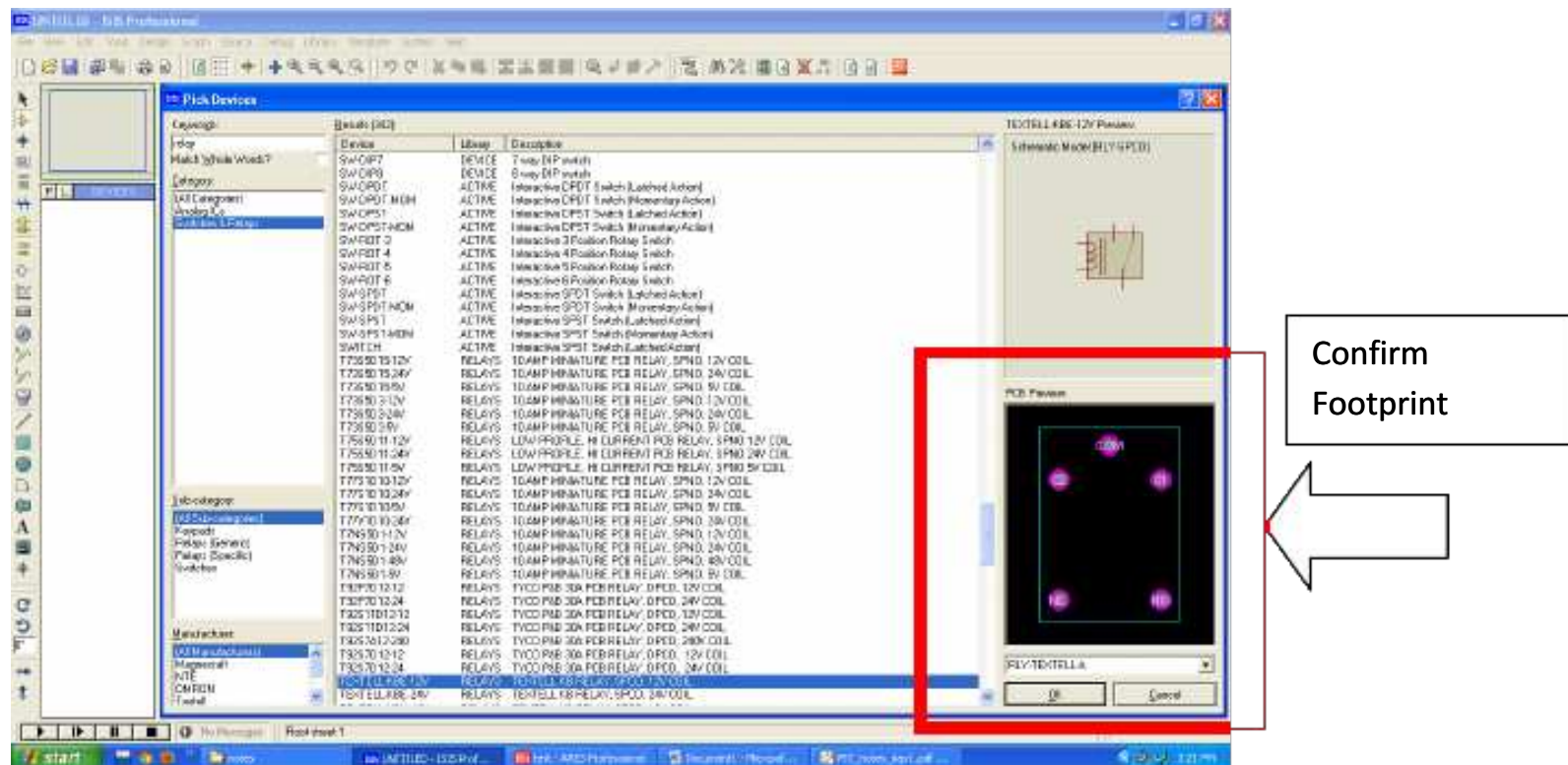


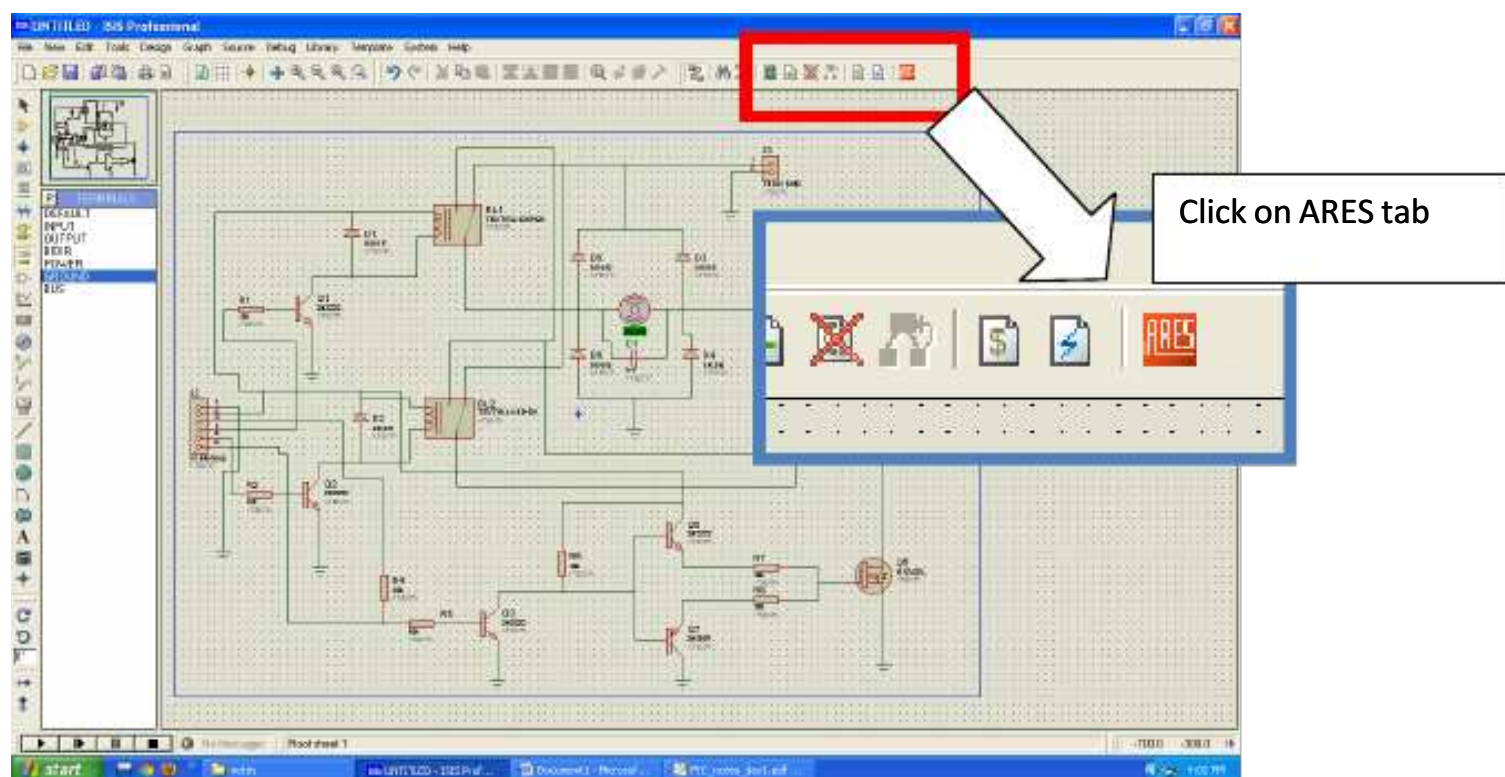
## PCB design using Proteus

**Prepared by : Mohamad Taib bin Miskon**

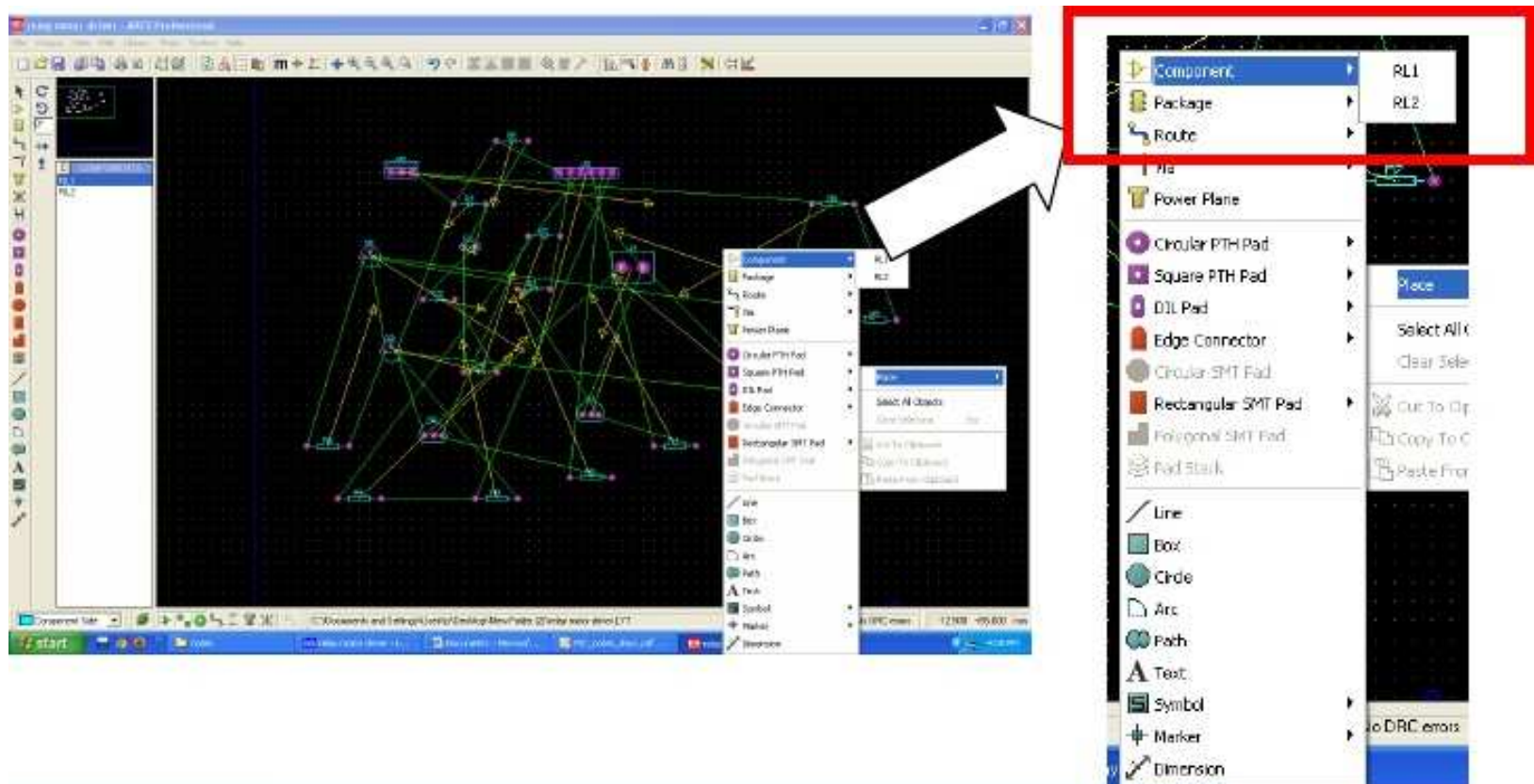
