

Easy-PC

Users Guide V28.0

Copyright ©1987-2024 by WestDev Ltd. All rights Reserved. E & O E

Number One Systems

20 Miller Court
Severn Drive
Tewkesbury
Glos, GL20 8DN
United Kingdom

Phone +44 (0)1684 296 501

Email: sales@numberone.com

Technical support: +44 (0)1480 382 538

Email: support@numberone.com

Web site: www.numberone.com

The product is licensed by WestDev Ltd to the original purchaser of the product for use only on the terms set forth in the WestDev Licence agreement. Copying, duplicating, selling or using the product contrary to these terms is strictly forbidden.

This Manual Copyright to WestDev Ltd. All Rights reserved. No part of it may be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine readable form without WestDev Ltd. prior written permission.

Information in this manual is subject to change without notice and does not represent a commitment on the part of the vendor.

Number One Systems, Easy-PC, Easy-Spice, Easy-Router, Pro-Router, Trace-Router, Pulsar, Layan, Analyser, MultiRouter, Filtech, Z-Match, StockIt and all variants thereof are Trademarks of WestDev Ltd.

All other trademarks acknowledged to their rightful owners.

Number One Systems, a trading division of WestDev Ltd.

While every care has been taken in the preparation of this manual WestDev Ltd cannot be held responsible for any errors or omissions within it. If informed we will add comments and features which you may like to see written which may help others using this manual. Please send your comments through the technical support desk.

Manual dated: 08/10/24

Contents

CONTENTS	3
CHAPTER 1. GETTING STARTED	9
Welcome	9
About this Manual.....	9
CHAPTER 2. INSTALLATION.....	11
Hardware Requirements.....	11
Installing the Product	11
Moving Your Installation.....	12
Updating Your Menus.....	13
Installing Cost Options.....	13
Installing Cost Option Libraries.....	14
Running The Program.....	14
CHAPTER 3. CONCEPT OF OPERATION	15
Introduction.....	15
Creating Designs.....	15
Projects (Overview)	16
Schematic Files	16
PCB Files	17
Libraries.....	17
PCB Manufacturing Files for Plotting.....	18
Job Files for Plotting.....	18
Panel Files for Plotting.....	18
Technology Files.....	18
CHAPTER 4. USING THE INTERFACE	19
Overview.....	19
The User Interface.....	19
Menus.....	20
The Online Help System	21
Status Bar	21
Opening Multiple Documents	22
Customising the User Interface.....	22
Toolbars	22
Commands and Shortcut Keys	24
Rulers.....	25
Dockable Windows	26
Component Bin	27
Add Component Bar	28
Colour Files Bar	29
Goto Bar.....	29
Layers Bar.....	31
World View.....	33
Shape Information Bar	33
Design Notes Bar	34
Properties Bar.....	35
Viewing the Design.....	36
Mouse Operation.....	36
Numeric Keys to Zoom In and Out	36
Frame View.....	36
Viewing all of the design	37
Pan	37

4 Users Guide

Auto Pan	37
Design Item Selection	38
Standard Selection Operations	38
Other Selection Mechanisms	39
Controlling the Selection	40
User Defined Preferences	42
Properties Dialog	43
CHAPTER 5. GENERAL UTILITIES	45
Undo and Redo	45
Interactive Operations	45
Moving and Placing Items	46
Mirror (Flip)	47
Rotate	48
Fix & Unfix Items	49
Cut, Copy, Paste and Duplicate	50
Replicate	51
Delete	52
Adding Text	53
Adding Barcodes	55
Mitres, Fillets and Arcs	57
CHAPTER 6. SETTING UP THE DESIGN	59
Projects	59
File Operations	60
Folders	61
Saving Your Files	61
Security Copies	61
Units and Co-ordinates	62
Grids	64
Change to the Grid Snap context menu	67
Changing the Working Area	67
Changing the System Coordinate Origin	67
Working in Absolute and Relative Modes	68
Using Technology Files	70
Overview of Technology Files	70
Styles in Technology Files	71
Merging Styles	74
The relationship between Net Names and Net Classes	75
Net Classes	76
Net Class to Class Spacing Rules	78
PCB Specific Items in the PCB Technology File	79
Changing Styles Globally and Locally	79
Track Length Rules on Net Classes	79
Defaults	83
Loading and Reusing Technology Files	83
Exporting & Importing CSV Files	85
Changing Colours	85
Working with Values	89
Drag and Drop Values	93
Using the Component Values Editor	94
CHAPTER 7. CREATING SCHEMATIC DESIGNS	96
Overview and Getting Started	96
Starting a New Schematic Design	96
Setting Up a Schematic Technology	98
Styles	98

Net Names and Net Classes	98
Drawing Borders.....	99
Changing the Colours of the Schematic	99
Adding Drawing Borders	100
Adding Components	100
Using the Component Bin	100
Using the Add Component dialog.....	101
Introduction to manipulating Components in the design.....	104
Using Decoupling Capacitors.....	105
Adding Connectors	106
Reloading Components	106
Adding Connections to the Design.....	107
Adding Connections.....	109
Dragging off Pins to Start Connections.....	114
Adding Nets that aren't shown on the design (Implied Nets).....	115
Naming and Renaming Nets	116
Power and Ground Symbols in Schematics.....	117
Cross Sheet References	117
Page Links.....	118
Using Busses.....	119
Adding more Sheets to the Project.....	123
Adding Circuitry from Other Designs.....	124
Resolving Net Names.....	125
Hierarchical Designs	125
Overview.....	125
Using Hierarchy	133
Adding Shapes	137
Schematic Design Rules Checking.....	138
Transferring Schematics into PCB Layouts	139
Exporting Netlists	140
Reverse Engineer	140
Printing the Schematics.....	141
CHAPTER 8. CREATING PCB DESIGNS.....	144
Getting Started with a PCB Design	144
Starting the Design.....	144
Design Setup for PCB layout	147
Setting up a PCB Technology	148
Pad Styles.....	148
Track Styles	149
Spacing Rules.....	149
Design Rules	150
Layers.....	151
Creating Layers.....	152
Layer Types	153
Layer Spans.....	155
Changing Colours in PCB.....	156
Translating from Schematics.....	159
Adding a Board Outline	159
Adding 'Free' Pads.....	161
Adding Components	161
Using the Component Bin.....	163
Updating Components.....	164
Replacing Components	166
Placing the Design Interactively.....	167

Adding Connections	172
Net Optimisation.....	174
Adding Tracks and Routing the Design	175
Editing Existing Tracks.....	181
Other editing options and techniques.....	182
Unrouting Tracks	183
Viewing the Grids.....	184
Snapping Tracks to Grid	184
Adding Stub Routes.....	185
Track Fattening.....	186
Automatic Track Necking into SMD Pads.....	187
Auto Neck Track.....	188
Swap Layers Mode	189
Display Clearances	189
Pull-Tight Mode.....	190
Online Design Rule Check (DRC).....	191
Differential Pairs.....	191
Copper Pouring.....	196
Copper Pour Areas.....	197
Pour Copper.....	197
Copper Pour Keepout Areas	201
Split Power Planes	203
Teardrops.....	203
Dimensions	205
Text Callouts.....	207
Measure	208
Design Rules Checking.....	210
DRC Sign-Off Checks	213
‘Accepted’ DRC Errors (PCB)	215
Adding Shapes.....	216
Component Height Checking.....	221
Integrity Checking	222
Bringing Design Changes Forward from Schematics	223
Connectivity Check.....	224
Back Annotation to Schematics	225
Using the Cross Probe Mode	226
Using the 3D Viewer.....	228
Defining the 3D Packages.....	229
System Reports	230
Component Positions Report	231
HTML Reports.....	231
Custom Reports	232
BOM (Bill of Materials) / Parts List Composer	233
CHAPTER 9. OUTPUT FOR MANUFACTURE	235
Overview	235
The Plotting and Printing Mechanism.....	235
The plotting ‘process’	235
Creating a Plot Job.....	236
Plot Jobs.....	237
Plots	238
Plotting & Printing Devices	238
Auto-generating Plots	239
Adding Plots to a Job.....	240
Editing Plots	241

The Options Dialog	247
Pre-Plotting Checks.....	251
Drill Drawings	251
Using Drill Tables in the Design.....	252
Power Planes.....	254
Plotting Solder Resist Layers	256
Viewing Power planes.....	259
Tented Vias	261
Split Power planes.....	263
Schematics Print/Plot Outputs.....	264
Windows and Pen Plotting	264
Printing Across Multiple Pages.....	265
CHAPTER 10. AUTOMATIC ROUTERS IN PCB	266
Trace-Router	266
Automatic Routers Overview	266
Trace-Router Overview.....	266
Preparing and Routing Designs.....	268
CHAPTER 11. LIBRARY CREATION	276
Understanding Libraries	276
Introduction to Schematic and PCB Symbol Libraries.....	279
Introduction to Component Libraries	279
Creating Schematic Symbols	280
Introduction to Library Creation Wizards	294
Creating PCB Symbols (Footprints).....	294
Additional Footprint information	300
Creating Surface Mounted Footprints	302
Creating Components.....	303
Adding More Component Detail.....	314
Adding Component Values	314
Adding and Removing Packages.....	315
Predefining Terminal Names	317
Pre-assigning Net Names	317
Mapping One Schematic Symbol Pin to Multiple PCB Pins.....	318
Checking and Verifying Component Pin Assignments	319
Library Manager Styles Editor.....	321
CHAPTER 12. MANAGING LIBRARIES.....	323
Managing Libraries	323
The Library Manager	323
Adding and Merging Libraries.....	324
Importing Libraries From Designs	324
Library Databook Tool.....	324
CHAPTER 13. LIBRARIES.....	327
Standard Supplied Libraries	327
Library Definitions.....	327
SnapEDA Component Browser	328
INDEX	329

Chapter 1. Getting Started

Welcome

Welcome to Easy-PC, the Windows generation of the original award winning EASY-PC that was launched in 1988. At that time the product was revolutionary in its affordability and usability and many tens of thousands of licences have been sold to customers in over 100 countries world-wide.

All features provided make the task of designing Schematics and printed circuit boards (PCBs) easier than ever before. Transparent links to Spice simulators and PCB autorouters provide an integrated solution. Web-based library content is instantly available for simply dropping into your designs.

The use of Technology Files, the equivalent of Style Sheets in a word processor, greatly reduce the set-up time for new projects and enable companies to standardise their operating procedures and design rules according to the type of product being designed or the manufacturing technology being used. Technology files also include company standard net names and the track widths and clearance rules to be applied to those nets. This will ensure that proven standards can be maintained over a range of similar products with the minimum of effort.

In addition to this, all the expected professional output and manufacturing features are provided, including Photo plotting outputs to Gerber and Excellon NC drill, ODB++, IPC-2582, Windows printers, PDF and pen plotters. Mechanical CAD programs are supported using the DXF and IDF formats. Additional reports for assembly machine drivers are provided as part of the standard product to further aid productivity.

About this Manual

This manual has been created to guide you through a reasonable 'normal' design flow. Obviously, what might be normal for one person is not normal for another. Don't be alarmed if you find that you have not used some of the options featured, you can skip a section without it being detrimental to the end result!

The manual doesn't take you through every small detail of the product but will guide you through the essentials to produce a design. It is directed at users who understand the fundamentals of designing PCBs, it is not intended as a course in designing PCBs.

Chapter 2. Installation

Hardware Requirements

The table below shows the hardware requirements. It shows both the minimum and the recommended specification. It should be said that this product will run pretty much on any off-the-shelf PC available in most computer stores.

	Recommendation
Processor	i5 processor or better, such as i7 or i9
RAM	4GB or more, such as 8 or 16GB
Hard disk space	1GB for product and library content
Operating System	Windows 10 or 11
Graphics	Better than 1024x768
Mouse	Wheel Mouse

* MAC O/S or Linux are not supported. All supported operating systems are supported in both 32 and 64-bit formats. Please visit our web site for the latest news on supported operating systems www.numberone.com/lifecycle.asp

Installing the Product

Regardless of whether you are installing a new system or updating your existing system, the procedure is the same. The setup program will automate as much of the process as it can for you with minimal input.

Download and Save

You would have received your installation via as a download link that enables you to access product from our web site.

Save this downloaded file, it is a self-extracting executable program. This is not a ghost installer; this is the whole program in one file. In the event that you require a backup copy of the installer, save this executable to a location on your system that gets backed up. It can then be recalled and used in future. It will be the current 'base' version at the time of download. To bring it up to date in the future, you would also need to apply any services patches that you would have also downloaded over the life time of this version. These should also be downloaded and saved in a backup location.

Installation

The Easy-PC install program is straight forward and efficient to use.

Double-click on the install program to run it once it has been downloaded to your local hard drive.

On running, it displays the **Welcome** dialog:



You will need enter each tab to provide the information required, such as agreeing to the **License Agreement**, your **Registration** details and product install code. The two other tabs inform you where the product will be installed (**Directories**) and changing file extensions (under **Other Options**).

You will be asked for your **customer number**, this would have been identified to you in the email with the download link.

You will be asked for your **Name** and **Company Name**. If you don't have a company, just type your name again.

For each major version of the product, you would also need to have your **16-digit installation serial** number to hand. This would have been provided to you when the product was purchased. This can be copied and pasted into the entry box or typed in. Care should be taken when typing in the Serial number sequence, as this causes many unnecessary support calls, especially regarding the I and l, 0 and o keys!! Make sure you use the correct code for the product that you are installing (another cause for support calls!). It is better to copy/paste if you can to save typing errors. Do not pass this number to anyone else, it does identify you and is traceable back to you.

Once the install process has been completed a '**Finished**' button appears, click this to exit. The product is now ready to run.

Moving Your Installation

Your product can be installed onto any logical drive that is part of your computers' environment. If you decide to relocate the installation once an initial install has been made, you may do this but you **cannot** simply move it. The product must be reinstalled using the downloaded executable file using the new location. As predefined Registry settings are used to set up many of the functions and to store data; the only way to restore these settings is to save the existing settings by saving a Configuration file.

Updating Your Menus

For existing users where the latest version of the software is installed alongside the old version, the installer is designed to make minimal disruptions to the existing installation. It is assumed that you have created the menu and toolbar set required so this is not changed.

To reset the menus and toolbars to the new set, go to the **Settings** menu and **Customise**. Use **Reset** on the **Menu** tab for the relevant menu type.

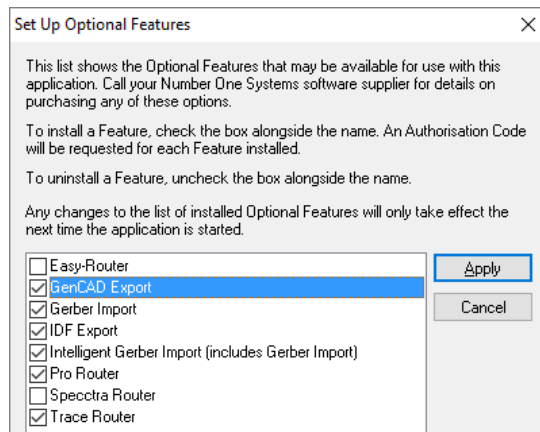
Installing Cost Options

Cost options are available to increase the capabilities of the product. Our web site also contains information about cost options.

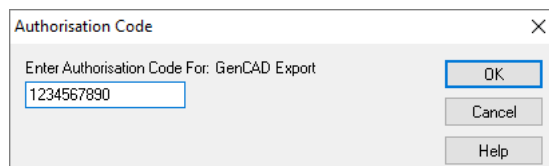
Some cost options are automatically installed during the installation of the main product. All cost option products except the **Pro-Library**, **Micro Library**, **Connector Library** and **IPC SM Library** options are installed. These libraries are installed using their own separate downloaded installations.

Each of the cost options will require an additional **serial number** supplied when purchased - **Trace-Router**, **Pro-Router**, **GenCAD Export** etc. These cost options are preinstalled and must be activated for use.

The main product must be installed and already running. Cost options are initialised through the **Settings** menu, **Optional Features** dialog. Each of the features can be selected for authorisation but the process of applying the code will be requested for each of the options required. The product serial number is supplied on the **License Authorisation** page or by email and are used for activation.



Select the relevant option from the list (you must have purchased this option first and been given the serial number to install it with). Click the **Apply** button. Type in the **serial number** provided to you (copy this carefully from the license page or copy/paste from the email).



With the correct serial number entered, a message will verify that the code number has been successful, click **OK** to accept it.

You can license more than one option in this session, license each option required.

You must restart the product to activate the options. Each cost option authorised is now available to run and will be indicated by the 'tick' in the product check box next to the product in the **Optional Features** list.

Installing Cost Option Libraries

Unlike other cost options, the **Pro-Library**, **Micro Library**, **Connector Library** and **IPC SM Library** are installed from separate install programs supplied to you as download links at the time of purchase.

On double-click of the downloaded install executable, the installer will check and verify an existing library, once found, it will offer this folder to install the optional libraries into. Once installed, the library will be immediately ready for use.

Running The Program

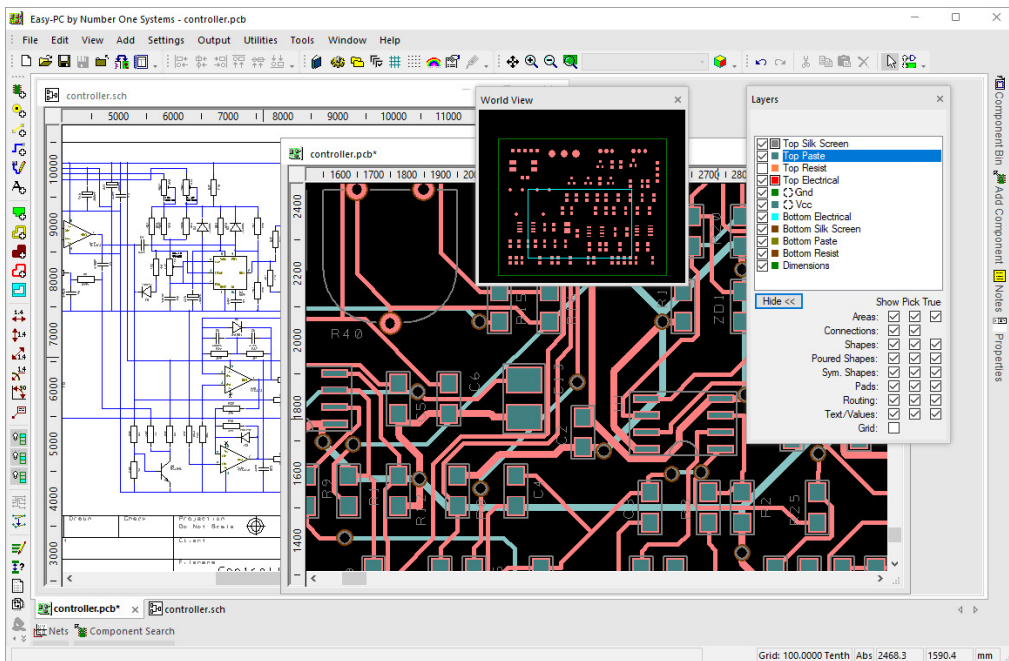
The program can be started using a button on the **Start** menu under **Programs** and **Number One Systems**. When selected, this will start **Easy-PC** in its default location. This mechanism is perfectly adequate, but for more frequent users who wish to run the program from their own location, a more convenient way of running the program is via a desktop shortcut that has been setup for you to include a suitable working folder.

Chapter 3. Concept of Operation

Introduction

Please note that this chapter contains important information relating to the concepts behind this program. It is highly recommended that all first time users read this chapter first.

This program has been designed to provide an integrated design environment for as many aspects of the design cycle as possible. Tools to assist this aim include project files, which coordinate all aspects of the design; Technology files, which standardise design parameters; and libraries, which provide easily managed component resources.



Creating Designs

The design process usually involves the creation of the initial Schematic design (although it doesn't have to be, you can create your design directly into the PCB design editor itself if you wish), the transfer of the schematic into the PCB design editor, and the layout of the PCB.

During this process you may need to create library items, modify existing ones or download available libraries off the internet.

A **Glossary of Terms** is available at the back of this manual to help you with electronics design terminology.

Creating designs – the 'basics'

As a very basic guide, this short section will help you understand the way some of the options are used.

'True' connectivity in Schematics are automatically transferred to the PCB design editor, this in-turn is transferred right through to a final plot or print thus retaining the original concept but in a physical layout.

The first of these concerns the ability to maintain the connectivity of your circuit when you move from drawing its Schematic diagram to laying out its Printed Circuit Board (PCB). This facility ensures that, when you start to layout the PCB, the connections on your board are the same as those in the circuit diagram. Obviously, as you work on the board layout, you have the power to alter the connections, for example, by adding or deleting tracks. An 'update' facility is included to make sure that the Schematic and PCB stay in-step electrically at all times. This where the program extracts the connection information from the PCB layout and compares it with similar information extracted from the Schematic. A full report warns you of any discrepancies between the two and will automatically update your PCB to bring them in line.

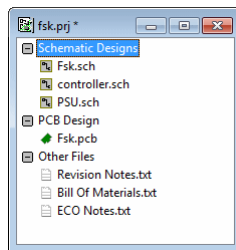
In the PCB design, the DRC (Design Rules Check) facility ensures that you have not violated the spacing rules defined, where tracks are crossed or they have been routed too close to other objects on the board for example. These methods of checking allow you to decide whether to make alterations to your circuit either on the Schematic or the PCB, and still ensure that the two parts of your design contain the same connective information.

Projects (Overview)

Projects can be optionally used to collate and present your design and associated files. You can decide to use them from the start, add them part way through a design or not use them at all.

Projects are used to tie single or multiple Schematic sheets with a PCB design. Each sheet of a Schematic is effectively a single design file. Projects can also contain blocks of hierarchical circuitry within a design, they can also contain associated files, such as BOMS, reports etc.

The use of a Project will enable net names and Component names to be tracked through each sheet so that duplicates are not produced. The Project file is also a neat way of grouping related design information in one file.



Schematic Files

Schematic files have the extension '.sch'. They represent the interconnections of the circuit in a logical way so that its operation becomes clear. The symbols used to represent components are highly stylised, rarely bearing any physical resemblance to the actual device. Single lines join these symbols, with the logical flow of information normally being from left to right. These signal lines may be named, and where several run together, they may be represented by a much thicker line known as a Bus.

A Schematic may be broken into several sheets for clarity, with signals crossing between sheets. The schematic editor allows multiple sheets, which will translate to a common PCB, to

co-exist in one working area. It is customary for each sheet to be surrounded by a drawing frame (or drawing blank or drawing border). This contains information such as the project name, design title, author, date, revision, and sheet number. If several Schematics are saved together as a project, they will merge together providing the Translate to PCB is performed within the project environment.

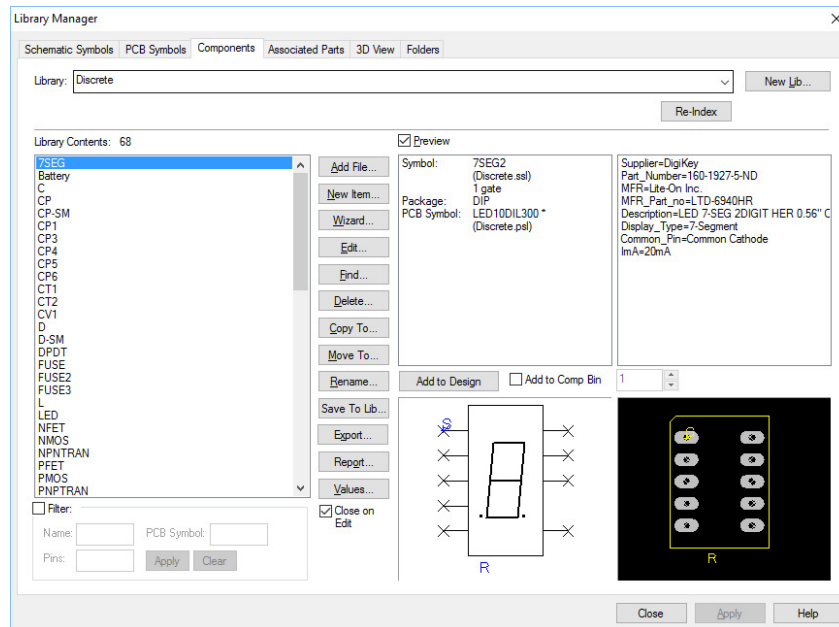
A copy of every style used, and every component used is saved in the file. Any component already in the file will be used in preference to importing a new definition from a library.

PCB Files

PCB or layout files as they are sometimes known, have the extension '.pcb'. They represent the physical layout of the interconnections between components. Generally, a standard view of the board is taken from the top side. The actual component footprints are used with associated silkscreen legends to aid placement. A copy of every style used, and every component used is saved in the file. When a component is requested, a check box gives you the option of choosing from the design, or browsing the current library. If a definition already exists in the file, you will be asked if you wish to update the existing component from the library.

Libraries

The library structure must combine the necessary information for both Schematic and PCB use tied in together with a Component. Each component normally combines at least two symbols; for the Schematic, and for the PCB. As extra package types are added, the number of PCB symbols increases correspondingly.



To support this structure, the symbols (which form the actual graphics seen in the design) are separate from the components (which are stored in a database handling labels, names etc., and specifies how the symbols should link). Every component is aware of each symbol it requires both by symbol name.

PCB Manufacturing Files for Plotting

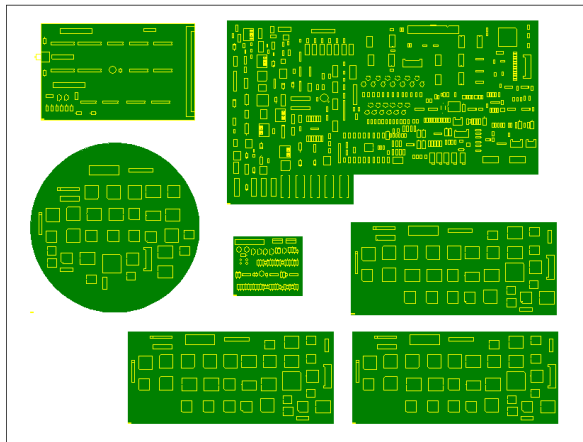
Plotting files for manufacturing are created using the **Plotting and Printing** mechanism. These consist of files exported to industry standard formats such as Gerber and Excellon but also include ODB++. Additional plotting and printing formats such as Windows and PDF are also exported so that all media requirements can be supported.

Job Files for Plotting

Job files can be created for use with the plotting mechanism. They provide you with a method of creating a set of plot options to plot your designs for manufacturing. These settings are stored in one Job file and can be loaded and applied to any design.

Panel Files for Plotting

Individual or multiple PCB designs can be added to a panel in the **Panel Editor** to create a single plot of different designs (or panelised designs of the same type). These panels can be saved to a file for reuse.



Technology Files

A technology file contains much the same information as other design files, with the exclusion of the actual design itself. It may perhaps best be viewed as a template, as standard design elements such as a Schematic drawing frame or board outline may also be included. One of the easiest ways of creating a technology file is to save a Schematic or PCB file as a technology file. The necessary information is automatically extracted.

The styles saved within the files are discussed in detail in the menu reference section under **Settings, Technology Files**. It is sufficient to note here that styles may be copied between technology files and the currently loaded design.

Chapter 4. Using The Interface

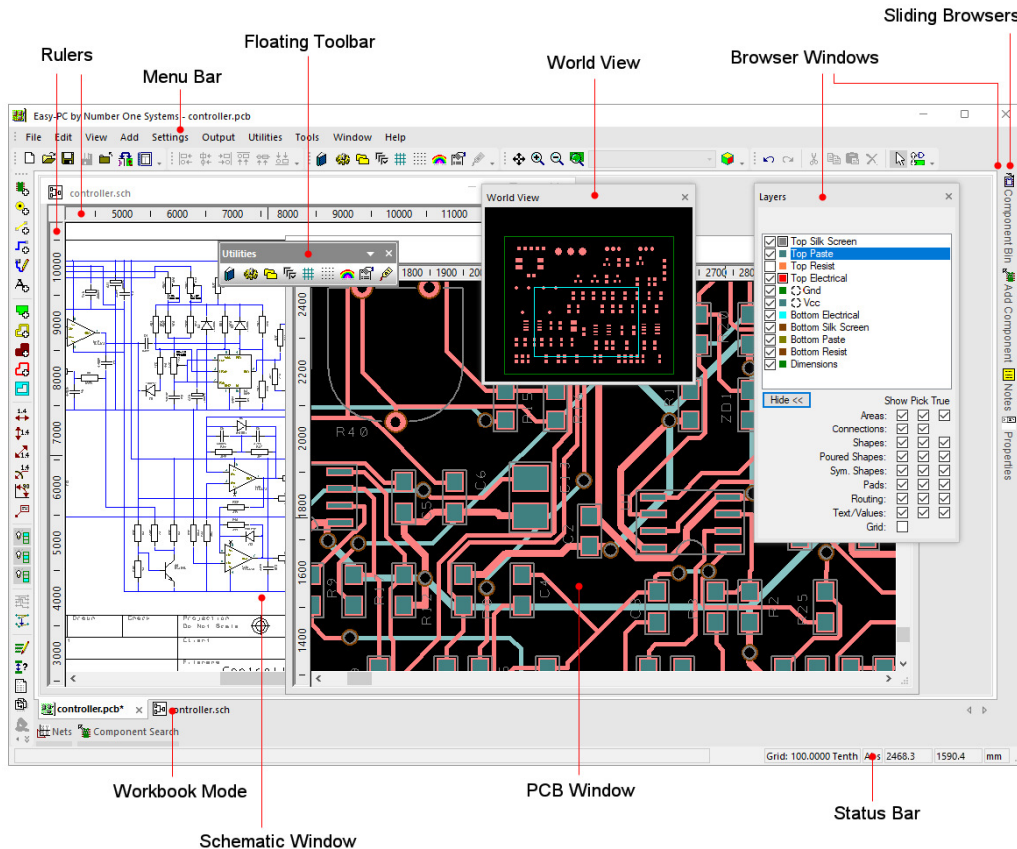
Overview

You are provided with an intuitive user interface that can be used to produce designs and multi-layer PCBs. The user interface means it is easy to learn and pick-up even when it hasn't been used for some time. Likewise, for the professional user, there are many shortcuts and context menus to assist and speed up the design process.

As with most Windows programs, the functions in this product can usually be accessed in many different ways - through the main menu, toolbars, context menus and direct shortcut keys. In this guide only one method is usually mentioned.

The User Interface

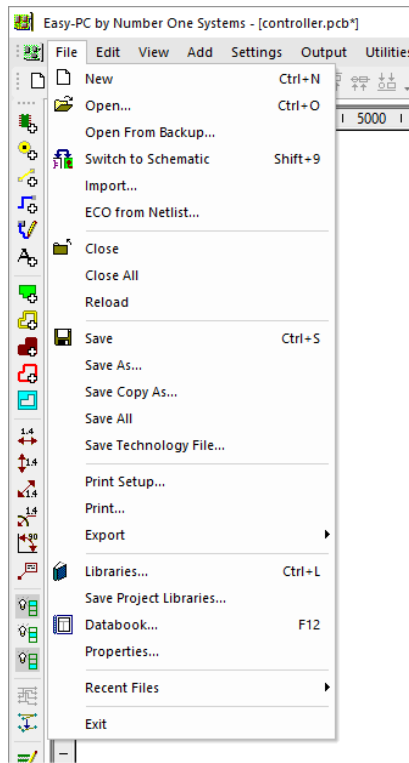
The interface uses standard Windows features and shortcuts wherever possible. This makes the learning curve for infrequent users and regular users alike as easy as possible, there is an immediate familiar feel with it. This standard will make you feel at ease with the layout and controls of this design environment within no time at all.



During its use, you are presented with terminology and commands that are as close to the Windows standard but with a PCB 'flavour'. The image above highlights some of the major framework features. Each of the features will be discussed in the following chapters.

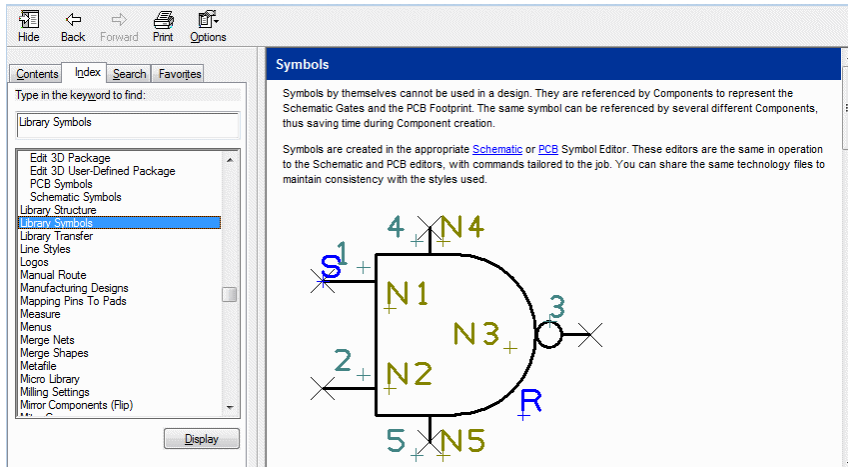
Menus

The majority of the program is accessed through menus, toolbar buttons or context menus. Most of the features have been located on the menus in the approximate order that they would be used during the design process moving from left to right across the menu bar. For example, **Open** is on the **File** menu (as it would be for any Windows product) and the **Undo** option is on the **Edit** menu.



The Online Help System

Help is supplied in the form of a HTML-based online help system. This provides detail about the options within the product.



Context sensitive help is available from all dialogs in the design by pressing the <F1> key on the keyboard, (you can redefine all the keys on the keyboard but it is recommended that you keep F1 as Help).

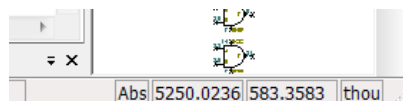
The **Index** tab has a list of all topics available from which you can select the option required. The **Search** tab allows you to locate items in the help system by typing in the word required. Word strings can be found by using double quotes, for example "Add Component".

Status Bar

The status bar at the bottom of the display gives you information about the current items selected, any prompts for input or messages and information about the cursor position.



As the cursor moves around the active drawing area (within the square border), the co-ordinate display at the bottom of the window changes. If you move the cursor from one side of the border to the opposite, you will see the co-ordinates go from 0.000 to approximately 39.000 (with Imperial units and 3 decimal points set). This represents the largest board that can be handled, 39 inches square or the metric equivalent. The status bar will also show you the **units** (**thou** in the example below). Changing between **Imperial** and **Metric** units is covered later.



The status bar shows more than just the X, Y coordinates, it provides instant information about any selected item(s). This information is the most commonly required during use, more detailed information is also available if the item is selected and the **Properties** option selected using the context menu accessed from the right mouse button.

During editing of items, the status bar is important. It continuously shows you information about the items you are editing and prompts you for action. If the operation fails, it will give you information as to why this has happened.

The status bar is also active. You can double-click on fields within it to reveal additional information. For example, double-clicking on the **Abs** field will change it to **Rel** (Relative). Likewise, double-clicking on a Line or Pad style will display the styles dialog for that item.

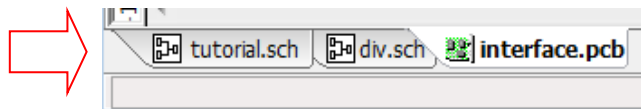
The status bar can be switched on and off from the **View** menu. You may switch it off to gain you valuable design space on your screen.

Opening Multiple Documents

You can operate in a number of different design modes and use the modes at the same time, depending on whether you are working on a Project, a PCB layout, a Schematic circuit diagram, a Technology file, or on a Component or Symbol.

Workbook Mode

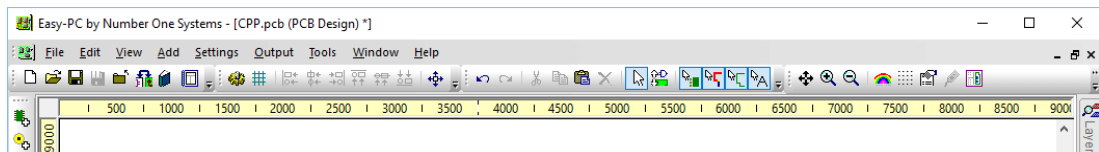
Use the workbook mode to switch between applications by simply selecting the tab required. The mode can be switched off using the **View** menu to maximise the design area.



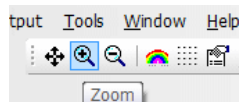
Customising the User Interface

Toolbars

The toolbar is divided into 'palettes' of commonly used tools. The toolbars can be moved and customised as you require. The example below shows the default set of toolbars in the docked position.



As you move your cursor over each button, if you pause momentarily, a small label will appear. This is called a **tooltip** and will provide you with the function of each button when selected.



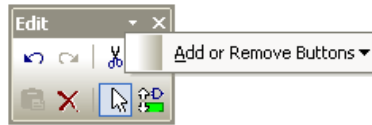
Each of the toolbars can be individually switched on or off and can be docked against the furniture or left floating in 'space'.

In **docked** mode, a toolbar is fixed along any of the four borders of the main window framework in either a horizontal or vertical position and will have a **Gripper** bar across the top or down the left. You cannot modify the size of a toolbar when it is docked.



The toolbar can be moved by selecting the window border and 'dragging' it away from its current location (by dragging the small dots on the left side of the toolbar). When dragging, it becomes detached from the framework. When the toolbar is then moved over another area of the framework to which it can be attached, it changes shape to indicate it can be docked there. You can also make it floating by double-clicking on the window gripper bar or in the toolbar area itself, but avoiding the buttons that will select an option.

The toolbars can also be **floating**. This is a toolbar that has been moved into free space on the design and outside the framework. This toolbar can now be reshaped to make it fatter or thinner by dragging the outer edge in or out. The toolbar can be shaped to be one tool wide or long by as many tools as are loaded on the toolbar in the opposite direction. By selecting the outer framework edges of the toolbar, a small arrow 'handle' will appear to indicate that the toolbar can now be reshaped. Drag the handle until you have the shape you require.



You can restore the toolbar to its last docked position by double-clicking on the name caption on the toolbar, in the example above, this would be the word *Edit*.

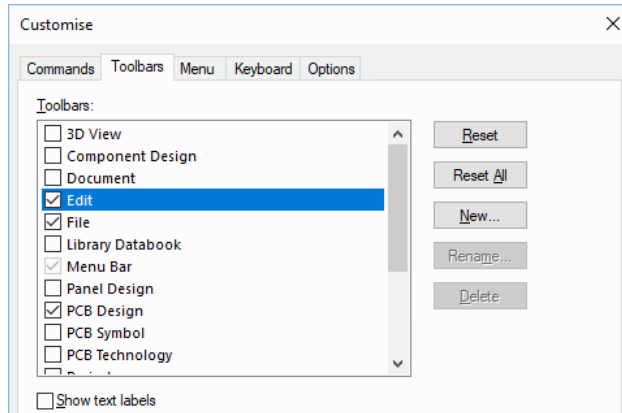
To move a floating toolbar, select and drag any part of the toolbar framework. The toolbar will move dynamically with the mouse movement. Release the mouse button to release the toolbar.

Customising Toolbars

Toolbars can be customised to contain the buttons (and underlying options) of your choice. You are provided with a default set of toolbars that cover different design types and some designs tasks, such as editing etc.

You can create toolbars for any task that appears on the menu or context menus.

To edit or create your own toolbars you must use the **Customise** option on the **Settings** menu. Once in this dialog you can move tools on existing toolbars or add/create your own. Other than in this dialog, you cannot add or remove tools from toolbars. Buttons can be moved by pressing the <Alt> key and dragging the button from one toolbar to another. This technique can also be used to remove the button from the toolbar. Press the <Alt> key and drag the button into free space away from a toolbar.



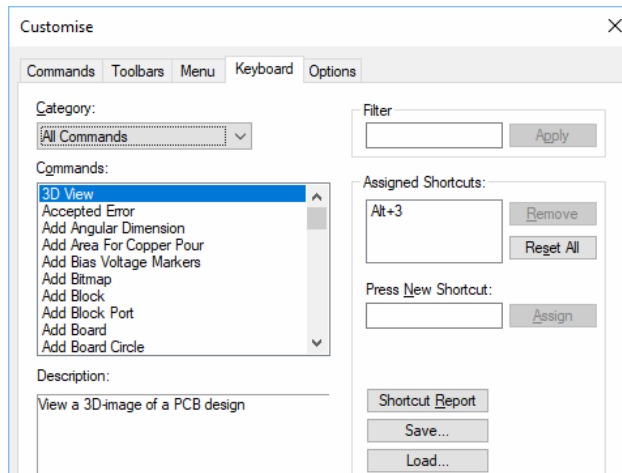
Commands and Shortcut Keys

As well as the toolbars, you have the ability to program your keyboard for commonly used options or commands. This means that once programmed, you can press a key or key combination to enter a function. For experienced users, this makes the program's operation far more efficient (and quicker).

When the program is first installed, you will find a set of shortcuts already defined for you. This is only a default set and can be modified to your own preference through the **Customise** dialog. Almost all of the keys on a standard keyboard can be utilised giving you extensive coverage of shortcuts.

► To define shortcuts

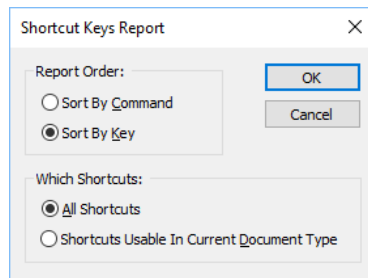
1. From the **Settings** menu, select **Customise**
2. Select the **Keyboard** tab.



3. Each of the currently assigned shortcuts are displayed when the command of the same name is selected. You may assign more than one shortcut to a command if required.

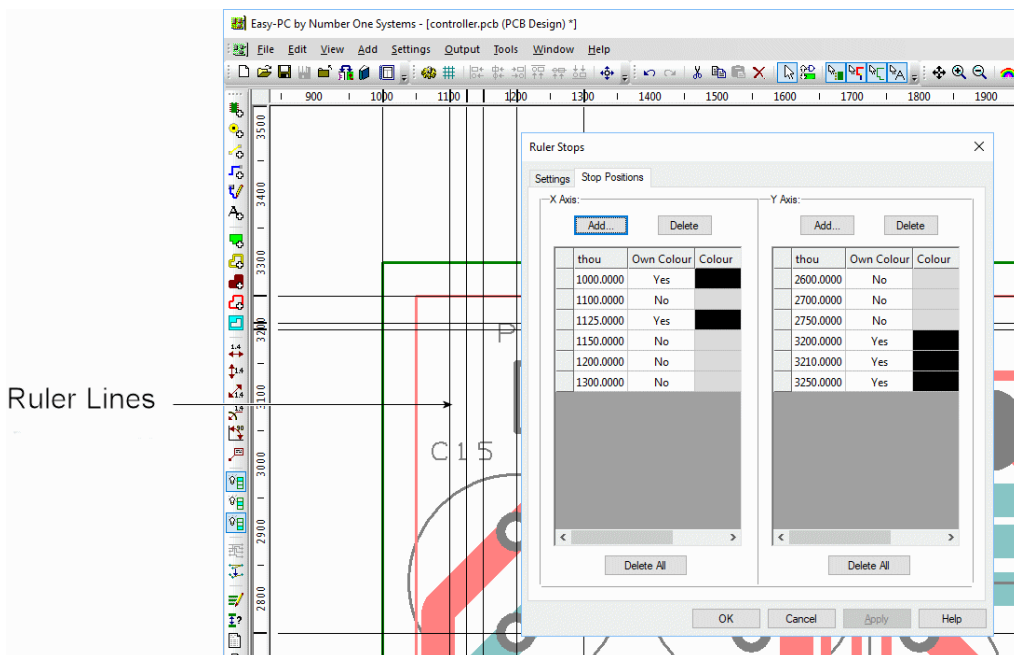
4. Select the Command that you wish to create a shortcut key for. The Category selection will help you choose the command. These are categorised by menu option with an <All Commands> option also available.
5. Click the shortcut key(s) that you wish to use.
6. The dialog will display any command that already uses this by name and the shortcut will be displayed in the **Press New Shortcut Key** box.
7. Click the **Assign** button to assign it.
8. Click **Close** to exit this dialog.

For a report of all shortcut keys defined, run the **Shortcut Keys Report** from the **Settings** menu, **Customise** and **Shortcut Keys Report**. The dialog will allow you to choose the sorting order and the shortcuts to report.



Rulers

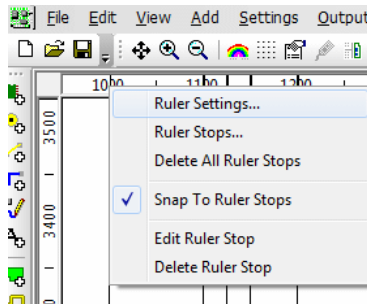
You can choose to have vertical and horizontal rulers displayed along the left and top margins of the design window. Rulers can be useful for sizing or aligning design items in terms of physical units such as millimetres, inches or thou.



Rule Stop settings

The ruler lines and snapping to the ruler lines is controlled through the **Ruler Stops Settings** dialog. To display the Ruler settings dialog, select **Rulers** from the **View** menu, or right click on a ruler bar to use **Ruler Settings** from the shortcut menu, or alternatively, you can use a defined shortcut key or toolbar button.

You can right click on the ruler bars and use a context menu to access the dialogs and also to perform some other operations on ruler stops.



- Use **Delete All Ruler Stops** to quickly remove all the stop positions from both axis.
- Use **Snap To Ruler Stops** to toggle interactive snapping of design items to stop positions.
- Use **Add Ruler Stops at Selection Origin** to add a horizontal and vertical stop position at the origin of the selected items. This command is available to put on a shortcut key, and is useful for displaying a line through particular items to use to line up other items.

If a ruler stop existed at the point that you right clicked on the ruler bar and one or more items of the same type are selected in the design, two **Align To Ruler Stop** options will be available. These allow you to **align** either the lower or upper side of each selected item to the picked ruler stop position.

If a ruler stop existed at the point that you right clicked on the ruler bar the **Edit Ruler Stop** and **Delete Ruler Stop** will be available to allow you to change the value of, or delete, the picked ruler stop. If no ruler stop was underneath the picked position, the **Add Ruler Stop** option is available. Use this to add a new stop at the position picked on the bar.

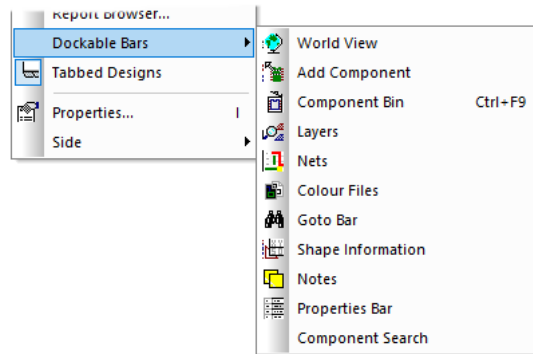
A new ruler stop can also be added by double-clicking on the ruler bar.

You can reposition a ruler stop by dragging it along the ruler by pressing and holding the left mouse button, release the button at the new location.

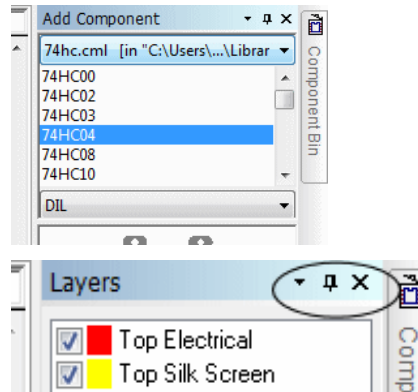
Dockable Windows

There are a number of dockable windows available for use. These will speed up your design work and provide you with convenient access to features: **Component Bin**, **Add Component bar**, **Layers bar**, **Colour Files bar**, **World View** and **Goto bar**. Each of these may be displayed, hidden, docked, floated, auto-hidden (so they can 'slide') or combined together in a tabbed bar.

To view any of these menus, either select **Dockable Bars >** from the **View** menu or right click on the menu bar or the framework to access the shortcut menu.



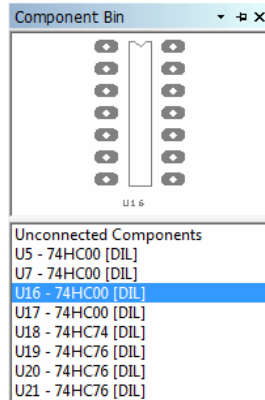
When docked, these bars can automatically hide when they go out of context and re-display when they come back into context. They come into context by hovering the mouse over the relevant option tab.



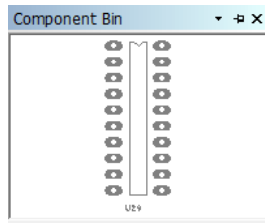
The three small icons (shown circled) on the top of the sliding bar indicate additional functionality. Moving left to right; these reveal the context menu, a pin to enable the dialog to be sliding to conceal it or docked to be permanently displayed. A cross indicates the Close option to dismiss the dialog altogether.

Component Bin

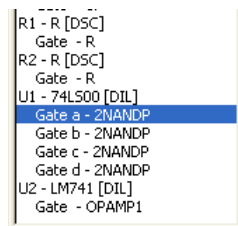
The **Component Bin** itself consists of the **Component Bin Preview** (showing both Schematic and PCB symbols) and the **Component Bin**. The component bin is provided to store and view unplaced components in the PCB design editor, and unused gates in the Schematic design editor.



The **Component Bin Preview** window shows the component currently selected in the bin, along with its pad numbers and names. This updates automatically as each component within the bin is selected.



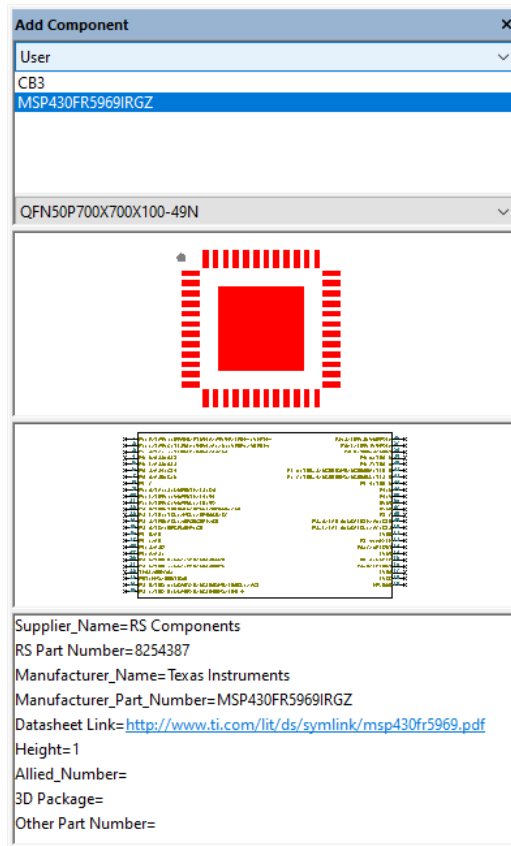
The **Component Bin** window lists the components currently in the bin by Component Name. This shows the component name and component type and is sorted alphabetically on the component name. For a Schematic design and for multi-gated components, the gate name and symbol name are listed for each gate of the component. The components (or gates) are selected by clicking in this list.



To use the component bin, select and drag the component required into the design. You will see the component on the end of your cursor in the design ready for placement.

Add Component Bar

The **Add Component** dockable bar is effectively an 'always-accessible' modeless dialog for adding components. This is available for a Schematic or PCB design.

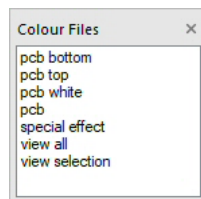


It allows a component to be 'dragged' from the selected library directly into the current design view. Alternatively, double-clicking activates the normal 'button-up' add component mode and allows multiple instances of the component to be added. The component package name is also displayed allowing an alternative package to be chosen.

This window is more basic than the **Add Component** dialog as there is no name filtering or Find option. However, because this uses the sliding bar, then it is accessible very quickly.

Colour Files Bar

The **Colour Files** bar will display a list of previously saved colour files in the colour files folder. The folder is predefined and named in the **Preferences** dialog on the **Settings** menu. Clicking on any of the files by name from the list will load that file and change the current display.

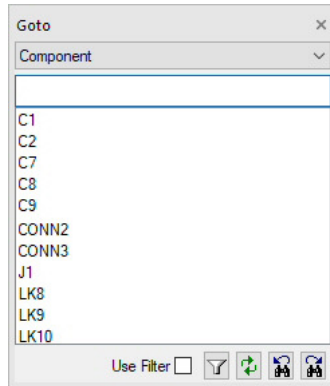


A basic selection of colours files is installed by default but the list can be added to and modified to your own requirements. Colour files are defined and saved from the **Colours** dialog.

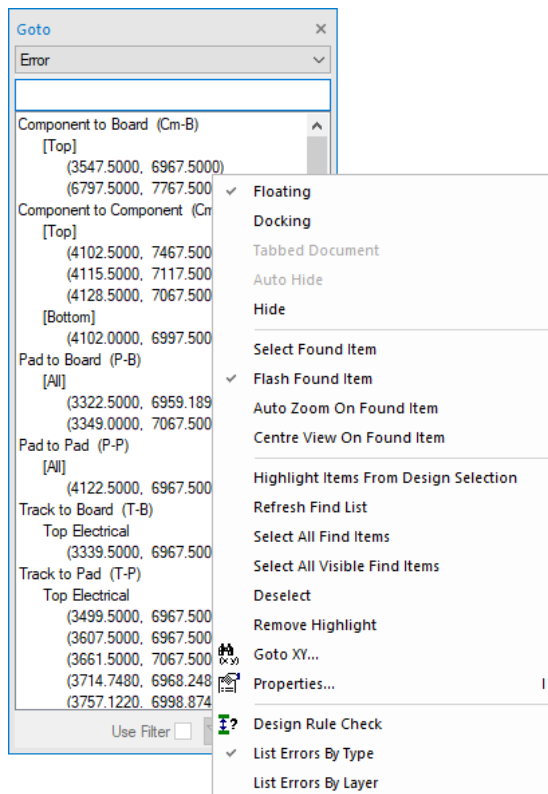
Goto Bar

Different design items can be located by using the **Goto Bar**. You can go to **Components** and **Nets** in both the Schematic and PCB design editors, as well as **Errors** in the PCB design

editor. Additional **Goto** options include Goto **Pad/Line/Track, Text** and **Text Styles**. The Goto **X,Y, Relative Origin, Selection Origin, System Origin** and **Previous/Next Error** options are available from the **View** menu.

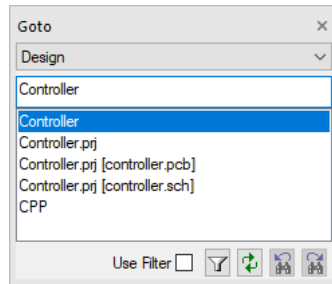


The **Goto Error** selection will show all the design rule errors in the design once the online DRC or batch Design Rules Checking options have been run. Clicking on any of the errors will 'pan' the display to the selected error. Errors can be sorted by **Type** of error and **Layer** of the error using the context menu in the window.



Design – Lists the names of all open designs, projects and library items. Also lists all schematic and PCB designs in a project even if they are not open.

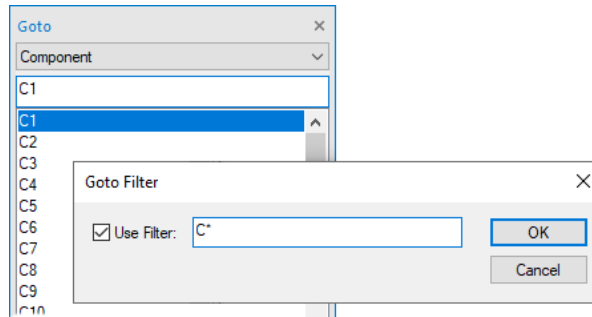
Select an entry from the list to switch to that design.



Goto Filter

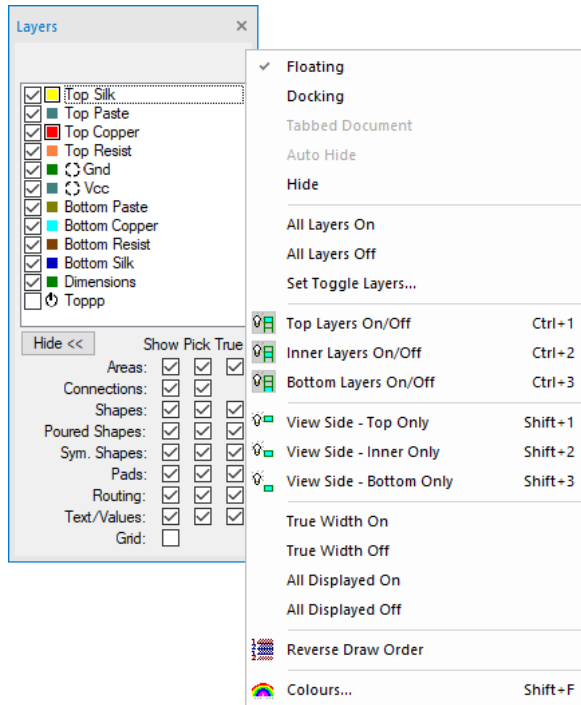
You can filter found items in the list by using the filter on the **Goto** dialog.

Check the **Use Filter** box to activate the filter. A dialog will be displayed into which you can type the name you wish to find. You can use standard wildcards (* and ?) in this field.



Layers Bar

The **Layers Bar** is used for switching named layers on and off. This is only available from within the PCB design editor and PCB symbol editor. This will list all the layers in the design (created in the **Layers** dialog on the **Settings** menu).



Each layer is shown with a check box. Clicking the check box toggles that layer's visibility on or off. A single click on the layer name highlights that layer in the list, whereas double-click will make just that layer visible and all other layers invisible. There are also context menu entries available to switch **All Layers On** and **All Layers Off**.

Defining Current Electrical and Documentation Layers

The colour squares next to the layer name have a black square outline to denote both the default electrical and documentation layers. Left click on the square to set that layer as the default. The black ring will appear next to that layer to indicate this. If there is no common default (i.e. all categories do not have their default set to the same layer) the black square is not drawn.



When starting a new track or adding a new doc shape, the default layer will be used.

Showing and Hiding Powerplane Layers

On the Layers Bar, power plane layers are indicated by a small thermal pad icon to the right of the coloured square. Powerplane layers are defined in the Technology file under Layers. These are layers that have a Layer **Bias** as **Power Plane**.



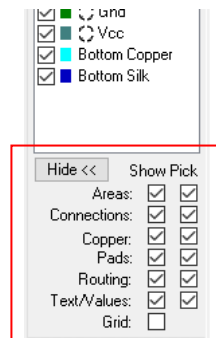
If the power plane is being displayed in the design the icon is in negative. Left click on the icon to toggle the display to show/hide the power plane.



Note, this option is a quick way to access the **View Powerplane** functionality. View Powerplane is still available from the **View** menu.

Categories

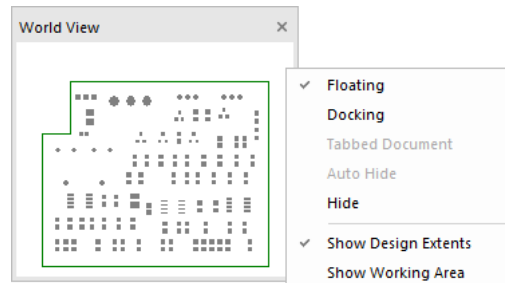
Within the Layers bar, you also have the details pane showing additional design categories. This is revealed using the **More>>** button. It can be closed using the **<<Hide** button. This allows you to **Show** items in the design as well as making them **Pickable** or not.



World View

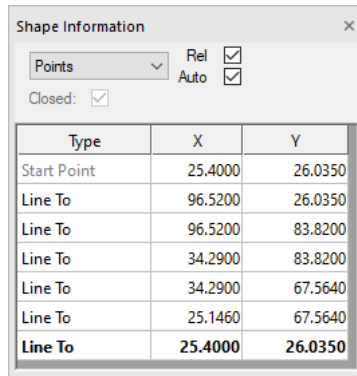
The **World View** provides you with a view of the whole design and working area.

For fast zoom and unzoom, the frame option can be used inside the World View window. To use this simply click and drag in the World View (like a Frame Select). The area drawn will define the zoom region. The design will automatically redraw at this size.



Shape Information Bar

The Shape Information Bar is available as a dockable bar to allow viewing and editing of the currently selected shape in your design. Shape coordinates, offsets and lengths can be edited for easy shape creation.



If the selected shape is edited interactively, the Shape bar will dynamically update to reflect the changing co-ordinate information for the shape being modified.

The **Rel** check box allows you to display the coordinates in either **Absolute** or **Relative** units.

When the **Auto** option is checked, this will cause the Shape bar to automatically switch to the most appropriate mode to display a selected shape, i.e. if a circle is selected it will be displayed in **Circle** mode. Polygon shapes will be shown in the most recently used of the possible modes.

The **Closed** option will be automatically set to indicate whether the selected shape is Closed (unchecked means the shape is Open). The check box allows the state of the shape to be toggled between the two states.

More information about the *Shape Information Bar* is available in the online help.

Design Notes Bar

PCB and Schematic designs can contain **Notes** organised within subject categories. Use notes to jot down questions, ideas, reminders, and anything you would write on paper. For example, to document the designs progress, highlight problem areas, document any design constraints, serve as reminders of how to do things, highlight where a workaround has been applied, and many more reasons.

Notes can be synchronised between a schematic and its corresponding PCB design and so can be used to pass instructions from the schematic engineer about things to consider when laying out the PCB. Also notes about problems can be back annotated to the schematic from the PCB design.

Diam (thou)	Qty	Id	Plated	Symbol
30.0	4	E	Y	●
35.0	182	B	Y	★
42.0	9	C	Y	+
50.0	3	D	Y	■

Unlike free text and callouts, notes cannot be displayed in the design, but can have associated markers placed in the design to show their area of interest.

There is a dockable **Notes Bar** containing a browser listing all the notes in the current design, sorted and filtered in various ways, and an edit window that can be used to view, edit and print the contents of each note.

What is a Note

A note contains a single line subject or title, and a multi-line text message. Each character in the text message can have its own colour, size and text font. The text message can be created in a text editor window by directly typing the text, or by pasting unformatted or Rich Text Format (RTF) text from another application. Each note is saved in the design it was created in.

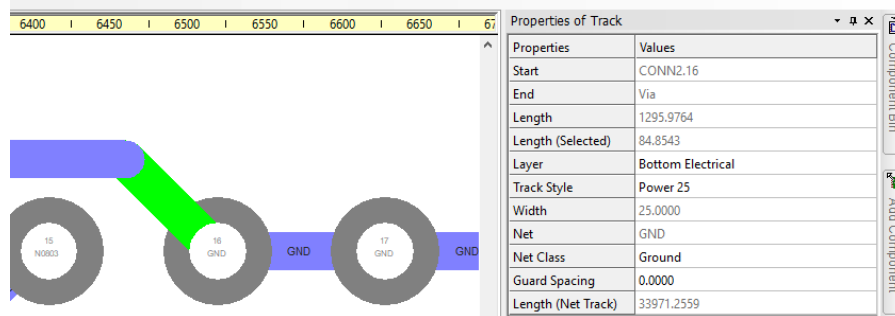
All notes belong to named note categories to help you organise and manage them. As well as a name, these categories contain information about the note markers. Each note can optionally have an associated marker placed in the design. By default, the marker appears as a yellow square. You can change the appearance of note markers, including the symbol, colour and size, by editing the note category.

Additional help on adding and using Design Notes is available in the online Help.

Properties Bar

The dockable **Properties bar** allows you to see the properties of an item or multiple items using a modeless window. Many of the features available to you from within the **Properties** dialog are available in this dockable option, helping you to view and edit values for items without accessing a different dialog. Note that not all the properties are available in the Properties bar. It is purely a faster, more convenient mechanism for accessing Properties on the fly.

The dockable **Properties** bar also provides the ability to show only the properties common to all the items in the selection.



To enable the **Properties Bar**, click on the **View** menu, **Dockable Bars**, and click **Properties Bar**. Alternatively, right click on the right-hand bar and select **Properties** as shown in the illustration to the right.

Because this is a floating bar, it can be undocked once selected for use.

Viewing the Design

Mouse Operation

The mouse can be used for many operations, not only for menu and option selection but also for interaction within the design. A single left button click can be for selection. Once selected, an additional operation can be performed on that item. By selecting an item (with the left button) and then right clicking, you can get access to the context menus. You can actually just use a right click on an item to get to the context menu, this has the same effect as left click then right click.

Functionality is also available through the use of the mouse wheel button. The wheel can be used in two ways, by rolling it backwards (and forwards), and by clicking it.

By rolling it forwards you can activate a **zoom out** on the design, the same as using the <U> key on the keyboard.

By rolling it backwards you can activate a **zoom in** on the design, the same as using the <Z> key on the keyboard.

By using a single click of the wheel, you can pan to the current cursor position. If you keep the wheel mouse pressed down and moving the cursor, a dynamic pan will be obtained. This has the same effect as using the <P> key on the keyboard.

All the Zoom commands are available on the **View** menu although you will find that you will use the mouse wheel or remember the keyboard commands more frequently.

As an alternative, you also have commands for **Zoom In/Out At Centre**. Instead of zooming in and recentring the selected zoomed item, the window will remain in its current position and zoom the item out.

Numeric Keys to Zoom In and Out

The numeric keys <1> to <9> and <0> along the top of the keyboard will give you predefined zoom levels. These keys change the zoom level immediately to correspond with the key pressed. For example, at zoom level 8, you will see the entire drawing area; at zoom level 1 the screen corresponds to an area of the drawing about half an inch (~15mm) across but will depend on another factor, the size of the defined working area.

Frame View

A **Frame** option is also provided to selectively zoom around the design into specific areas, the shortcut key for this is <Ctrl-F>. Click and drag a box around items within the design, it will then fill the screen with the selection.

A note to mention here is that quite often you will drag a box on the design and get a selection but no zoom, this will be because you have not selected the Frame function first.

Frame can also be used in the **World View** by simply drawing a box in the World View window. You don't need to be in Frame 'mode' to do this.

Viewing all of the design

At any time, you can press the <A> key to zoom to show all of the design. The 'design' means anything on the screen within the design area, e.g. board outline and/or drawing blank (if used), or the extremities of any entity regardless of where it is in the design area.

The **View All** option is also available on the **View** toolbar and the **View** menu.



Pan

You can use the zoom options to move around the design by successively zooming out and then zooming in on different areas. A **Pan** option is also provided to navigate your way around a design without having to zoom in and out. This can be done using the <P> key on the keyboard (this assumes you haven't yet switched on the **Autopan** feature which is discussed below). You will see that the screen pans to the previous position of your cursor.

By keeping the <P> key pressed down and moving the cursor, you can achieve a dynamic pan on the design.

By pressing the middle mouse wheel button on the mouse, you can also do a Pan. Keeping the wheel mouse pressed will enable you to move the mouse and dynamically pan around the design.

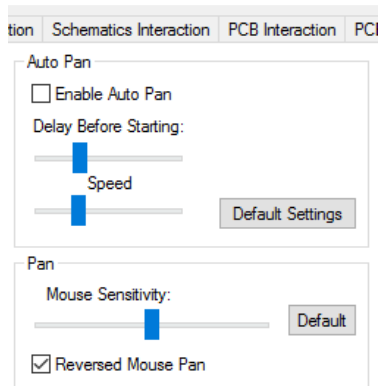
Pan can also be used in the **World View**. With the cursor in the World View press the <P> key on the keyboard, the design will pan in the direction at each key press.

As well as the Pan option, you can also use the **scroll bars** on the window to move from side to side or top to bottom.

Auto Pan

A smooth **Auto Pan** option is also provided. This operates when you move the cursor to within approx. 1/4 inch (0.7 cm) of the current design window edge (this value is approximate for a full screen design on a 15" monitor). The design pans until the cursor is moved outside the panning tolerance. Panning works with and without design items selected ensuring an optimum user defined view of the design at all times.

The switch enabling the **Auto Pan** control is selected through the **Preferences** dialog, **Display** tab on the **Settings** menu.



The delay before the pan is started can be also controlled on this dialog using the **Delay Before Starting** slider. Moving the slider to the right makes the autopan work sooner, whereas moving it to the left makes it less sensitive that means you have to leave it at the edge longer before it pans.

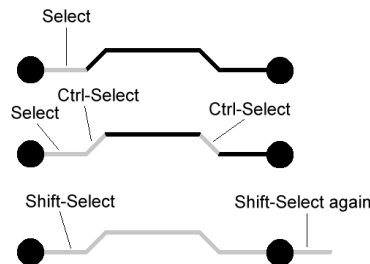
The **Default Delay** button setting will apply a built-in setting that will be a compromise between being usable and speed of operation.

Design Item Selection

Standard Selection Operations

There are a number of ways that items can be selected in your design and libraries.

- Single select
- <Ctrl> select (modify selection)
- <Shift> select (extended selection)
- <Ctrl-Shift> select



The diagram above shows how using the different combinations will allow you to select the entities of the design item.

Single select

A **single click** of the left mouse button will make a single selection of any item. By selecting another item, the first item will be deselected. Once an item is selected, you can add to that selection or remove the item from the selection (see the <Ctrl> and <Shift> selection sections below).

Control (Ctrl) select

Using the <Ctrl> key while clicking the mouse button will enable you to modify the selection. In other words, if you already have an item selected, you can add another item to the selection. It also allows you to remove items from a selection. This is a very useful option for making a selection of multiple items in the design.

Shift select

Using the <**Shift**> key while clicking the mouse button will extend the selection to the whole item. This could be used for selecting a complete track path between two pads on the same net for example. If the Shift-Select operation was executed twice on the same track, it would select and extend the selection to the first track path, then extend it further to include the whole net outside of the point-to-point connection path.

You can also use <**Shift-Select**> to select a whole shape, a copper shape for example.

Ctrl Shift select

By using the <**Ctrl-Shift-Select**> keys together you can extend the selection to the whole entity but then use this combination again to add to the selection. For example, for selecting a whole track path but then adding another whole track path to the selection of a different net.

Other Selection Mechanisms

There are also other modes of operation that can be accessed using the mouse: **Double-click** editing and **Frame select**.

Double-click editing

By using a **Double-click** of the mouse on an item, such as a connection in PCB, you will automatically be entered into **Edit** mode. This means that the connection is converted to a copper track and you can 'draw' in its electrical path using corners and changing layers.

Frame Select

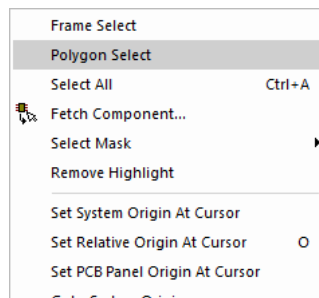
By dragging a box when in free space, if any items are enclosed within that area, they will become selected. This is called a **Frame-select**. Any item inside this selection will be drawn in the **Selection:** colour taken from the **Colours** dialog.

This type of selection is of use for making a selection of a group of items that could then be moved for example. You may also use this selection to **delete** sections of the design.

To deselect the selection, click the mouse in free space on the design or right click and choose **Deselect** from the context menu.

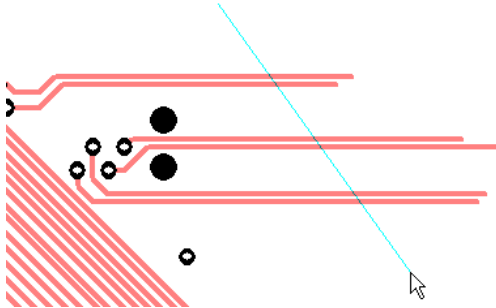
Polygon Select

From the context menu, you can choose the **Polygon Select** mode. In this mode you define a polygon by clicking at its corners. Use **backspace** to delete a corner and double-click to finish. Any item crossing the polygon will be selected.

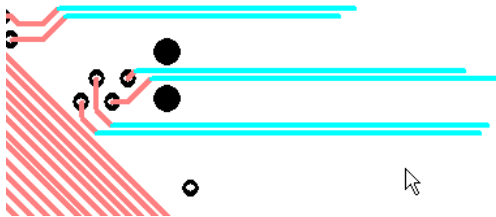


You can also use this mode as a single line for selection by dragging the cursor. This is useful for selecting angled tracks for example. This option ignores the **Select If Completely Framed** setting.

To do this, enter the **Polygon Select** mode, drag a line (not a polygon) across items you wish to select:



The items under the line are then selected once the cursor is released.



Like **Frame Select**, **Shift key down** means select whole items, and if the **Ctrl key** is down nothing is deselected before selecting the items with the shape.

Controlling the Selection

Cyclic Picking

Having selected a design item, it is possible to select the next nearest item within picking distance. This may be done either by using the **Select Next** option on the context menu, or by pressing the shortcut key <N>. Repeating **Select Next** will cycle through all items that are within close proximity. Once the required item is selected you can then execute another command on it.

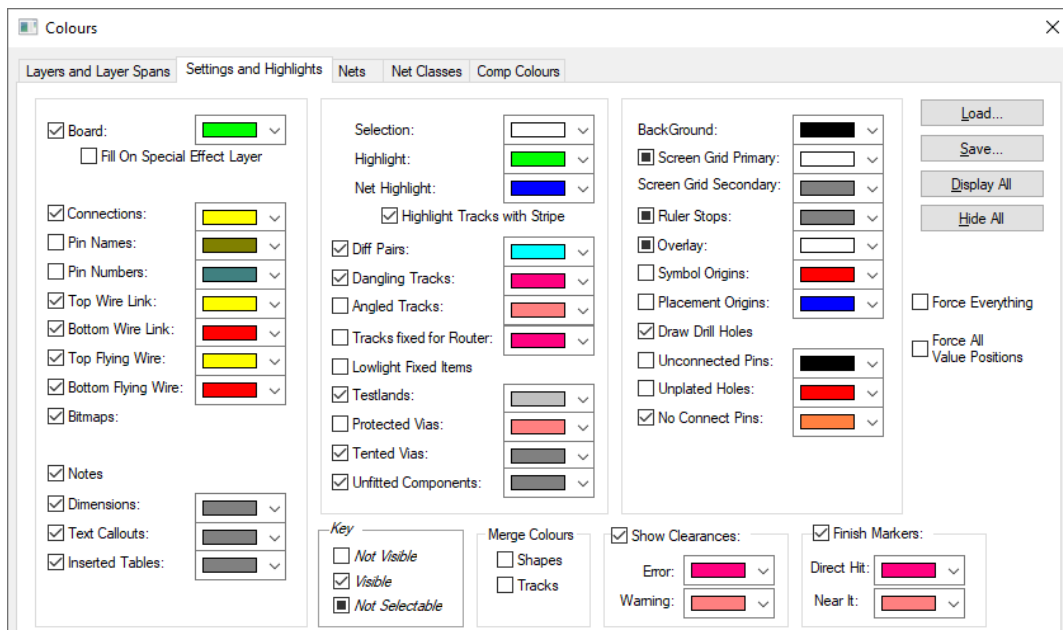
This is especially useful for selecting design items in a dense area. You can also use this effectively when the display of particular items is set to be non-selectable.

Colour selection flags

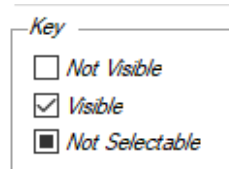
As well as the ability to select **Next** and **Previous** using the cyclic selection mechanism you can also make design items selectable and non-selectable from within the **Colours** dialog.

Layer	Display	Selectable	True W	Tracks	Areas	Pads	Shapes	Text	Sym. Pads	Sym. Shapes	Sym. Text	Value P
Displayed				Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Y
Selectable				Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Y
True Width				Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Y
[All]	Yes	Yes	Yes									
[Top]	Yes	Yes	Yes									
Top Electrical	Yes	Yes	Yes									
Top Silk Screen	Yes	Yes	Yes									

Within the **Layered Items** dialog, you can make an item not selectable by unchecking the **Selectable** check box. With this box unchecked, if you try to select that type of item, it will not be selected. Likewise, if you make a frame selection and the selection contains this item, again it will not be selected.

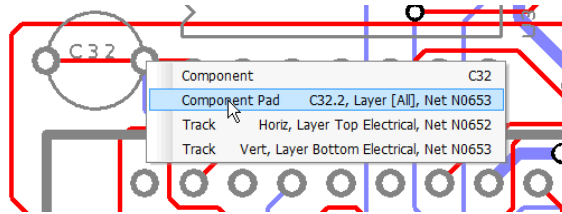


On the **Settings & Highlights** dialog you can also make non-layered items non-selectable by using the tri-state check boxes. The tri-states are unchecked (Not Visible but Selectable), checked (Visible and Selectable) and checked/greyed (Visible but not Selectable), these are shown on this dialog under **Key**.



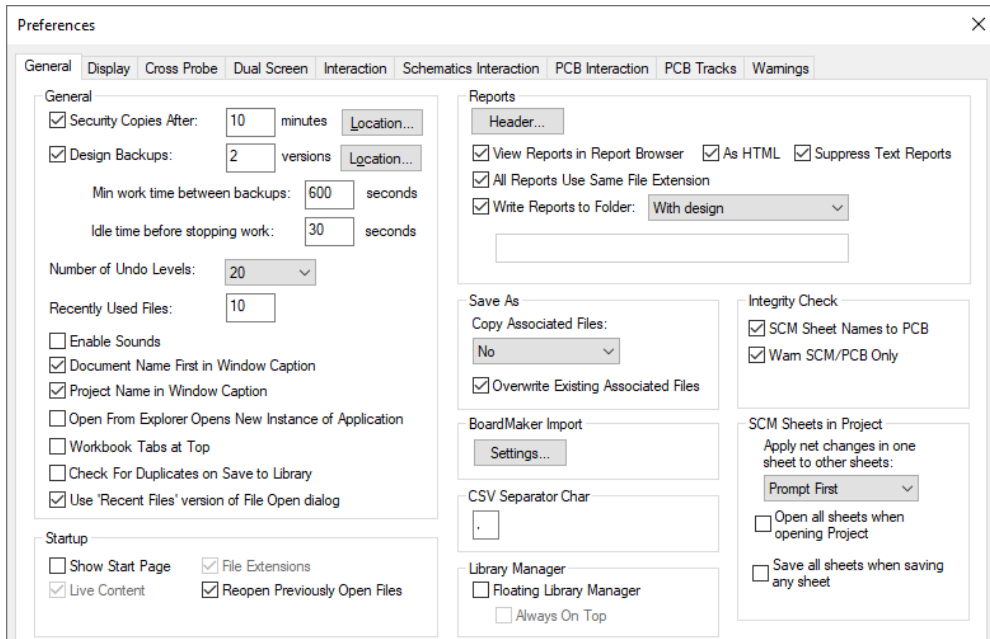
Selecting from a list

As an alternative to using the <N> key to cycle through the picking list, if you press the <Alt> key while selecting the required item, anything within the picking tolerance will be displayed on a small context menu. From this list pick the required item and it will be selected.



User Defined Preferences

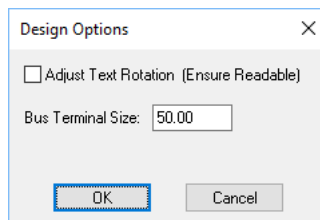
There are many aspects that can be tailored to suit your own needs. These options are defined in the **Preferences** dialog on the **Settings** menu.



Each tab on the dialog contains logical groups of functionality for different purposes. The **General** tab (shown above) is used to set general system information for example.

More complete detail for each of the options on these tabs is discussed during their use throughout this manual.

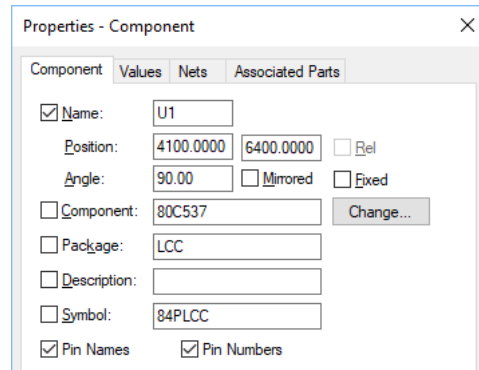
There is also a **Design Options** option on the **Settings** menu.



This has one option on it for setting an automatic text adjustment during text rotation so that it is readable from the bottom or right-hand side of a page.

Properties Dialog

Additional information can be obtained for any item using the item **Properties** option. You can get to the item Properties by selecting the item and using the shortcut key <Alt-Enter>, or selecting it from the context menu.

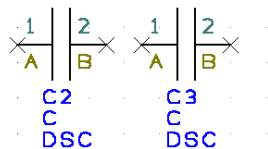


The **Properties** dialog shown above is an example for a **Component** in the Schematic design editor, it shows more detailed information about its **Position**, **Angle** and **Component** etc.

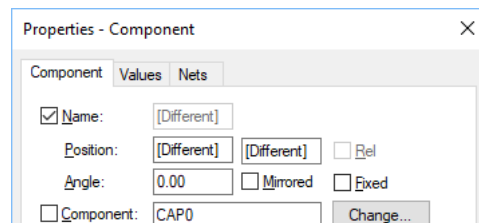
Much of the information within **Properties** can also be edited, a components' physical **Position** or **Angle** for example.

Multi-item Properties

The **Properties** dialog can be used for changing multiple selected items, components for example. When two or more similar items are selected and the **Properties** option entered, various fields will show that there are differences.



Using the above design example, the component **Properties** dialog looks like this:



If the **Component Position** in the X axis is different to the Y axis, this will be indicated in the dialog by the word **[Different]** in the coordinate field. Obviously, the component **Name** will also be different and is greyed out.

From here you could also apply the same **Rotation (Angle)** to all the selected components or **Mirror** them for example.

If you are currently using **Relative Coordinates** in the design, you can use the **Rel** button to switch between showing (and accepting) relative coordinates or absolute co-ordinates.

Values

From the **Properties** dialog you can also add **Values** to design items.



When a value is added, the name of the value and actual value is shown, in the example above, **C** is the value name and **100nF** is its *value*. You can display the value on the item in the design by clicking the check box next to the value.

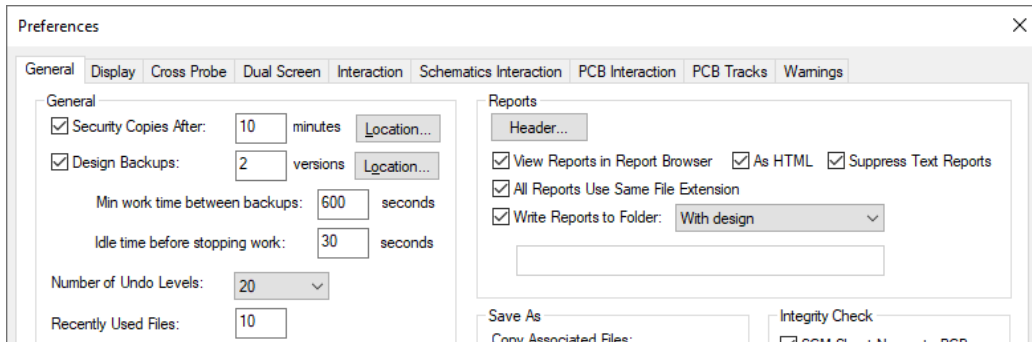
To add values to a component, use the **Value Positions** option from the **Add** menu. For more information, see the next chapter under *Working With Values*.

The **Key** shows the colour code for the entries in the list. So, a value just defined on the selected component instance is drawn in black, and a value inherited from the library component is drawn in light blue.

Chapter 5. General Utilities

Undo and Redo

Multi-level undo and redo facilities are provided for all major design items and Projects to a maximum of 50 levels. System memory (RAM and Virtual Memory) will have an impact on the larger undo levels available. The memory used for undo depends on the items and number of items preserved by the system.



The number of undo levels are set on the **Preferences** dialog on the **Settings** menu. Standard Windows functions for **Undo** and **Redo** are used as shortcuts, **<Ctrl-Z>** and **<Ctrl-Y>**, respectively, as well as the inclusion of menu options on the **Edit** menu for each item.

The use of undo and redo in design practice are invaluable. They allow you the flexibility of being able to try different iterations of design and restore the previous attempts if the effect isn't what you require, or if you, by any remote chance, make a mistake, undo allows you to recover the situation without incidence.

One last point about undo and redo, if you set your undo levels to 3, for instance, when level 4 is required by the system it will remove and overwrite level 1 and so-forth overwriting each previous level in turn and continually cycling through this sequence. If you rely on undo (as most of us do) set the levels to 50 and forget it! Check your memory and adjust to a level suitable for your system.

*Be aware though, once **Save** or **Save As** have been used, the undo buffer is reset to zero (0) levels available. This means after saving the design you cannot undo an iteration made before the save. From this point forward the undo buffer would start saving again.*

Do not rely on Undo or Redo as a substitute for regularly saving your work.

Interactive Operations

During the normal usage of the product you will need to make interactive operations to design items in order to manipulate them to where you require. The following sections outline the major interactive features of the product and explain their use.

Moving and Placing Items

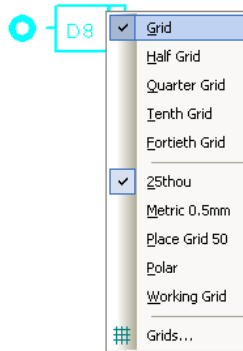
There are two interactive methods of moving items in your designs (don't forget you can also use **Properties** and **Position** to type in a location).

The first is using the standard Windows **drag and drop**, where an item is selected and with the mouse button still pressed, is dragged into its location and dropped. This is very fast and is modeless, it does not require that you are in any 'mode' of operation to perform this. When using drag, the amount of item that is moved will depend on how much of it is selected. A track segment for example can be 'stretched' until released. Where the stretching takes place from depends on how many segments were selected (you may have used the **Ctrl** or **Shift** keys to extend the selection). If a whole shape is selected, the shape will be moved rather than stretched. Selection of any part of a component (other than its name or values) will move that item. A component is effectively a small group created by the symbol or footprint library item and will act as a complete 'unit'.

The second method is by selecting the item and using the **Place** option from the context menu.

During both drag and Place, options available to you through the shortcut keys and context menus are the same. While moving an item, if you decide that it was the wrong item or the move is no longer required, press the **<Esc>** key to exit the mode.

When placing items, the grid used will be determined by the interactive grid setting. During move, use the shortcut key **<H>** to display the menu. Select the grid setting required.



The grids are taken as fractions of the current grid selected; this is set up in the **Grids** dialog on the **Settings** menu. You may create your own named grids, these will be displayed on the Grid Snap context menu when using Place. Grids are discussed later on in this chapter under the heading *Grids*.

Line segments that are mitred or fillets can also be moved but will retain their size and make the adjoining segments each side dynamic. If you wish to resize a mitre or fillet, use the edit facility (using double-click) and drag the corner to a new position.

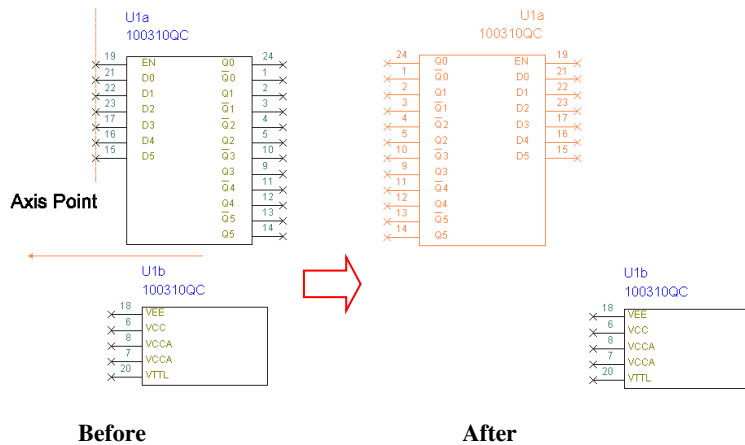
On straight line segments, you can pick the end of the line and drag it to a new position. The pick and drag must be continuous and must be accurate on the end of the line. If it isn't, you will move the whole line.

Mirror (Flip)

Mirroring of items is done using the **Flip** command. This can be performed on most design items. The command is available when placing or editing an item, or from the context menu. It can also be used by selecting the shortcut key <F>, and is available on some **Properties** dialogs, **Components** for example. It is not available for all design items.

Flip in Schematics

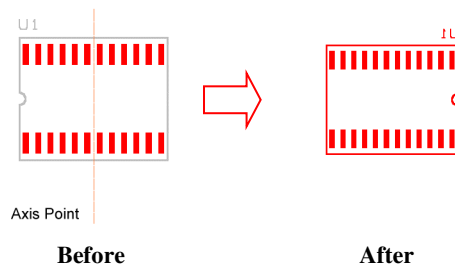
The **Flip** option on components in Schematics works by mirroring the symbol about its symbol origin.



In the above illustration, you can see that the gate U1a has been flipped about its origin axis (origin is on Pin 19 [top left pin]). You can see the effect in the *Before* and *After* pictures.

Flip components in PCB

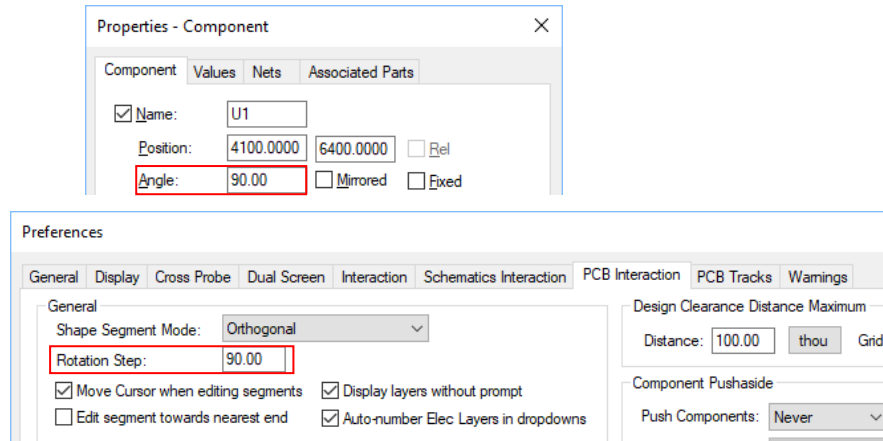
In PCB, Flip still works by mirroring the symbol about its symbol origin, but because PCBs are layer based the Flip command moves the item from one layer to its opposite outside layer. For example, a surface mounted pad on the Top layer would be Flipped to the Bottom layer, and vice-versa.



In the PCB example above using a surface mounted device (SMD), you can see that the axis point will be the centre of the device. By convention, all SMDs in the library have their origin defined in the footprint centre. Where text exists on the Top layer, this will also be automatically mirrored to the Bottom layer and will appear backwards.

