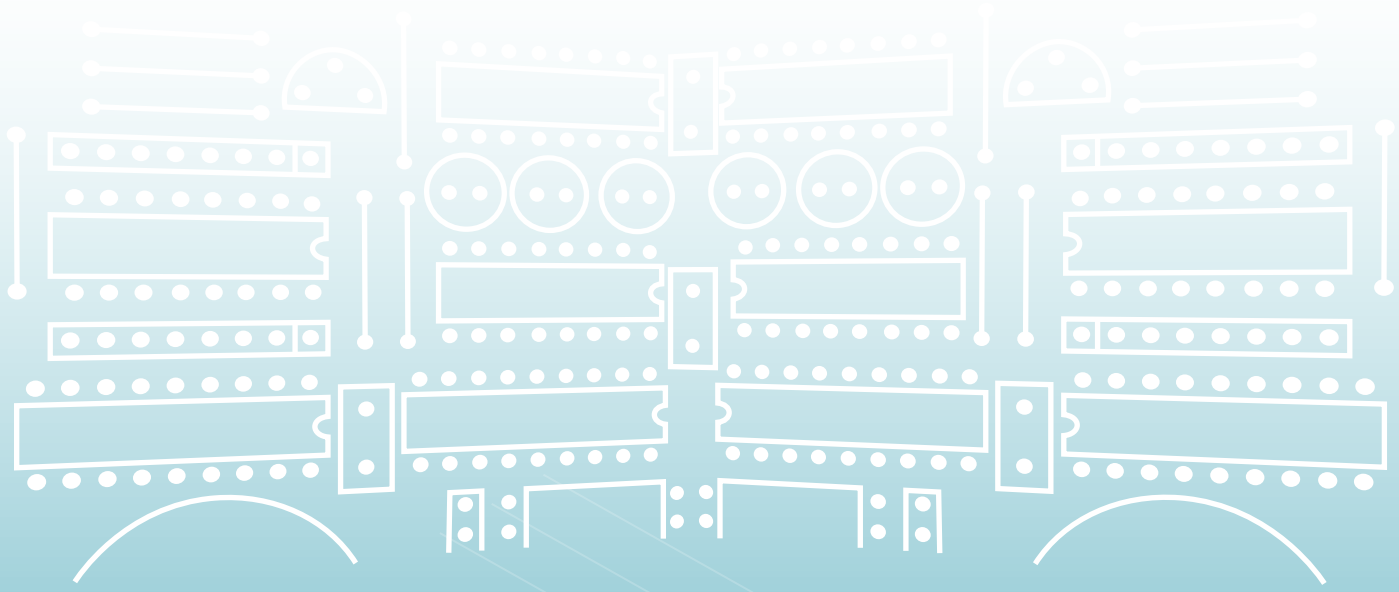
PROTEUS DESIGN SUITE

Getting Started Guide



Labcenter Electronics
www.labcenter.com

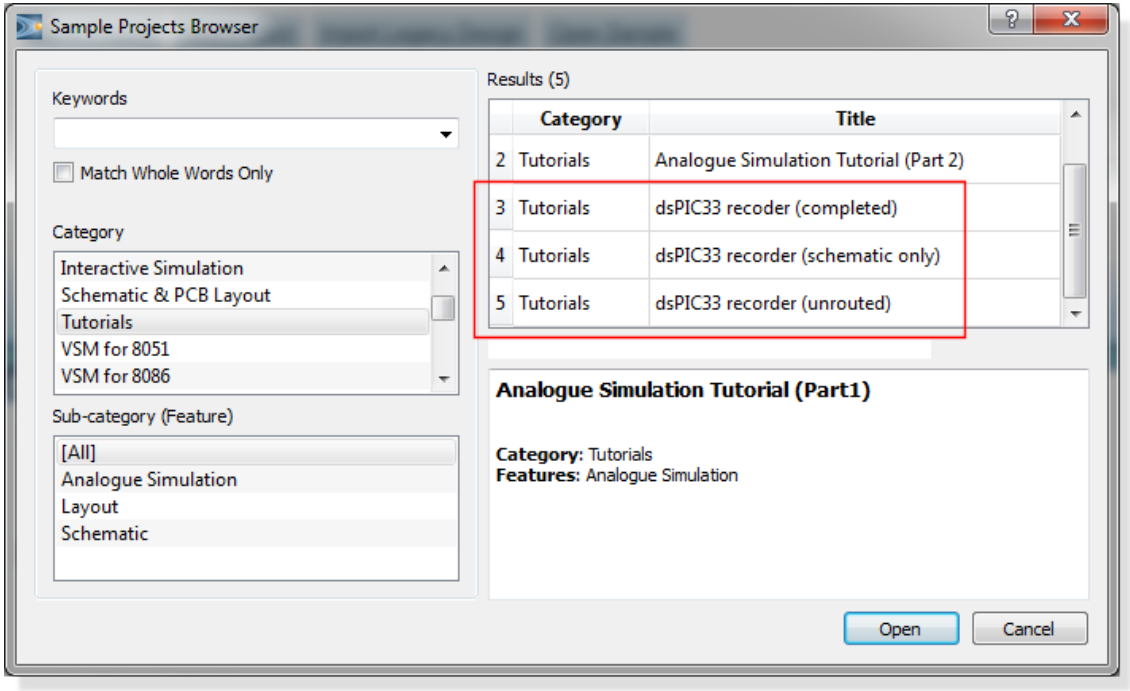
SCHEMATIC CAPTURE TUTORIAL

INTRODUCTION

The aim of this tutorial is to take you through the process of entering a circuit of modest complexity in order to familiarise you with the techniques required to drive the schematic capture module in Proteus. The tutorial starts with the easiest topics such as placing and wiring up components, and then moves on to make use of the more sophisticated editing facilities, such as creating new library parts.

An accompanying tutorial in the PCB module then continues the project development using the completed schematic drawn in this tutorial.

For those who want to see something quickly, DSPIC33_REC_SCHEMATIC.pdsprj contains the completed tutorial circuit but no layout while DSPIC33_REC_UNROUTED.pdsprj and DSPIC33_REC_COMPLETE.pdsprj both contain a completed schematic and a PCB. All of these projects can be loaded from the Open Sample command on the Proteus 8 home page under the tutorials category.



Note that throughout this tutorial (and the documentation as a whole) reference is made to keyboard shortcuts as a method of executing specific commands. The shortcuts specified are the default or system keyboard accelerators as provided when the software is shipped to you. Please be aware that if you have configured

your own keyboard accelerators the shortcuts mentioned may not be valid. Information on configuring your own keyboard shortcuts can be found in the General Concepts section of the Documentation.

Creating a New Project

We shall assume at this point that you have installed the Proteus 8 software package.

