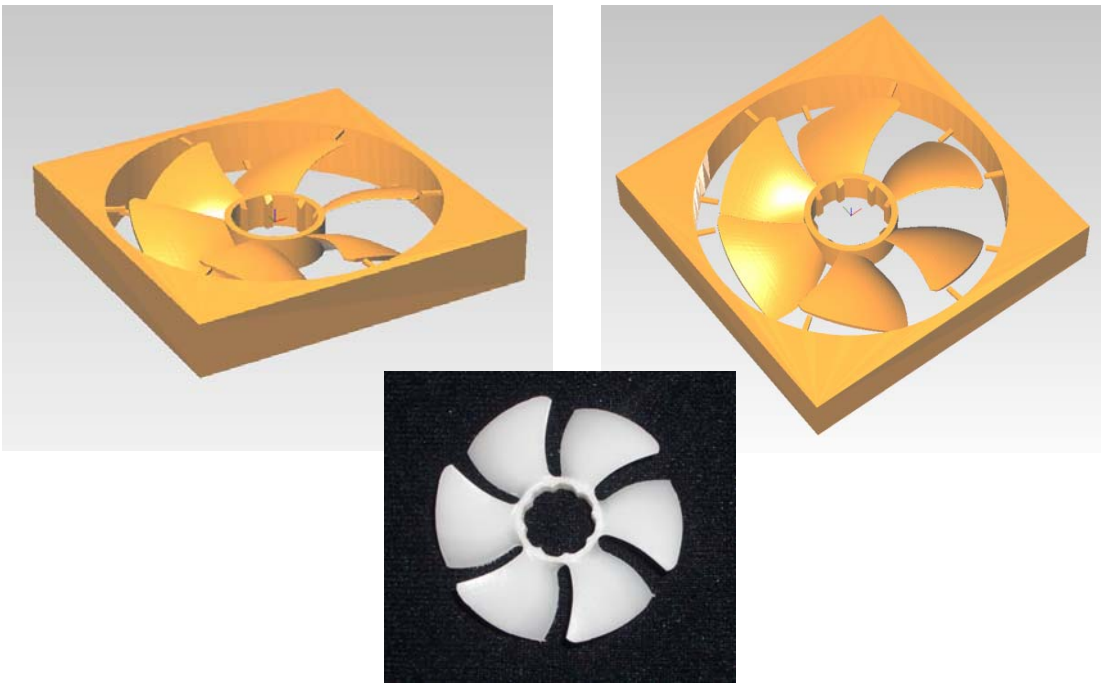
Fan



This tutorial will guide you through the various steps required of producing a double sided part using the MDX-40, ZCL-40, and Modela Player 4. The resulting functional part is a fan that can be used to blow chips away from the cutting surface.

Roland ASD 25691 Atlantic Ocean Drive Suite B-7 Lake Forest, CA 92630 949-655-5555

Materials Required:

•Delrin (Acetal/Polyacetal) Material Stock (min. 3.5” x 2.0” x .315”)

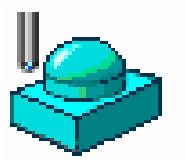
•Material Source: http://www.professionalplastics.com

•0.063 Ball End Mill (EMB-125-3F-063)

•0.125(3.175mm) Collet (ZC-23-3175, Included with MDX-40)

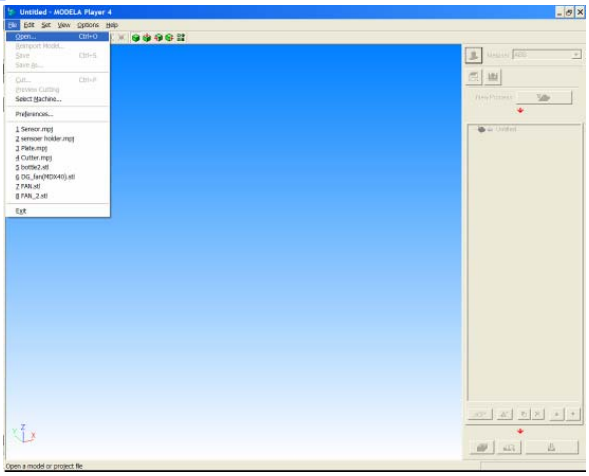
Starting Modela Player 4

1



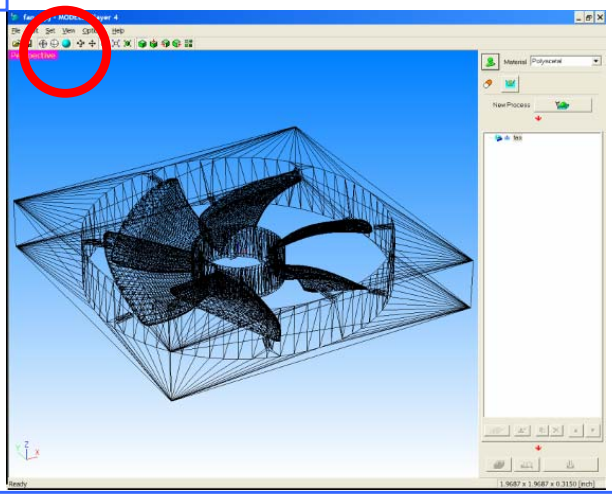
Start Modela Player 4. Either Click on the Desktop Shortcut or **Start** menu, **Programs** and **Roland Modela Player 4**.

2



Click on **File**, **Open**, and select the fan.stl file.

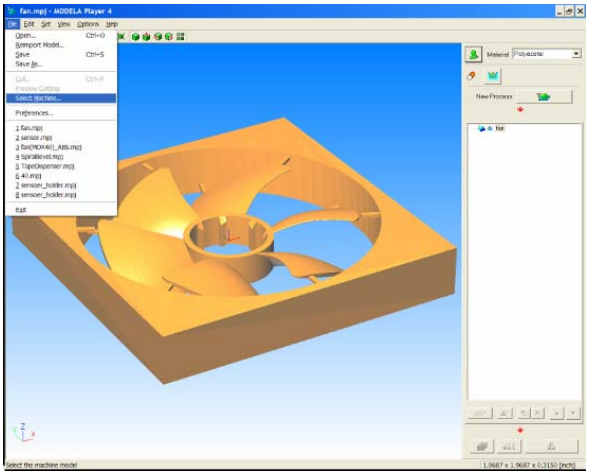
3



To view the part with shading, click on the shading button.

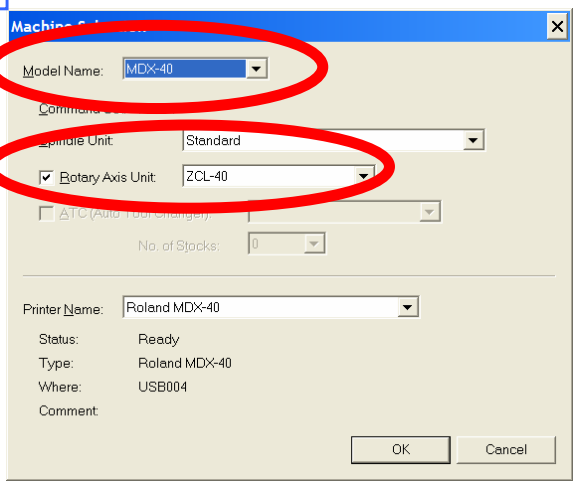
Machine Selection

1



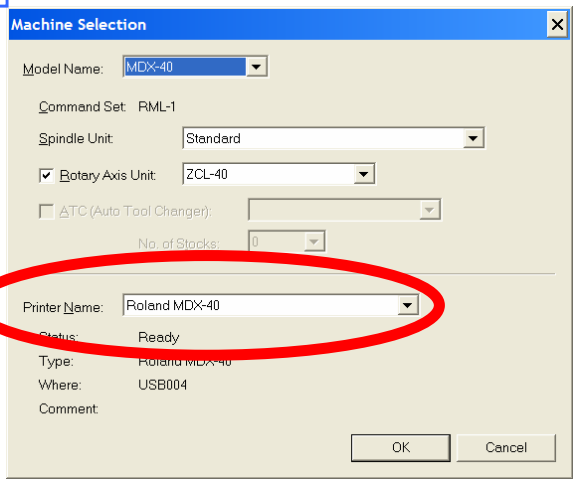
The proper machine and machine driver will have to be selected in Modela Player 4. Click on **File** and **Select Machine** to select the appropriate machine.

2



Select **MDX-40** under **Model Name** and select **Rotary Axis Unit**.

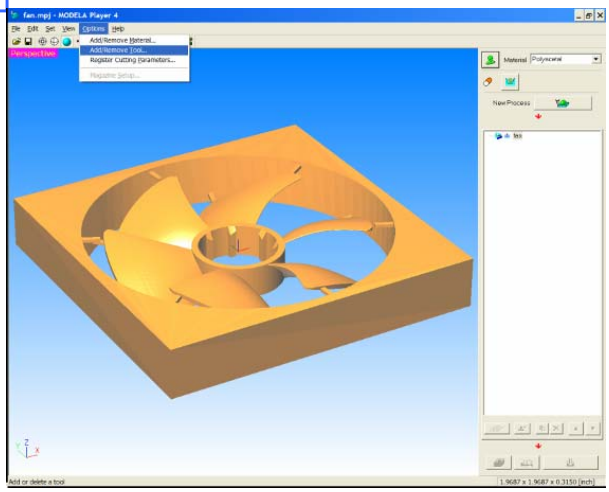
3



Select **Roland MDX-40** under **Printer Name** to select the correct Machine Driver.

