Contents

TECHNO / ENROUTE TRAINING

EnRoute: 1.1 - Starting a New File

- This section covers the procedure for opening a new file in EnRoute.

EnRoute: 1.2 – Initial Setup

- This section will go over the program preferences as well as setting up the correct machine driver.

EnRoute: 1.3 – Installing the Driver

- This section will describe how to install a new driver in the EnRoute software.

EnRoute: 2.1 – Layers

- This section shows how to use layers in EnRoute.

EnRoute: 2.2 – Tool Paths

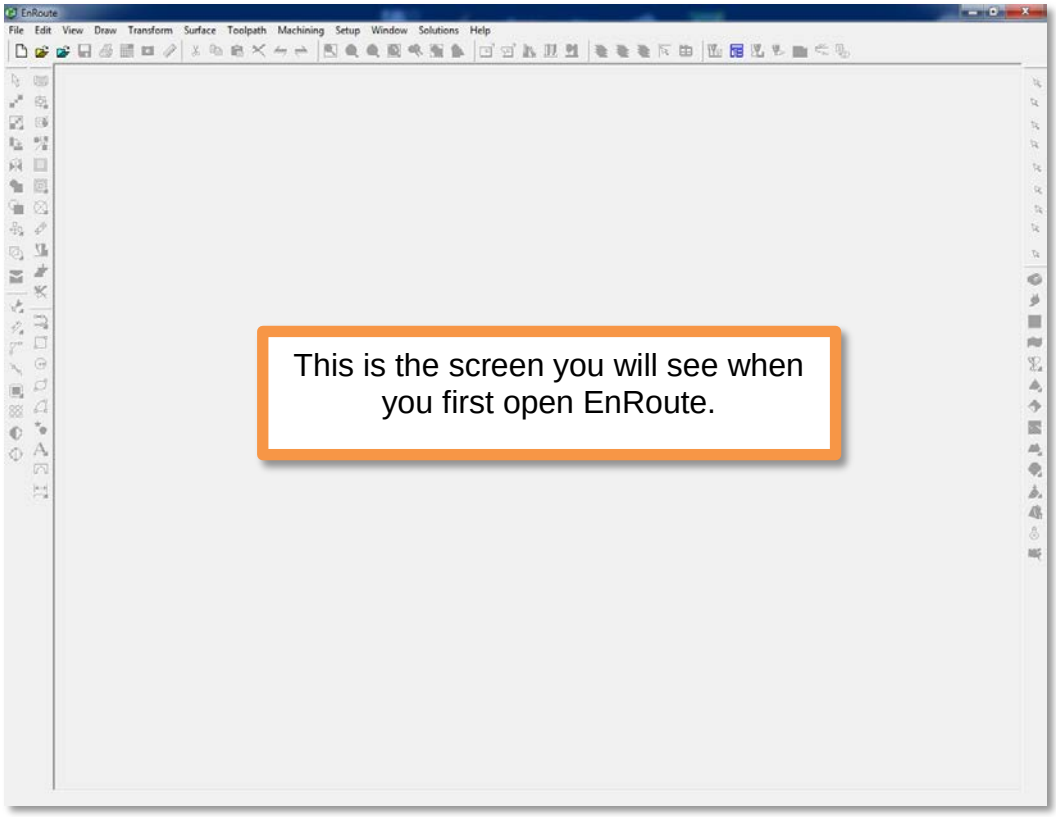
- This section covers the basics on common types of tool paths.

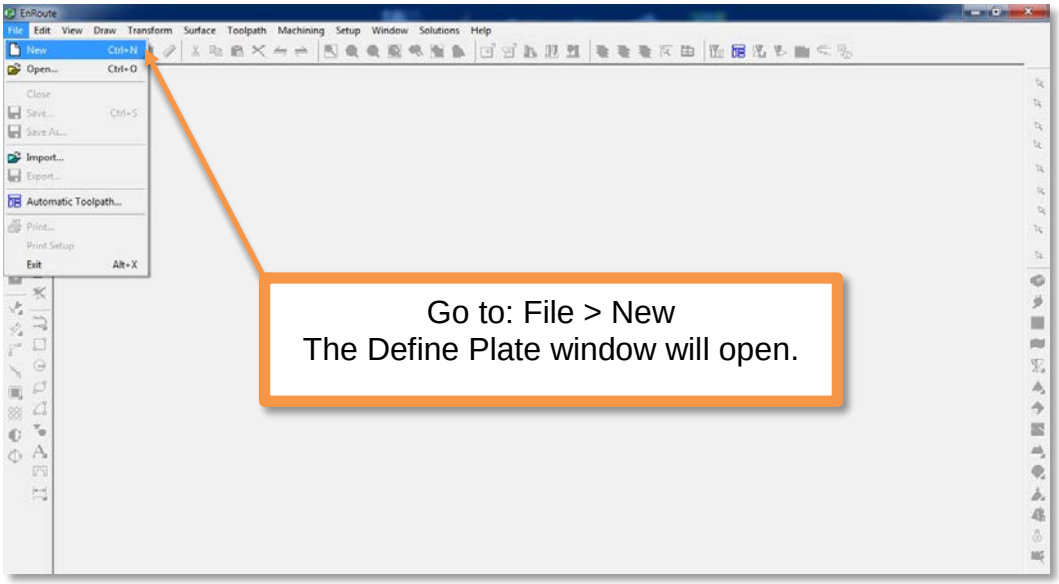
EnRoute: Project 1
EnRoute: Project 2
EnRoute: Project 3

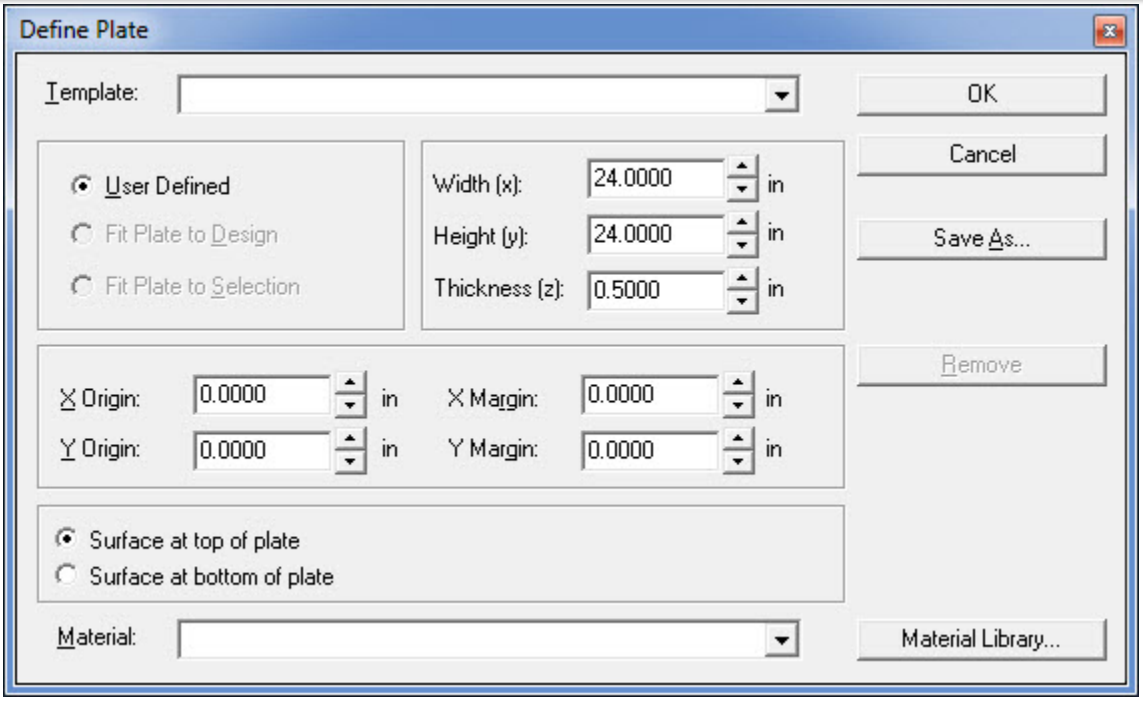
Safety Instructions / Proper Colleting

EnRoute: 1.1 - Starting a New File

This section covers the procedure for opening a new file in EnRoute.

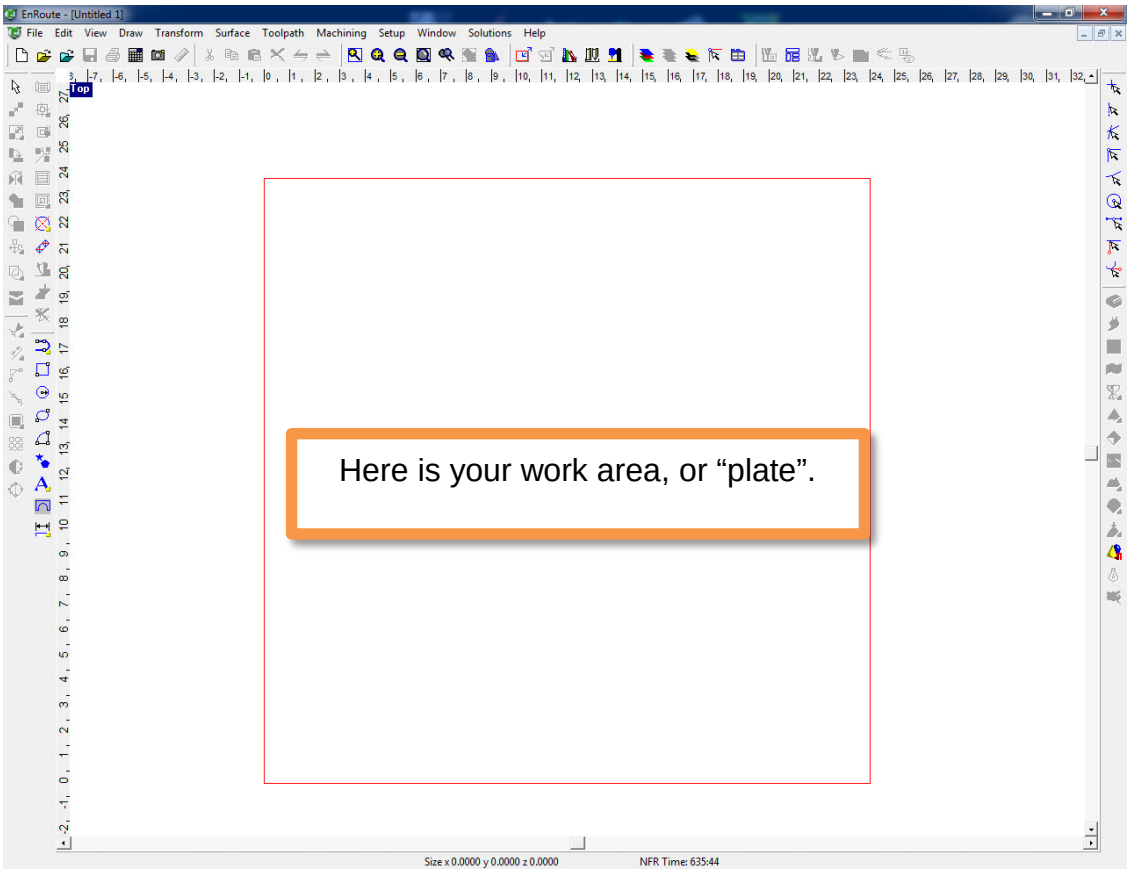


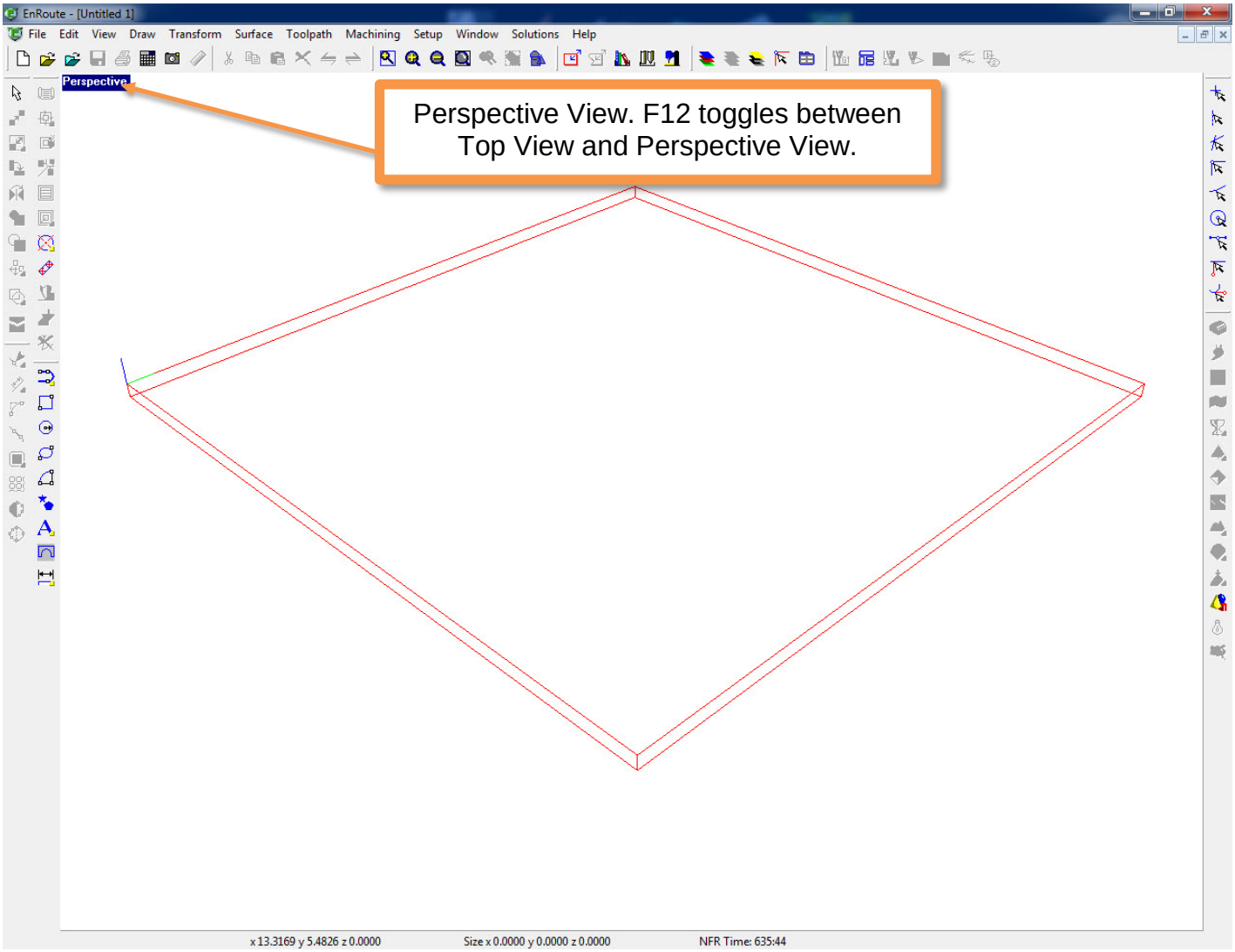




This box is where you will define your material size and origin position. For most cases, you will leave the X and Y origin at 0 and the surface at the top of the plate. This means that when you bring your output file to the machine, you will set your origin at the top of the material, in the lower left corner.

Note: The plate size does not limit your work area. It is there to help visualize where your cuts will be in relation to your material.



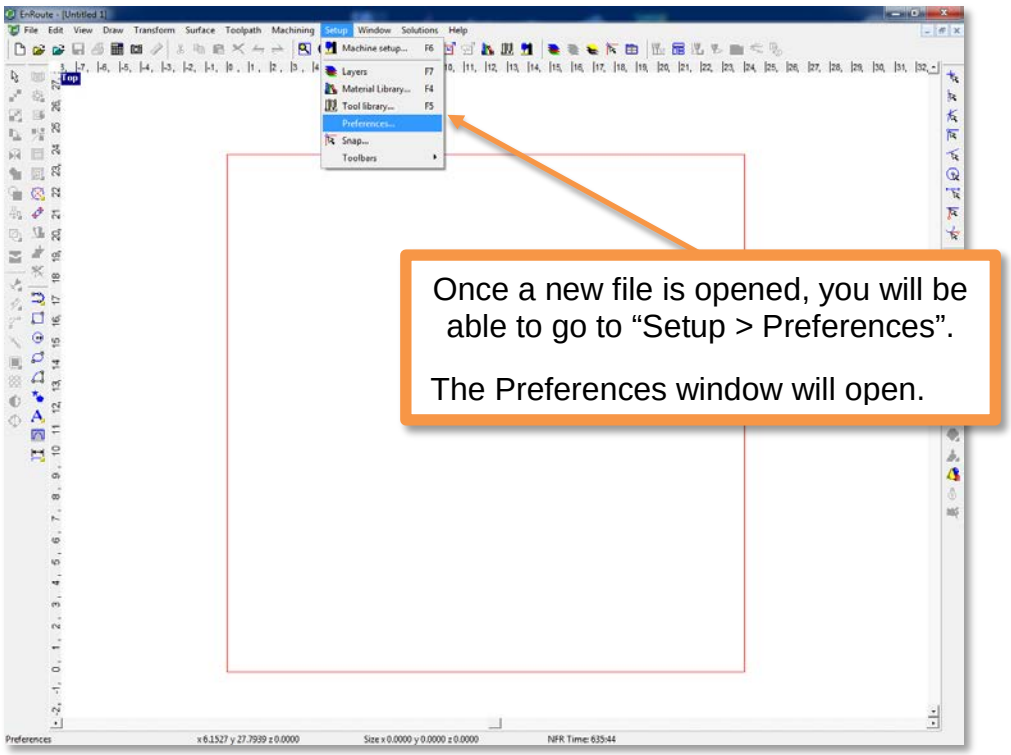


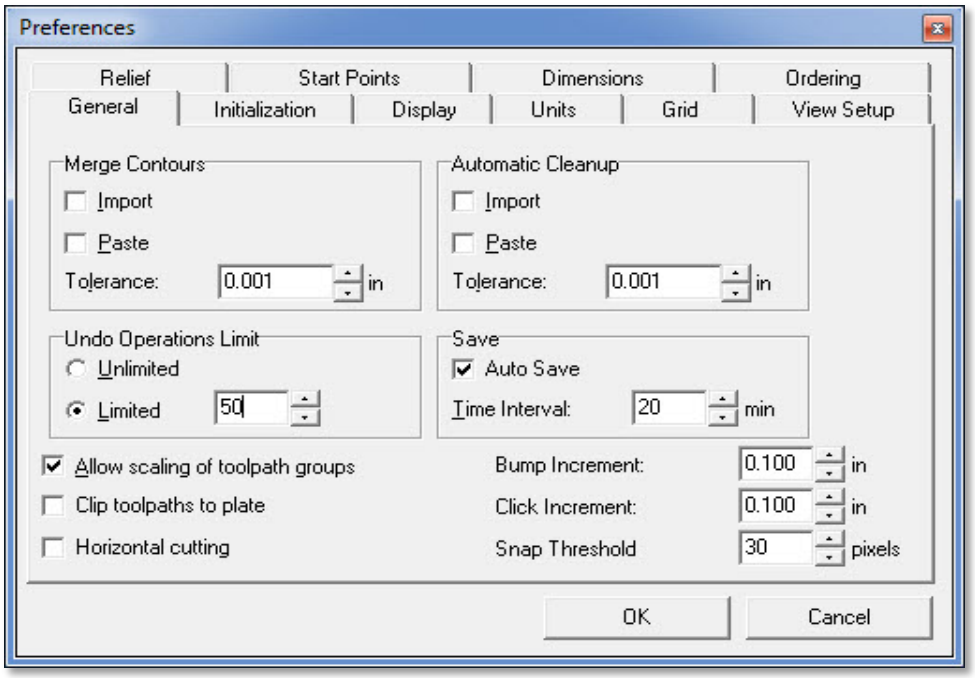
This will be the procedure each time you create a new file in EnRoute.
Plate definitions can be saved as templates for future use and materials can be set up for default cut values.

Next we will go over the initial setup of EnRoute.

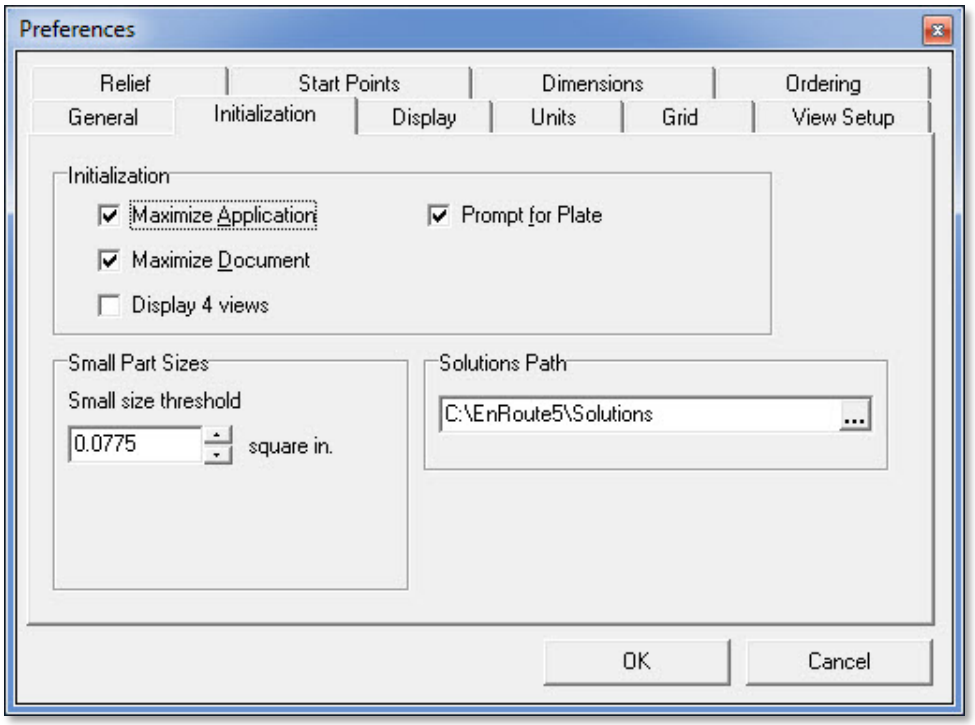
EnRoute: 1.2 – Initial Setup

This section will go over the program preferences as well as setting up the correct machine driver.

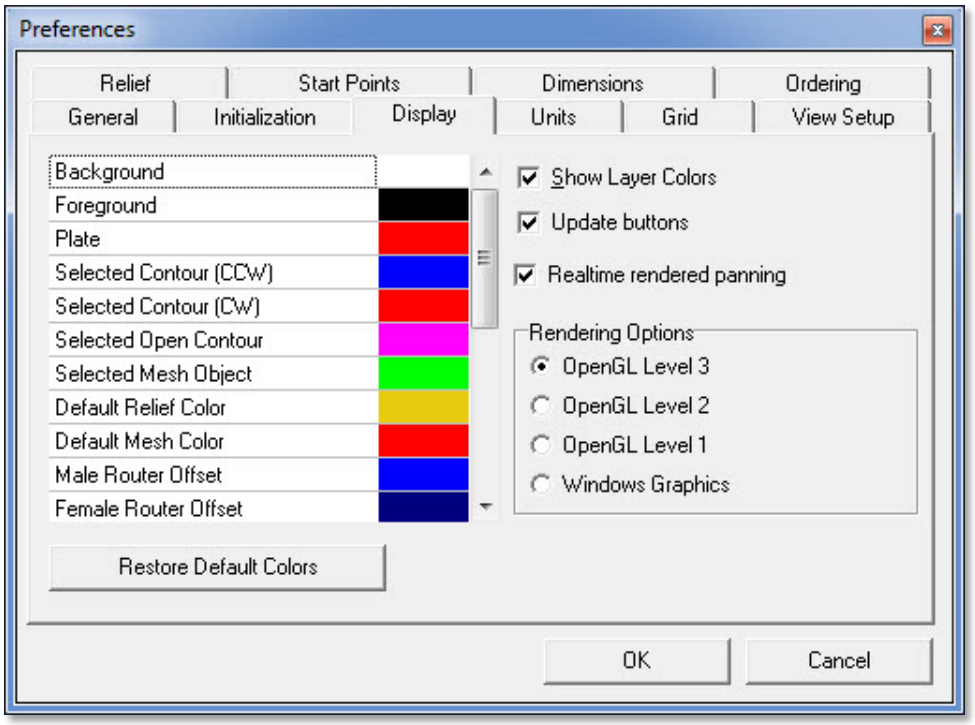




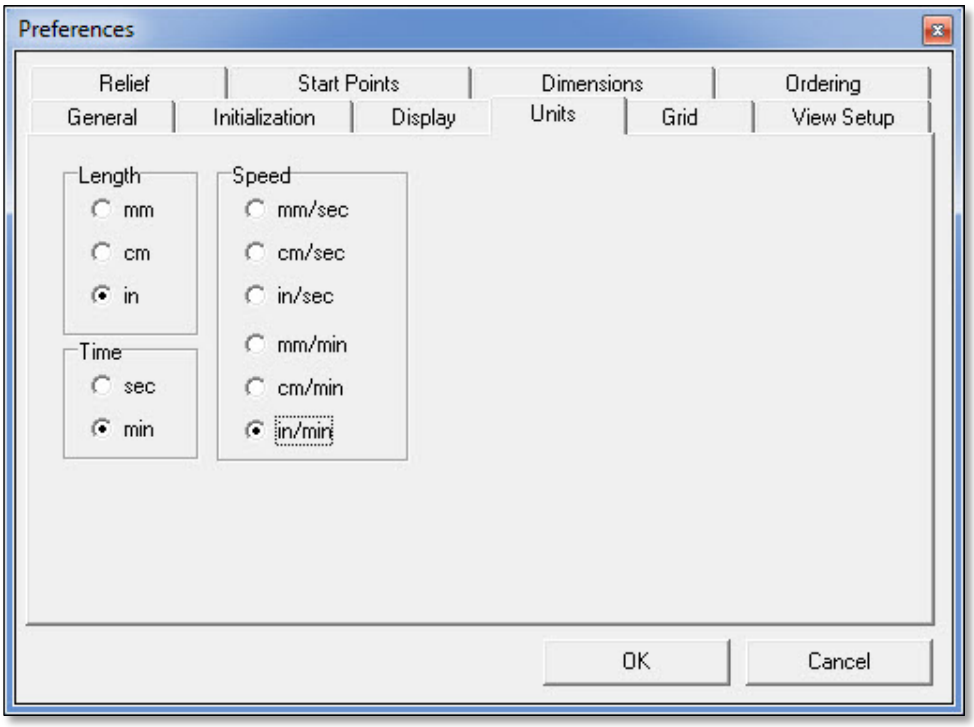
Here are the general settings



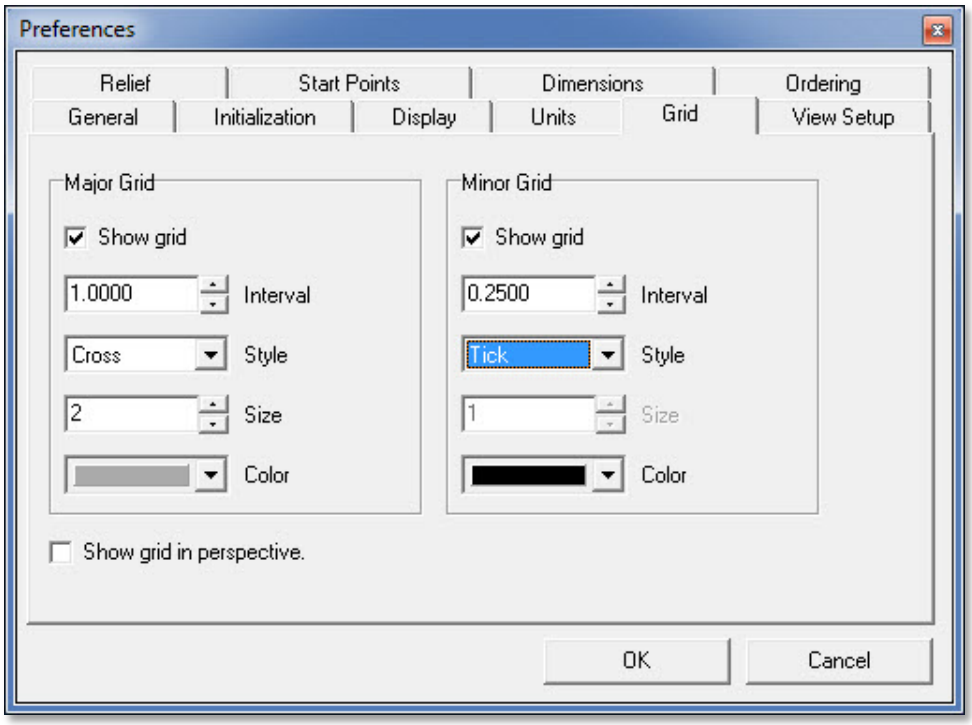
Initialization settings.



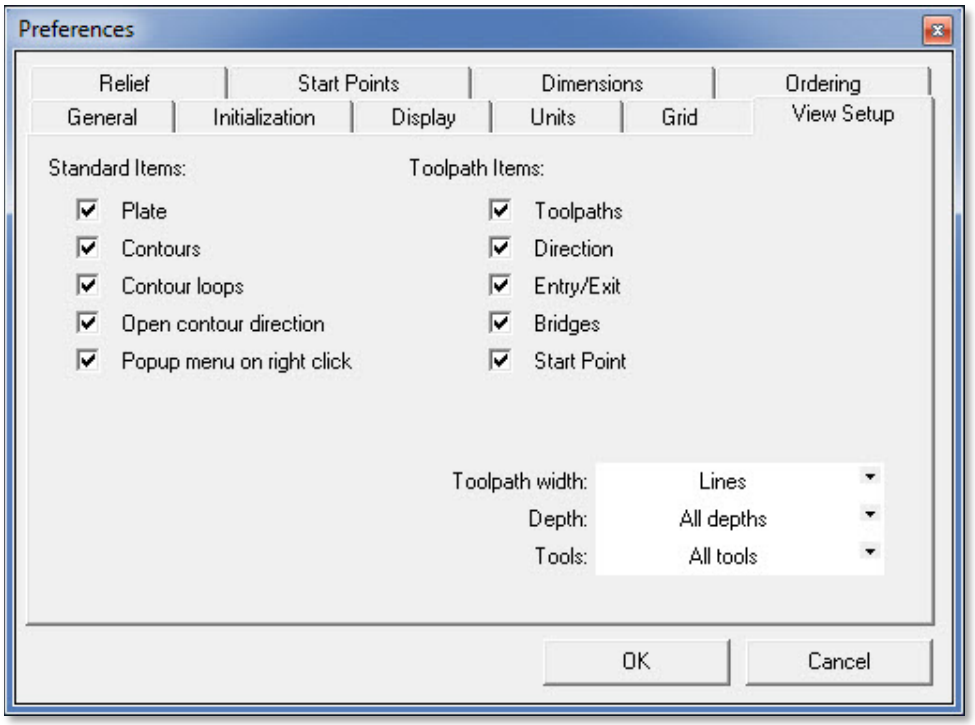
Display and graphics settings.



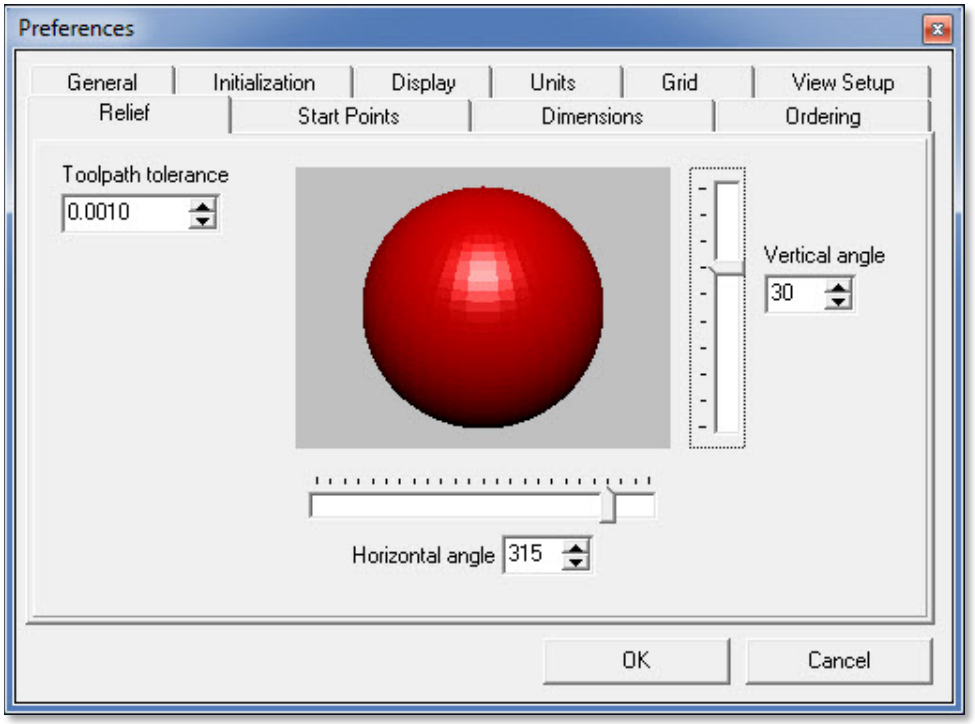
Output units. These should be set to inches and inches per minute.



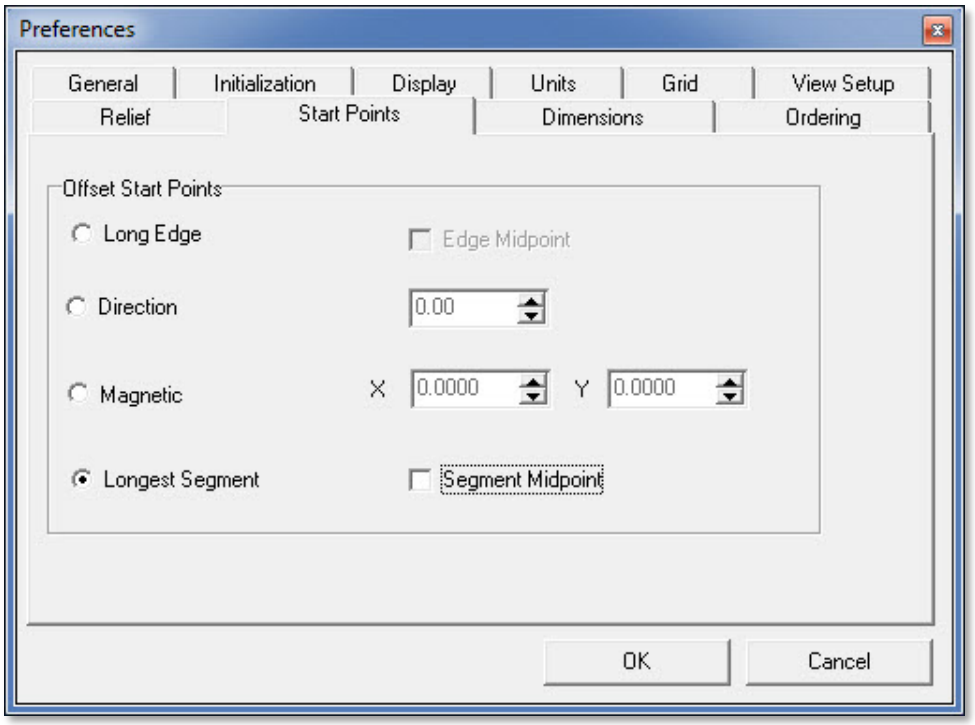
Grid settings.



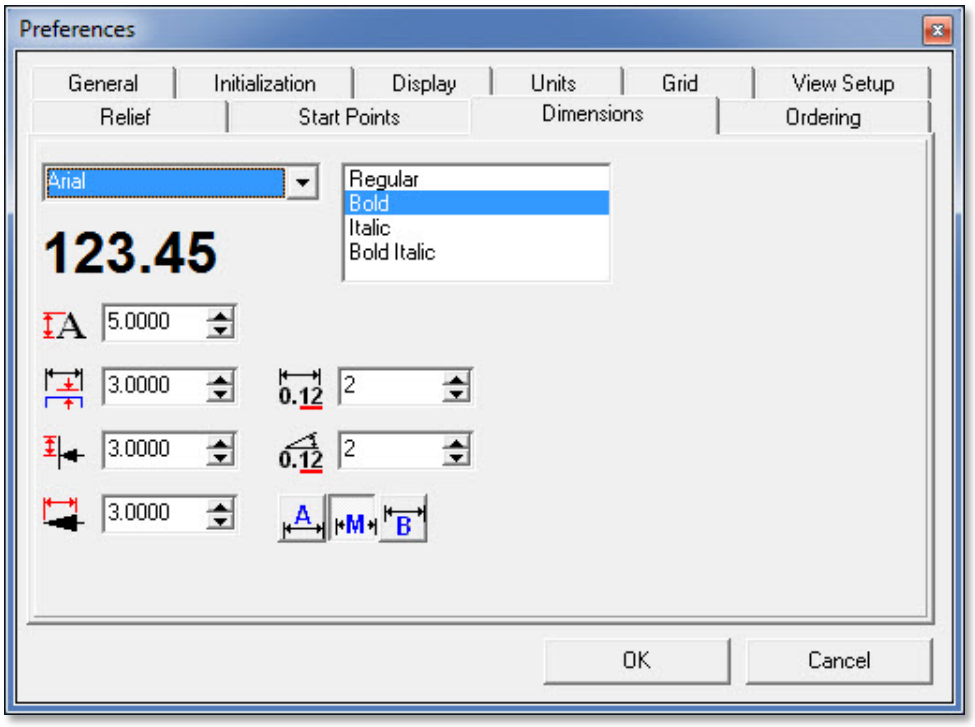
View settings. I prefer to have everything visible.



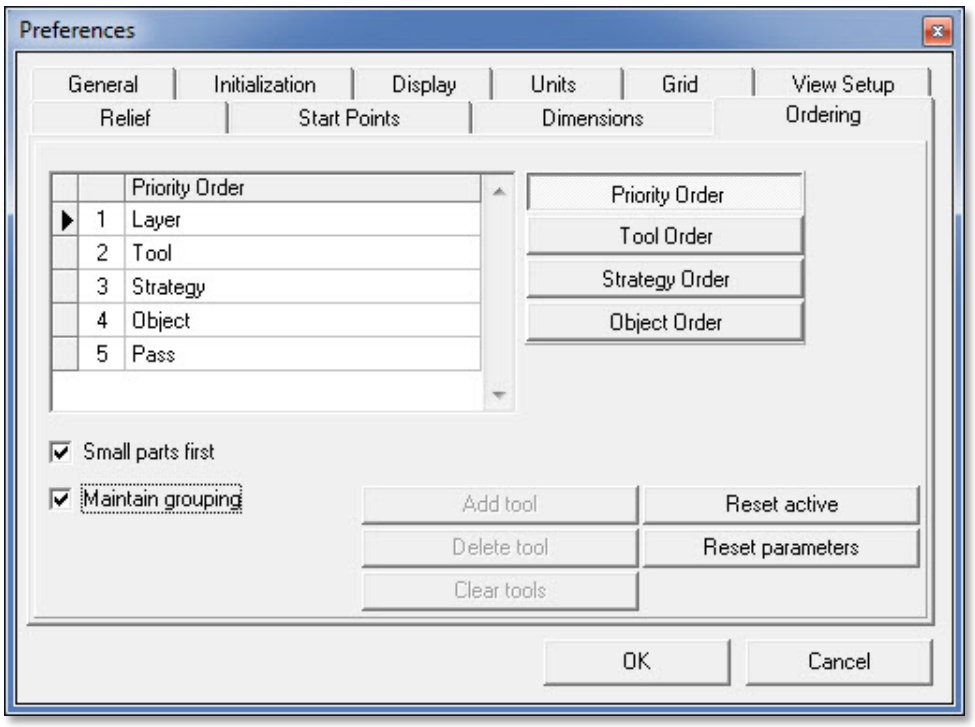
Rendering settings for 3d models.



Start points for tool paths.



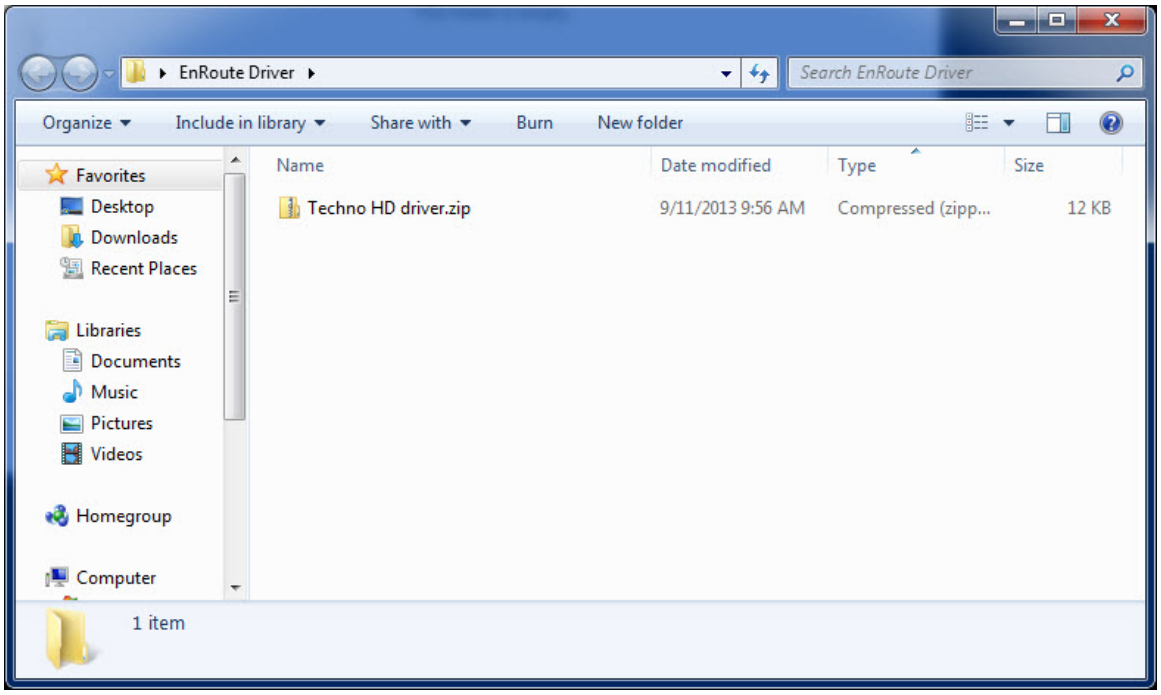
Settings for dimensions.



This screen sets the defaults for your output order. I prefer to output in layer order, so I drag “layer” to the top priority. In order to re-arrange the priority order you must click on the number, release the click, and then click a second time. You will see the mouse curser change to show a little white box under the pointer. At that point, you can drag to the new desired position.

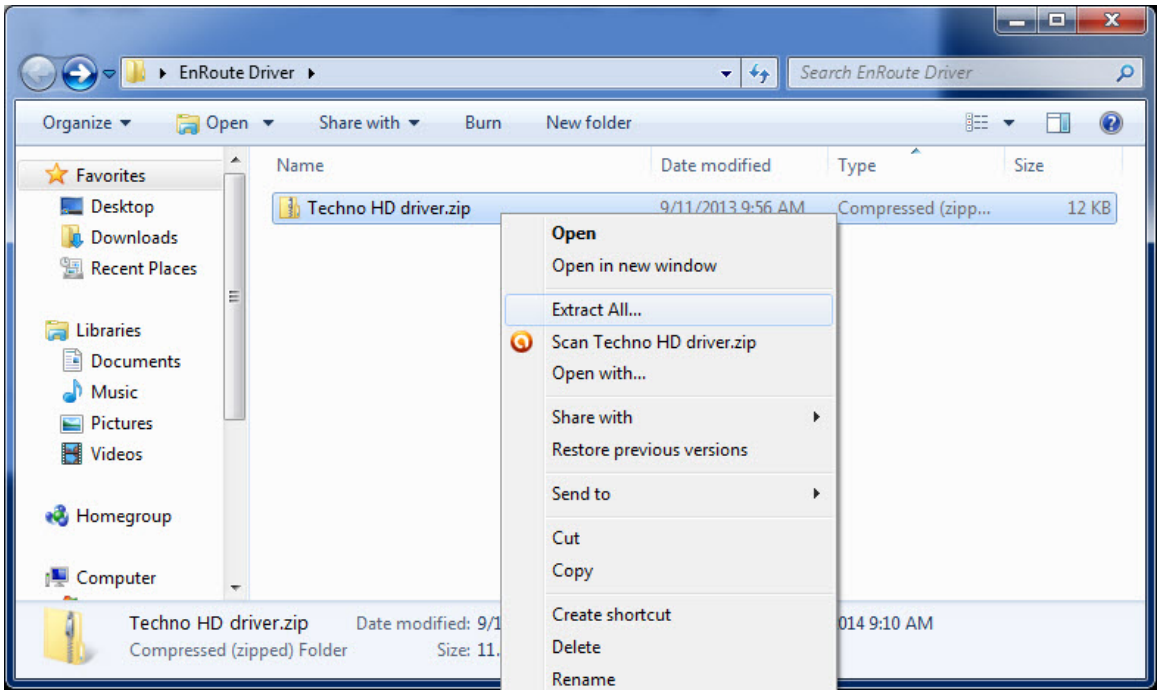
EnRoute: 1.3 – Installing the Driver

This section will describe how to install a new driver in the EnRoute software.

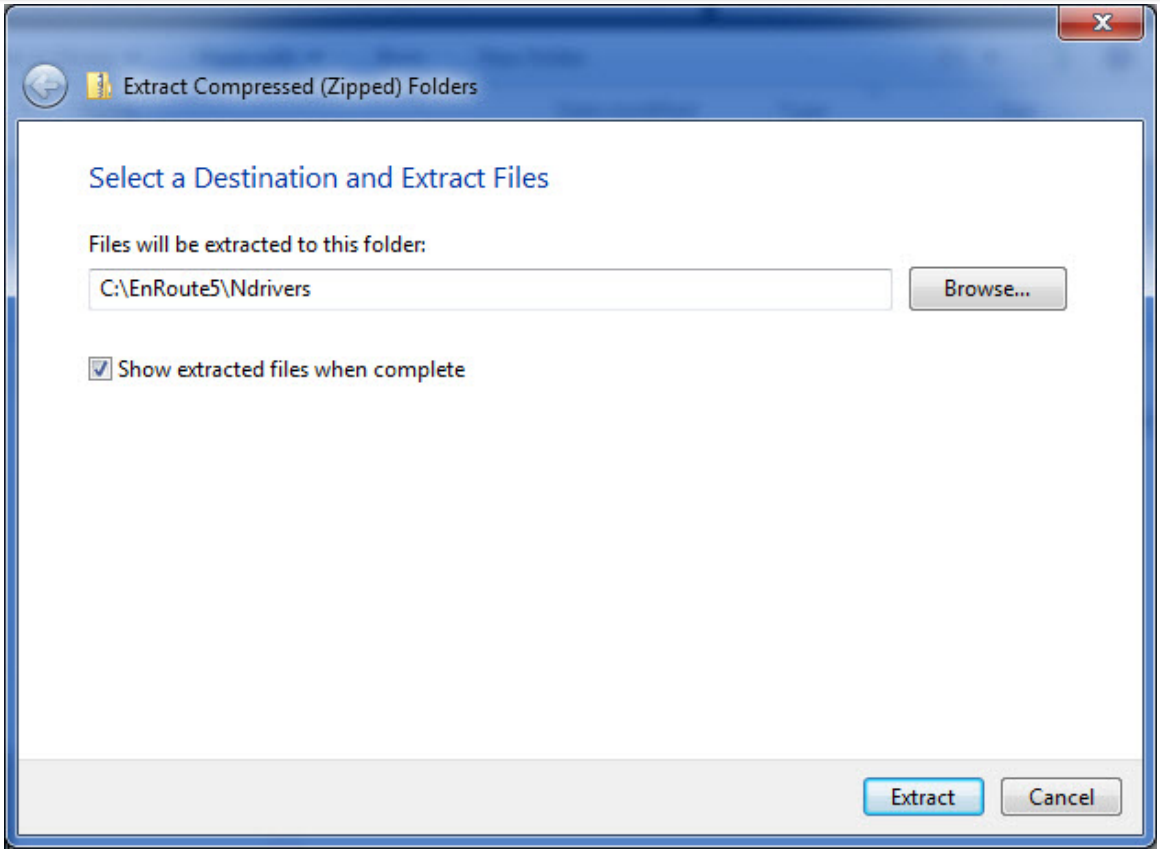


Download the HD driver from Techno's support site:

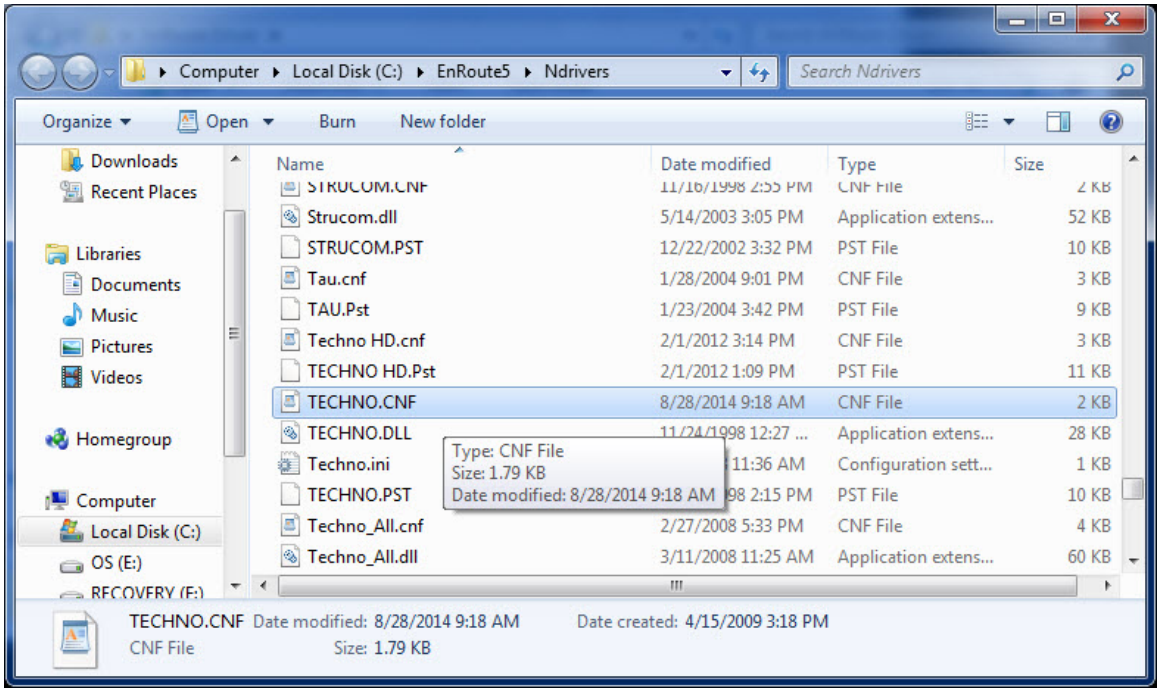
support.technocnc.com



Right click on the ZIP file and select "Extract All".

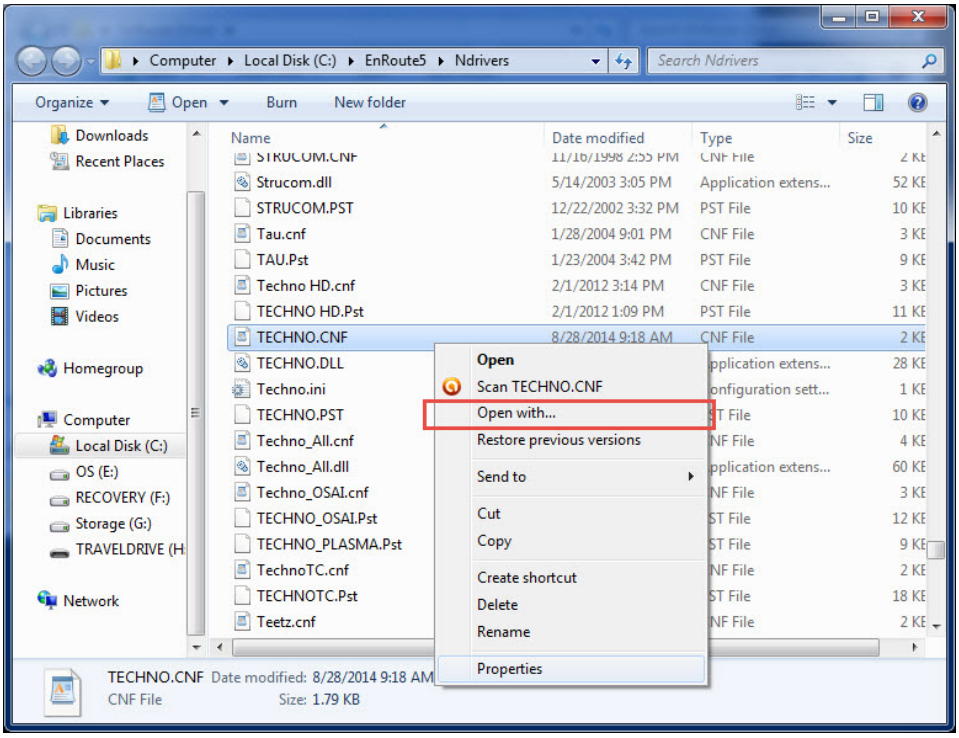


Click “Browse” and navigate to “C:\EnRoute5\Ndrivers” as the location to extract the driver to.

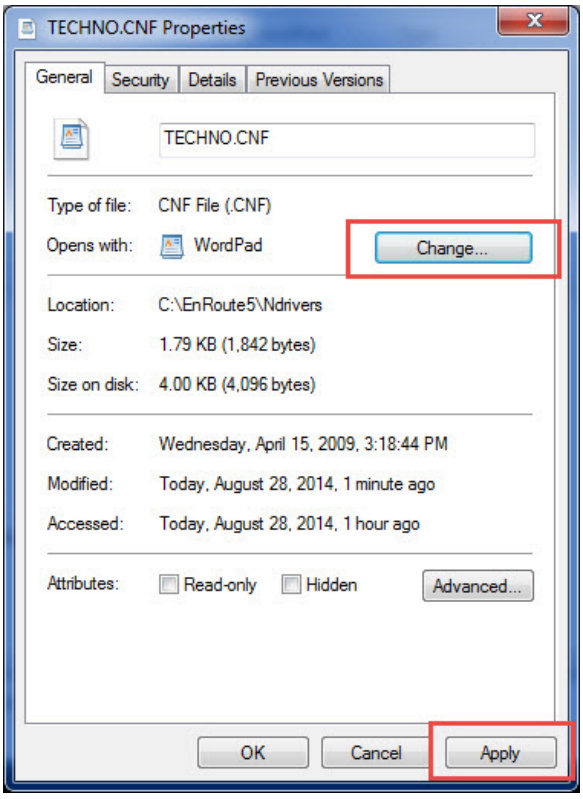


The file location should open in a new window. Locate the file “Techno.cnf”.

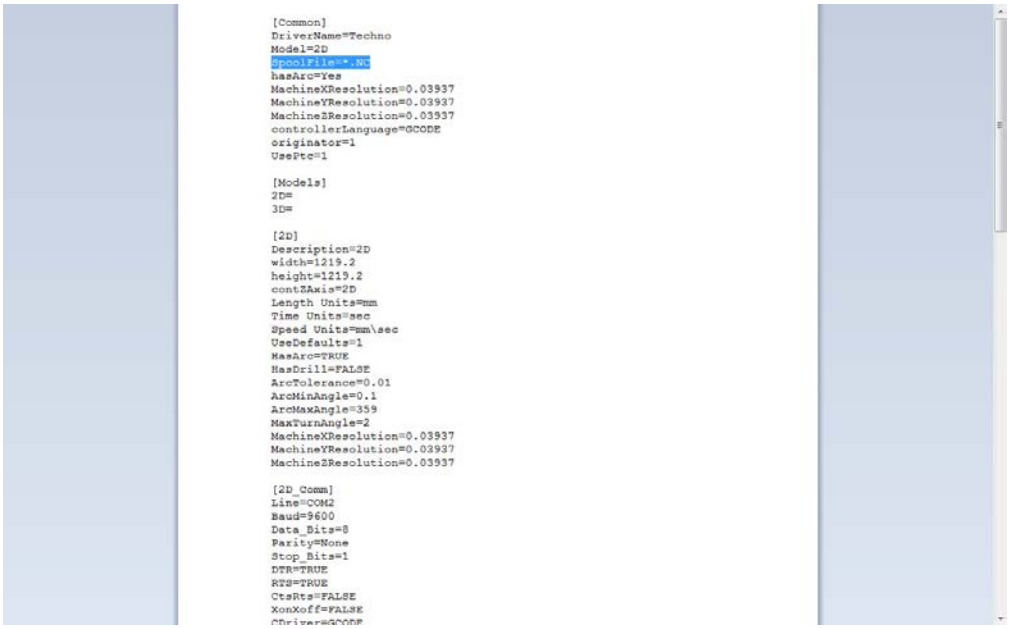
Note:
This file may not show an extension but may be labeled as a “speed dial” file.



Right click on the file and select “Open with…”



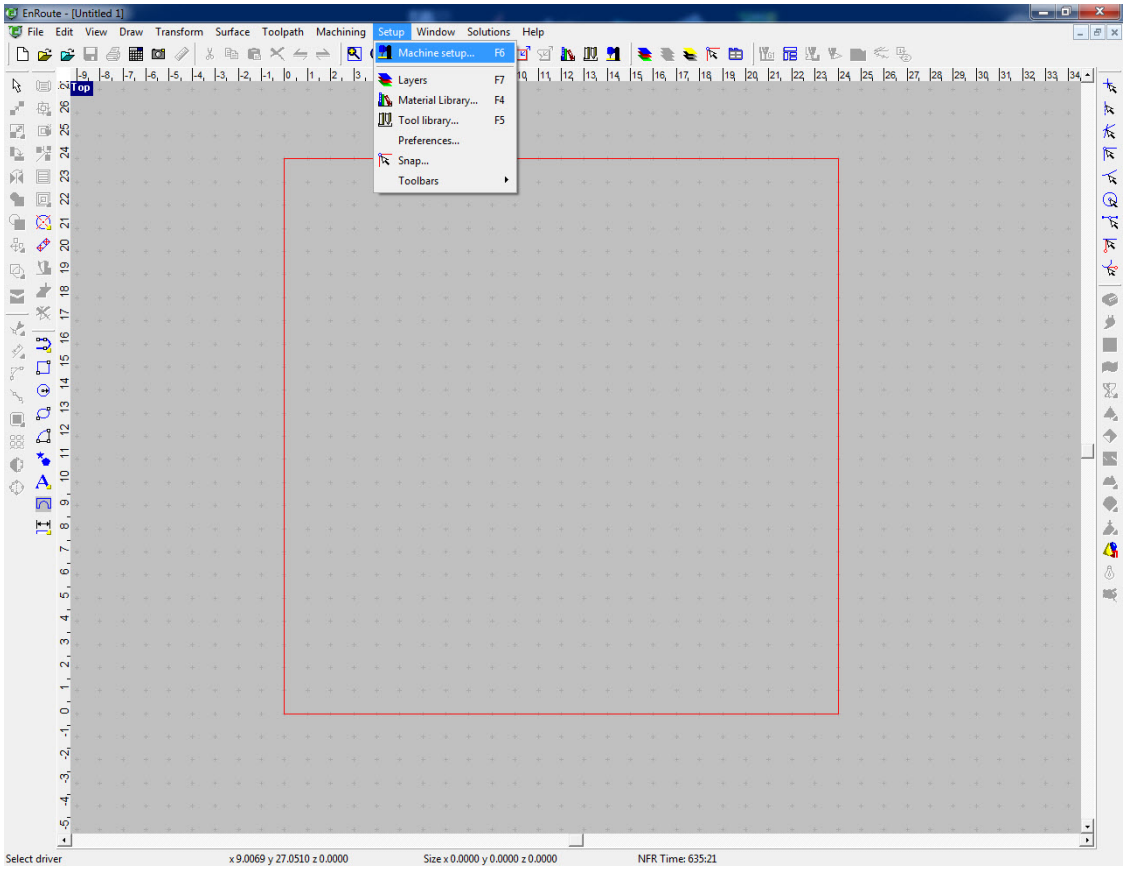
Click on “Change” and select Wordpad from the list of programs.
Then click “Apply” and “OK”. Double click the file to open it.



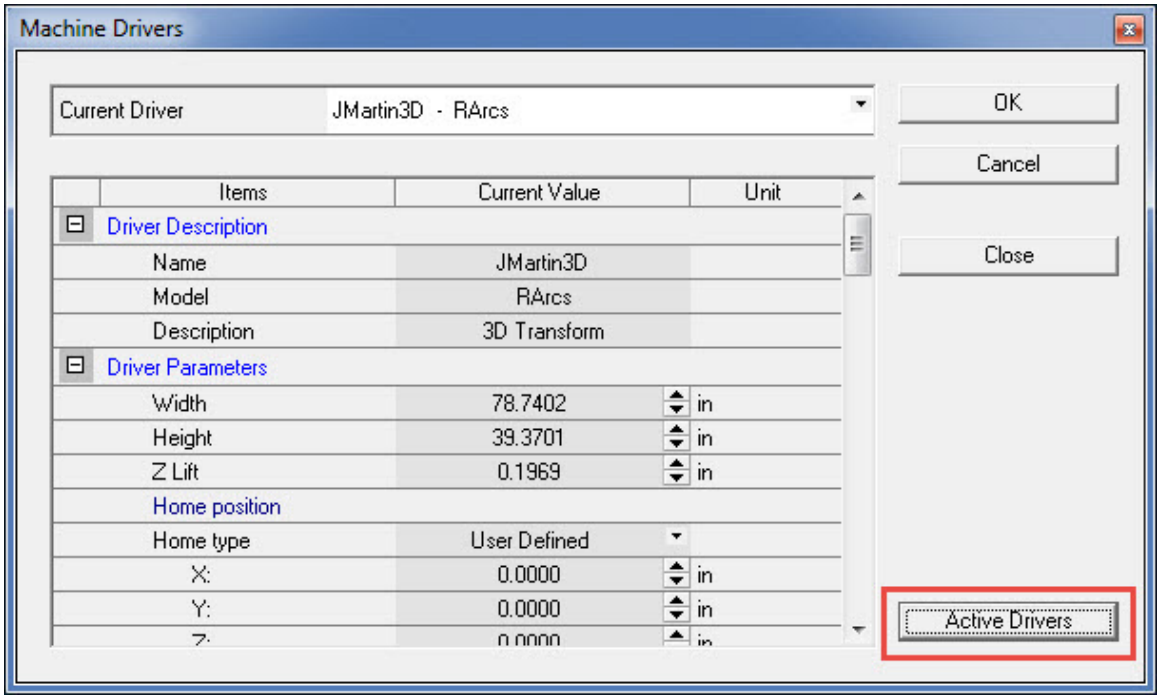
```
[Common]
DriverName=Techno
Model=2D
SpoolFile=*.NC
hasArc=Yes
MachineXResolution=0.03937
MachineYResolution=0.03937
MachineZResolution=0.03937
controllerLanguage=GCODE
originator=1
UsePtc=1

[Models]
2D=
3D=

[2D]
Description=2D
width=1219.2
height=1219.2
contZAxis=2D
Length Units=mm
Time Units=sec
Speed Units=mm\sec
UseDefaults=1
HasArc=TRUE
HasDrill=FALSE
ArcTolerance=0.01
ArcMinAngle=0.1
ArcMaxAngle=359
MaxTurnAngle=2
MachineXResolution=0.03937
MachineYResolution=0.03937
MachineZResolution=0.03937

[2D_Comm]
Line=COM2
Baud=9600
Data_Bits=8
Parity=None
Stop_Bits=1
DTR=TRUE
RTS=TRUE
CtsRts=FALSE
XonXoff=FALSE
```

Locate the line that reads “SpoolFile=*.NCD” and change it to “SpoolFile=*.NC” as shown. This changes the extension of the output file.

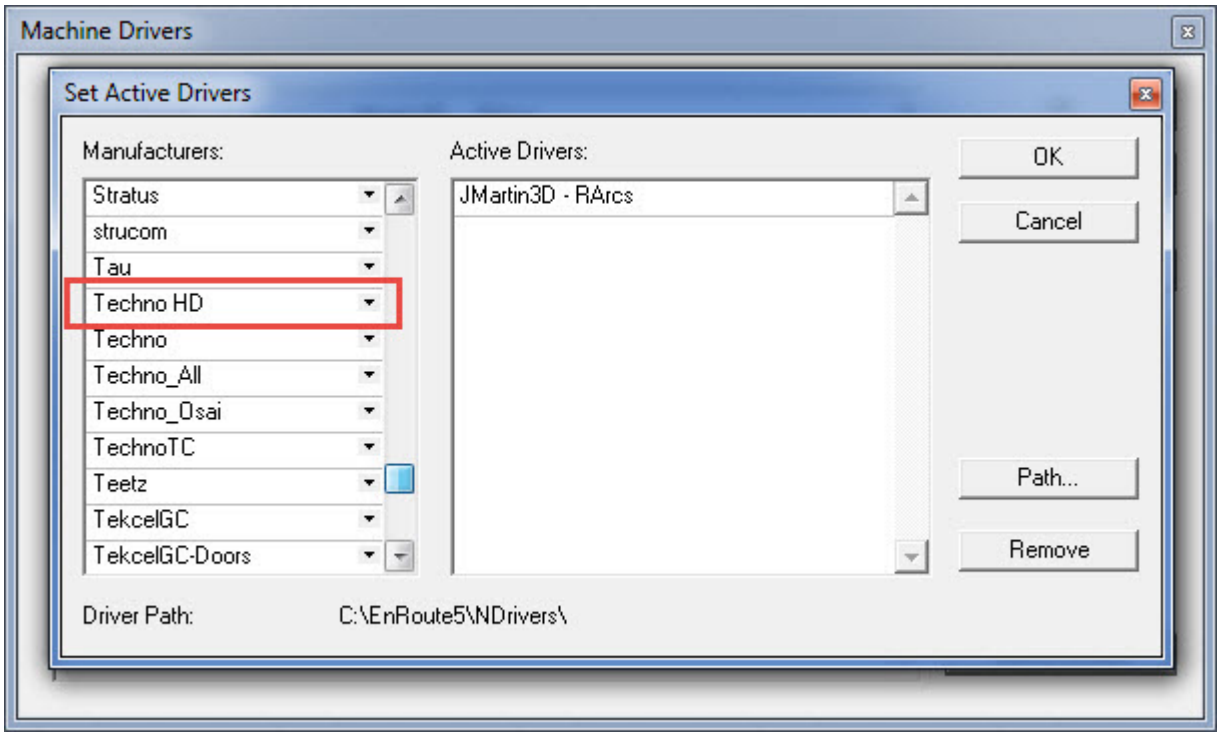
Note: This step should not be necessary if the driver was sent from tech support.



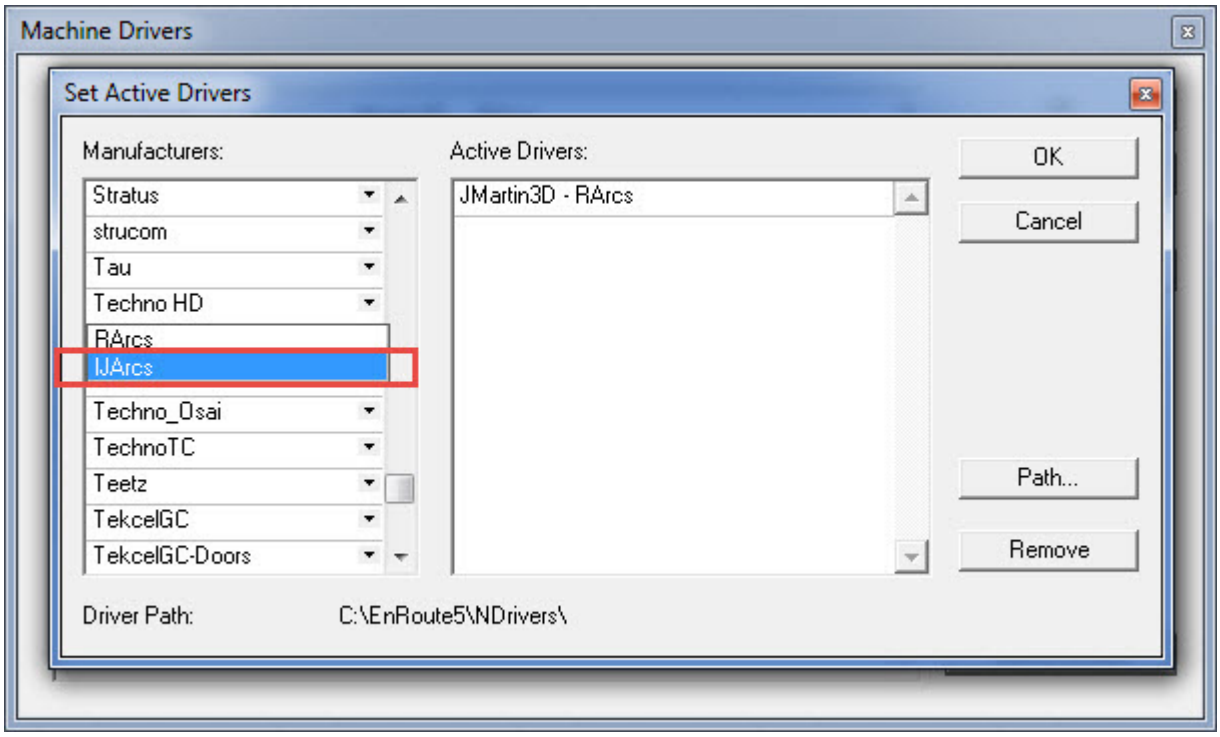
Now we have to make the new driver active. Shut and re-start EnRoute. Then open a new file and go to “Setup > Machine Setup”



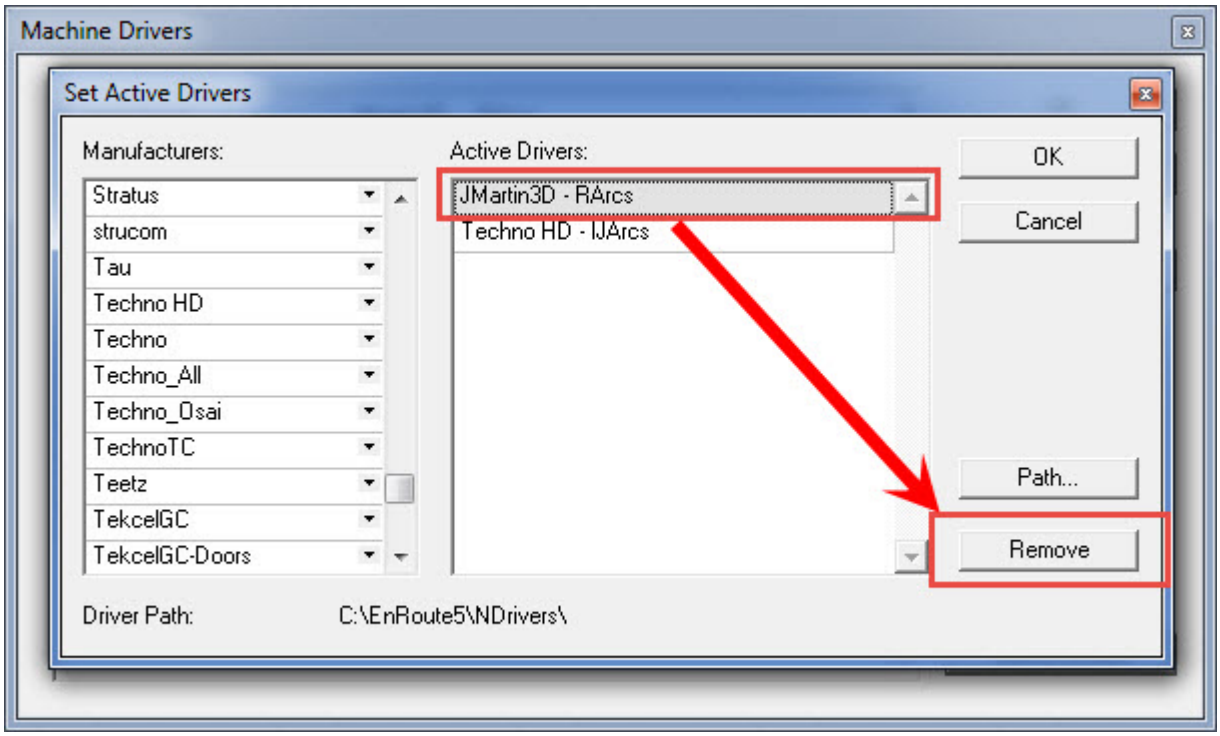
The “Machine Drivers” window will come up. Notice that the current driver is not correct. Click on “Active Drivers”.



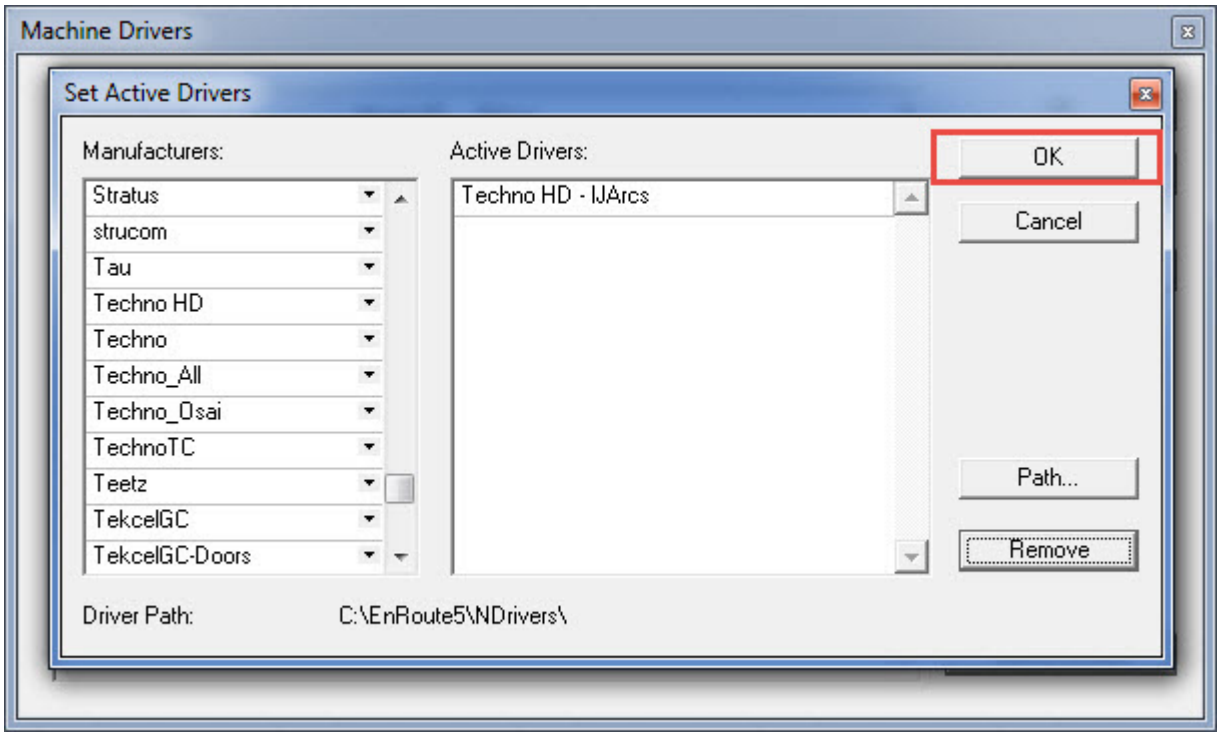
Locate “Techno HD” from the list on the left.



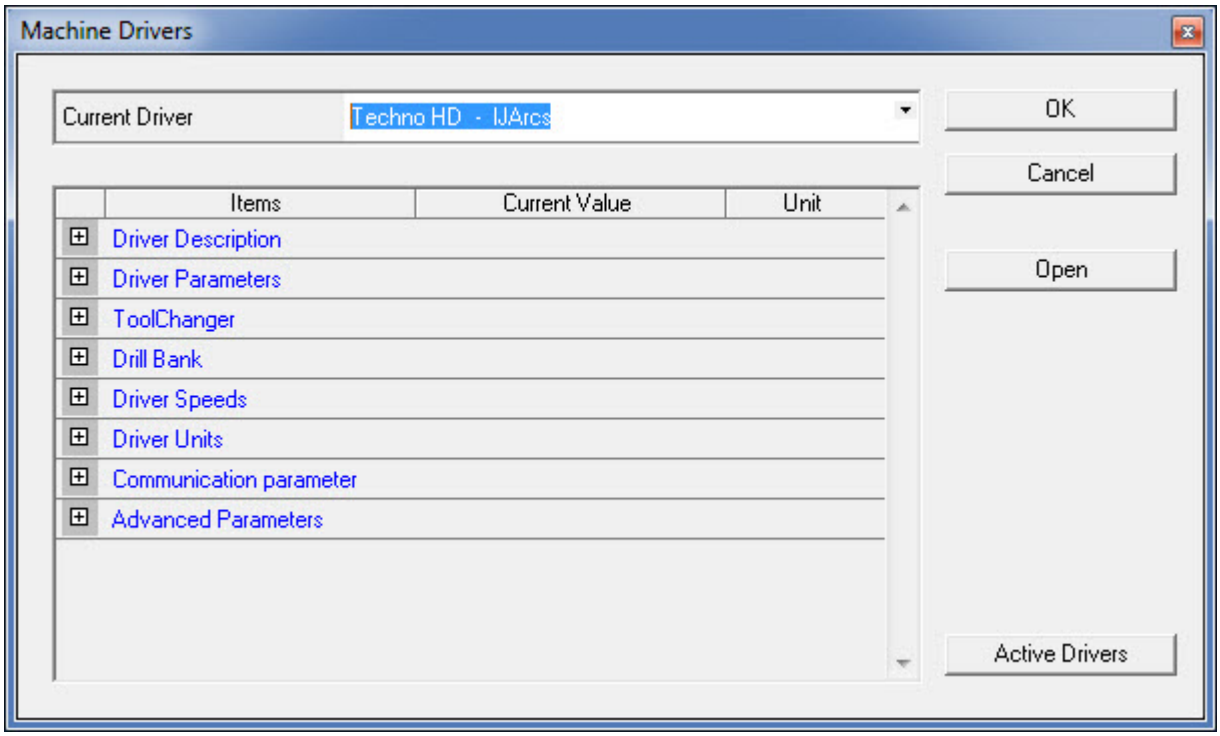
Click on the arrow and select “IJ Arcs”. The correct post will appear in the “Active Drivers” list.



