


## 2.5 D Celtic Knot



Here we will explore the use of the Hatch Fill Strategy using an engraving tool and 3D toolpaths. The drawing tools were used to create the contours we will use to create the 2.5 D 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.**

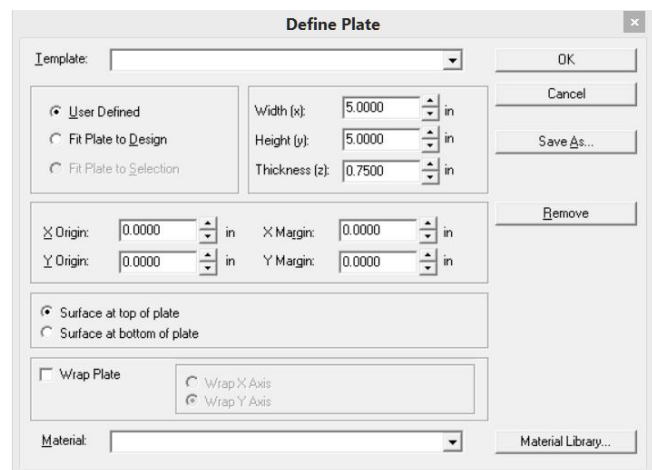
1.  Define the Plate – Enter these parameters and click OK.

Width 5.00

Height 5.00

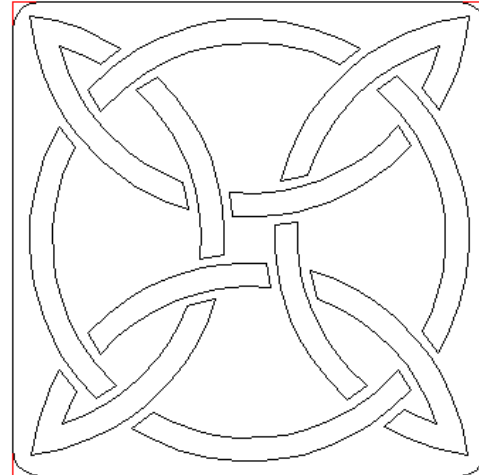
Thickness .75

Surface at the top of Plate.




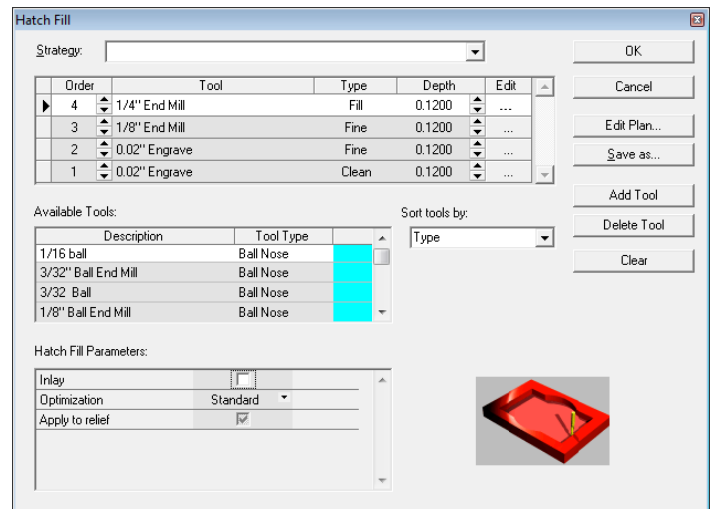
EnRoute Step-by-Step Series

2. This is the design that we are using for this example. Instructions for this design can be found in the Celtic Knot Introduction tutorial.



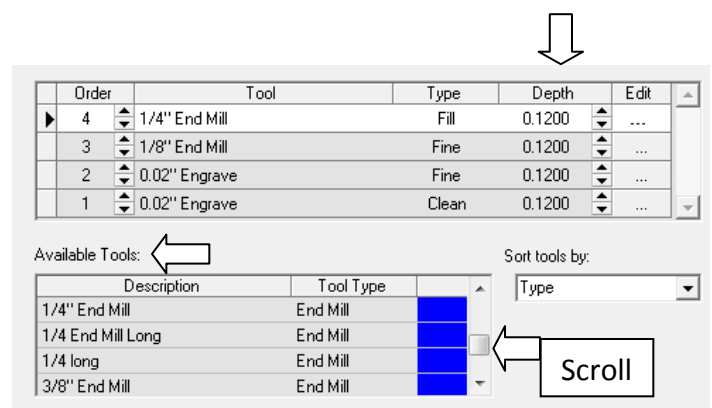
3. We want to create toolpaths that will mill down the background surfaces so that the Celtic Knot is the raised surface. We are also using an engrave tool with a 3D clean pass to put a bevel on the contours. To do this we will use the Hatch Fill Strategy.

