

MadCAM 2.0: Drill Pattern Toolpath

Digital Media Tutorial
2005 - 2006

MadCAM 2.0 can create a toolpath to drill holes directly into your material. The bit plunges in and out of the material without moving in the X or Y direction. This tutorial goes through the entire process of setting up your file and creating a toolpath for the large CNC mill.

MadCAM 2.0 can also create a toolpath to mill two dimensional curves into the material. See the 2D profile tutorial included with this tutorial.

Open the 2D File in Rhino

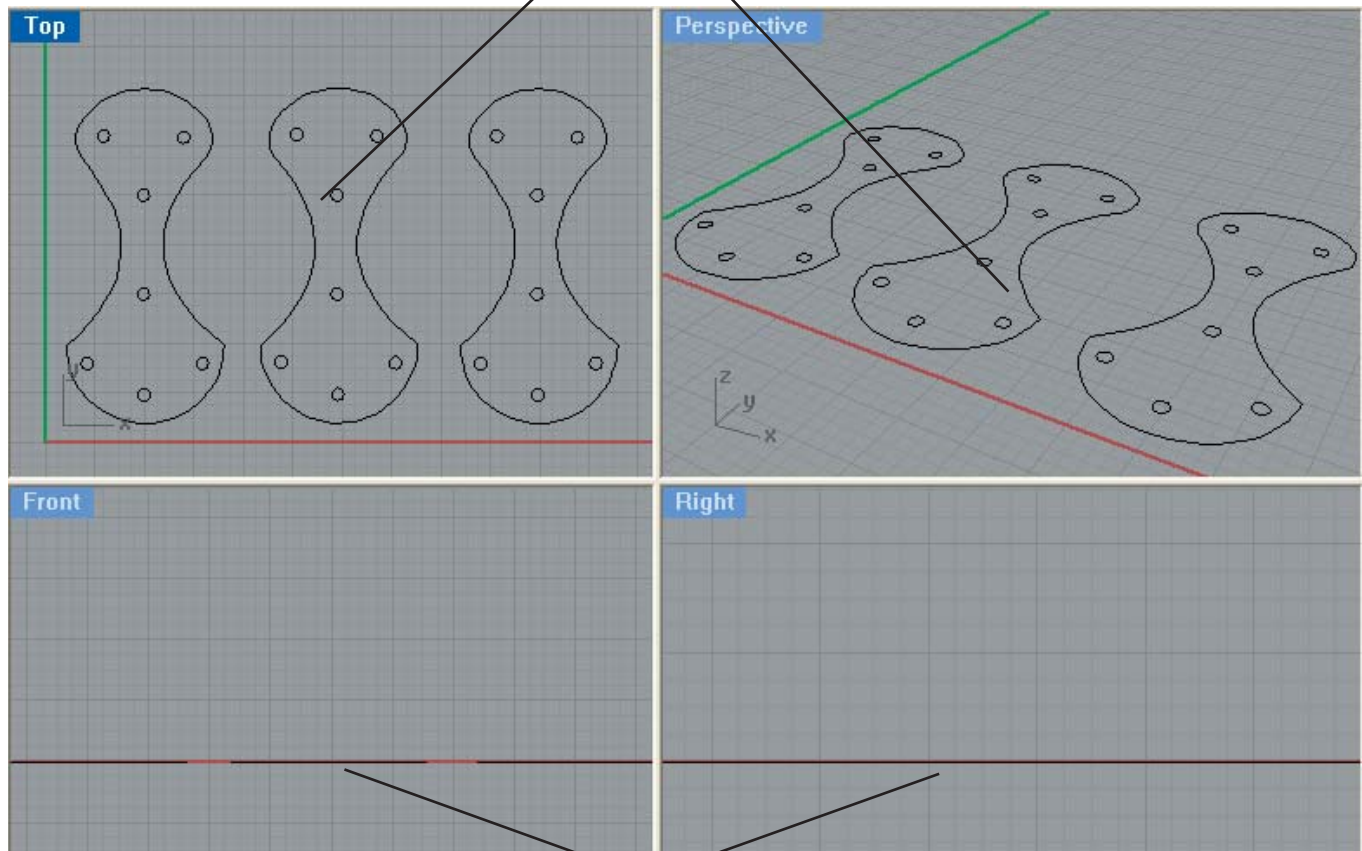
Step 1: Open your two dimensional file in Rhino.

MadCAM runs inside Rhino, and is automatically opened when Rhino is opened.

Step 2: Correctly place the profile in the modeling window.