Rotate

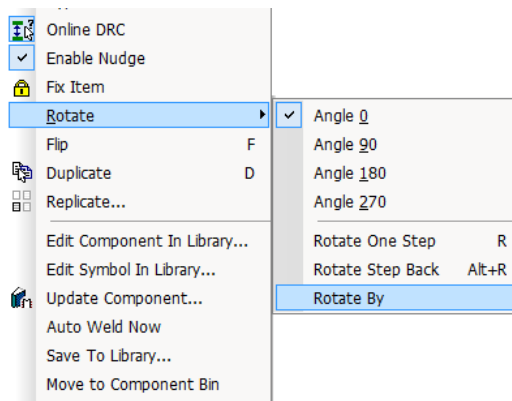
Rotate can be used on most items in the design. From selection or during **Place** or **Edit**, it is available on the context menu and by using the shortcut key <R>. Rotate is also available from the item **Properties** dialog. You can type the **Angle** of rotation into the box provided.



Rotation Steps are defined in the **Preferences** dialog and **Interaction** tab on the **Settings** menu. These are defined for the use in the Schematic and PCB applications. Rotation steps can be defined down to 1/100 of a degree (2 decimal places).

In the PCB design you should always bear in-mind your final manufacturing process, a rotation step of 32.75 degrees may cause unnecessary problems and costs!

As well as rotational steps, you can also use the **Rotate>** option on the context menu for predefined rotation steps.

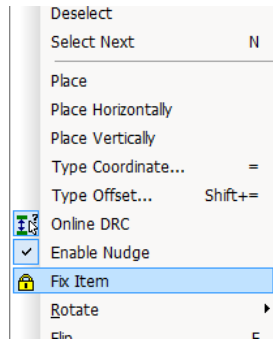


Rotation can be applied to multiple selected items using all the rotation methods above. This is useful if you wish to rotate all the resistors in your PCB design by 45 degrees for example.

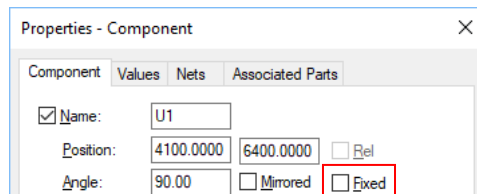
You can rotate one step back by using the shortcut key <Alt-R>.

Fix & Unfix Items

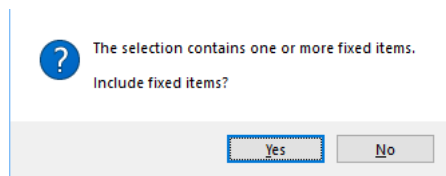
Design items can be fixed using the **Fix Item** command on the context menu. You must select the item first, then use **Fix Item** from the context menu.



You can also use the **Properties** dialog and the **Fixed** check box. The one shown below is for **Components**.



If you try and delete or move fixed items, the program warns you. You can override this by selecting **Yes**, or cancel with **No**. If you use **Yes**, once you have finished with that item the fix status is reapplied.

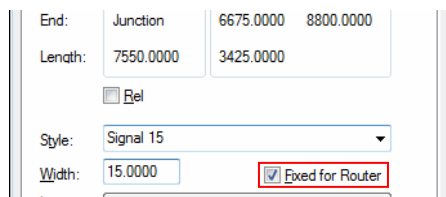


Multiple components can be fixed by making the selection using the **<Shift>** and **<Ctrl>** key option, and with the **Properties** dialog.

Fixed components cannot be interactively moved, flipped, rotated, changed or moved to the component bin without confirming the warning dialog.

Fixed tracks

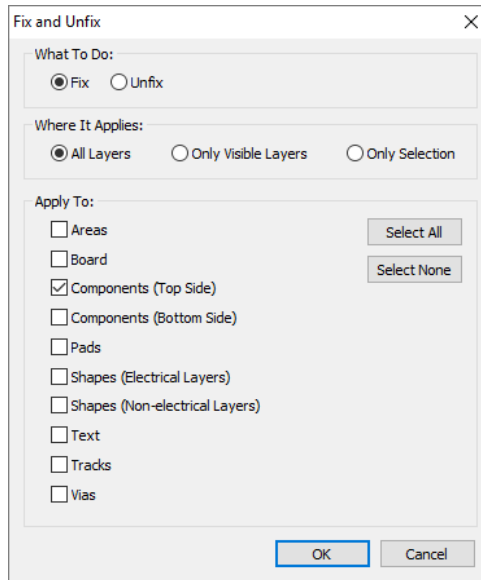
Tracks can be fixed for when using the router. For the selected track or track segments, use the **Properties** dialog and **Track Segments** tab has a **Fixed for Router** check box.



Using the Fix and Unfix Dialog

In addition to the interactive ‘one shot’ **Fix** and **Unfix** options, there is also a dialog-driven method of doing this too. Available within a PCB design on the **Edit** menu.

This allows you to fix or unfix multiple items by **layer**, on **only visible layers** or on **selection** of an item. The selection is then applied to an item category selected from the list. This can be items of different types, for example, **Areas**, **Components** and **Pads**.



With an item is fixed, a warning dialog will be displayed if you try to move, edit or delete it.

Cut, Copy, Paste and Duplicate

You can use **Cut**, **Copy** and **Paste** on design items to ‘cut’ (remove) and move them, or copy and paste as an additional copy into the design. The design items Cut or Copied are added to the Clipboard ready to be pasted into the design. They are also saved into the clipboard as bitmaps to be pasted into other applications.

Paste is used to retrieve design items from the clipboard and add them into the design. Only text cut or copied from another application can be pasted back in.

Duplicate is used to copy selected design items and paste them back into the design in one process. This works exactly the same as Copy/Paste but on one command.

These options are available on the **Edit** menu and use standard Windows shortcut keys that have been preassigned but which can be changed if you require.

Cut <Ctrl-X>

Copy <Ctrl-C>

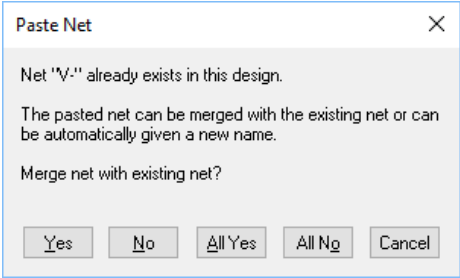
Paste <Ctrl-V>

Duplicate <D>

These options can be used on any selected items regardless of whether they are the same type of design item or not.

When pasting design items that contain net names, these must be resolved when being added to the design. As duplicate net names are introduced, you will be requested to either connect them to existing nets of the same name, or allow it to make them unique. Net names like GND, V- and +5v would normally be connected to each other, whereas net names like CLK or RESET may be renamed to be unique.

The **Paste Nets** dialog helps in the process of resolving these names.

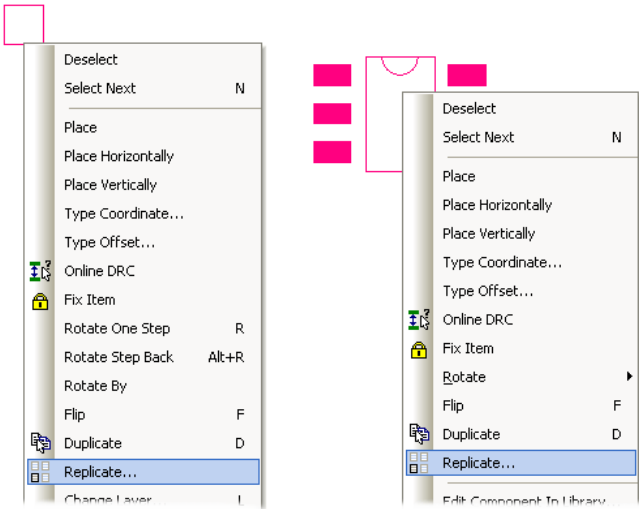


The options on this dialog allow you different choices for the nets in the paste group.

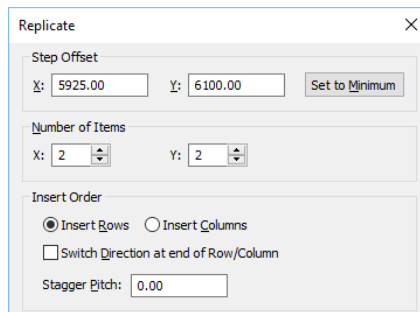
Replicate

The **Replicate** option will duplicate an item and then make a number of copies into an array. This is available for all design items, includes shapes and components.

From a selected item, a component or shape, from the context menu, select **Replicate**.

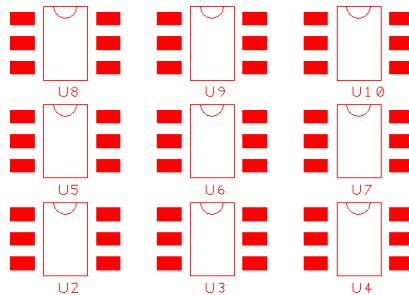


The **Replicate** dialog is displayed.



From this dialog set your required parameters and click **OK**.

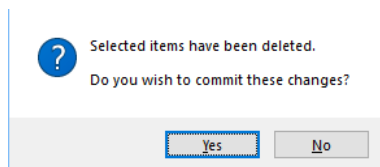
The pattern now looks like this:



Delete

Delete is available for almost every design item. It can be used by selecting the item then clicking the **Del** (or **Delete**) key on the keyboard, by selecting **Delete** from the **Edit** menu or by right clicking and selecting **Delete** from the context menu. Delete is also available on the **Edit toolbar** as an icon.

There is no confirmation for delete if an Undo level is set (in **Preferences** option under **General** tab) but deletions can be recovered by using the **Undo** option. If the Undo levels are set to **None**, a delete confirmation dialog is displayed from which to make absolutely sure you want to delete the item. It is recommended not to use this setting, keep Undo levels to at least 1 level!



Special cases of delete in Schematics

If deleting multi-gate components from the design, when a single gate is selected, the whole component is deleted not just the selected gate. Gates from the same component that are in the Component Bin are also deleted.

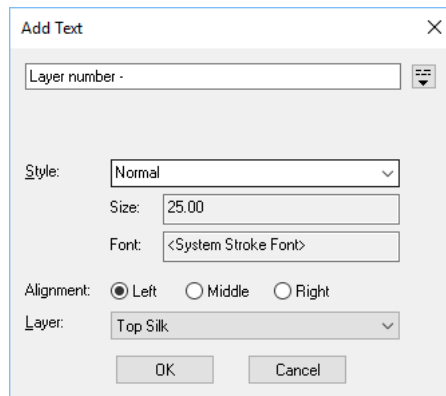
Special cases of delete in PCB

There are no special cases in PCB, you should be aware that when using delete on a track in PCB, the track and its connection path will be deleted from the design. This is a setting that can be switched in the **Preferences** option and **Interaction**. Select the **Delete Removed Attached** in, the underlying net path is still retained even when delete is used.

If you wish to delete the connection path as well you must make the deletion first then use **Nets> Change Net** and rename the other side of the net but ensure that the check box **Change Name Of Sub-Nets Only** is checked.

Adding Text

Text is added using **Add Text** from the **Add** menu or from the toolbar.



Type your text into the **text** field. If you wish to add multi-line text, click the small down arrow next to the text, this now allows you to use **<Enter>** to add multiple lines. Choose the text **Style** required and the **Alignment**. For a PCB design, you are also able to choose the **Layer**.

Once a text string has been entered click the **OK** button. The text will now be on the end of your cursor ready for placing within the design.

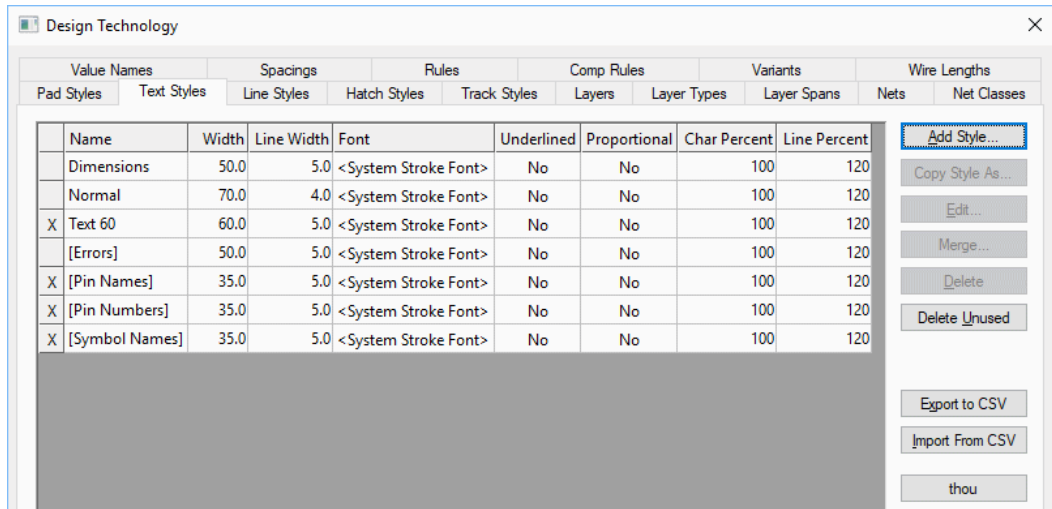
Once the text is in the design it can be edited or changed by selecting it. Right clicking now displays the context menu to change **Style** (text height, style and line thickness), to **Rotate**, change **Layer**, **Flip** (Mirror) or **Cancel**.

Rotate offers a choice of each of the four principal axes, or a rotation anticlockwise by the default step. This default rotation is defined using the **Preferences** option from the **Settings** menu.

Once released using a single click, the **Add Text** dialog is displayed again ready for the next text string, **Cancel** from the context menu ends the operation, you can also use the **<Esc>** button to cancel.

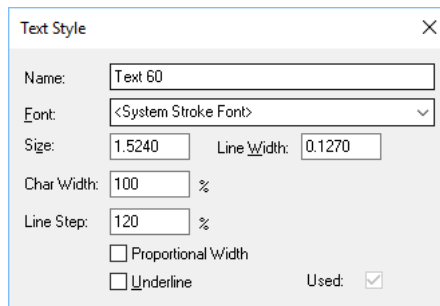
If you wish to edit existing text, double-click on the text and edit it in the edit box provided.

Text sizes are defined from the **Setting** menu under **Design Technology** and **Text Styles**. On the **Text** tab you can add text styles that will be required in the design or technology.



Within this tab there are a number of 'system' text styles which cannot be renamed but can be edited for different values. These are shown in square brackets, e.g. [Net Names]. Text style names should be reflective of a useable name for speed of recognition during use.

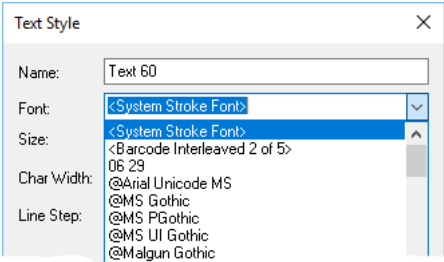
True type fonts are used as well as the original stroked font. These can be used by selecting the drop-down list box under **Font** on the main Text dialog, or by using the **Add Style** or **Edit** buttons. These will display the **Text Style** dialog.



From this dialog you can also make the style **Underline** when displayed by selecting the check box.

True type fonts can be used in both Schematic and PCB designs. When using these fonts in the PCB design, consideration must be made to how the boards will be manufactured.

If you wish to use the 'stroke' (or stick) font, you should select the **<System Stroke Font>** entry from the list of available **Fonts** in the drop-down list. This is shown in chevrons < > to indicate that it is the built-in font rather than a Windows system font.



Additional parameters on the edit dialog allows you to change the proportions of the Stroke Font. It can be **Proportional Width** to specify the text is to be spaced proportionally to close up the amount of white space between characters or you can change the **Character Width** or **Line Step**.

Adding Barred Text

To add text that is barred, add text as normal using the **Add Text** option but precede the characters you wish to be barred with the barred text delimiting characters which by default is the double underscore character (_ _).

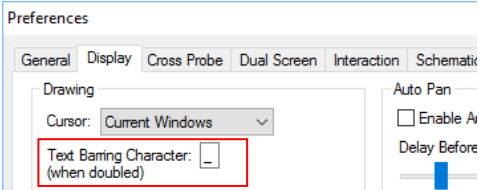
As an example, if you wish to have the word **barred** of **This is barred text** barred, you would define the text string as **This is _ _barred_ _ text**, (a space has been used in this example to show that two underscore characters have been used). This would then be written to the screen as.

This is barred text

If you wish to have barred text and use the underscore character you can either set the default delimiter character to another character or use an additional underscore character within the text string.

It is possible to change the barred text delimiting character via the **General** tab on the **Preference** dialog using the **Text Barring Character**.

The character specified as the **Text Barred Character** will cause all doubled occurrences of that character to enable or disable the barring of subsequent characters in that text.

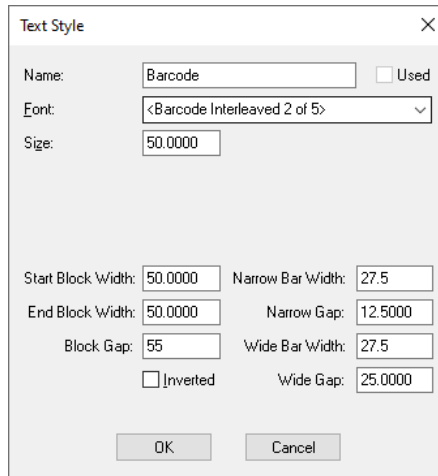


Note that this facility is provided mainly for use when importing other designs that already use other text barring delimiters and that changing the specified character will not convert existing barred text to the new delimiting characters.

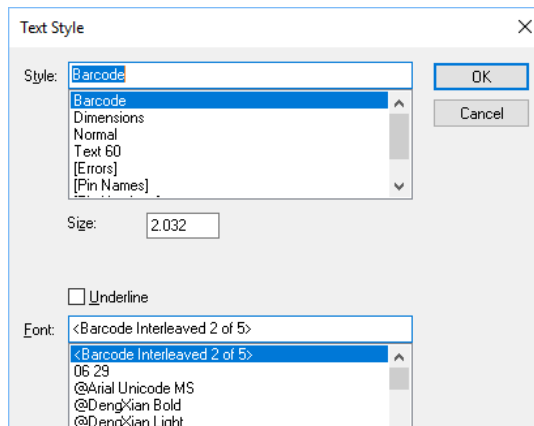
Adding Barcodes

You can define a barcode using a text style but only named barcode styles are allowed using the **Technology** dialog.

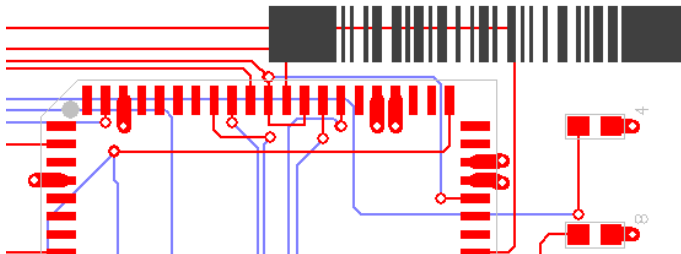
To enable barcodes, you must use the **Barcode Interleaved 2 of 5** text font. Once this is selected, you can then define the bars and gaps used for your barcode.



In the design, use the **Add Text** to add the numbers that will define the barcode, i.e. 05505223. Then change the **text style** of the barcode to your barcode font and the design will now display a barcode.



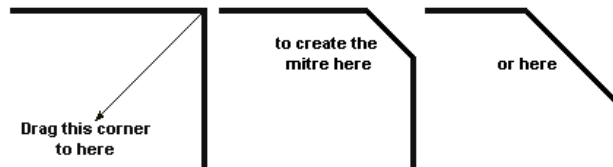
Until this special barcode font is used, the text will still appear as normal readable text. Detailed information about setting up bar widths and gaps can be found in the online Help.



Mitres, Fillets and Arcs

Mitres (45-degree corners) and **Fillets** (curved corners) can be added to any shape item in the design, this includes connections in Schematics and PCB tracks. Mitres can only be added to 90-degree corners on straight line segments. Mitres and Curves both behave in the same way and are added and edited the same.

The principle for editing corners to make them mitred (or filleted) is straightforward: you double-click on a corner and drag it into the position required. With the grid switched on, you can snap the diagonal corner to a grid point at varying mitre lengths. An **Edit Mitre/Fillet** option is available on the context menu as well for use.

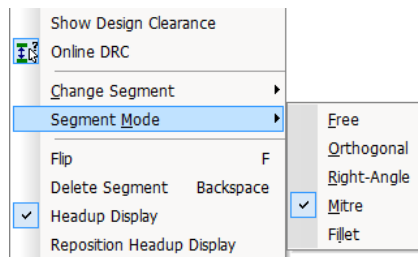


You can use the **Fillet** command from the context menu option to switch between Mitres and Fillets. You can also enter the radius using the **Enter Radius** command from the context menu for a selected Mitre.

Mitre Tracks

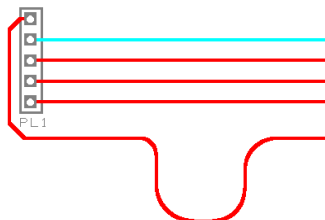
Mitred tracks can easily be inserted into the design using the **Mitre** segment mode. During addition of tracks or the editing of existing tracks, if you right click you can select **Segment Mode>** and then **Mitre**.

When adding or editing tracks, you will now be using orthogonal tracks in the X and Y bias but will use a **Mitred** corner on the direction change.



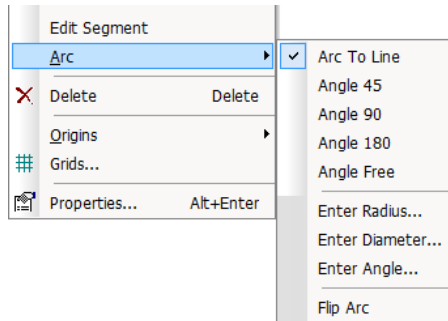
Curved Corners

When adding and editing corners, it is also possible to insert arcs and curves. While editing items you can switch the segment mode into **Fillet** (toggled using the <M> shortcut key). This will insert corners now as Fillets (Curves) instead of Mitres or 90 degree angles. The example below shows how a track segment might look using a filleted corner.



Adding Curves/Arcs

For an existing selected segment (Track or Shape), **Arc>** is available from the context menu.



The menu provides you with fixed angles but also **Angle Free**. During the **Angle Free** mode, the bottom status bar shows the **Angle** and **Radius**.

You can drag the arc out to the size required.



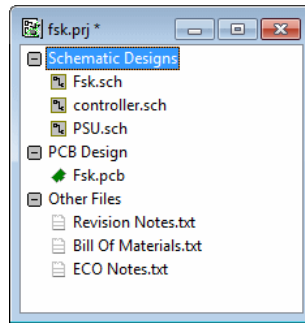
To use this, select a straight line segment and from the context menu, select **Arc>**. Using one of the edit arc options, create the arc or edit it using **Angle Free** and move the arc to required radius.

Chapter 6. Setting Up The Design

Projects

Projects are an optional feature; you can decide to use them from the start or not use them at all. If you decide you want them after the design has been started, then they can also be used.

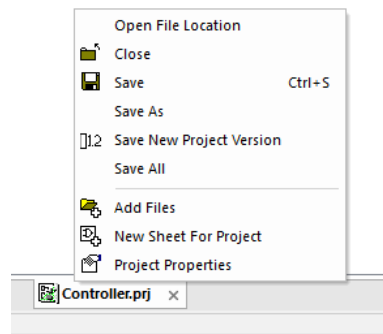
Projects are used to tie single or multiple Schematic sheets with a PCB design. Each sheet of a Schematic is effectively a single design file. The use of a Project will enable net names and Component names to be tracked through each sheet so that duplicates are not produced. The Project file is also used to group related design information in one file.



► To add files to a Project

Multiple Schematic designs can be added to the Project.

1. You can add files to a Projects with or without any design files open.
2. From the **Project** menu, select the **Add Files To Project** option. Or:
3. Right click with the cursor over the **Project** document tab at the bottom of the design window and select **Add Files** from the context menu:

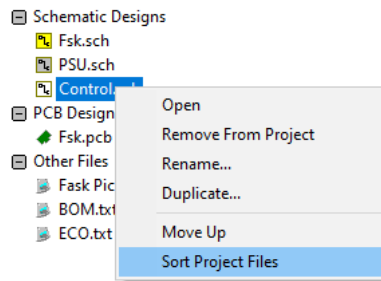


4. From the browser, select a design file to add to the Project. Other types of files, reports or Parts lists etc. can be associated with the Project as well using this option.

► To maintain a Project

You can do maintenance activities to a Project by using the options available on the context menu for a selected design file with that Project:

1. Open the Project.
2. Right click on a design.

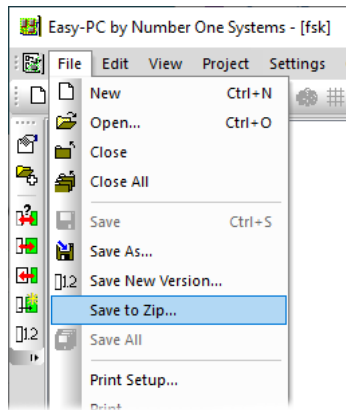


3. From the context menu, you can select **Open**, **Remove From Project**, **Rename**, **Duplicate** or **Sort Project Files** in the list.

► To save a Project to a Zip file

You can save the contents of a Project to a Zip file. This is useful if you wish to send the whole Project to another party:

1. Within the **Project**, from the **File** menu, select, **Save to Zip**.



2. From the **Save As** dialog, type the name and location you will save the file to.

File Operations

There are some basic commands available from the **File** menu that will help with the general upkeep of your designs:

Save/Save All allows you to quickly save all currently open designs and library items.

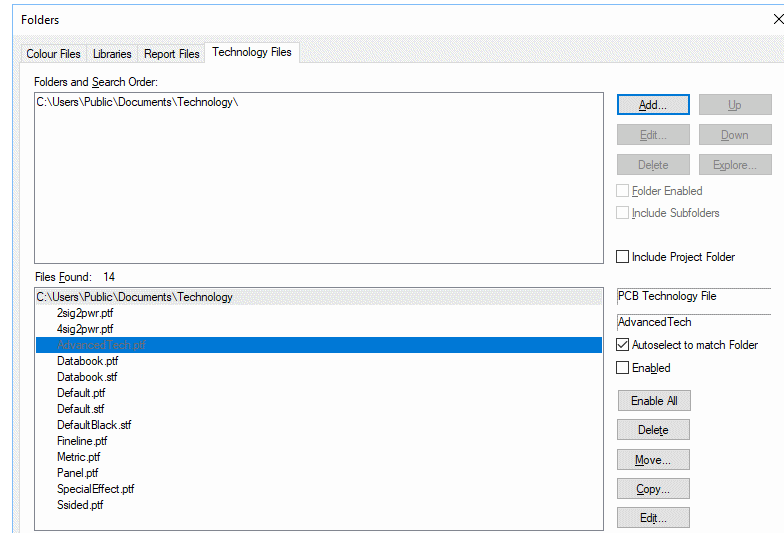
Close/Close All allows all open file designs to be quickly closed. If a design has not been saved, you will be warned.

Reload is used to discard the current edit and reload the design from the previously saved version on disk.

Open From Backup is used to open a design from a backup file if required. The backup file will be auto-named starting with a ~, i.e. ~design1.pcb

Folders

You can predefine where **Technology Files**, **Libraries** and **Colours Files** will be saved and loaded from. The **Folders** dialog is used to manage these from the **Settings** menu.



Saving Your Files

During usage you should regularly save your designs. You can use **Save** or **Save As** to do this. In addition to these features, you can also switch on the automatic security copies feature (see *Security Copies* below).

To 'force' a save or to save the design to a new name, you can use the **Save** command from the **File** menu, or the **Save As** command from the same menu.

The file extensions will automatically default to the type used for that application. e.g. .sch for Schematics and .pcb for a PCB design.

Save Technology on the **File** menu allows you to save the Technology within a design. This can be used again on a new design or applied to a different design to give it the same parameters.

The **Load** button on the **Colours** dialog will allow you to save Colour files, these can be applied to different designs.

Saving Libraries

When creating a Symbol (Schematic or PCB), you have the **Save To Library** option on the **File** menu as well. Once your new symbol is ready to save to the library, you should use the **Save To Library** option and **not** the **Save** or **Save As** options. This is covered later on.

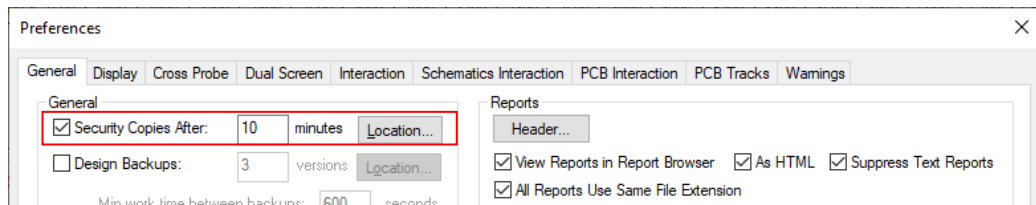
Security Copies

Making regular backups is a basic necessity when working with computers. A requirement, unfortunately, more honoured in the breach than in the observance. Hence, an automatic security backup service is provided. This can be switched on or off and set to specific intervals for use as required.

During use, the design is held in memory, if you make any mistakes the undo and redo functionality can save the situation by allowing you to recover commands previously executed (but only up to the number of **undo** levels set in the **Preferences** dialog). Major faults, like someone pulling the mains lead out of the computer will normally mean that you have lost any work back to the last save or security save, depending on which was the last used. Obviously, the more save is used the less design work is lost in one of these situations. Saving your work really is an essential part of the design process as well.

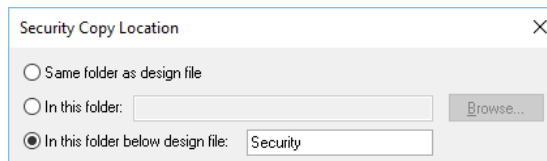
The status bar at the bottom of the screen will inform you that the application is making a security copy, this happens automatically at the time interval set. There is no user interaction, the security copies just happen.

An automatic security back-up facility is provided. This can be pre-defined for regular intervals. Back-up copies are automatically created of your design when the security interval is activated. The control you have for updating the security copy at intervals is determined by the **Security Copy Level** selection on the **General** tab of the **Preferences** dialog.



By making an entry of 0 (zero) minutes, the security copy is switched off. Any value greater than zero will activate the security copy.

The location of the security files can also be defined. By selecting the **Location** button, a dialog allows you to set the location to be **the same folder as the design** or one defined by you (**In this folder:**). The **Browse** button allows you to browse to a location to be used.



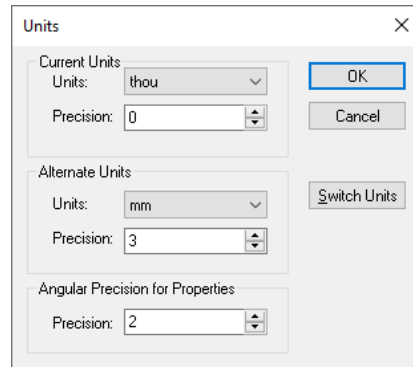
Security copy files are designated using the tilde (~) character. For example, opening 'Div.sch' will automatically produce a security file '~Div.sch' when the security interval is activated. You should not use these files but instead, copy them to a new name (removing the ~) and continue with the design. If you load a security file, you will be warned.

When using the product, you should not open files which start with the ~ character, this means you are opening your back-up which could be out-of-date with the design you intended editing. If you open the back-up file, you will notice that a back-up of the back-up will be made e.g. '~Design66.sch' will become '~~Design66.sch'

Units and Co-ordinates

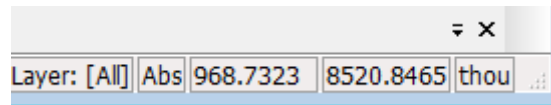
It is possible to change the display from **Imperial** units to **Metric** units and vice versa. This is done with the **Units** entry on the **Settings** menu. The database precision used is 1/10th Micron, this means there are true Imperial / Metric conversions at all times, with no rounding errors. This accuracy is essential in modern designs where mixed Imperial and Metric components are created and placed. You can also use the shortcut key <Shift-I> to toggle between Imperial and Metric units.

Changing the units does not affect the pitch of the grid but simply changes it to be the same size in both imperial and metric units.

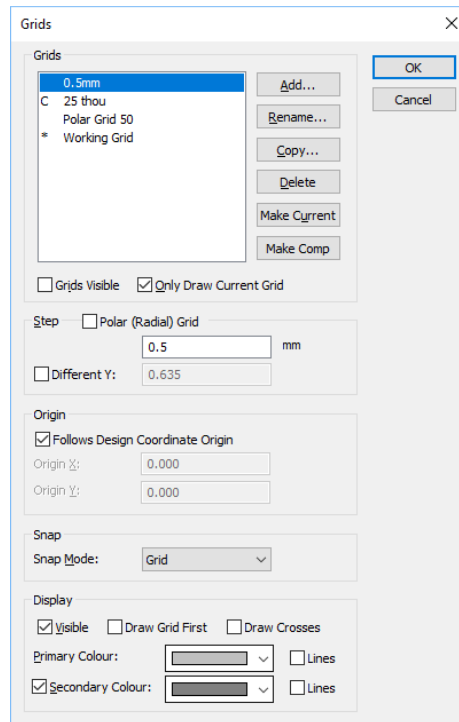


Imperial (Thou) (or Mils) units would normally be defined to 0 (Zero) decimal places, while **Metric (mm)** units would be defined to 2 or 3 decimal places. This is because of the rounding and accuracy required when using Metric units, this is especially important for the definition of drill sizes.

The **Units** used and **Coordinates** are shown on the status bar at the bottom of the design. You'll note that **Abs** (or **Rel**) are shown to indicate that you are working in Absolute or Relative modes. With the active status bar, double-clicking on these fields will display the relevant options.



Grids



Grids are used for both defining the points or steps that items will snap to when they are placed and for visible references in the design. For this, use the **Grids** dialog on the **Settings** menu. You can add and use your own named grids each with its own display settings, add and use polar (radial) grids, and have your own grids displayed in the **Grid Snap** context menu.

Grids

The first section of the dialog shows you a list of all the Grids currently defined in the design. From here you can carry out actions such as adding new grids or changing the name of an existing grid. Apart from adding a new grid, all other actions on this dialog will act upon the grid that is currently selected in this list.

Add : add a new grid to the list. You will be prompted to supply a new name for the grid, which must be different from the names of any grids that exist already.

Rename : change the name of the selected grid.

Copy : make a copy of the selected grid under a new name, copying across all the settings from the selected grid.

Delete : remove the selected grid from the design. Note that there always needs to be at least one grid, so the application will not let you delete the last grid from the list.

Make Current : sets the selected grid as the current working grid to use for placing items. The 'current' grid is marked in the list with an asterisk. As a shortcut to changing the current grid you can simply double-click on the required grid in the list. This will make that grid the current grid, and close the dialog.

Make Comp: sets the selected grid as the preferred grid to use for placing any selection containing one or more components. The 'component' grid is marked in the list with a letter 'C'.

Grids Visible : this is a global switch that toggles the display of grids on and off. This is the same as using the **Toggle Screen Grid** toolbar button or command. Note that each grid also

has a Visible check box, so that you can choose which of your grids should be drawn when grids are set as Visible.

Only Draw Current Grid : when drawing grids, normally all grids which have their Visible check box ticked are drawn. By checking the Only Draw Current Grid box, you can choose to draw only one grid, the grid that is set as the current working grid. Using it this way will mean that as you flip from one grid to another, the display changes to draw only the grid that is current (active).

Step

The first box is used to define the working grid step. Type the required size in the current design units.

The screenshot shows a dialog box with the following fields:

- Step: Polar (Radial) Grid
- 25.0000 thou
- Different Y: 25.0000

Different Y : check this box to use different grid step in X and Y axes. By default, the X and Y steps are the same.

Polar Grid : check this box to define a polar (radial) grid. When you check this box, some different settings appear on the dialog:

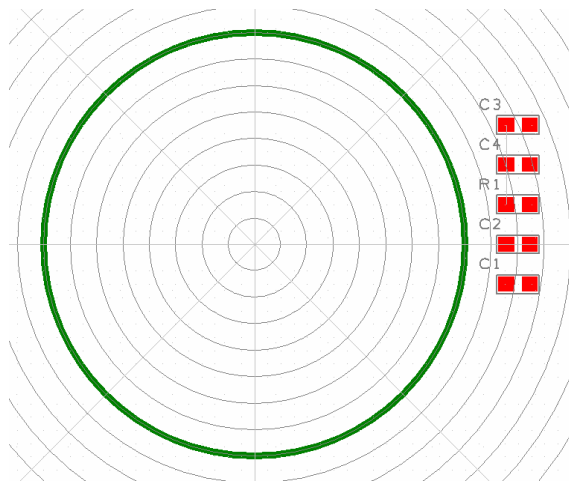
The screenshot shows a dialog box with the following fields:

- Step: Polar (Radial) Grid
- Concentric Step: 25.0000 thou
- Divide Circle Into: 8 steps

Concentric Step : this is the distance between each 'ring' of the grid.

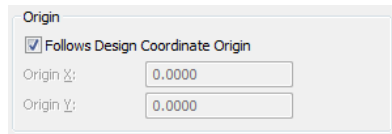
Divide Circle Into : this is how many 'slices' the grid should be divided into.

Here you can see an example of a Polar grid, in this case divided into 8 segments:



Origin

This section of the dialog allows you to specify the origin or 'starting point' of the grid.

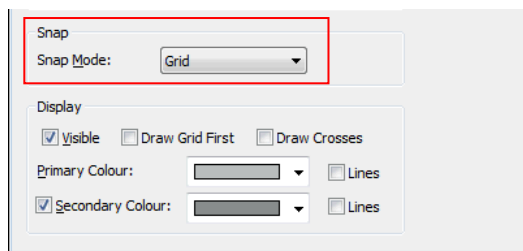


A Polar grid will be centred at the origin position, and a normal 'step' grid will have its first point on the origin and step outwards from there in each direction.

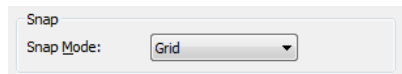
Follows Design Coordinate Origin : check this box to have the grid start from wherever the design System Coordinate Origin is placed.

Origin X and Y : here you can specify a particular location for the grid origin when it is not following the coordinate origin.

Snap



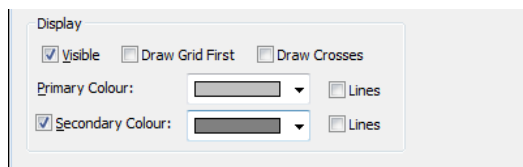
Snap Mode is used to further define the grid step that is actually used in interactive operations such as Place. Choose a value from the drop down list as follows:



If you choose a setting other than 'Grid', each step of the grid is divided according to the setting you select. Snap Mode is not available for Polar grids.

Display

This section allows you to control how each grid is displayed. As with the other settings, each grid has its own display values.



Visible : check this box to make this grid visible. Note that it will still only be drawn if the main **Grids Visible** check box is also checked.

Draw Grid First : check this box to draw the grid before drawing actual design items. This has the effect of drawing the grid 'underneath' everything else.

Draw Crosses : use this to draw small crosses instead of 'dots' or lines.

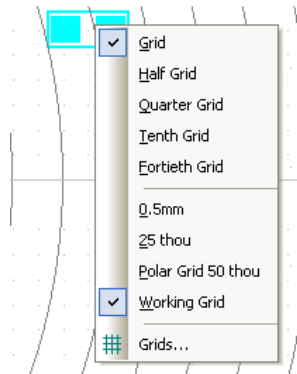
Primary Colour is used for every 10th grid point of a step grid, or for the radial 'arms' of a Polar grid. Check the **Lines** box to have the grid drawn as lines.

Secondary Colour is used for all other grid step points. As you use Unzoom to increase the amount of the design that is displayed you will reach a level where the secondary grid points

are too close together and they will no longer be displayed. The same is true for the Primary points for a step grid if you carry on with zoom out.

For a Polar grid, if you uncheck **Primary Lines**, the grid is drawn with a single dot at each place where the rings and the radial arms intersect.

Change to the Grid Snap context menu



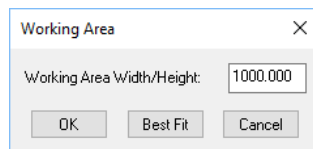
The **Grid Snap Mode** shortcut can be displayed during editing by selecting the direct command <H>. By selecting the snap mode setting you to choose current grid, and access **Grids** dialog. These can be changed interactively when editing or placing items in the design. The default shortcut for this is the <H> key. A small menu will be displayed which allows you to select the Snap Mode (for a step grid), to change the current working grid to a different grid (if more than one grid is defined), or to access the Grids dialog.

Shortcut for toggling grids

The **Visible** check box enables the grids to be displayed on or off, this can be set at any time in the design using the shortcut key <G>.

Changing the Working Area

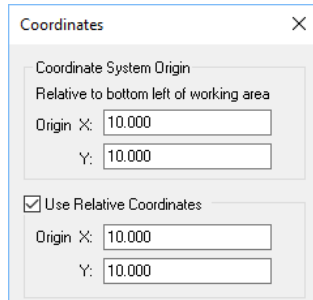
The **Working Area** option can be used to control the size of the design 'desktop' that your design is working in. By setting the working area to be slightly larger than your actual design it will enable you to zoom in closer and to use all of the Zoom shortcut keys on the keyboard. If the working area is set too large for a design, some of the shortcut keys will have no effect. When this is the case, select the Working Area option from the Settings menu. Unless you are familiar with the program use the **Best Fit** option.



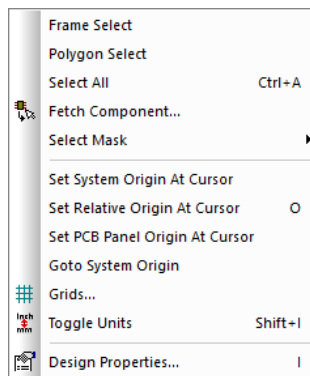
This will set the design area to make the best use of its size relative to the actual design or design items (this is the maximum area of design items, board outline or drawing blank).

Changing the System Coordinate Origin

You can change the system coordinate origin to be any other position within the design, the lower left corner of the board outline or design centre for example. By default, the coordinate system is defined as the lower most left corner of the working area (0,0). These are changed or set from within the **Relative Coords** option on the **Settings** menu.

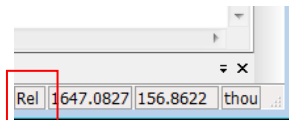


The system coordinates can be changed using the **Relative Coords** dialog on the **Settings** menu, or by using the option **Set System Origin** from the context menu (with nothing selected in the design). This option is also available for some selected items, such as pads or terminals.



Working in Absolute and Relative Modes

Two origins are maintained for the coordinate system, these are displayed at the bottom of the screen on the status bar. The status bar shows the mode as **Abs** or **Rel** for Absolute and Relative, the **Coordinates** and the **Units** (as previous discussed).



The **Absolute (Abs)** origin is fixed at the bottom left hand corner of the drawing area defined by the design area or border. All coordinates reported are positive in both X and Y directions.

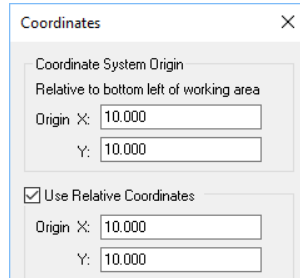
The **Relative (Rel)** origin can be set to any point within the design. The co-ordinates reported will always be relative to this position.

The **Relative origin** is very useful when it is necessary to position a pad or track accurately with reference to another pad or track in, for example, the case of component fixing holes. It is also an ideal tool for the measurement of distances and clearances.

Switching between Abs and Rel mode can also be done using the **Settings** menu and **Switch Rel/Abs Coords** option. This will be shown with a **tick** next to **Relative Coords** when in Relative mode.

Setting the Relative Origin

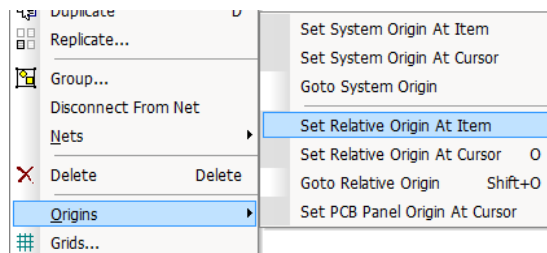
To use the Relative co-ordinate system, select the **Relative Coords** option from the **Settings** menu. From the dialog you have to select the **Use Relative Coordinates** check box to toggle to the Relative mode. You can enter the **Origin** position as an **X** and **Y** co-ordinate. You can also define the system origin, the relative origin is taken from this location.



This will toggle the co-ordinate display on the status bar between the **Relative** and **Absolute** origins, shown as **Abs** and **Rel** as the origin mode changes.

As well as selecting the **Use Relative Coordinates** check box to select the Relative mode you can also use the **<Shift-R>** shortcut key to toggle between the **Abs** and **Rel** modes. Once in **Rel** mode, to use this option, move the cursor to the origin point required and then click **<O>**.

You can also set the Relative Origin using the options from the context menu for a selected item:



Any mouse movement will now be shown relative to this position.

Using Rel mode

When using the Relative origin mode, you can go to the origin by using the **Goto>** and **Relative Origin** option from the **View** menu or by selecting the shortcut key **<Shift-O>**. There is also an option on the context menu with nothing else selected.

Mostly, you will use the co-ordinate system in **Absolute** mode which shows the cursor position relative to the **Absolute Origin**, but there are times when it would be more useful to be able to see the cursor position in relation to the Relative Origin, for example when we are laying down related pads, tracks or connectors.

Using Technology Files

Overview of Technology Files

Technology files are like the start-up files used by word processors, such as Word. These files contain groups of predefined styles that can be applied to design items, items such as tracks or connections or text for example.

These can be used on successive designs to greatly speed up the setting up for new designs of different types. The setup required for a four-layer, fine line surface mount device board with internal power and ground planes is totally different to how a low cost double-sided board might be. Technology files can also help standardise company standards or practices.

Technology files provide a means of setting most of the important parameters. They are read once when starting a new design and the design then inherits the contents of the technology file. The technology within the design can be edited and modified or added to. If you wish you can save the then edited technology back out to a new Technology file or overwrite an existing one. Having started a new Schematic or PCB, those designs will not be affected by subsequent changes to the technology files. There is no danger that subsequently modifying Technology files will affect previous good designs.

Existing Technology files can be applied to existing designs using the **Technology Files** option from the **Settings** menu. This is discussed later on.

Technology Files minimise errors and produce faster results, it is highly recommended that you use them.

Using Technology Files

Technology Files are design application specific, for example, you cannot use a Schematic Technology files in a PCB design vice-versa. However, you can use and should use a Schematic Technology File when creating Schematic Symbols. The Technology File that you expect to use for the design should be the same one used when creating the Symbols, this will help synchronise the names and values used for styles etc.

How to create and save Technology Files

Technology Files can be created as a specific non-design related task in their own editors, or created/modified from a design using a new or existing Technology File as the starting point.

If starting from new, you can use the **New** option from the **File** menu. On this dialog you can use the Technology tab. This will allow you to access the Technology editor application. This is like a design editor application but you can only edit Technology related information, such as **Styles** or **Net Classes** etc.

Once the Technology has been created (by editing the various entities of the Technology), use the **Save** option from the **File** menu. This will request a name for the Technology File and its location. By default, the location will be that previously defined in the **Preferences** dialog under **Directories** and **Technology Files**.

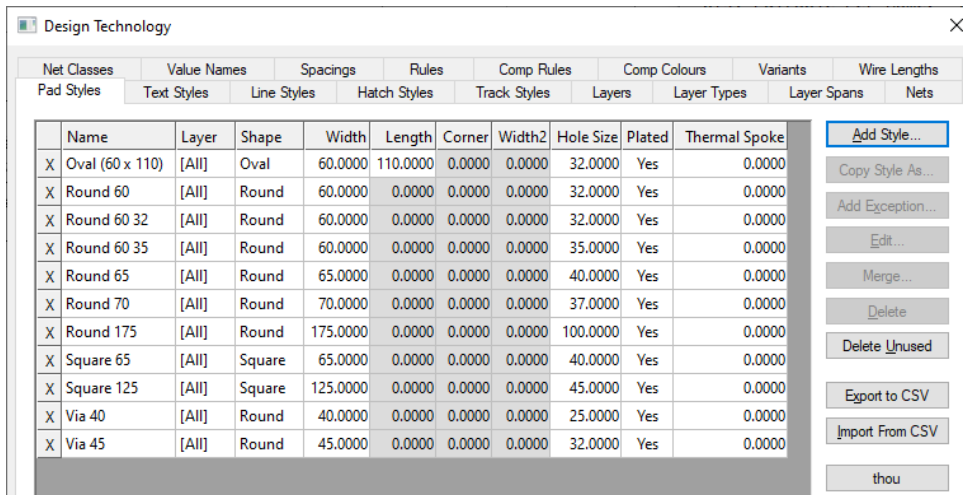
When saving a Technology file from within an existing design use the **Save Technology File** from the **File** menu. This will extract all the necessary entities from the design and save them to the Technology File.

Styles in Technology Files

One of the most important entities of the Technology is Styles. Styles are used to define the shape and size of items, such as Pads and Text. When design items have styles, it means that you can modify all the pads in a design or example by editing one value in the Pad Styles table. Obviously, local styles can be applied to one or more selected items in the design if required. All styles are defined in the **Design Technology** dialog under their respective styles.

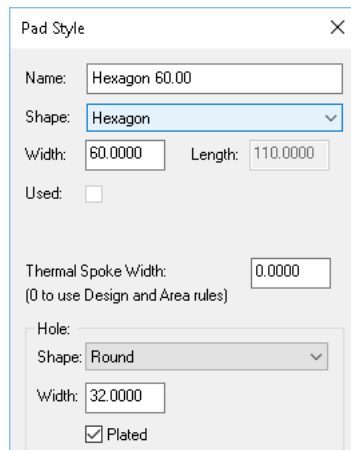
Pads

The Pad styles are used to define the style of Pads and Vias in a PCB design and Pin or Terminal styles in a Schematic design. The main difference between these two types of styles is that the Schematic pad style only defines the **Shape**, **Width** and **Length** of the pin, whereas the PCB pad style has a **Layer** and **Drill Hole** information as well.



Net Classes	Value Names	Spacings	Rules	Comp Rules	Comp Colours	Variants	Wire Lengths		
Pad Styles	Text Styles	Line Styles	Hatch Styles	Track Styles	Layers	Layer Types	Layer Spans	Nets	
Name	Layer	Shape	Width	Length	Corner	Width2	Hole Size	Plated	Thermal Spoke
X Oval (60 x 110)	[All]	Oval	60.0000	110.0000	0.0000	0.0000	32.0000	Yes	0.0000
X Round 60	[All]	Round	60.0000	0.0000	0.0000	0.0000	32.0000	Yes	0.0000
X Round 60 32	[All]	Round	60.0000	0.0000	0.0000	0.0000	32.0000	Yes	0.0000
X Round 60 35	[All]	Round	60.0000	0.0000	0.0000	0.0000	35.0000	Yes	0.0000
X Round 65	[All]	Round	65.0000	0.0000	0.0000	0.0000	40.0000	Yes	0.0000
X Round 70	[All]	Round	70.0000	0.0000	0.0000	0.0000	37.0000	Yes	0.0000
X Round 175	[All]	Round	175.0000	0.0000	0.0000	0.0000	100.0000	Yes	0.0000
X Square 65	[All]	Square	65.0000	0.0000	0.0000	0.0000	40.0000	Yes	0.0000
X Square 125	[All]	Square	125.0000	0.0000	0.0000	0.0000	45.0000	Yes	0.0000
X Via 40	[All]	Round	40.0000	0.0000	0.0000	0.0000	25.0000	Yes	0.0000
X Via 45	[All]	Round	45.0000	0.0000	0.0000	0.0000	32.0000	Yes	0.0000

The **Add Style** or **Edit** buttons provide you with a **Pad Style** dialog:



Pad Style

Name:

Shape:

Width: Length:

Used:

Thermal Spoke Width:
(0 to use Design and Area rules)

Hole:

Shape:

Width:

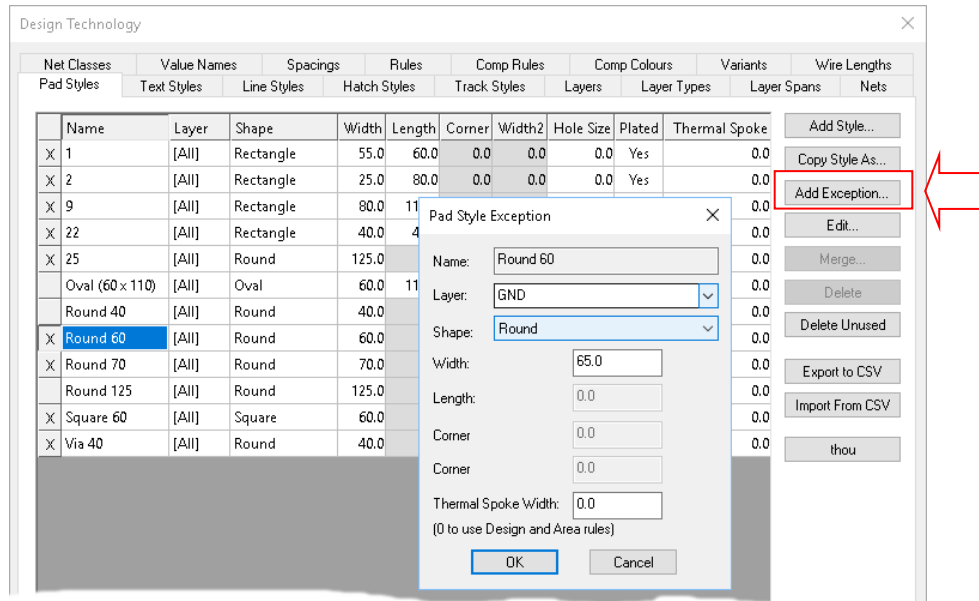
Plated

When defining pad styles in PCB, the style is assumed to be on all layers (shown by default as **Layer [All]**), this is the default pad 'stack'. The only time this setting changes is when a pad exception has been added to a particular pad style.

Pad Exceptions

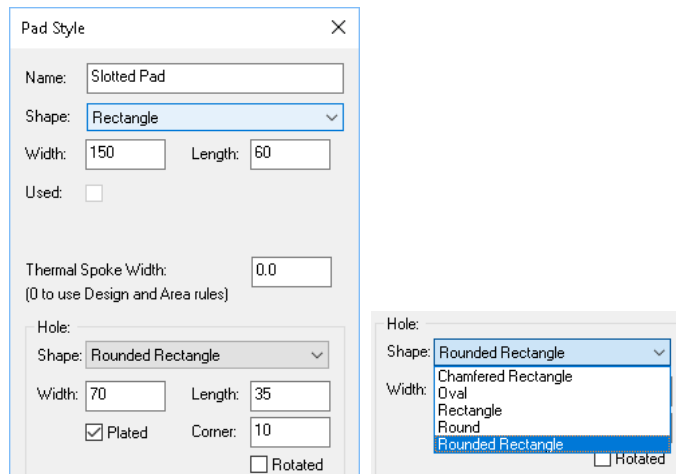
In the PCB Pad style dialog, it is also possible to give a pad a different style on a given layer using the **Add Exception** button.

Once the pad exception has been applied, the grid then displays a layer exception for the pad style changed, in the example below, this is shown on the pad style **Pad**. A layer exception for the **Bottom Electrical** has been added to increase the pad size and pad shape.



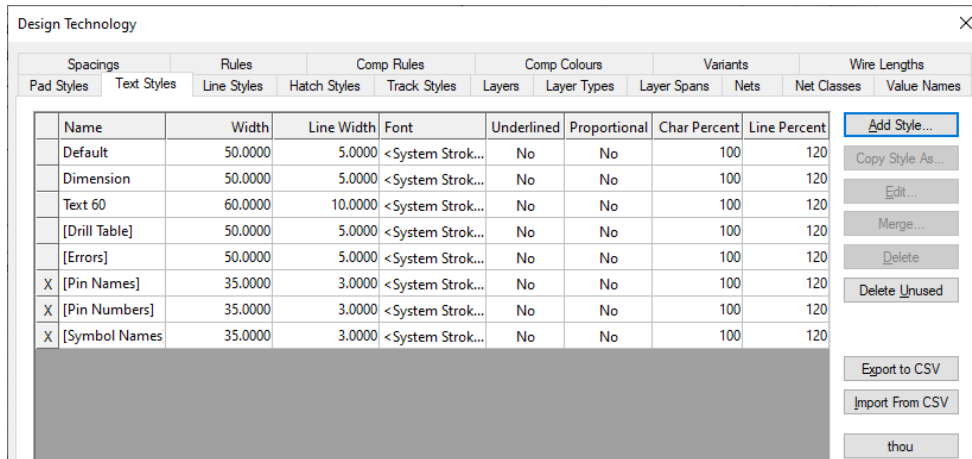
Slotted Pads

You can create slotted pads using the **Pad Styles** dialog, you can specify a **Hole** as being a non-round shape, a **Rounded Rectangle** for example. Using the non-round shape will create a slotted pad.



Text

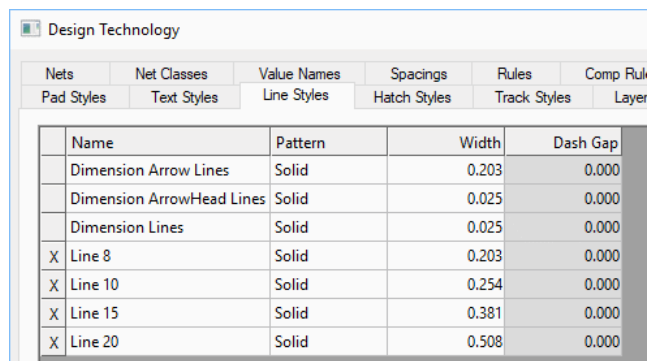
As with Pad styles, Text styles can be created and edited from the **Design Technology** dialog. Within the grid, predefined system names are shown in brackets, [Errors] and [Pin Names] for example. These can be edited but cannot be renamed or removed.



Text styles can be given a **Font** as well. The fonts will be selected from any Windows fonts that are installed and available in the drop-down list. If no font is selected then the **Font** box will show an empty grid box or **<System Stroke font>**, these are the same. The System Stroke Font is a built-in font made up of very simple lines.

Lines

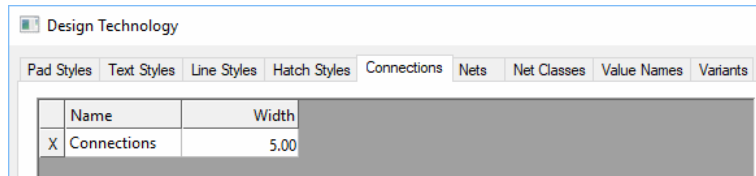
Line styles are used for line items within the design, items such as **Symbol outlines**, **Silkscreen outlines**, **Bus lines** and **Connections**. As well as the **Name** of the Line style, the **Width** of the line is defined. In a PCB design, Line styles will also be used for **Dimension Lines** and **Board Outline** as well.



Lines are not used for **Connections** or **Tracks**, these have their own style definitions.

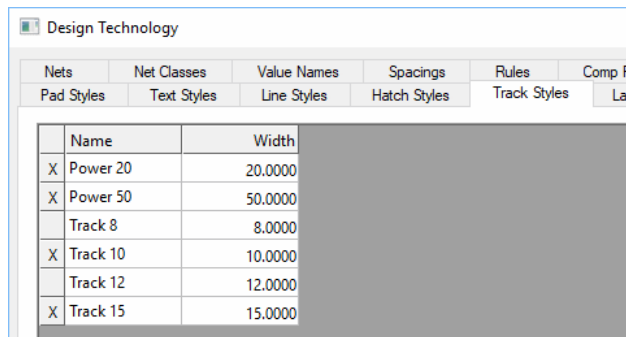
Connections in Schematic Technology Files

There are no **Track** styles in a Schematic design so the **Connection** style is defined. This style is used when adding connection to the design. By changing the style and thickness you could show different connections that are more or less significant than others.



Tracks in PCB Technology Files

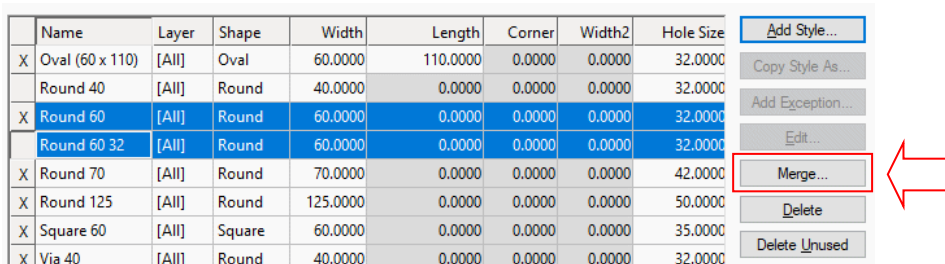
The **Track styles** dialog in PCB Technology files is used for defining the thickness of tracks. These can be used locally on specific nets or as part of a **Net Class** that defines the track width limits (and via style) when autorouting or manual routing.



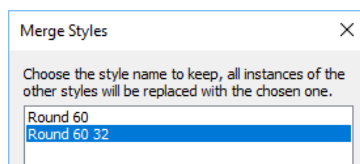
With a nominated net class on a net, from then on, it will automatically pick up all of the parameters of that net class. What is more, changing the net class in a design will automatically change all of the elements that have been given that net class. (Net Classes are discussed in more detail below).

Merging Styles

There is a **Merge** button on the **Styles** pages of the **Design Technology** dialog.



This allows you to select two or more rows in the table and merge them together. This will update the design to replace all instances of the 'merged' styles with the one style you have elected to retain.



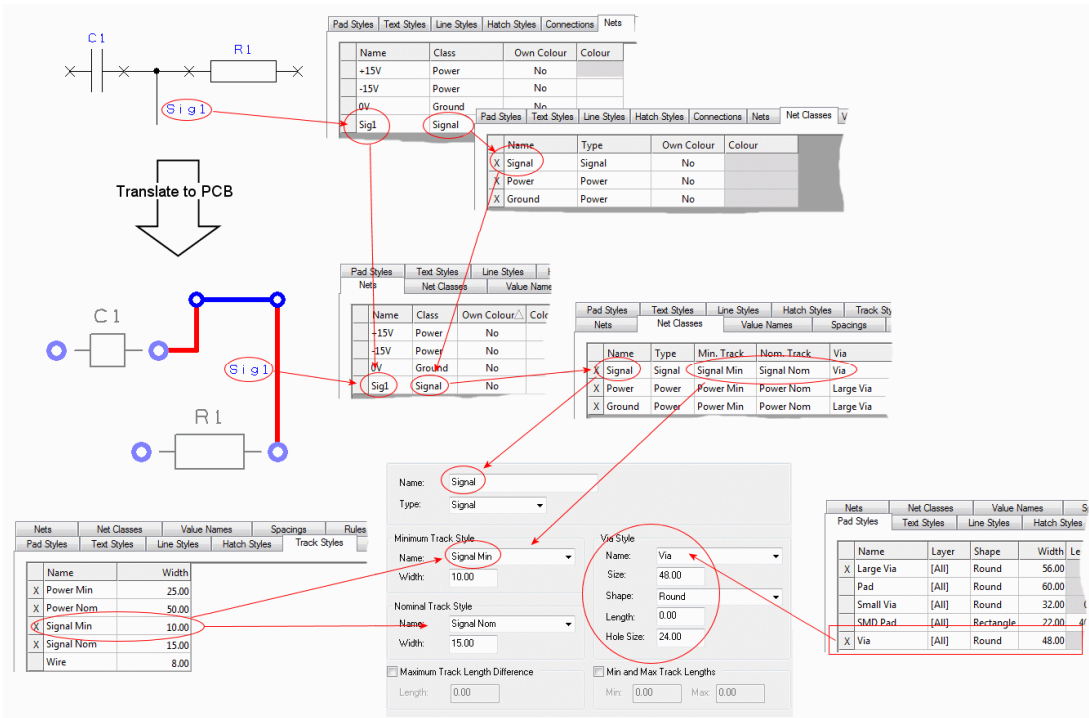
From the dialog, choose the style name that you wish to keep and click OK. All the other styles will then be merged with the selected style, including updating all references to that style in the current design. Note that you cannot remove built-in styles (those with names enclosed by square brackets), so if you select any of those you may not be able to perform the Merge action.

You may find the **Merge** facility useful if you have combined sections of data (components, connections, etc.) together from several different sources, where the styles used in those sections may not be consistent. Note that merging styles will only affect the current design, it does not go back to the library and rationalise style settings there.

*Note: The ability to select multiple rows required for **Merge** means that you can also use **Delete** on several entries at once.*

The relationship between Net Names and Net Classes

A Schematic design and PCB design would have the same Net Name for the equivalent design (in theory!). When designing a Schematic, you can assign a net a Net Class name. **Net Names** are stored on the **Design Technology** dialog under **Nets**.



When a Schematic is translated to the PCB, the matching net class is found (matched by name, the net has the net class assigned to it) and the PCB net properties applied to that net class. The Net Class in the PCB design has inherent properties that give the net more intelligence. For example, the Net Class will also contain track thicknesses and via styles that will be used once routing is started. The diagram below illustrates the relationship between the Schematic and PCB designs.

Net Names in Technology Files

Two types of naming are supported: the first is a default system supplied net name (an **auto-named** net), such as \$1, \$22, N0011 etc., the second is a user defined name, such as Gnd, Vcc, Sig1, Clock, Reset etc.

Once a net name is used in the design it then appears in the **Nets** dialog. The Nets dialog for Schematic and PCB Designs are slightly different. The Schematic Nets dialog shows the **Net Name**, the **Net Class** and a **Colour** for that net.

Net Names can be edited within this dialog and the new name applied to the design. Where a net name is shown as used (by a small **X** next to the name) it cannot be deleted from the design.

	Name	Class	Own Colour	Colour	Show Connectic	^
X	DRIVE	Signal	No		Yes	
X	Feedback	Signal	No		Yes	
X	GND	Power	Yes		Yes	
X	High_speed	Signal	No		Yes	
X	Low_speed	Signal	No		Yes	
X	V+	Power	Yes		Yes	
X	V-	Power	Yes		Yes	
X	N01	Signal	No		Yes	

You have a check box at the right of the dialog for **Showing Auto-Named Nets** in this grid, this helps clarify user defined and system named nets.

The colour can be edited by double-clicking on the colour box and selecting a colour from the drop-down list.

The colour shown in this dialog overrides the colour used for **Connections** in the **Colours** dialog but only if it is different.

The PCB Nets dialog shows the **Guard Spacing** and not the Colour. Guard spacings are used to add extra spacing to a net.

Net Classes

The **Net Class** is used to define a ‘group’ or collection of net names which have the same characteristics, i.e. track thickness or Via style.

In the **Design Technology**, on the **Nets** dialog, the **Net Class** name can be selected from the list. Only names that have been previously defined in the **Net Class** dialog are available in this list.

Name	Type	Min. Track	Nom. Track	Via	LenDiff Enabled	Length Diff	MinMax Enabled	Min	Max	Own Colour	Colour	Tented	Diff Pairs	Diff Pair Gap	Diff Pair Skew	Max Skew
Signal	Signal	Signal Nom	Signal Min	Via	No	0	No	0	0	No		None	No	39	No	0
Power	Power	Power Min	Power Nom	Large	No	0	No	0	0	No		None	No	39	No	0
Ground	Power	Power Min	Power Nom	Large	No	0	No	0	0	No		None	No	39	No	0

The **Edit Net Class** dialog enables you to make your selections:

Net Classes in the Schematic Design

In the Schematic design, Net Classes are defined as simply a name, there is no physical attributes at this point. The physical attributes of the Net Class are assigned in the PCB design **Net Class** dialog and are passed through when using **Translate to PCB**.

It is the Net Class name that is important, this is the name matched in the PCB design. From this Net Class name the physical information is assigned.

Net Classes in the PCB Design

The Net Class dialog in the PCB design contains the physical attributes of the net as well as its name. It is the Net Class name that makes the association between the Schematic and PCB design. The attributes defined will be used when the net is converted to a track using a manual or automatic routing process.

	Name	Type	Min. Track	Nom. Track	Via	Len
X	Signal	Signal	Track 10	Track 15	Via 40	
X	Power	Power	Power 20	Power 50	Via 40	
	Ground	Signal	Power 20	Power 50	Via 40	

By selecting the **Edit** button on this dialog, you can change the track and via styles used by that net class. You can also edit the grid directly by double-clicking on the name in the grid.

The styles used are taken from the **Track** and **Pad Styles** dialogs on the **Design Technology** dialog.

The Net Class page is also used to define **Diff Pairs** and **Tented Vias**, more about these later.

Net Class to Class Spacing Rules

You can define spacing rules which match by Net or Net Class name and Layer Name. These are used to produce extra clearances between items on different Net Classes, to allow for creepage for example.

	Tracks	Pads	Vias	Shapes	Text
Tracks	8.0000	8.0000	8.0000	8.0000	8.0000
Pads	8.0000	8.0000	8.0000	8.0000	8.0000
Vias	8.0000	8.0000	8.0000	8.0000	8.0000
Shapes	8.0000	8.0000	8.0000	8.0000	8.0000
Text	8.0000	8.0000	8.0000	8.0000	8.0000
Board	50.0000	50.0000	50.0000	50.0000	50.0000

Value Key
 Inherited value
 Overridden Value

Export to CSV
 Import From CSV
 thou

Rule Level
 Design Net Match

Match Net 1 (Row)	Match Net 2 (Column)	Match Layer	Bi-Directional
[Power]	*	Top Electrical	Yes
[Power]	*	*	No
High_speed	*	*	Yes
*	*	*	Yes

Add...
 Edit...
 Delete
 Delete Unused
 Up
 Down

Press the **Add** button to add a rule. You need to define three wildcard strings which will match two sets of nets (or net classes) and layers. (Note: if they are not wildcard strings (containing '*' or '?'), then they will explicitly match a single net or layer). You can match a set of net class names (instead of net names) by enclosing the Match Net string in [square brackets].

The **Bi-Directional** flag indicates that the rule applies to the design types of either net (so the Pads to Tracks spacing will always be the same as the Tracks to Pads spacing). This does not have to be the case. For example [Power] nets might need a large Pads spacing against Tracks on all other nets (*), but a smaller value is needed for the reciprocal Tracks on [Power] to Pads on the other nets.

Once a rule has been defined, it is added to the **Net Match** grid. You can move a rule **Up** or **Down** the list. The rules are applied strictly in the order that they are defined in the grid, until the first one which matches. So if you define the first rule as * to * on layer *, which matches everything, then none of the subsequent rules would ever be applied.

If you select a row in the Net Match grid, then the spacing grid above will show the values which apply. Initially, all values are inherited from the design level (indicated by grey text). Typing values into the grid will set an override value for this rule (indicated by blue text). You can also delete an override value to turn it back to the inherited value.

If you have defined the rule as Bi-Directional, then the top right half of the grid is greyed, because those values are the same as in the lower half of the grid. In the case when *Bi-Directional* is disabled and the Match Net criteria would match either way round, then the larger value is used. Note that the net string * also matches with items not on a net. If you use this string, then the Text and Board values become available. So, for example, you can define a specific value between net *Gnd* and the Board.

PCB Specific Items in the PCB Technology File

Layers

The **Layers** dialog (on the **Settings** menu) is where the ‘structure’ of the board is defined. This will define electrical, non-electrical and documentation layers as well as construction layers.

Spacings and Clearances

The **Spacings** dialog (on the **Settings** menu) is used to define the clearances used for design rules checking. These clearances will be required by your PCB Manufacturer, they are a set of rules or tolerances to which they can make your board.

How layers and Spacings are used is discussed in the PCB Design section later on.

Changing Styles Globally and Locally

Styles can be changed in several ways:

For new designs, changing the style in the Technology File will affect all subsequent designs.

For existing designs, a style can be changed either by editing the style using the **Style** dialog from the **Settings** menu, or directly on an object. Editing a style on an item is done using the **Properties** option from the context menu or by using **Change Style** from the context menu. Alternatively, double-clicking on a component or text will automatically open the **Properties** dialog.

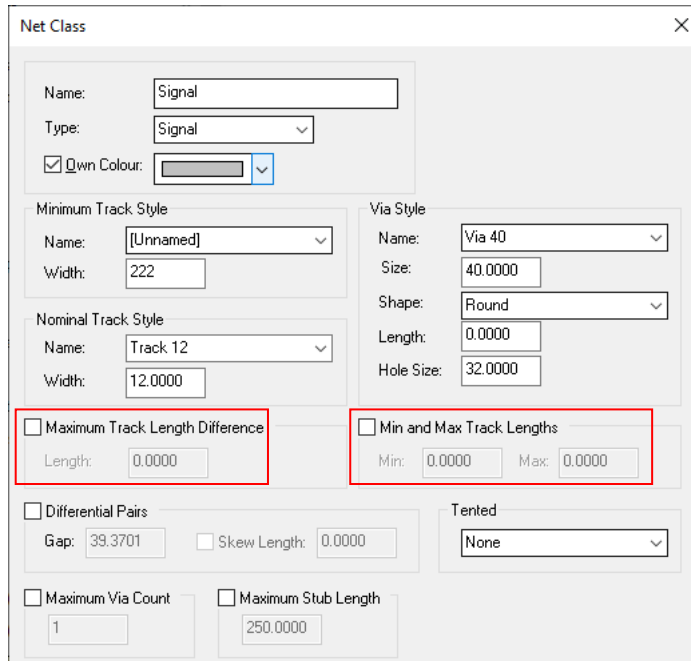
If you edit a style using **Properties** and give the changes a style name you are given the option of adding that named style to the design it will not be added to the Technology File, you must use the **Save To Technology File** option from the **File** menu to do this. If you change a style using the **Styles** dialog it will affect all objects in the design that use that style, i.e. the changes will apply globally to that design.

Track Length Rules on Net Classes

For advanced rules, you can add extra values on a net class:

- length difference value
- min length value
- max length value

You can optionally enable the **Maximum Track Length Difference** and **Min and Max Track Lengths** options within the net class.



Net Class

Name: Signal

Type: Signal

Own Colour: [Color]

Minimum Track Style

Name: [Unnamed]

Width: 222

Nominal Track Style

Name: Track 12

Width: 12.0000

Via Style

Name: Via 40

Size: 40.0000

Shape: Round

Length: 0.0000

Hole Size: 32.0000

Maximum Track Length Difference

Length: 0.0000

Min and Max Track Lengths

Min: 0.0000 Max: 0.0000

Differential Pairs

Gap: 39.3701 Skew Length: 0.0000

Tented: None

Maximum Via Count: 1

Maximum Stub Length: 250.0000

By adding these values and using the appropriate net class in your design, you can define a minimum and maximum length that the routed track must finish with. By adding a length difference value to the net class, you can then check that all the tracks using that net class will finish with their length within that tolerance.

These values are used by a **DRC** check (Net, Track Lengths), and a **Track Length Rules** report available from the **Output** menu and **Reports**.

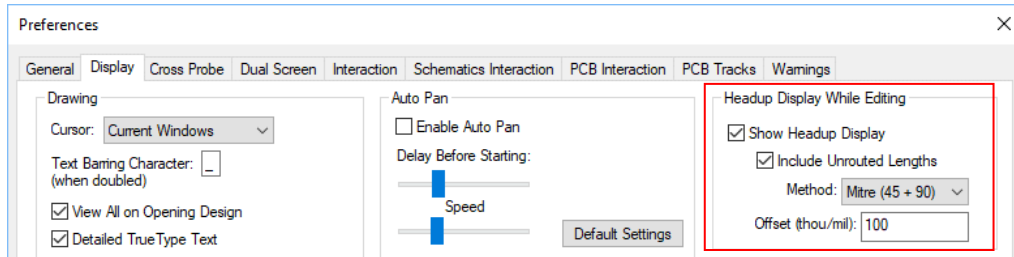
Headup Display

There is an interactive PCB tooltip based headup display to show track and net lengths as you add and edit track segments to the design.

This is available from the shortcut menu when using **Edit Track**, **Mitre Corner** and **Move** on track segments.

The display will also include rules from the **Net Class** if they are set up. The values in the headup display will change to red when the rule has not been met. There is also an option on the shortcut menu to reposition the headup display relative to the moving cursor.

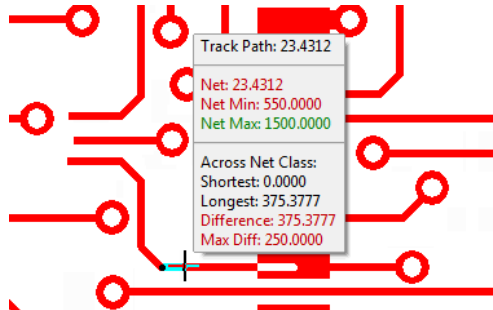
There are some options in the **Preferences** dialog and **Display** page to control how unrouted connections are treated when working out the track length of a net.



Using the Headup Display

Check the **Show Headup Display** box to show a tooltip based headup display of track and net length information when adding, moving or editing tracks (including Differential Pairs).

If the track being edited lies on a single path between two component pads, the names of the pads are included at the top of the headup display.



If **minimum** and **maximum track length rules** are defined on the **net class** of the track being edited, their values will be included in the display and will be drawn green if the net track length is within the defined limits. If the net track length is outside one of the limits it will be drawn with red text as well as the rule that failed.

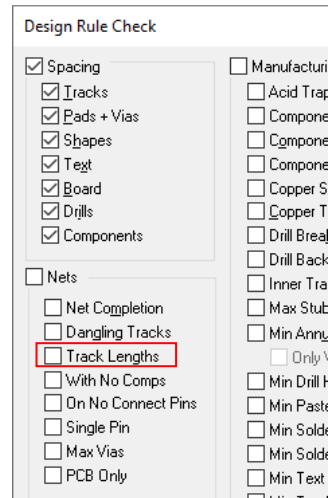
If the net being edited belongs to the same net class as other nets, and the **net class** has a **Maximum Track Length Difference** rule defined, the shortest and longest net lengths will be shown. If they are different, the difference between them will also be included along with the maximum rule limit. If the actual difference in net track lengths is greater than the limit it will be drawn with red text as well as the rule value.

Check the **Include Unrouted Lengths** box to include unrouted connections in the overall net track length. Choose the **Method** to be used when calculating the estimated track length for each unrouted connection. **Manhattan** is the minimum distance from one end of the connection to the other using just horizontal and vertical lines. **Mitre** is the minimum distance from one end of the connection to the other using horizontal, vertical and 45 degree lines. **Point to Point** is the length of the single straight line between the two ends.

You can also specify an **offset** here, which is the distance between your current cursor position and where the headup display is drawn. You can change the value here to anything between 0 (no offset) and 4000 (4 inches).

DRC Option

Within the DRC option, the **Track Length** check calculates length of all tracks in the class, finds the median value, then shows you any net that is too far either side of this value. The same check box enables the **Min/Max** check to simply verifies track length of each net against specified values, either value can be set to zero to 'ignore' that length. Nets with no tracks are always flagged as errors.



DRC Report

When the DRC option reports **Track Length Diff**s errors, the report typically might look like this:

```
Design Rule Check Report
-----
Report File      : c:\temp\ProDesign.txt
Report Written  : Thursday, July 04, 2022
Design Path     : c:\design\Job77Design_A.pcb
Design Title    : PSU Board
Created        : 22/04/2021 1:28:41 PM
Last Saved     : 04/07/2022 10:47:57 AM
Editing Time   : 6470 min
Units          : thou (precision 4)
```

Results

```
-----
```

```
Track length differs from other nets of same class : Net SIG009
too long, length is 602.7795 thou but needs to be no more than
600.0000 thou from rules in net class HS1
```

```
Net SIG009 too long, length is 602.7795 thou but needs to be no
more than 600.0000 thou from rules in net class HS1.
```

```
Number of errors found : 1
```

Track Length Rules

When run from the **Output** menu and **Report** option, an example **Track Length** report might look like this:

```
Track Length Rules
-----
Report File      : c:\temp\ProDesign.txt
Report Written  : Thursday, July 04, 2022
```

```

Design Path      : c:\temp\ProDesign.pcb
Design Title    : PSU Board
Created         : 22/04/2021 1:28:41 PM
Last Saved     : 04/07/2022 10:47:57 AM
Editing Time   : 473 min
Units          : thou (precision 0)

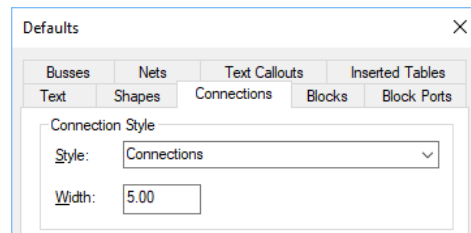
```

Net Class	Net	Length	Range from	to	Diff	Min	Max
HS1	FS1	602	590	605	15	580	600
	FS2	592	590	605	15	580	600
Power	-	-	-	-	-	-	-
Signal	-	-	-	-	-	-	-

Defaults

The **Defaults** entry on the **Settings** menu allows you to set up styles that will be used when first starting a design and when using design items such as text or tracks.

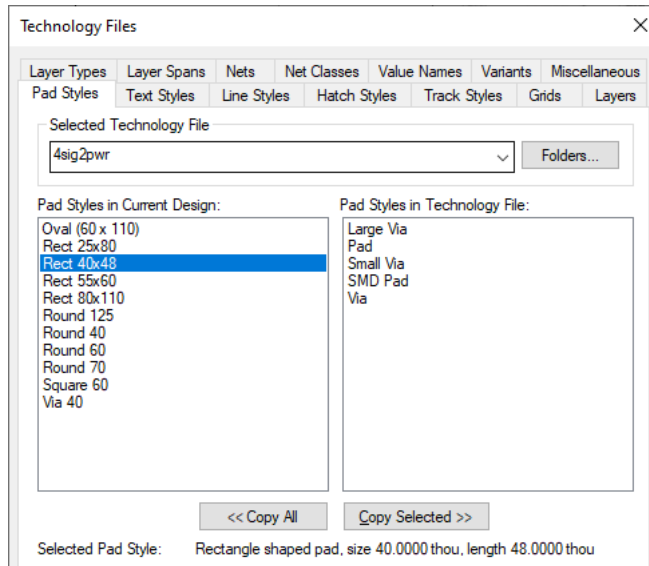
For example, after loading a few components to the **Schematic** and starting to add connections between them, the initial default connection width of those connections will be defined by the entry under **Defaults** and **Net Class** and **Defaults** and **Connection**. A new Pad added would only use the **Defaults** and **Pad** entry.



It will not be necessary if you normally translate from a Schematic. In this instance, the track styles used will be from the **Default Net Classes**. They can be changed after translation if required.

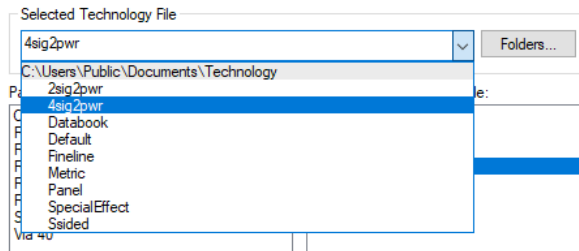
Loading and Reusing Technology Files

If a style has been added to a design and it is required for use in another Technology File for subsequent use in other designs - or if a Technology File contains a style that is required in a design, the **Technology Files** option on the **Settings** dialog provides a method of bi-directional transfer.

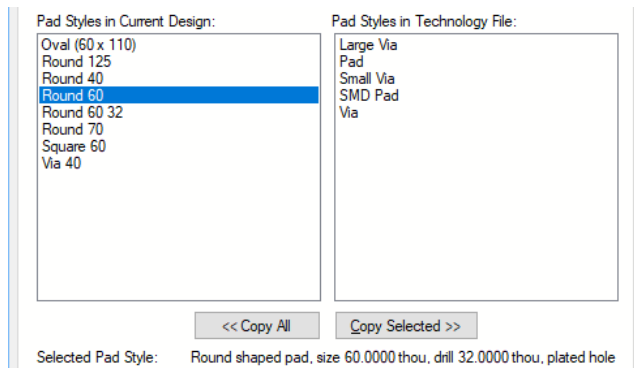


► **To transfer Technology items**

1. Select the required Technology file from the drop-down list next to **Name:**



2. Select the tab containing the required style type.
3. The main box on the left is the style in the **Current Design** and the box on the right is the contents of the **Technology File**.



4. Highlight the style or group of styles to be transferred by selecting them, then press the **<<Copy All** or **Copy Selected>>** buttons, depending on your selection.

5. The style(s) will be copied and shown in the other box.
6. If the Technology file has been changed you will be asked to confirm that the changes should be saved.

Exporting & Importing CSV Files

All tabs of the **Design Technology** dialog (except for **Wires** and **Rules**) have buttons to **Export To/Import From CSV** format files. The exported files can be easily edited in Excel for example, then imported back in with the new values.

During import, checks are made to ensure the right number of fields are imported and values are legal.

	Tracks	Pads	Vias	Shapes	Text
Tracks	8.0000	8.0000	8.0000	8.0000	8.0000
Pads	8.0000	8.0000	8.0000	8.0000	8.0000
Vias	8.0000	8.0000	8.0000	8.0000	8.0000
Shapes	8.0000	8.0000	8.0000	8.0000	8.0000
Text	8.0000	8.0000	8.0000	8.0000	8.0000
Board	50.0000	50.0000	50.0000	50.0000	50.0000

Export to CSV

Import From CSV

thou

Once exported into Excel, the file looks like this:

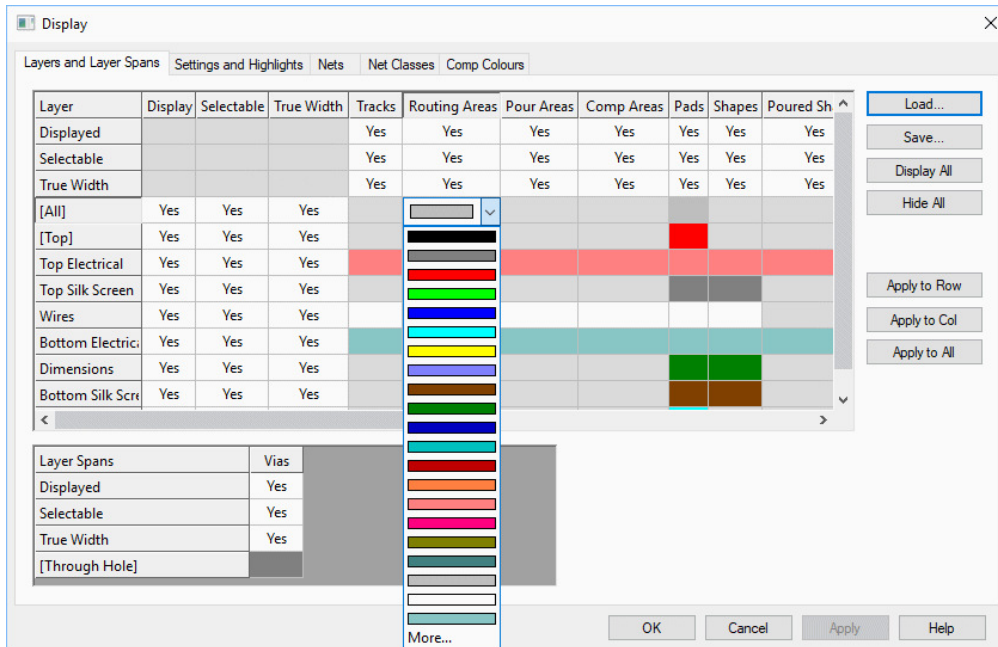
	A	B	C	D	E	F
1	Object Type	Tracks	Pads	Vias	Shapes	Text
2	Tracks	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou
3	Pads	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou
4	Vias	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou
5	Shapes	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou
6	Text	10.0000thou	10.0000thou	10.0000thou	10.0000thou	10.0000thou
7	Board	75.0000thou	75.0000thou	75.0000thou	75.0000thou	75.0000thou

All values are written in design units. If you change the Units in the CSV file for one of the values, to 1mm say, the imported value will be converted to the current design units.

The CSV file can also be edited in a text editor but care must be taken to retain the correct file formatting.

Changing Colours

Using the **Colours** dialog, you can customise the colours of almost all the design items. Although the PCB dialog is shown below, the Schematic one is more simplified as it has no layer structure to it.



To change the colour, double-click on the colour cell in the grid and pick a colour from the list. Each of the colour drop-down lists has a **More...** entry at the bottom. When selected this displays the standard Windows Colour selection dialog to allow a custom colour to be defined.

Multiple custom colours may be defined and will be retained in the Windows colours dialog.

Save & Load Colour Files

Colour settings can be saved to disk and later re-loaded, allowing you to set up commonly used groups of settings that you can then use to switch quickly between different ‘views’ of the design.

You might, for example, set up two colour files that only show top and bottom layers respectively, and use these to switch between a top and bottom view of your board.

You may also use the **Colour Files** bar to quickly switch between colour files.

To save the colour settings from the **Colours** dialog, click the **Save** button. When the file browser dialog opens, enter a name for the colour file and press **Save**.

To load a previously saved colour file, click the **Load** button and use the file browser choose the correct file. When the file is loaded, the design will be redrawn using the new colour settings, and the colours and visibility shown on the dialog will be updated to match.

Selectable Items

For most design items you can set how **selectable** items are. This is useful for restricting what may be picked to the particular type of items being worked on while still displaying other relevant parts of the design. It is available in PCB, Schematics, PCB Symbol and Schematics Symbol designs.

