

Seetrax

Computer Aided Engineering

Seetrax XL Designer

Reference Guide

About this manual

This manual is under constant revision, please check for updates from our website at: www.seetrax.com
It corresponds with version 2.25 – refer to the readme.txt in the ..\Seetrax\XLDesigner folder for later revision changes.

This manual is intended as a reference guide and describes each command in detail.

A separate manual contains a tutorial for new users and this can be viewed/printed when Seetrax XL Designer is running, from the Help command.

There are also videos available from our web-site for some topics: http://www.seetrax.com/xld_videos.htm

This manual is divided into chapters. The first chapter will describe general information, whilst remaining chapters will be laid out in “folder” order as seen in the Navigator pane. So for example, the chapter describing the Artwork appears after the chapter on the Board Profile.

If you have any queries or comments about this manual, please don't hesitate to let us know – contact details below.

Produced 16 September 2019

All brand and product names are trademarks or registered trademarks of their respective companies.

Seetrax have taken every effort to ensure the accuracy of this manual. If you should find any discrepancies, please notify us as soon as possible. If there are any inaccuracies, ambiguities or omissions in this document, Seetrax CAE and its consultants and distributors cannot accept responsibility for any loss or damage these errors may cause.

Seetrax CAE Ltd reserve the right to alter the specification.

Seetrax CAE Ltd, Woodstock, Hangersley Hill, Hangersley, Ringwood, Dorset, BH24 3JP, England

Telephone: 01425 489666 (overseas callers: + 44 1425 489666)

Fax: 01425 461641 (overseas callers: + 44 1425 461641)

Web-site: www.seetrax.com

Email: sales@seetrax.com
support@seetrax.com

Copyright Seetrax CAE Ltd 2019

Contents

Licensing Information	1
To use a License Key Code:	1
Store the License (a one off per machine operation):	1
Activate the Licence:	2
Updating to Windows 10:	2
De-activating a Licence:	2
Multiple License Counts:	2
Working in two or more locations:	2
What happens if my hard disk fails?.....	2
License Details:	2
Configuration Information	4
Initial Installation information.....	4
Sharing configuration/master libraries.....	4
Future updates	4
To access the System Setup	4
Configuration Paths (validated at startup)	5
Miscellaneous Configuration Paths	6
Installed Optional Products	6
Special Function keys	6
Design File Options	6
Fonts	7
Text report formatting	8
Temporary fixes.....	9
Diagnostics.....	9
Product License.....	10
Commands Common to All Editors	11
View commands.....	11
View > Pan	11
View > Zoom In	11
View > Zoom Area.....	11
View > Zoom Out.....	11
View > Previous View.....	12
View > Refresh	12
View > Toolbar	12
View > Status bar	12
View > Navigator/Browser/ Properties.....	12
View > Restore Default Layout > Main Frame Layout.....	13
View > Restore Default Layout > Editor Layout	13
Grid commands.....	13
Grid > On/Off	13

Grid > Inch.....	13
Grid > Metric.....	14
Grid > 0.1"	14
Grid > 1mm.....	14
Grid > User Defined.....	14
Grid > Snap to Grid	14
Grid > Select origin.....	14
Grid > Edit origin.....	14
Grid > Clear origin	14
Grid > Measure Distances.....	14
Window commands.....	14
Window > Cascade	14
Window > Tile Vertical.....	14
Window > Tile Horizontal.....	14
Edit commands	15
Edit > Units	15
Edit > Display Adjustments.....	15
Configuration Folder	18
Configuration folder – Colours	19
Configuration folder - Part prefix codes	20
To view/change the part prefix codes:.....	21
Effects of altering part prefix codes	21
Configuration folder - Size Tables, Pad Sizes	23
Master/Design Size Tables	23
To view/change the sizes table:	23
To restore the design’s pad/track sizes table to the master settings:.....	24
Notes about pad sizes:.....	24
Configuration folder - Size Tables, Heat-relief and anti-pads	25
To view/change the heat-relief and antipad sizes:	26
To restore the design’s pad/track sizes table to the master settings:.....	26
Configuration folder - Size Tables, Track sizes	27
To view/change the Track Sizes:	27
To restore the design’s pad/track sizes table to the master settings:.....	27
Configuration folder - Size Tables, Miscellaneous Line Widths	28
To view/change the Line widths:	28
Configuration folder - Manual routing parameters	29
To view/change the Manual routing parameters	29
Configuration – Rip Retry Autorouter costs	32
To view/change the Rip Retry Autorouter costs	32
Rip Retry Autorouter Setup Window	32
Configuration Folder - CNC Tools & Data Sets	40

CNC Tool Parameters	40
Drilling & Routing Datasets	41
Configuration folder - Power Names	44
Viewing/Editing the Power Names List.....	44
Configuration – Layer Assignments & Ordering.....	45
To view and/or change the layer assignments and ordering for a design:	45
Layer Assembly Order.....	49
Set empty copper layers to silk-screen	50
Notes for users of all previous versions of Ranger	50
Additional notes for Ranger1/Ranger2 users.....	50
Configuration folder – Via Hole Definitions	52
Points to Bear In Mind If Considering Using Blind/Buried Vias	52
Board Manufacture for Boards with Blind/Buried Vias	52
Drill set names.....	54
Drill/pad sizes for blind/buried via.....	54
Board layers - Assembly	54
Defining layer order within Seetrax XL Designer.....	54
Defining Blind/Buried Via stacks - Examples	55
Defining signal vias	64
Saving via stacks for use on other designs	64
Routing with user-defined vias	64
Blind/buried via colours and point count.....	65
Outputs	65
Configuration folder – Grid Autopitch Table & Axis Dot Limit	68
Grid autopitch table	68
Log Files Folder.....	69
Typical log files:.....	69
Documentation Folder	70
Creating a new documentation file.....	70
Importing a documentation file.....	70
Customising the file types available.....	70
Creating a Template for the file type	71
Standard Pads Folder	72
Custom Pad Folder.....	74
Opening/viewing the custom pads folder and its content.....	74
Custom Pad Information	74
Creating a custom pad, general guidelines.....	74
Individual commands in the custom pad editor.....	75
Right-click on custom pad folder.....	75
New - Creating a new custom pad	75
Paste	75

Right-click on a custom pad.....	75
Open.....	75
Cut.....	75
Copy.....	75
Delete.....	75
Rename.....	75
Datums commands.....	75
Tools > Set Pad Datum.....	75
Datums > Set Connection Point.....	76
Tools > Outline Definition.....	76
Tools > Slots & Extra Holes.....	76
Outline commands.....	76
Outline > Add Line.....	76
Outline > Add Arc.....	76
Outline > Add Rectangle.....	77
Outline > Corner.....	77
Outline > Adjust Arc.....	77
Outline > Delete Feature.....	77
Outline > Delete Point.....	78
DrillHoles commands.....	78
DrillHoles > Add.....	78
DrillHoles > Move.....	78
DrillHoles > Delete.....	78
Vias78	
Schematic Folder.....	80
Differences between Flattened & Hierarchical Schematics.....	80
Appearance.....	80
Connectivity.....	80
Old flat style schematic to hierarchical schematic conversions.....	80
Accessing the schematic tools.....	81
Right-click on the Schematic Folder, or Items within Sub-Folders.....	81
New Folder – applies to the Master Schematic folder only.....	81
New - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	81
Open - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	81
Copy - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	81
Paste - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	82
Delete - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	82
Rename - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	82
Attributes – schematic folder only.....	82
Purge Unused Blocks – schematic folder only.....	83
Edit Outline Names – schematic folder only.....	84

Edit Datasheet Addresses – Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol	84
Show Datasheet - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol	84
Show Dependancies - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol.....	84
Change Block Type – Split parts or Primitives	84
Contents of the schematic folder	85
Design sheets.....	85
Design Symbols	85
Parts	85
Split Parts	85
Primitives.....	85
Block I/O Ports	85
Commands available when a design sheet or part is open:	86
Edit > Preferences.....	86
View commands, schematic editor only	89
Structure commands	89
Identify commands	91
Find commands.....	92
Errorflag commands	93
Tools commands, schematic editor.....	93
Symbol commands.....	105
Wires commands.....	113
Alloc commands	120
Nonelec commands.....	122
Region commands	127
Commands available when a part, split part, primitive or block I/O is open.....	128
Tool bar commands.....	128
Outline commands	129
Terminals commands	131
Powerpins commands	135
Attributes	136
Subsymbol commands	142
Schematic configuration file – “schema_attribs.txt”	143
Content of schema_attribs.txt.....	143
Description of the entries within <i>schema_attribs.txt</i>	145
Adding User-Defined attributes as defaults.....	147
Component Outline Folder	149
Opening/viewing the component outline folder and its content.....	149
Component outline information	149
Viewing direction (from top or bottom)	149

Pads	149
How modifications to outlines in use on the artwork are implemented	150
Component outline families - introduction	150
Additional "copper"/free copper - introduction	150
Stringers	151
Dual outlines	151
Saving an outline	151
Creating an outline, general guidelines	151
Individual commands in the component outline editor	152
Right-click on component outline folder	152
New - Creating a new component outline	152
Paste	152
Manage Families	152
Edit Properties	154
Purge Unused Outlines	154
Right-click on a component outline	154
Open	155
Cut	155
Copy	155
Delete	155
Rename	155
View commands, outline editor	155
View > Show Outline Filled	155
View > Show Pin Numbers	155
View > Show Connection Points	155
View > Properties	155
Dialogue bar settings, outline editor	157
Pad Style	158
Pad Angle	158
Pad Size	159
Pin Scope	159
Next Pin Number	159
Outline/Copper Width	159
Free copper layer	160
Region Operations	160
Tools > Outline Definition	160
Tools > Slots & Extra Holes	160
Outline commands, outline editor	160
Outline > Add Line	160
Outline > Add Circle	160
Outline > Add Arc	161

Outline > Corner	161
Outline > Move Point	161
Outline > Adjust Arc.....	161
Outline > Delete Feature	162
Outline > Delete Point	162
Outline > Change Width	162
Outline > Text	162
Outline > Text > Add	162
Outline > Text > Move	163
Outline > Text > Edit.....	163
Outline > Text > Mirror	163
Outline > Text > Rotate	163
Outline > Text > Delete	163
Pad commands, outline editor	164
Pad > Add.....	164
Pad > Auto Add	164
Pad > Move	165
Pad > Key Move	165
Pad > Rotate	165
Pad > Change Style/Size	165
Pad > Delete.....	165
Pad > Assign	166
Pad > Auto-Assign.....	166
Pad > Describe.....	166
Copper commands, outline editor	167
Notes:	167
Copper > Add Line	167
Copper > Add/Move Corner	168
Copper > Add Keepout Line	168
Copper > Add Keepout Circle.....	169
Copper > Delete Point.....	169
Copper > Delete Line	169
Copper > Adjust Arc/Circle	169
Copper > Add Pad	170
Copper > Move Pad	170
Copper > Key Move Pad	170
Copper > Rotate Pad.....	170
Copper > Change Pad Style/Size.....	170
Copper > Describe Pad	171
Copper > Delete Pad.....	171
Copper > Free Copper Setup	171

Region commands, outline editor	171
Region > Move/Rotate.....	172
Region > Copy.....	172
Region > Delete.....	172
Autoplace commands, outline editor	172
Autoplace > Adjust Footprint	172
Creating and using Starpoints.....	173
What is a Star point?	173
Creating a starpoint outline	174
Using the starpoint outline	174
Logos Folder.....	175
Important Notes:	175
Creating the logo.....	175
Copying logos	175
Deleting logos	175
Renaming logos	175
Selecting multiple Logos	175
Editing/opening a Logo	175
Thresholding.....	177
Vector Generation	177
Parts & Nets List Folders.....	179
Viewing the list of parts or nets	179
Deleting the parts list	181
Cross-probing tools.....	181
Making Nets visible/invisible	181
Displaying particular nets (unroutes).....	182
Hiding nets (unroutes).....	182
Typing in a parts list.....	183
Modifying the parts list.....	184
Repeating parts	184
Parts list fields.....	184
Outlines are highlighted - typical causes of invalid entries in parts list editor	186
Correcting spelling mistakes	186
Correcting non-existent outlines.....	186
Typing in a net list.....	186
Modifying the wiring list	187
Net list fields.....	187
Nodes are highlighted - typical causes of invalid entries in net list editor	188
Correcting Pin 0's	188
Correcting non-pin 0's	189
Single node nets	189

Single node nets can be located as follows:	189
Unused pins list.....	190
BOM (Bill of Materials) output	190
Individual commands in parts/net list editor	190
File commands.....	190
File > Save Parts (or Nets) List As Text	190
File > Print	190
File > Close Parts (or Nets) List	190
Edit commands	190
Edit > Units	190
Edit > Find Part.....	190
Edit > Repeat Part.....	191
Edit > Allow Position Edit.....	191
Edit > Find Node.....	191
Edit > Find Signal Name.....	191
Edit > Find Unconnected Pins	191
View commands.....	191
View > Show Part UID's	191
View > Show Net UID Codes	191
View > Minimum Clearance.....	191
View > Bill of Materials	191
Right-clicking the Parts or Nets Folders.....	192
Delete All Parts and Nets	192
Delete All nets	192
Board Profile	193
Opening the Board Profile Editor (design or master)	193
Creating a board profile – general guidelines (incl. keepouts).....	193
Adding the board profile	193
Adding the keepout outline(s).....	194
Individual commands - board profile editor	194
Right-click on board profile/profile library in the navigator	194
New	194
Open.....	194
Copy	194
Paste	194
Delete	194
Rename.....	195
View Commands – specific to Board profile editor	195
View > Part Pins	195
Profile commands	195
Toolbar “Edit Mode” setting	195

Profile > Add Line	195
Profile > Add Arc	195
Profile > Add Circle.....	196
Profile > Add Rectangle.....	196
Profile > Corner	196
Profile > Move Point	196
Profile > Adjust Arc.....	197
Profile > Transfer to current layer.....	197
Profile > Delete Point.....	197
Profile > Delete Feature	197
Profile > Numeric Editor	198
Profile > Set XY Display Datum.....	200
Artwork Folder	201
Artwork	201
Right-click on Artwork	202
Open.....	202
Open Detached	202
Delete Tool	202
Optional autorouters	203
View commands.....	203
View > Artwork Flipped.....	203
View > Part Labels	203
View > Lines At Width	203
View > Pads Filled.....	203
View > Drill holes.....	203
View > Isolated Copper	204
View > Net Clearances.....	204
View > Negative Space Shaded.....	204
View > View Control Toolbar	204
View > Properties	212
View > Restore Default Layout > Main frame layout.....	213
View > Restore Default Layout > Editor layout.....	213
Identify commands.....	213
Identify > Part	213
Identify > Pin.....	214
Identify > Pad	214
Identify > Track.....	215
Identify > Feature	215
Identify > Track Measure.....	216
Tools > Placement & Routing command.....	216
Autoplacement (Tools > Autoplace).....	216

Autoplace > Control Orientation	218
Autoplace > Replace Part.....	219
Autoplace > Place Parts	219
Autoplace > Replace All	219
Auto-routing (Tools > Autorouter)	220
Preparations for auto-routing.....	220
Accessing the auto-router	221
Router dialogue bar.....	221
Copper Fill (Tools > Copper Fill).....	223
Boundary > Add Line.....	227
Boundary > Delete Line.....	227
Boundary > Corner	227
Boundary > Delete Point	228
StartFill > Set Datum	228
Start Fill > Connect To Datum	228
StartFill > Connect to Net.....	228
FilledCopper > Delete All.....	229
FilledCopper > Delete Area	229
FilledCopper > Delete Node	229
FilledCopper > Delete All Isolated Copper	229
FilledCopper > Save Copper	229
FilledCopper > Merge Copper	229
FilledCopper > Delete Saved Copper.....	229
Power planes (Tools > Powerplane).....	229
What is a power plane?	229
What is a Split Power Plane?	230
Power plane layers.....	231
Creating a power plane, or split power plane - basic procedure	231
Designs with split planes from earlier versions:	234
Adding text, logos, extra clearance areas, etc. to the power planes	235
Power plane commands	235
SplitBoundary commands	239
SplitBoundary > Add Line.....	239
SplitBoundary > Delete Line.....	240
SplitBoundary > Corner	240
SplitBoundary > Delete Point	240
SplitBoundary > Import Line	240
Powerplane commands (Setup & Generate).....	241
Slots & Extra Holes (Tools > Slots & Extra Holes commands)	241
Known limitations with this feature (in v2.25)	242
Autoplace dialogue bar.....	217

Opening older jobs (prior to v2.23).....	242
Slots & Extra Holes Dialogue bar	242
Slot Definition > Add Line	247
Slot Definition > Add Arc	247
Slot Definition > Add Circle.....	247
Slot Definition > Corner	248
Slot Definition > Move Point	248
Slot Definition > Adjust Arc/Circle.....	249
Slot Definition > Delete Feature	249
Slot Definition > Delete Point.....	249
Breakout commands	249
Slot Definition > Add Breakout	249
Slot Definition > Move Breakout.....	250
Slot Definition > Delete Breakout	250
Extra Holes commands	250
Extra Holes > Add	250
Extra Holes > Move.....	250
Extra Holes > Delete	250
Network commands	250
Tools > Network > Add Part	251
Tools > Network > Test Points Setup.....	251
Tools > Network > Test Points	252
Tools > Network > Add Link	253
Tools > Network > Delete Pin.....	253
Tools > Network > Move Node.....	253
Tools > Network > Pin Swap	254
Tools > Network > Set Net Name.....	254
Silk-screen	254
Silk screen information	255
Free layers	255
Generating the silk screen data and/or "free layer" data.....	255
Modifying the silk-screen and/or free copper	257
Modifications to completed designs	257
Tear drop pads.....	258
Creating the teardrops.....	258
Deleting teardrops	258
Parts commands	258
Parts > Place	258
Parts > Unplace.....	260
Parts > Move	260
Parts > Key Move	261

Parts > Rotate	261
Parts > Flip	261
Parts > Swap	262
Parts > Align X/Y	262
Parts > Highlight	263
Parts > Find	263
Parts > Change Outline	263
Parts > Gate Swap	263
Parts > Pin Swap	264
Parts > Dynamic Powernets	264
Parts > Reconnect Power	264
Parts > Auto Renumber	265
Parts > Manual Renumber	267
Parts > Status	268
Parts > Set Datum	268
PartFix commands	268
PartFix > Fix/Unfix Selected Parts	268
PartFix > Fix/Unfix All Parts	268
PartFix > Fix By Partcode	268
PartFix > Unfix By Partcode	269
PartFix > Enable Unflipped Parts	269
PartFix > Enable Flipped Parts	269
PartFix > Show Fixed Parts	269
Mroute commands	269
Mroute > Corner	271
Mroute > Enable Unroute Reconnect	273
Mroute > Move Segment	273
Mroute > Layer Swap	273
Mroute > Delete Point	274
Mroute > Convert Arc	274
Mroute > Fix 45	274
Mroute > Fix 90	274
Mroute > Neck	274
Mroute > Convert to Track	275
Mroute > Ripup	275
Mroute > Move Track	276
Mroute > Move Unroute	276
Amend commands	276
Amend Dialogue Bar	277
Amend > Enter Pads	278
Amend > Enter Tracks	278

Amend > Enter Arc Tracks	278
Amend > Enter Circle	279
Amend > Corner	279
Amend > Move Point	280
Amend > Delete Point	280
Amend > Delete Pad	281
Amend > Delete Track.....	281
Amend > Move Pad.....	281
Amend > Rotate Pad	281
Amend > Replace Pad	282
Amend > Set Size.....	282
Text commands	282
Text > Add	283
Text > Move.....	284
Text > Rotate.....	284
Text > Mirror	284
Text > Edit	284
Text > Add URL Anchor	284
Text > Display URL	285
Text > Delete	285
Text > Change Width.....	285
Text > Change Height	285
Text > Get Label.....	285
Logo commands	286
Logo > Place	286
Logo > Move.....	287
Logo > Rotate.....	287
Logo > Mirror	287
Logo > Edit	287
Logo > Copy	287
Logo > Delete	287
Region commands	288
Region > Show Rules.....	288
Region > Move/Rotate.....	289
Region > Copy.....	289
Region > Delete.....	289
Region > Unplace.....	289
Region > Macro Generate	290
Region > Macro Place	290
Deleting/Renaming macros	290
Check commands	291

Check > Angles 45 OK	291
Check > Angles No 45.....	291
Check > Connectivity.....	291
Check > Count Unroutes	299
Check > Next ErrorFlag.....	300
Check > Delete ErrorFlag.....	300
Check > Delete All ErrorFlags	300
Netfix commands	300
NetFix > Selected Nets.....	300
NetFix > By Pin Reference	301
NetFix > By Signal Name	301
NetFix > All Nets.....	301
NetFix > Fix Mode	301
NetFix > UnFix Mode.....	301
Hilight commands	301
Hilight > Selected Nets	302
Hilight > By Pin Reference	302
Hilight > By Signal Name.....	302
Hilight > Clear All.....	303
Specctra auto-router interface	304
Starting the router.....	305
Specctra configuration window.....	306
Electra auto-router interface	308
Electra router setup	308
Starting the router.....	309
Electra configuration window.....	310
Importing Files	312
Gerber Format File Import	313
Information:	313
Procedure:	313
Gerber Photoplot Importer window, <i>Setup</i> tab window settings:.....	313
Gerber Photoplot Importer window, <i>Dcode</i> tab window settings:.....	314
Viewing/editing the imported artwork	316
Typical problems when importing gerber files.....	316
The data cannot be seen when the artwork editor is opened:.....	316
All the pads appear to be too big, and too close together:	316
All the pads appear to be too small, and too far apart:.....	316
Some pads are too small, or too big, or the wrong shape:.....	316
Sample Gerber file	316
Regenerating a design from Gerber files	317
Import Assumptions:.....	317

Gerber file content and implications for importing:	318
Additional Data required to regenerate a design:	319
Verifying the design:	319
AutoCAD DXF File Input	320
Information:	320
Procedure:	320
Parts & Wiring List Import	322
Expected file format:	322
Parts list file/section.....	322
Power Rails file/section	323
Wiring list file/section.....	324
Example file containing all sections.....	324
Information:	325
Procedure:	325
Typical Warnings/Errors	327
Viewing/editing the imported parts/wiring list.....	327
Calay parts/wiring lists import	328
Expected Calay file format:	328
The Calay parts list.....	328
The Calay Wiring list	328
Useful Information:	329
Procedure:	329
Typical warnings/errors	331
Viewing/editing the imported parts/wiring list.....	331
Outputs Folder - Tasks	333
Outputs Folder - Tasks	333
What can be output	333
Creating a New Output Task.....	333
Renaming a Task.....	333
Copying an Output Task	334
Deleting a task	334
Opening a task.....	334
Layer selections	334
Data types/categories.....	334
Typical layer selections:	334
Configure Commands	335
Configure > Plot Source (with an Artwork Layers Task open).....	335
“Layer n1-n2” pages	337
Configure > Plot Source (with a Solder Mask Task open).....	338
Layer pages.....	339
Configure > Plot Source (with a Solder Paste Task open).....	341

Configure > Plot Source (with a Drill Sheet Task open).....	342
Configure > Plot Source (with a Schematic Task open).....	344
Configure > Output Filter	345
Configure > Output Filter - Photoplotter selected	346
Information on gerber files & Dcodes	347
Configure > Output Filter – 2D/3D Drawing selected	347
Configure > Output Filter – Windows Printer selected	350
Printer selection.....	350
Configure > Output Filter – Image File selected.....	351
Positioning and scaling the output data	351
Making the output visible.....	352
Configure > Move Plot.....	352
Configure > Keymove Plot.....	352
Configure > Rotate Plot.....	352
Configure > Mirror Plot	352
Configure > Scale Plot.....	352
Configure > Auto Centre.....	353
Plot commands	353
Plot > Execute	353
NC Drill & Rout Data Task	353
Producing the NC Drill/Router output files.....	353
Over-lapped drill holes	355
IDF Output Task.....	355
Producing the IDF Version 3 output files.....	355
Outputs Folder - Batches.....	358
Creating a batch process	358
Renaming a batch process	358
Copying a batch process	358
Deleting a batch process	358
Opening a batch process	358
Adding/removing Tasks from the batch process	358
Specifying filenames for the resultant output files	359
Specifying the folder for the resultant output files	359
Running the batch process or an individual task from the batch process	359
Saving the batch process	359
BSL (Bath Scientific Ltd) output files	360
Creating the Bath Scientific output file	360
Output file format.....	360
GenCAD output.....	361
Creating the GenCAD text file	361
Output file format.....	361

Licensing Information

Once XL Designer has been installed on a computer, a license key code is required to make it work. If you do not have the license key code, then please let us know.

To use the license key code, you will need to be a registered user on the XL Designer website (<http://www.xldesigner.com/downloads>) and know your login name (email address) and password (of your own choosing).

The license key can be *stored* on multiple machines, then *activated/deactivated* on each one as required from the *License Configuration* window.

Internet access is required during license activation/deactivation, but it is not required to *Store* the key or run XL Designer once it has been activated.

The license key code contains your Customer ID, the Number of systems that have a maintenance contract, the Date the license expires and the Date the maintenance contract ends at which time future updates will not be available.

The *License Configuration* window can be opened from the *Help > License Configuration* command (or the *File > System Setup* window, in which case select “*Click to Configure*” from underneath the *Product License* details).

The license configuration window is shown in Figure 1. (On a new system, the license details will be empty.)

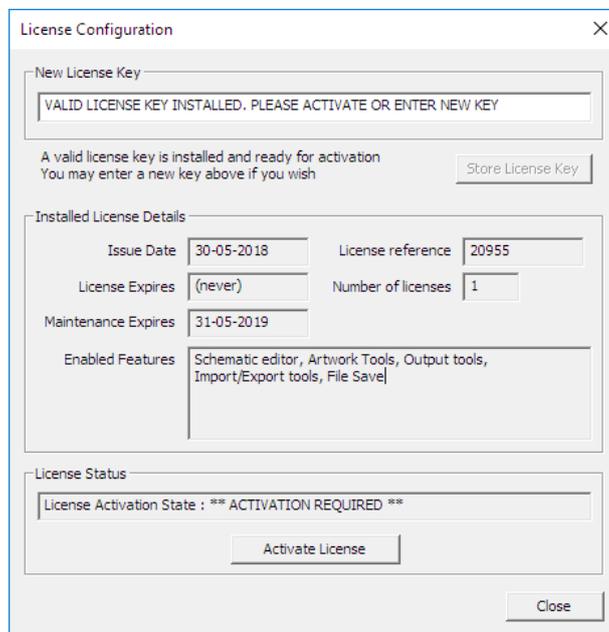


Figure 1

For security reasons, the license key code is never displayed.

To use a License Key Code:

Store the License (a one off per machine operation):

Storing the license key code can be repeated, now or in the future, on any machines that will run the software. It only needs to be done once and does not require internet access.

It does not “activate” the license key.

Assuming XL Designer V2.10 or later is installed, run up XL Designer. A window will appear requesting the license key. (If it doesn't, select *Help > License Configuration*.)

Copy and then paste the License Key Code, into the “*New License Key*” area of the window.

Select “*Store License*”. If the License Key is invalid an error message will appear, otherwise the license details will be shown in the window – ensure they match those in the license details that came with the license key code.

Once the License Key Code has been *Stored*, an “*Activate License*” button will appear at the bottom of the window.

Activate the Licence:

Assuming the license key has been stored as described above, open the *License Configuration* window if it isn't already open, then select the "Activate License" button from the window.

A new window appears. This should indicate that the "License Server Status" is "OK". If it isn't, then check there is an Internet connection established on your pc.

The window also shows the "Number of activations available". If 0 are available, it means all the licenses are in use and one will need to be deactivated before you can proceed.

Enter the email address and password you used to register on the XL Designer web-site. (Links in the window allow you to register or obtain a new password if required.)

Then select "Activate Now". A message will indicate the license has been *Activated*. Select OK/Close to continue. The software can now be used without an Internet connection until it is required to deactivate the license, for example if you wanted to use the license on another machine.

Updating to Windows 10:

If **updating** from Windows 7 or 8 to **Windows 10**, it is important that you **deactivate** the license **prior to the update**. If you don't, the license cannot be used and you will encounter delays and inconvenience whilst you contact us to request that the license is de-activated.

De-activating a Licence:

Once a License Key code has been activated, it has to be de-activated before it can be used elsewhere.

Run up XL Designer, select *Help > License Configuration*.

Ensure the "License Server Status" is "OK". If it isn't, then check there is an Internet connection established on your pc.

Select the "De-activate License" button from the window.

A new window appears, select the "Deactivate Now" button to proceed with the deactivation. A message will indicate the license has been De-activated.

Select OK/Close to continue.

Multiple License Counts:

Activation of a license reduces the number of available licenses (as defined in your license details) by one on the Seetrax License Server. Once zero licenses are available, an activated license would have to be deactivated in order for it to be used elsewhere.

Working in two or more locations:

"Store" the license key (as described above) on the machines you are likely to use when you are ready – it remains dormant until it is activated. You do not need to *Store* it again.

Once activated, it must be de-activated before it can be activated on another machine.

What happens if my hard disk fails?

You must de-activate the key in order to use it elsewhere, however hard-disk failures are difficult to predict! If the disk can be re-used, even if it has been re-formatted, you will be able to de-activate the license once XL Designer has been re-installed.

If the disk is not recoverable, then please contact your local Seetrax Support Office and they will assist in the recovery of the license key.

License Details:

The license details will be provided in the following format:

```
Customer:           Your Company Name
Serial Number:      1
Created:            08-19-2015
License Key:        ABCDE-FGHIJ-KLMNO-PQR12-STU34-VWXY
License Count:      1
License Expires:    31-12-2099
Maintenance Expires: 08-10-2016
Product Options:    Schematic, Artwork, Output Tools,
                   Import/Export Tools, File Save
```

<u>Customer:</u>	This will be your name or company name.
<u>Serial Number:</u>	The serial number of your license key.
<u>Created:</u>	The date and time the license key code was generated.
<u>License Key:</u>	This is the license key that has to be entered into XL Designer's License Configuration window. In this example the following would be copied/pasted: <div style="text-align: center;">ABCDE-FGHIJ-KLMNO-PQR12-STU34-VWXY</div>
<u>License Count:</u>	indicates the number of systems that were purchased and is the maximum number of machines that the software can be activated on at any one time.
<u>License Expires:</u>	indicates when the software will stop working. For example a 30 day license may be supplied for evaluation purposes, or whilst an invoice is outstanding. A permanent license (til 31-12-2099) will be provided once a purchase has been completed.
<u>Maintenance Expires:</u>	all the updates that became available whilst the maintenance period is current can be used with the license. If the expiry date is reached and the maintenance has not been renewed, then all subsequent updates will not work. The last update will continue working until the license expiry date. When the maintenance is renewed a new license will be supplied and once stored/activated the software can be updated. The old license key should no longer be used and will be deleted from our servers after a grace "changeover" period from renewal. So please ensure license key codes are replaced in a timely fashion to avoid any inconvenience and down-time once they are deleted.
<u>Product Options:</u>	indicates which options are enabled with the license. Currently a fully functional XL Designer has 5 product options as shown in the example above. If the option is not listed, then it is not enabled. For example a "Schematic Only" license would just show "Schematic, Output Tools & File Save".

Configuration Information

When XL Designer was installed it came with a set of defaults for file locations, auto-saves, etc. This information can be viewed and changed if required.

Initial Installation information

The supplied configuration information & default libraries will be installed into the following structure:

```
<TARGETDIR>/UserTemplate/Data  
<TARGETDIR>/UserTemplate/Libraries/PCB  
<TARGETDIR>/UserTemplate/Libraries/Schematic  
<TARGETDIR>/UserTemplate/DocTemplates
```

After installation, XL Designer never writes to the configuration files and libraries in this UserTemplate structure.

At initial startup of XL Designer, a personal configuration and libraries structure is created in the user's Documents directory, and this will be seeded with files from the UserTemplate structure.

The new personal files structure is configured as follows:

```
MyDocuments/Seetrax/XL Designer/Artmacros  
MyDocuments/Seetrax/XL Designer/Data  
MyDocuments/Seetrax/XL Designer/Libraries/PCB  
MyDocuments/Seetrax/XL Designer/Libraries/Schematic  
MyDocuments/Seetrax/XL Designer/DocTemplates
```

The seeding of any directory from the above structure will only occur if the directory is found to be missing, or is empty. Pre-existing non-empty directories will never have their contents overwritten with files from the UserTemplate area.

The *File > System Setup* window in XL Designer may be used to choose different locations for the various directories.

When a computer is shared between multiple users, the setup window allows individuals to edit/pollute their personal configurations and libraries without affecting other users of the system.

Alternatively, it's possible to configure the system to allow the master libraries/configurations to be shared amongst multiple users.

As long as these directories are not empty, files from the UserTemplate area will never be copied over the top of the shared libraries and configuration.

System managers will be able to produce their own sets of configurations and design libraries and copy them to UserTemplate area to be used as the initial defaults for all users.

Sharing configuration/master libraries

With XL Designer running, select *File > System Setup* and ensure that the *Configuration Data directory*, *PCB Masters* and *Schematic Masters* directory paths point to the shared content. As long as these directories are not empty, files from the UserTemplate area will never be copied over the top of the shared libraries and configuration files.

Please note that when running on Vista/Windows 7, administrative action will need to be taken to ensure that all users have appropriate access permissions to the shared content.

Future updates

When performing reinstalls or future upgrades, the *Custom Install* option permits the user to exclude installation of the default configuration and libraries. This would be desirable when the UserTemplate area had been customised.

Product *Uninstall* will never remove any files from the <TARGETDIR>UserTemplate structure.

Previously configured paths to user's configuration and library directories will be honoured, and never overwritten.

To access the System Setup

Select *File > System Setup*. A window similar to the one shown in Figure 2 appears.

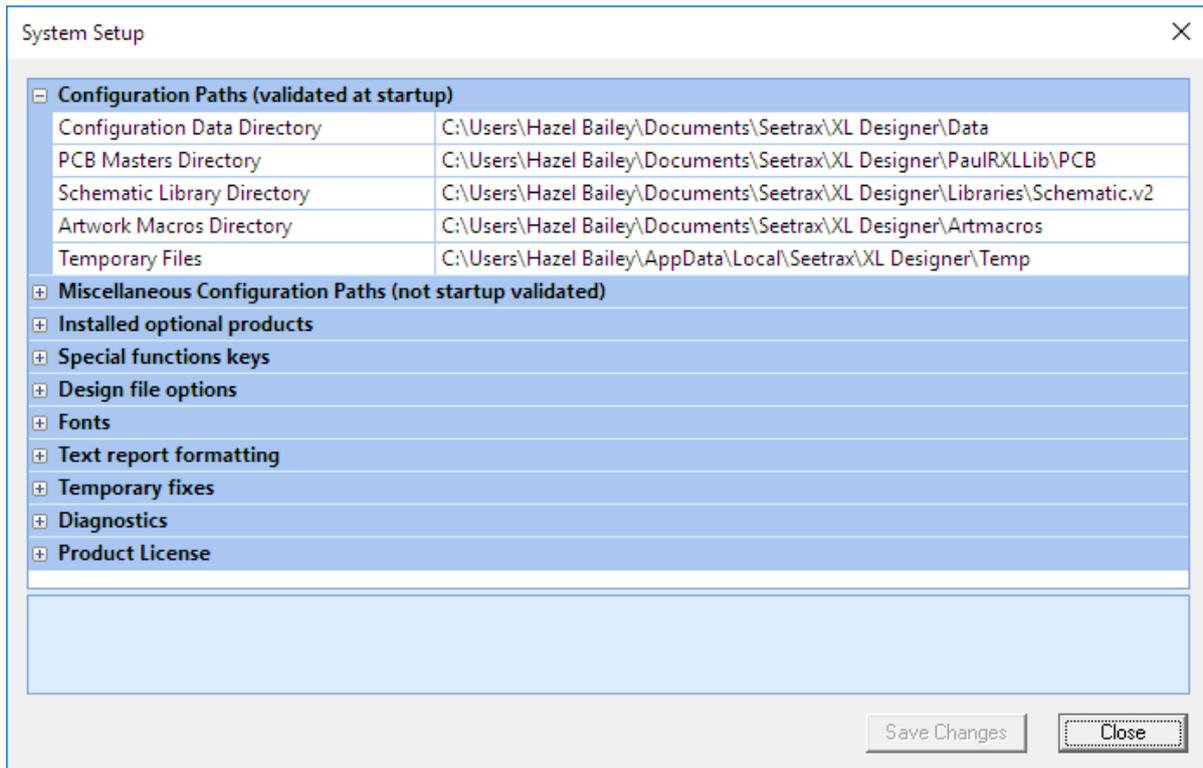


Figure 2

Each heading, like the *Configuration Paths* in Figure 2 can be expanded to display and allow editing of the values within. Each heading is described below.

If a change is made, select the *Save Changes* button to implement the changes, or the *Cancel* to abandon the changes (all of them). Only changes to the Configuration Paths require XL Designer to be closed and restarted – as indicated in the window.

Configuration Paths (validated at startup)

These paths are validated when XL Designer is run to ensure they exist and the required files located. These paths can be changed provided the full implications of the change are understood. For example moving/copying associated files.

If these paths are changed, then XL Designer has to be closed and restarted for the changes to take effect.

The browser buttons () will appear on the right-hand side of the entry when a folder location is selected and can be used to navigate around the directory structure.

Configuration Data Directory: indicates the path that should be used to locate the configuration files. If the path is changed then all the files that are found in the existing folder should be moved to the new folder.

PCB Masters Directory: indicates the path that should be used to locate the PCB library files which are: *outlines, datalib, profile and pads.*

If the path is changed, the files must be moved to the corresponding location (or the folder will be populated automatically from the user template).

The path may point anywhere on a network.

Schematic Library Dir: indicates the path that should be used to locate the schematic library files. There are 77 supplied library files with names reflecting the content of the files, such as 74xx, Capacitors, Resistors, etc., plus any files that have been created by the operator.

If the path is changed, the files must be moved to the corresponding location (or the folder will be populated automatically from the user template).

The path may point anywhere on a network.

Artwork Macros Dir: indicates the default path that will be used when saving and retrieving artwork macros. (It is possible to specify different folders when actually saving/retrieving the macro if required.)

Artwork macros are automatically given an extension to their name of *.amf* (artwork macro file).

Temporary Files: this indicates the place where temporary files are stored whilst XL Designer is running.

Miscellaneous Configuration Paths

These paths are NOT validated when XL Designer is run (so might not exist). You do not need to close/restart the program to make the change.

Text I/O Directory: indicates the default path that will be used when importing or exporting files. (When importing/exporting files, the Windows browser is also available to locate other folders as required.)

Generic Extractors: indicates the path that will be used to locate the generic extraction tools that are accessed from the schematic editor. If the path is set incorrectly, then the extraction tools will not appear and cannot be used.

Installed Optional Products

There are three products that can be used as integrated packages within XL Designer: the Cadence/CCT Spectra Autorouter, the Konekt Electra Autorouter and the Racal Redit Autorouter.

If any of these products are available, tick the appropriate box to enable the interface to that product within XL Designer. The interfaces are located by right-clicking on the *Artwork* in the Navigator, then selecting *Installed Optional Autorouters*.

Special Function keys

When working in the graphical editors, keyboard keys can be used to perform certain commands to make the design process quicker. For instance when zooming in and out, switching the grid on/off, etc.

Some of the keys were defined during installation and all of them can be changed.

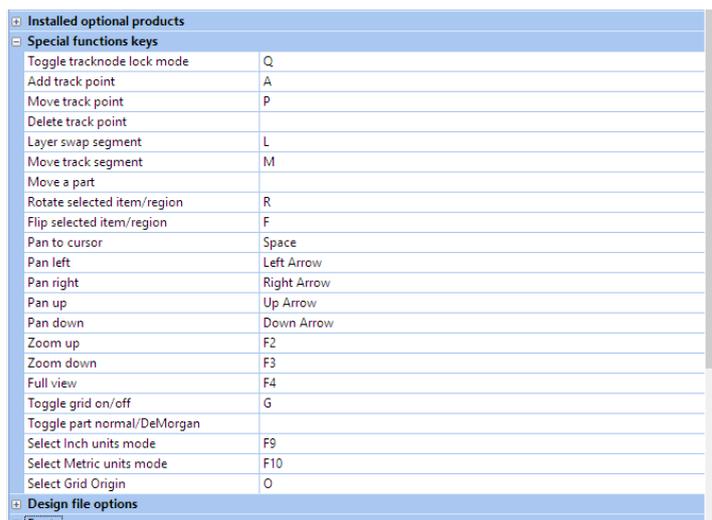


Figure 3

To change the key assigned to a command, select the current setting, then type in the new character. The keys are not case-sensitive so whether you work with CAPS LOCK On or Off makes no difference. Select *Save Changes* to close the window and implement all the changes.

Design File Options

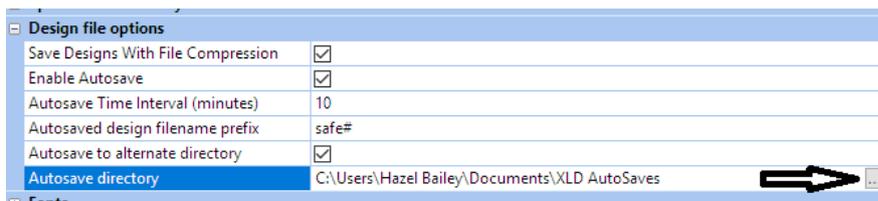


Figure 4

Save Designs With File Compression: Some designs can be too large to send via email, or take too long to save over a network, so it is possible to switch on file compression so that the design file is made smaller.

Tick the box to enable file compression. When the design is saved it will be compressed.

Compressed and uncompressed designs can be opened, regardless of this setting.

Enable Autosave: An auto-save feature can be switched on (recommended) to allow automatic backup saves of the design to be taken at specific time intervals.

Autosave Time Interval (minutes): Length in minutes between auto-saves (if auto-save is enabled).

