
Altium Designer Tutorial Notes

Revision: 1

Altium Version: Winter 2009

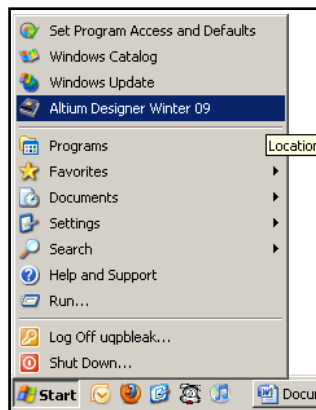
Introduction

Altium Designer is a software package which allows electronic circuit designers to design, draw and simulate electronic circuit boards. Altium is a vastly complex software design suite and these notes are designed to introduce the user to the fundamental principles and tools used throughout the package. These notes are designed to accompany the METR2800 tutorials provided by the University of Queensland and examples from that course are used to accurately describe certain tasks and processes. If you require any further details then please contact a member of the Instrumentation Support Group (Bd 45-103).

Getting Started

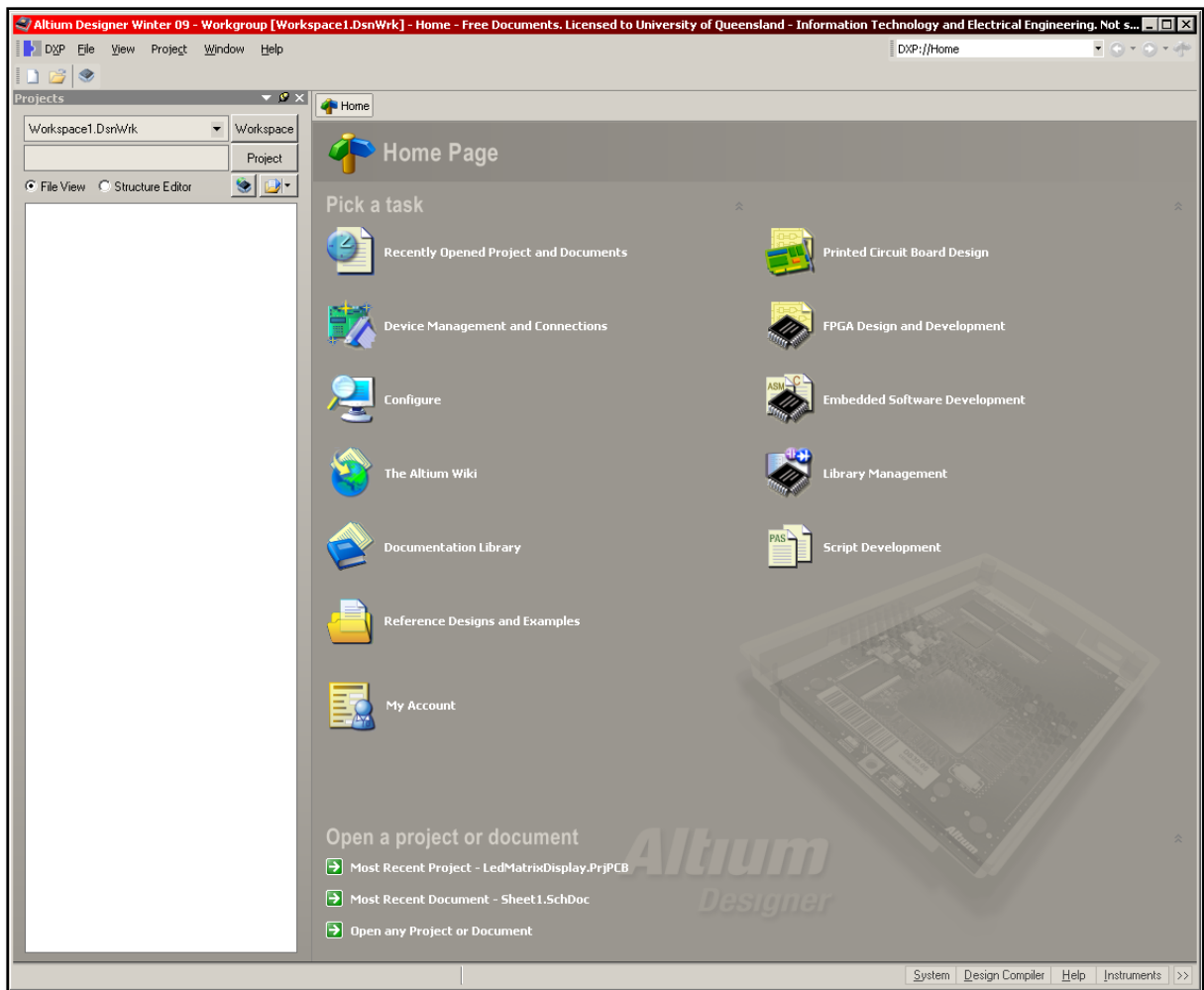
- Opening Altium Designer

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.

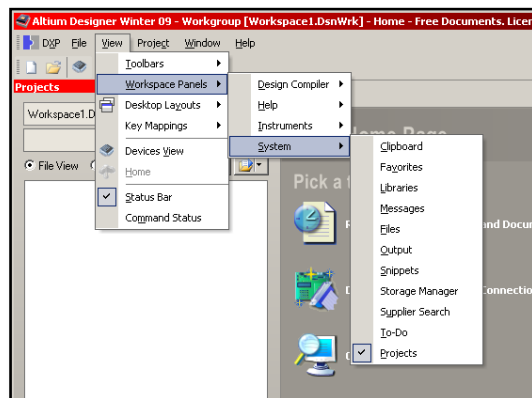


If no shortcut is available then the user should look in the 'Programs' menu to see if there is an Altium folder available. The Altium Designer Winter '09 shortcut should be available in this folder.

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 gray contains shortcuts to some of the more intricate parts of the Altium Designer Suite and will therefore not be explained in this section. The main area that needs to be explained is the white area on the left of the window. This is the Project Window and it is used to control the Design Workspace. If no Project Window is visible, go to the menu bar and click **View>>Workspace Panels>>System>>Projects**. This portion of the screen can have many windows open in it at once and the user is able to navigate between them using tabs at the bottom of the space. This will become apparent as the tutorial becomes more advanced.



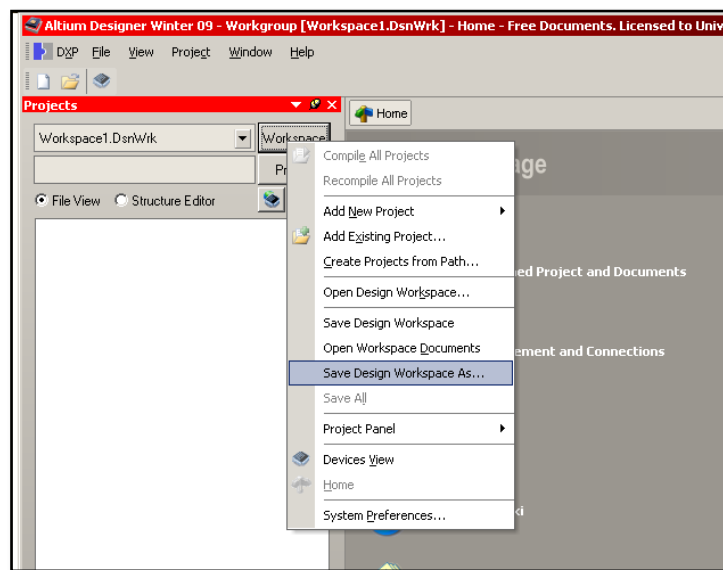
A) Setting up a Workspace and Project

In Altium Designer Suite, all the programs 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. Imagine this area as the empty drawer of a filing cabinet ready to take all the files produced by the software. It will be divided up into many different 'Projects', but the use of this word at this stage may be a little misleading. The design workspace will 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.

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

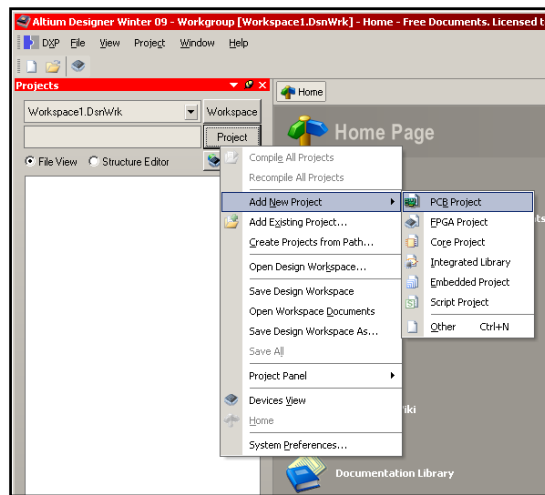
1. **Save the design workspace with an appropriate name.**

- 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.

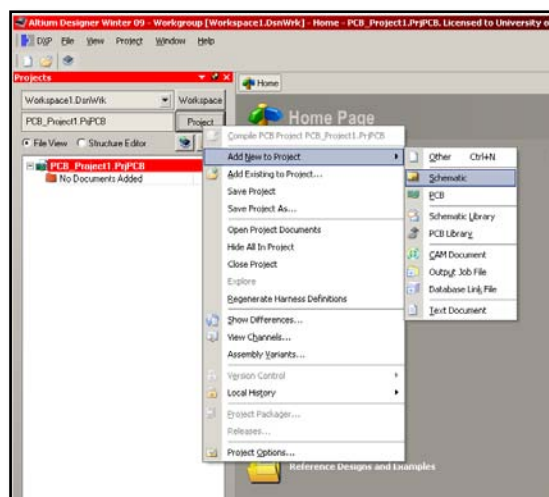


A Project icon will appear in the Projects window. The user is able to right-click on this icon to bring up a menu of functions. It is recommended that the user explores this menu in their own time as it can provide some very useful shortcuts for speeding up the design process.

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.

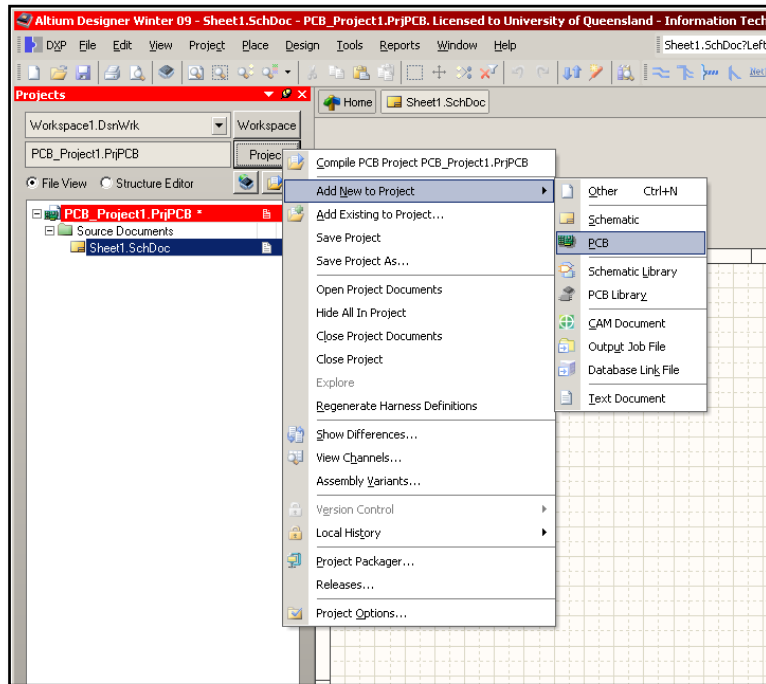
Click on 'Add New to Project>>Schematic' to add a blank schematic sheet to the project.