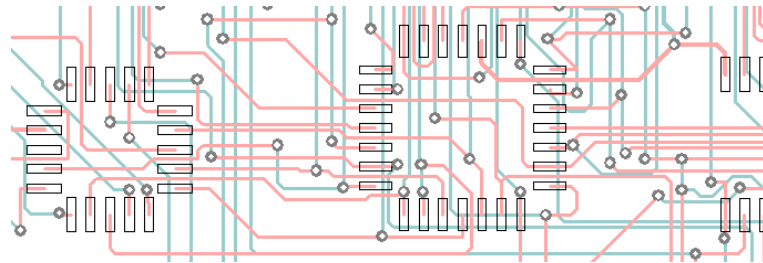
On the **Layered and Layer Spans** tab, an additional **Selectable** row and column allow selectability to be set for each item type or entire layers respectively. Each entry in these, like the existing 'Visible' row and column may be toggled between **Yes** and **No** as shown below:

Display						
Layers and Layer Spans Settings and Highlights Nets Net Classes Comp Colours						
Layer	Display	Selectable	True Width	Tracks	Routing Areas	Pour A
Displayed				Yes	Yes	Yes
Selectable				Yes	Yes	Yes
True Width				Yes	Yes	Yes
[All]	Yes	Yes	Yes			
[Top]	Yes	Yes	Yes			
Top Electrical	Yes	Yes	Yes			
Top Silk Screen	Yes	Yes	Yes			
Wires	Yes	Yes	Yes			
Bottom Electrici	Yes	Yes	Yes			

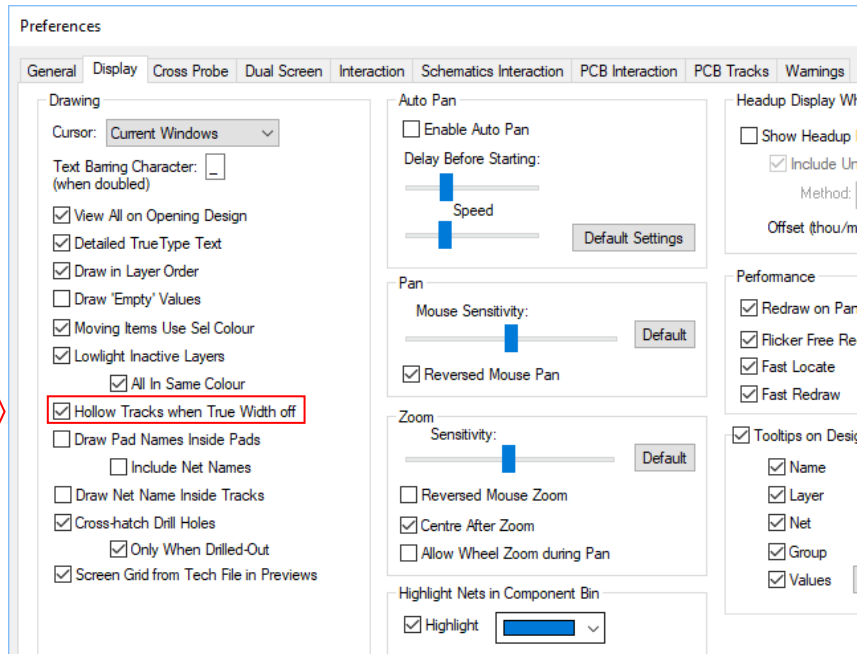
True Width display status in PCB

To aid the de-cluttering of PCB designs, you can switch on and off the true width of individual items. When switched off, the item is shown as a centreline object. The exception to this are **Pads** and **Vias** which are displayed as their outer edge (as an outline) and the pad diameter inside.

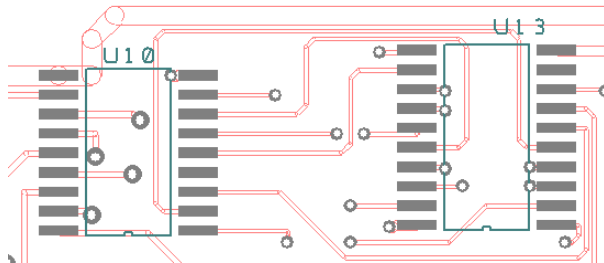
There are two extra fields next to **Display** and **Selected** named **True Width**. You can switch the true width of items on and off in columns for each layer or in rows for each item. For example, you could switch the true width off of all items on the top layer to make the lower layers more prominent. Likewise, you could switch the true width off of all pads in the design, making the tracks more prominent while routing.



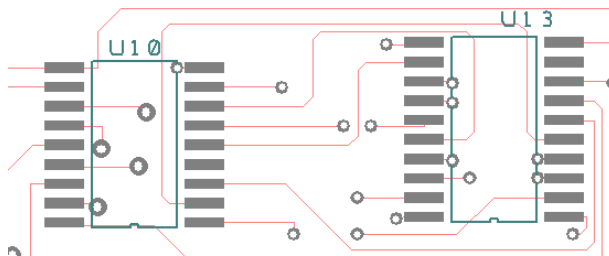
With **True Width** switched off in the **Colours** dialog, you can also display items as 'hollow' rather than centre line. To enable this feature, check the **Hollow Tracks when True Width off** box in **Preferences** and **Display**.



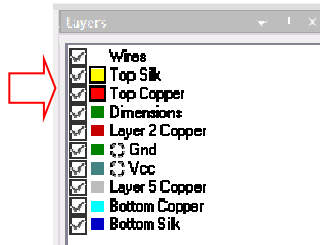
Tracks appear as 'hollow' when enabled:



In comparison, without this option selected, tracks appear as centre-line:



You can also switch the True width on/off quickly by pressing the **Ctrl** key and selecting the layer name in the **Layers Bar**.

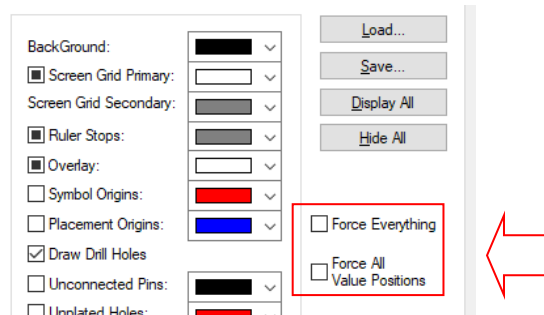


Force All Colours On

Switches are available in the **Colours** dialog, under **Settings & Highlights**:

- **Force Everything** on
- **Force All Value Positions** on

These switches work at a high level so they override all other switches in the Colours dialog. However, once used and set unchecked again, the previous colour settings are totally restored.



Use the **Force Everything** check box to force everything on so that it is visible, true size and selectable. This will be everything except Values which has its own check box.

Use the **Force All Value Positions** check box to force all Values in the design to be displayed.

Working with Values

What are Values?

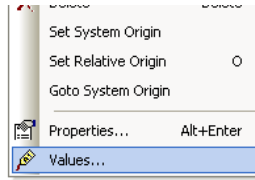
A **value** is an attribute that can be attached to a component to enhance the information that this item contains. The value is a **Name** that contains a **text** string (a piece of information).

You can use values on components for example, the value name could be **Height** with a value of 2mm, or **Tolerance** with a value of 1%.

Adding or Editing Values

Values can be added or edited using the **Properties** dialog of the component found on the context menu and the main **Edit** menu, the **Values** dialog of a component on the context

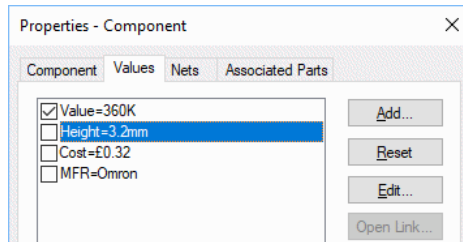
menu, and from the **Component Values** editor on the **Settings** menu. For a library, the dialog is accessed by clicking on the **Values** button on the **Components** page of the **Libraries** dialog.



Using the Properties dialog

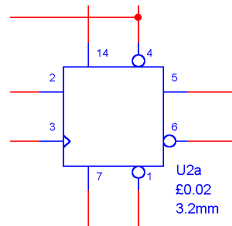
The **Component Properties** dialog has a tab for adding and editing **Component Values**.

Once a value has been added using the **Add** dialog, the window will indicate three pieces of information. The check box (selected by you) will indicate whether that value is **displayed** in the design or not (a value can be added but not displayed). The next item is the **Value** name, for example **Height**, followed by the value itself, in this example **3.2mm**



When you click the **OK** button, the values in this dialog are applied to the component(s) in the design.

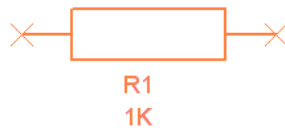
The **Values** dialog is much the same as the **Properties** dialog but without the extra **Component** tab. When displayed in the design the values would look something like this:



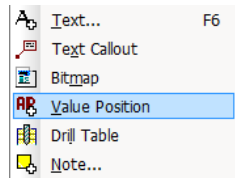
The position of the values is initially relative to the component name but these can be moved independently as required.

► To position and add values

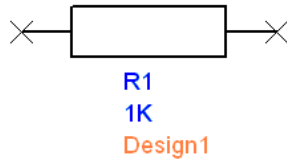
1. Select the component to add the Value to.



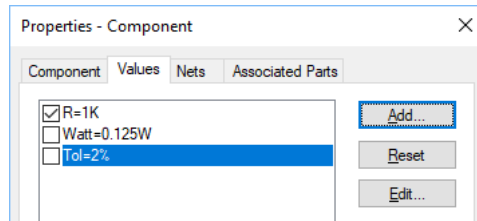
2. Values must already exist on the selected Component. Use **Properties** or **Component Values** editor first or add them to components in the component library.
3. Select the **Value Position** option from the **Add** menu.



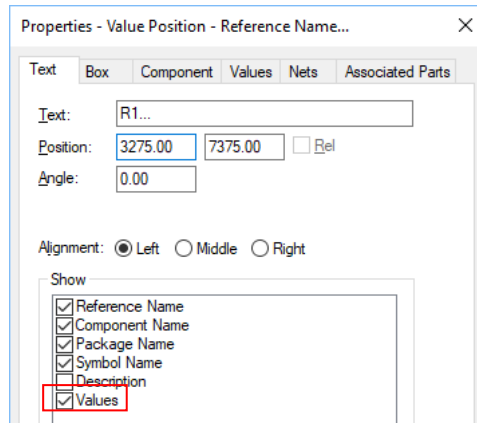
- Position the default name – Design1 (Design1 will be the design name before it has been saved).



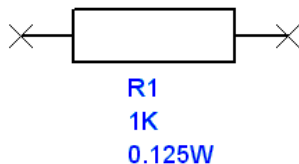
- Once positioned, select the value name and right click (you can also double-click on the name).
- From the context menu, select **Properties**.
- From the **Values** tab, switch on and off the values you require to view in your design.



- To switch all **Values** on or off, you can select **Values** to display from the **Text** tab.



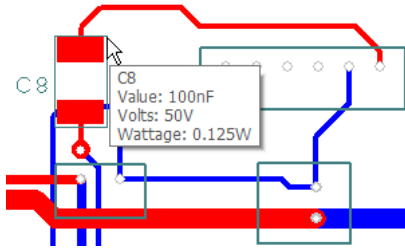
- Click **OK** to exit. The result might look something like this.



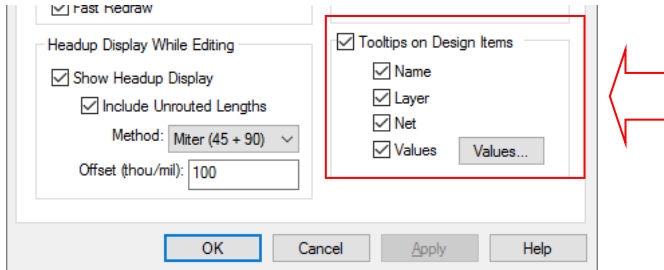
10. Use the **Add Value Positions** option to show more values on the component.

Tooltips Displaying Values

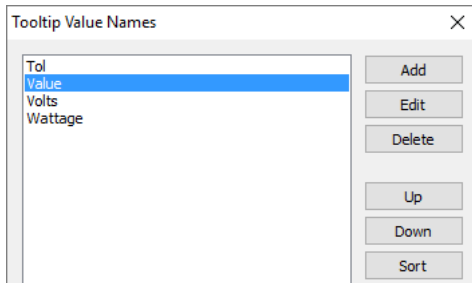
If you prefer not to show Values names in the design, **Design Tooltips** can display user-specified Values (Attributes) when hovering over components. The **Value Names** to display are configured on the **Display** page of the **Preferences** dialog.



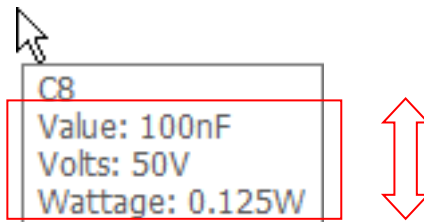
A **Values** check box and button appear on the **Preferences** dialog:



From the **Values** button, you can select the **Value Names** to display on the tooltip.

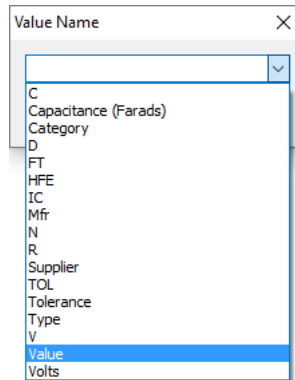


The **Up** and **Down** buttons will allow you to choose the order in which the values appear on the tooltip.



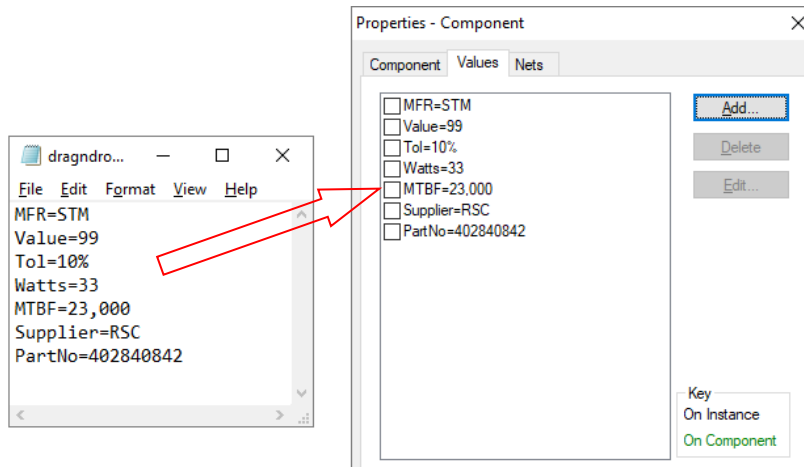
Pressing the **Sort** button will sort the list in alphabetical name order if required.

Using the **Add** button, choose the value required from the drop down list or type the value name you wish to use.



Drag and Drop Values

You can drag-and-drop one or more text files, or a block of text from a text application such as Word or Notepad2 to add Values to Components. This feature can be used from the **Values** tab on the **Properties** dialog for **Components** and the **Values** page in the **Component Editor** (available from **Libraries**).



It is hard-wired to expect the attribute text in **name=value** form, using = as the separator. Any line of text that does not fit with this form will be ignored. Any name that already exists, will have the value replaced without prompt or warning. The syntax will accept the name=value format with or without spaces but nothing else.

Using Drag & Drop Values onto a component

There are two methods to drag and drop Values onto a Component:

1. Using the Windows Explorer, you can select a text file containing legally formatted values text and drop it onto the **Values** tab of either the **Component edit** dialog or the **Component Properties** dialog, or;

2. Select portions of the required text from the file, again, formatted using the expected syntax, and **drag** it out of the text file and onto the Values page for the desired item, such as this below:

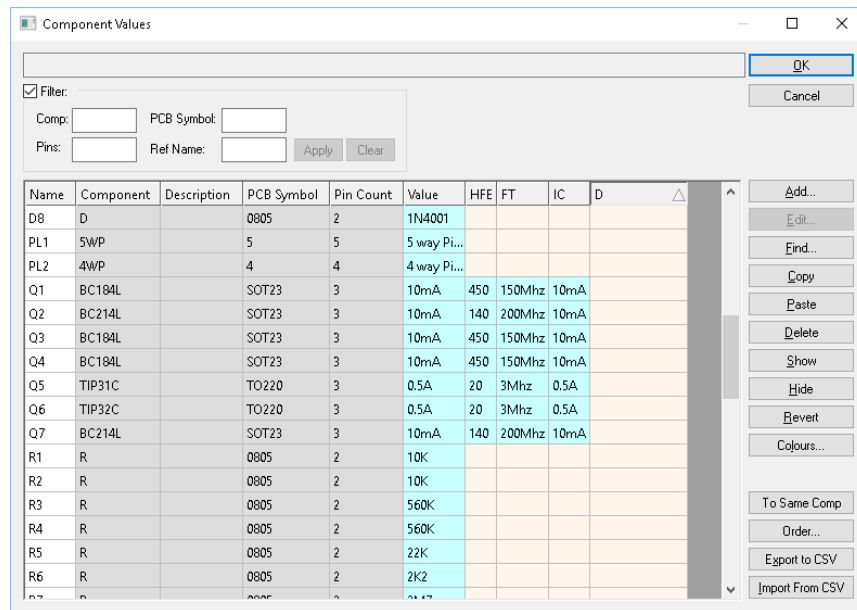
Note: **Copy Paste does not work** for this function, the text must first be selected, then **dragged** to the appropriate dialog.

Also note that the standard **Windows Notepad** program will **not** allow you to drag and drop individual text items due to the technical limitation of this basic program. Other enhanced text programs such as **MS Word** or **Notepad2** etc. do however allow this functionality.

Using the Component Values Editor

The **Component Values** editor on the **Settings** menu displays a list of the components, and a table or 'grid' of all the values on those components. For a design, the name shown on each row is the reference name of the component; for a library, it is the component name itself.

As well as the grid, there are a set of buttons down the right-hand side of the dialog that allow you to modify the values.



The cells in the table are colour-coded to show you the status of each value type. Four different colours are used, described below. The default colour of each is also shown, although you can change the colours used by clicking the **Colours** button on the dialog.

- **Undefined (cream)** - this value is not present on this component
- **Empty (pale green)** - this value is defined, but has no text
- **Hidden (pale blue)** - the value is defined, and has text, but is not displayed in the design
- **Visible (white)** - this value is defined, and has text, and is visible in the design

As well as changing the colours, you can also resize the dialog by dragging the corners or sides. The new size is remembered so the dialog comes back the same size next time you open it

To change the actual component values, you can use the buttons down the side of the dialog, click the right mouse button in the table and use the pop-up menu, or use the keyboard to type text directly into the grid cells.

Any changes are done to the selected part of the table, you will need to select the required value or values first and then click the appropriate button. For example, to set all the values for a given component to be visible, click the left mouse button in the first 'cell' of the row for that component, then drag across to the last cell and release the mouse button.

C191	100n	0805	X7R
C192	100n	0805	X7R
C193	100n	0805	X7R

Then click the **Show** button to set all those values visible.

C191	100n	0805	X7R
C192	100n	0805	X7R
C193	100n	0805	X7R

The cells will change colour accordingly.

Chapter 7. Creating Schematic Designs

Overview and Getting Started

Before we get started, you should be aware that a PCB design can be created straight into PCB design editor without creating the Schematic design first. You can also use a netlist from some other system and load this into the PCB design editor.

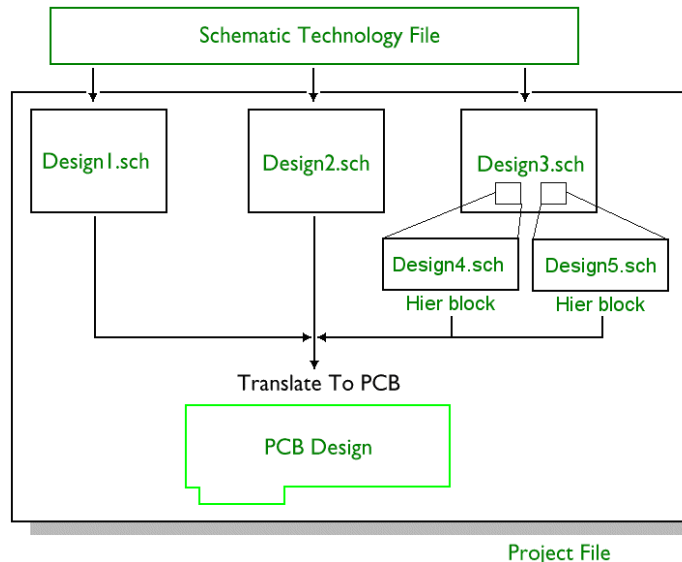
The working concept of the Schematic design editor and the PCB design editor are fundamentally the same. There are some small technical differences in the two editors; Schematic components are connected together by lines known as connections, rather than connections/copper tracks as they are in PCB, and there are no equivalents to PCB pad stacks in Schematics. Obviously, the PCB design contains 'layered' items whereas the Schematic is 'flat'.

The difference between Tracks and Connections is that Tracks are physical connections made in copper and used in the PCB design editor, whereas connections are logical connections between items only. The confusing part is that tracks also have underlying connections (the 'rats nest') until converted to copper tracking.

Starting a New Schematic Design

To use a Project file or not?

When starting a new Schematic, you may need to think about whether you need to use a Project or not. The decision for using a Project is governed mainly by how many Schematic sheets you intend creating and whether it has hierarchical blocks or not. If there is more than one sheet, then create a Project first and add all the design sheets to this Project. If there is one sheet, and only ever will be one sheet, you probably won't need a Project. A Project file enables you to maintain unique component and net names throughout all of the Schematic sheets without any duplications or errors.



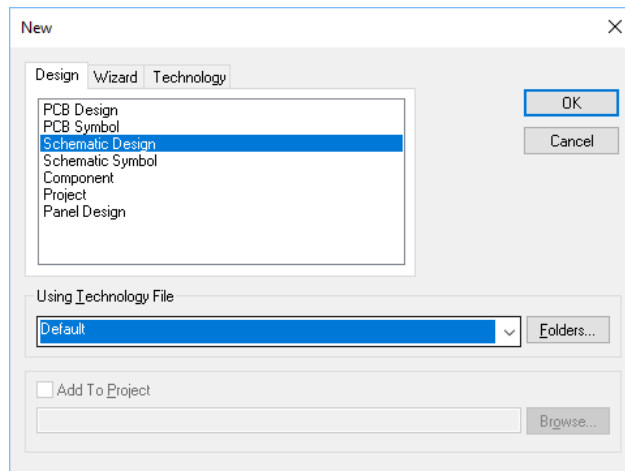
The Project file may contain many Schematic sheets but only one PCB design. Technology files are independent of the Project file and are used with the design files. A schematic design may also contain hierarchical blocks of 'lower' level functionality, these would also be contained within a Project file.

Starting the design

You can start your design using one of two methods; using the **New, Schematic Design** dialog or **New, Wizard, Schematic Design**. The **New** dialog for both options is located on the **File** menu.

Using the New dialog

The **New, Schematic Design** dialog will start a new **Schematic** design using just the **Technology File**. You can also elect to use no Technology File, [None], but we don't recommend this.



There are two Technology files supplied for Schematic designs, **Default.stf** that uses a white background and **Default (Black).stf** that uses a black background. You can of course, change the colours as required from within the Schematic editor.

You can also elect to add the new design to the Project file using the **Add To Project** check box and choosing its name.

If the design is new and no Project file exists, use the **Browse** button and locate the folder required. Now type a **Project** name and a new file will be created when you click **OK**. If one exists, select it by name.

Click the **OK** button to start the Schematic design editor.

Design Options for Schematics

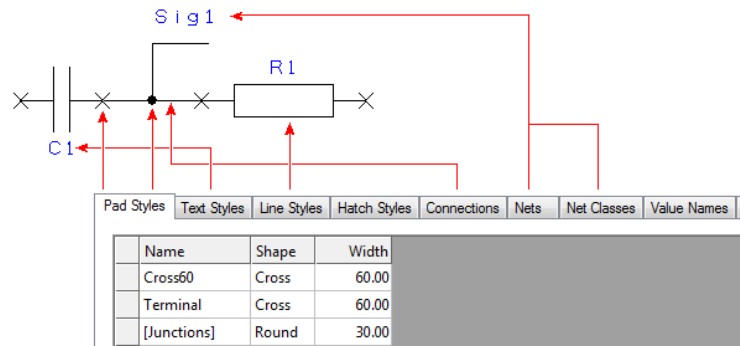
If you start a new Schematic design using the standard settings and one of the supplied Technology files, there isn't anything else essential to set up in order to get started. However, you can edit the user defined options in the **Preferences** dialog, the **Defaults**, the **Units** dialog and the **Folders** dialog. These are discussed in detail as used throughout this guide.

Setting Up a Schematic Technology

Styles

The diagram below shows the relationship between the major design items **Styles**.

The styles used in this dialog should be as required for your drawings and for any company standards that may exist. When choosing styles, consider how the drawing may be finally printed or plotted. The thicknesses used will have an impact on the look of the drawing.



Pads styles are used to define look of component terminals and junctions, where two connections are truly connected. Connections may overlap each other but will not be electrically connected.

Text is used to contain the styles of text items within the design.

Lines styles are used for the shapes and outlines of the symbols. They can also be used to define any detail within the drawing such as the drawing border.

Connections styles are used to define the thickness of 'real' connections within the design. These are not the same as Line Styles and should not be confused.

Net Names and Net Classes

When creating a new design, you should ensure that each **Net Name** on the **Nets** dialog has a **Net Class** associated with it. It is this association that will give you your track thickness in the PCB design. The Net Class dialog in the PCB design also has track and via styles associated with it.

When translating from the Schematic to the PCB design (using the **Translate To PCB** option), the Net Class name used on a net is also transferred, this then picks up the physical characteristics defined in the PCB Net Class section.

Design Technology										
		Pad Styles	Text Styles	Line Styles	Hatch Styles	Connections	Nets	Net Classes	Value Names	Variants
	Name	Class	Own Colour	Colour						
X	DRIVE	Signal	No							
X	Feedback	Signal	No							
X	GND	Power	No							
X	High_spe	Signal	No							
X	Low_spee	Signal	No							
X	V+	Power	No							
X	V-	Power	No							
X	N01	Signal	No							
X	N02	Signal	No							
X	N04	Signal	No							

Without a net class name, the net class used will be that of the **Defaults, Net Class** dialog. Net Names and Net Class names are described in more detail in the previous chapter.

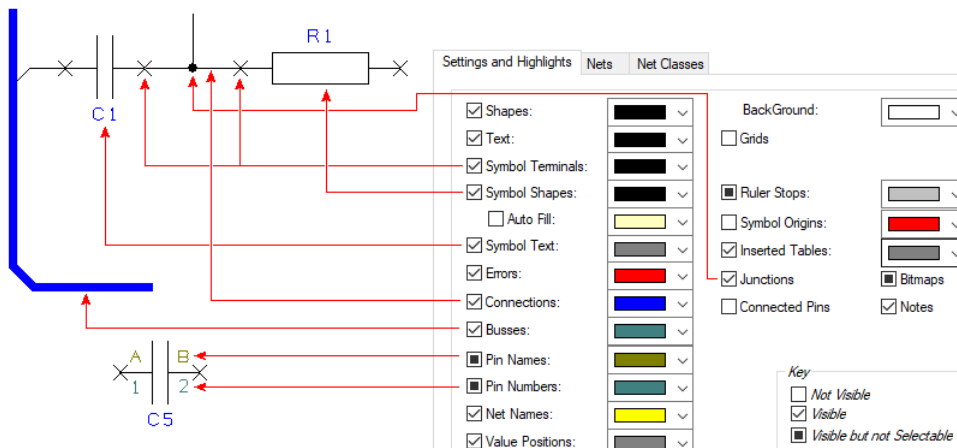
Drawing Borders

As well as **Styles** and **Colours**, Technology files can also contain drawing blank information and text. These are added as shapes and text from the **Add** menu. Drawing blanks that are created using a component cannot be saved in the Technology file. *See below.*

Changing the Colours of the Schematic

All of the colours within a Schematic design can be changed using the **Colours** dialog. This dialog has been discussed in the previous chapter.

The diagram below shows how the design is constructed using different colours for different design items.



Use the small arrow on the right side of the list box to drop down a colour palette when a colour change is required.



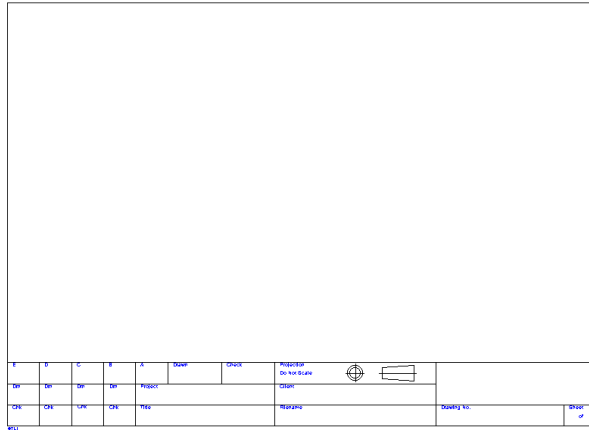
Some of the switches are slightly less obvious than others.

Connected Pins - Use the **Connected Pins** option to control display on component pins. Normally you would have the **Symbol Terminals** switch on and the **Connected Pins** switch off so that you only see the cross on unconnected pins. This makes it easier when adding connections to the schematic to see which pins are left to be connected.

The **Dangling Connections** option allows you to highlight unfinished connections. With the **Show All Dangling Connections** option checked, any connection which is not connected at one end is considered dangling. With the option unchecked, only such connections which are the only instance of that net on the current sheet are considered dangling.

Adding Drawing Borders

There are two methods of adding drawing borders to your Schematic design; using a component, or by drawing in the border using the shape drawing tools.



You will find a number of standard drawing borders in the component library, **schema**. See the next section in this chapter on how to add components to the design.

If you wish to add a drawing border using shapes, add the shapes to the design using the **Add Shape options** from the **Add** menu. The drawing border can be further enhanced by using free text from the **Add Text** option on the **Add** menu. Both Add Shape and Add Text are covered later on in more detail in this chapter.

Adding Components

Once components have been created in the appropriate library, they can be added to the design using the **Add Component** option from the **Add** menu or by using the **Add Component** icon on the **Schematic Design** toolbar.

During the Add Component process, you may decide to use the **Component Bin**, you don't have to but it does help simplify the design area by keeping components out of the way until needed.

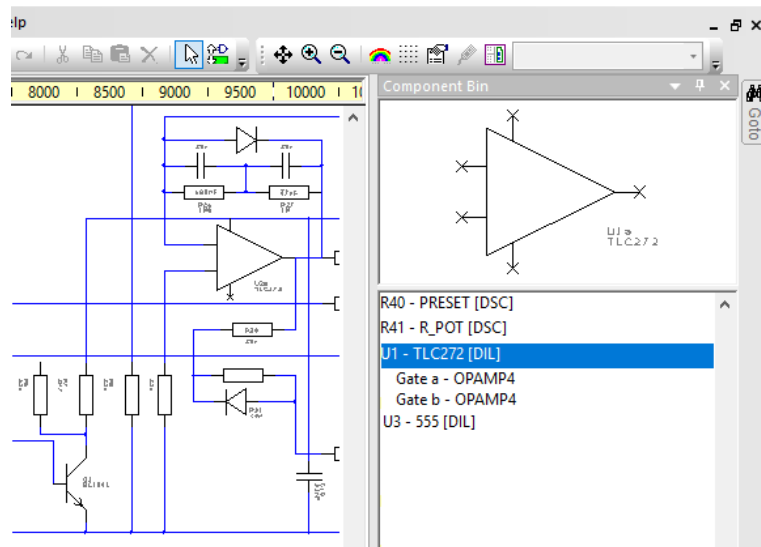
Using the Component Bin

The **Component Bin** bar has been described in detail in the previous chapter.

The component bin provides you with an off-screen location for holding components before placing them on the design, it is also used for holding unused gates on components that have more than one gate (called heterogeneous components).

Schematic components can be added directly to the component bin using Add Component and interactively by dragging components from the design into the bin.

When using the component bin, if you have components that are connected, if you drag them into the bin the connectivity will be lost when you bring them back out again. Of course, if you do this by mistake, you can use Undo to recover them.



Components placed in the bin will also be transferred to the PCB design during **Translate To PCB**.

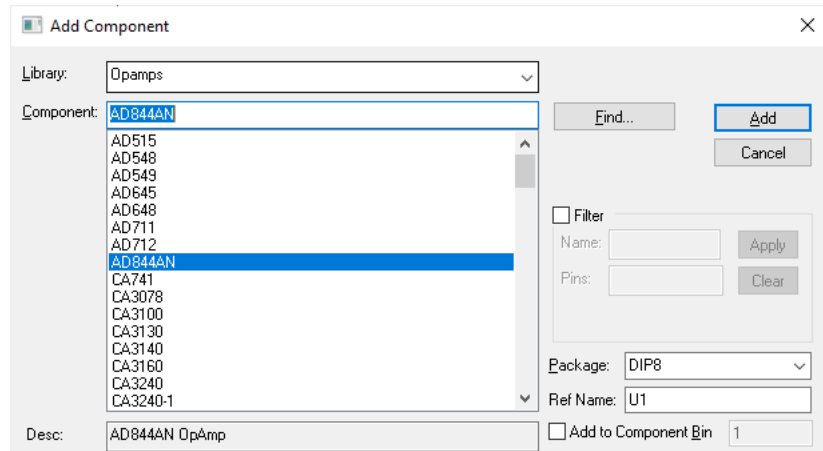
Using the Add Component dialog

Once the design preferences have been set and the drawing border added, the Schematic editor is ready for the design to be created.

One of the first major processes is the addition of components into the design; you are provided with the **Add Component** option for this purpose.

Components and Symbol contents are created and saved into corresponding libraries. The component entry contains a Component name, a PCB Symbol (Footprint) (optional if translate to PCB isn't required) and a corresponding Schematic Symbol or multiple symbols if the component represents a multi-gated component.

The **Add Component** dialog allows you to find and select components from your component libraries.



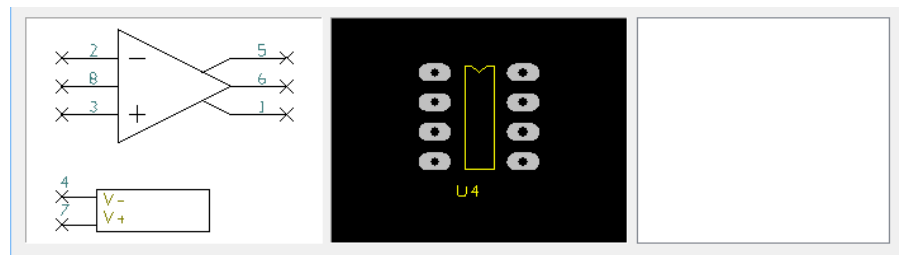
The dialog allows you to choose the library, find a component, choose the component, to preview it and to add it to the design or component bin.

The **Library** list allows you to choose the library to select components from. From the drop down list, you can also choose [All Libraries]. This will display the components from all of the libraries found in the search path defined in the **Library Manager** under **Folders**. The [Recent] list will display all component names that have been recently added to your design.

Once a library has been selected the **Component** list will update with all the components in the selected library. You can locate specific components by using the **Find** button and the options within this. The Find dialog allows you to search libraries and enter names based on wildcard names. This dialog is described in more detail.

The **Add** button will place it on the end of your cursor ready for positioning in the design. You will notice it will snap to the screen grid while placing. You can adjust the snap mode using the <H> key and by selecting one of the snap modes from the context menu. Alternatively, you can use the **Add to Component Bin** check box to send the selected component directly to the component bin. The **Count** entry will allow you to specify multiple copies of the selected component.

The **Preview** button can be used to show you the component before it is placed on the design, it also shows you a preview of the PCB Symbol. Once you create your own library you will probably not use this unless you need a reminder of what they look like.



Using the Find dialog

When selected, **Find** displays a dialog that allows an item name to be found in the libraries. It then searches the libraries for the given Component/Symbol and selects the appropriate library and item. It searches the current library first, then all the libraries in the library directory. This has been specified in the **Preferences** dialog under **Directories**.

Matching on the library item **Name** can be done in three different ways:

- **Is Exactly:** only matches items with exactly the name specified
- **Starts With:** only matches items that start with the given text
- **Contains:** only matches items that have the given text anywhere in their name

The **Number of terminals/pads** field allows you to limit the search to match only those library items with the specified number of pins.

For the **Value** field, when searching for components, you can also choose to search for items that have a particular value. Leave the 'is' field blank to match all items that have this value regardless of its actual setting, or enter the desired setting to search for a specific value.

On pressing **Find**, there may be a delay whilst any library index files are updated. Having done this, the libraries are searched for matching items. Any matches found are listed on the dialog, showing their name and library file and for components their 'package' as well.

To select the desired item, simply click on it in the list. As you select the item, the corresponding 'master' dialog (Libraries, Add Component, etc.) will update its details and preview to show you the selected item. To close the Find dialog, either click on **Close** or double-click the required library item in the list.

Value

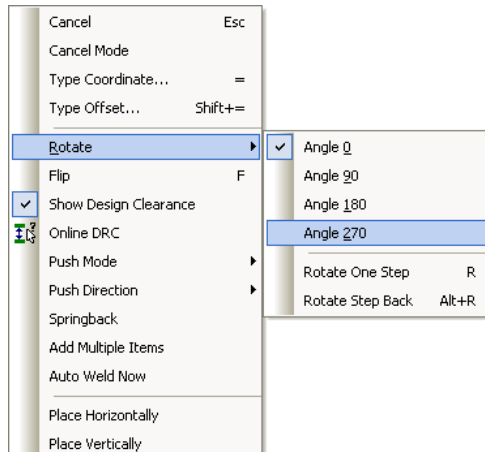
When searching for components, you can also choose to search for items that have a particular value or values. If you specify more than one value, only those components that match all the value criteria will be displayed. In other words, the value checks are combined with 'and' and not 'or'.

If you want to match items that have a particular value regardless of its actual text setting, set the 'operator' to 'Contains' and leave the value text field (the last field on the line) blank.

Interactively positioning components

Once the first component is positioned in the design, press the **Esc** key to cancel the addition of the next Component. Each time you add a component, notice that the component reference **Name** has incremented by one to the next number available automatically. You can overwrite this name and name stem if you wish but you will not be allowed duplicate component names on the same design, you will be prompted with a message box informing you of this.

While adding the component you can use the context menu to manipulate the component.



From this dialog you can **Flip** and **Rotate** the component to give it a more precise position. These options are also available by using shortcut keys.

The Styles used

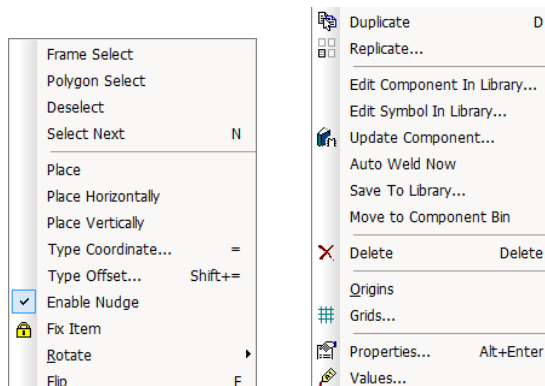
The values in the **styles** used for text, pads, terminals and shapes in the symbols from the library are retained when adding an instance of the component to a design. In this way the component will look exactly like it does in the library. This is managed by copying the styles from the symbol into the design. If the style name is already in the design, but contains different values, a new style name will be used.

Introduction to manipulating Components in the design

A component may be selected by clicking on any part of the outline or by pressing **<Shift>** and clicking on any part of the component. If a placement reference or any pin names or numbers are inadvertently selected, an attempt to drag the component will move the text instead. If this is a problem, use **<Shift>-click** to select the whole item.

Moving is done by dragging and dropping. Snap modes are adhered to at all times but can be changed by using the snap mode shortcut **<H>**.

Once a component is selected, there are several entries available on the context menu:



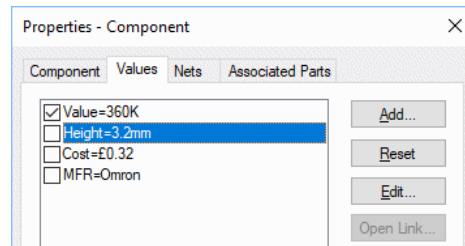
To rotate the component, either right click and select **Rotate**, or use the shortcut key <R>. From this menu you can also choose an alternative angle for rotation other than the default angle set for the single step rotation command.

Flip (shortcut <F>) will mirror the component. On the odd occasion that this is required in a Schematic design it will work by mirroring around the component origin. This has been discussed in detail in the previous chapter.

Each component in the design of the same type is referenced from the first one added. This is a copy of the item in the library that makes it independent of any changes to the source library. The Update Component function is provided to force a new definition into a design from the library. This intelligently replaces the component in the design with the new one from the library without affecting the connectivity.

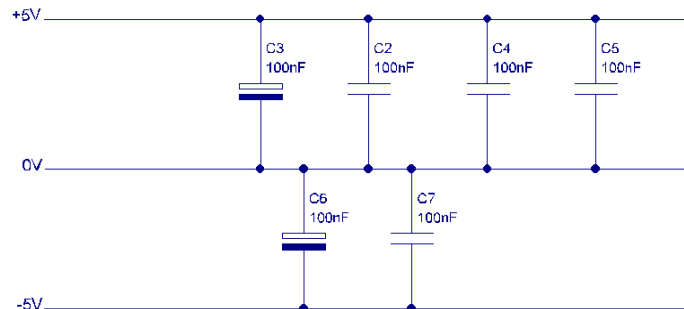
With connected components, you may also wish to ‘unconnect’ them, to change the component perhaps. For this you can use the **Disconnect From Net** option. This will keep the component in the same location but will disconnect all connections attached to it.

The values associated with a component have a variety of uses. Firstly, they can obviously be used to associate a supplementary label with the placement reference. As this is most commonly a value, it gives the function its name. Secondly, it provides a convenient way of specifying entries for export to parts lists, etc. These can be accessed through the **Properties** dialog or the **Values** option, both on the context menu.



Using Decoupling Capacitors

Decoupling capacitors, or by-pass capacitors as they are also known are usually used adjacent to ICs to ‘decouple’ the voltage rails to that device. These are usually shown as a ‘string’ of capacitors between the corresponding voltage rails without being shown in the proximity to the device they are intended for. In most cases the relationship between decoupling capacitors and the ICs is only that each IC is required to have a value (10pf for example) capacitor. In instances where there is a close relationship between the two, you should position the capacitor next to the device.

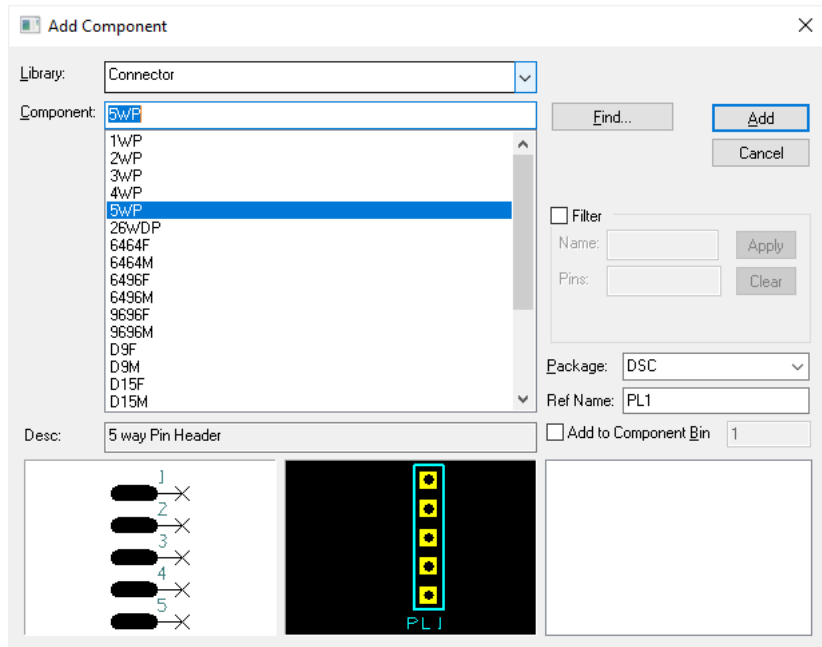


The picture above shows how the Schematic might look. This example shows positive, negative and 0V power rails.

Adding Connectors

Connectors are created as ‘normal’ components with associated Schematic Symbols and PCB Symbols, there are no special additional items to create.

Your only consideration for connectors is whether you have one multi-pin Schematic symbol representing a PCB footprint, or whether you have multiple single pin Schematic symbols representing a PCB footprint. The decision is based on whether you want to position the different Schematic Symbols across the design or have them clustered in one place as you would with a single Schematic symbol.



The **Add Component** dialog above shows an example of a single Schematic symbol. Where a connector Component is defined with multiple Schematic Symbols, each pin of the connector is added to the design individually. The creation of connectors is covered in a later Chapter on *Creating Components*.

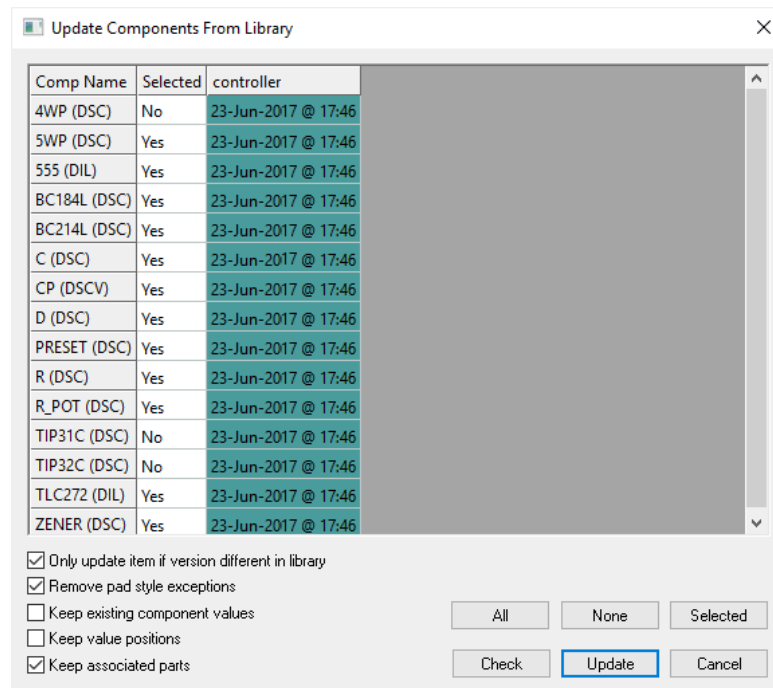
Reloading Components

Once components have been added to the design, you may need to change the Schematic symbol or the component definition and then reload it into the design.

You can reload components individually, or the whole design or by browsing and selecting the ones to update. There are three commands on the **Tools** menu under **Update Components** to do this.

You can also select a component or multiple components in the design then select the Update Components option from the context menu if you wish.

The dialog you are presented with allows a number of key functions to be performed.



You are presented with a list of all the components in the design, or across all sheets in the design if invoked from a Schematic within a Project.

This list shows you the timestamp on each component in the design as an initial guide to which library items are currently in use.

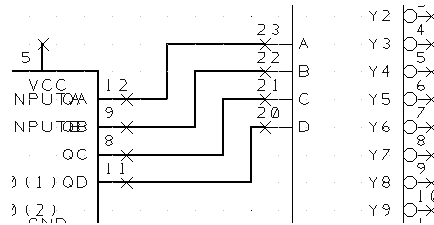
You can toggle the 'selected' state of each component, or use the All/None/Selected buttons to toggle all the states, as a quick way of choosing the components to be updated.

All components that use the same Component or Symbol will be reloaded, even if only one item was selected. You cannot have two different versions of the same Component or Symbol in the design.

Adding Connections to the Design

Connections are added to a Schematic design to make electrical connectivity between component pins. You can also add connections to other connections forming junctions or tie dots as they are sometimes called.

You are also allowed to add connections that don't connect to anything. These are called Unfished or dangling Connections and are useful during the design process but should be resolved before the translation to the PCB design. You may wish to start a connection path from one device without knowing where it connects to. Leaving it dangling will allow the path to be created ready for connection later on.

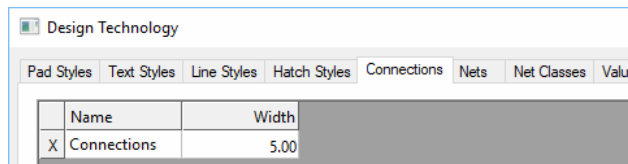


The **Undo** option can be used at any time to undo connection. While editing a connection, you can also use the <Backspace> key to undo the last connection segment.

Properties of a Connection

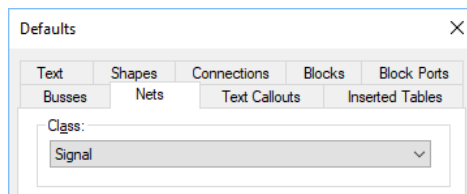
When you first start adding connections you can predefine the connection **Width** using a style **Name** (the physical size that will appear on the plot), and the **Net Class**. The Net Class is used to create a relationship between the logical Schematic connection, and the physical track thickness that is used in the PCB design once translated (see below).

There are two items to define in order to use the connection width that you require. The first is the style of the connection, the **Name** and the **Width**. These are edited in the **Design Technology** dialog under **Connections**. You can assign a sensible name for a connection style and a real width. A value between 5 and 8 thou is normally acceptable for most modern laser printers but you may decide that you want the connection on the paper thicker or fainter.

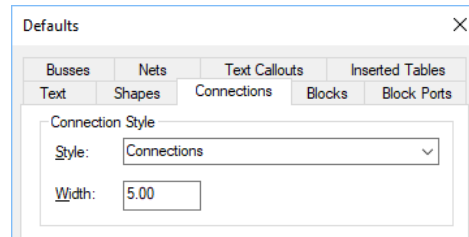


The next items to define are the defaults for the **Connection** and the **Net Class**. These are used to give the new connection a thickness (**Connection**) and a **Net Class** name for use when translated into the PCB design. Both of these items can be found on the **Settings** menu under **Defaults**.

As previously discussed in this chapter under **Setting Up A Schematic Technology**, the connection also carries a **Net Class**. This is used when the design is translated to a PCB design where it then picks up the PCB connection's physical properties. With the **Nets** selection, you choose a **Net Class** name from the drop-down list.



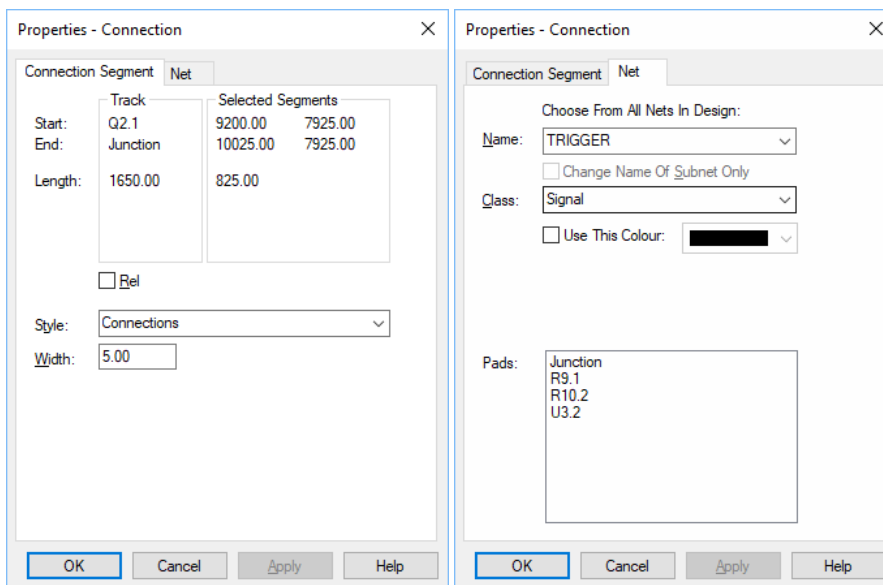
The **Connections** dialog is a list of Connection style **Names** with a given **Width**. The name chosen from this list will be the one first used when adding connections to the design.



When a connection is added to the design, it now has a drawn (and plotted) thickness, a Net Class Name and a Net Name (defined by you or a default net name).

Properties dialog

You can view and edit the connection's characteristics using the **Properties** dialog.



Changing the Connection style

During the addition of a connection you can change its style very easily. To do this just right mouse click and select **Style** from the context menu. You can also do this by using the shortcut key <S>.

Adding Connections

Connections can only be made by using the **Add Connection** option from the Add menu or from the **Add connection** button on the **Schematic Design** toolbar, or by double-clicking the mouse on an electrical terminal and then moving the mouse to draw the connection.

Double-click and move methods of adding connections are 'one-shot' modeless operations. It is fast but once the connection is completed, you are automatically put back into Select mode again. The Add Connection option allows you to continue 'stitching' in multiple connections until you exit this mode.

Connections cannot be added by adding a line item or bus item, they have to be proper connection items (using **Add Connection**). Likewise, just placing Components so that the connection points touch will not form a connection. Components must also have electrical terminals on them to connect to.

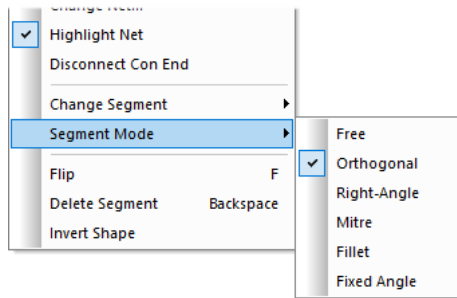
*To check that a connection really is a connection, select it in the design and view the status bar at the bottom of the window. If you have used a connection the status bar will say **Connection**.*

However, once a real connection has been made, moving that component will move the connections with it, a truly connective design structure is used throughout. You will see and understand the power of this as you design with the product.

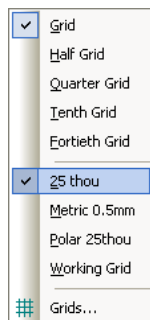
When adding connections, each click of the mouse will add a corner, from this you may make a direction change.

You will also find useful a switch in the **Colours** dialog which allows you to show **Connected Pins** only. When selected, only pins that are not connected to will show their **X** terminal position.

In Schematics, it is usually easier to add the connections if the **Segment Mode** is set to **Orthogonal**. This is set in the **Preferences** dialog **Interaction** tab on the **Settings** menu. The Segment Mode can also be changed while in **Add Connection** mode using the **Segment Mode** option on the context menu.



From this menu you can choose various options. In Schematics the **Orthogonal** mode is mostly used but other modes are available.



When adding connections, you can use the **Grid Snap** mode. While the connection is dynamic on the end of your cursor, click the **<H>** key. From the context menu you can select any of the grid settings and named grids.

Make sure that the connection begins and ends exactly on the electrical pins/terminals (using the Grid Snap mode makes this easier).

Adding Net Names to the connections

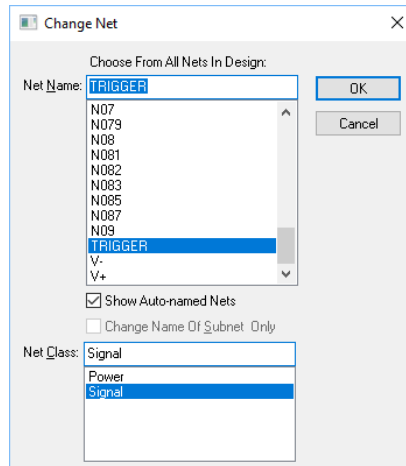
All connections are allocated a default net name and net class. It is a good idea to give significant signals a net name instead of the allocated name in the form of a number, such as N0022. It may also be desirable to reallocate the net class, by default each new connection acquires the same net class as the previous one.

Multi-sheet Schematic projects use common net names for the links that join them together. All net names are automatically associated with the Project and after first use can be selected from the drop-down list on each sheet rather than being remembered and re-typed. Predefined net names can also be picked from the external technology file associated with the design. It is worth the slight extra effort using meaningful net names rather than numbers.

Once a connection has been started (or once it is added to the design), you can give it a net name. If you don't give it a 'user defined' net name at the start, an 'auto-named' net name will be assigned for you. These will be sequential and unique, and will start with N, (N0010, N0008 etc.). They can still be renamed afterwards using the **Change Net** option.

The Net name does not appear automatically on the drawing, but it does always appear in the Status line at the bottom of the screen and can be viewed when the connection is selected. If you need to change the name of the net itself there are a number of methods for doing this.

You can use the **Properties** dialog from the context menu and **Net** page for a selected net, or you can use the **Change Net** option, also on the context menu.



The net name can also be changed from the **Nets** page of the **Design Technology** dialog on the **Settings** menu. This lists all the nets in the design and is shown below:

Name	Class	Own Colour	Colour
0	Power	Yes	Green
Q1_C	Signal	No	Grey
Q1_E	Signal	Yes	Blue
Q3_E	Signal	No	Grey
R1_N	Signal	No	Grey
R2_N	Signal	No	Grey
R3_N	Signal	No	Grey
R5_P	Signal	No	Grey
V2_P	Signal	No	Grey

Displaying Net Names on the design

To display the net names on the design you should select the net. From the context menu select the **Display Net Names** option. You will notice the net names appear on the Symbol terminal on one end of the connection. The terminal used for the name to be displayed is the closest terminal to the point on the selected net.

If you select a Component, you can also display all the Net Names associated with that component for connected pins. Use the **Display Net Names** command from the context menu with the component selected to do this.

Dangling Connections

Dangling connections are connections that start on an electrical point in the design but finish in 'free space'. You may do this naturally as you design so that the aesthetics are worked out as you design.

If you wish to add a connection and leave it dangling, double-click to end it in free space. To continue using that connection, double-click on the end of it and continue drawing in the connection path. You can also continue using it by using the Add Connection option and select the end of the dangling connection as the start point.

There are two features that will help you when using dangling connections, the first is the **Unfinished Connections** colour setting found on the **View** menu that can be chosen these connections, and the second is the **Unfinished Connections Report** which is located on the **Output** menu under the **Reports** option.

Starting a connection on a bus

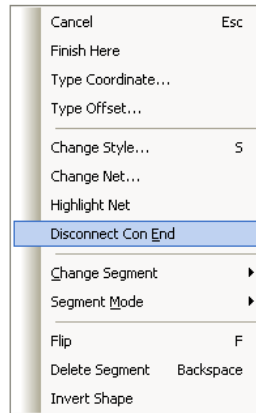
To start a connection on a **Bus** (Busses are described in more detail later on in this chapter) you must use the **Add Connection** option. You may however end on a bus using the modeless method.

Disconnecting connections from terminals

If you make a connection between two terminals and you decide that one end should have been connected to a different terminal, you can move it using the **Disconnect End Con** option.

To use this, edit the end which you wish to move (double-click on the connection). Once in edit mode, right click and select the **Disconnect End Con** option from the context menu. The connection will now be on the end of your cursor ready to connect to the new end point.

The context menu shows the **Disconnect End Con** option:

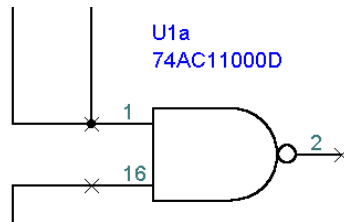


Showing Junctions over terminals

If two connections are attached to a component pin, a junction is only drawn if the **Display Implied Junctions on Pads** switch is on in the **Preferences** dialog under **Schematic Interaction**. By default, this is switched on. This only affects junctions on pins and not normal connection junctions.



The picture below shows how (on pin 1) the junction is shown on the component pin.

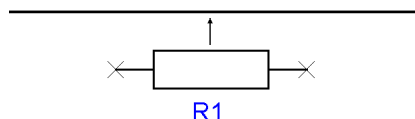


Inserting components to split a connection

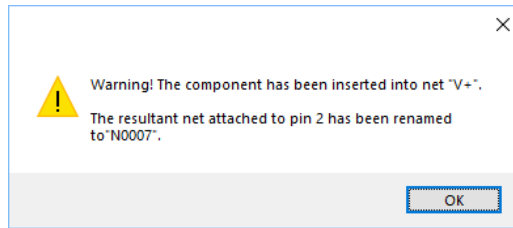
Once a connection has been added to the design, it is possible move or add a component over this connection and to split the net. This is useful if adding components such as resistors or capacitors to the design.

The situation can only occur with 2 pin components and where the connection is orthogonal horizontally or vertically.

First the component is selected or added, and moved over the connection.



Once the component has been moved over the connection and released, a warning dialog is displayed. This will warn you that two nets will be created and the name of the split net.

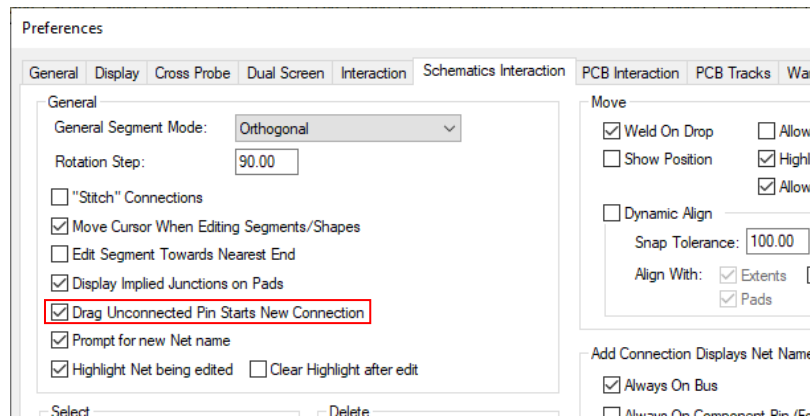


The finished result looks like this.

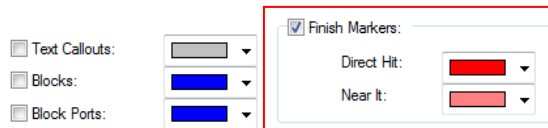


Dragging off Pins to Start Connections

An easier way to add connections is by 'dragging' the connection off the pin to start it. When enabled in the **Preferences, Schematics Interaction** tab, dragging from an unconnected component pin in schematics will start a new connection from that pin.

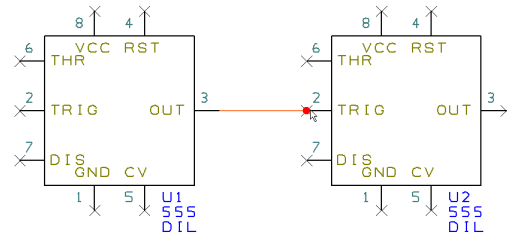


To aid the display of a legal landing site, finish markers can be enabled through the **Colours** dialog.



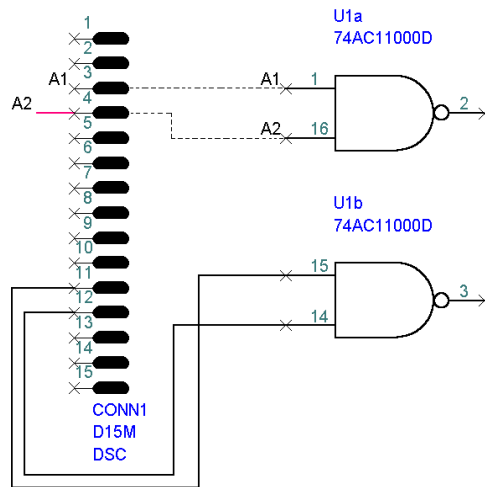
To use this mode, set the **Finish Markers** to on.

Drag off a source pin to start a connection or track, draw your corners and then move the cursor over the target pad. The **Finish Markers** will highlight to show that the connection can be finished there and the connection will finish.



Adding Nets that aren't shown on the design (Implied Nets)

When creating connectivity in the Schematic design you may not wish to show all the connections. Like the concept of a bus, or a power and ground nets, by using net names on the terminals of components, implied connections can be made. There must obviously be two or more net names of the same name in the design to make the connectivity.



The design above shows how the connectivity might look by using Net Names and implied connections, and physical connections. On the net marked A1, the net name has been applied directly to the pin using the context menu option Add To Net. For the A2 net name, a small dangling connection has been shown. This might be used to clear the component to make the connection clearer without drawing in the whole connection across the design. The dotted lines show the implied connections on this picture only, they wouldn't appear on the real design.

When the design is translated to PCB, any nets with the same name (even though there isn't a physical connection drawn) will be connected together. This helps keep the Schematic free from connections that don't need to appear. Likewise, this would happen also if two nets had the same name on different locations of the Schematic but weren't actually supposed to be joined. You can use the **Highlight Net** option to view the entire net.

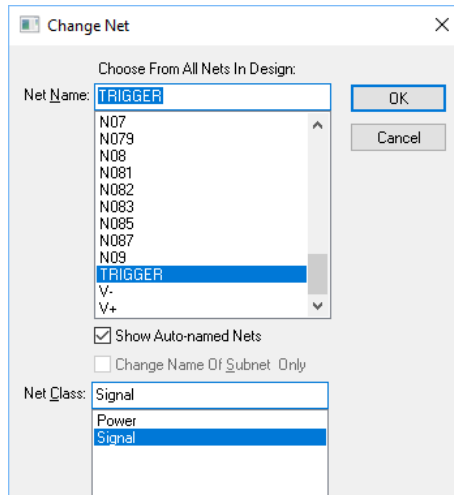
If you really wanted, you could create the whole design using this connectivity style.

Naming and Renaming Nets

At any point nets can be named using a user defined name or renamed from an existing name. When a connection is added to the design it will be assigned a 'system' name, such as N0009, N0001 etc. These names are unique but can be changed to one of your own during the add connection process or afterwards as you wish.

Renaming a net

A net name can be changed by picking the connection and selecting the **Change Name** option from the context menu.



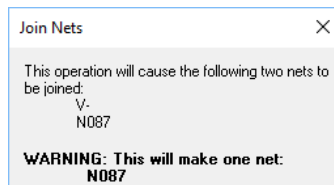
From the **Change Net** dialog, you can choose an existing net name to use or type in your own name into the **Net Name** box.

Connecting nets together

If two connections with different net names are connected together, the resulting single net will retain one of the original names. In this situation you should check the net name on the Status line at the bottom of the screen and change it if required. Use the **Change Net** option to do this.

Joining Nets

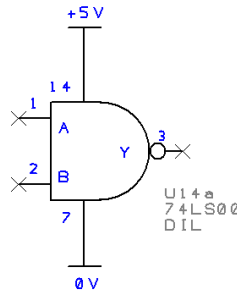
Where named connections are connected together, the **Join Nets** dialog will be displayed. This will inform you of the changes that will take place and to the name of the merged net. If the net name is not to your liking, use the **Change Nets** option to rename it.



For connections where one or both of the nets uses a default net name, no warning dialog will be displayed.

Power and Ground Symbols in Schematics

Power and ground supplies can be shown in various ways; they can be shown by adding connections or by using power symbols, or by simply naming the terminals using implied connections.



The most normal way is to use small power and ground symbols that use named nets to give implied connectivity.

Where there is a requirement to show power connections a symbol can be used. The symbol has no electrical significance and is not translated to PCB, so it is purely there as cosmetic representation on the final Schematic drawing. The importance in the symbol will be the net name used in the terminal of the symbol or the net attached to it.

These symbols are created as Schematic only components in the library. The supplied libraries contain a selection of basic power and ground symbols as well as generic symbols that can be copied and adapted for other power symbol representation. Because these symbols have a Schematic symbol and component representation but have no PCB symbol assigned to them, this means they will not be translated to PCB.

Alternatively, where terminals require power connections, these can be named with a nominated power name (using **Add To Net** from context menu) and all nets of that name will be connected together during Translate To PCB.

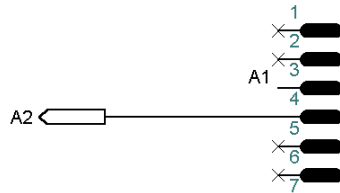
Component Editor

Within the component editor you can use the Net Class Name field for the signal pin to define the name of the net that will be pre-assigned when the component is added to the design. This is used to auto-connect this pin to the given net / net class when the component is added to a design.

Cross Sheet References

As with power and ground symbols, cross sheet signal references are purely cosmetic. They are created as Schematics only components that have a Schematic symbol and component representation but not PCB symbol assigned to them.

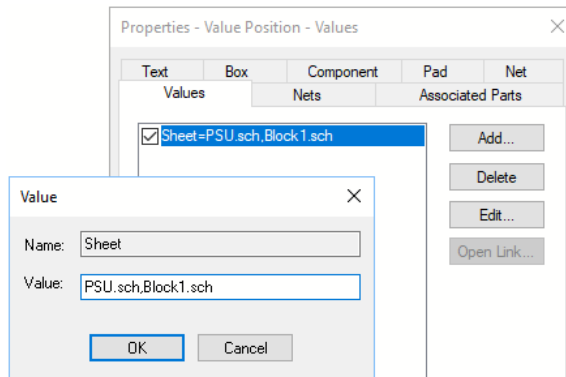
When this type of symbol is used on a sheet, it is the net name that carries the significance not the symbol. The same net names used will be connected together when the Schematic design is translated to the PCB design.



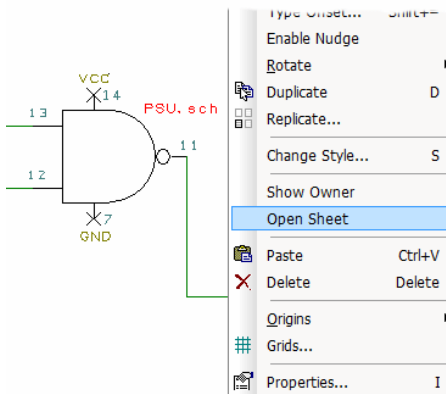
For the example above, all nets called A2 will be connected together regardless of which sheet they are on. The cross sheet reference is a neat symbol that denotes the connection.

Page Links

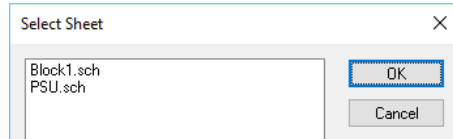
You can use any piece of text which names another sheet in the Project as a **page link**. The text must be the schematic design name and must have the .sch file extension, for example PSU.sch. The referenced design also must exist in the current **Project**. The name format can also contain a comma to create a list of sheets, for example, PSU.sch,Memory.sch,FPGA.sch. You could add a Value, such as PageLink, to an item and reference the sheet(s) as the value.



The value must be displayed in order to select it to use the **Open Sheet** option from the context menu.

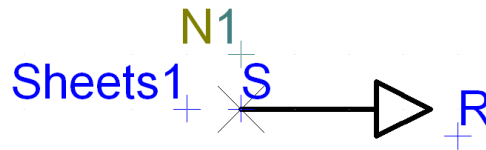


Where the net appears on multiple sheets, a dialog will be displayed from which you can select the sheet to browse to.



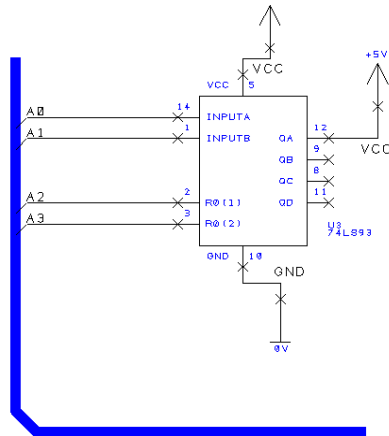
Page Link Components

This can be used in conjunction with the **Nets where sheets are used** attribute (see above) to create page link components. Add the **Nets where sheets are used** attribute to the Schematic Symbol to provide the link reference.



Using Busses

A bus is a shape that represents a collection of signals on the Schematic design. Instead of drawing all the connections across the design and connecting them to every pin required, you can make the design less cluttered by using busses. You can add connections from an electrical pin to the bus that carries the signal.



Open/Closed Busses in Schematics

There are **Open** and **Closed** busses.

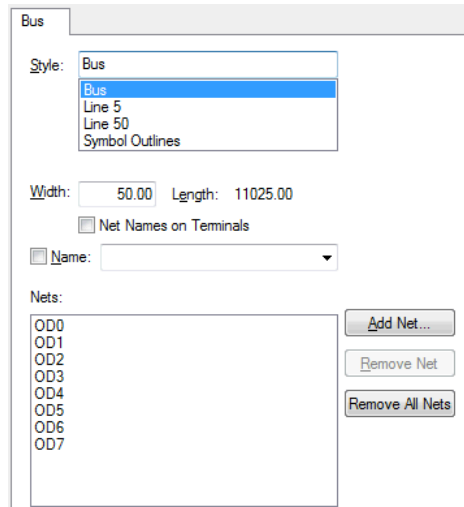
An open bus means that it can carry any net names. You simply attach the connection to the bus and the bus inherits that connection's net name. A closed bus is where all the net names are predefined in the bus (using Properties). When you add a connection to the bus, you are prompted with a dialog requesting the net name selected from a list.

Closed Busses

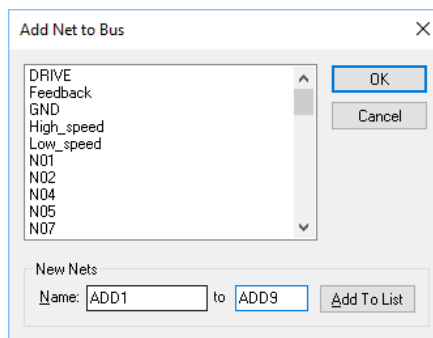
This is the process for defining a 'Closed' bus with the net names pre-defined.

► To define a Closed bus

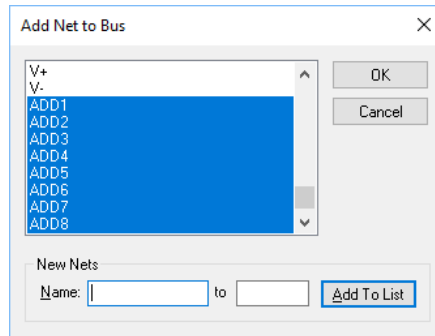
1. Using the **Add Bus** option from the **Add** menu, draw the shape of the bus. The Add Bus uses an open polygon shape.
2. You should use the **Segment Mode** of **Mitre** to add the mitred corner to any corners of the bus shape.
3. Exit **Add Bus** by double-clicking the mouse.
4. Select the bus and select **Properties** from the context menu.
5. Select the **Add Net** button.



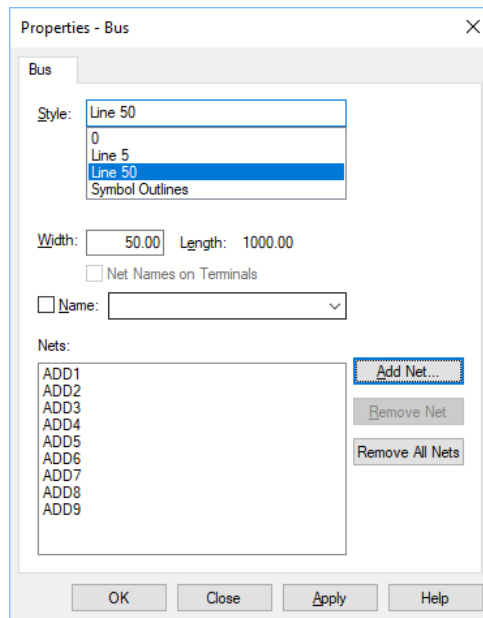
6. At the bottom of the dialog in the **New Nets** boxes, enter the net name range required in **Name** and **to**.



7. You can add a name range by entering the first and last numbers in the sequence, or a single name in the **Name** box and then selecting **Add to List**. This is the only place you can add bus net names for use with this bus (now a closed bus).
8. You must use **Add to List** to add the name to the list so that it can be selected and added to the bus.



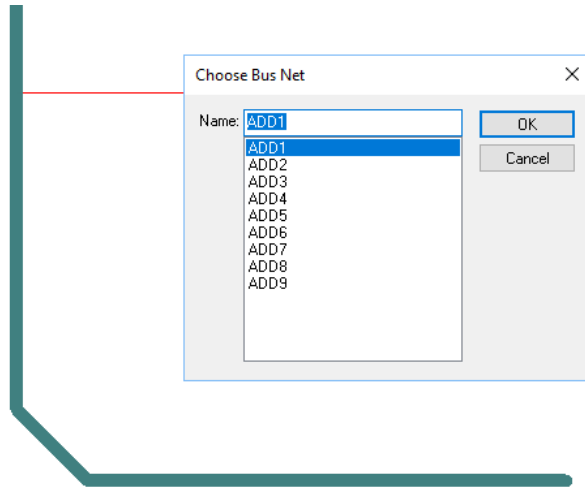
9. Select the names required from the list for the bus and click **OK**.
10. If you have missed a net name or wish to add others, enter the **Add Net to Bus** dialog and select or add the net name as before.
11. Press **OK** to exit the **Properties** dialog once all the net names are visible in the display window.
12. The dialog below shows the net names **A0** to **A15** entered.



Adding Connections to busses

► To add a connection to a closed bus

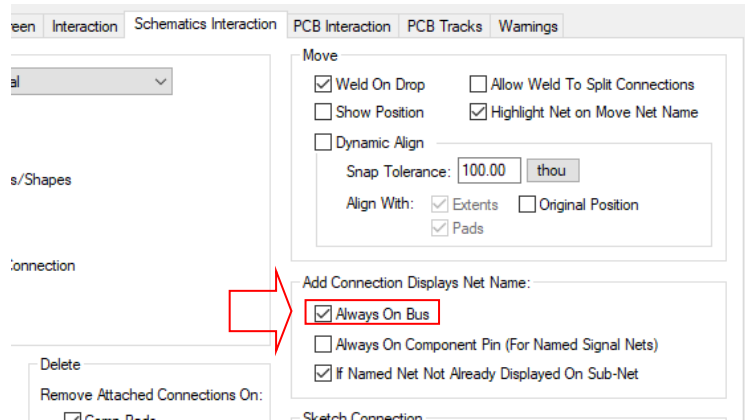
1. You have defined a closed bus (in the above description), now when you add a connection from a terminal and click on the bus to finish, the system will give you the choice of which net name to choose (taken from the bus net names list just entered).
2. Select one of the names and press **OK** or double-click to add it.



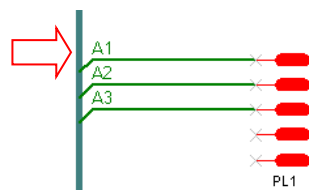
3. Any connection can be added to the bus but only one of the names from the list, no other name can be used without adding to the **Bus Properties** dialog first.
4. If two connections are added and both use the net name ADD1, for example, the **Translate to PCB** option will connect them both together, in other words the connection is implied by the name of the net.

Bus Net Names

When connecting to a bus you can choose to display the net name or leave it not shown. There is a check box in the **Preferences** dialog under **Schematic Interaction** that sets this, by default it is switched on.



The picture below shows the net name on and off for the three connections.



In summary:

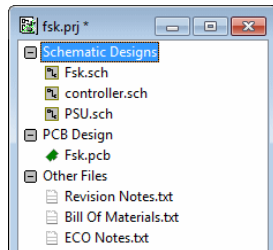
- Busses are cosmetic and have limited intelligence.
- Busses can be open or closed - closed is where the net names are predefined and internal to that bus.
- Connections are made by clicking on the bus which inserts a 45-degree bus terminal.
- Connectivity is made though implied net names, e.g. net names that are the same.
- Busses can be **named**, the name displayed and busses located using the **Goto** bar.

Adding more Sheets to the Project

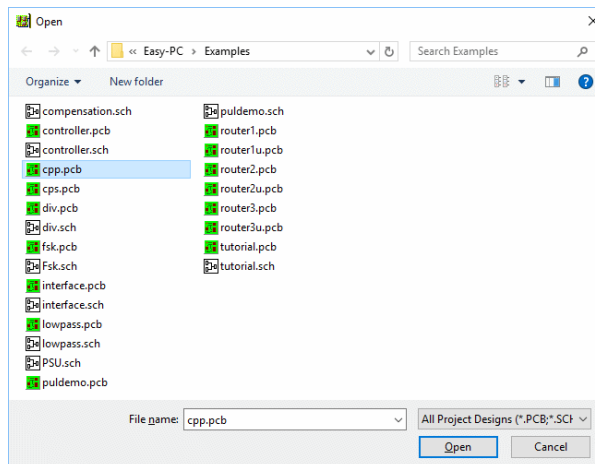
If more than one person is working on a design, you may split the schematic into a number of design sheets. When the sheets need to be combined you can add them to a Project. For multiple sheets you'll need to have them all in a Project so that they can be translated into the PCB design.

► To add a schematic design to a Project

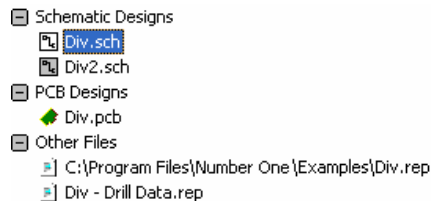
1. You should be on the Project page (if a Project has been defined). If a Project hasn't been defined you should create one.
2. An existing Project might look like this:



3. To add an extra sheet to it you must be showing the **Project** view (as above).
4. Click on the Project menu and select **Add Files To Project**. This dialog is displayed:



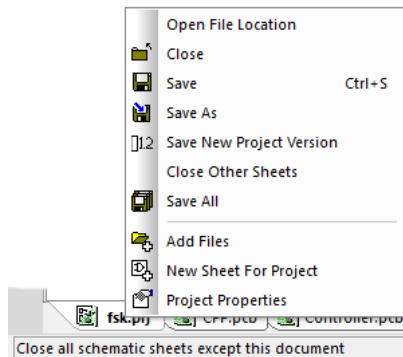
5. You can add Schematic sheets (designs) or a PCB design to the Project. You can only add one PCB design though, whereas you can add multiple Schematic designs.
6. Select the required design and select the **Open** button.
7. Your Project will now look like this. The extra Schematic design sheet – Div2.sch has been added.



8. Now when this Project is translated into a PCB design, Div and Div2.sch will become a single PCB.

► Alternative method to add a schematic design to a Project

1. As an alternative, you can also right click with the cursor over the Project document tab at the bottom of the design window and select Add Files from the context menu:



2. From the browser, select a design file to add to the Project. Other types of files, reports or Parts lists etc. can be associated with the Project as well using this option.

Splitting components across Schematic Sheets

If you wish to split multiple gated components across more than one schematic sheet then this is possible. As this is an infrequent operation, it is fully documented in the online help provided and installed with the product.

Adding Circuitry from Other Designs

Where previous designs already exist, you may find it more practical to copy a section of circuit and add it to your current Schematic design. Where proven circuits can be used this is highly desirable. This process is straightforward.

► To copy existing circuitry

1. Open the Schematic design containing the circuit to copy from.
2. Open the current Schematic.

3. In the existing circuit, drag a box to make the selection, or use the <Ctrl> and <Shift> key combinations to select parts of the circuit required.
4. Use the **Copy** command to copy the circuit. Copy is available on the shortcut keys <Ctrl-C>, or on the **Edit** toolbar or on the **Edit** menu.
5. Now go back to the current design (use the Windows menu and select the design from the list, or use <Ctrl-tab> keys to toggle through the open active windows).
6. Use the Paste command to add the circuit to the design. Paste is available on the shortcut keys <Ctrl-V>, or on the **Edit** toolbar or on the **Edit** menu.
7. During the addition of new circuitry to a design you may be asked to resolve duplicate net names. This process is described below.

Resolving Net Names

When using the **Paste** (or **Duplicate**) option to paste a copied circuit into your current design, if net names already exist in the design, you will be requested to resolve these. Component names will automatically be resolved by being incremented to the next unique name. Net names however need to be resolved.

The **Paste Net** dialog allows you to see and control how duplicate net names are resolved.



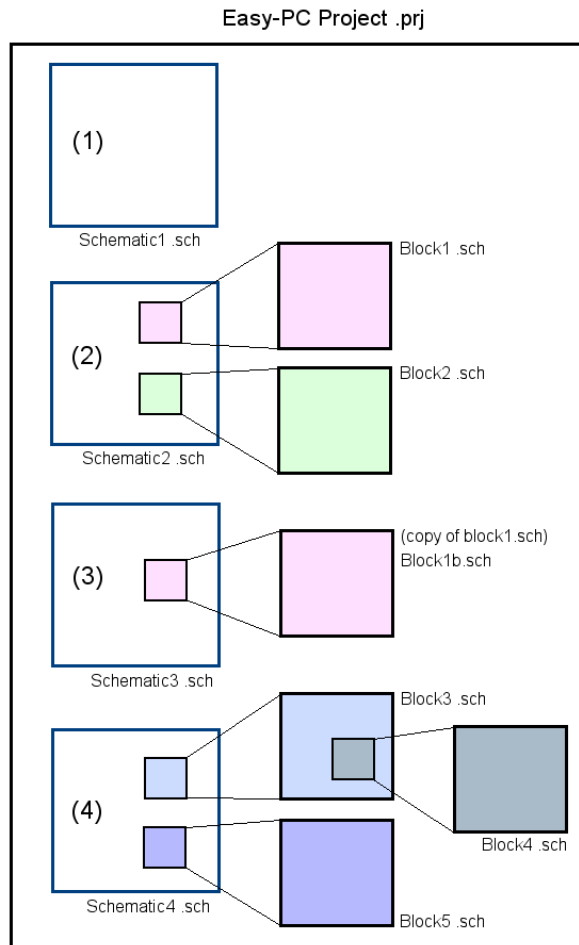
Once a net name is detected, you are prompted with the question - **Merge net with existing net?** Using the options on the dialog, you can elect to add the net name to the existing net or to automatically allocate a new name to it, therefore not connecting it into the existing net.

Hierarchical Designs

Overview

You can add multiple hierarchical schematic sheets to a project. This is different to multiple sheets in a Project in that you can add a block on a schematic sheet and 'push' into that block to reveal another page. For example, this could be detail that can be covered by an overview block, say PSU or MEMORY etc. but for which you can show the lower level detail on a separate sheet.

To use hierarchy, the designs must be contained within a **Project** file.

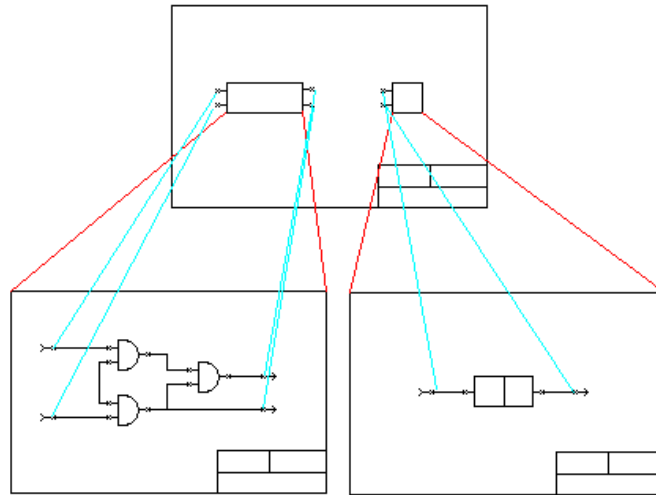


Hierarchical Building Blocks

The diagram above illustrates how various scenarios are possible when using hierarchy, all within the same Project file. A design sheet can contain multiple blocks (2) and multiple hierarchical sheets. In (3) an existing block can be copied and effectively used again, it is renamed though. Schematic (4) shows multiple blocks and a block inside a block (a nested block).

Overview

Schematic design sheets in a project can have a hierarchical structure. This is different to using multiple sheets in a project at the same level in that you can add a block on a schematic sheet and push into that block to reveal the block contents on another schematic design. For example, this could be detail that can be covered by an overview block, say PSU or MEMORY etc. but for which you can show the lower level detail on a separate sheet. To use hierarchy, the designs must be contained within a Project file.



The above shows a top level schematic design with two blocks. Under this, each block's contents is a separate schematic design within the Project.

There are two reasons for using a hierarchical schematic design structure.

- To allow a schematic design to be structured, so that it can be broken down into its functional elements. This promotes "Top Down" design; break down a problem into functional elements, then define the detail of each element.
- To facilitate the re-use of commonly used circuit elements. This promotes "Bottom Up" design; use predefined elements to build a solution.

In practice, you may want to use a combination of both "Top Down" and "Bottom Up" design.

What is a block?

A block consists of two parts. The first part is a block symbol, inserted into a schematic design to represent the block at the higher level. The second part is a complete schematic design containing the design elements that define the contents of the block, and this represents the implementation of the block at the lower level.

The two halves of the block can be defined separately, but are eventually linked within a project by using a common block name. The block symbol has a name, and a schematic design representing a block's contents can be allocated a name using the **Design Properties** dialog.

The block symbol can contain block pins that can have schematic connections attached to them to pass signals to a corresponding block port in the block's contents design. The block ports are single pin symbols that can be separately placed anywhere in the lower level design.

The link between a block symbol pin and its corresponding block port is set by both having the same port name.

Blocks are not copied to the PCB design, but exist to place a hierarchical structure on the schematic representation of the design. When translating the schematic into a PCB design a check will be made to ensure each block symbol pin and its corresponding block port are on the same net.

A hierarchical design is easier to navigate, because you can **Push** into the design representing the block contents, from its block symbol, and **Pop** out of the block design, back to the block symbol.

In a project tree view, schematic designs that represent the contents of blocks will have a blue icon and display their block name. If their block symbol is also defined, they are shown as an indented branch below the design that contains the block symbol. This way the hierarchical structure of the design is clearly shown.

Adding Blocks

There are three ways of adding a block into a design.

1. Add Block

Use this option from the Add menu to insert a new block into the design. A dialog will be displayed for you to define the block name, set the number of block ports, choose the block symbol and supply the name of the block contents design.

Define what the block symbol will look like by either using a schematic symbol from the library, or generating a standard rectangular shape using user specified parameters.

The block contents design can be a new design or you can hook up to an existing design either already in the project or a design file in the project folder.

The block symbol can be placed in the current design and the block design will be automatically added to the current project. Block ports will not be initially added to a new block design, but you can add them when editing the design by using **Add Block Port** when the port is needed.

2. Cut To Block

Use this if you want to take an existing schematic design and cut a section of it into a block. This option prompts you to draw an outline around the items to be pushed into a block, and across any connections that are to be passed down into the block.

The connections are cut at the outline and terminated on a block symbol pin. The other half is pushed to the new block design and terminated on a corresponding block port.

The drawn outline and new block pins become the block symbol, and all data within it is moved into the new block design. The block symbol can now be replaced with a more suitable shaped symbol using **Change Block Symbol**.

The new block design will take a default file name based on a name stem defined with the **Block Defaults** dialog. You can change this name in the project view by selecting the schematic design and using the Rename option from the context menu.

3. Duplicate Block

Once you have added a block using one of the two methods above, you can create multiple copies of it using the **Duplicate** command. This will allow you to copy the block symbol and the designs that it uses to define its block contents.

All component and block names used in the original block will be incremented in the duplicated block designs. User named nets will be merged with the original nets if they are used a higher level outside of the cut block, or will become new nets with incremented names if they are used just within the cut block.

Navigating a Hierarchical Design

To navigate the hierarchy of a project, you can either use the project view to switch to the design representing a block, or you can use the **Push** and **Pop** commands within a design.

To **Push** into a block, select any part of the block symbol and use the **Push Into Block** command from the context menu. The blocks design will be opened and if you had selected a block symbol pin the corresponding block port will be selected. You can use double-click on the block symbol outline, or on a connected block symbol pin to quickly perform the same operation.

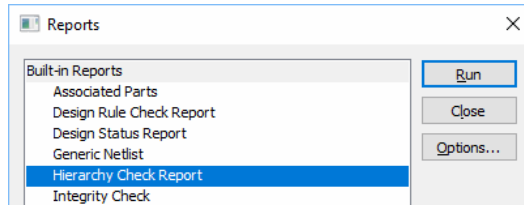
To **Pop** out of a block contents design, back to its block symbol, select the **Pop Out Of Block** option from the context menu. You do not need to select anything in the block design to do this, but if you select a block port the corresponding block symbol pin will be selected instead of the whole block symbol. You can use double-click on the block port outline to quickly perform the same operation.

Hierarchical Nets

The span of nets in a project containing blocks is no different to a project with separate schematic sheets in it. If the net in a block design has a user defined name, it may also be used anywhere in the entire design. The user named net can be used inside and outside of a block, and all items on the net in both designs will be connected together when a PCB design is created. Therefore, all user defined net names are always global in a project. Auto named nets are only unique within a single schematic sheet, and so these cannot be used to connect from a block symbol pin in one design to its corresponding block port in another design, as they would end up as different nets in the PCB design.

