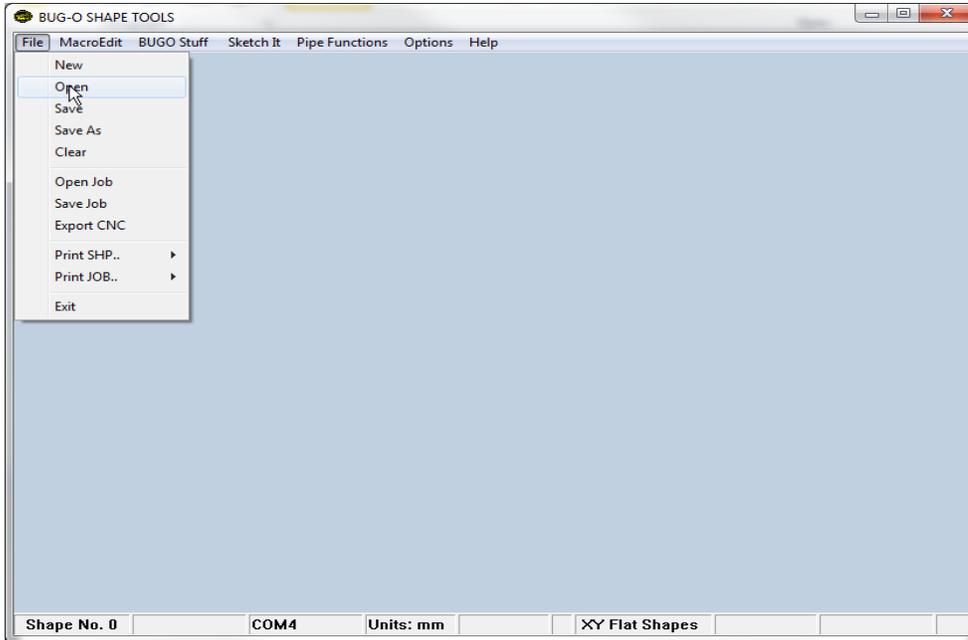
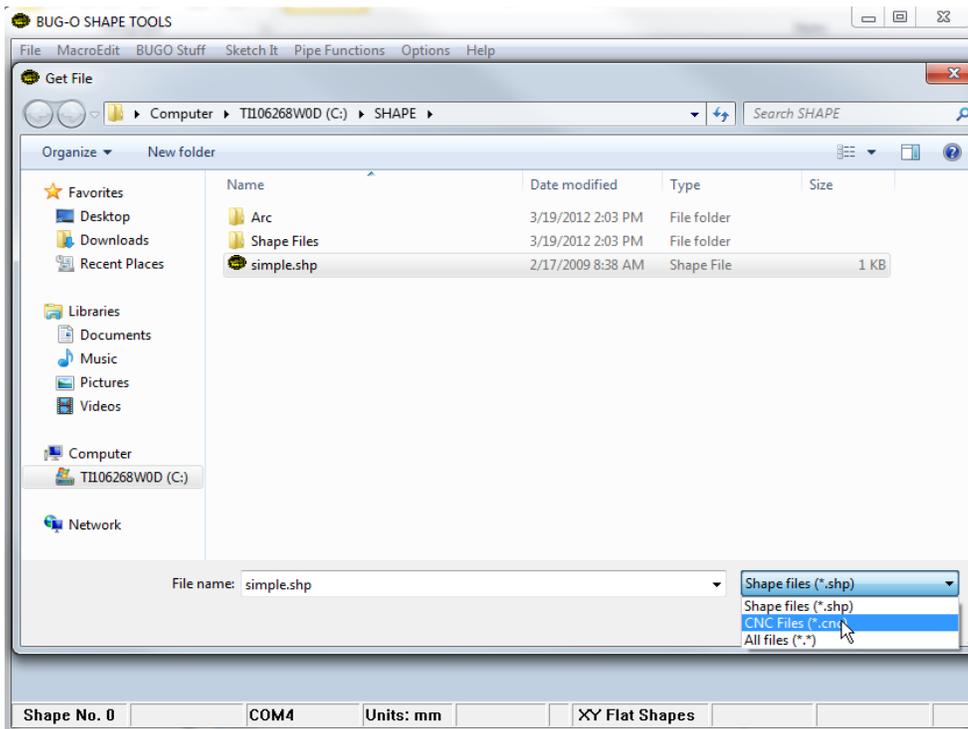


How to download a shape file to a Programmable Shape Machine using PC Shapes Software

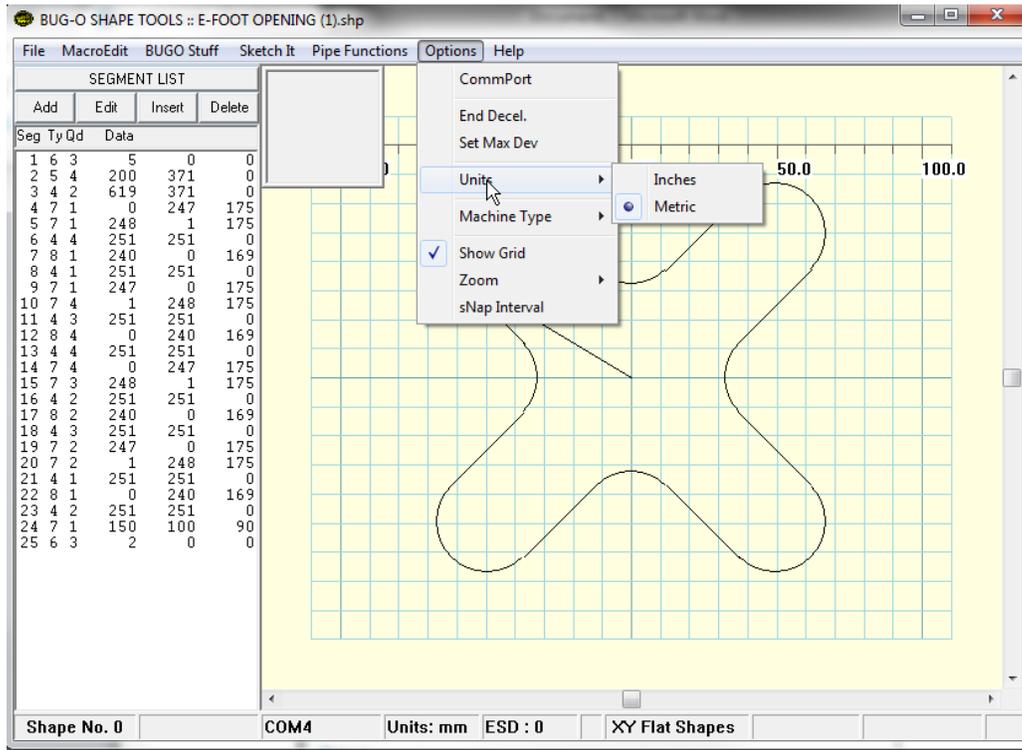
1. Open PC Shapes Program. Click File Open to find the shape you want to download to the machine.



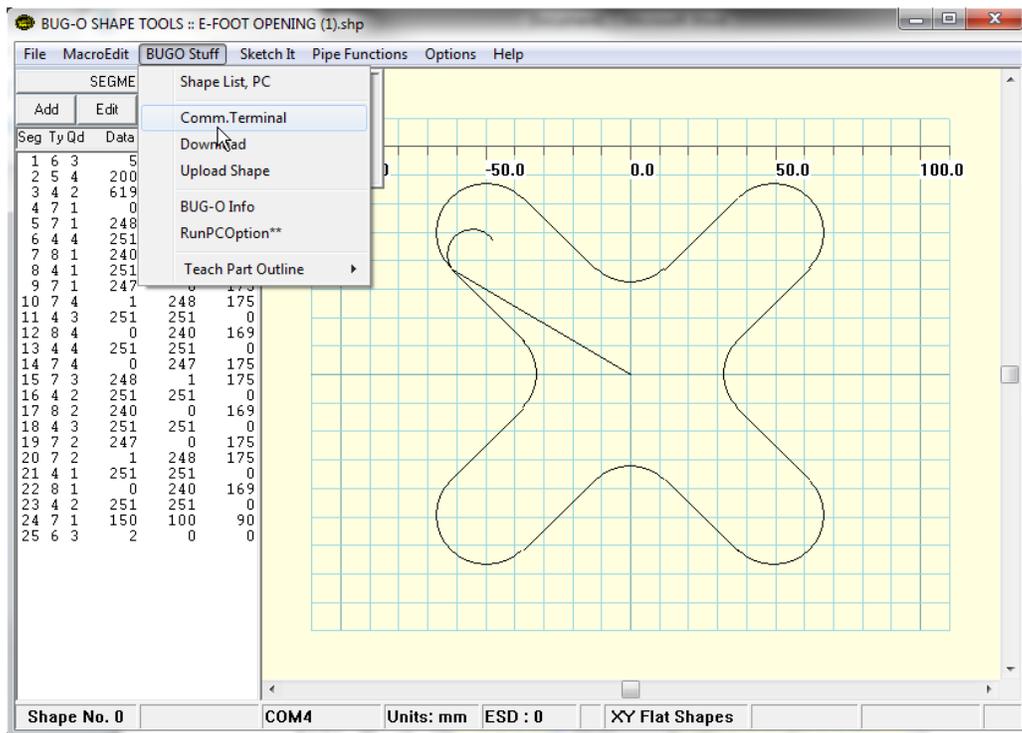
2. Click on the file you want to open from your computer and click Open. If you are working with a .cnc file, make sure to change the file type at the bottom of the screen to .cnc.