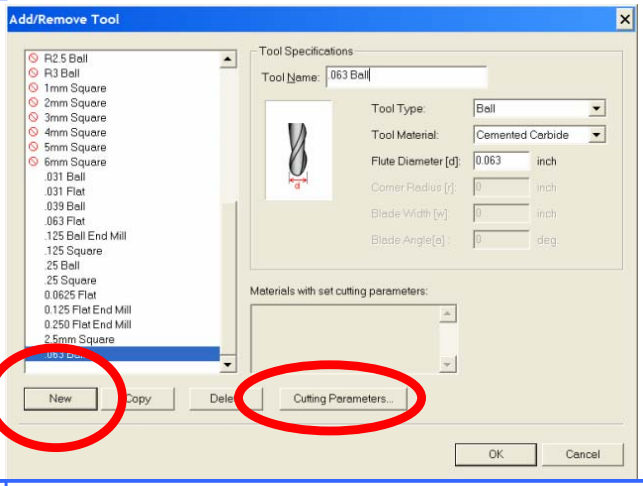
Add Tooling

1



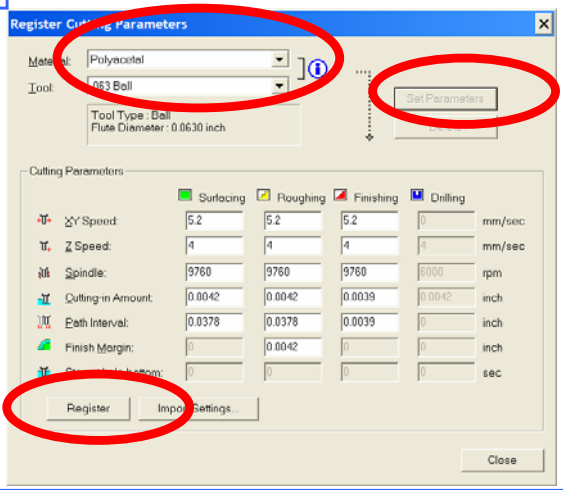
Click on **Option** and **Add\Remove Tool…** to add tools and material information.

2



Select **New** and type in **Tool Name, Tool Type**, **Tool Material**, and **Flute Diameter** for .063 Ball Tool. Try to use a name that will easily identify the tool, for example ".063 Ball". Select **Cutting Parameters** to start adding cutting parameters for that tool.

3

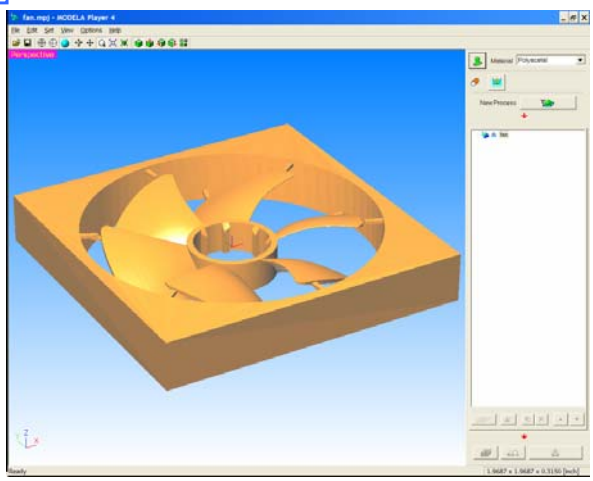


Select **Polyacetal** under Material (same as Delrin, acetal), click on **Set Parameters**, and **Register** to accept the cutting parameters.

Repeat for other possible materials you may use with this tool.

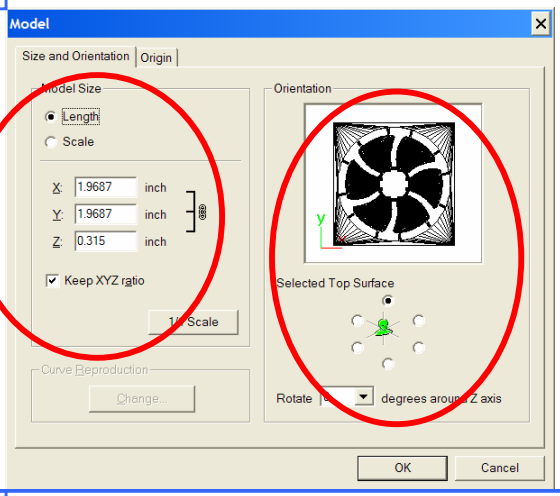
Model Set Up

1



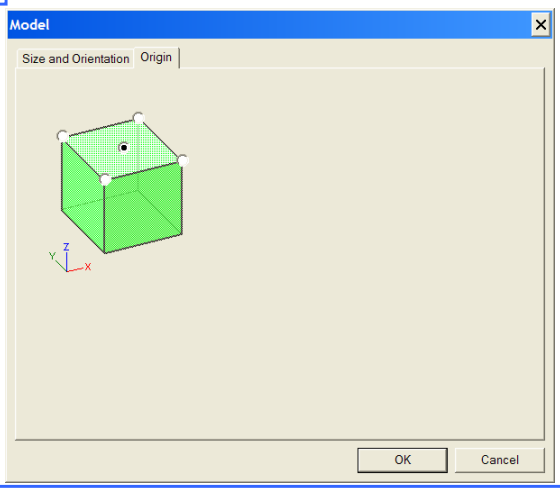
To check the dimensions and origin of the part, click on the **Model** button.

2



Check the size of the part as well as the orientation of the part. The view in the orientation window is from the top. If you need to change the orientation of the part, you would select the different surface view below the orientation view

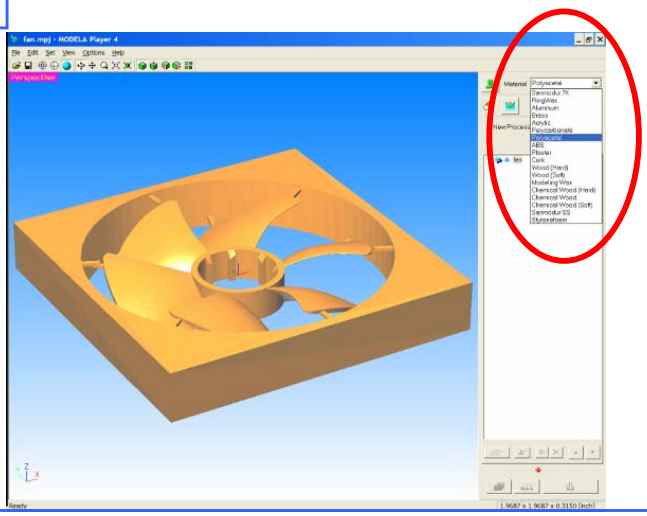
3



Click on the **Origin** tab and select the center of the part as the new origin. Click **OK** when finished.

Material Selection

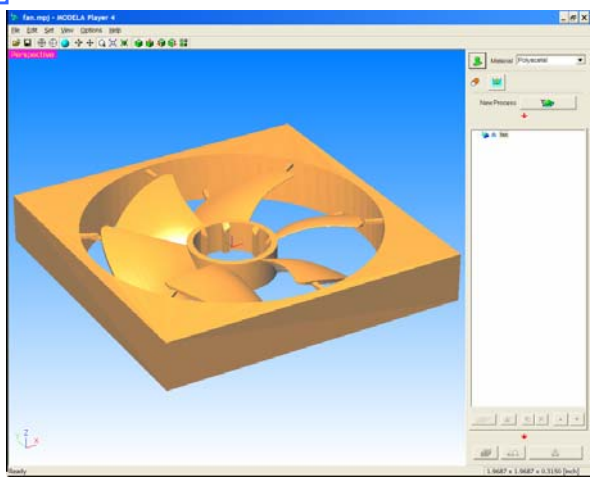
1



Select **Polyacetal** from the pull down material selection. This is the same as Delrin or Acetal.

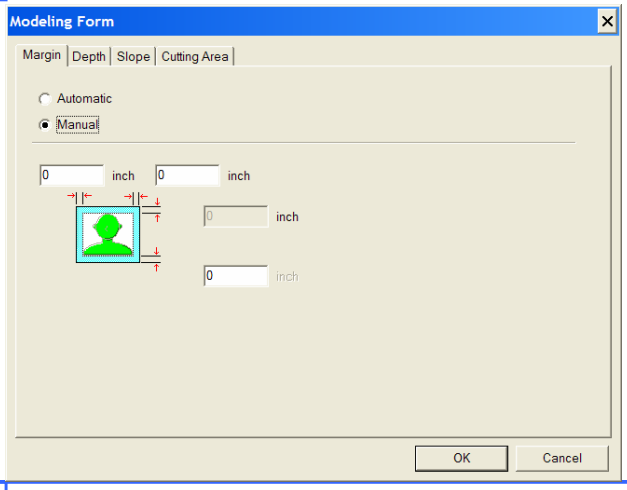
Modeling Form

1



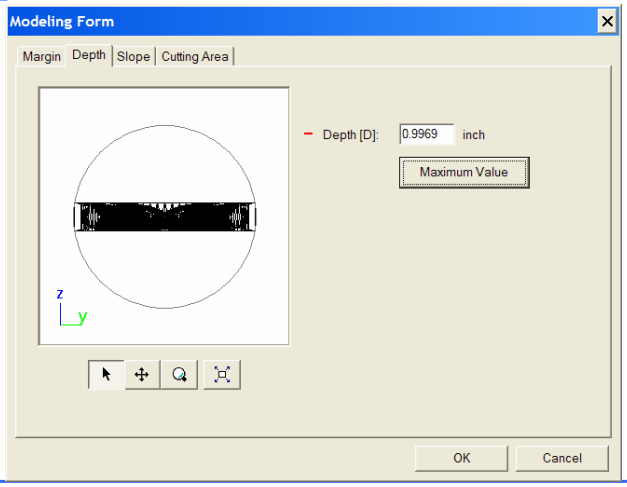
Select **Modeling Form**.

2



This will be the **Margins** around the part. For this example we will be using 0 around the part.

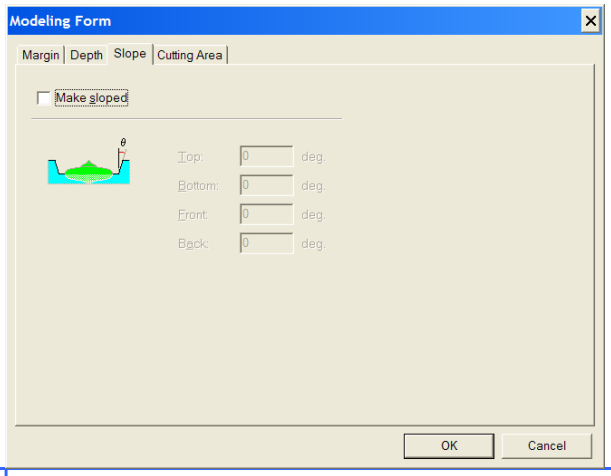
3

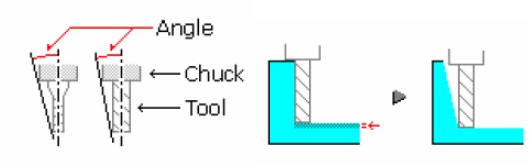


Click on the **Depth** tab. Click on the **Maximum Value** button to allow full milling of the object. Enter a smaller value will allow you to control the depth towards the center.

