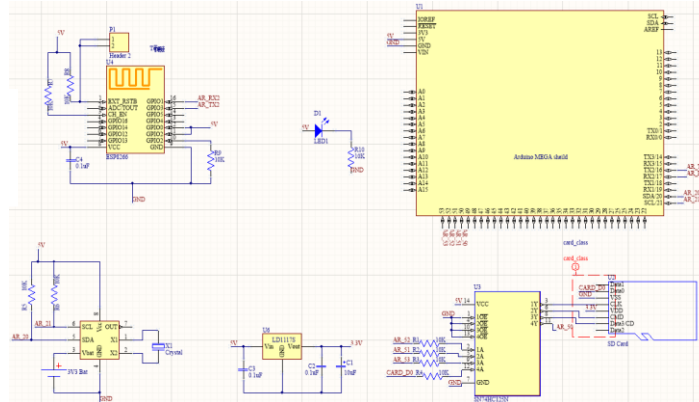


LAB #2

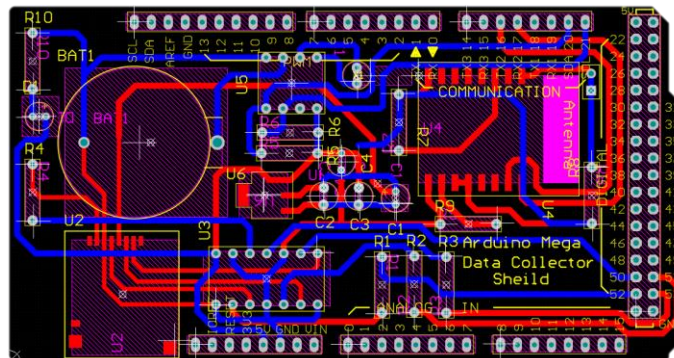
PCB Layout Basics

(Using Proteus Design Suite)

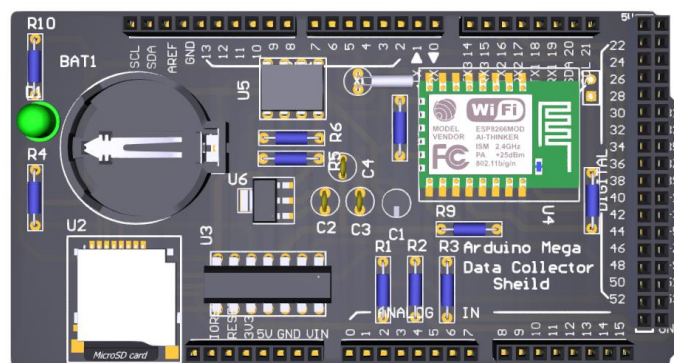
Schematic Capture



Components Placement
& Routing



Final Board



Contents

Objective	3
Equipments:.....	3
Introduction:	3
Basic Concepts:	3
Pcb Cores And Layer Stack-Up.....	4
Pcb Fabrication Process	5
Function Of Proteus Design Suite In The Pcb Design Process	7
Common Terms Used In Pcb Design:	7
Footprint (Land Pattern):	7
Via:	8
Trace:	8
Overview Of The Design Flow	8
Introducing Proteus Design Suite:.....	11
Schematic Capture Basics:	11
Guided Tour Of Isis Window:	14
Zooming:	14
Panning	15
Visual Effects:.....	15
Basic Schematic Entry:.....	16
Placing Objects On The Schematic:	19
Wiring Up:	21
Making Connections With Terminals	22
Part Labels And Annotation:.....	23
Preparing For Pcb Layout:	24
Packaging Considerations:.....	24
To Change A Package On A Component.....	25
Design Verification:	28
Bill Of Materials.....	29
Task #1:.....	29
Pcb Layout Basics:	30
The Main Window	30
The Control Bar	30
Component Placement.....	32

The Board Edge.....	33
Placing Components And The Rastnest.....	34
Routing The Board	36
Placing A Route Manually	36
3d Visualization.....	39
Basic Navigation.....	39
Appendix: Creating New Packages	40
Drawing The Footprint.....	40
Packaging The Footprint.....	45

OBJECTIVE:

- To introduce students with PCB design process.
- To get familiar with Proteus as a professional PCB design CAD tool.
- To design and Layout a simple audio amplifier board.

EQUIPMENTS:

- Computer with Proteus 8.8 or Later
- 5cm x 5cm single layer PCB board
- 1 LM386 audio Amplifier Board
- 1 Potentiometer (10 Kohm)
- 10 Capacitors
- 10 Resistors
- 2 Audio Plug
- 1 DC power jack
- 12V Power supply

INTRODUCTION:

Printed Circuit Boards (PCBs) are found everywhere around us, any electronic device such as your laptop or mobile must have at least one PCB. Therefore, it is clear that schematic creation and PCB layout should be an essential skill for an electrical engineer. In this lab, we will introduce the PCB Design Flow by an example. We will try to schematic capture and PCB layout a simple audio power amplifier. Throughout this lab, Proteus Design Suite is introduced as a CAD tool for schematic creation and PCB layout.

BASIC CONCEPTS:

A PCB consists of two basic parts: a substrate and printed wires (copper traces). The substrate provides a structure that physically holds the circuit components and traces in place and provides electrical insulation between conductive parts. A common type of substrate is FR4, which is a fiberglass/epoxy laminate.

PCB Cores and Layer stack-up

During manufacturing the PCB starts out as a copper clad substrate as shown in Figure 1-1

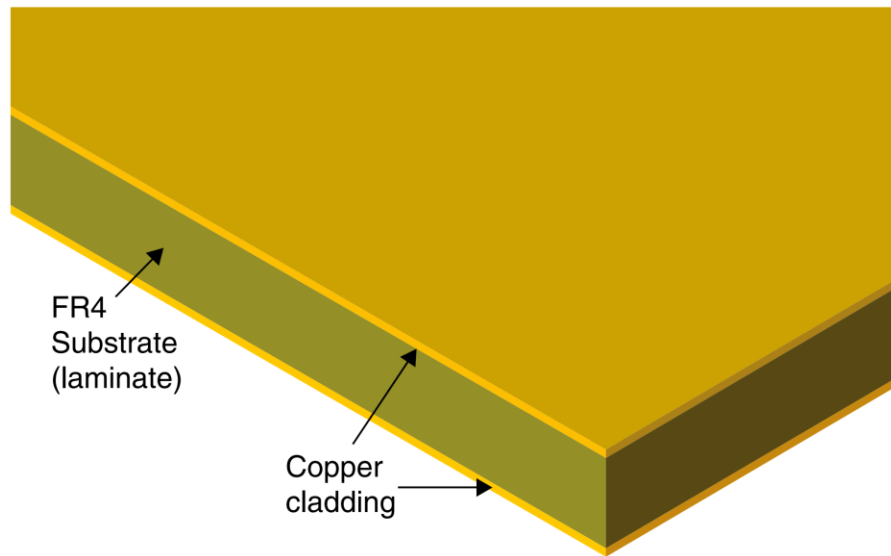


Figure1- 1

A substrate can have copper on one or both sides. Multilayer boards are made up of one or more substrates called cores. Figure 1-2 shows a cross-section of complex PCB stack.

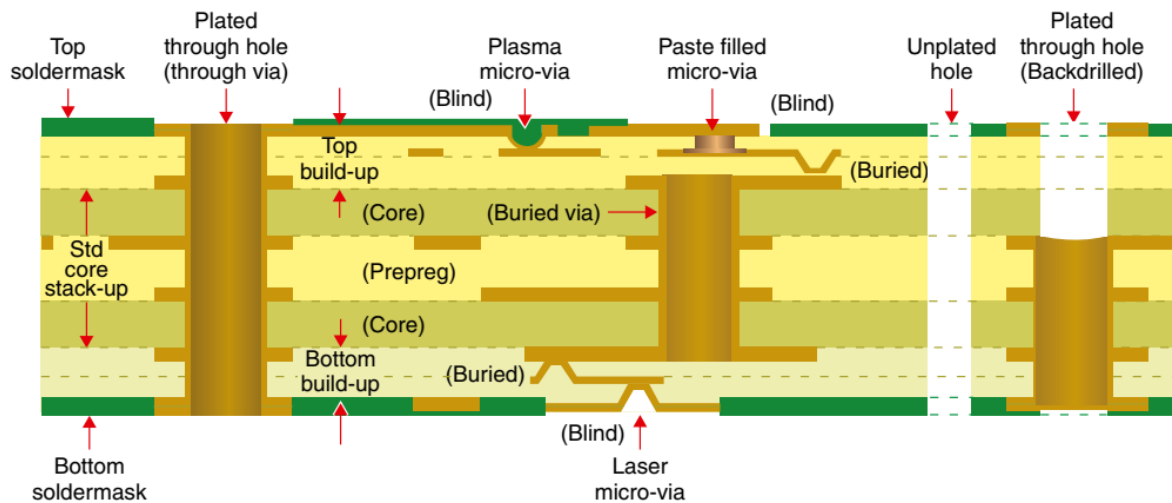


Figure1- 2

PCB fabrication process

The copper traces and pads seen on a PCB are produced by selectively removing the copper cladding. Two methods are commonly used for removing the unwanted copper: **wet acid etching** and **mechanical milling**. Figure1-3 shows the basic steps of acid etching process while figure 1-4 shows **mechanical milling process**.

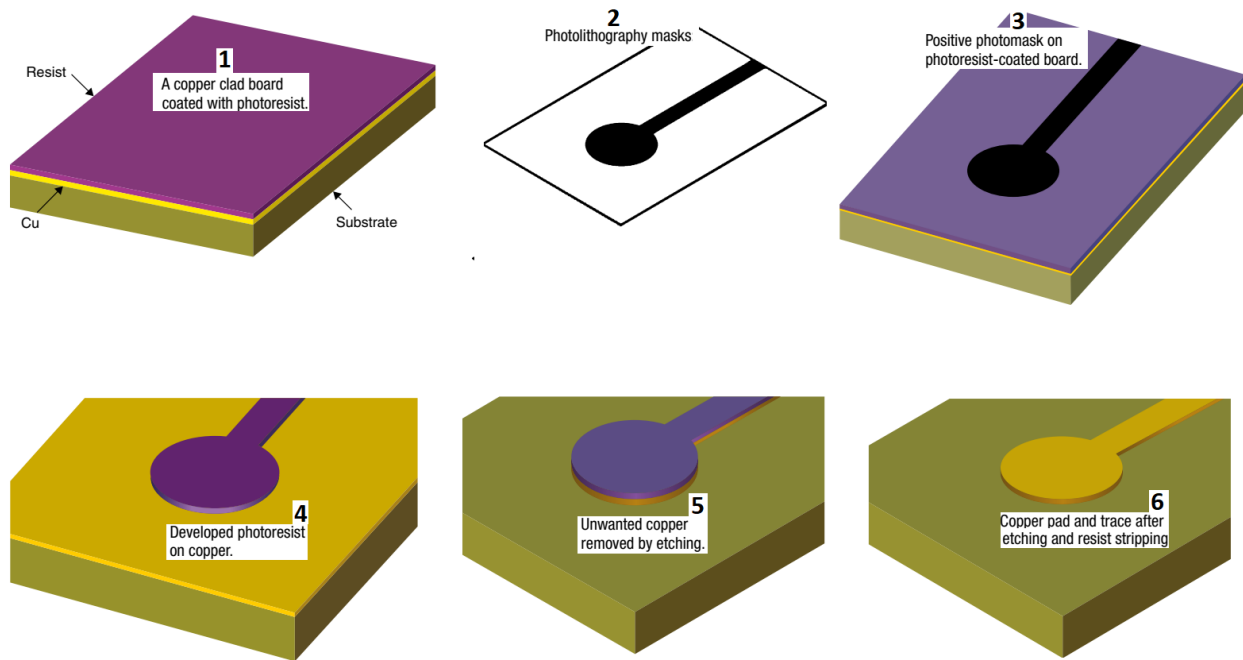


Figure1- 3

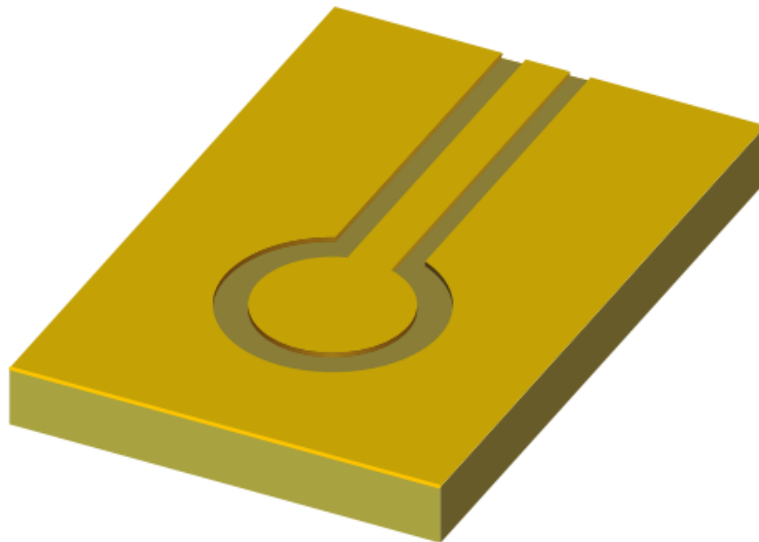


Figure1- 4

To make electrical connections between pads on different layers, the board is placed into a plating bath that coats the insides of the holes with copper, which

electrically connects the pads, hence, the term plated through holes. The cutaway view of Figure1-5 (right) shows a plated through hole on an internal layer of a PCB.

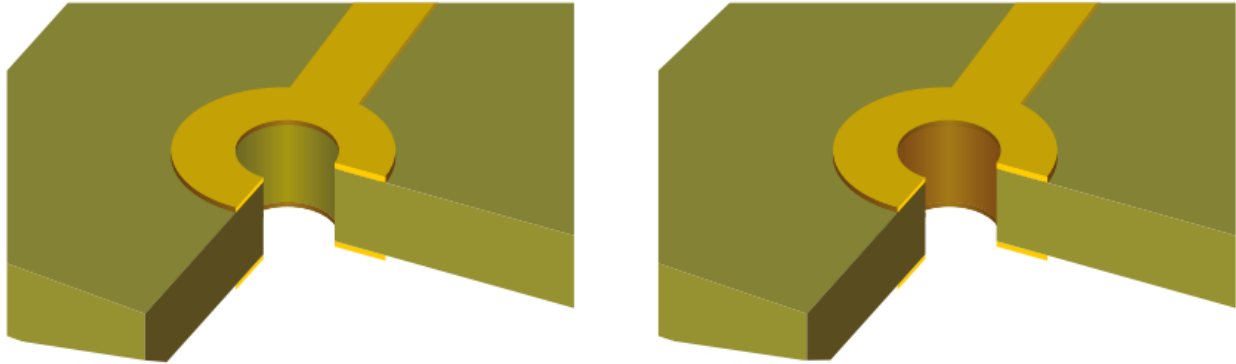


Figure1- 5

Next, a thin polymer layer is usually applied to the top and bottom of the board. This layer (shown in dark green in Figure 1-6) is called the solder mask or solder resist. The solder mask protects the top and bottom copper from oxidation and helps prevent solder bridges from forming between closely spaced pads.

