*.

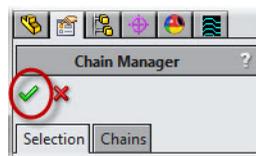
Exercise 2: Creating the Facing Toolpath

- 1 In the MastercamX9 CommandManager, click **2D Toolpaths**, and select **Facing**.

The Chain Manager displays, docked on the left edge of the SOLIDWORKS window.

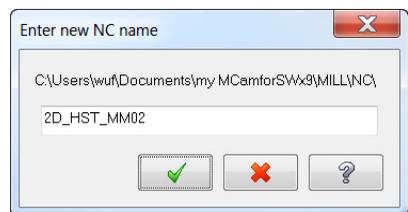


- 2 Click **OK** in the Chain Manager to chain based on the stock you defined.

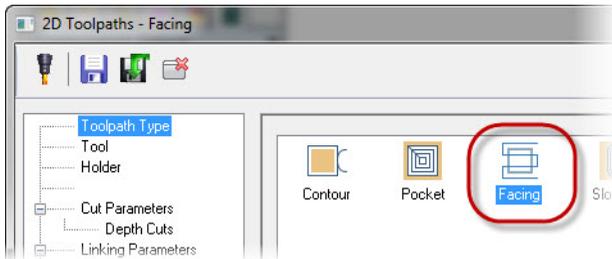


- 3 Click **OK** to accept the default name for the part's NC file if the **Enter new NC name** dialog box displays.

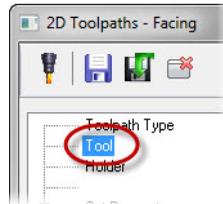
The 2D Toolpaths - Facing dialog box displays.



Notice that the dialog box starts off with the Facing toolpath selected.

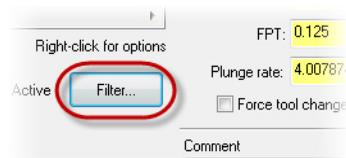


4 Select the **Tool** page.



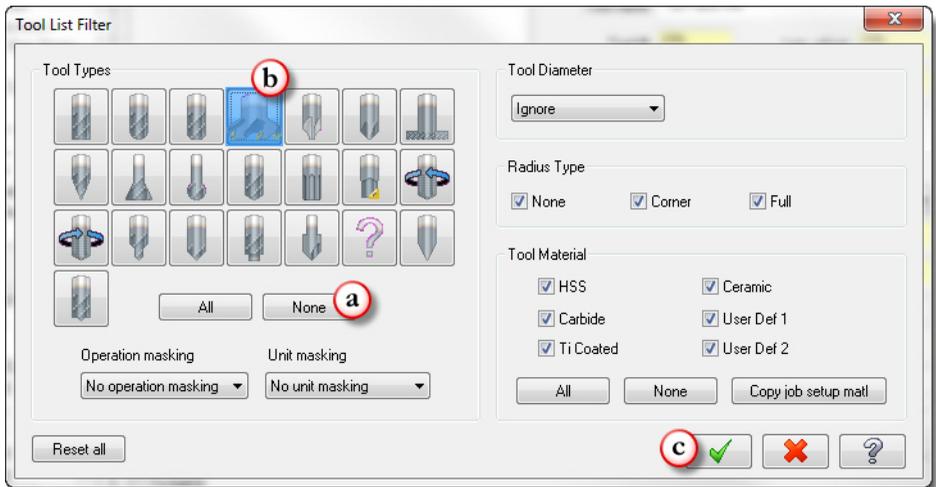
5 Click the **Filter** button.

The Tool List Filter dialog box displays.



6 In the dialog box, do the following:

- a In the Tool Types group, click **None**.
- b Select the Face mill icon. This will filter your tool list to only display Face mill tools.
- c Click **OK**.



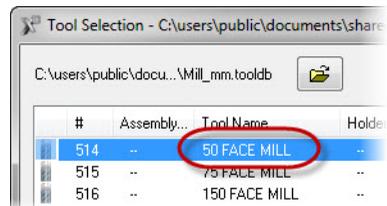
7 Click **Select library tool**.

The Tool Selection dialog box displays.

Select library tool...

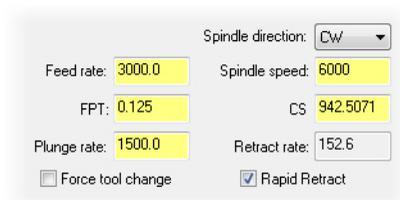
8 In the tool list, click the **50 mm Face Mill**, and click **OK**.

The selected tool displays in the tool list of the **Tool** page.



9 In the **Tool** page, set the following options:

- a** Set **Feed rate** to **3000**.
- b** Set **Plunge rate** to **1500**.
- c** Set **Spindle speed** to **6000**.
- d** If it's not already selected, turn on **Rapid Retract**.

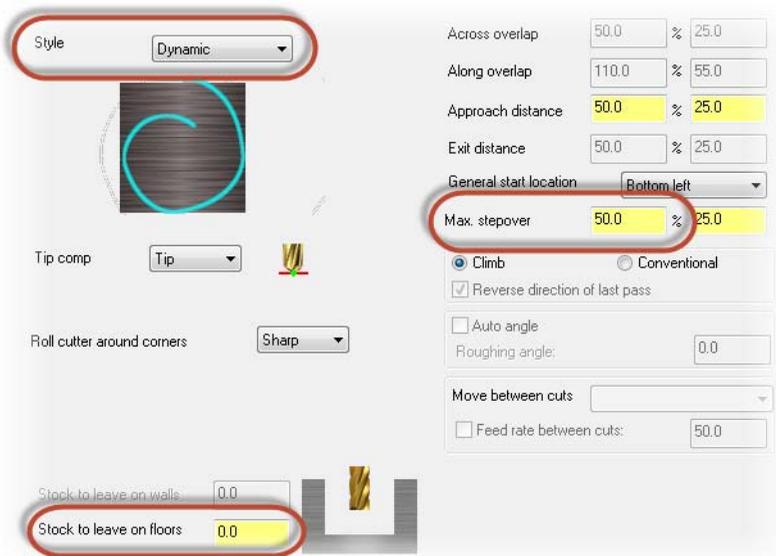


10 Select the **Cut Parameters** page, and then set the following options:

- a** Set the **Style** drop-down to **Dynamic**, as shown in the image below.

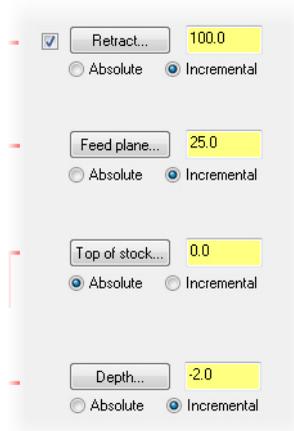
- b Set the percentage for **Max. stepover** to **50%**.
- c Set the **Stock to leave on floors** to **0.0**.

The Dynamic style creates a smooth controlled motion that cuts from the outside to the inside, while maintaining a constant load on the tool with minimal entries and exits.



