Tutorial: 99649 - Native American on Horse

Background

I was commissioned to carve a plaque that would be used as a gift for a person on the Rosebud Reservation of South Dakota. I chose design *99649 - Native American on Horse* from the 3DModelClub.com web site. After purchasing the design I downloaded the STL file according to the instructions that were e-mailed to me.

Carving Considerations

Carving 3D designs is time consuming even on modern CNC equipment. Expect machine times anywhere between one and forty hours depending on the design size, complexity, and materials. *(Most of my 3D work takes three hours for a finish pass.)*

Warping, twisting, and cupping can be an issue for deep relief carving in wood if the design is deeper than 50% of the material thickness. Ensure the material is twice as thick as the carving, or design the project to use a frame around it for support.

One of the appeals of the 3D Model Club designs is the significant amount of detail in the Z depth. For this particular design the default dimensions (which are scalable in Aspire) are: X 89.34 mm, Y 99.76 mm, Z 10.65 mm

Therefore the Z depth is about 11% of the maximum dimension of the model. This has to be taken into consideration. It means that if the design is 10 inches across the depth of the carving will be *more* than 1 inch thick.

While it is possible to decrease the Z dimension in Aspire, the design will lose visual impact and character as the design is flattened. For example:

Z @ 100% Z @ 75% Z @ 50%

Therefore care should be used in reducing Z height within the software.

Another method to reduce height without losing as much detail can be accomplished during the STL import process by moving the zero plane position in the model. This can be useful when the deepest parts of the design can be eliminated without loss of aesthetic appeal. This will be demonstrated in the STL Import section of this document.

The 3D Model Club web site lists the dimensions for many of the designs so you can determine if you need to make design changes such as thicker glue-ups, frames, or slicing.

For this particular project a smaller plaque size was appropriate. However, for larger designs consider the following:

- If the Z height was less than 1.5” then do a glue-up of the appropriate thickness. The reason for 1.5” is that the best ¼” and 1/8” tapered ball nose bits have a maximum cutting length of 1.5”.

- For designs with a Z depth ***greater*** than 1.5” inches use the **slicing** feature of Aspire to break the design into manageable Z depths. The pieces are then stacked and glued together for finishing.

Style

There are generally three styles for carving designs:

Onlay	**Dish**	**Scooped**
This style is used when the carved item is glued onto a larger piece such as a desk or cabinet	The dish style has a uniform concave appearance. It provides a uniform boarder around the design	The scooped style has more of a hand-carved appearance
		

Finish

Finally, finish work will make a significant difference on the appearance and impact. For projects carved in wood a glazing technique using glaze or gel stain to accent deeper features of the design works extremely well and is relatively simple.

This particular project was carved in hard maple and used a gel stain to accent the carving detail.

Common Steps for All Projects

Open a New Project in Aspire

For 3D carving it is customary to set the Z-zero point to the material surface.

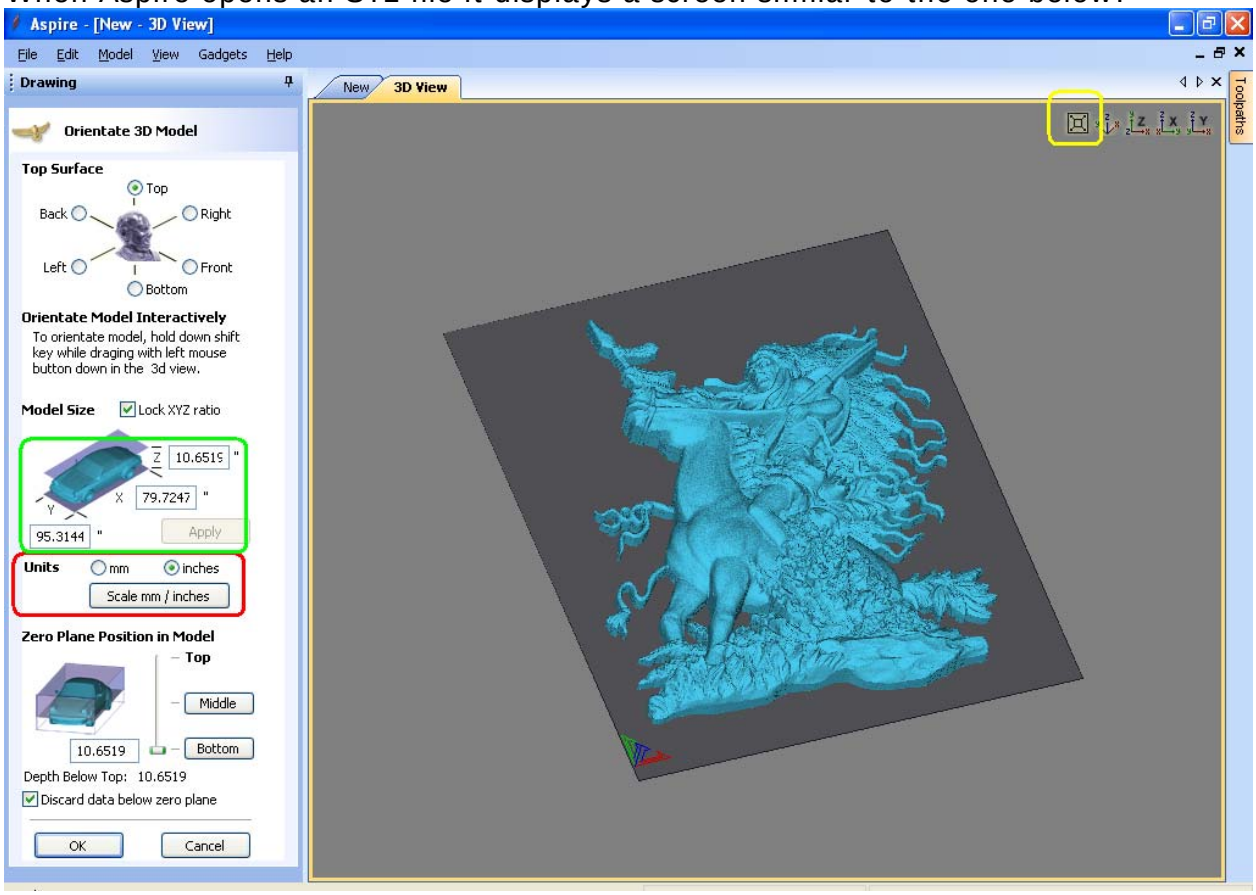
The Setting of the XY origin is a matter of preference. *(In this example the XY origin will be set to the lower left corner.)*

For this example set:

- Width: 12”
- Height: 12”
- Thickness: 1”
- Z-Zero: Top of Material
- XY Origin: Lower Left
- Units: Inches

Importing the STL File

When Aspire opens an STL file it displays a screen similar to the one below:



Chose Units (mm/inches)

The model will import with its dimensions in millimeters. If you typically work in inches a common error is to forget to have Aspire convert the model dimensions to inches and the model will appear huge in the design. The screen area highlighted in red indicated where this is done. Simply click on **mm** or **inches** and click on the **Scale mm/inches** button.

For this example set the scale to **inches**.

Adjust Model Dimensions

Aspire's STL import screen allows adjustment of the model dimensions. The screen section highlighted in green shows where this is done. As long as the **Model Size** checkbox **Lock XYZ ratio** is checked changing one number and clicking **Apply** will proportionately scale the other two dimensions. As the model size changes you may need to press the icon (highlighted in yellow) that resets the image size to fit within the display.

For this example set the Z dimension to 1” and click **Apply**.

