

MadCAM 5.0: Large & Medium CNC Tool Path Generator

Digital Media Tutorial
Written By John Eberhart

MadCAM is a tool path generator that works inside Rhino. It uses Rhino's interface to generate the tool paths that are exported as a text file to the mills.

3 Axis Milling

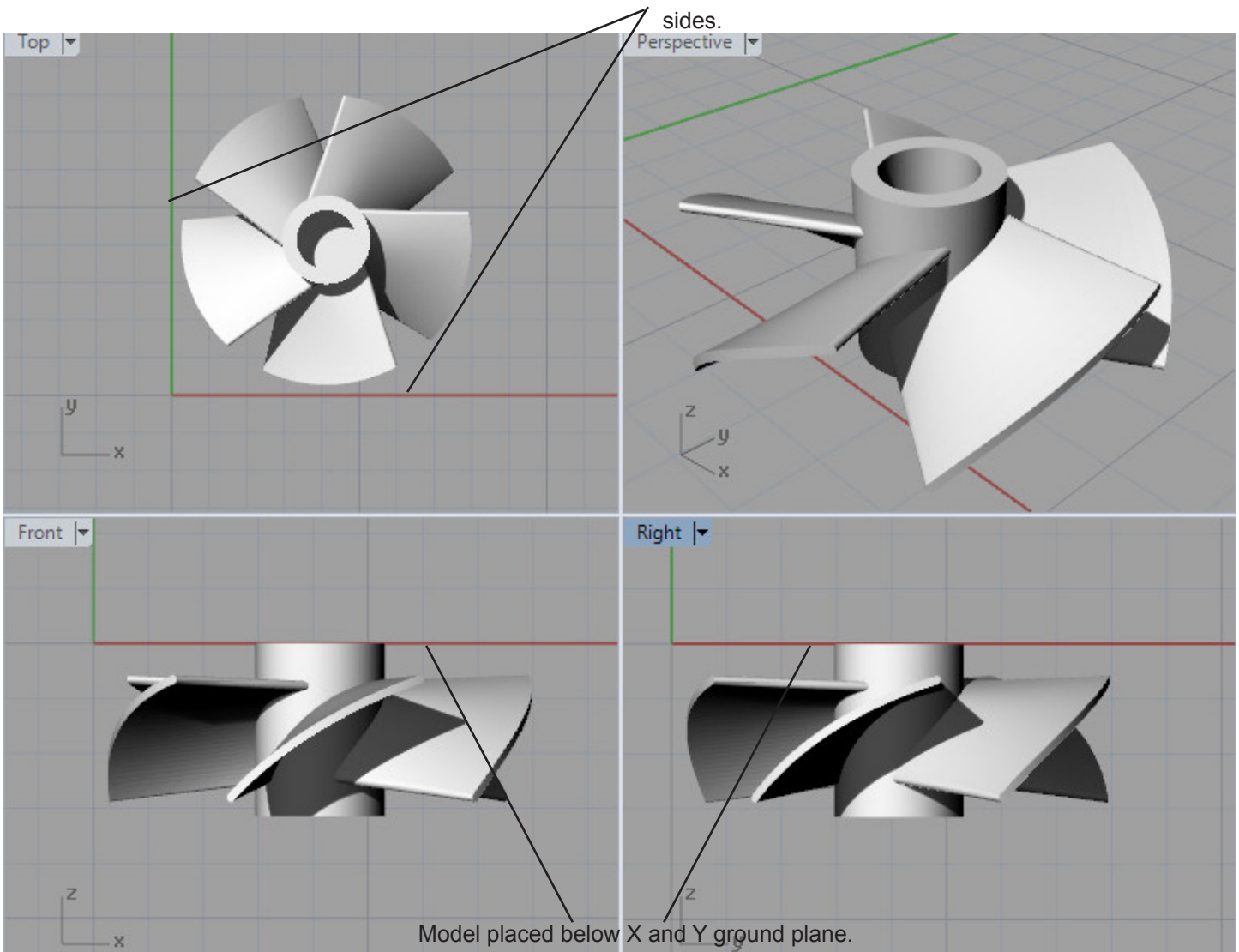
Step 1: Open your file in Rhino, MadCAM runs inside Rhino, and is automatically opened when Rhino is opened. If the MadCAM toolbar is not there see next page for instructions on loading the toolbar.

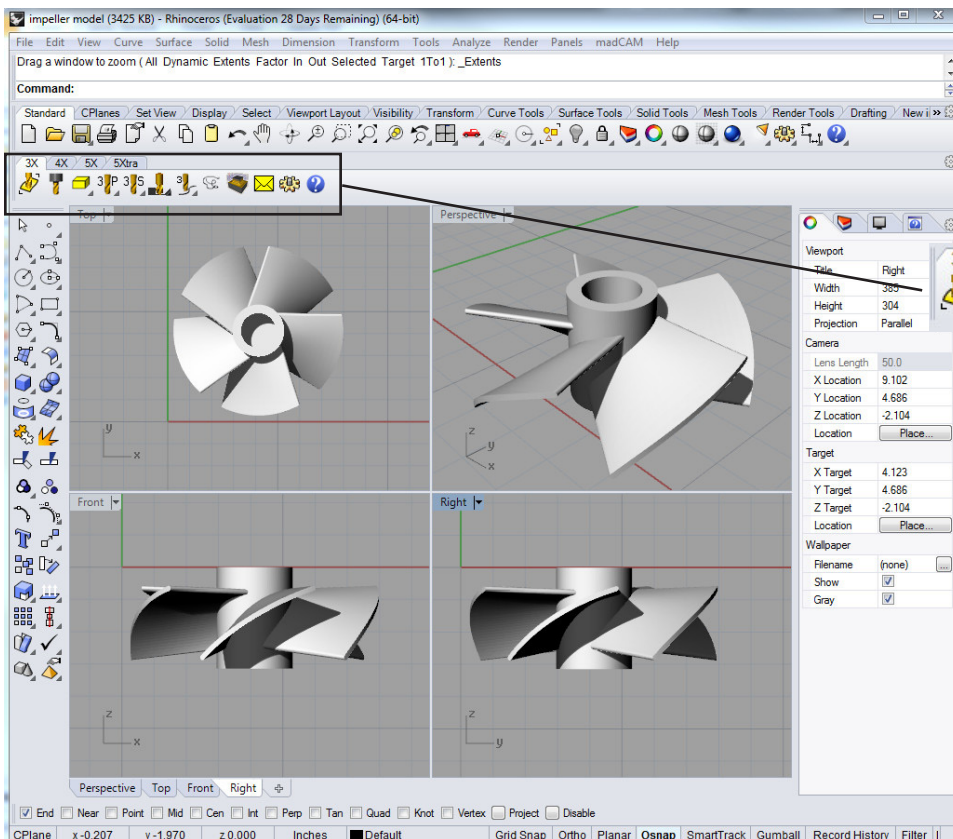
Step 2: Prepare Model. Correctly place your part in the modeling window. The mills use the model origin location as the start point for the mill. You need to move your part so that it is ***completely in the positive X and Y axis and BELOW the ground plane.***

NOTE: Model in Rhino MUST be scaled to match the actual part to be milled.

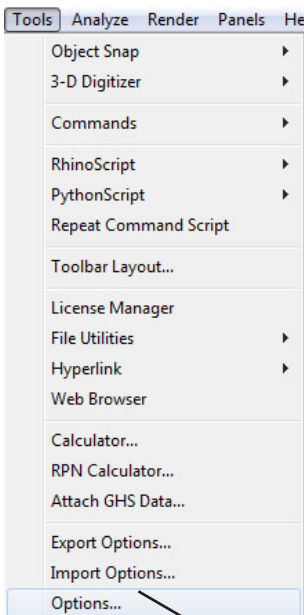
Model placed in the positive X and Y axis.

Note: It is advised to leave a little extra room on the sides.

