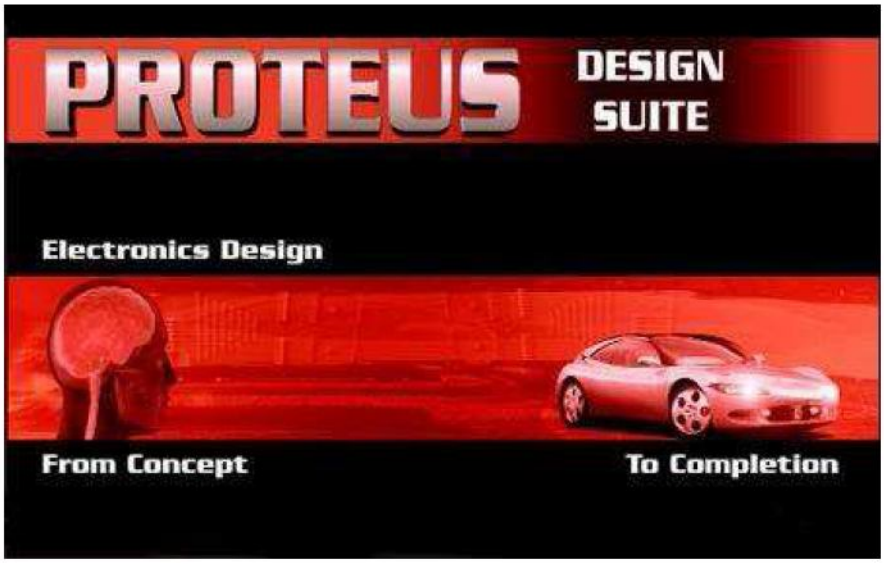


PROTEUS PROFESSIONAL DESIGN

Aminurrashid Noordin

Mohd Hanif Che Hasan

Mohd Razali Muhamad Sapiee

Sulaiman Sabikan

PCB DESIGN WITH PROTEUS PROFESSIONAL 7

Topic 1
Introduction to Proteus Professional

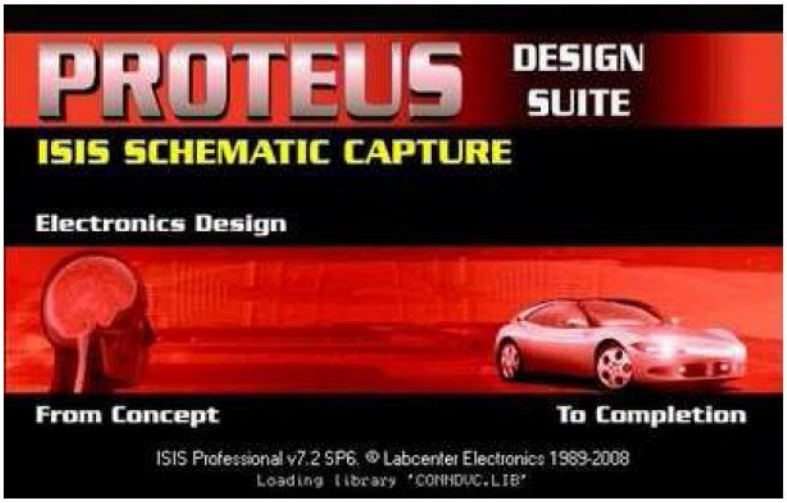
Proteus Professional design combines the ISIS schematic capture and ARES PCB layout programs to provide a powerful, integrated and easy to use tools suite for education and professional PCB Design.

As a professional PCB Design Software with integrated shape based auto router, it provides features such as fully featured schematic capture, highly configurable design rules, interactive SPICE circuit simulator, extensive support for power planes, industry standard CADCAM & ODB++ output, and integrated 3D viewer.

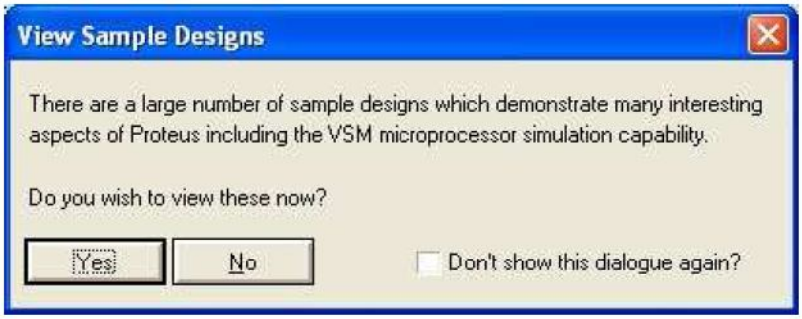
Topic 2
Designing Circuit using ISIS Professional

Following are the steps to be followed to start designing a circuit:

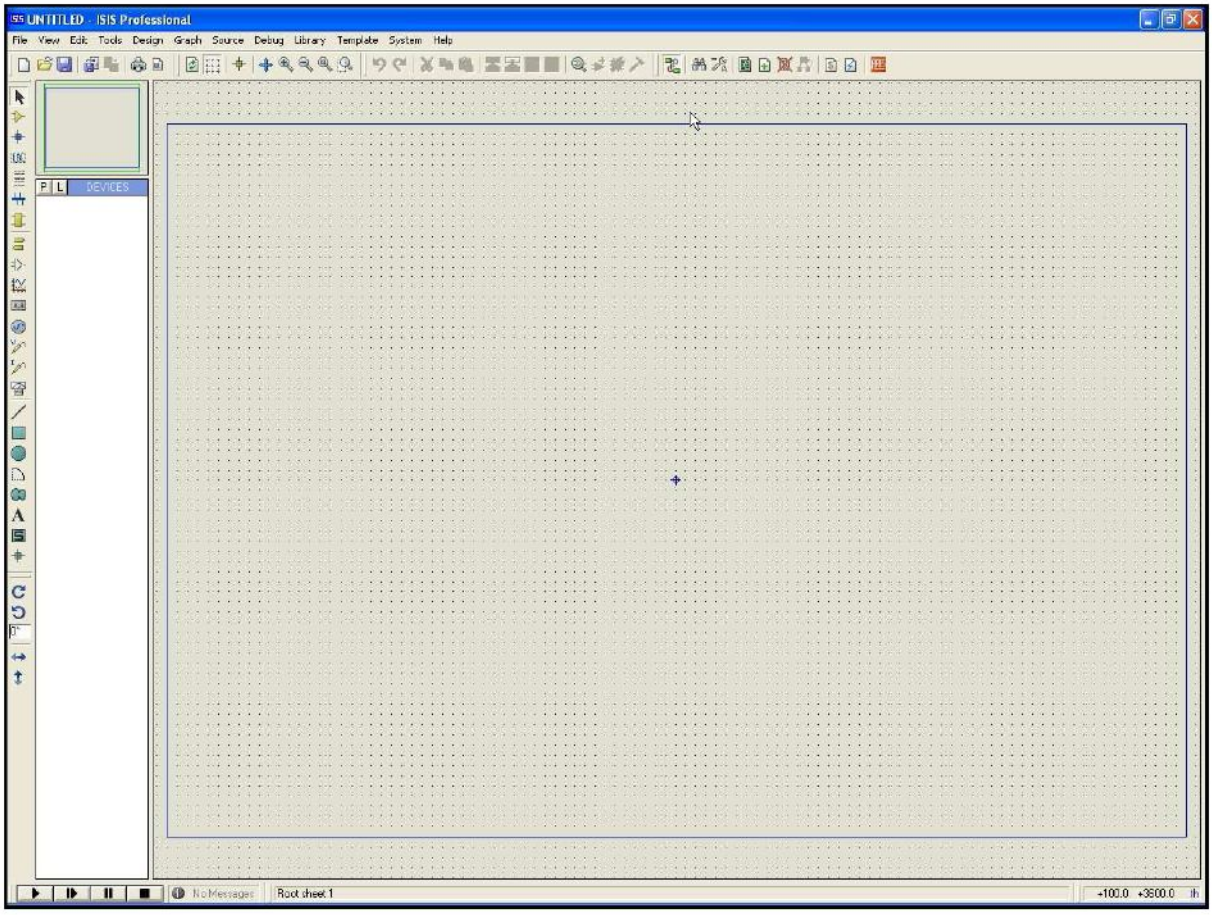
1. Run ISIS professional program by clicking ISIS Professional icon on desktop or go to start windows > all programs > Proteus Professional > ISIS Professional. A splash screen will appear.



