Celtic Knot – 2D Fill

One of the most common methods of milling a design on your router is to use a 2D Fill. The fill removes material, typically using an end mill tool. As demonstrated in the example, it can be necessary to use more than one tool in order to remove the material completely and efficiently.

The following steps demonstrate how to mill out the design to create a finished piece.

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.

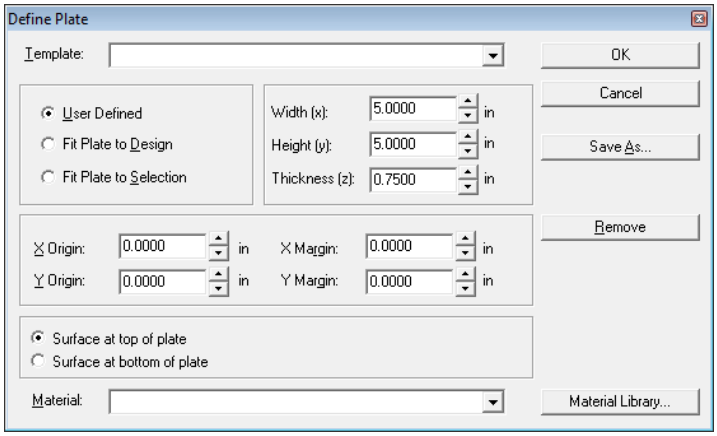
   Width 5.00

   Height 5.00

   Thickness .75

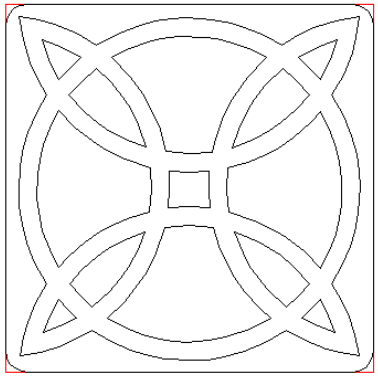
   Surface at the top of Plate.

   The plate is used primarily as a reference for the design.



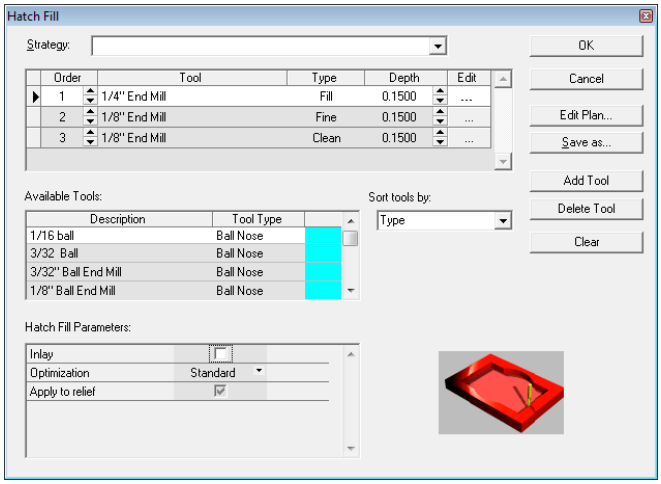
2. This is the design that we are using for this example.

   Instructions for creating this design are provided in the *Celtic Knot Introduction* tutorial.



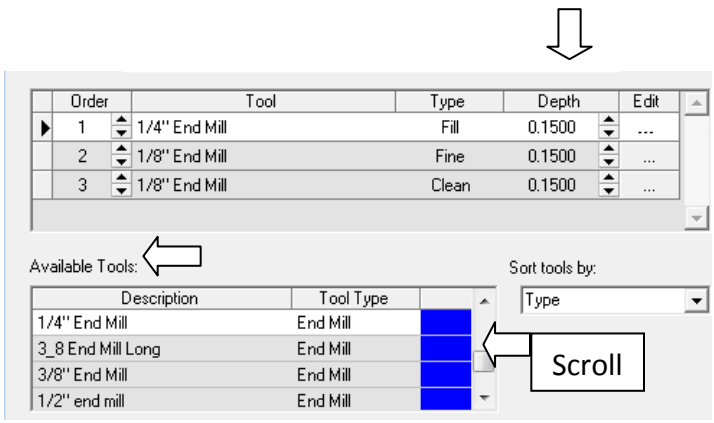
3. We want to create toolpaths that will mill down the background surfaces so that the Celtic Knot is the raised surface. To do this we will use the Hatch Fill Strategy.