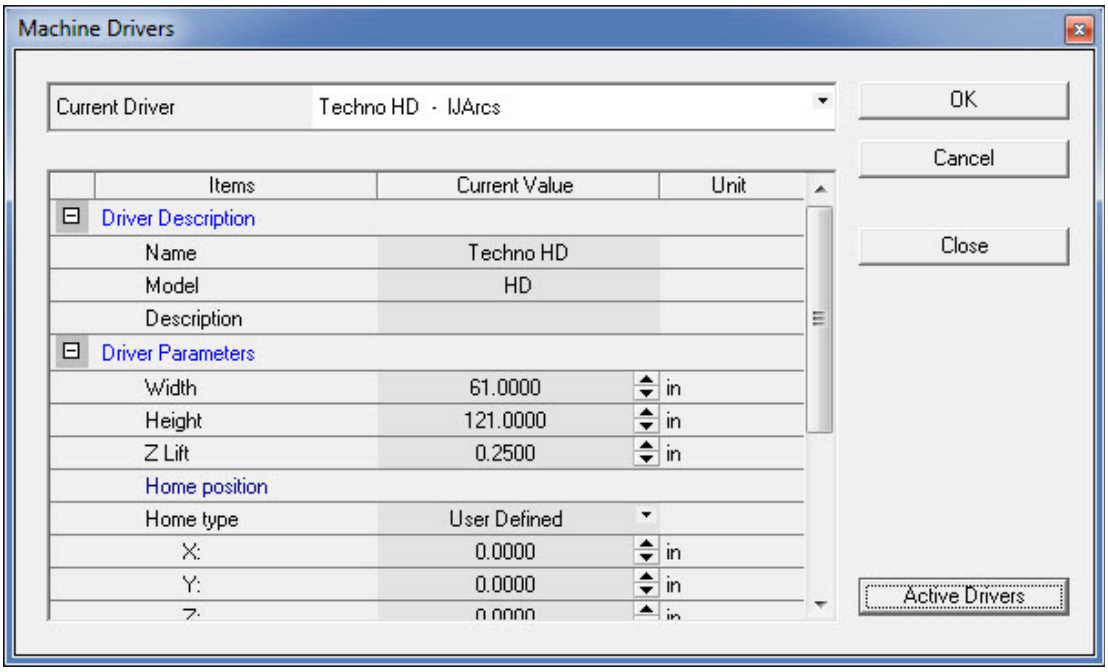
Now, remove the default JMartin3d post. Highlight it and click “Remove”.



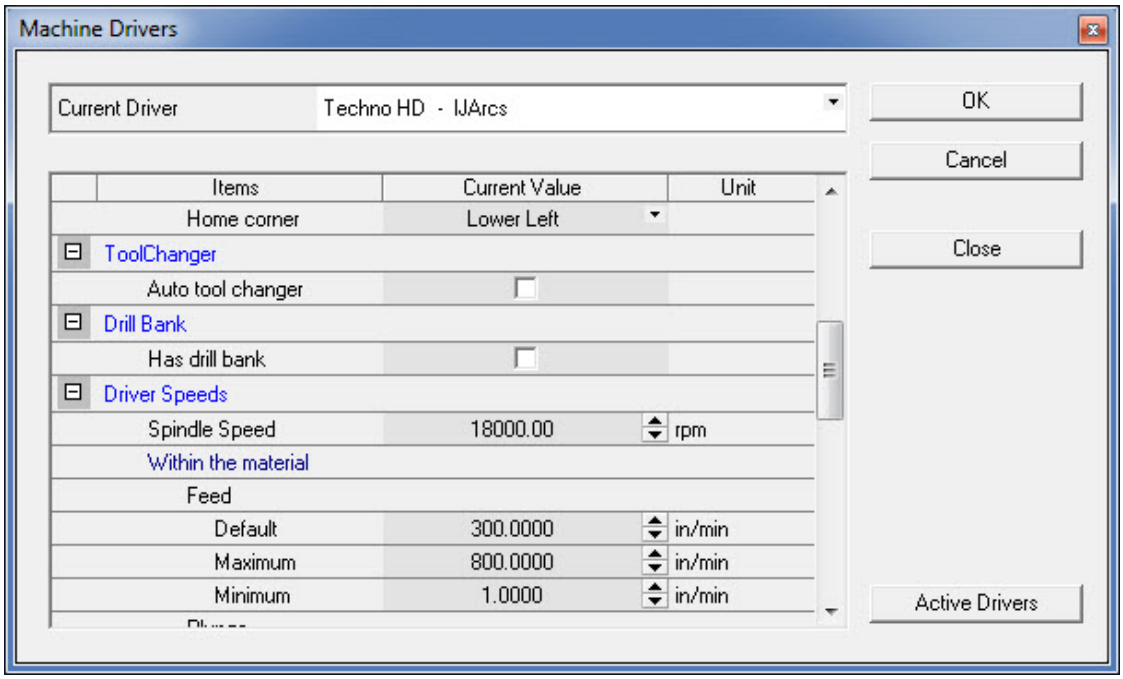
Click “OK”.



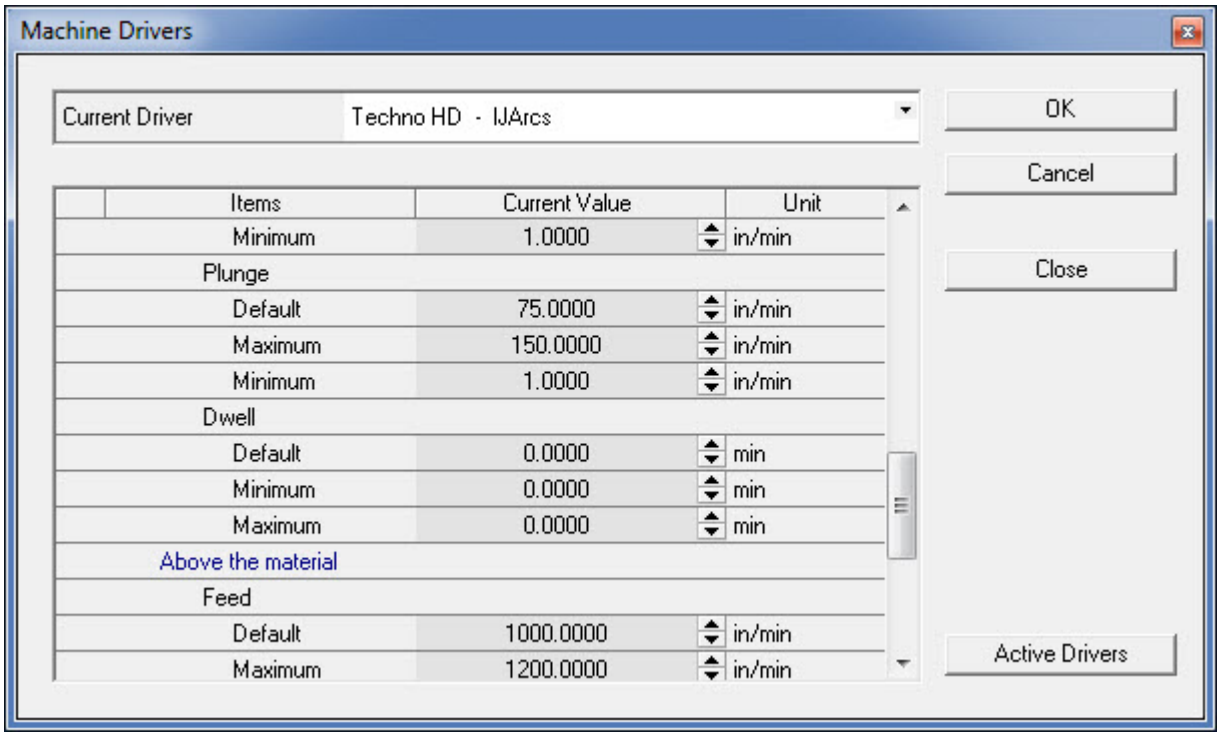
The correct driver is now active and we can check the parameters.



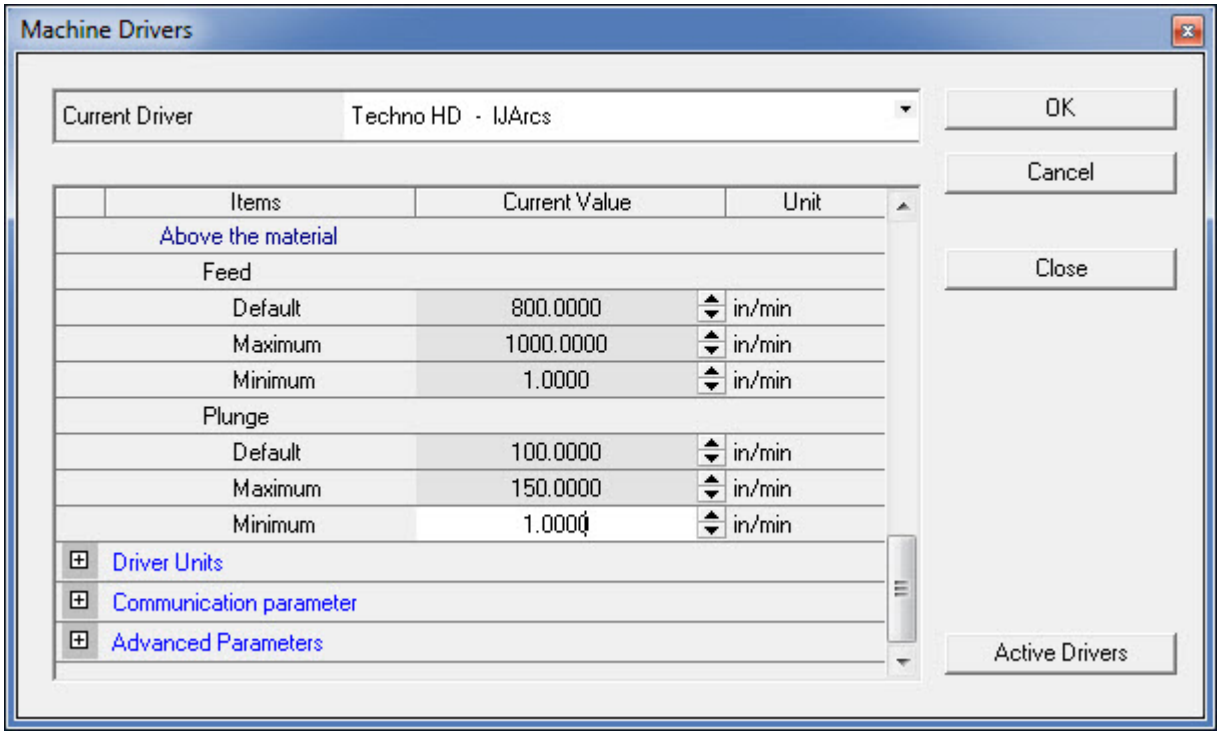
These next few screens show what the driver settings should be. The “Z Lift” parameter is variable. It defines the clearance height of the tool. This is how far above the material the tool will raise before traverse movements.

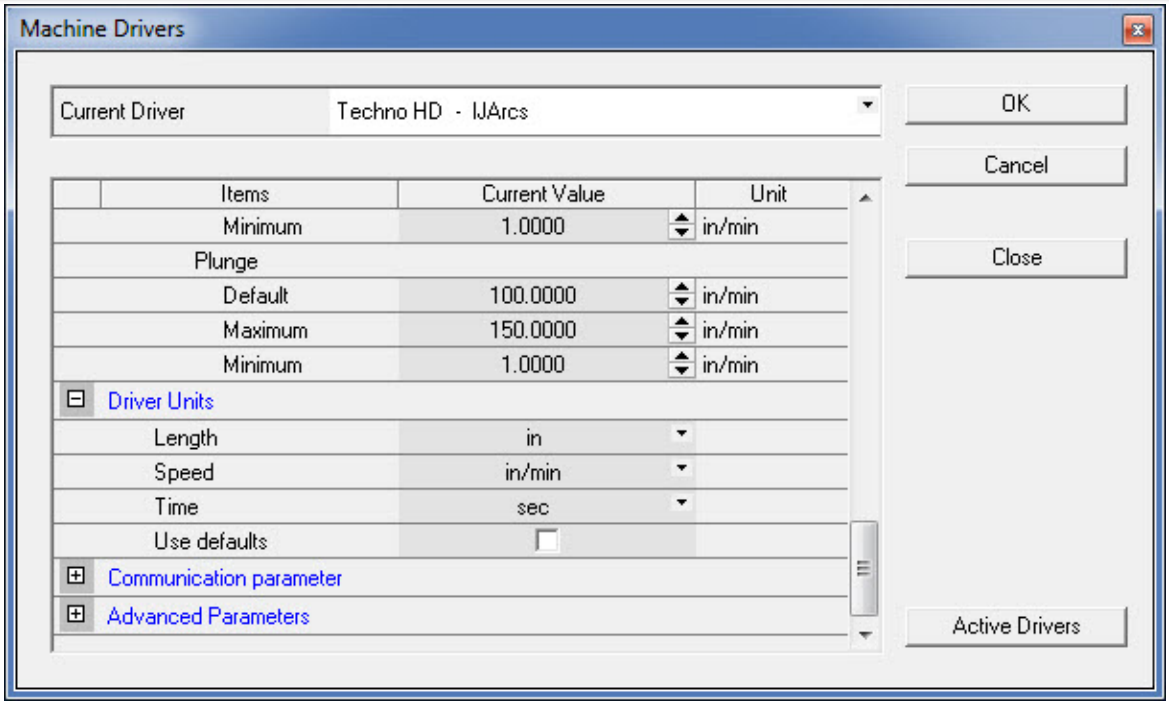


“Auto tool changer” and “Drill bank” should be unchecked as they do not apply to the HD machines.

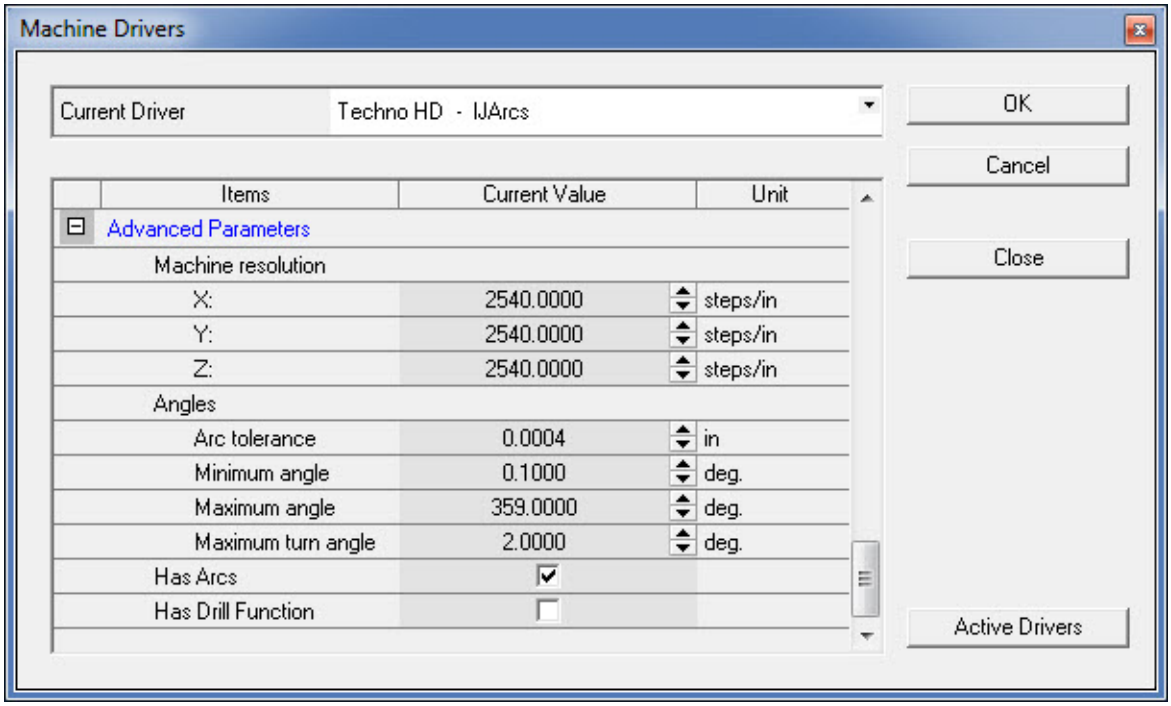


These default speed values will fill in the g code if no speeds are defined in the tool path.





Units should be set to: inches, inches per minute, and seconds.

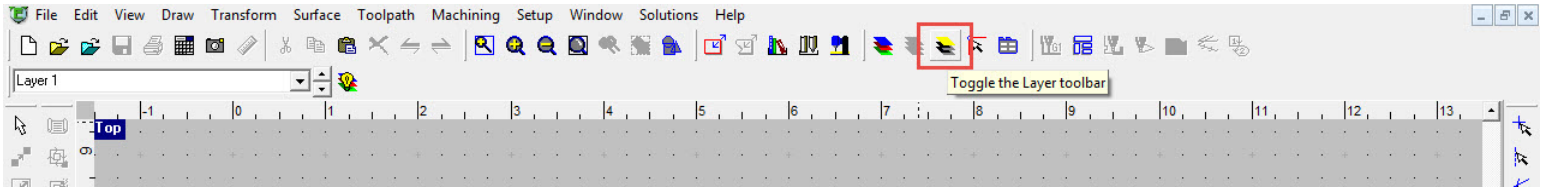


“Has arcs” should be checked and “has drill function” should not be checked.

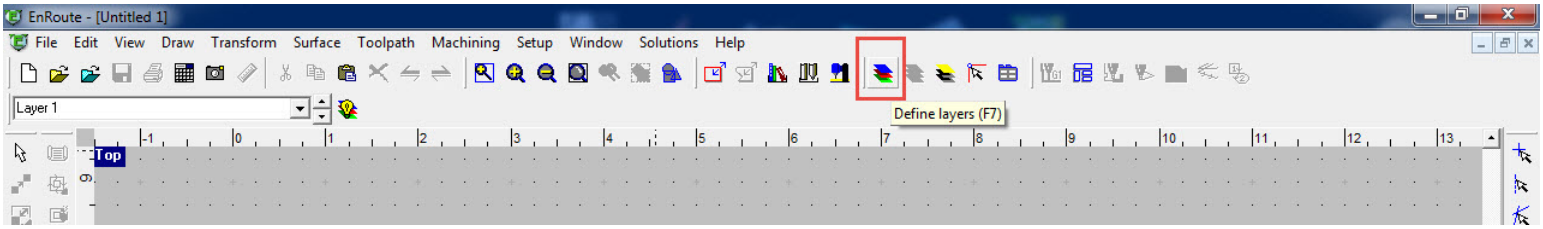
Once all of the settings are confirmed, click “OK”.
EnRoute is now ready to output files to run your HD machine!

EnRoute: 2.1 – Layers

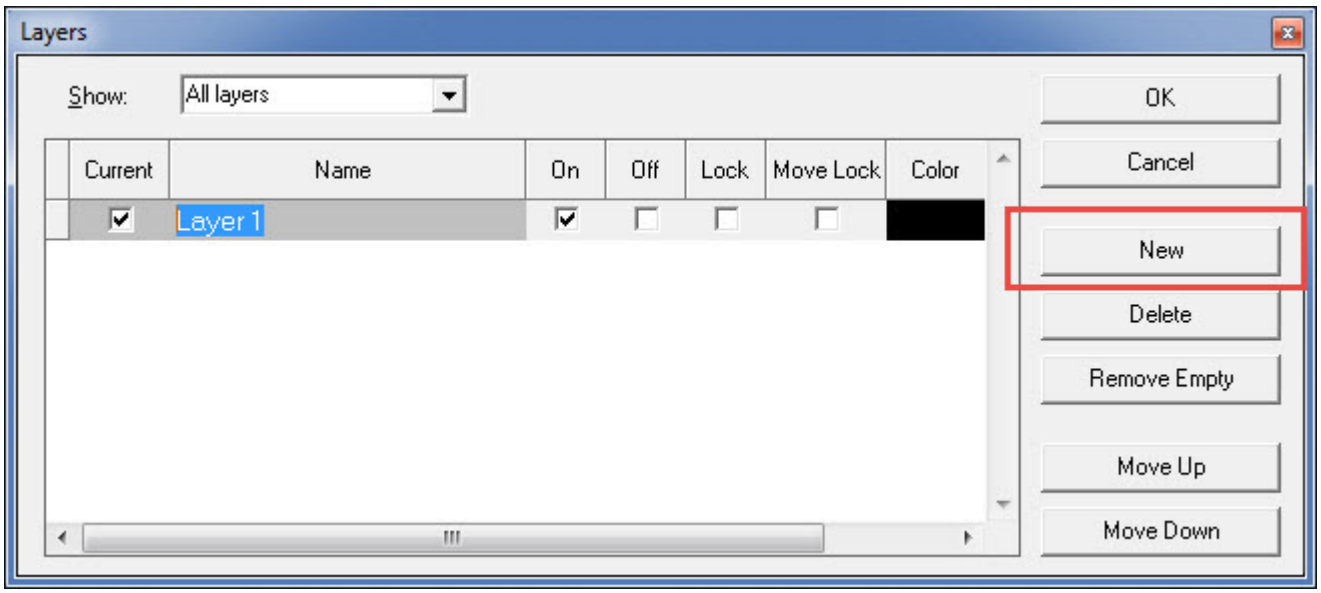
This section shows how to use layers in EnRoute.



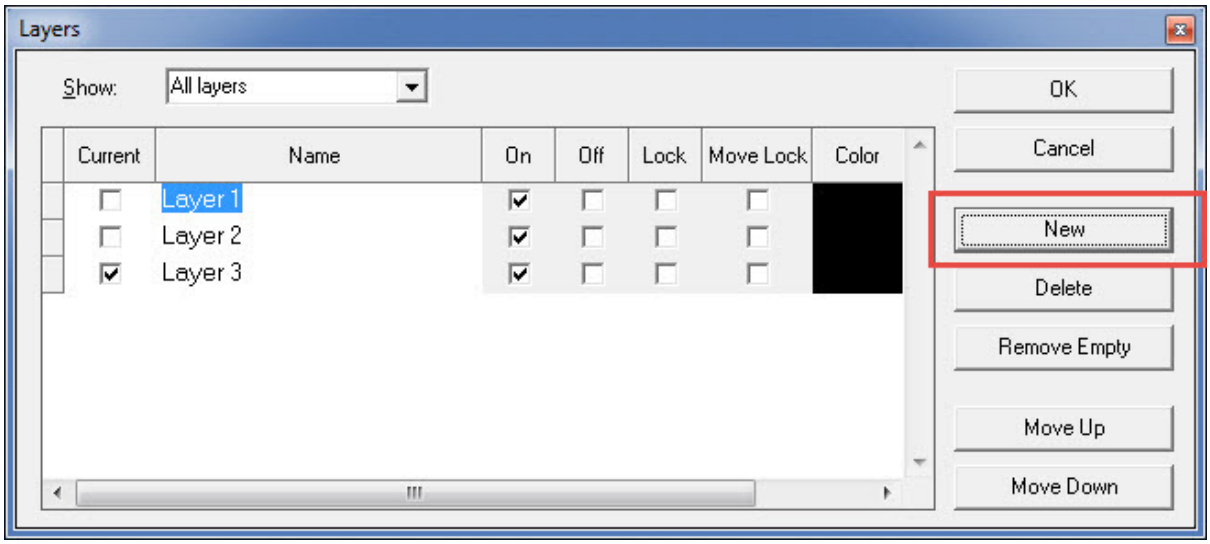
Locate and click on the icon to show the layer toolbar.



Click on the “Define Layers” button or hit “F7” to add and manage layers.



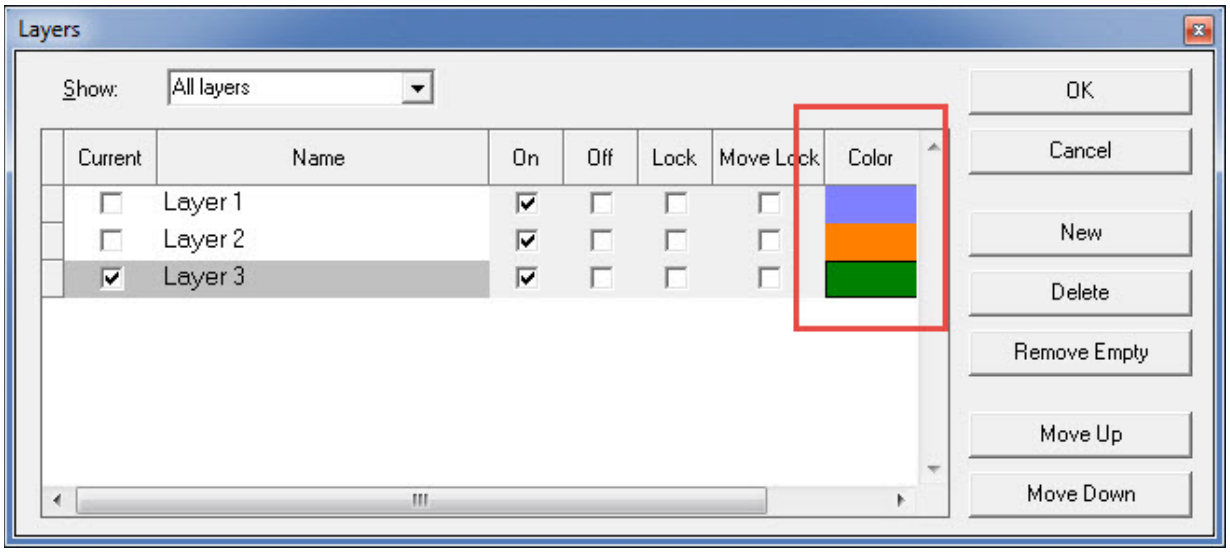
Click on the “New” button to create a new layer.



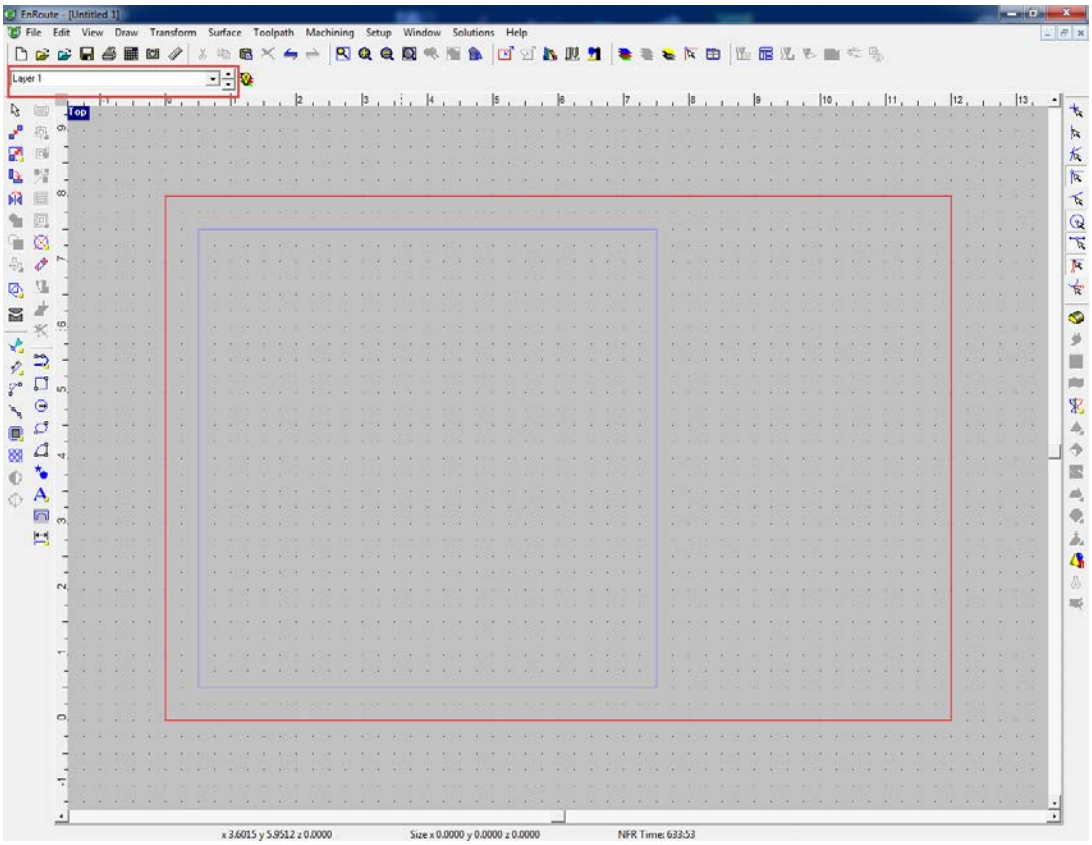
Repeat for as many layers as you need. Just click on the layer name and type a new name to keep the layers better organized.

They can be re-ordered using the “Move Up” and “Move Down” buttons.

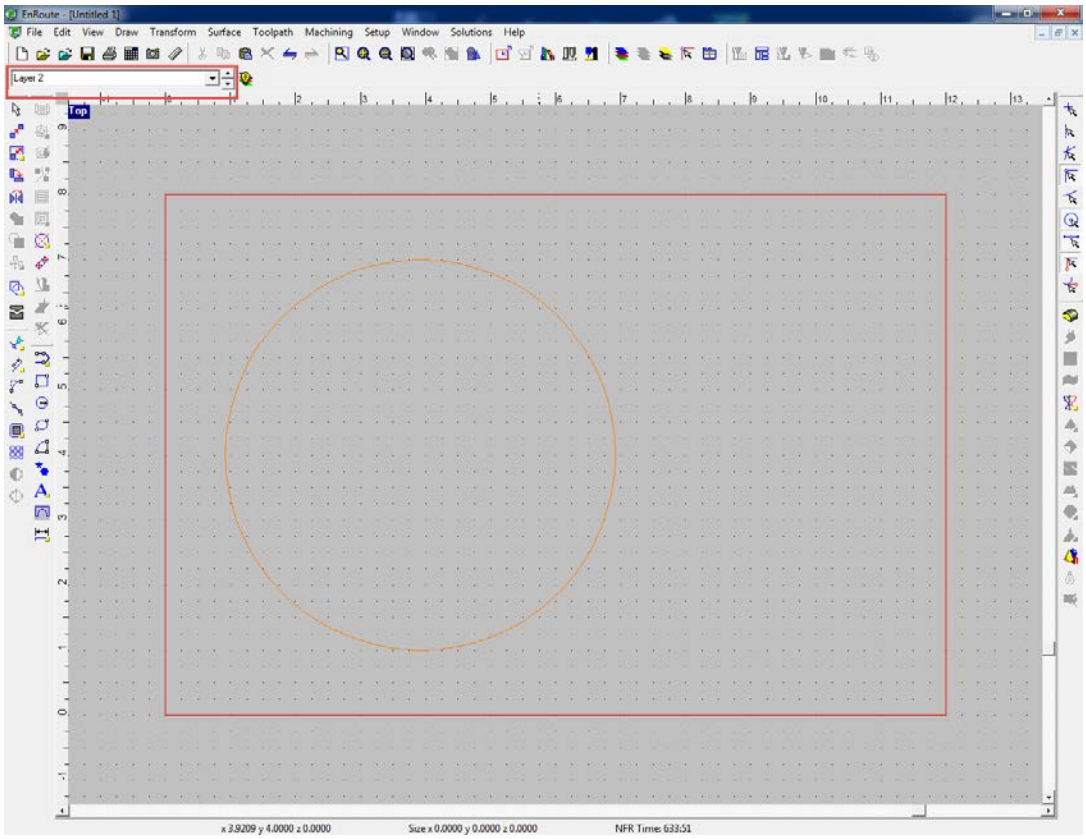
The order the layers are in from top to bottom will be the order that they output in if “Layer” is selected as the output priority. For example: I would use the last layer as my cutout, so all of the other cuts would happen first.

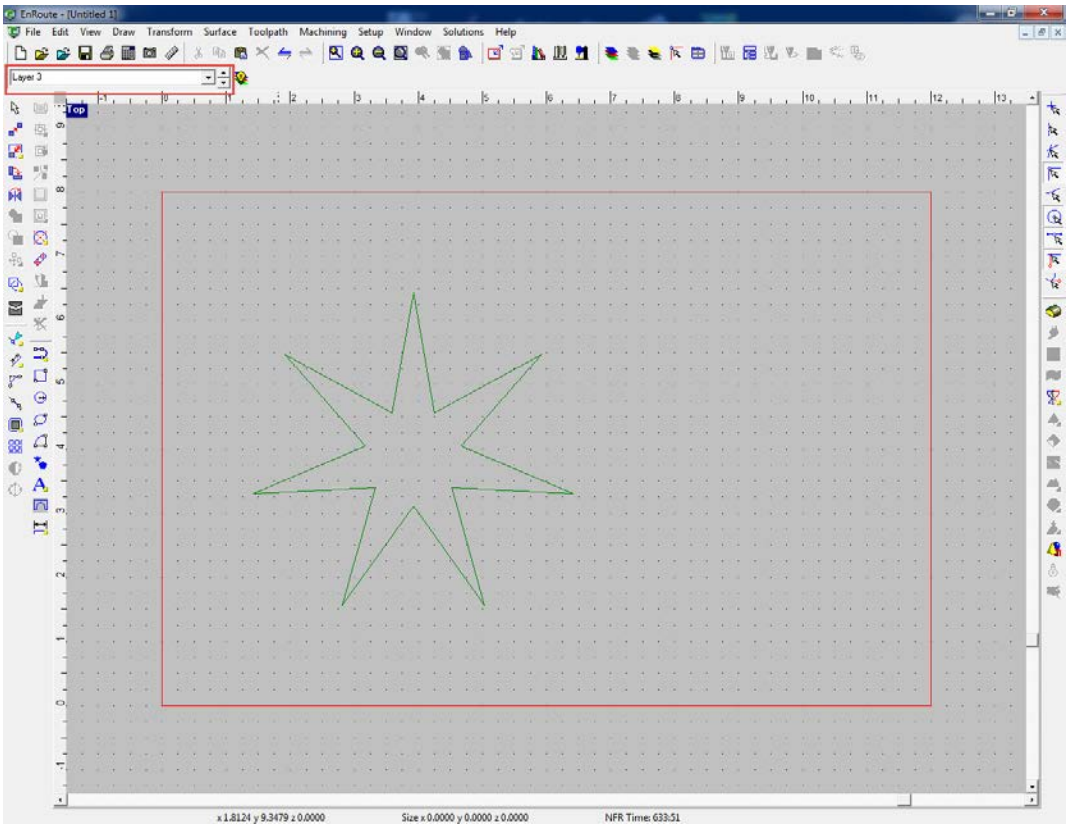


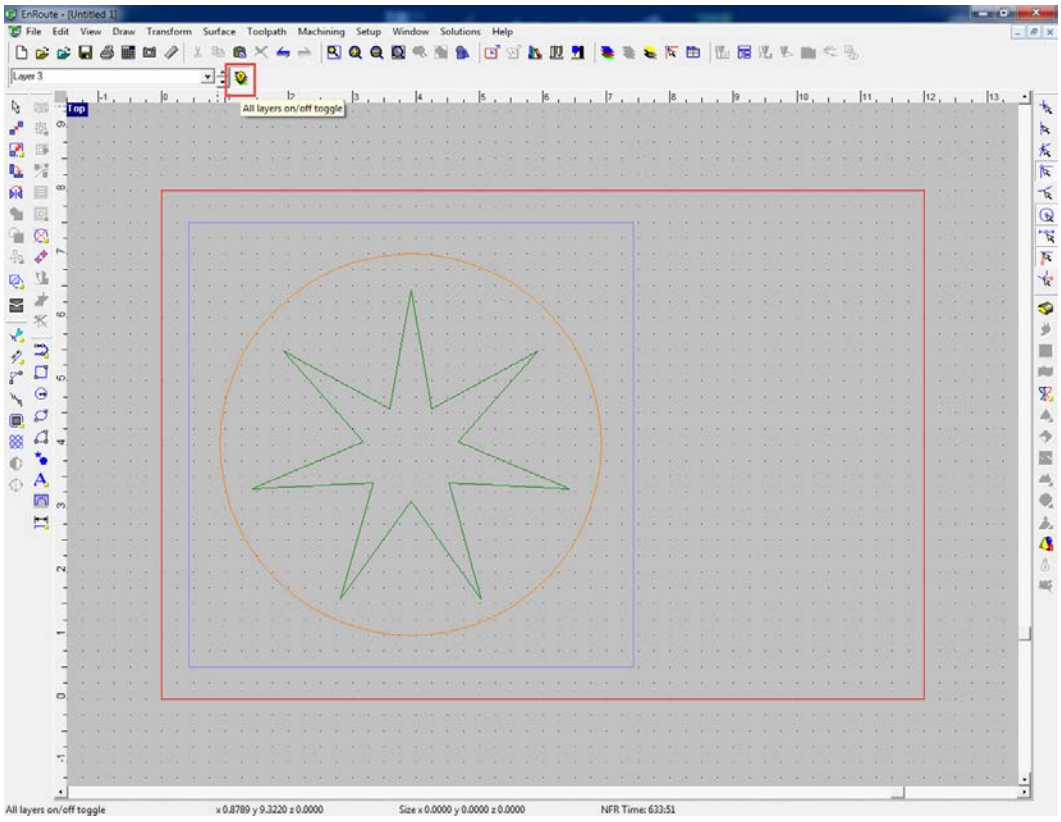
Double click on the color column to change the layer’s color. This comes in handy when there are multiple active layers.



Use the layer toolbar to choose the active layer.
All vectors will be the color of the layer that they are on.



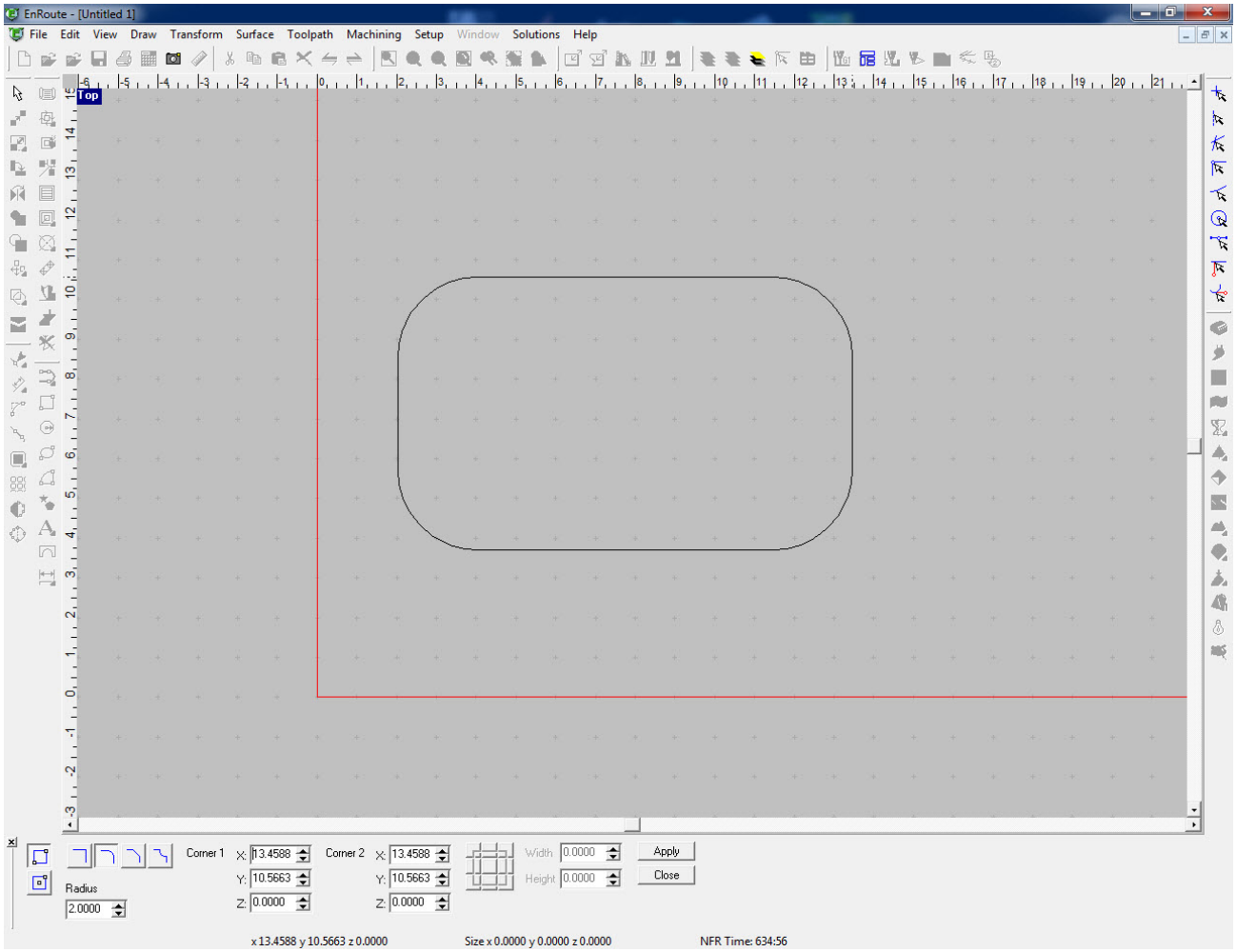




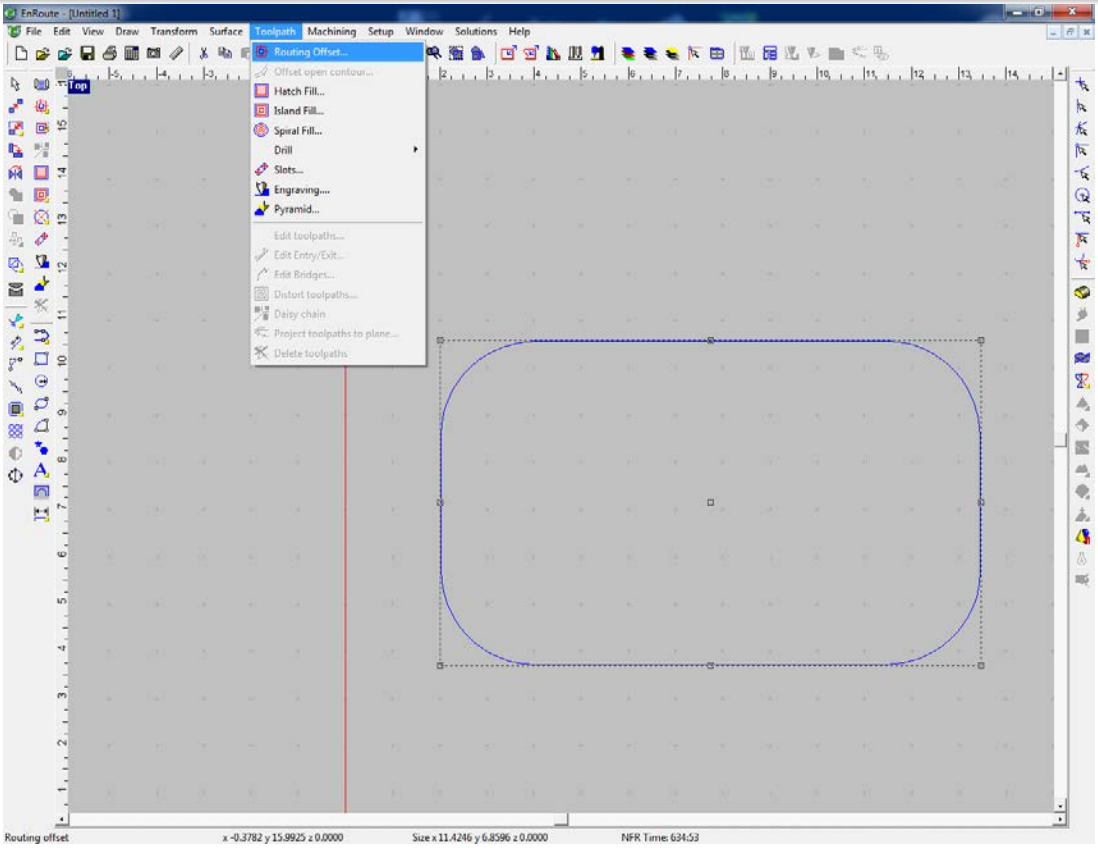
The little light bulb on the layer toolbar makes all layers visible

EnRoute: 2.2 – Tool Paths

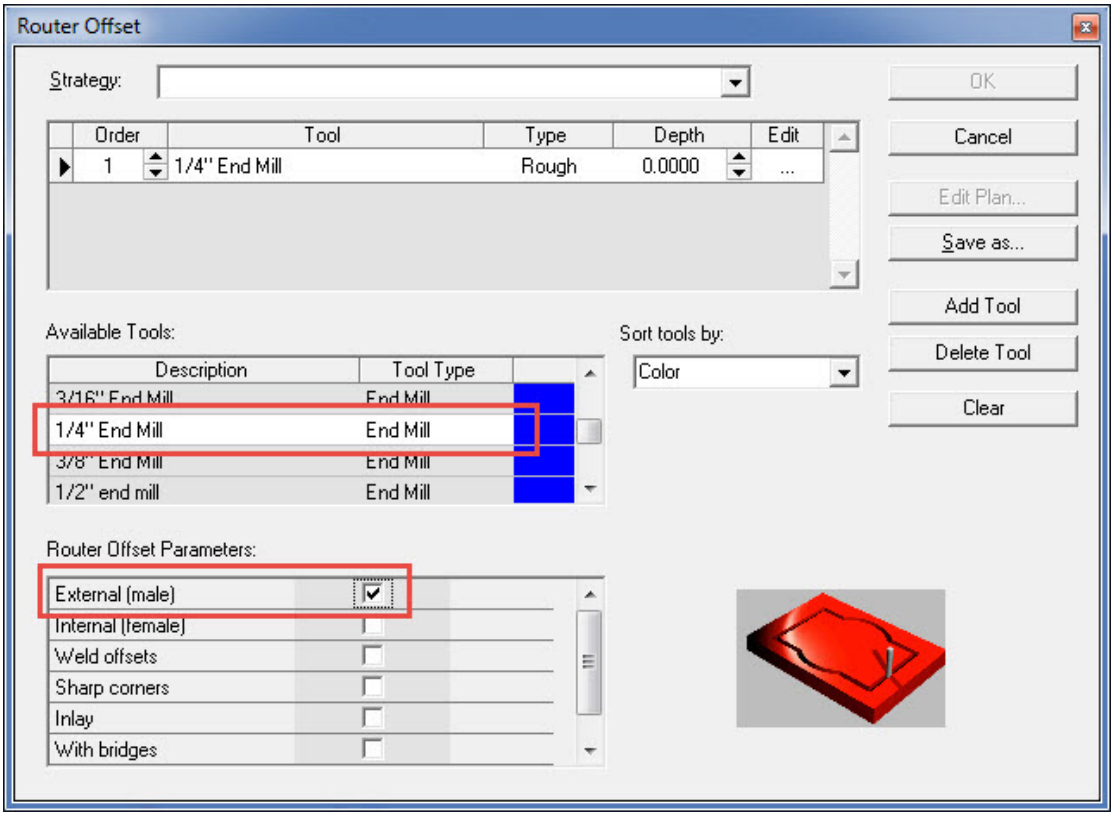
This section covers the basics on common types of tool paths.



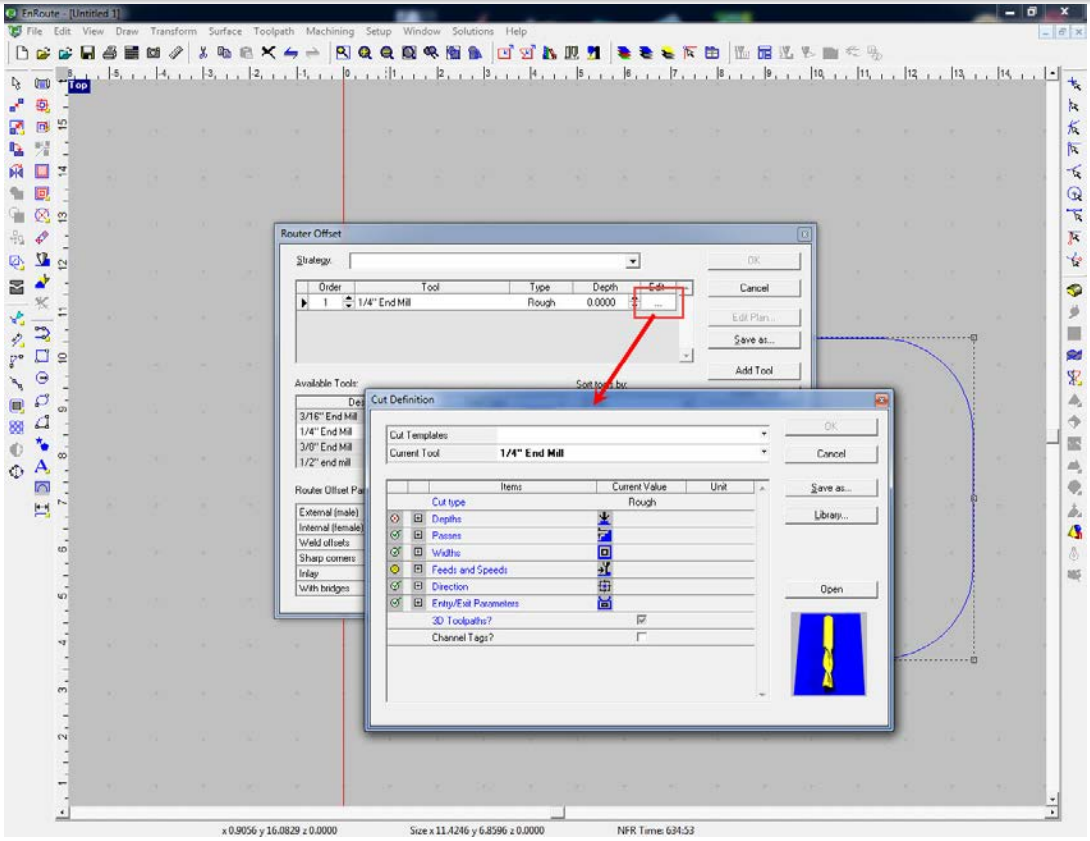
First, we'll start with a basic closed contour.



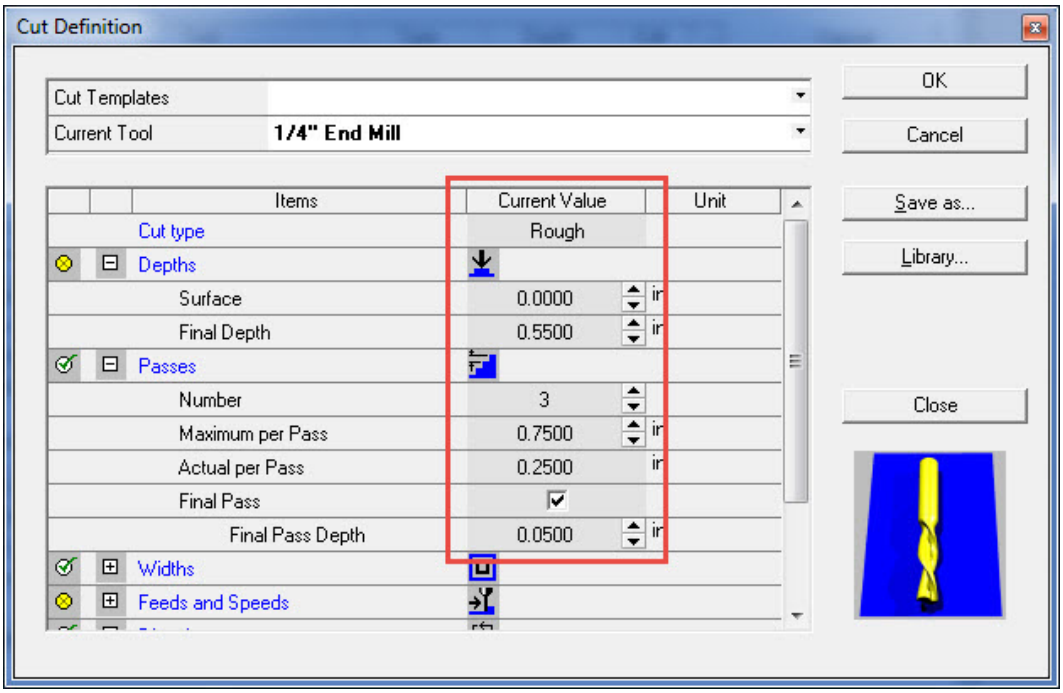
The most commonly used tool path is a cutout, or “Routing Offset”.
Note: Geometry must be selected before tool path is available.



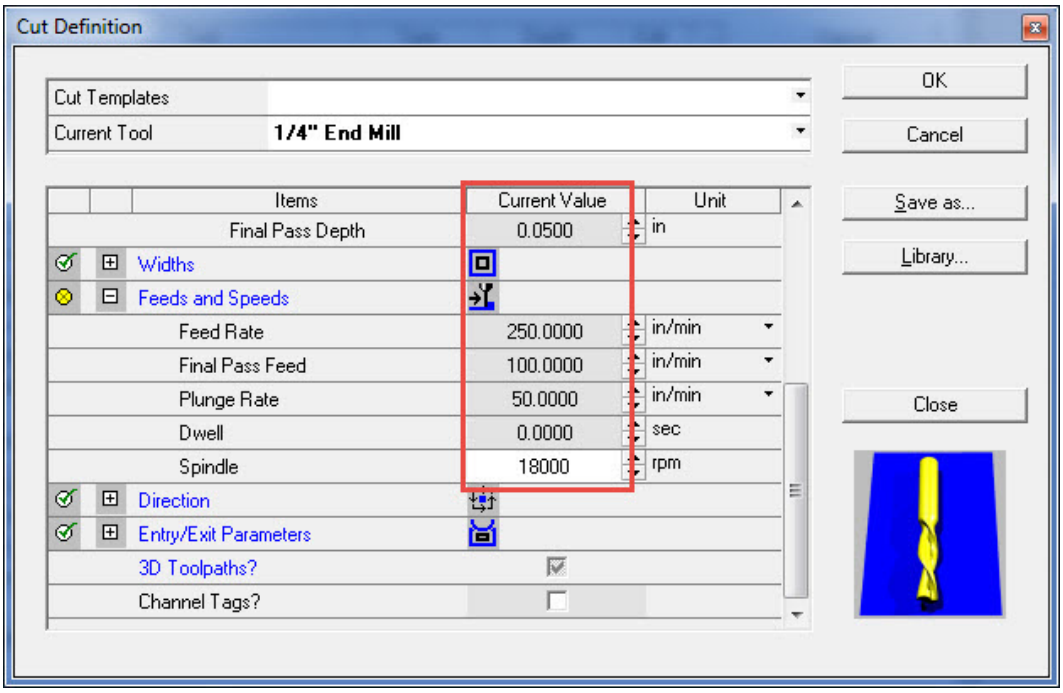
Select which cutter you want to use and whether it is an internal or external cut.



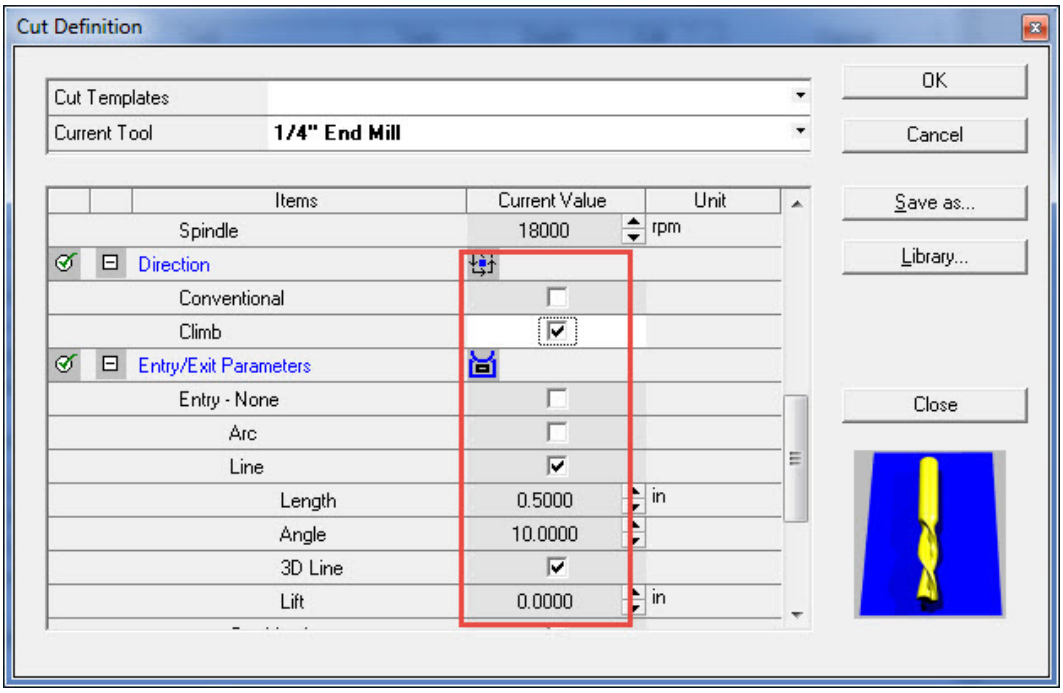
Click on the "…" under the edit column. This will bring up the "Cut Definition" window. This is where all of the tool path parameters are defined, such as the cut depth, number of passes, and speeds.



Just go from top to bottom, filling in the desired parameters. In this view we see the total depth of cut and the number of passes to get there. The "Final Pass" is independently definable with it's own depth and speed.



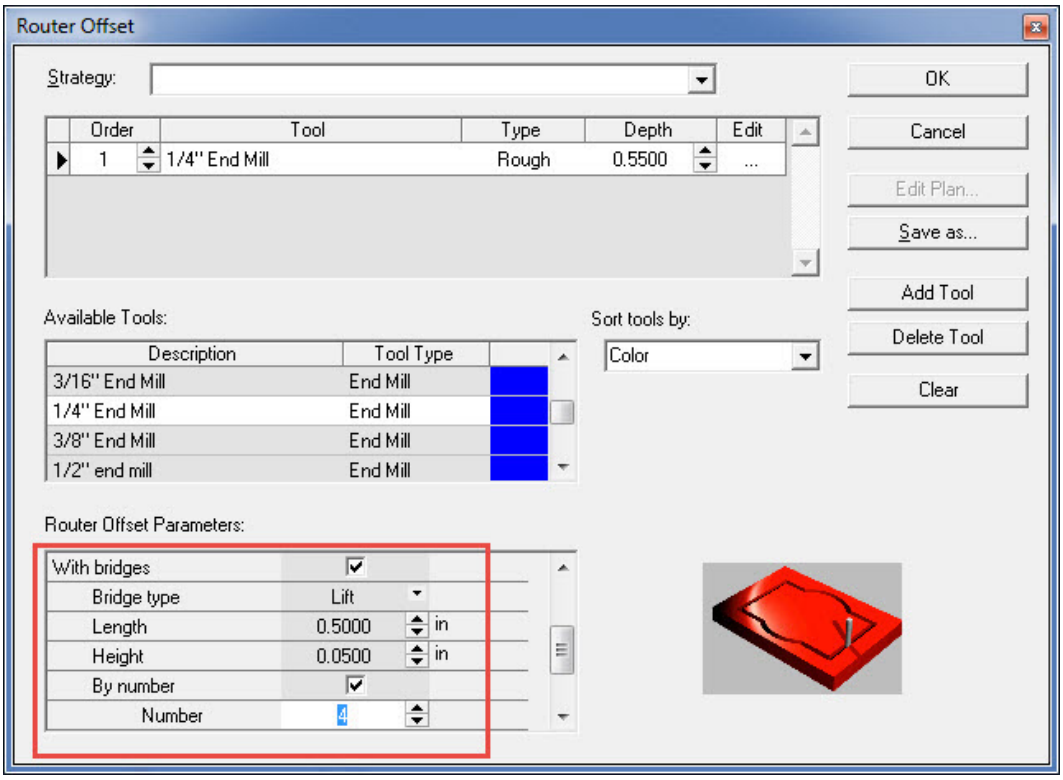
Fill in the feed rates and spindle speed.



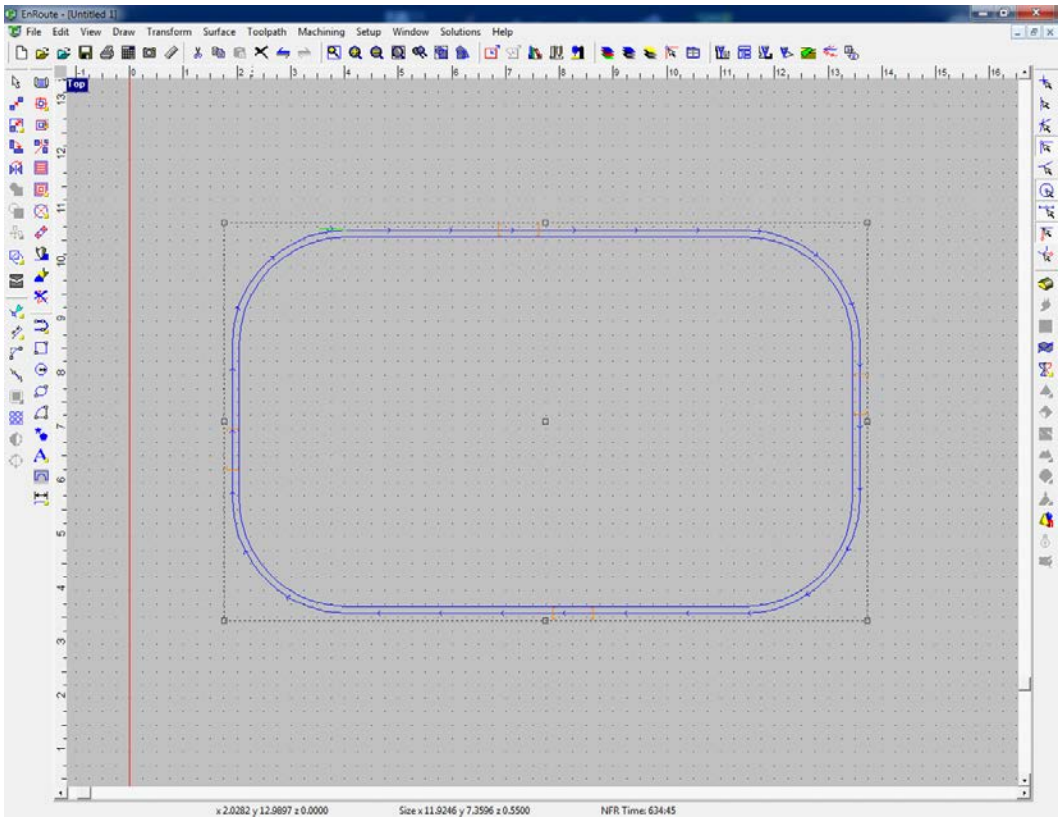
Choose a cut direction and an entry/exit If you are using a cutter with a flat bottom, it is recommended that you add a ramp-in move. “3D Line” will allow the entry move to ramp as it plunges down.

Click “OK” when all of the parameters are entered.

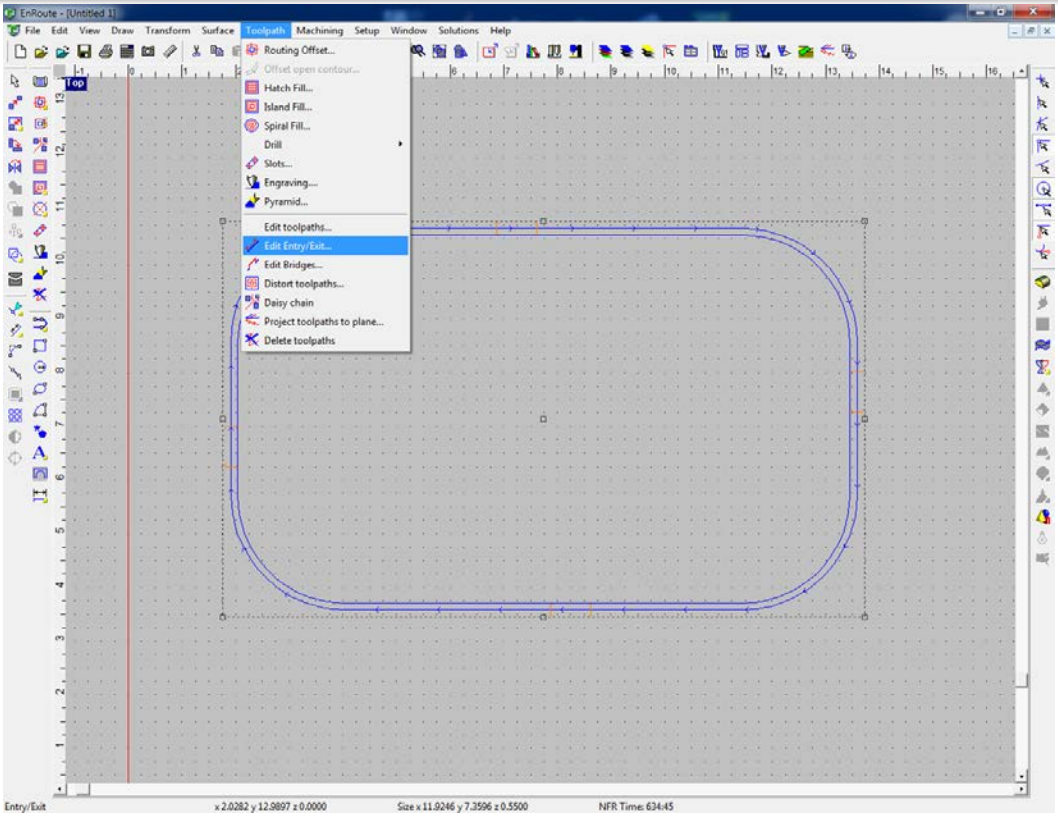
Tool paths can be saved as strategies to avoid having to re-enter the same parameters if the cuts will be similar.



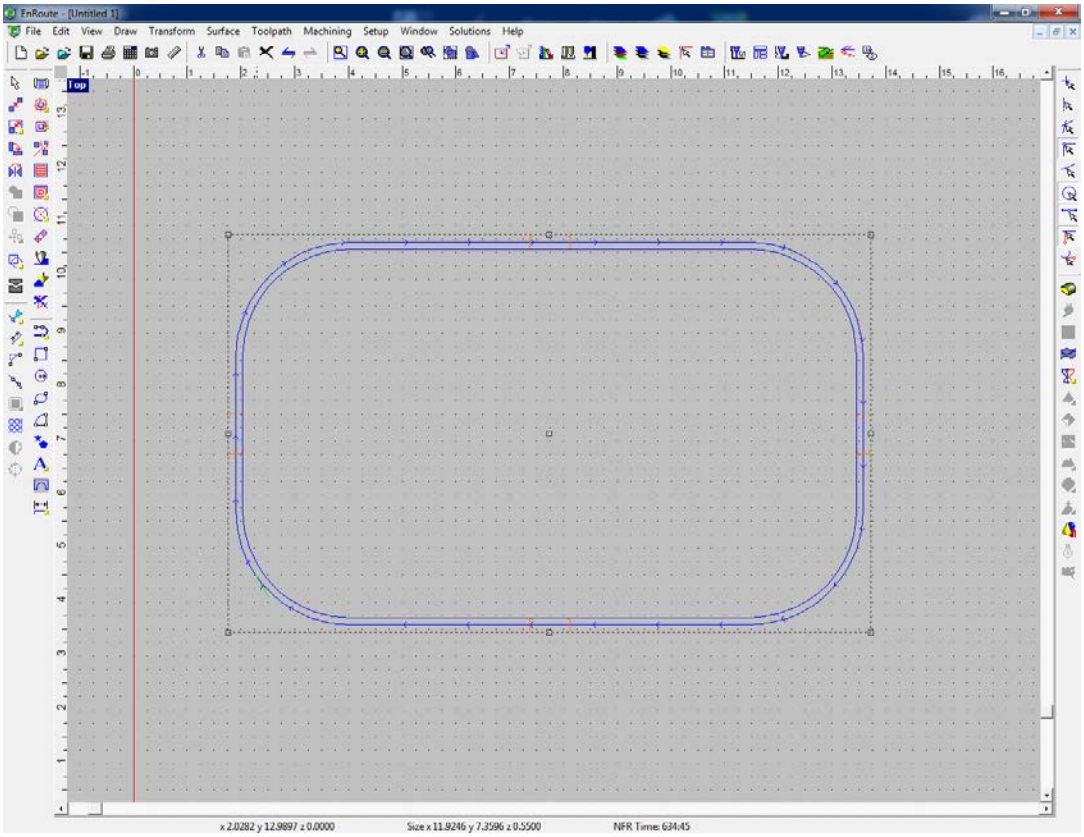
If bridges are needed to secure the cut part to the material, they can be defined here.

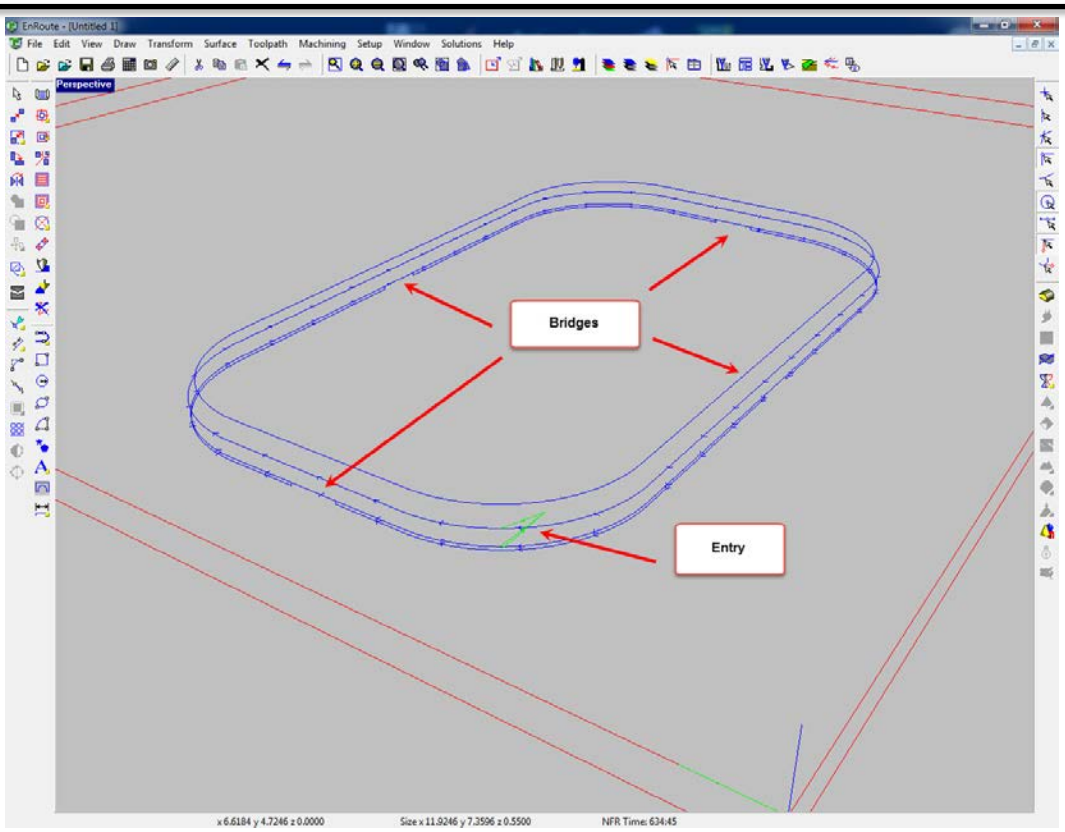


Here is a look at our tool path. The green lines are the entry and the orange brackets are the bridges.

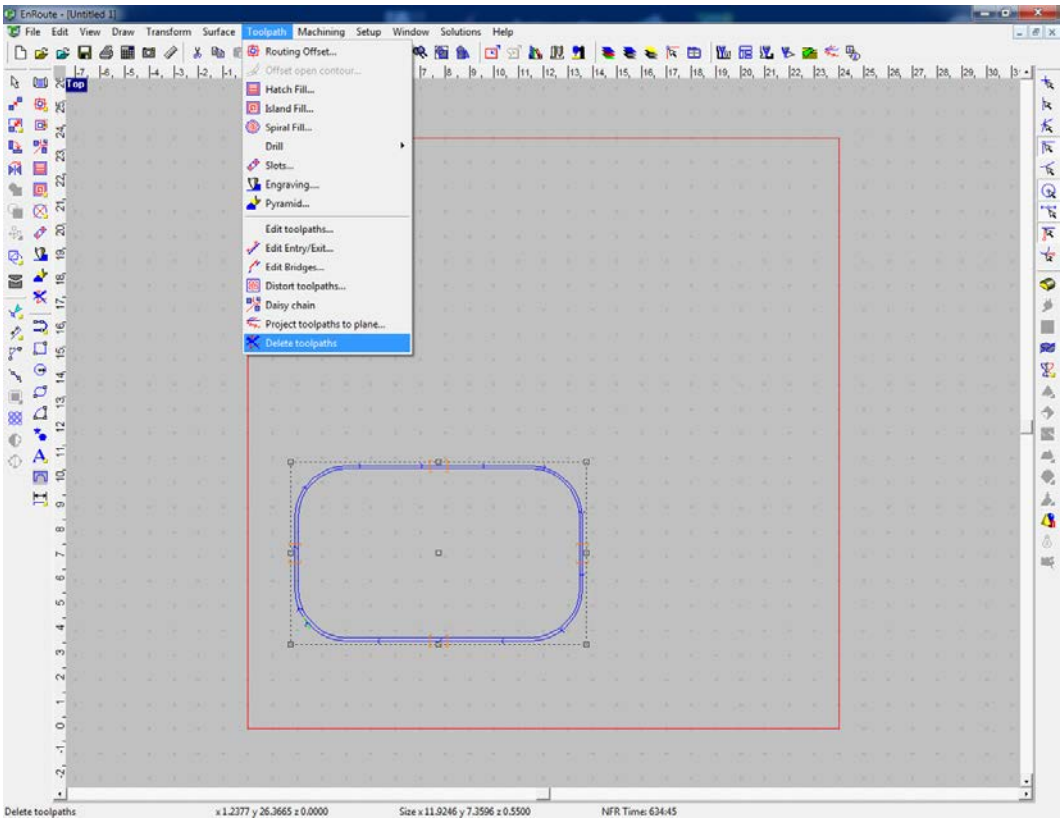


The positions of the entry and the bridges can be adjusted.
Go to “Toolpath > Edit…” and then just click and drag them to the desired position on the geometry.