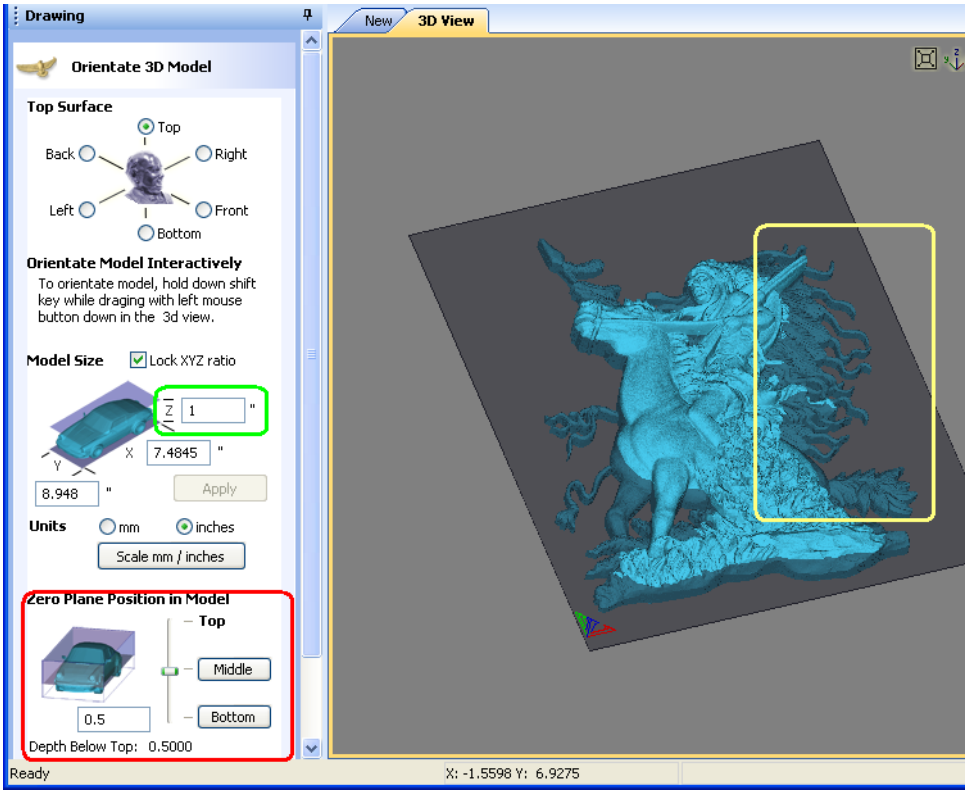
Optional: Adjust Zero Plane in Model

Note: Adjusting the Zero Plane is an optional step and is not needed with every project.

As mentioned previously, it is possible to reduce some of the thickness of the model by moving the zero plane up and discarding portions of the design that fall below that line. This could be important if the carving needs Z detail but is constrained by the thickness of the material. *The example below is extreme in order to make it obvious what portion of the design would be lost.*

In the screen shot below note the following:

1) The units have been converted to inches
2) The design has been scaled so that it is roughly 7.5”x9”x1”
3) The zero plane was then moved up so that parts of the design that are deeper than 0.5” will not be included.



Those portions that would *not* come in appear as a shadowed area. A few have been highlighted in yellow.

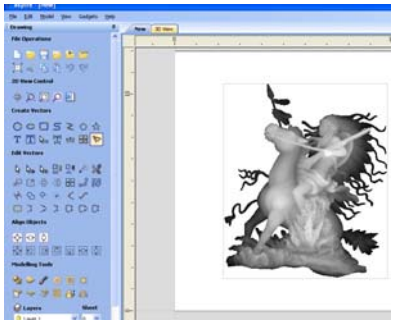
This technique is useful in retaining Z-depth detail of the main portion of the design but still reducing overall thickness of material needed for the carving.

For this example set the zero plane to **0.8”**

Click OK

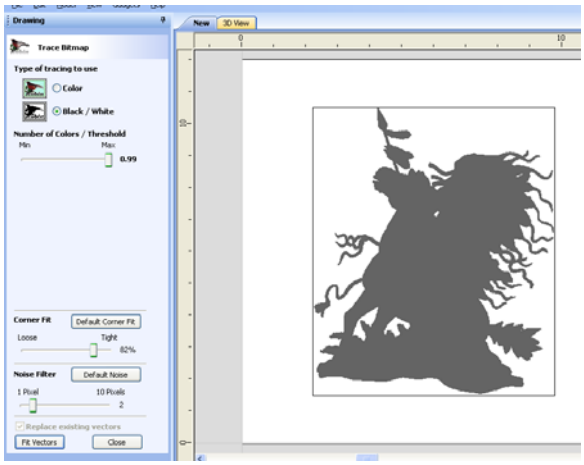
Click the **OK** button at the bottom of the screen when you are ready to have the model appear in the project.

Tracing Vectors Around the Design



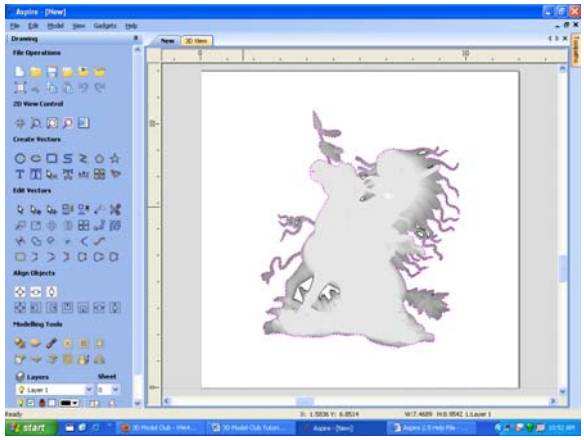
A useful step is to trace vectors around the design. These vectors can be used to constrain the toolpaths and can be offset to generate a scoop around the design.

With the 3D model selected *(it is **dark** when it is selected)* click on the icon under **Create Vectors** that does the **Fit Bitmap to Vectors** operation. The following screen appears:



Back on the main screen the vectors will appear as black lines next to the design.

When a vector is selected it will turn magenta colored.

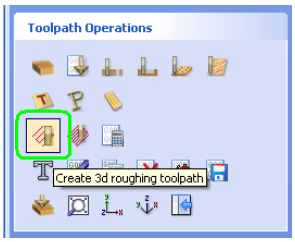


Carving a Design as an Onlay

If the design being carved is going to be an onlay then the next step *could* be to just toolpath it and carve it, but this would cause machine time to be longer than needed. A better approach is to select the outside vector that was created when the image was traced in the previous step.

With the outside vector selected press the **F12** key to activate the toolpath screen.

Roughing Pass



A roughing toolpath is usually done with an end mill to remove the bulk of the material in order to reduce stress on the finer ballnose bit that is will be used for the finish pass.

With the outer vector selected click on the Roughing Toolpath icon.

The Material Setup tab may display at this point. For now, just click **OK.**

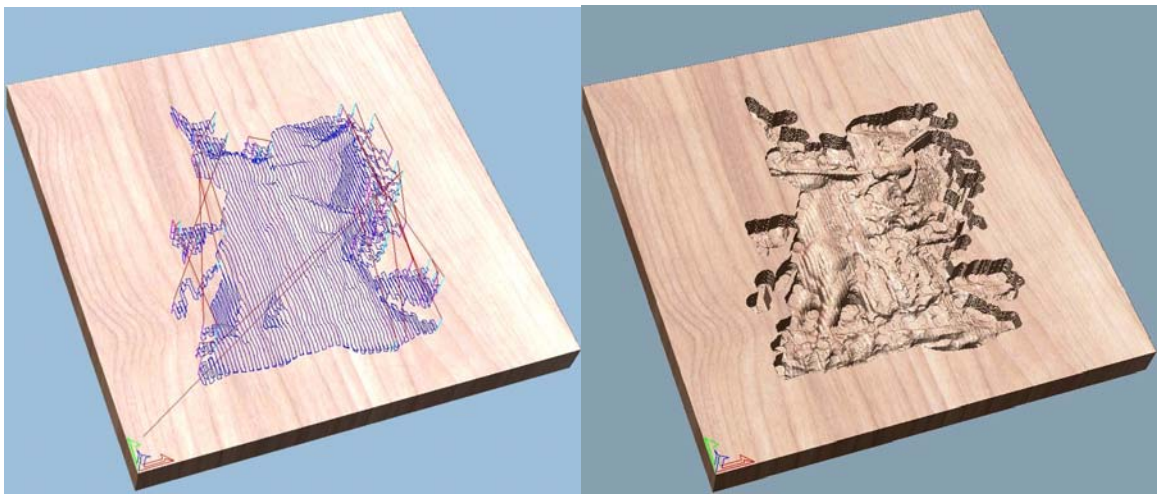
For this example select a 0.25” end mill.