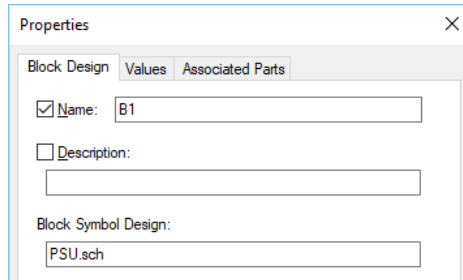
Each block symbol pin and its corresponding block port must therefore be on the same user defined net. If two different named nets are used, an error will be given when running the hierarchy report, or when attempting to create a PCB design or adding forward design changes to a PCB design.

To help keep these nets the same, when you start a new connection from a block port, its corresponding block symbol pin from the higher level design will be accessed to retrieve the name of the net attached to it. This net will then be automatically used for the new connection. Thereafter, though, the nets will not be kept in line automatically, but can be checked at any stage using **Design Rule Check** or the **Hierarchy Check Report**.



Creating Commonly Used Blocks

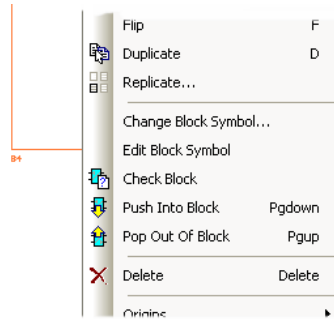
You may want to create a set of separate schematic designs representing commonly used blocks, such as power supplies or signal converters. These designs can contain block ports to terminate signals to be passed in and out of the block. It is then possible to copy these existing block designs into a project and link them to block symbols added to a higher level design in the project. This is done by using **Design Properties** (**View** menu, **Properties** with nothing selected in the design) to change the block name for the design. The block port names will have to be resolved at this stage to ensure the correct ports are linked to their corresponding block symbol pins. You must be pushed into the block to be able to do this.



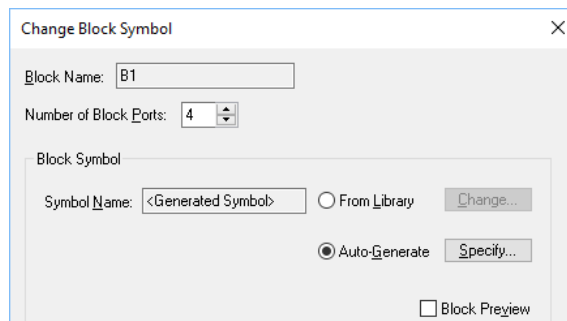
Editing Block Symbols

Blocks and block ports resemble schematic symbols, but are implemented as an internal "group" of shapes, text and pins representing the symbol. This enables them to be easily altered within the design, for example to have extra pins, a wider box, or descriptive text added. You can use **Edit Block Symbol** on a selected block symbol or block port symbol to open a new window to edit its contents, add shapes, pins and text etc. After edit, use **Save** to replace it in the design and it will again act like a single item. **Edit Block Symbol** is available on the context menu for a selected block outline.

When adding block symbols or block ports to a design, you can choose to auto generate them using some simple parameters, or use a schematic symbol from a library. The symbol from the library itself will not be used, but a copy of its elements will from the block symbol or port. This enables you to predefine standard block and block port symbols in your libraries to ensure a consistent interface. For example, you could define separate input and output block port symbols.



A block port created from a library symbol will remember its symbol name and can be replaced by a changed symbol from the library at a later stage using the **Change Block Symbol** function from the context menu, this is available for a selected block outline.



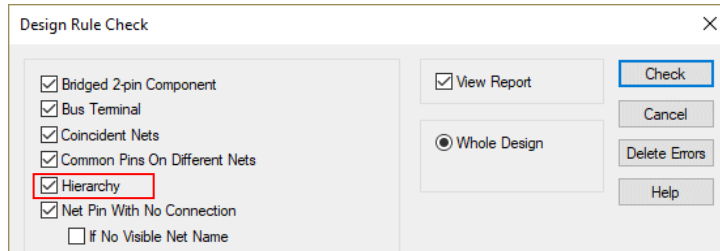
You can set up a highlight colour to force all Block Symbols and Block Ports to be the same colour, thus differentiating them from components in the design. Block symbols can also be individually changed to have their own colour using the **Properties** dialog.

Adding A New Connection To A Block

If you have a new signal that you wish to physically take down to an existing block design, you can start or end the connection directly on a block symbol outline. A new block symbol pin will be added to the outline to terminate the connection on, taking an automatically generated port name. You can then double-click on the block symbol to push down into the block design and use **Add Block Port** to add the port, from which you can start a connection to continue the signal in the block design. The net will be automatically set to the net used in the upper level.

Checking the Hierarchy

You can use the **Design Rule Check** option to check the consistency of the two halves of each block in a design or project. An error marker will be placed on a block symbol pin if it has no corresponding block port, or if its block port is connected to a different net.

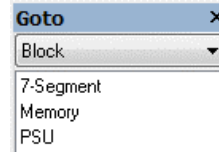
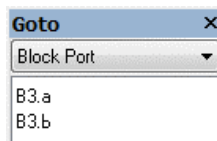


Alternatively, if you don't want to produce error markers, you can run the **Hierarchy Check Report** from the **Reports** dialog to get a report of any problems in the hierarchy. You can run this report on the whole project (from the project view), on a specific schematic design, or just on the selected block symbols from the context menu.

Hierarchical problems can be serious and difficult to fix once the PCB design has been created, so the **Translate To PCB**, **Integrity Check** and **Forward Design Changes** options will run the Hierarchy Checker prior to their operations and will not continue if there are problems.

Finding Blocks

You can use the **Goto Bar** to find any **block symbols** and **block ports** in the entire project.



Deleting Blocks

When block symbols are included in a selection that is being deleted you will be asked if you want to delete entire hierarchical blocks by removing all of their block designs from the project. At this stage you can also decide to just delete the block symbols and leave the block designs in the project. These designs can be re-introduced into the hierarchy at a later stage by linking by name to free block symbols, or can be deleted from the project using the project view.

Removal of entire blocks, including their contents, with the **Delete** option does not delete the actual design files from disc. The files can be restored to the project at a later stage or moved to another folder to be used in a different project.

Block Default Settings

The **Defaults** dialog has sections for setting the parameters that define what a generated **Block Symbol** and **Block Port** will look like. Here you can also set up the default name stems for block names, block design names, port names and default block net names.

Defaults

Busses Nets Text Callouts Inserted Tables
Text Shapes Connections Blocks Block Ports

Block Name Stem: B

Block Design Name Stem: Block

First Port Name: a

Block Net Name Stem: Hier

Generated Block Symbol

Pin Style: Block Pin

Outline Style: Block Outline

Name Style: Block Text

Gap Between Pins: 200.00 Show Port Names

Outline Width: 1000.00

Pin Leg Length: 100.00

Using Hierarchy

Starting with Top Down Design

► To create a top-down hierarchical design

1. Start with an open schematic from within a Project file.
2. We will first use a new block with no circuitry, we will assume the circuit will be added afterwards.
3. Select **Add Block** from the **Add** menu.

Add Block

Block Name: Memory

Description: 8Mb DRAM

Block Details

Symbol Name: <Generated Symbol> From Library Auto-Generate

Change... Specify...

Number of Block Ports: 0 Block Preview

Schematic Design for Block Contents

Use Schematic Design already in the Project

Fsk.sch

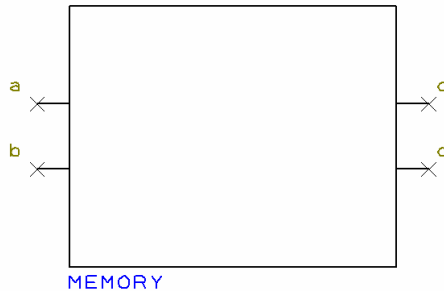
Add Existing Schematic Design to the Project

Browse...

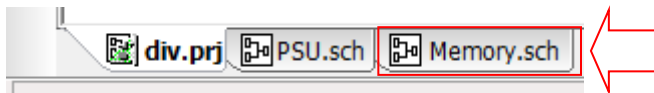
Generate New Schematic Design

Memory

4. The dialog is split into three sections; **Block Name** and Description, **Block Details** for the block 'outline' and the number of 'ports', and the bottom box containing the block schematic **contents**.
5. For **Block Name**: this identifies the block. It can be a system allocated name, like B1, or a user defined name, like MEMORY. A name to reference the actual contents of the block would be more useful.
6. For **Block Details**: we will select **Auto-Generate**. Once you have established some block shapes, you can save these to the library and reuse them using the **From Library**: option. The definition of the auto-generated block shape can be altered by selecting the **Specify** box. A dialog will allow you to create the auto-generated block using your own criteria.
7. For this example, make the **Number of Block Ports: 4**
8. We will create a new schematic design for the **Block Contents**.
9. Select the **Generate New Schematic Design**: radio button.
10. A default name is entered for you (Block1), change this to be more meaningful, we've typed **Memory**. This will be the new name for your lower level schematic design. It also matches the Block name but this isn't essential.
11. Click the **Add** button. This will add your new block symbol to the current design. Don't forget, it will also add you a new lower level schematic design named Memory.



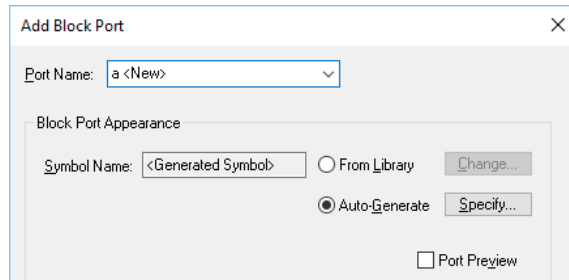
12. Once added, the **Add Block** dialog is displayed again to allow you to add another.
13. Click the **Cancel** button on the dialog to close it, we only need one block.
14. In the design, double-click on the block outline to push down into it. You could also select the outline and then select **Push Into Block** from the context menu. If you have a shortcut key allocated to this option, selecting the block and pressing the shortcut key is also a quick way to push into the block.
15. You'll notice the design name appears on a workbook tab at the bottom of the design area:



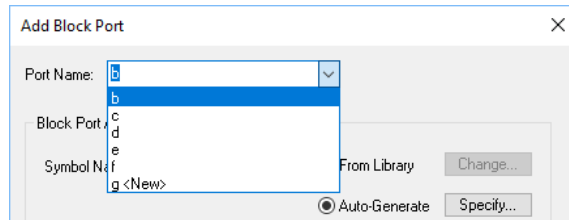
16. The lower level schematic will now come to the front and be editable.
You now need to add corresponding Block Ports (pins) to the schematic.

► **To add Block Ports to the lower level schematic**

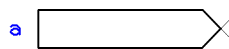
1. From the **Add** menu, select **Add Block Port**.



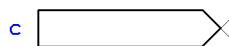
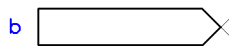
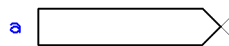
2. A list of **Port Names** is shown in a drop down list. This will be a list relating to the number of block ports shown on the top level block symbol.
3. You'll see that the list contains the same number of Block Ports plus one extra, shown as <New>. The list knows how many should be added to match the top level block.



4. The **Block Port Appearance** is the shape of the block port itself. This can be a symbol from the symbol library or an Auto-Generated one.
5. If using an **Auto-Generated** symbol, the **Port Style** and **Port Outline** can be defined using the **Specify** button and dialog box.
6. For now, we will keep things simple and leave the default Auto-Generated values.
7. Press the **Add** button to add a block port to the design.



8. Each block port is added to the design in sequence. Once one port has been released, you are returned to the **Add Block Port** dialog to add the next.



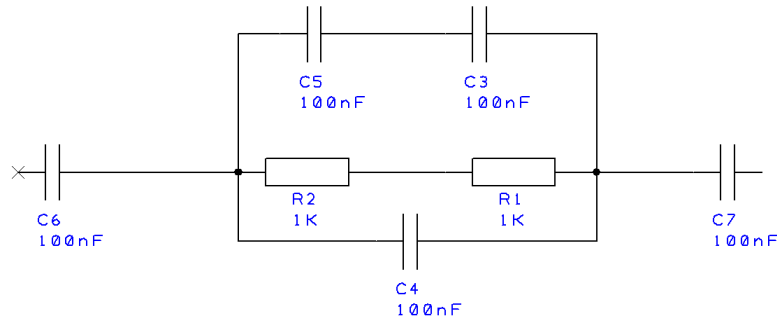
9. When the option gets to the first new port (the one marked as e <New> in our example, and the fifth port to be added, a warning dialog is displayed to inform you that there is a mismatch in numbers of ports between the top and lower levels.
10. Press the **Cancel** button to close the dialog.
11. Your block is complete with both upper and lower levels.

Cutting your design to block

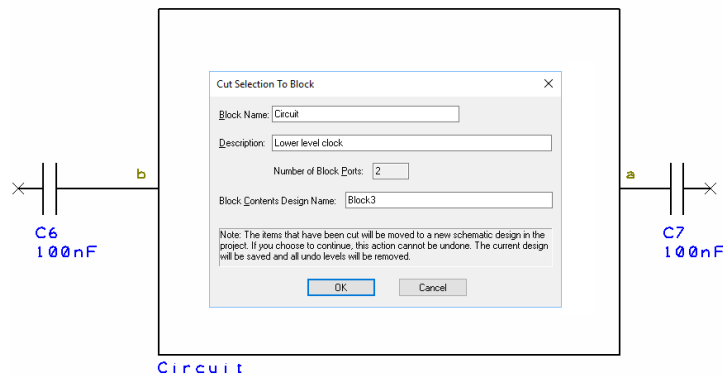
If you have an existing non-hierarchical design, you may wish to divide the design into blocks. The **Cut To Block** command allows you to select a piece of circuitry and create a block from it. Cutting a piece of circuitry to a block does not change the connectivity of the design.

► To cut a design to block

1. Select the **Cut To Block** option from the **Edit** menu.



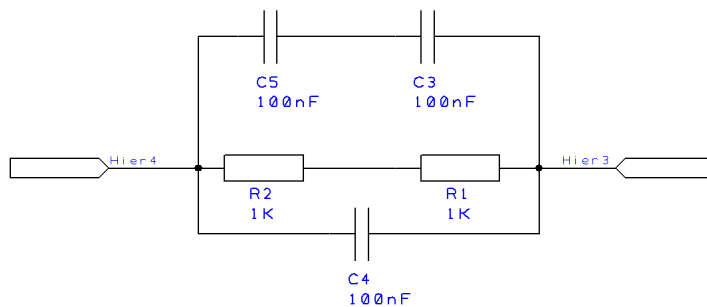
2. Draw around the circuitry you wish to cut into a block. By default, this is a rectangular frame, you can also use a polygon by selecting **Polygon Select** from the context menu.



3. Once the selection is completed, a block is created which contains all the items that were fully selected.
4. Each of the connections and busses are cut at the point they cross the selection shape. Connections are terminated on the upper level at a block symbol pin, and on the lower level by a block port.



5. If a cut connection does not have a user defined net name, its net is changed to a name generated using the name stem taken from the block defaults.
6. The block symbol outline on the upper level is created using the selection shape. The symbol line style and block pin style for this symbol are taken from the **Block Defaults**.
7. You can later edit this block symbol shape using **Edit Block Symbol** from the context menu, or generate a standard box shape, or use an existing schematic symbol, by using the **Change Block Symbol** option from the context menu.
8. Pushing into the block by double-clicking on its outline will reveal all the items removed from the upper level.

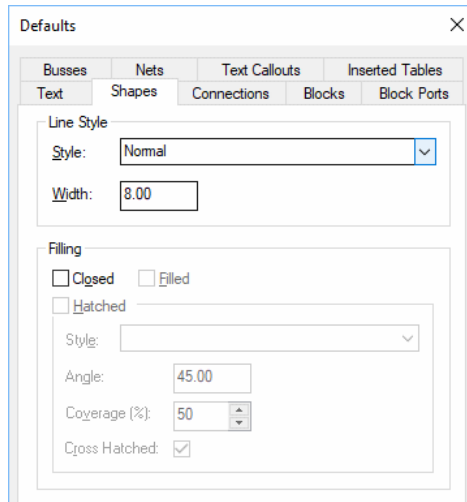


9. Connections which were cut will end on corresponding **Block Ports**.

Adding Shapes

When creating a Schematic design, you may need to add extra shapes to the circuit for annotation purposes. These shapes have no electrical significance but may be used for documentation purposes or suchlike.

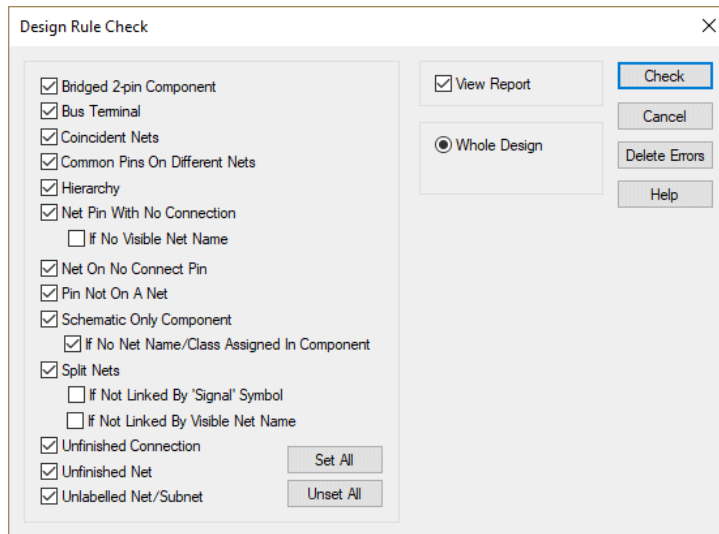
To add shapes, you can use the **Add shape** options from the **Add** menu, or use the **Add shape** buttons on the **Schematic Design** toolbar. When a shape is added, it uses the **Line Style** defined in **Defaults** and **Lines** dialog under the **Settings** menu.



Once a shape has been added, its style can be modified by using the **Properties** option on the context menu.

Schematic Design Rules Checking

From the menu option **Design Rule Check** on the **Tools** menu you can check using the electrical rules available. From the dialog, you can check across a Project, or on the current design using the options available.



All rules in this dialog are described in detail in the online help under the Index heading, *Design Rule Check (Schematic)*.

Transferring Schematics into PCB Layouts

In creating the Schematic, the names of all the components and the connections between them have been defined. The final process is the translation of the Schematic design into the PCB design environment.

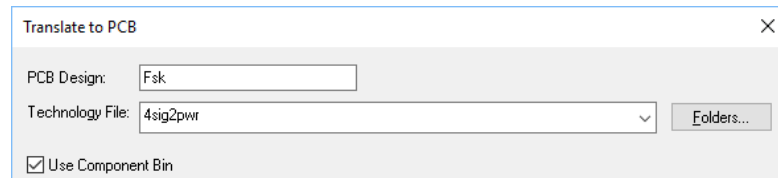
With the checking and procedures that are used, you should find that there are very few errors when translating to the PCB design. All errors must be resolved before proceeding; otherwise you may find your PCB design has errors. Warnings should be checked and action taken if required.

Before using the **Translate To PCB** option you should save your current work by selecting the **Save** or **Save As** option from the **File** menu.

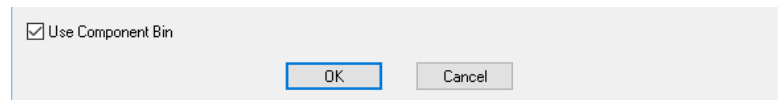
Running the Translate To PCB option

► To translate to PCB

1. Select the **Tools** menu and select **Translate To PCB**, the option will automatically cross reference the Schematic symbols to PCB symbols and will resolve the correct pin numbers for all the component pin names.
2. This process automatically collates the netlist and resolves nets to create the netlist.



3. You must give the PCB Design a name. This is normally the same name as the Schematic design but not necessarily.
4. You can use a **PCB Technology file** to start the PCB with. This is recommended as it ensures that layers and styles are setup for you. The file chosen here will contain your design requirements matching the type of design being generated e.g. SMT, double sided, single sided etc.
5. The check box to **Use Component Bin** may be selected. All components will then be sent to the Component Bin. This will make the start of the design process easier.



6. The alternative is to allow the components to be added directly to the design. These would be positioned in a stack near the bottom left hand corner of the design.
7. Once the **OK** button is pressed, the translate process will begin.
8. During this, any warnings and errors are displayed and saved to a report. You should correct any errors and observe and correct any warnings. The possible errors and warnings are described below along with detail of how to resolve them.
9. Once an error-free translation has taken place, you must remember to switch on the component bin in the PCB Design editor, use the shortcut key **<Ctrl-F9>**.

Resolving Warnings and Errors

A report is generated if there were any problems with the translation. This includes details of any connections made to only one terminal of a device, and any component for which a PCB symbol could not be found. The latter would be fatal and would need to be corrected.

Any problems reported following translation should be sorted out depending upon the severity.

- "Net NNNN contains a single pin"

 - This suggests a Schematic error where a connection should have been taken to a device pin, but failed to connect.
- "Could not load component XXXXX / package YYYYYY into PCB design"

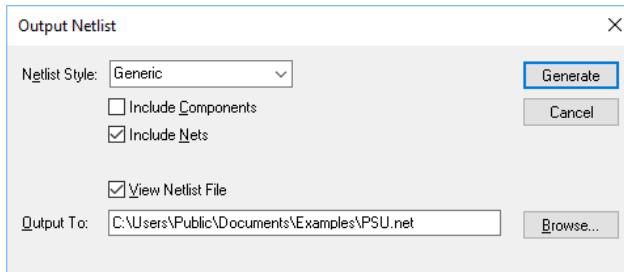
 - This suggests that a Schematic component has no corresponding PCB symbol allocated (at least for the chosen package type).

It is best to sort out any problems and re-translate, as it is less work to correct errors now.

Exporting Netlists

You can export a netlist in various neutral formats.

From the **Output** menu you can use the **Reports** option and select **Generic Netlist**. There is a limited amount of customisation that can be achieved using this option. The netlist style is fixed with the **Generic** format but you can include items by selecting or deselecting them.

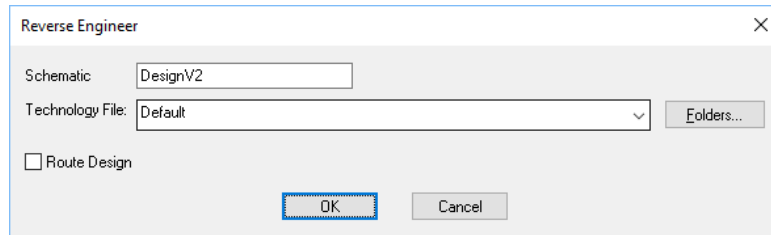


The netlist generated will also include any components in the component bin and nets that have been defined in the Nets dialog but not used in the design.

As well as the Generic Netlist output option, there is also the **Netlist** option that appears on the **Output** menu, this produces a native format netlist (this format is detailed in the online help).

Reverse Engineer

The **Reverse Engineer** tool is designed for use where there is no corresponding Schematic design for the existing PCB design. It is not intended for use where a PCB design is modified and the changes are to be reflected in the Schematic (the Schematic is always considered the 'master' design in this situation). This option is only available in the PCB design editor under the **Tools** menu and **Reverse Engineer**.



When a PCB design has been created, and if the PCB design is used to create 'local' libraries with the Schematic symbols added to the components in the new library, you must update the components into the PCB design first before using the **Reverse Engineer** option. Use the **Update Component** option from the **Tools** menu.

If you opt to use PCB only components, the Schematic Symbols/Components will be auto-generated for you when the Reverse Engineer option is selected. This method will work fine but will create dummy Symbols only, symbols that are shown as boxes with a representative number of terminals.

Using the Reverse Engineer dialog will enable you to select the new **Schematic** name, which library **folder** to use and the **Schematic Technology File** name. You can also elect to **Route the Design** automatically once autoplaced. Once back in the Schematic design, all Symbols are automatically placed.

Autorouting selected Symbols

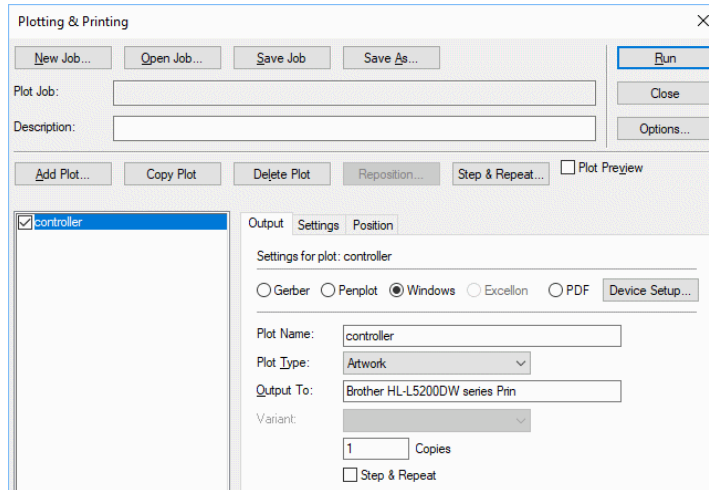
If you choose not to autoroute the design, you can selectively route them afterwards by using options on selected pins or components. These are available on the main menus and context menu.

It does 90% of the work in creating a Schematic. A bit of tidying-up is required of course but it is a big time-saver for those who choose to work this way.

Printing the Schematics

When you wish to print the design there are a number of options to choose from. The **Plotting & Printing** option on the **Output** menu gives you access to all printing options. This option allows you accurately scale and position the different plot types to the plotting media.

In the setup, you can choose from **Windows** printers, **HP pen plotter**, direct **PDF** and **Gerber** device drivers. (Note: The DXF output option is not part of this output mechanism and is located on the **Output** menu under **DXF Output**).



The dialog is split into two main areas; the top half allows plot 'jobs' to be created and saved. By saving a job file, you can apply the same parameters to a different design saving the setup time for each design. The bottom half of the dialog is used to create the contents of the plot.

Starting the Printing option

The process of setting up and using this dialog is much simpler than if creating a set of PCB design plot files, Schematics is usually only one sheet and not layer based.

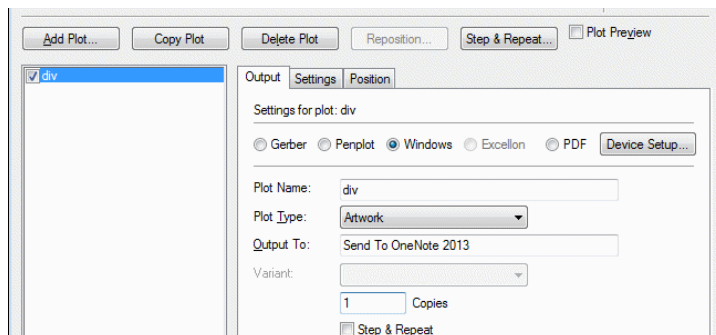
By default, when you first open the **Printing and Plotting** dialog, a single Windows plot scaled to fit the default print page size will be created for you. In theory, you simply click the **Run** button and a print will appear on your Windows printer. You don't need to create a job file or adjust any of the settings at to obtain this plot.

The creation of plot Jobs is detailed later on in the chapter headed *Output for Manufacture* as this is more relevant for PCB design plots.

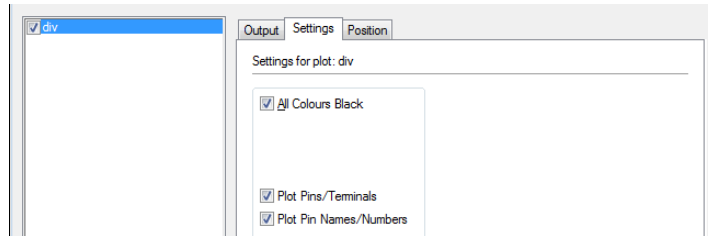
Setting up the dialog

There are three sub-pages to the dialog; **Output**, **Settings** and **Position**.

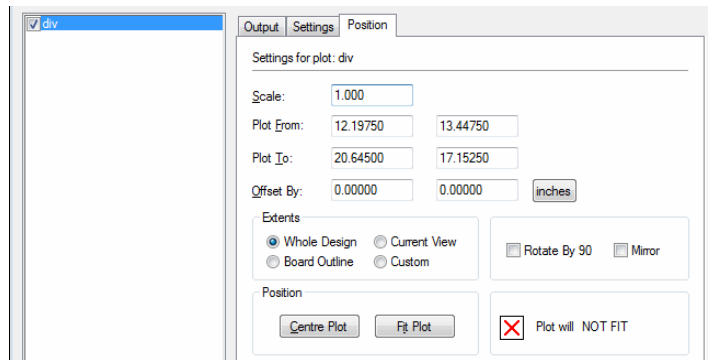
The **Output** page is used to define the device type and the plot type. It will also inform you of the device where a Windows printer type has been selected. This confirms that you are using the correct printer.



The **Settings** page provides you with general plot settings used when outputting the plot. Settings that can be used such as, using **All Colours Black** even when printing to a colour printer, plotting the **Pins/Terminals** and **Pin Names/Numbers**.

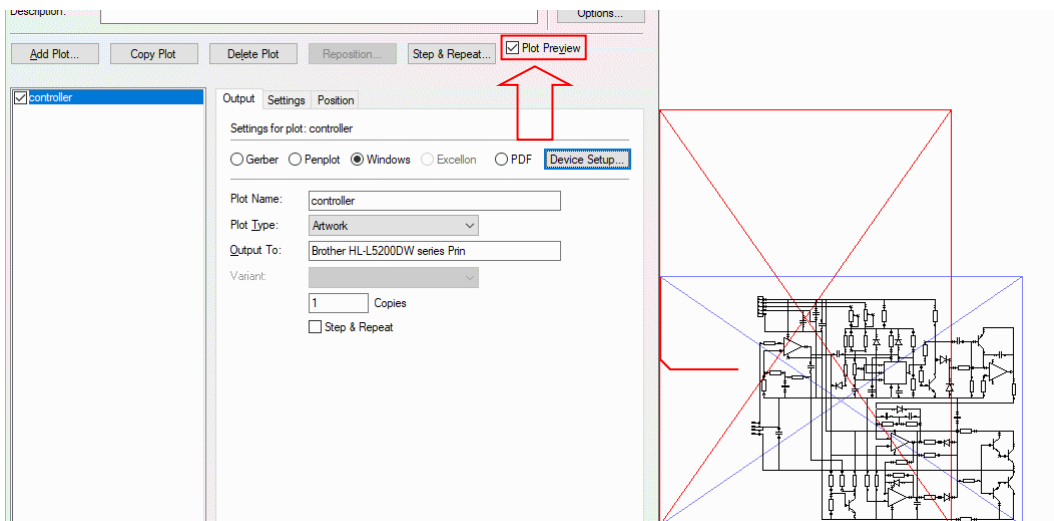


The **Position** page enables you to define how the plot will be positioned on the plot media.



For a Schematic design, clicking the **Centre Plot** and **Fit Plot** buttons will be sufficient. A plot will be generated that fits the page and is automatically scaled. If more precise prints are required, use the controls on this page.

The **Preview** check box can be used to display a preview of the selected plot.



Chapter 8. Creating PCB Designs

Getting Started with a PCB Design

There are a number of ways to start a new PCB design. The way in which you start will depend on whether you have a Schematic design, a netlist or a piece of paper with a schematic scribbled on it.

You can:

- Use the built in Schematic design editor to create the schematic and automatically translate it into the PCB design editor using the integrated libraries and translate mechanism.
- Use a netlist derived from another source, or even hand typed into a text file created following our netlist ASCII format.
- Use the PCB design editor interactively and create your PCB without using a schematic or netlist. The program is quite happy to allow you to do this; you cannot automatically create the equivalent schematic design from the PCB though.

The PCB design environment will allow you to take a 'logical' schematic design (we'll assume that under normal practice, the schematic is created first), translate it into the PCB editor, place the components, route it in order to get the 'logical' connectivity into a physically and electrically correct design and onto manufacture with the confidence that the design is correct.

In this chapter, we will also look at some of the interactive processes that may not be always applicable to the design process presented. For example, a Schematic to PCB design flow will not normally require the addition of component from within the PCB editor. This would be handled by the addition of new components in the Schematic design and forward annotated PCB. The process of adding components directly to the design will be covered to explain the process involved.

Starting the Design

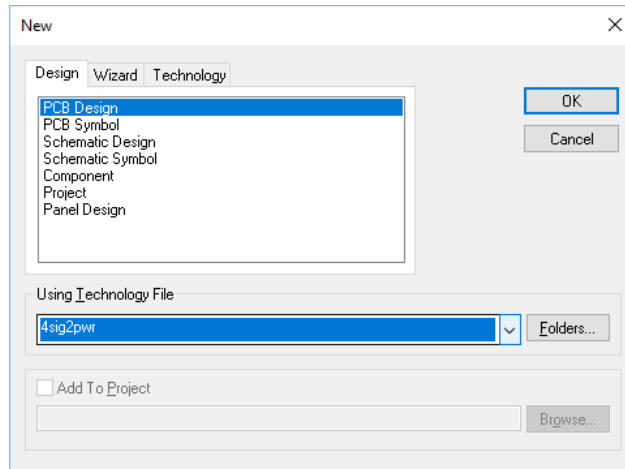
Regardless of where you start the PCB design from, you are required to provide basic information at the start point. This would normally be the **Technology filename** but this is optional. The Technology file is used to give you a fast-start of the design using predefined parameters, styles and settings.

Obviously, how you start the design is determined by the source of the component and net lists.

We will start with a new PCB design.

► To start a new PCB design

1. From the **File** menu select **New**



2. In the **Using Technology File** box, select a file from the drop-down list. Hovering your cursor over the name will reveal the full name.
3. There are a number of Technology files supplied, you can also create your own that can be added to this list. You may also choose [None] from this list.

If you are unfamiliar with this style of drop-down list, select the small black 'down' arrow on the right hand side of the box. This will 'drop down' a list of available files from which to select from.

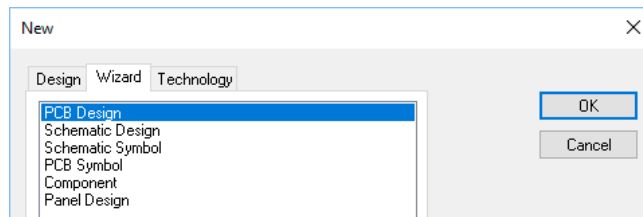
4. Click the **OK** button to start the new design.

The PCB design editor is now ready with a Technology file loaded ready for use. From here you may add a netlist or you may choose to start entering the design by hand on-the-fly by adding components and nets etc. We will look at starting the PCB design using **the Translate To PCB** option and from netlists later on in this chapter.

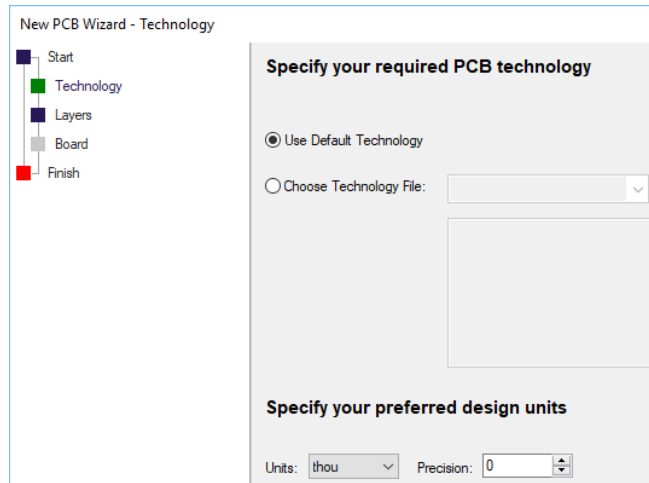
Alternatively, you can also start a new design by using the **New, Wizard, PCB Design**. This will guide you through each stage of the process.

► To start a new PCB design using the PCB Design Wizard

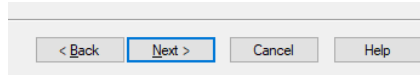
1. From **File** menu select **New, Wizard, PCB Design**



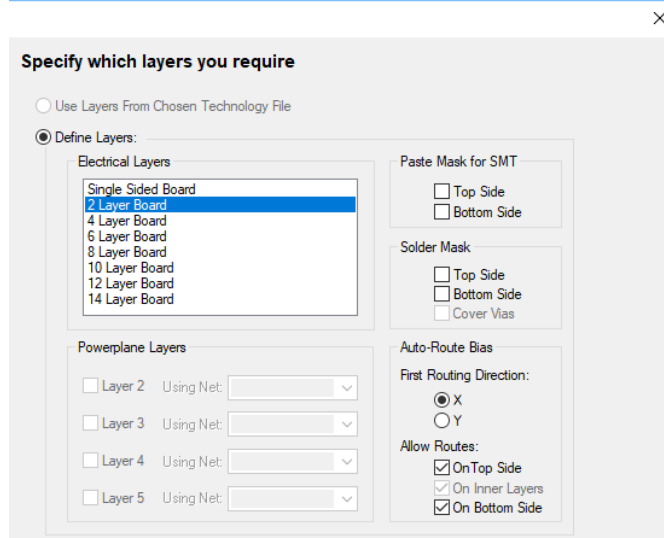
2. This will start the Wizard
3. The Technology page enables you to choose the **Technology** file required and the **Units**. Your Units would be set in the technology file anyway.



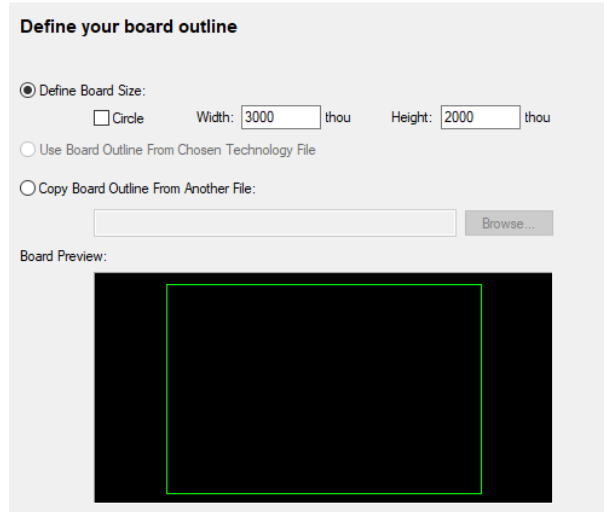
4. Use the **Next >** button to move onto the next page.



5. From the **Layers** page, either use the layers defined in your Technology file (selected on the previous page) or **Define** the **Layers** on this page (you may use a technology to set up styles only for example).



6. The **Board** page enables you to define the board outline that you require:

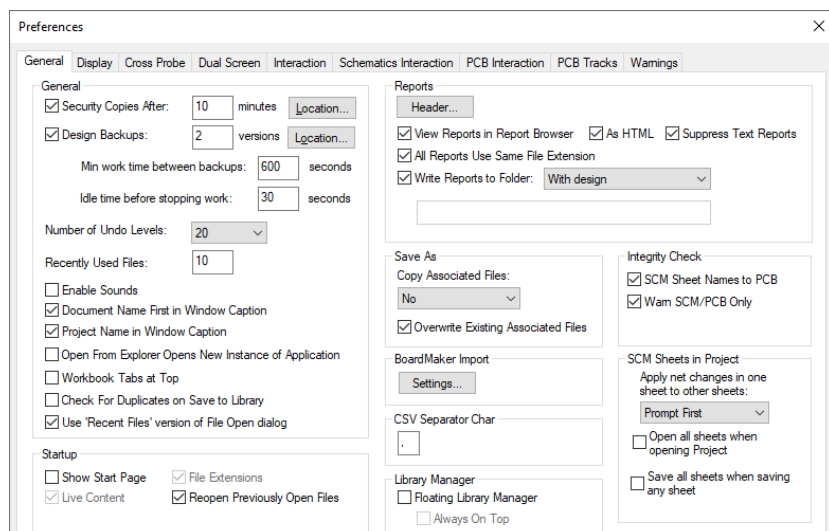


7. Define a basic Rectangle, Square or Circular board in your chosen units. You may also select the board outline from your Technology file if it had one defined or Copy a board outline from another design.
8. The **Board Preview** window verifies your selection.
9. The **Finish** page enables you to type a **Design name** and to **Save** it to your hard drive.
10. Once **OK** is pressed, your design will open using your selections.

Design Setup for PCB layout

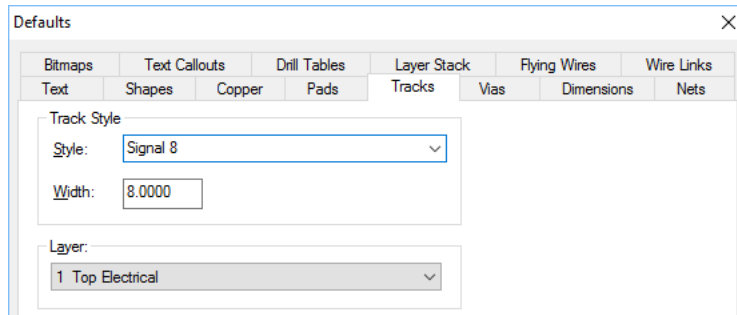
There are a small number of design settings that can be set before the design is started.

Many of the characteristics of the design items are set up in the **Preferences** dialog. You check settings in the **General**, **Interaction** and **PCB Interaction** pages.



Default Styles

The **Defaults** have been discussed previously but in brief, this dialog allows you to define how items with in the design will be added for the first time. Now is a good time to check and adjust them before starting your new design. They can be found on the **Settings** menu.

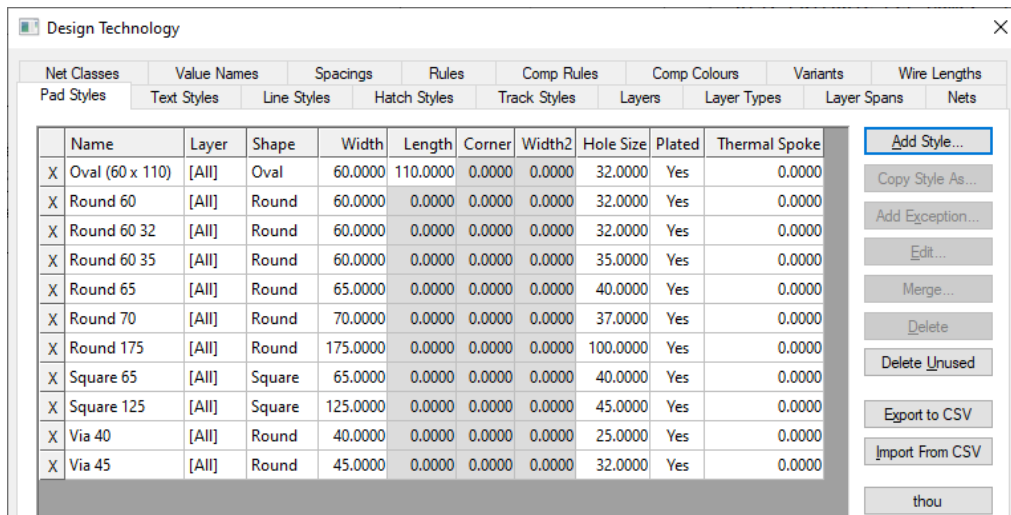


Setting up a PCB Technology

The concept of Technology files has already been discussed in previous chapters. This section details specific items for the PCB Technology file.

Pad Styles

Pad styles, regardless of their type, can be defined from the **Settings** menu, **Design Technology** under **Pad Styles**. It is recommended that the **Sizes**, **Shapes** and **Drill Holes** are defined as a style so that they can be reused for other Pads or Vias where the same pad sizes are required.



The names used for these pad styles should be something recognisable to you so they can be recalled and used without having to think what size etc. has been defined. For example, Vias could be called something like, **Via 45**, for a 45 thou diameter via, **Pad Round 35**, **Free Pad 60**, etc.

Where the layer is shown as **[All]** this means that the default pad 'stack' is shown for all layers except where an 'exception' is shown for the same pad.

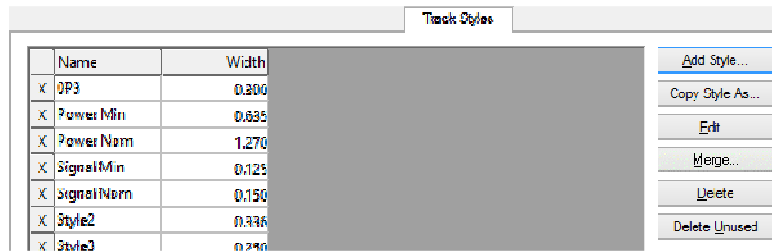
Where pads are defined, you define the pad as it appears on all layers – called the pad 'stack'. You may change part of the entity for that pad for a given layer, for example you want a pad stack of 60 thou, Round with a drill of 35 - this might have a pad size for the GND power plane layer defined as 80 thou instead of the 60 'default'.

The exception can be any of the shapes allowed and the size but not the drill size. The drill is always defined for the layer stack regardless of exceptions. The exception can contain zero sized pads on layers but will still be drilled (if a drill has been defined).

The pad length can only be adjusted for asymmetrical shapes, otherwise it is defined as zero size, shapes such as rectangle.

Track Styles

Track styles are defined from the **Settings** menu, **Design Technology** option and **Track Style**. However, track styles are used as part of **Net Classes** that are assigned to connections and tracks.



Track styles can be changed at any time during editing or on selection by selecting **Style** from the context menu or by using the shortcut key <S>. It is more organised to retain net classes and track styles rather than creating track styles on-the-fly but for expedience this can also be done. Any local track style added can be added to the Styles table anyway automatically. This is done when the track style is edited using the confirmation prompt.

Track style widths can be set to any value but it is better to use standard widths for the designs in hand, and also with the manufacturing rules and requirements in mind at all times.

Track styles and Net Classes

By default, the net class used for tracks and connections is taken from the **Settings** menu, **Defaults option** and **Net** entry. This means when a track or connection is added to the design, the Remembered Style for Net Class is used, which in-turn has a Track style defined within it.

Connections that are translated from the Schematic will generally (should) have a Net Class assigned to them, when don't, the Default for Net is used.

Spacing Rules

Spacings rules on the **Design Technology** define how close one type of item is allowed to be to another item. For example, Tracks to Pads or Vias to Pads. These are used by both the **Online Design Rules checking (Online DRC)** and (batch) **Design Rules Checking (DRC)** options.

They relate to the distance between objects in the design, a Track to a Pad for example. These rules relate to the possible tolerances used when actually manufacturing the PCB. If changing

these, it is highly recommended that you consult with the person manufacturing the PCB for you. If you change these clearances, you may not be able to get the board made!

	Tracks	Pads	Vias	Shapes	Text
Tracks	8.000	8.000	8.000	8.000	8.000
Pads	8.000	8.000	8.000	8.000	8.000
Vias	8.000	8.000	8.000	8.000	8.000
Shapes	8.000	8.000	8.000	8.000	8.000
Text	8.000	8.000	8.000	8.000	8.000
Board	50.000	50.000	50.000	50.000	50.000

Rule Level

Design
 Net Match
 Stop at first Net Match

Check Against Same Net

Use Board Centreline

Use Track spacings for Teardrops

Practical Use

When creating designs, 10 thou (0.254 mm) is a good relatively low cost clearance for all conductor patterns for photographically produced boards. For boards produced using more basic methods you may need to look at higher clearances, around 15-20 thou perhaps to ensure that copper is sufficiently clear. Allow at least 20 thou clearance to the board edge for tracks and 100 thou for component pads (which allows a margin for holding equipment in the assembly and flow soldering processes).

Design Rules

In addition to Spacing rules, you also have design Rules. These are available on the **Rules** tab under the **Design Technology**. Design rules are additional rules that need to be considered when manufacturing a PCB. They also relate to general design rules for use when actually design the PCB.

Power Planes

<Default>

Prefer Orthogonal Spokes

Isolation Gap: 0.254

Spoke Width: 0.254

Number of Spokes: 4

Minimum Spokes: 2

Pads and Drills

Drill Spacing: 0.000

Min Annular Ring: 0.127

Min Paste Size: 0.076

Min Via Annular Ring: 0.127

Min Hole Size: 0.000

Min Via Hole Size: 0.000

Via to SMD Pad: 0.000

Tracks

Minimum Track Width: 0.051

Min Neck/Fatten Length: 6.350

Component Height Checks

Top Max Allowed Height: 0.000

Bottom Max Allowed Height: 0.000

Value Name: Height

Component to Component Spacing

Top Side: 0.000

Bottom Side: 0.000

Boards

Board Spacing: 0.000

Panel to Board: 0.000

Test Land Checks

Minimum Pad Width: 2.540

Minimum Separation: 6.350

Probe Side: Bottom

Acid Trap Checks

Minimum Allowed Angle: 90.00

Minimum Allowed Gap: 1.270

Pad Style Exceptions

Mirror with Component

mm

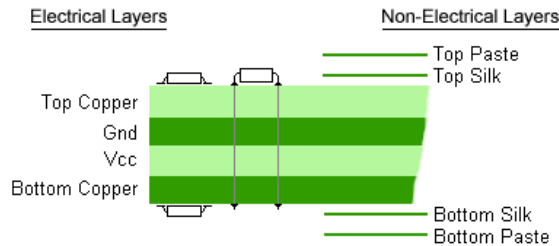
Layers

An unlimited number of layers is supported in a PCB design. This limit is regardless of whether the layers are construction, documentation, electrical or non-electrical or any combination of these. Layers are normally defined to detail the physical make-up of the PCB, the board that will eventually be manufactured.

As a flexible system, you can add or remove layers from a design as you require, but remove can only be used on unused layers. Generally speaking, the loading of your technology files will pre-load your layer choice, the number of layers that will be used. You can always add to this afterwards but it gives you a good starting position.

Layer construction

The picture below shows a 4 layer PCB showing the 4 electrical layers (Top Copper, Gnd, Vcc and Bottom Copper), plus the basic non-electrical layers. Non-electrical layers are used to define layer information for solder paste masks, or solder resist masks for example. Additional non-electrical layers can also be used for documentation purposes.



Layers that are defined as non-electrical can be defined as being on the Top side, Bottom side or Inner layer.

Any number of electrical layers can be contained within a design and it is not restricted. The number and type of layers that the PCB design editor uses is controlled by the use of either a technology file or by starting with (no technology file) loaded [None] and creating layers, styles, colours etc. from scratch. We have supplied a range of default technology files to aid you with the starting process.

If we take the most basic technology file (Default.ptf), this contains two electrical layers (Top and Bottom Electrical) and two non-electrical Silk Screen layers (Top and Bottom Silk Screen). This example is shown below:

Name	Type	Side	Usage	Bias	Net	Colour
Wires	Wire	Top	Wire	None		
Top Silk	Silk Screen	Top	Non-Electrical	None		Yellow
Top Copper	Electrical	Top	Electrical	X		Red
Dimensions	Dimensions	Top	Non-Electrical	None		Green
Layer 2 Copper	Electrical	Inner	Electrical	Y		Red
Gnd	Electrical	Inner	Electrical	Power Plane		Green

Layers can be named anything you like but it is better to give layers a sensible name, something meaningful and reflective of its function or its physical layer topology, e.g. Top Electrical, Top Silkscreen etc.

The example below shows a 4 signal and 2 power plane layer setup (6 electrical layers in total):

Name	Type	Side	Usage	Bias	Net	Colour
Wires	Wire	Top	Wire	None		
Top Silk	Silk Screen	Top	Non-Electrical	None		Yellow
Top Copper	Electrical	Top	Electrical	X		Red
Dimensions	Dimensions	Top	Non-Electrical	None		Green
Layer 2 Copper	Electrical	Inner	Electrical	Y		Red
Gnd	Electrical	Inner	Electrical	Power Plane		Green
Vcc	Electrical	Inner	Electrical	Power Plane		Grey
Layer 5 Copper	Electrical	Inner	Electrical	X		Grey
Bottom Copper	Electrical	Bottom	Electrical	Y		Cyan
Bottom Silk	Silk Screen	Bottom	Non-Electrical	None		Blue

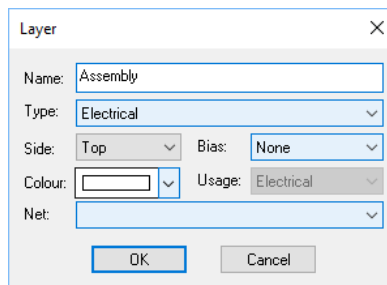
Layers are not numbered but are instead named using a name. It is normal to assign these with logical layer names. They should appear in the **Layers** dialog in the physical order in which they are manufactured. This logical approach will make it easier for you to quickly see and locate layers by name and sequence. Because many of the options that use layer names are drop-down lists or selections of lists, the names and length of names are unimportant; you do not need to remember their names!!

Creating Layers

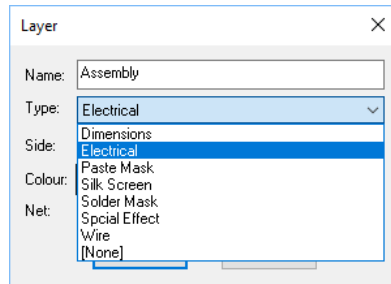
Before any design item (other than a board outline) can be added to a PCB design, there must be at least one layer available on which to place it. If a technology file is in use, this will normally define the necessary layers for you, otherwise they must be added to the design.

► To create new layers

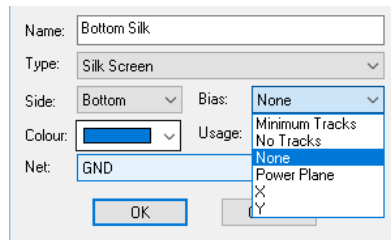
1. From the **Settings** menu select **Layers**.
2. Select the **Add** button.



3. Type in a **layer Name**. Usually this will be a name to define the layers functional in the final design, Ground, Solder Mask etc.
4. All the other options on the dialog use drop-down menus. They will reveal a list of options from which to select. To access this selection, click on the small 'down' arrow on the right side of the list box.
5. From the drop-down menu, select the Layer Type. The layer types offered will be as defined in the Layer Types dialog. A standard set will be offered if using one of the supplied technology files.



6. Choose the layer type from the list.
7. From the Side list, you must choose the side that this layer will be related to. You can only choose Top, Inner or Bottom.
8. The bias setting is principally used to describe layer properties for autorouting and for power planes.
9. From the **Bias** list, choose from Minimum Tracks, No Tracks, None, Power Plane, X or Y. For the layer to behave correctly as an intended power plane, the layer bias must be defined as Power Plane.



10. When a layer is defined as a power plane, it will be necessary at some point to define the net name associated with the plane.
11. The Colour selection is used for the new layer default (for all items on that layer), it can be changed afterwards in the **Colours** dialog.
12. Usage will be pre-defined from the **Layer Type** selection made previously. This box is simply a display of the setting and is not editable in this dialog.
13. Net is used to define a net name associated with the layer. This is only required where the layer Bias is defined as Power Plane.
14. When complete, the new layer is added to the list and other layers shuffled around it so that it slots in into its logical physical order within that category, but not outside it.
15. Click the **OK** button to save the Layer changes.

Remember that most designs will need at least one Electrical and one Silk Screen layer as a minimum design requirement.

Layer Types

The **Layer Types** page is accessed from the **Design Technology** dialog on the **Settings** menu. This allows you to define a type of layer and its usage for down-stream processes, e.g. colour display or manufacturing output selections etc.

	Name	Usage	Surface Pads	All Layer Pads	Surface Sym. Pad	All Layer Sym. Pad	Vias	Oversize	Size	Placement	Break Shapes	Break Gap
X	Silk Screen	Non-Electrical	No	No	No	No	No	None		No	Yes	9.00
X	Electrical	Electrical	Yes	Yes	Yes	Yes	Yes	None		No	No	0.00
X	Dimensions	Non-Electrical	No	No	No	No	No	None		No	No	0.00
X	Wire	Wire	Yes	Yes	Yes	Yes	Yes	None		No	No	0.00
	Special Effect	Special Effect	Yes	Yes	Yes	Yes	Yes	None		No	No	0.00
	Paste Mask	Non-Electrical	Yes	No	Yes	No	No	Absolute	-5.00	No	No	0.00
	Solder Mask	Non-Electrical	Yes	Yes	Yes	Yes	Yes	Absolute	5.00	No	No	0.00

The **Name** used for the layer type is the one that appears in the **Layers** dialog under **Type**, it is **not** the one used for the layer name, although they can be the same if required. **Layers** names are unique whereas **Layer Type** names can be used for many layers.

The Layer Types can be **Added**, **Edited** or **Deleted** using the three buttons on the dialog.

From the **Add** or **Edit** button you can make the changes:

The **Name**: field is the name of the Layer Type and denotes its usage, not the plot name.

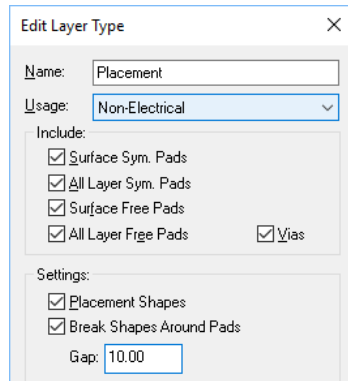
The **Usage** of the type can be **Electrical**, **Non-Electrical**, **Wire** or **Special Effect**. Electrical is only used for a physical electrical layer, for example Top Copper or Inner Layer 2 etc. Non-Electrical is used for layers such as Silk Screen or Solder Resist. Wire is used for displaying wires that are used for jumpers when the Wire option is used (these are not connections or nets). Drawing will be used when the Real Life PCB view is used.

The selections below Name and Usage define the display and output of pad types within the design and their eventual usage. These switches have an effect on the design view. They control what is viewed and in conjunction with the **Colours** option will provide control and views of the specified layer with the pads/vias showing on or off depending on the selections made.

When used with the **Output** menu option **Plotting & Printing**, the **Oversize** settings will affect the pads/vias output to printer or plotter. These settings for the specified layer override the **Plot Pads Only** switch settings in the **Output** options regardless of the switch settings.

Layer Types for placement areas

Any shape on the top or bottom layer can be defined as **Placement** using the **Layer Types** dialog. A check box under **Layer Types** for **Placement Shapes** should be checked to enable this feature.



Once checked, when the **DRC** option for **Spacing** and **Components** is selected, the shapes defined on that specific layer will then be checked. If a component does not have a placement area defined on one of the checked layers (top or bottom), the bounding box of the component extents will be used.

In the example above, we have created a new layer called **Placement** which will have shapes that will be added to the footprint and which will be used to check the placement rules. You could simply nominate the Silkscreen layers as **Placement Shapes** if you wish to use existing shapes.

Layer Spans

Blind and buried via technology is supported. This technology support allows two additional layer spans to be defined as well as the 'normal' through-hole span.

Blind vias are vias that span an external layer to an inner layer, for example Top Electrical to Inner Ground.

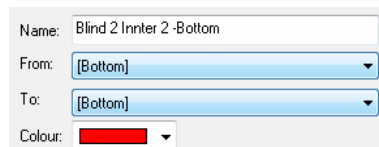
A **Buried via** is a via which spans two **internal** layers, for example Inner 2 to Inner 3.

Both, **blind** and **buried** vias are considered more advanced technology and should only be used after consulting your manufacturer as there may be cost implications when being produced.

To add and modify layer spans, use the **Layer Spans** dialog, this is available from the **Settings** menu.

Name	From Layer	To Layer	Colour
[Through Hole]	[Top]	[Bottom]	Yellow
Blind Top > Inner 2	[Top]	[Top]	Blue
Blind Inner 5 > Botton	[Bottom]	[Bottom]	Red

When the **Add** or **Edit** buttons are pressed, the **Layer Span** dialog is displayed:



If you enter an illegal or un-manufacturable layer span, the program warns you with a message:

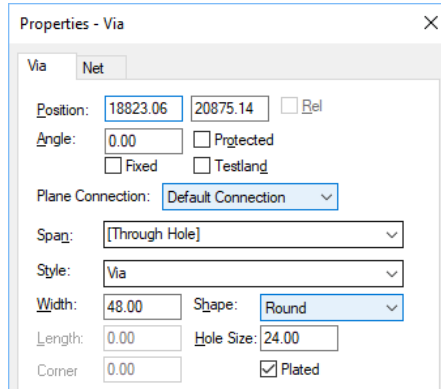


Layer span "Blind 2 Inner 2 -Bottom" starts and ends on layer "[Bottom]".

You should only use this arrangement if your particular manufacturing process requires it.

When using this dialog, the <Through Hole> span is automatically added.

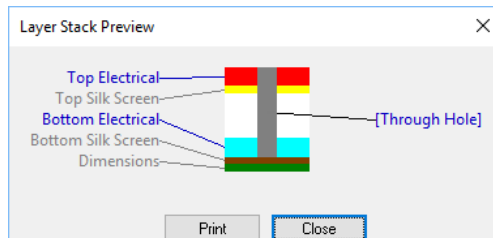
Via Properties allow the changing of layer span and finding the best fit span.



When editing a track, the interactive change of track layer will use the best via span.

Layer stack Preview

The **Preview** button on the **Layer Spans** dialog displays a graphical representation of the layers defined in the design.

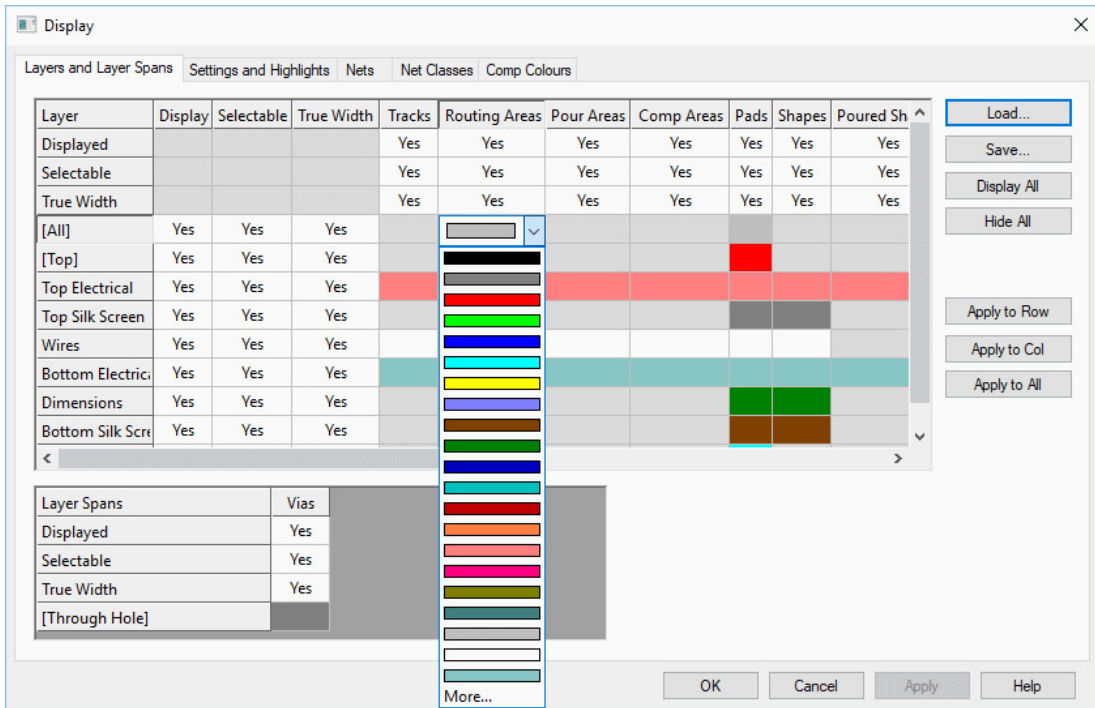


The **Print** button can be used to send this picture to a printer. You can also add this picture to the design using the **Add Layer Stack** option from the **Add** menu.

Changing Colours in PCB

Like Layers, the colours of items within a design should be assigned before commencing the design process. The colours can be edited from the **View** menu and **Colours** dialog.

This dialog is split into sections; the first, **Layer and Layer Spans** is for defining the colours of items that are layer specific, i.e. Pads, Tracks etc.

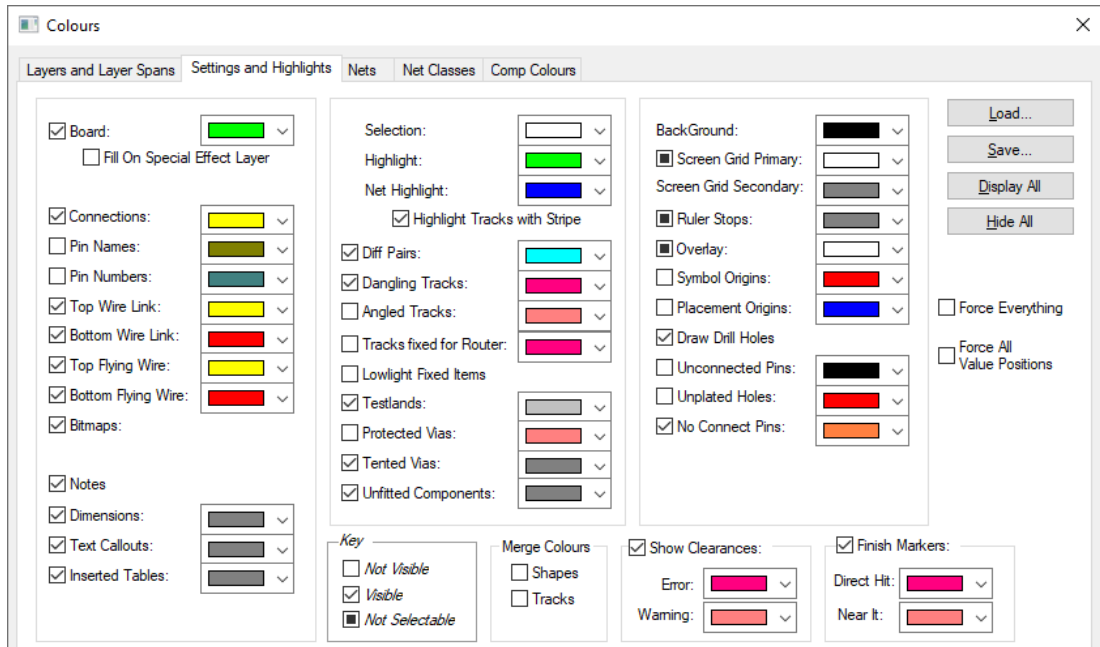


You can change the colours of each cell by double-clicking in the cell and selecting the colour from the drop-down list.

Colours should be created that contrast between different electrical layers, e.g. Top and Bottom electrical layers should be distinctive against each other. This contrast will make it easier to identify layer changes and electrical differences.

You can also specify that items are Selectable as well as being displayed. This means you can make items displayed but when you try to pick them they are not pickable. This could be useful if editing tracks and you don't want to pick pads for instance or text.

The second page, **Setting and Highlights**, is used to define non-layered items and generic design items, such as the board outline, connections, pin names etc.



The **Merged Colours** items will help you contact the layered items that ‘cross’ each other in the display. Obviously, they don’t physically cross because they are on different layers.

Colours files can be **saved** and **loaded** from this dialog. These are saved with the .ECS extension and when saved to the colour files path can also be recalled to the design using the **Colour Files** dockable **Preview** bar.

The **Nets** tab is used to define net settings. A net can be drawn in its **own colour** and selectively displayed using **Show Connection**.

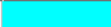

Name	Own Colour	Colour	Show Connection
+12V	Yes		Yes
-12V	Yes		Yes
A0	No		Yes
A1	No		Yes
A2	No		Yes
A3	No		Yes
A4	No		Yes
A5	No		Yes

The **Net Classes** tab is used to set the colour individual Net Classes within the design.

Name	Own Colour	Colour
Signal	No	
Power	Yes	
Ground	Yes	

The **Comp Colors** tab allows you to set up 'rule based' colours for the components in your design that will override the colours specified by the shapes, text or layers in the symbol definition.

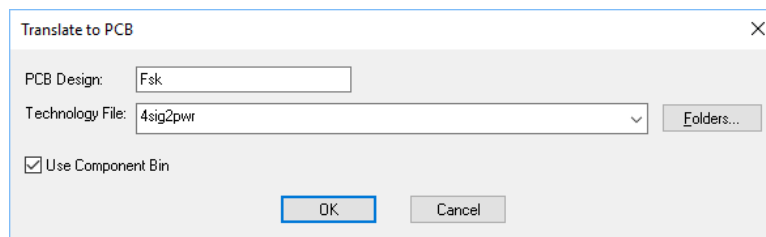
These can be used for example to construct an Assembly drawing where different categories have their own colour, or to draw components matching a particular Value in a bright colour to assist with things like component obsolescence.

Enabled	Value Name	Match	Type	Layer	Swap Layer	Colour
Yes	FRAGILE	*	All	Top Silk Screen	Yes	
Yes	OBSOLETE	*	Shape	Top Silk Screen	Yes	

Translating from Schematics

Within the integrated environment you can translate the Schematic design into the PCB layout editor with a simple button click. There are no netlists to use or other commands to use.

The **Translate To PCB** option is available from the **Tools** menu within the Schematic editor.



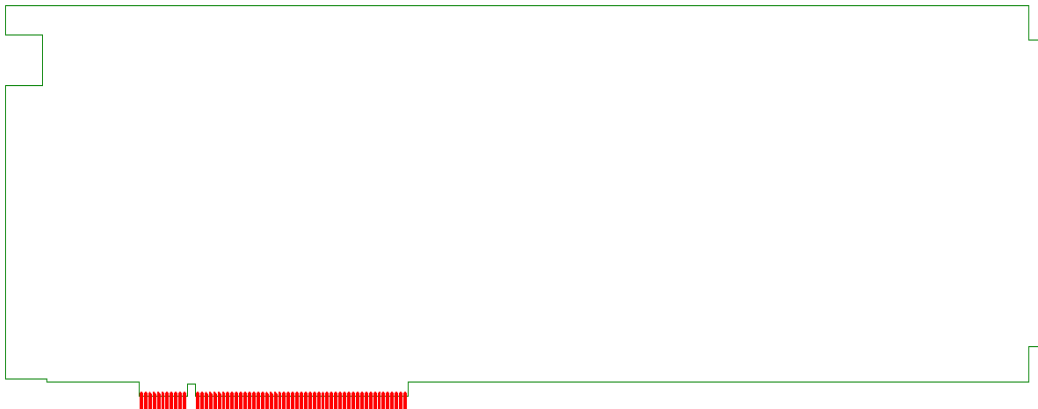
By setting up this dialog, this will give you the name of the PCB design, the Technology File and whether or not to use the Component Bin.

Once this is completed, you are ready to start the component placement and track routing.

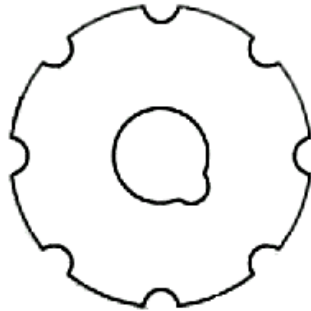
The **Translate To PCB** option has been discussed in detail at the end of the previous chapter, *Creating Schematic Designs* under *Transferring Schematics into PCB Layouts*.

Adding a Board Outline

The board outline (or Profiles as they are sometimes called) is used to define the edge of the PCB design. This will often be the shape or size required to fit the PCB into a box or enclosure.



You can create board outlines in a number of methods. You can use the **Add Board** option to simply create the shape or you can use the **Import DXF** option to use an existing shape created in a mechanical package such as AutoCAD where the shape maybe very complex.



This shows an example of a more complex board outline imported via the **DXF import** option.

Using Add Board

Board outlines can be created using the **Add Board Rectangle/Polygon/ Circle/Line** options from the **Add** menu. The board is added directly to the design and created on the fly. You can also add board using the Add Board option from the **PCB Design** toolbar. The **Line shape** style allows you to create a simple line board item that could be used for board detail for example.

Board outlines are created using an **Unfilled** shape that is normally **Closed**, i.e. the start and end point are the same, unless you use the Line shape style. They cannot be self-intersecting where part of the shape extends outside itself (the option will check this for you when you finish the shape). You may edit a shape after it has been added by double-clicking the shape segment.

Board outlines are not layer biased in that they appear on all layers.

The Board outline default style is taken from the **Defaults** dialog and **Lines** page on the **Settings** menu. Whatever is defined here will be used as the initial board outline. You can of course change this afterwards by selecting the shape and using **Properties** or **Change Style** from the context menu. **Defaults** are saved with the Technology file.

If you require cutouts in your board then you should use the **Add Board** option to create this cutout shape, or use **Add Copper** to define the area of cutout. The advantage of using copper is that other items placed near or on it will be checked during a Design Rules Check.

► To add a board outline

1. From the **Add** menu select **Board shape**. Select the shape type required.
2. Click to add the first corner or the centre point of a circle (if Add Board Circle was selected).
3. If Add Board Rectangle option was selected, move the mouse to create the opposite corner and click to finish adding the shape.
4. If the Add Board Polygon option was selected, click to add the first corner, each subsequent click will add another corner. You will notice that during the mouse movement, a trailing line segment joins your current position back to the start position. This indicates that the shape is closed.
5. To finish this shape, double-click the mouse.
6. If the **Add Board Circle** option was selected, click to start the circle at its centre point, then move the mouse to the required diameter. Click again to finish the circle.

7. At any point, if you want to exit or make a mistake and need to start again, press the <Esc> key on the keyboard.
8. During the addition of the board shape, if you right click you will see options available on the context menu.

It is recommended that you create board outlines in the centre of your working area although this can be changed using the **Working Area** option later on.

Adding 'Free' Pads

There are a number of pad types that can be used during your PCB design. For components, regular **Pads** are used. **Vias** are automatically added during a layer change when using **add** or **edit** track. Vias can also be added interactively to copper for example, if stapling is required (the process of attaching extra vias to an electrical item and plane).

Free pads are pads that are not part of components and are not vias but can be used as additional pads within the design to define special uses such as **mounting holes** etc. They should not be used for component pins or vias as they have no pad number and are not recognised by other options.

These pads take their style from the **Design Technology** option and the **Pad Styles** page, their styles can also be set locally as can other normal pads and vias. Local means a style on that item only and not added to the Design Technology.

Free pads can be connected to, but are not identified with a component reference number or pad number. If connected into the design, for example a mounting hole might be connected to GND to earth it, this additional connection will not be checked in the integrity check.

Free pads can be added to the design using the **Add Pad** option from the **Add** menu, or from the **Add Pad** option on the **PCB Design** toolbar.

When a free pad is selected, there are additional options on the context menu that can be used.

Adding Components

Once the technology file has been loaded and the board outline created, the next process to attempt is the insertion of components into the design. Components are a fundamental design item for PCBs. As with most design items there are numerous ways to work with them.

Components can be added interactively from the **Add Component** option, or loaded automatically from the library during the **Translate to PCB** phase without using an existing netlist or using a Schematic design.

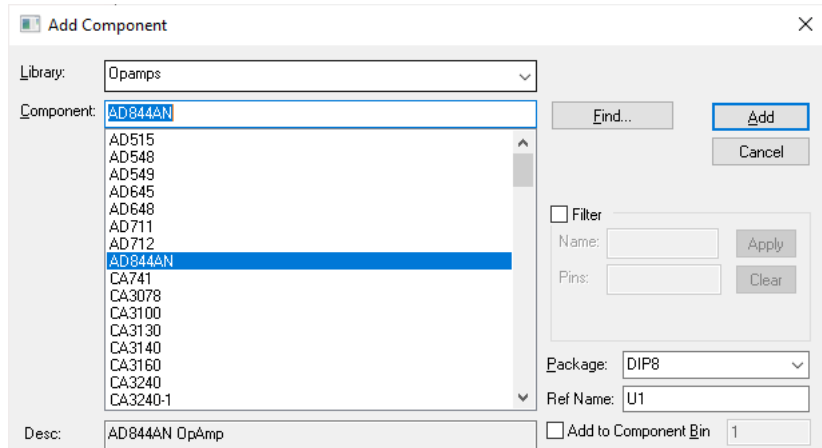
Components are used in both the Schematic and PCB design editors and are used to reference both the Schematic symbol and the PCB symbol depending which design editor is being used at the time. For components to be used, you must have a PCB Symbol. You may also have an associated Schematic Symbol with this component. The actual creation of components is covered later on in the chapter *Library Creation*.

Within the PCB design editor, PCB symbols are used. These are 'footprints' of the pad land patterns and Silk Screen details required for the placement of the actual devices onto the manufactured board. These symbols are made up of the footprint detail and stored as a single item within a PCB symbol library.

As well as adding components to the design, if you previously translated the design from the Schematic editor, you may have components located in the Component Bin ready for placement. This option will be covered a little later on in this section.

Adding Components Interactively

The **Add Component** option can be found on the **Add** menu, on the **PCB Design** toolbar and with the shortcut key <F8>.



This dialog allows you to search libraries and find components, then add them to the design or component bin.

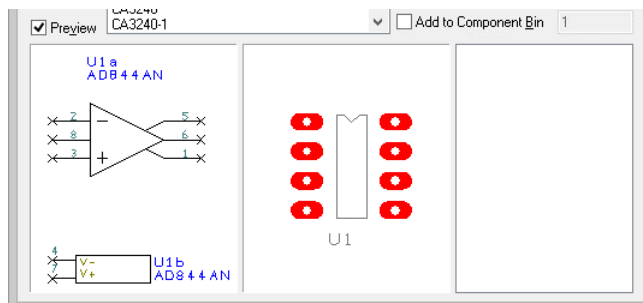
The **Library** list allows you to choose the library to select components from. From the drop down list, you can also choose [All Libraries]. This will display the components from all of the libraries found in the search path defined in the **Library Manager** under **Folders**.

Once a library has been selected the **Component** list will update with all the components in the selected library. If you wish to use specific search criteria for components, the **Find** button and options within this will enable you to do so. The Find dialog allows you to search libraries and enter names based on wildcard names. This dialog is described in more detail below.

The **Add** button will place it on the end of your cursor ready for positioning in the design. You will notice it will snap to the screen grid while placing. You can adjust the snap mode using the <H> key and by selecting one of the snap modes from the context menu.

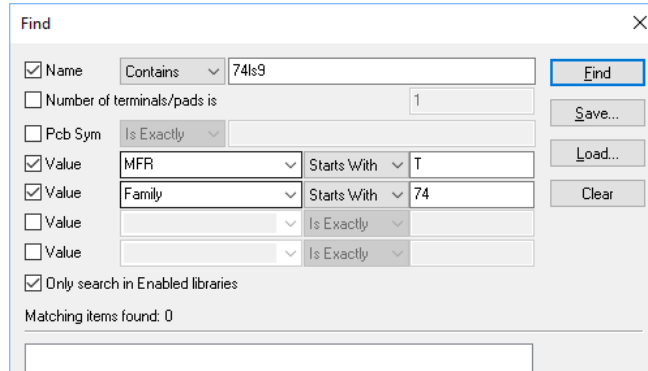
Alternatively, you can use the **Add to Component Bin** check box to send the selected component directly to the component bin. The **Count** entry will allow you to specify multiple copies of the selected component.

The **Preview** button can be used to show you the component before it is placed on the design, it also shows you a preview of the PCB Symbol. Once you create your own library you will probably not use this unless you need a reminder of what they look like.



Using the Find dialog

When the **Find** button is selected it displays a dialog that allows an item name to be found in the libraries. It then searches the libraries for the given Component/Symbol and selects the appropriate library and item. It searches the current library first, then all the libraries in the library directory.



Matching on the library item **Name** can be done in three different ways:

- **Is Exactly:** only matches items with exactly the name specified.
- **Starts With:** only matches items that start with the given text.
- **Contains:** only matches items that have the given text anywhere in their name.

The **Number of terminals/pads** field allows you to limit the search to match only those library items with the specified number of pins.

For the **Value** field, when searching for components, you can also choose to search for items that have a particular value. Leave the 'is' field blank to match all items that have this value regardless of its actual setting, or enter the desired setting to search for a specific value.

Choose **Only Search in Enabled Libraries** to refine the search into enabled libraries. If not selected, you will be able to search all of your libraries in the **Folders** path.

On pressing **Find**, there may be a delay whilst any library index files are updated. Having done this, the libraries are searched for matching items. Any matches found are listed on the dialog, showing their name and library file and for components their 'package' as well.

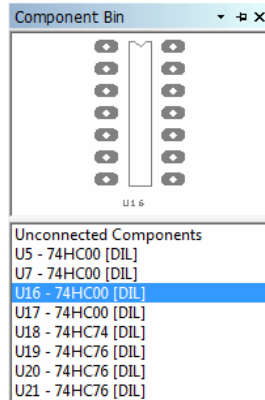
To select the desired item, simply click on it in the list. As you select the item, the corresponding 'master' dialog (Libraries, Add Component, etc.) will update its details and preview to show you the selected item. To close the Find dialog, either click on **Close** or double-click the required library item in the list.

Using the Component Bin

The **Component Bin** bar is available as a sliding window. This feature provides you with a facility for holding unplaced components in an off screen location. This frees up the design space while placing other components and generally makes it less cluttered, especially on dense designs.

The component bin has been discussed in detail in the previous chapter on *Using the Product*.

The component bin itself consists of the component bin preview and the component bin. It is used to store and view unplaced components in the PCB design editor.



The **Component Bin Preview** window shows the component currently selected in the bin, along with its pad numbers and names. This updates automatically as each component within the bin is selected.

The **Component bin** window lists the components currently in the bin. This shows the component reference name, the component name and component type and is sorted alphabetically on the component reference name. The components are selected by clicking in this list.

Placing Components on to The Design

To place components onto the design while they are located in the component bin you can click on the required component in either the Preview window or the Bin itself, and drag it across onto the design.

Note: the drag may be cancelled either by dropping the component back into the bin, by pressing the <Esc> key or once released, by selecting Undo (<Ctrl Z>).

In summary, components can be placed in the bin:

- From the **Add Component** dialog.
- By dragging the component from the design to the bin.
- By selecting a component in the design and choosing **Move To Component Bin** from the context menu.
- From the **Autoplace** option dialog.
- During the **Translate To PCB** option.

In a PCB design, **layers** have to be considered when adding components from the library. If the PCB symbol has shapes or text on layers that do not exist in the PCB design, these shapes and text items will be removed from the Component used in the PCB design. The matching is performed by layer type and side.

Updating Components

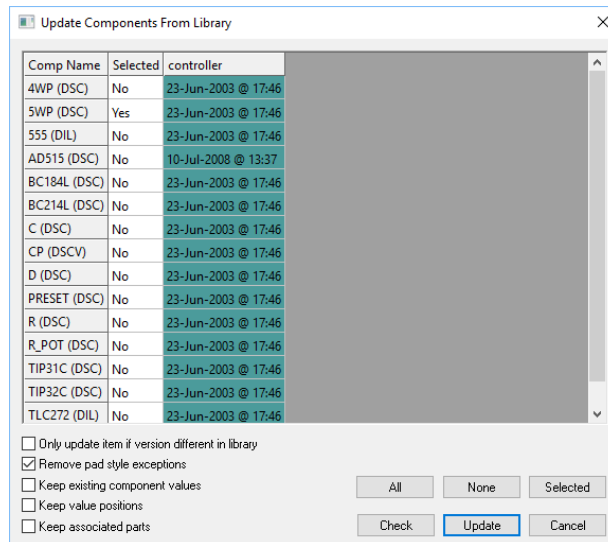
Components and Symbols can be updated or changed in the design using two features: **Update Component** and **Change Component**.

Update Component

Update Component is available on the context menu when a component is selected and also on the **Tools** menu. It is used to update the current component on the design to an updated component from the library. This feature can also be used to force component updates from the library if required, to globally replace all components to the latest ones that exist in the library.

Components within a design are always unique as are Symbol instances. A design cannot contain a DIL14 for example that has two different shapes or pad shapes they must always be identical. Once a component or symbol is added to the design, the next addition of that item will be duplicated from the design not accessed from the library.

On selection of the **Update Component** option, the dialog presented allows a number of key functions to be performed.



Firstly, it enables you to select particular devices to update, or you can choose them all using the **All** button. Then check boxes allow control of the facets within uploaded components:

Only update item if version different in library effectively allows you to 'force' an update even if the library items are the same. Leave the check box unchecked if you wish to force the update.

Remove pad style exceptions is used to control local pad style exceptions that have been added to components that are to be updated from the library. It is also used if the technology file used for the library creation and the design technology files are different.

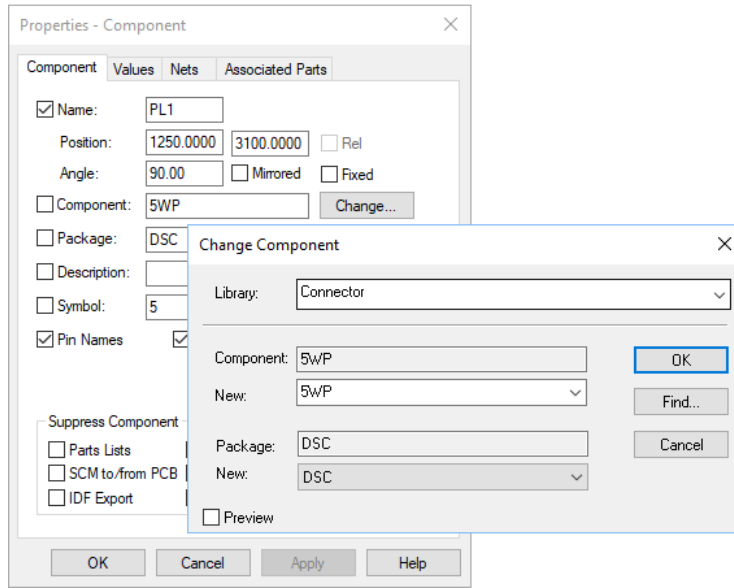
Keep existing component Values is used to control local attribute values which have been added to the components which are to be updated from the library.

Keep value positions If this is checked the positions and layers of the components value positions are retained. Otherwise the value positions will revert to the symbol or footprint defaults.

Keep associated parts If this is checked, any associated parts for that component are retained.

Change Component

Change Component is available from the **Properties** option on the context menu when a component is selected. By selecting this option, you are able to change components and/or Packages for that component in the design.



The dialog gives you options to change the **Component** or the **Package** from a selected library. The information displayed shows the item before and the changed item. **Find** allows you to select different library items using the Find facility.

The 'Delete' option is not to be used!

The net effect for both of these options (**Update Component** and **Change Component**) is that if a component or PCB symbol is incorrect in the design, instead of removing the device and its connections/tracks, you simply replace the component or symbol for the one that is required or updated.

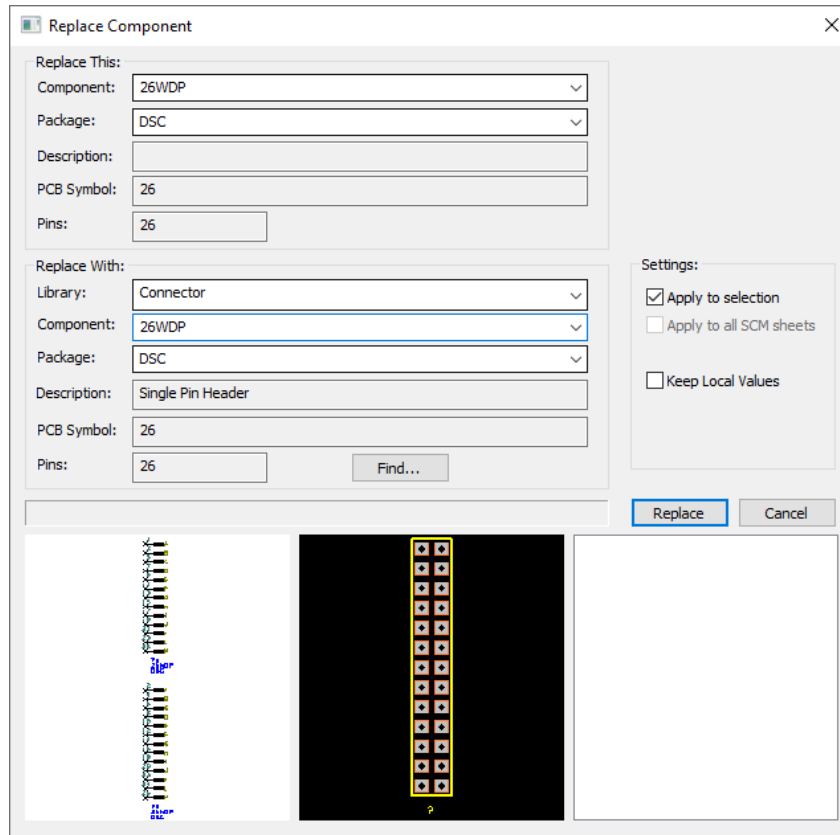
Replacing Components

The **Replace Components** option on the **Tools** menu enables you to perform 'global' find and replace on components in a design or project. It allows you to choose a replacement component to apply across the design or project.

Use the **Replace This:** selection of **Component** and/or **Package** to select the item to replace. The **Replace With:** option allows you to choose the **Library** and the **Component/Package** as the replacement. Once selected, the **PCB Symbol** name and number of **Pins** will be displayed as confirmation.

The **Preview** check box and windows allow you see previews of the new replacement **Schematic** and **PCB** symbols before replacing them in the design.

The **Find** button will display the standard **Find** dialog from which to make an advanced search and selection.



Settings switches are available to **Apply to selection** (rather than the whole design) and **Apply to all sheets** in a Schematic design.

Placing the Design Interactively

Once the components have been added to the design (by hand or during the translation from the Schematic) they will be either in the component bin or in the design unplaced. From here you have two options, to place them **interactively** by hand, or to autoplace them using the **Autoplace** option. For now, this section will concentrate on manual placement.

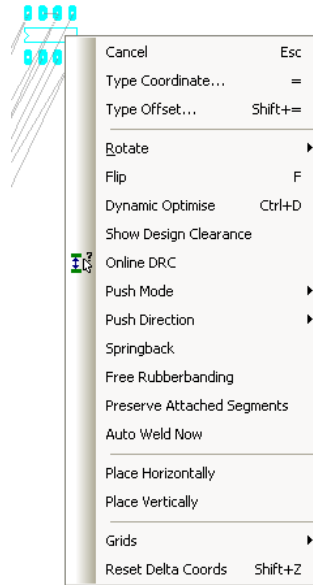
This product follows Microsoft standards, the movement of components is straightforward, you simply 'pick' the component and 'drag' it to the required location. As well as individual components you may also use the Frame Select mode to pick a group of components for moving (Frame Select is activated by dragging a box around the items required).

You may also put the component into **Place** mode, this means that the component is 'attached' to your cursor until released. This mode is available from the context menu of a selected component. It is sometimes more convenient to move the component like this without having to concentrate on keeping the mouse button pressed while dragging. This method is slightly slower as it requires more mouse button presses and is not dynamic.

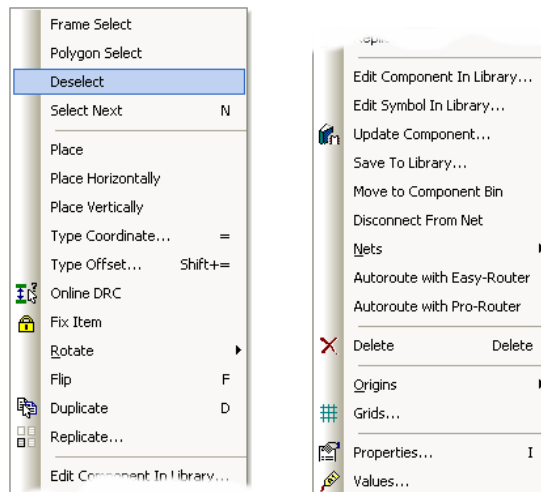
Using the Place mode, you can also use the arrow keys on the keyboard to make precise positional changes, it is more difficult to do this with the mouse alone.

