MadCam 4.2: Large CNC Tool Path Generator

Digital Media Tutorial

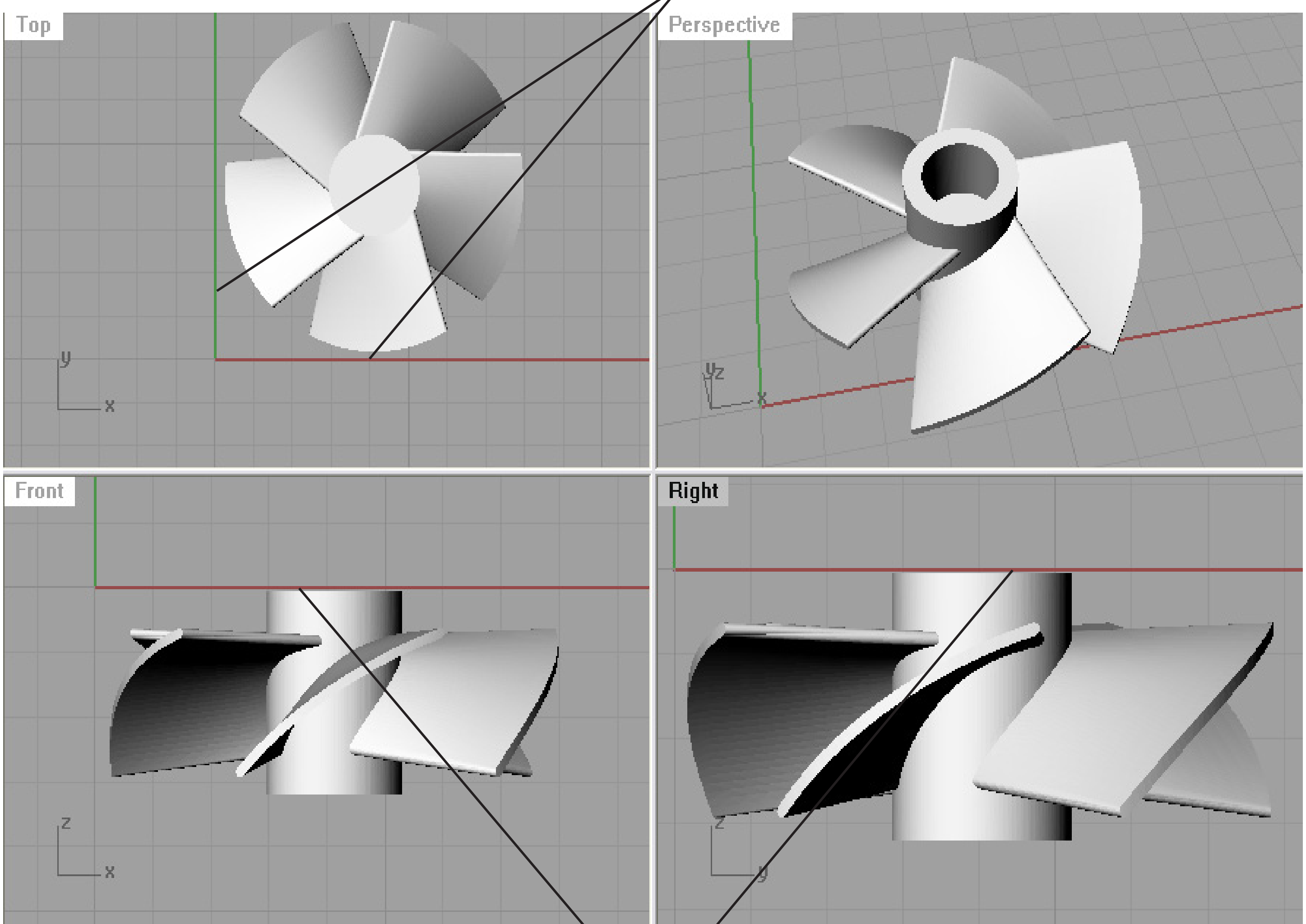
MadCAM is a tool path generator that works inside Rhino. It uses Rhino's interface to generate the tool paths that are exported as a text file to the large CNC mill.

Section 1: Axis Milling

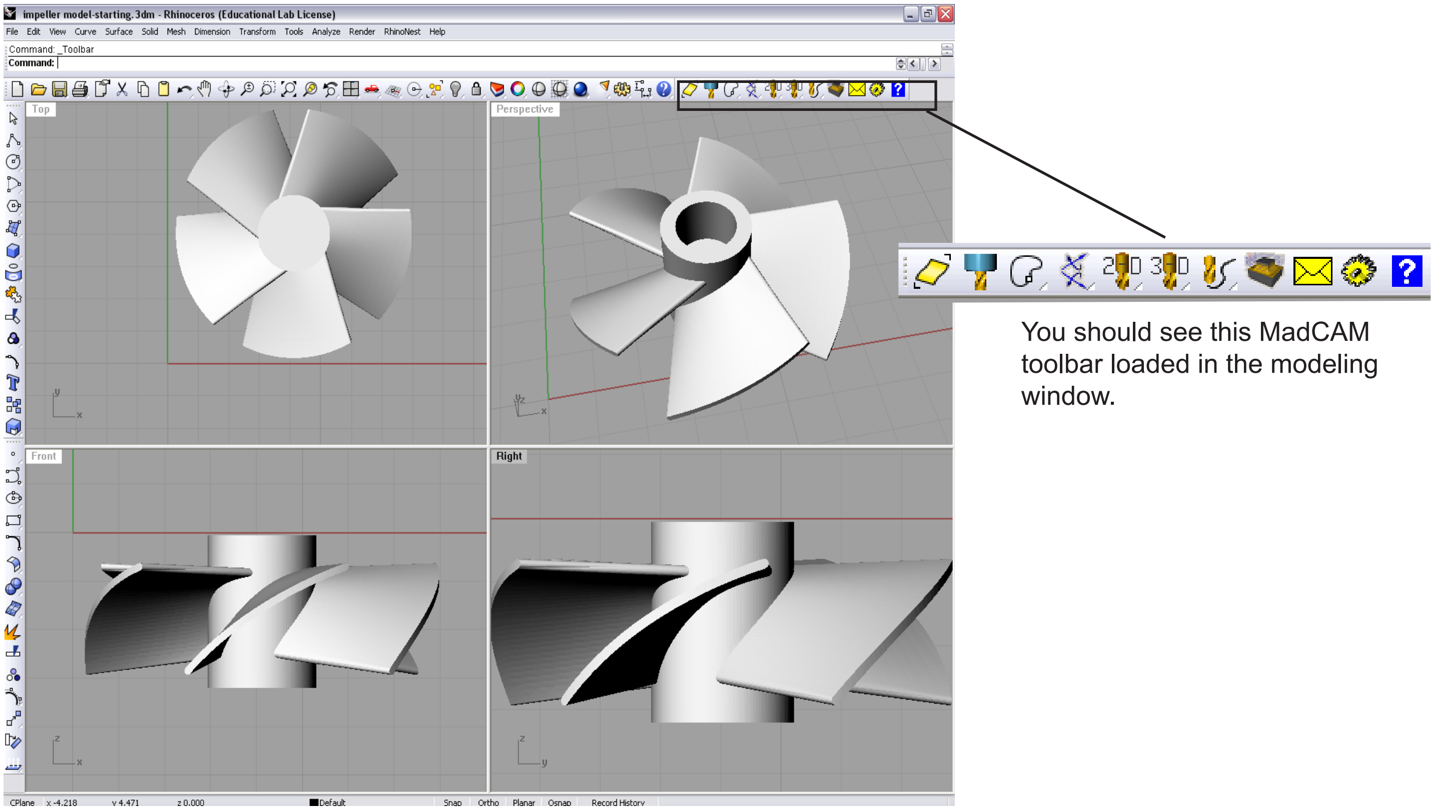
Step 1: Open your file in Rhino, MadCAM runs inside Rhino, and is automatically opened when Rhino is opened. If the MadCAM toolbar is not there see next page for instructions on loading the toolbar.