Auto-saved design filename prefix: The user can choose to overwrite the open design with the autosave, or to save the design with a different name (recommended).

If this field is left empty, the design will be auto-saved using the current design file name, so the file is over-written.

If a prefix is specified (*safe#* by default), this will be inserted at the front of the design file name before saving, so a separate backup copy is created.

Auto-save to alternate directory: When checked, autosaved designs can be stored in a different directory to the one in which the design file is located.

To select a destination folder, select the entry, then select the browse button (arrowed in Figure 4 above). Navigate to the folder required, or create a new folder.

For data security in the event of file system corruption or hardware failure, it is advisable to store autosaved work on a different file system to the one that holds the design file.

Fonts

The font used in the schematic editor, associated output tasks and printed text reports can be configured. The schematic editor can display all text using either a fixed pitch vector font or a selected Windows text font. Most Windows fonts use proportional spacing, so text strings will be slightly shorter in overall length than with the fixed pitch vector font. Text height remains the same and is still controlled by the height properties on attributes, non-electrical text items etc.

It is unadvisable to change between the two fonts due to the difference in text string length – so make a choice and stick with it. Older Ranger designs will have been designed with the vector font.

Editors & Output Tasks: If the vector font setting is not in use (see below) then the schematic editor will display all text using the selected Windows text font.

To change the font, select the setting so it highlights as shown in Figure 5, then select the browser button that appears on the right-hand side of the window (arrowed in Figure 5), choose from the options given.

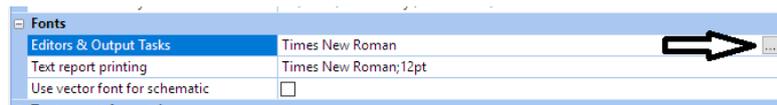


Figure 5

Because text height is specified by attributes within the editor, the 'point size' available when choosing the font is fixed at an arbitrary 16pt and has no bearing on the resultant text size.

With outputs such as PDF, this results in the text being searchable in the resultant document.

The onscreen preview of an output task automatically switches the display of text between Windows font and vector font, as determined by the capabilities of the output device. For example the Gerber output can only render text as vectors.

The output to image files (bmp, jpeg, gif, png) continues to output all text as vectors.

Artwork text is always vector based in the editor and outputs because of the strict requirements of dimensional control and clearance checking.

Text report printing: All printed reports (parts lists, bom's, extraction reports, etc.) will be output using the font and size selected. They have a consistent appearance and include a page header, footer and page number/total page count.

(The content of the header/footer left,centre & right text fields may be configured under the “Text report formatting” section of the system setup window.)

Printed text reports which have information organised into columns (like the parts list) automatically adjust column widths to obtain the best fit of the report for the paper size in use. When a large font point size is selected for text printout, some columns will need truncation to fit the output onto the page. The column sizing algorithm will resize the columns such that the minimum number of text strings are truncated in the printout.

Truncated text will be marked with a small red bracket at the point where it is truncated.

Use Vector font for schematic: When ticked, causes the schematic editor to use the vector text font.

Text report formatting

Printed text outputs now have a consistent appearance and can include a page header, footer and the page numbers with a total page count.

The content of the header & footer and its position can be configured from within the Text Report Formatting area of the System Setup window as shown in Figure 6:

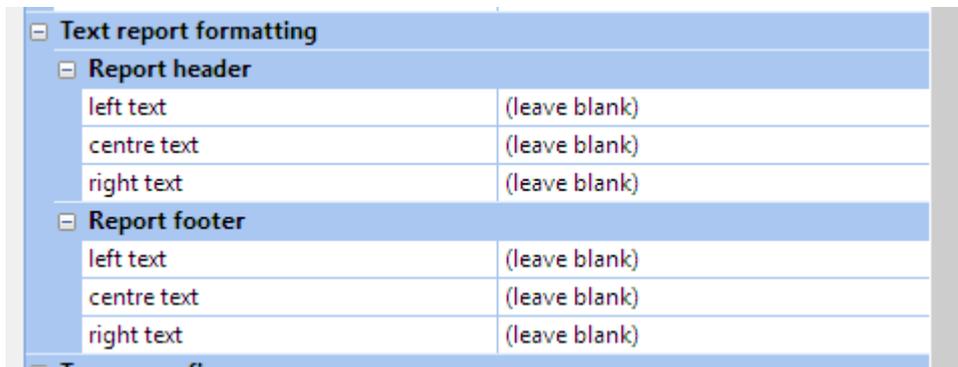


Figure 6

To alter any of the entries, first select the item to be altered so that it highlights – *left text* is highlighted in Figure 7, this will produce a selector for a pull down menu, shown arrowed in Figure 7

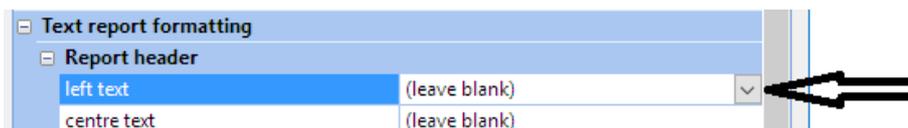


Figure 7

Select the pulldown menu and make your choice from the options given as shown in Figure 8.

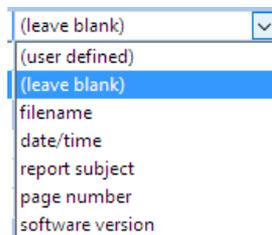


Figure 8

If the *user-defined* option is chosen, type in the required text string in the empty field that opens up underneath the setting once selected, arrowed in Figure 9.

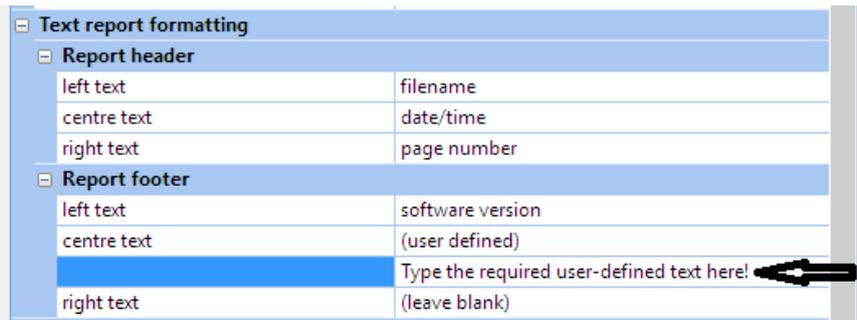


Figure 9

Temporary fixes

Reverse printed arc:

when printing a schematic and using *some* Windows printer drivers, arcs can be printed the "wrong way round". If this happens, tick the box. Arcs will be reversed on output and printed the correct way round.

Enable schematic block type change:

On older versions of the software, under certain circumstances after fetching a part into a design, the type of the part in the source library was inadvertently set as a *split part*. To assist in correcting the wrong block type on designs and libraries affected by this bug, a command "Change Block Type" menu has been added to the "right-click" context menu of schematic split parts in the navigator. This command is only available when this check box is ticked.

Note: the *Change Block Type* command is only intended for recovery of corrupted designs, so for this reason XL Designer always starts with this feature disabled.

After changing a block's type in this way, the design should be saved, closed and reopened in order for the parts to appear in their correct navigator folders.

Disable schematic Undo/Redo:

The schematic editor *Undo/Redo* functionality currently has performance problems when working on very large schematics. As a temporary measure to help customers' speed up work on these large schematics, it is now possible to switch off the undo/redo code.

Diagnostics

Crash Dumps:

This allows the user to enable the local storage of diagnostic information or "crash dumps" which can assist the product development team in the location of program errors.

To access this configuration tool, XLDesigner must be run as administrator.

Once crash dumps are enabled, the software can be restarted as a normal user and the crash dump files will be stored in the user-selected directory if an application crash occurs.

If the crash occurs again, then please send us the crash dump.

It is useful to have the crash dumps enabled at all times, they add no overhead to the program and will provide useful information in the event of a repeatable problem.

The window shown in Figure 10 controls the crash dump settings and should be configured as advised below.

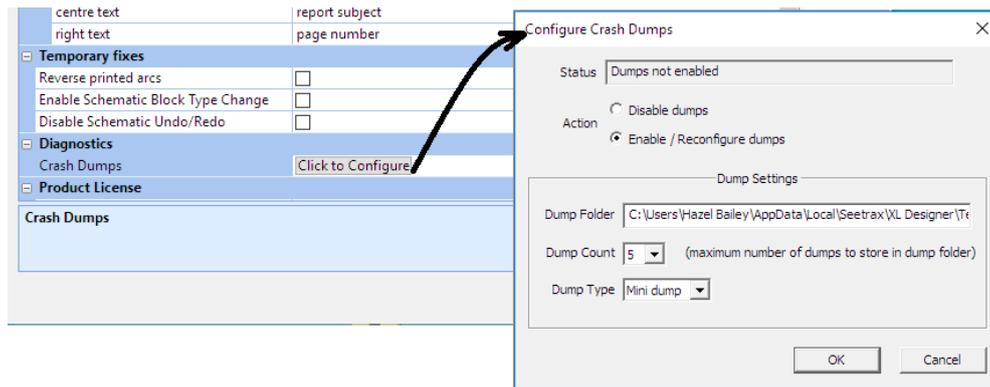


Figure 10

- Status:** shows the current setting for crash dumps, at the time the window was opened. This will not change until the setting is changed and the window closed/reopened.
- Action:** Choose which option is required.
If *Enable/Reconfigure dumps* is selected, then fill in the rest of the window as required.
- Dump folder:** Type in the path to the folder that any crash dumps that are produced will be stored in.
- Dump Count:** Specify the maximum number of crash dumps that will be stored.
Limit this number to avoid using lots of disk space. We typically require one crash dump in order to locate a problem. However it can be useful to have a couple of files if the crash is caused by a random event.
- Dump Type:** We typically require the Mini dump in order to locate a problem.
The Full dump is very large and usually too big to transfer easily.
We will request a full dump if required to help our investigations, but the mini dump should be sent in the first instance.
- Product License**
License Configuration: Refer to the heading “Licensing Information” within this guide (page 1).

Commands Common to All Editors

View commands

View > Pan

Used to view areas just outside of the current view.

Items that are being moved, added, etc. do not have to be released to allow this command to operate.

Once selected, point at the area of interest and click the left-hand mouse button. The area is re-drawn centrally on the screen.

Pan remains active until the right-hand mouse button is clicked to cancel the panning operation, when control returns to the previously active command.

Alternative (quicker) methods of panning

Press the user defined <pan to cursor > special function key (typically the spacebar) whilst the cursor is positioned over the item of interest.

Or press the <pan left>, <pan right>, <pan up> and <pan down> user defined special function keys (typically up/down/left/right arrow keys). The display is moved in the direction indicated, by one screen width.

If a mouse with a middle “scroll-wheel” button is in use the display may be panned by pressing and holding the middle mouse button, then moving the mouse in the direction required.

View > Zoom In

Used to increase the magnification of an area of interest. One level of magnification takes place about the cursor position. Once selected, point at and select with a click of the left-hand mouse button the item of interest.

Zoom In remains active until the right-hand mouse button is clicked to cancel the operation, when control returns to the previously active command.

Alternative (quicker) methods of zooming in

Press the user defined <zoom up> special function key (typically F2) whilst the cursor is positioned over the item of interest.

If a mouse with a middle “scroll-wheel” button is in use, by rotating the button, the display will zoom in or out, depending on the direction of rotation.

View > Zoom Area

Used to increase the magnification of an area of interest. A rectangle is defined around the area of interest, and that area is re-drawn to fill the screen. Items that are being moved, added, etc. do not have to be released to allow zoom area to operate.

Once selected, point at and select with a click of the left-hand mouse button one corner of the area of interest. A rectangle appears with one corner fixed in the selected position and the opposite corner attached to the cursor. Move the cursor until the rectangle encloses the area of interest, and release it with a second click of the same button. The area is re-drawn.

Zoom Area remains active until the right-hand mouse button is clicked to cancel the operation, when control returns to the previously active command.

View > Zoom Out

Used to reduce the magnification scale of the display.

Items that are being moved, added, etc. do not have to be released to allow this command to be selected.

Once selected, point at an area and click the left-hand mouse button. One level of reduction takes place about the cursor.

Zoom out remains active until the right-hand mouse button is clicked to cancel the operation, when control returns to the previously active command.

Alternative (quicker) methods of zooming out

Press the user defined <zoom down> special function key (typically F3) whilst the cursor is positioned over the item of interest.

If a mouse with a middle “scroll-wheel” button is in use, by rotating the button, the display will zoom in or out, depending on the direction of rotation.

View > Full - used to view the complete item being edited, i.e. schematic sheet, part, profile, outline, artwork, etc.

Items that are being moved, added, etc. do not have to be released to allow this command to be selected. Once selected, the display immediately re-draws to display the complete sheet.

Control returns to the previously active command once the screen has been re-drawn.

Alternative (quicker) methods of selecting Full

Press the user defined <full view> special function key (typically F4).

View > Previous View

Used to restore the previous picture to the screen. It can be used up to four times.

Items that are being moved, added, etc. do not have to be released to allow this command to be selected. When selected, the previous picture is restored to the screen and control returns to the previously active command.

View > Refresh

The screen is not automatically refreshed, so this command can be used to re-draw or refresh the screen.

For instance, if a symbol that is overlapping other items is moved, it looks as though the items that were underneath have disappeared or broken up. They are still there and *Refresh* restores the picture.

When selected the display is immediately refreshed, control returns to the previously active command.

View > Toolbar

The toolbar can be made invisible if required. This means that the icons will not be available for selection.

(Large or small icons can be selected from the *File > System setup* window.)

View > Status bar

The status bar (along the bottom of the XL designer window) can be made invisible if required. Warning This means that messages that normally appear in the status bar will be missed.

View > Navigator/Browser/ Properties

The *Navigator*, *Browser* & *Properties* windows can be made visible or switched off. When ticked in the pull-down menu, they are visible.

An extra facility, called *autohide*, allows them to be visible, but only when required. Typically it is more useful to have them visible, but hidden so they can be easily accessed when required and they don't clutter up the display.

Autohide

If the windows take up too much screen space, use the *Autohide* feature so they hide themselves when they are not used. They will shrink to the size of their tab, and will show themselves only when required.

To activate *Autohide* on a pane, select the pin button in the title bar (Figure 11(a) - the window is now reduced to a tab bar Figure 11(b). The actual position of the tab is related to the position of the window on the screen and can vary.

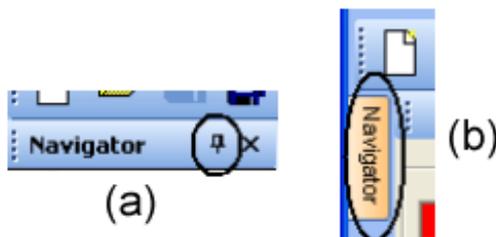


Figure 11

Moving the cursor over the tab re-opens the window. The window remains open whilst the cursor remains in the window. The pin is displayed rotated by 90 degrees () to indicate the window has been auto-hidden. The window closes as soon as the cursor is moved out-side of its area. This allows access to the window whilst achieving maximum editing area.

To un-hide the window permanently, select the rotated pin from the title bar of the window whilst it's open.

Docking & Merging

If the *Navigator*, *Browser* & *Properties* windows are visible and have not been auto-hidden (the pin should be upright in the title bar), they can be moved or "floated" around the screen as required, then released or "docked" in position.

To move/dock a floating window

Position the cursor over its title bar and hold down the left button. Drag the window to the location required.

A rectangle shows you where the window will be docked. It may try to snap to the edges of the screen

at a set size - the Ctrl key can be pressed when dragging the window to prevent it from trying to dock at these positions.

Release the mouse button to release the window.

The released windows can be moved as required, Figure 12.

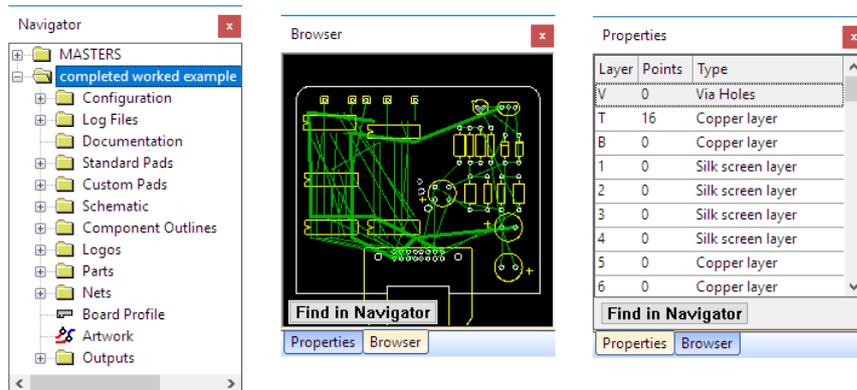


Figure 12

These windows can be re-sized by dragging the sides/corners of the window in the usual way.

To return them to their original positions without having to pick them up and move them, use the *View > Restore Default Layout > Main Frame Layout* command.

View > Restore Default Layout > Main Frame Layout

Used to restore the navigator, browser & properties panes to their original locations.

View > Restore Default Layout > Editor Layout

Used to restore the editor's dialogue bars to their original locations.

Grid commands

The grid commands are used to control the grid. The grid can be made visible/invisible, items can be made to snap to the grid/half grid as required and there are 6 grid settings available to cater for any size of grid required. The grid dot size and intensity can be adjusted from the *Edit > Display Adjustments* window.

The grid in the schematic/schematic parts editors may be displayed as lines or dots (*Edit > Preferences, Miscellaneous* window) and its colour can be altered (*Edit > Preferences, Colours* window).

The colour of the grid in the other editors (artwork, profile, pad and outlines) is controlled from the *Configuration > Colours* folder.

Grid > On/Off

Used to toggle the grid display on and off. A tick indicates the grid is visible.

At low zoom levels the grid pitch may be too fine to be displayed, this is indicated in the information bar:

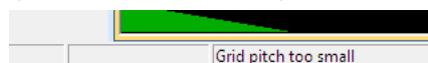


Figure 13

The current grid pitch is displayed in the status bar. The grid pitch is controlled by the selection made from the Grid menu.

Whether the grid is visible or invisible, the *Grid > Snap to Grid* setting controls whether items snap to the nearest grid or half grid setting.

Grid > Inch

Allows the user to specify a selection of grid pitches in inches, the most applicable one being chosen automatically for each level of zoom.

The selection of grids is defined in the *Configuration* folder of the design, *Grid Auto-pitch table*. For instance, grids of 0.2", 0.1", 0.050", 0.025" and 0.005" could be specified. When viewing the complete board, one of the coarser grids would be used as the finer grids would obliterate everything else on display. Likewise if a single item were filling up the screen, one of the finer grids would be used.

The threshold at which the grid pitch changes is controlled from the *Edit > Display Adjustments* window.

Grid > Metric

As *Grid > Inch* but using metric pitches

Grid > 0.1"

Gives a 0.1" grid at all zoom levels.

Grid > 1mm

Gives a 1mm grid at all zoom levels.

Grid > User Defined

Allows a specific grid to be defined which can be different in the X and Y axis. Once selected, type in the pitch required in both axes, select the units required then select **OK**.

Grid > Snap to Grid

Used to toggle grid snapping on and off. A tick indicates grid snapping is "on".

With grid snapping on, items that are moved snap to the nearest grid, or half grid point when they are released, whether or not the grid is visible. If snapping is switched off items are released wherever the cursor is positioned.

The current snap state gets saved with the design and the state is stored separately for the artwork, profile, outlines, custom pads and schematic views within the design.

Note: when using the region commands in the schematic editor, the region rectangle corners and move/copy destination will always snap to grid/half grid irrespective of this setting.

Grid > Select origin

Used to force the grid to radiate from a selected point. If the grid origin has been moved, then the information bar contains the words "Grid-shift".

In the component outline editor only a point (corner) of the outline or a pad datum can be selected.

In the artwork editor, component datum's can be selected as the origin, in addition to pads and track nodes on any visible layer.

In the profile editor only a point (corner) of the current category (profile, keepout or router) can be selected.

Grid > Edit origin

Used to force the grid to radiate from a specific position, using X-Y co-ordinates. If the grid origin has been moved, then the status bar contains the words "Grid-shift".

When selected, a window appears containing the current position of the grid origin, with respect to the original datum point of the outline. New co-ordinates can be typed in. Select **OK** to implement the change.

Grid > Clear origin

Used to restore the grid to its original location, the word *Gridshift* will be removed from the status bar.

Grid > Measure Distances

Used to display the co-ordinates of cursor selected positions with respect to the datum of the outline, and also the last position selected.

If grid snapping is switched on, the co-ordinates will refer to grid or half grid positions. The current grid is displayed in the information bar.

Once selected, select a position with a click of the left-hand mouse button. A window appears containing the XY position of the selected location. If another position is selected, the window contains its XY position, plus the distance in the X and Y axis (DX and DY) between the two selected positions. The straight-line distance is also given (L).

Window commands

Window > Cascade

Used to display the open windows in "cascading" mode from the top left of the SXLD window.

Window > Tile Vertical

Used to display the open windows in vertical columns in the SXLD window.

Window > Tile Horizontal

Used to display the open windows in horizontal rows in the SXLD window.

Edit commands

Edit > Units

Used used to change the selected units. Either metric or inch units can be selected.

The units selected control the X-Y readout, they do not control the grid which is controlled from the Grid command menu.

The selected units are used when displaying sizes or when sizes can be entered. Unless specified (" or mm) the decimal point indicates whether Imperial or Metric values are in use a . (dot) is used in Imperial values and a , (comma) in metric values.

The selected units mode gets preserved across program sessions.

Edit > Display Adjustments

Previous versions of Ranger and XL Designer had become very slow when refreshing the artwork display on Windows7 and later versions, which meant changes were necessary. XL Designer now uses the Microsoft Direct2D API for all graphical displays so that full speed advantage of the latest generation of graphic cards and processors can be utilised. This change has led to the loss of some features, such as colour mixing (when red and blue tracks overlapped they became purple) and the ability to control the thickness & height of items in terms of pixels when zooming in/out (so that labels/pin numbers didn't alter their height dependant on zoom level).

Because of these changes, there is no longer one setting that suits all hardware configurations or users, so new controls have been added to allow fine-tuning of the new-style display, for example to make lines and grid dots thinner/thicker/sharper, to make labels/pin numbers/grid dots bigger/smaller, etc. These will be found in the Edit > Display Adjustments window (also the *Opacity* control in the *View Control* dialogue bar).

A combination of changes may be required and if the display is not to your liking, some time should be spent adjusting the settings to obtain the best display possible. For example, it may be necessary to adjust the grid dot size and intensity to obtain grid dots that are comfortable to look at.

When making a change, it is a good idea to view the display with the minimum, then maximum values set – this provides a reference marker to allow finer-tuning to take place.

When selected a window similar to that shown in Figure 14 appears.

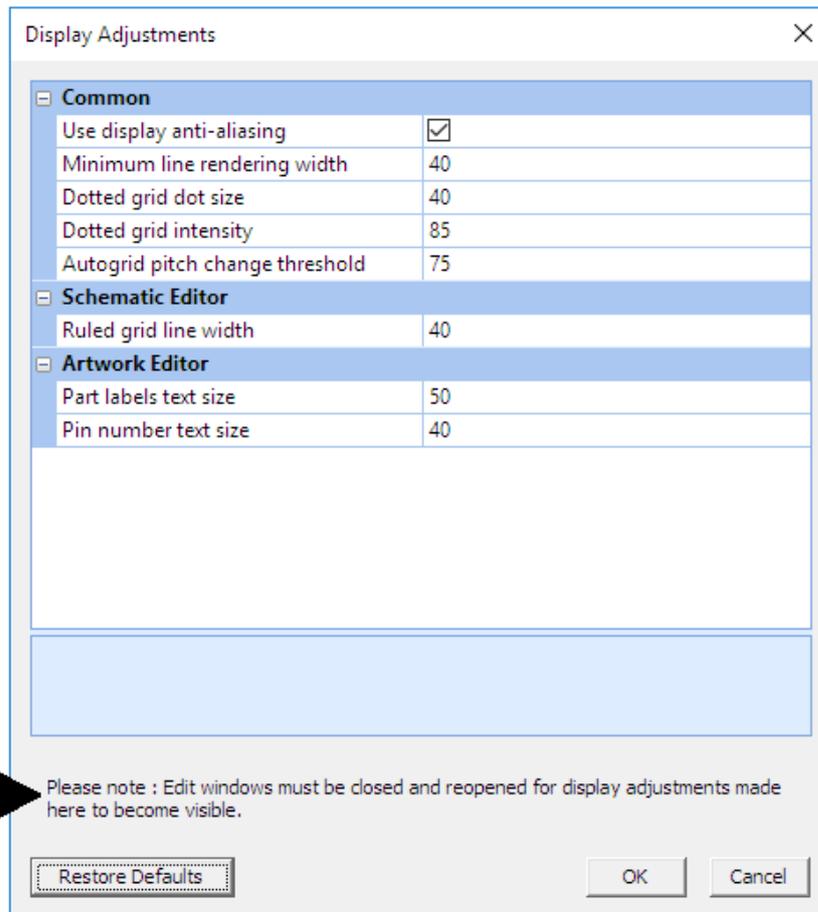


Figure 14

Pay particular attention to the note (arrowed in Figure 14) - if any adjustments are made to the settings, then the editor itself (schematic, outline, profile, artwork) MUST be closed and reopened for those changes to be implemented and therefore visible.

The first settings are common to all graphical editors:

Use Display Anti-aliasing:

The anti-aliasing feature of Direct2D means that the edges of displayed features are now much smoother – for example arcs/circles no longer have a jagged look to them, nor do non 0/90 degree lines. However this can be a disadvantage when trying to ensure that all lines are at 0 or 90 degrees, so you may prefer to have anti-aliasing switched off (unticked).

There is also some speed penalty with anti-aliasing switched on, though typically only noticeable on very dense artworks. A display rendering speed increase of 40% is typical when anti-aliasing is disabled.

Minimum Line Rendering Width:

Some lines can be too thin to be visible at some zoom levels (typically square/rectangular pads shown unfilled). This setting allows the thickness of the thinnest lines to be made visible at all zoom levels.

Note: when "display tracks at width" is selected, the tracks will never be rendered narrower than the size specified here.

Dotted grid dot size and Intensity:

It can be difficult to see the grid dots due to their size or intensity/brightness, or they can be too overpowering.

Use these two controls to make adjustments to the grid dot diameter and brightness.

Larger numbers increase the size of the grid dots and make them brighter.

Autogrid pitch change threshold:

By changing this value, the threshold at which the display switches from one autogrid pitch (Grid > Inch/Metric) to the next, can be controlled.

The auto-grid pitches are defined in the *Configuration > Grid Autopitch Table*.

Schematic Editor:

Ruled Grid Line Width:

The width of the ruled grid lines in the schematic editor can be set in the range 10 -100 where larger numbers result in thicker grid lines.

Artwork Editor:

Part Labels & Pin Number Text Sizes:

The size of text displayed for part labels and pin numbers can be set in the range 20-100 with larger numbers increasing the size of the text.

The size selected is a compromise as the actual height varies with zoom level, so adjust the size until a good compromise is selected. The more you zoom in, the smaller they appear.

Configuration Folder

The configuration folder contains fundamental information about the design. The majority of this information will be common across all designs, but individual designs may need slight variations.

For this reason, each design has its own configuration folder and settings. The information in the design's configuration folder is initially copied from a master configuration folder. The master configuration folder was created when the software was installed and it contains the default values for these settings.

The master configuration folder can be modified to suit individual company standards. It can be reviewed and altered as required, but bear in mind altering some values will affect other things. For example, altering the master pad sizes table will affect component outlines held in the master component outline library that use those pads.

Altering the master configuration folder has no effect at all on designs that already exist.

When a design is created it takes a copy of the master configuration folder contents, so the design's configuration folder is initially the same as the master configuration folder.

Once a design has been created, its configuration folder can be modified for the particular design rules in place for that design. Once created, each design's configuration folder is totally independent of the master configuration folder and every other design's configuration folder.

A design's settings can be restored to the values held in the master configuration folder by right-clicking on the design's configuration folder (Figure 15), selecting *Restore Design Defaults*, then selecting which items to restore to the current master design default values.

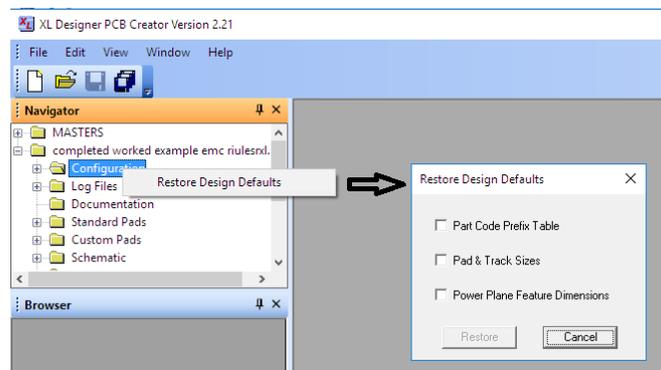


Figure 15

The configuration folders are accessed from the navigator pane as shown in Figure 16, from within the expanded Masters folder or the design's own folder.

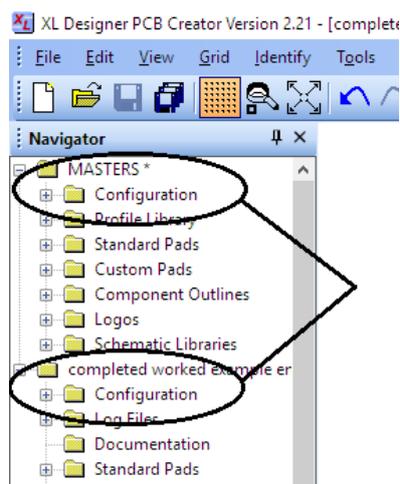


Figure 16

When the folders are opened, the settings appear. The content of a design configuration folder is shown in Figure 17. Each setting can be selected and opened.

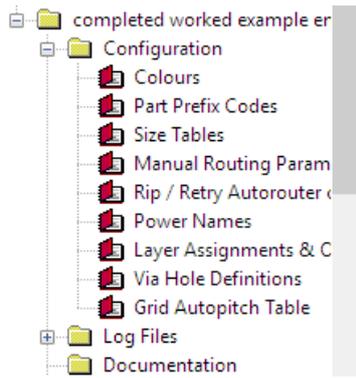


Figure 17

The master configuration folder does not include the settings for Power names, Layer assignments & ordering or Via hole definitions, as these will always be defined locally within the design.

Each of the settings is described in this chapter.

Configuration folder – Colours

When selected the window similar to that shown in Figure 18 appears.

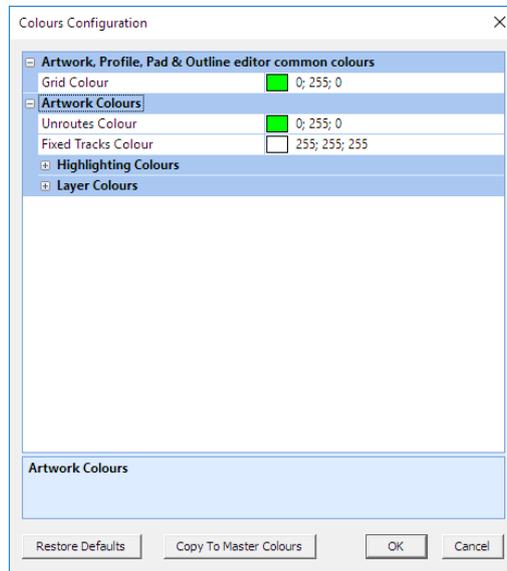


Figure 18

This permits the configuration of a set of colours for the items/layers/categories listed, in the specified editors. To make a change, select the category so that it highlights, at which time the browser selection button will appear, as shown arrowed in Figure 19.

Select this browser button to display the colour palette. Make a choice, then OK the window.

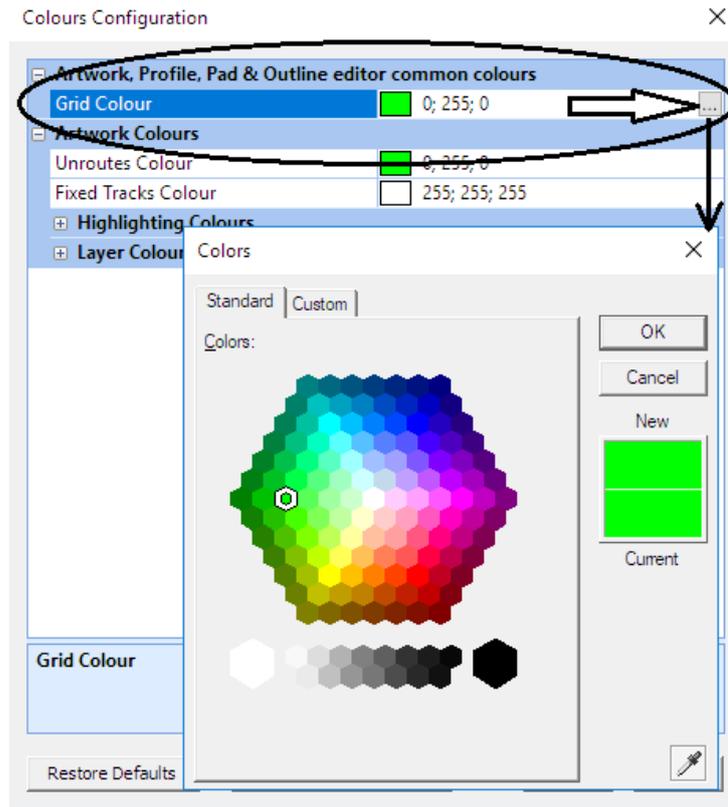


Figure 19

- Restore Factory Defaults: only available when the master colours window is open. It is used to replace the existing colour set with those that were selected when the software was installed.
- Restore Defaults: only available when the design's colours window is open. It is used to replace the existing colour set with those colours defined in the master set.
- Copy to Masters: only available when the design's colours window is open. It is used to copy the current design's colour set to the Master set.

Configuration folder - Part prefix codes

Every part within a design must have a reference designator, which consists of a part prefix code and a number. For example: IC1, IC2, R1, R2, R3, C1, C2, etc.

Upto 64 part prefix codes can be defined within a table, a sample part prefix code table is shown in Figure 20.

ID	Prefix	Description
1	IC	INTEGRATED CIRCUIT
2	TR	TRANSISTOR
3	D	DIODE
4	BR	BRIDGE RECTIFIER
5	XT	CRYSTAL
6	R	RESISTOR
7	VR	VARIABLE RESISTOR
8	C	CAPACITOR
9	L	INDUCTOR
10	PL	PLUG
11	J	SOCKET
12	TP	TEST PIN
13	SW	SWITCH
14	RL	RELAY
15		
16		
17		
18		

Figure 20

Up to 4 upper-case characters are permitted in a part prefix code. (Lower case characters are automatically changed to upper-case.)

An optional description can be assigned to each prefix, but the software does not refer to this data.

To save typing in a part prefix code table each time a design is created, a master table is provided. This table should contain the standard part prefixes used by your company. For example, IC or U might be used for integrated circuits, TR for transistors, R for resistors, etc.

When a design is created, a copy of the master part prefix code table is copied into the design, so the master table should contain all the prefixes commonly used by your company. Non-standard prefixes can be added to the design's part prefix code table as they are required.

To view/change the part prefix codes:

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Part Prefix Codes* with the left mouse button, or right-click it, and then select edit. The part prefix table appears.

To add or edit a part prefix or description, select the appropriate entry and type in the characters required. Use <tab> to move between entries.

Select OK to close the window and implement the changes.

Changes to the master part prefix code table only affect new designs, existing designs are not affected.

Changes made to a design's part prefix table do not affect other designs or the master part prefix table.

The design's part prefix table is automatically updated when a circuit schematic is compiled into a parts/wiring list or a parts/wiring list is imported, if new prefixes were used on the circuit.

The prefixes can be restored to the values assigned in the master part prefix table by right-clicking on the Configuration folder and selecting *Restore Design Defaults*, then choosing the option(s) required – but see the notes below prior to making the change.

Important Notes:

- DO NOT alter the order of the part prefixes in this list once a parts and wiring list exists - the change will affect the parts and wiring list and artwork. See *Effects of altering part prefix codes* below.
- New part codes should be added to the end of the existing entries.
- If an existing prefix is modified and a parts and wiring list exists, then all occurrences of the changed prefix will also be changed in the parts/net lists.

Effects of altering part prefix codes

The following information is supplied for reference purposes and explains why the order of part code prefixes should not be changed once a parts/wiring list has been created.

Internally, the software operates on a number it has assigned to part prefixes (from 0 to 63), not the prefix itself.

So for example, if the prefix IC is in position 0 in the part prefix code table, IC1 will be referred to internally as 01, IC2 will be 02, etc. If the prefix TR is in position 1, then TR1 is held as 11, TR2 as 12, TR3 as 13, etc.

In the parts list, each part is shown with its prefix, IC1, IC2, TR1, TR2, etc. and once the parts are placed on the artwork, their X/Y co-ordinates are also shown/stored. For example:

IC1	7400,74LS00	DIL14	1.0	2.0	0	(part held internally as 01)
IC2	7404,74LS04	DIL14	1.0	3.0	0	(part held internally as 02)
TR1	BC999,BC999	TO18	2.5	1.0	0	(part held internally as 11)
TR2	XX123,XX123	TO3	2.5	1.5	0	(part held internally as 12)

If the positions of the prefixes are reversed in the prefix code table, so that IC is now in position 1 and TR in position 0, the prefixes in the parts list also get reversed as shown below.

TR1	7400,74LS00	DIL14	1.0	2.0	0	(part held internally as 01)
TR2	7404,74LS04	DIL14	1.0	3.0	0	(part held internally as 02)
IC1	BC999,BC999	TO18	2.5	1.0	0	(part held internally as 11)
IC2	XX123,XX123	TO3	2.5	1.5	0	(part held internally as 12)

This is obviously incorrect and will affect the artwork if designed as well - TR1 and IC1 have swapped positions, as have TR2 and IC2. The tracks on the artwork will remain in their original positions, which will cause connectivity problems (shorts and unroutes).

If the prefix table is restored to its original settings, the parts/wiring list and artwork will also be restored correctly.

If the prefix IC was changed to ZZ, then all the IC's in the parts list would become ZZ's. On a completed design this would mean the silk-screen labels would be incorrect.

To sum up: do not alter the *order* of prefixes in the design's part prefix table unless the implications are fully understood.

The order of part prefixes in the master table can be changed without any effects.

Configuration folder - Size Tables, Pad Sizes

Four standard pad shapes are available, with up to 512 different pad sizes for each of those pad shapes. (Non-standard pad shapes are described in a separate chapter, entitled *Custom Pads*.)

The four standard pad shapes are:

- Round
- Square
- Round-ended-rectangular
- Square-ended-rectangular

(Heat-relief and anti-pads are also available for use on internal power planes and these are described separately later.)

Standard pads can have up to one drilled hole that will always be in the centre of the pad.

Connections/tracks are attached to the centre of standard pads.

To help identify the pads, each pad is assigned a number for reference purposes, from 0 to 511, which is known as its pad code.

When pads are used, they are called up by their pad code and the size of the pad is dependent on the dimension assigned to that pad code in the pad size table. This makes it easy to change the size of all the pads of a particular code within a design, by simply editing the pad size table – this also gives flexibility between designs.

Master/Design Size Tables

When the software was installed, a master pad sizes table was supplied. The pads used in the master component outline library use the pad sizes defined in the master sizes table.

When a design is created, a copy of the master size tables is copied into the design. The design's table can be edited without affecting the master sizes table or any other design.

The master pad sizes table should contain all the pad sizes in common use.

The design's pad sizes table in addition to the pad sizes in common use, should also contain any non-standard pad sizes required by that design.

Changes to the master sizes table only affects new designs, existing designs are not affected.

Changes made to a design's sizes table do not affect other designs or the master sizes table.

All the pad codes from 0 to 15 in the master pad sizes table had sizes assigned to them, whilst pad codes from 16 onwards were set to 0 unless they were in use by the master outline library.

(Pad codes from 0 to 15 that are not used in the supplied master outline library are set to 0.049" diameter/wide or 0.049" x 0.019".)

To view/change the sizes table:

With the Masters or Design folder open, depending on which set of pad sizes will be edited, double-select the *Configuration* folder, then either double-select *Sizes Table* with the left mouse button, or right-click it, and then select edit. The sizes table appears, similar to the one shown in Figure 21.

Size Code	Drill Diameter	Round Pad Diameter	Square Pad Width	Round Finger Length	Round Finger Width	Square Finger Length	Square Finger Width
0	0.0300	0.0550	0.0550	0.1000	0.0500	0.1000	0.0500
1	0.0000	0.0490	0.0490	0.0490	0.0190	0.0400	0.0100
2	0.0000	0.0490	0.0300	0.0490	0.0190	0.0600	0.0100
3	0.0000	0.0490	0.0350	0.0490	0.0190	0.0400	0.0110
4	0.0000	0.0490	0.0490	0.0490	0.0190	0.0600	0.0110
5	0.0350	0.0600	0.0400	0.0400	0.0190	0.0800	0.0240

Figure 21

If the design's sizes table is opened, this table was initially copied from the master sizes table when the design

was created.

This window has four tabs that can be selected, the Pad Sizes being one of them.

The list of pads can be scrolled through by selecting the scroll bar or arrows on the right side of the window.

The units in use can be set to inches or millimetres by selecting the *Units* command from the top of the window.

The pad sizes table is made up from rows and columns. Each row represents one pad code. The columns are used for the drill size and the size of each pad shape (rectangular pads have a length and width column).

The drill size relates to all pads with the same pad code.

Even if a drill size is specified in the pad, it may not be drilled when used. (Holes are only drilled if the pad is added to the inner layer of a pad stack, or to layer V in the artwork editor.)

Each column can be sorted in ascending/descending order by selecting the column headers.

Any of the values may be edited, by selecting the value and retyping. Note: changing the value of any pad/drill sizes that are in use within component outlines of the associated library (master outline library if the master sizes table is open, or the job library if the job's sizes table is open) will affect those outlines. Likewise if a job is open and the pad/drill size is changed, the change will affect those pad/drill sizes used in the artwork editor.

