DeskCNC setup and operation manual

This document explains how to install, setup and cut foam shapes using DeskCNC 4 axis foam cutting software.
The document will go through a step by step process of how to setup the software and make simple to complicated cuts to maximize the use of the software.

DeskCNC was designed for a 3 axis CNC router type machines, at a later stage it was modified to enable a 4th axis need to run an automated 4-axis foam cutter. If you have purchased DeskCNC for your foam cutter, you can also use it to drive a CNC router table or a similar 3-axis machine.

This document explains the following three chapters:

- DeskCNC Software Installation
- DeskCNC Software Setup
- Making Sample cuts tutorial plus examples

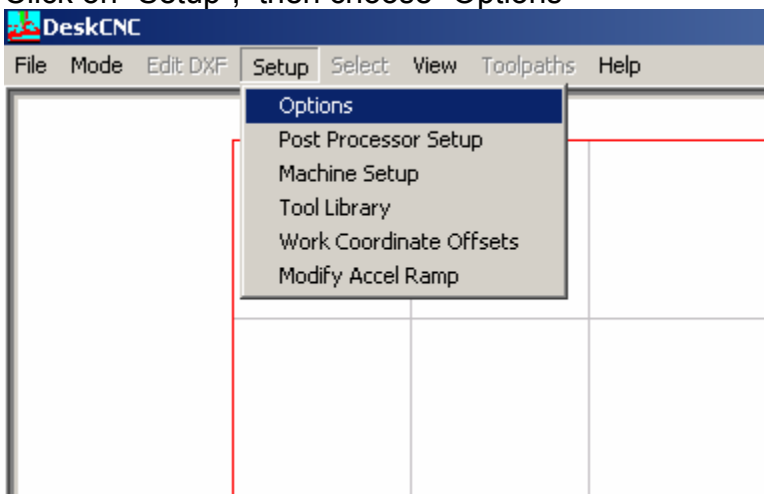
DeskCNC Software Installation
If your product did not come with pre-installed DeskCNC software, please follow these steps to install it on your Windows PC.

1. Download the latest DeskCNC software from this web site: www.deskcnc.com
2. Double click the DeskCNC file and continue the installation
3. Once the installation has finished, turn on the power to the controller box and click on the DeskCNC icon to run the program

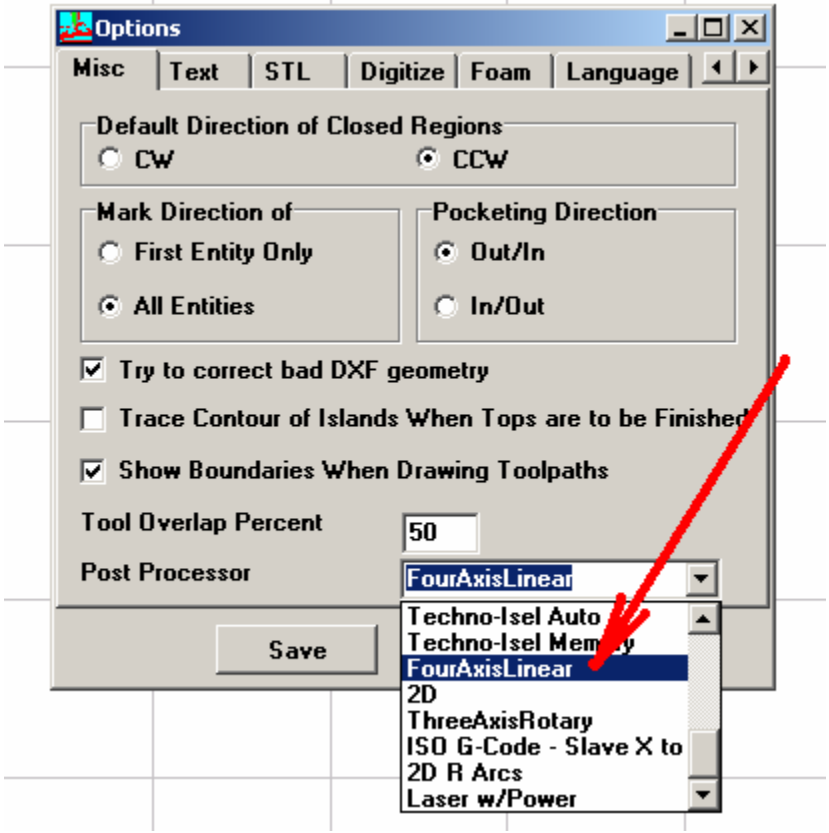
DeskCNC software Setup
Please follow the next steps very carefully in order to assure the correct operations of the machine and software.

1. Turn on the DeskCNC box or the driver box (which has the DeskCNC electronics inside
2. Click on the DeskCNC icon.
3. The 1st message on the screen should be “Hardware detected”
4. If you get “hardware not found” this means that the software on the PC does not recognize the DeskCNC electronics, and you will not be able to proceed without fixing this.
5. Usually this message will show if the DeskCNC electronics are not powered up or the serial cable is not connected between the PC and the DeskCNC electronics.

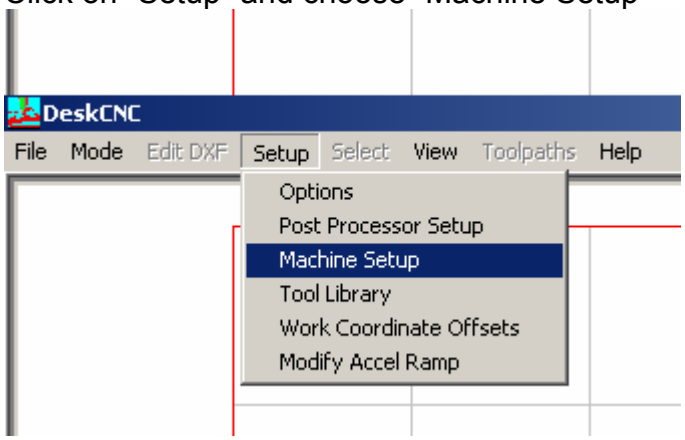
6. Click on “Setup”, then choose “Options”



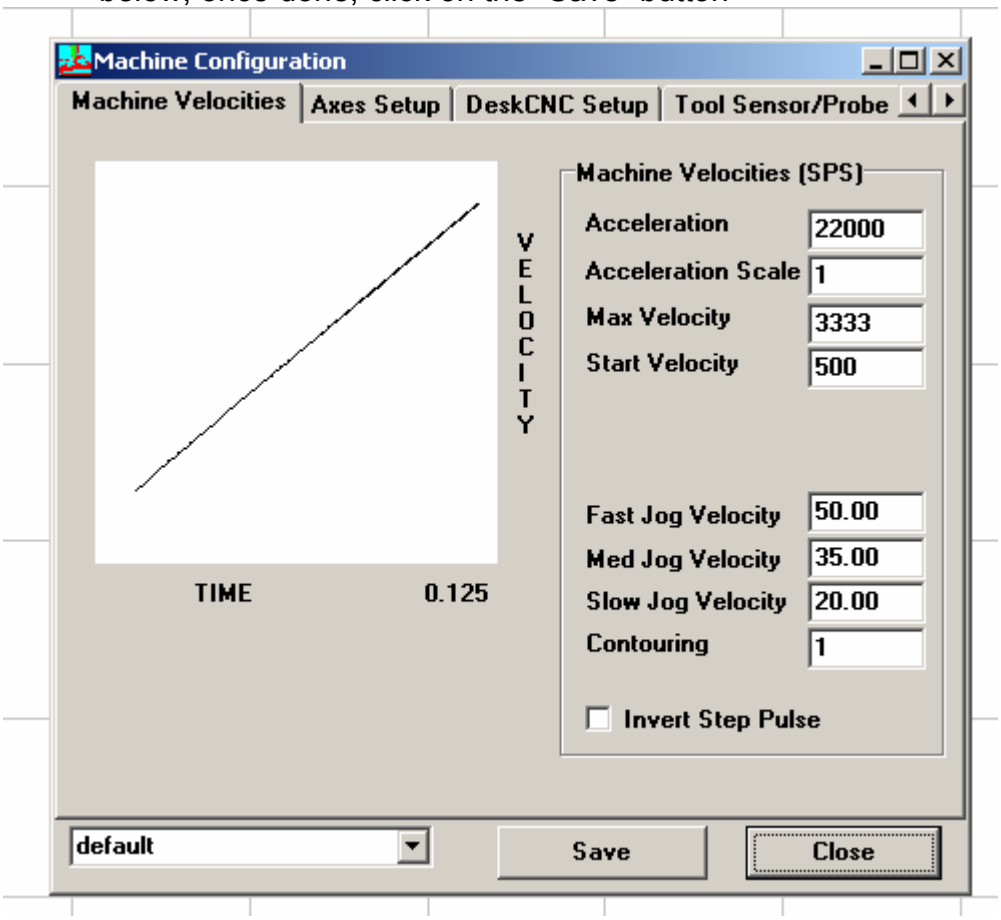
7. In the Options category, click on the Post Processor pull down menu and select the FourAxisLinear once selected, click on the Save Button



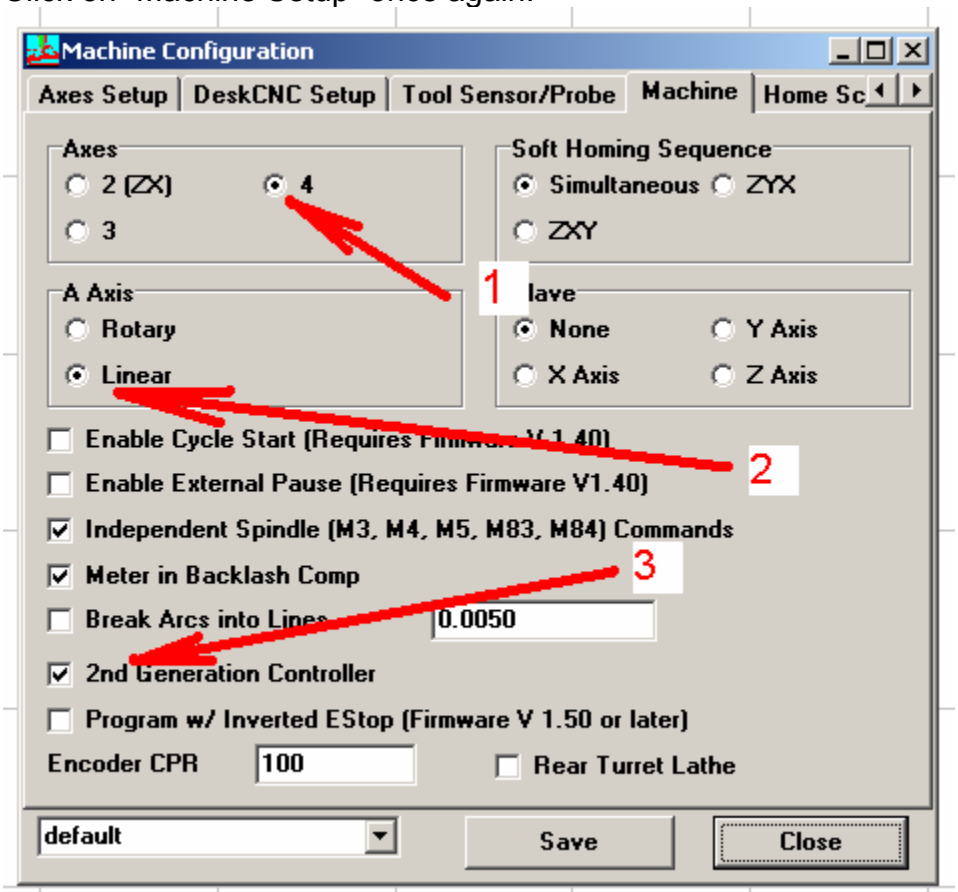
8. Click on “Setup” and choose “Machine Setup”



9. Make sure to set the values to the exact values shown in the screen shot below, once done, click on the “Save” button

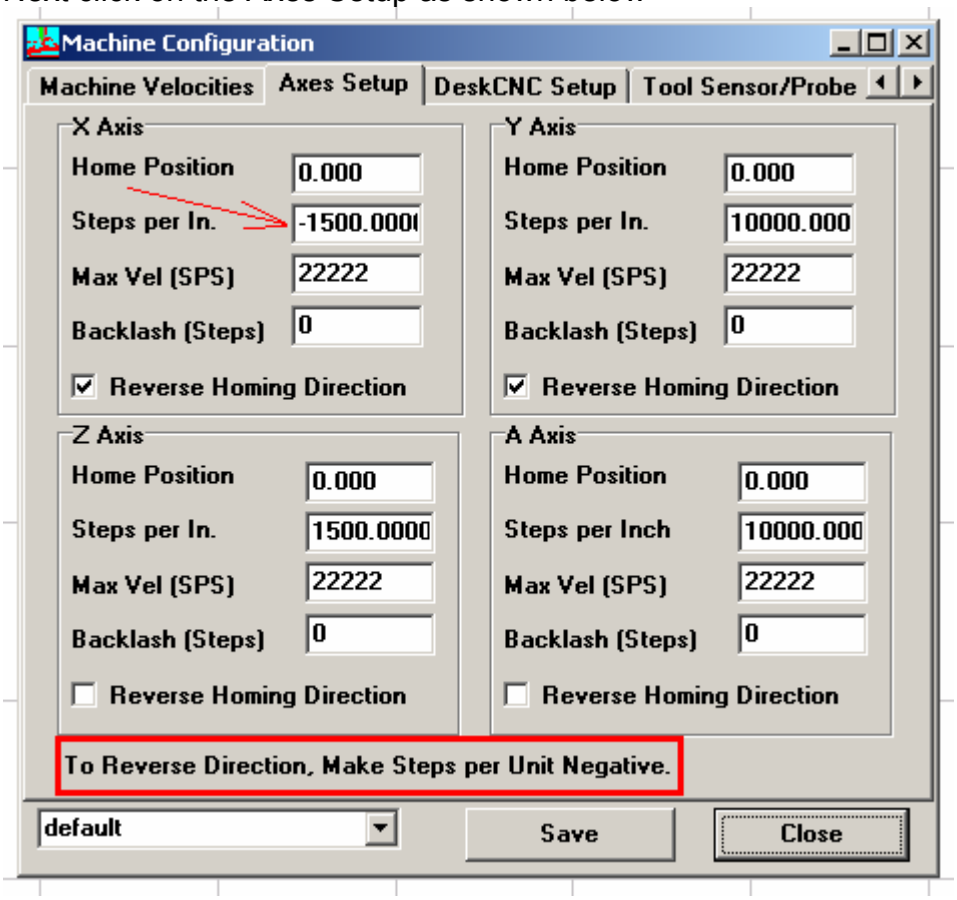


10. Click on “Machine Setup” once again.



Axes - Click on the “4” option with the number 1 arrow pointing to
A Axis - Click on “Linear” option with the number 2 arrow pointing to
Click the checkbox on the “2nd Generation Controller” with the number 3 arrow pointing to. Once done, click on the “Save” button

11. Next click on the Axes Setup as shown below



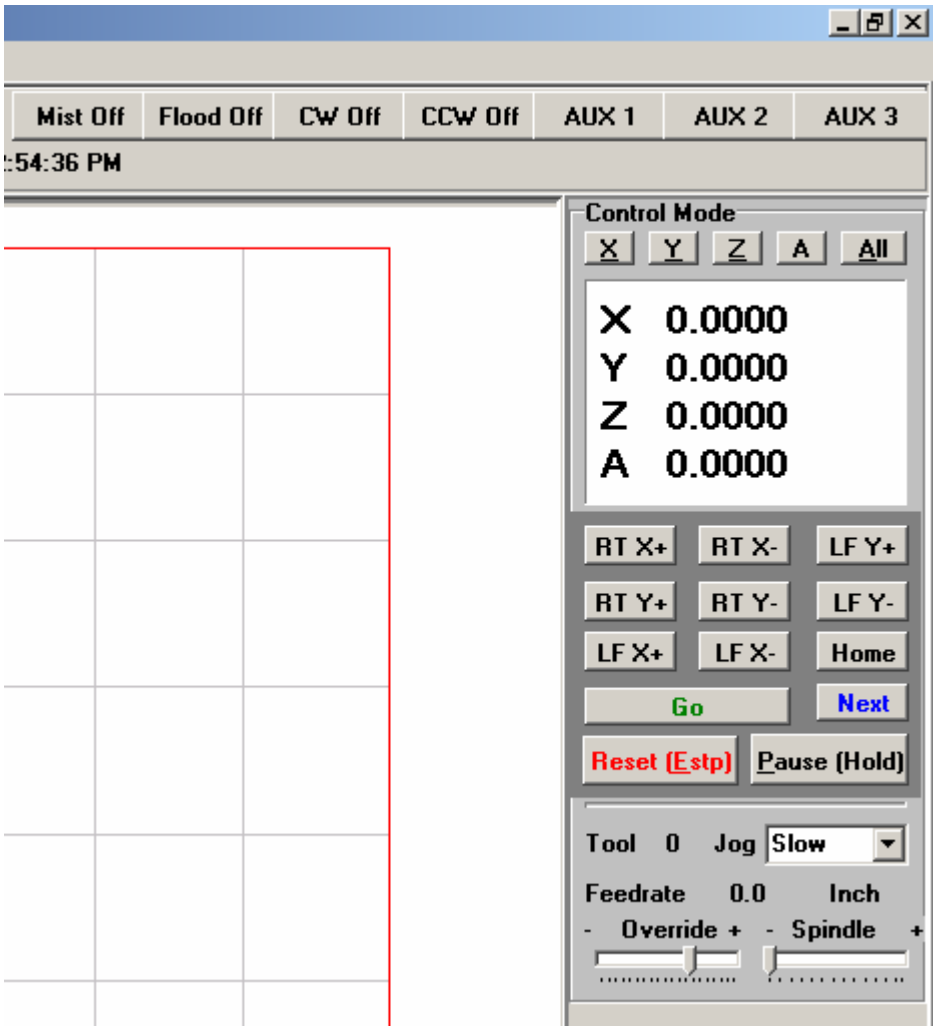
The steps per In. values will change from one machine to the other depending on the lead screw use – Please refer to the setup page which came with you machine

12. Notice that X-axis “Steps per In.” has a [-] in front of the 1500.000 this is intended to make this axis move the “other” direction. This is used when the machine was setup and each axis is manually moved for testing purposes. If an axis moved the “wrong” direction, place the [-] sign in front of the value. Once done, click on the Save Button
    Set all “steps per In.” to 1500 !!!
13. Note the red rectangle – To Reverse Direction, make Steps per in Unit Negative – as explained above in item 12

Setup of DeskCNC is now complete – you are ready to start cutting

Manual Control
Each of the 4 axis can be move independently.
In the screen shot below,
RT X+ : Click on this button and the Right X axis will move forward
RT X - : Click on this button and the Right X axis will move Backwards
RT Y+ : Click on this button and the Right Y axis will move Up
RT Y - : Click on this button and the Right Y axis will move Down

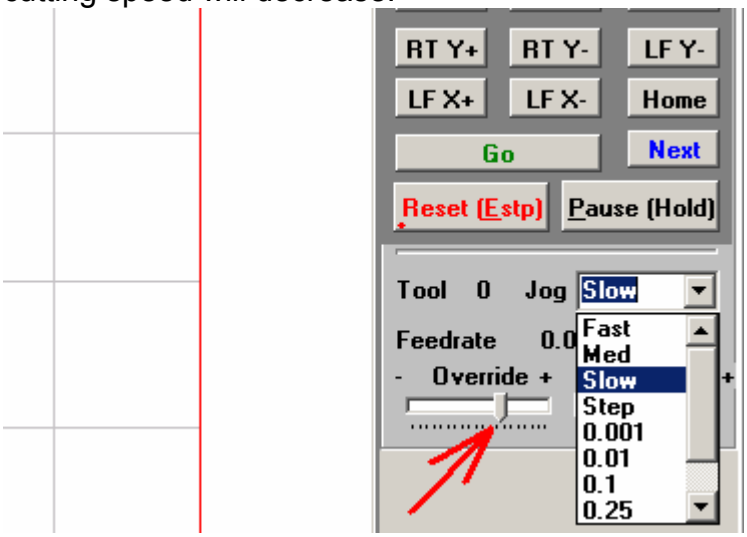
LF X+ : Click on this button and the Left X axis will move forward
LF X - : Click on this button and the Left X axis will move Backwards
LF Y+ : Click on this button and the Left Y axis will move Up
LF Y - : Click on this button and the Left Y axis will move Down



When trying out the machine – moving each axis, the above window will be shown. By default the machine will start when the emergency switch is open (the machine will not operate) In order to move each axis you must click once on the RESET (Estp) button shown in red text in the picture above

Feedrate Overrride

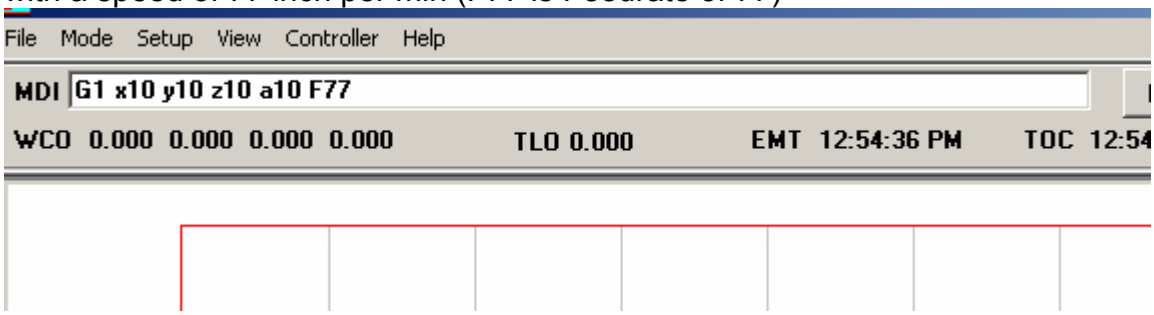
When the machine is running (cutting), the cutting speed can be increased or decreased. By holding down the override slide, and moving it to the left, the cutting speed will decrease.



