Mastercam X4

Getting Started Series Tutorials

Basic 2D Machining

mastercam x getting started series tutorials

Basic 2D Machining

August 2009

Be sure you have the latest information!

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

Mastercam X4

Mastercam® X4 Basic 2D Machining

Date: August 2009

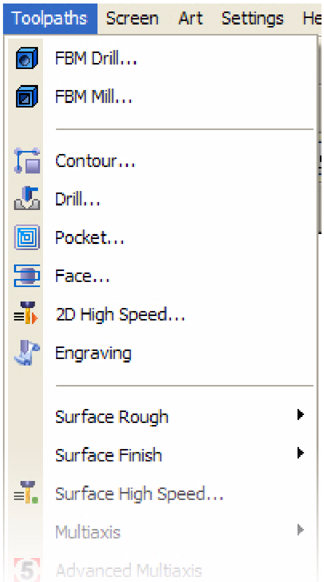
First Printing: August 2009
Software: Mastercam X4
Part Number: X4-PDF-TUT-2M

Contents

INTRODUCTION

This tutorial focuses on applying several mill toolpaths to a part previously designed in Mastercam. The tutorial then guides you through the steps to take that toolpath data and create NC code to machine the part.

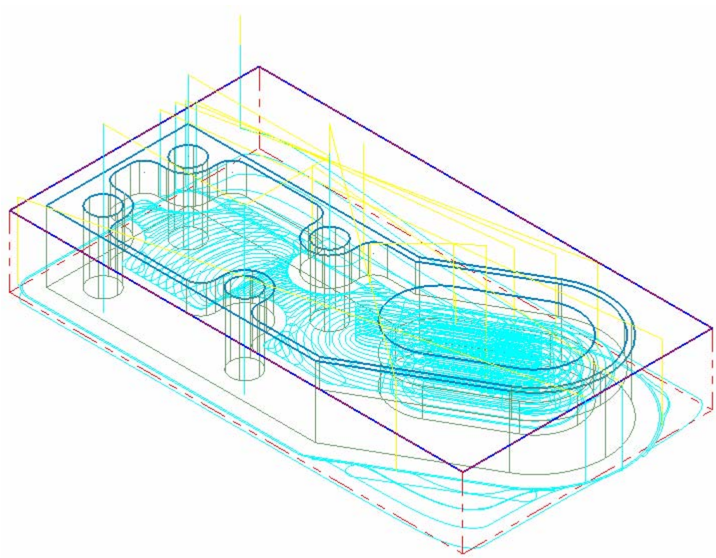
Mastercam offers a variety of toolpath types that let you quickly build toolpaths for specific applications. You access these toolpaths, such as the mill toolpaths shown here, through the Toolpaths menu.



After you create a toolpath, you can use Mastercam's backplot feature to preview its operation on the screen. Once you are satisfied with it, post it from the Toolpath Manager to generate the NC code for a specific machine tool.

When you first begin this tutorial, you will be prompted to assign a machine definition to the part. This tutorial does not go into any depth on machine and control definitions. However, the Help and other documentation installed with Mastercam provide comprehensive information regarding these Mastercam features.

When you finish this tutorial, your part will look like this:



If you would like to design (create) the part before you begin this tutorial, please follow the *Basic 2D Design* tutorial procedures (published separately in document #*X4-PDF-TUT-2D*).

Tutorial Goals

- Open a part file, assign a default machine definition, and set up stock.
- Create four drill holes with one toolpath (including selecting a drill point, choosing tooling, using tool tip compensation, and setting machining values).
- Rough the outside of the part (including chaining entities, selecting tooling, and setting machining values).
- Clean out the inside of the part (including chaining entities, choosing tooling, and setting machining values).
- View all toolpaths in the graphics window.
- View a specific toolpath by temporarily turning off the display of selected toolpaths.
- Backplot (view the path the tools take to cut the part) all toolpaths.
- Customize your backplot display.
- Simulate (verify) the machining of the part from a stock model display.
- Post all toolpath operations to an NC file, review/edit the code as necessary, and save the NC file.

Before You Begin

This is a module of the *Mastercam Getting Started Tutorial Series*, which introduce basic Mastercam skills. Other tutorial series, which cover more advanced skills, are:

- *Focus Series*—This series provides more in-depth training on specific or advanced Mastercam features and functions.
- *Exploring Series*—This series explores the application of a single Mastercam product, such as Mill, Wire, or Art.

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

Note: Screen colors in the tutorial pictures enhance image quality; they may not match your Mastercam settings.

General Tutorial Requirements

Because each lesson in the tutorial builds on the mastery of preceding lesson's skills, we recommend that you complete them in order. In addition, the tutorials in this series have the following requirements:

- You must be comfortable using the Windows® operating system.
- You must have a seat of Mastercam X4 Design or higher to complete most of the tutorials in the *Getting Started* series. The tutorials cannot be used with Mastercam Demo/Home Learning Edition.

- The *Basic 2D Machining* tutorial requires a seat of Mill Entry or Router Entry. Tutorials in other series may require higher level licenses.
- Part files may accompany a tutorial. They should be stored in a folder with the tutorial or in a location that you prefer.
- You must configure Mastercam to work in metric units. The next section includes instructions for setting Mastercam to metric.

Preparing for a Tutorial

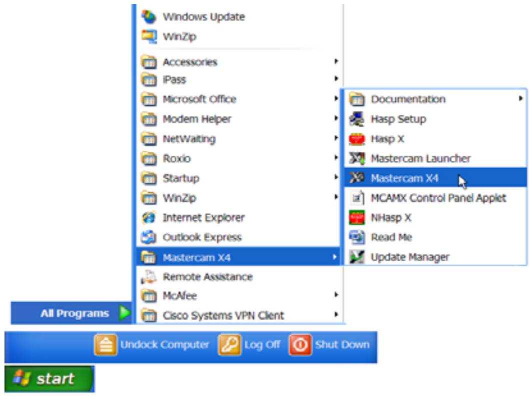
Before you start a tutorial, be sure you have completed the following tasks:

1 Start Mastercam using your preferred method:

- Double-click Mastercam's desktop icon.

  Or

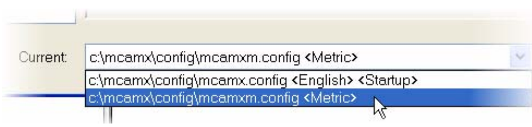
- Launch Mastercam from the Windows Start menu.



2 Select the metric configuration file:

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

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



c Click **OK**.

If You Need More Help

There are many ways to get help with Mastercam, including the following:

- *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.
- *Online help*—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.
- *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.

- *Documentation feedback*—For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable, online learning tool with more than 180 videos to let you master your skills at your own pace. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.

Additional Documentation

You can find more information on using Mastercam in the following materials, located in the `\Documentation` folder of your Mastercam installation:

- *Mastercam X4 Installation Guide*
- *Mastercam X4 Administrator Guide*
- *Mastercam X4 Quick Start*
- *Mastercam X4 Reference Guide*
- *Mastercam X4 Transition Guide*
- *Mastercam X4 Quick Reference Card*
- *Mastercam X4 Wire Getting Started Guide*
- *Version 9 to X Function Map*

Drilling Holes

The first step to machining the tutorial part is to drill the four holes. By beginning with drilling, you can then use the holes to fixture the part on the machine tool.

Lesson Goals

- Open a part file and assign a machine definition.
- Define stock.
- Create a drill toolpath (including selecting drill points, choosing tooling, and setting machining values).
- Use tool tip compensation.

Exercise 1: Assigning a Machine Definition

In this exercise, you open an existing part, assign a machine definition to the part, and save it under a new file name.

1 Start Mastercam.

2 From the Mastercam menu bar, choose **File**, **Open**.

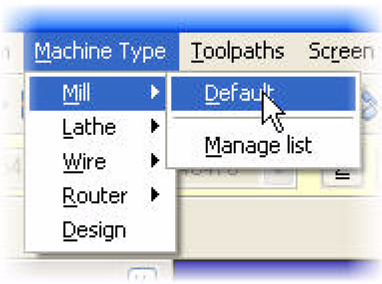
3 Open `BASIC_2D_MACHINING_START.MCX`, which was provided with this tutorial.

4 Click **OK** if prompted to switch to a metric configuration.

5 Press **[Alt +S]** to shade the part for easier viewing.

6 From the Mastercam menu, choose **Machine Type**, **Mill**, **Default** to open the default Mill machine definition.

In Mastercam, you select a machine definition before creating any toolpaths. The ***machine definition*** is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs.



Note: Parts that have previously been saved with a machine definition automatically load the associated machine definition.

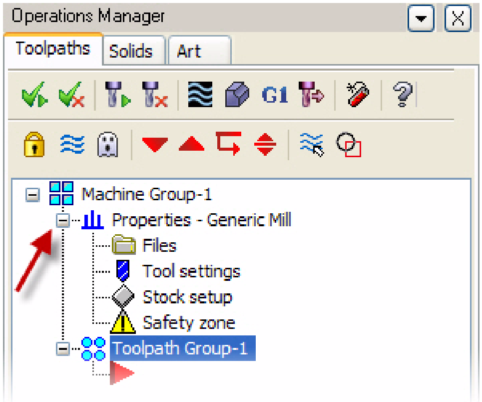
7 Choose **File**, **Save As** and save the part under a different file name than the original file. This will protect the original tutorial file from being overwritten.

Exercise 2: Setting Up Stock

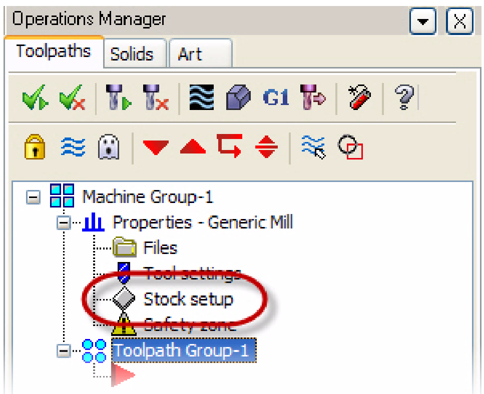
In this exercise, you define the stock model from which your part is cut. ***Stock models*** help you visualize your toolpaths more realistically. The stock model you define can be displayed with the part geometry when viewing the file or toolpaths, during backplot, or while verifying toolpaths.

1 In the Operations Manager, click on the plus sign (+) next to **Properties - Generic Mill**.

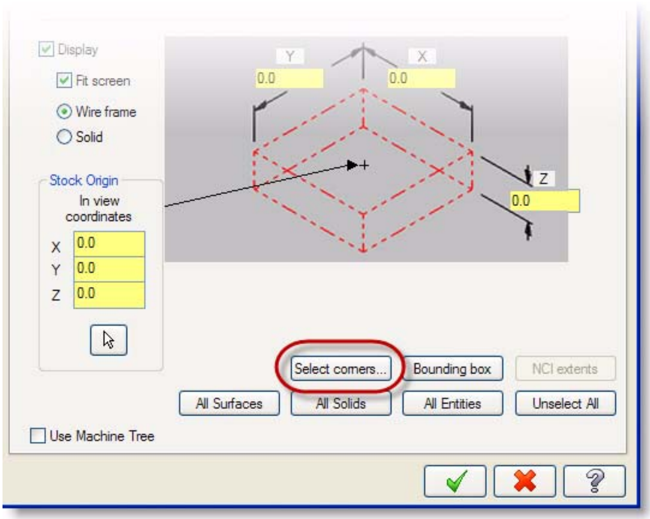
It changes to a minus sign (-) and displays the machine group properties. ***Machine groups*** are created automatically and displayed in the Toolpaths Manager when you select a machine from the Machine Type menu. They include job setup information for toolpaths, such as tool numbering, stock models, material selection, and toolpath defaults and libraries.



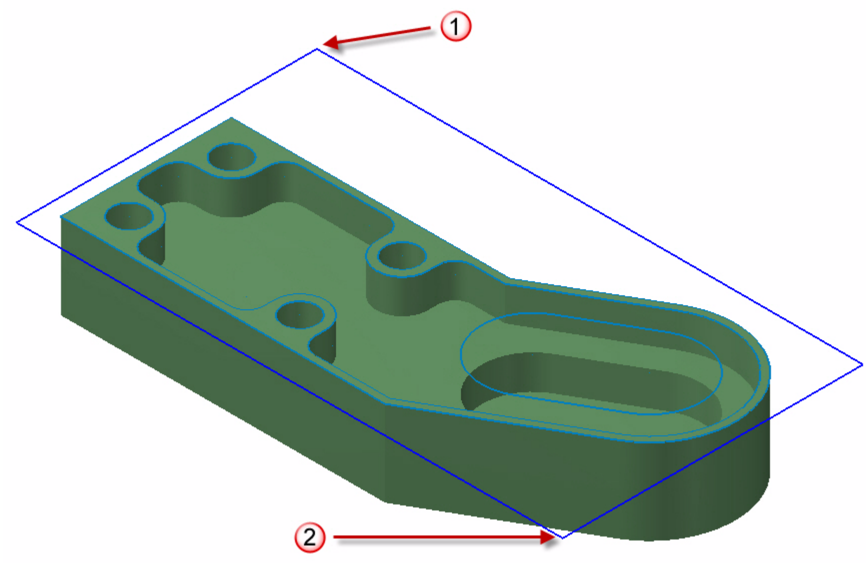
2 Click the **Stock Setup** icon to display the Stock Setup tab in the Machine Group Properties dialog box.



3 Click the **Select corners** button near the bottom of the dialog box. Mastercam brings you back to the graphics window to select the two opposite stock corners.



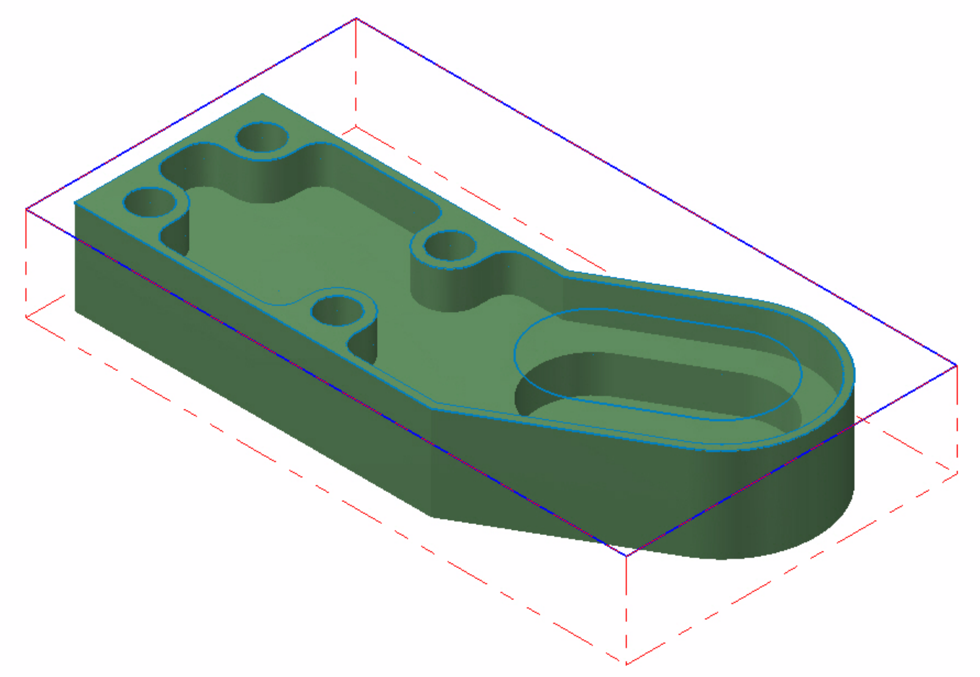
4 Click the two opposite corners as shown in the following picture. The Stock Setup tab displays again with X and Y coordinate values from the selected corners.



