Drilling Holes

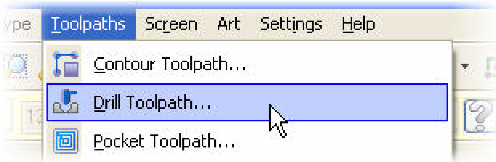
Lesson Goals

- Create a drill toolpath (including selecting a drill point, choosing tooling, and setting machining values).
- Use tool tip compensation.
- View all toolpaths.

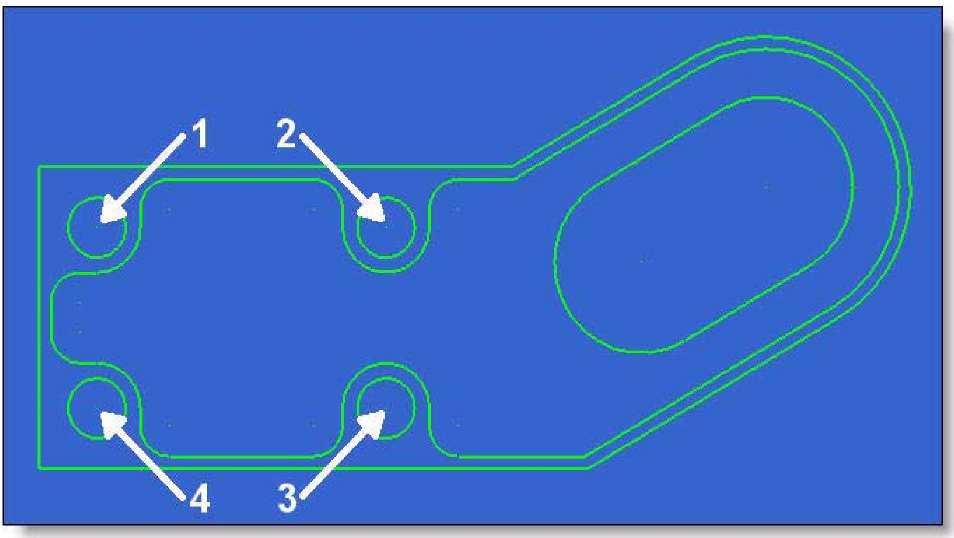
Exercise 1: Drilling Four Holes

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

1. Choose **Toolpaths**, **Drill Toolpath**. The Drill Point Selection dialog box opens.

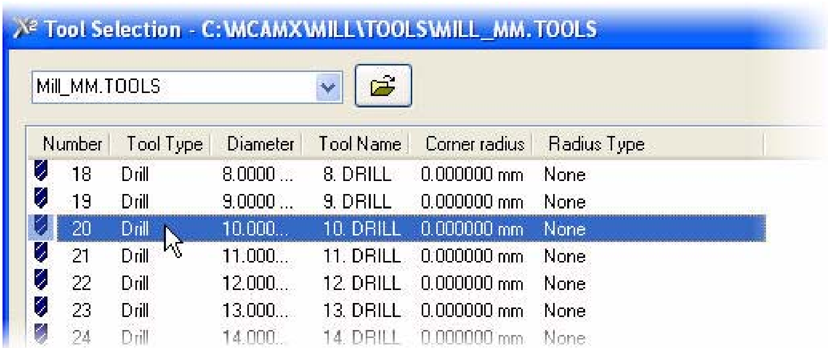


2. Click each of the center points of the four holes of the part. Choose them in a clockwise order from top left to bottom left as shown in the numbered sequence below.