Step 2: Prepare Model. Correctly place your part in the modeling window. The 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.**

Model placed in the positive X and Y axis.
Note: It is advised to leave a little extra room on the sides.

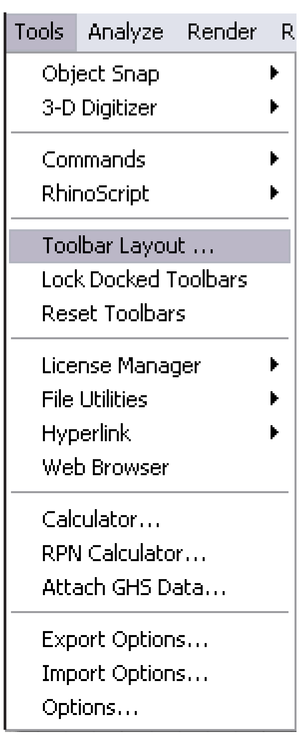


Model placed below X and Y ground plane.

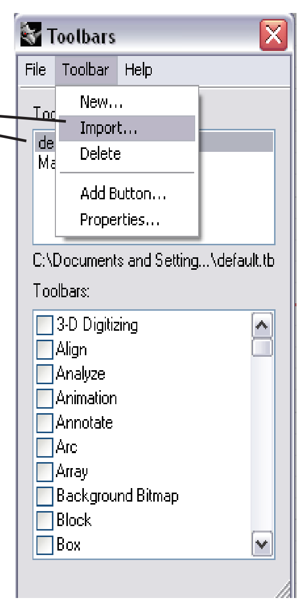


You should see this MadCAM toolbar loaded in the modeling window.

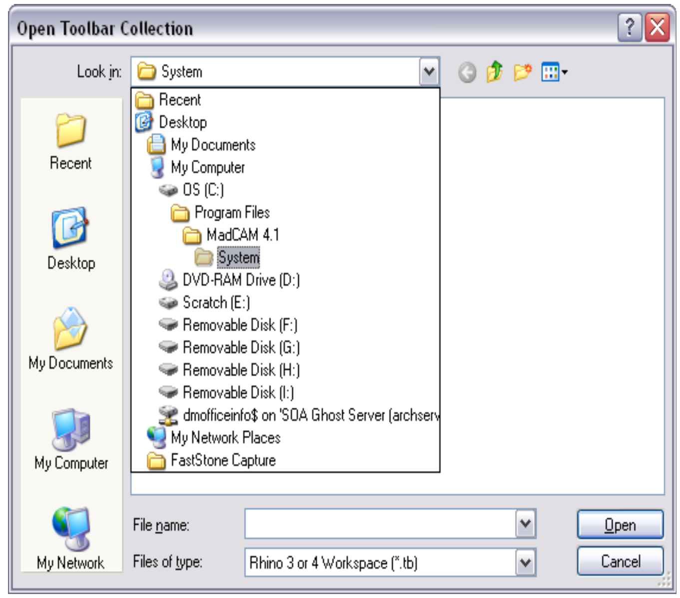
If the toolbar is not visible, it can be loaded by selecting **Tools>Toolbar Layout.**

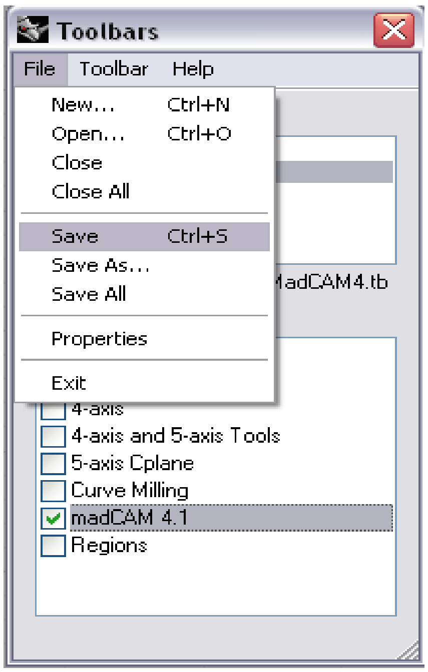


Highlight **Default** and choose **Import** from the Toolbar menu.



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 3D toolpath from nurbs models or meshes.

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

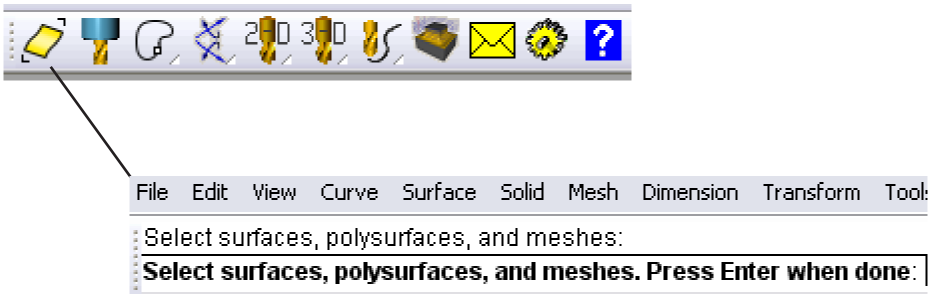
View the work piece shape after each toolpath operation.

Post the job to the CNC mill.

