North South University
Department of Electrical and Computer Engineering
CSE331L/EEE332L/ETE331L: (Sec-01)
Microprocessor Interfacing and Embedded Systems

Lab Week: 06   **Topic: Introduction to Proteus**

Proteus is a simulation and design software tool developed by Lab center Electronics for Electrical and Electronic circuit design. It also possess 2D CAD drawing feature. It deserves to bear the tagline "From concept to completion".

About Proteus

1) It is a software suite containing schematic, simulation as well as PCB designing.
2) ISIS is the software used to draw schematics and simulate the circuits in real time. The simulation allows human access during run time, thus providing real time simulation.
3) ARES is used for PCB designing. It has the feature of viewing output in 3D view of the designed PCB along with components.
4) The designer can also develop 2D drawings for the product.
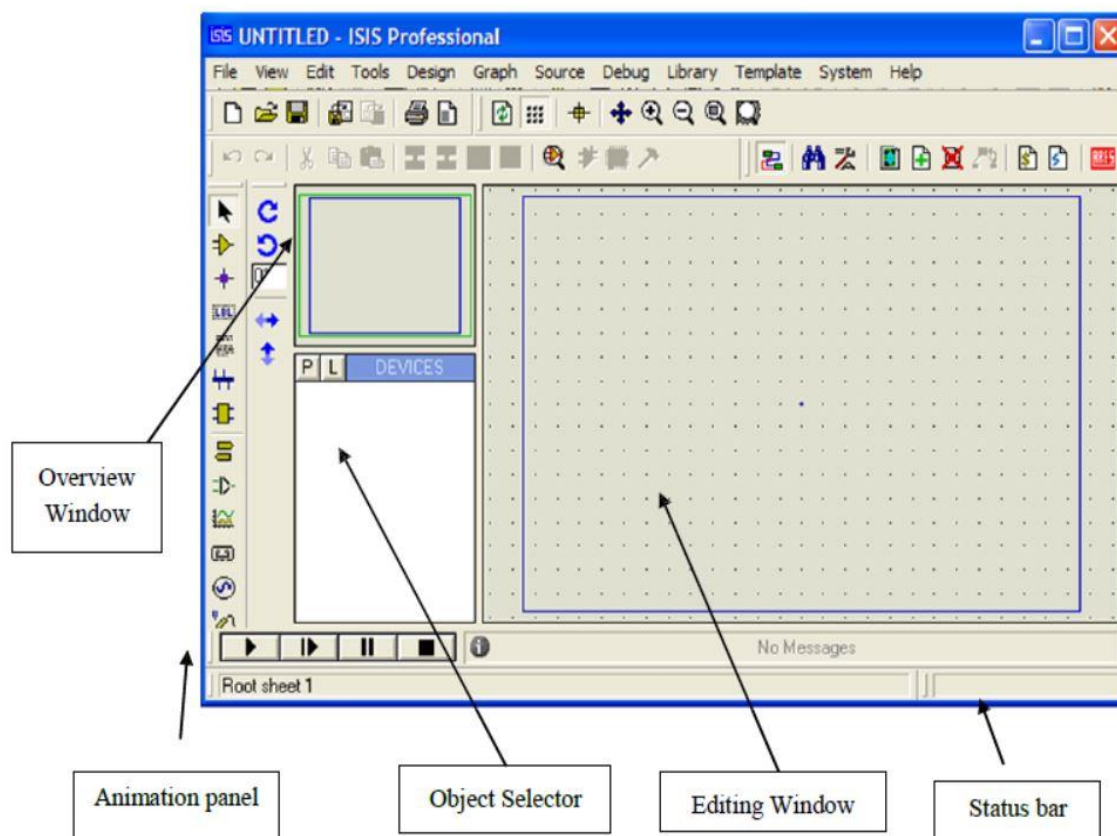
Figure 1. A screen shot of the Proteus IDE

Starting New Design

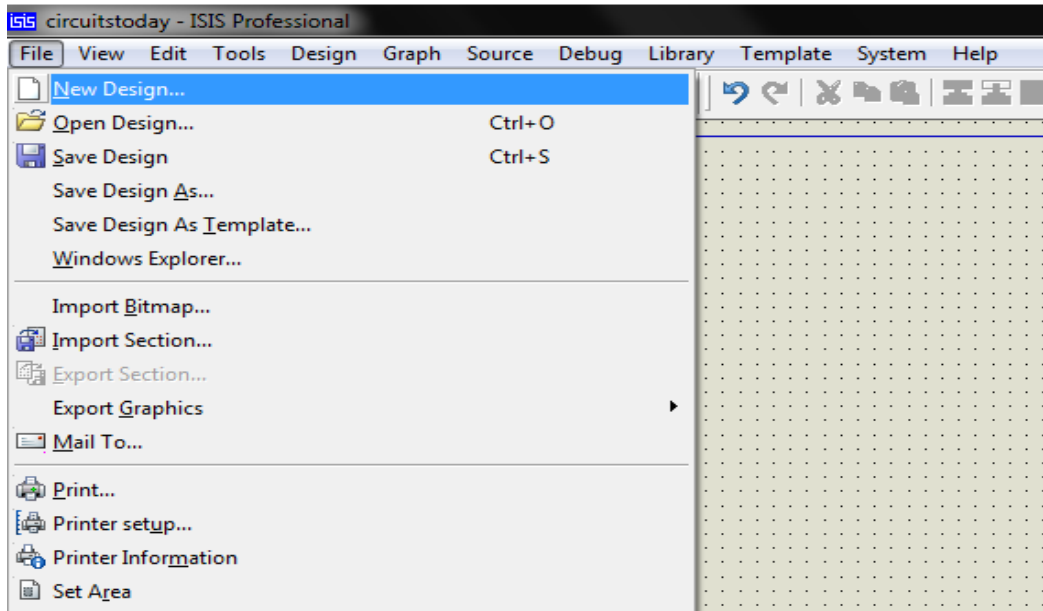**Step 1:** Open ISIS software and select new design in File menu