Select all of the contours and click on the Hatch Fill Icon.  This will open the Hatch Fill 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 1/4" End Mill, 1/8" End Mill tool and .02 Engrave 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. In this example we have loaded the 1/4" End Mill as the Fill tool and the 1/8" End Mill tool as a Fine tool and the .02 Engrave tool as the Fine and Clean tool.

Enter the **depth** of the cut (.12) by typing in the depth or using the arrows to select the depth desired.



## Fill Cut

Using the Hatch fill strategy, the fill cut will move back and forth across the object to the defined depth to mill the area. This tool is often a larger tool that will be defined to do most of the material removal. Additional parameters such as the angle of the cut and the overlap of the cut are defined in the cut parameters for the tool.

## Fine Cut

The fine cut is optional. If it is used, a tool that is smaller in diameter than the Fill tool is selected to fit into sections of the design that the fill tool was not able to fit into. This will be corners and any other thin areas that are too small for the Fill tool. The Fine cut uses an Island Fill strategy to fill in areas that require more than one width of the tool. It is necessary to define the amount of overlap between adjacent toolpaths.

## 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. Often the same size tool is used to do the Clean Cut and the Fine Cut.

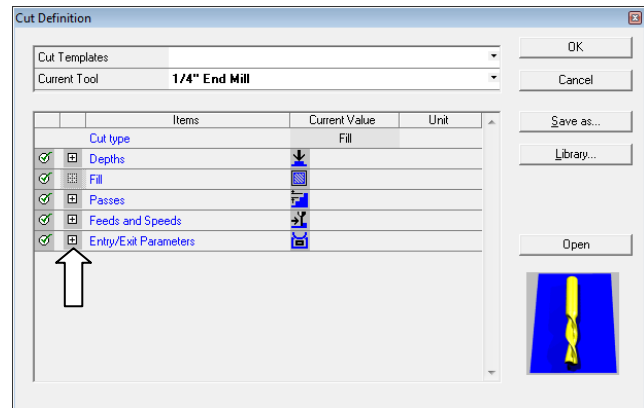
## 3D Clean Cut

A 3D toolpath may be created using a conic, engraving or tapered tool. The difference is that it uses the beveled shape of the tool to create a beveled edge. It also will move up into corners to get as far into the corners as the tip dimension of the tool will allow. The 3D option is a very powerful option, allowing a very different type of cut from any of the toolpaths that are strictly 2D.

5. The next step is to edit the parameters of each of the tools. Move the cursor to the edit box next to the tool you wish to set the parameters for and left click. This will open the Cut Definition Dialog box for that specific tool.

Fill	0.1200	▲▼	...
Fine	0.1200	▲▼	...
Fine	0.1200	▲▼	...
Clean	0.1200	▲▼	...

6. There are several parameters that can be defined for each tool. The type of material that you are using will have influence in determining the parameters used. If you are using HDU foam, it will cut a lot easier than wood or metal, so parameters can be set at a faster more aggressive pace. More specific information can be found in the EnRoute manual in the "Working With Toolpaths" section.



*EnRoute Step-by-Step Series*

- Enter the Parameters in the Cut Definition Dialog.  
Click on the + box to open the Fill category.

Enter Parameters:

Overlap = 87%

Hatch Angle = 0

Click on the + box to open the Passes category.

Enter Parameter:

Passes = 1

Fill		
Overlap	87.0000	▲▼
Hatch angle	0.0000	▲▼
Passes		
Number	1	▲▼

### Overlap

This parameter defines how much the toolpaths will overlap each other in the fill. The default value that is created when the Fill Cut is created is 50 percent. Values can be set between 0 and 99 percent. Softer materials can be set at a lower percentage than the denser materials. The overcut percentage can affect the look of the finished cut of the material. When using a harder material it may be necessary to have a higher overlap to avoid the appearance of the toolpaths.