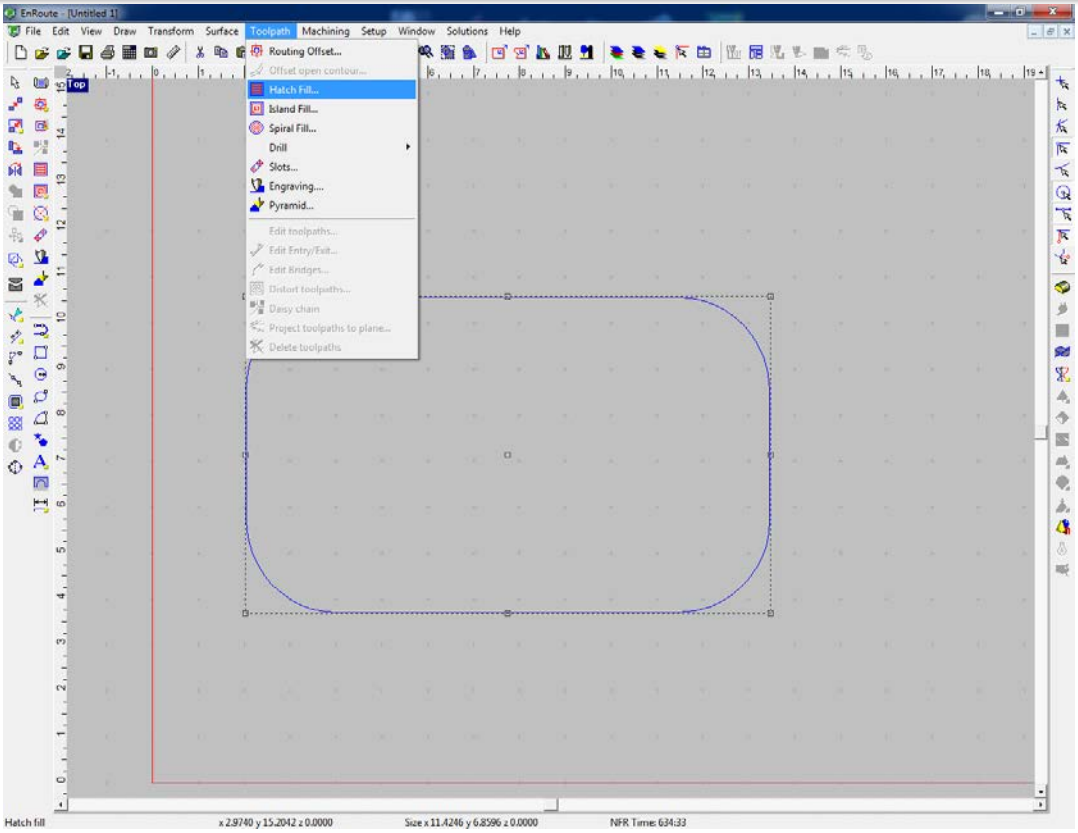




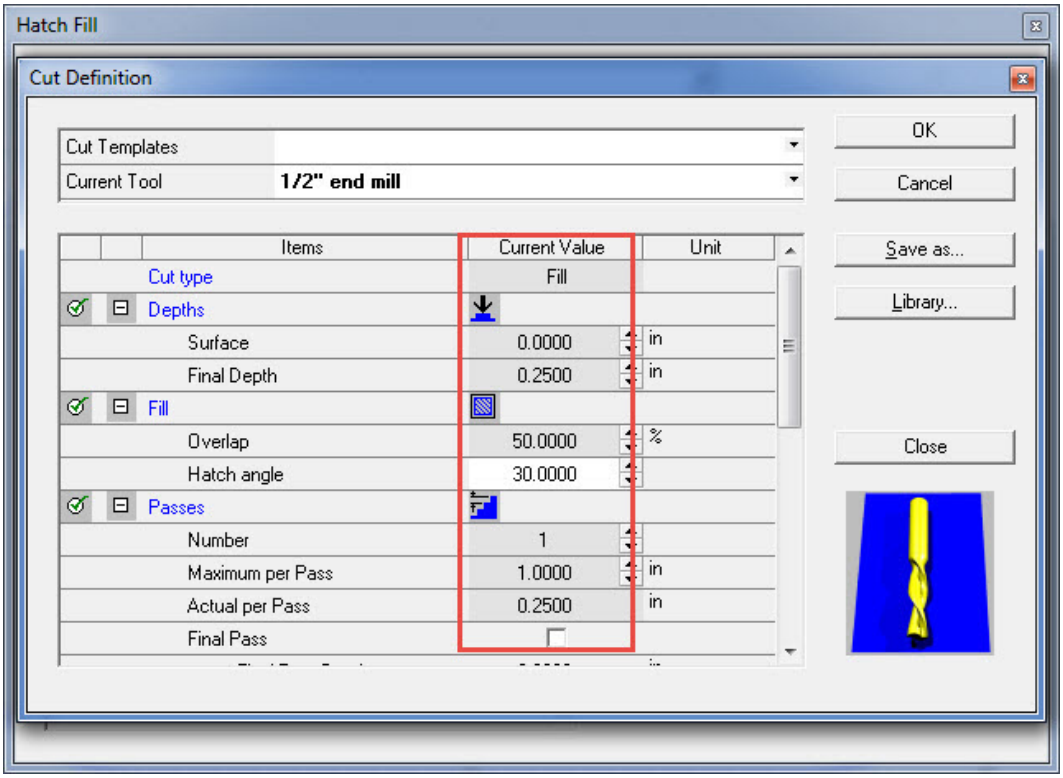
Hit F12 to toggle Perspective view.
You can see the different passes of the tool path as well as the cut direction, entry, and bridges.



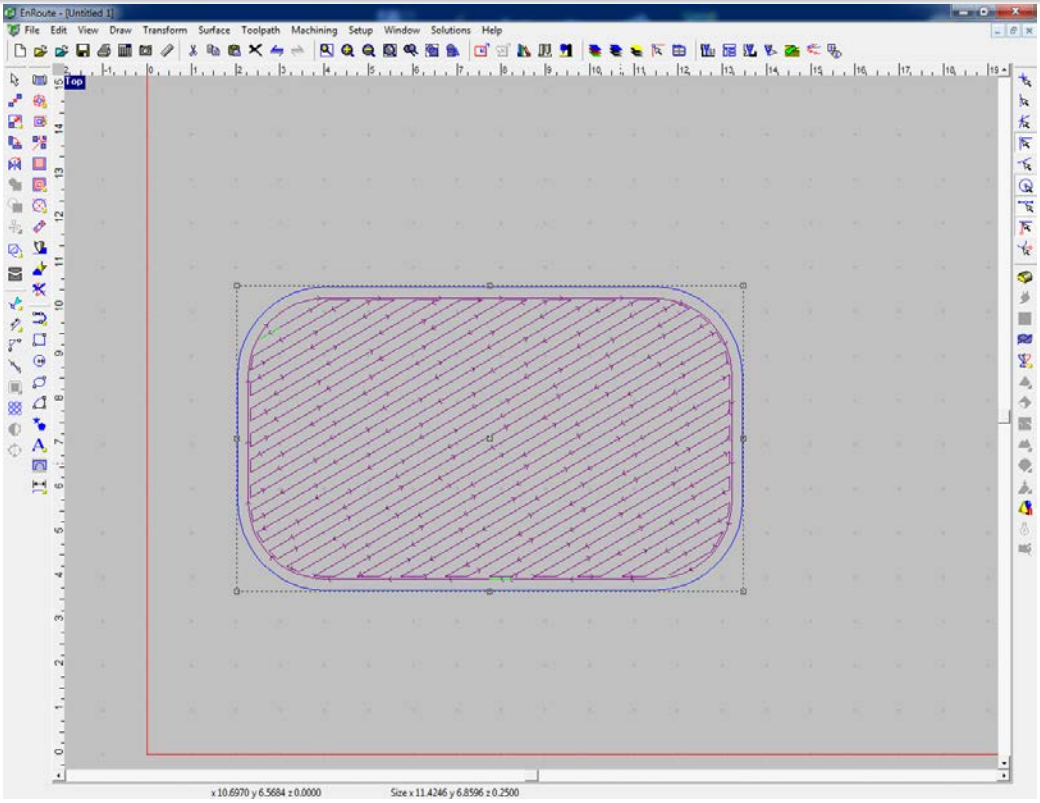
Now, lets take a look at some other tool paths. Let's delete this one.



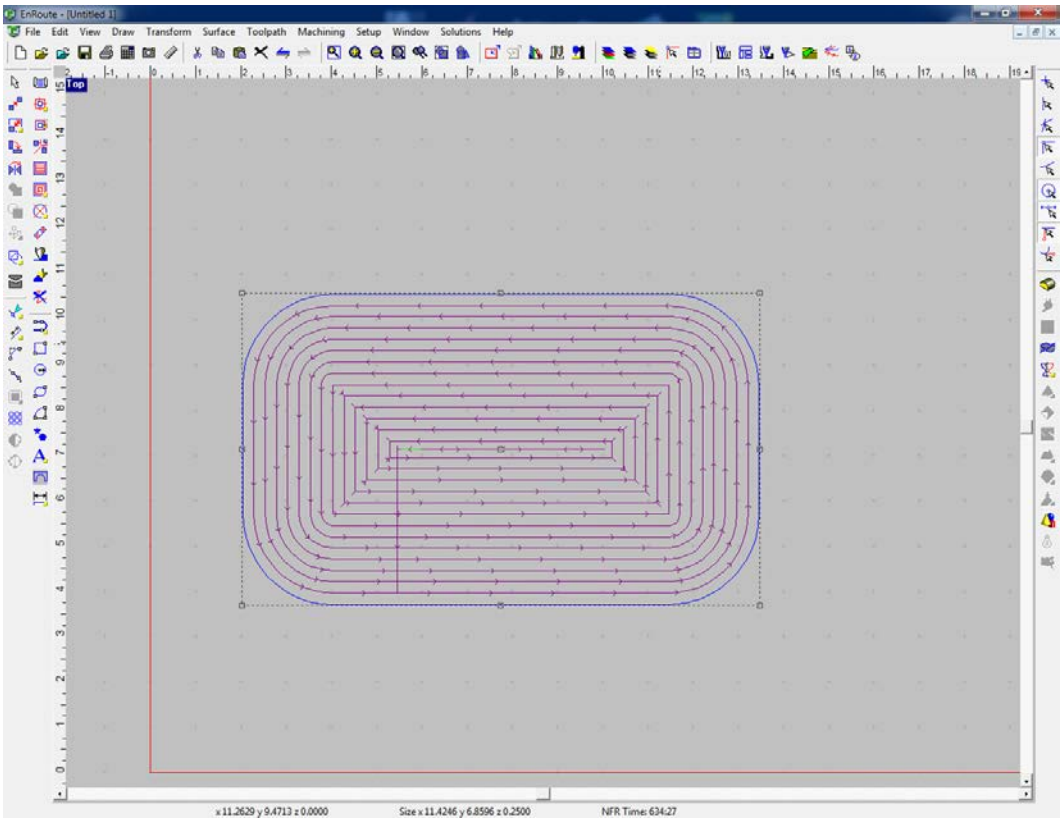
Highlight the geometry again and select “Hatch Fill” from the Toolpath menu. This is a pocketing tool path.



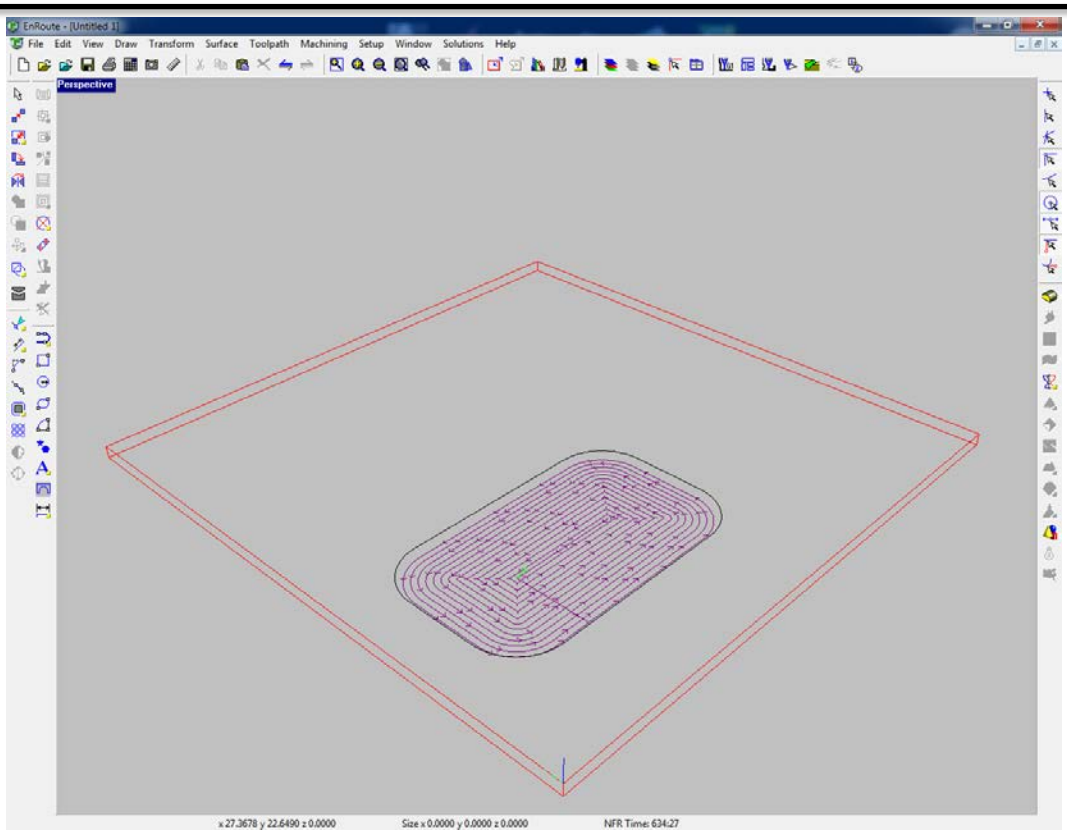
Most of the cut parameters are the same. Depth, feed rates, number of passes, etc… The hatch fill will move back and forth across the pocket. The hatch angle is defined here.



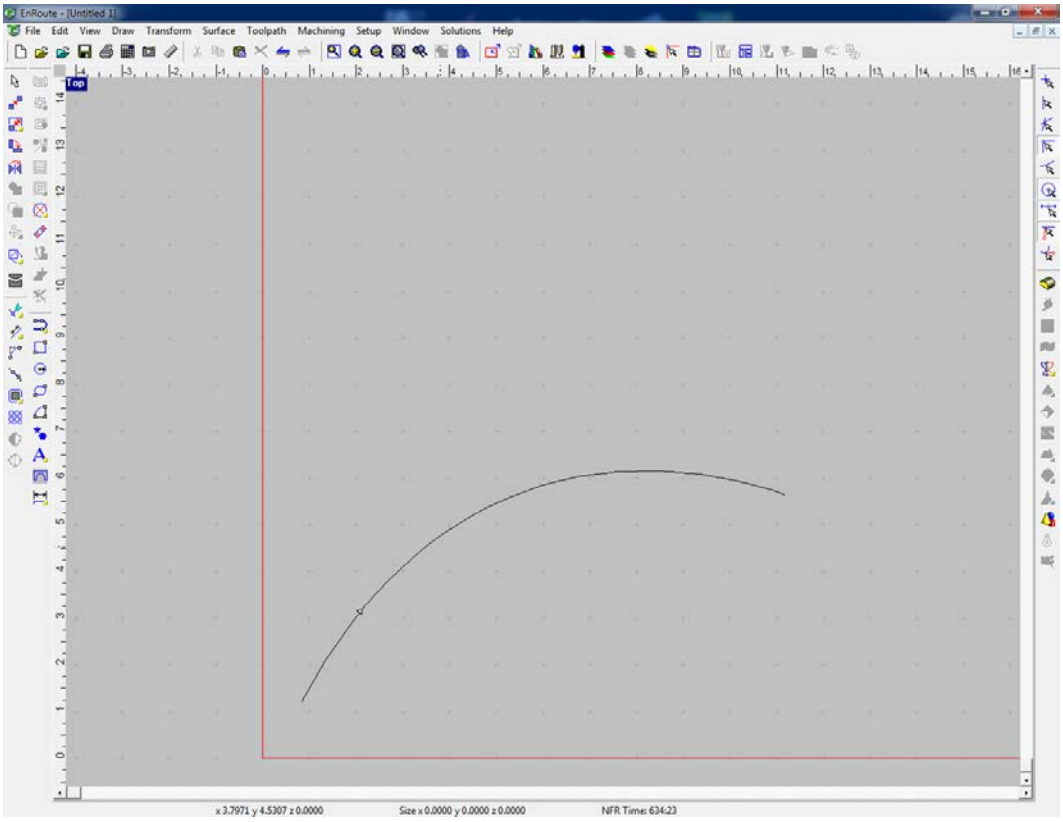
This is the top view of the Hatch Fill tool path. Note the hatch angle of 30 degrees.



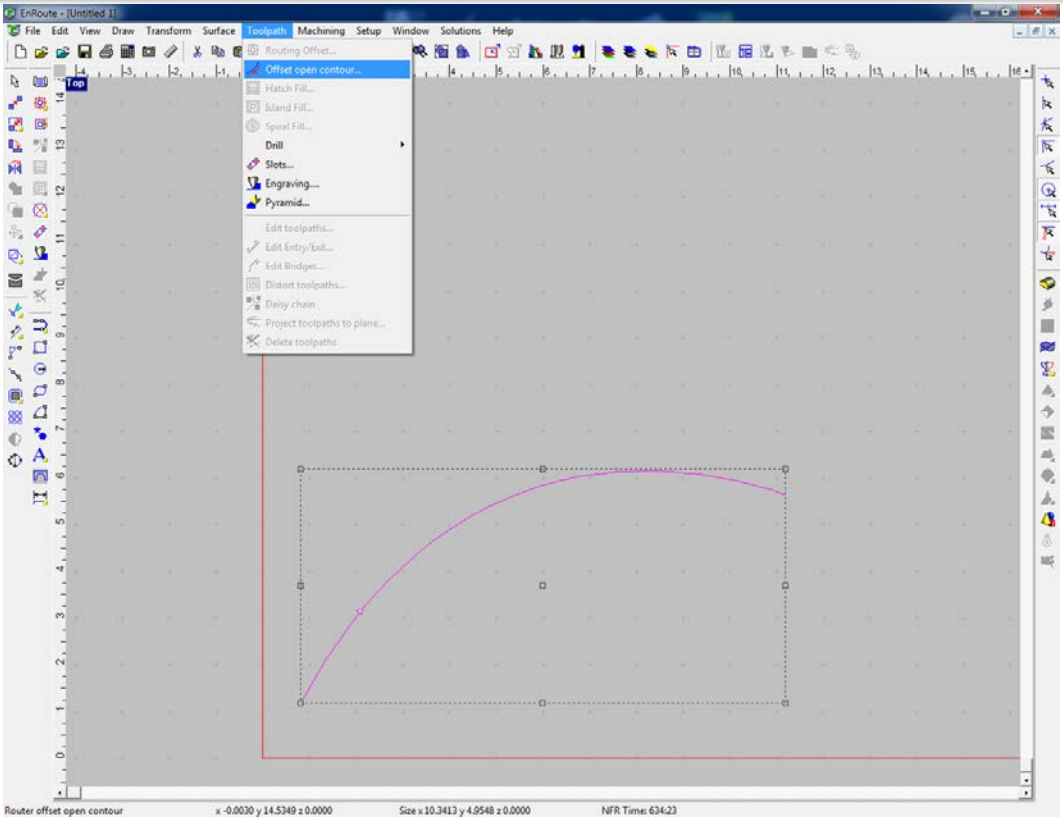
The “Island Fill” tool path will give you the same end result.
In this case, the path follows the contour of the shape.
This makes more sense than hatch filling in a lot of situations (like lettering, for example).



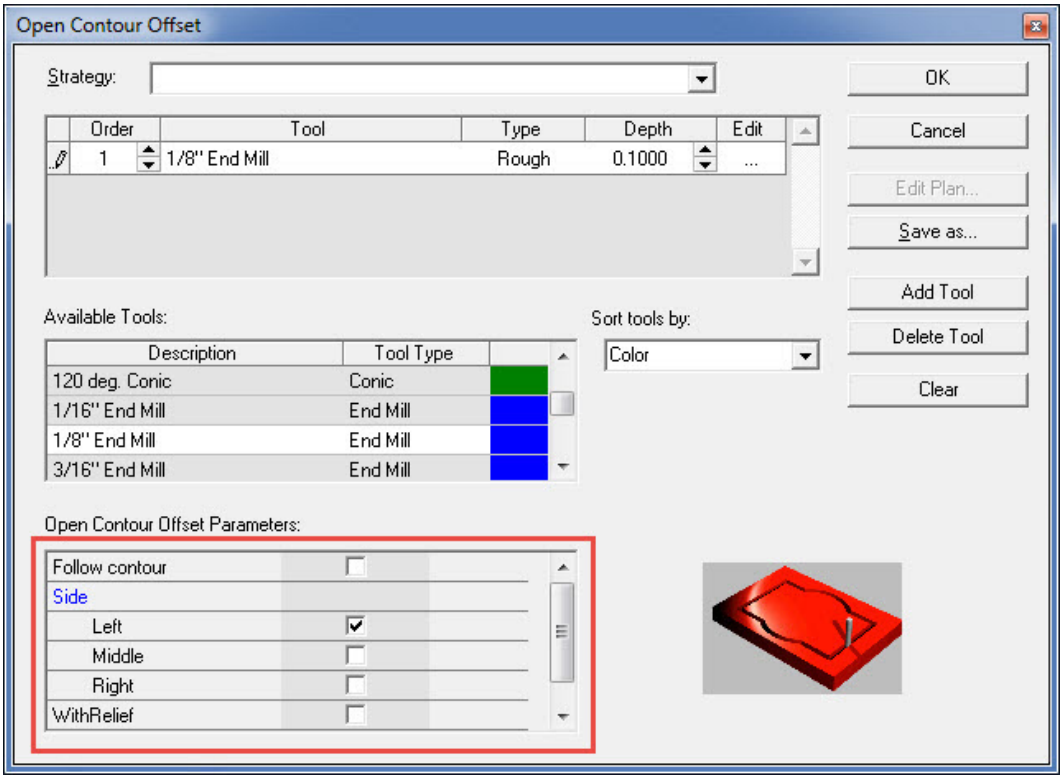
Here is a perspective view of the island fill.



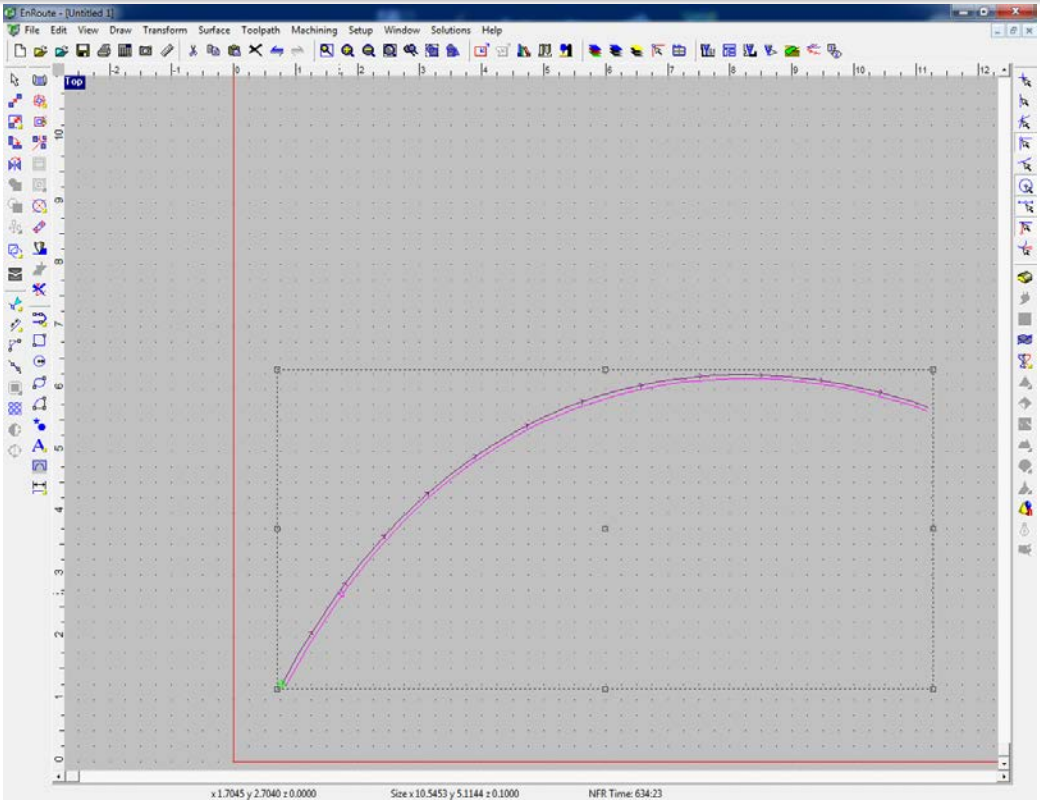
Now, let’s take a look at an open contour.



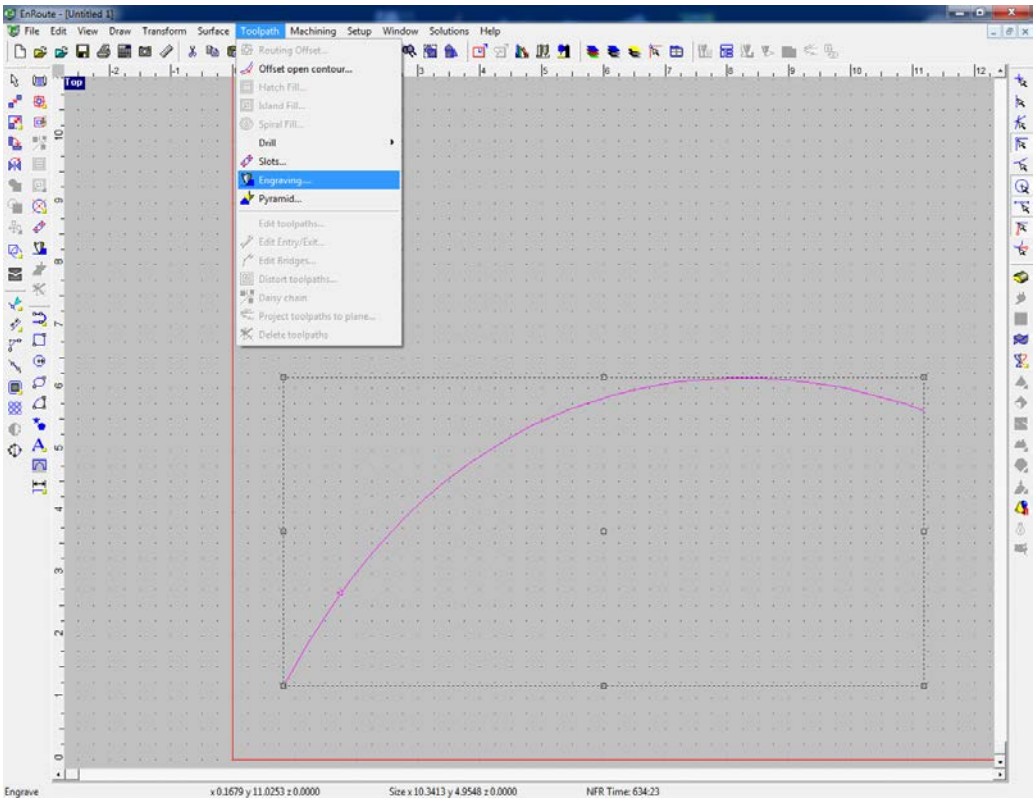
Some of the tool paths are only for closed contours, so only the paths that will work are available.



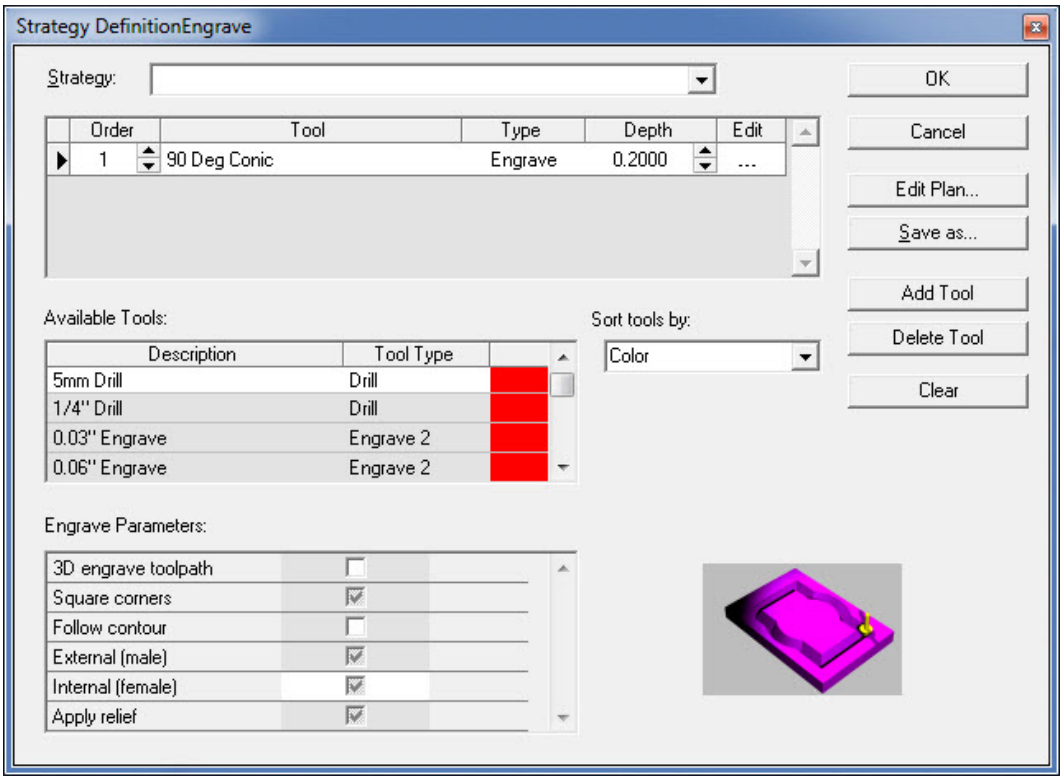
Choose whether to machine to the left, right, or center of the open contour.
The line direction will be visible on your top view.
Don't forget to edit your cut definition for depths, feeds, and speeds!



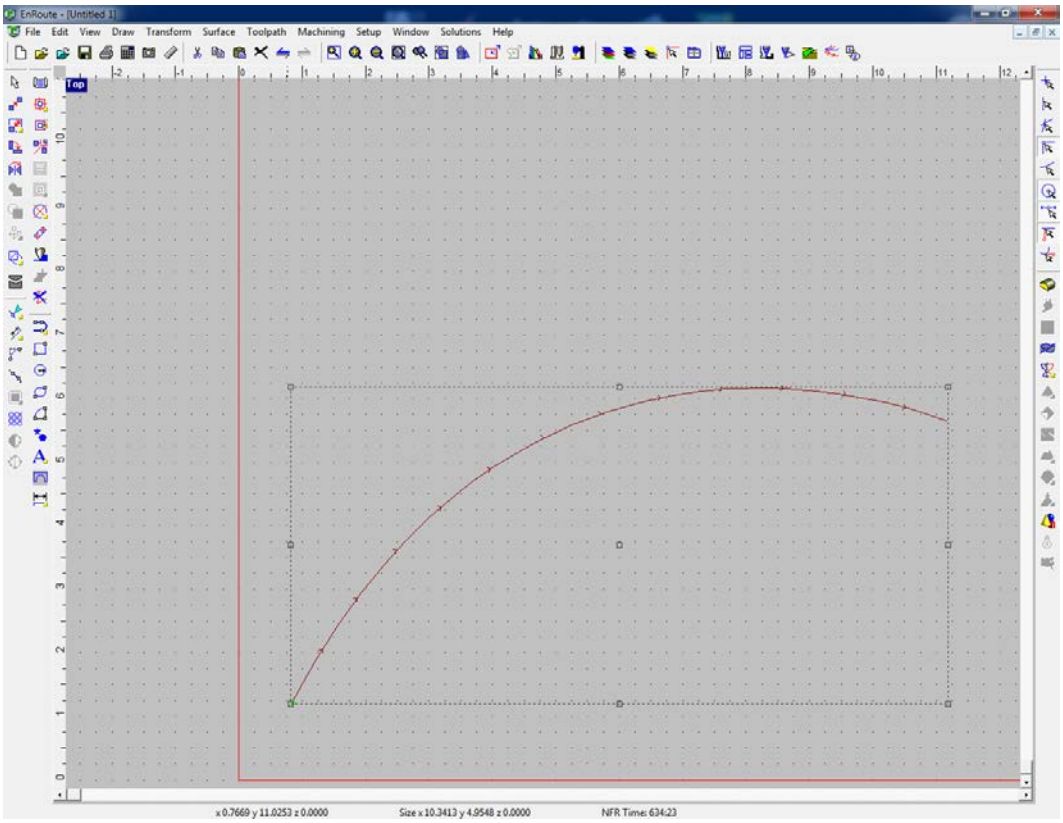
Notice how the cut is to the left of the line, relative to its direction.



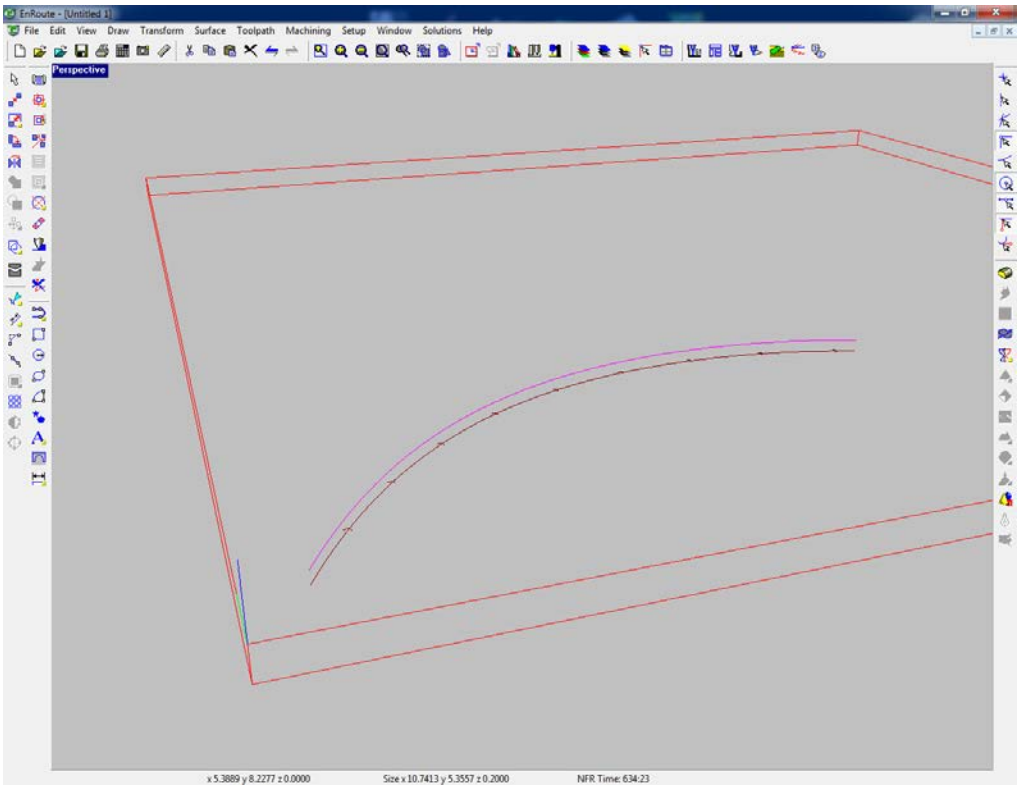
Another open contour tool path option is Engraving.
This will have the tool follow the line to a designated depth.
Cutting to the left or right of the line is not an option here.



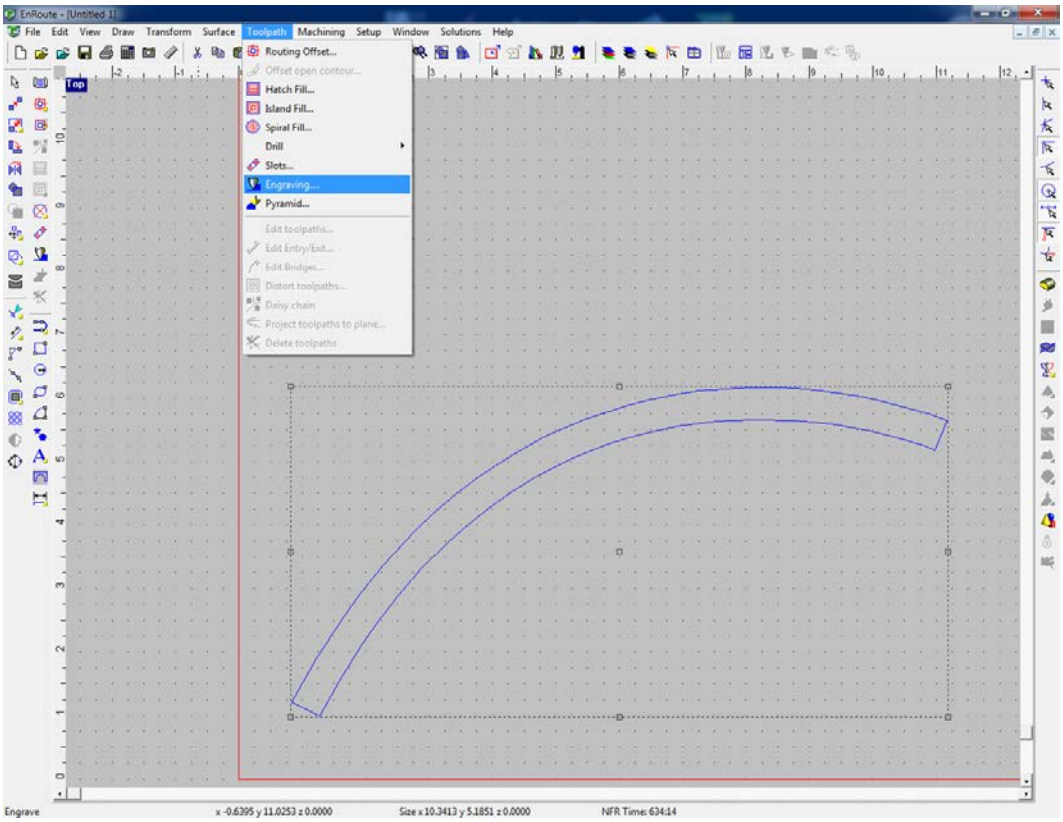
Select cutter and define cut as usual.



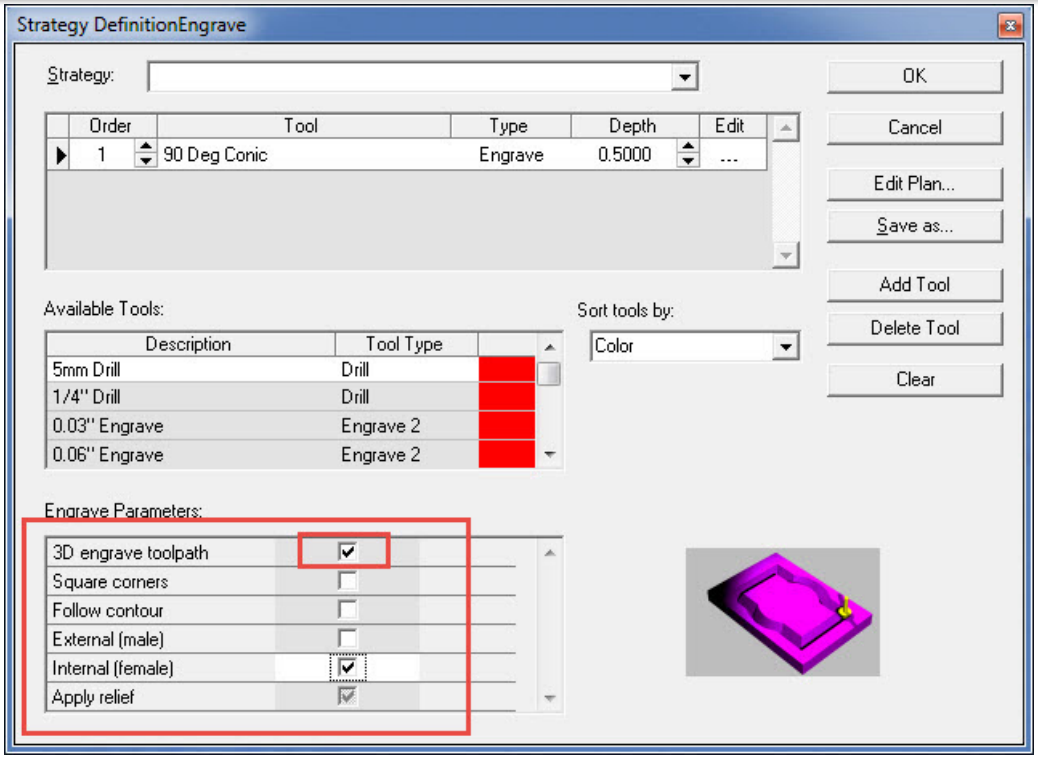
The cut follows the line.



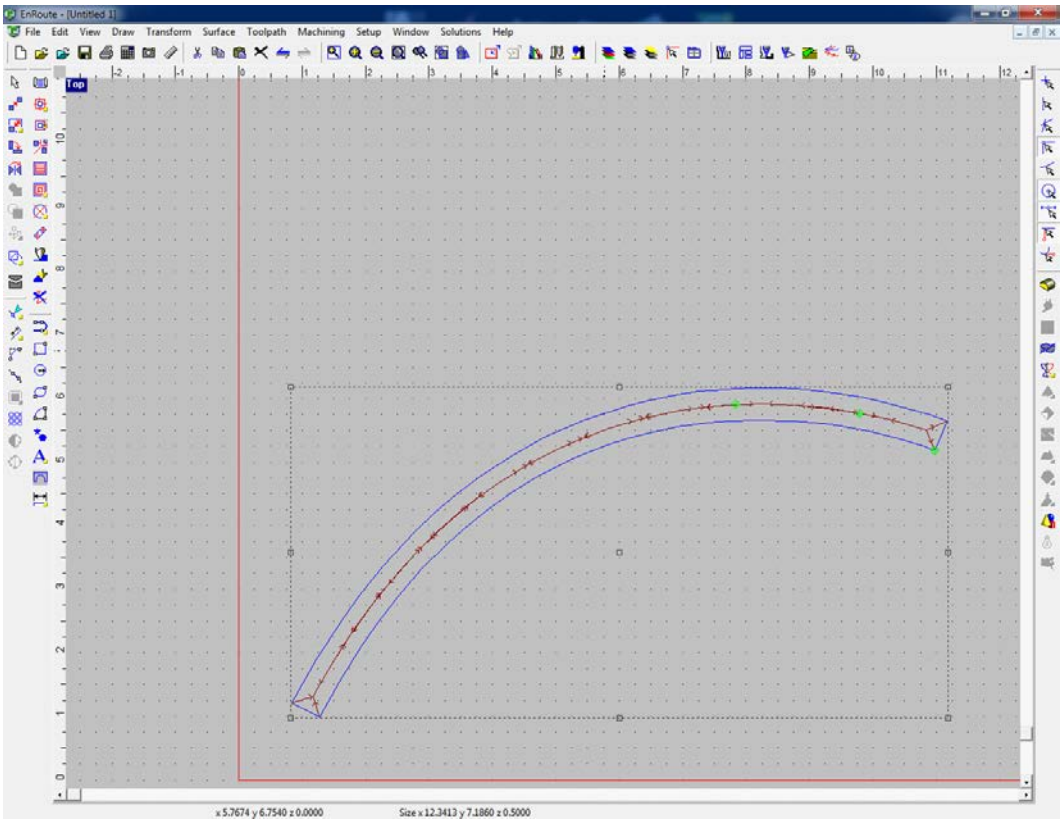
Perspective view showing the cut depth.



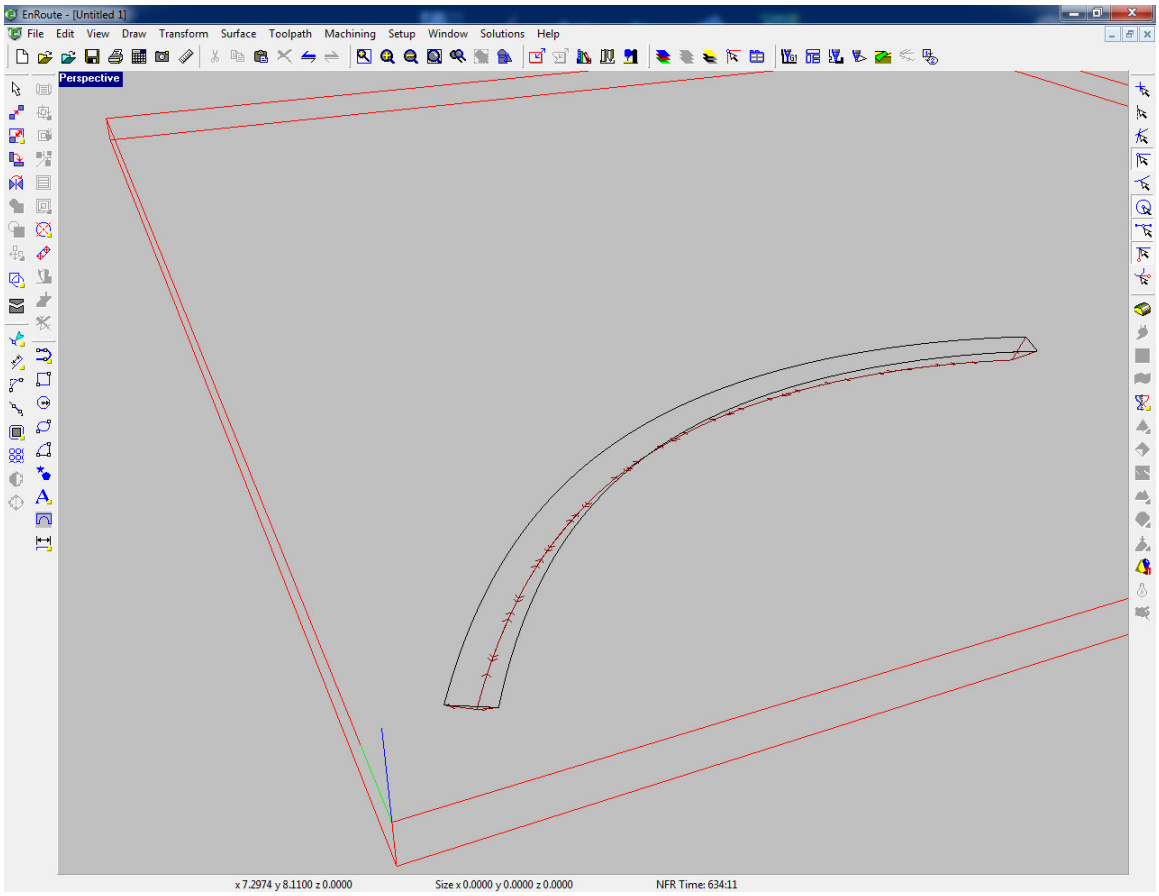
3D engraving or V-Carving is also available with the Engrave tool path.
A closed geometry is required.



The key to the carving is to check “3D engrave toolpath” in the cut definition. In this case, the depth of cut will be variable. You can set the max depth, but the cutter will lift and adjust based on the width of the contour it is carving. This allows sharp inside corners and fine detail.



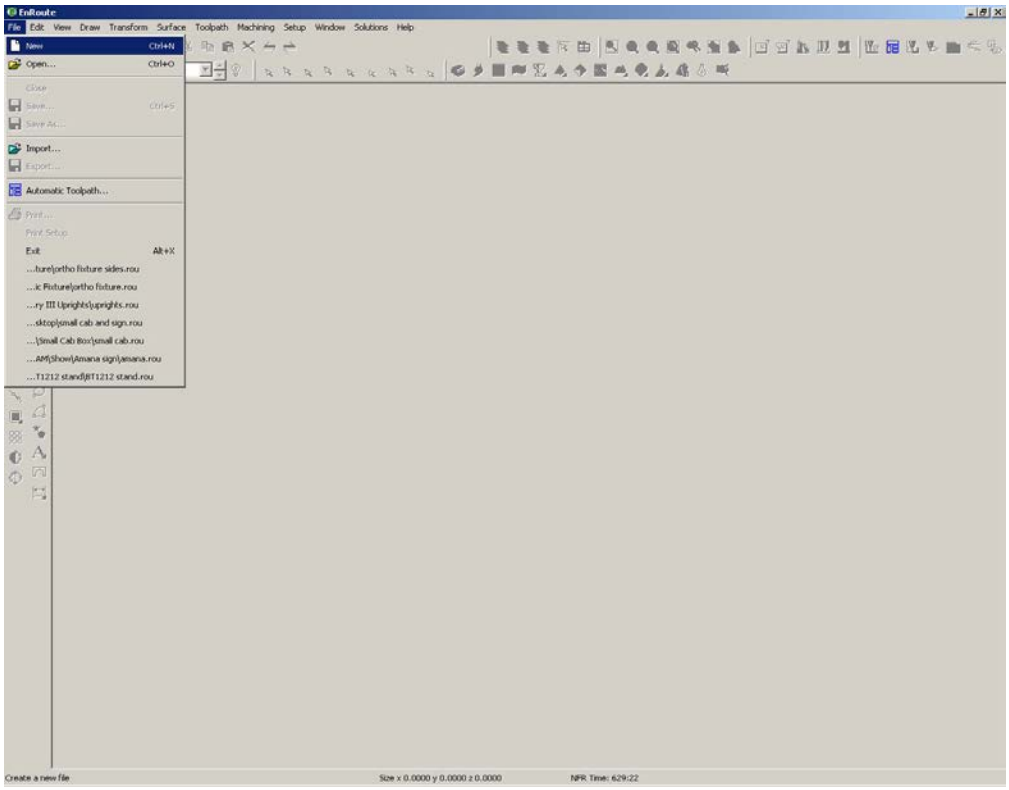
Top view of the 3D engrave tool path. Notice how the tool path is the centerline of the geometry.



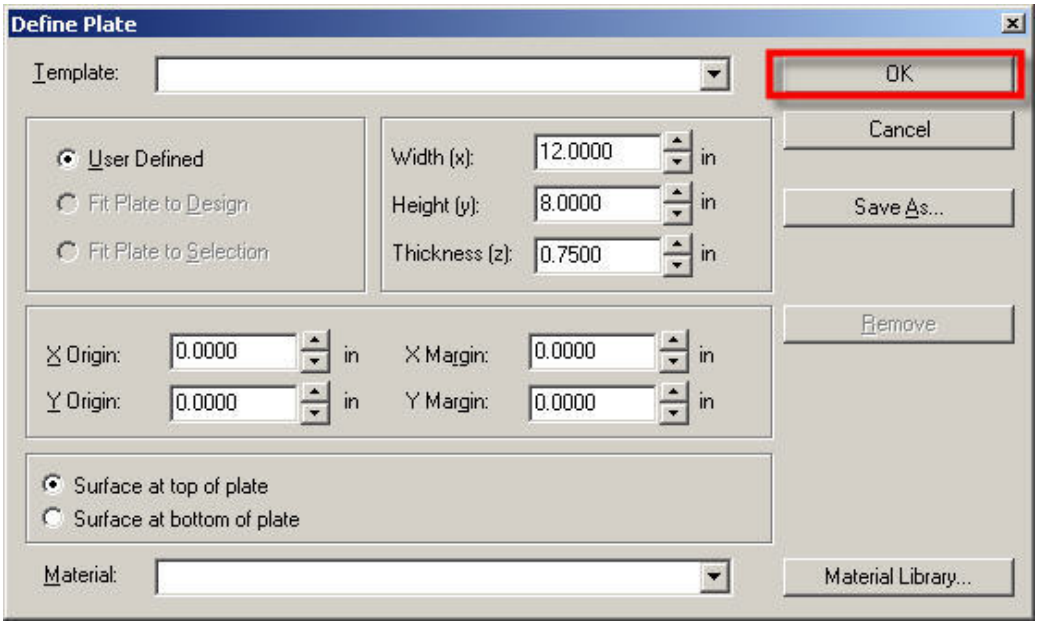
Here is the perspective view to illustrate how the cutter ramps in and out to give sharp inside corners.

EnRoute: Project 1

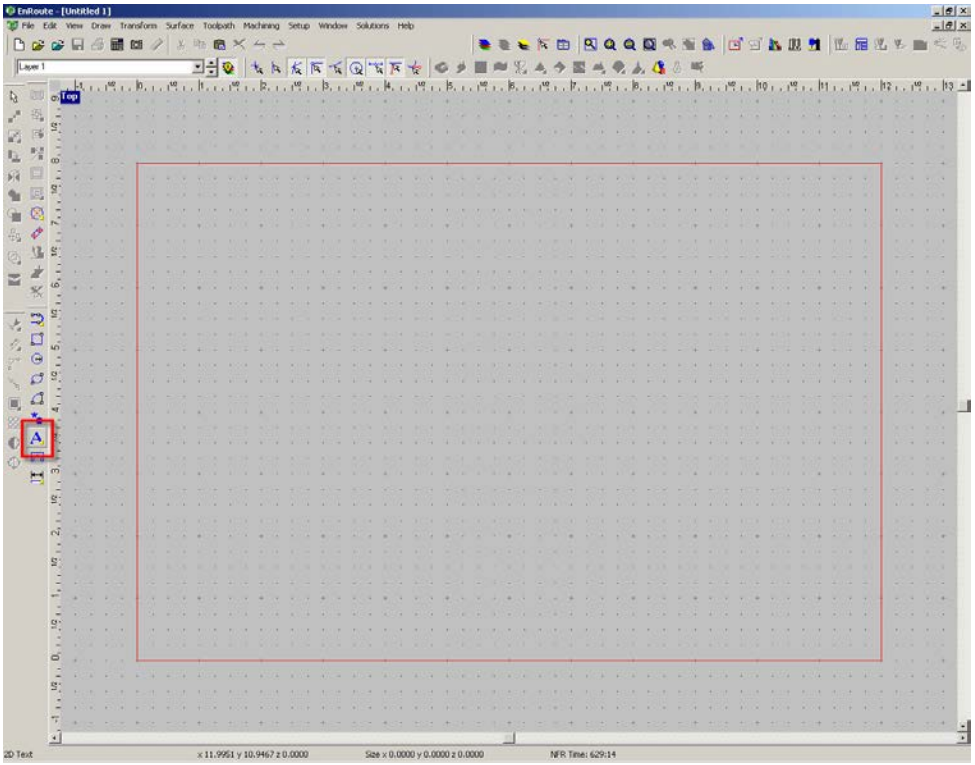
This section covers a nameplate project from start to finish.



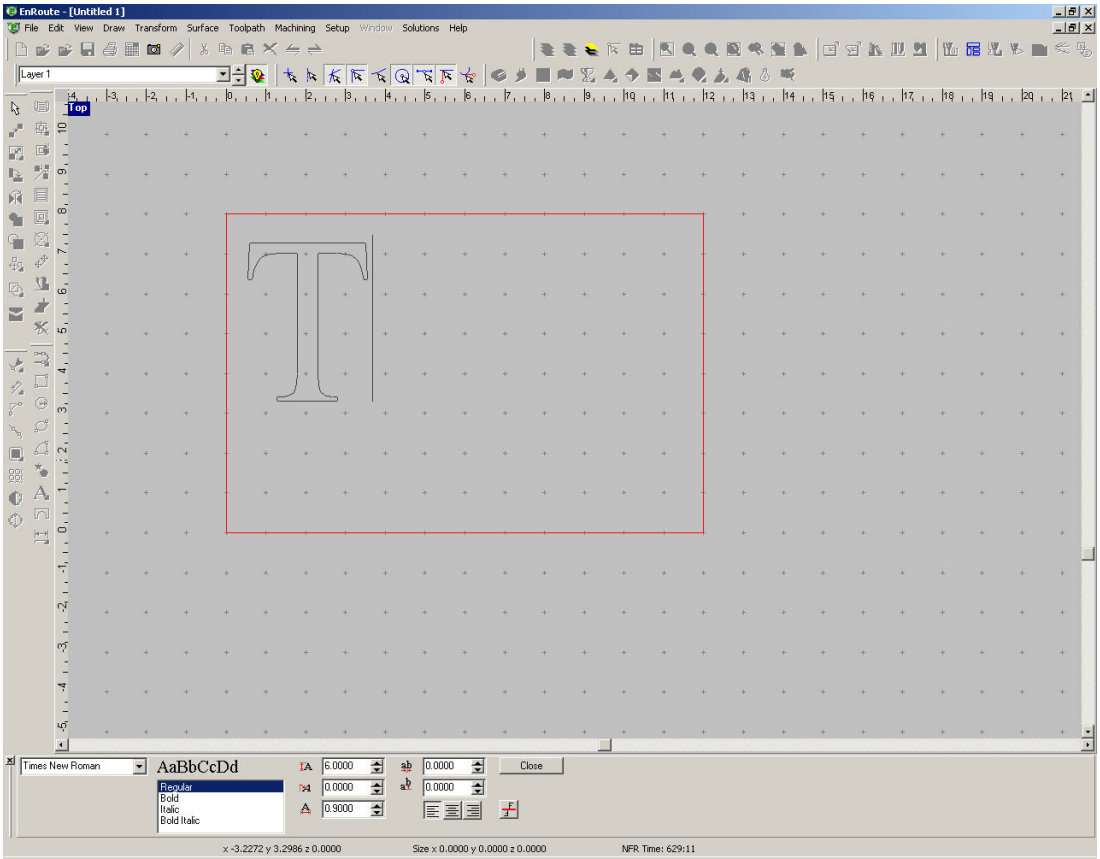
To start a new file, go to “File > Open”.



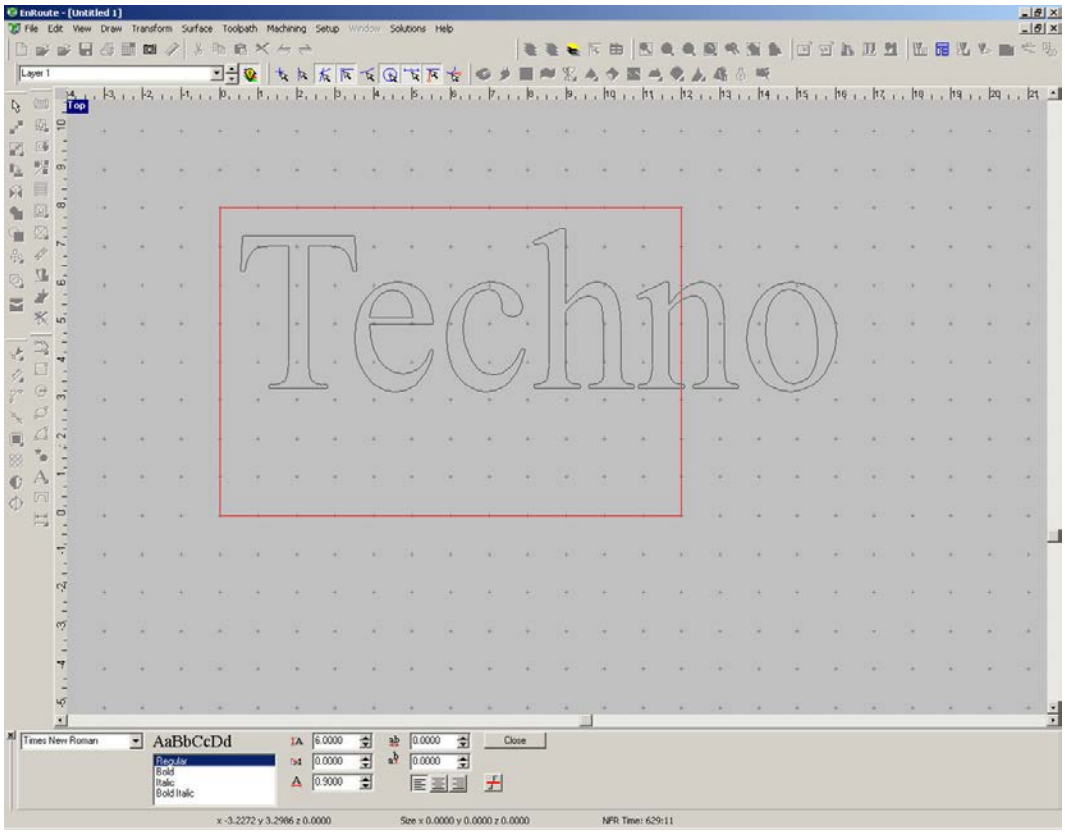
Enter your material size, make sure that the X and Y origins are set to 0 and the surface is at the top of the plate.



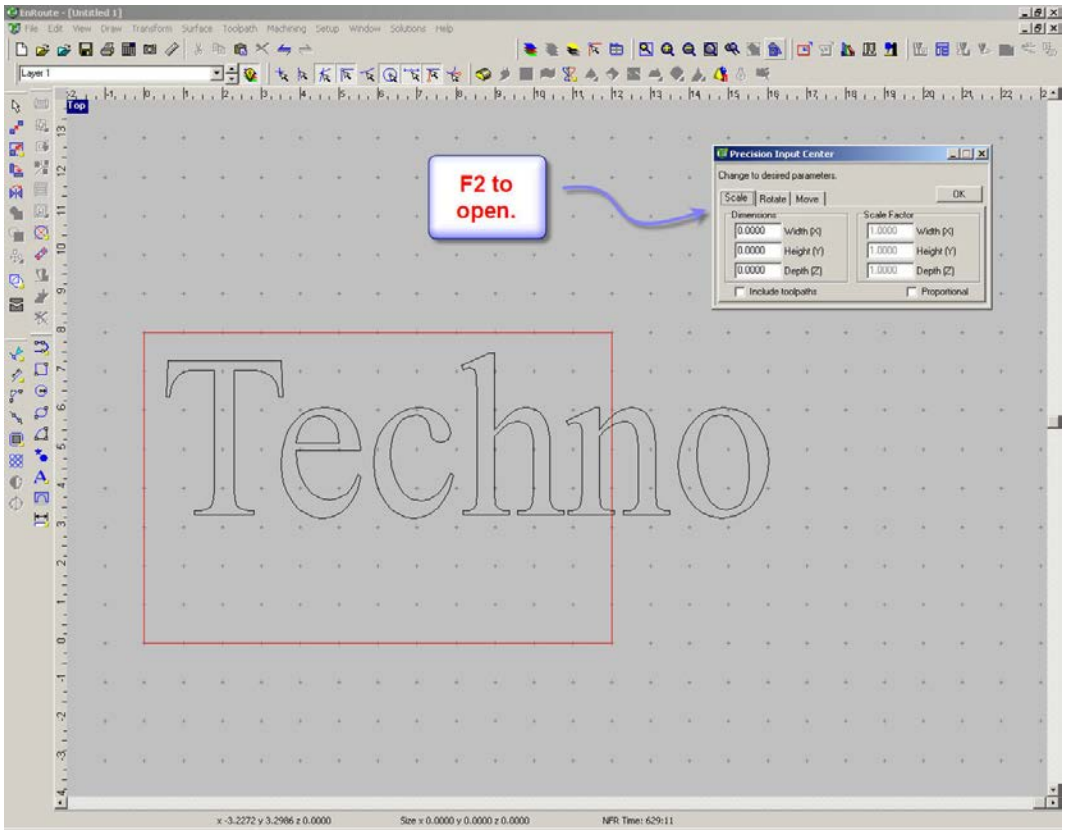
The first step of the drawing is to choose a font a write the nameplate text.



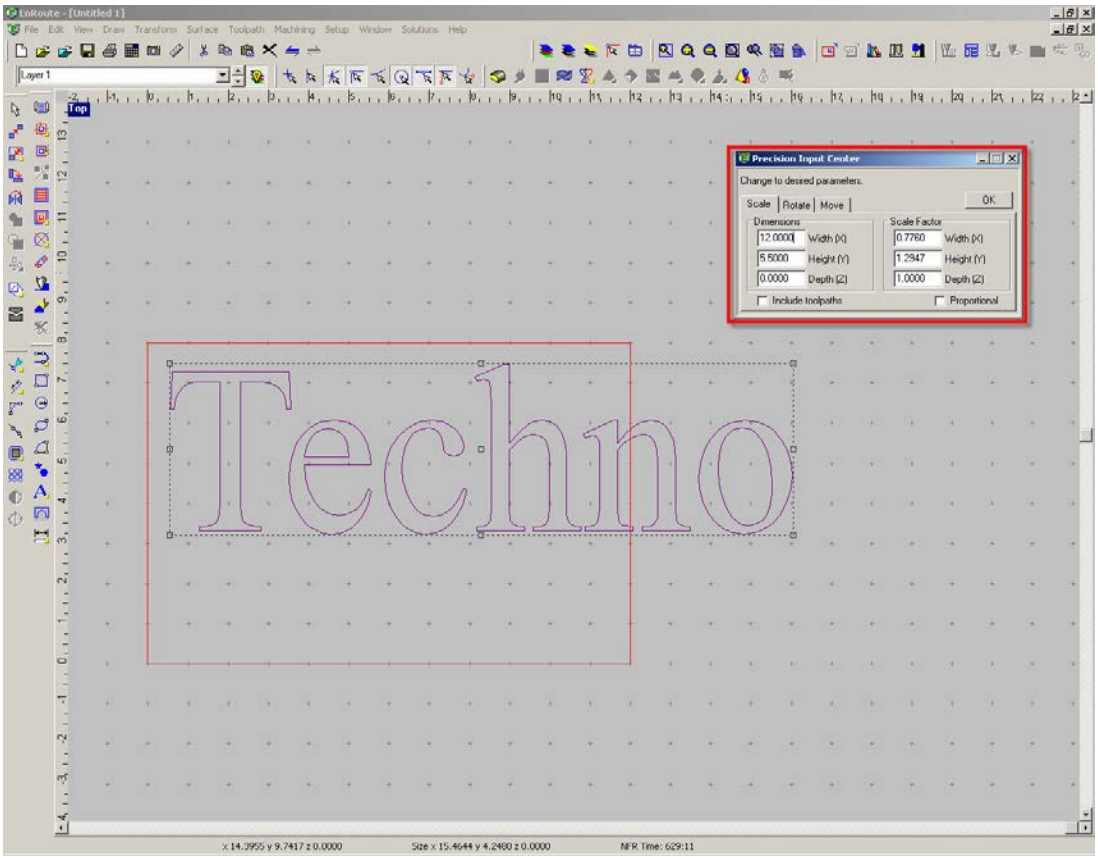
For now, just click anywhere and start typing. We will move the text into position later.



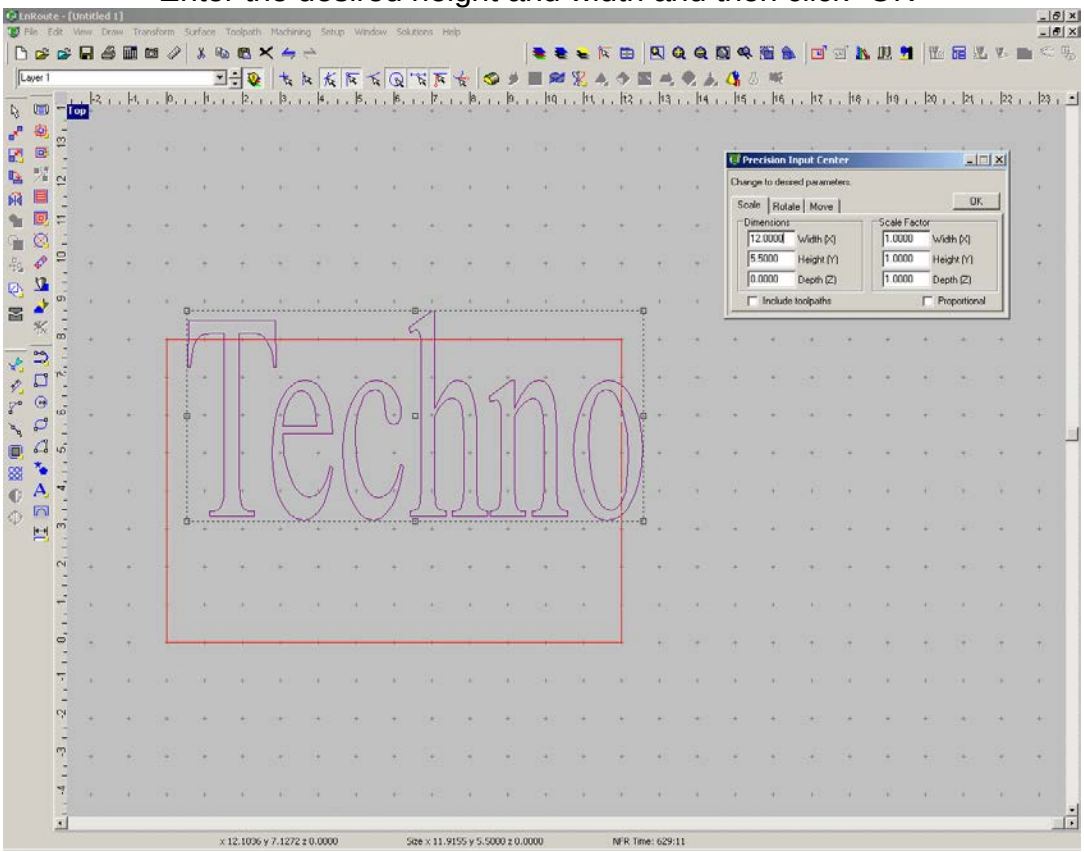
The finished text is too large for the plate. We will resize it.



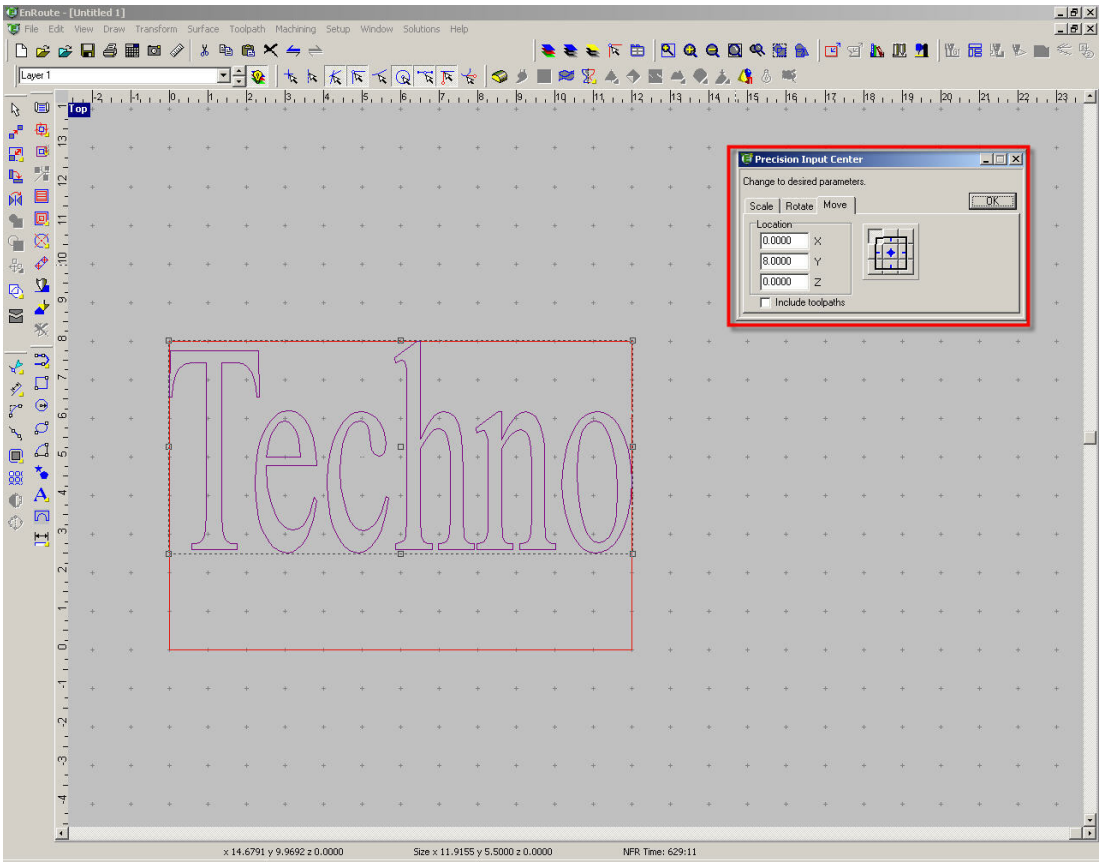
Hit F2 to open the Precision Input Center. This will be used throughout the project.



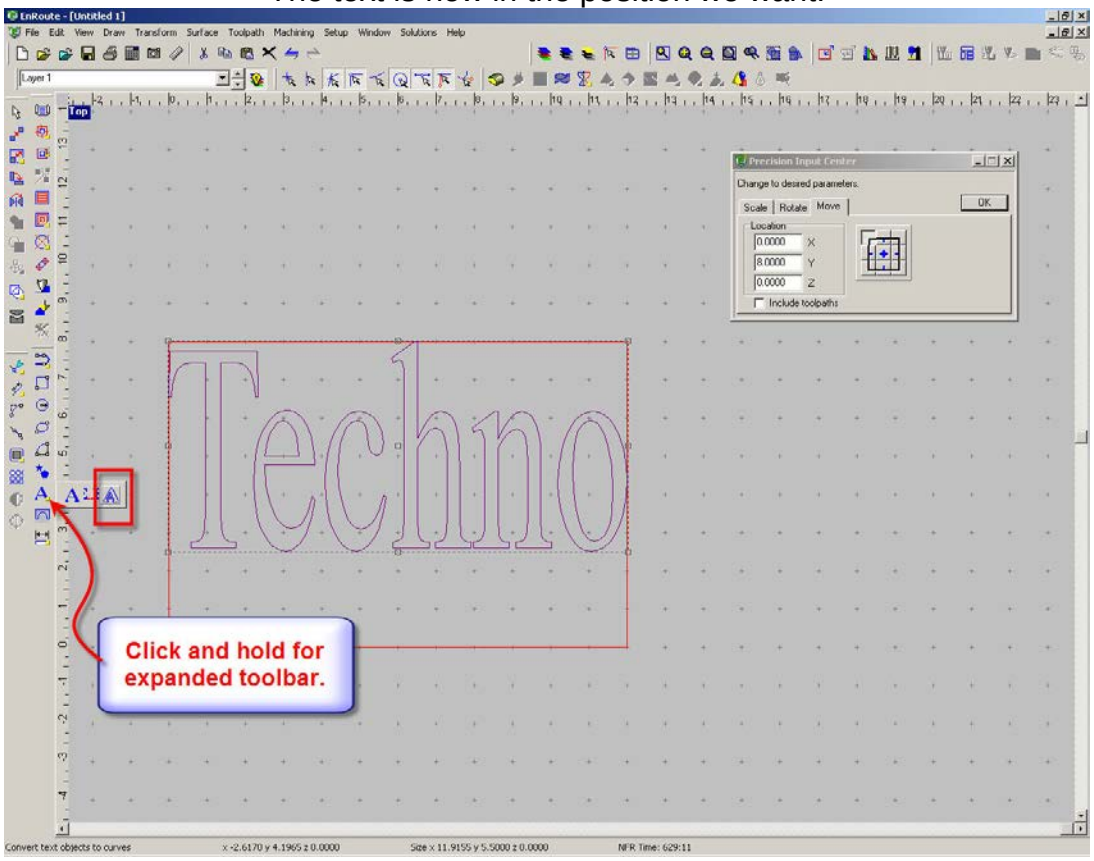
Uncheck “Proportional” so we can adjust the height and width separately. Enter the desired height and width and then click “OK”



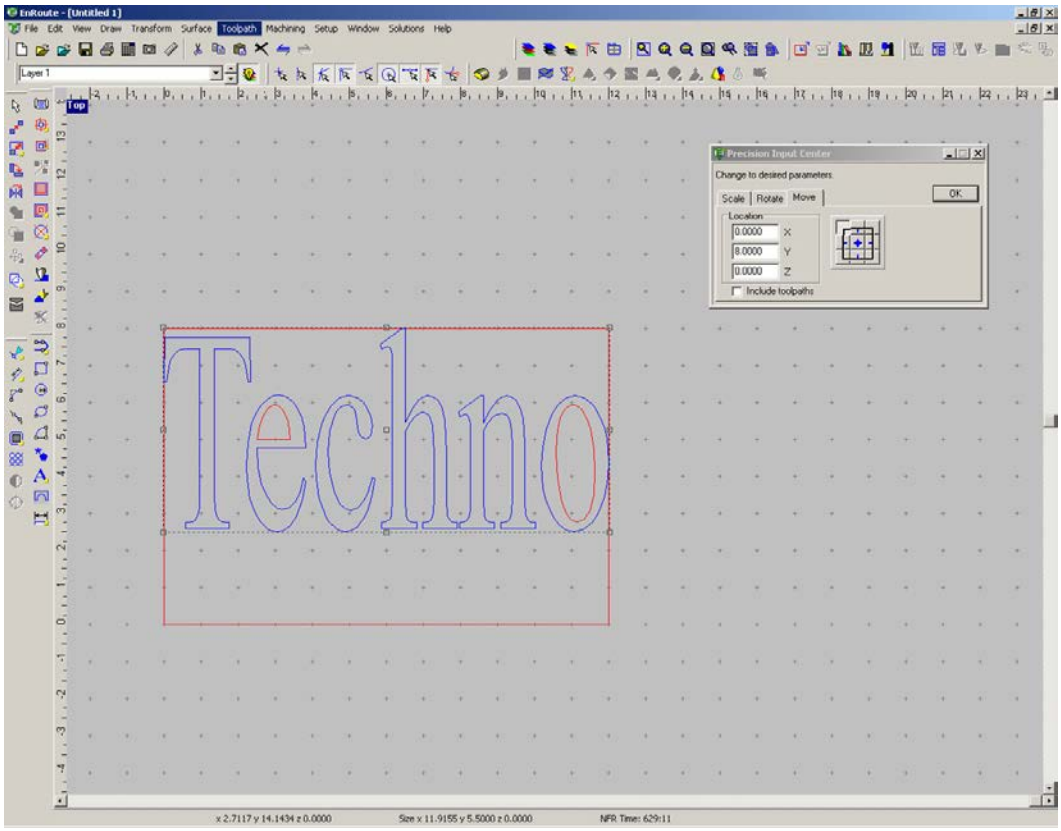
The text changes to reflect the new dimensions. Now we can move it into the correct position.



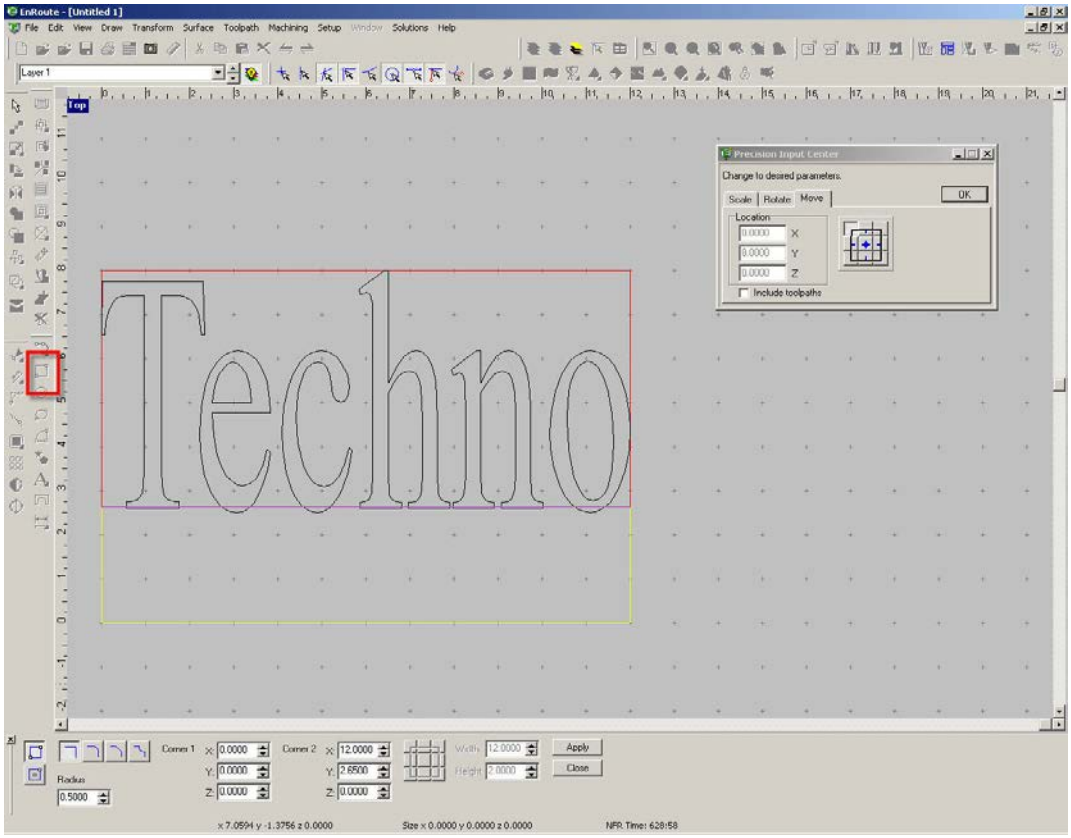
The text is now in the position we want.



