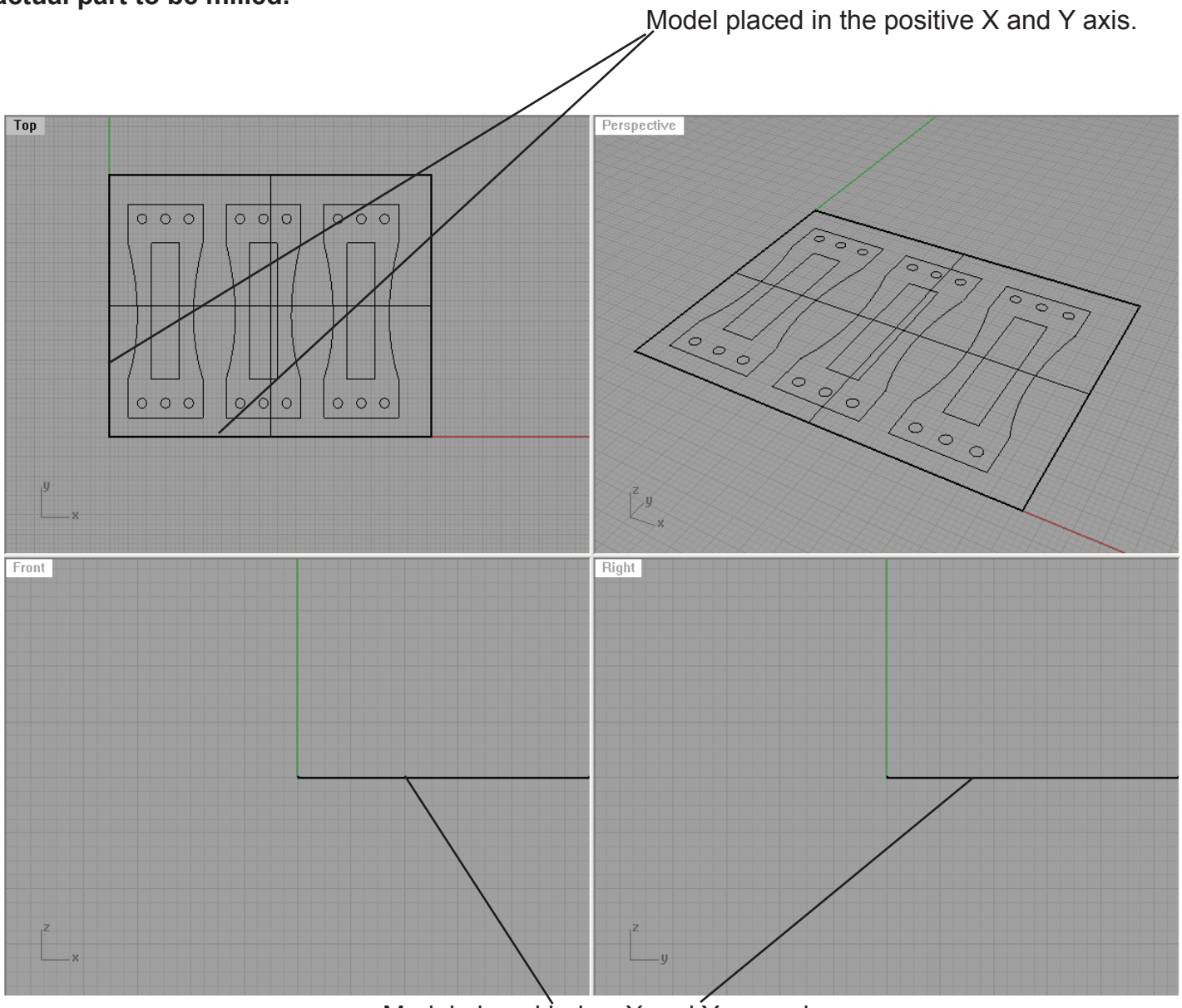
MadCam 4.2: 2D Profile Toolpath

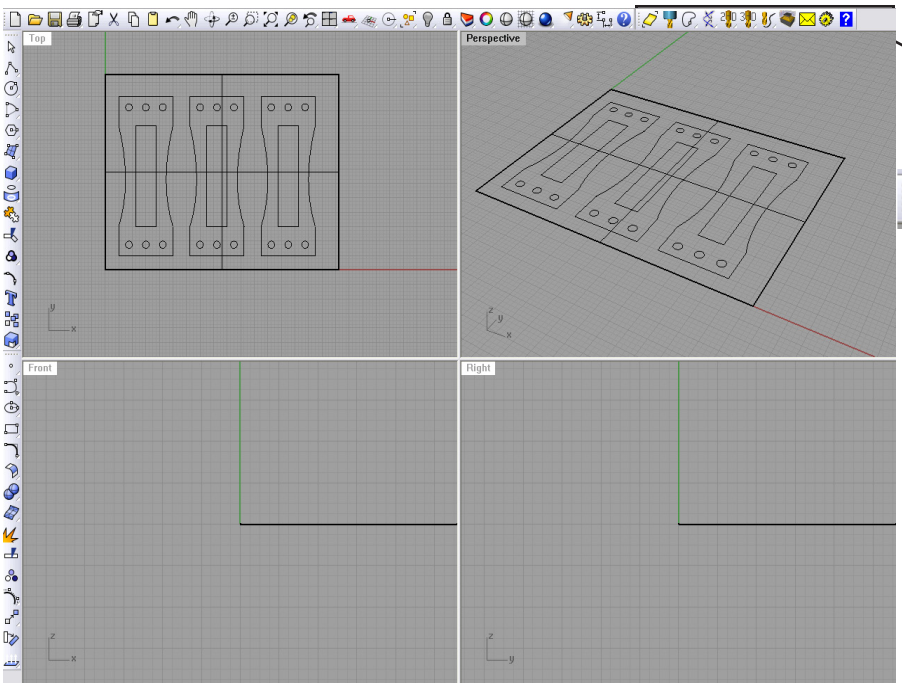
Digital Media Tutorial

MadCAM 4.2. can create toolpaths to mill two dimensional profiles in a range of material thicknesses. This tutorial goes through the entire process of setting up your file and creating a toolpath for the large CNC Mill.