Modeling Form

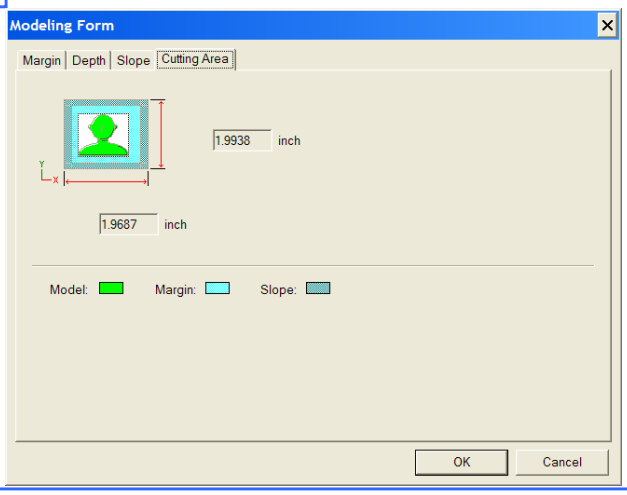
4





Slope will add a slight slope to the side of the material that you are cutting for extra tool clearance if necessary. We won't use this for this sample.

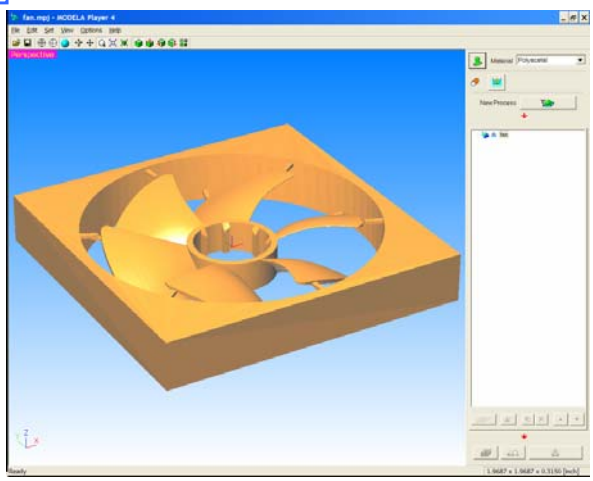
5



Cutting Area will let you know the total area to cut, including part and margins if any. You need to make sure that your material stock is larger than the cutting area.

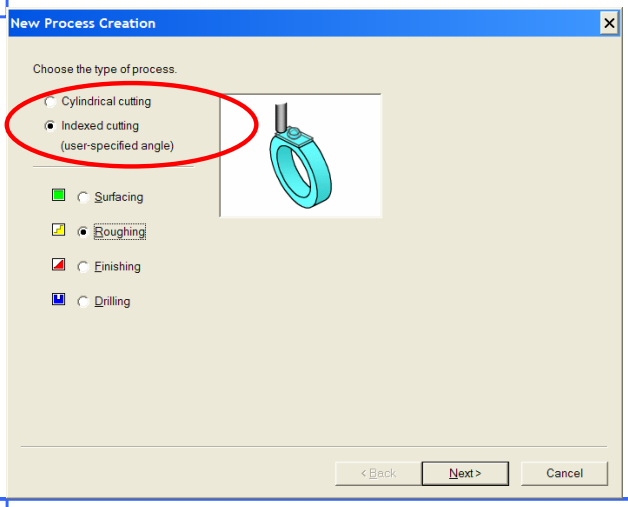
Adding Processes, Top

1



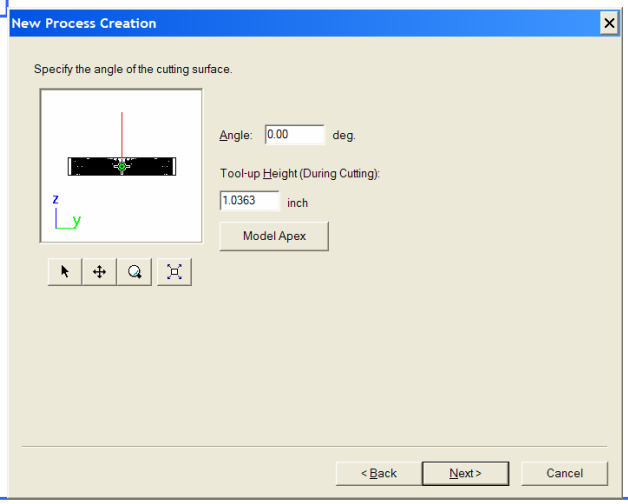
Click on **New Process** to start adding processes.

2



Select **Indexed Cutting, Roughing** and then click on **Next**.

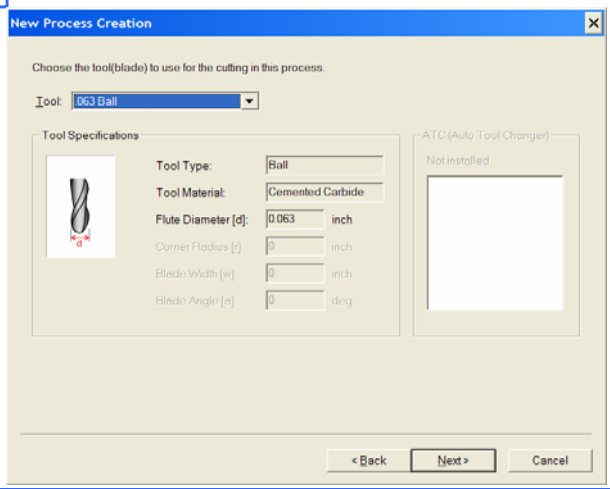
3



You will cutting on the Top (0deg) so enter 0 for the **Angle.**

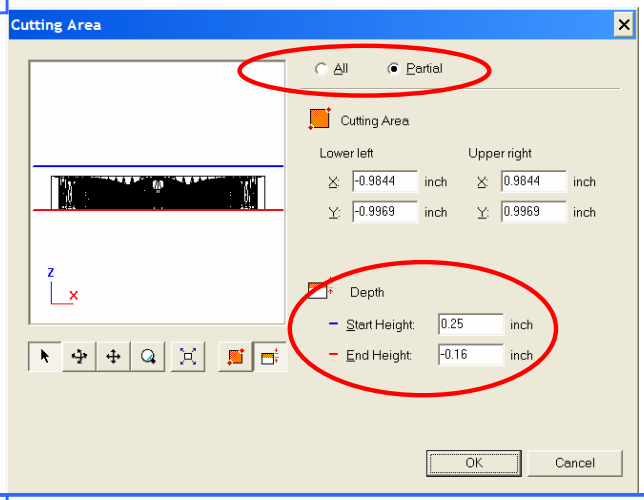
Roughing Process Top

4



Select your tool, or 0.063 Ball for this example. Click **Next** when finished.

5



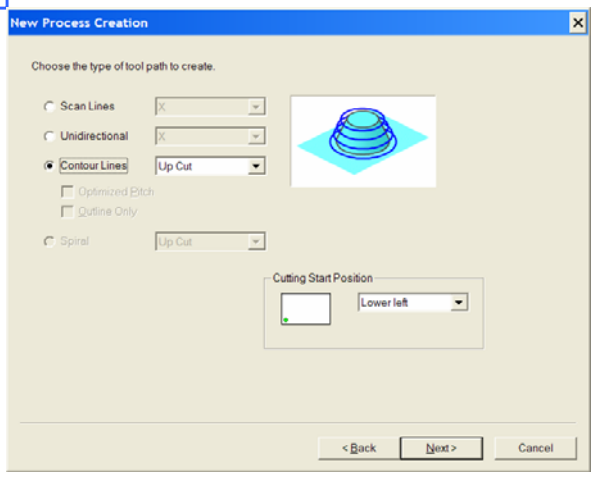
Select **Partial**. You need to change the Start Height of the cutting area, or where the end mill will start to cut.

(Material) / (2) = (Start Height)

0.500 / 2 = 0.250

We also want for mill to cut through the part, so enter an **End Height** value of -0.160

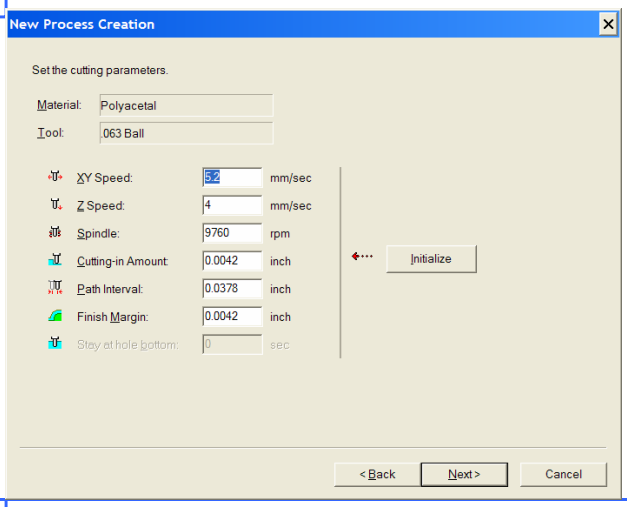
6



Select **Contour Lines** for the type of tool path.