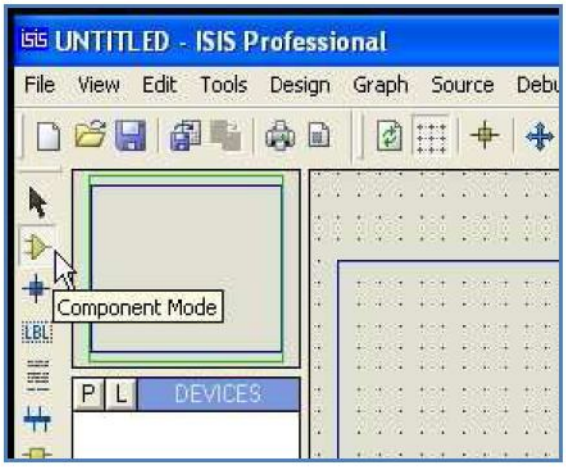
2. A window appears once ISIS professional is running, asking user whether to view example of functional circuits or not. Since we going to design a new circuit then choose No.



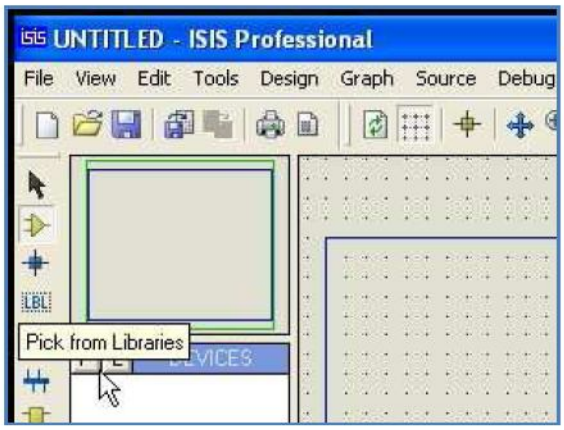
3. Next, a workspace with interface buttons for designing circuit will appear as shown in figure below. Note that there is a blue rectangular line in the workspace; make sure the whole circuit is designed inside this rectangular space.



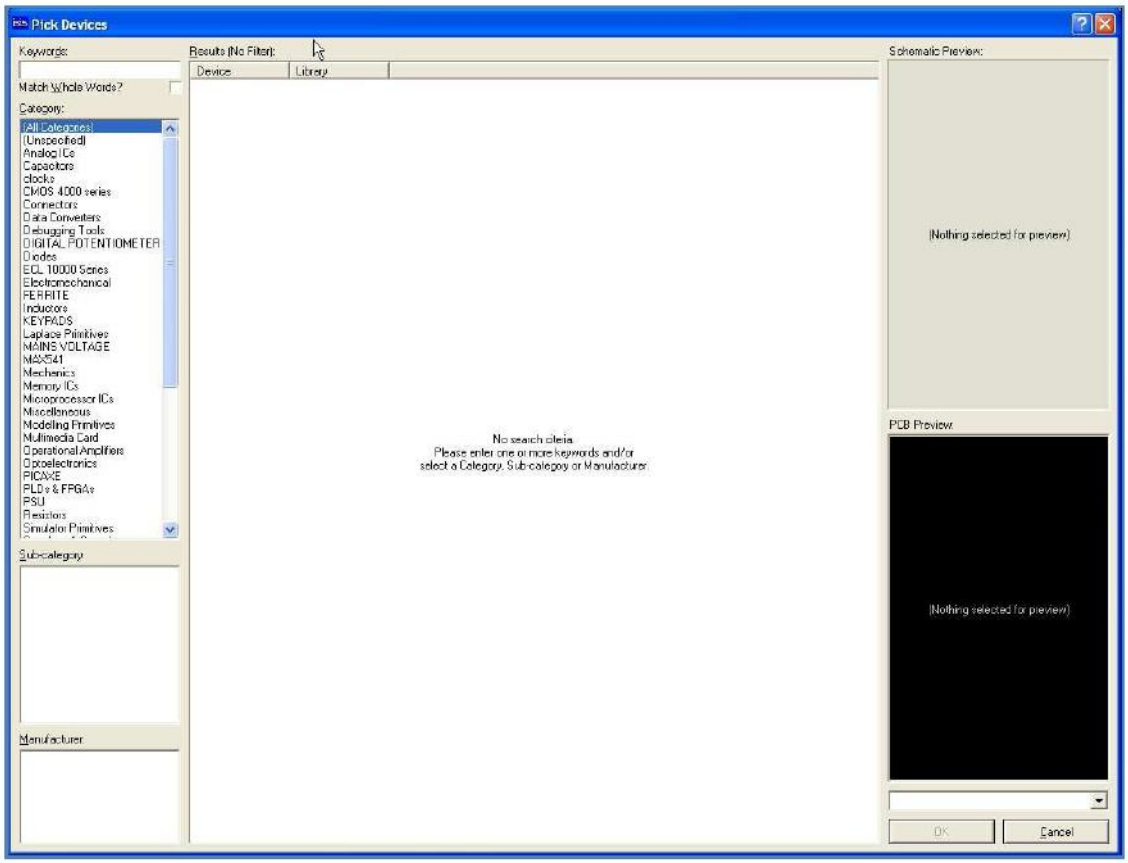
4. The next step is to select all components needed. To choose a component, click the icon button for component mode.



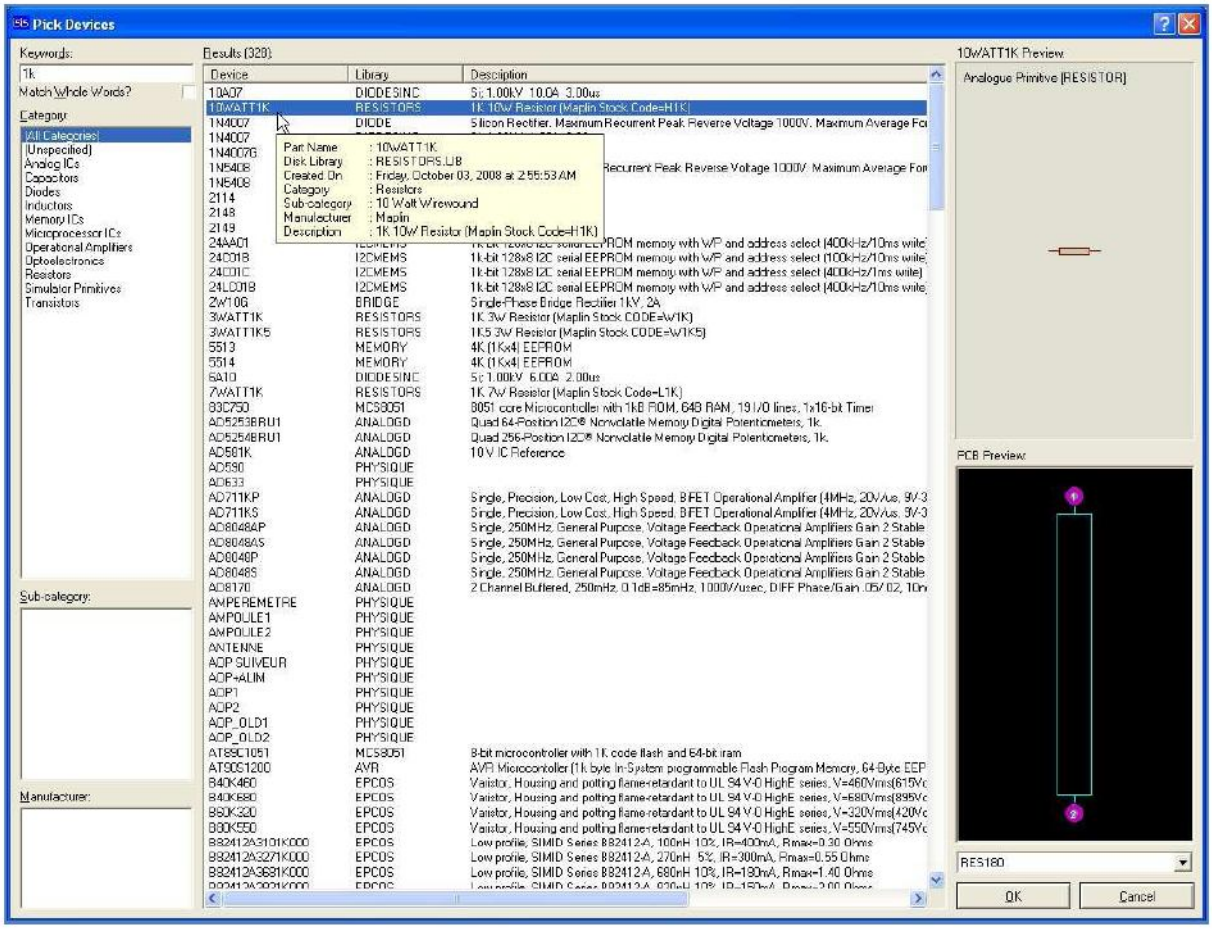
Then click on P to pick from library.