Component placement options

During component move you have a number of features available.

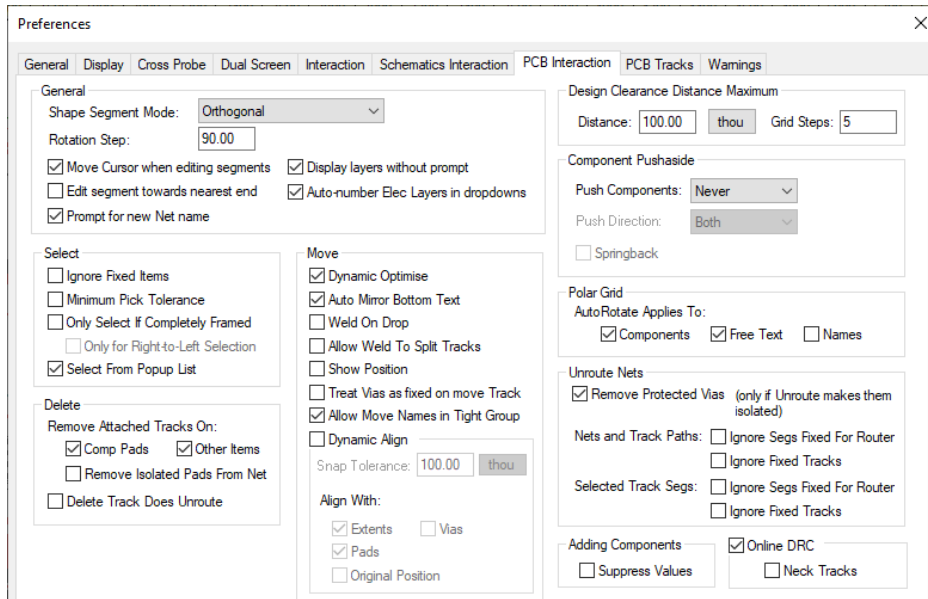


As part of the component interaction, you also have many features available to you that can be accessed on **component selection** from the context menu (in addition to those found on the **Place** menu).



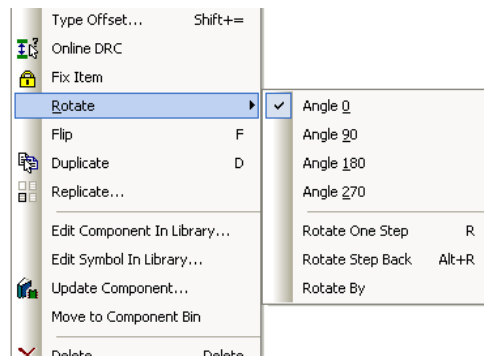
Setting up placement options

Most of the options available during Place can be pre-defined ready for use. These are located on the **Preferences** dialog (on the **Settings** menu), on the **PCB Interaction** tab.



Rotate

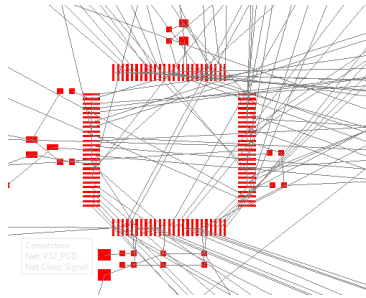
During move, press the <R> key and the component will be rotated by the default rotation step, this will be set to 90 degrees by default but can be changed in the **Preferences** dialog under **PCB Interaction**. This is also available on the context menu on **Rotate**>. On this sub-menu, you are also able to **Rotate By** and specify the rotation value.



Dynamic connections during move

When you place components, you'll notice how the connections move with the components; this is called 'rubber-banding'. Move of components is also equipped with a de-clutter mechanism for ensuring that the function is as smooth and accurate as possible. This means that during place (and is especially noticeable when using larger PCB components such as BGAs) that the entities within the component are only drawn if the system performance allows it. If the mouse is rested momentarily then the component will be fully redrawn.

On movement of a large pin device, you would only see the pads and connections drawn once the movement has stopped or paused momentarily.



Placing surface mounted components

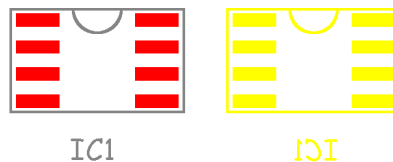
The placement of surface mounted components is no different to that of through-hole components except that they can exist on both sides of the design (Top and Bottom sides). When placing you may need to make consideration of any down-stream processes such as automatic assembly machines and their physical restraints. For example, the jaws of a placement machine may need to be accounted for when placing SMD components next to each other.

The only main difference is in the way they behave when the component is flipped (mirrored) but this is purely only for SMD components.

Flipping (Mirroring) Components

For SMD components in a PCB CAD system you need to be able to indicate that the component has been physically moved from one side of the PCB to the other. This entails the view of the component being mirrored from the Top layer to the Bottom layer, including all pads and text as well as moving drawing items such as Silk Screenidents. This is an easy process using the **Flip** option.

If a component is selected and the <F> key on the keyboard pressed, the component will flip to the other side of the board. This means that when its pads and silk screen detail (outline and component reference name) were on the **Top** side, they will now only exist on the opposite (**Bottom**) layer.

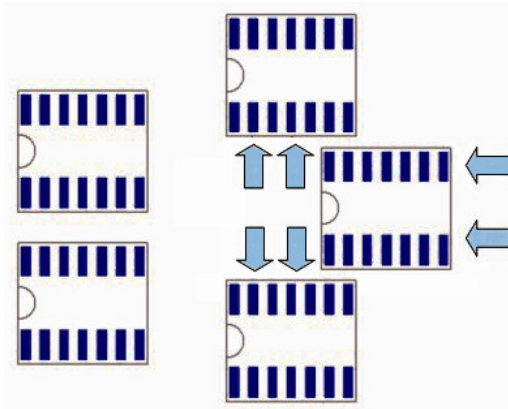


The picture above shows the effect of mirroring this device using the **Flip** command. Pin 1 is now shown on the top right hand side of the device on the right, whereas it is on the top left on the non-flipped device. You'll also notice the silk screen name, IC1, has also been mirrored.

Pushing Components

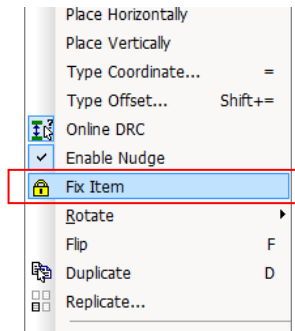
Components can be 'pushed' while interactively moving them. This feature is enabled through the **Preferences** dialog and **PCB Interaction**, or while actually moving the component using options on the context menu. Options are available for pushing to select the push mode as **Never**, **Continuous** and **On-drop**. The push direction can also be set to **Both**, **Vertical** and **Horizontal**. You can also set whether components use **Springback** (with Springback set the component being pushed will continually attempt to return to its original position before the

pushing started. This is particularly useful on a tightly packed board where space is at a premium).

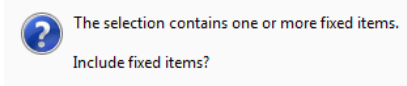


Fixing components

Components can be fixed in position. The **Fix Item** option is located on the context menu for a selected component.

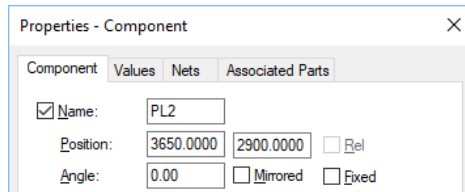


If they are selected and dragged for placement, a warning dialog is displayed. You can confirm that you want to move the device by clicking **Yes**, if you want to leave it fixed and not move it, click **No**. If **Yes** has been selected, the component can be moved. Once the move is completed, the fixed status will be reapplied.



Using the Properties option to place components

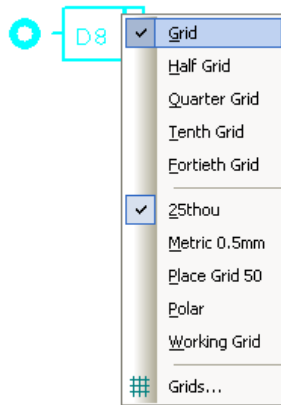
When a component is selected, the Properties option is available from the context menu. This displays information about the component including its **X** and **Y** position, **Mirror** status, **Rotation** and **Fixed** status.



From this dialog you can interactively change fields that are not greyed out.

Grid settings

During placement you can snap the component on a grid. By pressing the shortcut key <H> the grid setting menu will be displayed. You can then select the grid required for placement.



The grids here are taken from the **Grids** dialog on the **Settings** menu. From this dialog the **Grid Snap** mode can be set to the setting required. This will ensure that the placement is 'on-grid' and that the footprints are aligned.

From within the **Grids** dialog you can also set a grid to be the component grid by selecting it and pressing the **Make Comp** button. *This has been discussed previously under Grids.*

Adding Connections

Connections are used to indicate the point-to-point connectivity of the netlist. When designing PCBs on the fly directly into the PCB editor, you may wish to add connections to create the design. You may also add tracks directly to the design without adding connections; the track path still makes the connectivity but is not so easy to work with at the start of a design.

Where a design has been translated from the Schematic, the netlist will already be defined with the associated components and connections. In this situation all you need to do is to place the components and route the track paths in.

Connections or nets carry a number of properties:

- The net always has a net name, whether it is user-defined (like GND, CLK, RESET etc.) or a system-allocated net name (like N0001, N0011).
- A net class – this name provides an inherent set of physical characteristics to the net, such as track thickness and via sizes to be used when routed.

Each of these properties has already been discussed earlier in this chapter under the *Technology File* section.

Connections may be connected to any electrical items, such as copper, pads, vias and free pads.

Note: Guard Spacings can also be defined for connections, these are optional – an additional spacing value can be applied to a net where it requires more spacing over the design spacings or net class spacings. Usually not used now as Net Class to Class spacing rules are available.

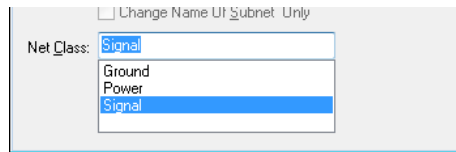
Note: there are also Wire items, these are not connections, they are used for adding jumpers and suchlike to the design.

Default Connection Settings

When adding a connection to the design, the net class of the net will be taken from the **Settings** menu, **Defaults** and **Nets**. The net class contains properties for setting the track and via styles when the connection is converted to a track.

If the design has been translated from a Schematic, the net class would have already been defined. The net class properties will be taken from the PCB technology file.

The **Net Class** can be changed using **Properties** of the net or by using **Change Net** for the selected **Net**.



Using Highlight Net with Connections

Connections can be highlighted at any time, during adding or during editing, you can use the **Highlight Net** option from the context menu. This will display any pads and connections on that net using the **highlight** colour defined in **Display** under the **Settings and Highlights** tab.

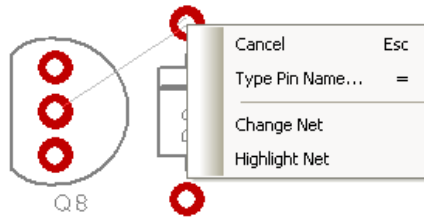
► To add a connection

1. This option can be used if starting your design from within the PCB editor. If starting with a design translated from the Schematics, then the netlist will already be defined for you.
2. From the **Add** menu, click **Connection**.
This option is also available from the PCB Design toolbar icon.
3. Click on the first pad to add the connection, click on the second 'target' pad. A connection will be made between the two pads.
4. The Add Connection option will work on 'pairs' of items. If you want to 'daisy chain' the pads together then switch on the **Stitch Connections/Tracks** option in the **Preferences** dialog, **Interaction** page.
5. Click <Esc> to exit the Add Connection mode.

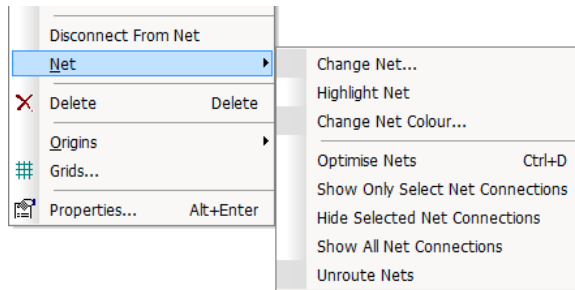
During Add Connection you can also use the context menu to change or highlight the net.

Change Net will enable you to add or edit the net name and add or edit the net class.

Highlight Net is used to highlight the whole net to make it stand out from the other connections in the design.



Once the connection has been added, if you select it, there are additional items on the context menu.



On the **Net**> sub-menu, there is **Optimise Nets**. This is used to optimise the selected net only (Optimise is the process of shortening the connection to the nearest connection point on the same net).

PCB Net Names

By default, all PCB nets (nets added to the PCB design, not ones originating in the Schematic editor) have a net name of Nxxxx, for example N001, N010 etc. This name can be changed during or after the Add Connection process using the **Change Net** option from the context menu.

Net Optimisation

Before starting to route the tracks, you can make use of the **Net Optimiser**. This is used to make the shortest possible connection lengths within the unrouted design (it only works with unrouted designs). When creating the logical net list in the Schematic design, it won't necessarily be the shortest connection path when passed into the PCB design editor for layout. The Net Optimiser enables you to get the most optimised connection length as you design.

The **Net Optimiser** is invoked from the **Tools** menu, **Optimise Nets**> option. As soon as the option is selected, the layout will be redrawn with all the nets optimised. You have three options for performing an **Optimise** on **All Nets**, **Browsed Nets** and **Selected Nets**.

You can perform a dynamic optimise during component move. This is enabled from the **Settings** menu by selecting the **Dynamic Optimise** switch on the **Preferences** option and **Interaction** tab.

As the **Net Optimiser** is a commonly used option, we suggest you set it on a shortcut key using the **Customise** dialog and **Keyboard** tab.

Adding Tracks and Routing the Design

Following on from component placement (and the addition of netlist/connections if you started in the PCB design itself), you will now need to convert the connections into copper tracks. Routing can be done both manually using the interactive tools provided, or automatically if the autorouter is available to you.

This section will guide you through the interactive manual routing tools. The automatic routing tool can be found in a later chapter *Automatic PCB Features*.

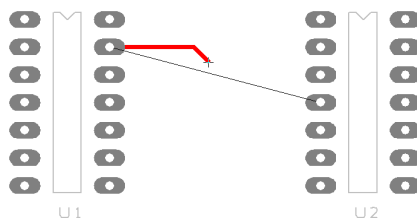
The design may also be created directly into the PCB. You can add connections to components in the design or you can add tracks directly into the PCB. By doing this, the underlying connectivity will be constructed from the track node to node connection (if you were to remove the tracks using Unroute, the connection base would still remain).

► To edit track manually

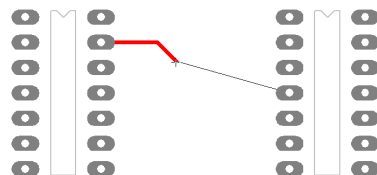
- To start editing a track (and subsequently convert them to tracks) you can:
 - Select **Add Track** from the **Add** menu and select the pad to start from, or:
 - **Double-click** on a connection.
- The **Add Track** mode can be started from both the **Add** menu and from the **PCB Design** toolbar.

When a track is referred to as being 'edited' this means double-clicking on the connection when it is then dynamically converted to a track

The picture below shows the **Add Track** method of adding tracks:



The picture below shows the alternative method of directly editing the connection showing the 'flying' connection from the end of the track to the target pad. This is the normal recommended method of manually editing connections to tracks.

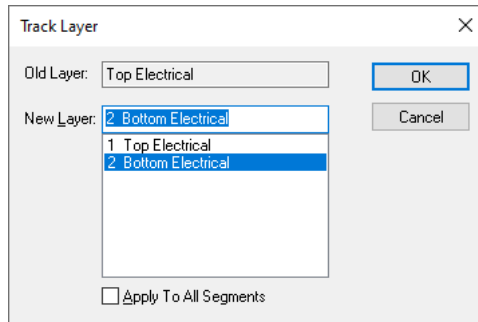


Overview of the basic modes of routing

The basic commands for editing tracks are:

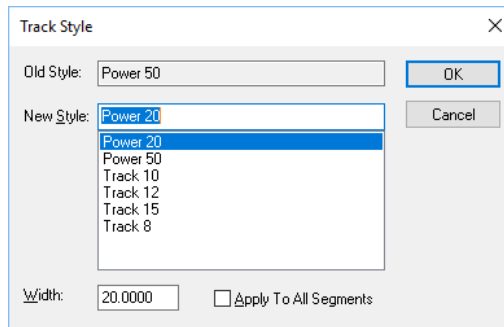
Single-Click, once editing a single-click will add a corner and enable you to change direction. In the appropriate mode, a change of direction will also add a mitre to the corner. When using **Add Track**, a single click over the picked item will end the track on that item.

<L> (Change Layer) followed by **<Enter>** will enable a layer change of the track to be made. This will automatically insert a via at the point of the layer change.



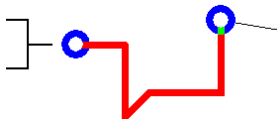
Layer changes automatically inserts a via and changes layer. The current track segment will now be on the new layer and will stay there until another layer change is made. If a new layer change is made another via is inserted. This rule is only applied if starting on a through-hole pad and finishing on a through-hole pad. If the track starts and stops on a surface mount pad a via will automatically be inserted regardless of any layer changes, obviously, this is required anyway to change the track to the correct layer for completion.

<S> (Change Style) followed by **<Enter>** will enable the style of the track segments from that point onwards to be changed. Choose from the styles presented in the dialog.

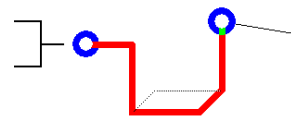


Double-click, during edit this will finish the track in mid-air with a connection back to the target pad.

<F> (Flip) enables the current segment and the previous one to be mirrored.

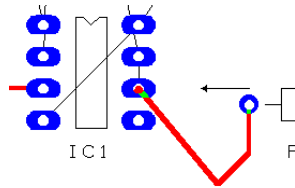
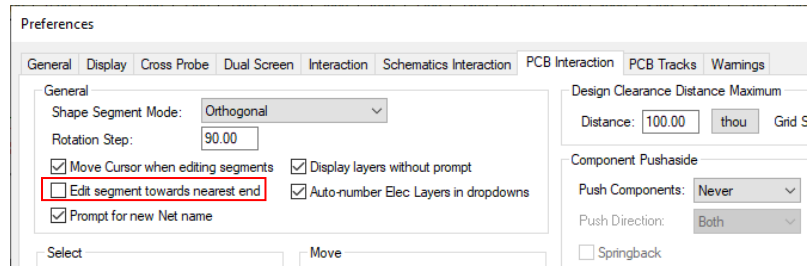


This shows the original track path.

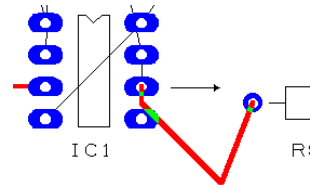


This is how it looks when the Flip command is used. The fine dotted line shows the original track path.

<I> (Insert shape), using this mode will enable you to determine which end of the net the track will start from. You can also fix this by setting the **Edit Segment Towards Nearest End** switch on the **Settings** menu, **Preferences** dialog and **PCB Interaction** tab.



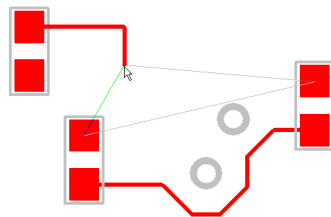
Editing from the resistor adds the vertical track and 45-degree segment with the flying track back to the IC.



Flipping the direction then starts the vertical track and 45-degree angle from the IC with the flying track back to the resistor.

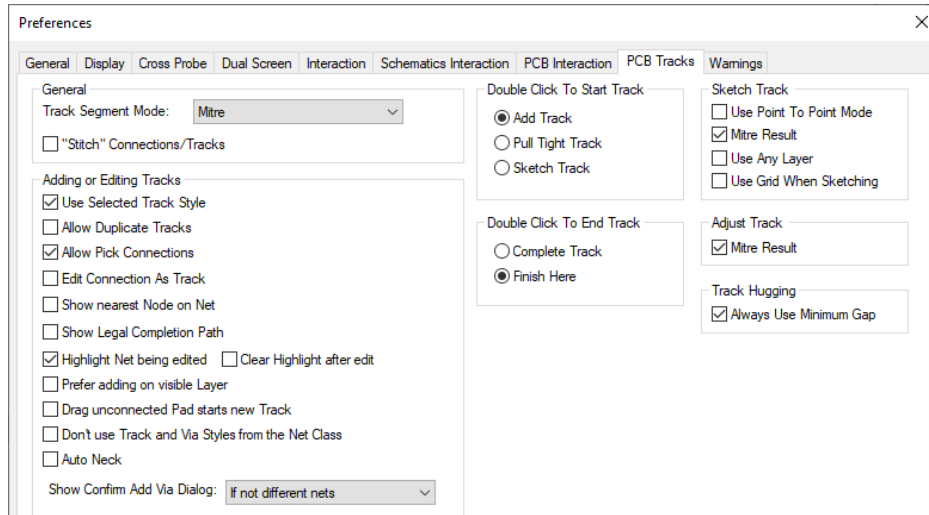
Show Nearest Node on Net

There is an option on the context menu when adding a track or sketching, called **Show Nearest Node on Net**. Use this option to show a line to the nearest item on the net to the end of the track being added, a dynamic optimise mode. The option can also be set on the **PCB Tracks** tab in the **Preferences** dialog.



Setting up preferences

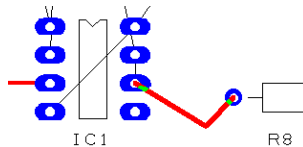
As well as the **General** preferences, there are also preferences specific to the PCB design editor on the **PCB Tracks** tab:



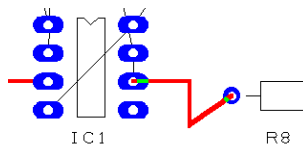
PCB Track Segment Mode – there are four modes available on the drop-down list, **Fillet**, **Free**, **Mitre**, **Orthogonal** and **Right Angle**.

For all of these modes, the ‘trailing’ segment back to the target pad is always left as a free-angle track. In the examples below, this is shown as the right most track segment back to the resistor leg of R8.

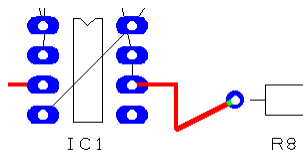
Free – the track will exit the pad or corner at any angle following the cursor until the next mouse click.



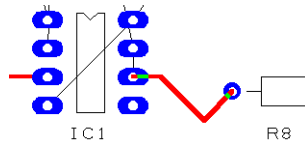
Orthogonal – the track will remain in 90 degree mode at all times during the addition of corners. Alternate track segments are horizontal and vertical with the previous segment being dynamic.



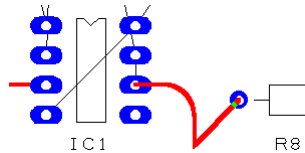
Right Angle – the track will remain in 90 degree mode at all times during the addition of corners. Alternate track segments are horizontal and vertical.



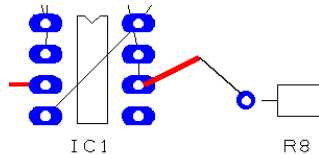
Mitre – following the orthogonal track, the corner added will be a 45 degree mitre. The size of the mitre will depend on the length of the movement away from the corner.



Fillet – following the orthogonal track, the corner added will be a filleted track with a radius. The size of the fillet will depend on the length of the movement away from the corner.



Fixed Angle – this segment mode can be used for adding single segments at a supplied angle. The default will be to add single segments at 45 degrees.



Selecting this mode again from the menu returns it to the previous segment mode you were using. Use **Enter Angle** from the context menu to change to the required angle, you will then be able to add segments at this angle and at 45 degree rotation increments from that angle.

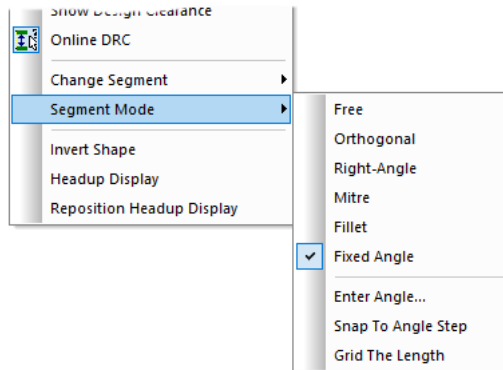


Select **Snap to Angle Step** from the context menu to snap to steps equal to the angle originally supplied. For example if the angle supplied was 15 degrees then you will be able to add segments at multiples of 15 degrees.

Select **Grid the Length** from the context menu to grid the length of the angled segments being added using the current grid value. For example, if your working grid is set to 100, then the length of the segment in this mode will be 100. If this feature is not selected the line ends will snap orthogonally to the nearest grid point.

Changing Segment Modes

These modes can also be set while editing the track using the context menu. The shortcut key for this mode is <M>.

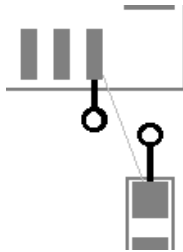


Advanced options on the context menu

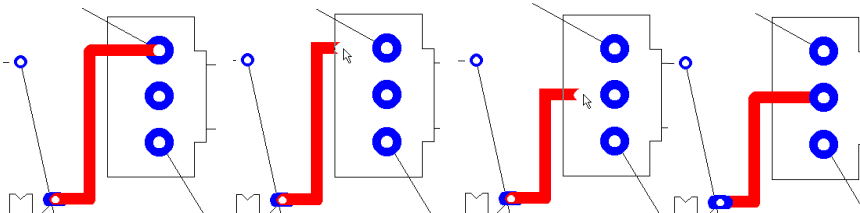
Some useful features to note on the context menu (note, not all are listed here):

Edit Segment – this option is available on the context menu when an existing track segment is selected. It has the same effect as if you double-clicked on the track segment directly. It allows you to edit the track path of the selected segment, you may also change the style and layer once in the edit mode.

End On Via – this option is used to end the track being edited on a via, effectively creating a ‘stub’ route. This is discussed further on in this section.



Disconnect Track End – this option is used for disconnecting tracks from pads when an end segment of a routed track needs to be connected to a different end location. This option is then available on the context menu. You can also use the shortcut key <E> when in edit mode on the track. This will leave the track end segment on the end of your cursor from which you can then select a new pad or land site for the segment.



The sequence above shows how this might be applied, moving from left to right.

If working with a master Schematic, you would edit the schematic and then run the Forward Design Changes option to apply the net changes to the PCB design.

Autoroute – for a selected connection or for a partially routed track, you can select Autoroute to finish the track. This is only available if you have the Easy-Router option installed.

Auto Adjust Track – this option will attempt to correct track errors but adjusting the track with minimum deviation from its path.

Smooth Track – for a selected track, the smoother will try and reduce the track length, number of vias and number of staircases. This option is described in more detail later on in this section.

Auto Mitre Track – for a selected track, a 45-degree mitred corner will be applied to it using the values set in the **Auto Mitre** dialog on the **Tools** menu.

Arc – when selecting an existing track segment, the **Arc** command converts a segment to curve. As well as the pre-defined arc angles of **45**, **90** and **180 degrees**, there is a **flip** command to reverse the sense of the arc. An **Angle Free Mode** option will enable an arc of any angle to be created using the cursor position to define the required size. This gives quite fine control, providing that the rotation step size has been set sufficiently small (0.1 degree is possible). There is one illegal value, 360 degrees. If an attempt is made to create effectively an infinite circle, the segment defaults back to a line. The **Arc To Line** mode provides you with an option to snap the arc back to a straight line segment.

Editing Existing Tracks

Where tracks have already been routed but the path needs to be adjusted, the manual editing feature is very easy and powerful to use. Some basic, (and standard Microsoft), techniques have been used to make the edits a familiar format.

Moving a Track

Moving a track segment is a simple process of selecting the track segment and **dragging** it to the required position. During this pick and drag you may also (from the context menu) change the tracks' **Style** and **Layer**, toggle the **Online DRC** on and off, and allow the tracks to have **free rubberbanding** (rather than be fixed orthogonally).

Editing and Selecting a Track

To edit an existing track, double-click the cursor while over the track, this then makes the segment 'flexible' so the path can be changed using add corners etc. To finish the edit, double-click again.

A **<Single-Click>** selects the tracks and then waits for something to be done with it, e.g. right-hand mouse key and change layer etc.

A **<Single-Click>** and drag will move the track segment, but will move it either orthogonally if the track is at 90 degrees, or if the track is on a free angle or 45 degrees, will move it parallel to its existing position.

Editing a Corner

The same principles will be applied to a corner track. One point to make is that the selection or editing of a corner track has to be reasonably accurate otherwise the track segment could be selected by mistake.

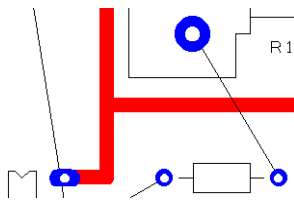
Double-Click on a corner will edit the corner, the corner must be orthogonal though. This means that the corner will automatically be edited to be a 45-degree mitre. Once editing, you move the mouse and the mitre will snap to the Working Grid.

A **<Single-Click>** and drag on a corner will move the corner retaining its orthogonality but will allow you to reposition the corner.

Other editing options and techniques

'T' Routes

When using the **Add Track** mode it is possible to connect a track into an existing tracks, effectively 'copper sharing'. This is particularly useful when a 'T' junction is required. A hidden connection will be automatically created to the nearest node on the net to retain its connectivity (this would only be visible if the track was unrouted).



Finishing Tracks

To finish the track on the target pad you simply Double-click the mouse over the pad. If you wish to finish the track before reaching the pad and then come back to it later and edit the remaining track path, click to add a corner, then Double-click to finish. An angled track will be drawn to the target pad. This track can be edited later on. **Finish** will always work from the current corner not the current cursor position (they could be the same though).

You may also use the context menu options to **Finish** - finish at the current corner and add orthogonal segments back to the target, **Autoroute** - allows the autorouter to complete the track, this will not be available if you don't have the Easy-Router option.

Adding Corners

During edit corners can be inserted in a track path by a single mouse click at any point. After the mouse click, move the mouse and continue with the track.

The shape of the corner will be affected by the **Segment Mode** that you have selected. When editing the track, select an option from the **Segment Mode>** option on the context menu.

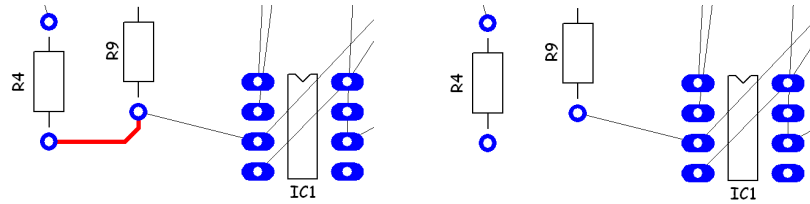
Exiting the edit option

If you get lost with the routing or make a mistake you can use the **<Esc>** key to cancel the edit. You can also use **Finish** from the context menu which will complete the track, then use the **Undo** option **<Ctrl-Z>** to remove it.

Using Delete to Unroute Tracks

If you wish to unroute a track back to its connection, use the **Unroute Track** option (see below). If you use the **** key on the keyboard there are two situations that you should be aware of:

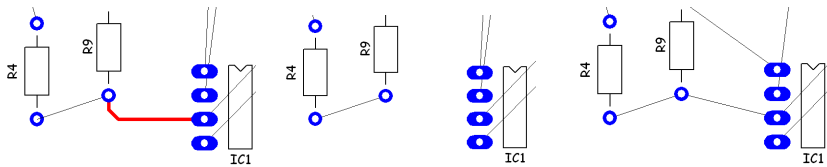
- Delete on a track that is in the 'middle' of a net will remove the track and the connection. However, when you run an **Optimise** on the design or that net, it will reinstate the connection path.



In the above example, with the end connection between R4 and R9 routed, this track is selected and the Del key pressed.

Even when **Optimise** is run, the connection is not reinstated.

- Delete on a track that is at the end of net will remove it completely from the design even if **Optimise** is run afterwards.



With the middle connection routed, this track is selected and the **Delete** key pressed.

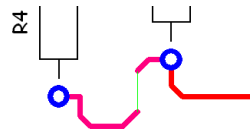
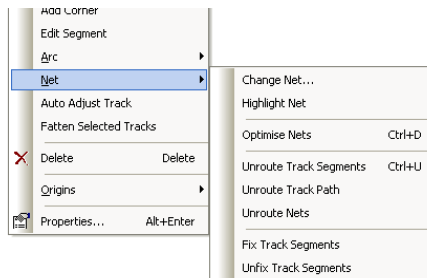
The track is deleted. The nets on each side of the deleted connection retain their original Net Name.

If **Optimise** is then run, the connection reappears

Unrouting Tracks

Tracks within a design can be unrouted back to their 'connection'. To do this, use the **Unroute Nets>** option from **Tools** menu and choose either **All Nets**, **Browse** or **Selected Nets** depending on the refinement of unroute that you require.

The context menu also contains unroute options under the **Nets>** option. From this menu you can choose **Unroute Track Segments**, **Unroute Track Path**, and **Unroute Nets**. This gives you control over unrouting the individual selected track segment(s), the point-to-point track path and the whole net.

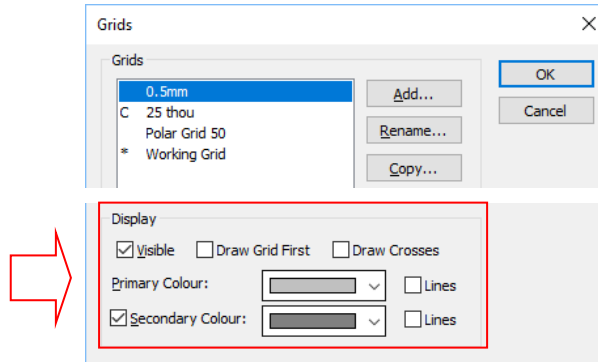


After selecting the track segment(s), choose **Nets>** and **Unroute Track Segment** from the menu.

The selected track segment only is unrouted.

Viewing the Grids

A visible grid is helpful when positioning tracks. If the grid dots are not displayed, press the <G> key on the keyboard. You may also switch on the grid dots from the **Settings** menu and **Grids**. Each named grid has its own visibility switch.



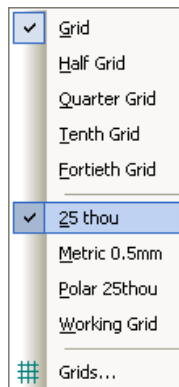
You can also make the grid dots displayed as **lines** on the design. Use the **Lines** check box next to the **Primary** or **Secondary Colours**.

By selecting the check box, **Draw Grid First**, you can control whether the grids are draw in front or behind design items. Selecting this box will draw them behind design items. This switch really depends on whether

The **Grid Snap** settings will use the grids defined in the Grids dialog. To access this menu, while designing, press the <H> key, see below for more information. Experienced users usually work with the grid displayed.

Snapping Tracks to Grid

The **Grid Snap** menu is accessed at any time during the design process using the shortcut key <H>. It can be changed for the placement of components and routing of tracks for example as well as all other interactive options.



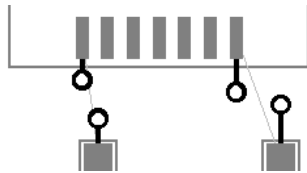
It also allows you to set fractions of the currently selected named grid using the shortcut key <H> while editing the design. By pressing this key, you will be shown a context menu that contains alternative grid modes. These grid snap modes are useful when routing fine tracks between the pins of ICs and where more than one track between is required.

For the neatest layouts, always try to work to the largest snap grid possible, use the snap modes where required and only move off-grid using freehand when absolutely necessary.

Using named grids allows you to swap grids between imperial and metric if you require mixed-unit designs. By defining a grid of 0.5mm for example, this will be exactly defined once Metric units are invoked (using the **Units** option and **Metric**). When using Imperial units, you can ignore this grid until it is needed.

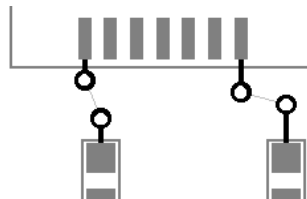
Adding Stub Routes

Stub routes are short segments of track, normally with a via at the end of them. These might connect a surface mount pad to an inner layer power plane. Because surface mount pads have no way of connecting to a power plane (unlike through hole pads which have plated drill holes), a track and via are required to make the connection as the picture below illustrates.



The nets act as an indicator of the connectivity and can be used later on to highlight gaps in the 'copper' path when the **Connectivity Report** is run (from the **Output** menu). This picture also shows the nets before **Net Optimisation** has been run, showing the actual connection path and not the nearest path.

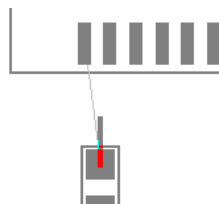
Once run, the nets now take the shortest connection path between the vias.



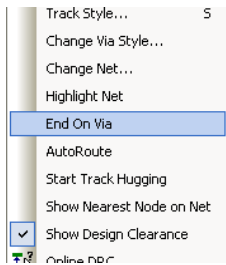
► To add a stub route

To add a stub route, you must be in **Add Track** mode and not be editing the connection directly.

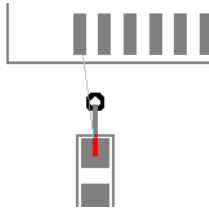
1. From the **Add** menu, click **Track**, you can also use the **Add Track** option from the PCB Design toolbar. As explained before, the connections and nets are only there for clarity of connectivity of the nodes in the design.
2. Select the pad to have the stub route, and click on the pad centre.
3. Drag the track to the length required as indicated below on the left hand surface mounted capacitor:



4. From the context menu select **End On Via**.



5. This will give you a dynamic via on the end of the track ready for placement. The via can be moved around to place it on a grid point or to the location required.



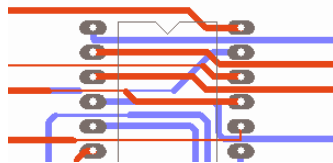
6. Double-click the mouse button to release the via in its current position. If the position of the via is critical and you are working on a very fine grid you can also use the keyboard keys to add corners and release the via. The up and down arrow keys can also be used to move the via. The spacebar will add a corner and the <Enter> key release the via.

Protected Vias

On the **Properties** dialog for selected vias, you have the ability use the **Protected** switch. This is used to lock a via against being automatically removed when a track attached to it has its layer changed so that the via is no longer required. Protected vias are never automatically deleted, and are usually used for stub routes from a surface mounted pad to reach an inner plane or the other side of the board.

Track Fattening

Tracks can be ‘fattened’ up allowing more track copper to be laid down on the PCB.



The **Net Class** track settings on the **Design Technology** dialog are used to define the **Nominal** and **Minimum** widths.

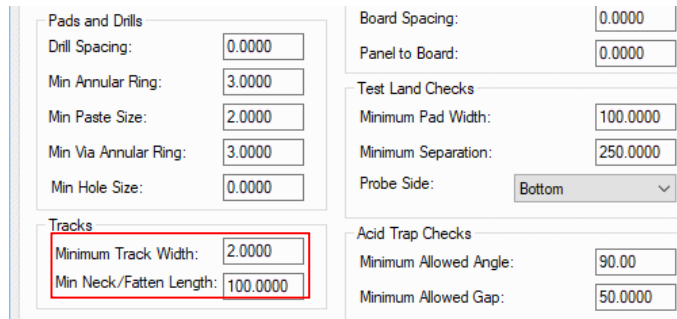
Where the Nominal width used causes a DRC violation and with the Neck Track feature enabled, the option will ‘neck’ down to the **Minimum Track Width** defined.

	Name	Type	Min. Track
X	Signal	Signal	Signal 10
X	Power	Power	Power 25
X	Ground	Power	Power 25

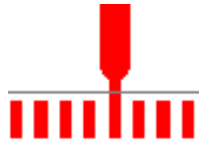
The option to **Fatten Tracks** is available on the context menu while editing tracks.



The minimum length setting can be defined in the **Design Technology** dialog under **Rules** and the **Min Neck/Fatten Length** rule.

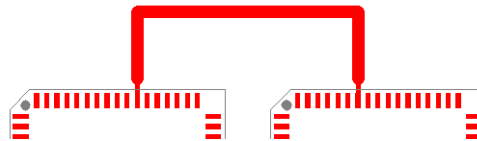


This rule is also used to define the 'stub' distance when routing and fattening. This is the distance between the pad and the start of the 'fat' track segment.



Automatic Track Necking into SMD Pads

While editing a track, it is possible to enable automatic track necking if the 'fat' track is too wide to enter a pad without causing a design rule violation.



Like the Fatten Track option, the **Net Class** settings are used to define the **Nominal** and **Minimum** widths. Where the Nominal width used causes a DRC violation and with the Neck Track feature enabled, the option will 'neck' down to the **Minimum Track Width** defined.

	Name	Type	Min. Track
X	Signal	Signal	Signal 10
X	Power	Power	Power 25
X	Ground	Power	Power 25

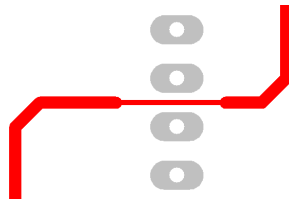
To operate in this mode, you need two features enabled. The **Online DRC** and **Neck Tracks** options should be enabled. Both of these features are enabled in the **Preferences** dialog under **PCB Interaction**.



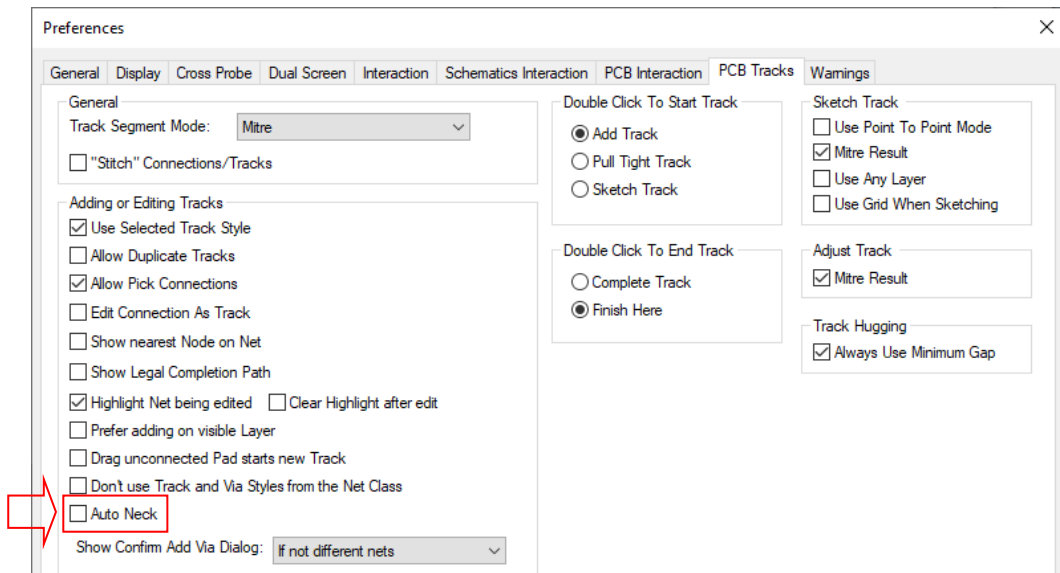
Auto Neck Track

The **Auto Neck Track** option allows you to dynamically track segments to automatically neck down to use a smaller track width to pass by obstacles when the normal width would produce a design rule error. When the obstacle has been cleared, adding a corner to the track will automatically swap back to the fatter normal width.

The necked track width will be taken from the minimum track style associated with the **Net Class** of the track being edited. Necking will not be performed if there is no minimum track style, or the current track width is not greater than the minimum from the net class.



Use the **Auto Neck** option on the context menu when adding or editing a track, this can also be set in the **Preference** dialog under **PCB Tracks**.



Swap Layers Mode

You may have a requirement for swapping track layers (this also works for shapes) from one side of the board to the other, the **Swap Layers** option allows you to interactively do this for selected items. This is generally useful for a post-routing clean up, you work your way around the design swapping items to the other layer to reduce the via count.

The **Swap Layers** option is available from the **Edit** menu in a PCB design.

As a quicker mode of swapping layers, it is easier to select the track and use <L> (**Change Layer**). It will do exactly the same as the Swap Layers mode including adding or minimising vias.

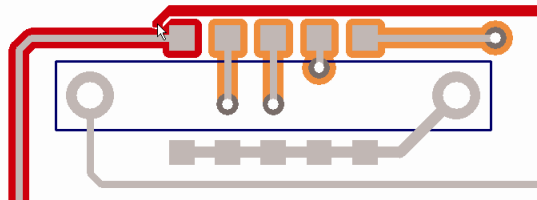
*Note: A certain number of vias can usually be removed automatically using the **Smooth Track** option (see the **Tools** chapter below).*

Swap Layers works much like the normal **Select** mode, including the ability to select single or multiple items, but the selection is restricted to the type of items that can be swapped. Once selected, the items are swapped to the opposite side of the board. When swapping tracks to the other layer, vias are automatically inserted into the ends of the tracks where they swap layers.

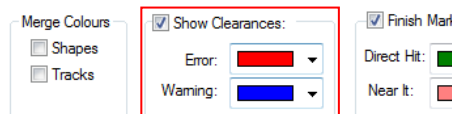
*Note: **Online DRC** is not carried out within this option, as it would prevent much of its usefulness in 'untangling' track patterns.*

Display Clearances

The **Display Clearance** option is used to show the **Spacing** clearances around items within the design.

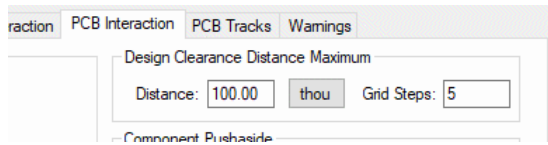


This is enabled through the **Display** option, **Settings and Highlights** page using the **Show Clearances** switch and the colour settings for **Error:** and **Warning:**

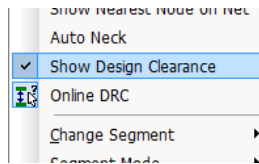


This will highlight potential obstacles when performing various editing options (such as track editing or moving a single item). You can specify how far away potential obstacles should be before they are highlighted, this can be specified as an absolute distance, or as a number of grid steps (the larger of the two is always used). If both values are 0, then obstacles are only highlighted if actually in error.

The maximum distance to a highlighted item is set in the **Preferences** dialog on the **PCB Interaction** tab using **Design Clearance Distance Maximum**.



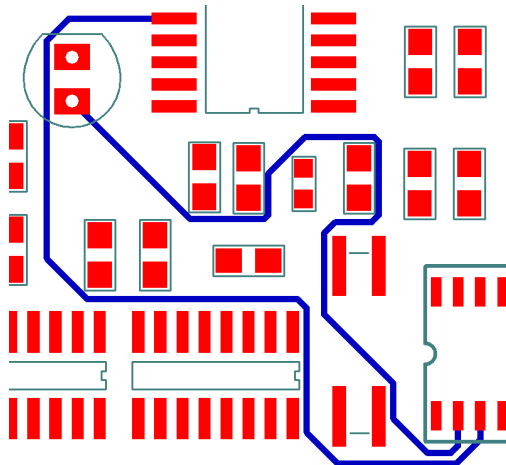
When adding a track for example, you are able to switch on the display clearances using the **Show Design Clearance** option from the context menu when you right click.



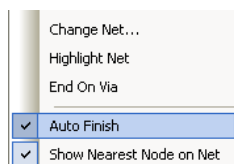
Pull-Tight Mode

The **Pull Tight Track** mode is available complement the **Trace-Router** autorouter and is available on the **Add** menu. Note, without Trace-Router enabled, this mode is not available. This works slightly different to the conventional manual routing and sketch track modes. Two things make it different; the track is 'pulled tight' between the source and target pads. Think of it as a piece of string if you like, being pulled tight around some obstacles.

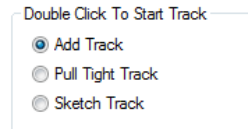
This is what the appearance of the completed track would look like, the shortest track length but pulled close to obstacles.



The second is the auto-finish option (switchable). When the track gets close enough to the target pad, it will auto-finish, ready to start editing the next connection. This mode is available on the context menu when in **Pull-Tight** mode.



Using the **Settings** menu and **Preferences, PCB Tracks** tab, you can define how double-click on a connection action will behave for **Pull Tight** mode.

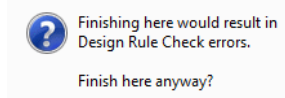


Online Design Rule Check (DRC)

Online Design Rules Check (DRC) will help you to detect design rule errors as they happen whilst you are editing your PCB design.

As each item (track, component, etc.) being added, placed or edited is released, the program checks for design rule violations against other items in the design. All affected items are checked, including for example, any tracks attached to a component you have moved.

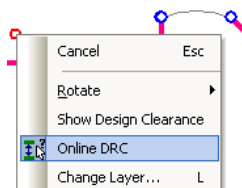
If errors are found by these checks, error markers are added to the design, and a message box is displayed prompting you to choose between finishing with errors, or resuming adding or editing the item.



To control this feature, the **Online DRC** check box is available on the **Preferences** dialog under the **PCB Interaction** tab.



A shortcut to this switch is also available on the context menu when in select or edit modes on a track and when placing items.



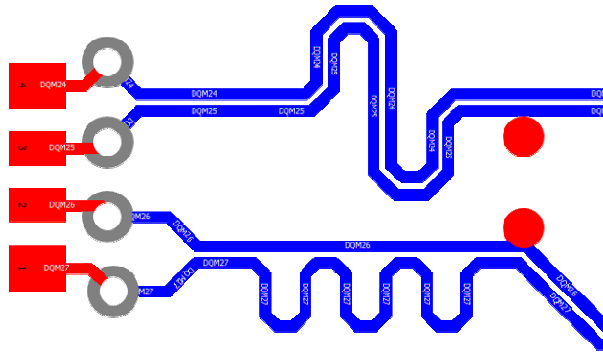
The **Online DRC** option switch is also accessible on the main **Tools** menu.

Differential Pairs

As part of your tracking or manual routing process, you can define and route in Differential Pairs.

What is a Differential Pair?

A Differential Pair is a term used in PCB design to define a pair of tracks that carry signals that are transmitted down them. These are usually tightly coupled tracks; one carries a positive signal and the other carries the returning signal, this tight coupling balances the pair.

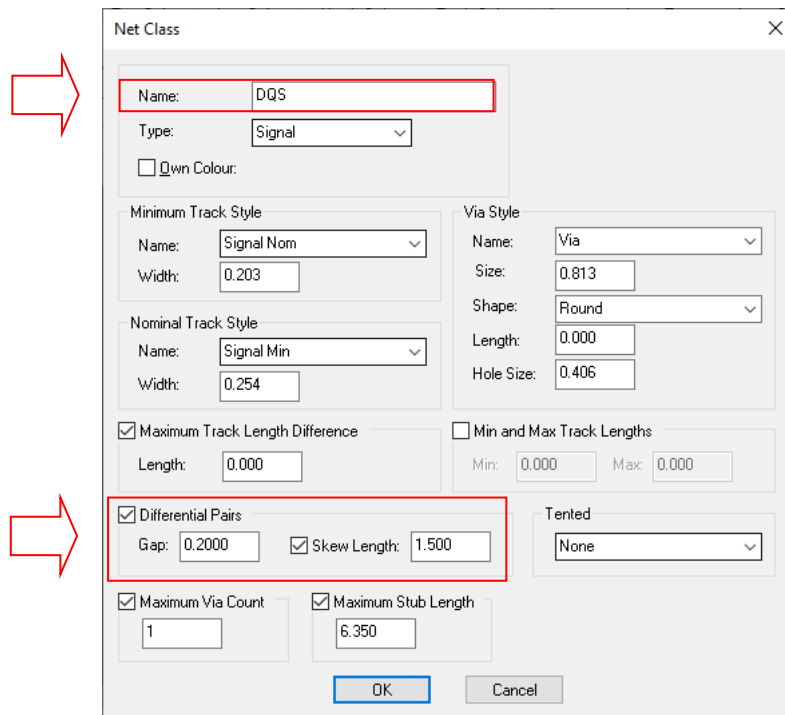


Setting up Diff Pairs

Diff Pairs are defined using **Net Class** settings within your **Design Technology** and they must be enabled for use by selecting the **Differential Pairs** check box.

If **Differential Pairs** are enabled on a **Net Class**, Easy-PC will automatically detect pairs of nets using this Net Class and then based on their **Net Names**, it searches for pairs of nets with the suffix **_P** and **_N** and automatically designates those as a pair, for example, DQS_P and DQS_N. If you had DQS1_P and DQS2_N for example, these would not match as a legal Diff Pair.

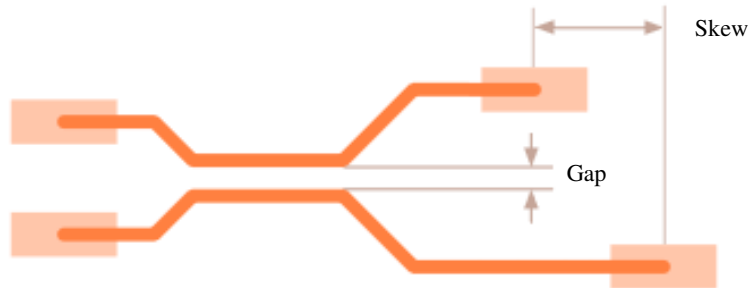
The Net Class dialog allows you to enable **Differential Pairs**:



Once enabled, controls are available for separate **Diff Pair Gap** and **Skew Length** for pairs on the Net Class.

A **gap** different to the main Track to Track Spacing rule may be required for specific Diff Pairs. This is the gap between the edges of the tracks of a pair and overrides the normal Track to Track Spacing rule.

The **Skew Length** value between each pair can be set, this effectively enables you to set a maximum delay distance. This can be as long or as short as required depending on your timing requirements.



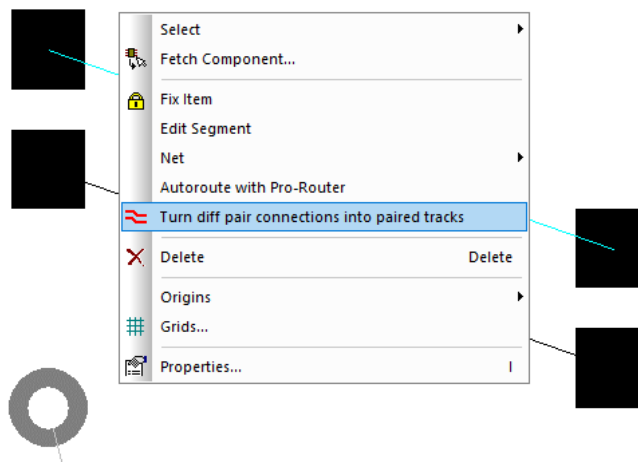
Other length rules for a Net Class can also be utilised within a Diff Pair, such as **Min/Max Track Lengths**, **Maximum Via Count** and **Maximum Stub Length**.

Creating Diff Pairs

When a Diff Pair is added to the design, it will be the actual centre of the two lines that is on grid, not the Diff Pairs themselves. This means you should try and use grid multiples when routing so that odd segments are avoided.

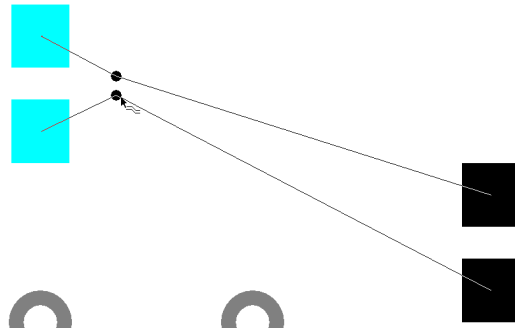
To start a Diff Pair, you can choose a number of options:

- Pick one of the Diff Pair connections and choose **Turn diff pair connections into paired tracks** from the context menu:



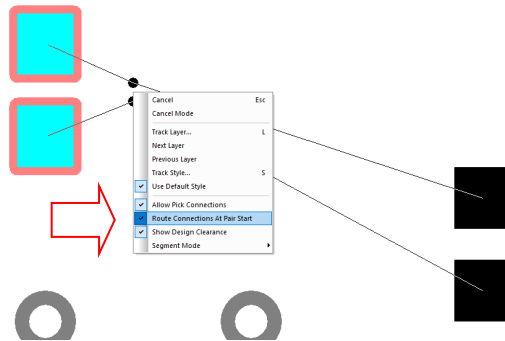
- Pick either the connection or Pad of the Diff Pair connections and select **Diff Pairs** from the **Edit** menu.

Once the **Diff Pair** option is selected you will see two ‘dots’, these present the start of the Diff Pair, a modal cursor indicated you are in ‘pairing’ mode.



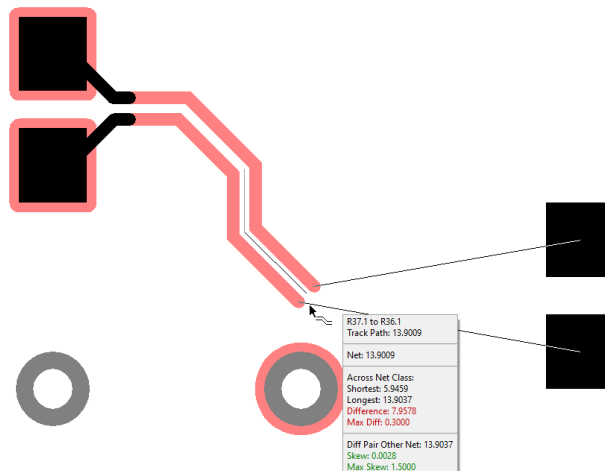
From here, you can 'draw' in your pair. This operates the same as when adding tracks with most tracking modes available; **Change Layer**, **Segment Modes**, **Display Clearances** etc. These are available on the context menu.

When you see the start points, you can also immediately right click to view the context menu. This contains the **Route Connection At Start Point** mode.



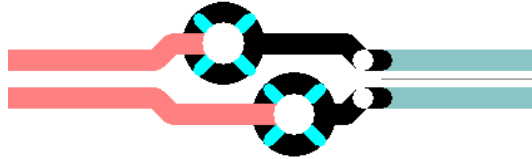
When this is selected, the start of the Diff Pair will be taken from the pads and will automatically route out (where it can) to the Diff Pair tracks.

Draw the tracking in as required:



During routing, this uses the **Diff Pair Gap** (on the Net Class, if defined) or if not defined, it uses the **Track to Track spacing** for the current Net Class and Layers (this way you can have different gaps on different layers).

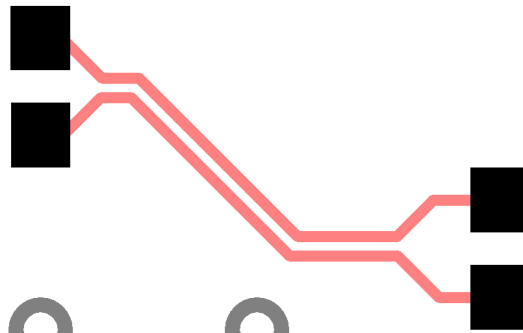
During editing, you can use **Change Layer** using the context menu option or <L> key. Two vias are inserted along with suitably spaced tracking. When using **Change layer**, if not **All Segments** then it inserts a 45-degree via pattern where vias are separated using the **Via to Via Spacing**.



Double Click to finish a Diff Pair

While editing a Diff Pair, it uses the **Edit Track** options in **Preferences** to decide what to do when **double-click** is used to finish, for example, **Finish Here** or **Auto Complete**.

Using **Auto Complete** will enable the pairing to finish on the target pads for you:



Partial routing of Diff Pairs

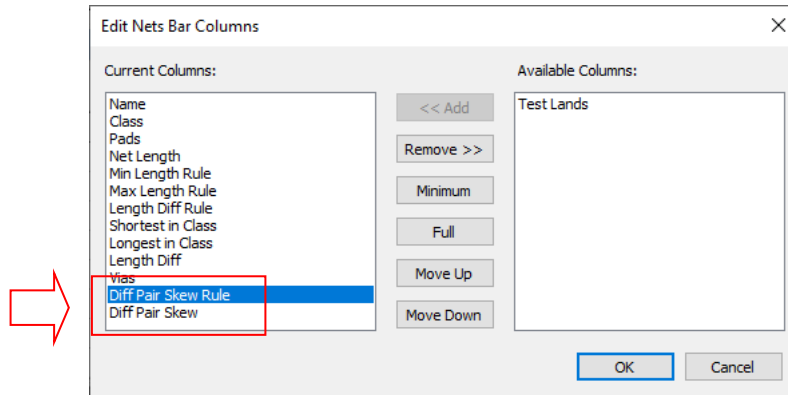
If a diff Pair has been partially completed, within **Add Diff Pair** mode, you can pick the two junctions at the end of the paired section to start from. Move the gather point back over the junctions to add paired segments to the existing ones and continue Diff Pair routing.

Diff Pairs – Nets Bar

Columns are available in the **Nets Bar** for Diff Pairs.

Name	Class	Pads	Net Length	Min Length Rule	Max Length Rule	Length Diff Rule	Shortest in Class	Longest in Class	Length Diff	Vias	Diff Pair Skew Rule	Diff Pair Skew
DQS_N	DQS	C16.1 Q2.1 R20.2 U1.6	33.0287			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DQS_P	DQS	C7.2 Q2.2 R22.2	29.1688			0.3000	3.9624	53.7648	49.8024	0	1.5000	3.8599
DRIVE	DQS	C12.2 PL2.1 Q5.3 Q6.3	30.4794			0.3000	3.9624	53.7648	49.8024	0		
EX_N	DQS	R36.2 R37.2	16.0547			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
EX_P	DQS	R36.1 R37.1	13.2607			0.3000	3.9624	53.7648	49.8024	0	1.5000	2.7940
Feedback	DQS	PL2.4 R1.1	46.8972			0.3000	3.9624	53.7648	49.8024	0		
High_speed	DQS	PL1.4 R40.1	19.6663			0.3000	3.9624	53.7648	49.8024	0		
Low_speed	DQS	PL1.3 R39.1	17.0361			0.3000	3.9624	53.7648	49.8024	0		

The **Edit Nets Bar Columns** dialog includes **Diff Pair Skew Rule** and **Diff Pair Skew**.



For a Diff Pair net, **Diff Pair Skew Rule** shows the rule on a net if its Net Class has a skew rule defined and **Diff Pair Skew** shows the actual difference between the lengths of the two nets forming the Diff Pair.

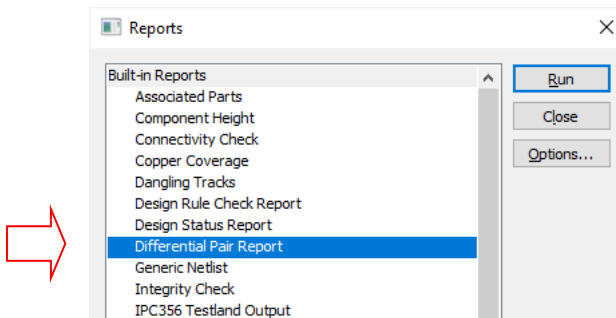
As with all Length columns in this dialog, the cells change colour to display whether the Skew is within the limits defined.

Design Rules Checking of Diff Pairs

The **DRC Track Lengths** check also checks and reports Diff Pair skew errors.

Diff Pair Report

A report is available in the **Output** menu and **Reports** called **Differential Pairs Report**.



Copper Pouring

As well as add basic copper shapes to the design, copper can also be used to produce partial copper planes or split power planes with intricate shapes and cutouts where the copper avoids obstacles such as pads and vias.

You should use the **Copper Pour Areas** and **Copper Pour** if you wish to:

- Add a copper shape with no net association (a shape with cutouts to avoid pads etc.);
- Add a copper shape with net association, for a small ground plane area for example. The net will be connected to the copper using thermal relief;
- Add a copper pour area to a layer defined as a power plane to create a split power plane (two nets associated with the same plane but not connected);

To achieve the above you must add a **Copper Pour Area** first to the design then 'pour' the copper into it.

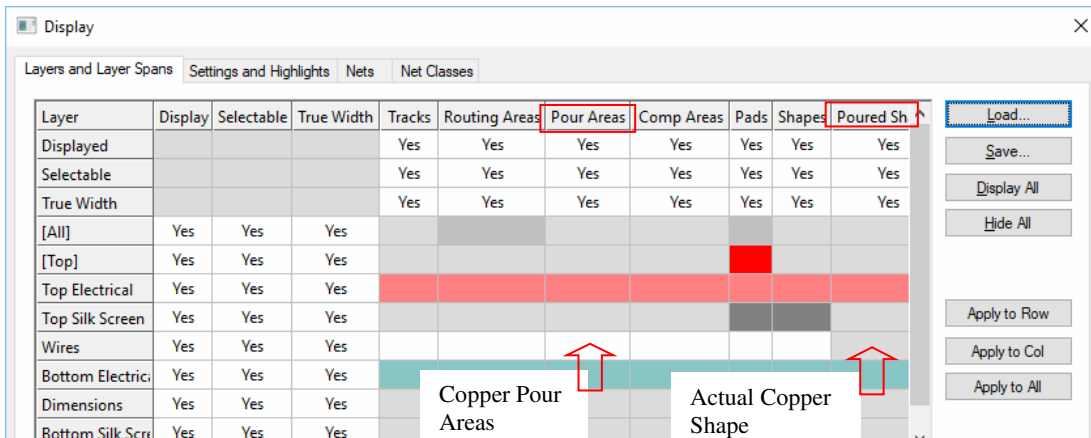
Copper Pour Areas

A **Copper Pour Area** is used to define a boundary in which the copper is to be 'poured'. Using these areas as templates for pouring provides the ability to re-pour areas and to clear blocks of copper within that area. It allows net association to be made where connection of the copper to a particular net, optionally with thermal relief, if required. It also allows properties or rules for that template to be defined.

The **Copper Pour Area** shape options are located on the **Add** menu. Choose from four shape types; Rectangle, Closed Shape, Open Shape, Circle, Triangle, Square and Single Line.

Copper Pour Areas can only be added to electrical or power plane layers. They cannot be added to non-electrical layers. When inserted, these areas are added to the current layer, this can be changed using the <L> key during insertion or after insertion.

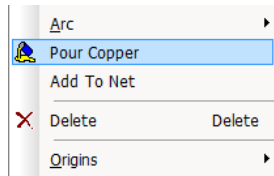
The **Copper Pour Area** is drawn using the **Pour Areas** colour in the **Colours** option on the **View** menu. The actual copper uses the **Poured Shapes** colour in the **Colours** dialog. These colours are layer specific depending on which layer the area or shape appears.



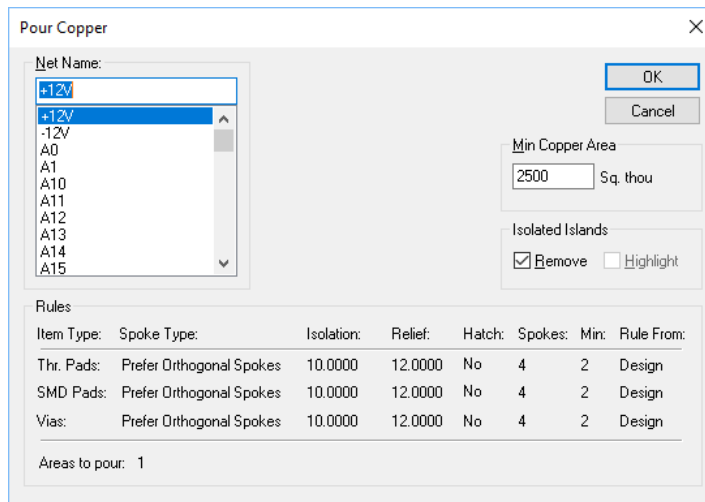
Pour Copper

► To add a copper pour area

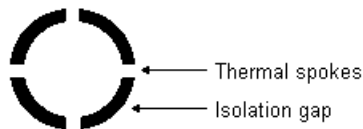
1. From the **Add** menu, select the **Copper Pour Area** shape required.
2. Select the area shape type and draw your shape.
3. Copper Pour Areas can be modified using the same editing methods as any other shapes or areas.
4. When the final shape has been completed, select it and from the context menu you will be presented with a context menu of commands applicable to Copper Pour Areas:



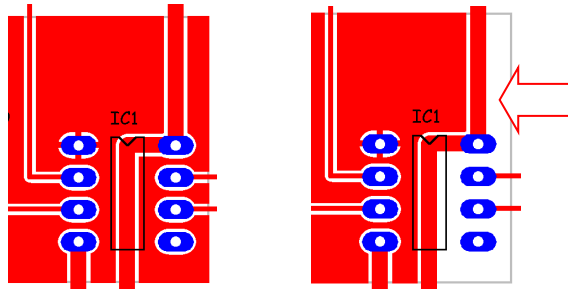
5. On this menu **Pour Copper** (and **Clear Copper** once the area is poured) are used.
6. As their names suggest, **Pour Copper** is used to create the poured area and **Clear Copper** is used to remove areas of poured copper from the design.
7. On selection of the **Pour Copper** option, a dialog is displayed.



8. This allows you to associate a **Net Name** to the copper, to control the **Minimum** area size of copper poured and the **Isolated Islands**. Values for **Thermal relief** and **Isolation gaps** used for the thermal 'connections' of the attached copper spokes are displayed. These are taken from the rules defined in the **Design Technology** dialog and **Rules** page.



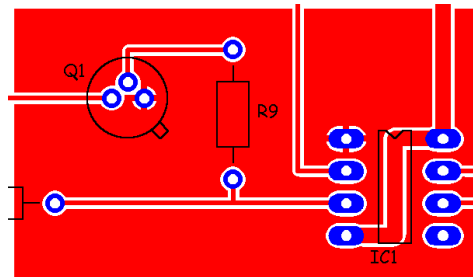
9. Check the **Isolated Islands Removal** button or the **Highlight** button if you wish to control how isolated islands are handled. Isolated islands are created when copper is required but there is not enough space to connect it to the next shape.



The picture above shows the effect with isolated areas retained.

This picture shows how the isolated area (on the right of the main copper area) has been removed because the check box is selected.

- Click the OK button to 'pour' the copper into the area.



Net Association

Poured copper areas can be made without any net association and are used to add more copper to an electrical layer if required avoiding the tracks, vias and pads etc. Net association will mean that thermal spokes and isolation gaps are automatically added to pads on that net during the creation of the copper area.

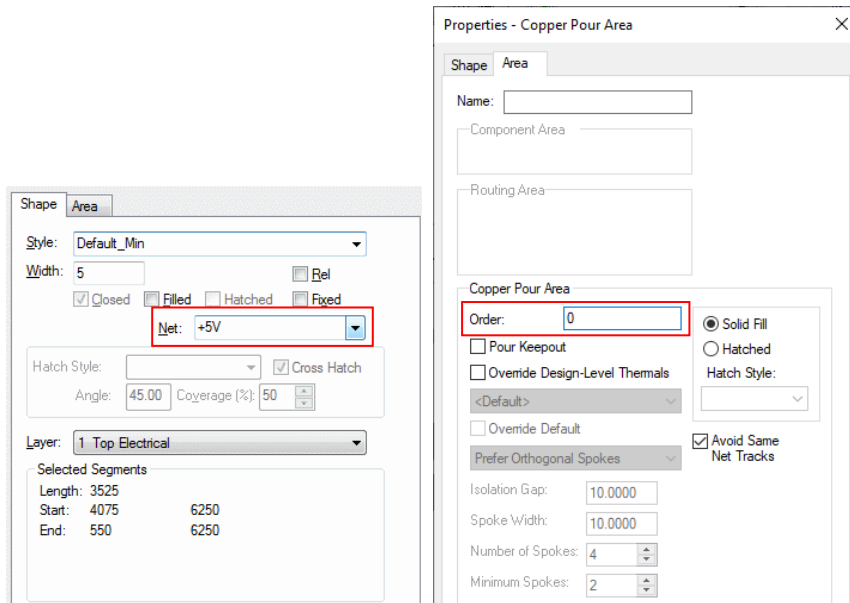
The spoke width will be defined using the width of the **Spoke Style** defined in the **Rules** of the **Design Technology** dialog:

<p>Power Planes</p> <p><Default></p> <p>Prefer Orthogonal Spokes</p> <p>Isolation Gap: 0.254</p> <p>Spoke Width: 0.254</p> <p>Number of Spokes: 4</p> <p>Minimum Spokes: 2</p> <p>Pads and Drills</p> <p>Drill Spacing: 0.000</p> <p>Min Annular Ring: 0.127</p> <p>Min Paste Size: 0.076</p> <p>Min Via Annular Ring: 0.127</p> <p>Min Hole Size: 0.000</p>	<p>Component Height Checks</p> <p><input type="checkbox"/> Top Max Allowed Height: 0.000</p> <p><input type="checkbox"/> Bottom Max Allowed Height: 0.000</p> <p>Value Name: Height</p> <p>Component to Component Spacing</p> <p>Top Side: 0.000</p> <p>Bottom Side: 0.000</p> <p>Boards</p> <p>Board Spacing: 0.000</p> <p>Panel to Board: 0.000</p> <p>Test Land Checks</p> <p>Minimum Pad Width: 2.540</p> <p>Minimum Separation: 6.350</p> <p>Probe Side: Bottom</p>	mm
--	--	----

When associated with a net, leaving 'Thermal Pads' unchecked will pour copper over the the top of the associated pads. This may be a requirement for some design applications.

During the pour of an area, all spacings used are as defined in the **Spacings** dialog. For each net the **Guard Band** spacing is also considered and added to the spacing already defined as a default. For example, a Guard Band spacing of 50 thou would be added to a default spacing of 10 thou to create an overall clearance between the copper area and a particular track/via/pad of 60 thou.

Copper Pour Areas can be assigned a **net name** and a **pour order** using the **Properties** dialog and **Shapes** and **Area** tabs. A Pour Order means that all pour areas in the design can be poured at once using the **Copper Pour** command from the **Tools** menu. They will be poured in numbered order in ascending order.

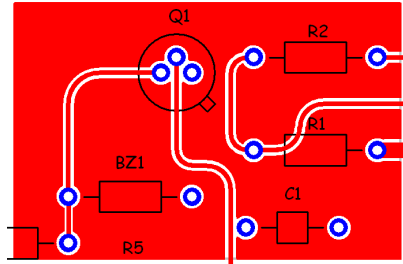


Warning: Do not pour copper onto layers which have been defined in the Layers page of the Design Technology as Powerplane. When plotted this can cause adverse results. See below for Split Power Plane usage.

Editing the copper poured shape

To edit a copper shape, **the poured copper should be unpoured first using Clear Copper from the context menu.** Do NOT use Delete to remove the copper shape. The **Copper Pour Area** template can then be edited and re-poured. This way all the spacings and net association are maintained.

If curved tracks or shapes are used, the shape looks like this:

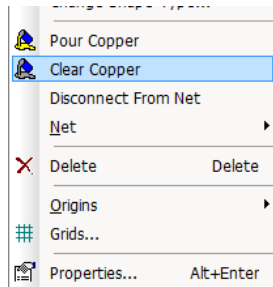


Clear Copper

Clear Copper is used to 'unpour' the copper area. This must be used to remove copper before editing or repouring the area. Any poured copper within a Copper Area will be removed when Clear Copper is selected. Copper which has not been poured but has been manually added will not be removed.

► To clear a copper pour area

1. Select the **Copper Pour Area** boundary to unpour the copper area.
2. From the context menu select **Clear Copper**.

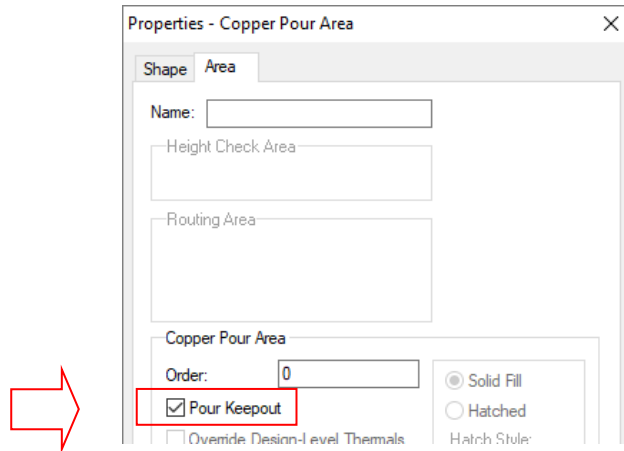


3. All the copper associated with that poured area will be removed including thermal spokes. Do not use delete on copper areas, this will not clear all the associated copper shapes created.

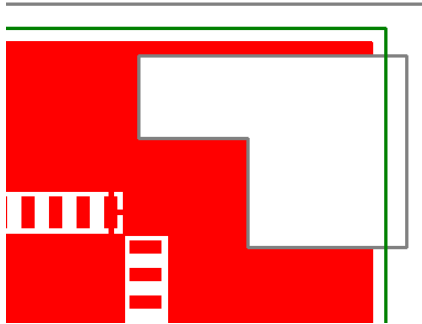
Copper Pour Keepout Areas

Adding a **Copper Pour Area** to a power plane layer and designating it as a **Pour Keepout** will cause that region of the layer to be kept clear of copper on the resulting plane.

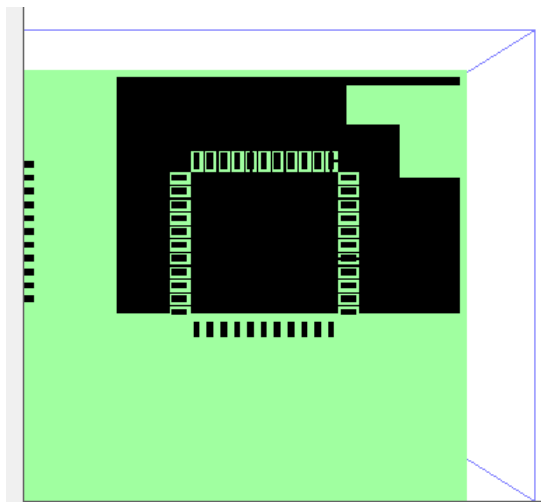
To designate the Copper Pour Area as a Keepout Area, select the area and then in **Properties**, select the **Pour Keepout** check box.



When the Pour Keepout area is added to the design, copper is avoided when poured.



The area will be shown on the **Preview** in the **Plotting & Printing** dialog to assist you in checking that the area is set up correctly.



Split Power Planes

The **Copper Pour** feature can also be used to create split power planes (split planes are two or more copper areas on the same layer, normally an inner layer that contain two different signals).

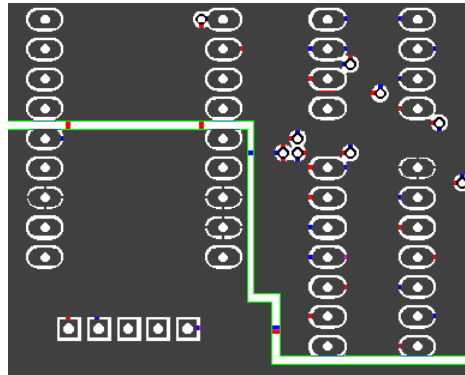
To create a split plane there are a number of rules that should be observed:

- The layer must be defined in the **Layers** dialog as an **Electrical** type.
- The layer can be defined as **Bias** of **Powerplane** but must not have a net associated with it.
- The **Copper Pour Areas** must be defined on that layer and must each be associated with the required power/ground nets that will connect to it.
- The Copper Pour Areas must be poured.
- The layer must be plotted as if plotting a normal electrical layer.

By using **Copper Pour Areas** on an inner power plane layer, and a layer nominated for plane use (although the layer doesn't need to be a type of Plane as long as it is an electrical layer) but not associated with any nets, a split plane can be generated.

Enough space should be left between each of the copper areas, although when poured the copper areas will be spaced to at least the distance of the **Copper to Copper** spacing rule defined in the **Spacing** dialog.

When poured, the split planes will have this effect:



This would then be plotted as you would a 'normal' electrical layer. The power plane has effectively already been generated using the copper pour and no other process is now required other than plotting it.

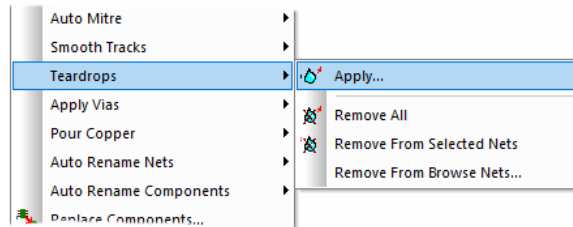
Teardrops

Teardrops are supported in the PCB design. A **Teardrop** is a copper shape which *smoothes* the entry of a track into a pad, causing the pad shape to resemble a *teardrop*. Adding teardrops would normally be done near the end of the layout process, when track positions have been finalised.

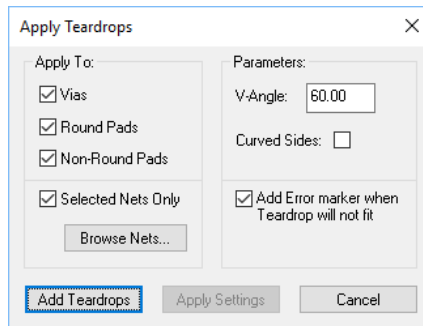
Use the **Apply Teardrops** command from the **Tools** menu to add teardrops to all or selected nets. A teardrop is just a copper shape which you can later apply most of the normal edit commands on. So, for example, you can select and delete individual teardrop shapes once

they have been added. A teardrop will not be added if there is not sufficient track length on the track segment leaving the pad, or if it would cause a design rule error.

These shapes are not dynamically changed, so they need to be deleted and regenerated if the design is changed.



When the **Apply Teardrops** option is selected, a dialog is displayed to allow you to define parameters of the teardrop.



You can control which types of pad receive teardrops using the **Apply To** check boxes. Check the **Vias** box if you want Vias to receive teardrops. Use the **Round Pads** and **Non-Round Pads** boxes to control if all pads other than Vias will receive teardrops. You may not want **Non-Round Pads** to receive teardrops because they do not produce the same smooth effect of round pads.

As well as controlling which types of pads receive teardrops, you can select which nets. If you had nets selected when you entered the dialog, then the **Selected Nets Only** box will be checked, and only those selected nets will be used. Uncheck this box if you want teardrops applied to all nets in the design. Alternatively, you can use the **Browse Nets** button to select which nets will be used, once you have done this, then the nets you selected in the **Browse Nets** dialog replace those initially selected when you entered the Apply Teardrops dialog.

You can control the appearance of new teardrops by changing the **Parameters**. The **V-Angle** is the angle formed at the *sharp* end of the teardrop. The smaller the V-Angle, the longer and more slender the teardrop will be. The default value of 60 degrees is a typical value. You can choose to curve the sides of the teardrop by checking the **Curved Sides** check box, this gives a better transition into the track itself.

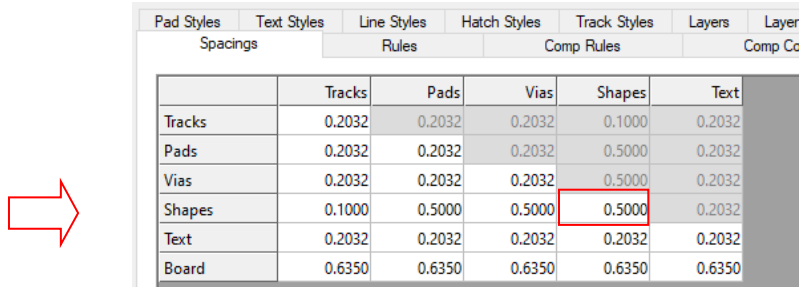
The **Add Teardrops** button will add teardrops to the design. Any existing teardrops will be removed from affected nets, before the new ones are added.

The **Apply Settings** button simply stores any changes made in the dialog but it does not add any teardrops.

DRC Track Spacings for Teardrops

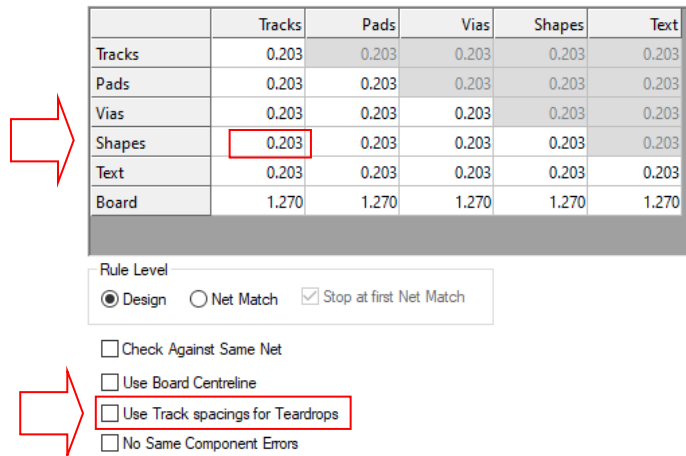
A check box on **Spacings** page of **Design Technology** dialog instructs Easy-PC to use the spacings defined for tracks when dealing with teardrops. This is used instead of the default setting which is to use the Copper **Shape** spacings.

The pouring will use the **Shape to Shape** spacing if this option is not checked:



Spacings		Rules		Comp Rules		Layer
	Tracks	Pads	Vias	Shapes	Text	
Tracks	0.2032	0.2032	0.2032	0.1000	0.2032	
Pads	0.2032	0.2032	0.2032	0.5000	0.2032	
Vias	0.2032	0.2032	0.2032	0.5000	0.2032	
Shapes	0.1000	0.5000	0.5000	0.5000	0.2032	
Text	0.2032	0.2032	0.2032	0.2032	0.2032	
Board	0.6350	0.6350	0.6350	0.6350	0.6350	

If the **Use Track spacing for Teardrops** check box is selected, it will use the **Shape to Track** spacing instead:



	Tracks	Pads	Vias	Shapes	Text
Tracks	0.203	0.203	0.203	0.203	0.203
Pads	0.203	0.203	0.203	0.203	0.203
Vias	0.203	0.203	0.203	0.203	0.203
Shapes	0.203	0.203	0.203	0.203	0.203
Text	0.203	0.203	0.203	0.203	0.203
Board	1.270	1.270	1.270	1.270	1.270

Rule Level
 Design Net Match Stop at first Net Match

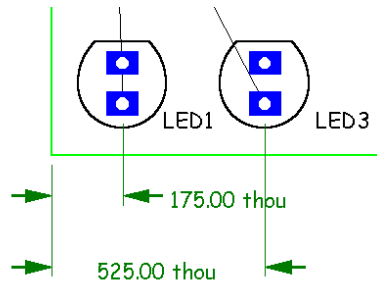
Check Against Same Net
 Use Board Centreline
 Use Track spacings for Teardrops
 No Same Component Errors

Removing Teardrops

You can remove individual teardrops by selecting and deleting them. There are three commands which allow you to remove them in batches. **Remove All Teardrops** will remove all teardrops from the whole design. **Remove Teardrops From Selected Nets** will remove all teardrops from the nets currently selected. **Remove Teardrops From Browsed Nets** will remove all teardrops from the nets selected from a dialog.

Dimensions

For documentation purposes in a PCB design, you can interactively create linear dimensions that show the distance between two chosen points.



On the **Add** menu in PCB Design you have the following options; **Horizontal Dimension**, **Vertical Dimension**, **Radial Dimension**, **Angular Dimension** and **Free Angled Dimension**. There are also three buttons on the PCB toolbar to add dimensions:



Selecting one of these options puts you in Add mode for the appropriate type of dimension.

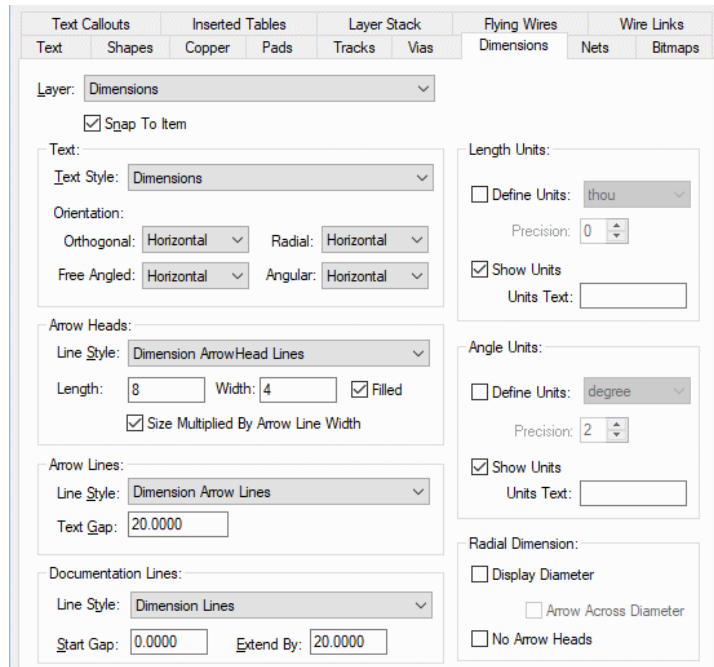
► To add dimensions

1. Select the **Add Dimension>** option from the **Add** menu.
2. The interaction is very simple; the first mouse click starts the dimension at the designated position.
3. The second click places the end position.
4. The third click fixes the length of the construction lines.
5. Drag can be used at each stage instead of click and move, if preferred.
6. The dimension text will be automatically updated to show the measurement between the dimension start and end arrows in current units. This will be automatically positioned between the arrows or to one end of the dimension as space dictates.
7. Once added, a dimension may no longer be manipulated as a composite item, only as individual shapes and text. For removal, rather than editing a dimension, it should be frame-selected, deleted, and replaced by a new one.

Dimension Defaults

Dimension settings may be modified by using the **Design Technology** dialog and the **Line Styles** and **Text styles** pages.

When adding Dimensions, their 'look' is defined from the **Defaults** page and **Dimensions**.

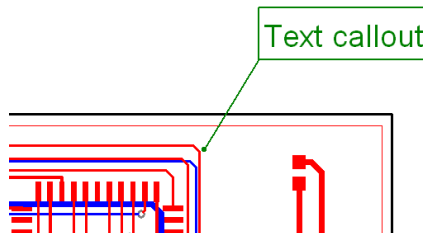


This dialog allows you to view and change the line styles, text style, layer and other settings used when dimensions are added.

The **Snap To Item** check box will cause the dimension start or end point to be at the closest edge of the item selected. If a pad is being dimensioned the snap point will be at the pad's centre. This option may also be toggled via the right mouse context menu when adding a dimension.

Text Callouts

There is a composite design item in both PCB and Schematic designs called **Text Callouts**, used for documenting points of interest.

