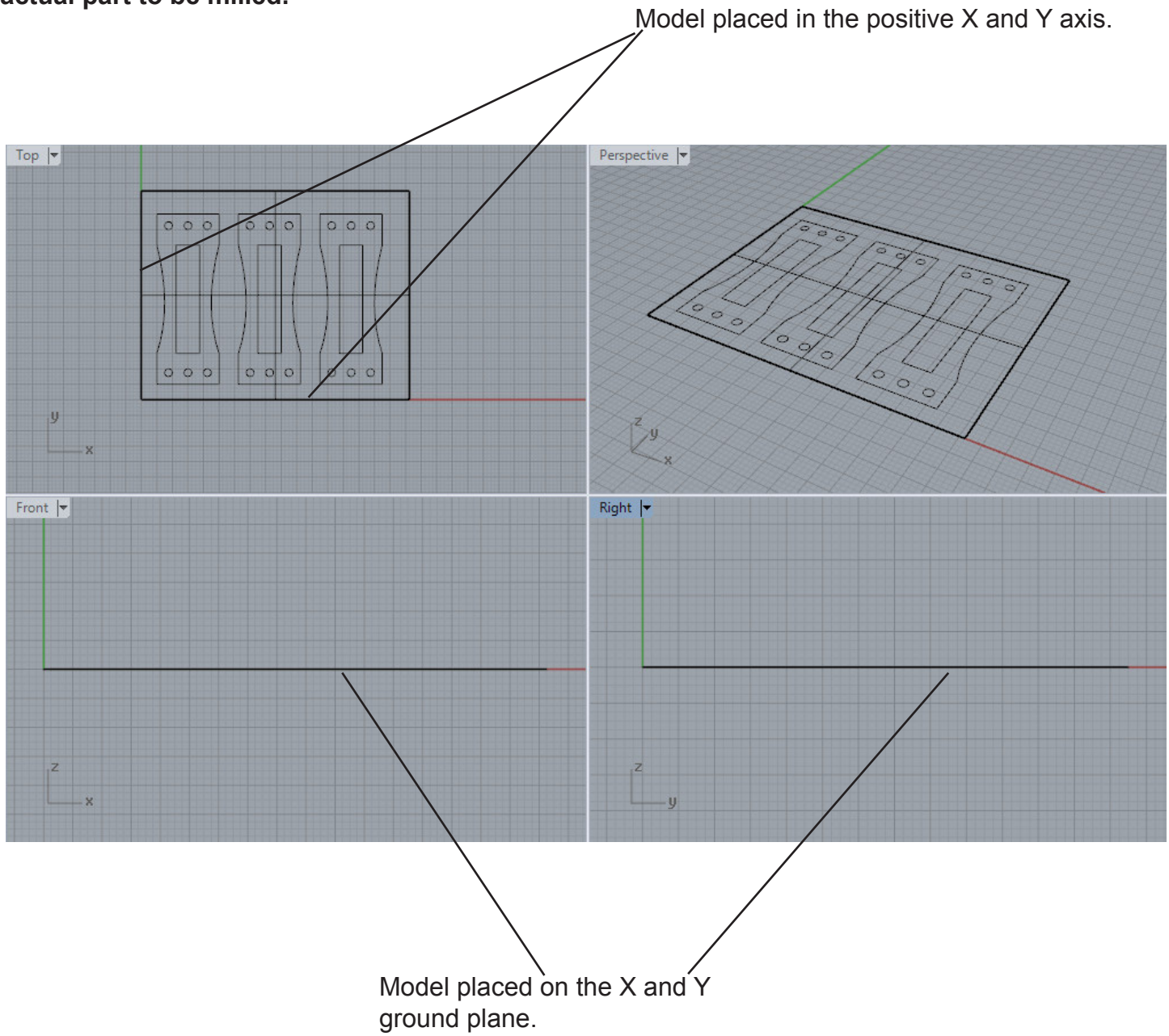
MadCam 5.0: 2D Profile Toolpath

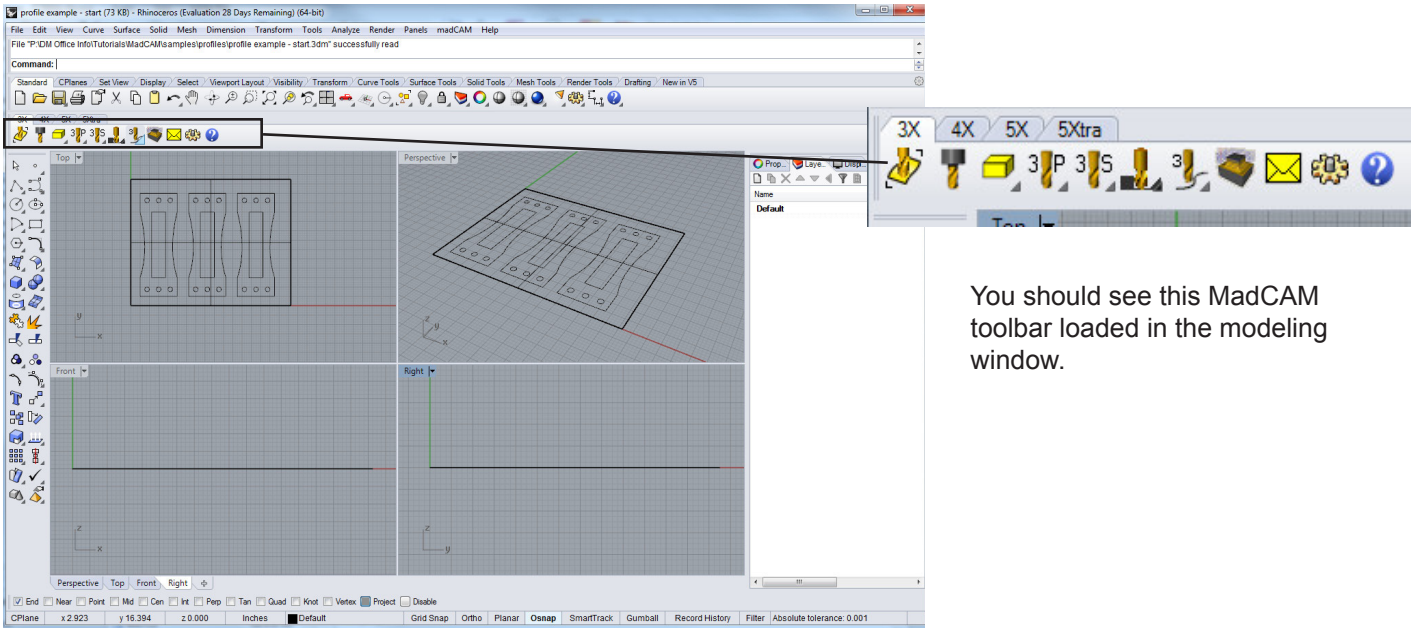
Digital Media Tutorial
Written By John Eberhart

MadCAM 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.

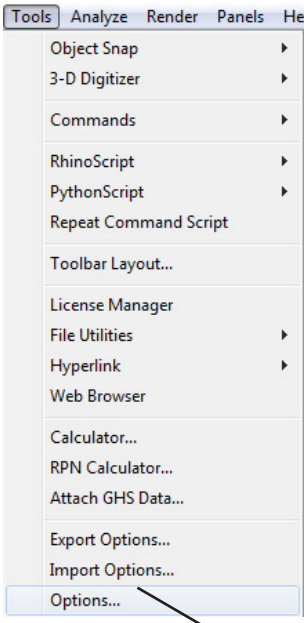
Step 1: Open or Create a 2D file in Rhino. MadCAM runs inside Rhino, and is automatically opened when Rhino is opened. (If the MadCAM toolbar does not open, see the next page for instructions on loading the toolbar).

