The PROTEUS Environment:

Proteus PIC Bundle is the complete solution for developing, testing and virtually prototyping your system designs based around the Microchip Technology series of Microcontroller. This software allows you to perform *schematic capture* and to *simulate* the circuits you design.
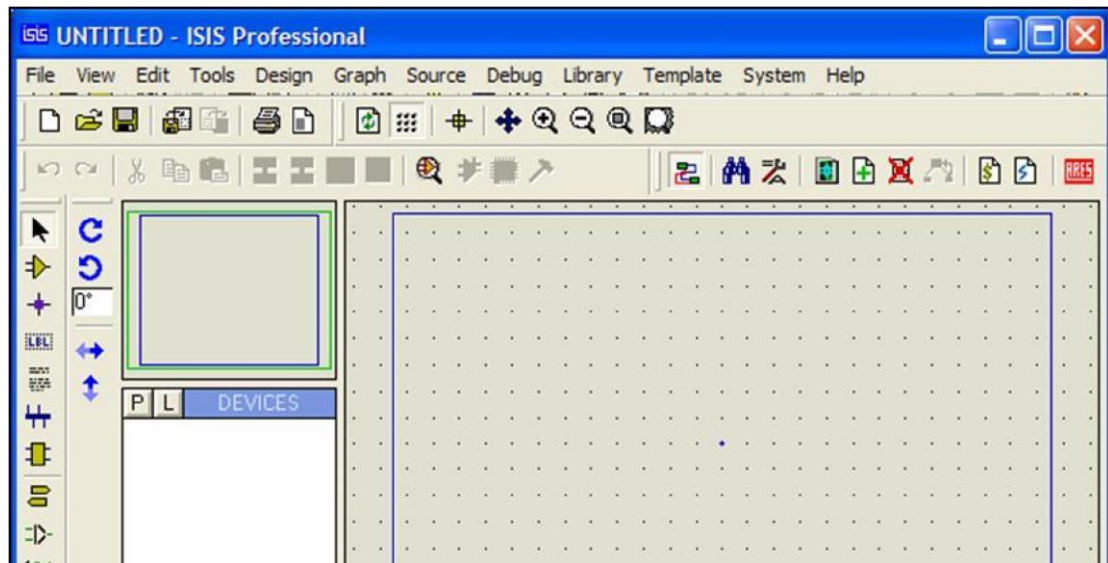

Figure 1: A screen shot of the Proteus IDE

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
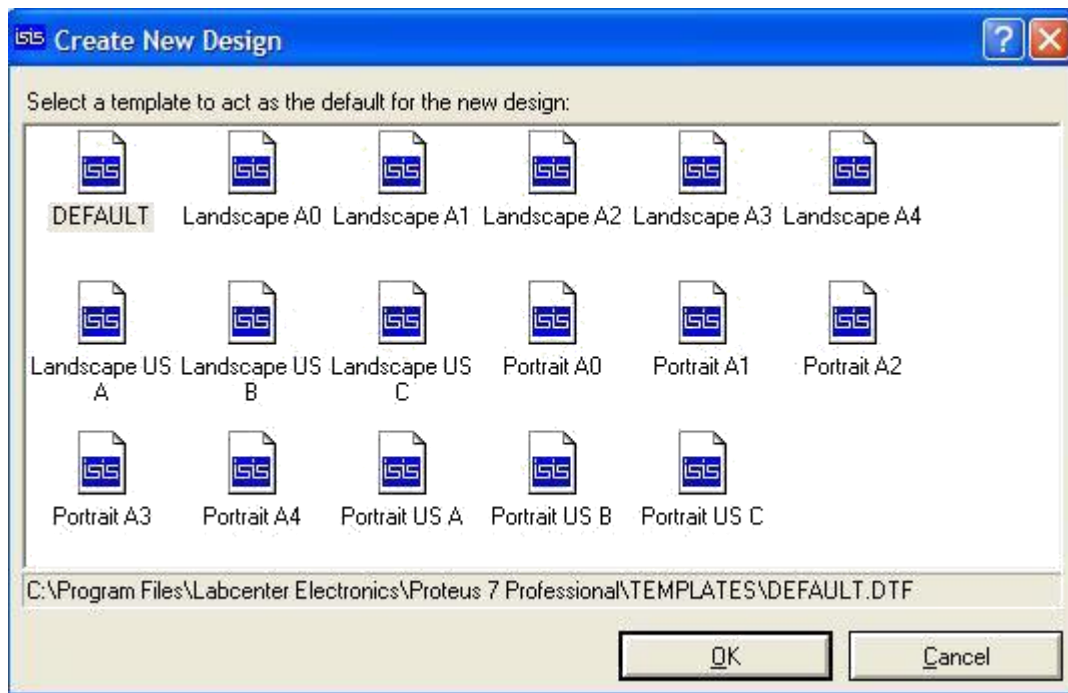