Set the **Machining Allowance** to 0.025 *(i.e. about 10% of the tool diameter).*

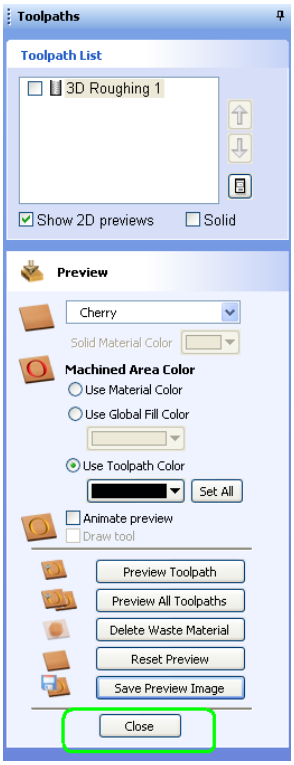
Select **3D Raster**. Whether you choose **Along X** or **Along Y** will depend on the direction the wood grain runs. *Your own experience will tell you if you want the toolpath to be with the grain or cross grain.* If you are machining material without a grain structure, such as HDU, then it really doesn’t matter which one you choose.

Click **Calculate**.

Aspire returns to the material preview but this time you will see blue lines indicating the travel of the end mill. Click **Preview Toolpath** to see what the design will look like after the roughing pass.



Click the **Close** button to return to the toolpath operations tab.



Note: As you become comfortable with 3D machining and how the CNC machine performs you *may* be able to skip the roughing pass. The sole purpose of the roughing pass is to reduce the load on the ballnose bit during the finish pass.

But if machine time is *not* an issue or you can't afford to break expensive ballnose bits then *always* do a roughing pass.

Finish Pass

The Finish Machining Toolpath is where the quality of the finished design is determined. There is a tradeoff: machine time vs. cut quality. The two items that control this are:

1) Bit size (a smaller bit provides better detail)
2) Stepover

Stepover

This determines how close together each pass of the bit is. The smaller the stepover the better the detail.

For example, consider someone mowing a golf course. They make on pass with the mower and then turn around and make the next pass, but they slightly overlap the first pass to ensure they don't miss part of the green. The more they overlap each pass the better the cut quality but it takes longer to complete cutting the golf course.

For 3D work a stepover of 8%-10% is typical. With a larger stepover you will begin to see prominent "raster lines" in your finished product. In some cases this is acceptable.

If you are doing a large design in HDU then you may wish to have the stepover as high as 15% because HDU sands fairly easily and the item may be mounted 20 feet off the ground and will not be viewed up close.

If you have an application where cut quality is critical then a stepover of 5% may be reasonable.

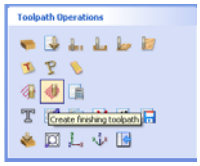
To help illustrate stepover's impact on machine time observe the following:

Stepover	**Machine Time**
5%	3 hours 13 minutes
10%	1 hour 37 minutes
15%	1 hour 6 minutes

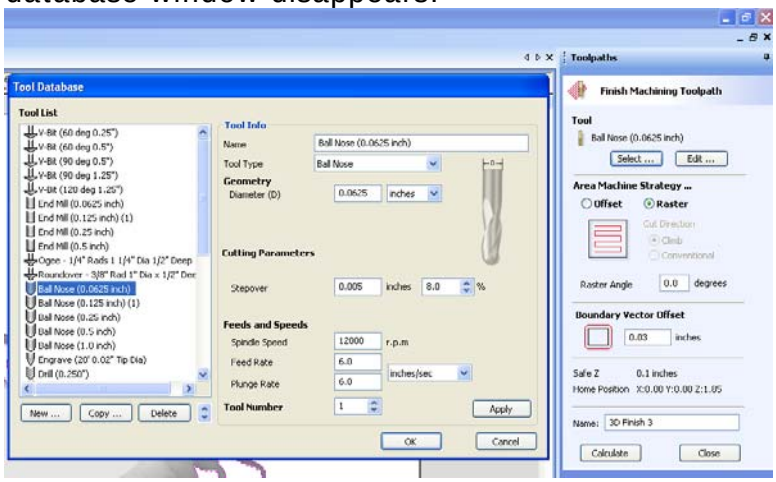
The above were calculated for my machine using a 0.0565" ballnose bit doing the design at 7.5"x9"x1".

For this exercise we will use a 10% stepover

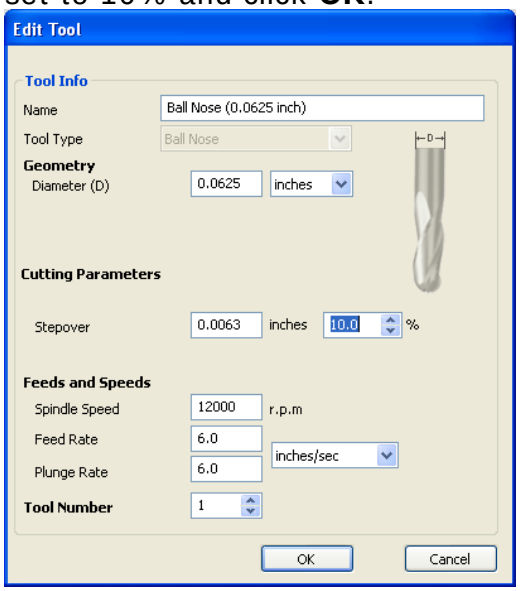
Click on the Finish Toolpath icon:



Click on the **Select** button and choose a 0.0625" ballnose and click **OK**. The Tool database window disappears.



In the finishing toolpath widow click the **Edit** button. Ensure that the stepover is set to 10% and click **OK**.



In the **Area Machine Strategy** choose **Raster**. With experience you will learn if you want the raster to go with the grain or cross grain.