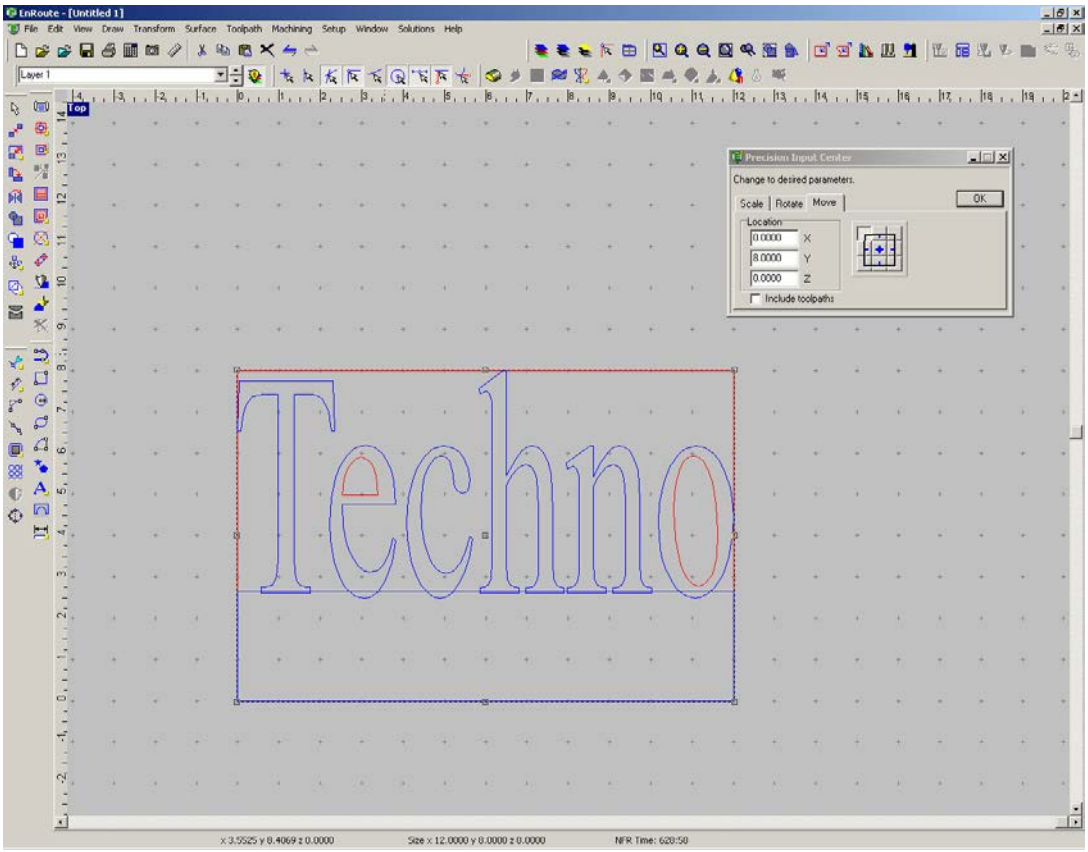
Click and hold the text icon to expand the toolbar.
Then click on the icon to change the text to curves.



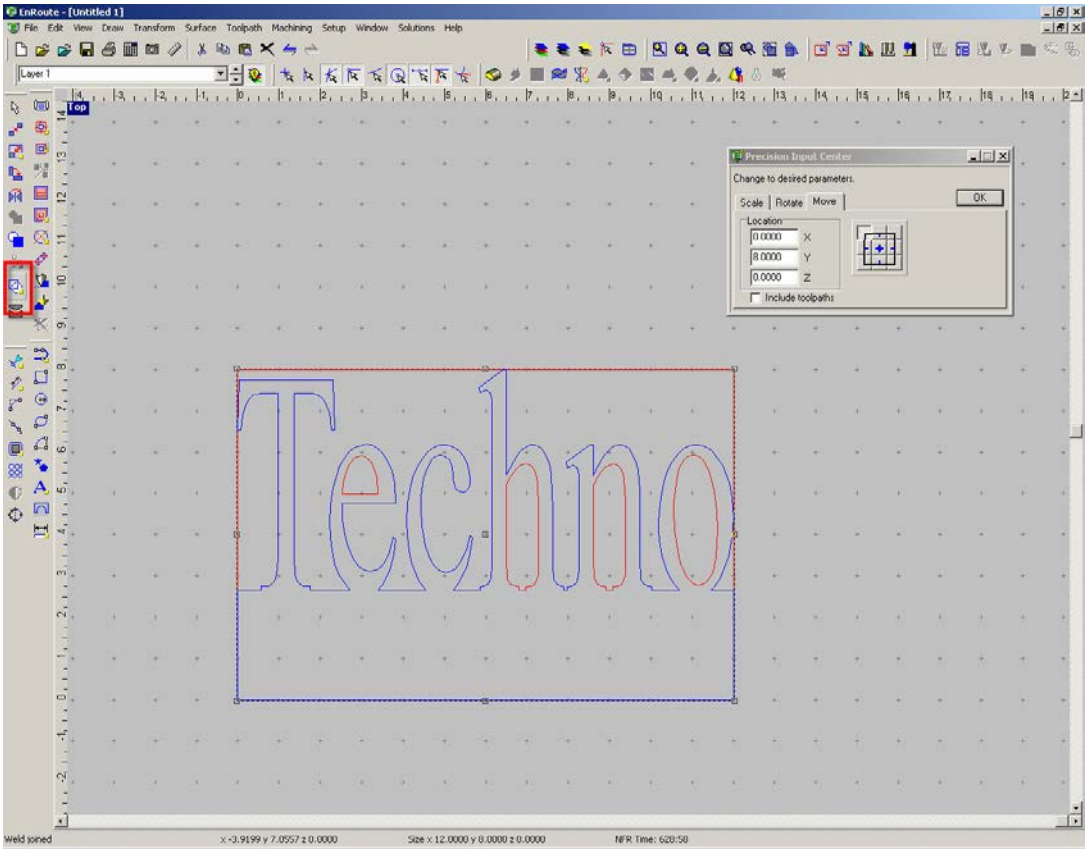
The text is now vectorized. Notice how the inside geometries are red. This is EnRoute's way of indicating that it recognizes a closed geometry inside of another closed geometry.



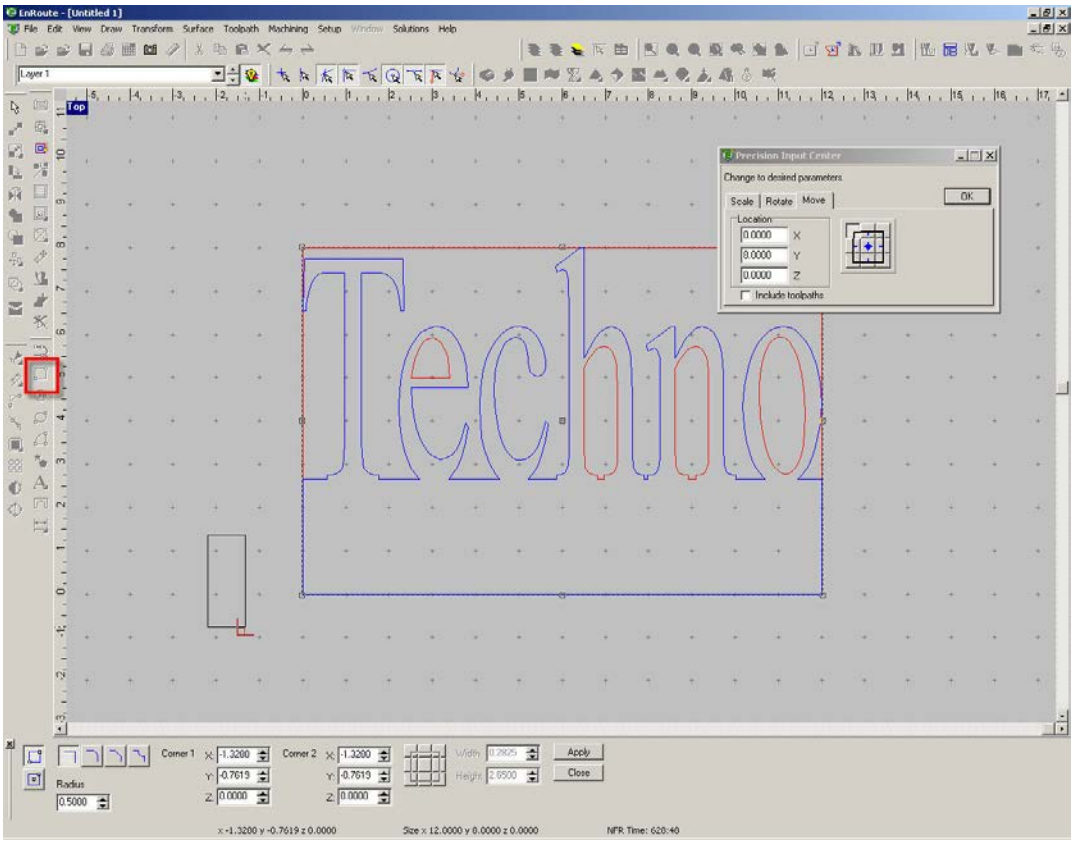
To make the base of the nameplate, select the "draw rectangle" tool.



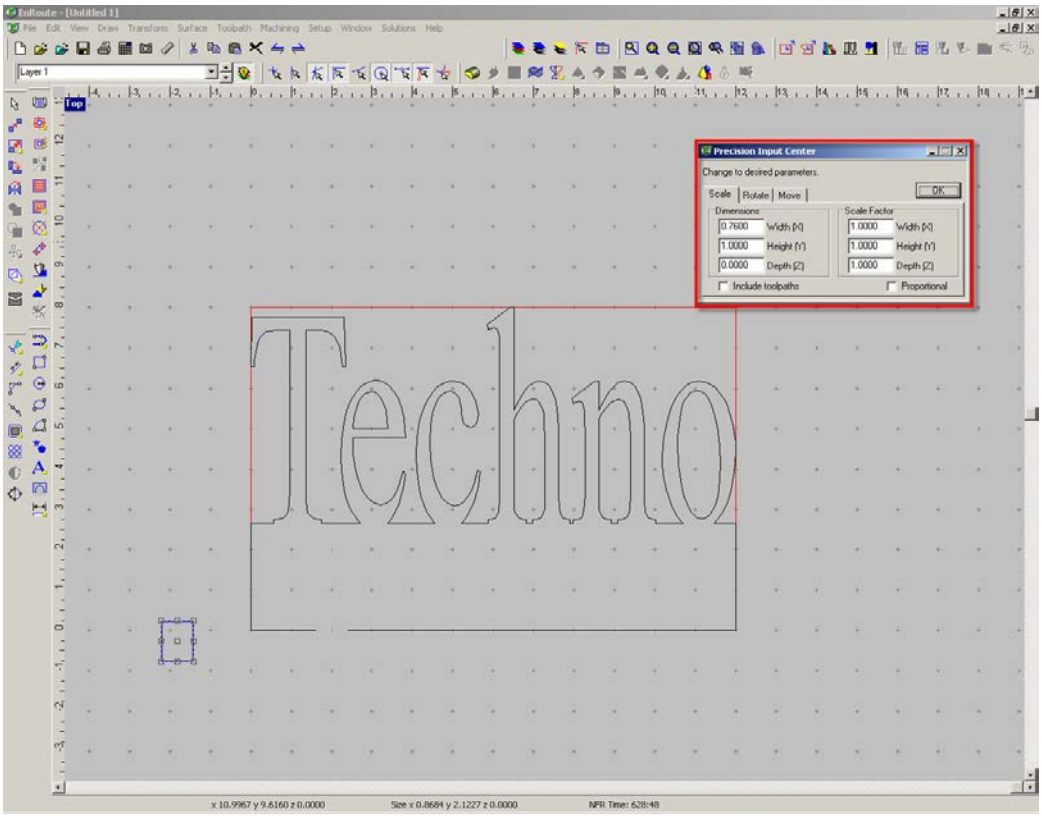
The rectangle should slightly overlap the bottoms of all the letters.



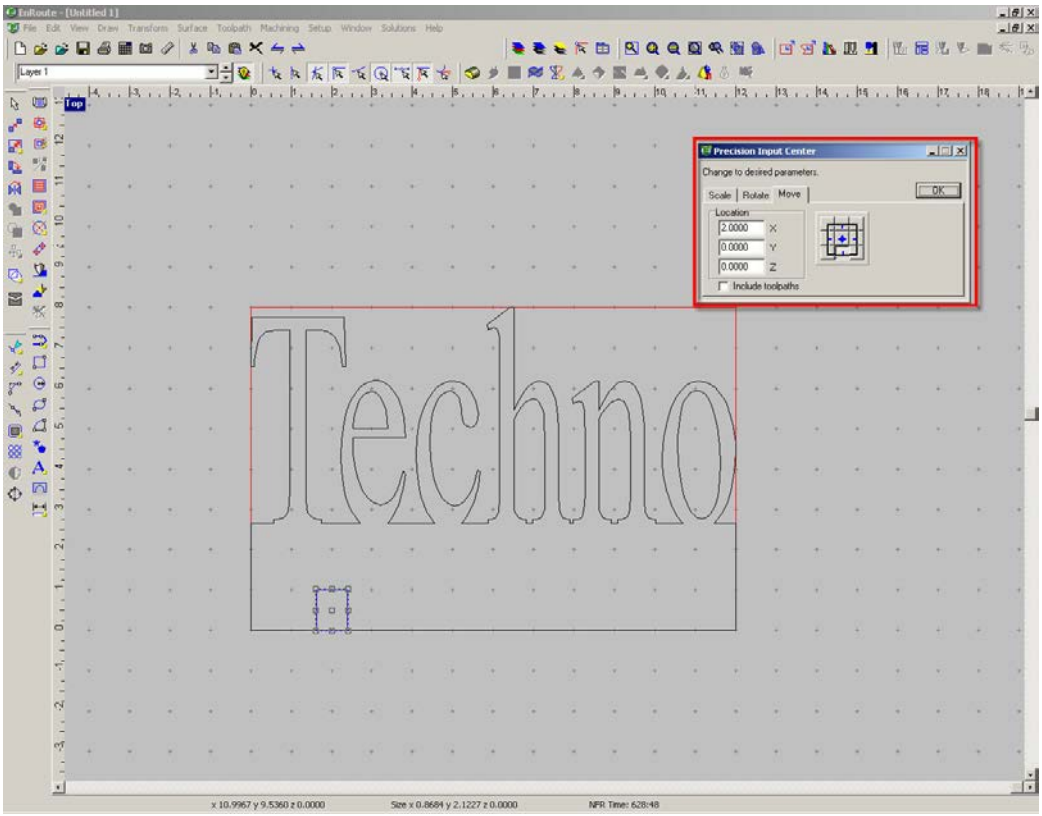
Highlight both the lettering and the rectangle.
Then click on the “weld joined” tool to automatically trim the overlapped lines.



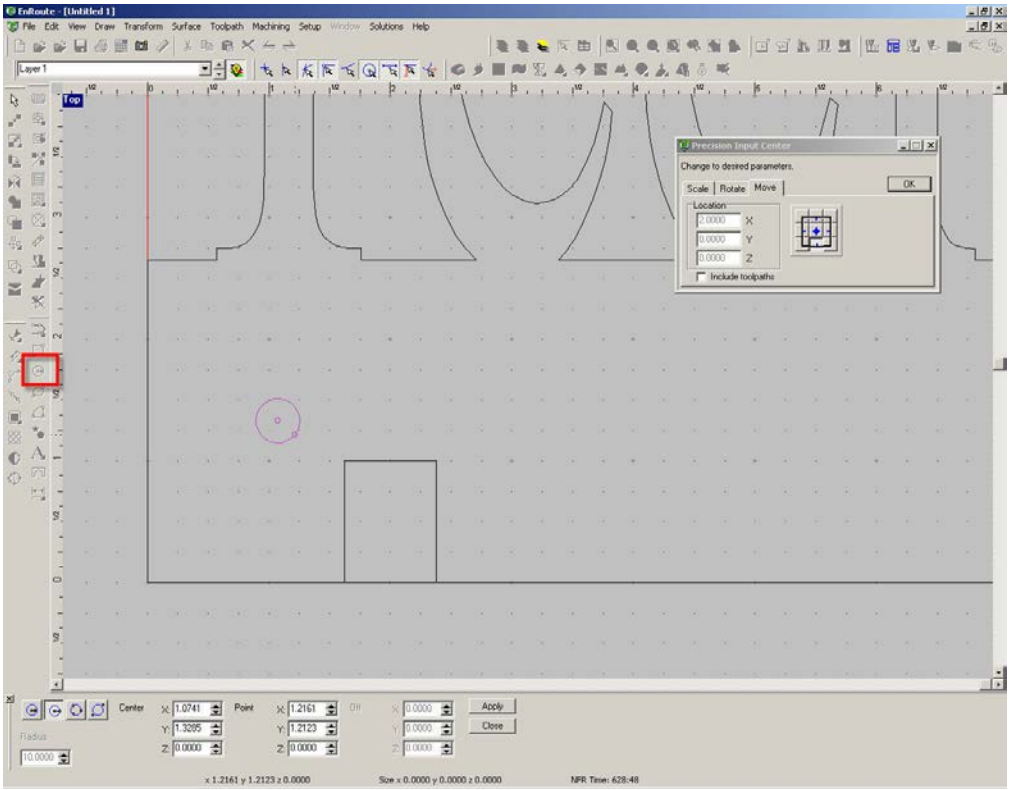
Now we have to add the slots for the nameplate feet.



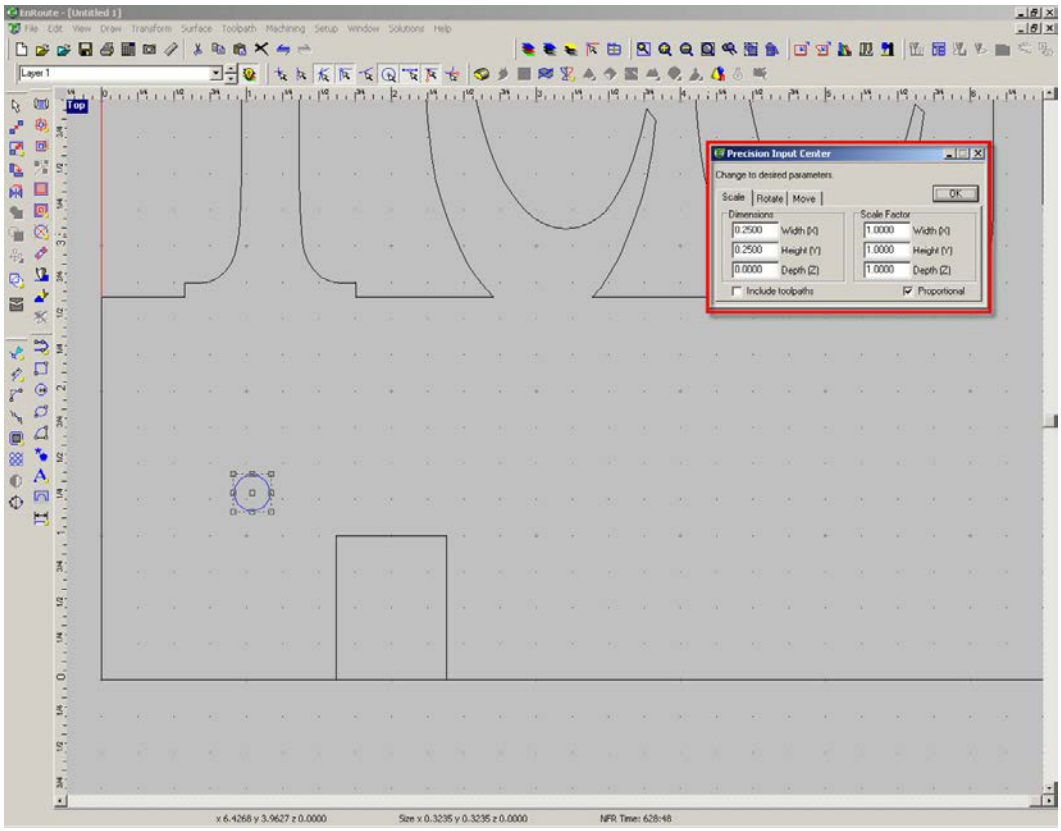
Using the PIC, resize the new rectangle to .76” wide by 1” high.
We want to make the slot slightly wider than the thickness of the material.



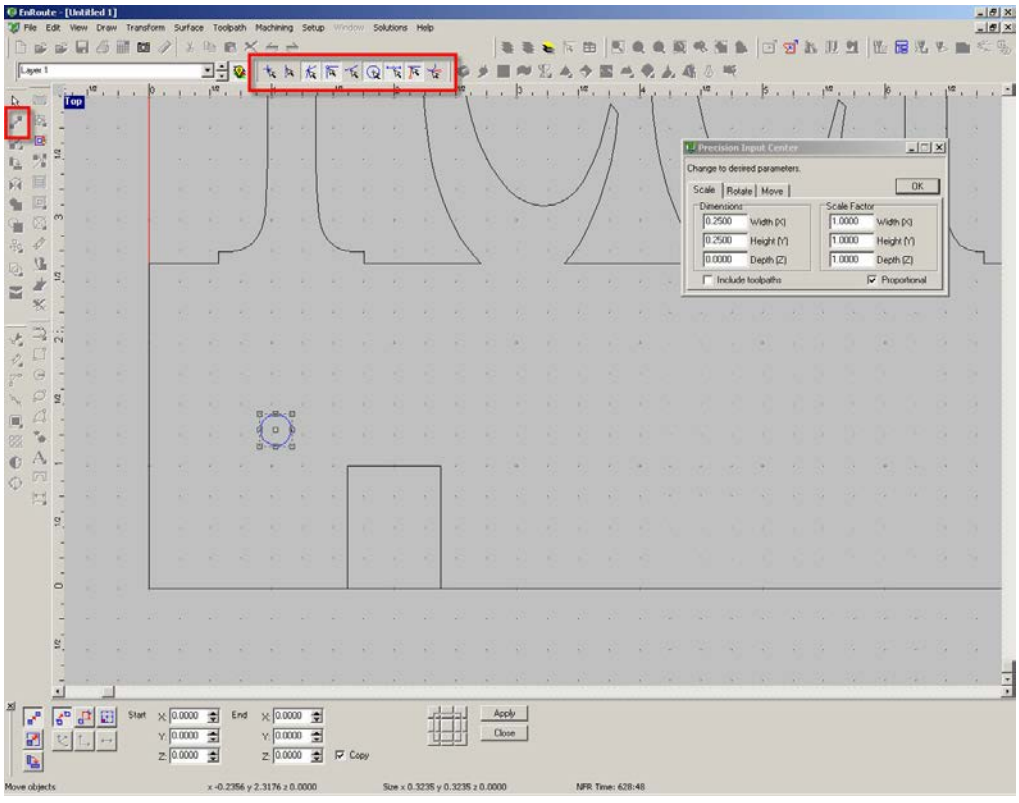
Now move the resized rectangle so that the bottom center is at X=2 and Y=0. The PIC grid selects the anchor points.



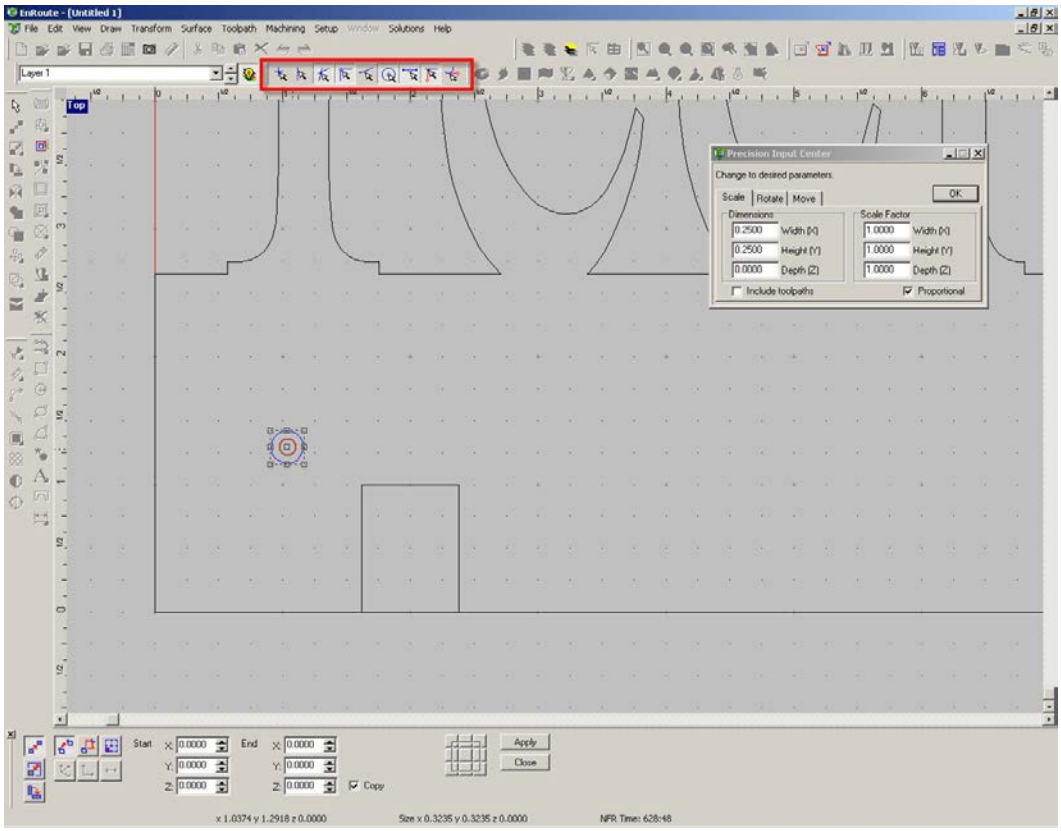
Now, so we don't have to worry about inside corners getting in the way later, we can add a dog-bone fillet to the slot.



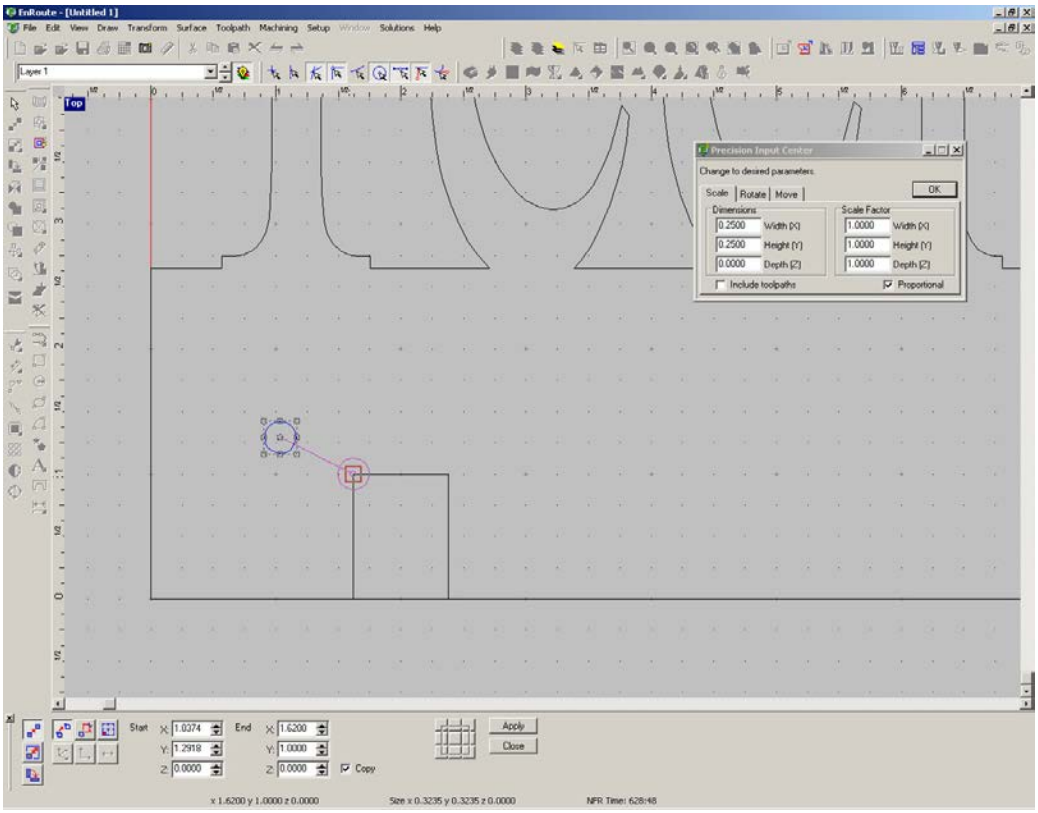
Create a circle and resize it to .25".



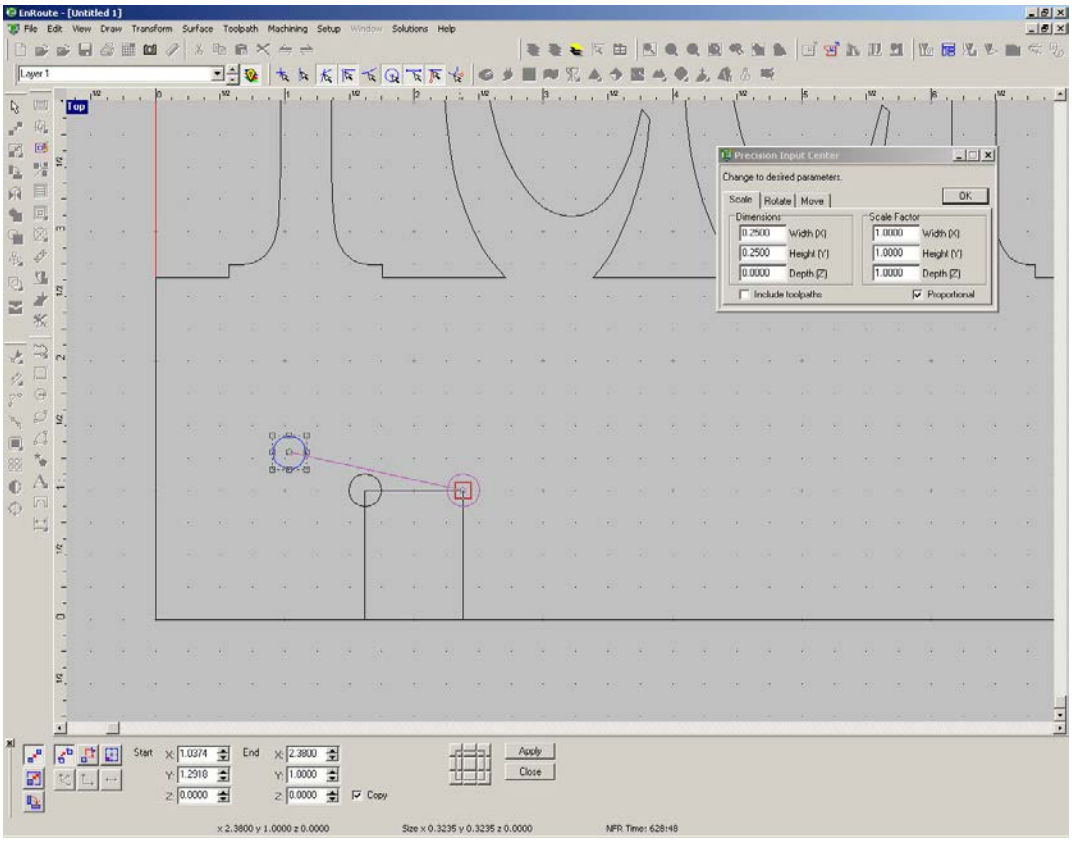
Now we can use our "move objects" tool to copy the circle to the corners of the slot. Make sure that you have the appropriate "snaps" turned on.



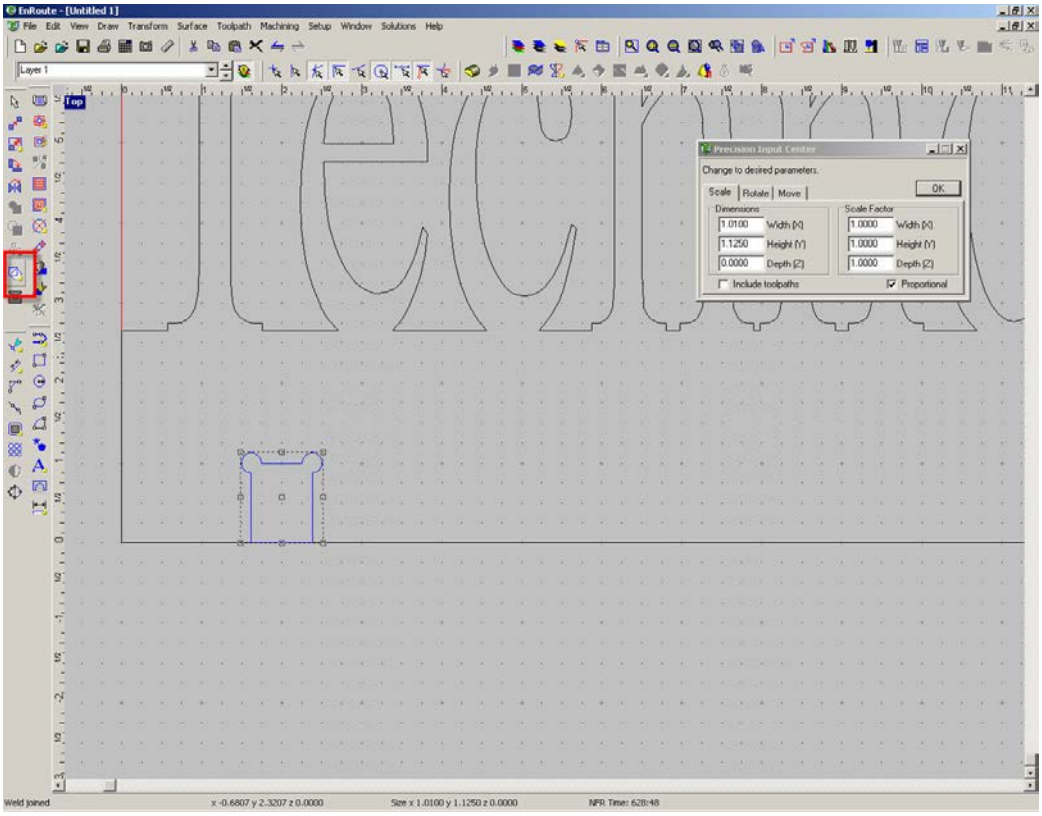
Grab the circle from the center and copy it to the first corner of the rectangle.



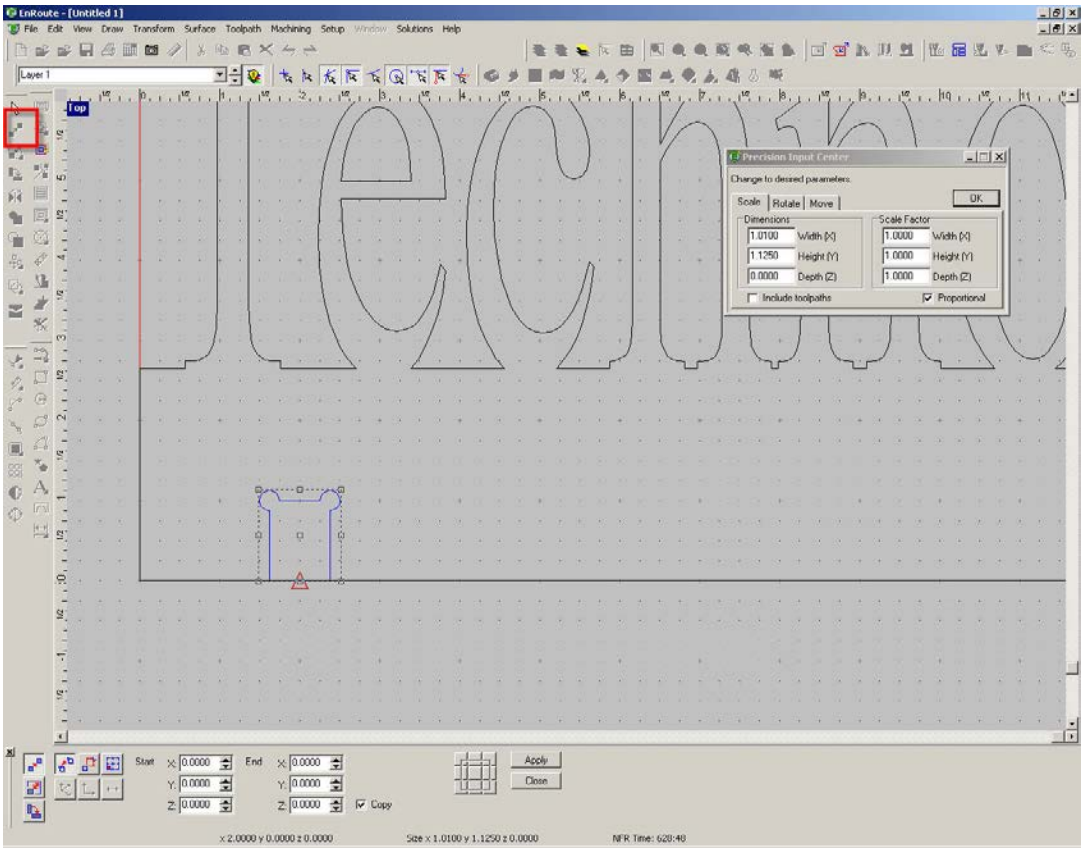
Click to apply.



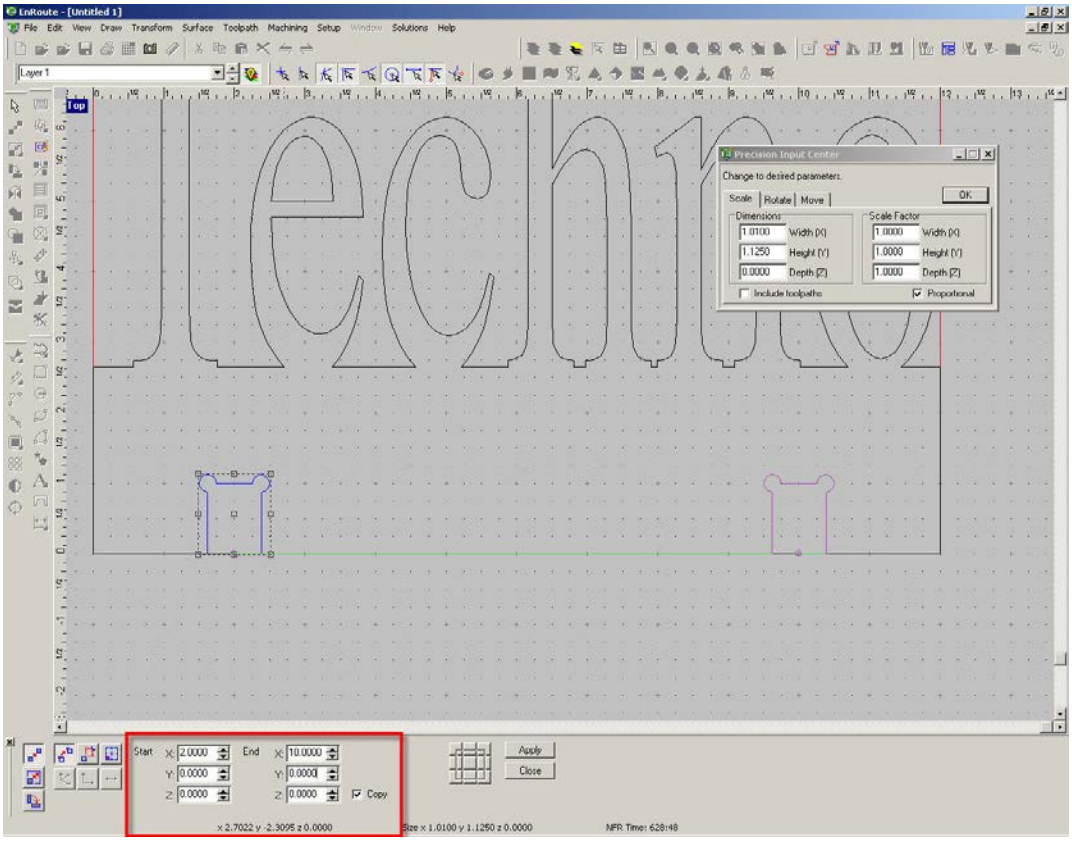
Repeat for the other corner and then delete the original circle..



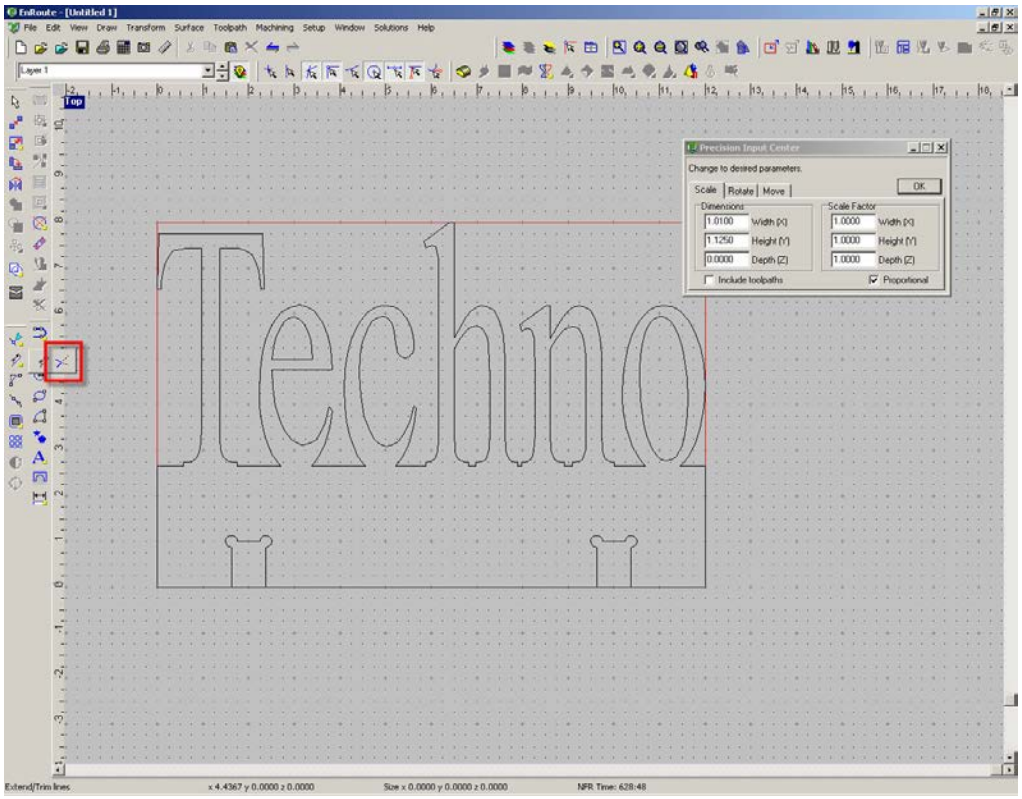
Select the rectangle and circles and use the “weld joined” tool again to trim the overlap.



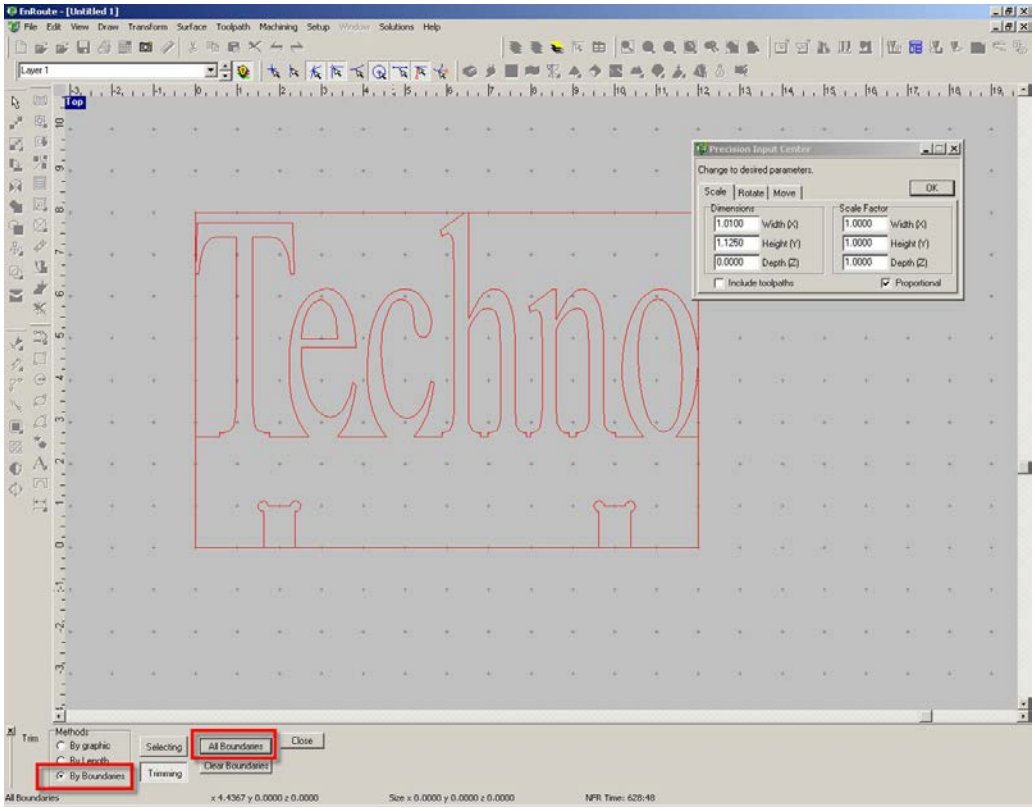
Now, we want to copy the new geometry to the other side of the nameplate for the second stand.



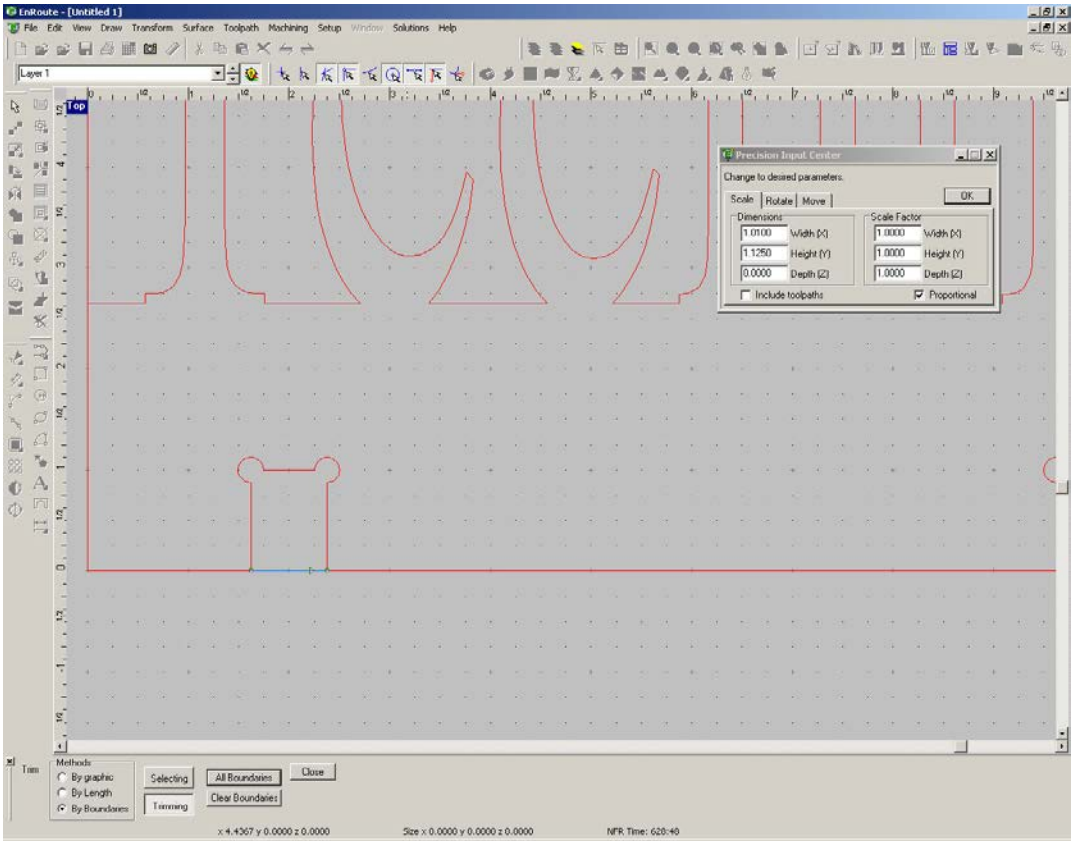
Using the “move selected” tool again, copy the slot so the center is at X=10, Y=0.



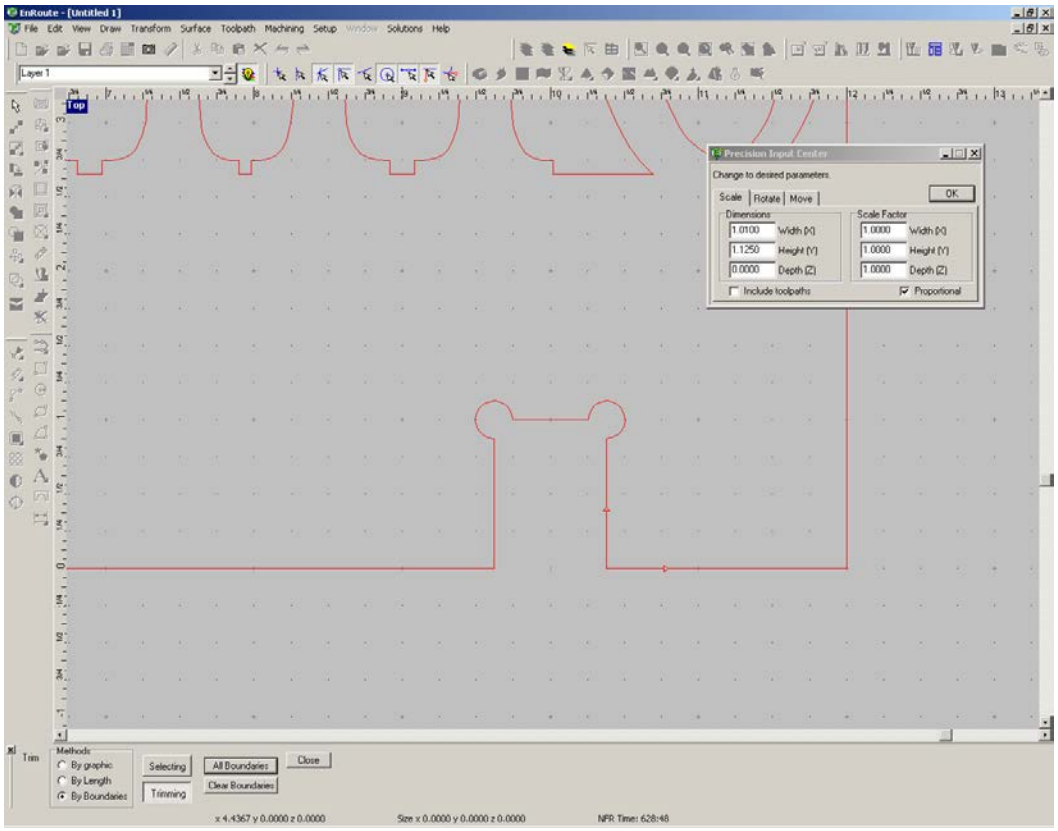
Now we will use the “trim” tool to open up the bottoms of the slots.



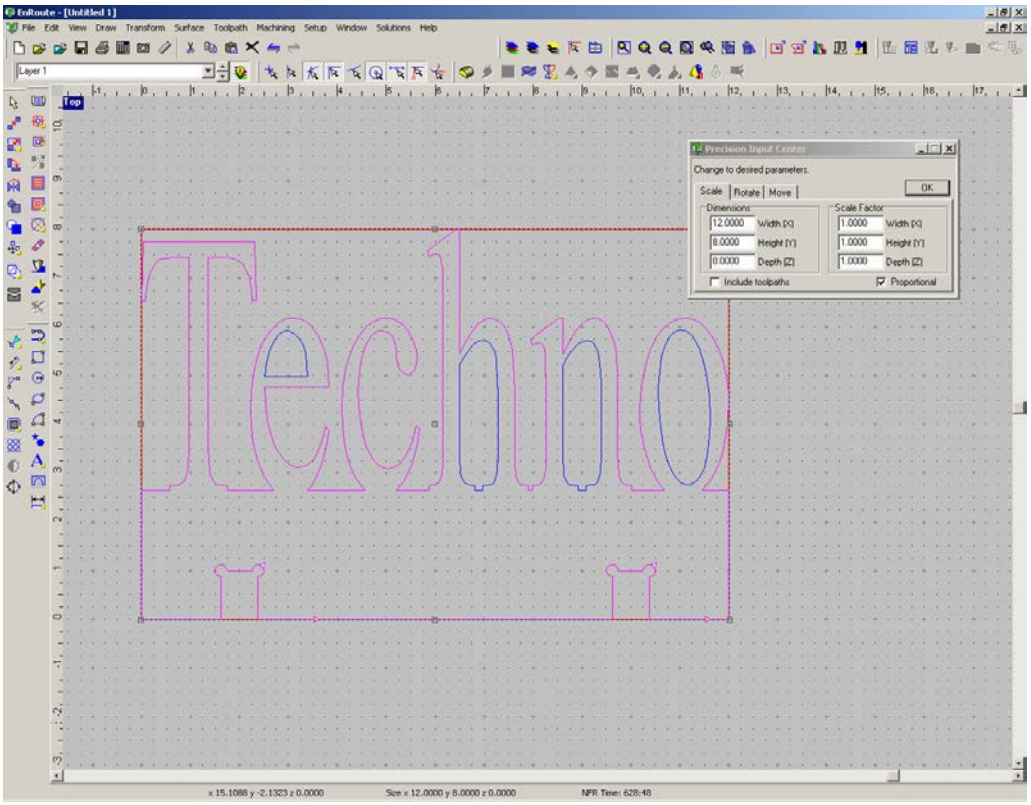
Select “by boundries” and “all boundries”.



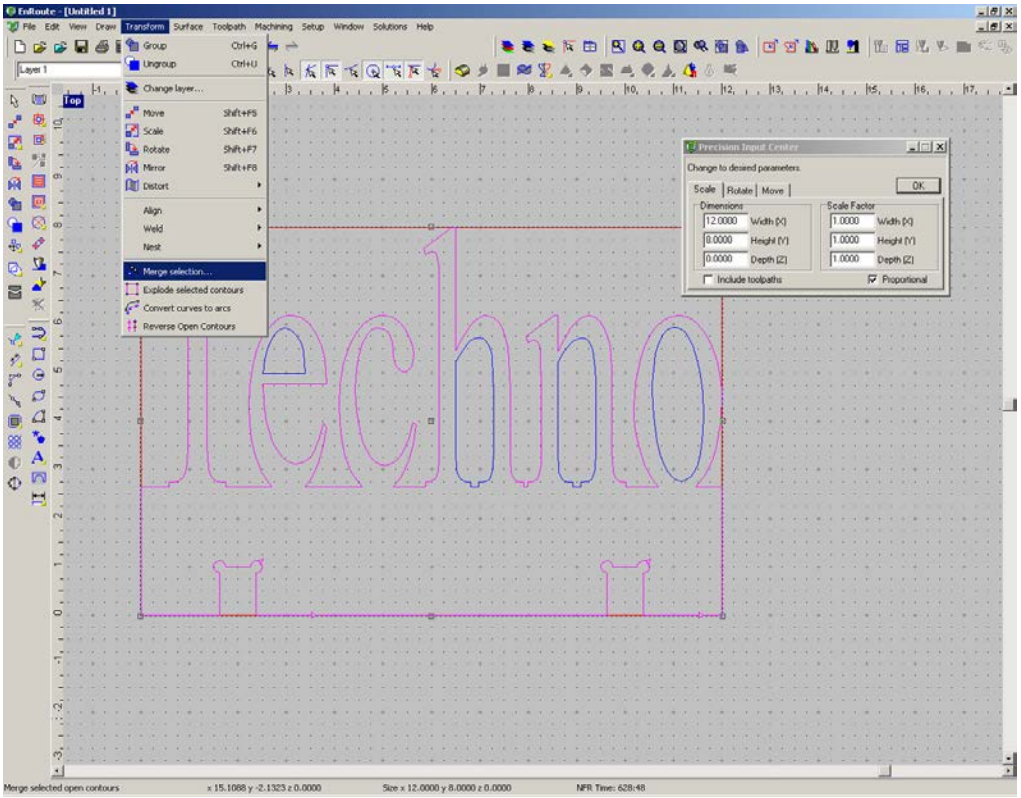
Then click on the segment to trim out. It will highlight a different color.



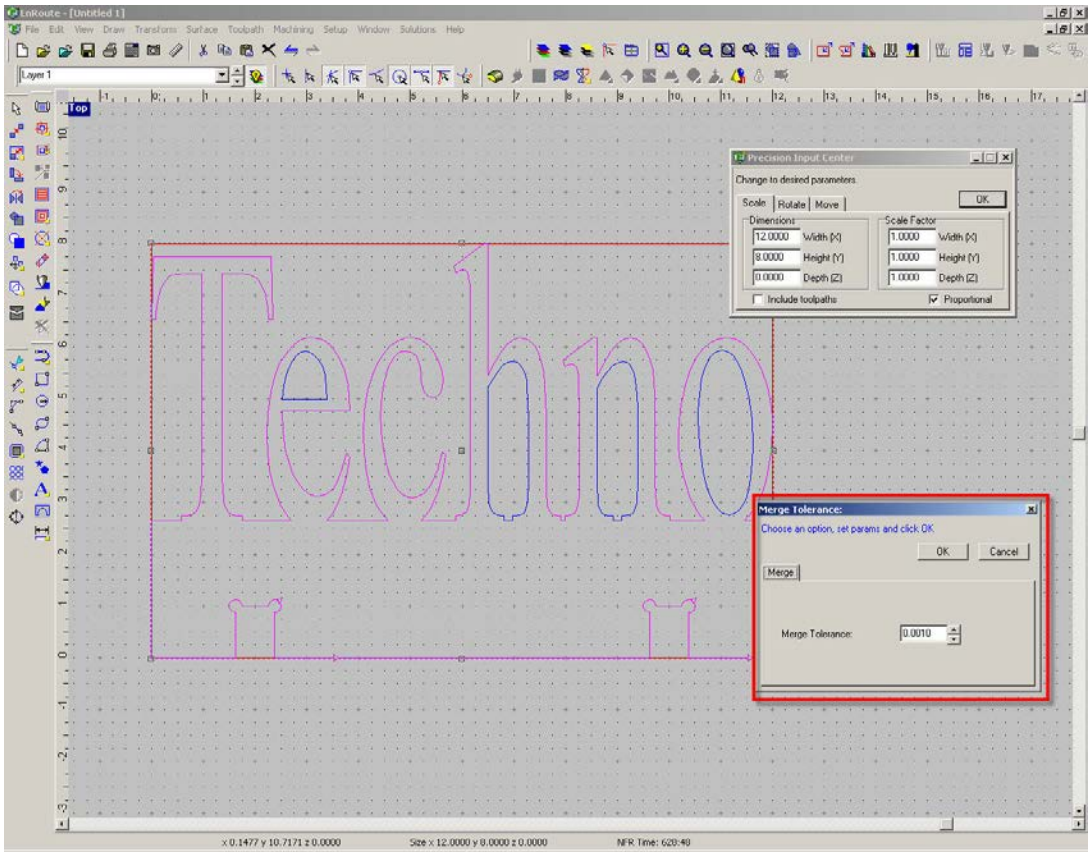
Repeat for the other side as well.



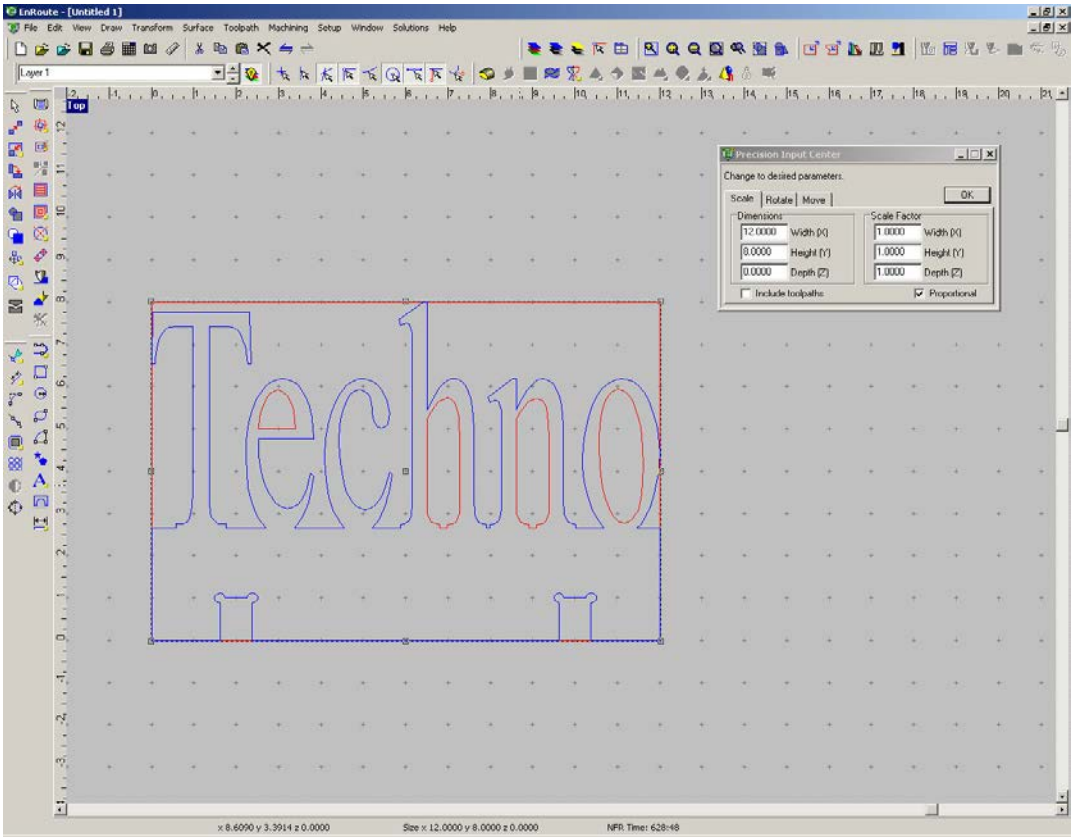
If you highlight the vectors you will notice that some of them are pink. That indicates that they are open contours and need to be merged together before you can apply a tool path.



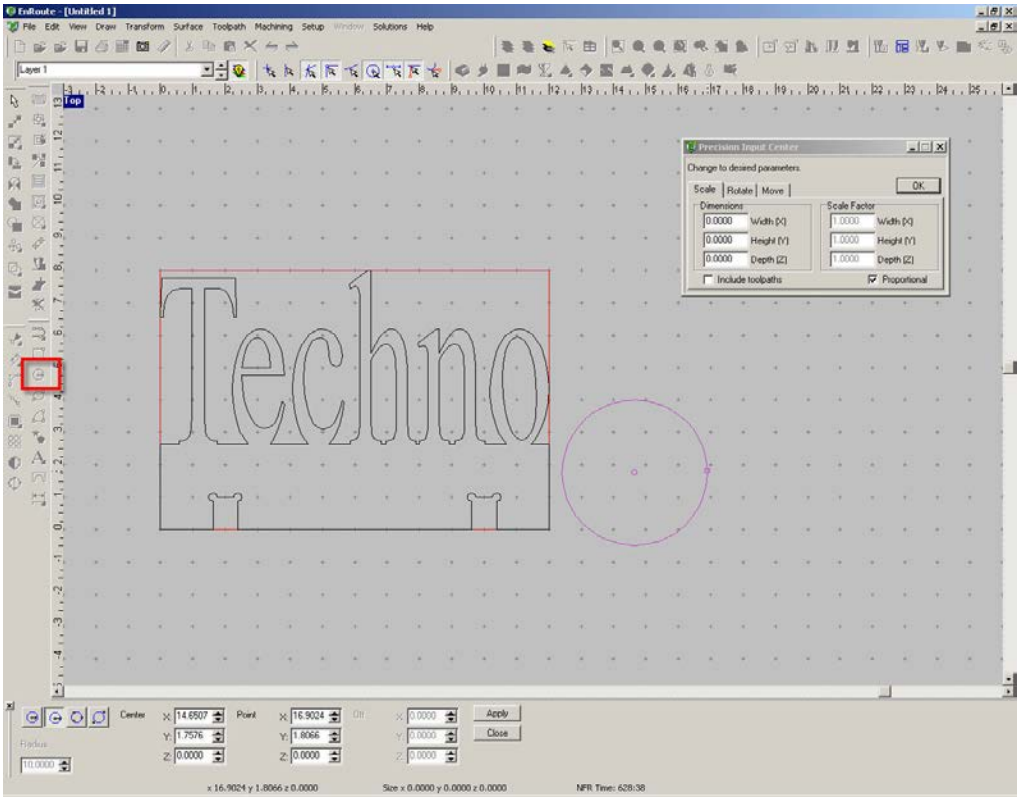
Go to “Transform > Merge Selection”.



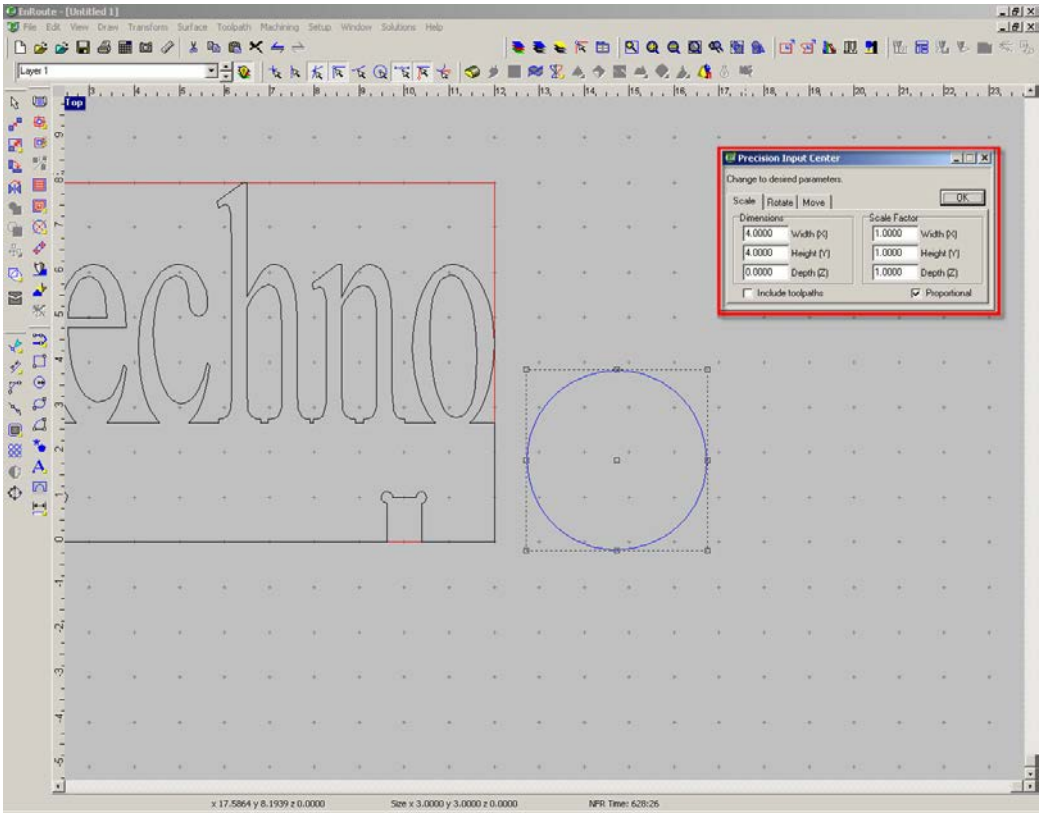
A merge tolerance window will pop up. Click “OK”.



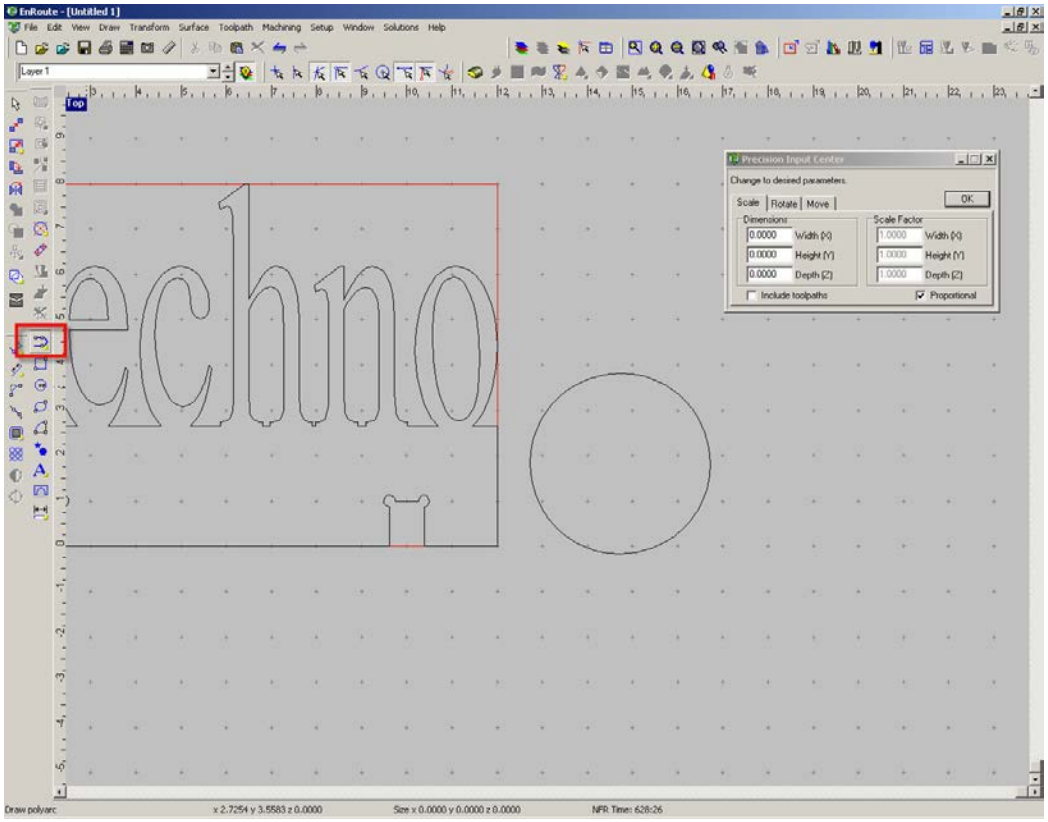
The re-merged outline.



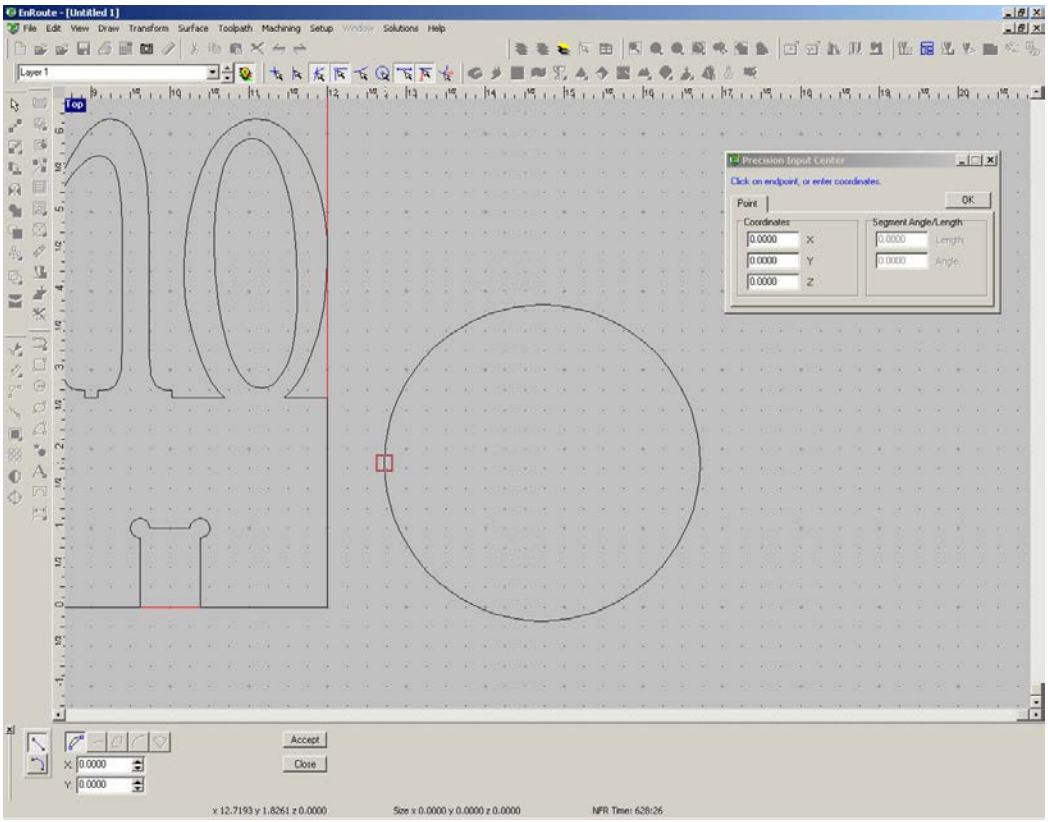
Now we need to make the feet. It is OK to work outside of our plate – it is only there for visualization. Create a circle.



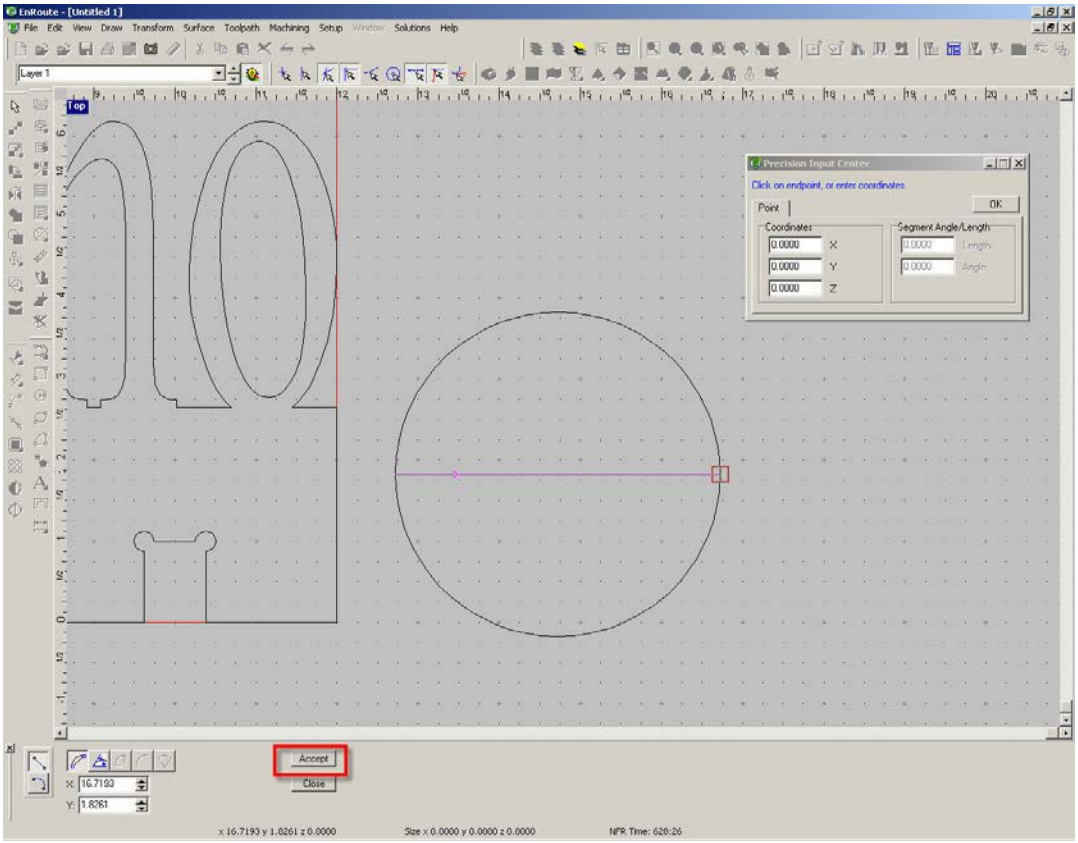
Resize the circle to a 4” diameter, using the PIC.

