



Altium Designer Suite Tutorial Notes

Notes to accompany METR2800 and ENGG2800 Altium Tutorials

Written by: Peter Bleakley

Instrumentation Support Group

Faculty Workshop Group

Faculty of Engineering, Architecture and Information Technology

University of Queensland, Brisbane, Australia

CRICOS Provider Number: 00025B

Contents

1.0	Introduction	3
2.0	The Project Design Space	3
2.1	Project Documents – Quick Reference	3
2.2	Getting Started.....	5
3.0	The Schematic Document	11
3.1	Basic Sheet Manipulation Tools.....	11
3.2	The Schematic Sheet – Quick Reference	13
3.3	Placing Components on the Schematic Document.....	13
3.4	Connecting Components Together on the Schematic Document	15
3.5	Place V _{CC} or GND Power Ports.....	16
3.6	Advanced Tools and Options	17
3.7	Add Components to the PCB document (from the schematic document).....	23
4.0	The PCB Document	25
4.1	The PCB Document – Quick Reference	25
4.2	Placing Components on the PCB Document	27
4.3	Designing the Board Shape	27
4.4	Design Rules	28
4.5	Placing Tracks or Routing the PCB	30
4.6	Text Strings and PCB Identification.....	34
5.0	Altium Component Libraries	34
6.0	Custom Component Design	36
6.1	Footprint Design.....	36
6.2	The Schematic Library and Symbol	41
7.0	Summary	44

1.0 Introduction

This document is designed to introduce readers to the Altium Designer software package. Altium is a Printed Circuit Board (PCB) design package used to easily and simply produce electrical and electronics design documents. This document serves as a basic introduction to the fundamental principles and tools used to capture circuit board design. These notes are designed to accompany the introductory Altium tutorials provided by the University of Queensland.

The current version of Altium that is in use at the University of Queensland is Altium Designer Summer '09. These tutorial notes may include some references to older and later versions than this, but most of the basic functions remain the same between versions. Altium 10 has some major changes in regards to layout of board options and menus, but these can be figured out by experienced users.

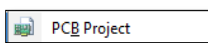
2.0 The Project Design Space

Altium is able to link documents together so that a single change on one document can be reflected in the second linked document with ease. This is a useful tool as, during the design process, a prototype design can change many times before even one circuit board is manufactured. The first section of this document relates to setting up a project in the correct way so that the most powerful features of Altium can be utilised with ease. Altium can work without the documents placed into a PCB project but the functionality will be greatly reduced.

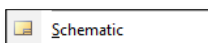
A quick reference guide is provided at the start of this section and then procedures are explained in more depth in the following chapters.

2.1 Project Documents – Quick Reference

There are five main items that the user should be familiar with:



PCB Project – The Project serves as a folder that links all the documents contained within it. All these documents should be able to pass information between each other. Any documents not contained within a project are unable to interact with documents within the project.

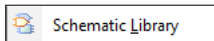


Schematic Document – This document is a simple, graphical explanation of the **function** of the circuit. Any electronics engineer should be able to look at a schematic document and understand the function of the circuit relatively quickly. The size of the symbols for the components is no indication

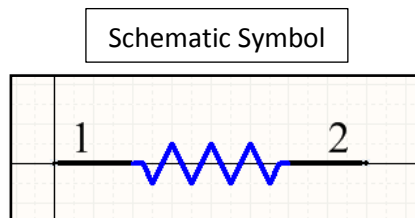
of the actual physical size of the components on the circuit board. The user should try to keep wires short and direct, avoiding cross-overs where possible.



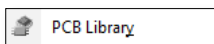
PCB Document – This document contains the **physical size** and shape of the Printed Circuit Board (PCB). It shows all components, tracks, labels, mounting holes, and the board shape in a scale diagram that can be manipulated and designed by the user. This document is what a manufacturer will attempt to replicate in fine detail when producing a PCB.



Schematic Library – This is a library of schematic symbols of components. It is common for libraries to group similar families of components (e.g. resistors, capacitors, microprocessors, etc.) together in separate libraries for quick reference. Altium has integrated libraries that group components by their manufacture as well as a couple of generic libraries of basic component shapes. A single schematic symbol may have a number of component footprints attached to it. E.g. For a resistor, the physical shape can vary between 5 Watt, 2 Watt, 1 Watt, 0.5 Watt, 0.25 Watt, 1206, 0805, 0603, 0402 and many others. All of these physical component shapes can be linked to one single schematic symbol for the resistor.



From left to right: 5W, 2W, 1W, 0.25W, 1206, 0805, 0603, 0402 resistors

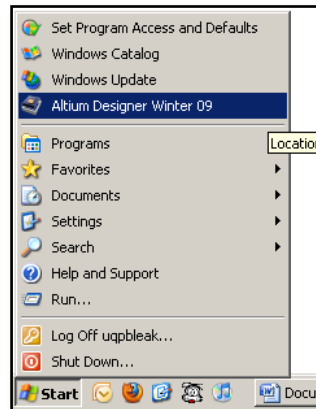


PCB Library – The PCB library contains the physical shape and size of each component, including any 3-dimensional component bodies that are drawn on the mechanical layers. Again, it is common for similar families of components to be grouped together in separate libraries for quick reference.

2.2 Getting Started

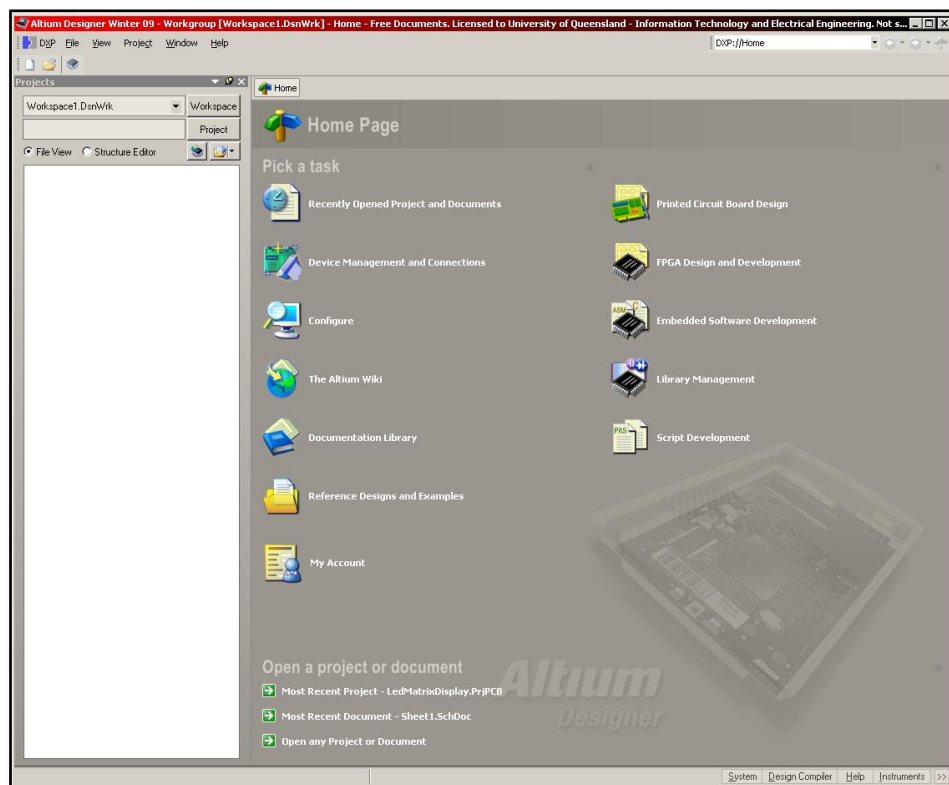
- Opening Altium Designer

Altium places its own shortcut into the Window's Start Menu. The simplest way to start Altium is to open the Start Menu and click on this shortcut.

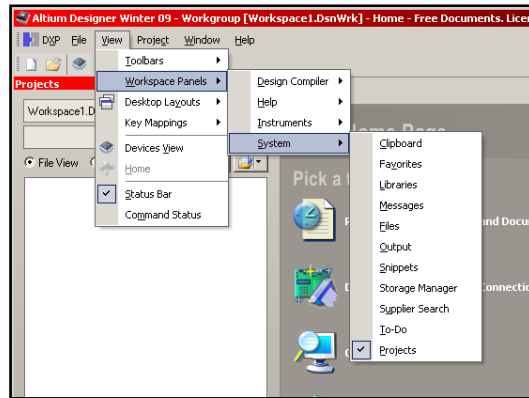


If no shortcut is available then the user should look in the 'Programs' menu to see if there is an Altium folder available. The Altium Designer Winter '09 shortcut should be available in this folder.

Upon opening the program, the user will be presented with an opening window similar to the one below. This is the home page and is used to set up the design workspace.



The main portion in grey contains shortcuts to some of the most frequently used parts of the Altium Designer. The main area of interest, however, is the white area on the left of the main screen. This area can be used for a multitude of tasks and normally has some tabs at the bottom which allow the user to view a number of different pages. The default page is the 'Files' page where a user can select to open new documents and projects, but the most useful page is the 'Projects' tab. When the 'Projects' tab is selected, the Project Window will be visible and it is used to control the structure of the Design Workspace. If no 'Files' or 'Projects' window is visible, the user can navigate the menu bar and click **View>>Workspace Panels>>System>>Projects**.



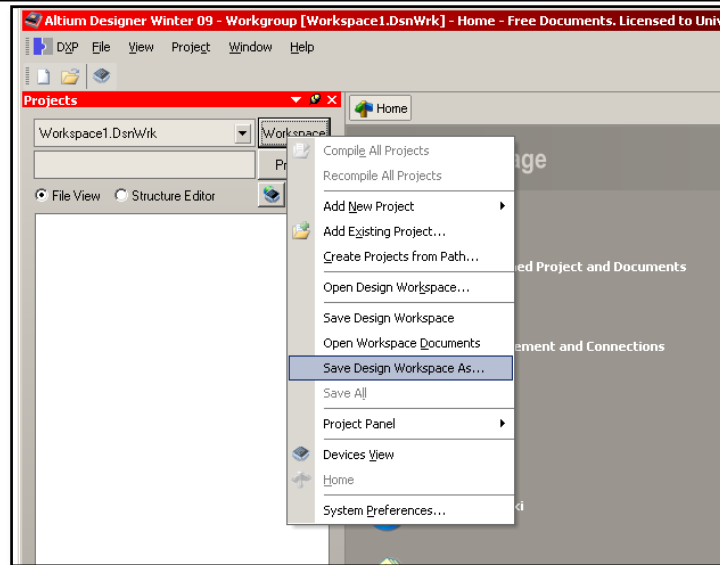
A) Setting up a Workspace and Project

In Altium Designer Suite, all the documents are able to interact and pass information between them. This makes designing a circuit board a lot less time-consuming as the software does most of the work. In order for the software to work correctly, the user must first set up a design workspace. The user can place many different 'Projects' in this area, but the use of this word at this stage may be a little misleading. The design workspace should contain all the documents relating to a single design project that the user is working on. For each new electronic design project there will be a new design workspace. Normally each new PCB project is used to represent a new revision or release of the circuit board.

