Shoulder Instructions

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.

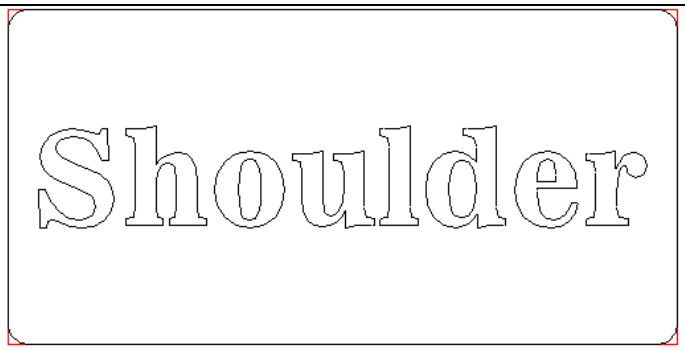
We are also demonstrating the use of the shoulder function. This provides a way to create a step, or lip around the perimeter of the design. When used as a push through piece, the lip will stop it from pushing through the face panel.

The Weld function is also demonstrated in this example. This allows the user to combine the lettering so that it is treated as one object and does not cut the letters individually.

1. Define the Plate – Enter these parameters and click OK. Width 10.00 Height 5.00 Thickness .125 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.

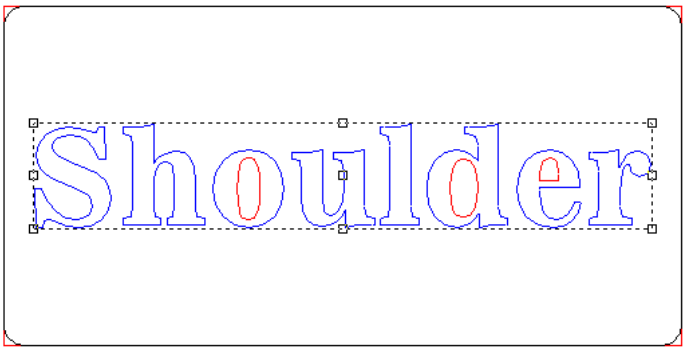


3. The Routing Offset Strategy is used to cut out the face panel in which the letters will be inserted through.

   Select the letter contours then click the Routing Offset icon to open the strategy.

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

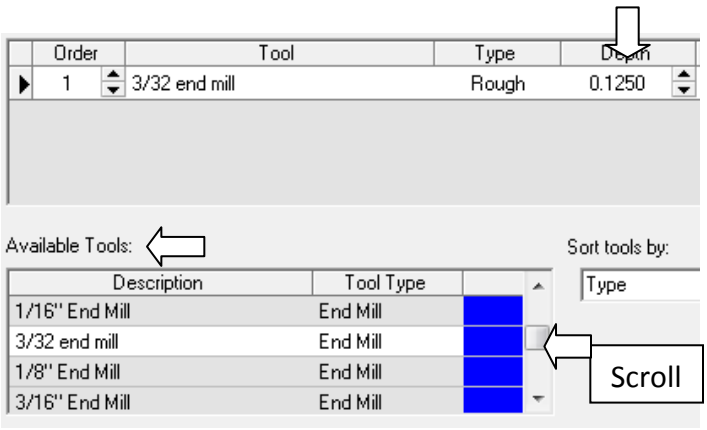
   1. Make sure that you use the same size tool in both the Routing Offset strategies for the face panel and the letters to cut out.
   2. Assign the same Inlay gap parameter to both of the strategies.



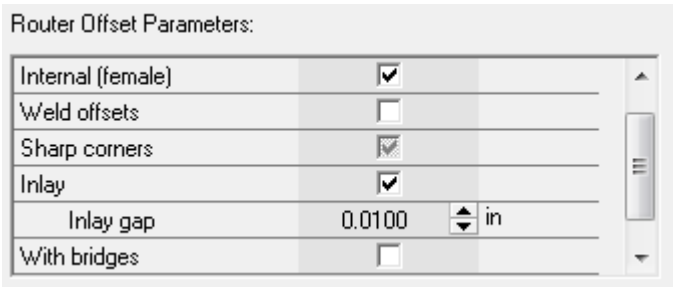
4. The next step is to select the tools that will be used to cut the material. In this example we have selected a 3/32" End Mill.