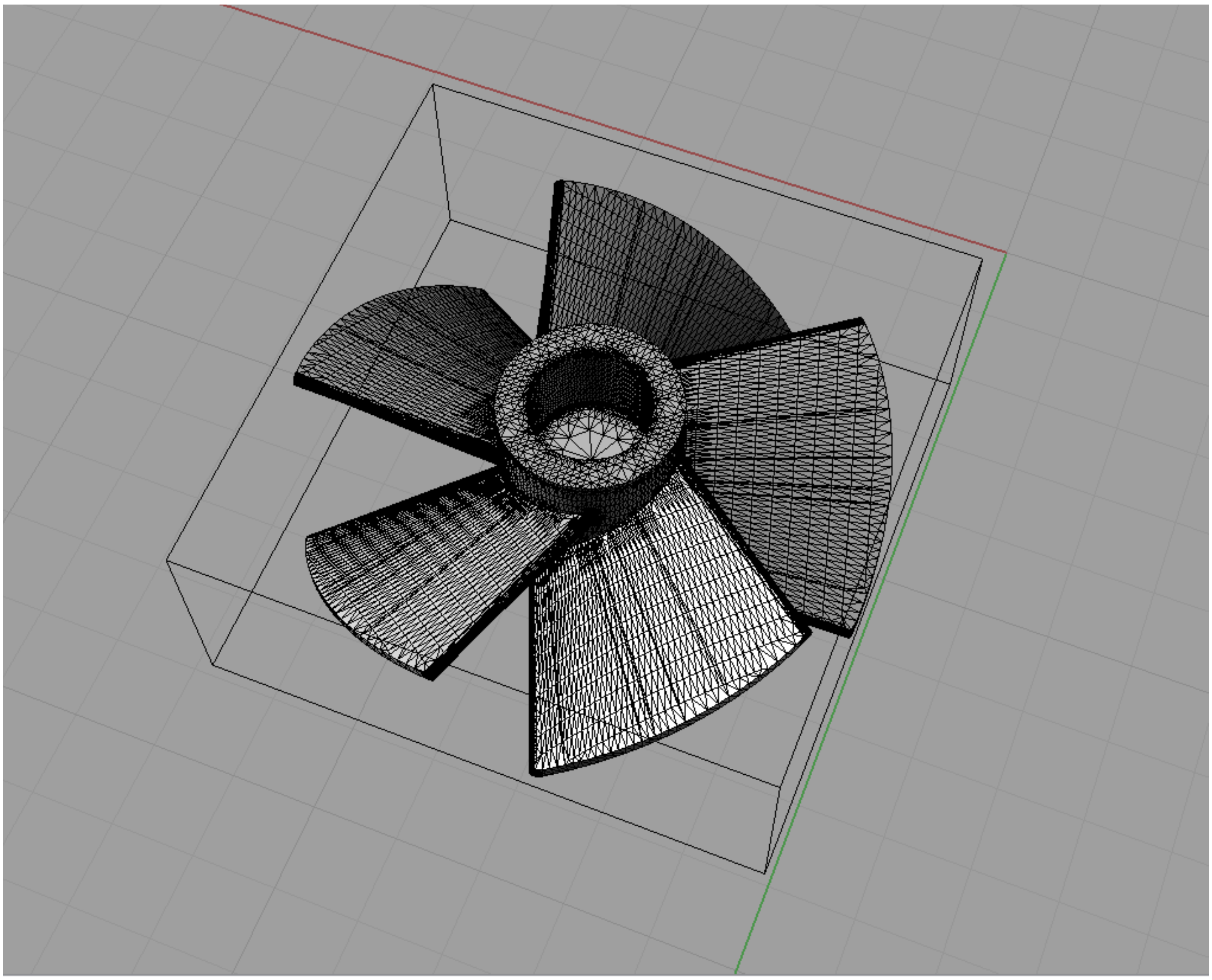
Create A Tool Path:

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.



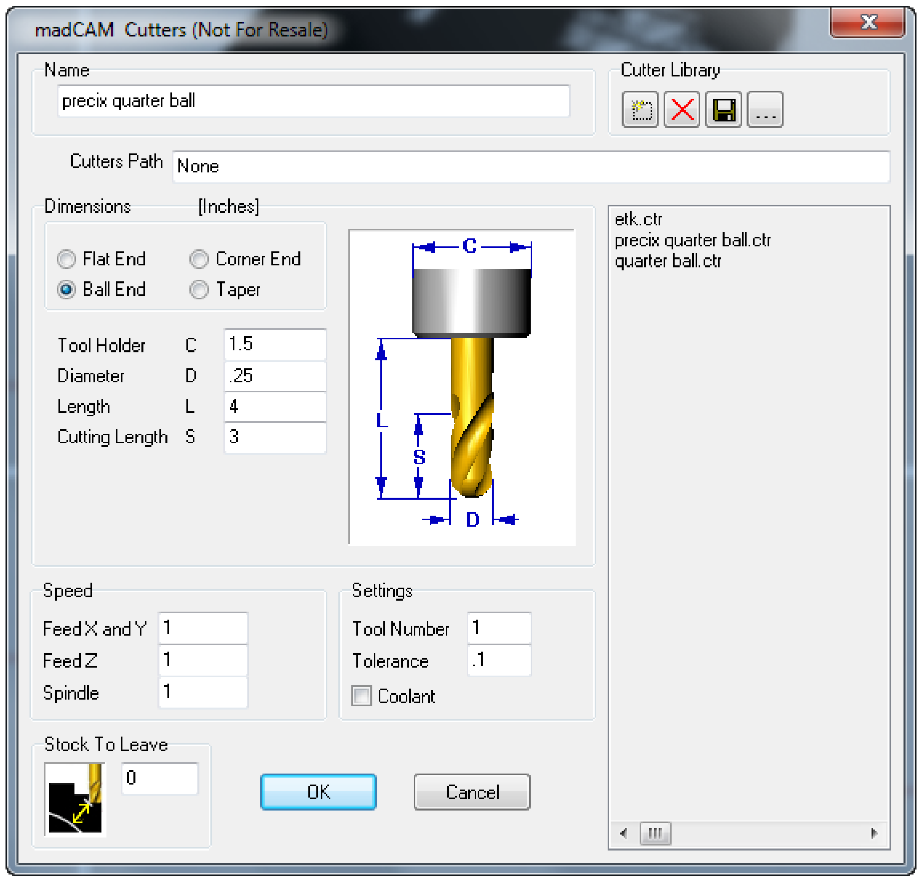
The loaded object will have a box around it. If there is not a box around the object, reload it in Mad-Cam.



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

Step 3: 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 if doing a finishing pass. Otherwise, set to same value as stepdown.

Tool Number: Set to any number.

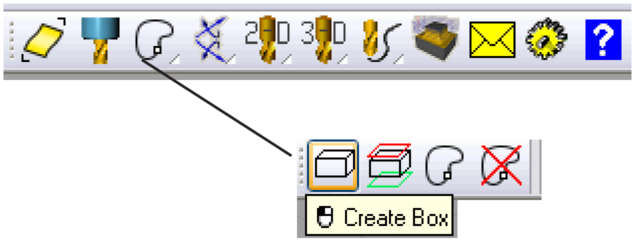
Tolerance: Use 0.1 - 0.01, depending on your material. The lower the tolerance, the longer your toolpath will take to process. Setting too low a tolerance can cause the program to run out of memory.

Coolant: Leave unchecked.

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

Click **OK** to select the cutter.

