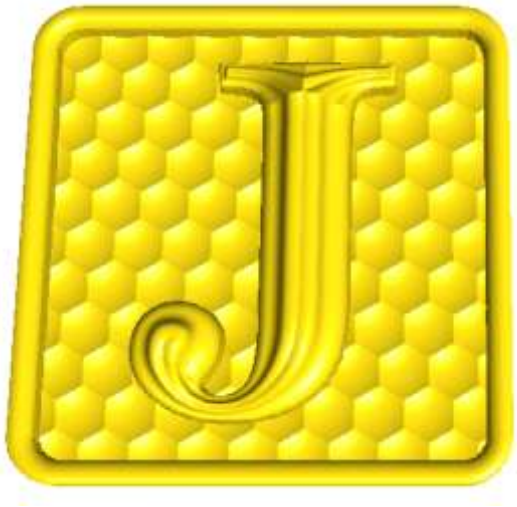
Chamfer with Texture

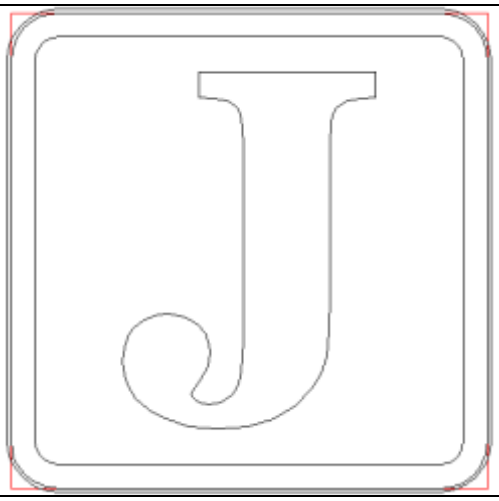
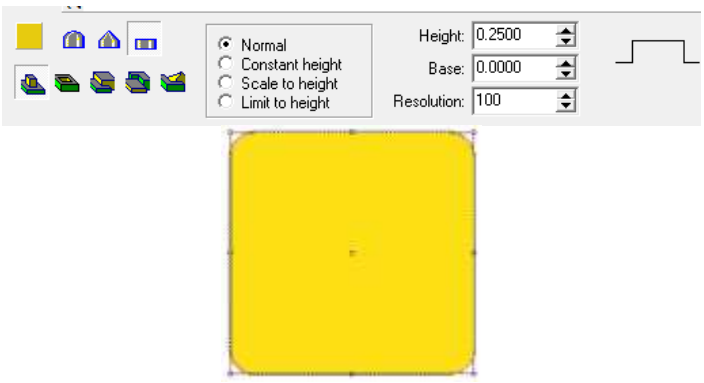
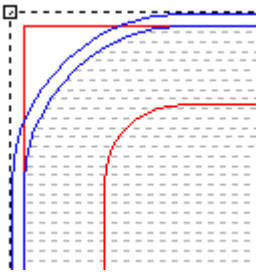
Each of the tutorials in the EnRoute Step-by-Step Series provides instructions for creating a piece using a single concept. Our goal is to provide examples that are relatively easy to follow and to accomplish, and that demonstrate an interesting element of EnRoute. We keep the design elements quite simple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

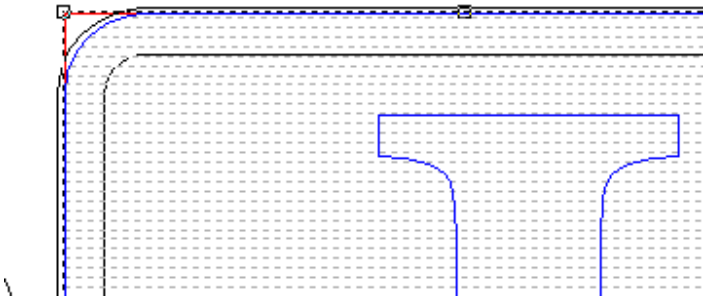
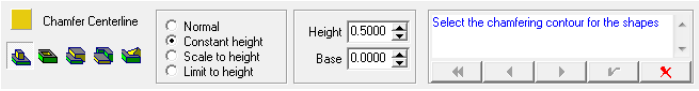
The Chamfer tools allow you to modify reliefs by applying profiles that can range anywhere from a simple rounded edge all the way to an intricate profile that changes the whole relief. The Chamfer works by first adding a height to a relief that is equal to the height of the profile, and then removing material around the perimeter of the relief that is the shape of the profile.