The user should follow the next few steps in order to set up a new PCB design workspace.

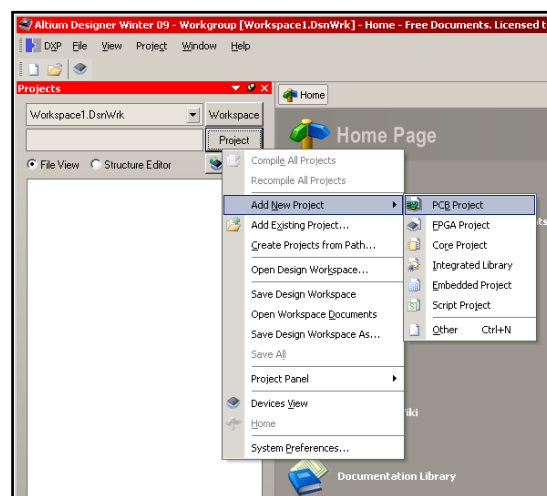
1. Save the design workspace with an appropriate name.

- Click on the button in the project window labelled 'Workspace'. This brings up a menu. Select 'Save Design Workspace As...' from the menu. Save the workspace to an appropriate folder on the computer.



2. Add a project to the workspace.

- Below the 'Workspace' button in the Projects window is a button called 'Project'. Click this button to open up the Project menu. From the menu select '**Add New Project >> PCB Project**'. This task can also be performed from the 'Workspace' menu if desired.

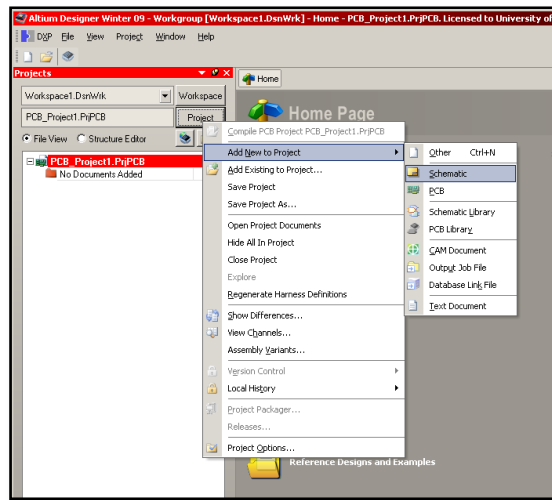


A blank 'Project' icon will appear in the 'Projects' window. The user is able to right-click on this icon to bring up a menu of functions.

3. Add a schematic sheet to the project.

- Click on the 'Project' button in the 'Projects' window to open up the Project menu. The user may notice that the menu has changed slightly in its layout. It is quite common in Altium Designer for the menus to change to give the user an appropriate list of functions that are possible at that moment in time. This helps guide the user throughout the design process and is very useful.

Click on '**Add New to Project >> Schematic**' to add a blank schematic sheet to the project. The user can also perform this task by right-clicking on the PCB project icon.

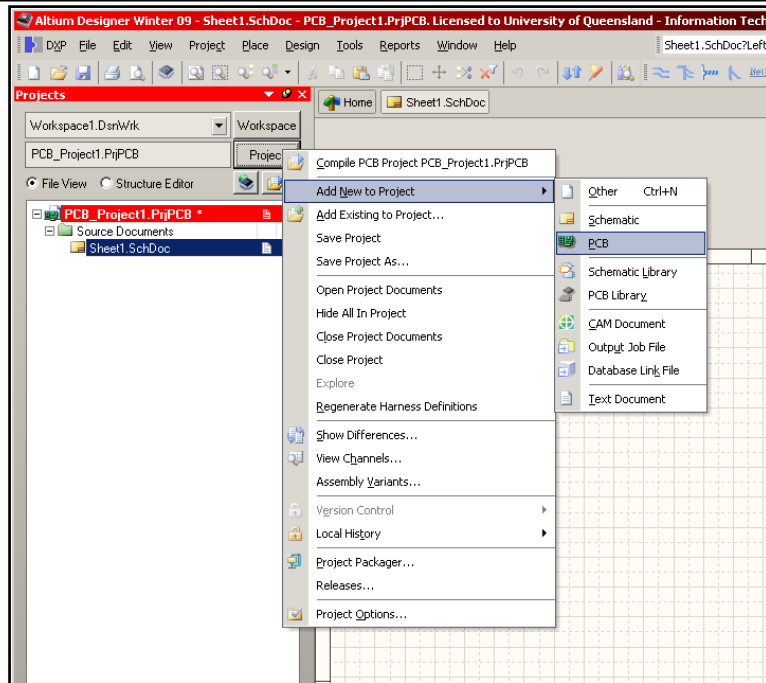


The new, blank schematic document will open in the main window of Altium and a schematic sheet icon will be added to the project in the 'Projects' window. Again, the user can right-click on this icon to perform useful shortcut functions and it is recommended that the user explore these shortcuts in their own time. The user will notice that the schematic icon in the 'Projects' window is indented slightly to illustrate that it is added to the PCB Project. If it is not indented then it may be under a heading of 'Free Documents'. This means that the newly created schematic document is not part of the PCB project. Dragging and dropping the schematic icon into the project will add it.

The user can add multiple schematics to a single project and link them all together. This is common for large projects but will not be covered in this tutorial.

4. Add a PCB document to the project.

- Similarly to the previous step, click on the 'Project' button in the 'Projects' window to open up the Project menu. Click on '**Add New to Project >> PCB**' to add a blank PCB sheet to the project.



Again, the new, blank PCB document opens in the main window and a PCB icon is added to the project in the 'Projects' window. The user will notice that the schematic and PCB documents have respective tabs at the top of the main area. The user will also notice that both document icons in the 'Projects' window are indented from the PCB project icon but are in-line with each other. This shows that they are both part of the same project at the same level and will be able to interact with each other.

Unlike schematics, it is not recommended to add more than one PCB document to a single project as this adds an extra layer of complication. It is best to keep one PCB per project and just open new projects for each separate PCB.



5. Add Libraries to the Project

When designing an electronic circuit, the components needed are stored in software libraries. These libraries can contain information about schematic symbols, pins, as well as a fully 3-dimensional mechanical representation of the component itself. Single components may have many different physical shapes (or packages), so multiple mechanical drawings of components can be linked to one schematic symbol. For instance, on the PCB document, the physical shape of the resistor may take the form of a through-hole component or a surface-mount component. There are even different sizes of resistor for different power ratings (see photograph on page 4 of these notes). However, on the schematic document, the symbol for the resistor would be consistently the same throughout the whole document as this is only a graphical representation illustrating the function of the component. Typically, a user will have one schematic symbol with a number of mechanical footprint models attached. The schematic symbol will be stored in a schematic library and the mechanical footprint model will be stored in a PCB footprint library.

Altium Designer Winter '09 and Summer '09 have some libraries supplied with the software design suite, but generally these are limited in their use and it is common for users to develop their own component libraries over time.

Altium 10 is only supplied with the 'Miscellaneous Components' and 'Miscellaneous Connectors' libraries. All other libraries are able to be downloaded by registered users from the Altium website. These component libraries are sorted by manufacturer and are generally kept up to date.

- All component libraries must link to the project in which they are going to be used. To add a schematic library to the project, the user will need to right-click on the Project icon in the 'Projects' window and select **'Add New to Project... >> Schematic Library'** from the project menu. Altium will add the schematic library to the project, organising it into a different folder structure within the project. Altium Designer will automatically sort these library files according to their type and will add an icon for each library to a folder in the 'Projects' window. The same process is used to add a PCB Library to the project. The user can now add components to these libraries but must first save the libraries before importing the components to the schematic or PCB documents.

More detailed notes about schematic and PCB libraries can be found towards the end of this tutorial.

6. Save all documents in an appropriate project folder.

- Click on the 'Workspace' button in the Projects window to open up the workspace menu. Click on 'Save Design Workspace'. This button will check all the documents throughout the design workspace and, if they are not saved, will open a dialogue box requesting the user to select an appropriate file name and location for each document.

NOTE 1: The user needs to take special note of the file named at the top of the 'Save As' window in order to establish which document is currently being saved as several different 'Save As' dialogue boxes will appear; one for each document, project or library that needs saved. The top bar of the 'Save As' dialogue box will clearly state which file is being saved.



NOTE 2: The folder structure shown in the Project window within Altium is not automatically replicated when the user saves files. Altium is able to let the user specify whichever folder they wish for each document's location. However, it is recommended that the user keeps the project file well organised and easy to navigate.

3.0 The Schematic Document

The schematic document is intended to quickly and concisely show any reader how the circuit is meant to function. The component symbol in this document does not relate to the mechanical size of the actual component but merely the function. The more information contained on the schematic, the more a reader will understand about the circuit and how it is intended to function.

The user must remember that Altium will, in the next stage, use the schematic document to generate the components and electrical connections for the PCB document. This can speed up the design process, but also means that there is added importance on the schematic design stage as any mistake at this point will be carried through the entire design process.

3.1 Basic Sheet Manipulation Tools

The following are some basic mouse movements and key presses that will help the user to manipulate the documents throughout the design stage. Most of these functions will work on both the schematic document and the PCB document:

Hand Tool – Move the Document

- **Click and hold the right mouse button** on the schematic or PCB document. The mouse pointer will change to a hand icon and the user can move the document in any direction using the mouse.

Zoom In and Out

- **Hold down the CTRL key on the keyboard and scroll the mouse wheel** in either direction. This allows the user to zoom in and out. This can also be done by clicking and holding both left and right mouse buttons at the same time and moving the mouse up or down. Also, pressing the PAGE UP key will zoom in and pressing the PAGE DOWN key will zoom out.

Select Multiple Components

- Similar to any Microsoft® software package, **holding down the SHIFT key on the keyboard and left-clicking on multiple components** allows the user to select many components at once.

Rotate Component or Change Direction of Wire

- When placing a component on the schematic or routing a wire (net) on either the schematic or PCB document, **pressing spacebar** once will rotate the component or wire by 90°.



Continual pressing of the spacebar will continue to rotate the component or net by a further 90°.

Quick Copy Component (Schematic document only)

- **Hold down the left SHIFT key and left-click, hold and drag any component** on the schematic document. The original component will stay and a copy of the component can be dragged to a new location by the user.

Move Component Keeping Connections (Schematic document only)

- **Hold down the CTRL key and left-click, hold and drag any component** on the schematic document. The component will attach to the mouse-pointer and can be dragged to a new location, but the wires attached to the component remain connected and will attempt to follow movement of the component.

Double-Click to Open Properties

- **Double click on any component, wire or object** in the schematic or PCB documents to open that item's properties dialogue window.

Change Properties while Placing

- While a component, wire or object is still attached to the mouse (just before placement on the document), **press the TAB key** on the keyboard to open that item's properties dialogue window. This is particularly useful when sequentially numbering or naming pins or nets (wires). It can also be useful when routing nets (wires) on the PCB document.

