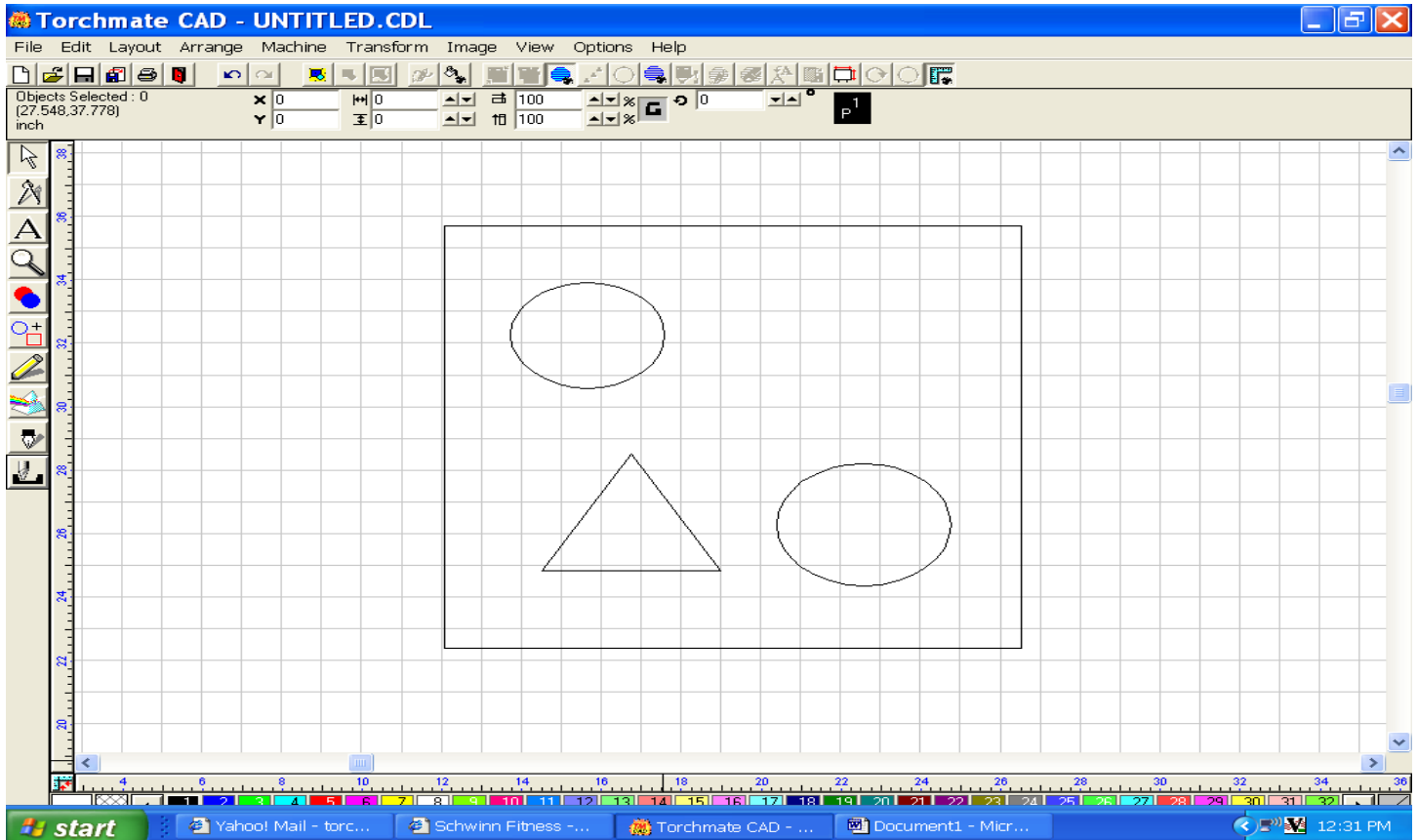
