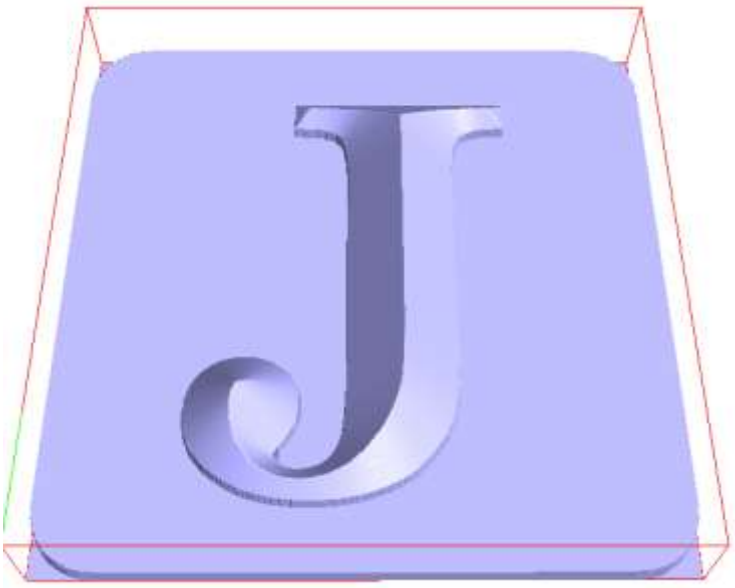
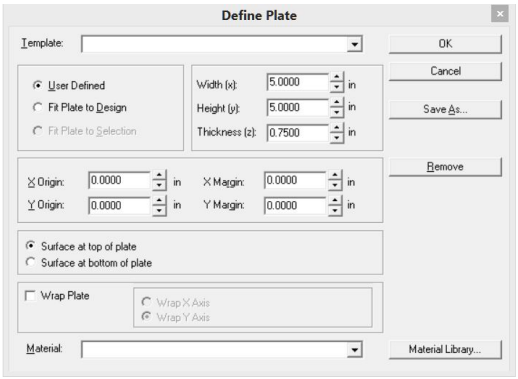
Pyramid

Each of the tutorials in the EnRoute Step-by-Step Series provides instructions for creating a piece using a single concept. Our goal is to provide examples that are relatively easy to follow and to accomplish, and that demonstrate an interesting element of EnRoute. We keep the design elements quite simple so that the focus is on the concept and not the design.



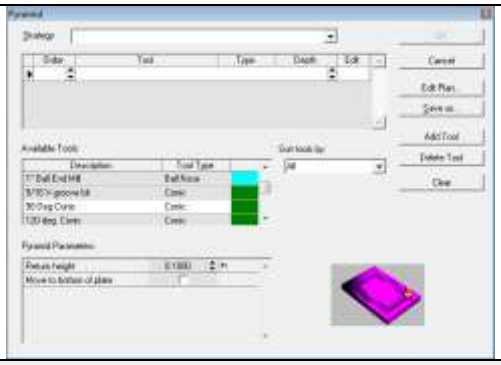
The Pyramid Strategy allows you to create a beveled type of artwork that is also commonly referred to as chiseled or sculpted artwork. The Pyramid Cut utilizes a conic tool to create a beveled effect. This cut is unique in that it is necessary for the software to calculate the depth of cut based on the selected tool, and the selected contours. The cut depth is based on the widest 'stroke' in the design. The software then places toolpaths at the proper depth based on the maximum cut depth it calculates.

1. Define the Plate – Enter these parameters and click OK. Width 5.00 Height 5.00 Thickness .75 Surface at the top of Plate. We will first be working with the face panel of the design that is stencil cut through the material.	
2. This is the artwork that we are using for this example.	