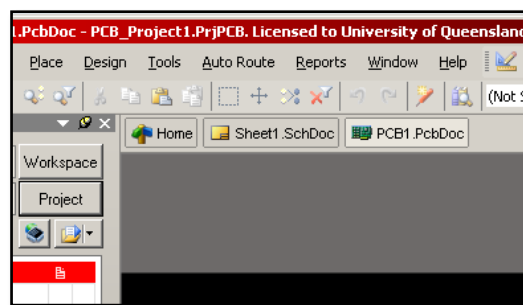
The new, blank schematic document will open in the main window of Altium and a schematic sheet icon will be added to the project in the 'Projects' window. Again, the user can right-click on this icon to perform useful shortcut functions and it is recommended that the user explore these shortcuts in their own time.

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 window.



These tabs can be used to quickly switch between the schematic and the PCB documents.

5. Add libraries to the Project

- When designing an electronic circuit, the components needed are stored in software libraries. These libraries contain information about schematic symbols, pins, as well as a fully 3-dimensional mechanical representation of the component itself. Single components may have different physical shapes (or packages), so multiple mechanical drawings of components can be linked to one schematic symbol. Altium Designer has some libraries that



come supplied with the software design suite, but generally these are very limited in their use and it is common for users to develop their own libraries over time.

All students studying METR2800 will have access to the component libraries used throughout this tutorial, but it is possible to design component libraries from scratch.

However, this tutorial will not go into the detail of how to produce custom libraries as this is considered to be an advanced skill.

All component libraries need to be linked to the project in which they are going to be used.

In order to accomplish this, the user will need to right-click on the project icon in the 'Projects' window and select 'Add Existing to Project...' from the project menu. A dialogue box will appear prompting the user to navigate to the component libraries. The user will need to change the file type (at the bottom of the 'Open file' dialogue box) to 'Library file'. The user will need to open any .PCBLIB and .SCHLIB files that they need for the project. Once added to 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.

Components can now be selected from these libraries for use in both the schematic and PCB documents.

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 ask the user for an appropriate file name for each document and will ask the user to save the document to an appropriate, user-specified location. **NOTE:** The user needs to take special note of the file named at the top of the 'Save As' window in order to see which document is being saved as several different 'Save As' dialogue boxes will appear; one for each document/project that needs saved. The top bar of the 'Save As' dialogue box will clearly state which document/project is being saved.

B) Design Capture using the Schematic Document

The schematic document is used to generate a detailed concept of how the electronic circuit is going to work. It is not meant to show the specific shape or functions of components but merely how they relate to each other and connect together. It is important to have a clearly laid out schematic diagram so anyone can pick up the schematics and quickly work out the function and purpose of the electronic circuit.

A schematic of a single circuit can be spread across many schematic sheets although it is beneficial to try and limit the number of sheets used. It is also useful to categorise the different sections of the circuit so they can be easily identified. Altium Designer has some useful tools that allow the user to produce clear and effective schematics relatively simply. The following guide illustrates how to generate a simple schematic and introduces the user to some of these tools.

Basic Sheet Manipulation Tools

The following tools are used throughout Altium Designer to view, move and generally manipulate the sheet that the user is working on. The context used in this instance is for the schematic document, but some of these tools are also available when the user is working on the PCB sheet. The user will frequently use these tools throughout Altium, so it is best to become familiar with them as soon as possible.

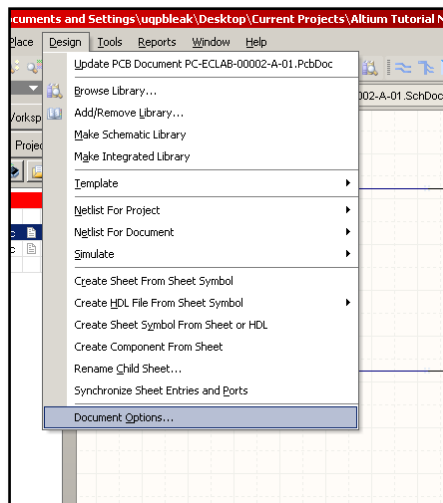
Hand Tool – If the user wishes to ‘grab’ the sheet and move it slightly, they simply need to click and hold the right mouse button on the sheet. The mouse cursor will change to a hand symbol and the user will be able to move the sheet in whichever direction they wish simply by moving the mouse.

Zoom Tool – If the mouse has a scroll-wheel, this can be used to zoom in and out of the sheet. If no scroll-wheel is available, the user can click and hold both mouse buttons together. The cursor changes to a magnifying glass symbol and moving the mouse up will zoom in and vice-versa.

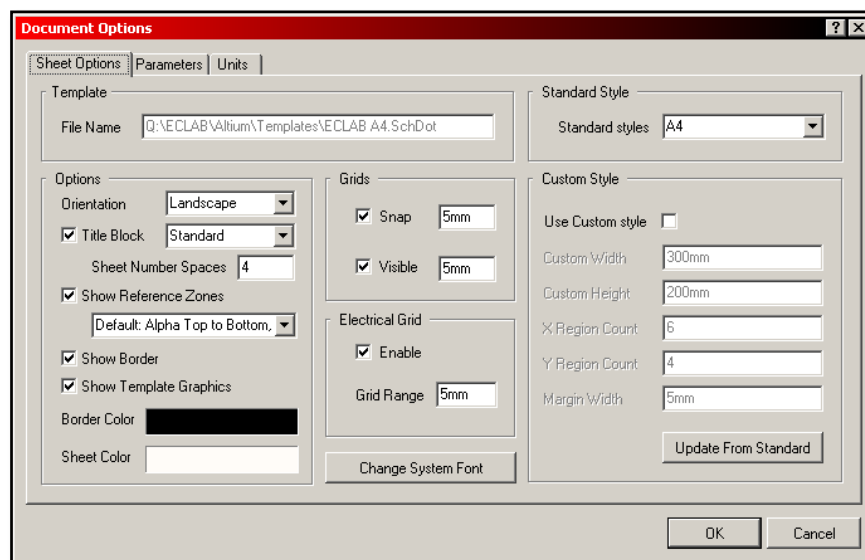
Select Tool – Clicking and holding the left mouse button will allow the user to drag a selection box over an area of parts, components or tracks. When the user releases the box, anything **completely** within the box area will be selected. This is useful for mass-deleting or moving.

Setting Up the Schematic Document

The user can change some of the default schematic document settings in order to make sheet easier to work with. This is done by changing the ‘Document Options’. To edit these options, the user should click on ‘Design>>Document Options’ from the menu bar at the top of the main window.



This will open the 'Document Options' window from which the user can change some of the basic parameters.



This window has 3 tabs at the very top, labelled 'Sheet Options', 'Parameters' and 'Units'. Each of these is explained below.

1. Sheet Options

- In 'Template' the user is told which template is being used for this file. The user cannot change this template here, but it is mentioned for reference. The template is changed by choosing 'Design>>Template>>Set Template File Name...' from the main schematic window menu options.
- In the 'Options' box, the user can change the basic details of the sheet shape, outline and colour.
- In the 'Grids' section, the user can change the resolution of the Snap and Visible Grids. These help with component placing and the drawing of wires (nets).

- d. In the 'Electrical Grid' section the user can enable or disable the electrical grid. This grid is used to help the user easily select the right pin or component. It makes the mouse pointer 'jump' to any component or track that is in range of the mouse pointer. The range is set by the user and this grid can be turned on and off depending on the user's preference.
 - e. 'Change system font' opens a new window that allows the user to change and manipulate the fonts applied to the schematic.
 - f. 'Standard Style' allows the user to change the sheet size and shape to one of the standard paper sizes available.
 - g. If the user wishes to apply a 'Custom Style' to the document then they should set this up using the final section.
2. **Parameters** – Parameters are text and numeric fields which give some form of extra explanation of the schematic to the user. They can be dates, names, or just descriptions but each one can be edited by the user depending on their needs. These text parameters can even be displayed in the title block in the corner of the schematic and can make updating and producing new schematics relatively quick. In a standard word-processor the equivalent would be the Properties of the document.
 3. **Units** – The 'Units' window is used to set up the schematic with either Imperial Units or Metric Units. A range of standard units can be chosen and these will be applied throughout the entire schematic.

When the user is happy with the schematic set up they can simply click on the 'OK' button.

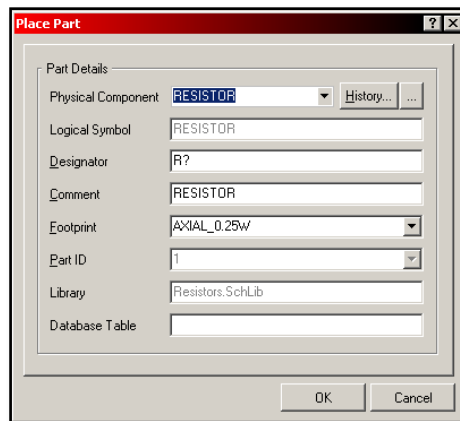
4. Placing Components (or Parts)

- In Altium Designer, components are referred to as 'Parts'. This is intentional and helps illustrate the concept that a single schematic symbol may in fact be made up of more than one component. Parts are stored in software libraries that are linked to the Altium Designer project. Users can create bespoke libraries of parts drawn to manufacturers' specifications. The appendix of this document outlines how to design individual parts and save them in software libraries.

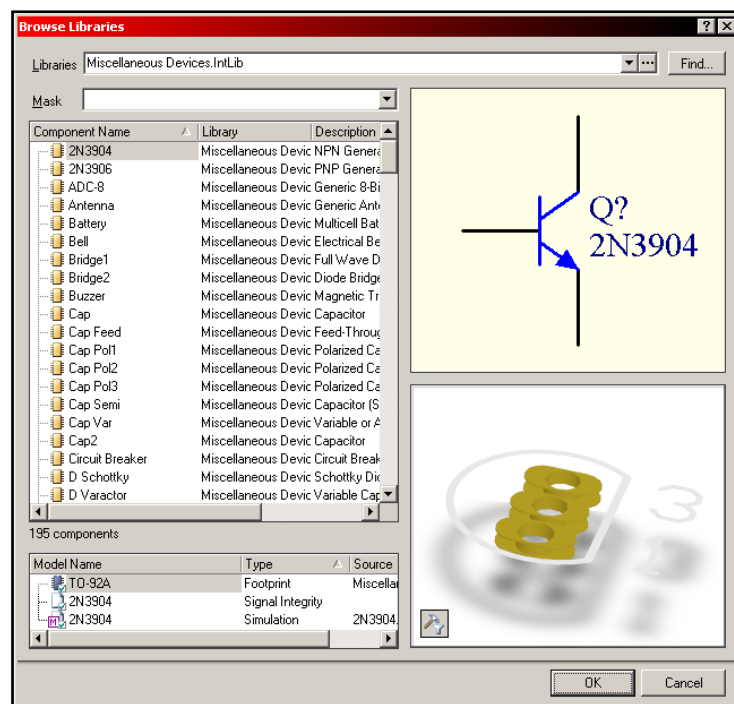
Once libraries are added to the project, the parts within the libraries are able to be placed on both schematic and PCB documents. This section is a guide on how to add parts to the schematic sheet, so make sure that the schematic sheet is selected in the main window. Next, simply click on the 'Place Part' icon located in the toolbar at the top of the main window.



A dialogue box will appear with some component details already entered.



This dialogue box is used mainly if the user has already placed a specific part on the schematic and wishes to add more of those parts to the schematic. This guide starts from the base of having no components on the schematic, so it is necessary to select a component from a library. To do this, click on the 'Browse' button which is indicated as a square button with three dots on it '...' next to the 'History' button. This brings up a new window called the 'Browse Libraries' window.



From this window it is possible to select any of the parts from within the installed libraries. The various installed libraries are able to be selected from the 'Libraries' box at the top of the window. Users should try to categorise their libraries when they are designing components and parts for ease of reference at this stage of the part selection process. Altium has two generic libraries already installed with a wide variety of parts in each.

The rest of the window is divided into 4 sections. Starting at the top-left and moving clockwise, the sections are:

- The Part List
- The Schematic Symbol
- The 3D Image of the Component Footprint
- Linked Document Details (a list of further documents or information that is linked to this schematic symbol).

Selecting a part from the part list will bring up the details of that part in the other 3 windows. A single part can have several footprints attached to it as well as simulation data (PSpice) and various other data documents.