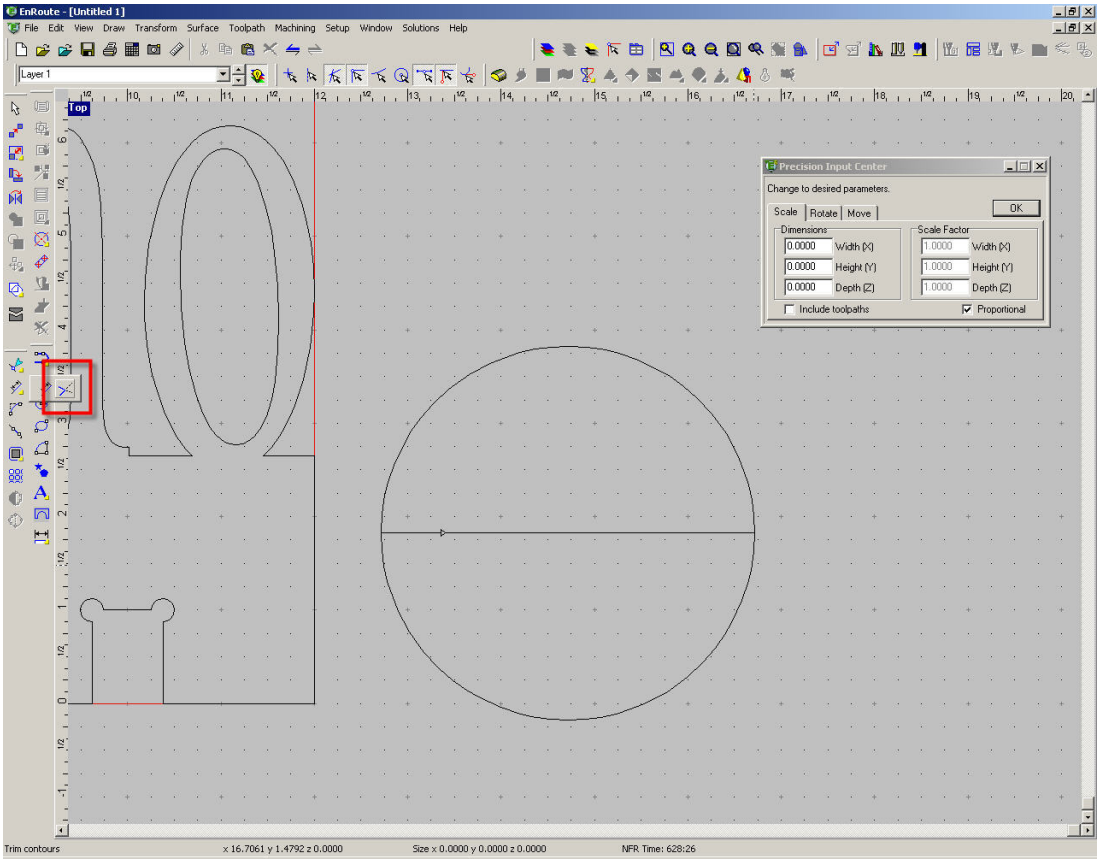


Select the “draw polyarc” tool to create a line.

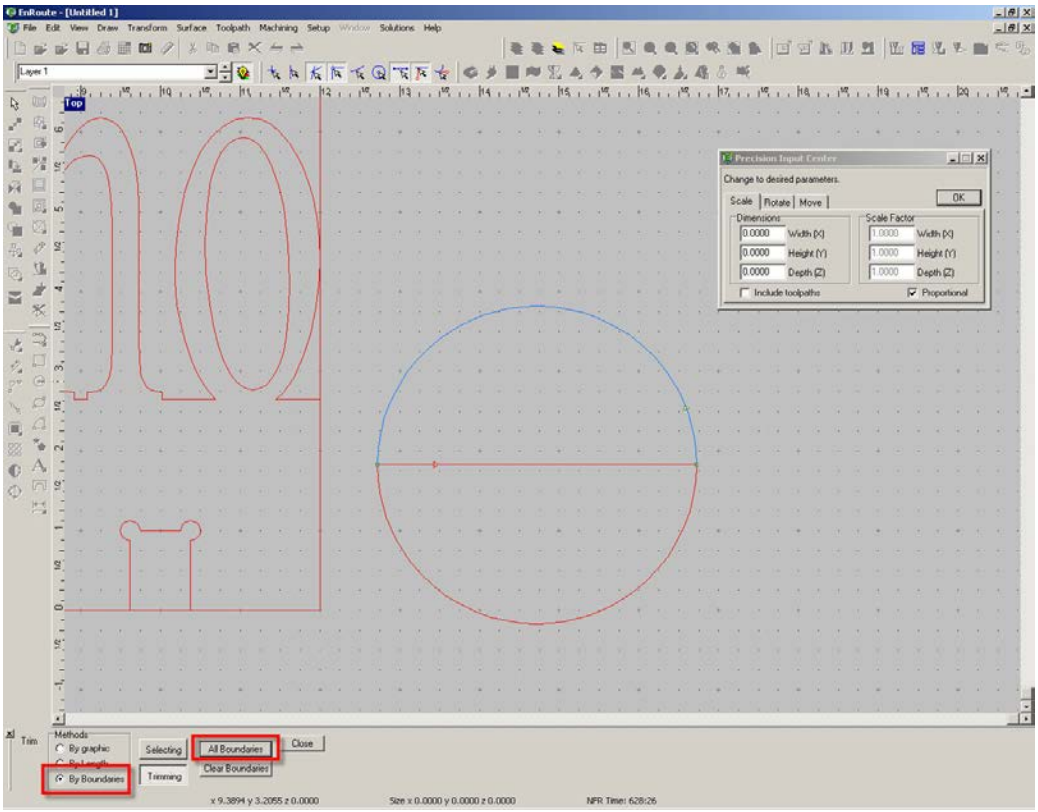


Using the snaps, draw a line bisecting the circle.

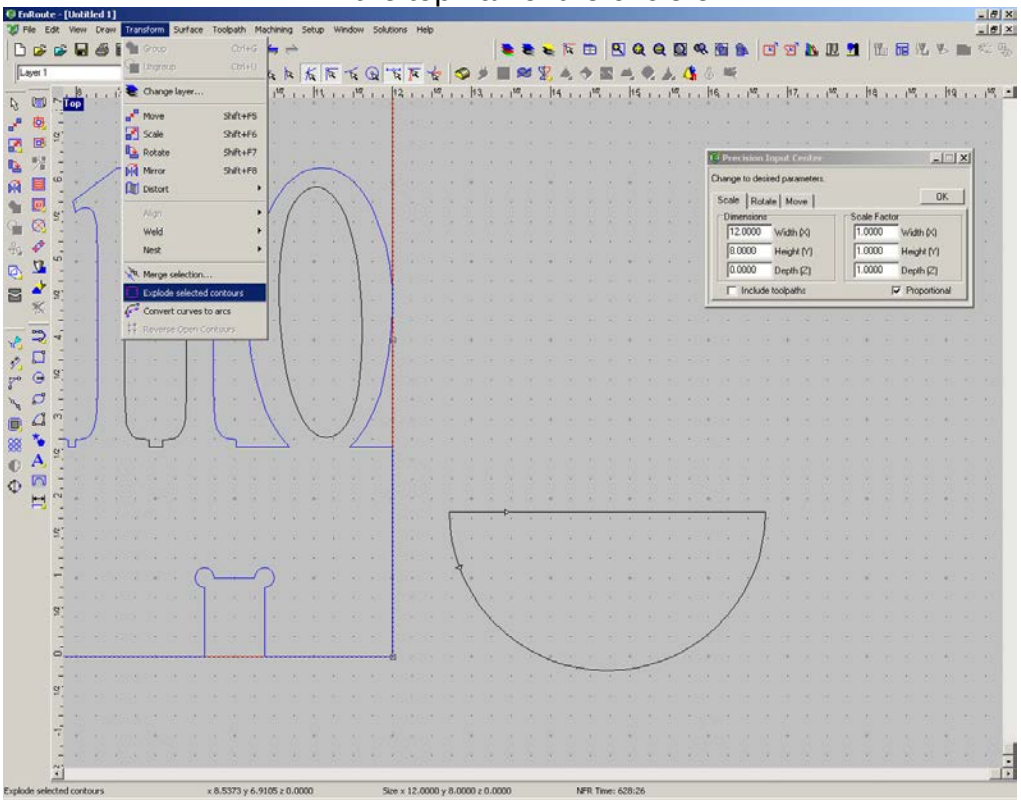




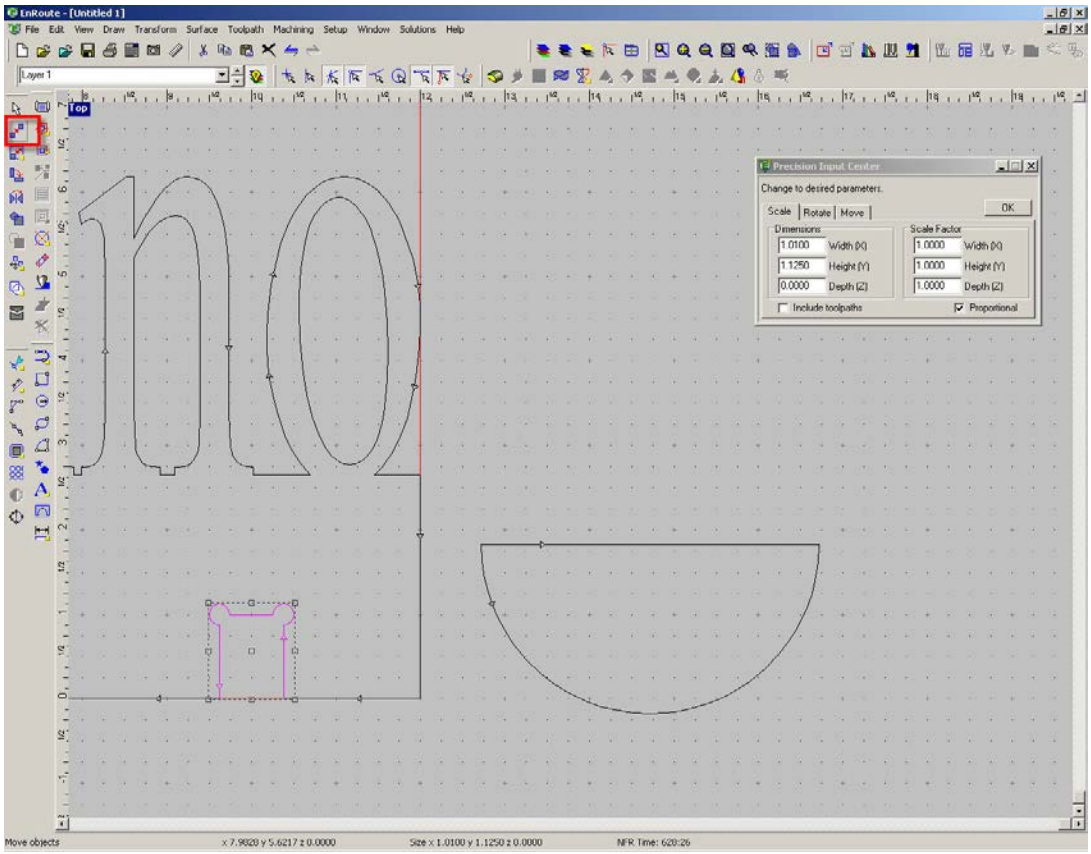
Now that the line is drawn, go to the “trim” tool.



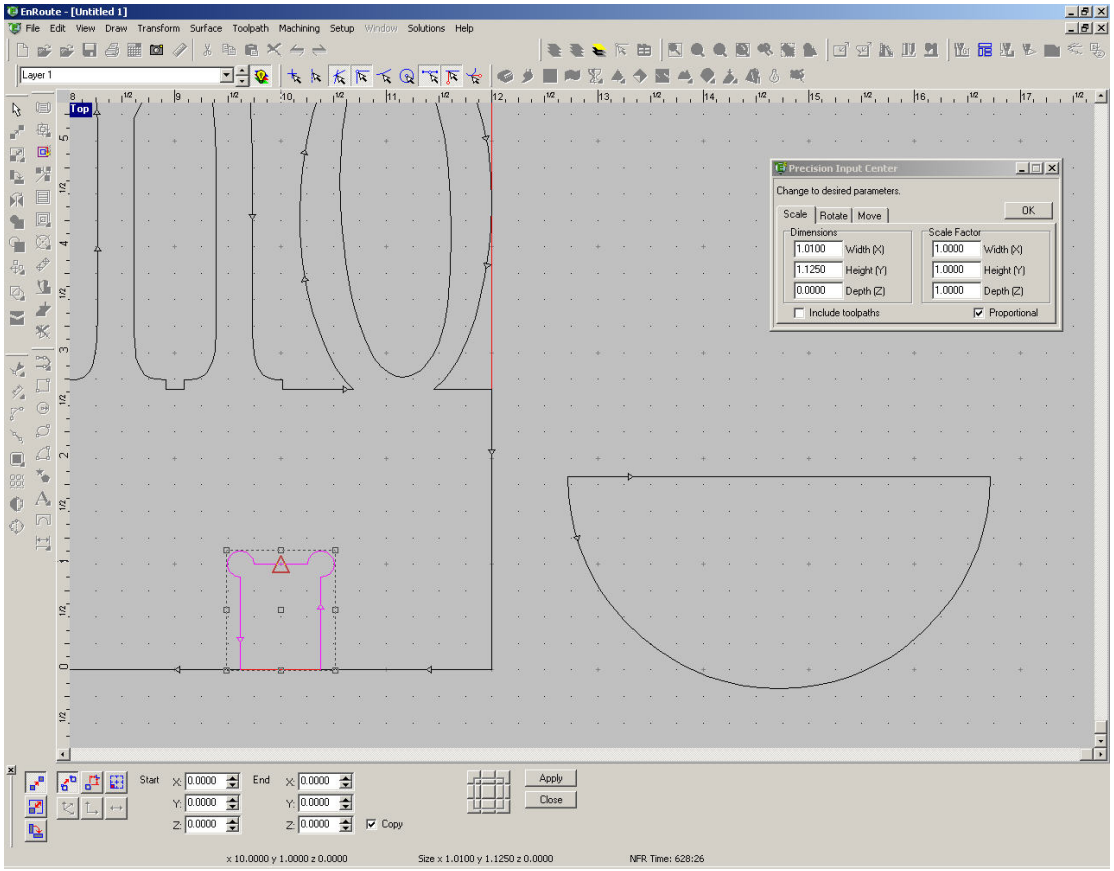
Trim the top half of the circle off.



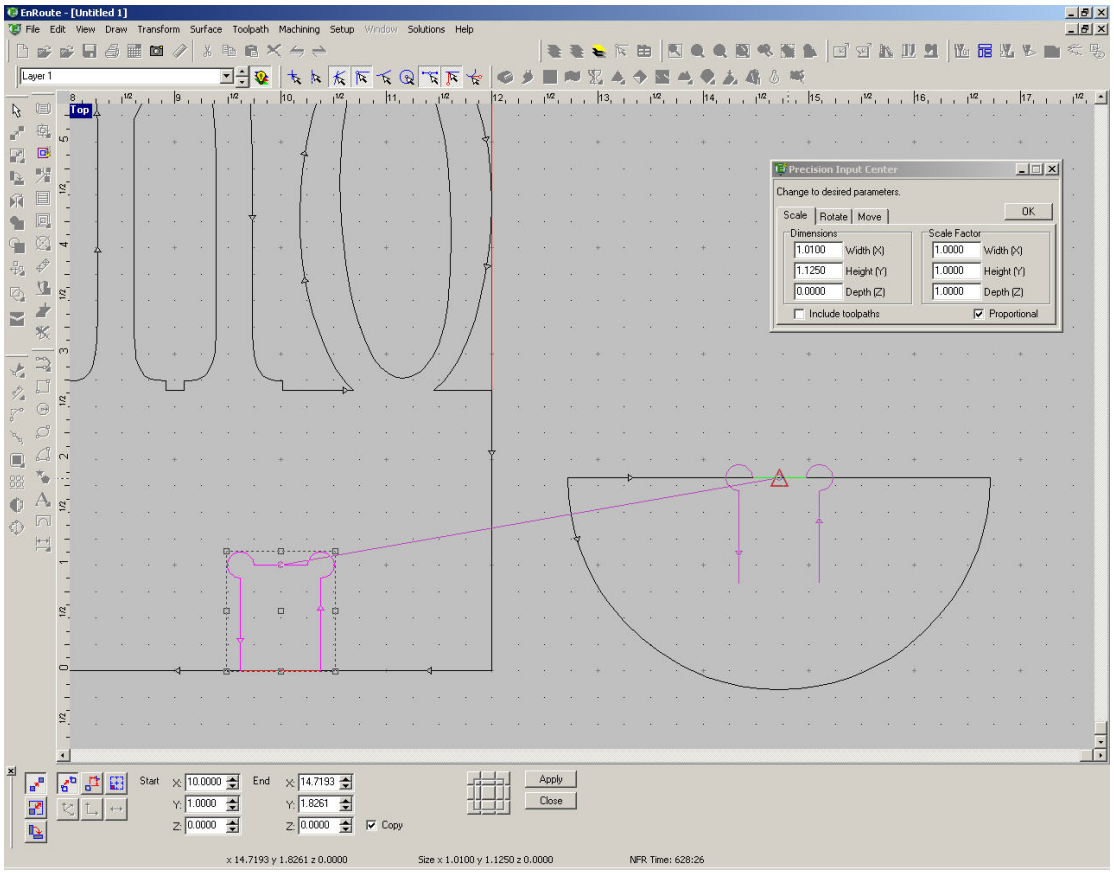
I'd like to use our slot from the nameplate to copy onto the new shape.
To do this, I can just explode the closed geometry back into segments.
Select the closed geometry, then go to "Transform > Explode Selected Contours".



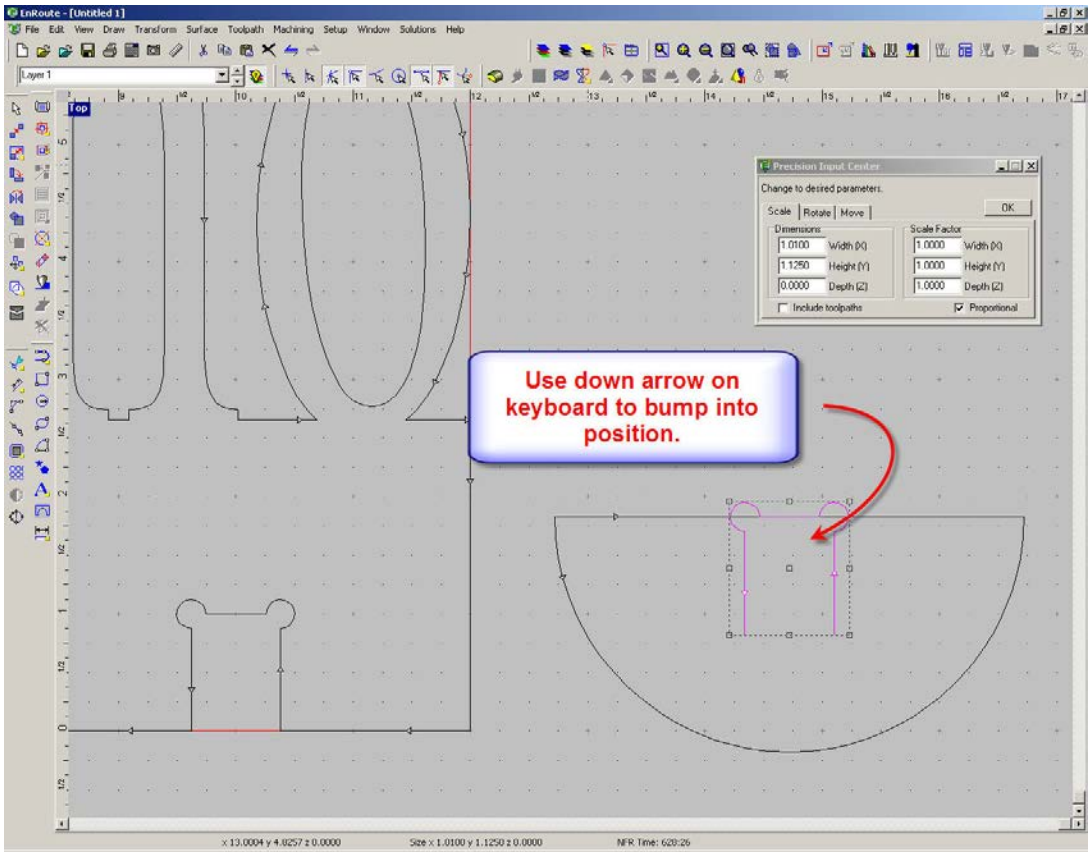
Now we can select the lines we want to copy and use the “move objects” tool.



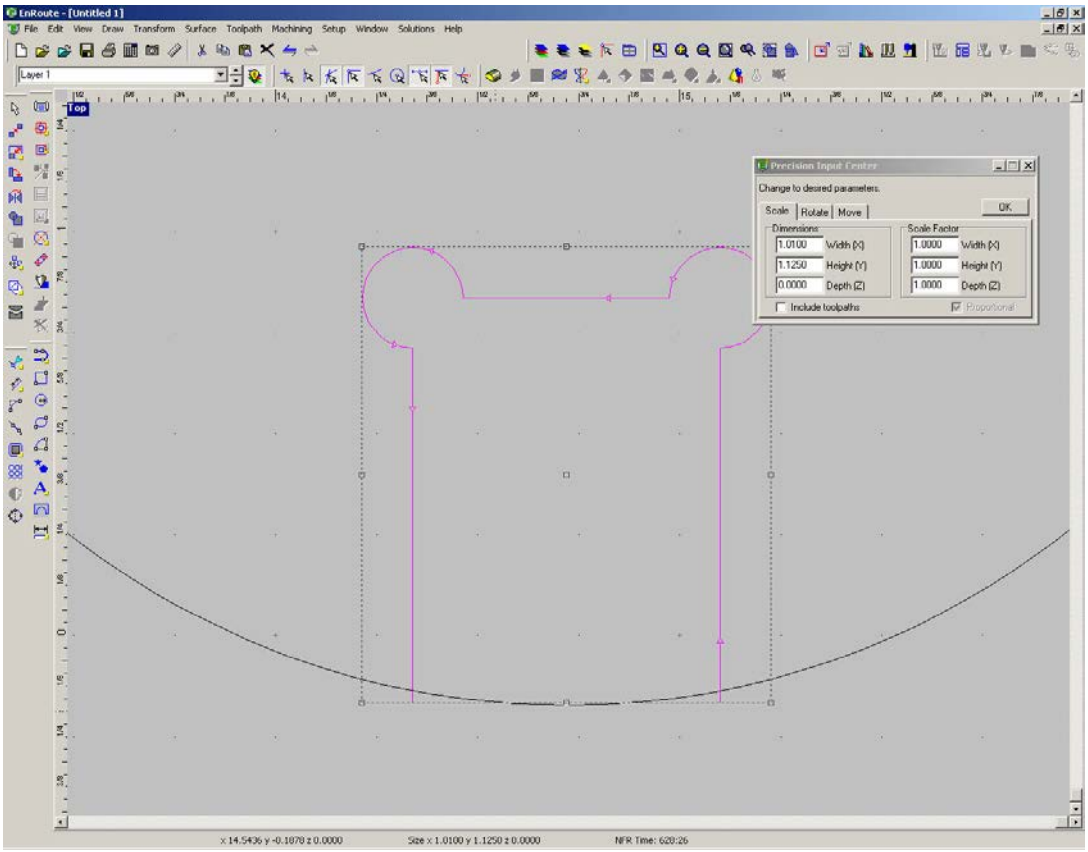
Copy from the center-point of the slot…



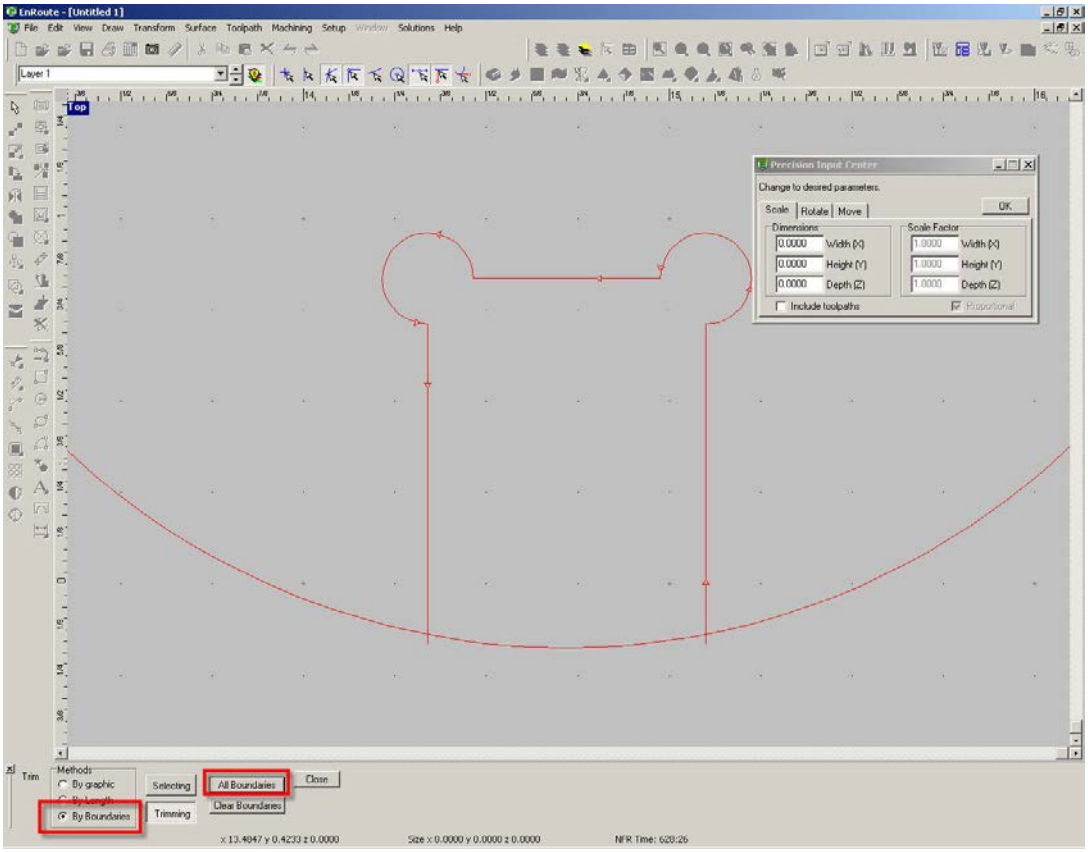
…and copy to the center-point of the foot. Don't worry about the Y position – we will move it.



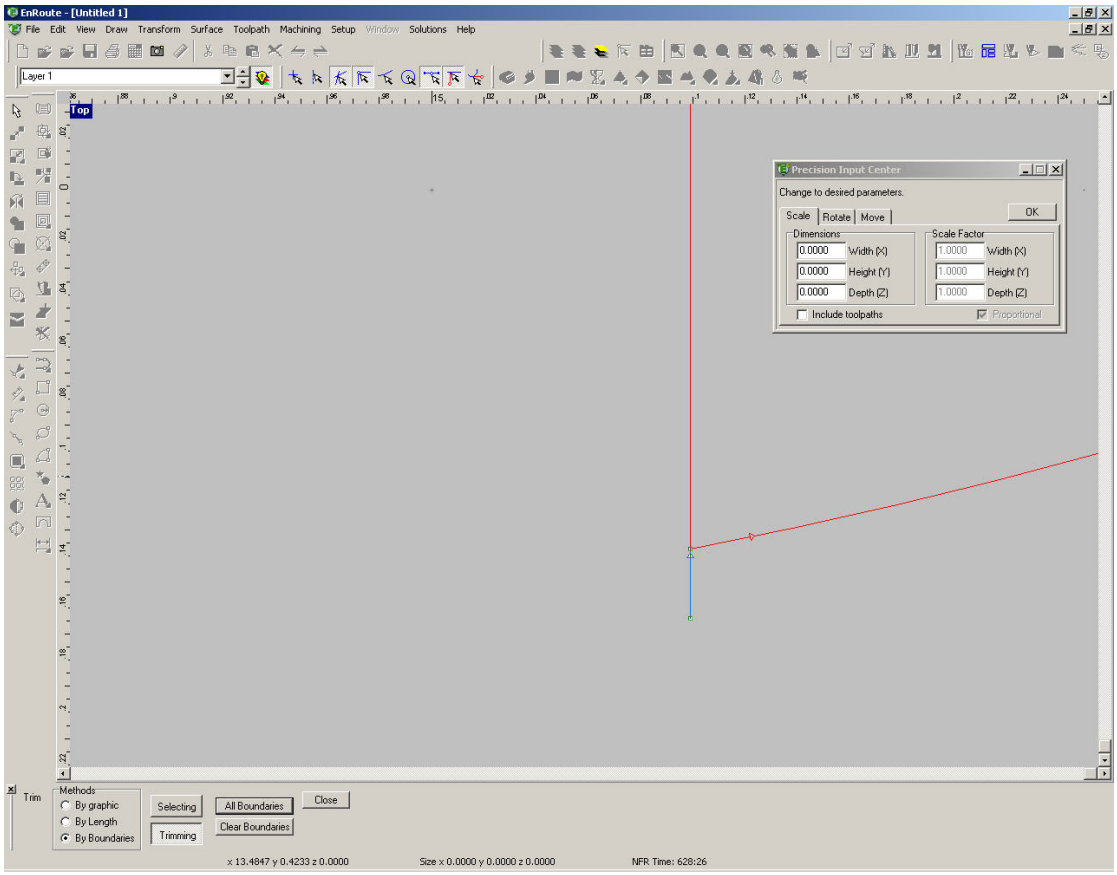
Use the down arrow on your keyboard to bump the selected lines down.

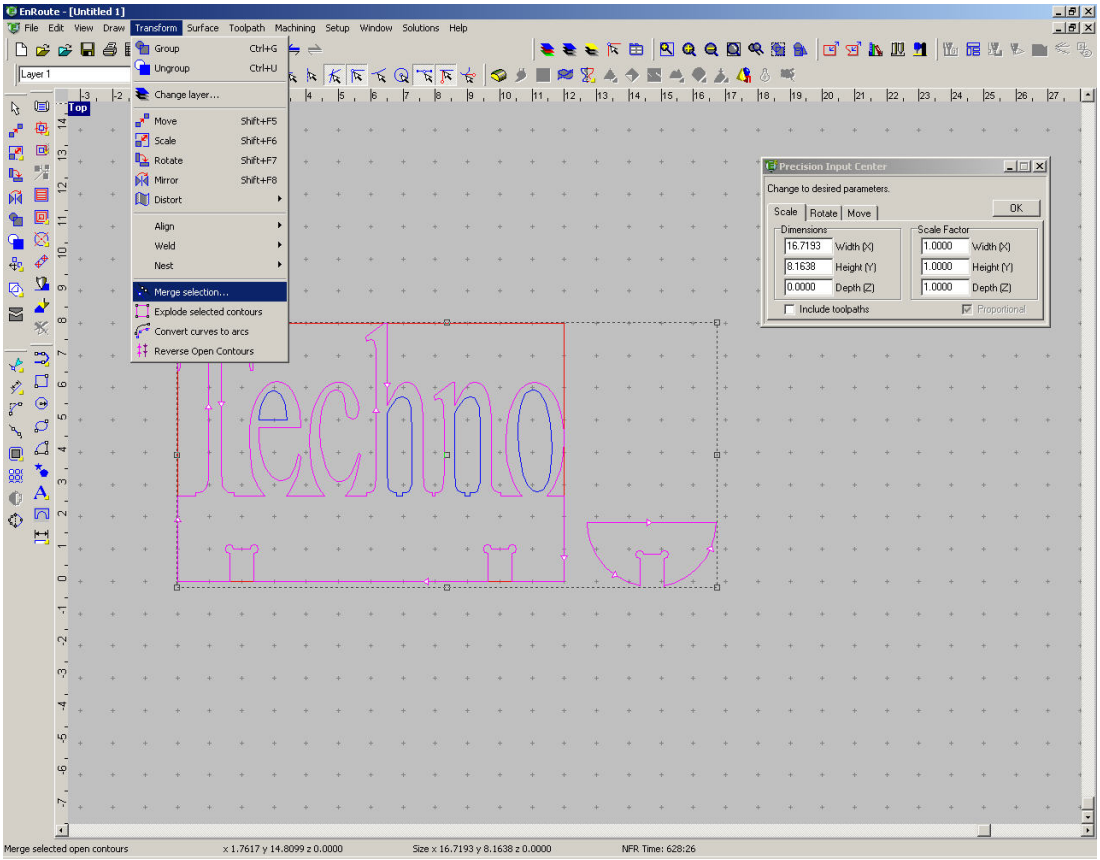


Move it until the lines slightly intersect.

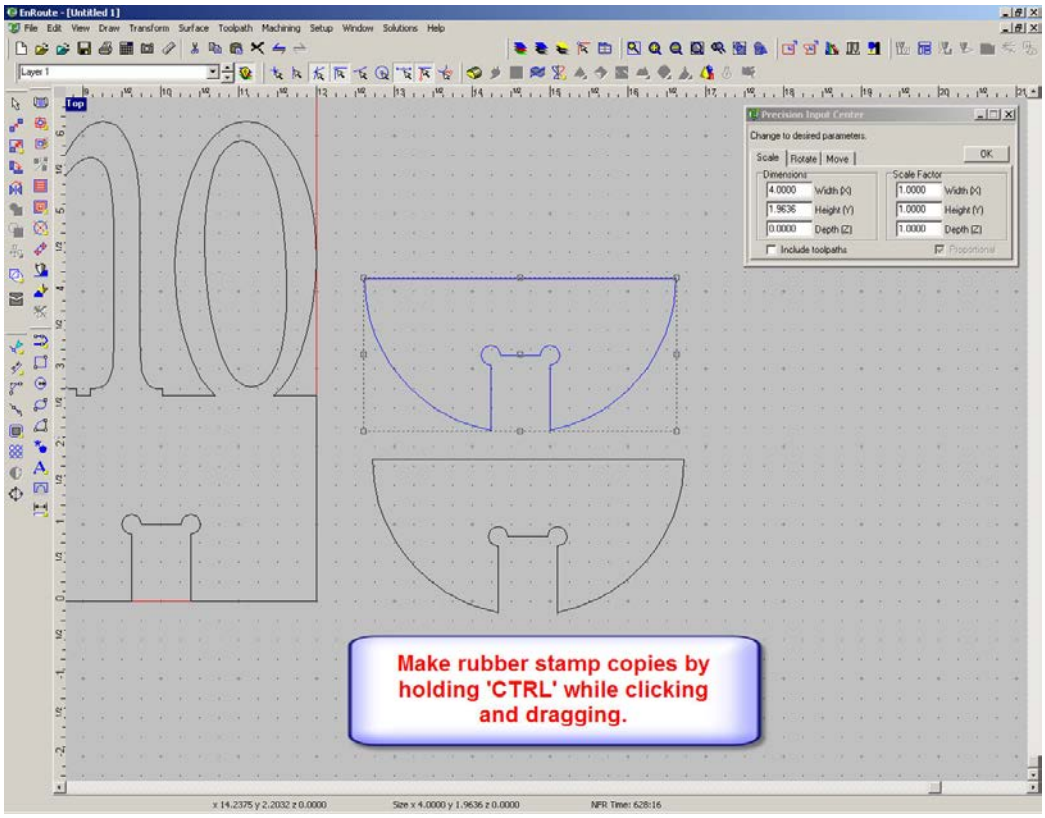


Go back to the “trim” tool and trim out the intersected lines.

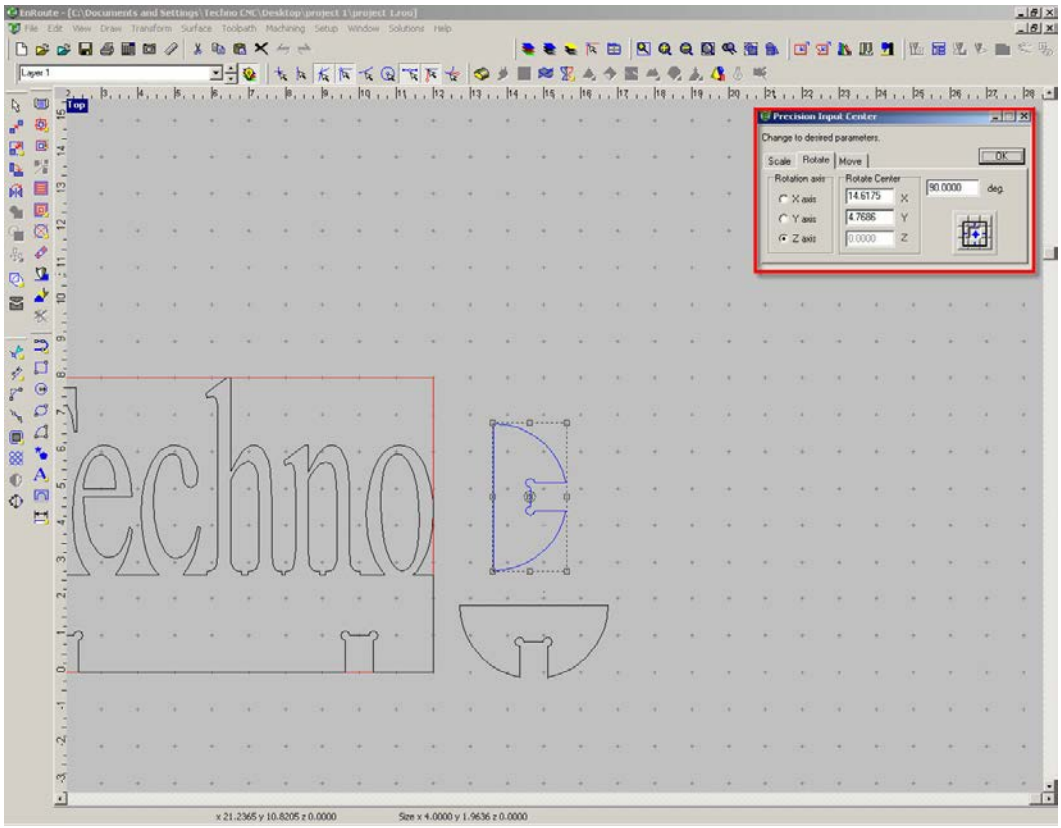




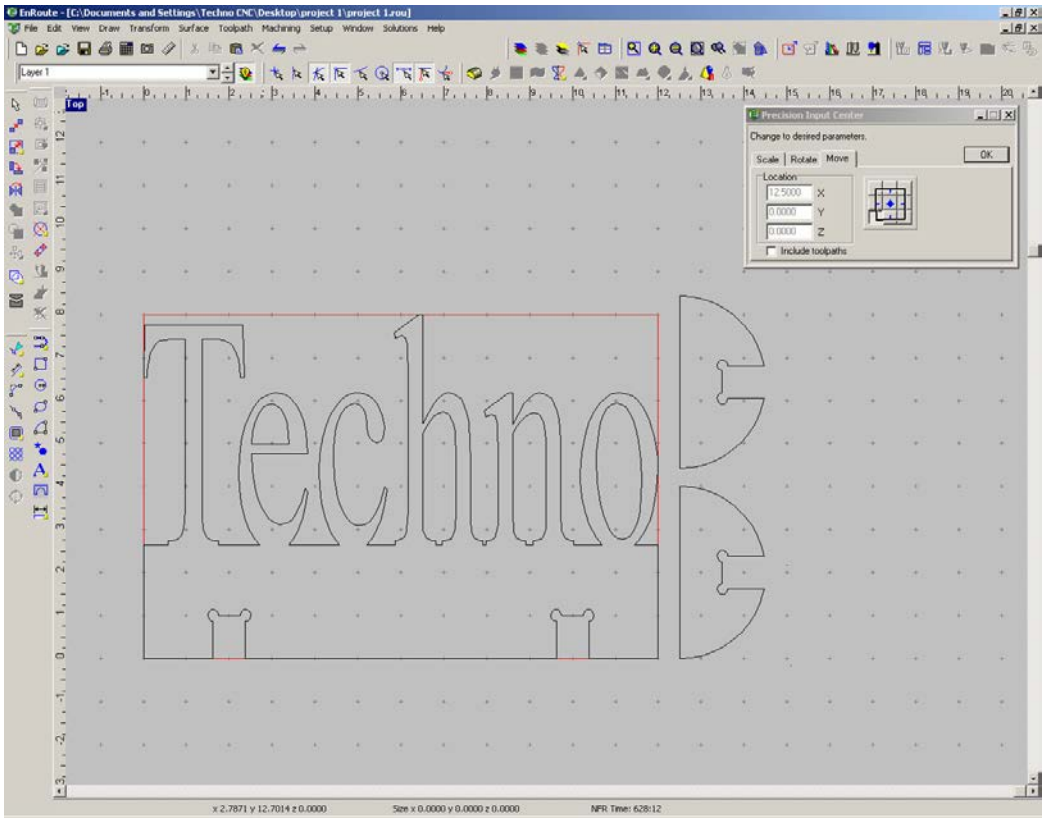
Now we can re-merge everything.



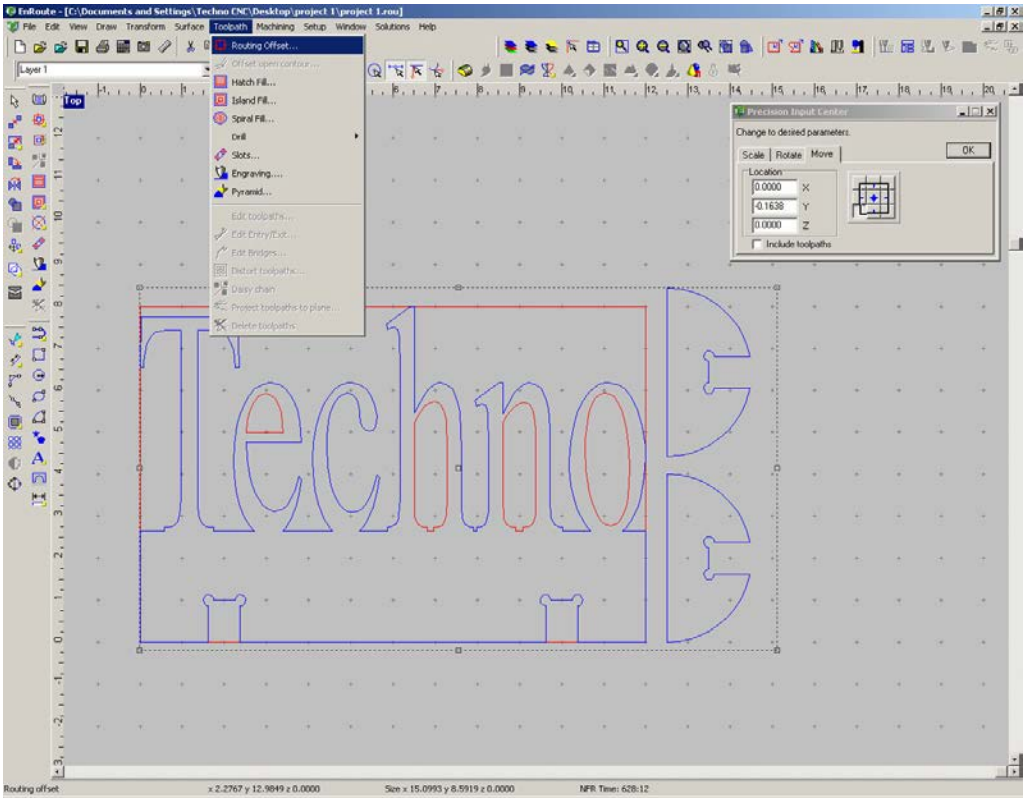
Make a copy of the foot by selecting it, holding “CTRL” on the keyboard and then clicking and dragging to a new location.



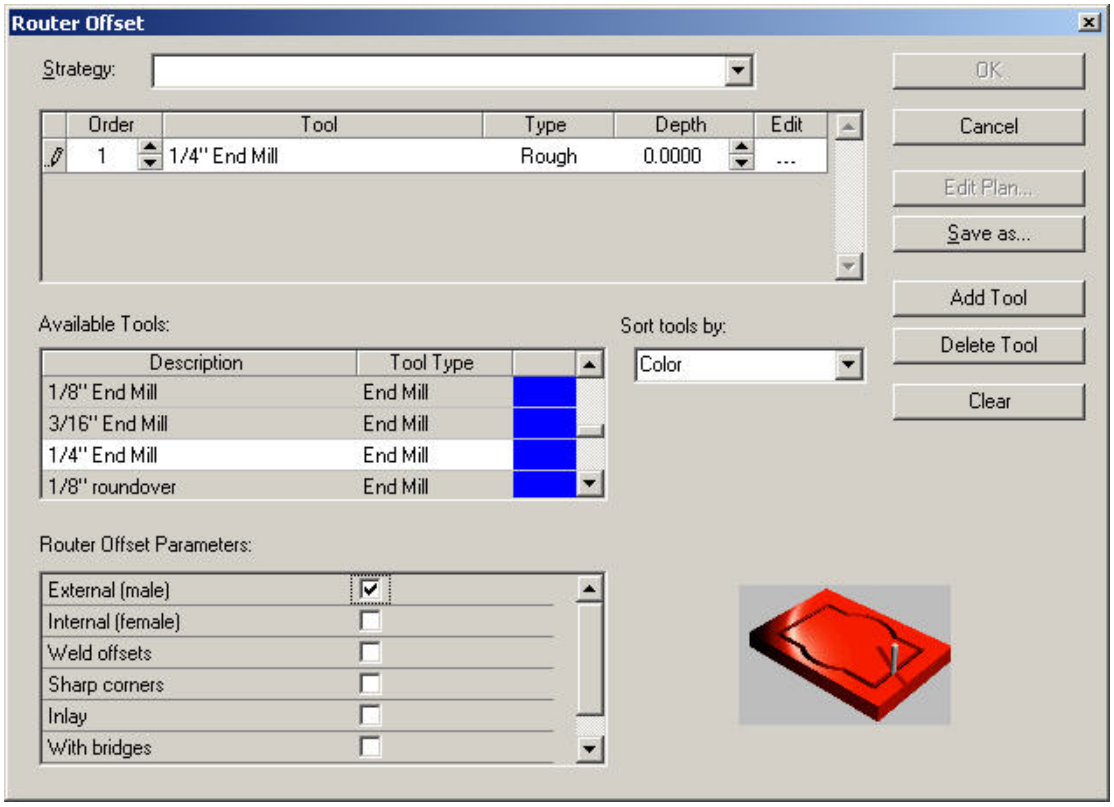
Now we can position the feet closer to the rest of the cuts so we don’t waste material.



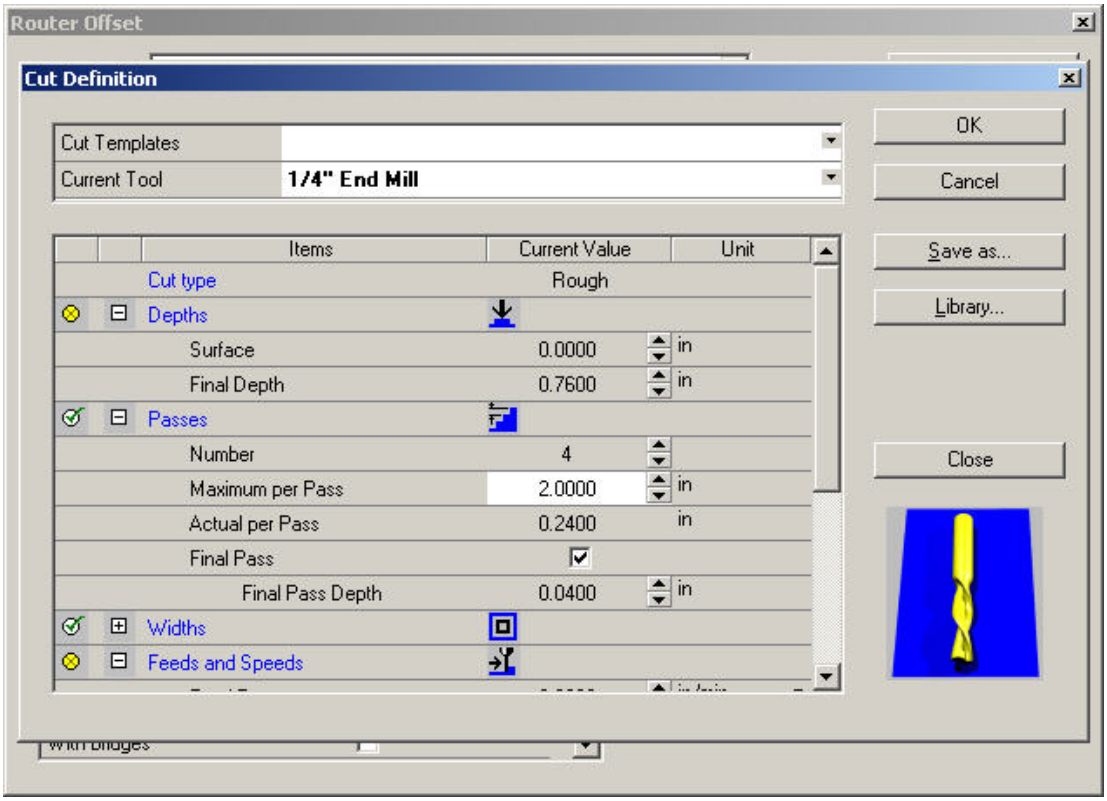
This is how it will look when it cuts.



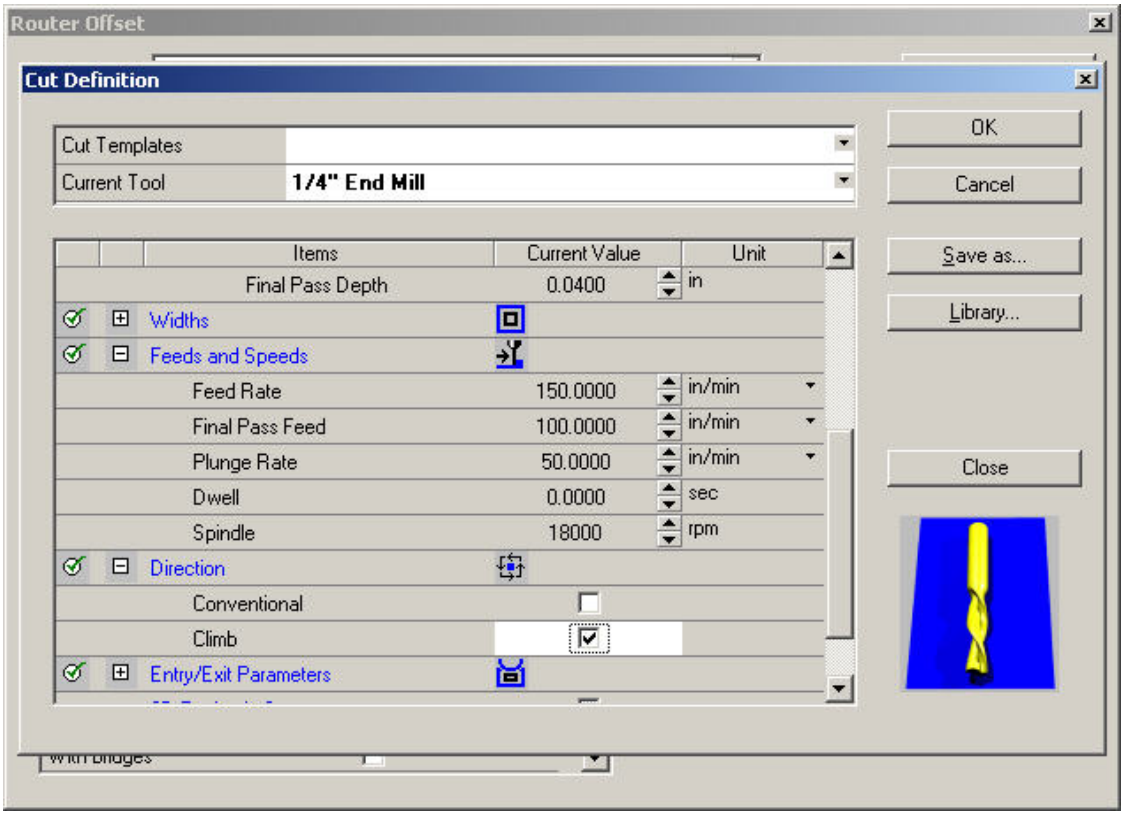
Time to add a tool path. Select everything and go to “Toolpath > Routing Offset”. This is the tool path that we want because it is a cutout.



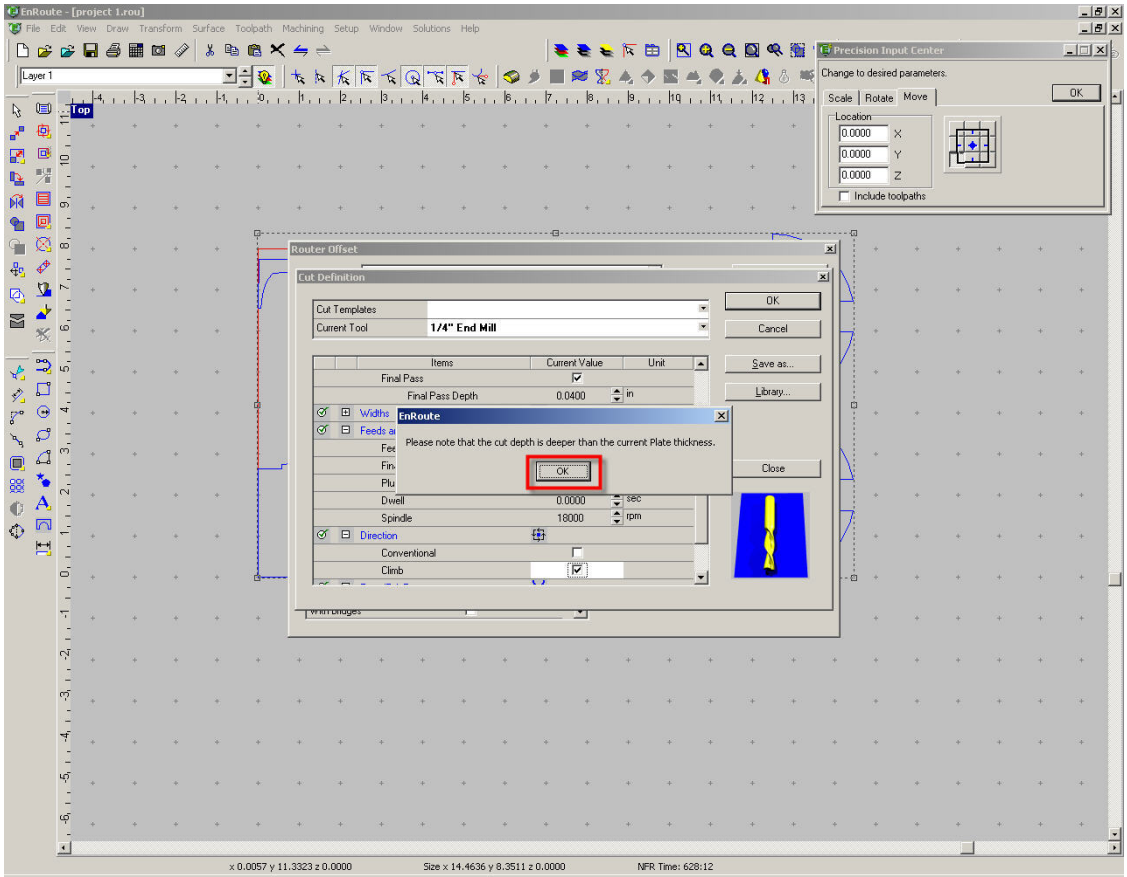
Select a ¼" endmill and then click on edit to add the cut parameters.



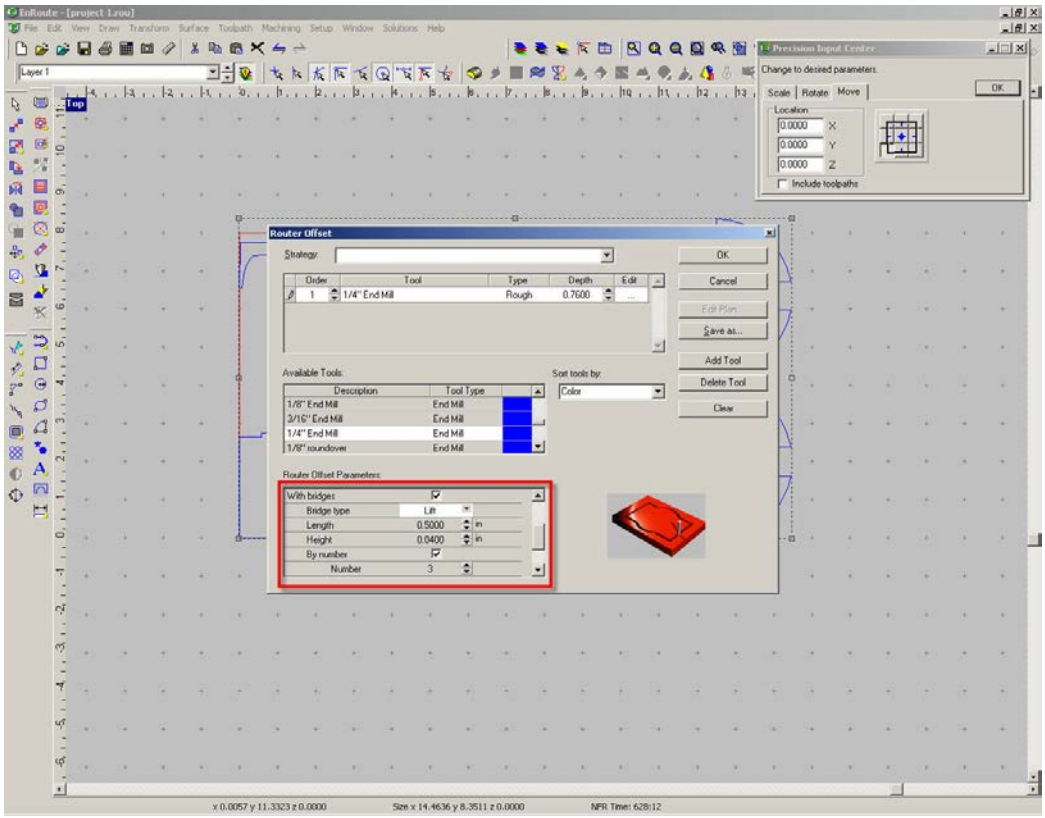
Note that the final depth of cut is a little deeper than the material thickness. This is to ensure that we cut all the way through. I'm also adding a "final pass" this allows the pass that actually cuts through the material is gentler and helps the objects stay in place.



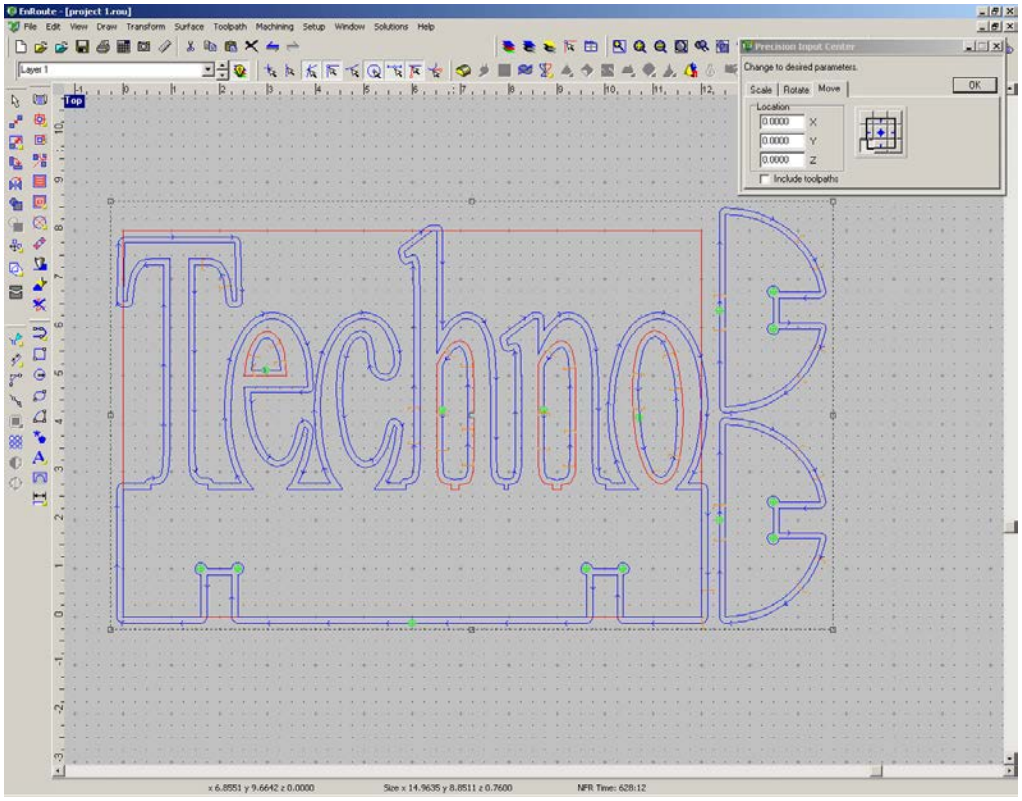
Enter the feed rates. The final pass feed rate is a little slower.



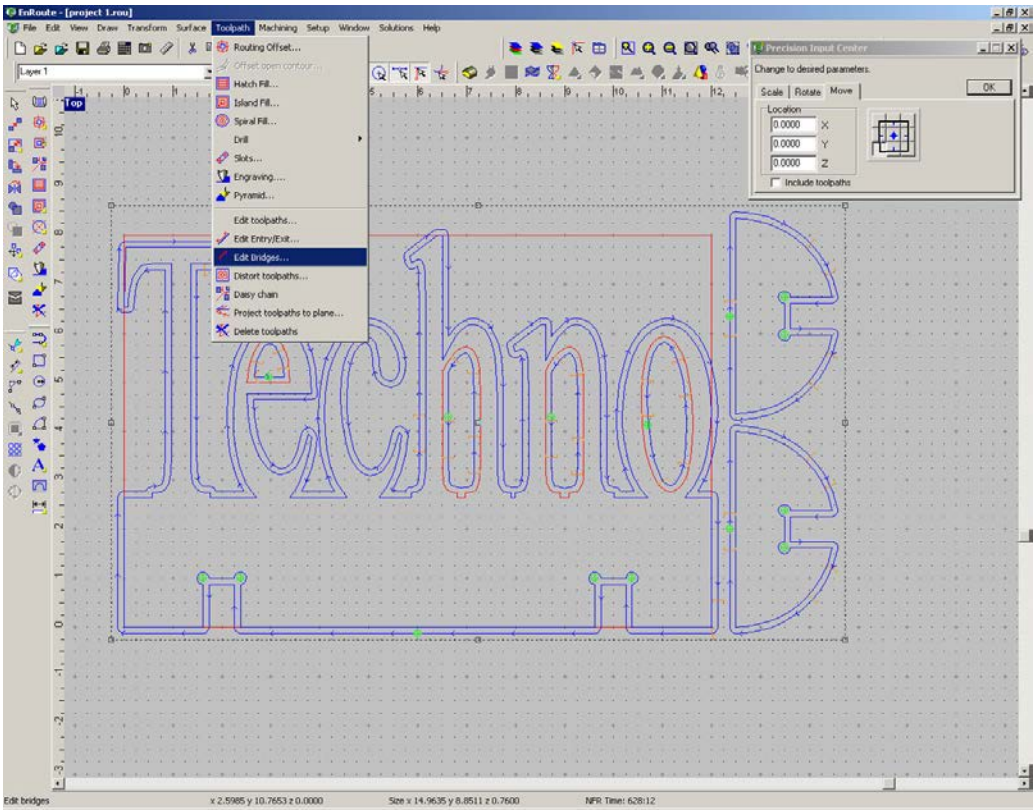
Click “OK”. You will get a message saying that the cut is deeper than the plate thickness. Click “OK”.



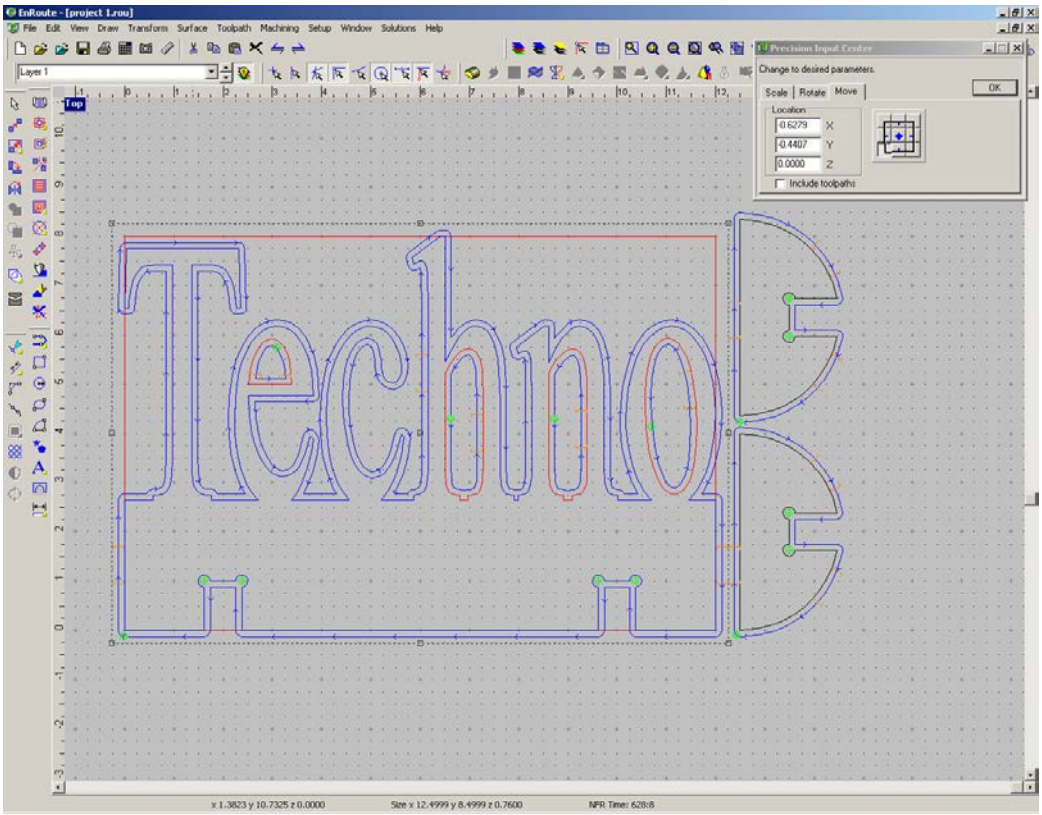
We can set up some bridges to hold the objects in the stock material. This is helpful with small parts or if you don't have vacuum hold-down.



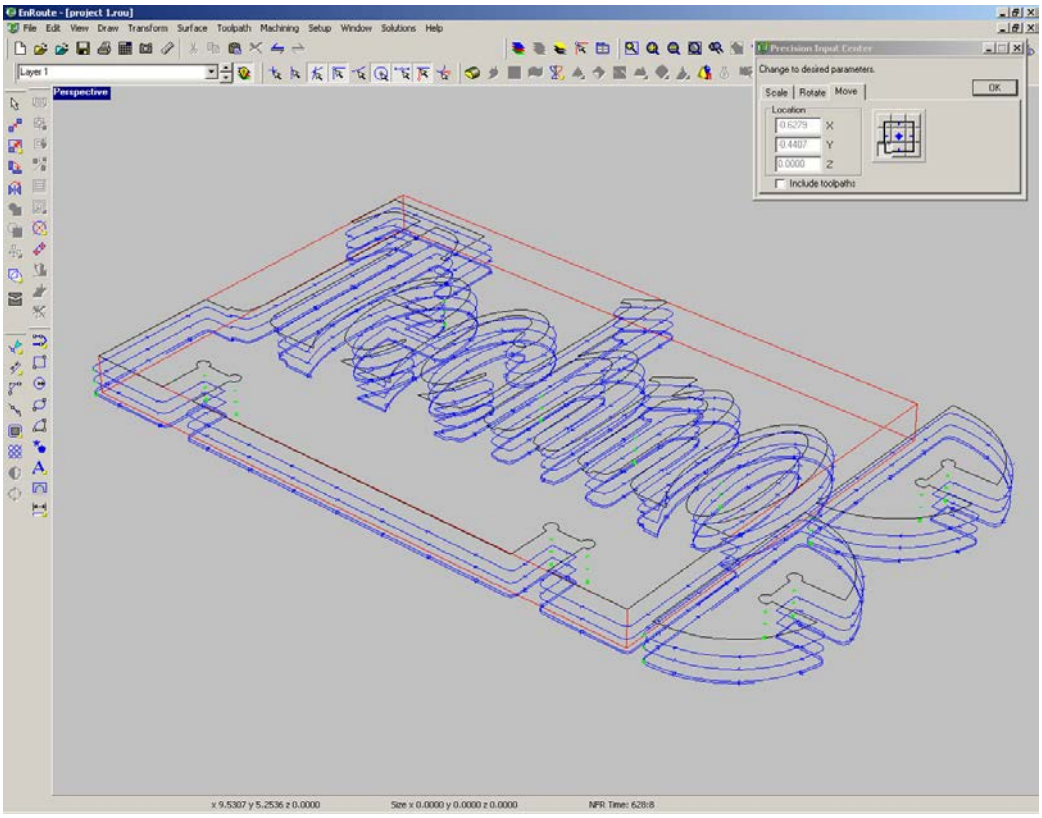
This is what the tool path looks like. I want to move the bridges a little bit.



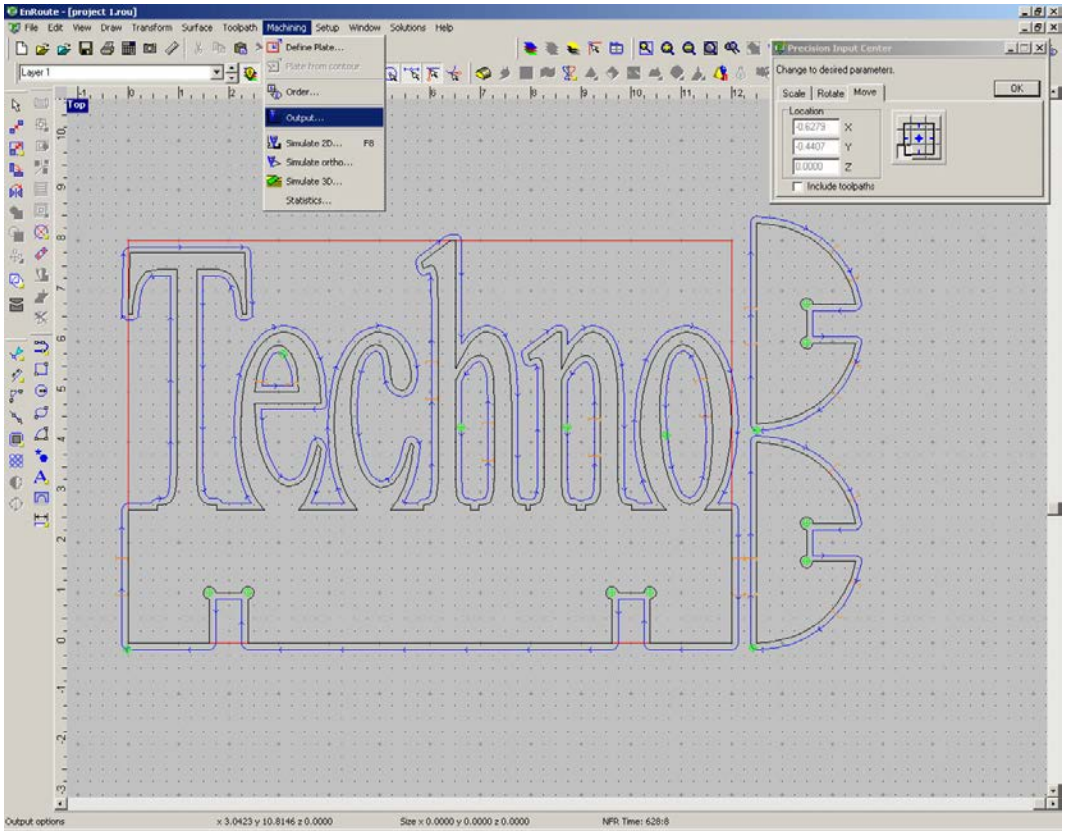
Go to “Toolpath > Edit Bridges” and then simply drag them to the desired location. You can right-click and delete any unwanted bridges or add more.



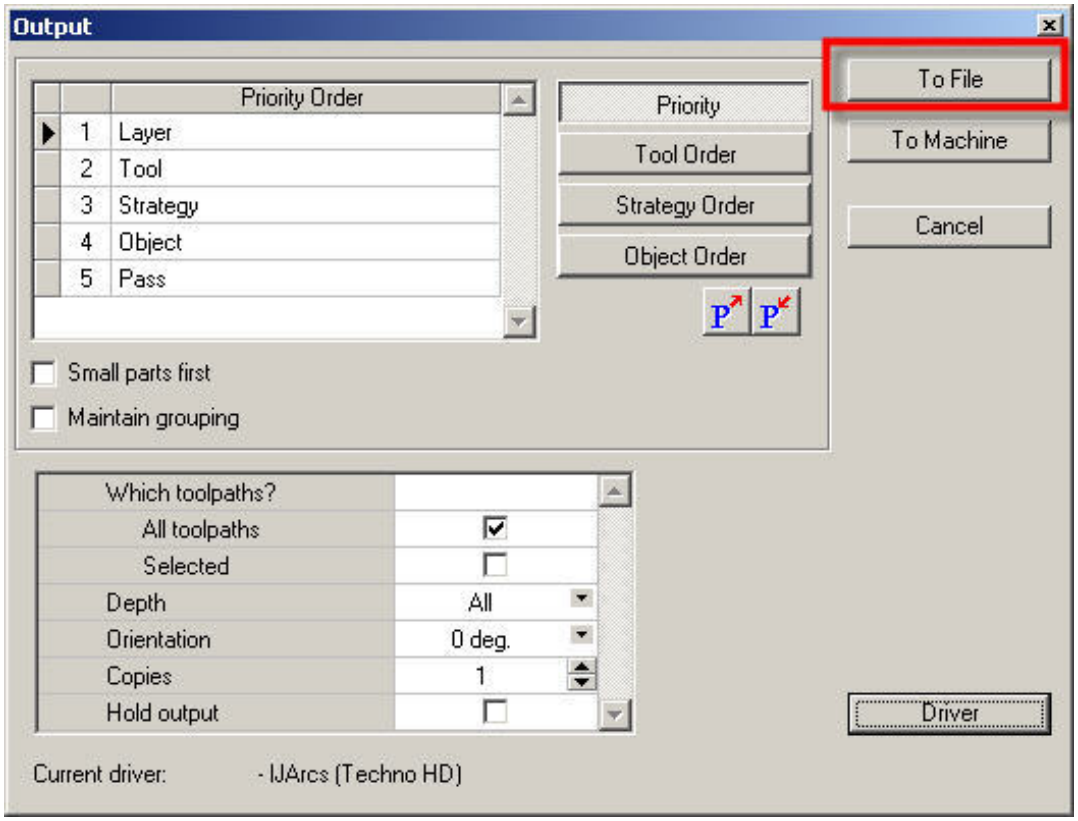
Here is the modified tool path.



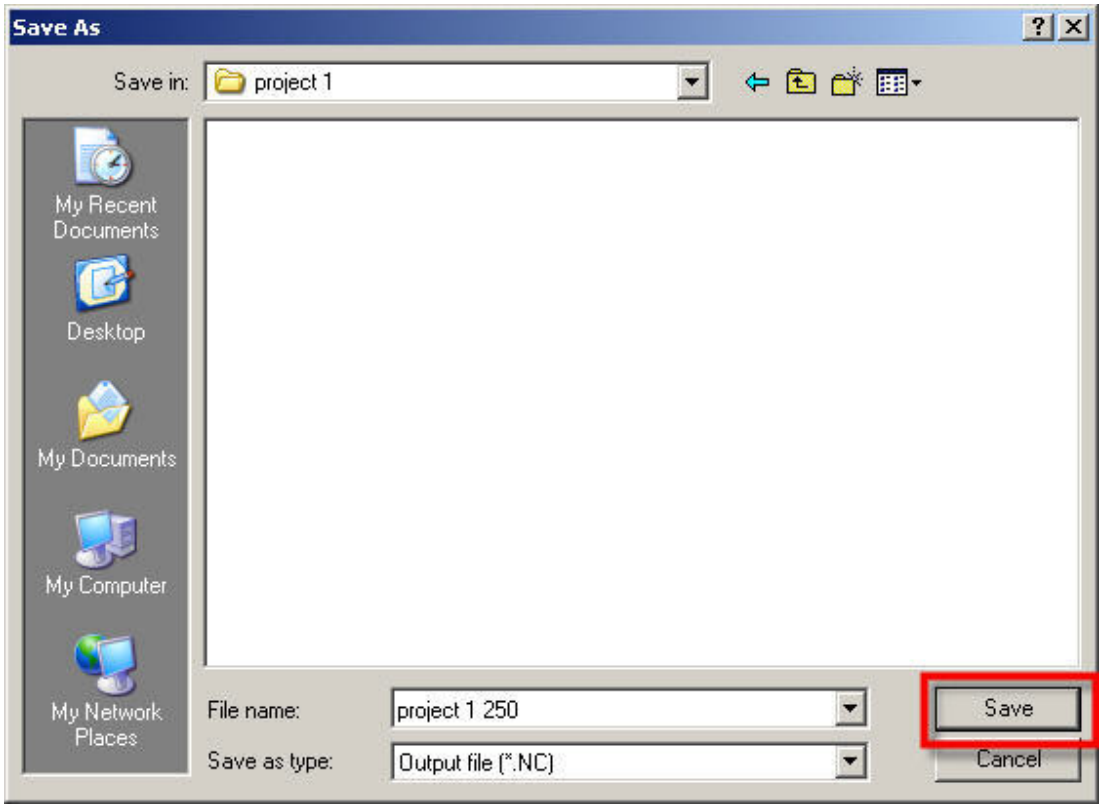
Perspective view to see the cut levels.



Now we can output this file and take it to the machine to cut!