   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 ¼" End Mill and ⅛" End Mill 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 ¼" End Mill as the Fill tool and the ⅛" End Mill tool as both the Fine tool and the Clean tool.

   Enter the **depth** of the cut (.15) by typing in the depth or using the arrows to set the depth.



Following is a brief description of the different cut types we are using.

Fill Cut

In 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

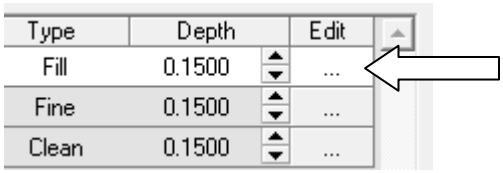
The fine cut is optional. If it is used, a tool that is smaller in diameter than the Fill tool is selected to fit into sections of the design that the fill tool was not able to fit into. This will be corners and any other thin areas that are too small for the Fill tool. The Fine cut uses an island fill 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.

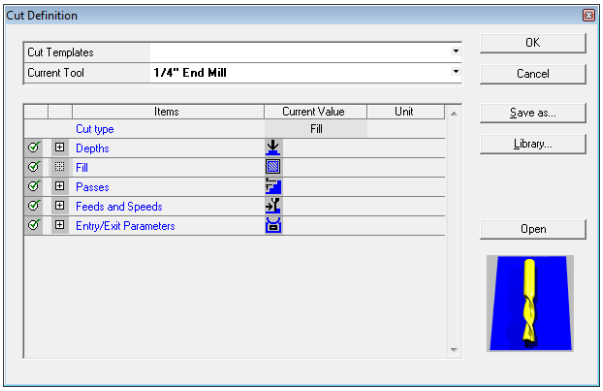
Clean Cut

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 tool is used to do the Clean Cut and the Fine Cut.

5. The next step is to edit the cut 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. There are several parameters that can be defined for each tool. The type of material that you are using will determine the parameters used. If you are using HDU foam, it cuts easier than wood or metal, so speeds can be set at a more aggressive pace.

   More specific information can be found in the EnRoute manual in the "Working With Toolpaths" section.





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

   Enter Parameters:

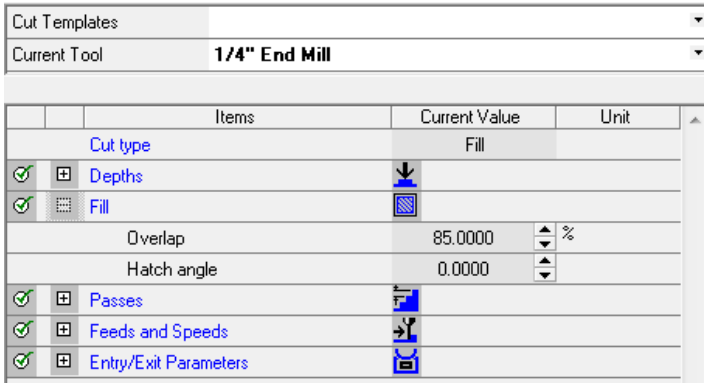
   Overlap = 85%

   Hatch Angle = 0 (If you were cutting wood, it may be desirable to cut with the grain of the wood. This parameter can be set so the direction of the tool travels along the grain of the wood.)

   Click on the + box to open the Passes category.

   Enter Parameter:

   Passes = 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 denser materials. The overlap percentage can affect the look of the finished surface of the material. When using a harder material it may be necessary to have a higher overlap to avoid chipping and to put less stress on the cutting tool.

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. This setting can be used to set the toolpaths to cut wood in the direction of the grain.

Passes

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

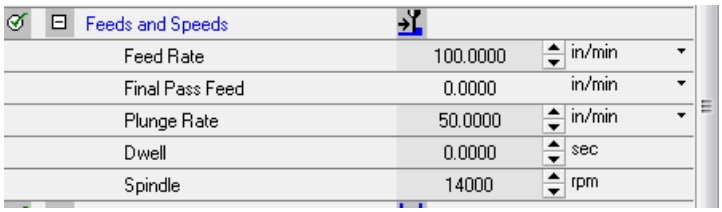
7. Click on the + box to open the Feeds and Speeds category.
   Enter Parameters:

   Feed Rate = 100.

