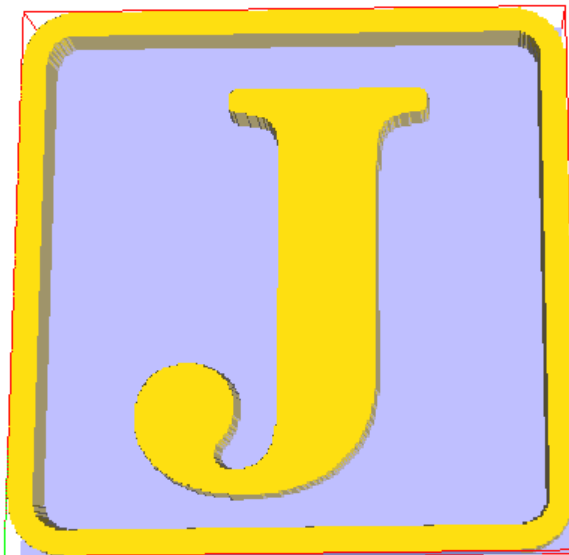


## Push-Through Letters

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.

Push-through graphics have many applications in the sign industry. The idea is to have the design machined out of two different substrates. One substrate serves as the background and the other is the graphic. A popular sign application is to create the graphic (often lettering) out of a clear or translucent acrylic and to create the background using an opaque material such as aluminum. The sign is then backlit so that the light comes through the graphic to create a very nice nighttime effect.

Push-through letters are created using a two part system. One part is a stencil cut face panel in which the openings for the letters are cut through the material. The second part is the letters that are cut of a contrasting material to be inserted through the panel. The Inlay feature in EnRoute enables you to create these letters very easily. In this example, we have milled out the blue portion of the piece so that the letter and the border of the piece will be elevated above the stencil cut face panel.



1. Define the Plate – Enter these parameters and click OK.

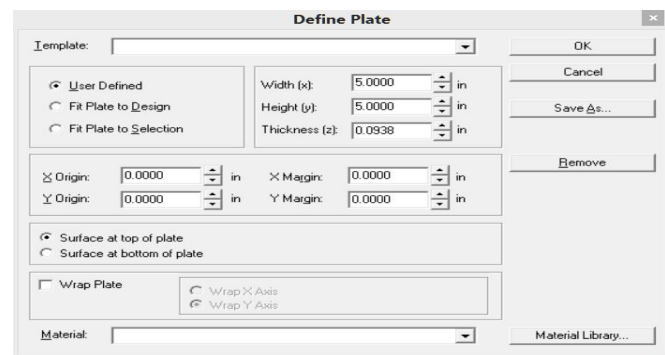
Width 5.00

Height 5.00

Thickness .0938

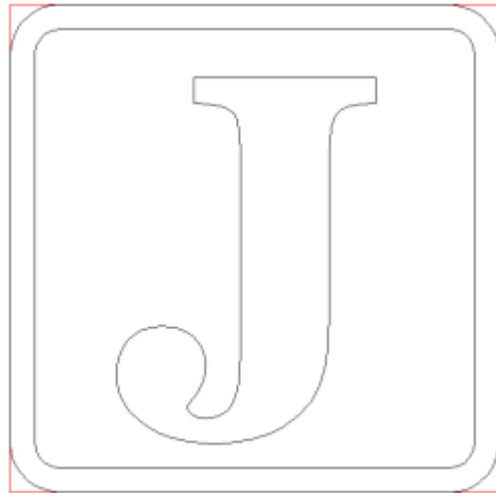
Surface at the top of Plate.

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.

The same artwork is used to cut both the male and the female parts of the push through letter.