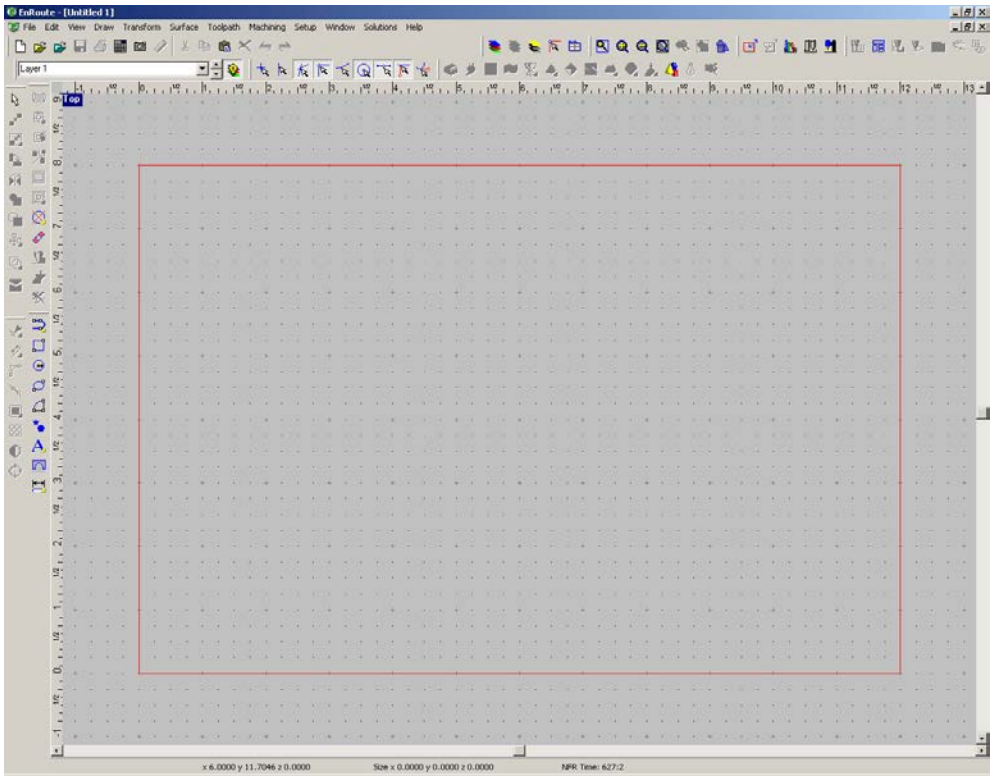
Click on “Machining > Output”. Then click “To File”.



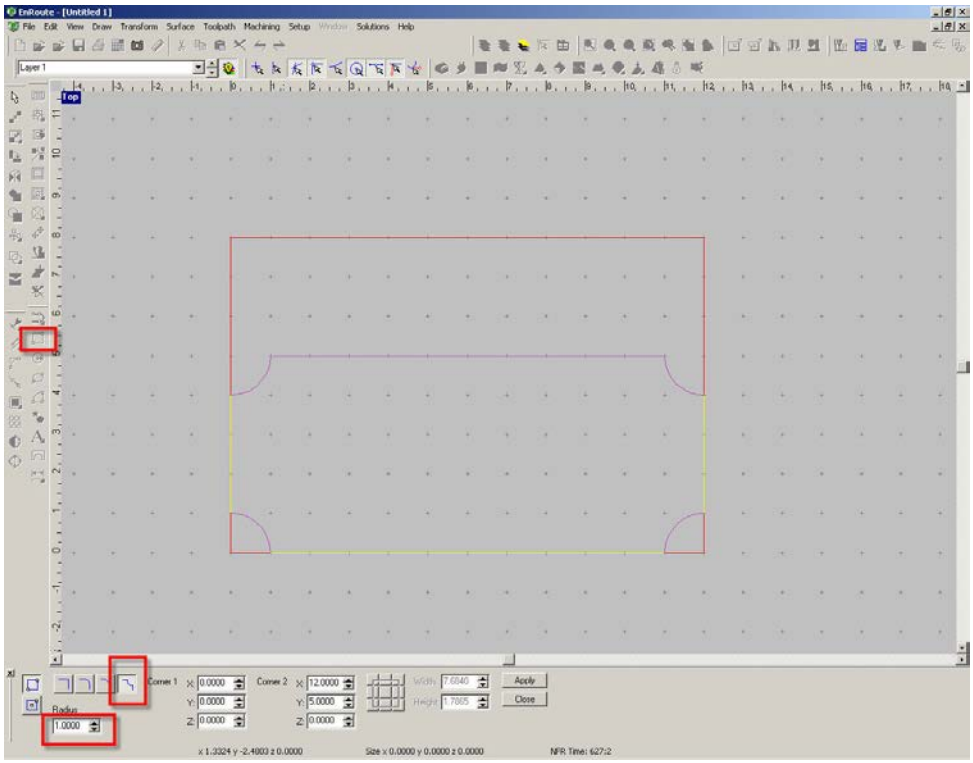
Name and save the file.
At this point, you can copy this file to a flash drive and take it to the machine.

EnRoute: Project 2

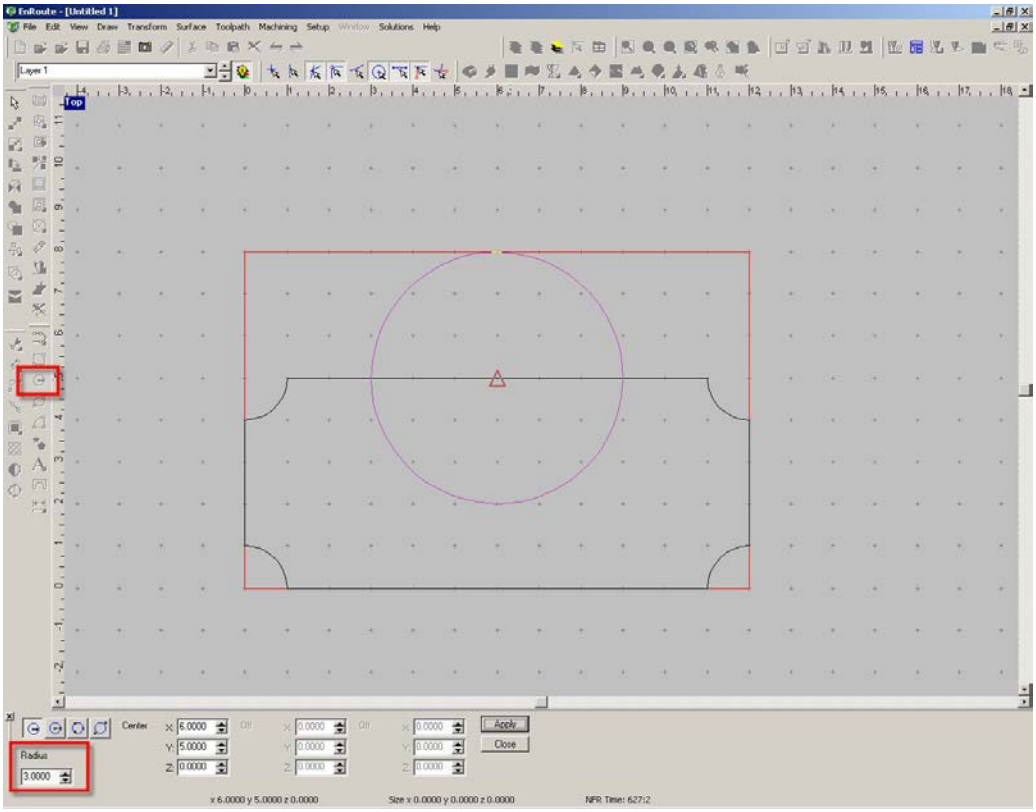
This section will cover a simple 3D engraving project.



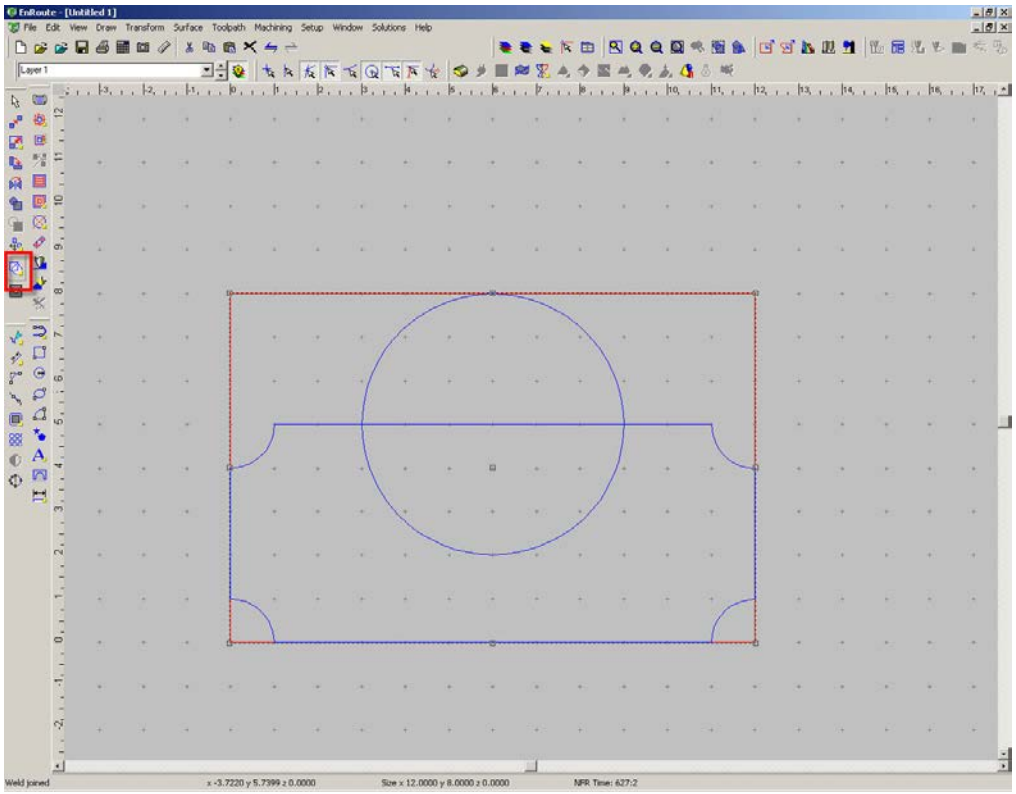
Let’s start with a new file. Set the plate size to 12” x 8” x .75”.



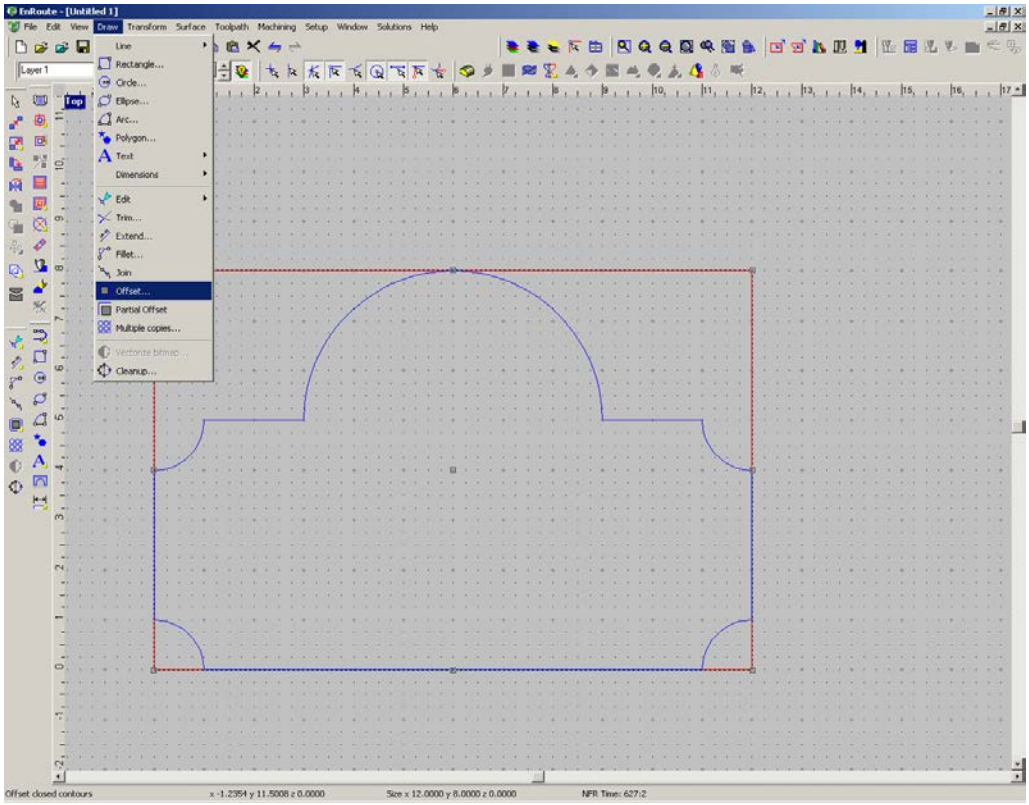
The first thing we’ll do is draw a rectangle with reverse filleted corners.



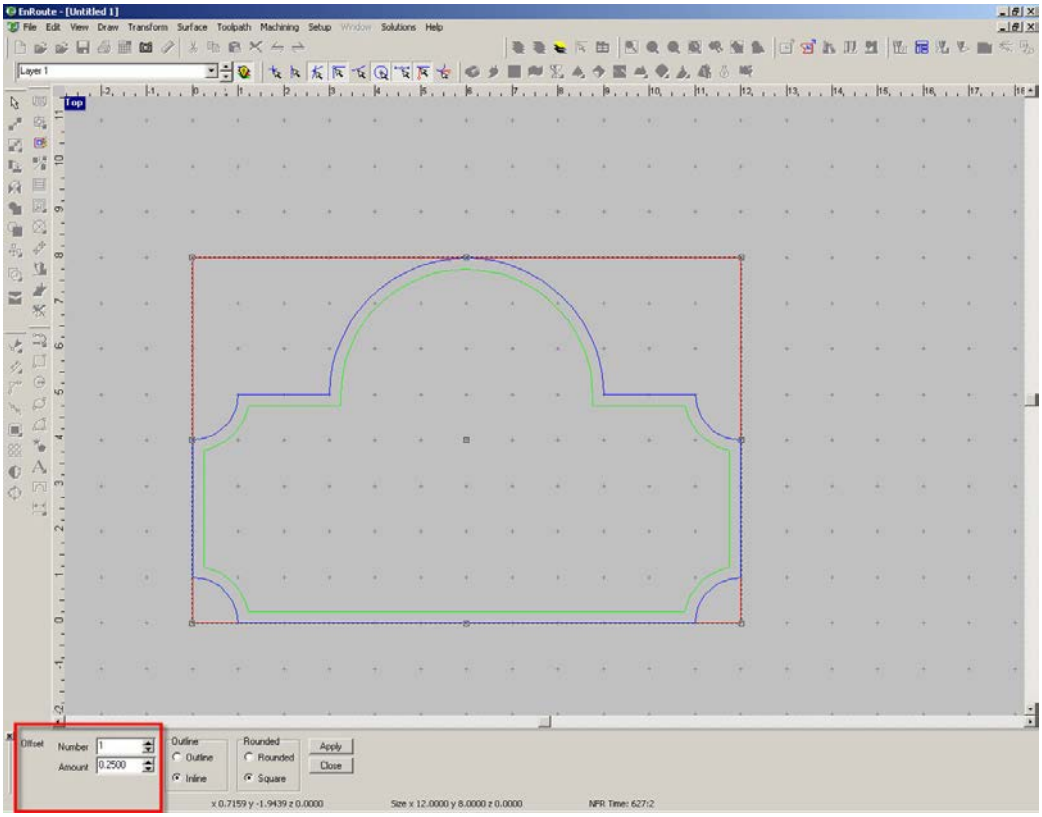
Next, let's add a circle, centered on the top side of the rectangle.



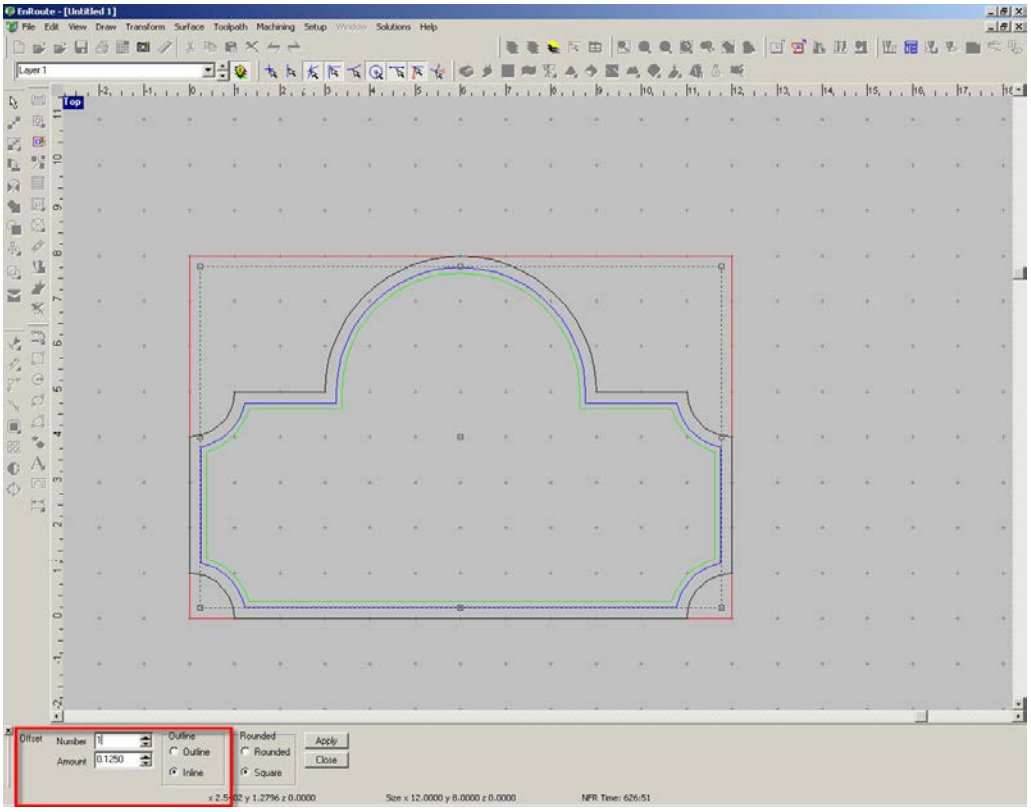
Highlight both objects and click the "weld joined" tool to trim out the intersection.



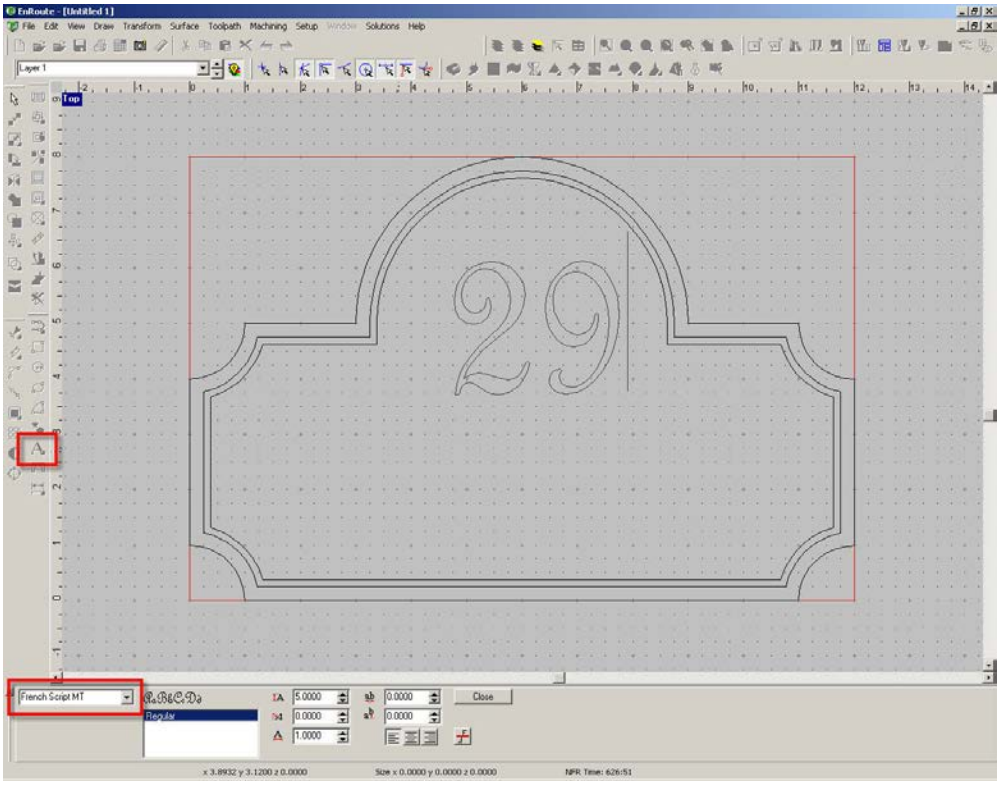
Highlight the new contour and go to “Draw > Offset”.



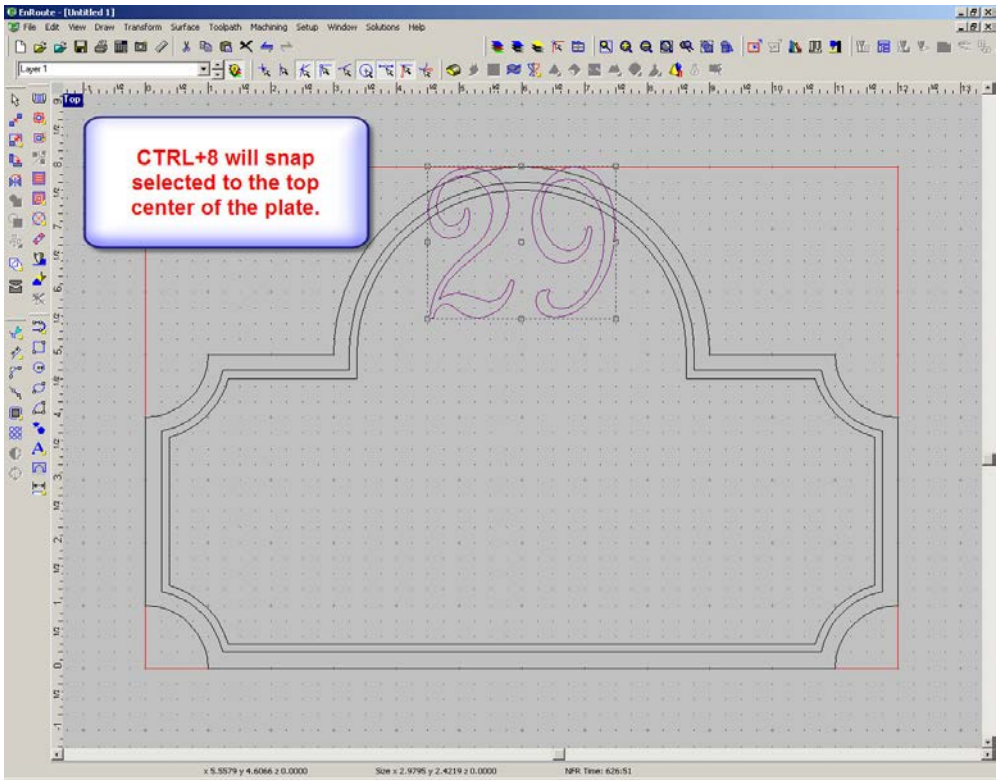
Now, let’s offset the contour in 1/4”.



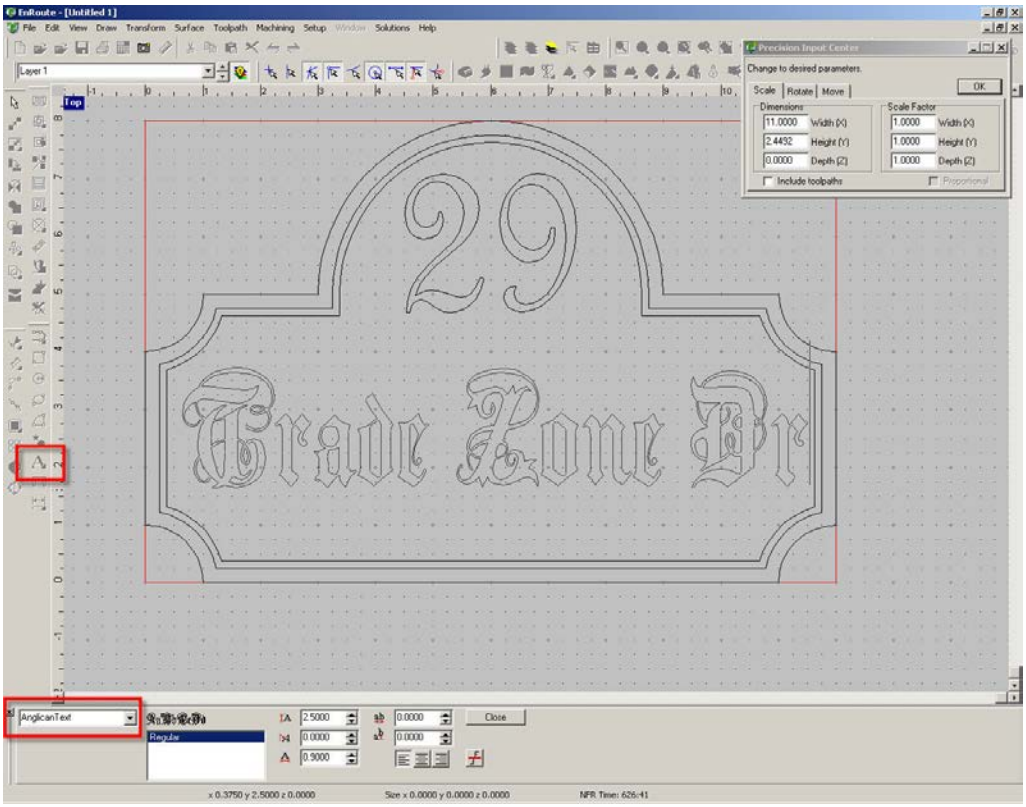
Then offset that line in an additional 1/8”.



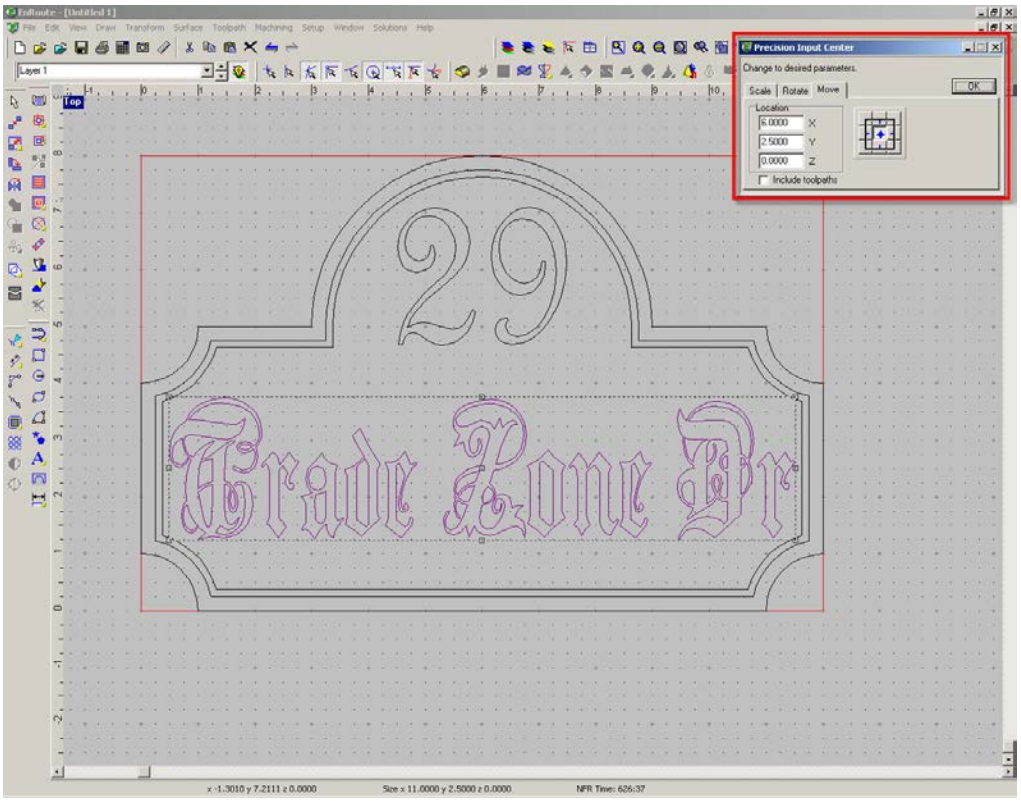
Now, add some text. We’ll make this an address plaque.



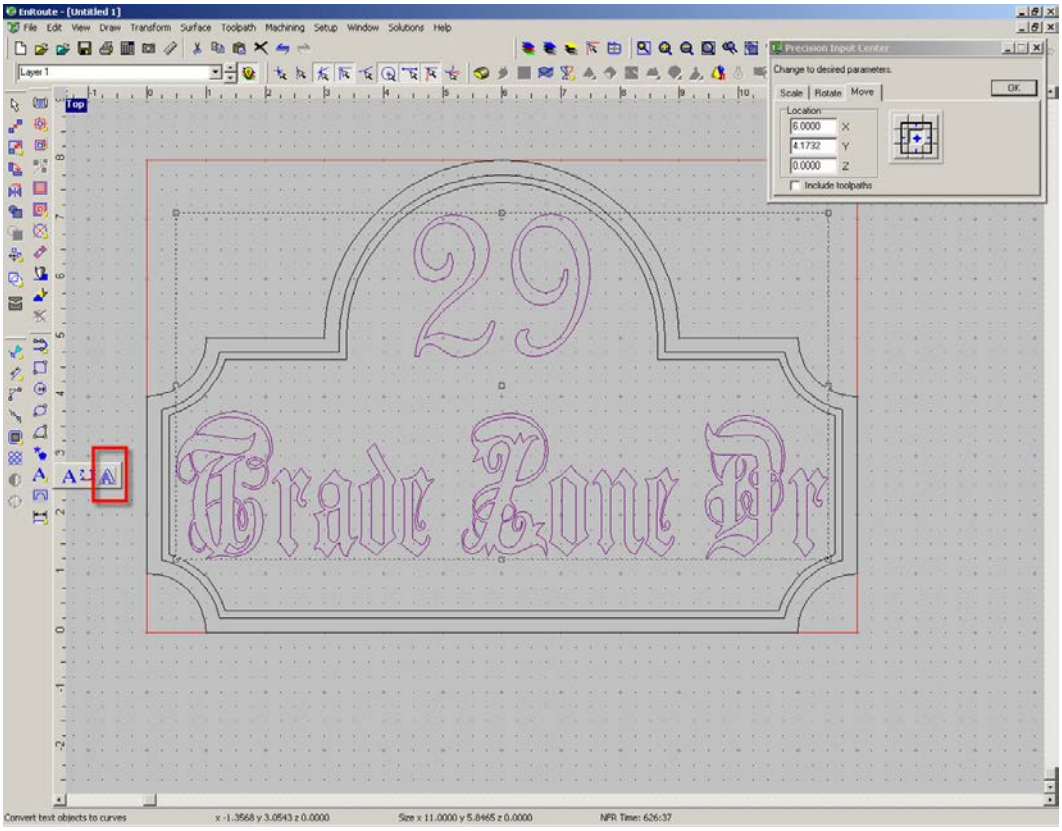
Click on the text and move it to the top center of the plate by holding CTRL=8. (You can snap to the 9 corresponding points on the plate with the different numbers on the keyboard's number pad.) Then, use the down arrow to bump the number into a good position. Of course you can also use the Precision Input Center (PIC) to move it as well.



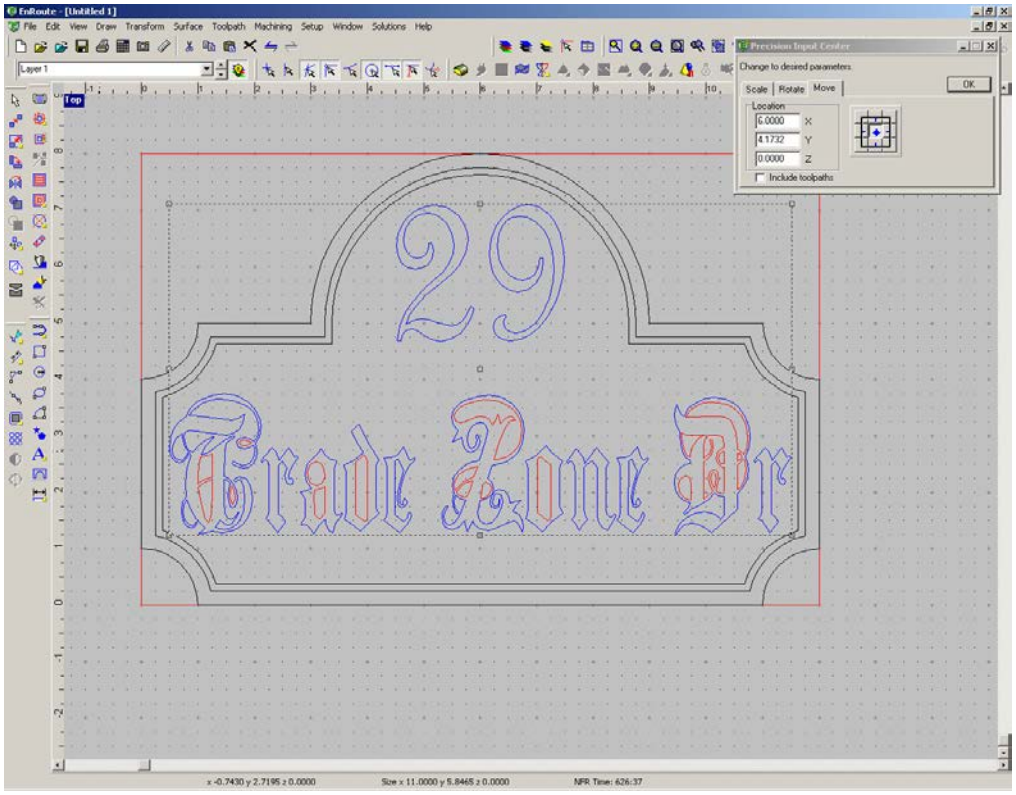
Now, go to the text tool again and type in an address.



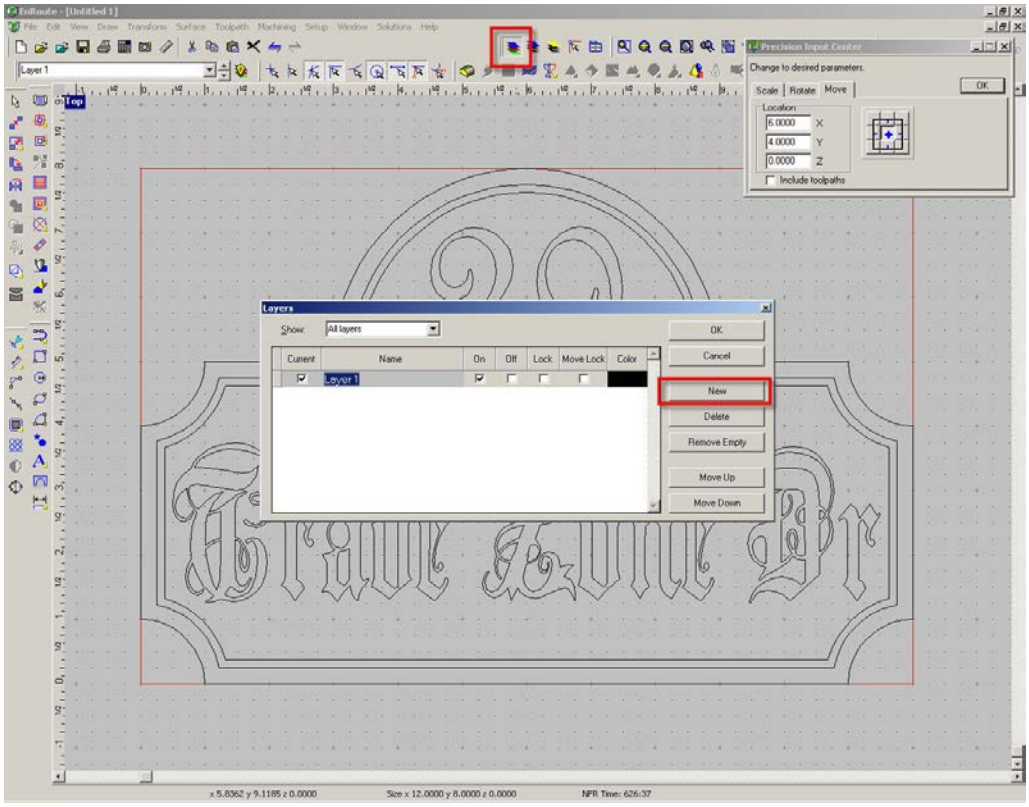
This time, we'll use the PIC to move the center of the text to the X=6, Y=2.5 – which will be centered in our initial rectangle.



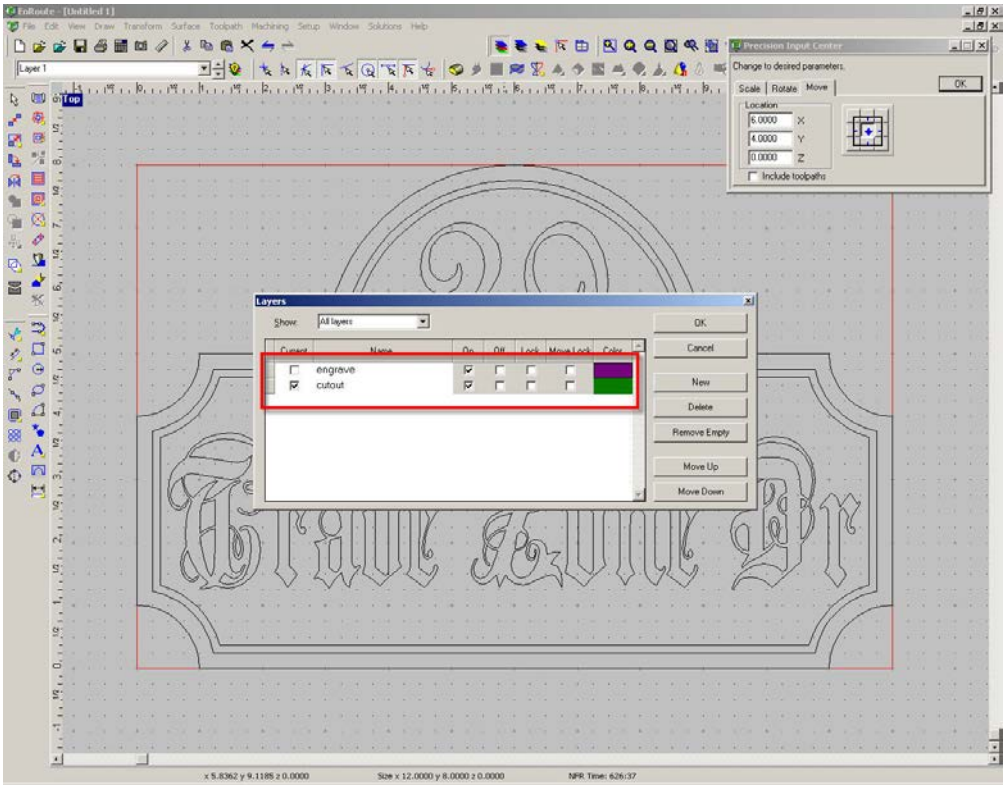
Now, convert the text to curves.



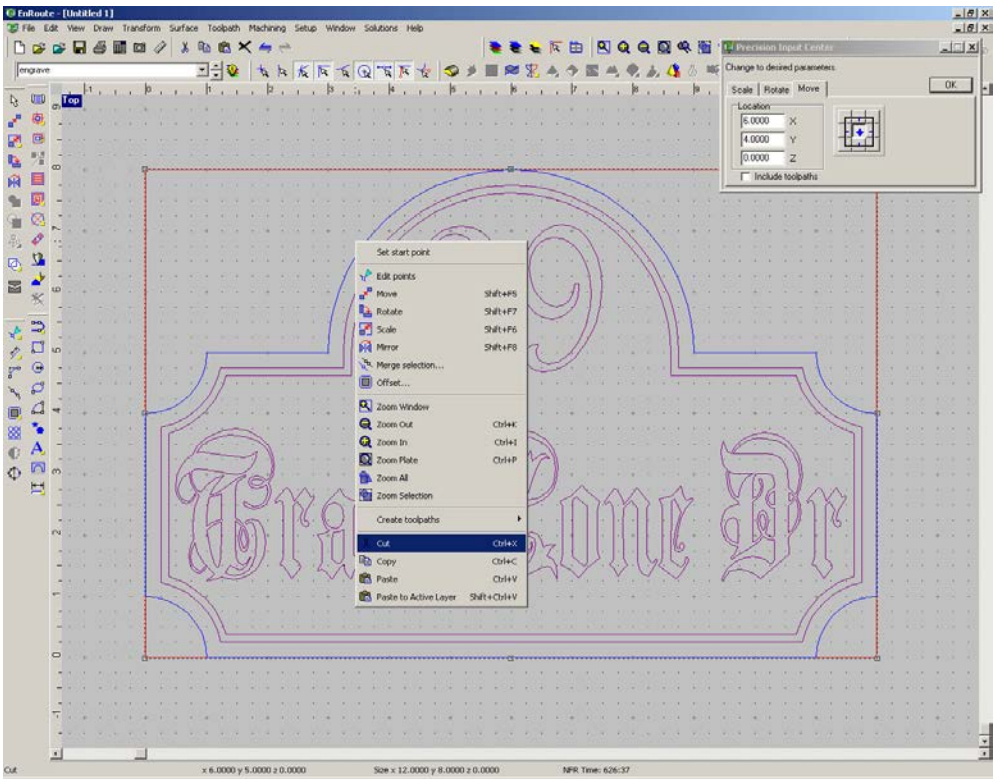
The drawing is complete. Now let's separate it into layers to make outputting the cuts easier.



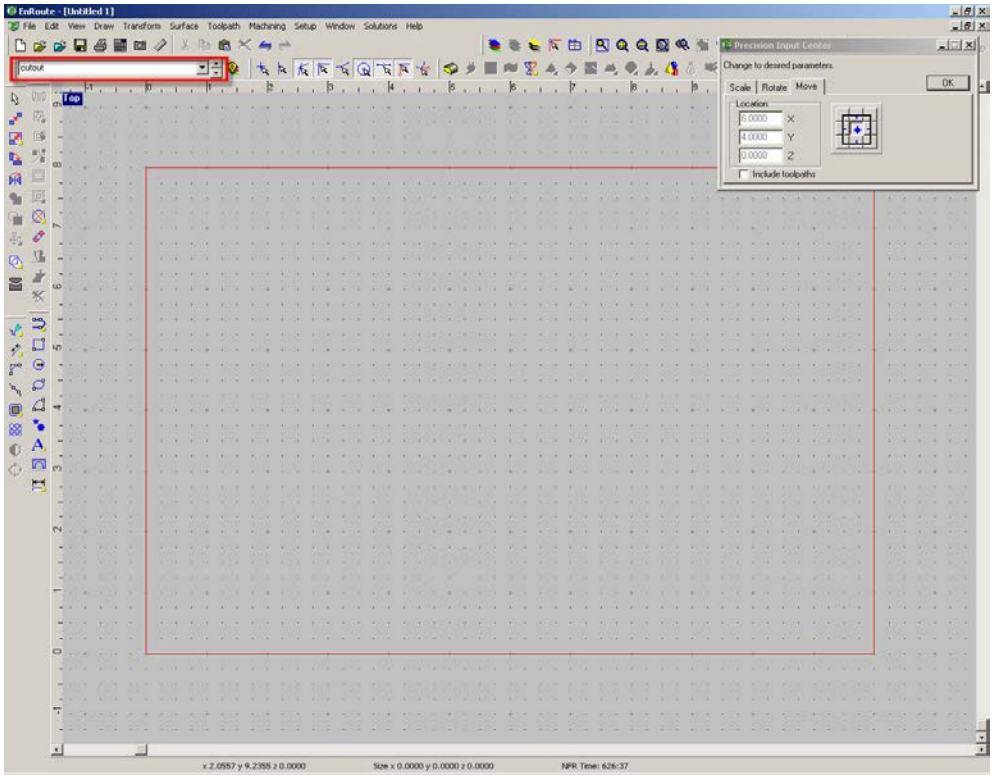
Click on "Define Layers" or hit F7 to open the layer manager.
Click the "New" button once to create one additional layer.



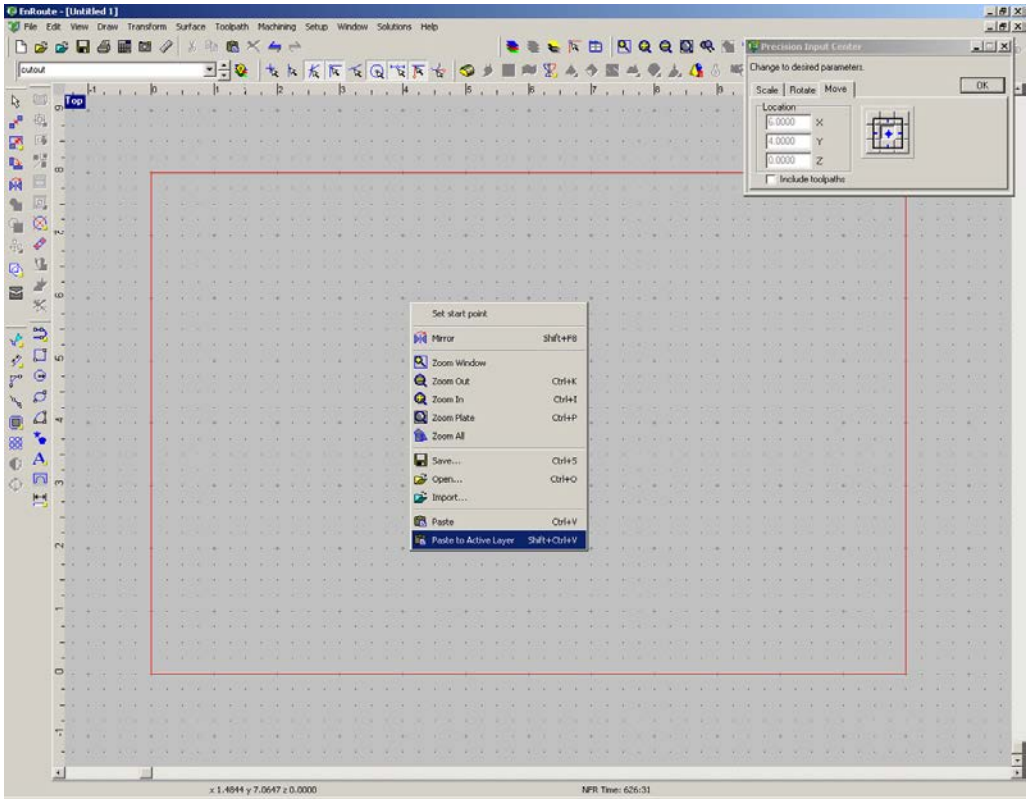
Now we have enough layers. Rename layer 1 to “engrave” and layer 2 to “cutout”. You can also change the layer color.



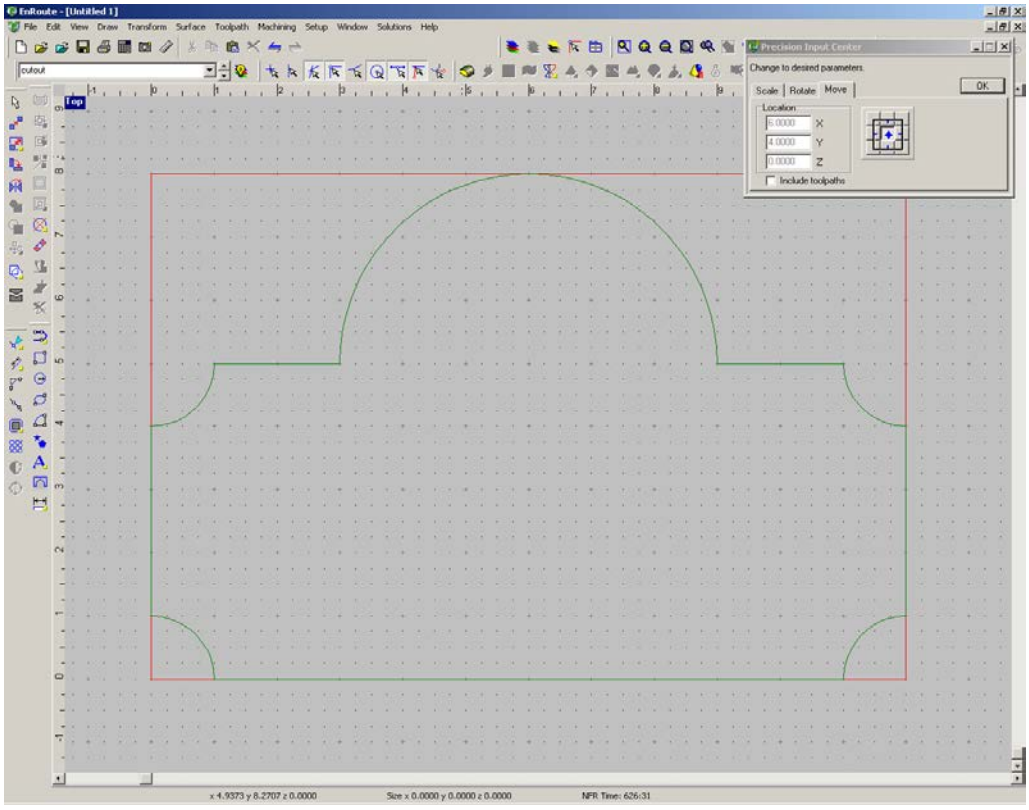
Close the layer manager and select the outer border. This will be our cutout line. Right click on it and select “Cut”.



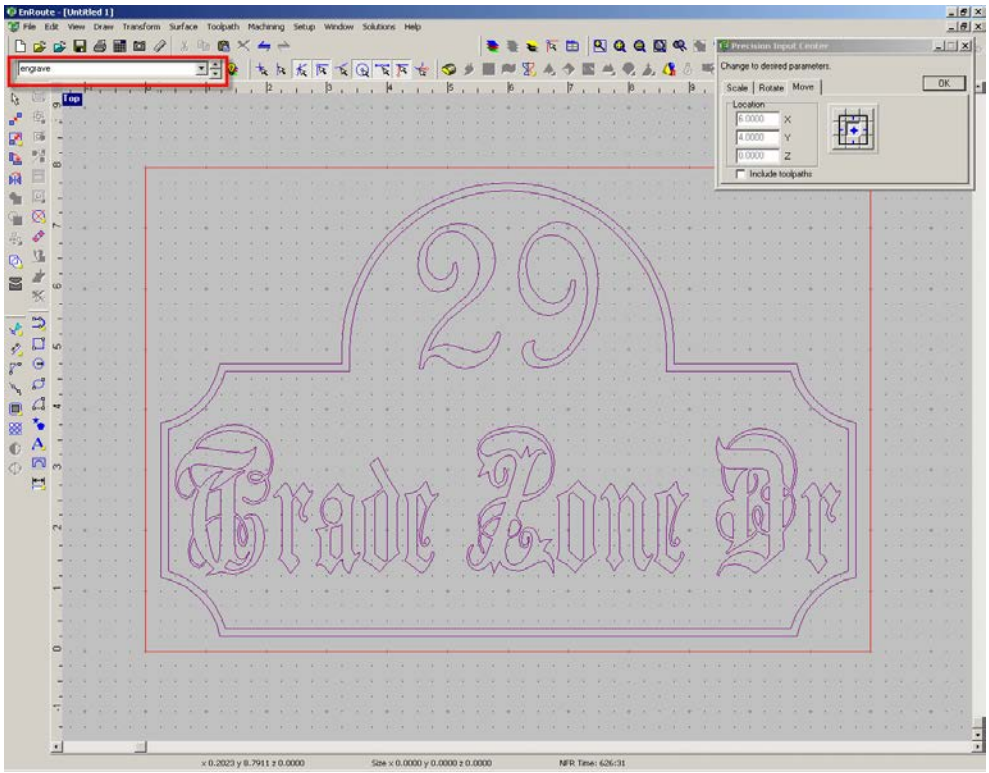
Now, using the layer toolbar, go to the “cutout” layer.



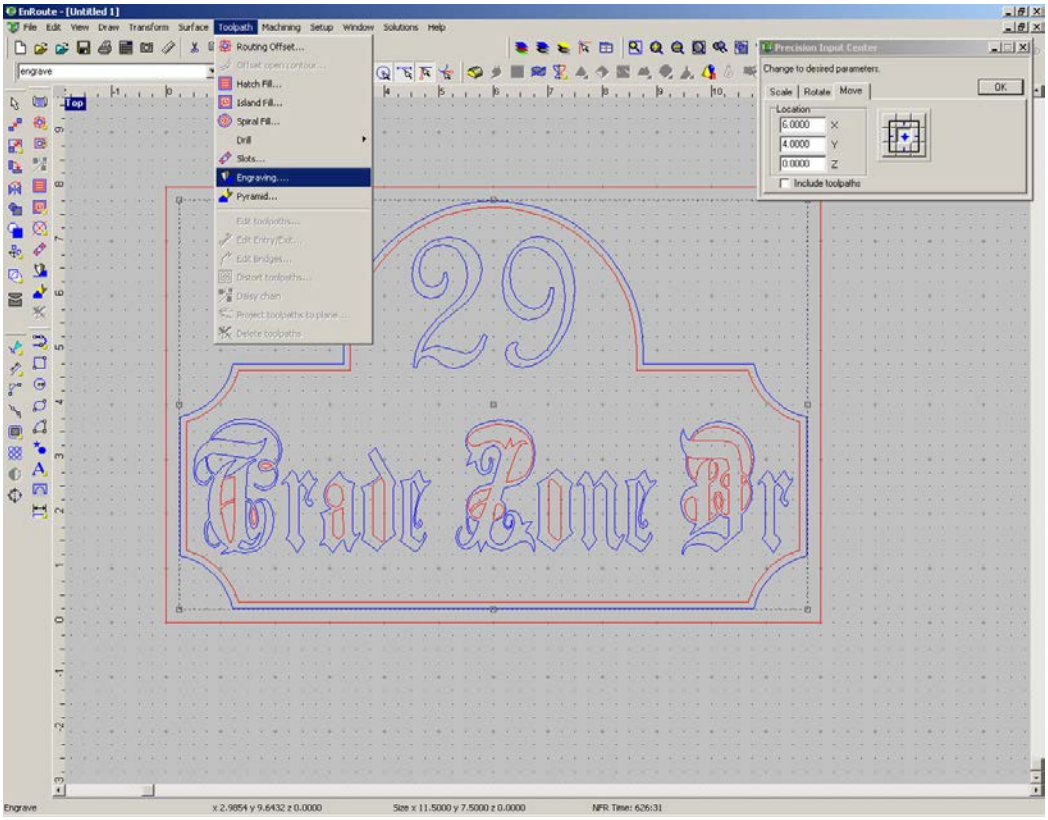
Right-click somewhere on the screen and then select “paste to active layer”.
Note: if you just select “paste”, the object will be pasted back on its original layer.



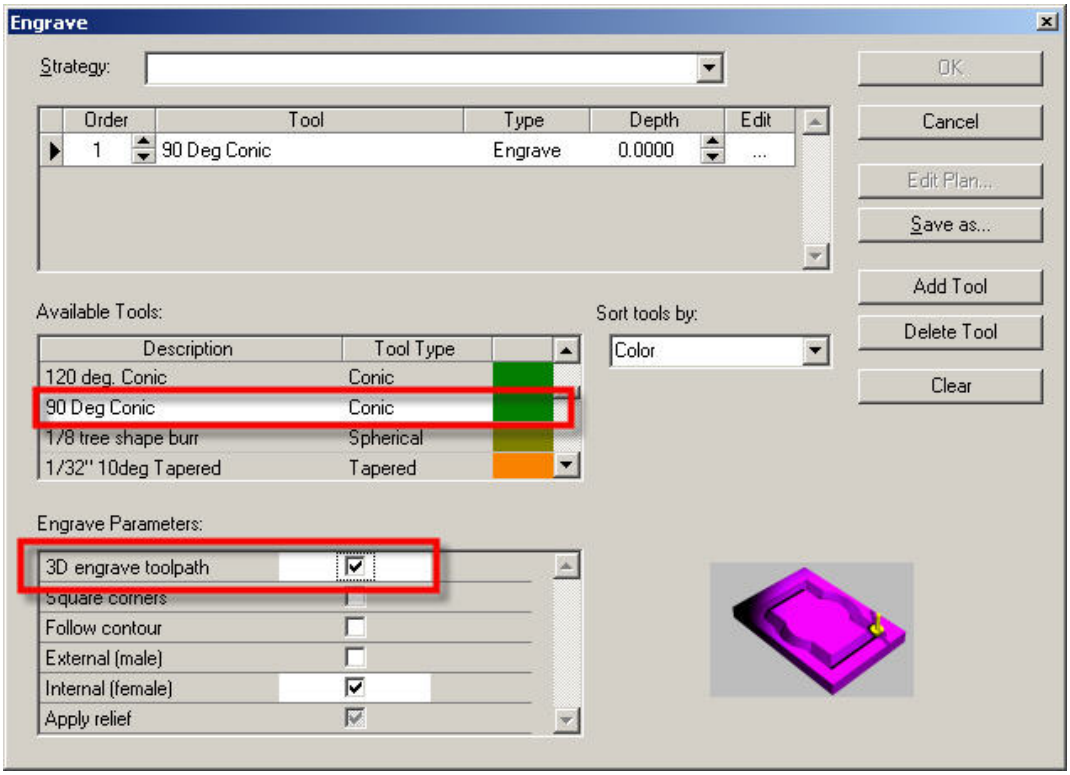
Now the cutout contour is on the correct layer.



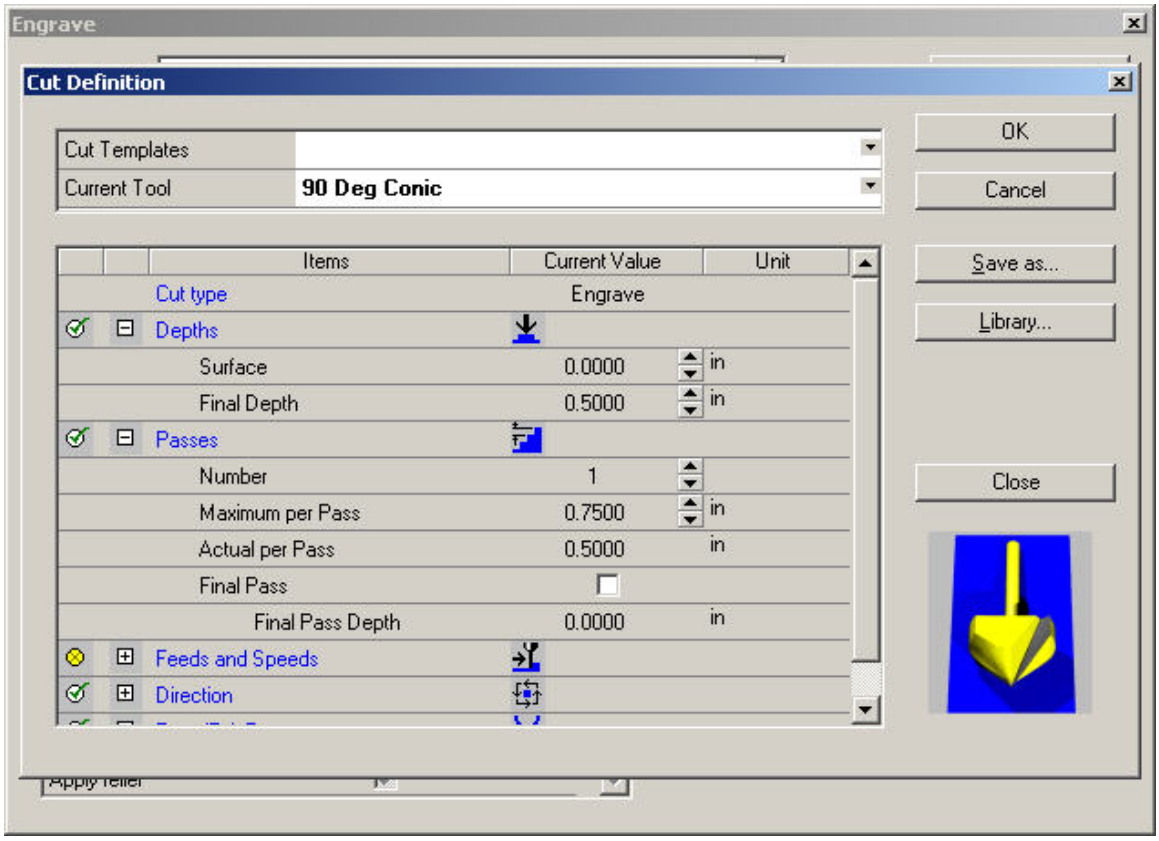
Use the layer toolbar and go back to the “engrave layer”.



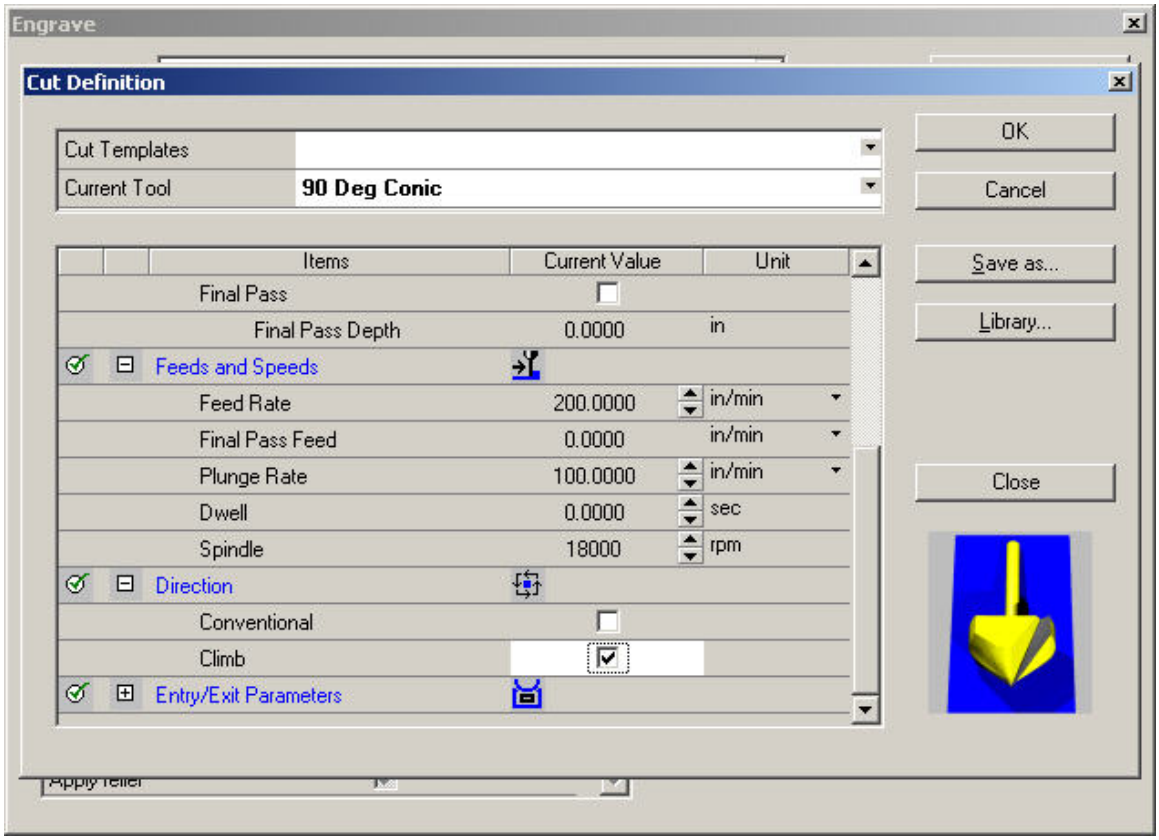
Select all and go to “Toolpath > Engraving”.

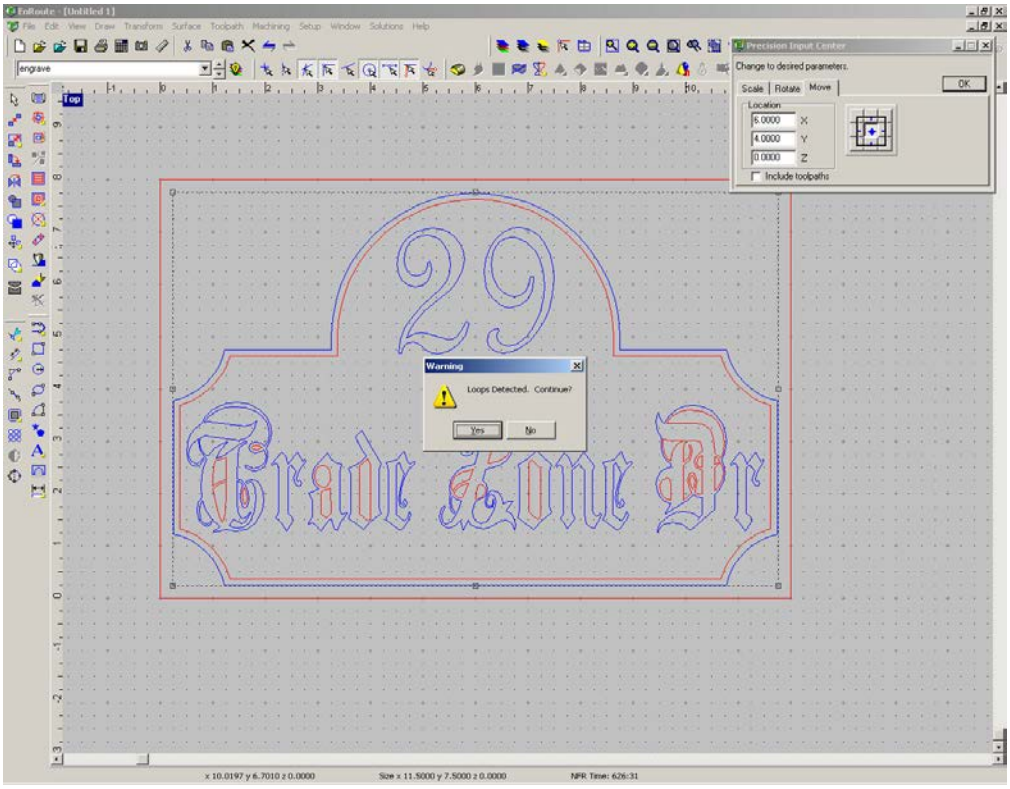


Select a 90 degree tool for the cut. IMPORTANT: If “3D engrave toolpath” is not selected, the lines of the selected contours will be engraved and the cut will be wrong. We want to engrave within the lines, not on them.

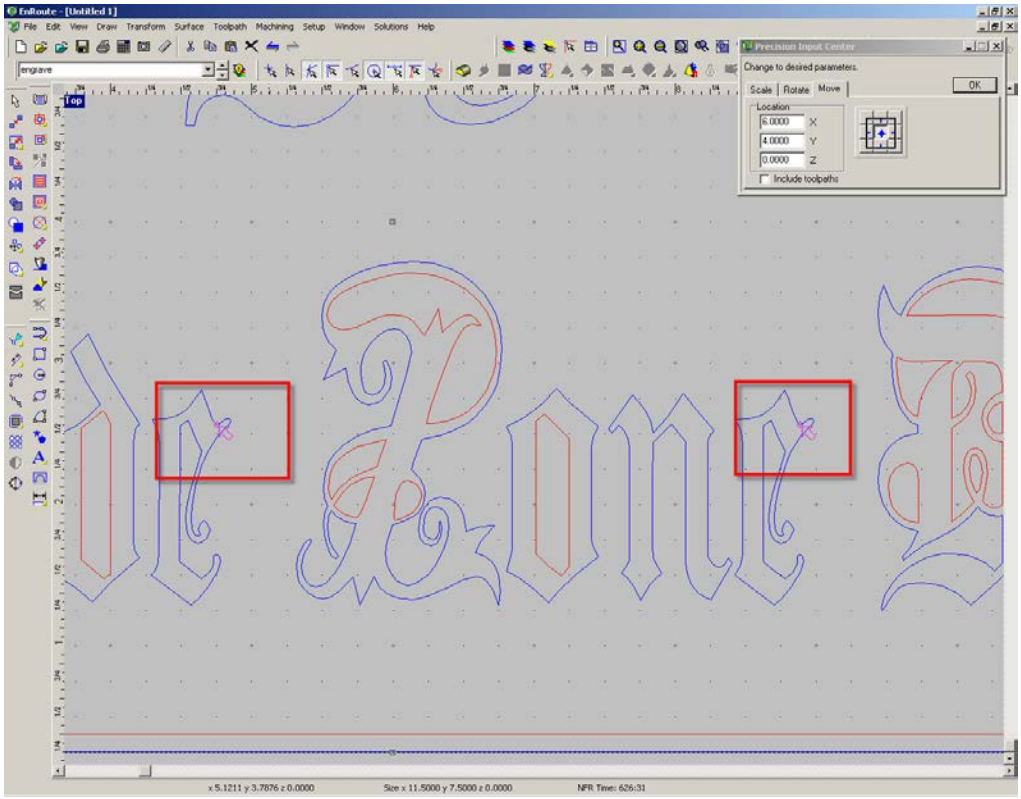


Fill in the cut parameters.

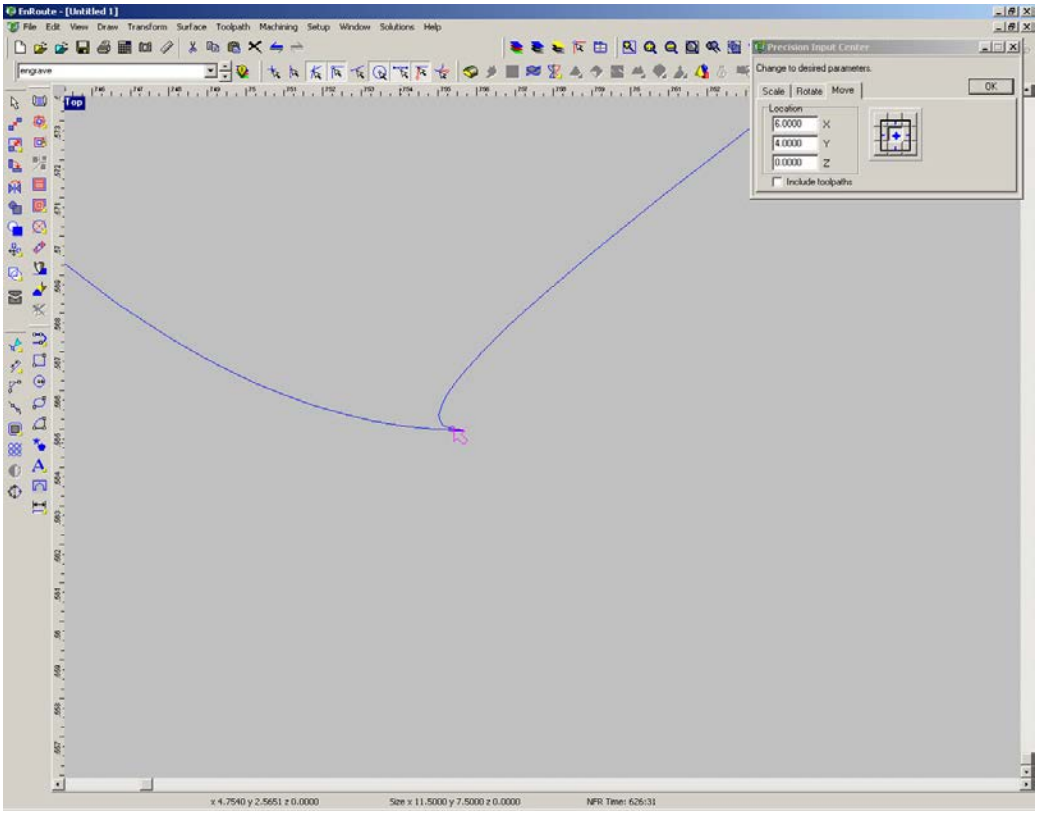




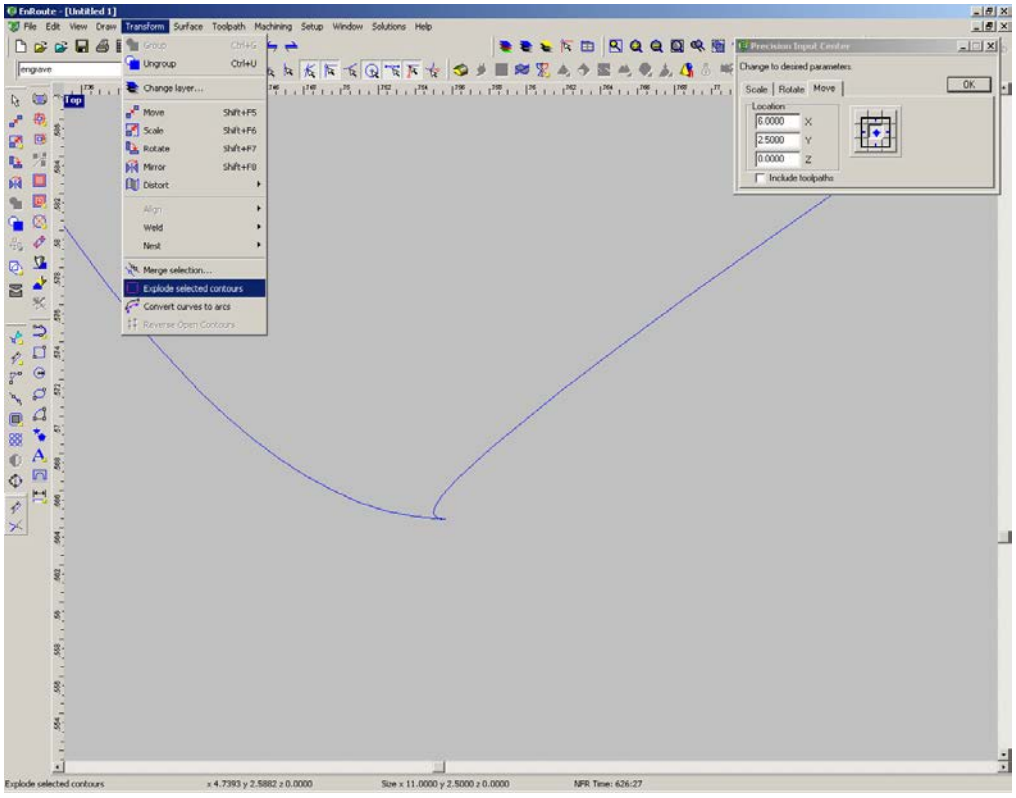
Loops detected. The may happen sometimes with engraving fonts due to how they import. Say “No” and we’ll have to clean up the drawing a little bit.



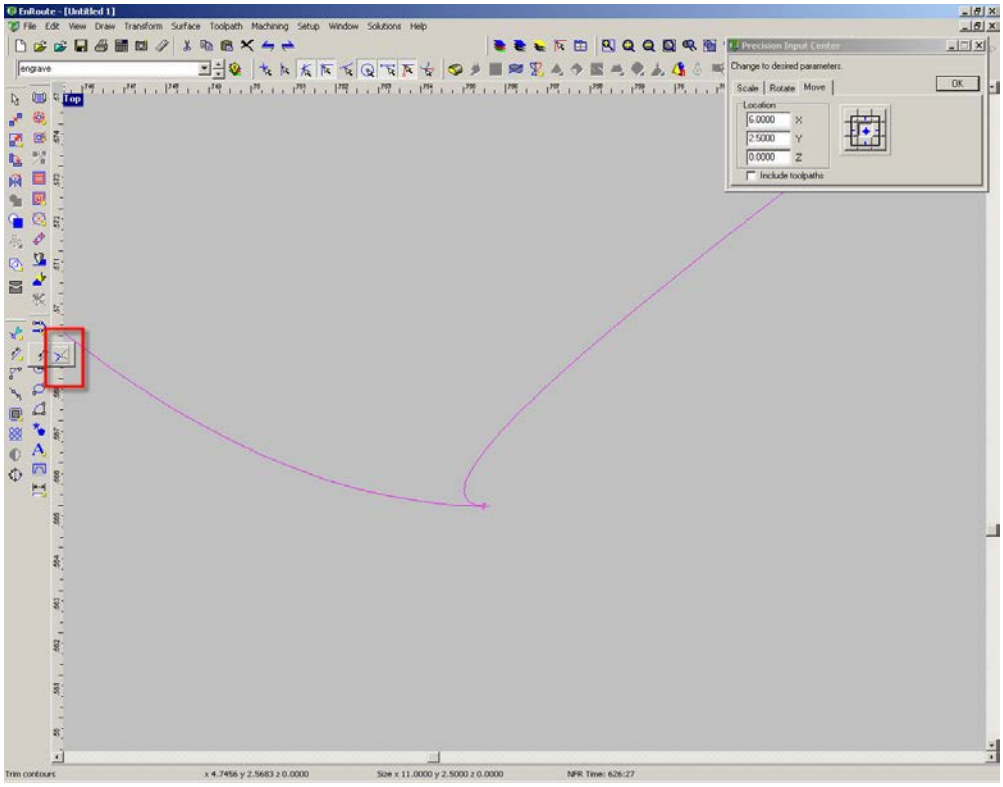
EnRoute points to the problem spots.