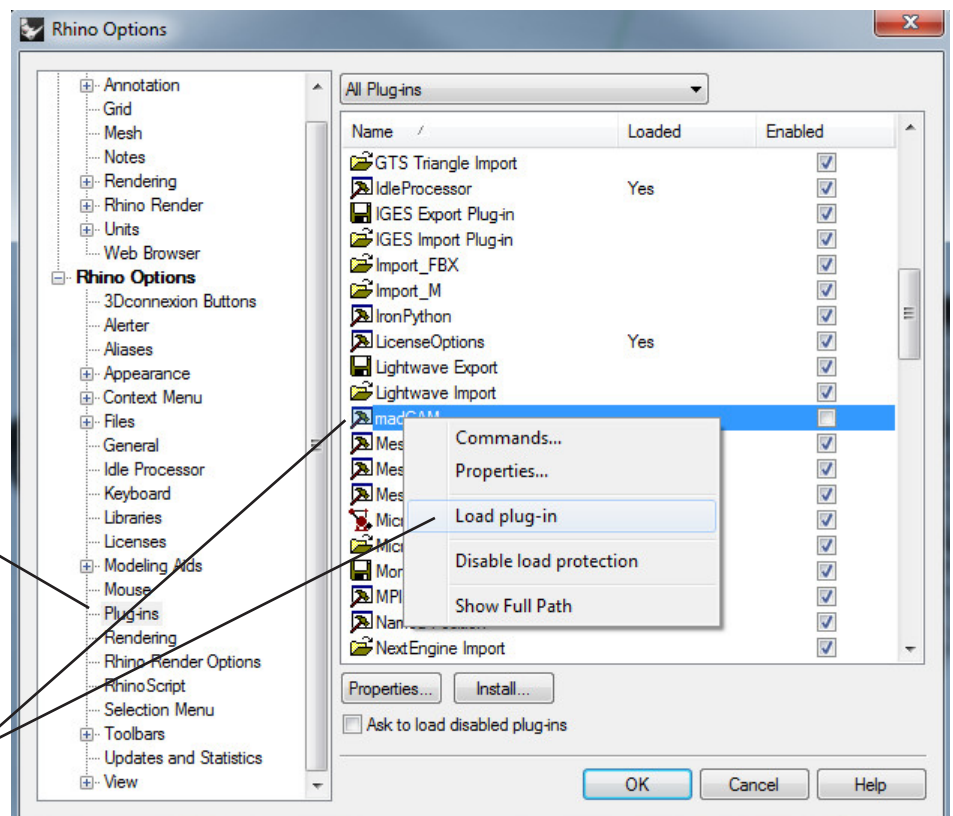




You should see this MadCAM toolbar loaded in the modeling window.



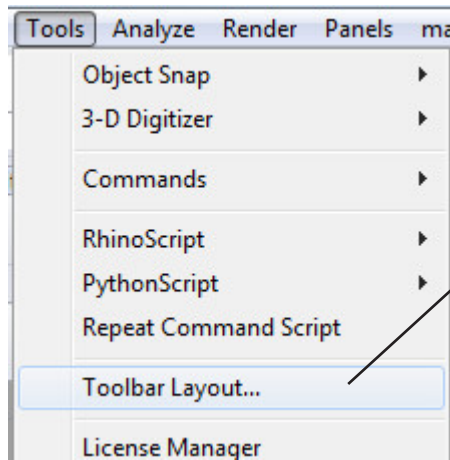
The first time you launch Rhino, the Madcam Plugin May have to be loaded. To load the MadCAM plugin, choose **Tools>Options**. The following window will appear.



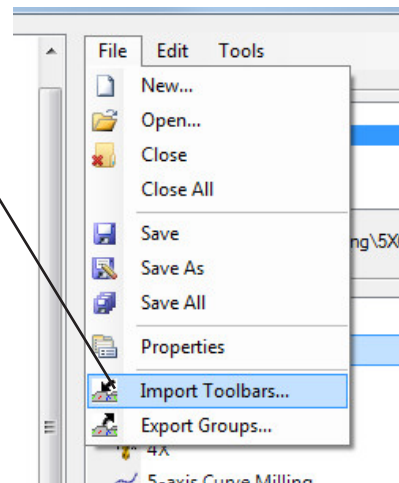
Click on Plugins and then select MadCAM from the list of plugins.

Right Click on MadCAM and choose Load Plug-in.

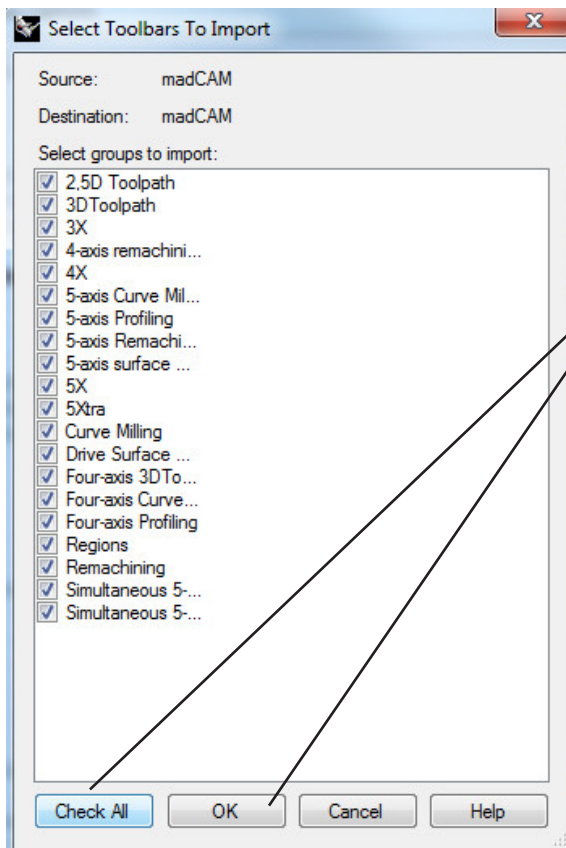
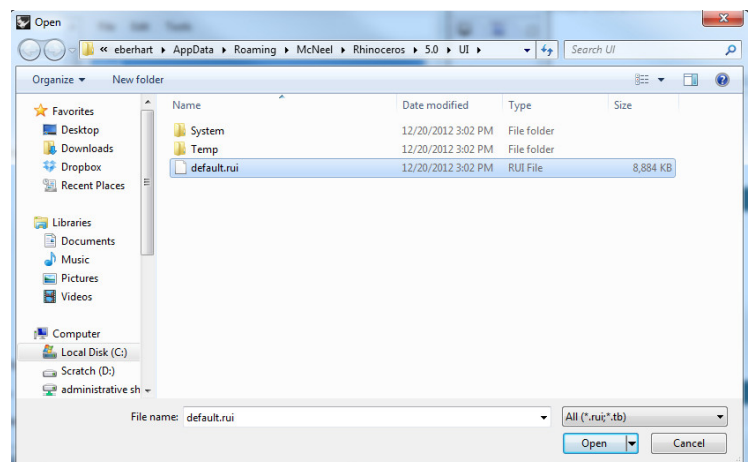
If the MadCAM toolbar is also not always visible, it can be loaded by selecting **Tools>Toolbar Layout**.



Highlight
Default and
choose
Import Tool-
bars...