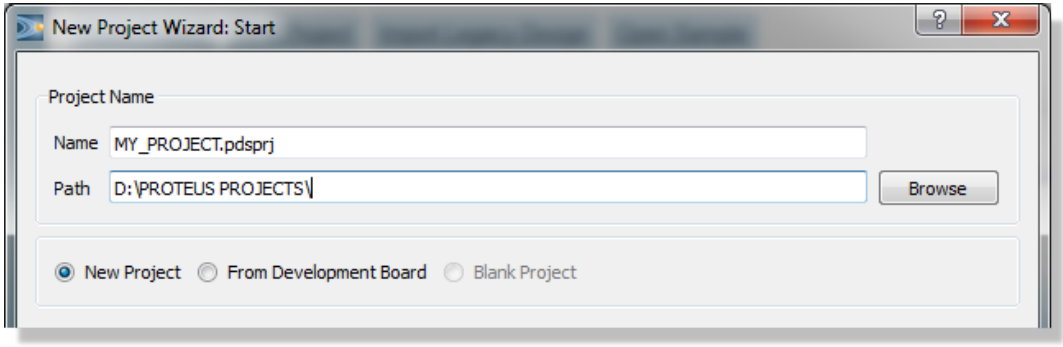
To start the software, click on the Start button and select Programs, Proteus 8 Professional and then the Proteus 8 application. The main application will then load and run and you will be presented with the Proteus home page.

If you have a Demonstration copy of the software you can start the Proteus application via the Proteus 8 Demonstration tab from the Start Menu.

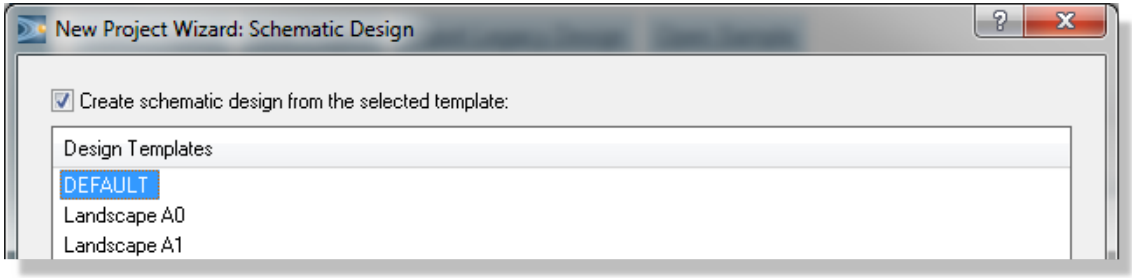
In order to create a schematic we must first create a project. Since this tutorial is partnered with the PCB tutorial we will create a project for schematic/PCB.

Start by pressing the new project button near the top of the home page in Proteus.

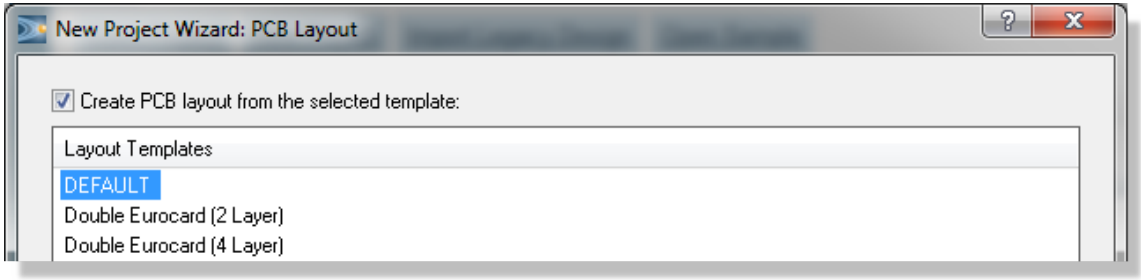
On the first page of the wizard specify a name and path for the project.



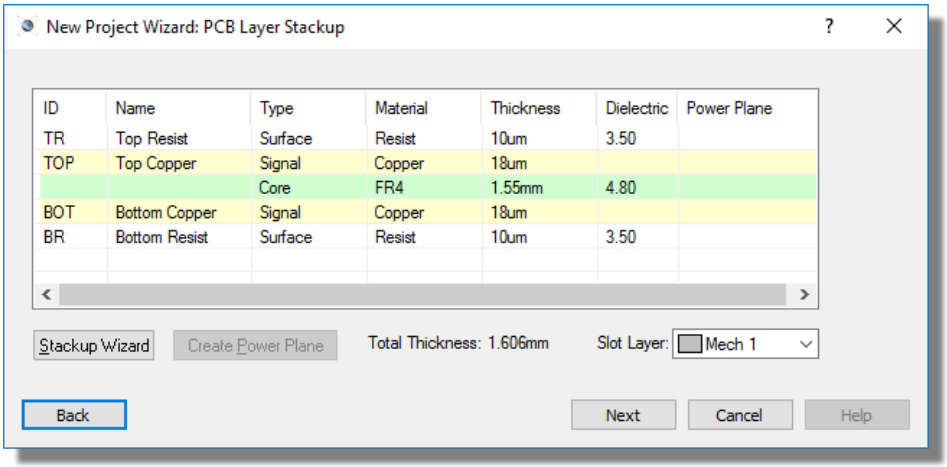
We need a schematic so check the box at the top of the next step and then choose the default template.



Similarly, we need a layout so check the box at the top of the layout page and again choose the default template.

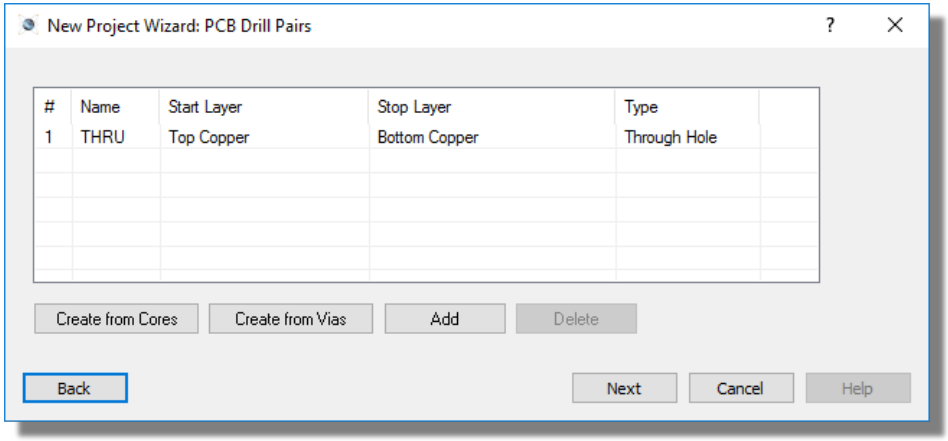


The next screen allows us to define the layer stack for our PCB. Since we will be designing a simple two layer board there is no configuration necessary here.