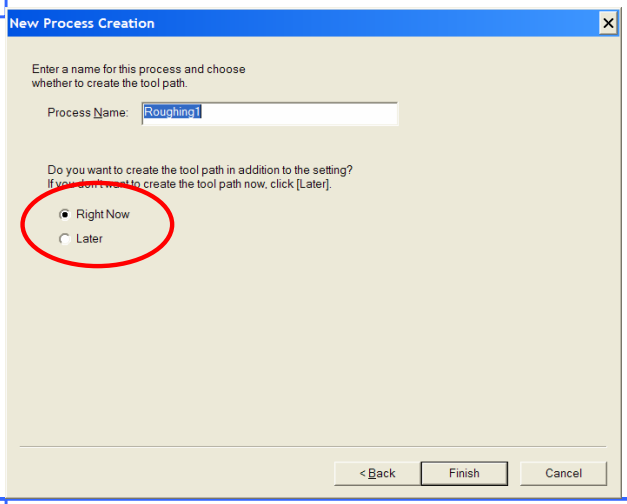
Roughing Process Top

7



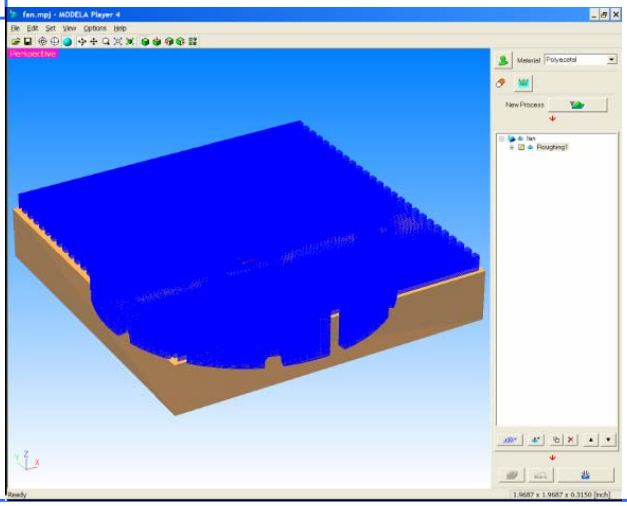
The cutting parameters will be displayed.

8



The Process name will be displayed and you can either process the tool path **Now**, or **Later**.

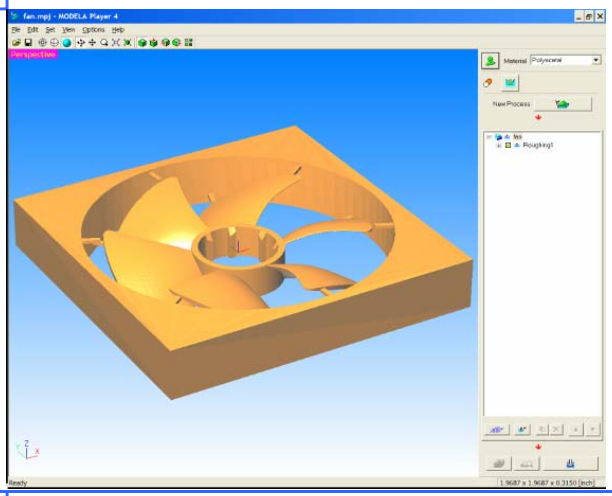
9



The roughing tool path displayed after processing.

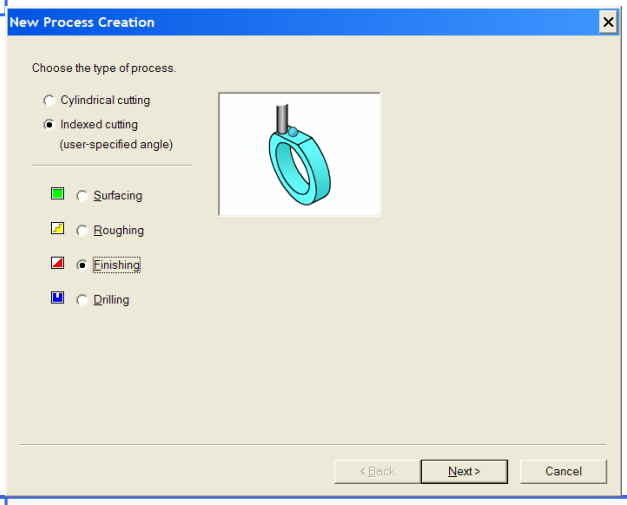
Finish Process Top

1



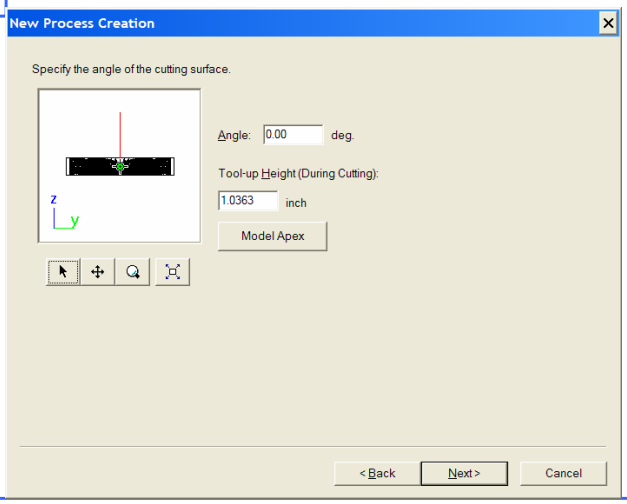
Next you will add the finishing process so click on **New Process** again.

2



Select **Indexed Cutting, Finishing** and then click on **Next**.

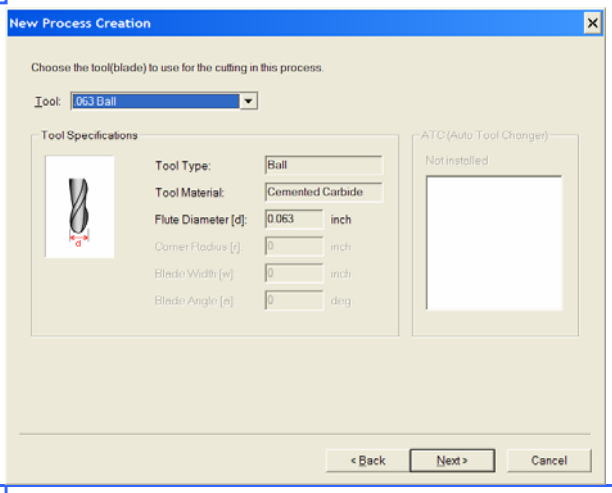
3



You will cutting on the Top (0deg) so enter 0.00 for the **Angle.**

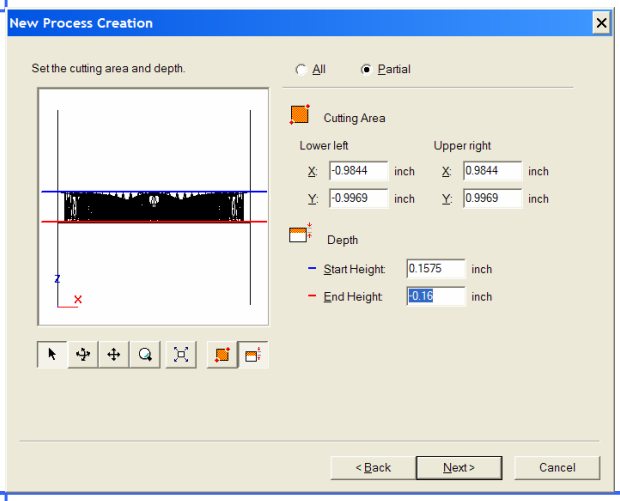
Finish Process Top

4



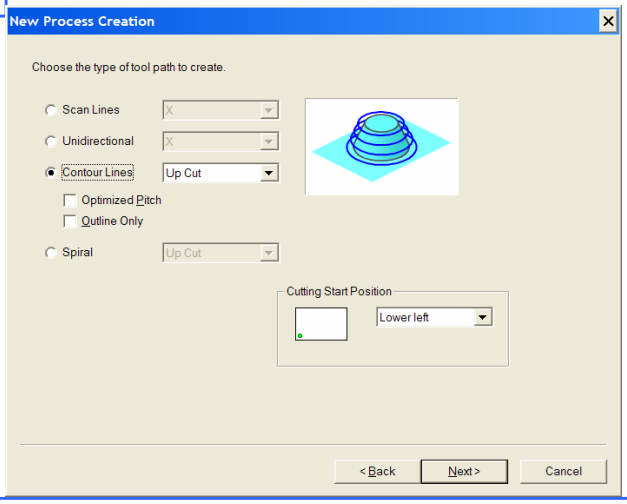
Select the .063 Ball

5



Select **Partial**. Again, we want for mill to cut through the part, so enter an **End Height** value of -0.160 We do not need to change the start height as the material above has already been removed.

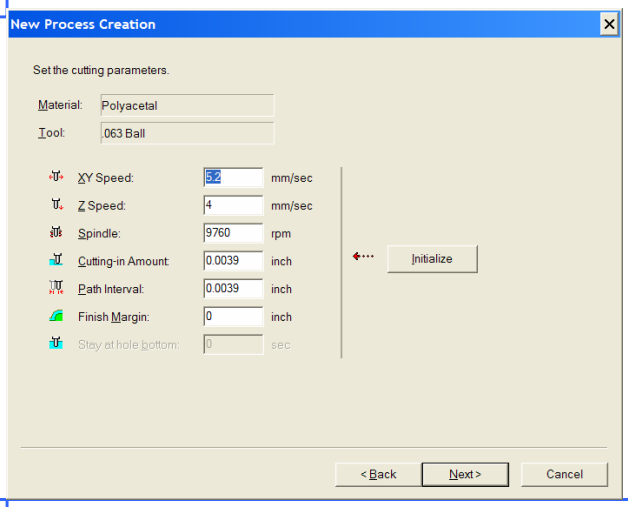
6



Again you are selecting **Contour Lines** for the Tool Path. It may take longer to mill than Scan Lines, but it yields better results.