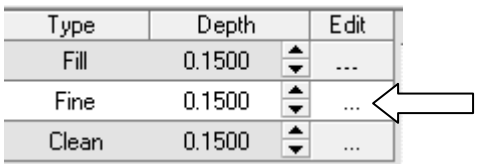
   Plunge Rate = 50.

   Spindle = 14000

   Choose parameters that are appropriate for your machine and material.



8. Once you have entered these parameters, Click OK. Then click in the Edit box for the next tool in the list.



9. Enter the parameters for the Fine Cut.
   Overlap = 70%

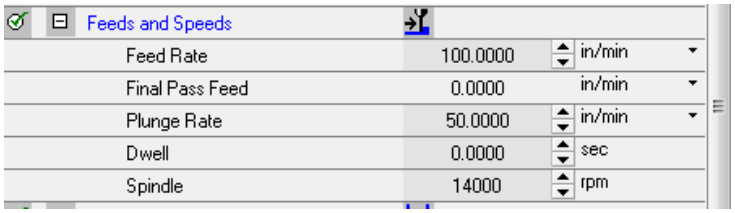
   Passes =1

   Feed Rate = 100.

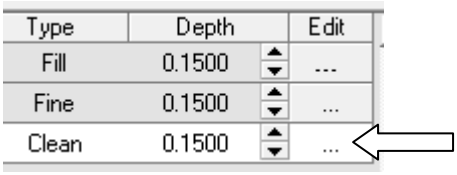
   Plunge Rate = 50.

   Spindle = 14000

   Choose parameters that are appropriate for your machine and material.



10. Click OK. Then click in the Edit box for the next tool in the list.



11. Enter the parameters for the Clean Cut and then click **OK**.
    Passes =1

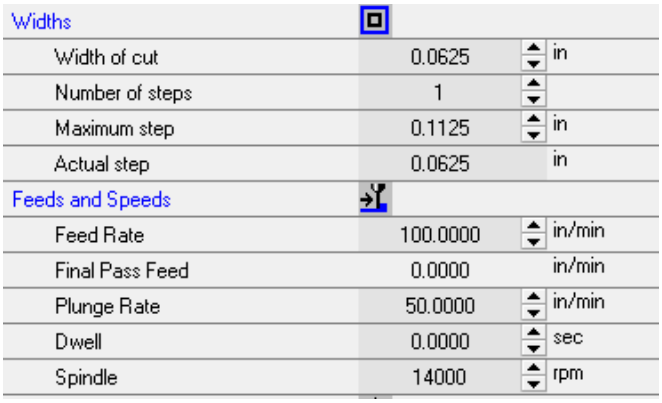
    Width = .0625

    Feed Rate = 100.

    Plunge Rate = 50.

    Spindle = 14000

    Choose parameters that are appropriate for your machine and material.



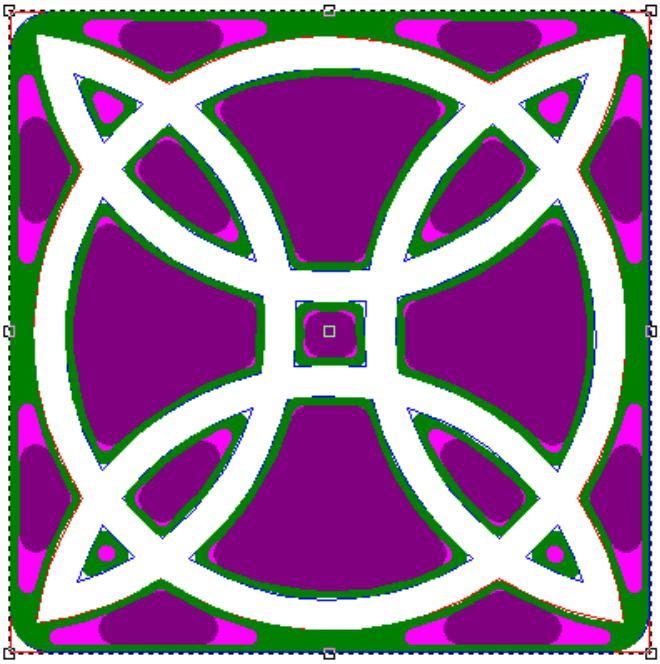
12. Click Ok in the Hatch Fill Dialog. EnRoute will then calculate the toolpaths.

    This image shows the toolpaths. Press F9 to show the thickness of the toolpaths.

