Celtic Knot Inlay

The information presented here will show how to create the design using an Inlay technique. The drawing tools were used to create the contours we will use to design the Inlay Celtic Knot.

Note: **These instructions include the selection of specific tools and setting cut parameters such as depth, passes, and feed rates. Our choice of these parameters was based on our machine, our tools, and the material we were cutting. You must always choose parameters that are appropriate for your machine, tooling and material.**

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

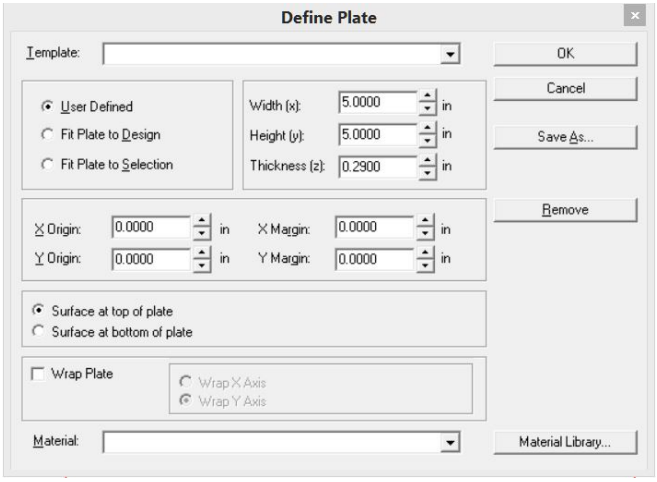
   Width 5.00

   Height 5.00

   Thickness .29

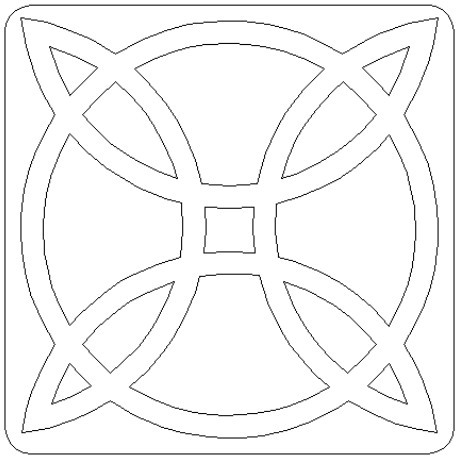
   Surface at the top of Plate.

   We will first be working with the male portion of the design. This part will be inserted into the base of the artwork.



2. This is the artwork that we are using for this example.
   It is the same artwork that was used to create the 2D sample. The difference will be in how the toolpaths are created.

   The same artwork is used to cut both the male and the female parts of the inlay.

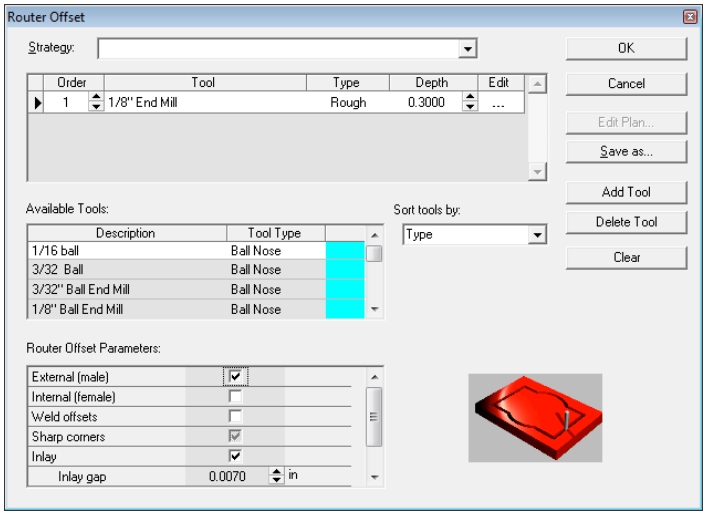


3. The Routing Offset Strategy is used to cut out the male piece that will be inset into the base or female piece or the inlay.