Once the jog speed is highlighted, you can move both axis stimulatingly. Simply highlight the Jog speed (in the example above) Slow, and then press the keyboard arrows (left right up or down)

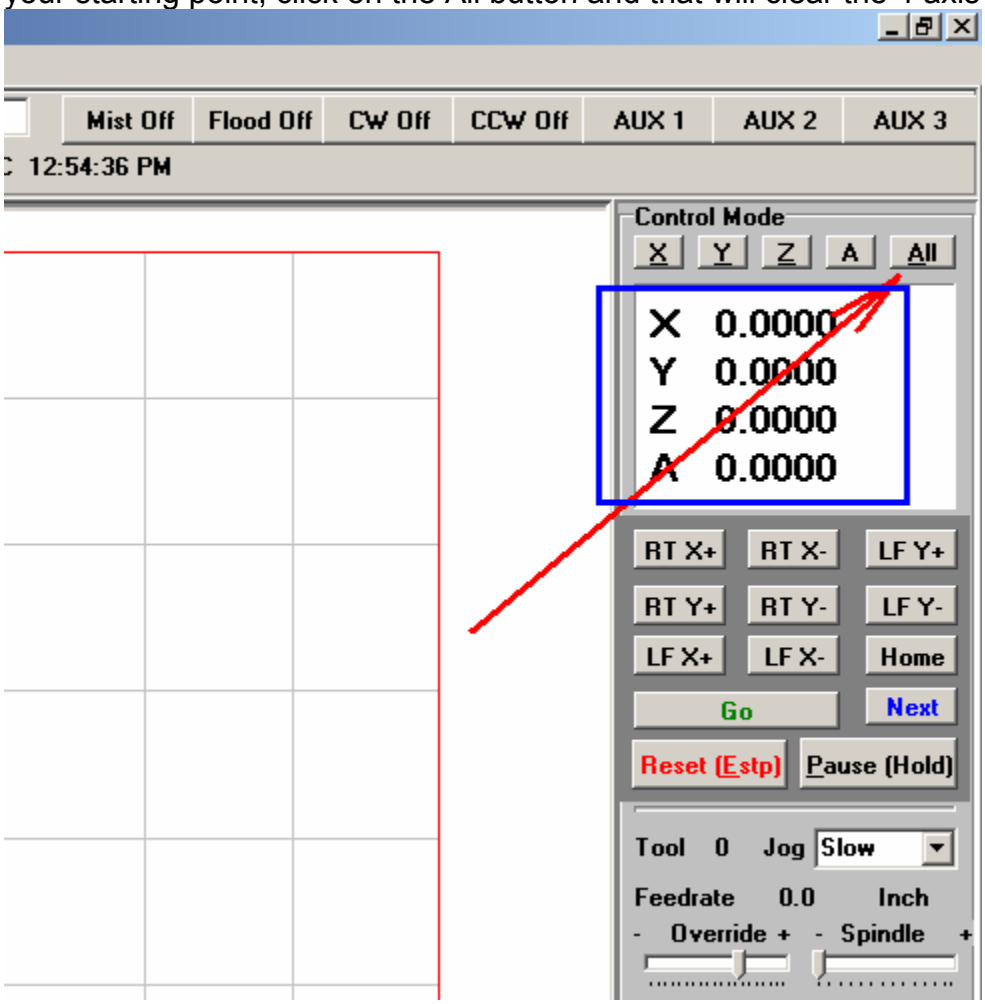
G code

G code can be entered (in a command line) and run in the top left corner of the screen as shown below. This is useful if the user wants to move the machine to a specific location. In the example below, the machine will move 10” on all 4 axis with a speed of 77 inch per min (F77 is Feedrate of 77)

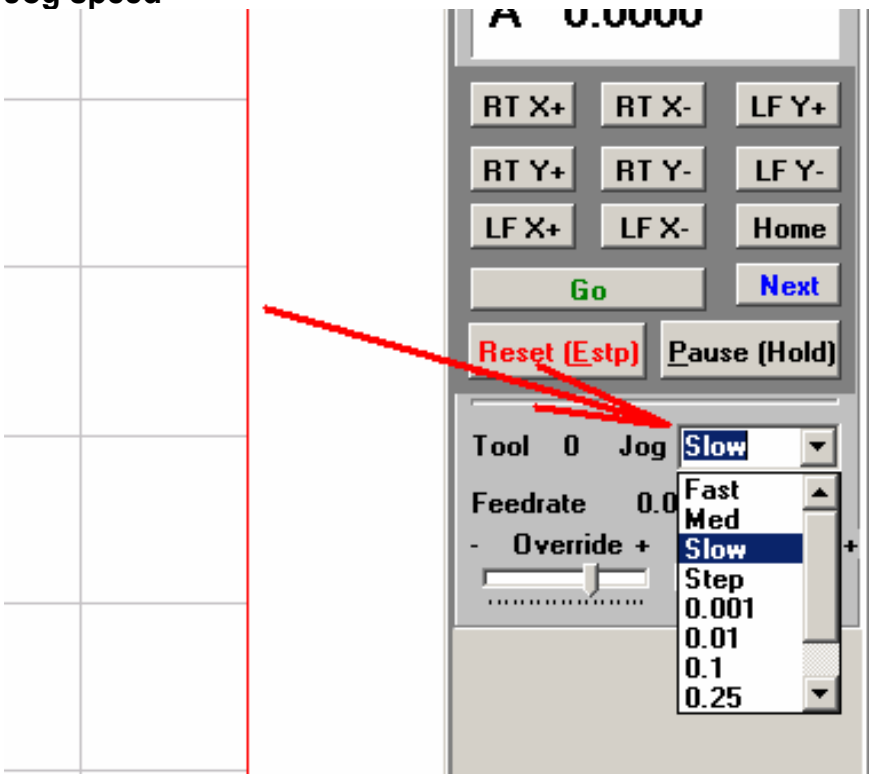


0 Axis

Click this button (the one with the red arrow pointing to) and the XYZA values will change to 0. In the screen shot there are already 0, but this feature is useful if you manually move any of the axis, and set your starting point. Once you set your starting point, click on the All button and that will clear the 4 axis values.

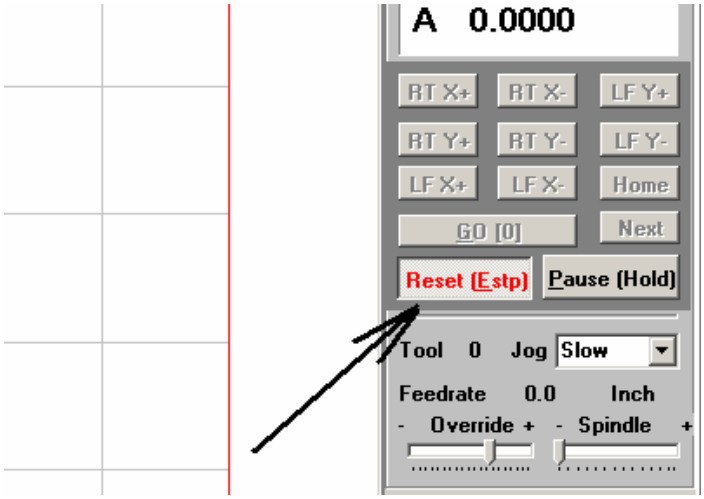


Jog speed

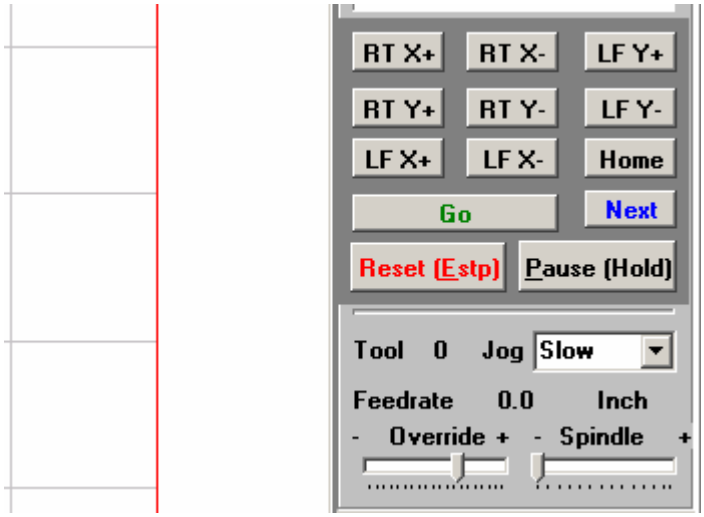


When manually moving any or all of the axis, jog speed can be set to different speeds.
Click on the Jog pull down menu, and set the desired speed.
Once the desired speed is set, click on the RT X+ or any of the other buttons to move the axis with this new speed settings.

Reset – Estp (Emergency Stop)
By Default, when the software is started, the Reset button will be “pressed” and the Axis buttons are “grayed” out – no active.
Click once on the Reset button and the buttons will become active.
If during a cut or an automatic movement of any of the axis, you wish to stop the machine, simply click once on the Reset button and all movements will stop



Screen shot below - Reset button enabled



Making sample cuts

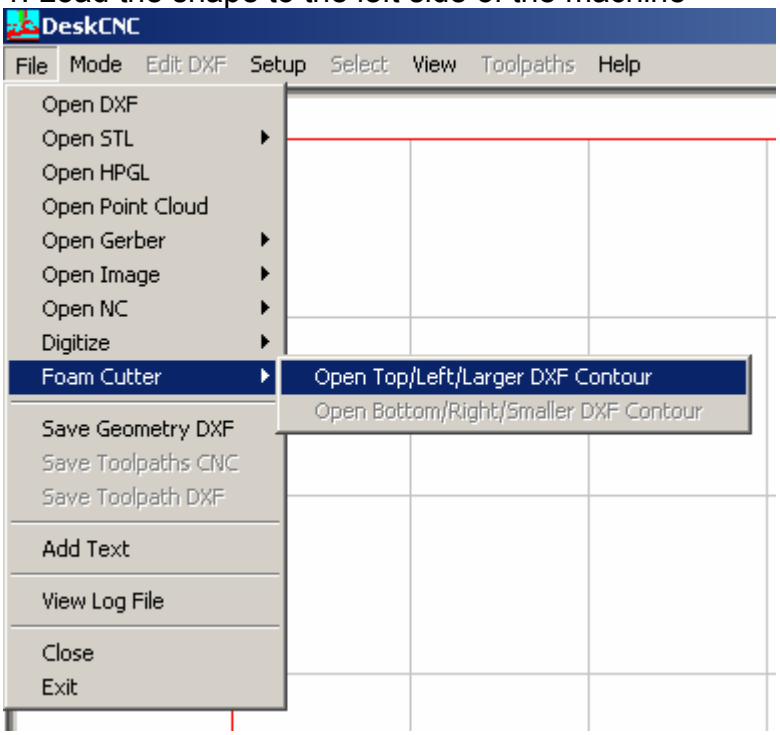
Single shape cut:

In this example we take a single shape and prepare it for a cut.

DeskCNC will accept DXF file format for its foam cutter operation.

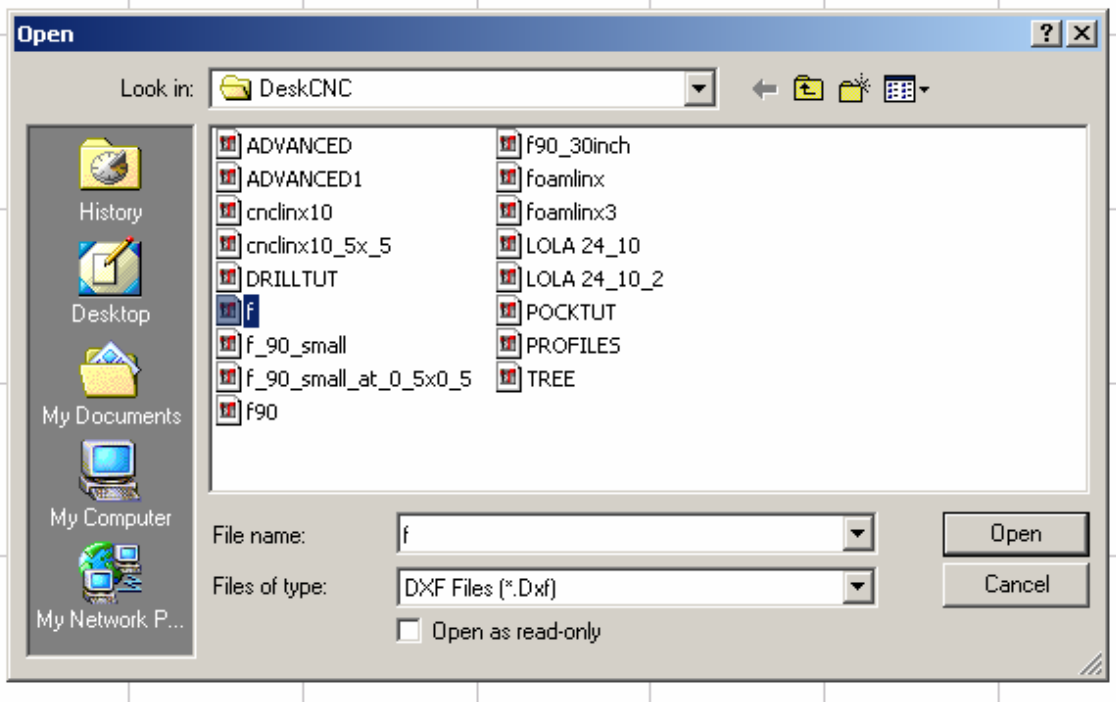
Since the machine is a true 4 axis machine – it can cut one shape on the left side of the machine and a different shape on the right side. For an example. A square on one side and a circle on the other. In the below example, both shapes are the same. AutoCAD was used to generate the shape in the example

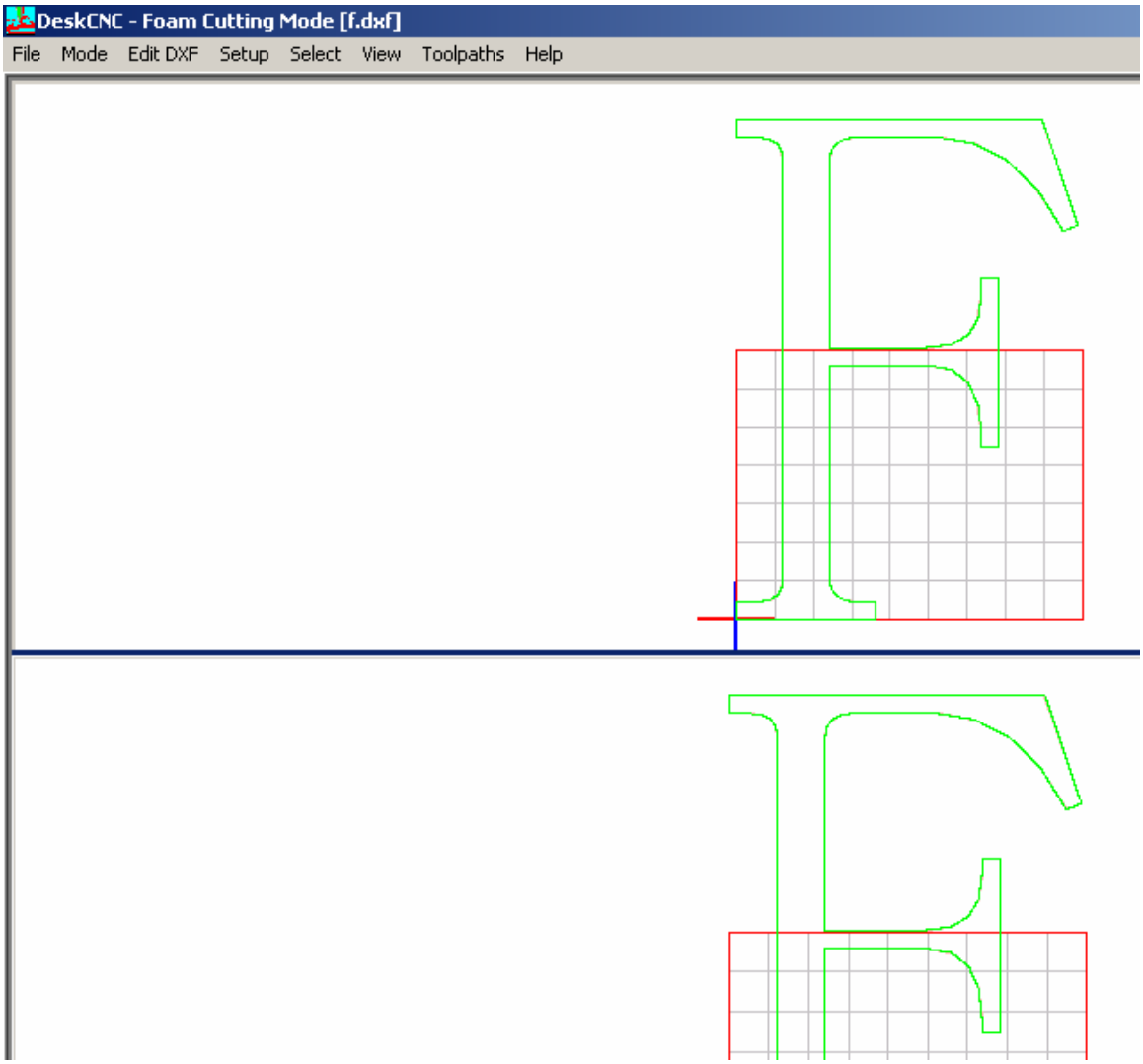
1. Load the shape to the left side of the machine



By default, DeskCNC will place the same shape on both sides, but if a different shape is needed for the Right side of the machine, simple click on Foam Cutter once again and this time choose the Right side and load the desired shape