Select **OK** to close the window and implement the changes.

If the changes were made whilst the outline or artwork editors were open, the screen will need to be refreshed to update the displayed pads/drill sizes.

To restore the design's pad/track sizes table to the master settings:

The design's size table can be restored to the values assigned in the master sizes table by right-clicking on the Configuration folder and selecting *Restore Design Defaults*, then choosing the option(s) required.

Notes about pad sizes:

- round or square code 0 pads are always used for standard vias so this size code should not be used for anything else. If a component pad will require the same pad size as that assigned for code 0, then define another pad code with that same size.
For example code 0 and 9 round pads are both assigned as 0.055" (different drill sizes though) in the master table. Code 0 will be used for vias, and code 9 should be used for component pads. If a different hole size is required, define a new round pad.
- Pads that have been used on the artwork have an * (asterisk) next to them in the design's pad sizes table. (The * does not appear in the table until the outlines containing the pads have actually been placed on the artwork and the artwork saved.)
- In the master pad sizes table, any pad sizes that have a size assigned (with the exception of those set to 0.049" or 0.049" x 0.019") are used in the master component outline library. If those sizes are changed, they will affect the supplied master component outline library.
- round-ended or square-ended pads cannot have a pad width that is the same size or larger than the pad length defined – the width must always be smaller

Configuration folder - Size Tables, Heat-relief and anti-pads

Heat-relief pads and anti-pads are only used if power planes or split-power planes are generated on an inner layer of a design.

A power plane layer is an inner layer made from a solid sheet of copper. All holes that pass through the layer are connected to it, and therefore to each other, until they are isolated by anti-pads.

Two types of pad are used on power planes, **anti-pads** and **heat-relief pads** and they are shown in Figure 22, the anti-pad is on the left. The red background represents copper.



Figure 22

Anti-pads are used to isolate the holes from the copper plane. Anti-pads are in fact "not copper", they are the negative/opposite of a copper pad as they remove copper from the plane. When placed over a hole (that can't be seen in Figure 22) on a plane of copper, they form a clearance area around the hole. The hole is therefore disconnected from the plane.

Heat-relief pads are used around holes that should be connected to the plane; they are not strictly required as the hole is connected to the plane even if they are not used. However, problems with heat transfer could be experienced when soldering the board if they are not present. Figure 22 shows the heat-relief pad on the right, surrounding the hole which can be seen.

The heat-relief pad maintains the connectivity to the hole.

The sizes of these pads are related to the size of the hole passing through the board, and never related to the size of pads used on other layers of the board.

Different anti-pad and heat relief pad sizes can be defined for different hole sizes.

On the screen and during the plotting processes, a reverse image of the power plane is seen (except when using "Preview" mode in the power plane tools commands). For example, on the screen the anti-pads and heat-relief pads will appear as copper, but in reality they are not copper, the rest of the layer is copper and they form clearance areas.

Heat relief and anti-pads are defined in a table as shown below.

Master dimensions					
File Units					
Drill Size Range		Heat Reliefs		Antipads	
Min.	Max.	Track Width	Clearance	Channel Width	Clearance
Min	0.0300	0.0150	0.0150	0.0150	0.0150
0.0310	0.0500	0.0150	0.0150	0.0150	0.0150
0.0510	0.0700	0.0150	0.0150	0.0150	0.0150
0.0710	0.0900	0.0150	0.0150	0.0150	0.0150
0.0910	0.0910	0.0150	0.0150	0.0150	0.0150
0.0910	0.0910	0.0150	0.0150	0.0150	0.0150
0.0910	Max	0.0150	0.0150	0.0150	0.0150

Figure 23

The sizes can be displayed/entered entered in imperial or metric after selecting the *Units* command from the top of the window.

The sizes in each row can be changed by selecting the value to be changed and then typing in the new value. The <tab> moves the text entry cursor between fields. The window is divided into areas as follows:

Drill size range - Min and Max Sixteen ranges of hole sizes can be defined. This allows different heat-relief and anti-pad sizes to be defined for different hole sizes. The ranges should not overlap, e.g. if the minimum and maximum values for one range of holes were set to 0.020" and 0.035" respectively, the previous maximum value should be 0.019", and the next minimum value should be 0.036".

Heat-relief pads Three parameters are defined for heat-relief pads, as shown in Figure 24 and they are: **Track width**, **Clearance** and **Channel width**. The sizes entered

Anti-pads

relate to the range of hole sizes on the same horizontal line.

One parameter is defined for anti-pads as shown in Figure 24. It is the isolation distance between the edge of the hole and the copper plane.

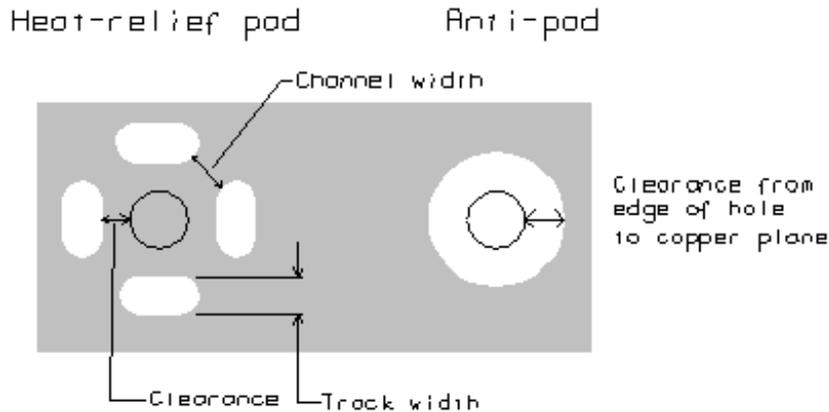


Figure 24

A master heat-relief & anti-pad table is provided. When a design is created, a copy of the master heat-relief & anti-pad table is copied into the design. Therefore, the master table should contain the standard requirements for heat-relief & anti-pads. Design specific requirements can be included in the design's heat-relief & anti-pad table as they are required.

To view/change the heat-relief and antipad sizes:

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Size Table* with the left mouse button, or right-click it, and then select edit. The sizes table appears, with the Pad sizes visible. Select the tab titled Heat-Relief & Anti-Pads to display the heat-relief & anti-pad size table. A window similar to the one shown previously in Figure 23 appears.

Changes to the master Heat-Relief & Anti-Pads table only affect new designs, existing designs are unaffected.

Changes made to a design's Heat-Relief & Anti-Pads table do not affect other designs or the master Heat-Relief & Anti-Pads table.

Heat-relief pads and anti-pads are only used on layers defined as power plane layers.

Selecting **OK** closes the window and implements the changes.

If the changes were made whilst the artwork editor was open, the screen will need to be refreshed to update the displayed heat-relief and anti-pads.

To restore the design's pad/track sizes table to the master settings:

The design's size table can be restored to the values assigned in the master sizes table by right-clicking on the Configuration folder and selecting *Restore Design Defaults*, then choosing the option(s) required.

Configuration folder - Size Tables, Track sizes

Each design within Seetrax XL Designer can use upto 512 different line thicknesses or track sizes.
(For profile, keepout, etc. line sizes, refer to the *Miscellaneous Line Widths* tab in this same window.)

Although referred to as the *Track sizes table*, any lines added to the artwork also use the sizes in this table, for example lines that form the silk-screen outline shapes.

Each track size is assigned a number for reference purposes, from 0 to 511, which is known variously as its width code, track code or size code.

A master track sizes table is supplied, with codes 0 to 15 having commonly used sizes assigned to them, the remainder being set to zero.

Each time a design is created, the sizes from the master table are copied into the design, so the design starts off having the same sizes assigned. The master table should therefore contain the standard track sizes used by your company. Non-standard track sizes can be added to the design's track size table as required.

Changes to the master sizes table only affects new designs, existing designs are not affected.

Changes made to a design's sizes table do not affect other designs or the master sizes table.

When tracks/lines are added, the size of the line is dependent on the size defined in the size table. This makes it easy to change the thickness of all the tracks/lines of a particular code within a design, by simply editing the track size table. It also allows flexibility between designs.

Notes when changing sizes:

- code 4 tracks are used extensively in the master outline library for silk screen outlines.
- code 2 is the default size code used when silk-screen labels are added
- codes 4 and 11 are the default sizes assigned for signal and power connections
- track/line sizes that are used in the artwork editor of the design have an * (asterisk) next to them in the design sizes table. (The * does not appear in the table until the tracks/lines have actually been used on the artwork and the artwork closed.)

To view/change the Track Sizes:

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Size Tables* with the left mouse button, or right-click it, and then select edit. The size tables appear, with the Pad sizes visible. Select the tab titled Track Sizes to display the track size table. A window similar to the one shown in Figure 25 appears

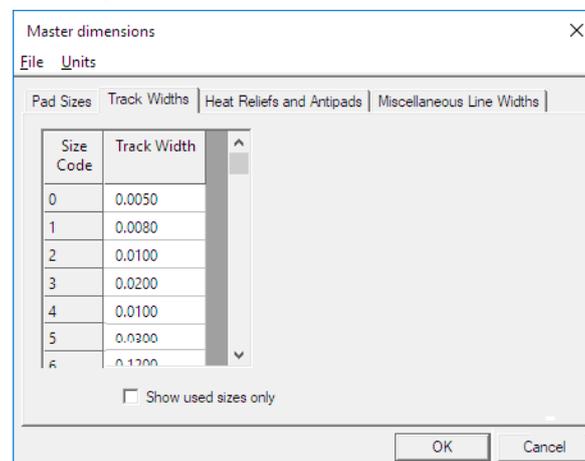


Figure 25

The units in use can be set to inches or millimetres by selecting the *Units* command from the top of the window.

To edit a track/line size, select the track size from the list, then type in the new size.

Select *OK* to close the window and implement the changes.

If the changes were made whilst the outline or artwork editor was open, the screen will need to be refreshed to update the displayed track/lines.

To restore the design's pad/track sizes table to the master settings:

The design's size table can be restored to the values assigned in the master sizes table by right-clicking on the Configuration folder and selecting *Restore Design Defaults*, then choosing the option(s) required.

Configuration folder - Size Tables, Miscellaneous Line Widths

This window controls the size of board profile and keepout region boundary lines and any other features that do not have a specific thickness assigned.

Please note that the values specified here only influence the display of features in the output task preview window and in resultant outputs. They do not influence the display within the outline, artwork or board profile editors.

To view/change the Line widths:

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Size Tables* with the left mouse button, or right-click it, and then select edit.

The size tables appear, with the Pad sizes visible. Select the tab titled *Miscellaneous Line Widths* to display the window similar to the one shown in Figure 26.

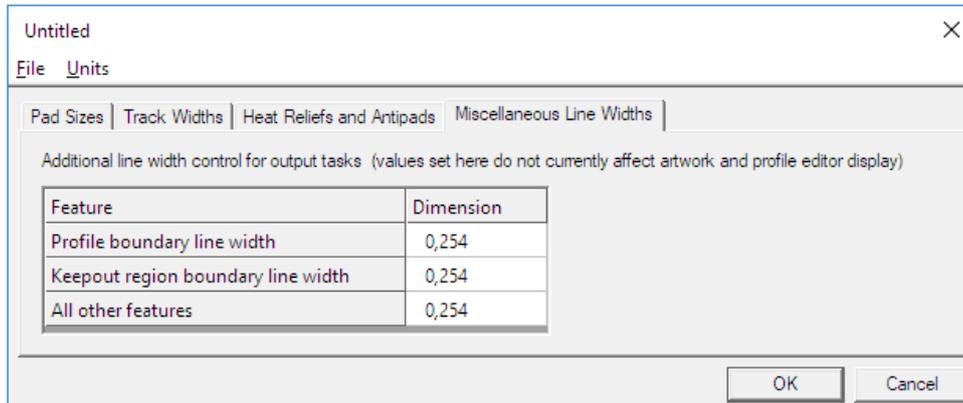


Figure 26

The units in use can be set to inches or millimetres by selecting the *Units* command from the top of the window.

To edit a size, select the size from the list, then type in the new value.

Select *OK* to close the window and implement the changes.

If the changes were made whilst the output task editor was open, the task will need to be closed/reopened to update the displayed lines.

Configuration folder - Manual routing parameters

This window contains the settings that are used when manually routing connections or tracks in the artwork editor.

A master set is stored and can be changed. The master set is copied into each design when it is created.

Changes to the master set do not affect existing designs.

The parameters within each design can be modified as required without affecting other designs or the master set.

To view/change the Manual routing parameters

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Manual Routing Parameters* with the left mouse button, or right-click it, and then select edit. The manual routing parameters window appears.

To edit a parameter: some have values that can be typed in, in which case select the entry and type in the value required. Use <tab> to move between entries. Other parameters have a pull down menu, from which an option can be selected.

Select *OK* to close the window and implement the changes.

When selected, a window similar to the one shown in Figure 27 appears. The parameters are described below.

Figure 27

Power & Signal Width Used to specify the default width codes used for power and signal connections/tracks.

The actual thickness assigned to the codes is displayed underneath the setting, which is controlled from the track sizes table. The codes can be changed by using the spin-controls alongside.

If the codes are changed and the artwork editor is open, the screen has to be refreshed to implement the change. The change does not affect existing tracks – only unroutes are affected.

Note: the schematic/wiring list can be used to specify particular width codes for individual nets if required and these values will over-ride the default settings on those nets.

DRC Clearance

When manually routing tracks in the artwork editor, it is possible to switch on, on-line dimensional rule checking (DRC) (using the tick box in this window).

If it is switched on, any gaps that are smaller than the sizes specified here when manually routing are flagged as clearance errors. Different minimum clearances can be specified for power and signal tracks.

The artwork-checking window is automatically updated to reflect the values assigned in this window.

Note: the schematic/wiring list can be used to specify particular minimum clearance requirements for individual nets if required and these values will over-ride these default settings on those nets.

The units (inches or millimetres) are defined before opening the window using the *Edit > Units* command.)

Active DRC

When manually routing tracks in the artwork editor with this setting ticked, clearance or short circuit errors are flagged as they occur. An error flag temporarily appears next to the offending clearance error or short circuit error, it doesn't stop the error occurring. The error can be left for later modification.

The minimum clearance requirements for signal and power tracks are defined in this window (*Default width codes and DRC clearance*).

It is important that the artwork checking routines are always performed on the finished artwork design in case flagged errors are forgotten.

With *Active DRC* unticked, clearance or short circuit errors can be made when manually routing and no warning is given.

Unroute reconnect

This setting is only used if parts are mounted on both sides of the board. It specifies the order required when performing the reconnect function on signal connections. It can be set to *Side-biased* or *Shortest*.

On layouts where all the parts are mounted on one side of the board, signal reconnection can only take place on that one side, irrespective of whether *Side-biased* or *Shortest* is selected.

Where parts are mounted on both sides of the board, the shortest path between connected pads may well be through the board, but this could prove costly in terms of the number of vias required.

The alternative would be for the reconnection to take place on each side of the board first, and then join through the board.

The user chooses either *Side-biased* or *Shortest*.

Example - There are eight surface mounted devices on the layout Z1 to Z8, and all pin one's are connected together. Z1 to Z8 are placed in a horizontal line across the top of the board. Then Z2, Z4, Z6 and Z8 are flipped on to the bottom of the board.

If side-biased reconnection is used, the connections will connect all the parts on the top of the board, followed by those on the bottom of the board, with one connection joining the two sides, i.e. connections will appear between Z1, Z3, Z5, Z7, a connection will join the top and bottom parts together, i.e. Z7 to Z8 with connections between Z2, Z4, Z6 and Z8. This would result in one via being required between Z7 and Z8.

If *Shortest* reconnection is used, then the connections will connect between Z1 to Z2 to Z3 to Z4 to Z5 to Z6 to Z7 to Z8, even though the parts are on different sides of the board. This means that 7 vias will be required to route the connections.

Unrouted power bias

Used to specify the default reconnection direction for power connections, as the shortest path is not always the most appropriate. It can be set to *Horiz*, *Vert* or *None*.

Horizontal bias will bias the power connections, when they are reconnected, in the horizontal direction, even if the distance between connected pins is greater than taking the most direct route between connected pins, as shown in Figure 28

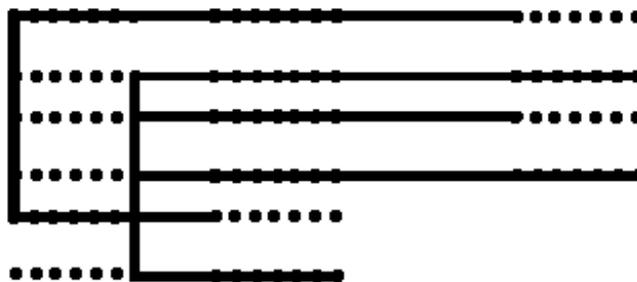


Figure 28

Vertical bias works in the same way as horizontal bias, except the direction is vertical, as shown in Figure 29.

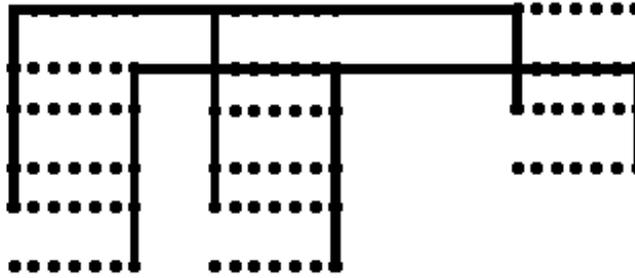


Figure 29

- No-bias (none) results in the most direct path being chosen for power connections when they are reconnected.
- Via shape* Selections are *Round* or *Square*. Standard vias are inserted with a code zero pad whose shape is defined here. Existing vias are unaffected by this setting, so an artwork could have both round and square code 0 vias in use.
- Segmove mode* Selections are *Free* or *Axis-locked*. This setting applies when the *Mroute > Move Segment* command is used. *Free* allows track segments to move freely in any direction. *Axis-locked* restricts horizontal and vertical track segments to movement in one axis only. Note: angled track segments always move freely.
- Unroute convert* Selections are *Convert all* or *Convert partial*. This parameter applies when using the *Mroute > Corner* command in the artwork editor to insert corners into unroutes, which subsequently turns the unroute into a routed track. It controls whether one or both segments on either side of the corner are converted into a track.
- With *convert all* selected, when a corner is inserted into the unroute, the unroute becomes a routed track on either side of the corner, i.e. both segments are routed.
- With *convert partial* selected, when a corner is inserted into the unroute, the shortest segment is converted to a routed track, whilst the longer segment remains as an unroute. Because the unroute is not complete (ie it has been partially routed) it is shown as a dashed line to distinguish it from complete unroutes that are shown as solid lines and stretch between pins.
- Note: if the "longer" segment is shorter than the value inserted in the *Unroute Minimum Length* parameter, the segment is also converted to a routed track.
- With *convert partial* selected, the track segment being added can be placed on any of the active manual routing layers by pressing the user defined "layer swap" key. Vias are inserted as required. The layer that the track will appear on is displayed in the Manual routing dialogue bar.
- Unroute min length* If *Unroute Convert* is set to *Convert Partial*, this parameter is used, it is otherwise ignored. When a connection is released, if the length of the unrouted segment is less than the length specified here, it is converted into a track.
- Select the box and type in the value required.
- Corner insertion* Selections are *Select\drop* or *Multinode*. This setting is used when the *Mroute > Corner* command is used in the artwork editor.
- If *Select\Drop* is selected, connections and tracks are released from the cursor once a corner has been inserted. If *Multinode* is selected, the connection or track remains attached to the cursor until the opposite mouse button is clicked, provided the connection or track segment selected is attached to a pad at one end.
- Select **OK** to close the window and implement the changes.

Configuration – Rip Retry Autorouter costs

This window contains the settings that are used when auto-routing connections in the artwork editor.

A master set is stored and can be changed. The master set is copied into each design when it is created.

Changes to the master set do not affect existing designs.

The parameters within each design can be modified as required without affecting other designs or the master set.

To view/change the Rip Retry Autorouter costs

With the Masters or Design folder open, depending on which will be edited, double-select the *Configuration* folder, then either double-select *Rip Retry Autorouter Costs* with the left mouse button, or right-click it, and then select edit. The rip retry autorouter setup window as shown in Figure 30 appears. The window is described below.

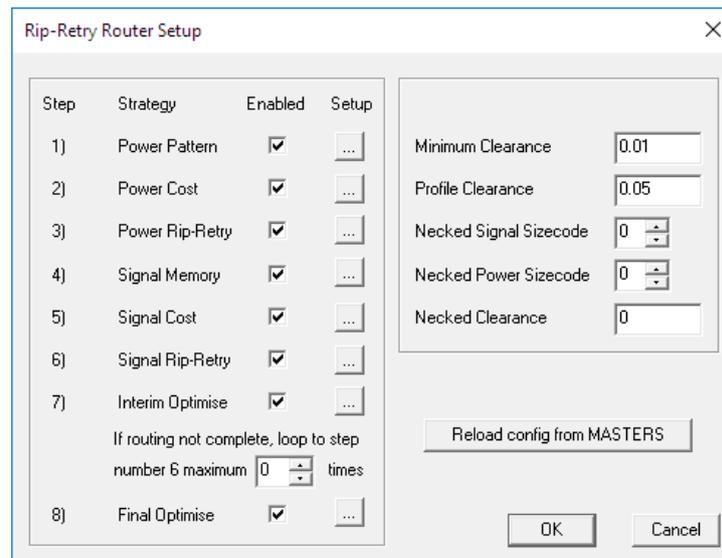


Figure 30

Select OK to close the window and implement the changes.

Rip Retry Autorouter Setup Window

The auto-router has 8 routing strategies available for use and they are numbered as steps from 1 to 8 in the setup window.

The strategies can be selected or *enabled* for use by the auto-router as required, by ticking the boxes in the *Enable* column alongside the strategy. The enabled strategies are always used in numerical order, from step 1 to 8.

Each routing strategy has parameters that are accessed by selecting the box in the *Setup* column alongside the strategy. When the box is selected, a window appears containing the parameters for that routing strategy.

Different types of layout may use different combinations of the strategies with different settings assigned to the parameters.

The master set should be set to suit most designs (as supplied). Before starting the auto-router, select the settings you think most appropriate from the information given below.

Any changes that are made are saved with the design. The master default settings can be restored at any time by selecting the "*Restore Default Config*" button from the setup window.

Power pattern strategy and setup

If enabled, this strategy will be the first to be used when auto-routing. It operates on the connections that have been defined as power rails in the configuration folder's, power names table.

This routing strategy has two objectives. First, to insert "stubs" (tracks with a via) from surface mounted component pins that should be connected to internal power planes

Secondly, if the board does not have power planes defined, it attempts to route the power connections without inserting via holes. The strategy runs as follows:

Power "buses" are placed horizontally and vertically across the board with "spurs" connecting to the appropriate pads. The minimum distance between buses connected to the same rail, and the maximum length of the spurs is defined in the setup window.

Power rails are routed one after the other, as listed in the wiring list.

Often, there will be pads connected to the power rails that are "out of reach" of the power buses because of the values assigned in the power pattern setup window. They will be attempted by the "costed power router", whose strategy and setup is described later.

Minimum bus spacing This setting is used to reduce the number of power buses on the layout. For example, the power routing in Figure 31, would usually be preferred to that in Figure 32.

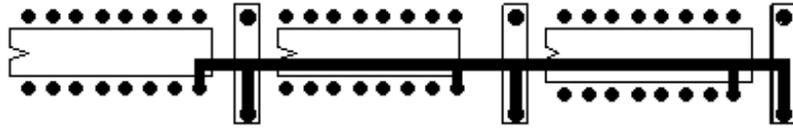


Figure 31



Figure 32

Before routing the power connections, the router draws imaginary straight lines, across all pads on the same power connection, i.e. all pins connected to the GND signal as shown in Figure 33.

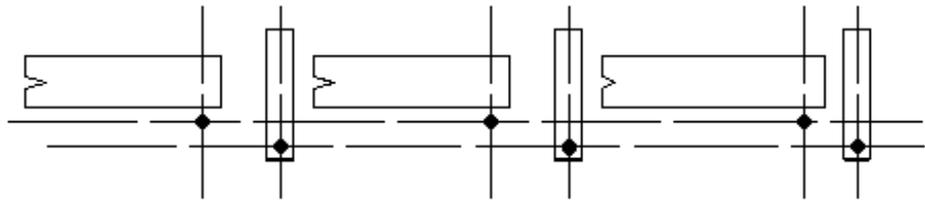


Figure 33

Lines that only pass through one pad are ignored.

The direction chosen for the bus lines depends on the number of pads each bus line passes through. The ones passing through the most pads are chosen.

The distance between these imaginary lines is the "bus spacing". If the lines are closer together than permitted by the *minimum bus spacing*, lines are removed one at a time until those that are left do not violate the spacing requirement. The lines that are left form the basis of the power buses. These lines obviously need to be adjusted to clear the pads that are not connected to the power rail, see: *Bus to pin spacing*

Bus to pin spacing

This value defines the distance between the imaginary bus lines that were drawn and maintained as described for the *Minimum bus spacing*, and the centre of the actual routed power bus rails. Spurs are then used to join the bus and pads together, as shown in Figure 34.

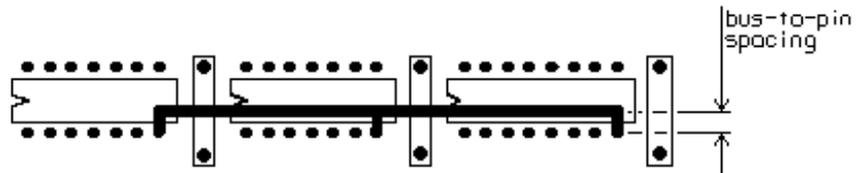


Figure 34

Off-bus capture

The router scans either side of the imaginary bus lines, for the centre of any pads that are also connected to the same power rail. The scanning distance is defined by the *Off-bus capture*. Any pads connected to the same power rail that are found within this scanning area will also be connected to the power bus with spurs.

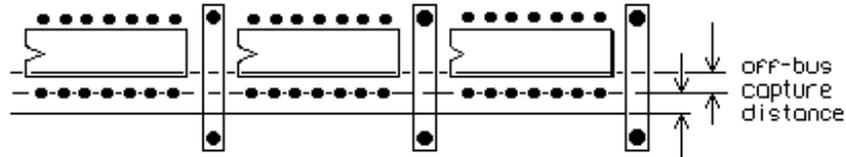


Figure 35

Increasing this value results in longer spurs. If a double sided board is being routed, with horizontal tracks on one layer and vertical tracks on the other, these longer spurs will block routing channels as they will be running at right angles to the preferred routing direction.

DIL Preference in Figure 34 the power busses have been inserted under the dual-in-line components. This setting controls whether the power busses are placed underneath or outside the dual-in-line components.

Note: the component outline's autoplacement footprint as defined in the component outline is used to define the boundary of the component outline for the purposes of the *under* or *outside* setting.

Fix routes Tracks that are routed using the power pattern router can be fixed in position if required. They will not then be ripped up by the power or signal rip-up routers if they are used.

Typically power tracks routed with this strategy are fixed in position.

Power cost strategy and setup

This routing strategy is designed to route the remaining power connections after the *power pattern* routing strategy has been used. It will however attempt to route all the power connections if the power pattern strategy was not enabled. The shortest connections are routed first.

This strategy evaluates all the possible paths that each track can be routed along. The path that is chosen is determined by its "cost". The costs are defined in the setup window and the lowest cost path is chosen.

This routing strategy can be repeated up to 3 times automatically, with different settings assigned to each routing pass. Each connection is attempted once per pass. Typically each pass is less restrictive than the previous pass to achieve more routed connections. In the early restrictive passes, more direct routing will be achieved, but not all the connections will be routed.

The settings for this strategy are described now.

Direction change Due to obstacles (pads, tracks, vias, etc.) the track may need to change direction to reach its destination pad. Every time it does this, the cost assigned to this setting is added to the cost of the route.

Note: only 90 degree direction changes are taken into account.

A higher value will make it less likely that direction changes will be inserted. This may lead to lower completion levels as it is less likely that paths will be found around obstacles. However on early routing passes the cost could be higher to achieve more direct routing.

Away from target The strategy searches for a path between the source and destination pads using an iterative process. If at any time during a search for a solution, the distance from the current search point to the destination pad is greater than, or equal to, the distance from the previous search point to the destination pad, the router assumes it is straying away from the target and adds the "away from target" cost to the solution cost at that point.

A higher cost will make it less likely for the router to find a way around obstacles, however on early routing passes the cost could be higher to achieve more direct routing.

Wrong way for layer Because of obstacles (pads, tracks, vias, etc.) the router may need to route horizontally on the side assigned for vertical tracks, or vice versa. Every time it does this, the cost assigned is added to the cost of the route.

Note: tracks inserted at 45 degrees are not taken into account.

A lower cost will result in more "wrong way routing" which will reduce the via count, but it may lead to lower completion levels.

Via hole To achieve horizontal tracks on one side of the board and vertical tracks on the other, as required on most double-sided boards, a via should be inserted every time the track changes direction.

Every time a via hole is added, the cost assigned is added to the cost of the route. A lower cost will result in more vias and less wrong way routing.

Maximum route cost If the total cost of a route between two pads exceeds the value entered here, the connection will not be routed. Subsequent routing passes will attempt to route it.

A lower maximum cost will make it less likely for the router to find a satisfactory way around obstacles. However on early routing passes the maximum cost could be set lower to achieve more direct routing.

Gravity distance & Gravity strength - The router will normally find a minimum cost route between the source and destination pads. This will sometimes result in a layout where tracks are too far apart, consequently wasting routing channels. A higher completion rate would be achieved if the tracks had been forced to pack tightly together. The Gravity distance and Gravity strength parameters control this tendency for tracks to pack closer together.

The Gravity distance parameter controls how close a route search must approach an existing route before it gets pulled and held close to the existing route's path.

It is measured in routing grid units. Valid entries are 1 through to 10. For instance with a routing pitch of 0.025", an entry of 6 would be 0.150". If the routing pitch were 0.020" an entry of 6 would be 0.120". A higher number will pack tracks closer together than a lower number.

The Gravity strength parameter is a measure of the difficulty of breaking away from an existing route's path, after having been drawn towards it.

Valid units are from 1 through to 50. A higher number makes it more expensive for the track to move away from the existing route.

Routing pitch Defines the minimum grid over the board on which the centre line of tracks and vias can be placed.

The grid radiates from the origin of the board which is defined via the *Grid* commands in the artwork editor. It is suggested that the position of the grid origin is maintained between part placement and routing.

Finer grids will result in more success. Before assigning a very fine grid (0.001") bear in mind that finer grids require a lot more memory (RAM) (this will also be affected by board dimensions and layers) and that the resultant tracks will be placed on that grid. If manually adjusting tracks afterwards, this can prove frustrating when trying to maintain horizontal and vertical tracks.

Major axis overshoot An imaginary rectangle is drawn to enclose the two connected pads that are currently being evaluated for routing. The chosen route path must lie completely within the rectangle, or the connection will be left unrouted.

The rectangle can be made bigger by a specific amount as defined by this parameter. The longer side of this imaginary rectangle is defined as the "major axis".

If an overshoot of 0.2" is entered the rectangle is increased by 0.1" at each end of the major axis.

Typically no overshoot is permitted on the initial routing pass. The overshoot generally increases with each pass. The routing time will increase as this rectangle is made larger.

Minor axis overshoot - This parameter operates in the same way as the Major axis overshoot as explained above, except it is the shorter side of the imaginary rectangle that is enlarged.

Track neck and bend Selections are *None* or *Neck*.

If *Neck* is selected, the router can neck tracks in order to find a path between obstacles, see Figure 36. The neck size code, and clearance requirements for necked tracks, are defined in the main setup window (Figure 30).

The necked section of track extends for two routing pitches, either side of the obstacle. If a track is necked down, and the end of the necked segment of track coincides with a corner (direction change), the necked segment extends beyond the corner for one routing pitch.

If *None* is selected, the strategy cannot use necking and the tracks have to be routed at their specified size.

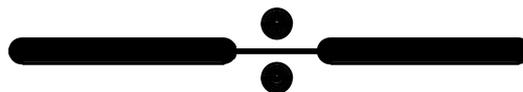


Figure 36

Pass number The values assigned to the settings in this window apply to the pass number shown here. Use the arrows to display the other pass settings. With this strategy, 3 sets of values can be applied for pass 1, 2 and 3.

Start pass the number shown here determines which set of values is used for the first pass.

N passes When this routing strategy is used, it attempts to route every power connection with the values assigned to the first pass as defined by the Start pass parameter. If some connections remain after the first routing pass has been completed, the router can

attempt the remaining connections with further passes. Enter the maximum number of passes permitted.

Bear in mind that pass 3 will be repeated as many times as is necessary to reach the number of passes required (unless routing is complete when it will stop). Repeating pass 3 with this strategy is not recommended - if the strategy cannot complete a connection the first time it uses pass 3, it won't manage it the second time because nothing will have changed.

Power Rip-retry strategy and setup

The Power Rip-retry routing strategy is designed to route the remaining power connections after the *Power Pattern* and *Power Cost* strategies have been used. It will however attempt to route all the power connections if the *Power Pattern* and *Power Costs* strategies are not enabled.

Whilst attempting to route the remaining connections, this strategy can remove (or rip-up) existing unfixed tracks in order to complete remaining connections. Tracks that have been ripped-up are added to the list of remaining connections and will be attempted again.

This strategy also has settings that can be "costed" by the user to force the router to use user-preferred routing options.

Ripped up tracks are re-routed with the costs assigned to the current routing pass and not those they were originally routed with. If the track has to be replaced in its original position because a new path could not be found, the costs are not taken into account, i.e. it will always be allowed back to its original position, even if that position would not be valid with the current costs.

This routing strategy can be repeated as many times as required. It uses the same parameters as the *Power Cost* strategy, plus the following:

Ripsearch Max violate This parameter can be used to allow the auto-router to *temporarily* make violations (short circuits or clearance errors) on the board in order to route more successfully.

For example, with this parameter set to 1, in order to route a remaining connection known as A, an existing track known as B, would be violated. The solution for connection A is stored, and track B is ripped out and reverts back to a connection. Connection B is added to the list of routes still to be completed. This list will be referred to as the "to-do" list. The top entry in the "to-do" list is taken and a solution is searched for that does not violate any other track. If no solution is possible, a solution is sought that will not violate more than the permitted number of rip-search violations. Any new solution is placed on the artwork, and violated tracks added to the "to-do" list. This process continues until one of the following conditions is met:

- a) The "to-do" list is empty. In this case, the original route and all ripped-up routes, were routed without violation.
- b) A route is taken from the "to-do" list and no solution can be found within the permitted "max violation depth" range. In this case the original routing is restored and the route attempt fails.
- c) Too many iterations have been attempted. This cut-off point is set at two times the max violation depth iterations. If the "to-do" list contains only one entry, the new situation is accepted as no worse than the original situation, otherwise the original routing is restored. In either case the route attempt fails.

Maximum rip-up depth The number inserted here determines the number of existing routed connections that can be removed in order to route one of the remaining connections. These "ripped-up" connections are added to the list of remaining connections, the "to-do" list.

Rip-up fixed routes selections are *Yes* or *No*. Selecting *Yes* allows the strategy to rip-up any tracks that are fixed in order to find room for tracks. (Including tracks fixed manually in the artwork editor or automatically whilst the router was routing.)

Pass number The values assigned to the settings in this window apply to the pass number shown here. Use the arrows to display the other pass settings. With this strategy, 3 sets of values can be applied for pass 1, 2 and 3.

Start pas the number shown here determines which set of values is used for the first pass.

N passes When this routing strategy is used, it attempts to route every power connection with the values assigned to the first pass to be used. If some connections remain after the first routing pass has been completed, the router can attempt the remaining connections with further passes. Enter the maximum number of passes permitted.

Pass 3 will be repeated as many times as is necessary to reach the number of passes required (unless routing is complete when it will stop). Repeating pass 3 may improve routing success - the strategy will be ripping-up and re-routing so the result of each pass will be different.

Signal Memory strategy and setup

If all the strategies are enabled, the memory router is used after the power strategy but before the signal strategies. It should always be run before attempting to route the rest of the signal connections, and its results fixed in position.

Memory type connections are routed efficiently, without vias and will cause the least obstruction to subsequent tracks. If the memory connections are left until later on, their optimum paths will probably be blocked and they will prove very difficult to fit in. Fixing them in position means that the rip-retry strategy will not waste time trying to find a better path for them.

The Signal Memory strategy is designed to route any signal connections that fall within the "memory type connection" category.

What is a "memory type connection"?

Take any signal connection and enclose the two connected pads within a rectangle, as shown in Figure 37.

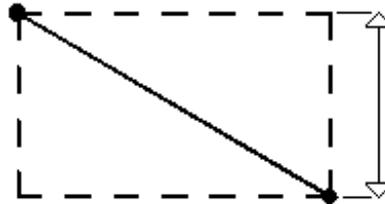


Figure 37

If the shorter side of the rectangle is equal to, or less than, two memory routing pitches, as defined in the setup window, the connection is regarded as being a memory type connection as far as the strategy is concerned.

The memory routing strategy is so called because of the pattern formed by the routed tracks, as shown in Figure 38.

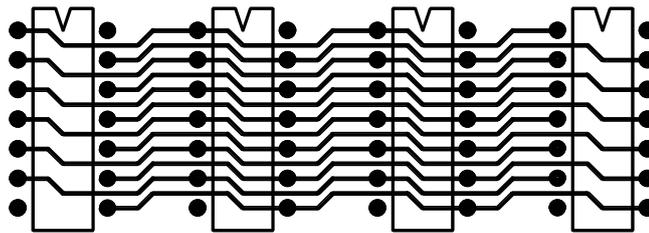


Figure 38

The success of the memory router depends upon the routing pitch selected. On conventional boards, a pitch of 0.050" will give the best results.

When the memory router is enabled it assesses every signal connection on the board, from left to right, top to bottom, and any connections that fall within the "memory" category are attempted.

The connections have to be routed using a maximum of five segments on one side of the board, vias are not permitted. 45 degree track segments are permitted. Tracks that are routed vertically are placed on the side designated for vertical tracks and horizontal tracks are placed on the side designated for horizontal tracks.

Routing pitch Defines a grid over the board on which the centre line of tracks can be placed. To route memory type connections between the pins of dual-in-line components where the pin pitch is 0.1", the routing pitch should be set to 0.050". If the pin pitch is halved, the routing pitch should also be halved.

The grid radiates from the origin of the board, which is defined via the Grid command in the artwork editor. It is suggested that the position of the grid origin is maintained between part placement and routing.

Fix routes It is suggested that memory tracks are fixed in position (box ticked) to save time on subsequent routing and optimising passes.

Signal Cost strategy and setup

This signal routing strategy is designed to route the remaining signal connections after the memory routing strategy has been used. It will however attempt to route all the signal connections if the memory routing strategy was not enabled. The shortest connections are routed first.

This strategy evaluates all the possible paths that each track can be routed along. The path that is chosen is determined by its "cost". The costs are defined in the setup window and the lowest cost path is chosen.

This routing strategy can be repeated up to 6 times automatically, with different settings assigned to each

routing pass. Each connection is attempted once per pass. Typically each pass is less restrictive than the previous pass to achieve more routed connections. In the early restrictive passes, more direct routing will be achieved, but not all the connections will be routed.

This strategy uses the settings described previously for the *Power Cost* strategy, plus the following:

Routing at 45 degrees In some situations the router may need to insert 45 degree tracks instead of using horizontal or vertical ones in order to move around obstacles.

Every time a 45 degree track is inserted, the cost assigned by the user multiplied by the length of the segment, is added to the cost of the route.

The length of a 45 degree segment is determined as follows:

a rectangle is drawn around the 45 degree segment and the length of one side measured in increments of 0.005". The number obtained is multiplied by the cost assigned by the user, which is then added to the cost of the route.

Longer 45 degree track segments act as bigger obstacles to subsequent routing, hence their increased cost.

When working on double-sided boards, 45 degree tracks act as obstacles to both horizontal and vertical tracks so they should be avoided in the early stages of routing. However 45 degree tracks on single-sided boards tend to give higher completion rates.

A higher cost will make it less likely that 45 degree tracks are used.

Track neck and bend Two additional selections are available, *Bend* or *Neck and Bend (N+B)*.

If *Bend* is selected, the autorouter can locally change the selected routing pitch, in order to route two tracks between adjacent pads. This option is typically used when trying to find a path in or out of congested areas, i.e. when trying to route to the third row of a 96-way connector. It still has to maintain the clearance requirements set by the user.

Note: bending will only be used if a routing pitch of 0.025" is selected.

If *N+B* (neck+bend) is selected, the autorouter can use either necking and/or bending as explained previously, to find ways through congested areas.

Pass number The values assigned to the settings in strategy window apply to the pass number shown here. Use the arrows to display the other pass settings. With this strategy, 6 sets of values can be applied.

Start pass the number shown here determines which set of values is used for the first pass.

N passes When this routing strategy is used, it attempts to route every signal connection with the values assigned to the first pass to be used. If some connections remain after the first routing pass has been completed, the router can attempt the remaining connections with further passes. Enter the maximum number of passes permitted.

Bear in mind that pass 6 will be repeated as many times as is necessary to reach the number of passes required (unless routing is complete when it will stop). Repeating pass 6 with this strategy is not recommended - if the router cannot complete a connection the first time it uses pass 6, it won't manage it the second time because nothing will have changed in between.

Signal Rip-retry strategy and setup

The Signal Rip-retry routing strategy is designed to route the remaining signal connections after the *Memory* and *Signal Cost* routing strategies have been used. It will however attempt to route all the signal connections if the Memory and Signal Cost strategies are not enabled.

Whilst attempting to route the remaining connections, this strategy can remove (or rip-up) existing unfixed tracks in order to complete remaining connections. Tracks that have been ripped-up are added to the list of remaining connections, the "to-do" list and will be attempted again.

This strategy also has parameters that can be "costed" by the user to force the router to use user-preferred routing options. All these settings have been described above.