Figure 2

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.
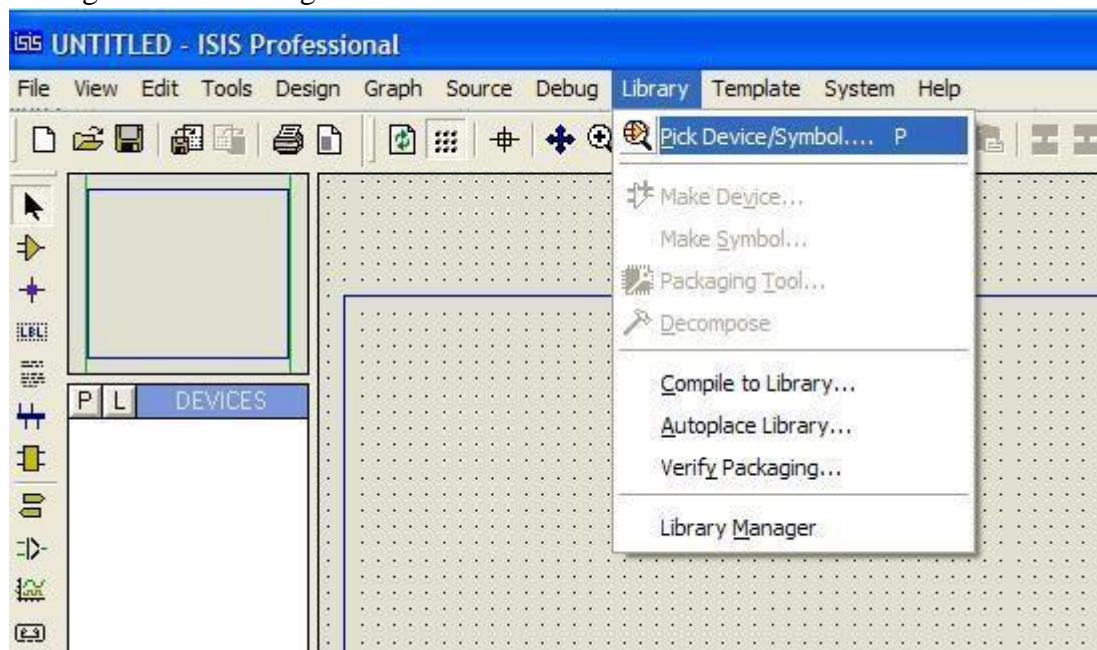