Step 1: Open or Create a 2D file in Rhino. MadCAM runs inside Rhino, and is automatically opened when Rhino is opened. (If the MadCAM toolbar does not open, see the next page for instructions on loading the toolbar).

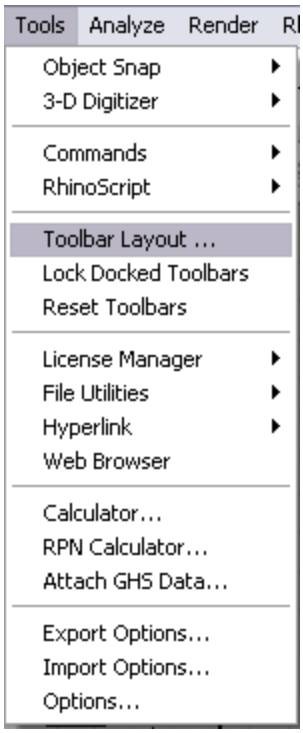
Step 2: Prepare Model. Correctly place your part in the modeling window. The large CNC mill uses the model origin location as the start point for the mill. You need to move your part so that it is completely in the positive X and Y axis. **NOTE: Model in Rhino MUST be scaled to match the actual part to be milled.**



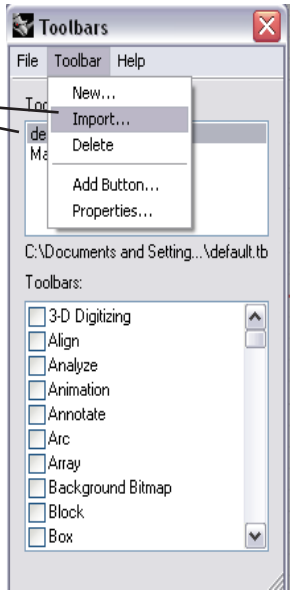


You should see this MadCAM toolbar loaded in the modeling window.

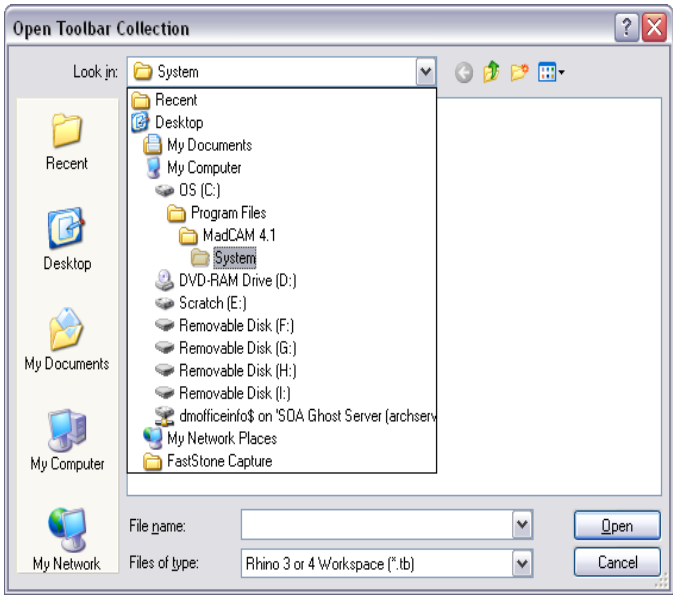
If the toolbar is not visible, it can be loaded by selecting **Tools>Toolbar Layout.**

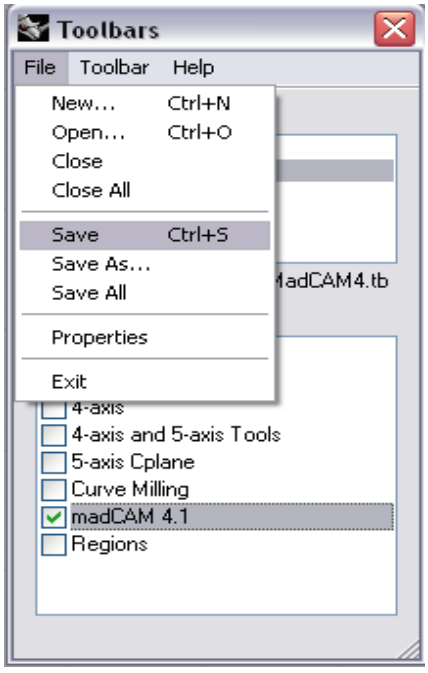


Highlight **Default** and choose **Import** from the Toolbar menu.



Find MadCAM4 by following the path:
C:\Program Files (x86)\MadCAM 4.2\System\ MadCam4.tb.





Once you choose **MadCam4** it will take you back to the Toolbars window. Highlight **MadCam4** in the upper window and check **madCAM 4.2** in the lower window. Then go to **File>Save** and the tool bar will pop up. You can drag this Toolbar and place it anywhere on or around your workspace.

The MadCAM Toolbar:

Select surfaces.

Create and select the cutting tool.

Create or modify workpiece, regions, or clipping planes.

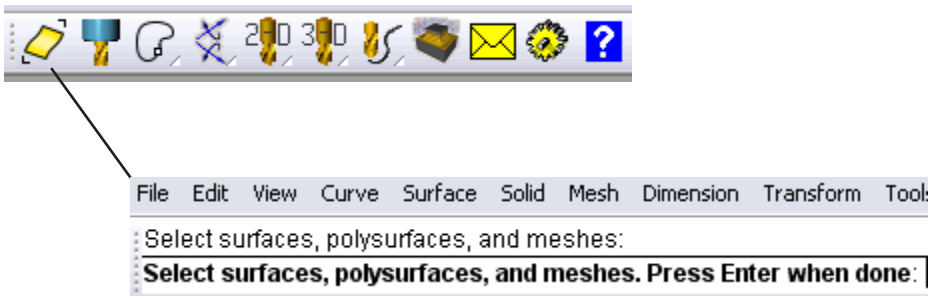
Create 2D toolpath from curves or edge curves.

Select the type of cut to perform: Roughing, Z-level and Planar finishing, and Pencil Tracing.