1. **Open ISIS professional and create a new file**
2. **Start drawing your schematic. Make sure all components have PCB footprint or you might need to consider replacement component with similar footprint.**



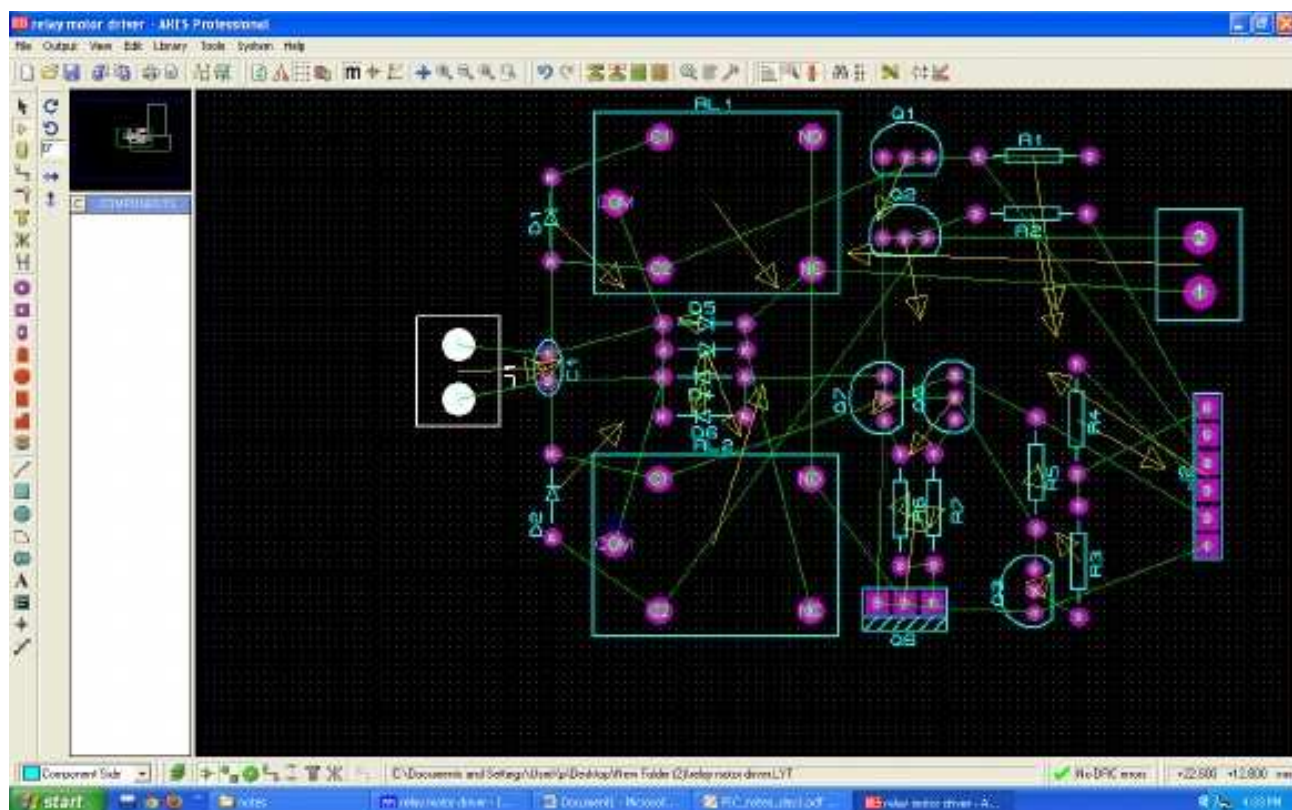
- 3. Once completed your schematic, click on ARES tab at the upper right of ISIS workspace**