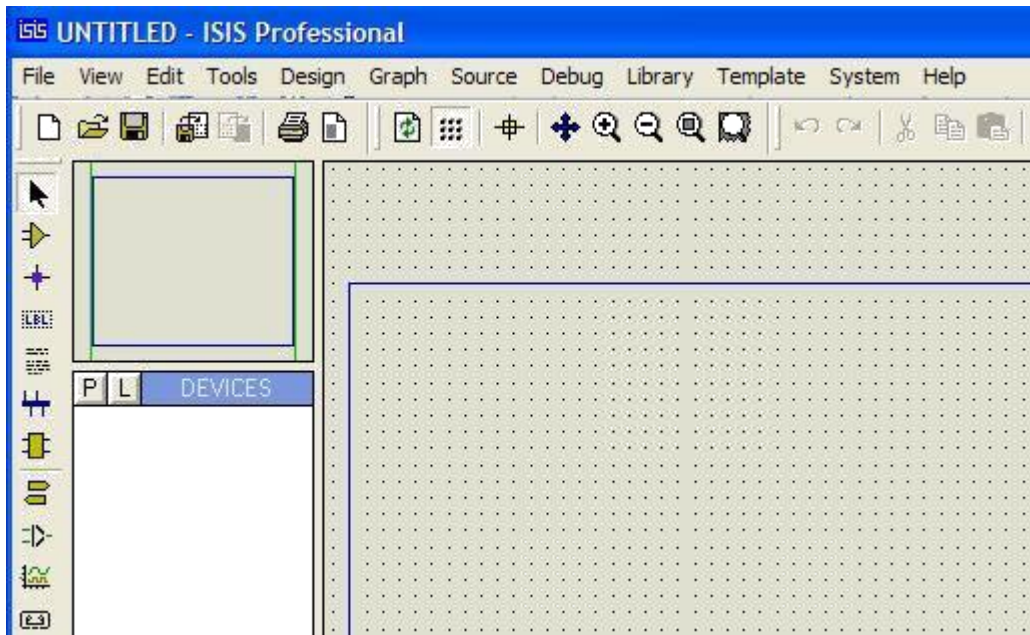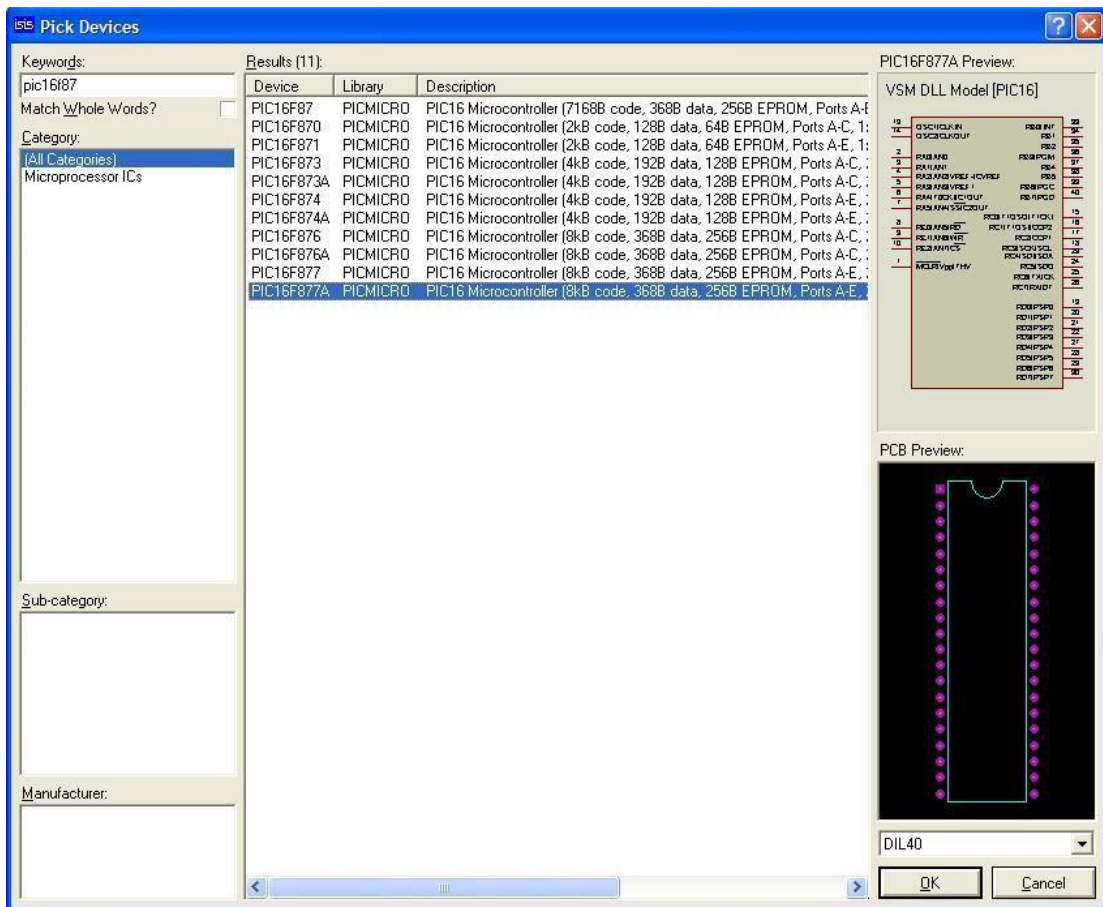