   Select the contour and then click the Routing Offset icon to open the strategy.

   There are two important things to remember when creating toolpaths for an inlay design.

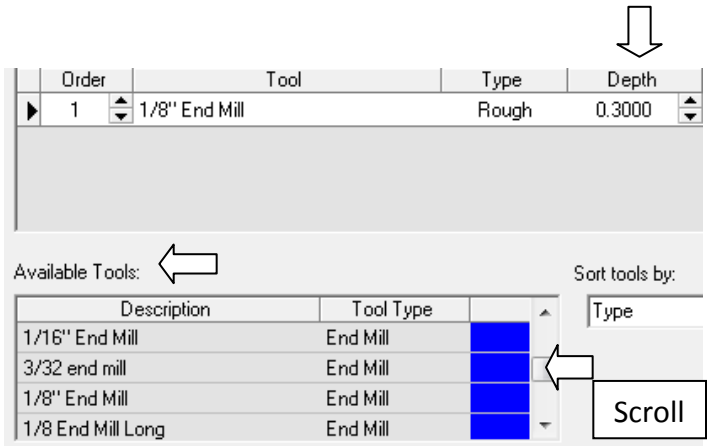
   1. Make sure that you use the same size tool to cut out the male pieces and to mill out the female piece.
   2. Assign the same Inlay gap parameter to both of the strategies.



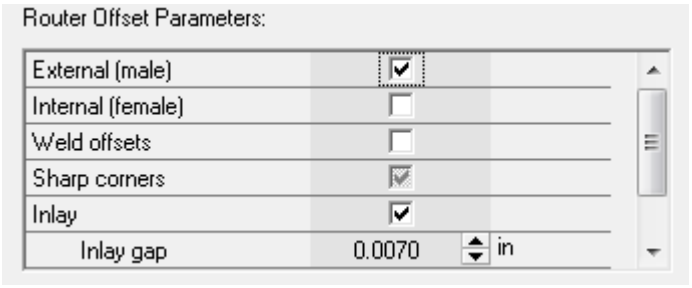
4. The next step is to select the tools that will be used to cut the material. In this example we have selected a 1/8 End Mill.

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

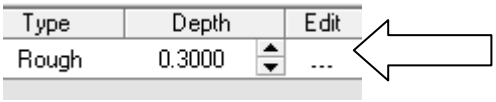
   In this example, the depth is set just slightly deeper that the material thickness. This will insure that the tool will cut cleanly through the material.



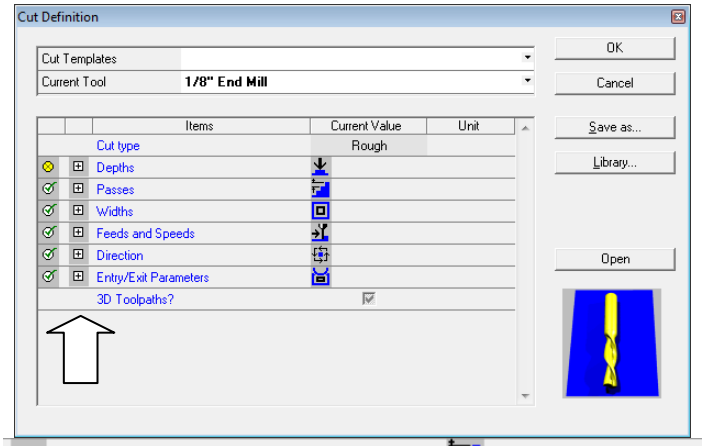
5. At the bottom of the Routing Offset Dialog, there is a checkbox for the Inlay feature. When you place a check in this box, the Inlay gap parameter box will open. Enter in the amount of the gap. This is the parameter that will define the space between the male and the female portions of the inlay.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



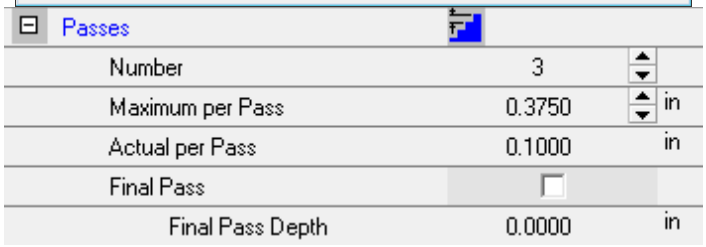
6. Click in the Edit box next to the tool to open the Cut Definition Dialog.