4. A new ARES workspace window shall appear. Right click>Component>chose the component number and place one by one

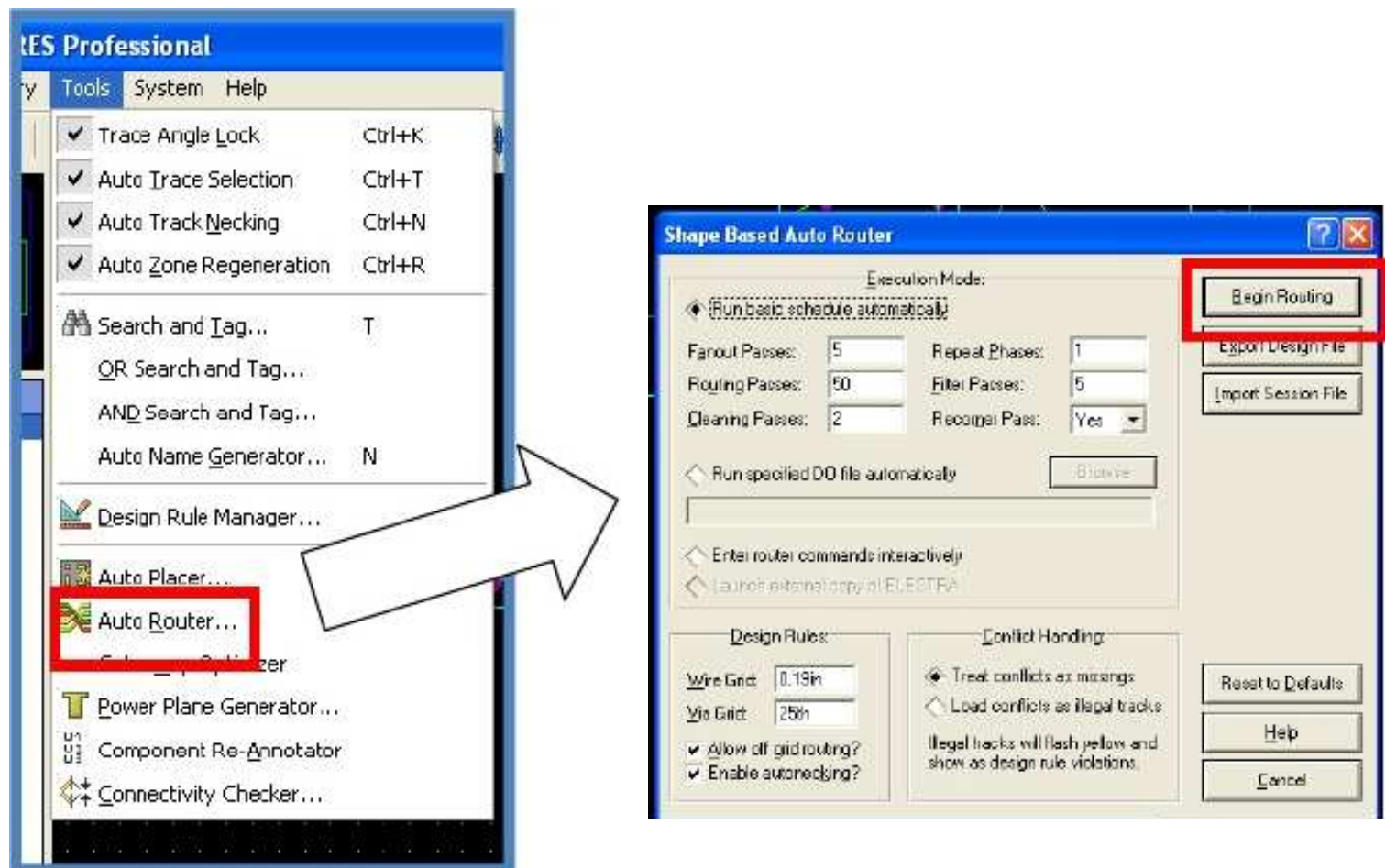


5. Rearrange your component