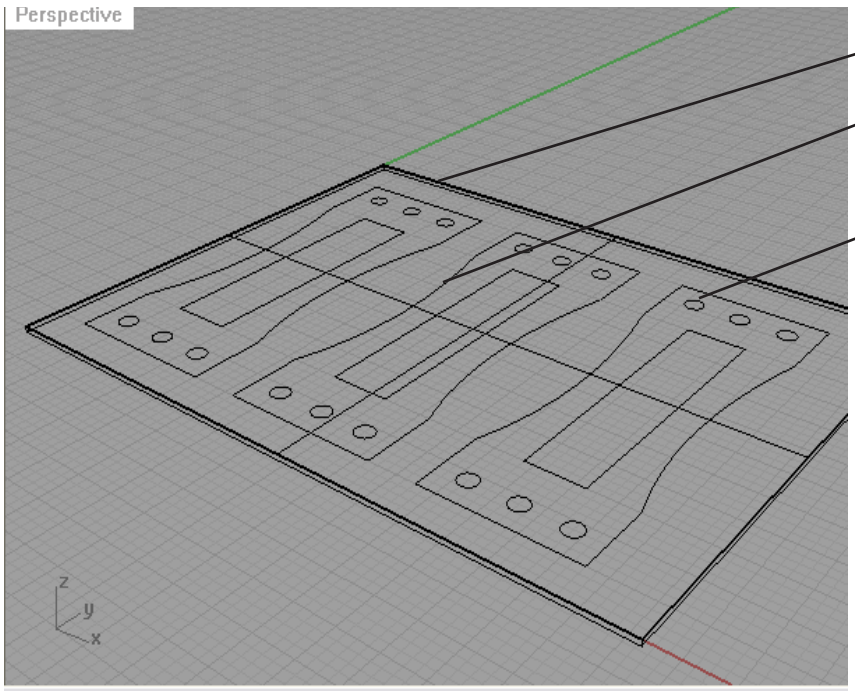
View the work piece shape after each toolpath operation.

Post the job to the CNC mill.

Step 3: Load the geometry into MadCAM. Click on the **Surfaces** icon, and select the 2D surface to load.Follow the prompt box when loading the object.



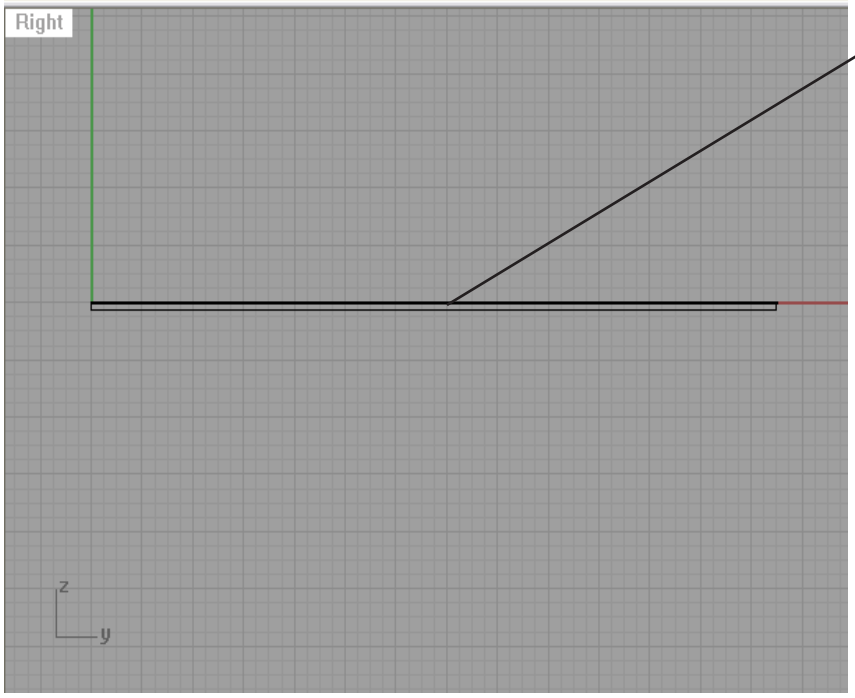
The loaded object will have a box around it.



2D Surface: Represents material to be cut.

Profile Curve: Cuts on the outside of the curve. (Must be a closed curve).

Pocket Curves: Cuts on the inside of the curve. (Must be a closed curve).



2D surface box loaded.

Step 4:Create and load a cutting tool. Click the **Create Cutter** button to create a new cutter or or load an existing cutter.

Step 5: Cutter parameter window: Load a predefined cutter by selecting it from the menu on the right side or create a 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: Squared end.
•Ball End: Rounded end.
•Corner End: Chamfered end.

Diameter: Diameter of bit.

Length: Overall length of bit.