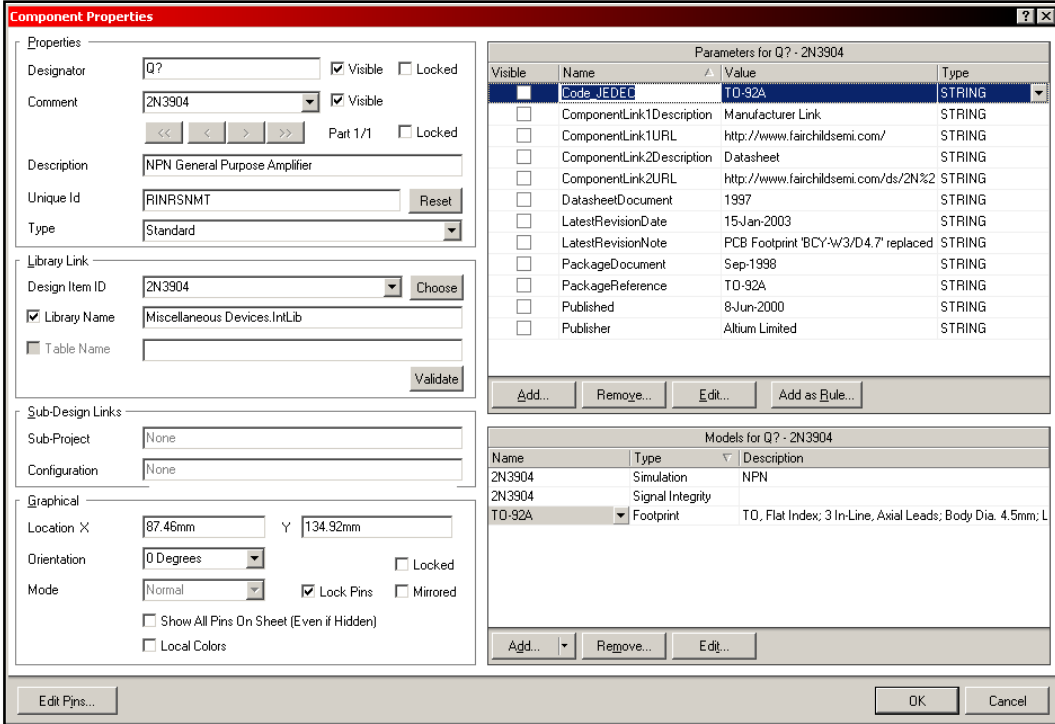
To select a component from these libraries, the user simply needs to highlight it in the part list (top-left window) and then press 'OK' at the bottom of the window. The 'Browse Libraries' window will disappear and the part's details will be loaded into the 'Place Part' dialogue box. The user should just click 'OK' in the 'Place Part' dialogue box. The box will disappear and the user will notice that a transparent image of the part is attached to the mouse pointer. The user can position the image in a suitable portion of the schematic document and left-click the mouse to place the component.

The transparent image of the component attached to the mouse pointer will not disappear, even though the part has been placed. This allows the user to place the part more than once, depending on how many times the part is used throughout the circuit. Once all the instances of the part have been placed, the user can right-click on the schematic or press 'Escape' on the keyboard. This will bring back the 'Place Part' dialogue box to allow the user to select another component to place.

When all the components have been placed on the schematic, the user should press 'Cancel' on the 'Place Part' dialogue box to exit 'Place Part' mode.

5. Part Properties

Double-clicking on any part placed on the schematic sheet will bring up the 'Properties' window of that part.



Component Properties

Properties

Designator: Q? ☒ Visible ☐ Locked

Comment: 2N3904 ☒ Visible

Description: NPN General Purpose Amplifier

Unique Id: R1NRSNMT

Type: Standard

Library Link

Design Item ID: 2N3904

☒ Library Name: Miscellaneous Devices.IntLib

☐ Table Name:

Sub-Design Links

Sub-Project: None

Configuration: None

Graphical

Location X: 87.46mm Y: 134.92mm

Orientation: 0 Degrees ☐ Locked

Mode: Normal ☒ Lock Pins ☐ Mirrored

☐ Show All Pins On Sheet (Even if Hidden)

☐ Local Colors

Parameters for Q? - 2N3904

Visible	Name	Value	Type
<input checked="" type="checkbox"/>	Code JEDEC	T0-92A	STRING
<input type="checkbox"/>	ComponentLink1Description	Manufacturer Link	STRING
<input type="checkbox"/>	ComponentLink1URL	http://www.fairchildsemi.com/	STRING
<input type="checkbox"/>	ComponentLink2Description	Datasheet	STRING
<input type="checkbox"/>	ComponentLink2URL	http://www.fairchildsemi.com/ds/2N%2	STRING
<input type="checkbox"/>	DatasheetDocument	1997	STRING
<input type="checkbox"/>	LatestRevisionDate	15-Jan-2003	STRING
<input type="checkbox"/>	LatestRevisionNote	PCB Footprint 'BCY-W3/D4.7' replaced	STRING
<input type="checkbox"/>	PackageDocument	Sep-1998	STRING
<input type="checkbox"/>	PackageReference	T0-92A	STRING
<input type="checkbox"/>	Published	8-Jun-2000	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING

Models for Q? - 2N3904

Name	Type	Description
2N3904	Simulation	NPN
2N3904	Signal Integrity	
T0-92A	Footprint	T0, Flat Index; 3 In-Line, Axial Leads; Body Dia. 4.5mm; L

Initially, this looks like a very complicated window, but that is due to the amount of information on display. In reality, the user does not need to change much of this data, but can learn a lot about the component being placed. The window is divided into 2 columns; the left-hand column containing 4 boxes, and the right-hand column containing 2 boxes. Starting at the top of the left-hand column, the boxes are: Properties; Library Link; Sub-Design Links; and Graphical.

Properties – This is where the user can find out the most useful information about the part that they have placed. Each part is given a unique alphanumeric identifier in Altium, called a 'Designator'. This term is used frequently throughout Altium, so it is best to become familiar with it. When a component is first placed, the designator will only have letters at the start of it and a question mark at the end. The letters usually are a reference to the type of component that is being placed (i.e. 'R' for a Resistor, 'C' for a capacitor, 'L' for an inductor, and so on). The question mark is later replaced by a number when it comes time to annotate the completed schematic, but this will be covered later in the tutorial.

Other information that is available in the Properties section includes a 'Comment', 'Description', 'Unique ID' and 'Type'. Normally there is a specific model or manufacturing number which will clearly identify the part in the 'Comment' or 'Description' box. The 'Unique ID' is usually automatically filled out by Altium and is used as an internal program reference. The 'Type' field rarely needs to be changed by the user.

If the component is a multi-part IC (like a quad package operational amplifier) then the user can select which part is represented by this schematic symbol by using the left and right arrows situated in the 'Properties' section.

The 'Library Link', 'Sub-Design Links' and 'Graphical' boxes all contain useful information but they are not necessary for basic operation.

The right-hand column of the 'Properties' window is more frequently edited by advanced users. It contains two boxes; 'Parameters' and 'Models'.

Parameters – This box is used to attach text parameters to the individual part or component. This information can be used to keep tidy and accurate libraries as well as highlight any component quirks or special cases. It usually depends on company policy or author's professionalism as to what, if any, data is stored in this section.

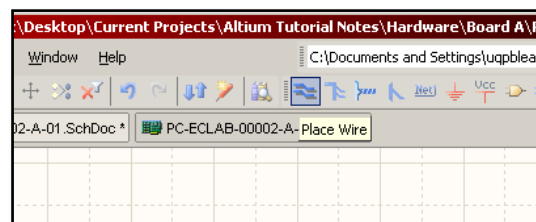
Models – This box is a link to all the external data files associated with this part or component. Usually, this is where the mechanical footprint will be available for selection, as well as any simulation files or extra data. In the 'Footprint' row there will be a drop-down menu. If more than one mechanical footprint is attached to a schematic symbol then the user will be able to select which footprint they wish to use for this component. It is important that the user selects the correct footprint when placing the component so that errors in the PCB do not arise later in the design process.

Once the user has set up the 'Properties' of the part, they should close the 'Properties' window.

6. Connections (or nets)

Once all the components are placed on the schematic, the user will need to electrically connect the pins together. There are several ways to do this, and it is important to learn a variety of methods of connecting pins as this will assist the user in producing neat, easily readable schematics.

1. **Direct Connection** – Click on the 'Place Wire' icon located on the top toolbar above the main window.



This allows the user to directly link two pins together. The user will notice a cross-hair appears on the mouse pointer when it is in the main schematic sheet area. This is used to precisely click on the component pin. The user only needs to click once to start a wire and click again on another component pin to finish the wire. Altium can route the wire automatically for the user, but often this is not the neatest or shortest wire. To manually choose the path of the wire, the user can click on

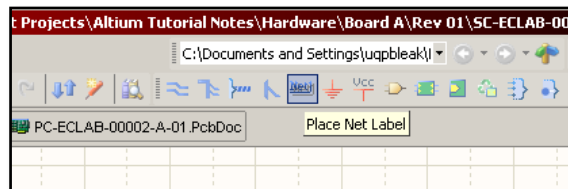
points in between each pin connection to set the wire in place as they route. Each time the user clicks on the schematic sheet, the wire will be set in place and the direction of the wire after the 'click-point' will alter by 90° . Also, if the wire is being drawn in the wrong plane, the user can change the direction of the wire by 90° by pressing the spacebar on the keyboard.

When the crosshair cursor is positioned on a pin, a red 'X' will appear on the crosshair. This indicates that if the user clicks, the wire will attach to the pin. Care needs to be taken when connecting wires to pins as a frequent mistake is connecting near, but not directly on a pin. This can have major, knock-on repercussions when it comes to laying out the PCB as wires and connections can be overlooked and missed.

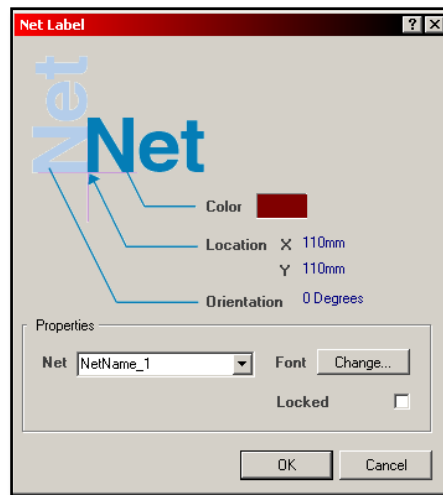
Drawing wires on a schematic sheet is a common task in Altium and it is recommended that the user experiments with this function to fully understand it.

2. **Naming Nets** – Firstly, Altium commonly refers to connections between pins as 'nets'. In the same way that Altium has to give all components a unique name to identify them, it also has to give each net between each component a unique name. This is normally automatically done within Altium and the user should not over-complicate the design process by trying to allocate a name to each net, but it is sometimes useful for the user to name nets directly. For instance, if there are many connections to be drawn on the schematic then drawing a wire can sometimes be messy, especially if there are long distances between the pins. A better course of action is for the user to give the wire, pin or net a name.

Click on 'Place Net Label' in the toolbar at the top of the main window.



A cross-hair appears on the mouse pointer when it is in the main schematic sheet area, in the same manner as to when the user was drawing a manual net. This time, a name also appears beside the crosshair. This name is generated automatically by Altium and can be changed to something more user-friendly. A general rule throughout Altium is that if the user is currently performing a task, they can press the 'tab' button on the keyboard to bring up a window of properties related to that task. By pressing 'tab' while placing a net name, the user will bring up a window in which they can change the name that is being placed.

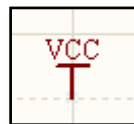


Here the user can change a number of properties about the net name, including its position on the schematic, orientation, font style, colour, and name. The user simply needs to type a new name into the 'Net' text box and press 'OK' in order to change the name allocated to the net.