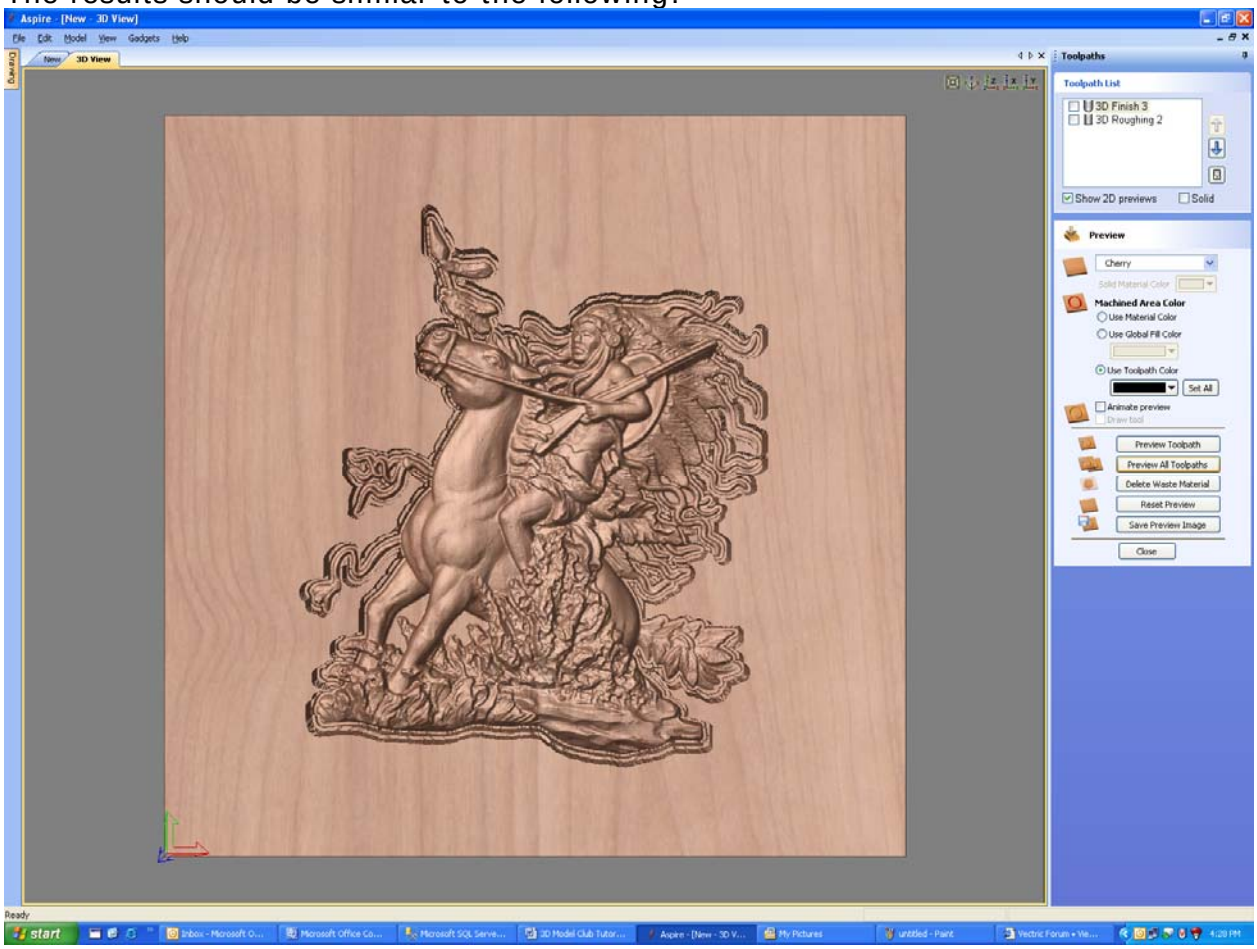
Set the Boundary Vector Offset to 0.03 (about half the bit diameter). This tells Aspire to machine slightly past the edge of your design to eliminate a rough edge.

Click **Calculate**.

Similar to the roughing pass, blue lines display for the finish toolpath.

Click on the **Preview** icon (if you aren't already in the previewer) and click **Reset Preview** and then **Preview All Toolpaths** to see what the finished result will look like.

The results should be similar to the following:



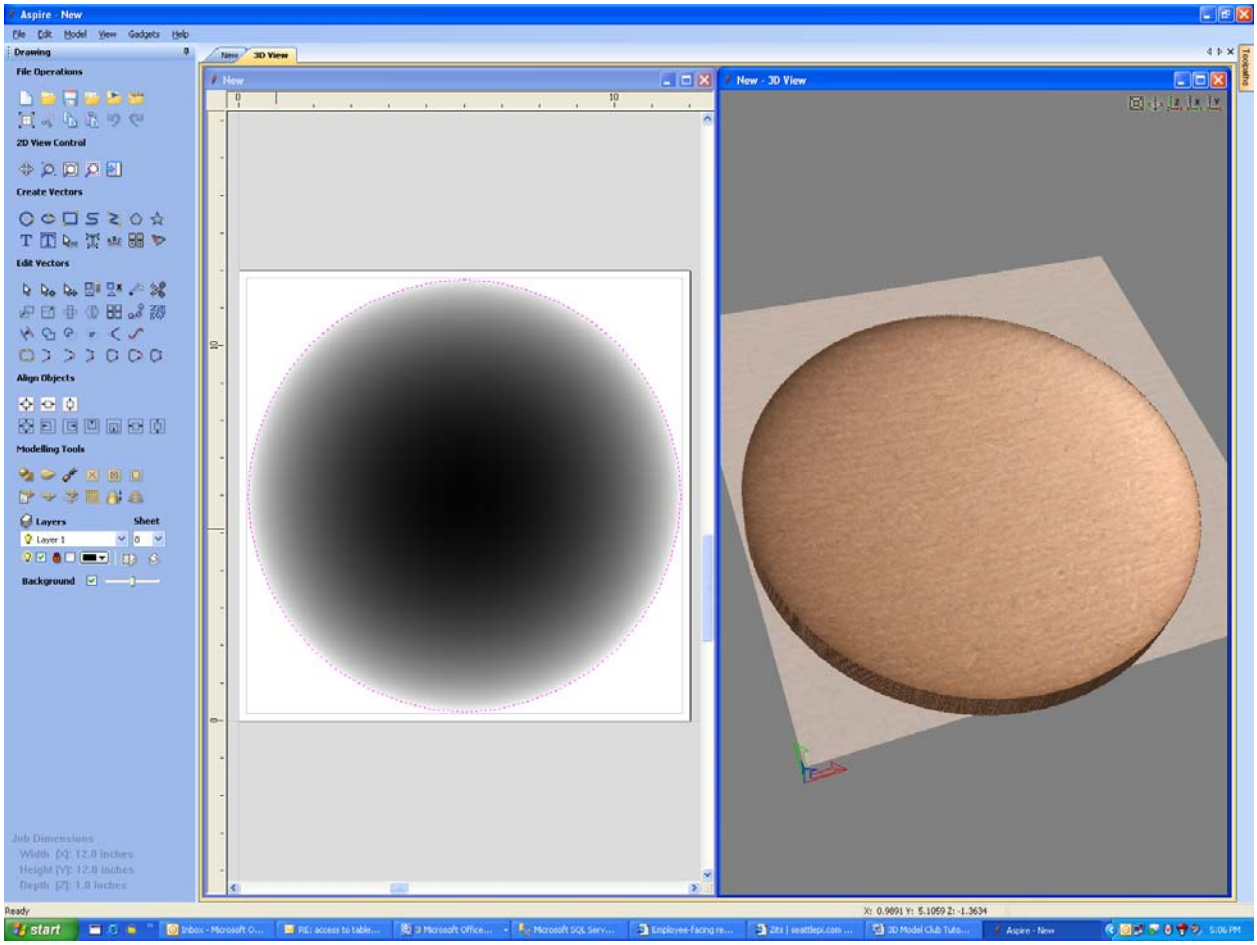
Cutout

Finally a cutout pass is needed.
Select the outside vector that was created earlier.
Choose the Profile toolpath icon.
The cut depth should match the thickness of the material.
Choose machine *outside* the vectors and click Calculate.
Preview all the toolpaths.
Choose Delete Waste Material to see the result.