### Hatch Angle

This angle determines the angle that the tool will cut. The usual settings are either 0 degrees for horizontal toolpaths or 90 degrees for vertical toolpaths.

### Passes

This is the number of passes used to cut the material to the assigned depth.

- Click on the + box to open the Feeds and Speeds category.

Enter Parameters:

Feed Rate = 100.

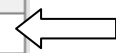
Plunge Rate = 50.

Spindle = 14000

Feeds and Speeds		
Feed Rate	100.0000	in/min ▼
Final Pass Feed	0.0000	in/min ▼
Plunge Rate	50.0000	in/min ▼
Dwell	0.0000	sec ▲▼
Spindle	14000	rpm ▲▼

- Click OK. This will bring you back to the Hatch Fill Dialog. Then click in the Edit box for the next tool in the list.

Type	Depth	Edit
Fill	0.1200 ▲▼	...
Fine	0.1200 ▲▼	...
Fine	0.1200 ▲▼	...
Clean	0.1200 ▲▼	...



*EnRoute Step-by-Step Series*

10. Enter the parameters for the Fine Cut.

Overlap = 70%

Passes =1

Feed Rate = 100.

Plunge Rate = 50.

Spindle = 14000

The same parameters are used for the .02 Engrave Fine tool.

Depths		
Surface	0.0000	in
Final Depth	0.1200	in
Overlap	70.0000	%

11. Click OK. This will bring you back to the Hatch Fill Dialog. Then click in the Edit box for the .02 Engrave Clean tool in the list.

Then click in the Edit box for the .02 Engrave Clean tool in the list.

Type	Depth	Edit
Fill	0.1200	...
Fine	0.1200	...
Fine	0.1200	...
Clean	0.1200	...

12. Enter the parameters for the Clean Cut.

Passes =1

Width = .001

Feed Rate = 100.

Plunge Rate = 50.

Spindle = 14000

Check the 3D Toolpaths box

Widths		
Width of cut	0.0100	in
Number of steps	1	
Maximum step	0.0180	in
Actual step	0.0100	in
+ Feeds and Speeds		
+ Direction		
+ Entry/Exit Parameters		
3D Toolpaths? <input checked="" type="checkbox"/>		

13. Click OK.

Then Click Ok in the Hatch Fill Dialog. EnRoute will then calculate the toolpaths.

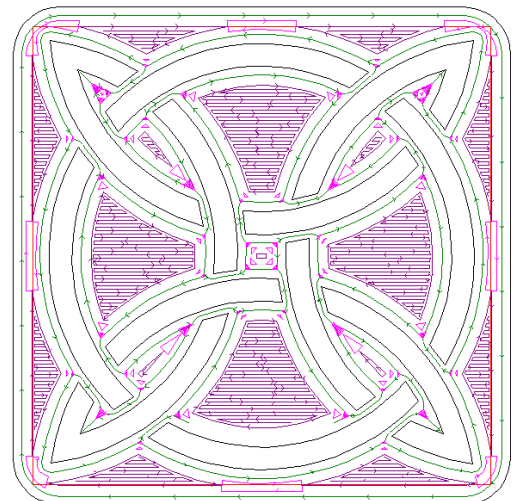
This image shows the toolpaths.

Purple = 1/4" End Mill

Pink = 1/8" End Mill Fine Cut

Smaller Pink = .02 Engrave Fine Cut

Green = 1/8" End Mill Clean Cut

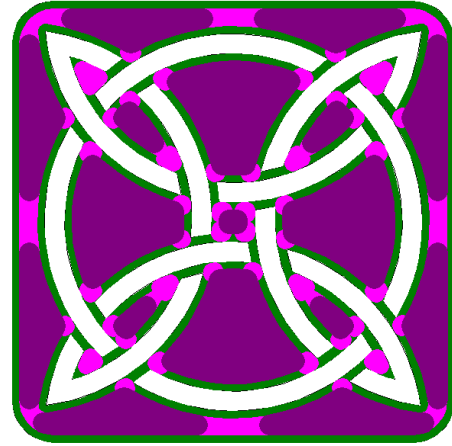


Press F9 to show the thickness of the toolpaths.