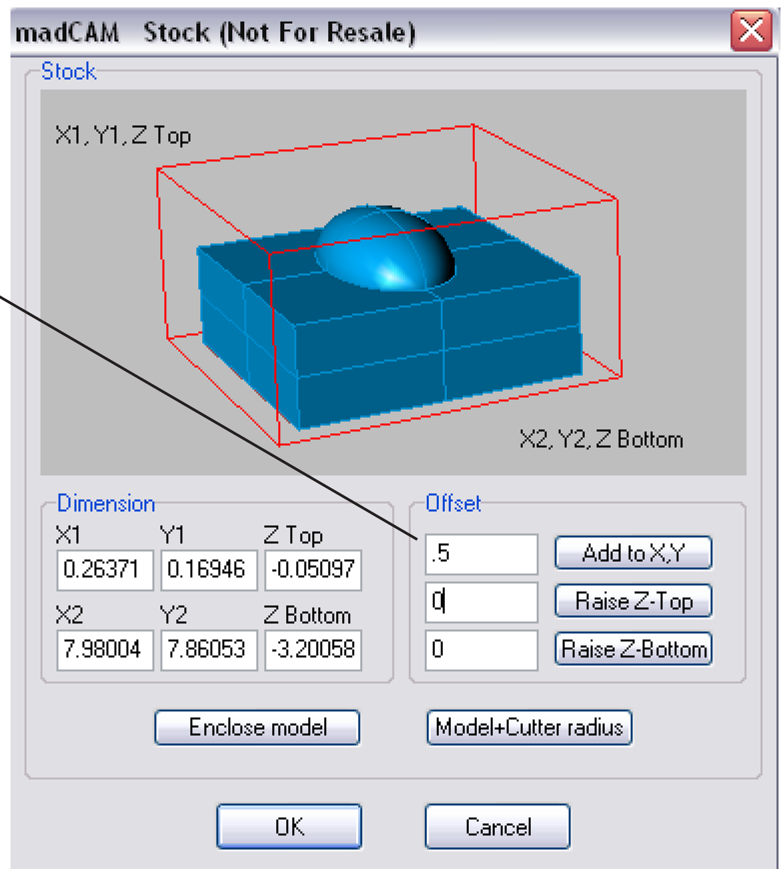
Step 4: Set the properties of the bounding box. The bouding box represents the “Virtual Stock Piece” that simulates the material that is cut and removed by the mill. Click on the **Regions** icon which pops up an additional window to select the **Create Box** button.

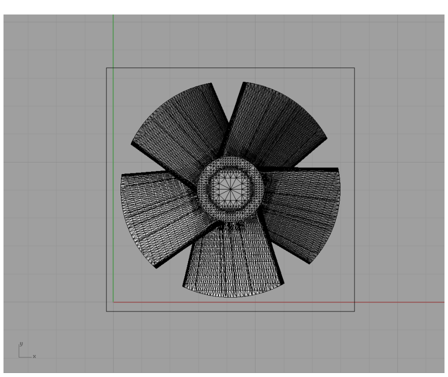


If for some reason you need to create a “wall” of unmilled material around your model, such as when flip-millling, you can increase the bounding box size in the XYZ axes. For example, to make the bounding box larger by 0.5in in XY, enter **0.5** in the X,Y Offset box, then click the **Add to X,Y** button.

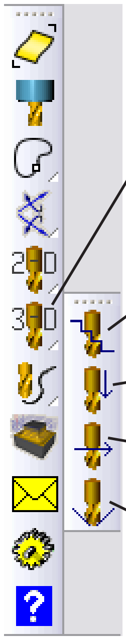
It is usually not necessary to increase the size of the bounding box, and you should be aware that doing so can cause collisions between the spindle and the wall of your material that’s created by doing so.

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:

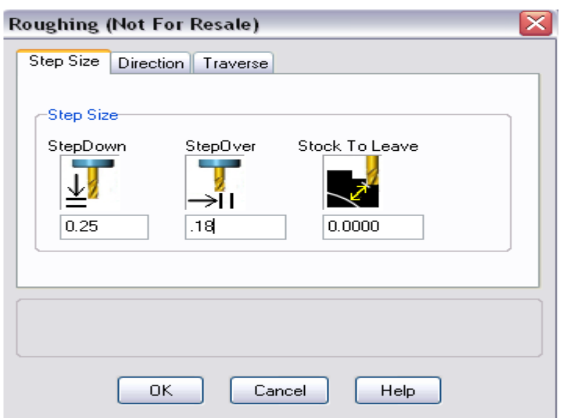
Roughing: This is a rough cut, where extra material is removed from the work piece to prepare it for the finishing cut. A rough cut should always be cut first if cutting wood or material.

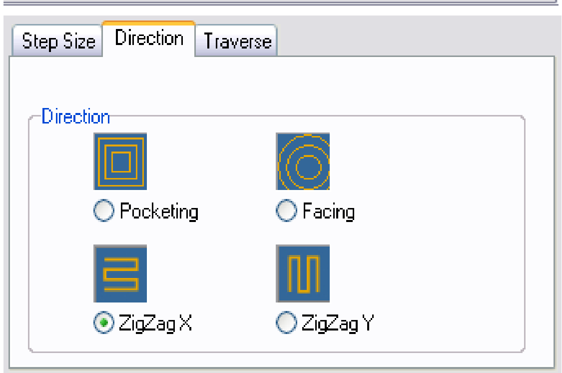
Z-level finishing: Cuts the material with a constant cutting depth (Typically is not used with our mill setup).

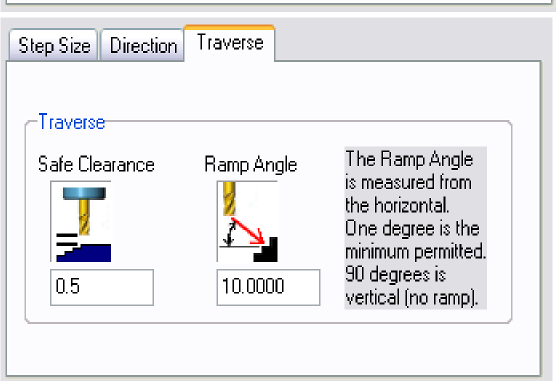
Planar finishing: A finishing cut. The bit follows the surface of the part. It should be used AFTER performing a Z Rough cut.

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

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 stepovers 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 ZigZagX

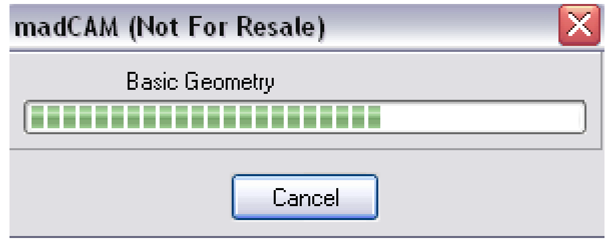
Safe Clearance Distance: The height the bit will raise to clear the material.

Set this to 0.25 - 0.5

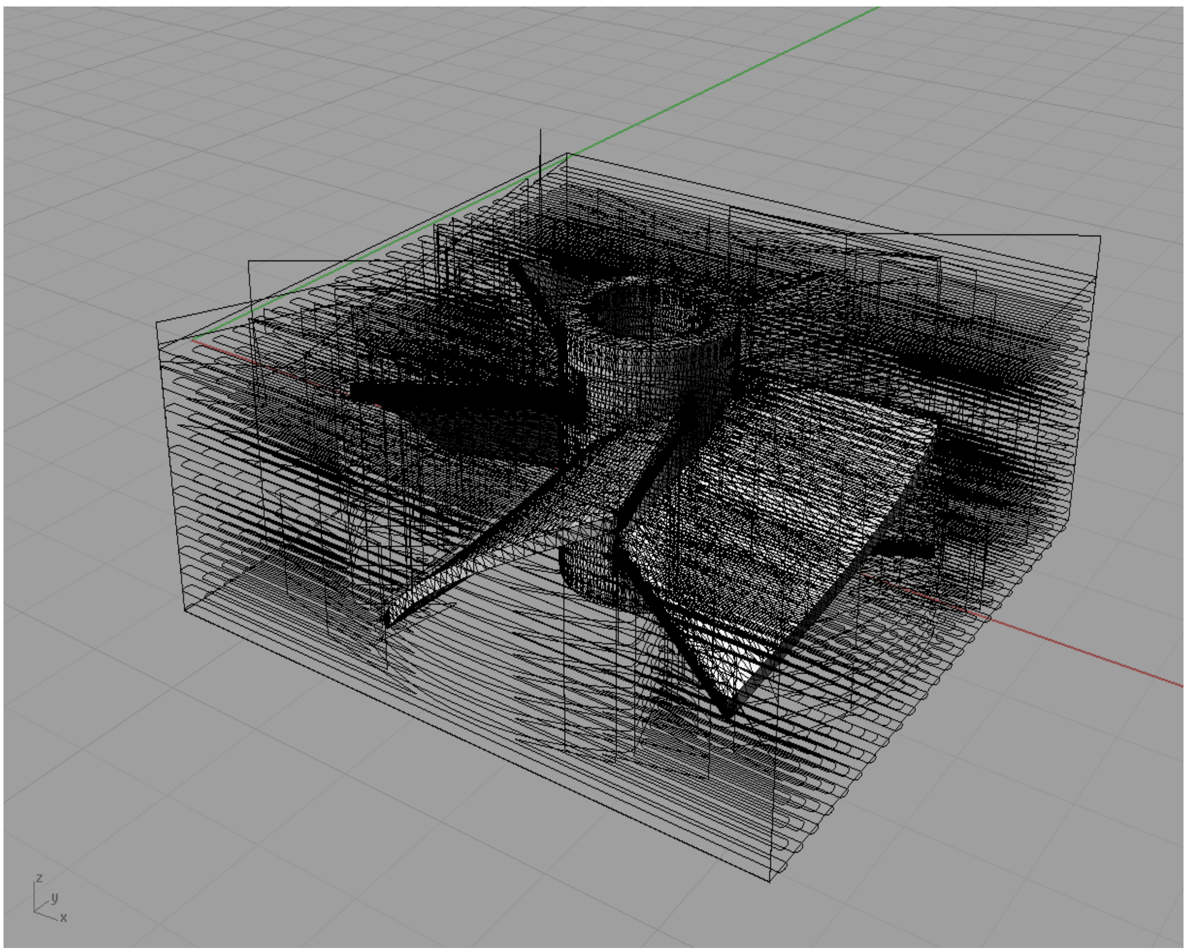
Ramp Angle: Set to 10 by default.

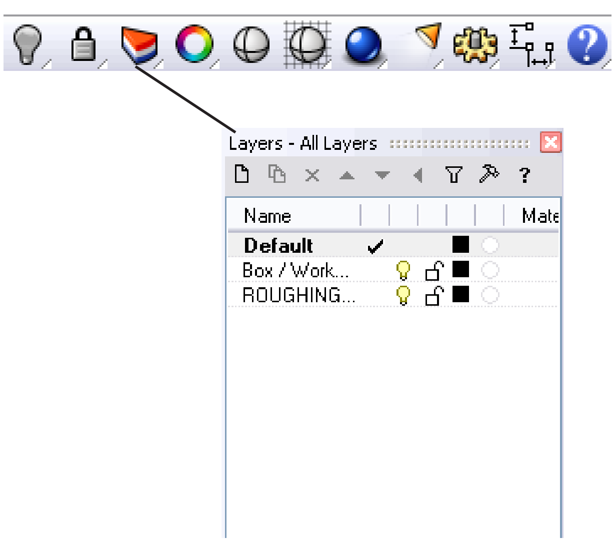
Click **OK** to calculate the toolpath.

MadCAM will calculate the basic geometry and toolpath.



The calculated toolpath:

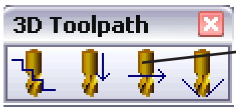




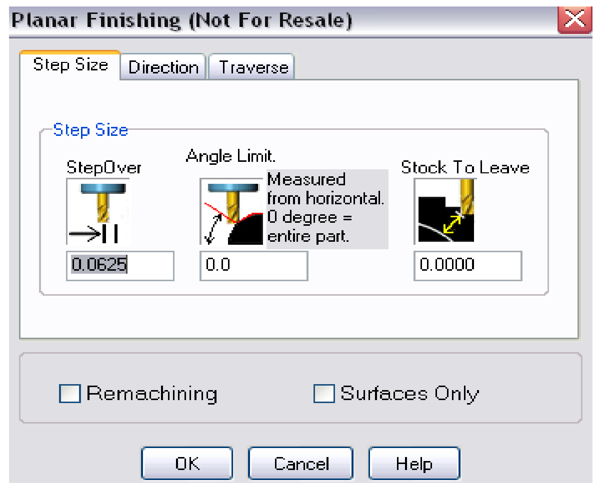
MadCam stores toolpath data in Rhino’s layers window, allowing you to create multiple toolpaths and post them all at once (provided you keep the same sized bit).

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.



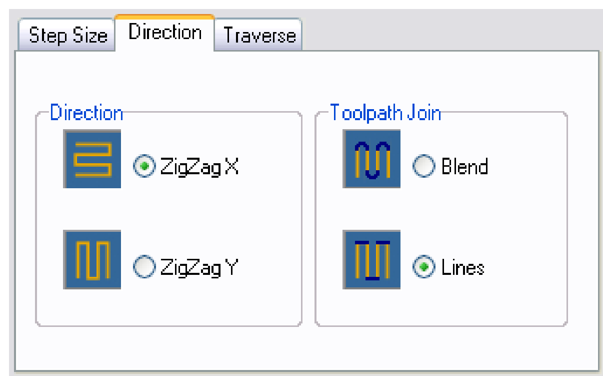
Select the **Planar Finishing** toolpath.



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 = .18" StepOver

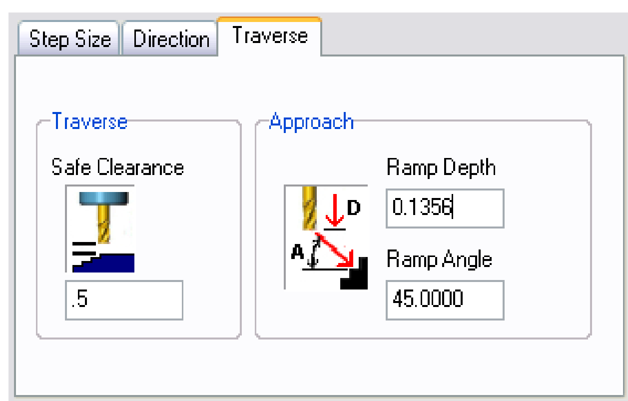


Angle Limit: Set to 0.

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

Set to 0 by default.