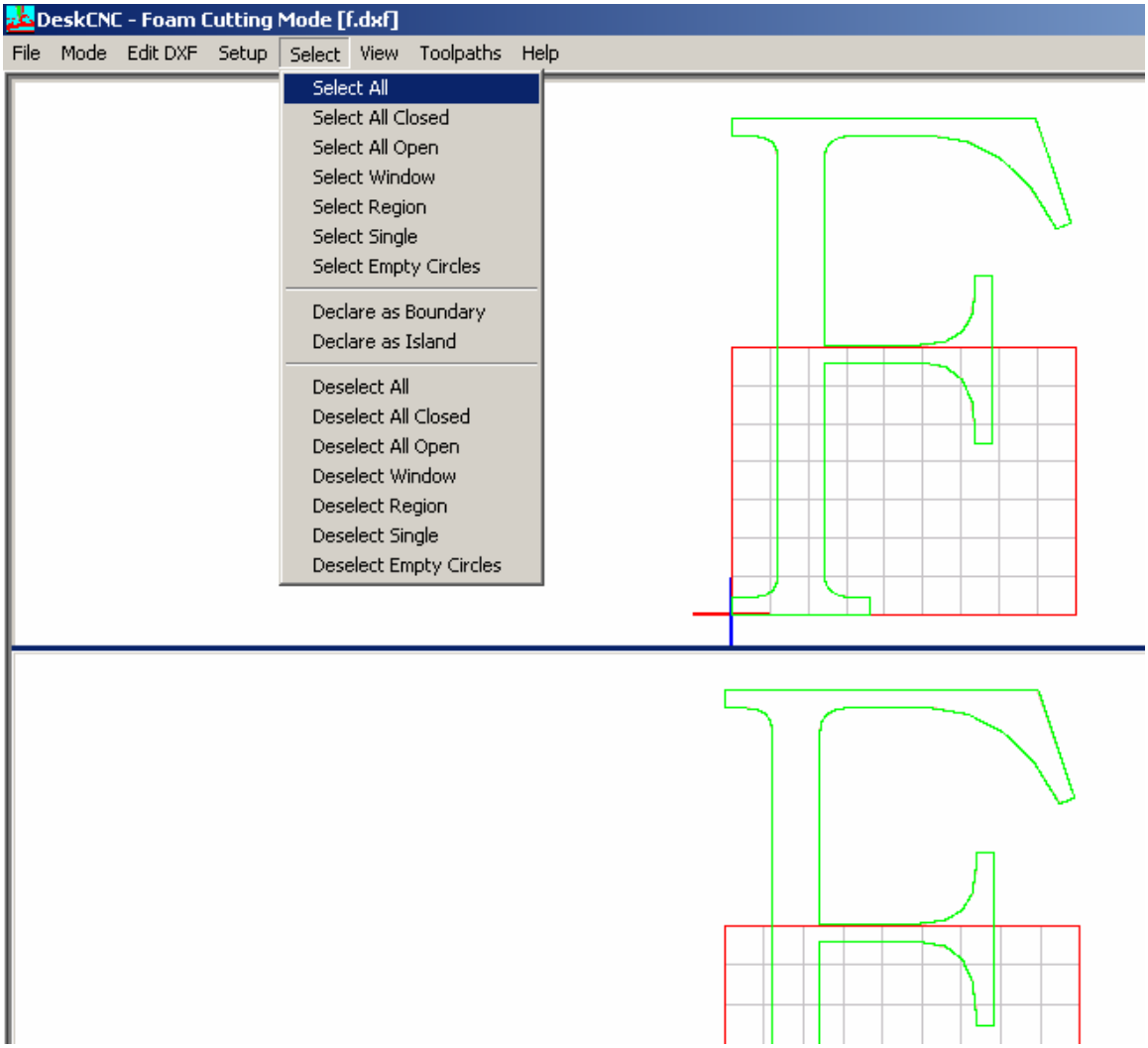
2. Click on the file that you wish to load, in this example we load the Shape with name of “f”





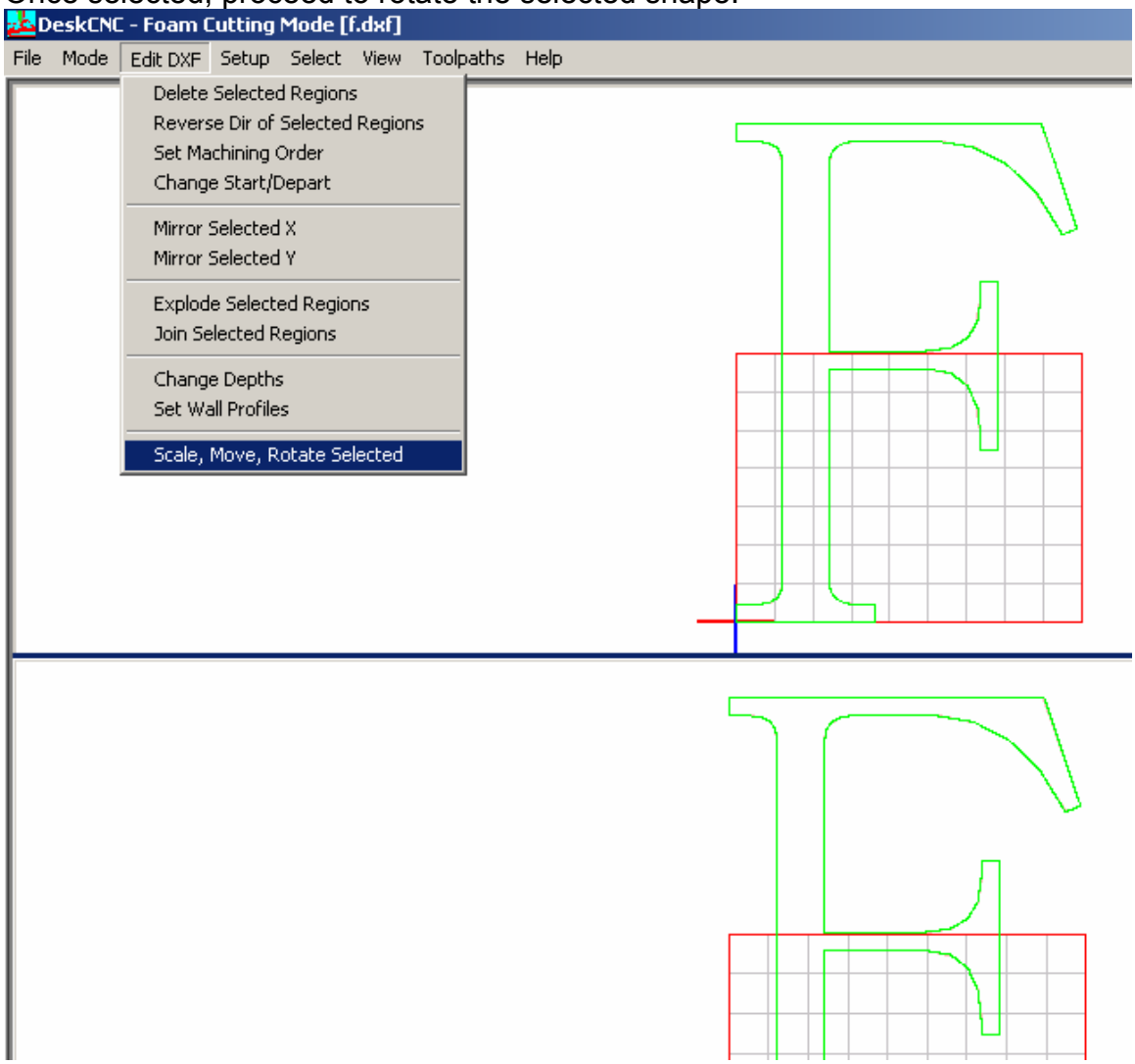
In the above screen shot, the shape of a letter F was chosen.
Suppose the letter must be rotated to fit into the block of foam – in the following steps we will rotate, resize and position the shape as needed.

1. Rotate the shape:
Before rotating the shape, DeskCNC must know what you want to rotate,
In this example we will rotate the entire shape. In order to rotate the entire shape we muse first select it.

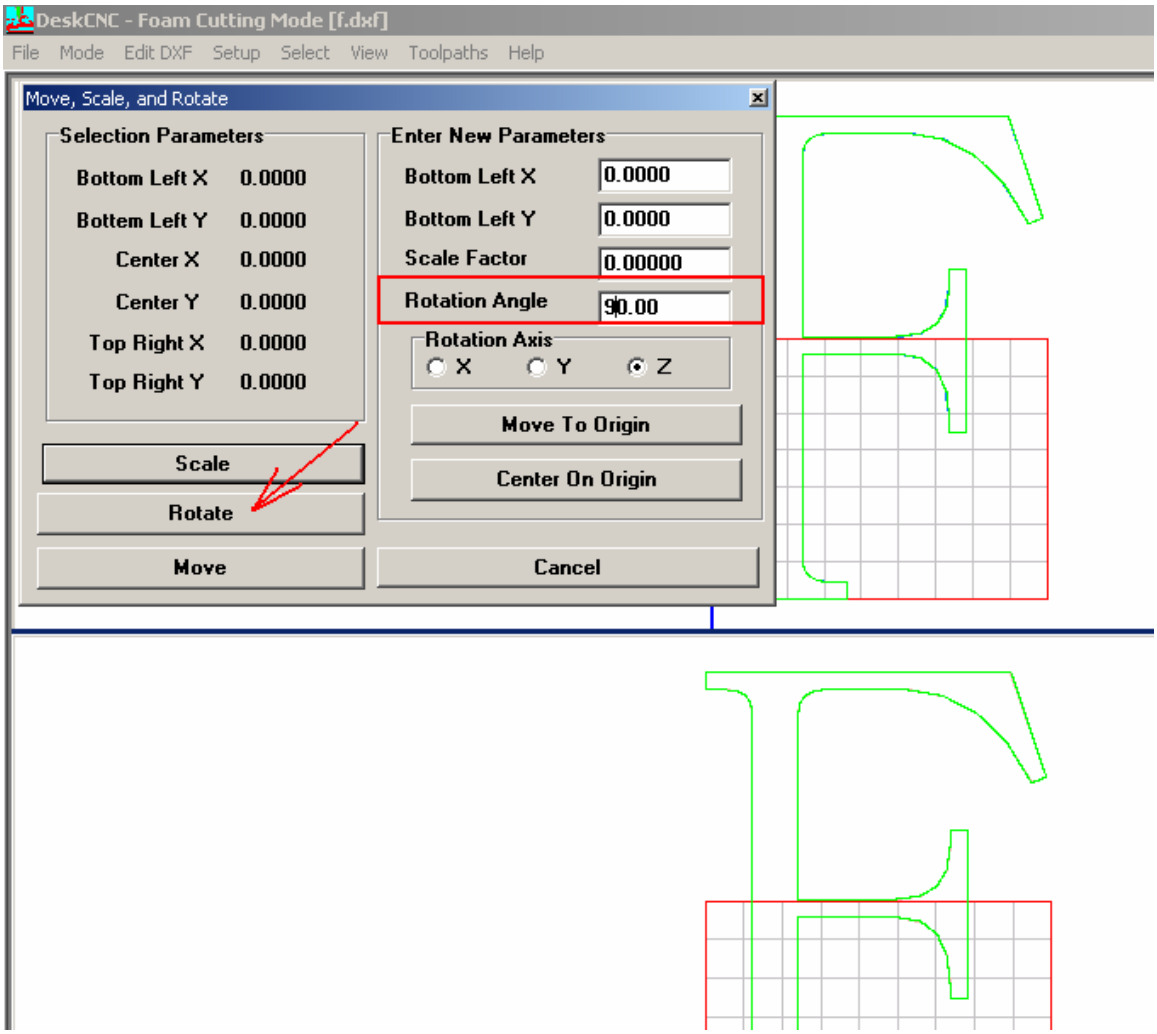


Click on Select, and then choose “Select All” – as shown above.
If this step is skipped, once in the rotate menu, an error message will pop stating that nothing was selected, and the rotation can’t be done

Once selected, proceed to rotate the selected shape.

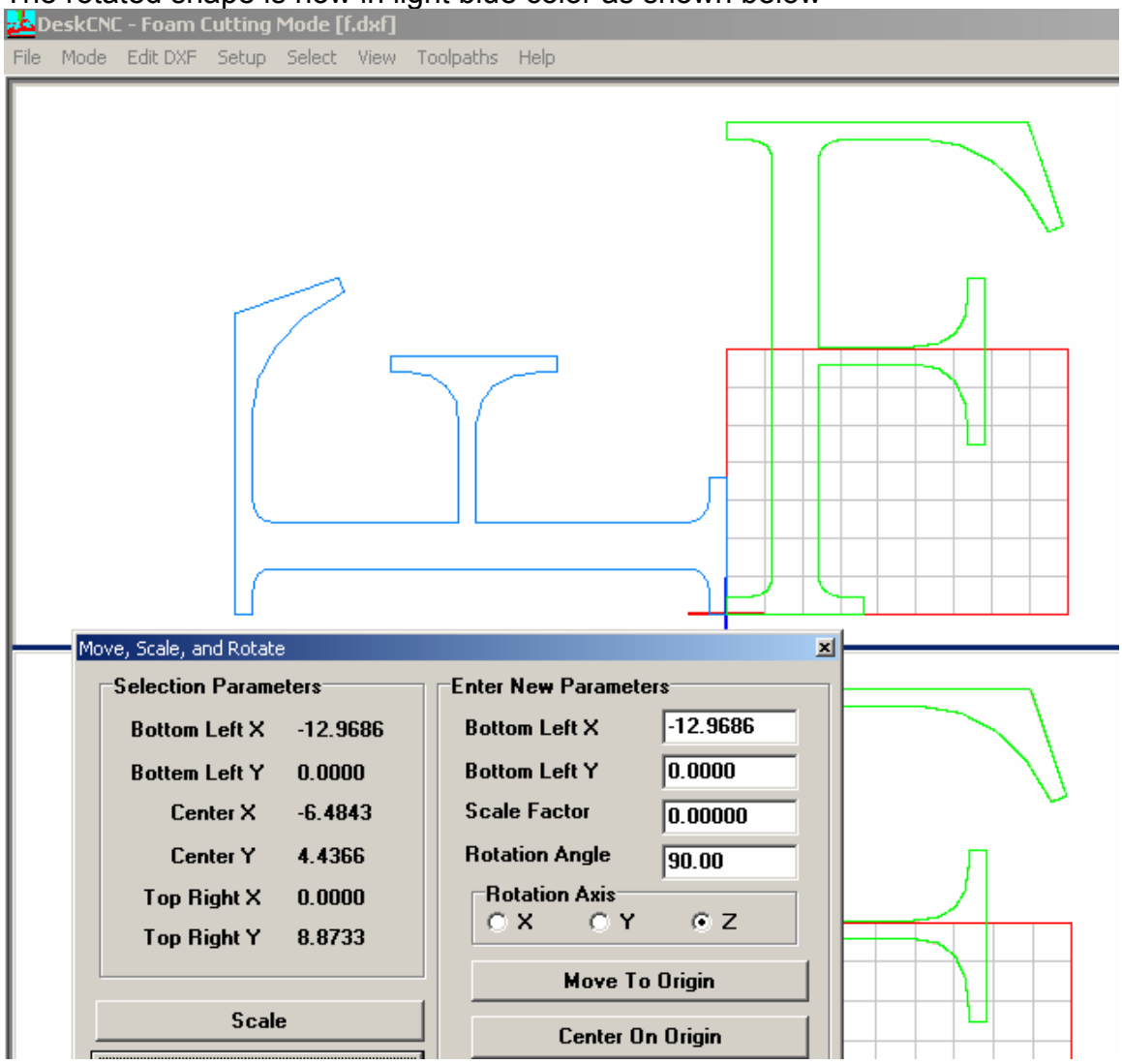


Click on “Edit DXF”, then choose the last option in the pop down menu “Scale, Move, Rotate selected”

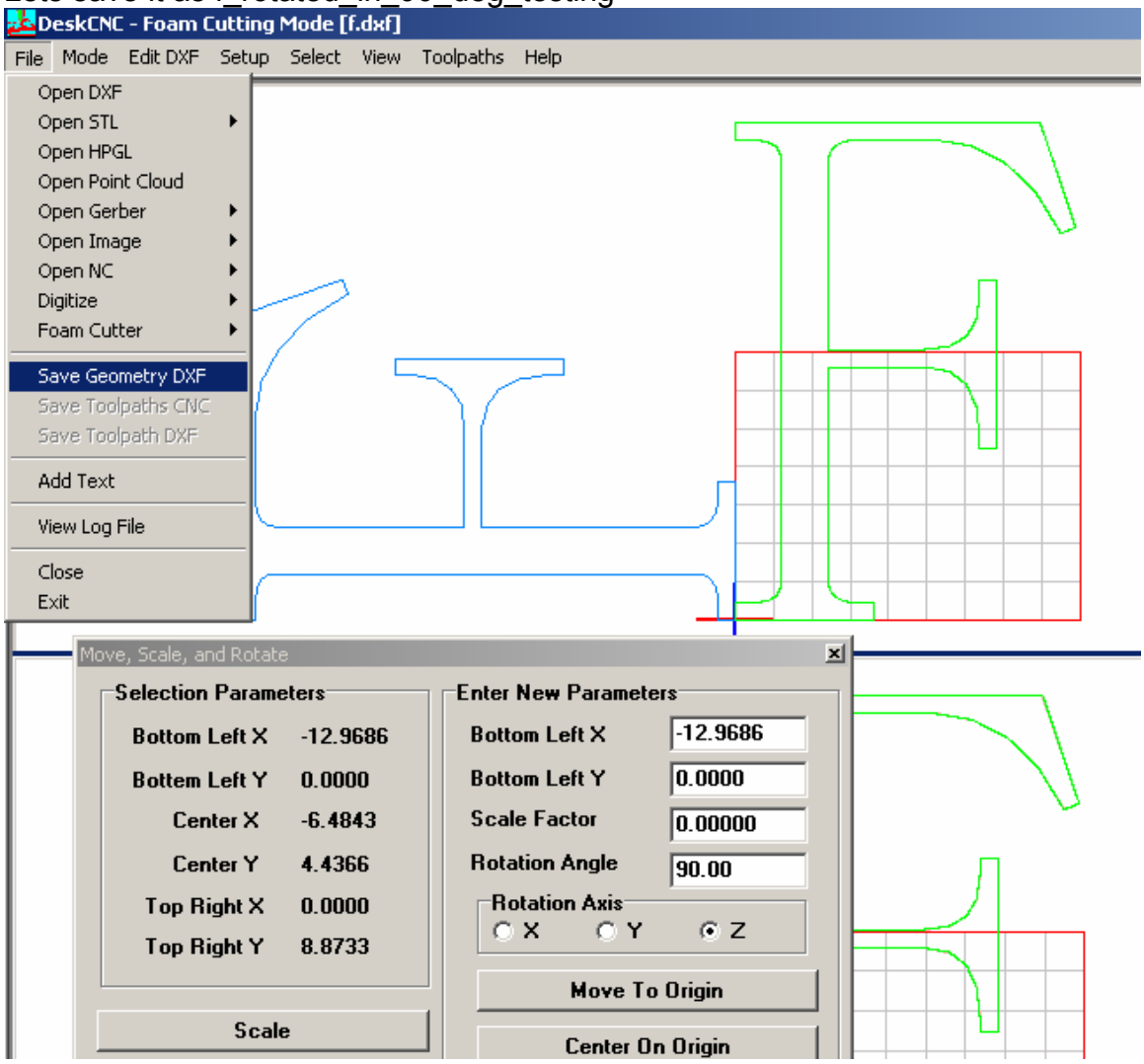


In this example, we will rotate the shape in 90 deg angle. Enter the deg of rotation and then click on the Rotate button

