Single Sided PC Board Layout
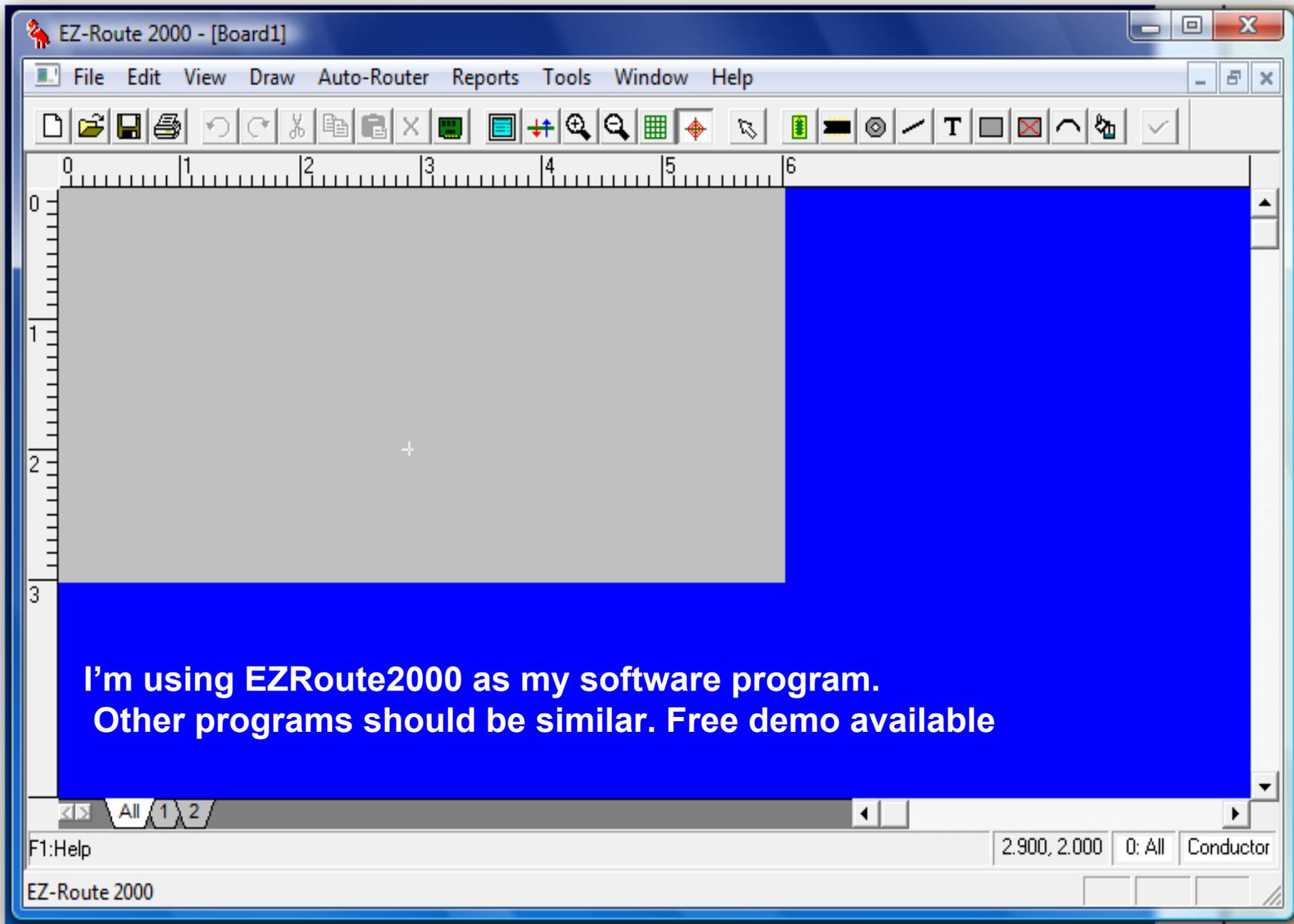