Ripped up tracks are re-routed with the costs assigned to the current routing pass and not those they were originally routed with. If the track has to be replaced in its original position because a new path could not be found, the costs are not taken into account, i.e. it will always be allowed back to its original position, even if that position would not be valid with the current costs.

This routing strategy can be repeated as many times as required. It has 3 sets of values and repeating pass 3 is worth while as during rip-up cycles, the existing tracking will change, so the results will be improved.

Interim Optimise strategy and setup

The *Interim Optimising* strategy can be used for two purposes:

- (i) to "tidy up" a partially routed layout in order to assist subsequent routing passes
- (ii) to restore vias on a completed layout to assist in the routing of modifications that have been made to the completed layout.

For example, once a layout is complete, it is usual to have reduced the number of vias used on the layout. If new connections are added to that layout, they will prove difficult to route because of the amount of "wrong way" routing, which acts as obstacles. Hence the need to restore vias.

NB: The interim optimiser is only used if there are connections still waiting to be routed, in which case it is used after all the other enabled routing strategies have been completed, but before step 6 is repeated.

The Interim Optimiser assesses every routed track and removes "kinks" in tracks that were originally used to move around tracks that were subsequently ripped-up and routed elsewhere.

The optimiser will restore vias to tracks if doing so decreases the total route cost. Once all the routed tracks have been assessed, the optimiser then attempts to route any remaining connections.

The interim optimiser has one additional setting that is only used by the optimising strategies as follows:

<i>Optimisation order</i>	this setting controls the order in which tracks are evaluated for optimisation. Options are:
S-L	short to long
L-S	long to short
S-L/L-S	alternate between short to long and long to short on subsequent passes
L-S/S-L	alternate between long to short and short to long on subsequent passes

Final Optimise strategy

The final optimising pass is performed when all the other enabled routing strategies have been completed and repeated as many times as specified. All the connections will not necessarily have been routed.

It may be wise not to enable this final optimisation pass until all the connections have been completed.

Otherwise remaining connections could be difficult to route afterwards, as this pass is typically run to try and decrease the number of vias in use.

The final optimisation pass evaluates each routed track and tries to re-route them with a lower routing cost.

Typically vias are made expensive, so that it tries to re-route each track with fewer vias. If vias are removed, the amount of wrong way routing will be increased, i.e. horizontal tracks will appear on the side assigned for vertical tracks and vice versa.

The number of optimising passes can be specified. Three optimisation passes should be sufficient to improve the design. More passes can be used but diminishing returns will be seen.

- holes - choose from the drop-down list.
- Diameter The diameter of the tool – in inches or mm (change units using the Units command from the top of the window).
- Pre-drill Diameter If the tool “Type” is set to Router and pre-drilling is required, specify the diameter of the pre-drilled hole.
Note: it is also necessary to add a tool for the drill of the pre-drilled hole, which should match the pre-drill size.
- Drill Infeed Rate & Routing worktable rate (ipm) These will generally be left unassigned and controlled by the manufacturer.
- Spindle RPM This will generally be left unassigned and controlled by the manufacturer.

The table can be sorted in various orders, using the **Reorder** button, which helps identify sizes already defined in the list to avoid repetition, Figure 41.

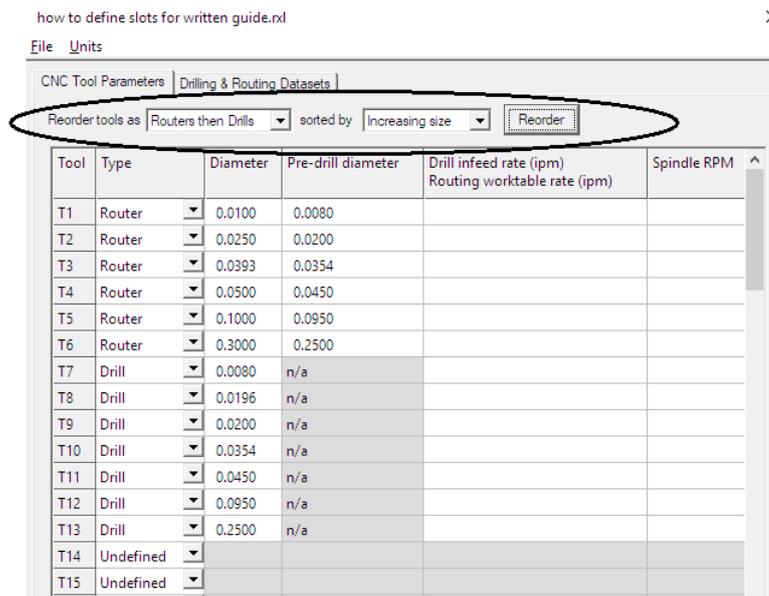


Figure 41

Once this table has been filled in and saved, the slot diameter and drill diameter settings in the **Slots** dialogue bar (Artwork editor, Tools > Slots & Extra Holes) can be selected from a drop down list, Figure 42.

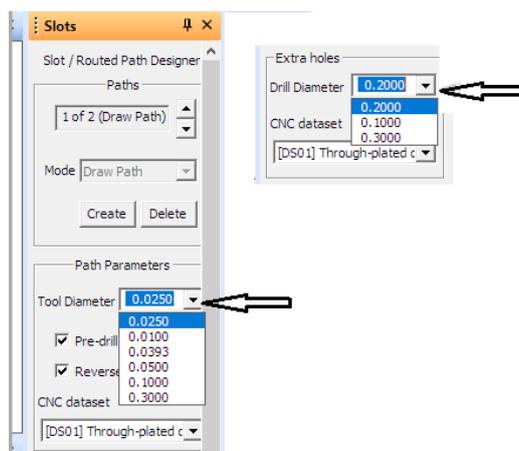


Figure 42

Drilling & Routing Datasets

The **Drilling & Routing Datasets** table is where names & descriptions can be assigned to the different sets of

drilling data that will be required for a board, such as plated and unplated holes, and/or blind/buried via holes. If a board has plated and non-plated holes, then at least two drilling output files will be required. Additional files are required for each blind/buried via stack.

Two pre-defined datasets exist, these are:

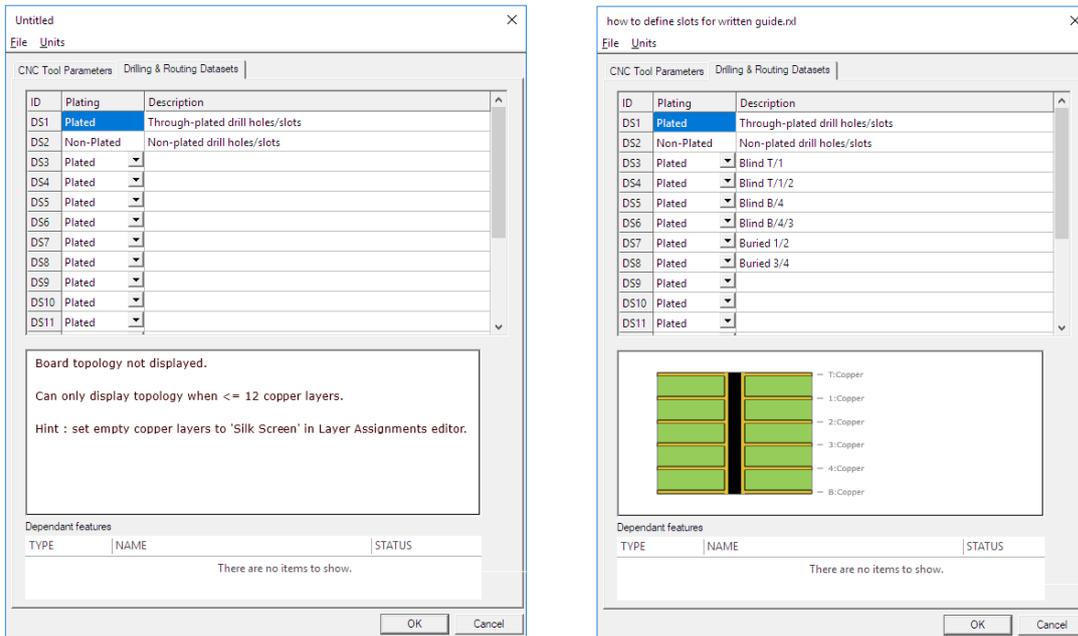
- DS1 for through plated drill holes/slots
- DS2 for Non-plated drill holes/slots

Any slots or holes that should be output alone should have their own datasets.

All the slots/drills in the same dataset will be output together.

Figure 43 shows the table for a new design on the left and for a 6 layer design with blind and buried vias on the right.

Most single/double-sided designs contain through the board holes, either plated or non-plated, so typically just two drill datasets are required, one for each type.



Drilling & Routing Dataset for a new design

Drilling & Routing Dataset for an existing 6 layer design with blind and buried vias

Figure 43

- ID** A unique number assigned to each dataset. DS1 and DS2 are pre-defined and cannot be changed.
- Plating** Whether the holes/slots included in this dataset will be plated or unplated.
- Description** A description to aid recognition of the dataset by the user.
- Board topology** The board topology can only be displayed when there are less than 12 copper layers. Most designs generally have all unused layers defined as copper layers, so there are too many to display. (This can be changed from the *Configuration* folder, select *Layer Assignments and Ordering*, then select the *Set empty copper layers to silkscreen* button.) The topology of a double-sided design with a through-plated and non-plated through hole is shown in Figure 44.

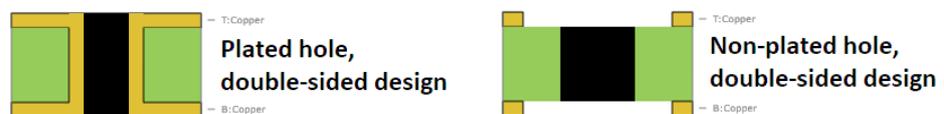


Figure 44

(Figure 43 shows the topology of a through plated hole in a 6 layer design.)

If there are additional copper layers, and a dataset other than DS1 or DS2 is selected, the top and bottom of the drilled zone can be moved up and down in the board topology using the **Layer Span** buttons, identified in Figure 45.

Figure 45 shows DS4 selected, which is being used for a blind via from the top of the board through to layer 2.

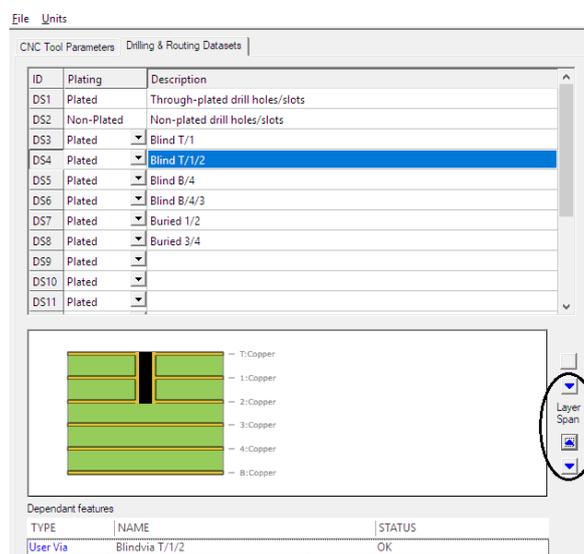


Figure 45

In version 2.25 this graphic is just an advisory to help the designer understand what the various drilling and routing sets of data are being used for, but eventually it will also guide the validation of the board.

Configuration folder - Power Names

A list of power names is defined within each design so that the power connections can be differentiated from other connections/tracks. When a design is created this list is empty.

Power connections (by default, though this can be changed by the user) are given a different track code to the signal connections, can be given a different minimum clearance requirement, they have their own auto-routing strategy and can be converted into a power plane/split power plane.

The power names list should include all the names of connections that should be treated as power connections by the artwork layout and checking routines.

Up to 256 different power names can be entered. Upper or lower case characters, or a mixture of the two can be used in a power name. However, do not enter the same name more than once with different combinations of upper and lower case characters. i.e. do not add GND and Gnd, or GND and gnd, etc.

If a circuit schematic has been drawn and a parts/wiring list extracted, the power names (as defined implicitly in used schematic parts) are automatically added to the power names list.

Additional named connections can be added to the list as required, so that they are treated as power connections, even though they might not technically be considered as power connections.

Power names can also be removed from the list if they should not be treated as power connections. (If the tracks have already been routed, de-classifying them will not affect their track sizes.)

If a parts and wiring list has been or will be typed in or imported, the power names have to be entered manually into the list. (Some of the importers can add names to the power names list, but always check the list to ensure the correct names are present.)

Viewing/Editing the Power Names List

With the Design folder open, double-select the *Configuration* folder, then either double-select *Power Names* with the left mouse button, or right-click it, and then select edit. The power names list appears with any power names already defined, present, similar to the one shown in Figure 46.

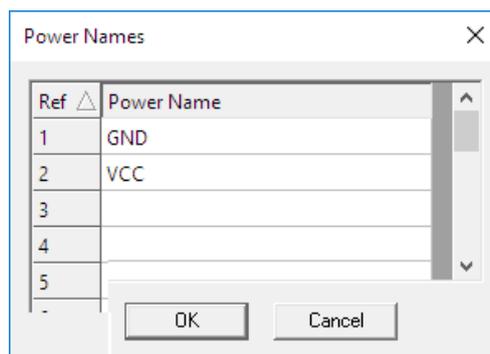


Figure 46

Each power name is given a numerical reference number (from 1 to 256) that is used internally.

Power rail names should be added one after the other, starting from the top of the list.

The columns can be sorted in ascending/descending order by clicking on the column headers.

To add a power name, select the next empty slot (alongside 3 in the example shown above), then type in the new power name. Select another slot or press <tab> to continue. Repeat for each different power name.

To edit a power name, select the power rail name edit or replace the name as required. Select another slot or press <tab> to continue.

Select OK to close the window.

When the net list is open (from the design's net list folder), power names appear highlighted.

Configuration – Layer Assignments & Ordering

Each artwork design in Seetrax XL Designer has 63 layers in total available for copper (vias/pads/tracks), power planes, split-power planes, silk-screen and documentation layers - typically only a small number of them are actually used.

The board profile, keepout information and router information are stored in their own categories and not stored as "layers", so do not appear in the layer assignments window.

When a design is created, it has three pre-defined layers that cannot be changed, the two outer copper layers known as the *Top & Bottom* layers, and a layer reserved for holes, usually vias, known as the *Via* layer - even though these layers are always defined, they do not have to be used.

The top and bottom layers must always be used as the outer layers of the board because the pads of surface mounted components will only appear on those layers.

The remaining layers are defined as inner copper layers but they can be changed as required to become silk-screen layers, power/split power planes or documentation layers.

When a component is placed on the artwork, its pads will appear on the top, inner copper and bottom layers (as defined in the component outline).

With these default settings, a single, double-sided or multi-layer artwork can be designed with no further adjustments to the layer assignments.

Layers defined as silk-screen or documentation layers are ignored by the connectivity checking routines.

When a design is started, it may not be known how many layers, or what type of layers will be required so the layer assignments can be changed at any time.

It is recommended that all designs maintain the same layer definitions to aid compatibility between designs and multiple users. (For example, layers 1 & 2 for the top silkscreen labels and outlines, 3 & 4 for the bottom silkscreen labels and outlines, 5 - nn for internal power planes, and so on. Which layers are chosen is irrelevant - just try to maintain the same layer types between jobs.

It is recommended that each layer is assigned a name or "title" to aid its recognition throughout the design.

When the layer assignments are changed, if data already exists on the changed layer, it will not be lost but it may be inappropriate. For example if a silk-screen layer with outlines and labels present were re-assigned as a copper layer, the silk-screen data would be regarded as copper and would probably cause short-circuits, therefore additional changes would be required.

The assembly order of the inner layers in a multi-layer design that does not include blind or buried vias, is irrelevant as far as the design's connectivity is concerned. (There may be practical reasons for specifying that particular layers such as ground planes are inserted in a particular position in the layer stack, but their position will not affect connectivity.)

However, if blind/buried vias are used in a design, the order of the layers in the layer stack becomes critical for connectivity. It is therefore essential that the layers in the board layer stack are given a position in the board layer stack and this order is adhered to throughout the design and manufacturing process.

To view and/or change the layer assignments and ordering for a design:

With the Design folder open, double-select the *Configuration* folder, then either double-select *Layer Assignments & Ordering* with the left mouse button, or right-click it, and then select edit. The layer assignments/ordering window appears, as shown in Figure 47.

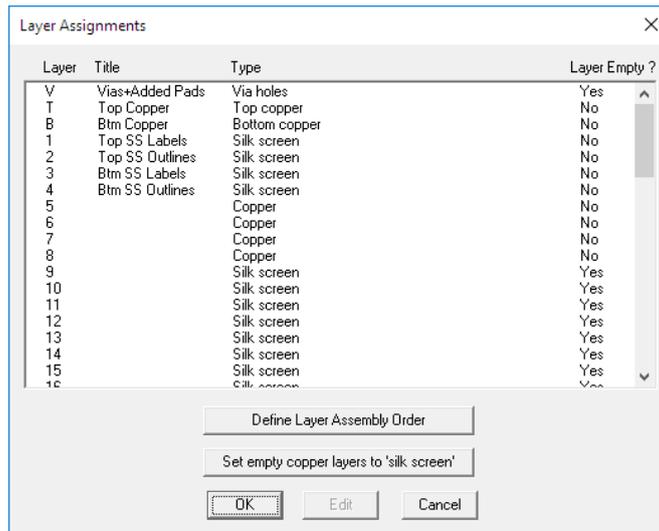


Figure 47

This window lists the layers of the board with their user-defined titles (names), layer types, i.e. copper, silk-screen, etc. and whether the layers are empty.

Layer

This lists the layers, starting with the the layer reserved for Vias, followed by the two outer board layers (T for Top and B for Bottom) then layers numbered from 1-63 which can be used for silkscreen, documentation, copper or power planes. These cannot be changed.

Title

To aid recognition and speed design work up, it is highly recommended that the layers are given names or “titles”, so that it isn’t necessary to keep cross-referncing which layer is used for what.

Examples: Vias & Added Holes, Top Copper, Bottom Copper, Topsilk Labels, Topsilk Outlines, Btmsilk Labels, Btmsilk Outlines, GND Plane, Split (0V, -5v, Vbb), Innercopper 1, Innercopper2, etc.

The titles will be displayed in dialogue boxes where appropriate, example shown arrowed in Figure 48, and tooltip balloons will also provide the information, shown in Figure 49.

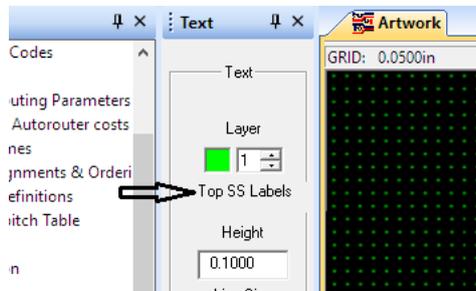


Figure 48

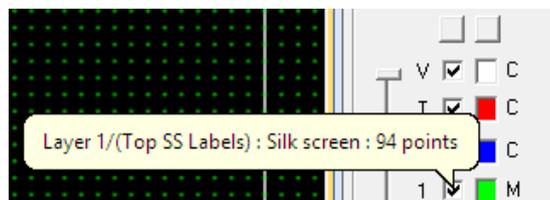


Figure 49

Type

V - Via holes

This layer is reserved for pads with holes - typically "vias", including blind and buried vias. Vias are pads that are drilled & plated and used purely to link tracks from one layer to another to maintain connectivity between copper layers.

Vias (including blind and buried vias) do not appear on any other layers. So when producing output plots, the Via layer should be included with any other layers if the pads with holes/vias should be included.

A pad added to the V layer in the artwork editor (using *Amend > Enter Pads*) will be drilled.

Everything added to the via hole layer is checked by the artwork checking routines.

T - Top copper This layer is reserved for use as the top, outer copper layer of the board. It is often referred to as the "top, component side".

Component pads added to the top layer in the outline's pad stack will appear on this layer (unless the outline has been flipped, in which case they will appear on the Bottom layer).

Everything added to the Top copper layer is checked by the artwork checking routines.

B - Bottom copper This layer is reserved for use as the bottom, outer copper layer of the board. It is often referred to as the "bottom, solder side".

Component pads added to the bottom layer in the outline's pad stack will appear on this layer (unless the outline has been flipped, in which case they will appear on the Top layer).

Everything added to the Bottom copper layer is checked by the artwork checking routines.

Layers 1 to 61 These layers will be assigned as one of the following types:

Copper: these are the inner copper layers, used in a multi-layer design. They always contain the "inner pads", as defined by the component outline pad stacks.

Typically tracks are added to inner copper layers to form the connectivity required between component pins when there is insufficient room on the outer layers.

Copper layers can also contain text and extra pads.

Everything (pads, tracks, text) added to a copper layer is regarded as being a conductor of electricity and checked by the artwork connectivity checking routines.

Silk screen: these layers are empty until data is specifically added to them.

Anything added to silk-screen layers is assumed to be non-conducting by the artwork connectivity checking routines.

These layers typically contain everything for the silk-screen ident of the board.

If parts are mounted on both sides of the board, separate silk-screen layers are required for the top and bottom silk-screen ident.

Tracks and pads can be added to silk-screen layers but they will be considered as non-conducting by the artwork connectivity checking routines and ignored.

Documentation: these layers are empty until data is specifically added to them.

Anything added to documentation layers is assumed to non-conducting by the artwork connectivity checking routines.

These layers are typically used for creating manufacturing drawings or for data that has no electrical significance.

The output tasks can be used to send data to a documentation layer, for example the standard drill drawing. This allows extra data, for example manufacturing notes, to be added.

Tracks and pads can be added to the documentation layers but they will be considered as non-conducting by the artwork connectivity checking routines and ignored.

Named power: Note: A layer can only be defined as a named power layer after the power rail has been added to the power names list in the design's configuration folder.

When a layer is assigned as a named-power layer, the layer becomes a solid sheet of copper (a plane) that will cause short-circuits to all the holes that pass through the layer. The power plane tools in the artwork editor are used to isolate the appropriate holes from the plane and when complete, the layer will contain heat-relief and anti-pads around the appropriate holes to ensure the connectivity of the named power net.

If the artwork checking routine is run before the power plane tools have been used, then every drilled hole through the plane will be reported as causing a short circuit to the plane.

Any of the named power rails can be selected in order to define the layer as an individual power plane for that power rail.

A power rail can be defined on multiple power plane/split plane layers if required.

Auto-generated power plane layers are included by the artwork connectivity checking routine, with some exceptions, so refer to the artwork Check > Connectivity command for full details.

Split power:

When a layer is assigned as a split-power layer, the layer becomes a solid sheet of copper (a plane) that will cause short-circuits to all the holes that pass through the layer. One power rail is chosen as the *primary* power rail and this covers the entire layer. Other power rails are chosen as *secondary signals* and these (using the power plane tools in the artwork editor) will be embedded into the primary plane as islands of copper surrounded by an isolation gap to ensure correct connectivity.

When complete, the layer will contain isolation lines and heat-relief and anti-pads around the appropriate holes to ensure the connectivity of the named power nets.

The artwork checking routine cannot be run until the split-plane has been generated (using the power plane tools in the artwork editor).

Auto-generated split power plane layers are included by the artwork connectivity checking routine, with some exceptions, so refer to the artwork Check > Connectivity command for full details.

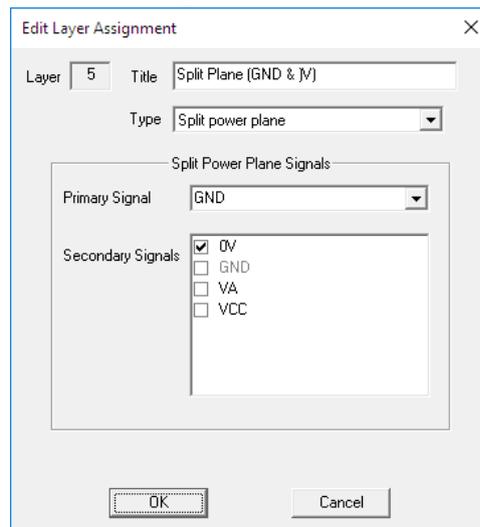


Figure 50

Once the layer has been selected as a Split power plane (Figure 50), choose the power rail that will become the Primary Signal by selecting the arrow alongside the *Primary Signal* setting and selecting the power name from the list that appears.

Note: the power names only appear after they have been added to the power names list in the design's Configuration folder.

Once the primary power rail has been selected for the layer, select the power signals that will be embedded in the primary plane, by ticking the check boxes alongside their names in the *Secondary Signals* list.

Select OK to continue.

Empty Layers

An empty layer is one that has not had pads/copper/silk-screen added to it – component pads are not included, though they will be present on the top/inner/bottom copper layers as defined in the outline library shape.

So for example once parts have been placed, the copper layers will be “empty” but they will have component

pads on them.

Changing the layer titles/types

Layers V, T and B are pre-defined layers and cannot be re-assigned, but they can be given titles.

The remaining layers, 1 to 61 can be defined as inner copper, silk-screen, power/split-power plane or documentation layers.

To change the type of layer for layers 1 to 61, or assign a Title to any of the layers, double-select the layer and a window similar to the one shown in Figure 51 appears. The lower part of the window lists the power rails that have been assigned in the design and becomes active if the layer type is altered to a *Split Power Plane*.

Type in the text to be assigned to the layer Title, then select the down-arrow alongside the current *Type*, then choose the new layer type required from the list that appears.

Select *OK*. The layer type is changed. Repeat as required.

Note: to re-assign all empty layers as silk-screen layers in one step, see the heading: *Set empty copper layers to silk-screen* below.

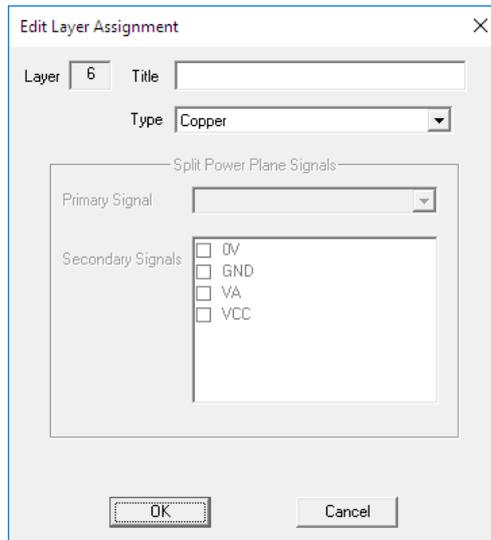


Figure 51

Layer Assembly Order

This button is greyed out until at least one inner copper layer (1 to 61) is defined.

For designs that do not use blind/buried vias, the layer assembly order is not important for connectivity purposes - provided the outer layers are on the outside of the board and inner layers are on the inside of the board when assembled, the connectivity will be correct provided the artwork checks reported no errors. It is not necessary to define the assembly order for this type of board.

If the design uses blind/buried vias, then the order that the board layers are assembled in, becomes very important. If the board is assembled in a different order to that which it was designed and checked against, then it will be incorrect - tracks will become open-circuit or shorted.

The artwork checking routines perform the connectivity checking on the artwork using the assembly order specified here. It is important that the layer assembly order is defined prior to routing and maintained through to board assembly.

Note: once the artwork has been started, changing the order of the layers will affect connectivity if blind or buried vias are used, so it is not recommended.

Defining the Layer Assembly Order

From the *Layer Assignments* window, select *Define Layer Assembly Order*, a window similar to the one shown in Figure 52 appears.

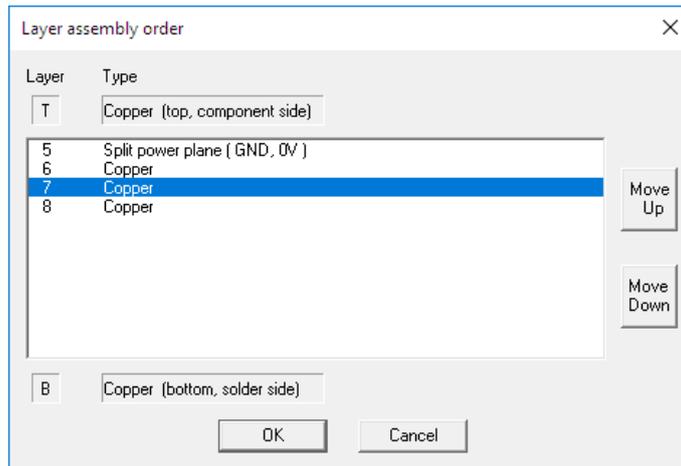


Figure 52

Only the copper and power plane layers defined in the design appear in this list, so more or less layers may appear.

The order of the inner layers can be changed by selecting a layer, then moving it up or down, using the *Move Up* or *Move Down* buttons. The outer layers T and B remain fixed as the outer layers.

It seems logical to define the layers in numerical order, but this is entirely at the designer's discretion.

Once the order has been defined, select *OK* to continue.

Set empty copper layers to silk-screen

This utility provides a quick method of assigning all unused copper layers to silk-screen layers. When using blind/buried vias and to avoid confusion, it is advisable to set all the unused layers to silk-screen layers so that they are not included in the artwork connectivity check.

From the *Layer Assignments* window, select *Set empty copper layers to silk-screen*, all the inner copper layers that are empty will be set as silk-screen layers.

Notes for users of all previous versions of Ranger

- If the Ranger system you were using referred to layers 0 onwards, then the following will apply:
 Layer 0 is now referred to as V (Vias/added Holes)
 Layer 1 is now referred to as T (Top copper)
 Layer 2 is now referred to as B (Bottom copper)
 Layers 3 to 63 are now layers 1 to 61 (user-defined layers)
 This may initially cause confusion but the decision has been made deliberately in order to support blind/buried vias in a logical way.
 Old layers 3 to 63 will remain set to the same type, but their numbers will decrease by 2, for instance old layer 3 will become layer 1, old layer 12 will become layer 10, etc.
- Users who defined their own component outline library with layer 1 as the bottom of the board and layer 2 as the top of the board will have to remember that their top and bottom layers will be reversed if they continue to use their own outline library.

Additional notes for Ranger1/Ranger2 users

The following differences should be taken into account when older designs are opened in Seetrix XL Designer:

- Pad stacks:
 Surface mounted component pads were added to the outer layers of the board (layer 1 or layer 2), these are still added to the outer layers of the board, which are now called T and B.
 Drilled component pads were added to layer 0.
 However, drilled *component* pads in Seetrix XL Designer are made up from a "stack of pads". This allows different sizes/shapes of pads to be used on the top, inner copper and bottom layers of the board.
 Outlines that have been converted from Ranger1/2 with pads on layer 0, will have "padstacks" in Seetrix XL Designer instead of a pad on layer 0 (or new layer V). Each pad in the stack uses the same size/shape of pad as was previously used on layer 0. For instance if a code 6 square pad was placed on layer 0 in Ranger1/2, this will be replaced by a code 6 square pad on the top, inner and bottom

layers of the stack in Seetrax XL Designer.

In Seetrax XL Designer, only *vias* and *added pads* appear on layer V (old layer 0) - there are no drilled *component* pads on this layer. Instead, the component pads appear on each individual copper layer. This means they are not always displayed in white - they will be displayed using the colours for the visible layers. So for instance with only layer T visible (red) the component pads will be red. With T and B visible (red and blue), if the top and bottom pads are the same size and shape they will be displayed in pink/purple (red and blue make pink/purple).

Configuration folder – Via Hole Definitions

User-definable vias can be defined to allow the use of blind and/or buried vias and to allow different via sizes for power tracks.

Code 0 round/square pads will continue to be used for vias unless user-defined vias are defined and used.

User-defined via examples are given later in this description.

The following information is supplied as a general guide but no liability is accepted for incorrect information - please talk to your board manufacturers for current advice and information.

Points to Bear In Mind If Considering Using Blind/Buried Vias

Cost of manufacture:

For users that have never designed with blind/buried via types before, it is worthwhile investigating the cost of manufacturing these types of boards - they are more expensive than conventional multi-layer boards, although prices may start to come down as the technology improves. Blind vias seem to be in fairly common use, but buried vias are quite rarely used. Find out before incorporating either of them in your design.

Manufacturing techniques:

Find out how the actual boards will be assembled, as this will affect how the board is designed. It would be quite easy to design a board that can't be made with the current technology.

What Seetrix XL Designer can do:

Blind vias, buried vias and through-board power via stacks can be defined within Seetrix XL Designer. The design can be auto-routed using the Spectra or Electra auto-routers which will use these vias. The Seetrix auto-router will not use user-defined vias.

The artwork checks take into account and fully check the user-defined via stack definitions.

The resultant artwork can be output with the usual output tools. The drilling output routines (drill sheet drawing and NC Drill) have been designed to incorporate the separate output files required for the different drill datasets of the design.

The manual routing facilities within the artwork editor allow user-defined vias to be moved (*Mroute > Corner*) or replaced with standard vias (*Mroute > Layer Swap*). User-defined via stacks can be entered, moved, replaced and deleted using the *Amend* commands.

Board Manufacture for Boards with Blind/Buried Vias

Before describing what goes on within the software, let's have a look at how the board layers can be assembled - there are other methods in common use and they may affect how the blind/buried vias and drill sets are defined within Seetrix XL Designer - discuss this with your board manufacturer.

Four layer board

In Figure 53 below, the cross-section view of a 4 layer board is shown.

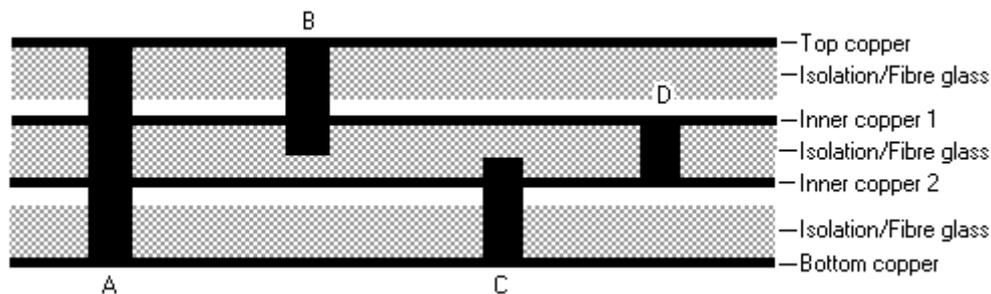


Figure 53

The copper layers are called Top, Inner1, Inner2 and Bottom, with fibreglass/isolation between them, as shown.

Inner layers 1 and 2 are made as a double-sided board, then the outer layers added and the final assembly etched as a double-sided board. In this board there are 4 via types:

- A is a **conventional** (standard code 0 pad) via that is drilled through all layers.
- B is a **blind** via. It is drilled after the board layers have been assembled, from the **top** of the board to a set depth so that the top copper and inner copper layer 1 are joined.
- C is also a **blind** via. It is drilled after the board layers have been assembled, from the **bottom** of the board to a set depth so that the bottom copper and inner copper layer 2 are joined.
- D is a **buried** via. It can only be drilled before all the board layers are assembled.

Inner copper layers 1 and 2 have to be made as a double-sided board and the buried vias drilled/plated at that stage. All the layers are then assembled and the blind and conventional holes drilled. (It is this double-process

(effectively making two, double-sided boards) that makes boards with buried vias expensive to make. They also have higher failure rates.)

From this information, it should become apparent that four sets of drill data outputs will be required.

- i) one for conventional via holes (this file would also include all conventional component holes as well) where the drill, drills straight through the board
- ii) one for buried holes between the two inner layers
- iii) one for blind holes drilled from the top of the board
- iv) one for blind holes drilled from the bottom of the board

Six layer board

In Figure 54 below, we see the cross-section view of a 6 layer board.

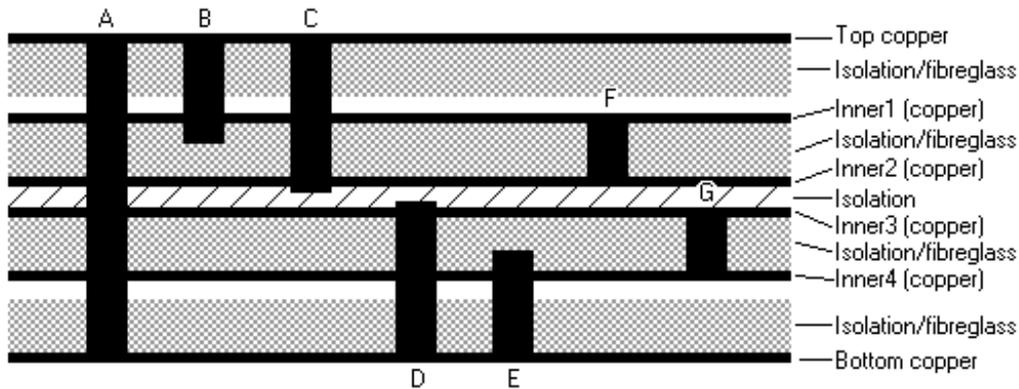


Figure 54

The copper layers are called Top, Inner1, Inner2, Inner3, Inner4 and Bottom, with fibreglass/isolation between them, as shown.

Inner layers 1 and 2 are made as a double-sided board, as are inner layers 3 and 4. These two "boards" are sandwiched together then the outer layers added and the final assembly etched as a double-sided board. In this board there are 6 via types:

- A is a **conventional** (standard code 0 pad) via that is drilled through all layers.
- B is a **blind** via. It is drilled after the board layers have been assembled, from the **top** of the board to a set depth so that the top copper and inner copper layer 1 are joined.
- C is a **blind** via. It is drilled after the board layers have been assembled, from the **top** of the board to a set depth so that the top copper and inner copper layers 1 and 2 are joined.
- D is a **blind** via. It is drilled after the board layers have been assembled, from the **bottom** of the board to a set depth so that the bottom copper and inner copper layers 4 and 3 are joined.
- E is a **blind** via. It is drilled after the board layers have been assembled, from the **bottom** of the board to a set depth so that the bottom copper and inner copper layer 4 are joined.
- F & G are **buried** vias (between different layers). They can only be drilled before all the board layers are assembled.

Inner copper layers 1 and 2 have to be made as a double-sided board and the buried via "F" drilled at that stage. Inner copper layers 3 and 4 are also made as a double-sided board and the buried via "G" drilled at that stage. These two "boards" and the remaining (outer) layers are then assembled and the blind and conventional holes drilled. (In this case, three double-sided boards are effectively made.)

Seven sets of drill data outputs will be required for this board.

- i) one for conventional via holes (this file would also include all other conventional component holes as well).
- ii) one for buried holes between inner layers 1 and 2.
- iii) one for buried holes between inner layers 3 and 4.
- iv) one for blind holes drilled from the top of the board, to reach inner copper layer 1.
- v) one for blind holes drilled from the top of the board, to reach inner copper layer 2.
- vi) one for blind holes drilled from the bottom of the board, to reach inner copper layer 4.
- vii) one for blind holes drilled from the bottom of the board, to reach inner copper layer 3.

It would be possible to define a buried via between inner layers 1, 2, 3 and 4 but this could add extra expense - talk to your board manufacturer.

Eight layer boards, onwards

Use the information for a 6 layer board, simply adding extra layer pairs as required and adjusting the names

accordingly.

Note: there is a limit to the depth that blind via holes can be drilled, the hole diameters also tend to have to increase with increasing depth, so this should be discussed with the manufacturer before defining deep, blind vias.

Drill set names

The drill datasets are numbered from 1 to 18 which are listed as DS01 through to DS18.

The first two datasets are reserved for through the board holes which are plated (DS01) and non-plated (DS02). To avoid confusion, it makes sense for the other drill datasets to be given names in order to differentiate them from one another.

Logical names should be assigned to assist recognition - upto 20 characters are permitted within Seetrax XL Designer for these names.

Drill set names example: - Four layer board

Blind T/1	for blind holes drilled from the top of the board to inner layer 1 (B in Figure 53).
Blind B/2	for blind holes drilled from the bottom of the board to the first inner layer (from this direction), inner layer 2 (C in Figure 53).
Buried 1/2	for buried holes between inner layers 1 and 2 (D in Figure 53).

Conventional via holes (through the board) are usually output in drill dataset one (DS01).

Drill set names example: - Six layer board

Blind T/1	for blind holes drilled from the top of the board to the first inner layer, inner layer 1 (B in Figure 54).
Blind T/1/2	for blind holes drilled from the top of the board to the first and second inner layers, inner 1 and 2 (C in Figure 54).
Blind B/4	for blind holes drilled from the bottom of the board to the first (from this direction) inner layer, inner 4 (E in Figure 54).
Blind B/4/3	for blind holes drilled from the bottom of the board to the first and second inner layers (from this direction), inner layers 4 and 3 (D in Figure 54).
Buried 1/2	for buried holes between inner layers 1 and 2 (F in Figure 54).
Buried 3/4	for buried holes between inner layers 3 and 4 (G in Figure 54).

Conventional via holes (through the board) are usually output in drill dataset one (DS01).

Notice that the drill set names have changed for the holes drilled from the bottom of the board, when compared against the 4 layer board, because of the extra layers.

Drill/pad sizes for blind/buried via

The following sizes were supplied as **minimum** guidelines from one manufacturer, individual manufacturers may have different sizes that can be achieved - if slightly bigger holes/pads can be used then the costs may come down.

Blind vias can be "drilled" up to 14 layers deep. The deeper the blind via, the larger the drilled hole has to be, this will obviously affect the pad size as well. As a rule of thumb, the hole size increases by approximately 50% for each extra layer (because of problems plating deep, narrow holes).

Minimum drill/pad sizes from one manufacturer

- Blind via from an outer layer to 1 inner layer, 0.004" hole and 0.012" pad.
- Blind via from an outer layer to 2 inner layers, 0.006" hole and 0.012" pad.
- Blind via from an outer layer to 3 inner layers, 0.008" hole and 0.015" pad.
- Buried vias, 0.012" hole and 0.020" pad.
- Ranger's default blind vias have a 0.010" hole and 0.020" pad defined.
- Ranger's default buried vias have a 0.015" hole and 0.025" pad defined.

Board layers - Assembly

When the layers of the above boards are assembled, they **MUST** be assembled in the same order as they were designed, so it is essential that a system of naming/numbering the layers is maintained to avoid confusion and boards that don't work.