'Find Similar Objects' and 'SCH Inspector' or 'PCB inspector'

- These are more complicated tools and instructions on how to use them are located in a later section of these notes. It is important to become familiar with these tools as they will speed up design work in Altium considerably. These tools are noted here simply to make the user aware that they are part of the everyday toolset needed to successfully navigate Altium Designer. 'Find Similar Objects' is accessed by **right-clicking on any component** on either the schematic document or the PCB document. It should be the top tool listed in the resulting menu. It is used to select and highlight a number of components that have the similar properties.

The SCH or PCB Inspector can be accessed in two different ways. It can be automatically run after using the 'Find Similar Objects' tool provided the correct box is checked, or it can be accessed at any time by **opening either the 'SCH' or 'PCB' menu located in the bottom right** of their respective windows. The Inspector allows the user to change properties which are common to the selected components. I.e. if the user has selected a number of tracks all with a width of 0.3mm, then the Inspector will allow the user to change the width of all the selected tracks to whatever value they choose.

3.2 The Schematic Sheet – Quick Reference

The following tools from the schematic document toolbar are used to place components on the schematic document and electrically join them together with wires. These tools are explained in more detail in the next section.



Place Part – Select a component from an installed library and place on the schematic document.



Place No ERC – Use this symbol to signify that a pin or wire has been intentionally left unconnected.



Place V_{CC} – Place a power connection symbol onto a wire.



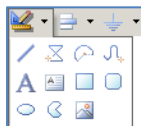
Place GND – Place a ground or 0V reference symbol onto a wire.



Place Wire – Electrically join two points together using a standard wire.

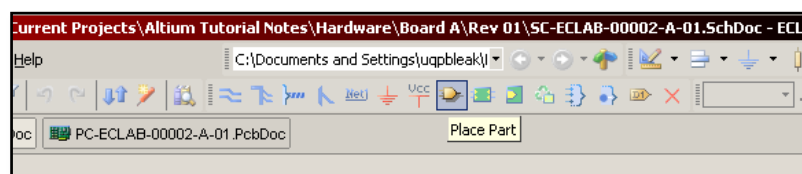


Place NetLabel – Label a wire with a custom name to electrically join it to other wires with the same name.



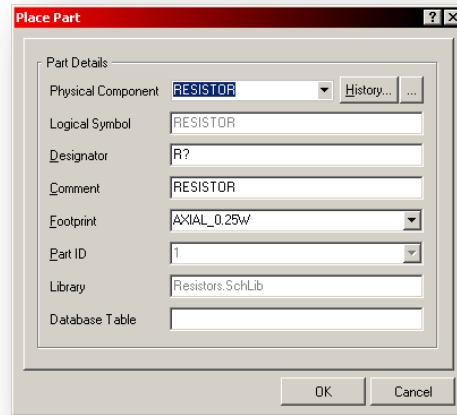
Drawing Tools – Place shapes or text onto the schematic document. These tools have roughly the same functions as Microsoft® Paint.

3.3 Placing Components on the Schematic Document



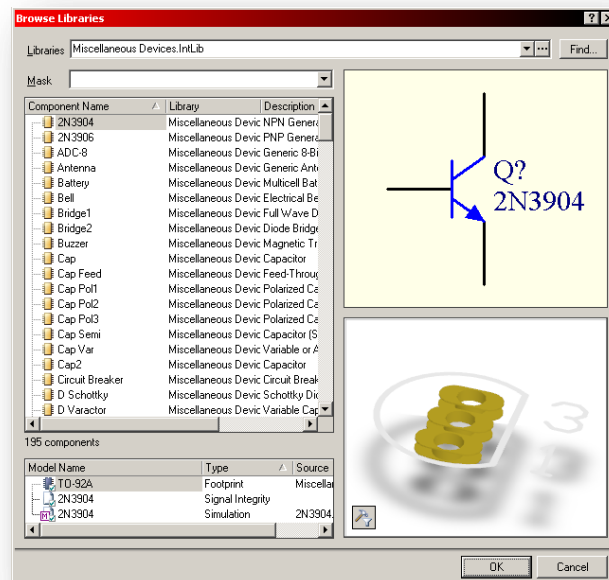
Altium refers to components as 'Parts'. Parts are stored in libraries and Altium contains two basic libraries which are built into the software. The in-built libraries of Altium can be quite limited, so it is common for users to create their own components and store them in custom libraries. PCB libraries contain the physical model of the component whereas the schematic library contains the functional representation of the component. The two are linked within the project by placing the schematic and PCB libraries within the same project as the schematic document and PCB document.

With the schematic document open and selected, clicking on the 'Place Part' icon (shown above) will open up a dialogue window (below).



This window shows some of the basic properties of the component that Altium last placed on the schematic document. This window can serve as a quick-placement tool if the user simply wishes to replicate the last component placement that they carried out. However, it is more common for the user to skip past this window and browse the libraries to select the new component that they wish to place.

Clicking on the button with '...' on it opens the 'Browse Libraries' window (shown below). In Altium Designer 10, this button is labelled 'Choose'.



Using this window, the user can select which component they wish to place on the schematic document. It is also possible to browse all the libraries installed within the project using the drop-down menu at the top of the window (labelled 'Libraries'). The library will only appear in this drop-down menu if it is saved at a specified location on the computer or network. If the library has not yet been saved, it will not appear in this list.

NOTE: The user must be careful as any new libraries can appear at the very top of the drop-down menu or at the bottom. Use the scroll bar at the right-hand side of the menu to navigate to the correct library.

When the user has selected which component they wish to place from the library, they should click the 'OK' button. This will load the component with its properties into the 'Place Part' window. The user should always check the component properties before they click the 'OK' button as they may wish to change something before the component is placed on the schematic. If the user is satisfied with the component properties then they may proceed to click the 'OK' button.

The schematic symbol of the selected component will appear on the schematic document as if it is attached to the mouse pointer. At this point, the user will notice that they can rotate the component by 90° by pressing the spacebar. Clicking the left mouse button will place the component onto the schematic document, but Altium will keep the symbol attached to the mouse pointer so that the user can place as many components on the schematic document as they wish. Clicking the right mouse button will cancel the 'Place Part' operation and return the user to the 'Place Part' window in order for them to select a new component to place on the schematic.

Once the user has finished placing all components onto the schematic, they should click cancel on the 'Place Part' window to return to the schematic document.

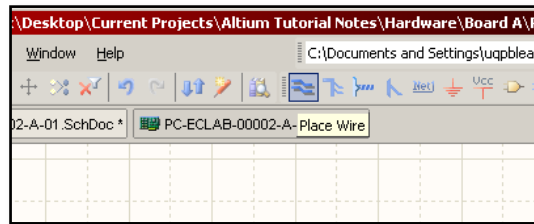
3.4 Connecting Components Together on the Schematic Document

Although the most obvious way of connecting components together on the schematic document is to use a wire, this can often lead to messy and difficult to read schematics with long wires intersecting and having no labels. It is for this reason that Altium has three different ways of connecting components together and these techniques help keep the schematic document neat and easy to read.

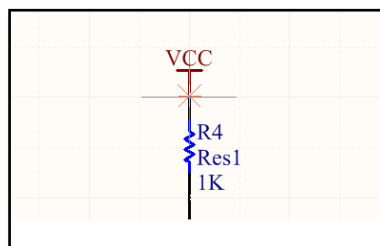
Wires or Nets

The terminology for a wire within Altium is a 'Net'. Nets are used to make simple connections between components that are close to each other. They are simple to understand and show a direct and obvious connection.

To place a wire, click the left mouse button on the 'Place Wire' tool in the top toolbar (see below).



The mouse pointer in the schematic document will become a cross-hair. Placing this cross-hair over the free tip of a component leg will change the cross-hair slightly. A red 'X' will appear (as shown below) and this indicates that an electrical connection between the component and the wire can be made if the user clicks the left mouse button.



Clicking the left mouse button will commence drawing the wire and will electrically connect it to the component pin. Any subsequent left-click of the mouse will place a 90° corner on the wire. The user can press spacebar at any time while placing a wire to change the direction of the wire corner. When the user has navigated to the component that they wish to connect with the wire, they will notice that holding the mouse pointer over the tip of the pin will again change the cross-hair to a red 'X'. This shows that an electrical connection can be made when the user left-clicks the mouse. Doing so will complete the wire and the two points will be electrically connected together.

If no red 'X' appears then no electrical connection will be made, even though a wire may be touching a pin. It is very important to make sure that all wires are electrically connected to the pins as Altium does not search for this type of error and assumes that the user has intentionally left pins unconnected.

3.5 Place V_{CC} or GND Power Ports



The wires connecting power and 0V reference (also called ground) to each component on the schematic are usually the longest wires in the whole circuit as nearly every component needs powered and also needs a reference voltage. This means that these wires can become particularly messy and may overlap other nets on the schematic, which is far from ideal. A more common method of notation for schematics is to use a symbol to represent the Vcc or 0V wires or nets. There are a few different symbols that can be used, but the most common in electronics is the 'T' bar. For a positive power voltage the 'T' symbol is upright and for 0V and negative voltages, the 'T' is placed upside-down.



It is not, however, the symbol which designates which net the point is connected to, but the name or label attached to the symbol. If +5V and +12V are being used in a circuit, it is important to label these different wires as such. Power wires can be positive or negative. Double-clicking on a power port will bring up that port's Properties and the name of the port can be changed. Make sure to label connected ports exactly the same name as Altium will distinguish between, for instance, +5V and 5V.

If using two different ground planes, i.e. analogue and digital, then using two different symbols to annotate them is recommended.

Place NetLabel

Similar to power ports, Net-Labels can be used to connect two component pins together without directly using a wire.

Clicking on the 'Place NetLabel' button will place a line of text on the user's mouse pointer. This text can be changed to any word, so the user can appoint descriptive names to wires, e.g. MOSI, MISO, CLK, etc. The word itself needs to be electrically connected to a wire or pin. By default, the electrical connection is the centre of the mouse pointer and will turn to a red 'X' when the mouse pointer is placed over a point to which it can connect. It is common for the designer to place a wire under the full length of the word as this signifies to the user that the word pertains to a wire or net.

The connection is completed when the user places the identical name on another net or pin somewhere in the circuit. The two words need to be identical or else Altium will not recognise the connection.

3.6 Advanced Tools and Options

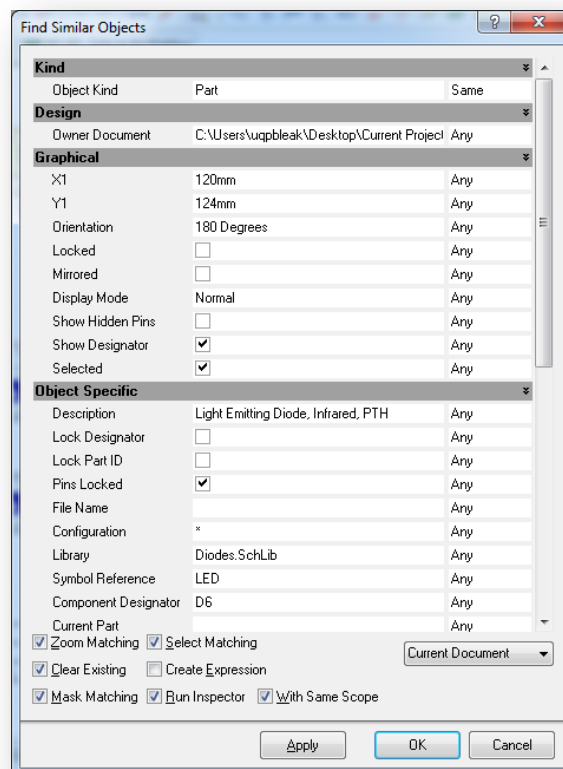
This section contains instructions on the following tools:

- Find Similar Objects
- Inspector
- Sheet Options
- Annotate Schematics

'Find Similar Objects' and 'Inspector'

These two tools are more advanced but can help with navigation in Altium a great deal. It is worth taking the time to understand these two tools as they speed up the design process immensely.