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.**

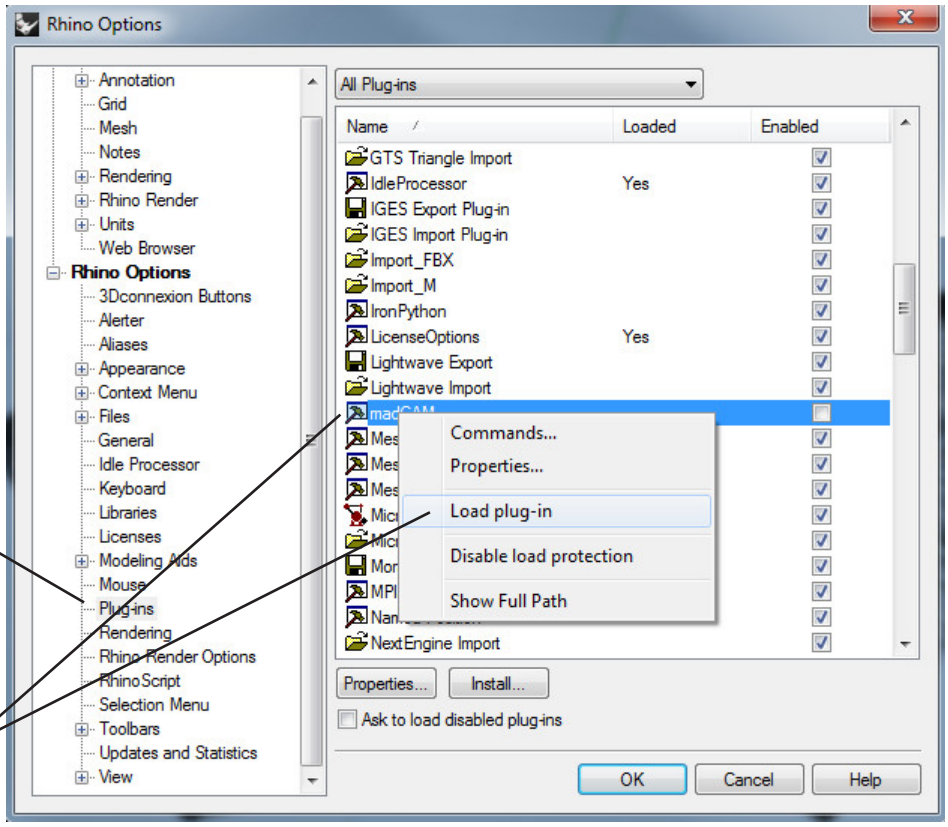




You should see this MadCAM toolbar loaded in the modeling window.



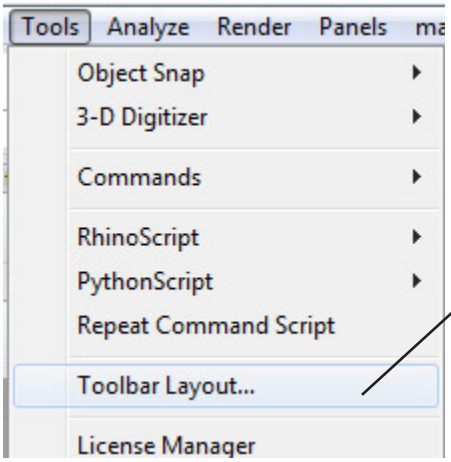
The first time you launch Rhino, the Madcam Plugin May have to be loaded. To load the MadCAM plugin, choose **Tools>Options**. The following window will appear.



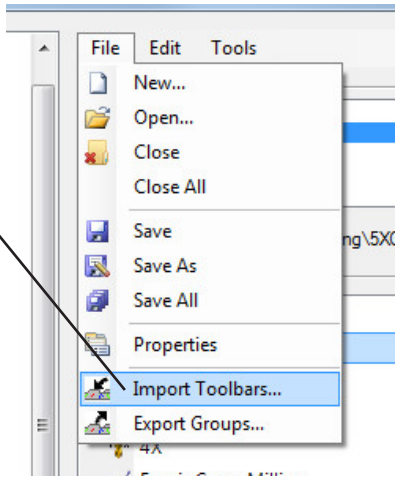
Click on Plugins and then select MadCAM from the list of plugins.

Right Click on MadCAM and choose Load Plug-in.

If the MadCAM toolbar is also not always visible, it can be loaded by either typing in a MadCam command like **MC_Options or MC_SelectModel**:

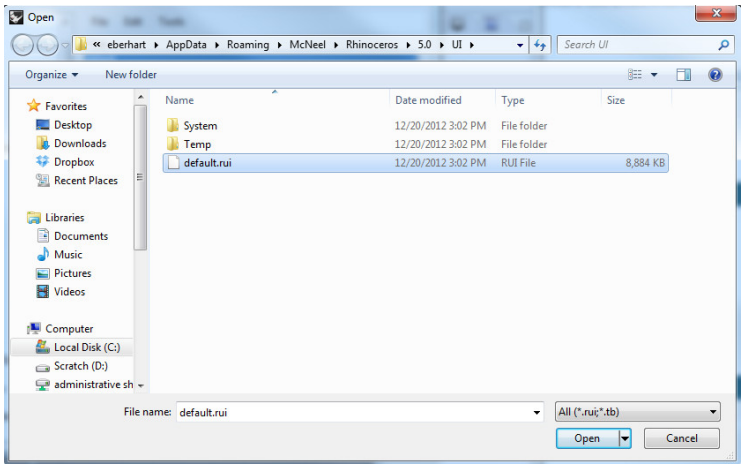


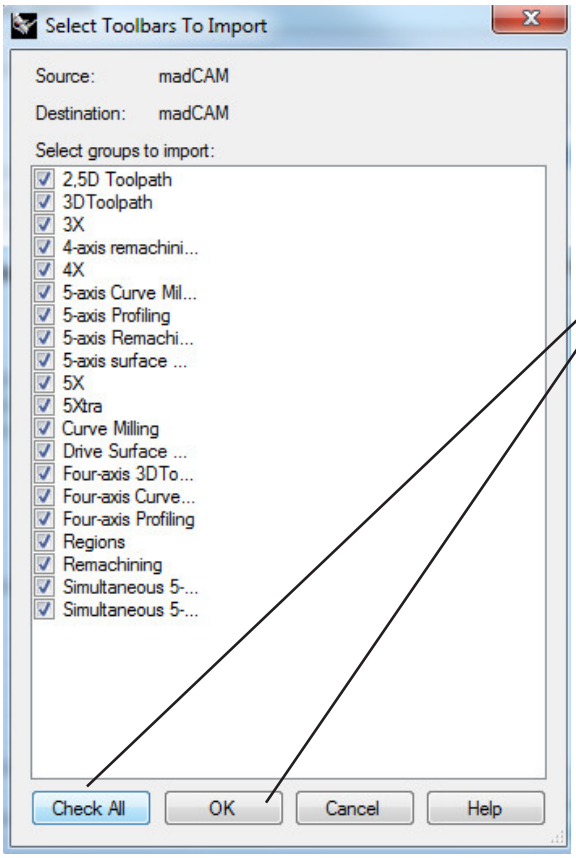
Highlight **Default** and choose **Import** Toolbars...