Defining layer order within Seetrax XL Designer

Layers **Top** and **Bottom** should always be the **outer** copper layers of the board, as surface mounted component pads appear on them.

All the numbered copper layers within Seetrax XL Designer will be internal layers and the numbers start at 1. It

is therefore logical to use sequential layer numbers so the board will be assembled as T, 1, 2, 3, 4, 5 (etc.) B. On designs made with previous versions of Ranger, layers 3 to 63 will become layers 1 to 61 - they will maintain the same usage, ie silk-screen, power-plane, etc. Users who defined their own outline library with layer 1 as the bottom of the board and layer 2 as the top of the board will have to remember that their top and bottom layers will be reversed.

When designing a board without blind or buried vias, the order in which the layers are assembled is not crucial, provided the T(op) and B(ottom) layers are used as the outer layers and inner copper layers/power plane layers are in between them in any order - the electrical connectivity will be maintained/correct.

However, when designing an artwork that includes blind or buried vias, the software has to know the order the board layers will be assembled in, in order to maintain connectivity when routing the tracks and then to check the artwork correctly.

If they are assembled in a different order to the order in which they were designed and checked, then connectivity will be lost, the artwork checking routine results will be invalidated and the board will not be correct.

Setting the layer order

With the design open, double-select the *Configuration* folder, then either double-select *Layer Assignments & ordering* with the left mouse button, or right-click it, and then select edit. The *Define Layer Assembly Order* button is used to define the order in which the layers will be designed and checked - it is inoperative until there are some inner copper/plane layers defined. This window is described under the heading: *Configuration – Layer Assignments & Ordering*

Boards with power planes

Depending on where the plane layer is positioned in the board assembly "sandwich", heat-relief or anti-pads (as appropriate) will be added to the hole on the plane layer, if the hole reaches the plane layer. (There is an option of not using heat-relief pads on the blind/buried hole definitions if preferred - the holes are usually quite small so shouldn't cause a problem with heat dissipation.)

Defining Blind/Buried Via stacks - Examples

Open the design where the blind/buried vias will be used.

With the design open, double-select the *Configuration* folder, then either double-select *Via Hole definitions* with the left mouse button, or right-click it, and then select edit. A window similar to the one in Figure 55 appears.

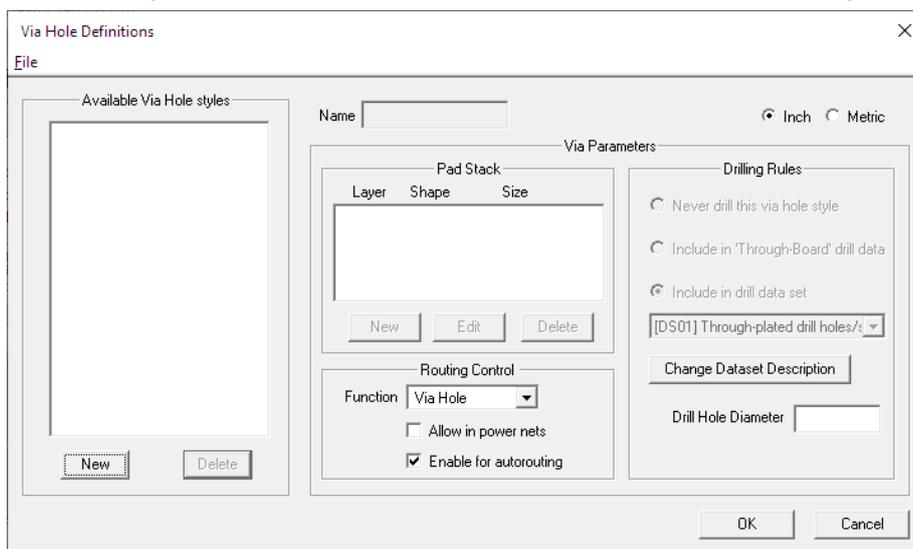


Figure 55

There are four steps to define a blind or buried via in this window, as follows:

- Step 1: Define the "*Via Hole Styles*"
- Step 2: Define the "*Pad Stack*"
- Step 3: Define the "*Routing Control*" information for the Specctra/Elecctra auto-router
- Step 4: Define the "*Drilling Rules*"

Examples follow, describing how to define a variety of blind and buried vias. Example 1 provides a full explanation; the other examples follow on from this information so less descriptive information is given.

Note: All the examples assume that inner copper layers/power plane layers have been defined, the *Via Hole Definitions* window is open and the inner layer order is in numerical order starting from 1.

Example 1: Defining a blind via to be used from the top copper layer to the first inner layer, inner layer 1.



Figure 56

Figure 56 shows a cross-section of the via to be defined. This via could be used on 4, 6, 8, 10, etc. layer boards.

Step 1: Via Hole Styles

Select the *New* button from the "Available Via Hole Styles" area of the window, then enter a name for the blind via - as a suggestion for this example, use *Blindvia T/1* (blind via from the top layer to layer 1) then select *OK*.

(The via can be renamed if required - select the name then re-type it from within the box above the *Pad stack* information. Press the <tab> key to implement the change.)

Step 2: Pad Stack

Select the *Blindvia T/1* from the "Via Hole Styles" list. This name now appears above the "Pad stack" information and the pad stack can be defined.

Select the *New* button from the "Pad Stack" area of the window, the window shown in Figure 57 appears.

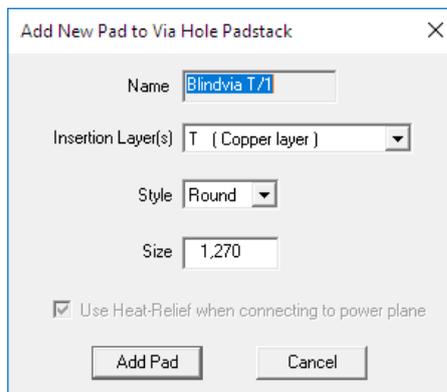


Figure 57

The *Insertion layer* should be set to one of the layers the pad will appear on, **T** for the top copper layer in this example.

Choose a *Style* (round or square) for the pad and change the size of the pad as required.

Select *Add Pad*. The window closes and the details appear in the pad stack definition. (If a mistake was made and the details need to be changed, select them then press the *Edit* button and make the changes.)

With *Blindvia T/1* from the "Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 1 (the first inner layer), set the style/size as required, then select *Add Pad*.

If layer 1 was defined as a power plane layer, then the *Style* and *Size* settings are greyed out - a heat-relief or anti-pad will be used as appropriate. The "Use Heat-relief when connecting to power plane" check box becomes selectable. If ticked, a heat-relief pad will be placed over the hole on the power plane layer if the hole should be connected to it. If un-ticked, a heat-relief pad is not used.

This completes the *Pad stack* for this blind via style *Blindvia T/1*, the pad stack information will look something like that shown in Figure 58.

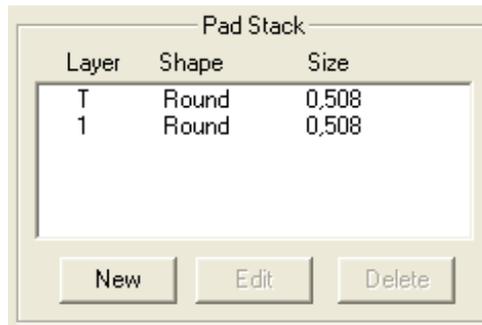


Figure 58

Step 3: Routing control

For a blind (or buried) via, the *Function* should be set to *Via Hole* as shown in Figure 59.

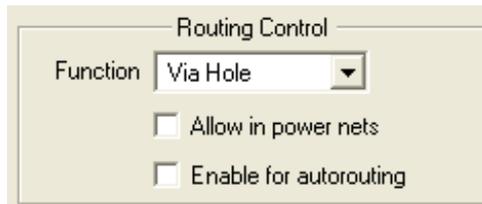


Figure 59

The tick boxes should be set as required:

Allow in power nets indicates whether the Spectra/Elecetra auto-routers are allowed to use the blind/buried via stack in power tracks. If the *Enable for auto-routing* setting is not selected, then this setting is ignored as the stack will not be used.

Enable for auto-routing indicates whether the Spectra/Elecetra auto-routers are allowed to use this blind/buried via stack when auto-routing. If this is disabled, then the *Allow in power nets* and *Allow in SMD* settings are ignored.

Use heat-relief when connected to power plane (inoperative unless a power plane is selected as the insertion layer) indicates whether a heat-relief pad should be used when the via is used to make a connection to the power plane. If unticked, a heat-relief pad is not added around the hole.

These design rules may vary with each design.

Step 4: Drilling rules

For blind or buried vias the *Drilling Rules* should be set to "Include in drill data set", as shown in Figure 60. When selected, the other settings become editable.

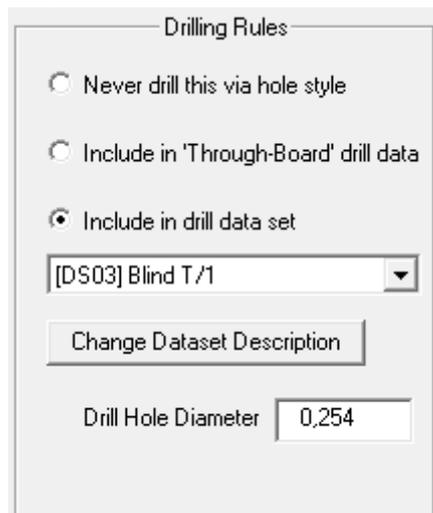


Figure 60

Drill sets are defined so that individual drill output files can be produced for each different set of blind/buried holes as described above. For instance, drill information for blind vias between the top and first inner layers should not be included with the drill information for blind vias between the top, first and second inner layers.

The drill set names are numbered from DS01 to DS18 (DS01 and 02 are reserved for through board

holes).

It's suggested the names are changed to something easily recognisable, for instance *Blind Top* or *Blind Top/1* for blind vias drilled from the top of the board to the first inner layer. Use the *Change Set Name* button to implement this. The number/name is not important but which drill set each blind/buried via hole is added to, is important - the boards will be drilled incorrectly if a mistake is made.

Set the drill hole diameter as required for this pad stack - talk to your manufacturer regarding the diameter of blind vias that they can achieve.

This completes the definition of this blind via stack. If there are more blind/buried vias to be defined, leave the *Via Hole Definitions* window open, if not, select *OK* to continue.

Example 2: Defining a blind via to be used from the top copper layer to the first and second inner layers, inner layers 1 and 2.

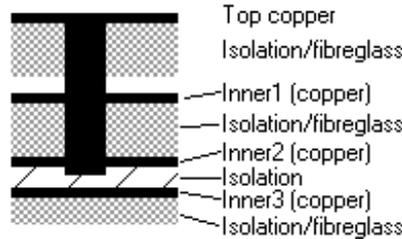


Figure 61

Figure 61 shows a cross-section of the via to be defined. This via could be used on 6, 8, 10, etc. layer boards.

Step 1: Via Hole Styles

Select the *New* button from the "Available Via Hole Styles" area of the window, then enter a name for the blind via - as a suggestion for this example, use *Blindvia T/1/2* (blind via from the top layer to layer 1 and layer 2) then select *OK*.

Step 2: Pad Stack

Select the *Blindvia T/1/2* from the "Available Via Hole Styles" list. This name now appears above the "Pad stack" information and the pad stack can be defined.

Select the *New* button from the "Pad Stack" area of the window, the *Add New Pad to Via Hole Padstack* window appears.

The *Insertion layer* should be set to one of the layers the pad will appear on, *T* for the top layer in this example.

Choose a *Style* (round or square) for the pad and change the size of the pad as required.

Select *Add Pad*. The window closes and the details appear in the pad stack definition.

With *Blindvia T/1/2* from the "Available Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 1 (the first inner layer, which could be a power plane layer), set the window as required, then select *Add Pad*.

With *Blindvia T/1/2* from the "Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 2 (the second inner layer, which could be a power plane layer), set the window as required, then select *Add Pad*.

This completes the *Pad stack* for this blind via and the pad stack information will look something like Figure 62.



Figure 62

Step 3: Routing control

For a blind (or buried) via, the *Function* should be set to *Via Hole* as shown in Figure 63.

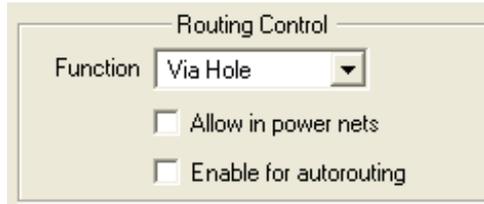


Figure 63

Set the tick boxes as described in example 1.

Step 4: Drilling rules

For blind or buried vias the *Drilling Rules* should be set to "Include in drill data set". When selected, the other settings become editable.

It's suggested that this Drill data set name be changed to something easily recognisable, for instance *Blind T/1/2* (*blind* via drilled from the *top*, through layer *1* and *2*). Use the *Change Set Name* button to implement this.

Set the drill hole diameter as required for this pad stack.

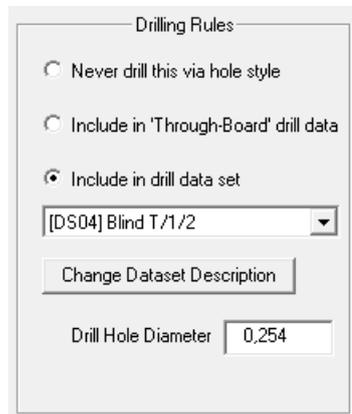


Figure 64

This completes the definition of this blind via stack.

Example 3: Defining a blind via to be used from the bottom copper layer to the next two inner layers from the bottom of the board (inner layers 3 and 4 in a 6 layer board)

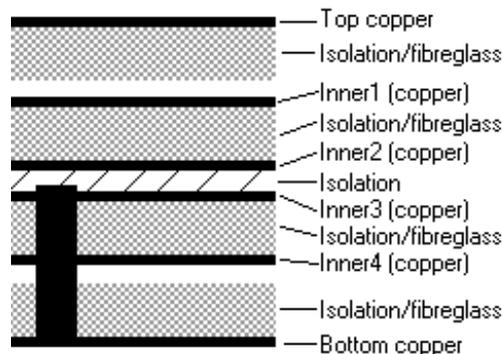


Figure 65

Figure 65 shows a cross-section of the via to be defined.

When defining blind vias that are drilled from the bottom of the board, take care to use the correct inner layer numbers when defining the padstack - these will depend on the number of layers in the board.

For example, on a 6 layer board as shown here, the pads are added to the bottom layer, layer 4 and layer 3, but on an 8 layer board they would be added to the bottom layer, layer 6 and layer 5.

Step 1: Via Hole Styles

Select the *New* button from the "Via Hole Styles" area of the window, then enter a name for the blind via - as a suggestion for this 6 layer design example, use *Blindvia B/4/3* (*blind* via from the bottom layer to layer 4 and layer 3) then select *OK*. (On an 8 layer design the name could be *Blindvia B/6/5* (*blind*

via from the bottom layer to layer 6 and layer 5)

Step 2: *Pad Stack*

Select the *Blindvia B/4/3* from the "Via Hole Styles" list. This name now appears above the "Pad stack" information and the pad stack can be defined.

Select the *New* button from the "Pad Stack" area of the window, the *Add New Pad to Via Hole Padstack* appears.

The *Insertion layer* should be set to one of the layers the pad will appear on, *B* for the bottom layer in this example.

Choose a *Style* (round or square) for the pad and change the size of the pad as required.

Select *Add Pad*. The window closes and the details appear in the pad stack definition.

With *Blindvia B/4/3* from the "Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 4 (the first inner layer from the bottom of the 6 layer board, which could be a power plane layer), set the window as required, then select *Add Pad*.

With *Blindvia B/4/3* from the "Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 3 (the second inner layer from the bottom of the 6 layer board, which could be a power plane layer), set the window as required, then select *Add Pad*.

This completes the *Pad stack* for this blind via and the pad stack information will look something like Figure 66. In this example the inner pads are smaller than the one used on the outer layer.

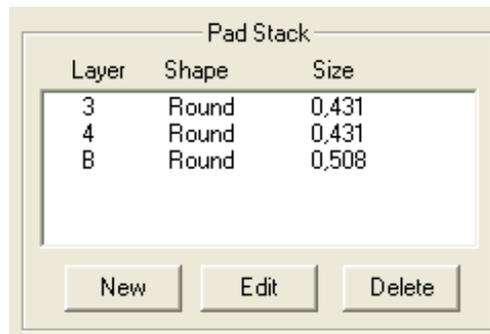


Figure 66

Step 3: Routing control

For a blind (or buried) via, the *Function* should be set to *Via Hole* as shown in Figure 67.



Figure 67

Set the tick boxes as described in example 1.

Step 4: Drilling rules

For blind or buried vias the *Drilling Rules* should be set to "Include in drill set data". When selected, the other settings become editable.

It's suggested that the Drill data set names be changed to something easily recognisable, for instance *Blind Bottom/4/3* or *Blind B/4/3* (*blind* via drilled from the *bottom*, through layer 4 and 3). Use the *Change Set Name* button to implement this.

Set the drill hole diameter as required for this pad stack.

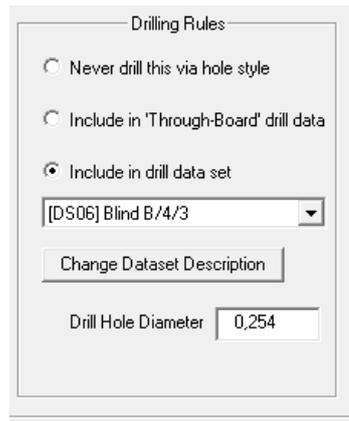


Figure 68

This completes the definition of this blind via stack.

Example 4: Defining a blind via to be drilled from the bottom copper layer to the next inner layer in from the bottom of the board, inner layer 4 in a 6 layer board

The same procedure as in example 3 should be followed with the following differences:

The *Via hole style* name could be *Blindvia B/4*.

The *padstack* should only include pads on the bottom layer and inner layer 4.

The *Routing control* should be set as required.

The *Drill rules* should be set for a new drill data set, which could be called for instance *Blind Bottom* or *Blind B/4*.

Example 5: Defining buried vias

Buried vias are defined in a similar way to blind vias. Buried vias should be defined between pairs of internal layers. In theory a buried via could be defined between more than two layers however this would increase the board manufacturing costs so it is not recommended.

In Figure 69, three buried vias are shown, between inner layers 1 and 2, inner layers 3 and 4 and inner layers 5 and 6. First a via between inner layers 1 and 2 will be defined.

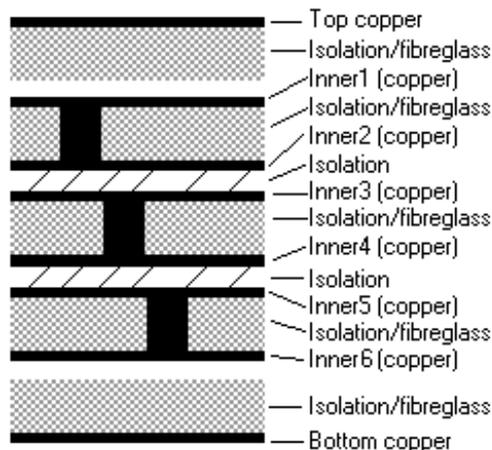


Figure 69

Step 1: Via Hole Styles

Select the *New* button from the "Via Hole Styles" area of the window, then enter a name for the buried via - as a suggestion for the via between layers 1 and 2, use *Buriedvia 1/2* (buried via between inner layer 1 and 2) then select *OK*.

Step 2: Pad Stack

Select the *Buriedvia 1/2* from the "Via Hole Styles" list. This name now appears above the "Pad stack" information and the pad stack can be defined.

Select the *New* button from the "Pad Stack" area of the window, the window shown in Figure 70 appears.

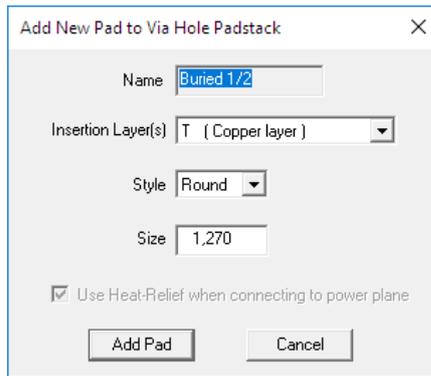


Figure 70

The *Insertion layer* should be set to one of the inner layers the pad will appear on, **1** for the first inner layer in this example.

Choose a *Style* (round or square) for the pad and change the size of the pad as required. Pad sizes for buried vias are usually quite small - talk to your board manufacturer about what they can achieve cost-effectively.

Select *Add Pad*. The window closes and the details appear in the pad stack definition. (If a mistake was made and the details need to be changed, select the pad then press the *Edit* button and make the changes.)

With *Buriedvia 1/2* from the "Via Hole Styles" list still selected, select the *New* button from the "Pad Stack" area of the window again.

This time, set the *Insertion layer* to 2 (the second inner layer), set the style/size as required, then select *Add Pad*.

This completes the *Pad stack* for this buried via style *Buriedvia 1/2*, the pad stack information will look something like that shown in Figure 71.



Figure 71

Step 3: Routing control

For a buried via, the *Function* should be set to *Via Hole* as shown in Figure 72.



Figure 72

Set the tick boxes as described in example 1.

Step 4: Drilling rules

For blind or buried vias the *Drilling Rules* should be set to "*Include in drill data set*", as shown in Figure 73. When selected, the other settings become editable.

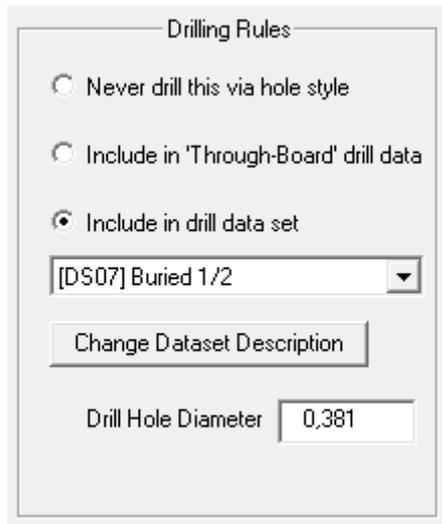


Figure 73

Drill sets are defined so that individual drill output files can be produced for each different set of blind/buried holes as described above. Each buried via layer pair has to be output separately from the others.

The drill data set names are numbered from DS01 to DS18 (DS01 and 02 are reserved for through board holes).

It's suggested the names are changed to something easily recognisable, for instance *Blind Top* or *Blind Top/1* for blind vias drilled from the top of the board to the first inner layer. Use the *Change Set Name* button to implement this. The number/name is not important but which drill set each blind/buried via hole is added to, is important - the boards will be drilled incorrectly if a mistake is made.

Set the drill hole diameter as required for this pad stack. Again, talk to your manufacturer about the diameter of buried vias that they can achieve.

This completes the definition of this buried via stack. If there are more blind/buried vias to be defined, leave the *Via Hole Definitions* window open, if not, select *OK* to continue.

Buried vias between inner layers 3 and 4

Buried vias between inner layers 3 and 4 should be defined in the same way as above except the names and layers used would be adjusted each time. For example, the *Via hole style* name could be *Buried 3/4*. The *padstack* should only include pads on inner layer 3 and inner layer 4. The *Routing control* should be set as required. The *Drill rules* should be set for a new drill set, which could be called *Buried 3/4*.

The same would apply for a buried via between layers 5 and 6, etc. (*Buried 5/6*).

If all the examples have been created as described, then the *Via Hole Definitions* window will look similar to the one shown in Figure 74.

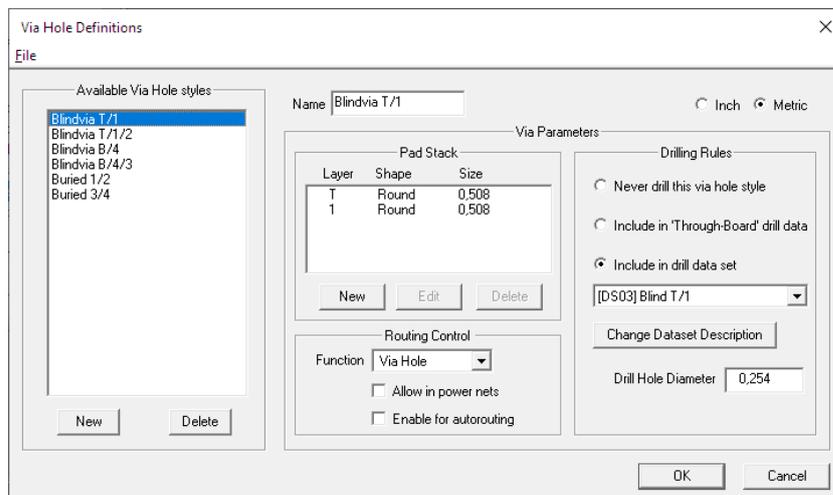


Figure 74

If each *Via Hole Style* is selected in turn, the settings for that via pad stack can be checked and changed if necessary. Select **OK** to complete the via hole definitions.

Example 6: Through board via stacks (for power tracks) – Power Via

Typically code 0 round pads are used for all through-board vias. However, because of the higher current carrying requirements of power tracks when compared to signal tracks it is often a requirement to use larger vias for power tracks. They can be defined as follows:

Step 1: Via Hole Style

Select the *New* button from the "Via Hole Styles" area of the window, then enter a name for the power via - as a suggestion for this example, use *Power via* then select *OK*.

Step 2: Pad Stack

Select the *Power via* from the "Via Hole Styles" list. This name now appears above the "Pad stack" information and the pad stack can be defined.

Select the *New* button from the "Pad Stack" area of the window, the *Add New Pad to Via Hole Padstack* appears.

The *Insertion layer* should be set to *All copper layers* (layers can be defined individually, but this would be time-consuming).

Choose a *Style* (round or square) for the pad and change the size of the pad as required.

Select *Add Pad*. The window closes and the details appear in the pad stack definition.

Step 3: Routing control

For this via, the *Function* should be set to *Via Hole*.

The *Allow in power nets* tickbox must be ticked to allow these vias to be used in power tracks.

Step 4: Drilling rules

For these vias the Drilling Rules should be set to "Include in "Through board" drill data".

Set the drill hole diameter as required for this pad stack.

This completes the definition of through board power via stack.

Defining signal vias

It is possible to create a via stack specifically for signal tracks, although code 0 is usually set to the size required.

If an additional via stack is required, it can be defined in the same way as the power via, but do not enable it for use in power tracks.

Code 0 vias can still be used by the auto-router, but they can be disabled if required (in the Spectra/Elecetra auto-router setup window).

Saving via stacks for use on other designs

Once via stacks have been defined they can be saved to a file and then loaded into other designs. This allows a set of blind/buried vias to be defined for each number of board layers, 4, 6, 8, etc.

Saving via stacks

From the *Via Hole Definitions* window, select *File > Save Via Definitions*. Use the explorer window that appears to locate the folder the file should be saved in, then supply a name for the file (the extension must be .uvd).

Loading via stacks

From the *Via Hole Definitions* window, select *File > Load Via Definitions*. Use the explorer window that appears to locate the folder the file has been saved in, then select the file required.

A default set of blind/buried via stacks for a 6 layer board have been supplied in the file *default_user_vias.uvd* which will be found in the ..\XL Designer\masters\pcb folder. If used, the layer assignments will need to be adjusted accordingly.

Routing with user-defined vias

The parts should be placed in the normal way and any split power planes created.

If vias are allowed in some surface mounted component pads, then edit the properties of those component outlines and enable the *Allow via holes at SMD pads* setting. This will allow vias (blind/buried and through board) to be placed in the smd pad by the Spectra/Elecetra auto-router.

Submit the design to the Spectra/Elecetra auto-router. Once the design has been routed, re-create any power planes and when complete, check the artwork.

The artwork can be manually adjusted if required.

Manually routing with user-defined vias

When manually routing, user-defined vias will be inserted automatically if the "Via Autoselect" checkbox in the Mroute dialog box is ticked. The most appropriate user-defined via will be chosen, according to the layers the via has to connect through. If there isn't a user-defined via available that would maintain connectivity between the layers, then a standard code 0 via will be used.

Blind/buried via colours and point count

In the artwork editor, blind and buried vias are visible if the via layer and the layer they appear on, are visible.

The point count for layer V includes blind and buried vias, they are not included in other layer point counts.

Outputs

All the output files should be generated in the usual way whilst taking into account the following information.

Artwork layers output

When outputting artwork layer information, user-defined via pads are output if the layer they are included on, has Pads included, along with pads on layer V.

"Standard" vias (code 0 pads) are output if the pads on the V (via) layer are included.

Figure 75 shows the typical selections for an inner layer 1 output which would include all standard vias (V layer), inner pads of through-hole components and blind/buried pads on layer 1 (pads on layer 1) and tracks/text on layer 1 (copper on layer 1).

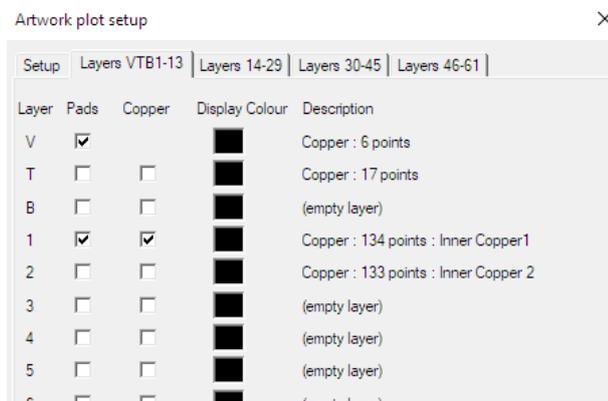


Figure 75

Note: When outputting inner power plane/split plane layers, only the pads and copper from the power plane layer should be output. Layer V should NOT be included.

NC Drill Output

A separate NC drill file will be required for conventional through hole vias/component holes, for each set of blind vias and for each set of buried vias.

When the NC Drill output task is opened, window similar to the one shown in Figure 76 appears.

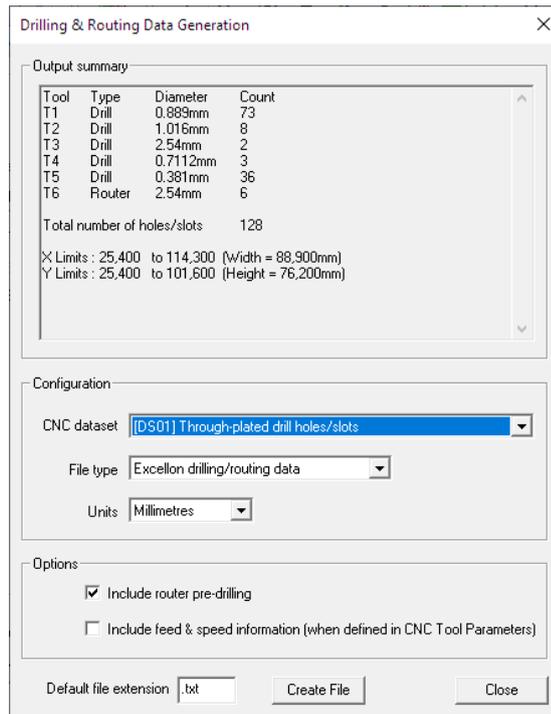


Figure 76

As the window is set at the moment, with *CNC Dataset DS01* selected, the drill output file produced (when the *Create File* button is selected) will contain drilling information for drilled components and "standard vias" - vias that go all the way through the board.

To output the blind or buried via hole information, change the *CNC Dataset* so that the drill dataset required is selected (by selecting the down-arrow alongside the *CNC Dataset*, then select from the list, for instance *Blind T/1* (or whatever it was called)). The tooling/hole information changes accordingly and the drill file can be produced. Repeat this for each Drill dataset used.

When the drill datasets are used for drilling buried vias, it is essential that the correct file is used for the correct buried via layer pairs.

When the drill datasets are used for drilling blind vias, it is essential that the holes are drilled from the correct side of the board (top or bottom) and to the correct depth. Bear in mind that there could be more than one set of blind holes (drill datasets) drilled from the same side of the board, but to different depths.

Drill sheet (drawing) output

If drill sheet drawings are required, a separate drill sheet drawing output will be required for conventional through hole vias/component holes, for each set of blind vias and for each set of buried vias.

Open the output task for the drill sheet, then open the drill sheet setup window, which is shown in Figure 77.

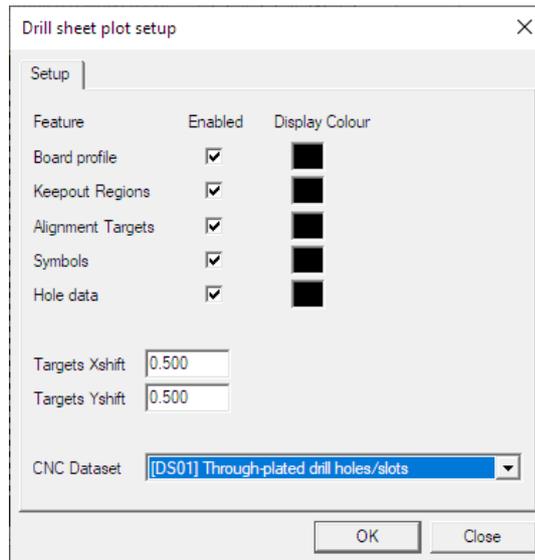


Figure 77

As the window is set above, with the *CNC Dataset* set to "DS01 Through plated drill holes/slots", the drill sheet output will contain hole information for drilled components and "standard vias" - vias that go all the way through the board.

To output the blind or buried via hole information, change the *CNC Dataset* so that the drill dataset required is selected (by selecting the down-arrow alongside the *CNC Dataset*, then select from the list, for instance Blind T/1 (or whatever it was called)). Repeat this for each Drill dataset used. Figure 78.

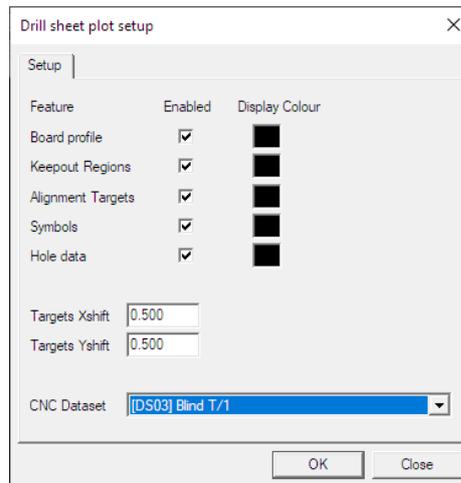


Figure 78

Ensure that each drill drawing is associated with the correct layers and that the holes are drilled from the correct side of the board (top or bottom) and to the correct depth if buried vias are in use. Bear in mind that there could be more than one set of blind holes (drill datasets) drilled from the same side of the board, but to different depths.

Configuration folder – Grid Autopitch Table & Axis Dot Limit

Grid autopitch table

When working in the graphical editors a grid is used to help position items. On many occasions it is useful if the grid gets finer or coarser automatically, depending on the level of zoom. This table allows the user to define the grid pitches that XL Designer can select from when an auto-grid is in use.

When an auto-grid is in use, the most appropriate grid pitch is chosen automatically for each level of zoom. The table, shown in Figure 79, contains two sets of values, one in mm and one in inches.

When viewing the complete board, one of the coarser grids would be used as the finer grids would obliterate everything else on display. Likewise if a single item were filling up the screen, one of the finer grids would be used.

These grid pitches are only used when *Grid > Inch* or *Grid > Metric* grid is selected.

Inch	Metric
0.2000	20.0000
0.1000	10.0000
0.0500	5.0000
0.0200	2.0000
0.0100	1.0000
0.0050	0.5000
0.0010	0.2500
n/a	n/a

(Enter 'zero into a cell to disable it)

Figure 79

The pitch chosen is determined by how many grid dots would be drawn along the longest axis of the workarea window if that pitch were chosen. The grid pitch that fits the \leq "Autogrid pitch change threshold" setting (in the Edit > Display Adjustments window) along the longest axis of the workarea is the grid pitch that is chosen.

Log Files Folder

The Log Files folder is empty initially. It will contain auto-generated reports that contain information associated with specific processes or tasks. The information is supplied for future reference purposes.

The names of the log files should be self-explanatory but if further information is required, open the file to view the log report itself.

They can be opened by double-selecting them (or by a right-click/*Open*).

They can be deleted by a right-click/*Delete*. Care should be exercised when deleting the log reports as the information may be relevant to future processes.

Typical log files:

flatcir_conversion	report provided from the conversion of a circuit drawn using the old-style flat schematic editor to the hierarchical style editor. It is suggested that this report is not deleted as it may be useful for future reference in the event of a problem.
compile	report created by the extraction of a parts/wiring list from the schematic.
circuit_plotfilter	report created when the schematic was output.
art_plotfilter	report created when the artwork was output.
electra_export.txt	report created when the Run Interactively, Run Router or Export commands are used from the Electra interface window.
electra_import.txt	report created when an attempt has been made to import a routed file using the Electra interface window.
specctra_export.txt	report created when the Run Interactively, Run Router or Export commands are used from the Specctra interface window.
specctra_import.txt	report created when an attempt has been made to import a routed file using the Specctra interface window.
renumber	a history of the part renumbering and/or gate/pin swapping that have taken place.
netmods	a history of the changes made in the artwork editor using the Tools > Network commands.

This is not a definitive list and there are many other log files that can be created.

Documentation Folder

The documentation folder allows external file types, such as Spreadsheets, Word documents, etc., to be stored within the design. User-defined file types can be specified and templates created that load automatically when the file type is selected.

The file is stored within the design and can only be opened from within the navigator. Note: if the application required to open the file is removed from the system, then it will not be possible to edit that file until the application is re-installed.

Important Note: The target application (Excel, Word, etc.) should not be running when a new file is created or a file modified as the changes made from within XLD will not be saved.

Creating a new documentation file

From the Navigator, right click on the *Documentation* folder within a design, select *New*, and then choose the file type from the list that appears.

The list of file types available is configured in the <INSTALL_DIR>\UserTemplate\DocTemplates\index.txt file (see *Customising the file types available* below).

The file is added to the documentation folder with the name <Untitled.ext>, an opportunity is given for the file to be re-named when it's created.

Once created, it can be renamed/deleted by right-clicking the file and selecting *Rename/Delete*.

Open the file by double-selecting it, or right-click/*Open*. Once opened, the *Open, Rename & Delete* commands are greyed out until the file is closed.

Importing a documentation file

Any existing file can be added to the documentation folder of the design. Right click on the *Documentation* folder within a design, select *Import*. Use the browser window to locate and select the file required.

Once imported, it is a copy of the original file that is held in the design, not the original itself. If the file is modified from within the design, the original file is unaffected. Likewise if the original file is edited, this does not affect the file held in the documentation folder.

Customising the file types available

The file types that are presented in the *Documentation > New* menu can be customised as required by editing the *index.txt* file in the <INSTALL_DIR>\UserTemplate\DocTemplates folder. This file should be edited whilst Seetrix XL Designer is closed.

This file can be edited provided basic rules are obeyed. The file takes the following form:

Extension1	Description1
Extension2	Description2
Extension3	Description3

An example is shown in Figure 80.

txt	Text Document
doc	Microsoft Word Document
xls	Microsoft EXcel Worksheet

Figure 80

The text string in the description field will appear in the *Documentation > New* menu as shown in Figure 81.

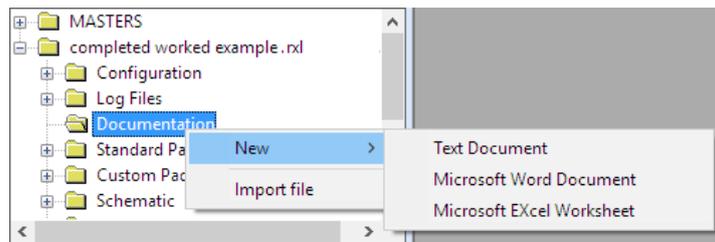


Figure 81

When editing this file, the file extension and description fields must be separated by a single tab character.

Creating a Template for the file type

For each file type defined in the `index.txt` file, if a corresponding file named: `templates.{extension}` is created in the `DocTemplates` folder, then that file will be used as the initial file content for newly created documentation file.

If a template file does not exist for a file type, then the `New` file action will create an empty (0 bytes long) file.

Standard Pads Folder

This folder provides access to the four standard pad shapes and associated sizes. Each pad shape has its own folder. When the folders are opened, a list of the sizes for that shape is presented in the navigator, as shown in Figure 82.

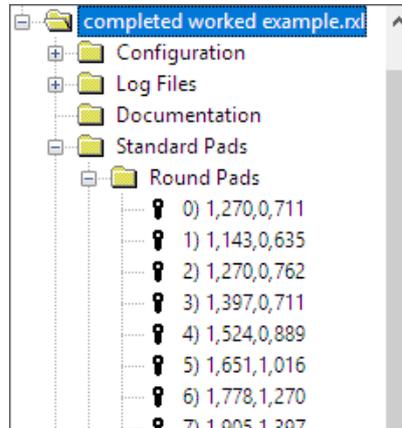


Figure 82

The list only includes the pad codes that have a size assigned to them. The first dimension provided is the pad diameter, or length of the pad if a rectangular shape is selected, followed by the drill size.

If an individual pad is selected, it is displayed in the Browser pane (if visible – View > Browser setting), Figure 83.

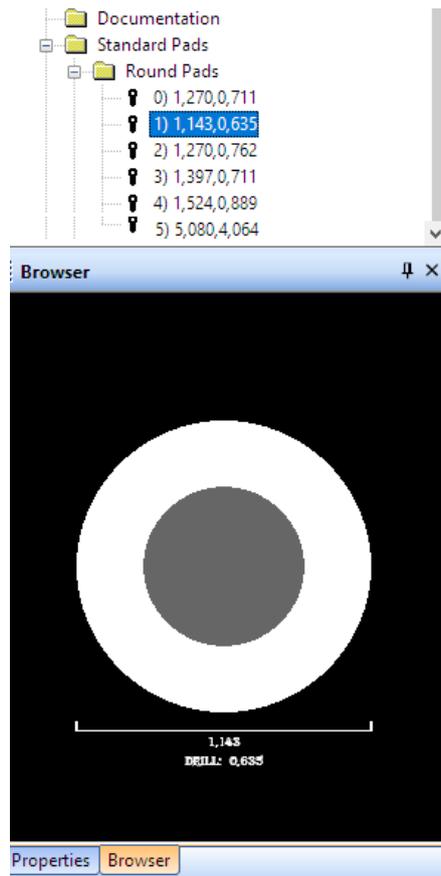


Figure 83

If the Standard Pads folder or any of the pad shape folders, or a pad from the list is double-selected the sizes table appears and the complete list can be viewed and the sizes altered if required.