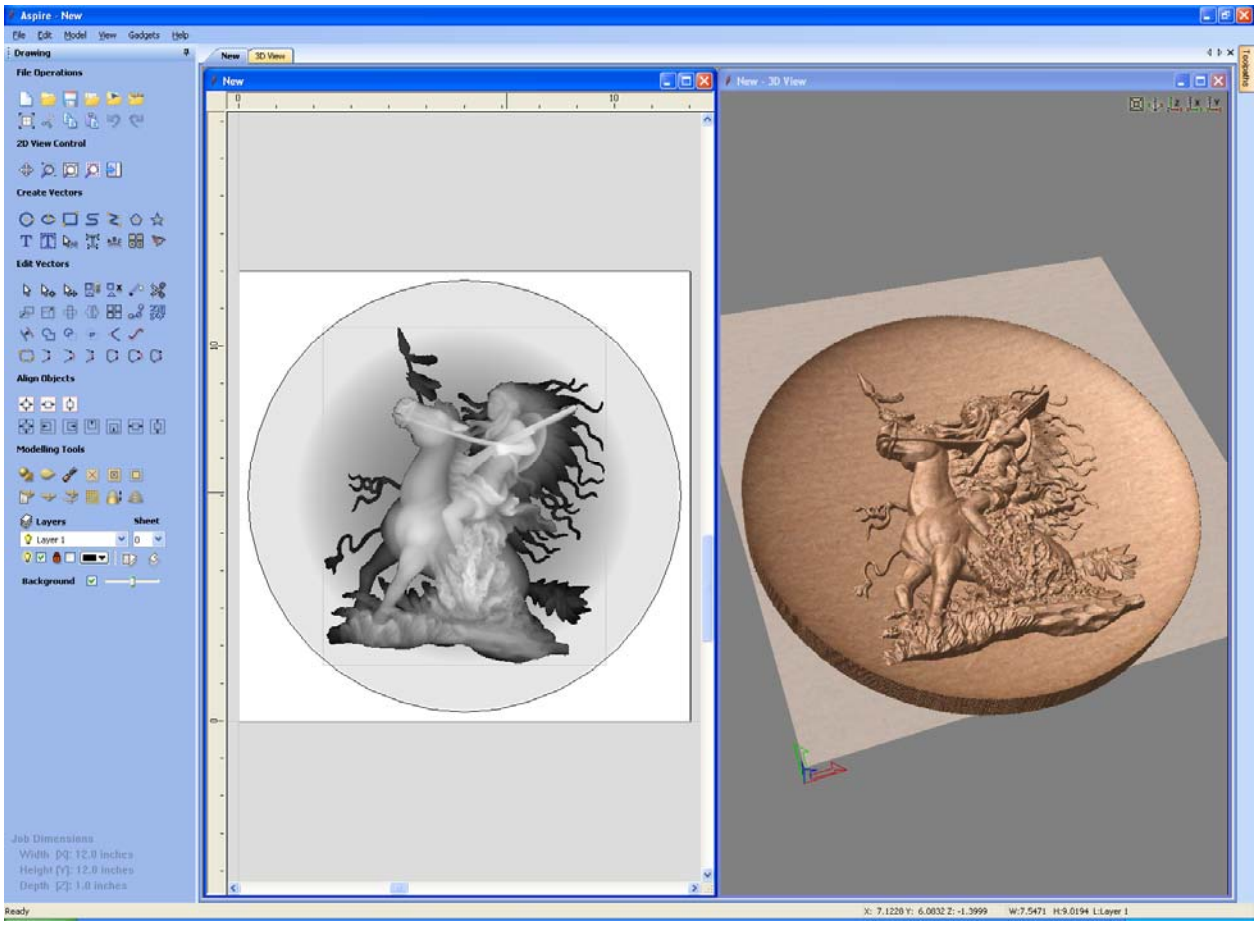
Dish Style

The dish style is similar to making the onlay except that first a dish is made and then the design is imported and laid on top of the dish design.

For this exercise create a circle that is 11.5” in diameter and center it in the job.
Choose **Create Shape From Vectors** under the **Modeling Tools**.
Choose a Dome style with an 80º angle
Set the **Base Height** to 1”
Select **Scale to Exact Height** and set it to 0.95”
Click **Subtract**
This creates a dish

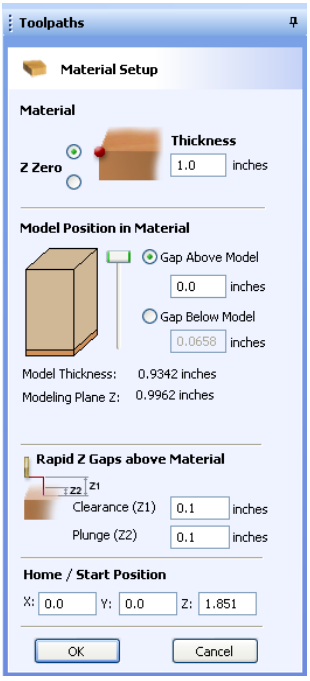


Import the STL file from 3D Model Club design just as you did in before.



Select the circle vector and press **F12** to switch to the toolpath pane.

The material setup box appears. The slider for Model Position in Material should be set to the top.



You can create a roughing toolpath if you wish but dish designs lend themselves to just being cut with a finish toolpath if you use a raster pattern for the finish toolpath.

Choose the Finish Toolpath and be sure to select **Raster**. This ensures that the tool starts cutting at the edge of the design where the cutting depth is quite shallow. A stepover of 8%to 10% works well for dish designs.

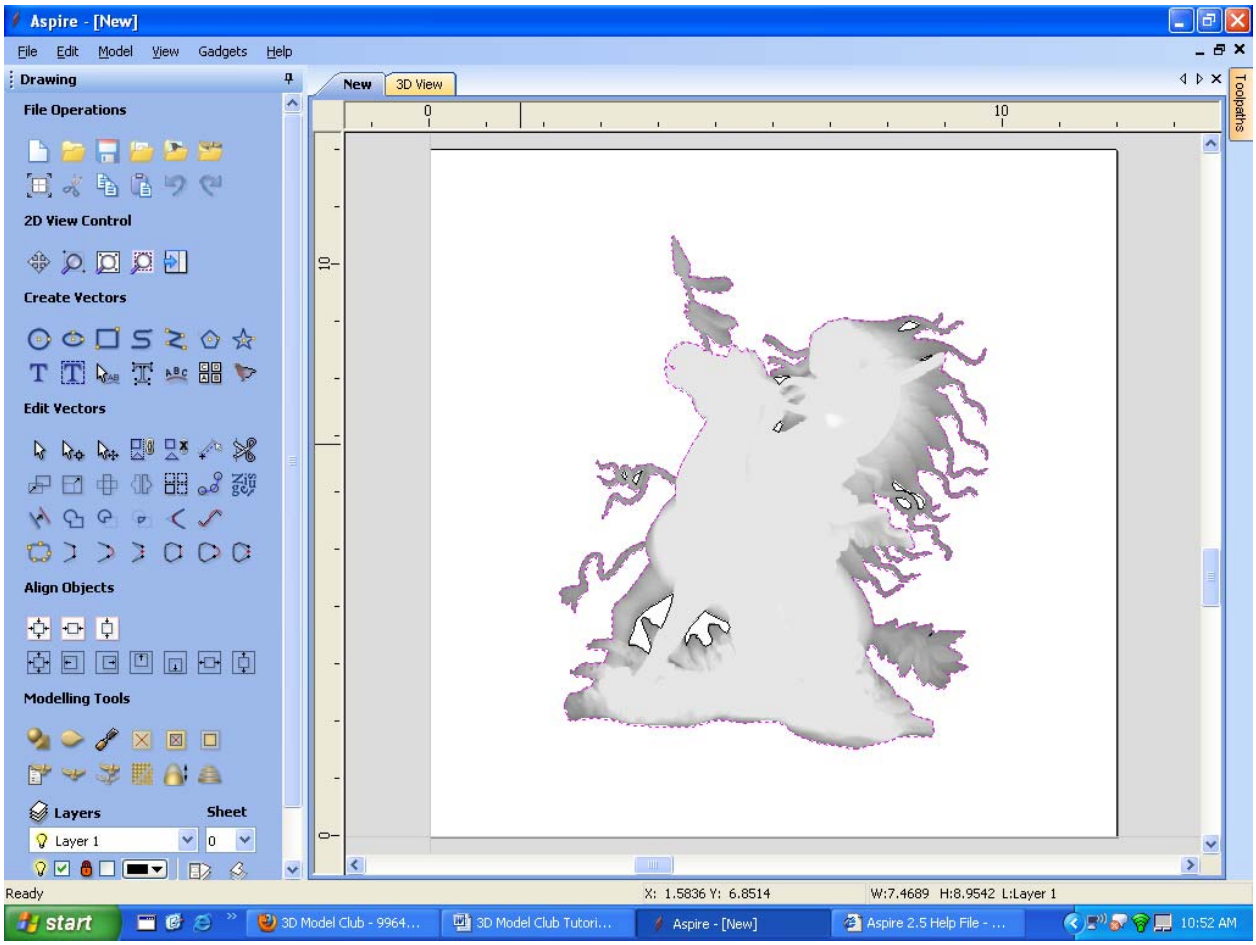
Click **Calculate**. (This can take a long time)

Preview the toolpath.

