Introduction to Proteus

The PROTEUS Environment:

Proteus PIC Bundle is the complete solution for developing, testing and virtually prototyping your embedded system designs based around the Microchip Technologies ™ series of microcontroller. This software allows you to perform *schematic capture* and to *simulate* the circuits you design.
A demonstration on the use of *PROTEUS* will be given to you on this lab session, after that; you are encouraged to learn to use the software interactively.

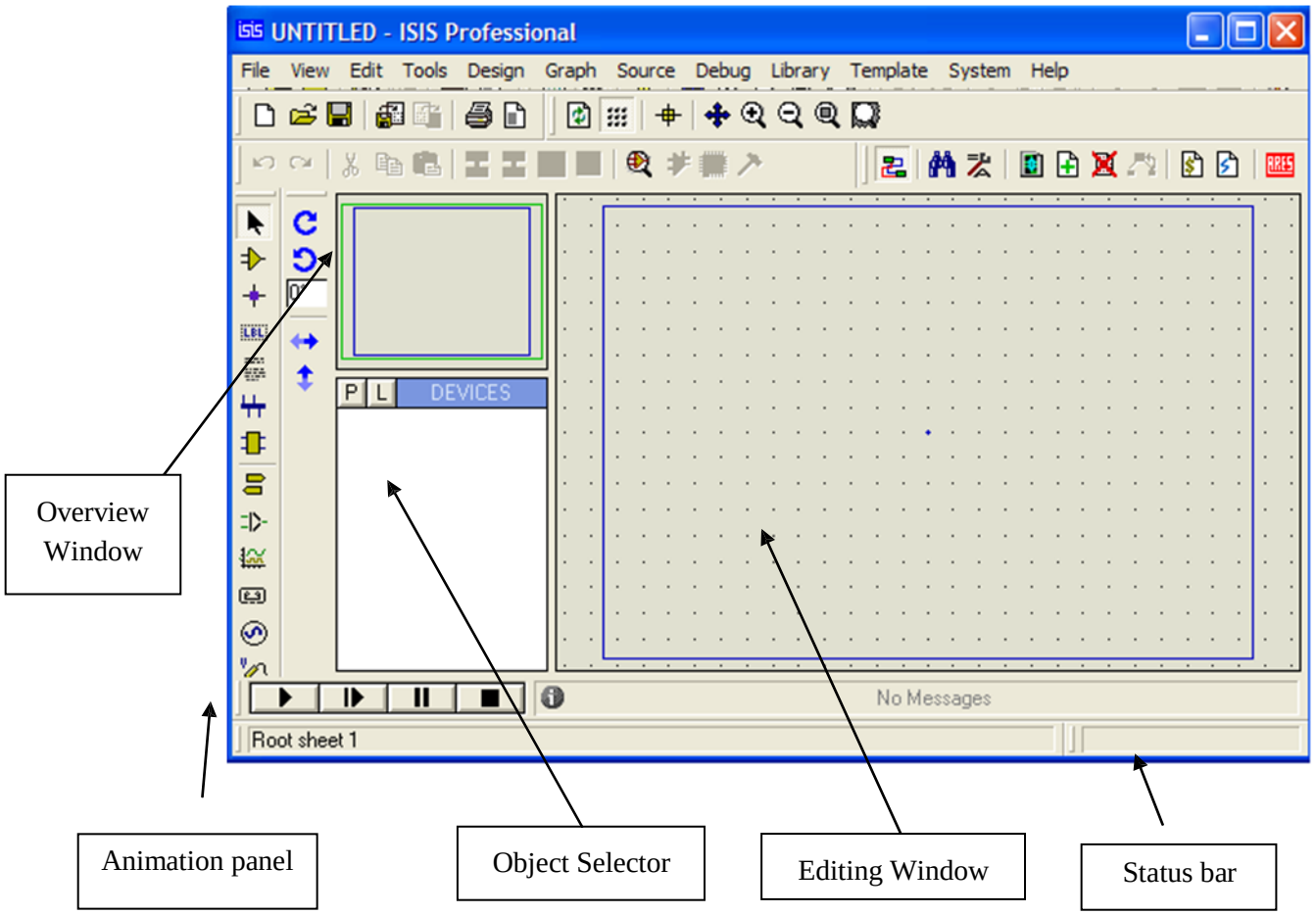

Figure 1. A screen shot of the Proteus IDE

Proteus How to Start

Drawing the Circuit

Start a fresh design, select New Design from File menu then the Create New Design dialogue now appears as shown in Figure 2 and 3. Select Default and press OK.

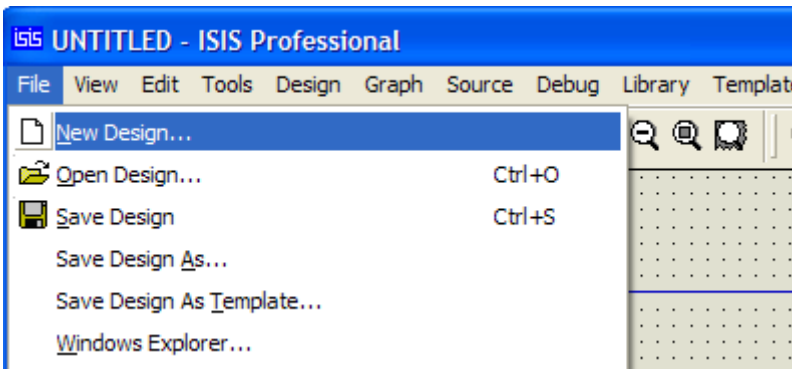

Figure 2

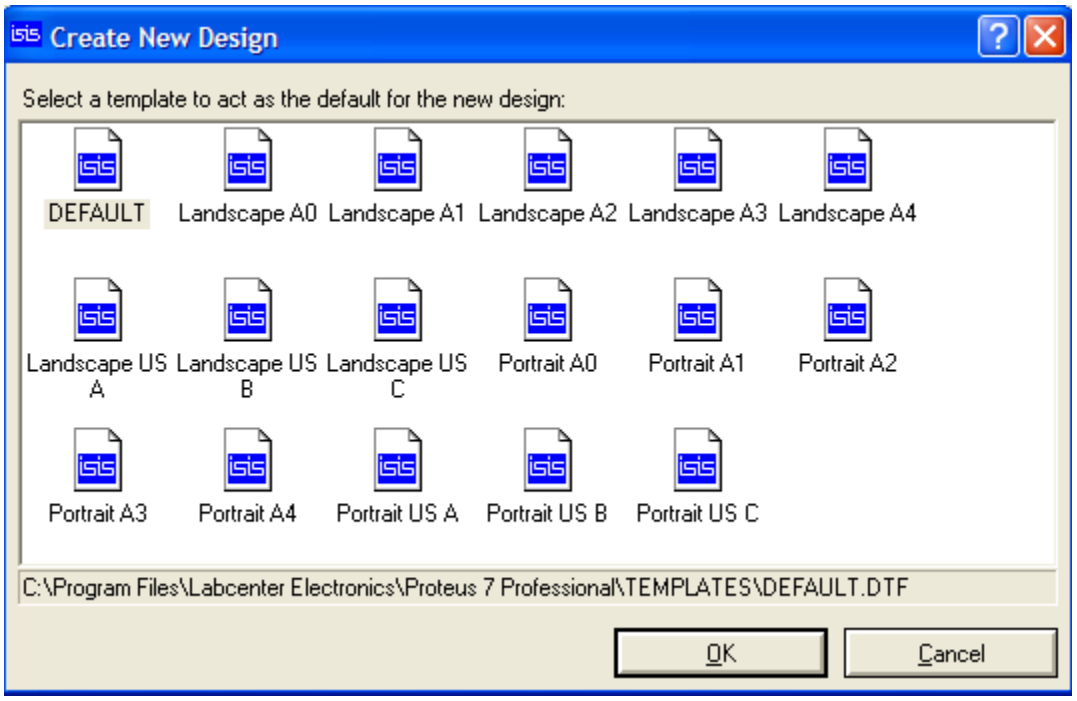

Figure 3

From the Library menu select Pick Device/Symbol see Figure 4 or Left click on the letter 'P' above the Object Selector as shown in Figure 5 to launch the Library Browser or Press the 'P' button on the keyboard. The Library Browser will now appear over the Editing Window see Figure 6.