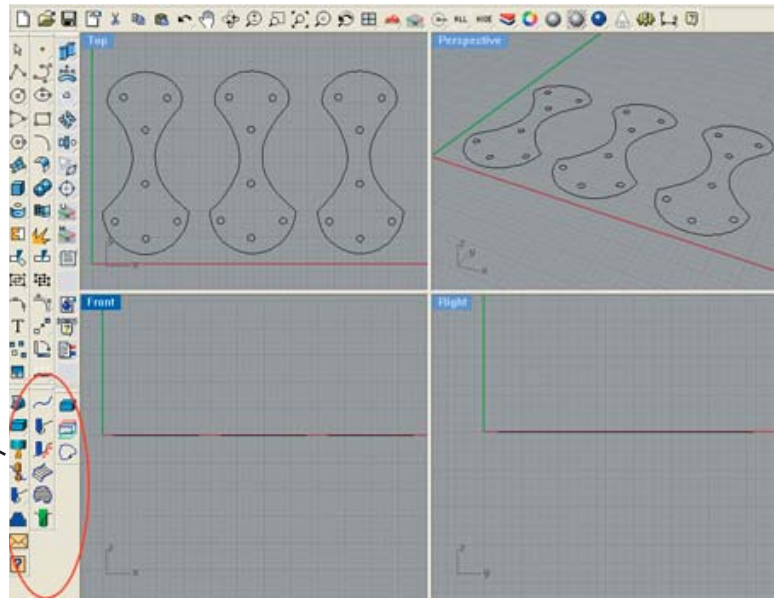
The large CNC mill uses the profile location relative to Rhino's modeling window. The profile should be at the X and Y ground plane, and completely within the positive X and Y axis.

Model placed in the positive X and Y axis.

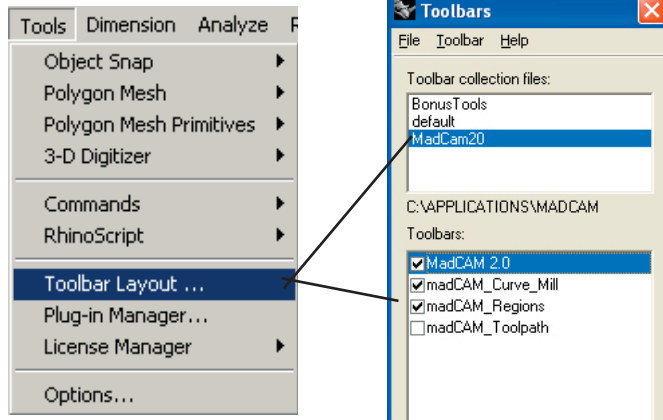


Model placed at the X and Y ground plane.

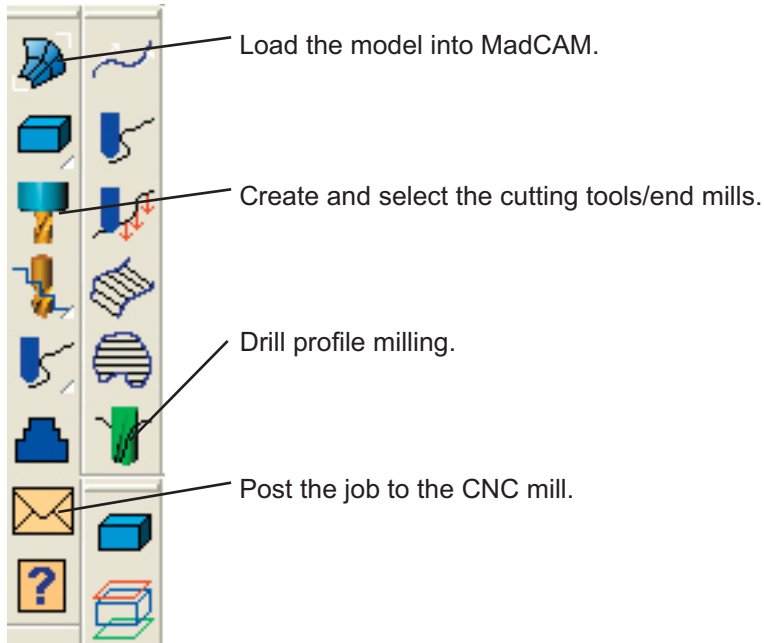
You should see the MadCAM tool bars loaded in the modeling window.



If the toolbars are not visible, they can be loaded by selecting **Tools>Toolbar Layout**. Select MadCAM, and check on the following toolbars.