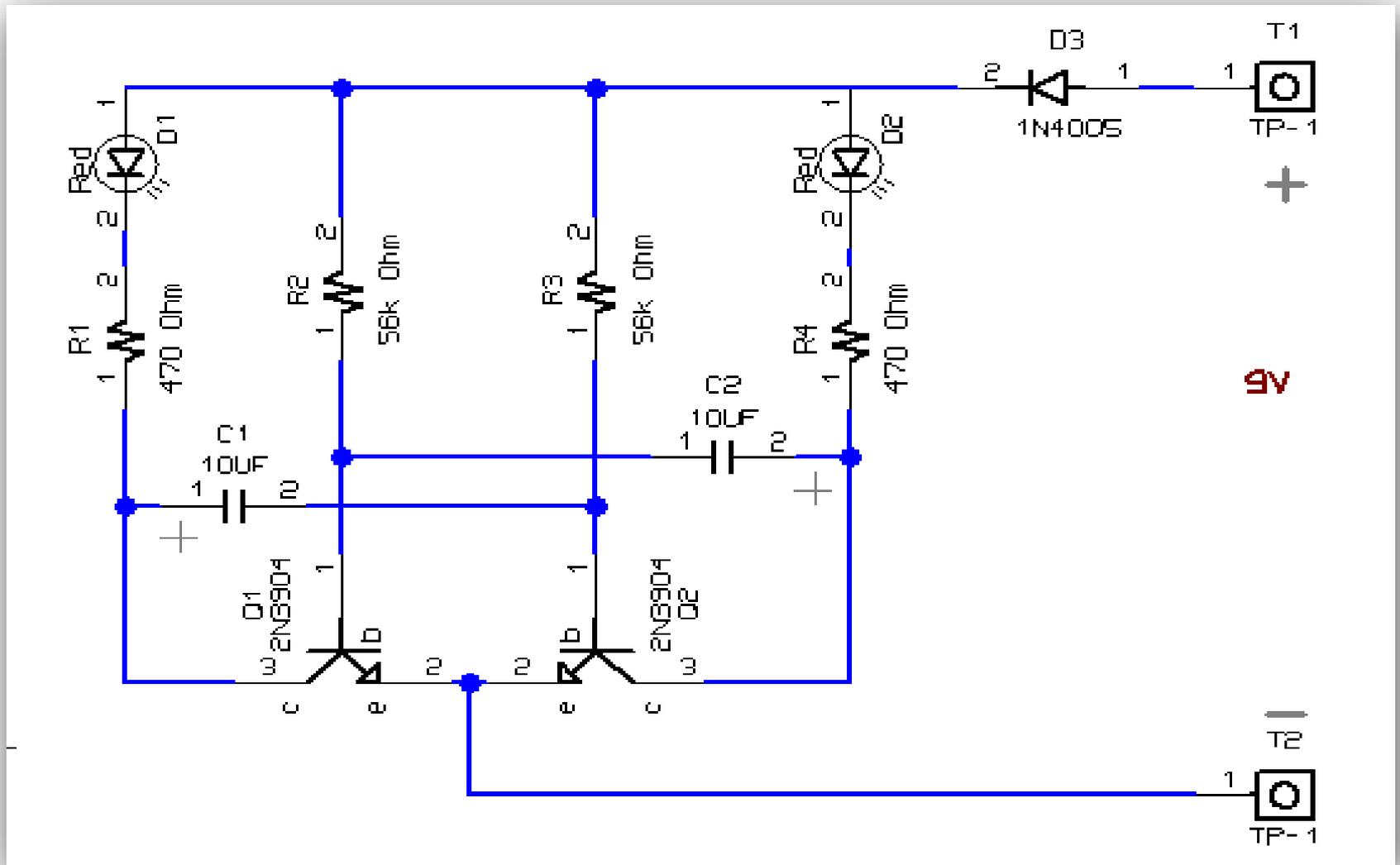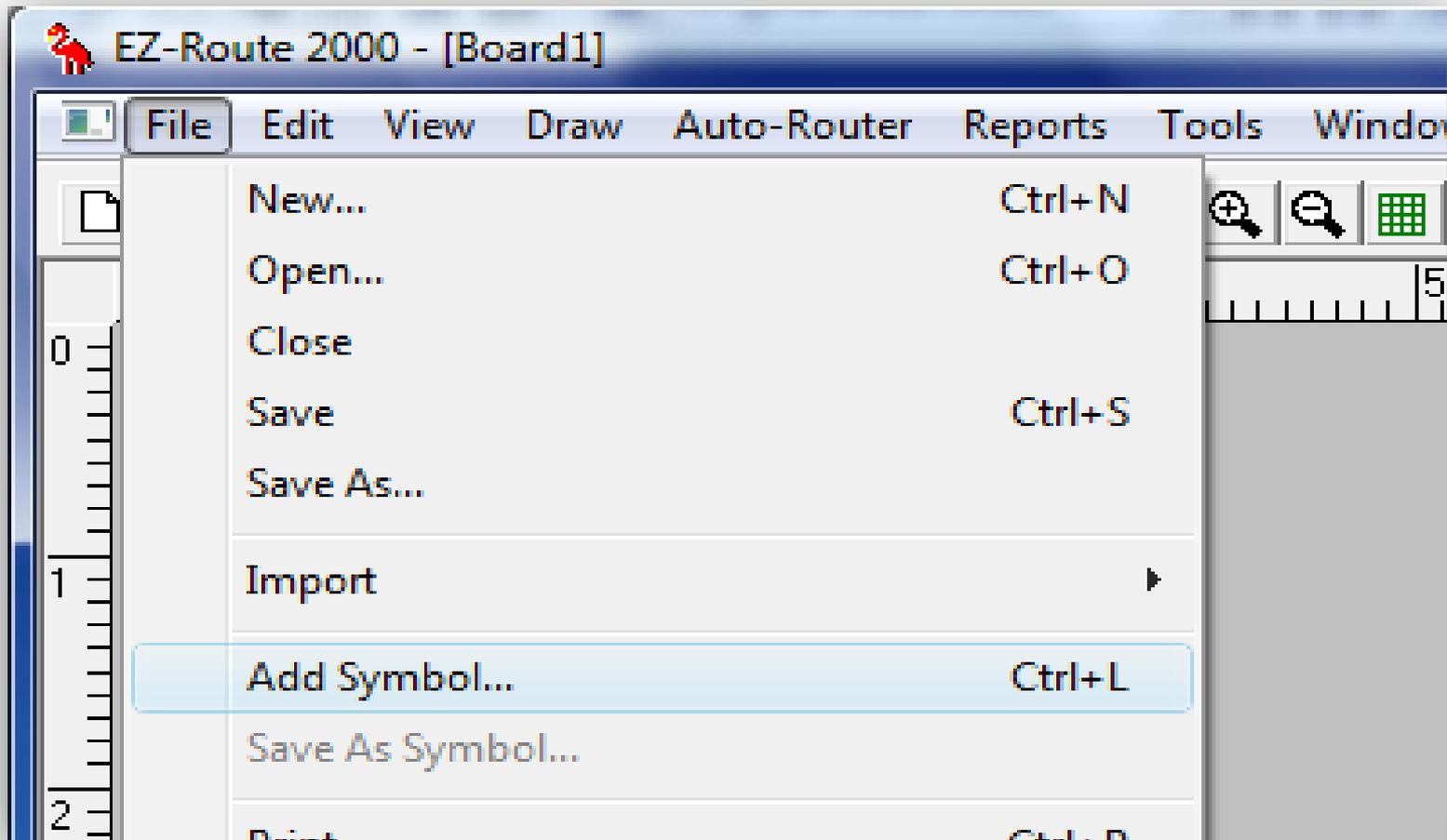