Scoop Around the Design

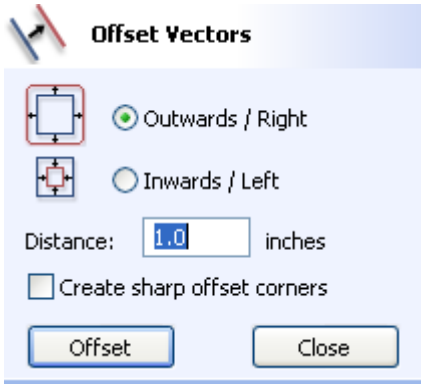
A scoop around the design lends to a hand-carved look. It is slightly more complicated to set up than a dish design. The following is a fast and simple method to generate a scoop in Aspire that compliments the design.

Create a new project
Import the STL file from 3D Model Club using the method documented previously.
Use the trace bitmap tool to create vectors around the design.
Click on one of the outside lines to select it.

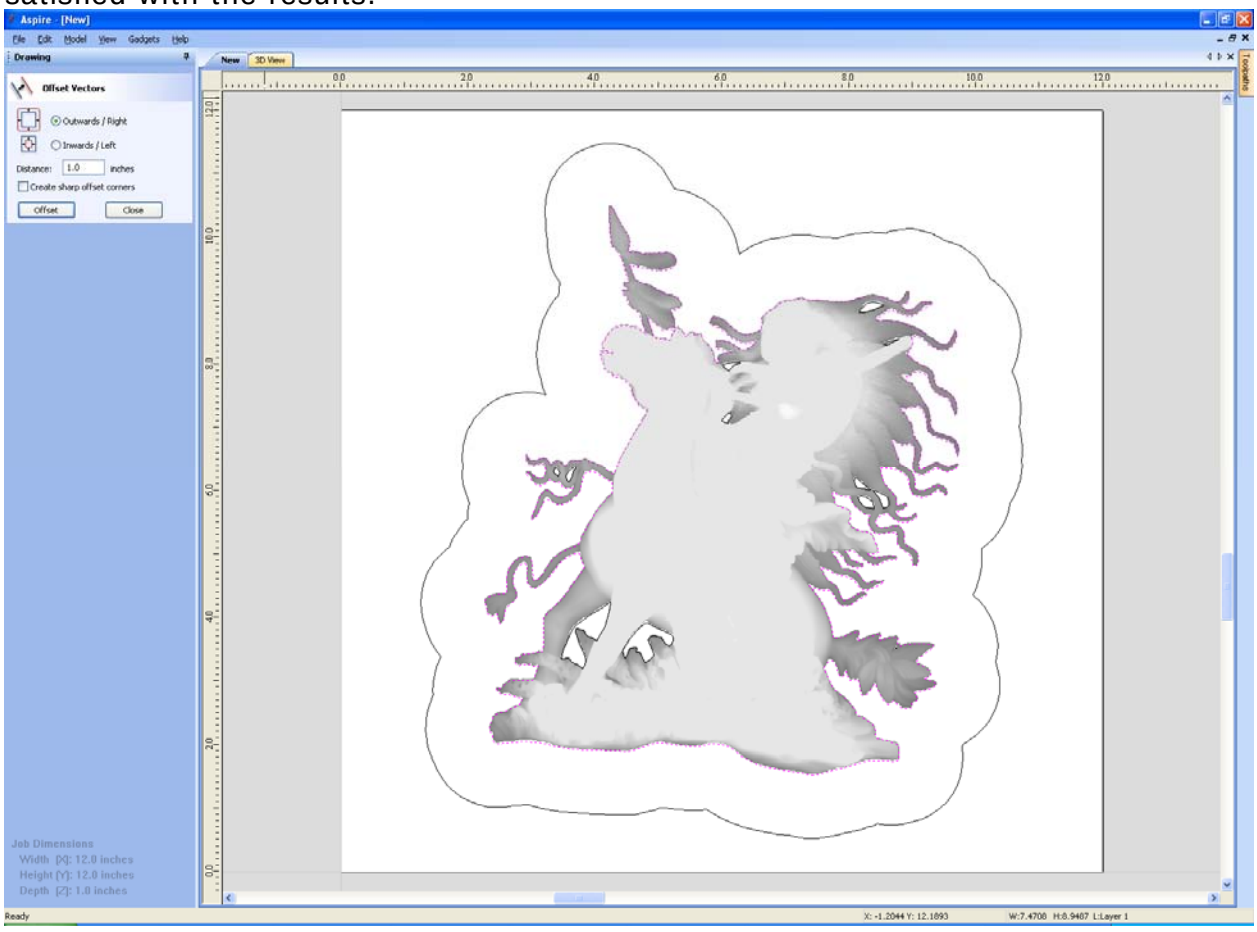


Next choose the **Offset selected vectors** tool:

Offset outward using a distance that makes sense for the project. This example uses 1 inch.



Uncheck **Create sharp offset corners** and click **Offset** and then **Close** if you are satisfied with the results.



Choose **Create Shape From Vectors** under the **Modeling Tools**.

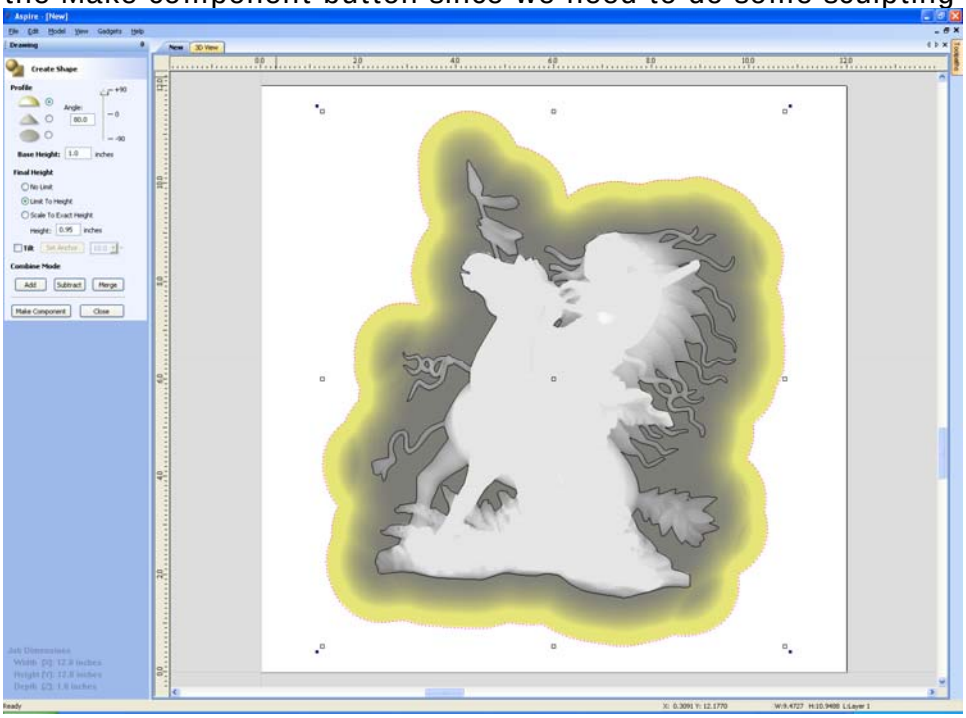
Choose a **Dome** style with an 80º angle

Set the **Base Height** to 0”

Select ***Limit*** **Scale to Exact Height** and set it to 0.95”

Click **Subtract**

This creates a scoop.
There are some interference lines that will need to be sculpted out. Do NOT click the Make component button since we need to do some sculpting first.