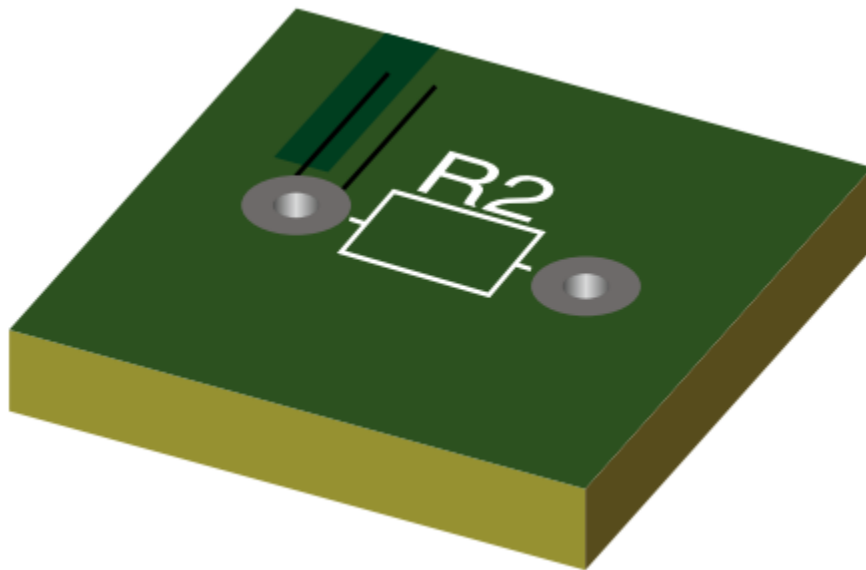


Figure1- 6

Finally, markings (called the silk screen) are placed on the board to identify where components are to be placed. The silk screen is shown in white in Figure1-6.

Function of Proteus Design Suite in the PCB design process

Proteus PCB Editor (ARES) is used to design the PCB by generating a digital description of the board layers for photoplotters and CNC machines, which are used to manufacture the boards. Separate layers are used for routing copper traces on the top, bottom, and all inner layers; drill hole sizes and locations; soldermasks; silk screens; solder paste; part placement; and board dimensions. Figure1-7 shows routed layers (top and bottom and an inner, for example), the background is black and the traces and pads on each layer are a different color to make it easier to keep track of visually.

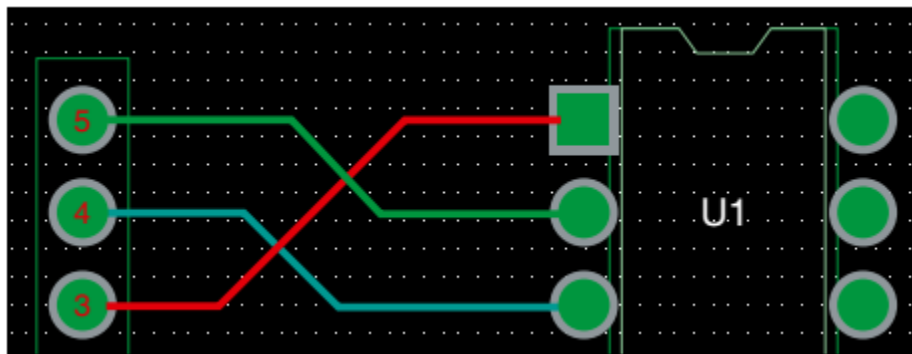


Figure1- 7

Common terms used in PCB Design:

Footprint (land pattern):

Footprints define the physical interface between the PCB and the real component. It consists of arrangement of pads that used to physically attach and electrically connect a component to a printed circuit board (Figure1-8). It consists of several layers of information, including the copper pads, silkscreen (the text and shapes you see when looking at a PCB), solder mask and solder paste.

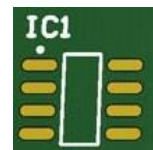


Figure1- 8

Pad:

you can think of pad as a piece of copper where the pins of a component are mechanically supported and soldered. There are two types of pads; thru-hole and smd (surface mount)

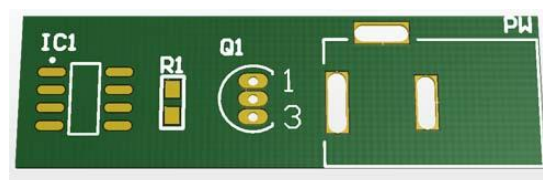


Figure1- 9 SMD and Thru-hole Pads

Via:

A plated hole that electrically connects two or more traces on two or more separate layers (Figure1-10).

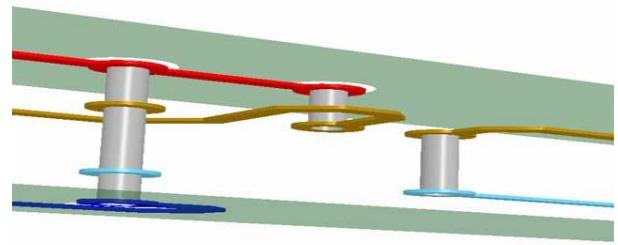


Figure1- 10

Trace:

A track is conductive path used to connect 2 points in the PCB (Figure1-11).

For example, two pads, a pad and a via, or two vias. The tracks can have different widths depending on the currents that flow through them.

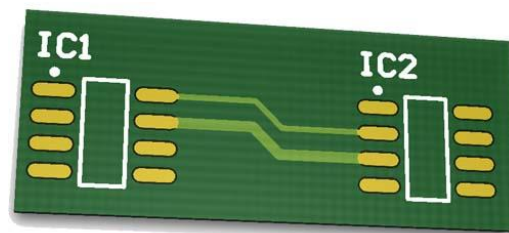


Figure1- 11

Overview of the design flow

Regardless of which type of board is made, certain steps must be executed in the design flow process. The following is an outline of the process. Remember that in this lab we will deal with simple circuits so some steps in the design flow may not be applied.

- **Initial design concept and preparation**
- **Set up the design project in Proteus**
- **Define the board requirements**
- **Basic board setup**
- **Pre-route specific nets using manual routing:**
- **Auto-routing**
- **Finalizing the design**

Detailed description for each of the above steps as following:

1. Initial design concept and preparation:

- a. Generate initial drawings (handmade designs).
- b. Collect data sheets.
- c. Take inventory of packaging and footprint needs.
- d. Search through the Proteus libraries to find the parts. For any parts that are unavailable, manually construct these parts from scratch.

2. Set up the design project in Proteus:

- a. Draw the schematic (placing and connecting parts).
- b. Perform an annotation to clean up numbering.
- c. Make sure multipart packages are properly utilized.
- d. Make sure global power nets are properly connected.
- e. Assign related components to groups to aid in part placement in PCB Editor.
- f. Generate a bill of materials to identify PCB assigned and missing footprints.
- g. Search through the PCB Packages libraries to find and assign footprints. For any footprints that are unavailable, obtain the data sheets for recommended footprints and design these footprints from scratch.

3. Define the board requirements:

- a. Board dimensions and mounting holes locations.
- b. Part placement considerations (height restrictions, assembly method).
- c. Noise and shielding requirements.
- d. Component mounting technology (SMT, THT).
- e. Trace width and trace spacing requirements.
- f. Required vias and fan-outs (size and tenting, etc.).
- g. Number of Power/Ground planes and Routing layers.

4. Basic board setup:

- a. Physical:
 - i. Create the board outline.
 - ii. Place mounting holes.
 - iii. Define part and routing restriction areas.
 - iv. Add dimension documentation (optional).
- b. Preliminary parts placement:
 - i. Place selected parts and groups.
 - ii. if DRC reports errors, fix it

- c. Layer setup:
 - i. Set up Power and Ground planes.
 - ii. Set up Routing layers.
 - iii. Assign ground and power nets to Plane layers.
 - iv. Define thermal relief parameters.
 - v. Set which vias to use for fan-outs, routing vias, jumpers, and the like.
 - vi. if DRC reports errors, fix it
- d. Final parts placement:
 - i. Make sure spacing rules are not violated.
 - ii. If using split or moated Plane layers, make sure parts are placed accordingly.
 - iii. Check orientation of polarized components (caps, diodes, etc.).

5. Pre-route specific nets using manual routing:

- a. Perform power and ground fan-outs.
- b. Pre-route critical nets manually.
- c. If DRC reports errors, fix it.

6. Auto-routing:

- a. Set up the autorouter.
- b. Run the autorouter.
- c. If DRC reports errors, fix it.

7. Finalizing the design:

- a. Post-routing inspection:
 - i. Sharp (acute) angles.
 - ii. Long parallel traces (cross-talk issues).
 - iii. Via locations.
 - iv. Silk-screen markings.
- b. Board cleanup:
 - i. Unroute and then reroute problem traces.
 - ii. Perform a final DRC.

8. Generate Manufacturing files

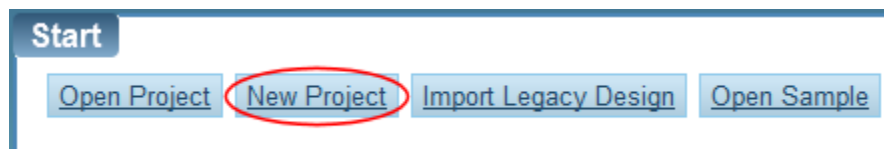
Introducing Proteus Design Suite:

Proteus 8 is a single application with many service modules offering different functionality (schematic capture, PCB layout, etc.). In this Lab, we will introduce you with two pieces of Proteus 8 called ISIS and ARES through an example. ISIS is used for schematic capture while ARES is used for PCB layout.

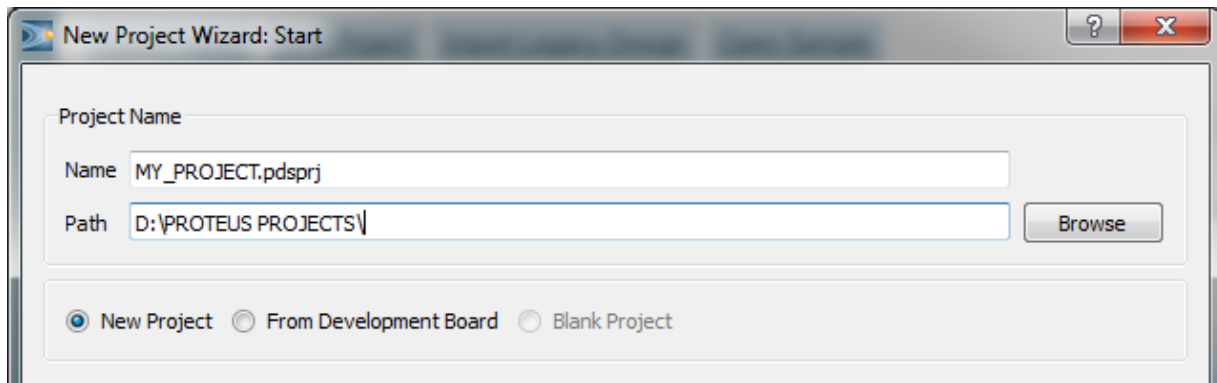
SCHEMATIC CAPTURE BASICS:

To start with ISIS you need to create a new project as following:

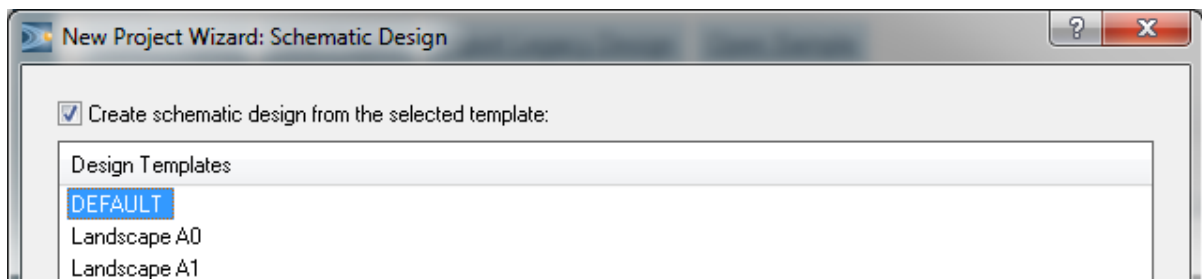
- Start Proteus 8 from Desktop shortcut or from start menu. The main application will then load and run and you will be presented with the Proteus home page.
- Start by pressing the new project button near the top of the home page in Proteus.



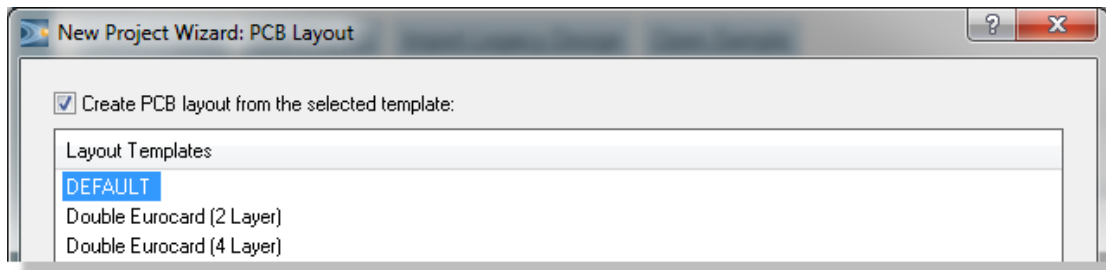
- On the first page of the wizard specify a name and path for the project.



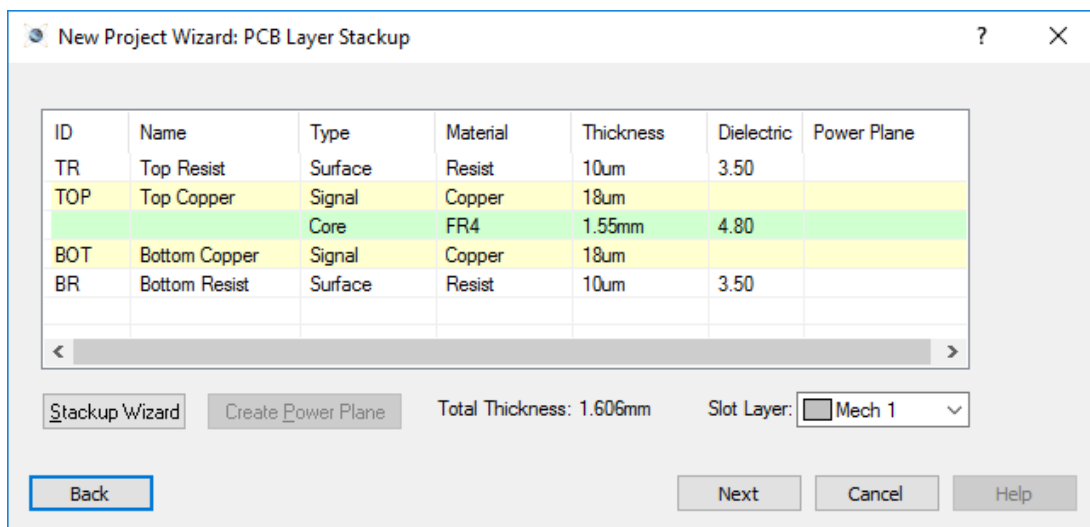
- We need a schematic so check the box at the top of the next step and then choose the default template.



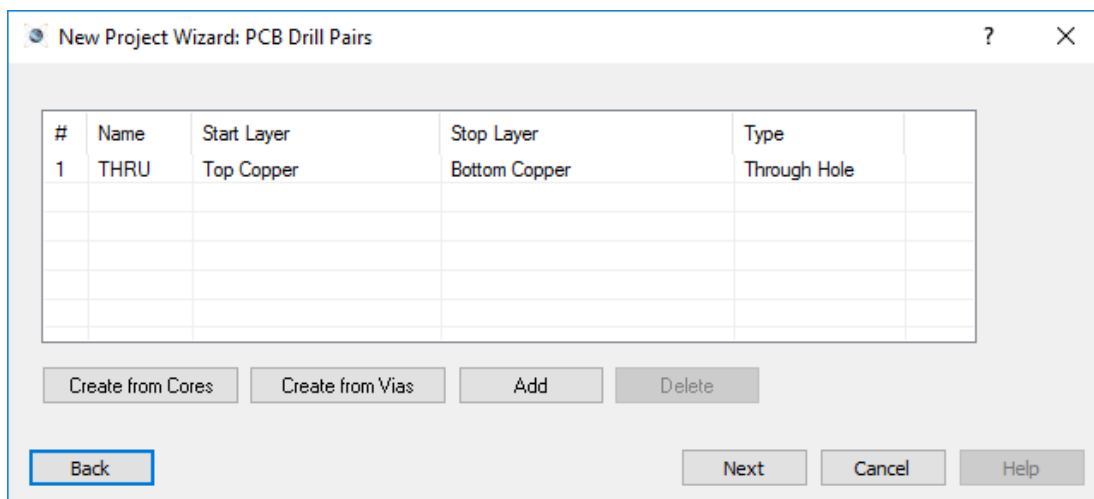
- Similarly, we need a layout so check the box at the top of the layout page and again choose the default template.



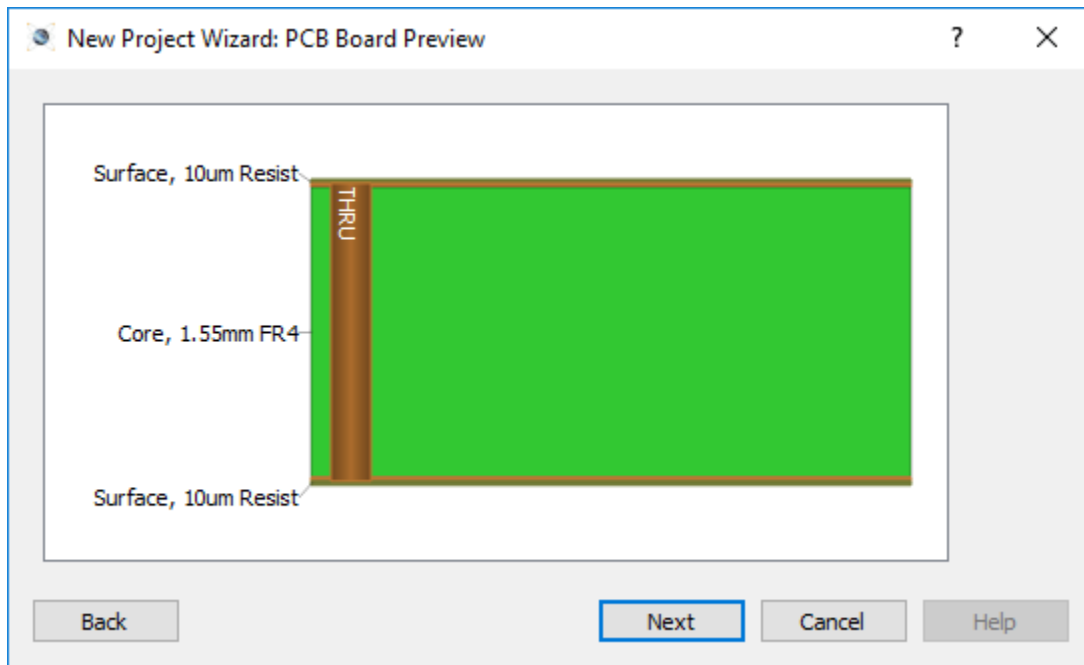
- The next screen allows us to define the layer stack for our PCB. Since we will be designing a simple two layer board there is no configuration necessary here.



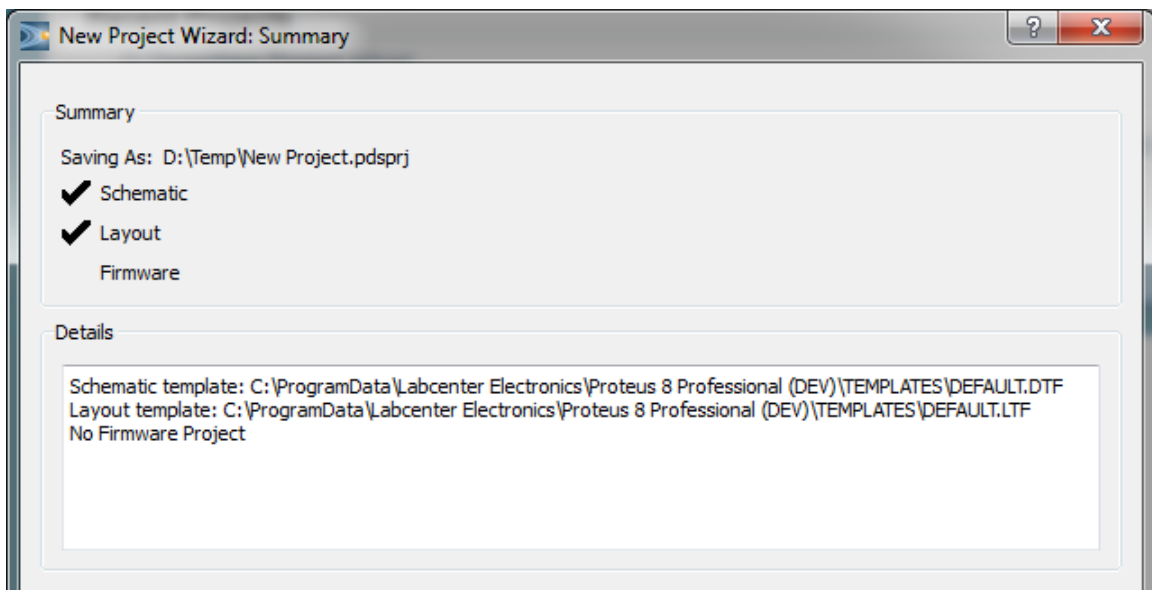
- The next screen is for configuration of drill spans. Again, for our proposed 2-layer board the only possibility is thru-hole so there is no action required.



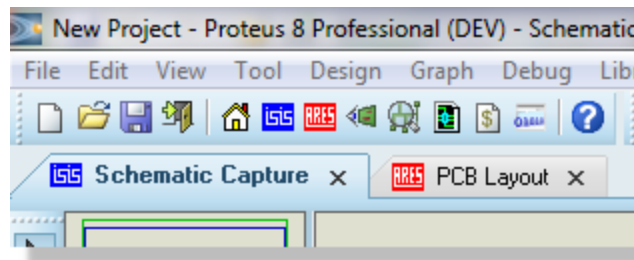
- The final screen in the PCB configuration is simply a preview of a PCB cross section that displays visually what has been set up in the previous screens.



- 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
- 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 editor to the foreground.

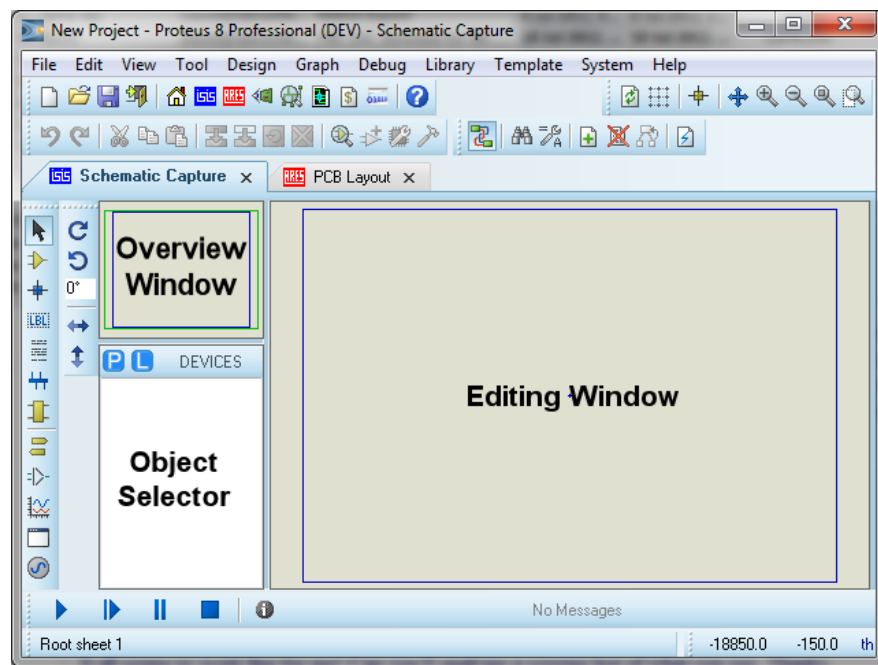


Guided Tour OF ISIS WINDOW:

The largest area of the screen is called the Editing Window. 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 an overview of the entire drawing. However, when a new object is selected from the Object Selector the Overview Window is used to preview the selected object.

Below the Overview Window is the Object Selector which you use to select devices, symbols and other library objects.



Zooming:

There are several ways to zoom in and out of areas of the schematic :

Point the mouse where you want to zoom in and out of and roll the middle mouse button (roll forwards to zoom in and backwards to zoom out).

Point the mouse where you want to zoom in or out of and press F6 or F7 keys respectively.

Use the Zoom In, Zoom Out, Zoom All or Zoom Area icons on the toolbar.



The F8 key can be used at any time to display the whole drawing.

Panning

As with zooming, there are a number of options for panning across the editing window.

Click on the middle mouse button to enter track pan mode. This puts the schematic in a mode where the entire sheet is picked up and will move as you move the mouse. Left click the mouse again to exit track pan mode.

Hold the SHIFT key down and bump the mouse against the edges of the Editing Window to pan up, down, left or right. We call this Shift Pan.

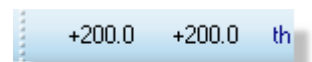
the quickest method is to simply point at the of the new area on the Overview Window and click left.

It is well worth spending a few moments familiarizing yourself with navigation on the schematic - it is after all one of the most common operations you will perform. In particular, learning to use the middle mouse button both for track pan and for zooming will save you time during schematic design.

Grid and snap configuration:









A grid of dots or lines can be displayed in the Editing Window as a visual aid using the **Grid command on the View menu**, or by **pressing 'G'** to toggle the grid from 'dots', 'lines' or 'off', or by clicking the **Grid Icon on the toolbar**

Finally, at the bottom of the screen is the co-ordinate display, which reads out the co-ordinates of the mouse pointer when appropriate. These coordinates are in 1 thou units and the origin is in the centre of the drawing



Visual Effects:

objects are encircled with a dashed line or when the mouse is over them and mouse cursors will change according to function. Essentially, the dashed line scheme tells you which object the mouse is over (the 'hot' object) and the mouse cursor tells you what will happen when you left click the mouse on that object. A summary of cursors used, together with their actions, is provided below:

Cursor	Description
	Standard Cursor - Used in selection mode when not over a 'hot' object.
	Placement Cursor - Placement of an object will start on a left click of the mouse.
	Hot Placement Cursor - Appears green when placement of a wire is available on left click of the mouse.
	Bus Placement Cursor - Appears blue when placement of a BUS is available on left click of the mouse.
	Selection Cursor - Object under the mouse will be selected on a left click of the mouse.
	Move Cursor - The currently selected object can be moved.
	Drag Cursor - The wire or 2D graphic can be dragged by holding the left mouse button down.
	Assignment Cursor - When over an object (having set the Property Assignment Tool) You can assign the property by left clicking the mouse button.

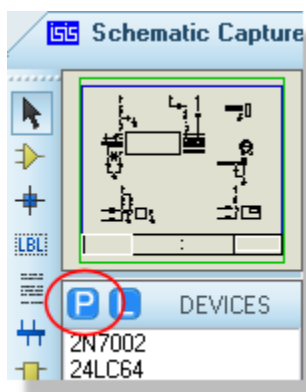
Basic Schematic Entry:

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.

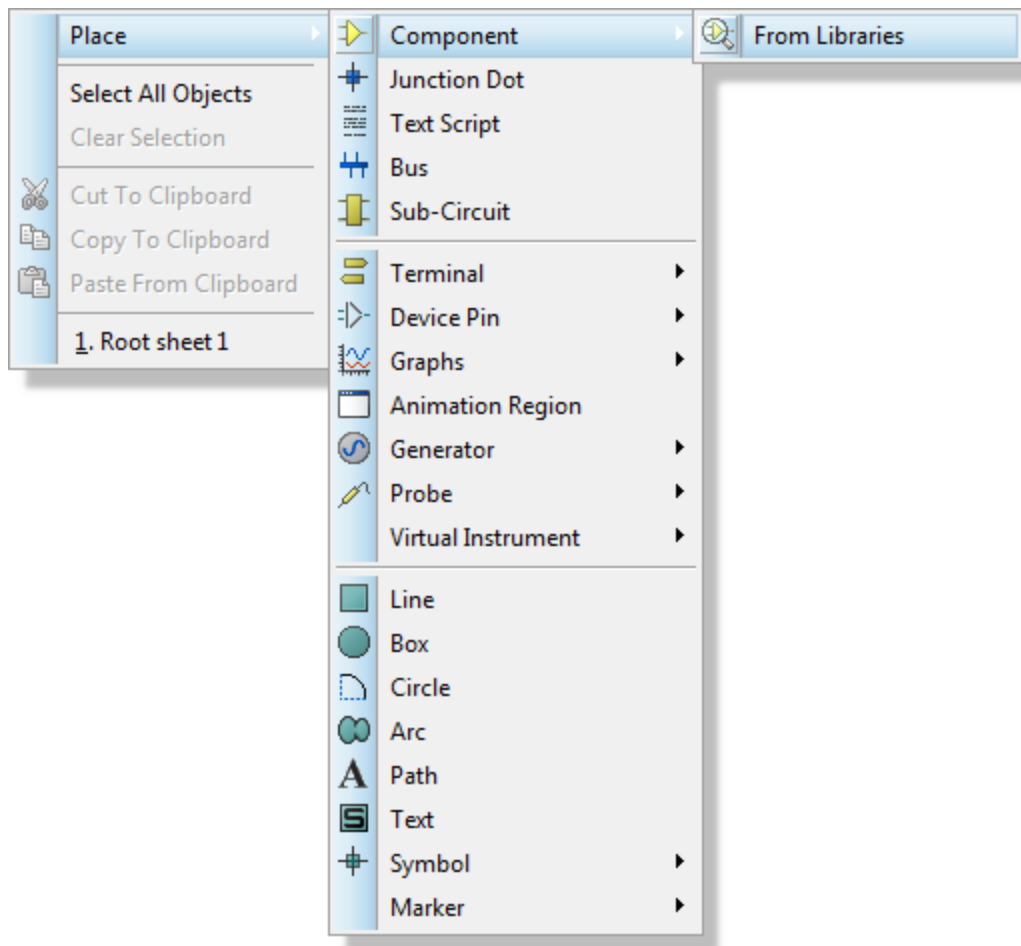
Selecting Parts from the Library

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 keyboard shortcut for this command (P key on the keyboard).

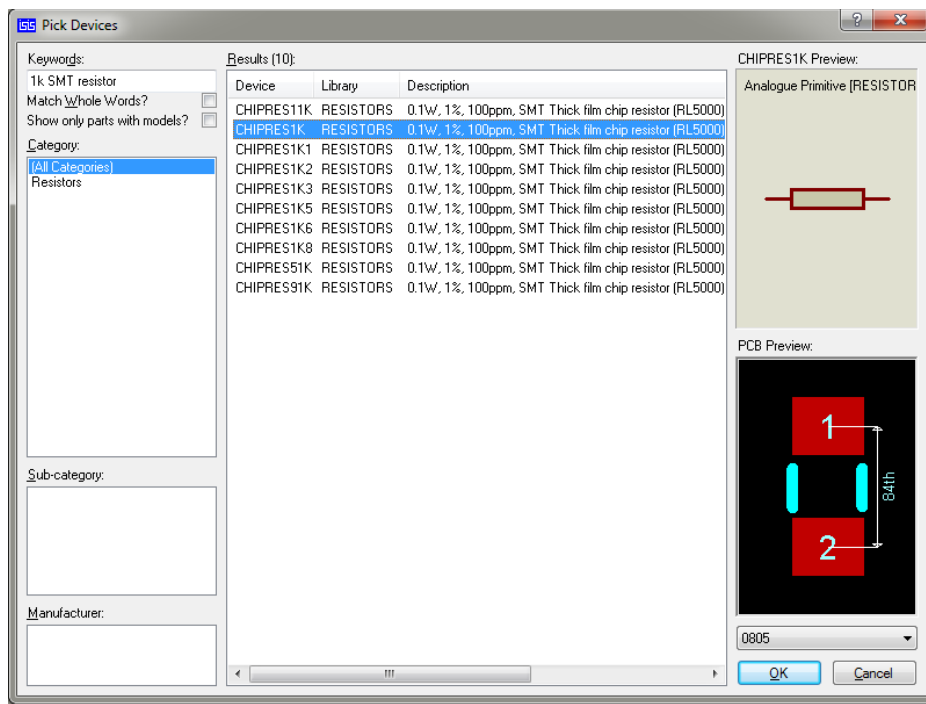


Right click the mouse on an empty area of the schematic sheet and select Place – Component → From Libraries from the resulting context menu as shown below :

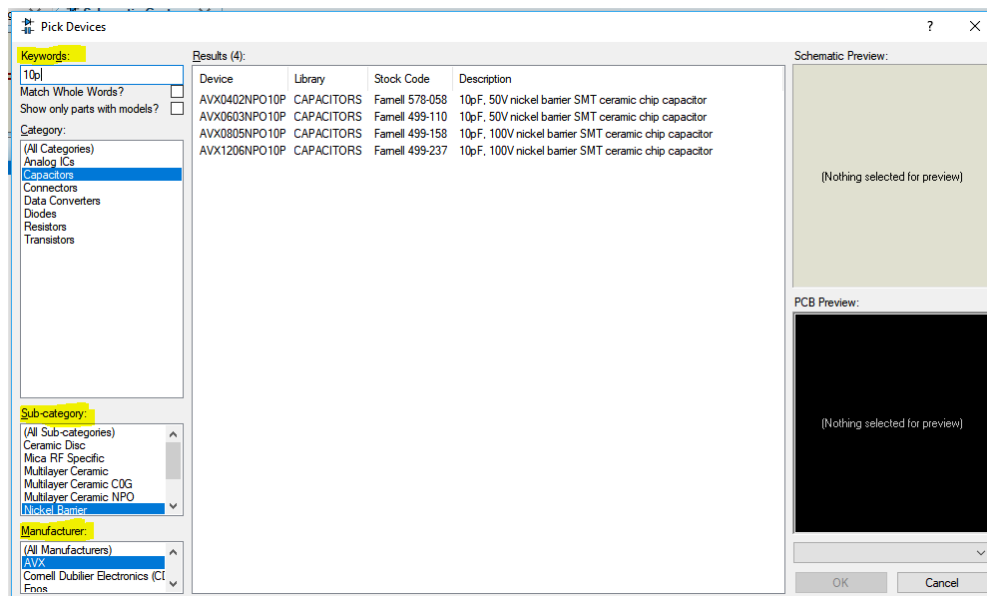


Either of these two methods will cause the Device Library Browser dialogue form to appear.

Given that we know the names of all the parts we want, we could simply proceed by using this technique to bring in all the components we need. However, this may not always be the case and Proteus provides several methods for finding parts in the component libraries. One of the most intuitive is to use the library browser a little like an internet search engine, typing in descriptive keywords and then browsing the results to find a specific part. Try this now with the resistors, typing in '1k SMT resistor' in the keywords field of the library browser dialogue to locate the CHIPRES1K component (double click on the part in the results list to import into the schematic). We could similarly search for '10k SMT resistor' to find and insert the CHIPRES10k component and so on.



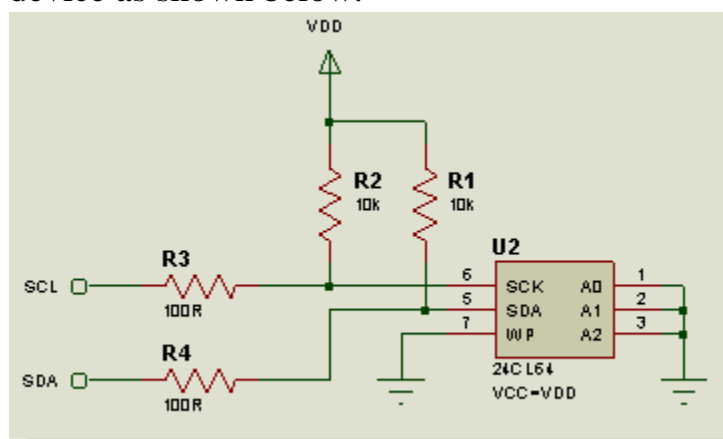
It may be that we simply want to browse for a specific type of part that available from a specific manufacturer. To take an example, clear out the contents of the keyword field and then select the Capacitors Category. In our design, we are looking for some Nickel Barrier caps from AVX so we can further filter the results set by selecting Nickel Barrier from the sub-category field and AVX from the Manufacturer field. There are still a large number of caps available so we might type in '22p', '1N', etc. in the keywords field to isolate and select the particular components we require (AVX0805NPO22P, AVX0805X7R1N and so on).



When you are finished you should have all of the required components in the Object Selector as shown in the following screenshot for example.

Placing Objects on the Schematic:

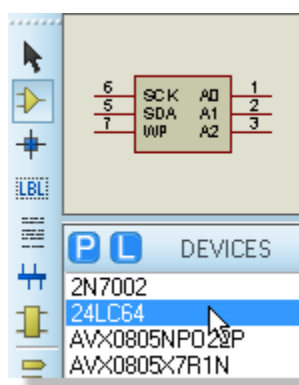
Having selected the parts we need the next thing is to actually place them on the drawing area – the Editing Window – and wire them together. We are going to start off simply and complete the block of circuitry comprising the I2C Memory device as shown below.



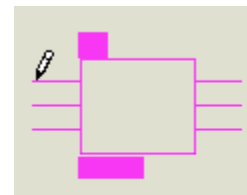
P	L	DEVICES
		2N7002
		24LC64
		AVX0805NPO22P
		AVX0805X7R1N
		AVX0805X7R10N
		AVX0805X7R100N
		B82432T1103K000
		BAS40
		CHIPRES1K
		CHIPRES10K
		CHIPRES20K
		CHIPRES68K
		CHIPRES100K
		CHIPRES100R
		CHIPRES200K
		CONN-H2
		CONN-SIL4
		CONN-SIL6
		CRYSTAL
		DSPIC33FJ12GP201
		JUMPER
		LED-RED
		LM285-2V5
		LM358
		MAX1724EZK50
		MPX4115
		SHT15
		TAWD106M025R0600

Begin by placing the I2C memory device as follows:

Select the 24CL64 device from the Object Selector.



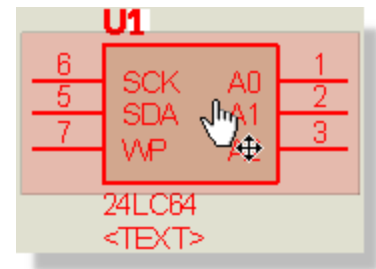
Left click on the schematic to enter placement mode.



Move the mouse to the desired location for the part, then left click the mouse again to 'drop' the part and commit placement.

If you need to move parts or blocks of circuitry after placement, select the object(s) we want to move, left press the mouse, drag to the new location and finally release the mouse to drop.

There are several ways we can select objects:



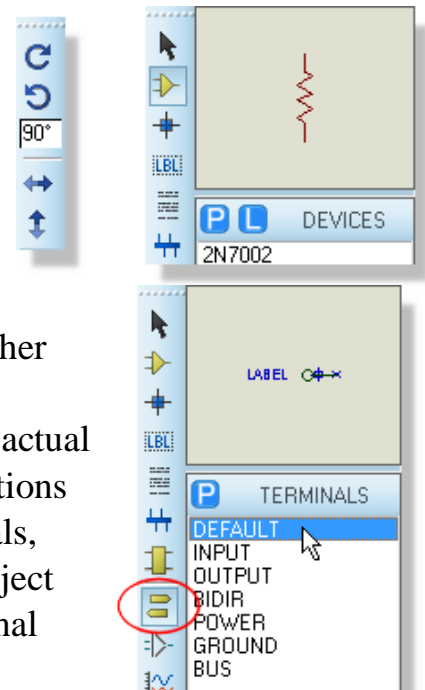
Choose the Selection Icon and then left click on the object you want to select. Right clicking the mouse on an object will both tag the object and present a context menu containing available actions on that object.

Draw a tagbox around the object by depressing the left mouse button and dragging the mouse to form a box encompassing the object to be selected. This is the technique that should be used for moving multiple, connected objects or blocks of circuitry.

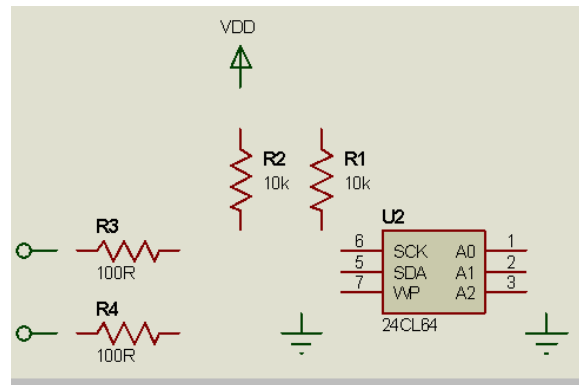
Having placed the memory device, we now need to get the peripheral circuitry down and oriented correctly. We are going to need two 10k pull up resistors and two 100 Ohm resistors for the data and clock lines. Additionally, we are going to need to use terminals to achieve connectivity with power, ground and other sections of circuitry. Begin by selecting the CHIPRES10k device and click left once on the anti-clockwise Rotation icon (shown below); note that the preview of the resistor in the Overview Window shows it rotated through 90°.

You can also rotate parts 'live' when in placement mode. Left click the mouse once to enter placement mode (at this point you will see the component outline following the mouse) and then use the '+' and '-' keys on the numeric keypad to rotate the component as you are placing it. Left click again to commit the placement in the normal way.

We use terminals in schematic design simply to terminate a wire and assign a connection. Often this connection is to either power or ground but it can just as easily be to another wire elsewhere on the circuit. Terminals allow us both to reduce actual wiring (avoiding spaghetti schematics) and to make connections between different sheets on the schematic. To place terminals, start by selecting the terminal mode; this will switch the Object Selector and provide us with a listing of the available terminal types.

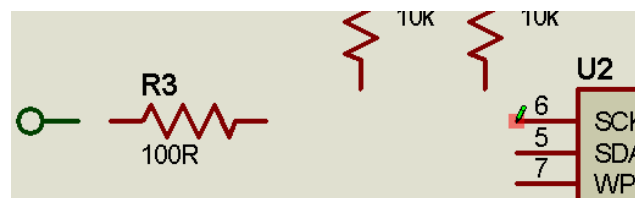


We need a power terminal, ground terminal and also two default terminals for the connections on the I2C bus. From this stage, placement and orientation are identical to any other object and should now be quite familiar. Place the appropriate terminals in their approximate locations now, such that the area around the memory device now looks something like the following

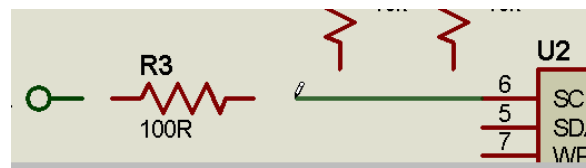


Wiring Up:

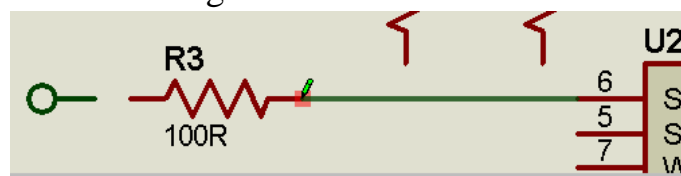
The basic procedure for placing a wire between two pins is given below, using the connection between the SCK pin of the memory device and the 100Ohm resistor as an example:



Move the mouse over the SCK pin on the memory device - the cursor will change to a green pen.



Left click the mouse and then move it to the left until it is over the pin of the 100 Ohm resistor. The wire will follow the mouse and the cursor / pen is white during wiring. Left click the mouse again to commit the connection and place the wire.

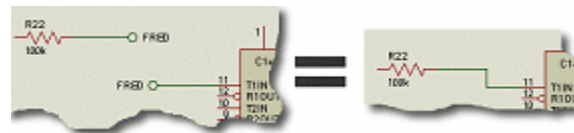
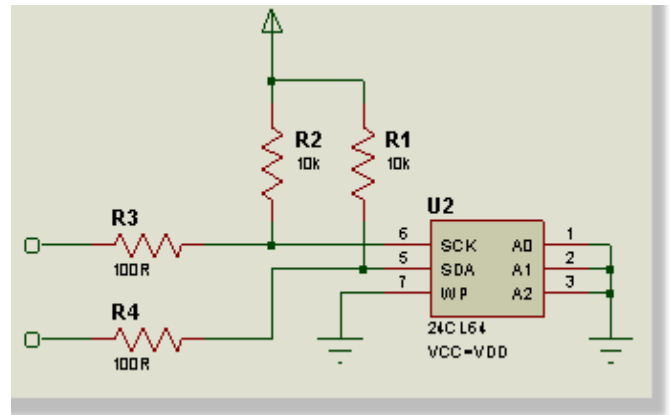


Armed with the above you should now be able to connect up all of the circuitry, so that your schematic now looks something like this:

Making Connections with Terminals

The final thing we need to do to complete this block of circuitry is label the terminals.