Or by selecting **Tools>Toolbar Layout.**

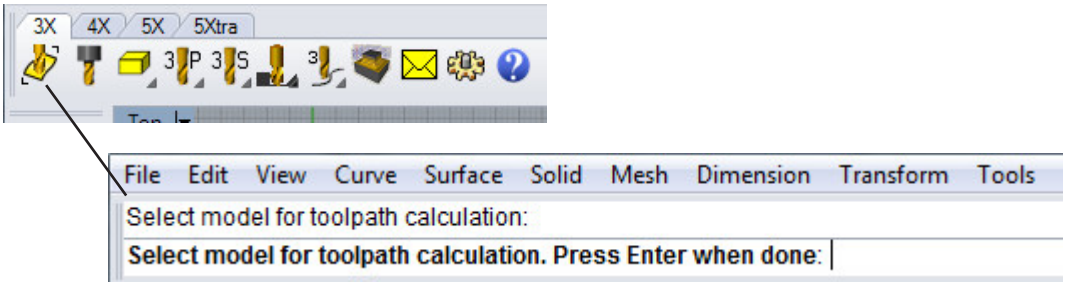
Find MadCAM5 by following the path:
C:\Program Files\madcam 5.0 (64-Bit)\UI\madcam.rui



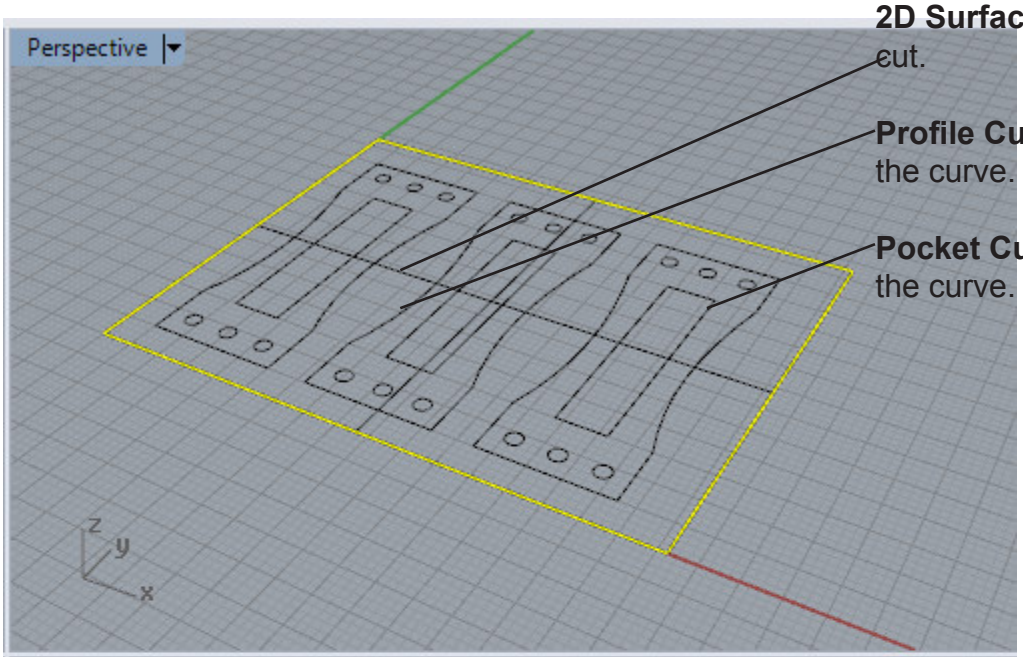


Once you choose the MadCAM toolbar file, it will take you to this window. Click **Choose All** and then click **OK**. You have now loaded the Madcam toolbars.

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.



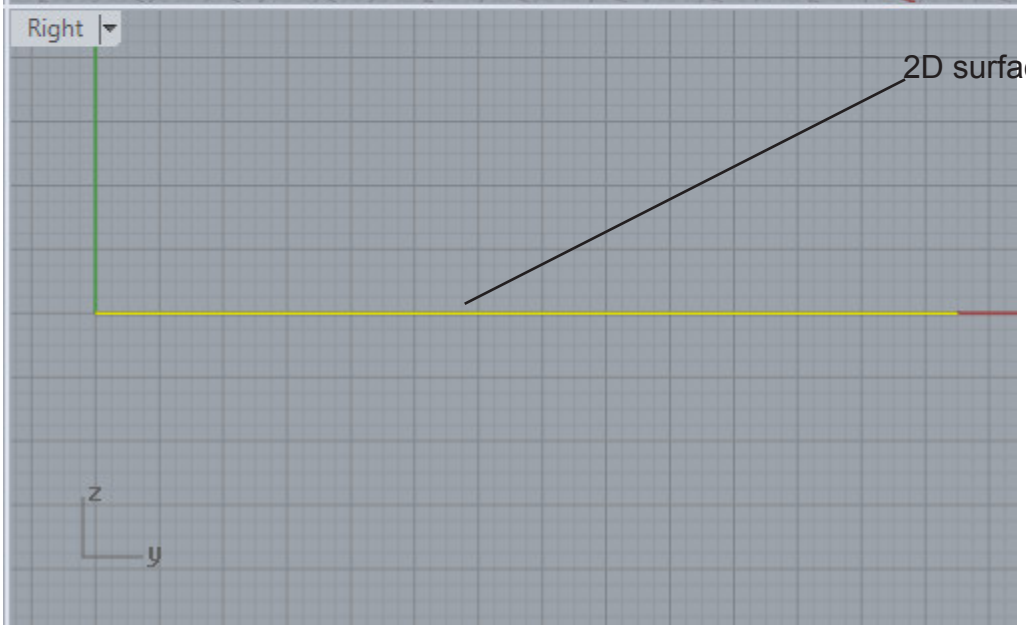
The loaded object will have a box around it.



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 and or load an existing cutter.

Step 5: Cutter parameter window: Load a predefined cutter based on the material you want to cut by selecting it from the right side. This will give you a cutter with predefined Feed and Spindle Speed rates.

Name: Give a unique name.

Type of Bit:
Choose the bit that best represents your bit shape.

Diameter: Should be set

Length: Measure while in a collet and input length

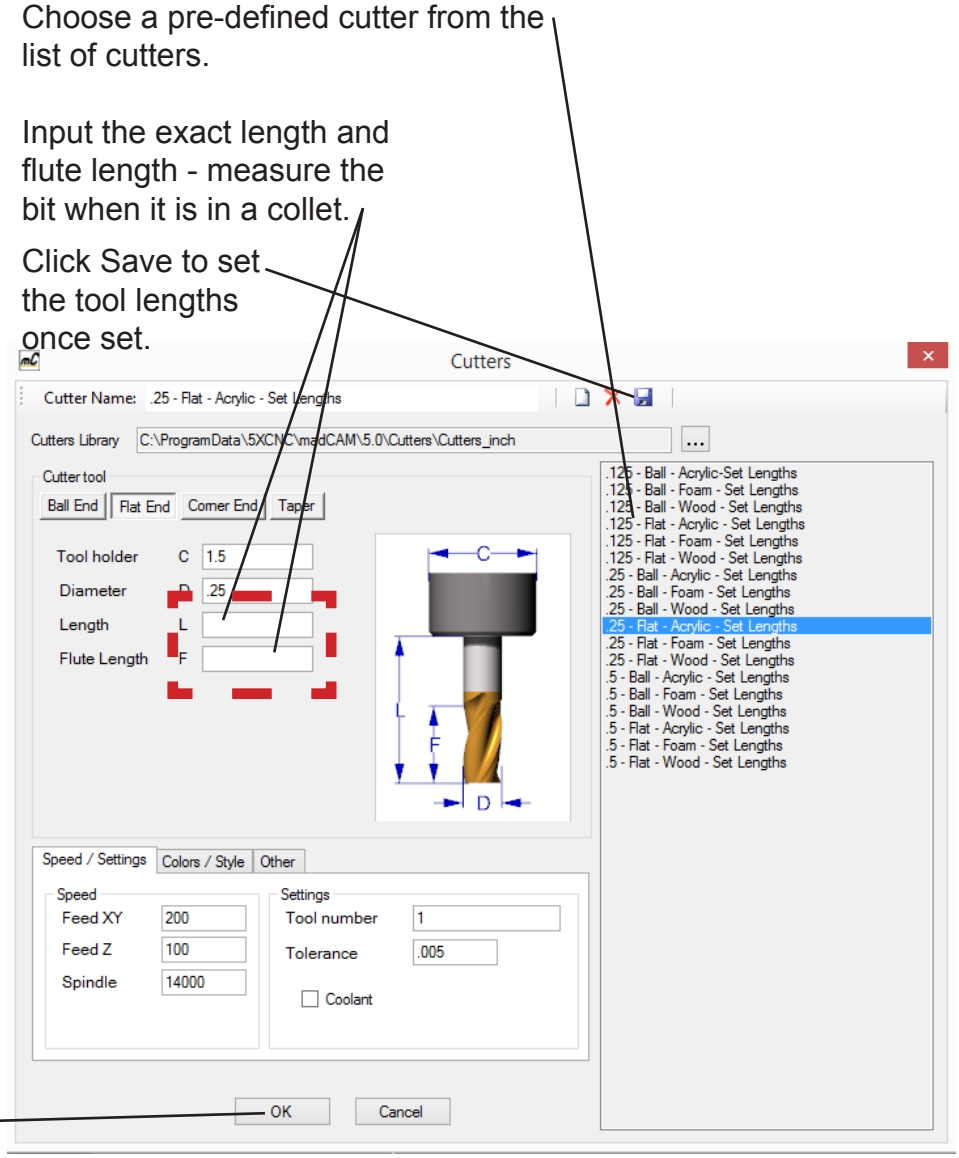
Cutting Length: Measure and input length