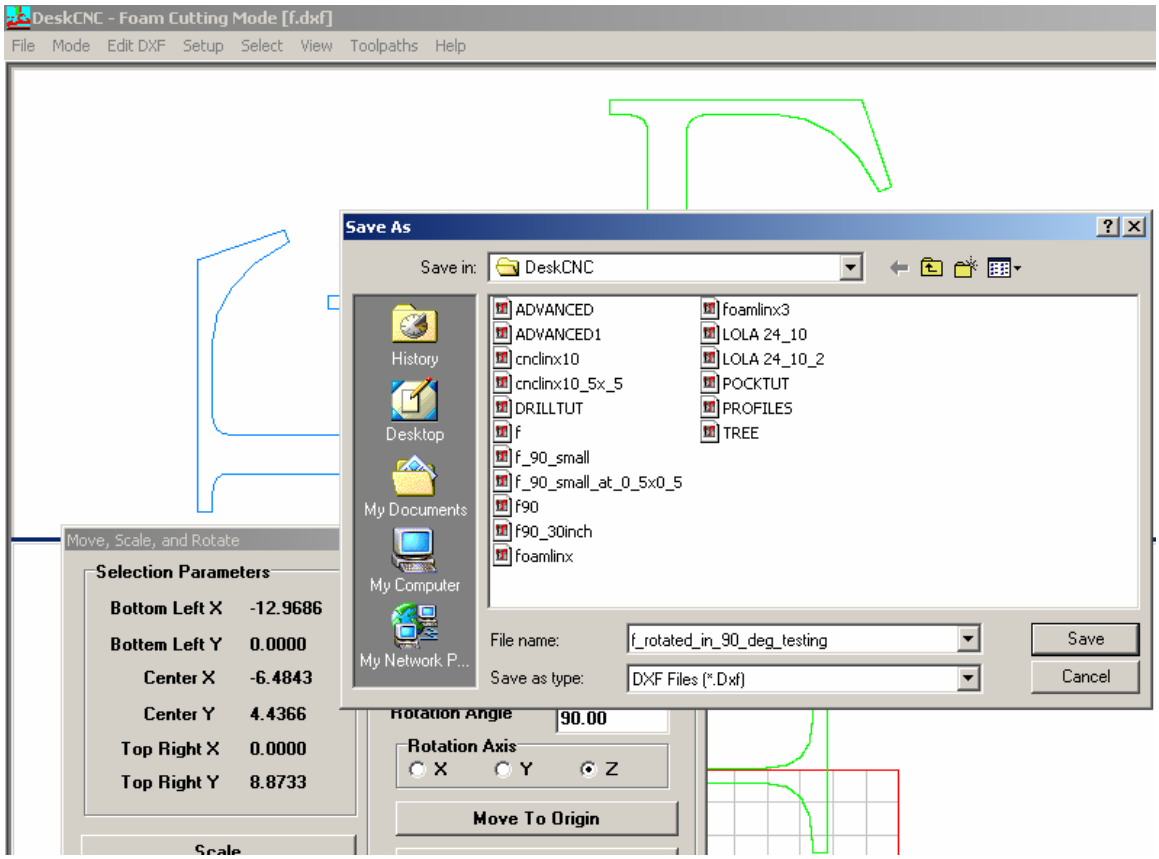
The rotated shape is now in light blue color as shown below



The new shape (the rotated one) should now be saved under a different name, Lets save it as f_rotated_in_90_deg_testing



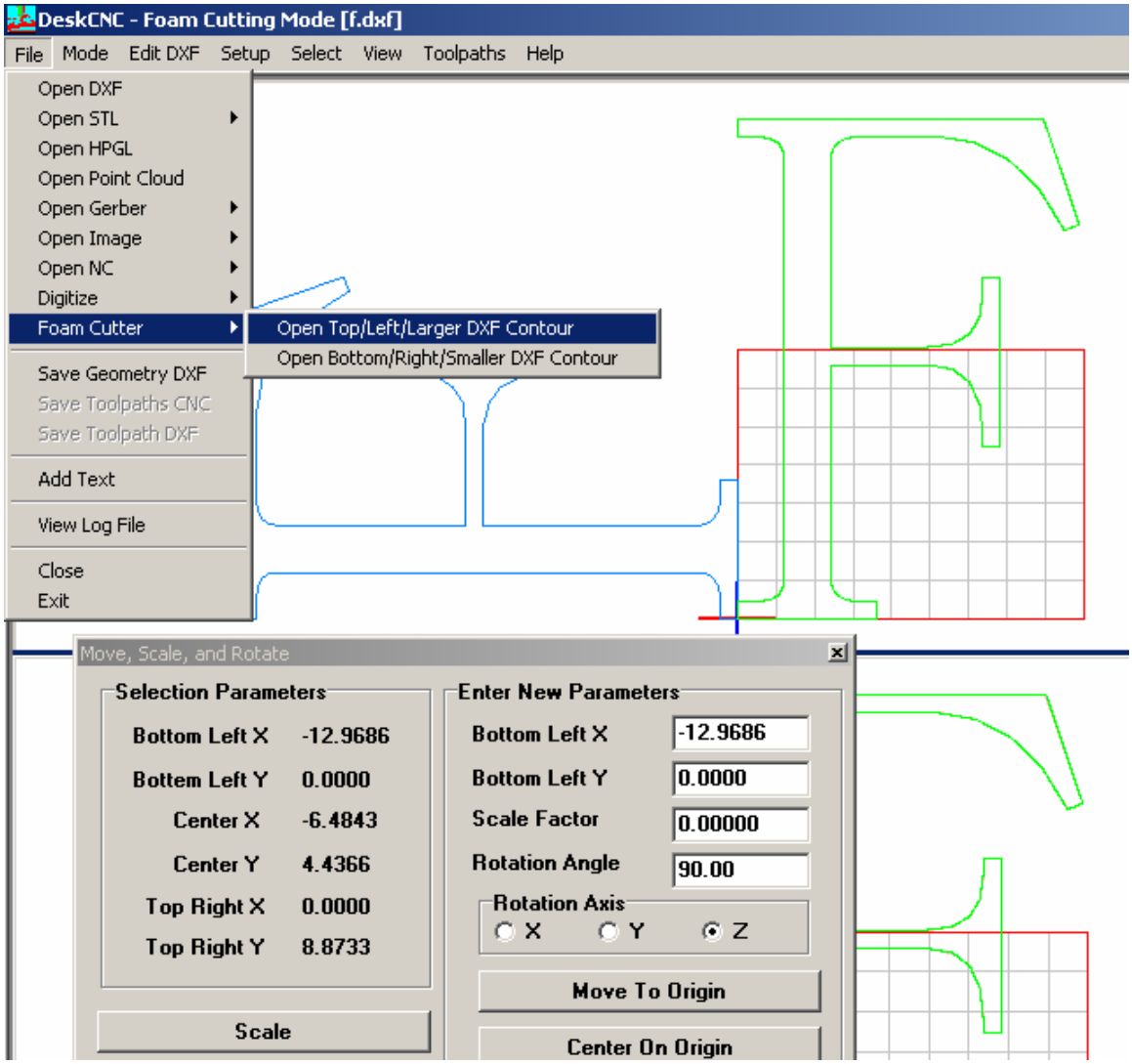
Now enter the name of your new file - f_rotated_in_90_deg_testing



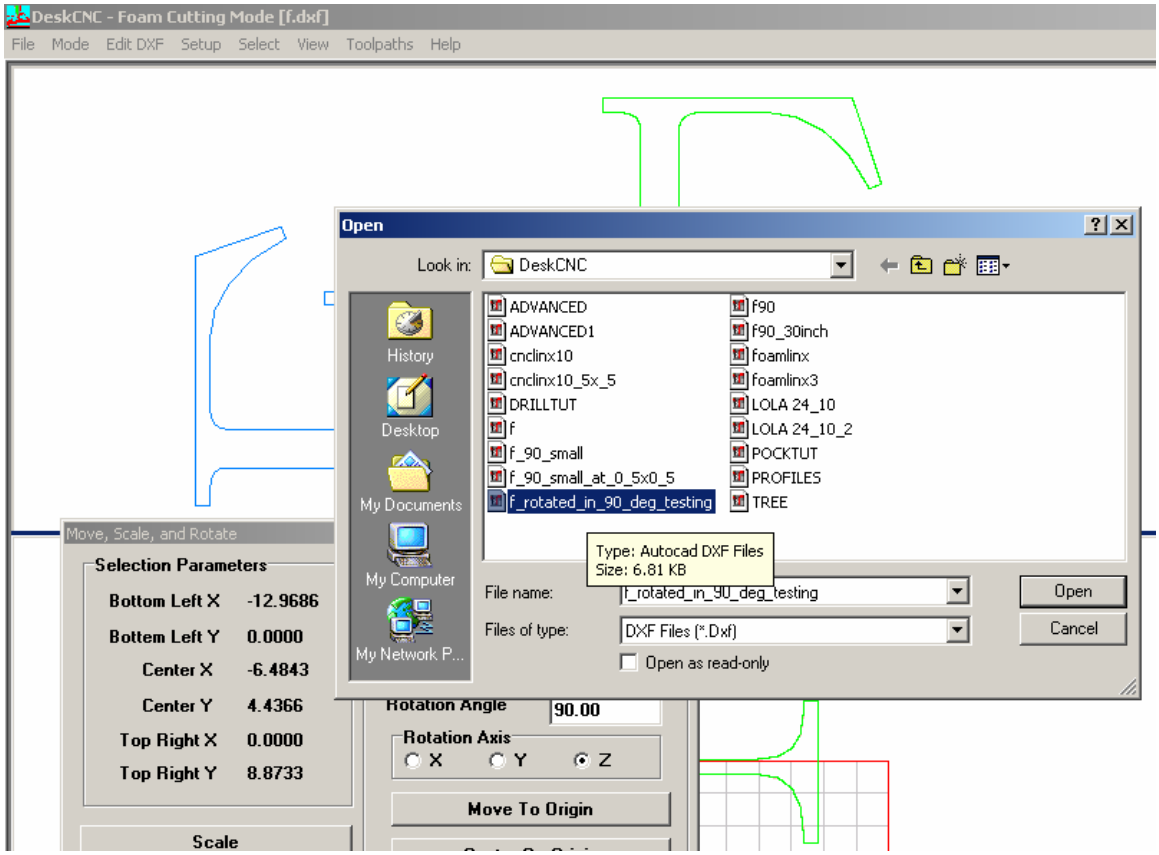
Enter the new name and click on the save button

At this stage you should have a new DXF file with the name f_rotated_in_90_deg_testing.

Now we need to load this new shape for further manipulations.

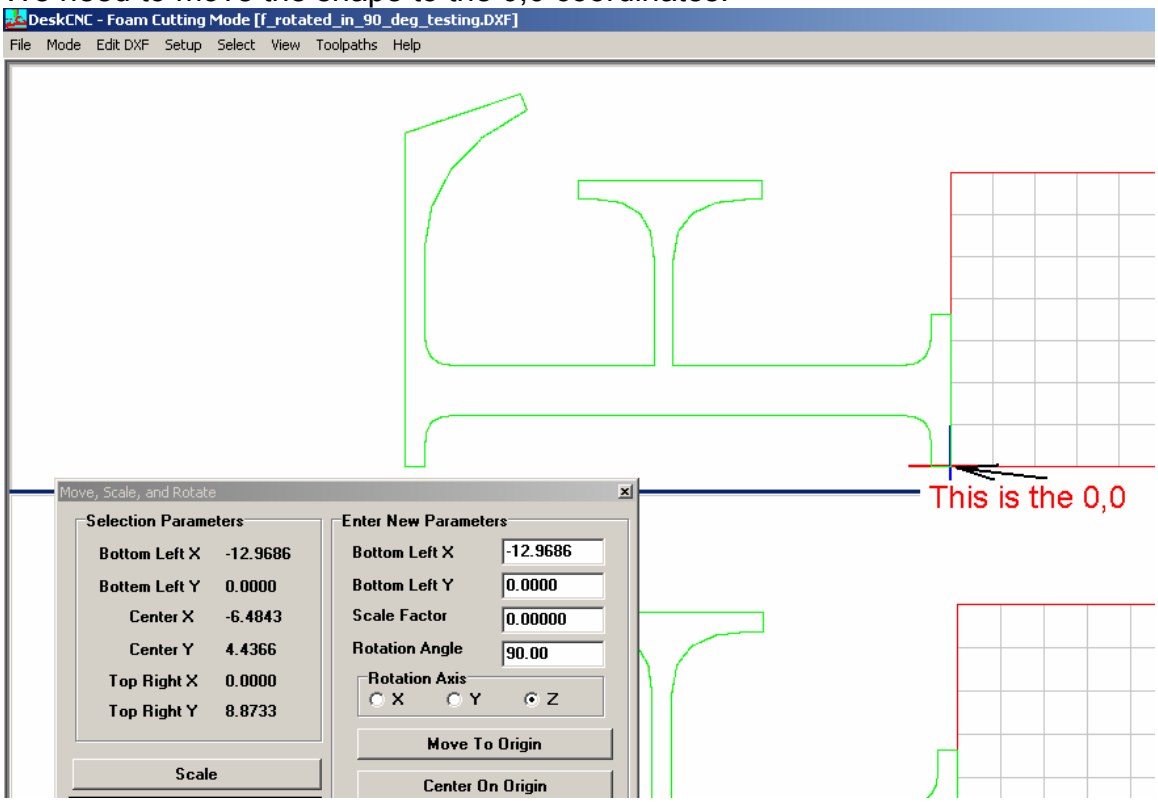


Load the new image by clicking on the “File”, then “Foam Cutter”, then “Open Top/Left/Large DXF Contour” as shown above



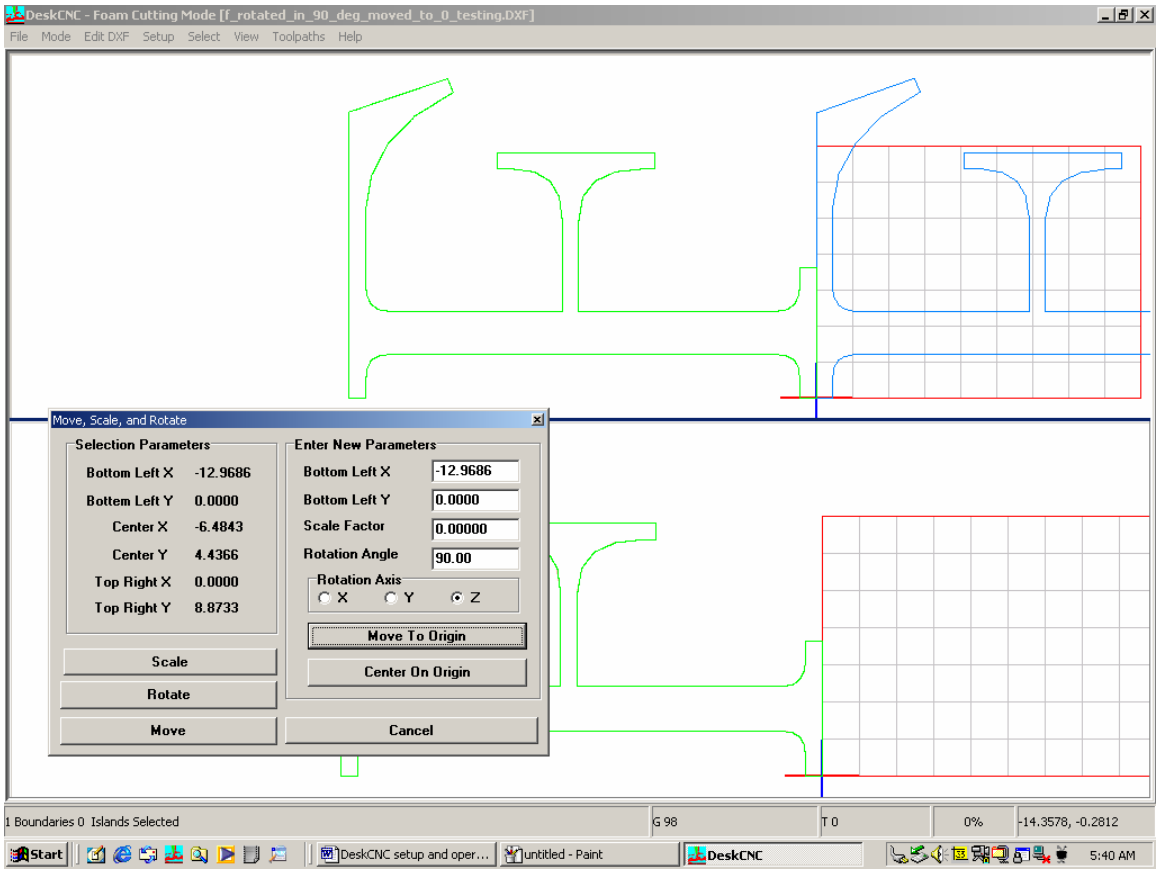
Choose the new file name and click on the “Open” button

The new shape is now loaded but it is on the left side of your grid – in other words, if you try to cut this shape, you will notice that the cut will be outside the foam.
We need to move the shape to the 0,0 coordinates.



In order to move the new shape we must first select it – as before click on the “Select” Menu, then select All.

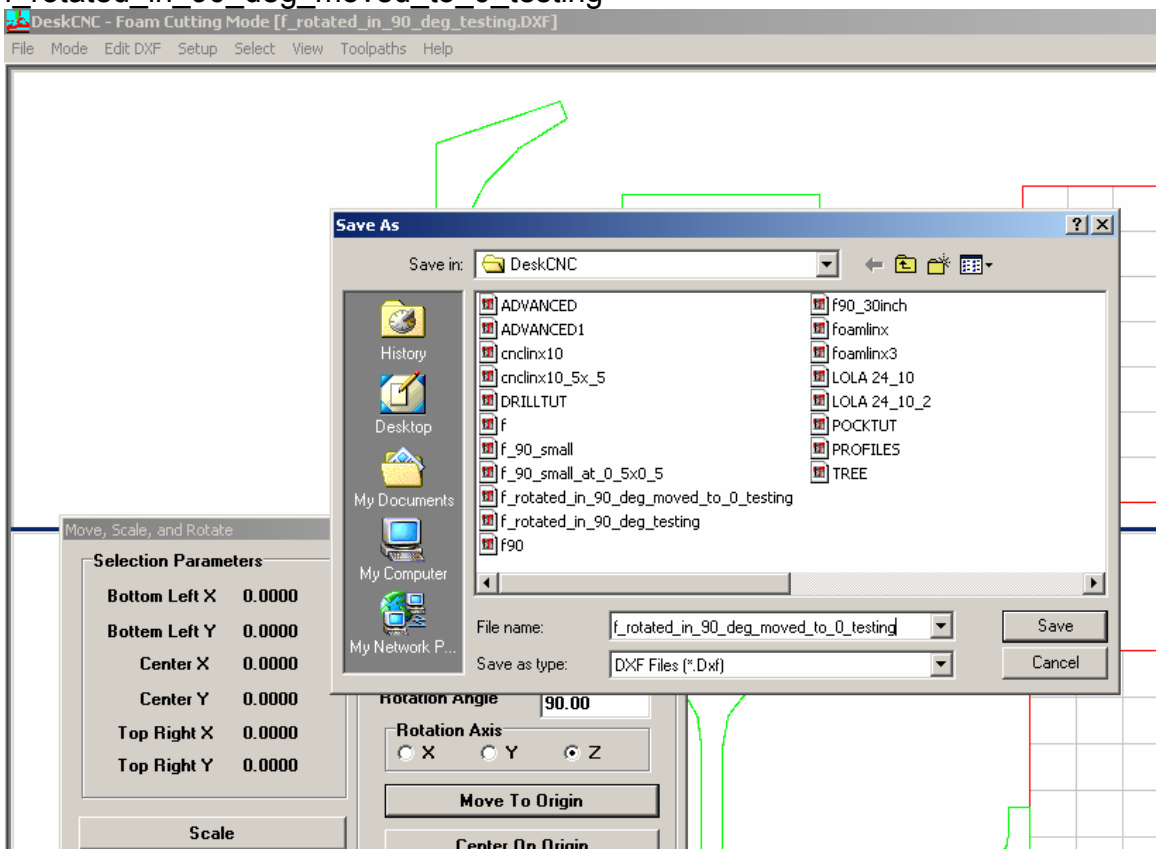
Once selected, click on the “Edit DXF” and choose the “Move, Scale and Rotate” option



Click on the “Move to Origin” button on this new window as shown above and you will see that the new shape (now in Blue) has moved to the right side of the coordinates.