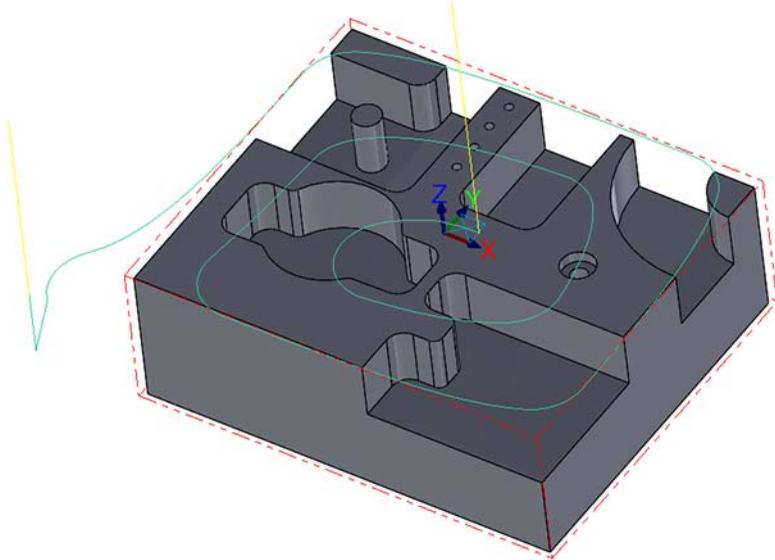
11 Select the **Linking Parameters** page, and then set the following options:

- a Set **Retract** to **100.0**.
- b Set **Feed plane** to **25.0**.
- c Set **Top of stock** to **0.0**.
- d Set **Depth** to **-2.0**.
- e Ensure that all values are set to **Incremental**, except **Top of Stock**, which should be **Absolute**.



- 12 Click **OK** to finalize your choices.

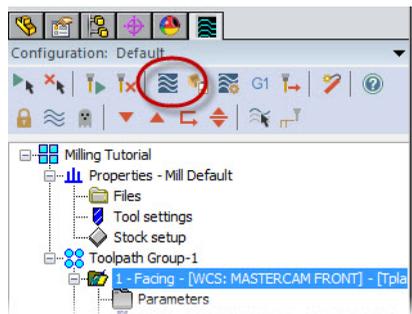
Mastercam for SOLIDWORKS creates the toolpath shown in the following picture:

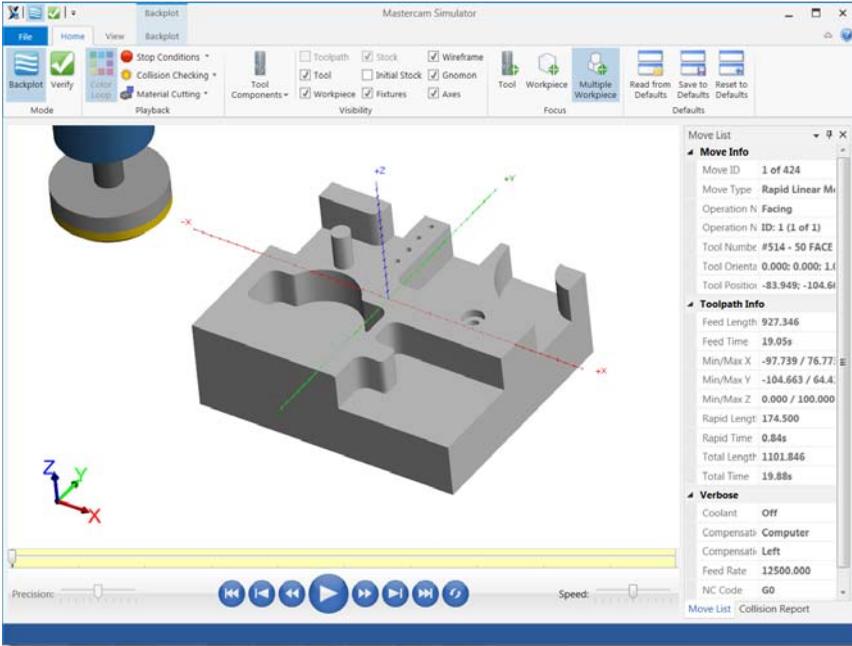


Exercise 3: Backplot and Verify the Facing Toolpath

- 1 In the Toolpaths Manager, click the **Backplot selected operations** button.

Mastercam Simulator opens in a new window.





- If it's not yet selected, check the **Workpiece** box.

Mastercam Simulator displays the part's final shape.



- Click the **Play** button to watch Mastercam Simulator backplot the toolpath.



TIP: Use the slider to adjust the backplot speed.

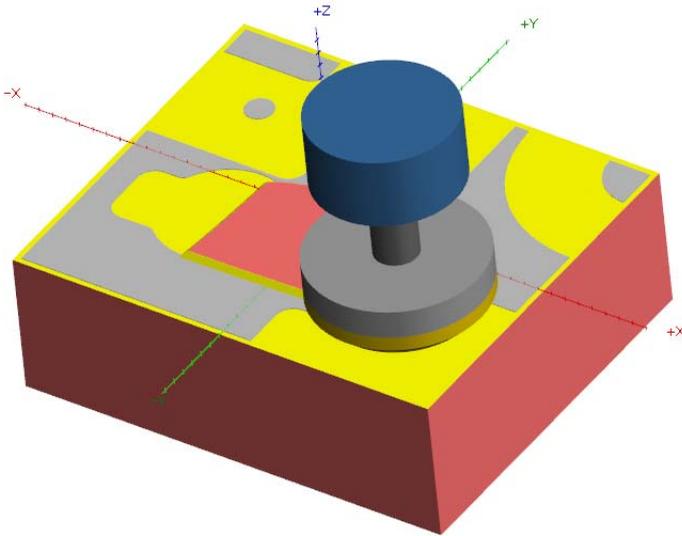
- When Backplot finishes, click the **Verify** button on the Home ribbon bar.



- 5 Click the **Play** button to start Verify.



Mastercam Simulator shows the cutting operation, as shown in the picture below.



- 6 Close the Mastercam Simulator window, and then save your work.

LESSON 3

The Dynamic Contour Toolpath

The next step in toolpathing your part is to contour the sides to their specified dimension. In this lesson, you will add a Dynamic Contour operation to the part file.

Lesson Goals

- Add the 2D Dynamic Contour toolpath.
- Backplot the toolpath.
- Verify the toolpath.

Exercise 1: Creating the Dynamic Contour Toolpath

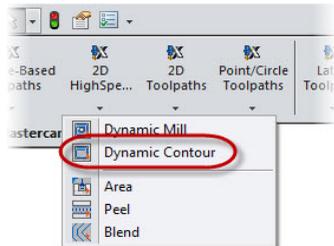
- 1 If necessary, open the `2D_HST_MM_Tutorial.SLDPRT` file that you worked on in the previous lesson.



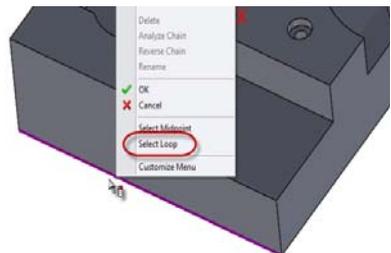
TIP: You can also begin this lesson by loading the file `2D_HST_MM03.SLDPRT`, which is provided with this tutorial.

- 2 In the MastercamX9 CommandManager, click **2D HighSpeed Toolpaths**, and select **Dynamic Contour**.