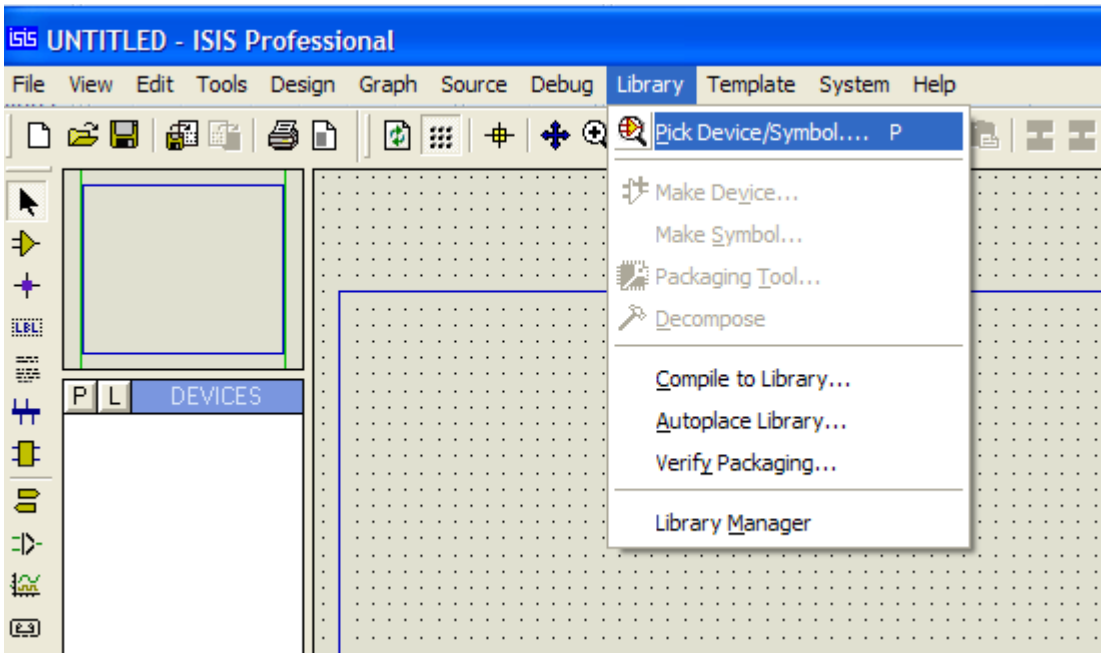

Figure 4

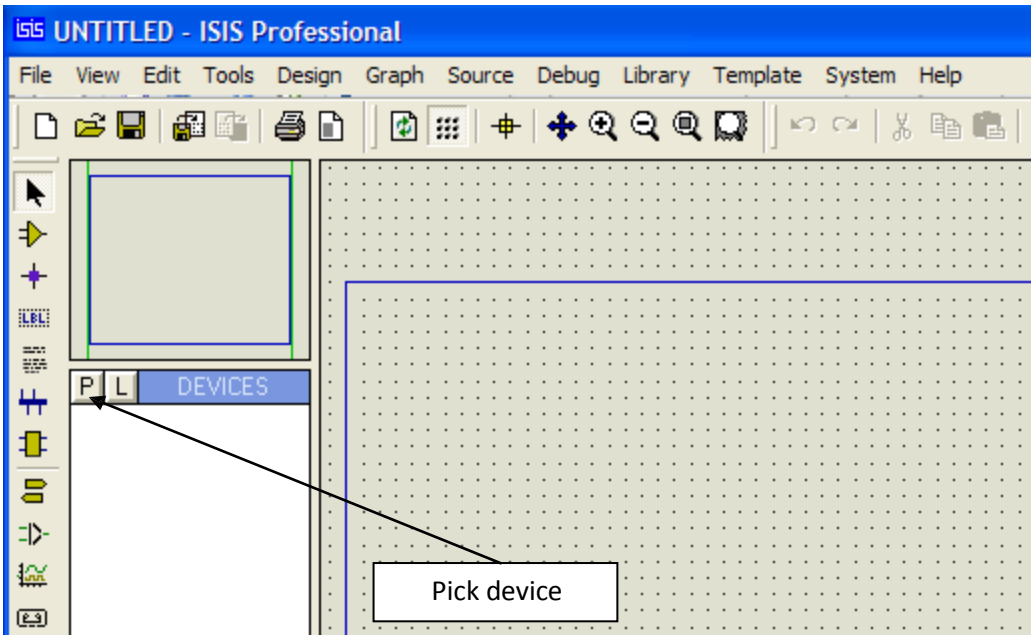

Figure 5

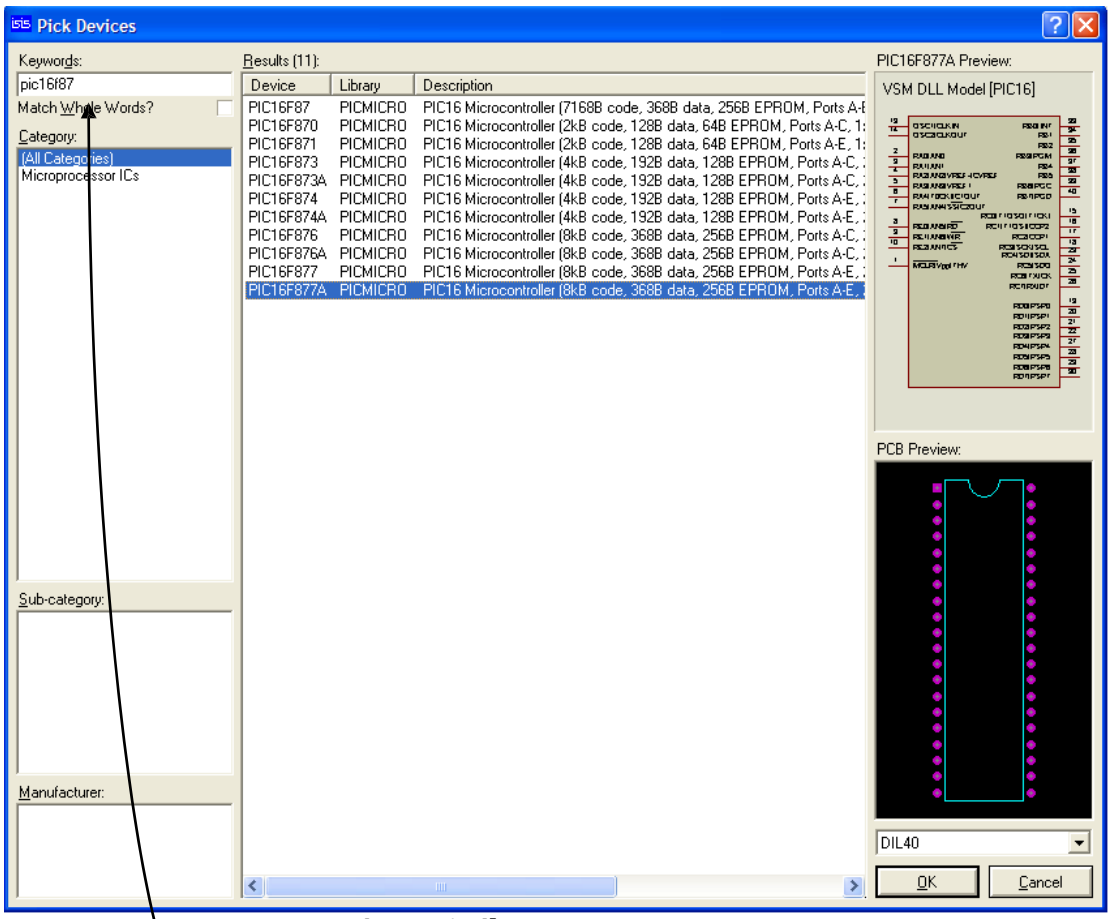

Figure 6 Library Browser

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

Type ' PIC16F877A ' in the Key words field and double click on the result to place the PIC16F877A in to the Object Selector. Do the same for the LEDs, Buttons, Crystal oscillator, capacitors, 7 SEG-COM-Cathode, Resistors.

Once you have selected all components into the design close the Library Browser and left click once on any component in the Object Selector
(This should highlight your selection and a preview of the component will appear in the Overview Window at the top right of the screen see Figure 7). Now left click on the Editing Window to place the component on the schematic - repeat the process to all components on the schematic.

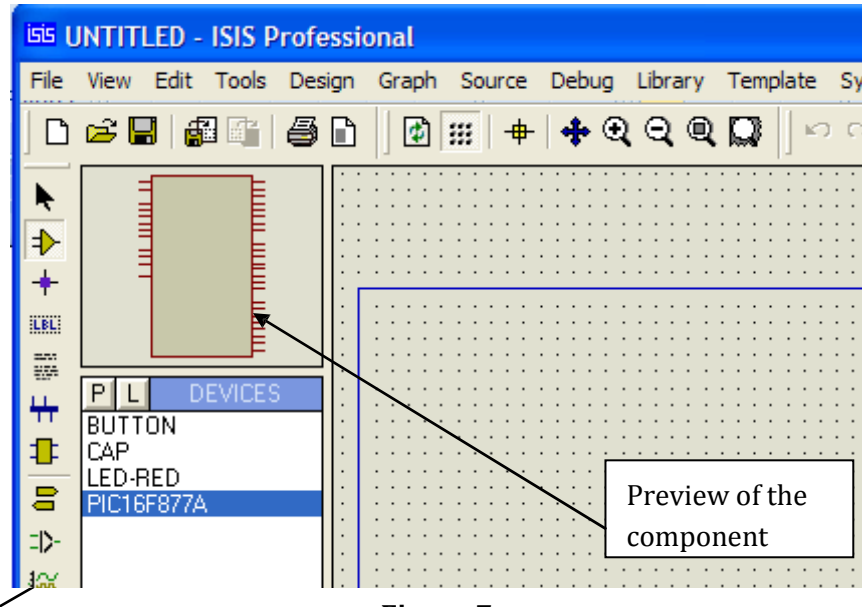

Figure 7

When you click left on any component in the Object Selector, a preview of the component will appear in the Overview Window

In order to place ground or 5 voltage right click on the Editing Window , select place then terminal then select ground (0 V) or power (5 V).

Connect the components to obtain the circuit you need.

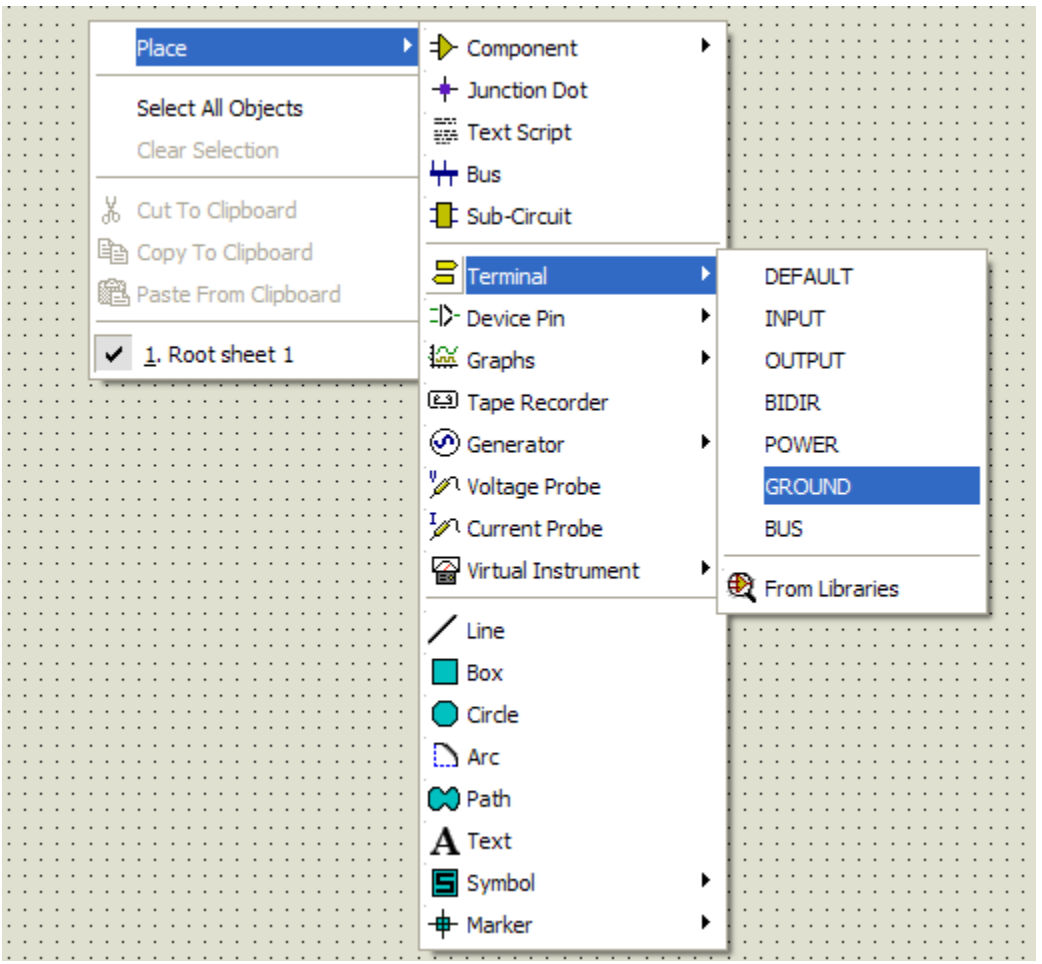

Figure 8

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 .

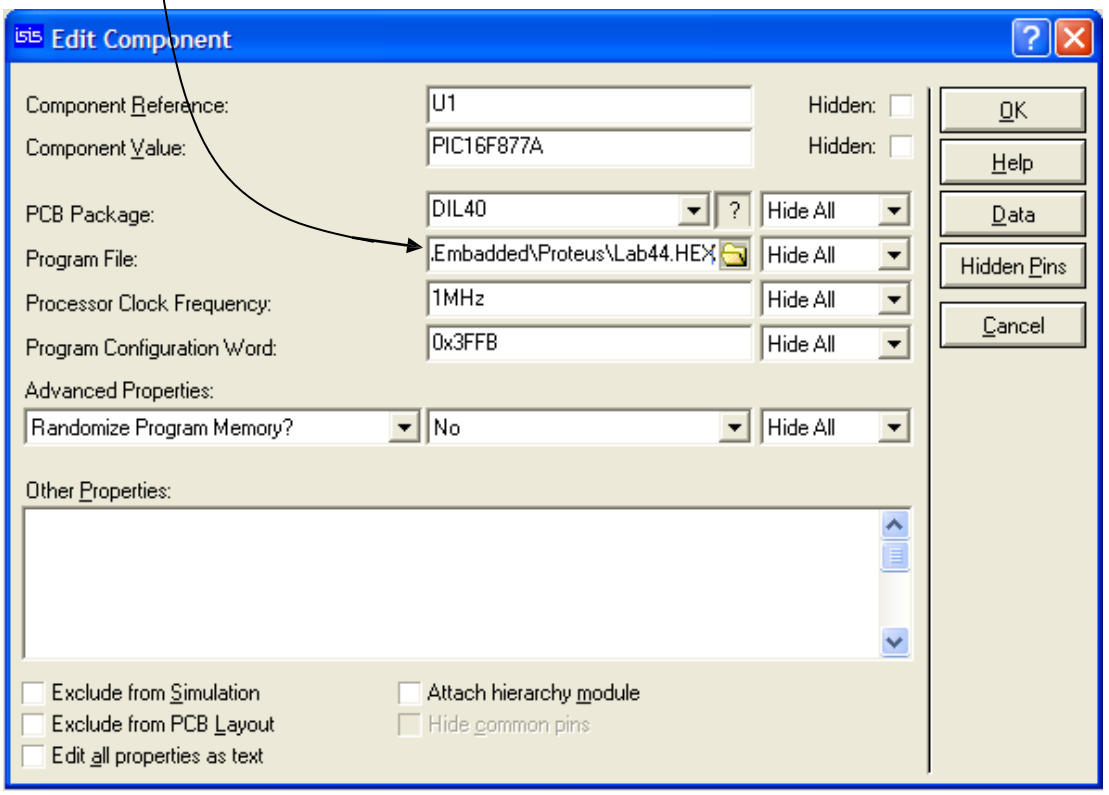

Figure 9

Debugging the Program (Simulating the Circuit)

In order to simulate the circuit point the mouse over the Play Button on the animation panel at the bottom right of the screen see Figure 10 and click left. The status bar should appear with the time that the animation has been active for.

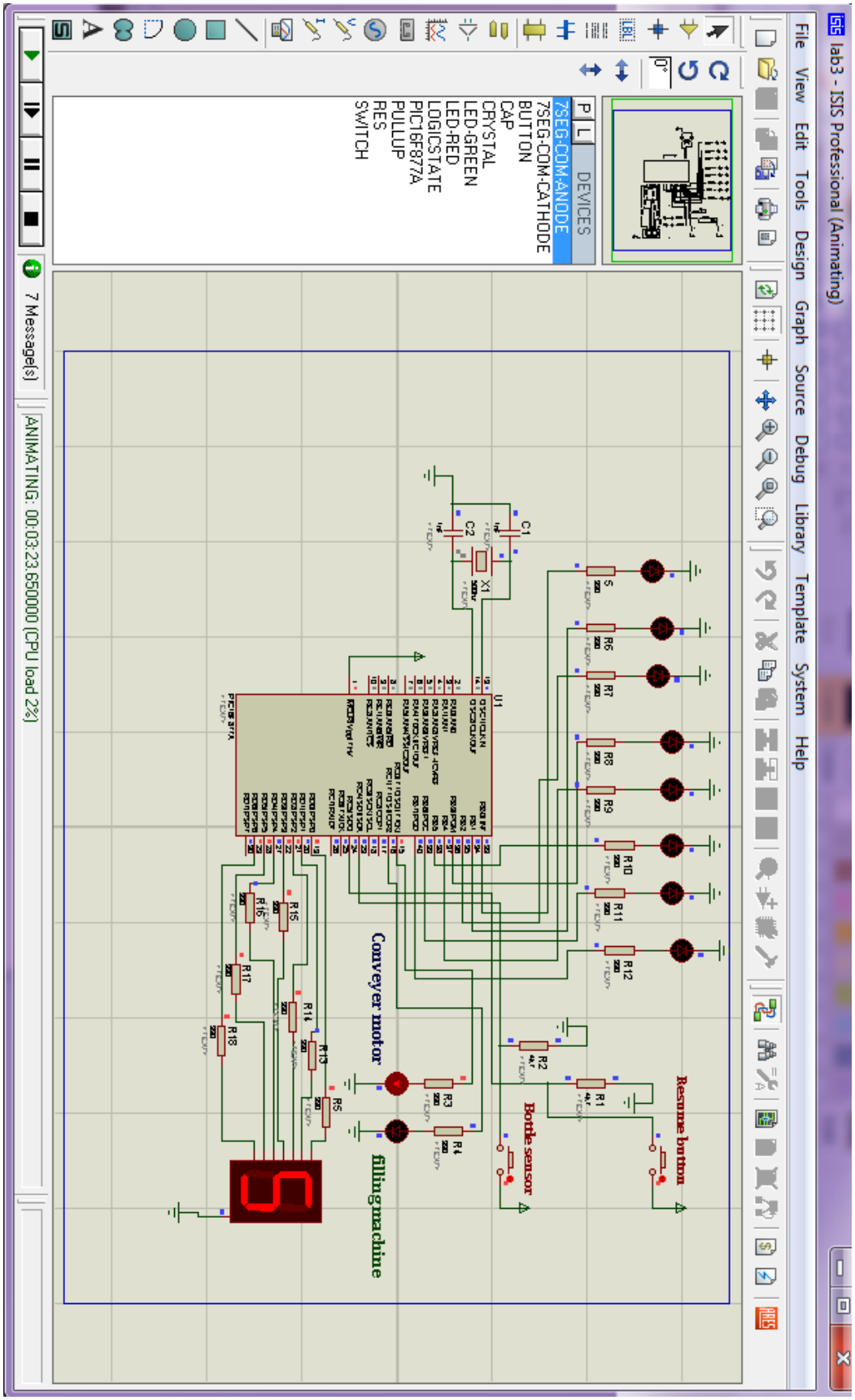

Figure 10: The Filling Machine Circuit