

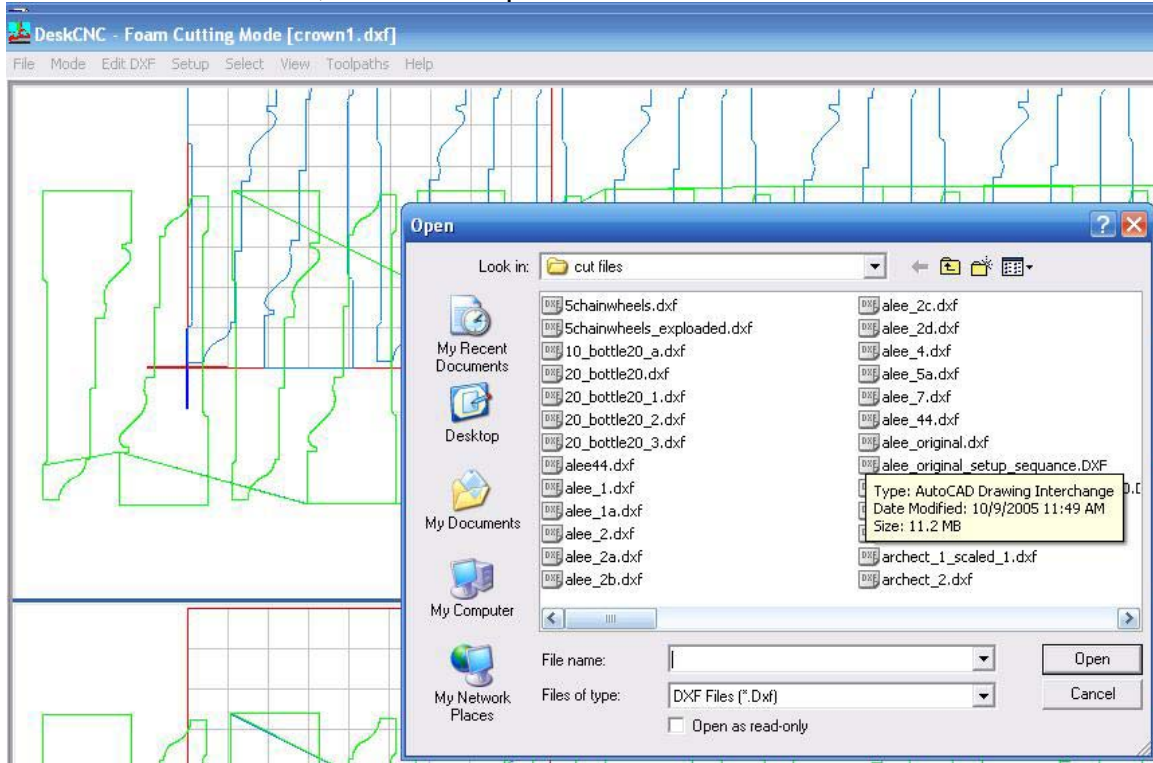
## Cutting multiple shapes with DeskCNC software

This document describes step by step how to cut multiple shapes using DeskCNC software

It is very important that you follow each step exactly as described in this document.

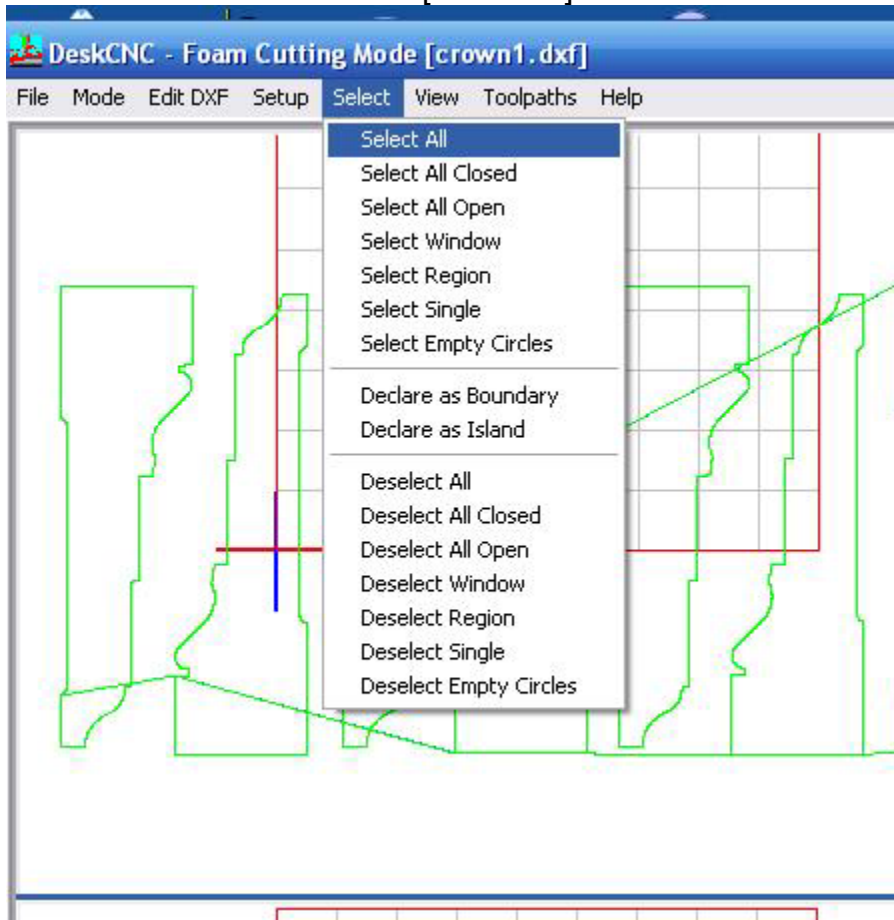
The shapes in this example can be found on the CD included with your machine. This will work for DeskCNC only !!!!

Load the desired DXF, in this example we will use the DXF named crown1.dxf

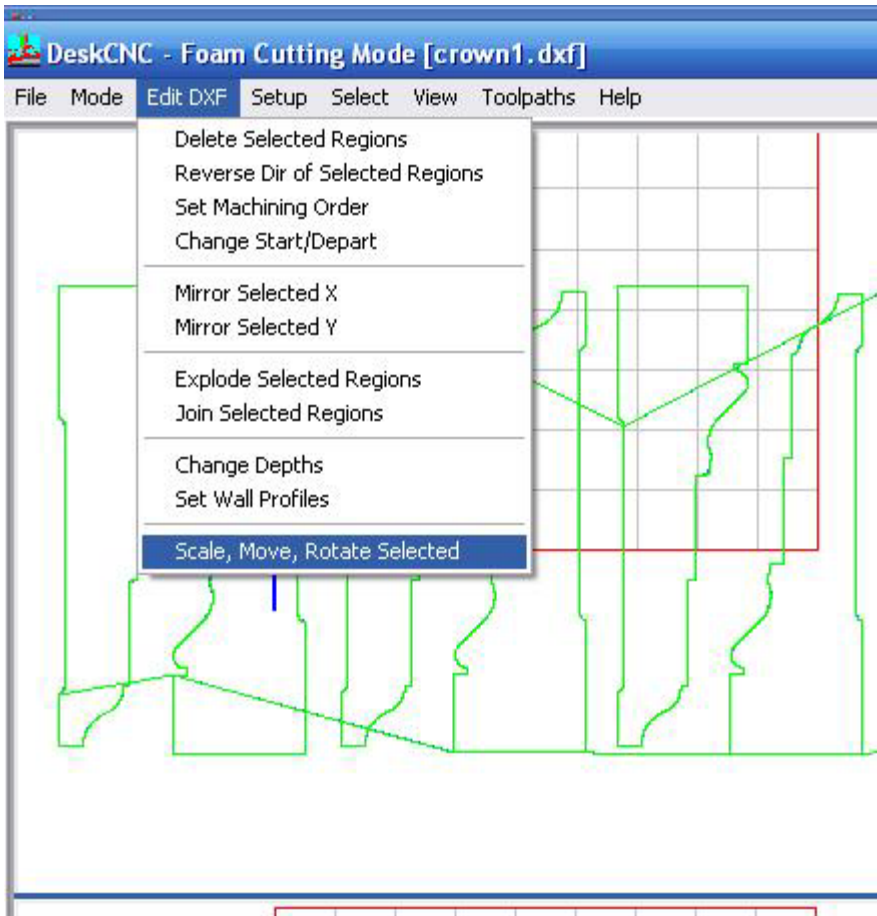


Notice that the shapes are not in the 0,0 position but are offset. The next step would be to move the shapes to the home position (0,0)