Now save the new image with a new name say f_rotated_in_90_deg_moved_to_0_testing

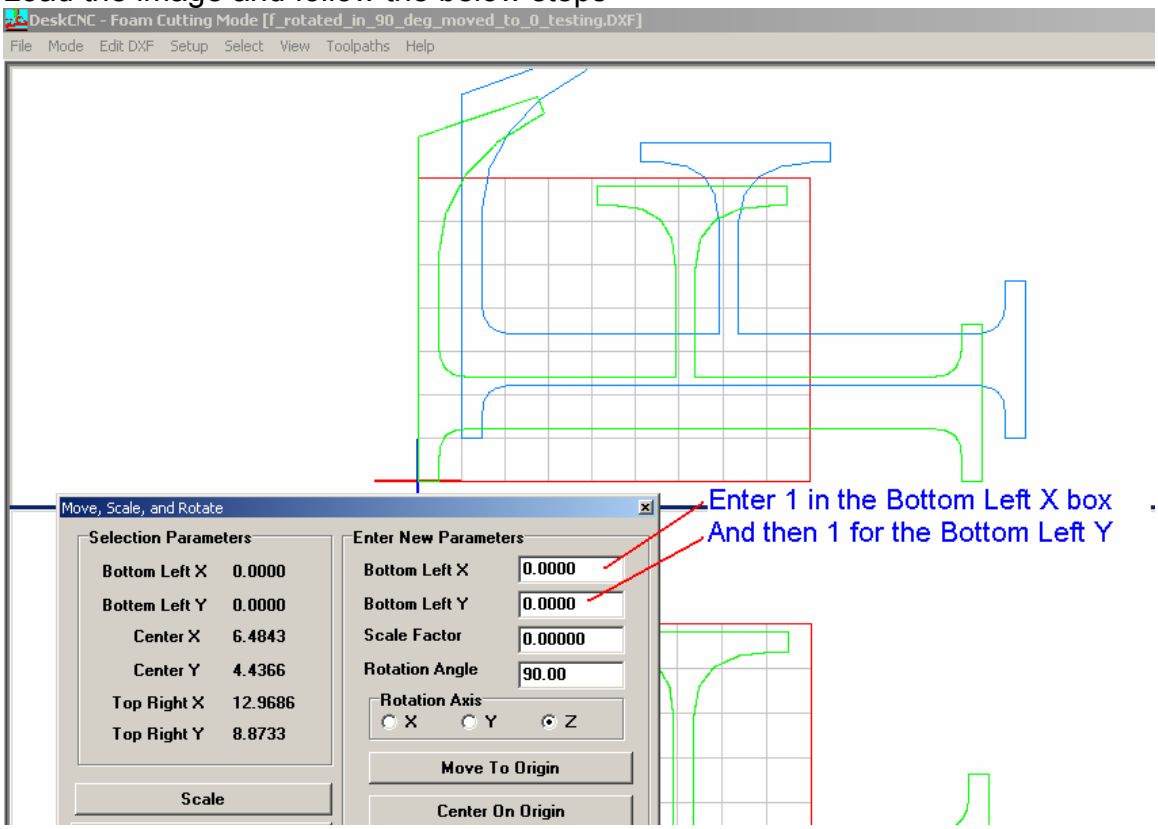
Like before, save the new shape by clicking on the “File” menu, then “Save Geometry DXF” option, and give the new name,,, say f_rotated_in_90_deg_moved_to_0_testing



Now click on the “Save button”

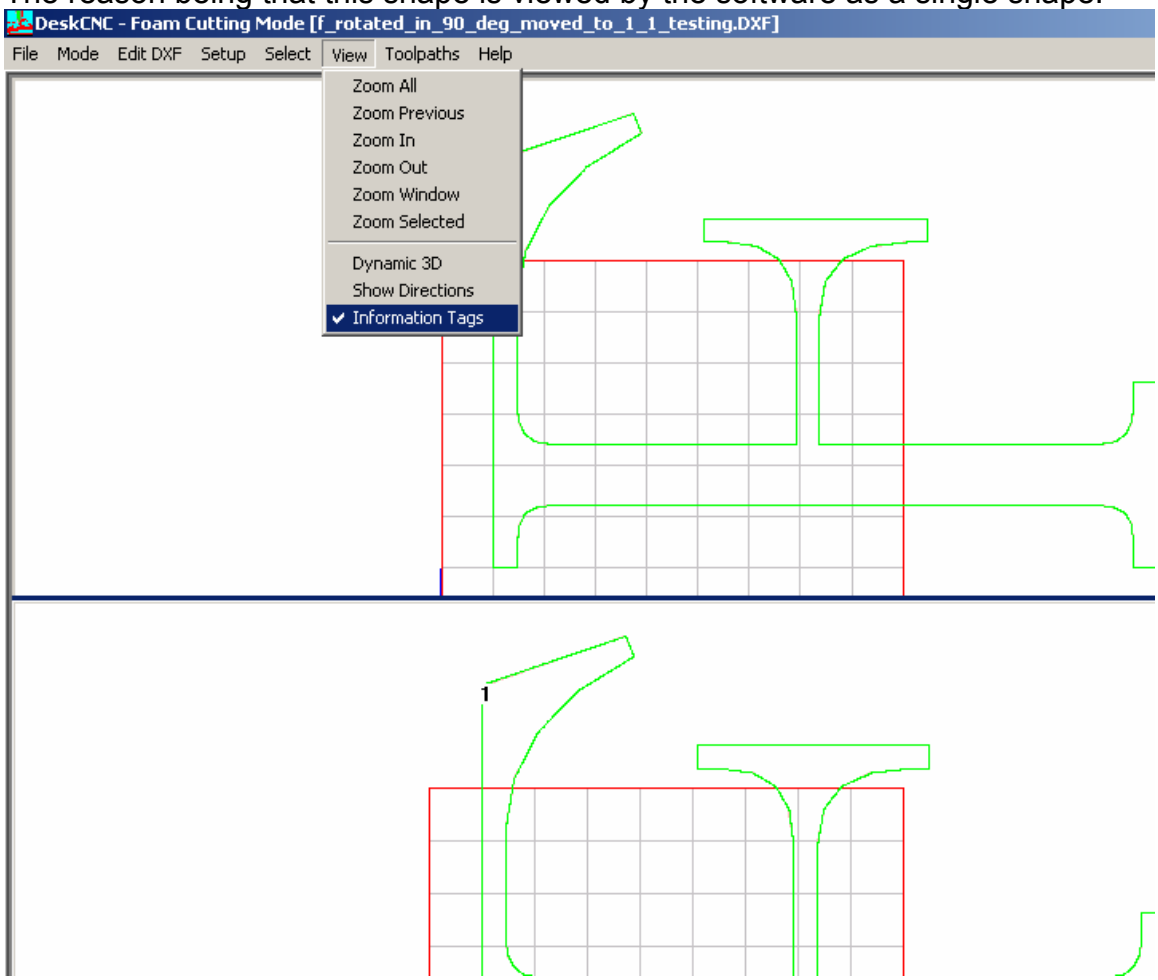
There are a few more options that can help you manipulate the shape, like resizing, and moving, we will not show all here, but try to experiment with these options to get familiar with them. We find that the most useful features are moving scaling and rotating the shapes

Now lets move the shape to 1,1 coordinates
Load the image and follow the below steps



Once you entered 1 for the bottom left x box and 1 for the bottom left Y box, click on the “Move” button, and you should see that the new shape now in blue has moved.
Like before, save this new shape under a different name, and then load this new image

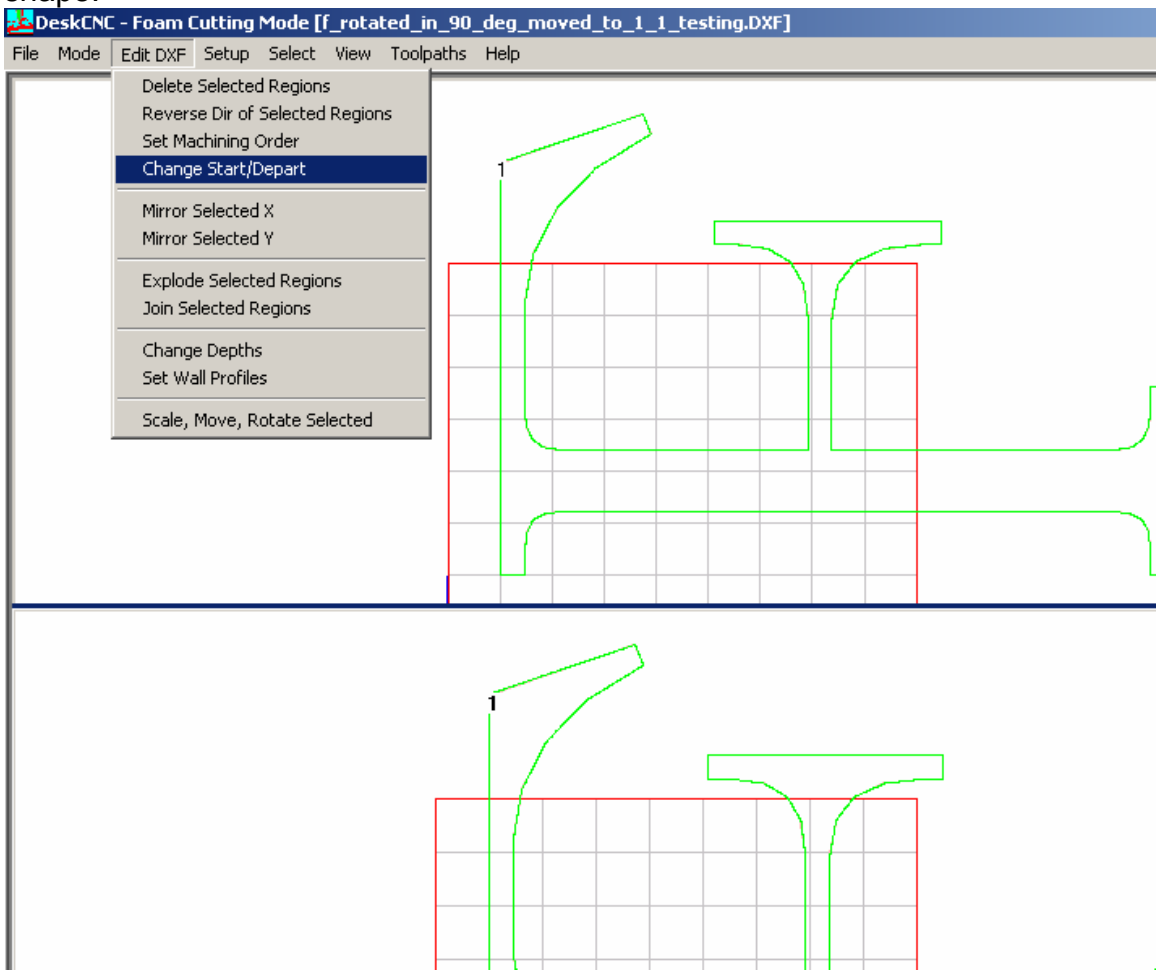
Now load this new shape:
Once the new shape is loaded, click on the “View” button and choose the “information Tags”.
Information are numbers that tell you where the machine will start cutting and where it will move next. In our example you will notice that is only one number. The reason being that this shape is viewed by the software as a single shape.



The number 1 also tells you that the software chose this point and it will start cutting from that point and finish there as well.

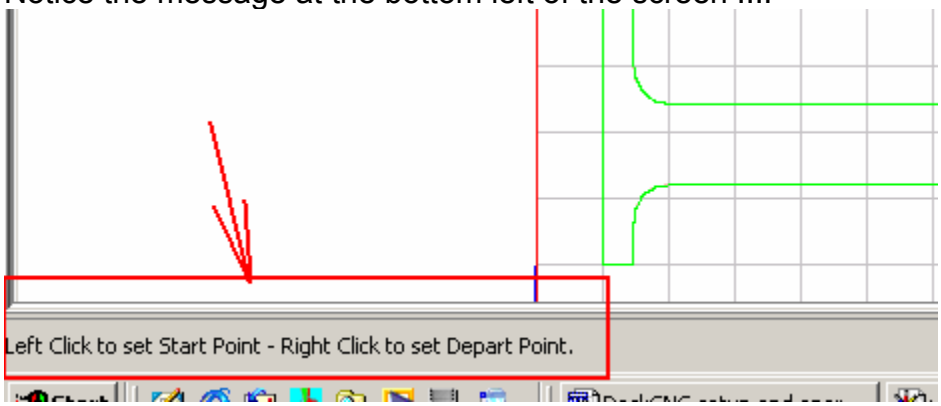
At this point we would like to move this point to a different position. The reason for moving this point is that the machine always start cutting from 0,0, and we would like it to start the actual cut as close as possible to the starting position.

In this next step we will set the new starting position at the bottom left side of the shape.



Click on “Edit DXF” and choose “Change Start/Depart”

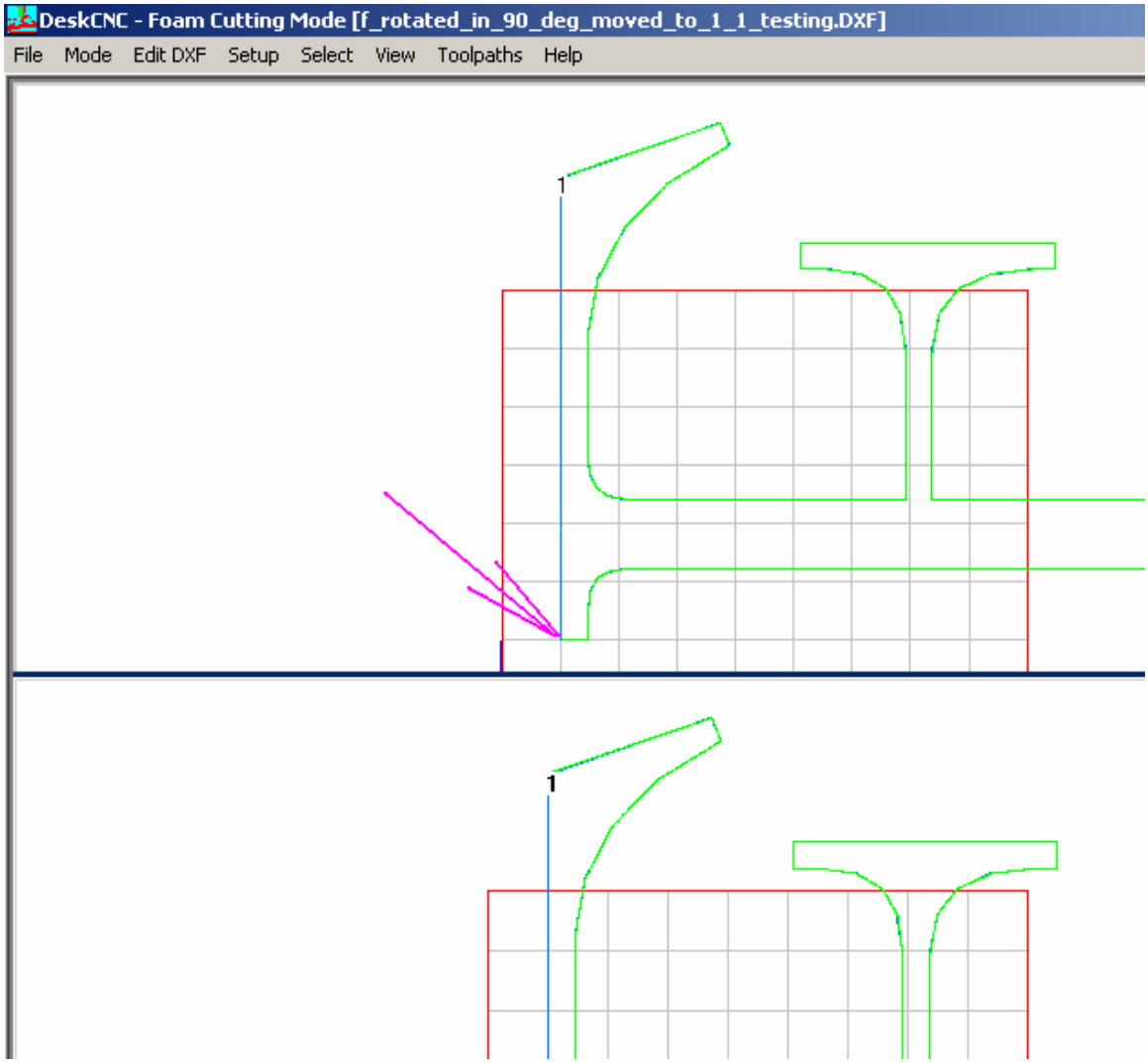
Notice the message at the bottom left of the screen !!!!



This step is very important!!!
You now have to use the two mouse buttons, the right mouse button and the left mouse button.

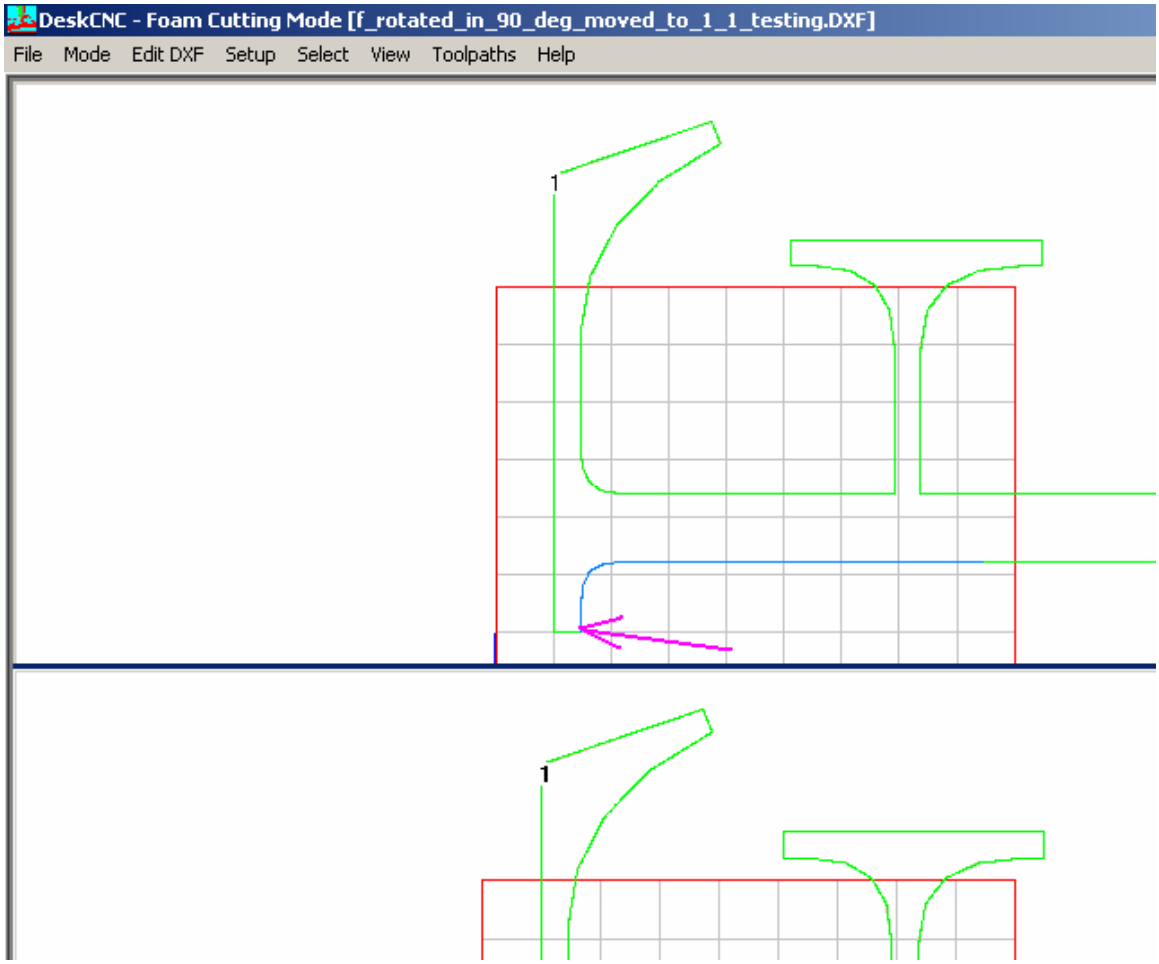
Like in the message in the screen shot, To set the staring point use the left mouse button and the set the depart point click on the right mouse button.