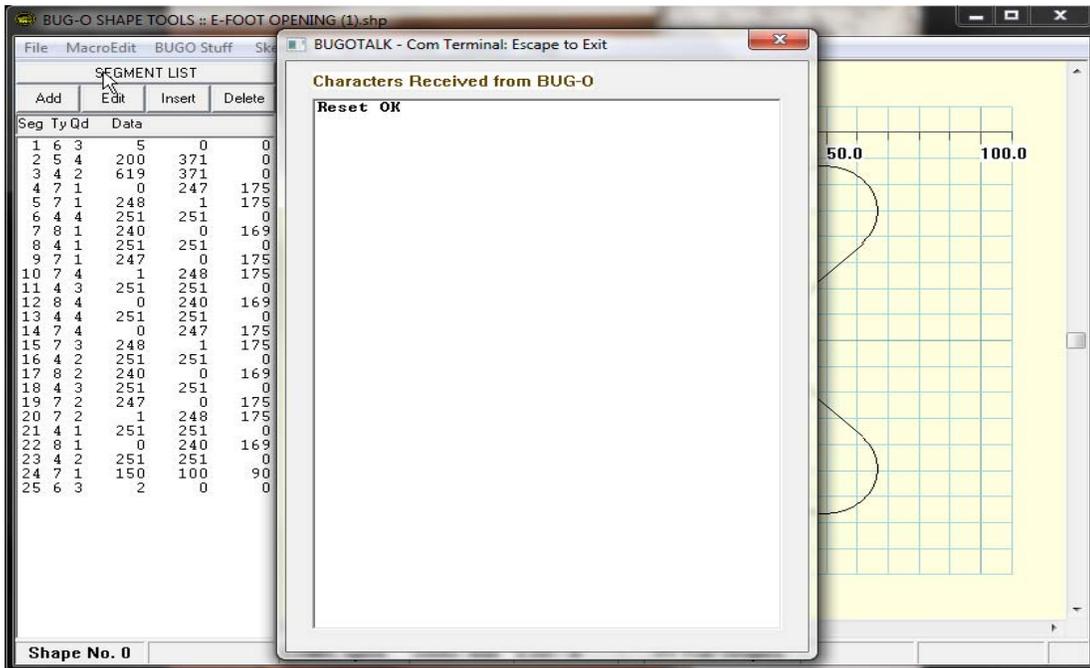
3. Make sure Shapes program is set for proper units (in/mm) for the shape you are loading. You can change the units by clicking Options Units.



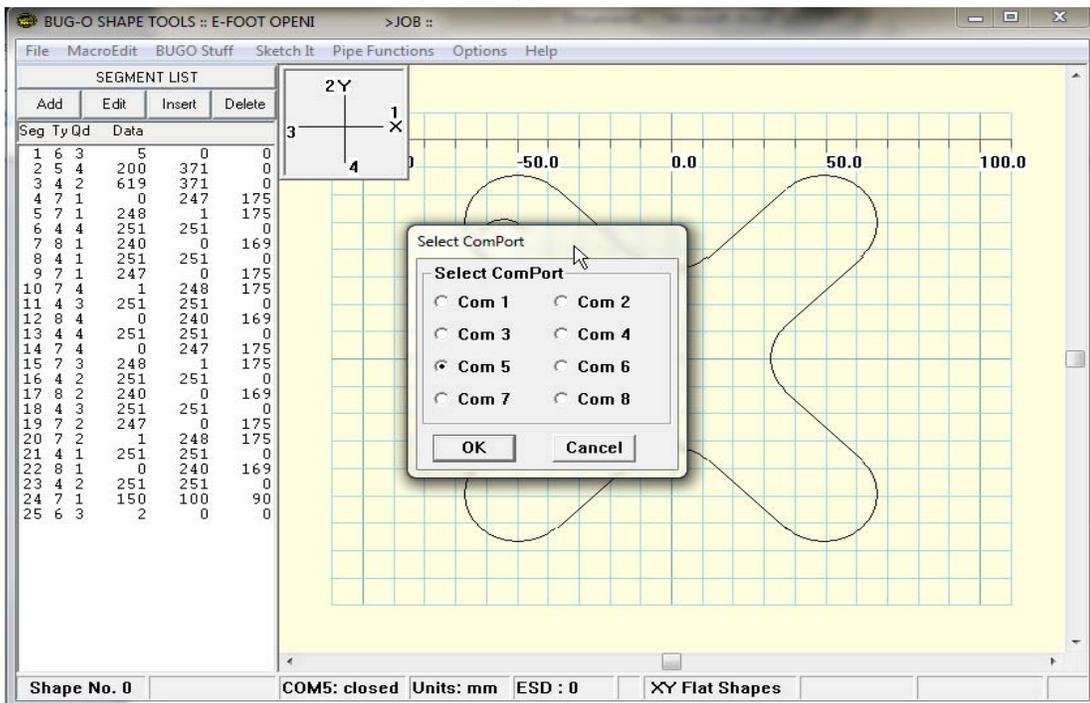
4. Next, check to make sure you are communicating with the machine. Plug the cable into the programmable shape machine and the computer you are working with. Click BUG-O Stuff Comm Terminal.



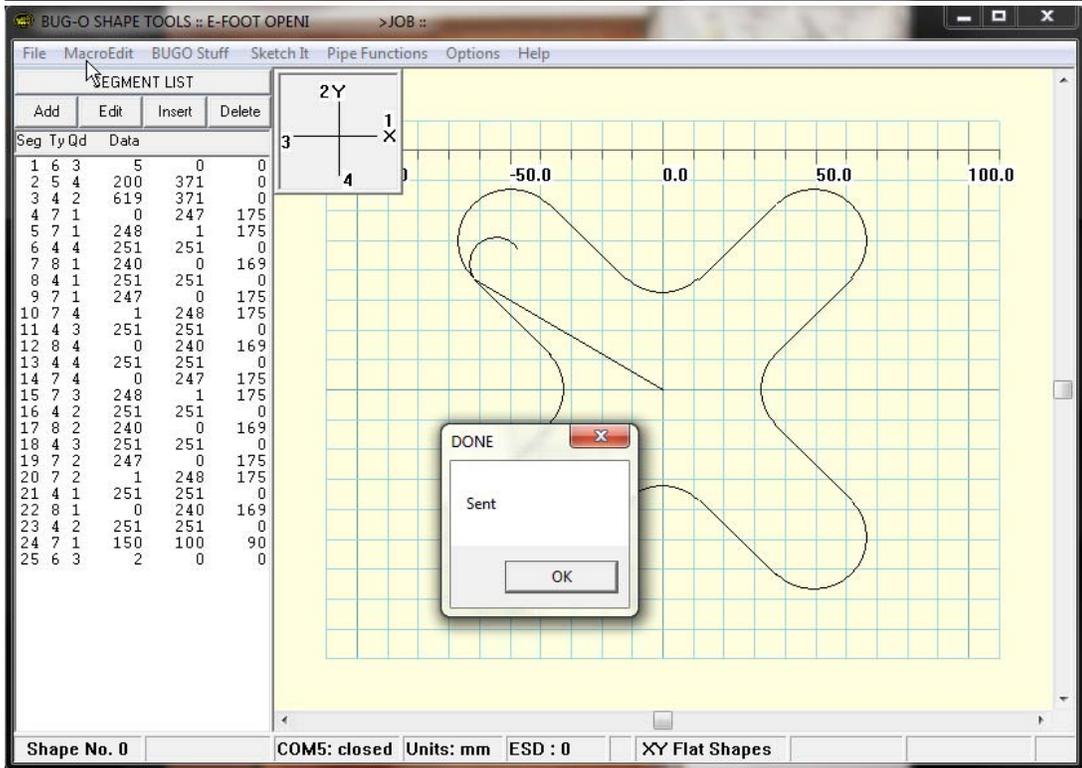
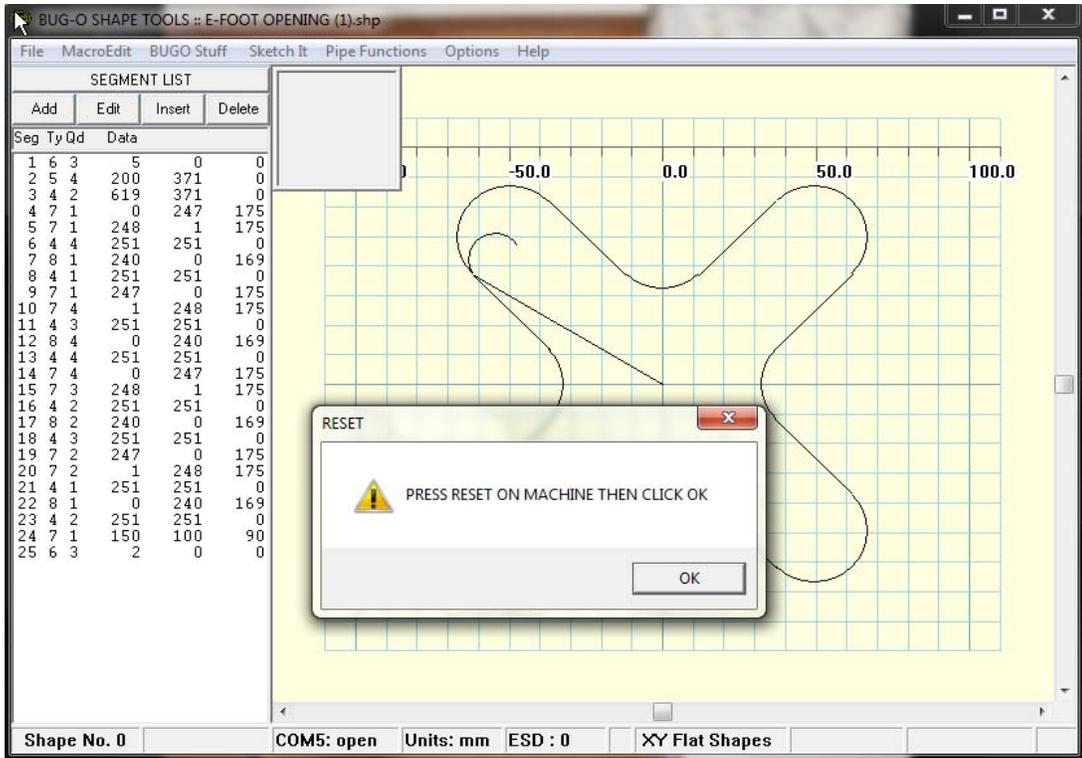
- The Comm Terminal will open a new window. If you are using the correct Comm port, when you press the black reset button you should see Reset OK on the Comm Terminal. This tells you that you are talking to the machine properly.



- If you DO NOT see Reset OK on the Comm Terminal when you press the black reset button on the machine, you will need to change the comm port. Click Options Comm Port to select the proper port. The software should tell you which ports are valid. Click on a valid comm port and click ok. Go back to BUG-O Stuff Comm Terminal and press reset on the machine. If you see Reset ok, move to step 7. If not, repeat step 6.



- Now we are ready to download the file to the machine. Click BUG-O Stuff Download. You will get a window that tells you to press reset on machine then click ok. Press black reset button on machine then click ok. When the download is done, you should see a window that says Sent. Click OK and the machine should be ready to run.



How to convert a .dxf file through DeskCNC:

Installation of DeskCNC:

1. Insert CD labeled BUG-6241 into computer
2. Run DeskCNC.exe from the CD
3. Enter the user name and key code written on the CD when prompted the first time you run DeskCNC.
4. Copy the file "posts.cfg" from the CD to the following location on your computer:

My documents\DeskCNC\config

- Overwrite the file that is currently in that location with the file that came on the CD. This will allow the DeskCNC conversion file to be read by the BUG-O PC option

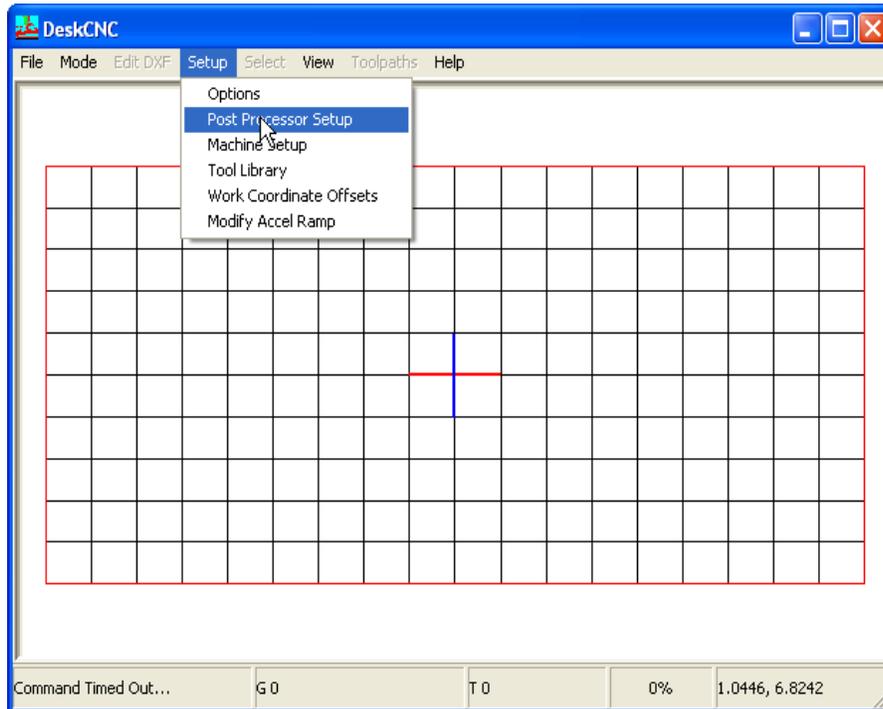
Requirements for .dxf file are as follows:

1. Drawing MUST be only 2D
2. Drawing must be cleaned to only show the actual cut lines to be cut. All reference marks and extra lines must be removed. This can be done in DeskCNC or in the original CAD drawing.
3. All blocks and Splines must be exploded completely before loading the .dxf file on DeskCNC

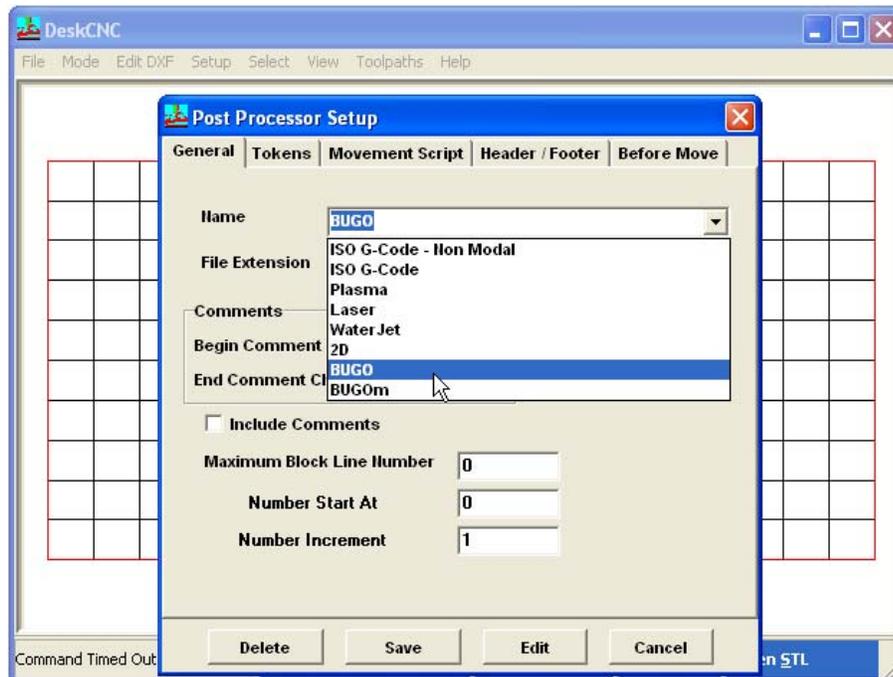
How to configure DeskCNC:

*****This MUST be done every time you first open the program to set the configuration properly****

1 . Click the setup tab and click post processor setup

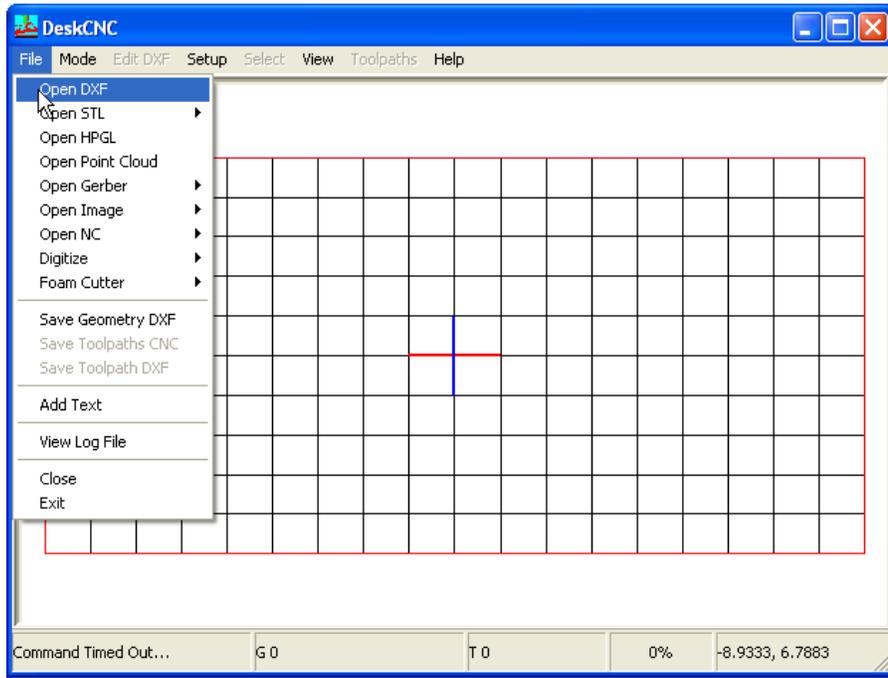


2. On the general tab, click the dropdown name box and click BUGO. Click Edit to adjust the settings for the program, then click cancel to go back to the main program.

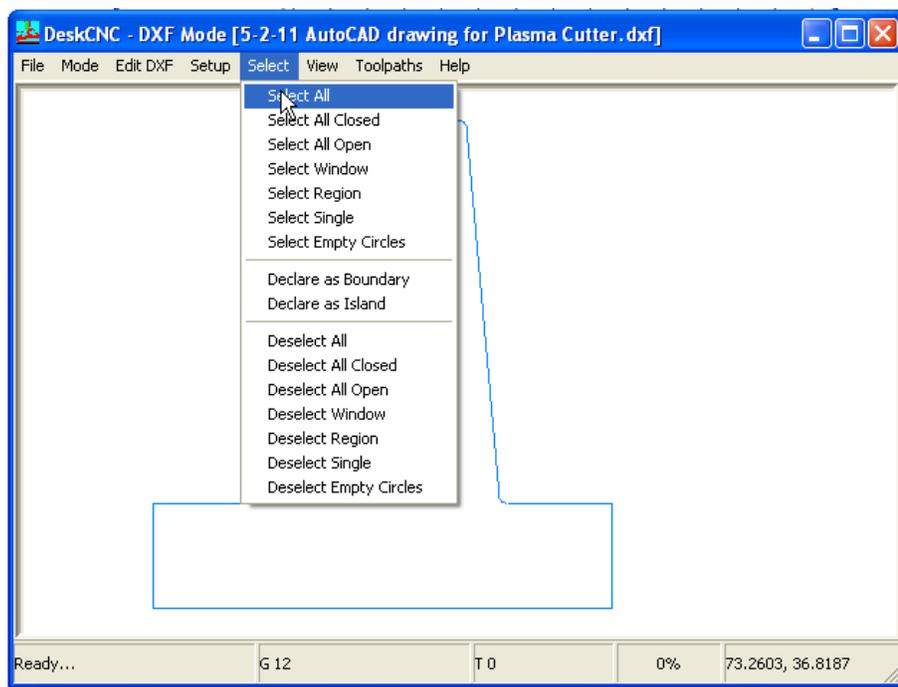


How to convert a .dxf file to a CNC file in DeskCNC:

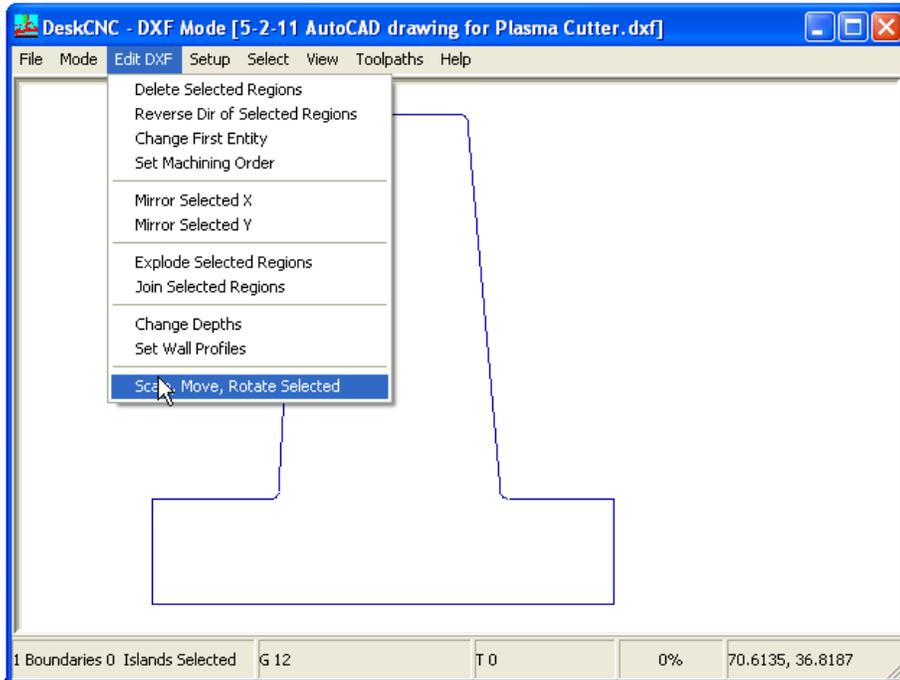
1. Open .dxf file in DeskCNC



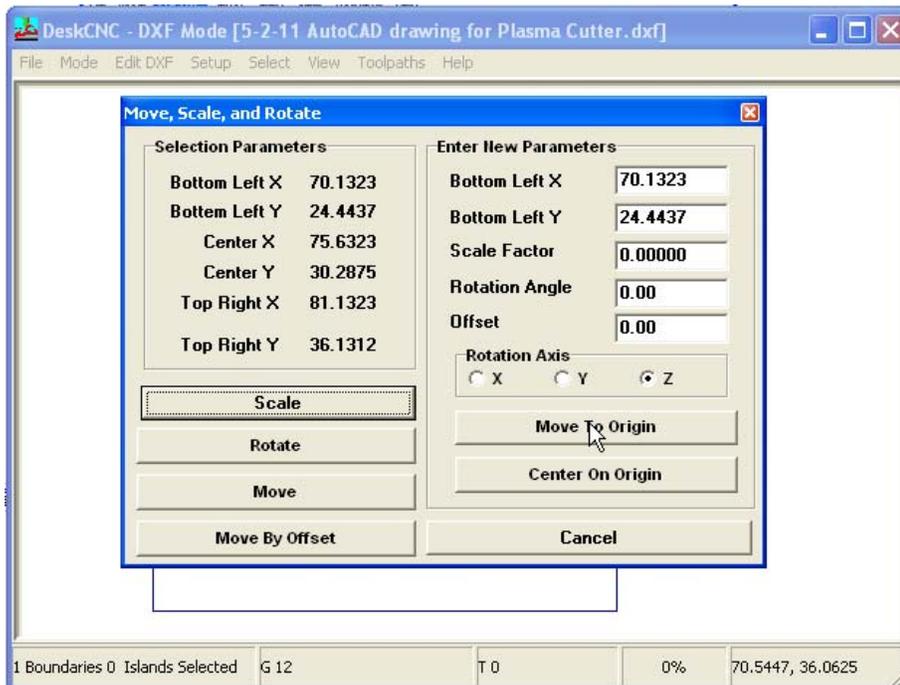
2. If using a clean drawing, click select menu and click select all.



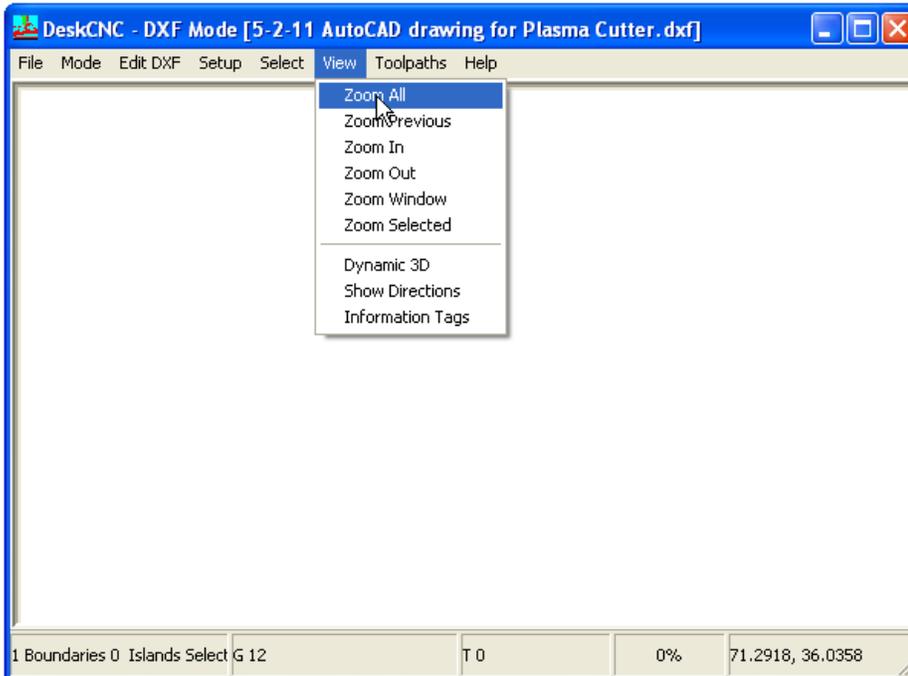
3. Click the edit DXF menu and click scale, move, rotate selected.



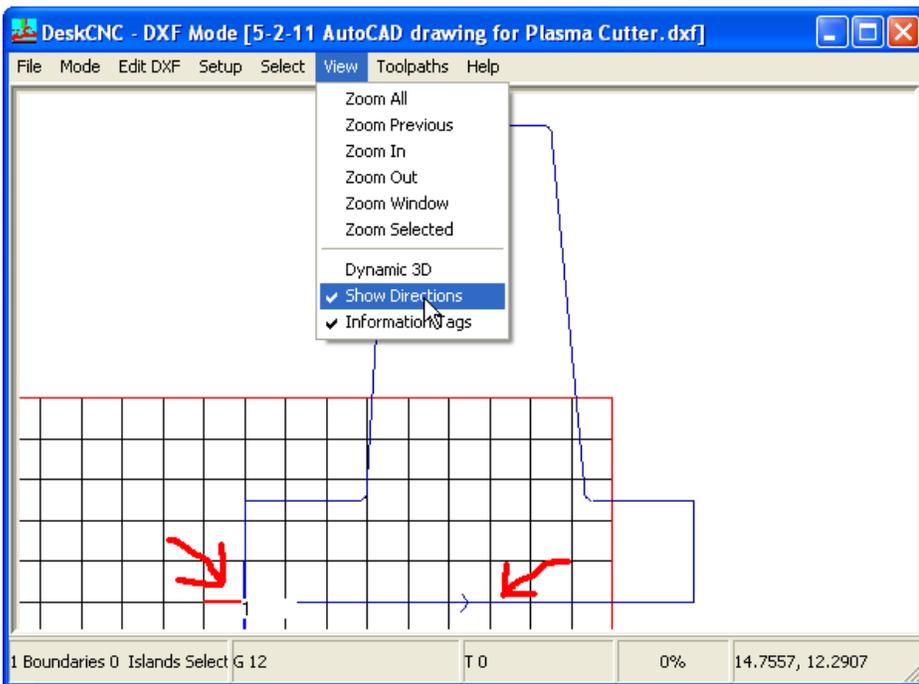
4. Click move to origin to move the piece to the center point of the grid. Then click the cancel button. At this point you will likely not see anything on your screen.



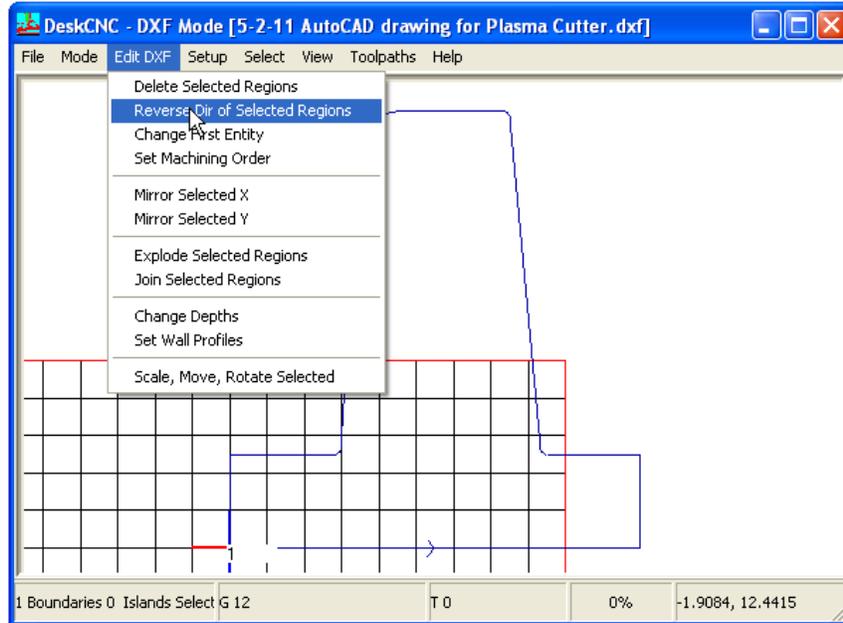
5. Click the view menu and click zoom all. This will bring the full drawing back into view.



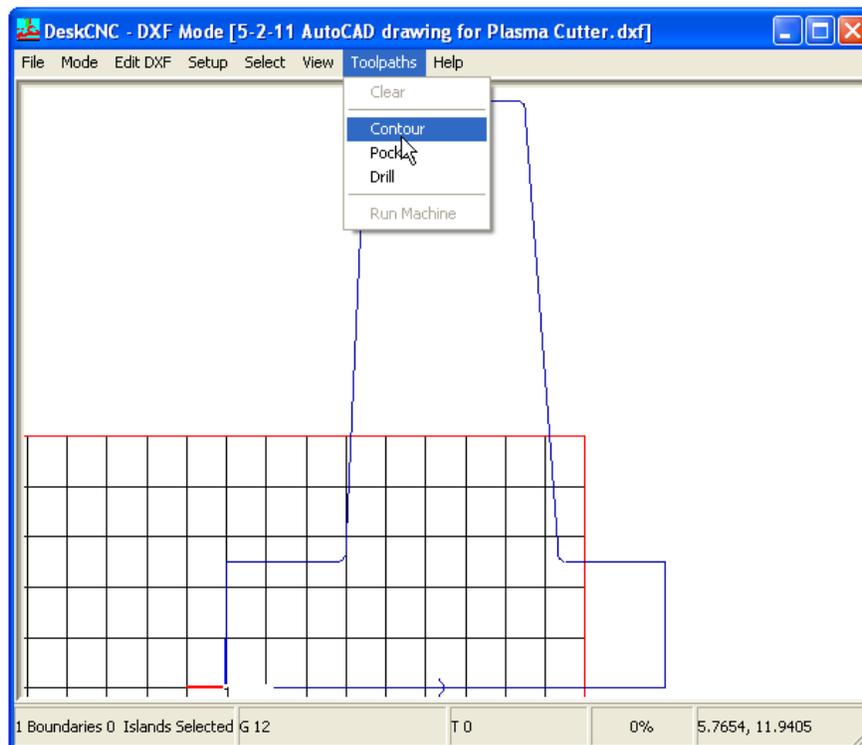
6. Click on view menu and make sure that Show Directions and Information tags are both checked to determine number and direction of cuts.



7. To change the direction of the cut, make sure your drawing is still selected by clicking select and select all. Then click Edit DXF then Reverse Dir of Selected Regions.



8. Once you have the cuts set the way you want, you are ready to make toolpaths for the machine to read. Click Select menu and select all (If the shape is no longer highlighted), then click Toolpaths menu and Contour.

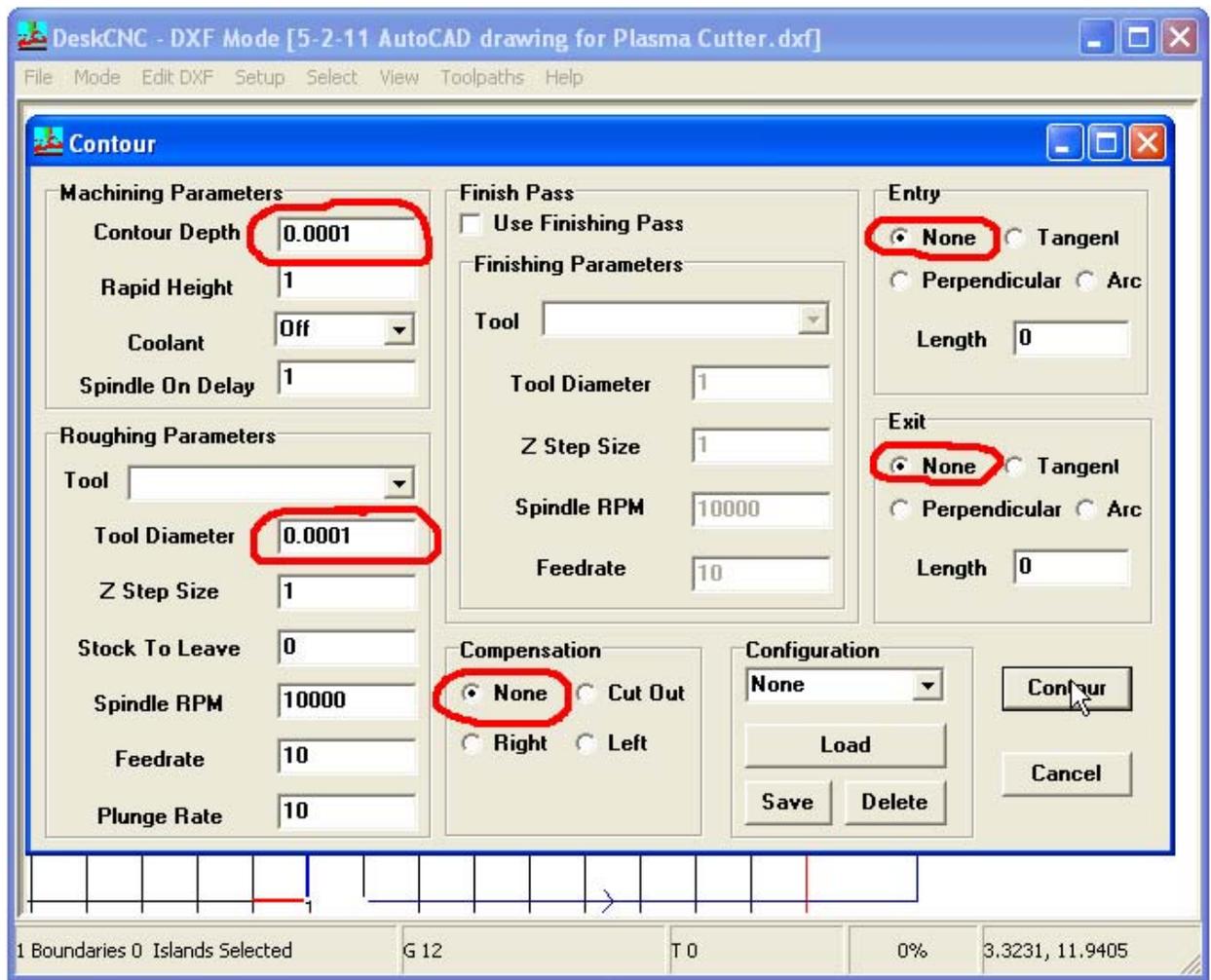


9. In the contour menu, there are a few things that need to be changed. Under machining parameters, the contour depth should be 0.0001. This must be done in order for DeskCNC to create a toolpath. The following parameters deal with kerf for plasma cutting as well as lead in and exit.

Assuming zero kerf and no lead in or exit, the following settings should be applied:

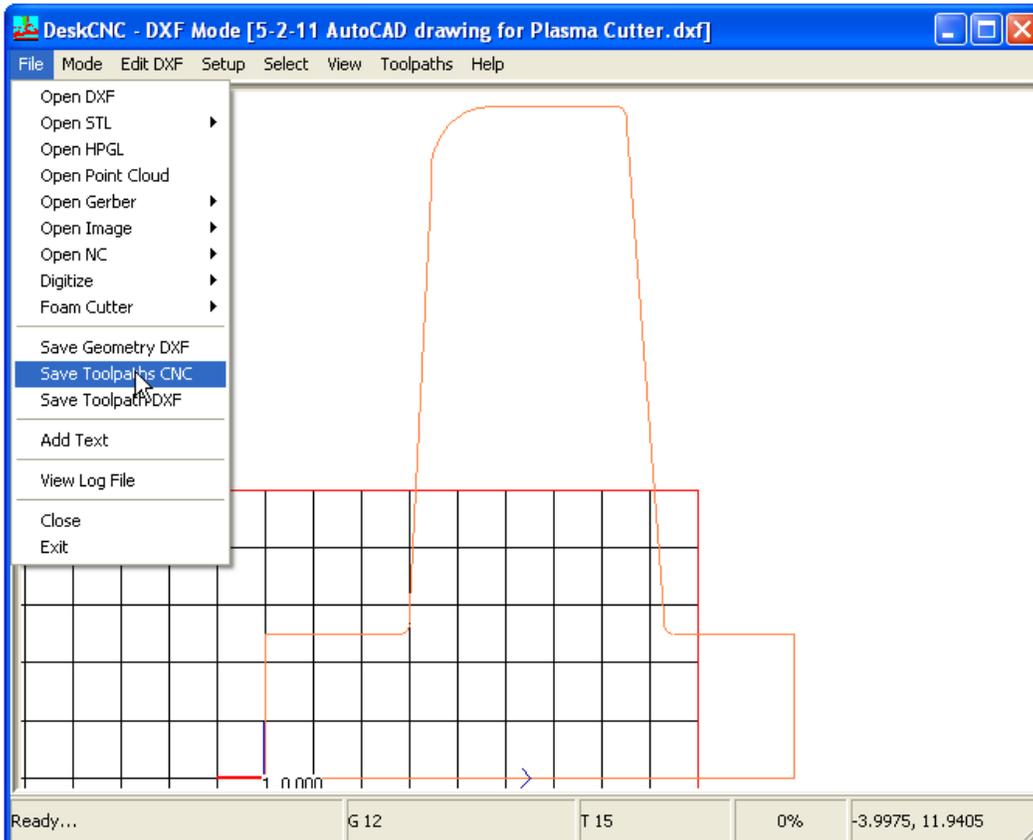
Under Roughing Parameters, the Tool Diameter should be 0.0001. Under Compensation, make sure None is checked. Under Entry, make sure None is checked. Under Exit, make sure None is checked.

After all the settings are correct, click the contour button to generate your toolpaths.



10. Once you have created your Toolpaths through the contour menu, you are ready to save the file in a format to load in the PC option. Click the file menu and click Save Toolpaths CNC. Save the file somewhere that you can easily get to later.

****For the purpose of using this file in the PC option, the easiest place to save the file is in your Shape directory on your computer which by default is C:\Shape.**



11. Now you are ready to open your CNC file in the PC option. Once you have it loaded, the PC option can be used normally to download. The only thing different you have to do is change the file type to be opened to CNC files and then open your file. The PC option will build your program automatically from there.

Cutting Pipes with DeskCNC and Shape using sample.dxf

Installation:

1. Run DeskCNC.exe
2. Copy posts.cfg to C:\Program Files\DeskCNC, overwriting the old file
3. Run the setup file for the Shape program

Prepare your DXF

- Remove lines that are not part of the shape (I left the top and bottom lines to show how to remove them in DeskCNC)
- Make sure dimensions are 1:1



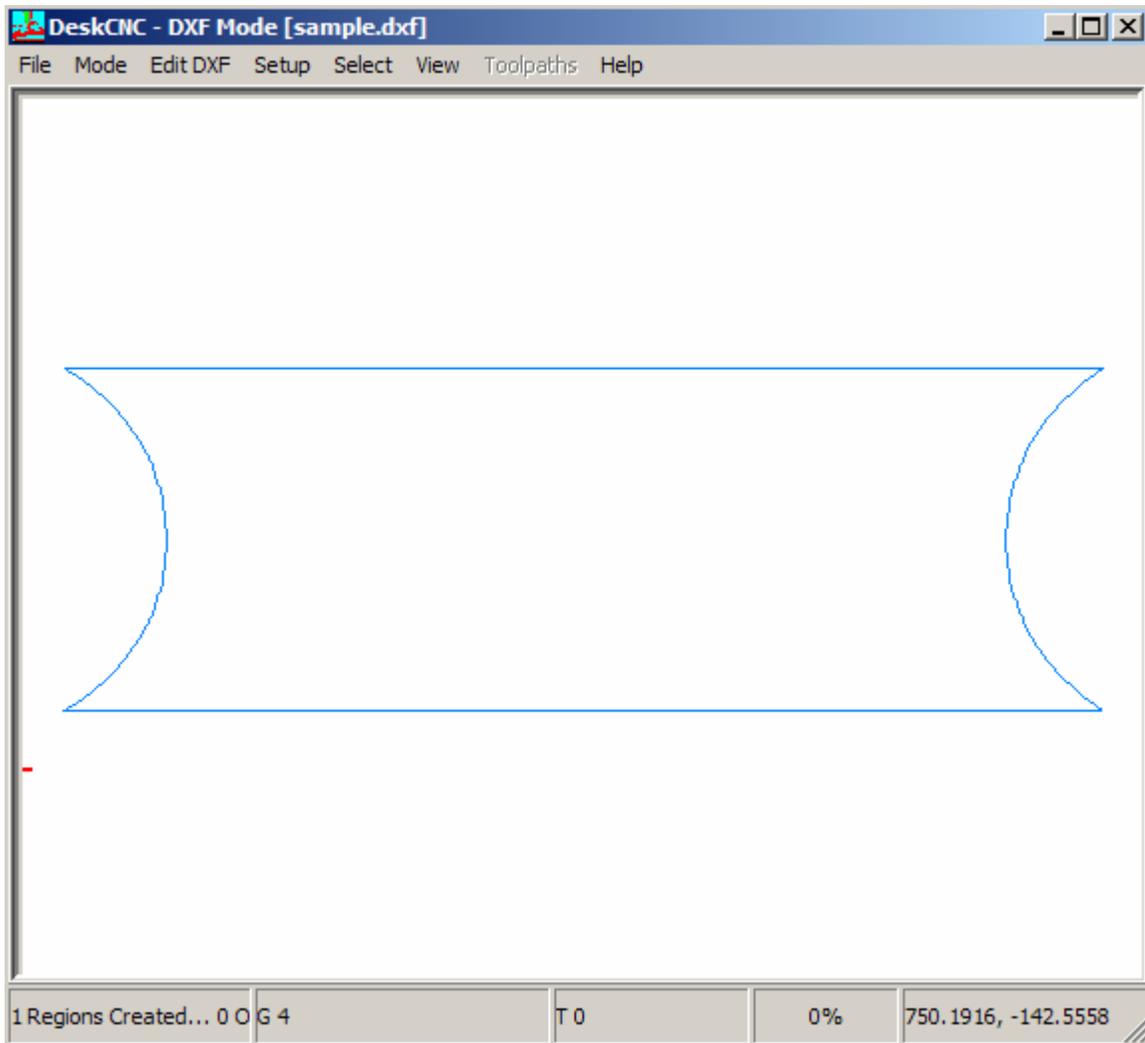
Configure DeskCNC.

1. Machine Setup
 - a. Click Setup->Machine Setup
 - b. Click DeskCNC Setup
 - c. Set the Units to match your units
 - d. Click Save
2. Post Processor Setup. **This must be done every time the program is started**
 - a. Click Setup->Post Processor Setup

- b. Click General
- c. Click the Name dropdown and choose BUGO
- d. Click Edit to load the configuration
- e. Click Cancel

Load the DXF

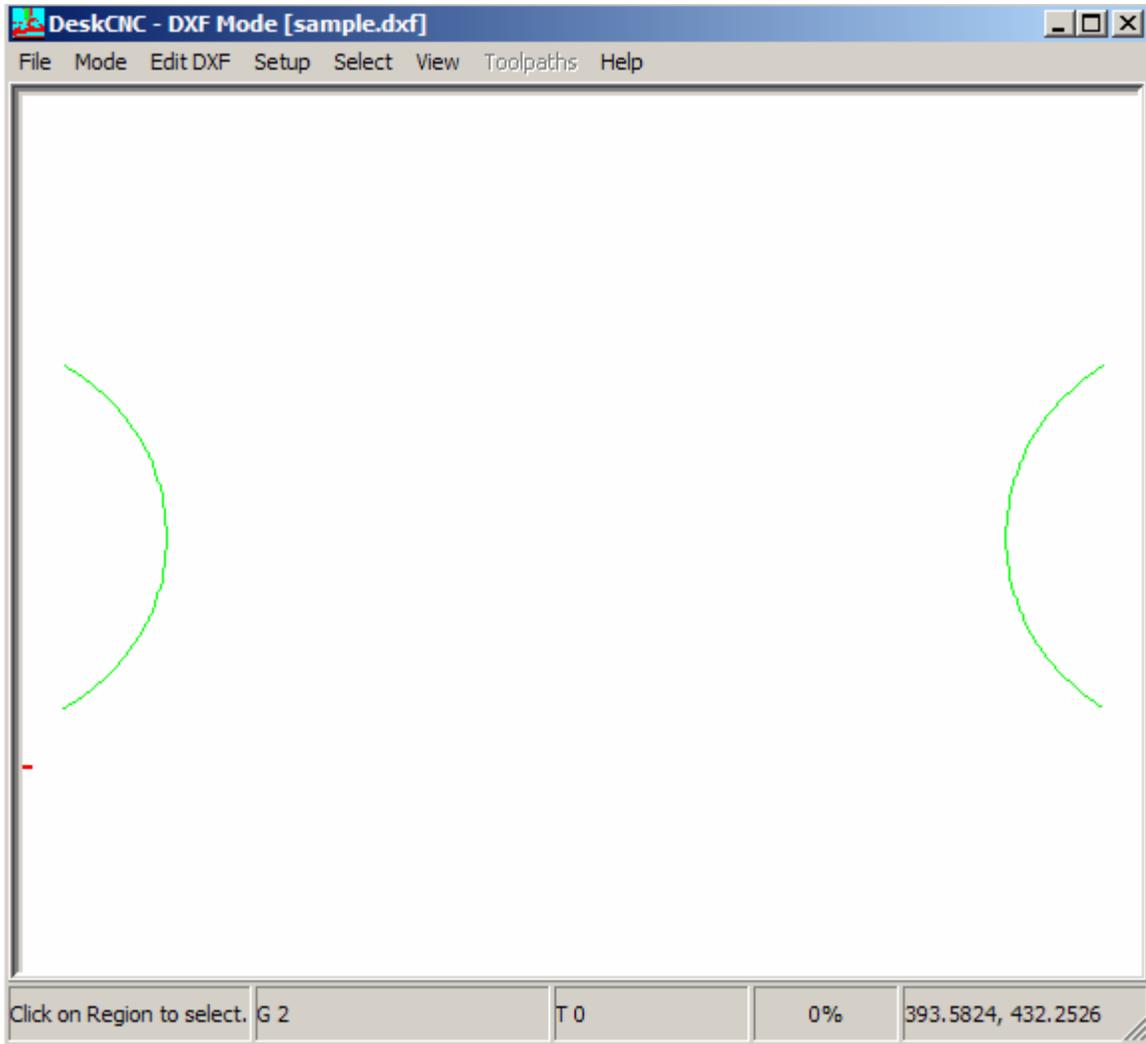
1. Click File, Open DXF
2. Choose
sample.dxf



Edit the shape

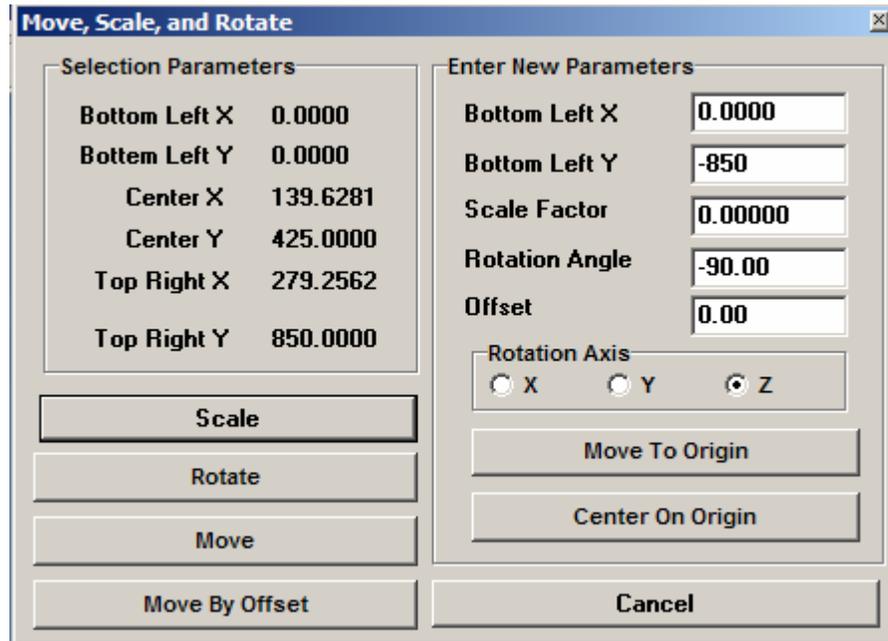
You will have to make changes to the loaded DXF file to tell the machine how to cut it. At the minimum, you will have to move it to or below the origin and rotate it properly. The select menu is at the top of the screen, but can also be opened by right clicking on the drawing.

1. Remove any extra lines
 - a. Select->Select all
 - b. Edit DXF->Explode Selected Regions
 - c. Select->Select Single
 - d. Click each extra line that must be removed
 - e. Edit->Delete Selected Regions
 - f. Click OK
 - g. Select->Select All
 - h. Edit DXF->Join Selected Regions



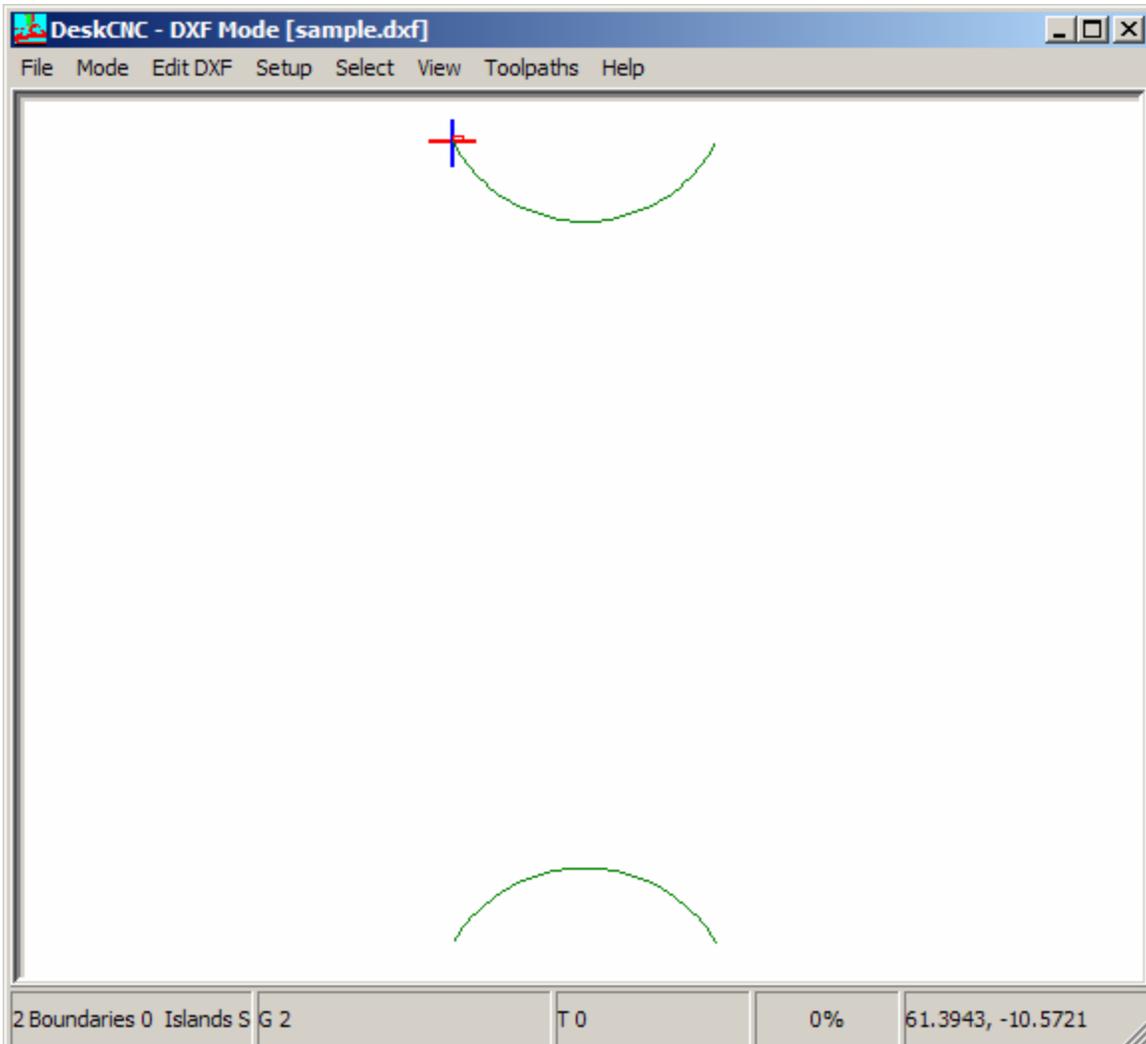
2. Move and Rotate the Shape. The shape to cut must be vertical and at (below) the origin. If it is not, it must be rotated and moved.
 - a. Select->Select All
 - b. Edit DXF->Scale, Move, Rotate Selected
 - c. Enter 90 in Rotation Angle to rotate left, -90 to rotate right.
 - d. Click Rotate
 - e. Click Move To Origin. If you leave the shape like this, the track will have to start close to the machine, move away from it to the top of the shape, then begin cutting. To make this faster, you want to move the shape below the X axis.
 - i. Take the number displayed next to Top Right Y. Enter the negative of this number in Bottom Left Y.

- ii. Click
Move



- iii. Top Right Y should now display 0

- f. Click View->Zoom All



Setup Cutting Paths

1. Enable path info display
 - a. View->Show Directions
 - b. View->Information Tags



Picture the MM1 at the bottom left of the image. The left side of the image is the starting point for rotation. This shows that the tractor will move from its origin toward the machine until it reaches point 1. The arc will turn on at point 1, cut the pipe near the machine, turn off, travel to point 2 at the end of the pipe, turn on, and then cut while turning in the opposite direction. This won't work because the pipe will be cut and unable to rotate. You must change the direction and order of the cuts

1. Change the order of the cuts
 - a. Edit DXF->Change Machining Order
 - b. Click Ok
 - c. Click each cut in the correct order
2. Change the direction of the cuts
 - a. Select->Select Single
 - b. Choose each cut that needs reversed
 - c. Edit DXF->Reverse Dir of Selected Regions

Now the machine will start from its origin at point 1, make the top cut, then move to the bottom for cut 2. It is ready to be turned into a toolpath.

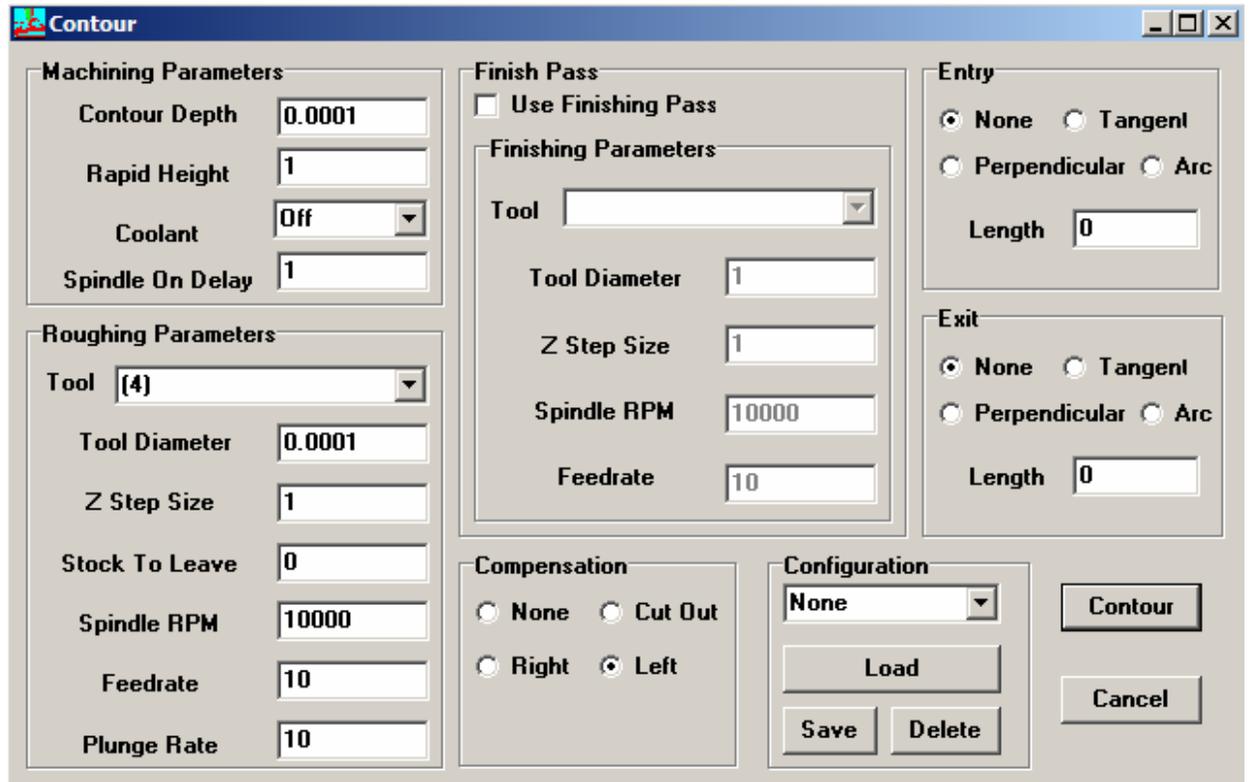


Creating Toolpaths

You must create the toolpath and save the resulting cnc file that will tell the machine how to move

1. Select->Select All
2. Toolpaths->Contour
3. Change Contour Depth to .0001
4. Change Tool Diameter to .0001
5. Change Compensation to Left

6. Click Contour

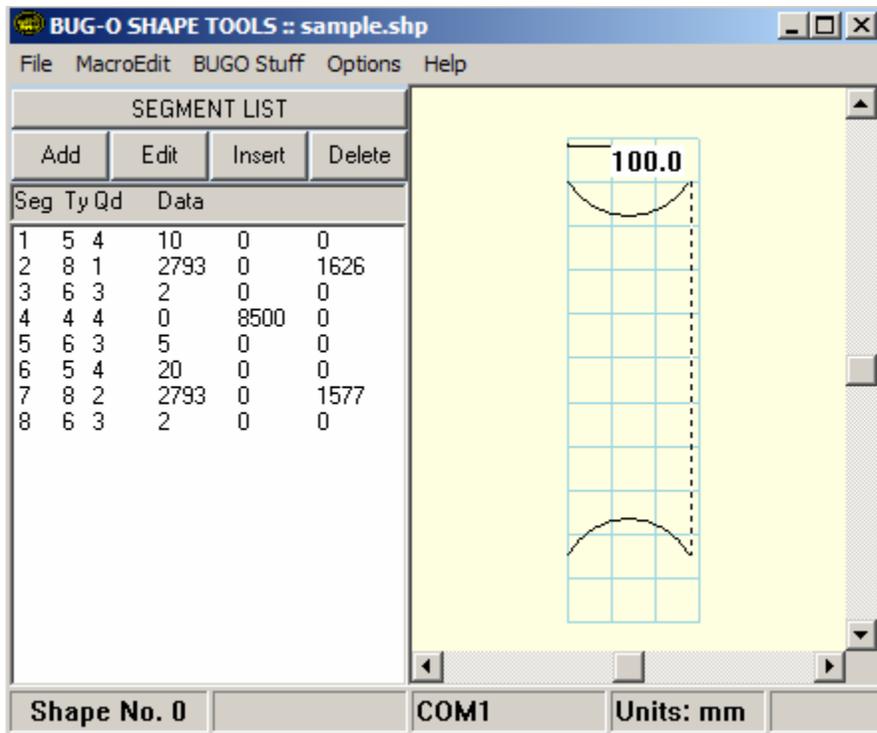


7. Click File->Save Toolpaths CNC.
8. Make sure the Save as type is BUGO
9. Save the file. It can now be imported into the Shape program.

Importing a shape

You must import the shape into the Shape program before it can be cut.

1. Start the Shape program
2. File->Open
3. Change Files of type to CNC Files
4. Find and open the CNC file you created in DeskCNC



You can now work with the Shape program like normal