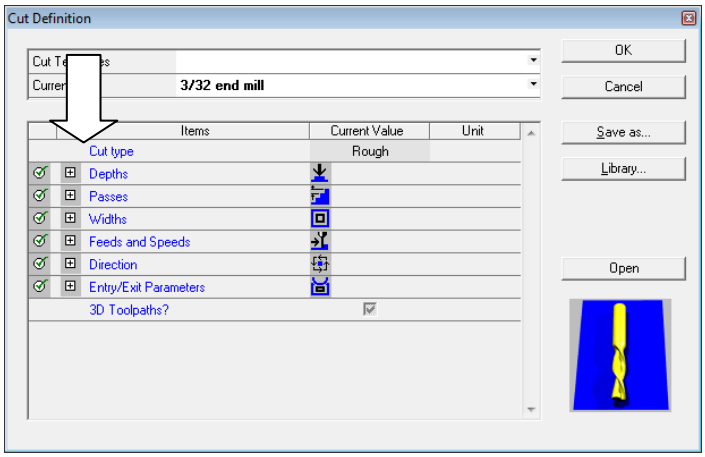
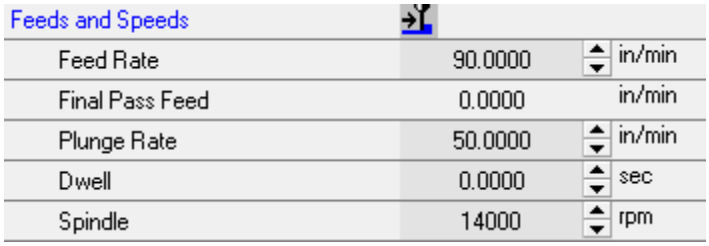
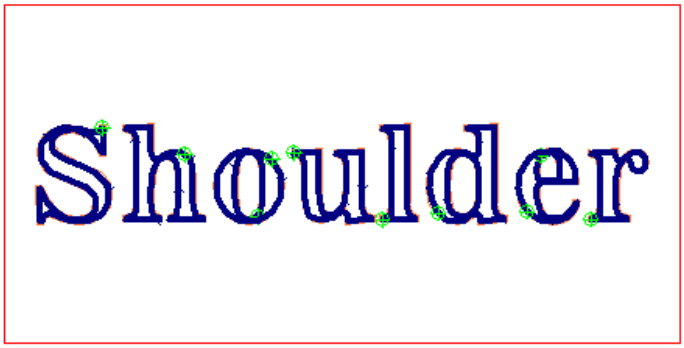
   Select the tool by scrolling through the available tools until you find the 3/32" End Mill. Double click to select the tool.

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

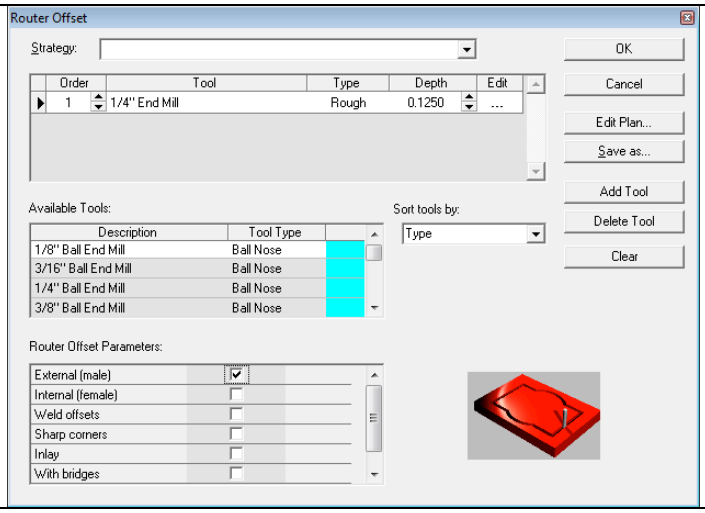


5. At the bottom of the Routing Offset Dialog enter the Routing Offset Parameters. Select the **Internal option** so the tool will cut to the inside of the letters.
   Select the **Inlay feature**. When you place a check in the inlay box, the Inlay gap parameter box will open. Enter in the amount of the gap. In this example we used a .010 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.

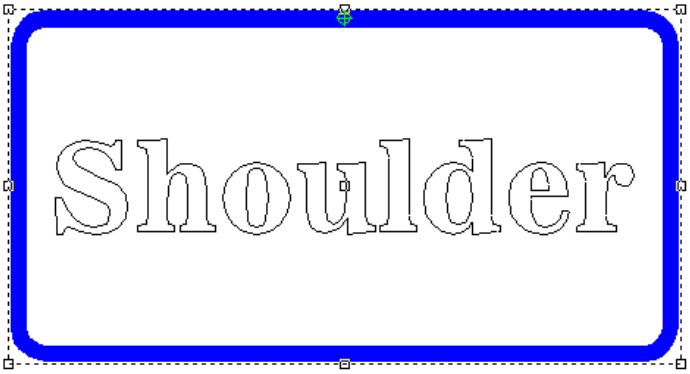


6. Click in the Edit box next to the tool to open the Cut Definition Dialog.	
7. Click on the + box to open each area of the Cut Definition Dialog.	
8. Enter these parameters: Passes = 2 Feed Rate: 90.0000 Plunge Rate: 50.0000 Spindle Speed 14000	
9. Once you have entered all of the parameters, Click 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.	