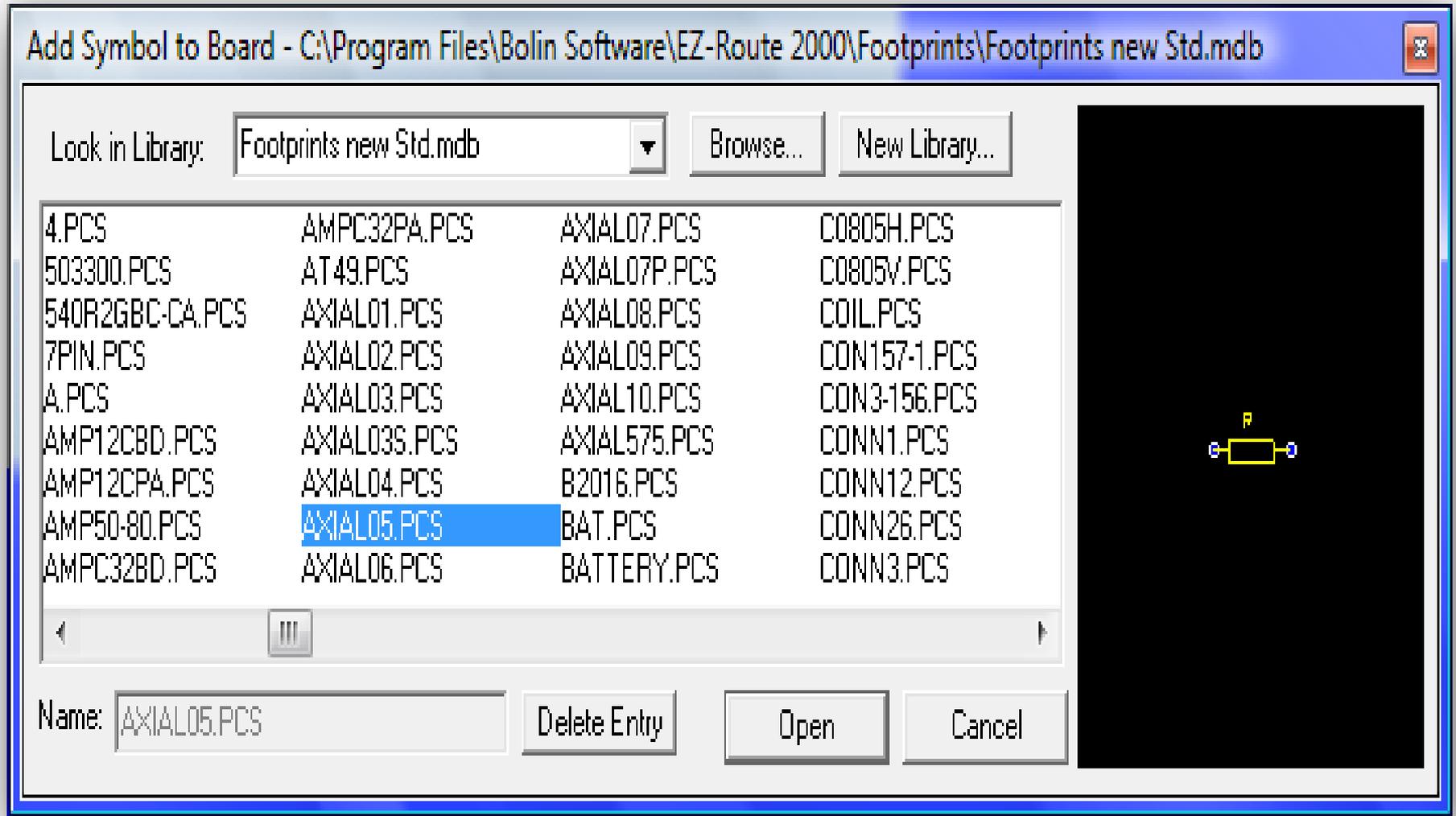
I'm using EZRoute2000 as my software program.
Other programs should be similar. Free demo available