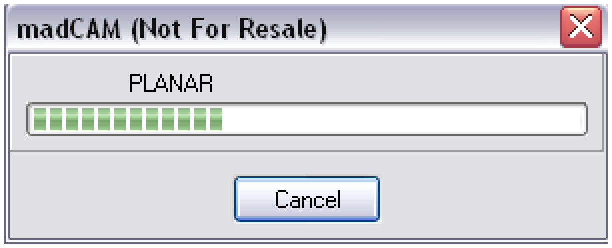
Direction: The direction of the passes the bit will make as it cuts. You can select either X or Y axis direction. Toolpath is set to Lines by default.



Safe Clearance: The height the bit will raise to when moving diagonally. Set to 0.25 - 0.5.

Ramp Angle: Set to 45.

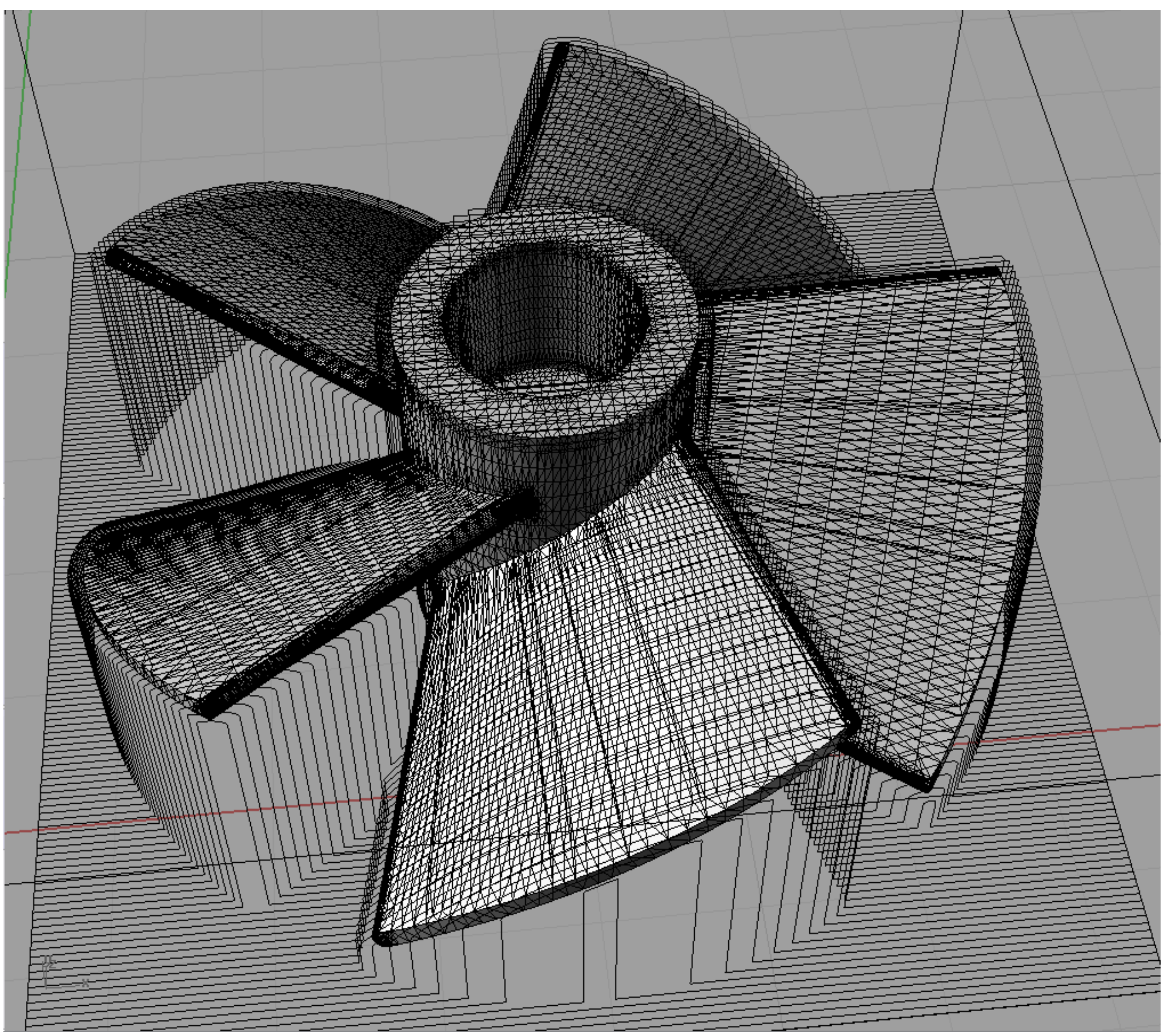
Click **OK** to calculate the toolpath.



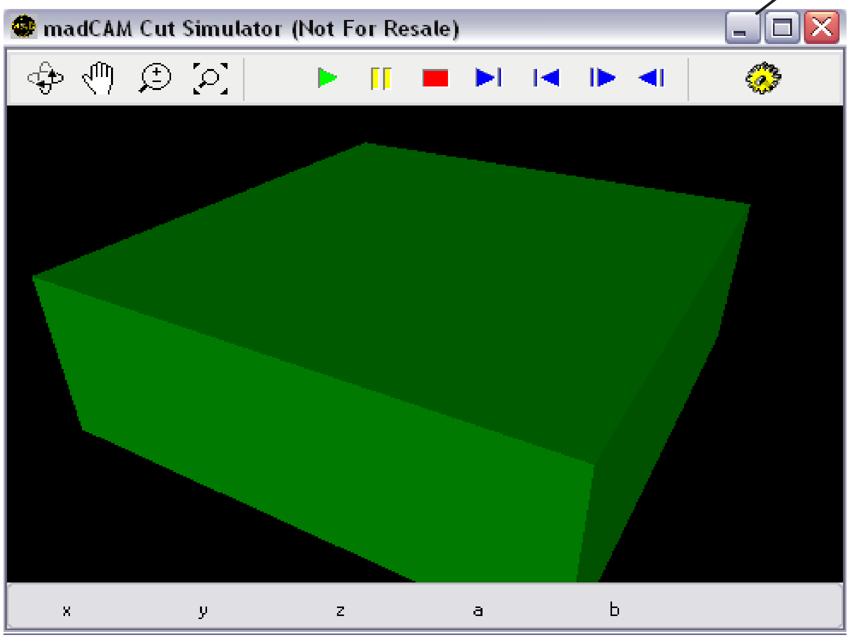
Planar toolpath calculation.