Place the mouse curser at the point where the purple arrow is pointing and click on the left mouse button



At this point you should see that the left side of the shape is now Blue.

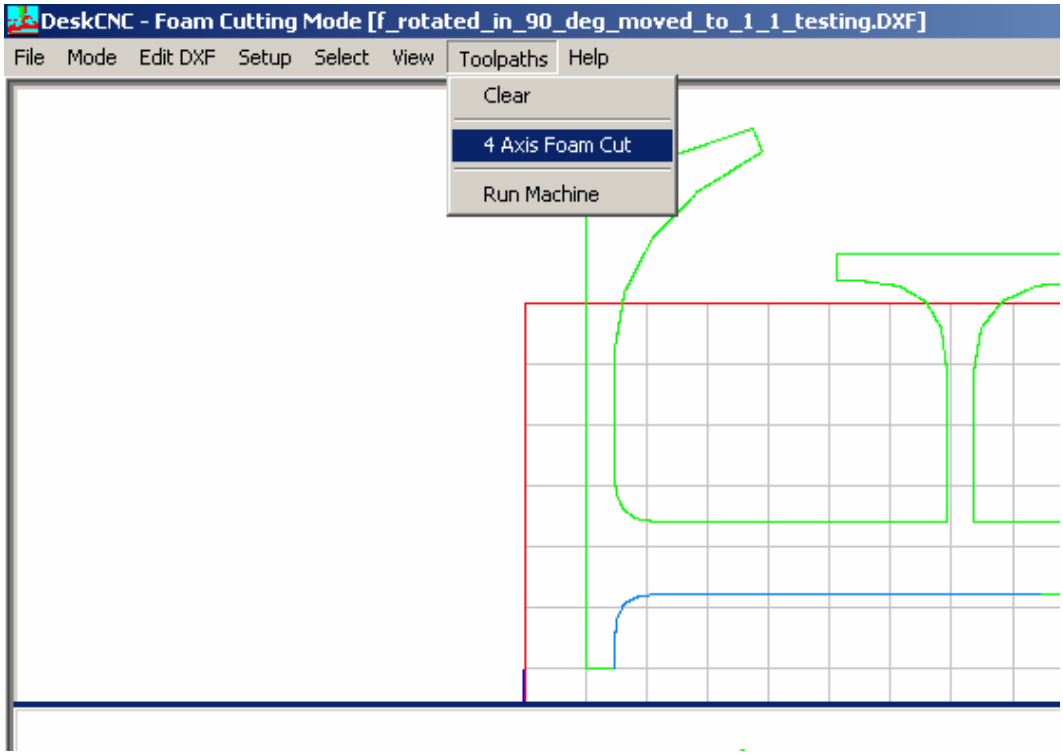
Now set the mouse curser to where the purple arrow is pointing to as shown below



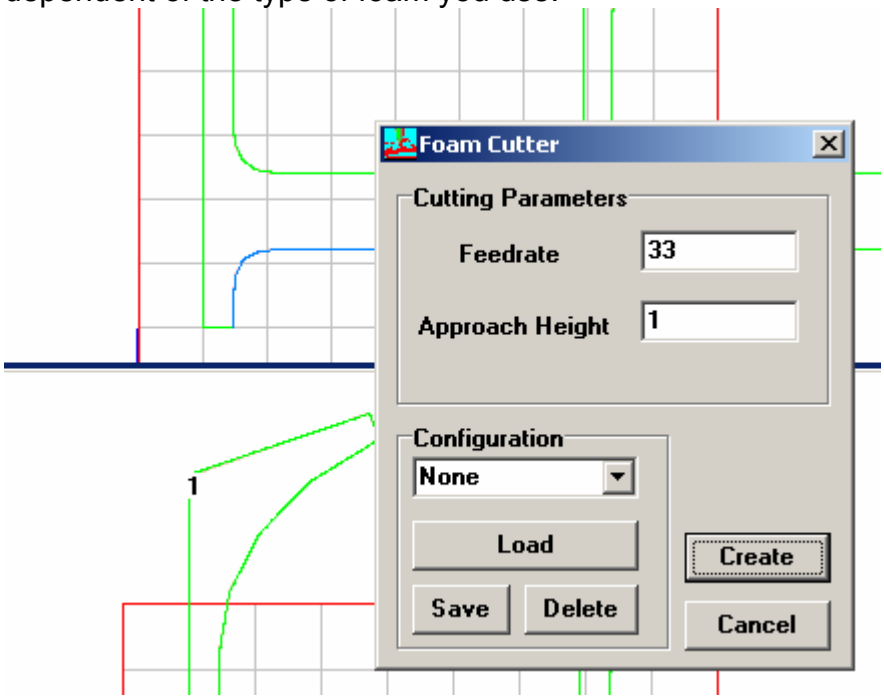
Notice that the bottom side of the shape is now Blue.
The above two steps told the machine to start from the bottom left of the shape and depart from the bottom right side of the shape (where the above purple arrow is pointing at)

We are now ready to create the tool path – tool path is the G code that runs the machine and tells it where to go step by step

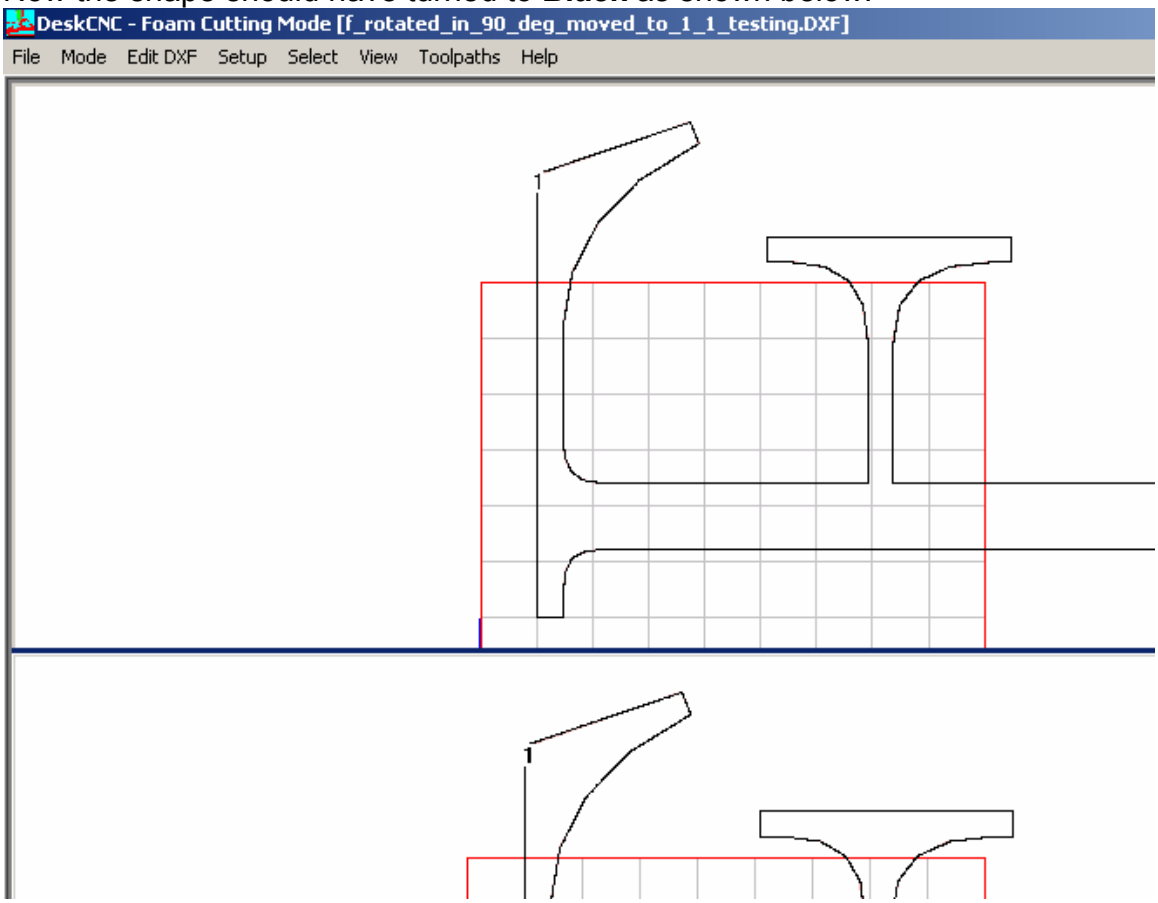
To create the tool path click on the “Toolpath” button and choose “4 Axis Foam Cut”



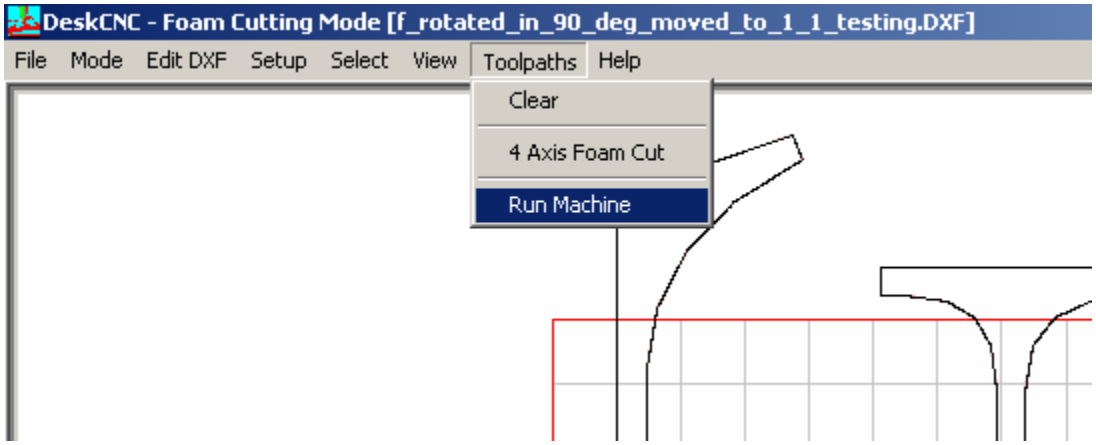
Enter the Feed rate – in this example we used 33 inch per min, Feed rate is dependent of the type of foam you use.



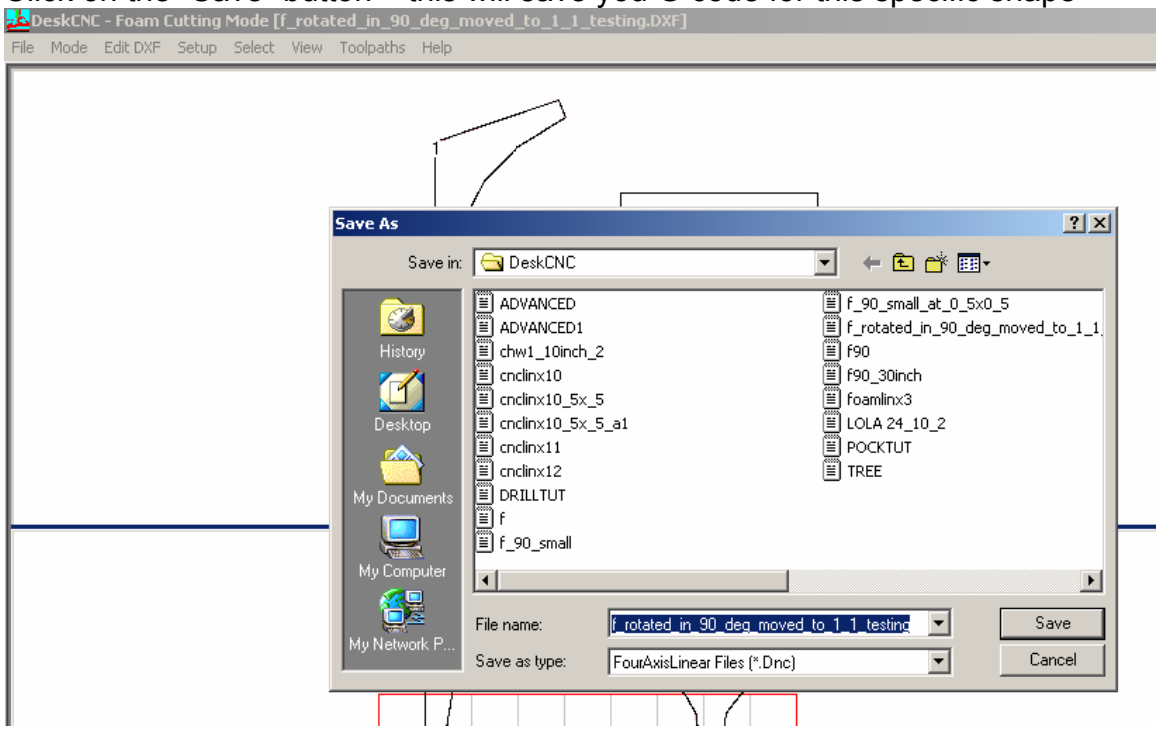
Click on the “Create” button.
Now the shape should have turned to **Black** as shown below.



Now click on the Toolpaths button once again and choose “Run Machine”



Click on the “Save” button – this will save you G code for this specific shape



Cutting multiple shapes – nesting

Nesting is used when there are more then a single shape to be cut.

The goal is to place the shapes as close as possible to each other in order not to waste more material then needed.

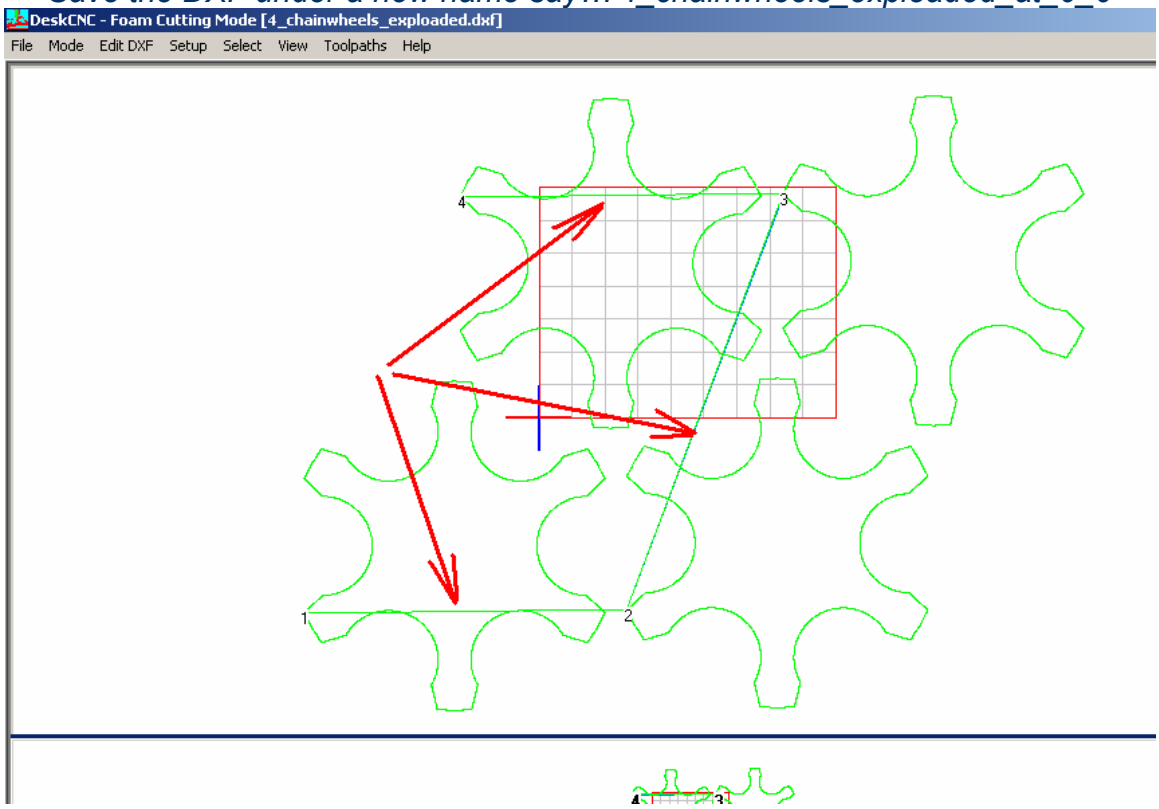
Placing the shapes is done with the design software – AutoCAD, CorelDraw or any other drawing software that can save its output as DXF file.

Once the file containing multiple shapes, load it into DeskCNC just as a single shape was done.

In the following example, we take a file containing 4 shapes, and prepare it for the cut.

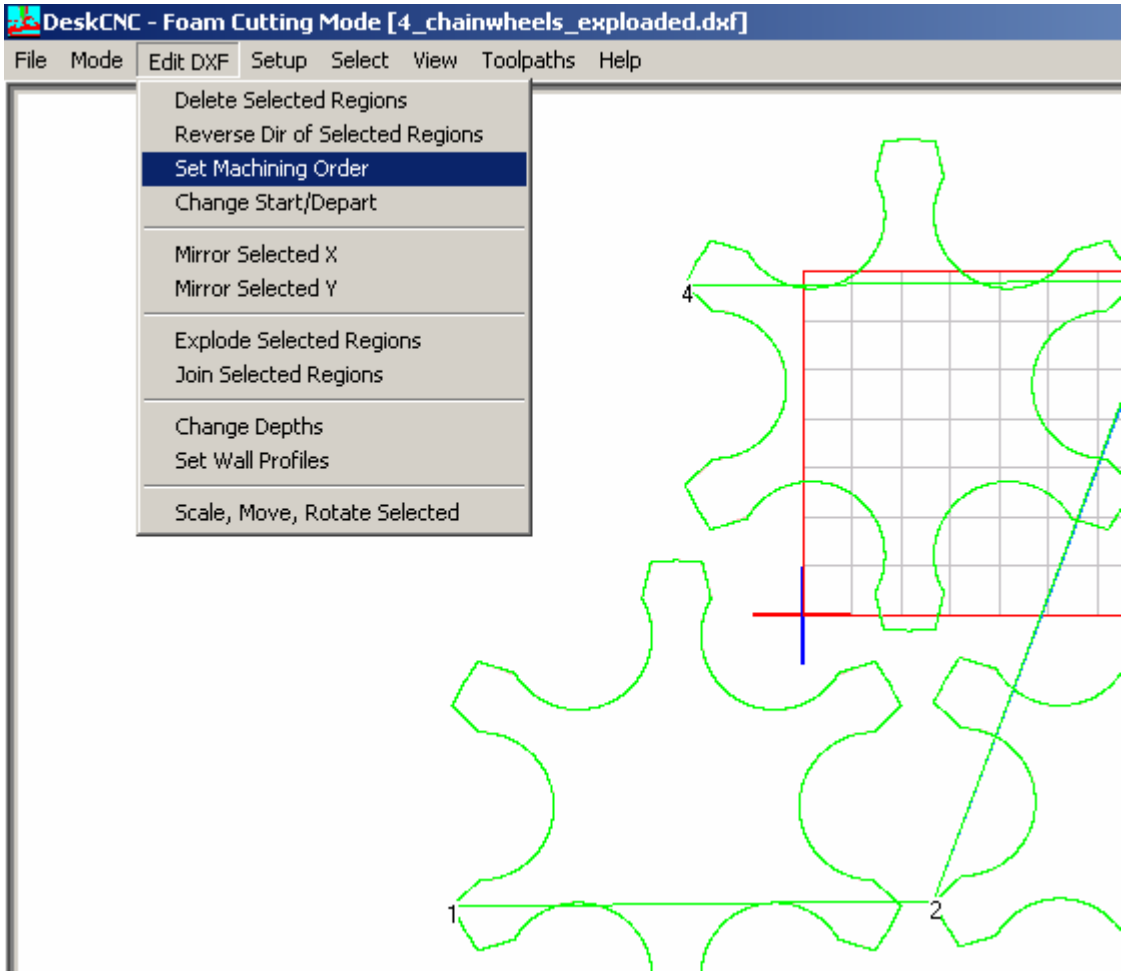
In this example we will load a DXF file which has 4 shapes:
From the “File” menu choose “Foam Cutter” and load the file 4_chainwheels_exploaded.dxf, this file can be found on the CD included with your machine.
Software detected 4 shapes and it numbered then 1,2,3,4