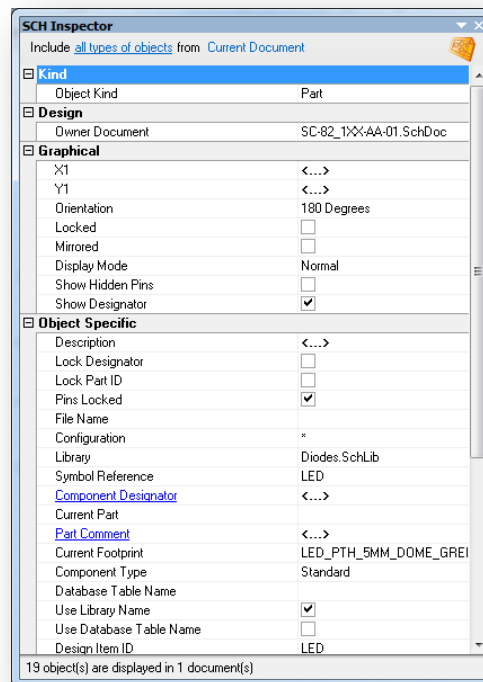
'Find Similar Objects' is used to select a number of components with similar properties. It is accessed by right-clicking on any component and this will open a menu. 'Find Similar Objects' should be located at the top of this menu. Selecting this tool opens the following window:



This window lists every single property relating to the selected component. By default, all the 'matching criteria' are set to 'Any'. This means that this specific property is not important and will not be used to refine the search. The user needs to figure out which property is common between all the components they wish to select and set the matching criterion to 'Same'. For Instance, if the user wished to select all LEDs in the schematic, they would right-click on an LED, select 'Find Similar Objects' and could choose the 'Same' matching criterion for either 'Description', 'Symbol Reference', or even 'Current Footprint'. The more criteria that are chosen, the more refined the search can be, but the user needs to understand what property it is that they are using to select the components and need to be sure that it is actually common to all the components that they wish to select.

There are a number of checkboxes at the bottom of the window that the user can select to tell Altium what they wish to do with the selected components. The most common and important checkbox is the 'Run Inspector' box. This will open the 'Inspector' after the user closes the 'Find Similar Objects' window.

The 'Inspector' is used to change any component properties that are common to any two or more selected components. 'Find Similar Objects' does not necessarily need to run first but it is a common method of selecting multiple components.



When the 'Inspector' is first opened, a list of all the properties common to the selected components is displayed. In the inspector window above, some LED schematic symbols have been selected. The user can browse all the common properties and change any property. The change of the property will be reflected in every selected component once the user presses the 'Enter' key or clicks on a new property.

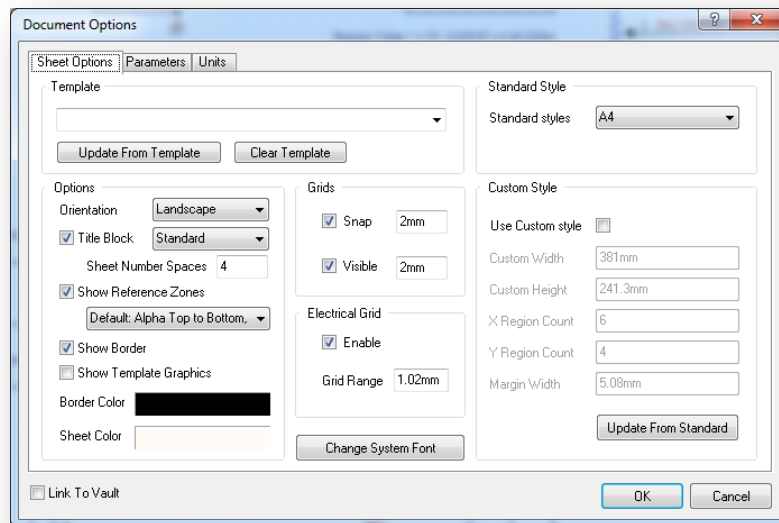
When the user is finished changing properties, they simply need to close the Inspector window.

The function of 'Find Similar Objects' and 'Inspector' are common to both the schematic document and the PCB document. The Inspector can be run at any time by opening either the 'SCH' or 'PCB' menu located in the bottom right of their respective document windows. The window will appear and the user can then return to the open document and manually select multiple components, wires or objects that they wish to change the properties of.

Change Sheet Options

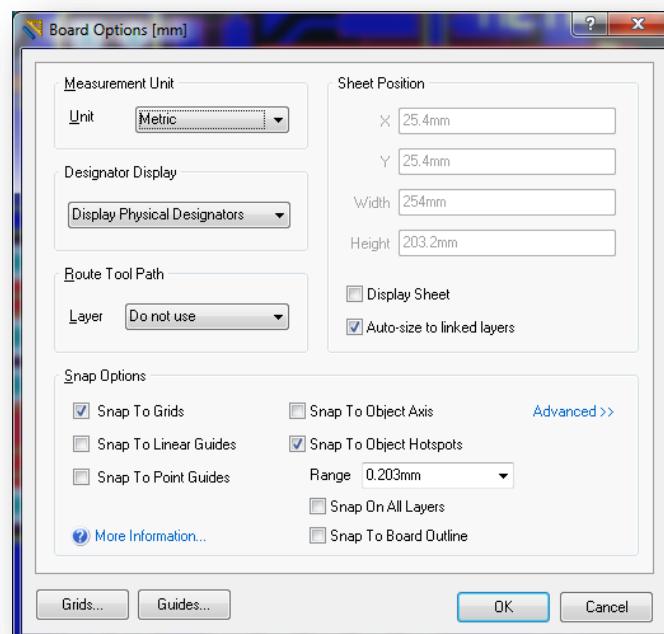
In the same way that components have properties, the sheet or document in which the components are placed also has properties which can be changed to suit the user's needs. Although Altium Designer names these options differently depending on which document the user is working on, the type of parameters or options that the user can access are very similar.

Schematic Document Options – Entitled 'Document Options', these are accessed by clicking on the menu **Design >> Document Options**. This menu brings up the following window:

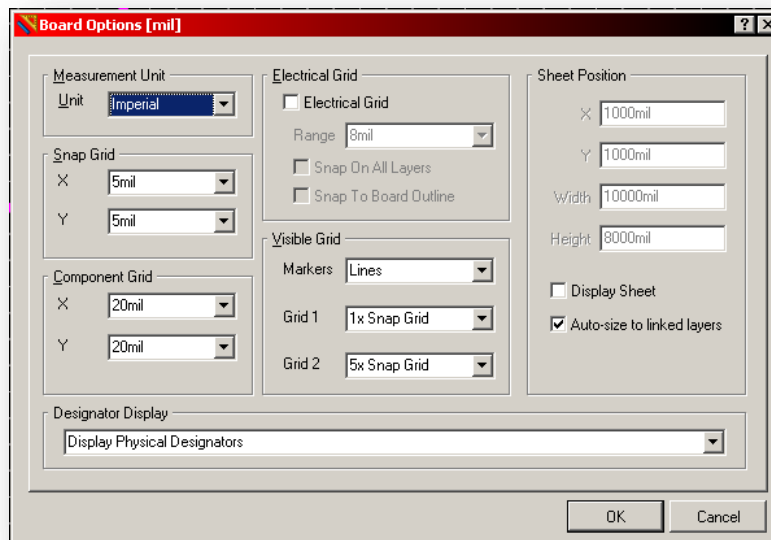


Here the user can change the grid size and the other basic properties relating to the schematic sheet. Using the tabs at the top of the window, the user can change the units between Imperial and Metric and can also add descriptive parameters which can be displayed on the sheet itself.

PCB Document Options – Entitled ‘Board Options’, these are accessed by clicking on the menu **Design >> Board Options**. This menu brings up the following window:



The above window is from Altium Designer 10 and is quite different from the display in Altium Designer Winter 09 (shown below).



The function of both windows is the same and they both allow the user to control the same options, but the layout of the options appears very differently.

The user can again change the units between Imperial and Metric and also change the grid properties. It is common to change the snap grid and component grid regularly throughout the design of a PCB, especially where component placement is critical to the end product. Increasing and decreasing the grid size and speed up component placement as the grid allows the user to accurately place multiple components at equal spacing.

In the Schematic and PCB libraries, these sheet properties can also be changed in much the same way. The only difference is that the name of the menu is 'Document Options' in the schematic library and 'Library Options' in the PCB library and the properties relate to the entire library, not just a single component.

Annotate Schematics

This tool is used to set each component with a unique and relevant designator. When each component is placed on the schematic, it is given a designator which usually contains one letter with a question mark after it (i.e. 'R?'). Altium requires that every component has its own unique designator which acts as an identifier for that component. This designator is also used to identify which footprint in the PCB document relates to each component on the schematic document.

It is not necessary for the user to change each designator manually. Altium uses a tool called 'Annotate Schematics' to give each component a unique designator. This tool can be found in the menu **'Tools >> Annotate Schematics'**.

When selected, the following window appears:



Proposed Change List					
Current			Proposed		Location of Part
Designator	/	Sub	Designator	Sub	Schematic Sheet
<input type="checkbox"/> C?		<input type="checkbox"/>	C1		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> C?		<input type="checkbox"/>	C2		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> CON?		<input type="checkbox"/>	CON2		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> CON?		<input type="checkbox"/>	CON3		SC-82_1XX-AA-01.SchDoc
<input checked="" type="checkbox"/> CON?		<input checked="" type="checkbox"/>	CON4		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> CON?		<input type="checkbox"/>	CON1		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D5		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D6		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D7		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D8		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D9		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D13		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D10		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D11		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D12		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D2		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D3		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D4		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D14		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D15		SC-82_1XX-AA-01.SchDoc
<input type="checkbox"/> D?		<input type="checkbox"/>	D16		SC-82_1XX-AA-01.SchDoc

Annotation Summary
Annotation is enabled for all schematic documents. Parts will be matched using 2 parameters, all of which will be strictly matched. (Under strict matching, parts will only be matched together if they all have the same parameters and parameter values, with respect to the matching criteria. Disabling this will extend the

If the user is satisfied that the new designators are correct then they should click the 'Accept Changes (Create ECO)' button. A new window will open where the user can select the 'Execute Changes' button and Altium will change all the designators according to the changes specified by 'Annotate Schematics'. When the user closes the windows and returns to the schematic document they will see that every designator of each component has changed.