In the following lesson, the Chamfer Centerline tool is demonstrated and then the texture tool is also used to show just how easy it is to be creative using the EnRoute 4 tools.

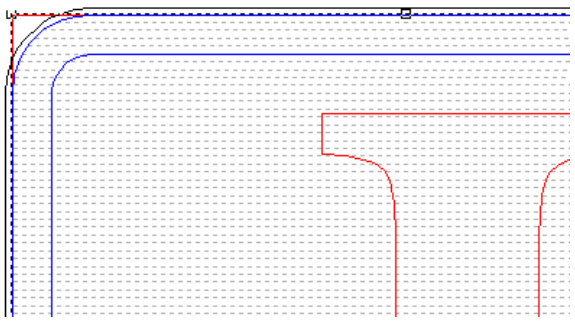


1. Define the Plate – Enter these parameters and click OK. Width 5.00 Height 5.00 Thickness .75 Surface at the top of Plate.	

2. This is the artwork that we are using for this example. We will create a relief using the Chamfer tools and the Texture tools. Notice that we have used the Offset Tool to create an offset for the outline of the design.	
3. Start with a flat relief with a height of .25 to create the base of the design. Enter Parameters: Add Flat Normal Height = .25 Click Apply.	
4. Add the rounded border to the design. Select the Offset contour, the Flat relief and the inside contour. Click on the Add Relief Icon. 	
5. Select Parameters: Add Rounded Normal Angle = 75	

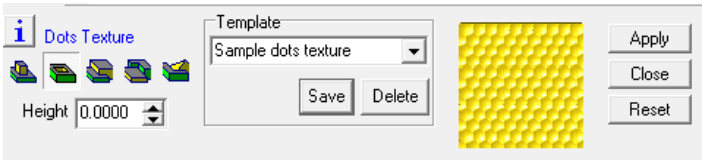
6. By selecting the outside offset along with the relief and the inside offset, the rounded portion is only added to the border area.	
7. The next step is to apply a profile to the letter using the Chamfer Centerline tool. To do this, we need to create the profile to be applied. Use the drawing tools to create the contour that you wish to use with the chamfer tool.	
8. Use the Chamfer Centerline tool to add a contour to the letter. Select the Relief. Select the Letter Contour. Click on the Chamfer Icon.  This will open the Chamfer Centerline Dialog.	
9. Enter parameters: Add Constant Height Height = .5 Select the contour that you created. Click on the Green arrow.	
10. This is the results of the Chamfer Centerline.	

11. Next, we will apply a texture to the background area. To do this we will use the letter contour as a mask so that the texture is only applied to the background area.
    Select the Relief.
    Select the inside offset contour.
    Select the letter contour.



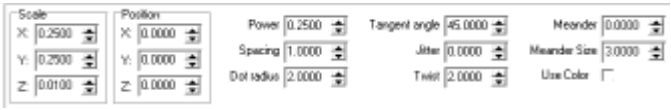
12. Click and hold the Basic Relief Icon to open the Texture Flyout Menu.