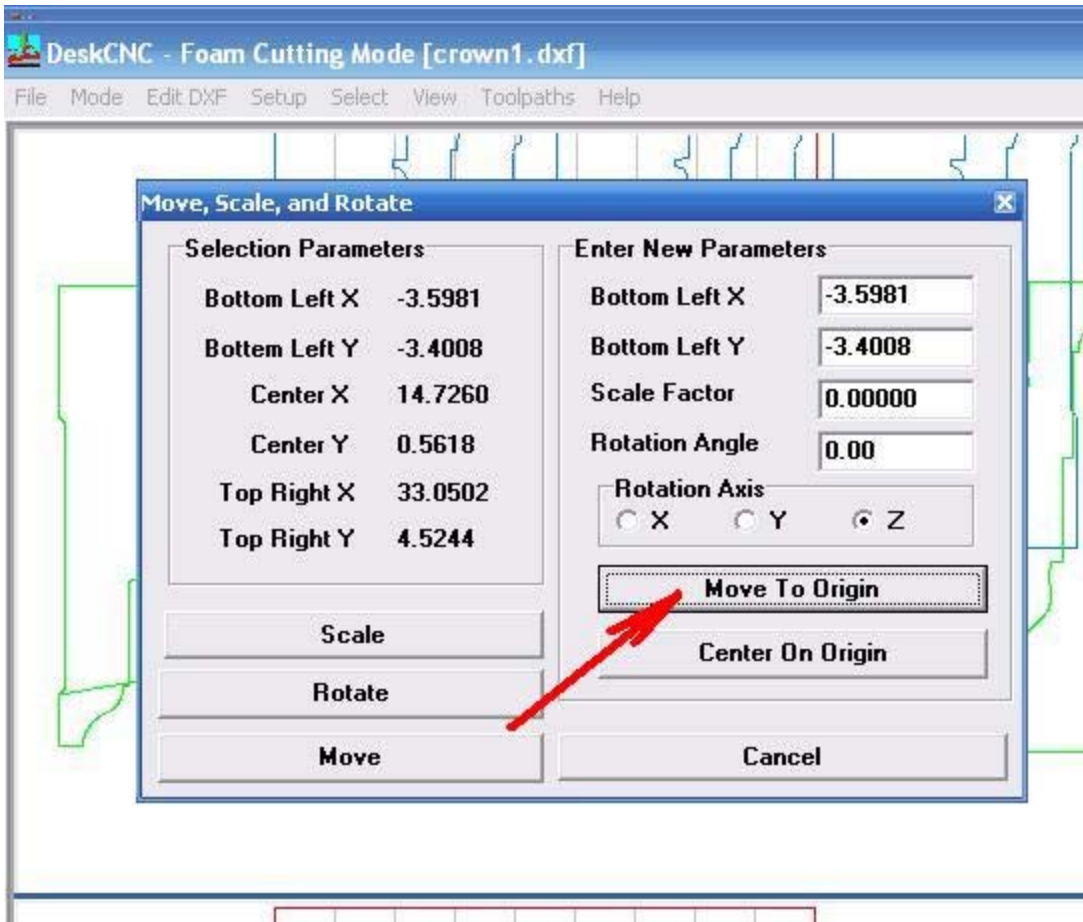
From the Select menu select [Select All] as shown below



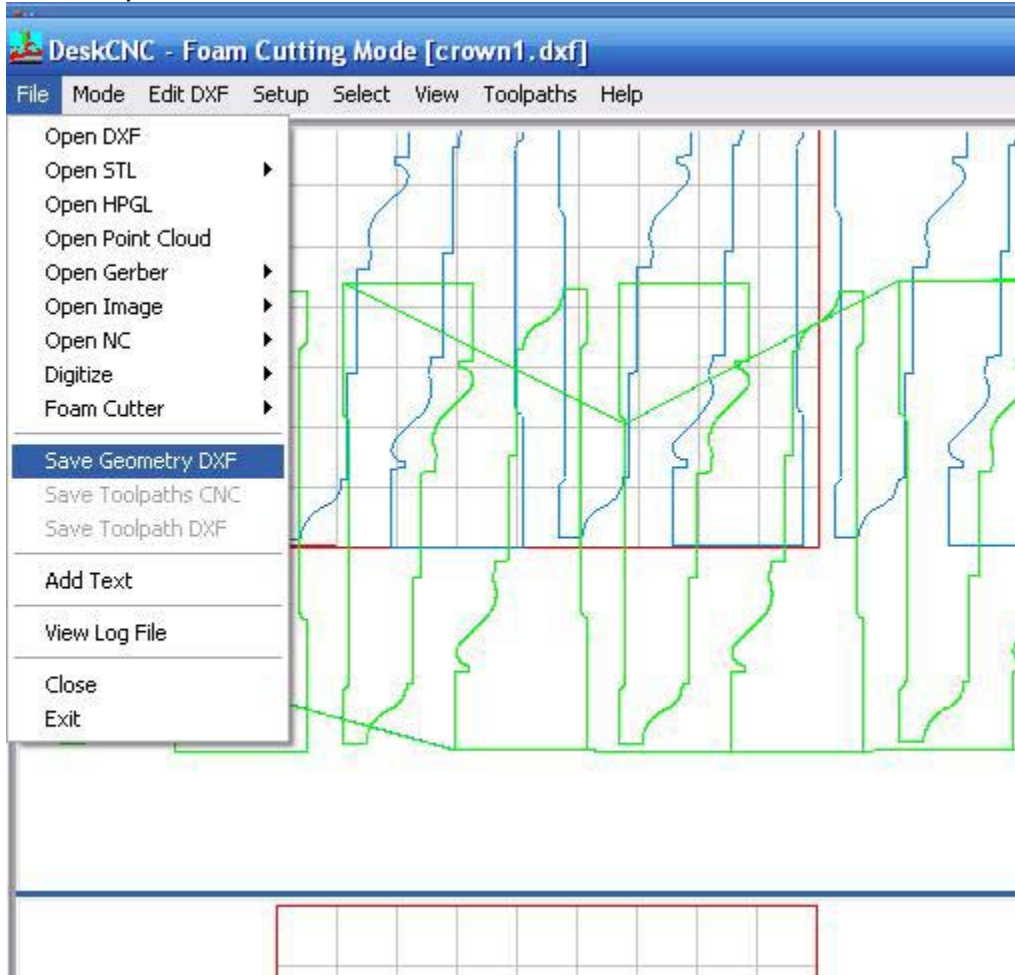
In the Edit DXF menu select [Scale, Move, Rotate selected] as shown below



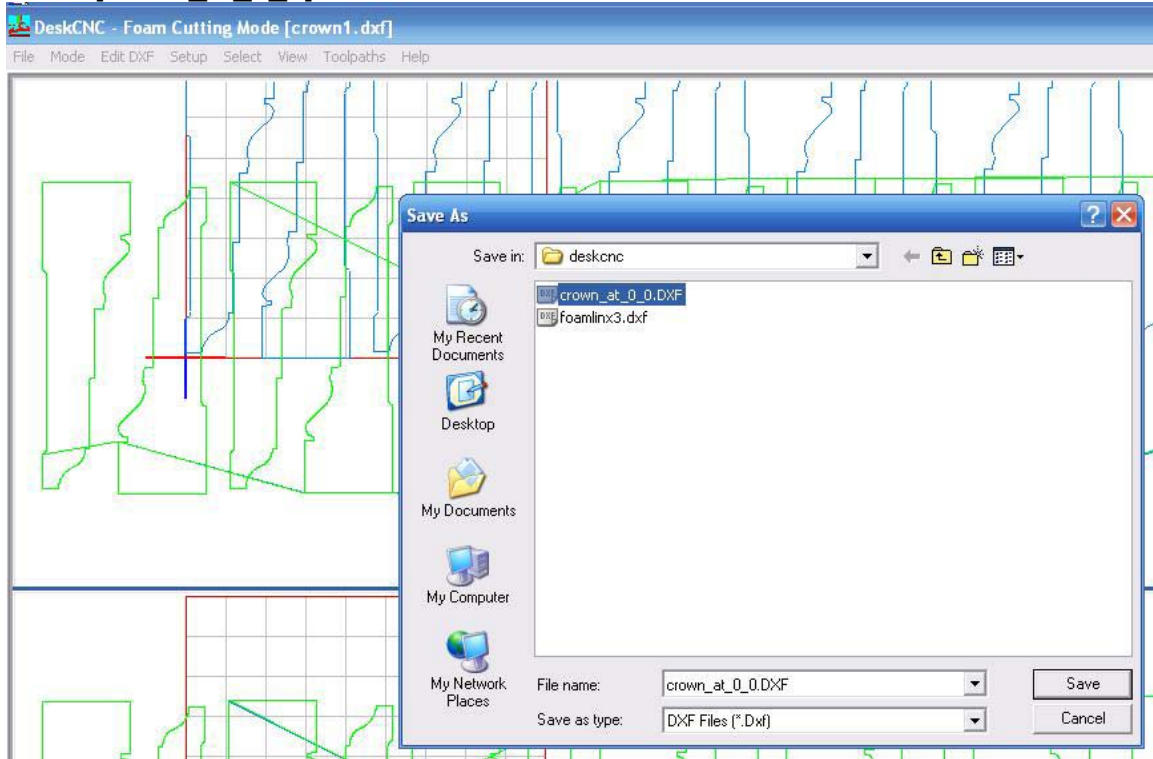
Click on the [Move to Origin] button as shown in the red arrow below