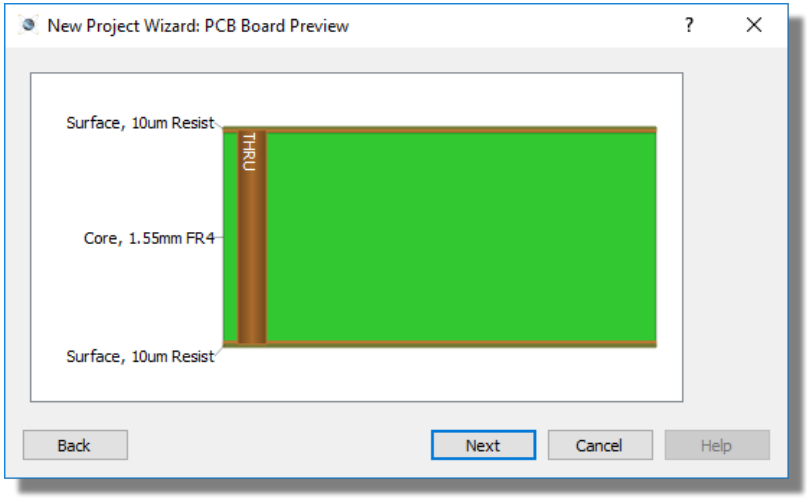


For multi-layer PCB's the stackup wizard button would be used to define the number of copper layers, cores and pre-preg's. This is discussed in more detail in the accompanying PCB tutorial.

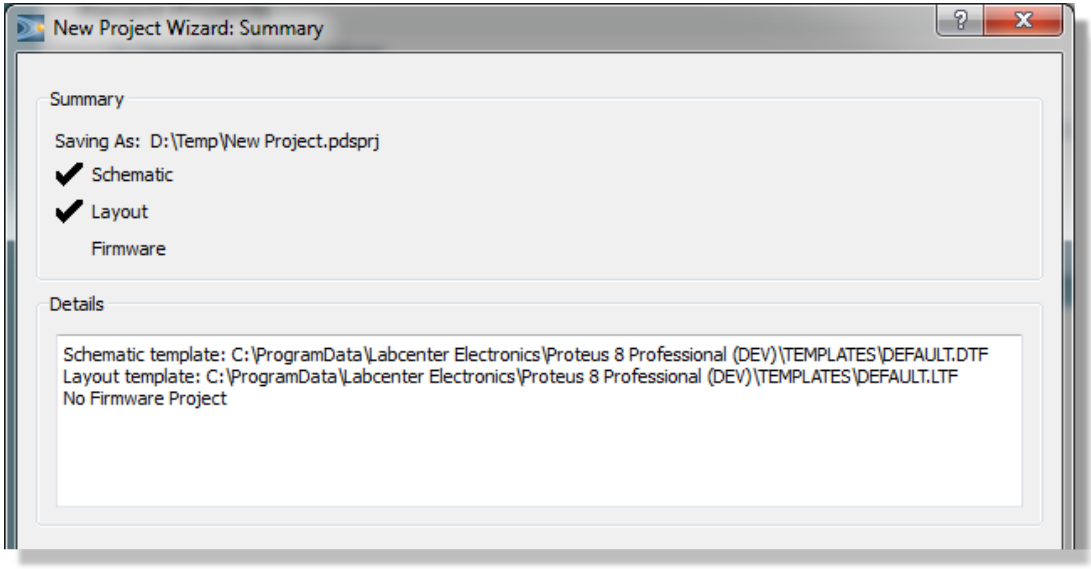
The next screen is for configuration of drill spans. Again, for our proposed 2-layer board the only possibility is thru-hole so there is no action required.



The final screen in the PCB configuration is simply a preview of a PCB cross section that displays visually what has been set up in the previous screens.



We are not simulating the design so leave the firmware page blank and continue on to the summary which should look like the following:



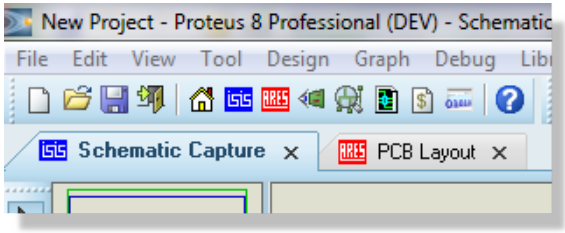
Click on the finish button to create the project.

- A schematic template can contain sheet size, colour scheme, company logo, header block and various other aesthetic presets. Further information can be found in the Templates chapter of the reference manual.

- A PCB template can contain board edge, mounting holes, design rules, layer stack and various other technology information. Refer to the Templates chapter in the PCB documentation for more information.

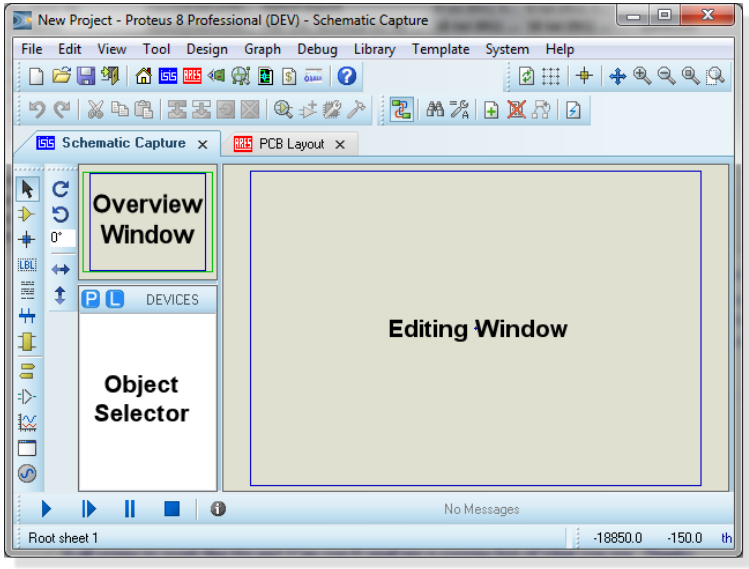
- The configuration of the Layer Stack and Drill Spans is really important for multi-layer PCB's and is discussed in some detail in the reference manual.

The project will open with two tabs, one schematic capture and the other for PCB layout. Click on the schematic tab to bring the Schematic module to the foreground.



Guided Tour

The largest area of the screen is called the Editing Window, and it acts as a window on the drawing - this is where you will place and wire-up components. The smaller area at the top left of the screen is called the Overview Window. In normal use the Overview Window displays, as its name suggests, an overview of the entire drawing - the blue box shows the edge of the current sheet and the green box the area of the sheet currently displayed in the Editing Window. However, when a new object is selected from the Object Selector the Overview Window is used to preview the selected object - this is discussed later.