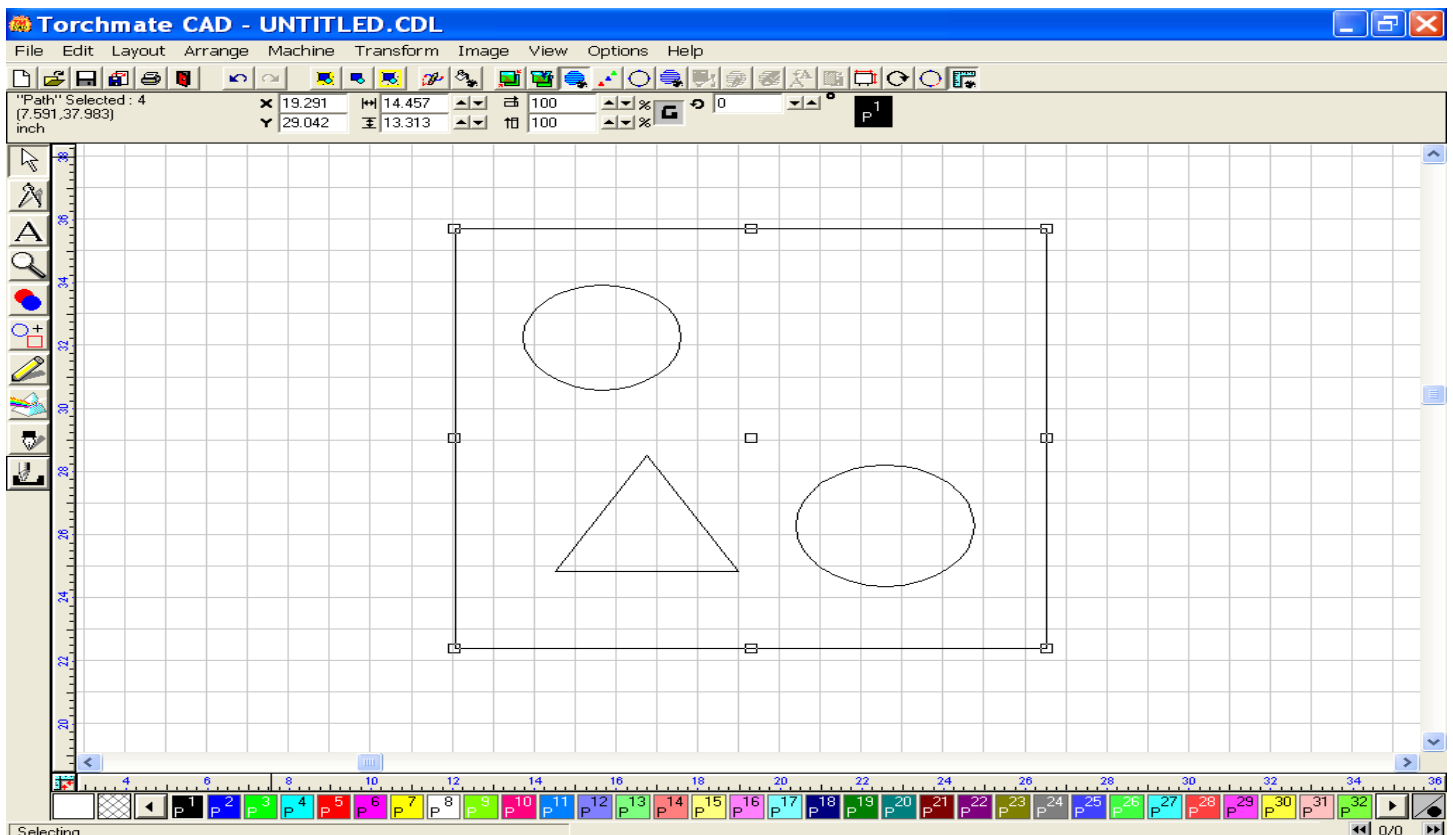