Terminal naming is extremely important as it denotes the connection to be made. We could name the terminals in any fashion we liked but sensible names make the schematic more legible and easy to understand.



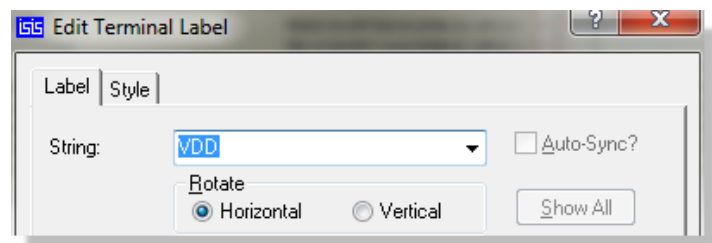
Power, Ground and NC terminals are the exception to this rule, although there is no reason not to label them; an unlabelled power terminal is assigned to the VCC net and an unnamed ground terminal will be assigned to net GND.

Essentially what we are doing by labeling a terminal is making a connection to somewhere else on the schematic (a terminal with the same name) without placing a physical wire between the two objects.

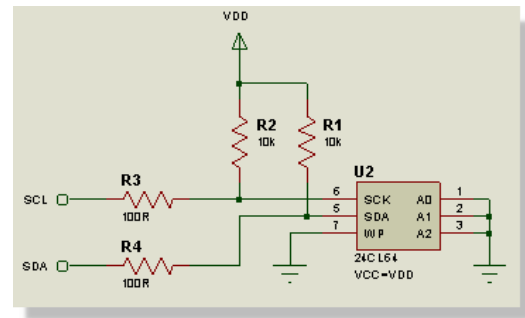
As discussed previously the Proteus schematic is flexible enough to present you with several methods for editing parts - choose your preferred method for editing the terminal from the following:

33Double left click on the terminal.

Right click on the terminal to select it and launch the context menu and then use the Edit Properties menu option.



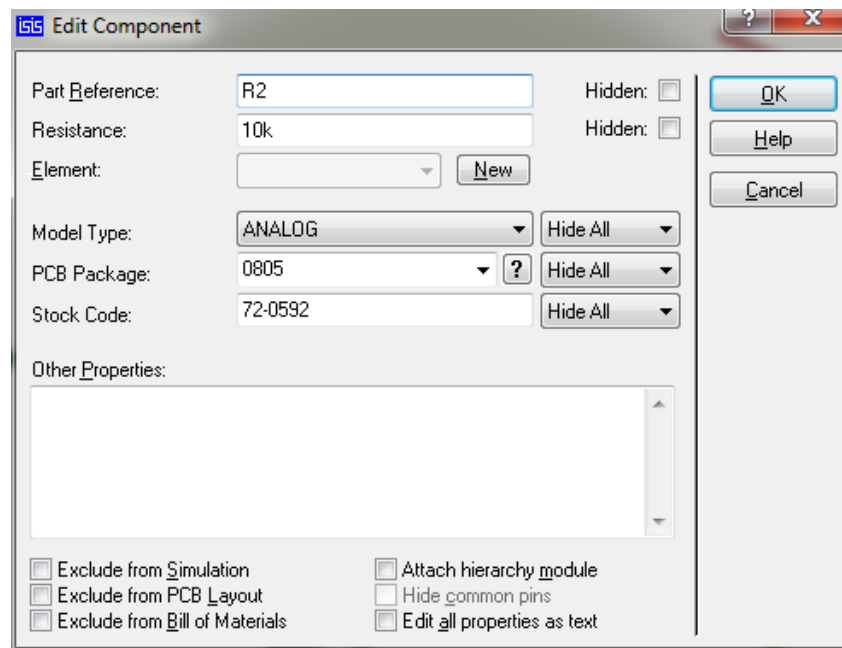
Finish the job now by editing the other terminals and labeling them appropriately such that your completed circuit block now looks like this :



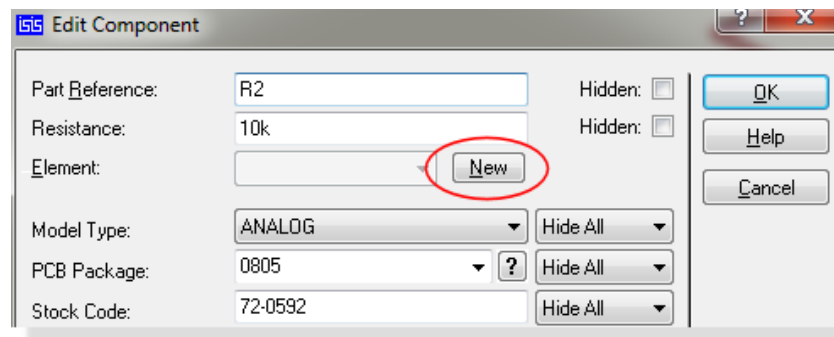
Part Labels and Annotation:

You should see that all the parts you have placed have both a unique reference and a value. A unique and sequential annotation is assigned to components as you place them on the schematic, although you can re-annotate manually if you need to.

You can edit both these fields and their visibility via the Edit Component dialogue form. Launch this dialogue form now by double clicking the left mouse button over the resistor



If, for example, you change 'R1' to be 'R2' then you will have two parts with the same reference on the schematic. This will cause netlist errors when working in the PCB module. If, however, you re-annotate using the new button on the edit component you are guaranteed a unique part reference.



Preparing for PCB Layout:

Assuming Now that we have completed the Schematic connection we need to give some thought to PCB Layout and the information that we are going to provide from the schematic.

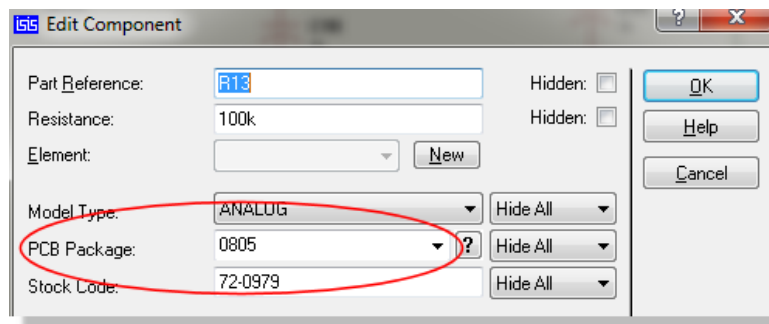
Packaging Considerations:

Ideally, each component on the schematic should be associated with a footprint in the PCB editor. While you can do this at the point of placement in the layout editor best practise would be to sort this out in the schematic. Fortunately, the Proteus system provides a large set of pre-packaged components where this work is all done for us. As a case in point, the schematic we have just created requires no alteration or packaging work as the parts we have selected from the libraries already have footprints assigned to them.

However, it is possible that you may want to change the footprint associated with a part (e.g. PTH to SMT). We'll use the current schematic as a playground to explore how we can view and change packages to components.

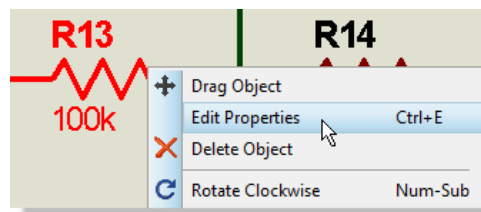
To view the package associated with a component:

right clicking on one of the resistors on the schematic and selecting Edit Properties from the resulting context menu. You should see that there is a PCB Package property on the dialogue form and that the part is packaged with a standard 0805 footprint.

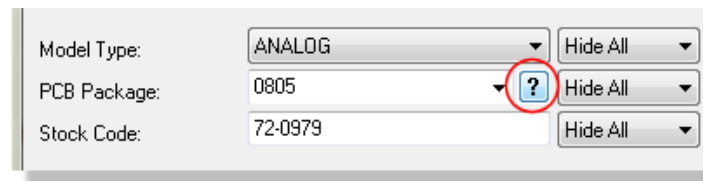


To change a package on a component

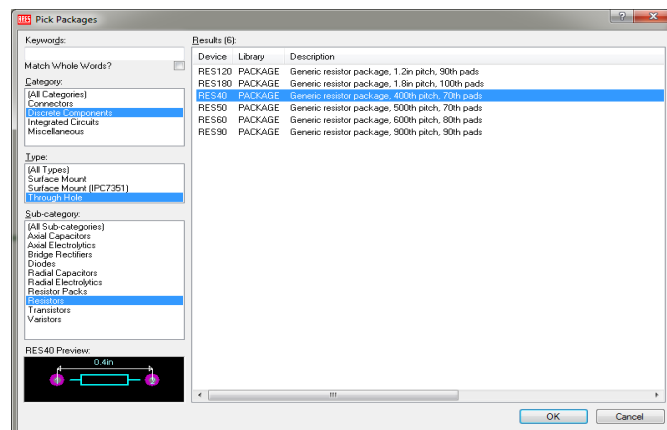
Launch the Edit Component dialogue form by right clicking on the resistor and selecting Edit Properties from the resulting context menu.



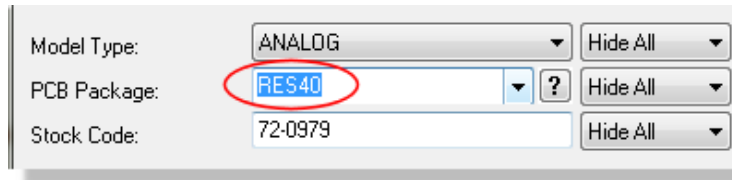
Click on the question mark to the right of the package property to launch the footprint browser



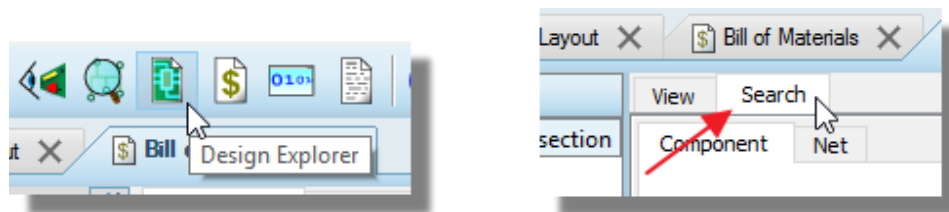
Clear out the text from the keywords field and then use the filters on the left hand side to narrow down the selection. We will want to select the 'Discrete Components' category with type 'Through Hole' and Sub-Category 'Resistors'. We can then select for example the 'RES40' footprint and click OK to commit.



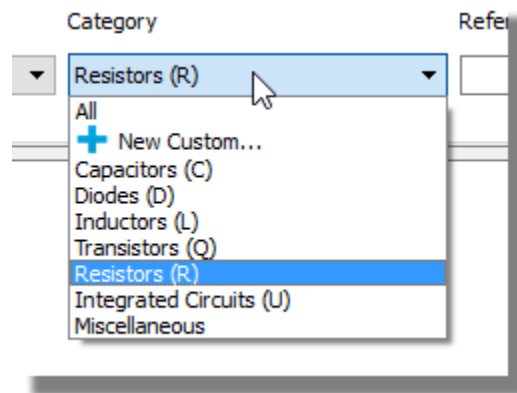
You should now see that the RES40 is listed as the PCB Package for the component in the schematic.



If we want to make a wholesale change to parts already placed we can do so from the design explorer. First, open the Design Explorer via the icon on the main toolbar and then switch to the search tab:



Following on from our example above, let's set category to be Resistor and hit search



You should now see a filtered list of all the resistors on the schematic in a grid view. We could filter further by entering a resistor value.

View Search			
Component		Net	
Sheet		Tagged	
Analog, Reference & Power		<input type="checkbox"/>	
Reference	Type	Value	Package
▶ R11 (20k)	RES	20k	0805
▶ R12 (20k)	RES	20k	0805
▶ R13 (100k)	RES	100k	0805
▶ R14 (68k)	RES	68k	0805
▶ R15 (100k)	RES	100k	0805
▶ R16 (100k)	RES	100k	0805
▶ R17 (100)	RES	100	0805
▶ R18 (100)	RES	100	0805
▶ R19 (200k)	RES	200k	0805
▶ R20 (1k)	RES	1k	0805
▶ R21 (100)	RES	100	0805

To change the package, simply type in the field at the top and then, much like Excel, drag the value down the other entries in the column.

Apply Changes Discard Changes		Value	Package
▶ R11 (20k)	RES	20k	0603
▶ R12 (20k)	RES	20k	0603
▶ R13 (100k)	RES	100k	0603
▶ R14 (68k)	RES	68k	0603
▶ R15 (100k)	RES	100k	0603
▶ R16 (100k)	RES	100k	0603
▶ R17 (100)	RES	100	0603
▶ R18 (100)	RES	100	0603
▶ R19 (200k)	RES	200k	0603
▶ R20 (1k)	RES	1k	0603
▶ R21 (100)	RES	100	0603

Hit Apply Changes to commit the change.

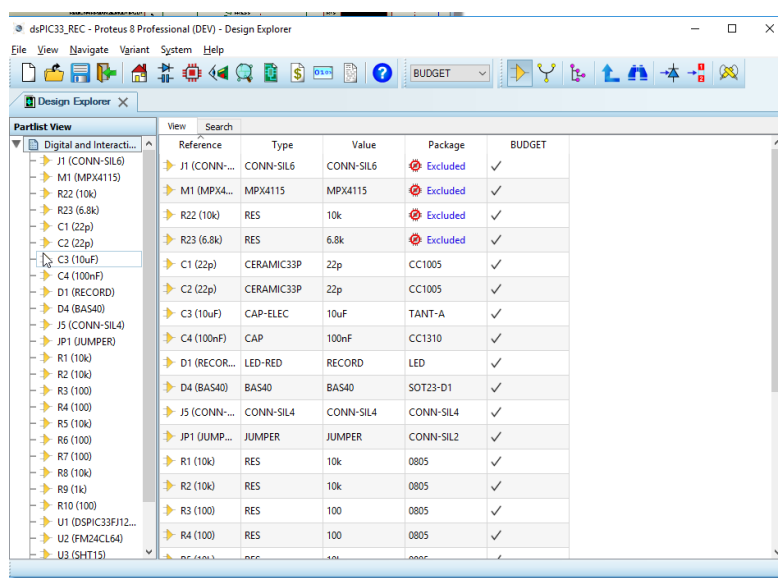
Apply Changes Discard Changes		
▶ R11 (20k)	RES	20k
▶ R12 (20k)	RES	20k

Design Verification:

It is always a good idea to spend a little time checking the schematic before we move through to PCB layout. Proteus provides a unique and extremely powerful tool in the form of the Design Explorer that will help us catch any errors before we sign off on the schematic phase of design

The Design Explorer

The Design Explorer can be launched from top level application module toolbar and will appear as a separate tab inside the Proteus application.



The Design Explorer shows us the bigger picture and allows us to make a global check of the packaging on the design at a glance. All of the footprints used are displayed beside their schematic reference at the far right hand side of the right hand pane so all we need to do is scan down the list to make sure everything is packaged, switch sheets (via the left hand pane) and repeat the process. If a part on the schematic did not have a footprint assigned it would have bright red text stating 'MISSING' in this field to highlight the potential problem. Locate the missed footprints and try to assign them as discussed previously.

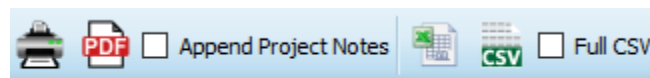
Bill of Materials

Constructing a Bill of Materials is an often necessary but frustrating task at the end of the schematic design phase. Fortunately, Proteus provides a completely flexible scheme which allows you to include as much or as little information as required.

You can launch the Bill of Materials module from the application module toolbar at the top of Proteus.

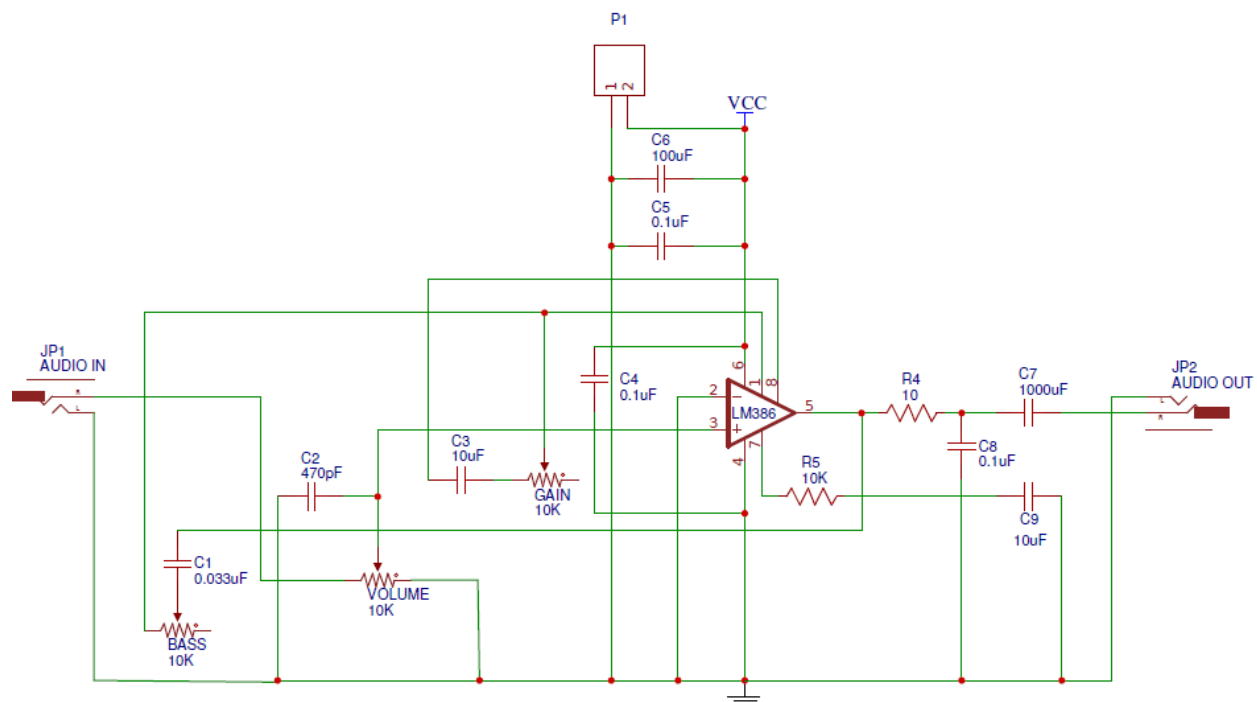


Having configured the information on the BOM you can generate the report from the export options from the icons at the top.



Task #1:

Each students has to create a new Schematic/ PCB project as described earlier, complete the schematic for the audio amplifier circuit shown, and generate Bill of material for that schematic.



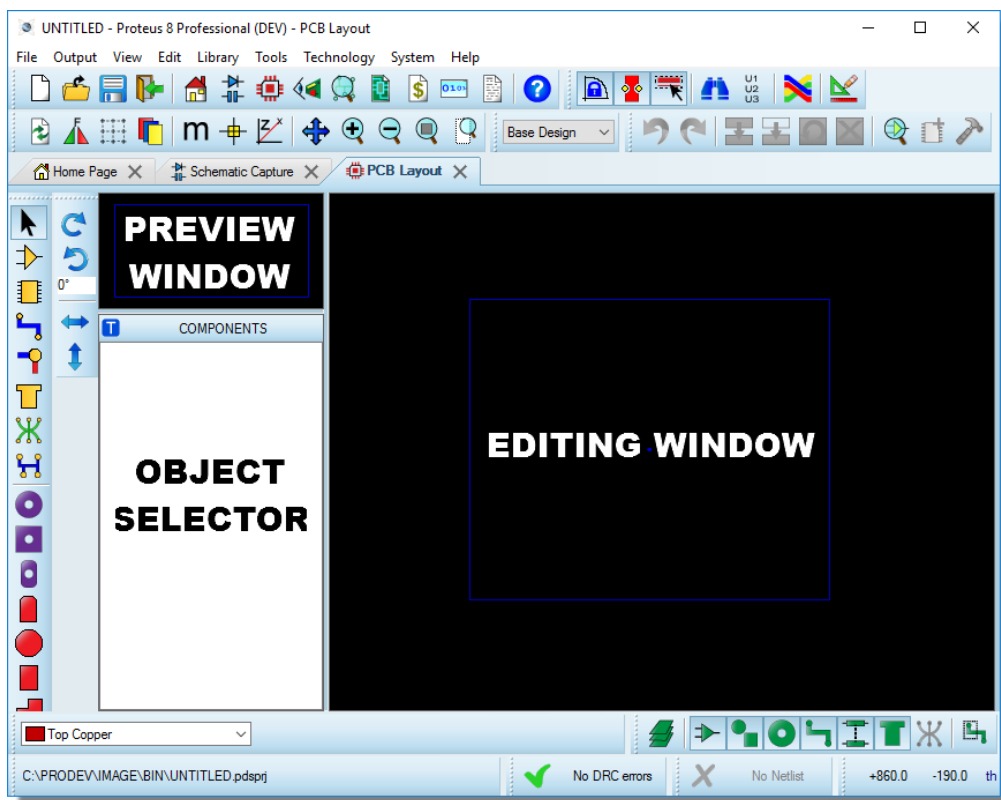
PCB Layout BASICS:

You can start the PCB layout module from the application module toolbar at the top of the Proteus application.



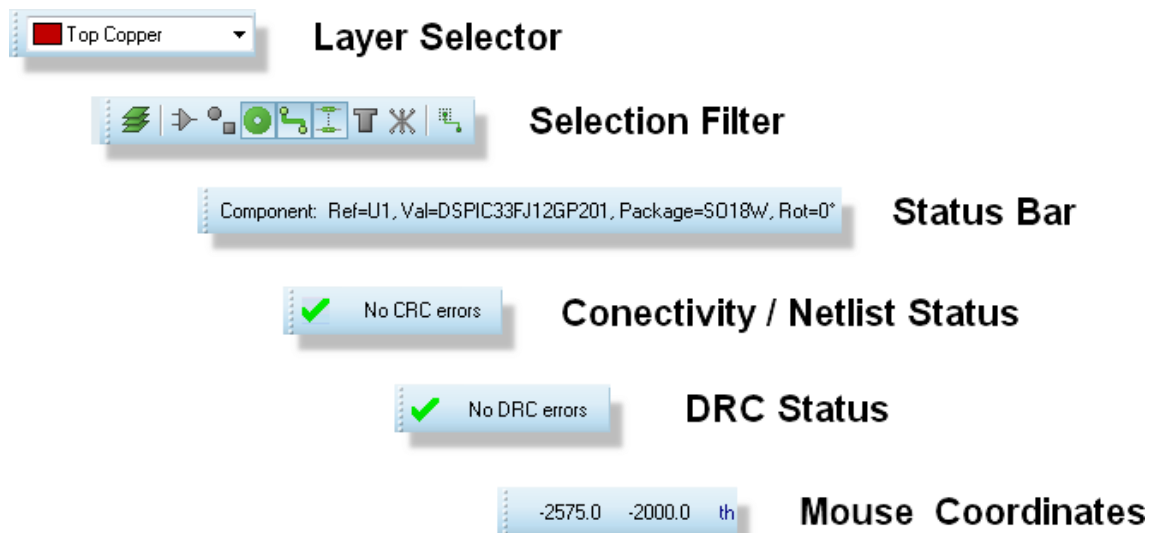
The Main Window

The largest area of the screen is called **the Editing Window**, - 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 an overview of the entire drawing. However, when a new object is selected from **the Object Selector** the Overview Window is used to preview the selected object.

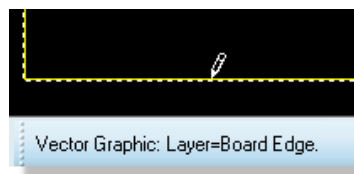


The Control Bar

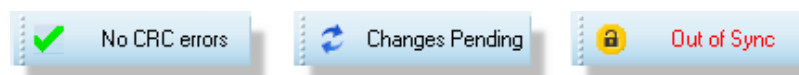
The control bar at the bottom of the application is different from what we have seen in the schematic capture module and essentially splits into five sections:



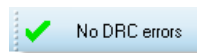
In the middle is the Status Bar which provides textual 'hints' on the object currently under the mouse. This is particularly useful when you hover a mouse over a pad for example, as it will inform you which net the pad is on.



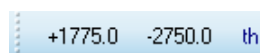
Next, we have the connectivity status which controls the syncing of the PCB to the schematic and - when everything is synchronised - doubles as a live connectivity checker.



Next to the connectivity status is the live Design Rule Checker. This will report any physical design rule violations that occur while the board is being designed. A left click on this will launch a dialogue detailing the violations with the further option of zooming in to examine a particular error.



Finally, at the far right hand side is the co-ordinate display which reads out the position of the cursor when appropriate. These reflect not the exact position of the pointer but the location to which it has been snapped. The co-ordinates can be in imperial or metric units as set by the Metric (default key mapping 'M') command.



Component Placement

Proteus 8 works with a live netlist so the layout module already has much of the information we need to start the layout process. In particular, we have specified which footprints are associated with each schematic symbol and so Proteus can therefore pre-select these for us ready for placement. This brings us to an important distinction in the software; the difference between a component and a package.

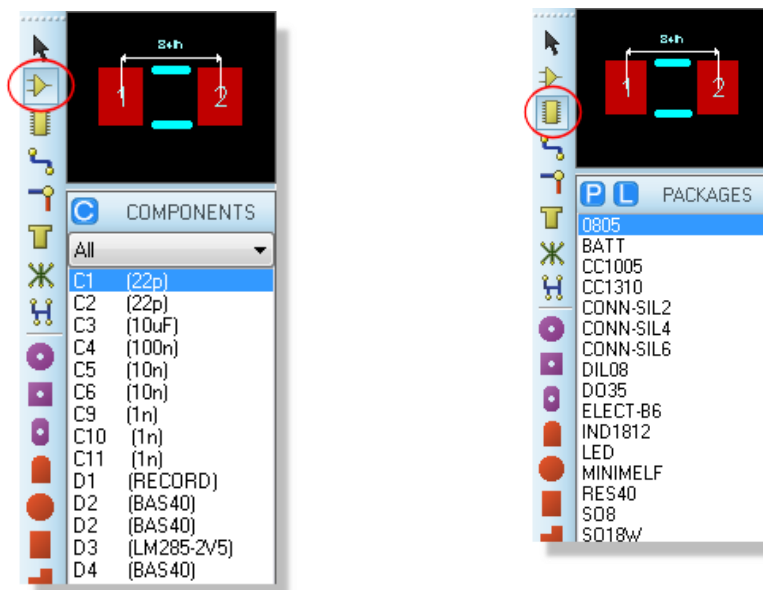
Components and Packages

A **component** is an instance of a footprint that has been netlisted through from the schematic.

A **package** is a physical footprint that exists in the PCB Layout Libraries.

Selecting component mode will access footprints which have been specified as relating to parts in the associated schematic and carry connectivity information whereas selecting package mode will access ‘unbound’ physical instances of a footprint. When working with a layout which is driven from a schematic we therefore exclusively use component mode.

The **Component Mode** icon is second from the top directly underneath the Selection icon. Clicking on this will display a list of items in the Object Selector which correspond directly to the parts on the schematic.



The **Package Mode** icon is directly underneath the Component mode icon and clicking on this will show us the physical footprints corresponding to the components in the layout.

When placing, routing and laying out a PCB based on a schematic (such as with this project) we will be working with components.

The Board Edge

Before we can place the components on the board we need to define what shape and size the board will be. For simple project we need only a simple rectangular board edge but we do want to control the dimensions of the board (Ex: 115mm wide and 40mm high).

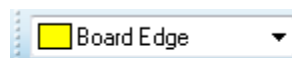
The first thing to note here is that the layout module will operate in either imperial or metric units and you can switch between modes either by toggling the Metric command on the View menu or via the 'M' keyboard shortcut. You may need to switch units for the placement of the board edge.

You can set your default or preferred units from Technology Menu - Grid Configuration command.

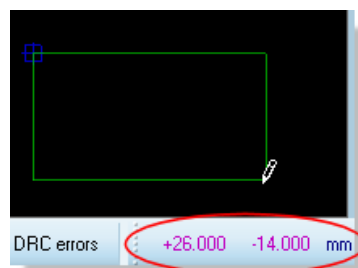
To start placing the board edge, select the 2D Rectangle icon from the left hand side of the application.



Next, change the Layer Selector to the Board Edge Layer.

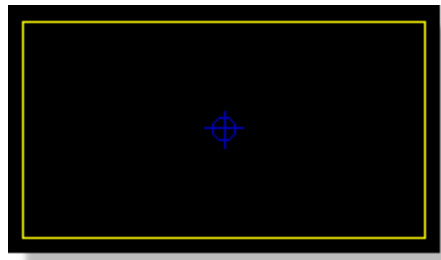


Move the mouse to the approximate starting point (e.g. top left) for the board edge. Now, hold the mouse still and press the 'O' key on the keyboard to set the origin of the co-ordinate system to the point under the mouse. This will be reflected in the co-ordinate display at the bottom right of the application. Left click to start placement and drag in the normal way. The co-ordinate display at the bottom will show you the dimensions of the board edge as you drag.

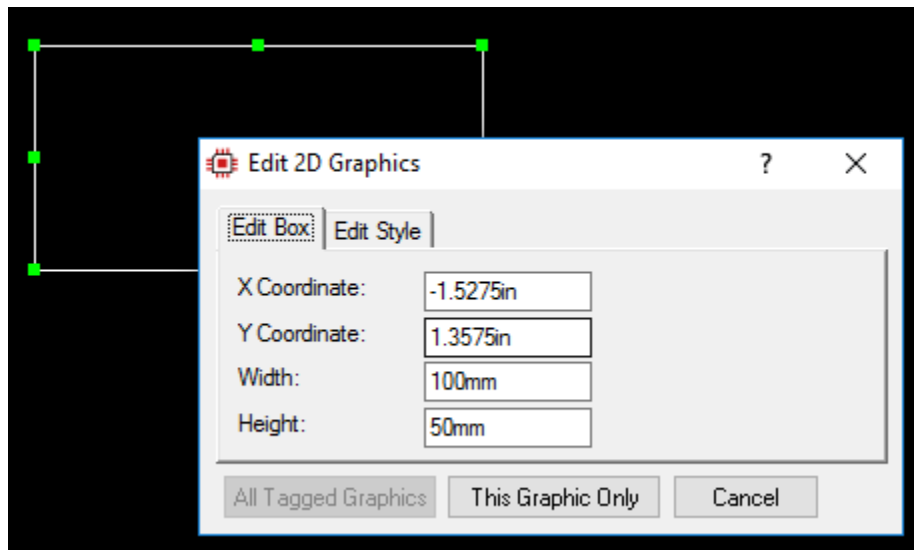


Drawing a board edge

Press the 'M' key on the keyboard to toggle between metric and imperial units as required. Once you have the desired size of board edge click left again to commit placement. Don't worry about where in the Editing Window you have placed the board edge – we will move it to the center of the world area shortly.