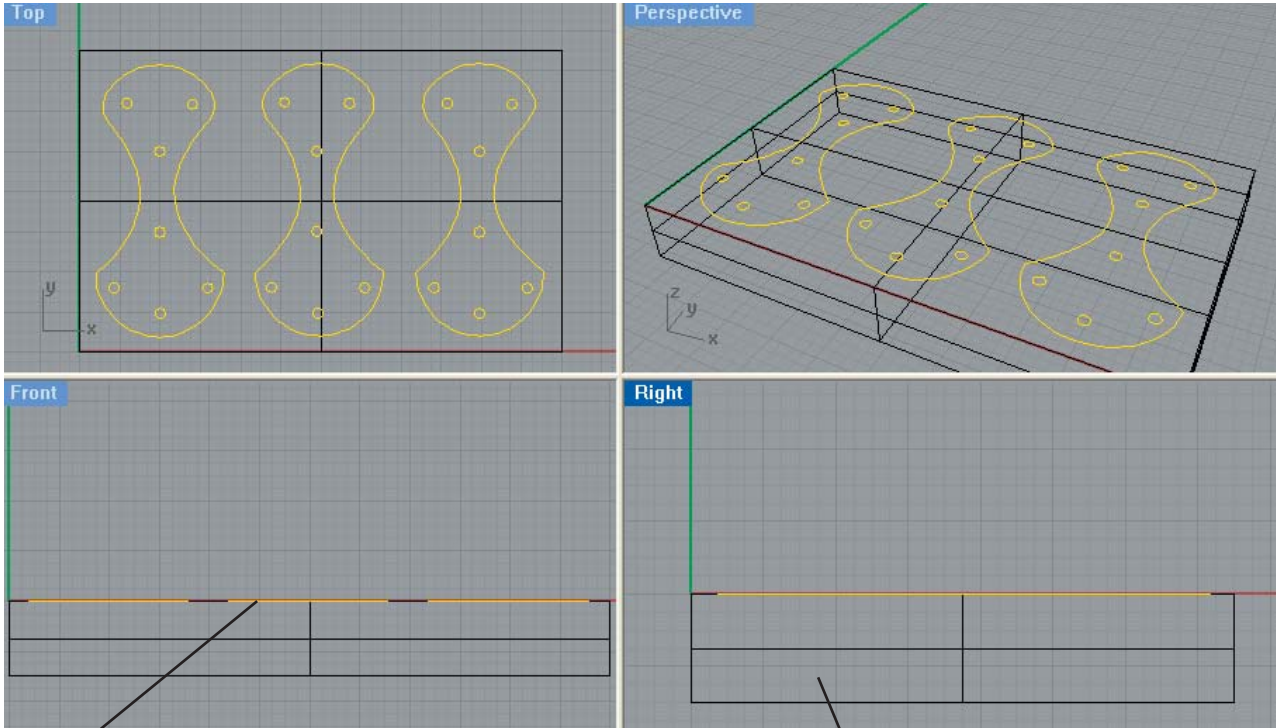
The MadCAM Tool bars:



Configure Your File To Create A Drill Pattern Toolpath

Step 1: Create a bounding box to represent your material.

Draw a box to represent the material you will be milling. The width, height, and length of the box should match the material exactly. The box should be below the X and Y ground plane, and the profile curves should be on the top surface of the box.



Profile curves on top surface of box.

Box placed below X-Y ground plane.

Create A Drill Pattern Toolpath

In order for MadCAM to correctly generate a toolpath, the process must be followed exactly. If a mistake is made, it is best to start over and reload the file.

Step 1: Load the geometry into MadCAM.

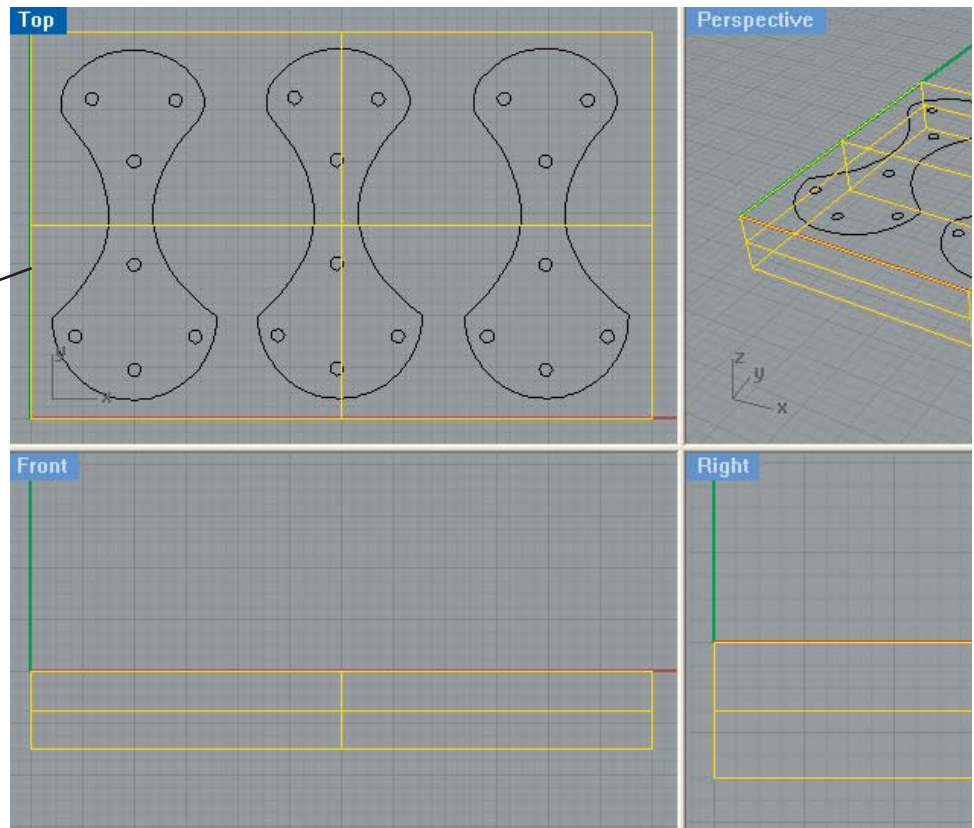
Click on the MadCAM icon. You will be prompted to select the surfaces.



Select surfaces, polysurfaces, and meshes to export:
Select surfaces, polysurfaces, and meshes to export. Press Enter when done:

Select the box created to represent the material you are cutting.

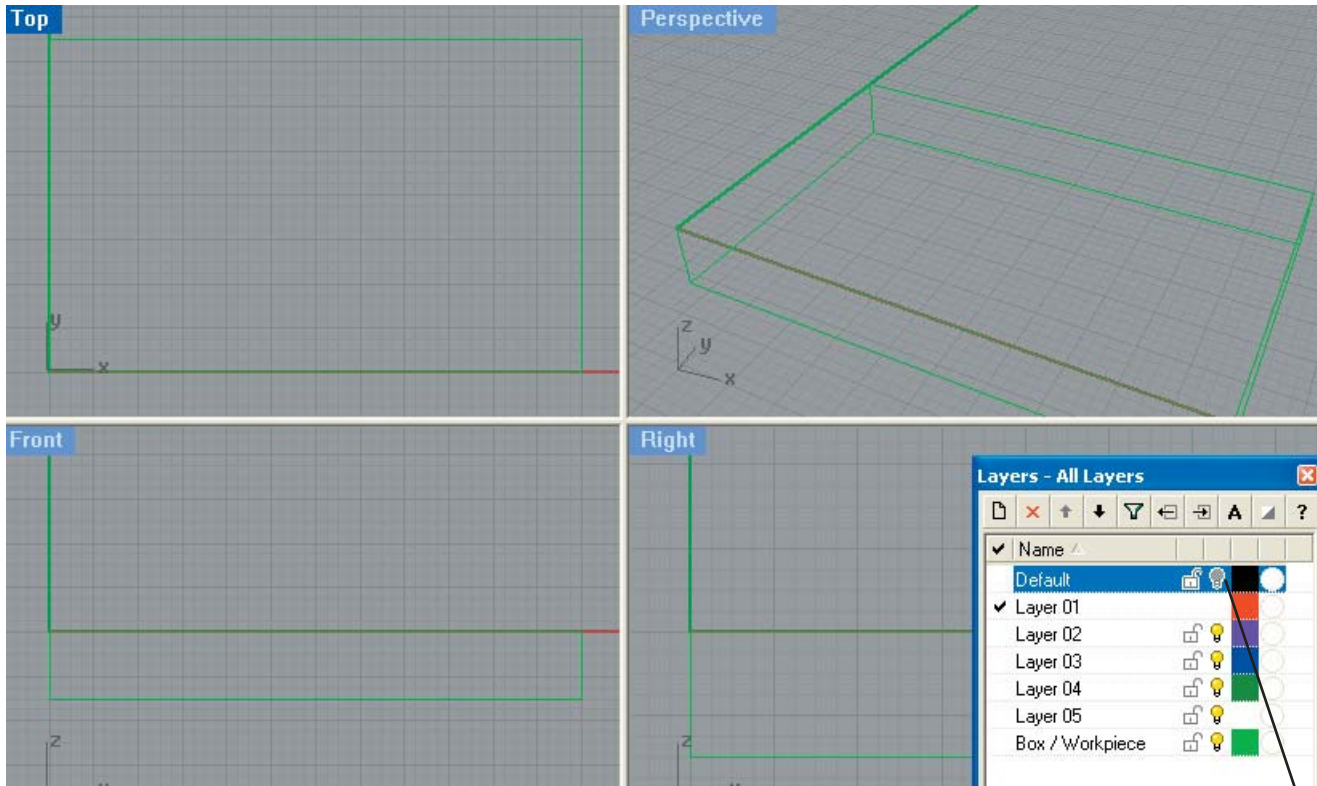
Press **Enter** when done.



Step 2: Verify the bounding box.

When an object is correctly imported to MadCAM, a green bounding box will appear about the outside of the surface selected. The bounding box represents the “Virtual Stock Piece” that simulates the material that is cut by the mill.

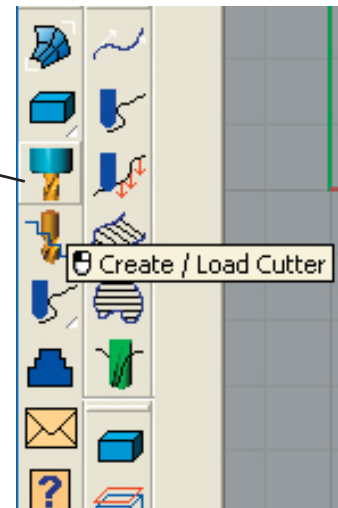
The bounding box may be difficult to view, as it may be on the same lines as the surface box created earlier to represent the cutting material. Temporarily turn off the layer that contains the surface to view the bounding box.



Main layer turned off to view the bounding box.

If there is no green box appearing, reload the surface in MadCAM.

Step 3: Create and load a cutting tool. Click the **Create Cutter** button to create a new cutter or load an existing cutter.



Step 4: Cutter parameter window. Load a predefined cutter by selecting it from the menu on the right side, or create a new cutter by inputting the various sizes for the cutter. You can find the sizes by measuring the bit.

Name: Give a descriptive name for the cutter.

Type of Bit:

Flat End: Most common, with a squared end.

Ball End: Rounded end.

Corner End: Chamfered end.

Diameter: Diameter of the bit.

Length: Overall length of the bit.

Cutting Length: Length of the cutting edges of the bit.