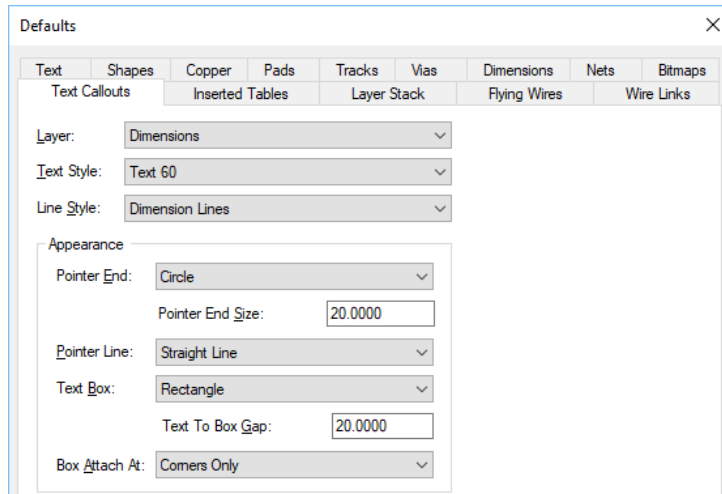


Text callouts consist of a text item, a shape around the text (the text box), a line from the text to the location of interest (the pointer), and a shape at the end of the line (the pointer end). These items are invisibly grouped so that if any one item is selected, the whole callout is selected, and so the callout is always considered as a single item.

These are available on the **Add** menu, **Add Text Callout** and on the PCB toolbar.

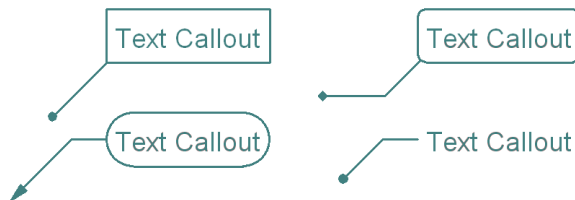
► To use Text Callouts

1. First select the pointer end position, you will be presented with a dialog to supply the text and layer. The layer can only be a non-electrical layer. After this the callout is displayed on the screen and you can place the text item at the required position.
2. The defaults for the styles, layer and appearance of new callouts are defined in the **Defaults** dialog. You can override this by pre-selecting a Text Callout or **Free Text** item before using the **Add Text Callout** function.



3. Once added, you can alter callouts by dragging the text item, dragging the pointer end, moving the whole callout by dragging any other part of it, and changing cutouts appearance by double-clicking on it and using the **Properties** dialog. You can change the appearance of each of the three shapes.
4. By default the text and shapes are drawn using the normal text and shape colour, but you can also highlight all callouts in a single supplied colour using the colours dialog. Individual callouts can also have their own colour using the properties dialog.
5. Different styles of text callouts can be achieved by changing their **Properties**.

The illustration below shows the various callout options:



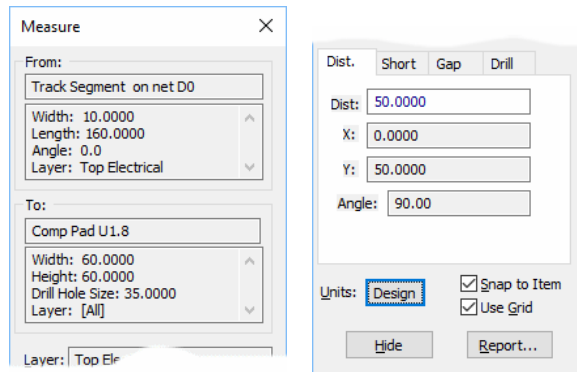
Measure

This is an interactive option to **measure** between two design items.

► To use measure

1. Select the **Measure** option from the **Tools** menu.

2. The **Measure** option appears, this is a floating dialog.
3. This prompts you to select the first item to be measured. This window can be moved if it obstructs the items you wish to measure.

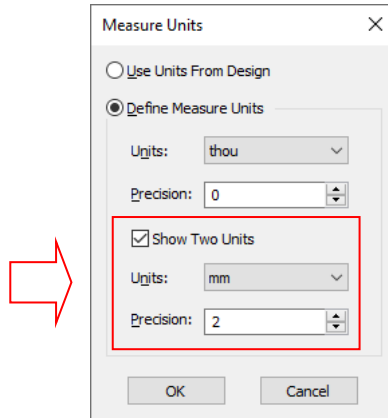


4. Select an item, then a second item.
5. The results window shows various distances between the two items.
6. This is the real distance between the points as would be calculated through the DRC option and is measured against the closest point of each item selected. The electrical items selected will be highlighted in the selection colour to show which are chosen. A small line also indicates the points chosen for the measurement as confirmation.
7. The window can be cleared using the small cross [X] or by selecting <Esc> on the keyboard. This will also cancel the option when you have finished.
8. Sometimes there is no measurement that can be shown, for example if the two items are track segments on different layers; this is indicated in the result window too.
9. Click the cross in the top right corner of the dialog to exit this mode or press the <Esc> key on the keyboard.

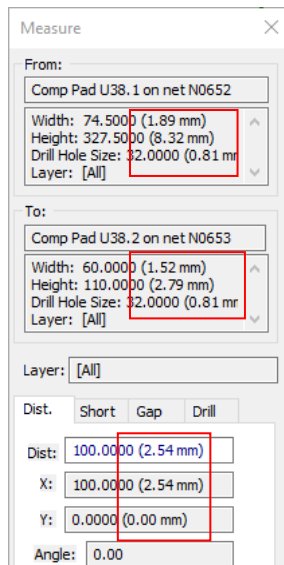
Dual Design Units

The **Measure** dialog/panel can display lengths/areas in two different units. From the main **Measure** dialog, click on the **Units:** button (the one that shows the current units) to access the **Measure Units** dialog.

This dialog has an extra check box **Show Two Units** and a second pair of controls to let you set up the secondary units.



Once this is set up, any measurements you do will be displayed in both sets of units. The second unit set being shown in brackets () on the Measure dialog:

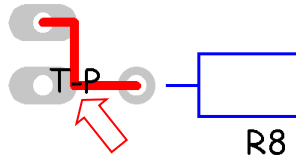


Design Rules Checking

The **Design Rules Checking** option will allow you to check your design prior to sending it away to be manufactured. The rules checked against are set up in your Technology file and passed into the design in the **Spacings** dialog available from the **Design Technology** dialog on the **Settings** menu.

While it is perfectly acceptable for unrouted tracks (connections) to criss-cross the board, routed tracks must not cross each other, and should be routed with a certain minimum clearance between each other and between tracks and pads.

When this option is run, if DRC errors are found, the error will be highlighted using an **error marker** with an appropriate code for that rule violation. This will help you identify and correct the error.



Setting up clearances

The clearances used for the check are set up from **Spacings** page on the **Design Technology** dialog. All combinations of tracks, pads, vias, shapes, text and board outline can be specified independently.

	Tracks	Pads	Vias	Shapes	Text
Tracks	10.0000	10.0000	10.0000	10.0000	10.0000
Pads	10.0000	10.0000	10.0000	10.0000	10.0000
Vias	10.0000	10.0000	10.0000	10.0000	10.0000
Shapes	10.0000	10.0000	10.0000	10.0000	10.0000
Text	10.0000	10.0000	10.0000	10.0000	10.0000
Board	10.0000	10.0000	10.0000	10.0000	10.0000

Rule Level
 Design Net Match

The **Rules** page of the **Design Technology** dialog is used to define rules for other design-based objects such as **thermal relief on power planes** and copper pour and **Component Height Checks**.

Power Planes

<Default> ▾

Prefer Orthogonal Spokes ▾

Isolation Gap:

Spoke Width:

Number of Spokes:

Minimum Spokes:

Pads and Drills

Drill Spacing:

Min Annular Ring:

Min Paste Size:

Min Via Annular Ring:

Min Hole Size:

Min Via Hole Size:

Component Height Checks

Top Max Allowed Height:

Bottom Max Allowed Height:

Value Name:

Component to Component Spacing

Top Side:

Bottom Side:

Boards

Board Spacing:

Panel to Board:

Test Land Checks

Minimum Pad Width:

Minimum Separation:

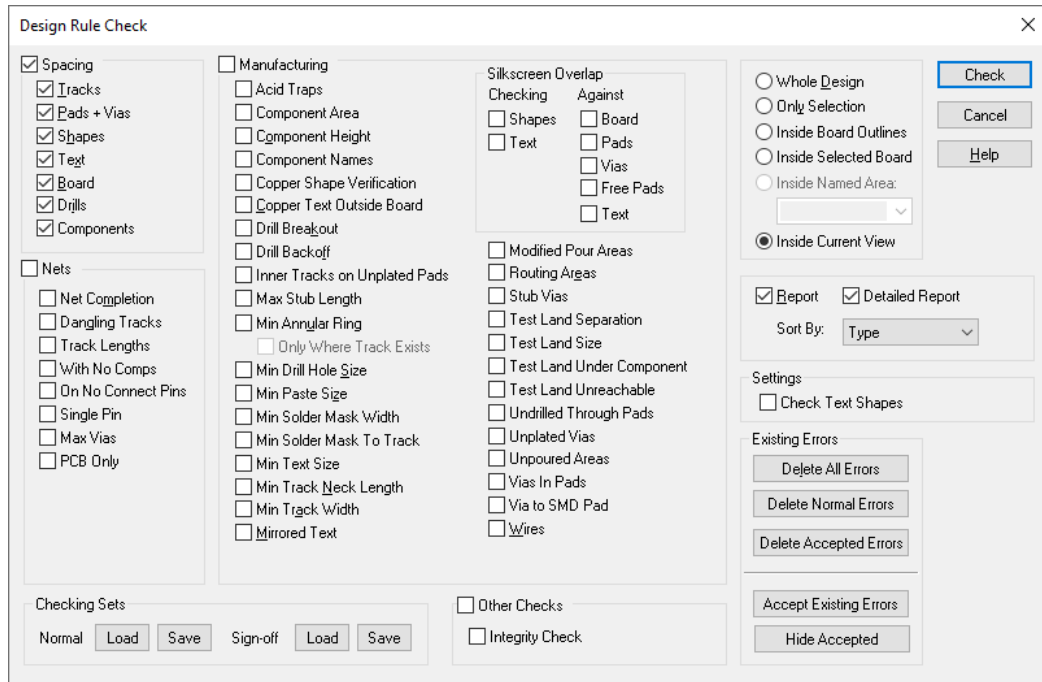
Probe Side:

Acid Trap Checks

mm

Running the DRC option

The **Design Rule Check** option is located on the **Tools** menu.



A dialog controls that items are included in the check. If an area on the design has been selected, it is also possible to check just this selection by checking the **Selection** check box.

The dialog is split into areas of functionality to check – **Spacings**, **Nets** and **Manufacturing**.

The design rule checker can be configured at any time to suit your design and manufacturing requirements. This information is stored in the Technology File.

Resolving Errors

When a Design Rule Check has produced errors, you have a choice of either correcting the error by re-positioning the offending elements, or of relaxing the minimum specified clearances in order for the check to succeed.

Once you have made the appropriate edits you must re-run the checker, or to remove them temporarily (to view a clear design for example) select the **Delete Errors** button on the dialog.

Removing the error marker from the design by using delete or by switching it off in **Colours** does not remove the error as well. Beware, this is dangerous in-case you forget to re-run it before plotting.

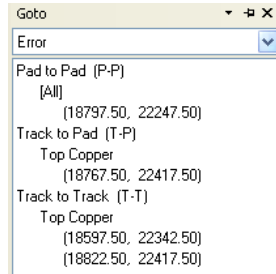
Note that if a group of items are selected, the Design Rule Check can be applied to the selected items only by choosing the Selection option rather than the design option from within the Design Rules Check dialog.

Finding the error markers

Once a Design Rule Check has been completed and one or more errors have been reported, you can move through the errors in turn using the **Goto>** option from the **View** menu. Use the **Goto> Next Error** and **Goto> Prev Error** options. Alternatively, you can use the shortcut keys of <V> and <T> for **Previous** and **Next Error** respectively.

At each error the display will pan and zoom if necessary to show the offending elements in the centre of the screen with a line linking them together. There will also be a text item indicating the type of error, e.g. T-T is a Track to Track error marker.

You may also use the **Goto Browser** and the **Errors** option. Each error can be selected from the list and the design will pan to the centre of the screen showing that error marker.



DRC Error report

The format of the report is similar to most of the others in that it starts with a header block giving information about the design file, followed by a list of all the errors found. Each error is described by type, location and layer, and a summary of the errors found.

Note: If there are several errors between two items, only the first is reported. The occurrence of several errors is most likely with track to track faults. There is no risk in this as re-running the checker will always report the next error until all are cleared, and on large designs it helps keep the report to a manageable size.

DRC Sign-Off Checks

Two sets of DRC settings are stored in the design – one for ‘normal’ checking and the other for running as acceptance checking within the plotting dialog before manufacturing.

The **Checking Set** is used at the start of the **Plotting** process to check your design before producing outputs, these form the **Sign-Off** checks. The Sign-Off checks are used as a final verification that your design conforms to the rules you’ve specified. It highlights important information, errors and warnings the moment before you send off your plots to be manufactured. It will ensure you remember to run the DRC checks and will highlight the errors to you at a critical time in the design process.

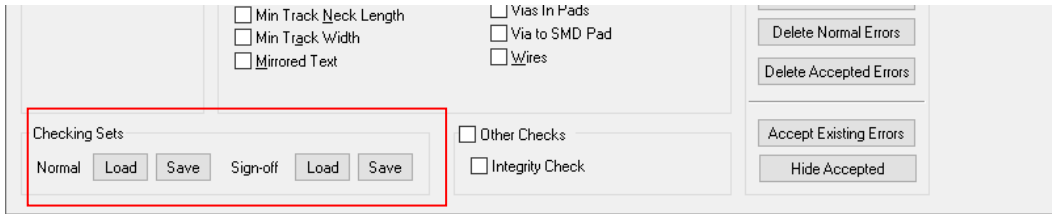
DRC option

There are two sets of DRC settings stored in the Design. You can use the **Load** and **Save** buttons in the lower left corner of the dialog under **Checking Sets** to load either of the two sets into the dialog, and save the current dialog settings away to either set.

- **Normal** - The first is the 'normal' set that is used when running **Design Rule Check**.
- **Sign-off** - The second is a 'sign-off' set that is used when running the Sign-off Checks prior to producing plots.

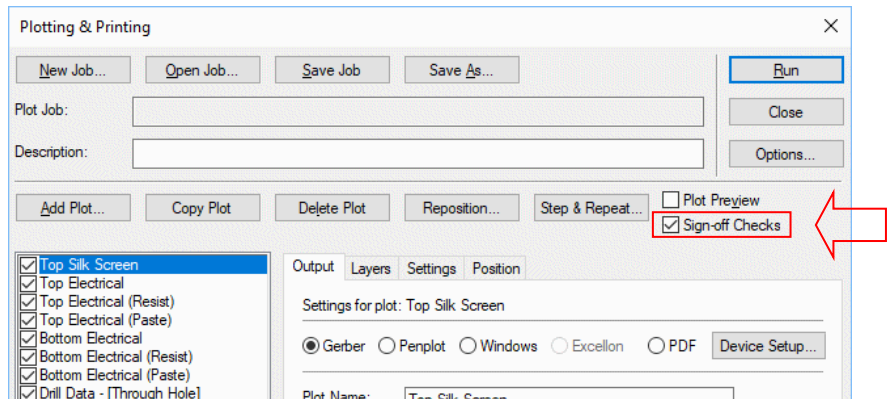
Note: When you click the **Run** button on the **Design Rule Check** dialog to start the checking, whatever settings are present in the dialog at that time will be saved to the 'normal' set.

The two sets of **Load** and **Save** are used for **Normal** and **Sign-off** checking sets.

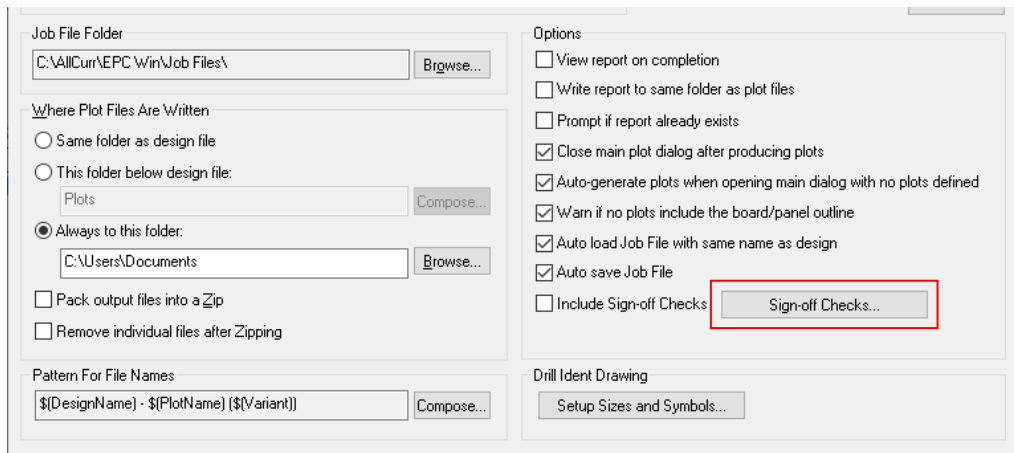


Sign-off checks in the Plotting dialog

A check box on the **Plotting Options** dialog is used to enable the **Sign-Off Checks** when the plots are generated.



A button on the **Options** dialog to setup the **Sign-Off** checks. This is a reduced version of DRC dialog showing only settings, and main DRC dialog has controls for saving and loading both sets of settings.



The **Sign-off Checks** button displays a reduced set of **DRC** options to enable you to select the rules to check.

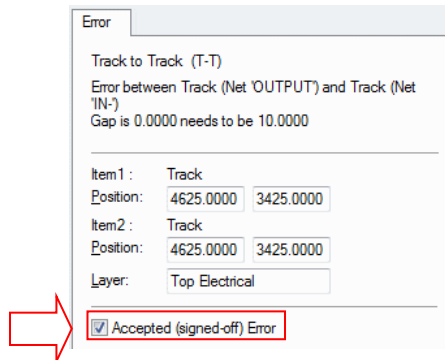
'Accepted' DRC Errors (PCB)

Error markers can be 'accepted' (as 'signed-off'). You may find that DRC finds errors which you are prepared to accept (or 'sign off'). This might happen for example if you have a set of 'finger' pads touching the edge of the board. DRC will flag these with 'Pad to Board' errors, which you can then set as 'Accepted' so that there is a positive indication that you have examined these errors and agree that they are not faults that need to be fixed. Once you have resolved all the faults in the design except those that you are prepared to sign off, you can use the **Accept Existing Errors** button to mark all remaining errors as accepted. Alternatively an error marker can have its Accepted property toggled using the Properties dialog.

Accepted error markers are only deleted by you, either by deleting them interactively in the design, or using the **Delete Accepted Errors** button on the DRC dialog. When you run **Design Rule Check**, any existing accepted errors are left alone.

Accepting errors

Individual errors can be checked as **Accepted** on the **Properties** dialog.



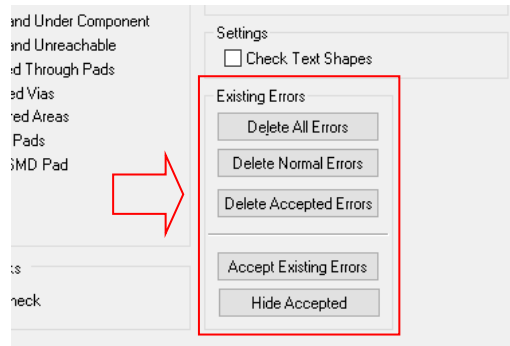
DRC dialog

The **DRC** dialog has buttons to help with clearing and accepting existing errors. These are listed under the **Existing Errors** section:

The buttons in this section are used to deal with existing error markers in the design.

- **Delete All Errors** - removes all existing error markers from the design.
- **Delete Normal Errors** - removes all existing error markers that are not set as 'Accepted' errors.
- **Delete Accepted Errors** - removes only existing error markers that are set as 'Accepted' errors.
- **Accept Existing Errors** - use this button to mark all existing error markers as 'Accepted'.
- **Show/Hide Accepted Errors** – use button to show and hide accepted errors in the design.

On running **DRC**, the option will only delete 'normal' errors and will leave **Accepted** errors in-place.



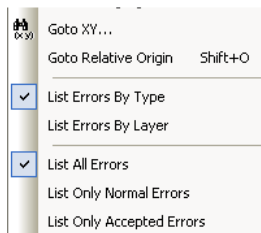
Colours dialog

There are different colours and visibility settings on the **View** menu and **Colours** dialog under **Layers and Layer Spans** and **Accepted Errors**.

	Text	Sym. Pads	Sym. Shapes	Sym. Text	Value Positions	Errors	Accepted Errors
Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes

Find Error types

From the **Goto** bar, on selection of Errors, you can list errors by type. From the context menu within the Goto bar, you can select **List All Errors**, **List Only Normal Errors** and **List Only Accepted Errors**.



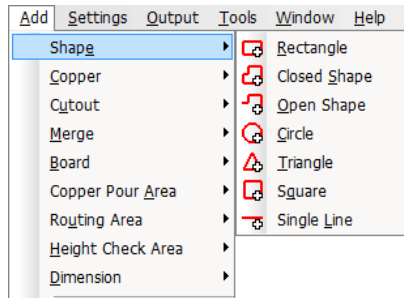
When showing a list of all errors, they are separated into two; **Normal** and **Accepted**, so they are easily visible.

Adding Shapes

You can add shapes both electrical and non-electrical layers. The type of shape that is added to any layer is dependent on the layer type itself. If the layer is electrical, the shape will be copper, if the layer is non-electrical e.g. Silk Screen, then the shape will be a non-electrical silk screen shape. A shape type of *Shape* can be added to electrical and non-electrical layers. It will appear as documentation, silkscreen or copper depending on which layer it has been added to.

You can add **Shapes** (to documentation layers), add **Board** to create a board outline, add **Copper Pour Areas** (to electrical layers) for use with the pour copper option and add **Routing Areas** (to electrical layers) for use with the autorouter. You can also add **Cutouts** to all shapes.

To add a shape, you can use **Add Shape** from the **Add** menu, or from the **PCB Design** toolbar.



Once the shape has been started you can use the context menu to control the **Style** and **Layer** of the shape, and whether it is **Closed** or **Filled**. The tick next to the Closed and Filled entry shows that it is used; by not selecting one of these items shows that it is **Open** or **Not Filled**.

There are no specific rules for the polygon shape other than when filled it must have at least 3 sides, and all segments can only be on the same layer. All segments must have the same width and style, and for polygons, they are not allowed to self-intersect.

Copper shapes on electrical layers can be associated with a net. Copper areas with contoured shape following and net association are detailed in the following section.

Shape Defaults

When adding a shape to the design, the default style used is taken from the **Design Technology** dialog and **Line Styles** page.

	Name	Pattern	Width	Dash Gap
X	0	Solid	5.00	0.00
X	Dimension Arrow Lines	Solid	8.00	0.00
X	Dimension ArrowHead Lines	Solid	1.00	0.00
X	Dimension Lines	Solid	1.00	0.00
X	Line 5	Solid	5.00	0.00
X	Line 10	Solid	10.00	0.00
	Normal	Solid	5.00	0.00
	Outlines	Solid	8.00	0.00

Hatch Defaults

You can make shapes hatched using **Hatch Styles** defined on the **Design Technology** dialog.

	Name	Angle	Coverage	Cross-hatched
	Cross Hatched	45.0	50	Yes
	Hatched	45.0	50	No

The dialog allows you to define hatch styles for hatch (single angled lines) or cross hatch (two sets of crossing angled lines). The angle can be changed as well as the coverage. Changing the coverage allows you to define what percentage of the hatched area has copper on it. A higher percentage will apply more copper to the area. A 50% coverage is normal.

Available Types and their usage

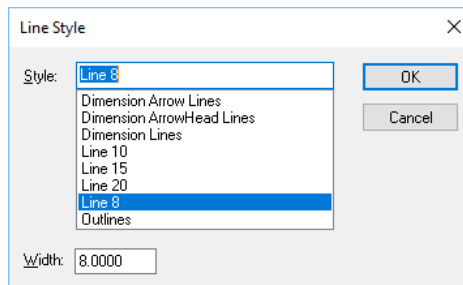
The table below shows the shape type available on the **Add** menu, the shape available for it and its layer usage and usage.

Shape Type	Shape	Layers/Usage
Shape	Rectangle, Closed Shape, Open Shape, Circle, Triangle, Square, Single Line	Electrical and Non-electrical layers, Closed, Open, Filled, Hatched
Copper	Rectangle, Closed Shape, Open Shape, Circle, Triangle, Square, Single Line	Electrical layers, Closed, Open, Filled, Hatched
Board	Rectangle, Shape, Circle, Square	Only <Through-board> (all layers), only Closed, cannot be Filled or Hatched
Merge	Rectangle, Shape, Circle, Triangle, Square	Electrical and Non-electrical layers, Closed, Open, Filled, Hatched
Cutout	Rectangle, Shape, Circle, Triangle, Square	Electrical and Non-electrical layers, Closed, Open, Filled, Hatched
Copper Pour Area	Rectangle, Shape, Circle, Square	Electrical layers, Closed, Open, Filled, Hatched
Copper Pour Keepout Areas	Rectangle, Shape, Circle, Square	Electrical layers, Closed
Routing Area	Rectangle, Shape, Circle, Square	Electrical layers, Closed, Open, Filled, Hatched
Component Area Height Area	Rectangle, Shape, Circle, Square	Electrical layers, Closed, Open, Filled, Hatched

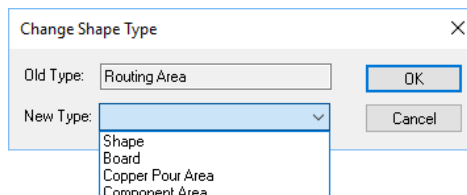
Shape Usage

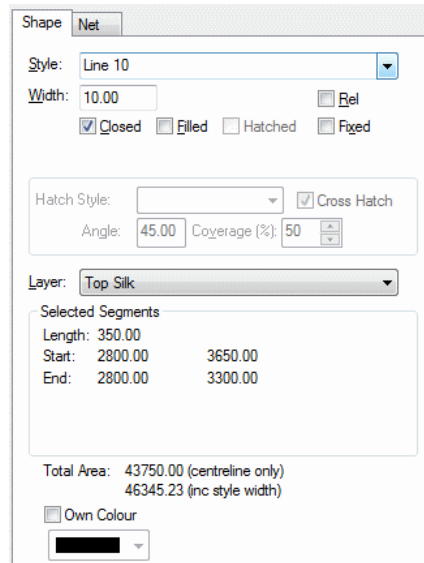
Once added to a design, the **Layer** of a shape can be changed by selecting it choosing <L> **Change Layer** from the context menu.

The shape **Style** can be changed by selecting the shape and using **Change Style** <S> from the context menu.



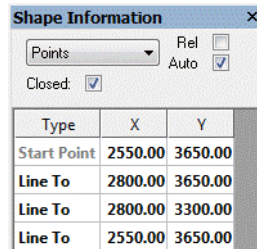
If you need to change the shape type, from Silkscreen to Board for example, use **Change Shape Type** from the context menu.



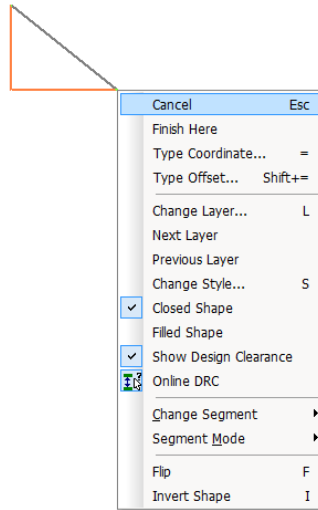


If you wish to change a shape's usage (Closed/Open, Filled/Not Filled, Hatched/Cross Hatched/Not Hatched, Fixed/Not Fixed etc.) select it and choose **Properties** from the context menu.

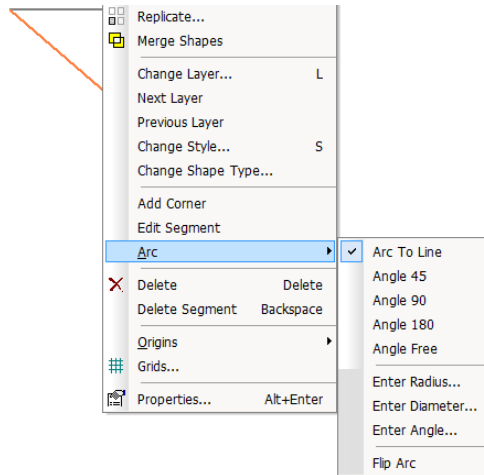
To aid the drawing of shapes, you can use the **Shape Information** browser available on the **View** menu.



You can also edit the shape during addition or interactively after adding, by selecting options from the edit context menu. **Type Coordinate** and **Type Offset** can be used to place corners and shapes whilst editing.

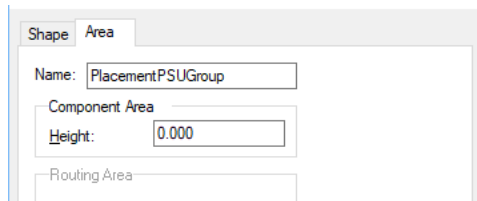


The **Add** and **Delete Corner** options are available also for selected segments when editing shapes as is the **Arc** option to make the segment curved.



Named Areas

You can name some areas using the **Properties** dialog.



Naming an area allows it to be referenced by name in the **Arrange Components**, **Autorouting** and **Autoplace** options, as well as the **Height Checking**.

Component Height Checking

From within the PCB design editor Height Checking is available for Components. This allows you to define a special height area on the design with a maximum allowed height. The maximum height allowed can be assigned on the whole board (using the **Design Technology** dialog and **Rules**) or on an area basis (using **Component Areas**).

Using Height Checking

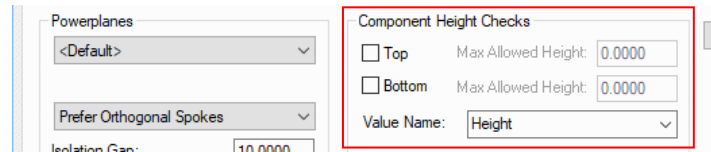
The process for using this option is:

1. Define the Height **Value** to be used in the **Rules** page of the **Design Technology**.
2. Optionally, define the **Side** to be checked and the **Max Allowed Height** for that layer in the Spacings dialog.
3. Add a **Value of Height** to the components to be checked in the design. Do this in the component library or in the design itself (using **Properties** or **Component Values** editor) on a component basis (it is more efficient to do this once in the component library for each component type).
4. Run the **DRC** option and select **Height** under **Manufacturing** rules.

Rules dialog

On the **Rules** dialog on the **Design Technology** dialog, there are controls for setting up the maximum **Component Height Checks** for Top and/or Bottom layers, and choosing the default **Value Name** for components to define their height.

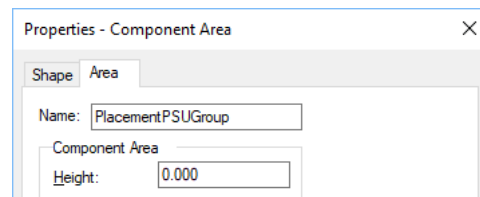
The values used are in your current design units.



Height Check Areas

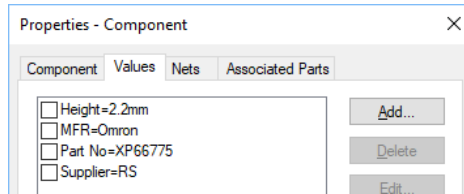
On the **Add** menu, there is an option to add **Component Areas**. These areas are layered and have a maximum **Component Area Height** setting available in their **Properties**.

When the area is selected, **Properties** will allow you to change the area **Height** for checking. That is, components within this area will have their **Height Value** checked against the maximum **Height** defined for the Area (defined in the **Spacings** dialog).



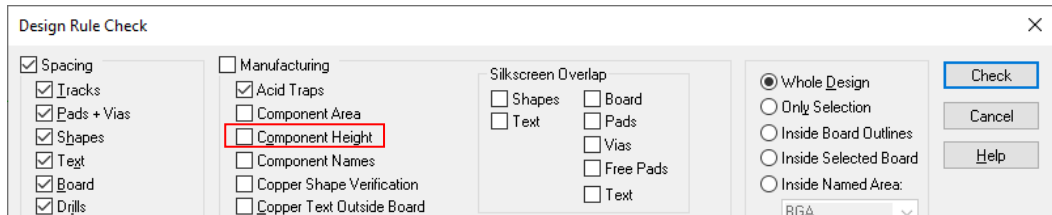
Component Properties

For **Properties** of **Components**, you should add a **Height** value (Height is the default value but you can define and use any **Value** you wish). This value can be assigned locally within the design on a component basis but it is more efficient to define this in the **Component library**.



Design Rules Checking

On the DRC dialog, a check box for **Component Height** under **Manufacturing** is used. Use this for checking the Values and Spacing Heights of components.



Integrity Checking

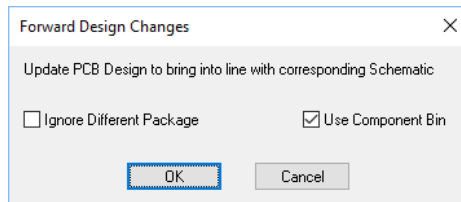
The **Integrity Checker** is used to check that the Schematic and PCB are 'in-step' electrically. Any problems with the design not being synchronised can be corrected using the **Forward Design Changes** or **Back Annotation** options. This is an important option.

The **Integrity Check** option is run from the **Tools** menu. It begins to run immediately, displaying its progress in a window in the screen.

On completion, it reports success or failure in the form of a **Design Integrity Status report**.

If any **Back Annotation** changes are pending, you will be warned. These must be back annotated to the Schematic design prior to using the **Forward Design Change** option.

The **Integrity Checker** works by extracting two connectivity netlists, one from the Schematic, the other from the corresponding PCB design. The two netlists are compared and any differences reported. If any discrepancies are found, you can bring the PCB in-line with the Schematic by using the **Forward Design Changes** option which is automatically offered.



To make the Schematic design match the PCB you will have to note the changes or print the report and make them by hand manually. Design changes should always be made to the Schematic and brought forward using the **Forward Design Changes** option.

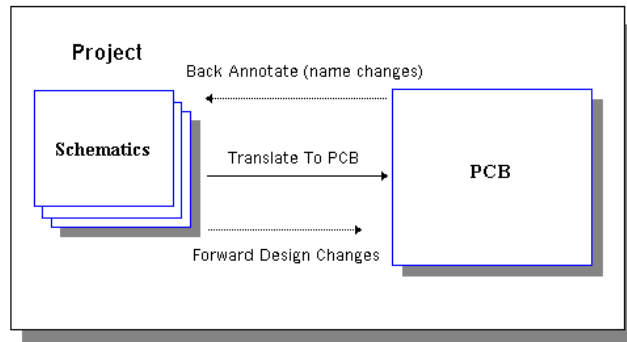
The **Integrity Check** option can also be run as part of your **DRC** checks or **Sign-off Checks**.

Bringing Design Changes Forward from Schematics

Both **Forward Design Changes** and **Back Annotation** are related although they are used independently. They both work to achieve design synchronisation between related Schematic and PCB designs.

Forward Design Changes executes a partial netlist translate from SCM to PCB to update the design. This means that design changes made in the Schematic can be forwarded into the PCB design automatically.

The diagram below indicates the 'flow' of information between the applications:



Design changes which are 'pushed' forward include:

- Net name changes/additions/deletions.
- Connection/Net changes/additions/deletions.
- Net Class changes/additions/deletions.
- Component changes/additions/deletions.
- Component Values changes/additions/deletions.
- Package changes/additions/deletions.
- Pin changes additions/deletions*.

* disconnects track from pad end using the corner of the last segment inserted and reconnects to new pad without altering track path significantly.

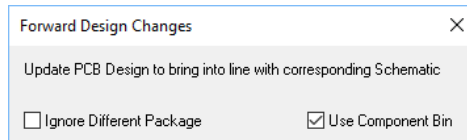
Undo levels are retained for all forward design changes (restricted within number of **Undo levels** defined in **Preferences**).

Note: Forward design changes are often called Engineering Change Orders or ECOs and is a terminology normally used in larger companies to describe a 'note' to inform the PCB layout engineer of changes to the design being laid out. Forward Design Changes are used to best describe the function being performed. It is this function used if an ECO is raised to modify the design.

To use this option the design should have been translated from Schematics to make a PCB design first. The PCB design doesn't necessarily have to be placed, simply having the design components and connections in PCB is enough to be able to apply further design changes from the Schematics. Components may also reside in the component bin.

The **Forward Design Changes** option is located on the **Tools** menu in Schematic or PCB design editors, or in Project. Before attempting to perform changes to the PCB, it is advised that the Schematic design be saved though this is not essential.

When this menu entry is selected you are prompted with the dialog shown.



The **Component Library Directory** browser will be used to locate the library for when loading new components into the PCB design. Although stored in the actual application it provides you the opportunity to change the location using the **Browse** button. The **Ignore Different Package** check box will ensure that the component package remains the same even if it has changed on the Schematic. **Use Component Bin** will add any new components to the component bin and will make the bin visible when the design changes are made.

When the **OK** button is clicked, changes are made automatically and a report of the changes and/or errors is produced. The **Undo** option can be used to revert the changes made to the PCB if required.



The PCB design has renames that have not been back annotated to the schematic design.

Use Back Annotation option to apply these renames prior to running forward design change option.

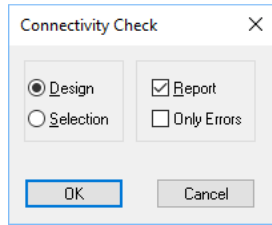
If the design has any name changes which have not been back annotated to the Schematic before a **Forward Design Changes** process is attempted an error message is displayed. Click on **OK** to clear the message and use **Back Annotate** before retrying **Forward Design Changes**.

Connectivity Check

The connectivity is a vital part of a design. When a Schematic is being designed, the connectivity is generated automatically as the interconnections are drawn. After Translation to PCB, or in the case of a PCB only design, the connectivity becomes vulnerable to editing operations. It is all too easy to decide that a net requires to be routed the long way round, and remove the link before manually re-routing it. This is no problem. What does cause a problem is when part of this route is accidentally omitted.

Integrity check compares net lists. A design can pass perfectly and still be faulty because there are two clusters of tracks, electrically isolated, but with the same net name. This is a common result of the above scenario. The **Connectivity Check** detects all such instances, and generates a detailed report to enable corrections to be made. It will also detect places where a net has been started but not finished, so that only one connection is present.

To generate the report, a PCB must be the active design. The report is run from the **Tools** menu and **Connectivity Check**.



The dialog displayed contains a check box for **Report**, which should be checked. You can also choose whether the whole **Design** should be checked, or only **Selected** items. If you choose only selected items, all nets with the same name as any of the selection will be checked, not just the specific highlighted items.

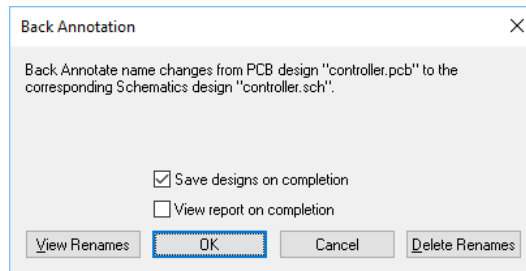
The report comprises of header information for the design, followed by a detailed report of problems found. The last few lines contain a summary, giving a quick overview as to the extent of any problems.

Back Annotation to Schematics

Back Annotation provides a facility to make changes in the PCB design to the net names and component reference names, either manually or in the case of component names automatically, and to then apply those changes back to the master or original Schematic design.

Any changes made to the PCB are automatically stored with the PCB design, there is no separate or additional file to retain with the design. This means that the associated Schematic design will be able to detect whether back annotation changes are required when the option is selected.

Back annotation is run by selecting the **Back Annotation** option in the Schematic design editor or from the PCB design editor from the **Tools** menu. When run from the PCB design editor without the associated Schematic or project open, the system opens the Schematic design automatically ready for the back annotation.



With the Back Annotation dialog you have the option to **View Renames** before they take place and to click **OK** to perform the changes. **Cancel** will exit you from the option without performing Back Annotation. This gives you the facility to check the any back annotation changes as you progress with your design.

```

Back Annotation Renames Report
-----
Report Written : Thursday, June 05, 2022
Design Path   : C:\Examples\Design99.pcb
Design Title  :
Created       : 05/05/22 11:58:05
Last Saved    : 05/07/22 12:00:56
Editing Time  : 48 min
Outstanding PCB renames awaiting back annotation to the
Schematics design(s).
    
```

```

Rename Component from "U2" to "U1".
Rename Component from "U1" to "U2".
End of Report.

```

The report for **View Renames** shows the steps which need to be taken by the program to change the Schematic so that it is in-step with the PCB design.

When the **OK** button is clicked you are presented with a warning dialog which allows you to confirm the back annotation changes. This is your final chance to escape the changes before they are made permanent.

The changes are made permanent with no undo facility available because back annotation is seen as a 'down-stream' design process after the initial PCB design and one which is synchronising the Schematic and PCB designs.

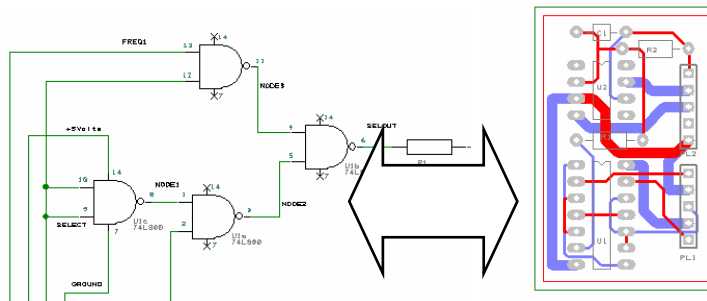
Once the changes are made to the Schematic, the design should be saved.

Back annotation should be used after any changes to PCB net names, after using the automatic component rename option, after making any manual component changes, and before making any changes to the Schematic (after making net or component name changes in PCB). If back annotation changes are pending for the Schematic and a **Forward Design Changes** is attempted, you are prompted to use the **Back Annotation** facility first.

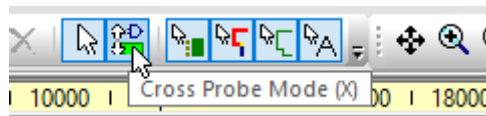
Be warned, if Back Annotation is not run to make necessary changes, the Schematic design can easily become out of synchronisation with the PCB design.

Using the Cross Probe Mode

This feature allows you to probe and locate items in a known design. It enables bi-directional **Cross Probing** between corresponding Schematic and PCB designs, between Schematic designs within a Project or between any open designs. This enables better integration of design data between each stage of the design process.



To use this option, a toolbar button is available to put the applications into the Cross Probe 'mode' prior to probing. There is also an entry on the **Edit** menu, **Cross Probe**, which selects the cross probe mode.



Once this is selected the designs can be cross probed.

Items which can be probed

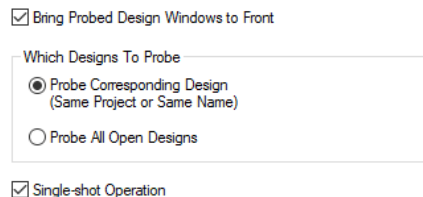
Pins/Terminals	By picking the pin.
Components	By picking a component or symbol gate.
Nets	By picking connections/tracks/net names or copper.
Closed busses	By picking the closed bus containing the same net names (good for viewing all similar closed buses).

Where multiple sheets have been combined to make one PCB, the sheets must be encompassed by a Project in order for probing from the PCB design to search all design sheets. Where Projects are not used and a single Schematic sheet is probed with a corresponding PCB design the design names for both must be the same. These rules apply if you wish to make sensible use of the probing mechanism and can be controlled by the settings defined on the **Preferences** dialog (detailed below).

If different designs are in the same Project and do not have the same name, they can still be probed using the **Probe All Open Designs** switch from the **Settings** menu and **Preferences** option (detailed below).

Cross Probing Schematic to Schematic is useful for checking for multiple component references and symbol gates not located together as well as checking nets spanning multiple sheets. Again though, Schematic to Schematic sheets must be included under a single project.

The **Cross Probe** tab on the **Settings, Preferences** dialog enables Cross Probing to be tailored to be your own preferences.



As its name suggests, the **Bring Probed Design Windows to Front** will ensure that the probed design is always visible by forcing it to the front as the active design.

The two other options under the **Which Designs To Probe** heading allow you to force intelligence into the cross probing. **Probe Corresponding Design** means only designs under the **Same Project or Same Name** will be probed, any other design names will be ignored. If probing a PCB design, a Schematic design not containing probed net names will be opened automatically. If probing a Schematic design, the PCB design will be opened and probed but only the Schematic designs already opened will be probed.

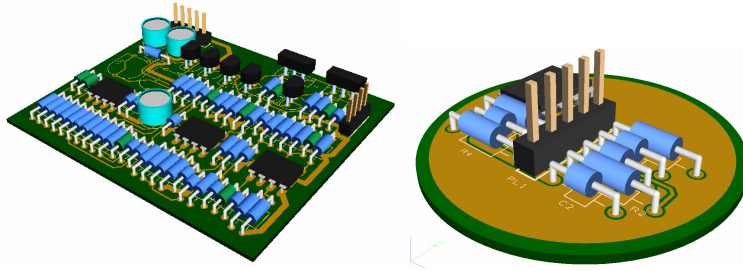
Probe All Open Designs has the opposite effect and allows any design to be probed regardless of its name and even if it doesn't correspond to the other open design - care should be taken using this option as it could mean an irrelevant design is accidentally probed!!

Single Shot Operation enables cross probing

Using the 3D Viewer

The 3D viewer allows the 'flat' view of the PCB to be annotated so that it can be seen as it would be when populated with components after assembly. A menu option **3D View** on the **View** menu in the PCB editor is used to send the current design into the viewer.

The appearance of components in the 3D view is controlled using the 3D Package library type edited from the **Library Manager**.



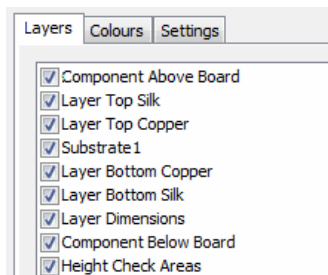
Using the 3D Viewer

The 3D Viewer option is easy to run, use the **3D View** option from the **View** menu and a new window is opened containing the design in a 3D view. Options within this window allow you to customise the look of the design. Packages must be created in the library to represent the 3D view of the PCB footprints within the design. The standard library contains 3D Packages for all supplied PCB symbols but more can be added of your own.

Within the viewer you can manipulate the view by using the zoom and unzoom shortcut keys, **<Z>** and **<U>** respectively. By clicking and dragging the mouse you can roll the PCB in three axis to show all views of it.

Options within the 3D Viewer

The **Options** dialog is available on the **3D** toolbar and on the **View** menu. The tabs within it allow you to define how the 3D view will look.



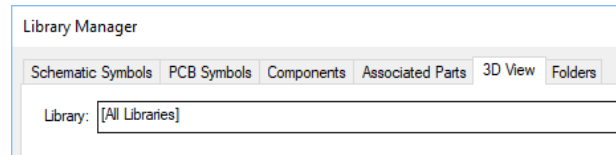
The **Layers** tab will enable you to switch on and off the design layers within the view. Note this is not displayed when viewing 3D Settings in the PCB Editor, only available in 3D Viewer.

The **Colours** tab allows you to setup preferred colours for the 3D view.

The **Settings** tab is used to define view parameters such as the thickness of the board in the 3D view. Each of the entries for **Sizes** defines a thickness that can be created and viewed. Generally speaking, **Board** and **Copper** thicknesses would be consistent, whereas you may change the **Gap** thickness to show internal layers more clearly, perhaps to show the internal via stacks.

Defining the 3D Packages

In order to view components in the 3D viewer, you must define the package 'shape' in the library. There is a tab on the libraries dialog called **3D View**, use this to define package libraries.



The package libraries have the file extension .pkg. The contents of the package libraries have definitions which relate to a footprint or groups of footprints, for example DIP*. The supplied libraries are defined by functional groups i.e. connector, CPS, SMT, DIP

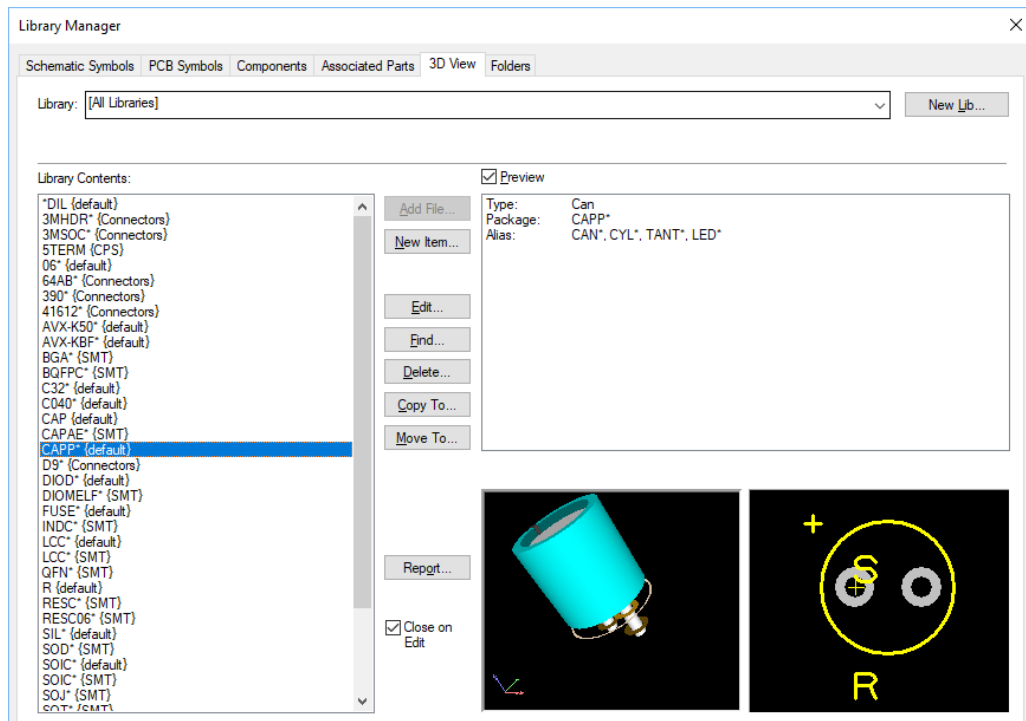
Package Overview

3D Packages define how PCB symbols are represented in 3D View images of your PCB designs. Each 3D Package has a 'style' (DIL, Quad, Axial, and so on), and various settings that control how the 3D image is generated. It also has one or more names that specify which PCB symbols the 3D Package applies to, allowing one package to be used to generate the 3D image for many PCB symbols. These names can be specific symbol names, or they can include 'wildcards' to match multiple symbol names.

To generate the 3D view of the actual components on your PCB design, the set of package libraries is scanned to find the first package that matches each of the PCB symbols used in the design. Because each package can have a basic name and a set of 'aliases' both of which can include wildcards, it is possible to create packages that can auto-generate the 3D view for many PCB symbols.

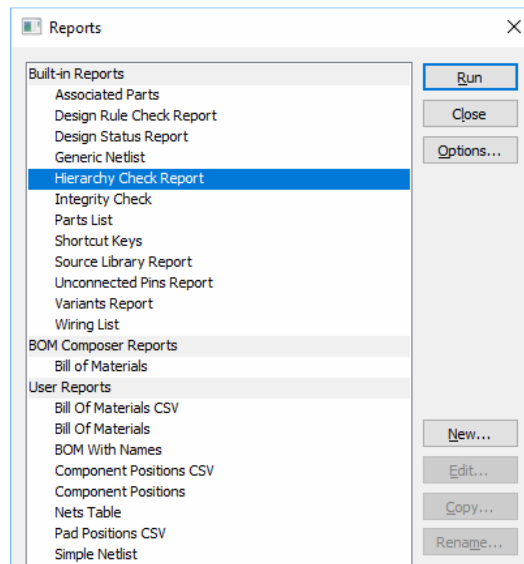
3D Edit Dialog

The **3D View** tab gives a summary of the selected item, and small pictures in 3D and 'normal' view of a corresponding footprint. The PCB symbol that is used for this dialog will be the first one found that matches the rules defined by the selected package.



System Reports

There are a number of report files that can be generated. Some of these are error and status reports, while others give information about the content of the design files.



When created, all the reports automatically open in your nominated viewer for immediate display, with the exception of the comma separated values (CSV) parts lists.

All of the supplied system reports can be accessed from the **Custom Reports** dialog on the **Output** menu.

Component Positions Report

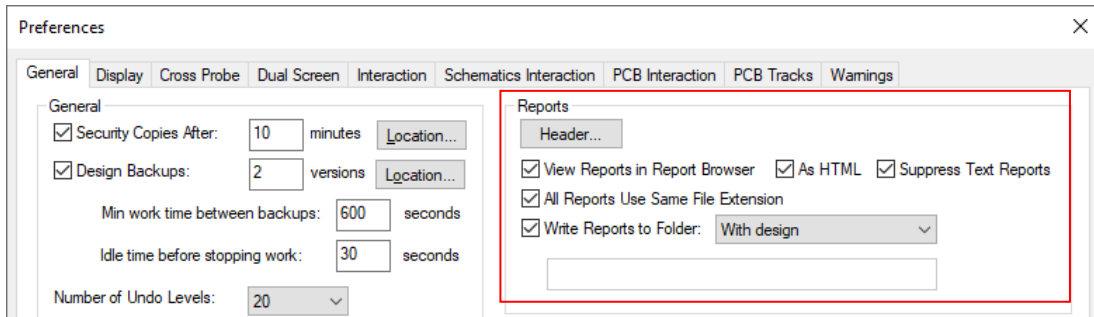
One report of special mention is the **Component Positions Report**. This report will output information suitable for assembly machines. Because this is created in the **User Report** section, it can be edited to your own specification.

```
Component Positions Report
Units : mm (precision 4)
Name      Component      Side      Place X      Place Y      Rotation
J1        6-174681-6      Top       75.0452     174.6250     90.00
J2        6-174681-6      Top       75.0452     98.4250     90.00
U24       74AC174SJ       Top       158.7500    154.9400     90.00
U25       74AC174SJ       Top       189.2300    154.9400     90.00
U11       74LS151         Top       87.6300     91.4400     180.00
```

HTML Reports

An integrated report viewer is provided, this includes 'live' HTML for some reports with links back to design data. The links back enable you to probe report items within the design.

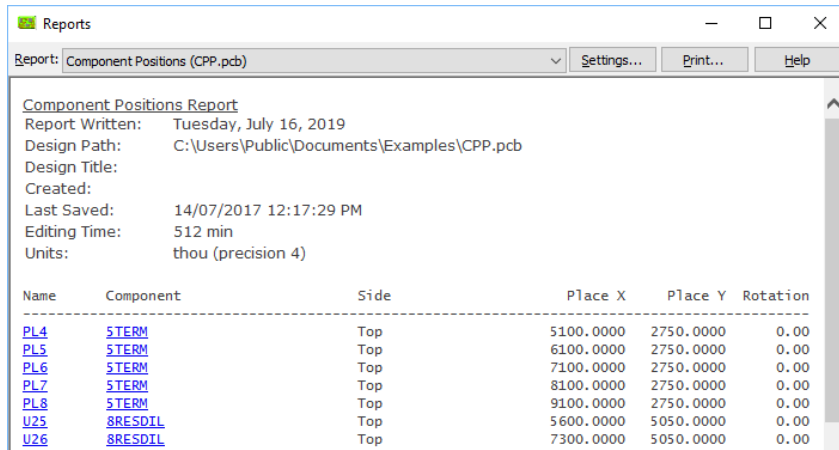
To activate the HTML reports, from within the **Preferences** option (**Settings** menu) and **General** page. Under **Reports**, there are two check boxes: **View Reports in Report Browser** and **As HTML**



If **View Reports in Report Browser** is enabled, any TXT or HTM report that would normally be sent to Notepad or your normal web browser will instead be sent to the **Report Browser**. The only files that are sent outside are be CSV files.

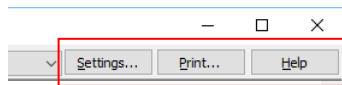
Report Browser

The Component Positions report would look like this for example. Where items are shown as underlined hyperlinks, they can be queried back to the design.



Browser Controls

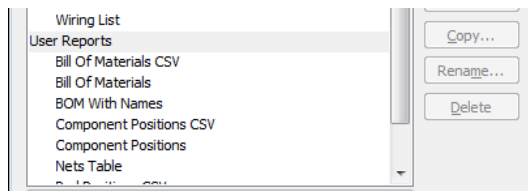
Browser controls along the top of the report allow you to configure the report or print them.



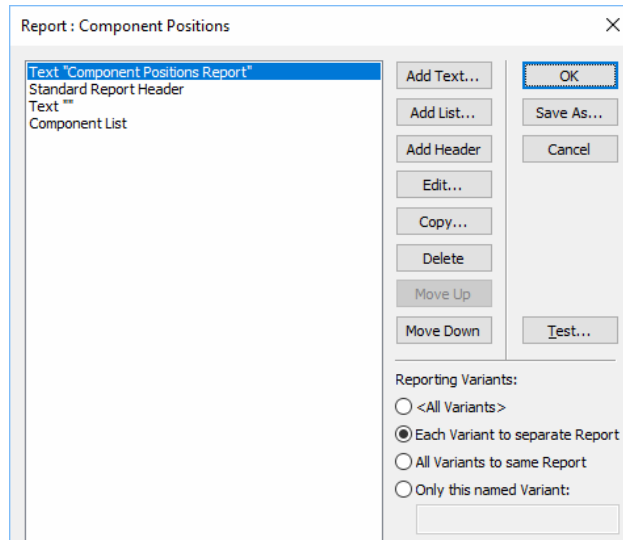
Settings takes you to your **General Preferences**. **Print** - reports in the report browser can be printed to your Windows printer.

Custom Reports

As well as system reports, the **Custom Reports** dialog can be used to create advanced Parts lists.

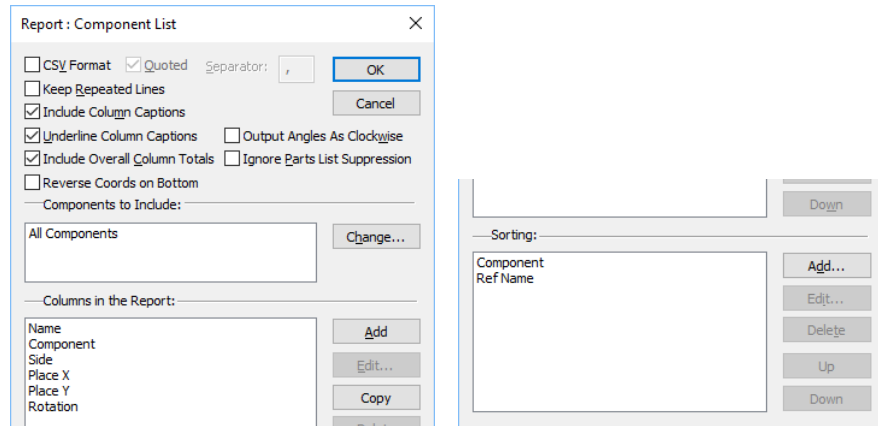


By editing the selected report, you are able to add build information to the report.



This dialog allows you to edit the contents of a component list contained in a user-defined parts list. The settings on this dialog control how the output will be formatted, which components should be included, and in which order they are listed.

Each field can be edited to add detail for when formatting the report.



BOM (Bill of Materials) / Parts List Composer

The **BOM Composer** allows you to create BOM reports and Parts Lists using a simple point-and-click WYSIWYG (what you see is what you get) dialog.

Available on the **Output** menu the **BOM Composer** option opens up a new set of dialogs to control the BOM (Bill of Materials) creation with various levels of user control over formatting.

Using example templates, a built-in default setup, or building your own template from scratch, the whole process is quick and easy. What's more, you can immediately see the effect of each change you make to the template, right in front of you on the screen.

The BOM Composer dialog box is shown with the following settings:

- Sections:** ICS, Capacitors, Diodes, Connectors, Resistors, Transistors, Associated Parts, Others (all checked).
- Columns:** Line Item, Component, Names, Value, Part Number, Quantity, Cost (all checked).
- Group By:** Line Item (checked).
- Sort By:** Line Item (checked).
- Options:**
 - Generate from PCB (checked)
 - Generate from SCM/PRJ (unchecked)
 - Separate Top and Bottom (checked)
 - Separate each Variant (checked)
 - Include SCM/PCB-only (unchecked)
 - Write CSV (unchecked)
 - Collate by ref name (checked)
 - Collate using name-range (checked)
 - Sub-total sections (checked)
 - Include standard header (checked)
 - Extra header (unchecked)
 - Show Tooltips (checked)
 - Auto Save BOM (checked)
- Filter:** Comp Name: [], Pins: [], PCB Symbol: [], Apply, Clear
- Single Variant:** <Current Variant>

The main BOM table is as follows:

Line Item	Component	Names	Value	Part Number	Quantity	Cost
1	555	U3			1	0.00
2	TLC272	U1-2			2	0.00
Sub-total:					3	0.00
Capacitors						
Line Item	Component	Names	Value	Part Number	Quantity	Cost
3	C	C2	0.1nF		1	0.00
4	C	C4	47pF		1	0.00
5	C	C16	330pF		1	0.00
6	C	C5, C7, C10-11	33nF		4	0.00
7	C	C9	47nF		1	0.00
8	C	C1	0.1uF		1	0.00
9	C	C6, C12	100nF		2	0.00
10	C	C3, C8	680nF		2	0.00
11	CP	C13-15	100uF		3	0.00
Sub-total:					16	0.00

When the BOM Composer is first run, the dialog will look something like the picture shown above.

Along the top are some controls that are used to load, save and generate BOM templates and reports. Below this are some lists and buttons which allow you to control what appears in the BOM and how it is built and formatted.

Finally, the main body of the dialog shows you the BOM as it is generated based on the current settings. At the top right is a button that lets you 'fold up' the dialog to hide the lists and other editing buttons. Once you have got your BOM formatted how you wish, you can hide away the formatting controls to give you more space to view the actual BOM.

For more detail about configuring and running the BOM Composer, please refer to the On-Line help.

Chapter 9. Output for Manufacture

Overview

There are a full range of output capabilities supplied for printing, plotting and manufacturing; these are accessed from the **Output** menu:

- Gerber RS-274-D, RS-274-X and X2 format files
- Excellon compatible NC drill files
- PDF format files (using a built-in driver that is supplied for you)
- Windows printer devices
- HPGL compatible pen plotter files
- ODB++ format files
- IPC-2581 format files
- DXF files (AutoCAD compatible)
- IDF 2.0 and 3.0 format files (cost option export)
- GenCAD format files (cost option export)
- Pick & Place Coordinate lists (from **Reports** dialog)

Each of these outputs is described on the following pages. For printing and plotting outputs, the dialogs use a common interface. This interface is discussed first followed by specific information about the different formats and outputs required for manufacturing.

*Note: ODB++, IPC-2581, DXF, IDF and GenCAD are output from their respective menu options on the **Output** menu and **not** from the **Plotting and Printing** option.*

The Plotting and Printing Mechanism

For Gerber, Pen and Windows outputs the process is fundamentally the same with small variations to cater for the formatting specifics of the file type to be output.

The **Plotting and Printing** option on the **Output** menu encapsulates all the necessary plots into a single, re-usable plot 'job'. These plot jobs can be used to reliably recreate your plots, not just for one design but also across all designs using the same technology.

The plotting 'process'

For the initial plot session, you will need to create a plot job. After the initial session you can reuse plot jobs and edit existing jobs, although the jobs are very simple to create using the automated process of the option.

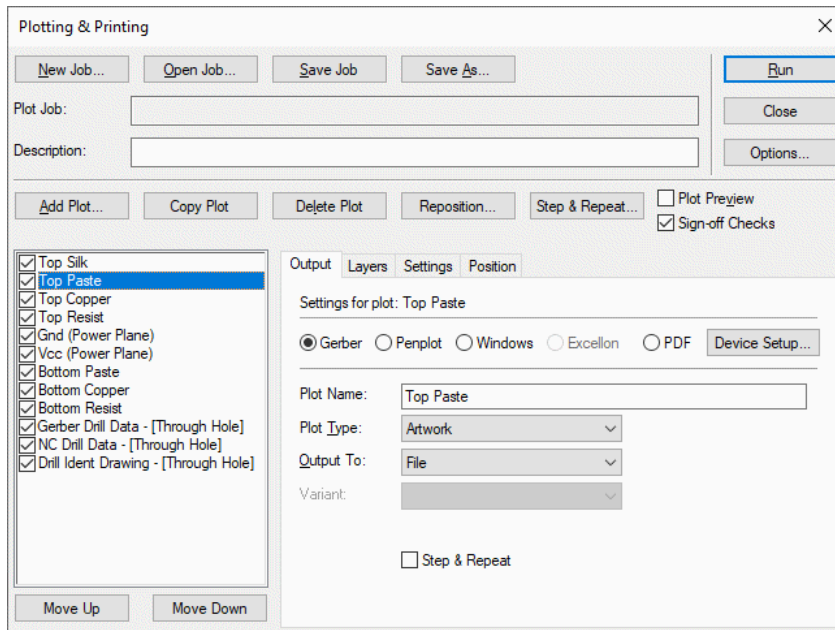
The flow of the process is to start or open a new job, choose the device type, auto-generate the plot job, select the plot(s) required, make adjustments to the plot and run the output to generate the plot set.

Creating a Plot Job

When you select **Plotting & Printing** from the **Output** menu, the main plotting dialog will open. This option can also be accessed from the toolbar using the **Plotting** button.

You will note that the whole plotting mechanism is the same for both Schematic and PCB design, except that for Schematics there is no layer information which makes the process much more simplified.

The dialog is divided into two main areas. The first deals with the current Plot Job file, and the second shows the contents of that Plot Job. Buttons in the top right allows you to run the plot job and to access various subsidiary dialogs to view or change device options.



Depending on the general plotting settings, you may find that some plots have already been generated for you. If not, the next step is to create the plots you need for your design.

A typical plot job

If we consider an example 4 layer PCB design with:

- two signal layers
- two power planes (for power and ground)
- through-hole components on top side

The following steps would be all that is required to create a plot job for our design:

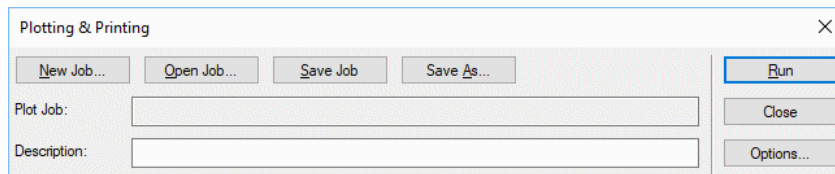
1. Select **New Job**, click on **Gerber** (or whichever format is required).
2. Under **Layer/Plot** types, uncheck **All**, and check all other plot types except **undersized**.
3. Under **Side**, check '**All**'.

4. Click **OK**.
5. The plot list is now completed with a plot for each layer and plot type requested.

Plot Jobs

Plotting works around the concept of a 'Plot Job'. Each job defines a list of plots, with each plot specifying the type of output device and what is to be included on the plot. Plot Jobs can be created, saved, edited and re-used, and are independent of any particular design file. If you are reasonably consistent about the technology settings you use for your designs, you should be able to create a small set of Plot Jobs that can be re-used time and time again to generate the plots for all your designs.

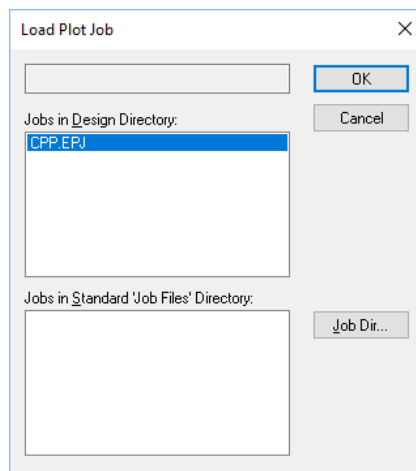
The row of buttons along the top of the **Plot Job** area allows you to manage the current plot job:



The functions of these buttons (from left to right) are:

- **New Job:** create a new job with a set of plots.
- **Open Job:** load an existing job file.
- **Save Job:** save the current job.
- **Save As:** save the job under another name.

When opening a Plot Job, the program looks in two places. The first is the Job Files directory, and the second is the directory where the current design is stored. The **Open Job** dialog lists the jobs it finds in these two locations, and displays the description (if any) for any job file you select.



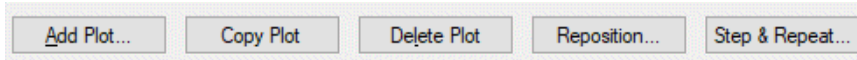
The **Job Dir...** button will allow you to choose the default Jobs folder.

Plots

Each plot in a job specifies the type of device it will output to, the layers to be included in the plot, and a range of other settings depending on the type of plot (such as pen numbers, inclusion of pin names and numbers, etc).

There are two lists in the lower (plots) area of the dialog. The left-hand list shows you the plots that are already defined for this job. The right-hand list shows the items (layers etc) that don't have plots defined for them in this job.

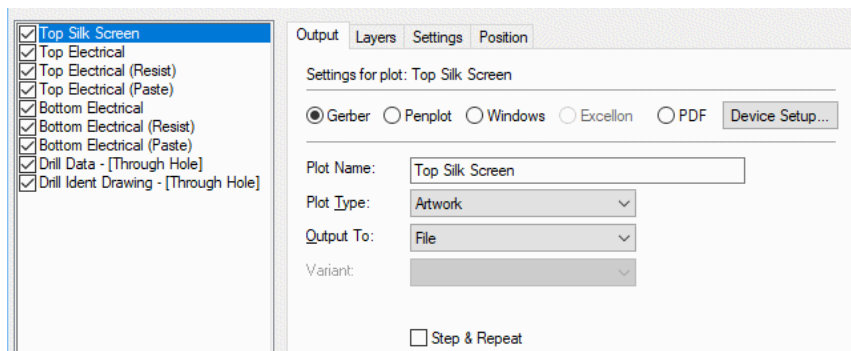
Plots are managed using the buttons provided:



- **Add Plot:** add a new plot to the job.
- **Copy Plot:** add a new plot by copying an existing one.
- **Delete Plot:** delete the selected plot.
- **Reposition:** reposition plots to re-align with selected plot.
- **Step & Repeat:** use this to create the rules used to export multiple plots.

Plotting & Printing Devices

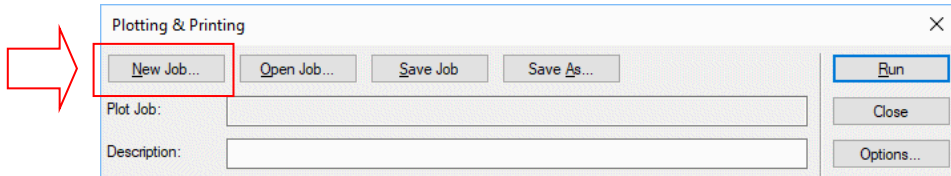
For each plot a device is selected. Initially, this might be the default device/layer combination choice made when generating the Plot Job. This can be edited using the selections on the tab dialog, this also includes the device type. Additional plots can be added to this Plot Job even if they are similar to other plots which already exist. For example, a full set of Gerber plots for final manufacturing can be copied and changed to also output to a Windows printing device for checking the plots before they are sent away.



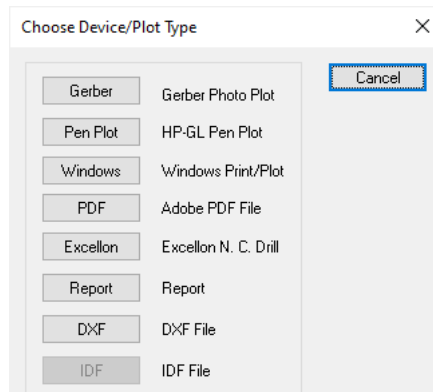
Each of the tabs is discussed in more detail later on in this section.

Auto-generating Plots

The best way of creating plots is to let the program do it for you! From the row of buttons at the top of the dialog, click on the **New Job** button.

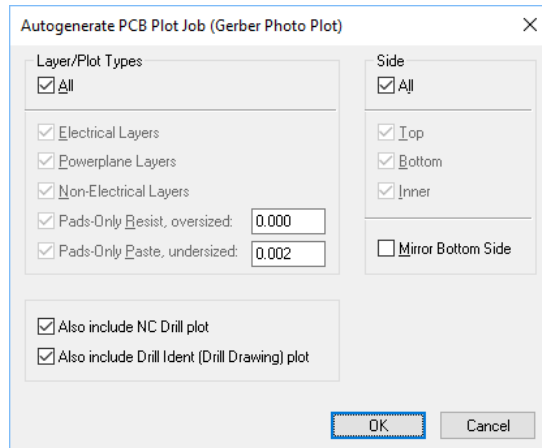


Auto-generation of plots for a design depends on the type of design and the type of the output device selected. On choosing to create a new set of plots, the first thing you need to do is decide on the output device type.



In a schematic design or a symbol, this is all you need to do. The single plot needed to print or plot your design is created and you are returned to the main Plotting dialog.

For a PCB design, selecting **Gerber** will also enable you to create an **Excellon** plot for the NC Drill data if you select the **NC Drill** the check box. You can also produce a **Drill Drawing** plot by selecting the **Drill Drawing plot** check box. For other output types in a PCB design, you should use one of the other options available for the plots it should cover.



There are two main selections, the **Layer/Plot Type** and the **Side** to plot.

Layer/Plot Type relates to the **Layer Type** specified in the **Layers** dialog on the **Settings** menu for **Electrical** and **Non-electrical** layers, as well as **Power plane** layers and special plots for pads-only.

Side allows you to choose the side required for plotting.

Powerplane Layers

For power plane layers, the program will automatically select the appropriate plot type for the plots it generates.

Additionally, if a power plane layer is found to contain actual design items (perhaps you have some tracks on the power plane), two plots are created, one to produce the negative oversized plot for the power plane itself, and one positive plot to be super-imposed to get the items 'embedded' on the power plane layer.

Pads-Only Plots

The Pads-only **Oversized** and **Undersized** plots are for layers which can contain pads and can therefore be plotted as pads-only. Pads-only plots are typically used to create solder mask (resist) and solder paste plots. The Auto Generate dialog allows you to choose one or both of these plot types which will be used for each side of the board, as well as allowing you to specify the amount by which these plots should over or under size the pads as they are plotted.

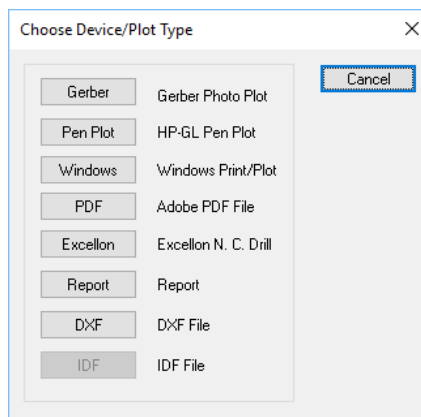
Values for the over and under size are defined in the boxes adjacent to the plot type and are defined in the current units.

Generating the Plot Job

When you have chosen what plots you want your plot job to contain, click the **OK** button to add the relevant plots to your plot job.

Adding Plots to a Job

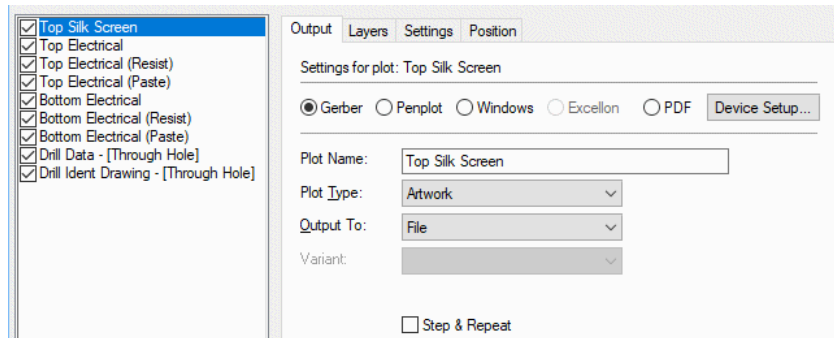
As well as auto-generating a whole set of plots, you can add individual plots by clicking on the **Add Plot** button. This will bring up the **Choose Device Type** dialog.



Once selected it opens up the relevant plot 'details' dialog to allow you to specify what to plot or output, as is the case of **Reports**, **DXF** or **IDF**. This dialog is described in the next section.

Editing Plots

The details of each plot are easily accessed. The plot details are displayed for the currently selected plot.

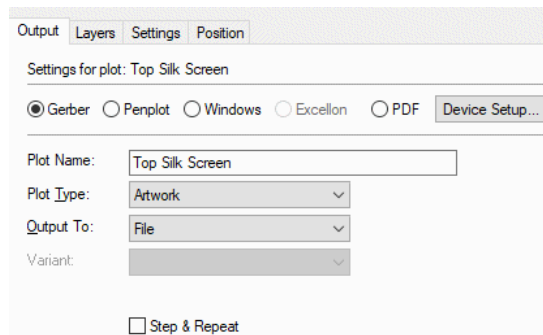


To edit an existing plot, click on the plot name in the list of plots. The plot detail has four ‘tabbed’ pages. The exact contents on some of the pages depend on the device type for this plot, but the basic operation is the same for all of them.

The pages described below are almost the same for each plotter or output type. Where differences appear, these are highlighted.

Output dialog

The **Output** page is used to specify the name of the plot, its type and the output mechanism.

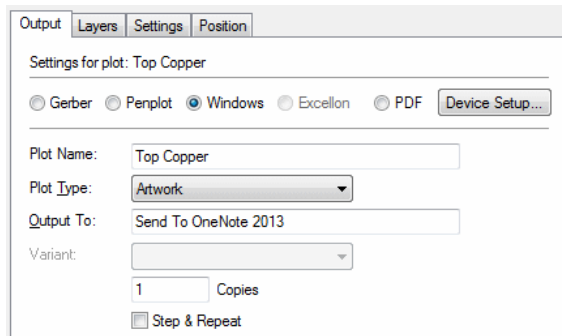


Plot Name is a name you type in. By default, the option will give you Plot 1 as the name but it is better to give the plot a more meaningful name.

Plot Type enables an artwork plot or an outline or check-plot to be specified for output. Mostly, **Artwork** will be used, but a quick check-plot may be clearer, and will use less ink, if **Outline** is chosen. For Excellon, the output is set to **Drill Data**.

Output To is used to show the direction of the output. For Pen Plotters there is a choice of File, Com1/2, File or LPT1/2/3. With the Windows printers this is set to your current default Windows printer. With the Windows printers, the name of the print device selected is shown but greyed out. This device can be changed using the **Device Setup** dialog. The destination will be one of the ports supported by the computer. When using serial (COM) ports, it is important that the communications settings used by the operating system correspond with those set for the plotter. A browser allows you to select a destination file if you have chosen File as the output. Gerber and Excellon plotters can only be directed to File.

With the **Windows** option you are able to specify the number of **Copies** to print at a time.

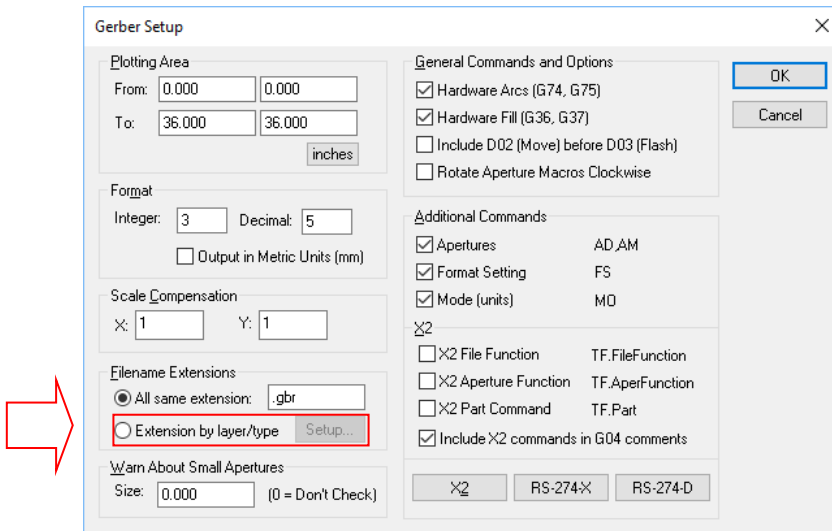


This saves time in setting up the dialog and in the load time to the printer. Once the initial setup to the printer has been completed and loaded into the printers' memory, additional copies are simply the time taken to put the print to the paper itself.

The **Step & Repeat** button allows you to panelise your design, in other words to add multiple copies of the same design onto a panel when it is plotted or printed. This is discussed below in more detail.

Gerber File Extensions

When setting up Gerber files, you can also customise the file extension for each plot. To do this, select the **Setup** button on the setup page.



This opens the **Gerber File Extensions** dialog from where you can customise the file extensions.

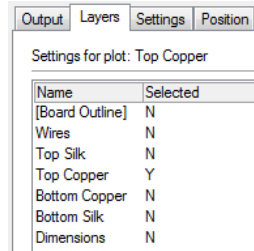
Step and Repeat Plotting

Included in the main **Plotting & Printing** option is the ability to **Step & Repeat** plots. This allows you to print multiple copies of the same plot, side-by-side on the paper or artwork without using an additional post-processing tool or process. Using this mechanism is an alternative to that of using the **Panel Editor** option.

The **Panel Editor** is a more comprehensive mechanism and is the preferred method of doing this. The Panel Editor not only allows you to panelise one design into many arrays, but also enables you to panelise different designs (provided they have the same layer count).

Layers dialog

The Layers dialog displays a list of layers in the design to select for plotting. The Excellon output doesn't use this table and is therefore not presented here.

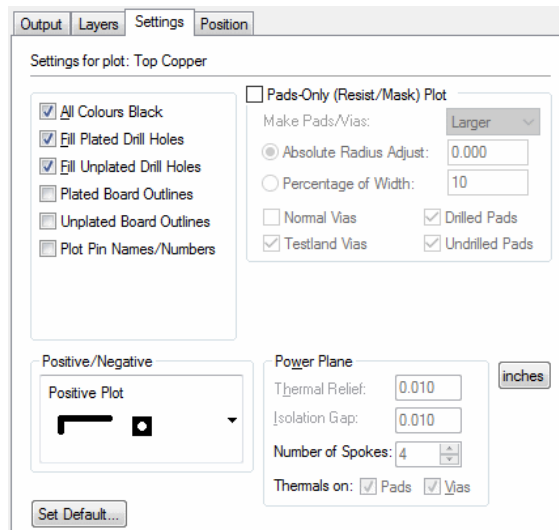


By selecting more than one layer you are effectively creating a combined plot. Remember that even though you might combine a number of layers, these are still only one single plot. To plot all the different layers shown on this dialog, you will need to create a number of different plots.

This gives control over the finished print. It is often useful for a check-plot to output the layers together in colour, but for prototyping (for instance) they would need to be separate. It also allows for 'composite' plots to be generated. That is, plots which require two sets of data which have been produced over two layer spans, or where two layers are required using perhaps two different pen colours to show both the top and bottom layers. By toggling the **Selected** box to **Y**, you will make that layer selection active and therefore output.

Settings dialog

The **Settings** page is the same for Gerber, Pen and Windows outputs. Excellon has no **Settings** dialog. This dialog is used to define the contents of the plot.



Fill Drill Holes enables an output to have any drilled hole to be solid. When prototyping. It can be useful to have all drill holes filled, as then a slight drill misalignment becomes unimportant.

All Colours Black is used for the Windows printers where all colours are printed to black regardless of their on-screen colour. Without this box checked, a coloured design on a colour printer will be printed in full colour, and a coloured design printed to a black and white printer will become greyscale. It is often a necessary function when generating printouts which will be used for prototyping to ensure that they are as black as possible to give the maximum effect.

Plot Board Outline is used where you wish to include the board outline in the plot. This isn't always required and so is available as a check box option. The additional check boxes allow you to specify which type of plate status you need to plot. This allows you to plot board cutouts that are plated to highlight these to your manufacturer.

Include Drilled Out is used if you have pad styles where the drill size is larger than the pad, this produces an effect called 'drilled out pads' where any copper land for the pad would be completely removed by the drilling process. If you want to suppress the pad land from these drilled out pads, then uncheck this check box.

Plot Pin Names/Numbers, again this is an option for the instances where these items may be required for checking purposes for example. Under normal circumstances these are not normally required.

If a **Pads Only** plot is selected, then a pads-only output will be generated for the layer. This activates the **Pads Only Plot** section in the top right hand corner of the dialog which allows you to make the pad selection and oversize selection for the output.

A solder resist mask consists of a pads only plot, with all of the pads being both filled and also oversize (or undersize) by the amount specified in the 'Oversize' box. For a solder resist mask it would be normal to oversize the pads to allow for any slippage over the actual size used. For a solder paste mask used on SMT pads it would be normal to undersize the pads to shrink the actual size used.

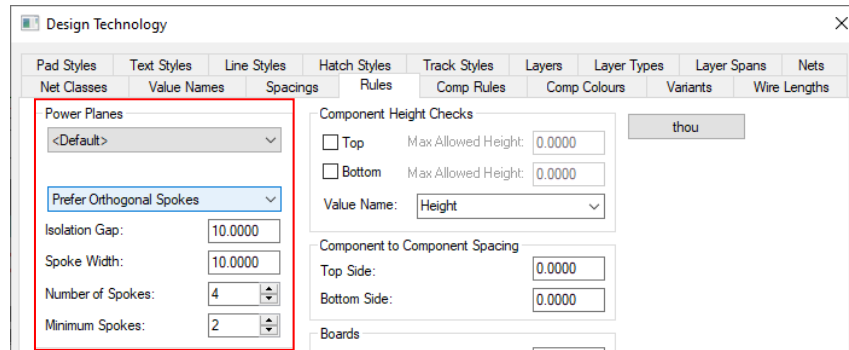
All Colours Black is used for the Windows printers where all colours are printed to black regardless of their on-screen colour. Without this box checked, a coloured design on a colour printer will be printed in full colour, and a coloured design printed to a black and white printer will become grey-scale. It is often a necessary function when generating printouts which will be used for prototyping to ensure that they are as black as possible to give the maximum effect.

Exclude Tented Vias is used to exclude any vias that are designated as Tented by the settings on their **Net Class** or **Via Properties**.

If plotting a power plane to a Windows printer, you may plot positive and negative plots together to give you a composite plot. This would be used if the power plane contained track items on it.

Power Plane

When generating power plane plots, **Thermal Relief** and **Isolation Gap** 'spokes' and isolation gaps are plotted for pads and vias on the appropriate power plane net. are used with power planes in PCB designs. The settings for these are taken from the **Design Technology** and **Rules** dialog.



These values are used for layers which are defined with a **Bias** of **Powerplane** in the **Design Technology** and **Layers** dialog and which are selected as 'Yes' in the selected entry at the top of the **Layers** dialog.




Thermal Relief and **Isolation Gap** can also be overridden on a per-area basis for **Copper Pour Areas**.

When a through hole is connected to a power plane, it is necessary to 'slot' the continuous copper sheet adjacent to any drill hole (otherwise too much heat is required for soldering). The program puts in four profiled slots (Isolation) for each such hole, positioned round the outside of the pad so as to leave bridges (thermal relief) on the major axes. It is the width of these bridges that is specified under 'Thermal Relief'. The isolation gap is similar, but in this case, it is the width both of the thermal relief slot and the extra clearance allowed round an isolated pad where it passes through the power plane.

The generation of power planes is based on selections shown as small pictorial representation to display the intended output.

Plot combinations are possible using the switches but also final composite plots to give the full plane required in the correct positive/negative film for use directly with Windows printers and photo-sensitive films.

When the **Positive/Negative** list box is selected, four types of plot outputs are displayed:

- Negative (only plot gaps)**

 This is used to produce a plot of all the oversize in the plane, mainly for the Gerber output, to be reversed and combined with a Positive (only plot items) plot.
- Negative (plot isolation)**

 This is used to create the full power plane with isolation gaps, thermal relief and embedded tracks (with isolation) but in negative mode. To create the power plane from this plot, the image would need to be photographically reversed. Producing this type of plot saves printer toner and can be photographically reversed to get the plot image required. This option is not available for Pen plot outputs.
- Positive (only plot items)**

 This is used if the power plane contains 'embedded' tracks or items. This plot is the item itself without any isolation or thermal relief. This would be used in conjunction with the Negative (only plot gaps) plot to produce a power plane with embedded items for a Gerber plot.

Positive (plane & items)



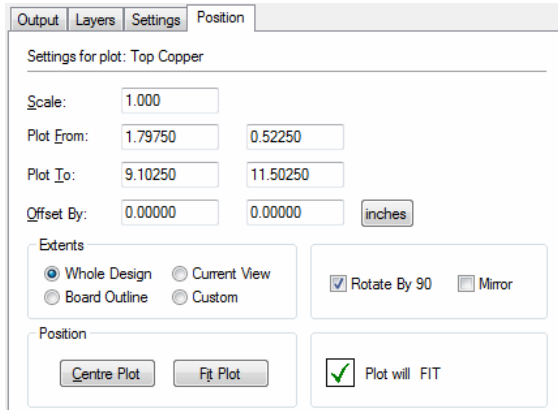
Positive (plane & items)

This is the full power plane output with isolation gaps and embedded tracks with isolation clearances and thermal relief where connected to the plane. This is used to create the full power plane with everything plotted as it would be when manufactured

This option is not available for Pen plot outputs.

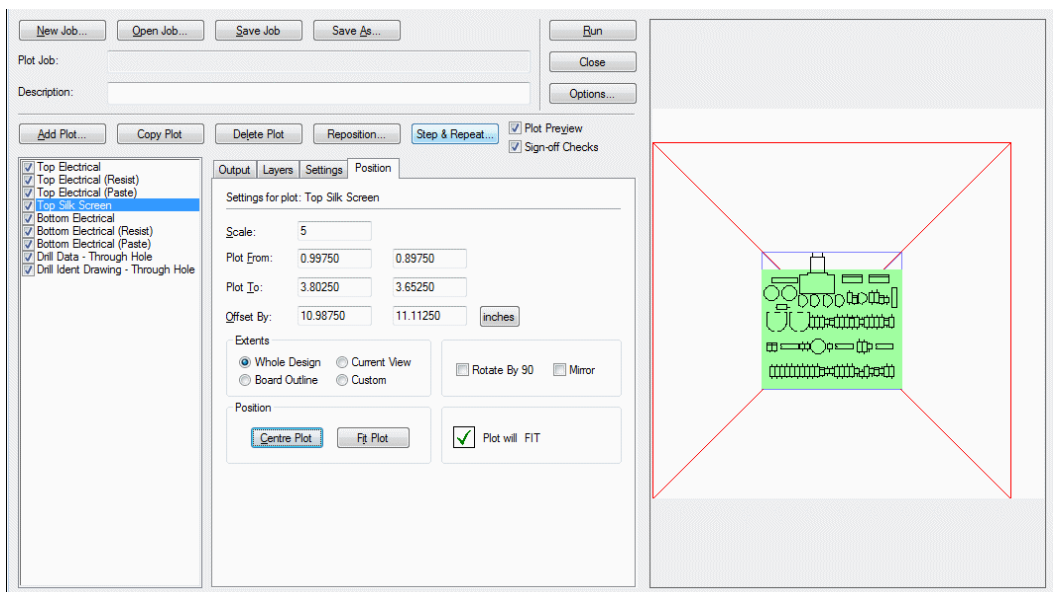
Position dialog

The **Position** dialog is used to set the position and scale of the plot on the plot bed.