Simple circuit as my example

Use Add Symbol to find your footprints library

Add Symbol to Board - C:\Program Files\Bolin Software\EZ-Route 2000\Footprints\Footprints new Std.mdb

Look in Library: Footprints new Std.mdb

Browse...   New Library...

4.PCS	AMPC32PA.PCS	AXIAL07.PCS	C0805H.PCS
503300.PCS	AT49.PCS	AXIAL07P.PCS	C0805V.PCS
540R2GBC-CA.PCS	AXIAL01.PCS	AXIAL08.PCS	COIL.PCS
7PIN.PCS	AXIAL02.PCS	AXIAL09.PCS	CON157-1.PCS
A.PCS	AXIAL03.PCS	AXIAL10.PCS	CON3-156.PCS
AMP12CBD.PCS	AXIAL03S.PCS	AXIAL575.PCS	CONN1.PCS
AMP12CPA.PCS	AXIAL04.PCS	B2016.PCS	CONN12.PCS
AMP50-80.PCS	AXIAL05.PCS	BAT.PCS	CONN26.PCS
AMPC32BD.PCS	AXIAL06.PCS	BATTERY.PCS	CONN3.PCS

Name: AXIAL05.PCS

Delete Entry   Open   Cancel

**AXIAL05 is my resistor footprint, it's shown on the right**

Place each footprint in a column. I find it better to load all the footprints needed before I start.
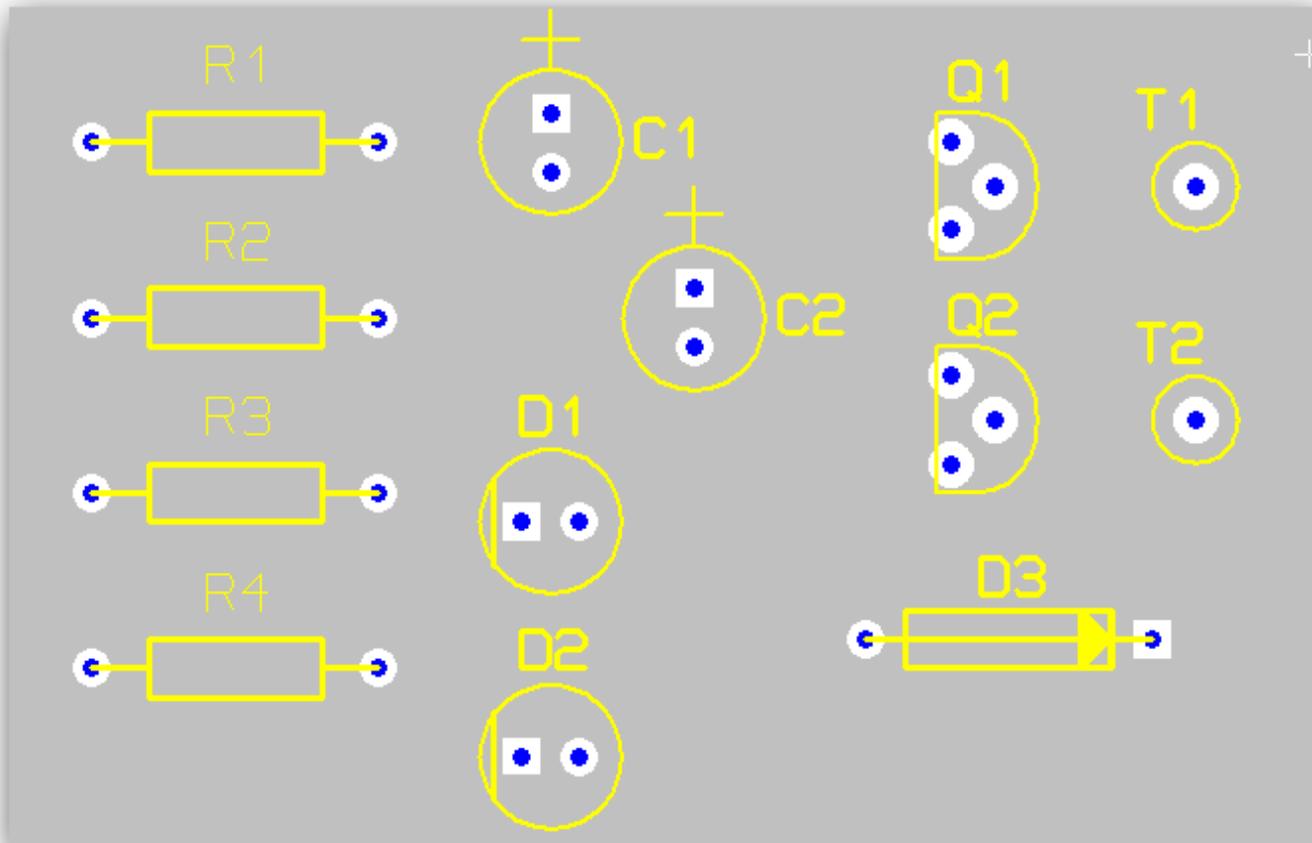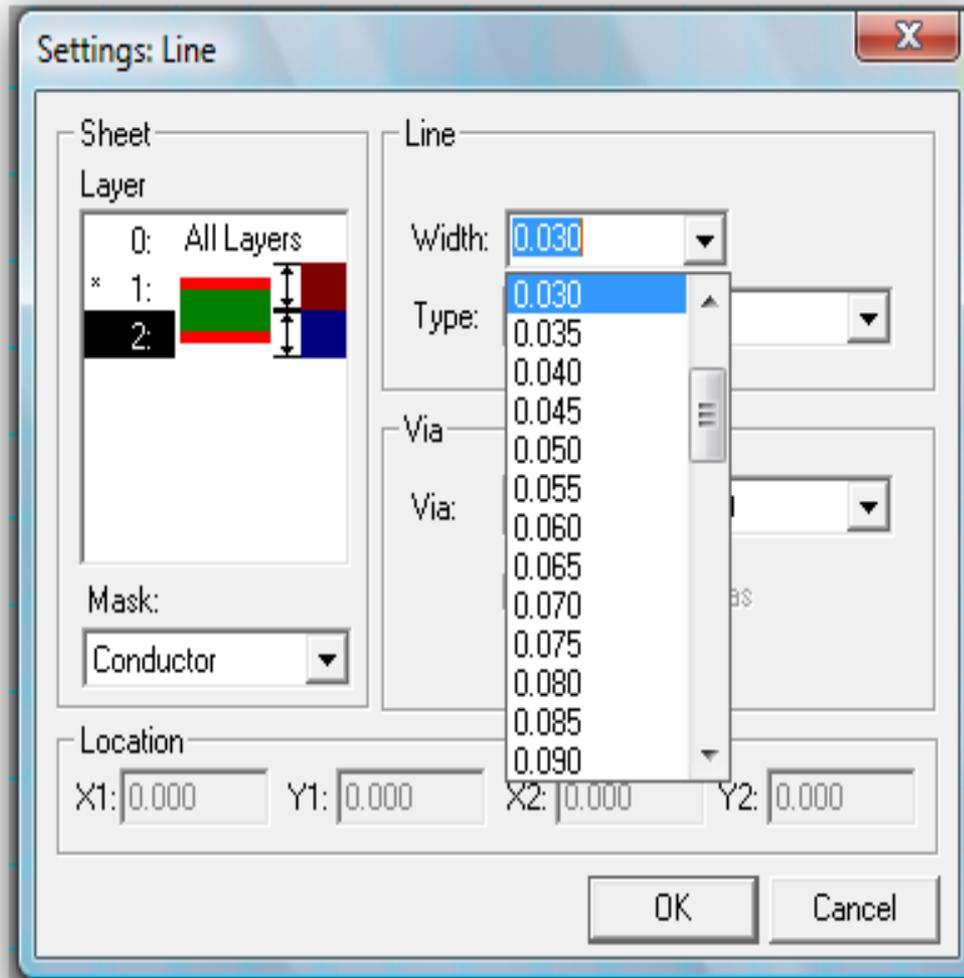
Use the on-screen grid to place each footprint in-line.

As you see I have followed the schematic symbols to roughly position each part.