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



8. Enter in 3 passes or use the arrows to select the number of passes.

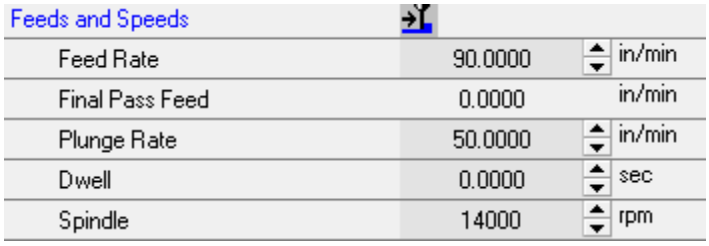


9. Enter these parameters:

   Feed Rate: 90.

   Plunge Rate: 50.

   Spindle Speed 14000

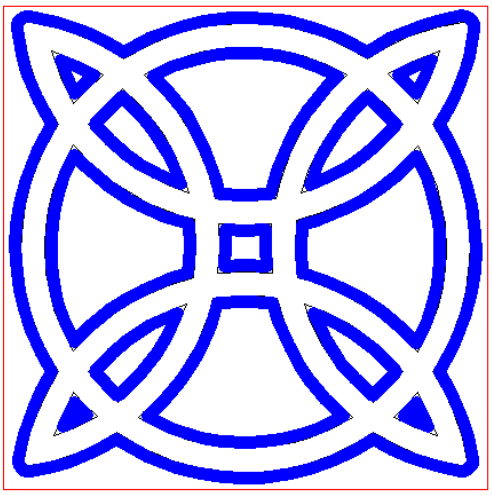


10. Once you have entered all of the parameters, Click OK in the Cut Definition Dialog.

    Click Ok again in the Routing Offset Strategy Dialog.

    The toolpaths will then be created.

    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.



Island Fill

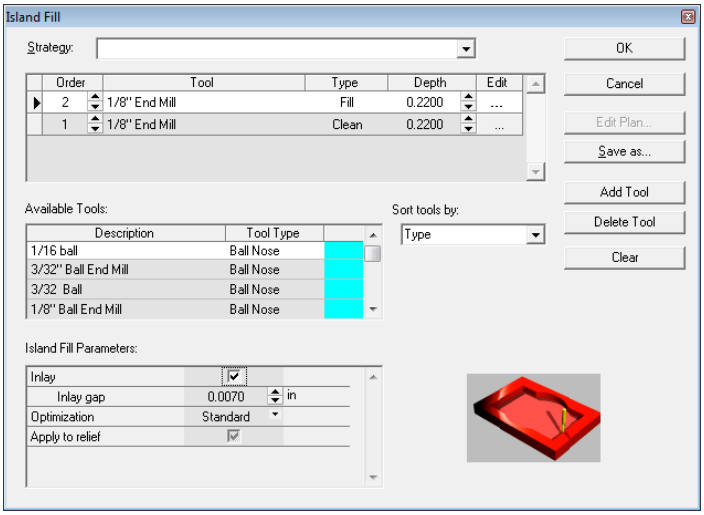
The Island Fill strategy is used to mill a design that is made up of long, thin sections. The Island Fill will follow the shape of the contours which can save time in milling the piece.

11. The second part of the inlay is to cut the female portion. To do this we will use the Island Fill Strategy, and a Routing Offset around the perimeter to cut the piece out.