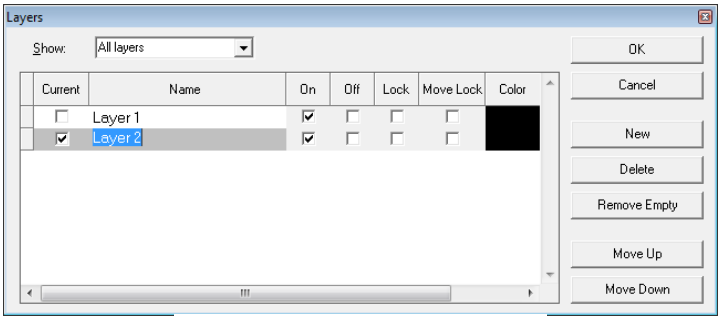
    Purple = Fill Cut

    Pink = Fine Cut

    Green = Clean Cut



13. Create another layer and copy the outline of the design to that layer.

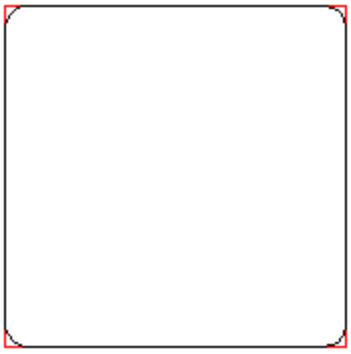


To create a new layer, click on the Layers Icon.  This will open the Layers Dialog.

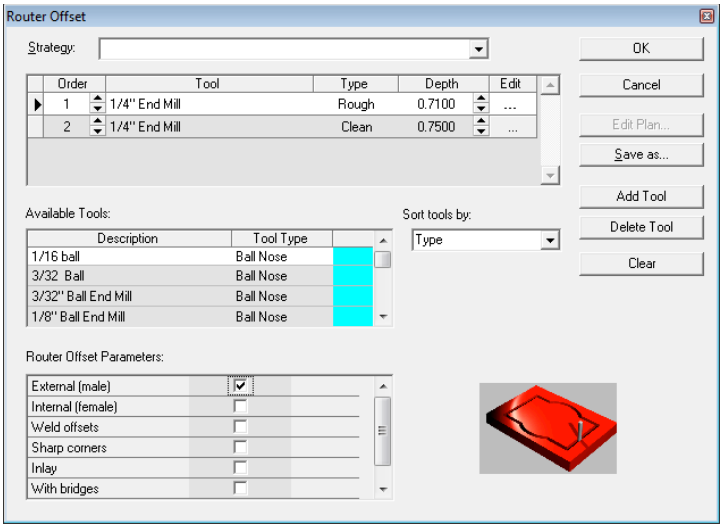
Click on the New button. Notice that a new layer will be created and named Layer 2.

Highlight the name. Type in the new name for this layer.

Click OK to accept the changes.

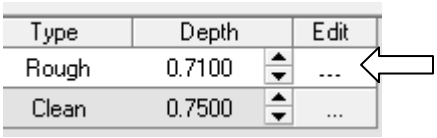


14. To cut the piece out, a Routing Offset Strategy is used with a clean cut. The clean cut is used to improve the quality of the finished cut. Select the outline contour and click on the Routing Offset icon to open the Dialog. 



A ¼" End Mill was used to cut out the object. 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 same ¼" End Mill tool for 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. This will leave just a small amount of material to cut in the Clean cut pass and therefore will help to hold the object in place while that pass is cutting.



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

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

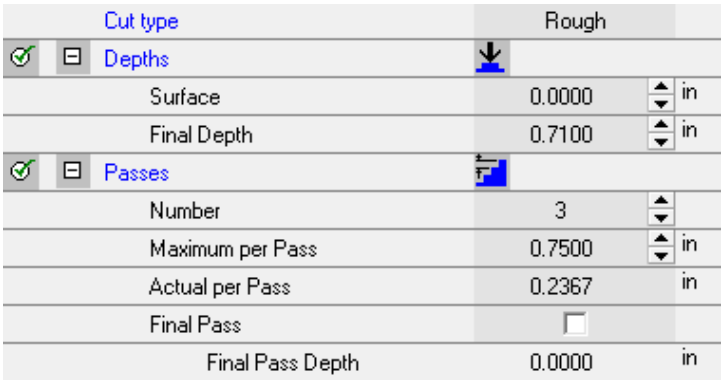
    Feed Rate = 100.