Feed X and Y, Feed Z, Spindle:

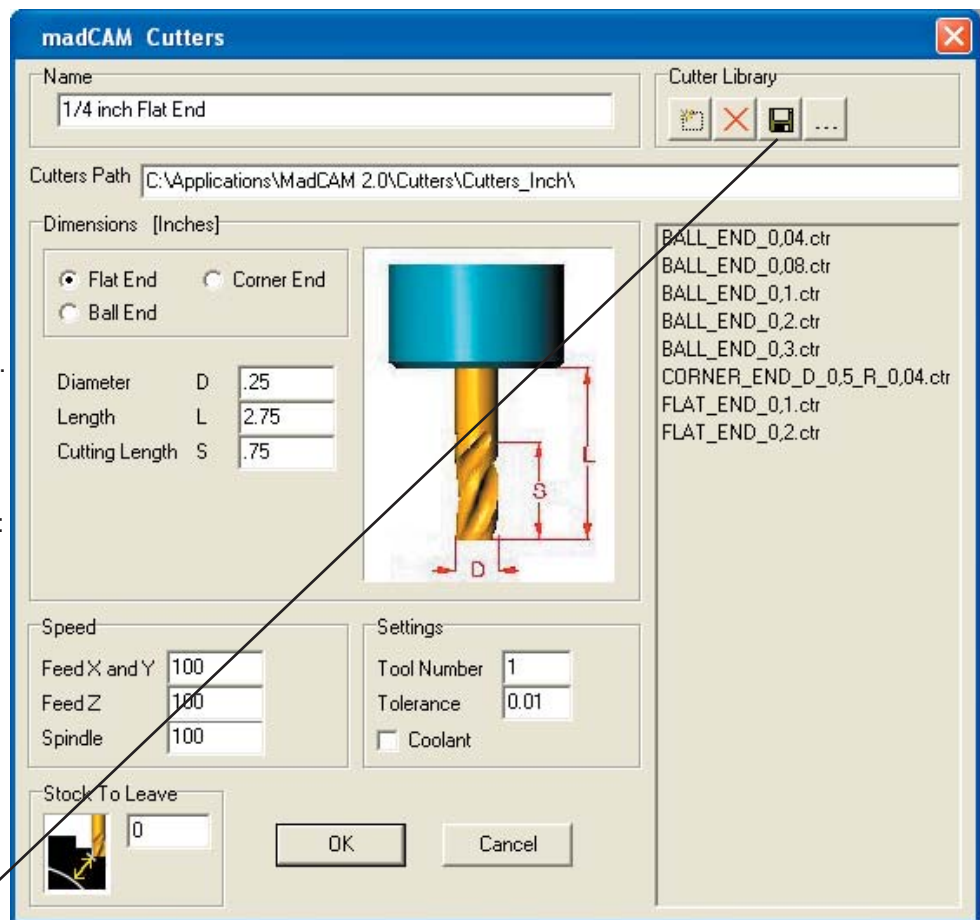
Set to 100 by default. The mill overrides this setting.

Stock To Leave: Set to 0.

Tool Number: Set to any number.

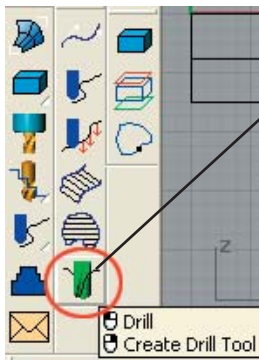
Tolerance: Set to 0.01 by default.

Coolant: Leave unchecked.



Click the **Save** button to save your settings.

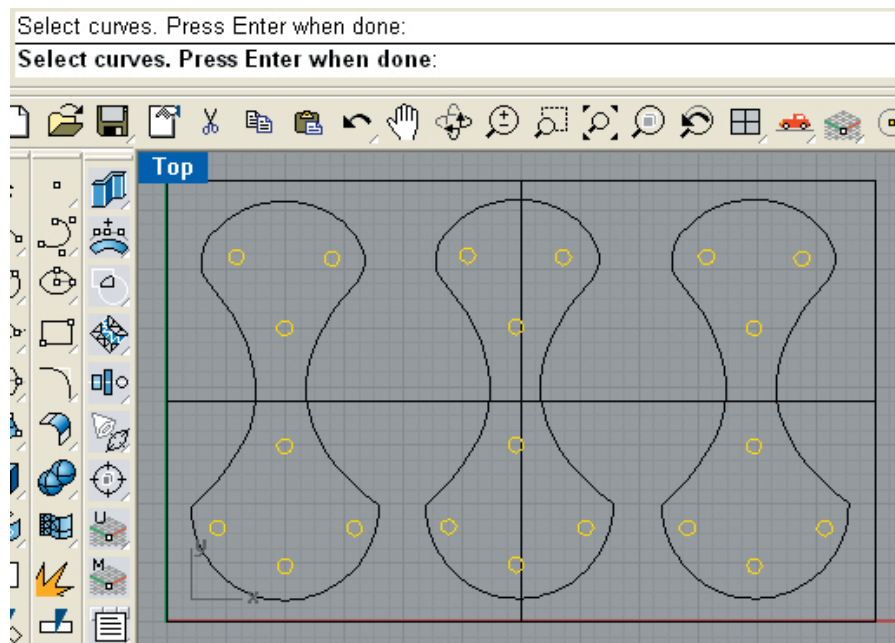
Click **OK** to select the cutter.



