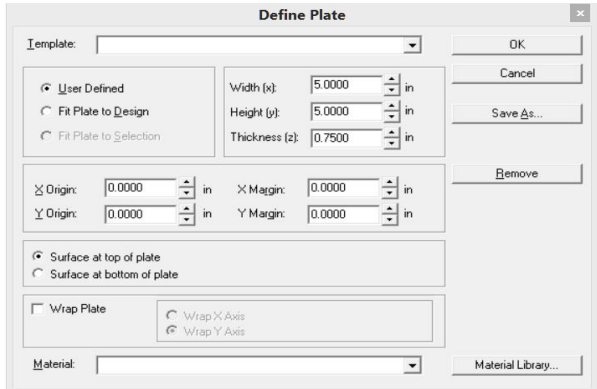
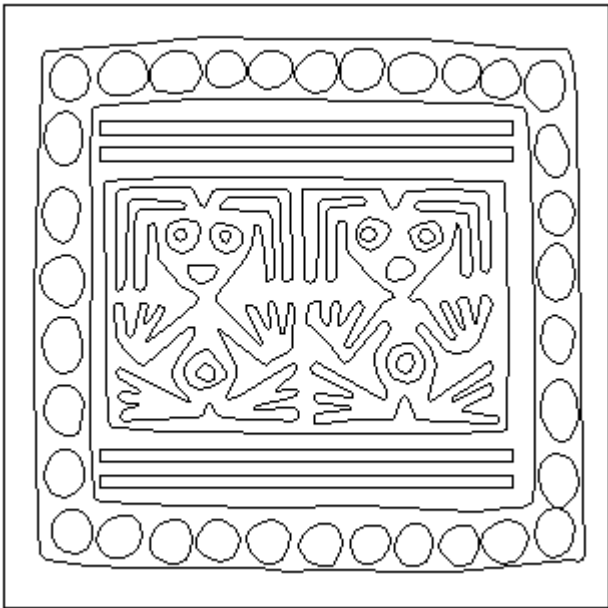


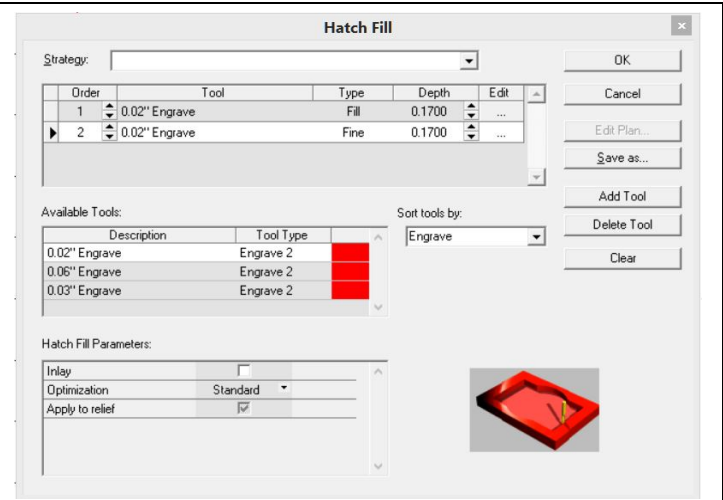
2_5 D Aztec Design

The inspiration for a design can come from almost anywhere. This was found in one of the Dover Clip Art books. This lesson is part of a series of lessons using the same design and EnRoute to create many different looks. Here we will explore the use of the Hatch Fill Strategy using an engraving tool and 3D toolpaths. The drawing tools were used to create the contours we will use to create the 2_5 D Celtic Knot.

<p>1. Define the Plate – Enter these parameters and click OK.</p> <p>Width 5.00</p> <p>Height 5.00</p> <p>Thickness .75</p> <p>Surface at the top of Plate.</p>	
<p>2. This is the design that we are using for this example.</p>	

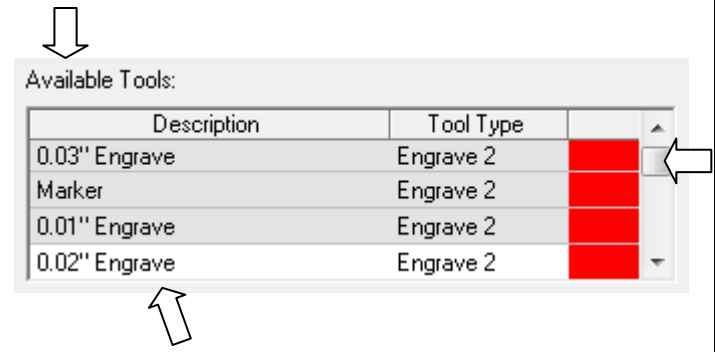
3. We want to create toolpaths that looks as if the piece has been carved. To do this we will use an engrave tool and the Hatch Fill Strategy with a 3D clean pass. The piece is small, so we will use the Engrave tool for the fill also.

Select all of the contours and click on the Hatch Fill Icon. This will open the Hatch Fill Dialog.