Schematic Capture Window

If you don't like the default layout of the toolbars you can pick them up and dock them on any of the four sides of the application. Similarly you can move the Object Selector & Overview Window pane across to the right hand side of the application by dragging the end of it all the way across to the other side.

> Toolbars and menu options will switch according to which tab is active (at the front). Throughout this tutorial when we refer to an icon or a menu command we are assuming that the schematic tab is active.

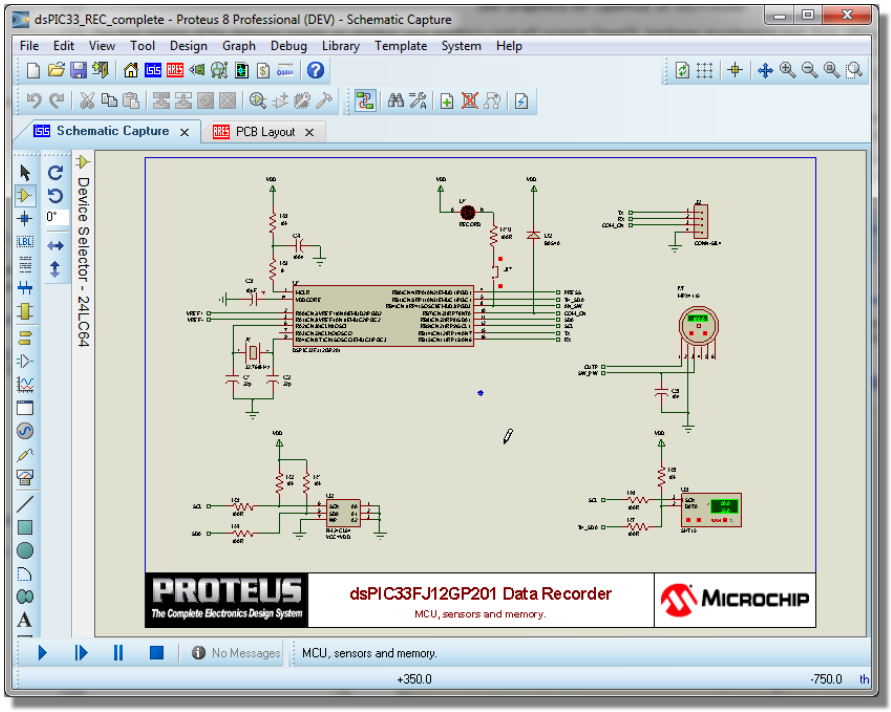
Right clicking the mouse either in the Object Selector or in the Overview Window will provide a context menu, including the option to 'auto hide' the left hand pane. This is extremely useful if you want to maximise the editing area of the application. When enabled the Object Selector and Overview Window will be minimised to a 'flyout bar' at the left (or right) of the application by default and will appear either when the mouse is placed over the bar or when the mode of operation is changed by selecting a different icon.

display issues. We would suggest you being with multi-sampling off if you choose to work in OpenGL mode.

- Configuration of colours and styles in Proteus takes place from the Template Menu. This allows to change everything from paper, grid and highlight colours to the thickness and colour of all the object types used in a design. Please refer to the reference manual section on Templates for more information.

Design Overview

The circuit we are going to draw is shown below. This is reasonably straightforward schematic that will nonetheless allow us to cover most of the major features of the Proteus schematic capture package. Being a real-world design, it will also allow users to follow the design through the PCB phase of development via the PCB tutorial.



The dsPIC33 Data Recorder Sample Design

General Description

The dsPIC33 Data Recorder Tutorial circuit is a solid state recorder designed to collect three of the most important environmental variables:

- Atmospheric pressure
- Ambient temperature

- Relative humidity

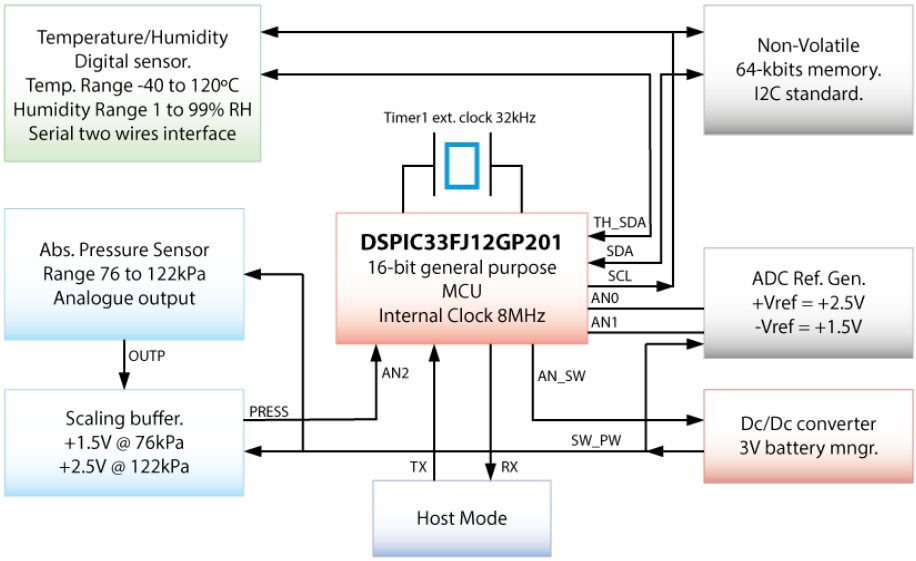
A number of low cost silicon transducers with built-in signal conditioning or digital interfaces have been used for measurement. This reduces the number of components to the minimum required for dsPIC33 interfacing.

All measurements get periodically stored into a non-volatile, low power memory in row binary format for later transfer to a host PC either by using a serial terminal or, more properly, a simple program (not supplied with the project). For the purpose of this project a serial terminal is used.

The circuit is designed to work with a battery for relatively long period in an unattended environment and, as such, a particular emphasis has been given to optimizing the power consumption and consequently the battery life. The recorder will sleep most of the time and will be woken up periodically only to accomplish the required recording operations.

Circuit Description

The following is a block diagram showing the behaviour and interaction of the circuit components.



Block diagram of the dsPIC Data Reorder Design

We'll cover the basics of the design construction as we progress through the tutorial but in order to keep the focus on the practical aspects of using the software, design decisions and theory will not be discussed in this document.

Basic Schematic Entry

We'll start the tutorial by familiarizing ourselves with the basics of schematic design; picking components from the libraries, placing them on the schematic and wiring them together.