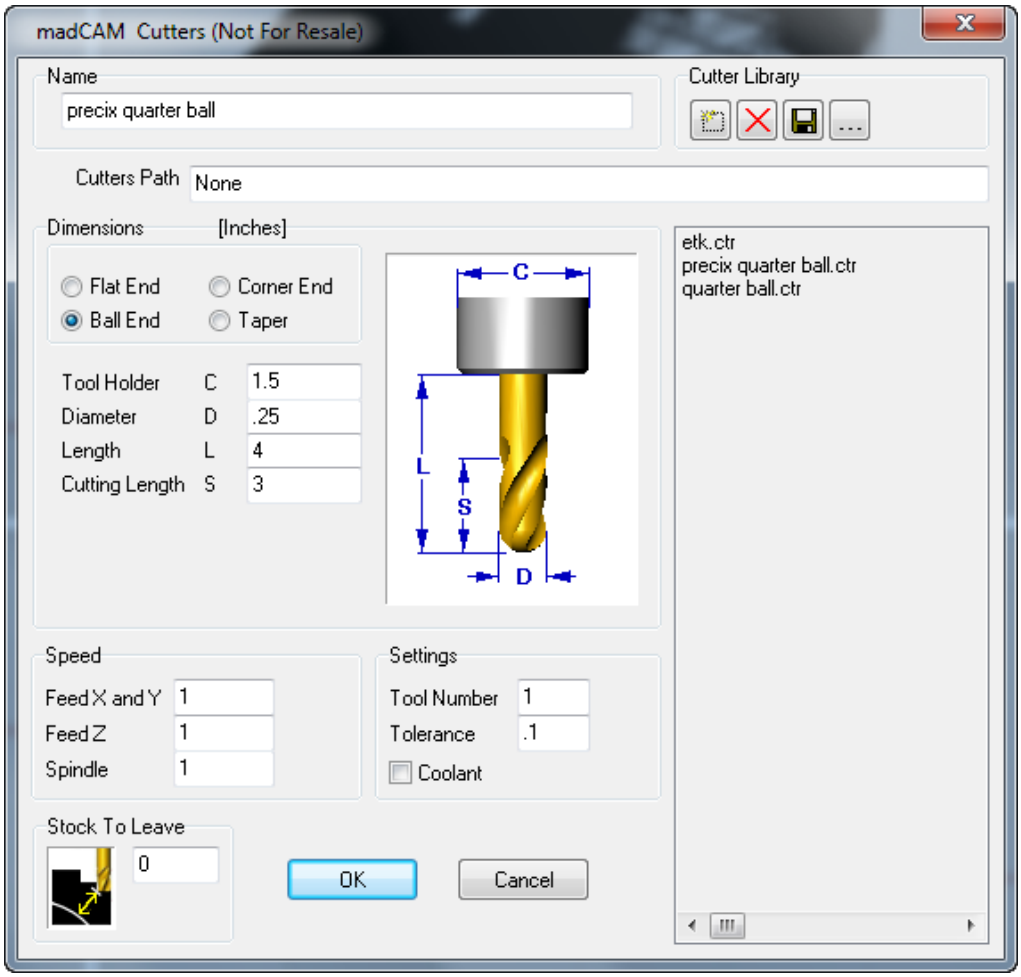
Cutting Length: Length of the cutting edges of the bit.

Feed X and Y, Feed Z, Spindle: Set to 1

Stock to Leave: Set to 0.

Tool Number: Set to any number

Tolerance: Use 0.1 - 0.01, depending on your material. The lower the tolerance, the longer your toolpath will take to process. Setting too low a tolerance can cause the program to run out of memory.



Coolant: Leave unchecked.

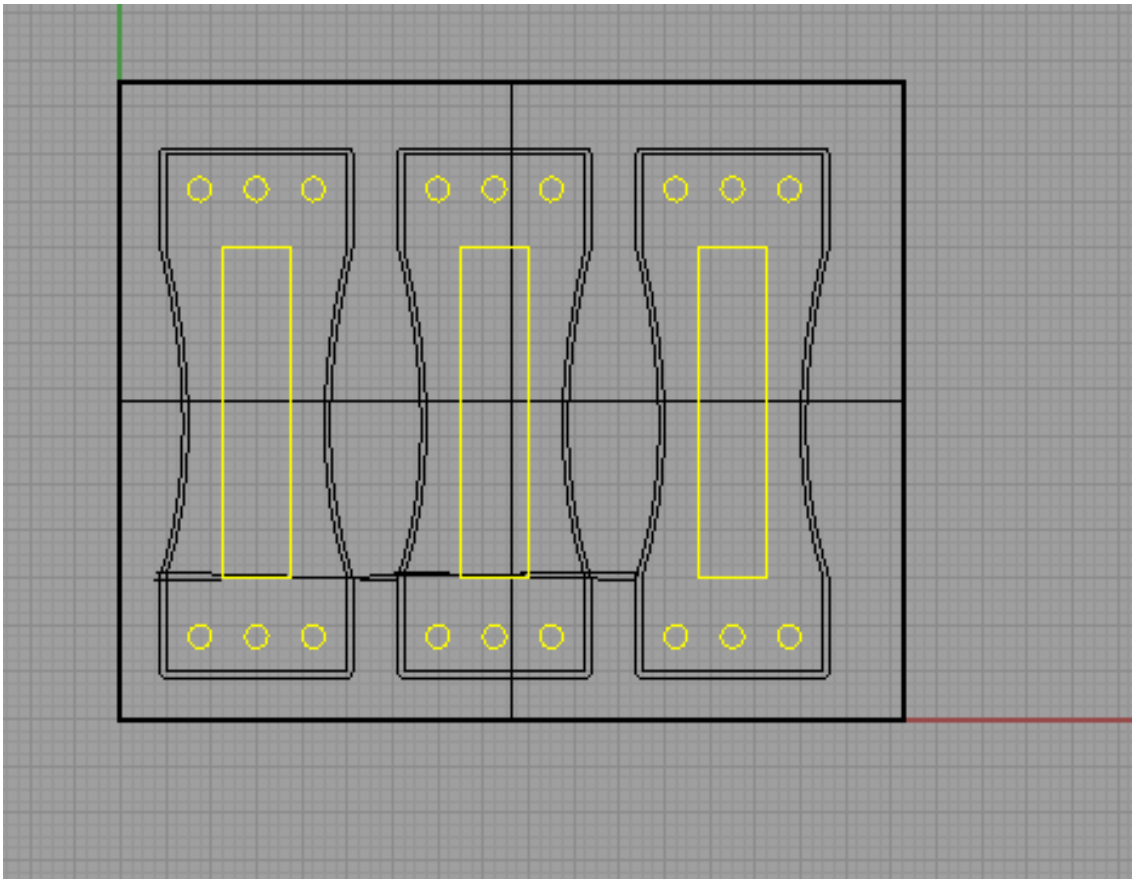
Click the **Save** button to save your settings.

Click **OK** to select the cutter.

Step 6: Pocketing. In order to pocket cut, your curves must be completey closed. Go to 2D toolbar and select **Pocketing.**

Note: You should perform all pocket cuts **BEFORE** cutting any profile cuts.

Select the closed curves and hit **Enter**.



Step 7: Pocket Toolpath Setup:

Material Top: Set to 0 by default.

Material Bottom: The thickness of the material. In this case set this to -.75 (3/4” plywood)

StepDown: How thick of a layer the bit will cut as it mills down the part. Rules of thumb:

Foam: Max StepDown = cutting edge length
Wood: Max StepDown = 1/3 diameter of bit
Metal: Max StepDown = 1/4 diameter of bit

Step Over: 2/3 diameter of bit for softer materials; 1/3 for denser materials and faster feed rates.