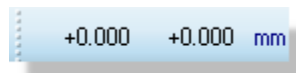


If you would like the board edge to be a specific size, Edit the board edge graphic and set the dimensions to your requirements:



Setting the dimensions of the board edge

Finally, restore the co-ordinate system to it's global origin by pressing the 'O' button on the keyboard again. The colour of the co-ordinates will change from Magenta to black to indicate that the global origin is now in use.

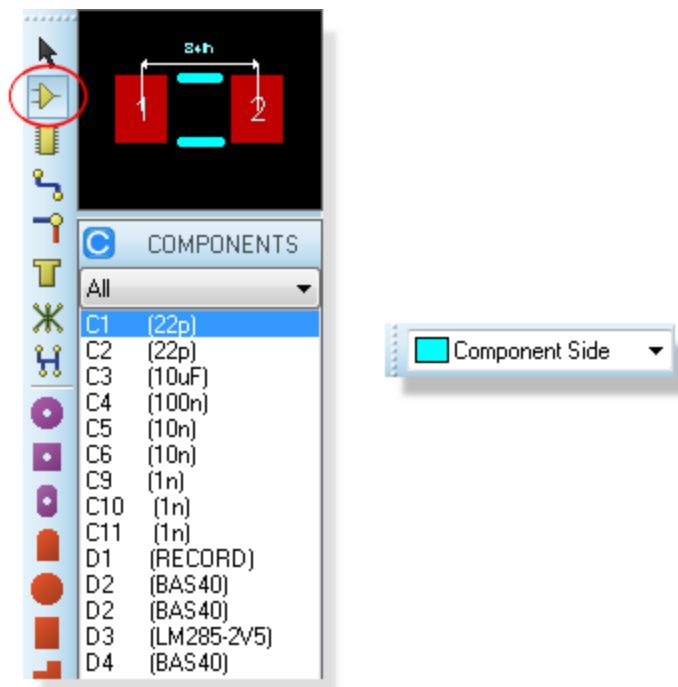


Board Edges of any complexity are best brought in from the MCAD tool via the Import DXF command on the File Menu (and placed onto the board edge layer !).

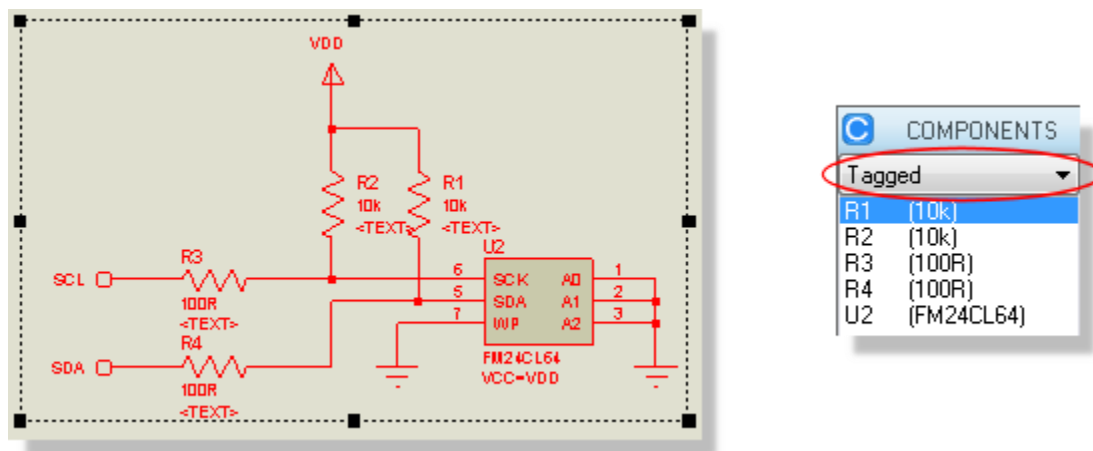
Placing Components and the Rastnest

Placing a component on the layout is very similar to that in the schematic module.

Start by selecting Component Mode from the left hand side icon set and then ensure that the layer selector is set to Component Side – we will not be placing any solder side components in this project.



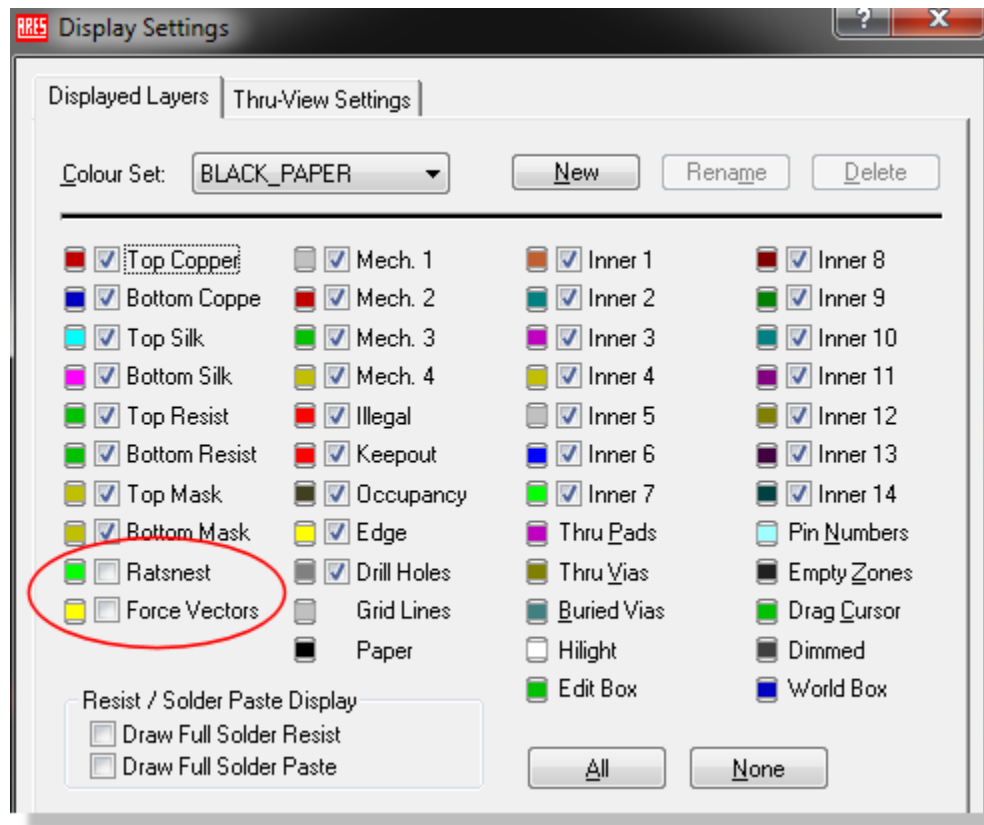
The object selector by default will contain a list of all parts to be placed. For more complex boards however it is worth noting that you can restrict the display to only items selected on the schematic or only those on the currently selected schematic sheet. This allows you to select parts of your design in the schematic and then work on placement and routing of them on the layout without the clutter of other components in the object selector.



You should notice both during placement and afterwards that there are green 'elastic' lines between the two components and also a yellow arrow from each component. The green lines are ratsnest lines and indicate connections that have to be made between the two parts, whilst the yellow arrows (named 'force vectors') indicate a preferred position for the part to minimize ratsnest distance. The force

vectors are provided as guidance only and are based solely on logic to reduce ratsnest lines. You can disable these lines if it distract you as following

From the View Menu, select the Edit Layer Colours/Visibility command. The resulting dialogue form shows all the available layers 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.



Routing the Board

Having configured the board constraints we can now move on to actually making connections and routing the board.

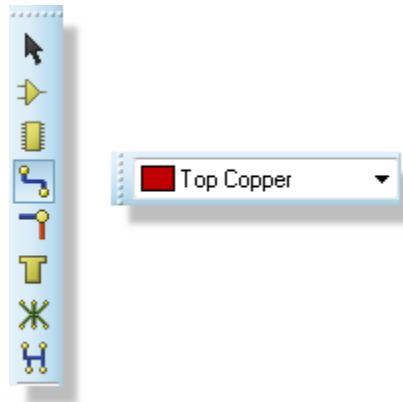
Placing a Route Manually

Before we start, we will want the ratsnest lines turned back on. If you have them disabled (can't see any green lines between pads), turn them on from the View Menu - Displayed Layers dialogue.

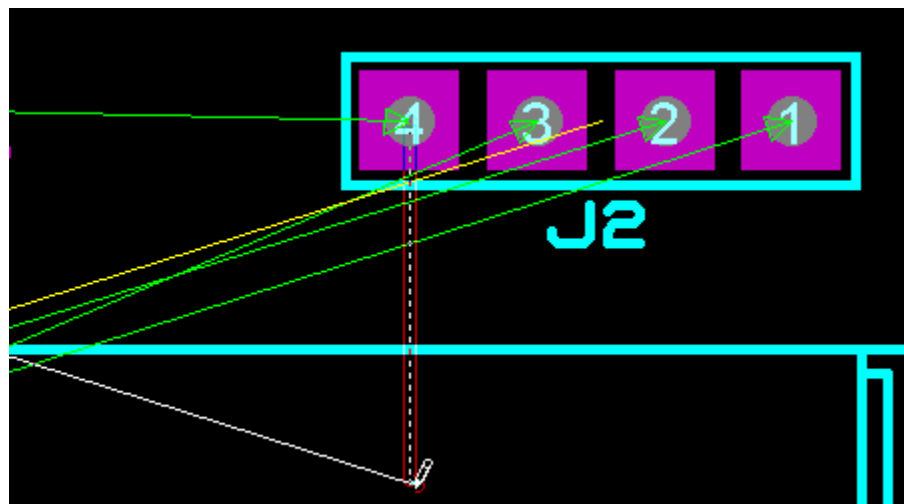
Let's begin by manually placing some tracks on the board. Typically, we would manually lay out connections where a specific path is desired for the track or where we need greater control over track position. On our layout we want to make

sure that our connections from connector J2 follow a sensible path around the board so we can start here and route them manually.

Start by selecting track mode at the left hand side of the Object Selector and changing the layer selector to be on Top Copper.



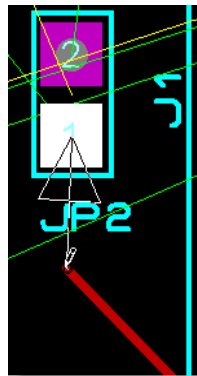
Proteus features a sophisticated 'follow me' routing algorithm for manual routing in which the route being placed will follow the path of the mouse as best it can while still obeying all of the design rules for the board that we set up previously. You start track placement by left clicking the mouse on pad 4 and then move the mouse downwards.



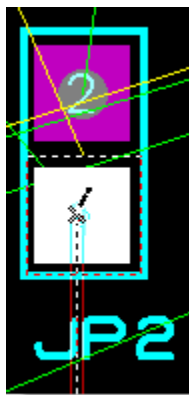
Routing J2 Pin 4

You should see that the closest legal destination for the track is now highlighted in white. This will update as we move the mouse and since we are not going to route to this destination we can ignore it for now. When the mouse approaches the bottom of the board, you could just change direction to the left and the track will corner to follow the mouse. However, since we want a tighter corner it is better to left click the mouse to place an anchor and then change direction to the left.

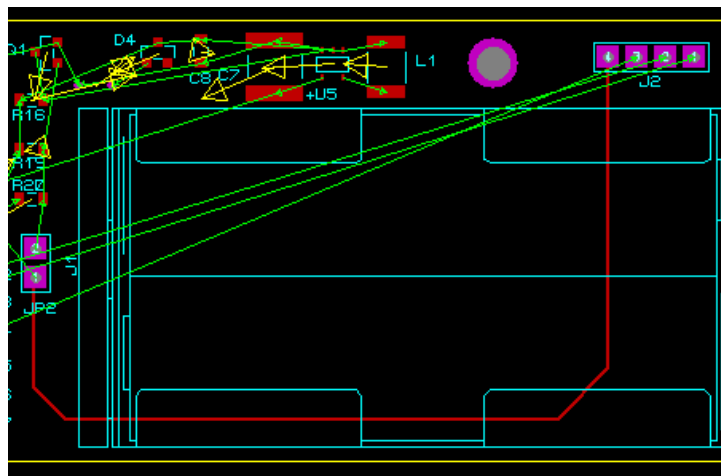
You will see that as we move further to the left the ratsnest guide will change to show us that we can terminate the track on pin 1 of jumper JP2.



Once we are underneath the jumper we can change direction and move upwards towards the destination pin



Finally, we can clicking left over the pin to finish placing the route. This will both commit the route and remove the ratsnest line corresponding to this connection.



Note that the width of track to route has a default value because it is configured for you automatically in Design Rule Manager. Manual routing is probably the most

common action you will perform with the software and it is vital that you understand how it works. The basic rules of operation are :

- Left click on pad, track or zone border to begin routing from that object.
- Left click at any point during routing to commit the route up to the mouse position (we call this anchoring).
- Right click to terminate the route at it's last committed / anchored point.
- ESCAPE key to abandon route entry completely.
- SPACEBAR to float a via on the end of the route and left click to then place the floating via.
- Double left click to drop a via at the current mouse position.
- Move the mouse backwards over existing tracking to rub-out.
- SHIFT key toggles the neck style on/off (SHIFT key down = neck style on, SHIFT key up = neck style off).
- CTRL key toggles curved tracking mode on/off (CTRL key down = curved tracking, CTRL key up = curved tracking off)

3D Visualization

Now that the board is now routed and ready for production we first want to examine it in 3D in order that we can properly preview how it will look in real life and possibly make final design alterations prior to prototyping. Start by invoking the 3D Visualization Engine from the module toolbar at the top of the application

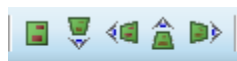


This will open and load the 3D view of the board in a separate tab. As with any tab, you can drag onto a separate frame if you want to look at both the 2D layout and the 3D view simultaneously.

Basic Navigation

The first thing we can do is view the board from different preset angles. Five preset views are supplied: top view, front view, back view, left view and right view and these are accessible via any of the following methods:

- Menu options on the View menu in the 3D Viewer
- From the navigation toolbar at the bottom of the 3D Viewer
- From keyboard shortcuts F8 through F12 whilst in the 3D Viewer.