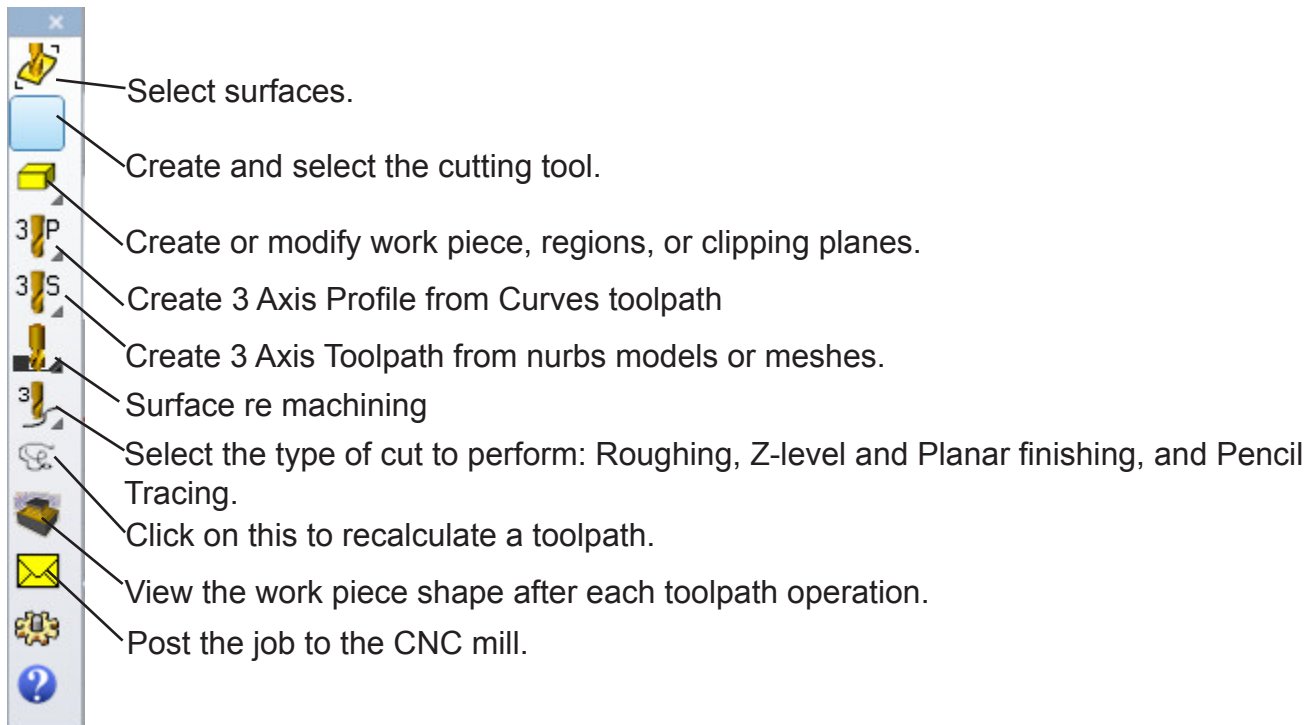


Find MadCAM5 by following the path:
C:\Program Files\madcam 5.0 (64-Bit)\UI\madcam.rui



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.

The MadCAM Toolbar:



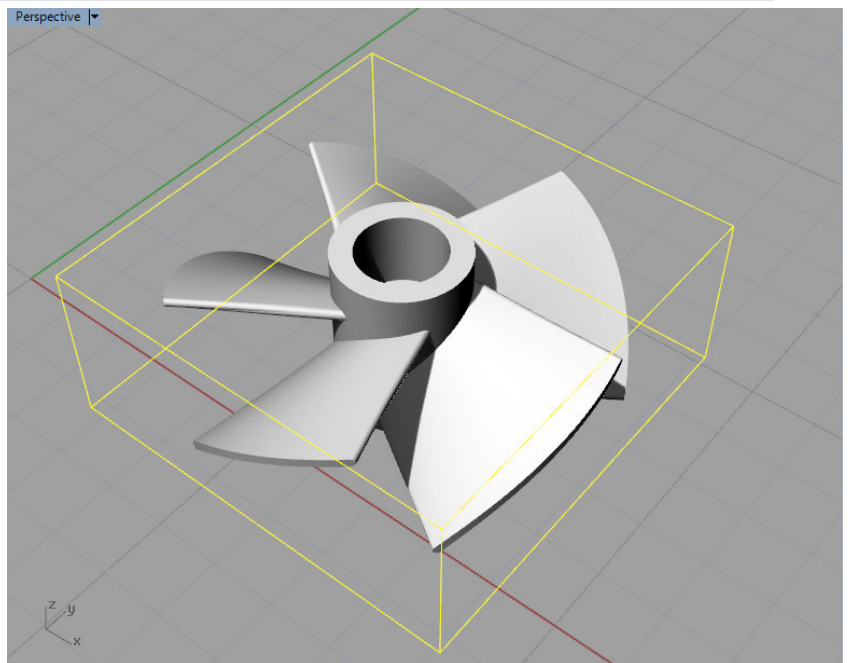
Create A Tool Path:

In order for MadCAM to correctly generate a tool path, the process must be followed precisely. If a mistake is made, it is best to start over and reload the geometry.

Step 1: Load the geometry into MadCAM. Click on the surfaces icon, and select the entire object to load. Follow the prompt box when loading the objects.



The loaded object will have a bounding box around it. If no bounding box appears, reload it in MadCAM.



Step 2: Create and load a cutting tool. Click the **Create Cutter** button to create a new cutter and or load an existing cutter.



Step 3: 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.

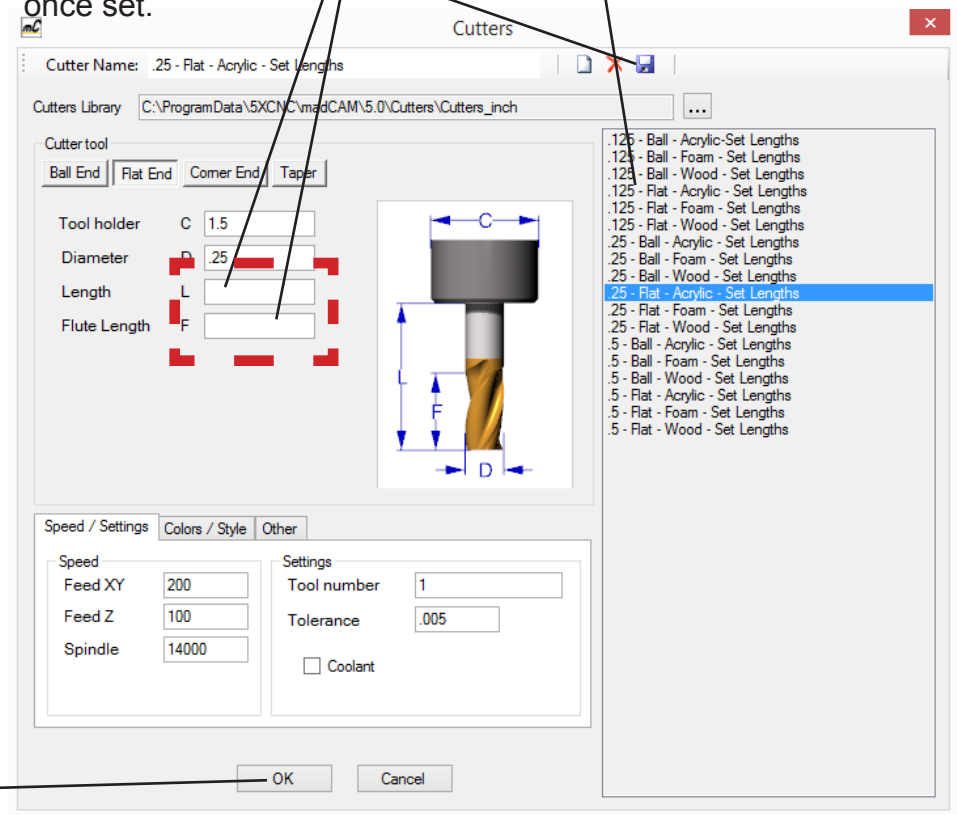
Coolant: Leave unchecked.

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

Choose a pre-defined cutter from the list of cutters.

Input the exact length and flute length - measure the bit when it is in a collet.

Click Save to set the tool lengths once set.



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. For your tools, it should not be set to **more than 280** inches/min. This speed can be adjusted at the machine