4. The next step is to select the tools that will be used to cut the material. In this example we have selected a .02 Engrave tool to cut the design. Go to the **Available Tools** area of the dialog and scroll down to locate the tools. Double click on Engrave tool twice to load it first as a Fill tool, and next as a clean tool.

Enter the **depth** of the cut by typing in the depth or using the arrows to select the depth desired.



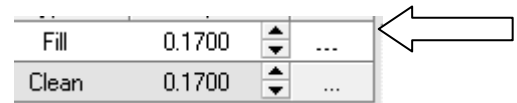
Fill Cut

Using the Hatch fill strategy, the fill cut will move back and forth across the object to the defined depth to mill the area. This tool is often a larger tool that will be defined to do most of the material removal. Additional parameters such as the angle of the cut and the overlap of the cut are defined in the cut parameters for the tool.

3D Clean Cut

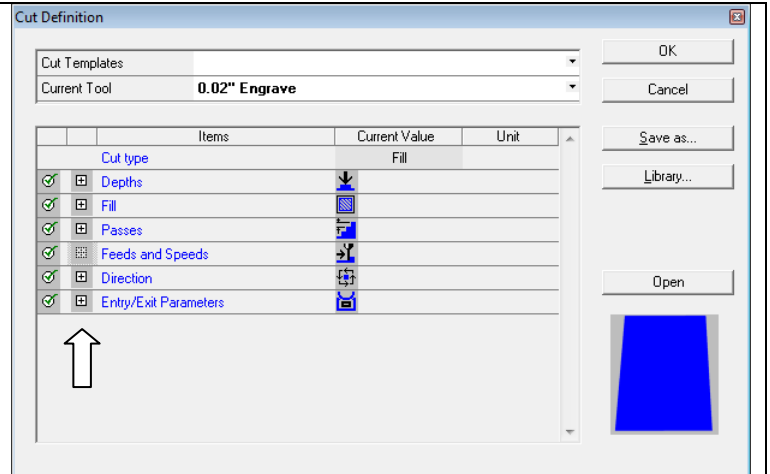
A 3D toolpath may be created using a conic, engraving or tapered tool. The difference is that it uses the beveled shape of the tool to create a beveled edge. It also will move up into corners to get as far into the corners as the tip dimension of the tool will allow. The 3D option is a very powerful option, allowing a very different type of cut from any of the toolpaths that are strictly 2D.

5. The next step is to edit the parameters of each of the tools. Move the cursor to the edit box next to the tool you wish to set the parameters for and left click. This will open the Cut Definition Dialog box for that specific tool.



There are several parameters that can be defined for each tool. The type of material that you are using will have influence in determining the parameters used. If you are using sign foam, it will cut a lot easier than wood or metal, so parameters can be set at a faster more aggressive pace.

More specific information can be found in the manual in the working with toolpaths section.



6. Enter the Parameters in the Cut Definition Dialog.
Click on the + box to open the Fill category.

Enter Parameters:

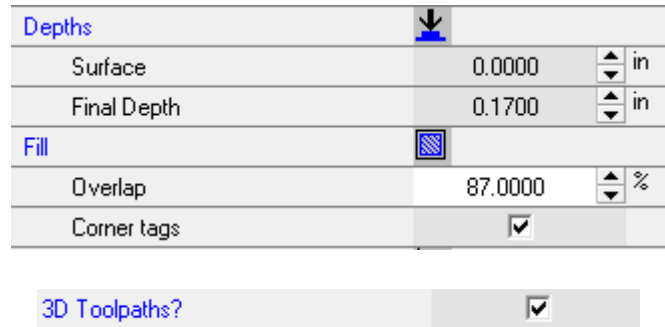
Overlap = 87%

Hatch Angle = 0

- Click on the + box to open the Passes category.

Enter Parameter:

Passes = 1



Overlap

This parameter defines how much the toolpaths will overlap each other in the fill. The default value that is created when the Fill Cut is created is 50 percent. Values can be set between 0 and 99 percent. Softer materials can be set at a lower percentage than the denser materials. The overcut percentage can affect the look of the finished cut of the material. When using a harder material it may be necessary to have a higher overlap to avoid the appearance of the toolpaths.

Passes

This is the number of passes used to cut the material to the assigned depth.

7. Click on the + box to open the Feeds and Speeds category.

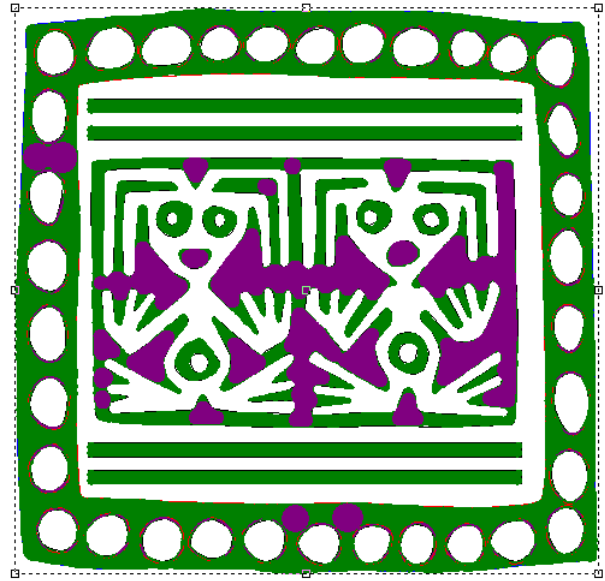
Enter Parameters:

Feed Rate = 100.0000

Plunge Rate = 50.0000

<p>Spindle = 14000</p>	
<p>8. Once you have entered these parameters, Click OK. This will bring you back to the Hatch Fill Dialog. Then click in the Edit box for the .02 Engrave Clean tool in the list.</p>	
<p>9. Enter the parameters for the Clean Cut. Passes =1 Feed Rate = 100.000 Plunge Rate = 50.0000 Spindle = 14000 Check the 3D Toolpaths box</p>	
<p>10. Step 12: Once you have entered the parameters for the clean pass. Click OK. Then Click Ok in the Hatch Fill Dialog. EnRoute will then calculate the toolpaths.</p> <p>This image shows the toolpaths. Purple = .02 Engrave Fill Cut Green = 1/8 End Mill Clean Cut</p>	

Press F9 to show the thickness of the toolpaths.
 Purple = .02 Engrave Fill Cut
 Green = .02 Engrave Clean Cut



Using the Simulate Ortho Tool allows you to see a rendered image of the toolpaths that you have applied. This is often a good way to see a preview of the toolpaths. If there are any noticeable errors you can then correct them before you actually cut your piece.

This is also a good way to set the cut order of the toolpaths. The simulation allows you to see the progress of the toolpaths in several different ways.



11. Create another layer.
 Paste a copy of the outline contour to that layer.
12. Select the contour and apply the Routing Offset toolpath.

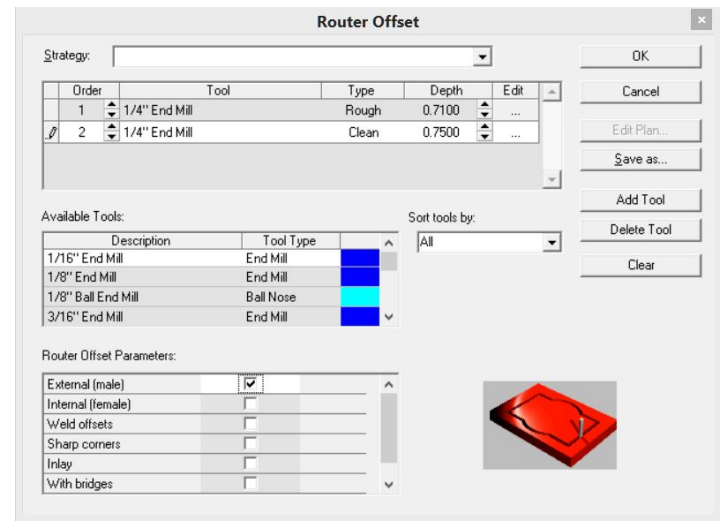


13. A Routing Offset was used to cut the relief out. Select the relief.

Click on the Routing Offset Icon. This will open the Routing Offset Dialog.

Load the 1/4 End Mill tool by selecting it from the Available Tools section of the dialog. Scroll down to locate the tool and then double click on it to load it. In this example, we have used the 1/4 End Mill tool as the Rough cut and the Clean cut, so you need to load the tool twice.

Enter the Depth of cut. For the Rough tool the Depth is .71. For the Clean cut set the Depth at .75.



14. Click in the edit box next to the Rough tool. This will open the Cut Definition Dialog for this tool.

15. Enter the parameters for the Rough tool:
 Passes = 3

Feed Rate = 100.0000

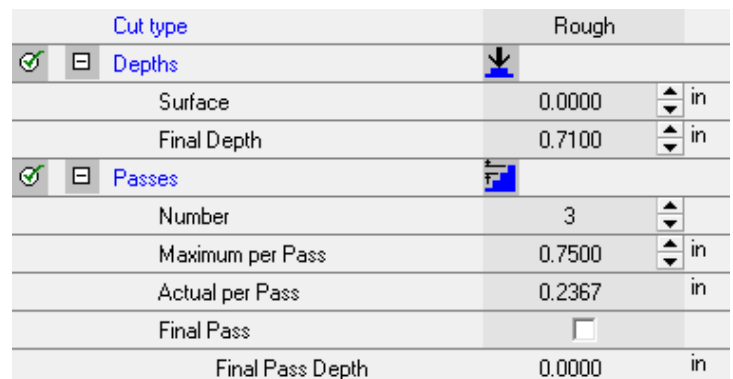
Plunge Rate = 50.0000

Spindle Speed = 14000

Click OK. This will bring you back to the Routing Offset Dialog.

Set the parameters for the Clean Tool.

Click in the Edit box for the Clean Tool. This will open the Cut Definition Dialog.



16. Enter these parameters.

Passes =1

Width of cut =.02

Feed Rate = 100.0000

Plunge Rate = 50.0000

Spindle Speed = 14000

Click Ok.

Click Ok again in the Routing Offset Dialog to process the toolpaths.