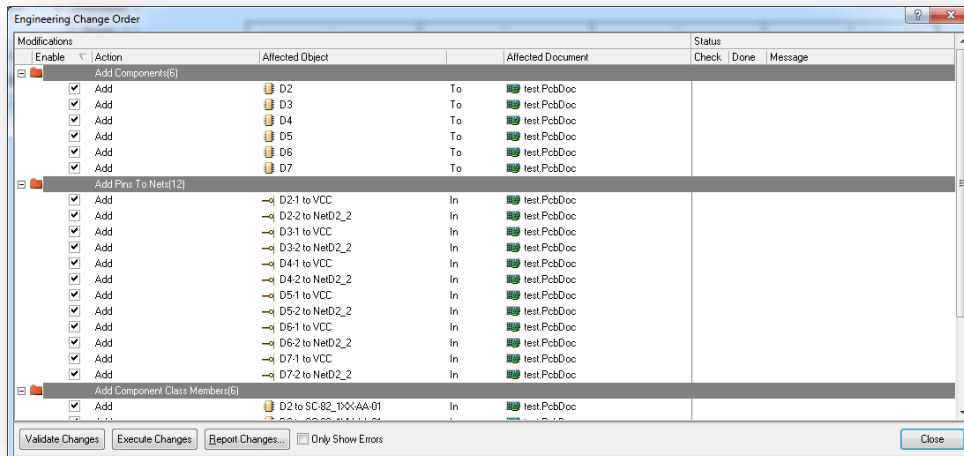
Note: The only cautionary note for this tool is for a special situation that occurs when components have already been added to the PCB and changes need to be made to the schematic. If the designators for the components change on the schematic then the PCB designators will also need to change. Altium can usually pick this up easily, but if multiple changes occur then the user may need to update the component links using the tool in the PCB document window entitled 'Component Links'. This is located in the Project menu of the PCB document window and opens a window showing which components on the schematic and the PCB document are not linked together. It asks the user to match the components and update the links. This can be a complicated tool and should only be used by experienced users of Altium Designer.

3.7 Add Components to the PCB document (from the schematic document)

This tool illustrates the power and functionality of Altium Designer. A major strength of Altium is that it can use the schematic document to automatically update the PCB document with the correct components and also indicate to the user how these components should be connected together. This is just about as much as Altium can be expected to do as the rest of the design is up to the user's specifications and understanding. It does, however, save the user a lot of time and effort.

When the user is satisfied that the schematic document is finished, annotated, within a project and that a PCB document is open and added to the current project, then they should be ready to update the PCB document by adding nets and components to it.

In the schematic document, click on the menu **'Design >> Update PCB Document >> PCB document name.PcbDoc'**. In the background, Altium will, by default, compile and check the schematic document for errors. It will notify the user if any are found, but if not, it will proceed with the update process by opening a 'Engineering Change Order' (ECO) window.



This window details the changes that Altium wishes to make to the PCB document. It is usually a list of components and nets that need added to the PCB document.

Note: Altium groups components on multiple schematic sheets into sections called rooms. At this stage of the process, Altium suggests that it will add a room to the PCB document. This can be a handy tool for more advanced users, but it is common to simply not add the room to the PCB (see the bottom of the list) by de-selecting the check-box.

The user simply needs to click on 'Execute Changes' to proceed with adding the components to the PCB document.

Altium will add the components and nets one-by-one, verifying the component before it is added. The user is notified of any errors and must check the list to make sure that every component has been placed. If an error occurs at this stage, it is often because of a missing footprint or because the component is not found in the installed component libraries. If this occurs then the user should go back to the schematic document and check the problem component and that it can be found in the libraries.

If not errors occur then the component have been successfully transferred to the PCB document and the document will display automatically.

The components are often placed to the bottom left of the black space in the PCB document.

4.0 The PCB Document

The PCB document is the mechanical representation of the exact shapes and sizes of the PCB and the components. This is an exact representation of how the circuit board will look when it returns from the manufacturer. Any PCB is made up of a number of layers which are listed in tabs at the bottom of the PCB document window. The following list explains the general functions of some of the most commonly used layers:



- | | |
|-----------------------|---|
| Top Layer | – Tracking layer for connecting component pads together. |
| Bottom Layer | – Tracking layer for connecting component pads together. |
| Mechanical 1 | – Used for mechanical features of the PCB like 3D bodies, the board shape and any enclosure features that need to be accounted for. |
| Top Overlay | – Used to illustrate the component body boundaries to avoid component collisions when soldering components to the PCB, or used to place labels on the PCB. The top overlay appears on the top layer of the PCB. |
| Bottom Overlay | – Same as Top Overlay, but on the bottom layer of the PCB. |
| Keep-Out Layer | – A line on the keep-out layer will ensure that all other layers keep a set clearance away from the line. It is used to keep ground planes away from noisy components or to mark out mounting holes that are isolated from the PCB. |

4.1 The PCB Document – Quick Reference



Fit Document – This tool is used to fit all the components on the document within the space of the user window. It is useful if the user loses track of where some components are located on the PCB document as it will automatically show the location and layout of all components.



Fit Selected Area – Use this tool to fit a user-specified area to the PCB document view window. This is mostly used to view a selected area in close detail.



Select Inside Area – Drag a box over a number of components to select them.



Move Selected Objects – Use this tool to move the selected objects to a new location. The user must select the objects before activating this tool and must then click a point on the PCB to use as a reference for moving the objects.



PCB Drawing Tools – These tools are used to draw simple, non-electrical shapes on the PCB. They can be used for drawing the board outline, any keep-out lines, or marking dimensions for mechanical manufacture. The origin or reference point of the board can also be changed using the tool in this set.



Interactively Route Connections – This is the tool used to electrically connect two points on the same net together. It will draw a track on the selected layer from the initial starting point to any same net that the user specifies. It allows the user to track by any possible route, although it is beneficial to keep tracks to the shortest length possible. Pressing spacebar while tracking will change the direction of the track currently being drawn. Pressing the * key on the numeric keypad while routing will change between top and bottom layers using a via (pressing the shift + number 8 key will not work).



Place Via – A via is a hole that passes through the circuit board from the top to the bottom layer. It does not normally have any pin passing through it, so by default it is plated (electrically connected around the inside of the hole so that the top pad is electrically connected to the bottom pad). Vias can be placed automatically while tracking or by using this tool. Vias will always have some form of electrical connection in the form of a track or net.



Place Pad – A pad is similar to a via but may or may not have an electrical connection. They can be used as mounting holes for attaching a circuit to a mechanical structure. They can also be used as test points. Pads can also be used to mount heat-sinks and other mechanical structures onto the PCB.



Place Polygon – A polygon is a large, flat track of copper that can be any shape or size (in 2 dimensions). 'Place Polygon' is normally used to create ground planes or large, high-capacity tracks. Pressing this button will open a window prompting the user to set up the polygon pour. The user needs to set up the polygon pour and, upon closing the window, will then draw the outline of the polygon.



Place String – The user can place text on any layer of the PCB. This can be used to annotate the PCB and give it serial numbers, revision numbers and engineering information.



4.2 Placing Components on the PCB Document

Altium automatically places components onto the PCB document, but in no order. It is up to the user to place the components correctly. A good rule of thumb is to initially place the components in similar fashion to how they are on the schematic. This will ensure that the connections are as short as possible and that the PCB design follows some sense of logic. If connections are kept as short as possible, there is less chance of noise from other signals being picked up. Short tracks are also easier to draw and space is kept to a minimum.

If using a combination of through-hole and surface-mount components, it is common to place the through-hole components on one side of the board and the surface-mount components on the other side. This means that all the low-profile components are on one side of the board and the tall components are on the other side. This should allow for easier mounting within an enclosure. The layer on which components are placed can be changed by double-clicking on the component to open its properties.

There are a number of rules-of-thumb when it comes to component layout and these rules are too numerous to put all of them in a brief tutorial. However, the PCB designer should be aware of electronic noise and should try their utmost to avoid transference between noisy components and tracks on the circuit and the signals which need to be clean (like analogue signals). A very general guide is listed in the following section. Using separate ground planes for digital signals and analogue signals is a common technique. Separated ground planes should be connected at a single point close to the power input of the circuit, i.e. near the regulator or near the power connector.

Sources of Noise

- Any quartz crystal oscillator;
- Any high frequency tracks (radio or PWM);
- Microprocessors (avoid tracking underneath the microprocessor where possible);
- High current tracks (larger magnetic fields);
- Regulators and power supply components;

4.3 Designing the Board Shape

Before placing tracks on the PCB, the board shape must first be defined. The size and shape of the PCB is normally dictated by the shape and size of the enclosure, but this can be a trade-off between the mechanical requirements of the project and the functional requirements of the electronics. It is best to design the function of the circuit board (schematics) and get a rough idea of what board size this is expected to take up before selecting the mechanical enclosure. Selecting the right enclosure at this stage will set the boundaries and rules for the next phase of the PCB design.

Once the mechanical enclosure has been selected, the board shape can now be drawn, leaving room for mounting holes and physical component bodies. The board outline, along with any other mechanical aspect of the PCB, is drawn on the layer labelled 'Mechanical 1'. There are a number of mechanical layers available for drawing mechanical structures. In actual fact, the entire enclosure

that the PCB resides in can be imported into Altium if the user has time. This can help with component placement if space is of particular importance.

In the PCB document, with the 'Mechanical 1' layer selected (tab at the bottom of the window), the user should click on the 'Utility Tools' icon in the top toolbar. The user can use these tools to draw most shapes and dimensions as well as edit the position of the origin. Most simple board shapes are drawn using the 'Place Line' tool. The outline of the PCB must not have any gaps in it and should be as accurate as possible. No lines should extend beyond the boundary as this will cause conflicts to occur when the board shape is defined.

Hint: It is best to set the origin of the PCB to a corner of the board outline. This allows the user to accurately set the dimensions of the PCB using the XY coordinate system built into Altium.

Hint: Remember to set the 'Board Options' to allow simple placement of the components using the snap and component grids.

Once the outline of the PCB is drawn, the user should select each line of the outline and then select '**Design >> Board Shape >> Define from Selected Objects**' from the PCB document menu. This will limit the black design area to within the board outline which helps the user throughout the process of component placement.

4.4 Design Rules

The user would not wish to get a PCB back from the manufacturer only to find that the components do not fit and that some tracks are short-circuited together. It is for this reason that Altium has 'Design Rules' built into the software that constantly monitor the design as it is happening. These rules speed up the design process by simply limiting the user to component and track placements that are physically possible for the manufacturer to make.

The limits of the design rules need to be dictated by the manufacturer as different manufacturers have different manufacturing capabilities. These limitations are normally listed on the manufacturers' websites or can be obtained through conversation with the manufacturer. PCB's made in-house at the University of Queensland have very different rules to those made externally as the internal manufacturing process is significantly different. The design rules pertaining to the ETSG within UQ are listed on their website.

The design rules are accessed from the menu '**Design >> Rules**'. As a starting base, some of the more generic design rules are listed below.