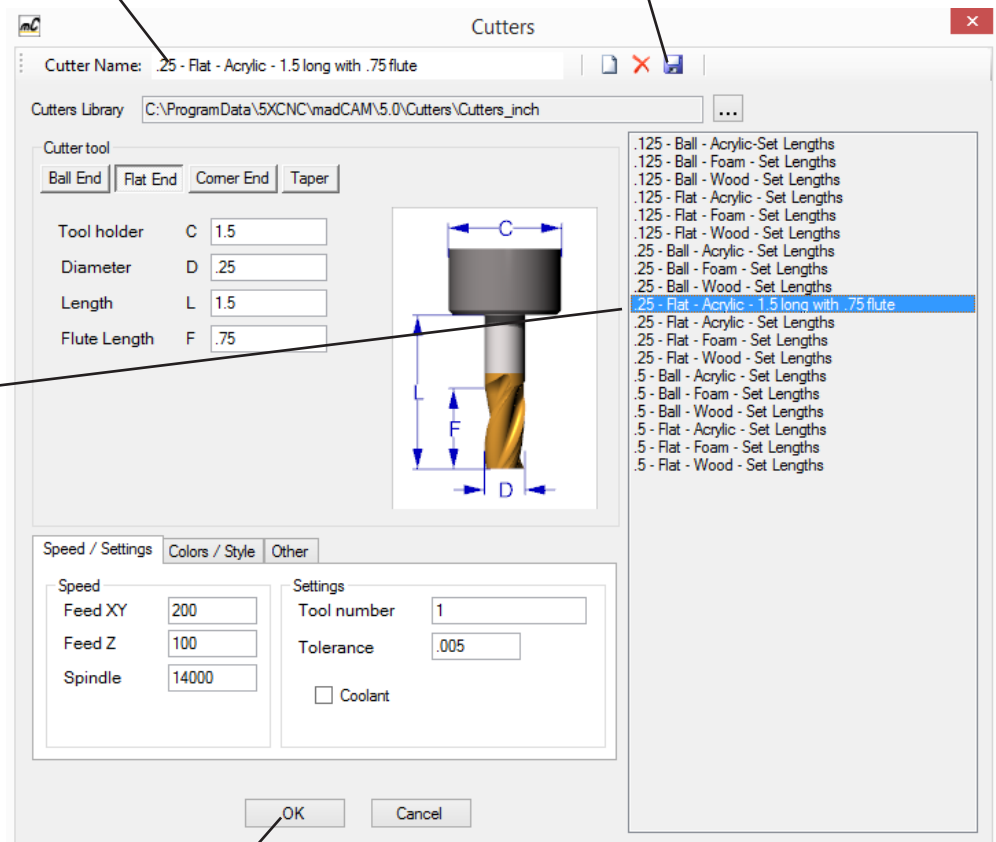
Feed Z - Set this to a **max of 100** inches/min as this can be hard on the bit

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:

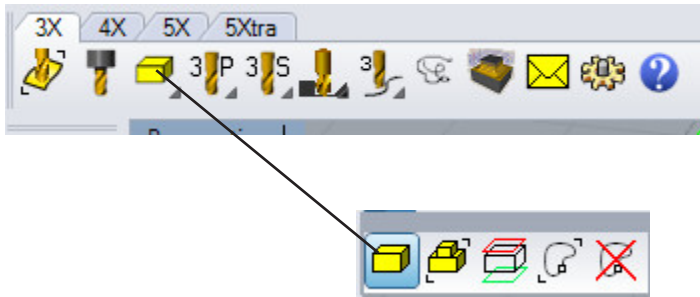
Click the "Save" icon to save the tool to the library

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



Click OK to make that tool active

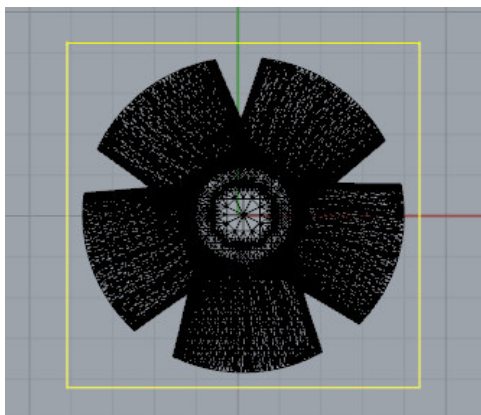
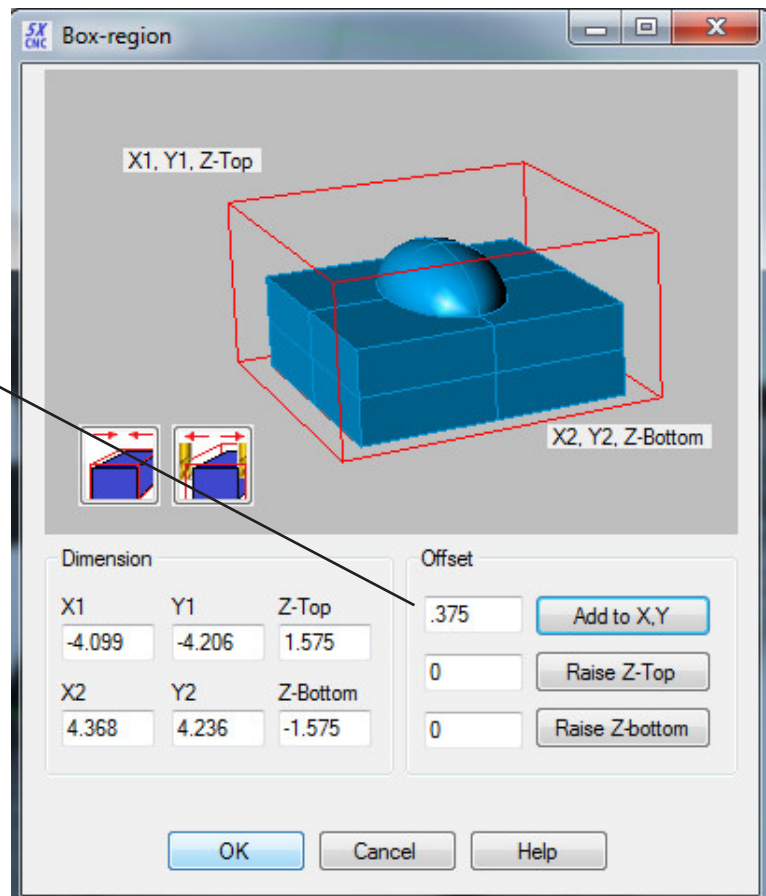
Step 4: Set the properties of the bounding box. The bounding box represents the “Virtual Stock Piece” that simulates the material that is cut and removed by the mill. Click on the **Regions** icon which pops up an additional window to select the **Create Box** button.



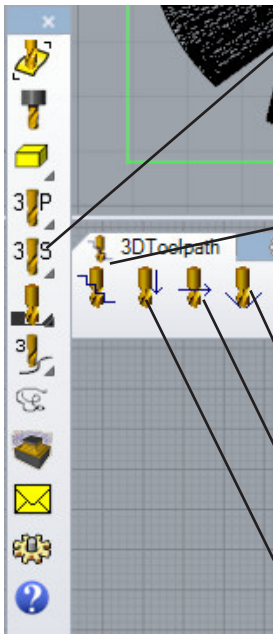
If for some reason you need to create a “wall” of unmilled material around your model, such as when flip-milling, you can increase the bounding box size in the XYZ axes. For example, to make the bounding box larger by 0.5in in XY, enter **0.375** in the X,Y Offset box, then click the **Add to X,Y** button.

It is usually not necessary to increase the size of the bounding box, and you should be aware that doing so can cause collisions between the spindle and the wall of your material that’s created by doing so.

Click **OK** when done.



Object within specified bounding box.



Step 5: Set up a toolpath. Click the **Create 3D Toolpath** button. The toolpath window will appear.

Four types of toolpaths are available:

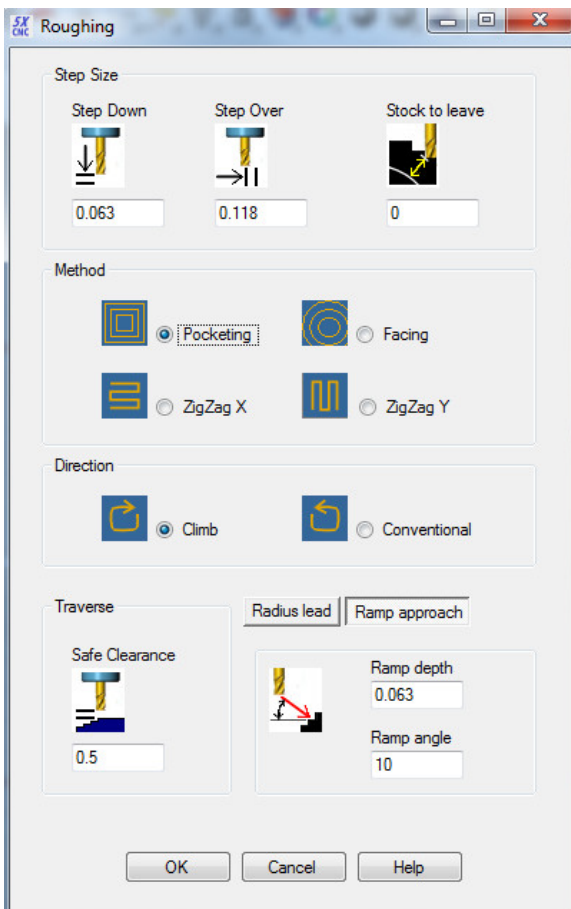
Roughing: This is a rough cut, where extra material is removed from the work piece to prepare it for the finishing cut. A rough cut should always be cut first if cutting wood or material.