    Plunge Rate = 50.

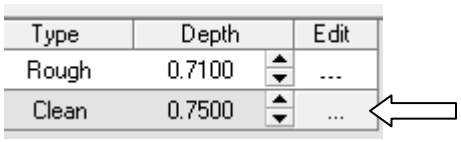
    Spindle Speed = 14000

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

    Choose parameters that are appropriate for your machine and material.



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



17. Enter these parameters.
    Passes =1

    Width of cut =.02

    Feed Rate = 100.

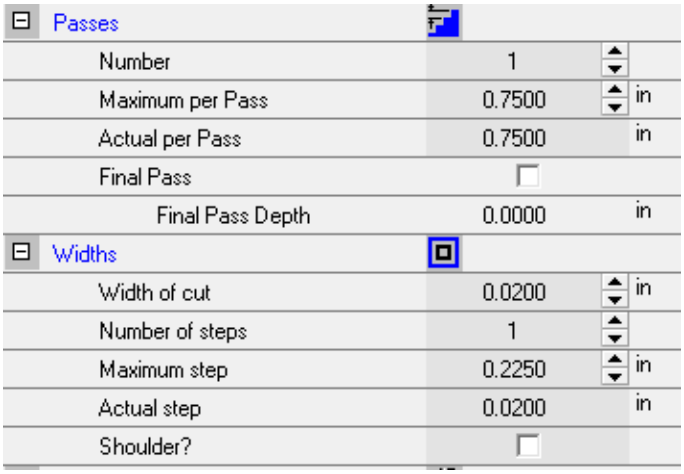
    Plunge Rate = 50.

    Spindle Speed = 14000

    Click Ok.

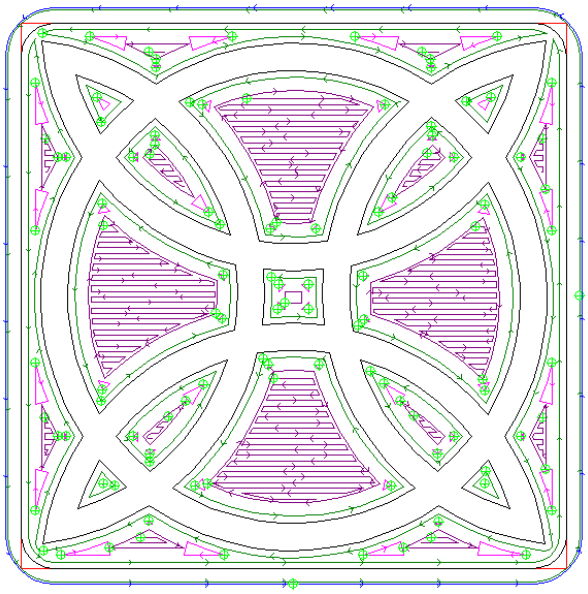
    Click OK again in the Routing Offset Dialog to process the toolpaths.

    Choose parameters that are appropriate for your machine and material.



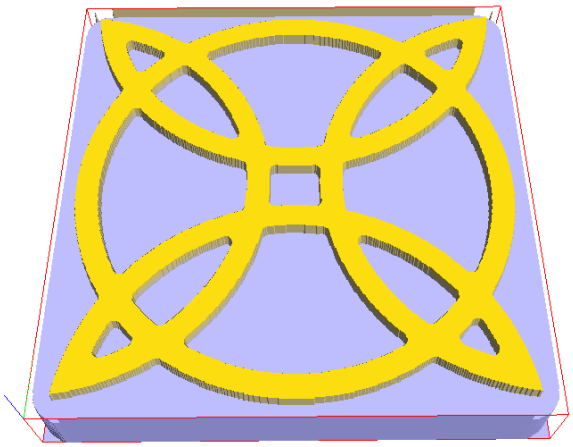
18. This image shows all of the toolpaths.

    Note the added Routing Offset toolpaths around the perimeter.