5 Enter **25** for the Z coordinate in the stock diagram to provide some depth for your stock.

6 Select the **Display** check box to see the stock model boundaries in the graphics window.

7 Click **OK** to complete the stock setup. The stock displays as dashed red lines around the part.

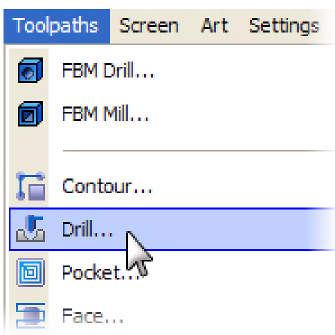


Exercise 3: Drilling Four Holes

In this exercise, you create a drill toolpath that drills all four holes of the part.

Selecting the Drill Holes

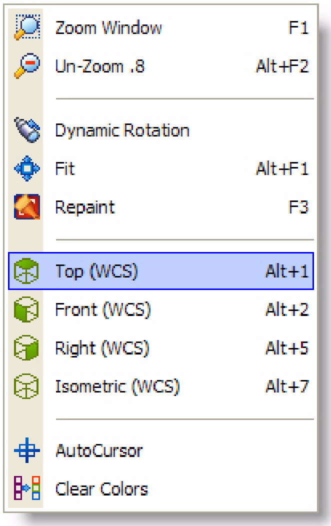
1 From the Mastercam menu, choose **Toolpaths**, **Drill**. The Drill Point Selection dialog box opens.



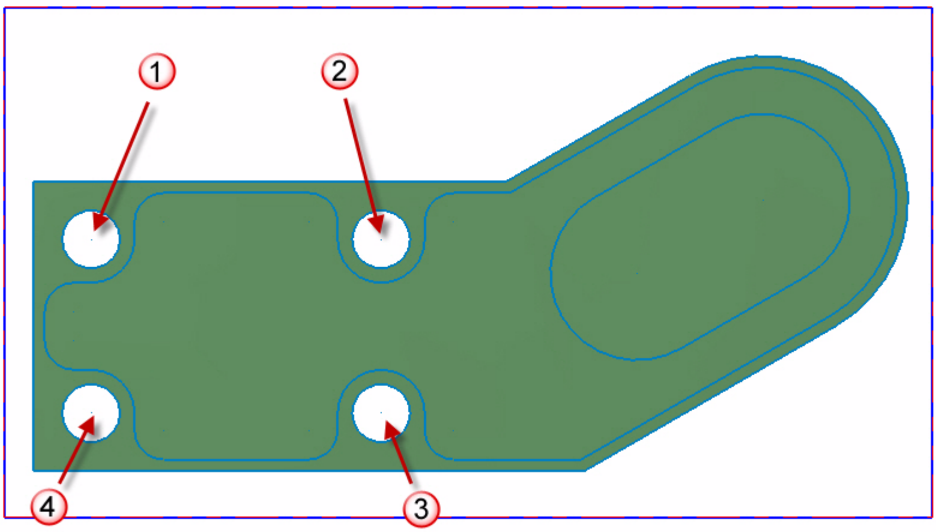
Note: You may be prompted for an NC file name. If prompted, click **OK** *to confirm the default NC file name, or overwrite the file name and click* **OK** *to modify the default file name.*

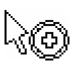
---

2 Right-click in the graphics window and choose **Top (WCS)** to switch to a Top graphics view. This will make it easier to select the drill holes.



3 Click each of the center points of the four holes of the part. Choose them in a clockwise order from top left to bottom left as shown in the numbered sequence below.



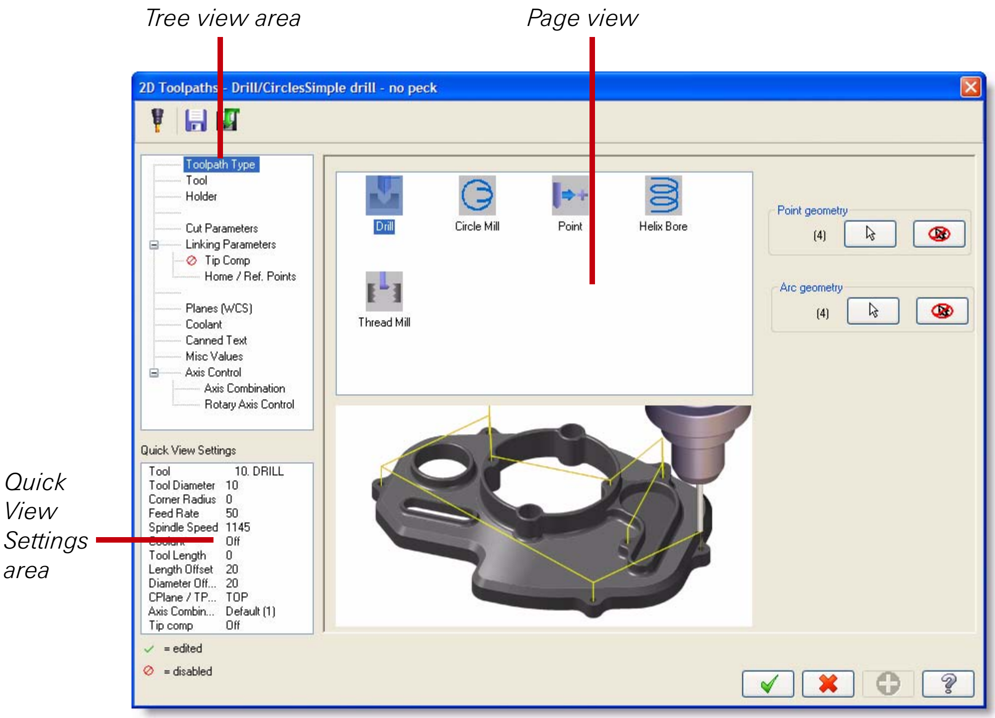
TIP: When you get close to each hole's center point, Mastercam displays a Visual Cue that indicates the center point. 

4 Click **OK**. The 2D Toolpaths - Drill/Circles dialog box opens.

Exploring the Toolpath Dialog Boxes

Many Mastercam toolpaths, including Drill, use a tree-style dialog interface made up of three distinct areas:

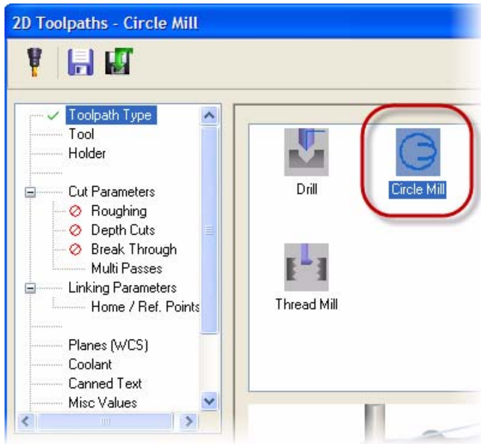
- *Tree View* - Displays a list of all the available dialog box property pages.
    - Inactive pages are marked with a red circle and a slash.
    - Edited pages are marked with a green check mark.
    - Some pages may have a plus or minus symbol, indicating that the page has subpages and can expand and contract to either display or hide the subpages.
- *Page View* - Changes with each selection you make in the Tree View area.
- *Quick View Settings* - Summarizes key toolpath information from parameters you set on the different pages. It updates automatically as you make changes in the pages, and is always visible.



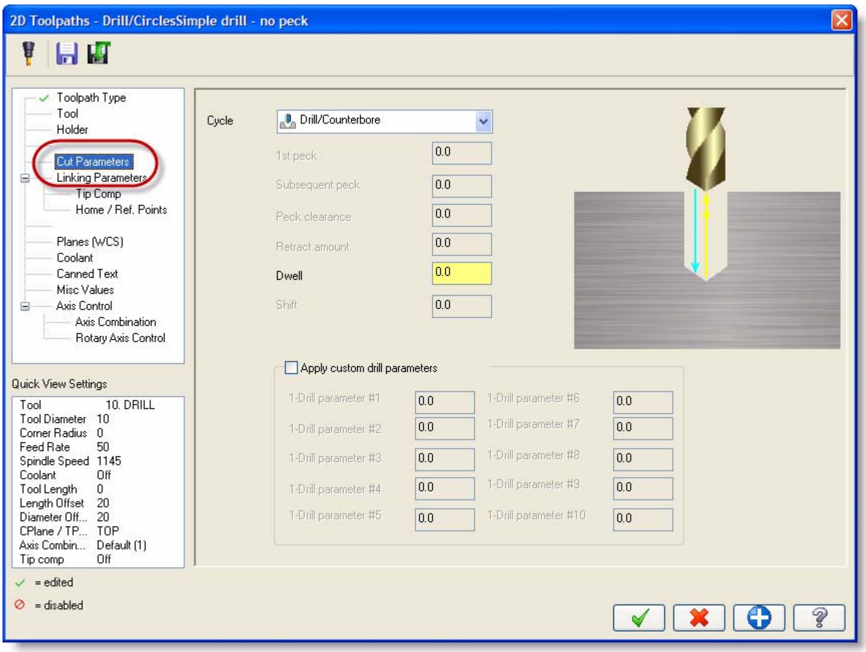
1 Click on the **Circle Mill** toolpath type icon. This changes the toolpath to a circle mill toolpath and changes the pages listed in the Tree View area.

Notice that the Toolpath Type page in the Tree View area now has a green check beside it, indicating that you made a change on that page.

2 Click the **Drill** toolpath type icon to change back to a drill toolpath.



3 Click the **Cut Parameters** page in the Tree View list. The Page area changes to list cut parameters for the toolpath.



Setting Drilling Parameters

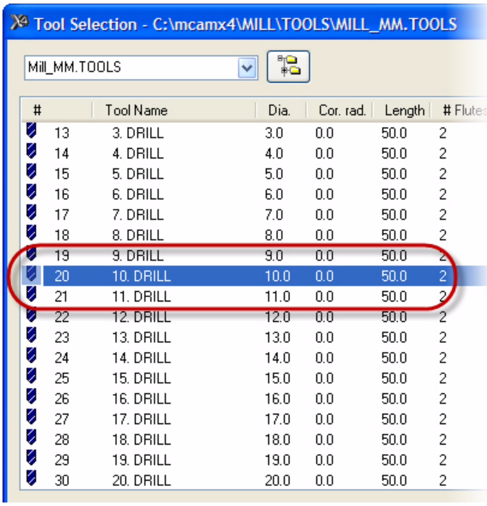
1 Click the **Tool** page in the Tree View list to pick a tool for the drill toolpath.

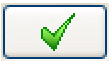
2 Click the **Select library tool** button to pick a tool from one of Mastercam's tool libraries.

The Tool Selection dialog box opens.

3 Select the **10mm** diameter drill from the tool list.

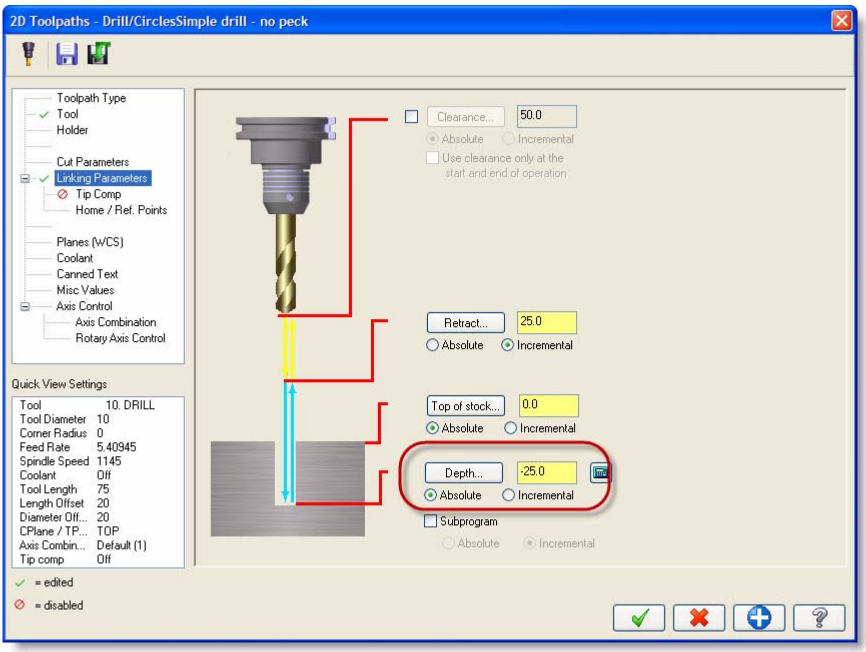
Note: Make sure that the `Mill_MM.TOOLS` *tool library is selected at the top of the dialog box. If not, click the drop-down arrow and select it from the tool library list.*



4 Click **OK**  to return to the Tool page.

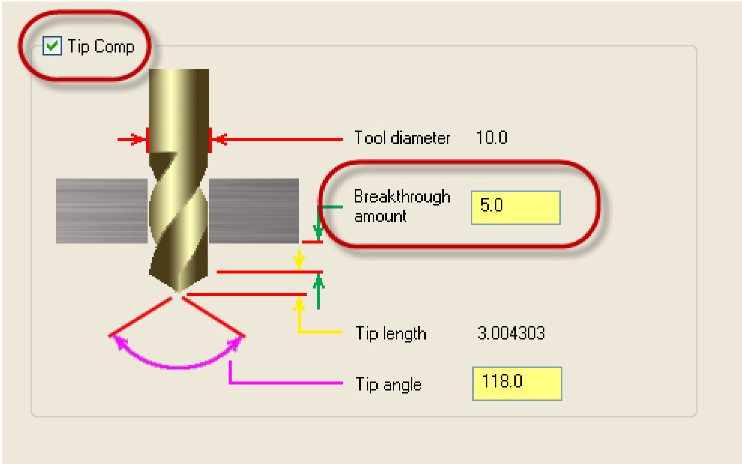
5 Click the **Linking Parameters** page in the Tree View list.

6 Enter **-25.0** in the **Depth** field.



7 Click the **Tip Comp** page in the Tree View list. Notice that this page is off by default.

8 Select the **Tip Comp** check box to turn on this feature and enter **5.0** in the **Breakthrough amount** field.

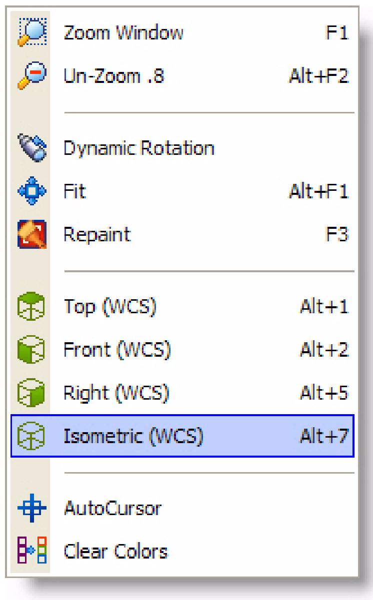


TIP: The drill tip compensation tells Mastercam how far to drill past the final depth to break through the stock. Enter a positive number only. Entering a negative value results in the drill retracting before the desired depth is reached.