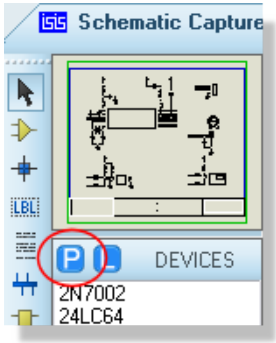
The design in question is relatively large and there is therefore a reasonable amount of drawing involved. We provide a completed schematic at the end of this section so, if you feel that you have mastered the basics at any point, there is no need to continue with drawing the remainder of the circuitry. We do however urge you to read through the full contents of the documentation as we introduce important features throughout.

The first thing we need to do is to get the parts from the libraries that we need in our schematic.

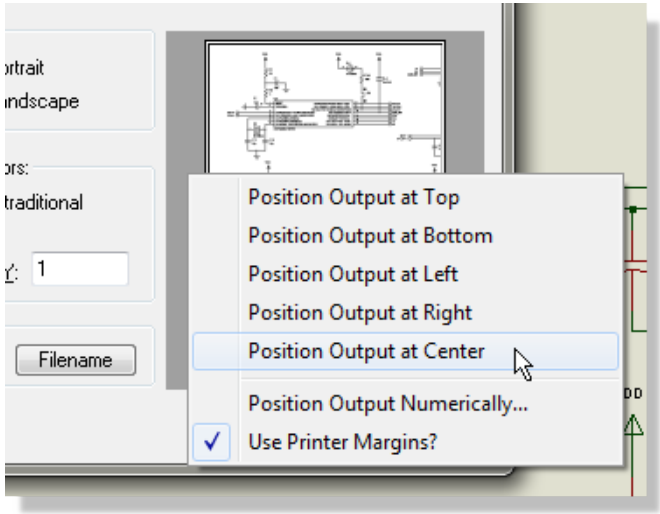
Selecting Parts from the Library

You can select parts from the library in one of two ways:

- Click on the P button at the top left of the Object Selector as shown below. You can also use the Browse Library icon on the keyboard shortcut for this command (by default this is the P key on the keyboard).



- Right click the mouse on an empty area of the schematic and select Place – Component - From Libraries from the resulting context menu as shown below:



You can also export the schematic directly in PDF format from the Output Menu (no driver installation required).

The current schematic is now ready for board layout so if you have purchased a Proteus PCB Design package you can move straight onto the accompanying tutorial booklet for PCB Layout.

Library Parts

Introduction

The Proteus Design Suite comes with a significant installed base of schematic components, almost all of which are already packaged with the correct PCB footprint. However, with the number of new parts entering the market every day and the vast choice available to engineers it is inevitable that users will have to either import library parts into Proteus or create library parts inside Proteus.

Importing Library Parts

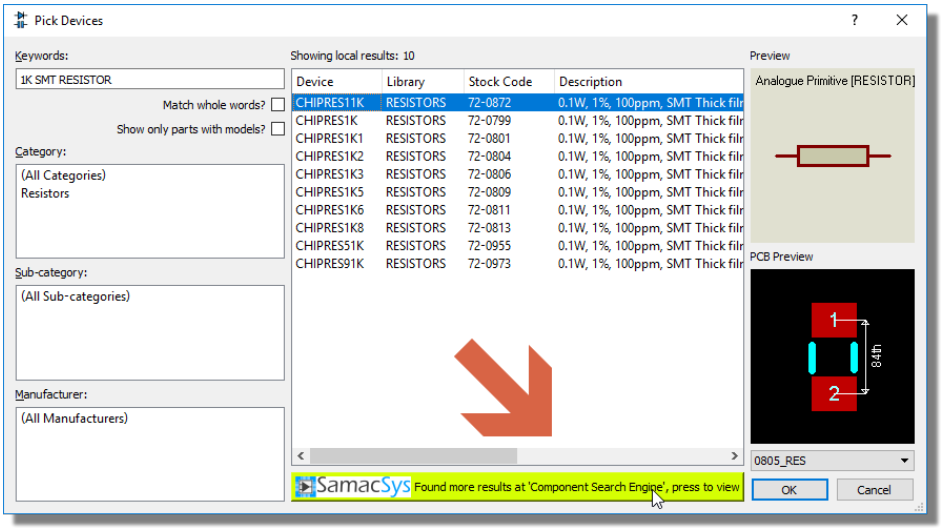
Importing Library Parts is by far the preferred option. It is both much faster to do, is far less error prone and you can import both the schematic part and the layout footprint at the same time. Proteus supports two ways to import parts:

- Integrated Web Search and Import (Requires a valid USC)
- Manual Library Part Import Dialogue.

Short tutorial videos on working with both methods can be found on our Youtube Channel

Web Search Import

The integrated web search import works directly from the library picker dialogue. You simply type the part name you want and, after searching the installed libraries, you can search a database of over 15 million parts from our partner Samacsys. This service requires a free account to be registered with Samacys and a valid USC to import the parts.



Library Part Import Dialogue

This method enables you to manually import via the Import Part command on the Library menu in either the schematic of the layout editors. This import will work with all major vendor files such as those generated by Ultra-Librarian, Samacys, SnapEDA and PCB Library Expert as well as through common supplier portals such as Digikey or RS Components. It does not require a valid maintenance contract to work but does involve slightly more effort.

PCB LAYOUT TUTORIAL

Introduction

The purpose of this tutorial is to familiarize you as quickly as possible with the main features of the PCB Layout, to the point that you can use the package for real work. Users with modest computer literacy should find it possible to learn the package and produce their first board within a day or two.

The tutorial proceeds by taking you through worked examples involving all the important aspects of the package including:

- Basic techniques for placement and routing.
- 3D Board Visualisation.
- Netlist based design including both manual and automatic routing.

More advanced editing techniques such as block editing and route editing.

- Report generation.
- Hard copy generation.
- Library part creation.

We do urge you to work right the way through the tutorial exercises as many things are pointed out that if missed will result in much wasted time in the long run. Also, having worked through the tutorial and thus got a basic grasp of the concepts behind the package you will find it much easier to absorb the material presented in the reference chapters.