13. Click on the Dots Texture Icon  to open the Dots Texture Dialog.
    Click on the Subtract Icon.

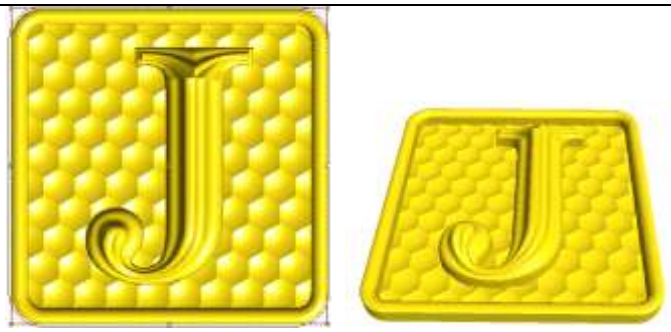


14. Enter these parameters:
    Scale X = .25
    Scale Y = .25
    Scale Z = .01
    Position = 0
    Power = .25

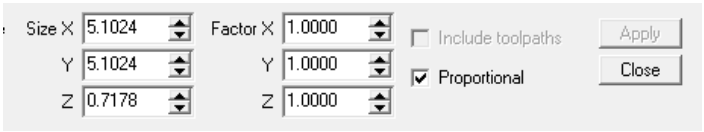
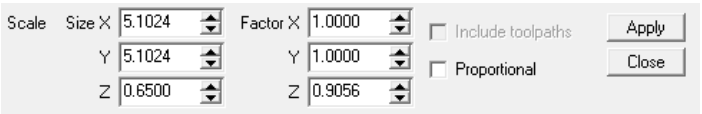
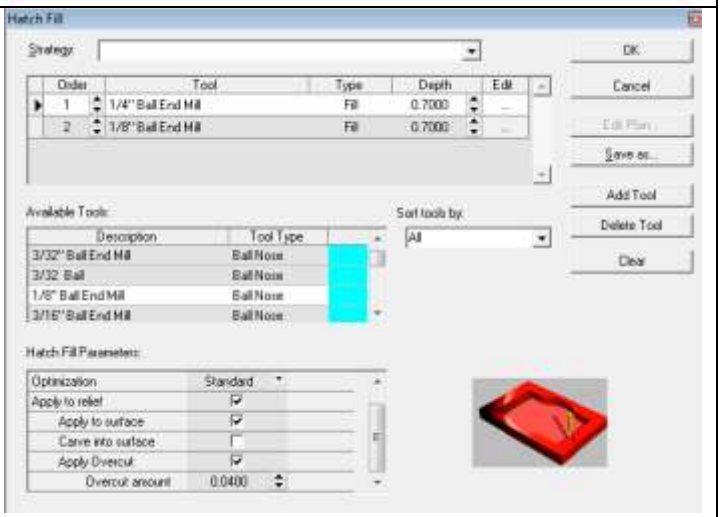
    Spacing = 1.00
    Dot Radius = 2.00
    Tangent Angle = 45
    Jitter = 0
    Twist = 2.00
    Meander = 0
    Meander Size = 3.00



15. Click Apply.
    The texture has only been applied to the background area of the design. The relief was selected so that the texture would be applied to it. Then the inside offset contour and the letter contours were selected so that the texture would only be applied to the area within.

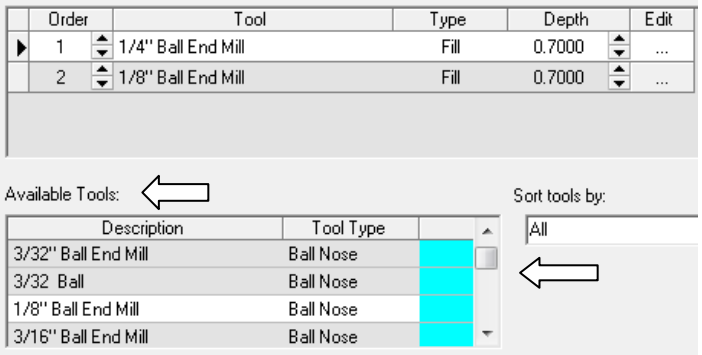


16. After designing the relief, you can now apply the toolpaths to cut the design.
    First, you must move the relief into the plate.

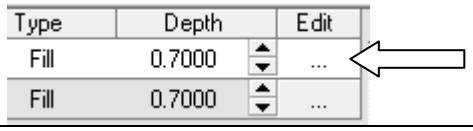
Click on the relief. Click and hold the align reliefs icon. This will open the flyout menu. Click on the Align to top Icon.	
17. This is the front view of the relief that has been moved to the top of the plate. When you look at your relief from the front view, you can see that it is almost the full depth of your plate. This will not allow much material to be left for the textured bottom. So we will use the scale tool to change the depth of the relief.	
18. Click on the relief in the top view. Click on "Front" to change to the front view. Click on the Scale icon to open the dialog. You will notice that the dimensions of the relief are calculated.	
19. Uncheck the proportional box. Change the Z parameter to .65. This will allow enough material to be left at the bottom of the relief.	
20. Click Apply. This is the results. Notice that the relief has been changed to the .65 depth.	
21. We will use the Hatch Fill Strategy to apply the toolpaths to the relief. 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. Select the relief and click on the Hatch Fill Icon. This will open the Hatch Fill Dialog.	
22. The next step is to select the tools that will be used to cut the material. In this example we have selected a ¼ Ball End and 1/8 Ball End tool	