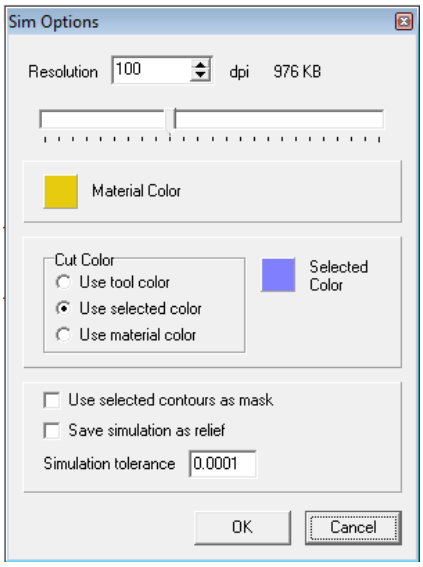
19. Using the Simulate Ortho Tool allows you to see a rendered image of the toolpaths that you have applied. This is a good way to see a preview of the toolpaths. If you notice an unintended result in the simulation piece, you can then correct it before you actually cut your piece.

    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.

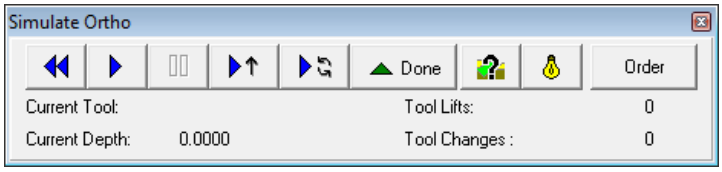


20. Click on the Simulate Ortho Icon.  This will open the Simulate Options Dialog.
    Enter the Simulation Options:
    In this example we have set the resolution at 100dpi. The Selected color has been set to a blue color, and the Cut color parameter has been set to Use selected color. The simulation tolerance is set at .0001.

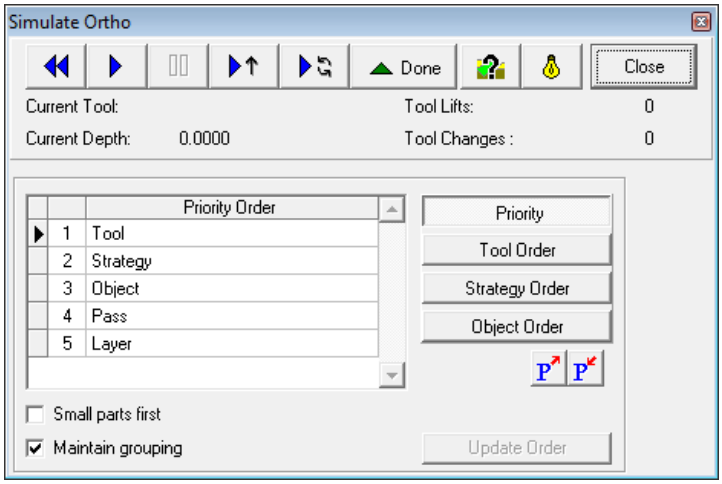
    Click OK. This will bring up the Simulate Ortho Dialog.



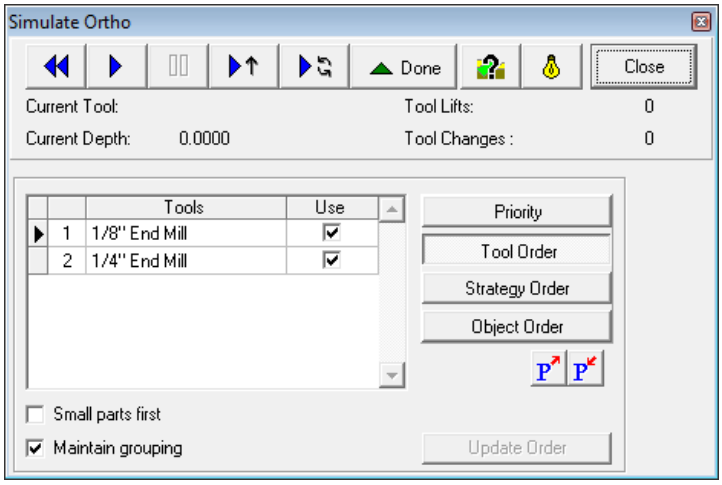
21. To set the cut order, click on the Order button to open the dialog.