Figure 4

Figure 5

Figure 6

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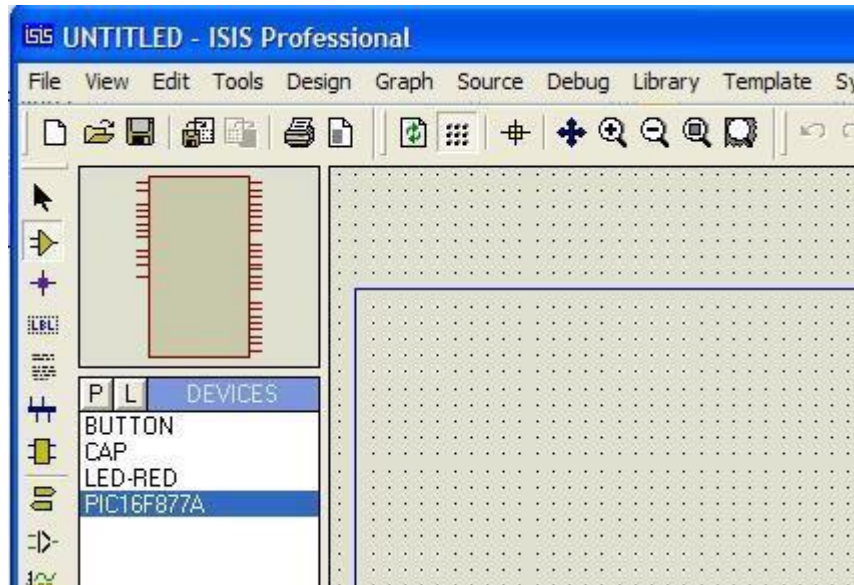

Figure 7

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 (5V).Connect the components to obtain the circuit you need.
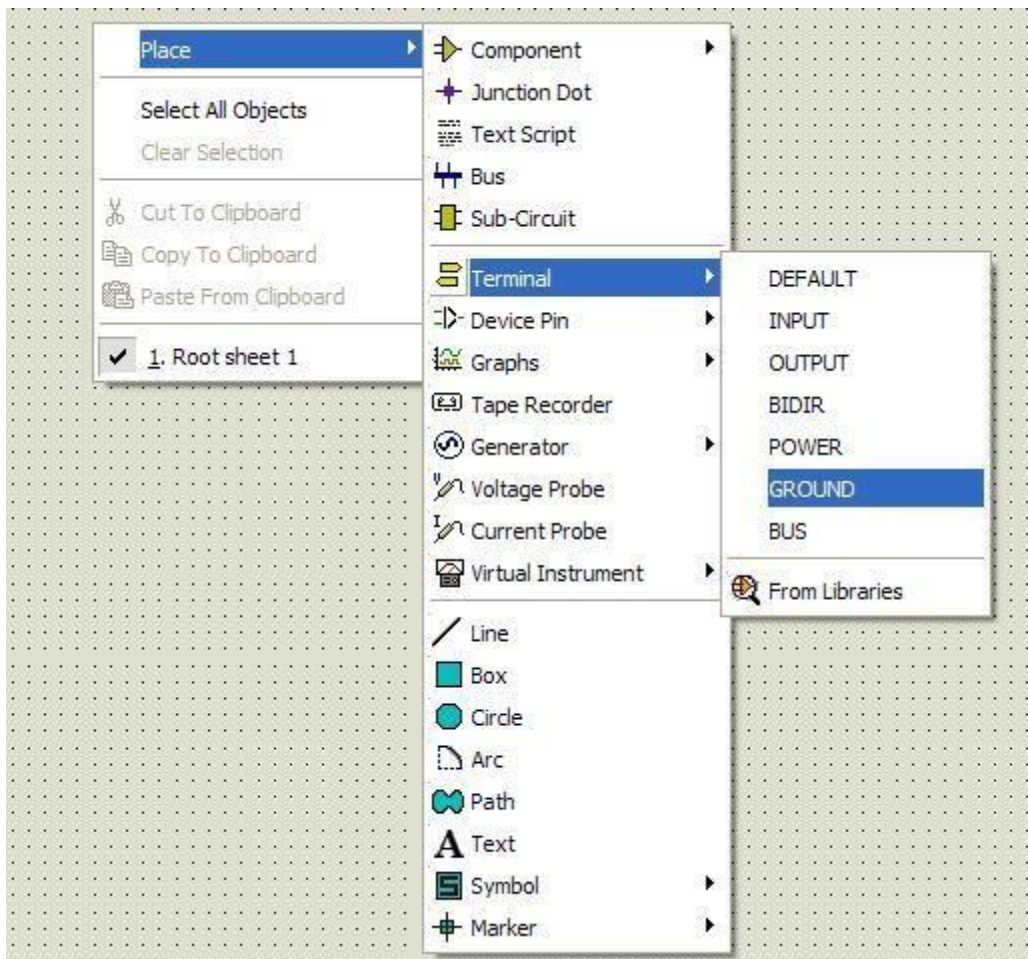
Figure 8