Pencil Tracing: Dual contact cutting. Used as a compliment to clean up the overlap between Z-level finishing and planar finishing.

Planar finishing: A finishing cut. The bit follows the surface of the part. It should be used AFTER performing a Z Rough cut.

Z-level finishing: Cuts the material with a constant cutting depth (Typically is not used with our mill setup).

Step 6: Roughing Toolpath: Click the roughing toolpath, the following window will appear:



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

Foam: Max StepDown = length of cutting edge

Wood: Max StepDown = 1/3 diameter of bit

Metal: Max StepDown = 1/4 diameter of bit

StepOver: How much the bit will shift over after each cutting pass. Rule of thumb: StepOver = 2/3 bit diameter for software materials, 1.3 bit diameter for harder materials and faster feed rates

0.25" Dia bit = 0.08" StepOver

Ball ends require even smaller stepover to account for the smaller point of contact, which is often 0.06" or less

Stock To Leave: How much material is left on each cut.

Set to 0 by default

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

Select Pocketing for most efficient cutting.

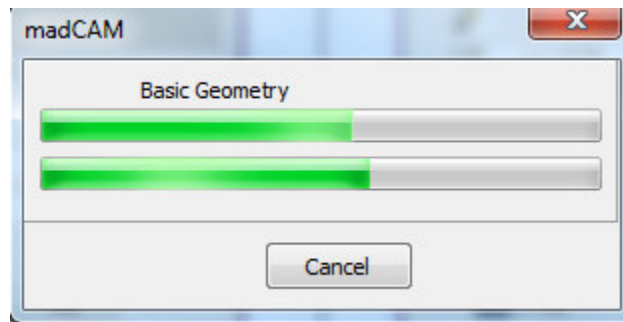
Safe Clearance Distance: The height the bit will raise to clear the material.

Set this to 0.5

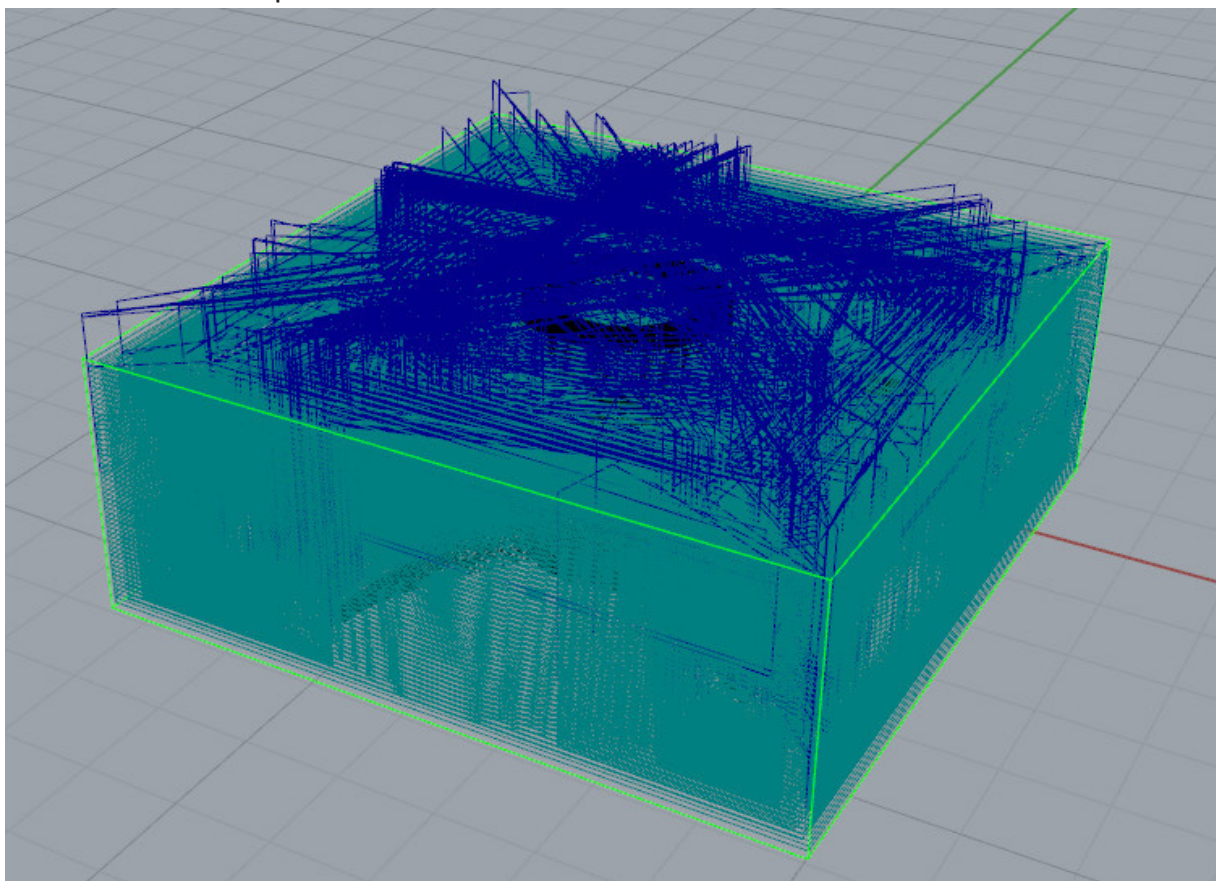
Choose Ramp Approach: Set to 10 deg by default.

Click **OK** to calculate the toolpath.

MadCAM will calculate the basic geometry and toolpath.

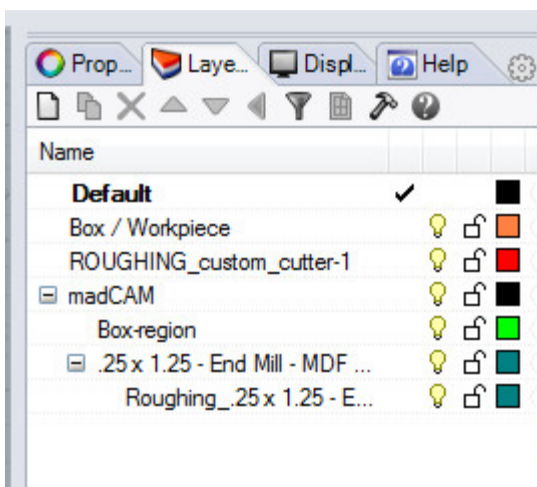


The calculated toolpath:



Blue Lines-Safe clearance moves

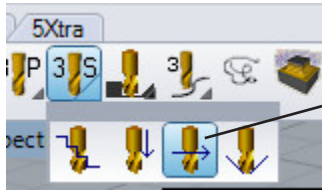
Green Lines-Tool Cutting lines



MadCAM stores toolpath data in Rhino's layers window, allowing you to create multiple toolpaths and post them all at once (provided you keep the same sized bit).

Step 7: Finish Cuts using Planar, if needed.

Planar cuts are used to refine and smooth out your surfaces. Often you create a rough cut first, to remove large amounts of material. Planar cuts are used to refine and bring out the details in your surfaces. It is typical to have multiple planar cuts.



Select the **Planar Finishing** toolpath.

This has to be set to Zero.

If not, the planer cut will not cut along steep angles.

Planar Finishing Toolpath Setup:

StepOver: How much the bit will shift over after each cutting pass. Rule of thumb: StepOver = 2/3 bit diameter for softer materials, 1/3 bit diameter for harder materials or faster feed rates.

1/4" Dia Bit = .188" StepOver

Angle Limit: Set to 0.

Stock To Leave: How much material is left on each cut.