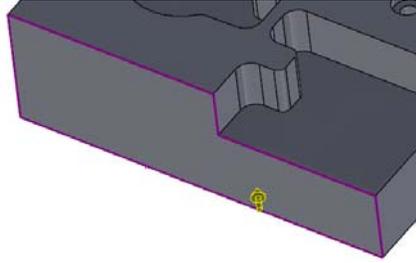
The Chain Manager dialog box displays.



- 3 Right-click the bottom edge of the part, and choose **Select Loop** from the pop-up menu.

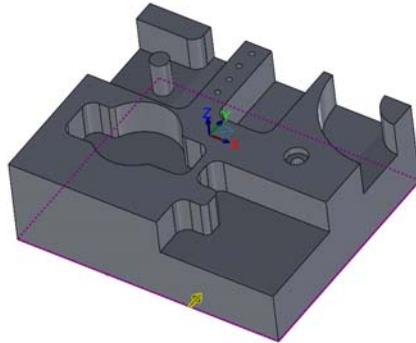


SOLIDWORKS chains the edges of the side face.

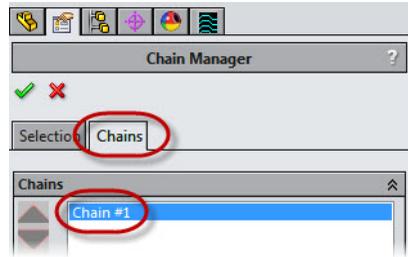


- 4 Click the yellow arrow to switch the chain from the side face to the bottom of the part.

SOLIDWORKS chains the bottom of the part.



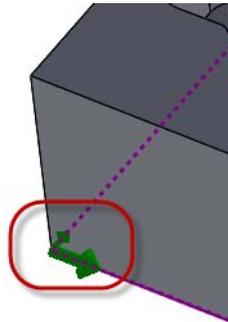
- 5 Click the **Chains** tab, and select **Chain #1**.



Two green arrows display.

- The larger arrow indicates cut direction.
- The smaller arrow shows the cutter compensation.

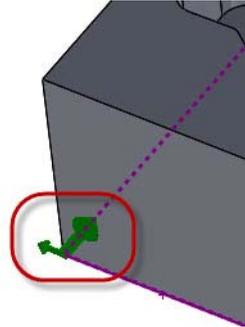
In this case, the small arrow points inwards, which is the wrong direction.



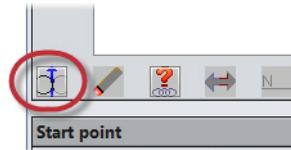
- 6 To change the both the cut direction and the compensation direction of your selection, click the **Reverse Chains** button.



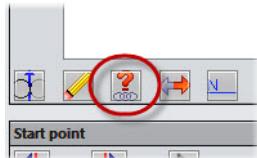
The green chaining arrow changes orientation.



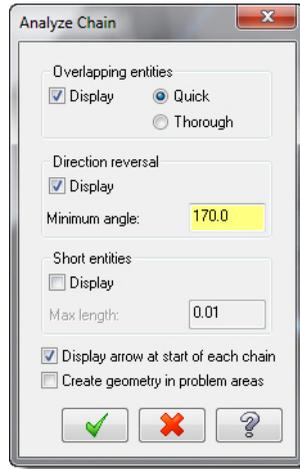
TIP: Click the **Change sides** button to reverse only the compensation direction of all chains in an operation.



- 7 Click the **Analyze Chain** button.

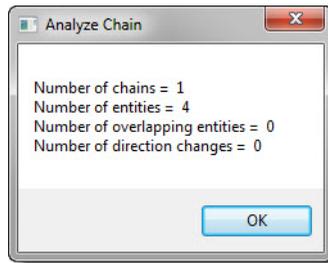


The Analyze Chain dialog box displays. You can use this dialog box to select problems—such as overlapping entities, direction reversals, and short entities—that you want Mastercam to look for in selected chains.



- 8 Click the **OK** button.

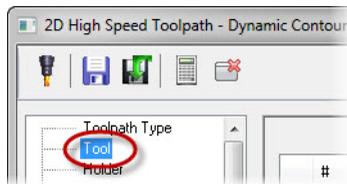
Mastercam for SOLIDWORKS displays information about the selected chain.



- 9 Click **OK** to exit the Analyze Chain dialog box, and click **OK** in the Chain Manager to finalize your selections.

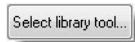
The 2D High Speed Toolpath - Dynamic Contour dialog box displays.

- 10 Select the **Tool** page.



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

The Tool Selection dialog box displays.

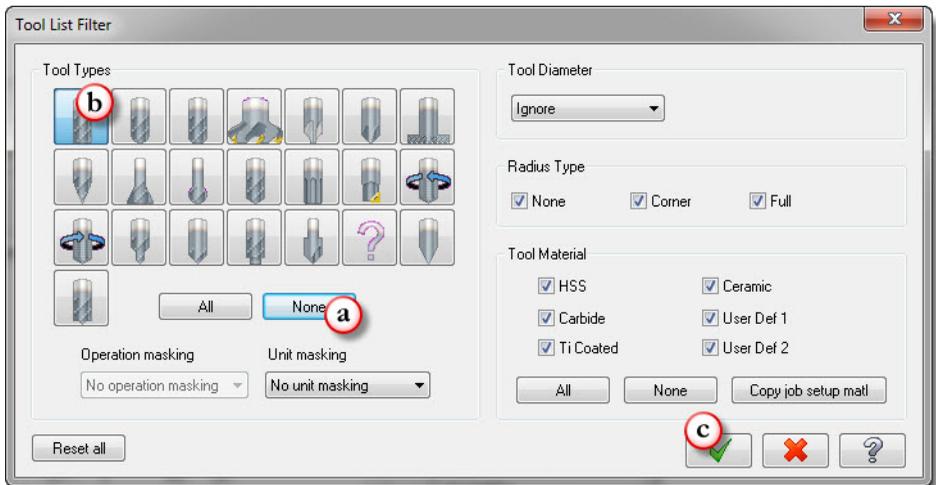


- 12 Click the **Filter** button.



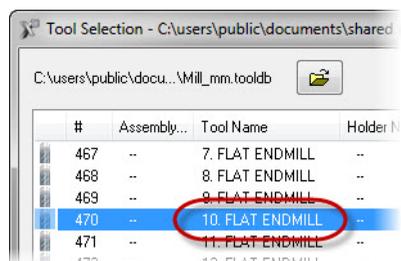
The Tool List Filter dialog box displays.

- 13 In the dialog box, do the following:
- In the Tool Types group, click **None**.
 - Select the flat endmill's icon.
 - Click **OK**.