Attaching the Source File

The next stage is to attach the source file to our design in order to successfully simulate the design. We do this through the commands on the Source Menu.

Go to the Source Menu now and select the Add/Remove Source Files Command as shown down.
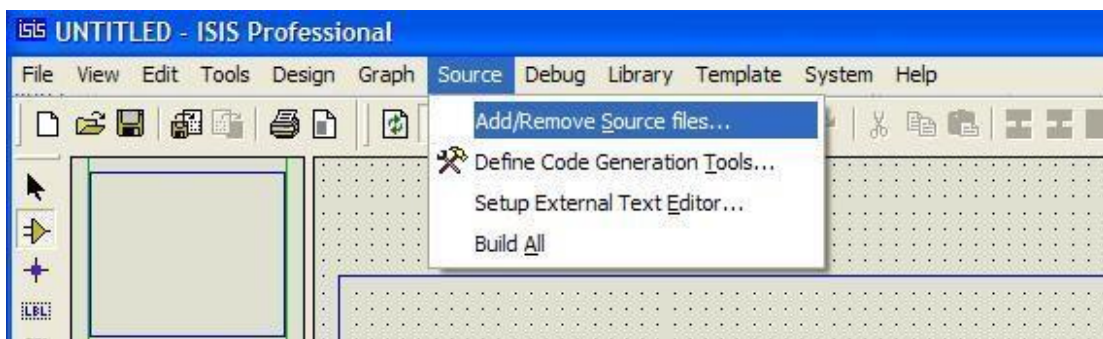

Figure 9

Click on the New button, browse until you reach the desired directory and select your ASM file.
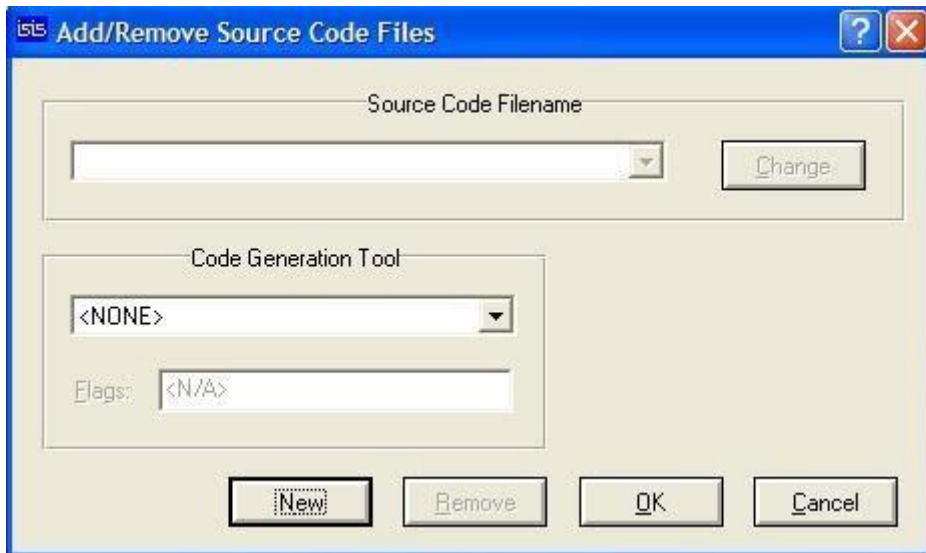
Figure 10