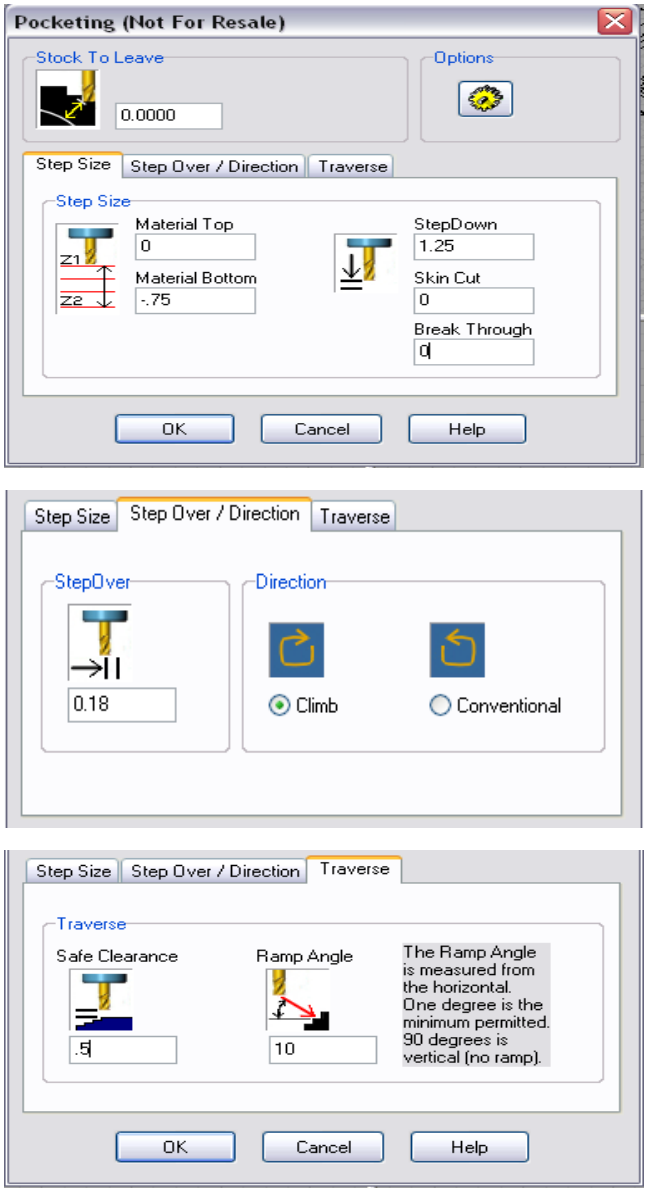
Direction: The direction of the passes the bit will make as it cuts.

Climb: Used for cutting wood.

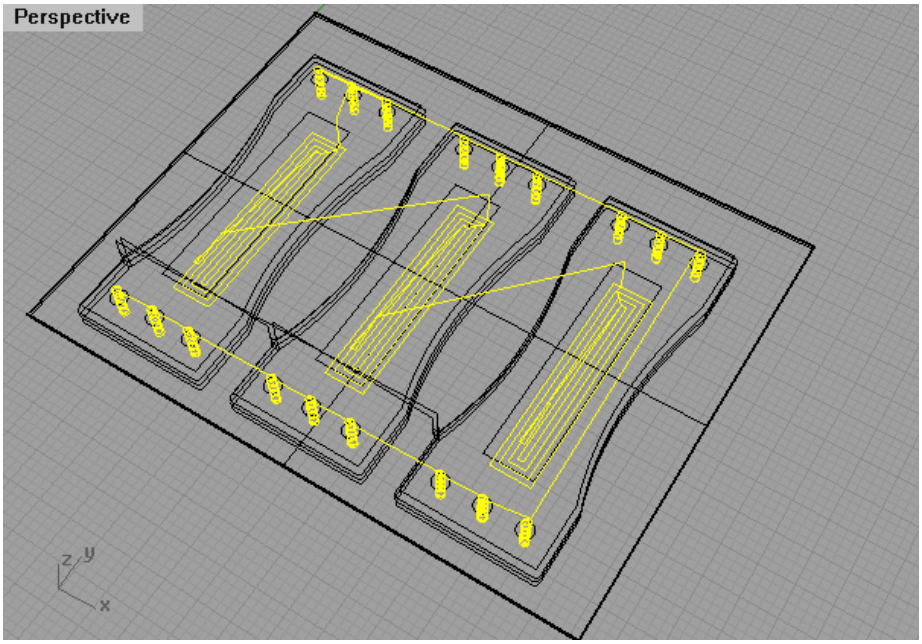
Conventional: Used for monolythic materials.

Safe Clearance: Set to 0.25 - 0.5.

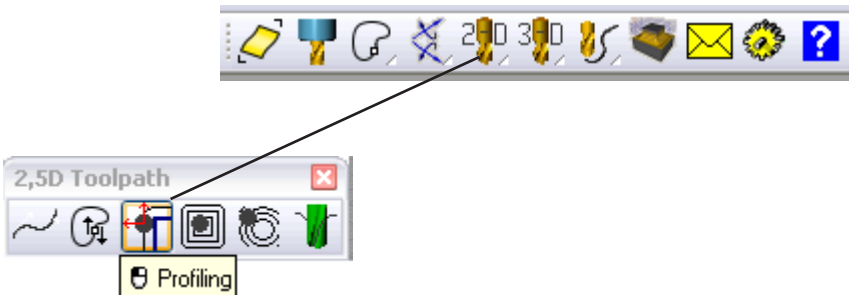
Click **OK** to calculate toolpath.