Choose **Close**

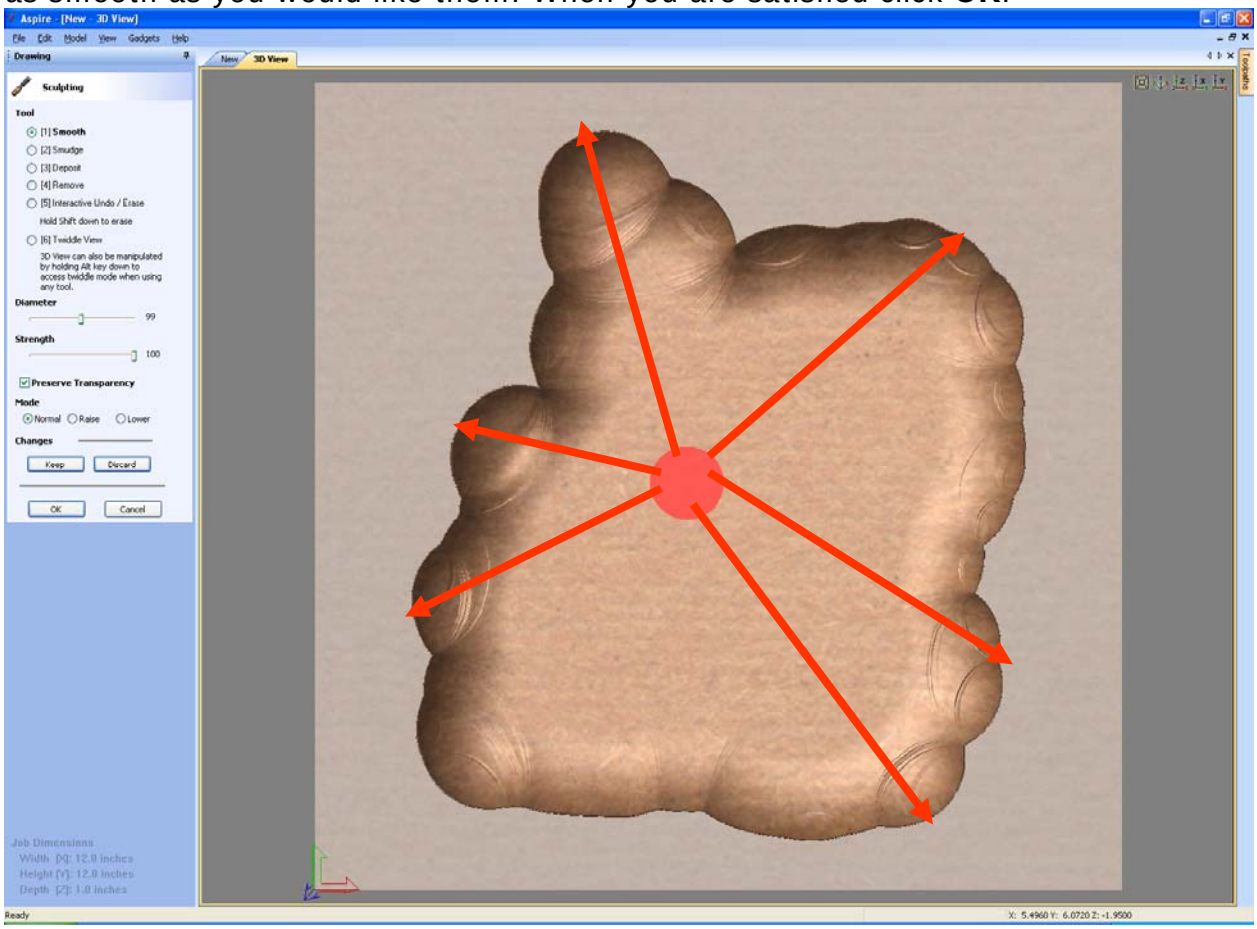
Press the **F8** button to display the component manager.
Uncheck the model of the Warrior. We are turning it off so we can sculpt the scoop design.

Click the **Hide** button on the component manager.

Sculpting

Click on the Sculpting icon in the Modeling Tools section:

When the sculpting window appears choose the **Smooth** tool.
Start with the **Diameter** set in the middle and the **Strength** set to maximum.
Ensure the **Preserve Transparency** is checked.
Position the mouse pointer (the red circle) in the center of the design.
While holding down the left mouse button drag the red circle over the rough areas until the circle reaches the edge of the scoop design. Repeat this until the areas are as smooth as you would like them. When you are satisfied click **OK**.



Check the height of the scoop using the **Scale Z height of Model** tool and adjust to 0.95” if needed.
Click the icon to **Make the Working Model Into a Component**.

Press **F8** to bring up the Component manager. The Scoop model will display, the original model is not visible because the
Click the check mark to turn on the model in the display.

In 2D mode select the outer vector.

Roughing Pass

With the outer vector selected click on the Roughing Toolpath icon.

The Material Setup tab may display at this point. Ensure that the slider is set to the top of the material**.**