Creating Tool Paths

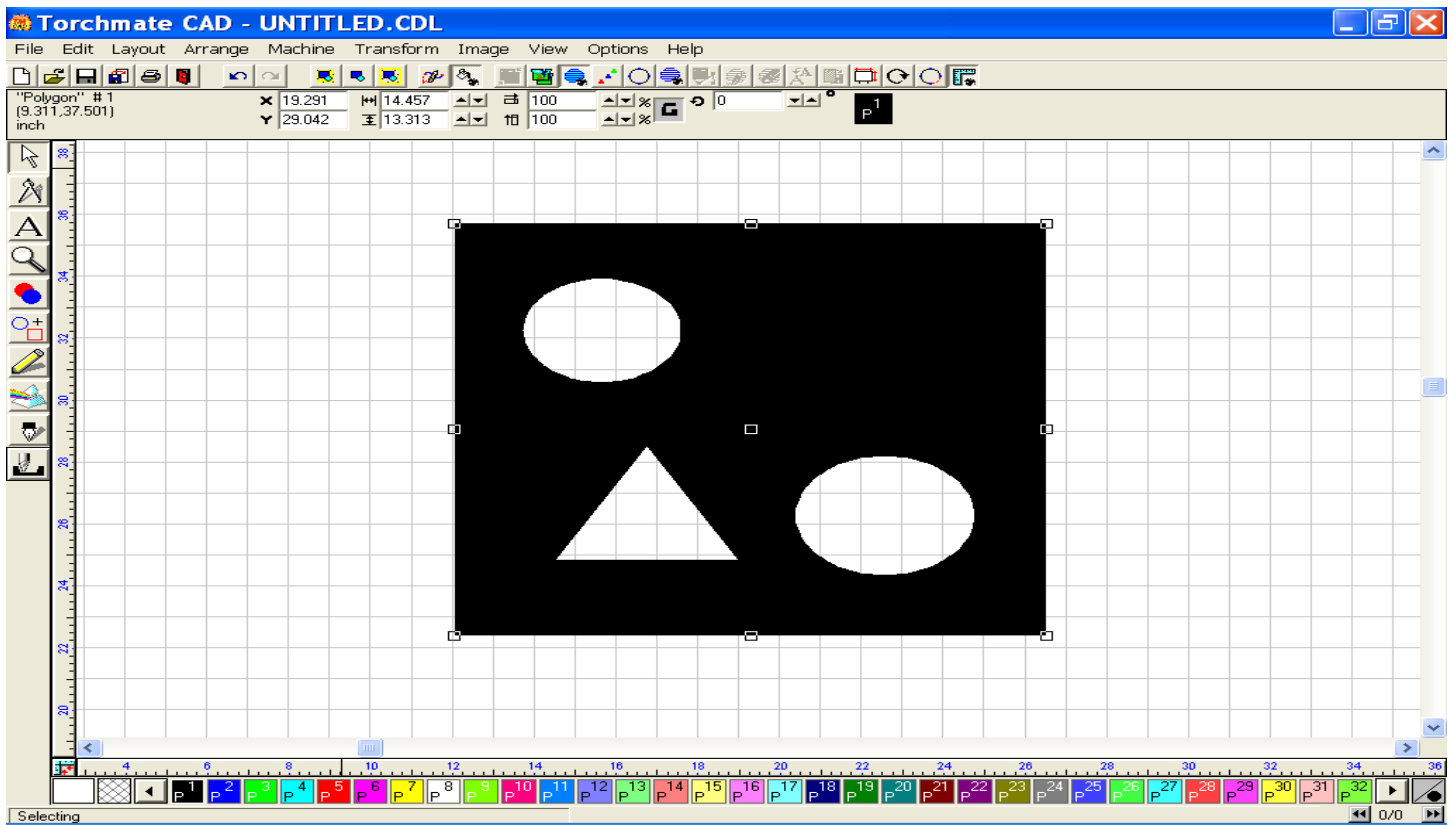
Start by drawing the square shape, and make it large enough for 3 smaller images to be fit inside. Draw two circles and a three sided polygon (Triangle), and place them inside the larger square shape.