The sizes table can also be accessed from the *Configuration* folder and it is described in the chapter titled *Configuration Folder – Size Tables, Pad Sizes*.

When adding pads to component outlines, the pad to be added can be dragged from the Standard Pads Folder (unless a *Custom Pad* is being added).

Custom Pad Folder

The *Custom Pad Folder* contains pads that do not conform to the four standard pad shapes.

Standard pads are defined as round pads, square pads, rectangular pads or round ended finger pads that have one hole (if required) in the centre and the connection point also in the centre.

Custom pads are not supplied with the software so the master and design custom pad folders will initially be empty.

Only pads that cannot be defined as standard pads should be added to the custom pad folder. It is inefficient to define standard pads using custom pads in terms of time and space.

Each design has its own custom pad folder, which will contain the non-standard pads used on that particular design, plus any other pads the user may wish to include.

Custom pads can be copied/pasted to/from other custom pad folders (other designs or the master folder), using the right-click Copy/Paste tools. Custom pads can also be created directly in the folder.

Once custom pads have been copied into the design, changes to them do not affect the original pads, other designs or the master custom pad folder.

Changes to custom pads in the master custom pad folder do not affect any other designs.

The custom pad folder is automatically updated if outlines containing custom pads are copied to the design/master outline folder - providing the pads are available in the source file.

It is suggested that custom pads are created within a design folder and verified before they are copied to the master folder. Once custom pads are in the master folder it is assumed they are correct.

Opening/viewing the custom pads folder and its content

Locate the custom pads folder required from the navigator window - the master custom pads folder is held inside the *Masters* folder, whilst each design's custom pads folder is held inside the design folder.

Once found, double-select select the *Custom Pads* folder (or select the + sign alongside it) to open it. If custom pads exist in the folder they will be listed below the open folder in the navigator pane.

The custom pads folder of a new design will be empty.

If custom pads are listed they can be selected in turn to browse through them. They are shown graphically in the *Browser* window and their properties are displayed in the *Properties* window (if those windows are open).

A custom pad can be opened for editing by double-selecting it.

To close the folder, either double-select the folder (or select the - sign alongside it).

Custom Pad Information

The pad **MUST** be defined as an enclosed area, with **ONE** continuous line.

Corners in the line should only be added in order to change direction.

The start point and end point of the line **MUST** be in the same position to ensure an enclosed polygon is formed.

The line **MUST** start and stop at a corner in the pad, not in mid-segment.

Areas within the pad should not be defined.

Failure to follow the above will result in problems when creating the gerber photo-plotter files.

When a custom pad is created, it does not "belong" to a particular layer of the board. It is added to the appropriate layer(s) when the component outline (footprint) is made, or the pad is added to a particular layer in the artwork editor.

A hole or holes can be defined in the pad, but it is the layers the pad is used on that determine whether it will be drilled.

The position of the pad's datum, the centre of the drilled hole(s) and the connection point can be selected.

It is suggested that custom pads are created within the design custom pad folder and verified before they are transferred to the master folder. Once pads are in the master folder they are available to all users and expected to be correct.

If custom pads in a design are modified, they are automatically updated on the associated artwork.

Creating a custom pad, general guidelines

With the design open, create a pad and give it a name by right-clicking the Custom Pad folder, then selecting *New*. Double-select (or right-click/Open) the pad to open it for editing.

Check the grid is set to suit your requirements (*Grid* commands).

Draw the shape of the pad using the *Outline* commands - take note of the information given above.

Position the datum of the pad (*Tools > Set Pad Datum*), typically in the centre of the pad.

Position the connection point of the pad (*Tools > Set Connection Point*), typically in the centre of the pad.
Add the drill hole(s) if one or more are being added (*DrillHoles > Add*), their size is defined in the toolbar.
The pad is ready to be used in an outline or on the artwork.

Individual commands in the custom pad editor

An overview of the custom pad editor can be found in the *Installation & Getting Started Guide*. Here we describe each of the custom pad editor commands in detail.

Right-click on custom pad folder

Right-clicking on the Custom Pad folder in the navigator window introduces the following commands:

New **Paste**

These commands are described below.

New - Creating a new custom pad

A new custom pad can be created by right-clicking the *custom pad folder* in the navigator window and selecting *New* from the options that appear. A new custom pad is added at the bottom of the list of custom pads in the folder, called *Unnamed*. It can be renamed at that point as it is highlighted ready for editing. However, if <enter> is pressed or the cursor is selected elsewhere, the original name (unnamed) will remain. It can be renamed by right-clicking on the outline and selecting *Rename*.

Paste

A custom pad that has been copied to the paste buffer can be pasted (copied) to the custom pad folder. Right-click the *custom pad folder* in the navigator window, then select *Paste* from the options that appear. The custom pad is added to the bottom of the list of custom pads in the folder, it will take its original name unless a custom pad with that name already exists in the folder, in which case it will be named "*Copy of <original name>*"

It can be renamed by right-clicking on the outline and selecting *Rename*.

(To copy a custom pad to the paste buffer, right-click on the custom pad, then select *Copy*.)

Right-click on a custom pad

Right-clicking on a custom pad in the navigator window introduces the following commands:

Open **Cut** **Copy** **Delete** **Rename**

These commands are described below.

Open

When selected, the custom pad is opened for editing.

Cut

Currently (version 1.68) operates in the same way as *Copy*, though this may change in a future release – refer to the readme.txt in the ..\Seetrax\XL Designer folder for update information.

Copy

When selected, the custom pad is copied to the paste buffer, overwriting anything previously added to the paste buffer. (To paste the custom pad from the paste buffer to any custom pad folder, right-click on the appropriate custom pad folder (which might be the same design, another design or the master custom pad folder), then select *Paste* from the options that appear.)

Delete

When selected, the custom pad is deleted from the folder.

Rename

Used to alter the name of a custom pad. Once selected, the name is highlighted and the usual editing keys (backspace, delete, left/right arrows, etc.) can be used to rename it. It is possible to rename a custom pad that is used in an outline or the artwork editor.

Datums commands

Tools > Set Pad Datum

Used to define the datum of a custom pad.

When a pad is selected in the outline or artwork editors, it must be selected on its datum position.

The appearance of the datum is shown at the bottom left-hand corner of the screen to avoid confusion with the connection point, which is also displayed.

Once selected, point at the position required for the datum then click the left-hand mouse button. The datum moves.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The X-Y readout (shown in the information bar) of the cursor position is displayed with respect to the position of the datum point.

The *Set Pad Datum* command remains active until the right button is clicked to cancel the operation, or another command is selected.

Datums > Set Connection Point

Used to define the point at which connections terminate on the pad.

The appearance of the connection point is shown at the bottom left-hand corner of the screen to avoid confusion with the datum, which is also displayed.

Once selected, point at the position required for the connection point and click the left-hand mouse button. The connection point moves.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The *Set Connection Point* command remains active until the right button is clicked to cancel the operation, or another command is selected.

Tools > Outline Definition

Once Tools > Slots & Extra Holes has been selected, use this command to revert back to the main Pad editor commands.

Tools > Slots & Extra Holes

This command provides access to the *Slots and Extra Holes* commands which allow routed slots and/or additional holes to be added to an individual custom pad.

Their use is identical to the same commands in the Component Outline and Artwork editors. The commands are therefore described under the heading *Slots & Extra Holes (Tools > Slots & Extra Holes commands)* on page 241 of this manual.

Outline commands

These commands are used to define the external shape of the pad. Only one line should be added and it should form an enclosed polygon shape. No other lines should be added.

Tip: start off with a simple enclosed shape, then add corners and convert the segments to arcs as required. The starting point of the line should coincide with the end point of the line, there should not be a gap between the two. This point should also be on a change of direction in the line, not mid-segment.

Outline > Add Line

Used to add a line that will define the external shape of the pad.

Point at the location where the line will start and click the left button. Move the cursor, a line appears attached to it. Click the left button to insert corners in the line. Release the line from the cursor by clicking the right-hand mouse button. The end of the line must be terminated where the line was started from, so that it forms an enclosed shape.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The line can be modified using the *Outline > Corner/Adjust Arc/Delete point*, etc. commands.

Outline > Add Arc

Used to define the external shape of the pad, typically when it is made from a series of curves. Only one line should be added and it should form an enclosed polygon shape. The end of the line must be terminated where the line was started from, so that it forms an enclosed shape.

Point at the location where the curved line is to start and click the left button. Move the cursor and its attached line to the position where the curve should end, and click the left button again. A straight segment is produced that bends as the cursor is moved. The shape of the arc is dependent on the position of the cursor. Click the left button to release the curve in its current position. The line remains attached to the cursor, allowing a series of curves to be added.

Once the arc has been released, clicking the right-hand mouse button releases the line from the end of the cursor.

If the right-hand mouse button is clicked whilst the curve is being stretched, a straight-line segment is

introduced, thus allowing a mixture of curved and straight lines to be added.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Arcs can be modified with the *Outline > Corner/Adjust Arc* commands.

Outline > Add Rectangle

Used to add a rectangle. Once added it can be modified using the *Outline > Corner/Adjust Arc* commands.

The size of the rectangle can be specified in either inches or millimetres. The units in use are controlled by the *Edit > Units* command.

When selected, a window appears allowing the *Width* and *Height* of the rectangle to be typed in. Select *OK* to add the rectangle.

The rectangle appears attached to the cursor, move it into position and click the left button to release it. Another rectangle appears on the end of the cursor until either another command is selected, or the right-hand mouse button clicked when the window reappears, and *Cancel* should be selected.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Once in position, the rectangle is treated as a line, which can have corners added, deleted or moved or converted to arcs.

Outline > Corner

Used to add and move existing corners (points) in lines or arcs.

Corners are **added** and released with clicks of the **right-hand** mouse button. Existing corners are **moved** with clicks of the **left-hand** mouse button. Once the corner has been selected, clicking the opposite mouse button cancels the operation.

right button = **add** corner
left button = **move** corner
(Opposite button cancels)

Adding a corner to a line or arc:

Point at the line or arc and click the right button. Move the new corner and release it with another click of the same (right) button. Clicking the opposite (left) button before the corner is released cancels the new corner.

Moving an existing corner in a line or arc:

Point at the corner in the line or arc and click the left button. Move the corner and release it with another click of the same (left) button. Clicking the opposite (right) button before the corner is released, cancels the move.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Outline > Adjust Arc

Used to convert straight lines into curved lines, curved lines into straight lines, and to change the shape of existing curved lines.

Straight lines into arcs:

Point at a line segment and click the left-hand mouse button. Move the cursor, the segment is replaced with an arc attached to the cursor. Once the arc is in the required position, click the left-hand mouse button to release it.

Arcs into straight line segments:

Point at the arc and select it with a click of the left-hand mouse button. Follow this with a click on the right-hand mouse button to convert it to a straight-line segment.

Changing the shape of an arc:

Point at the arc and select it with a click the left-hand mouse button. Move the cursor with the arc attached, then click the left-hand mouse button when the arc is in the desired position.

Clicking the right-hand mouse button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically and can be used to convert 45-degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Outline > Delete Feature

Used to delete a line or an arc.

A complete line is deleted between its start and end points, not just a segment between points.

Point at the line or arc to be deleted and click the left-hand mouse button.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Outline > Delete Point

Used to remove corners from lines or arcs.

Move the cursor over a point (corner) in a line or arc and click the left-hand mouse button. The corner is removed.

If the start or end point of a line is deleted, the segment is removed.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

DrillHoles commands

These commands are used to define the position and size of the drilled holes. The layer(s) the pad is used on in the artwork editor control whether the holes are used.

Drill diameter (in tool bar) - Used to specify the size of drilled holes before they are added.

If the diameter of an existing hole is incorrect, the hole should be deleted and a new one added at the correct size.

Hole sizes are usually specified as finished hole sizes, i.e. after plating. Drilled hole sizes are not usually given, as the thickness of copper in the hole, and consequently the finished hole size, is determined at the manufacturing stage.

Hole sizes can be specified in either inches or millimetres if the appropriate units are selected (*Edit > Units*).

Select the *drill diameter* dialogue in the toolbar, then type the hole size required. (Once the value has been selected for editing, keep the cursor in the dialogue or the value cannot be changed.)

New holes will be added at the size specified.

DrillHoles > Add

Used to add a drilled hole to a pad. More than one drill hole can be added to a pad.

Whether the pad is actually drilled is determined by which layer(s) the pad is used on in the outline or artwork editors.

The diameter of the hole is specified before it is added, by the *Drill diameter* size setting in the tool bar.

Select *DrillHoles > Add*, move the cursor and a "hole" appears attached to it. Position the hole as required and click the left-hand mouse button to release it. More holes appear on the end of the cursor until another command is selected.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

DrillHoles > Move

Used to move an existing drilled hole.

Select *DrillHoles > Move*, point at the centre of a hole and click the left-hand mouse button. Move the cursor taking the hole with it, to its new position, then release it with a click of the left-hand mouse button.

Clicking the right-hand mouse button whilst the hole is being moved returns it to its original position.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

DrillHoles > Delete

Used to delete a drilled hole from a pad.

Select *DrillHoles > Delete*, point at the centre of a hole and click the left-hand mouse button. The hole is removed immediately.

Refer to the grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Vias

Unless other via types are specifically defined, code 0 round or square pads are always used for vias.

Whether round or square code 0 pads are inserted in the artwork is controlled from the *Manual Routing Parameters* window, in the *Configuration* folder.

Other via types, such as blind/buried vias, or different sizes of via for use in power tracks have to be defined by the user, using the *Via Hole Definitions* window in the *Configuration* folder.

Schematic Folder

The schematic folder contains the tools to design circuit diagrams on a single sheet, or multiple sheets using a flattened or hierarchical structure. The tools are also used to create/edit the parts used on the circuit diagram and to extract a parts and wiring list from the circuit diagram.

Typically designs start off with one sheet, then more sheets are added as required – these are referred to as *flat* or *flattened* schematics.

Hierarchical structures are employed when the design is more complex and symbols can be used to represent sheets of circuitry in order to make the diagram easier to follow, or perhaps when there are multiple channels of similar circuitry.

Flattened schematics are easier to design and there are important differences that should be taken into account when choosing which method to use – it is easier to convert a flat schematic into a hierarchical schematic than the other way round.

Differences between Flattened & Hierarchical Schematics

Once a design has more than one sheet, it can be drawn as a “flat” or “hierarchical” schematic. The main differences between a hierarchical & flattened schematic are:

- appearance
- connectivity - the way in which connections are linked between sheets

Note: in older versions of Ranger a different schematic editor was supplied, this was known as the “old, flat-style schematic editor” which should not be confused with flat schematics designed in XL Designer. If a design with a schematic drawn using the “old flat-style schematics” is opened in Seetrix XL Designer the schematic is automatically converted to the new-style editor, in flattened mode – see *Old flat style schematic to hierarchical schematic conversions* below.

Appearance

In a flat schematic the circuitry is simply added to as many sheets as required.

In a hierarchy, each schematic sheet can be represented by a symbol. This symbol can be used on other schematic sheets to represent the circuitry from its associated sheet.

Connectivity

Connections may be added from pin to pin on the sheet or they may be added to a pin and given a signal name. All connections with the same signal name on the same sheet (local connections) are automatically connected together in the resultant wiring list.

When multiple sheets are used there is usually a requirement that some connections are connected between sheets.

To cater for this, the wiring list can be extracted in one of two modes - either “flattened” or “hierarchical”.

Flattened Mode

In this mode, any connection with the same signal name, from any sheet is connected to ALL other like-named connections on any other sheet.

Hierarchical Mode

In this mode, connections with the same name, on different sheets are NOT connected together in the resultant wiring list. A conscious decision has to be made to make the connection between sheets. This is achieved through the use of ports.

The only exception to this, are any implied power rails within schematic library parts – these automatically become “global” connections, i.e. they connect across all sheets without the use of ports. Note: this does not apply to Primitives that are used for signal names, they are not global connections.

Ports are added to the sheet as required to act as a connection point between sheets and the levels in the hierarchy. The ports have to be connected through the hierarchy in order to make the connection between sheets.

Old flat style schematic to hierarchical schematic conversions

The first time a design is loaded that has an old style flat schematic (for example from a design created with Ranger1/2/3), then the schematic and its associated device library entries are converted into the hierarchical schematic editor format.

Due to major differences in the way that “bus” signals were handled in the old flat schematic editor compared to the new editor, all imported “bus width” wires are converted to non-electrical features.

Because of this, schematics with busses may need some manual editing after loading, if a parts/wiring list extraction results in errors because multiple signal names are found on a net. It will be necessary to edit the busses using the *Wires > Flat Circuit Import Assist > Toggle Wire Section To Non Electrical* command

which is described later in this manual.

Accessing the schematic tools

The master schematic library will be found under the *Masters* folder in the navigator. Its commands work in exactly the same way as those found in the design's schematic folder.

When a master schematic library folder is selected, its path/location is shown in the *Properties* pane, along with the number of parts held in the folder.

When schematic parts are selected in the navigator, the *Properties* pane displays its attributes and power pin information.

To open the design's schematic library, assuming the design has been opened and expanded in the navigator pane, select the + sign from alongside the *Schematic* folder to open it and view the items making up the schematic part of the design, which includes parts and design sheets (circuit diagrams).

Expand the appropriate folders depending on the task in hand - in a new design all the schematic folders will be empty to start with, they become populated with the parts used on the design – either by being placed on a sheet, copied from the master library or other jobs, or created within the design itself.

To create a new item, right-click the appropriate folder and select *New* - an opportunity is provided to supply a name. Double-click on the item to open its editor. Right-clicking the item in the navigator pane produces other commands which are detailed below.

(A worked example of the schematic editor can be found in the Seetrix XL Designer Manual, available from the Help command when the program is running.)

Right-click on the Schematic Folder, or Items within Sub-Folders

New Folder – applies to the Master Schematic folder only

The master schematic library is divided into folders so that parts can be located easily, for example resistors in one folder, capacitors in another and so on. Each of those folders can contain sub-folders, or nested folders to allow further separation of the parts, for example, smd's in one, through-plated parts in another.

A selection of folders containing parts/sheets was provided when the software was installed and these folders can be added to, by either right-clicking on the master schematic folder to add a new top level folder, or one of its sub-folders to create a nested library within that folder.

New - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Creates a new item of the type selected and places it in the appropriate schematic folder for the item selected. Each of the types is described below under the heading: *Contents of the schematic folder*

Open - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Opens the selected item for editing, see following pages for details on the commands within the individual editors.

Copy - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Copies the selected item(s) into the "paste" buffer for subsequent "pasting" (copying) to an appropriate folder. For example, a schematic part into a schematic part folder, primitive into a primitive folder, etc., – items cannot be pasted into the wrong folder type.

Items can be copied into folders within the same job, other jobs or the master library.

If a schematic part is copy/pasted between jobs or the master library, then the component outline(s) required for the copied schematic part(s) will be imported into the target design's component outline library at the same time, from the same source (job or master) library, provided it exists.

Multiple items can be selected, using the Shift/Ctrl keys in unison with the right-click selection.

Drag-drop functionality – for moving/copying

In addition to the *Copy* command, "drag & drop" is available to move schematic library parts between the various folders in the master library and between designs.

Parts that have dependencies on other library entries (like split parts and their dependent primitives) cannot be moved or copied by drag/drop.

When dragging parts between the master schematic library folders, the default action is "Move". Hold down the *Ctrl* key to change the action to "Copy".

When dragging parts from a design into another design or master library, the default action is "Copy". Hold down the *Shift* key to change the action to "Move".

Drag/drop "Move" functionality is not available when removal of the part from the drag source would break design dependencies.

Paste - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Pastes the content of the Paste buffer into the selected folder, provided the items are compatible, for example. a primitive could not be pasted into a block i/o port folder, design sheet folder, etc. See details above, regarding the Copy tool.

Delete - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Deletes the selected item(s) from the folder. Items that are in use within the design cannot be deleted and an appropriate information message is displayed. Multiple items can be selected, using the Shift/Ctrl keys in unison with the right-click selection.

Rename - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

Used to Rename the selected item. All occurrences of the item that are in use in the design are updated.

Attributes – schematic folder only

Each schematic part has a set of attributes associated with it. This command allows the attribute fields for all parts within the currently open job, or a selected master library folder, to be exported to a text file, edited off-line, then imported back to the design. This provides a simple way of editing the attribute fields within each part without having to open each part for editing, for example to update all the current price attributes.

Attributes > Export

Used to export attributes from the selected schematic or master schematic library file. Exported attributes may be saved into tab separated (.txt) or comma separated (.csv) files. To export a file suitable for editing in Microsoft Excel Spreadsheets, it is recommended that the attributes are output as a tab separated (.txt) file. On loading, Excel will display an import Wizard where the type of all imported columns should be changed to "text" type. If this is not done, the strings of digits will be treated as numbers, which is a problem if some of the attributes have all numeric names with leading zeros, for example "0805". The first row in the created file will always contain the column names for the data in the subsequent rows. The first column is always the name of a part in library. Each additional column represents an attribute setting. The columns between rows may not necessarily line up, as can be seen in the following samples, but there will always be a tab or comma between entries – including blank entries.

Sample attribute output file (tab delimited):

Name	\$3 (Outline)	\$2 (Value)
7400	DIL14	7400
7401	DIL14	7401
7402	DIL14	7402

Sample attribute output file (comma delimited):

Name,\$4 (Order code),\$3 (Outline),\$2 (Value)
7400,DIL14,7400
7401,DIL14,7401
7402,DIL14,7402

When editing the file, be sure to leave just one tab or comma between fields. When selected (right-click appropriate folder, then *Attribs > Export*), the window similar to the one shown in Figure 84 appears, with any user-defined attributes that are in use listed (none in this example).

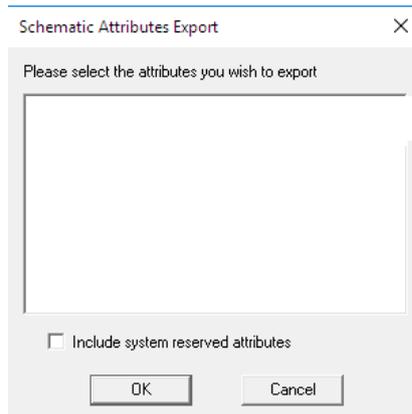


Figure 84

To exclude any of those attributes in the output file, untick the checkboxes alongside.

If some/all of the system attributes are also required, then tick the *Include system reserved attributes* checkbox. This will produce the list as shown in Figure 85.

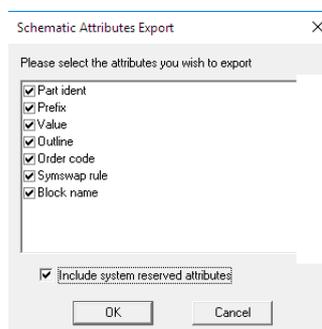


Figure 85

Any of the attributes that are not required in the output file should be unticked. Select OK to produce the output file, the Windows browser appears, allowing you to specify where the output file should be created and its name.

Select the *Type* of file required – either *Text (Tab delimited)(*.txt)* or *CSV (Comma delimited) (*.csv)*.

Select *Save* to save the file. A window indicating the export has taken place will appear.

Once the file has been created, the values assigned to the attributes can be edited. The file can then be imported back using the *Attributes > Import* command.

Attributes > Import

This command is used to import a file containing the attribute value fields associated with a library file or job back into the library or job.

When importing attributes, the schematic editors should not be open.

The importer will be using commas or tabs to identify attributes – lines will be ignored where the incorrect number of tabs or commas exists.

The first row in the created file will always contain the column names for the data in the subsequent rows.

The first column is always the name of a part in the library which should exist in the library.

Each additional column represents an attribute setting. The columns between rows may not necessarily line up, but there will always be a tab or comma between entries. If some columns have a blank entry, ensure the tab or comma for the entry is present.

Right-click the schematic folder, then select *Attributes > Import*. The Windows browser appears allowing the file to be imported to be located. Select the file and also the file type (csv or tab) required and the attributes will be imported.

A report window will appear listing any problems. These should be investigated and appropriate action taken.

Purge Unused Blocks – schematic folder only

Used to delete unused items from the design's schematic library.

For example, parts may have been placed on the schematic and then every occurrence of them deleted from it. The parts are therefore no longer required in the library, but they will still be there.

Note: there is no "undo" once the parts have been deleted.

Once selected, a window appears listing all the unused parts and split parts. Select the parts that should be deleted by ticking the box alongside them, then select *Delete*.

The *Select All* and *Deselect All* buttons in the window can be used to quickly select/deselect all the parts.

Once the required parts have been selected, select *Delete* to purge the library of the selected parts.

Note: once split parts have been deleted, there may be primitives in the primitive folder that are no longer used. Use the command a second time to identify and remove these items in the same way.

Edit Outline Names – schematic folder only

When selected, an editor opens that allows the modification of the *outline* attribute field within all the library parts or split library parts in the library.

This command is used to quickly update the outline attribute fields in parts.

When selected, a window appears, listing all the library parts or library split parts present in the design, along with their default outline name and number of pins.

All the parts must have a valid outline name in order to extract a usable parts and wiring list suitable for board layout.

Changes that are made to attribute fields from the schematic, using the *Symbol > Attributes > Edit* command take precedence over changes made in the library parts. So for instance, any parts that have had their outline name attributes changed from the schematic sheet, prior to using this command will not be updated. As an example: one part appears on the schematic 10 times, called R1 to R10. Its outline name is set to ????. R4's outline attribute has been manually changed using the *Symbol > Attributes > Edit* command to RESA10. When the OUTLINE window is opened, the outline name will be set to ??? as the actual part has not been modified, just one instance on the schematic. If an outline name is added to the OUTLINE window for the part and Proceed selected, the outline attributes within the part and consequently for R1 to R3 and R5 to R10 would be changed, but R4 would remain as it was.

Edit Datasheet Addresses – Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

When selected, an editor opens that allows the modification of the *Datasheet address* attribute field within all the library parts or split library parts in the current design/library.

The *Datasheet address* attribute field is typically used to define a web address (URL) or file path for parts. This allows access to data-sheets or information associated with the parts.

Double clicking on a part type entry brings up another dialog where the web address or file path may be edited.

The address window in the *Datasheet Address* edit dialog will accept a "drag and dropped" file, so you can for example drag a .pdf format datasheet from Windows Explorer into the address window, and the file path will be set.

Once the attribute field has been defined, the datasheet/file can be viewed by right-clicking the part in the folder and selecting *Show Datasheet* (greyed out if the attribute has not been defined), or once on the sheet using the *Symbol > Show Datasheet* command.

Show Datasheet - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

When selected, opens the file/web-page associated with the datasheet address of the selected item, provided a path has been defined in the Datasheet attribute of the item.

Show Dependancies - Design Sheet/ Part/ Split Part/ Primitive Symbol/ Block I/O Symbol

When selected, lists the items that are used by the selected item.

Change Block Type – Split parts or Primitives

This command is not normally visible, as it is only intended for the recovery of parts whose block type has become corrupted. The corruption occurred when a schematic part or userblock was fetched into a design from a master library or another design; during this process the type of the block in the source library was inadvertently set as a split part. If the source library or design were subsequently edited and saved, then the block became stored as a split part regardless of its correct type. The part entered into the destination design received the correct block type.

To enable the command, open the *File > System Setup* window and tick the "Enable Schematic Block Type Change" setting under the *Temporary Fixes* section. The program always starts with this feature disabled, so the checkbox state is not remembered.

Once the feature has been enabled, right-click on the split part or primitive to be recovered. Choose the appropriate submenu type to alter the selected split part/primitive to be a userblock, part, split part, primitive or blockio symbol.

After changing a block's type in this way, the design should be saved, closed and reopened in order for the parts to appear in their correct navigator folders.

Contents of the schematic folder

Within the schematic folder will be found:

Design sheets

Design sheets are used to define a page of a schematic diagram.

Each schematic page can be represented by a *Design Symbol* that is used in a hierarchical schematic to represent the contents of the sheet, on another sheet. Connectivity between sheets in a hierarchy is achieved through the use of *ports*. Each port on a sheet has a corresponding pin on the sheet's associated symbol. Each port and its associated pin have the same name.

Note: It is not possible to open the *Design sheet* and its associated *Design symbol* at the same time.

Design Symbols

Design Symbols are used to define the symbol associated with a design sheet, for use in a hierarchical schematic.

This symbol can be used on other schematic pages to represent the circuitry from its associated sheet - this creates a hierarchical schematic.

Pins can be added to the symbol, in which case an associated *port* with the same name is added automatically to the associated sheet.

Note: It is not possible to open the *Design sheet* and its associated *Design symbol* at the same time.

Note: If the symbol of a sheet is placed on a higher level sheet, and the pin name attribute is edited, the new name will appear against the block i/o pin when descending into the schematic sheet associated with the symbol.

An iopin attribute that is overridden by an attribute on a higher level instantiation will always show the overridden value when the sheet is displayed. The value may still be edited on the sheet, but the override value will still be displayed.

Switching to the symbol of the sheet will show the attribute value without the effect of any higher level override. The value displayed in this mode is the value displayed against the pin for any new instantiations of the symbol.

Over-riden attributes can be reset to their original values, using the *Symbol > Attributes > Reset* command. When re-setting the attributes of design symbols in a hierarchy, the attributes should be reset at the upper level.

Parts

Parts are used to define single element parts, or multi-element parts where all the elements are the same, for use on the schematic sheet.

Examples of single element parts are diodes, resistors, capacitors and connectors that are shown as one component. Examples of multi-element parts are 7400's, 7402's, resistor packs and connectors where the pins are added individually.

Split Parts

Split parts are used to define a multi-element part where the elements are **not** all the same. The elements that make up the part have to be made from *primitives*.

Examples are a relay containing a coil and contacts, a part containing two nand gates and two inverters.

Primitives

Primitives are used to define the elements for a *split part*, a *power symbol*, or a *symbol to show connectivity*. Also used to define a *simulation probe* for use with the PSpice output.

Examples are the coil and contact from a relay, logic gates from *mixed* logic parts, power symbols for logic packages, symbols used to show connectivity and probes for simulation.

Block I/O Ports

Block I/O Ports are used to define a port to ensure connectivity between sheets in a hierarchical schematic – they are not required in a flat schematic.

Block I/O ports are not physical parts, they simply allow connections to be made from one hierarchical sheet to another.

There are two types of port, each with a single pin. One type allows single connections to be attached, the other allows bus connections to be attached. The port type is determined by the pin that is added, it's either a *net-pin* or a *bus-pin* – so the port is either a net-port or a bus-port.

When block I/O ports are placed on the schematic design sheet, a pin is automatically added to the associated design symbol, with the same name. The pin that is added is either a net-pin or a bus-pin, depending on the

type of port that was added to the sheet. Like-wise, if a pin is added to a design symbol, its type (net or bus) determines the type of port that is added to the associated sheet.

Commands available when a design sheet or part is open:

Edit > Preferences

Used to control colours, line widths of items, sheet and symbol attributes, versions, function keys and default information within the editor.

When selected, the window contains varying information depending on the type of block being edited. Tabs allow access to particular information, such as Colours, Part Attributes, etc.

The decimal point in numeric values indicates whether the size is displayed in inches or millimetres. A dot (.) is used in inches, and a comma (,) in millimetres.

All the information the *Edit > Preferences* window can contain is described here.

Edit > Preferences - Miscellaneous tab

Junction blob diameter - controls the size of junction blobs which are used when connections are connected to one another. Existing blobs are updated automatically, but the screen must be refreshed to see the change.

Auto-symgen pin gridpitch - controls the pitch of pins that are added automatically to the symbol of a sheet when the *Tools > Auto Symbol Generate* command is used (or the design symbol is opened for the first time after ports have been added to a sheet). Existing symbols are unaffected.

Extract pin-pin capture distance - determines whether a connection has to be physically added between pins in order for them to be connected in the wiring list. For example, some designers like to be able to place pins on top of one another to make a connection, whilst others prefer to add connections between pins.

The distance specified indicates how close pins have to be to one another before they are connected. For instance with 0.005" set, any pins that are touching one another or within 0.005" of one another will be connected in the wiring list.

To disable pin-pin connectivity, enter any negative value. For instance -0.001. This means that connections must be added between pins.

Drawing grid display mode - controls how the grid is displayed when it is switched on.

Ruled: the grid is shown as a solid lined grid.

If the ruled schematic grid is too faint, or not visible, the width of the grid lines can be controlled from within the *Edit > Display Adjustments* window.

Dotted: only the intersection points of the grid are shown, by dots. The colour of the grid is controlled by the *Edit > Properties, Colours* tab window.

Negation bar prefix - to obtain a negation bar over signal names, precede the signal name with the character specified here. The bar appears on the display and all plots.

The negation bar is not used in the wiring list or in the artwork editors, when the negation bar prefix character appears in front of the signal name.

Note: negation bars cannot be displayed or plotted if the *Rotated text display mode* is set to *Vertical*. In this mode, the signal names will be displayed and plotted with the prefix character, whether or not they have been rotated.

To change the character, simply select the box and type a new one. Do not change this character once used as a negation prefix within the schematic, or existing negation bars will be lost.

Version control character - used to define the character used within version control strings. Refer to the *Edit > Preferences, Version* tab for more details.

Maximum window mode dynamic nodes - controls the maximum amount of data that can be dynamically moved using the *Region* commands. Moving large amounts of data (nodes) dynamically can be slow and cumbersome on some machines, so experiment and choose a number that suits you and your machine. (The status bar indicates the number of nodes within a window when *Region > Copy* is used.)

Outline default prefix (i.e. DIL) - used to define a default outline name prefix for use with the *Block > Outline* command.

Upto 12 characters can be used in the outline prefix, or it can be left blank. This allows upto 4 extra characters to be added to the outline name by the *Block > Outline* command.

Rotated text display mode - defines which way rotated attribute text is presented (not non-electrical text). The setting affects all rotated attribute strings in the schematic. Options are:

Right edge rotated text strings appear vertically, the characters rotated anti-clockwise, so the first character appears at the bottom of the column. (Tilt head to the left to read.)

Left edge rotated text strings appear vertically, the characters rotated clockwise, so the first character appears at the top of the column. (Tilt head to the right to read.)

Vertical rotated text strings appear vertically, the characters are not rotated, the first character appears at the top of the vertical column. Signal name negation bars cannot be displayed whilst in vertical mode, the *negation bar prefix character* will be displayed and plotted irrespective of whether the signal names have been rotated.

Trap distance in 'absolute' mode - providing the "*Trap distance mode*" parameter is set to "*Absolute*", this parameter specifies the maximum absolute distance the cursor can be from an item, in order to select it. Default value is 0.1000".

Trap factor 'n' in width/n mode - providing the "*Trap distance mode*" parameter is set to "*Screenwidth/n*", this parameter specifies the maximum distance the cursor can be from an item as a proportion of screen width, in order to select it. Default is 50. i.e. the cursor has to be within 1/50th of the screen width to select an item. The smaller the number, the further away the cursor can be.

Trap distance mode - when items are selected, the cursor has to be within a certain distance of the item before it will select it. Options are *Screenwidth/n* and *Absolute*.

With *Absolute* selected, the cursor has to be within a specific distance of the item in order to select it. The distance is defined by the parameter, *Trap distance in absolute mode*.

With *Screenwidth/n* selected, the cursor has to be within an area of the item based on a proportion of the width of the screen. The actual area is defined by the parameter, *Trap factor 'n' in width/n mode*.

Pickup wires during symbol move - determines whether pins connect to existing connections when they are released over the connection, when parts are moved, rotated, flipped, etc. For instance, if a part pin is moved on top of a connection, this parameter controls whether the pin is connected to the connection or not. When enabled, the connection connects to the pin otherwise the connection remains unconnected.

Allow connection start in mid-air - determines whether connections are allowed to start in mid-air. When enabled the connection can start anywhere. When disabled, the connection must start on a pin or existing connection.

Specifying that connections must start on a pin or existing connection means there is never any doubt that the connection is attached to the pin or connection.

Show symbol datum points - determines whether the datum points of symbols are visible. It is useful to be able to see the datum points of symbols so that you know where to select them when moving, rotating, etc. them.

Edit > Preferences - Line Widths Tab

Any of the values can be changed. The screen will need to be refreshed to see the changes.

Normal connections - controls the width of connections as seen/printed on the schematic sheet (not the thickness of the corresponding tracks on the board which is controlled by the *Wires > Attributes* commands). (Refer to the *Edit > Preferences, Miscellaneous* tab to control junction blob sizes.)

Bus connections - controls the width of buses as seen/printed on the schematic sheet (not the thickness of the corresponding tracks on the board which is controlled by the *Wires > Attributes* commands).

Symbol outlines - controls the width of lines used to draw symbols. All symbols are updated.

Attribute text - controls the width of line used to form characters in attribute strings.

Non-electrical, drawing sheet lines - controls the width of lines used to draw the auto-generated drawing border.

Non-electrical, drawing sheet text - controls the width of line used to form characters in the auto-generated drawing border.

Non-electrical, annotation lines - controls the width of lines used to draw non-electrical items added in annotate mode.

Non-electrical, annotation text - controls the width of line used to form characters in non-electrical text strings added in annotate mode.

Edit > Preferences - Colours Tab

This window controls the colours used to display the various items in the schematic. To change the colour associated with an item, select the coloured box alongside the item with a click of the left-hand mouse button. The *Colour editor* window appears. Select the new colour required, followed by *OK*. When the *Properties* window is closed the screen will be re-drawn to reflect the changes.

Set classic scheme - when selected, the colours assigned to features will be the same as those assigned in the (old-style) flat style schematic editor – to make Ranger1, 2 or 3 users feel "at home".

Set modern scheme - when selected, the colours assigned to features will be those assigned by default for the hierarchical schematic editor.

Restore defaults - should be selected to restore the default colours as defined in the colours file.

Edit > Preferences - Version Control

Used to define "versions" within the schematic. Up to 10 versions can be defined. Each version is given a name. Up to three upper or lower case characters can be used in the version name (case sensitive). For example: UK, USA, NZ, etc.

One of the versions is the default or current selection as defined by the button alongside being enabled.

"Versions" are used if:

- specific sheets should be excluded when the schematic is compiled to a parts/wiring list.
For example, the same schematic could be designed for UK or USA use. Because of the different mains voltages in use, you may require two mains input blocks, one for each version of mains input. The USA mains input sheet will need to be excluded when the UK version is compiled, and the UK sheet excluded from the USA version.
- different value fields, outline fields, etc. are required on specific parts, connections or non-electrical text when the schematic is compiled to a parts/wiring list or plotted.
In this example, the mains input sheet is essentially the same for the UK and USA version, but some or all parts may need to have different values assigned. In this case parts can be assigned a UK and USA value. When the UK version is selected, the schematic is displayed, compiled and plotted with the UK version attributes. When the USA version is selected, the schematic is displayed, compiled and plotted with the USA version attributes.
- the same sheet will be repeated on an upper level in the hierarchy, but different values (or outlines, etc.) are required on parts in each instance.
For example, the circuit may consist of three identical channels of circuitry, for three different frequency ranges Low, Mid and Hi. The only differences between each channel would be the values assigned to parts. The channel could be drawn once and placed three times on an upper level sheet. Three versions could be defined, called Low, Mid and Hi. Each instance of the channel would be assigned to a version (using the *Structure > Set Version Override* command), either Low, Mid or Hi. When viewing or plotting individual instances of the sheet, the values assigned will correspond to the values assigned for that particular version.
For instance, if a resistor was assigned a value of 1K for the Low version, 10k for the Mid version and 100k for the Hi version, when viewing or plotting the channel assigned as the Low version the resistor value will be set to 1k. When viewing or plotting the channel assigned as the Mid version the resistor value will be set to 10k, and when viewing or plotting the channel assigned as the Hi version the resistor value will be set to 100k.

The *Edit > Preferences, Compile Version Control* window indicates whether each block is included or excluded from the compile process. The *Symbol and Wires > Attributes > Edit* commands or *Non-Elect > Add Text* commands are used to define different values for different versions of part, connection, non-electrical text, etc.

Edit > Preferences - Sheet size

This window only appears if a sheet is being edited.

Sheet size - used to select a sheet size for the circuit schematic. Standard sheet sizes are listed, or a sheet size can be specified if *User-defined* is selected.

A drawing border is automatically added to the sheet. It is made from non-electrical lines and text in *Drawing sheet mode*, which can be modified using the Non-Elec commands. If the border is removed, it is automatically re-generated the next time the circuit is modified. (The output routines use this border for scaling and selection purposes.)

Sheets have been defined and added to the master library which contain additional border information. They can be copied to the design and the circuit diagram drawn on them, or the additional data pasted into the current block using the *Region* commands.

Once the schematic has been started, the sheet size can be increased or decreased at any time. If the sheet is decreased, data may appear outside of the sheet. This data cannot be selected until the sheet size is increased again.

Edit > Preferences - Compile Version Control

Upto 10 versions of a schematic can be defined. This table indicates whether the current sheet should be included or excluded when the schematic is compiled to a parts/wiring list, depending on the version selected. The version names are defined via the *Edit > Preferences, Version Control* window. The names are "undefined" until a name is entered.

Example: the same basic circuitry could be used in the UK and the USA, but the mains input circuit would have to be different to cater for the different power supplies in use. In this situation, a user block could be created for each of the input circuits, a UK input and a USA input.

In the UK input SETUP, Sheet window, the block would be included in the UK version, but excluded from the USA version.

Likewise in the USA input Setup, sheet window, the USA input would be included in the USA version,

but excluded from the UK version.