to cut the design. Go to the **Available Tools** area of the dialog and scroll down to locate the tools. Double click on each tool to load it.

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



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



25. Enter these parameters:
    Offset from surface = .1

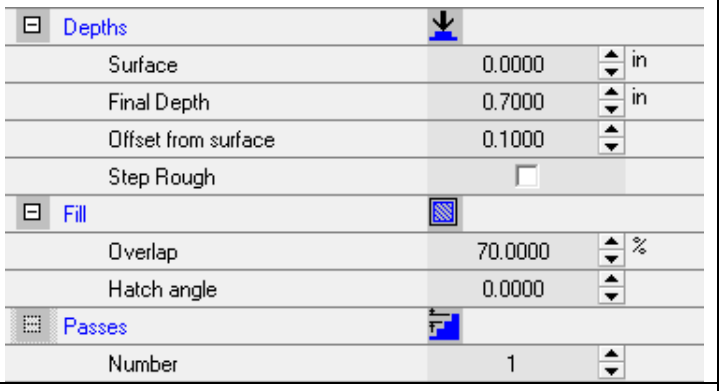
    Overlap = 70%

    Passes = 1

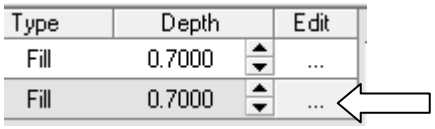
    Feed Rate: 100.0000

    Plunge Rate: 50.0000

    Spindle Speed 14000



26. Enter the parameters and Click Ok. This will return you to Hatch Fill Dialog.
    Click in the Edit box next to the 1/8'" tool. This will open the Cut Definition Dialog.