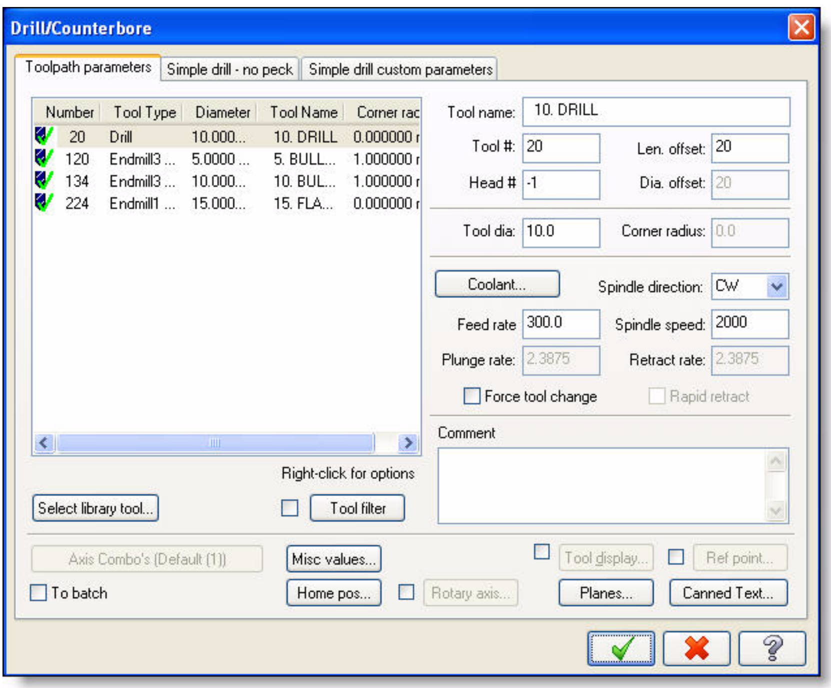


3 Click **OK**. The Drill/Counterbore dialog box opens.

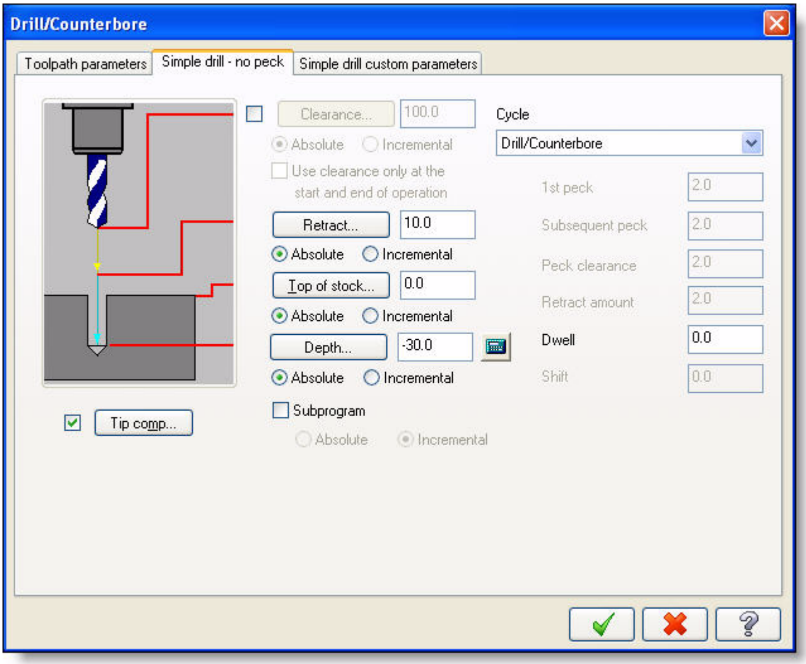
4 Click the **Select library tool** button, select the 10mm diameter drill, and click **OK**.



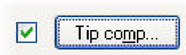
5 Enter the toolpath parameter values as shown here.

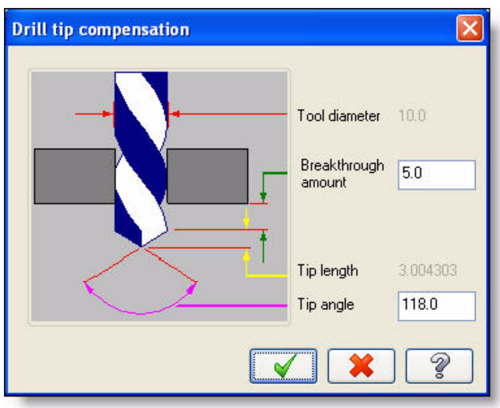


6 Click the **Simple drill - no peck** tab and enter the values as shown here.



7 Click the **Tip comp** check box and button, enter values as shown here, and click **OK**.





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 will result in the drill retracting before the desired depth is reached.