Click open and the file should now appear in the Source Code Filename drop down listbox.
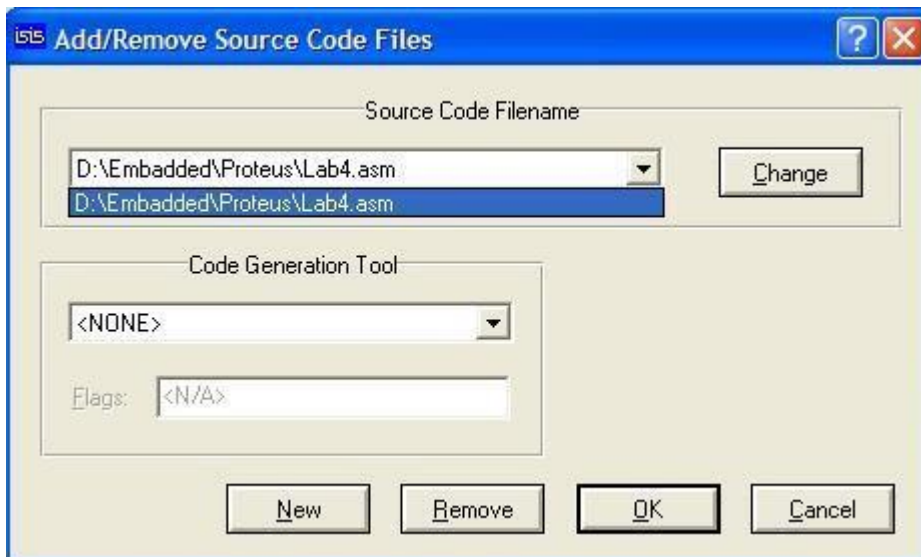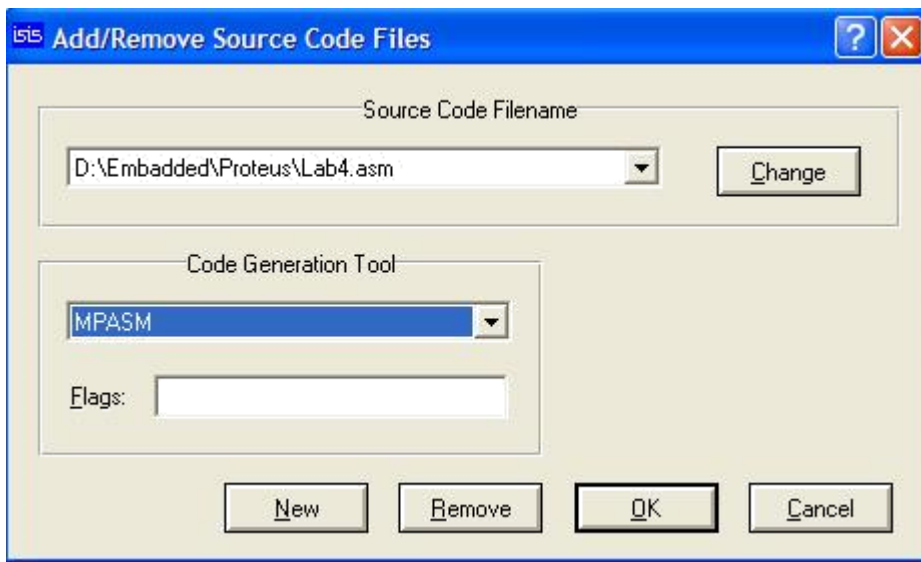

Figure 11

Figure 12

We now need to select the code generation tool for the file. For our purposes the MPASM tool will suffice. This option should be available from the drop down listbox and so left clicking will select it in the usual fashion.