Step 5: Setup the toolpath
Click the **Drill** button.

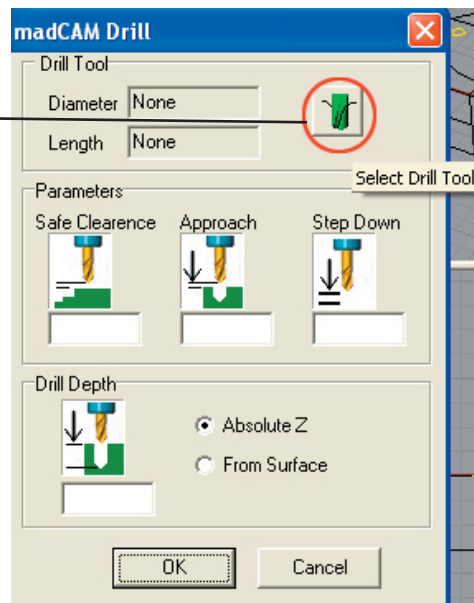
Step 6: Select the curves

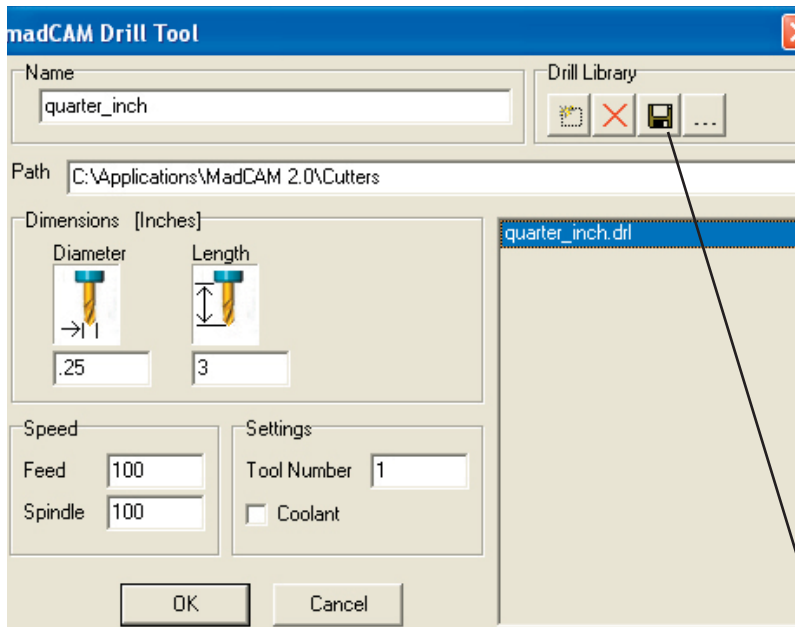
You will be prompted to select the curves to create a toolpath from. Select all curves you want to drill. Press Enter when done.



Step 7: Drill options:

Click on the **Select Drill Tool** to input end mill cutting parameters.





Step 8: Drill Tool Setup:

Input the parameters for the end mill you will be using.

Name: Give the end mill a descriptive name.

Path: Verify the path is set to **C:\Applications\MadCAM 2.0\Cutters**

Dimensions: Input the diameter and length of the bit.

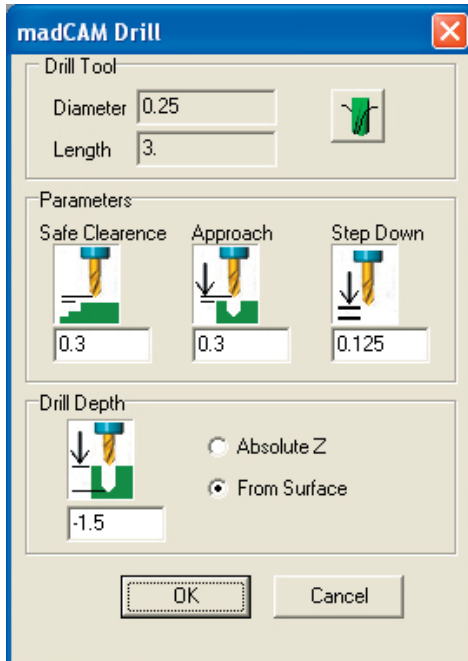
Feed, Spindle: Set to 100 by default. The mill overrides this setting.

Tool Number: Set to any number.

Coolant: Leave unchecked.

Save the end mill information.

Select **OK** when done.



Step 9: Drill Setup:

Input the parameters for the toolpath.

Drill Tool: The diameter and length should match the tool you just configured. If it does not, repeat step 8.

Safe Clearance: Set to 0.3.

Approach: Set to 0.3.