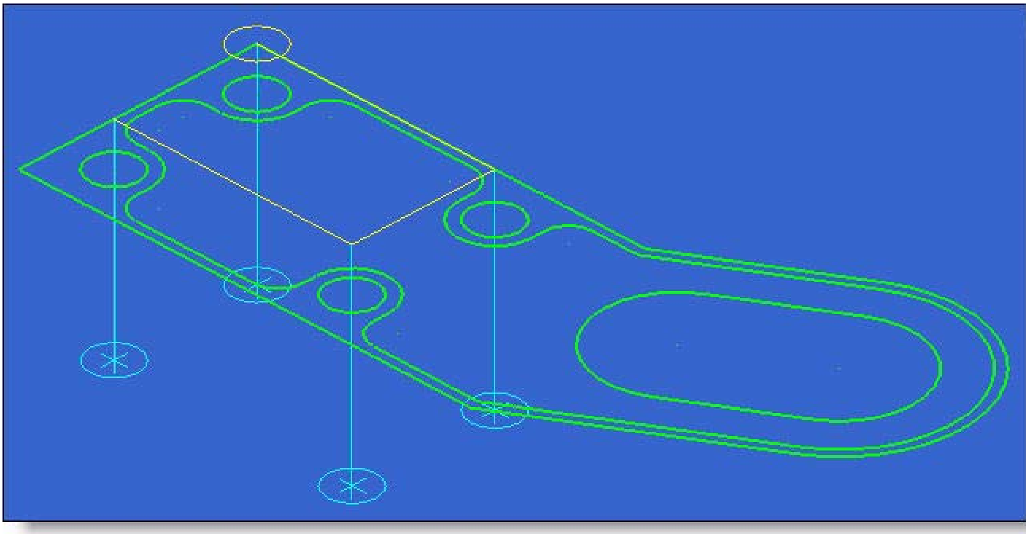
---

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

9 In the Toolpath Manager, make sure the current drill toolpath is the only toolpath set to display in the graphics window.

10 Right-click in the graphics window to bring up the pop-up menu, and choose **Isometric Gview** to view the part and toolpath in the isometric view.

Your toolpath should look like this.



11 Right-click in the graphics window again to bring up the pop-up menu, and choose **Top Gview**.

12 In the Toolpath Manager, turn off the toolpath display for all toolpaths.

13 Save your part.

Backplotting Toolpaths

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

Lesson Goals

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

Exercise 1: Backplotting All Toolpaths

In this exercise, you backplot all toolpath operations for this part.

Note: This lesson assumes that you have successfully completed Lessons 1 through 4 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.*

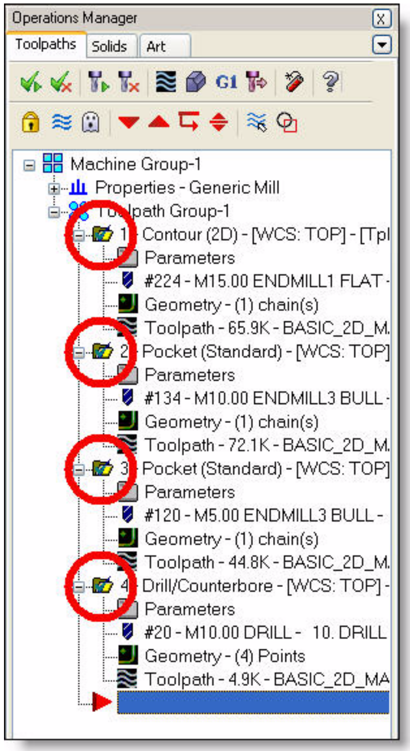
---

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

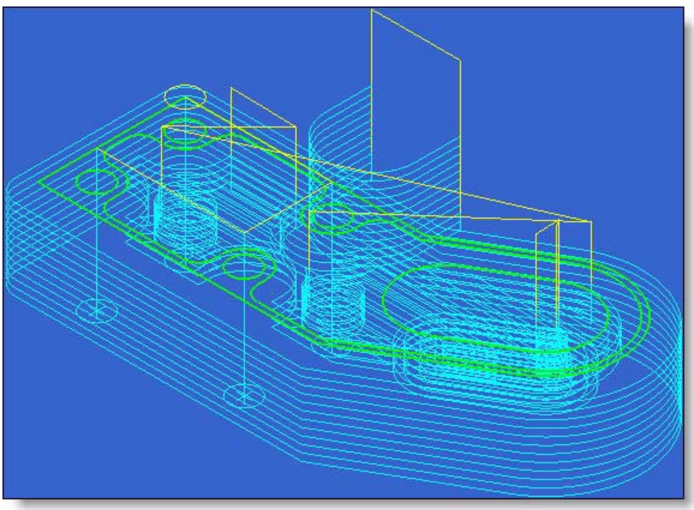
2 In the Toolpath Manager, click the **Select all operations** button.

All four toolpath folders display a green check mark as circled in the graphic here.