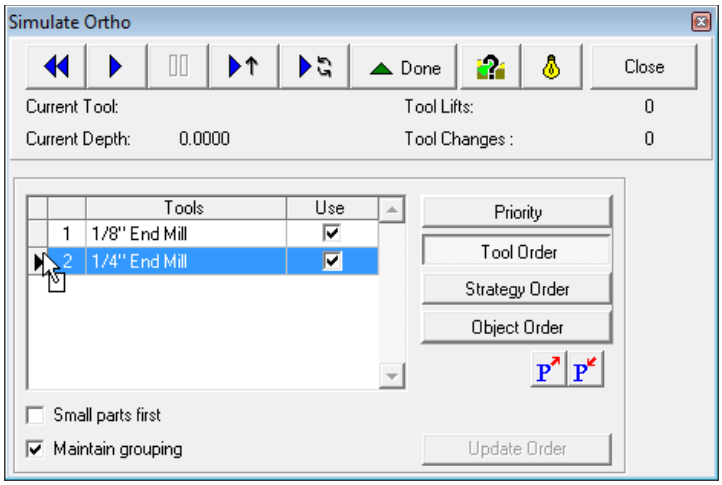
22. The software determines the cutting order based on a number of different parameters. You can read in detail about these parameters in the "Output to Machine" section in the EnRoute manual.
    Clicking on the Priority button displays the Priority Order list: Tools, Strategy, Object, Pass, and Layer. The order of the list determines how EnRoute will calculate the order of the cut. In this example, the tool order will have top priority.



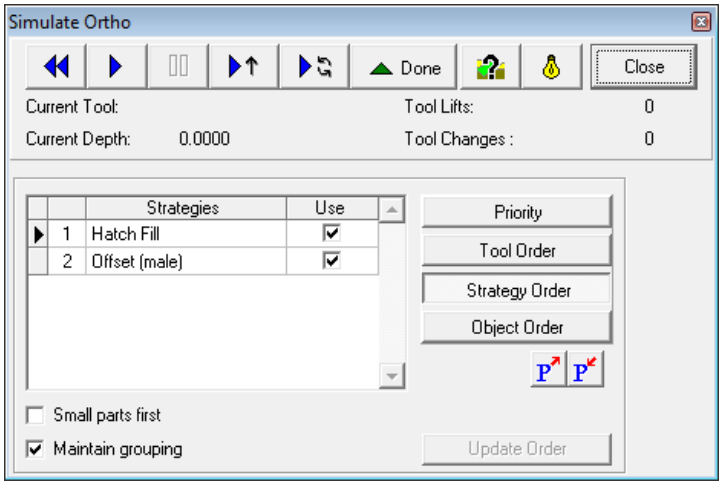
23. Click on the Tool Order button to open the section that displays the order of the tools. In this example the ⅛ “tool is listed first. All of the toolpaths associated with this tool will come first in the order of cut.



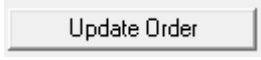
24. If you wanted to change the order of the tools, click on the row header on the left edge of the entry to select, (the row will highlight) then click and drag the entry up or down by the header.



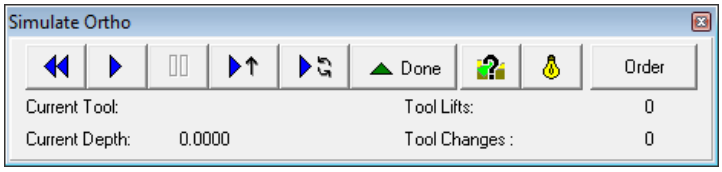
25. The Strategy Order button allows you to decide the order that you would like the strategies to cut.



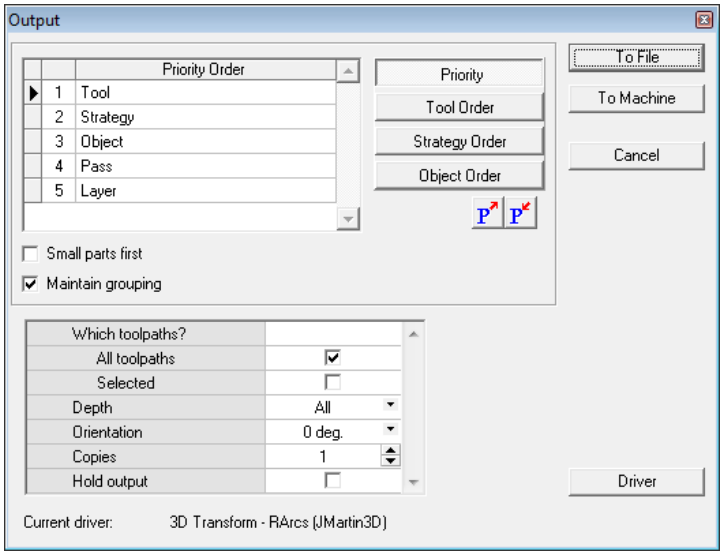
26. Once you have made the changes to the order of cut, click on the Update Order button.