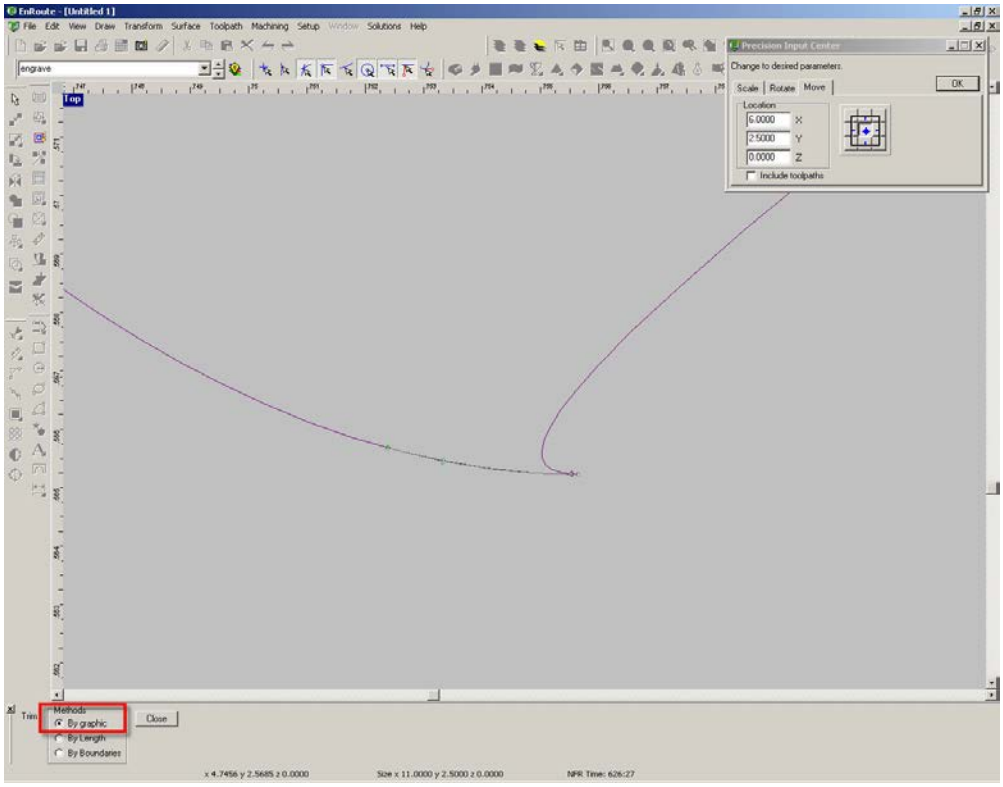
Zoom in close to get a better look.



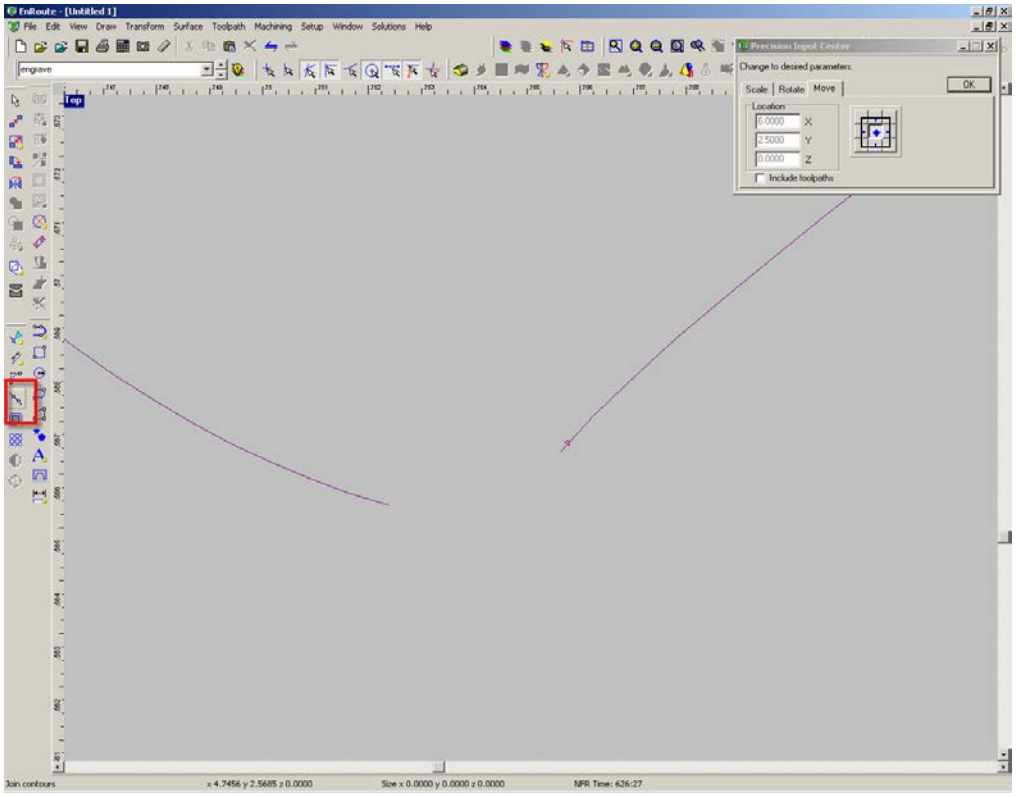
You have to trim out the loop. First explode the contour.



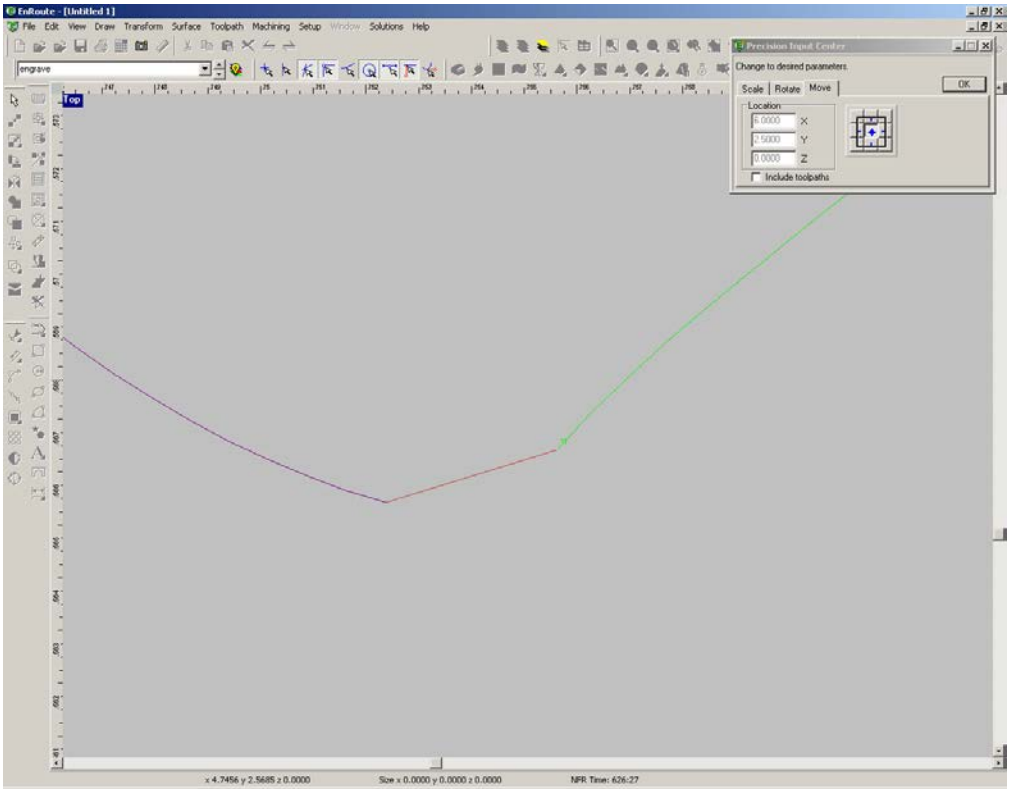
Then, go to the trim tool.



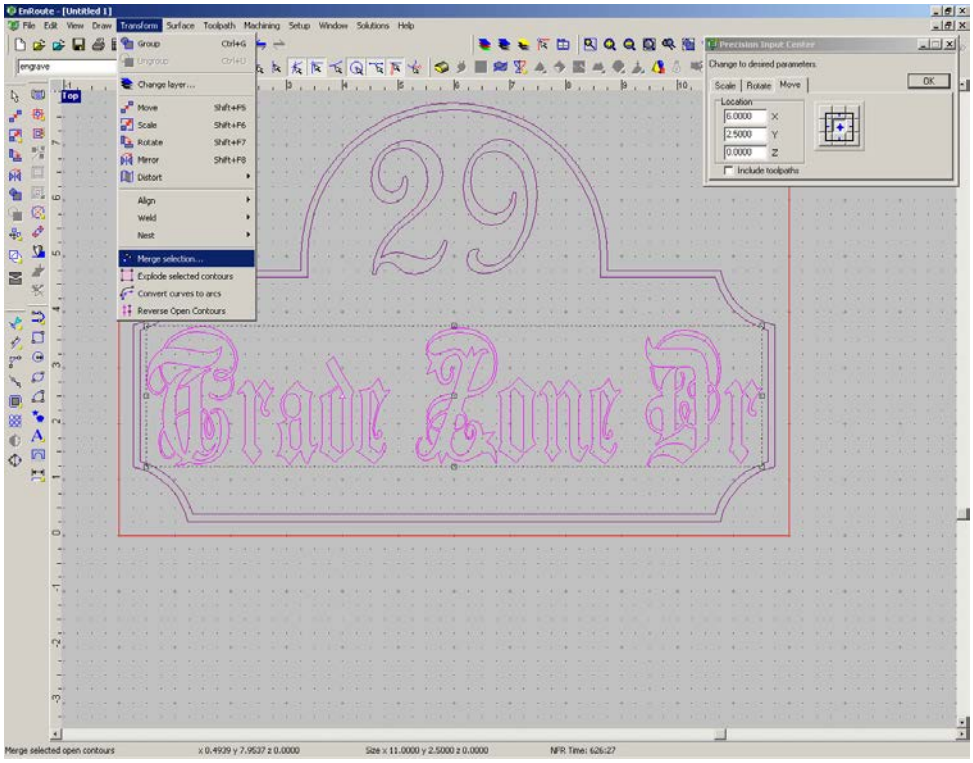
You can trim by graphic. This is a very small area and doesn't have to be that precise.



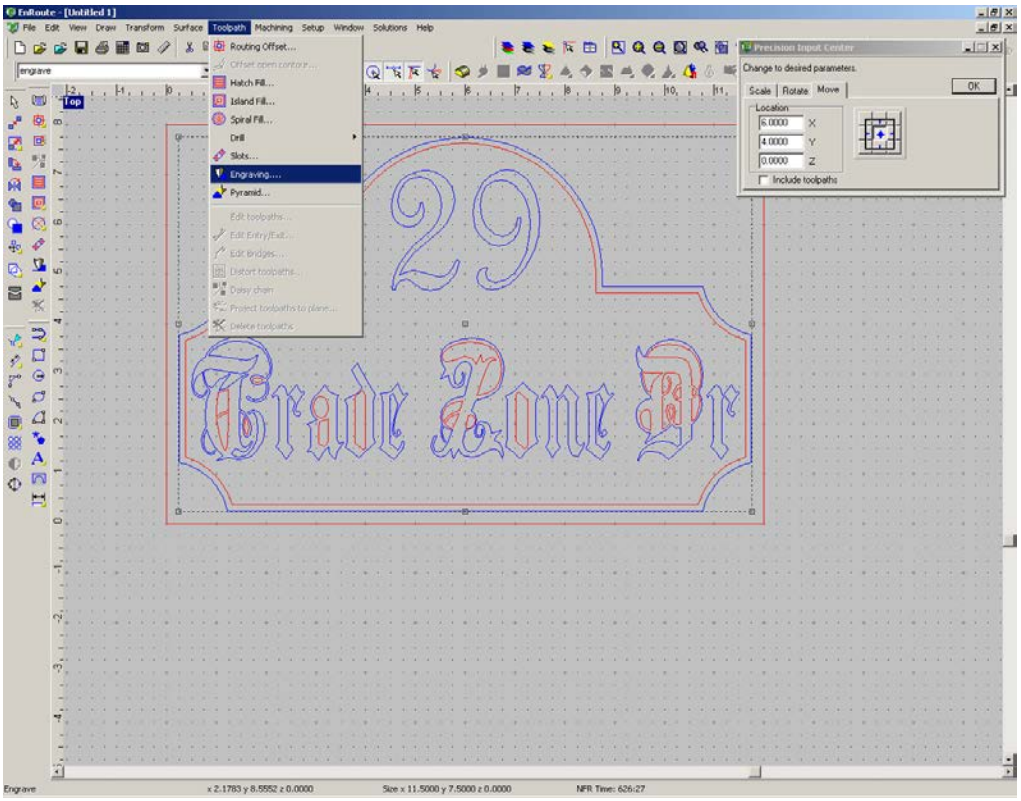
Once the loop is trimmed out, select the “Join Contours” tool.



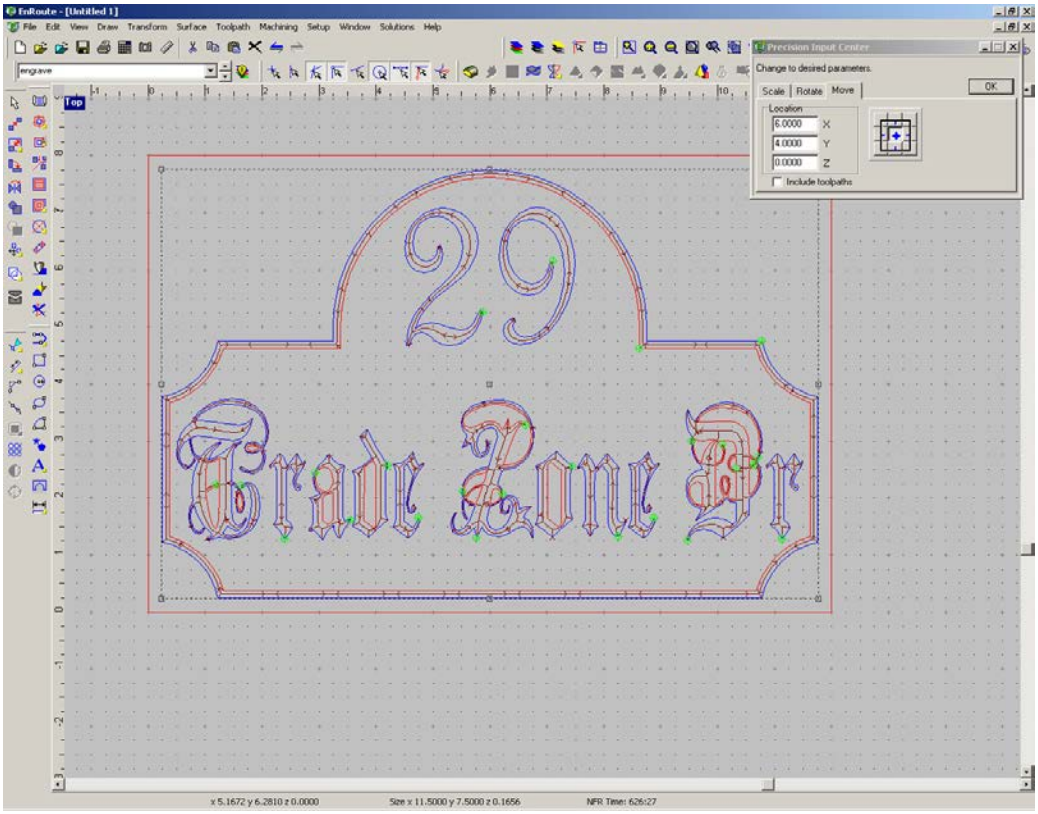
Click on one line, then the other and they will connect.



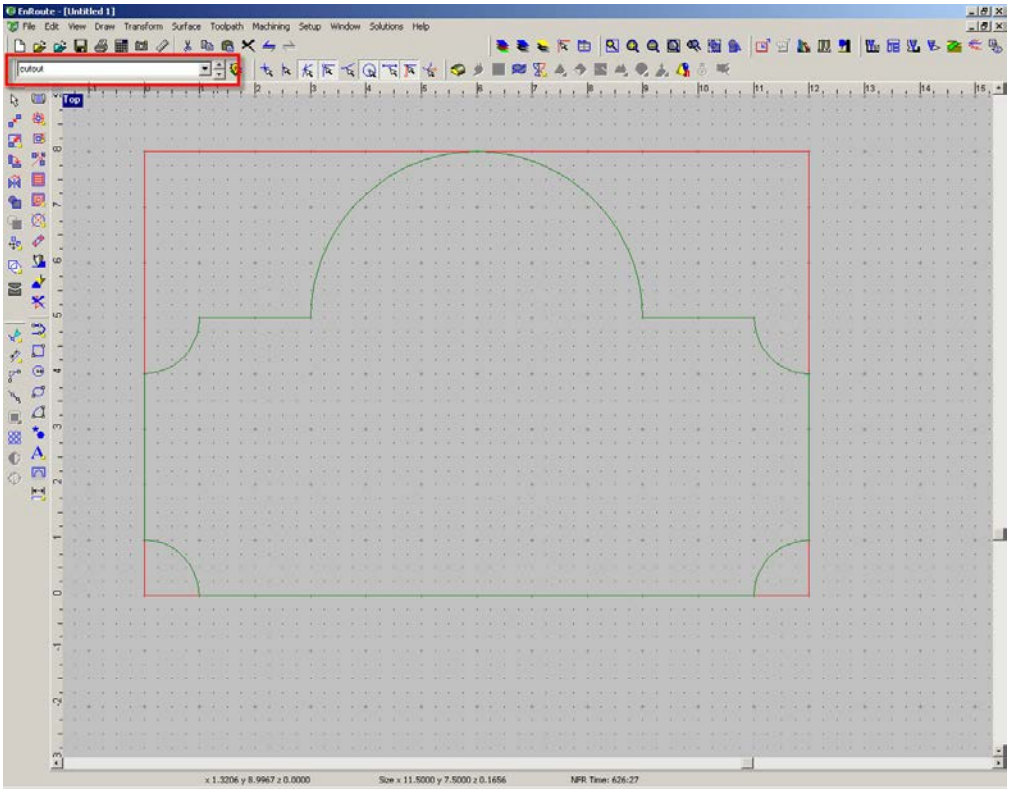
Now, we have to re-merge the contours.



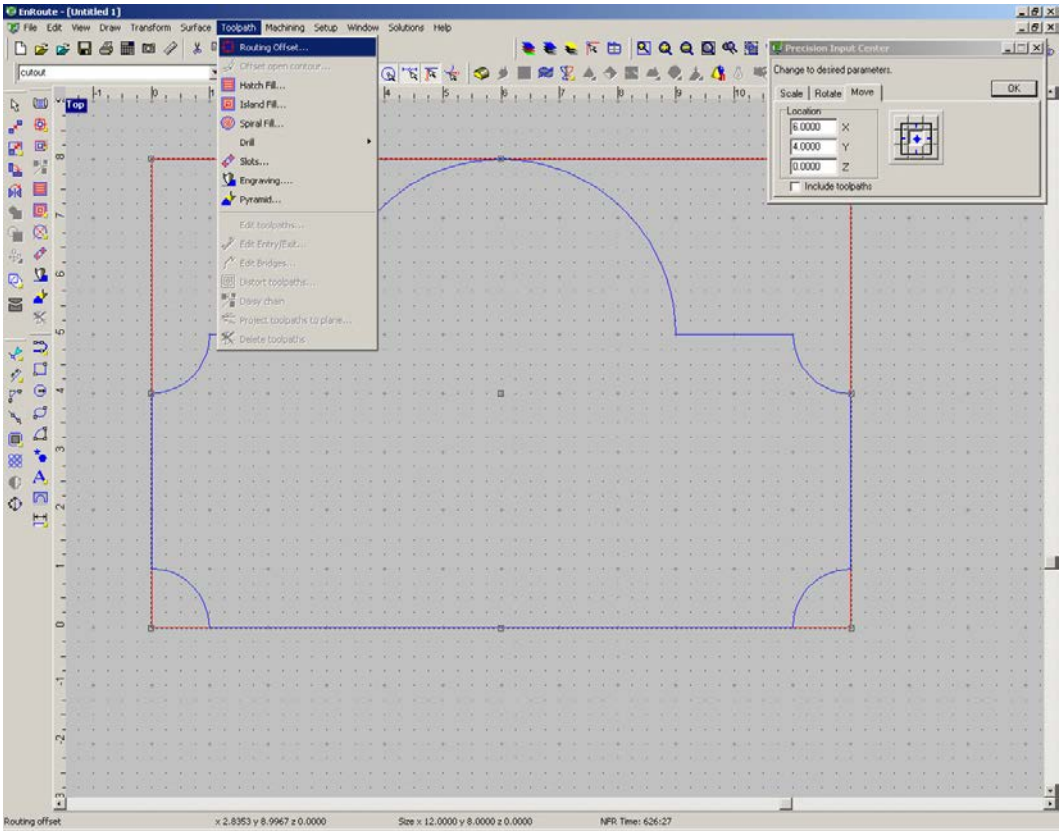
Once they are merged, select all again and re-apply the engrave tool path.
All of the settings we entered earlier are still there, so you can just click “Ok”.



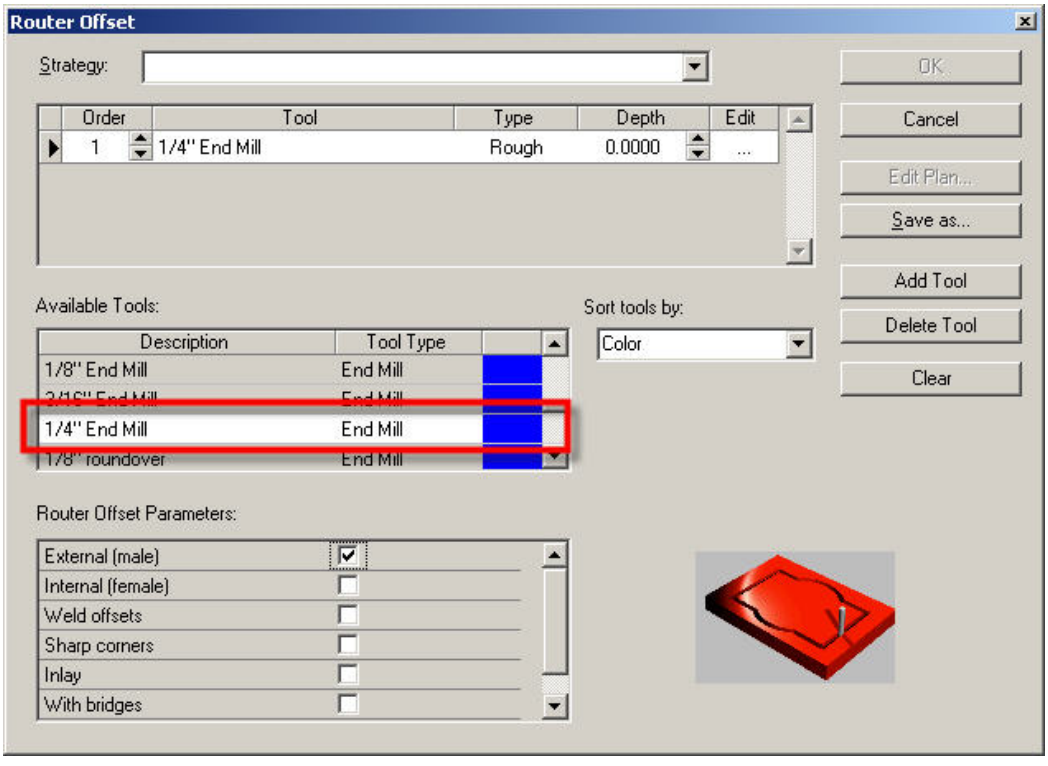
Here is the 3D engrave tool path. Notice how all of the cut lines are inside the contours.



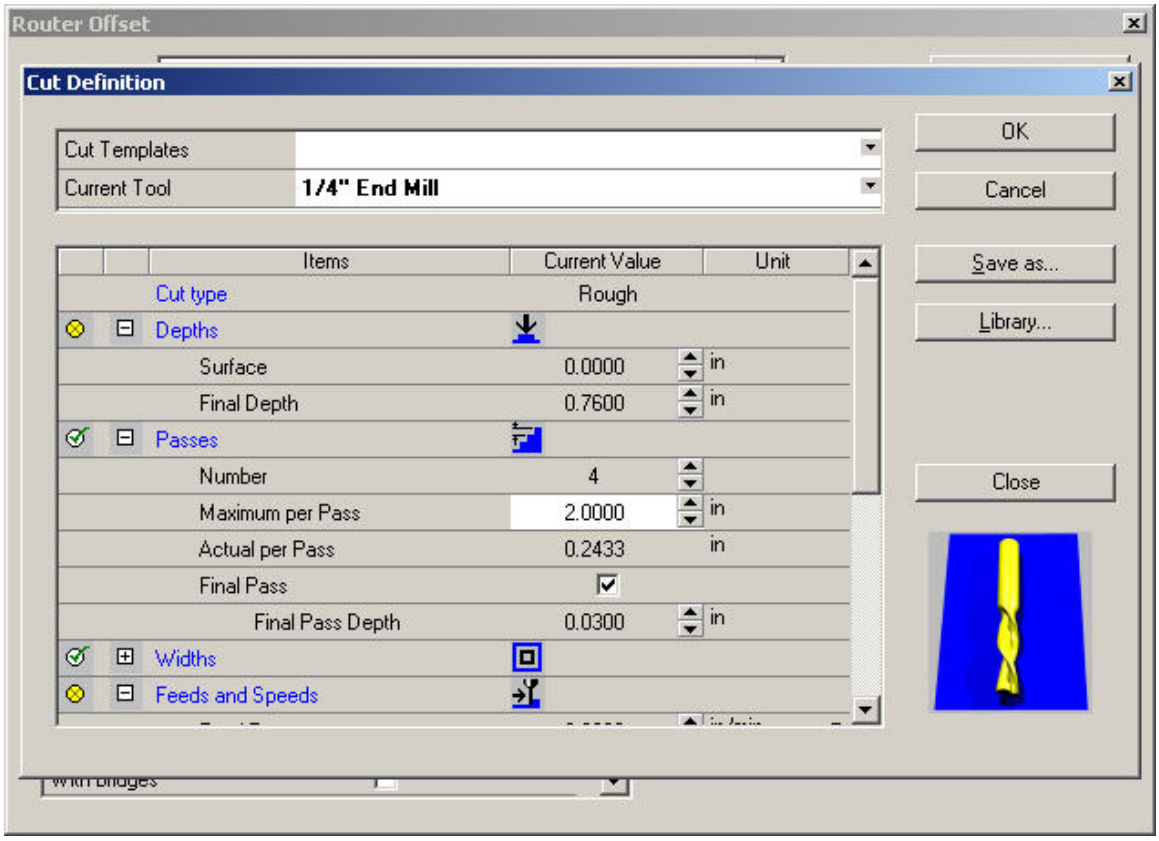
Now, switch to the “cutout” layer and add the offset toolpath.



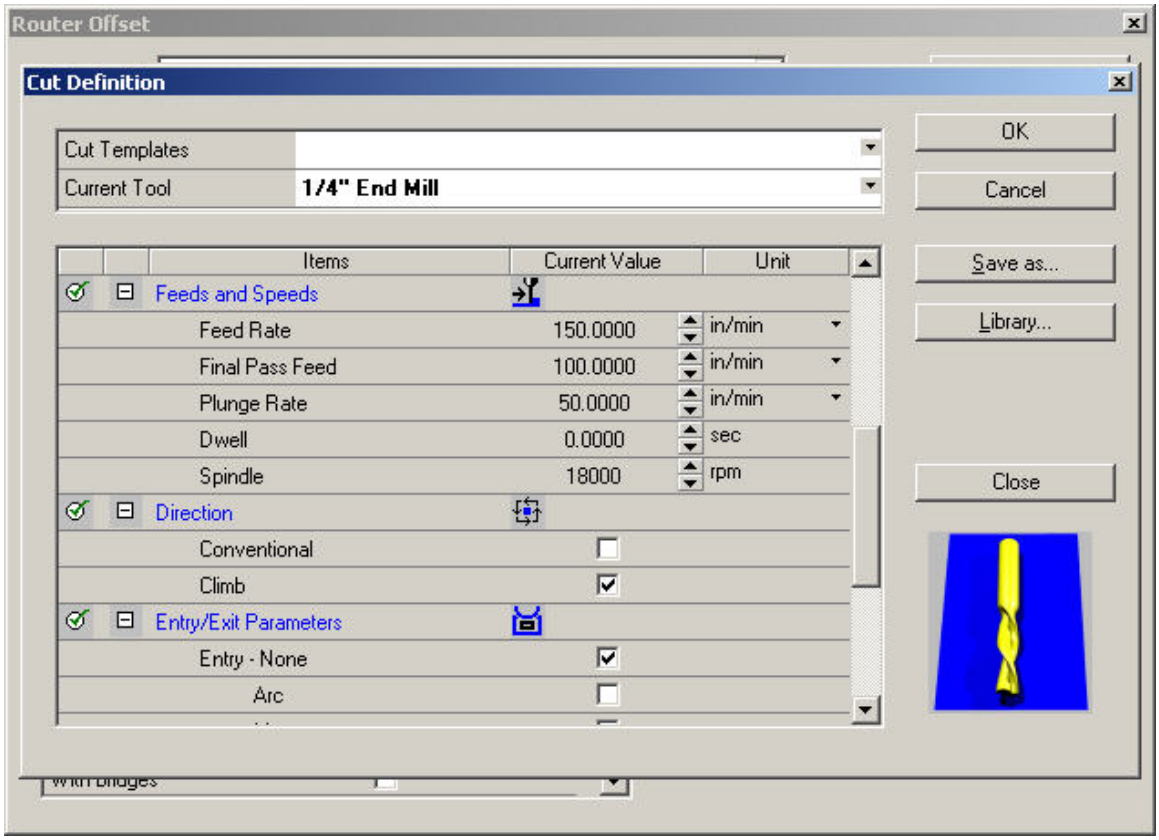
Select the line and go to “Toolpath > Routing Offset”.

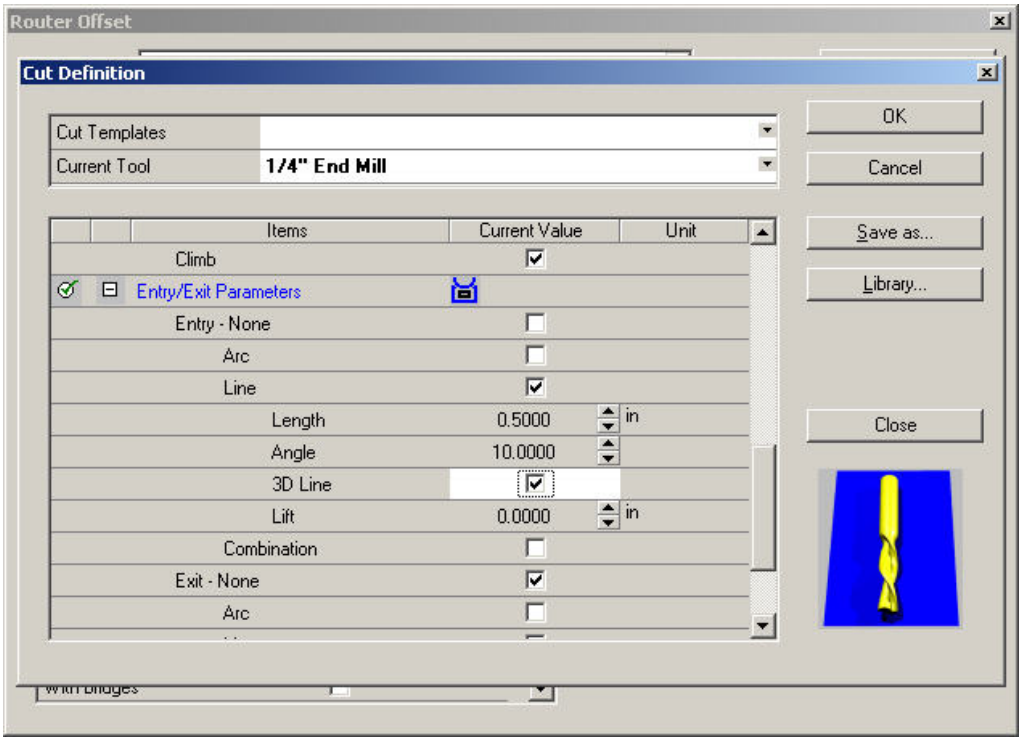


Select a 1/4" endmill and an external cut.

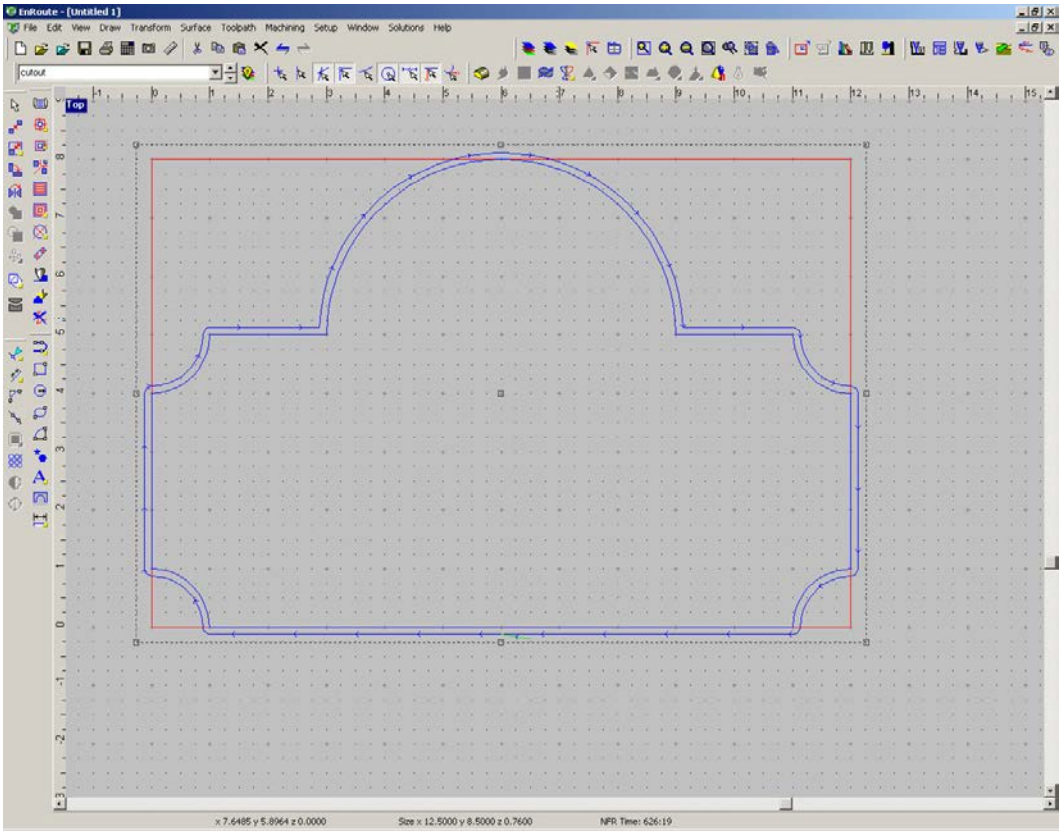


Add the cut parameters as usual.

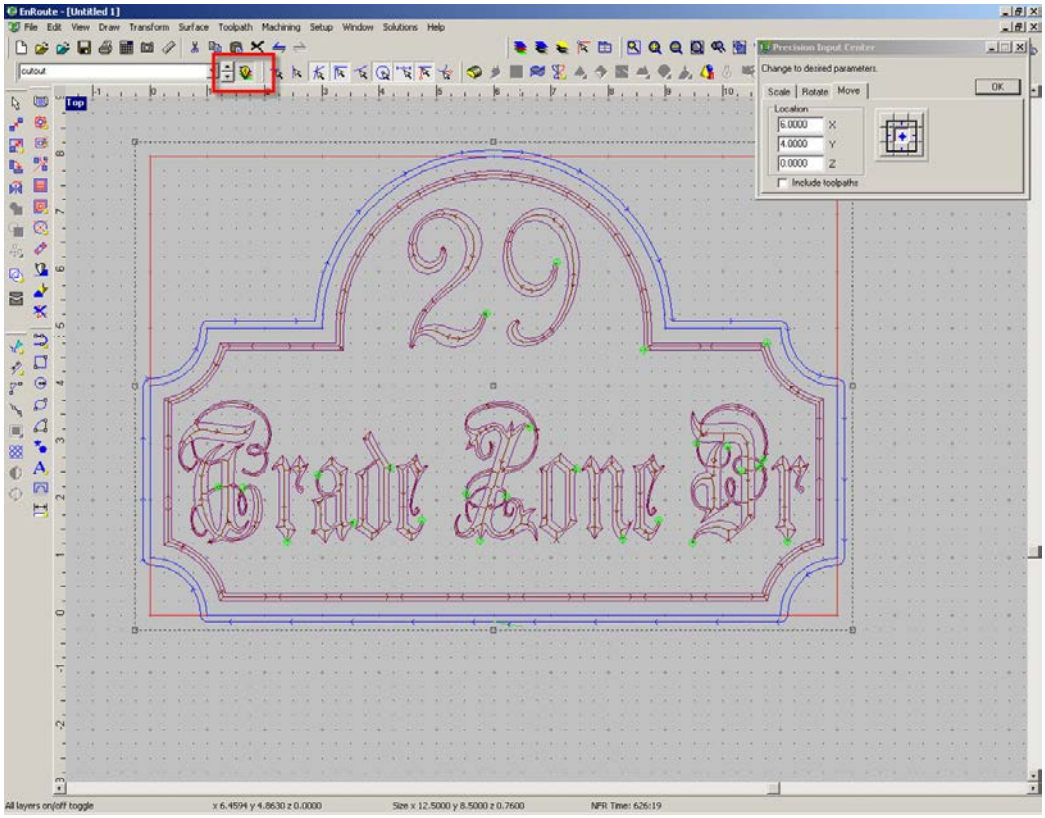




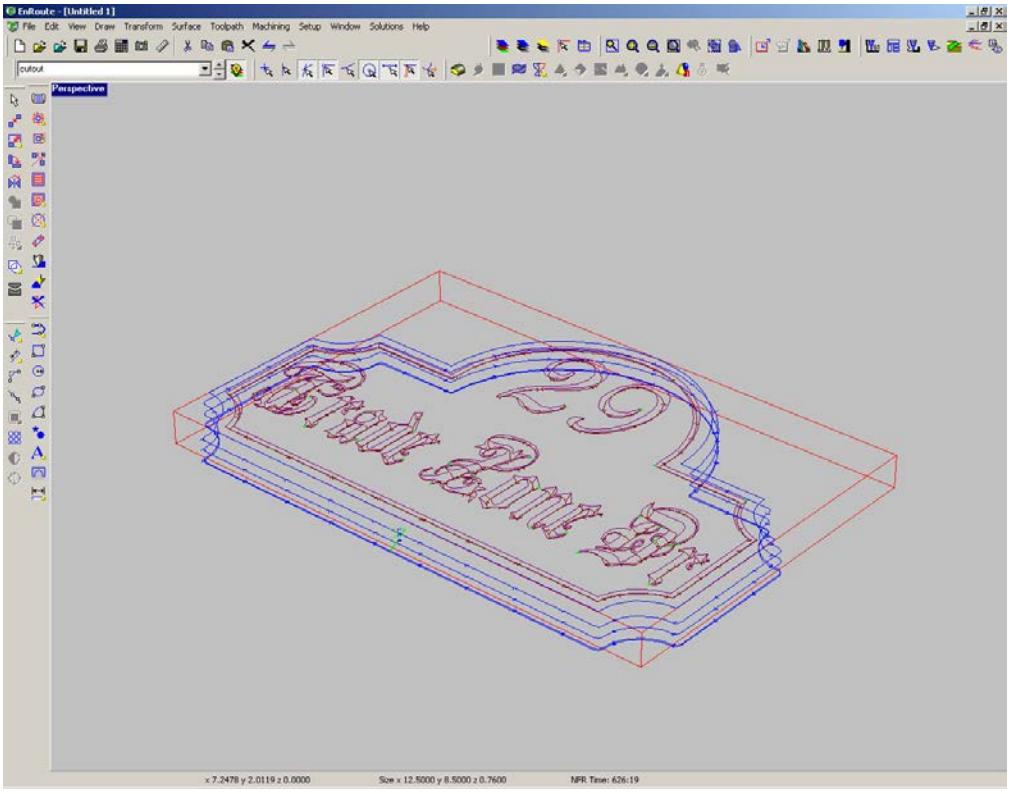
We can put a 3D entry in, so that the cutter doesn't plunge straight in. The entry angle is 10 degrees off of the cut line.



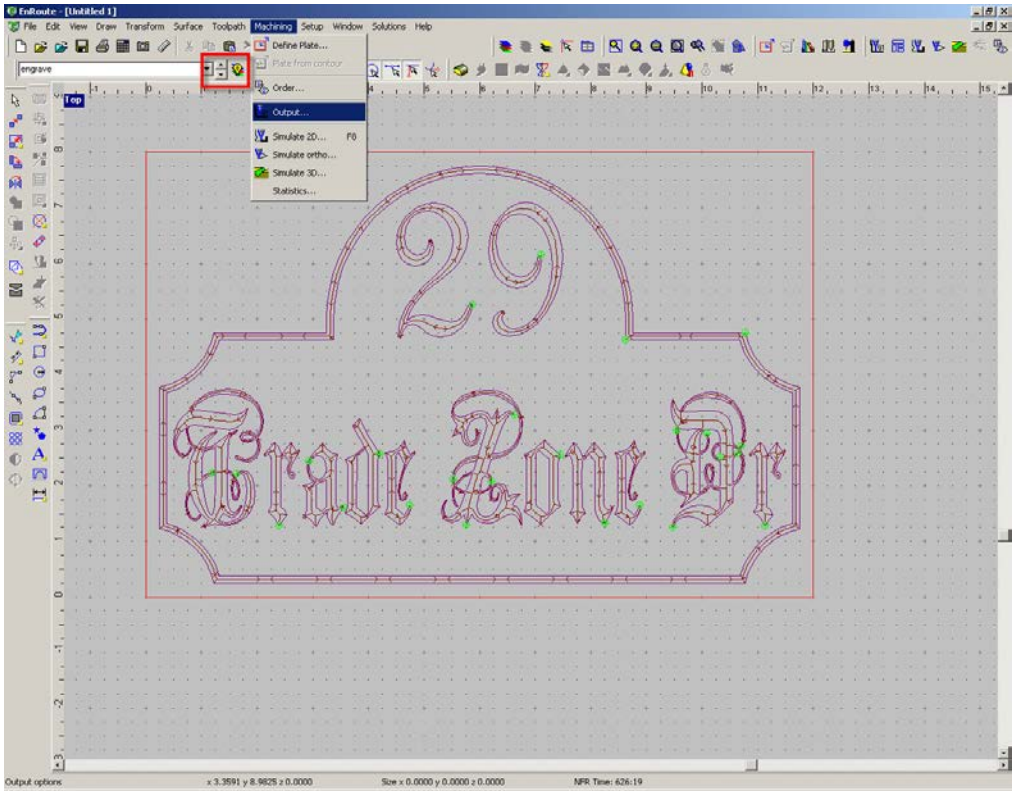
Here is the cutout tool path.



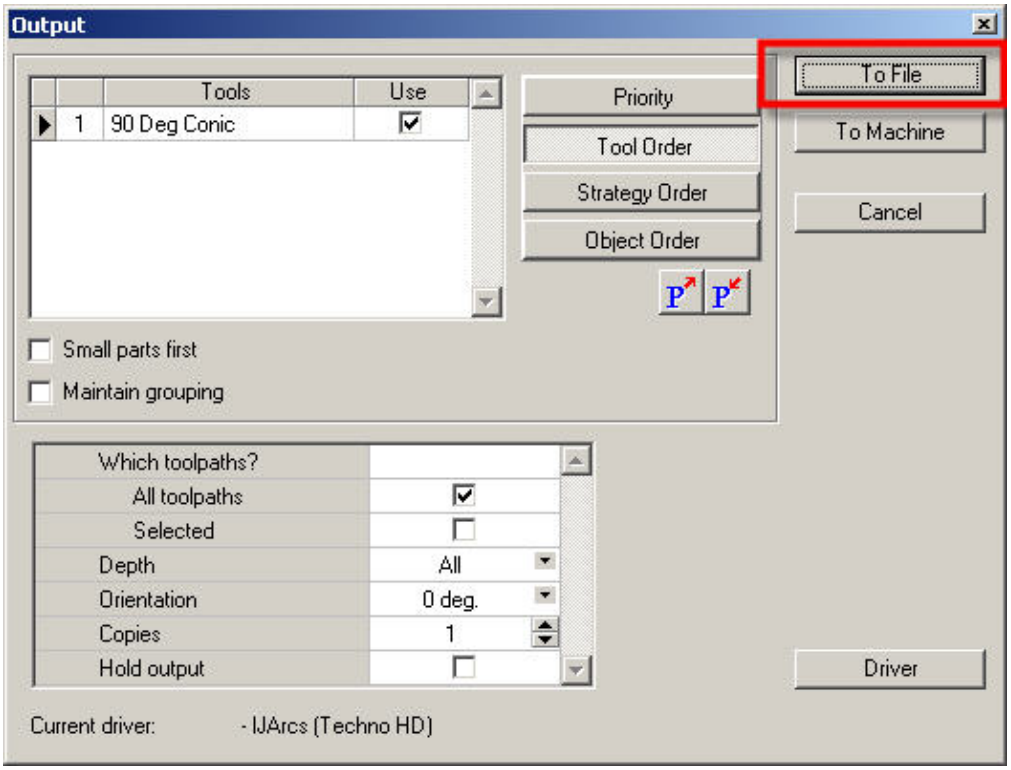
Click on the little light bulb on the layers toolbar to show both layers at once.



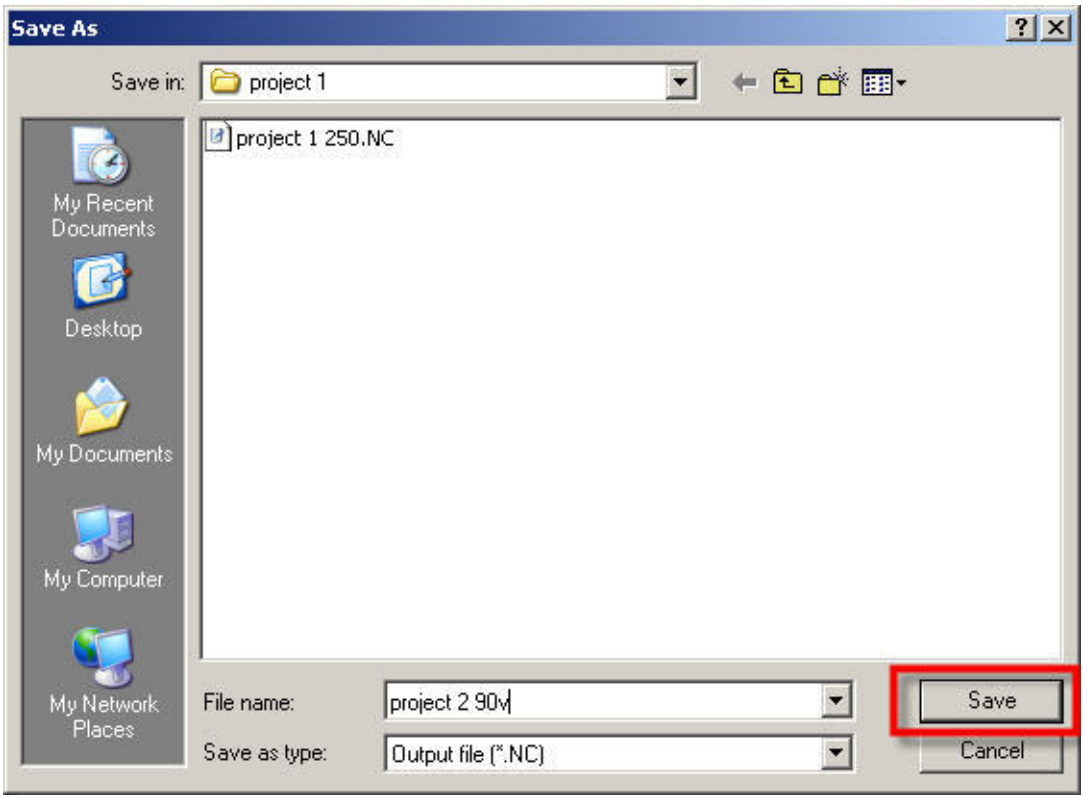
Perspective view of all toolpaths.



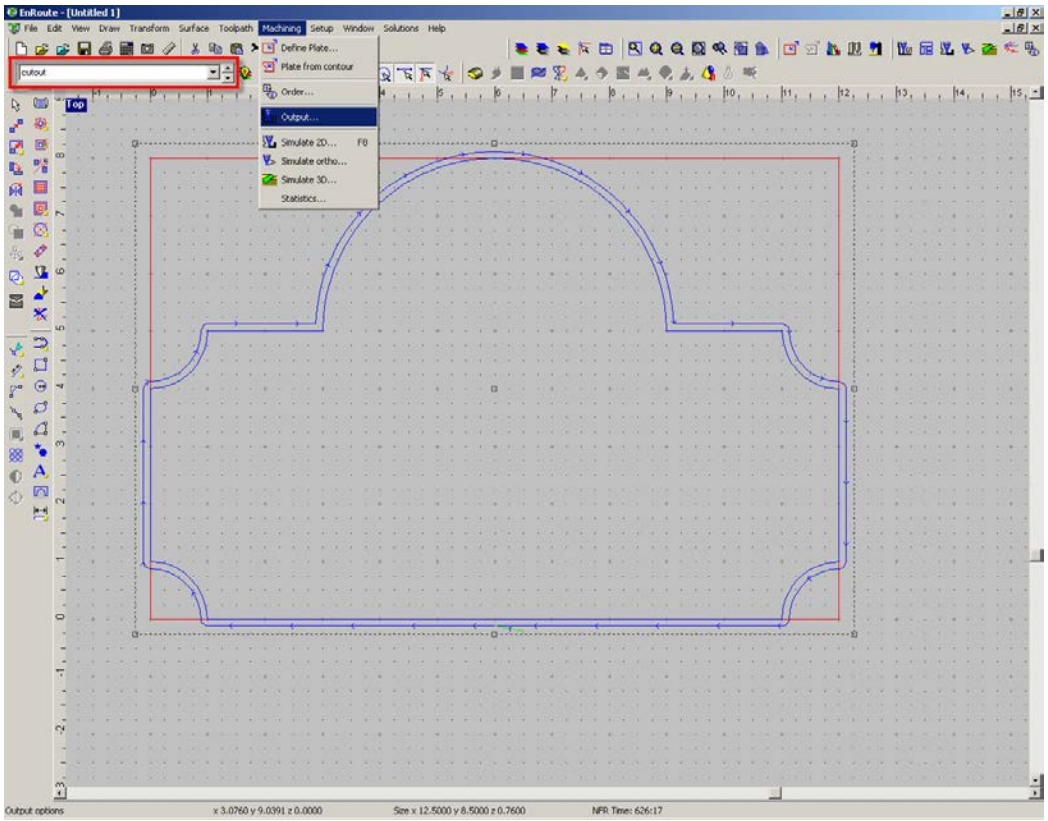
Time to output the files. Turn off the light bulb and go to the "engrave" layer. Go to "Machining > Output".



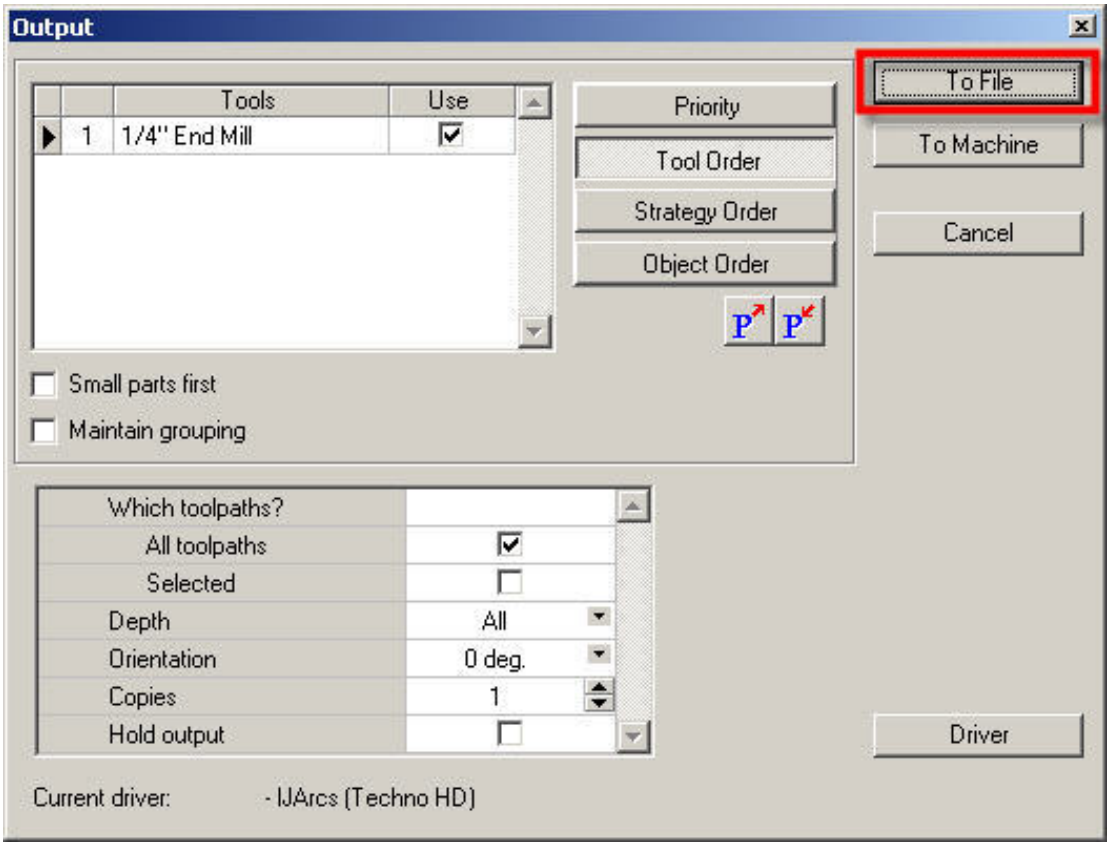
Click "To File".

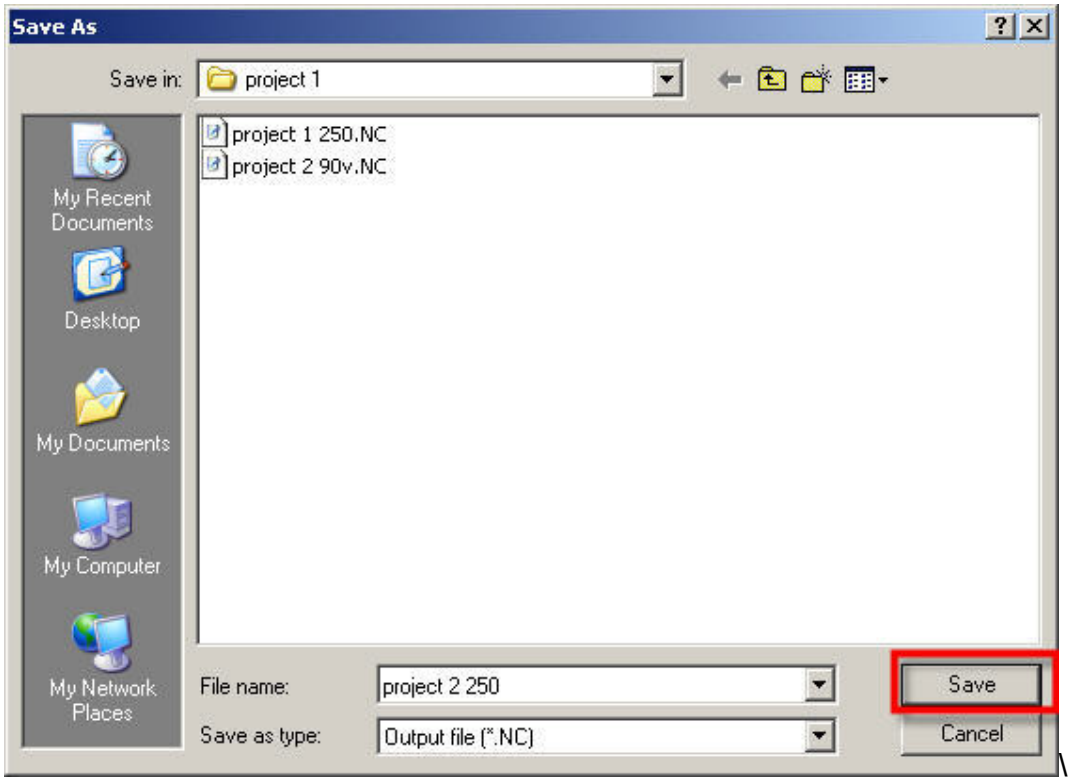


Name the file and “Save”.
I like to name the files with the cutter included so I know which is which.



Now go to the “cutout” layer and do the same.

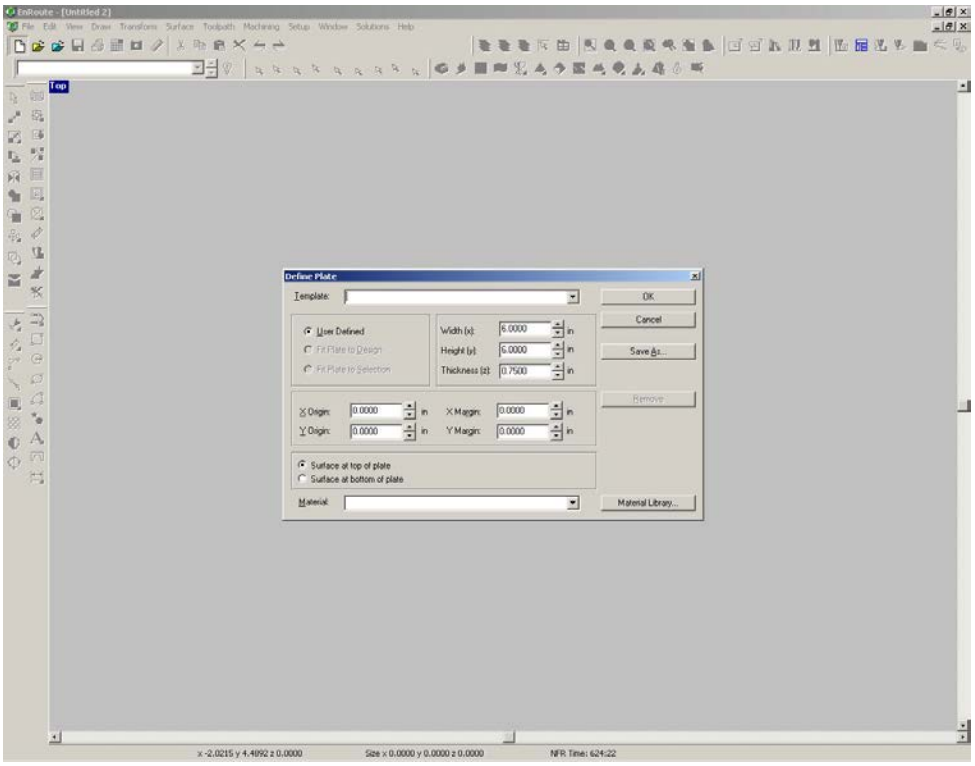




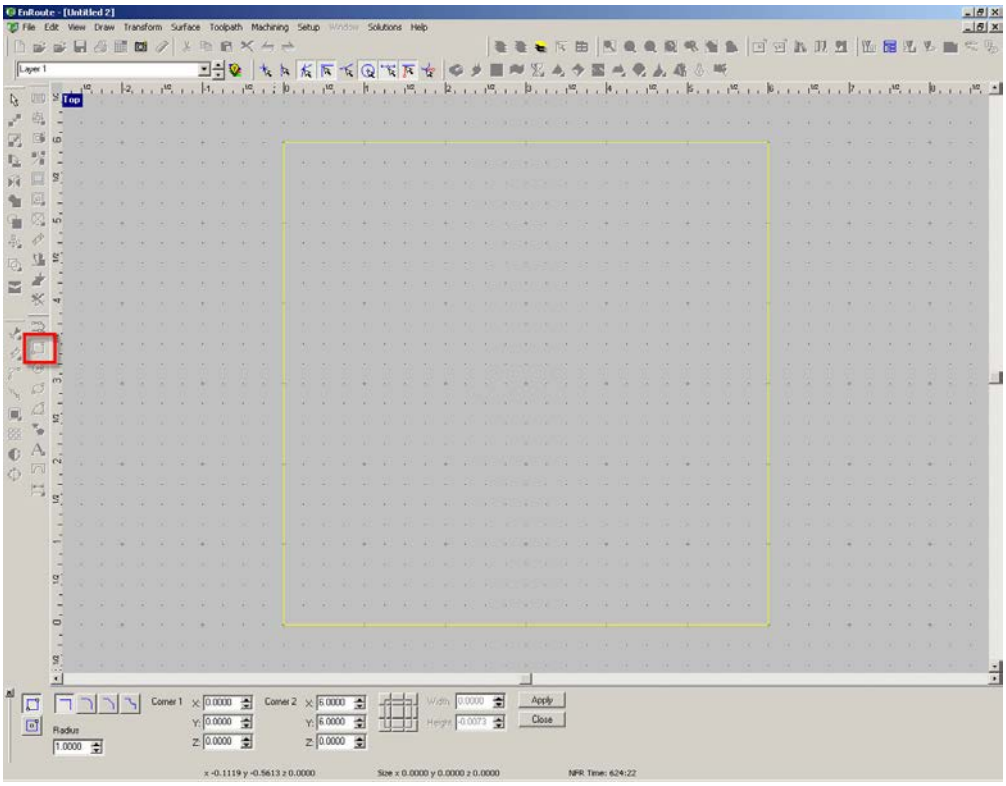
This project has 2 files that will go to the machine.
Run the engrave file first, and then the cutout.

EnRoute: Project 3

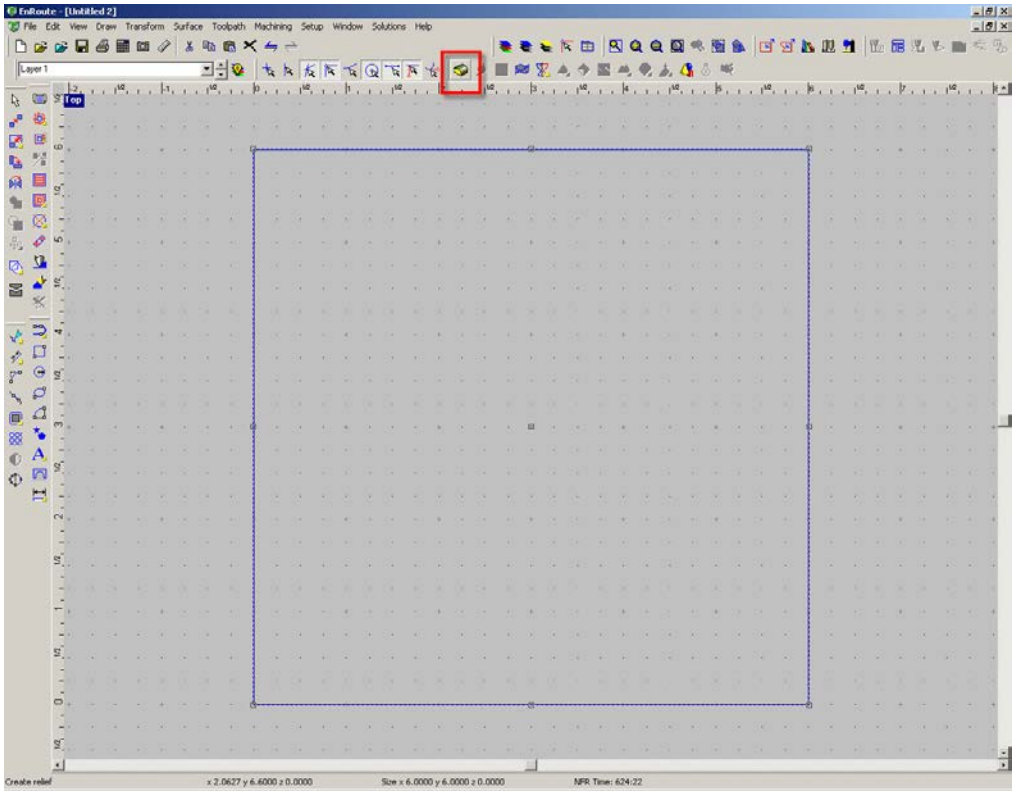
This section covers setting up a basic 3D texture.



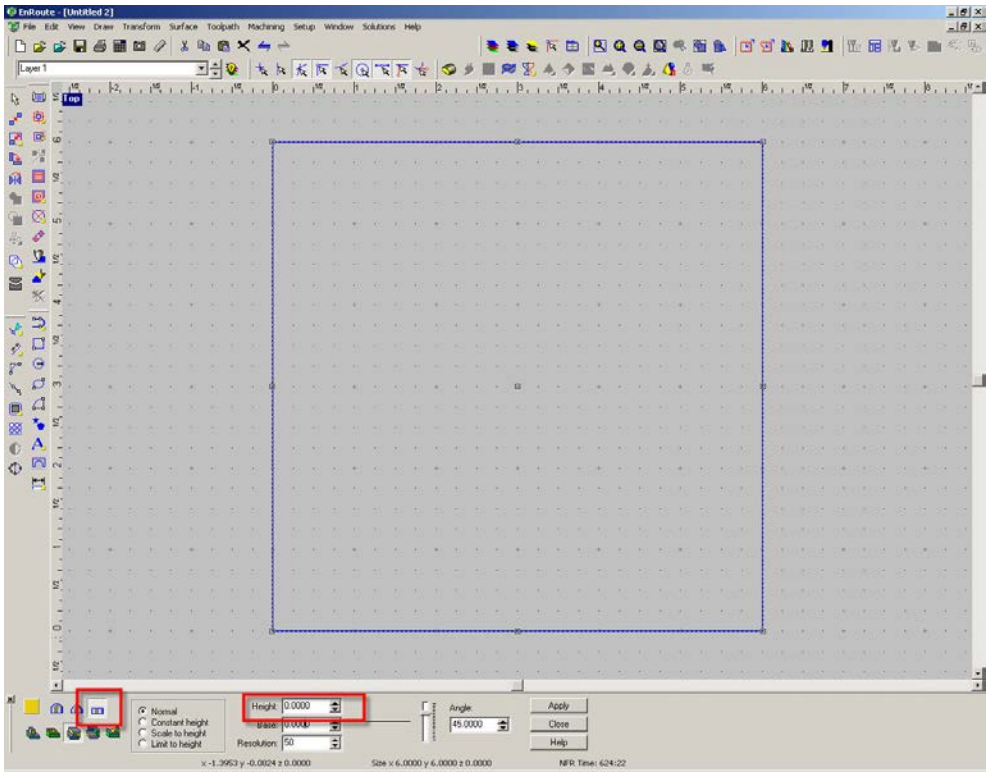
First, let's start with a 6" x 6" x .75" plate.



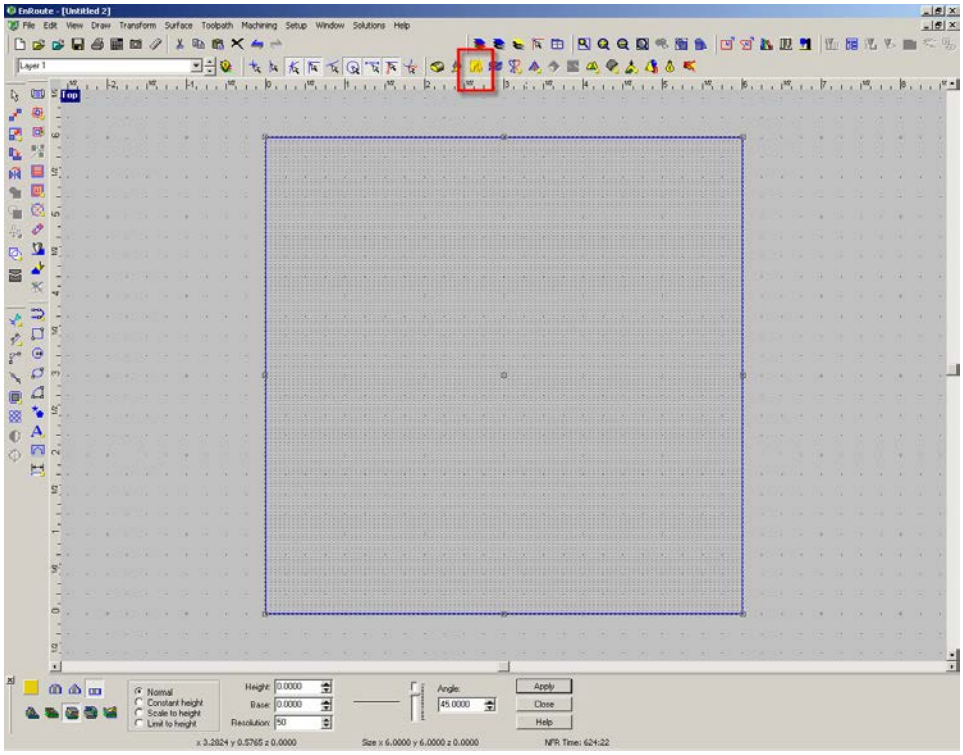
Draw a 6x6 square.



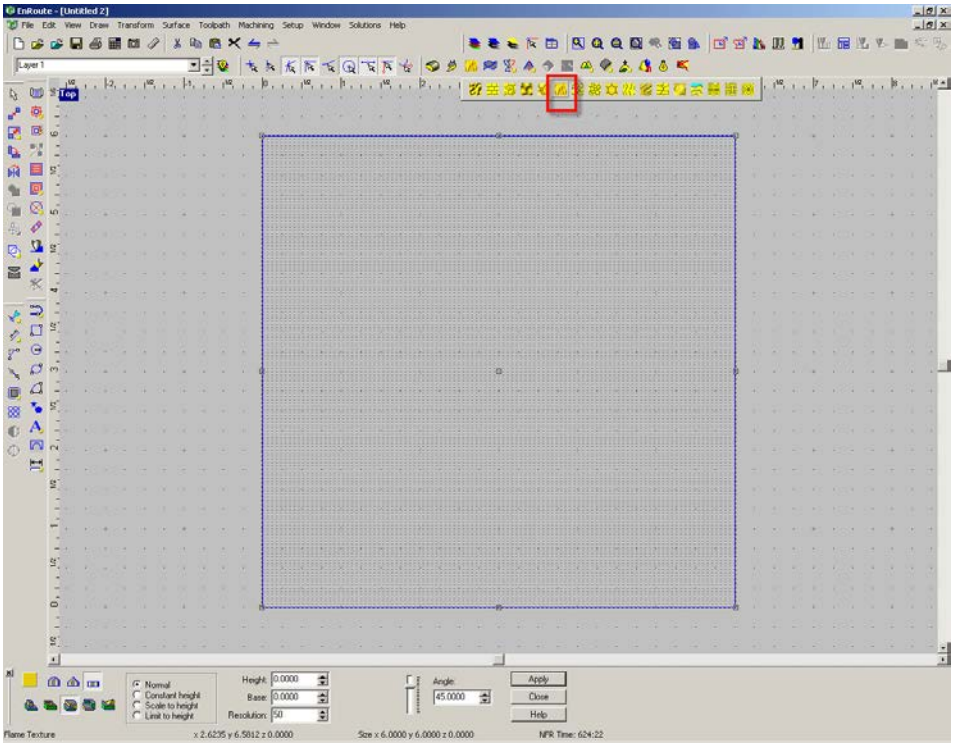
Highlight the square and click the “Create Relief” icon.



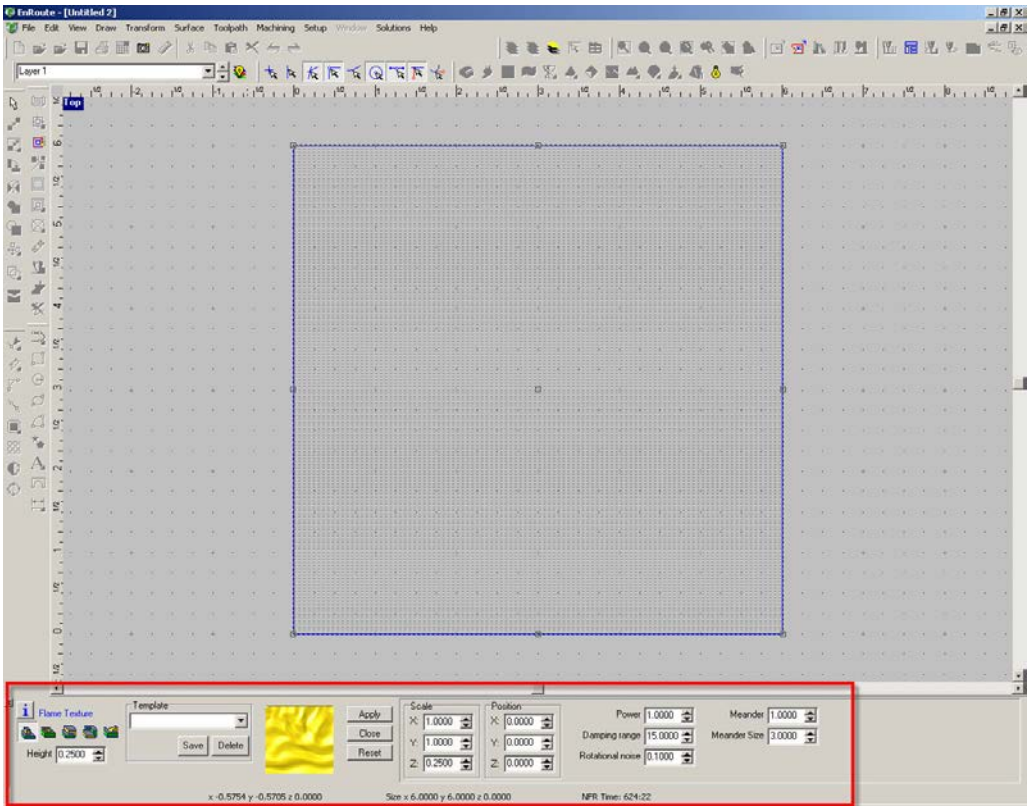
Create a flat, zero height relief.



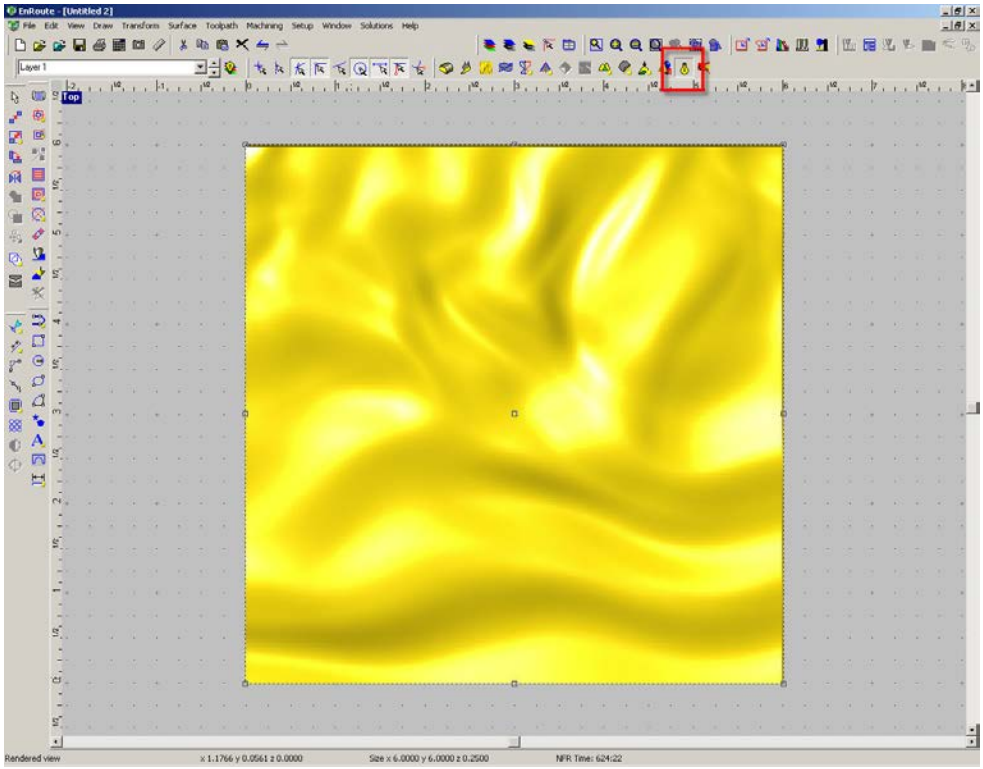
Click and hold the “Texture” icon to expand the toolbar.