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.

If no changes are made, click **OK** to load the tool.



You can adjust the settings of a tool to fit your particular needs. Do this only if you are comfortable changing the settings.

Setting the Feeds and Spindle Speeds - Rules of thumb:

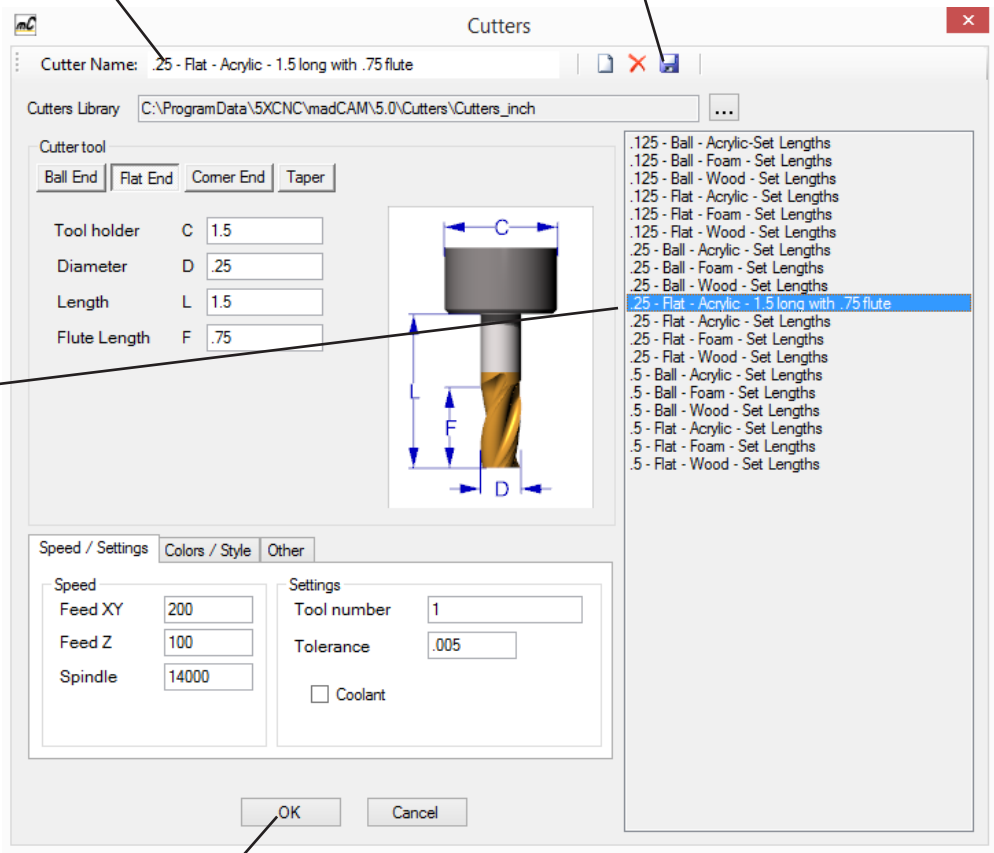
Feed X,Y - This is how fast the bit is pushed through the material and is based on a number of factors. Fior you tools, it should not be set to **more than 280** inches/min. This speed can be adjusted at the machine

Feed Z - Set this to a **max of 100** inches/min as this can be hard on the bit

Spindle Speed - Set to **Max of 12,000 rpm** (except for metal) this can be adjusted at the machine.

If you want to make your own tool, make changes to the tool settings and change the name of the tool under cutter name:

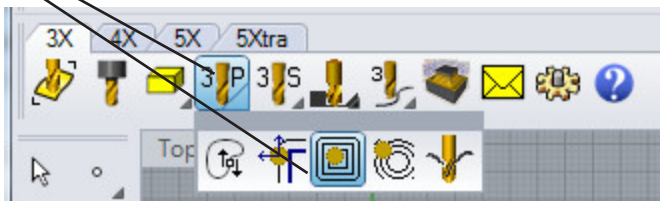
Click the “Save” icon to save the tool to the library



Your cutomized tool will be saved to library for use down the road.