Fig: Proteus File Menu

Step 2: A dialogue box appears to save the current design. However, we are creating a new design file so you can click Yes or No depending on the content of the present file. Then a Pop-Up appears asking to select the template. It is similar to selecting the paper size while printing. For now select default or according to the layout size of the circuit.
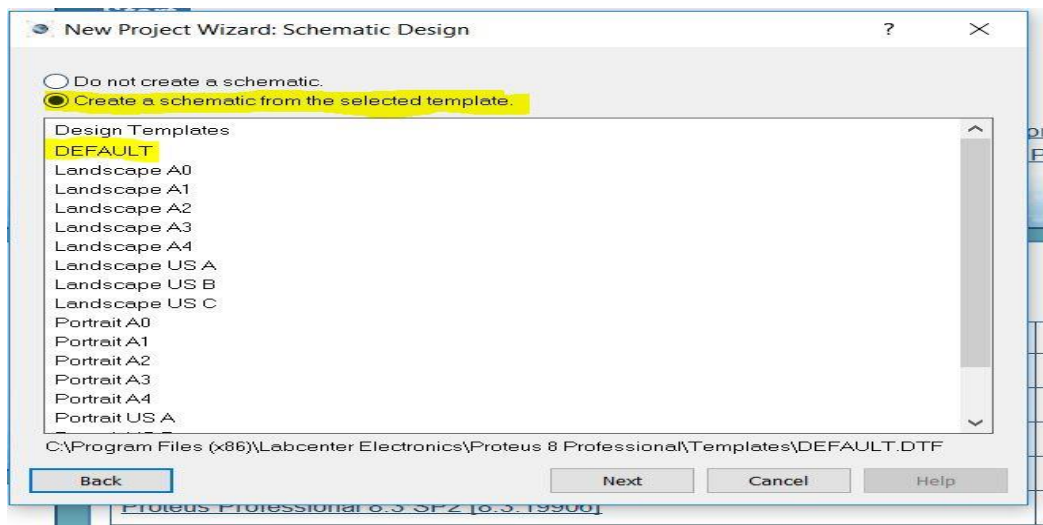

Fig: Proteus Default Template Select

Step 3: An untitled design sheet will be opened, save it according to your wish, it is better to create a new folder for every layout as it generates other files supporting your design. However, it is not mandatory.
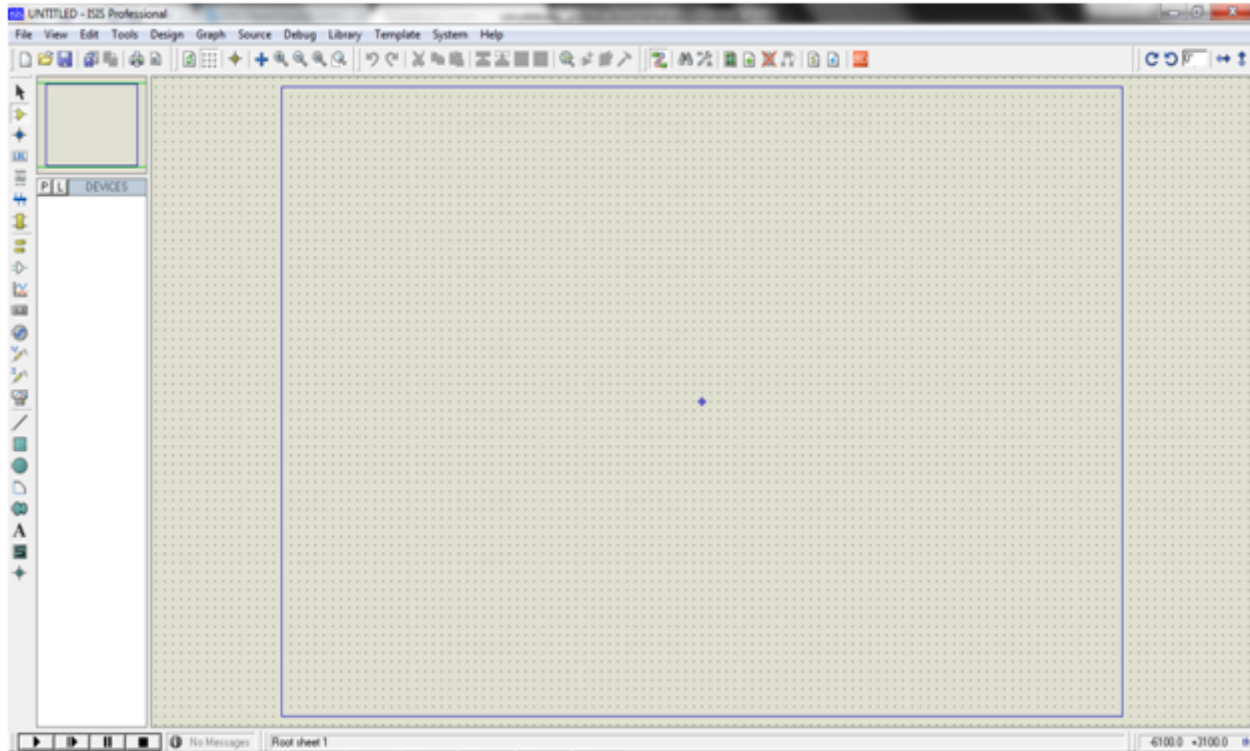

Fig: Proteus Design Sheet

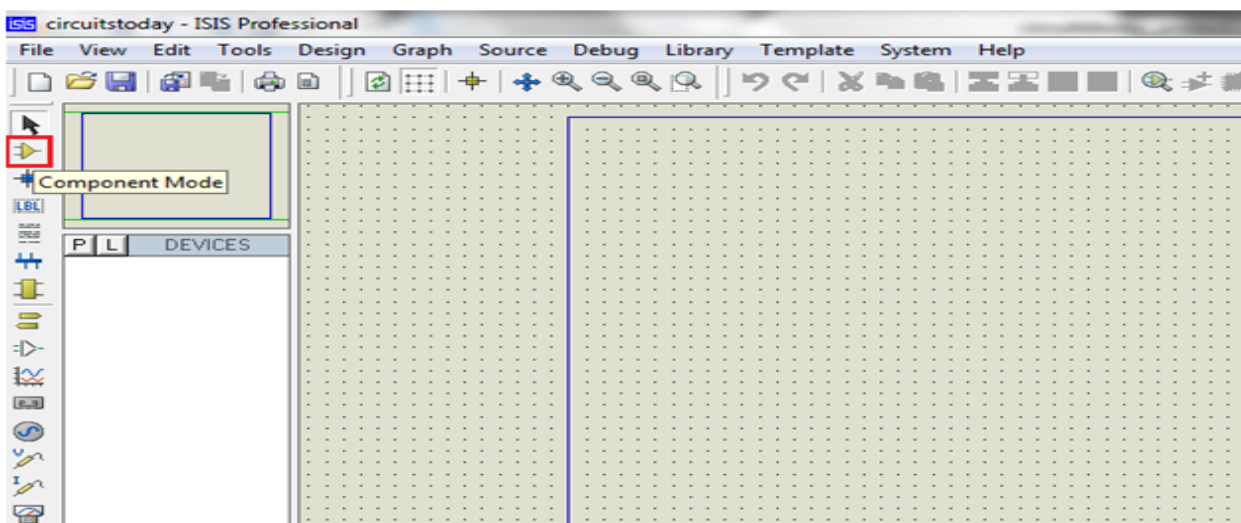**Step 4**: To select components, Click on the component mode button.

Fig: Component Mode

Step 5: Click on Pick from Libraries. It shows the categories of components available and a search option to enter the part name.
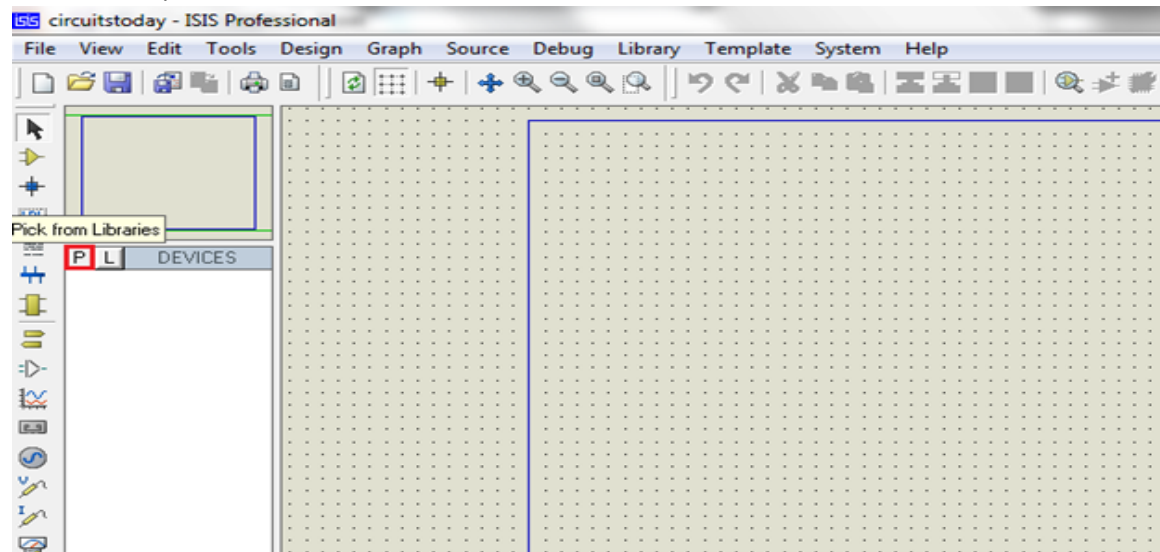

Fig: Pick from Libraries

Step 6: Select the components from categories or type the part name in Keywords text.
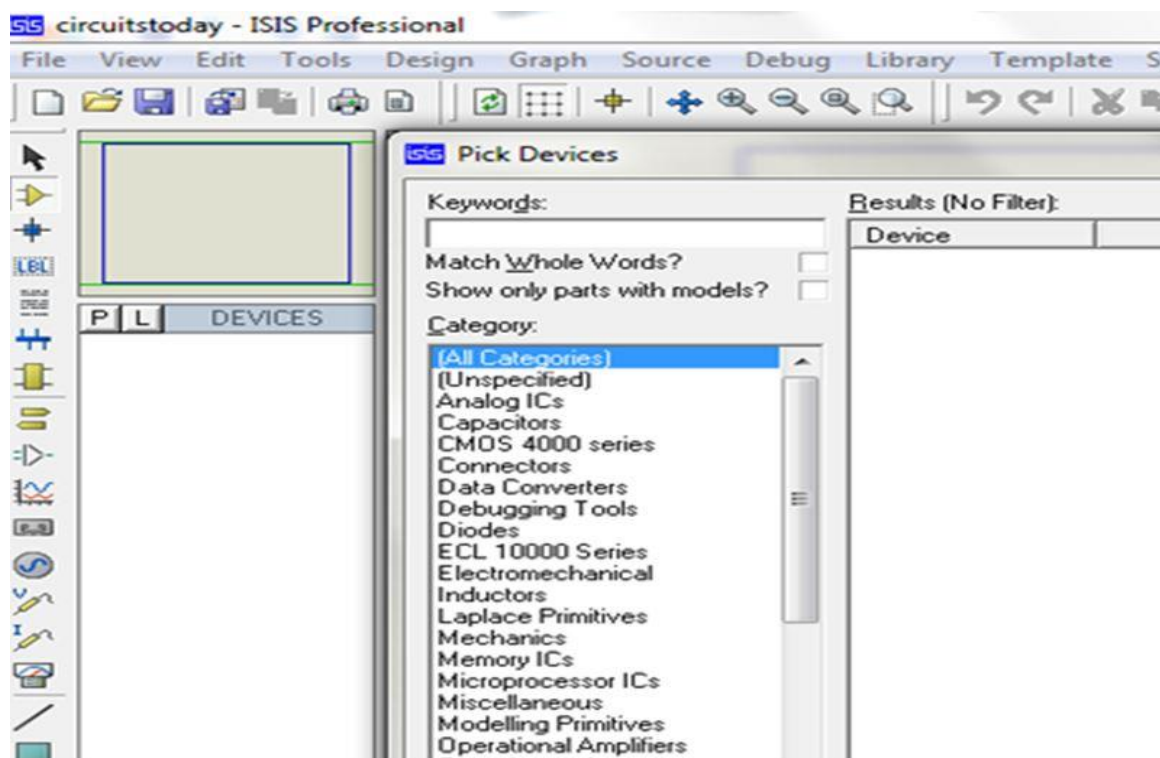

Fig: Keywords Textbox

Example shows selection of push button. Select the components accordingly.
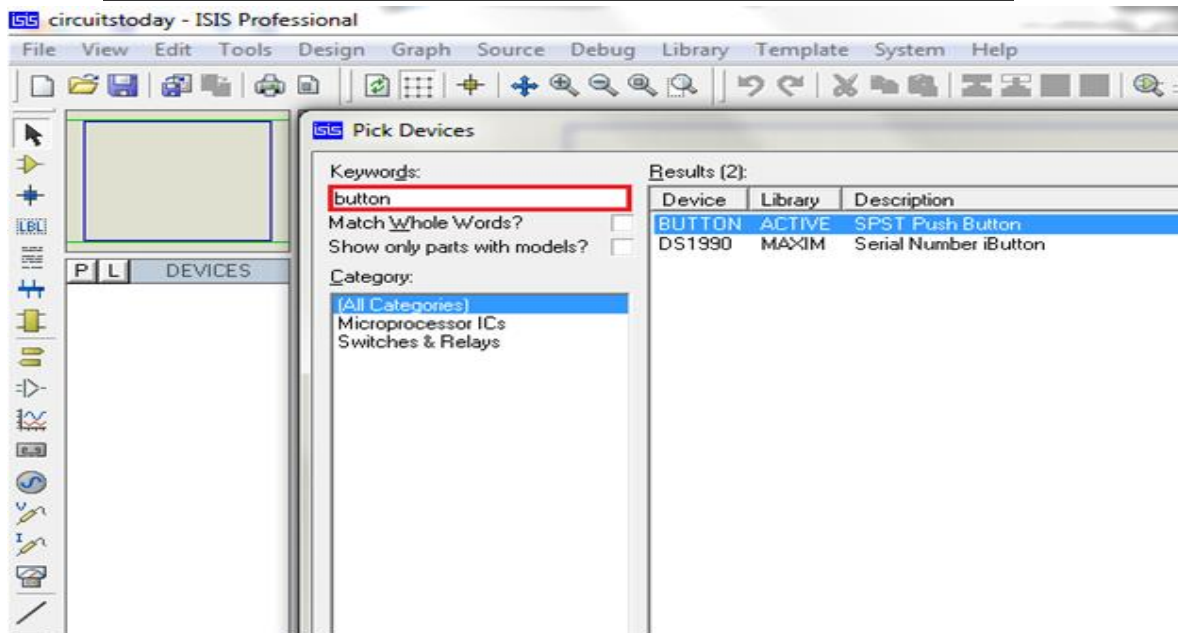

Fig: Push Button Selection

Step 7: The selected components will appear in the devices list. Select the component and place it in the design sheet by left-click.
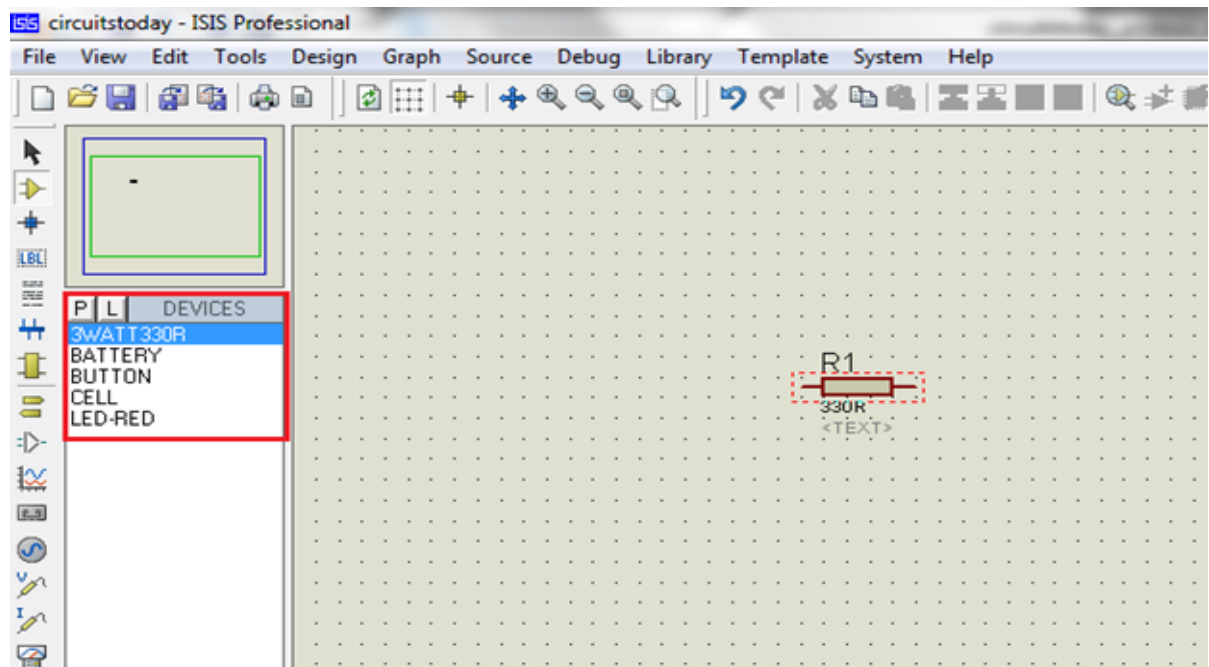

Fig: Component Selection

Place all the required components and route the wires i.e., make connections.

Either selection mode above the component mode or component mode allows to connect through wires. Left click from one terminal to other to make connection. Double right-click on the connected wire or the component to remove connection or the component respectively.
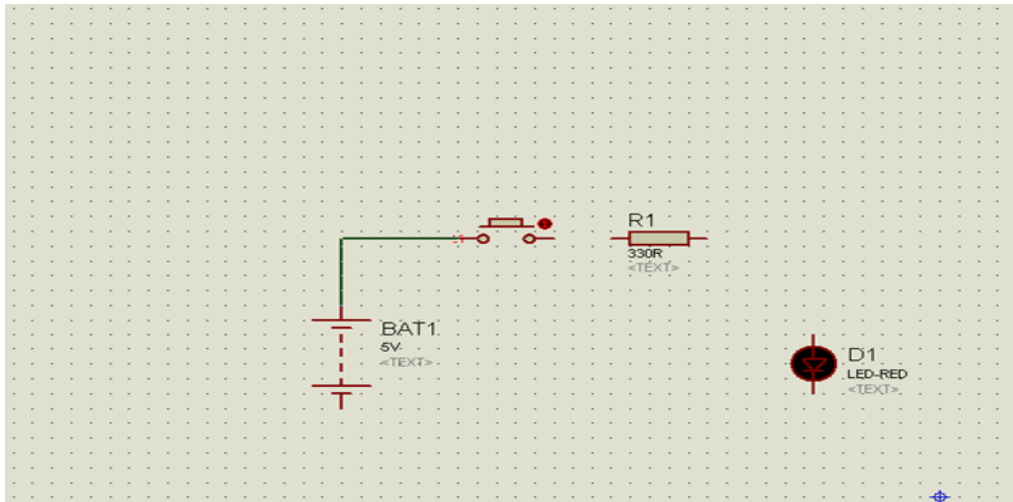

Fig: Component Properties Selection

Double click on the component to edit the properties of the components and click on Ok.
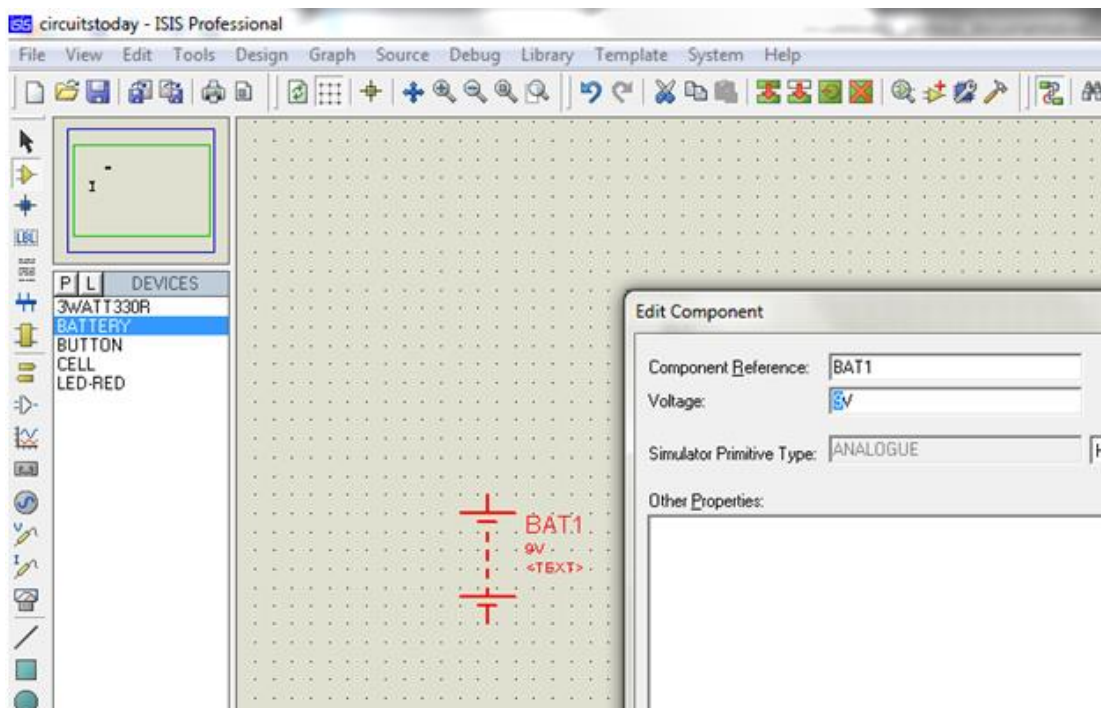

Fig: Component Properties Edit

Step 8: After connecting the circuit, click on the play button to run the simulation.
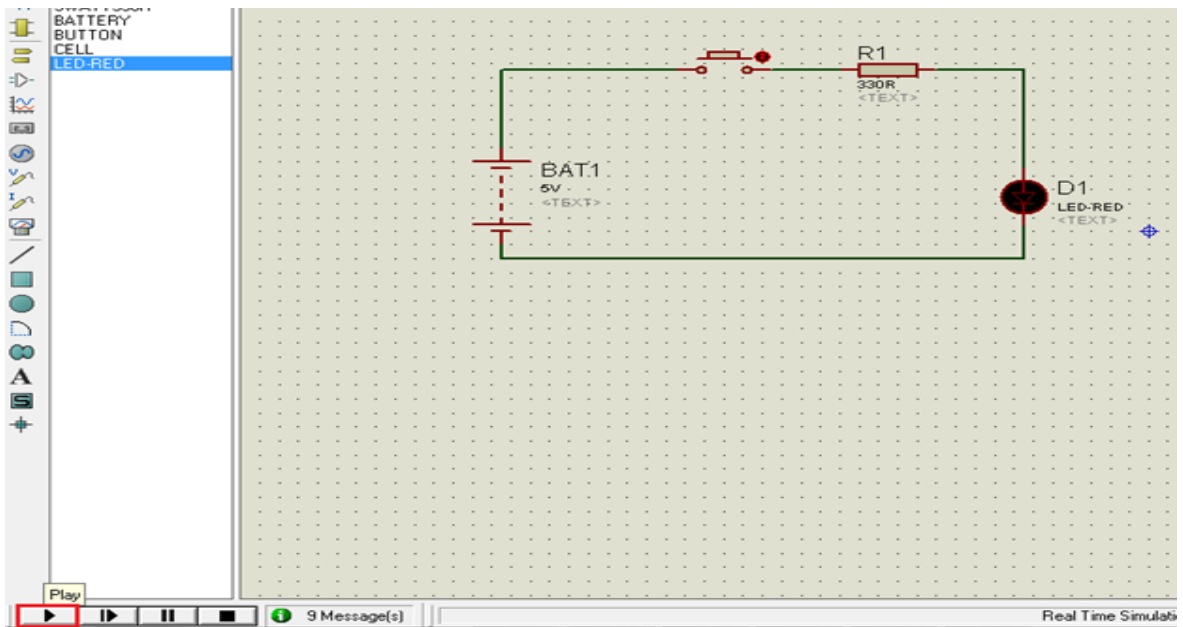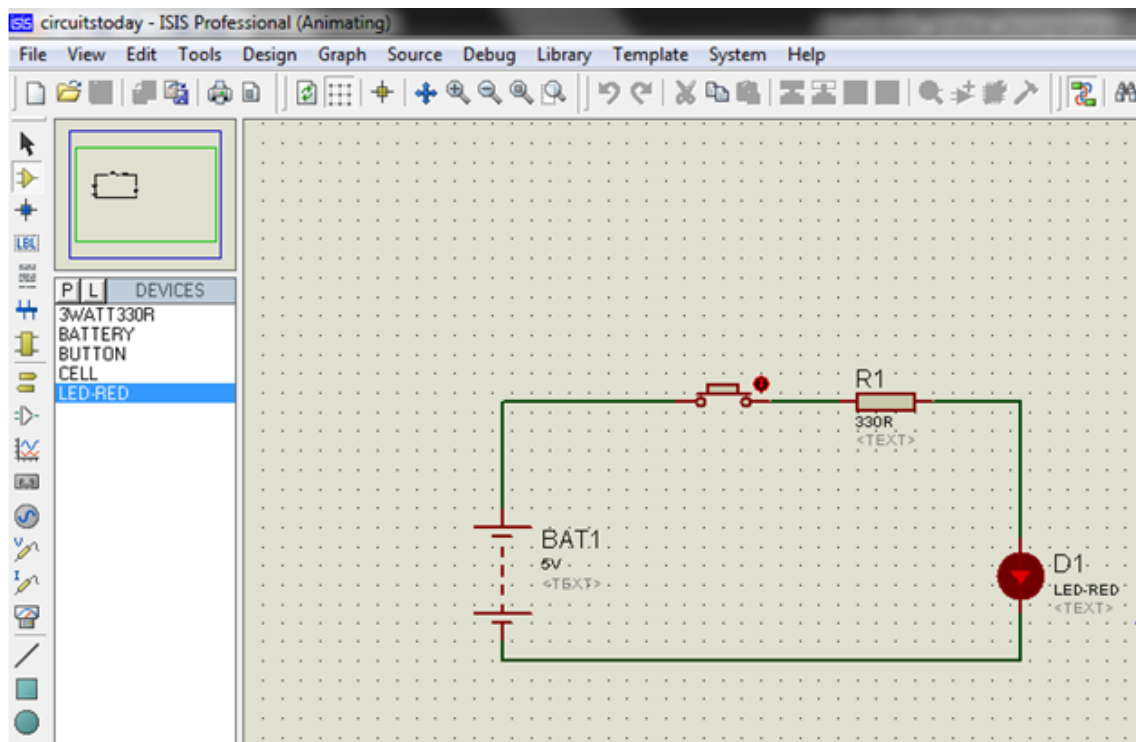

Fig: Simulation Run

Fig: Simulation Animating

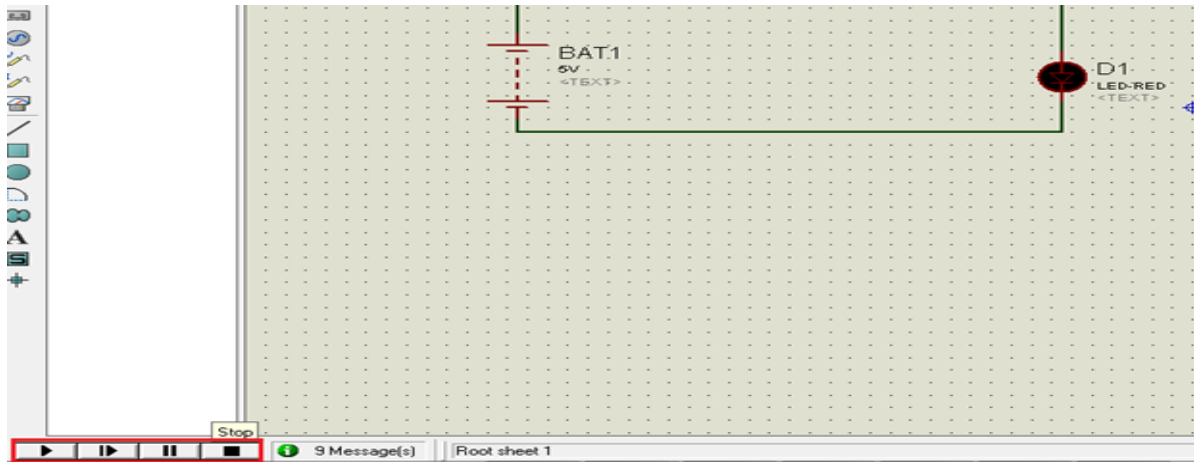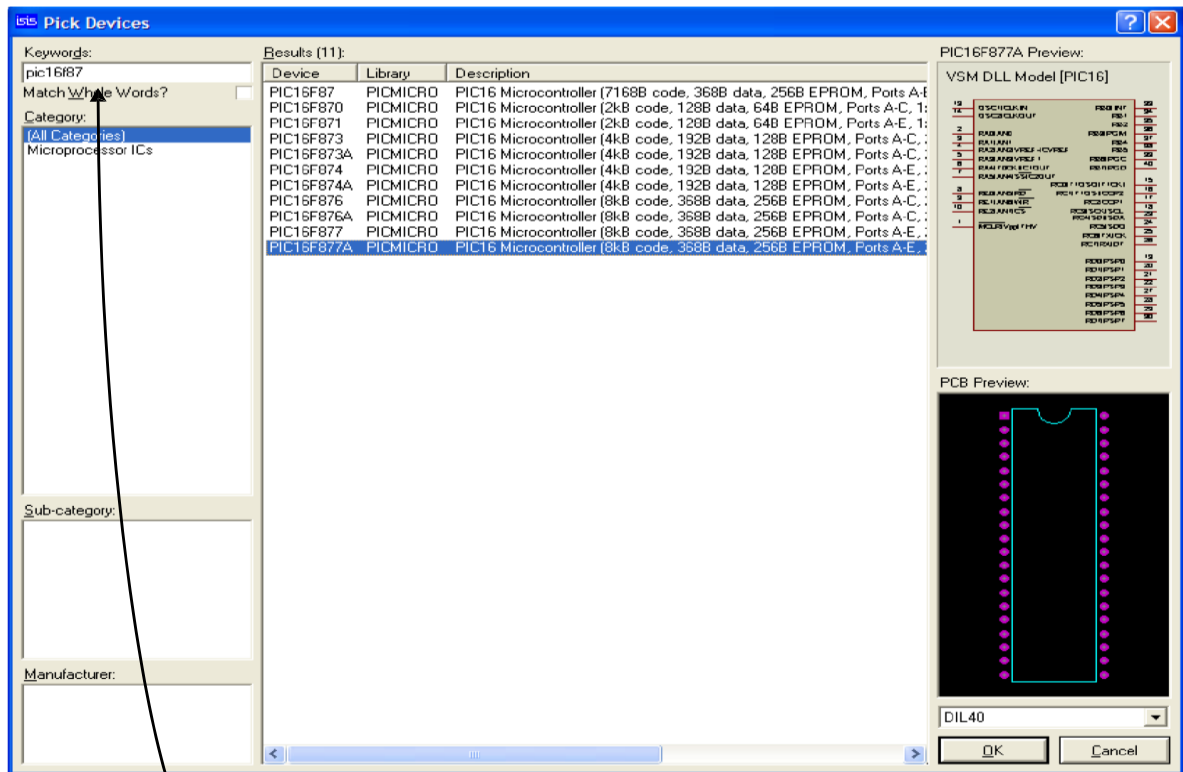Simulation can be stepped, paused or stopped at any time.

Fig: Simulation Step-Pause-Stop Buttons

How to Place a PIC microcontroller in Proteus

Type ' PIC16F877A ' in the Key words field and double click on the result to place
the PIC16F877A into the Object Selector.
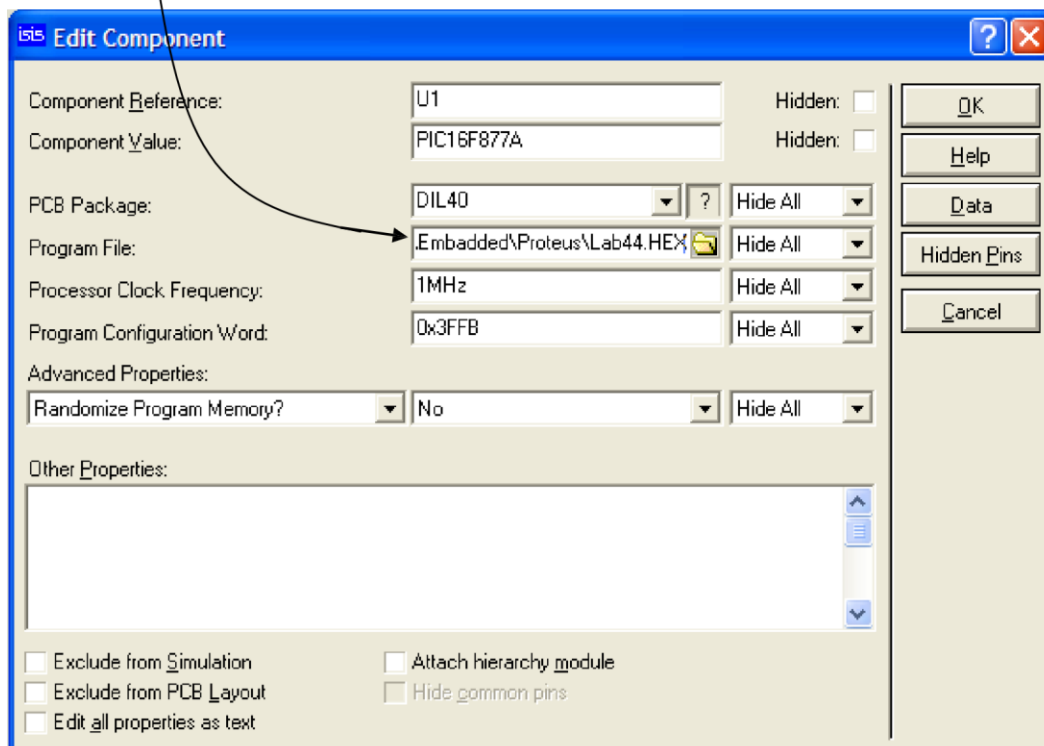
Attaching the HEX File

The next stage is to attach the HEX file to our design in order to successfully simulate the design. We do this through the following steps.

It is necessary to specify which file the processor is to run. In our example this will be filename.Hex (the hex file produced from MPASM subsequent to assembling filename.asm).

To attach this file to the processor, right click on the schematic part for the PIC and then left click on the part. This will bring up the Edit Component dialogue form which contains a field for Program File. If it is

not already specified as filename.hex either enter the path to the file manually or browse to the location of the file via the 📁 button to the right of the field. Once you have specified the hex file to be run press ok to exit the dialogue form.
We have now attached the source file to the design .

Reference: [1] http://www.circuitstoday.com/proteus-software-introduction.

[2] Open source documents.