Once the toolpaths are claculated, 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 seperately, allowing for each job to stop when complete in order to change the bits.*

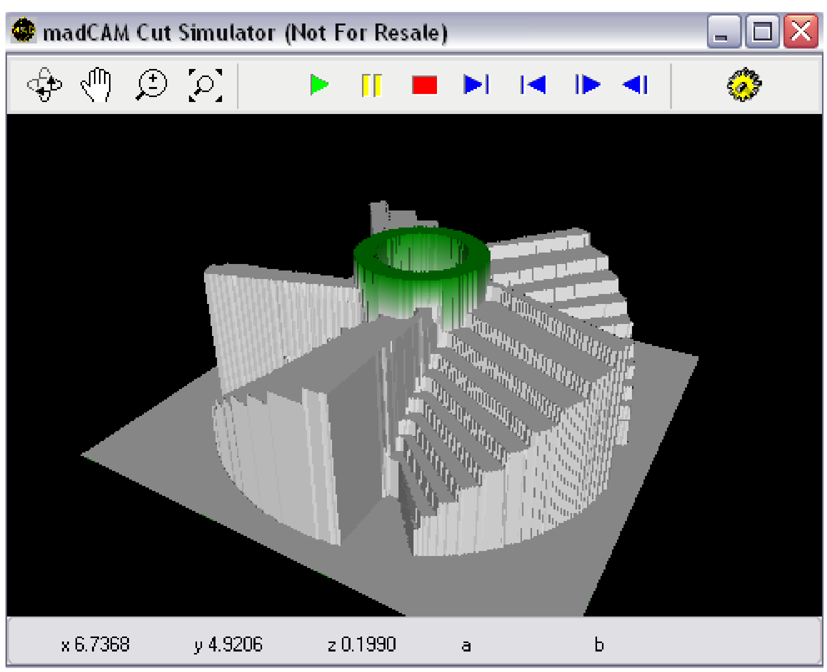


Step 8: Simulate Cutting Job. Click the **Simulate** button. The Cut Simulator Window will open.



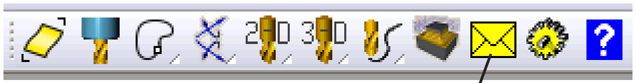
The buttons at the top of the window are used for controlling view, cut simulation and cut simulation settings.

Press **Play** to start simulation.



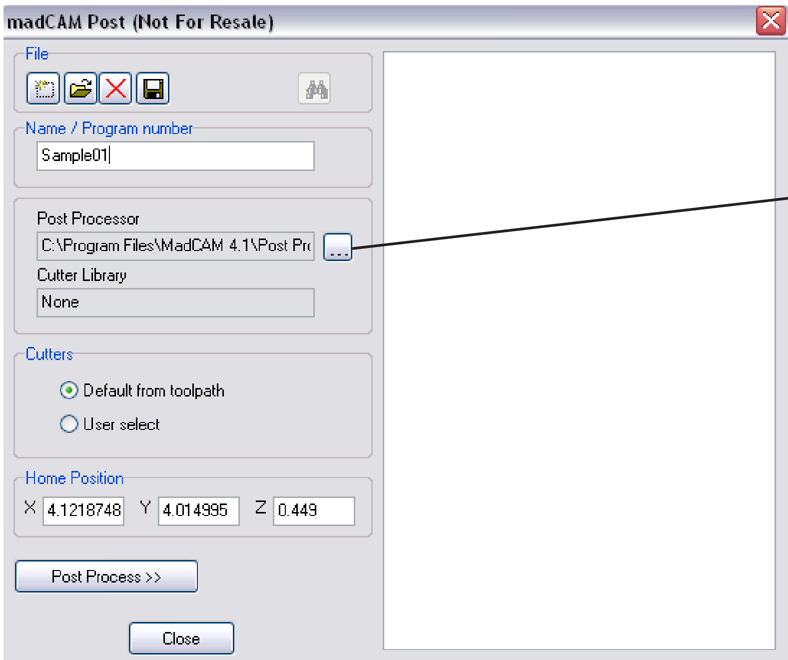
What you see here is what you will get! If you do not like what you see, then you will need to add additional toolpaths.

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



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

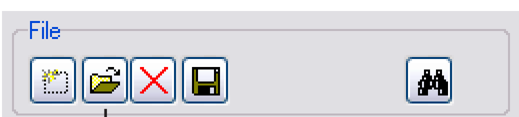
Step 10: Verify the post processor and cutter library settings are correct.



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

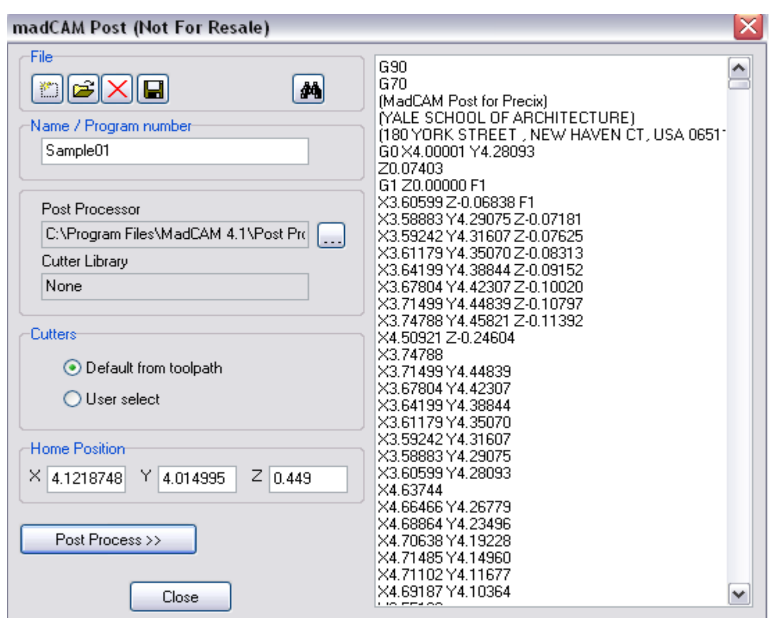
Default from toolpath should be selected for Cutters.

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.

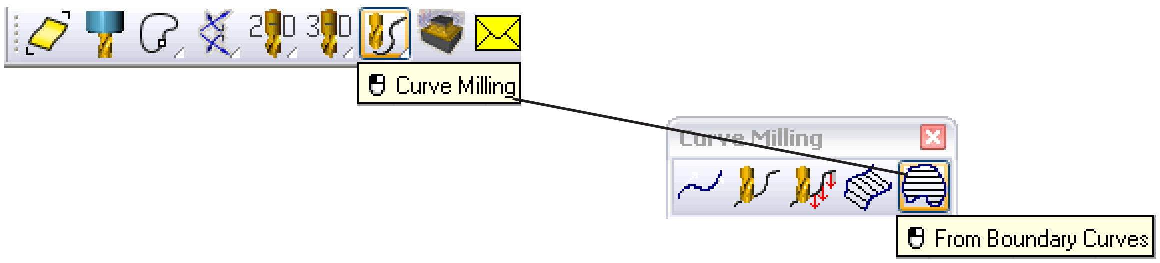
The posted file.



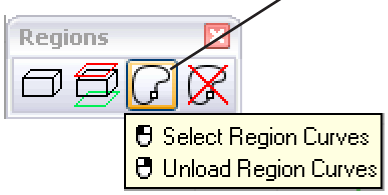
Section 2: Setting a Defined Boundary for Toolpath Calculation

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: Choose the **Curve Milling** button from the MadCAM toobar. Select **Boundary Curves** and create the curve area.