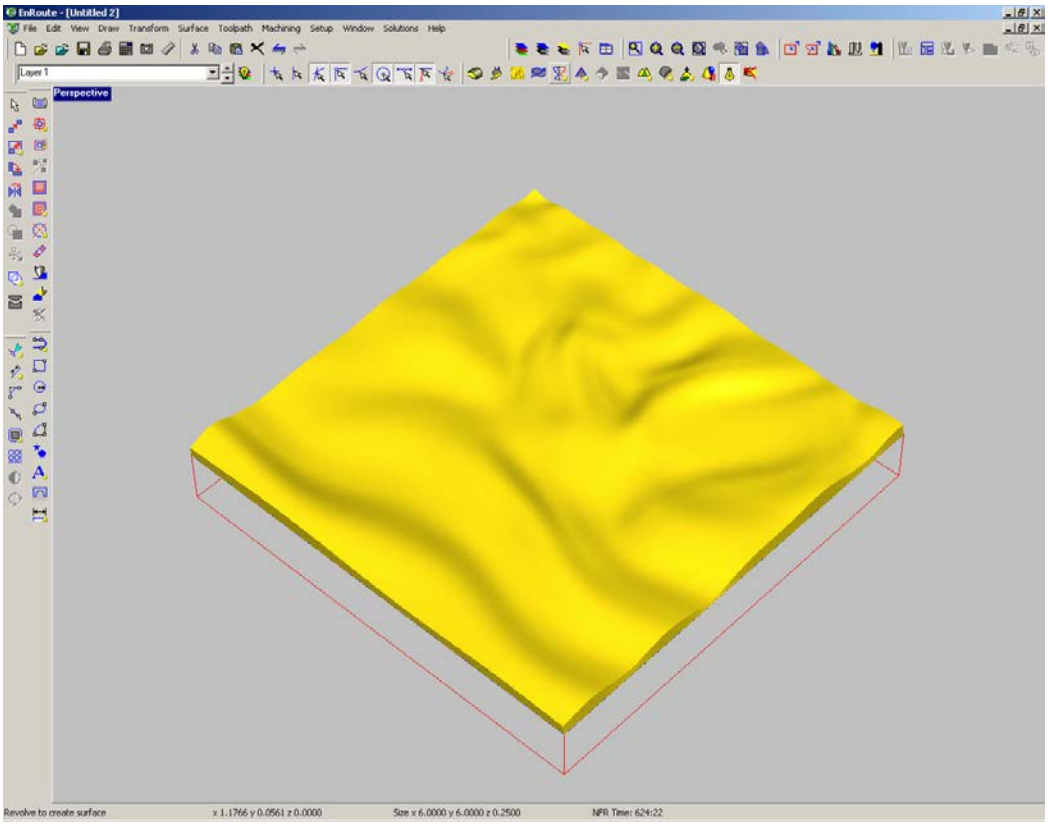
Select the “flame” texture.



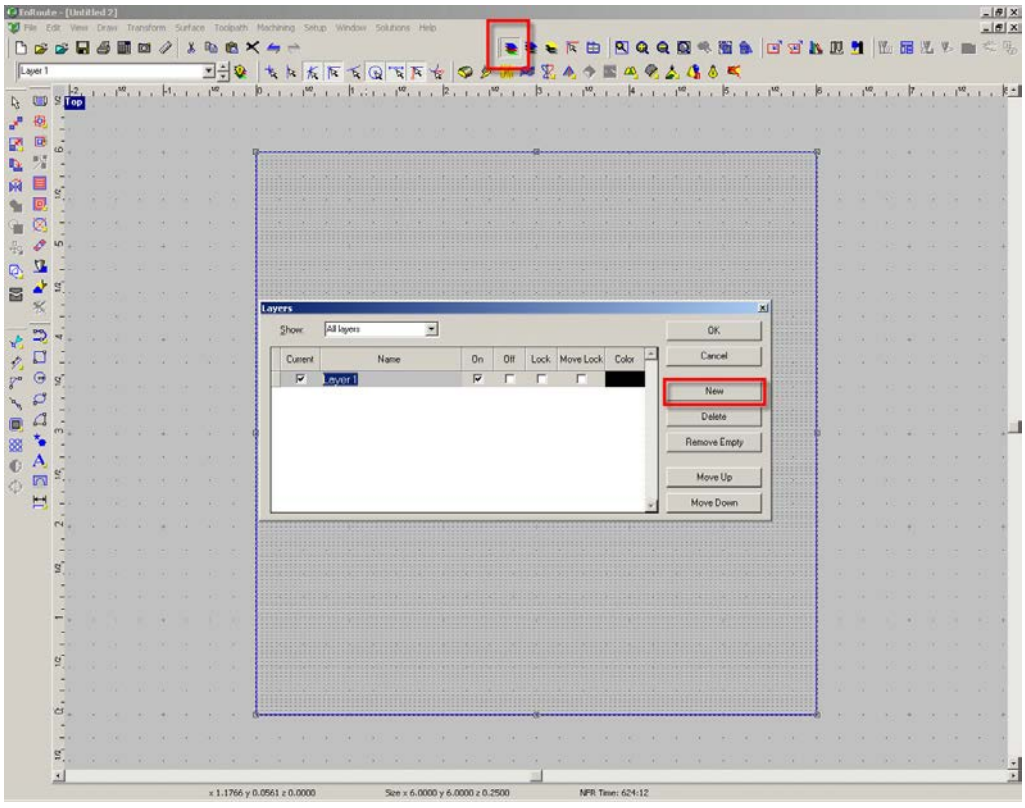
Adjust the parameters to change the texture as desired and click “Apply”.



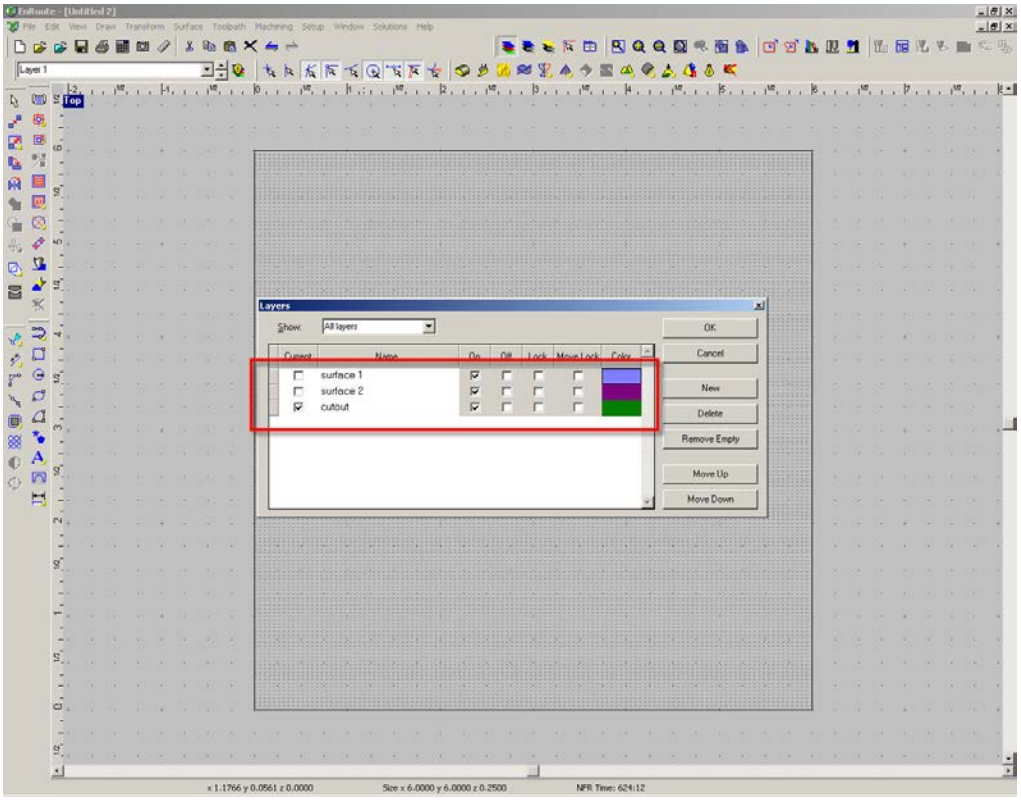
Click the light bulb on the surface toolbar to render the surface.



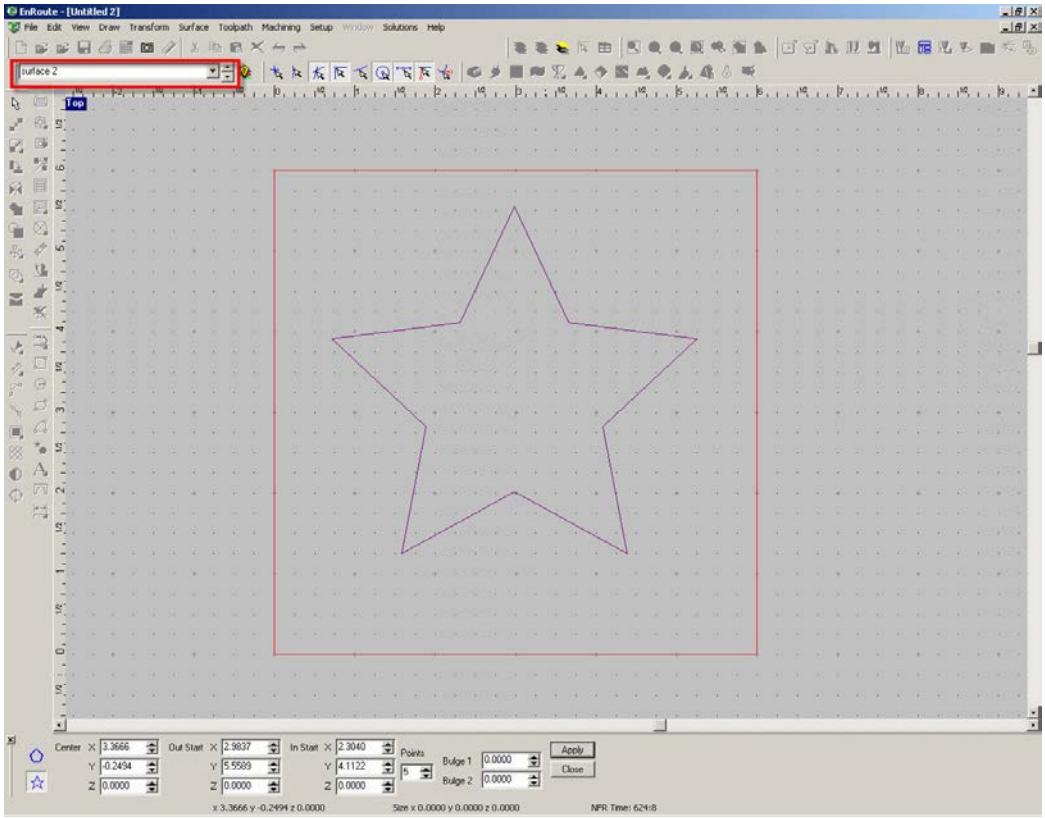
Perspective view of the texture.



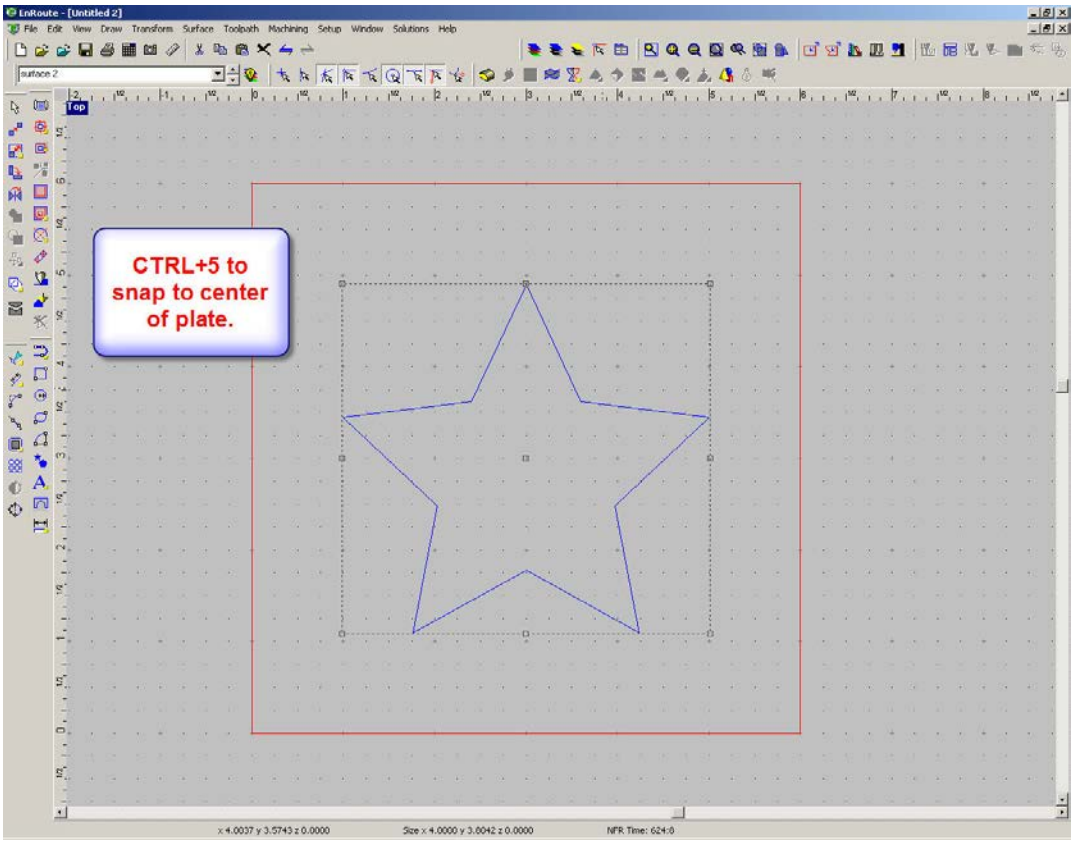
Now, let's add some layers.



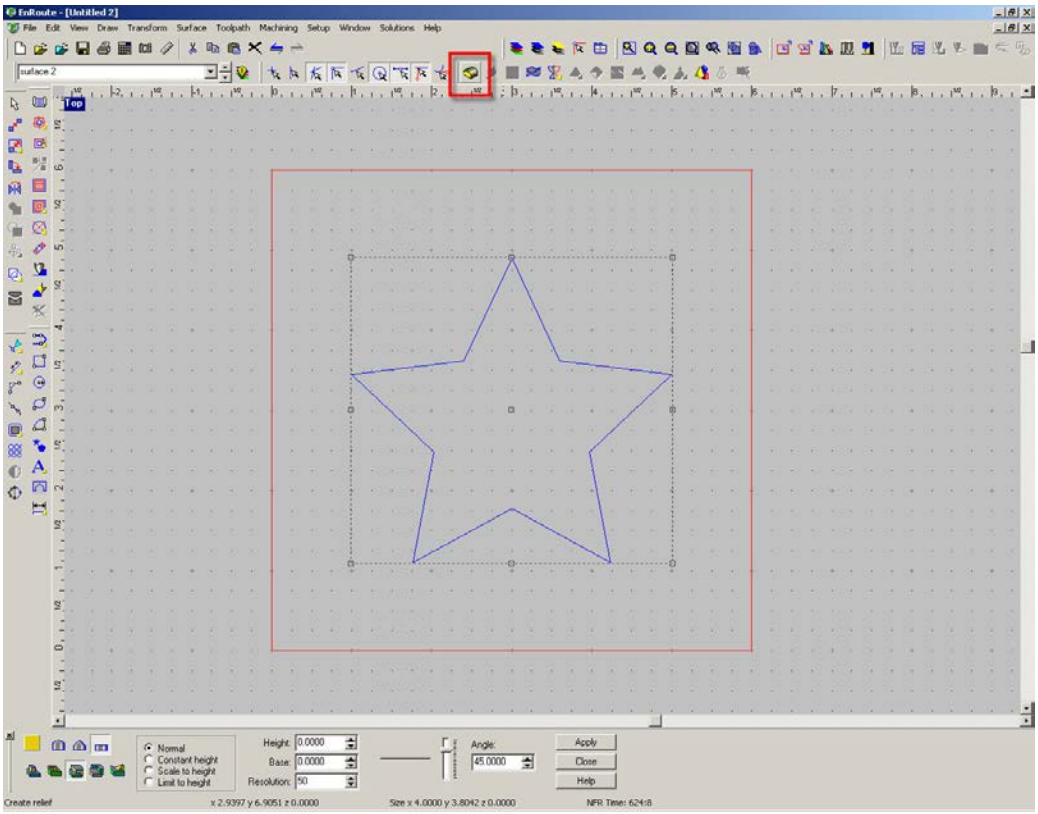
I'm going to add 2. Rename them to "surface 1", "surface 2" and "cutout".



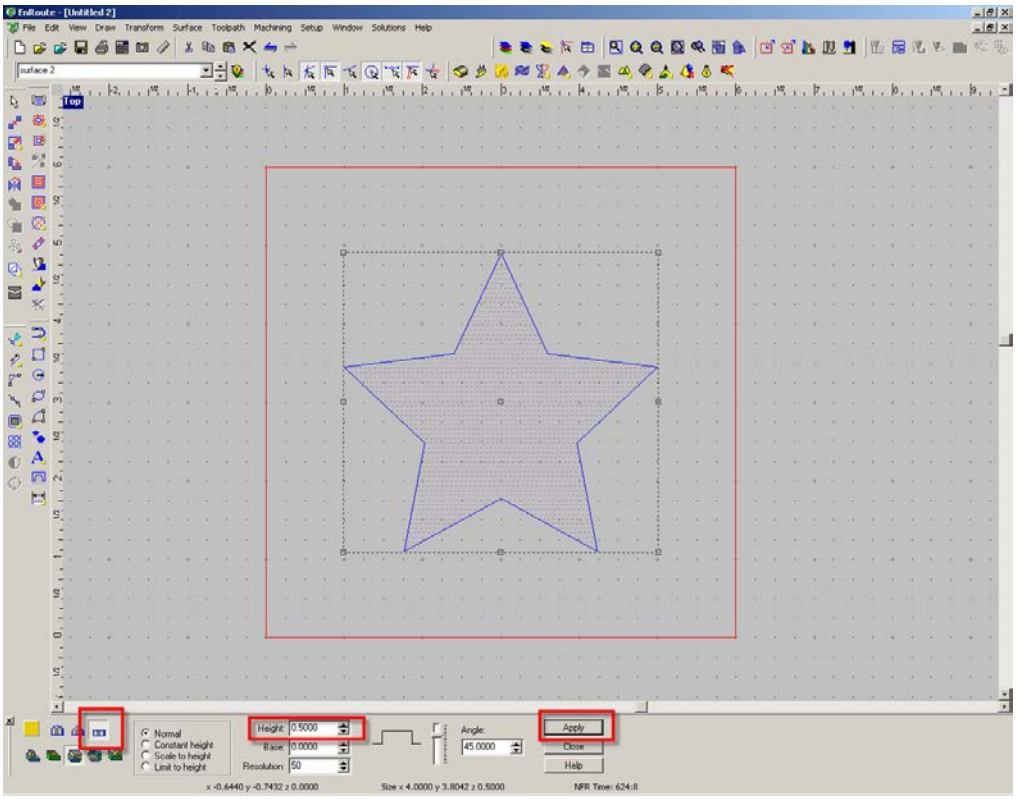
Go to the "surface 2" layer and use the "Draw Polygon" tool to create a star.



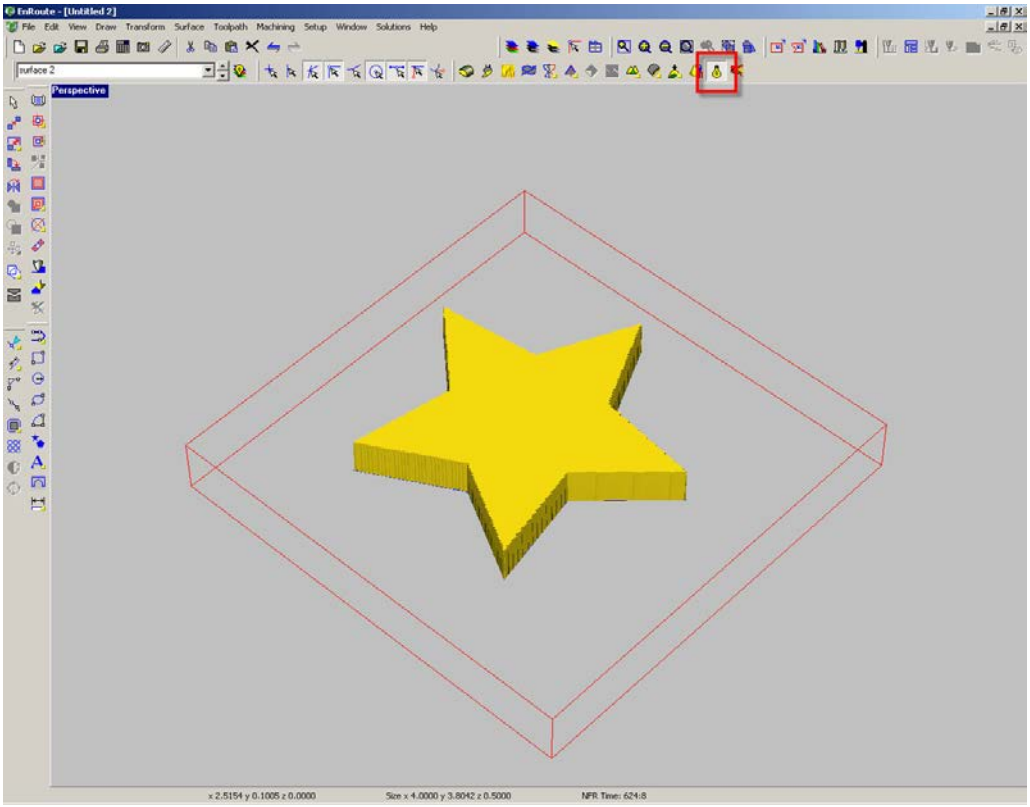
Center it in the plate.



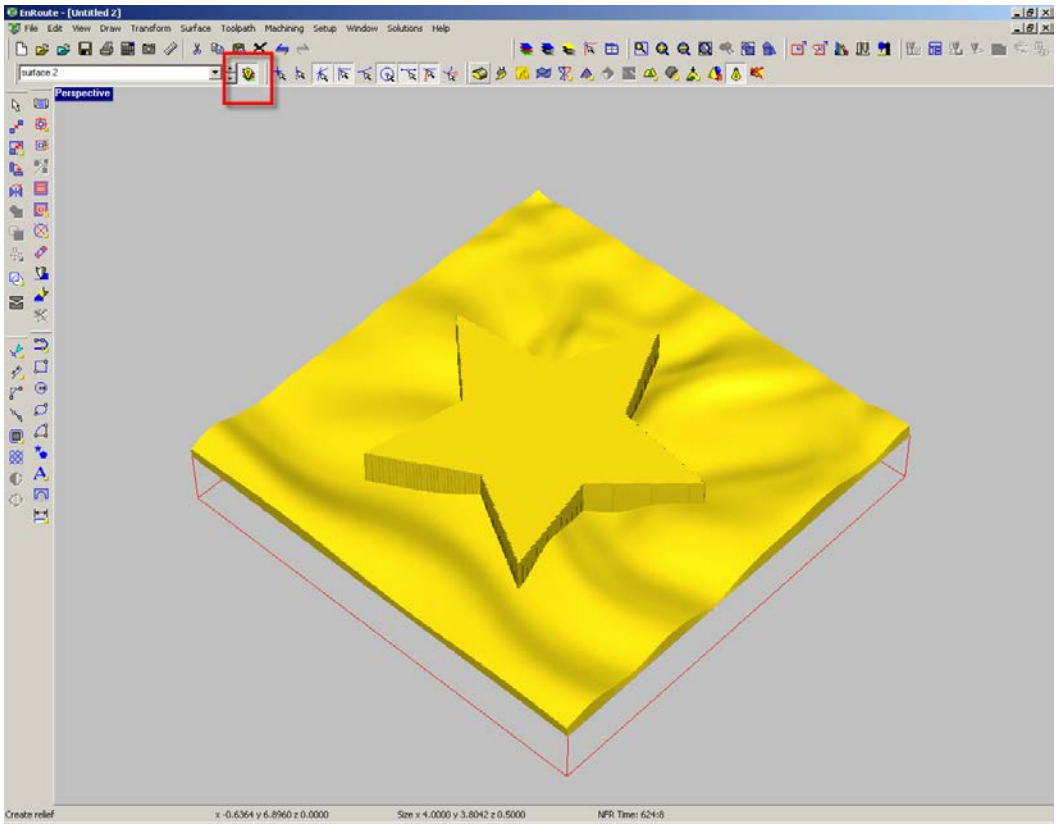
Highlight the start and click “create relief” again.



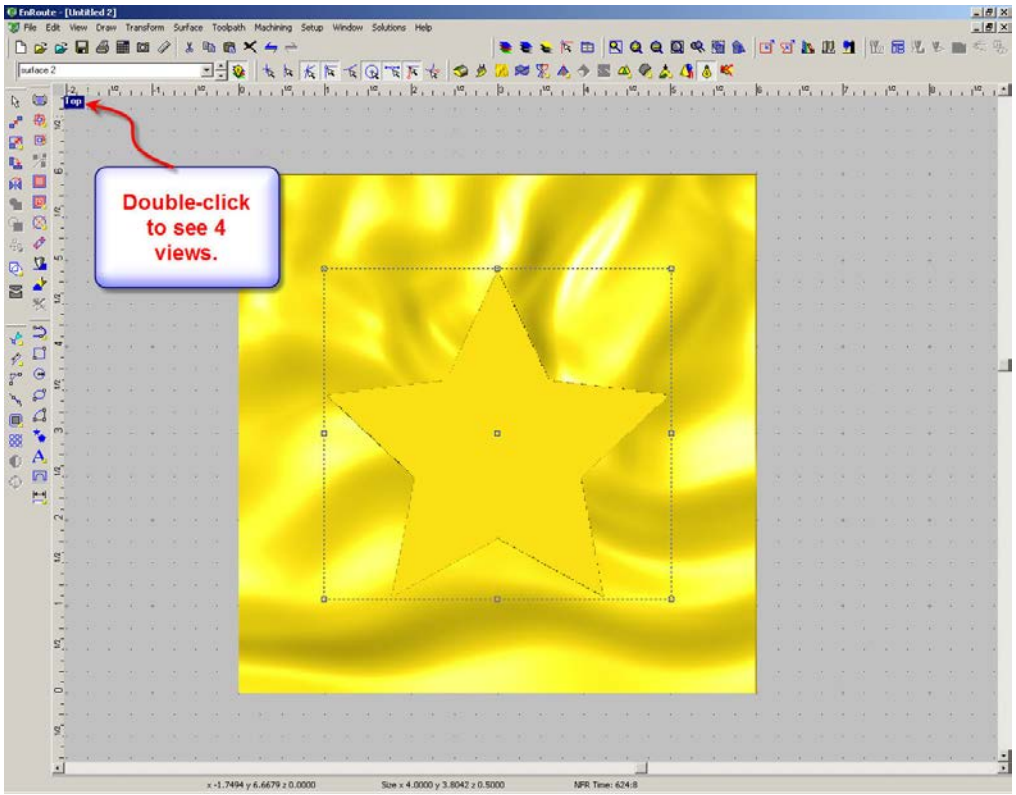
We're going to do another flat surface, only this time we'll give it a .5" height.



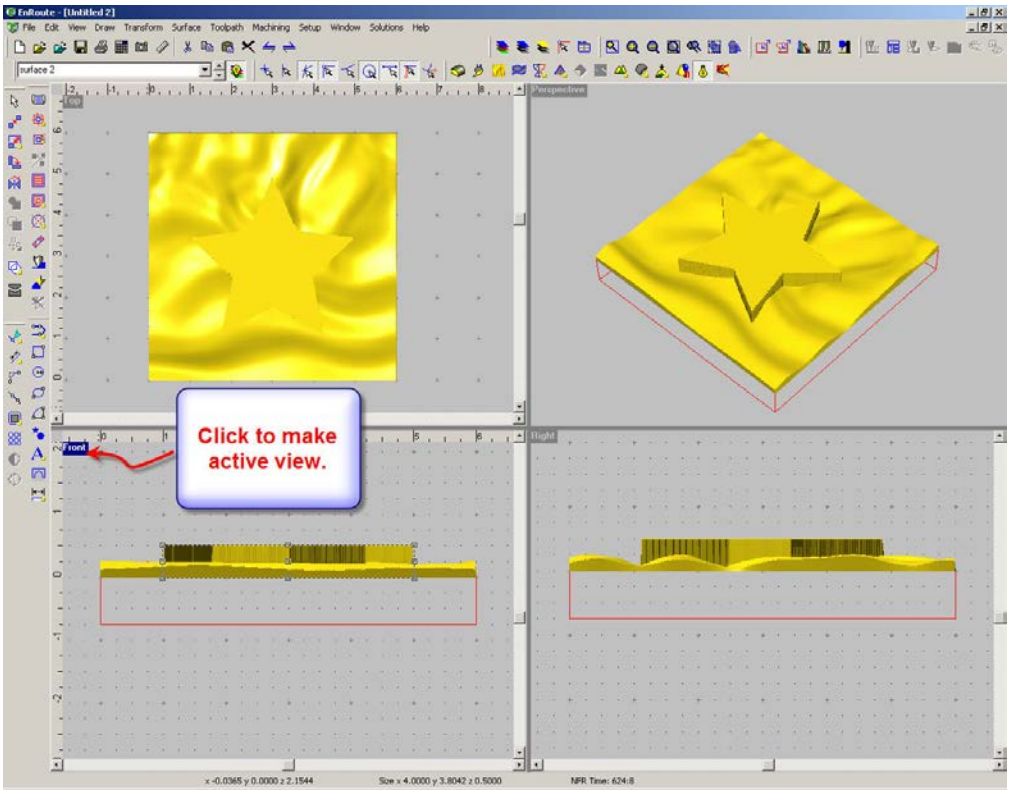
Rendered perspective view of Surface 2.



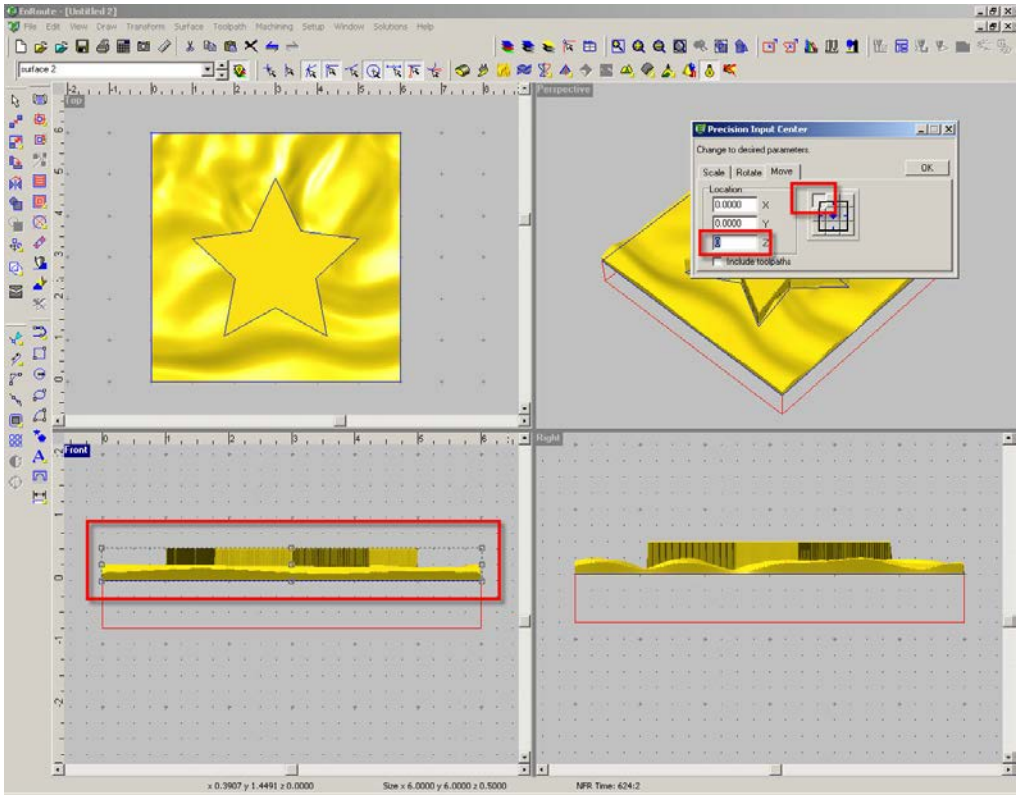
Both surfaces together.



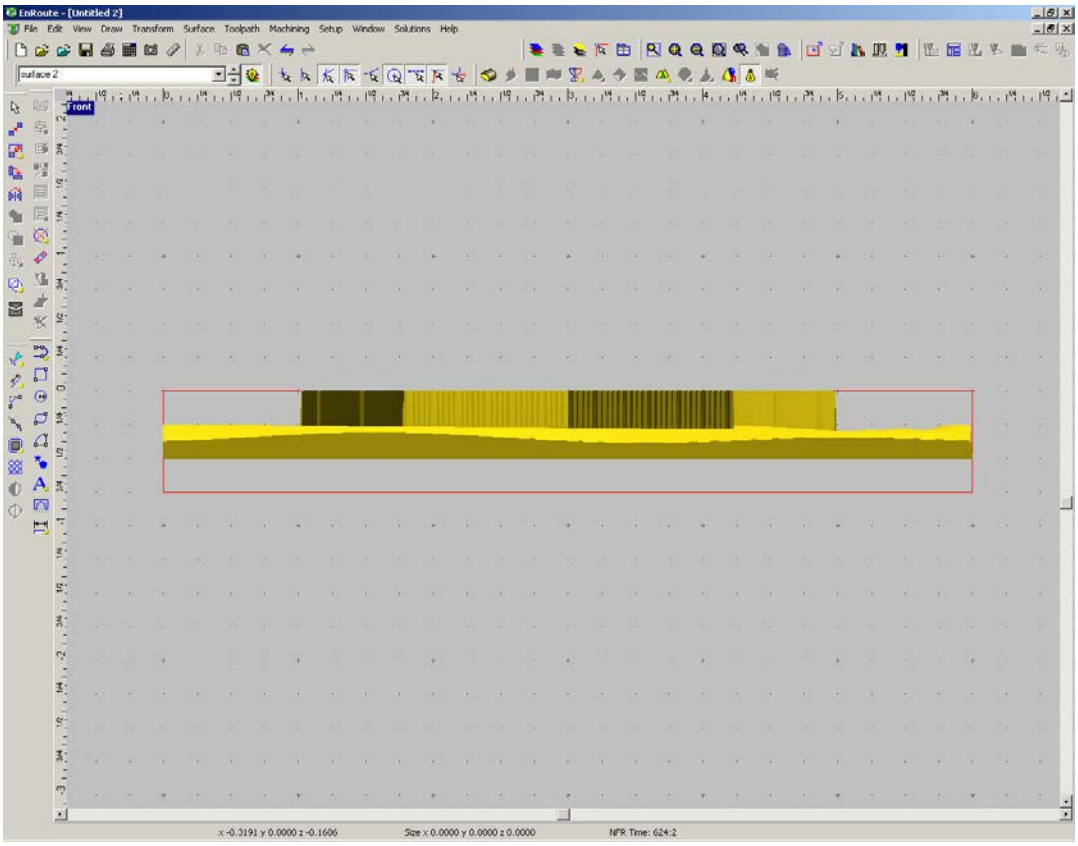
Double-click on the view name to make all views visible.



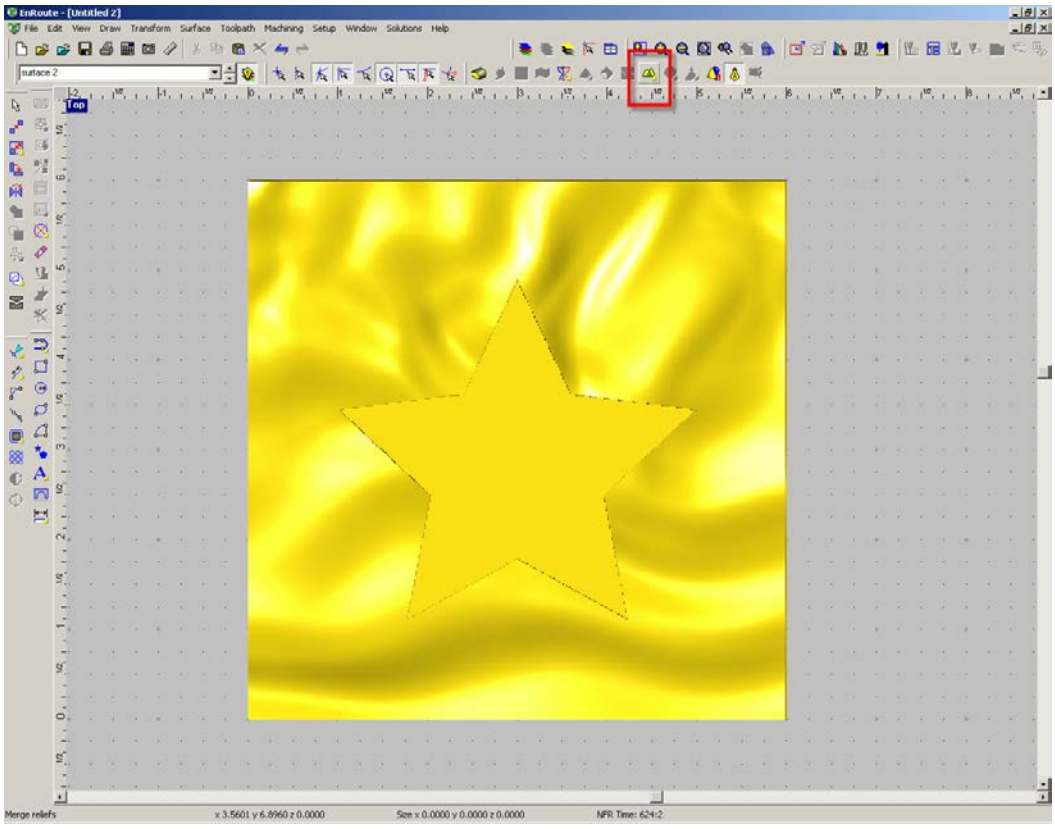
Click on the front view so that the name is highlighted.



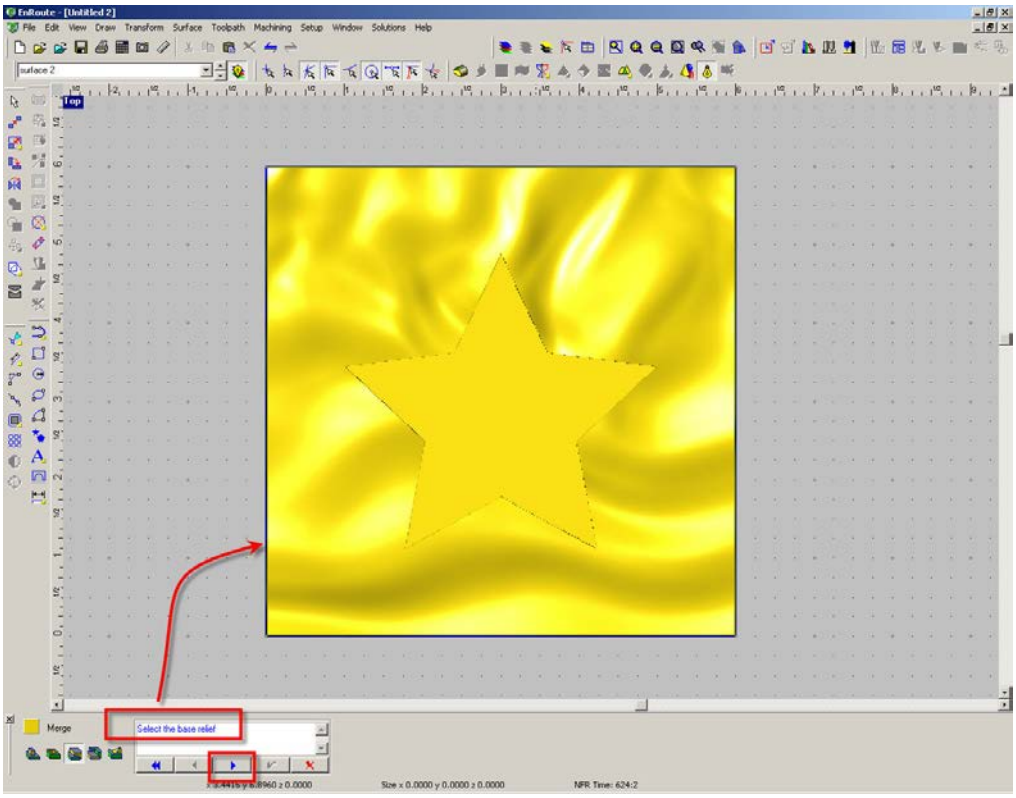
Now open the PIC (F2), select both surfaces together in the front view and move the top of the surfaces to Z=0.



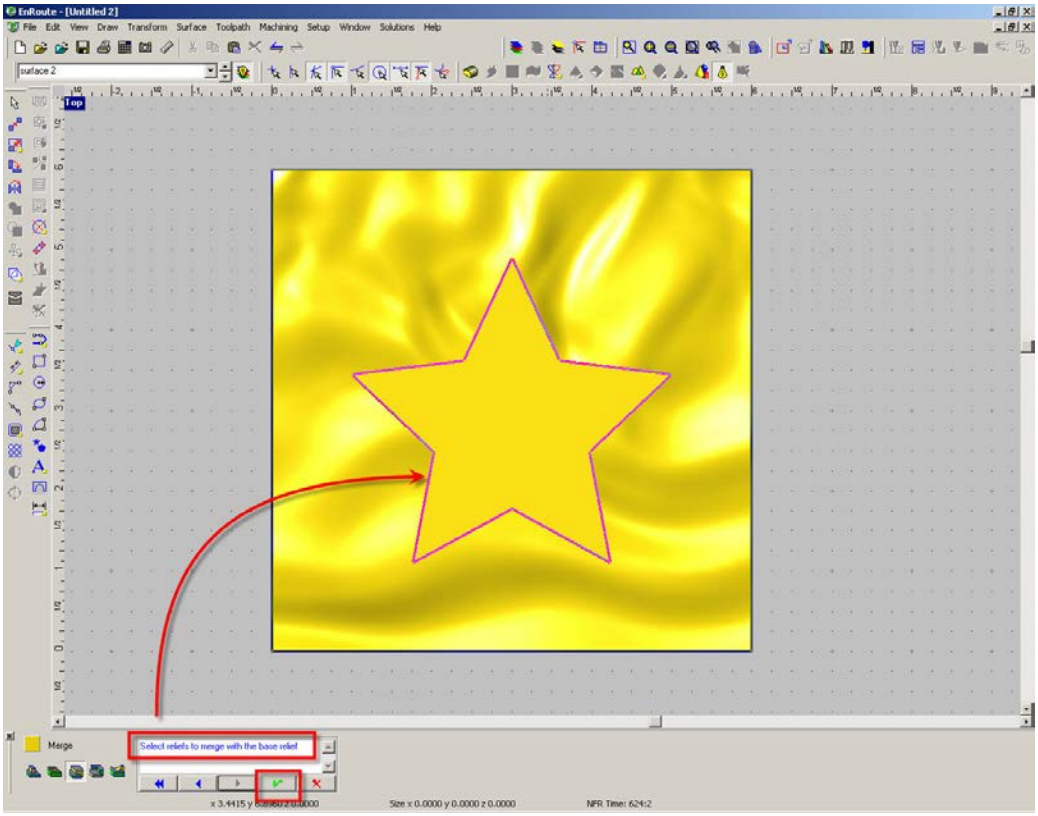
The surfaces moved into the plate.



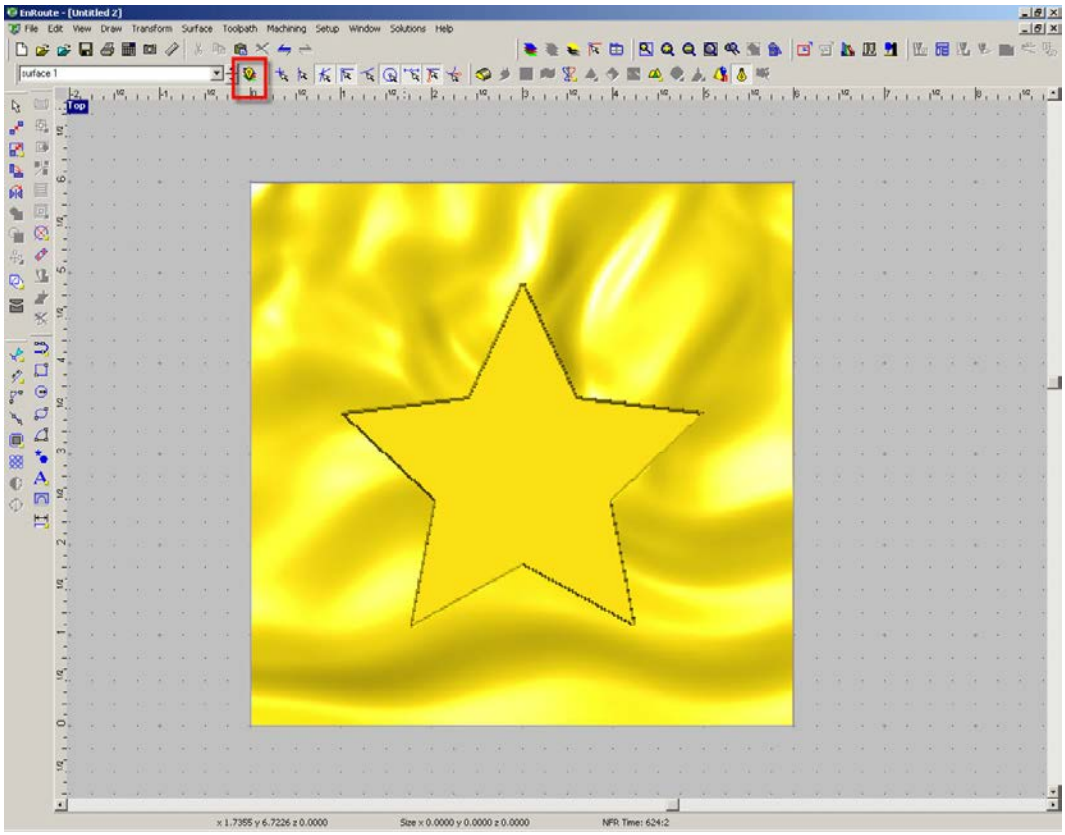
Go back to the top view. Now we can merge the two reliefs together.



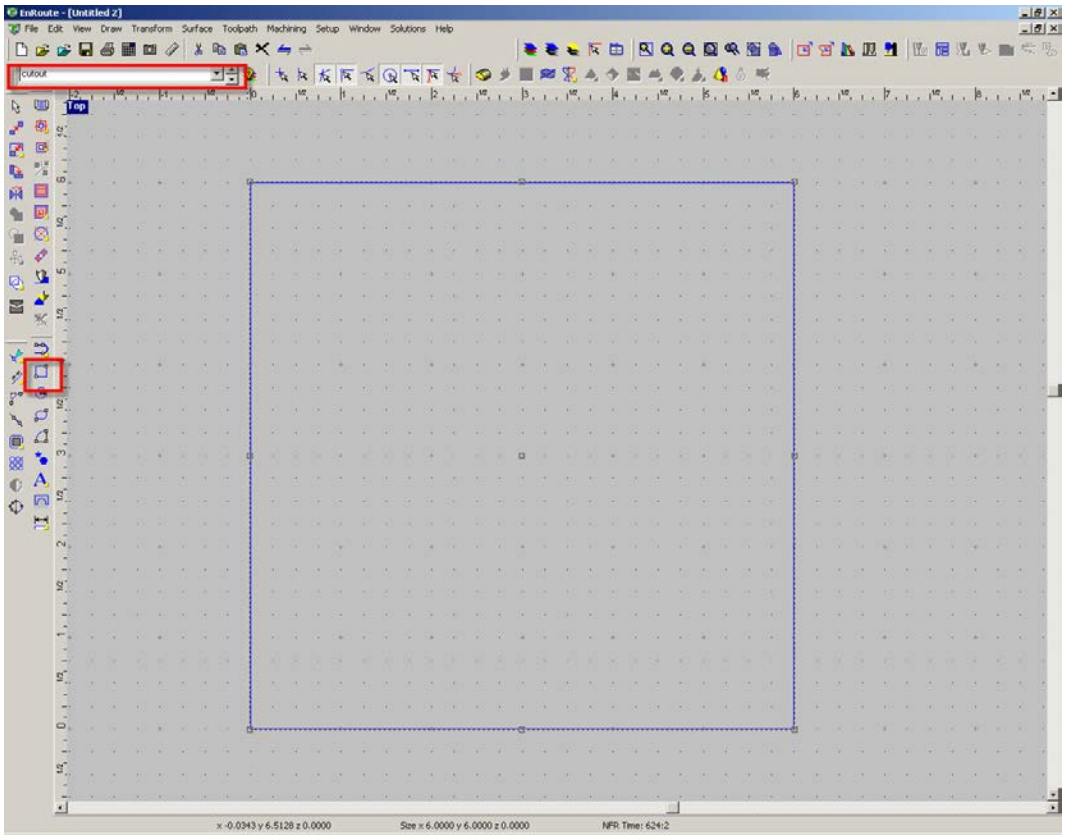
Click the “Merge Reliefs” tool and follow the instructions.
First, select the base relief then click the blue arrow.



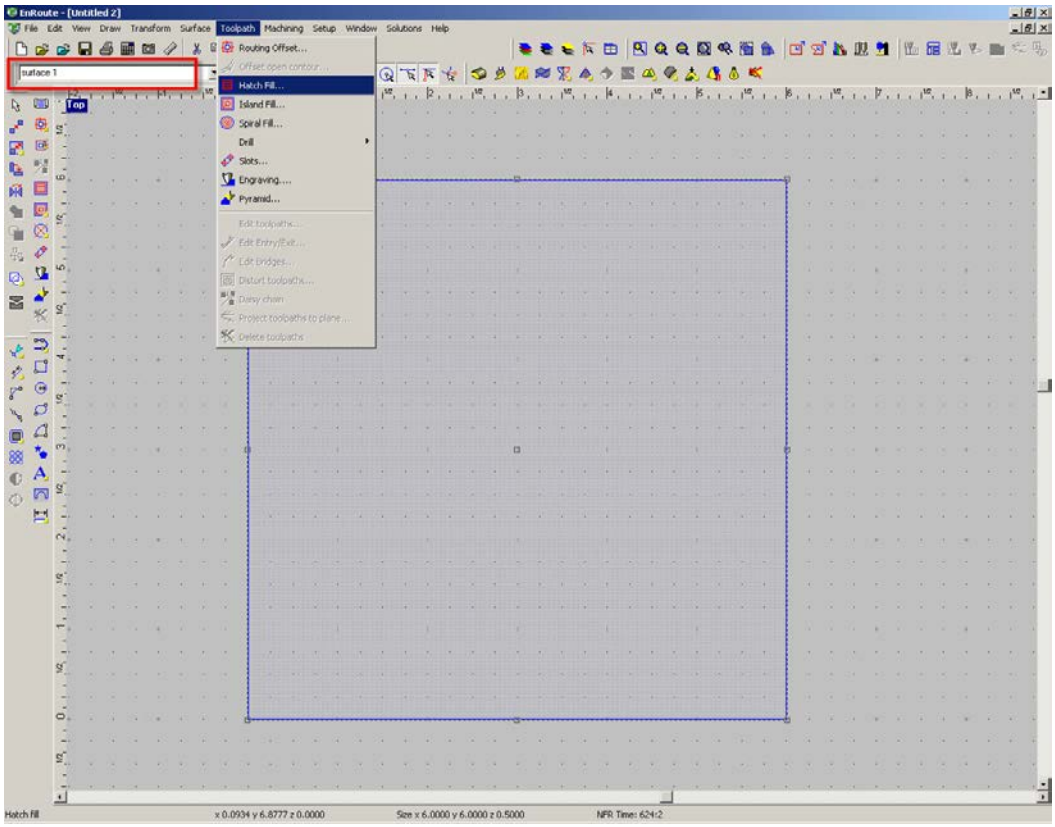
Then select the relief to be added and click the green check.



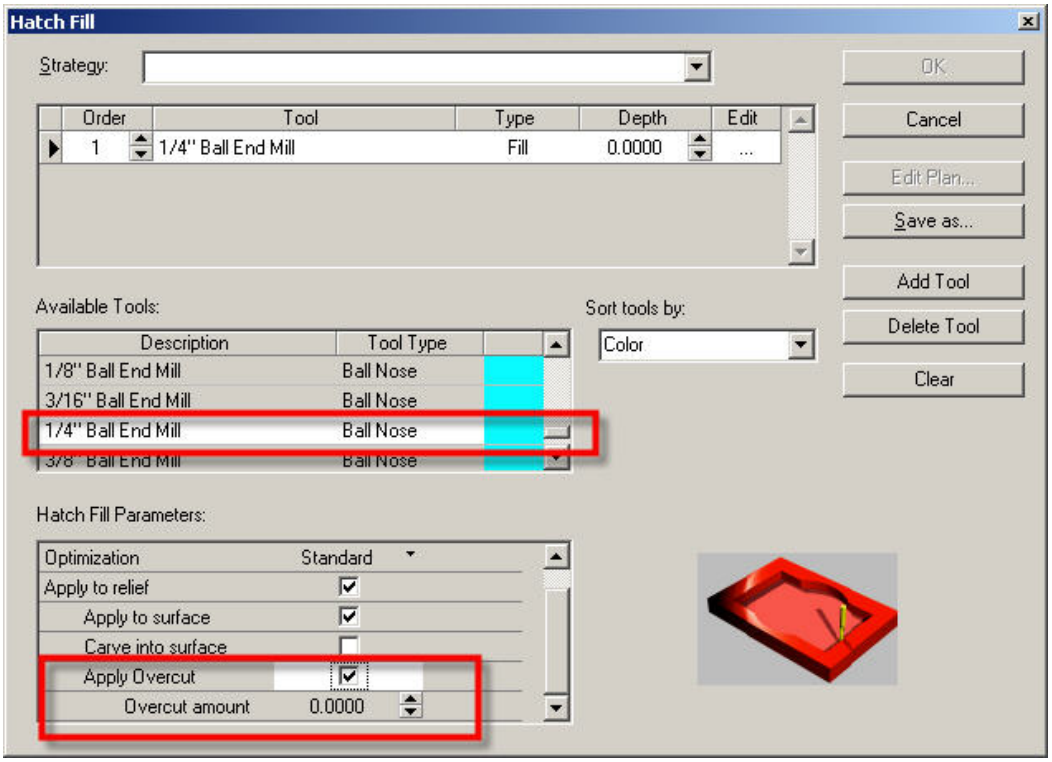
The reliefs are now combined together on layer 1. We don't need the "surface 2" layer anymore.



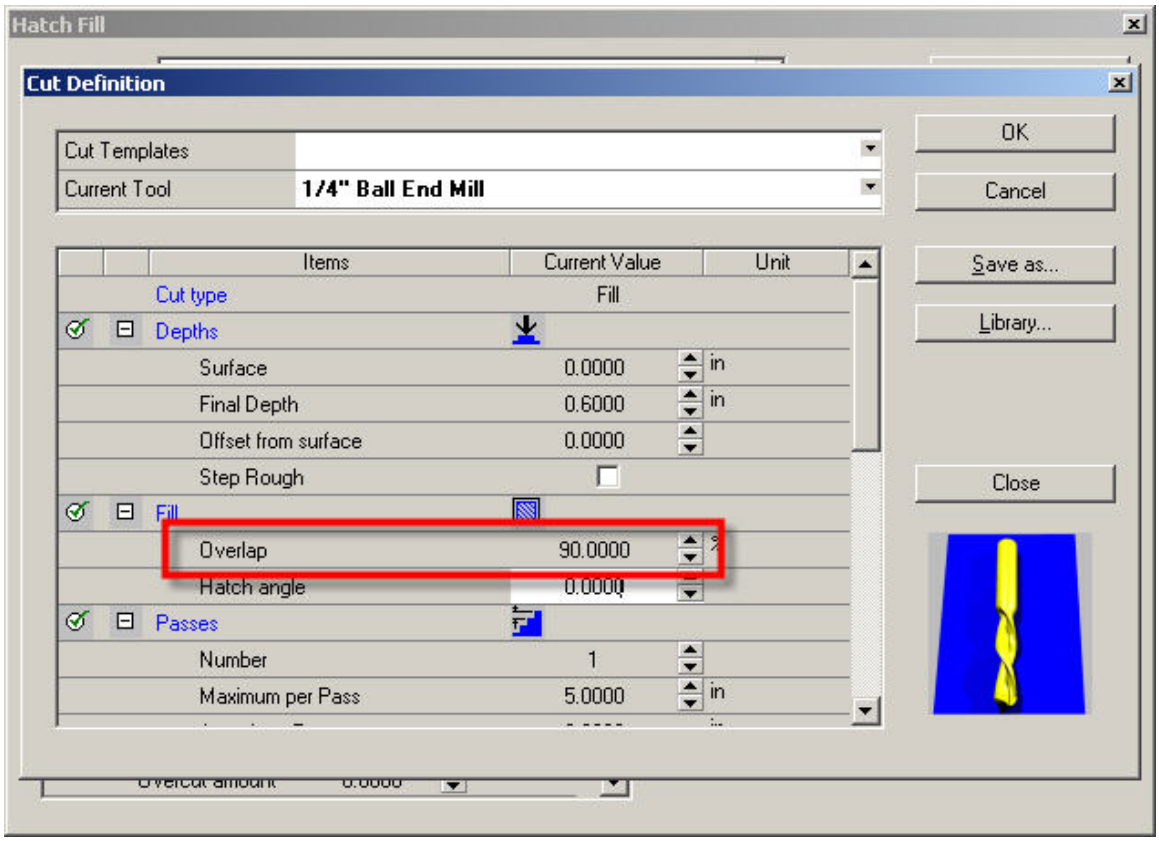
Go to the "cutout" layer and draw a 6x6 box like we started with.



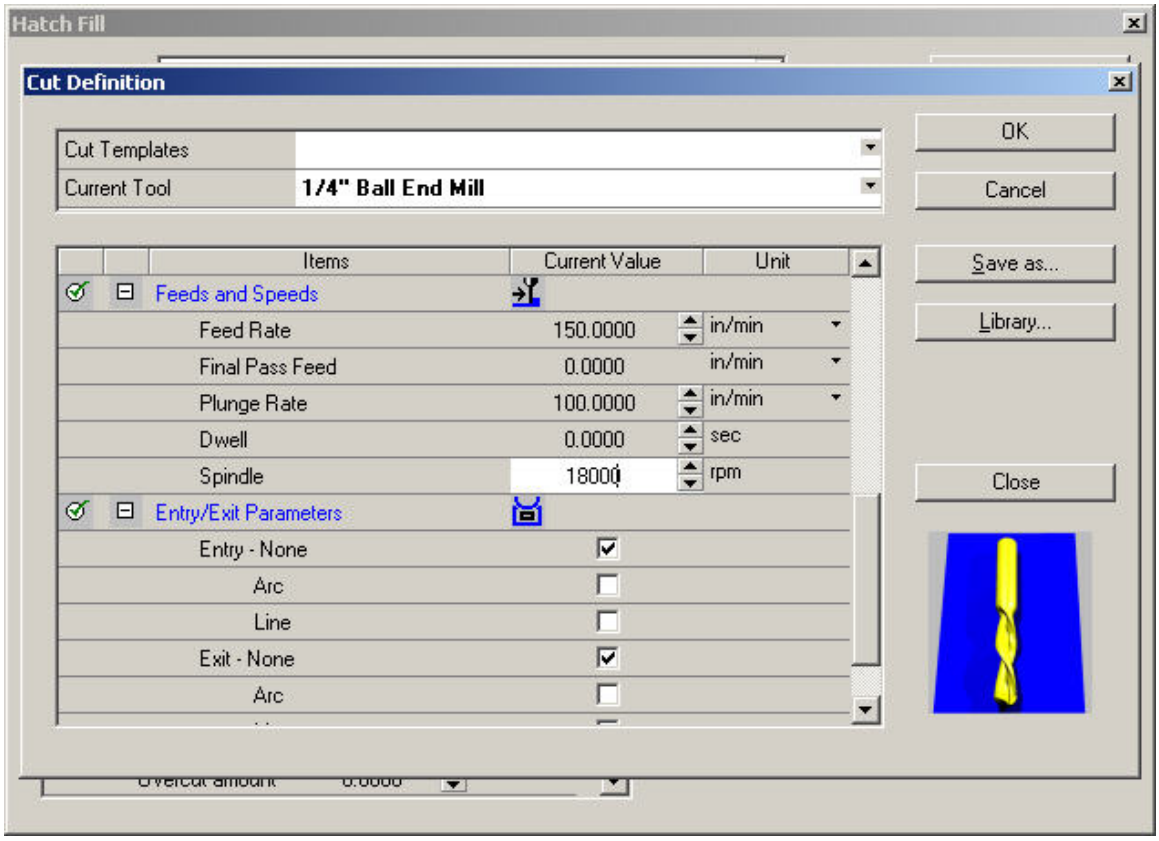
Now go back to the surface layer and select a “Hatch Fill” tool path.



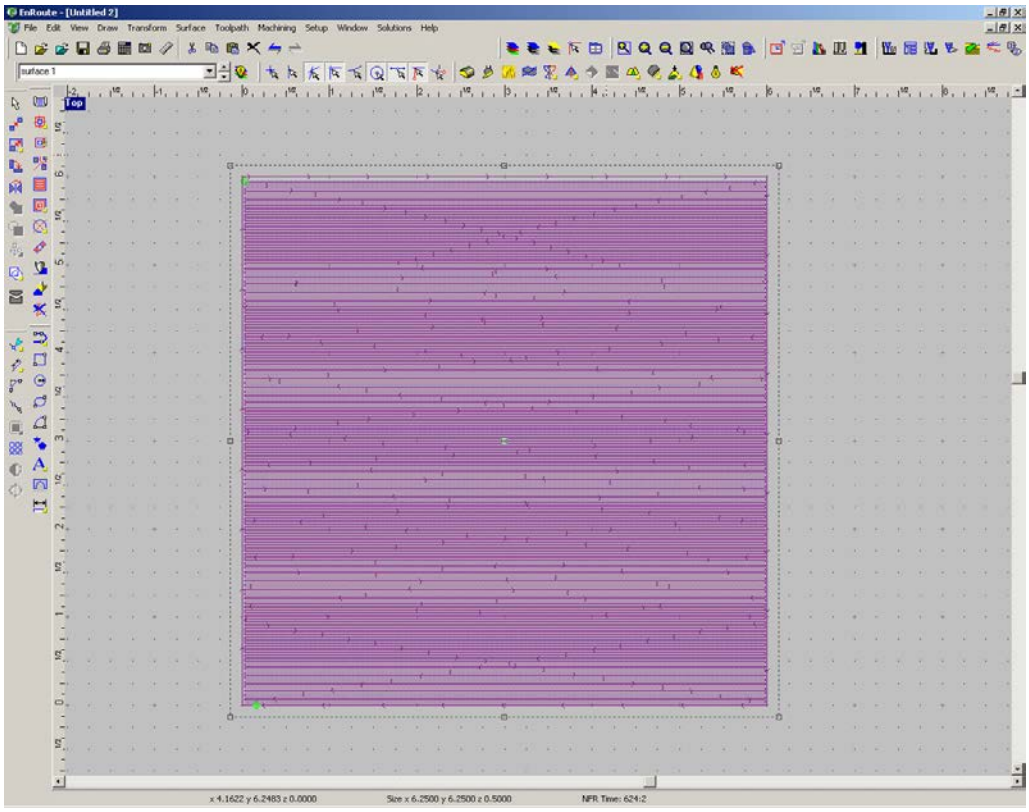
For any 3D surface, you want to use a ball end mill.
You can apply an overcut so that the cutter reaches the edge of the surface.



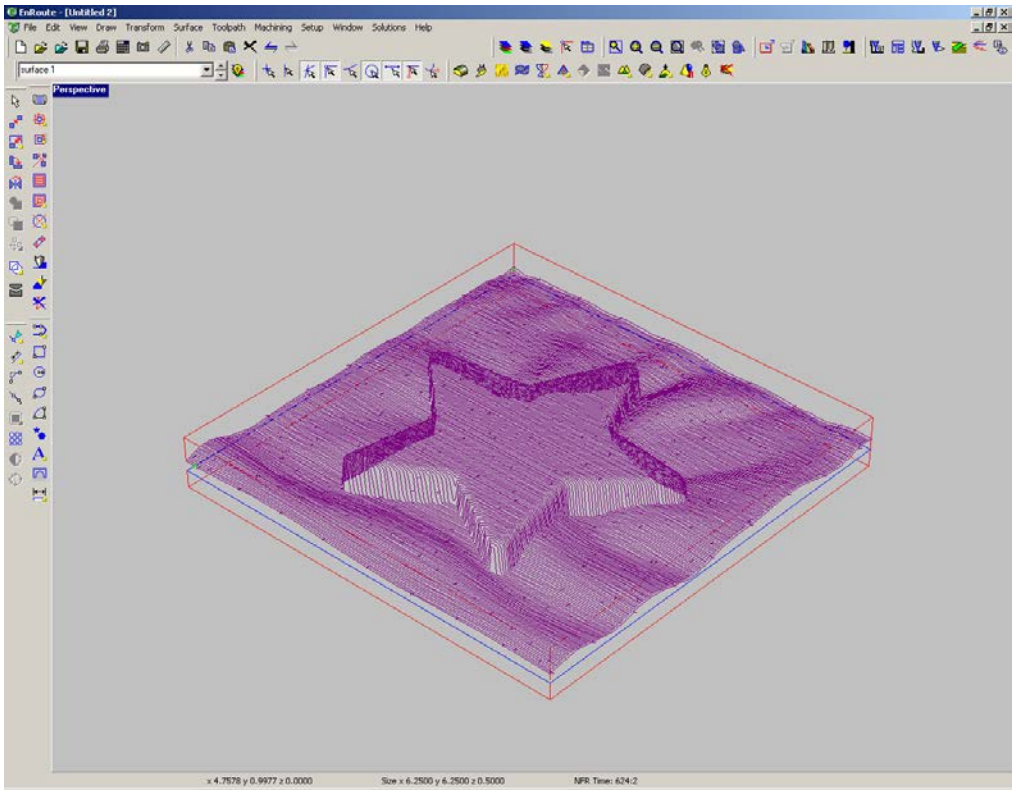
A large overlap is important for 3D – otherwise, you will see a scalloped surface.



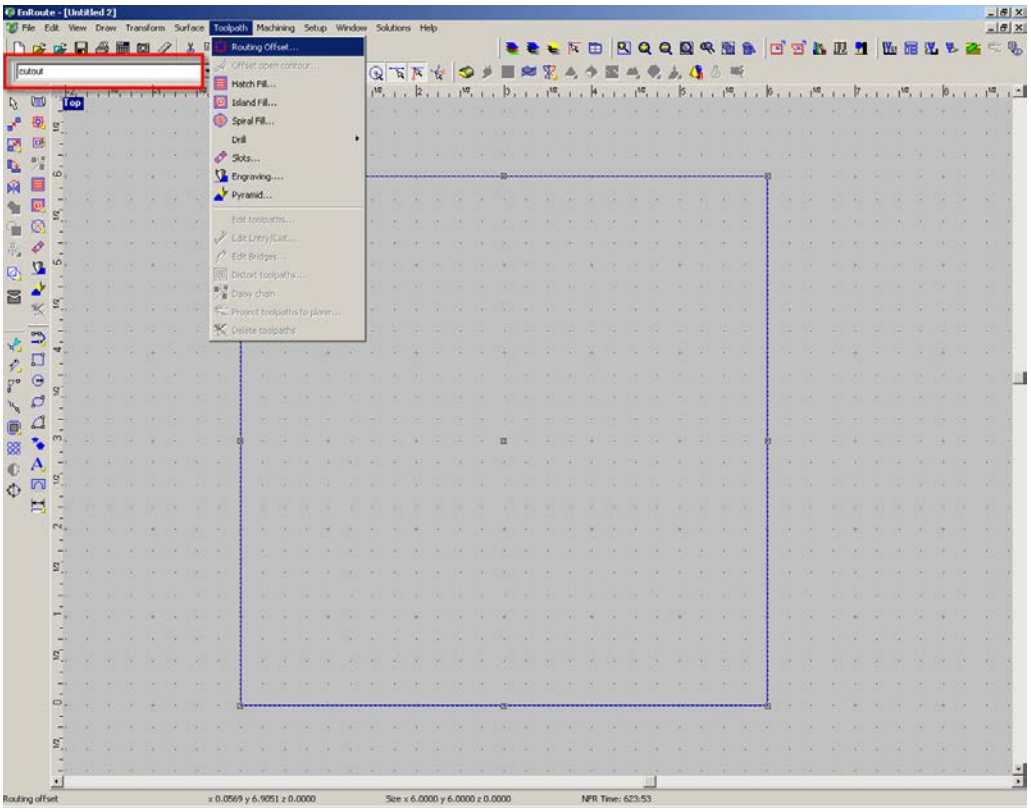
Fill in the speeds as usual.



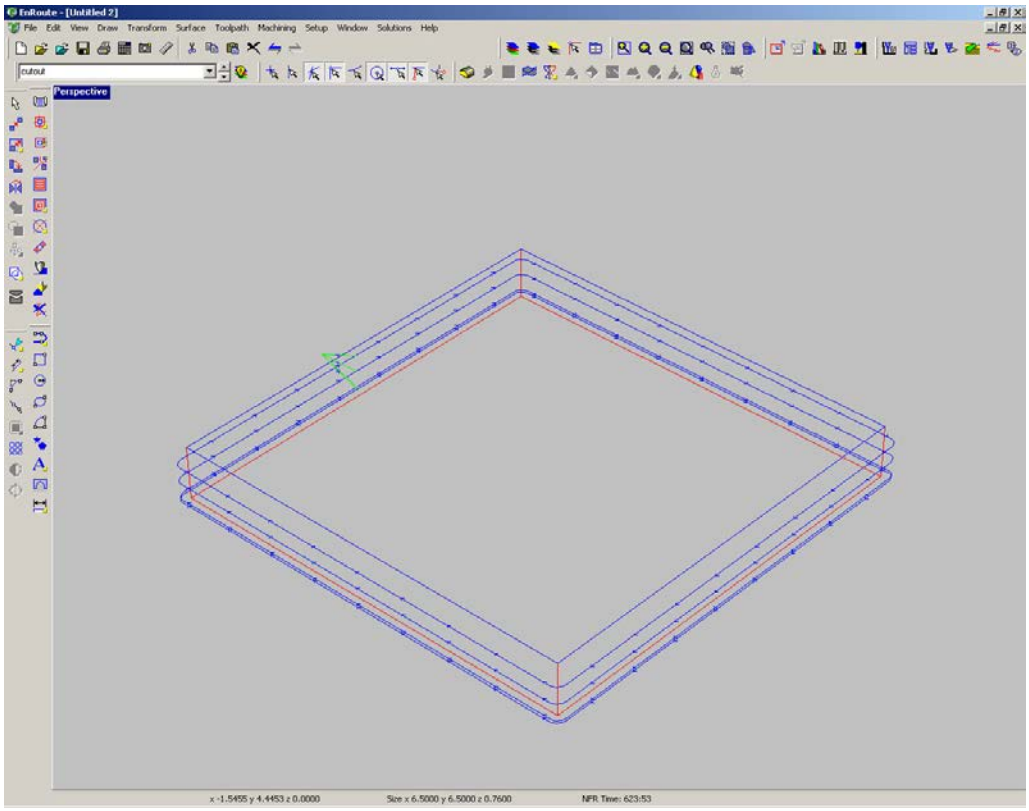
Top view of the tool path. Note how close the path lines are.



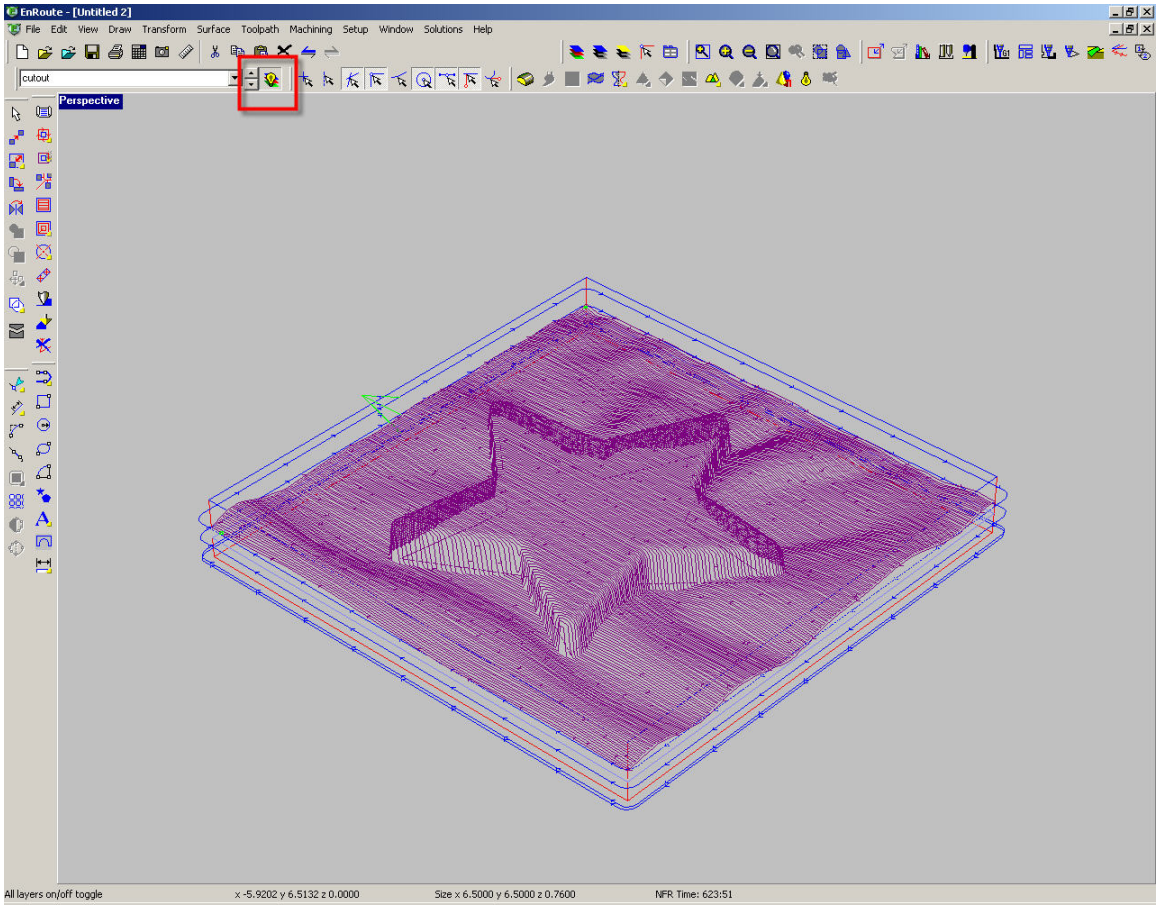
Perspective view of the surface.



Go to the “cutout” layer and apply a “routing offset” tool path for our cutout.



Perspective view of the cutout.



Both toolpaths.

At this point you are ready to output your files and take them to the machine.

Safety Instructions / Proper Colleting

Techno CNC Systems

READ THESE INSTRUCTIONS THOROUGHLY *BEFORE* OPERATING MACHINE. DO NOT OPERATE MACHINE IF YOU ARE UNFAMILIAR WITH THESE SAFE OPERATING INSTRUCTIONS. DO NOT OPERATE MACHINE WITHOUT KNOWING WHERE THE EMERGENCY STOP SWITCH IS LOCATED.

WARNING: IMPROPER OR UNSAFE OPERATION OF THE MACHINE WILL RESULT IN PERSONAL INJURY AND/OR DAMAGE TO THE EQUIPMENT.

1. Keep fingers, hands, and all other objects away from machine while power is on.
2. Disconnect power to all system components when not in use, when changing accessories, and before servicing.
3. Do not loosen, remove, or adjust machine parts or cables while power is on.
4. Exercise care with machine controls and around keyboard to avoid unintentional starting.
5. Make sure voltage supplied is appropriate to specifications of components.
6. Machines must be plugged into three-pronged grounded outlets. Do not remove the grounding plug or connect into an ungrounded extension cord.
7. Keep cables and cords away from heat, oil, and sharp edges. Do not overstretch or run them under other objects or over work surfaces.
8. Use proper fixtures and clamps to secure work. Never use hands to secure work.
9. Do not attempt to exceed limits of machine.
10. Do not attempt to use machine for purposes other than what is intended.
11. Use machine only in clean, well-lit areas free from flammable liquids and excessive moisture.
12. Stay alert at all times when operating the machine.
13. Always wear safety goggles and dust mask.
14. Do not wear loose-fitting clothing or jewelry when operating machine. Long hair should be protected.
15. Always maintain proper balance and footing when working around the machine.
16. Maintain equipment with care. Keep cutting tools clean and sharp. Lubricate and change accessories when necessary. Cables and cords should be inspected regularly. Keep controls clean and dry.
17. Before using, check for damaged parts. An authorized service center should perform all repairs. Only identical or authorized replacement parts should be used.
18. Remove any adjusting keys and wrenches before turning machine on.
19. Do not operate the machine unattended.
20. Follow all safety instructions and processing instructions in the MSDS for the material being processed.
21. Use proper precautions with dust collection systems to prevent sparks and fire hazards.
22. Make sure to have proper fire extinguishing equipment on hand at all times.

PREVENT FIRE HAZARDS by using the proper feeds, speeds, and tooling while operating your Techno machine. For example, setting feeds and speeds too low and/or using dull tool bits creates friction at the material. The friction generates heat which can result in a fire that can be drawn through the vacuum table or dust collector without warning. Fire hazard from friction heating caused by dull tools is possible when cutting certain materials, especially composite material such as wood composites, MDF and Particleboard.

Techno CNC Systems

WARNING!

THE SPINDLE WILL BE DAMAGED
IF UNBALANCED EQUIPMENT IS USED.

AIR SUPPLY MUST BE FILTERED AND DRY.

COLLETING GUIDELINES

WRONG!

This picture shows an improper assembly. Notice the gap and angle of the collet in relation to the nut. The collet is not flush to the end of the collet nut. Correct this assembly before using.

RIGHT!

The picture above is how your collet nut assembly should look: the end of the collet is flush with the bottom surface of the collet nut. You will hear and feel a "SNAP" as the collet properly goes into the collet nut. Once it is assembled, then "SCREW" the nut onto the threaded spindle end.

DO NOT PUSH THE COLLET INTO THE SPINDLE AT ANY TIME!

Only the proper assembly should be screwed onto the spindle.

FOR TOOLCHANGE AND FIXED COLLET SPINDLES:

ONLY USE TOOLHOLDERS, COLLET NUTS AND TOOLS THAT ARE BALANCED TO MEET OR EXCEED THE MAX RATED SPEED OF THE SPINDLE.