When the schematic is compiled to a parts\wiring list, depending on which version is selected, only one of the input sheets will be used. (Refer to *Edit > Preferences, Version Control* to define which version is used when compiling.)

View commands, schematic editor only

View > Fast Part Tray

Used to switch the *fast part tray* on/off. When switched on, the *fast part tray* appears to the left of the schematic editor window.

It can contain a selection of parts chosen by the user for fast access during part placement. Parts do not have to be added to the fast part tray in order to be placed on the sheet, they can be placed directly from the library.

When the *fast part tray* is switched on, four buttons at the top of the tray control what it contains as follows:

- Add** Used to add parts to the fast part tray. Note: split parts cannot be added to the fast part tray.
To add a part, drag it from the master or job library from the navigator pane, into the fast part tray.
When a part is added to the tray from the master library it is copied into the design, even though it may not be added to the schematic.
- Delete** Used to remove parts from the fast part tray. Select the part from the fast part tray, then select the *Delete* button. The part is removed from the tray.
The part is not removed from the design library, it can be re-added to the tray using the *Fast Part Tray, Add* command.
- Empty** Used to remove all the parts from the fast part tray. The parts are not removed from the design library.
- Sort** Used to sort the parts in the fast part tray into numeric-alpha order.

Using Parts from the Fast Part Tray

Parts from the tray are selected when using the *Symbol > Place* command. The part appears attached to the cursor. Position and release it with a click of the left button. More parts appear until another part is selected from the tray, the right-hand mouse button is clicked or another command selected.

Structure commands

Used to control and view information relating to the overall schematic design. The commands available are context sensitive, so not all commands described will be available at the same time.

The structure of the schematic can be displayed graphically by selecting *Hierarchy* from the schematic folder in the navigator window. A window opens showing a block diagram, where a block represents each schematic sheet. Any of the blocks can be selected to open them (or bring forward if already open). Note - if the design symbol is already open, then that will be brought forward as the symbol and its associated sheet cannot be open at the same time.

In a "flat" schematic, the blocks are shown side by side, all on one level.

Where a hierarchy has been created, the blocks are shown in their positions in the hierarchy. An example of a hierarchical design is shown in Figure 86.

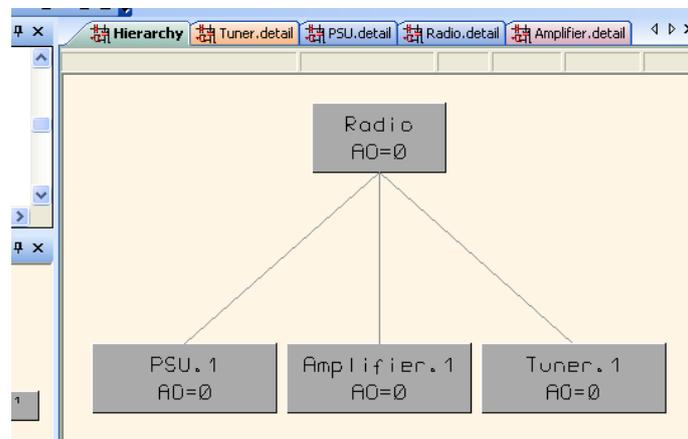


Figure 86

Each box contains the name and identifier of the block it represents. For example if a block called *amplifier* occurred three times in the hierarchy, the first occurrence would be called *amplifier.1*, the second *amplifier.2* and

the third *amplifier.3*.

The boxes also contain the *allocation offset* for each block and the *version override* if assigned. By default the allocation offsets are set to zero (AO=0), refer to the *Structure > Set Allocation offset* command for details. A version override is only displayed if one has been assigned, refer to the *Structure > Set Version override* command for details.

Structure > Traverse Up (not operational on current version)

Used to travel *up* through the hierarchy one level at a time. (Greyed out unless a schematic sheet is being edited.)

When selected the next level up in the hierarchy is opened/displayed and can be edited. (If the top level in the hierarchy is being displayed, nothing happens when selected.)

When travelling *up* through the hierarchy there is only one possible path that will eventually return to the top level of the hierarchy.

Use the *Show Hierarchy* icon to travel quickly up or down through many levels of the hierarchy. (Or select *Hierarchy* from the schematic folder in the navigator.)

Structure > Traverse Down

Used to travel *down* through the hierarchy one level at a time. (Greyed out unless a schematic sheet is being edited.) Because there are (or could be) many routes down through the hierarchy, the sheet to be viewed has to be selected.

Once selected, select the symbol of the sheet you wish to view with a click of the left-hand mouse button.

Use the *Show Hierarchy* icon to travel quickly up or down through many levels of the hierarchy.

Structure > Locate in hierarchy tree (not operational on current version, select Hierarchy from the schematic folder in the navigator)

Used to select and view/edit any sheet from the overall picture of the hierarchy. This method of travelling through more than one level in the hierarchy is quicker than using the *Structure > Traverse Up/Down* commands.

Once selected, select the block that represents the sheet to be viewed/edited. Note - if the design symbol of the selected sheet is already open, then that will be brought forward as the symbol and its associated sheet cannot be open at the same time.

Structure > Set Version Override

This command is only available when the hierarchy block diagram is open. It is used to control the *versions* of repeated instances of sheets, in order to allow variations for value fields, outline fields, etc., within each instance.

For example, one *design symbol* might be placed many times on an upper level sheet. Although the actual circuitry is the same in each associated *design sheet*, the values assigned to certain parts may need to vary within each instance. The *Set Version Override* feature allows one sheet/symbol to be created, but multiple values to be assigned to parts within it.

First a version name has to be assigned for each variant required (refer to *Edit > Preferences, Version Control*).

Within the sheet, each part that requires a different value for each instance, should be assigned all the values, as described under the *Symbol > Attribute > Edit* command. Once this has been performed, proceed as follows:

With the Hierarchy window open, select *Structure > Set Version Override*. A window appears listing all the versions that have been defined. Select the one required, followed by *OK*. A flag appears on the end of the cursor containing the selected version name. Point at and select the sheet that should be assigned this version name. The words VO=<versionname> will appear inside the sheet instance.

Repeat this for each version required. When each of the sheet instances are viewed or plotted, they will display the values assigned according to their version name.

Structure > Set Allocation Offset

This command is only available when the hierarchy block diagram is open.

Used to control the starting number for part references within a block, provided allocation has not already been performed. By default, parts are numbered or "allocated" sequentially from 1 upwards.

There is sometimes a requirement to number parts within blocks from a specific number. For example, in a multi-channel design where each of the channels has been drawn in a separate sheet, part numbers in the first channel could start from 101, part numbers in the second channel from 201, part numbers in the third channel from 301, etc. This numbering system would help in part location, etc. Use the *Alloc* commands to actually allocate the part numbers.

To set the allocation offsets, with the Hierarchy window open, select *Structure > Allocation Offset*. Fill in the values, using the following information to help you.

<i>Starting ident</i>	the first part number required in the first block selected.
<i>Increment per block</i>	the number to be added to the starting ident number for subsequent block

selections.

To use the assigned values, select *Auto-incrementing Assignemnt* - when selected a flag appears attached to the cursor containing the *starting ident* number. Select the block to be assigned that starting ident number. The allocation offset for that block is assigned and *AO=0* is replaced by the appropriate number. The number in the flag on the end of the cursor is increased by the number specified in the *increment per block* parameter. Continue selecting blocks to assign the allocation offsets. Click the right-hand mouse button to cancel further allocation offsets.

To use your own values, select *Typed*, a message along the bottom of the screen requests you to select the block that you want to set the allocation offsets for. Select one of the boxes. A window requests the *Block allocation offset number*. Type the starting number required followed by OK. The allocation offset for that block is assigned and *AO=0* is replaced by the appropriate number. Continue selecting blocks and entering the numbers required.

Identify commands

Used to display an information balloon for selected items.

Buttons within the balloon (where appropriate) allow the selected item to be located in the parts/wiring list or artwork editors.

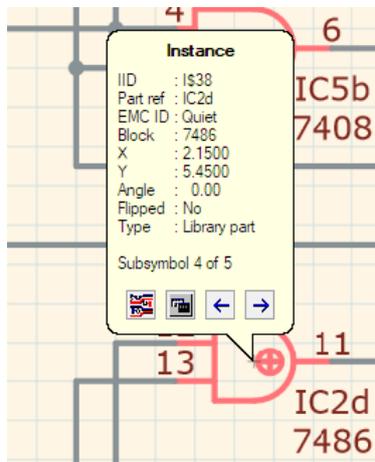


Figure 87

Identify > X/Y

Used to indicate the X and Y co-ordinates of a selected position. Once selected, move the cursor to the required position, then click the left-hand mouse button to display the X and Y co-ordinates of that position. (X = horizontal, Y = vertical.) Further positions can be selected, or the right-hand mouse button clicked to cancel the command.

Identify > Instance

Used to display information about selected parts or "instances". Once selected, select a part. A balloon appears pointing to the selected part, similar to that in Figure 87.

If the instance has been allocated and also appears in the parts list and artwork editors, then a button in the balloon allows the part to be located in those editors.

If the instance has been allocated and is an element from a multi-elementy part, arrow icon(s) appear in the balloon to allow other elements from that package to be located, including power symbols if used.

The balloon is replaced by another, when another part is selected, or closed when the right mouse button is clicked to cancel the command, or another command is selected.

IID The internal identification number (or instance number) of the part. This number is assigned and used by Ranger in reports, in order to identify parts that may not have been allocated.

Part ref If allocated, the part reference.

EMC ID The EMC part group to which the part belongs if defined.

Block The name of the block.

X and Y The X and Y co-ordinates of the part's datum point on the sheet.

Angle The part's rotation on the schematic sheet, with respect to its original library definition.

Flipped Whether the part has been flipped on the schematic sheet, with respect to its original library definition.

Type The type of schematic item (library part, primitive, etc.).

Further parts can be selected, or the right-hand mouse button clicked to cancel the command.

Identify > Part Pin

Clicking on a symbol pin will display the identity of the pin if the part is allocated and the net it is connected to. If the part pin can be located in the net list/artwork, then the popup balloon will contain buttons as shown in Figure 88, that allow the part pin to be located in the artwork and netlist editor views.

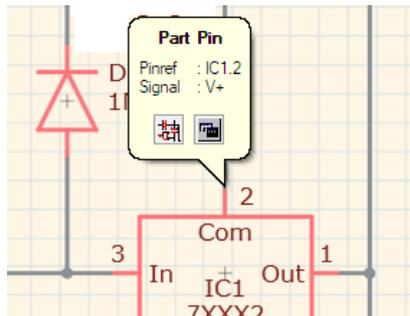


Figure 88

Identify > Connection

Clicking on a connection segment that connects directly to an allocated symbol that can be located in the netlist (parts/netlist has to have been extracted), then a balloon window will display the net name and the net's properties will be displayed in the properties pane as shown in Figure 89.

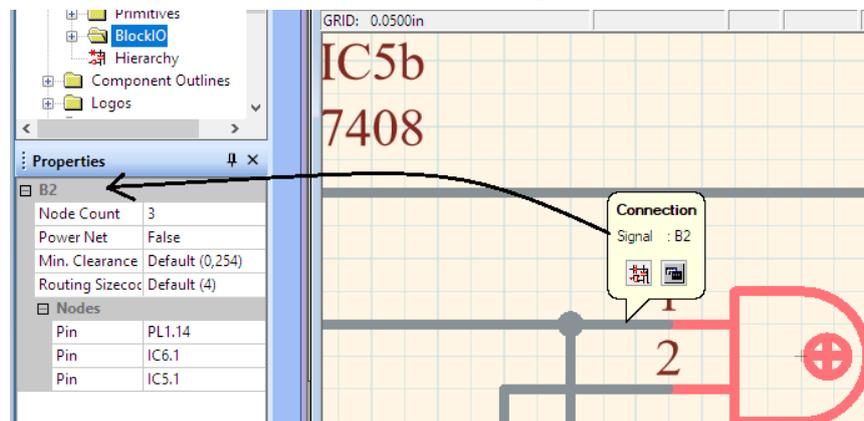


Figure 89

If the net can be located in the net list/artwork, then the popup balloon will contain buttons which will locate the associated net in the artwork and netlist editor views.

Find commands

Used to find specific items and X/Y positions.

Find > X/Y

Used to find a specific X and Y co-ordinate. When selected a window appears requesting the co-ordinates. Type them in, followed by OK. A balloon indicates the specified X and Y position.

As soon as the left-hand mouse button is clicked, or the screen is re-drawn the balloon disappears.

Find > Instance

Used to find a specific part or "instance" by entering its internal identification number (IID). Instance numbers are typically used in parts/wiring list compilation report windows to identify un-allocated parts.

When selected a window appears requesting the IID. Type it in followed by OK. The specified instance appears on the screen with a balloon to identify it.

The balloon provides information about the instance – refer to the *Identify > Instance* command for details on the balloon. It remains open until closed by selecting the close X.

Find > Part

Used to find a specific part by its reference designator. Pin or gate references can be included in the name, in order to find a specific gate.

For instance, if IC7 were a 7400, entering **IC7** would find the first gate within IC7. Entering **IC7c** would find the third gate within IC7, whilst entering **IC7.5** would find the gate containing pin 5 and its associated pins.

A balloon identifies the specified part - refer to the *Identify > Instance* command for details on the balloon. It

remains open until closed by selecting the close X.

Find > Signal Name

Used to find a specific wire attribute with the name specified. The search is not case-sensitive, so entering CLOCK will find CLOCK, clock, Clock, etc.

Note: this command finds wire attributes with the specified name on the schematic. This is different to the *Find in schematic* command that can be used by right-clicking on a net in the Nets folder from the navigator, which finds the nodes (IC1.5, etc.) in a particular net on the schematic.

A balloon identifies the specified part - refer to the *Identify > Instance* command for details on the balloon. It remains open until closed by selecting the close X.

Errorflag commands

Used to find, describe and/or delete, error or warning flags posted by the electrical rules checking routine. (Electrical rule checking is accessed from the *Tools > Electrical Rules Check* command when a design sheet is open.)

Errorflag > Next

Used to locate the next occurrence of an error or warning flag. The flags are not necessarily found in the order listed in the electrical rule check report.

When selected, the next error or warning flag is displayed on the screen and momentarily flashes. The flag remains highlighted until a mouse key is clicked or the screen refreshed. Each flag is highlighted in turn when *Errorflag > Next* is selected, until each one has been shown. If *Errorflag > Next* is selected again, the flags are highlighted in order again until they are removed.

Use the *Errorflag > Describe* command to display the error/warning text.

Errorflag > Delete

Used to delete individual error or warning flags posted by the electrical rules checking routine. Deleting the flag does not clear the error it is indicating.

When selected, point at the end of the stem of the flag and click the left-hand mouse button. The flag is deleted. *Delete* stays active allowing further flags to be deleted until the right-hand mouse button is clicked or another command is selected. Flags can only be restored by running the electrical rules checking routine again.

Errorflag > Describe

Used to display the error/warning the flag (posted by the electrical rules checking routine) is indicating.

Once selected, point at the end of the stem of the flag and click the left-hand mouse button. The description of the error or warning appears in a window above the schematic. *Describe* stays active allowing further flags to be selected and described until the right-hand mouse button is clicked or another command is selected.

The window containing the description is closed automatically when the right-hand mouse button is clicked or another command is selected.

Errorflag > Delete All

Used to delete all the error and warning flags posted by the electrical rules checking routine. (Existing flags are automatically removed when the electrical rules checking routine is performed.)

When selected, all the flags are deleted.

Tools commands, schematic editor

Tools > Electrical Rules Check

Used to identify electrical rule violations on the active (open for editing) schematic sheet. If the check is performed on the top level sheet in a hierarchy then the check is performed on all sheets below it as well.

When parts are made, each pin can be assigned one of thirteen *logic types*. The electrical rules checker is used to flag instances where pins with a certain logic type are connected to pins with the same, or a different logic type.

An individual pin's logic type can be defined from within the schematic using the *Symbol > Attributes > Edit* command, but it is more efficient to define the logic types within the part itself.

As one example of a check that can be performed, output pins connected to output pins can be flagged if required. The user chooses which checks are run as follows:

Select *Tools > Electrical Rules Check*. A window appears, similar to the one in Figure 90, showing a table with the thirteen logic types listed down one side and along the top to form rows and columns. The box or "cell" that intersects the row and column indicates whether or not a check is performed according to its colour as follows:

Yellow	-	Check, and flag a warning message
Red	-	Check, and flag an error message
Blue	-	Do not check

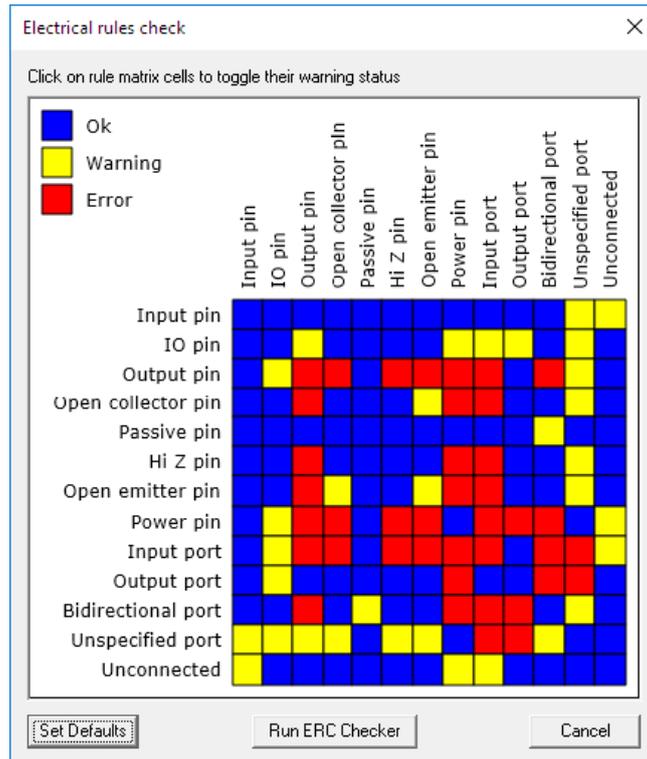


Figure 90

For instance, the cell that intersects the **output pin** row and the **output pin** column is set to **red** by default. This means that when the check is run, if an output pin connected to another output pin is detected, an error message is given and an error flag placed on the offending connection.

If the cell were changed to yellow, by selecting the cell, then a warning message and warning flag would be posted. If the cell were changed to blue, then the checker would not look for output pins connected together.

By pointing at the cells and clicking the left-hand mouse button, the colours change. Set the colours of the cells according to the checks required. The original (or default) checking selections can be restored by selecting the *Set Defaults* button.

Note: when checking for unconnected pins, Ranger is looking for a connection on the same sheet. This means that named connections that are only connected to a pin on another sheet will be flagged as unconnected, i.e. they are unconnected on the current sheet.

When you are ready to run the checks, select the *Run ERC Checker* button. After a while, depending on the size and complexity of the circuitry on the sheet, a report appears on the screen listing details of the problems found. This list can be printed if required by selecting *File > Print*, or saved to a file by selecting *File > Save As* from the top of the window. Select *File > Close* to close the window.

Use the *ErrorFlag > Next* command to locate the error or warning flags and the *ErrorFlag > Describe* command to display the actual error or warning message.

Each time the check is performed, the previous error and warning flags are removed automatically. (They can be deleted manually if required using the *ErrorFlag > Delete/Delete All* commands.)

To check for unused gates, use the *Alloc > Show Freelist* command.

Unused pins can be listed from the wiring list editor.

Tools > Parts/Netlist Extraction > Setup

Used to define the rules that the *Ranger Parts + Nets extraction* routine (*Tools > Parts/Netlist Extraction > Ranger Parts + Nets*) uses.

It is important that these options are set to suit your requirements or the resultant parts/wiring list may not be what is expected. When selected a window similar to that in Figure 91 appears.

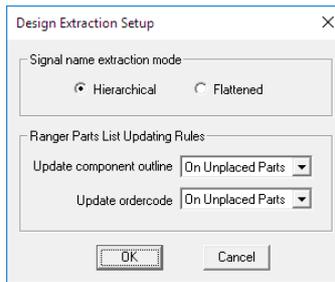


Figure 91

Signal name extraction mode:

Flattened when the parts/wiring list is extracted, signal names assigned to connections will be treated as global connections, so all connections with the same signal name will be linked together, including across sheets.

Hierarchical when the parts/wiring list is extracted, signal names assigned to connections will be treated as local to the sheet they are used within, unless block I/O ports have been attached to them and the corresponding symbol pins connected at the upper level in the hierarchy.

Ranger parts list updating rules:

When a parts list is produced from the schematic for the first time (or if the parts list has been deleted), then the parts list contains the outline and ordercode field attributes as assigned within each part on the schematic sheet.

However, **on subsequent extractions** it is not always desirable that this happens. In order that the user controls what happens, the following choices are available.

Update component outline

Never with *Never* selected the component outline field is not updated when the parts list is extracted. This means the outline called up in the parts list may be different to that called up by the part on the schematic. If there is a difference a warning is posted when the parts list is extracted.

On unplaced parts with *On unplaced parts* selected the component outline field is not updated when the parts list is extracted if the part is already placed on the artwork (the part's X/Y co-ordinates in the parts list are not set to 0,0). However, if a part has not been placed on the artwork (X/Y co-ordinates are set to 0,0) then its outline field is updated.

Always the component outline field is always updated when the parts list is extracted. This means the outline called up in the parts list at parts list extraction time will be the same as that called up by the part on the schematic. (It is possible to change the outline in the parts list or artwork after the extraction, but it will always revert back to the outline defined by the schematic part when the parts list is extracted.)

Update ordercode

Never with *Never* selected the ordercode field is not updated when the parts list is extracted. This means the ordercode called up in the parts list may be different to that called up by the part on the schematic.

On unplaced parts with *On unplaced parts* selected the ordercode field is not updated when the parts list is extracted if the part is already placed on the artwork (the part's X/Y co-ordinates in the parts list are not set to 0,0). However, if a part has not been placed on the artwork (X/Y co-ordinates are set to 0,0) then its ordercode field is updated.

Always the ordercode field is always updated when the parts list is extracted. This means the ordercode called up in the parts list at parts list extraction time will be the same as that called up by the part on the schematic.

Tools > Parts/Netlist Extraction > Ranger Parts + Nets

Used to extract a parts and wiring (net) list from the schematic for use by the Ranger layout routines. There is a setup window (*Tools > Parts/Netlist Extraction > Setup*, described above) that controls how the extraction routine works and this **MUST** be set to suit your requirements prior to using the extractor to ensure the extracted parts/wiring list meets your requirements.

Note: Once the parts/wiring list has been extracted it is always worthwhile opening the parts and wiring lists from the Navigator, as this will highlight errors that must be corrected before starting the artwork layout.

If versions have been defined for this design ensure the required version is selected before extracting the parts/wiring list (*Edit > Preferences, Version Control* window).

Also ensure all the sheets are included or excluded as required (*Edit > Preferences, Compile Version Control* window).

Only the parts and connections from **allocated** parts are extracted. A warning is posted and added to the extraction report if unallocated parts are found. (The extraction report will be found in the design's *Logfiles* folder, it's called *compile*.)

The *Alloc > Unallocated Check* command can be used to reveal unallocated parts prior to the extraction.

The parts and connections lists are updated automatically, so there is never any need to delete the parts and wiring list. Note: once a layout has been started, the parts list contains the positional information of the parts on the layout, so NEVER delete the parts list unless it is intended to start part placement from scratch again.

The parts and wiring lists produced from the schematic should always be checked to ensure they contain what you think they contain.

When the parts/netlist is extracted, the required component outlines (that are not already in the job) are imported to the job from the master component outline library.

To extract the parts/wiring list, select *Tools > Parts/Netlist Extraction > Ranger Parts+Nets*.

If any of the following conditions occur, a report is posted but the parts/wiring list is still produced. It is up to the operator whether any remedial action is taken and a parts/wiring list extracted again.

- * Unallocated parts - the unallocated parts are not included in the parts/wiring list - investigate and correct if appropriate.
- * An outline that is called up for a part that is different to an outline already placed on the layout for that part. Typically caused when the layout has been started and an outline has been swapped on the layout. The *Tools > Parts/Netlist Extraction > Setup* window controls whether outlines are updated - investigate and correct if appropriate.
- * Pin numbers that appear more than once and are connected to different connections – investigate and correct.
- * A connection that has different signal names assigned to it – investigate and correct.
- * Signal names, outline fields or value fields that are too long for the Ranger layout facilities. The names are truncated - investigate and correct if appropriate.
- * Single node nets (connections that start, but do not become attached to another pin) – investigate and correct.
- * Allocated parts that have a different pin number allocation to the parts defined in the library - de-allocate, then re-allocate those parts.
- * Allocated parts that have a different number of pins or gates to the parts defined in the library use the *Symbol > Replace* command to update the part.
- * Excluded blocks - nothing from excluded blocks will appear in the parts\wiring list – check this is what is required.
- * Version - if versions have been defined, the version used is reported – check this is what is required.
- * Signal names that have been hidden (can cause connectivity issues that you may be unaware of), always best to keep them visible.

If a warning window is posted, the warnings should be investigated and corrected where necessary, then the parts/wiring list extracted again.

The parts and netlist (wiring list) can be viewed/edited from the navigator pane, select the Parts & Nets folders from the design.

Tools > Parts/Netlist Extraction > User Defined Extraction

Used to extract attribute information from the schematic in a user-defined format.

A program must be written in the Seetrax generic design extraction language (Genex) that will extract the information from the schematic, for instance a list of the part names and order codes. This program must have a filename with a **.GEX** extension and exist in the sub-directory: `..\seetrax\XL Designer\genex`

This program produces an ASCII output file.

The programme can also be configured to call up a secondary program that can be written in any language familiar to the user, for instance 'C', BASIC, PASCAL, Assembler, etc. The secondary program will use the ASCII data produced and manipulate it into any format required and produce a second output file. The secondary program must also exist in the `..\seetrax\XL Designer\genex` sub-directory.

Some examples are provided, these are written in 'C'. These examples can be used as a guide when creating your own programs.

Probes can be added to the schematic in order for simulation type outputs to include probe information. An example of probe construction is given in the worked example of the user guide. Only the `PSPICE.GEX` program that is supplied (not `EXAMPLE3.GEX`) uses the probe information.

To use these examples, select *Tools > Parts/Netlist Extraction > User Defined Extraction*. All the programs with a **.GEX** extension found in the `genex` folder are listed in this window. Select the program or "filter" required followed by *Select*. A window appears indicating which filter was selected and where the output files will be located. The *View config* button can be selected to view the program.

A name for the output file must be specified in this window and then *Run* selected. A window appears showing the data extracted. A file will also have been generated and placed in the folder indicated in the window.

The Seetrax generic design extraction language

The generic design extractor is an interpretative program module embedded within the hierarchical circuit schematic editor. The extractor allows the user to gain access to most design features and output them in any desired format.

Selection and formatting of data for output is controlled by user written configuration files (.GEX file). As well as selecting and formatting output data, the extractor configuration files can also specify that a secondary output filtering program be executed. These secondary filter programs can be used to perform additional data processing, formatting, statistics generation etc., which cannot be performed by the built in low level interpreter.

Configuration files - file names and location

The generic design extractor configuration files must all end with a file name extension of '.gex'. By default, the extractor expects to find configuration files in the MyDocuments/Seetrax/XL Designer/genex folder. By editing the **.genex_path** path directive in the circuit schematic configuration file (*schema_attribs.txt*), it is possible to relocate the generic extractor files to anywhere within the file system.

Configuration files - file sections

The configuration files consist of two main sections:

Header	This section contains directives for file path selection, secondary filter control, menu selection titles, etc.
Code	This section contains interpreted program code used for primary level access to the design data.

Configuration files - file header directives

The following directives may be included within the header section of a configuration file.

.title	The text string that follows this directive is displayed within the filter selection window when the user selects the User Defined extraction routine.
.scope	The .scope directive is used to determine the application package that the .gex file belongs to. Future releases of Seetrax XL Designer will contain generic extractor capability within other areas of the design system. The .scope directive enables an application to choose only the .gex files appropriate to itself. Currently, the only argument that may be given to the .scope directive is capture . If a configuration file does not have a .scope directive, the file is always made selectable.
.filter	By giving the name of an executable file as an argument to this directive, the extracted data can be additionally filtered and formatted into the required final format. External filter programs can be written in any language familiar to the user. If the filter cannot be found using the given name then the extension .exe will be added. External filters are invoked with the following sequence of parameters:- raw_inputdata_path, output_paths, user_defined_args.
.targetdir	This directive controls where the extracted design data should be stored. The file path argument may contain environment variable references (\$xx) and by using @name notation, it is also possible to refer to entries in the file called filepath.
.outprompt	The user is normally prompted for the name of the file to be created by a standard prompt saying 'Output filename:'. The .outprompt directive permits the user to assign different prompt messages. Also, by including more than one .outprompt directive, the user will be prompted for multiple output file names. This may be needed by external filter programs requiring more than one target filename. No more than two output file names are permitted in the current release of the software.
.userarg1(2)	These directives allow the user to pass up to two additional arguments to external filtering programs.
.format	This directive marks the start of the interpreted code section. The interpreted code language is described further on.
.exdepth	The .exdepth directive controls the type of net extraction performed on the design. Permitted values are 'parts' and 'primitives'. The default value is parts . In parts mode, extracted nets will not contain nodes that are associated with library primitive symbols. primitives mode outputs all nodes regardless of type.
.endformat	This directive marks the end of the interpreted code section.

Interpreted code - concepts

Before the language of the interpreted code section can be described, the reader must understand some of the terminology that will be used.

context	Within the generic design extractor, the term 'context' means a collection of data items of identical type. For example, in the schematic editor the 'instances' context means a list of all instances (symbols) within the design tree being extracted.
context_converter	A context converter takes data of one type and creates another context from it. Special keywords in the interpreted language cause data in the current context to be converted to a new context for further data extraction.
iteration	Because a context may contain many data items, when a context conversion is requested, the code interpreter will iterate on each data member of the source context list converting the data item to the target context and executing the associated code block for each converted item.
datavar	A datavar keyword is a reference to a data member within a context. Datavar keywords are encountered in data output statements and conditional skip statements.

Interpreted code - language syntax

The interpreted code has the following structure:

```

{ outer_level_context_key
  { code_block
  }
}

```

A code block is formatted as follows:

```

data_output_statements
context_conversions
skip conditions
.
.
data_output_statements
context_conversions
skip conditions

```

A data output statement is formatted as follows:

```
output "format_string" datavar datavar .. datavar
```

A context conversion is defined as follows:

```

context_conversion_key
{ code_block
}

```

A skip condition is defined as follows:

```

skipif datavar == "stringvalue"
or
skipif datavar != "stringvalue"

```

A true skip condition causes execution to skip to the end of the currently executing code block. The == operator will cause a true skip condition on equivalent strings. The != operator will cause a true skip condition on differing strings.

If you need to skip execution to a point before the end of the currently executing code block, place a { character before the **skipif** directive, and a } character just before the statement you wish to skip to.

The C escape sequences are:

\a	alert (bell)	\v	vertical tab
\b	backspace	\'	single quotation mark
\f	formfeed	\"	double quotation mark
\n	newline	\\	backslash
\r	carriage return	\ddd	ASCII character in octal notation
\t	horizontal tab	\xdd	ASCII character in hex notation
%%	a single % sign		

A format specification has the following form:

%[flags][width]type

Each optional field of the format specification is a single character or a number signifying a format option. These fields control width and other aspects of formatting.

Flags The flags determine justification of output and control the printing of signs, blanks, decimal points, and octal and hexadecimal prefixes, as shown in the following list.

Flag	Meaning
-	Left-justify numeric output. Default is right justification.
+	Precede output value with sign (+ or -) if output is of signed type. By default, a sign appears only for negative signed values (-).
0	Write zeroes until a minimum width is reached. If 0 and - appear, the 0 is ignored. If 0 is specified with an integer format (i, u, x, X, o, d), the 0 is ignored. The default is no padding.
blank (' ')	Precede the output value with a blank if the output is signed and positive. The blank is ignored if both the blank and + flags appear. The default is no blanks.

Width The width field contains a non-negative decimal integer that specifies the minimum number of characters printed.

If the number of characters in the output is less than the specified width, blanks are added to the left or the right of the output value (depending on whether the - flag is used) until the minimum width is reached. If width is prefixed with a 0, zeroes are added until the minimum width is reached. (This is not useful for left-justified numbers.)

The width specification never causes a value to be truncated. If the number of characters in the output value is greater than the specified width, or width is not given, all characters of the value are printed.

Type The type character, which appears after the last optional format field, determines how the associated argument is converted:

d,i	=	integer : signed decimal
u	=	integer : unsigned decimal
o	=	integer : unsigned octal
x	=	integer : unsigned hexadecimal using lowercase for 'a' - 'f'
X	=	integer : unsigned hexadecimal using uppercase for 'A' - 'F'
s	=	string

Interpreted code - reserved words

The interpreter reserves only two keywords for special use. These are the data output keyword **output**, and the conditional skip function **skipif**. All context conversion and data reference key words are defined by the application from which data is being extracted.

Interpreted code - data output directive

The data format string for the **output** directive is formatted identically to the format strings used by the 'C' language **printf** function. Each **datavar** to be printed must have a corresponding format conversion character in the format string. The format conversion character must correctly match the type of data being referenced. Unexpected results will occur if there is a mismatch. The interpreter does not check for syntactically correct output format strings. The user must refer to the data variables section of the context descriptions to ascertain the type of a data variable.

Interpreted code - skip directive

The conditional skip directive always compares data values as character strings. If a data variable is of integer type or double precision type, it is converted to a string before comparison with the right-hand string argument. Because of round off errors etc. it may be difficult to achieve correct comparison of double precision data variables.

Circuit schematic data extraction - contexts

The following contexts are defined for the circuit schematic editor.

design	The 'design' context represents the entire schematic design hierarchy. The 'design' context keyword is the only keyword permitted in the outer_level_context_key position of the interpreted code body. It defines the starting point for all subsequent context conversions and data extractions.
instance	The 'instance' context consists of all symbols instantiated within the design hierarchy. This includes library primitives, placed power blocks and library component symbols. In the current

	version of the schematic capture program, user block symbols do not appear in this context.
parts	The 'part' context is a list of all library part/split part items formed into a physical parts list.
net	The 'net' context is all the net data for the design hierarchy.
netnode	The 'netnode' context is all the nodes that make up one connection within a 'nets' context.
attrib	The 'attrib' context is a list of attributes attached to another design context. Attribute lists may be extracted from most other contexts.

Circuit schematic data extraction - context conversions

The following context conversions are defined as valid by the schematic capture system.

From-context	To-context	context_conversion_keyword
design	instance	instances
design	part	parts
design	net	nets
instance	attrib	attribs
part	attrib	attribs
net	attrib	attribs
net	netnode	nodes
netnode	attrib	attribs
netnode	part	part

Circuit schematic data extraction - data variables within each schematic context

Instance context

Name	Type	Function
iid	int	Internal identifier for the instance.
ident	int	Allocated part number.
symtype	int	Symbol type code.
powerblock_flag	int	Set to '1' if the symbol is a placed power block.
npins	int	Number of pins on symbol.
blockname	string	The block name for the symbol.

Part context

Name	Type	Function
iid	int	(as for instance)
ident	int	(as for instance)
symtype	int	(as for instance)
powerblock_flag	int	(as for instance)
blockname	string	(as for instance)
prefix	string	Part prefix code (e.g. IC, R etc.)

Net context

Name	Type	Function
netref	int	Internal net reference code.
nnodes	int	Number of nodes on the net.
nattribs	int	Number of attributes on the net.

Netnode context

Name	Type	Function
iid	int	Instance identifier for netnode.
pinref	int	Pin reference on the instance.

If the 'iid' value for a netnode is -1, then the net node relates to a part power pin that does not appear on the schematic diagram. In this case, the 'pinref' value has no useful function. The netnode can however be context converted to 'attribs' type for extraction of the outline pin reference attribute, and also converted to 'part' type for determining the associated part

prefix, ident, etc.

Attrib context

Name	Type	Function
atrid	int	Attribute name code.
atname	string	Attribute name text.
atrvalue	string	Attribute value text.

All attributes within the schematic capture package have an integer code associated with the attribute name text. The schematic capture system reserves attribute codes 0-31 for attributes having special predefined meanings. The attribute codes currently assigned within the reserved area are:

- 0 Part identifier. Only used to determine where the attribute text will be displayed on the schematic sheet. The actual value is calculated from the contents of the part prefix and allocated part ident attributes.
- 1 Part code prefix. Holds the prefix text that forms the start of a part ident.
- 2 Component value. Holds the assigned value for resistors etc. When a design is extracted for Ranger, this field is appended to the end of the component name text and the resultant string becomes the part description field in the extracted parts list.
- 3 Component outline.
- 4 Bill of materials / part order code.
- 5 Symbol swapping rule. Syntax not yet defined.
- 6 Block name. Place holder for the displayed identity text for user defined blocks only.
- 7 Symbol pin name.
- 8 Pin number on a component outline.
- 9 Pin swapping rule. Syntax not yet defined.
- 10 Pin logic type used by the electrical rules checker.
- 11 Data bus signal subscript number (bus member number) for bus ripper symbols.
- 12 Signal name.
- 13 Ranger signal routing width code.

When using the skipif condition to extract specific attributes from an attribute block, the **atrid** reference should be used to access attributes with reserved functions. For user defined attributes, the **atname** string format should be used.

Circuit schematic data extraction - examples

The circuit schematic package is supplied with several example generic extractor configuration files, together with the source and object code for their associated external filter programs.

Example 1, a component package counter.

This example simply extracts a list of component outline attributes for all parts in a design. The list is then passed to a small external filter program where the list is totalled according to package type.

The related files are example1.gex and ex1filt.c in the genex directory.

Example 2, a part ordering list.

This example will extract a list of components together with their associated order codes. The list is then processed by an external filter to yield a part ordercode / quantity type list.

The related files are example2.gex and ex2filt.c in the genex directory.

Example 3, a Spice nodal format extractor.

This example shows how a list of symbol instances and a net list can be extracted from a design. The data is subsequently passed to an external filter where it is reorganised into a symbol nodal type format similar to that used by a Spice simulator.

The related files are example3.gex and ex3filt.c in the genex directory.

Example 4, a parts and wiring list extractor.

This example extracts the design part and net data, and then formats it into a layout similar to that presented within the Ranger parts and wiring list editor.

The related files are example4.gex and ex4filt.c in the genex directory.

Tools > EMC Attribute Tags

Used to add EMC type attributes to parts, part pins and connections on the schematic.

This allows part pins and nets to be tagged with groups of attributes that allow more precise control over the Specetra and Electra auto-routers with regard to routing order, lengths, widths, clearances etc.

Also, parts may be tagged with an attribute to define whether they are susceptible to interference, produce interference or are quiet. They can then be shown in different colours in the artwork editor to assist manual part placement.

The ascii file *emcmemb.txt* held in the *..XL Designer\data* folder lists all the attributes currently understood by the Specetra/Electra router interface. The keyword names (in the left-hand column), and the declared variable type words (dimension, int, double, stringlist etc.) should not be edited by the user, but the attribute descriptive text and default values may be freely edited. The information in this file appears when defining emcgroups.

The file *emcgroup.txt* held in the *..XL Designer\data* folder may be edited to contain default groups of attributes that are available to new jobs. The supplied file currently only contains definitions for part and partpin attribute groups. Each design receives its own copy of the *emcgroup.txt* file as soon as the hierarchical circuit editor is activated. This can be viewed from the navigator pane, right-click the design name, select *Design File Content Browser*. A list of files will appear, locate: */config/emcgroup.txt*

Only the *emcgroup.txt* file currently being accessed is updated, so a change made within a job is not reflected in any other jobs or the master library. Likewise, changes made to the master library are not made to existing jobs.

Tools > EMC Attribute Tags > Select Attribute

Used to select and edit the EMC rules that will be assigned to either *Parts*, *Part pins* or *Nets*. When selected, the *EMC Attribute selection & editing* window appears.

The window contains varying information, depending on whether *Parts mode*, *Part pins mode* or *Nets mode* was active. The content of each window is explained later on in this description.

Figure 92 shows the Parts window.

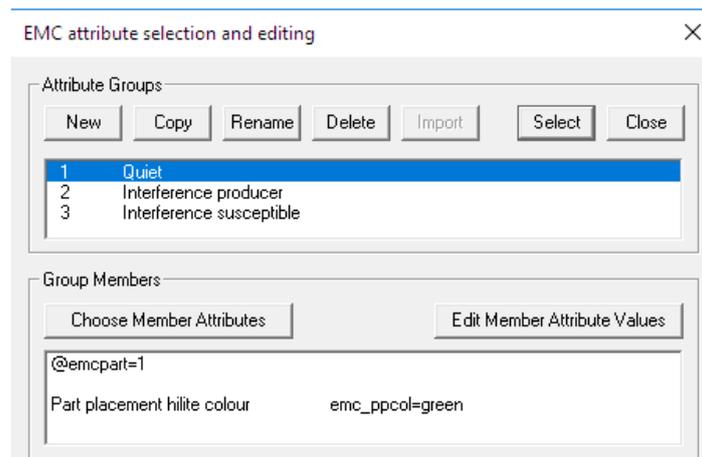


Figure 92

The window is always divided into two, the top part lists the *Attribute Groups* that have been defined. The lower window lists the *Group Members* assigned to the selected group.

To select a group, point at the emc group required in the upper part of the window, then select it with a click of the left mouse button.

Buttons in the window allow new groups to be created, edited, deleted, etc. as follows:

Close Used to close the window. Any group\attribute changes will be saved, but the active group is not changed from when the window was opened. (Use *Select* to change the active group.)

Select Used to select one of the groups as the active group.

The active group is the group applied to parts, pins or nets when they are selected with the *Tools > EMC Attribute Tags > Apply* command.

Select the group to be made the active group, followed by the *Select* button.

Use the *Tools > EMC Attribute Tags > Apply* command to add the selected group to a part, pin or net.