Select the Build All command from the source menu as shown in Figure 13, this provides an excellent way to check that everything has built without errors or warnings.

Figure 13

Finally, 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 and specified which Code Generation Tool will be used.
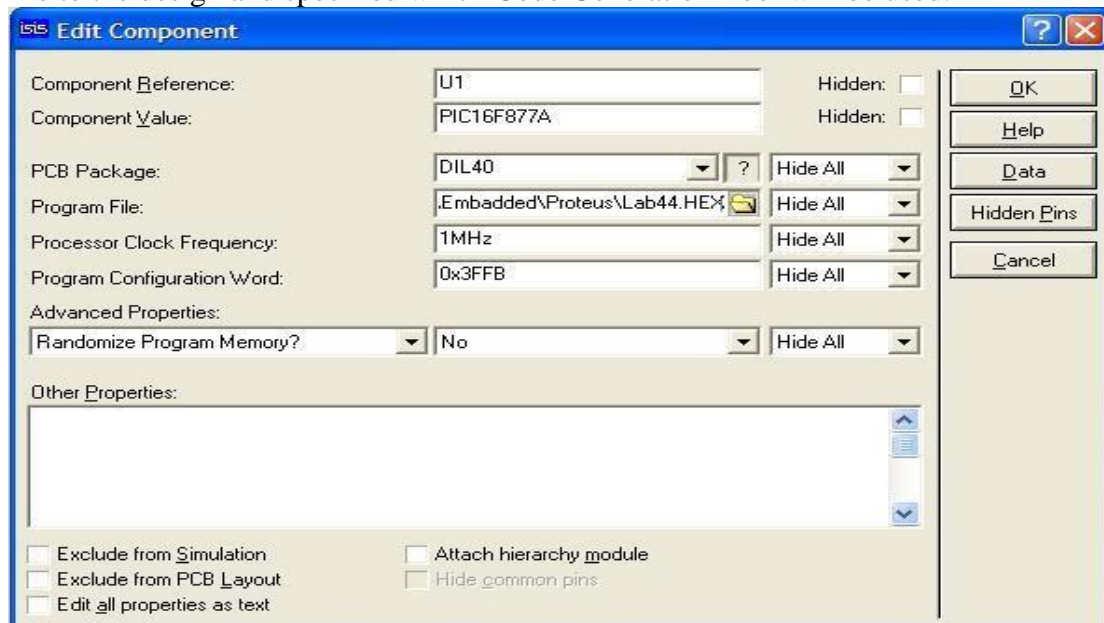

Figure 14

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 1 and click left. The status bar should appear with the time that the animation has been active for.

Working on your Source Code

To edit a source f ile:

- Select the source file it will be Filename.asm from the Source menu. ☐
  ☐ Edit the source file. ☐
- Select the Build All command from the Source menu. ☐

You should see a small window showing details of the compilation process and the following message "Source code build completed OK".
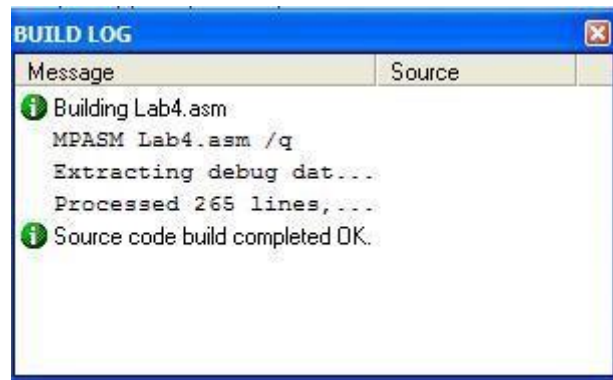

Figure 15