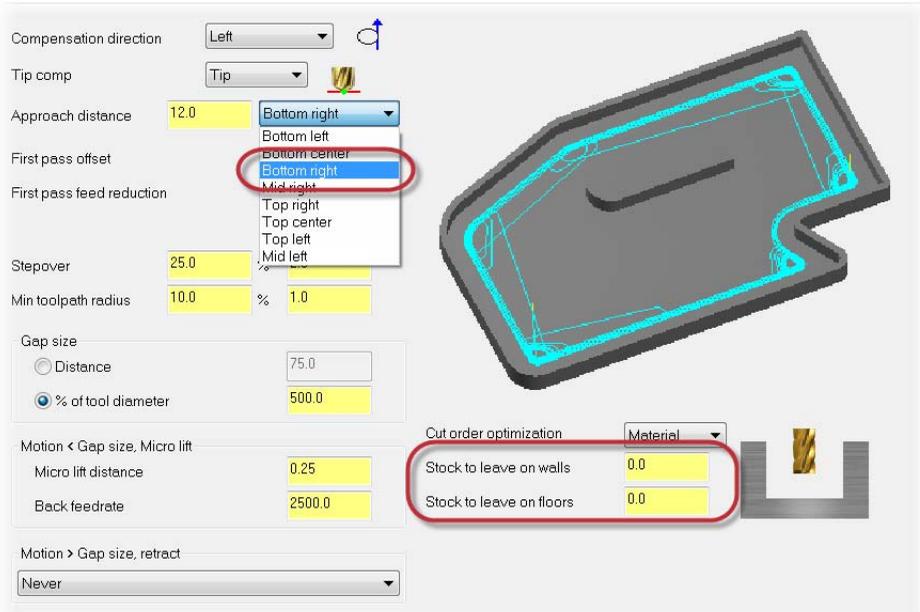
- 14 In the tool list, click the **10 mm Flat Endmill**, and click **OK**.

The selected tool appears in the tool list of the 2D High Speed Toolpaths - Dynamic Contour dialog box.



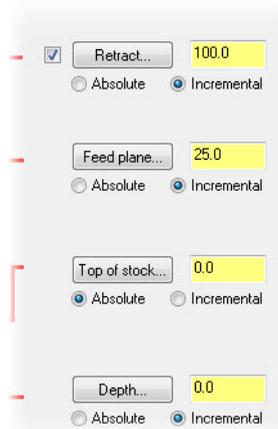
- 15 Select the **Cut Parameters** page.

- Choose **Bottom right** from the General start location drop-down menu.
- Set **Stock to leave on walls** and **Stock to leave on floors** to **0.0**.



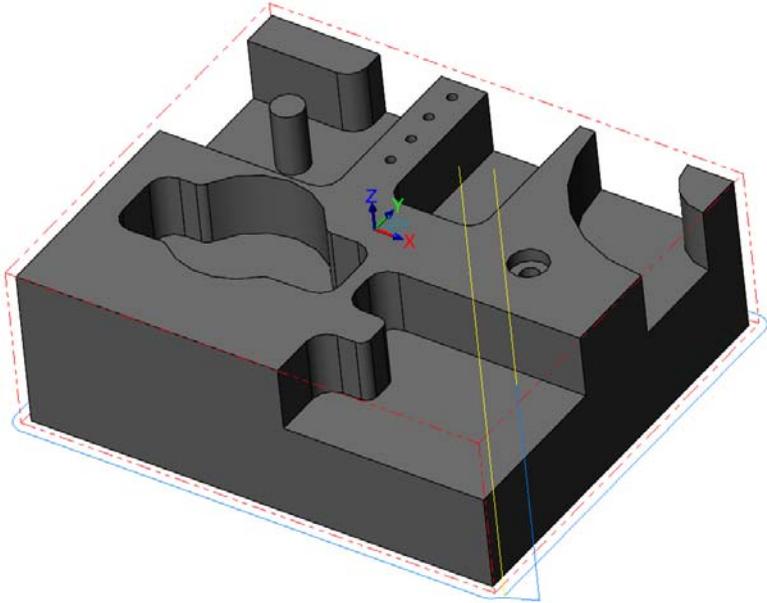
16 Select the **Linking Parameters** page, and then set the following parameters:

- a** Set **Retract** to **100.0**.
- b** Set **Feed plane** to **25.0**.
- c** Set **Top of stock** to **0.0**.
- d** Set **Depth** to **0.0**.
- e** Ensure that all values are set to **Incremental**, except **Top of stock**, which should be **Absolute**.



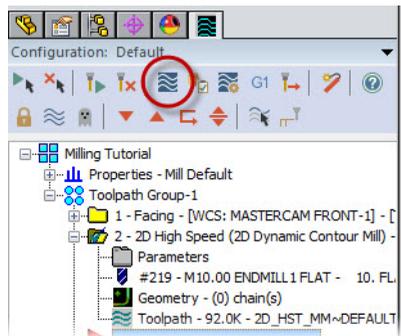
17 Click **OK** to finalize your choices.

Mastercam for SOLIDWORKS creates the toolpath shown in the following picture. (The picture does not show the facing toolpath.)



Exercise 2: Backplot and Verify the Dynamic Contour Toolpath

- 1 In the Toolpaths Manager, click the **Backplot selected operations** button.



- 2 Click the **Play** button to watch Mastercam Simulator backplot the toolpath.



30 MASTERCAM FOR SOLIDWORKS X9/ *The Dynamic Contour Toolpath*

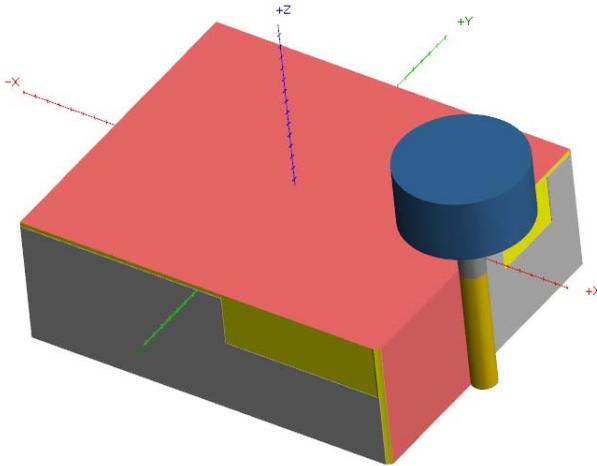
- 3 When Backplot finishes, click the **Verify** button.



- 4 Click the **Play** button to start Verify.



Mastercam Simulator shows the cutting operation, as shown in the picture below.



- 5 Close Mastercam Simulator.
- 6 Save your work.

LESSON 4

The Dynamic Mill Toolpath

Your part has five pockets, one closed and four open, that must be cut. In this lesson, you will add two Dynamic Mill toolpaths with different machining region strategies to machine the closed pocket and the open pockets.

Lesson Goals

- Add a Dynamic Mill toolpath with the Stay Inside machining region strategy.
- Add a Dynamic Mill toolpath with the From Outside machining region strategy.
- Backplot the toolpaths.
- Verify the toolpaths.

Exercise 1: Creating the Dynamic Mill Toolpath (Stay Inside)

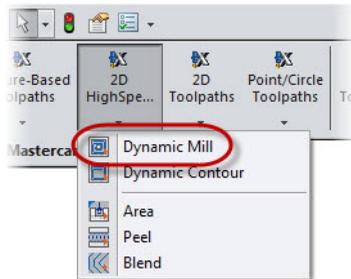
- 1 If necessary, open the 2D_HST_MM_Tutorial.SLDPRT file that you worked on in the previous lesson.