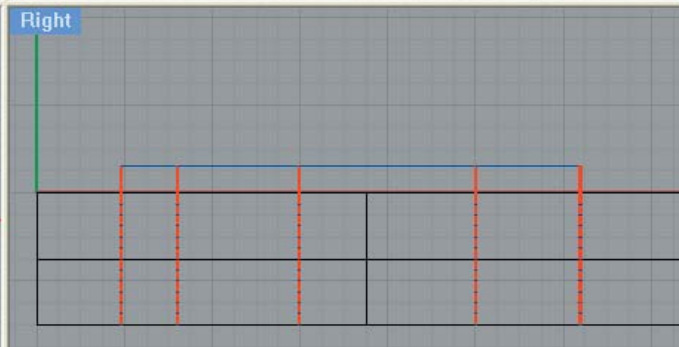
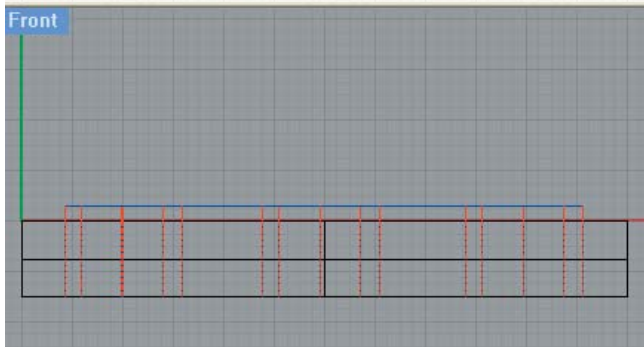
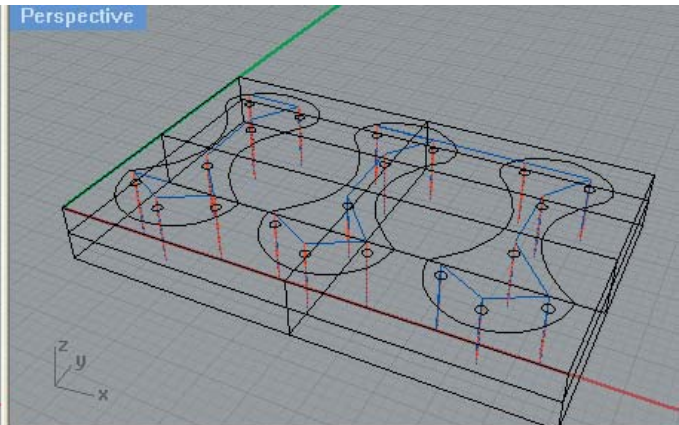
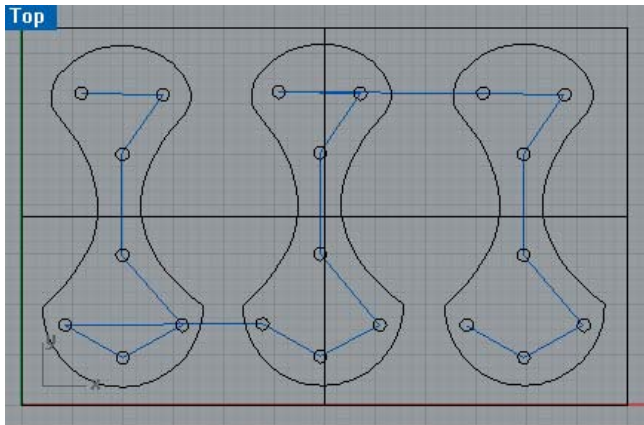
Step Down: Set to 1/2 the diameter of the end mill. (1/4 inch bit = 0.125 step down)

Drill Depth: How far the end mill will drill down. *Do not exceed the depth of your material!*

Select **From Surface**.

Select **OK** when done.

Step 10: Verify that the toolpath. The red lines represent the drill path and should be placed exactly on the curves in your file.

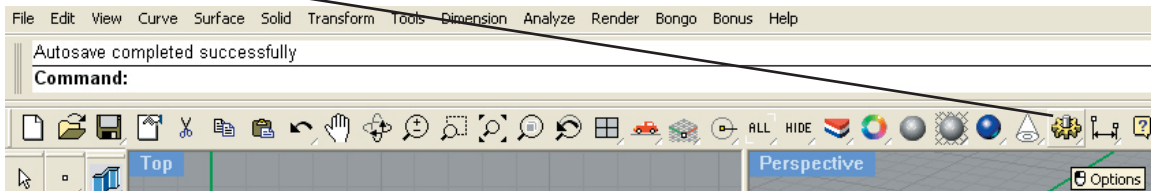


Post the Post-Processor File

Step 1: Verify MadCAM Settings.

Verify the correct settings before posting the toolpath file to the large CNC mill.

Click on the **Options** button.



Step 2: Select MadCAM in the left menu.