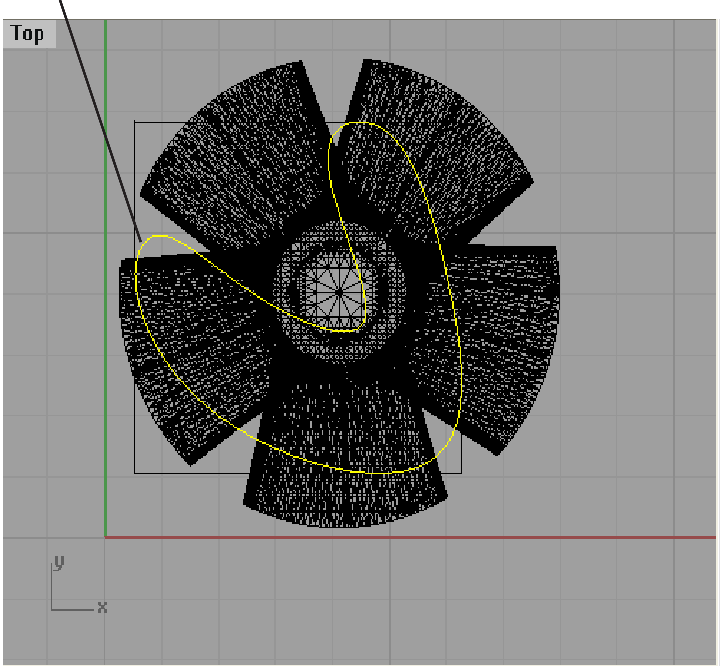


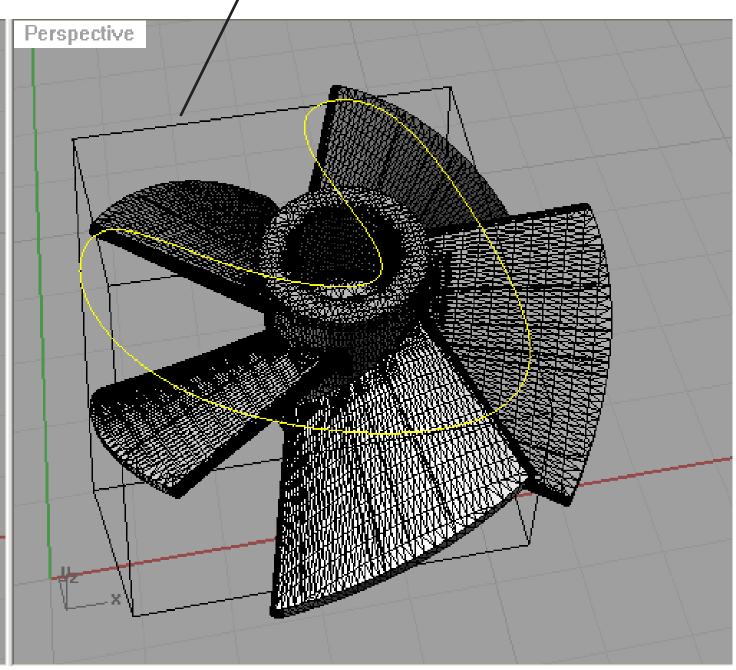
Step 2: Go to the **Regions** button and choose **Region Curves**. Select the curve or curves and press **Enter.** The bounding box will now snap to the chosen curve.



Selected Region Curve.

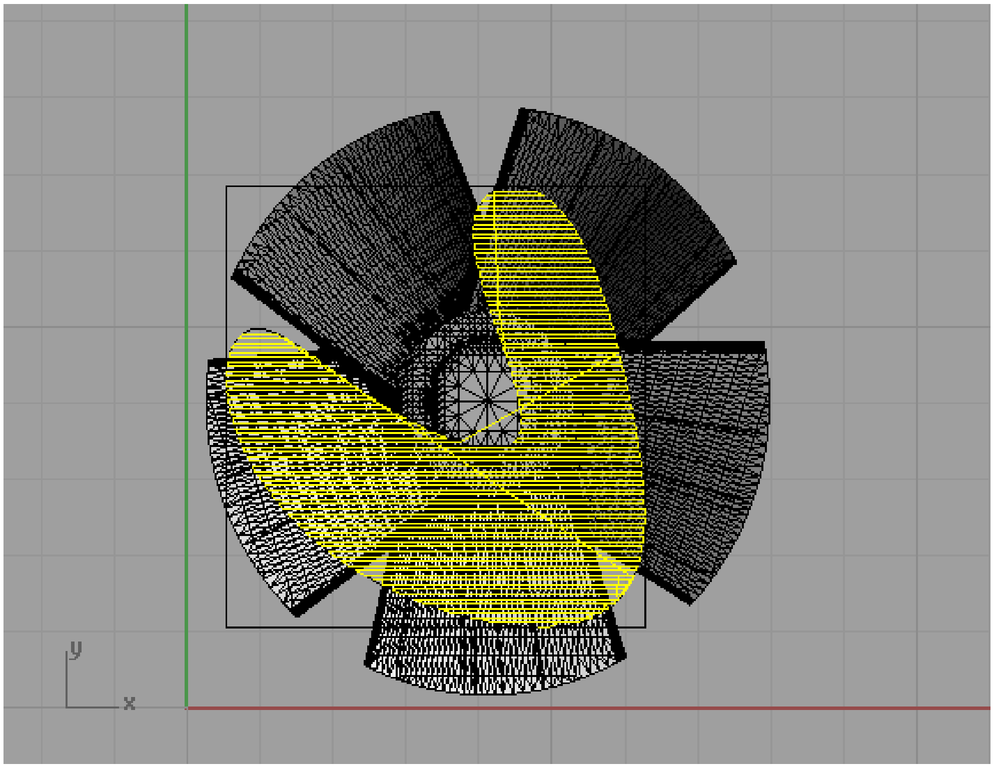
New Bounding box.

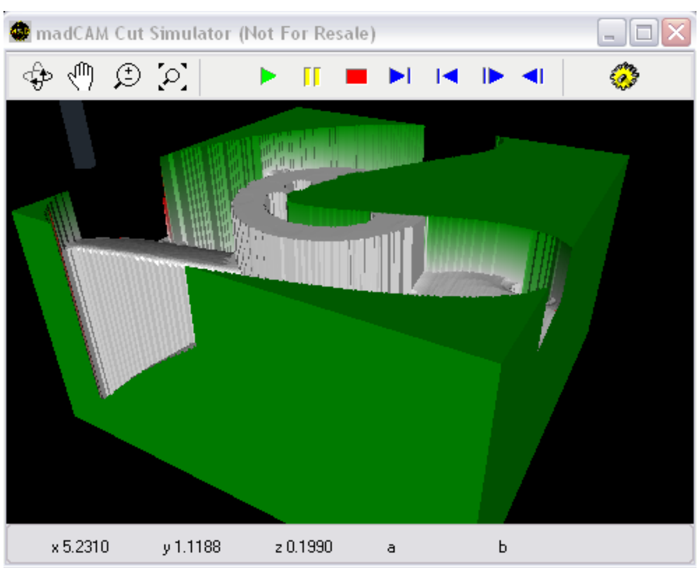




Step 3: Choose a finishing cut. Performing a finishing cut will only cut within the boundary curve you choose.

Example of a Planar Cut:





Step 4: Simulate cutting job.

Step 5: Post the toolpath to the Yale Precix Processor.