To place the net label on a wire/pin in the schematic, the user needs to place the crosshair cursor over the wire/pin to which they wish to attach the net label and make sure that the red 'X' appears, indicating that the net label will attach to the pin/wire. Once the user is confident that the net label is correctly placed, they simply need to left-click once to place the net label.

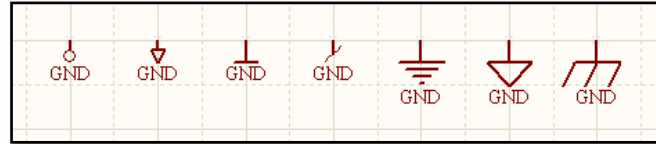
If the user wishes to change the net label at any stage then they can double-click on it to bring up the net label properties window. This is the same as the net label properties box that appears when pressing 'tab' before the net label is fully placed.

3. **Net Symbols** – The third and final way of connecting pins together is similar to that of giving the net a name, except this time the net is given a symbol representing what the net does. This method is used extensively to illustrate power pins and ground terminals as it would be tedious and messy to use wires or net labels to illustrate these connections. Usually, for a DC power pin, the symbol is similar to this:



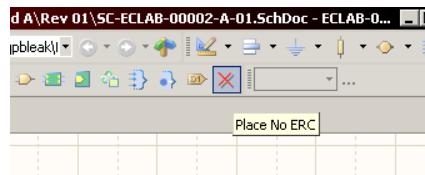
This symbol can have different names in circuits that use a multitude of power supply rails and often has the text +5V, +12V, or -3V above it instead of VCC. This symbol may even be in a different colour, depending on designer preferences.

Another commonly used symbol is the one used to represent the Ground or Common connection. This symbol can have many different forms and meanings, depending on the circuit being designed. Some of these symbols are shown below:



In DC circuits, the most commonly used symbol is the bar (similar to the DC power pin symbol). This is a small and neat symbol that helps keep the schematic sheet tidy. These symbols can also be used for power rail inputs although this is not very common practice.

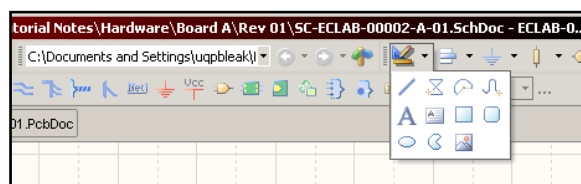
No Connection – If no connection exists on a pin and it is to be left floating, it is preferable to tell Altium that this is the case. In order to do this, there is a tool called 'Place No ERC'.



This simply allows the user to place a red 'X' on the pin that is to be left un-connected. It is advisable to use this tool as it signifies that the pin has been intentionally left un-connected, rather than simply missed during the design process.

4. Drawing Tools

Altium Designer has drawing tools available to the user for drawing simple shapes on the schematic sheet. The drawing toolbar is located at the very top of the main window.

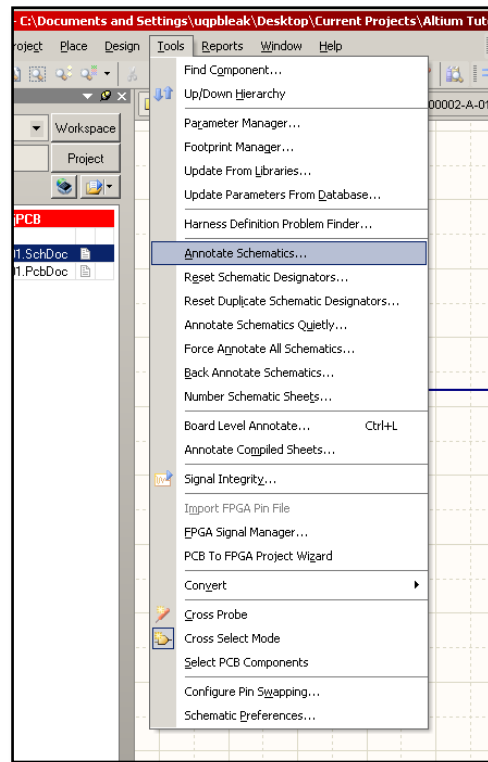


In this toolbar there are tools for drawing lines, polygons, circles and arcs, multi-point lines, letters, text boxes, quadrangles, circles, ellipses and many other tools. It is recommended that the user explores this toolbar in their own time as many of the tools have extra functions that can assist in developing explanatory notes and borders of the schematic.

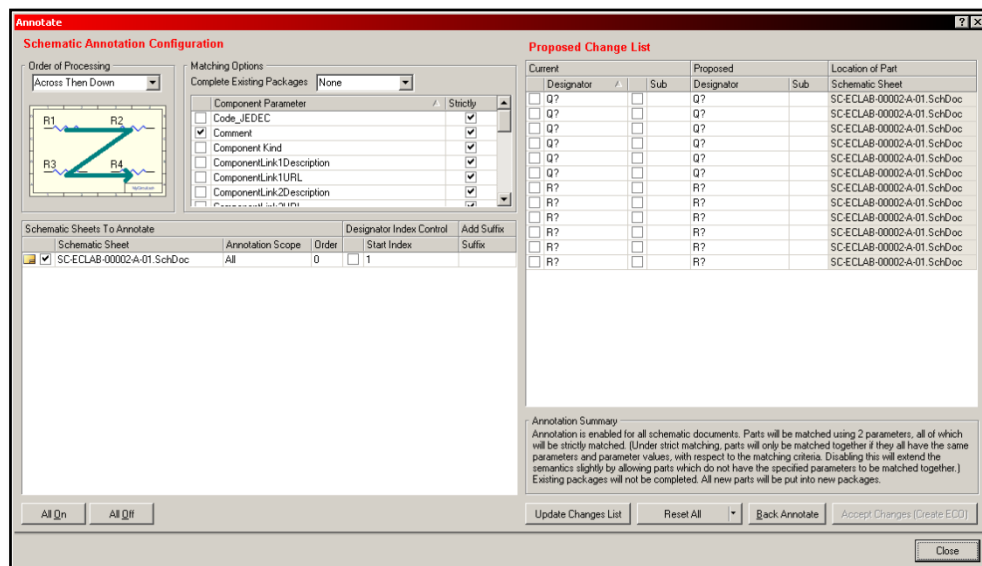
7. **Part Annotation**

When the schematic is complete and all components are placed and wired correctly, the final task is to annotate the components. Altium is able to do this automatically with a tool called 'Annotate

Schematics...'. This is located in Menu Bar at the top of the main window under **'Tools>>Annotate Schematics...'**



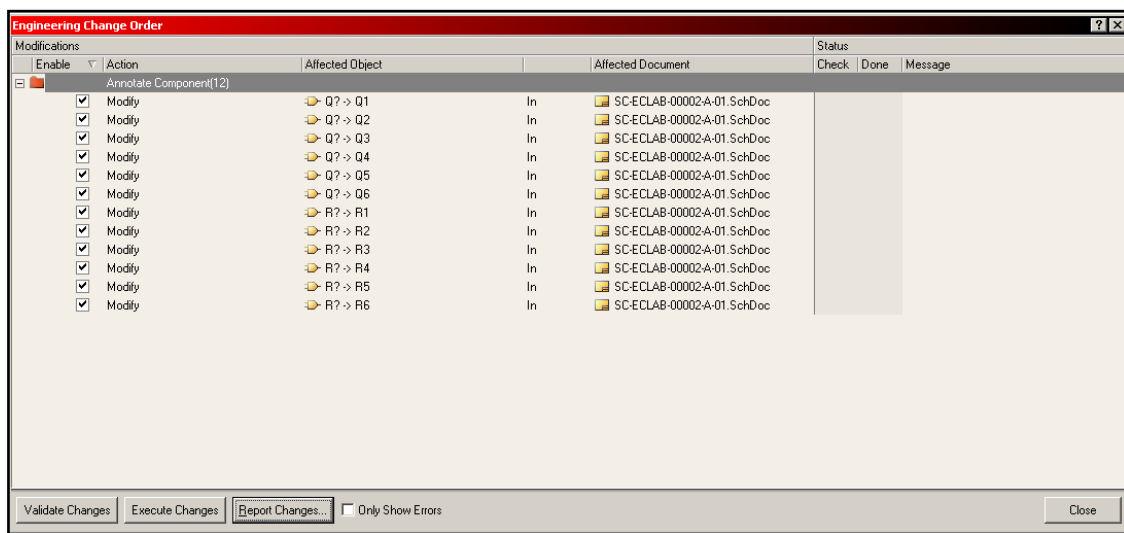
Clicking on this tool opens a new window.



The left-hand side of the window is able to set up *how* Altium will annotate the schematic sheet and also *which* schematic sheets (if there are more than one) are to be annotated. The user can also select different parameters of the component to update, although this tool is not used very frequently.

The right-hand side of the window lists all the parts and components in the selected schematic sheets. Initially, every single component has a letter with a question mark beside it. The question mark will be changed to a number, giving each component a unique identifying code called a 'Designator' (see earlier notes).

The user simply has to click the 'Update Changes List' button at the bottom of the right-hand side of the window. This will show the user how Altium intends to update the designators. If the designators are not labelled to the user's satisfaction, they can change the settings on the left-hand side of the window to label the components differently, but they must press the 'Update Changes List' button every time they change the settings. If the user is happy with the changes, they will need to press the 'Accept Changes (Create ECO)' button. This brings up a second window called the 'Engineering Change Order' (or ECO) window.



This window is simply a confirmation screen informing the user of the changes that are about to be implemented. It allows the user to carry out these changes in a visually transparent manner. This is extremely useful, especially when schematics cross over many different sheets and are the result of many different revisions. Initially, however, this is a simple task. The user simply needs to click the 'Execute Changes' button and the component names will update. The 'Validate Changes' button is useful if the user wants to see what the implications of the changes would be without actually changing the part designators. 'Report Changes' produces a text document of what changes have occurred. This is mainly used in companies that have a rigid documentation procedure in place.

When Altium is finished executing all the requested changes, the user simply needs to click 'Close' in the bottom right-hand corner of the window. This will return the user to the main schematic. The user is now ready to import the parts and components into the PCB document.

C) Designing and Producing a PCB Document

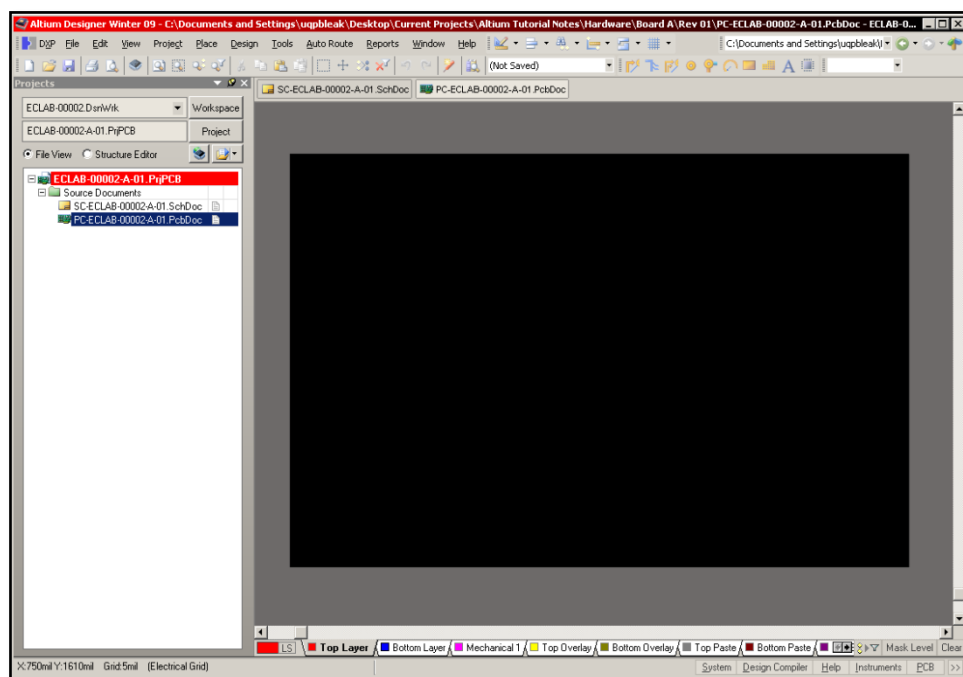
The Printed Circuit Board (PCB) document is used to produce a mechanical software model of the PCB that you wish to produce. All components are accurately placed on the PCB and tracks can be drawn between them in order to electrically connect the components together. The board shape is drawn in this document and all layers can be accessed and manipulated to correctly 'lay out' the full PCB. The latest version of Altium Designer can model the components and circuit board in 3D and this can be imported into a mechanical CAD software package to test if the circuit board will fit into its enclosure.