Purple = ¼ End Mill Fill Cut

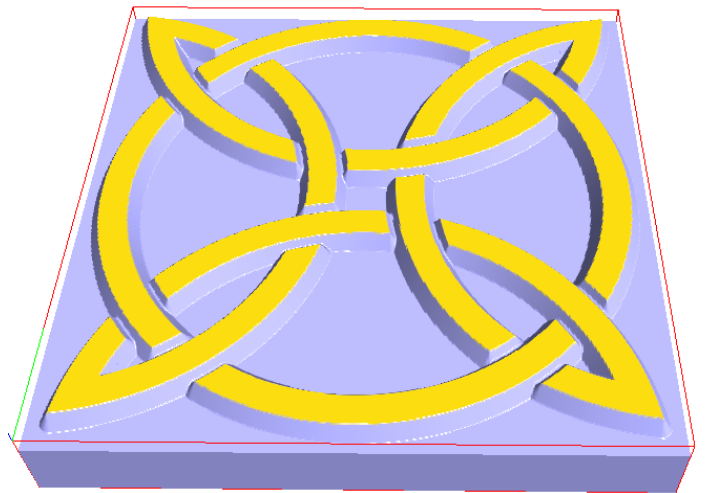
Pink = ⅛ End Mill Fine Cut

Green = .02 Engrave Clean Cut

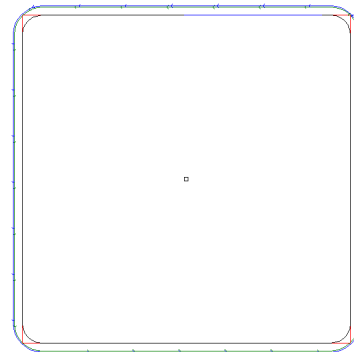


Using the Simulate Ortho Tool allows you to see a rendered image of the toolpaths that you have applied. This is often a good way to see a preview of the toolpaths. If there are any noticeable errors you can then correct them before you actually cut your piece.

This is also a good way to set the cut order of the toolpaths. The simulation allows you to see the progress of the toolpaths in several different ways.



14. The last step is to cut out the piece from the material.  
 Create another layer.  
 Paste a copy of the outline contour to that layer.



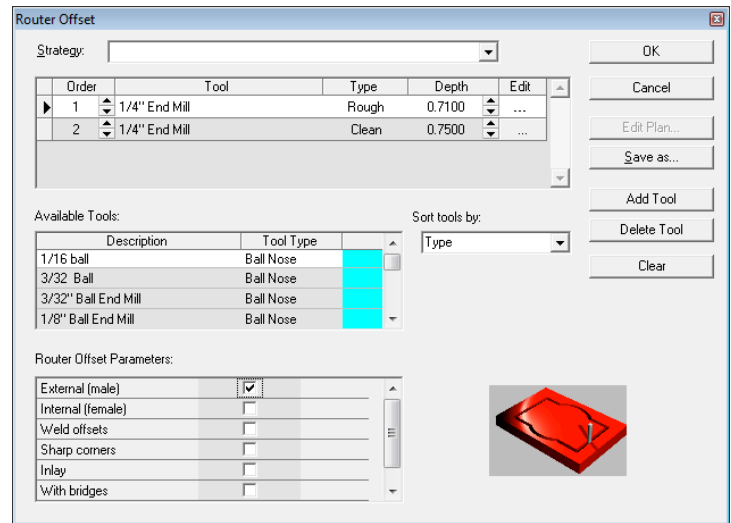
EnRoute Step-by-Step Series

15. A Routing Offset was used to cut the relief out. Select the relief.

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.

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. This will open the Cut Definition Dialog for this tool.

17. Enter the parameters for the Rough tool:

Passes = 3

Feed Rate = 100.

Plunge Rate = 50.

Spindle Speed = 14000

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

Set the parameters for the Clean Tool.

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

Cut type		Rough
<input checked="" type="checkbox"/>	<b>Depths</b>	
	Surface	0.0000 in
	Final Depth	0.7100 in
<input checked="" type="checkbox"/>	<b>Passes</b>	
	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

18. Enter these parameters.

Passes =1

Width of cut =.02

Feed Rate = 100.

Plunge Rate = 50.

Spindle Speed = 14000

Click Ok.

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

<input checked="" type="checkbox"/>	<b>Passes</b>	
	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 checked="" type="checkbox"/>	<b>Widths</b>	
	Width of cut	0.0200 in
	Number of steps	1
	Maximum step	0.2250 in
	Actual step	0.0200 in
	Shoulder?	<input type="checkbox"/>