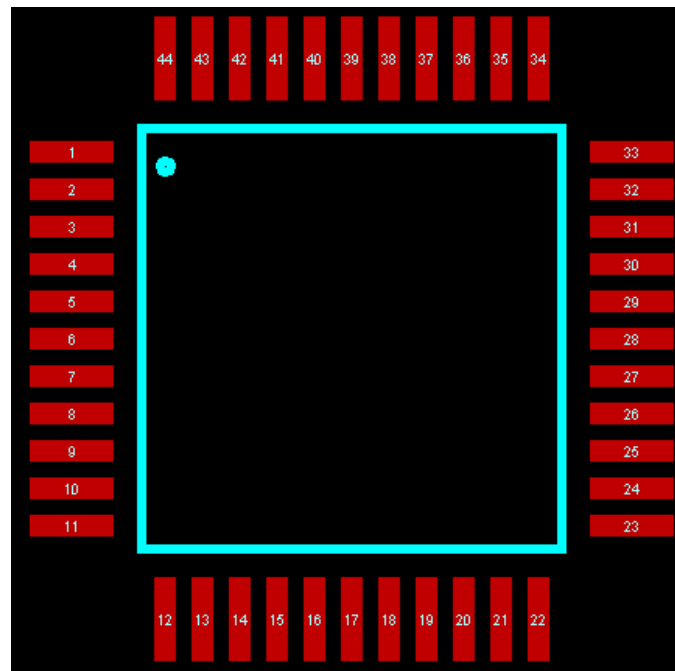


APPENDIX: Creating New Packages

Proteus comes pre-supplied with a large quantity of footprints, and we have seen previously how to select and place these parts on to a layout. However, it may be necessary at times to create your own custom footprints or symbols – also a simple task with Proteus – and this process is detailed below.

Drawing the Footprint

In our example, we will create a SQFP44 footprint with 0.8mm pitch and 12mm width.



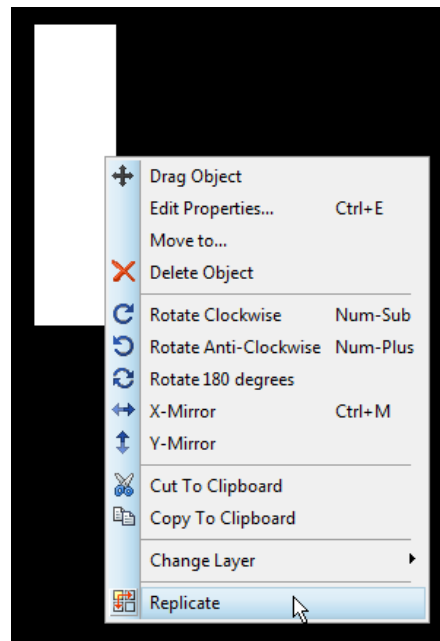
Start by selecting rectangular SMT mode. We want a pad 0.5mm by 1.8mm that should already exist as M0.5x1.8.



If the pad style does not exist you can easily (Click on the 'C' button above the Object Selector) and specify the pad properties (i.e. name, width, and height).

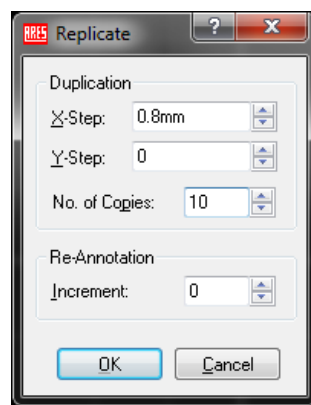
Make sure that the Layer Selector is on Top Copper and place one of the pads in the usual way. Right click to cancel placement immediately after placing a single pad and then left click on the pad to highlight it. With the pad highlighted, go to

the Edit Menu and select the replicate command. Alternatively you can right click on the pad and select replicate from the resulting context menu.



The replicate command will action on tagged objects so you should ensure that you have only the placed pad highlighted before invoking this command.

There are 11 pads on each side of the footprint so we need to replicate 10 times with an X-Step of 0.8mm as per the following screenshot.

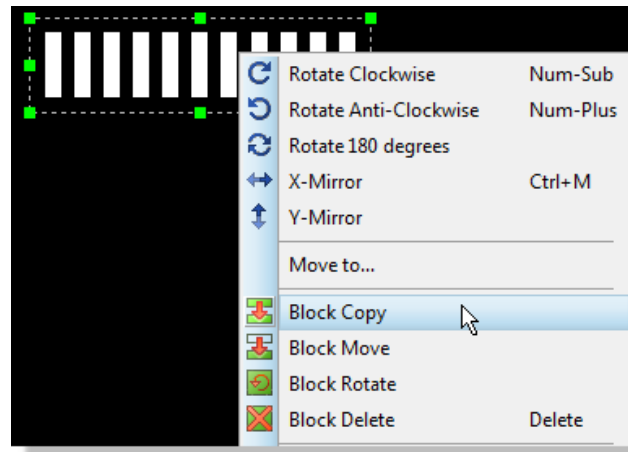


Note that you need to include the units of either millimetres (mm) or Thou (th) when entering values in to the X/Y-step fields.

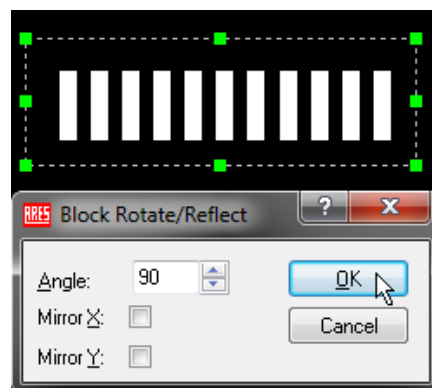
This will give us a single row of pads of the correct pitch (zoom in, change the snap settings and measure if you would like to confirm). The next step is to duplicate this row of pads to form the bottom of the footprint.

Start by right dragging a tagbox around the entire row of pads and then go to the Edit menu and select replicate. This time we want only one copy, 12mm down (or up) from the current row. Use negative co-ordinates if you want the duplicated row beneath the current one or positive co-ordinates if you want it above.

We now need to repeat the process with the other two rows. Drag a tagbox around a full row, right click inside the tagbox and select Block Copy from the resulting context menu.

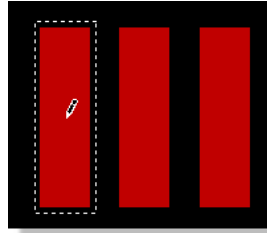


Move the mouse and drag the copy to a free area, left clicking to place and then right clicking to exit copy mode. Now, drag a tagbox around the row, right click and select the Block Rotate option, specifying a 90 degree rotation to align the pads correctly.



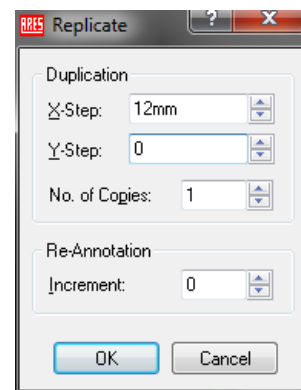
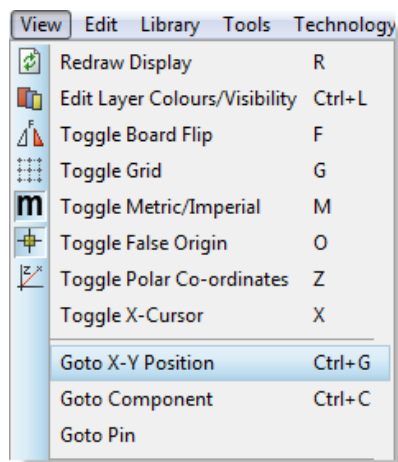
In order to position this row of pads correctly it's best to first set place a marker on the location we want to move the row of pads to. For this footprint the center of the top pad on the left hand side is 2mm below and 2mm to the left of the center of the left hand most pad on the top row. This gives us enough information to accurately position the row.

Select marker mode in preparation and then move the mouse over the left hand most pad on the top row until it is encircled. You'll want to be on a fairly precise snap setting for this (e.g. 0.5mm or F2 shortcut key).

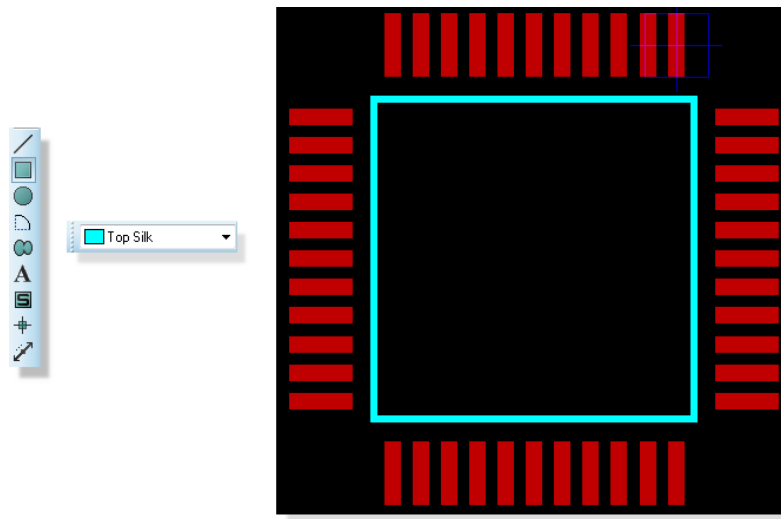


If you find your snap settings in thou you need to switch to Metric mode, either by hitting the 'M' key on the keyboard or via the Metric option on the View Menu.

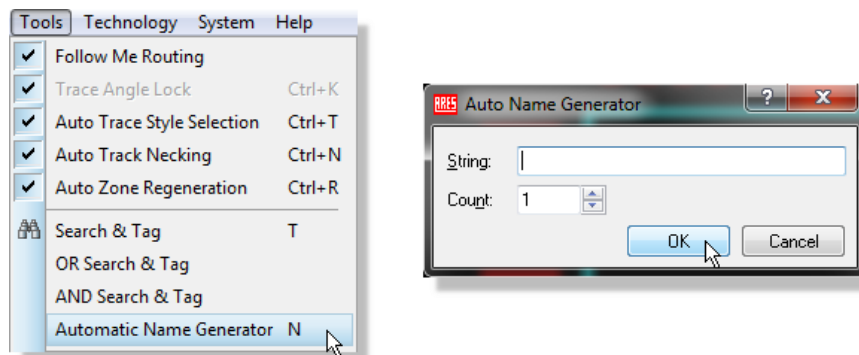
Now, hit the 'O' key on the keyboard to set a false origin at this location and then invoke the Goto-XY command from the *View* Menu.



Adding the silkscreen graphics is now straightforward. Select the 2D Graphics Line icon, make sure the Layer Selector is on top silk and place four lines along the inside edges of the pads to form a box. You'll find this much easier to do if you change the snap settings upwards (e.g. F2).



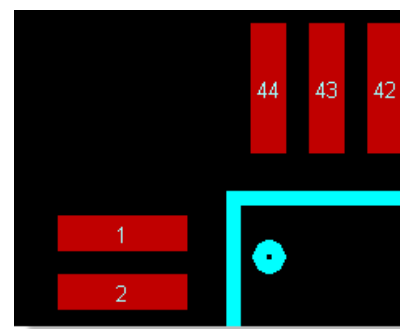
The next job we have is to number the pads. Start by invoking the Auto Name Generator from the Tools Menu. We don't need to enter anything in the string field here; simply leave the defaults and click on the pads consecutively to number them, Number 1 is the top pad on the left hand row.



Remember to hit the escape key on your keyboard when you have finished numbering pin 44 to exit assignment mode.

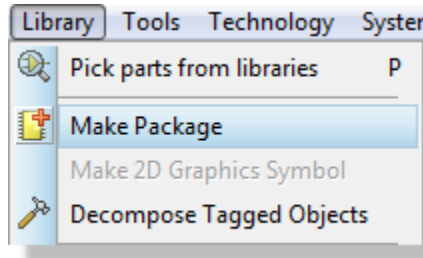
It's common to place a small dot beside pin one and you can do this via the 2D Graphics circle icon. Make sure that the Layer Selector is on Top Silk and turn the snap setting down to minimum (CTRL+F1) to achieve finer control over the size.

Place a small dot where Pin 1 is



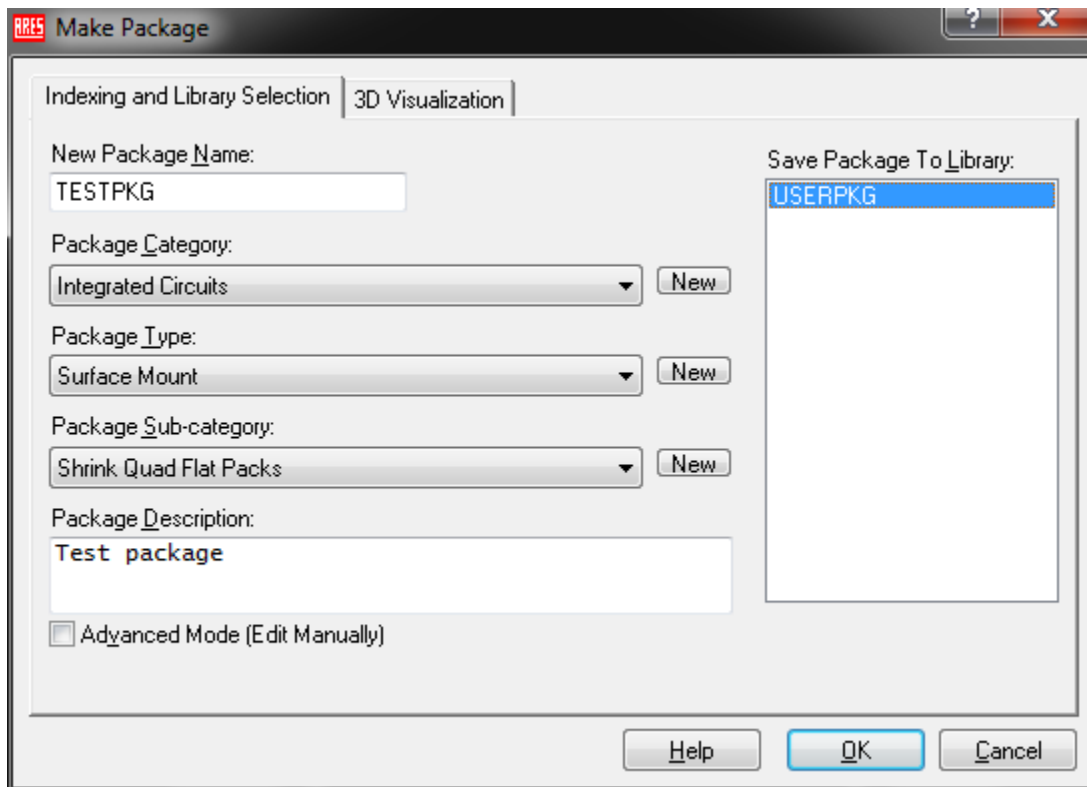
Packaging the Footprint

Drag a selection box around the entire footprint and then select the Make Package command from the Library Menu.



Launch 'Make Package' from the Library menu

The first screen is fairly self-explanatory and similar to that we've seen in the schematic application. Note that the package description is searchable when we are browsing for footprints so a little effort to make this as descriptive as possible is worthwhile. You will also want to create the part in the USERPKG library which is the default library supplied for user footprints. For our purposes, we'll call this package TESTPKG and give it some basic entries.



Task #2:

Each student should begin with schematic created in task #1 to properly PCB layout the audio amplifier circuit into real board.

➤ **Note:**

Please watch lab#2 tutorial on YouTube for more details on how to use Proteus Design suite. It will summarize all basic concepts covered through this lab Documentation.