In this example we did not move the 4 shapes to Origin.
In order to move the shapes to Origin, please follow the steps in the previous example - From the Select menu choose “Select All”
From the Edit DXF menu select “Scale move rotate selected”
Click on the “move to Origin” – at this point you should see the new 4 shapes in blue color moved to the origin.
Save the DXF under a new name say… 4_chainwheels_exploaded_at_0_0

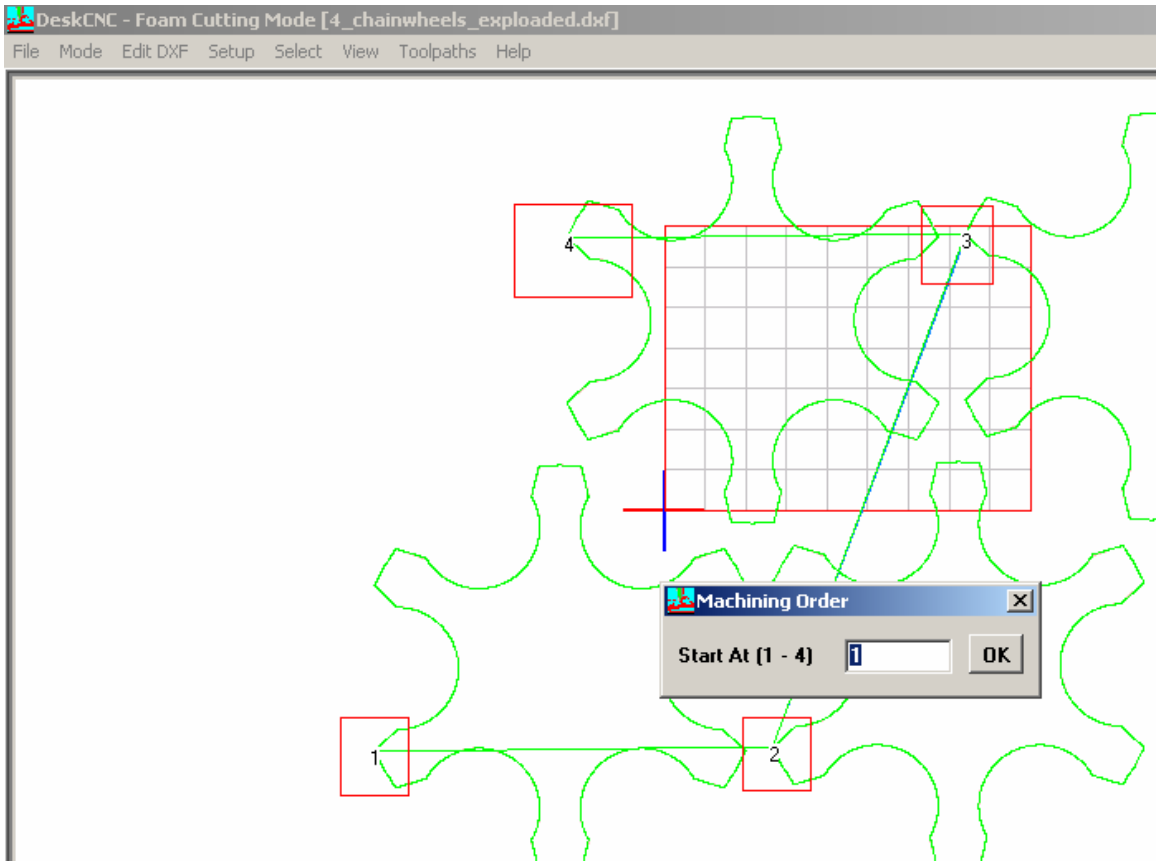


In the screen shot above, the Green lines with pointed by the red arrows show the path that the software decided to run between one shape to the other, if you would to cut these shapes with this configuration, the shapes would be cut, but in addition to the shape, the lines pointed with the red arrows would be cut as well, ruining the project.
To fix this we would need to set the order of the cut as well as to tell the software where to start cutting each shape and where to exit each shape.

Changing the machining order – this step will tell the software which shape will be cut 1st and which 2nd and so on



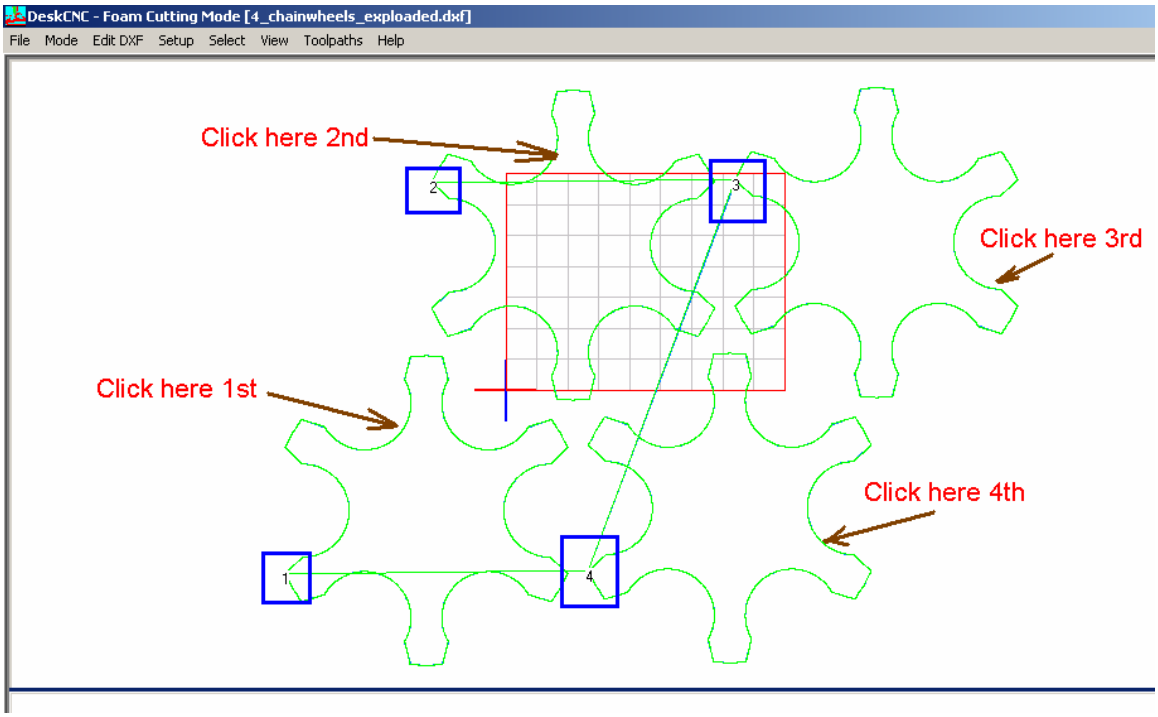
Click on “Edit DXF” and choose “Set Machining Order”



Notice the numbering with the red square around them – these will change in the next steps.

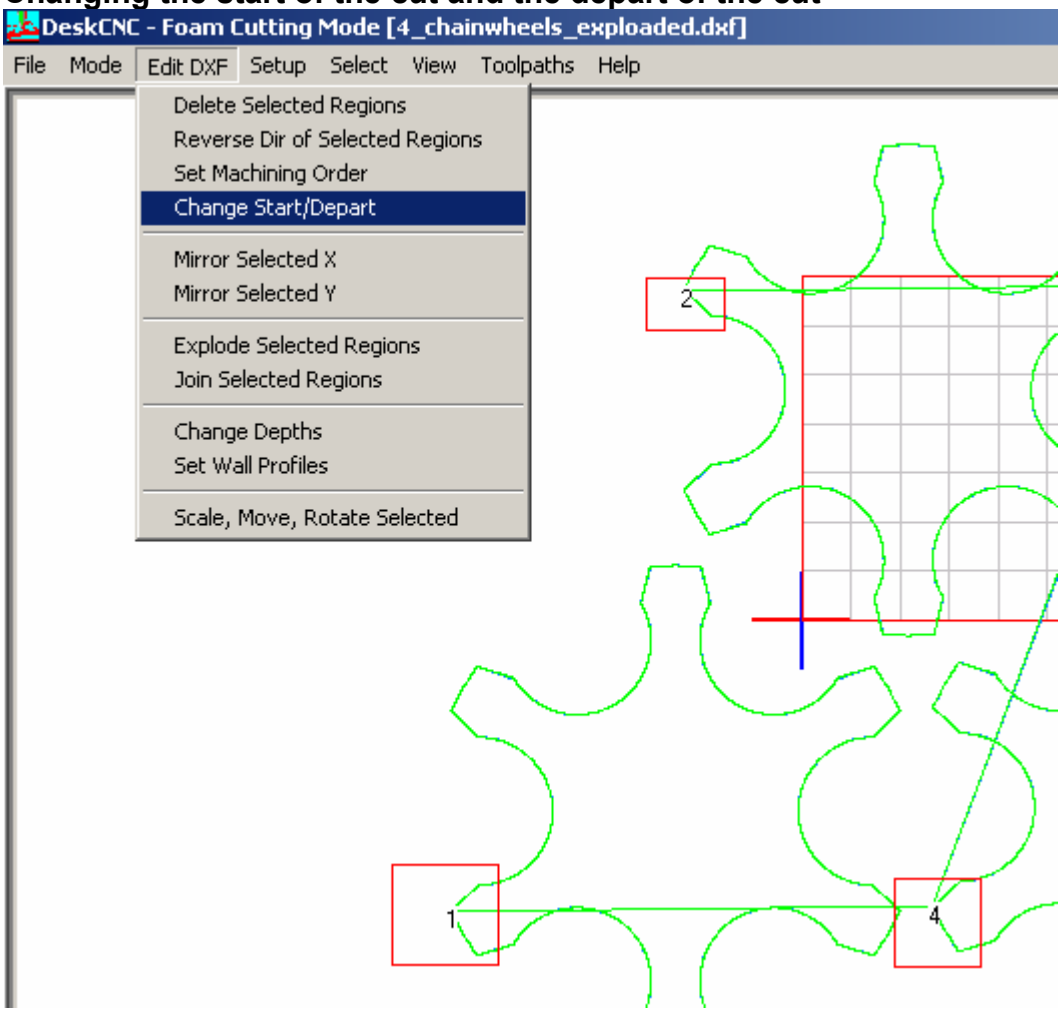
Software detects 4 shapes – click on the OK button

Click on each shape (it does not matter where on the shape you click).
Each click will change the number on the shape indicating the sequence it will be cut



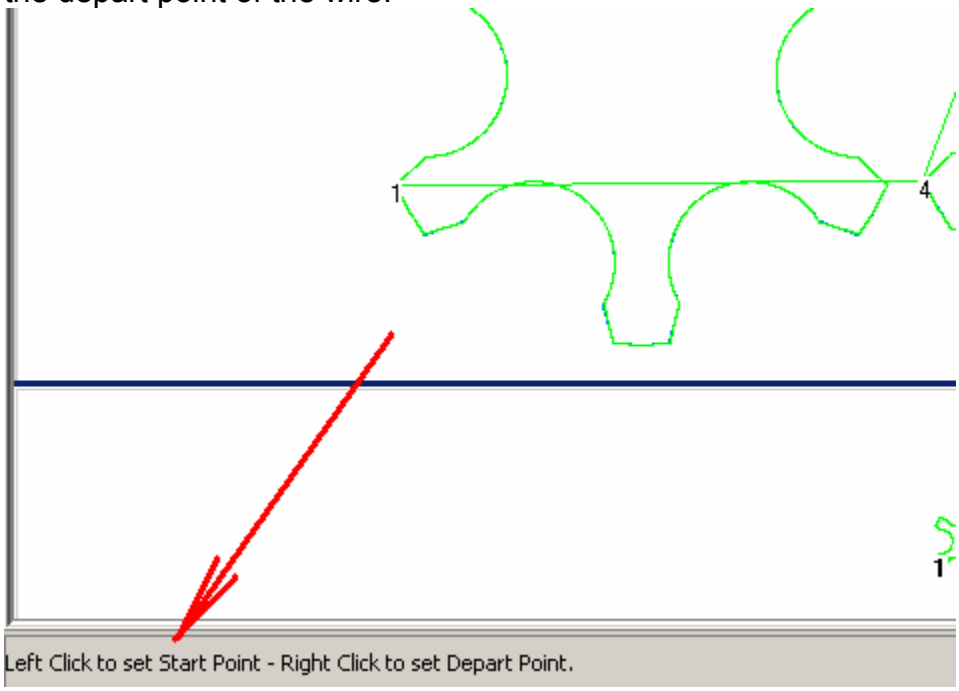
Notice that after this stage the sequence is a bit different then the sequence that the software decided once the file was loaded. The new sequence is now marked with blue squares around the new numbering.
You can set any sequence you like, there is no wrong sequence, but the idea is tell the machine to move from one shape to the other in the shortest route.

Changing the start of the cut and the depart of the cut

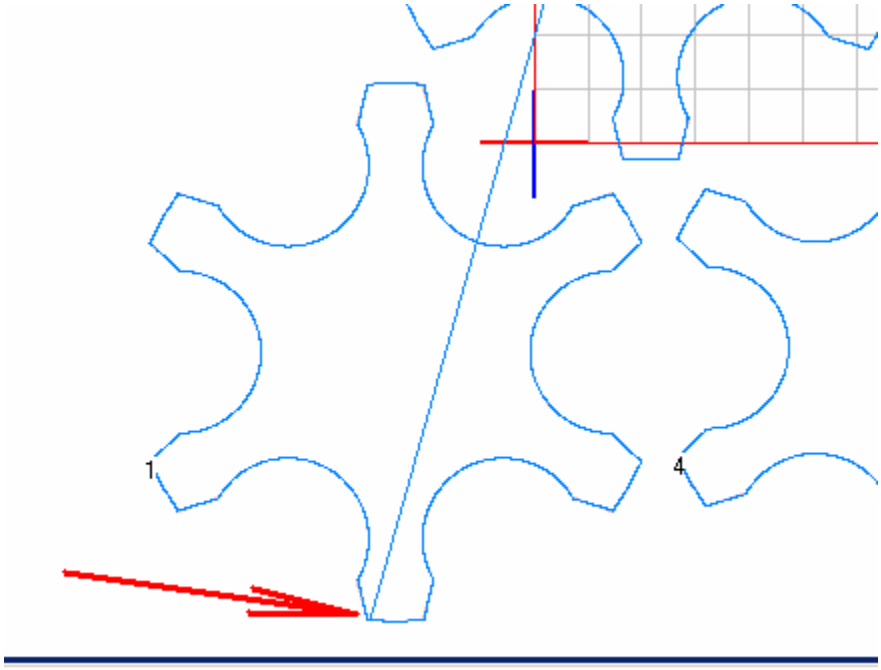


In the next few steps, we will change the start of the cut for each shape as well as the depart point of each shape.
The Start point is where the wire will first hit the shape and start the cut, the depart is where the wire will exit the shape in order to move on to the next shape.
The goal is to get the wire to move from one shape to the other in the minimum distance.

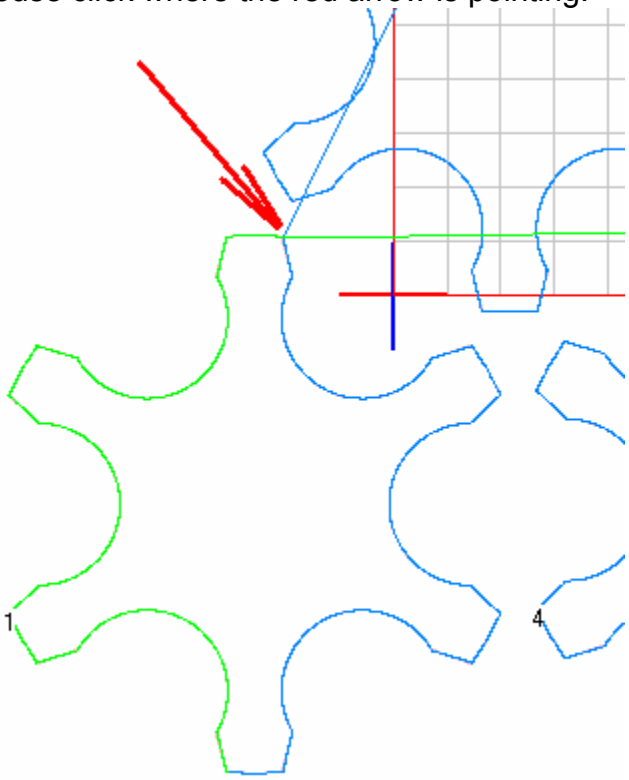
Once you click on “Change Start/Depart” option you will get a message on the bottom left side of the screen. Two of the mouse buttons will be used in this phase. The Left mouse button to set the start and the Right mouse button to set the depart point of the wire.



Left mouse Click at the point where the arrow is pointing as shown on the screen shot below. Notice that the line from shape number 1 is now connected to shape number 2

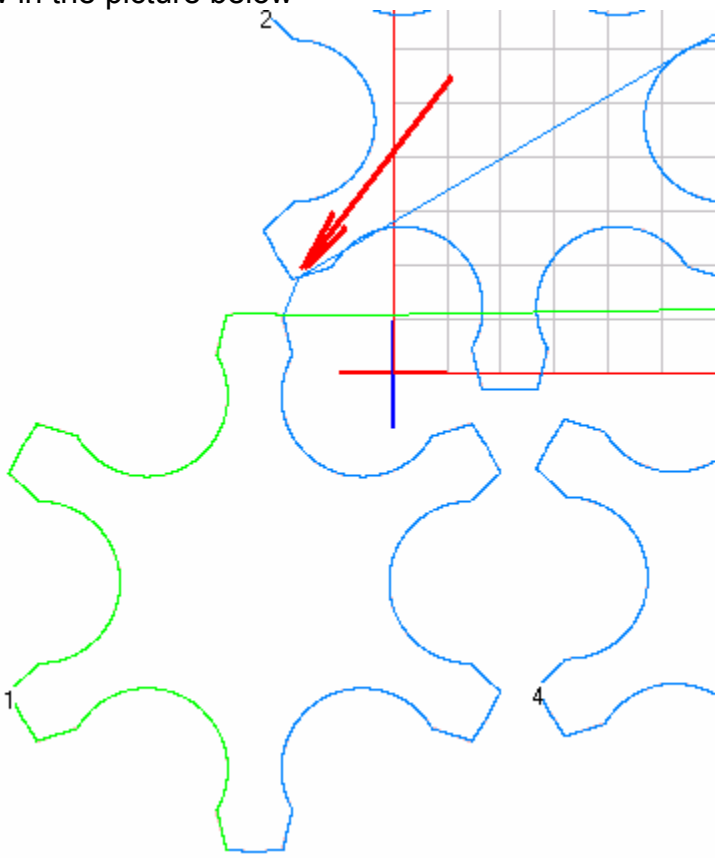


Now we need to set the depart point out of shape number 1
Right mouse click where the red arrow is pointing.