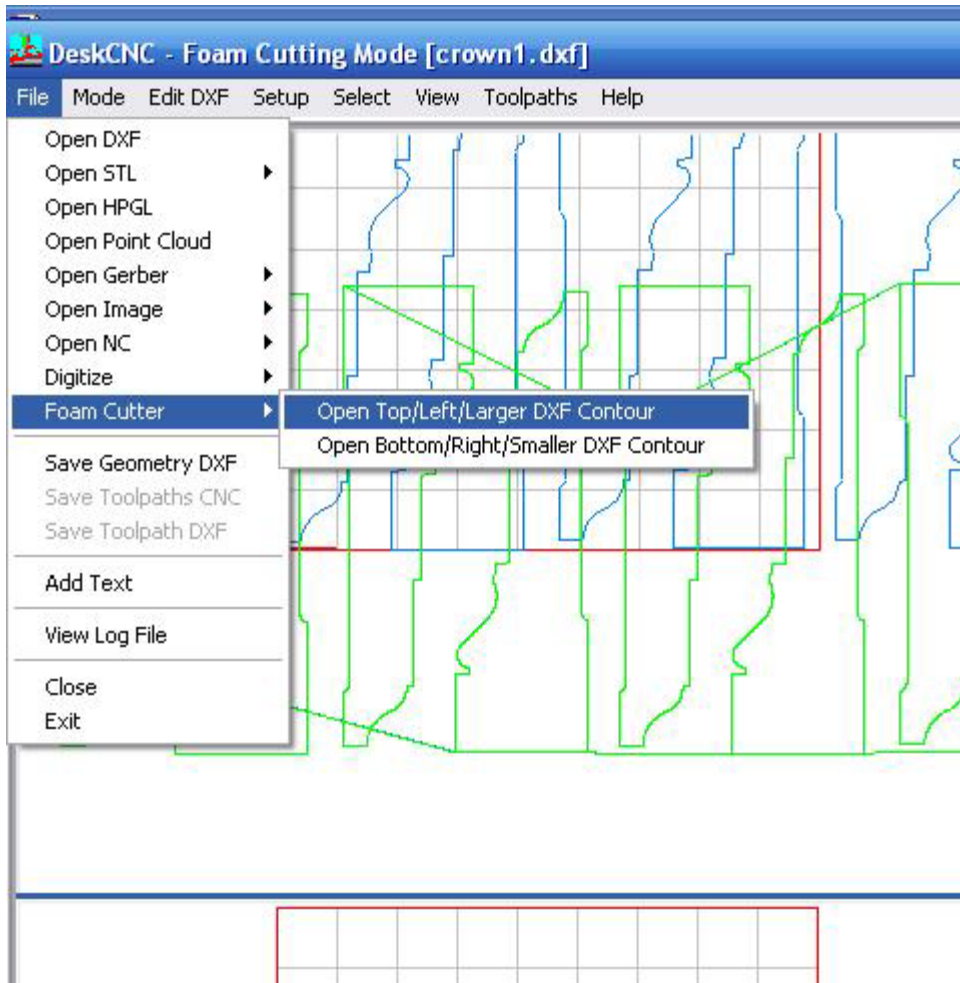
Notice that the original shapes (in green) stay at their original position, and the new shapes now in blue are at the 0,0



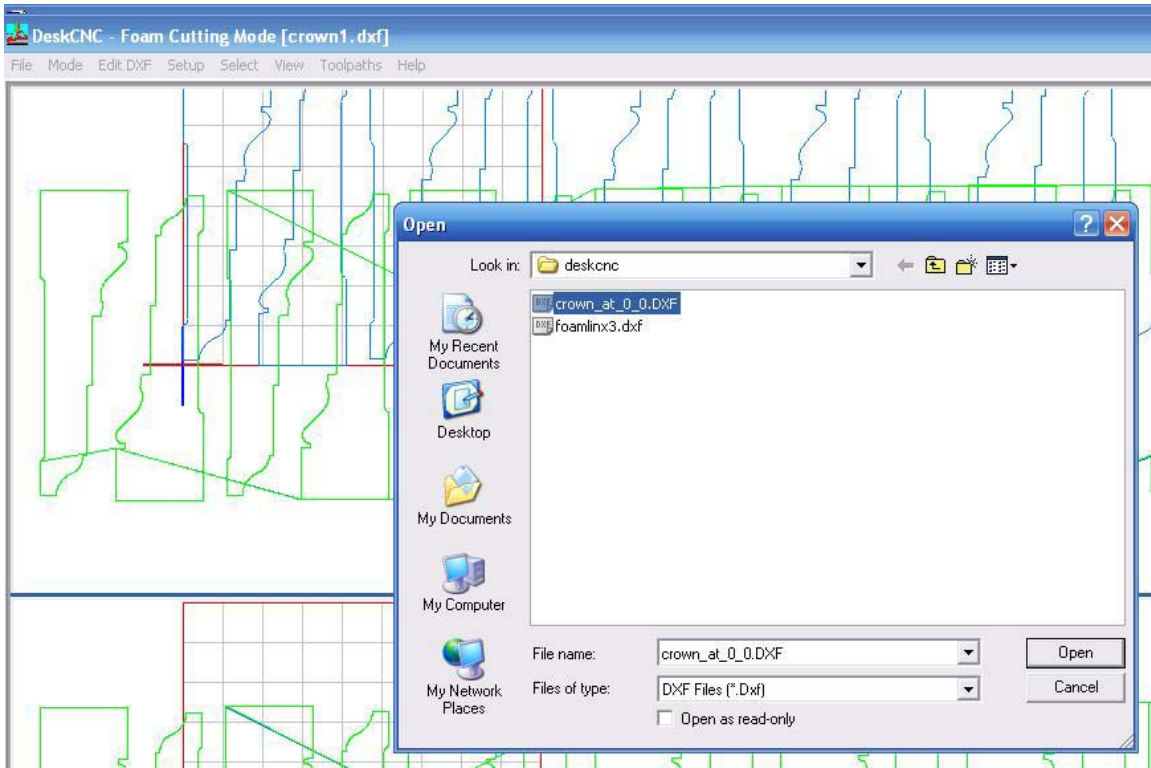
Now save the new shape at the 0,0 position. In this example we will use a similar name [crown\_at\_0\_0]



Now load the new shape we just saved,  
From the File Menu select [Foam Cutter], and then [Open Top/Left/Larger DXF Contour] as shown below