Design Rule Suggestions

- >> Electrical
 - Clearance
 - Minimum Clearance = 0.3mm
- >> Routing
 - Width
 - Min Track Width (both top and bottom layers) = 0.3mm



- Preferred Track Size (both top and bottom layers) = 0.5mm
- Max Track Width (both top and bottom layers) = 2mm

- Routing Via Style

- Min Pad Size = 1.3mm
- Max Pad Size = 3mm
- Preferred Pad Size = 1.3mm
- Min Hole Size = 0.7mm
- Max Hole Size = 1.5mm
- Preferred Hole Size = 0.7mm

>> Plane

- Polygon Connect Style

- Conductor Width = 0.5mm

>> Manufacturing

- Hole Size

- Min Hole Size = 0.7mm
- Max Hole Size = 4mm

- Hole to Hole Clearance

- Hole to hole clearance = 0.15mm (maybe less if datasheet of component dictates)

- Minimum Solder Mask Sliver

- Minimum Solder Mask Sliver = 0.1mm

- Silkscreen Over Component Pads

- Silkscreen over component pads = 0.01mm

- Silk to Silk Clearance

- Silk to silk clearance = 0.01mm

>> Placement

- Component Clearance

- Min Vertical Clearance = 0.3mm
- Min Horizontal Clearance = 0.3mm

If an element of the PCB design is in violation of the design rules, it will show up as a bright green colour on the PCB document. The user can remove the violation notification by either clearing the violation (moving the track or component so that it is no longer in violation of the rule) or by resetting the error marker (select 'Tools >> Reset Error Markers' from the menu bar). With the latter case, the error will still remain but the user will not be notified about it unless they change the component in some way.

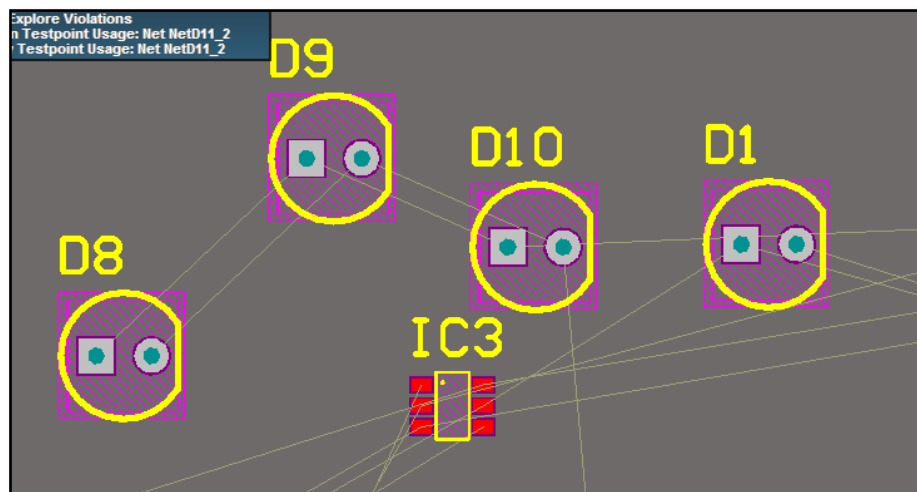
Right-clicking on a component or track with an error present will bring up a menu. The user can select 'Violations' from that menu and a list of the design rules that are being violated will appear. This will help the user determine whether or not the rule needs to change or if the component or track needs to be re-designed.

4.5 Placing Tracks or Routing the PCB

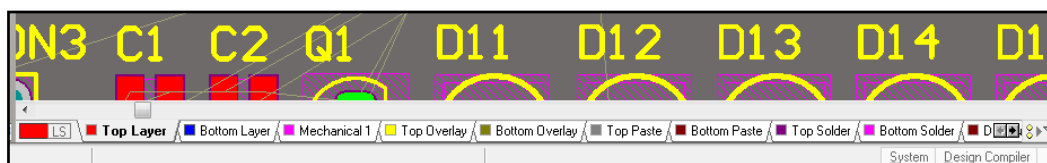
With the board shape defined and the design rules set, the user can now place components within the PCB area. Once this is complete, the user can then start to 'track' or 'route' the PCB.

Note: Auto-route can be a useful tool if correctly set up. It is considered an advanced tool within Altium and could take up a chapter of these notes on its own. Altium requires very specific design rules and auto-route rules for this tool to work effectively and therefore it is not recommended as a tool for Altium beginners. It may be tempting to use this as a means of saving time, but it will normally cause more design problems than it solves.

Altium will already have placed direct lines on component pads showing which pads are electrically connected together. It is up to the designer to draw the physical tracks which connect the pads together.



A number of tabs representing layers can be found at the bottom of the PCB document window.



The user must select which layer they wish to place a track on before they begin routing that track. The most common layers for tracking are the top and bottom layers. More complicated boards will use 4 layers or more, but most simple circuits can be tracked using just two layers.

For PCBs that are to be manufactured by a professional PCB manufacturing company, it is recommended to route most tracks on the top layer and leave as much space on the bottom layer for a ground plane.

Note: For PCBs that are to be made in-house within UQ, it is recommended that all tracks connecting to through-hole components are started on the bottom layer. This is because the ETSG workshop



does not have the capability to plate holes through the PCB, so soldering the components to the board on the bottom layer is recommended. Vias will also need to be hand-soldered using a wire through the centre of the via and soldered on both the top and bottom layers.

When the user is ready to track or route the PCB, they should click on the 'Interactively route connections' button found close to the middle of the toolbar.



- Interactively Route Connections

Any subsequent click in the PCB document area will commence drawing a track. The track will be electrically connected to the net that the user clicked on. If the user clicked on a blank space, the track will not be connected to any net. If the track is started on the top layer, it will be coloured red. If the track is started on the bottom layer, it will be coloured blue. The track will attempt to follow the path of the mouse, but the user can left-click at any time in the tracking process to 'lock' a track in place. This usually occurs at bends and corners in the track.

Note: It is best to avoid 90° angles in PCB routing as this will increase the electromagnetic noise generated by the circuit, especially in high frequency and digital tracks.

The user will notice that the mouse cursor will 'lock on' to the centre point of any component pads. This feature is called the electrical grid and is set within the document options window. It can be turned off or the range altered if necessary.

When the user has completed a connection between two components the faint grey line that was automatically generated by Altium will disappear as the track is now electrically joined. Altium will continue to route the connection however, until the user right-clicks the mouse. One right-click will cease drawing that track and will allow the user to select another connection to route. Two right-clicks will stop drawing tracks altogether.

Note: A general method for selecting which tracks to place first is to start with the shortest and most obvious connections, working up to the longest tracks with the most connections. Leave VCC (or any power nets) and 0V (GND) to the end. VCC and any other power lines will be the last tracks drawn and 0V may be drawn using a polygon pour (explained in a later section of this tutorial). Using this method will speed the process up considerably and reduce the chances of mistakes.

Track Properties

While routing a track, the user can press the 'Tab' key on the keyboard to bring up the track properties. This allows the user to change the track width and via properties. All track widths and via sizes must be within the limits of the design rules, otherwise Altium will flag this and issue warnings.

The user may also wish to open the track properties by double-clicking on a section of a track that has already been placed on the PCB. Instead of bringing up the full track routing properties, this properties window is specific to the section of track that was double clicked. The user can set the length, width and net name of the track using this window.

Vias

Vias are a tool used to allow a track to jump from one layer to another while maintaining the electrical connection. It is simply a plated hole through the board and both top and bottom layers, and any other tracking layers present on board, are connected. This is a common item to see on any circuit board as tracks often need to overlap or cross over and this tool allows this to happen using the fibreglass PCB as a means of electrically separating the overlapping tracks.

A via can be placed manually by selecting the correct tool from menu bar (as shown below), or by pressing the '*' key on the keyboard while routing a track.



- The 'Place Via' tool

Since a via does not have any pin passing through it, it does not need such a large hole in the middle. It also will not need to be soldered if manufactured outside of UQ, so it does not need a large pad either. As a general rule of thumb, keep the via hole size at a minimum of 0.7mm and the via pad size at a minimum of 1.3mm.

Polygon Pours

A polygon pour is simply an area of copper that has the same effect as a track, but can be any shape or size. It is common to use a polygon pour at the end of the PCB design to connect all the 0V (or GND) connections together. This will save a lot of time for the user, rather than trying to join all these points with tracks. It may also have the effect of reducing the amount of noise throughout the system as a ground plane or solid reference plane will provide a low impedance path for any signals to return to ground.

Ground planes can be quite complicated and all the specific details will not be covered in this tutorial, but some general rules are mentioned to guide the user.

Drawing the polygon pour should be left to the end of the tracking process, but it may take a couple of attempts to produce a pour that is solid and has no narrow bottlenecks. The process is started by clicking on the 'Place Polygon Plane' tool (shown below)



- Place Polygon Plane tool

This will open a window displaying the polygon plane's properties. As a general rule, set up the options as below:

- Fill Mode = Solid (Copper Regions)
- Name = 0V or GND
- Layer = Bottom Layer
- Connect to Net = 0V or GND (or whatever the name of the ground connection is)
- Pour over all same net objects
- Remove Dead Copper (Check box to make sure dead copper is removed)

The other options do not matter so much, so just leave them in the default state.

Click the 'OK' button to close the properties window and begin to draw the outline of the polygon plane by right-clicking on the PCB area.

Note: Do **NOT** draw the polygon pour outside the PCB area as this may cause the plane to be removed at some point in the manufacturing process.

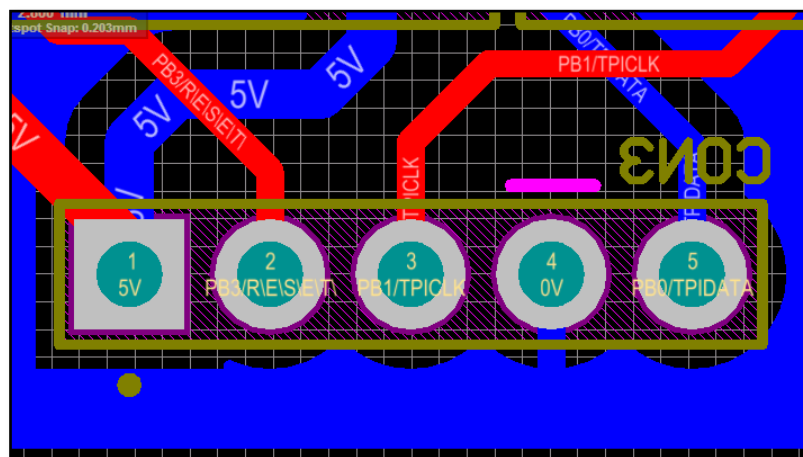
The user can right-click at any time to finish drawing the polygon. Altium will then join the starting point to the last point with a straight line and place the solid polygon on the PCB and connect it to any nets that are supposed to be connected. It will keep any non-ground nets separate from the ground plane.

The polygon pour can be re-drawn at any time by right-clicking on it and then selecting '**Polygon Actions >> Re-pour**'.

Keep-out Layer

As much as a ground plane is useful in reducing the level of electronic noise within a circuit, it can also pick up noise fairly easily and transmit it to other components in the circuit. It can be highly beneficial to keep the ground plane away from known sources of noise (e.g. the quartz crystal oscillator).