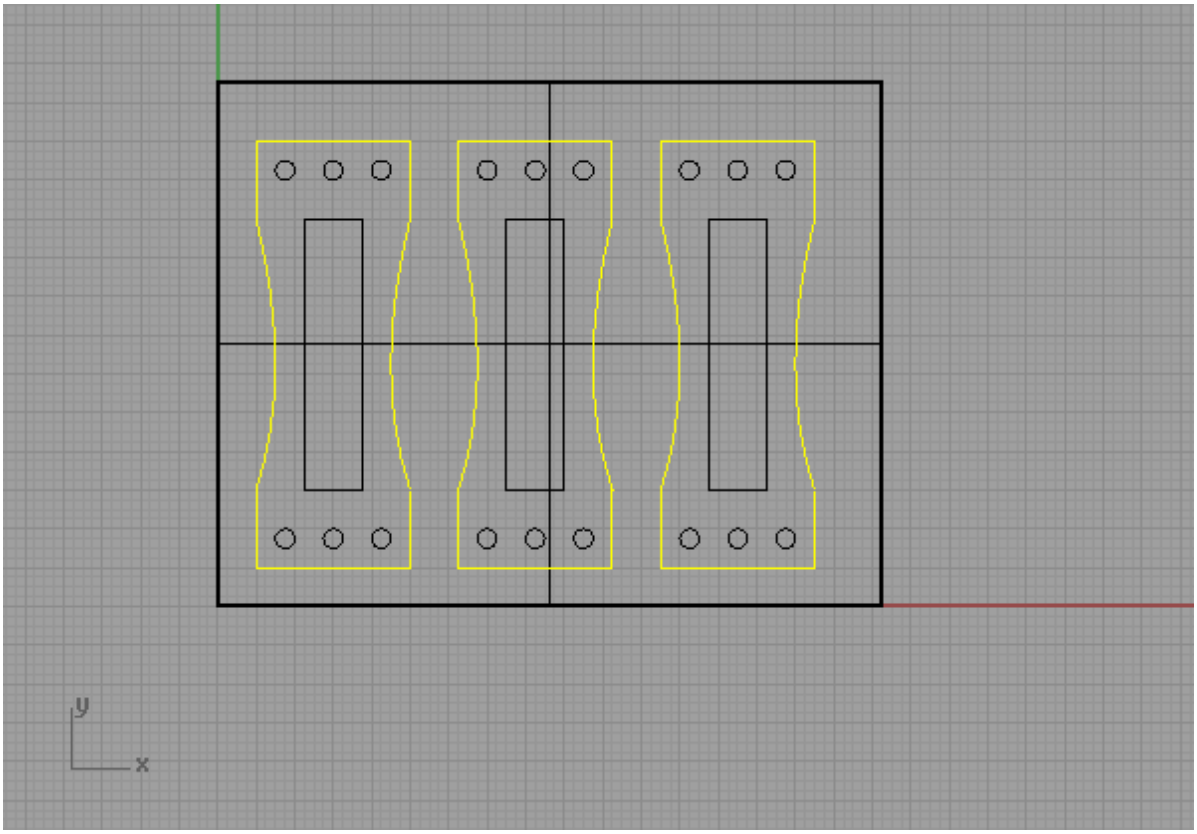
Completed Toolpath.

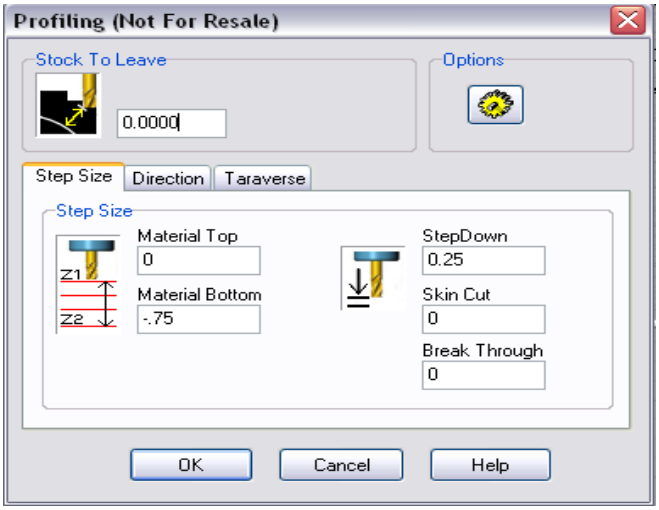


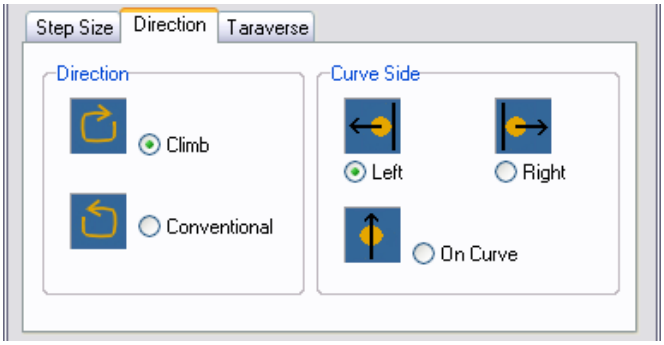
Step 8: Set up a toolpath. Click the **Create 2D Toolpath** button. Choose **Profiling** from the 2D Toolpath menu.

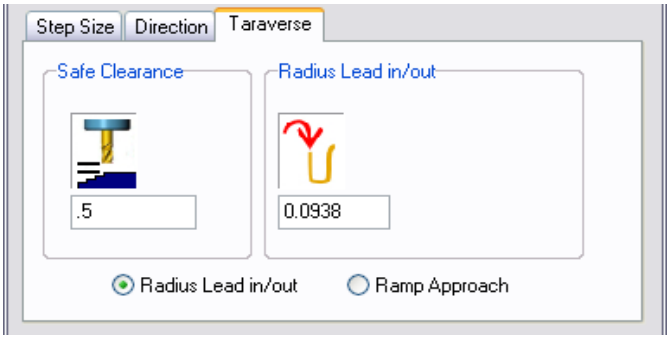


The prompt instructs you to select the profile curves and press **Enter**. Note: Try to select curves in clockwise order.









Step 9: Profile Toolpath Setup:

Material Top: Set to 0 by default.

Material Bottom: The thickness of the material.
In this case we set to -.75 (3/4 plywood)

StepDown: How thick of a layer the bit will cut as it mills down the part. Rules of thumb:

Foam: Max StepDown = cutting length of bit
Wood: Max StepDown = 1/3 diameter of bit
Metal: Max StepDown = 1/4 diameter of bit

Direction: The direction of the passes the bit will make as it cuts.

Climb: Used for cutting wood.