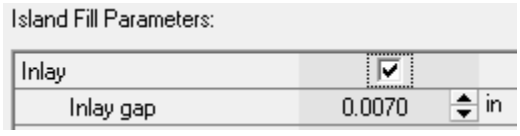
    A 1/8 End Mill tool is used to mill the area for the inlay piece to fit into. Remember that it is necessary to use the same size tool as you did to cut out the male piece.

    Select the contour and click on the Island Fill Icon.

    This will open the Island Fill Dialog. 

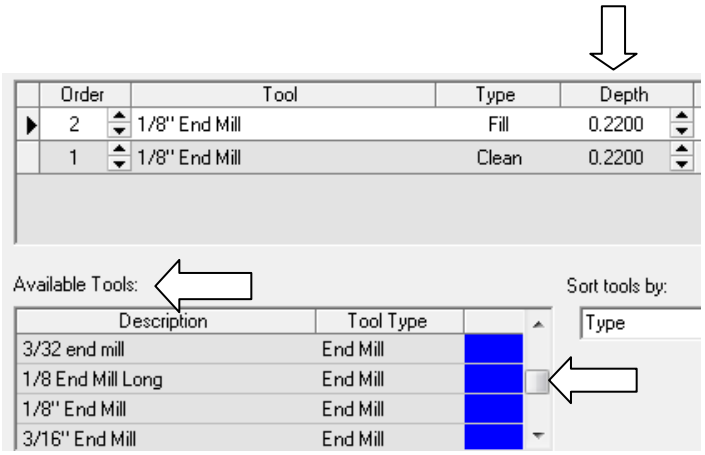


12. Check the Inlay box located at the bottom of the Island Fill Dialog. Once you have checked this box, the Inlay gap field will open for you to enter in .007. This value is the same in both the Island Fill strategy and the Routing Offset strategy. This will assure that the two pieces will fit together correctly.



13. Select the 1/8 End Mill tool.
    Go to the available tools section and scroll down until you find the 1/8 End Mill. Double click on the tool to select it.

    In this example we are also using the 1/8 End Mill tool to do a clean pass so we want to double click to load the tool again.



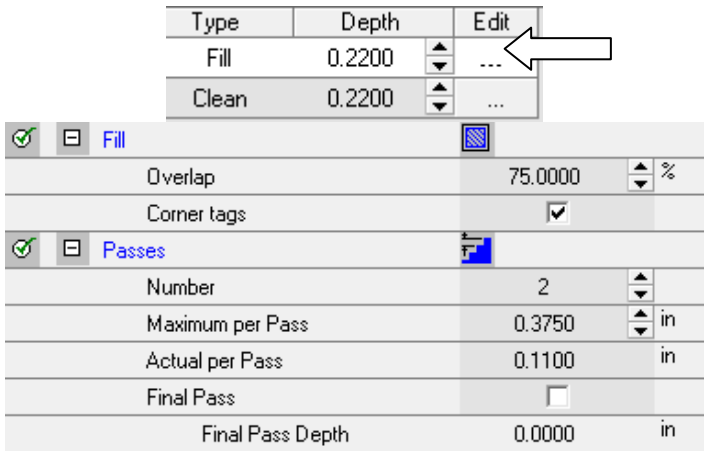
14. Click in the edit box to open the Cut Definition Dialog.

15. Click the + box to open each category to enter the parameters.

    Overlap = 75%

    Check Corner tags box = yes

    Passes = 2



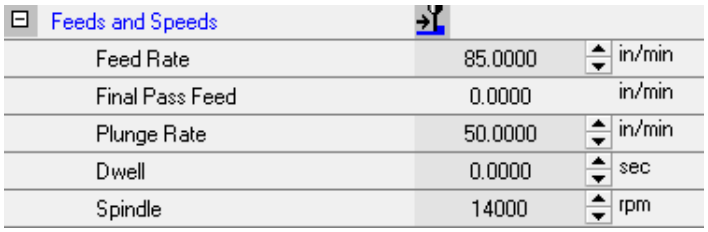
16. Enter the parameters for the Feeds and Speeds:

    Feed Rate: 85.0000

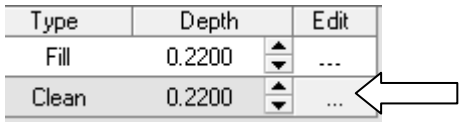
    Plunge Rate: 50.0000

    Spindle Speed 14000

    Click Ok to accept these parameters for this tool.