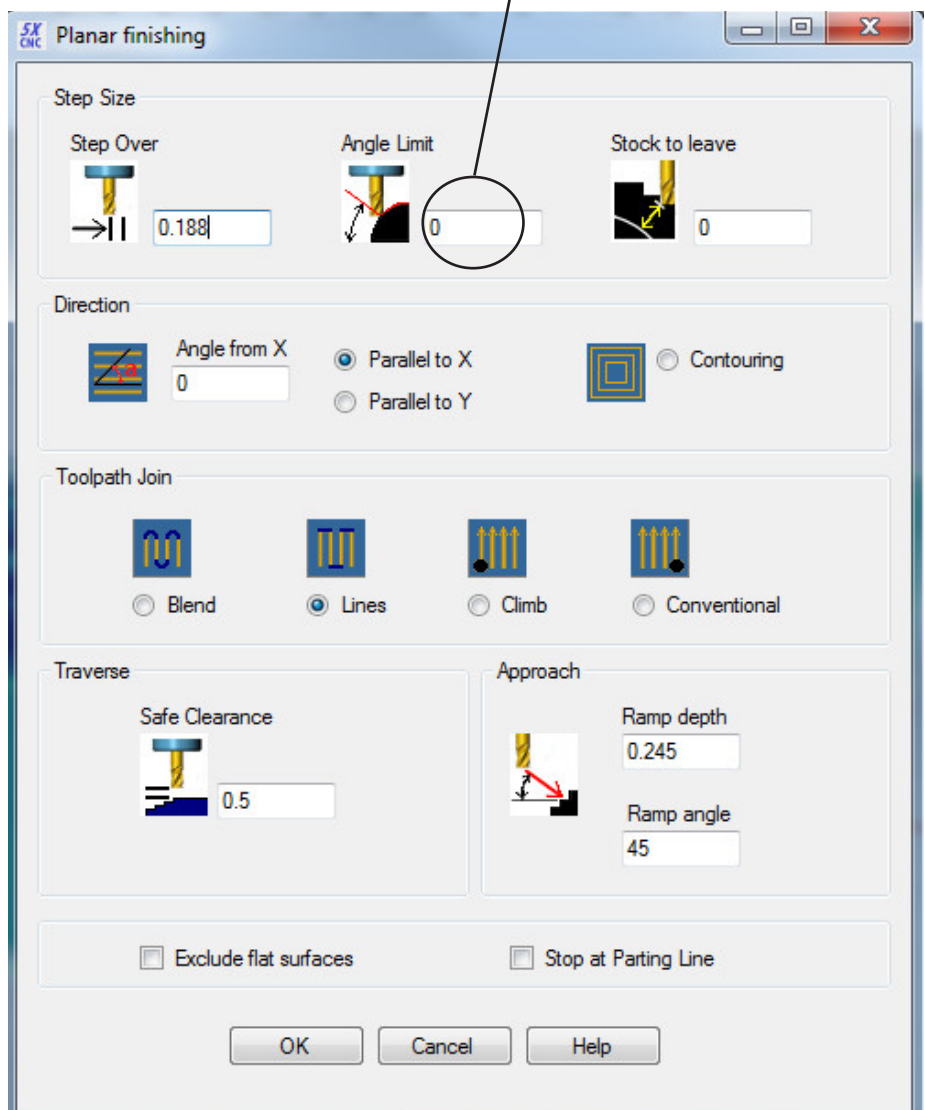
Set to 0 by default.

Direction: The direction of the passes the bit will make as it cuts. You can select either X or Y axis direction. Toolpath is set to Lines by default.

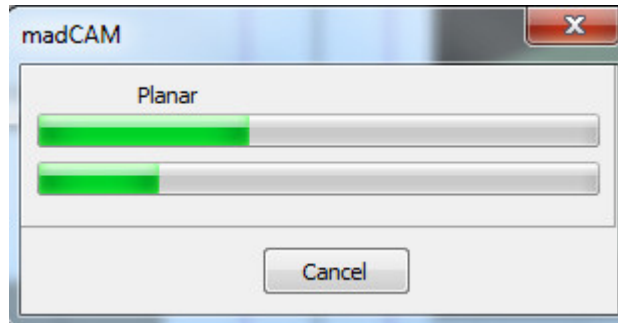
Safe Clearance: The height the bit will raise to when moving diagonally. Set to 0.5.

Ramp Angle: Set to 45.

Click **OK** to calculate the toolpath.

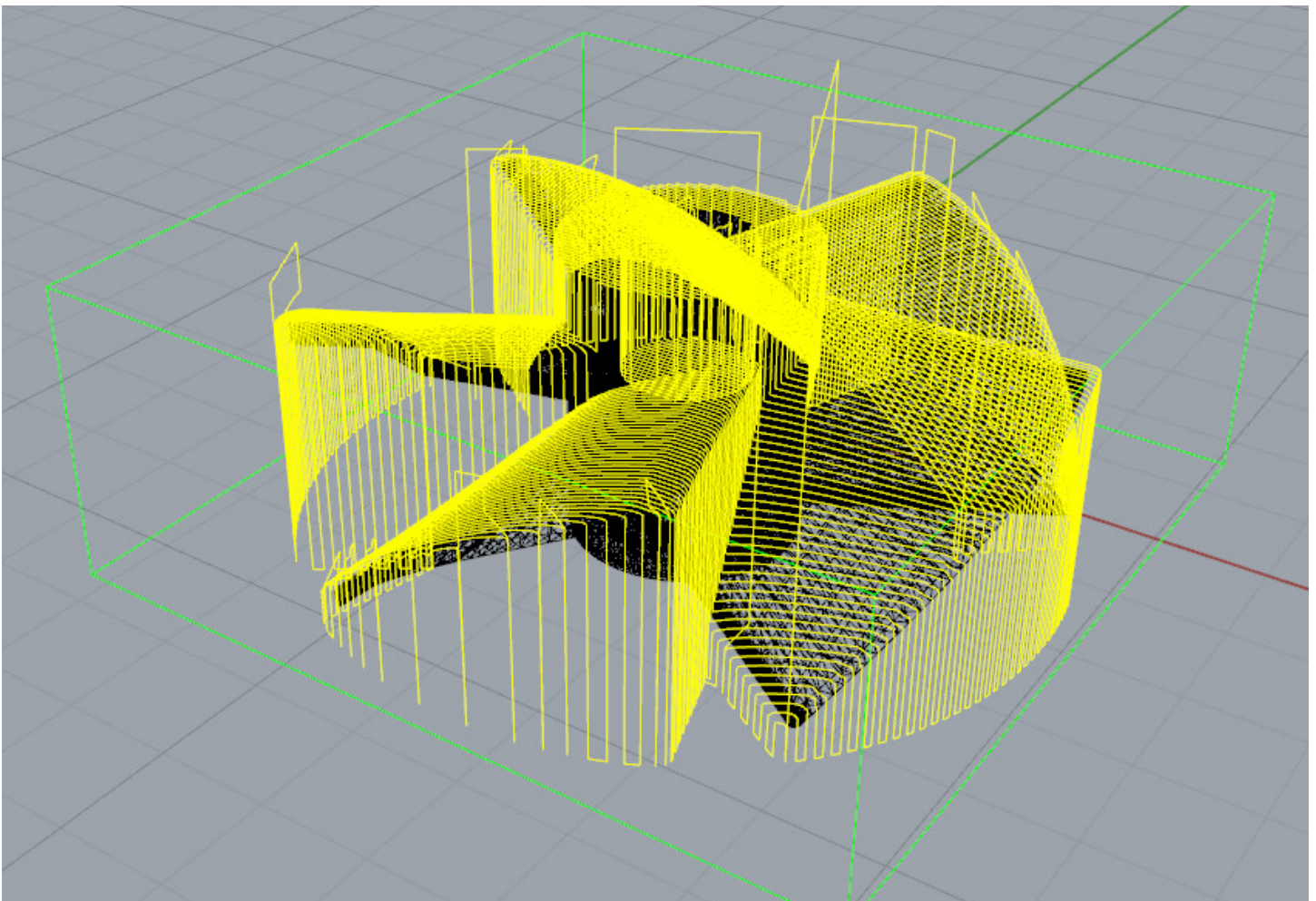


Planar toolpath calculation.