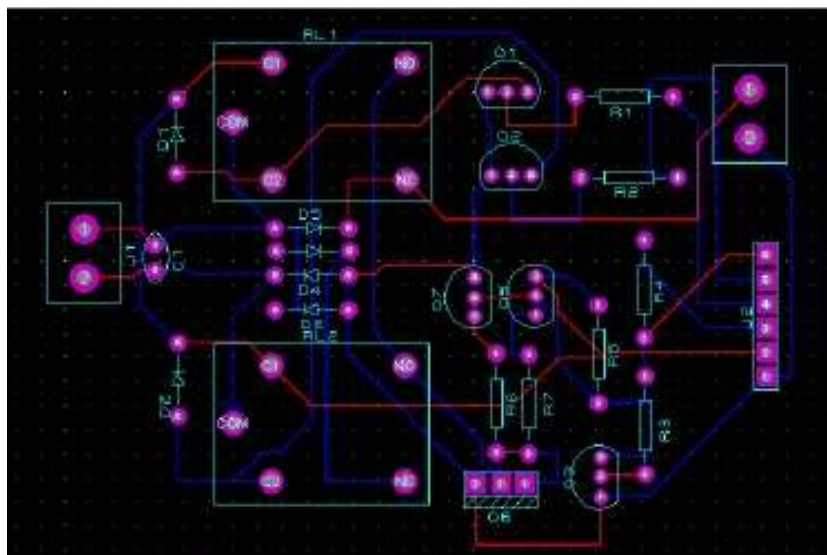




6. Click Tools>Auto Router. A shape based auto router window will appear and click Begin Routing

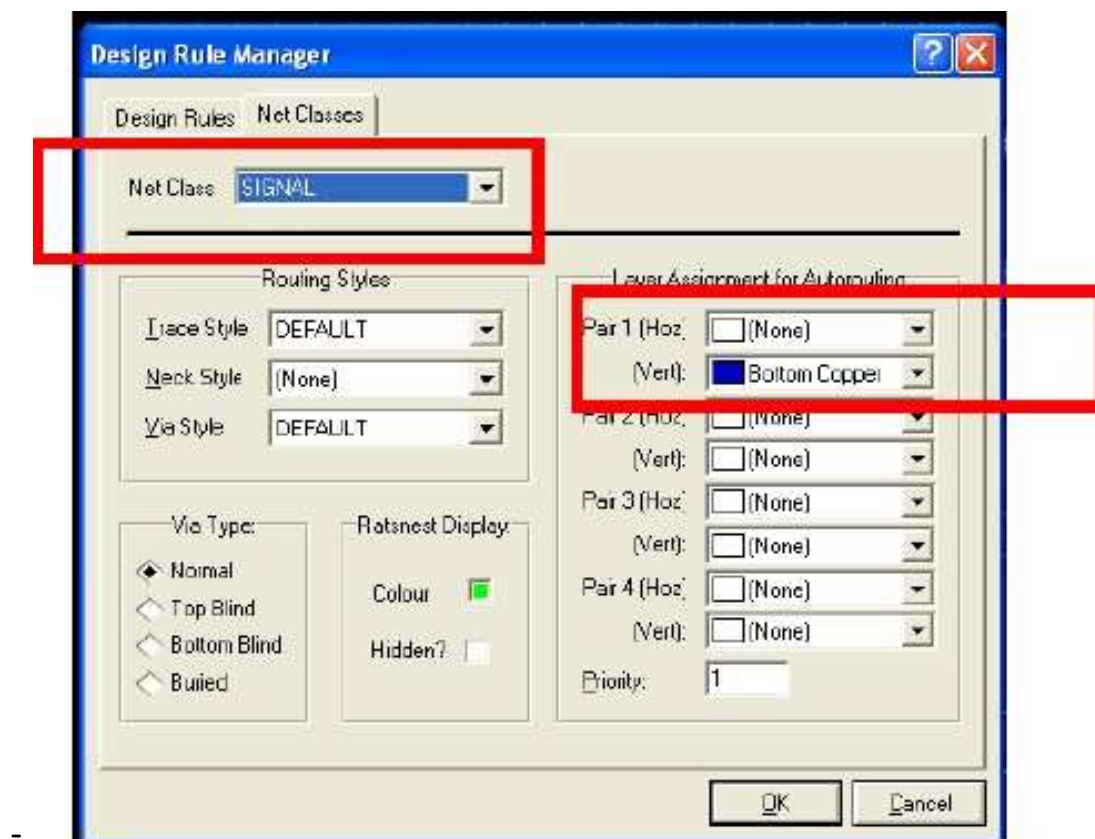
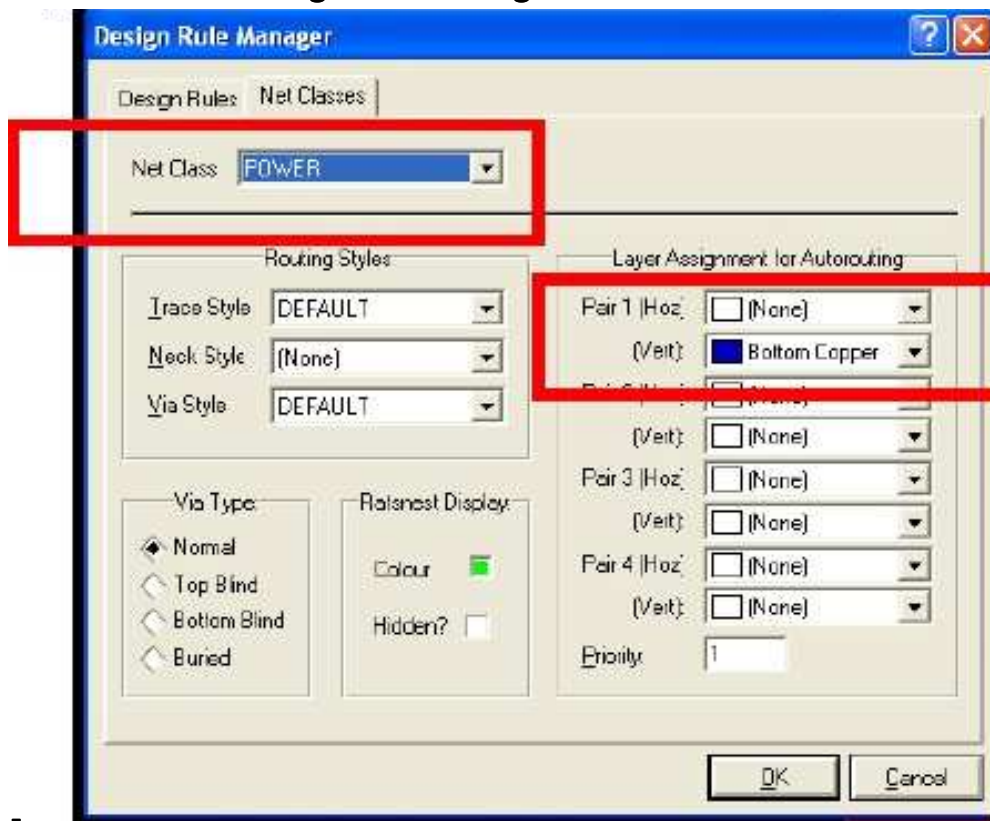