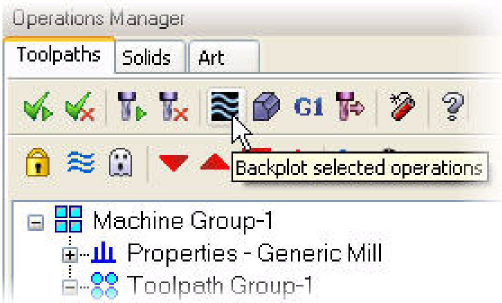
Make sure that all toolpaths are set to display in the graphics window.



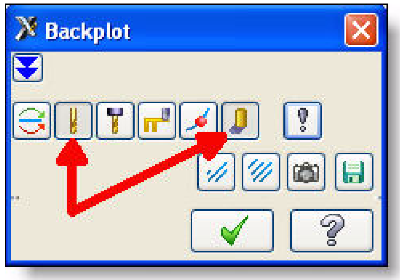
3 Right-click in the graphics window and choose **Isometric Gview** from the menu. The part will display as shown here.



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

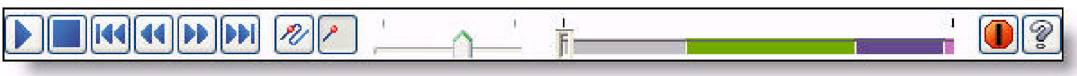


5 In the Backplot dialog box, Select the **Display tool** and **Quick verify** buttons. These options will display a simulation of a tool and shade the toolpath during backplot.



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 options 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 (shown to the right) 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

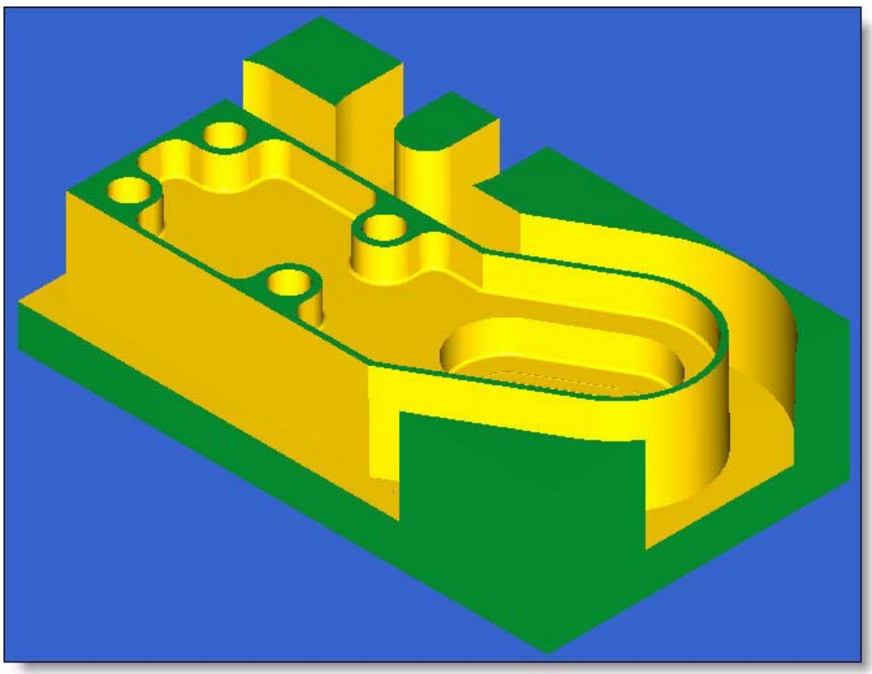
In this exercise, you simulate (verify) the machining of the part from a stock model display.

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.

When the verification process is complete, the part should look like this. Here it is shown without the tool displayed.



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

Posting Toolpaths

Lesson Goals

- Post all toolpaths to create NC files.

Exercise 1: Posting All Toolpath Operations

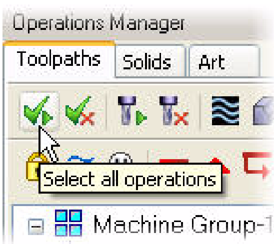
In this exercise, you post all toolpath operations for this part to an NC file, review/edit the code as necessary, and save the NC file.