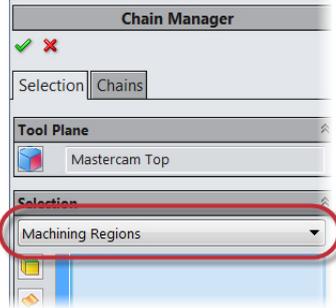
TIP: You can also begin this lesson by loading the file 2D_HST_MM04.SLDPRT, which is provided with this tutorial.

- 2 In the MastercamX9 CommandManager, click **2D HighSpeed Toolpaths**, and select **Dynamic Mill**.

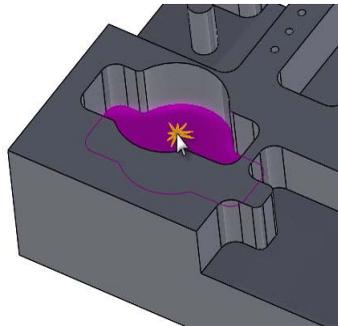
The Chain Manager displays, docked to the left of the SOLIDWORKS window.



- 3 In the Selection tab, select **Machining Regions** from the **Selection Type** drop-down.



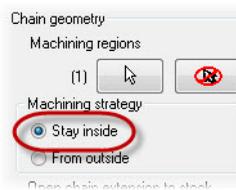
- 4 Click the floor of the closed pocket.
The selected face displays in the Chain Manager's selection list.



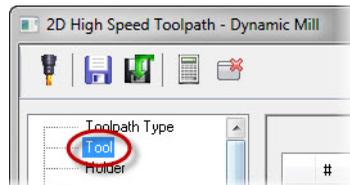
- 5 Click **OK** in the Chain Manager to finalize your selection.
The 2D High Speed Toolpath - Dynamic Mill dialog box displays.

- 6 In the **Toolpath Type** page, ensure that the **Machining strategy** is set to **Stay inside**.

This strategy keeps the toolpath inside the selected machining region.



- 7 Select the **Tool** page.



- 8** Click the **Select library tool** button.

The Tool Selection dialog box displays.

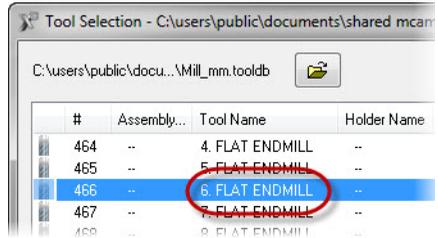


- 9** If necessary, filter the tools to show only flat endmills.

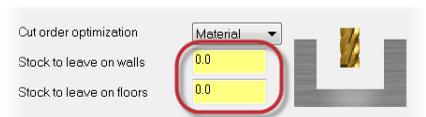


- 10** In the tool list, click the **6 mm Flat Endmill**, and click **OK**.

Mastercam for SOLIDWORKS assigns the tool to the Dynamic Mill toolpath.

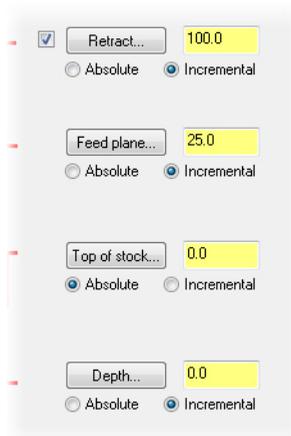


- 11** Select the **Cut Parameters** page, and change **Stock to leave on walls** and **Stock to leave on floors** to **0.0**.



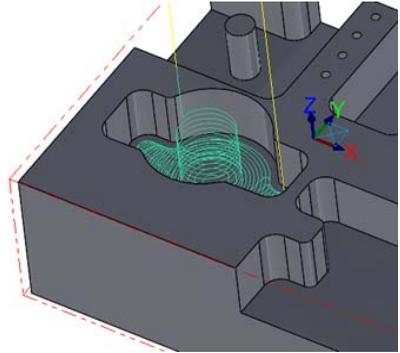
- 12** Select the **Linking Parameters** page, and then:

- a** Set **Retract** to **100.0**.
- b** Set **Feed plane** to **25.0**.
- c** Set **Top of stock** to **0.0**.
- d** Set **Depth** to **0.0**.
- e** Ensure that all values are set to **Incremental**, except **Top of stock**, which should be **Absolute**.



- 13** Click **OK** to finalize your choices.

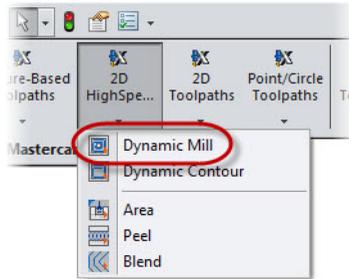
Mastercam for SOLIDWORKS creates the toolpath shown in the following picture. (The picture does not show the toolpaths you created in previous lessons.)



Exercise 2: Creating the Dynamic Mill Toolpath (From Outside)

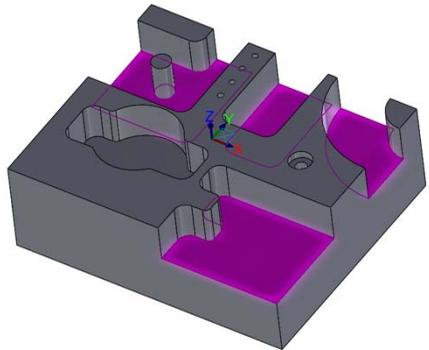
- 1 In the Mastercam X9 CommandManager, click **2D HighSpeed Toolpaths**, and select **Dynamic Mill**.

The Chain Manager displays, docked to the left of the SOLIDWORKS window.

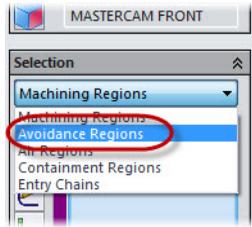


- 2 With the **Selection Type** set to **Machining Regions**, click the floor of each open pocket.

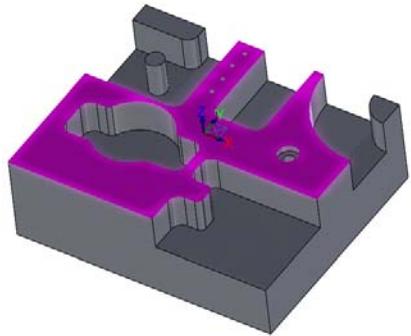
The selected faces appear in the Chain Manager's selection list.



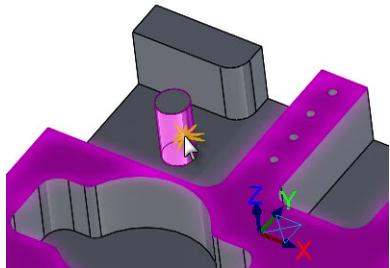
- 3 In the Selection tab, select **Avoidance Regions** from the **Selection Type** drop-down.



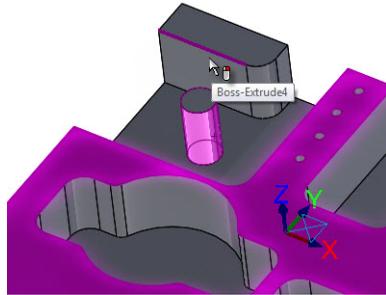
- 4 Select the following:
- a Click the part's upper face.



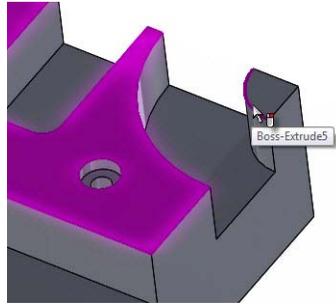
- b Click the round boss, shown in the picture to the right.



- c Right-click the upper edge of the boss next to the round boss, and choose **Select Loop** from the pop-up menu.

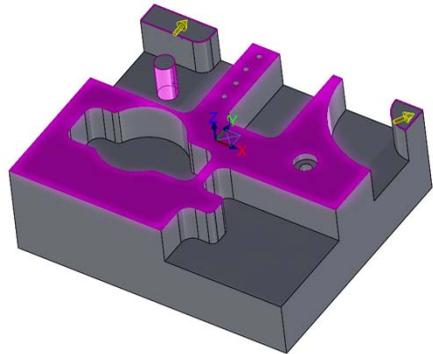


- d Right-click the upper edge of the remaining boss, and choose **Select Loop** from the pop-up menu.



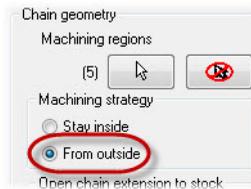
- e Verify that your avoidance selections match the picture to the right.
- f Click **OK** in the Chain Manager to finalize your selection.

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

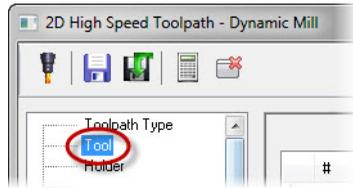


- 5 In the **Toolpath Type** page, ensure that the **Machining strategy** is set to **From outside**.

This strategy causes the toolpath to start machining from the outside and work its way in.



6 Select the **Tool** page.



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

The Tool Selection dialog box displays.

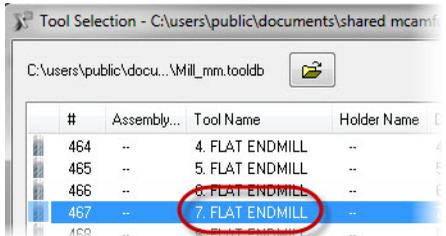


8 If necessary, filter the tools to show only flat endmills.



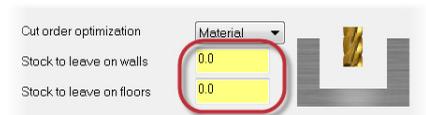
9 In the tool list, click the **7 mm Flat Endmill**, and click **OK**.

Mastercam for SOLIDWORKS assigns the tool to the Dynamic Mill toolpath.



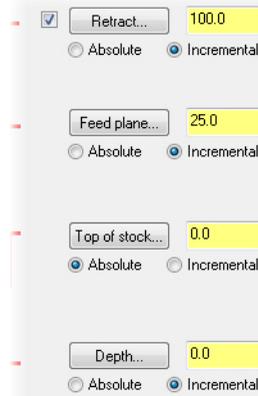
10 Select the **Cut Parameters** page, and ensure that the **Stock to leave on walls** and **Stock to leave on floors** is set to **0.0**.

Mastercam will remember the previously set values, unless the software has been closed and reopened. This also applies to other pages.



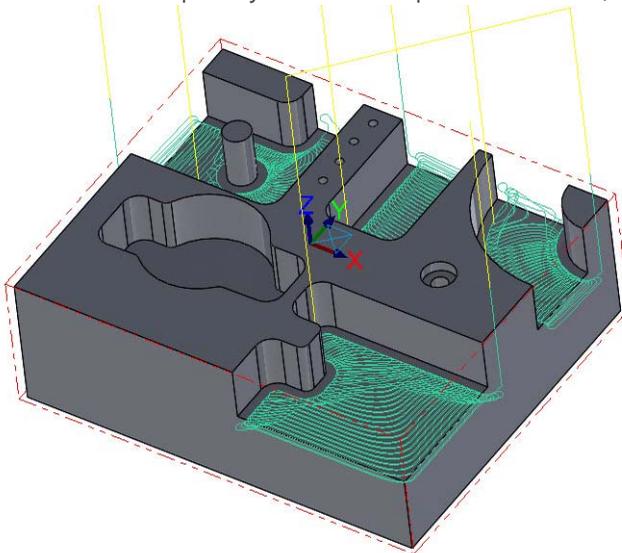
11 Select the **Linking Parameters** page and ensure that the parameters are set to the following:

- a Set **Retract** to **100.0**.
- b Set **Feed plane** to **25.0**.
- c Set **Top of stock** to **0.0**.
- d Set **Depth** to **0.0**.
- e Ensure that all values are set to **Incremental**, except **Top of stock**, which should be **Absolute**.



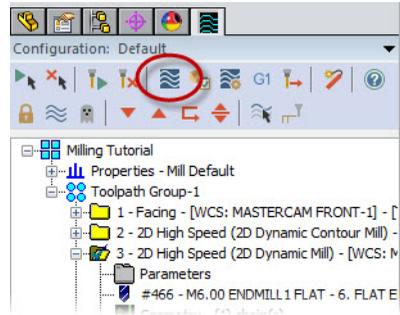
12 Click **OK** to finalize your choices.

Mastercam for SOLIDWORKS creates the toolpath shown in the following picture. (The picture does not show the toolpaths you created in previous lessons.)



Exercise 3: Backplot and Verify the Dynamic Mill Toolpaths

- 1 Select both toolpaths in the Toolpaths Manager by selecting the first 2D High Speed (2D Dynamic Mill) toolpath and then [**Ctrl + click**] the second 2D High Speed (2D Dynamic Mill) toolpath.
- 2 Click the **Backplot selected operations** button.



- 3 Click the **Play** button to watch Mastercam Simulator backplot the toolpaths.



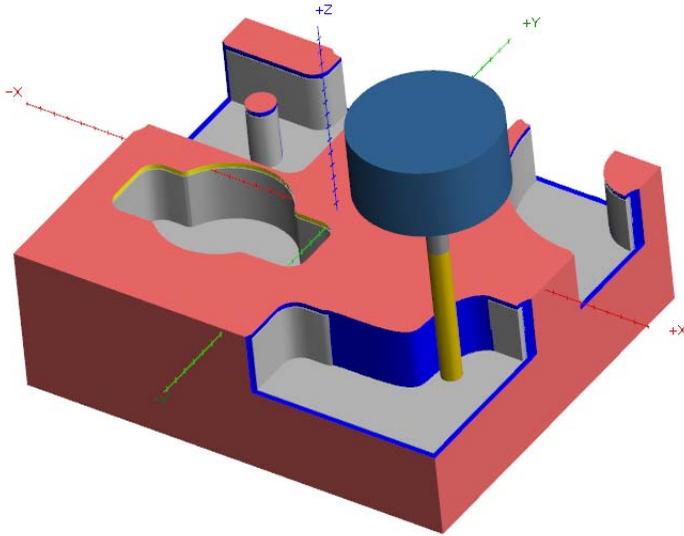
- 4 When Backplot finishes, click the **Verify** button.



- 5 Click the **Play** button to start Verify.



Mastercam Simulator shows the cutting operations, as shown in the picture below.



6 Close Mastercam Simulator.

7 Save your work.

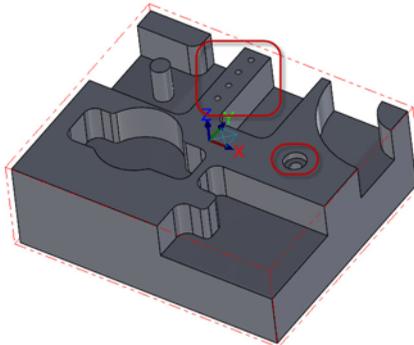
LESSON 5

The Drill Toolpaths

Wizard Holes is a Feature Based Machining (FBM) strategy that can automatically generate a complete series of drill operations with data *exclusively* from pre-selected holes created by the SOLIDWORKS Hole Wizard. With Wizard Holes you can:

- Preview toolpath operations and make additional changes before they are generated.
- Automatically generate a complete series of drill operations for the selected features.

In this lesson, you use Wizard Holes to generate toolpaths to drill the holes indicated in the following picture.



Lesson Goals

- Add the Drill toolpaths.
- Backplot the toolpaths.
- Verify the toolpaths.

Exercise 1: Creating the Drill Toolpaths

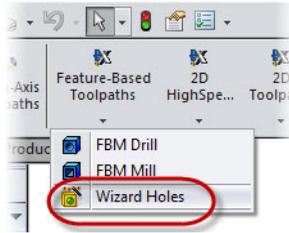
- 1 If necessary, open the `2D_HST_MM_Tutorial.SLDPRT` file that you worked on in the previous lesson.



TIP: You can also begin this lesson by loading the file `2D_HST_MM05.SLDPRT`, which is provided with this tutorial.

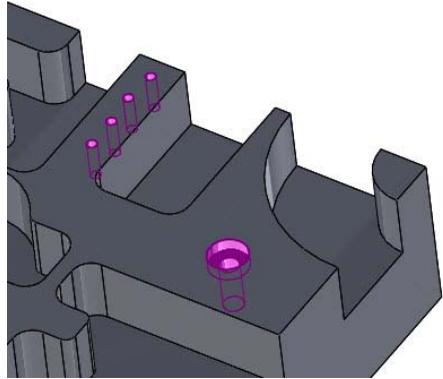
- 2 In the MastercamX9 CommandManager, click **Feature-Based Toolpaths**, and select **Wizard Holes**.

The Selection dialog box displays, docked to the left in the SOLIDWORKS window.



- 3 Select the holes shown in the picture to the right. The Wizard Hole Selection list will display the following selections:

- Ø3.0 (3) Diameter Hole1
- CBORE for M5 Hex Head Bolt1



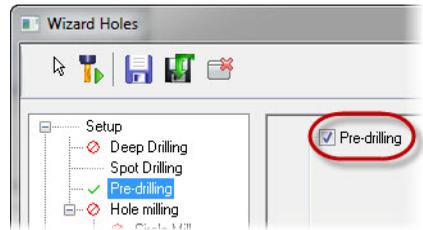
TIP: You can select Hole Wizard features from the graphics area or from the Flyout FeatureManager Design Tree.



- 4 Click **OK** in the Selection dialog box. The Wizard Hole dialog box displays.

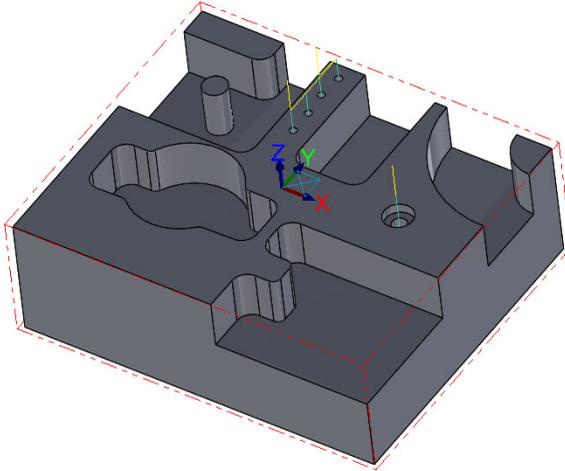
- 5 Select the Pre-drilling page. Enable the page by selecting the Pre-drilling option.

Pre-drilling creates (roughing) cycles that rough out the drill holes before the finish drill cycles.



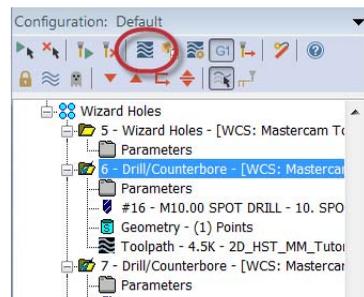
- 6 Click **OK** to close the Wizard Holes dialog box and create the drill toolpaths.

Mastercam for SOLIDWORKS generates the required drill toolpaths. (For clarity, the picture shows none of the toolpaths from the previous lessons.)



Exercise 2: Backplot and Verify the Drill Toolpath

- 1 In Toolpaths Manager, click the **Backplot selected operations** button.



44 MASTERCAM FOR SOLIDWORKS X9/ *The Drill Toolpaths*

- 2 Click the **Play** button to watch Mastercam for SOLIDWORKS backplot the toolpath.



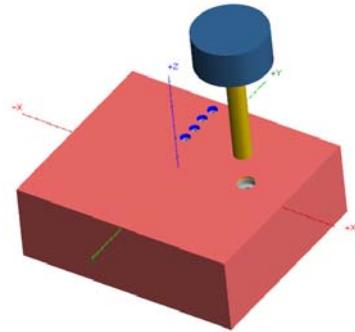
- 3 When Backplot finishes, click the **Verify** button.



- 4 Click the **Play** button to start Verify.



Mastercam Simulator shows the cutting operation, as shown in the picture to the right.

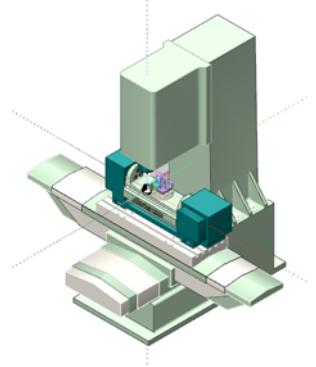


- 5 Close Mastercam Simulator.
- 6 Save your work.

LESSON 6

Final Verification and Posting

Before you send your part's machining data to your machine control, you must first verify that all of the operations are correct and then generate the part's NC file, which contains the machining instructions. In this lesson, you backplot and verify the entire set of toolpaths, as well as generate the post file. Along the way, you use Machine Simulation to view your machine's interaction with the part.



Lesson Goals

- Backplot all toolpaths.
- Verify all toolpaths.
- Run Machine Simulation on toolpaths.
- Post toolpaths.

Exercise 1: Backplot the Part's Toolpaths

- 1 In the Toolpaths Manager, click the **Select all operations** button.



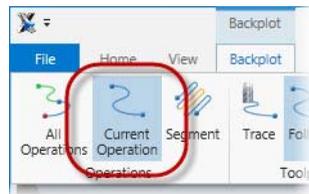
A green check appears next to each toolpath.

- 2 Click the **Backplot selected operations** button.



- 3 On the Backplot tab in Mastercam Simulator, click the **Current Operation** button.

Backplot now displays only the toolpath for the operation being backplotted.