7. Result:



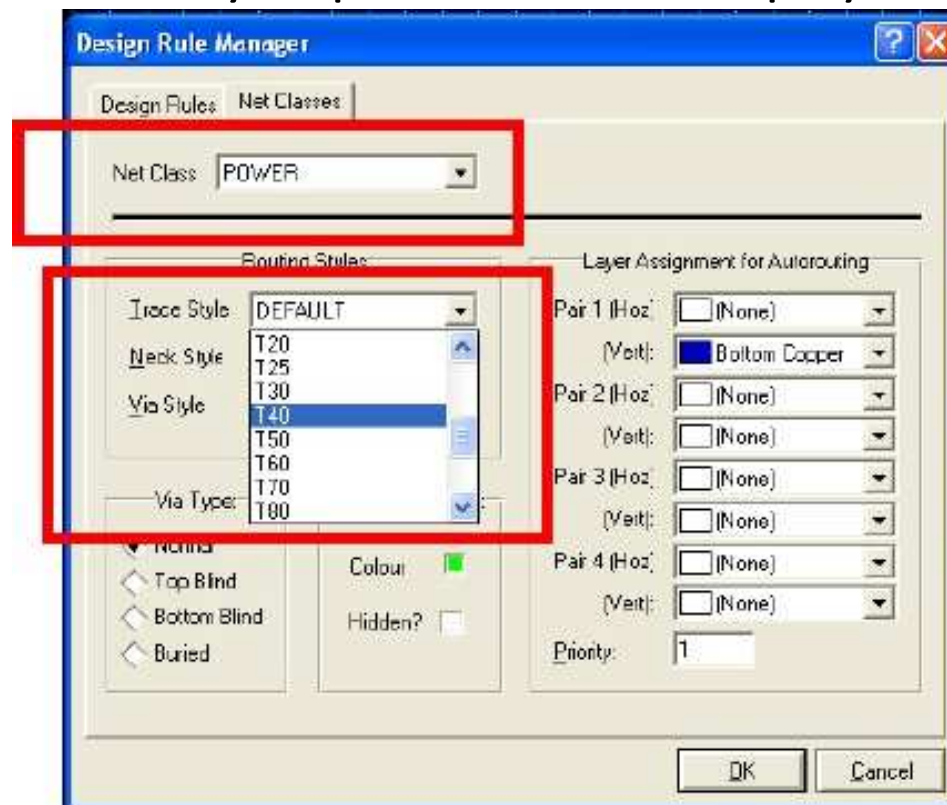
8. If you are using a single layer PCB:

- Before routing, you need to set the layer assignment for bottom copper only.
- Click Tools>Design Rule Manager>Net Classes



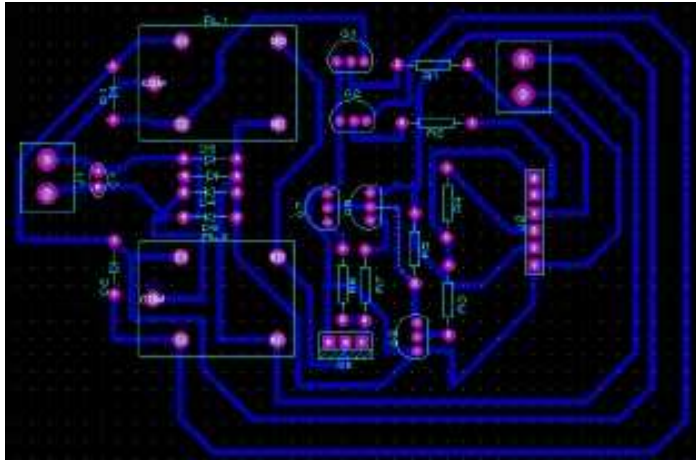
9. If you want to adjust the trace width:

- Click Tools>Design Rule Manager. Click on Net Classes tab. Select your desired trace style on Trace Style Drop down menu. You need to specify the trace style for both Net Class.