One of the tabs displayed at the bottom of the PCB document window is labelled 'Keep-out'. The user is able to draw lines on this layer that can be used to keep the ground plane (or other nets) away from a specific area. The keep-out lines are used to sever small necks off the ground plane to prevent it from entering an area. The ground plane can then be re-drawn and the plane will no longer try to enter past the keep-out line. E.g.

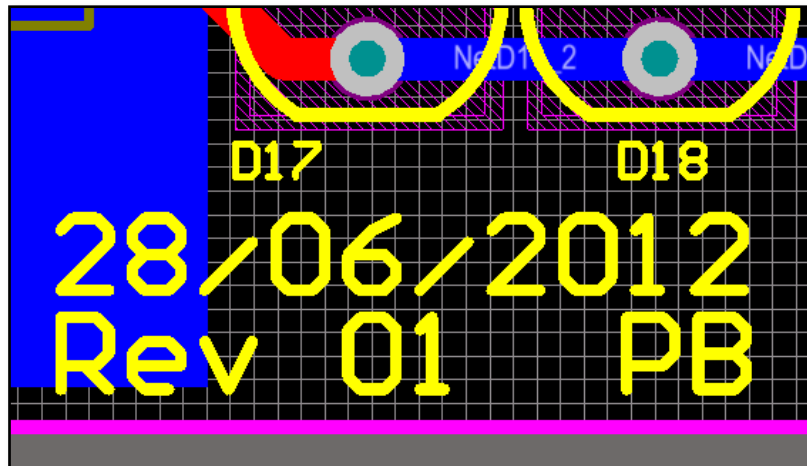


In this example, the 0V plane, coloured blue, would try to flow through pin 4 of the connector and fill the area just above the pin. There is no need for this to happen as there is no 0V connection in this area. Placing a keep-out line, coloured pink, just above pin 4 prevents the ground plane from flowing through the pin.

This method also reduces the risk of manufacturing short circuits that occur when solder accidentally crosses from one net to the ground plane.

4.6 Text Strings and PCB Identification

Text can be placed onto any layer of the PCB as a means of identifying the board, which project it belongs to, and which revision it is.



The 'Place String' tool for placing text onto the PCB document can be found in the PCB document toolbar.



Select which layer on which to place the text first and then click on this tool. The user will then be asked to place a default text string onto the PCB document by right-clicking anywhere on the PCB.

Once the text is placed, the user can double-click on the text to edit the string and any of its properties.

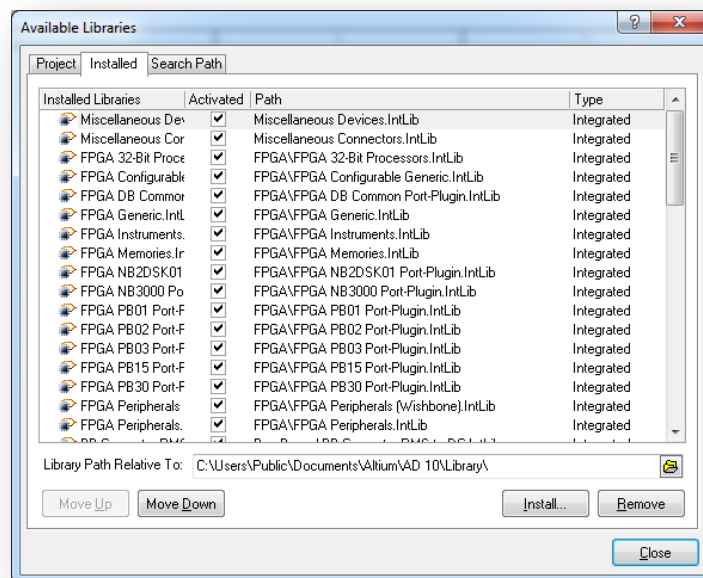
It is common to use this tool to place PCB identifiers onto the top or bottom overlays, but if a PCB is being manufactured within US, then place the text on either the top or bottom layer as PCBs built in-house will not have a top or bottom overlay.

5.0 Altium Component Libraries

As well as the default libraries containing miscellaneous devices and miscellaneous connectors, Altium Winter '09 and Summer '09 have a number of manufacturer-specific libraries containing schematic symbols and component footprints. These are mainly comprised of amplifiers, digital logic, microprocessors, as well as other forms of integrated circuits.

In Altium 10, these libraries are not included but users can download them from the Altium website. In Altium Winter '09 and Altium Summer '09 these libraries are included in the Altium 'Program

Files' folder. There are a number of folders organised by manufacturer, and each one contains a number of libraries. These component libraries must be added to the project in which the user is working. This can be done by the user clicking on the 'Project' button in the Project Window and selecting 'Add Existing to Project...', or by the user clicking on the menu **'Design >> Add/Remove Library'**. The Add/Remove Library window has three tabs at the top. The 'Project' tab will display all the libraries currently installed in the project. The 'Installed' tab will show all the currently installed libraries (see diagram below). The 'Search Path' tab shows any libraries that are found under the current list of search paths (if any have been added).



At the bottom of the 'Installed' tab is a button labelled 'Install'. Pressing this button will prompt the user to select a new library for use within the project. Even though the library may not appear in the Project Window, the components can still be added to the schematic document. This is often a more tidy way of adding libraries than the 'Add Existing to Project...' method.

Using the supplied libraries allows the user access to a wealth of pre-designed components and can save time, but it is still useful for the user to understand how to draw their own components. The next section will explain the basics behind this process.

6.0 Custom Component Design

Altium has extensive libraries supplied with the software and while these libraries have numerous components built into them, they are not all-encompassing and sometimes do not contain 3D component bodies. It is common for users to draw their own components on an as-needs-require basis. The final section of this tutorial instructs the user on how to create their own components.

6.1 Footprint Design

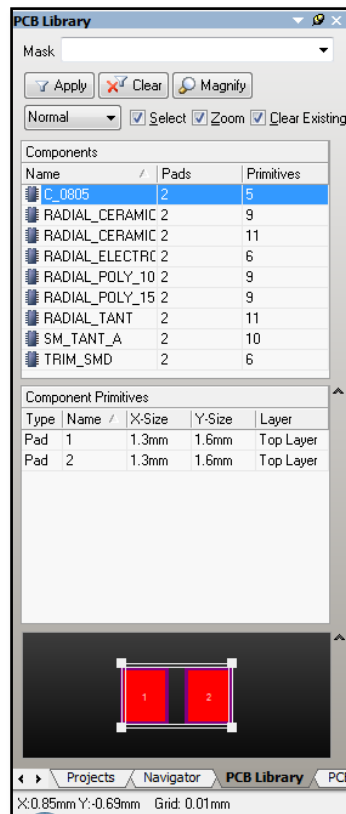
When drawing any component, it is best practice to draw the mechanical footprint first and then draw the schematic symbol. This will avoid confusion and incorrect linkage between the schematic symbol and the footprint.

Mechanical footprints must be drawn in a PCB library and the PCB library should be placed in a location that can be accessed by all the projects created by the user. The user can choose to add a PCB library to the project if they wish or they can leave it as a free document, but install it using a search path or by installing the library manually into each project (see section above). Only installed libraries can be used within a project.

A PCB library is created by selecting '**File >> New >> Library >> PCB Library**' from the menu bar. If a PCB project is currently open, then the PCB library will automatically be added to the project, but the user can right-click the library and select 'Remove from Project' to change the document to a 'Free Document' if they so desire. If this is the case, the PCB library will need to be installed into the project once it has been saved to a specific location.

The PCB library is very similar to the PCB document editor window. Many of the tools are similar and the layout should be quite familiar to Altium users.

At the bottom of the project space on the left-hand side of the screen are a number of tabs. Upon opening the PCB library a 'PCB Library' tab will appear amongst these tabs. Clicking this tab will bring up the PCB library window from where the user can select, add or browse properties of the components contained within the library.



The top tools in the window allow users to perform actions upon selection of a component. The section below this is a list of all the components contained within the library. Selecting one of the components from the list will display all of that component's properties and its current footprint in the sections below.

If the user wishes to create a new component, they can right-click the mouse in the 'Components' area and select 'New Blank Component' from the resulting menu. This window allows the user to place many footprints within one library.

Double-clicking the name of any one footprint will bring up that footprint's name and description properties.

Pad Placement

The user should start the component design by placing the pads of the component.

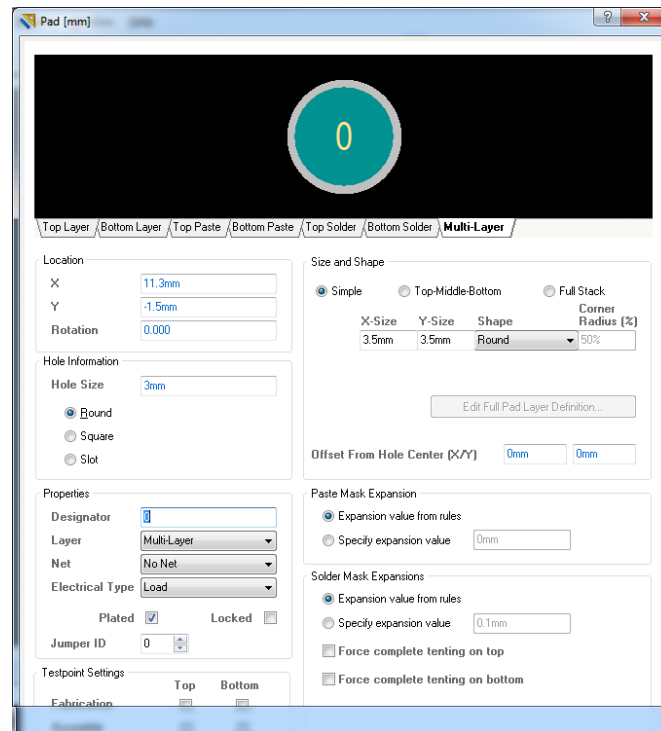


- Place Pad Tool

Clicking on the 'Place Pad' tool from the menu bar (see symbol above) will place a pad onto the mouse cursor. The user can then place pads at any position on the PCB library window. The goal at this point is to map out the physical pins of the component, whether it is a surface-mount component or through hole. The dimensions of the pins of the device can be obtained from the

device's datasheet. This is normally supplied on the distributor's website and can be quite complicated, especially for more advanced components.

Pressing tab at any time while placing pads will bring up the 'pad properties' window.



The user can change the individual properties of the pad using the option within this window. The most important properties to alter are the designator (all pad names should start from 1, not zero) and the hole and pads sizes. If the component is surface mount, it should be drawn on the top layer and will most likely have no hole through the centre of the component (set hole size to 0mm). If the component is through-hole then the pad should be drawn on multi-layer with a hole through the centre big enough to take the widest dimension of the component pin.

The spacing of the pins (or the 'pitch') is usually set very precisely by the manufacturer. In the majority of cases, the pitch is used as a reference dimension for the rest of the component body. This is why it is often best to start by placing the pads of the component and then draw the body. The user can set the snap and component grids to assist with accurate placement of the pads by setting them correctly in '**Tools >> Library Options**'. Picking quite a large grid (usually of the order of 100mil) will enable the user to quickly and accurately drop the pads in the correct position.