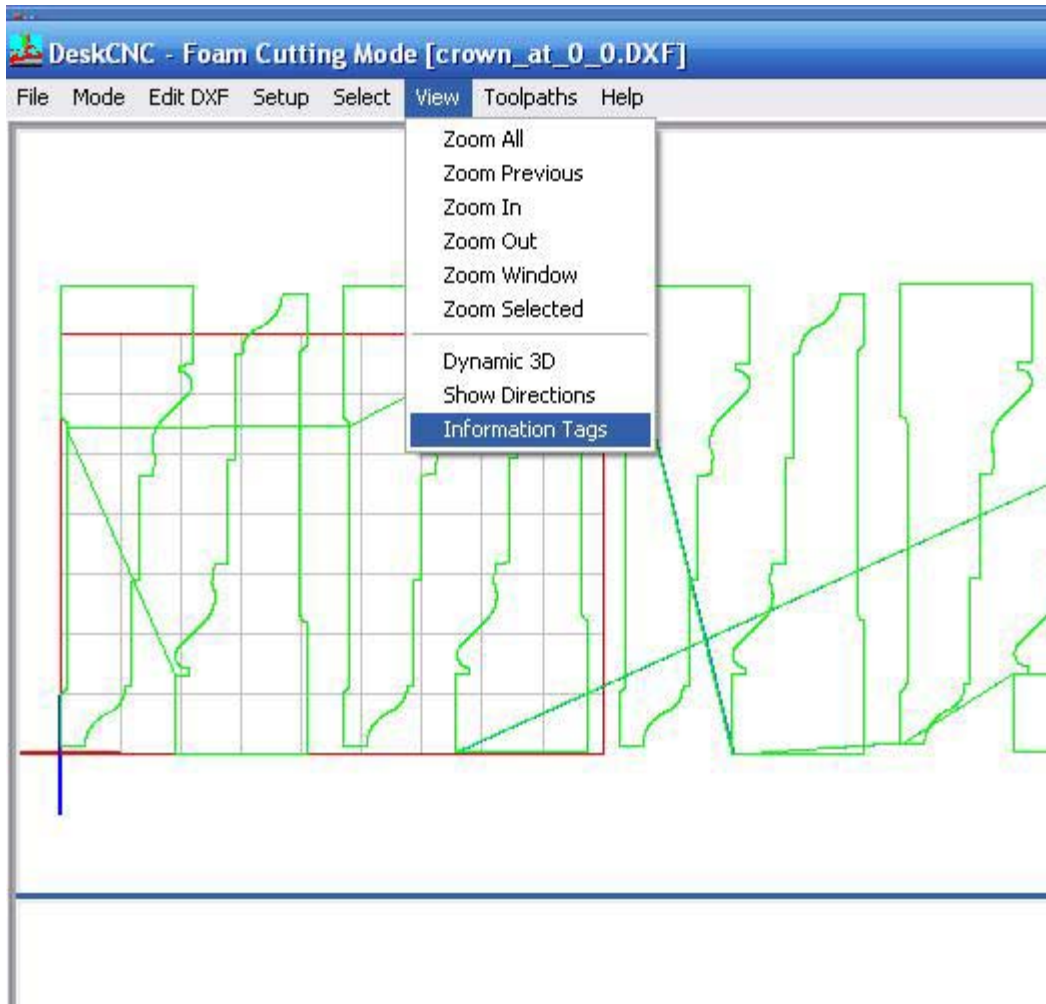


Choose the new file name (crown\_at\_0\_0.DXF)



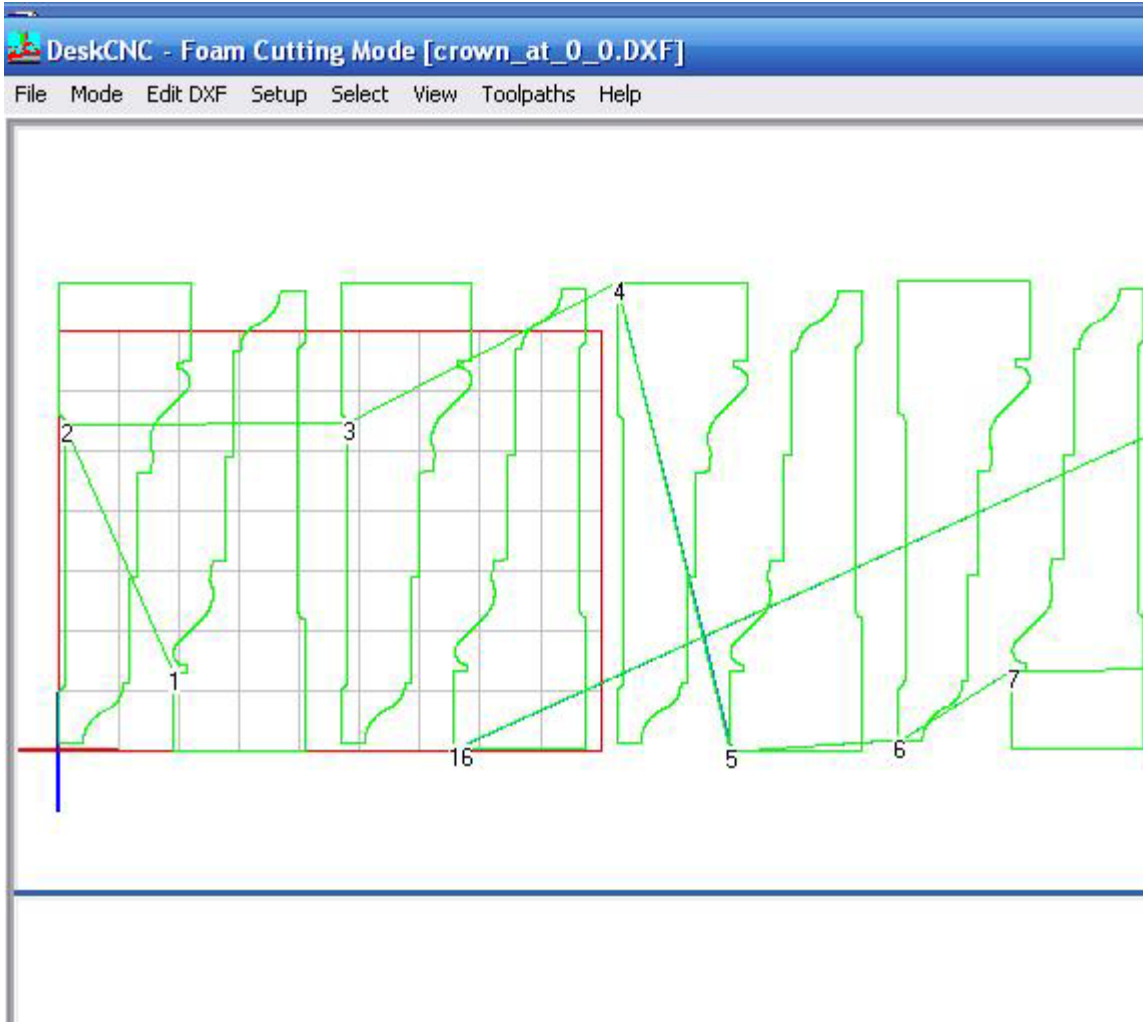
Notice that the new shapes are at 0,0 position

The green lines between one shape to the other are lines where the hot wire will travel from one shape to the other.