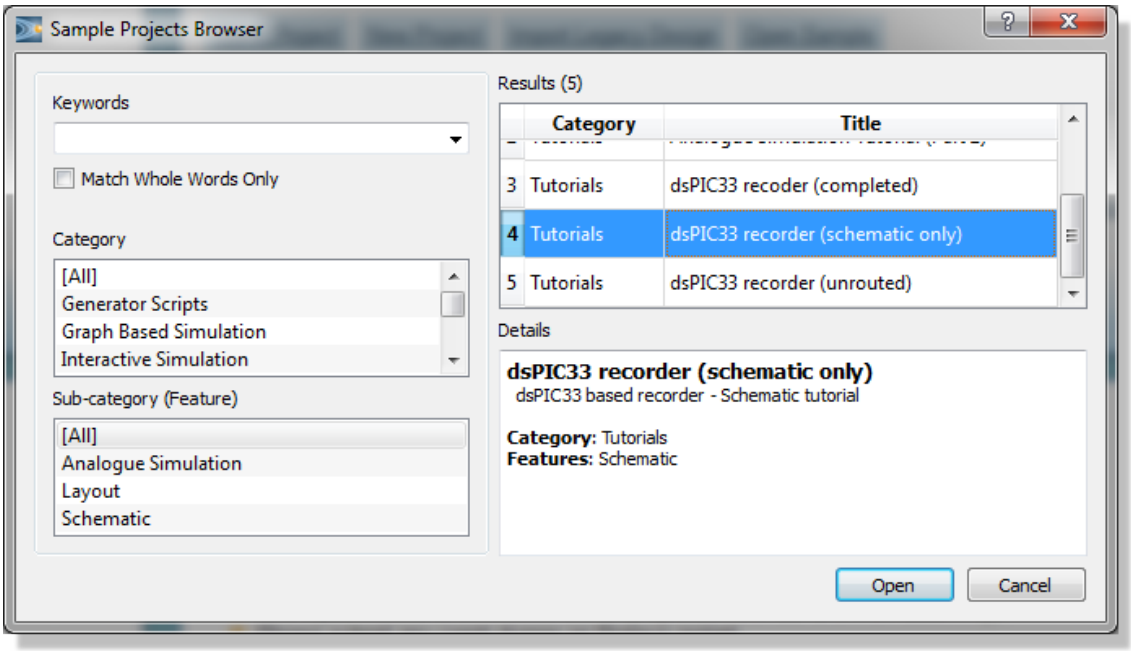
> Note that throughout this tutorial (and the documentation as a whole) reference is made to keyboard shortcuts as a method of executing specific commands. The shortcuts specified are the default or system keyboard accelerators as provided when the software is shipped to you. Please be aware that if you have configured your own keyboard accelerators the shortcuts mentioned may not be valid. You can configure your own keyboard shortcuts via the System - Set Keyboard Mapping command.

> Menus toolbars and icons all switch when you change between tabs to reflect the functionality of the module you are working on. When we talk about menu commands or icons in this tutorial we assume that the PCB layout tab is selected. The menu contents will be very different if the schematic tab is selected!

Overview of the Layout Editor

We shall assume at this point that you have installed the package, and that the current directory is some convenient work area on your hard disk. This tutorial is a direct continuation of project we started in the schematic capture tutorial so we will start by loading the project file with the completed schematic.

From the home page in Proteus click on the Open Sample button, filter to the tutorials category and then select the dsPIC33 Recorder (schematic only) sample.



Opening a Tutorial sample design

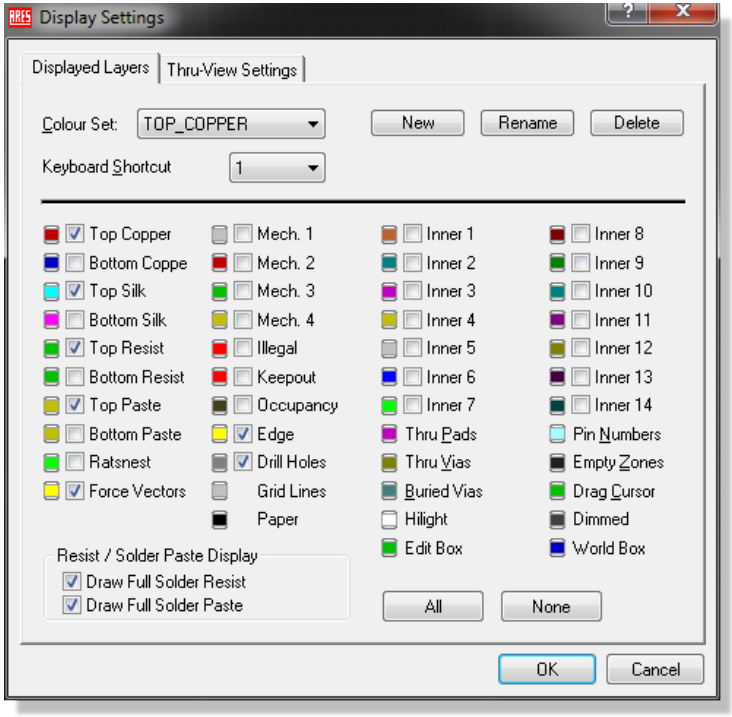
This will open the tutorial project with the completed schematic from the Schematic Capture tutorial. We can start the PCB layout module from the application module toolbar at the top of the Proteus application.

If you are working on two monitors or have free screen space you can drag and drop one of the tabs to view both modules simultaneously. If not, you can switch between the tabs with the mouse or via the standard windows CTRL+TAB shortcut key.

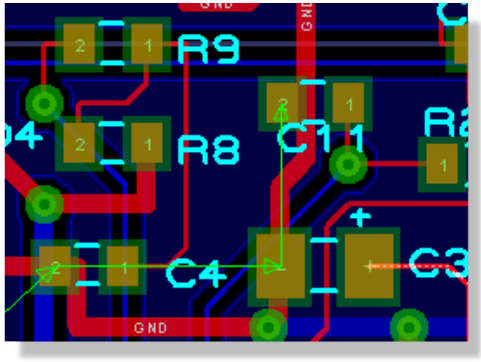
Much of the look and feel of the application is similar to Proteus and hopefully now familiar, although there are some important differences.

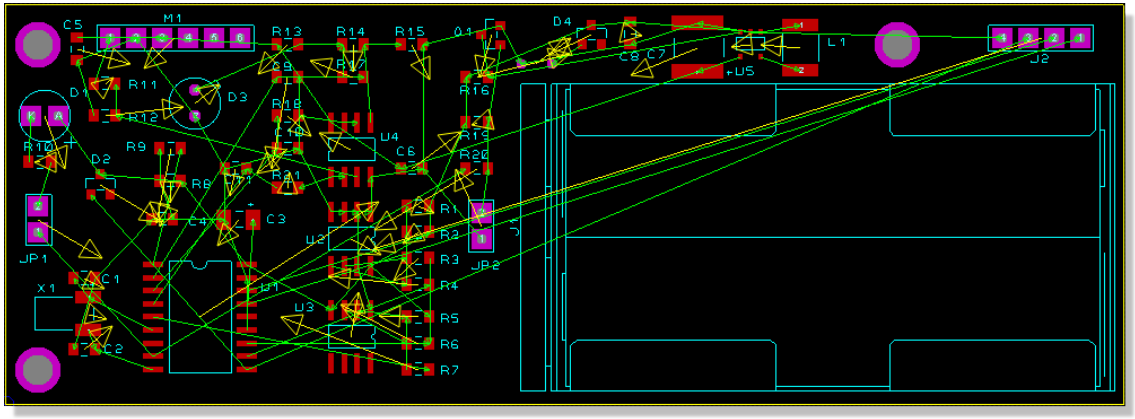
The Main Window

The largest area of the screen is called the Editing Window, and it acts as a window on the drawing - this is where you will place and track the board. The smaller area at the top left of the screen is called the Overview Window. In normal use the Overview Window displays, as its name suggests, an overview of the entire drawing - the blue box shows the edge of the current sheet and the green box the area of the sheet currently displayed in the Editing Window.