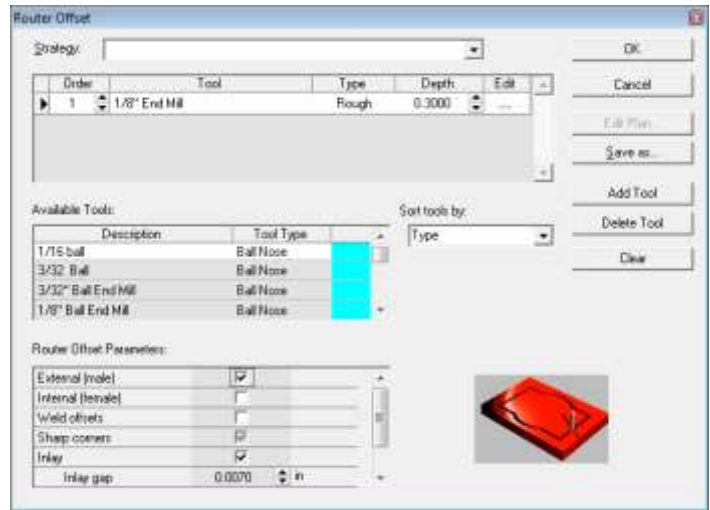


3. The Routing Offset Strategy is used to cut out the face panel in which the letters will be inserted through.

Select the letter contour and the inline contour of the border then click the Routing Offset icon to open the strategy.

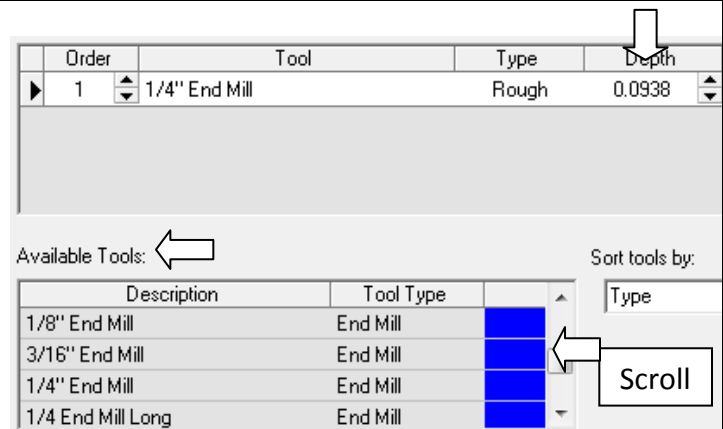
There are two important things to remember when creating toolpaths using the inlay feature.

1. Make sure that you use the same size tool in both the Routing Offset Strategy and the Hatch Fill or Island Fill Strategy.
2. Assign the same Inlay gap parameter to both of the strategies.

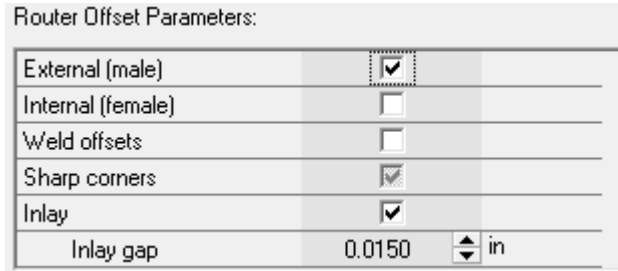


4. The next step is to select the tools that will be used to cut the material. In this example we have selected a 1/4" End Mill.

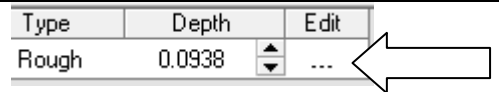
Enter the **depth** of the cut by typing in the depth or using the arrows to select the depth desired.



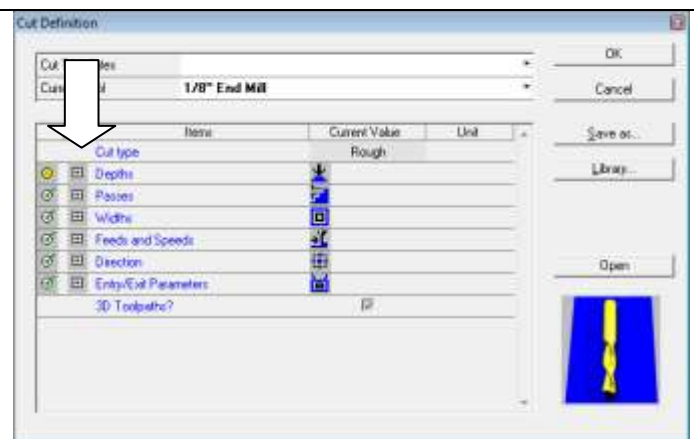
5. At the bottom of the Routing Offset Dialog, there is a checkbox for the Inlay feature. When you place a check in this box, the Inlay gap parameter box will open. Enter in the amount of the gap. In this example we used a .015 gap. This is the parameter that will define the space between the male and the female portions of the inlay. A gap is needed to allow the two parts of the inlay to fit together.