You can make your pcb as big or as small as you want. Remember that the yellow silk screen lines should show each part in it's correct size.
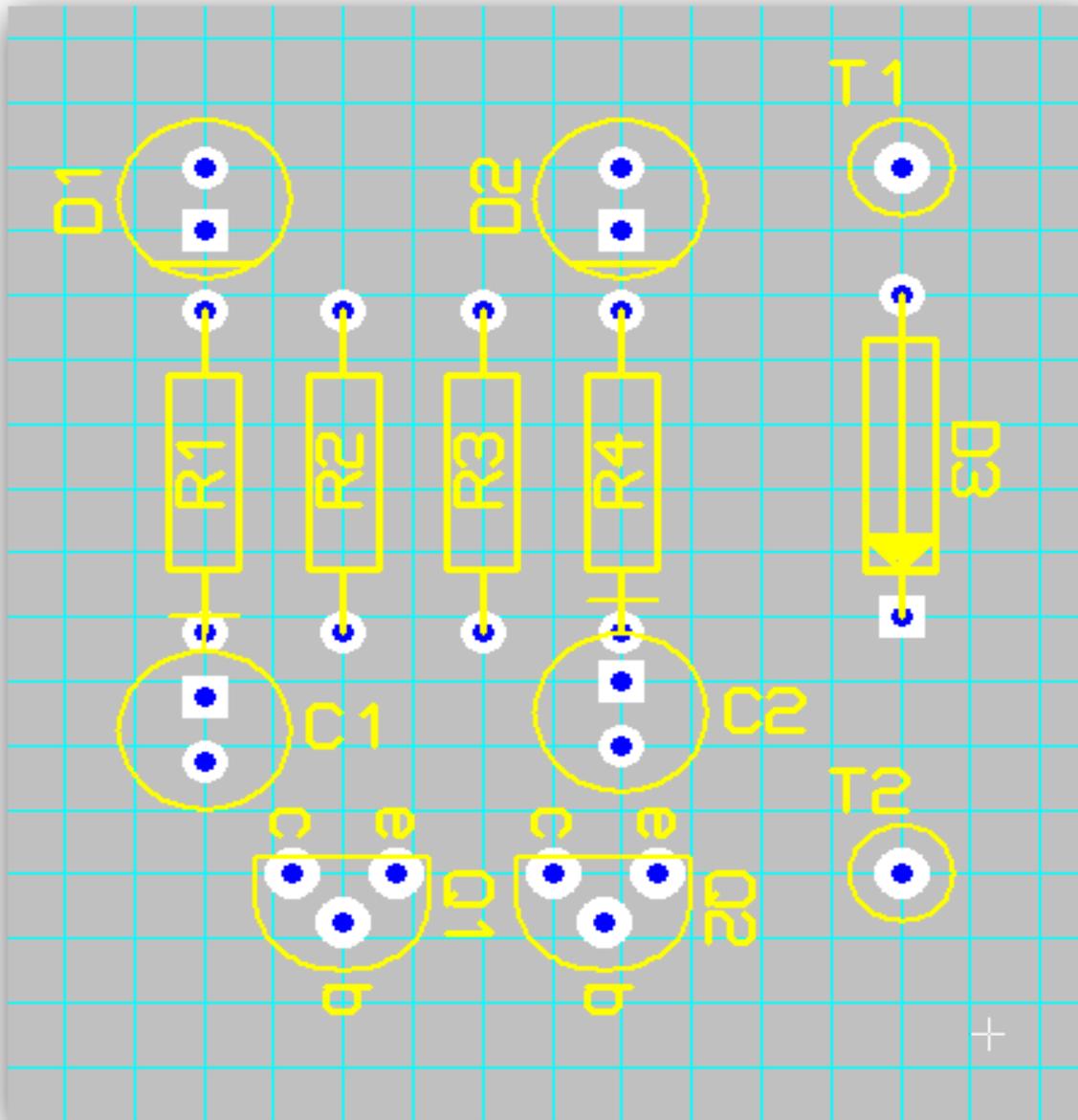
I have selected line settings and can change my line width, type of line   ie conductor, silk or many others. I have selected my line width as .030" and Layer 2 which is my solder side.
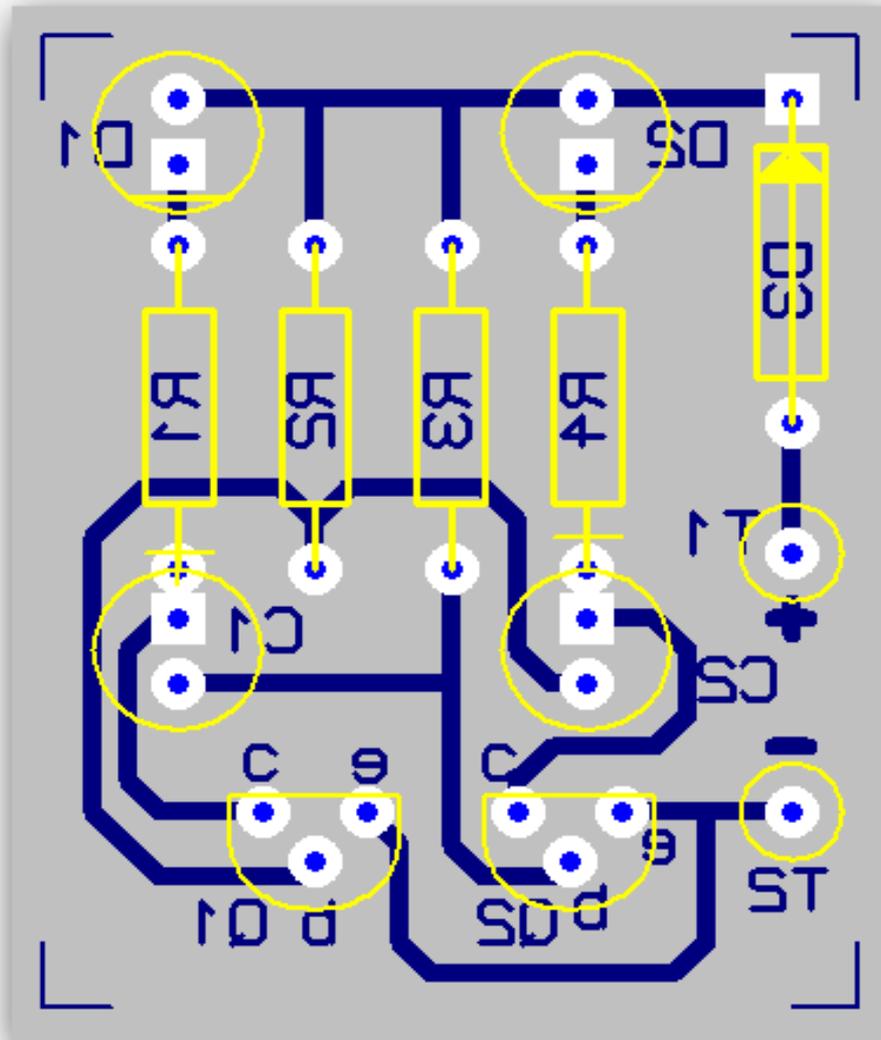
Since this is the solder side, I have mirrored all the text and changed it from silk screen to solder. This way I have my parts identified to help in assembly.

If you use Toner Transfer to make your boards, just print this side as it is. This will place the traces directly onto the paper.

 I have used blue as my solder side trace color and if I was making a component layer, it would be red. This is a normal standard and should be followed. (You can of course use what ever color is best for you)

As you see, I'm using a pcb program which allows for many layers but will be making a single sided board.

Using a program with at least two layers will give you a silk screen to actually screen print the component side part identities onto the board. You can also silk screen the solder side as well. This layer also gives you board assembly documentation.

Silk Screening is reasonably easy, a little messy though and a process used for making many boards. It would be a pain to make only one.

If you want to, you can use the silk screen layer, with modifications to make a copper silk screen instead of silk screening with paint by using a double side pc board.

Contact me at   wrj60@telus.net
for more information on this process or other board making information.