27. Click on the arrows to simulate the cut order. There are several choices for the simulation. Experiment with the different options to see the results of the tool order.

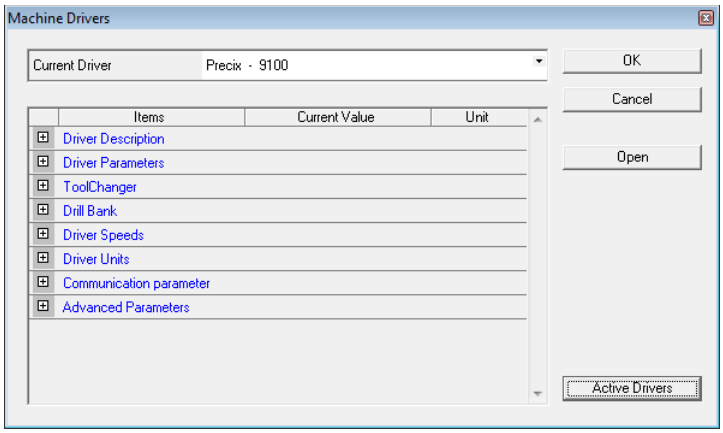


28. Once you have established that the tool order is the way you would like it, you can then output the toolpaths. Click on the Output Option Icon. This will open the Output Dialog. The ordering changes that we just described can be made here also. In this dialog, you can also choose which toolpaths to output. Under "Which toolpaths?" Choose either All Toolpaths or Selected Toolpaths. More specific information about selecting the Output parameters can be found in the EnRoute manual in the "Output to Machine" section.
    At the bottom of the page, the Current driver is listed.

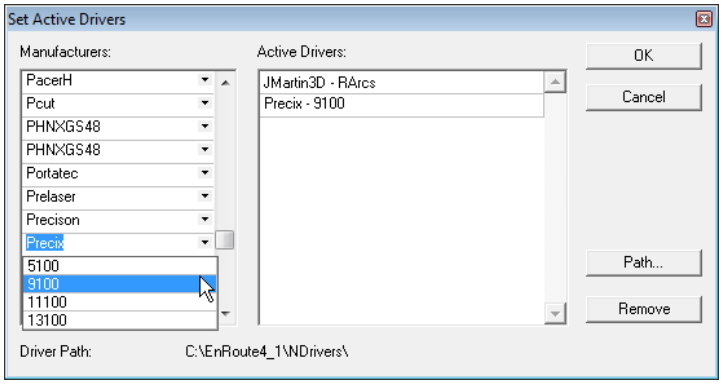


29. If this is not the driver for your machine, click on the Driver button to select the correct driver.

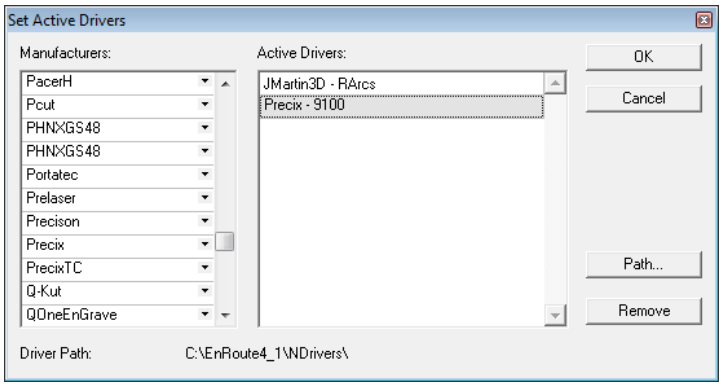
30. This will open the Machine Drivers Dialog. Click on the Active Drivers button at the bottom right to select the correct driver for your machine.



31. The Set Active Drivers Dialog will open. Scroll down the left side of the dialog until you locate the correct driver. Double click to add it to the list of Active Drivers.



32. In the Active Drivers list, highlight the correct driver and click OK.



33. Click Ok again in the Machine Drivers dialog. You will then notice that the correct driver has been updated at the bottom of the output dialog.

    Once you have made the toolpath selection, click on the To File button. Select the file where you want your toolpaths to be saved and Click OK.