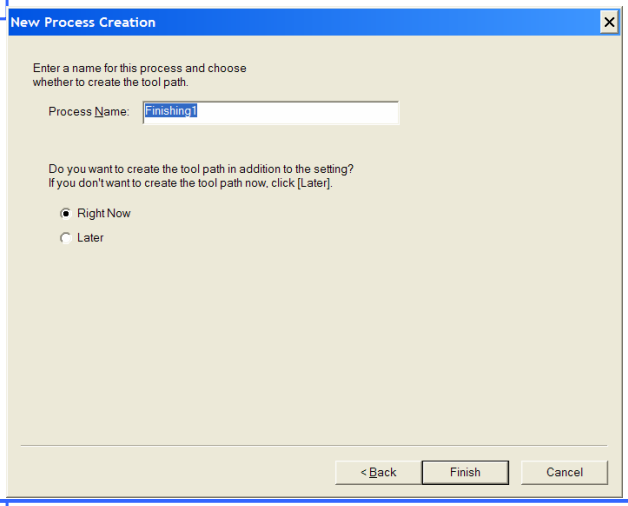
Finish Process Top

7



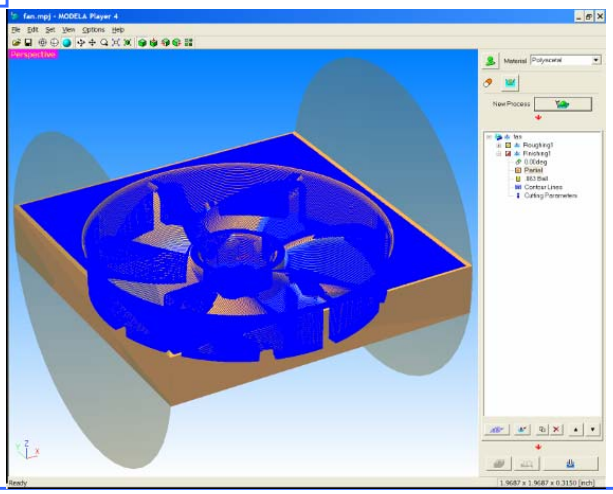
Again use the default settings.

8



Process the finishing process right now.

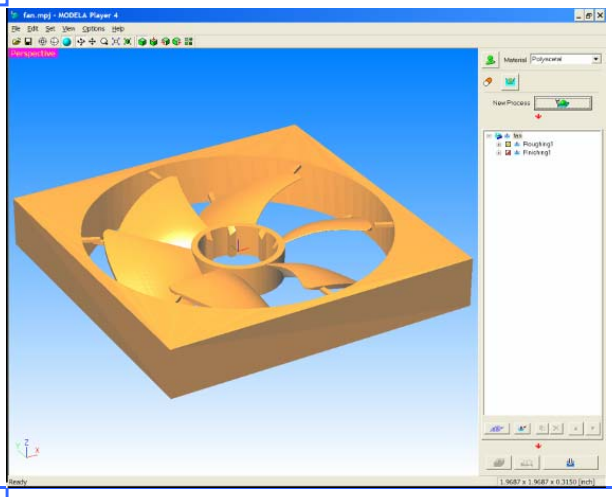
9



Finishing Process tool paths.

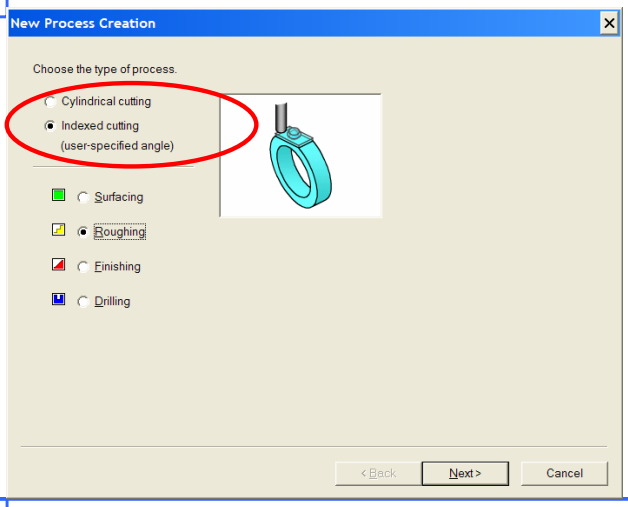
Adding Processes, Bottom

1



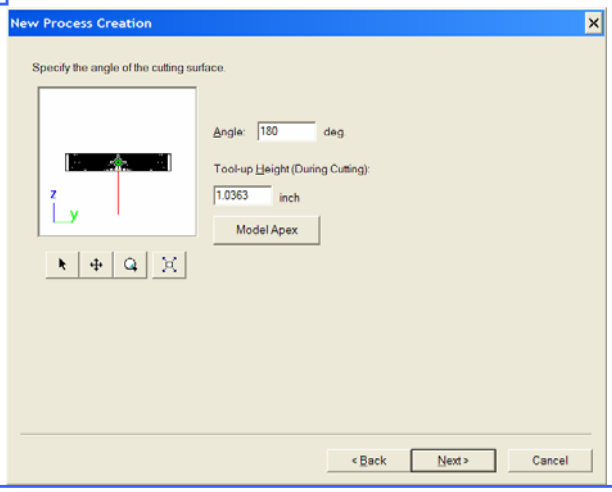
Click on **New Process** to start adding processes.

2



Select **Indexed Cutting, Roughing** and then click on **Next**.

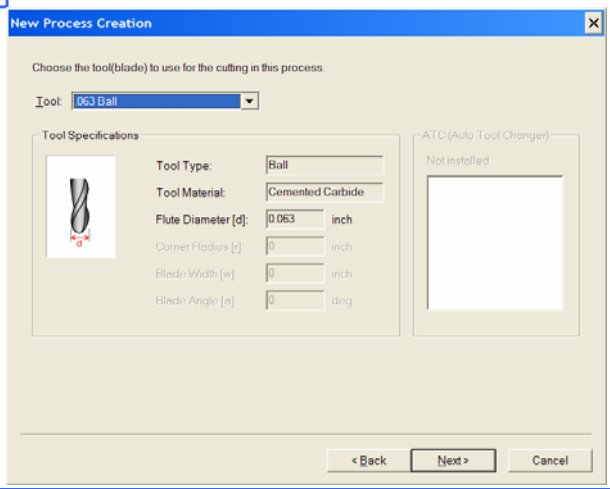
3



You will cutting on the Bottom (180deg) so enter 180 for the **Angle.**

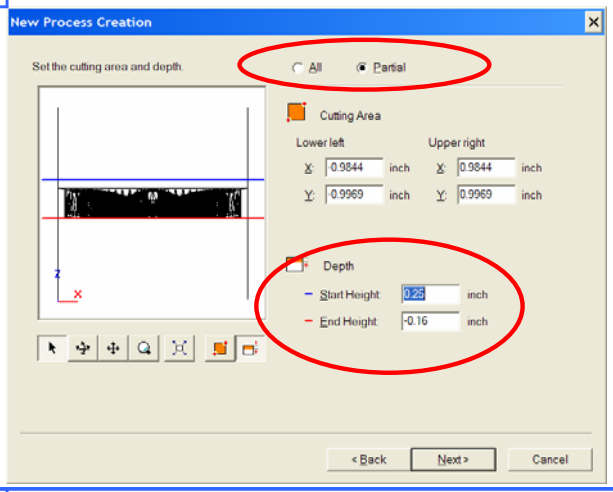
Roughing Process Bottom

4



Select your tool, or 0.063 Ball for this example. Click **Next** when finished.

5



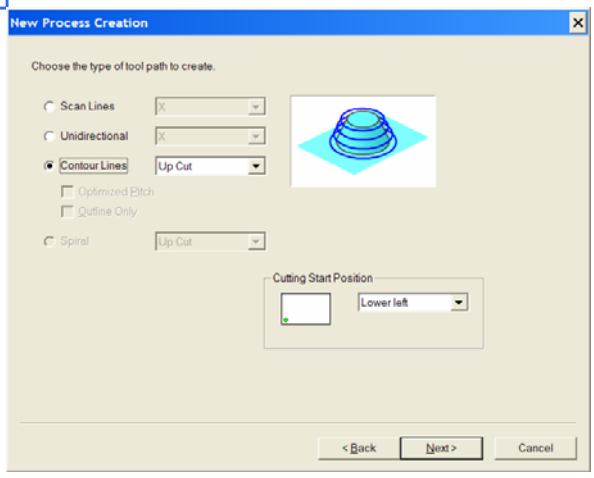
Select **Partial**. You need to change the Start Height of the cutting area, or where the end mill will start to cut.

(Material) / (2) = (Start Height)

0.500 / 2 = 0.250

We also want for mill to cut through the part, so enter an **End Height** value of -0.160

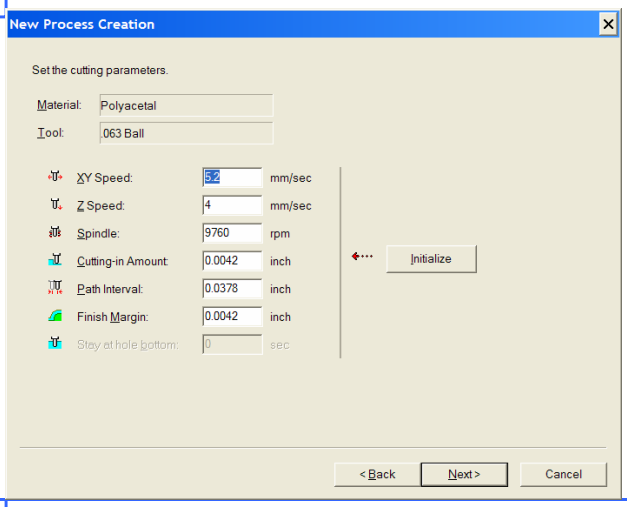
6



Select **Contour Lines** for the type of tool path.