Click OK to make that tool active

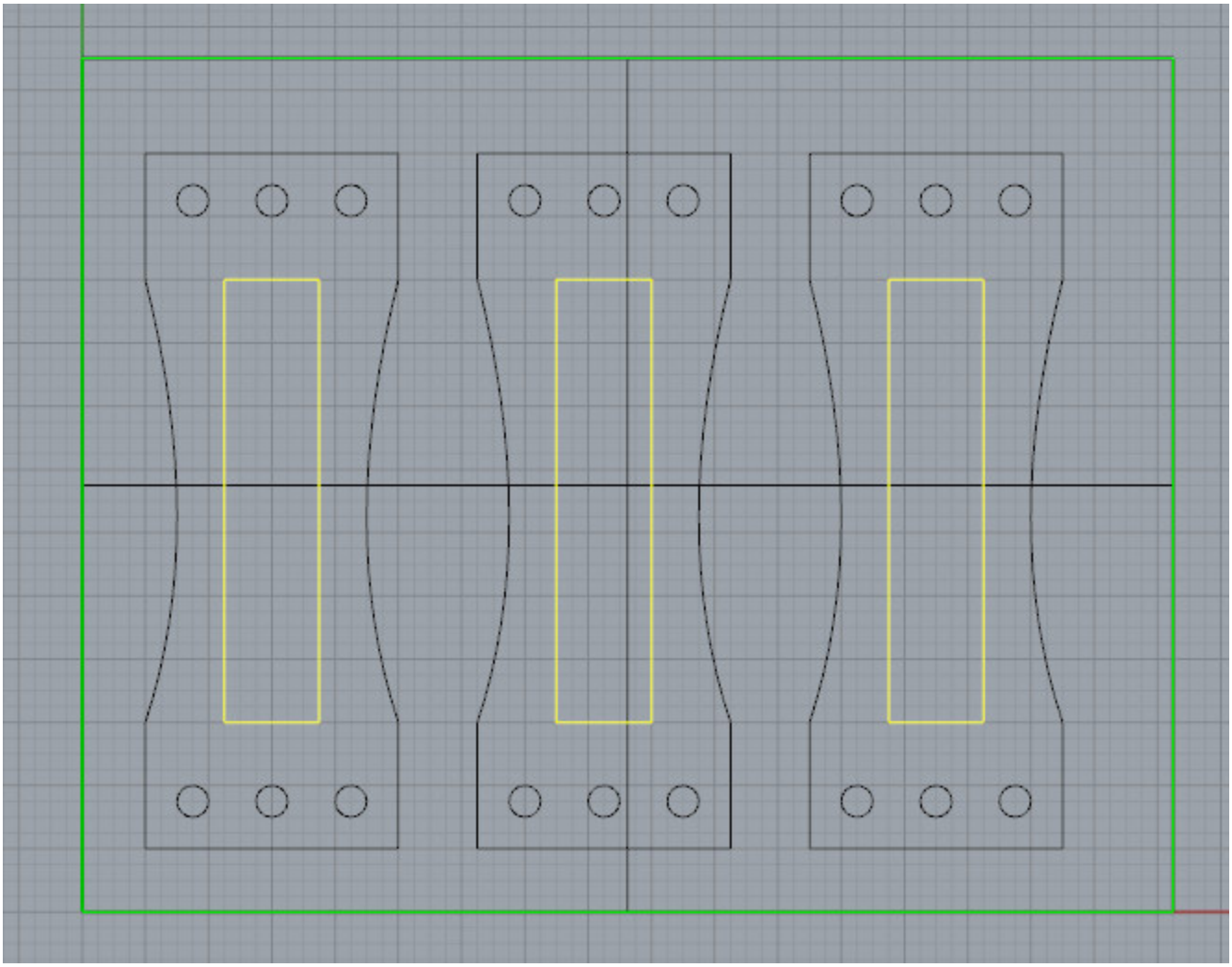
Step 6: Pocketing. In order to pocket cut, your curves must be completely closed. Go to 3P toolbar and select **Pocketing.**



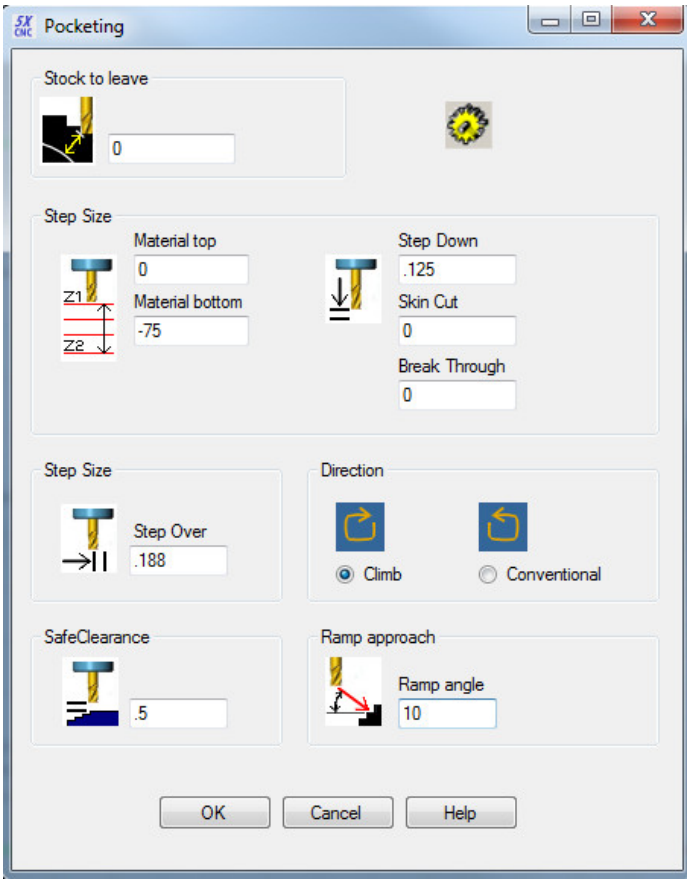
Pocketing is best for removing a depth of material inside a boundary curve. If you want to cut all the way through, you should used profile cutting.

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

Select the closed curves for the pocket cut 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.

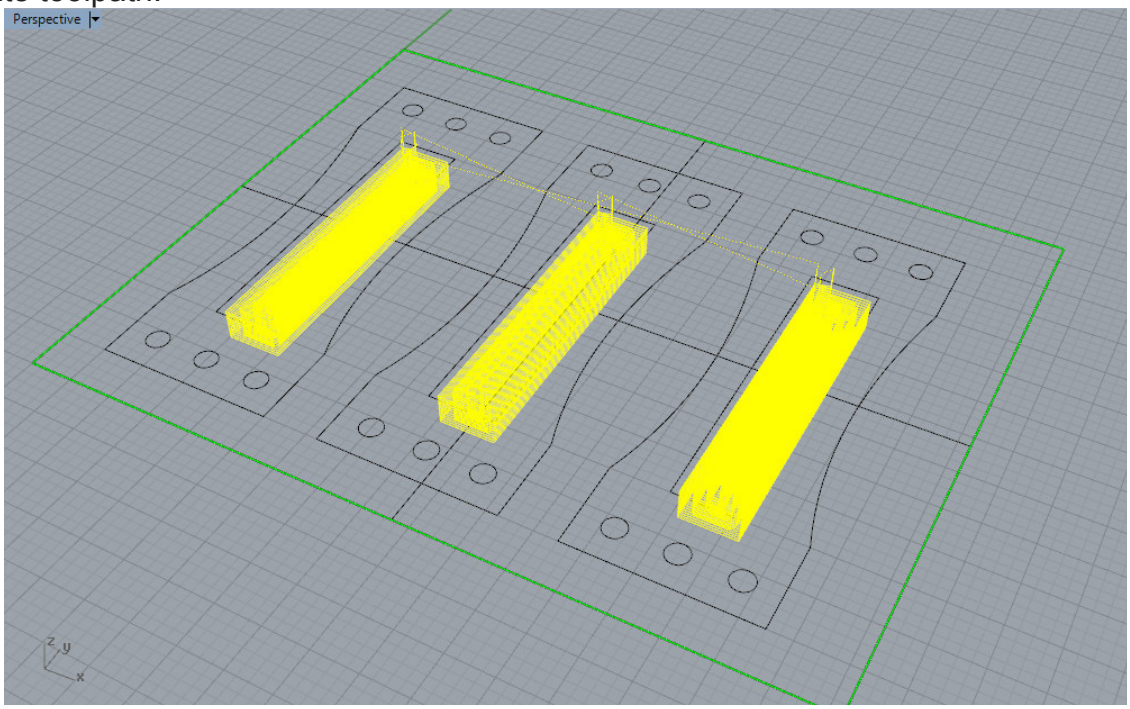
Climb: Used for cutting wood.

Conventional: Used for monolithic materials.

Safe Clearance: Set to 0.5

Ramp Angle: Set to 10 deg.

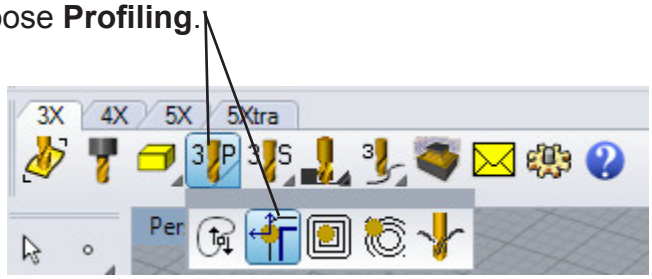
Click **OK** to calculate toolpath.