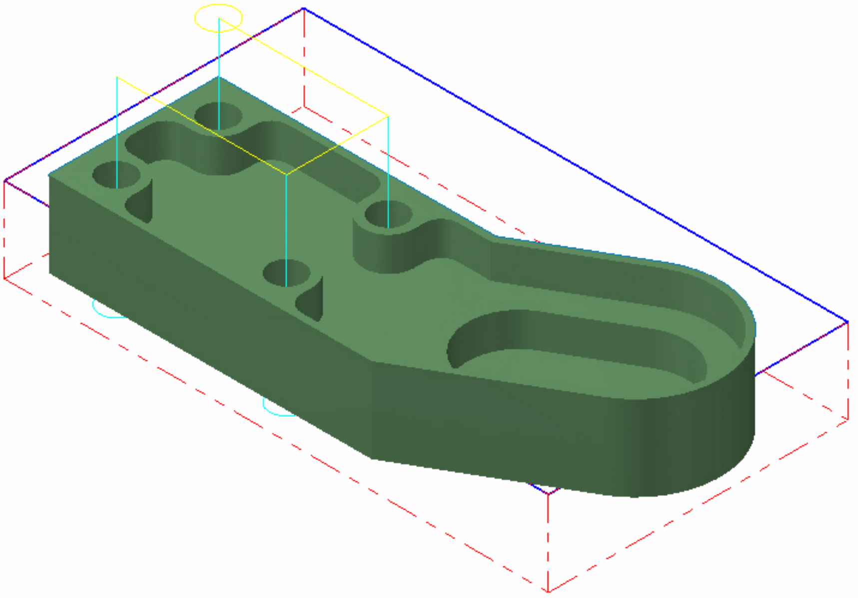
9 Click **OK** to generate the drill toolpath for the four holes.

10 Right-click in the graphics window and choose **Isometric (WCS)** from the menu to view the part and toolpath in the isometric view.

You may need to center the part in the graphics window to see it. The easiest way to do this is to use the graphics window right-click menu and select **Fit** to fit the part in the graphics window, then unzoom by pressing [**Alt+F2**]. You can also use the fit/zoom/unzoom buttons in the View Manipulation toolbar.



Your toolpath should look like this. The cyan lines are feed motion and the yellow lines are rapid motion.



11 Right-click in the graphics window again, and choose **Top (WCS)**.

12 Choose **File**, **Save** from the Mastercam menu or click the **Save** button  to save your part.

Now that the holes are drilled, you can move forward with removing stock from the outer area of the part.

Roughing Outside the Part

The next step to machining the tutorial part is to remove the bulk of the stock from the outside. You will use a couple of toolpaths to machine this area.

Lesson Goals

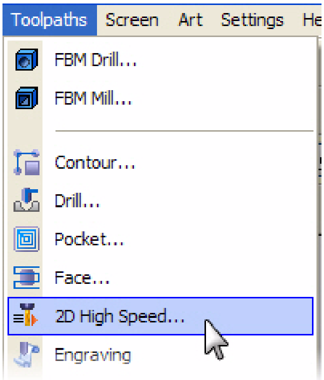
- Create a dynamic mill toolpath (including chaining entities, choosing tooling, and setting machining values).
- View a specific toolpath by temporarily turning off the display of selected toolpaths.
- Create a contour toolpath (including chaining entities, choosing tooling, and setting machining values).

Exercise 1: Creating a Dynamic Mill Toolpath

In this exercise, you create a dynamic mill 2D toolpath to clear away much of the stock from the outside of the tutorial part. Dynamic mill toolpaths, part of the 2D High Speed toolpath suite, utilize the entire flute length of their cutting tools to produce the smoothest, most efficient tool motion for high speed pocketing and core milling.

1 From the Mastercam menu, choose **Toolpaths**, **2D High Speed**. The Chaining dialog box opens.

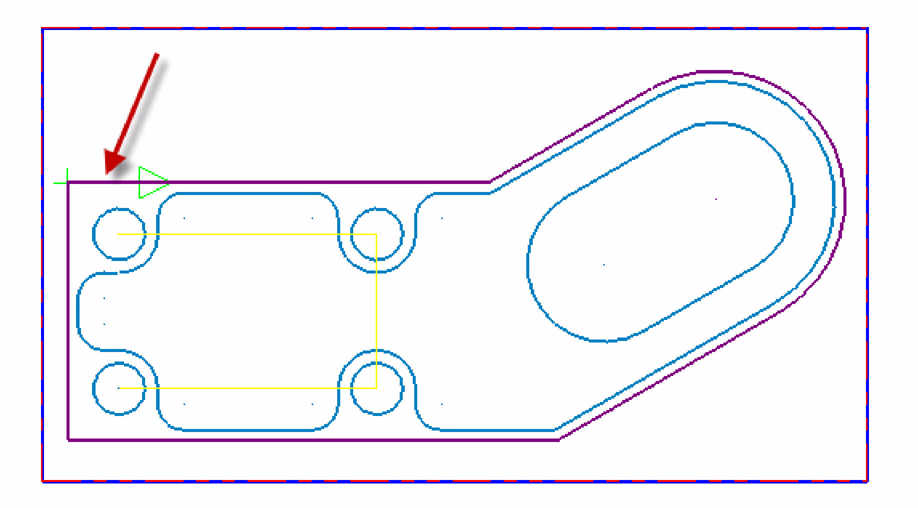
  Chaining is the process of selecting one or more entities and linking them together in a specific order and direction. The entities must have adjoining endpoints and may be open or closed shapes.



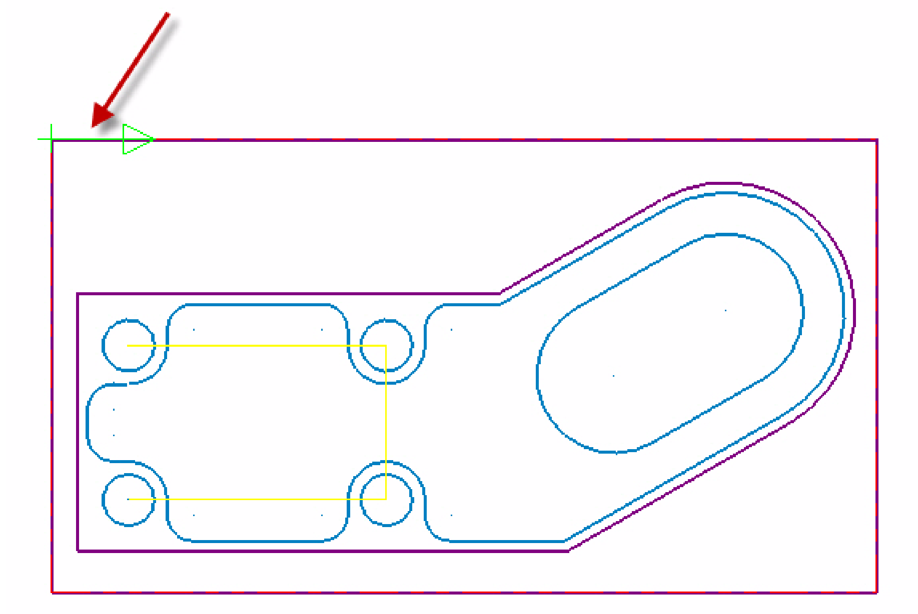
2 Press **[Alt+S]** to turn off shading and make it easier to select the chains.

3 Click the outside contour of the part to chain it. The chaining arrow should point clockwise. If the chaining direction arrow is pointing counterclockwise, click the **Reverse**

 button on the Chaining dialog box.

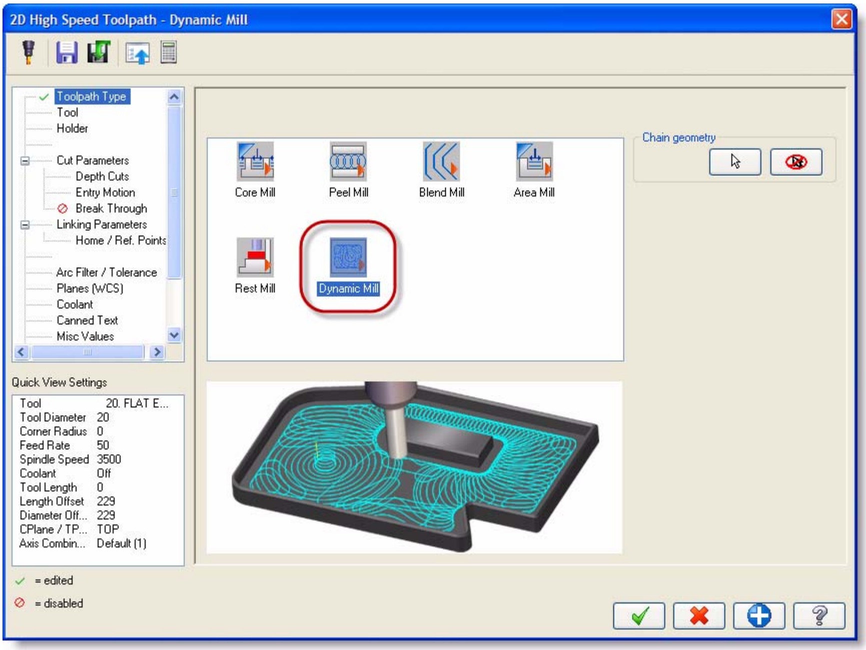


4 Click the top of the stock boundary as the second chain. This second chain represents the size of the stock material. The chaining arrow should again point clockwise.



5 Click **OK** on the Chaining dialog box to chain the part. The Chaining dialog box closes and the 2D High Speed Toolpath dialog box opens.

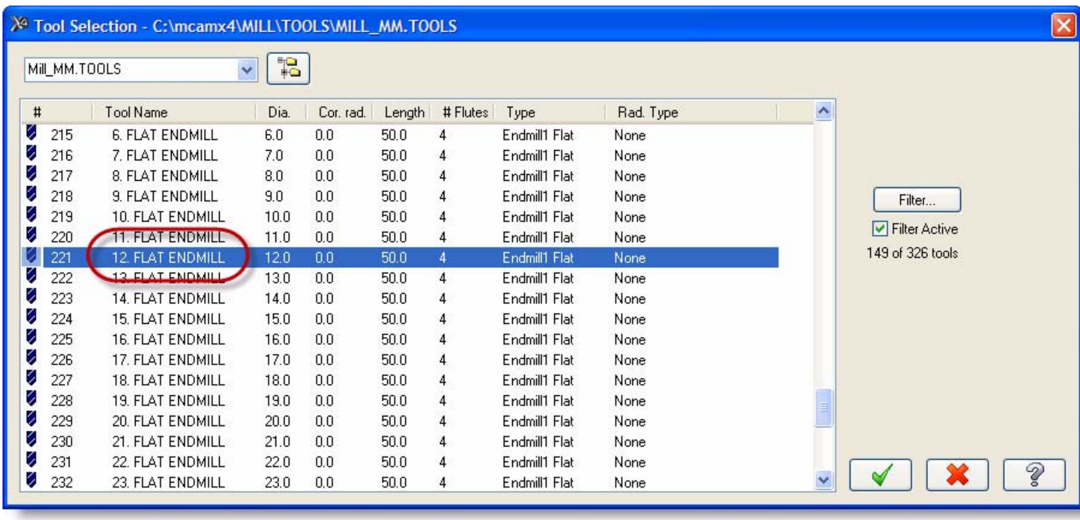
6 Select the **Dynamic Mill** toolpath type. The Tree View area updates with a list of parameters for dynamic mill toolpaths.



7 Click the **Tool** page in the Tree View list to select a tool for this toolpath.

8 Click the **Select library tool** button. The default metric tool library opens.

9 Select the **12mm** diameter flat endmill, and click **OK**.

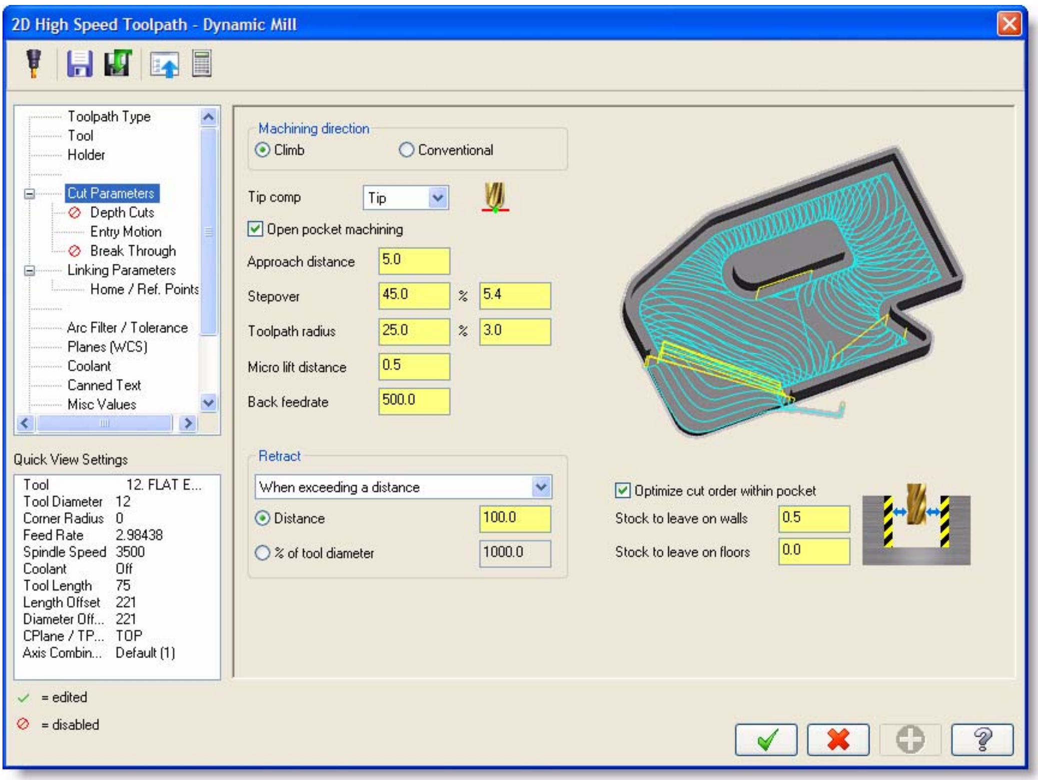


Setting the Dynamic Mill Toolpath Parameters

1 Click the **Cut Parameters** page in the Tree View area to enter values for different cutting parameters and compensation options.

2 Set the following parameters:

- Select the **Open pocket machining** check box (**Use dynamic core mill passes** in X4 MU1). When selected, Mastercam considers the largest outermost closed chain as the edge of material that can be removed by the toolpath. All other inner closed chains are machined as standing core features; they are considered islands to avoid.
- Enter **5.0** for the **Approach distance,** which adds the specified absolute distance to the beginning of the toolpath's first cut.
- Enter **25%** for the **Toolpath radius**. This parameter is used to calculate 3D arc moves and reduce sharp corner motion between passes.
- Enter **0.5** for the **Micro lift distance**. *Microlifts* are slight lifts during back moves that help clear chips and minimize excessive tool heating.
- Select **When exceeding a distance** from the **Retract** drop-down list, select the **Distance** radio button, and enter **100.0**. Mastercam adds retract motion when the next cut begins at a distance greater than the distance you define.
- Enter **0.5** for the **Stock to leave on walls**.This leaves 0.5mm of stock on the outer walls.



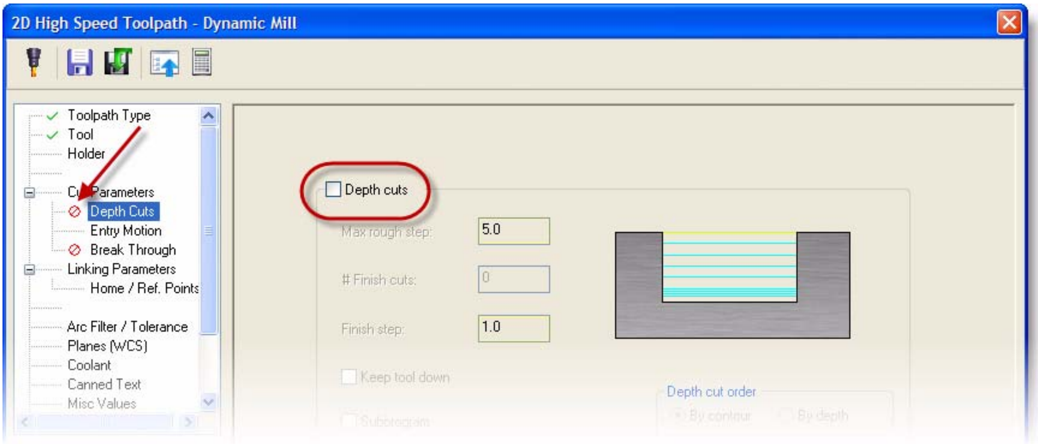
3 Click the **Depth cuts** page in the Tree View area.

TIP: Depth cuts divide the total depth of a toolpath into smaller Z-axis cuts to reduce tool wear. You can enter a maximum rough step, and Mastercam divides the total depth into equal steps. Or you can enter the exact number of finish steps and the size of each finish step. For more details, please refer to the Mastercam Help.

---

4 Deselect the **Depth cuts** check box to turn off depth cuts for this toolpath. Notice the red circle and a slash that displays next to the Depth Cuts page, indicating that the options on this page are all turned off.

Dynamic mill toolpaths utilize the full flute length of your tool, so dividing the toolpath into smaller cuts is not necessary.



5 Click the **Entry Motion** page in the Tree View list to set how and where the tool enters the stock.

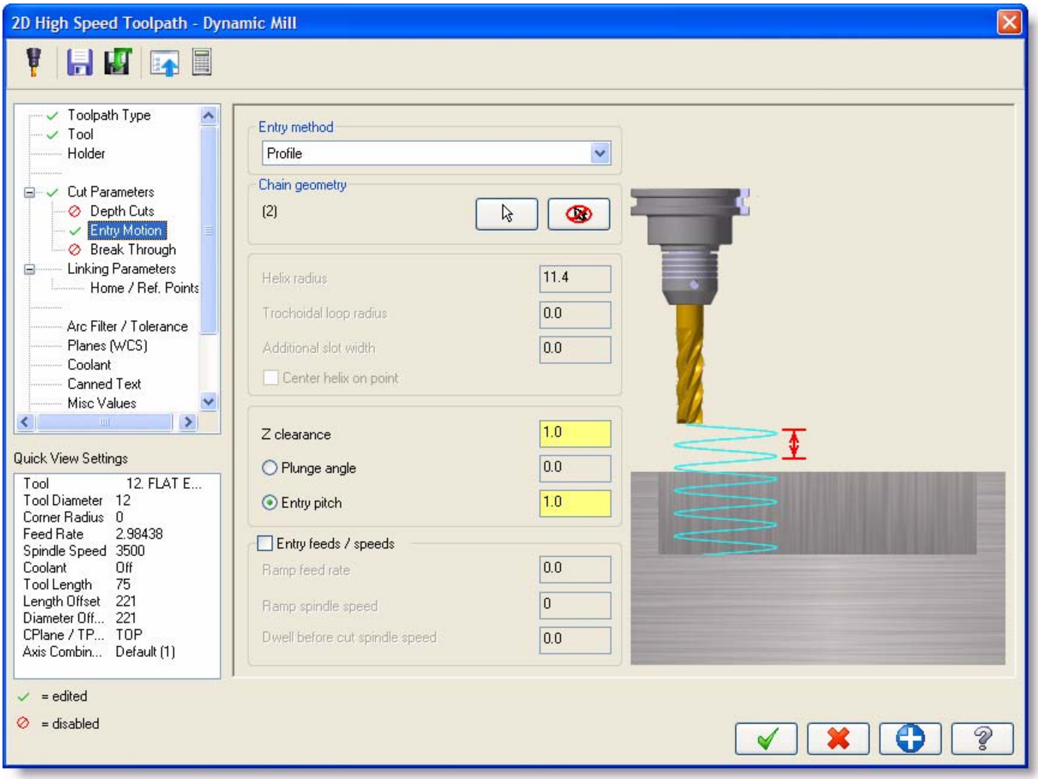
Note: The open pocket machining method enters from outside the material - these entry parameters are only used if Mastercam encounters a closed pocket.

---

6 Set the following parameters:

- Select **Profile** from the **Entry method** drop-down list. This entry method creates a ramp entry based on the shape of the offset pocket. The slot is cleared by taking lighter cuts in the Z axis. Subsequent cuts are properly engaged at the full cut depth.
- Enter **1.0** for the **Z clearance**. This is an extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material, so that it enters the material smoothly at the plunge angle.
- Select the **Entry pitch** radio button and enter **1.0** for the distance. This controls the entry descent by the pitch value you define. Mastercam lowers the tool by the pitch

value for every complete revolution of the entry motion, ensuring the tool is never buried by more than the specified pitch.

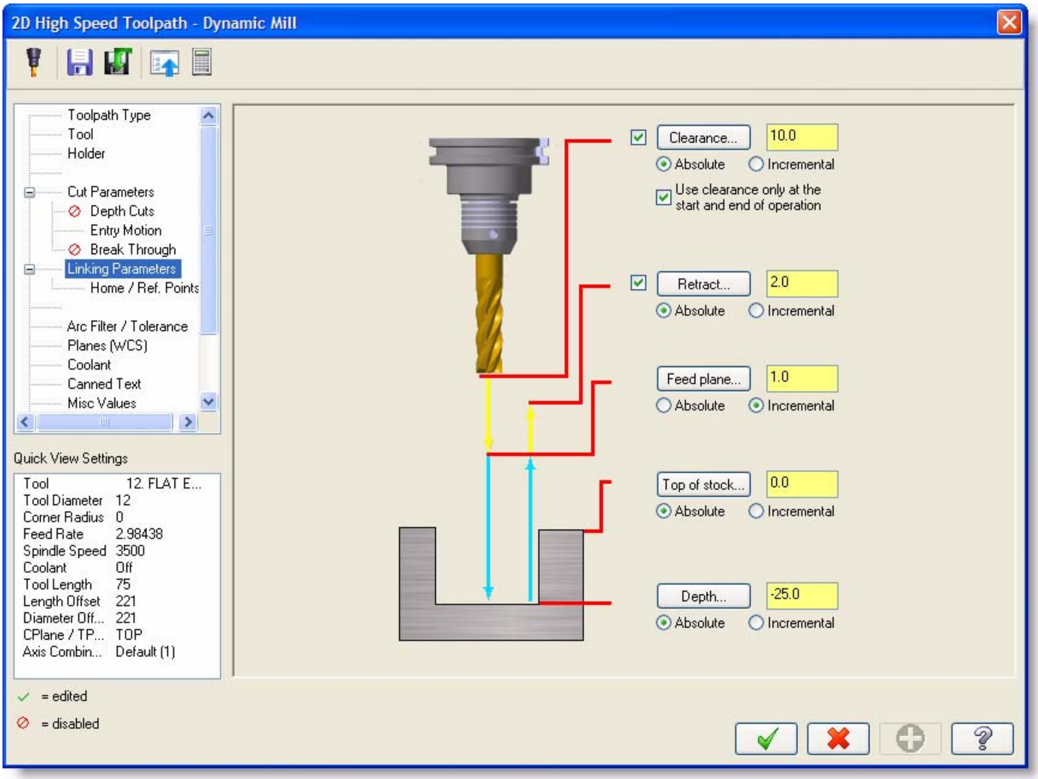


7 Click the **Linking Parameters** page in the Tree View list to set important heights such as clearance, retract, and feed plane, as well as the final toolpath depth.

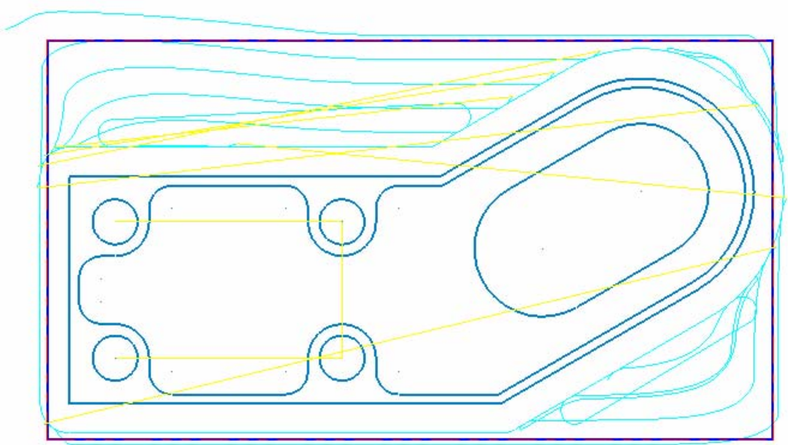
8 Set the following parameters:

- Select the **Clearance** check box and enter **10.0** for the height, which is the height at which the tool moves to and from the part.
- Select the **Use clearance only at the start and end of operation** check box, so the toolpath rapids to the clearance height only at the start and end of the toolpath.
- Enter **2.0** for the **Retract** height and select the **Absolute** radio button. This is the height that the tool moves up to before the next tool pass. By selecting Absolute, the retract height is always measured from 0,0,0, not relative to any selected geometry.
- Enter **1.0** for the **Feed plane** height, which is the height that the tool rapids to before changing to the plunge rate to enter the part.
- Enter **-25.0** for the **Depth**. This value determines the final machining depth and the lowest depth that the tool descends into the stock. In this case, the depth is -25mm, or 25 mm below the top of the part.

TIP: The default depth is the depth of the selected geometry.

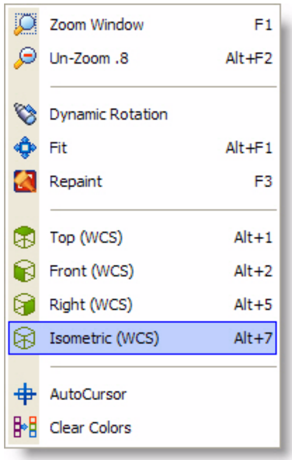


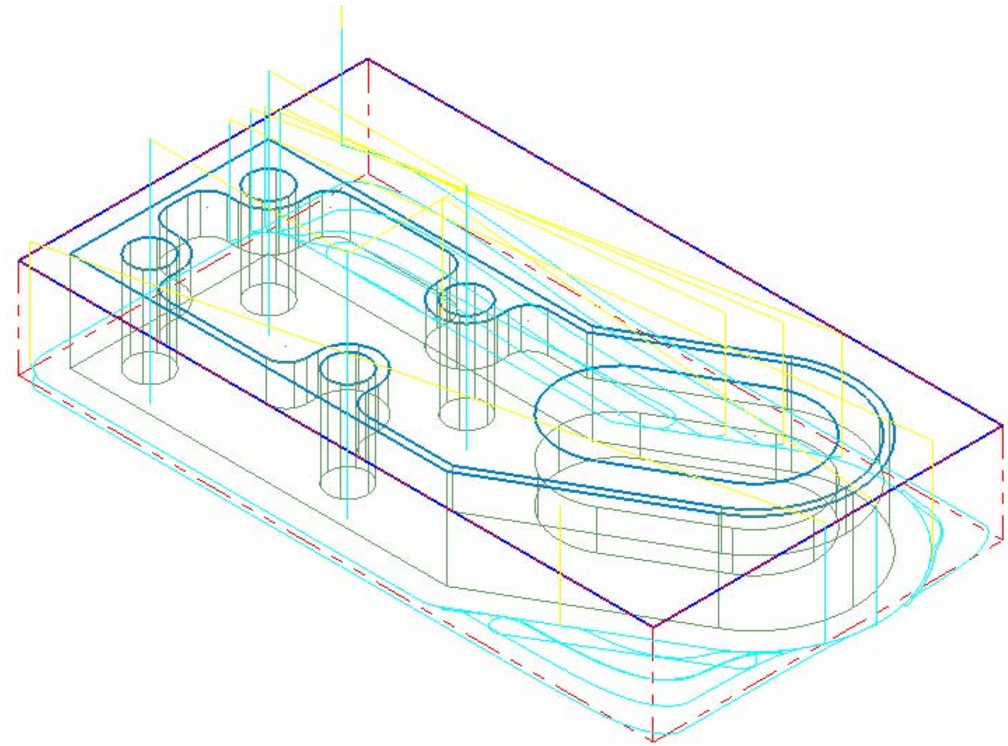
9 Click **OK** to generate the toolpath.



10 Right-click in the graphics window and choose **Isometric (WCS)** from the menu to view the part and toolpath in the isometric view.

The toolpath cleans most of the material outside the part using smooth, efficient tool motion.





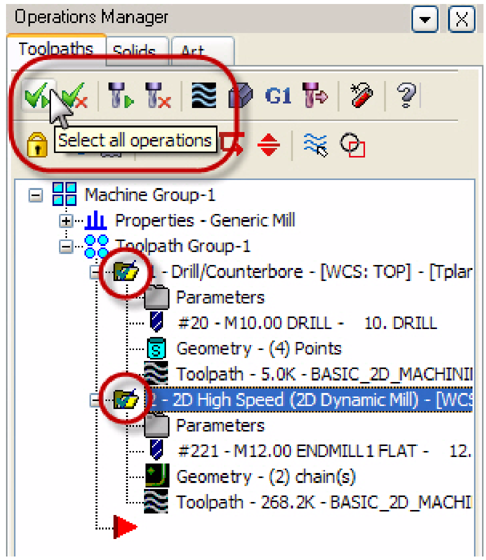
11 Right-click in the graphics window again and choose **Top (WCS)** from the menu to view the part and toolpath in the top view.

12 Choose **File**, **Save** from the Mastercam menu or click the **Save** button  to save your part.

Exercise 2: Viewing Your Toolpaths

In this exercise, you will temporarily turn off the display for the drill and dynamic mill toolpaths so that you easily see any additional toolpaths you create.

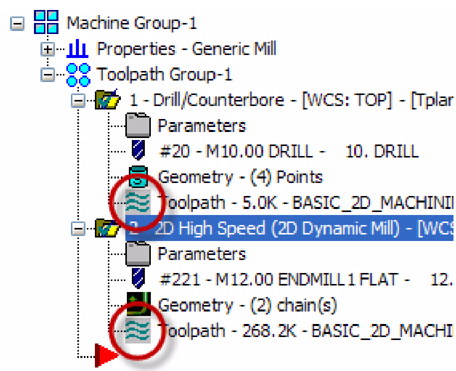
1 In the Toolpath Manager, click the **Select all operations** button. Check marks display on the yellow folders of each toolpath, which indicates that they are selected.



2 Press [**T**] to toggle off the toolpath display of both toolpaths in the graphics window.

This function toggles the visibility of toolpaths on and off in the graphics window so that you can view only specific toolpaths.

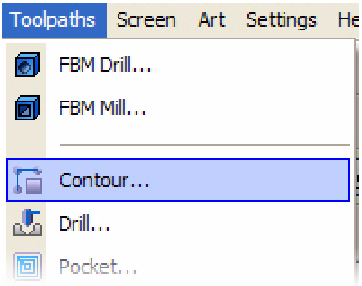
The **Toolpath** icon changes to gray when the toolpath display is toggled off.



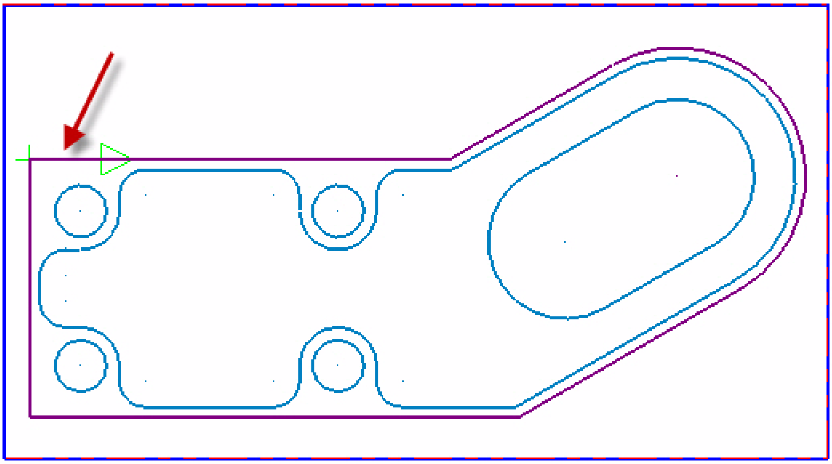
Exercise 3: Creating a Contour Toolpath

To get the final finish on the outer walls with no stock remaining, you will create a contour toolpath. Contour toolpaths remove material along a path defined by a chain of curves. Contour toolpaths only follow a chain; they do not clean out an enclosed area.

1 From the Mastercam menu, choose **Toolpaths**, **Contour**. The Chaining dialog box opens.



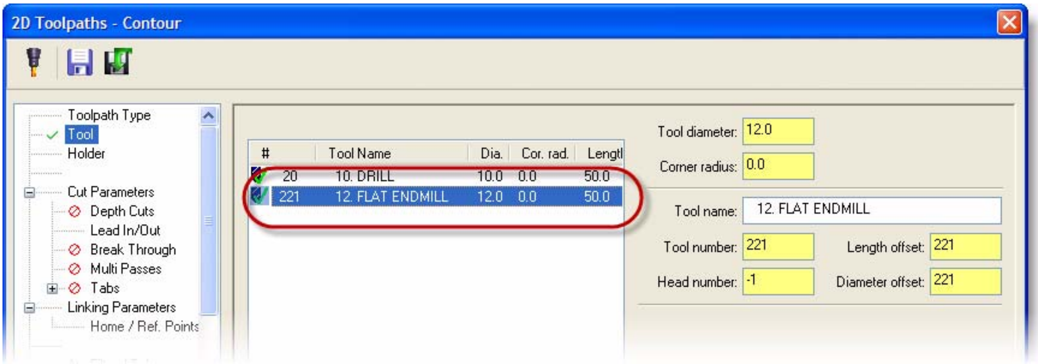
2 Click the outside contour of the part to chain it. The chaining arrow should point clockwise. If the chaining direction arrow is pointing counterclockwise, click the **Reverse**  button on the Chaining dialog box.



3 Click **OK** on the Chaining dialog box to chain the part. The Chaining dialog box closes and the 2D Toolpaths - Contour dialog box opens.

4 Click the **Tool** page in the Tree View list to select a tool for this toolpath.

5 Select the **12mm** diameter flat endmill you used for the dynamic mill toolpath.



Setting the Contour Toolpath Parameters

1 Click the **Lead In/Out** page in the Tree View area to enter values for the entry and exit moves.

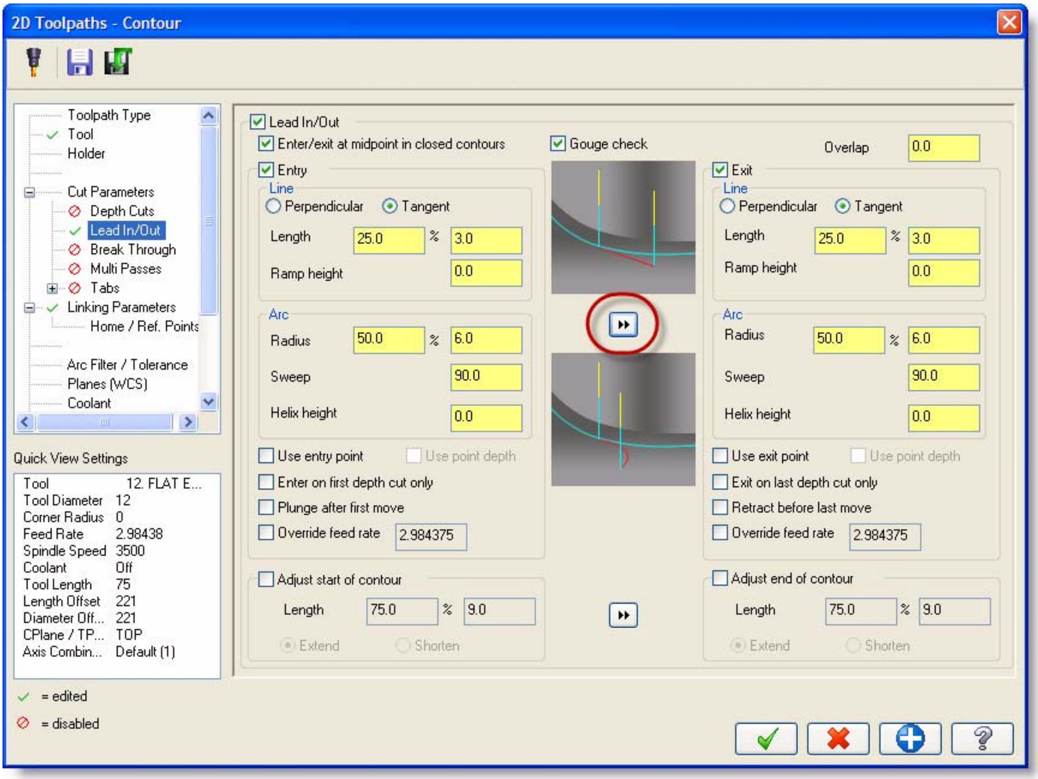
Lead in/out or entry/exit moves are a combination of lines and arcs at the beginning and end of a 2D or 3D contour toolpath. They control how the tool approaches and pulls away from the toolpath.

Mastercam places entry and exit lines relative to the entry and exit arcs. If both an entry line and entry arc are defined, the line is machined first. If both an exit line and exit arc are defined, the arc is machined first. The lead in/out entry points may differ based on where you clicked the geometry to chain it.

2 Set the following parameters:

- Enter **25%** for the **Entry Line Length**.
- Enter **50%** for the **Entry Arc Radius**.

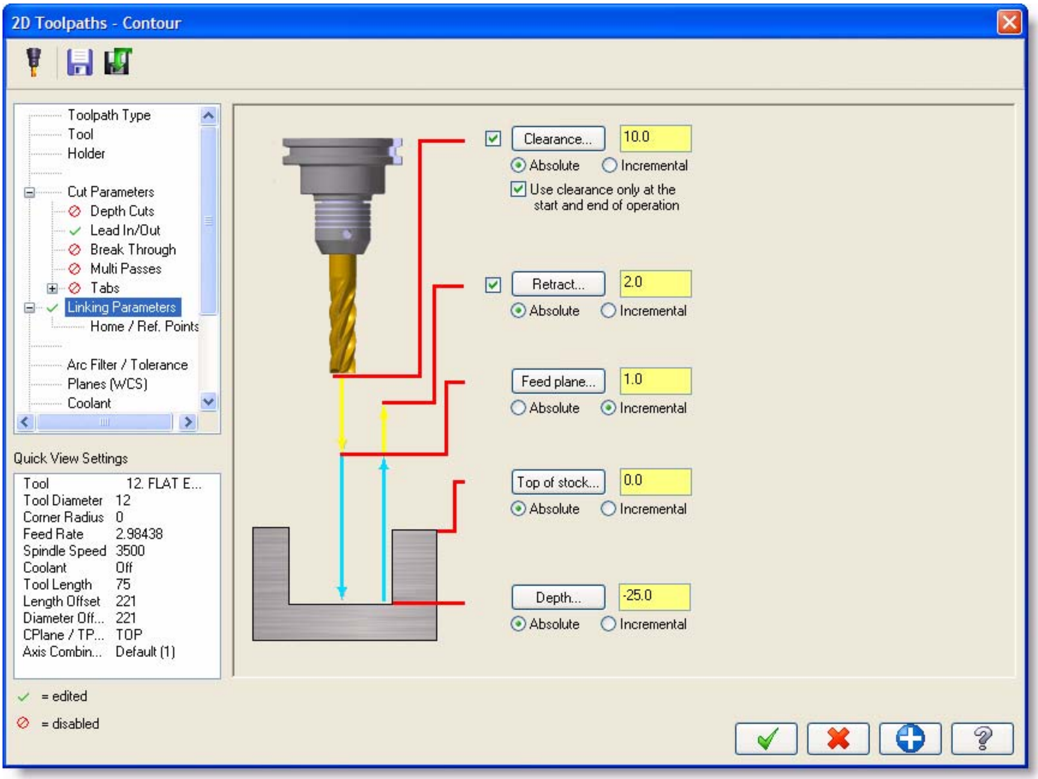
- Click the arrow button in the center of the page to copy the values to the Exit parameters.



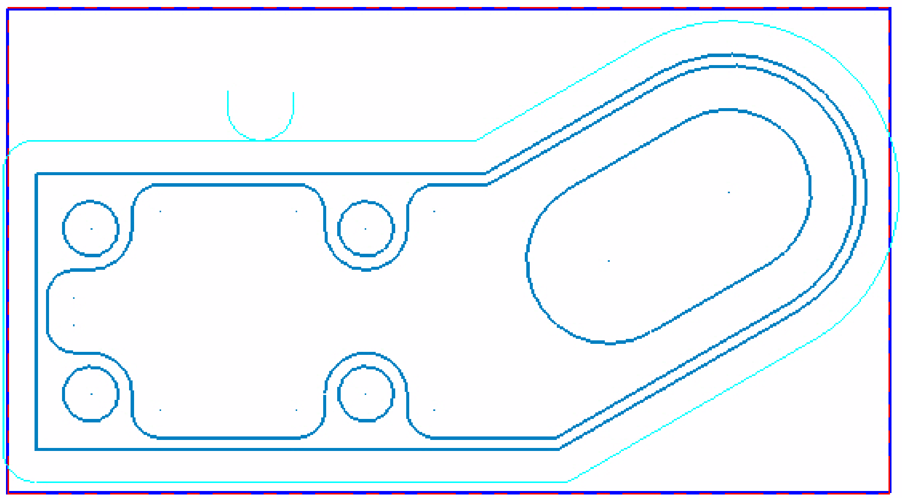
3 Click the **Linking Parameters** page in the Tree View list.

4 Set the following parameters:

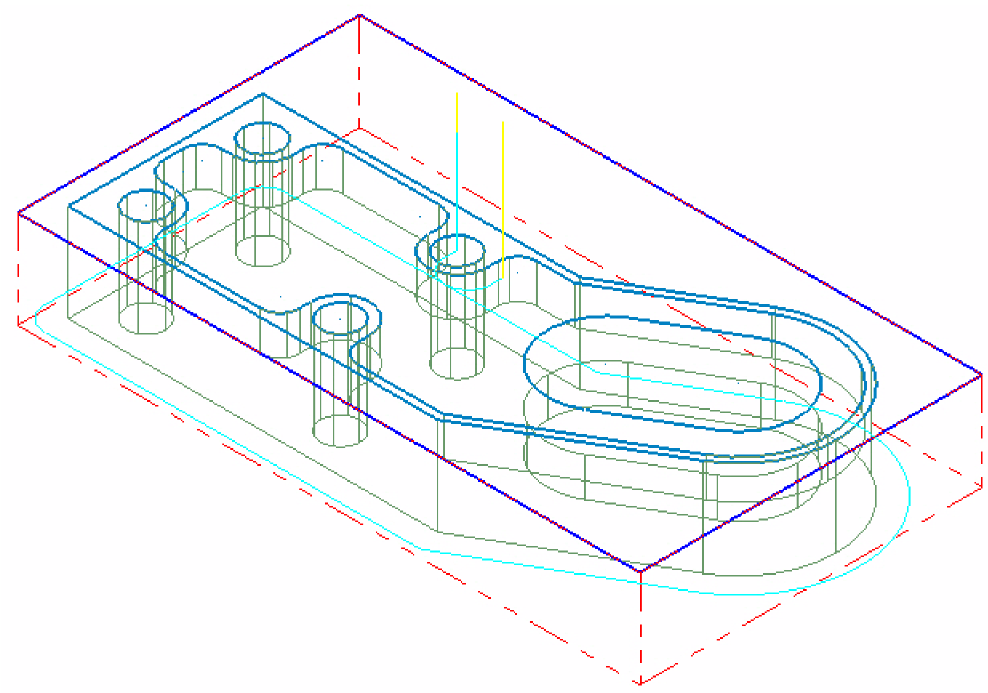
- Select the **Clearance** check box and enter **10.0** for the height.
- Select the **Use clearance only at the start and end of operation** check box.
- Enter **2.0** for the **Retract** height and select the **Absolute** radio button.
- Enter **1.0** for the **Feed plane** height.
- Enter **-25.0** for the **Depth**.



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



6 Right-click in the graphics window and choose **Isometric (WCS)** from the menu to view the part and toolpath in the isometric view. The contour toolpath removes the 0.5mm left behind by the dynamic mill toolpath and machines the outer wall to its final size.



7 Right-click in the graphics window again and choose **Top (WCS)** from the menu to view the part and toolpath in the top view.

8 Choose **File**, **Save** from the Mastercam menu or click the **Save** button  to save your part.

The stock has been removed from the outer areas of the part. In the next lesson, you will remove material from the inner areas of the part, including the slot.

Machining Inside the Part

The next step in machining the tutorial part is to clean out material from the inside. You will use two toolpaths specifically geared towards the areas inside the part.

Lesson Goals

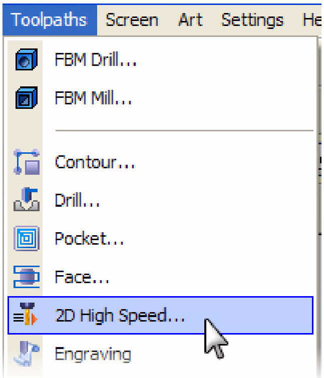
- Create a dynamic mill toolpath (including chaining entities, choosing tooling, and setting machining values).
- Create a slot mill toolpath (including chaining entities, choosing tooling, and setting machining values).

Exercise 1: Creating a Dynamic Mill Toolpath

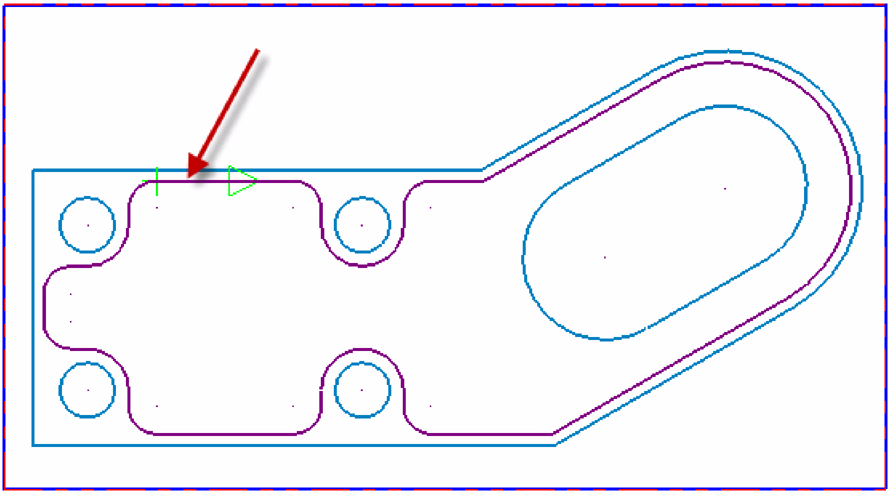
In this exercise, you create a dynamic mill toolpath to clean out the large interior shape of the tutorial part. This toolpath type works well for all pocket shapes because the tool goes right to the final depth, and you can cut using the whole flute length instead of stepping down.