There are many tools available to the user for accurate PCB design and this tutorial will explain some of the basic tools. However, it is not possible to cover all aspects of PCB design in this tutorial. There are some links to PCB design websites in the appendix of these notes, but most good PCB design practice is learned through work experience over a number of years. This tutorial is merely concerned with giving students an understanding of the basic tools needed to use Altium.

1. Preparing a PCB document

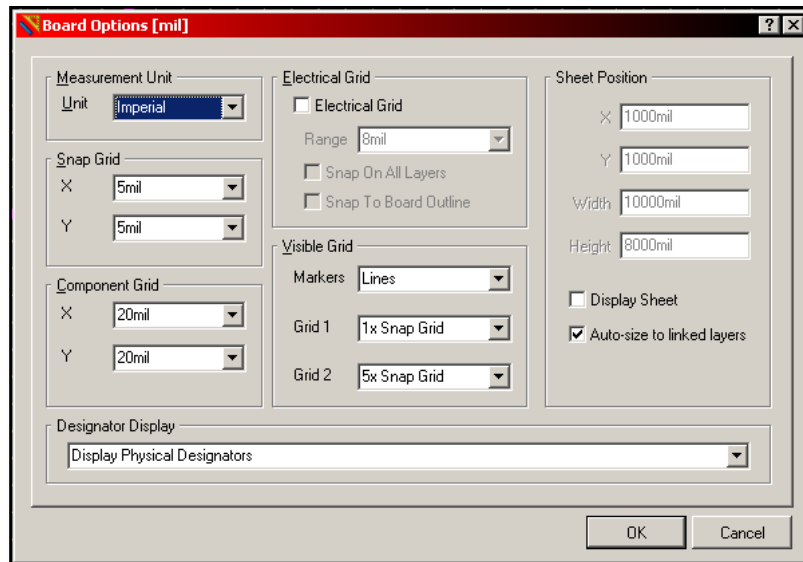
The first step in designing a PCB is to define the board space on which the components are going to be placed. This is potentially the most critical stage of PCB design as a simple mistake at this stage will be costly if the board is manufactured and does not fit its allocated space. A good rule is to check any measurements at least 3 times, preferably from multiple angles.

To start with, make sure that the PCB document is open, located within a project, and selected.



Setting up the PCB document

Altium Designer is set up for sale in the American market, and as a result, uses Imperial Units as its default unit of measurement. This can be easily changed using the 'Design Options'. This is also where most of the default settings for the PCB set up can be found. The user can open the 'Design Options' by clicking on '**Design>>Board Options**' from the menu bar at the top of the main window. This brings up the following window:



Here, the user can change the Measurement Unit between Imperial and Metric. It is also possible to change the 'Snap Grid', 'Component Grid', 'Electrical Grid' and 'Visible Grid'.

Snap Grid – This is an invisible grid that the mouse can automatically snap to. It is used when placing tracks to either increase accuracy (fine snap grid) or conform to a 'chunky' set of measurements. It is a very useful tool and can only be properly understood after some experience using it. Users are encouraged to spend some time investigating this tool and to become familiar with it.

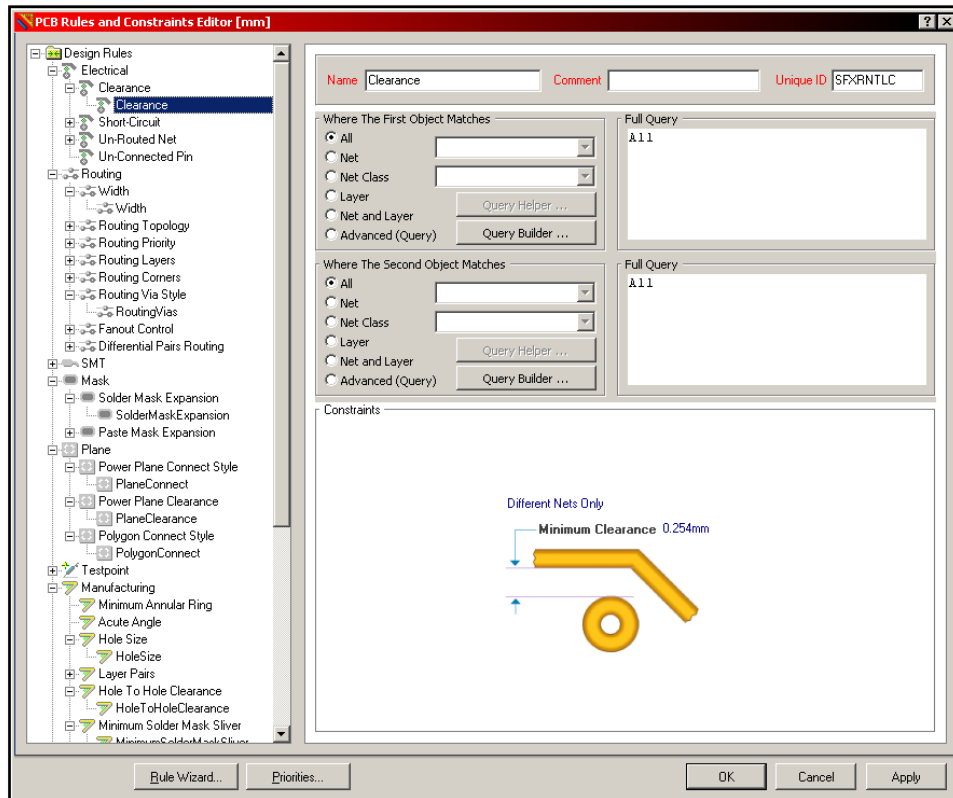
Component Grid – Like the Snap Grid, this is also an invisible grid, but this time it is used for the easy placement of components. It is good design practice to leave enough space around components to allow for easy tracking and simplistic design and this grid, if set correctly, will allow the user to do this.

Electrical Grid – This grid is used to help the user easily select the right track or component on the right layer. It makes the mouse pointer 'jump' to any component or track that is in range on the selected layer. The range is set by the user and this grid can be turned on and off depending on the user's preference.

Visible Grid – This grid is the visible grid that the user can see in the background of the main PCB design window. It has two elements to it; a primary grid and a secondary grid, both visible depending on how far the user is zoomed in on the PCB. Usually grid 1 is set to a low, blocky resolution and grid 2 is set to a high, fine resolution.

Design Rules

Another important section of Altium PCB Designer is the section that deals with 'Design Rules'. The Design Rules should be set up right at the start of the PCB design so that it is designed well from the start. Setting the Design Rules at the start will save a remarkable amount of time later on in the design process and will help the user with positioning and tracking of components. To access the Design Rules, the user should click on '**Design>>Rules**' from the menu bar at the top of the main window. A new window will appear detailing all the Design Rules applicable to the current PCB design.



On the left-hand side of this window is a list of categories. Some of the categories can be expanded, illustrating the point that there can be more than one rule per category. Realistically, the user should familiarise themselves with these rules and learn what each one means, so they can set each one specifically. The more rigid the rules, the more rigid the design of the PCB will be. However, the user can get by with just setting up some of the main rules, which are: clearance; width; polygon connect; and component clearance.

Clearance – This dictates the minimum clearance allowed between differing nets

Width – This rule allows the user to set the maximum and minimum width sizes of the tracks on all tracking layers. It also enables the user to set a 'Preferred Size' which is the width of track that the user believes will be most commonly used throughout the PCB.

Polygon Connect – This rule comes into play when the user is designing and connecting ground planes on the PCB. It is used to establish the clearance between the plane that the user is putting onto the PCB and the other nets available.

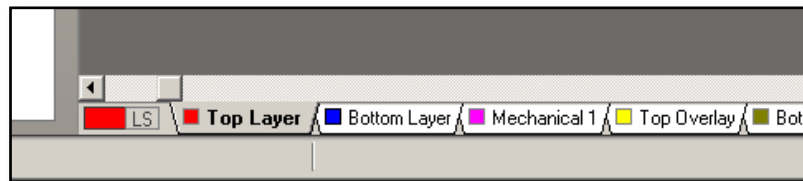
Component Clearance – Similar to the ‘clearance’ rule, this is used to dictate the minimum clearance between different components.

Once the rules have been set up, Altium will run an online check to see if all the rules have been obeyed. If there is a violation of the rules, the pad, track or component will be highlighted with a lime-green colour. The violation will continue to be highlighted until the user fixes it. Right-clicking on the violating part will bring up a menu. In this menu, under ‘Violations’ the user is able to see what rule is being broken and can take action to correct it. Altium should also ‘not allow’ the user to violate the rules directly.

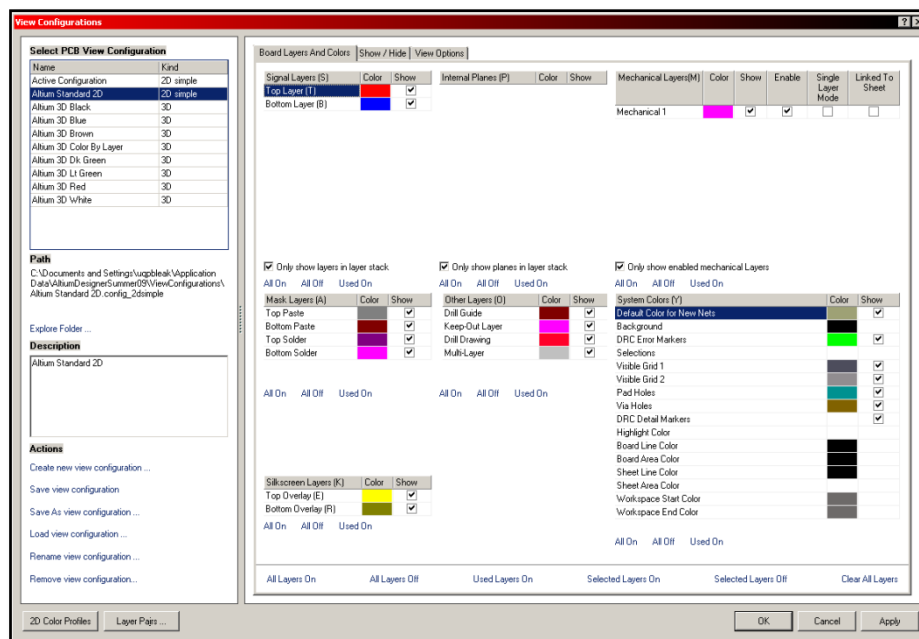
Board Shape

Layers

The next step is to outline the basic board shape. The PCB can have many layers and each layer has its own purpose. Any physical portion of the PCB should be drawn on what is called the ‘mechanical layer’. Each layer can be accessed using a series of tabs at the bottom of the main PCB document.



Layers can also be turned on and off using the ‘View Configurations’ window. This can be accessed either by pressing the ‘L’ on the keyboard or by selecting ‘**Design>>Board Layers & Colours**’ from the menu bar at the top of the main window.



Although appearing quite detailed, this window is fairly simplistic. Each layer is named and the colour representing it is placed beside it. To turn off or disable a layer, the user simply needs to un-check the box beside the colour. To re-enable the layer, the user must re-check the same box.

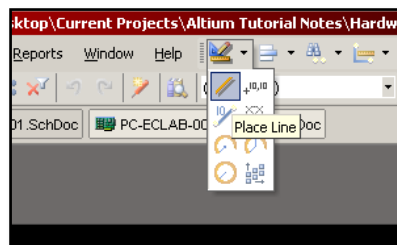
The representative colours can be changed, if the user desires, by clicking on the colour. This brings up a new window from which the user can select a new colour.