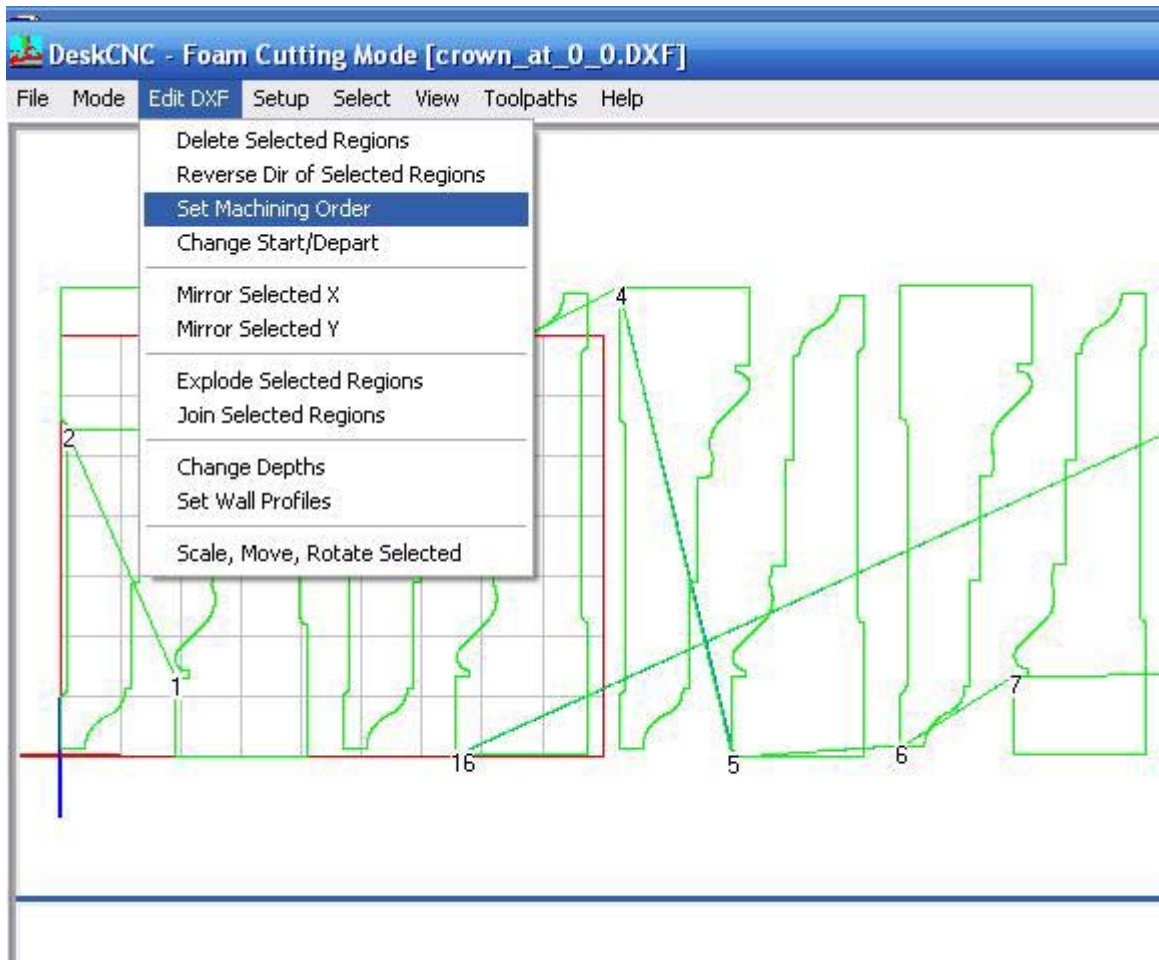


We must tell the software not to go through the shapes.  
Please follow these steps carefully.  
From the View menu shown above, choose [Information Tags], this will show each shape its serial number

In the screen shot below, note that the software chose to cut the first shape number 2, and then shape number 1.  
This is not good, and we need to tell the software to start from the left (now shape 2) and move towards the right



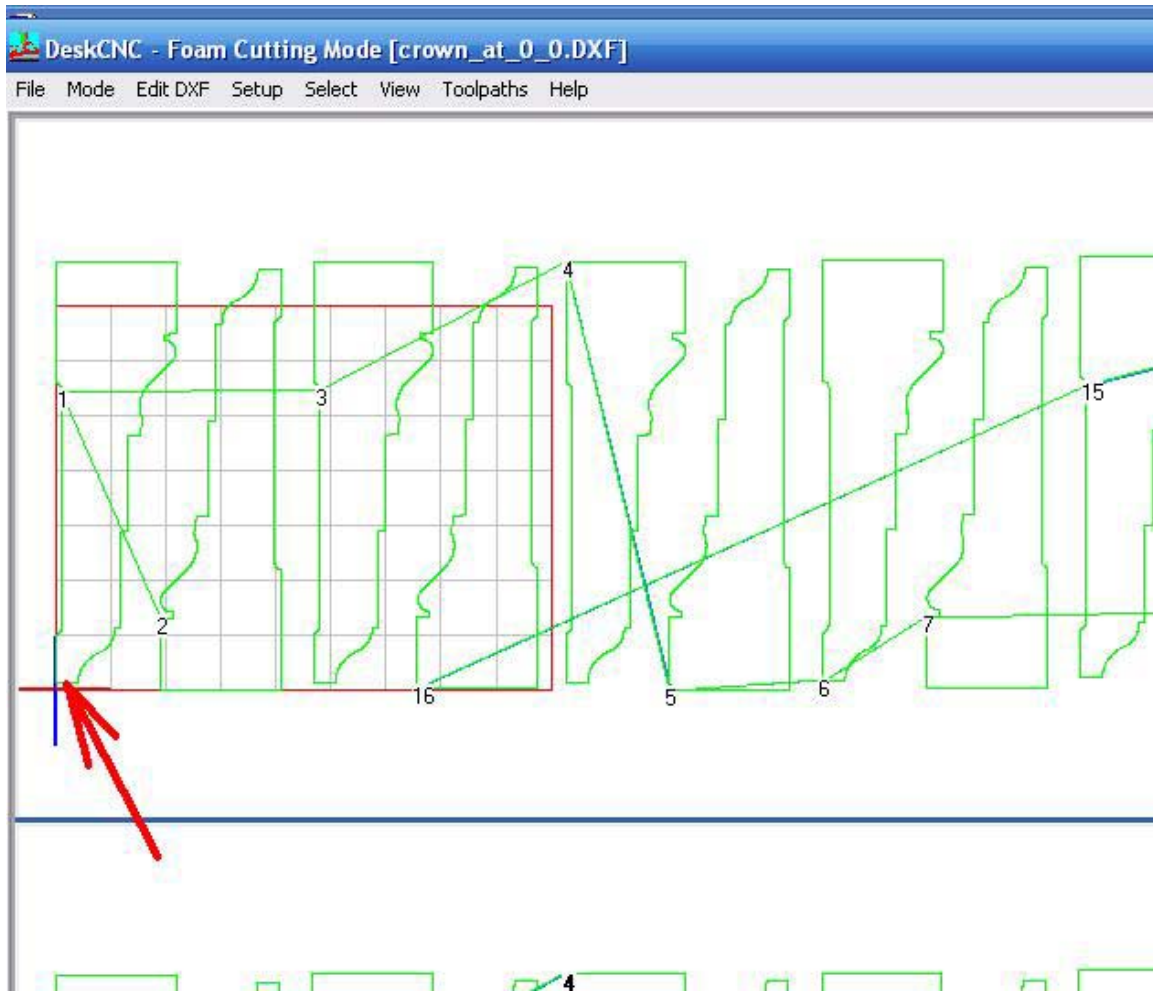
From the Edit menu choose [Set Machining Order]



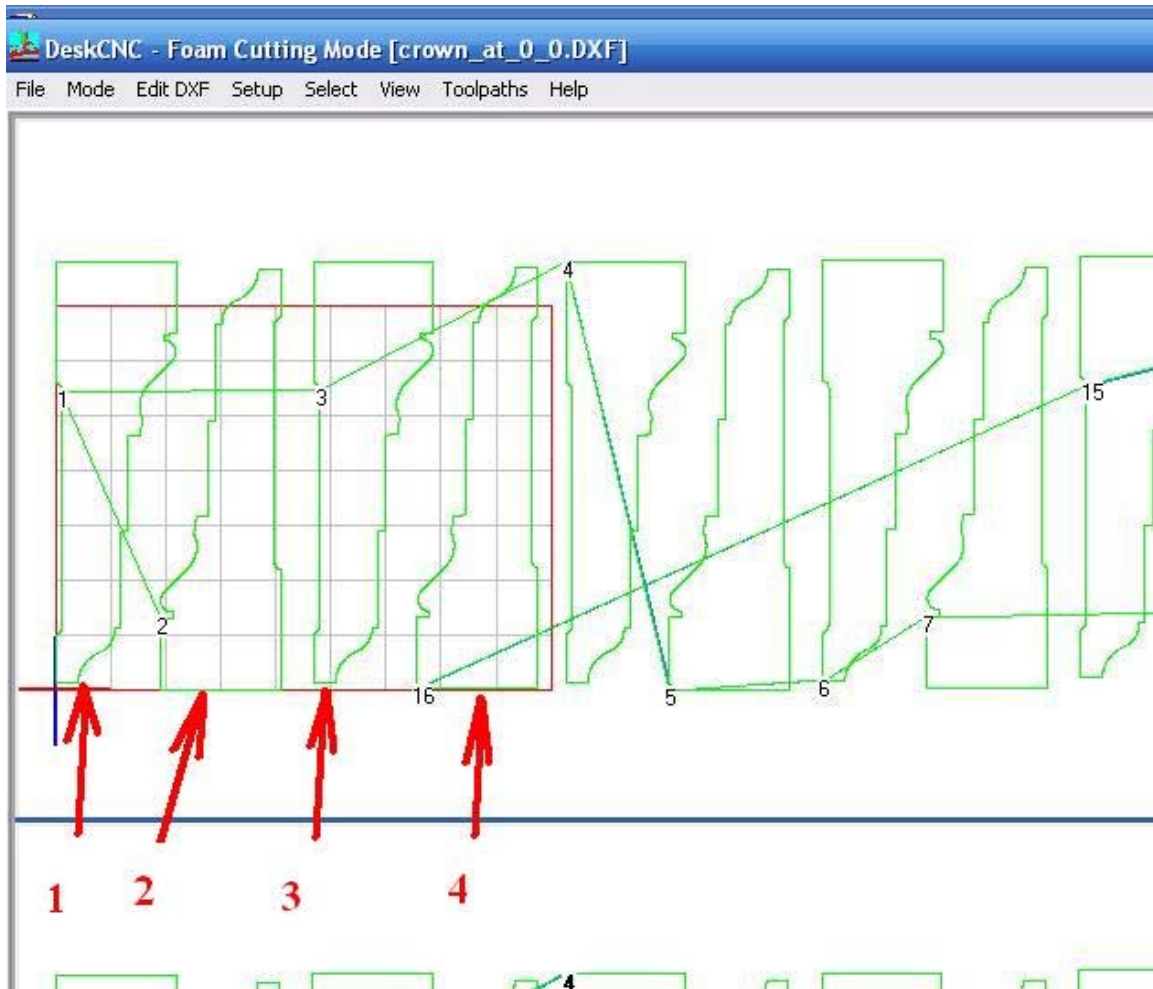
A small screen will pop and click on the OK button