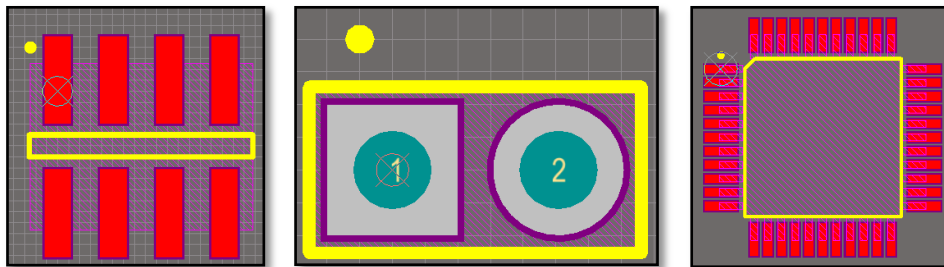
When the user has finished placing all the pads so that they are in the same pattern as shown on the datasheet, they should right-click on the mouse to finish the pad placement tool.

Drawing the Component Body

The next step is to draw the body of the component. Although this step is not essential, it will greatly assist any person soldering the components to the PCB as it will help the person place the components in the right holes and in the correct orientation.

The outline of the body is usually drawn on the 'Top Overlay' layer and appears in Altium as a bright yellow line. When the PCB is manufactured, the top overlay is drawn onto the PCB over the top of the top layer. If the PCB is manufactured within UQ, the top overlay will not be drawn on the PCB.

The user should use the 'Place Line' and 'Place Arc by Centre' tools to draw the lines that comprise the body of the component. No lines on the top overlay should overlap or come close to pads, so the user needs to carefully think about how to best represent the component body. The main purpose of drawing the component body is to allow the user to place components on the PCB without them bumping into each other. The second purpose is to show how the component actually sits on the PCB and which pin is designated pin 1. Therefore the user should make sure that the outline covers the maximum possible size that the component could be. The user should also place a dot on pin 1, normally outside the body of the component so that it is simple to distinguish which pin is pin 1, even when the component is soldered onto the board.

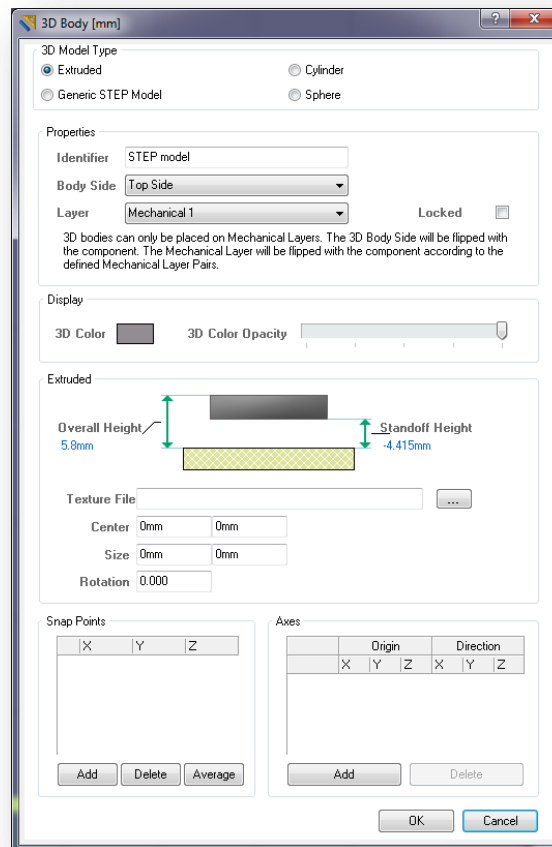


Component Footprint Examples

Adding 3D Component Bodies

The user can place 3D bodies of components onto the 'Mechanical 1' layer at this stage of the component design. 3D bodies can be imported from a CAD package if the model has been saved as a STEP file. Mostly, the 3D step files can be downloaded from supplier websites like RS Components and Element 14, or from the manufacturers' website.

If the component body is simple, the user may choose to draw it themselves. This can also be achieved using the 3D body tool which is found in the menu 'Place >> 3D Body'.



Clicking on the 3D Body placement tool opens a window. The user can import a STEP file using this window, or they can draw a simple shape themselves using the dimensions from the datasheet. Multiple 3D bodies can be placed on the footprint, but each must have its own individual name which should be descriptive of the part being drawn. The user should explore the properties in this window and view the effects that it has on the component body.

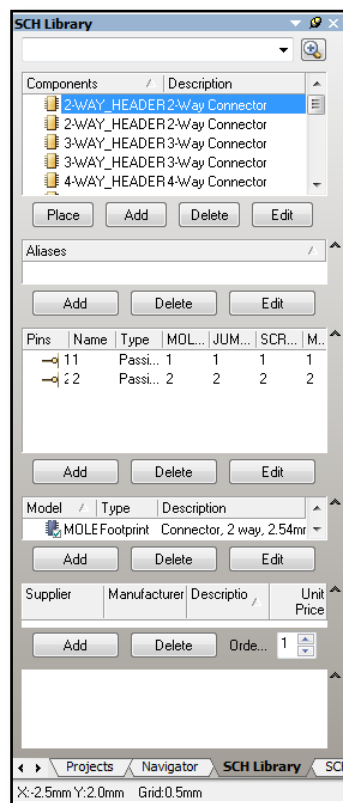
Note: To swap between 2D and 3D view in either the Footprint library or the PCB document, the user simply needs to press the number 3 on the keyboard to enter 3D mode, or the number 2 to enter 2D mode.

Once the user is satisfied that the component footprint is the best possible representation of the physical shape of the component, the footprint is complete. The next stage is to draw the schematic symbol of the component and link it to the footprint.

6.2 The Schematic Library and Symbol

The schematic symbol for each component is contained within a schematic library. The schematic library functions in much the same way as the PCB library in that it can contain a number of schematic symbols within the one library.

Upon opening a schematic library, the user will notice that a 'SCH library' tab appears at the bottom of the navigator window on the left-hand side of the screen. Upon clicking on this, the following window will appear.



The top part of this window displays a list of all the symbols contained in this schematic library. Individually clicking on each symbol will display that component's properties in the sections below. Double-clicking on any symbol name will bring up that symbol's properties window. Clicking the 'Add' button just below the component list will add a new blank component to the library.

Drawing the Schematic Symbol

The most common way of drawing schematic symbols is to use the drawing tools provided on the tool bar.

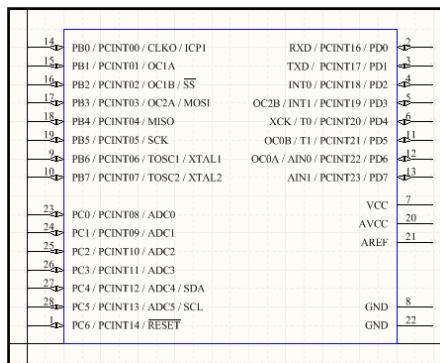


- SCH Library drawing tools

Size and shape do not matter as much when drawing the schematic symbol of a component. The user is simply trying to draw a best representation of the function of the component, rather than an accurate 2-dimensional shape. Therefore the user normally starts by drawing the body of the component which is the electrical symbol for that device (if it has one) or simply a rectangular box (mostly used for IC's).

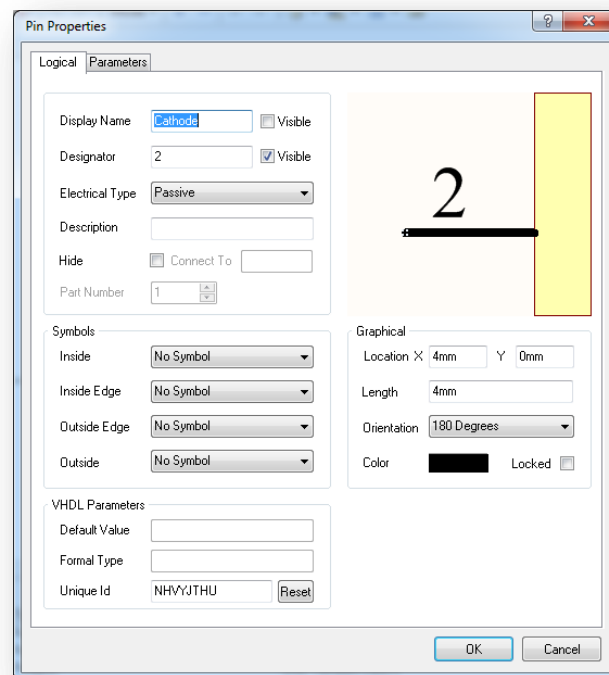
The user then has to place pins onto the symbol. The pins are the legs or leads of the component to which electrical connections are made. Every component will have at least one or two pins, but some IC's can have over 100.

Pins do not necessarily need to be placed numerically beside each other. In fact, for large microprocessors, it is common to group ports, VCC's and GND's together in sections.



When the user clicks the 'Place Pin' tool, a pin is attached to the mouse pointer. The end at the centre of the mouse pointer with the 'X' through it is the electrical connection end of the pin. This end should be outside the body of the symbol as the user will have to connect wires to it. The other end is the label end of the pin. This should touch the body of the symbol and can have visible pin labels or invisible labels.

Before clicking the pin and locking it in place, the user should press the 'tab' key to edit the pin properties. It is in this window that the user can change the name and designator of the pin.



Note: Pin designators are used to match schematic symbol pins to their respective pads on the mechanical footprint. Pin designators should start at 1 and numerically increment as required. Designators should not be anything other than numbers as this will confuse the linkage between the schematic symbol and the PCB footprint. If the user wishes to give the pin a label, they should put this in the 'Display Name' box within the pin properties window.

Hint: Placing a '^' after a letter in the 'Display Name' box will place a line above that letter. This is used for normally high pins like RESET.

Linking the Schematic Symbol and the PCB Footprint

When the schematic symbol is complete, the user should then link the schematic symbol and the PCB footprint of that component. If the schematic symbol has more than one footprint, all those footprints should be linked at this stage. Component footprints are stored in the PCB footprint libraries (see previous sections within this tutorial).

To add a footprint to the schematic symbol, the user should click the button at the bottom of the schematic library window labelled 'Add Footprint'. The user can then navigate to the correct library and add the component footprint to the symbol. Only libraries installed within the project will be visible to the user at this stage. Repeat this process to add multiple footprints. Each footprint added will be shown at the bottom of the schematic library window.

The user should then save the schematic library and it will then be available for use within the project.

7.0 Summary

This tutorial has covered the most basic functions of Altium Designer, however there are many more complicated functions available to the user. Altium Designer is a complicated software package that offers the PCB designer many tools for PCB design. The user is encouraged to explore more of the functions within Altium as this exploration will develop knowledge, understanding and experience. Altium itself offer some very useful tutorials on their website and the user is encouraged to view these and learn more.

However, if further help is required, please contact a member of the Instrumentation Support Group within the University of Queensland using the following details:

E-mail: instrumentation@eait.uq.edu.au

Room: Building 45 (Mansergh-Shaw), room 103 – please wear enclosed footwear if entering the ground floor of this building