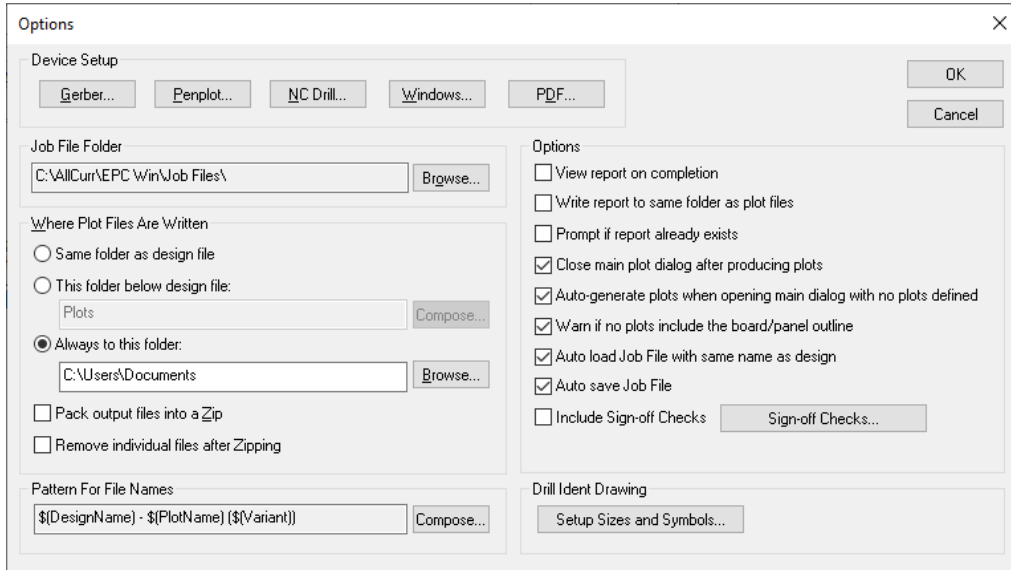
The best view of this dialog is when using the **Preview** option. Check the **Preview** check box to reveal an additional preview pane. Using any of the options on the dialog for scaling and positioning the plot will change and update the **Preview** view.

When viewed, the red outer box shows you the plot bed or paper size. The blue box is the full design extents; that is everything in the design, including the drawing border. The green box is the board extents.

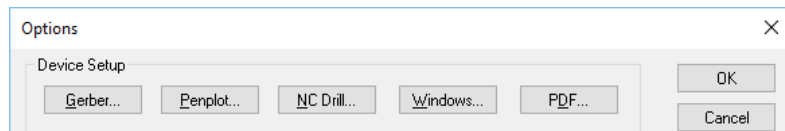


The Options Dialog

The **Options** dialog is used to define the **Device Setup**, the default Job folder and where the plots are written. To open this dialog, click on the **Options** button on the main **Plotting** dialog.

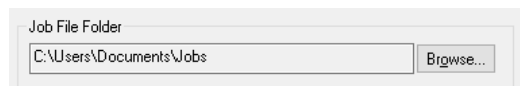


The buttons along the top of this dialog allow you to access the device-specific setup dialog for each type of output device. These settings are independent of any particular plot job, and are generally stored in the system registry (and hence 'persist' between design sessions).

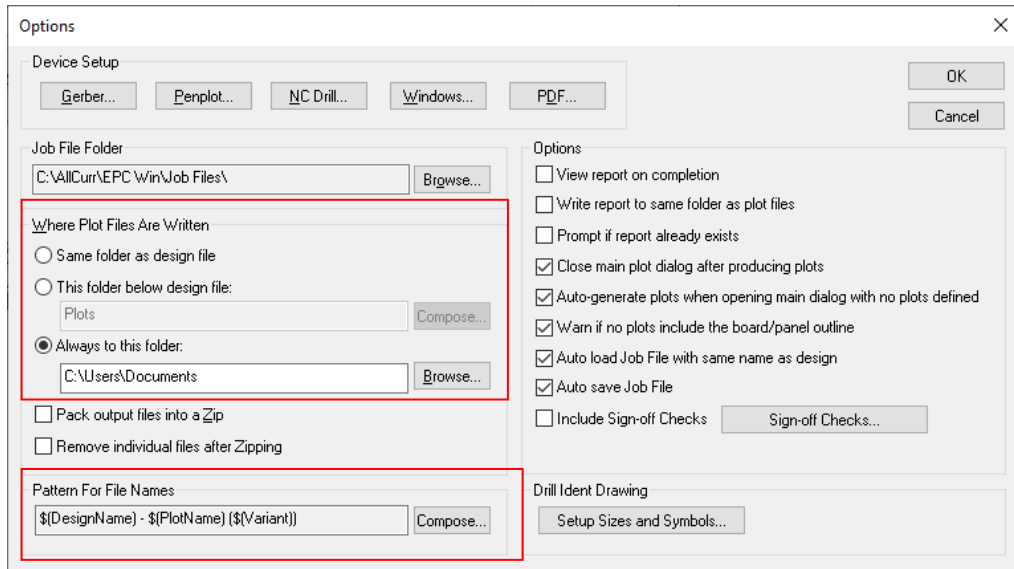


Each plot has a name, which is used to give the resulting plot output file a name (for those plots which are actually output to a file).

The **Job Files** area of the dialog allows you to view (or change, using the **Browse** button) the directory where the 'master' job files are stored.



Plot files are written out to the location specified in the next area of the **Options** dialog. This should be set up to fit in with the way you want to organise your files.



Customisation of Plotting File and Folder Names

User-defined formatting of **Folder** and **File** names in the **Options** dialog of **Plotting and Printing** is available. This uses a ‘tag’ scheme with user-defined key words to format the output filename. This applies to the folder name when the folder is set to ‘this folder below design’, and to file names for each plot.

Tags can include Design Name, Plot, Plot Job and Project Name, Device Type, etc. To help you construct the tag pattern, syntax and creation, there is a new **Compose** dialog.

Where patterns are used

These patterns are used to construct the folder name for storing plots and other files generated during **Plotting and Printing**, and also to construct the actual names for these files. This allows you to apply some quite sophisticated rules to name and organise your output files.

When composing the pattern for the output folder, the default value is simply the text "Plots" which will place the output files in the Plots folder below the design file.

When composing the pattern for actual output files, the default value combines the design name, the plot name, and (if variants are used) the variant name.

By selecting the **Compose** button, this reveals the **Compose File Name Pattern** dialog. From here, you can make choices that will customise your output filenames.

The screenshot shows a dialog box titled "Compose File Name Pattern". At the top, there is a text input field for "File Name Pattern" containing the text "\$ (DesignName) - \$ (PlotName) \$ (Variant)". To its right are "OK" and "Cancel" buttons. Below this is an "Example Name" field containing "Design - Plot (Variant)". The main area is divided into two sections: "Insert Elements" and "Edit Elements". The "Insert Elements" section contains several buttons: "Job Name", "Plot Name", "Plot Device", "Design Name", "Variant Name", "Date", "Project Name", "Design Type", "Time", and a dropdown menu for "Project Author" with a "Value" button. The "Edit Elements" section contains buttons for "Select Element", "Delete Selected", "Revert to Default", and "Undo Changes".

The **File Name Pattern** box on the dialog shows you the current pattern. These patterns are made up of 'tags' or 'fields' that are replaced at runtime with the relevant values, as well as any text required to complete the name.

Example Name: will display a preview that will show you what the final composed name might look like when the current pattern is applied. This will help you to quickly see the results of manipulating the pattern by altering the text or adding or removing elements.

Click on any of the buttons in the **Insert Elements** area to add the relevant item to the pattern at the current location of the text cursor.

When you click any function button that is not a text character, you will see the relevant tag added to the pattern. The 'character' buttons are there for convenience to simply save you typing the character yourself. They do not form part of the element but are used for filename formatting.

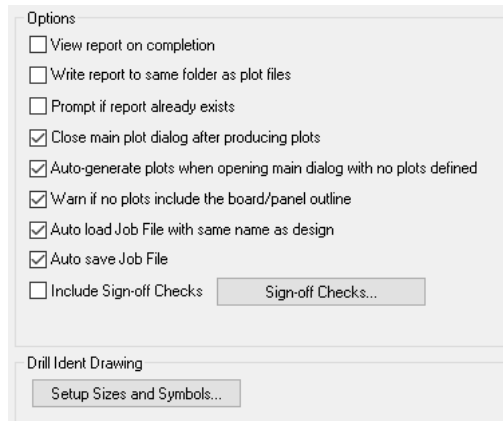
As well as design elements, you can also choose to add a **Value** to the pattern. The drop down list will provide you with all the value names defined in your design. Not only does this give you system values but also component values should you so wish to use them.

To remove elements from the pattern, click within the tag on the pattern box, then click the **Select Element** button. The text for that tag will become selected, and you can then either click one of the **Insert** buttons to replace that tag with the one you really want, or click **Delete Selected** to remove it from the pattern.

You can also edit the pattern by hand, to insert, alter or remove plain text items.

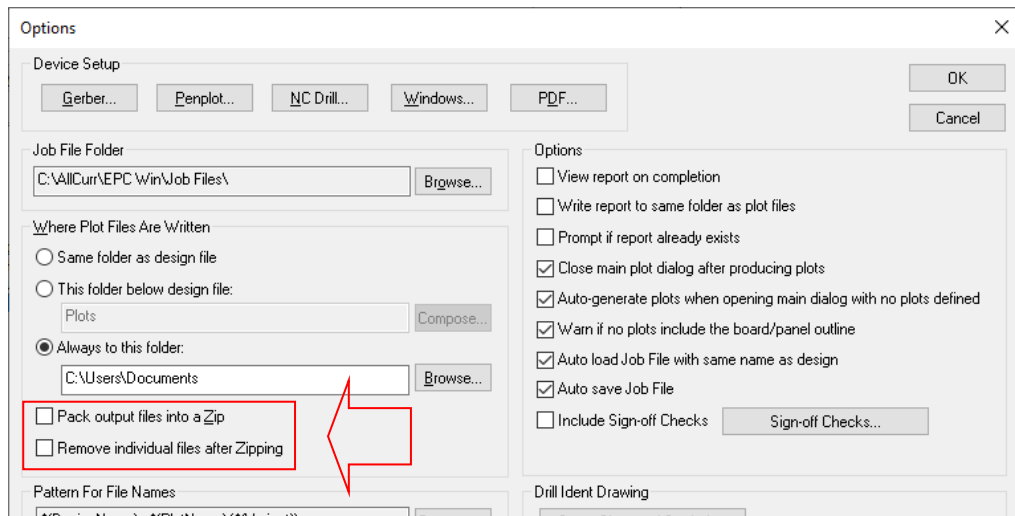
The Options Section

The **Options** section contains settings that you can use to control the way the plotting works.



Output Plots to Zip file

There are two check boxes on the **Options** dialog of the Plotting dialog. This allows you to automatically save the generated output files (Gerber, NC Drill, Reports, etc.) directly into a **Zip** file, and optionally **remove** the individual files afterwards.



Pack output files into a Zip – output files are exported as normal, then saved into a Zip file. Without the Remove check (see below) selected, all the files will remain saved on your hard drive along with the Zip file also. The Zip file will be named *DesignName -.zip* following your file name pattern for plots.

Remove individual files after Zipping – if the check box to Zip the output files has been selected, selecting this option will then remove them from your hard drive. All plots will be removed and just the Zip file (containing the plots and reports) plus the Plot Report will remain.

Pre-Plotting Checks

Before opening the main **Plotting** dialog, the program will check your design. If it is a PCB design with components still in the **Component Bin**, a warning message may be displayed.

This gives you the option of continuing with plotting, but it does at least warn you about potential problems before you produce plots that may be incomplete.

Drill Drawings

A drill drawing can be output, this is used to inform the PCB manufacturer of which holes to drill at specific sizes. This is important when drill plated and non-plated through holes. .

Use the **Drill ID Setup** dialog accessed from the **Plotting Options** dialog, from the **Settings** tab for a selected **Drill Drawing plot** and from the **Settings** menu and **Drill Ident** to define the drill hole ID/Shape relationship. This information is used for the **Drill Symbol table** used in the design and for **Drill Ident Drawings** used for manufacturing.

The screenshot shows the 'Drill Ident Setup' dialog box. It has three main sections:

- Drill Ident Text:** 'Use Text Style:' is set to 'Drill Id'. 'Offset Text:' has X: 50.00 and Y: 0.00. There are 'OK' and 'Cancel' buttons.
- Generating Table Entries For Drill Idents Not Already Defined:** 'Symbol Shape:' is set to 'Round'. 'Symbol Size:' is 100.00. 'Start Text From:' is 'G'.
- Drill Ident Table:** A table with columns: Drill, Plated, ID, Shape, Size.

Drill	Plated	ID	Shape	Size
32.00	Yes	A	Round	100.00
35.00	Yes	B	Round	100.00
40.00	Yes	C	Round	100.00
37.00	Yes	D	Round	100.00
100.00	Yes	E	Round	100.00
45.00	Yes	F	Round	100.00
25.00	Yes	G	Round	100.00

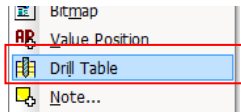
 Below the table are buttons: 'Add...', 'Edit...', 'Delete', 'Open...', 'Save As...'. At the bottom, it says 'File: DrillIdent.did'.

When setting up the Plotting dialog for drill drawings, select the **Plot Type** as **Drill Ident Drawing** and output to the required format.

The screenshot shows the 'Plotting' dialog box with the 'Output' tab selected. The 'Settings for plot: Drill Ident Drawing - Through Hole' section is visible. The 'Plot Type' is set to 'Drill Ident Drawing' and 'Output To' is set to 'File'. There are radio buttons for 'Gerber', 'Penplot', 'Windows', 'Excellon', and 'PDF', with 'Gerber' selected. A 'Device Setup...' button is also present.

Using Drill Tables in the Design

You can a drill table of drill sizes & counts in the design itself by selecting the **Drill Table** option from the **Add** menu. .



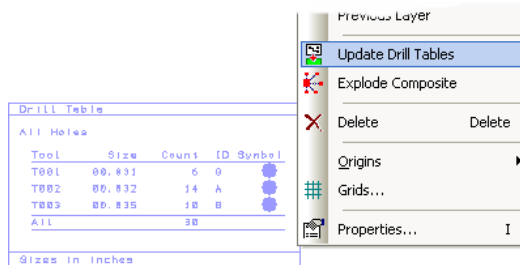
The drill table will list all drills required, separating plated and unplated if this is enabled in the NC drill options dialog, with a separate section for each layer span. Each entry in the table lists tool number, size, count, ID (letter) and the drill-drawing symbol. Boxes and lines are used to divide the table into sections.

Drill Table					
All Holes					
Tool	Size	Count	ID	Symbol	
T001	00.020	400	AD	●	
T002	00.024	324	O	●	
T003	00.028	4	K	●	
T004	00.030	55	A	■	
T005	00.031	42	B	◆	
T006	00.032	98	E	.	
T007	00.035	26	D	.	
T008	00.035	1	AC	●	
T009	00.035	18	I	●	
T010	00.036	10	N	●	
T011	00.039	46	C	×	
T012	00.040	3	L	●	
T013	00.047	58	F	●	
T014	00.059	24	AE	●	
T015	00.062	2	M	●	
T016	00.100	12	J	●	
All		1123			

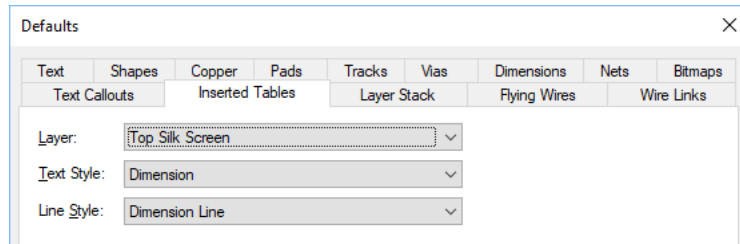
Sizes in inches
Machine units are Thou

When using the blind and buried via technology, you can specify a specific layer span for the inserted drill table, or set it to show all spans.

If a drill output is run, or if the **Update Drill Tables** option is used (from the context menu), or if **Properties** is used on it the table will change to the new style with lines.

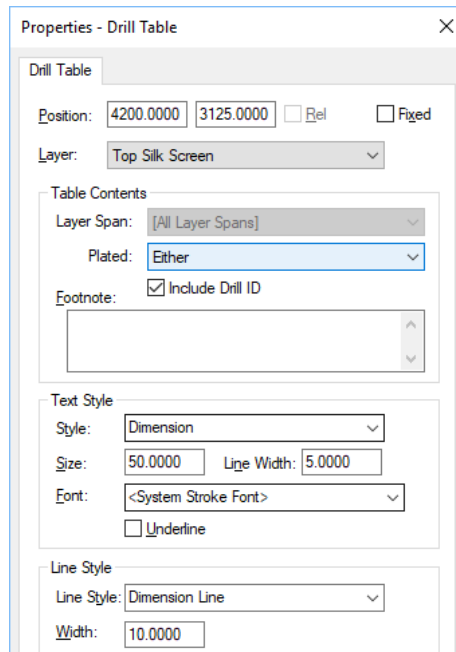


The **Add Drill Table** option on the **Add** menu in PCB generates the table for the current drills, and allows you to place it. The defaults for the text style, line style and layer are defined in the **Defaults** dialog under the **Inserted Tables** page.



You can define the symbols used by editing the **Drill ID Setup** option on the **Settings** menu.

Once added, you can use **Properties** to change the layer, text style and line style, and add an optional 'footnote' field that will be added at the end of the table (could be used for example to add manufacturing notes like tolerances etc.).



By default, the drill table text, lines and pads are drawn using the normal text, shape and pad colour, but you can also highlight all drill tables in a single supplied colour using the colours dialog.

Power Planes

The generation of power planes is provided by an automatic mechanism. It is of course true by definition that a power plane is dedicated to one signal but it could also be a split plane where multiple signals are contained on the same 'plane' layer. Within your Layers dialog in the Design Technology, you can name plane layers and make the name association. Inclusion of a net name in a layer definition automatically defines that layer as a power plane regardless of other settings.

	Name	Type	Side	Usage	Bias	Net	Colour
X	Top Silk	Silk Screen	Top	Non-Electrical	None		Yellow
	Dimensions	Dimensions	Top	Non-Electrical	None		Green
	Top Paste	Paste Mask	Top	Non-Electrical	None		Teal
X	Top Copper	Electrical	Top	Electrical	X		Red
	Top Resist	Solder Mask	Top	Non-Electrical	None		Orange
	Layer 2 Copper	Electrical	Inner	Electrical	Y		Red
X	Gnd	Electrical	Inner	Electrical	Power Plane	GND	Green
	Vcc	Electrical	Inner	Electrical	Power Plane	+5V	Teal
	Layer 5 Copper	Electrical	Inner	Electrical	X		Grey
X	Bottom Copper	Electrical	Bottom	Electrical	Y		Cyan
	Bottom Paste	Paste Mask	Bottom	Non-Electrical	None		Olive
X	Bottom Silk	Silk Screen	Bottom	Non-Electrical	None		Blue
	Bottom Resist	Solder Mask	Bottom	Non-Electrical	None		Brown

There are a number of considerations when using power planes. Tracks from other non-related signals can be used on a power plane layer, and will generate a clearance round them, BUT such clearances can reduce or defeat the effectiveness of a power plane as a low impedance high frequency path, so they should be kept to a minimum. You can also add Text to a power plane layer for documentation purposes perhaps.

Power planes are not shown during editing unless the **View Power Plane** feature has been used. Rather they are generated during the plotting process.

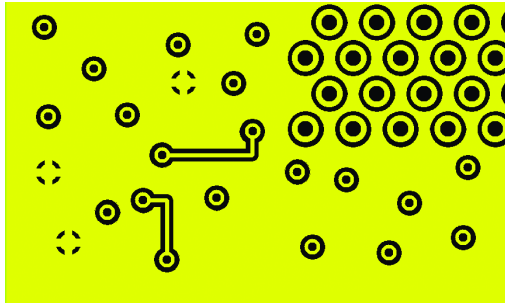
Normal Power Planes

A 'normal' copper power plane (a plane without tracks or text) only requires one file to create all the necessary thermal relief and isolation gaps. This would require the **Negative (plot only gaps)** selection to be made within the **Power Plane** section of the **Settings** page on the **Plotting** dialog. This would then be processed by the plot house and reversed to produce a 'gaps' only positive plot.

The negative plot is initially generated as you then only plot the items which will not exist in the final plot, this generates a much smaller plot. In most situations up to 10-20 times smaller! The file sizes are not a problem in themselves but their transportation on regular media or electronic mechanisms may cause a problem, especially if the file is upwards of 10Mb in size.

Power Planes with embedded items

If the plane includes embedded tracks or text, then the rules change and a different mechanism is required to process the plot.



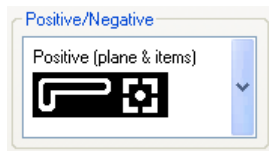
There are two mechanisms available depending on how you want to work or your level of competence.

1. Create a 'composite plot of the final plane with the tracks and text already embedded in the plane. This would show all the gaps for the plane isolation of connected nets and gaps for tracks and text.
2. Create two files; a negative plot of the plane with the thermal relief, isolation gaps and oversize for tracks plus spacings, and a positive plot of the copper items, the tracks etc. These two files are then composite together in a Gerber viewer/editor and the composite plotted as a final plot. Usually, this is done by the manufacturer.

Mechanism 1. Creating a composite plane

This is the easier of the two methods and is more complete if you have embedded tracks or text on your plane layers. However, the file sizes are much larger because the files contain a lot more information.

You would use the **Positive (plane & items)** selection from within the **Plotting** dialog and **Settings** page.

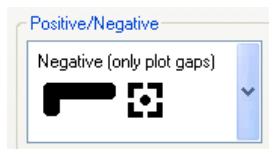


Mechanism 2. Outputting two files to create a composite plane from

Two files are required for one composite power plane with embedded items.

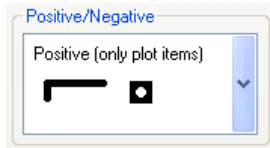
The first plot is a negative of the required copper, and defines the holes in a continuous copper plane (thermal relief, isolation gaps and oversize for tracks plus spacings).

You would use the **Negative (only plot gaps)** selection from within the **Plotting** dialog and **Settings** page.



The second is a positive, which defines all copper items (e.g. tracks) appearing inside the holes defined by the first file.

You would use the **Positive (only plot items)** selection from within the Plotting dialog and Settings page.



By reversing the sense of the first, and adding it to the second, the required composite power plane is generated complete with embedded items. This process would normally be completed by the plot house using the plotters own 'front-end' software to create the require effect. If can be done using commercially Gerber preview/editing software but you should really understand what you are doing before attempting to create this composite plane.

A word about settings, the clearance specified as “Isolation Gap” not only describes the separation between pads and power plane, but also specifies the width of the thermal relief profile round connected pads. The "Thermal Relief" figure specifies the width of the four spokes connecting appropriate pads to the power plane. Clearly it will normally be somewhat larger than the isolation gap.

When plotting power planes, ensure that there are no spurious tracks left on the plane layer. Run the **Dangling Tracks** option from the **DRC** option to check this.

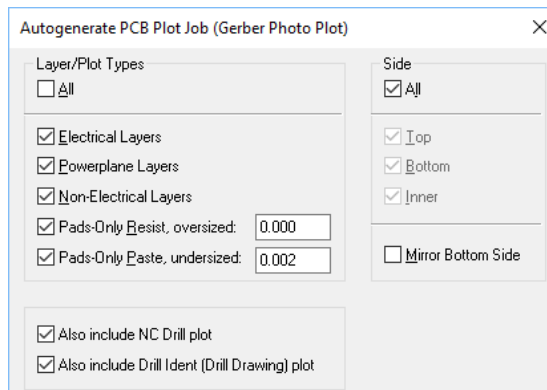
Plotting Solder Resist Layers

Solder resist plots are generally a pads only plot which is oversized by a small amount, usually 5 thou or thereabouts. This can easily be done using one of two methods.

Plotting the Solder Resist layer

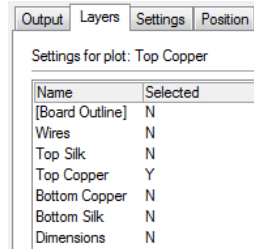
The first method is done using the **Output** menu and the **Plotting and Printing** option. You should generate a Pads only plot of the Top (and Bottom, if required) electrical layers.

When creating a plot, you must first select the **Oversize Pads-Only** plot entry using the **New Job** button (then select one of the: Gerber, Pen, Windows buttons).

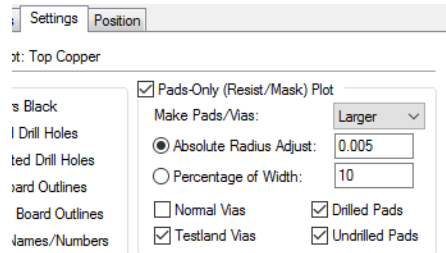


This can be selected as part of a new plot job or added to the plot selection for an existing job. Once selected, this will allow you to specify the oversize required by typing in the value.

When the **Plot Type** is selected, the layer required for the solder mask will need to be selected. Change one of the layers to be selected (usually the Top Electrical). This will be one of the electrical layers (Top or Bottom Electrical) which contain pads, or a specifically named Solder Resist layer if one has been created.



Because a layer has been selected which contains pads, the **Pads-Only (Resist/Mask) Plot** option is enabled on the **Settings** dialog.

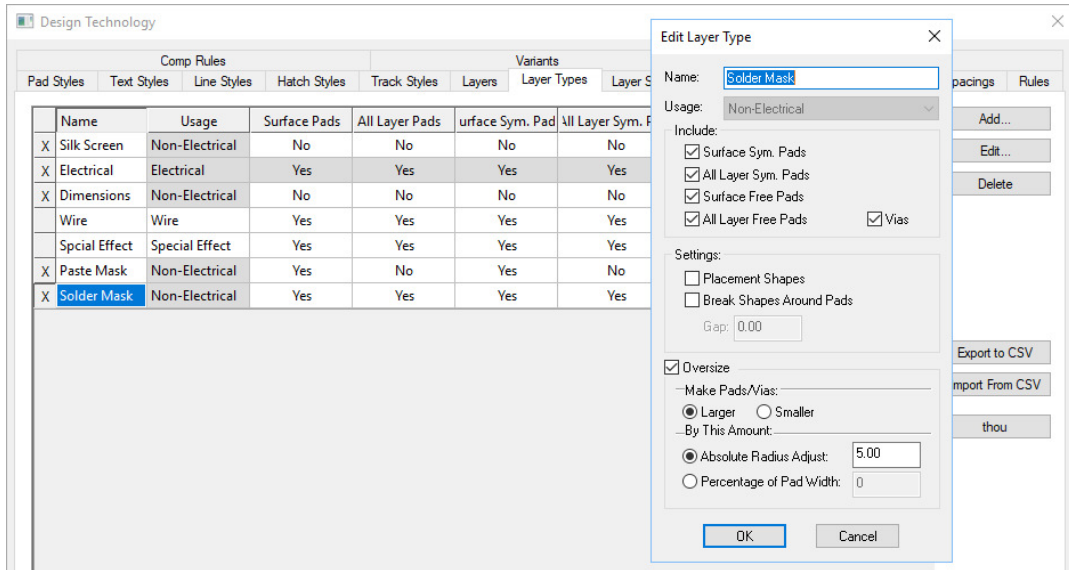


For a solder Resist plot, the size option **Make Pads/Vias** is normally set to **Larger** and the value specified. For a solder Paste mask this would be set to smaller and the value to show and undersize of the plot. This can be defined as an **Absolute** value or a **Percentage**.

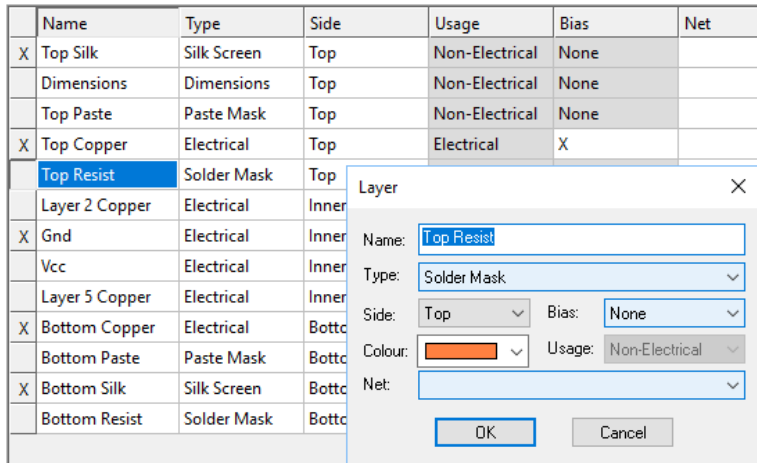
You should select the check boxes for the plot type required.

Creating a new Solder Resist layer

The second method is to create a new layer called **Solder Mask** using the **Layers** option from the **Design Technology** dialog and to setup this new layer using the **Layer Type** option.

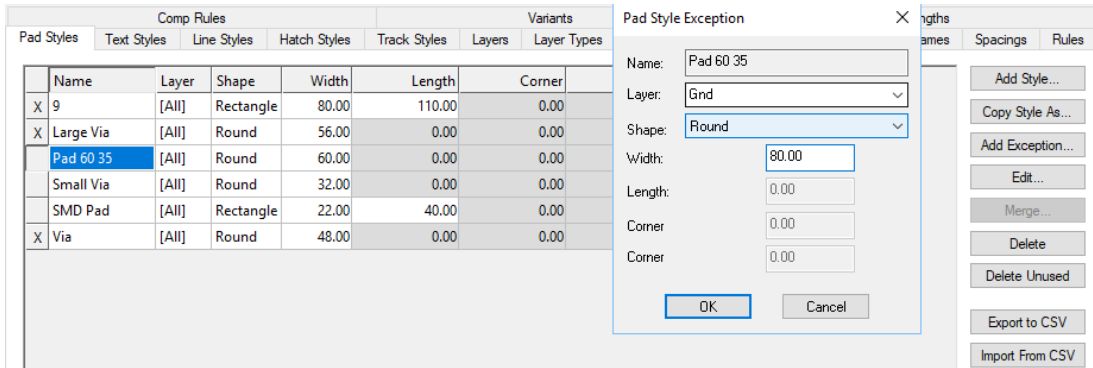


By setting up the layer type to have all pads except **Vias** available for plotting or display, these will be used for the solder resist. The **Layer** must then be created to have this new **Layer Type**.



By using this method, it enables the pad sizes to be controlled specifically on a one to one basis using Pad exceptions.

From the **Design Technology** dialog, select the **Pad Styles** page.

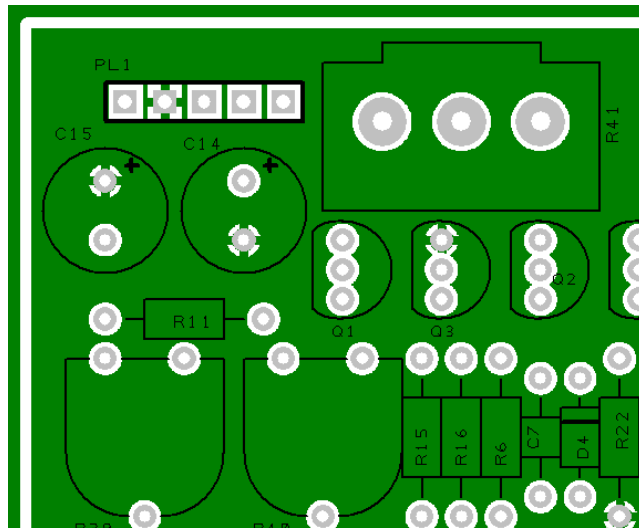


Select each of the pad styles in turn which will appear as a different size to that of the default pad stack. Select the **Add Exception** option and select the **Layer** to change (Ground) and the new **Width** and **Length** to be used.

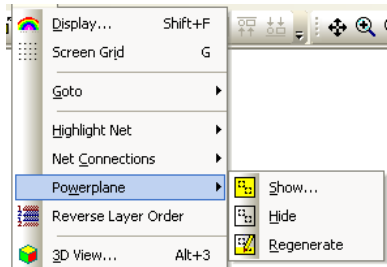
When a Solder Resist layer is now processed in the **Printing and Plotting** option you do not need to oversize the pads.

Viewing Power planes

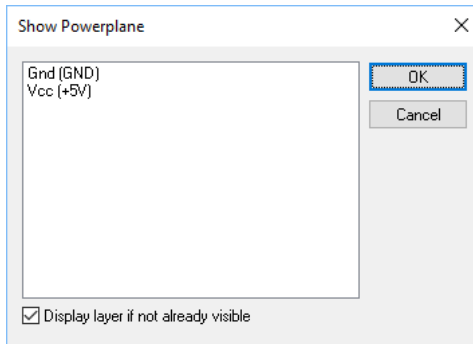
This is used as a visual on-screen representation of the final power plane for full power plane layers. This is generated using the same output mechanism used to generate the manufacturing plots rather than a simple 'bitmap' image.



This option is available from the **View** menu, there is a menu option **Powerplane** with three sub-menu options – **Show**, **Hide** and **Regenerate**.



Show displays a dialog allowing the selection of which power plane to view. If only one power plane is present in the design, it will not display this dialog but go straight into the display of the power plane.



Hide turns off the visible power plane mode and resumes the ‘normal’ view mode.

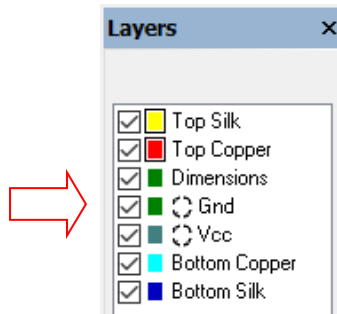
Regenerate, this recalculates the visible power plane items and redraws the picture.

The full power plane displayed using this option cannot be plotted directly without using the plotting modes on the **Output** menu.

View Powerplanes using the Layers Bar

As well as using the **View Powerplanes** option, you can also access the view of power planes using the dockable **Layers Bar**.

Powerplane layers are indicated by a small thermal pad icon to the right of the coloured square.



If the power plane is being displayed in the design the icon is in negative. Left click on the icon to toggle the display to show/hide the power plane.



Note, this option is a quick way to access the **View Powerplane** functionality. **View Powerplane** is still available from the **View** menu.

Tented Vias

What is a Tented Via and why/when should it be used?

Tenting a via refers to covering via with solder mask to enclose or skin over the opening. A via is a hole drilled into the PCB that allows multiple layers on the PCB to be connected to each other. A non-tented via is just a via that is not covered with the solder mask layer.

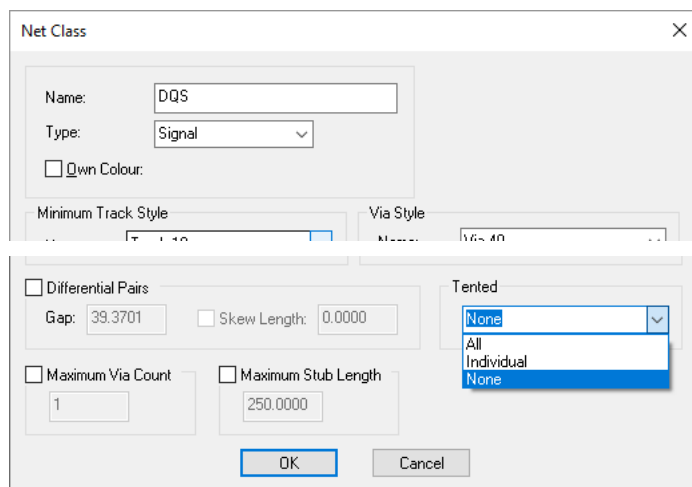
The role of the PCB solder mask is to protect the via (and copper traces) from damage, oxidation, and solder bridging. Tented vias will be more resistant to physical damage and electrical shorts by protecting the plating inside the via hole.

Whether or not you should use tented vias is down to your preference and that of your PCB manufacturer. If you contact them, they can advise you.

Defining Tented Vias

This is a simple method of specifying whether or not vias are to be 'tented'. A tented via is one that is covered with solder resist during the manufacturing stage. In order for this to happen, the via that is to be tented must be identified in the design and a specific output generated.

The **'tented'** property is defined in the **Net Class**. When vias in the design use this Net Class, the output file generated in the **Plotting and Printing** dialog can use this flag as a selection option.



Each Net Class has a property of **Tented** which can be set to one of three values:

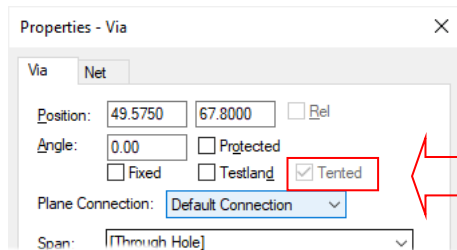
None: vias on this Net Class are automatically not tented so they will appear on plots as normal.

All: all vias on this Net Class are automatically classified as tented.

Individual: the Tented flag is set by you as required. This is set on **Via Properties** for each via

Via Properties

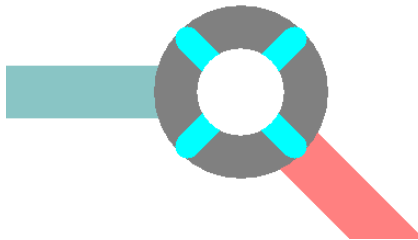
When **Tented** is set to **None** or **All** on the Net Class, the tented state of a via is displayed on the **Via Properties** dialog but the check box is greyed out because it is inheriting its setting from the Net Class.



When the **Tented** property is set to **Individual**, this check box then becomes available for setting.

Tented Vias in the Design

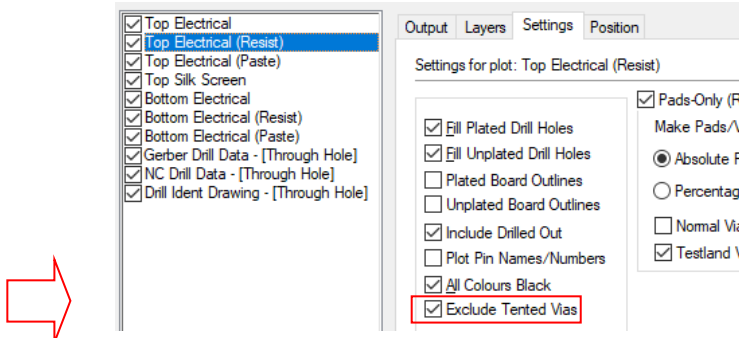
If this highlight is enabled (checked) in **Colours**, any via that is tented (either individually on **Properties** or from its **Net Class**) will be marked with an X (cross) in the same way as we do to highlight unconnected pads:



Note, this is the only visual indication of the ‘presence’ of the via shape on a resist layer, the rest of the processing is done at the plotting stage. In other words, if you display only your top resist layer, vias that are marked as tented will still be drawn.

Plotting Tented Vias

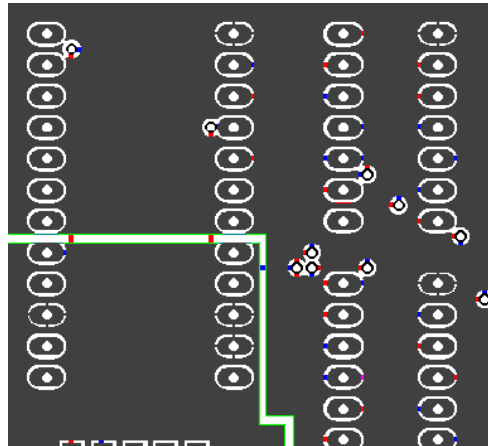
In **Plotting and Printing**, there is a new check box on the **Settings** tab for each plot that allows you to decide whether or not to exclude tented vias from that plot.



The default setting for this is unchecked (plots include tented vias) but when auto-generating plots it will try to determine which plots are **Resist** and will thus tick this box for you. This setting is also saved in **Plot Jobs** so it is easy to prepare a preferred set of plots and settings and save it for future use.

Split Power planes

The creation of split power planes has been covered previously in the section *Copper Pouring*. Although the split planes would appear on what would normally be full power planes they are output somewhat different to conventional power planes. They are output as a 'normal' plot layer but because the net association has been removed from the plane layer to allow for the split plane, they are no longer doesn't recognised as a power plane layer.



To output split power planes on internal layers, ensure that the net association in the **Layers** dialog for this layer has been removed to show **[None]**.

	Name	Type	Side	Usage	Bias	Net
X	Top Silk	Silk Screen	Top	Non-Electrical	None	
X	Top Electrical	Electrical	Top	Electrical	X	
	Wires	Wire	Top	Wire	None	
	VCC	Electrical	Inner	Electrical	Power Plane	VCC
	GND	Electrical	Inner	Electrical	Power Plane	[None]
X	Bottom Electrical	Electrical	Bottom	Electrical	Y	



This internal layer is then output by selecting the layer name in the Output dialog and treating it as a 'normal' positive layer (whereas power planes are normally output as negative layers to save plotter file sizes).

Some checks to make before plotting split power planes.

- Check that the net name association has been removed from the Layers dialog for this internal plane layer.
- Check that no components have been moved after pouring the plane(s). Check by re-pouring if necessary.
- Check that thermal relief and isolation gaps have been added to nets which need to be connected to the planes. A visual check in the design using the Highlight Net feature is a good start.
- Check that the splits are in the correct position.
- Check that the Copper to board spacing is a sensible distance and not too close e.g. approx. 75 thou rather than 8 thou (but use your own values based in manufacturing requirements).

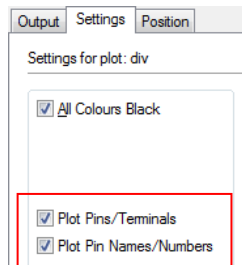
Schematics Print/Plot Outputs

Not strictly a PCB manufacturing output but plotting or printing the Schematic is a vital part of the whole process and completion of the design.

Windows and Pen Plotting

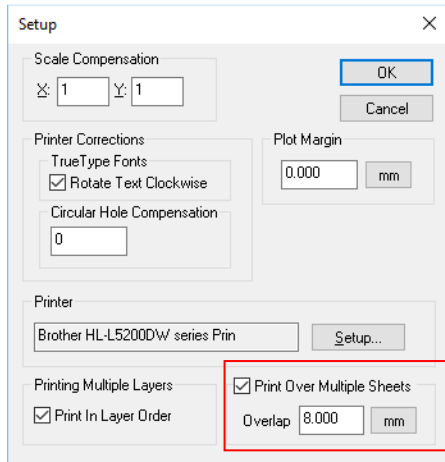
The Windows and pen plotting dialogs are much simplified to that used in the PCB design editor. Obviously, within Schematic there are no Layers to plot or power planes to consider. For Schematics, the ability to **Plot Terminals** and plot **Pin Names/Numbers** is available for extra control over the 'look' of the final plot.

The **Output**, **Settings** and **Position** pages are the same as in the PCB Editor.



Printing Across Multiple Pages

You can print across multiple pages (with an optional overlap), this is enabled on the **Printer Setup** dialog within the **Plotting and Printing** option. The **Print** option will always print across multiple pages, if required.



The **Overlap** value enables you to specify an overlap to allow for print quality degradation at the edge of the sheets.

Note: This option is available for both the Schematic and PCB design editors.

Print Overlap

Chapter 10. Automatic Routers in PCB

Trace-Router

Automatic Routers Overview

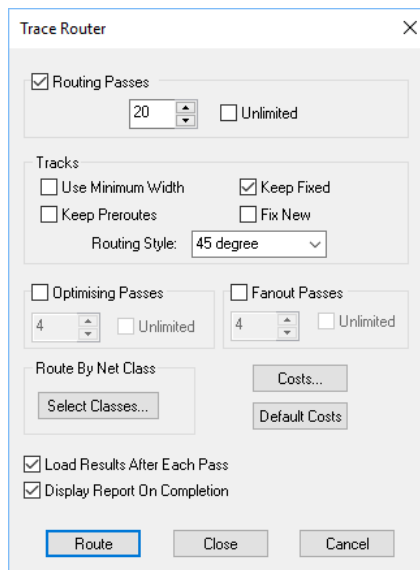
An autorouter is a set of software routines that automatically determine the routes of interconnections on a printed circuit board. Using an autorouter to route even part of your PCB will give you a significant saving in time (and hence money!), and by combining the power of an autorouter with your experience in PCB layout, the resulting design can beat even the most labour-intensive manual layout results in terms of manufacturability, cost and even aesthetics.

An autorouter takes sets of 'nodes' (connection points) and works out physical paths (where possible) between those nodes whilst at the same time obeying the design rules needed for error-free manufacture. During this task it makes many decisions about what to do when tracks cross, directions of track segments on different layers, design rule spacings between different kinds of items, and so on.

Trace-Router Overview

The Trace-Router is a modern shape-based router optimised to produce low track lengths and low via counts as well as a hand-routed looking result. It uses multi-pass algorithms to route and resolve tracking errors during its iterative process. By utilising rip up and retry technology, it can improve on its own previous pass results without major disruptions to existing tracking. The Trace-Router updates the design view as it runs allowing you to see the results as they are generated. You can also stop the router at any time, modify tracks yourself, and then re-start the router to complete the task.

The **Trace-Router** option is run from the **Tools** menu and **Trace Router**



Routing Passes

Routing This invokes a number of routing passes which will route either all nets in the design or the specified nets, depending on which option started this dialog. You can specify the number of passes that you want. The more your design the larger number of passes that you will want to give it. If it routes to 100% before the number of specified passes then no more passes will be attempted.

Tracks

Use Minimum Width means that if a net has a net class defined the router will use the minimum track style as opposed to the nominal track style.

Keep Preroutes means that existing routing is protected against being ripped up by the autorouter.

Keep Fixed means that only routes that are already fixed will be protected against being ripped up by the autorouter.

Fix New means that tracks routed by the autorouter will become fixed and will remain unaffected by the autorouter if a subsequent routing pass is made, changing the route parameters.

Routing style allows you to specify the angle of tracks, either **90 degree** (orthogonal), or **45 degree** (angled tracks).

Optimising

This invokes re-routing passes which attempt to reduce the overall track length and via count once the Routing passes are complete. You can specify the number of passes that you want. Be aware though, engaging this option can adversely affect the run times of Trace-Router and may significantly increase them.

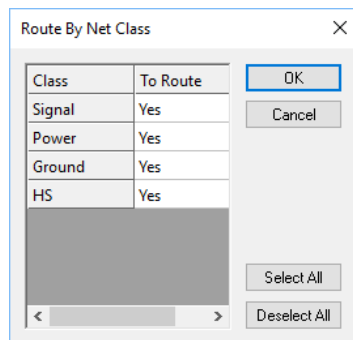
Fanout

This invokes a number of passes that routes short escape tracks to vias from SMD pads. This is done before the Routing passes. Fanout passes are not necessary in order for the router to function, and is more useful as an initial phase before manual routing. You can specify the number of passes that you want.

Any of the above pass options can be disabled by unchecking the appropriate box. You can also specify unlimited passes, in which case, the router will continue until the board is complete, or until it is cancelled.

Route By Net Class

This dialog is used to control the nets which are selected for routing.



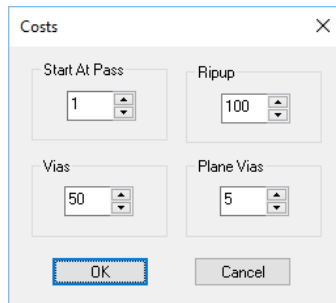
For each net required for routing or not required for routing, a net class must be specified and assigned to the net(s) concerned. Nets which are power or ground, or which have specific properties e.g. high speed or memory etc. should be assigned their own net class.

Load Results – Check this option if you want the display refreshed with the latest results after each pass.

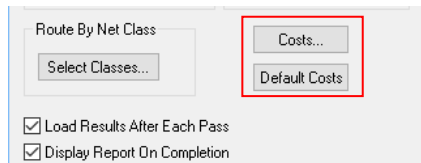
Display Report On Completion - Check this option if you want the Auto Router to produce and display a status report when it finishes.

Costs

This will invoke the **Costs** dialog from where you can change the **Cost Factors** used by the Auto Router. You can use this dialog to change the costs involved with Ripping up existing tracks and inserting vias. You can also restart the Router at a specified Pass.



Default Costs changes the costs back to their default values.



The **Route** button will start the Auto Router using the values specified in this dialog. **Close** will exit this dialog without starting the Auto Router but will store any changed values.

Preparing and Routing Designs

Before running the router, there are certain rules and guidelines that need to be considered in preparing the design for routing. This is especially important if the router is to produce the best possible results.

Design Considerations

To allow the router to work correctly and produce the best possible results, it is important to understand how the various rules and items within the design affect the routing.

This section details rules and guidelines that need to be considered before routing a design. Not all of these are “hard and fast” rules, but a good understanding of how the router uses and interprets these rules and the design data will help you to achieve the best possible results with Trace-Router.

Layers Used

Trace-Router works on designs with any number of electrical layers, adding tracks on these layers and inserting vias to drop from one layer to the other when it needs to.

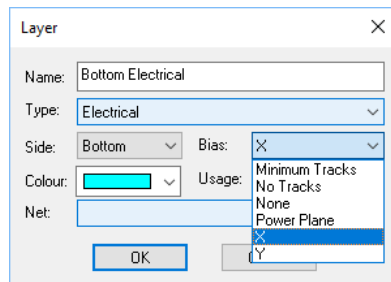
In addition to these electrical “signal” layers, the design may also contain any number of power (or ground) planes. As well as routing on the signal layers, Trace-Router knows how to route power and ground nets to the appropriate power plane layer.

Layers are added and deleted through the **Layers** dialog on the **Settings** menu.

	Name	Type	Side	Usage	Bias	Net	Colour
	test		Top	Non-Electrical	None		
	Wires	Wire	Top	Wire	None		
X	Top Silk	Silk Screen	Top	Non-Electrical	None		Yellow
X	Top Copper	Electrical	Top	Electrical	X		Red
X	Bottom Copper	Electrical	Bottom	Electrical	Y		Cyan
	Bottom Silk	Silk Screen	Bottom	Non-Electrical	None		Blue
X	Dimensions	Dimensions	Bottom	Non-Electrical	None		Green

Routing Direction (Bias)

Under most circumstances (especially when routing surface-mounted components), the predominant direction of signal flow on each layer of the board is important. This routing flow is controlled by the **Bias** setting for each layer on the Layers dialog and should be defined in consideration of the general connection flow of the design and the required routing bias on the given layers.



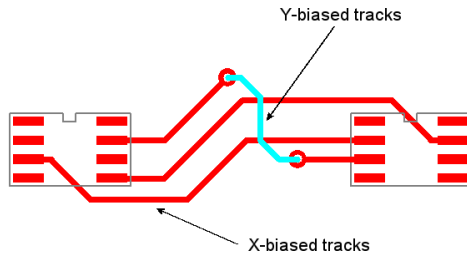
If you are routing on two layers, you should set the Bias to X on one side and Y on the other. The choice of which layer is X and which is Y will depend on the side of the board on which your components are placed, the component pad orientation, and other such considerations. You may need to experiment with these settings to find out which one gives you the best result for the kind of board you are designing.

When routing on a single layer, there are special considerations to take into account. You can use a wire layer to allow the router to add flying wires where it cannot route.

Bias values

X or Y

Tracks should generally run in the X (or Y) direction on the layer.



Tracks may be run in the 'wrong' direction, but the router treats these tracks as very 'expensive' and will try and restrict them to a minimum to avoid blocking other potential routing paths.

None

No restriction on the direction of tracks on the layer. The router is free to run tracks in any direction as it requires.

No Tracks

No tracks will be placed on this layer by the autorouter. If you accidentally set one of the outer layers to **None**, no routing will occur if an SMD component resides on that layer.

Minimum Tracks

The router will try and avoid placing tracks on the layer, but may do so if it has to. Tracks on this layer are again treated as 'expensive'.

Power plane

The layer is a power or ground plane, to which stub routes may be routed.

Nets, Net Classes and Styles

Each connection and track in the design belongs to a **net**. Each net belongs to a **Net Class**. These are used to specify a pad style to be used for vias, and two track styles to be used for routing, these are defined in the **Design Technology**.

	Name	Type	Min. Track	Nom. Track	Via	LenDiff Enabled	Length Diff	MinMax Enabled	Min	Max	Own Colour
X	Signal	Signal	Track 10	Track 15	Via 40	No	0.0000	No	0.0000	0.0000	No
X	Power	Power	Power 20	Power 50	Via 40	No	0.0000	No	0.0000	0.0000	No
	Ground	Signal	Power 20	Power 50	Via 40	No	0.0000	No	0.0000	0.0000	No

Trace-Router uses the **Via** pad style for vias that it inserts, and either the **Min. Track** or the **Nom. Track** style for tracks created during routing. The track style 'type' to be used, either **Min. Track** (Minimum) or **Nom. Track** (Nominal) is controlled by the **Minimum Width** switch on the main routing dialog.

Net Classes are important for defining different widths required for different 'classes' of connections or tracks. For example, the default technology supplied has a net class for Signals and one for Power type connections with values of 10/15 thou and 25/40 thou respectively. These values allow different track packing and different current carrying capabilities.

Spacings and Track Sizes

Trace-Router uses the same spacings as used by the **Design Rule Check** option. These define the gaps required between tracks, pads, vias, and so on.

It is important to set up the right combination of spacings (gaps) and track sizes if you are to maximise the amount of space available for routing. You will find that a little arithmetic can improve the routing immensely. For instance, when manually routing, if a track grid of 25

thou is used with 15 thou track, and the track-to-track spacing is also 15 thou, then the minimum legal pitch between adjacent tracks is 30 thou. This potentially wastes around half the possible track positions. If the track-to-track spacing is reduced to 10 thou, this will allow twice the available track positions.

	Tracks	Pads	Vias	Shapes	Text
Tracks	8.0000	8.0000	8.0000	8.0000	8.0000
Pads	8.0000	8.0000	8.0000	8.0000	8.0000
Vias	8.0000	8.0000	8.0000	8.0000	8.0000
Shapes	8.0000	8.0000	8.0000	8.0000	8.0000
Text	8.0000	8.0000	8.0000	8.0000	8.0000
Board	75.0000	75.0000	75.0000	75.0000	75.0000

Note: Depending on your manufacturing process, some processes, and especially less professional ones, may tend to over-etch the copper areas of the board. This means that tracks can be eroded during manufacture. It is important therefore to consider how the board will be manufactured when choosing the spacings and track sizes, and if need be consult with the person making the boards to see what is acceptable to them.

Board Outline

The design should contain a board outline to constrain the area for routing. The thickness of the board outline line style (as well as the track- or via-to-board spacing) is taken into account when deciding how close the router can go to the edge of the board. A typical board outline thickness would be 10 thou (provided it isn't being used for profiling or as an inner layer copper boundary for power plane clearances). A typical *item* to board spacing might be in the order of 50-75 thou (or even more) depending on the manufacturer. For power plane layers, these clearances are certainly near the correct value but again, consult with your board manufacturer.

Trace-Router will not cross the board outline, nor will it cross any routing area. This means that if there are areas of the board in which you do not want any tracks, you can add routing areas to the design in the required location to keep the router from running tracks into those areas. If you don't want tracks on either side in a given part of the board, you can add a routing area on each of the two routing layers. Routing areas are not used for routing into.

As you might expect, if you don't have a board outline in your design, the router will simply run the tracks out as far as it needs to go in "open space". There may be instances in which this would be useful, perhaps when making prototypes. In this case you may not care how large the board should be, so you can let the router do its work and then enclose the resulting routing pattern with a board outline once the routing is complete.

Routing Grids

The Trace-Router is a gridless autorouter, it does not use any grid to run. It simply uses the track thickness, via style and spacing rules defined to make the best possible use of routing space available. Correct consideration for the 'gaps' available should be made up-front.

Component Placement

Placement of components has a significant effect on the routability of any particular design. There will usually be some components that have to go in specific positions, such as connectors, but you can help the router to achieve better results by taking a little care over placement of other components on the board.

In general, you should place components so that the density of the unrouted connections (after using the **Optimise Nets** function) is reasonably even. This should allow the router as much choice of routing paths as possible. Over-congested or tightly-packed boards are clearly much harder to route successfully.

Although important for most boards, achieving adequate spacing and even distribution of nets across the board is vitally important if you are intending to route on a single side only. Single layer routing will be much easier if the design has a minimum number of crossing nets and a generous border around the edge.

As well as net density, it also helps routing (as well as build and test later on) if components are placed in neat rows and/or columns, on a regular grid. If component pads are 'off grid', the router will need extra effort to route into or out of these pads.

You can also help the router (as well as keeping down manufacturing costs and errors) by keeping component orientations consistent. For example, all IC's should be placed in the same orientation, as should other 'like' groups of components (transistors, diodes, polarised capacitors, etc.).

Where surface mounted devices have been used, and especially when they are used on both sides of the board, you should attempt to overlap pads (when they appear on the top and bottom sides obviously) so that the router has the maximum amount of 'free space' available. Normally, we call free space the area on the design which when viewed is black (assuming you have a black background). This is the space which the router can use for tracks and vias and should be kept to a maximum if possible. Where devices don't have equal amounts of pads, still line up rows of pads with each other, this all helps create free space.

Net Optimise

Trace-Router will attempt to route the connection pattern as it is defined in the design when routing starts. It is generally good practice to optimise the nets in the design before routing, to minimise the overall connection lengths of each net and generally reduce connection density.

To optimise your connection pattern, select **Optimise Nets** then **All Nets** from the **Tools** menu.

The autorouter will run **Optimise** after each routing pass (including the final or only pass), to optimise any remaining unrouted connections.

Copper Areas and Text

Shapes on electrical layers are treated as obstacles and should therefore be avoided. If obstacles appear on non-electrical layers they are ignored. Another word of caution for **copper shapes**, if they appear on power plane layers the router will avoid putting vias through them, so their use on these layers should be carefully controlled.

The same applies to **Text** which appears on electrical layers, the router will avoid it. Text on non-electrical layers is ignored.

Checking the Design before Routing

Finally, before actually running the autorouter, run the **Design Rule Check** option from the **Tools** menu. If you run the router before running a Design Rule Check and there are errors you will waste valuable routing time and may get an unexpected routing result for the wrong reasons!

Run and correct any design rule errors before running the router. Also check the **Spacings** set to validate that they are relevant to the board and design technology being used.

Schematic Designs and the Autorouter

Schematic designs or the original netlist are not required when autorouting with the Trace-Router. The PCB design is self-contained and holds all the information required by the router to run successfully, a PCB design which follows the basic rules defined in this manual is all that is required.

Interrupting Routing

At any time during the autorouting, you can interrupt the processing by pressing the **Cancel** button on the progress dialog.

Another dialog is displayed, allowing you to decide whether or not to keep the routes that Trace-Router has already created.

Press **Yes** to keep these tracks, or **No** to return to the connections as they were before this routing operation started. Tracks that are in error are always unrouted.

Resuming Routing

You can at any time during the design process go back into the routing dialog and use Trace-Router on the remaining connections. This applies equally well if you have interrupted the router to modify some of the tracks that the router has inserted, or if you have routed critical nets first (as described below). All you need to remember is to “check” the option to **Keep Preroutes**, and any existing routes will be left alone by the router.

Free Pads and Component Pads

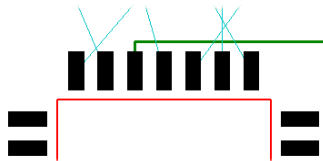
Free pads and component pads are treated the same by the router, in that they can be used in place of vias when routing tracks on different layers, and they form obstacles to routing when they are on a different net to the connection being routed (or not on a net at all).

Free pads are not removed by Trace-Router. They are treated as a fixed part of the board just the same as ‘true’ component pads.

Routing SMD Components

There are special considerations which apply when routing designs with surface mount components. Some surface mount packages can give a low success rate when routed using a single routing pass.

Consider the example shown below. The track has been routed according to the layer bias, but it has blocked connections from being routed up and out of their respective pads as they would need to be for these long rectangular pads.



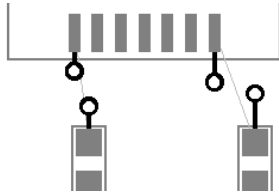
To overcome this problem, you will need to let the router complete its processing, interactively move some of the problem tracks, and then use the autorouter to finish off the rest.

Stub Routing To Power Plane Layers

Where surface mounted pads need to be connected to power planes, Trace-Router can do this for you by inserting what are called “stub routes”. These are short track paths ending in vias,

allowing the signal from the power plane to be connected up through the via and along the track to the pad.

Before the router can insert stub routes, the power plane layers must be correctly set up (with their Bias set to power plane, and with the appropriate Net assigned to the layer).



Trace-Router can then route any connection on that net by inserting a stub route, rather than by routing a complete track path between the pads at each of the connection.

Stub routes are only added to surface mount pads. Through-hole pads on power plane nets are of course already connected to the power plane through their pin, so they don't require stub routes with vias.

The distance between the pad and the via is controlled by the pad-to-via spacing, the larger the spacing, the longer the stub track.

Single Layer Routing

Single-sided boards are widely used, as they are generally significantly cheaper than double-sided and multi-layer plated-through boards. The principal areas where single sided boards are competitive are low cost products (especially high volume) and some circuits dissipating high power.

However, these benefits don't come without a cost. The restriction of routing to a single layer often means that track paths are a great deal longer (although this may not always be a problem), and links are often needed to achieve routing completion. Component placement and the resulting connection cross-over and density also becomes much more critical in its effect on routing completion rates.

Component type also has an effect on the routing results. Boards with mostly IC components often have much higher connection densities than those with mostly discrete components, and it may be unreasonable to expect good completion rates on such boards.

One of the key factors in achieving good completion rates on single-sided boards is good component placement with plenty of free space, especially if you can leave a wide "border" around the edge of the board. It is usually better to start with components well spread out, and only move them together if you find that the routing result allows you to do this.

To make the router operate on a single layer it is only necessary to set up the appropriate Bias on the electrical layers. The layer on which the tracks are to be routed should be set with a Bias value of 'None'. This allows the router to run tracks in any direction on the routing layer. The other electrical layer (if there is one) should be set with a Bias value of 'No Tracks', to prevent the router from placing tracks on that layer.

There is a variation on this setup that can be used if you find that there are a few connections that simply cannot be routed on the single layer. By setting the bias of the other layer to 'Minimum Tracks', the autorouter can now place tracks on the "non-routing" layer, but will try and keep them to a minimum. As with all attempts to perform single layer routing, you may find that you need to experiment with the settings to achieve the best result. It may be best only to allow this minimal tracking on the "other" side for a final pass with those connections which won't route on the preferred side.

Alternatively, you can use a Wire layer and manually add wires to jump gaps in the tracking, this is a design process following automatic routing.

Post-Routing Cleanup

Depending on the density and complexity of the board, there may be some connections that Trace-Router has not routed. This is frequently due to the restrictions on track direction on the routing layer(s), or congestion around the start or end pads of the connection. It is often possible with a little careful manual editing to clear enough space to allow the router to find a suitable routing path for these unrouted connections.

If there are a lot of unrouted connections in a particular area, this can tend to point towards component positioning as the problem. Rather than struggle to route the connections, it is often worth moving (or rotating if necessary) some of the components in this area and trying the router again. Don't forget to use Optimise Nets after moving any components, again this will help to reduce connection density. You could even consider alternative components to make more board space available for routing. You could perhaps use surface mounted decoupling capacitors even on a through-hole board (although this will of course affect the manufacturing cost).

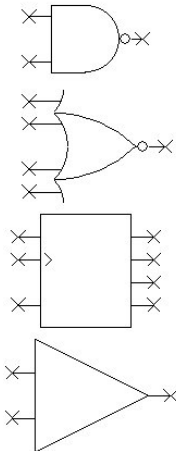
Chapter 11. Library Creation

Understanding Libraries

The next few chapters concentrate on getting you started with creating your own libraries. After the chapter on designing Components, there is more detail about how to use the Library Manager at the end of these chapters.

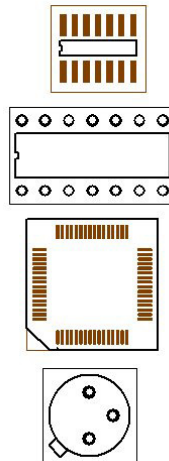
Libraries are easy to understand and use once you grasp the basic concepts. The library structure is made up of three individual libraries. There is a **Component library** made-up of elements from the **Schematic library** and the **PCB Symbol** (footprint) library.

Schematic Symbol Library



Symbols are often called 'gates'. These have generic names like: NAND, Buffer, OPAMP, Res, Cap

PCB Symbol Library



These have generic names like: DIP14, SOIC14, QFP44

Component Libraries



These normally use the manufacturer's proper component names like:

SN7400
SN74LS10
SN74HCT4002
SN74LS258
CA3450

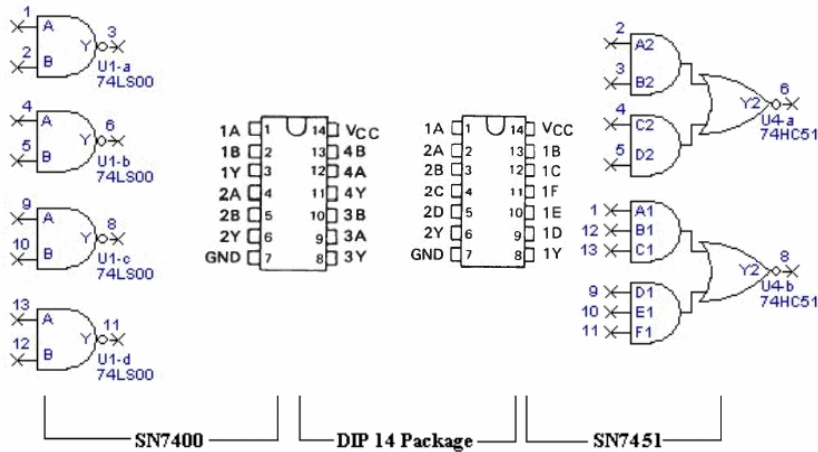
These symbols are used in the Component libraries to produce Components



Only the names from this library is used in both Schematic design and PCB layout.

The benefits of separate Schematic Symbols, PCB Symbols and Component Libraries

In electronic design we have Schematic Symbols and we have PCB Symbols (or footprints). Commonly, integrated circuits and other components use the same physical footprint, but have completely different technical specifications and internal digital or analogue circuitry. Take for example, the standard 14-pin DIP footprint:

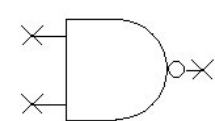


The 14-pin DIP (Dual Inline Package) is used on 100's of different components. For efficiency, we only want to store the 14-pin DIP PCB Footprint once and then use it repeatedly to build different Parts or components that specify a 14-pin DIP footprint.

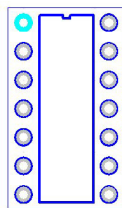
By the same token we only want to store a Schematic Symbol once and use it repeatedly in various configurations to produce different components. Hence, we have a **PCB Symbol library** to store a 14-pin DIP footprint plus any other footprints and a **Schematic Symbol library** for a 2-Input NAND gate plus any other Schematic symbols. These can then be used on their own or be brought together (Schematic Symbol plus PCB Footprint) in many different permutations to create a **Component Library**. You also have multiple component libraries to make the categorisation of components clearer, these can be by manufacturer, type etc.

Key Point: It is only the component name taken from the **Component library** that is used in the Schematic Design and PCB layout

Here's how we go about creating a simple 7400 component for example, to use in a Schematic design and the PCB layout:

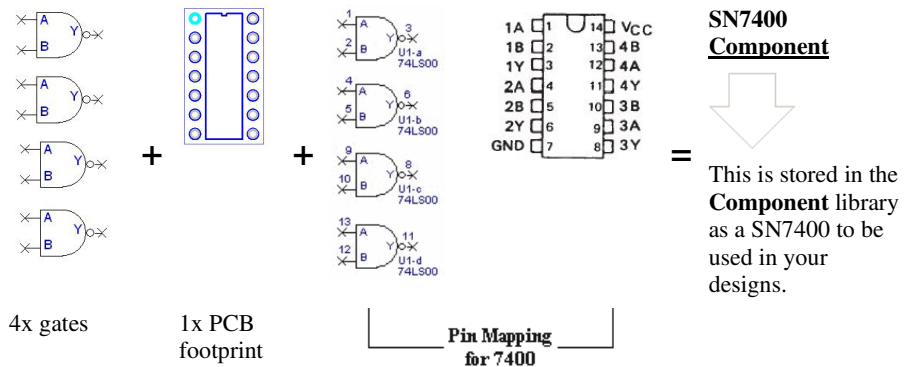


4x 2-Input NAND gate symbol from the **Schematic Symbol library**.



1x 14-pin DIP footprint from the **PCB Symbol library**.

We can create a new 7400 Part by selecting 4x 2-Input NAND gates from the **Schematic Symbol library** and a 14-pin DIP footprint from the **PCB symbol library**. By combining these in the component editor and inserting the correct pin mapping as per the data-book specification, we produce a SN7400 **Component**. This is then stored with a unique component name (SN7400 in this example) in the **Component library** ready to use in your designs.



Can I create a Schematic symbol and then select it for use in the Schematic design without creating a PCB symbol?

You can create a new Schematic symbol and store it in the Schematic symbol library. But before you can use it in a design you have to create a **Component** to 'hold' the Symbol and store this in the **Component library** for use in your designs. A Schematic only component could be used as power and ground symbols for example, or may just be this way for now until you decided which footprint to use in the PCB design. The PCB symbol information can be added to the component later on in the design process.

Why is there this added step of creating a Component - why can't I just use the symbol immediately?

Working this way, any Schematic symbols you produce are stored with an appropriate name in the Schematic symbol library. It can then be reused multiple times to create different Components using the manufacturer's component name. Most Components you produce will be complete with Schematic symbols and PCB symbol.

Is this the same for PCB symbols? Can I create a PCB symbol without a Schematic symbol?

This is exactly the same for PCB symbols as with Schematic symbols. Before you can use your new PCB symbol in a design you have to produce a component even though this component may not have a Schematic symbol associated with it. A PCB-only component could be added to the design as a drawing blank or test-pad probe point for example.

On the following pages is a tutorial that goes through the steps of creating Schematic symbols, PCB symbols and a component to tie them together.

Note: Standards for symbols vary from company to company. The tutorial is presented only as a means to understand the process of design and management and how to use the tools available.

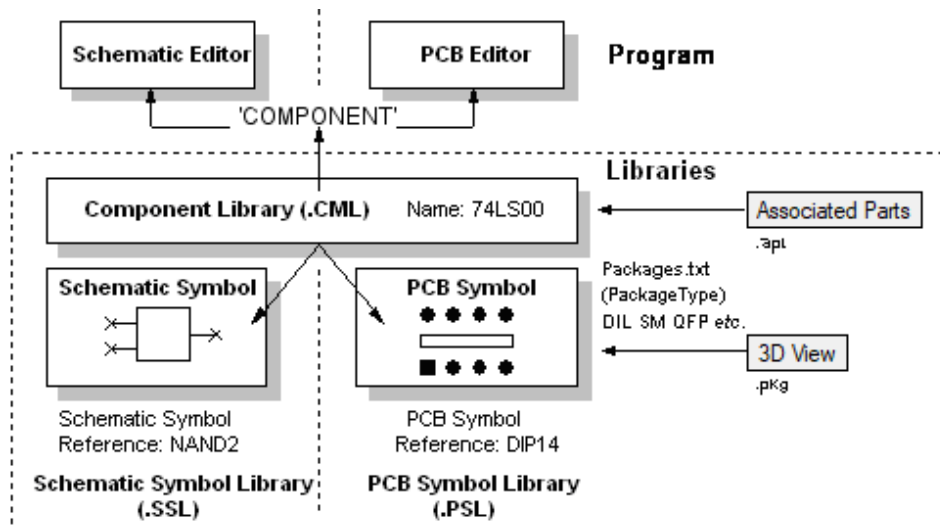
Introduction to Schematic and PCB Symbol Libraries

Two types of symbol libraries exist as discussed previously, those for Schematic symbols with the file extension `.ssl`, and those for PCB symbols with the file extension `.psl`. There is no limit how many symbols may be kept in a given library.

Introduction to Component Libraries

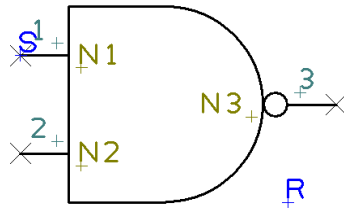
Component libraries can be identified by the extension `.cml`. Each component in a library is aware of the symbol name and library names of its constituent symbols. The components can contain several Schematic symbol gates in each component, and a number of PCB footprints, each with its own symbol. For example, a component might have 3x NAND gates and a power gate for use in the Schematic design, a DIL, and an SMD footprint for use in PCB.

The diagram below shows how the component will access both the Schematic symbol library and the PCB symbol library as required in the relevant design editor.



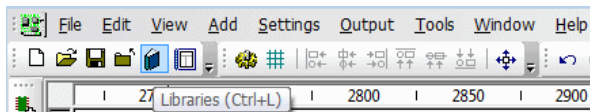
Creating Schematic Symbols

For this tutorial, you will be guided through a basic symbol example, a 2-Input NAND gate. This symbol is already available in the standard library but it gives us chance to use some of the shape editing features to create the symbol body.

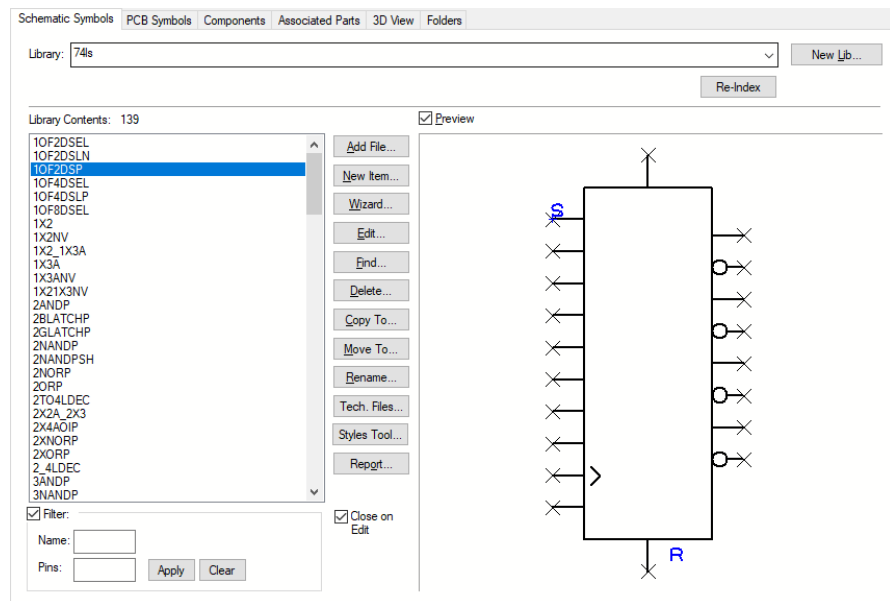


► To open the symbol editor

1. Click the **Libraries** icon on the **File** toolbar (the one that looks like a 'book'), shortcut **<Ctrl-L>**. You can also do this by clicking on the **File** menu and selecting **Libraries**. For this tutorial, we will use toolbar icons where possible.

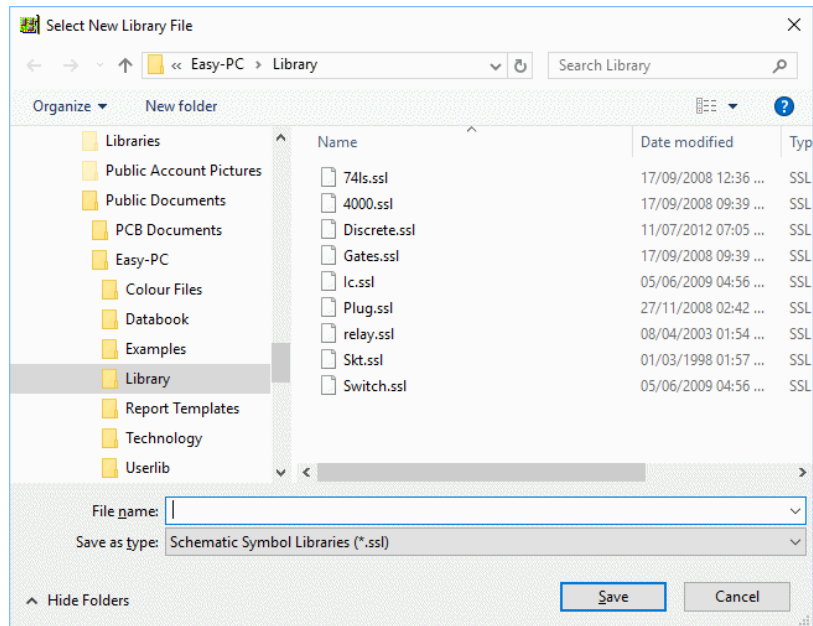


2. From the **Library Manager** dialog select the **Schematic Symbols** tab.

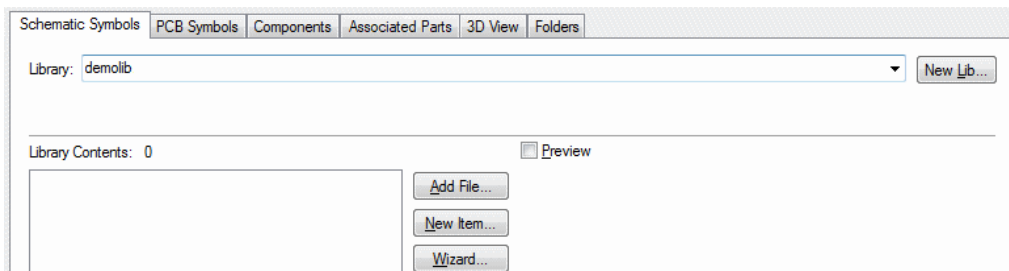


3. On the **Library**: drop down list, [**All Libraries**] will be selected.
4. The Library Contents list will show Schematic symbols that already exist.
5. We will create a new symbol and add it to a new library called Demolib.
6. For now, we will leave the Technology file to the one defined.

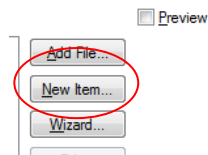
7. Click the **New Lib** Button.
8. In the **File Name** entry, type **demolib** and **Save**.



9. The **Library** entry will show **demolib** with no contents.



10. Click the **New Item** button.



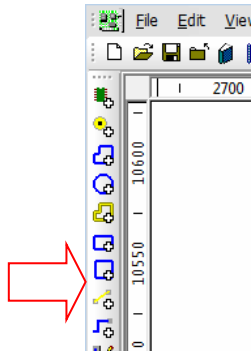
11. The Symbol editor opens. You will see that it looks similar to the Schematic design editor. It is pretty similar except it is limited to functionality required to create Schematic symbols.

► Zoom in to view the grid dots

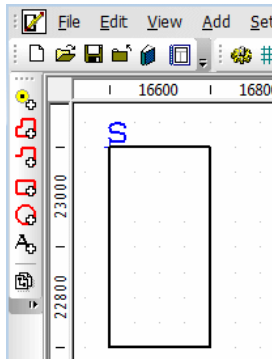
1. You will need to zoom in a bit (roll the mouse wheel or click the <Z> key) so you can see the **Screen Grid** displayed.
2. The grid is set to 50 thou increments (as defined in the selected Technology file).

► Adding the symbol detail

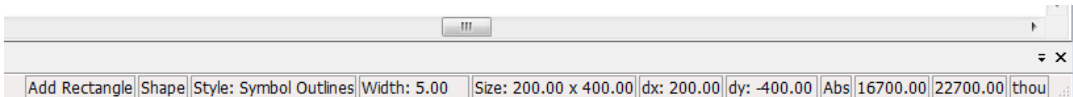
1. From the Schematic Symbol toolbar (the vertical toolbar on the left side of the screen) select **Add Shape Rectangle**.



2. Choose a starting point on a grid dot, hold down the left mouse button drag a rectangle down to create a shape 200.00 x 400.00 thou.



3. The status bar at the bottom of the screen will display the **size** being drawn as you draw it. It will show the **Size:** value changing dynamically as you edit the shape.

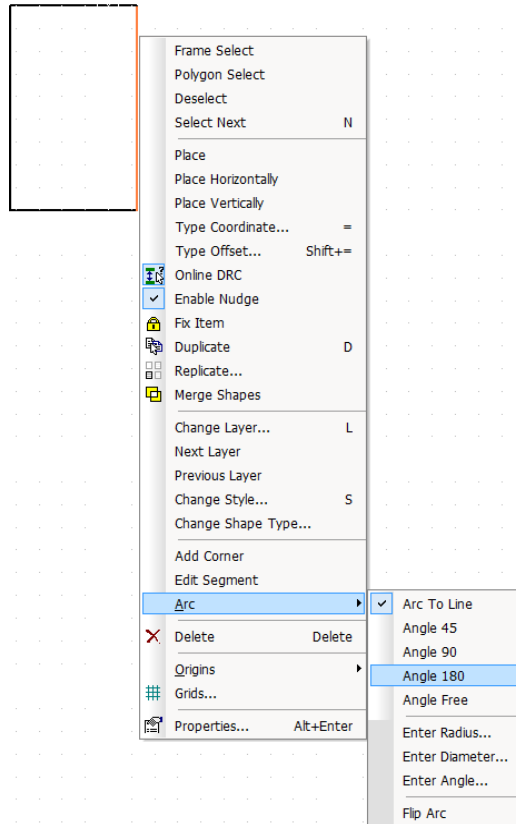


4. The Symbol origin point 'S' is at the top left corner of the shape. This is always added automatically to the first item when creating a symbol. We will move this later on to a more relevant location.
5. When the shape is the size required, click the mouse button once to release it and press the <Esc> key to cancel this option.
6. The first stage of your Symbol should look like the picture above. If it isn't to quite the right size you can easily click and drag the shape segments that make up the rectangle until it is 200 x 400.

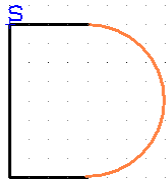
► To create an arc

The shape needs an arc on one side of it. Again, the tools are on-hand to create this easily.

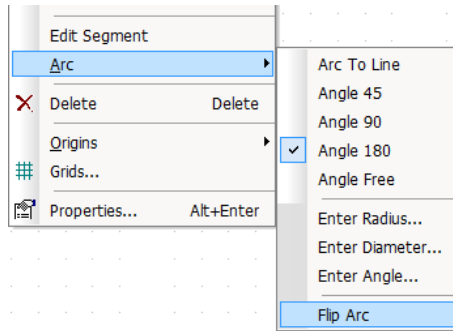
1. Right click the mouse on the right hand vertical shape segment (this is a standard Windows command and emulates a left mouse click to select then right click to view the context menu, and is quicker).
2. From the context menu choose **Arc>** and then **Angle 180**



3. The segment selected now becomes an arc.



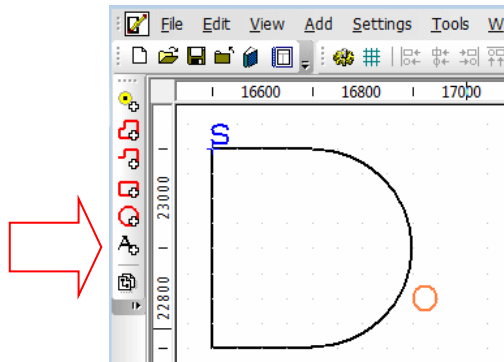
- If your arc is pointing the wrong way, right click on the shape again and choose **Arc>** and then **Flip Arc** from the context menu.



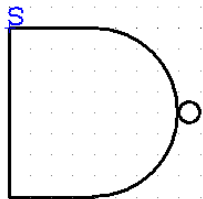
► Adding the inverter circle

We need to add an 'inverter' circle to the output pin of the symbol.

- From the Schematic Symbol toolbar select **Add Shape Circle**.



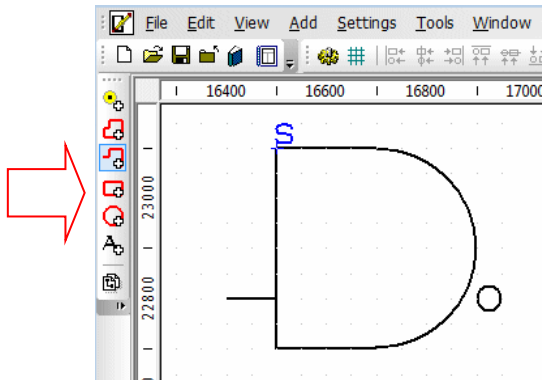
- In free space, click and drag open the circle to a radius of 25 thou, this will be displayed on the status bar.
- Release the mouse key to finish the circle.
- Click <Esc> to cancel the Add Shape Circle mode.
- Select and drag the circle, position it at the end of the arc.



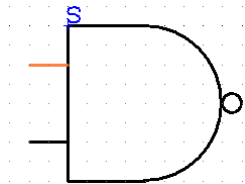
► Adding more symbol detail

- Pan to the right so that you have a bit more space at the sides of the symbol. Use the <P> key to pan or use the scroll bars.

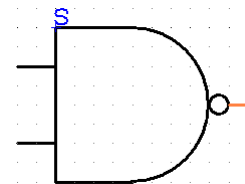
- On the Schematic Symbol toolbar, select **Add Open Shape** or use **Add Shape Single Line** from the **Add** menu.



- On a start point 100 thou (2 grid dots) above the bottom left hand corner of the shape, drag a line 100 thou long (2 grid dots) to create a horizontal 'leg'.
- Double-click to finish this line.
- Add a second line 100 thou (2 grid dots) from the top left corner of the main shape.
- Create the line 100 thou long as shown below. Double-click to finish.



- Click the <Esc> key to cancel.
- Now add the third leg to the right side of the symbol.
- This shape will be only 50 thou long because of the inverter circle already positioned.

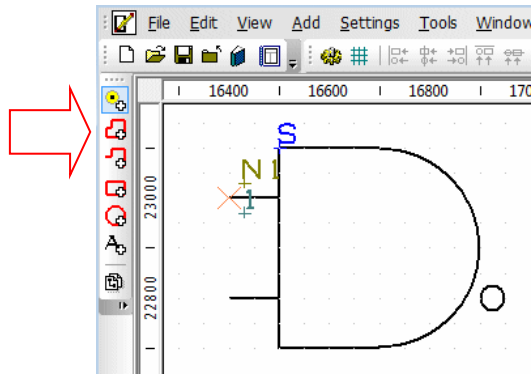


- At this stage the lines are not recognised as electrical connect points or pins. We are now going to insert these as Pins.

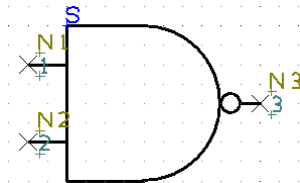
► Adding pins

Pins need to be added to the Symbol to give 'real' electrical points on which to make connections in the design. This gives its unique true electrical connectivity at all times.

- On the Schematic Symbol Toolbar, select **Add Pad**.



2. Drag each pin onto the symbol and position it on the end of the lines so that it looks like the example below.



3. You will see that these are auto-numbered as you proceed. After the last pin has been added, click the <Esc> key to cancel.
4. The pins must be numbered sequential overall but the numbering sequence on the symbol doesn't matter, they are there for reference only.
5. Click the <A> key to position the symbol to fit the window.

► Information on the pins

Each Pin contains three pieces of information: the big cross marks the location of the pin, **N1**, **N2**, **N3** etc. which marks the location of the **pin name** of the component when used in the design, and the **pin number** position marker (**1**, **2**, **3** etc.).



The pin name **N1** will only be fulfilled once the component is used in the Schematic design, until then you only see the name position marker.

You will also see the **S** character on Pin 1 (if the pin was the first item to be added to the Symbol before the Symbol Shape or if you have subsequently moved it).

The style of the pin can also be changed using the **Properties** dialog for a selected pin.

Style: 2
 Width: 60.00 Shape: Cross
 Length: 0.00
 Corner: 0.00 Corner: 0.00
 Pin Name Pin Number Net Name
 Net Sheets

By default, the pin styles is taken from the **Setting** menu, **Defaults** and **Pads**. Pad styles are defined in the **Pad Styles** dialog under **Design Technology**.

The **Pin Number** is automatically assigned a number. The number is shown in the position it will appear in when used in the design. This can be moved in the Symbol editor but cannot be moved once placed in the design.

If you select the N1 and choose **Properties** from the context menu, the following dialog is displayed:

Text: N1
 Position: 2175.00 3125.00 Rel
 Angle: 0.00
 Alignment: Left Middle Right
 Show
 Pin Name Pin Number Net Name Net Sheets

You can change the text alignment that will be used once the symbol is placed in the design, you can opt to align it Left, Right or Centre

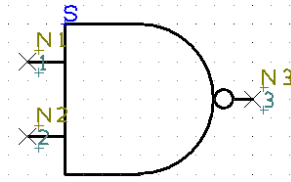
*Tip: When adding terminals, if you have moved the sequence number and the location or alignment of the pad name text, if you wish to use this on subsequent terminals, select it first and then press **Add Pad**. This will take the style of the selected terminal and add the next one in exactly the same way. If you don't select one first, the **Remembered Style** for **Pads** is used.*

► Moving the symbol origin

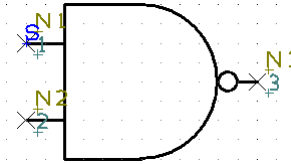
The Symbol origin is the symbol's reference point used when moving it in the Schematic design. For consistency we always place this on the pin 1 of the symbol.

The symbol origin would have been added with the first design item. In our example, this was the outline shape but it could have been the first pin added.

1. Select the origin on the symbol (the small 'S' marker). This is currently on the top left side of the symbol shape.
2. If you don't select it the first time, click the <N> key on the keyboard to **Select Next**. This will pick the next closest item to the one selected. Click <N> until you select the symbol origin marker.



- Pick and drag the symbol origin to pin 1 as shown below.