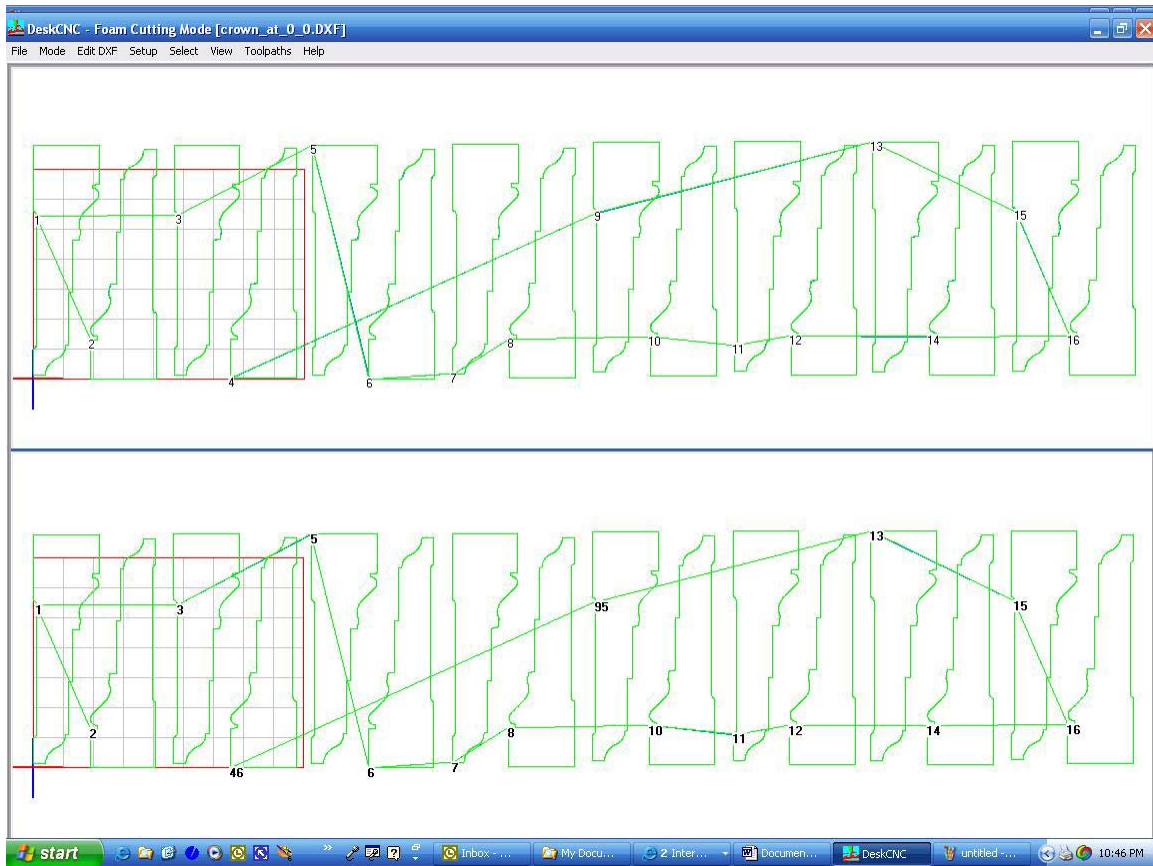
Click on the first shape from the left (where the red arrow is pointing) – this shape used to have the number 2 next to it, we now told the software that this shape is the first to be cut.



Now click on the shape next to it and move on towards the right side of the screen

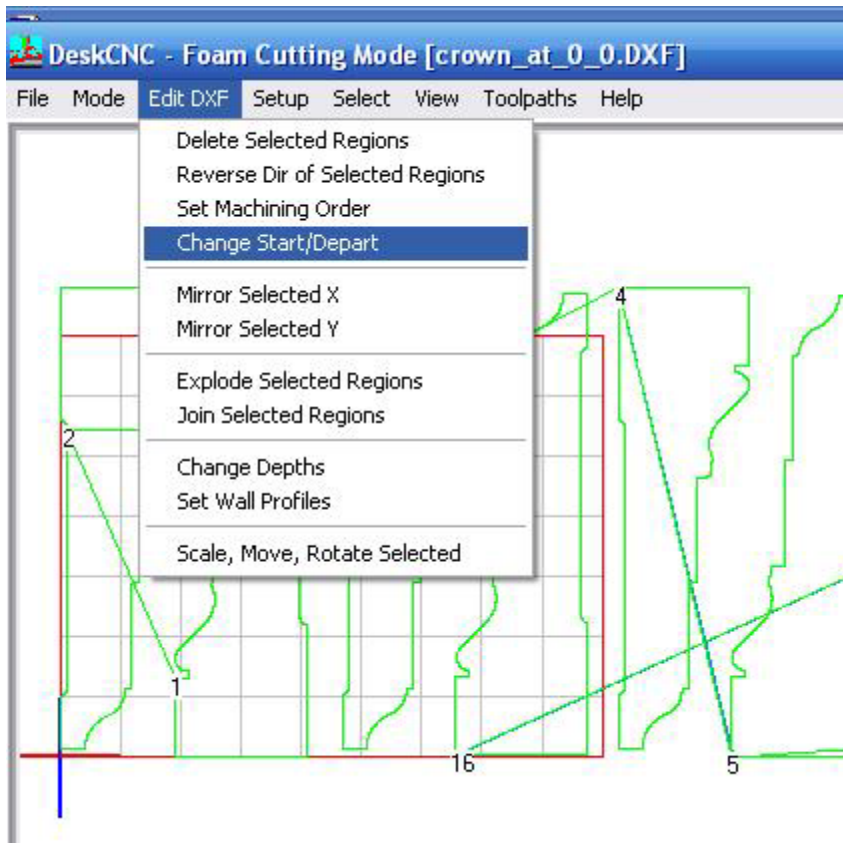


Once you have finished all the shapes, you should see the numbers from 1 to 16 in the correct order as shown below. If the numbers are not in the correct order, simple start again by choosing the Edit Menu, and then [Set Machining Order]



Although the shapes are now in the correct order, the software still does not know where the hot wire enters the 1<sup>st</sup> shape, where it should exit the 1<sup>st</sup> shape, and where to enter the 2<sup>nd</sup> shape.