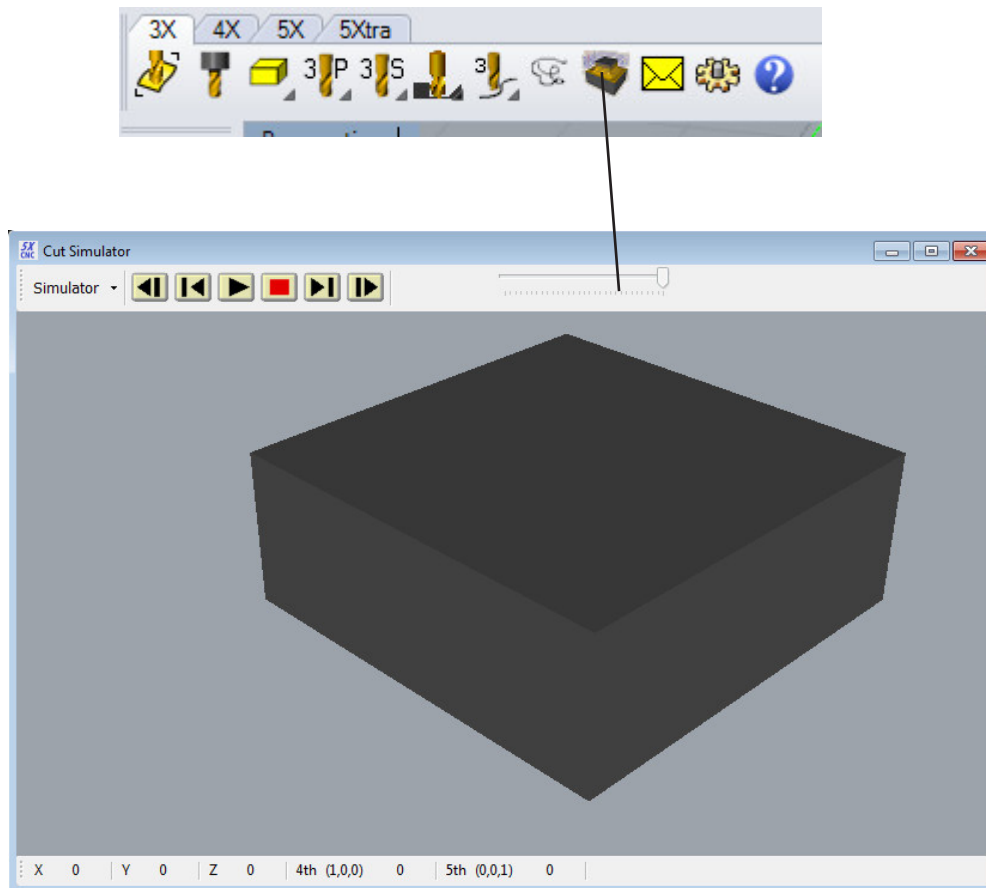


Once the toolpaths are calculated, make all layers visible and organize them in the correct order. For example: Rough cut first then finish cuts. Post the toolpaths to the mill in the same post-process file.

Note: Posting more than one toolpath requires the same bit to be used for all cuts. If you change to a different bit, the toolpaths will have to be posted separately, allowing for each job to stop when complete in order to change the bits.



Step 8: 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.

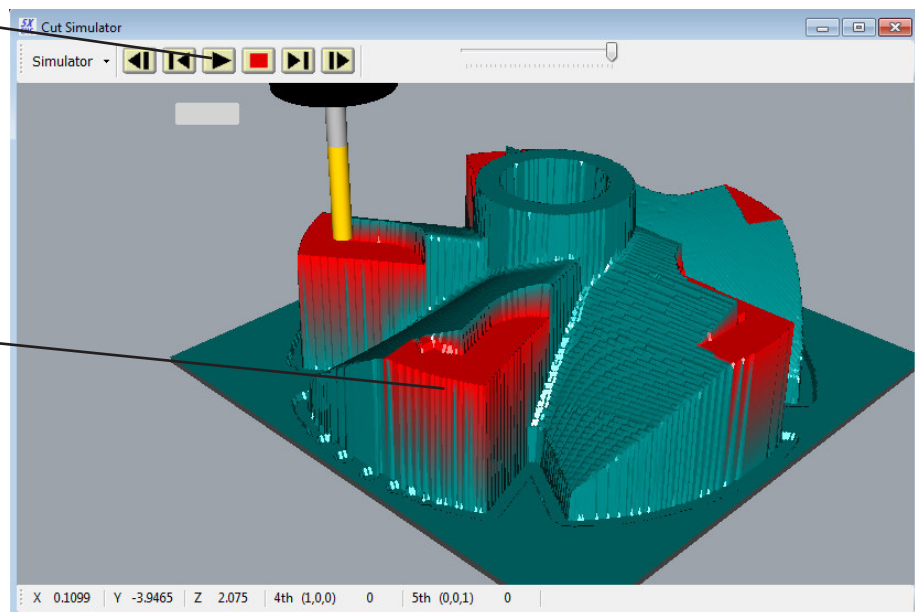
Press **Play** to start simulation.

What you see here is what you will get! If you do not like what you see, then you will need to add additional toolpaths.

If you see Red in the simulation, it means the depth of the cut is LONGER than the tool you have loaded.

You need to address this issues. Either limit the depth of the cut or get a longer bit.

If you see RED...You need to address it in BEFORE you try to cut the part.

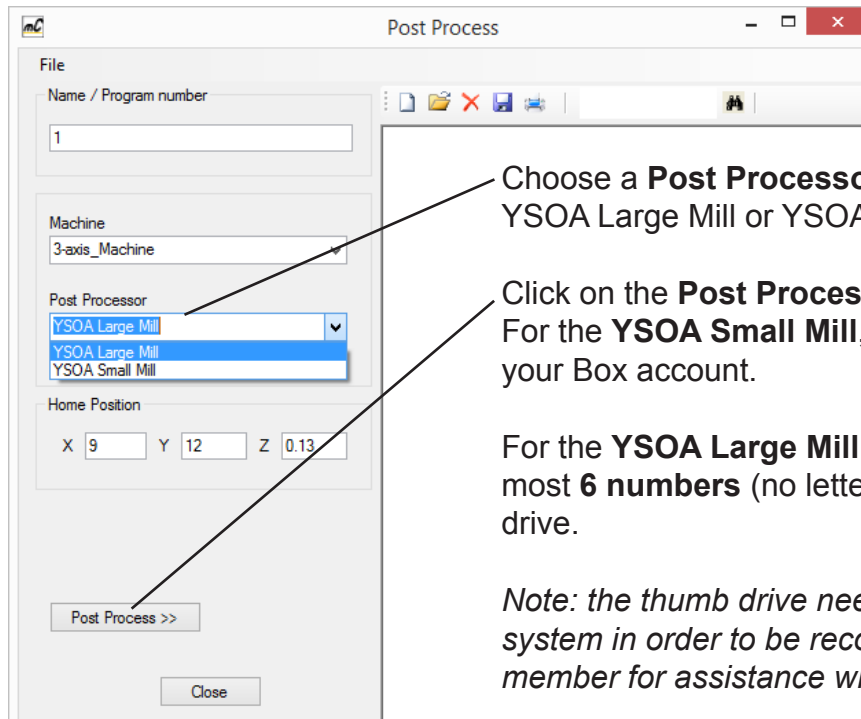


Step 9: Post the toolpath to the Yale Post Processors.



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

Step 10: 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.

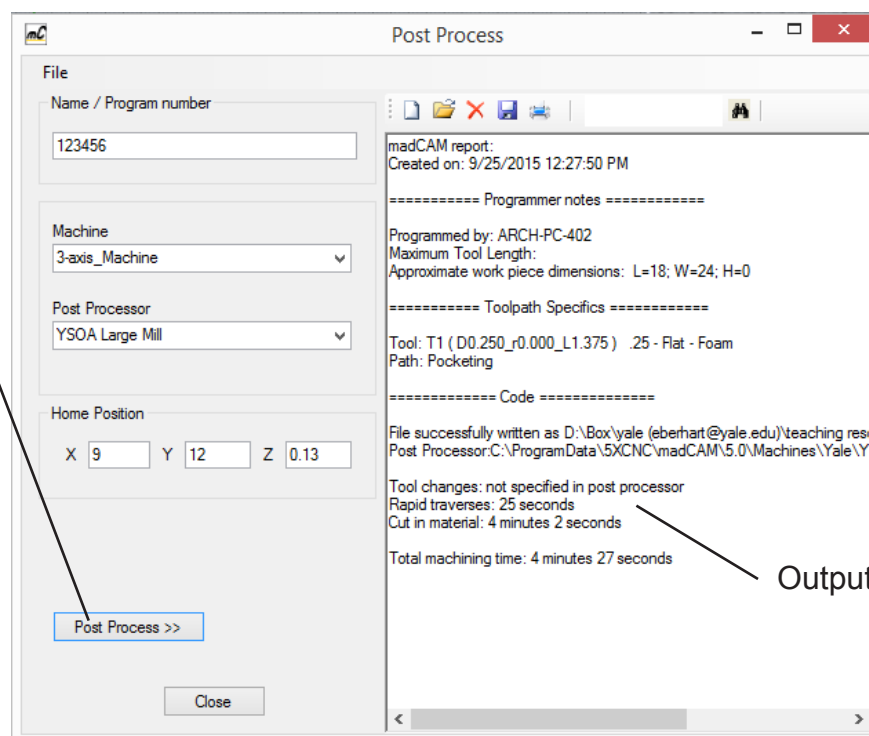
Click on the **Post Process** button:
For the **YSOA Small Mill**, you can give it any name and save it to your Box account.