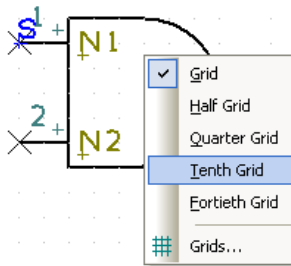
► Moving the pin name and number positions

The default position of the pin names and numbers is not what we would like. There is no right position, it's a personal preference to make the drawing more aesthetically pleasing when viewed.

You are still working using a 50 thou grid. This should be changed to a finer grid so that more precise name placement can take place.

► Changing the snap grid

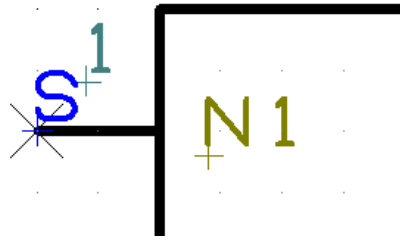
- With nothing selected, in 'free' space press the <H> key.
- From the context menu select **Tenth Grid**, this will set the **Working Grid** step to 5 thou (a tenth of the 50 thou grid).



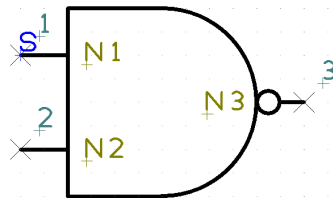
- Now that the grid step is set, anything you place or move will be on this new grid.

► To move the pin names and number positions

- Zoom in by rolling the mouse wheel or use the <A> key on the keyboard so more detail of the symbol is visible.
- Select and drag the pin names, **N1** and number, **1**, into position as shown below. You may wish to position them to your own requirements, doing this has no impact on this tutorial but allows you to try it.



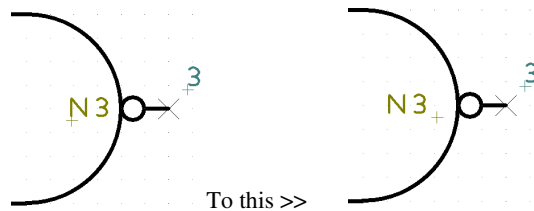
3. Note the position of the pin name **N1**. The small plus + sign sits below the centre line of the pin. It is the pin name **N1** position that represents the actual pin name position in the design.
4. The name and number positions now look like this:



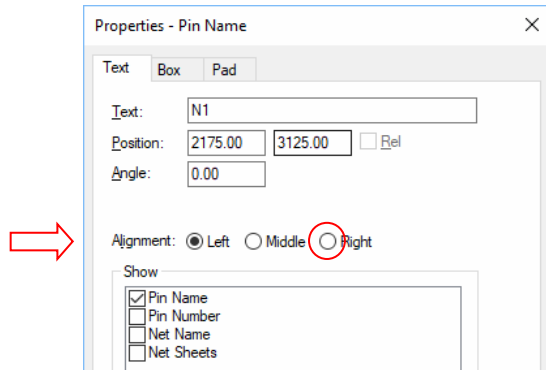
5. One small detail to change before we move on is the alignment of the pin name position for pin 3.
6. Currently, the pin name is aligned to read right to left. For this pin name position to the right side of the symbol, ideally it should read left to right. We will change this to demonstrate the ease at which it can be done.

► Changing the pin name text alignment

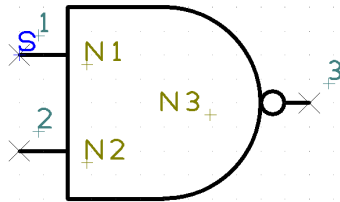
We want to change the alignment on pin 3 from this:



1. Select the pin name marker **N3**.
2. Right click and select **Properties**.
3. From the **Alignment:** list select **Right**. This will change it from its current position of Left.



4. Click **OK**.
5. The pin name would have moved left on the symbol.



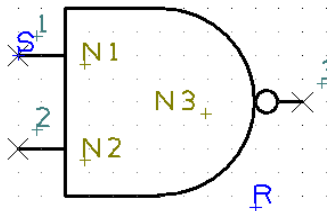
6. Drag it back to be closer to its host pin, pin 3.

► Adding the reference origin

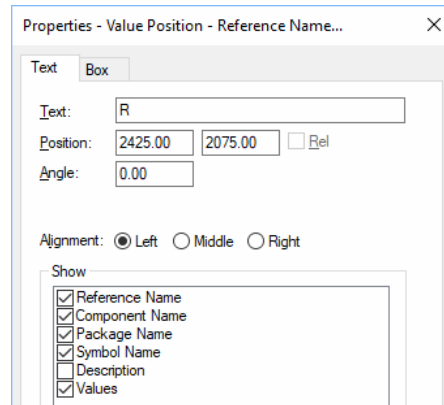
The component Reference Origin is the position that the component name (U1, IC2, R3 etc.) resides. Without a Reference Origin, the component name used in the Schematic design will automatically be on the Symbol Origin. By adding a specific Reference Origin you can make the place Component name in a location of your choosing.

By default, the libraries have the **Reference Origin** set to be the bottom right side of the Symbol.

1. From the **Add** menu, select **Reference Origin**.
2. Place the 'R' marker on the Symbol.



3. The first time you add a Reference Origin, you should edit its properties so that the **Reference Name** only is displayed when used in the design.
4. Select the **Reference Origin** and from the context menu select **Properties**.



5. In order for other information such as Component Name or Values to be displayed, you must add **Value Positions**. This is described below.

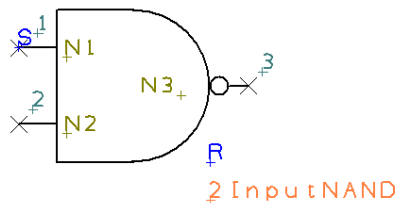
► Adding additional reference origins to display values

Value Positions are used principally for positioning the component **Values**. You can also use it for displaying Component Name, Package Name and PCB Symbol Name etc. if required.

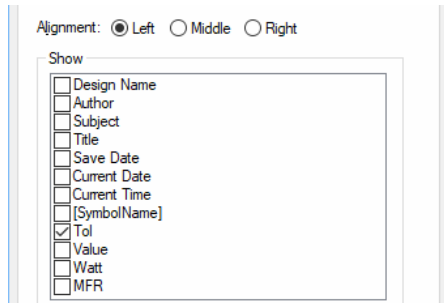
If user defined values are to be used and their position is specific, then the value names should be added to the same **Technology file** that you will use to edit both the Schematic design and the Schematic Symbol.

► Adding Value Positions

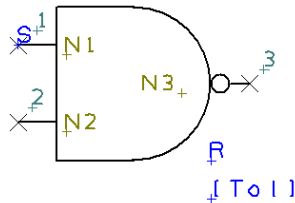
1. From the **Add** menu, select **Value Position**
2. You must add position markers for each value field that you wish to display on the symbol. Value fields for **Watt** and **Value** for example would require two **Value Positions**. These Values would have been previously defined in your **Technology** and will be then available for use.
3. With the value position marker on the end of your cursor, position the first marker where required, we've positioned our example under the **Reference Origin**.



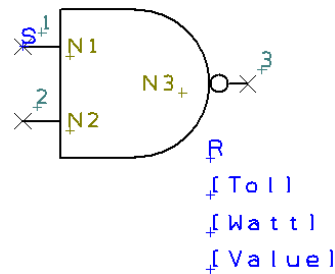
4. Notice that the value position name has defaulted to [2InputNAND, this is the design name value (a symbol is a design when editing the symbol, hence why it is displayed as 2Input NAND). This value can be changed.
5. Select the name value and from the context menu, select **Properties**.
6. Select the **Value** you require. We have selected **Tol** as our example. Click **OK**.



7. In the symbol, **Tol** looks like this:



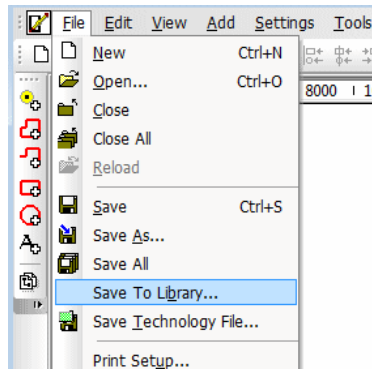
8. Using the same method, we will now add **Watt** and **Value** to the symbol.



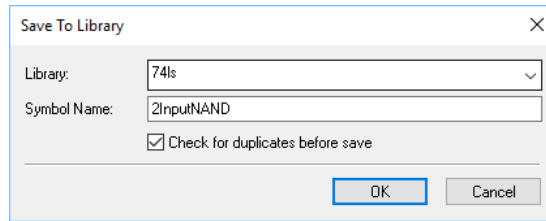
9. You are now ready to save the symbol into the library.

► **Saving the symbol**

1. From the **File** menu, select **Save To Library**.



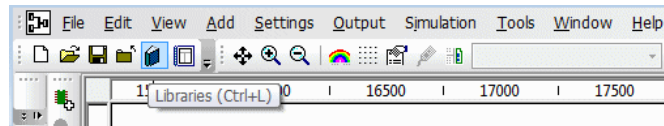
2. The **Save To Library** dialog opens.



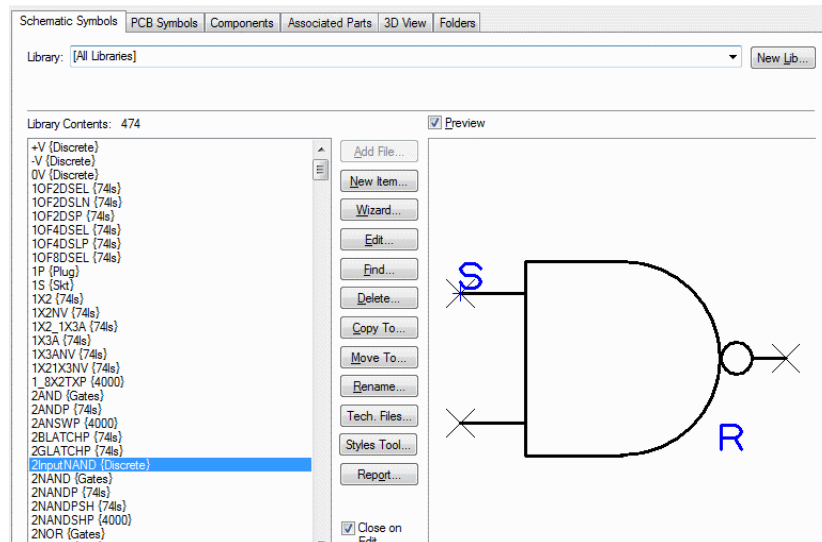
3. Choose the **Discrete** library from the list and give the symbol the name **2InputNAND**, click **OK**.
4. The symbol is saved. You can close the symbol editor now using **Exit** from the **File** menu.

► Viewing the symbol in the library

1. Now that the symbol has been created and saved, let's view it in the library manager.
2. Select the **Libraries** icon on the **File** toolbar.



3. Select the **Schematic Symbols** tab if it isn't already selected.
4. Check the **Preview** box.
5. Select the **2InputNAND** symbol from the list of **Library Contents**: it should look like the one below.



6. If you don't see your symbol in the list, you may have saved it to another library by mistake. You can use the **Find** button on the library manager dialog to locate it.

Introduction to Library Creation Wizards

The library creation wizards provide a quick method of creating symbols and components using a 'guided' system. Complex footprints are especially suited to the wizard as it contains parameter fields matched to the manufacturer's datasheets which when completed, creates a footprint as per the specification.

Wizards are provided for **Schematic Symbol** creation, **PCB Symbol** (footprint) creation and **Component** creation. These can all be accessed from each of the library tabs from within the **Library Manager** itself.

More specific detail on each of the wizards, including how to set them up and use them can be found in the online help.

We will use the footprint wizard to create our tutorial footprint.

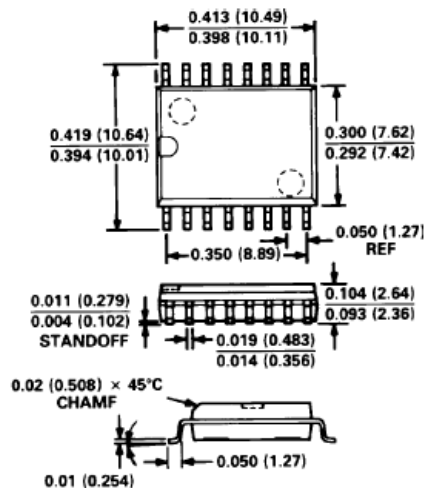
Creating PCB Symbols (Footprints)

You can create footprint symbols using the footprint wizard or interactively using the editor. Once created using either method, you can then use the footprint editor to change the footprint detail.

For this tutorial, we will use the footprint wizard to create the symbol and then discuss some of the editor details later on.

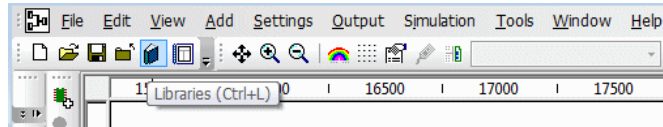
We create this footprint even though it already exists in the supplied library:

16-Pin SOIC (R) Package

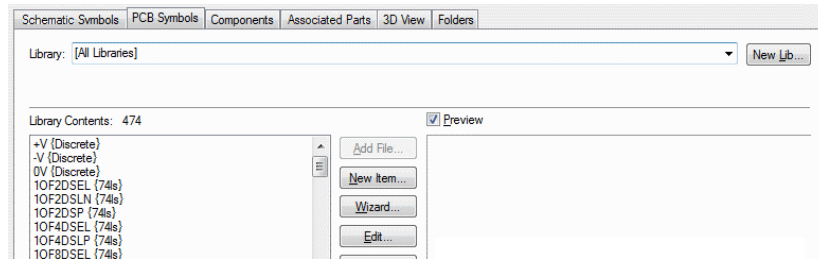


► To open the symbol editor

1. Click on the **Libraries** icon on the **File** toolbar.

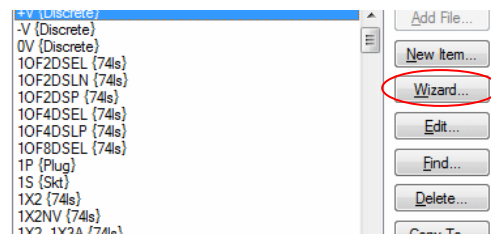


2. From the **Library Manager** select the **PCB Symbols** tab.



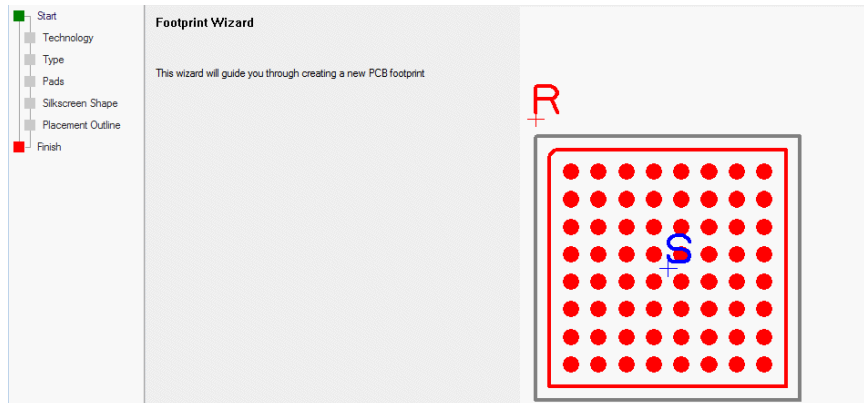
► To start the footprint wizard

1. Click the **Wizard** button from the **Library Manager**.

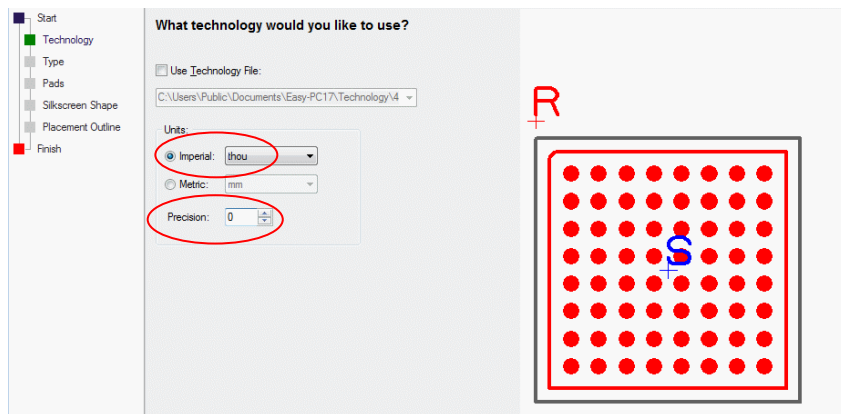


For the purpose of clarity in this tutorial we have changed the colours of the items and the background so they contrast better when printed or viewed as a PDF file. If you prefer, you can change the colours to be the same using the Colours dialog.

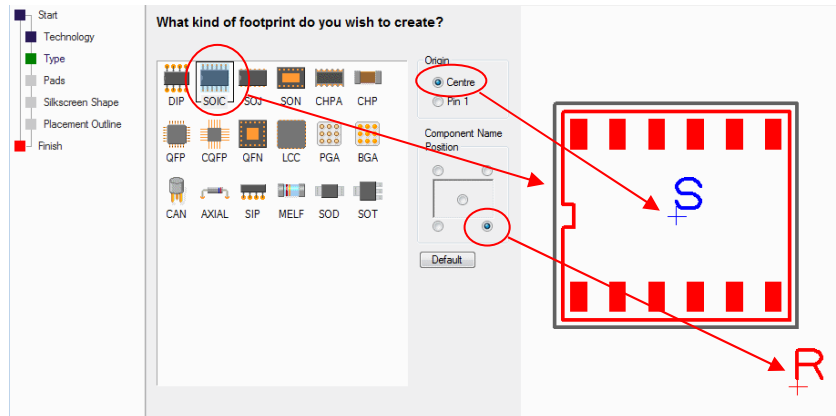
2. Before we start, you will notice that from the typical manufacturers datasheet shown above as our tutorial example, the number ranges supplied are pretty vague. The creation of footprints is not an exact science and depends largely on the final manufacturing process.
3. The wizard opens and the **Start** page is displayed.



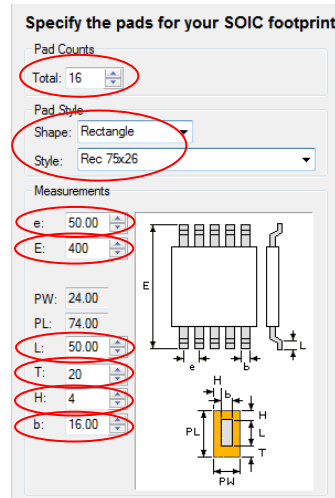
4. This page indicates the start of the creation process.
5. Click the **Next>** button to move forward to the next page. At the bottom of the page, you can navigate the dialog page by page backwards and forwards. If you wish to skip pages or go back more than one page, use the navigation bar along the left side of the dialog.
6. The next page allows you to specify a **Technology** file to be used if required. For this tutorial, we will use leave this and create our styles when required.



7. For this example, change **Units** to **Imperial** and **Thou**, and a **Precision** of **0**
8. Click the **Next>** button
9. This page allows you define the **type** of footprint you wish to create. This sets up subsequent pages based on your selection.
10. Select **SOIC** as the type.
11. Click **Centre** for the symbol **origin**. We have used Centre for surface mount devices and Pin 1 for through-hole device by convention in the supplied libraries.
12. Click the bottom right radio button for the **Component Name** position. You will see the preview update as you make the changes.



13. Click the **Next>** button
14. The **Pads** page is used to define the overall footprint layout. This is one of the most important pages of the wizard as it allows you to map the manufacturers' datasheet specification to your footprint.
15. We need to change quite a few of the parameters in this dialog.

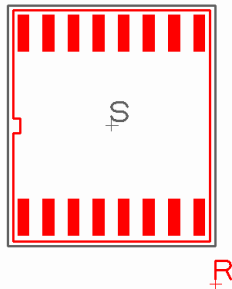


16. Change the **Pad Count:** to **16**.
17. Change the **Pad Style: Shape** to **Rectangle** (this should be the default anyway)
18. We will be changing the pad style dimensions to you need to edit the **Pad Style** name to be something more meaningful, type **Rec 75x24**
19. Under **Measurements**, type the following for each category:

e	50
E	400
PW	Automatically calculated by the wizard
PL	Automatically calculated by the wizard

L	50
T	20
H	4
b	16

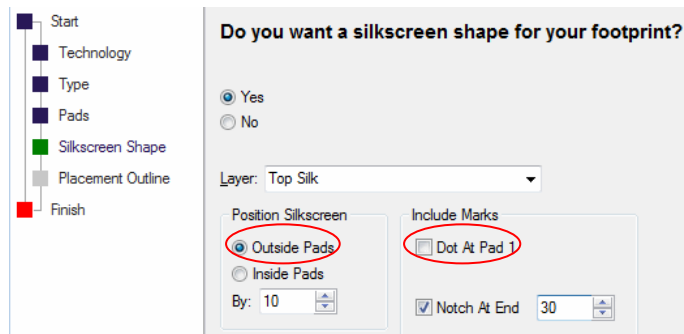
20. The preview will update each time and will now look like this:



21. Ensure that the **Pin 1** position is the **bottom left** corner, the **Pin Numbers** are **hidden** and the **Pin Numbering** is **Anti-Clockwise** (Counter-Clockwise).

22. Click the **Next>** button

23. The **Silkscreen** page allows you to say whether you require a silkscreen outline and how it will look.

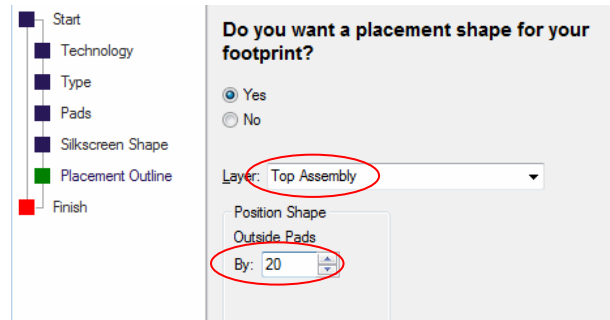


24. You will require one for this tutorial.

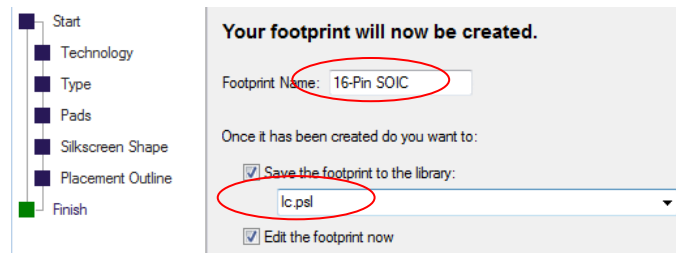
25. The only parameters to edit are the **Position Silkscreen Outside the Pads** by **10** and include a **Notch At End** with a size of **30**.

26. Click the **Next>** button

27. The **Placement** page allows you to say whether your footprint will have a placement outline or not. Placement outlines can be used for assembly purposes or to define the maximum bounding box size of the component.



28. We will add a placement outline to our example.
29. From the **Layer:** drop down, choose **Top Assembly**. You don't want the placement outline on the silkscreen layer as it will then be plotted on the finished PCB.
30. In the **Position Shape Outside Pads** box, type **20** as the value.
31. Click the **Next>** button
32. The **Finish** page lets you type the **footprint name** and choose which **library** to save it to.

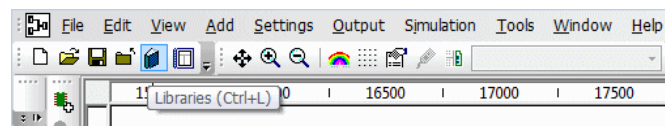


33. In **Footprint Name** type **16-Pin SOIC**.
34. Check the **Save the footprint to the library** box and select the **IC.psl** library from the drop down list.
35. Click the **Finish** button to save your footprint and exit the dialog.

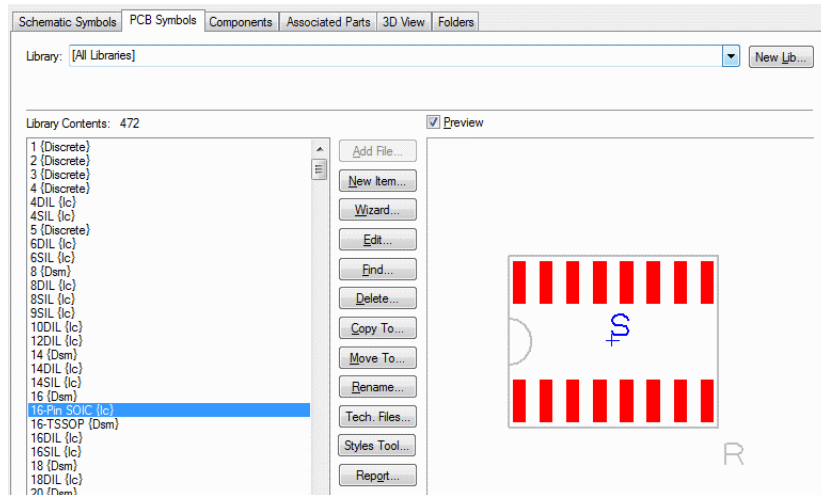


► Viewing the symbol in the library

1. Once the symbol has been created and saved, let's view it in the library manager.
2. Click the **Libraries** button on the **File** toolbar.



3. Check the **Preview** box.
4. Select the **16-Pin SOIC** symbol that you just created, it should look like the one below.



Additional Footprint information

Using the wizard will create you a finished footprint. This footprint can be edited in the footprint editor to change features or add additional information. Where footprints required do not fall into one of the 'types' available in the footprint wizard, you would then use the Footprint Editor to interactively create them. Much of the footprint editor is fundamentally the same as the editor used to create Schematic symbols previously.

The short sections below highlight some of the important areas of using the footprint editor.

► Information on the pads

When pads are added in the footprint editor (using **Add Pad** from the **PCB Symbol** toolbar), each pad has three pieces of information

- The 'S' character on pad 1 (if the pad was the first item to be added to the new symbol)
- The pad name marker **N1** which denotes the location of the pad name once used in the Component. The pad name **N1** will only be fulfilled once the component is used in the PCB design, until then you only see the name position marker.
- The Pad Number **1**. This is the pad number used in the design. This can be moved or renamed in the symbol editor but cannot be moved once placed in the design when used in a component.



One other vital piece of information for the pad is its layer. The pad will be placed on [All] layers by default, indicating that it is a through-hole pad spanning all layers. This can be changed to [Top] or [Bottom] to make it surface mounted by selecting the pad and choosing **Change Layer** from the context menu (this is covered later on).

► Adding mounting holes to your symbol

If your symbol requires additional pads to be added for use as mounting holes (lie on a D-type connector for example), you should use the **Add Pad** option from the **PCB Symbol** toolbar.

Although these pads will be sequenced along with the symbol pads, it does make it much easier if they exist outside the main pad sequence so they can be ignored. You can have a different number of PCB pads to Schematic symbol terminals. The mapping of terminals and pads will be covered in the next section on *Creating Components*.

► Changing Pad Styles

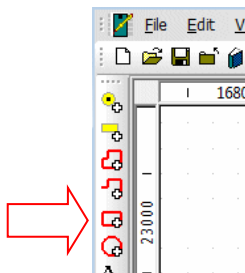
Pad styles are easy to change.

1. Select the pad.
2. From the context menu choose **Change Style** (shortcut key 'S').
3. From the **Pad Style** dialog, choose the new style required from the list.
4. You can also create your own style form here and give it a name.
5. As a tip on changing pad styles, before you add all your pads to the symbol, decide on the style you need before you start.
6. Create the style you require up-front and add it to the **Pad Styles** dialog.
7. Add the first pad using the **Add Pad** option.
8. Select this pad and use **Change Style**, change it to the style required. Use **Change Layer** to make it through-hole [All] or surface mounted [Top].
9. With the first pad still selected, add another pad. Because the first pad is selected, the second pad will pick up this style.

► Adding extra shape information

Shapes can be added to other layers, some for documentation purposes and some to decorate the footprint, like silkscreen shapes for example.

1. Use the **Add Shape options** from the **PCB Symbol** toolbar.

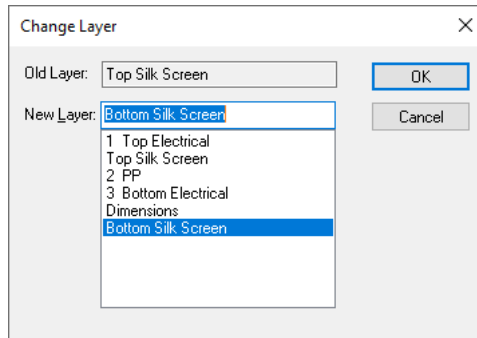


2. When adding the shape or editing it afterwards, there are various options on the context menu to select from.
3. You will mostly use **Change Style** and **Change Layer**.

► Changing layers of items

1. To change the shape layer, select the shape, or right click while adding it.
2. From the context menu select **Change Layer** (shortcut 'L').

- From the drop down list, select the layer required. The **Old Layer** is displayed to show you your original reference layer.



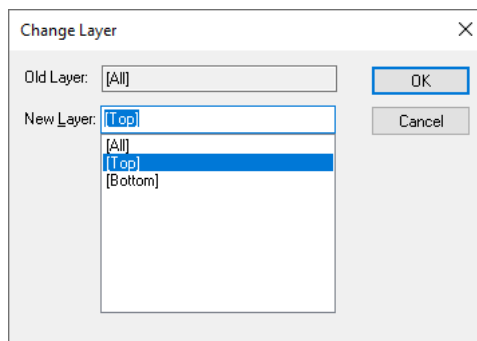
Creating Surface Mounted Footprints

Before moving onto the design of Components, one final consideration needs to be made for surface mounted pads of the PCB Symbol.

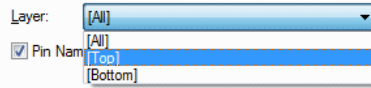
The only consideration needs to be for the side that the pads reside on. For normal pads, this would be the [All] layer. This means that the pad occupies all of the layers in the design. Whereas a surface mounted (SM) pad would only occupy one of the outer layers [Top] or [Bottom].

► Changing the layer of a pad

- During the addition of a pad or once the pad has been added, the layer can be changed. You may also find that you need to change the pad style as well.
- Select the pad(s) and from the context menu select **Change Layer**.
- From the dialog, select the **New Layer** from the drop down list. The **Old Layer**: box will display the previous layer used. The example below shows a through-hole pad on [All] layers which is to be changed to [Top].



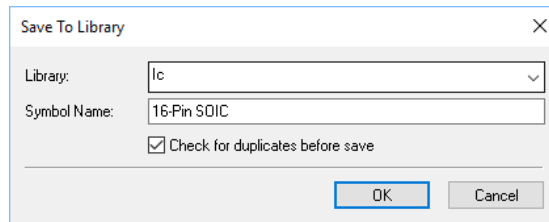
- Click **OK** to make the layer change.
- For SM pads you should make the pad side the [Top] layer. Once the footprint is used in the design, it can be mirrored to the other side of the board where the pads will also flip to the other side as well.
- You can also use the **Pad Properties** dialog to change the pad layer.



7. The **Layer:** drop down list will contain legal layers.
8. When changing a pad from [All] to [Top] layers, you may also need to change the pad style as well. Use the **Change Style** option or **Pad Properties** dialog to do this.

► Saving the symbol

1. From the **File** menu, select **Save To Library**.
2. The **Save To Library** dialog opens.



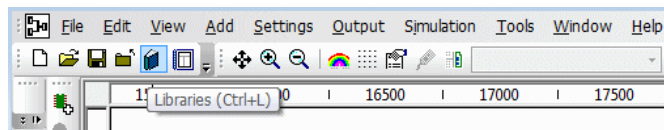
3. Choose the appropriate library from the drop down list.
4. In **Symbol Name:** type the name of the footprint that you wish to use and click **OK**. You have now saved the footprint in the Library.

Creating Components

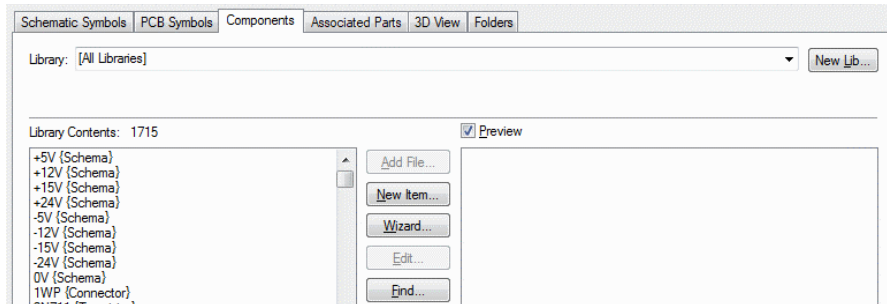
This tutorial uses the Schematic Symbol and PCB Symbol (Footprint) created in the previous tutorials, but equally applies to creating a Component using other existing Schematic Symbols and PCB Symbols.

► To open the Component editor

1. Click the **Libraries** icon on the **File** toolbar.



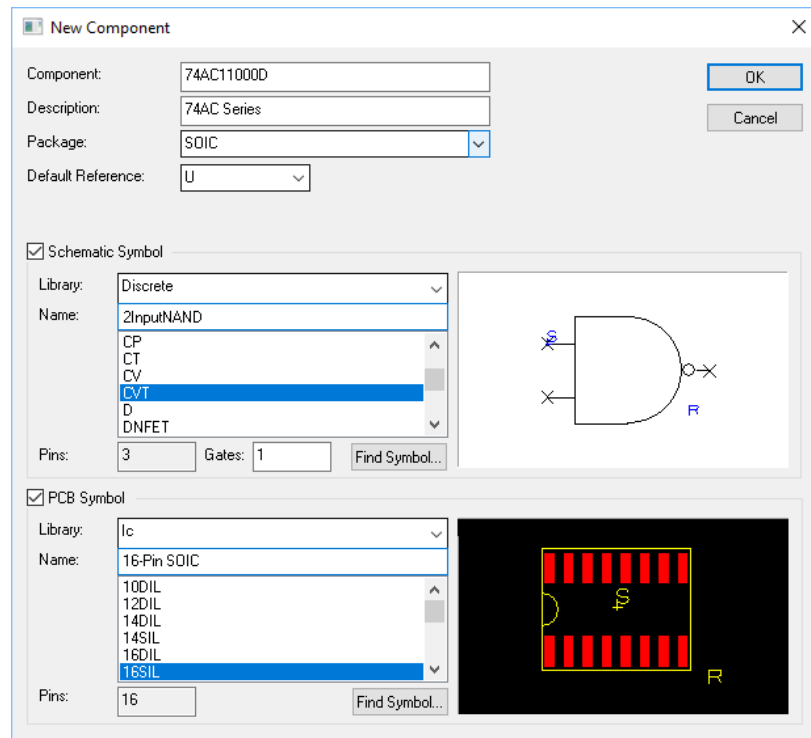
2. From the **Library Manager** select the **Components** tab.



3. Click the **New Lib** button.
4. In **File Name**, type in **Demolib** and click **Save**.
5. From the list of **libraries** available, this will now show **Demolib** and an empty library.
6. We need to create a new component so click the **New Item** button.
7. The **New Component** dialog opens.

8. There are 5 main items of information required.
 - **Component:** is the name of the Component. This is the name used in the Schematic and PCB designs to call the relevant Symbols from their respective libraries.
 - **Package:** this identifies the package type; multiple package types can be used for selecting alternative footprints.
 - **Default Reference:** this is the name stem used in the design, e.g. U, IC. Type **U** into the list box.
 - Schematic Symbol Name.
 - PCB Symbol Name.
9. For the **Component** name, type **74AC11000D**
10. For **Description:** type a suitable name, type **74AC Series**
11. Type **SOIC** for the **Package** name
12. In **Default Reference:** type **U**

13. For **Schematic Symbol**, select the library **Discrete** from the drop down list.
14. From the **Name** list, select your Schematic symbol previously created **2InputNAND**.
The number of Pins: will show **3**.
15. For **Gates**: type in **4**. Your component should contain 4x 3-pin gates plus a power gate.
16. For **PCB Symbol**, select the library **Ic** from the drop down list.
17. Select your PCB symbol **Name**: previously created **16-Pin SOIC**.
18. Ensure the **Pins** box shows **16** to verify that your PCB symbol is correct.
19. Your dialog will be complete and should look like the one below.



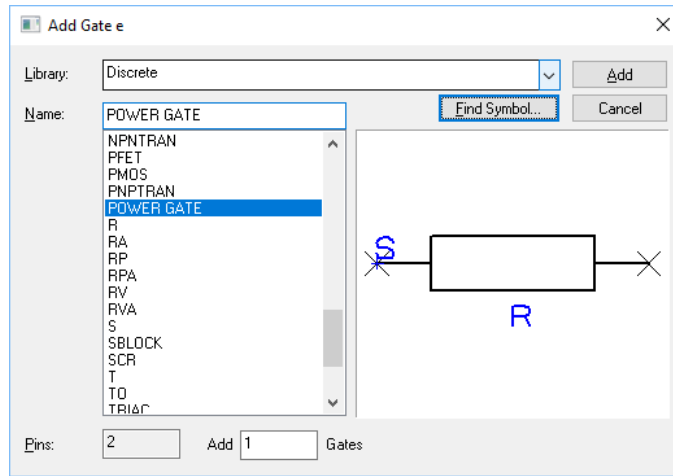
20. Once it does, click **OK** to continue.
21. The **Component Editor** shows the items added from the **New Component** dialog.

Gate Name	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Class) Name
a	2InputNAND		1			
			2			
			3			
b	2InputNAND		1			
			2			
			3			
c	2InputNAND		1			
			2			
			3			
d	2InputNAND		1			
			2			
			3			

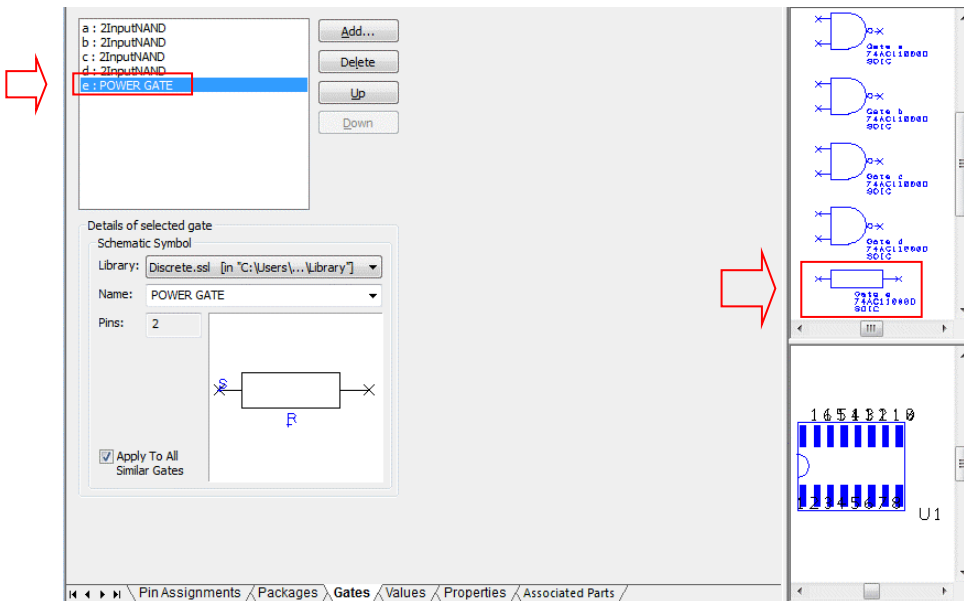
► Adding an extra gate

1. A power gate is required for this component. This is added before allocating pins to the device.
2. Click on the **Gates** tab at the bottom of the **Component Editor** page.

3. The power gate should be added to the bottom of this gates list.
4. Select gate **d**: in the list so it is highlighted blue.
5. Press the **Add** button.



6. Using the Discrete library, choose the POWER GATE symbol from the **Name**: list and click **Add**. Ensure that you only have **1** (Gates) selected in the **Add Gates** box.
7. The power gate name will appear at the bottom of the gates list. A preview of the symbol can also be seen in the gate symbols window to the right of the dialog.



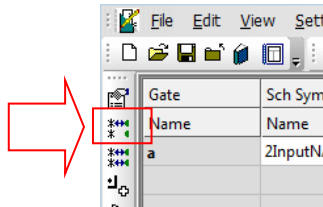
8. Now what we need to do is add more detail to the component and to 'map' the pins between the Schematic symbol and the PCB symbol.

► Mapping pins to pads

For a Component to work, it needs to know how the pins used on the Schematic symbols relate to the pads used in the PCB symbol and to the component when used in the design. These pin assignments are entered into the grid presented.

There are three methods of assigning pins:

- You can directly type the PCB pad numbers for the respective gates
 - You can use the **Assign Pins** mode that lets you click on the Schematic symbol pin first and then the equivalent PCB pad.
 - You can use the **Assign Pins 1-1** option to automatically assign all your pins for you. This is useful for devices such as BGAs and connectors where the pins are a straight 1-1 relationship.
- For our tutorial, we will use the **Assign Pins** option.
 - Click on the **Assign Pins** button on the **Component toolbar** or select it from the **Edit** menu.



- Select **Pin 1** on the **Schematic symbol**.
- Note, you can zoom in or out of the symbol preview windows using the shortcut keys **<Z>** and **<U>**.
- Now select **Pad 1** on the **PCB symbol**.
- The grid above the symbols will display the pin numbers in the **PCB Symbol Pad Number** cell and the **Component Pin Name/Number** cell.
- The illustration below shows how selecting the output pin of the symbol (pin 3) and the pad 2 of the PCB symbol when in Assign Pins mode, will put the pin number 2 into the Pad Number and Component Pin cells.
- It also shows how this has been completed to number 3.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2InputNAND		1		1	1
					16	16
				3	2	2
b	2InputNAND		1			
			2			
			3			
c	2InputNAND		1			
			2			
			3			
d	2InputNAND		1			
			2			
			3			
e	POWER GATE		1			
			2			

9. Back when you created your symbols, you would have added the pins and they would have displayed a pin number. It is this 'real' pin number which you now see in the component editor.
10. You can see that the pin number order doesn't matter because you can map them easily in this editor. If they are numbered logically when creating the symbols, it makes life easier now for you though.

Gate	Sch Symbol	Sch Symbol	Sch Terminal
Name	Name	Terminal Name	Number
a	2InputNAND		1 2 3
b	2InputNAND		1 2

11. You should complete all pin to pad mapping so that looks like this:

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2InputNAND		1	1	1	
			2	16	16	
			3	2	2	
b	2InputNAND		1	15	15	
			2	14	14	
			3	3	3	
c	2InputNAND		1	11	11	
			2	10	10	
			3	6	6	
d	2InputNAND		1	9	9	
			2	8	8	
			3	7	7	
e	POWER GATE		1	4	4	
			2	12	12	

► The Assign Pins 1-1 option

When mapping pins, if the pins and pads do not have a 1:1 relationship then you cannot use this option. **The Assign Pins 1-1** option is particularly useful when there are hundreds of pins!

For example, if we had used this option on our tutorial, the pin mapping would look like this below. As you can see, it is not suitable.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2inputNAND		1	1	1	
			2	2	2	
			3	3	3	
b	2inputNAND		1	4	4	
			2	5	5	
			3	6	6	
c	2inputNAND		1	8	8	
			2	9	9	
			3	10	10	
d	2inputNAND		1	11	11	
			2	12	12	
			3	13	13	
e	POWER GATE		1	7	7	
			2	14	14	

If the mapping is not at a 1:1 ratio, you can edit the grid boxes to assign the corresponding pins to pads.

► Assigning pins manually

To assign pin manually, you should not be in any 'mode'.

You should edit the component grid directly by double-clicking in the **PCB Symbol Pad Number** cell and then typing in the pin number required. This will map the **SCH Terminal Number** in the grey cell to the **PCB symbol Pad Number** that you just typed in. If you make a mistake, double-click again and retype it or press the **delete** key on the keyboard to clear the cell. When editing the grid, the corresponding pins on the symbols will be highlighted to guide you.

► Assigning pins alphanumeric pin names

By manually assigning pin numbers, you can also use this method to give the pins alphanumeric pin numbers as well. This is useful for devices such as BGAs or Connectors which may require this.

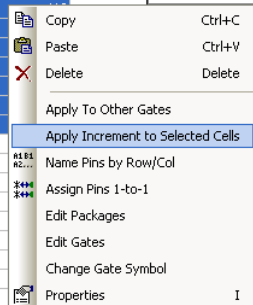
Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Number	Pad Number	Name/Number	Name
1	1	A1	
2	16	A2	
3	2	A3	
1	15	B1	
2	14	AA1	
3	3	3	
1	11	11	
2	10	10	

As you can see in the above example, you can also mix pin numbers with alpha and numeric, and alphanumeric numbers.

► Using Apply Increment to Selected Cells to create alphanumeric names

The **Apply Increment to Selected Cells** option is available on the context menu on a selected cell(s) for **Pin Name/Number** cells in the **Component Editor**. This can be used to name one cell and create an incremental sequence from it to each subsequently selected cell.

Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Terminal Name	Number	Pad Number	Name/Number	Name
MPLIB_CF4	_RESET	1	106	J10	
	XTAL	2	120	K12	
	XTAL	3	108		
	FB_CLK	4	121		
	_RSTOUT	5	144		
	_RCON	6	28		
	DRAMSEL	7	94		
	_BE/BWE0	8	101		
	_BE/BWE1	9	51		
	_BE/BWE2	10	124		
	_JRQ1	11	18		
	_JRQ4	12	30		
	_JRQ7	13	5		
	QSPI_DIN	14	19		
	QSPI_CS2	15	8		
	QSPI_CLK	16	31		
	QSPI_DDOUT	17	7		
	_TA	18	83		
	_UOCTS	19	52	E4	
	_UORTS	20	41	D5	



► To use Apply Increment to Selected Cells

1. Name the first cell, A1 for example.
2. Select the first cell and the next 14 cells by dragging the mouse over them.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	MPLIB_CF4	_RESET	1	106	J10	
		EXTAL	2	120	K12	
		XTAL	3	108	A1	
		FB_CLK	4	121	L1	
		_RSTOUT	5	144	M12	
		_RC0N	6	28	C4	
		DRAMSEL	7	94	H10	
		_BE/BWE0	8	101	J5	
		_BE/BWE1	9	51	E3	
		_BE/BWE2	10	124	L4	
		_IRQ1	11	18	B6	
		_IRQ4	12	30	C6	
		_IRQ7	13	5	A5	
		QSPL_DIN	14	19	B7	
		QSPL_CS2	15	8	A8	
		QSPL_CLK	16	31		
		QSPL_DOUT	17	7		
		_TA	18	83		
		_UOCTS	19	52		
		_UORTS	20	41		
		UORTV	21	120		

3. Select **Apply Increment to Selected Cells** option from the context menu.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	MPLIB_CF4	_RESET	1	106	J10	
		EXTAL	2	120	K12	
		XTAL	3	108	A1	
		FB_CLK	4	121	A2	
		_RSTOUT	5	144	A3	
		_RC0N	6	28	A4	
		DRAMSEL	7	94	A5	
		_BE/BWE0	8	101	A6	
		_BE/BWE1	9	51	A7	
		_BE/BWE2	10	124	A8	
		_IRQ1	11	18	A9	
		_IRQ4	12	30	A10	
		_IRQ7	13	5	A11	
		QSPL_DIN	14	19	A12	
		QSPL_CS2	15	8	A13	
		QSPL_CLK	16	31	A14	
		QSPL_DOUT	17	7	A15	
		_TA	18	83	G11	
		_UOCTS	19	52	E4	
		_UORTS	20	41	D5	
		UORTV	21	120	L10	

4. The **Name/Number** range will now show A1, A2, A3, A4, A5 etc. in the selected cells.

► Using Copy/Paste in the Component Editor

It is possible also to Copy and Paste data between the Component Editor grid and external applications such as Microsoft Excel and vice versa. A single cell or a range of cells can be selected for copying or pasting. It is also possible to paste the same value in to a range of cells.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	MPLIB_CF4	_RESET	1	106	J10	
		EXTAL	2	120	K12	
		XTAL	3	108	J12	
		FB_CLK	4	121	L1	
		_RSTOUT	5	144	M12	
		_RCON	6	28	C4	
		DRAMSEL	7	94	H10	
		_BE/BWE0	8	101	J5	
		_BE/BWE1	9	51	E3	
		_BE/BWE2	10	124	L4	
		_IRQ1	11	18	B6	
		_IRQ4	12	30	C6	
		_IRQ7	13	5	A5	
		QSPI_DIN	14	19	B7	
		QSPI_CS2	15	8	A8	

Using Copy and Paste To Edit Values

The values in the **Pin Assignments** grid can be manipulated using the normal Copy and Paste techniques associated with spreadsheet applications such as Microsoft Excel. This includes the ability to copy and paste multiple rows and/or columns of data. One possible use would be to copy the pin data into an external spreadsheet to take advantage of advanced features such as sorting and filtering, then paste the modified data back in to the Part Editor. When pasting from an external source it is important to ensure that the data is arranged in the same order as the cells in the grid.

The **Copy** and **Paste** commands are available from the context menu. Alternatively, the defined shortcut keys, by default <Ctrl-C> and <Ctrl-V> may be used.

Selecting cells to copy or paste

A single grid cell or a range of grid rows and/or columns may be selected for copy or pasting. Clicking on a single cell will select just that cell. If, rather than clicking, the mouse is dragged while the left button is held down a range of cells may be selected, indicated by being shown in reverse highlight. This range of cells can now be copied or pasted as a whole. If a single cell has already been selected by clicking, it is possible to extend this into a range of cells by clicking with the <shift> key held down. To quickly select an entire column, click in its header cell.

Pasting Values into the Grid

When data is pasted into grid the values are subject to the normal constraints of the column into which they are being inserted and are checked in the same manner as when values are typed in. For example, a value pasted into the **Sch Symbol Name** column will be checked to ensure that it is a valid name of a symbol defined in the library.

The **Gate Name** and **Sch Symbol Terminal Name** columns are read only and as such may not have values pasted into them although they may be selected and copied.

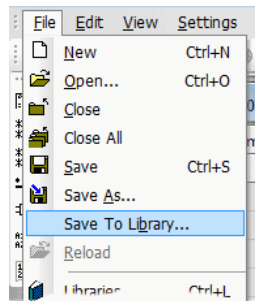
If, when pasting a range of values, a single cell is selected in the grid, the cells affected will match the range of the paste values subject to the constraints of the rows and columns affected.

By copying a single value then selecting a range of cells, it is possible to paste the same value into multiple cells. All cells in the selected range of cells will be set to the pasted value, subject to the column constraints.

All paste operations may be reversed using the standard **Undo** facility.

► Saving your Component

1. From the **File** menu, select **Save To Library**.

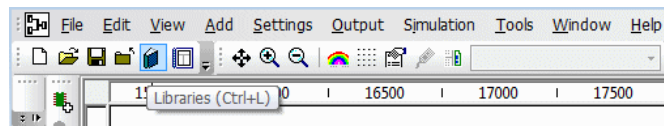


2. The **Save To Library** dialog opens.
3. In **Component Name:** type **74AC1100D** and click **OK**. You have now saved the Component in the Library.

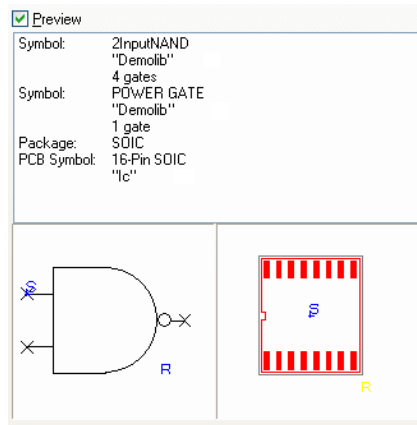
► Viewing the Component in the library

It is always worth checking the component in the Library Manager before saving closing it. The Component Library Manager displays details of symbols and also previews this on selection of the component in the Contents list.

1. Select the **Libraries** icon on the **File** toolbar.



2. Check the **Preview** box.
3. From the **Library Contents:** list select the **74AC1100D** component that you created, it should look like the one below:



This completes the minimum amount of work required constructing a component for it to be successfully used in the **Schematic editor** and passed through into the **PCB layout** using the **Translate To PCB** option.

Adding More Component Detail

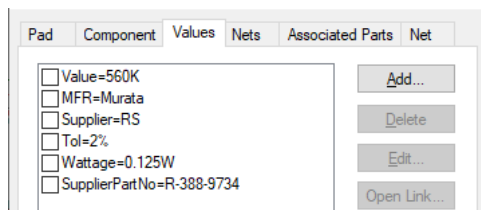
Adding Component Values

Although your basic Component has been saved it can be enhanced further by the use of **Values**. Values are an important aspect of components. They are the means by which, for example, the value, rating, type or tolerance of a component (such as a resistor or capacitor) can be stored. These can be displayed in a Schematic or PCB design.

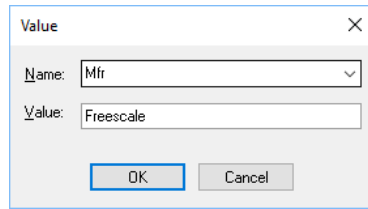
A further and most important application is that values are the means by which part number and value information can be stored for inclusion in Parts lists. It is also the mechanism used for passing essential values to the Spice simulator.

► Adding Values to your Component

1. In the **Component editor** you can add or edit values. Select the **Values** tab.
2. Click the **Add** button to create a new value.



3. From the **Add** dialog you can enter the **Name** and **Value**:



4. Click **OK** to confirm the selection.

Adding and Removing Packages

A component can contain more than one Package (footprint). This may well be the case if different technologies are to be used, e.g. DIL or SMD, and if SMD, then reflow or IR soldering or vapor-phase, all of which could have different pad sizes and hence package types.

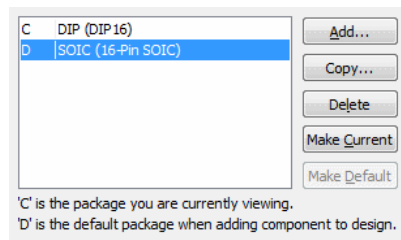
You can set the current package type used by the component using **Package** tab.

► Setting the Current packages

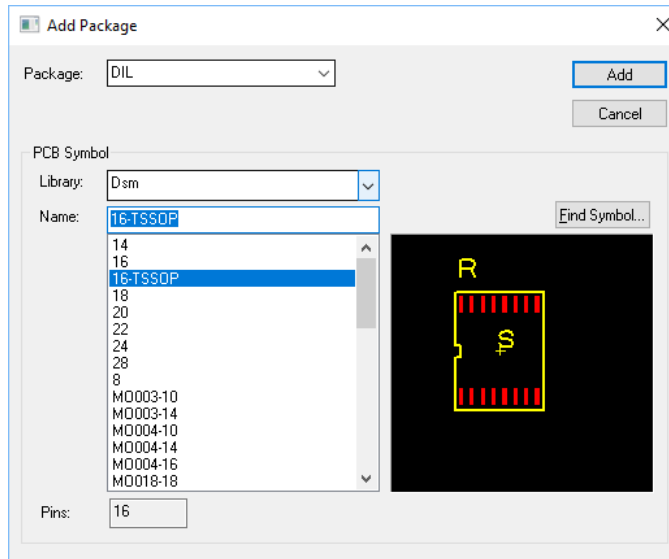
The **Packages** tab is used to add or remove packages for your component. This is an important dialog as it allows you to manage your packages.

Each package in the component can be selected as being the **Default**, the one added first to the design (you can swap to the alternative packages after the first package has been added). This is indicated with a '**D**' on the **Packages** dialog.

The second status it can contain in the component editor is whether it is **Current** or not. The 'Current' status means that this is the package currently selected for being edited. Different values or alternative symbols for example, can be assigned to each package in a component. This provides you with a mechanism for being able to select it in the editor. This is indicated with a '**C**' on the Packages dialog.



To define a new package, use the **Add** option using the **Add** button.

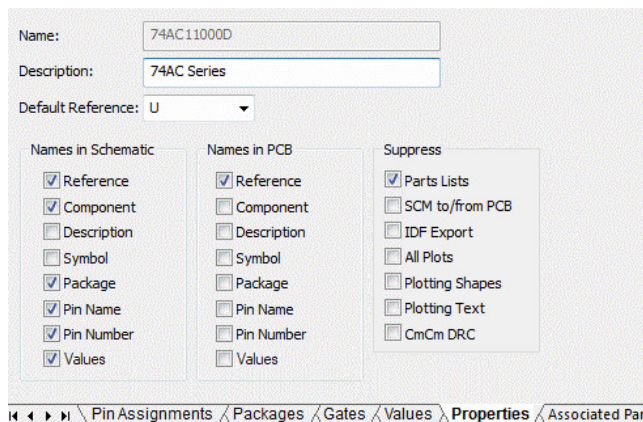


For each Package you add you will need to provide it with a unique Package name. In the above example, we have used **DIL** to define the **16-TSSOP** symbol used on our 74AC11000D component. The preview allows you to make the correct choice. When each symbol is selected in the list, its Pin number is displayed, providing you with additional confirmation of your choice.

Clicking the **Add** button will add it to your component.

► Component Properties

The Component **Properties** tab is used to hold information about the component, such as the display fields for the component in the Schematic and PCB editors.



Component tab

Through this page, you can edit the component description. You should also provide a **Default Reference**, which is the default text added to the front of the name each instance of the component (for example *U* gives *U1*, *U2*, etc.).

There are two columns of check boxes which set the default for what **names** are shown when the component is used in a Schematic or PCB design.

Predefining Terminal Names

You can predefine a terminal name so that when used in the design editors they will be visible and instantly recognisable as to their function or logic. This is usually more applicable to symbols in a Schematic design.

► To define terminal names

1. Use the **Sch Symbol Terminal Name** cell on the grid to edit in the name.
2. Select the cell using a click and type in the name required. To move down to the next cell and to save the previous name, click the down arrow on the keyboard. You can also use **<Enter>** to move down to the cell below.
3. The name is now displayed in the component grid under **Sch Symbol Terminal Name** and adjacent to the selected terminal number.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2InputNAND	A	1	1	1	
		B	2	16	16	
		Y	3	2	2	
b	2InputNAND	A	1	15	15	
		B	2	14	14	
		Y	3	3	3	
c	2InputNAND	A	1	11	11	
		B	2	10	10	
		Y	3	6	6	
d	2InputNAND	A	1	9	9	
		B	2	8	8	
		Y	3	7	7	
e	POWER GATE	GND	1	4	4	
		VCC	2	12	12	

Pre-assigning Net Names

Net Names can be predefined in the component so that when used in the design, the pin automatically has a net name assigned to it. As well as 'normal' components, this method is also extensively used for components such as power and ground symbols.

► To define terminal names

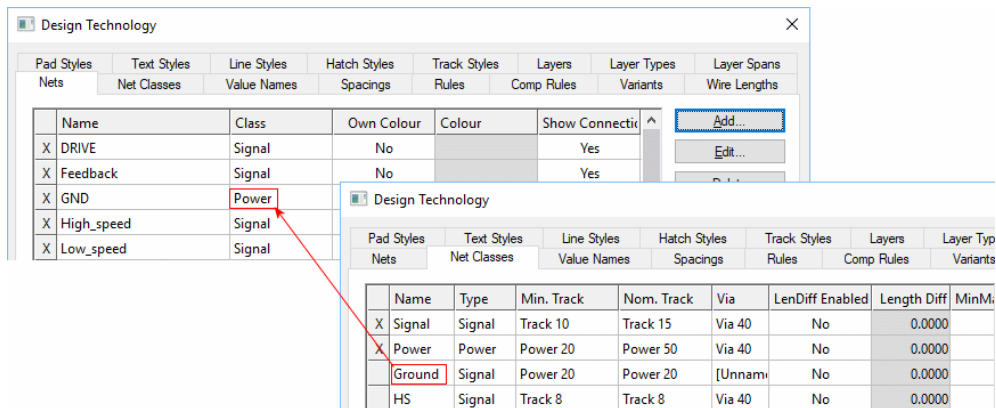
1. Use the **Net (Class) Name** cell on the grid to edit in the name.
2. A Net Name typed into this cell will then be used on the terminal pin in the design.
3. The pin in the Schematic design will be shown as assigned.

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Class)
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2InputNAND	A	1	1	1	
		B	2	16	16	
		Y	3	2	2	
b	2InputNAND	A	1	15	15	
		B	2	14	14	
		Y	3	3	3	
c	2InputNAND	A	1	11	11	
		B	2	10	10	
		Y	3	6	6	
d	2InputNAND	A	1	9	9	
		B	2	8	8	
		Y	3	7	7	
e	POWER GATE	GND	1	4	4	GND
		VCC	2	12	12	VCC

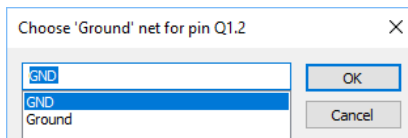
4. If you wish to assign a pin which can be used for multiple net names or where it can be assign using a choice (e.g. Vcc, +5V, +12V), then you can assign this a Net Class name in brackets (), for example (Power).

d	2InputNAND	A	1	9	9	
		B	2	8	8	
		Y	3	7	7	
e	POWER GATE	GND	1	4	4	GND
		VCC	2	12	12	(Power)

5. When the component is used in the design, the **Net Name** and **Net Class** are checked.



6. If more than one exists, all the names with that **Net Class** are displayed in a dialog for selection.

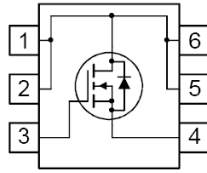


7. You choose the **net name** and click **OK**. The component is added to the design.

Mapping One Schematic Symbol Pin to Multiple PCB Pins

Using the Component editor within the Library Manager, you can map one Schematic symbol pin to multiple PCB footprint pins.

This is best demonstrated with a real example. The example used here is the Fairchild FDC2512 part. The datasheet looks like this:

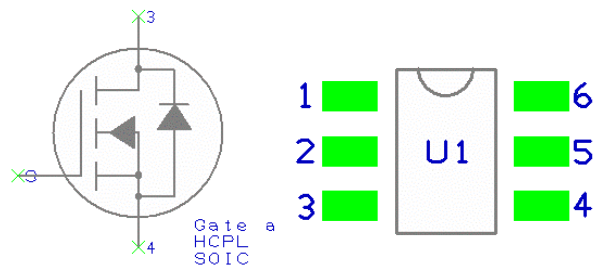


The internal mapping of the PCB footprint means that pins 1, 2, 5 and 6 are common to the one schematic symbol pin. The schematic symbol contains 3 pins overall, the same as the PCB footprint.

In the Library Manager the mapping is done using the **PCB Symbol Pad Number** column. Use a comma character to indicate multiple PCB pin numbers. In the example below, the Schematic symbol pin, pin 1, is mapped to PCB footprint pins 1, 2, 5 and 6. You can also use pin name ranges, such as 1-5,7,9, to map 1,2,3,4,5,7,9.

When this option is used, you must have a letter in the **Component Pin Name/Number** column for these corresponding pins.

Gate	Sch Symbol Name	Sch Symbol Terminal Name	Sch Terminal Number	Pcb Symbol Pad Number	Component Pin Name/Number	Net (Class) Name
a	hcpl		1	1,2,5,6	1,2,5,6	
			2	3	3	
			3	4	4	



Pin Mapping and Connections

There are two formats which can be used for denoting multiple mapped pins; you can use commas and the plus (+) sign.

If a comma is used, the PCB pins will be mapped externally and **Optimise** will show connections to each of these pins, all of the designated pins will be on the same net.

If a + sign is used, the connections are made internally and connections are not shown to each of the pins. However, they will still be connected to the same and with **Optimise** working in the normal way.

Checking and Verifying Component Pin Assignments

Visual and report-based verification is available during the assignment of pins in the Component Editor. This is more essential when handling large pin devices such as BGAs but will also help you with smaller pinned devices.

Pins Report

From within the **Component Editor**, you can select the **Pins Report** on the **View** menu. This lists key information about the pin assignment in the current Component. This includes details of common pins, multiple mapped pins, and any pins that are not assigned.

An example report looks like this:

```
Component Pins Report
-----

Report File:          C:\MicroLib\Rabbit (CML - Component Pins Report).txt
Report Written:       Friday, July 15, 2022
Design Path:          C:\MicroLib\Rabbit.cml
Design Title:
Created:              15-Jul-22 3:28:25 PM
Last Saved:
Editing Time:         0 min
Units:                mm (precision 3)

PCB SYMBOL PIN NAMES APPEARING MORE THAN ONCE
=====
None
COMPONENT PIN NAMES APPEARING MORE THAN ONCE
=====
None
GATE A
=====
Symbol: 101-1006_A
Pins: 64
GATE B
=====
Symbol: 101-1006_B
Pins: 64
End Of Report.
```

Highlight Creation Issues in Component Editor

Within the **Component Editor**, grid cells include small coloured bars at their left edge to highlight issues. These will highlight items such as unassigned, multiple mapped, and common pins. The bar is shown in the left hand side of the cell to which it represents.

Below is an example of the unassigned pins using a red 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	R	A	1			
		B	2			

Below is an example of the multiple mapped pins using a light blue 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	R	A	1	1	1	
		B	2	1	1	

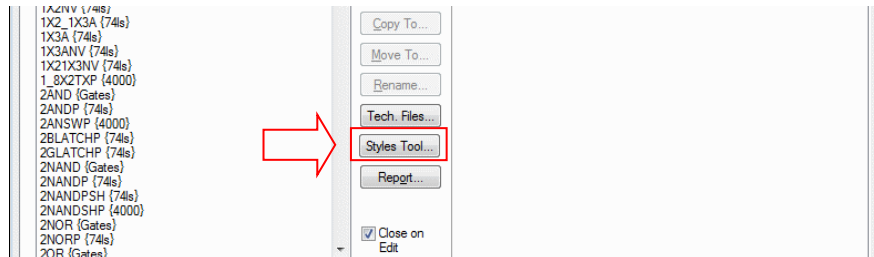
Below is an example of the multiple mapped pins using a double, purple/blue 'bar':

Gate	Sch Symbol	Sch Symbol	Sch Terminal	Pcb Symbol	Component Pin	Net (Clas
Name	Name	Terminal Name	Number	Pad Number	Name/Number	Name
a	2NANDP	A	1	1	1	
		B	2	2	2	
		Y	3	3	3	
		VCC	4	14	14	
		GND	5	8,9	8,9	
b	2NANDP	A	1	4	4	

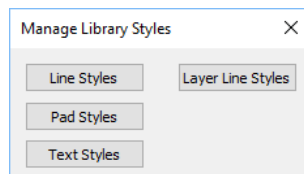
Once the issue has been corrected, the bars are removed dynamically. There is no manual user update to make this process is automatic.

Library Manager Styles Editor

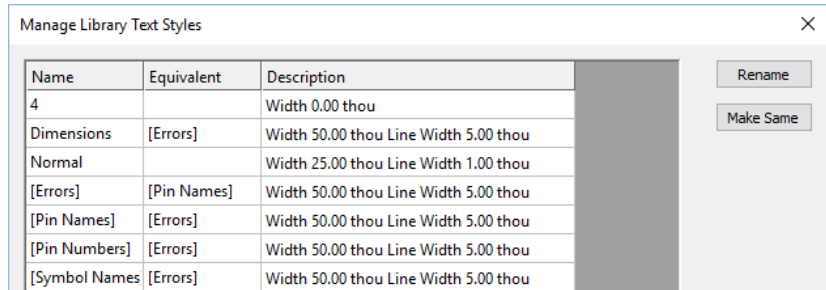
On the **Schematic Symbols** and **PCB Symbols** page of the **Library Manager** dialog, there is a **Styles Tool** button.



When selected, you are presented with the **Manage Library Styles** dialog:



Buttons on this dialog allow you to manage different style styles and **Layer Line** styles. Each dialog will display which styles are used and the ability to rename them or merge them with other styles.



Where a style matches by its value, the style name will be displayed in the **Equivalent** cell next to it.

In the example above, style name **Dimensions**, **[Errors]**, **[Pin Names]**, **[Pin Numbers]**, **[Symbol Names]** all use the same width. This is perfectly acceptable if your wish to differentiate between different items using the same style value or thickness but where common items use different names, you may wish to consolidate them.

Where style names exist but the name is meaningless, for example, Style1 or 4, you can select the style name and press the **Rename** button. Type in the new name, this might be **Pins Names** for example. To consolidate names, you select the cell of the style name you wish to keep. If we wish to keep **Normal**, and make **Line 5** the Normal named style, you would select **Normal** from the list and press the **Make Same** button.

Chapter 12. Managing Libraries

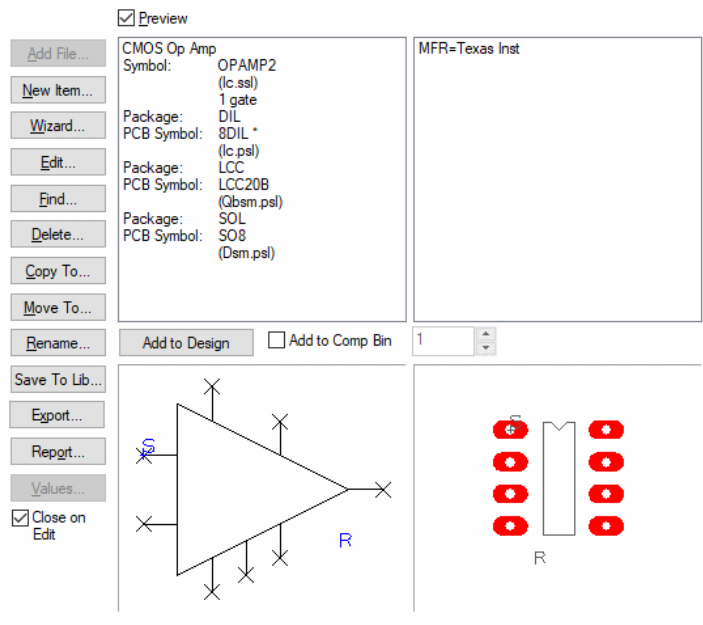
Managing Libraries

The Library Manager

The **Library Manager** is used to manage your libraries. Other than for backing up libraries for security purposes, you must always use the Library Manger for maintenance tasks, tasks such as copying and merging libraries.

Creating Libraries

From the Library Manager, you can create new libraries by selecting the **New Lib** dialog and providing a name in the dialog. The file extension will be supplied in the current library format. The current library selected is displayed in the **Library:** drop down list. You can use this list to swap to another library.



Managing Library Items

Use the navigation buttons in the centre of each of the dialogs to **Add File**, **Create New Item**, **Edit**, **Find**, **Delete**, **Copy**, **Move** and **Rename** library items.

Use the **Copy To** button to make copies of existing library items into other libraries or to the same library but under a different name. Once copied, use the **Delete** button to remove the original library item. This is more secure than providing a Move button as it always retains a copy of the original item.

Adding and Merging Libraries

Libraries may be combined using the **Add** button in the **Library Manager** dialog, you may add one library to another. Besides copying to alternative libraries, this function has an important second function; components can have many different package types, each one corresponding to an alternative PCB footprint. When components are added together, their package types merge. This means that an extra package type for a component can be built and tested using a working library, then only added to a main library after checking.

When two libraries are merged, you are always prompted before a component is changed. You should be aware of components (with the same name) using similar symbols from different libraries as it is easy to inadvertently associate the wrong symbol definition with a component.

Importing Libraries From Designs

New libraries can also be created from designs with the importer ignoring any design information. This means even if you don't have the library used to create an old design or a design from someone else which requires library modifications, you can still get the library back. Each design contains enough library information for which to fully rebuild the component library and appropriate symbol library; Schematic design to Schematic symbols and PCB design to PCB symbols.

If using this procedure, you should always rebuild the symbols first, followed by the component library data.

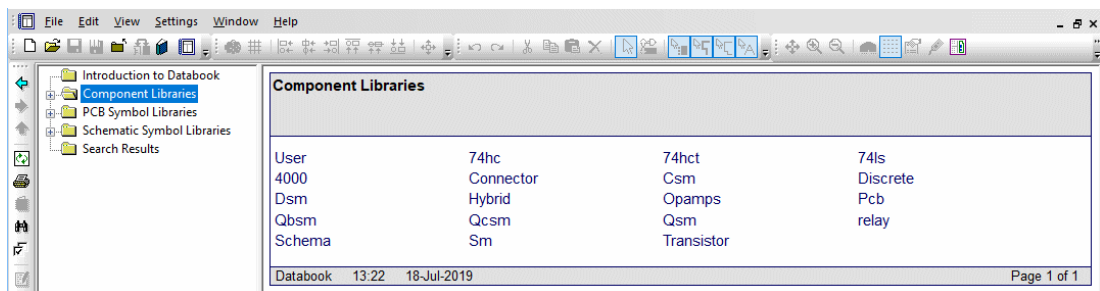
► To import designs as libraries

Use the **Add** button on the **Library Manager** dialog to import the files as libraries. Simply select the design and click **Open** to start the import. The **Library Contents**: list will update with just the library information; all design information will be ignored.

Library Databook Tool

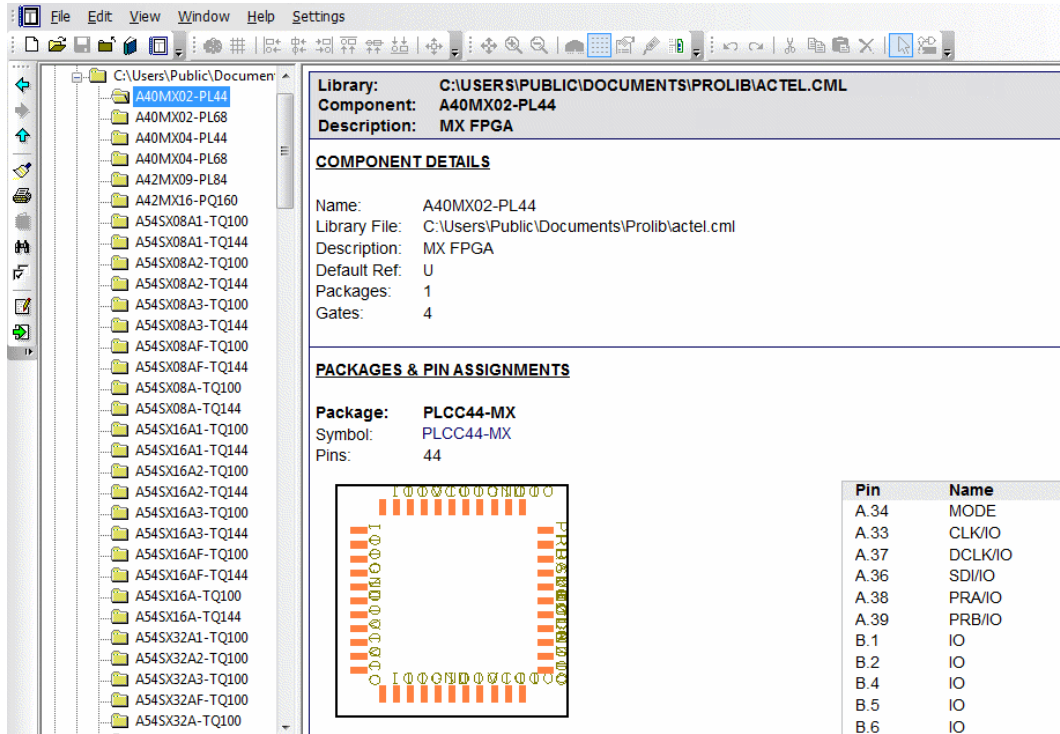
You can produce professional style documentation for your libraries using the Databook option. This provides you with template driven automatic generation of datasheets on-the-fly to keep up-to-date documentation of your library contents.

The **Databook** option is located on **File** menu. The Databook opens as a separate 'top level' window, separate from the application window. The shortcut key <F12> toggles back and forth between **Databook** and the main window.



The Databook window is divided into two sections; on the left-hand side is a ‘tree’ view of the libraries and their contents, whilst the right-hand side is used to display the **datasheet** for the currently selected item.

The picture below shows a component library selected in the left-hand view, and the names of all the items in the library arranged in the right-hand view.



Navigation and management of the Databook items can be done using the toolbar to the left side of the window.

Items found in the Databook can be added to the design using the **Item to Design** button on the **Add** menu (shortcut <Alt-F8>).

More detailed information about this feature can be found in the online help by pressing the <F1> key.

Chapter 13. Libraries

Standard Supplied Libraries

The standard library contains a selection of different components from a range of manufacturers. Supporting these components are their associated Schematic symbols and PCB symbols (footprints). The components cover a broad range to which you can edit or add your own components and symbols.

A word of caution, we **strongly** recommend that no alterations or additions be made to the standard libraries. If you create new libraries of your own, and especially if you modify the standard libraries, copy the Component, Footprints and Symbols from the standard library to your own library. You can do this on a Component by Component basis or do sets of libraries as you require.

Always ensure you backup your libraries as part of your standard operating routings and processes. By retaining the standard libraries as originals, you can always reinstall these if lost for some reason. Your own created libraries are more precious of course.

Errors and Omissions

We have taken great care in the preparation and checking of our libraries, but it is possible that there may be the odd undiscovered error. Please check your work carefully before releasing it for production. The inclusion of a device in this library does not imply that it is available, that it is suitable for new designs, or that all footprints or packages included are available. We cannot accept any responsibility for any loss or damage resulting from the use of any of our libraries.

Please let us know if you do find a problem, or if you have reason to suspect the accuracy of any components.

Using the Standard Library Components

Please note that components in the standard SCHEMA library are Schematic only, and will not translate to PCB. Similarly, components in the PCB library are PCB only, and will give an error if an attempt is made to import them into the Schematic editor.

Although there are no reasons why very large libraries cannot be handled, experience has shown that it is more efficient, and easier to manage a number of smaller ones instead.

Adding Your Own Components

Any of the supplied symbols may be used in your own components. It is always advisable to record the symbols were used where, and, if creating your own symbols, the dimensions used. It is also a good idea to use styles with meaningful names when creating symbols, as it makes it easier to change them if required in a design and if possible reuse existing styles.

Library Definitions

Component Definitions (*.cml)

Each of the sections below is headed **Library:** followed by the library name and a general description of its contents. Each of the library names assumes the file extension of **.cml**

PCB library symbols (*.psl)

These libraries contain the PCB symbols that are used within each of the component libraries.

Schematic library symbols (*.ssl)

These libraries contain the Schematic symbols that are used within each of the component libraries.

Associated Parts Libraries (*.apl)

These libraries contain Associated Parts that can be optionally 'attached' to components within the component libraries. These items add extra detail to a component and are used when the Bill of Materials (BOM) is created.

3D Packages (*.pkg)

These libraries contain the 3D packages used in the 3D Viewer option from within the PCB design editor.

Library Finder

Details of all our standard libraries and their contents can be found on our web site www.numberone.com/libfinder.asp You can also use the **Library Finder** which will allow you to search a database of all the library parts to find the part you require. It will also inform you of the appropriate library that the part can be located in.

SnapEDA Component Browser

As well as the standard and packaged libraries available, a built-in SnapEDA Component Search bar is provided. This allows you to search for Parts from inside Easy-PC and download them to the end of cursor in the design for insertion. This item is also saved to your libraries at the same time for use later on. Please see the online help for further information.

Availability

The SnapEDA Component Search bar is always only available for the current major version of the product.

Component Search Engine Web Site

You must register to use the SnapEDA Component Search bar using the web site:

<http://www.snapeda.com>

Once registered, you will be able to search and retrieve live components without logging in again. The setting is saved inside the option within Easy-PC.

INDEX

- .
- .apl, 328
- .cad, 235
- .cml, 327
- .dxf, 9
- .gbr, 235, 242
- .idf, 9
- .pcb, 17
- .pkg, 328
- .psl, 328
- .sch, 16
- .ssl, 328
- .stf, 97
- .tgz, 235
- 3**
- 3D Packages, 229
- 3D View, 228
- A**
- Absolute origin, 68
- Accepted errors, 215
- Active PDF, 141, 235
- Add block, 128
- Add block port, 131, 135
- Add board, 160
- Add component, 100, 101, 161
- Add component bar, 28
- Add copper pour area, 197
- Add dimension, 206
- Add drill table, 252
- Add pad, 161
- Add pad exception, 72
- Add shape, 137
- Add to net, 117
- Adding connections, 108, 110, 172
- Adding connectors, 106
- Adding corners, 182
- Adding new components, 327
- Adding shapes, 216
- Adding text, 53
- Adding tracks, 175
- All colours black, 244
- Alphanumeric pin names, 310
- Apply increment to selected cells, 310
- Arc, 57, 181
 - tracks, 58
- Auto finish, 190
- Auto neck tracks, 188
- AutoCAD output, 235
- Autogenerate plots, 239
- Autopan, 38
- B**
- Back annotation, 225
- Backing up libraries, 327
- Barcodes, 55
- Barring character, 55
- Best fit, 67
- Blind and buried vias, 155
- Block details, 134
- Block name, 134
- Blocks, 126
- Board
 - outline, 160
 - shape, 218
- BOM composer, 233
- Bus net names, 122
- Busses, 119
- Bypass capacitors, 105
- C**
- Change block symbol, 131
- Change component, 166
- Change net, 111
- Change net class, 173
- Clear copper, 201
- Clearances, 79, 211
- Close all, 60
- Closed busses, 119
- Colour files, 29, 86
- Colour selection flags, 40
- Colours, 85, 99, 156
- Colours bar, 29
- Commands, 24
- Component areas, 218, 221
- Component bin, 28, 101, 139
- Component definitions, 327
- Component editor, 303
 - copy/paste, 312
- Component height checking, 221
- Component libraries, 276, 279, 327
- Component positions report, 231
- Component search bar (SnapEDA), 328
- Component values, 90, 314
 - editor, 94
- Components, 104, 303
 - fix, 49

- pushing, 171
- reloading, 106
- rotate, 169
- updating, 164
- Concept of operation, 15
- Connections, 16, 108, 172
 - styles, 73, 98
- Connectivity, 16
- Connectivity check, 224
- Connectors, 106
- Coordinate system origin, 67
- Coordinates, 68
- Copper
 - pour area, 197, 218
 - pour keepout area, 201
 - pouring, 196
- Copper shape, 218
- Copy, 50
- Copy/paste in component editor, 312
- Corner editing, 182
- Cost options
 - installion, 14
- Creating designs, 15
- Creating layers, 152
- Creepage, 78
- Creepage rules, 78
- Cross probe, 226
- Cross sheet references, 117
- Ctrl select, 38
- Ctrl-Shift select, 39
- Curved tracks, 57
- Customise, 24
- Cut, 50
- Cut to block, 128, 136
- Cutout shape, 218
- Cyclic picking, 40

D

- Dangling connections, 107
- Databook, 324
- Decoupling capacitors, 105
- Default block settings, 132
- Default styles, 148
- Delete, 52
- Deleting blocks, 132
- Design notes bar, 34
- Design options, 42
- Design rules check, 16, 210, 212
 - Schematic, 138
- Design technology, 70
- Designing components, 303
- Dimensions, 206
- Direct PDF, 141, 235
- Disconnect end con, 113

- Disconnect end track, 180
- Display, 156
 - true width, 87
- Display clearances, 189
- Displaying net names, 112
- Dockable windows, 26
- Double-click, 39
- Drag & drop, 46
- Drawing borders, 100
- DRC, 210
 - accepted errors, 215
 - error report, 213
 - online, 191
 - sign-off checks, 213
- Drill drawings, 251
- Drill table, 252
- Duplicate, 50
- Duplicate block, 129
- DXF, 9, 141, 235
- DXF import, 160

E

- Edit block symbol, 130
- Edit menu, 45
- Editing tracks, 181
- End on via, 180, 186
- Enter angle, 179
- Error markers, 212
- Excellon, 235
- Exporting csv files, 85
- Exporting netlists, 140

F

- Fattening, 186
- Fillet, 57, 179
- Filletted tracks, 57
- Find component, 103
- Find dialog, 163
- Finish markers, 115
- Finishing tracks, 182
- Fixed angle, 179
- Flip, 47, 105, 170
 - components, 47
- Flying connection, 175
- Fonts, 54, 73
- Force everything on, 89
- Forward design changes, 223
- Frame, 36
- Frame select, 39
- Free angle, 178
- Free pads, 161

G

- Gerber, 235

Gerber file extensions, 242
 Global styles, 79
 Goto, 30
 block, 132
 component, 30
 design, 31
 error, 30, 213
 Graphical user interface, 19
 Grid the length, 179
 Grids, 64, 172
 snap menu, 67

H

Hardware supported, 11
 Headup display, 80
 Height checking, 221
 Hierarchy, 125
 Hierarchy check, 132
 Hotkeys, 24
 HPGL output, 235
 HTML reports, 231

I

IDF 2.0/3.0 files, 235
 Imperial units, 62
 Implied junctions, 113
 Implied nets, 115
 Import DXF, 160
 Importing csv files, 85
 Installation, 11
 Installing cost options, 13
 Integrity check, 222, 224
 IPC-2581, 235
 Isolated island removal, 198
 Isolation gaps, 198, 256

J

Job files, 18
 Junctions, 113

K

Keepout area copper pour, 201

L

Layers, 79
 bar, 260
 construction, 151
 layers bar, 31
 real life, 154
 spans, 155
 stack preview, 156
 types, 153, 257
 Layout files, 17
 Libraries, 17

Library, 276, 327
 FAQ, 278
 finder, 328
 reference data, 327
 Lines styles, 73, 98
 Load colour files, 86
 Local styles, 79

M

Mapping one scm pin to multiple pcb pins, 318
 Measure gap, 208
 Menus, 20
 Merge shape, 218
 Merged colours, 158
 Merging designs, 124
 Merging libraries, 324
 Merging styles, 74
 Metric units, 62
 Minimum clearance, 210
 Mirror, 47, 170
 Mitre, 57, 179
 Mitre tracks, 57
 Mounting holes, 301
 Moving components, 104
 Multi-line text, 53
 Multiple document interface, 22
 Multi-sheet schematics, 16

N

NC drill files, 235
 Neck tracks, 186
 Necking, 187
 Negative plots, 245
 Net class rules, 79
 Net class to class rules, 78
 Net classes, 75, 111, 149
 Nets
 busses, 122
 joining, 116
 names, 75
 naming, 111
 optimisation, 174
 optimise, 174
 renaming, 116
 New component, 304
 New PCB, 145
 Next error, 212
 Notes, 34
 Numeric keys, 36

O

ODB++ output, 235
 Online DRC, 191
 Only plot items, 245

Open busses, 119
Open sheet, 118
Optimise nets, 174
Optional features, 13
Origins, 68
Orthogonal tracks, 178
Outputs for manufacture, 235

P

Pad exceptions, 258
Pad styles, 71, 148
Page links, 118
Pan, 37
Panel editor, 18, 242
Parts List, 233
Paste, 50
Paste nets, 51
PCB only library, 327
PCB symbol library, 276
PDF output, 235
Pen plotter output, 235
Photoplot output, 235
Pick and place report, 231
Pin mapping, 319
Pin styles, 98
Pins report, 320
Place, 46, 167
Placing SMDs, 170
Placing the PCB, 167
Plane & items plot, 246
Plot isolation, 245
Plot type, 241
Plots, 238
Plotting, 235
 devices, 238
 files, 18
 layers, 243
 settings, 243
Polygon select, 39
Pop out of block, 129
Positioning components, 104
Pouring copper, 196
Power & ground symbols, 117
Power planes, 254
Predefined pin names, 317
Previous error, 212
Printing, 235
Printing schematics, 142
Profiles, 159
Projects, 59, 123, 126
 overview, 16
Properties, 43
 bar, 35
 bus, 121

 connections, 109
 values, 44
Protected vias, 186
Pull tight routing, 190
Push into block, 129

R

Real life layer, 154
Redo, 45
Reference origin, 90, 289, 290
Rel / Abs, 69
Relative coordinates, 67
Relative origin, 67, 68
Reload, 60
Reloading components, 106
Remove teardrops, 205
Renaming nets, 116
Report browser, 231
Reports, 230
Resist layer, 257
Resolving net names, 125
Rotate, 48, 105
Routing area, 218
Rubberbanding, 169
Rulers, 26

S

Save, 61
Save all, 60
Save As, 61
Save technology, 61
Save to library, 61
Saving components, 313
Saving designs, 61
Schema library, 327
Schematic design flow, 97
Schematic files, 16
Schematic only library, 327
Schematic symbol library, 276
Schematics plots, 264
Scroll bars, 37
Security copy, 62
Segment mode, 110
Select, 38
Select from list, 41
Selectability flags, 40
Selection methods, 38
Set origins, 68
Shape information bar, 34
Shapes, 137
Shift select, 39
Shortcut keys, 24
Shortcut keys report, 25
Show clearances, 189

- Show headup display, 81
 - Show powerplane, 260
 - Sign-off checks, 213
 - Sliding windows bar, 26
 - Slotted pads, 72
 - SMDs, 170, 302
 - Snap grid, 67
 - Snap mode, 110
 - Snap to angle step, 179
 - Snap to grid, 184
 - SnapEDA interface, 328
 - Solder resist
 - layer, 257
 - plots, 256
 - Spacings, 79, 211
 - rules, 149
 - Special effect layer, 154
 - Split planes, 203
 - Split powerplanes, 203, 263
 - Splitting schematic nets, 113
 - Status bar, 21
 - Step & repeat plotting, 242
 - Stub routes, 185
 - Styles, 71
 - editor, 321
 - local & global, 79
 - merging, 74
 - Surface mount devices, 170, 302
 - Swap layers, 189
 - Symbol libraries, 279
 - System origin, 67
 - System reports, 230
 - System stroke fonts, 54
- T**
- Teardrops, 203
 - Technology files, 18, 70
 - Tented vias, 261
 - Text
 - barred, 55
 - fonts, 54
 - rotate, 53
 - styles, 53, 73, 98
 - Text callouts, 207
 - Thermal
 - pads, 256
 - relief, 198
 - spokes, 200
 - Toolbars, 22
 - customising, 23
 - docking, 23
 - floating, 23
 - Tooltips, 22
 - Track
 - adding, 175
 - editing, 181
 - fattening/necking, 186
 - necking, 187
 - stubs, 185
 - styles, 74, 149
 - Track length rules, 79
 - Translate to PCB, 139
 - True type fonts, 54
 - True width on/off, 87
- U**
- Undo, 45
 - Unfinished connections, 107
 - Units, 62
 - status bar, 21
 - Unpour copper, 201
 - Unrouting tracks, 183
 - Update component, 105, 164
 - Update drill table, 252
 - User defined parts list, 233
 - User interface, 19
 - Using busses, 119
- V**
- Values, 89, 105, 314
 - Values tooltips, 92
 - Vias, 155
 - protected, 186
 - View all, 37
 - View menu, 36
 - View powerplanes, 260
- W**
- Windows bar, 26
 - Windows output, 235
 - Working area, 67
 - World view, 33