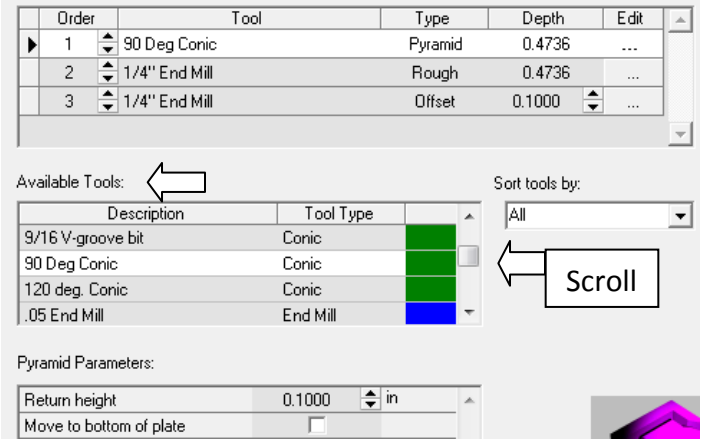
3. The Pyramid Strategy is used apply the bevel cuts to the artwork.

   Select the letter contour and then click the Pyramid Icon. 

   This will open the Pyramid 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 90 conic tool and ¼”End Mill.



Scroll down in the **Available Tools** section and double click the **90 degree Conic** tool for the bevel tool.

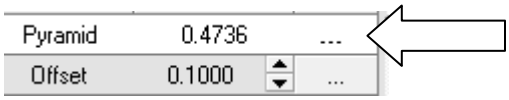
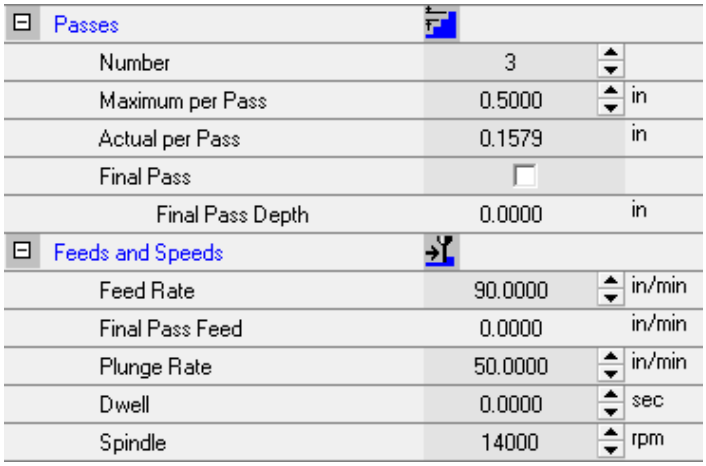
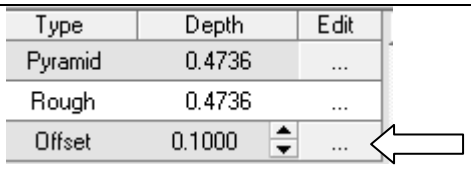
EnRoute automatically calculates the depth of the cut. This depth is determined by the size of the object and the tool selected to make the cuts. This will set it up so that the highest point of the pyramid will be at the surface of the material. To control where it is placed in the material, use the **Return Height** and **Move to bottom of Plate** parameters. When you select the Move to bottom of Plate parameter, EnRoute will place the lowest pass of the Pyramid cut at the bottom of the defined plate. If the Move to bottom of plate is not checked, the peak of the Pyramid on the widest stroke will be at the top of the material.