The user can change many colours configurations throughout the PCB editor from this window and these stylised configurations can be saved and re-used by utilising the controls at the left-hand side of the window. However, this is considered an advanced topic and will not be explained in this tutorial.

- Drawing the Board Outline

Click on the tab called “Mechanical 1” to select this layer for drawing. There can be multiple mechanical layers, depending on the complexity of the circuit being designed.

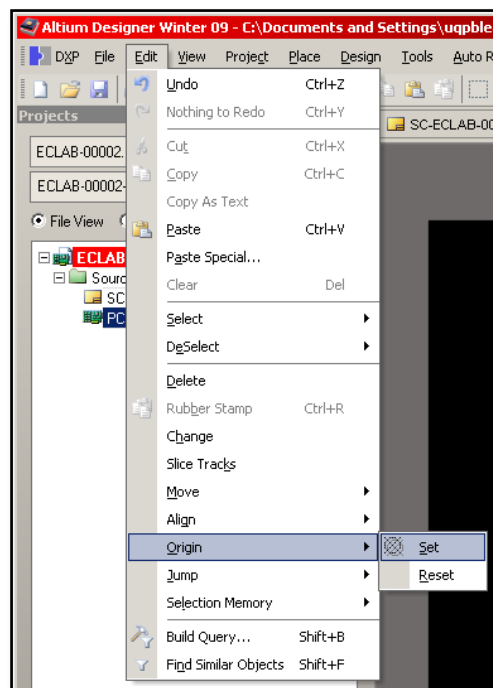
Next, click on ‘Place Line’ in the drawing toolbar at the top of the main window.



The user will notice that the mouse pointer becomes a cross-hair. Left-clicking anywhere in the main PCB window will start drawing a line. The line being drawn is a magenta colour, signifying that it is

being drawn on the mechanical layer. Once the line has been started, any further left-clicks will set a way-point and Altium will change the direction of the line by 45° . If the user wishes to change the angle that the line is being drawn at, they should press spacebar on the computer's keyboard. The user should be able to use this tool to fully draw a simple rectangle or polygon. When the basic board shape is complete, the user should right-click on the screen and the line will be finished. After right-clicking, the mouse pointer is still a cursor, indicating that, should the user wish to continue drawing lines, they simply need to left-click on the main screen again. If the user is finished drawing lines then they can right-click again to finish using this tool. The mouse pointer then reverts back to the normal arrow.

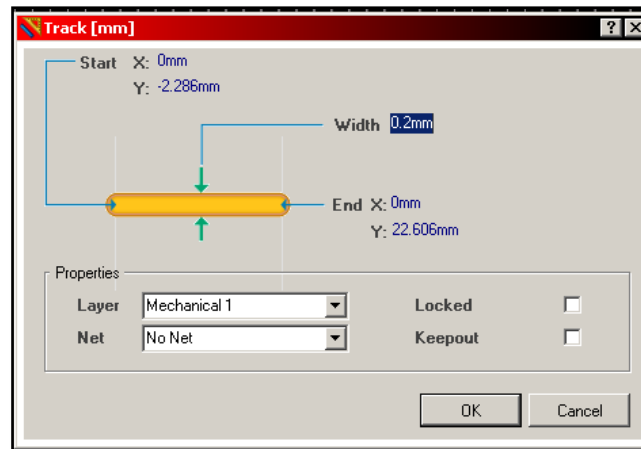
Although there is now a board shape on the screen, it may not be dimensionally accurate. This can be easily remedied but some form of reference needs to be set first. The origin for dimensions of any PCB in Altium can be changed as and when the user needs. This is done by selecting **Edit>>Origin>>Set** from the menu bar at the top of the main window.



Again, the mouse pointer becomes a cross-hair and the user can accurately pick a point that they wish to reference all measurements from (usually, when starting a new PCB, this is the bottom left-hand corner of the PCB).

The user should make sure to click on the exact point of the bottom left-hand corner of the PCB, and Altium facilitates this by (providing the 'Electrical Grid' is ON) automatically jumping to a line which is on the current drawing layer (at this stage the current layer should still be Mechanical 1). When the user is confident that the cursor is in the bottom left-hand corner of the PCB, they should left-click and the origin will be set to this point.

The user can now double-click any of the lines that have been drawn on Mechanical 1 to bring up their properties. A typical 'Properties' box looks like this:



The user is able to change any of the information displayed in this window by clicking on it and typing in new information. The user can also 'Lock' the line in place if it is important that the line does not move. Inputting the exact positions of lines and wires in this window is the most accurate method of positioning available in Altium, and it is recommended that this is used for dimensionally sensitive operations. However, for track placement, this is less commonly used as it is time-consuming to input the dimensions of every track.

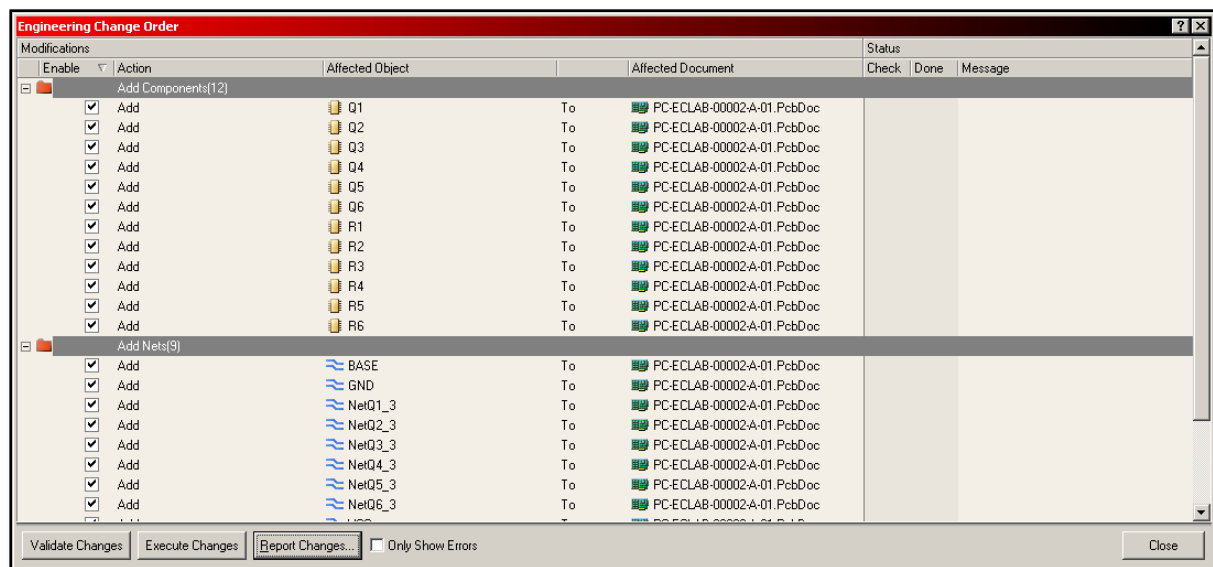
When the user is happy with the dimensions of the outline of their PCB, they will need to crop, or cut-back the PCB sheet to fit inside the outline. This is done by selecting the entire outline (use the 'Select Tool' to drag a box over the entire outline) and clicking on '**Design>>Board Shape>>Define from selected objects**'. This cuts back the black design space to fit inside the PCB shape that the user has drawn. If there are any gaps or breaks in the board outline then Altium will either flag an error or it will not cut back the black design space as requested.

The user can add holes to the board and other mechanical features at this stage if they desire, but often this is left until later in the design process.

Now that the board shape is defined, the user can add the components into the PCB design space. The following procedure illustrates how to do this automatically.

2. Transferring Parts and Components from the Schematic to the PCB

The user will need to have both the schematic and the PCB document open and both placed in the same project. The user should have the schematic sheet window selected and it should be complete with components, fully annotated and checked to make sure that all wires, components and names are correct. When the user is completely satisfied that their circuit is correct, they should click on '**Design>>Update PCB document <document name>**'. This will open a new window, similar to that of the 'Engineering Change Order' window brought up by the 'Annotate Schematics' command. This window is also called an 'ECO' window, but has a more liberal purpose than others.



The buttons have the same functions as before, but there are many more tasks in this window. Initially, this is because users must add all the component footprints, all net connections, all component classes, and all rooms to the PCB document. If the user wishes to update the schematic at a later stage and then, as a result, change the PCB, they can also use this tool to make small changes. This tool is one of the major links between the schematic document and the PCB document and is a very powerful and useful tool.

When the user is ready to add all the components, they should click 'Execute Changes' and Altium will carry out all the requested tasks. When it is finished and the user closes the window, they will be transferred automatically to the PCB document to allow for checking and placement of components.

3. Component Placement

Directly after adding the components to the PCB document, the user will be able to see all the components lined up beside their PCB. There is no real order to the components and it is up to the user to place them within the PCB boundary. This is done very simply by dragging each component on to the board. The user can click and hold the left mouse button over any component and drag it onto the PCB. While dragging, if the user presses spacebar, the orientation of the component will be changed by 90°. If the user presses 'tab' while dragging, the component's properties will be displayed.

A general tip is to closely reference the schematic layout when placing components on the PCB. This will allow the user to place components close together and have short connections between each one. This method gives the user a very general idea of where the components can be placed, but it is up to the user to implement good design practices to avoid noise and cross-talk. This tutorial does not discuss 'good PCB design practice' as this could be the subject for a single tutorial all of its own! The purpose of this tutorial is to give students the tools to be able to use Altium Designer.

A second, general tip is for the user to place the fixed components first. This means that, if there are components that absolutely **must** be placed in a specific area (i.e. an LED that protrudes through the mechanical enclosure), they should be placed first and locked in place. All mounting holes and cut-outs should be placed before putting any components onto the board as this will save time. It is a good idea to 'Lock' all the mechanical components (using the line or part properties) once they have been accurately placed on the PCB as this will prevent any accidental movement of the feature.

4. Placing Tracks (Routing)

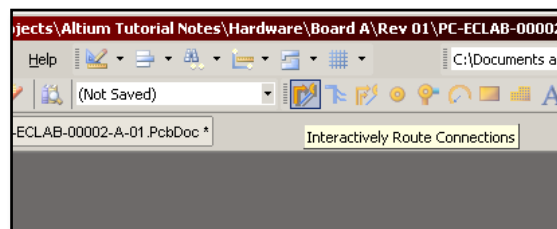
There are two ways to route tracks on a PCB; Auto Route and Interactive Routing. Although Auto Route is a useful tool, the purpose of this tutorial is to educate readers in how to use Altium from basic principles. Therefore it is deemed more useful to explain Interactive Routing and leave Auto Route for more advanced users. Auto Route is also prone to creating errors in the overall design of the PCB if it is not correctly set up. Sensitive analogue nets can be placed under noisy digital clocks and other such errors are quite common as Auto Route does not have the natural understanding of an experienced electronics designer. Therefore it is better to understand how to route the PCB manually as this helps the student develop an understanding of board layout and component placement.

Once the user is satisfied with the component layout, they will want to begin routing the PCB. To do this, they will need to understand the architecture of the PCB. A basic PCB is made up of just 2 layers, both of which can have tracks on them. Usually the bottom track will have a large ground plane and minimal tracks, whereas the top layer will have most of the tracks for connecting components. More complicated boards can have 4 or more layers, of which 2 would be power planes (Ground and Voltage Supply).

At all times the designer must keep in mind that there needs to be some method of soldering the components on to the board. It is very easy to forget about the physical construction of the board when dealing with virtual components and dimensions.

It is common for designers to have to move components during the tracking process, so be prepared for this to occur.

In order to begin tracking, the user has to click on the 'Interactively Route Connections' icon, located in the top tool bar above the main window.