1 Select the **Contour** toolpath in the Toolpath Manager and press **[T]** to turn off the toolpath display. This makes it easier to see the new toolpaths you create.

2 From the Mastercam menu, choose **Toolpaths**, **2D High Speed**. The Chaining dialog box opens.

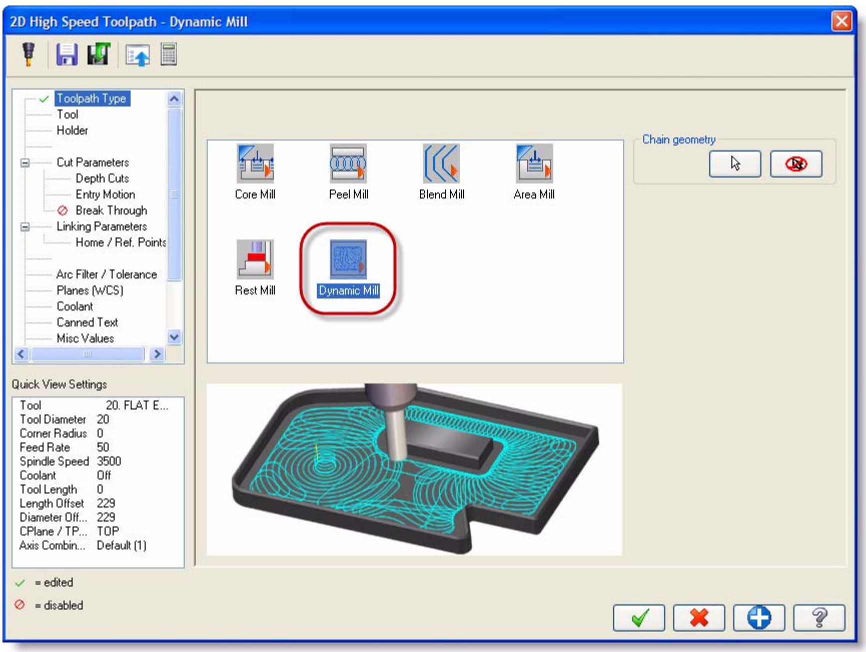


3 Click the inner contour of the part to chain it.



4 Click **OK** on the Chaining dialog box to chain the part. The Chaining dialog box closes and the 2D High Speed Toolpath dialog box opens.

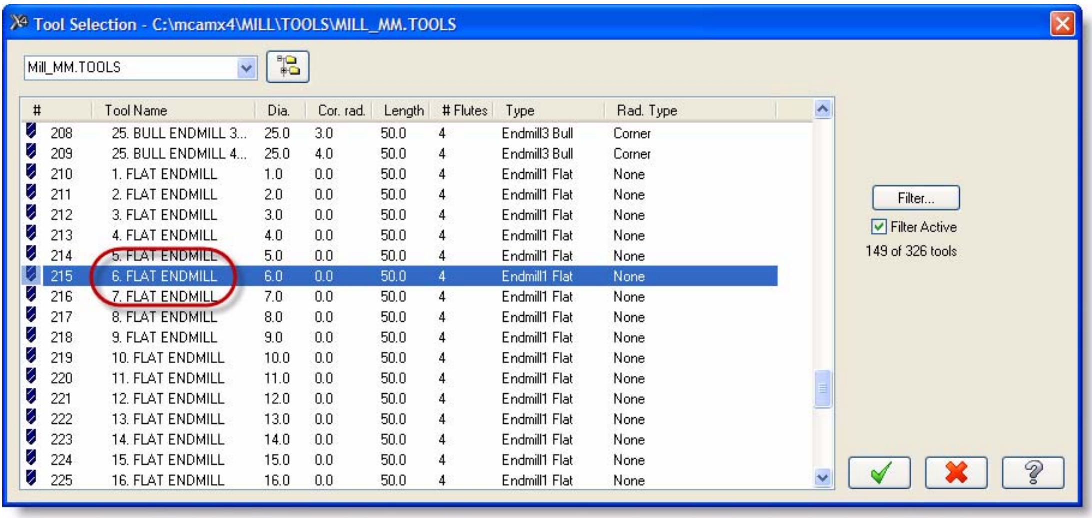
5 Select the **Dynamic Mill** toolpath type. The Tree View area updates with a list of parameters for dynamic mill toolpaths.



6 Click the **Tool** page in the Tree View list to select a tool for this toolpath.

7 Click the **Select library tool** button. The default metric tool library opens.

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



Setting the Dynamic Mill Toolpath Parameters

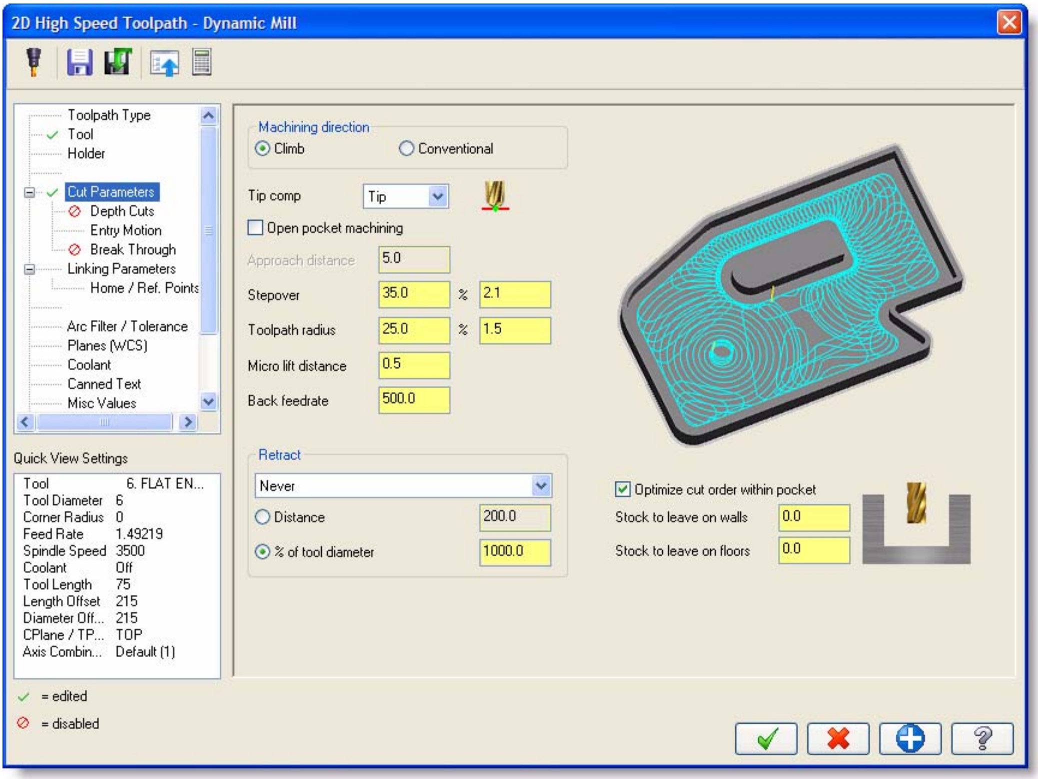
1 Click the **Cut Parameters** page in the Tree View area to enter values for different cutting parameters and compensation options.

TIP: Notice that the parameter values are the same as the last dynamic mill toolpath you created. Many of the Mastercam dialog boxes retain their previous settings, saving you from having to reenter data, reselect function buttons, or reselect options in a drop-down list. The settings stay in their “last used” state for the remainder of the Mastercam session or until you change them.

2 Set the following parameters:

- Deselect the **Open pocket machining** check box (**Use dynamic core mill passes** in X4 MU1).
- Enter **35%** for the **Stepover**.
- Select **Never** from the **Retract** drop-down list.

- Enter **0 (zero)** for the **Stock to leave on walls**.

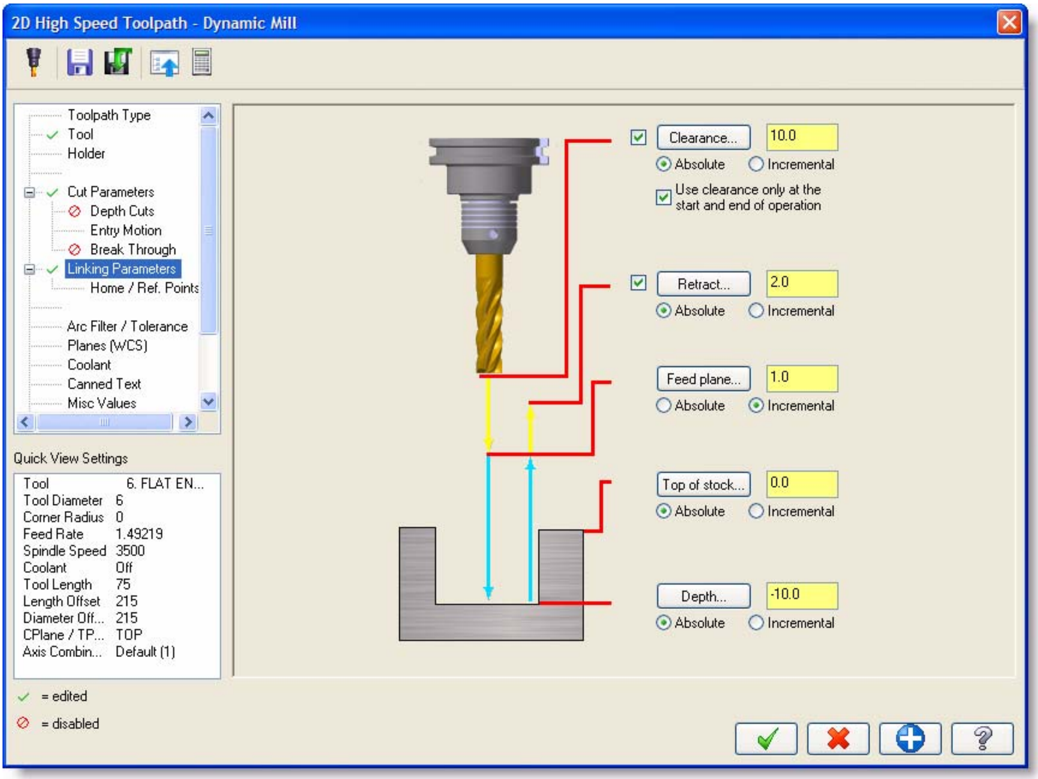


Note: See ***Setting the Dynamic Mill Toolpath Parameters*** *on page 18 for more information on these parameters.*

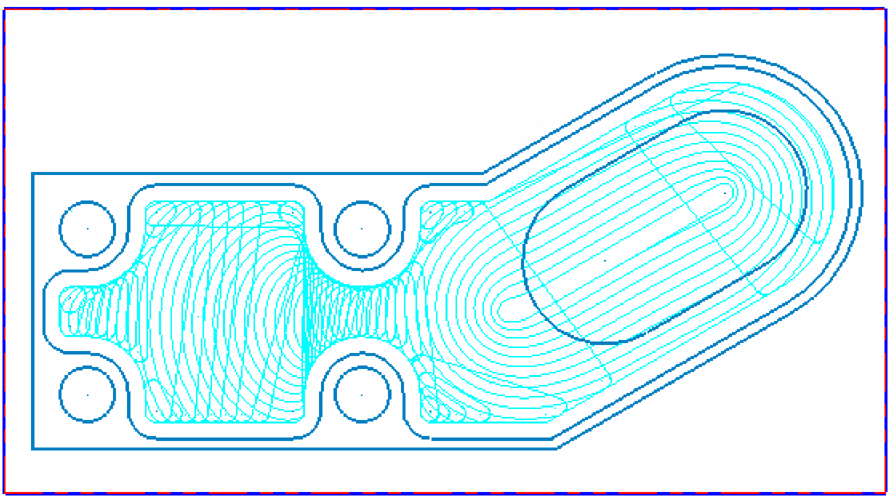
---

3 Click the **Linking Parameters** page in the Tree View list to set important heights such as clearance, retract, and feed plane, as well as the final toolpath depth.

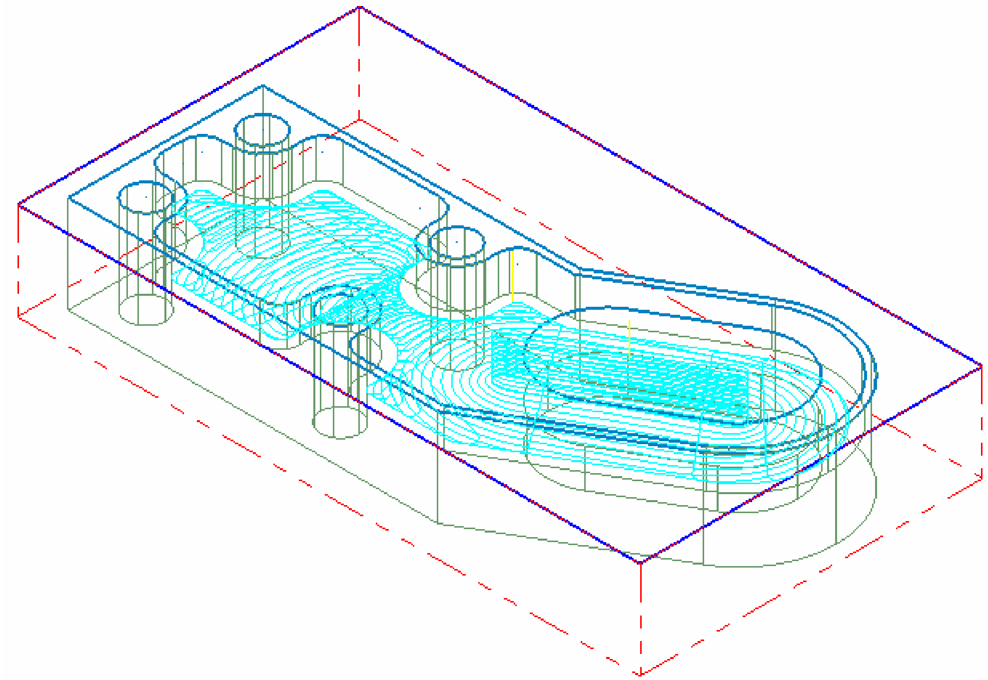
4 Enter **-10.0** for the **Depth**.



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



6 Right-click in the graphics window and choose **Isometric (WCS)** to view the part and toolpath in the isometric view. The toolpath cleans off the floor of the part, but doesn't completely machine the additional slot.



7 Right-click in the graphics window again and choose **Top (WCS)** from the menu to view the part and toolpath in the top view.

8 Choose **File**, **Save** from the Mastercam menu or click the **Save** button  to save your part.

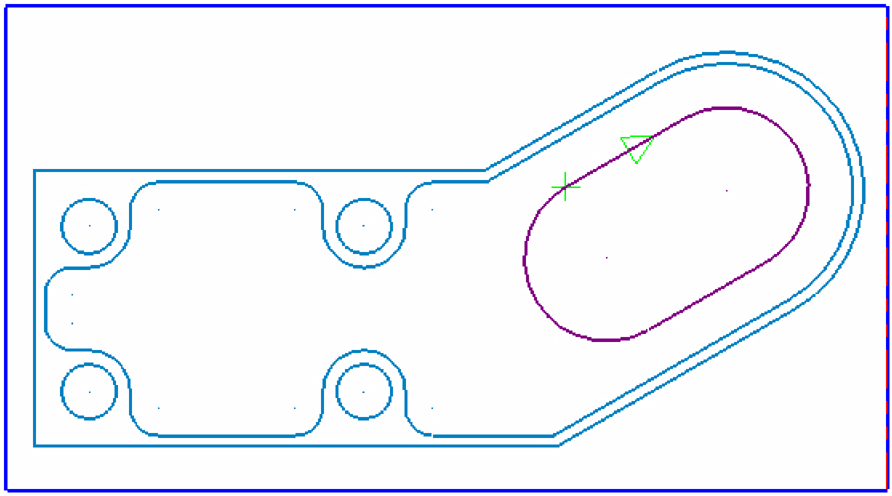
Exercise 2: Creating a Slot Mill Toolpath

Slot mill toolpaths are designed to efficiently machine obround slots. These are slots that consist of 2 straight lines and two 180-degree arcs at the ends, such as the remaining slot on your part.