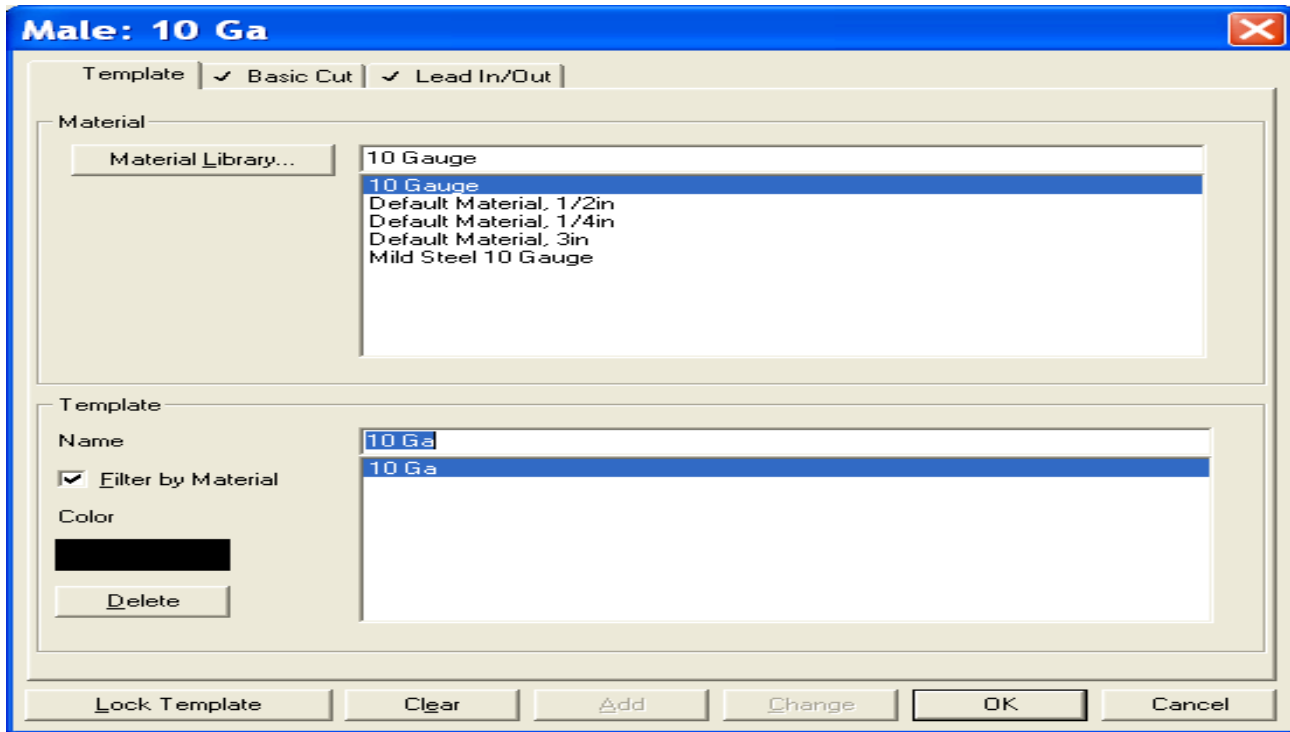
Using the Edit Menu + Select All, select the objects. Go to the Arrange menu and select Make Path.



When the make path is complete it will recognize the images as one part. To verify this, go to the View menu and click on Show Fill. The image will turn solid black, except for the pieces to be cut out.