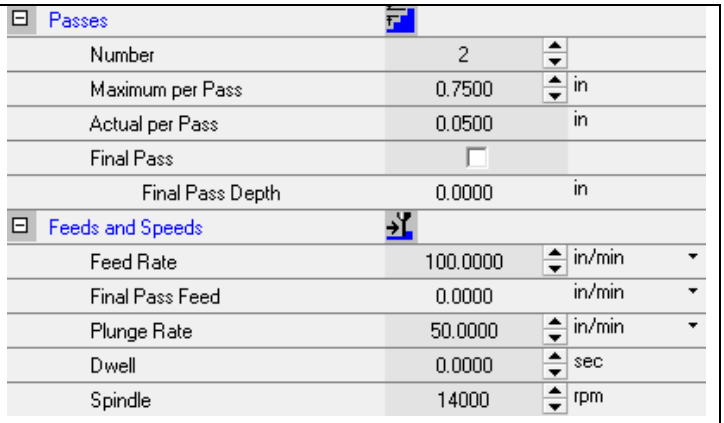
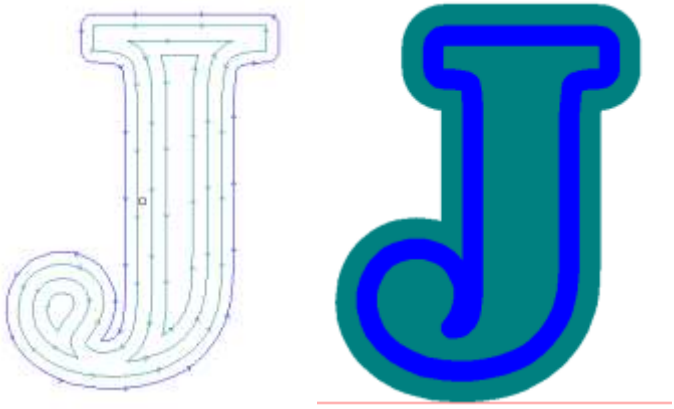
Set the Return Height Parameter at .1 inches.

Then next step is to select the ¼” End Mill for the Rough tool and Offset tool.

Scroll down in the **Available Tools** section and double click on the **¼ End Mill**. EnRoute determines that you are adding a roughing tool and it calculates the depth to be the same as the pyramid tool. It also calculates the number of passes to match the pyramid tool. This will ensure that the roughing tool will clear enough material away so the conic tool will have room to work. The conic tool could be damaged if we don’t do this.

Repeat to load the tool for the Offset cut. The Offset cut will cut around the parameter of the Pyramid cut at the assigned Depth.

5. Click in the Edit box next to the tool to open the Cut Definition Dialog.	
6. Click on the + box to open each area of the Cut Definition Dialog.	
7. Enter these parameters: Passes = 3 Feed Rate: 90.0000 Plunge Rate: 50.0000 Spindle Speed 14000 The cut parameters will vary depending on the type of material that you are cutting. A good rule is that the harder your material, the more passes you want to have. This will eliminate possible damage to your cutting tool.	
8. Enter the parameters and Click Ok. This will return you to Pyramid Dialog. Click in the Edit box next to the ¼" Offset tool. This will open the Cut Definition Dialog.	

9. Enter Parameters: Passes = 2 Feed Rate = 100.0000 Plunge Rate = 50.0000 Spindle = 14000	
10. Once you have entered all of the parameters, Click OK in the Cut Definition Dialog. Click Ok again in the Pyramid Strategy Dialog. The toolpaths will then be processed. This image shows the toolpaths that you have created. The second image shows the thickness of the tool assigned. Press F9. The teal color represents the conic tool and the blue represents the 1/4” tool.	

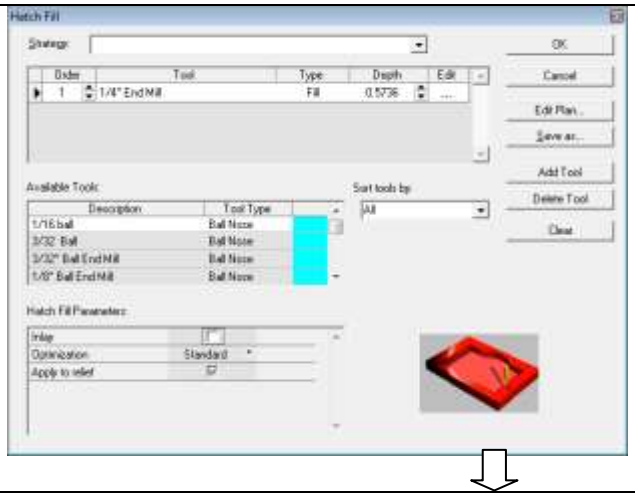
If you were cutting the letter out individually, you would move the letter to the bottom of the plate and set the return height to the height that you have determined for the sides of the letter. In this example, we are placing the pyramid letter on a background that will be milled down. We will use the Hatch Fill Strategy to mill the background.