Completed Toolpath.

Step 8: Set up a Profile Curve Toolpath.

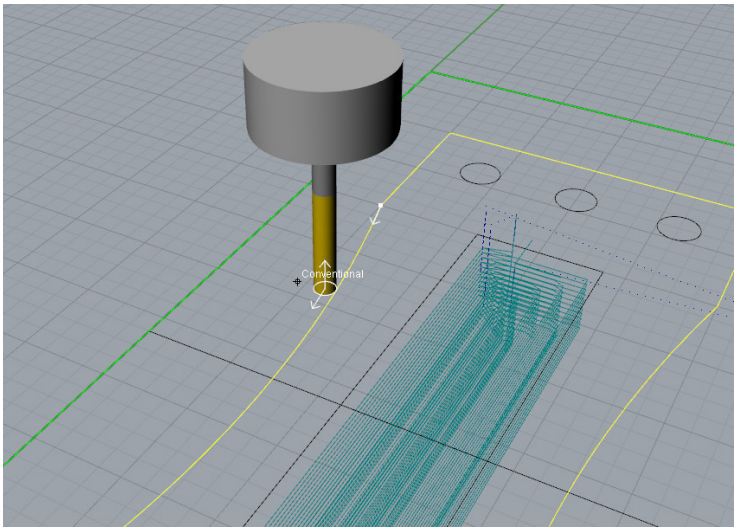
Click the **Create 3P Toolpath** button. Choose **Profiling**.

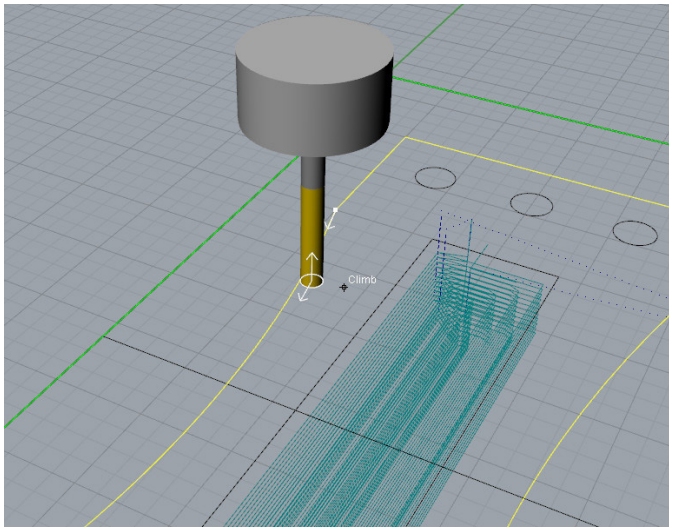


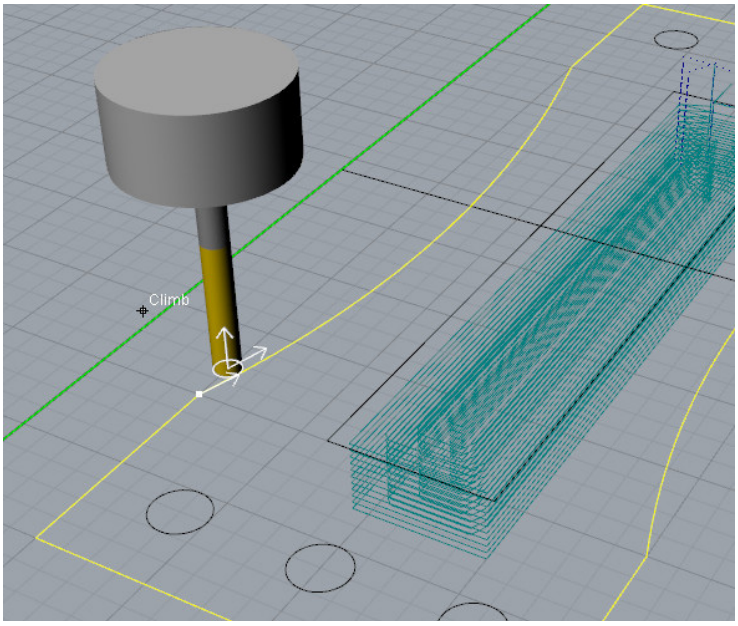
The prompt instructs you to select the profile curves and press **Enter**. Note: Try to select curves in clockwise order.

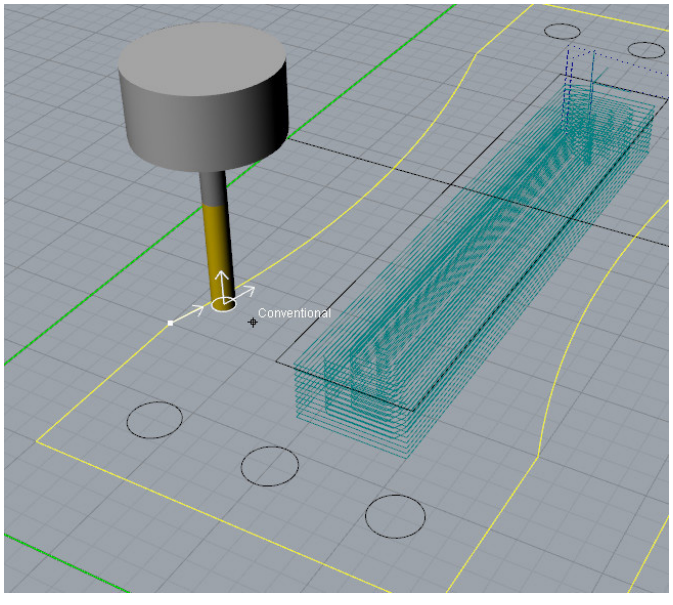
Once you hit Enter, you will be prompted to select the side of the line to cut on and if you want to do a climb or conventional cut.

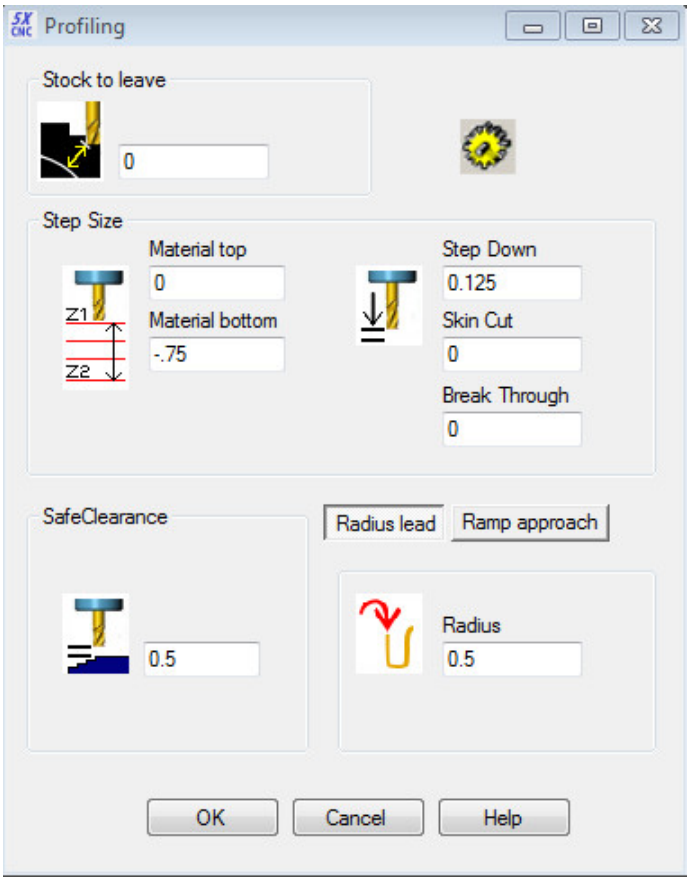
Climb Cutting-Used for material with a grain like wood and some plastics.
Conventional cutting-used for monolithic materials such as metal and MDF