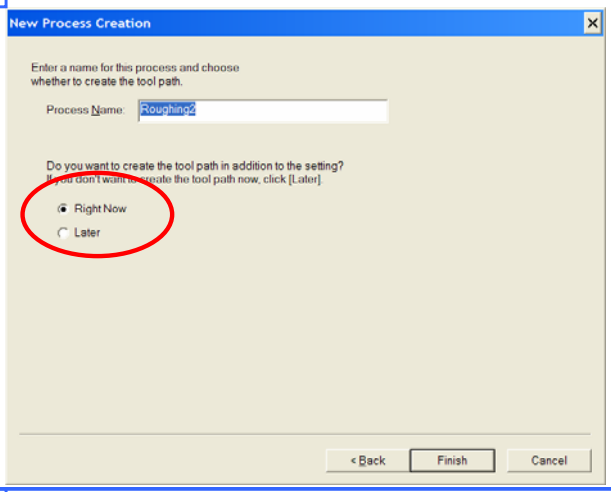
Roughing Process Bottom

7



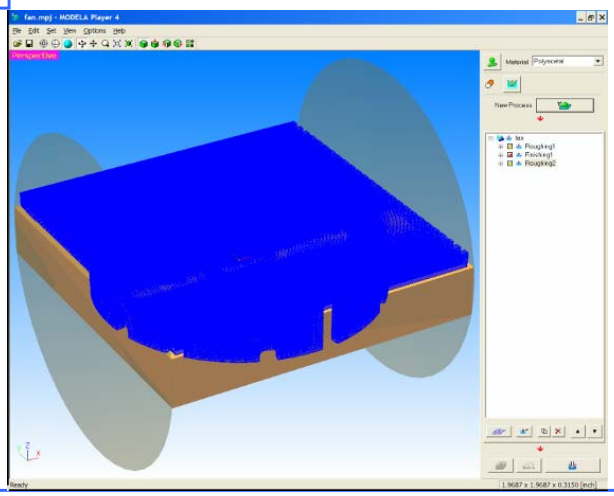
The cutting parameters will be displayed.

8



The Process name will be displayed and you can either process the tool path **Now**, or **Later**.

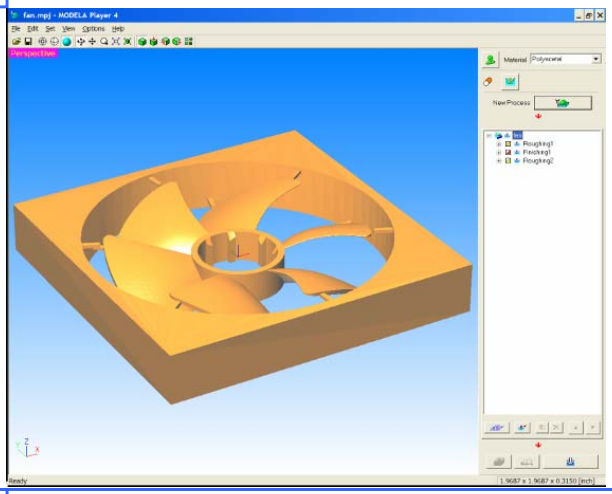
9



The roughing tool path displayed after processing.

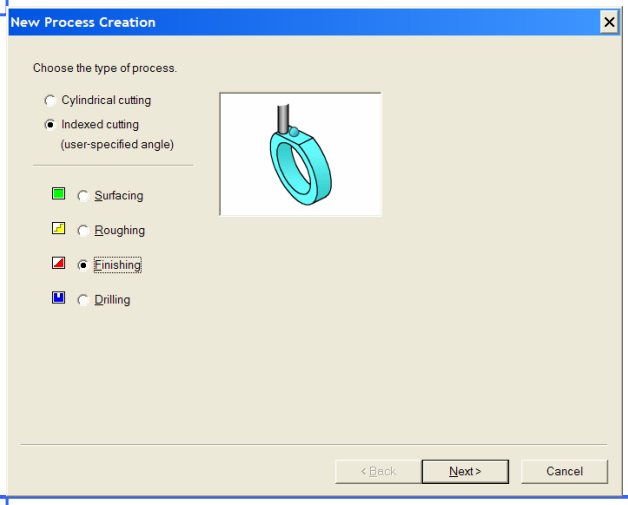
Finish Process Bottom

1



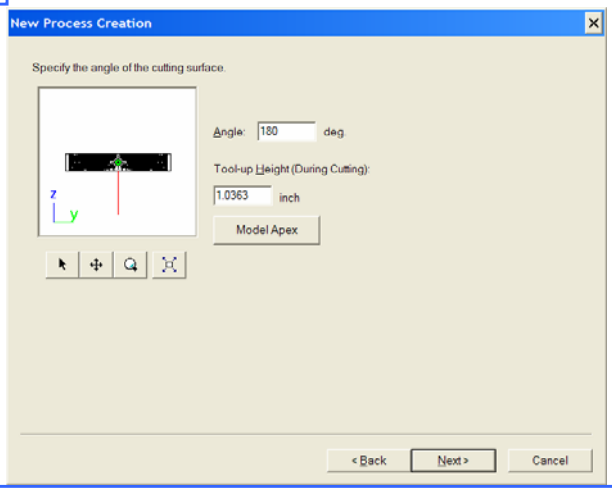
Next you will add the finishing process so click on **New Process** again.

2



Select **Indexed Cutting, Finishing** and then click on **Next**.

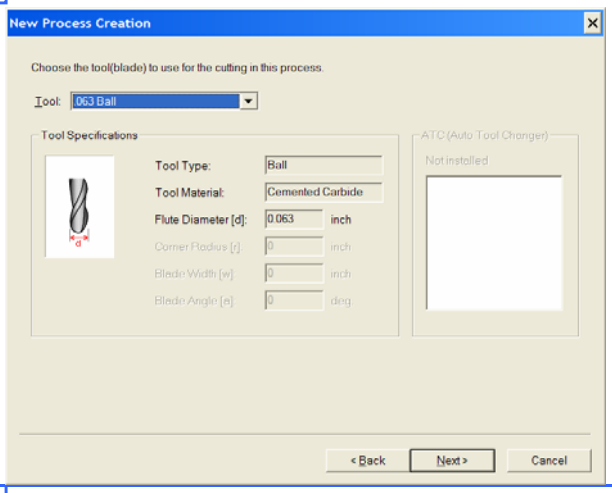
3



You will cutting on the Bottom (180deg) so enter 180 for the **Angle.**

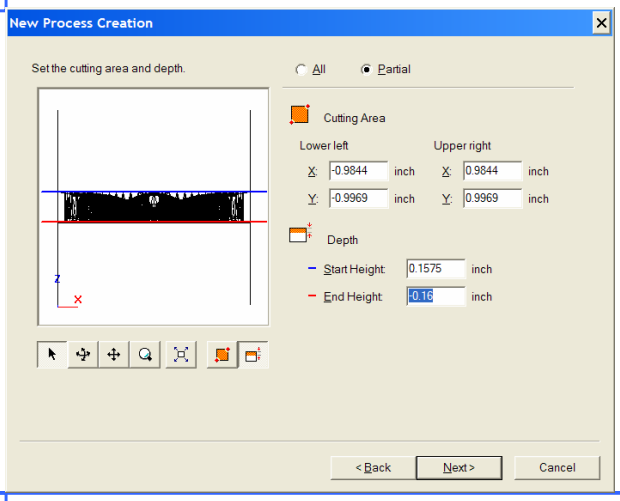
Finish Process Bottom

4



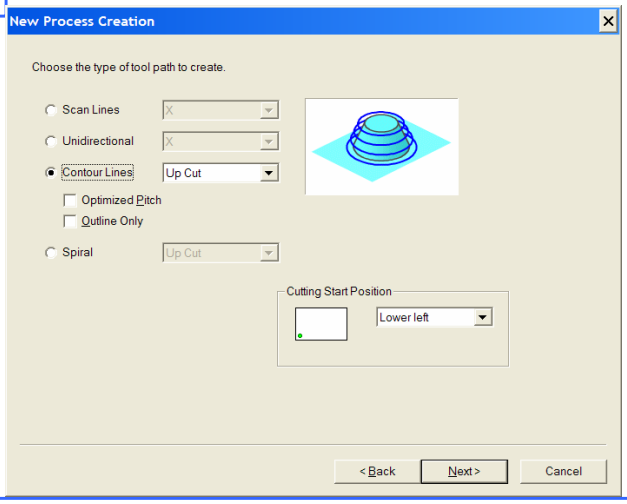
Select the .063 Ball

5



Select **Partial**. Again, we want for mill to cut through the part, so enter an **End Height** value of -0.160 We do not need to change the start height as the material above has already been removed.

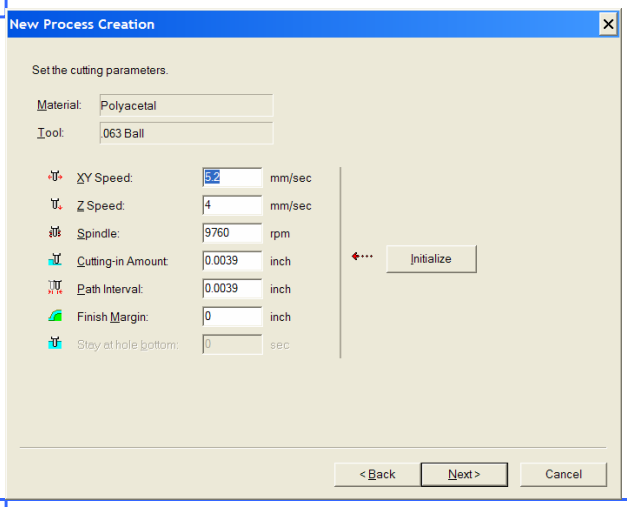
6



Again you are selecting **Contour Lines** for the Tool Path. It may take longer to mill than Scan Lines, but it yields better results.

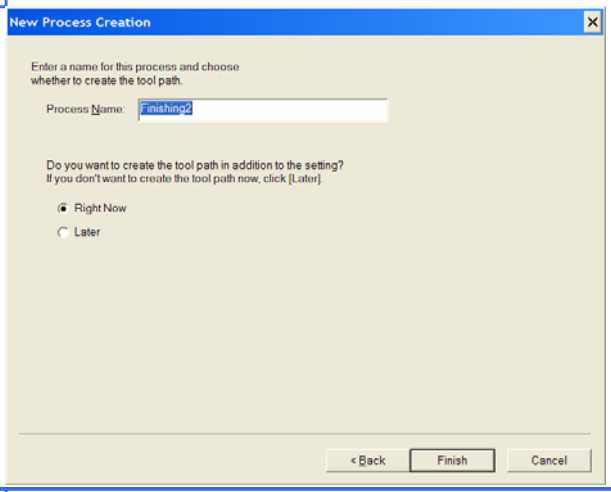
Finish Process Bottom

7



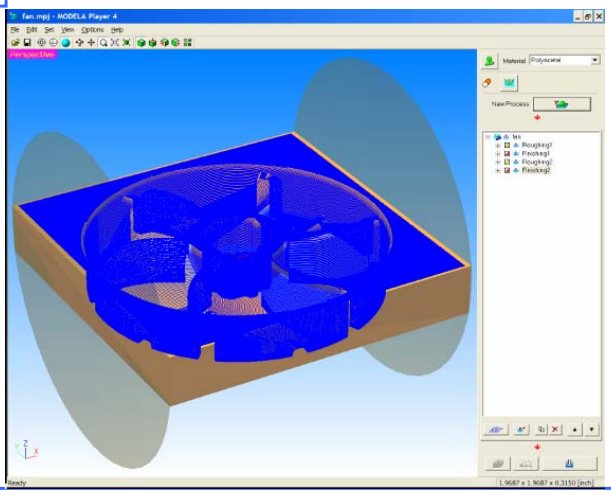
Again use the default settings.