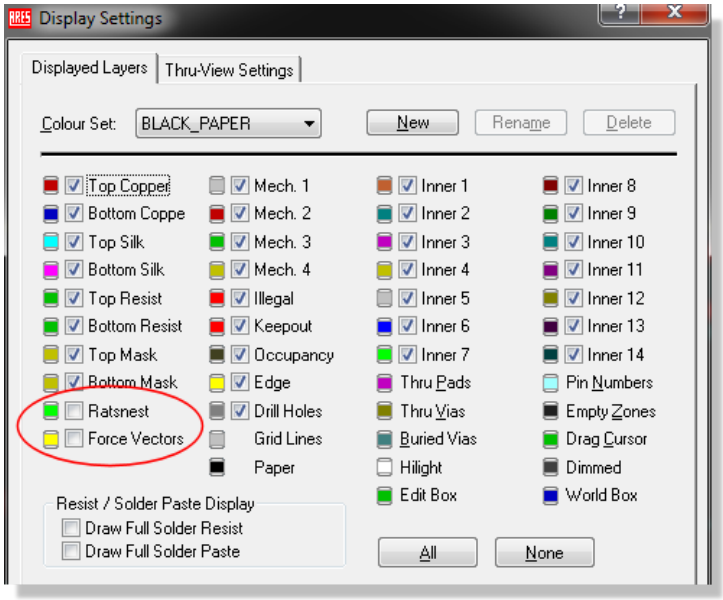
Finally, if you are working in OpenGL or Direct2D mode, the Resist/Paste Mask Display options allow you turn on full display of these layers on the board, showing the resist and paste coverage around pads and vias. When enabled you can change the intensity of these layers by switching to the Thru-View settings tab and adjusting the appropriate slider controls.





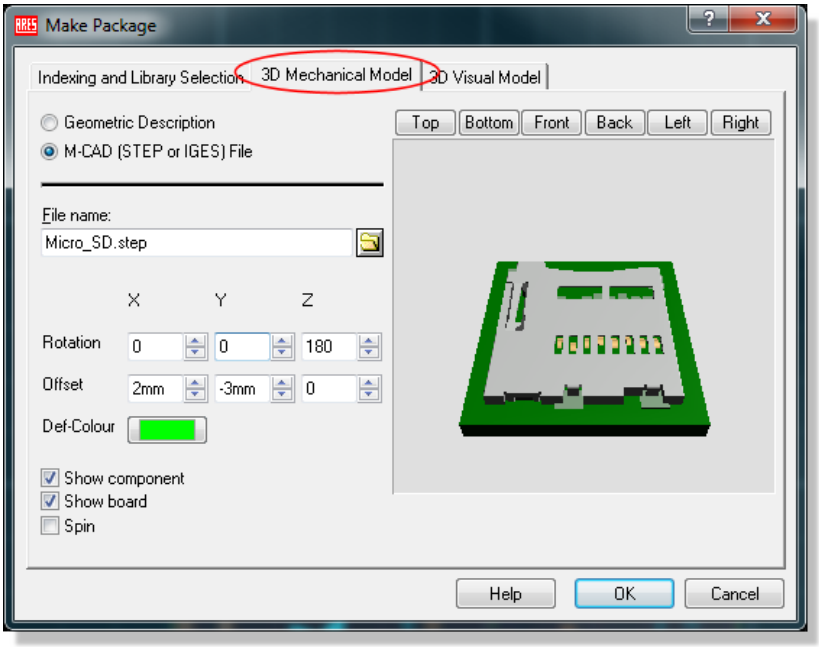
dsPIC33 board as you see it having placed all the components

From the View Menu, select the Edit Layer Colours/Visibility command. The resulting dialogue form shows all the layers in the layout with colour and visibility configuration options. All we need do here is deselect the checkbox for the 'Vectors and Ratsnest' layer and exit the dialogue.



The Ratsnest and Force Vector layers turned off

- You can also launch this dialogue form by clicking the mouse on the status reporting bar at the bottom of the application.
- It is important to remember that this dialogue form controls visibility only; to control whether objects on a layer were selectable/editable we would use the Selection Filter which is discussed in more detail later in the documentation



4) Type the name of the STEP file and then the rotation and offset required to align the STEP file with the pads on the board.

For geometric models, the difference is basically that you describe the part in terms of size and shape via a simple scripting language. This is covered in the reference manual.

Live Update

Like any other module in Proteus, the 3D Viewer will update live as changes are made to the PCB layout module. In practice, the usefulness of this will depend very much on the power of your machine (number of cores, memory, etc.) and the complexity of the board.

On a board of modest complexity and a modern computer the redraw time is not significant and - particularly on split frames - is very useful for movement and positioning.