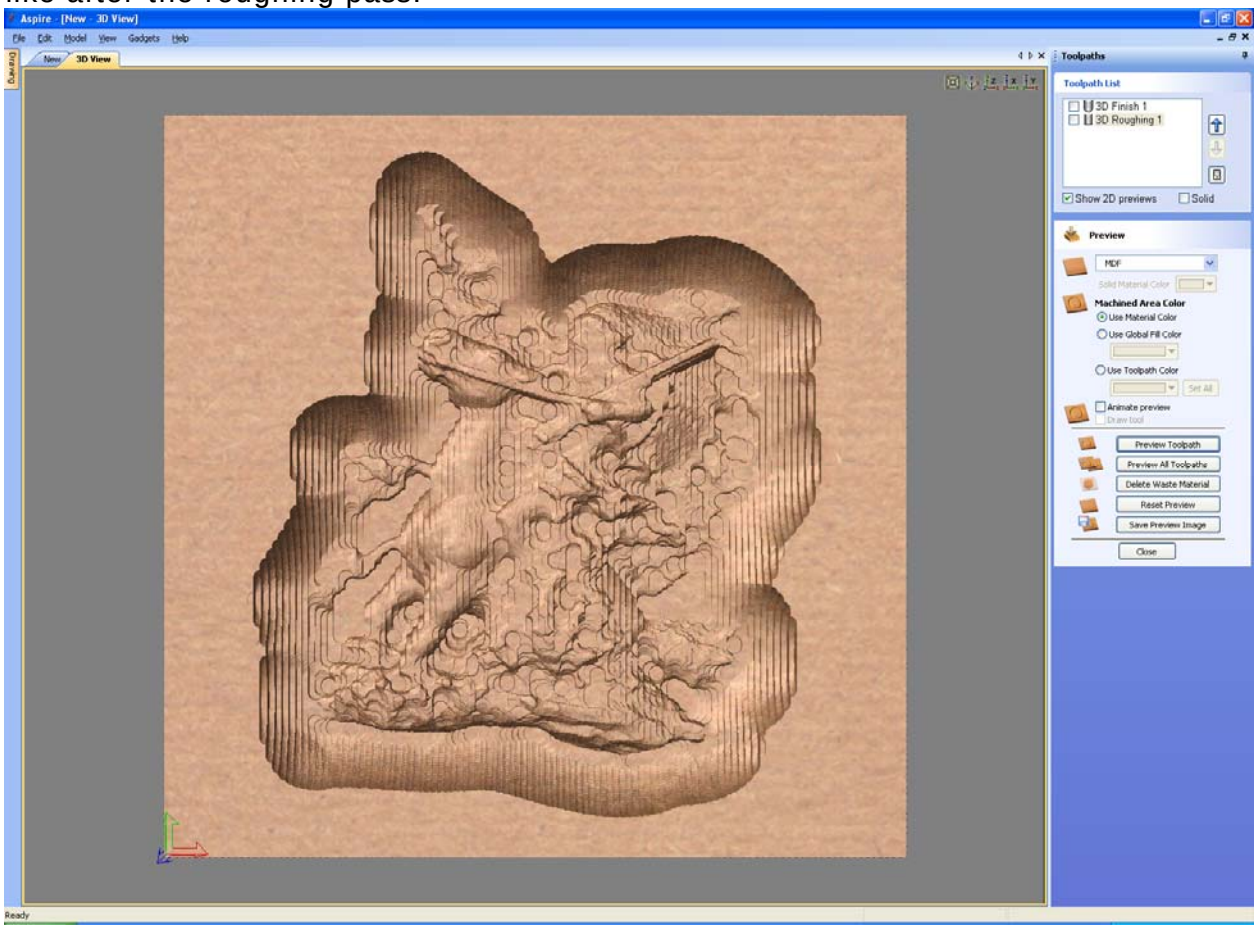
For this example select a 0.25” end mill.

Set the **Machining Allowance** to 0.025 *(i.e. about 10% of the tool diameter).*

Select **3D Raster**. Whether you choose **Along X** or **Along Y** will depend on the direction the wood grain runs.

Click **Calculate**.

Aspire returns to the material preview but this time you will see blue lines indicating the travel of the end mill. Click **Preview Toolpath** to see what the design will look like after the roughing pass.



Finish Pass

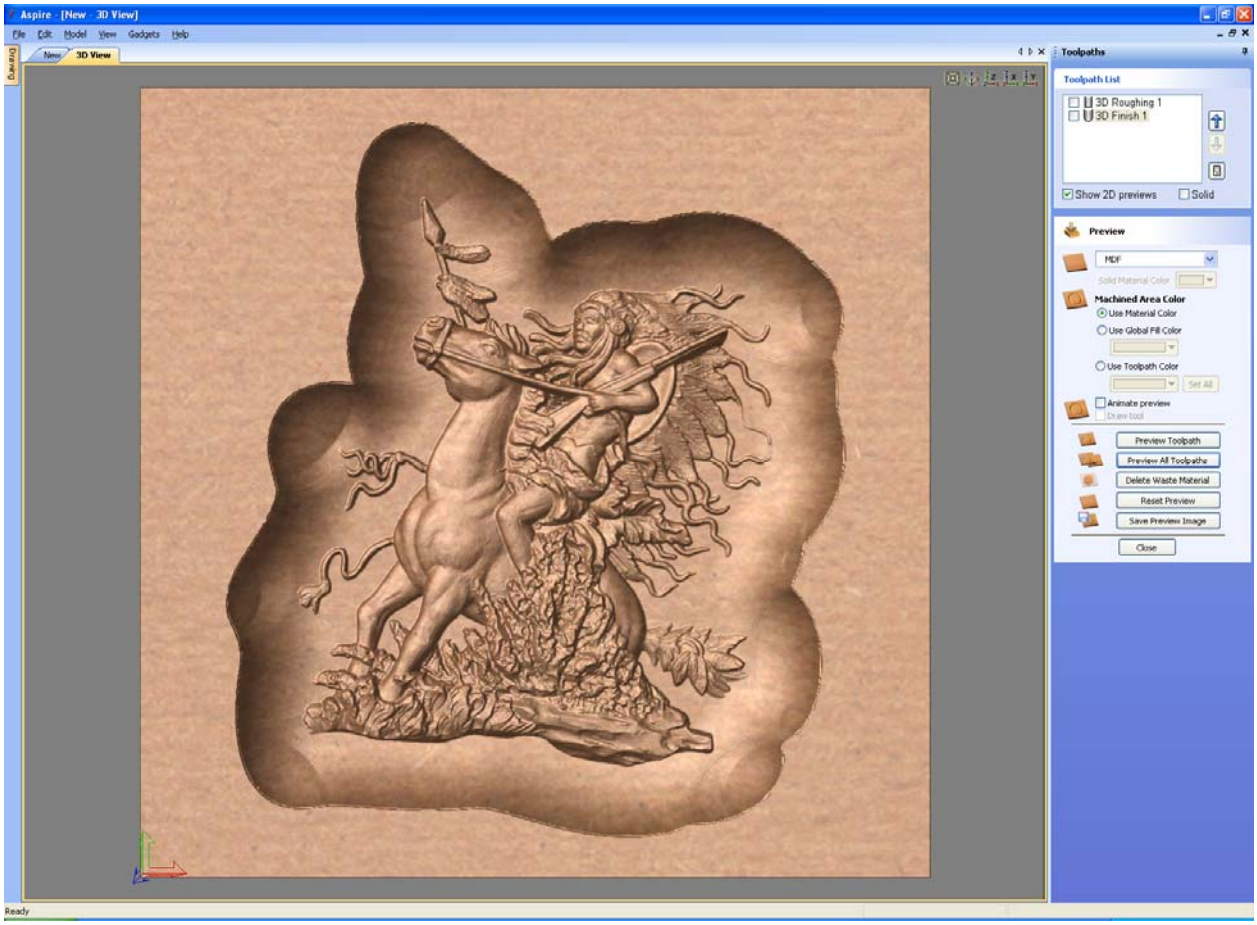
Click on the Finish Toolpath icon:

Click on the **Select** button and choose a 0.0625" ballnose and click **OK**. The Tool database window disappears.

In the finishing toolpath widow click the **Edit** button. Ensure that the stepover is set to 10% and click **OK**.

Set the Boundary Vector Offset to 0.0.

In the **Area Machine Strategy** choose **Raster**. Try machining with the grain. In materials such as HDU and MDF it doesn't matter which direction is chosen.



Click **Calculate**.

Click on the **Preview** icon (if you aren't already in the previewer) and click **Reset Preview** and then **Preview All Toolpaths** to see what the finished result will look like.

Sanding

Once you have machined the design you may need to do a bit of sanding. For 3D designs tools such as sanding mops and Dremel abrasive brushes and disks works well for detailed areas. Larger areas clean up well using a 3M ScotchBrite wheel mounted in an electric hand drill. A profile sander also works well.

Glazing

Glazing is a method of using a gel stain to darker the deeper areas of the carving to improve contrast. Briefly the process is:

1) Coat the material with a *light* coat of finish or sanding sealer
2) Apply and wipe off a gel stain.
    a. If it isn't dark enough then reapply the gel stain and let it sit longer.
    b. If it is too dark then sand off the excess.
    (Hint: try a lighter stain for learning the process)
3) Apply another coat of finish.

Tim Merrill has written an excellent tutorial on glazing that is available on the Vectric Forum (http://www.vectric.com/forum/viewtopic.php?f=28&t=6622).