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.

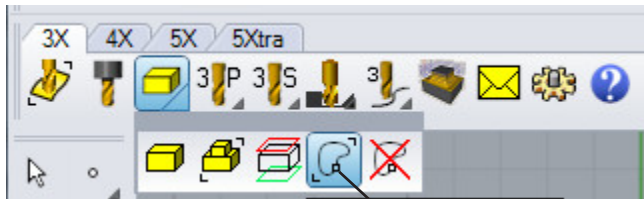


Outputted File

Additional Options: Setting a Defined Boundary Curve for Toolpath generation

You can refine the area to be milled using Boundary Curves. You can create a closed 2D curve that restricts the toolpaths to the inside of the curve. This allows you to use specific cuts on specific areas of the job. Greatly reducing the overall time to mill.

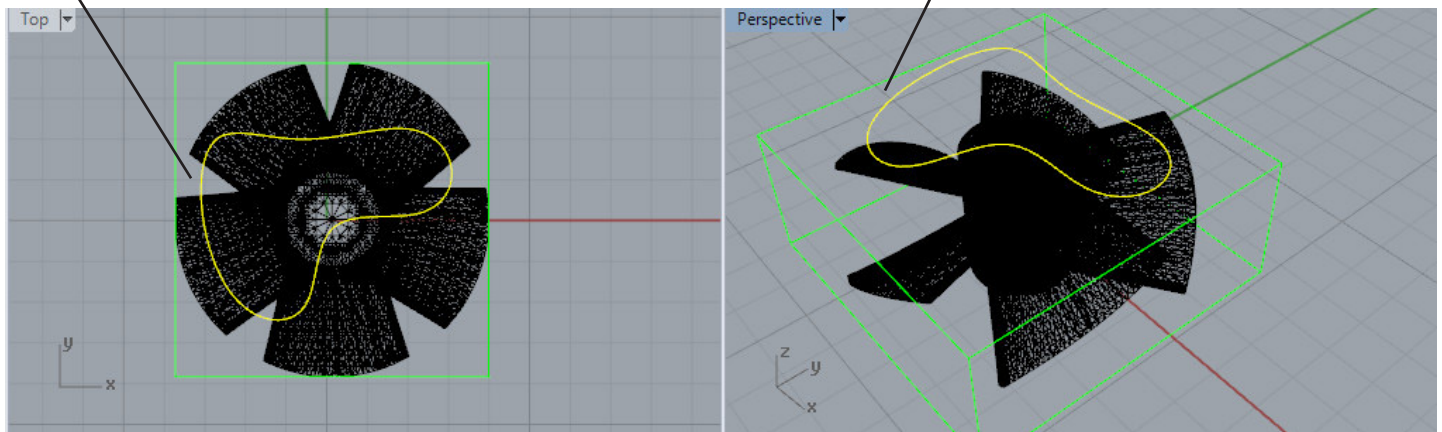
Step 1: Under the Create Box Icon, in the fly out tool bar choose “**Select Region Curves**”



Choose Select Region Curves

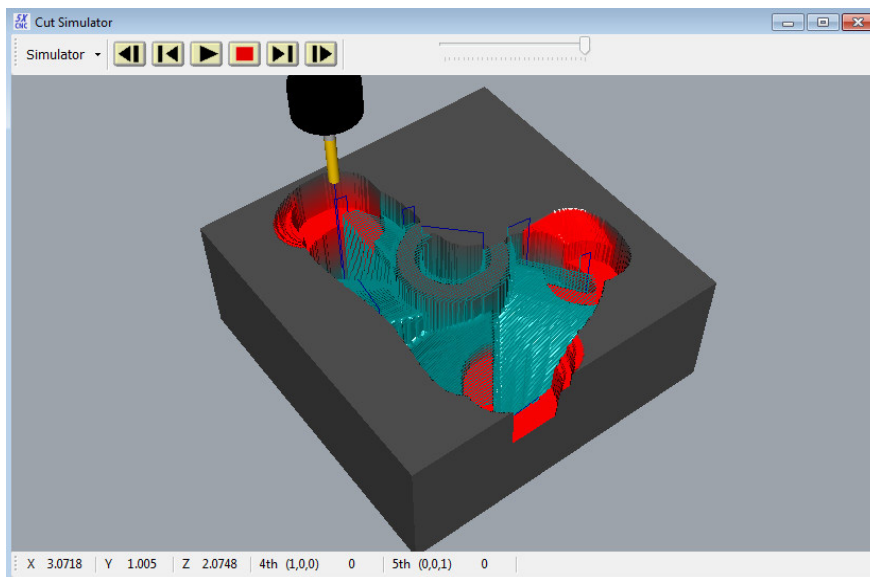
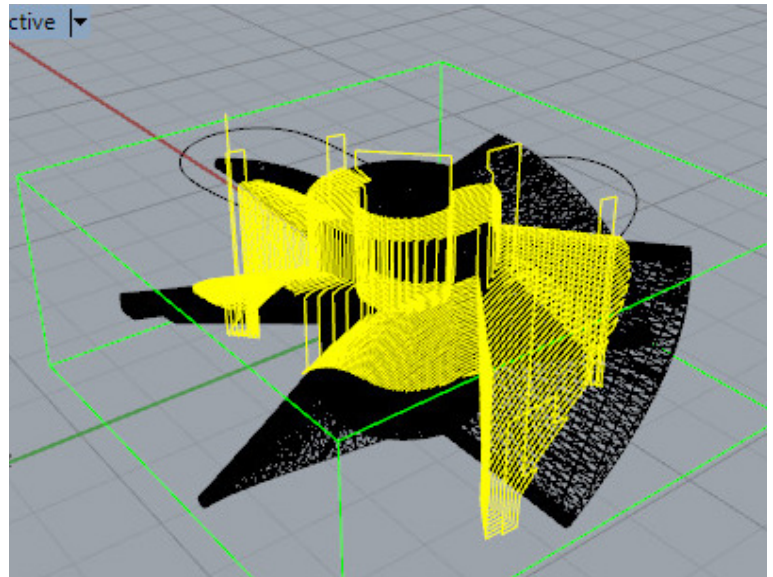
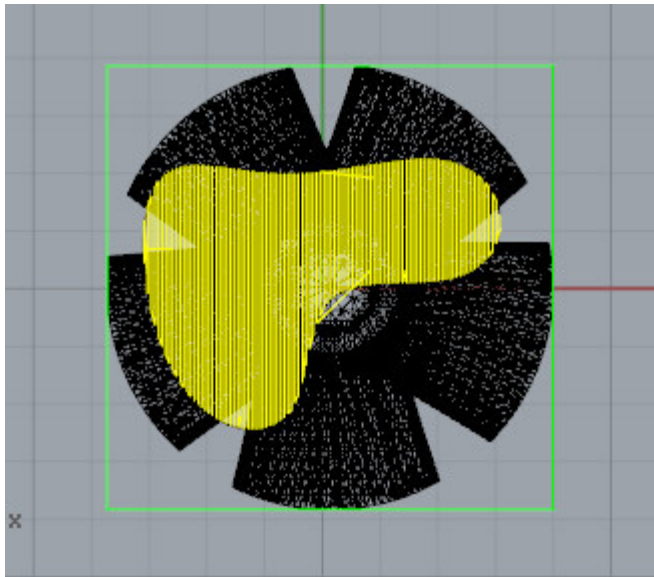
New Bounding box.

Selected Region Curve.



Step 2: Choose a finishing cut. Performing a finishing cut will only cut within the boundary curve you choose.

Example of a Planar Cut:



Step 3: Simulate cutting job.

Step 4: Post the toolpath to the Yale Post Processor.