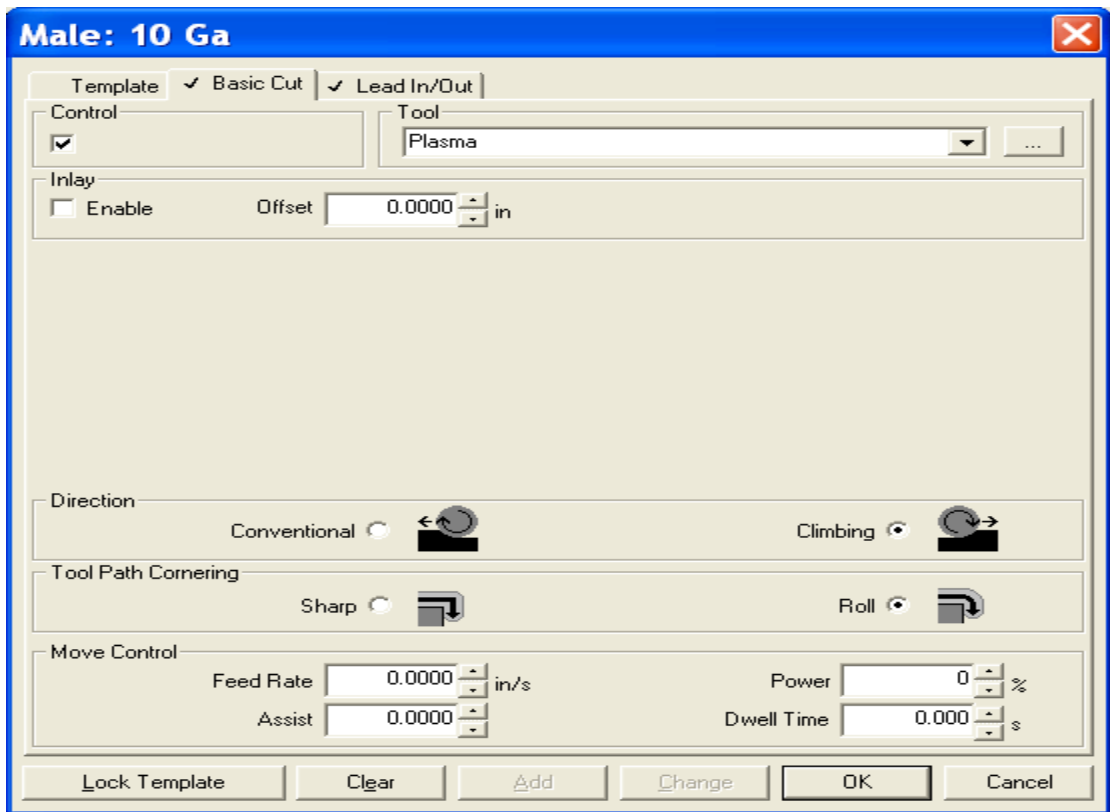


Now open the Machine menu, and select Create Tool Path + Male.