Clicking this button will convert the mouse pointer to a cross-hair and the user will be able to accurately left-click on the point at which they wish to start their net. Users should note that a faint

gray line indicates the shortest path between component pads that are to be joined by a track. If the user is drawing a track on the top layer, by default the track will be coloured **red**. If the user is drawing a track on the bottom layer, the colour of the track will be **blue**. Every different tracking layer should be automatically assigned a different colour to distinguish which layer it is on.

Similar to the schematic document, left-clicking anywhere on the PCB document will set down a way-point from which the user can change direction. Also, while drawing the line, the user can press 'spacebar' on the keyboard to change direction or orientation, and they can also press 'tab' to bring up the net (or wire or track) properties. Left-clicking on the destination pad will not finish the wire. To do this the user will need to right-click. The user is able to start another wire straight away as 'Interactive Routing Mode' is still enabled. Right-clicking again will exit this if the user has finished routing wires.

As stated before, most basic PCBs have 2 layers, both of which are available for tracking. Altium Designer has the ability to draw tracks on both of these layers and it is possible to swap between the layers while drawing a track. As previously stated, Altium will automatically draw on whichever layer is selected, and layers are selected using the tabs at the bottom of the PCB document. To start a track on the bottom layer, the user must select the bottom layer **before** clicking on the "Interactively Route Connections" icon. When the user starts drawing, the track will be coloured blue and will be placed on the bottom layer.

If the user wishes to swap tracking layers in the middle of drawing a track, they simply need to press the asterisk key on the keyboard (*). This is usually located on the number pad at the right-hand side of the keyboard or is accessed by holding the shift key and pressing the number 8 at the top of the keyboard. Pressing this key will place a 'Via' on the PCB. A via is merely a plated through-hole that connects the track on the bottom to the track on the top. The size and the shape of the via are set up by default in Altium, but can be changed by the user if they desire by double-clicking on the via or by pressing tab before the via has been fully placed. The user should remember that vias need more room than a track and also represent a potential source of noise or fault, and therefore their use should be minimised. Vias are, however, a necessary and useful tool to be used throughout PCB design.

5. Polygon Pours, Keep-Out Layer, and Power Planes

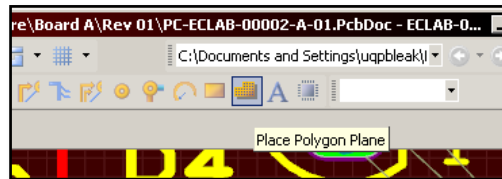
Usually, the final step in any PCB design is to place a ground plane. Again, ground plane design could be the topic of a separate tutorial as ground planes should be designed to reduce potential noise and ensure a solid return path to ground.

The user must take all steps necessary to reduce the possibility of a short-circuit to ground. It is therefore a good idea to increase the 'clearance between nets' (use the Design Rules) before placing the ground plane.

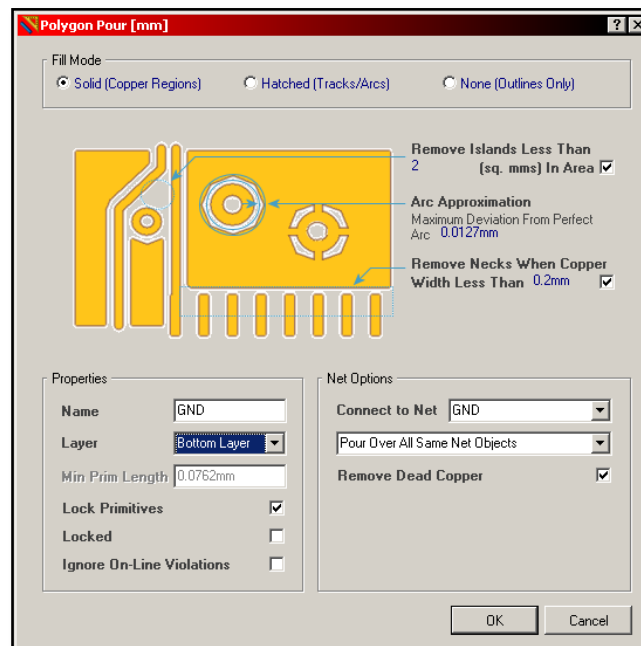
A second tip is to place a 'Keep-Out' line around the outline of the PCB. Keep-out lines are used to restrict the areas that the user can draw and they interact with tracks and planes on **every** layer. They are particularly useful when using Auto Route as they can tell the computer which areas to

avoid. They are also useful for defining the area in which a ground plane can be placed and for 'cropping' long fingers of ground plane that serve no purpose other than to introduce noise! Keep-out lines are, by default, the same colour as the mechanical layer, so it is easy to get confused between them.

Placing Keep-out lines around the outline of the PCB will make placing the ground plane a simple process. The user should click on the 'Place Polygon Plane' icon, located in the menu bar at the top of the main PCB design window.



This will open up the 'Polygon Pour Properties' window.



This window is used to set up the polygon pour before it is actually placed. This helps Altium decide which nets to connect it to and which nets to avoid. It also dictates how precise the polygon pour should be and what the limits are going to be.

Properties

- Name – The user can give the ground plane any name they desire, but it is recommended to stick to something specific and descriptive.
- Layer – Most ground planes will go on the bottom layer of the PCB. Use this drop-down selection box to select which layer you wish to place the plane on.



- Lock Primitives – A plane is made up of many lines and polygon shapes. If the user wishes to be able to individually select the lines and polygons that make up the ground plane, they should un-check this box. Otherwise it should be left checked to keep the ground plane as one whole unit.
- Locked – Checking this box locks the ground plane in place so that it cannot be moved or changed
- Ignore On-Line Violations – Checking this box will allow the polygon pour to break the Design Rules.

Net Options

- If producing a ground plane, the user should select GND from the 'Connect to Net' drop-down menu. This will allow the polygon pour to connect to all the ground nets on the PCB.
- The user will want the ground to connect to and pour over all the ground connections possible, so they should select 'Pour Over All Same Net Objects'.
- 'Dead Copper' is any part of the plane that is not connected to any net. It is best to remove dead copper to reduce the risk of short-circuits or to reduce the risk of noise in the system.

Once all these options have been set, the user should click 'OK' and will then be prompted to draw the outline of the polygon shape. If the user has outlined the board with keep-out they simply need to draw their polygon pour shape outside the outline of the board. The user should pick a starting point and left-click once on the PCB design area at this point. This will start the outline process and will give the user a polygon guide as to where the next outline border will be placed should they left-click again. The user should click at the four corners of the PCB to produce an outline of the polygon pour. When this is complete, they can right-click to exit the tool. Altium Designer will automatically connect the final point that the user clicked at with the first point, completing the polygon. It will then draw the polygon using the rules that the user has set up.

D) Exporting PCB's into Mechanical CAD Software

For the subject of METR2800, students have been given fully functional 3D libraries that contain all the 3D components they need to draw the required PCB. This also enables students to view and use the 3D functions of Altium Designer.

Altium is able to save and export 3D models of finished PCBs in to any mechanical CAD software. This is extremely useful in the design process as designers are able to test if their circuit boards are going to fit into the mechanical enclosure long before the circuit is even manufactured. This saves a lot of time and money.

Changing to 3D Mode

Users can view their PCB in 3D at any time by simply pressing the '3' key on the keyboard. Pressing '2' will revert back to 2D mode.

Using 3D Mode

Changing and moving the PCB in 3D mode is quite tricky, as 3-dimensions are very hard to represent on a 2-dimensional surface such as a computer monitor. Altium is able to rotate the PCB around the 3 axes' fairly easily but it is very easy to lose perspective. For this reason, Altium has buttons to return the user to a default, best-fit view.

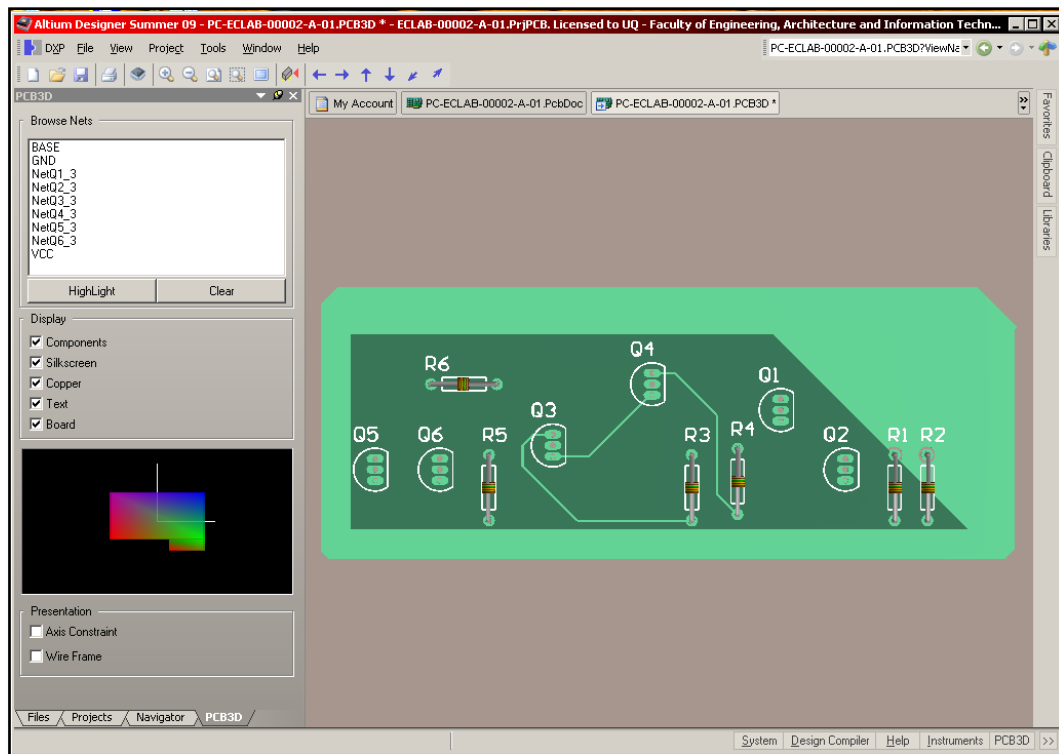
Pressing '0' at any time will return the PCB to its default, flat view (the same orientation as 2D mode). Pressing '9' will rotate the PCB 90 degrees clockwise. Other view functions are available in the View menu.

To rotate the PCB in any direction, the user needs to hold down the SHIFT key on the keyboard and press and hold the right-hand mouse button. This will bring up a spherical cross-hair with a circle around it. The user should be able to use this spherical pointer to gauge the depth and angle of rotation of the PCB. Altium will try to assist in this by locking the PCB to one axis of rotation at a time. The PCB will also lock to a plane if it is positioned in a close enough orientation to the plane. Although this form of manipulation is tricky to get used to, it serves its purpose. There are other tools available to flip and rotate the PCB automatically, and most of these are in the View menu at the top of the main window.

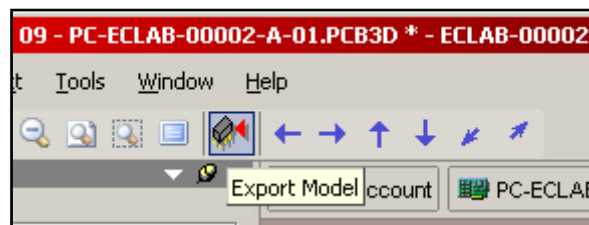
Exporting to Mechanical CAD Software

Altium uses a portion of legacy 3D software to export 3D drawings to mechanical CAD software. It can produce *.iges or *.step files which can then be opened using the CAD software.

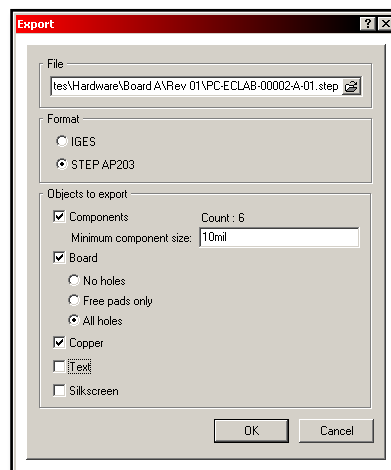
In PCB mode, the user needs to click on '**Tools>>Legacy Tools>>Legacy 3D View**' to open the legacy 3D viewing software. This will open a new window, as shown below.