27. Enter Parameters:
    Offset from surface = 0

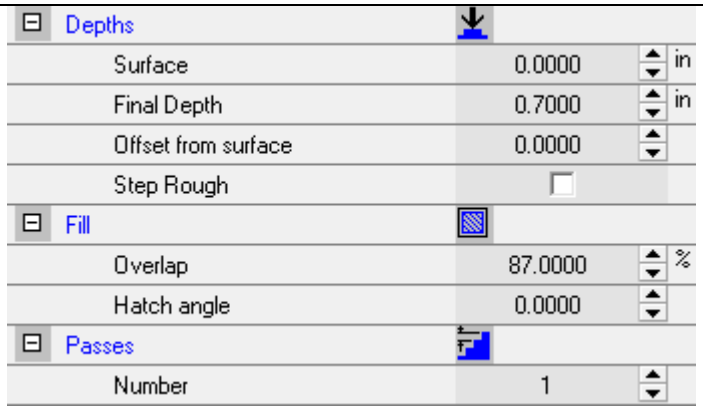
    Overlap = 87%

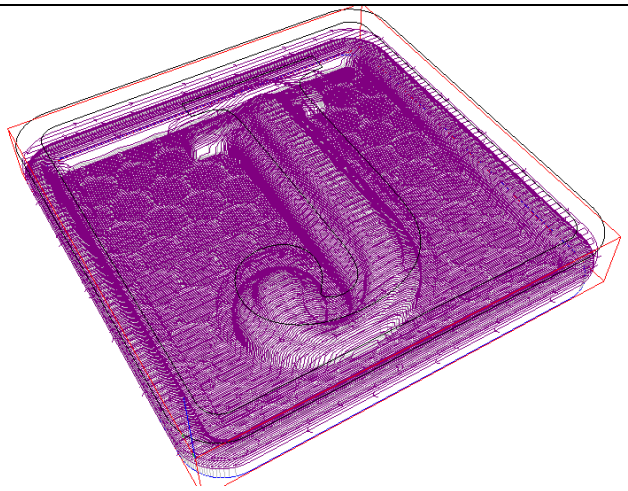
    Passes = 1

    Feed Rate = 100.0000

    Plunge Rate = 50.0000

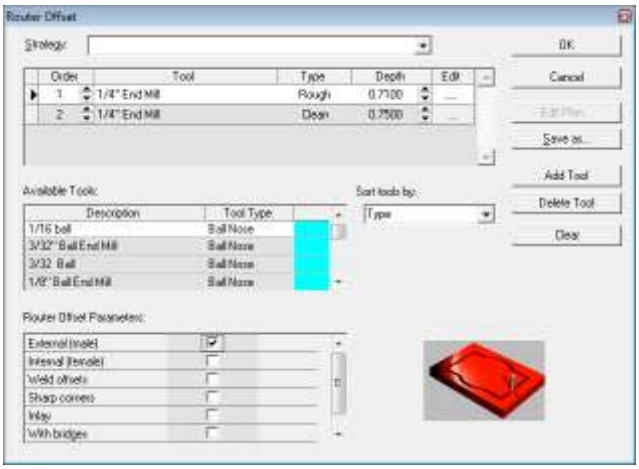
    Spindle = 14000



28. Once you have entered all of the parameters, Click OK in the Cut Definition Dialog. Click Ok again in the Hatch Fill Strategy Dialog. The toolpaths will then be processed. This image shows the toolpaths that you have created in the perspective view.	
29. The final step is to create the toolpaths used to cut out the piece.	

Routing Offset

The Route Offset strategy creates a toolpath that runs either inside or outside the selected closed contour. The toolpath is offset from the contour by the radius of the cutting too. So that it will trim right up to the edge of the contour.

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

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

    Feed Rate = 100.0000

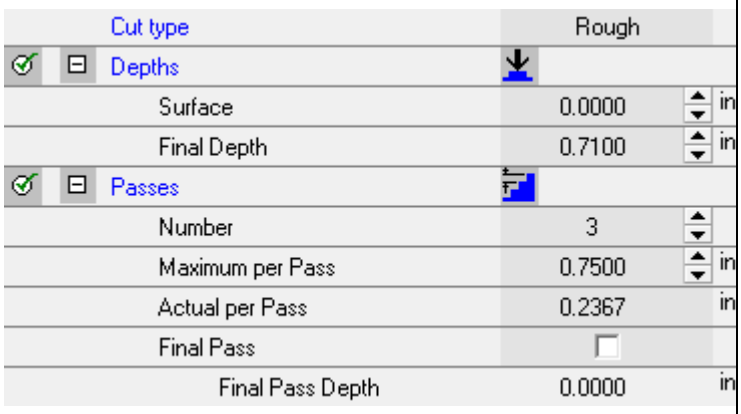
    Plunge Rate = 50.0000

    Spindle Speed = 14000

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

    Set the parameters for the Clean Tool.

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



34. Enter these parameters.
    Passes =1

    Width of cut =.02

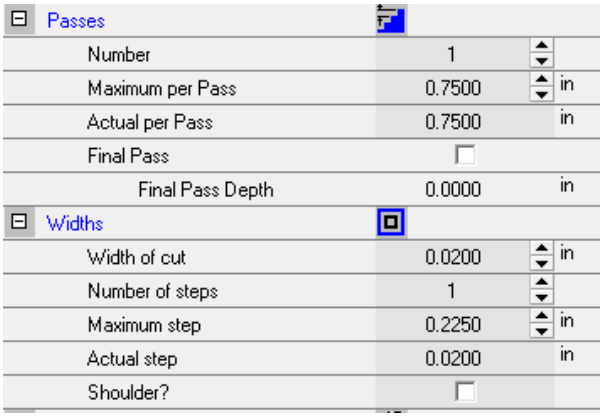
    Feed Rate = 100.0000

    Plunge Rate = 50.0000

    Spindle Speed = 14000

    Click Ok.

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



35. This is a rendered view of all of the toolpaths.