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:

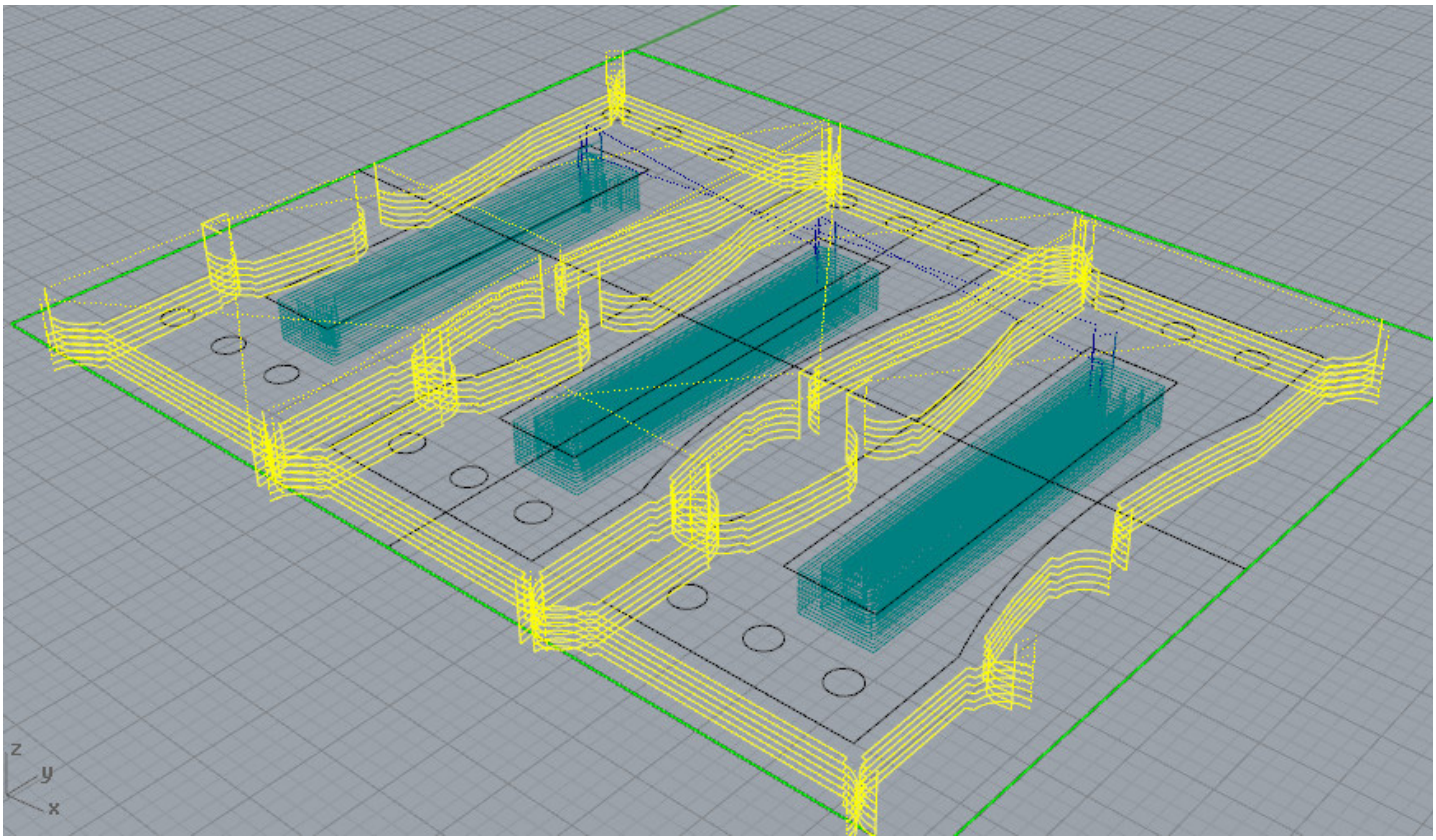
Foam: Max StepDown = cutting length of bit
Wood: Max StepDown = 1/3 diameter of bit
Metal: Max StepDown = 1/4 diameter of bit

Choose Radius Lead Approach

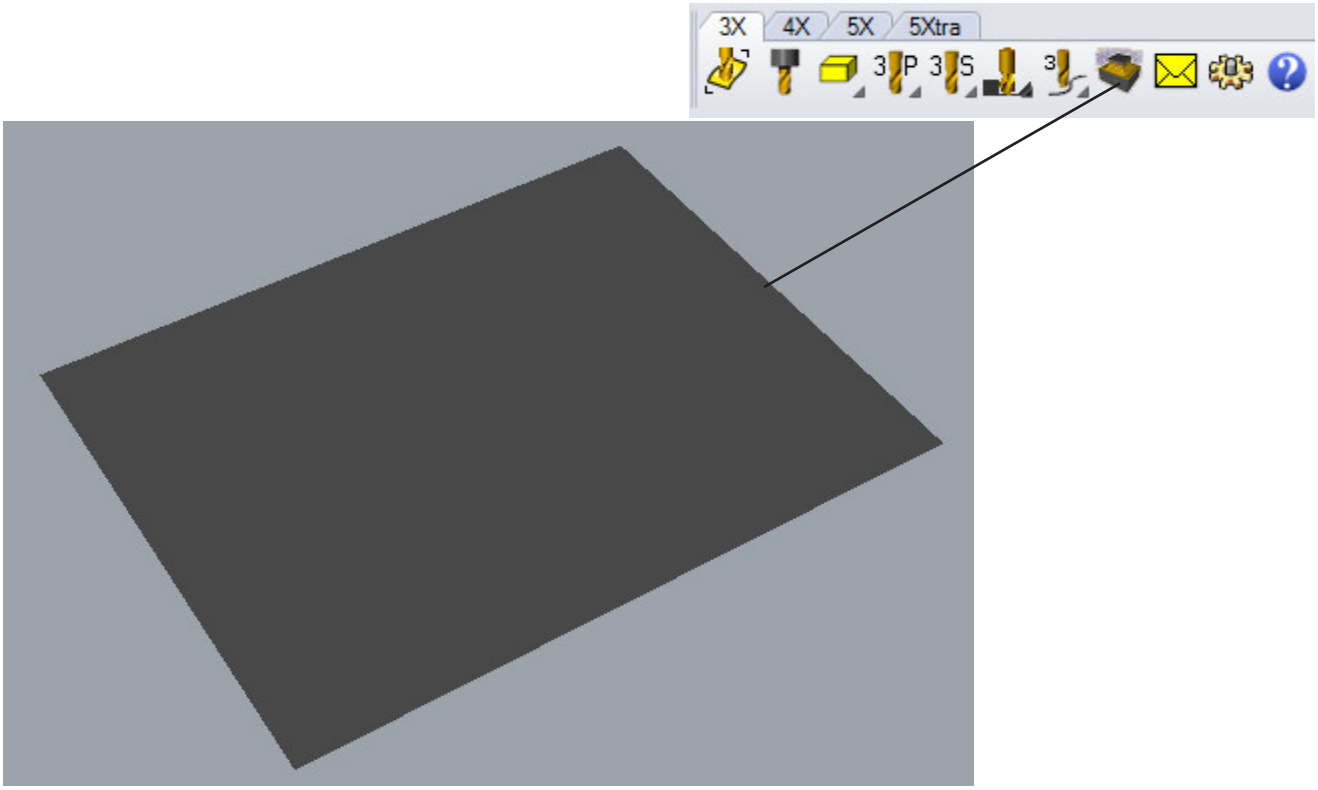
Safe Clearance: Set to 0.5.