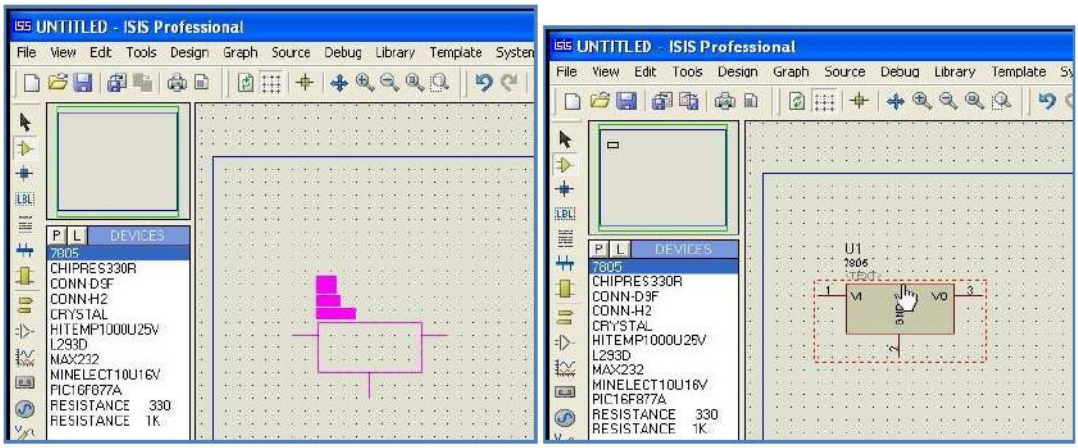
A window will open where we can browse and choose components from the library browser.



5. In this component library browser, type any character of component and it will appear automatically. For example; to find 1k ohm resistor, type 1k in keyword box and all related components will appear. Then double click on resistor to select. The selected component will appear in devices box. After all components have been selected, click OK to close the component library browser. Make sure the components selected have PCB view otherwise we need to draw them manually.



6. To place the components, click on a selected component list in devices box then single click on the drawing area. The image of the selected component will appear. Then just click on any part of drawing area to place the component.

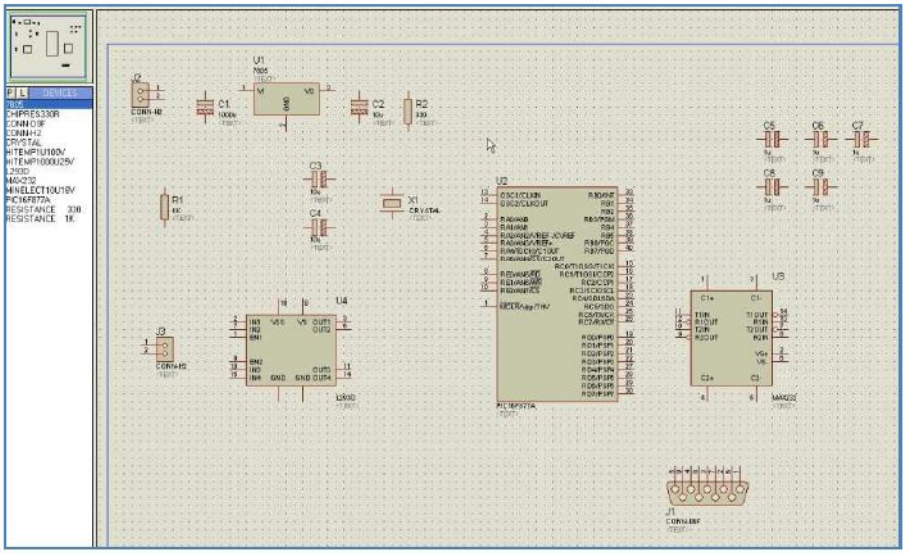


7. To move a placed component just place the cursor on the component until the move icon appear, then click hold and move the component anywhere in drawing area.

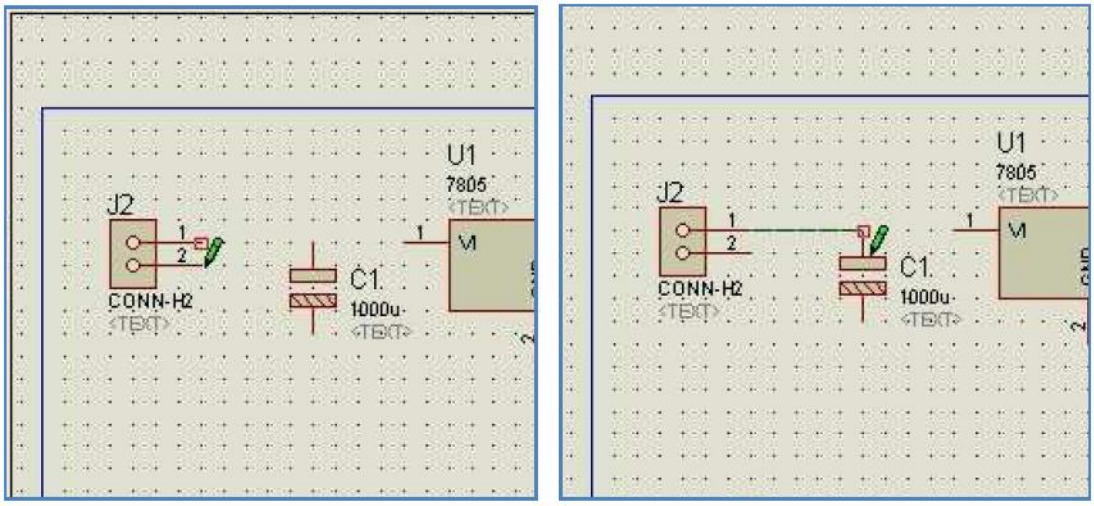
8. To change the component orientation, just right click on the component and chose the orientation you want.



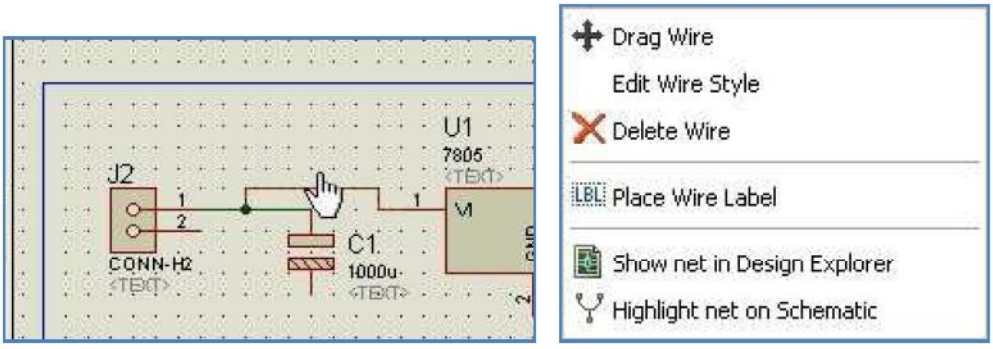
9. Now place all the components needed in drawing area with right quantities to design circuit.



10. To make wire connection, just place the cursor on component pin until red square appear, then click on the component pin and wire mode start. Click again on the pin of other component to join the wire. Join the wires to all components according to the circuit design.



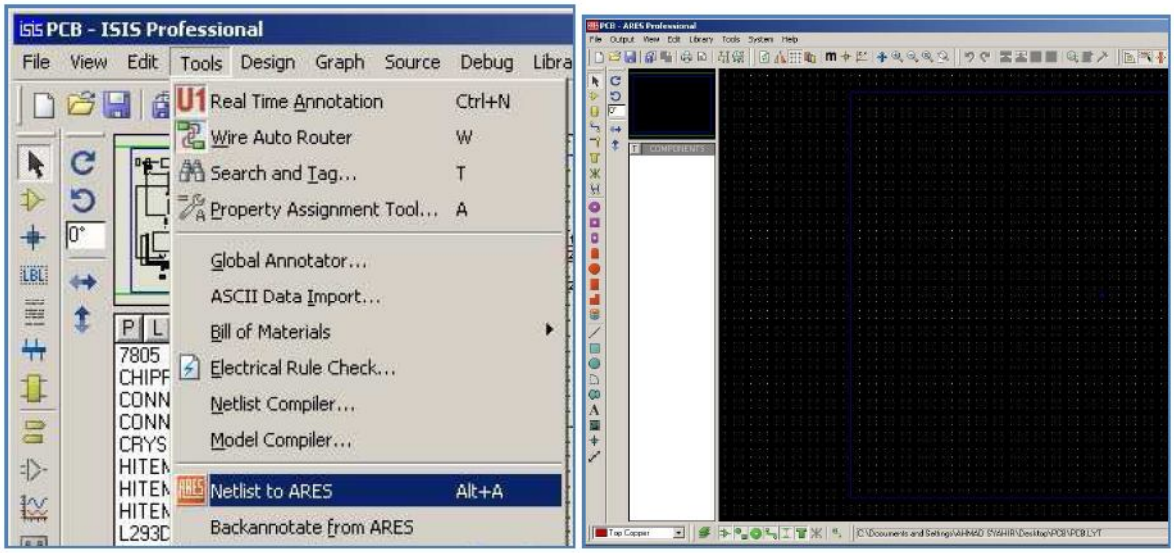
11. To remove the wrong connection made, right click on the wire connection and choose delete wire or just double right click on the wire.



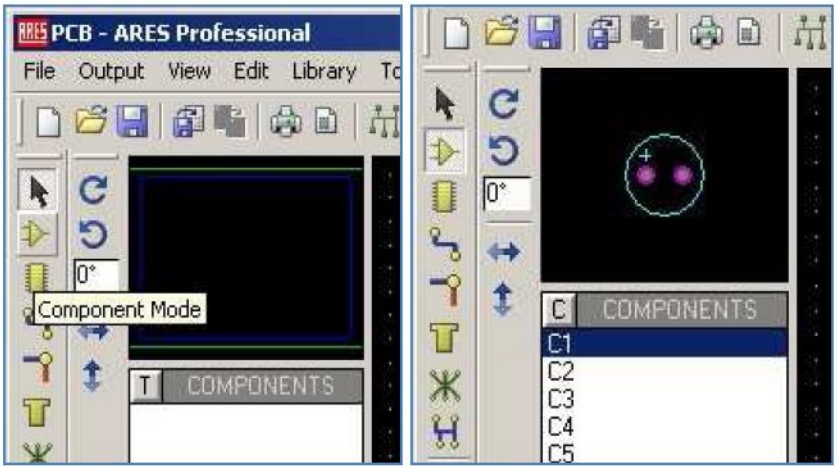
12. To edit the wire, just select the selection mode and select the wire. You can drag and drop like in Photoshop.



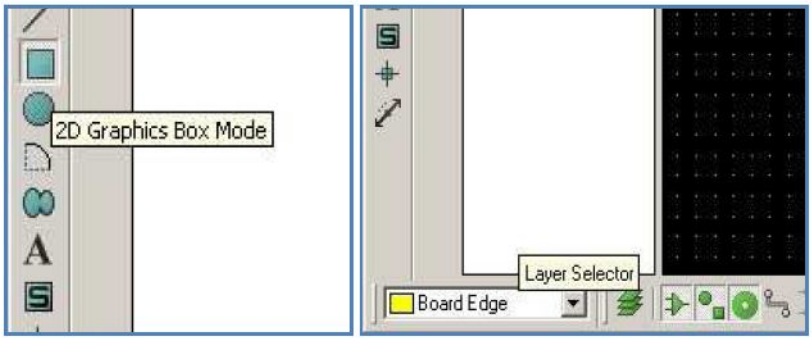
13. After done editing the design, you must save it. Now the ISIS design is ready for making PCB design. To making PCB design, select tool and then select Netlist to ARES and click. ARES professional window will soon after you click on it.



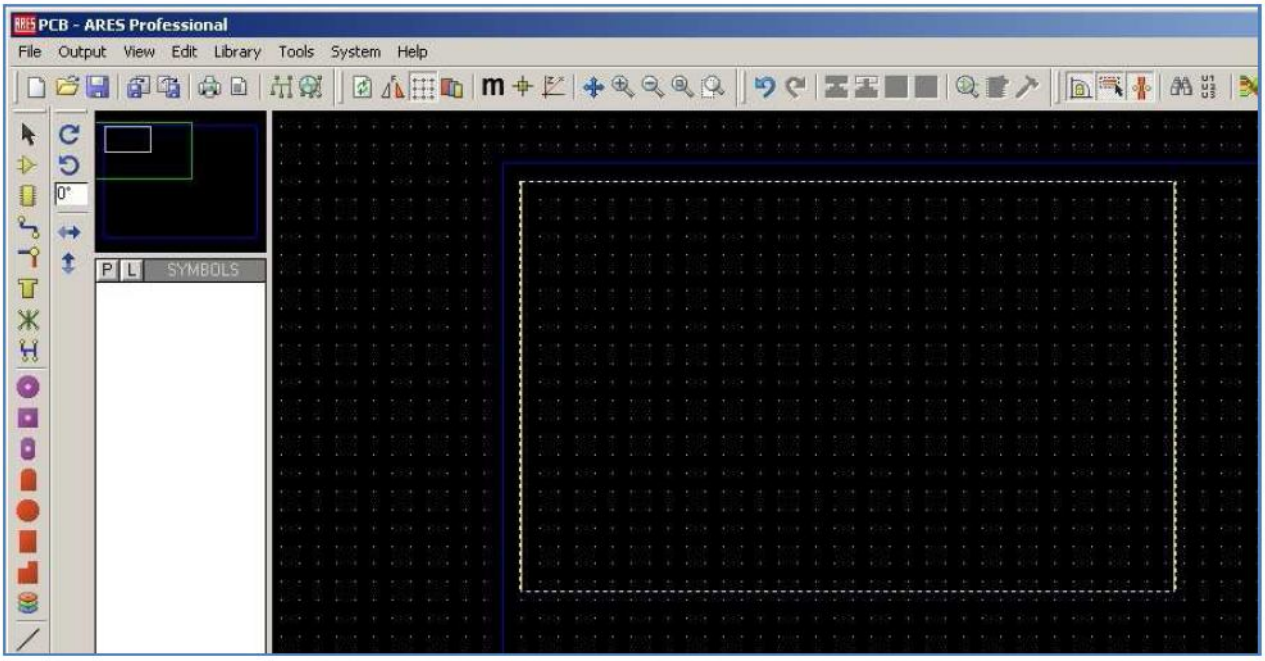
14. A blank workspace will appear in ARES professional. To import all components from ISIS design, select component mode on ARES professional. All components used in ISIS design will be listed in components box. A listed component will disappear from components box once you place it in the workspace.



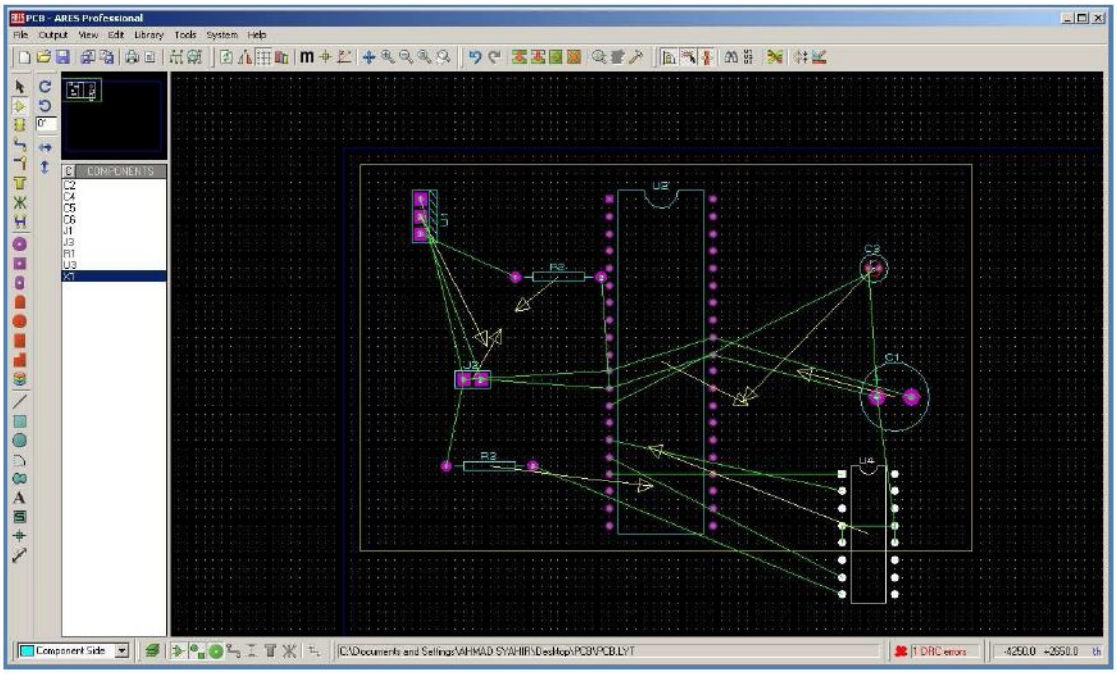
15. To begin with, first you must set the border for PCB design. To do so select 2D graphic board mode and go to layer selector and choose board edge.



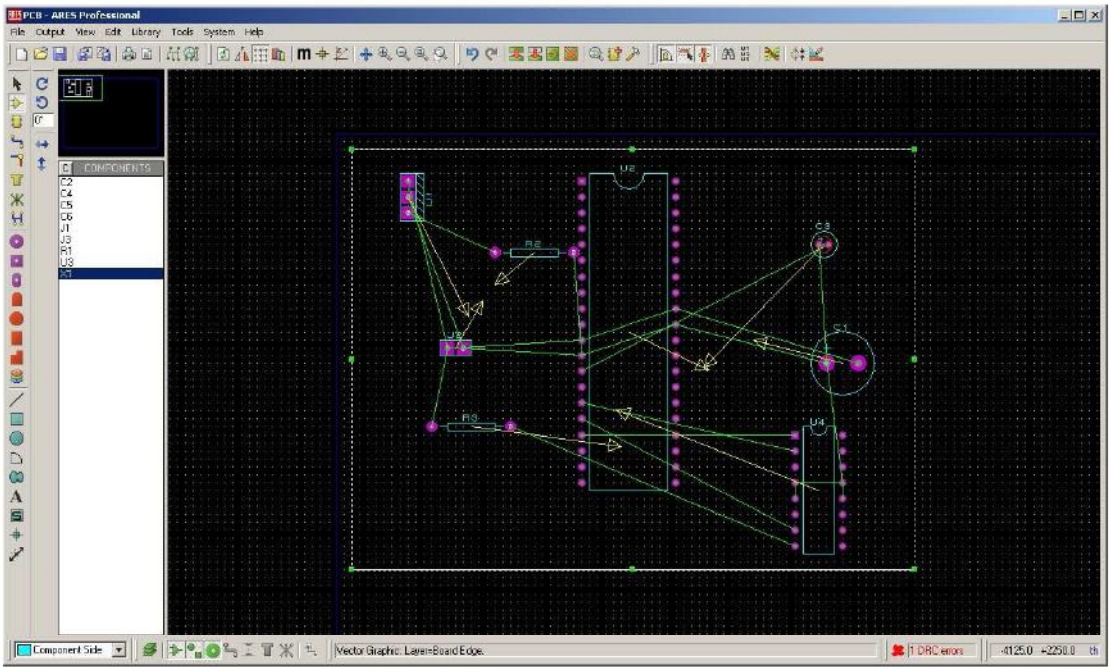
16. Then draw the border inside the ARES professional workspace. Do not worry about the size of the board you draw because it can be adjusted afterward.



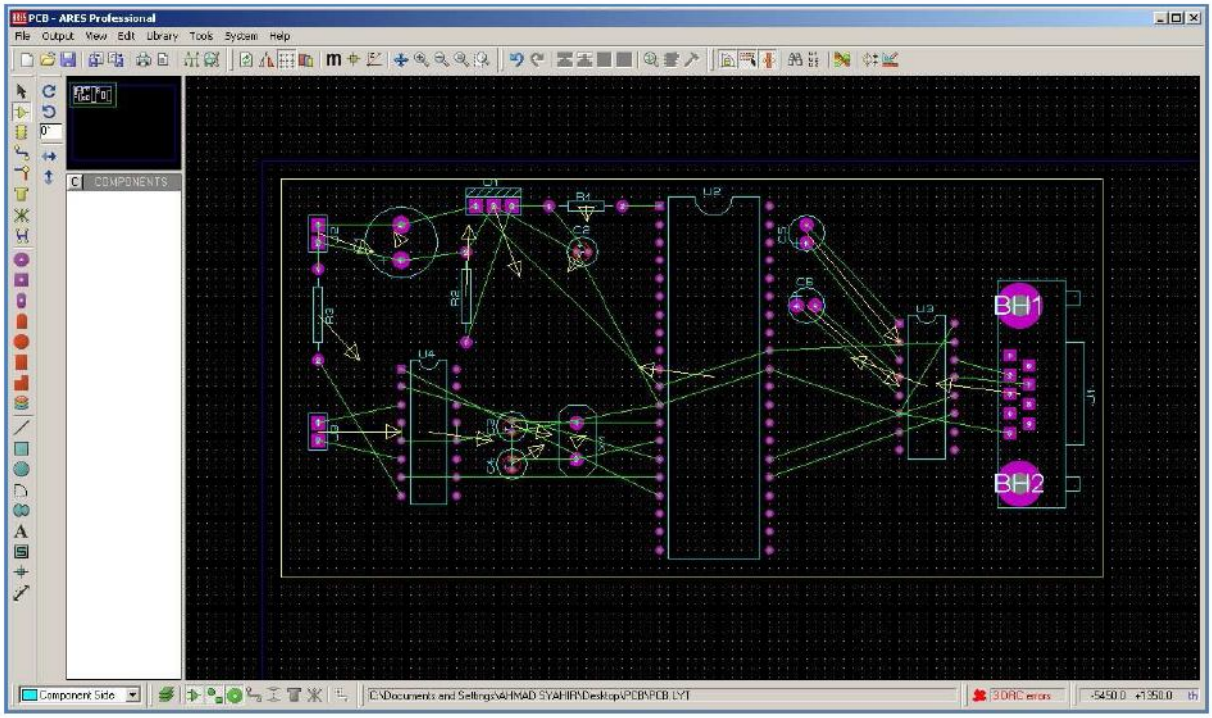
17. Now select component mode again, then select the component and place it in the board edge we have drawn earlier. To place, delete and edit position of components, the method is the same as in ISIS. Green lines show the connection between components that has been designed in ISIS.



18. If the drawn board border is not big enough, you can edit the size just by clicking on the border line until green rectangles appear on the border line. Click, hold, drag and release the line. To exit border editing mode just click on the workspace.



19. After placing all components, try to design a more suitable design by repositioning the components and minimizing the space.

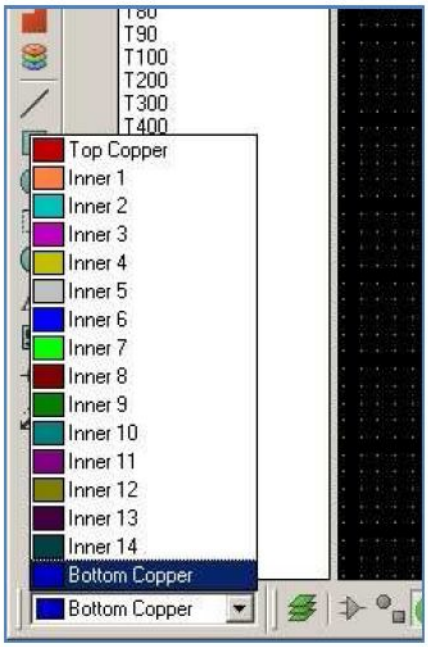


20. Now we have to route the design. Select track mode, then you can place the copper connection in the design. The width of copper traces connection can be edited by selecting the traces type in traces box by clicking E(edit) or right click mouse on selected trace type and choose edit. A editing windows will appear. To create a new type of trace just click C(create). A creating window will appear.

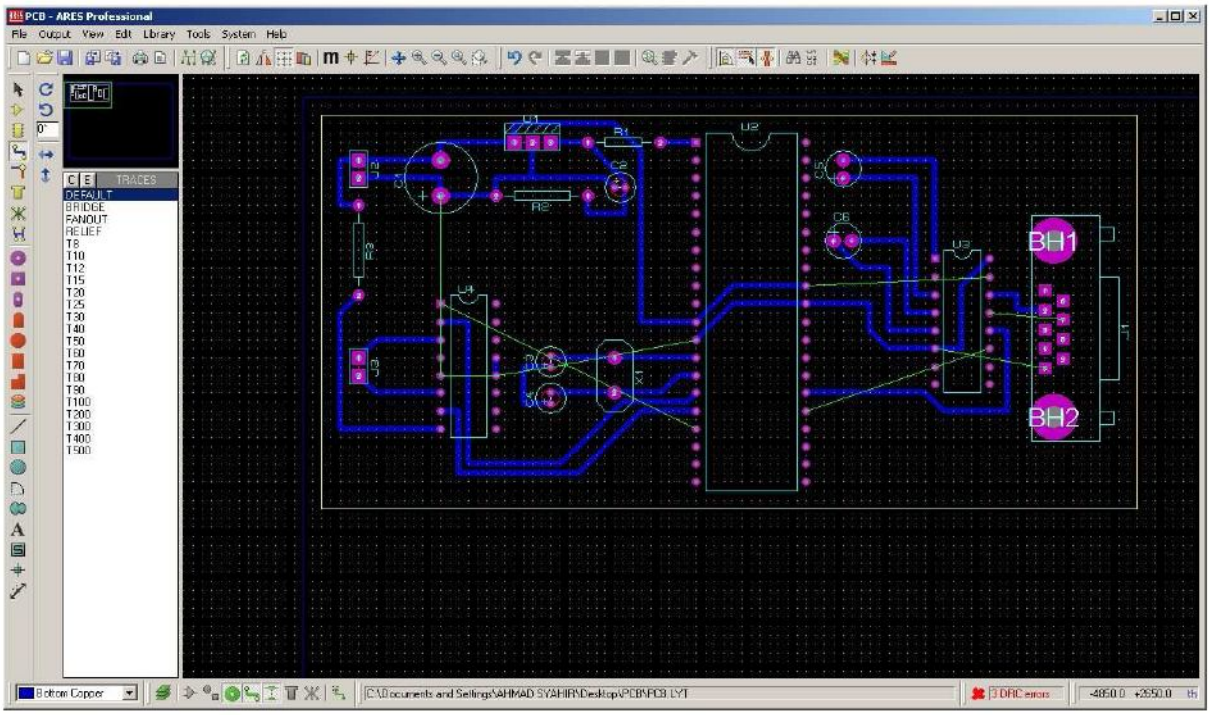




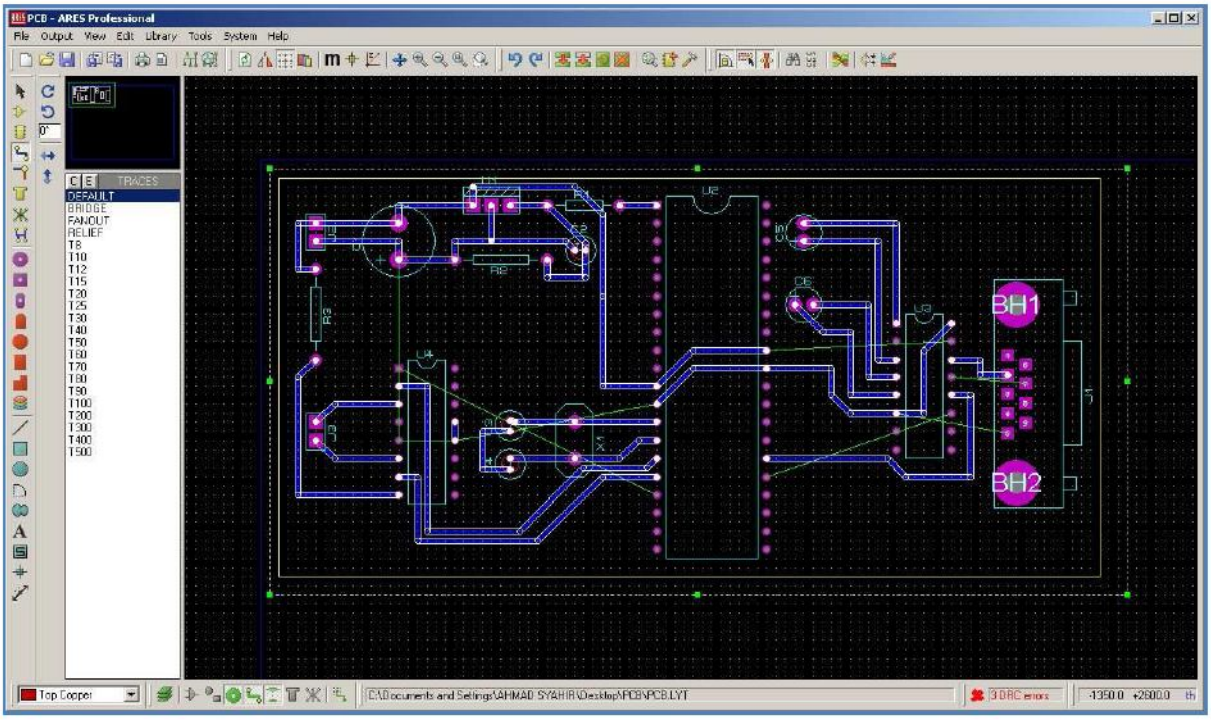
21. Make sure to change the traces selector to bottom copper for single layer PCB board.

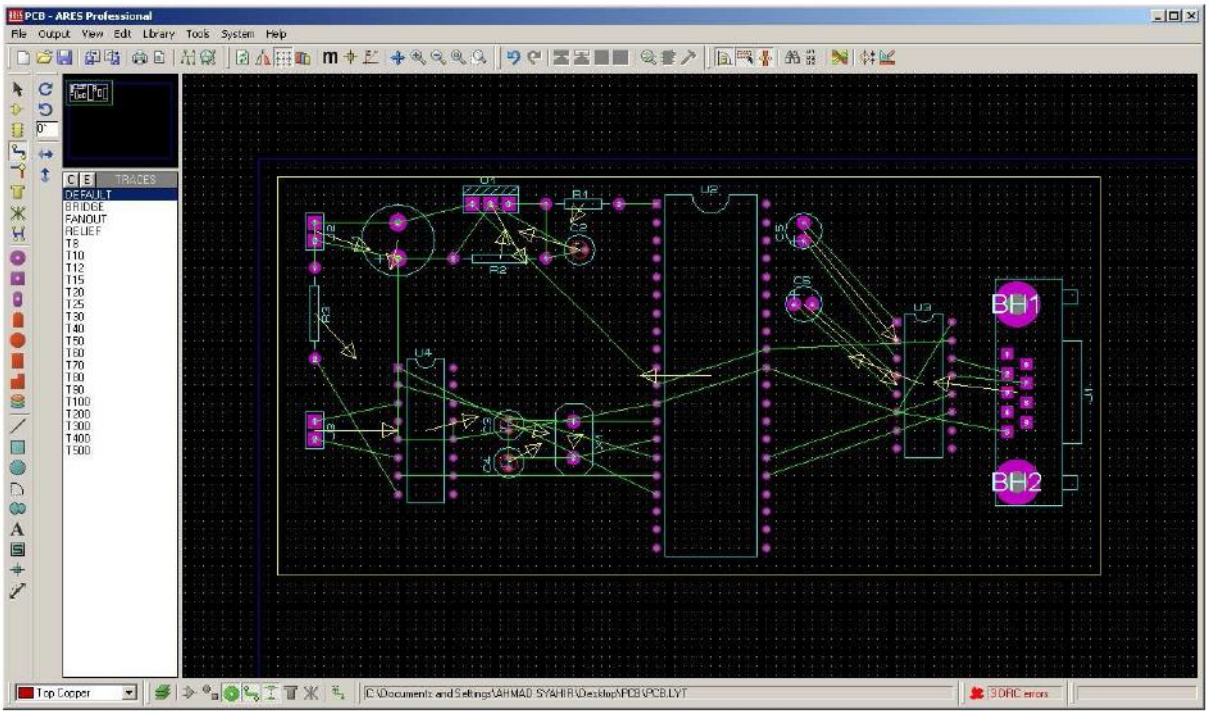


22. Select the traces type you want and make the copper connection route. The method is similar to wire connection in ISIS. Make sure the copper connections do not intercept each others. Use the green line as a guide to make copper connection.

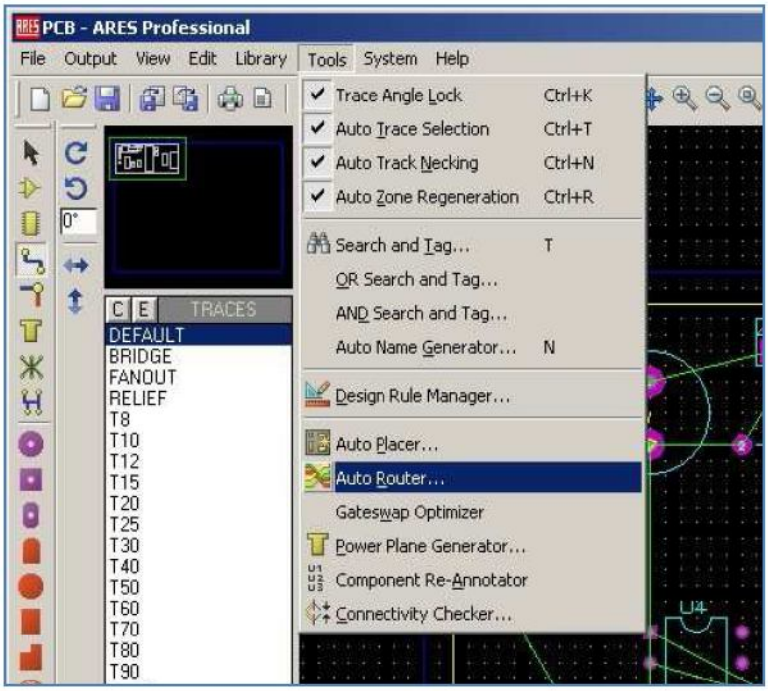


23. If you want to delete the entire route just click outside the board border and select all design, then press delete button on keyboard.

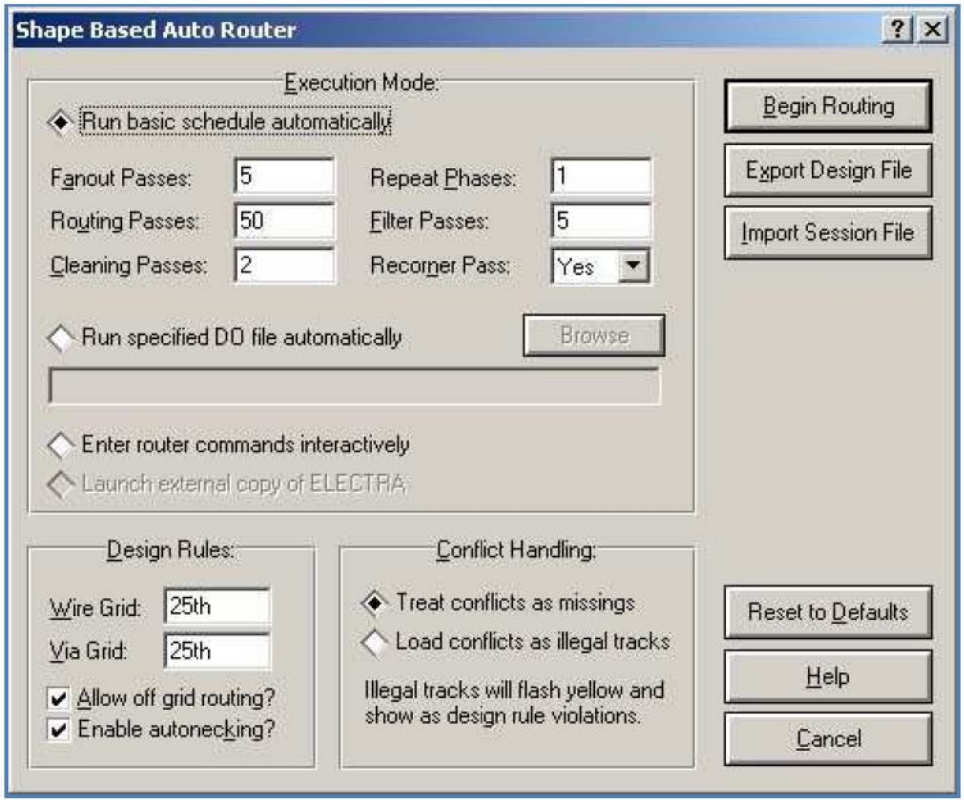




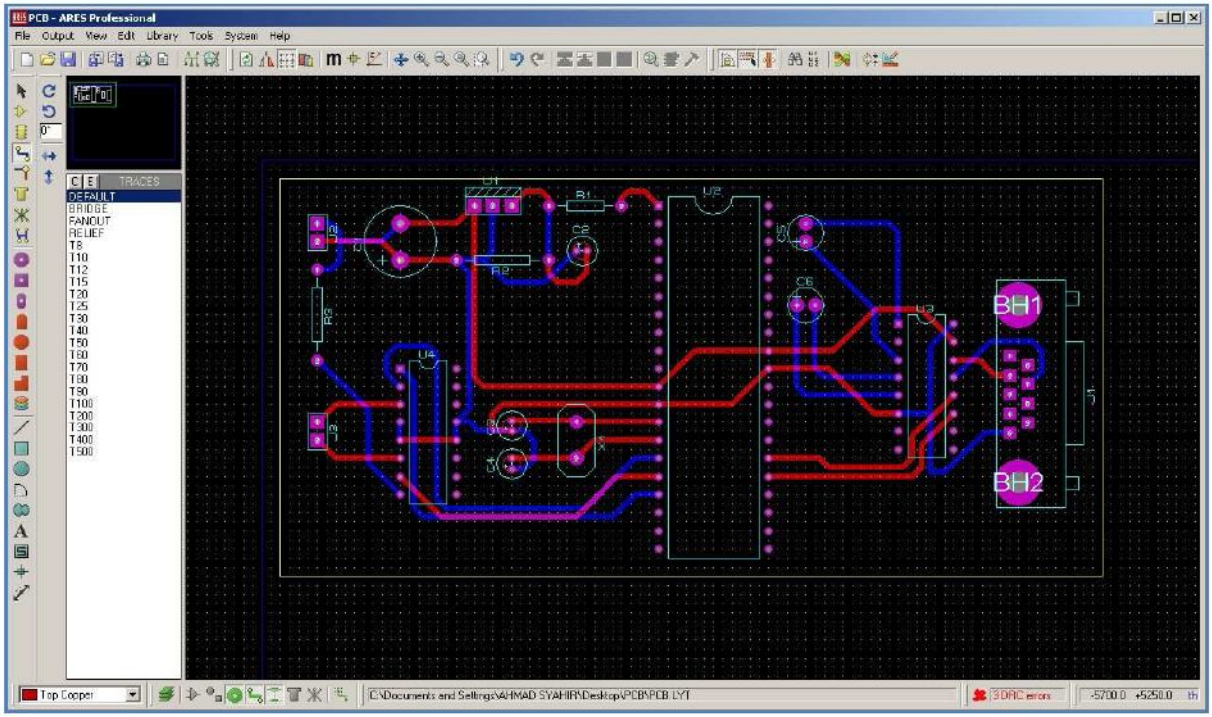
24. After deleting the entire route, you can begin to reroute the copper connection again. There is auto routing function to automatically route all the copper connections. Select Tools on menu bar and click auto router.



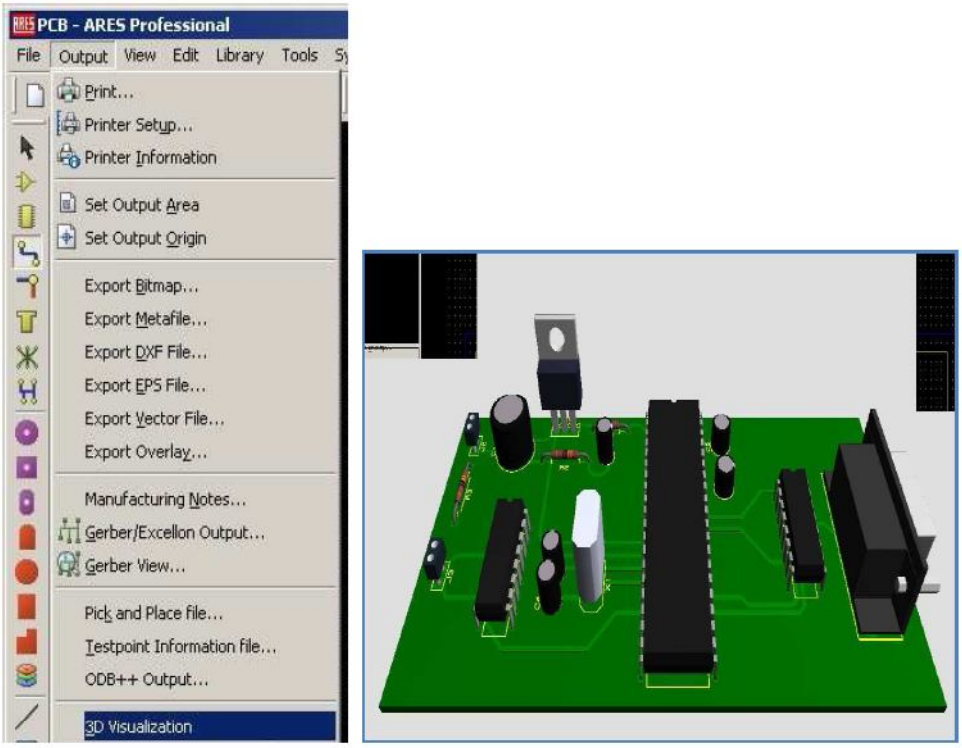
25. Shape based auto router window will appear and you can set the setting for the copper traces and click begin routing.



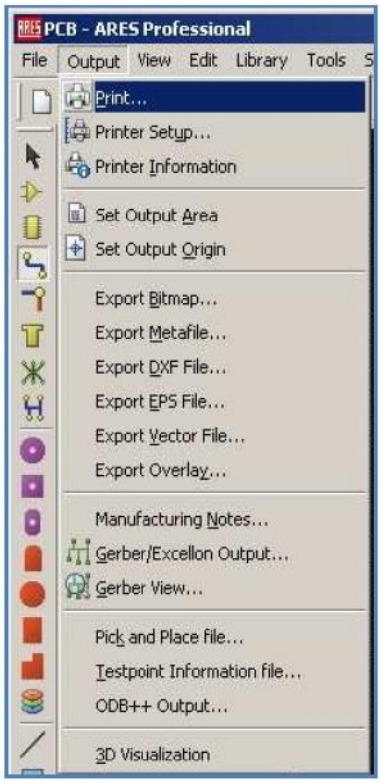
26. The copper traces using auto router is most likely for use in double layer PCB design, therefore auto router is not recommended for routing the single layer PCB design.

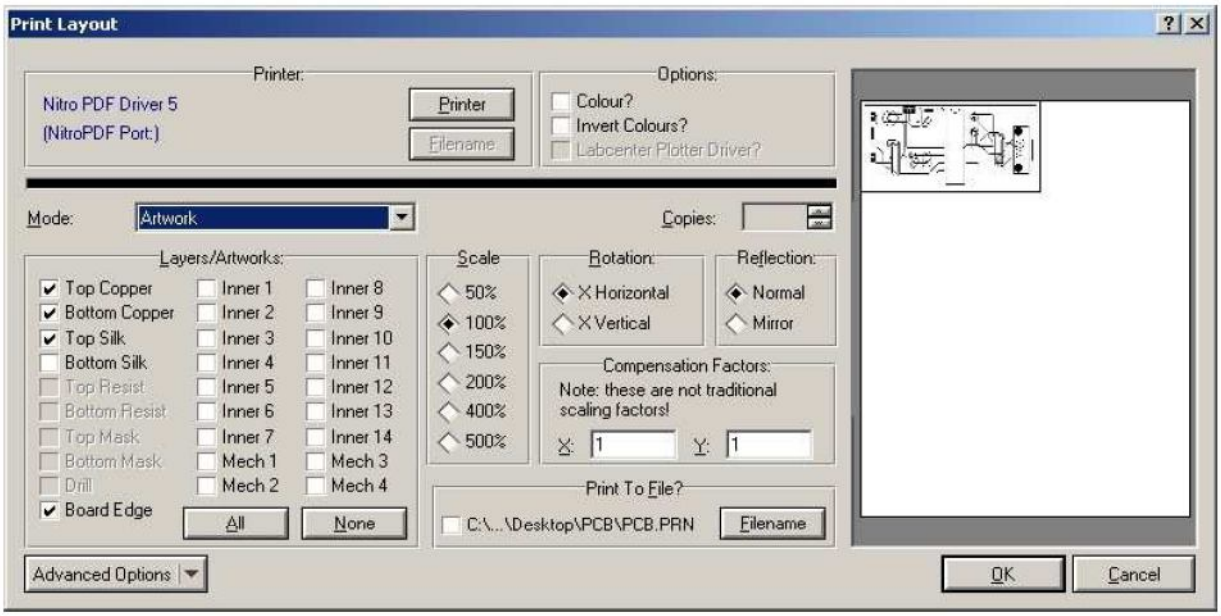


27. To view how your PCB will look like, select Output on menu bar and click 3D visualization. Just press Esc on keyboard to exit 3D visualization mode.



28. To print your PCB select Output and click Print. Print Layout window will appear. Then setup the print setting based on the PCB printer.





29. Now your PCB design is ready to be transferred to PCB board by way of chemical etching. GOOD LUCK!