1 Select the second **Dynamic Mill** toolpath in the Toolpath Manager and press **[T]** to turn off the toolpath display. This makes it easier to see the new toolpaths you create.

2 From the Mastercam menu, choose **Toolpaths**, **Circle Paths, Slot Mill**. The Chaining dialog box opens.

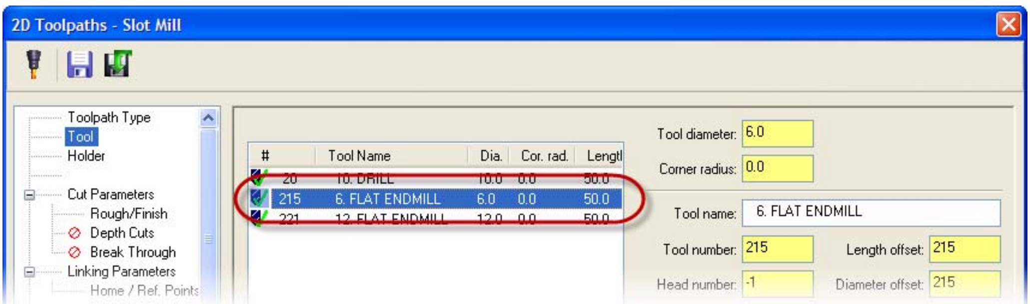
3 Click the slot to chain it. The chaining arrow should point clockwise.



4 Click **OK** on the Chaining dialog box to chain the part. The Chaining dialog box closes and the 2D Toolpaths - Slot Mill dialog box opens.

5 Click the **Tool** page in the Tree View list to select a tool for this toolpath.

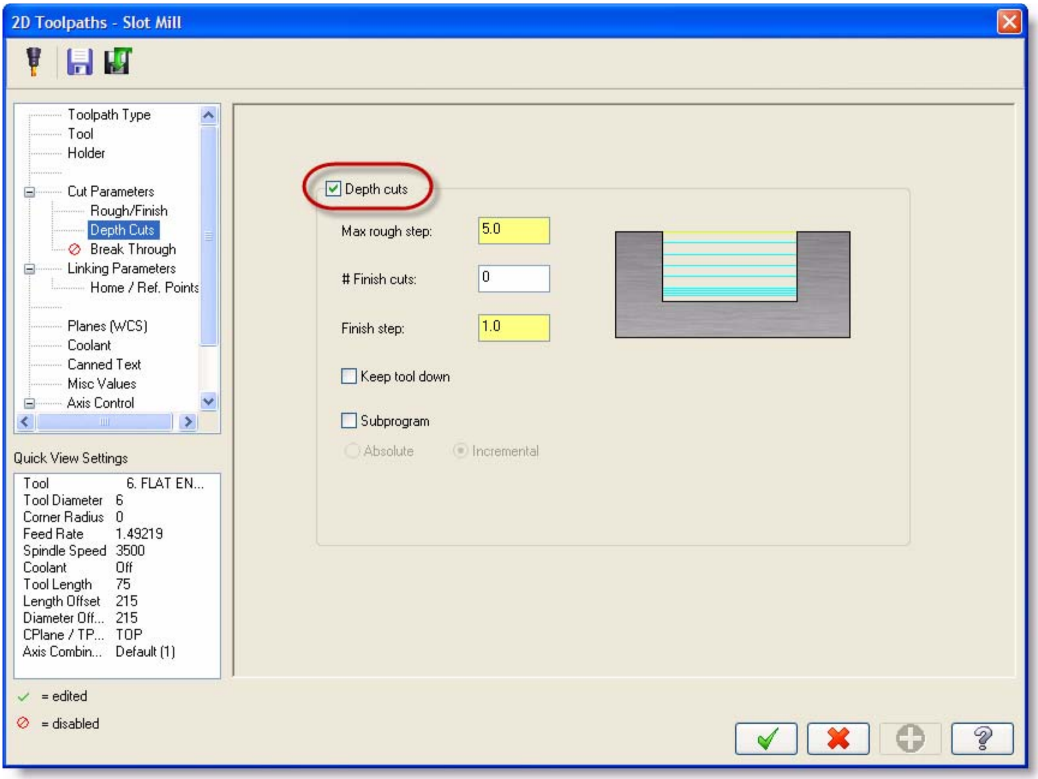
6 Select the **6mm** diameter flat endmill you used for the dynamic mill toolpath.



Setting the Slot Mill Toolpath Parameters

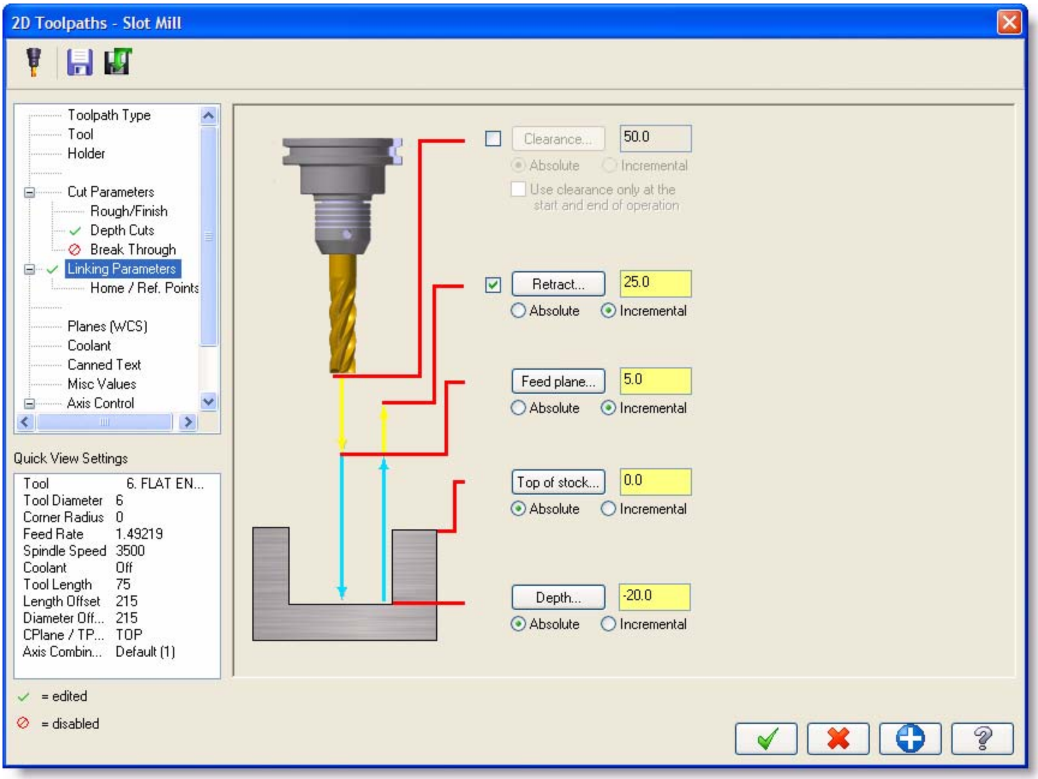
1 Click the **Depth Cuts** page in the Tree View area to enter values for different cutting parameters and compensation options.

2 Select the **Depth cuts** check box to turn on depth cuts. The default parameter values are used for this toolpath.

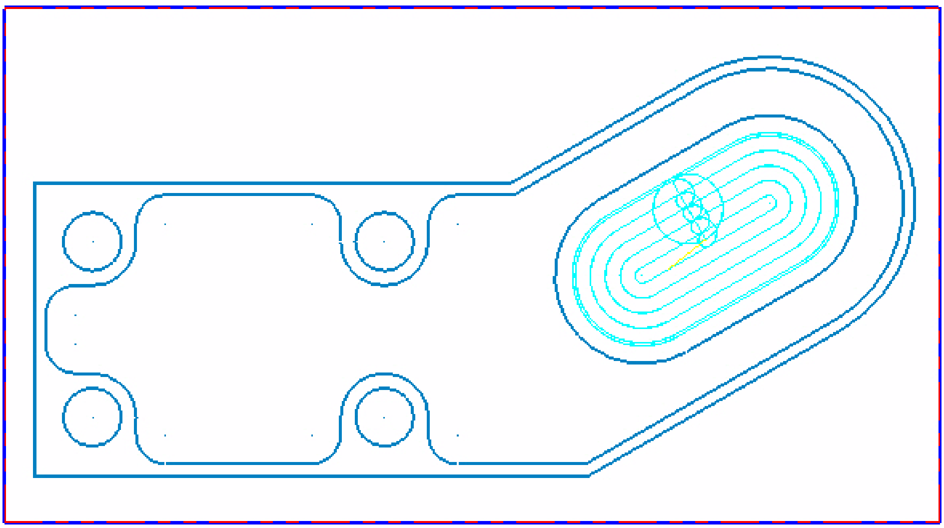


3 Click the **Linking Parameters** page in the Tree View list.

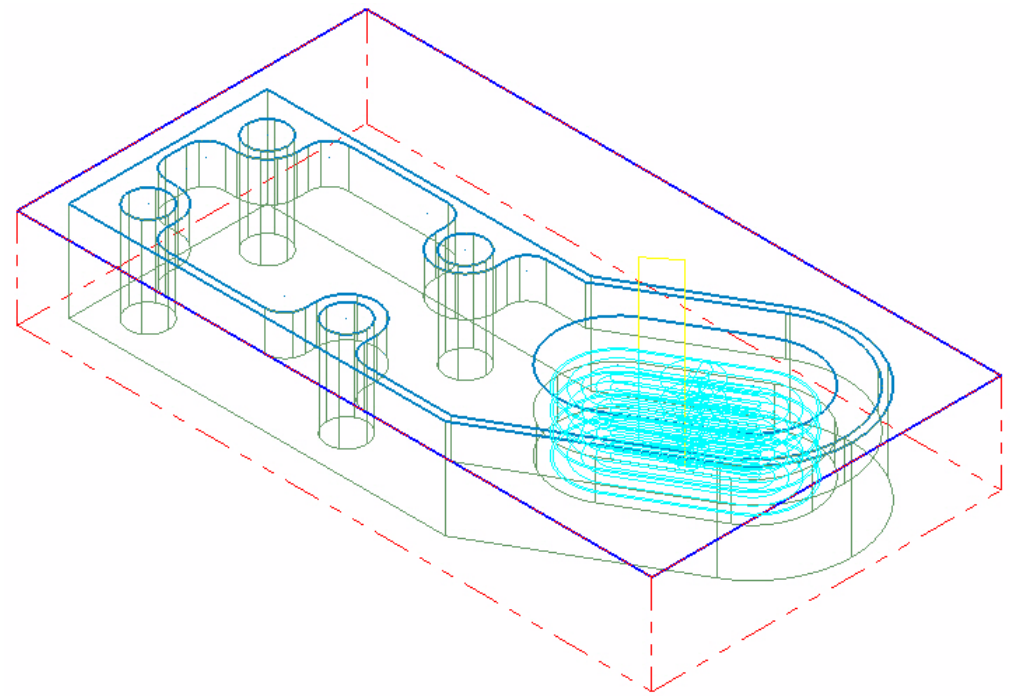
4 Enter **-20.0** for the **Depth**.



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



6 Right-click in the graphics window and choose **Isometric (WCS)** to view the part and toolpath in the isometric view. The slot is machined in four depth cuts.



7 Right-click in the graphics window again and choose **Top (WCS)** from the menu to view the part and toolpath in the top view.

8 Choose **File**, **Save** from the Mastercam menu or click the **Save** button  to save your part.

Now that you have machined all areas of the part, you can use Mastercam's Backplot and Verify functions to check your toolpaths before sending them to the machine tool.

LESSON 4

Previewing Toolpaths

Mastercam has several ways of viewing your toolpath motion before you actually machine your part. Visualizing the machining process for this part is an important step before sending the program to your machine control.

Lesson Goals

- Backplot all toolpaths.
- Customize your backplot display.
- Verify all toolpaths.

Exercise 1: Backplotting All Toolpaths

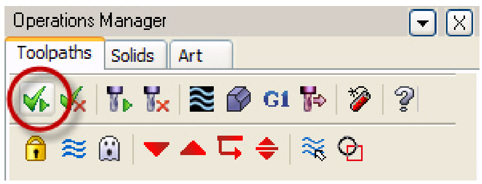
Backplot is a Mastercam function that allows you to see the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part.

Note: This lesson assumes that you have successfully completed Lessons 1 through 3 of the Basic 2D Machining tutorial and have saved the MCX file. If you have not, or if you think your completed part file is incorrect, open the `BASIC_2D_MACHINING_FINISH.MCX` *file provided with this tutorial.*

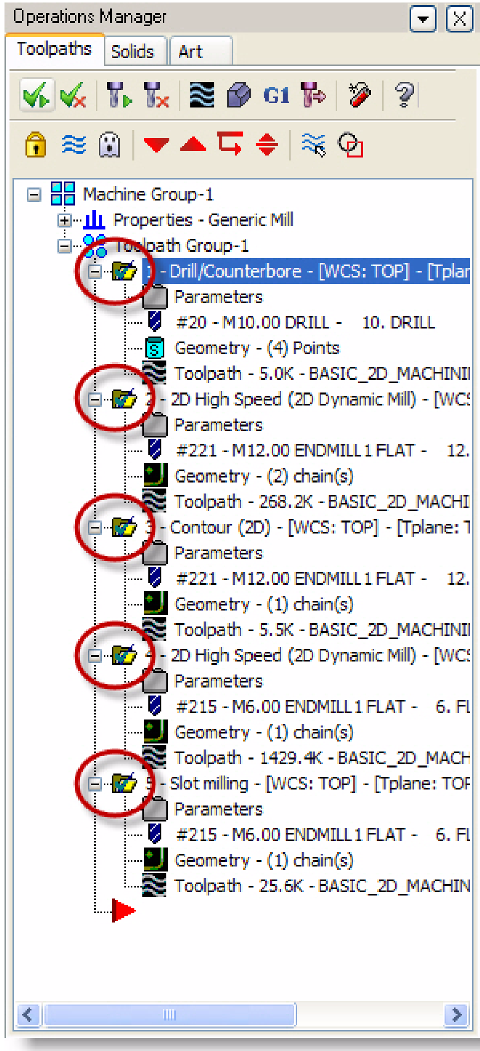
---

1 If necessary, open Mastercam and your part (see note above).

2 At the top of the Toolpath Manager, click the **Select all operations** button.



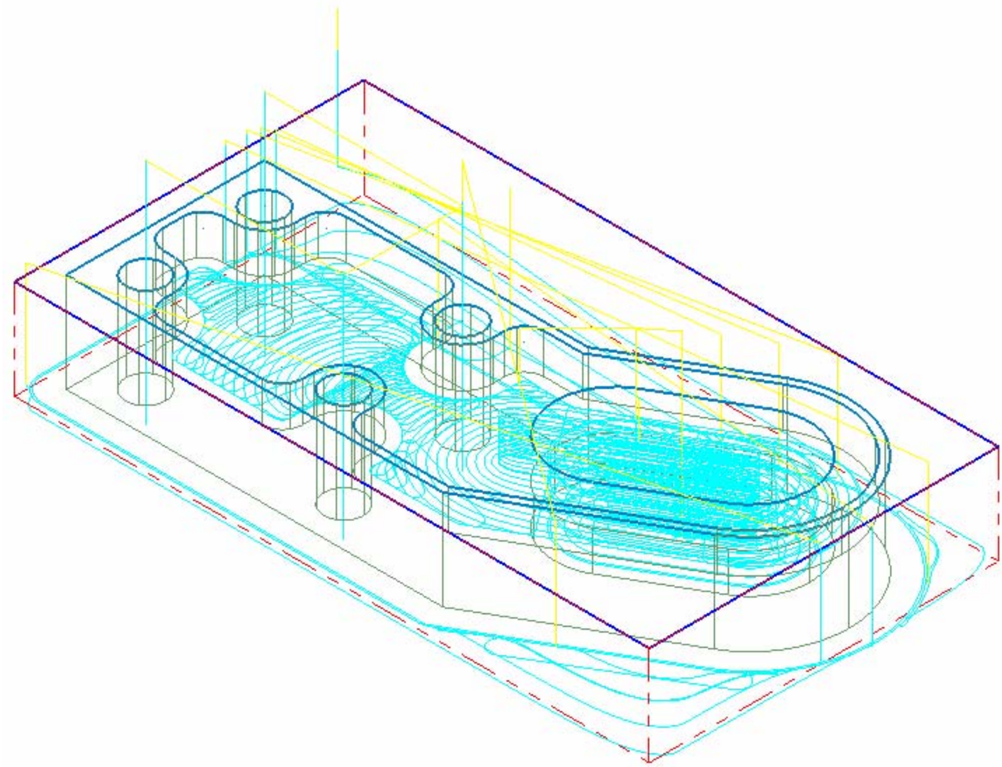
All five toolpath folders display a check mark.



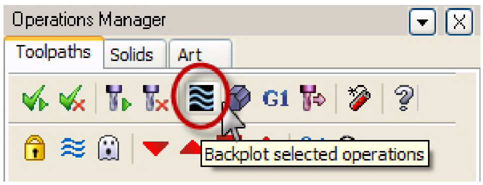
Note: Make sure that all toolpaths are set to display in the graphics window.
The Toolpath icon for each toolpath should be black.
If any of the toolpaths are not displayed, press **[T]**.

---

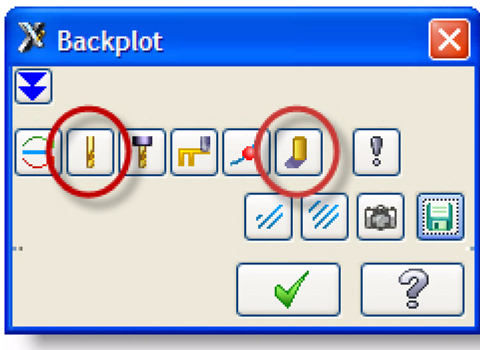
3 Right-click in the graphics window and choose **Isometric (WCS)** from the menu to view the part and all toolpaths in the isometric view.



4 In the Toolpath Manager, click the **Backplot selected operations** button. The Backplot dialog box and Backplot VCR bar open.

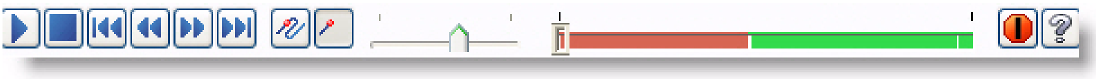


5 In the Backplot dialog box, select the **Display tool** and **Quick verify** buttons. These options will display a simulation of a tool and shade the toolpath during backplot.



TIP: To further customize your backplot display, choose other buttons on the Backplot dialog box. For example, choose the **Options** button  to open the Backplot Options dialog box. This dialog box lets you set various backplot parameters such as tool display, holder display, and tool motion colors.

6 Use the buttons and sliders on the Backplot VCR bar to backplot the operations. The **Play** button  begins the backplotting action. Click the **Help** button  on the VCR bar for more information on each of the controls.



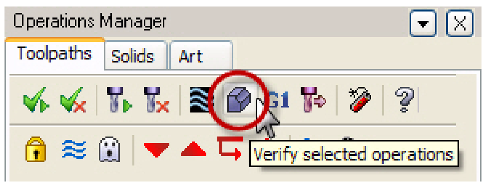
7 When finished, click **OK** on the Backplot dialog box to exit the backplot function.

TIP: The backplot display is easily customizable. See the Mastercam Help for details on each of the buttons, fields, and display options in the Backplot and the Backplot Options dialog boxes.

Exercise 2: Verifying All Toolpaths

Mastercam's ***Verify*** utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist. In this exercise, you simulate (verify) the machining of the part from a stock model display.

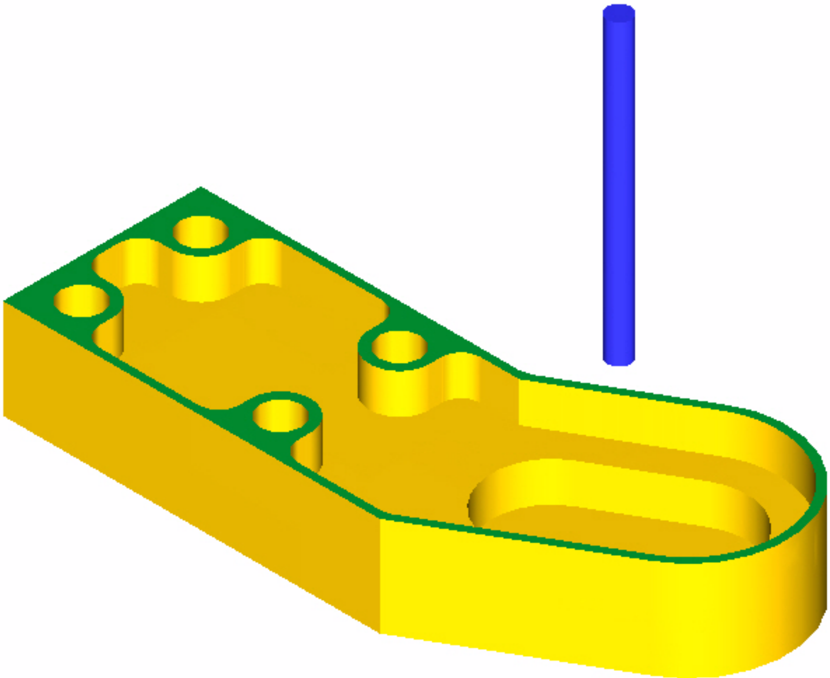
1 In the Toolpath Manager, make sure all operations are selected and click the **Verify selected operations** button. The Verify dialog box opens.



2 In the Verify dialog box, select the **Machine** button. The part, stock, and toolpaths are simulated.

TIP: Use the buttons, fields, and controls in the Verify dialog box to customize and manage the toolpath verification process. Click the **Help** button on the dialog box for details.

The following pictures shows the part after the verification is complete.



3 When finished, click **OK** on the Verify dialog box to exit the function.

You have verified that the toolpaths are correct, so now you can send the toolpath data to your machine tool in preparation for machining your part.

Posting Toolpaths

Post processing, or ***posting***, refers to the process by which the toolpaths in your Mastercam part files are converted to a format that can be understood by your machine tool's control (for example, G-codes). A special program called a ***post processor***, or ***post***, reads your Mastercam file and writes the appropriate NC code. Generally, every machine tool or control will require its own post processor, customized to produce code formatted to meet its exact requirements.

Lesson Goals

- Post all toolpaths to create NC files.

Exercise 1: Posting All Toolpath Operations

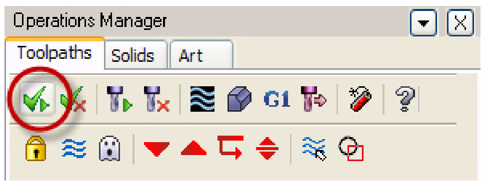
In this exercise, you post all toolpath operations for this part to an NC file, review/edit the code as necessary, and save the NC file.

Note: This lesson assumes that you have successfully completed Lessons 1 through 3 of this Basic 2D Machining tutorial module and have saved the MCX file. If you have not, open the `BASIC_2D_MACHINING_FINISH.MCX` *file provided with this tutorial.*

---

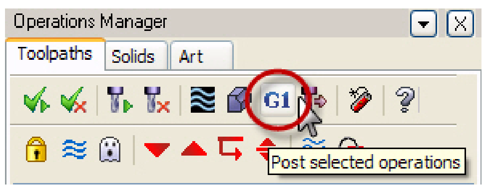
1 If necessary, open Mastercam and your part (see note above).

2 In the Toolpath Manager, click the **Select all operations** button.

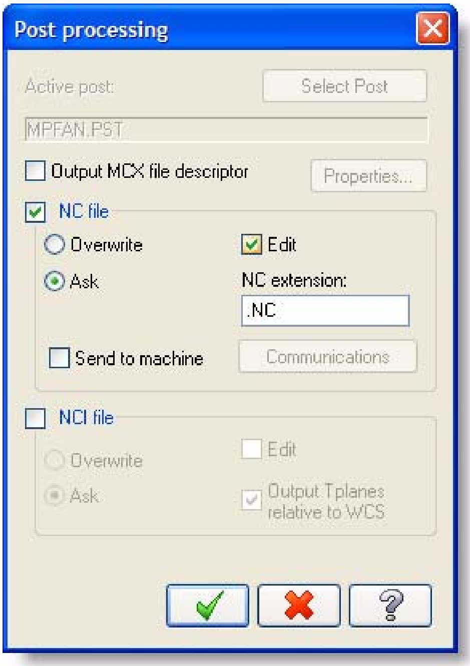


All five toolpath folders will display a check mark to indicate they are selected.

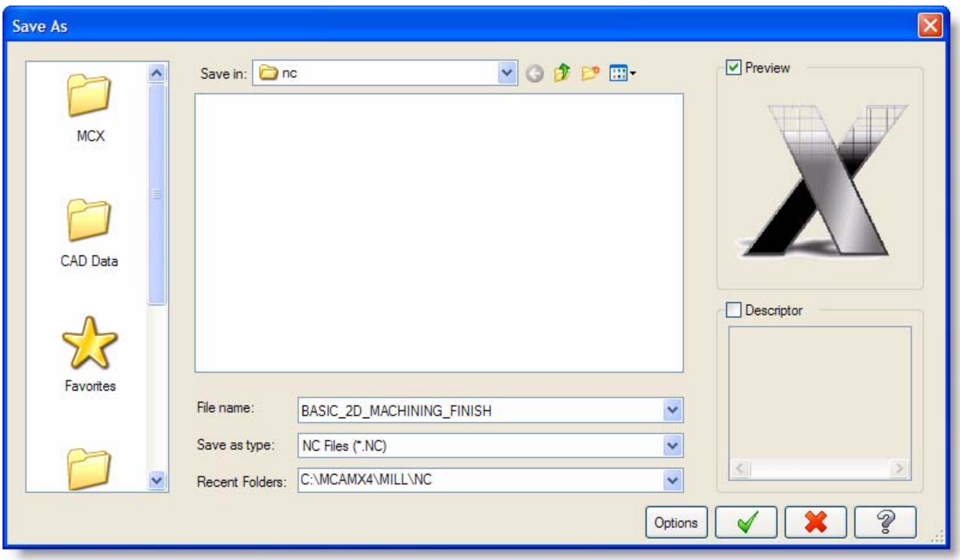
3 In the Toolpath Manager, click the **Post selected operations** button. The Post processing dialog box opens.



4 Set the post processing parameters as shown. These settings will ask if you want to save the NC file and will display the resulting file in your default text editor.



5 Click **OK**. The Save As dialog box opens.



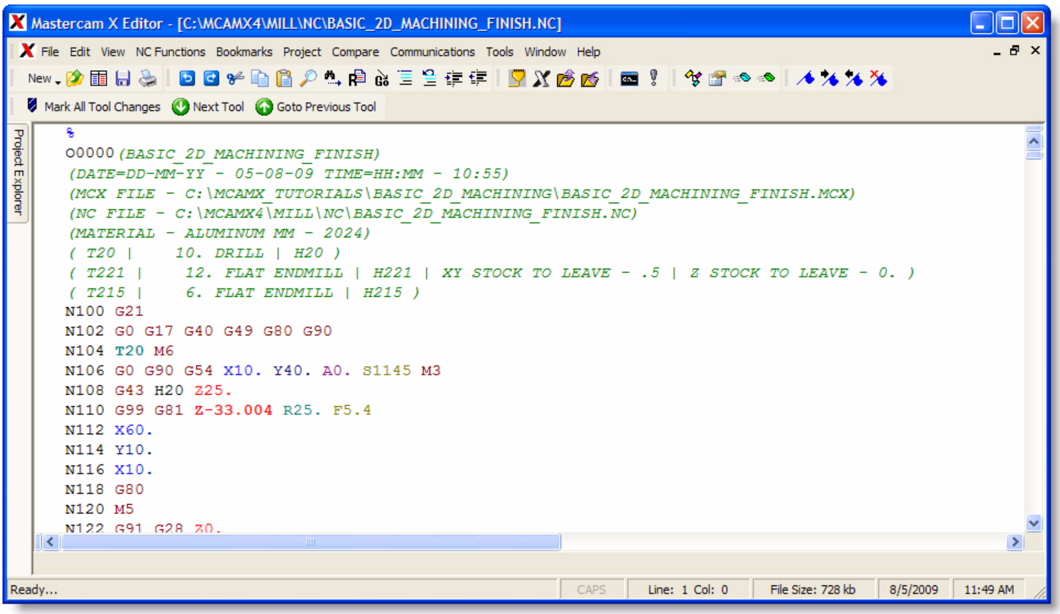
6 Click **Save** to save the NC file in the default location with the recommended file name.

Notes:

- *Posting may take several minutes. When it is complete, the NC code will display in your default editor/communicator. This tutorial uses Mastercam Editor to display the NC code.*
- *Producing the correct NC code for your machine and application depends on properly configuring the machine definition, control definition, and .PST file. For detailed information on machine definitions, control definitions, and posting, please see the following documentation supplied with Mastercam:*
  - Mastercam Help
  - *Mastercam X4 Reference Guide* (choose **Reference Guide** from the Mastercam Help menu)
  - *Mastercam X4 NCI & Parameter Reference* (in the `Documentation` folder under your Mastercam installation folder)

---

7 Your selected editor opens (in this case, Mastercam Editor), displaying the posted NC code.



8 Scroll through the NC code to verify that each line of code meets your expectations. Edit and save as necessary.

Post Processing Summary: Sending NC Files to Machine

IMPORTANT: This tutorial is based on the Mastercam Mill Default machine definition for training purposes only. It is not possible to provide a step-by-step procedure for sending the NC code to your machine control because machine setups are customizable and most likely different from the machine definition used here. Following is a general description of how the NC code is communicated to machines and their controls for machining.

After the NC file is reviewed, edited, and saved, you can set up your machine control to accept the NC file. This is done according to your machine and control manufacturer's procedures.

When the machine control is ready to receive the NC file, configure your preferred editor or communications program to communicate with your machine control. Refer to your communications program documentation for details.

Send the NC code to your machine control according to your machine and control manufacturer's documentation. Once you start the communication process, the send/receive data processing is mostly managed by your machine control.

Contact your local Mastercam Reseller for customized machine/control definitions, post (PST) files, and support.

Conclusion

Congratulations! You have completed the *Basic 2D Machining* tutorial. Now that you have mastered the skills in this tutorial, we encourage you to explore Mastercam's other features and functions. Additional tutorials may be available in this or other series. Please contact your authorized Mastercam Reseller for further training.

cnc software, inc.

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

Printed in the USA
Mastercam X4 Getting Started Series - Basic 2D Machining: Part X4-PDF-TUT-2M