10. To cut out the face panel, select the outside contour and click on the Routing Offset icon to open the strategy.
    Select a ¼ "End Mill tool to cut out the face plate.
    Enter a Depth of .125.
    Make sure the External parameter is checked.
    Click on the Edit box to open the Cut Definition Dialog for this tool.
    Enter the parameters that are appropriate to the type of material that you are using.

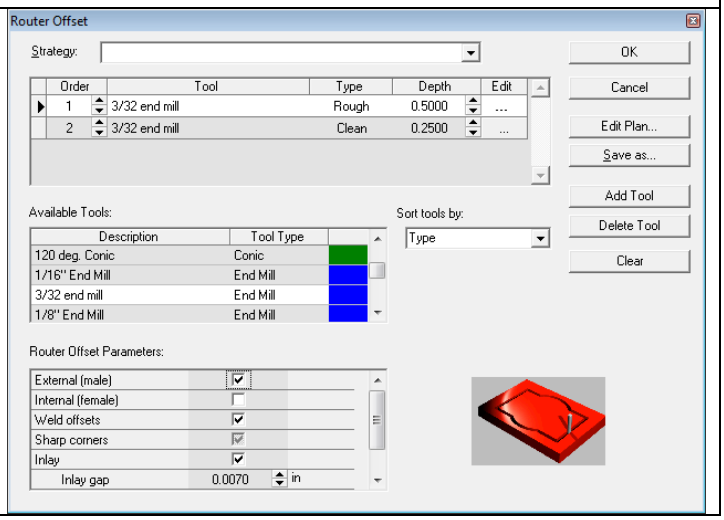


11. This is a screen shot of the toolpaths used to cut out the face panel.



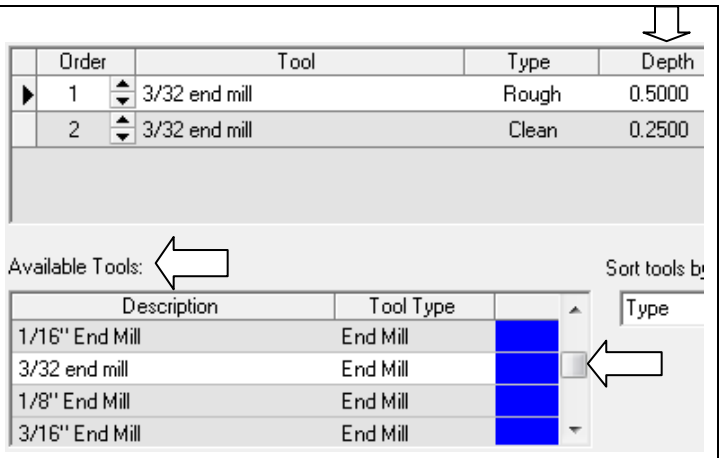
12. This is a rendered view of the face plate toolpaths.

13. The second part of the push through is to mill the letters to extend above the face panel. To do this we will use the Routing Offset Strategy.

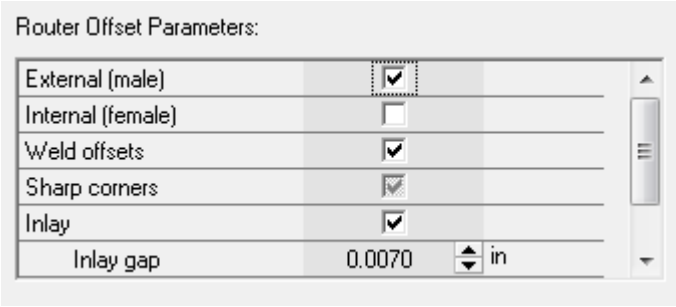


14. Select the 3/32” End Mill tool from the available tools section. Scroll down to locate the tool and then double click to load the tool. We are using the 3/32” End Mill for both the Rough and Clean cuts. Enter the 3/32” End Mill tool a second time.
    Set the Depth of the Rough tool at .5 inches.
    Set the Depth for the Clean tool at .25 inches.