At the top of the window is an icon that looks like an IC with a red arrow pointing at it. This is the tool that allows the user to export their PCB model to a mechanical CAD software package.



Clicking this icon will bring up another window which helps the user set the parameters for the file export.

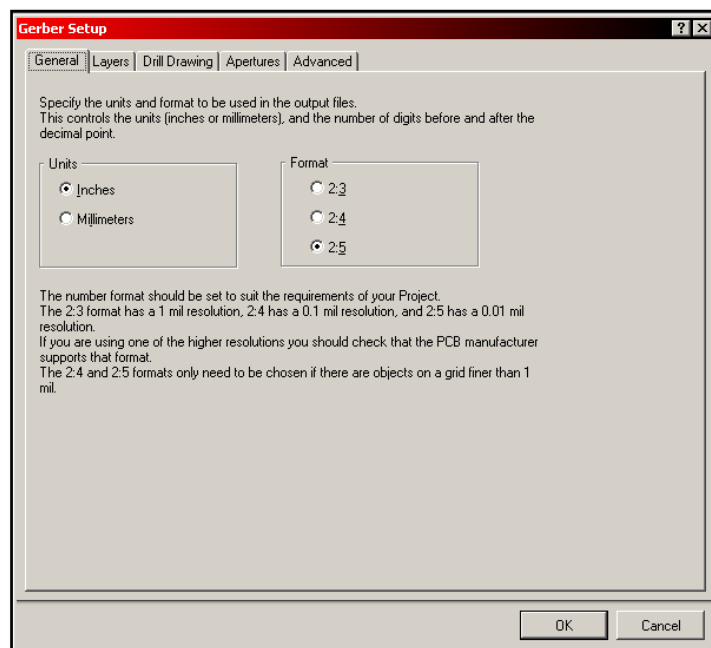


The user can choose what they want to export and into which format they wish to save the created file. Exporting all the text, copper and silkscreen can mean that the created file is extremely large. If the user is merely trying to test the size of the PCB and its components then it is only recommended that they export the board with its components in place.

Generating Gerber Files

Gerber files are sometimes required by PCB manufacturers when it comes time to fabricate the PCB. They are highly accurate, scale drawings of each and every layer on the PCB, usually separated into individual files for ease of reference. Altium Designer is able to generate these for the Electronics Designer with relatively little effort.

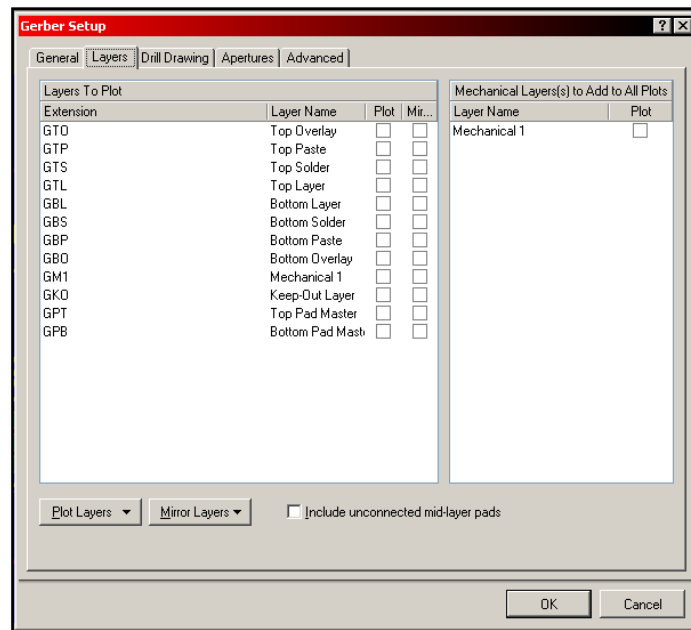
Once the user is satisfied with their circuit board in both 3D as well as 2D, they should save all their work. They can then click on **'File>>Fabrication Outputs>>Gerber Files'** from the menu bar at the top of the main window. There are other tools in the 'Fabrication Outputs' menu which the user should explore as they can help to produce some useful documents that can be used throughout industry. However, clicking on the 'Gerber Files' tool opens a new window.



This window has a number of tabs along the top labelled 'General', 'Layers', 'Drill Drawing', 'Apertures', and 'Advanced'. Very often, it is the manufacturers who specify the set up details of these parameters, so this tutorial will attempt to guide the user in what each parameter means.

General – This tab is used to specify the units and format of the output files. The 'Units' box on the left is where the user specifies whether to produce the Gerber Files in Inches or Millimetres. The 'Format' tab specifies the resolution or level of accuracy of the drawings (i.e. how many decimal places the measurements are correct to)

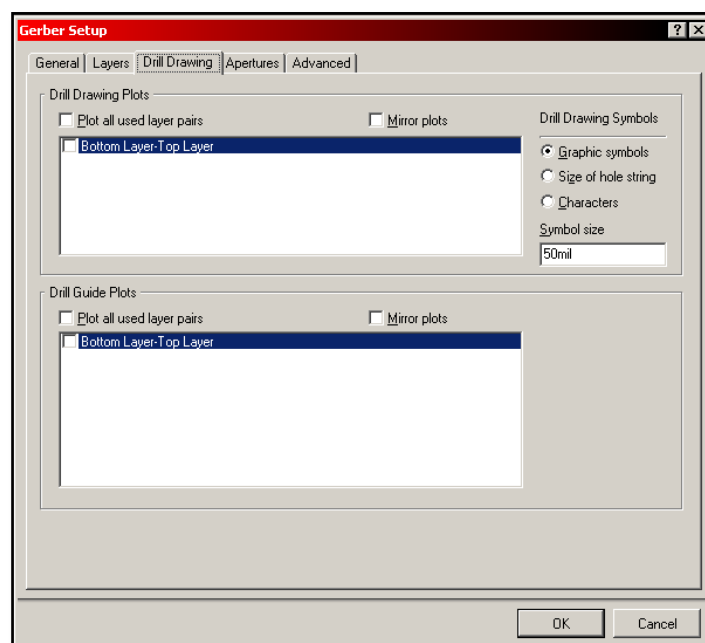
Layers



The 'Layers' tab allows the user to select which layers are converted into Gerber files. It is common to select all the layers individually as the manufacturer will need as much detail as possible.

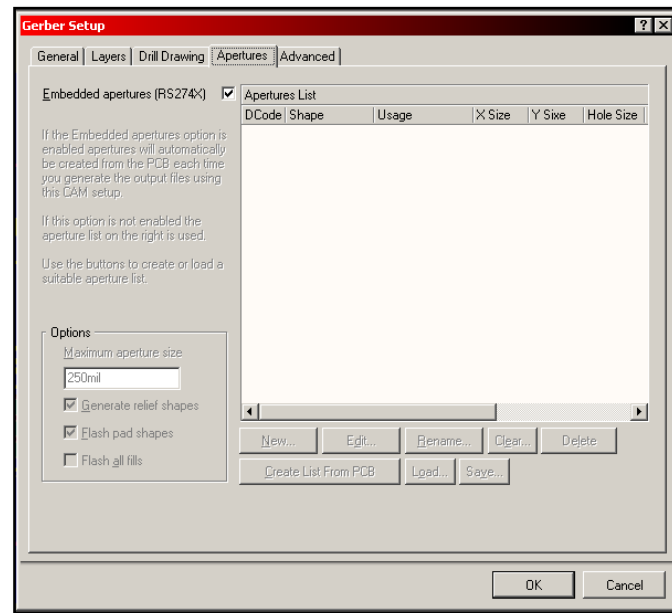
It is not advisable to put the mechanical layer on every layer of the PCB unless the user has some specific reason for this. Mirroring the layers is sometimes useful depending on the reasoning behind the user producing the Gerber files. The boxes at the bottom of the window give the user some short-cut menu options to allow them to speed up the process of selecting the layers.

Drill Drawing



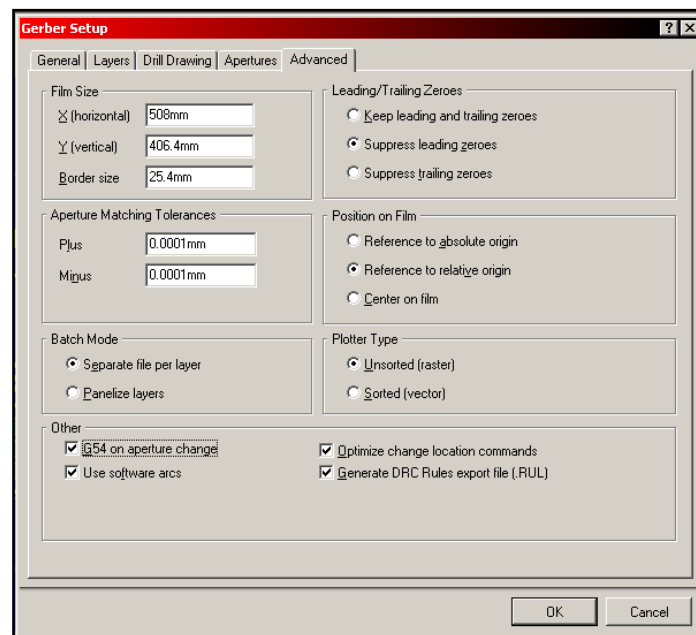
As part of the Gerber Files, a drill drawing can be produced. If the user is sending this to a professional manufacturer, they will have some kind of automated drilling machine that will drill the holes of the PCB very precisely. This drill drawing file will detail the co-ordinates and sizes of each hole on the PCB and will be able to be read by the drilling machine. Normally, the user will select 'Plot all used layer pairs' in both 'Drill Drawing Plots' and 'Drill Guide Plots'.

Apertures



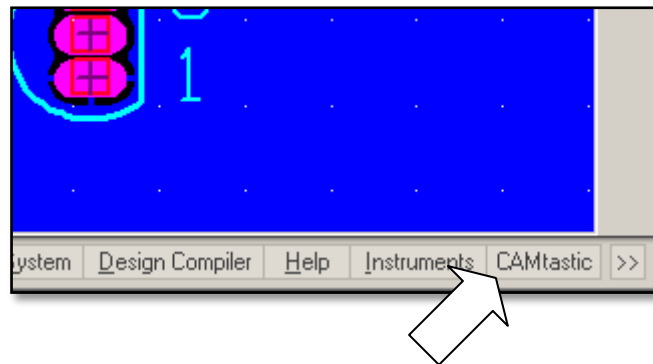
This tab is usually left with the 'Embedded apertures (RS274X)' box checked.

Advanced



The 'Advanced' options are more for advanced users and don't need much change as they should be set up automatically by Altium. The only check that the user should definitely perform in this window is within the 'Batch Mode' box. Here the user can choose to either have one separate file for each layer of the PCB, or they can choose to panelise the whole set of Gerber files onto one large sheet. For ease of reference, 'Separate file per layer' is usually selected.

When the user clicks the 'OK' button, a new program is launched within Altium called CAMtastic. This program is used to edit, analyse and export the Gerber files in the correct format for production. To view the layers and open and close each one, the user needs to select the CAMtastic window from the bottom right of the screen.



Clicking this icon brings up a list of all the layers that have been plotted and the user can select and de-select each one individually. The user should save these files and will notice that they are stored in a single with the file extension *.cam . This can be opened from within Altium or each file can be sent to the manufacturer individually (if the user navigates to the project folder they will see each individual file within the 'Project Outputs' folder).

Summary

The tools described in this tutorial should give the user a basic understanding of how to produce schematic and PCB documents in Altium Designer. Further functions and processes within Altium may be explained in an appendix to this document; however these add more detail than is necessary at this stage.

If there are any comments or feedback on this document that would be useful for future revisions, then please forward them to p.bleakley@uq.edu.au or contact a member of the Instrumentation Support Group on +61 (0)7 3365 3600