Click **OK** to calculate toolpath.

In this case I joined each profile and created a toolpath for each profile. This makes milling more efficent on the mill as it will cut each profile completely before moving to the next profile.

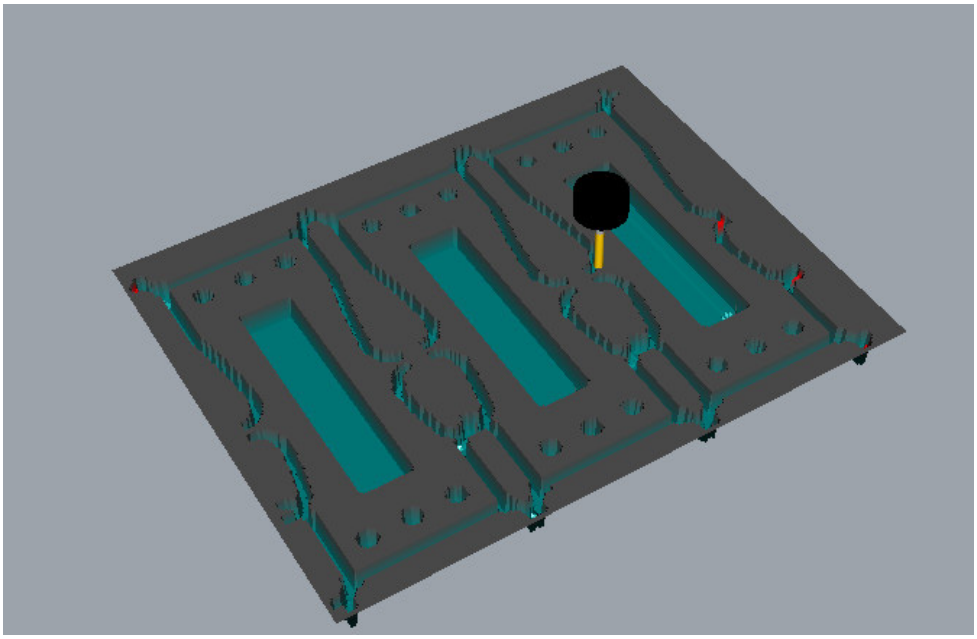


Step 10: Simulate Cutting Job. Click the **Simulate** button. The Cut Simulator Window will open.



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

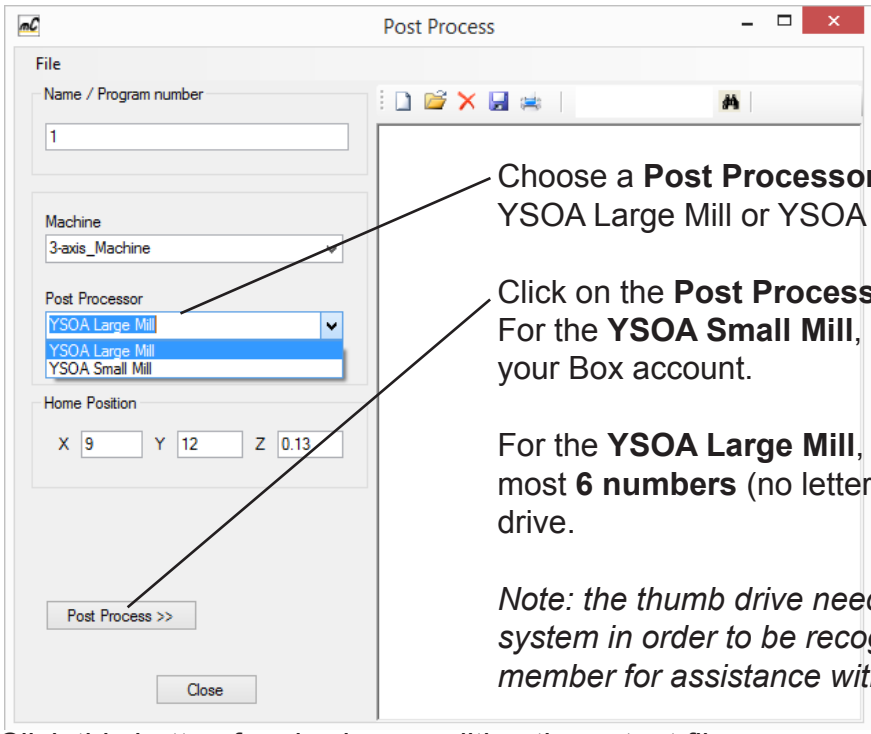
What you see here is what you will get!



Step 11: Post the toolpath to the Yale Post Processors.

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

Step 12: Verify the post processor and cutter library settings are correct.



Choose a **Post Processor** for the machine you want to use: YSOA Large Mill or YSOA Small Mill.

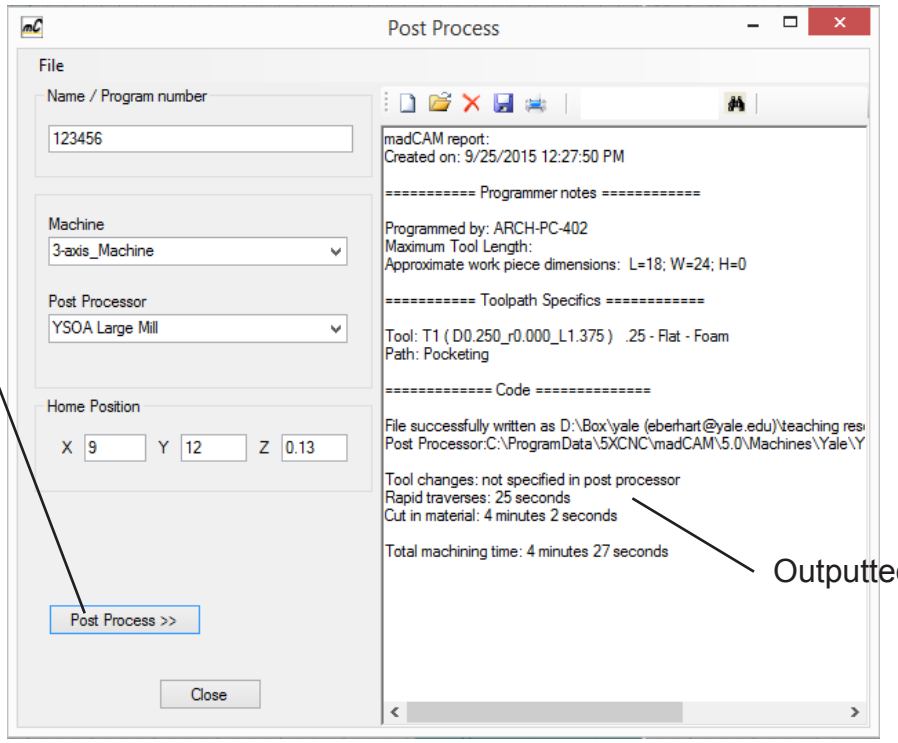
Click on the **Post Process button**:
For the **YSOA Small Mill**, you can give it any name and save it to your Box account.

For the **YSOA Large Mill**, you need to name the file using at the most **6 numbers** (no letters or spaces). Save the file to a thumb drive.

Note: the thumb drive needs to be formatted with a "fat 32" file system in order to be recognized by the mills. See a DM staff member for assistance with this if needed.

Click this button for viewing or editing the output file.

Click on the Post Process button to write your file.



Outputted File