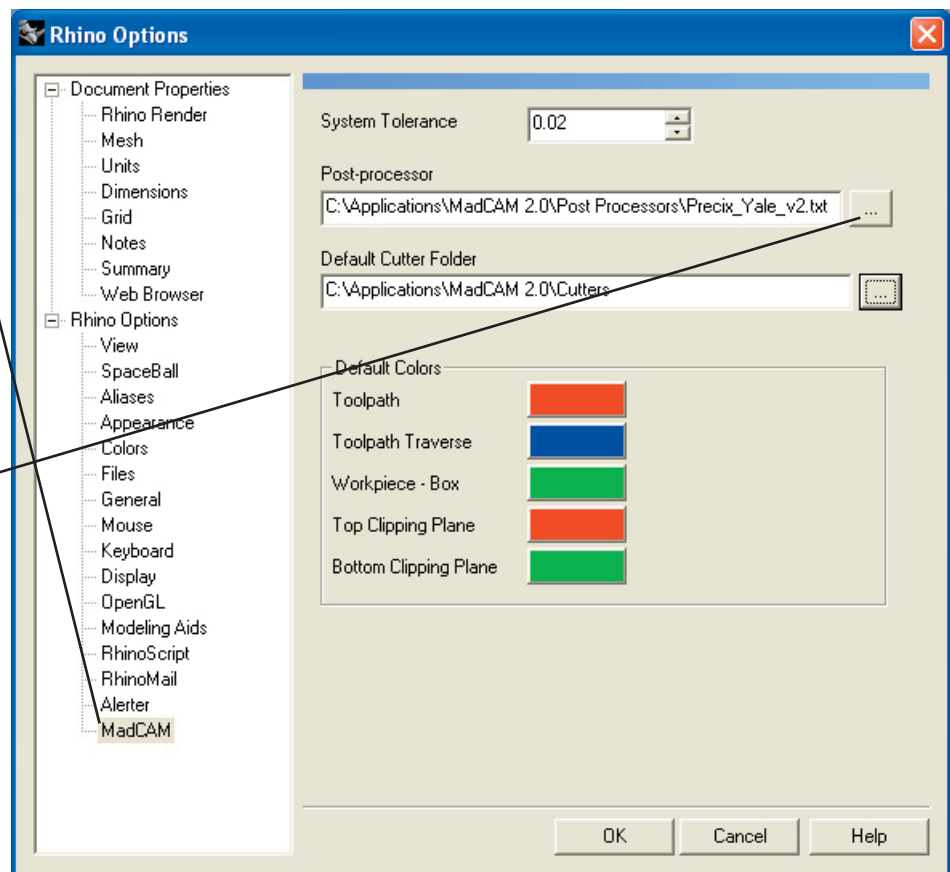
Set the System Tolerance to **0.02**.

Verify that the post-processor is set to **Precix_Yale_v2.txt**.

If the Precix_Yale_v2.txt post-processor file is not selected, click on the ... button to select it.

Verify the default cutter folder is set to **C:\Applications\MadCAM 2.0\Cutters**

Click the **OK** button once all the settings are correct.



Step 3: Post the toolpath to the Yale Precix Postprocessor.

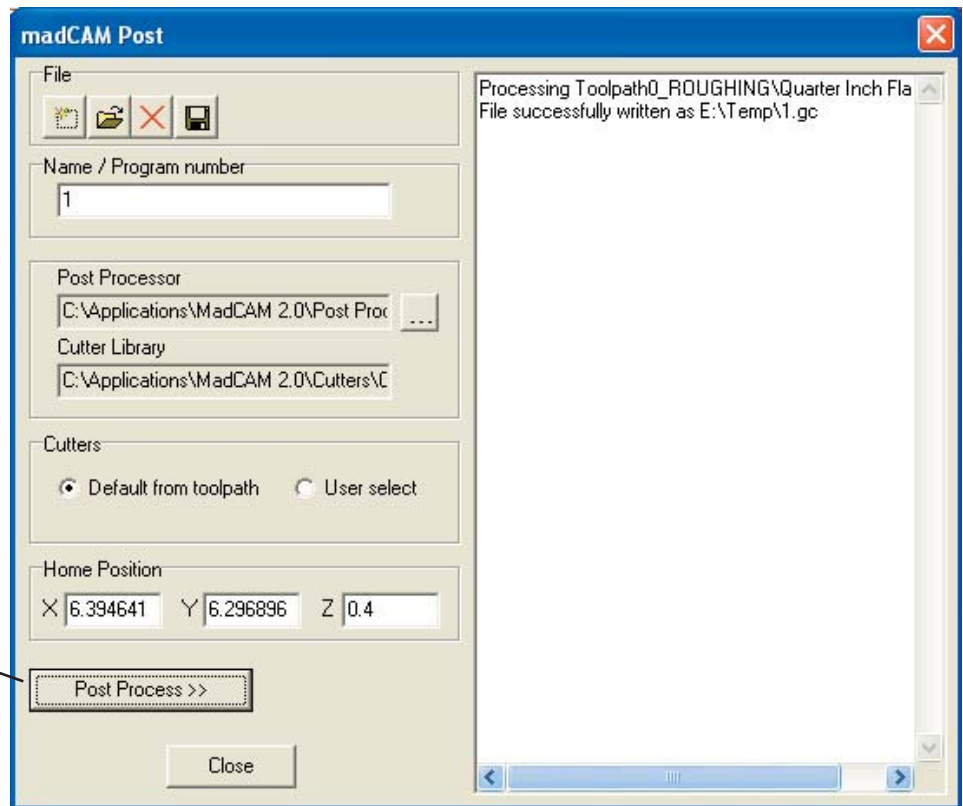


Click the Postprocess button. The MadCAM Post window will appear.

Verify the post processor and cutter library settings are correct.

Default from toolpath should be selected for Cutters.

Click on the **Post Process** button. Save the .gc file to any location, or directly to the zip disk for the mill.



The posted file.