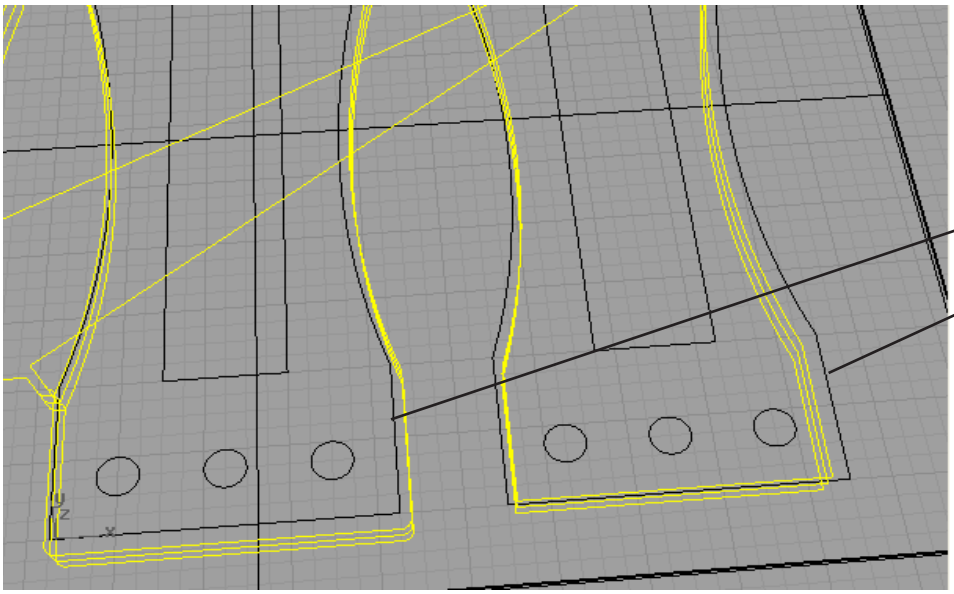
Conventional: Used for monolythic materials.

Curve Side: If curves have been chosen in clock-wise order choose left. If you offset curves ahead of time compensating for bit diameter, then you can choose On Curve.

Safe Clearance: Set to 0.25 - 0.5.

Click **OK** to calculate toolpath.

Note: Check to make sure profile cuts are on the right side of the curve.

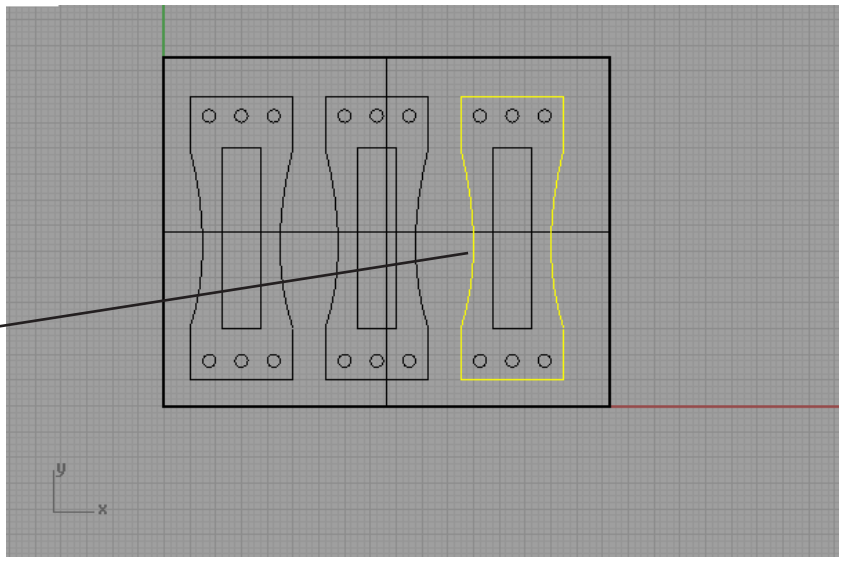


If everything is on the correct side then you can continue to **Step 10.**
If cut is on the wrong side you need to reverse the direction of the curve.

To reverse the curve direction:

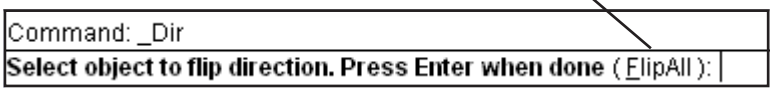
Delete profile cuts by **highlighting cuts** and pressing **Delete**.

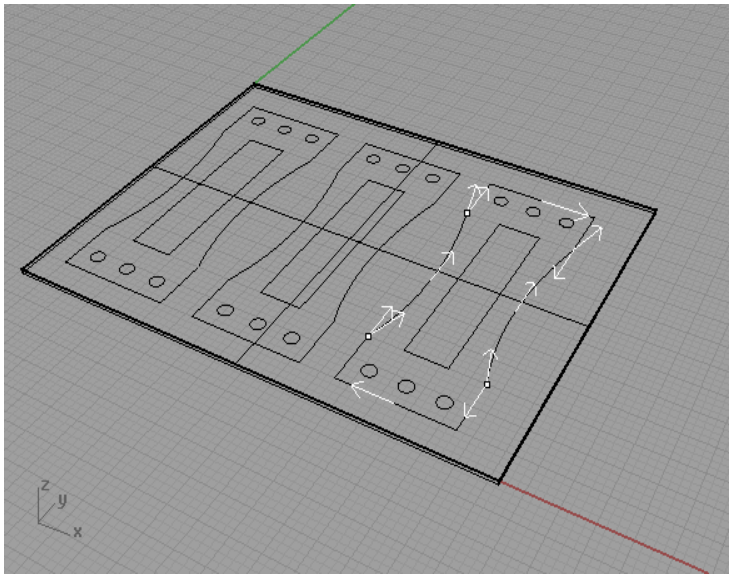
Shift Select curves in a clockwise direction on the incorrect profile.



Go to 2D tool path and click **Change Curve Direction**.