Hatch Fill

The Hatch Fill strategy is used to create toolpaths to mill a surface down to a defined depth using toolpaths that move back and forth across the area to be milled. In this example the background of the letter is milled down so that the letter stands out from the base.

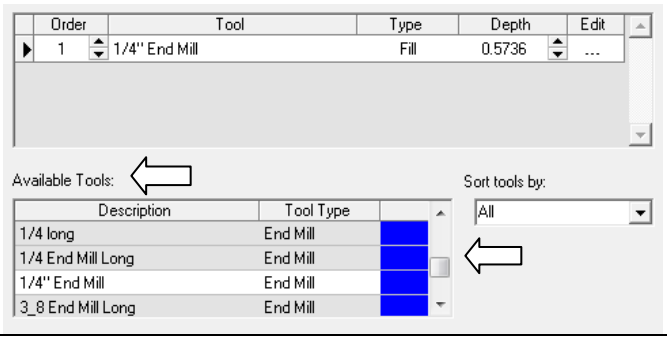
To mill down the background area that the letter is on, select the letter contour and the outside contour and click on the Hatch Fill Icon. This will open the Hatch Fill Dialog  .

A ¼" End Mill tool is used to mill this area.



11. Select the ¼" End Mill tool.
    Go to the available tools section and scroll down until you find the ¼" End Mill. Double click on the tool to select it.

    Enter in the Depth of .5736. This is the depth of the Pyramid cut plus the return height of .10.



12. Click in the Edit box next to the ¼" tool to open the Cut Definition Dialog.

13. Enter in these parameters:
    Overlap = 85%

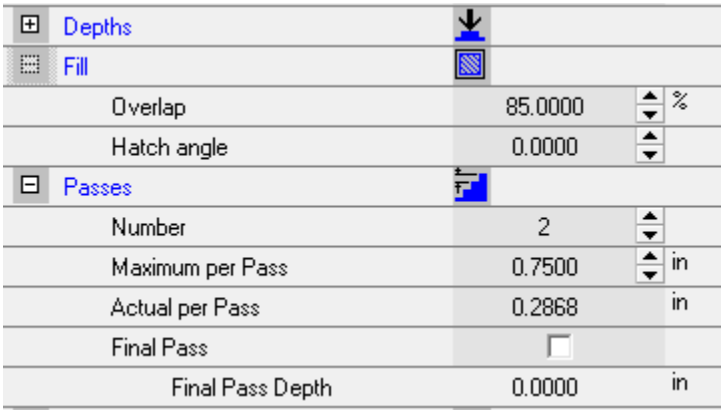
    Passes = 2

    Feed Rate: 100.0000

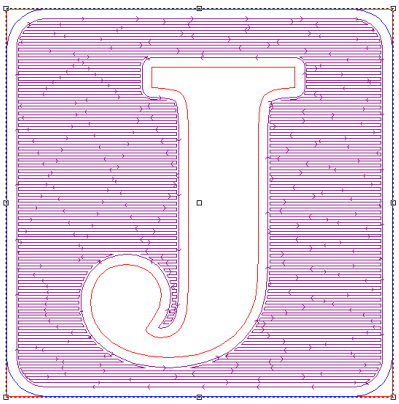
    Plunge Rate: 50.0000

    Spindle Speed 14000

    Click Ok to accept these parameters for this tool.