6. Click in the Edit box next to the tool to open the Cut Definition Dialog.



7. Click on the + box to open each area of the Cut Definition Dialog.



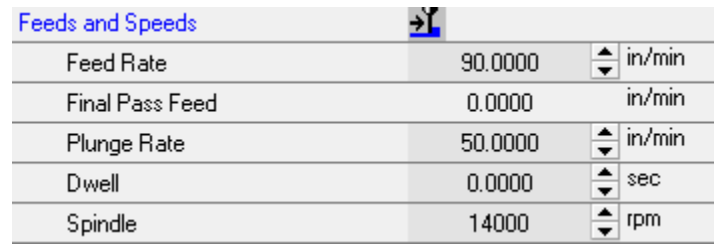
8. Enter these parameters:

Passes = 2

Feed Rate: 90.0000

Plunge Rate: 50.0000

Spindle Speed 14000

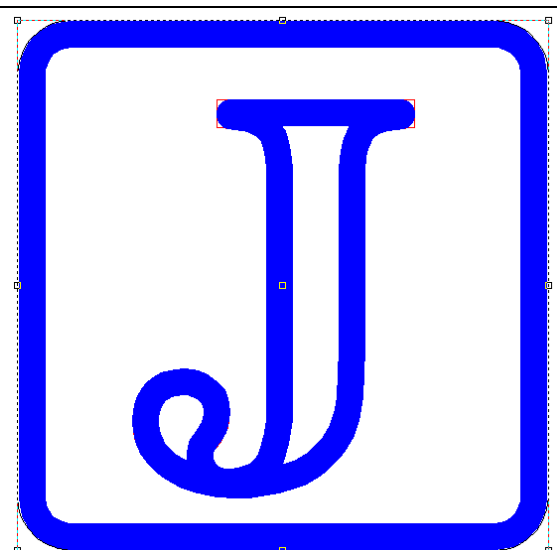


9. Once you have entered all of the parameters, Click OK in the Cut Definition Dialog.

Click Ok again in the Routing Offset Strategy Dialog.

The toolpaths will then be processed.

This image shows the toolpaths that you have created. When you press the F9 key, EnRoute displays the toolpaths showing the thickness of the tool assigned.




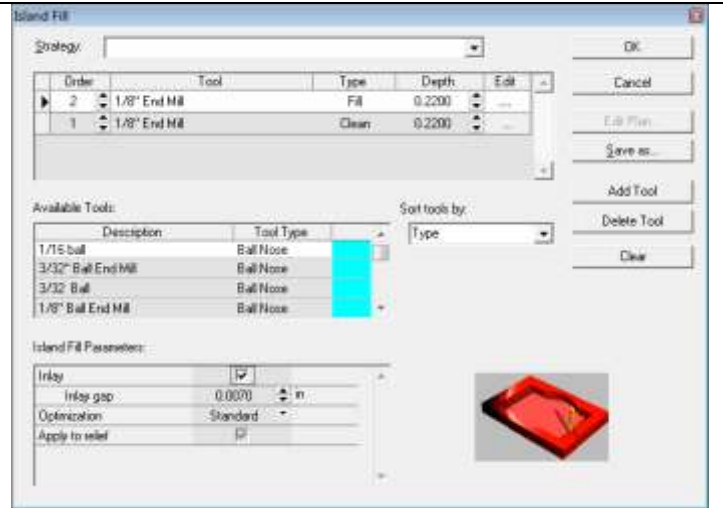
## Hatch Fill

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.

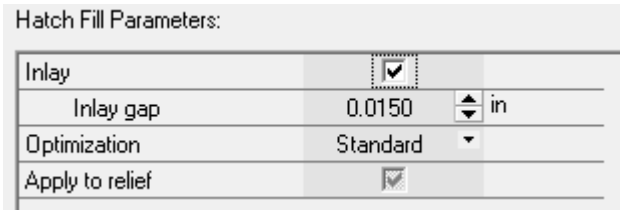
10. The second part of the push through is to mill down the portion that allows the letter and the border to extend above the face panel. To do this we will use the Hatch Fill Strategy, and a Routing Offset around the perimeter to cut the piece out.