Note: This lesson assumes that you have successfully completed Lessons 1 through 4 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.*

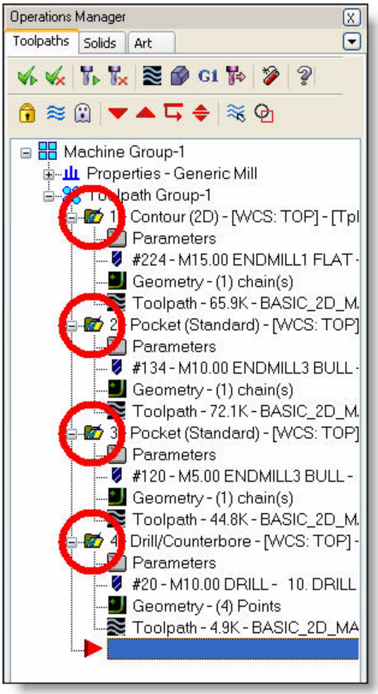
---

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

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

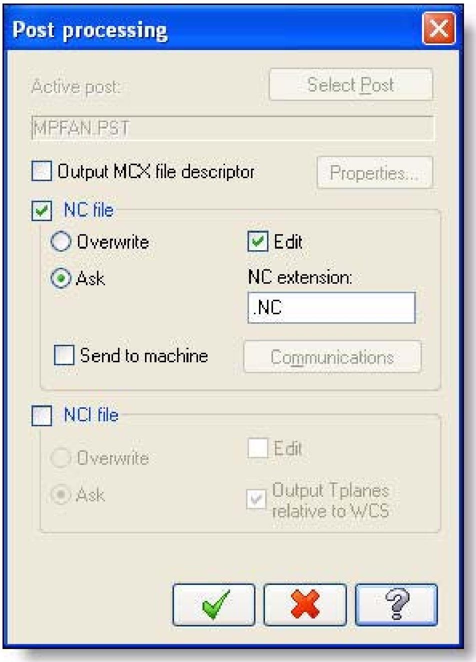


All four toolpath folders will display a green check mark as circled in the graphic below.



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

4 Select the post processing settings as shown here.



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

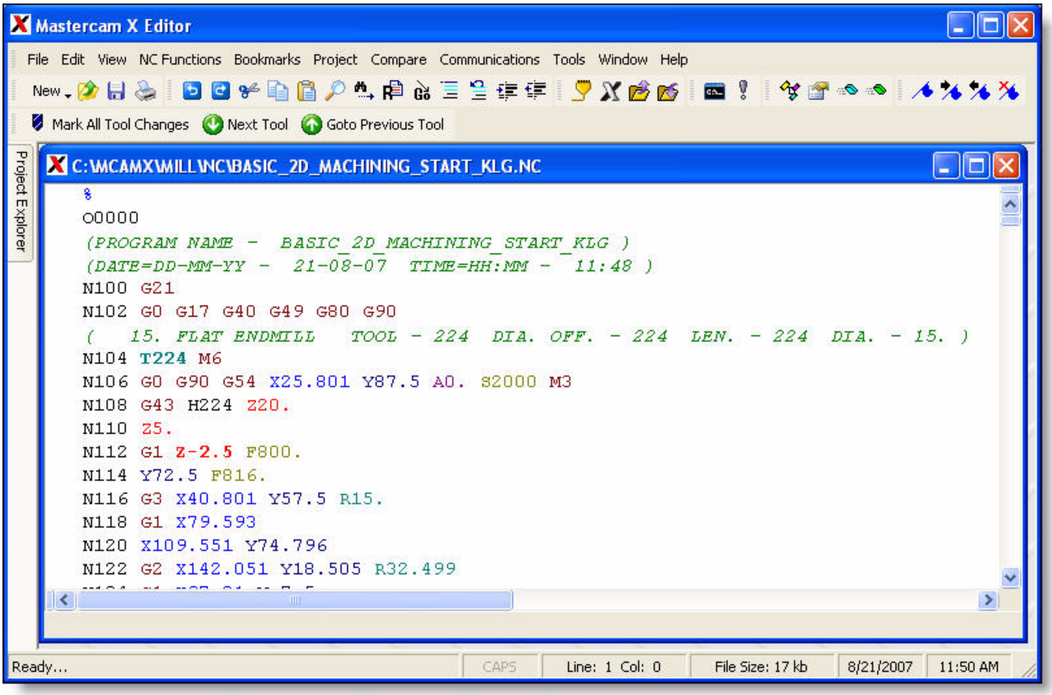
6 Click **Save** to save the NC file in the default location with the recommended file name.

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 X2 Reference Guide (choose Reference Guide from the Mastercam Help menu)*
  - *Mastercam Post Parameter Reference Guide (in the Documentation folder under your Mastercam installation folder)*

---

7 Your chosen editor opens (in this case, Mastercam Editor), displaying the posted NC code as shown below.



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 X2 MR2 Basic 2D Machining 1-883310-81-4