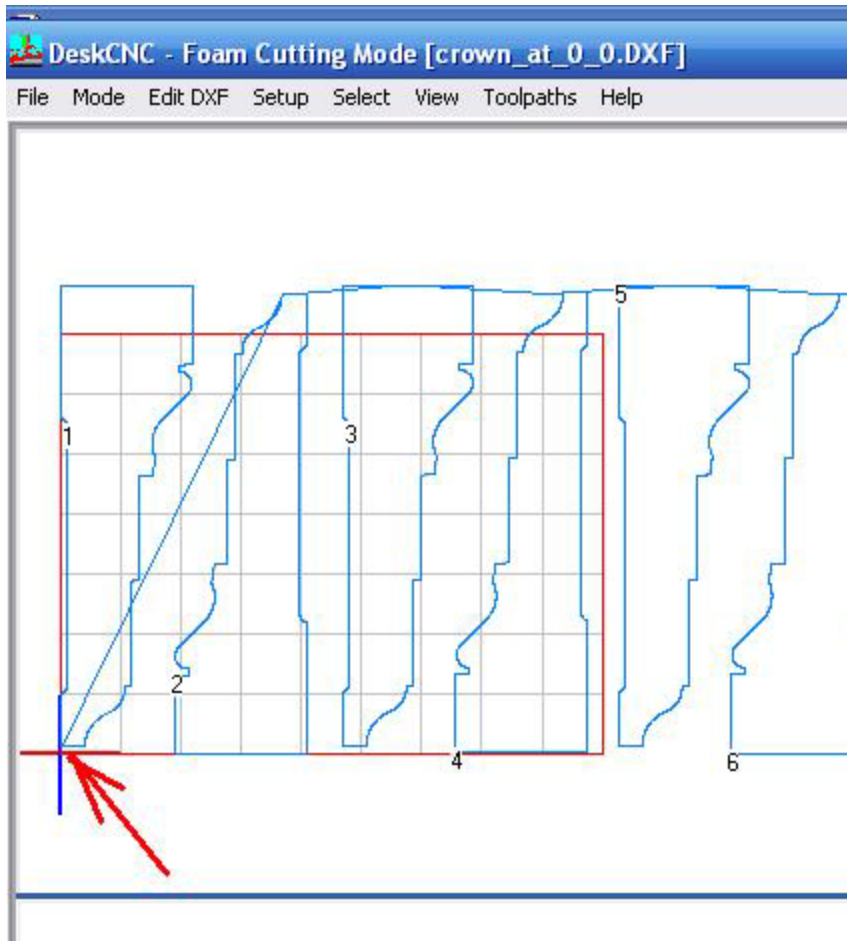
We will not set the start and the depart points for each shape



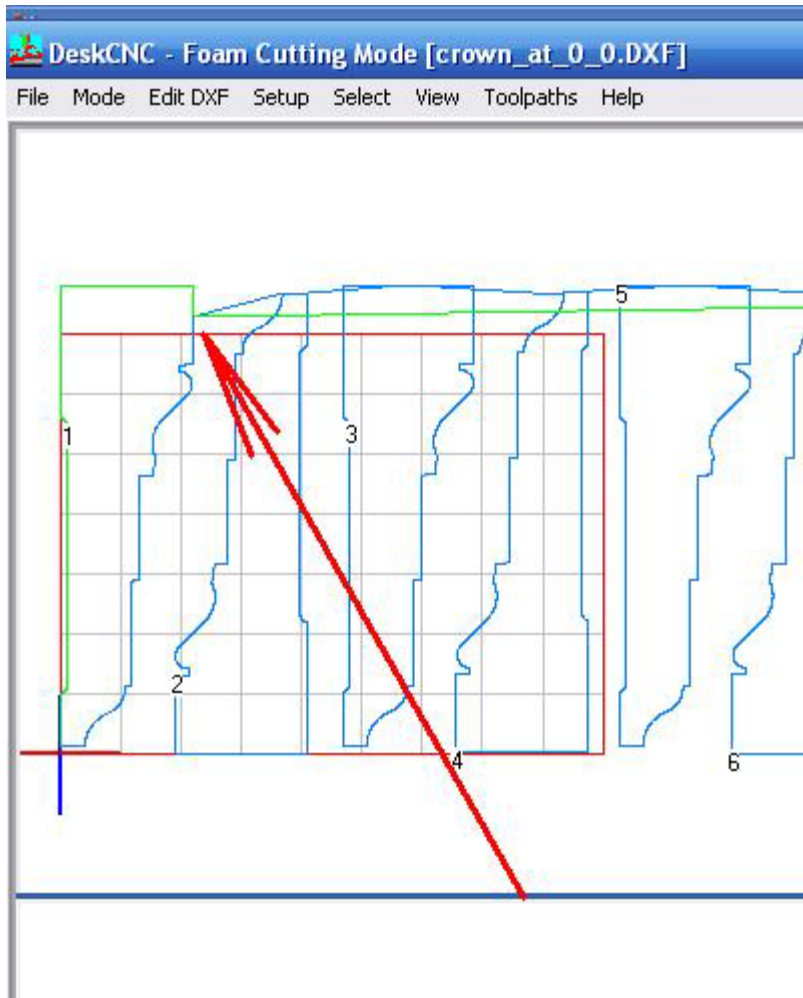
From the Edit DXF menu choose the [Change Start/Depart] as shown above

Left mouse click on the 1<sup>st</sup> shape exactly where the red arrow is pointing (in the picture below)

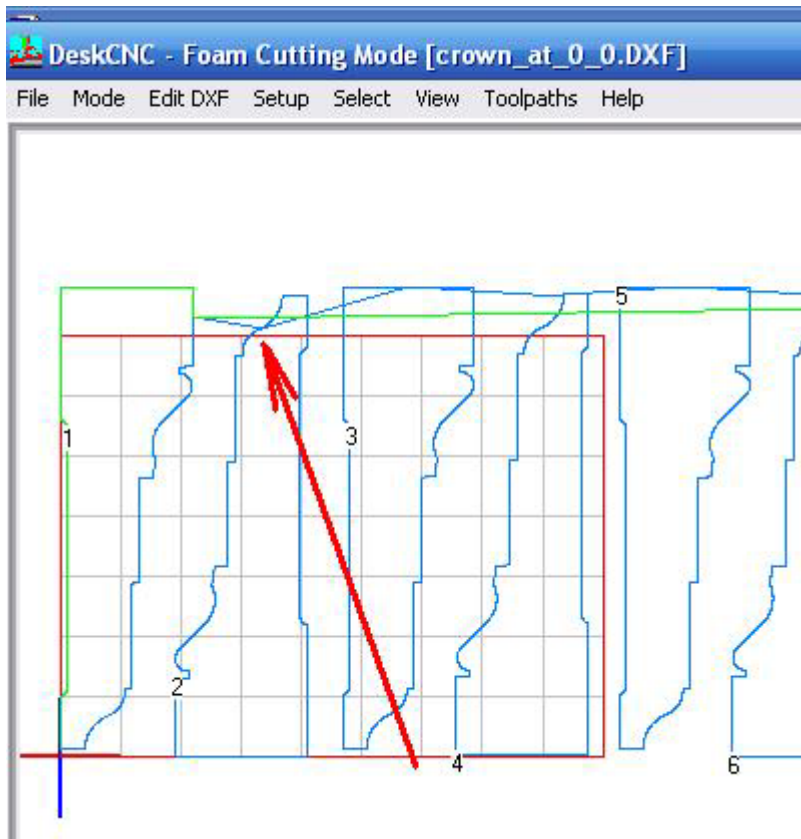
This would be the entrance point that the hot wire will go to the 1<sup>st</sup> shape



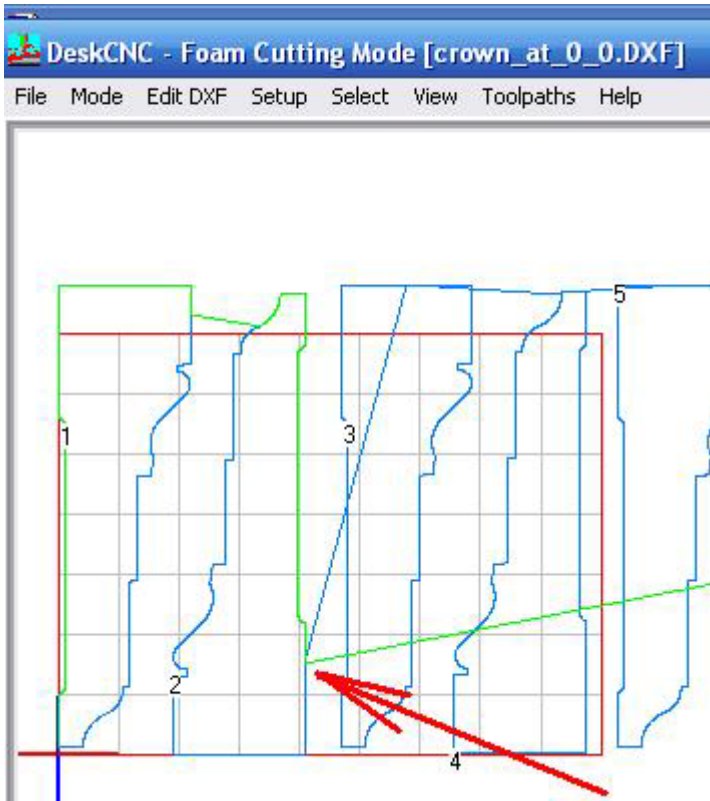
Now we need to set the exit point (depart) of the 1<sup>st</sup> shape.  
Right mouse click on the exit point shown with the red arrow in the picture below