17. This will bring you back to the Island Fill Dialog. Next, we will enter the parameters for the Clean Tool. Click in the edit box next to the clean tool.



Clean Cut

The Clean Cut defines a tool that creates toolpaths that are offset from the contours. The Clean Cut is used to improve the edge quality of the finished cut around the perimeter of the design. It helps to use a Clean Cut when creating an Inlay design because you want the edges of the milled area to cut as nicely as possible.

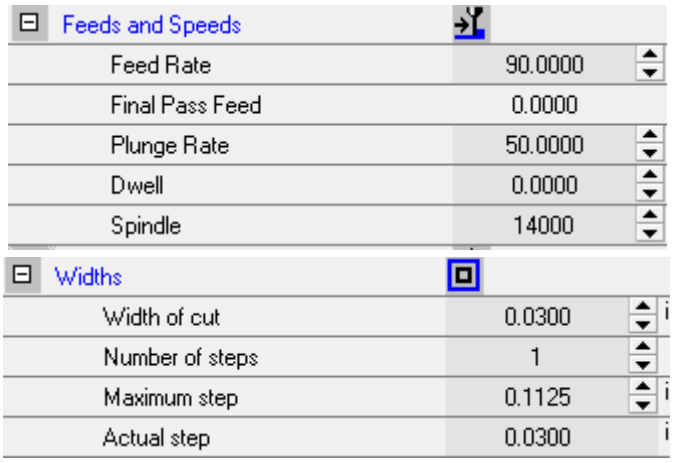
18. Enter the Parameters:
    Number of Passes = 2

    Feed Rate = 90.0000

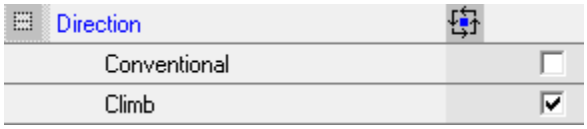
    Plunge Rate =50.0000

    Spindle Speed = 14000

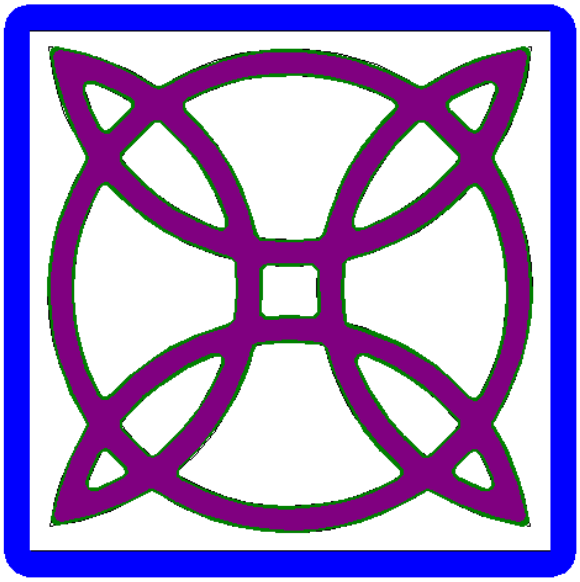
19. Enter the width of cut of .003.



20. In the Direction section check the Climb box. This will reverse the direction that the tool cuts. This is helpful when doing a clean cut because it will reduce chipping of the edges.



21. Once you have entered the parameters for the clean cut, click OK. This will bring you back to the Island Fill Strategy.
    Click OK again to create the toolpaths.