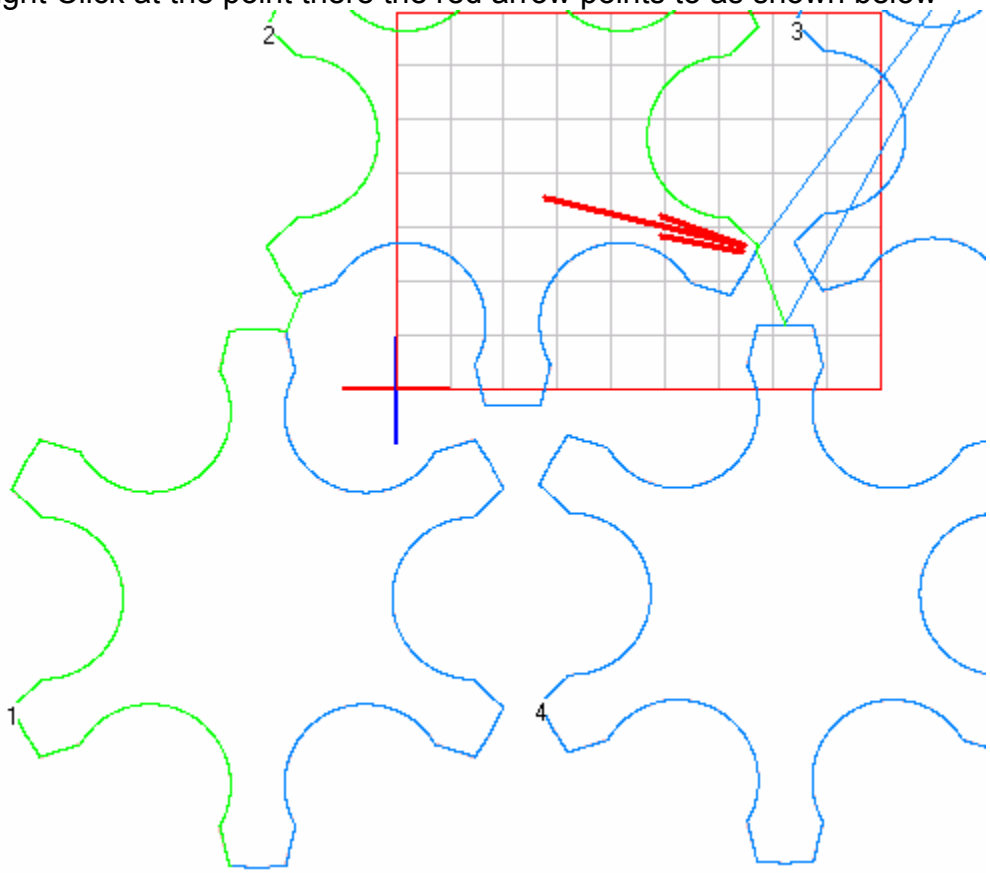


At this stage we set the depart point of the wire from shape number 1.
The next step is to tell the software where we want to set the start point of shape number 2.
We will try to get the start point of shape number 2 to be as close as possible to the depart point of shape number 1

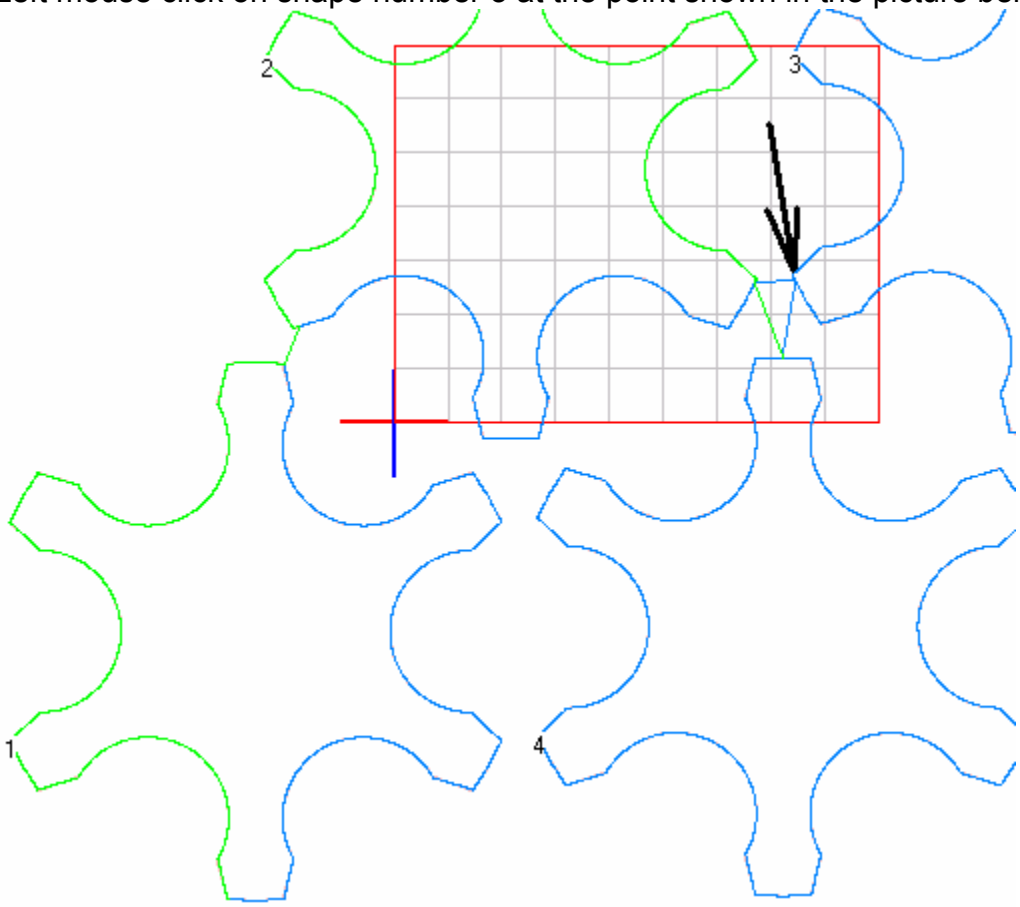
To set the start point of shape number 2, click on the point shown with the red arrow in the picture below



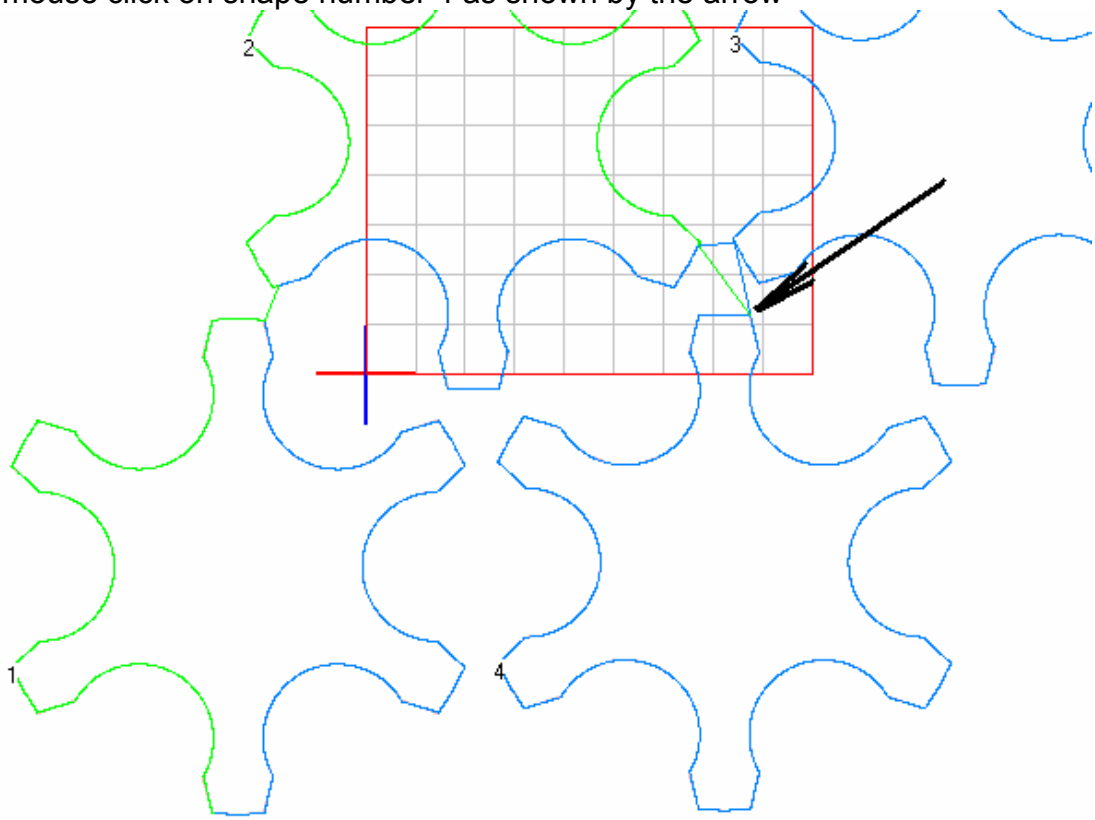
We will now set the depart point of shape number 2.
Right Click at the point there the red arrow points to as shown below



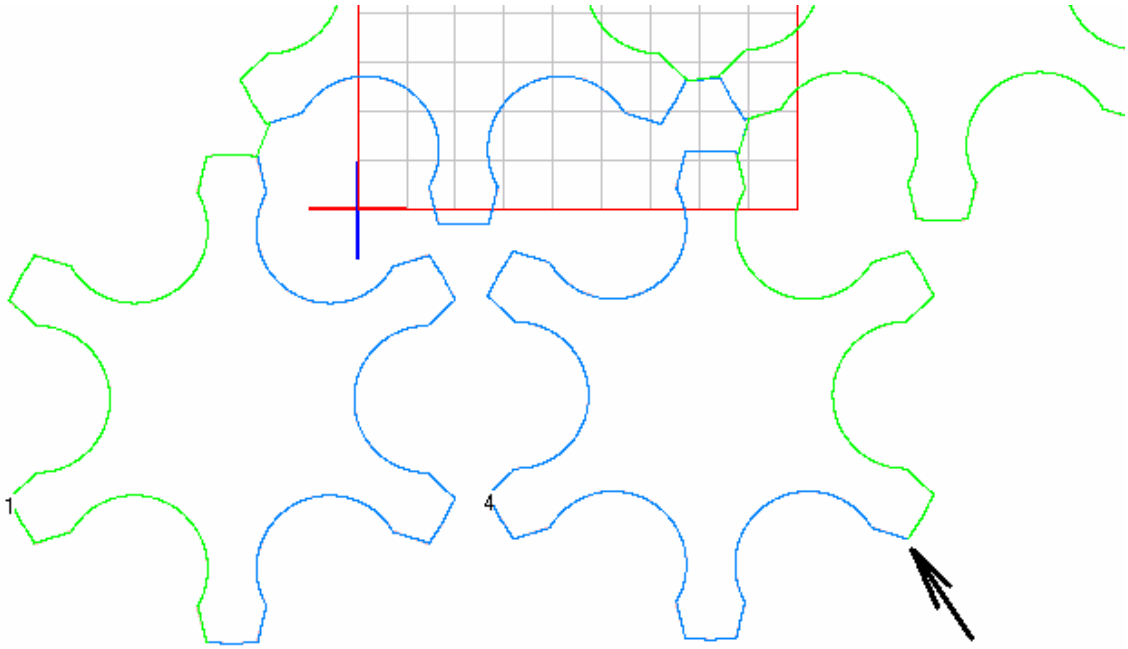
Now set the start point of step 3, again the goal is to have the shortest route connecting shape number 2 to shape number 3
Left mouse click on shape number 3 at the point shown in the picture below



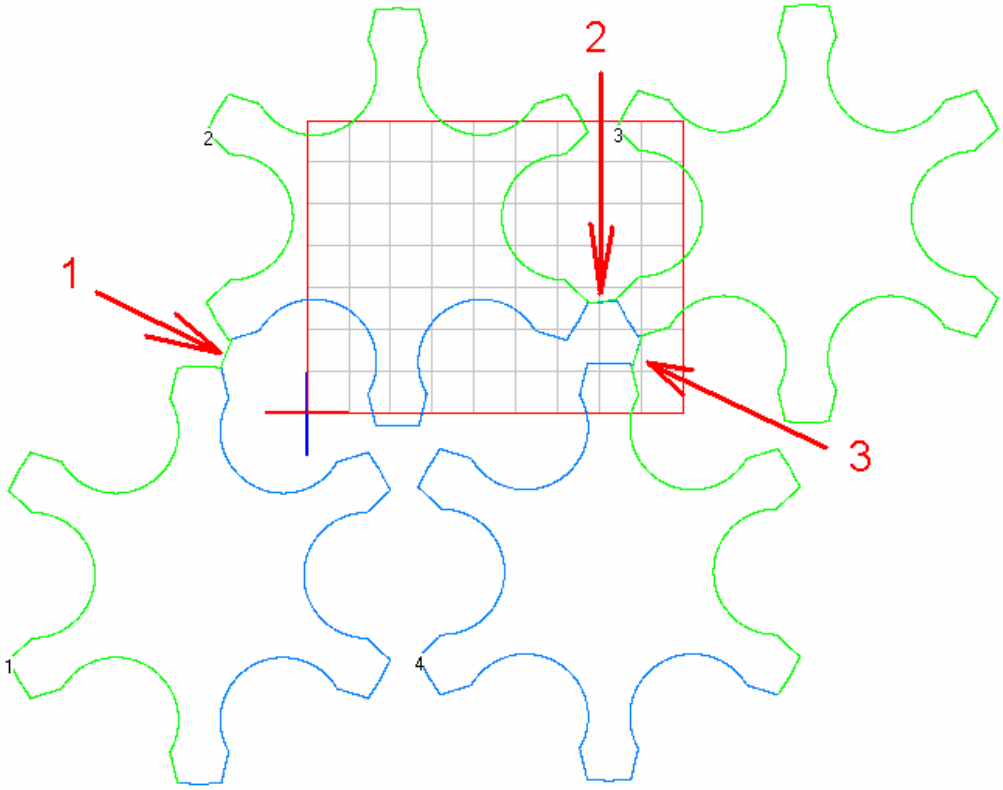
Now we will set the start point of shape number 4
Left mouse click on shape number 4 as shown by the arrow



The last step is to set the depart point of shape number 4
Right mouse click on the shape number 4 at the point shown with the arrow below

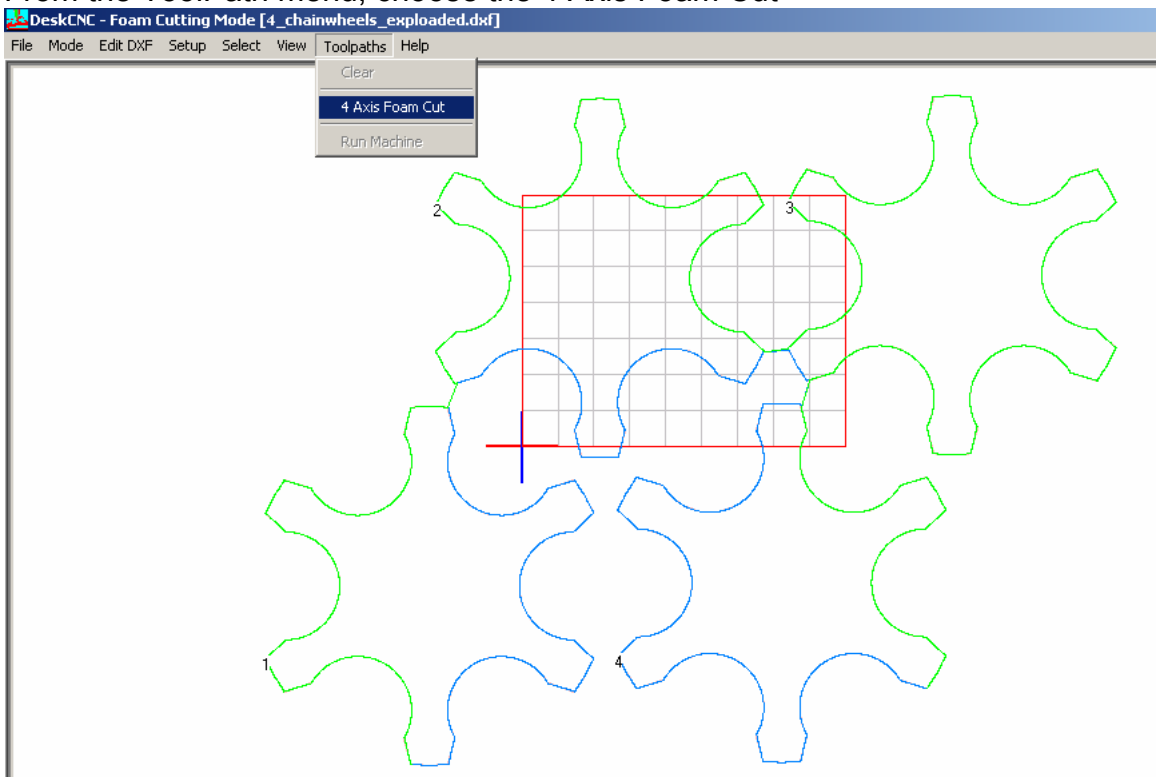


All in all there should have been 8 mouse clicks. Two for each shape.
A start click for each shape and a depart click for each shape.
In the picture below notice the red arrows – they show how and where the wire will exit one shape and go to the next shape

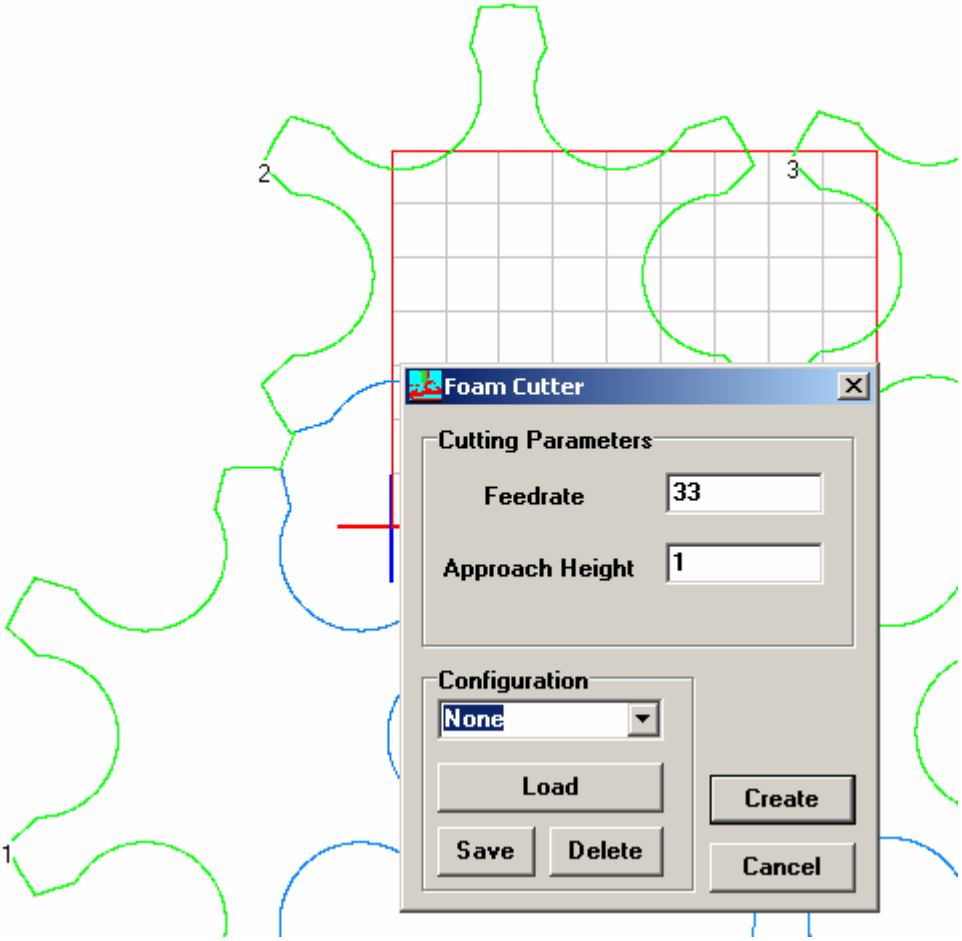


Notice that each shape has two colors, blue and green.
When the machine will start cutting, it will start from the bottom left side of the shapes, with shape number 1, it will follow the green path and continue to shape number two following the green path, it will continue to shape 3 and 4 on the green path.
Once in shape 4, it will start going back to the original starting point, but this time following the blue path .

The next and last step is to cut the shapes
From the ToolPath menu, choose the 4 Axis Foam Cut



Set the feed rate according to the foam used



Click on the “Create” Button, and save the file.
At this stage, the file name is the file containing the 4 axis G code that will run the machine.