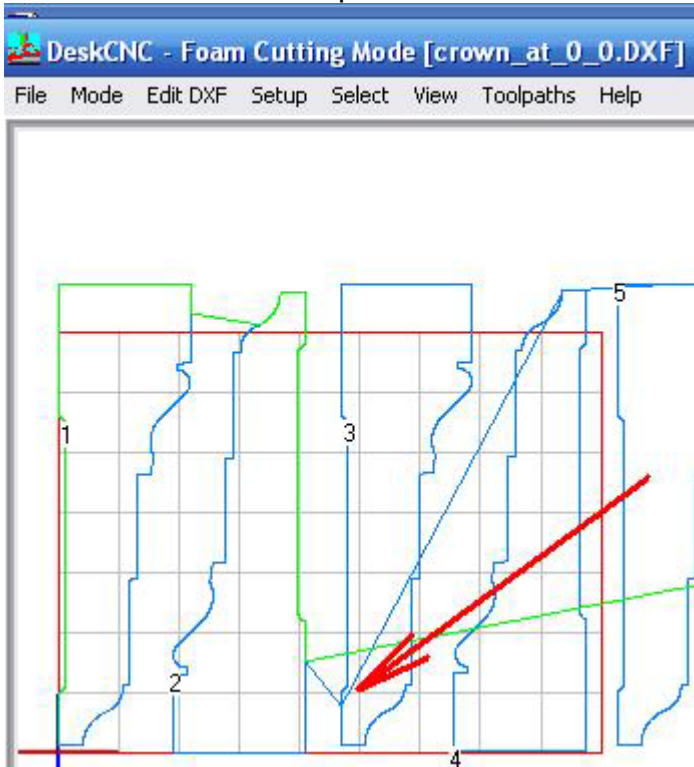
Now we will set the start point of the 2<sup>nd</sup> shape, this will automatically be connected to the depart point of shape 1  
Left mouse click where the red arrow is pointing in the picture below



And once again, we will set the depart point from shape number 2.  
Right mouse click where the red arrow is pointing as shown in the picture below

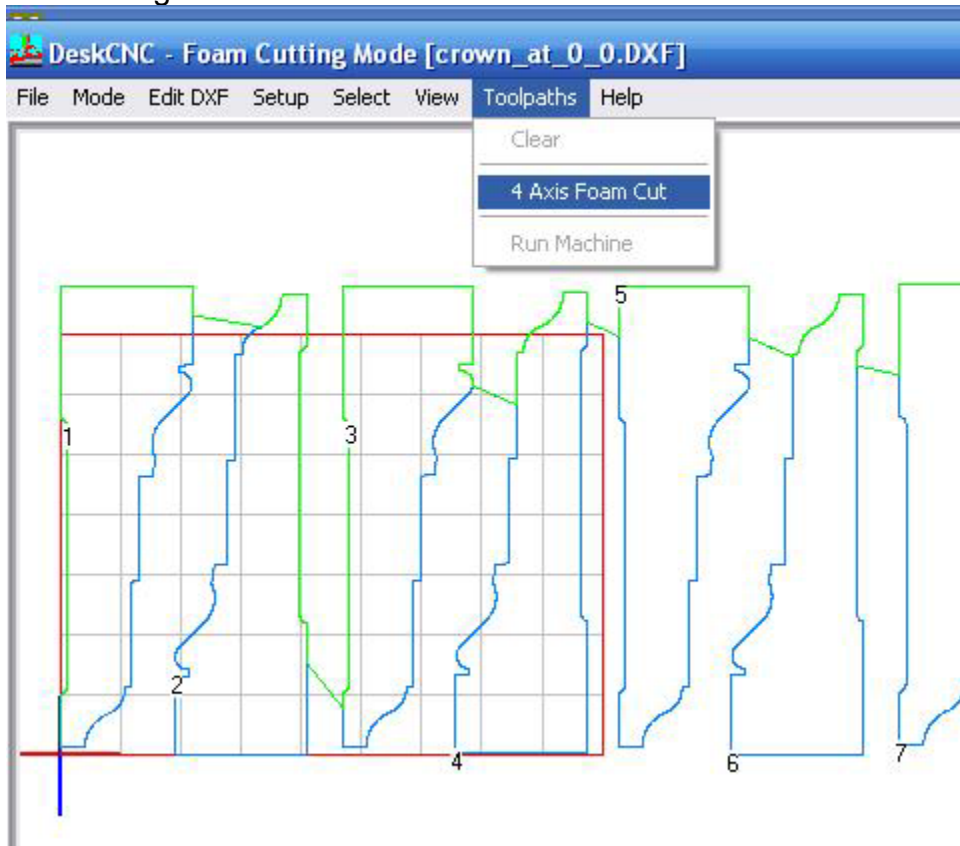


And again lets set the start point of shape number 3.  
Left mouse click on shape number 3 as shown in the picture below.

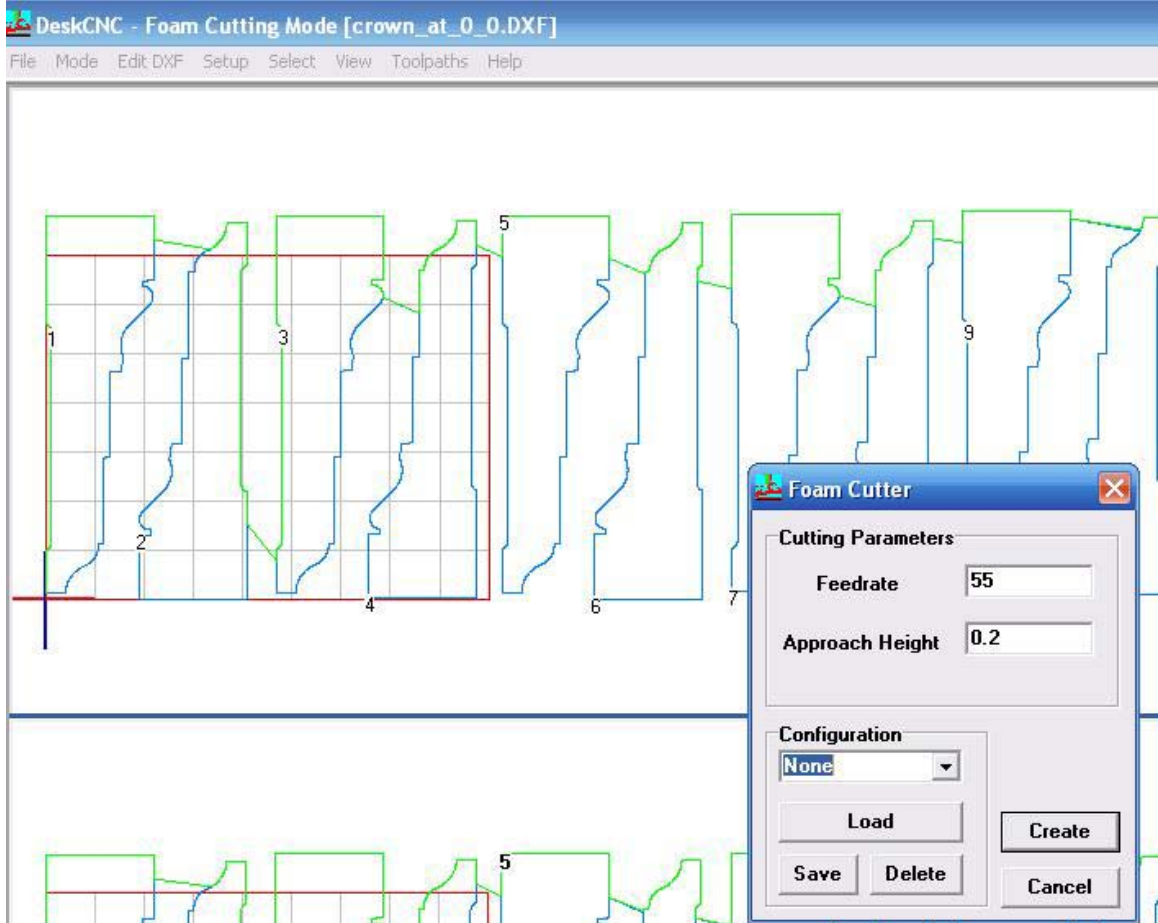


Continue till all the shapes are connected between one another without any lines going through any of the shapes (as shown below)

Its time to generate the G code that will run the machine



As shown in the picture above, click on the Toolpaths menu and choose 4 Axis Cut



Enter the feed rate – in the example above we choose 55"/min