8



Process the finishing process right now.

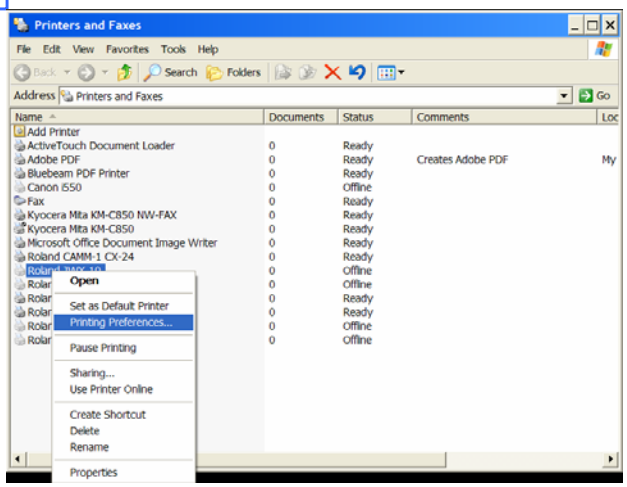
9



Finishing Process tool paths.

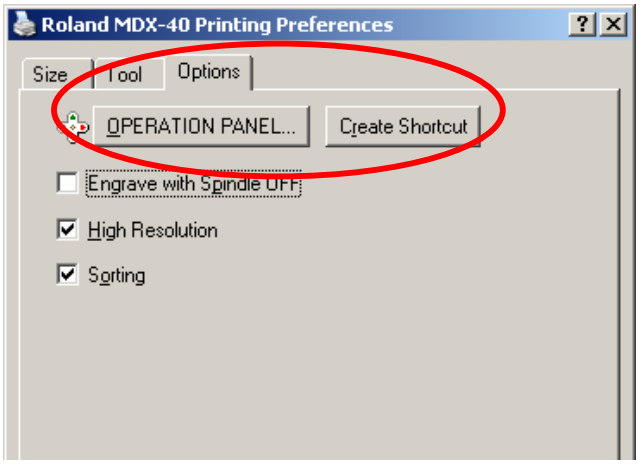
Virtual Control Panel

1



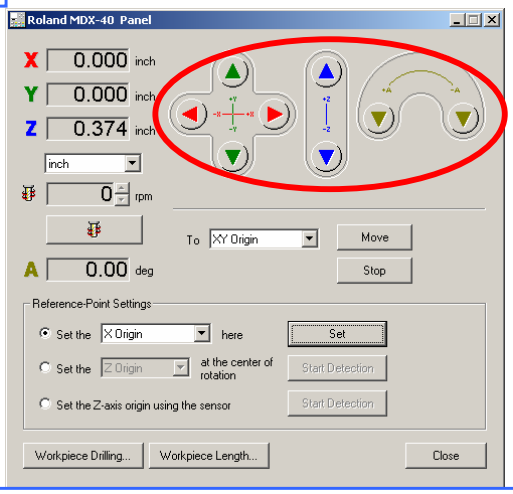
To load the Virtual Control Panel, click the Windows **Start** menu, **Settings**, and **Printers and Faxes**. Right click over the **Roland MDX-40** and select **Printing Preferences.**

2



You can open the Virtual Control Panel by clicking on **OPERATION PANEL**, or you can also Create a Shortcut onto your desktop.

3



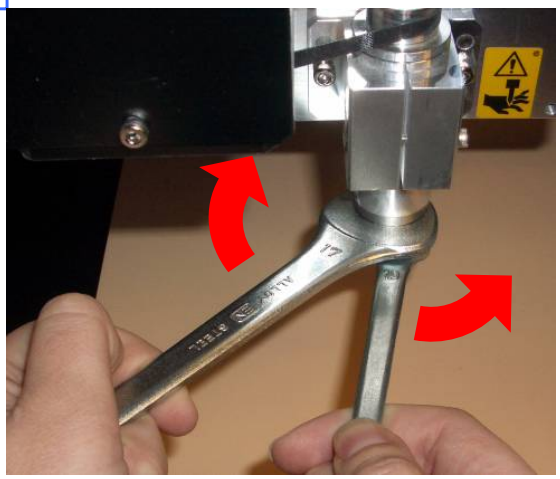
You will use the Virtual Control Panel to move the machine in the X,Y, Z, and A direction. You will also use this to set the various origins.

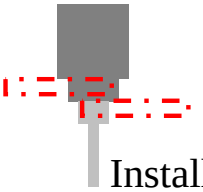
Installing Tool, Material

1

Select your 1/8” collet and 1/16” (.063”) Ball End Mill.

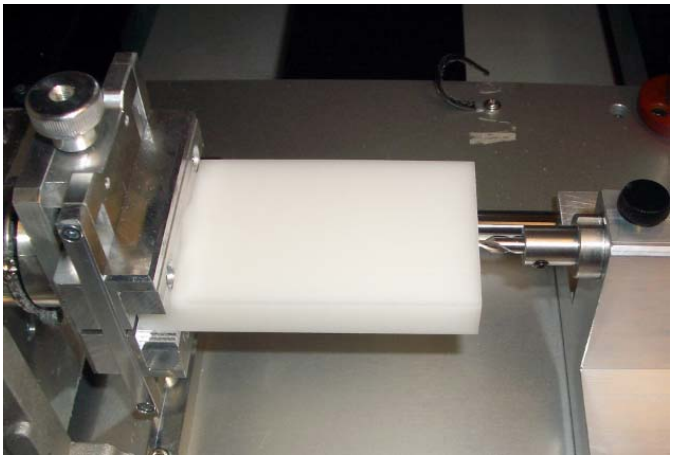
2





Install the collet and tool in the spindle. Place the 17mm tool in your left hand 1st and over the spindle nut as shown. Place the 10mm tool in your right hand and over the collet. To tighten, pull the wrenches away from each other.

3

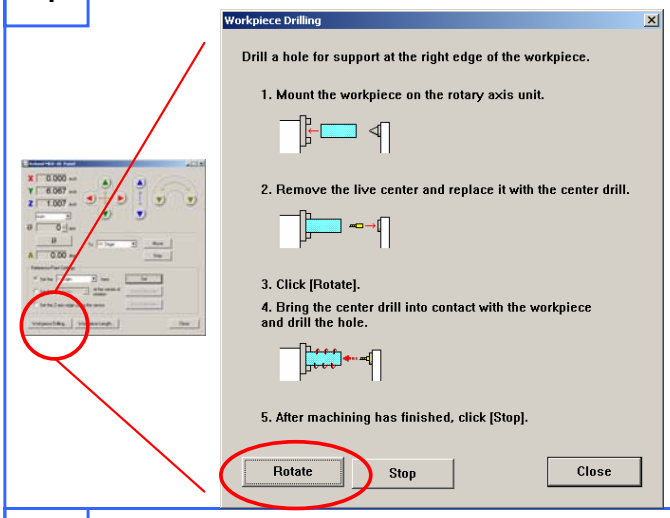


Place material into the self centering clamp. Install center drill and place next to end of material. The center drill will be used to place a small hole for the live center to placed into.

Use the lever to tighten and loosen the live center support

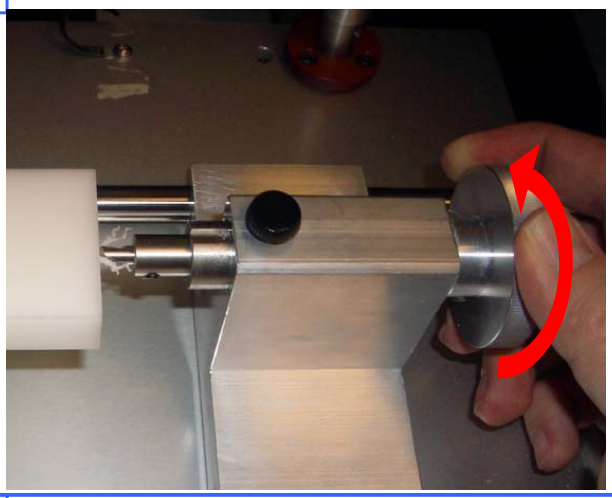
Installing Tool, Material

4



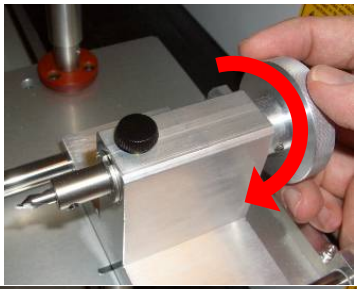
From the Virtual Control Panel, Click on the "Workpiece Drilling" Function. Click on the Rotate button to start the A-Axis spinning. Please make sure the machine is not in "view" mode.

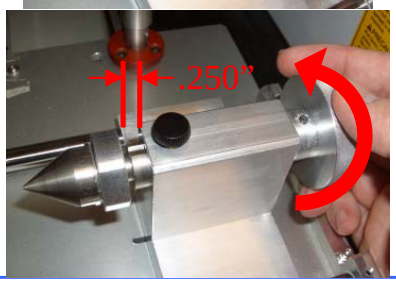
5



As the A-Axis rotates, **slowly** feed the center drill deeper into the part by turning the core support adjuster on the 4th axis support. Press the stop button when the drill cone is about half ways in the material.