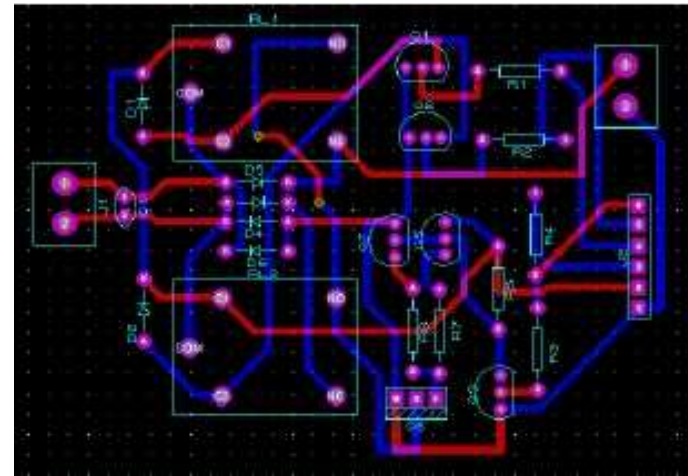




## 10. Next, begin autoroute



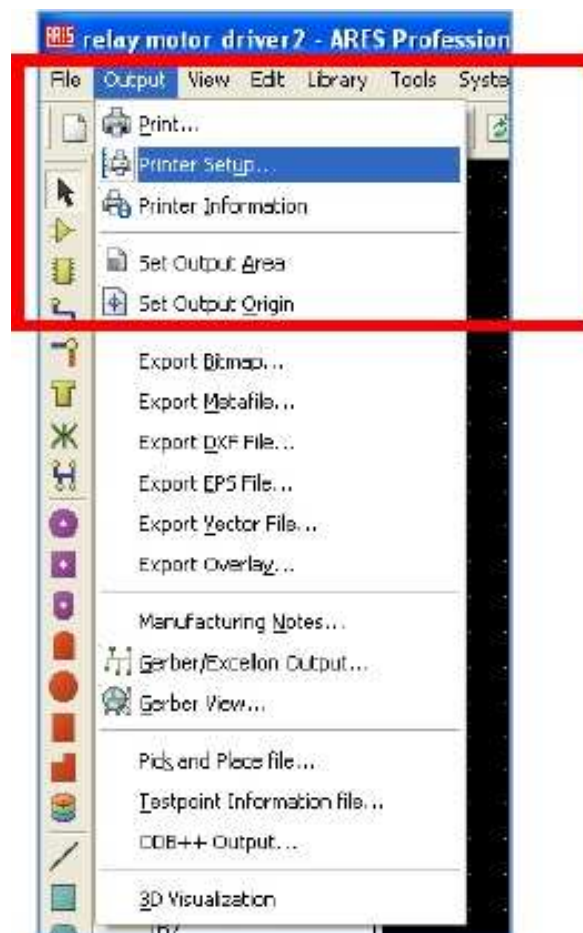
Result if you choose single layer

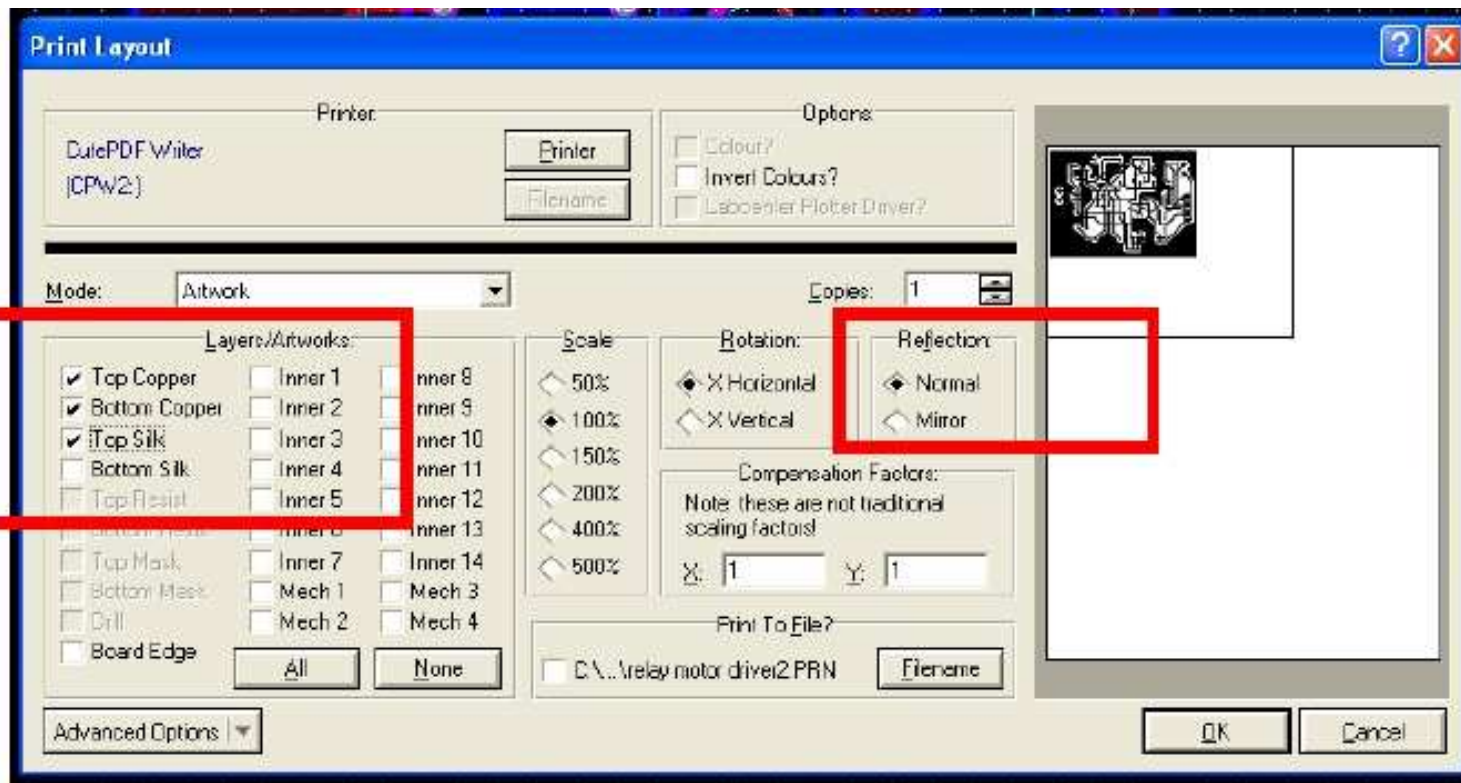


Result if you choose double layer

## 11. Printing the layout :

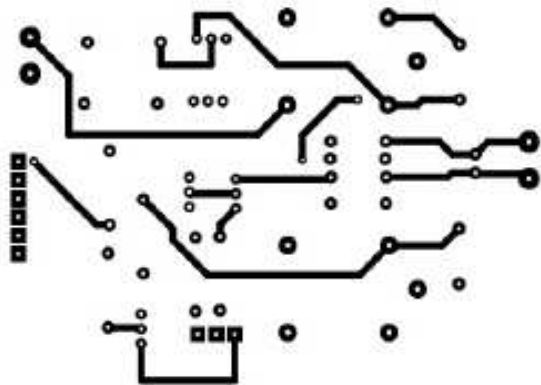
- Click Output > Print





Select the layer that you want to print and reflection type

Copper top:



Copper bottom:

