

I
S
I
S



Starting with proteus professional

Introduction

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 CAD/CAM and OD++ output , and integrated 3D viewer .

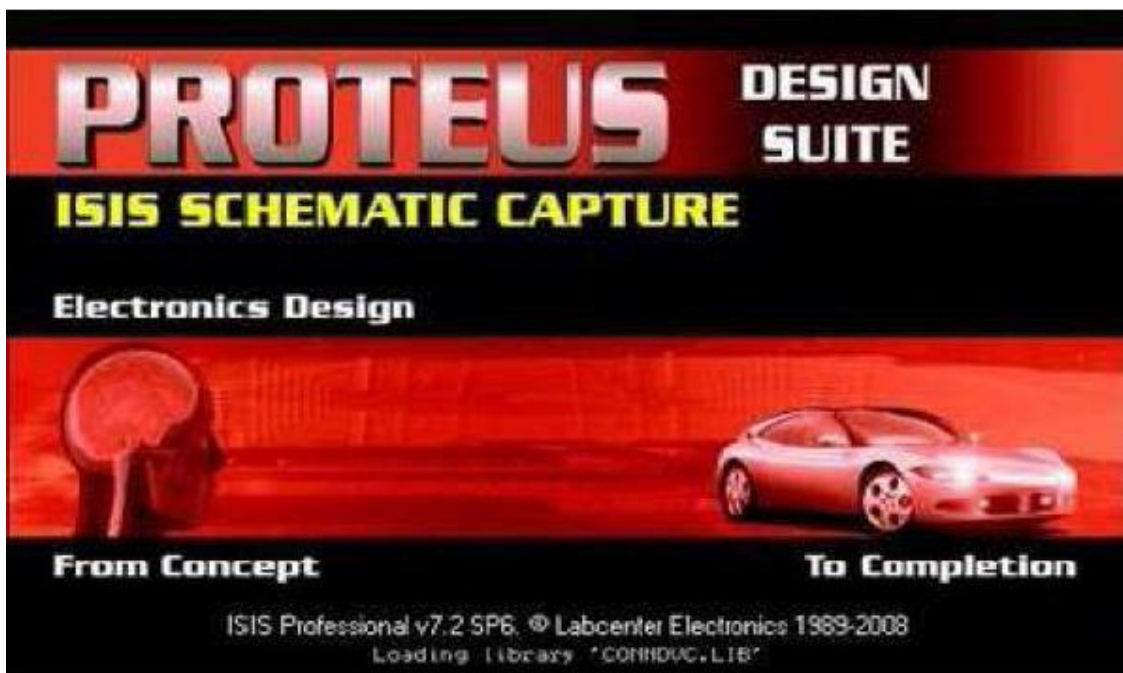
We will use ISIS for simulating PIC response, it has many variety modeling libraries, and its powerful concentrates in MCUs and MPUs modeling, along with wide range of supporting chips such that AVR MCU series, 8051 MCU series, Basic stamp, HCL1 MCU series, ARM CPU, Z80, Motorola 68K CPU, and most PIC's families, also it has a debugger, registers contents viewer and many other features.

Designing circuit using ISIS

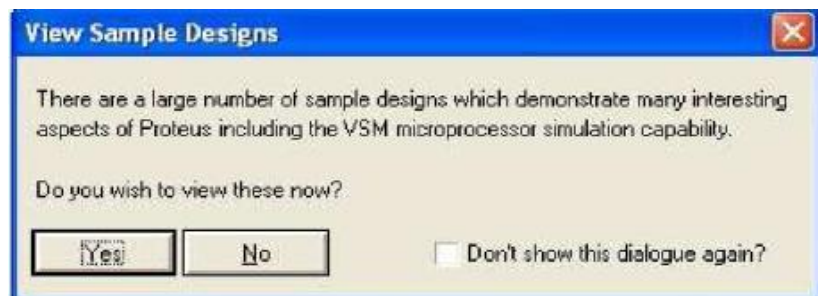
To be able to start the simulation you must build your schematic circuit on **ISIS professional**.

The following steps are used to design your circuit:

- ✚ 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.



- ✚ 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**.



✚ 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.

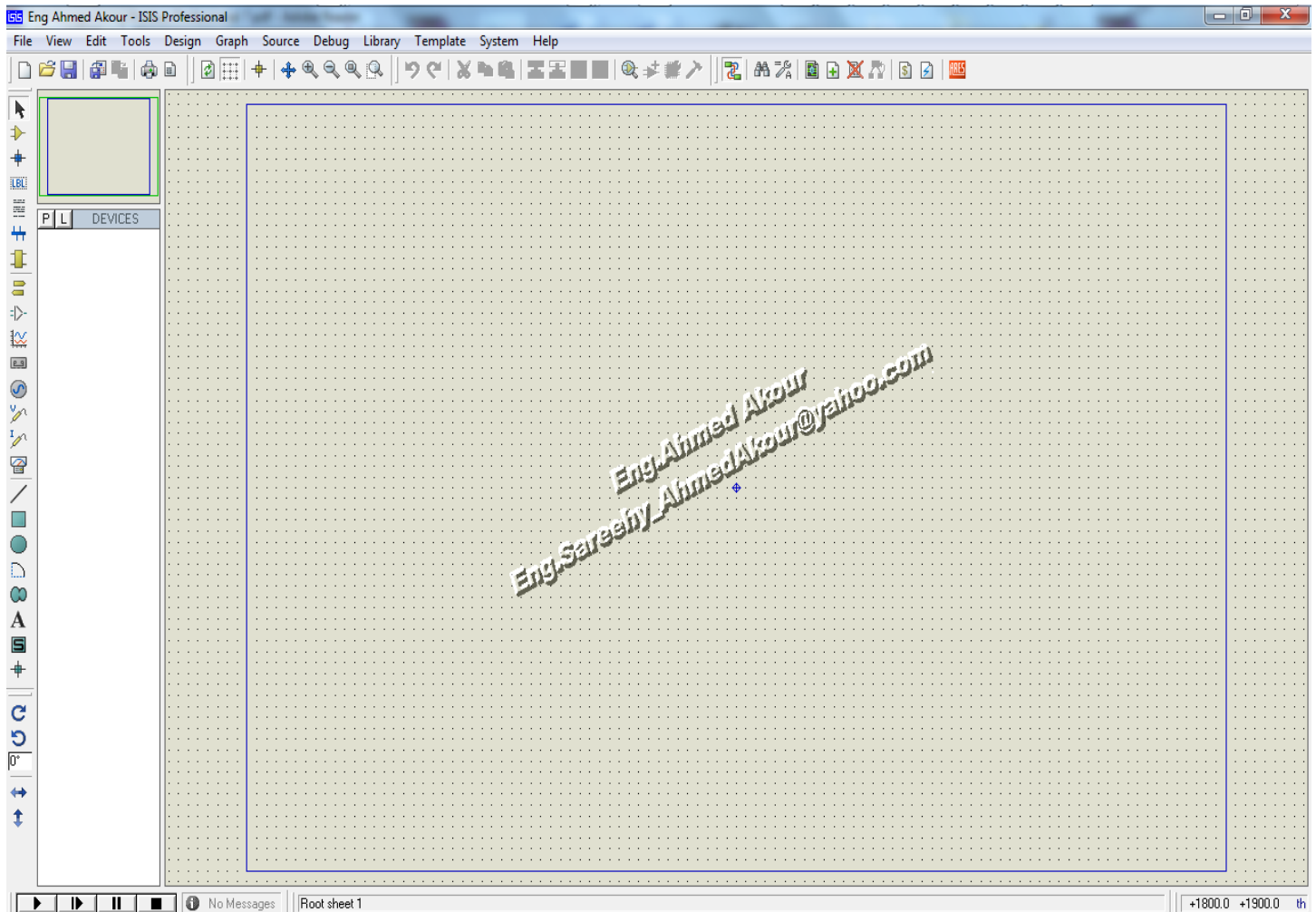
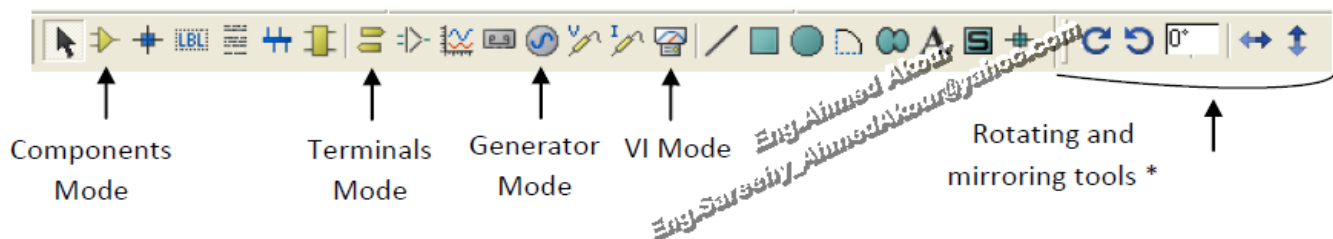






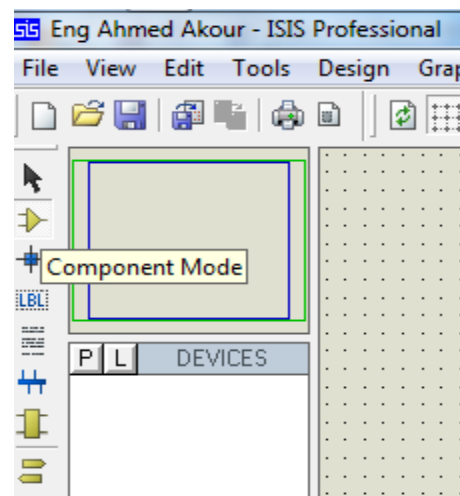
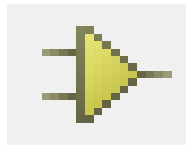
Figure below shows the design tool bar.



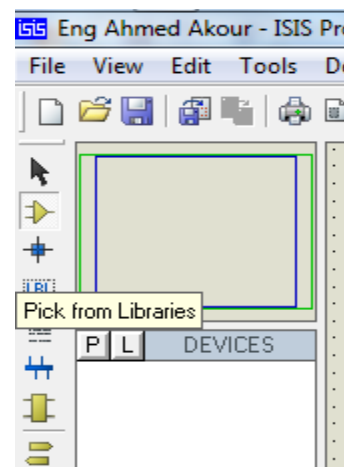
Tool's name	Tool's functions (The most important).
 Components Mode	Import MCUs, CPUs, Resistors, Sensors, Motors, etc...
 Terminals Mode	Power (V_{cc}), Ground (GND).
 Generator Mode	Sinusoidal, Square, External Audio file, Pulse, etc....
 VI Mode	Oscilloscope, Virtual terminal, Voltmeter, Ammeter.

* The rotating and mirroring tools are used to change the direction for any component.

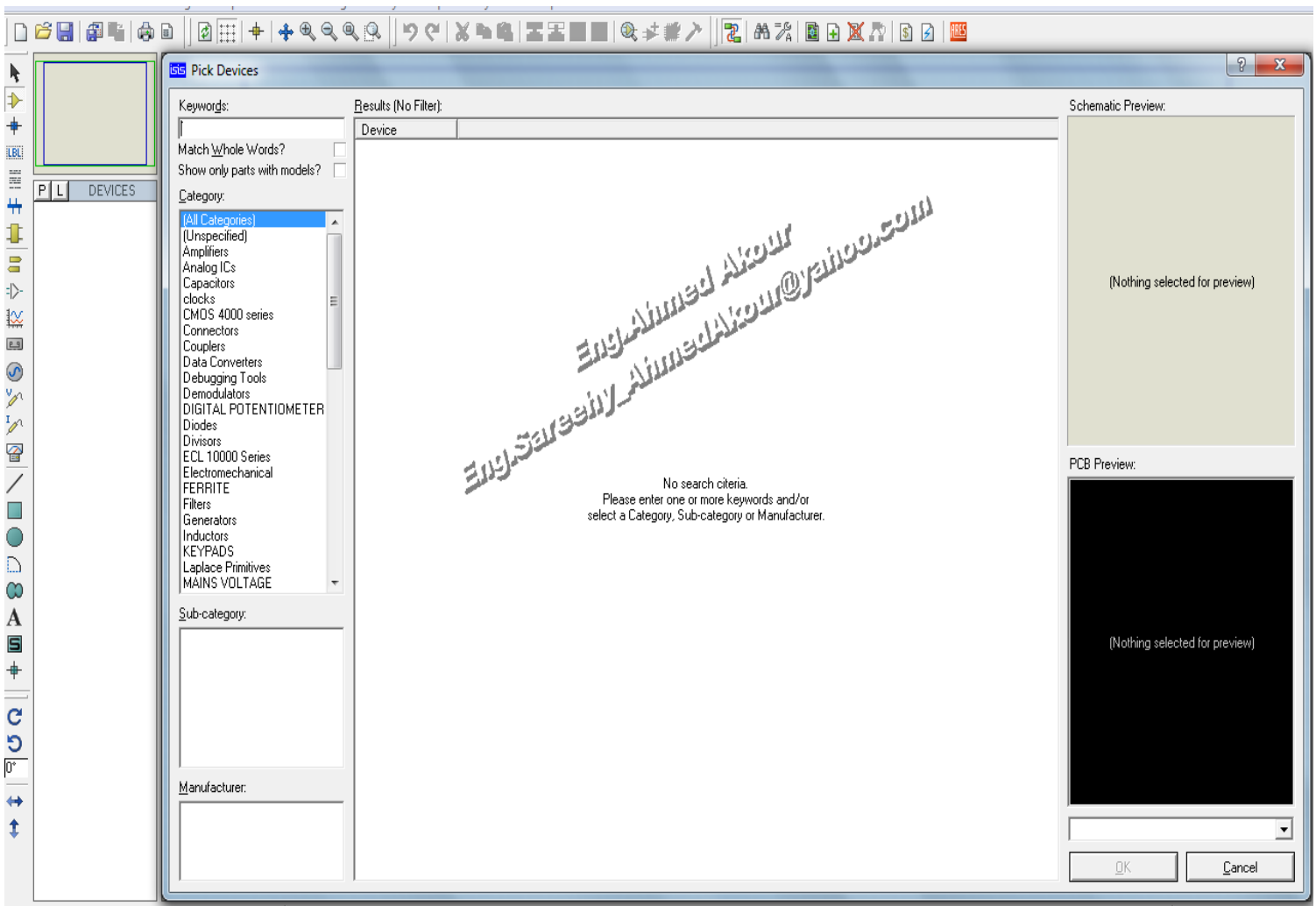
 **The next step is to select all components needed, to choose a component click the icon button for component mode.**



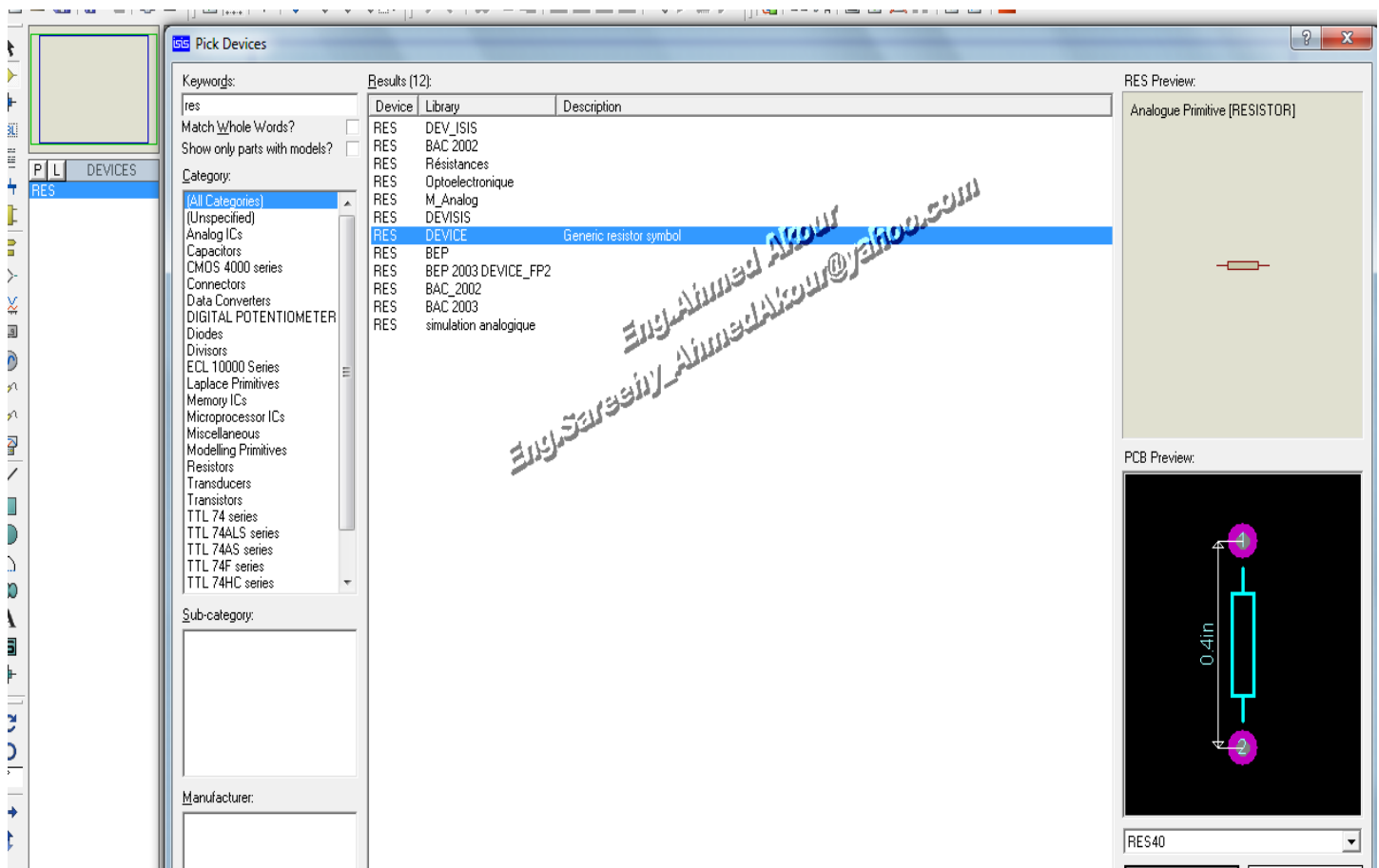
Then click on P to pick from library



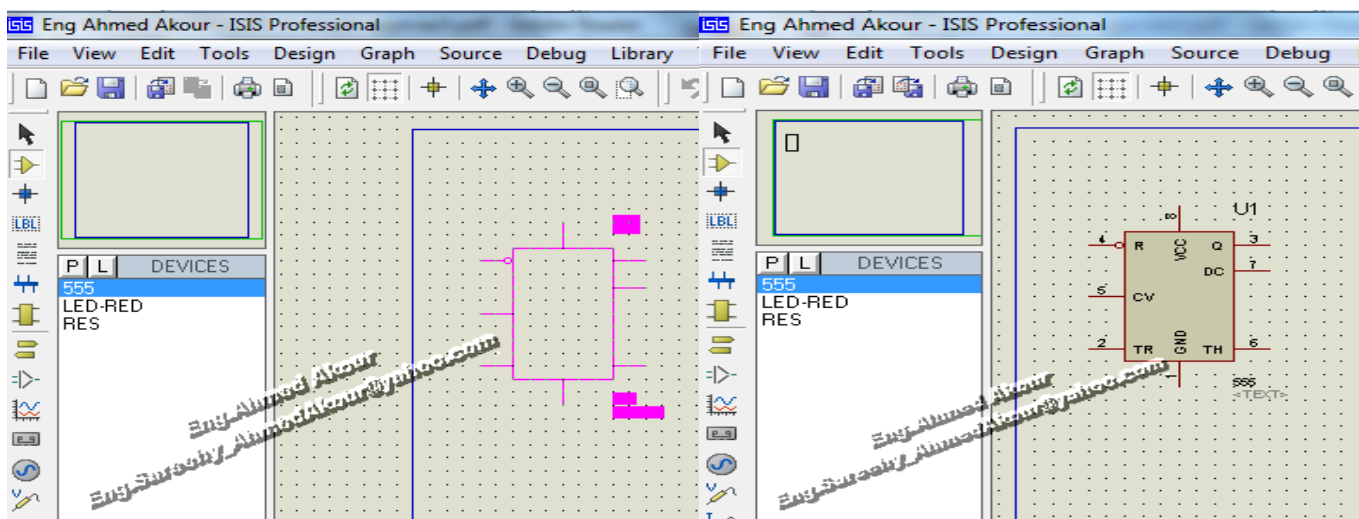
✚ **A window will open where we can brows and choose components from the library brows.**



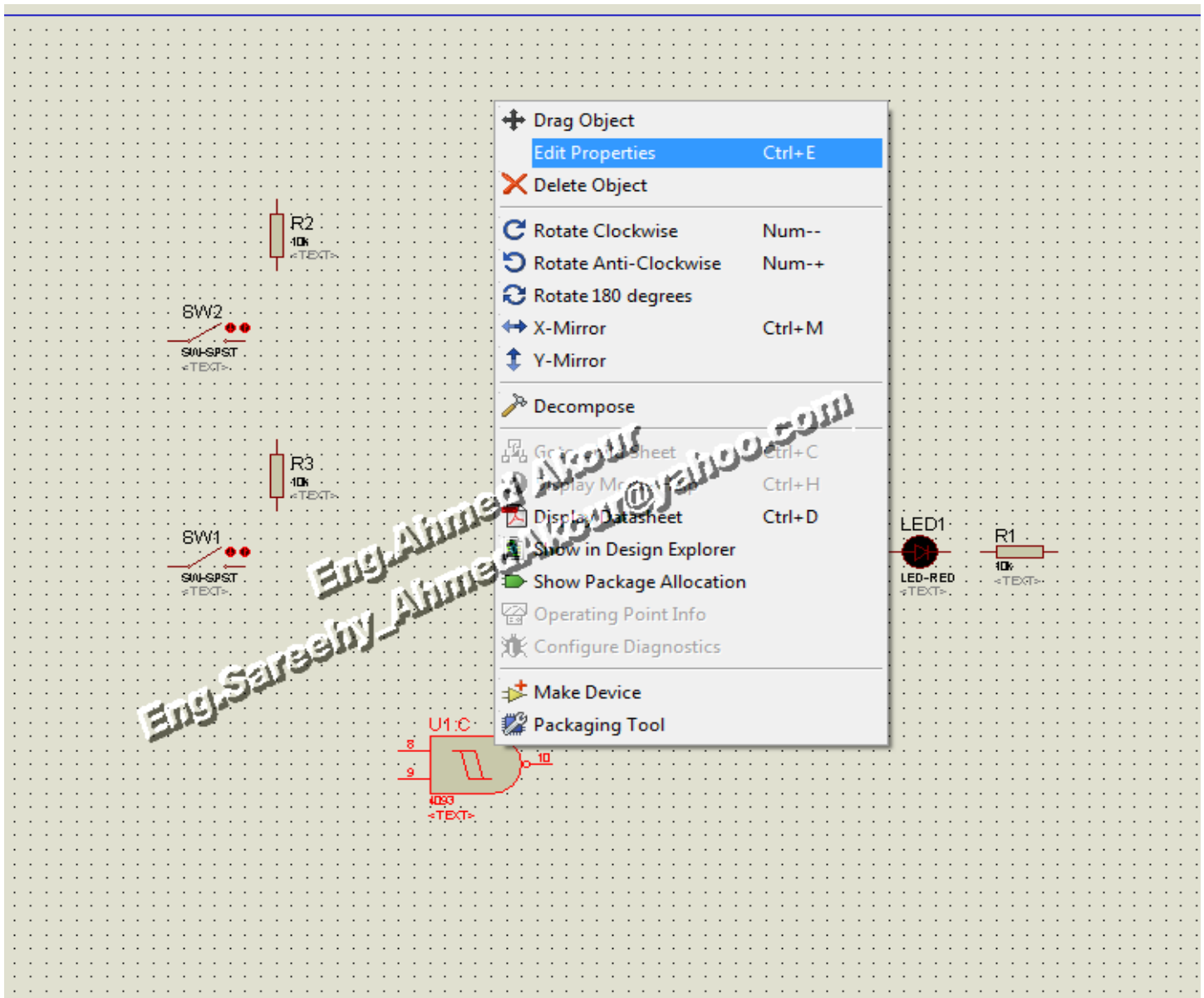
✚ **In this brows component library browser, type any character of component and it will appear automatically. Then double click on the component to select. The selected component will appear in devices list. After all components have been selected, click OK to close the component library browser.**



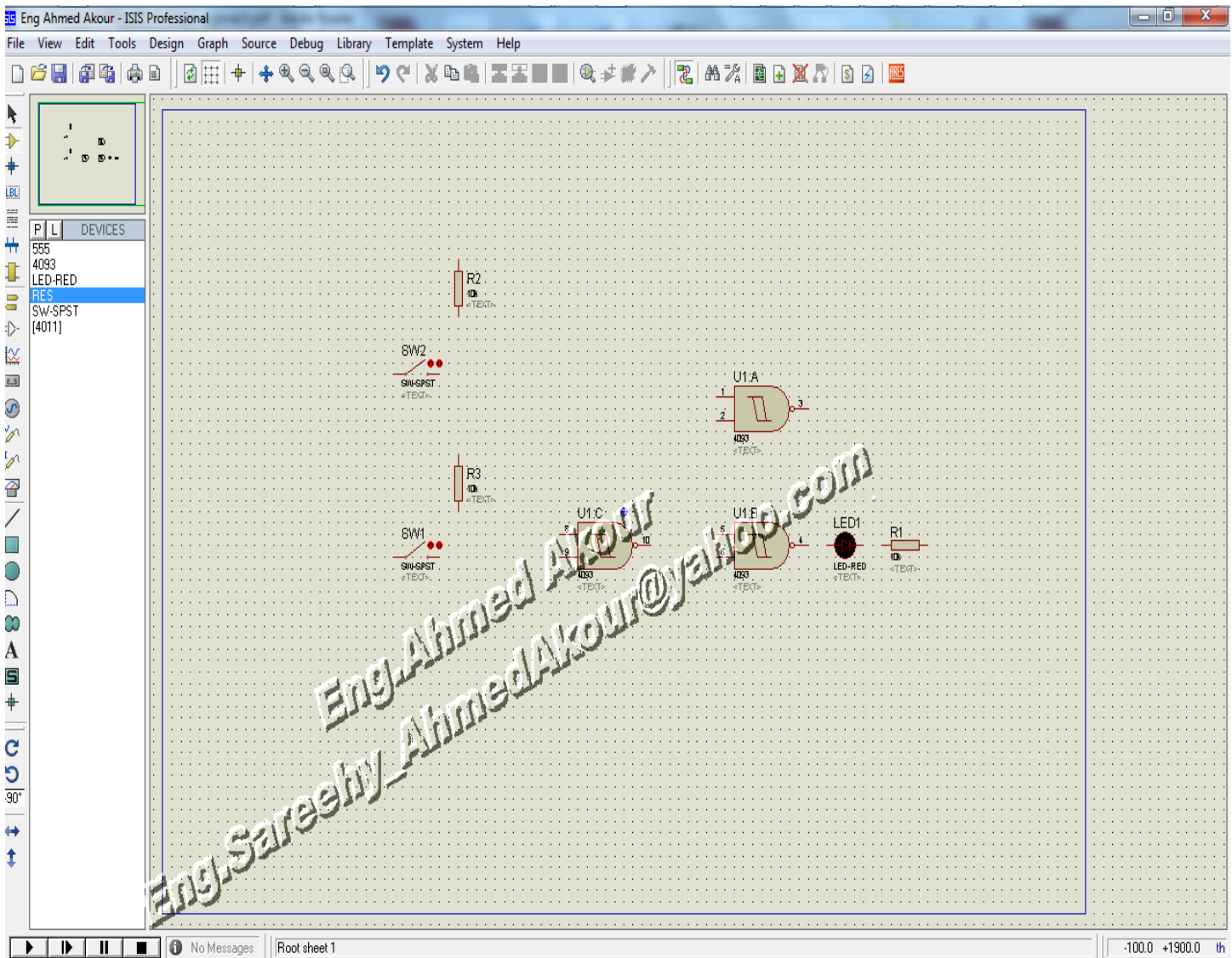
✚ To place components, click on the selected component list in devices box then single click on the drawing area. The image of the selected component will appear. Then just click on any part of drawing area to place the component.



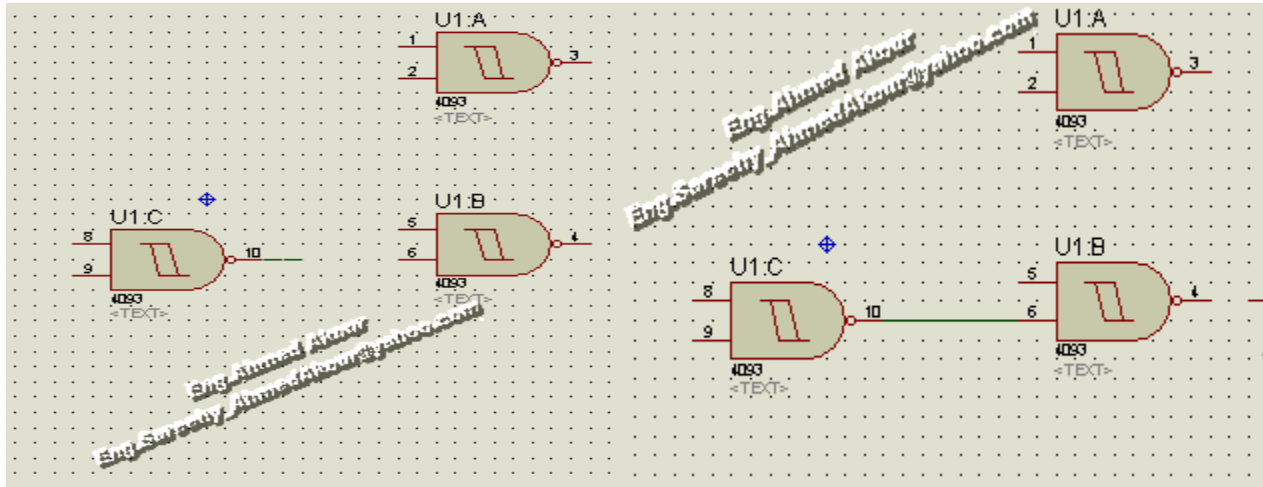
- ✚ To Move a placed component just place the cursor on the component until the move icon appear, then click hold and move the component anywhere area .
- ✚ To change the component orientation, just right click on the component and chose the orientation you want.
- ✚ To change the component properties such as the value, just right click on the component and chose the edit properties and then change the property you want.



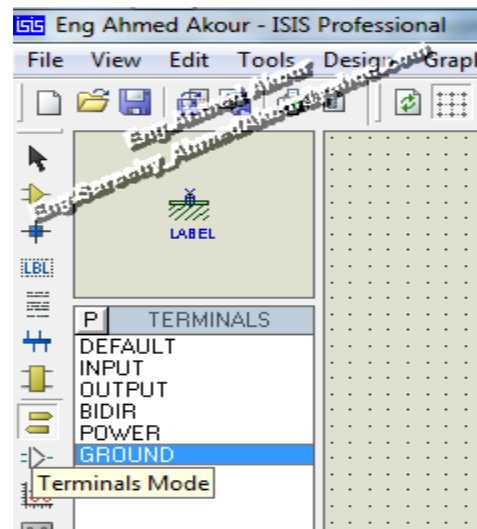
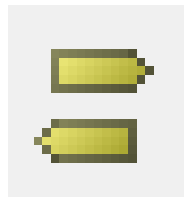
- Now place all the components needed in drawing area with right quantities to design circuit.



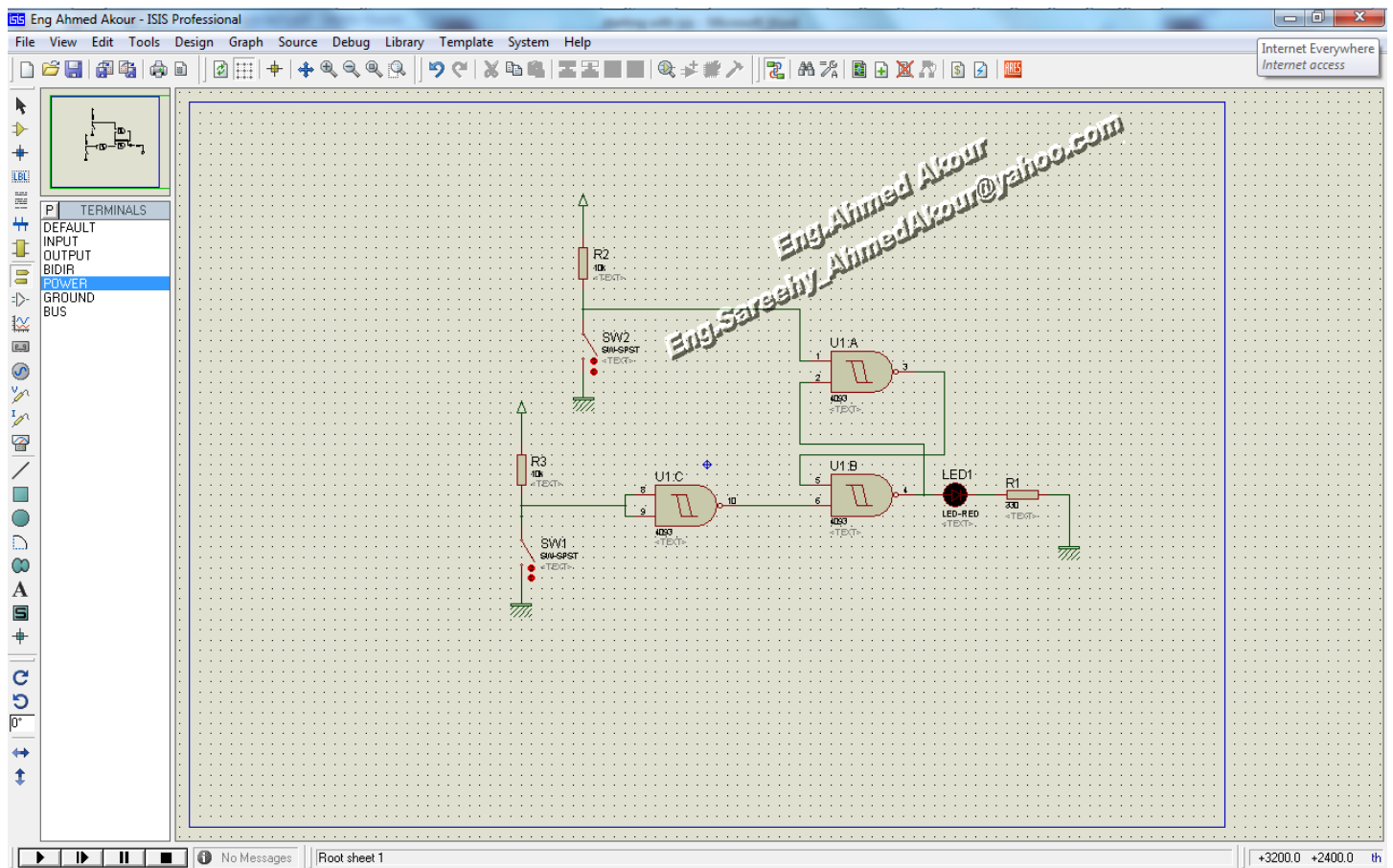
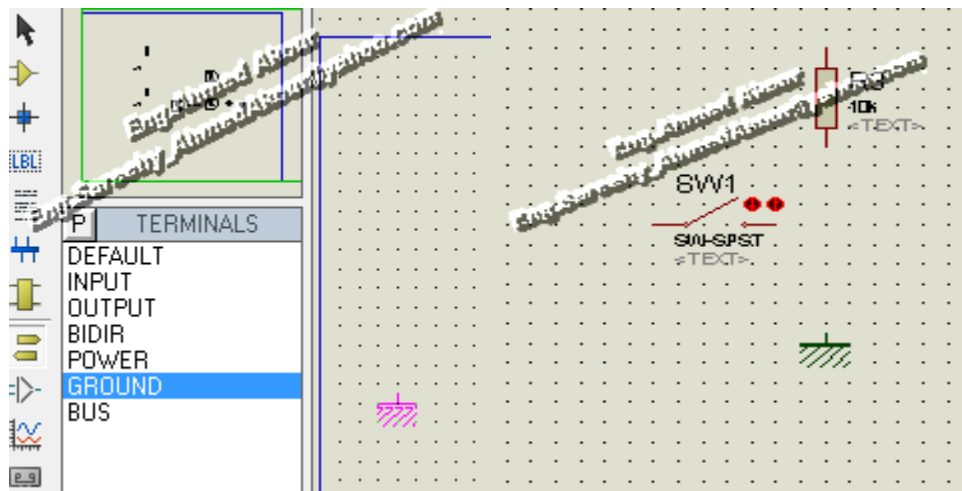
- To make wire connection, just place the cursor on component pin until red square appear, then click on the component to join the wire. Join the wires to all components according to the circuit design.



- ✚ To remove the wrong connection made, right click on the wire connection and choose delete wire or just double right click on the wire.
- ✚ The next step is to select power and ground terminal needed, to choose a terminal click the icon button for Terminal mode.

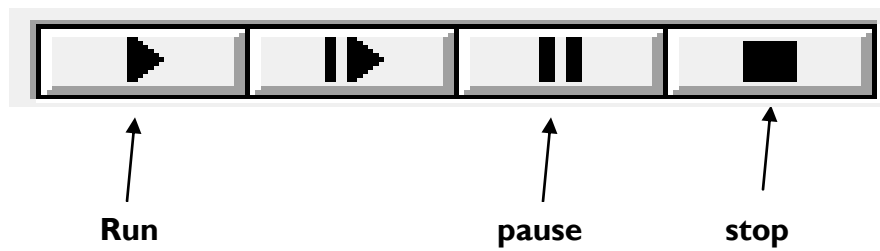


- ✚ To place terminals, click on the selected terminal list in terminal box then single click on the drawing area. The image of the selected terminal will appear. Then just click on any part of drawing area to place the terminal.

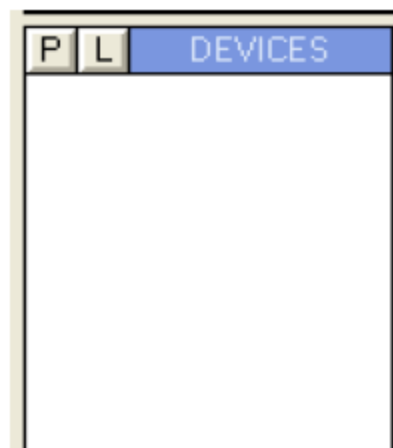


After you finish drawing the schematic circuits you want to start simulation process to test your circuit design. To start the simulation process just click on play button on simulation bar and show the response of your circuit design.

To stop the simulation just click on stop button on simulation bar.

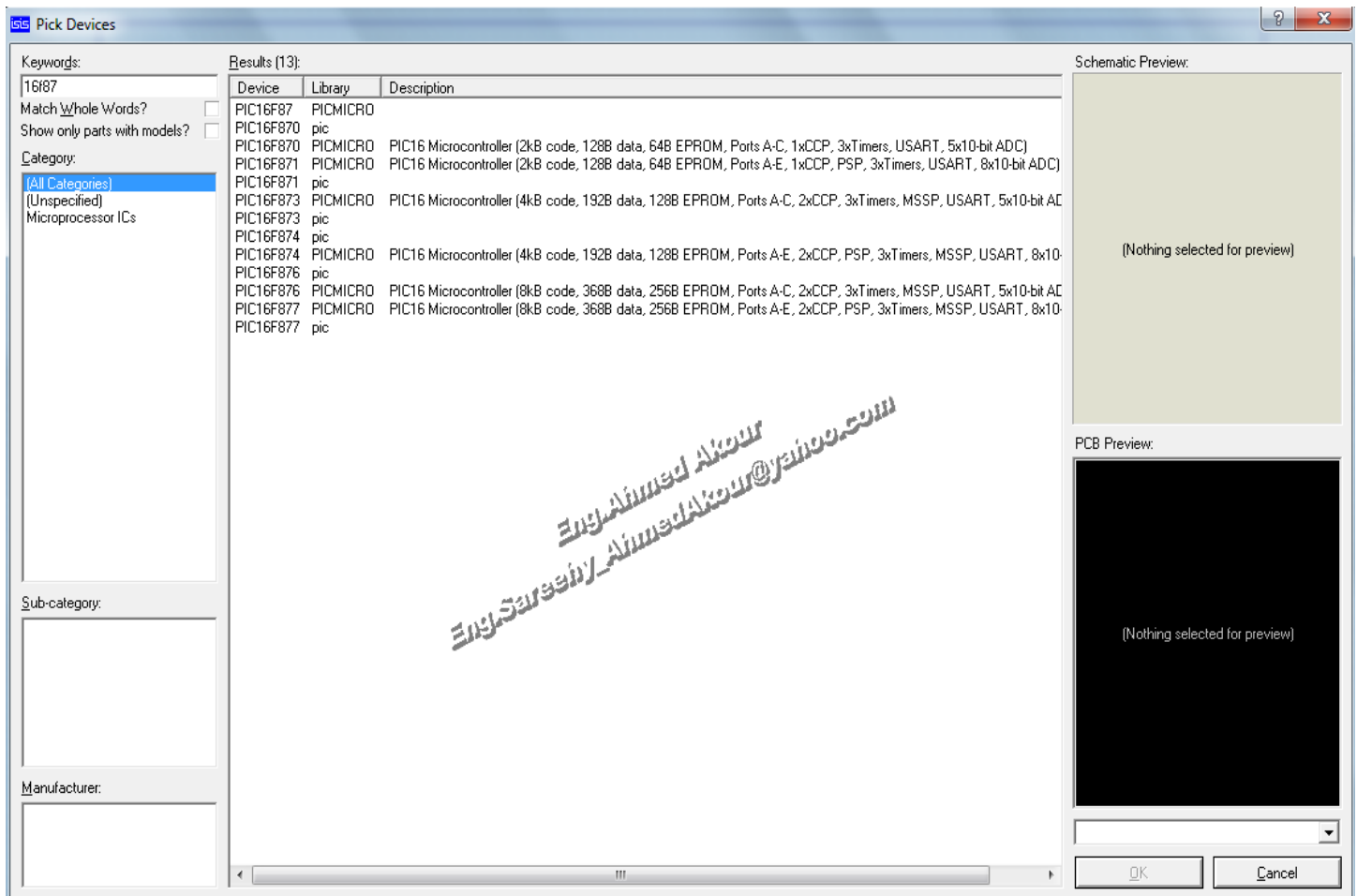


- **Eg:** Suppose now that you want to choose PIC16F877A to make a simulation then you must follow the following steps:
1. Click on Components Mode.
 2. After that click on letter *P*, *P* stands for Pick, show figure below.

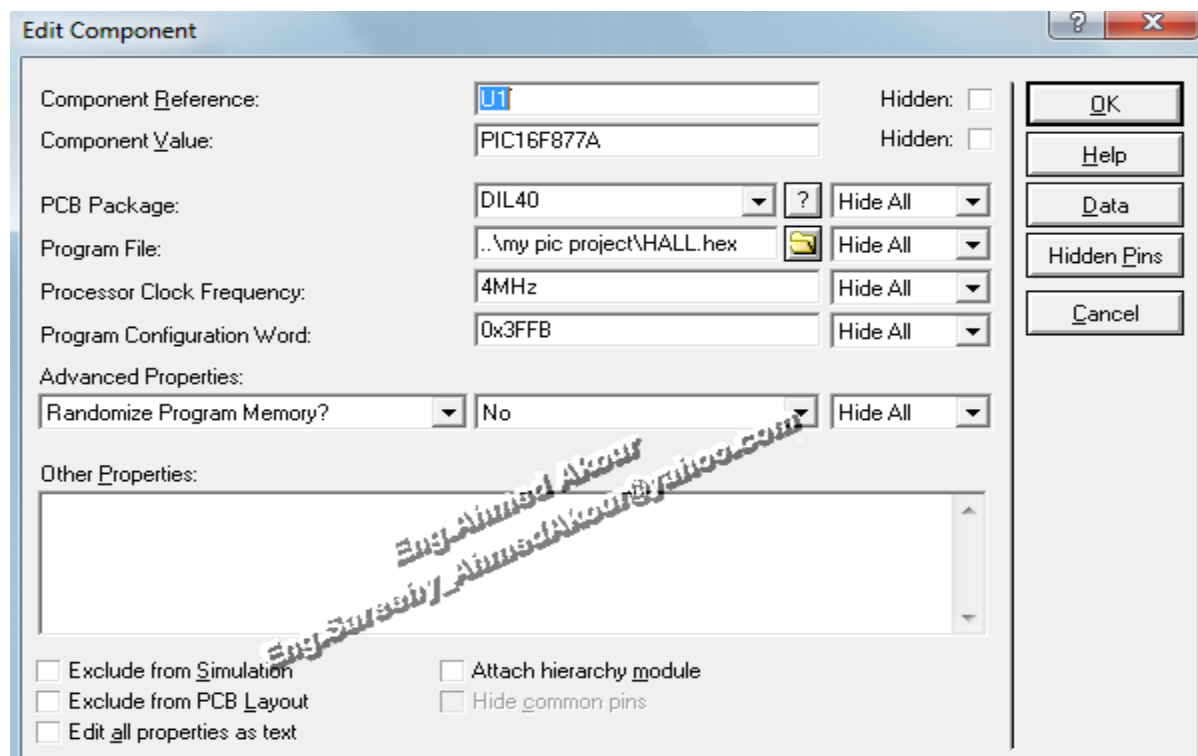


Empty component list, click on P to pick a device.

3. A new window will be displayed; it contains all components, with a powerful search engine, as shown in figure below.
4. In the Keyword fields, type 16f877a as a whole word, or simply type 16f87 to list all PIC16F87xx proteus supported devices in this range, show figure in the next page.
5. In Results list, make sure to highlight PIC16F877A, then press OK button. If you want to pick another component, without pressing OK button; simply double click on PIC16F877A will be listing it automatically into devices list, and then delete the contents of the Keyword text box and make your new search again.



- ❖ **After choosing all components for your design, put the components on the model screen, simply by choosing any components and clicking anywhere on the sheet, after that you must wire all components as drawn in your schematic, note that if your simulation contains MCUs, MPUs, FPGAs... or any programmed device; then you must load the firmware or program into the device, for MCUs say PIC16F877A; double click on it, or right click on the device and choose Edit Properties (Ctrl+E); after that load the HEX file from Edit Component window, show figure below.**



Also from the same window i.e. **Edit Component** shown above, you must set the used MCU's frequency clock for example 1 MHz, 4 MHz, 32 KHz, etc...; the crystal is connected by default.

Very important note: In PIC16F877A pin_1 (MCLR; Master clear) is used for reset. It is active low and this means that it must be connected to 5 volt (V_{CC} , V_{DD}) in normal mode, and to make a reset for the PIC it must be connected to 0 volt (GND, V_{SS}). In simulation you have a choice to connect it or not (for PIC16F only), but in reality you must connect it. In the same way the crystal oscillator (XTAL) on pins 13 and 14, it is optional in Proteus simulation but a must in reality. Finally, don't forget to feed the PIC with 5 volt and GND on pin_32 (V_{DD}) and pin_31 (V_{SS}) respectively. In Proteus simulator they are connected by default!

GOOD LuCK

Eng Ahmed Akour