6

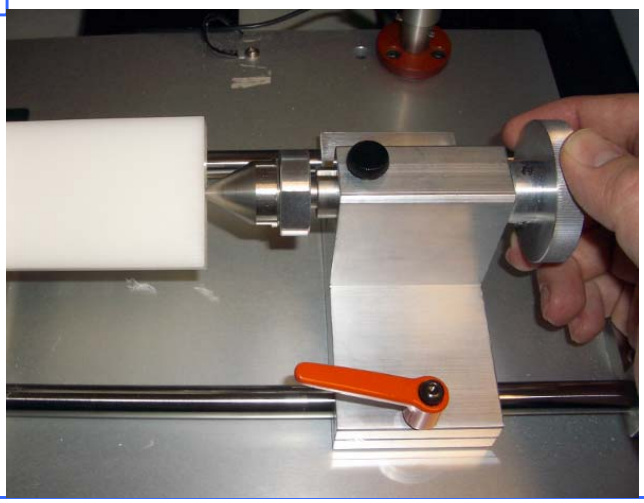




Remove the center drill by turning the core support adjuster until the center drill is free. Remove and place live center in the 4th axis support. Turn the core support adjuster until the support is about .250" out.

Installing Tool, Material

7



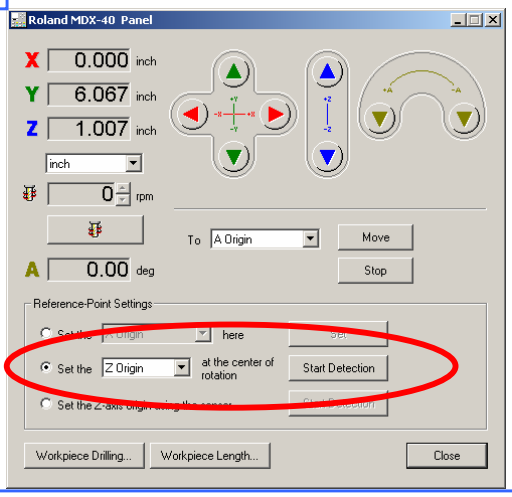
Move the live center support until the live center is touching the material stock. Turn the core support adjuster to ensure that the material is firmly supported. Turn the black adjuster lock to lock the core support adjuster.

Z-Origin

1

Ensure that the sensor cable is connected to the Z-Origin sensor.

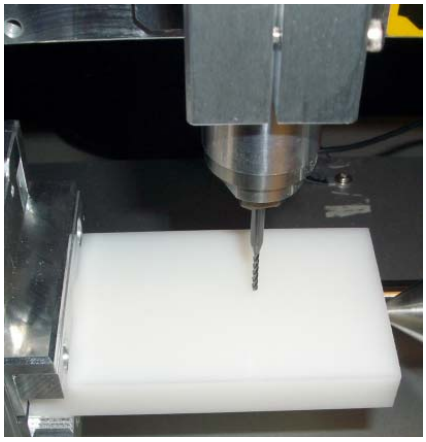
2

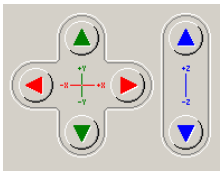


Select the **Z-Origin** for the "Set the Z-Origin at the center of rotation" reference point settings. Click on the **Start Detection** button. The Tool will move down, touch the Tool Sensor several times, and the Z-Origin will be set after this.

X Origin

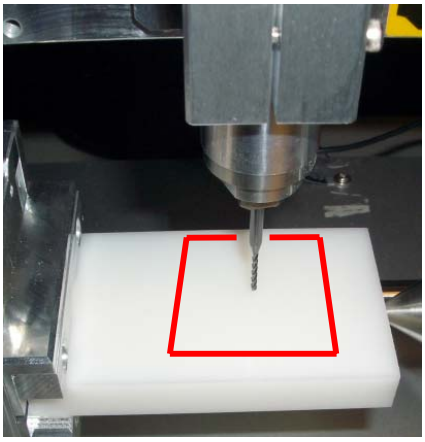
1

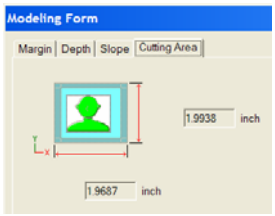


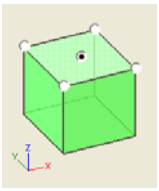


Since you are using a 4th axis, Y will always be 0. You will only be changing X when using the 4th axis. To set the X origin, use the virtual control panel to move the tool to the approximate center of the cutting area. You also need to ensure that you are far enough away from the clamp to avoid hitting it during milling.

2

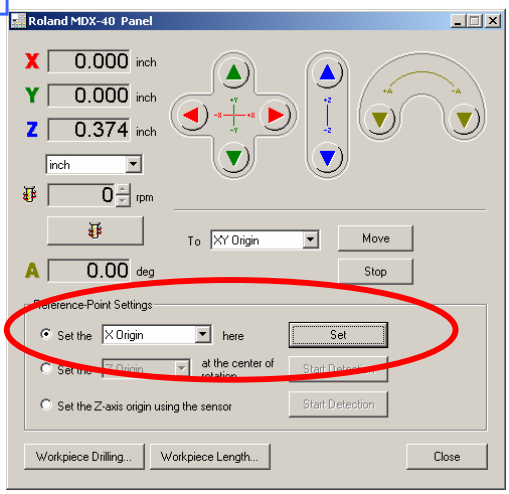






The material stock needs to be larger than the cutting area specified under Modeling Form, Cutting Area. Also note that the origin used under Model, Origin, center is how you are locating the cutting area on the material.

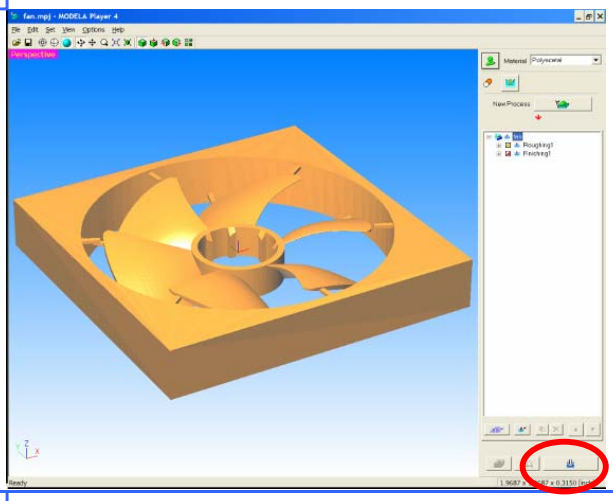
3



Set the X Origin by selecting the **Reference Point Settings** and selecting **X Origin**, and clicking on Set to set the X Origin. You will notice that the Virtual Control Panel will now display X 0.00.

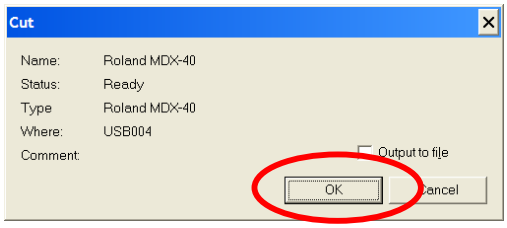
Cutting!

1



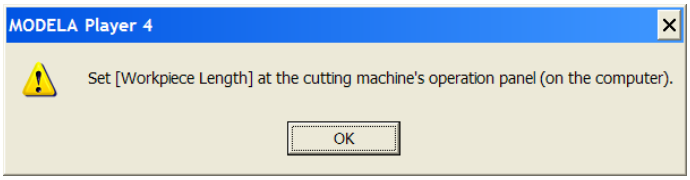
Click the **Cut** button to send the program to the machine.

2



You will get Cut screen confirming the Machine Name and Printer Driver selected. Click **OK**.

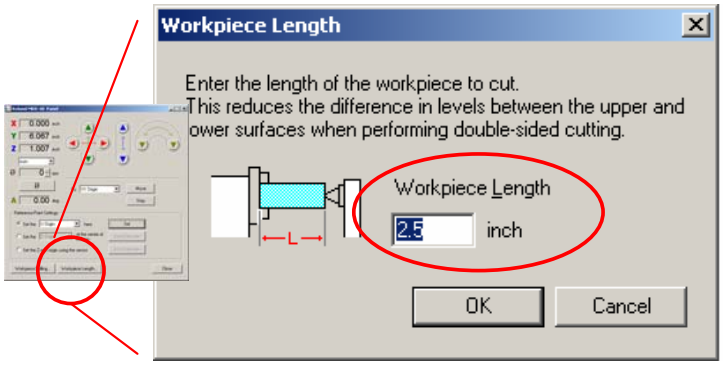
3



A message will appear for entering the material stock (workpiece length) on the virtual control panel.

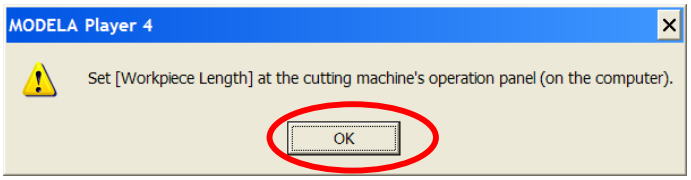
Cutting!

4



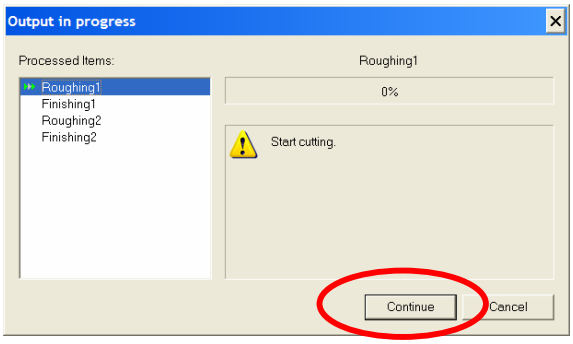
Click on the Workpiece length and enter the length of your material stock. Click **OK** when finished

5



Click **OK** when you have entered the workpiece length.

6



Click **Continue** to start sending the program to the machine.

Finished Functional Part

1