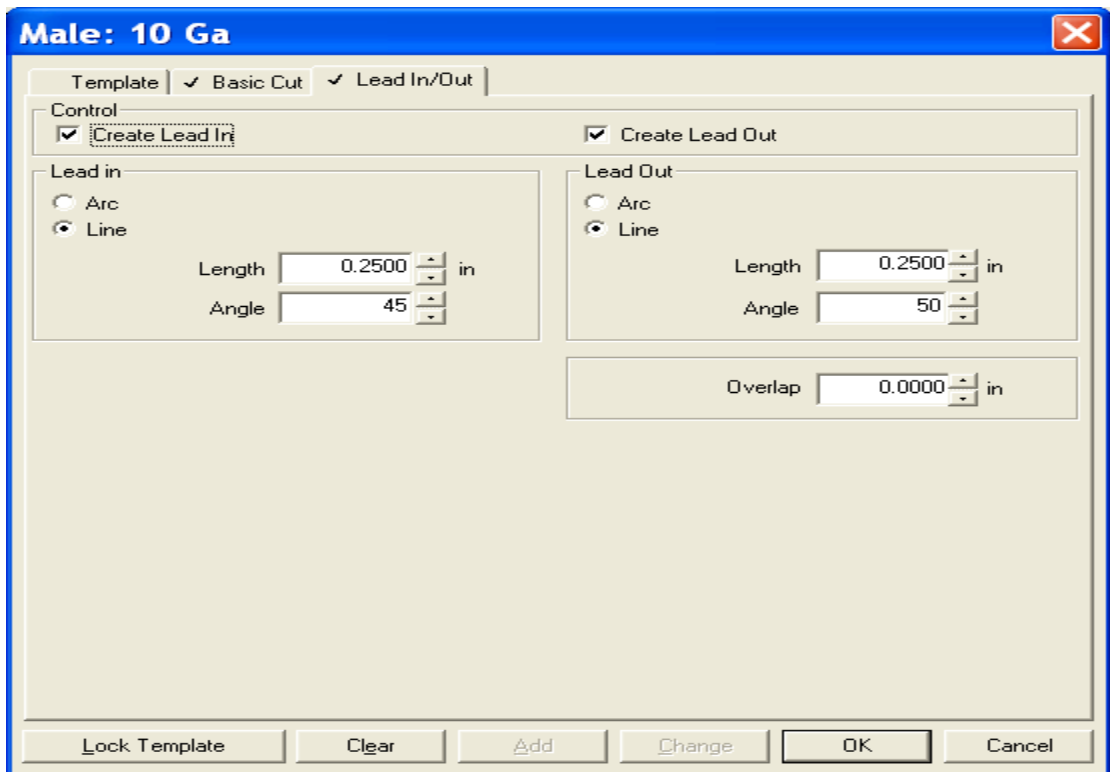


The Male default template window will appear. The only two pages you need to be concerned with are the Basic Cut & Lead In/Out pages (notice check marks).

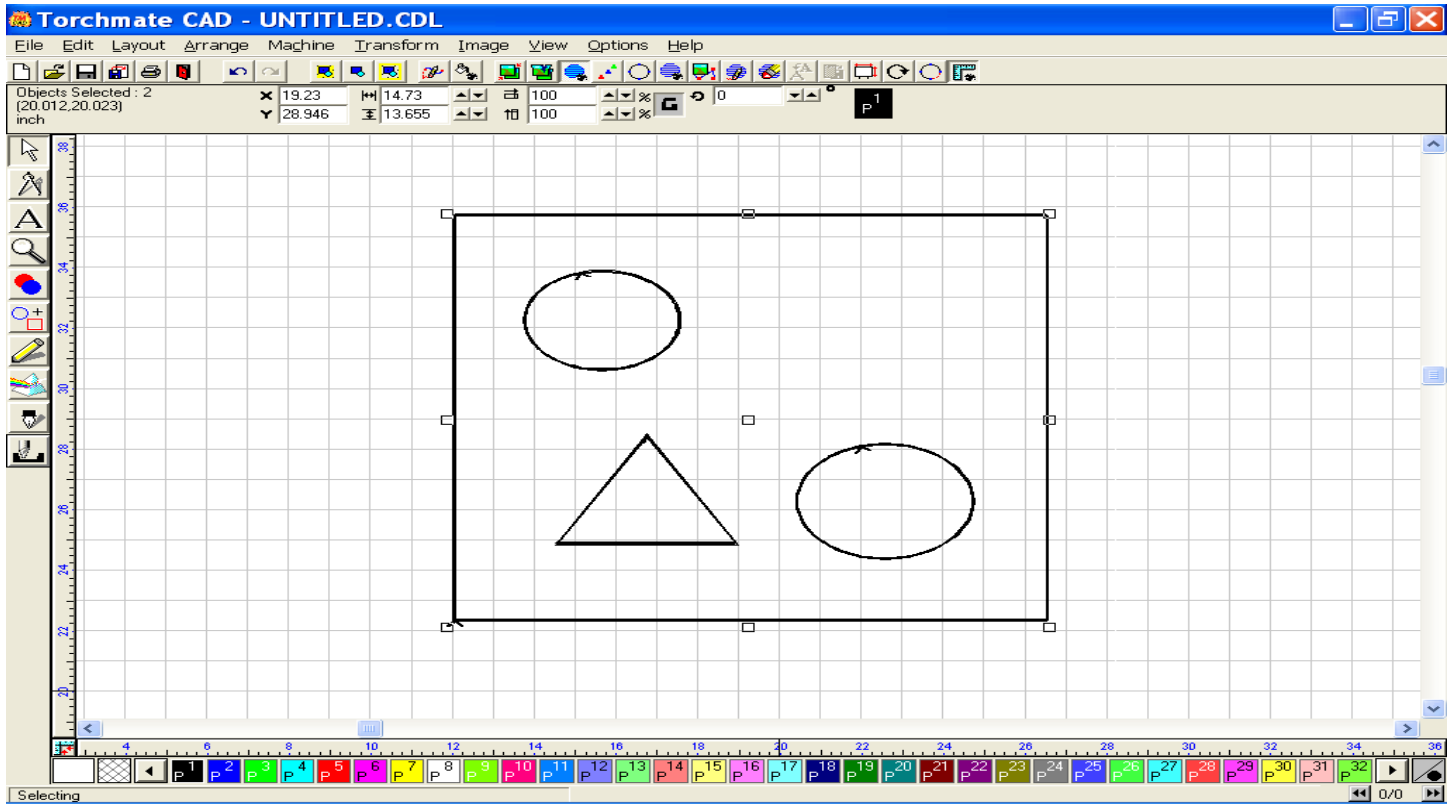
For Basic Cut, check Control, Tool = Plasma, Select your Direction, and Tool Path Cornering method. Now click on the Lead In/Out page.