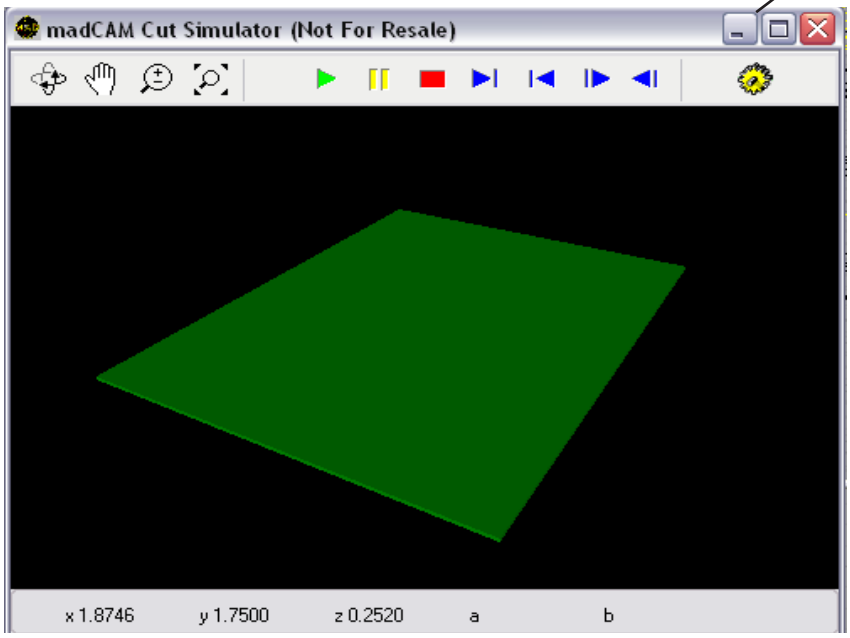
Choose **FlipAll** in prompt, hit **Enter**.





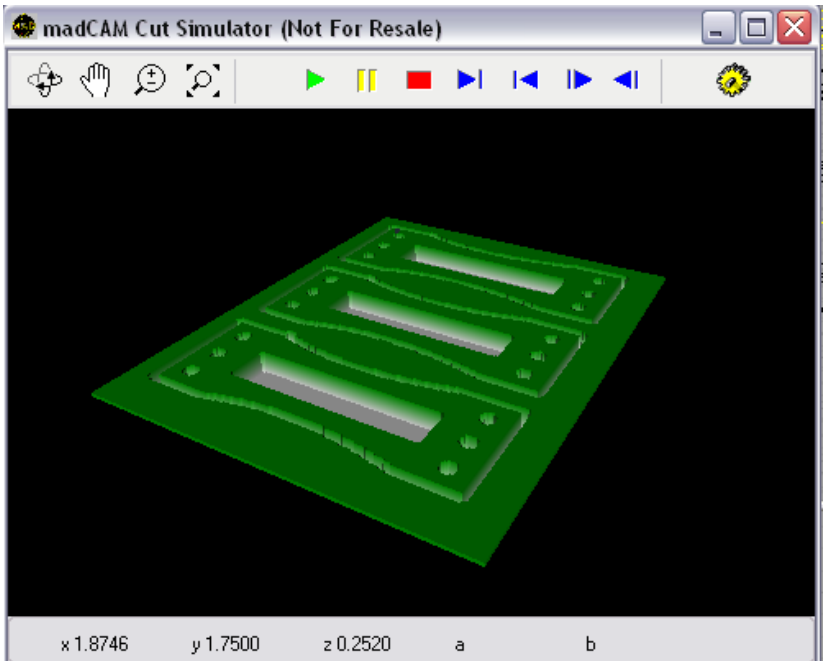
Curve direction has now been corrected.
Repeat **Steps 8 & 9**.

Step 10: Simulate Cutting Job. Click the **Simulate** button. The Cut Simulator Window will open.



The buttons at the top of the window are used for controlling view, cut simulation and cut simulation settings.
Press **Play** to start simulation.

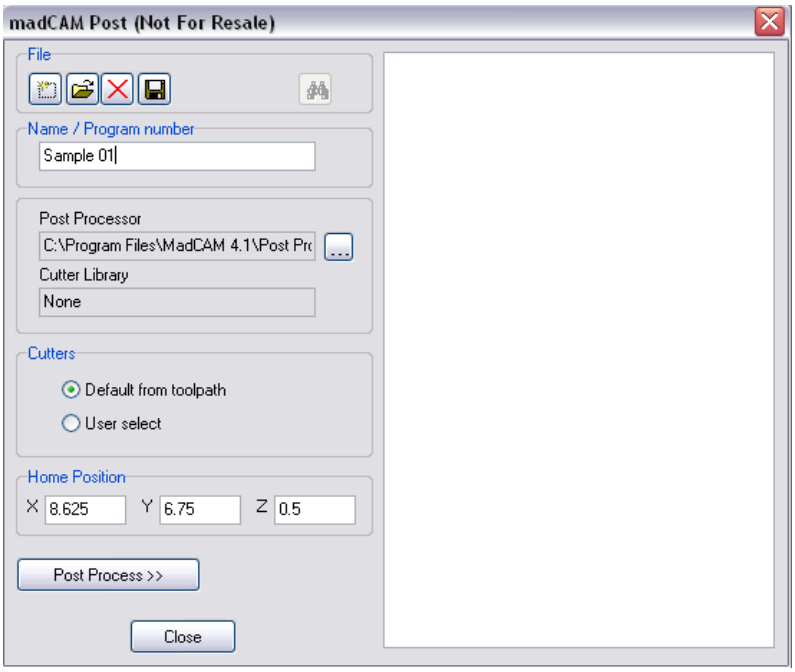
What you see here is what you will get!



Step 9: Post the toolpath to the Yale Precix Processor.

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

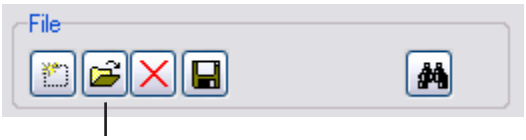
Step 10: Verify the post processor and cutter library settings are correct.



Browse and select the **Post Processor** by following the path: ***C:\Program Files (x86)\MadCam 4.2\Post-processors\Precix_Yale_v2.***

Default from toolpath should be selected for Cutters.

Click on the **Post Process button**. Name file using atleast **8 characters** and add **.gc** as the file extension. Save the file to your user account or directly to the thumb drive for the mill.



Click this button for viewing or editing the output file.

The posted file.