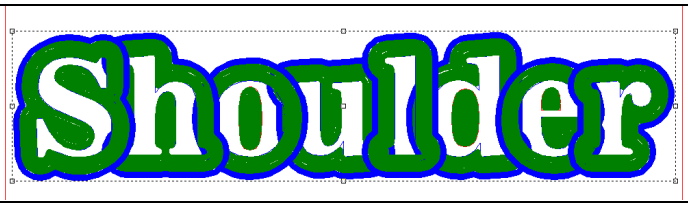
    The face panel we used is .125 thick. This will leave 1/8” of the letters to extend above the face panel.



15. Enter the Router Offset Parameters.
    Check **External**
    Check **Weld offsets**. This will allow EnRoute to treat the letters as one object. If we did not check this, each letter would be toolpathed individually.
    Check the **Inlay** box.
    Enter the Inlay gap of **.007**. This is the same gap as was used to cut the face panel. This will insure that the two pieces will fit.



16. This is a rendered view of the toolpaths if we did not check the weld offsets box. Each of the letters would be toolpathed individually. In this example, we want the letters to cut out together as one piece.

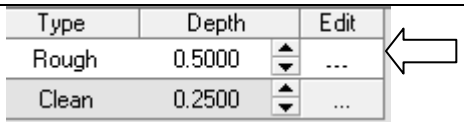


17. With the weld offsets checked, the toolpaths cut the letters as one object.

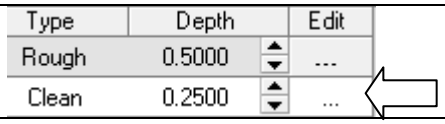
18. Click on the edit box next to the Rough tool to open the Cut Definition dialog. Since this is a small tool and it is going .5 deep, we have assigned it 4 passes.
    Feed Rate: 100.0000

    Plunge Rate: 50.0000

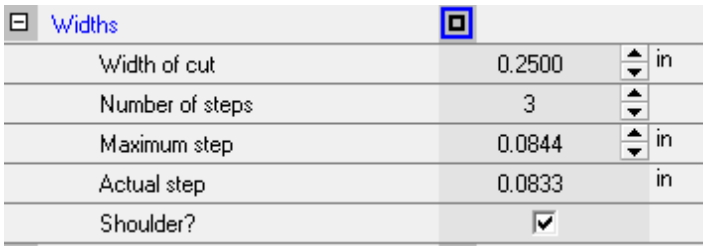
    Spindle Speed 14000



19. Click on the edit box next to the Clean tool to open the Cut Definition dialog for this tool.



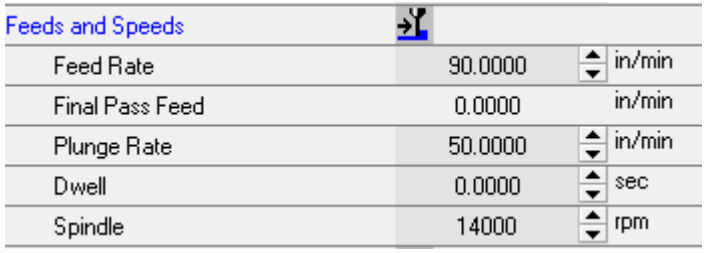
20. Enter the parameters:
    Passes =2
    This is where the shoulder feature is activated.
    In this example, we have entered a width of cut of .25. This will allow a .25" lip around the letters so that they will not push all the way through the opening of the face panel.
    The program will calculate the number of steps.
    Check the Shoulder option.



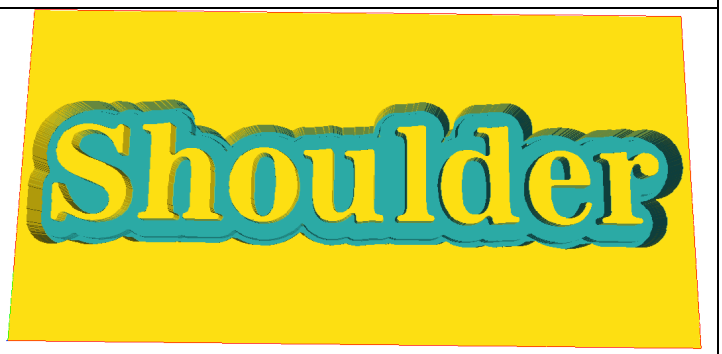
21. Enter these parameters:
    Feed Rate: 90.0000

    Plunge Rate: 50.0000

    Spindle Speed 14000



22. Once you have entered your parameters, click OK in the Cut Definition Dialog and then Ok in the Routing Offset Dialog. EnRoute will calculate the toolpaths.
    We have simulated the toolpaths using the Simulate Ortho tool.



23. This is a close-up view of the rendered toolpaths.
    You can see that the clean cut in this view.