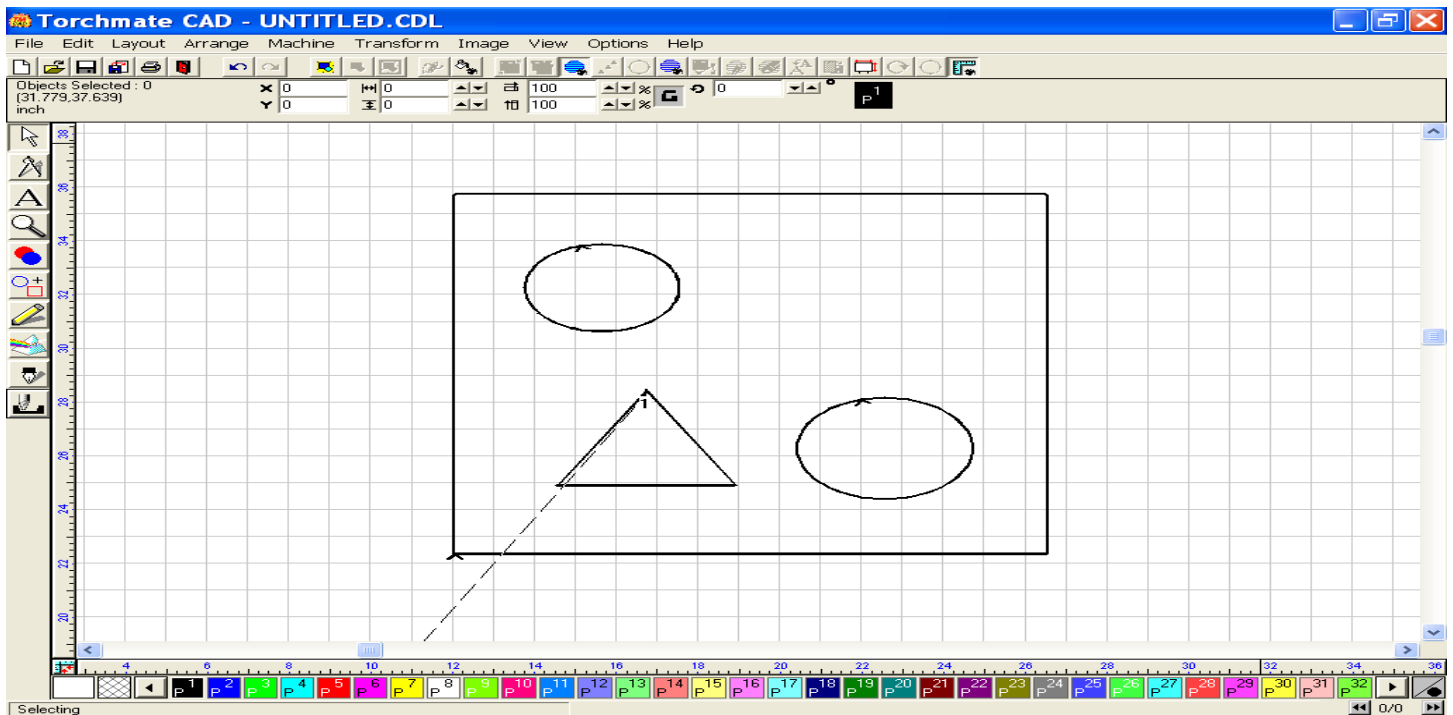
If you want a Lead In and a Lead Out, put a check mark next to each. Choose between using an Arc or a Line. If you choose to use Arc, you will be asked what radius of the arc you want to use, and you will enter the amount of radius. If you choose to use Line, you will be asked what length of line, and you will enter the length of line to be used. You will also be asked what angle of Lead In or Out to utilize. I tend to use Lead angle of 45 and 55 respectfully, as it will clearly show two leads, as opposed to having them the same angle, and thus created on top of each other.