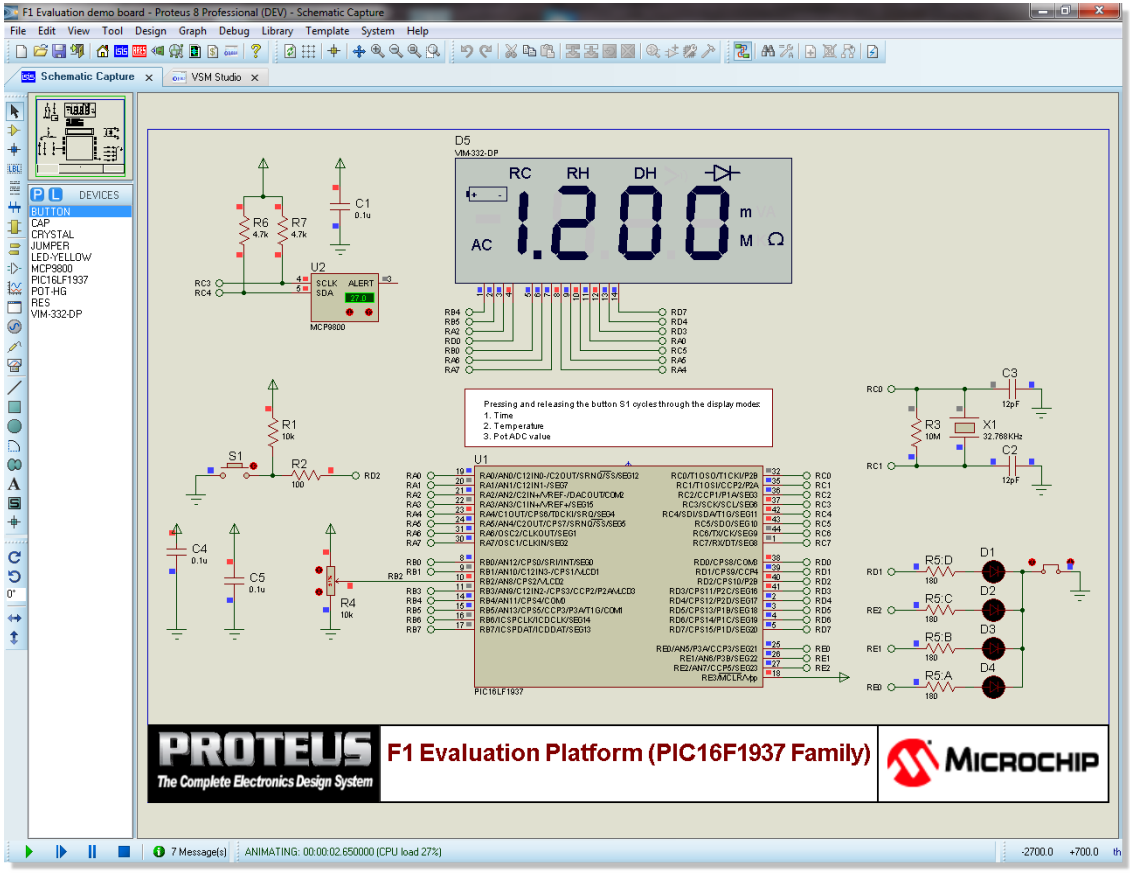
More information on 3D Visualization including creation your own 3D models, exporting 3D STEP Files for MCAD import, customizations and applying 3D data to legacy designs can be found in the 3D Viewer section in the online reference manual.

VSM TUTORIAL (Interactive)

Introduction – Interactive Simulation

The purpose of this tutorial is to show you how to conduct an interactive simulation with a microcontroller using Proteus VSM and the VSM Studio IDE. The emphasis will be on practical usage of the simulator and IDE, with more detailed coverage of each topic being available in the reference manuals. This tutorial does not cover schematic entry; if you are not familiar with drawing in Proteus then you should take the time to work through the tutorial content in the Proteus reference manual.

We will use a pre-drawn schematic of the Microchip F1 Evaluation Board as shown below.



Starting from the basics of driving the simulation from the VSM Studio IDE we will then look at some of the various debugging and measurement tools available inside the Proteus VSM software.

```
switch(display_mode)
{
    case MODE_TIME:
        display_time();
        break;
    case MODE_TEMPERATURE:
        display_temp(mcp9800_get_temp());
        break;
    case MODE_POT:
        display_pot(input_pot());
        break;
    case MODE_TEST:
        display_test();
        break;
    default: display_mode = MODE_TIME; break;
}
```

The last step is to write a function that does something when we enter our mode of operation. We'll add this at the bottom of the file.

```
void display_test(void)
{// Do Something...
}
```

You can experiment and add anything you like here but we'll do something simple, writing a value to the display and adding a little binary counter on the LED's.

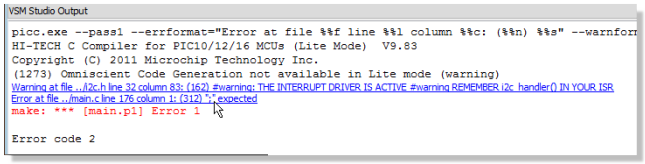
```
void display_test (void)
{ static int i,d;

  display_int(1111);
  if (++d > 400)
  { d = 0;
    PORTE=i;
    i = (i+1) % 8;
  }
}
```

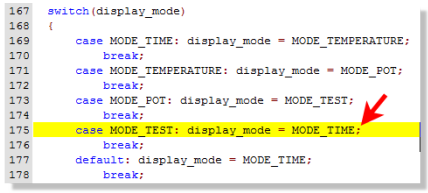
Since we are using PORTE to write the LED's here we also need to configure the port pins. We'll set up ANSELE in the peripheral config section at the top of the main function.

```
    ANSA1 = 1;
    ANSA3 = 1;
    ANSB3 = 1;
    ANSB1 = 1;
    ANSELE= 0;
```

The final step is to build the project and launch Proteus as discussed before. If you have made a mistake (as we have here) you will get a compiler error in the output window. Clicking on this

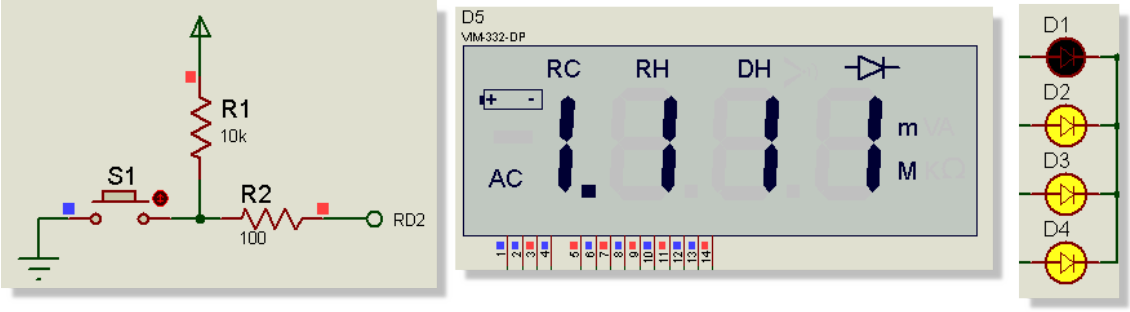
error will navigate to the problem. In our case we have missed a semi-colon from the end of the line.





Some errors - such as linker errors - will obviously not be navigable. In these cases you may need to change the options/includes via the project settings command.

When we run the simulation this time we can use the button to cycle through the modes of operation until our own is reached (time -> temperature -> test). We should see our displayed number on the display at this point and a byte cycle count on the LED's



Again, press the stop button on the animation control panel to stop the simulation.

This example is a little contrived and not terribly exciting but it does show how you can quickly write and simulate code on virtual hardware.

Active Popups

So far, we have been writing our code in the VSM Studio tab and simulating on the schematic capture tab. There is nothing wrong with this and it can work very well on two monitors if you drag one of the tabs onto another screen.

When debugging however, you are typically more interested in stepping the code and looking only at small sections of the board for verification. The active popup system in Proteus is designed to do exactly that - bringing defined sections of the schematic into VSM Studio during simulation.

In our case, the temperature sensor is a good example of something we might want to see and interact with during debugging. To add it as an active popup:

1) Switch to the Proteus tab and selecting the Active Popup icon.