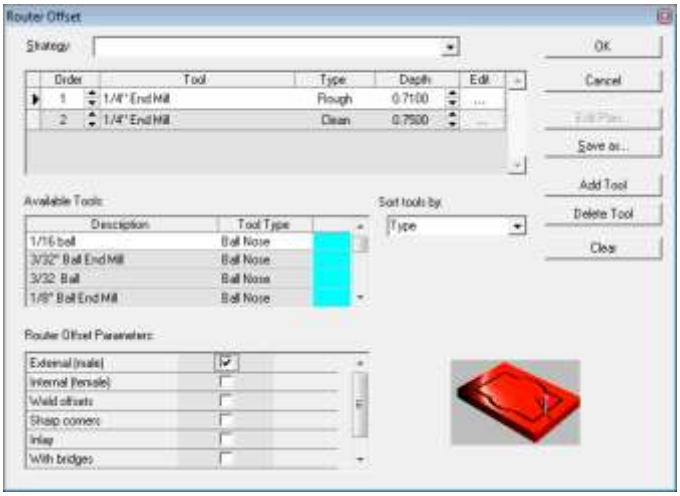
14. This will bring you back to the Hatch Fill Dialog. Click Ok in the Hatch Fill Dialog to process the toolpaths.



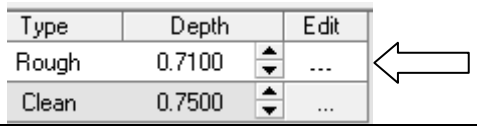
15. We now want to cut out the sample piece. Use the Routing Offset Strategy to do this.

    Click on the Routing Offset Icon. This will open the Routing Offset Dialog.
    Load the ¼ 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 ¼ 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.



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



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

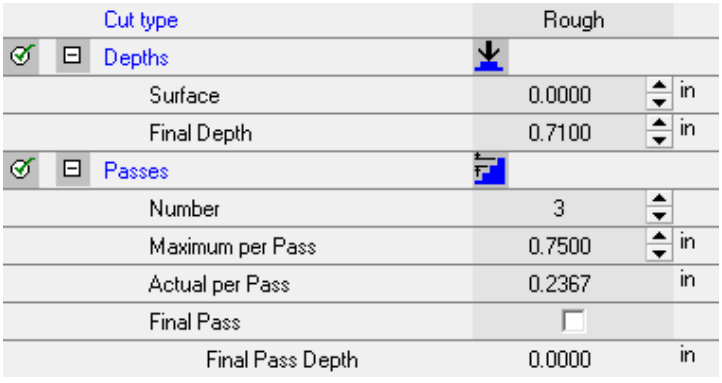
    Feed Rate = 100.0000

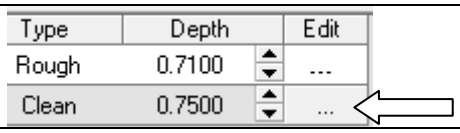
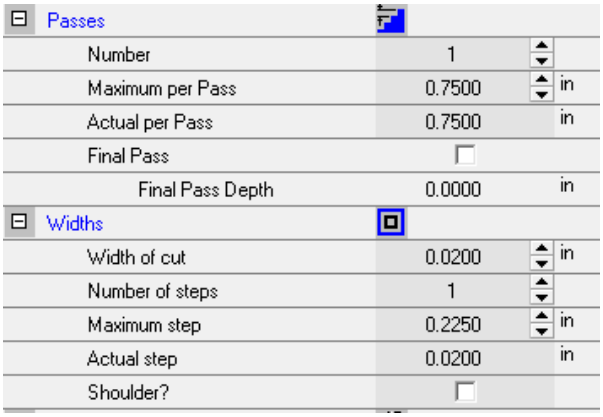
    Plunge Rate = 50.0000

    Spindle Speed = 14000

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

    Set the parameters for the Clean Tool.



18. Click in the Edit box for the Clean Tool. This will open the Cut Definition Dialog.	
19. 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.	
20. This is a rendered view of all of the toolpaths.