This view shows the created Tool Path with lead In/Out (notice the v shape inside the circles and on the outside of the square). If you do not see this view, go to the View menu and select Show Tool Paths. Before we can export this image, or part, we will need to delete the original image (otherwise you will send two tool paths to the driver software controlling the gantry). Turn OFF the Show Tool Path option in the View menu, click on the thin line of the original image, and then press the Delete key on your keyboard. The image should vanish. Return to the View menu to turn the Show Tool Path option back ON.



If you want to check the sequence of your cuts, go back to the View menu and select the Show Tool Path Viewer option. A small window will appear where you may specify the use of Show Direction and Show Order. Select Show Order. A dotted line should appear, traveling toward the first cut, which in this case should be to the inside of the triangle (1). This verifies that the first cut is to be on the inside of the part. The other two interior images will also be cut prior to the exterior shape, or tool path. Turn of the viewer before proceeding.



Export the image (DXF) to a folder of your own choosing, but some place easy to find, like My Documents. Minimize the Torchmate cad window.

Open the Torchmate Driver software (versions 1.42, 1.61, 2.09, or 2.10) and import the DXF file. As you can see below, the created program is cutting all the interior shapes prior to the exterior shape.

The screenshot shows the Torchmate software interface. The main window displays a CAD drawing on a grid. The drawing consists of a square with a red outline and a blue outline. Inside the square, there are two circles and a triangle, all outlined in blue. A dotted blue line indicates the tool path, starting from the origin (0,0) and moving to the top-left corner of the square, then to the top-right corner, then to the right side of the square, then to the bottom-right corner, then to the bottom-left corner, and finally back to the origin. The G-code program listing is shown in the bottom-left corner, with line 97 highlighted. The program coordinates are shown in the top-right corner, with X=14.6235 and Y=13.5793. The tool is set to None, and the offset is X=0.0000, Y=0.0000, Z=0.0000. The compensation is set to Positive (G43) and the diameter is 0.0000. The status is Offline.

Torchmate - CTP.fgc - Torchmate.stp - Torchmate.tlg

File Configuration Controller View Coordinates Help

Program Coordinates Set 0

X 14.6235
Y 13.5793

Tool: None
Offset: X 0.0000 Y 0.0000 Z 0.0000
Comp: Positive (G43) Diam: 0.0000

Offline

G-Code Jog Point Home Aux

Jump To Line Faster Program Feedrate: 100.0
Cancel Go Slower 100 % Override: 100.0

Run Time: 00:01:04
To Go: 00:00:17

Reset Step Continuous
Start Feed Hold (Any Key)

```
91 N870 G02 X 0.1250 Y 0.2174 I 0.0001 J 0.0377
92 N880 G01 X 0.1232 Y 0.2290
93 N890 G01 X 0.1237 Y13.5479
94 N900 G02 X 0.1491 Y13.5777 I 0.0378 J-0.0065
95 N910 G01 X 0.1607 Y13.5795
96 N920 G01 X14.6236 Y13.5790
97 N930 G02 X14.6534 Y13.5536 I-0.0065 J-0.0378
98 N940 G01 X14.6552 Y13.5420
99 N950 G01 X14.6547 Y 0.2231
100 N960 G02 X14.6293 Y 0.1933 I-0.0378 J 0.0065
101 N970 G01 X14.6177 Y 0.1915
```

This file should help to clarify the procedures for creating tool paths. If you find that you are still having difficulties, please don't hesitate to contact Torchmate Tech Support.