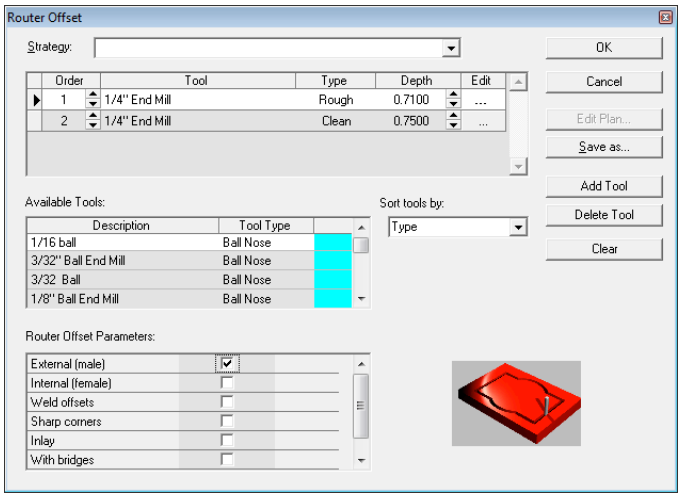


This image is also showing the Routing Offset strategy applied to the outside contour to cut out the piece.

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 to cut the female object from.

22. 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.



23. Click in the edit box next to the Rough tool. This will open the Cut Definition Dialog for this tool.
24. Enter the parameters for the Rough tool:
    Passes = 3

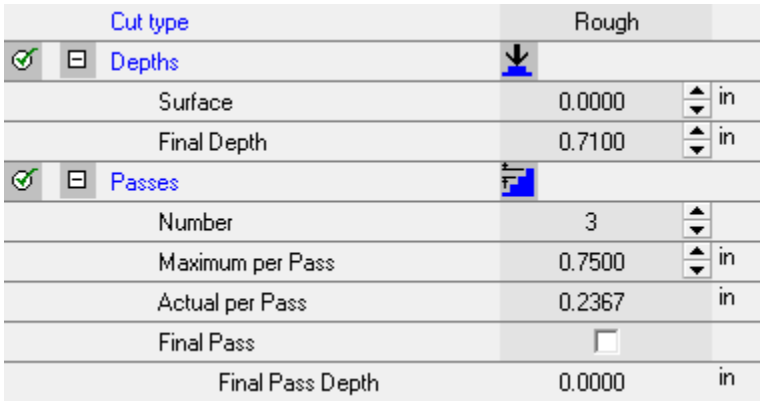
    Feed Rate = 100.0000

    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.



25. Click in the Edit box for the Clean Tool. This will open the Cut Definition Dialog.
26. Enter these parameters.
    Passes =1

    Width of cut =.02

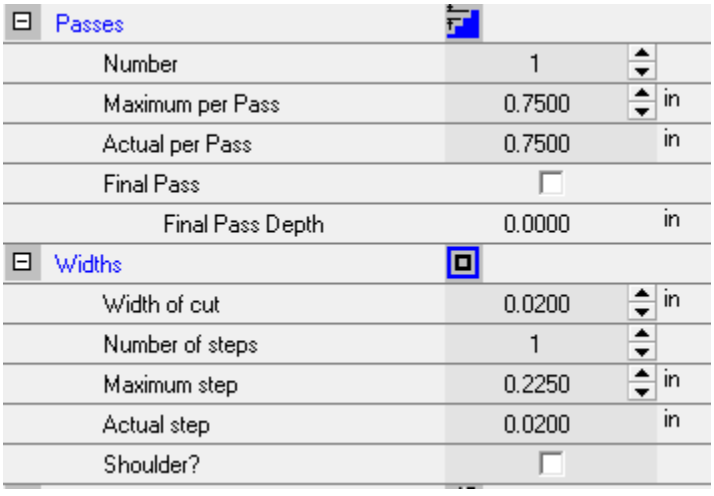
    Feed Rate = 100.0000

    Plunge Rate = 50.0000

    Spindle Speed = 14000

    Click Ok.

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



27. The two different strategies were created on two different layers. When you turn on both of these layers, you can get a good idea of the actual cut of the toolpaths. If there has been an error made you will be able to notice it when you do this step as a check.