A 1/8" End Mill tool is used to mill the area for the face panel to fit into. Remember that it is necessary to use the same size tool as you did to cut out the face panel.

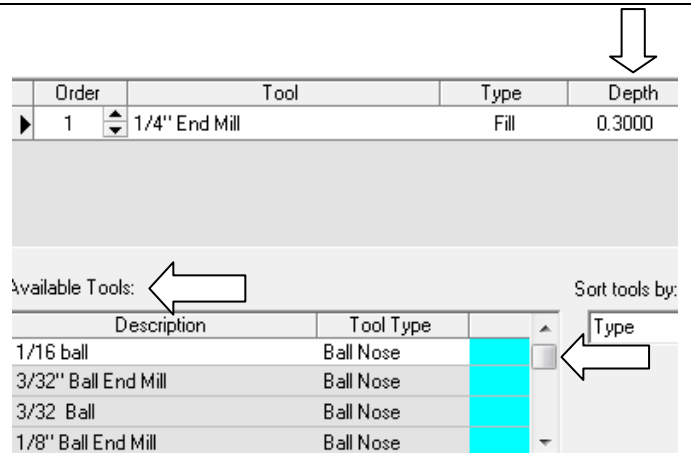
Select the letter contour and the offset contour and click on the Hatch Fill Icon. This will open the Hatch Fill Dialog. 



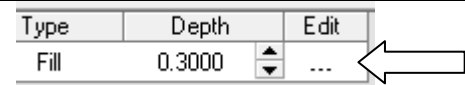
11. Check the Inlay box located at the bottom of the Island Fill Dialog. Once you have checked this box, the Inlay gap field will open for you to enter in .015. This value is the same in both the Hatch Fill strategy and the Routing Offset strategy. This will insure that the two pieces will fit together.



12. Select the 1/4" End Mill tool.  
Go to the available tools section and scroll down until you find the 1/4" End Mill. Double click on the tool to select it.  
Enter in the Depth of .3.



13. Click in the edit box to open the Cut Definition Dialog.



14. Enter the parameters  
Overlap = 80%  
Feed Rate: 100.0000



<p>Plunge Rate: 50.0000</p> <p>Spindle Speed 14000</p> <p>Click Ok to accept these parameters for this tool.</p>	
--	--

<p>15. This will bring you back to the Hatch Fill Dialog. Click Ok in the Hatch Fill Dialog to process the toolpaths.</p> <p>A 1/4" End Mill was used to cut out the piece. Remember to change the Plate parameters to reflect the thickness of the piece of material that you are using.</p>	
---	--

<p>16. 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.</p> <p>Enter the Depth of cut. For the Rough tool the Depth is .71. For the Clean cut set the Depth at .75.</p>	
<p>17. Click in the edit box next to the Rough tool to open the Cut Definition Dialog for this tool.</p>	

18. 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.  
 19. Click in the Edit box for the Clean Tool. This will open the Cut Definition Dialog.

Cut type		Rough
<input checked="" type="checkbox"/>	<input type="checkbox"/> Depths	
	Surface	0.0000 in
	Final Depth	0.7100 in
<input checked="" type="checkbox"/>	<input type="checkbox"/> Passes	
	Number	3
	Maximum per Pass	0.7500 in
	Actual per Pass	0.2367 in
	Final Pass	<input type="checkbox"/>
	Final Pass Depth	0.0000 in

20. 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.

<input type="checkbox"/>	<input checked="" type="checkbox"/> Passes	
	Number	1
	Maximum per Pass	0.7500 in
	Actual per Pass	0.7500 in
	Final Pass	<input type="checkbox"/>
	Final Pass Depth	0.0000 in
<input type="checkbox"/>	<input checked="" type="checkbox"/> Widths	
	Width of cut	0.0200 in
	Number of steps	1
	Maximum step	0.2250 in
	Actual step	0.0200 in
	Shoulder?	<input type="checkbox"/>

21. This is a rendered view of the Hatch Fill and Routing Offset.