- 4 Click the **Play** button to watch Mastercam Simulator backplot the toolpaths.



Exercise 2: Verify the Part's Toolpaths

- 1 Click the **Verify** button on the Home ribbon bar.



TIP: Turn the **Color Loop** on to view each operation as a different color.

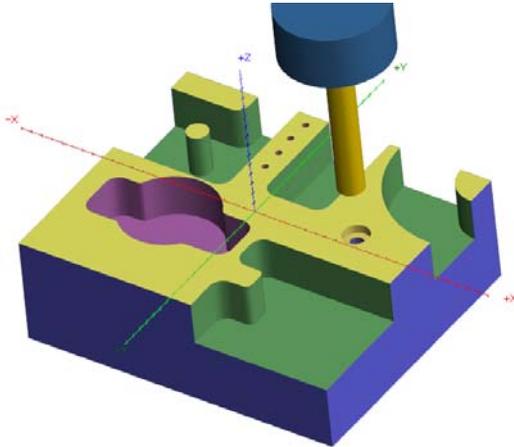


- 2 Click the **Play** button.

Mastercam Simulator runs Verify on all toolpaths.



The picture below shows the part following complete verification.



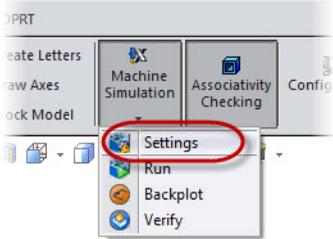
TIP: Drag the **Progress** slider left or right to move quickly from one operation to another.

- 3 Close Mastercam Simulator.

Exercise 3: Machine Simulation

- 1 From the MastercamX9 CommandManager, select **Machine Simulation, Settings**.

The Machine Simulation dialog box displays.

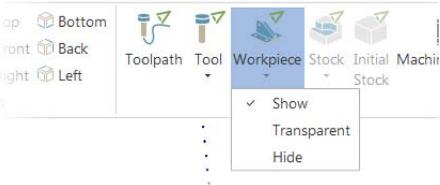


- 2 To start the simulation, click **Simulate**.

The Machine Simulation window displays.



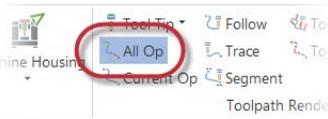
- 3 Click the drop-down list below to the **Workpiece** button (in the Visibility group on the ribbon bar) and select **Show** to display the workpiece on the machine.



- 4 Click the **Toolpath** button on the ribbon bar to display the tool motion.



- 5 From the Toolpath Rendering group, click **All Op** to display all of the selected toolpaths.



- 6 Click the **Run** button located on the ribbon bar in the Control group.

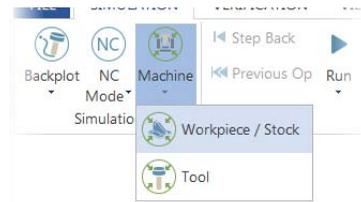
Machine Simulation simulates cutting the part. Red areas on the part indicate collisions.



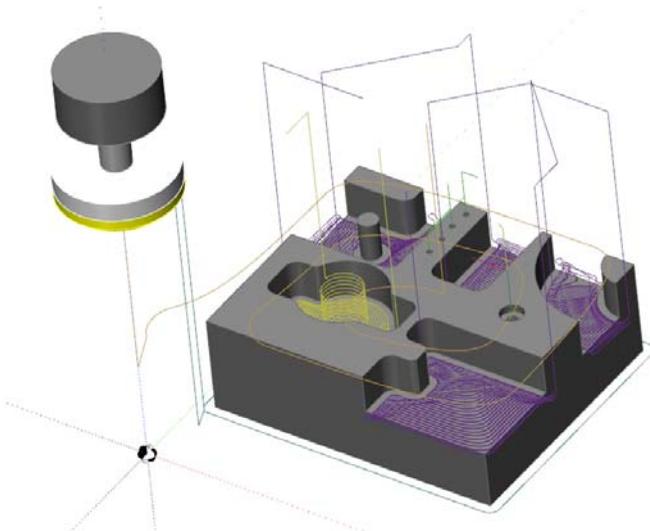
- 7 To stop the simulation, click the **Stop** button.



- 8 From the Simulation group, click the drop-down list below to the **Machine** button and select **Workpiece/Stock**.



Machine Simulation zooms in on the part.



9 Click the **Restart** button.

Your part "rewinds" back to the first operation.

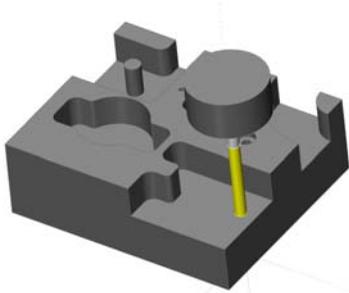


10 Run the simulation again to get a better view of the part being cut.

11 To get a better view of the finished part, turn off the **Toolpath** button.

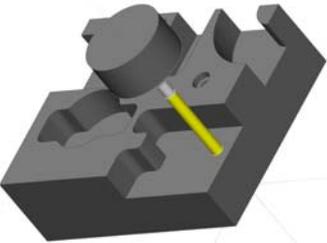


Machine Simulation removes the toolpaths from the part.



12 Place your mouse cursor in the graphics window, and drag with the left mouse button.

Machine Simulation rotates the part as specified by your mouse movement.



13 Click the **Exit** button.

Machine Simulation closes.



Exercise 4: Post the Part's Toolpaths

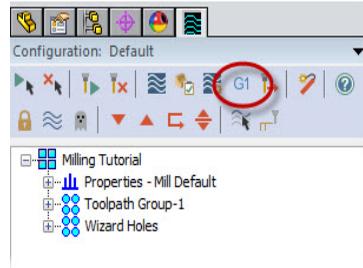
- 1 In the Toolpaths Manager, if necessary, click the **Select all operations** button.



A green check appears next to each toolpath.

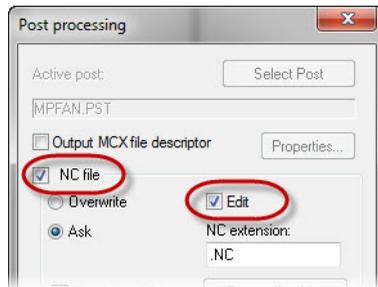
- 2 Click the **Post selected operations** button.

The Post processing dialog box displays.



- 3 If they are not already selected, select the **NC file** and **Edit** options, and then click **OK**.

The Save As dialog box displays.



- 4 Enter a file name for the NC File, and click **Save**.



Mastercam for SOLIDWORKS analyzes all operations and creates the NC file for the part. Because you selected the **Edit** option, the NC file displays in Mastercam Code Expert Editor, as shown in the following picture.

- *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 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 for SOLIDWORKS Documentation

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

- *What’s New in Mastercam for SOLIDWORKS X9*
- *Mastercam for SOLIDWORKS X9 Installation Guide*
- *Mastercam for SOLIDWORKS X9 Administrator Guide*
- *Mastercam for SOLIDWORKS X9 Transition Guide*
- *Mastercam for SOLIDWORKS X9 Quick Reference Card*
- *Mastercam for SolidWorks Tutorial (Mill)*
- *Mastercam for SOLIDWORKS 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.**

cnr software, inc.

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