Parts mode selection/editing window

There are three default groups defined for parts:

- 1 = Quiet
- 2 = Interference producer
- 3 = Interference susceptible.

When one of these groups is selected, the lower part of the window lists the attributes associated with that group, for instance the *Part placement hi-lite colour*. Any parts assigned to this group will have the colour specified when placed on the artwork.

This facility indicates to the board designer which parts (by their colour) are susceptible to noise, which ones produce noise and which are quiet.

The groups can be selected, modified, renamed, copied, deleted, or new groups defined using the commands across the top of the upper window. (However, new parts groups will have no practical use at the current time as there are no other attributes available.)

Part pins mode selection/editing window

There are three default groups defined for pins, they are:

- 1 = Load
- 2 = Source
- 3 = Terminator

When a group is selected the lower part of the window lists the attributes associated with that group.

In the case of the *Part pins* groups, there is only one attribute, *Net node type*. This is used by the Specctra auto-router, to control the order in which nodes on a net are routed. Refer to the Specctra Reference Guide for details on this feature (Rule, <reorder_descriptor>).

The groups can be selected, modified, renamed, copied, deleted, or new groups defined using the commands across the top of the upper window. (However, new part pins groups will have no practical use at the current time as there are no other attributes available to be inserted into the groups.)

Nets mode selection/editing window

There are no default groups defined for nets because of the variety of attributes that can be defined, so they have to be user-defined. Use the *New* button to create a group.

New Used to create a new emcrules group. Currently this command should only be used whilst in *Nets Mode*, as the other types (parts & part pins) do not have a variety of attributes available.

When a group is created, it can have a variety of attributes that control things like minimum track length, maximum track length, routing width, clearance, etc. When the group is assigned to a connection, the Specctra auto-router will obey the rules defined by the attributes within the group.

Creating a new group:

With *Nets Mode* active, select *Tools > EMC Attribute Tags > Select Attribute*, then *New* from the window. A window appears requesting a name for the new group. Enter a name required, followed by *Create*.

Select the new group from the upper part of the window, then select the *Choose Member Attributes* button, to add members (attributes) to the group. A list of all the members appears, a small selection is shown here:

Name	Description	Man ref	Active
cct_minclr	Minimum clearance	6-138 (STD)	No
cct_rwidth	Routing width	6-150 (STD)	No
cct_minrlen	Min routed length	6-19 (FST)	No
cct_maxrlen	Max routed length	6-19 (FST)	No
etc.			

All the attributes that you require to be part of the group should be "activated" by double-selecting them. *Yes* appears in the *Active* field.

For instance the new group could be required to define minimum and maximum track lengths for connections, in which case the "Min routed length" & "Max routed length" attributes should be made active.

Name is the internal name used by Ranger when referring to the attribute.

Description describes what the attribute is used to define.

Man ref indicates which page in the *Specctra User's Reference* manual (Nov 1995) describes this command. It also indicates which version of Specctra is required to implement the command. (STD=standard version, DFM=STD + design for manufacturability, FST=STD + fast circuit rules, ADV=STD + advanced rules)

Active indicates whether the attribute should be included as part of the group – select as many as required.

Select *OK* when all the required attributes have been selected. The lower part of the window lists the attributes selected and the values assigned to them. These values can be changed as required, by selecting the *Edit Member Attribute Values* button then typing in the values required for each member. Once the window is set as required, select *Close*.

To apply an Attribute group to a connection, select the attribute group required, and once highlighted, select the *Select* button. The window closes, and you are ready to *apply* the selected group to a

connection.

Choose Member Attribute - Used to select the attributes within a previously defined group. With *Nets Mode* active, select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups appears. Select the group to be edited, followed by *Choose Member Attribute*. The list of attributes appears from which attributes can be made active or inactive. Select *OK* to implement the changes.

Edit Member Attribute Values - Used to edit the values of attributes in a previously defined group.

With *Nets Mode* active, select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups appears. Select the group to be edited, followed by *Edit Member Attribute Values*. A window appears allowing the values to be changed. Select *OK* to implement the changes.

Copy - Used to copy a previously defined group.

With *Nets Mode* active, select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups appears. Select the group to be copied, followed by *Copy*. A window appears requesting a name for the new group. Select *Create* to copy the group.

The groups can be modified using the *Edit Member Attribute Values* and *Choose Member Attribute* commands.

Delete - Used to delete a previously defined group.

If the group has been assigned on the schematic, then a warning appears on the screen indicating that if you proceed with the deletion, all the occurrences of the group will be removed from the schematic. You are given the options whether to proceed, to replace all the effected instances with another attribute, or to cancel the deletion.

Select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups appears. Select the group to be deleted, followed by *Delete*. A window appears to confirm which group was selected. Select *Yes* to remove the group or *No* if you chose the wrong group.

If the group is currently in use on the schematic, then a warning appears to this effect. Follow the instructions in the window to either, cancel the deletion, proceed with the deletion and remove all the instances of that group from the schematic, or to proceed with the deletion and replace all the instances of that group with another group. Close the window as required.

Rename - Used to rename a previously defined group. All occurrences of the group name in use on the schematic are also changed on the schematic.

Select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups appears. Select the group to be renamed, followed by *Rename*. A window appears showing the old name and requesting a new name for the group. Select the empty box and type the new name. Select *Rename* to rename the group. (Select *Cancel* to cancel the rename.)

Import - Used to import the groups from the master library or another job, to the current schematic.

When a job is created, a copy of all the emc groups in the master library are copied into the job. If the master library is updated after a job has been created, the jobs are not updated. Likewise new groups can be defined within jobs, which will not appear in the master library.

When importing groups, all the groups within the selected job or library are imported unless they contain the same members and values. If a group has the same name, but different members, or different values assigned to the same members, then the group is imported. It would be wise to rename one of the groups after the import to avoid confusion.

Select *Tools > EMC Attribute Tags > Select Attribute*. The list of groups in the current schematic appears (if any have been defined). Select *Import*. A window appears listing all the jobs in the current jobs directory that have emcgroups defined, followed by the master library (unless the master library is being edited).

Select the job, or master library whose emcgroups you wish to import, followed by *Import*. All the emcgroups associated with either Parts, Part Pins or Nets, depending on which is selected, will be imported to the current schematic.

Tools > EMC Attribute Tags > Parts Mode

When selected, indicates that you wish to define or edit EMC attributes associated with parts. Any emc attributes that have already been assigned to parts will appear as flags attached to those parts.

Tools > EMC Attribute Tags > Part Pins Mode

When selected, indicates that you wish to define or edit EMC attributes associated with pins. Any emc attributes that have already been assigned to pins will appear as flags attached to the pins.

Note: the *Part Pins* attributes can only be used by the Specetra auto-router.

Tools > EMC Attribute Tags > Nets Mode

When selected, indicates that you wish to define or edit EMC attributes associated with nets (connections). Any emc attributes that have already been assigned to connections will appear as flags attached to the connections.

Note: the NETS attributes can only be used by the Specetra auto-router, although the *Check artwork* routine

does take into account the minimum clearance parameter on individual nets when specified.

Tools > EMC Attribute Tags > Copy

Used to copy an EMC rule flag currently visible on the screen, then apply it to another part, pin or net, depending on which mode is active.

Ensure the appropriate emc rule mode is selected, i.e. *Parts mode*, *Part pins mode* or *Nets mode*, then select *Tools > Copy*. Locate and select the flag required (point at the end point of the tail extending from the flag). The *Tools > EMC Attribute Tags > Apply* command becomes active, so simply select, with another click of the left button, the part, pin or net that the copied attribute should be applied to.

Tools > EMC Attribute Tags > Apply - Used to add the EMC attribute currently selected, to the selected part, pin or net.

Ensure the appropriate emc attribute type is selected, i.e. *Parts mode*, *Part pins mode* or *Nets mode*.

Ensure the correct emc attribute has been selected, using the *Tools > EMC Attribute Tags > Select Attribute* or *Copy* command.

Point at the part, pin or net that should have the emc rule added, then select it with a click of the left-hand mouse button. A flag containing the emc rule appears, with a tail extending to the selected part, pin or net.

A Ranger format parts/wiring list should be extracted from the schematic diagram in order for the emc rules to be passed through to the artwork and Spectra auto-router.

Tools > EMC Attribute Tags > Delete

Used to delete an EMC rule currently visible on the screen.

Ensure the appropriate emc attribute type is selected, i.e. *Parts mode*, *Part pins mode* or *Nets mode* then select *Tools > EMC Attribute Tags > Delete*. Point at the end point of the tail extending from the rule required, then select it with a click of the left mouse button. The rule is deleted.

Tools > Auto Symbol Generate

Used to automatically generate the design symbol that would represent the current sheet in a hierarchy.

Note: it is unwise to use this command if a symbol already exists and it is connected to, on an upper level sheet. Regenerating the symbol will probably move existing pins and cause them to become disconnected on the upper level sheet.

If a symbol already exists, a window will query whether or not you want the existing symbol overwritten.

To view the symbol, close the design sheet, then open the design symbol that will be found in the navigator, in the design's Schematic, Design Symbols folder.

The sheet must contain at least one **block I/O port**, or a symbol will not be created. The auto-generated symbol will contain one pin for each block I/O port on the sheet. The *Edit > Preferences, Miscellaneous* window controls the initial pitch of the pins in the symbol.

(A symbol is automatically generated when the design symbol is first opened if ports have been added to the design sheet. If extra ports have been added to the sheet since the symbol was initially generated, the extra pins are added alongside the symbol when the symbol is opened. *Tools > Auto Symbol Generate* will redefine the symbol if permitted, and place the ports appropriately.)

If ports are deleted from the sheet, the corresponding pins are automatically removed from the symbol.

Symbol commands

The symbol commands are used to add symbols (parts) and their associated attributes to the schematic sheet and then manipulate them.

Symbol > Attributes

Attributes are the text strings associated with parts, pins and connections. Changes made to part or pin attributes using these commands do not alter the attributes within the library part itself, they only affect the selected instance on the sheet.

An internal flag is assigned to the attribute on the selected instance, so that if the library part itself is subsequently altered, these changes are not overwritten. If you want the changes to be overwritten use the *Symbol > Attributes > Reset* command. This also means that if parts have been allocated on the schematic and pin numbers are changed within the library part, the part has to be de-allocated, then re-allocated for the changes to be implemented.

Once a part has been placed on the schematic sheet, changes made to attributes on the schematic take precedence over changes made to the library parts themselves.

When *Symbol > Attributes* is selected, the following commands appear.

Symbol > Attributes > Edit

Used to edit an attribute (or attributes) associated with a part or a part pin and the signal names within a bus.

Once selected, prompts appear in the status bar to indicate how to select items and provide messages when appropriate.

Using the left button to select an attribute string allows just that string to be edited, whilst using the right-button to select a part by its datum or a part pin, introduces a window containing all the attributes associated with the selected part or pin.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

If changes are made within the window select *OK* to implement the changes, or *Cancel* to forget the changes.

Part attribute window

The part attribute window (Figure 93) lists the attributes that have been defined for the part, the values assigned to the attribute, whether they are visible and their height.

If the field is optional, it may be deleted if required.

Not all of the attributes have values assigned to them.

If an asterisk (*) precedes a Name field entry (not the Value field), that attribute is a system attribute and should not be deleted as it is required by the program elsewhere. i.e. all the entries in Figure 93.

If an asterisk (*) precedes a value field entry (not the Name field), that entry cannot be changed from the window. i.e. the *Part ident* in Figure 93.

Changes made in this window do not affect any other parts.

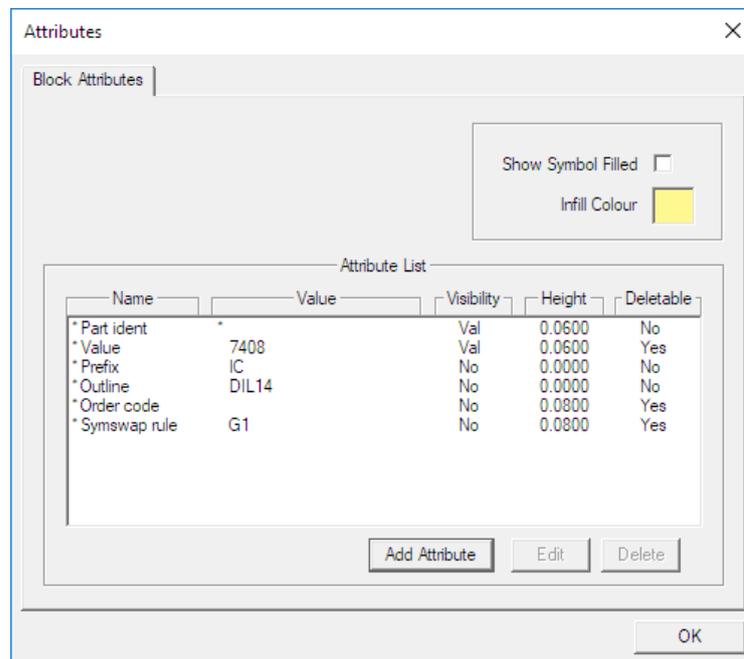


Figure 93

Show Symbol Filled:

Symbols can be displayed with a solid infill colour to make them stand out on the sheet by ticking this box.

(If an outline does not appear filled when this mode is activated, edit the symbol outline shape to ensure that it forms a closed path.)

Infill Colour:

If the symbol will be shown filled, choose the infill colour by selecting this box, then select the colour required from the colour palette window that appears.

The infill will be displayed and output using this colour, whereas the symbol outline colour is controlled in the schematic output task configuration window.

Attribute List Columns

Name contains the reference name of the attribute. If a * precedes the name, this is a system attribute and used elsewhere in the program – removing it (if permitted) would have implications for other tools, such as the extracted parts list, gate swapping in artwork, etc.

Value contains the text string assigned to the attribute. The * in this field means it cannot be edited from this window – the values for part ident's are controlled by the Prefix and Allocation commands.

More than one version can be assigned to a value field. For instance, it may be required that a resistor has a value of 220k in UK designs, but 100k in USA designs. Alternatively different values may need to be assigned within each instance of a repeated sheet.

The version names, and which version is currently active are defined in the *Edit > Preferences,*

Version Control window. The *Structure > Set Version Override* command is used to assign different values in repeated sheets.

The value should take the following format when specifying multiple values:

versionname!attribute_value versionname!attribute_value versionname!attribute_value

where:

versionname is the version name assigned in the *Edit > Preferences, Version Control* window.

! is the character defined for version control in the *Edit > Preferences, Miscellaneous*, window.

attribute_value is the value required for that version.

Examples:

UK!220k USA!100k

This will give a value of *220k* when displayed, plotted or compiled with the *UK* version selected and *100k* with the *USA* version selected.

Low!1k Mid!10k Hi!100k

This will give a value of *1k* when displayed, plotted or compiled with the *Low* version selected, *10k* with the *Mid* version selected and *100k* with the *Hi* version selected.

It is possible to specify a default version value using the version name ***. So for example:

UK!220k USA!100k *!500k

will produce *220k* if the *UK* version was selected, *100k* if the *USA* version was selected and *500k* for all other versions.

If the version control character is not used, then those attributes are unaffected by the version selection mechanism.

If a normally visible attribute has no value for a selected version, i.e. it has no *!* value* then (*null*) will be displayed on screen where the value would normally be. The (*null*) string will not appear when the sheet is plotted.

The version control character may be used in most part attributes (outline name, ordercode, value, etc.) in all user-defined attributes, in connection attributes (signal name) and in non-electrical text strings.

Visibility this field controls whether the attribute is visible or not when the part is used on the sheet.

Options are:

Not visible the attribute is not visible.

Value only the attribute's value field is visible

Name +Value the attribute's name and value field is visible

Height controls the height of the attribute when it is visible.

Deletable indicates whether the attribute is permitted to be removed from the part – this setting cannot be changed.

Note: if a system attribute (indicated by a *** - value, outline, order code, etc.) is deleted, adding a user-defined attribute with the same name does not make it a system attribute, so it will not be recognised as such. For example, the value assigned to a user-defined attribute named "Order code" will not appear in the parts list. To add a system attribute, use the export/import of attributes commands (right-click on the schematic folder).

To edit any of the above fields (except *Deletable*) select the attribute then the *Edit* button. Change the details as required followed by *OK*.

To delete any of the deletable attributes, select the attribute then the *Delete* button.

Default part attributes

The default settings for new parts are controlled from the *schema_attrbs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

Part ident refers to the reference designator (e.g. R1, R2, IC1, etc.). The actual characters are not displayed in this window as they are controlled from the *Alloc* menu.

Prefix controls the letters in front of reference designators, for example IC, R, C, etc. A prefix must be supplied. Upto 16 alpha-numeric characters can be used, in any order. This field is not normally visible as the part ident is more useful.

Outline controls the physical outline that is used for the part in board layout. An outline name must be given in order for a parts and wiring list to be produced in the correct format for board layout. Upto 16 alpha-numeric characters can be used, in any order. The outline can be created just prior to artwork design.

Value this field is optional, but typically contains the part type, i.e. 74LS00, 100k, etc. When filled in, the gate and pin swapping routines use this field to determine equivalent parts. Upto 16 alpha-numeric characters can be used, in any order.

Symswap this field is optional, but must be filled in if gate swapping will be required on the layout. Equivalent symbols are usually defined within the library part, using the *Attributes > Symbol Swap Rule* command - this field will therefore be filled in automatically. However, the field is either left blank or set to:

- G0** gate swapping not permitted.
- G1** gate swapping permitted between equivalent symbols within the part.
- G2** gate swapping permitted between equivalent symbols within the part and with other identical parts.

Order code this field is optional. It is inserted into the parts list if present.

Any other attributes are user defined.

User-defined attributes are added by selecting *Add attribute* button, then filling in the appropriate details.

To force a user-defined attribute to appear in the part attribute window when a part is created, edit the *schema_attribs.txt*, held in the *Configuration Data Directory* (MyDocuments/Seetrix/XL Designer/Data by default). This file also controls the default height, visibility and names for all attributes.

Pin attribute window

A pin attribute window (Figure 94) lists the attributes that have been defined for the pin, the values assigned to the attribute, whether they are visible and their height. If the field is optional, it may be deleted if required.

If a * precedes the Name, this is a system attribute and used elsewhere in the program – removing it (if permitted) would have implications for other tools, such as the electrical rules checker.

If an asterisk (*) precedes a Value field entry (not the Name field), that entry cannot be changed from the window. i.e. the *Outline pinref* value in Figure 94 which is controlled by the Allocation commands.

Changes made in this window do affect any other parts.

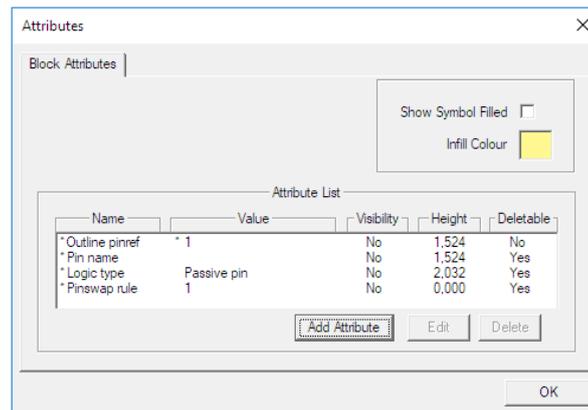


Figure 94

Note: If the symbol of a sheet is placed on a higher level sheet, and the pin name attribute is edited, the new name will appear against the block i/o pin when descending into the schematic sheet associated with the symbol.

An iopin attribute that is overridden by an attribute on a higher level instantiation will always show the overridden value when the sheet is displayed. The value may still be edited on the sheet, but the override value will still be displayed.

Switching to the symbol of the sheet will show the attribute value without the effect of any higher level override. The value displayed in this mode is the value displayed against the pin for any new instantiations of the symbol.

Over-riden attributes can be reset to their original values, using the *Symbol > Attributes > Reset* command. When re-setting the attributes of design symbols in a hierarchy, the attributes should be reset at the upper level.

Attribute List Columns

- Name** contains the reference name of the attribute. If a * precedes the name, this is a system attribute and used elsewhere in the program – removing it (if permitted) would have implications for other tools, such as the electrical rules checker.
- Value** contains the text string or value assigned to the attribute.
- Visibility** controls whether the attribute is visible or not when the part is used on the sheet. Options are:
Not visible - the attribute is not visible.
Value only - the attribute's value field is visible
Name +Value - the attribute's name and value field is visible

Height controls the height of the attribute when it is visible.
Deletable indicates whether the attribute is permitted to be removed from the part – this setting cannot be changed.

Note: if a system attribute (indicated by a *, i.e. Pin Name, Logic type) is subsequently deleted, adding a user-defined attribute with the same name does not make it a system attribute, so it will not be recognised as such. For example, the value assigned to a user-defined attribute named “*Logic type*” would not be recognised by the electrical rules checker.

To edit any of the above fields (except *Deletable*) select the attribute then the *Edit* button. Change the details as required followed by *OK*.

To delete any of the deletable attributes, select the attribute then the *Delete* button.

Default pin attributes

The default settings for new pins are controlled from the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

Outline pinref indicates the pin number of the pin. This pin number is linked automatically to the physical pad with the same pin number. Pin numbers can only be changed from within the library part or by allocation. Where the pin selected is an element from a multi-element part, then all the pin numbers for that pin are shown, separated with a | sign.

Pin name indicates the optional name assigned to the pin, for instance WE, CS, DI, etc. Upto 16 alpha-numeric characters can be used, in any order.

Logic type indicates the logic type of the pin. By default it is set to *Unspecified port*. The electrical rules checker refers to this information when checking the schematic for errors. Selection is made from the arrow alongside.

Pin swap rule indicates the pin group the pin belongs to. On the layout connections can be swapped between any pins within a gate or part that belong to the same group. Group numbers run from 0 to 6. Pins within group 0 are not swappable. Equivalent pins are typically defined within the library part, using the *Terminals > Set Pin Swap Rule* command.

Any other attributes are user defined. User-defined attributes are added by selecting *Add attribute* button, then filling in the appropriate details.

To force a user-defined attribute to appear in the pin attribute window when a part is created, edit the file *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

This file also controls the default height, visibility and names for all attributes.

Symbol > Attributes > Move

Used to move an attribute associated with a part or a part pin.

Once selected, point at the attribute to be moved, then click the left-hand mouse button. As the cursor is moved, the selected attribute is seen attached to it with a line extending to the part or pin it belongs to. Position the attribute and release it with a click of the left-hand mouse button.

As the attribute is being moved, it can be rotated by 90 degrees by pressing the *<rotate part>* special function key.

Clicking the right-hand mouse button whilst the attribute is being moved, restores it to its original location.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it.

Symbol > Attributes > Rotate

Used to rotate an attribute associated with a part or a part pin. Attribute strings can either be placed horizontally or vertically.

Once selected, point at the attribute to be rotated, then click the left-hand mouse button. The attribute string rotates by 90 degrees. If it is rotated again, it returns to its original orientation.

Attribute strings can also be rotated as they are being moved, by pressing the *<rotate part>* special function key.

The *Edit > Preferences, Miscellaneous* window, *Rotated text display mode* parameter controls the orientation of rotated characters.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it

Symbol > Attributes > Hide

Used to hide (make invisible) an attribute associated with a part or a part pin.

Once selected, point at the attribute string to be hidden, then click the left-hand mouse button. The attribute is hidden.

It can be restored to view if the *Symbol > Attributes > Edit* command is used on the part or pin, and the attribute made visible again.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Symbol > Attributes > Delete

Used to delete an attribute associated with a part or part pin.

Once selected, point at the attribute string to be deleted, then click the left-hand mouse button. The attribute is deleted. User-defined attributes can be added again using either the *Symbol > Attributes > Add* or *Symbol > Attributes > Edit* commands. System attributes can be restored using the *Symbol > Attribute > Reset* command.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Part references (part idents) or pin numbers (outline pinrefs) cannot be deleted.

Symbol > Attributes > Add

Used to add a user-defined attribute to a part or a part pin.

More than one version can be assigned to the attribute. Refer to the *Symbol > Attributes > Edit* command for details.

Once selected, point at and select the part or pin to which the attribute will be added. A window appears, enter the name of the attribute and the associated value field. Change the height and visibility of the attribute as required. (Refer to the *Symbol > Attributes > Edit* command for details on these attributes.) Select *OK* to close the window. As the cursor is moved, the new attribute is seen attached to it with a line extending to the part or pin it belongs to. Position the attribute and release it with a click of the left button.

As the attribute is being moved, it can be rotated by 90 degrees by pressing the *<rotate part>* special function key.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it.

Symbol > Attributes > Reset

Used to reset the attributes of the selected symbol to those found in the library part.

When re-setting the attributes of sheet symbols, the reset command should be performed on the symbol in the upper level in the hierarchy, not on the block i/o ports in the schematic sheet.

Part allocation information is not affected by this action.

Symbol > Place

Used to select parts from the current job, master library or other jobs ready for addition to the sheet.

Once selected, scroll through the Navigator window, until the schematic folder containing the part required is located - this might be from the current job, the Masters folder or from another open job.

If a part is selected with a click of the left button, it will be displayed in the *Browser* pane allowing it to be viewed and identified. Once selected, moving the cursor across to the sheet will show the selected part and it can be released with a click of the left mouse button.

Whilst being moved the part can be rotated in 90 degree increments or flipped by pressing the *<rotate part>* or *<flip part>* special function keys.

After releasing the part, another part appears on the end of the cursor, which can be released in the same way, or the right hand mouse can be clicked to cancel it.

If parts from the master library are placed on the sheet, copies of them are transferred into the job file automatically.

The name of the part is displayed along the bottom of the screen.

Commonly used parts can be added to the *fast part tray* for easy access. Refer to *View > Fast Part Tray* for details on the tray and its use.

Once a part has been added to the sheet, it can be copied using the *Symbol > Copy* command.

Refer to the *Grid* command menu to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up, etc. is enabled.

When the part is placed on the sheet from the master library, its associated component outline is not imported to the job until the parts/netlist is extracted from the schematic.

Symbol > Move

Used to move parts that are on the sheet. Buses can also be moved using this command.

Once selected, point at the datum of the part to be moved, then click the left-hand mouse button. As the cursor is moved, the selected part and its connections are seen attached to it. Position the part and release it with a click of the left-hand mouse button. Clicking the right-hand mouse button whilst the part is being moved restores it to its original location.

As the part is being moved, it can be rotated in 90-degree increments or flipped by pressing the *<rotate part>* or *<flip part>* special function keys.

Note: connections can get scrambled if *wire pickup* is enabled when the part is released (use Undo, then disable wire pickup).

To move a bus, point at its datum, which is displayed as a small circle or cross, and select it with a click of the left-hand mouse button. Move it as required, the adjoining connections stretch, then release it with a click of the same button. Clicking the right-hand mouse button before the bus is released cancels the move.

Refer to the the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up is enabled, how close you have to be to an item to select it, etc. and the *Grid* menu command to find out whether grid snapping is active.

Symbol > Rotate

Used to rotate parts that are already on the sheet.

Once selected, point at the datum of the part to be rotated, then click the left-hand mouse button. The part rotates by 90 degrees anti-clockwise each time it is selected. Note: connections can get scrambled if *wire pickup* is enabled during the rotation (use Undo, then disable wire pickup).

Parts can also be rotated as they are moved, if the *<rotate part>* special function key is pressed.

Refer to the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up is enabled and how close you have to be to an item to select it.

Symbol > Flip

Used to flip parts that are already on the sheet.

Parts are flipped about their original y-axis, i.e. parts are turned left to right about their datum unless they have been rotated by 90 or 270 degrees first, in which case they are turned top to bottom about their datum.

Once selected, point at the datum of the part to be flipped, then click the left-hand mouse button. The part flips each time it is selected.

Parts can also be flipped as they are moved, if the *<flip part>* special function key is pressed.

Refer to the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up, etc. is enabled and how close you have to be to an item to select it.

Symbol > Delete

Used to delete parts from the sheet.

If an allocated part is deleted, existing part allocations are unaffected. Subsequent automatic allocations will use the spare part names and gates in numerical order.

The part's connections remain after it has been deleted from the sheet. Another part could be moved and released with its pins over the ends of the connections and joined if wire pick-up is enabled in the *Edit > Preferences, Miscellaneous* window.

Once selected, point at the datum of the part to be deleted, then click the left-hand mouse button. The part is removed from the sheet.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Busses can also be deleted using this command, but only the signal names and angled section of the bus and connecting line are removed. Point at the bus datum, which is displayed as a small circle or cross, and select it with a click of the left-hand mouse button.

Points to bear in mind when deleting parts from a design where the parts have been placed on the artwork

When removing parts from the schematic, **never** de-allocate the remaining parts in order to re-allocate using consecutive numbers. This will have the effect of swapping parts around on the artwork **without** their tracks (if routed). Instead, use part renumbering from the artwork editor, which will back-annotate to the schematic.

Once a part has been removed from the sheet and the parts/wiring list extracted, the part will be removed from the artwork but its tracks and silk screen data (if present) remain in position on the artwork.

If a part is removed from the sheet and another part is inserted with the same reference designator and different connections, the new part's datum will appear in the same position on the artwork as the old part's datum, along with its new connections. The old part's tracks and silk screen data (if present) will remain on the artwork until they are manually deleted.

Symbol > Copy

Used to copy a part that is already on the sheet.

If attributes have been added/changed on the selected part, they are transferred to the copied part, including any EMC rules that have been defined for the part or part pins. (They can be reset using the *Symbol > Attibs > Reset* command.)

Once selected, point at the datum of the part to be copied, then click the left-hand mouse button. As the cursor is moved, a copy of the selected part is seen attached to it. Click the right-hand mouse button to cancel the copy, otherwise position the part and release it with a click of the left-hand mouse button. Another part appears

on the end of the cursor, release it in the same way if it is required.

Once the first copy has been released, its name, and the library it came from is displayed along the bottom of the screen.

As the part is being moved, it can be rotated in 90 degree increments or flipped by pressing the *<rotate part>* or *<flip part>* special function keys.

Click the right-hand mouse button to stop further copies appearing on the end of the cursor. *Copy* remains active until another command is selected, so another part could be selected and copied if required.

Refer to the *Grid* menu commands to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up, etc. is enabled and how close you have to be to an item to select it.

Symbol > Power Place

Used to add power symbols to the sheet. Power symbols are associated to packages and can only be added after the package has been allocated.

Once selected, a selection window appears to allow specific categories of parts to be added. Once selected and as the cursor is moved, a power symbol is seen attached to it. The part to which it belongs is highlighted in the Part Powerblocks list on the left-hand side of the schematic window.

If you do not want to use the power symbol for that particular part, move the cursor across to the list and select one of the other parts with a click of the left-hand mouse button. The power symbol for that part appears attached to the cursor.

Position the power symbol and release it with a click of the left mouse button. The next power symbol in the list appears on the end of the cursor, release it in the same way if it is required, or select another from the list.

Power symbols appear on the end of the cursor until another command is selected or there are no more parts in the list.

As the power symbol is being moved, it can be rotated in 90 degree increments or flipped by pressing the *<rotate part>* or *<flip part>* special function keys.

Refer to the *Grid* menu commands to find out whether grid snapping is active.

Notes on power symbols

By default power symbols are not required. Pins that are defined as power pins within the part's attribute window are automatically connected to the appropriate power rails when the parts and wiring list is extracted from the schematic, even if a power symbol is specified within the part.

However, it is often useful to view the schematic and be able to see which part pins are connected to the power rails. There are also requirements to use the same part on a schematic but connected to different power rails, for example op amps, or to connect a power pin via a pull-up or down resistor to the power rail. In these situations *power symbols* are used.

A power symbol is created as a *library primitive* and called up from within the *library part*. Refer to the worked example in the manual for details on creating and using power symbols.

N.B. If a power symbol is shown on the schematic, its pins must be connected to something, or the pins will be unconnected in the wiring list and on the layout.

Symbol > Add to Artwork Tray

When this mode is selected, clicking on schematic symbols that are related to parts list entries will cause those parts to be added to the artwork placement tray, whether or not the artwork editor is open.

In this mode, all symbols related to parts already placed on the board, or already in the placement tray will be shown with a dotted outline. Symbols for parts that can be placed in the tray will be shown with a solid outline.

Note: once added to the placement tray the parts can only be removed using the artwork editor tools.

Symbol > Alignx/Aligny

Used to align parts in either the X or Y axis by their datum. Each command operates in the same way, except for the axis of alignment; the X axis is the vertical axis, and the Y horizontal.

Once selected, locate the cursor in a position suitable for aligning the required parts and click the left-hand mouse button. A dashed line appears that represents the axis of alignment. Select each part to be aligned with a click of the **right-hand** mouse button. The parts move so that their datum lies over the axis line.

The alignment axis can be moved by selecting another position with a click of the left-hand mouse button.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up, etc. is enabled and how close you have to be to an item to select it.

Symbol > Replace

Used to replace a part on the schematic for another.

This command is typically used to update blocks (parts) on the sheet that have been modified since they were

added to the sheet. (Parts on the sheet are not automatically updated when the part is modified.) This command can also be used to replace one part for a different part.

Connectivity is maintained provided the old and new pins are in the same location. If the new pin(s) lie on top of a connection then they will become attached, irrespective of the setting for "Pick up wires during symbol move" in the *Edit > Preferences, Miscellaneous* window.

Part allocation information is lost.

Attribute fields are reset to the values defined by the part in the library.

Once selected, select the replacement part from the schematic folder in the navigator (from any design or the master library), it appears attached to the cursor. Move the part over the datum of the part to be replaced, then select it with a click of the left-hand mouse button. The part is replaced.

Symbol > Logic Type

Used to replace the symbol of a part with its DeMorgan graphical alternate - which is identical apart from its graphical representation. The alternative symbol is defined from within the part (*Type* setting in the toolbar).

Once selected, point at the part whose symbol is to be replaced, then click the left-hand mouse button, the symbol of the part is changed.

If a message *No alternative symbol available* appears in the status bar, then the part does not have an alternate outline defined.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Symbol > Show Datasheet

Used to display the web-page or file associated with the *Data-sheet Address* attribute for the selected part.

Once this command has been selected, select a part on the sheet, the associated page will appear (assuming an Internet connection is available for access to web-based pages). If a page has not been associated with the part, a message indicating this will appear in the status bar.

Wires commands

These commands are associated with adding connections to the schematic along with connection attributes such as signal names, track widths and minimum clearance values for the artwork.

Wires (or connections) can be made by inserting a connection between pins, or by attaching a connection to a pin and then adding a signal name attribute to it (*Wires > Attributes > Add*).

Connectivity can also be formed by laying pins on top of one another or close to one another. The *Edit > Preferences, Miscellaneous* window controls whether or not this method of making a connection is enabled, and how close pins have to be to one another before the "connection" is made.

When wires are attached to symbol terminals, the terminal cross disappears to indicate the connection is "made" or attached.

When the parts/wiring list is extracted or *compiled* from the schematic (*Tools > Parts/Netlist Extraction*), a list is created of all the connected pins. It can either be compiled as a *flattened* schematic or a *hierarchical* schematic – this setting controls connectivity between sheets.

If compiled as a *flat* schematic, every connection with a signal name is automatically connected to every other connection with the same name on the same sheet and any other sheet within the design.

If compiled as a *hierarchical* schematic, then connections with signal names are regarded as making "local" connections on that sheet. They connect all connections with the same name together, but only within a sheet. They do not maintain connectivity between sheets. To maintain connectivity between sheets, block I/O ports have to be added to the connections on the sheet, and a connection has to be made between the appropriate pins on the symbol representing that sheet on the upper level in the hierarchy.

For example, take four different sheets that are used in a hierarchical schematic. Each sheet contains connections with a signal name *ABC*. To link all the *ABC*'s between sheets together, a block I/O port has to be added to one of the *ABC* connections within each sheet, and given a name, for instance *ABC* (its sensible to use the signal name to avoid confusion). A design symbol has to be made for each sheet, which will contain a pin called *ABC* automatically (or whatever name was given to the port).

Another sheet should be created (the top level sheet), and the symbols representing the four sheets placed on it. The pins called *ABC* on the symbols should be connected together to complete the global connections. If these connections were not added, there would be four separate *ABC* connections in the wiring list.

Implicit power connections defined within parts make global connections, providing the library part is used somewhere in the design.

Negated signal names can be added by preceding the signal name with the *negation bar prefix character* as defined in the *Edit > Preferences, Miscellaneous* window.

Wires > Attributes commands

Connection attributes are the text strings associated with connections, such as signal names, width codes.

Note: signals names within a bus can only be modified with the *Symbol > Attributes > Edit* command.

Wires > Attributes > Edit

Used to edit an attribute, or attributes associated with a connection. (Bus signal names can only be modified using the *Symbol > Attributes > Edit* command.)

Once selected, if the attribute string to be edited can be seen, point at it then select it with a click of the left-hand mouse button.

If the attribute string can't be seen, point at the end or a corner of the connection, then select it with a click of the *right-hand* mouse button.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

If an attribute string was selected with a click of the left-hand button, a window appears containing the attribute string that can be modified.

If a connection was selected with a click of the right-hand mouse button, then a window appears with all the attribute information associated with the connection. If changes are made within the window select *OK* to implement the changes, or *Cancel* to forget the changes.

Connection attributes window

The connection attribute window (Figure 95) lists the attributes that have been added to the connection, the value assigned to them, whether they are visible, their height and whether they can be deleted.

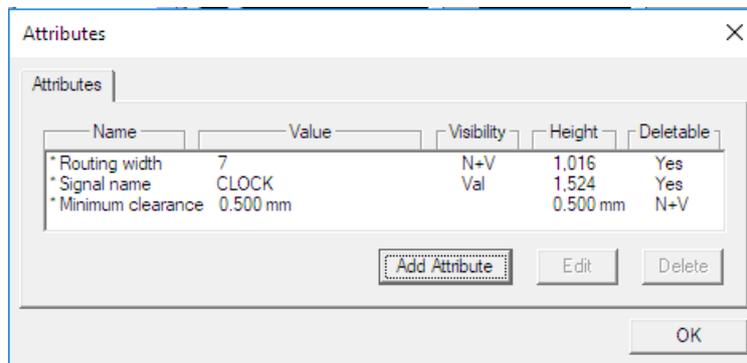


Figure 95

Attribute Fields

Name contains the reference name of the attribute.

If an asterisk (*) precedes a Name field entry, that attribute is a system attribute and reserved for use by the software. If deleted and then subsequently required, they should be restored using the *Wires > Attributes > Add* command (not the *Add Attribute* button from the Attributes window which is used to add user-defined attributes.)

Value contains the text string or value assigned to the attribute. More than one version can be assigned to a value field, for instance to assign a signal name of *CLOCK* in *UK* designs, but *URE* in German (*GDR*) designs.

The version names, and which version is currently active are defined in the *Edit > Preferences, Version Control* window. The value should take the following format when specifying multiple values:

versionname!attribute_value versionname!attribute_value versionname!attribute_value

where:

versionname is the version name assigned in the *Edit > Preferences, Version Control* window.

! is the character defined for version control in the *Edit > Preferences, Miscellaneous* window.

attribute_value is the value required for that version.

Example:

UK!CLOCK GDR!URH

This will give a value of *CLOCK* when displayed, plotted or compiled with the *UK* version selected and *URH* with the *GDR* version selected.

It is possible to specify a default version value using the version name *. So for example:

GDR!URH *!CLOCK

will produce *URH* if the *GDR* version was selected, but *CLOCK* for all other versions.

If the version control character is not used, then those attributes are unaffected by the version selection mechanism.

If a normally visible attribute has no value for a selected version, i.e. it has no *! value then (*null*) will be displayed on screen where the value would normally be. The (*null*) text will not appear when the sheet is plotted.

The version control character may be used in most part attributes (outline name, ordercode, value, etc.) in all user-defined attributes, in connection attributes (signal name) and in non-electrical text strings.

Visibility controls whether the attribute is visible or not. Options are:

Not visible - the attribute is not visible. Note: connection attributes are assigned to a segment of wire, for this reason they can be difficult to restore to visibility as the same segment has to be selected to change the setting.

Value only - the attribute's value field is visible

Name+Value - the attribute's name and value field is visible

Height controls the height of the attribute when it is visible.

Deletable indicates whether the attribute is permitted to be removed from the part – this setting cannot be changed.

Note: if a system attribute (indicated by a *, i.e. Signal name) is subsequently deleted, adding a user-defined attribute with the same name does not make it a system attribute, so it will not be recognised as such. For example, the value assigned to a user-defined attribute named "Signal name" would not be recognised by the parts/wiring list extractor. Use the *Wires > Attributes > Add* command instead.

To edit any of the above fields (except *Deletable*) select the attribute then the *Edit* button. Change the details as required followed by *OK*.

Default connection attributes

Routing width allows a width code to be assigned to the connection. Only numbers from 0 to 511 should be used. These numbers correspond to the track widths table for the artwork design, so track widths can be defined at the schematic stage.

By default width code 4 is used for signal connections and width code 11 for power connections.

Track widths can also be changed at the layout stage if required.

Signal name allows a name to be given to a connection. All connections with the same signal name on the **same sheet** are automatically connected together in the wiring list and therefore the layout. This means that connections do not have to be drawn on the sheet, making a less cluttered drawing. Upto 16 alpha-numeric characters in any order can be used.

When the circuit is compiled into a parts and wiring list, it is configured as a *flat* or *hierarchical* schematic. If *flat* is selected, then all connections with the same name on all sheets are linked together in the wiring list. If *hierarchical* is selected, then like-named connections between sheets are **only** connected if block I/O ports have been attached to them and the corresponding symbol pins connected at the upper level. (Refer to the *Tools > Parts/wiring list extraction > Setup* command for more details.)

Minimum clearance allows a minimum clearance to be applied to the connection. The online DRC tools in the artwork editor and the artwork checking routines will use this value instead of the default minimum clearance applied to the job.

Note: when the attribute is added the units will be included, for example: "*Minimum clearance = 1.25 mm*" they should not be removed if the value is subsequently modified as the wiring list extractor requires them to ensure the correct value is passed to the artwork editor/checking routines.

Any other attributes in the window have been user defined. More attributes can be added if *Add attribute* is selected, but only the user-defined extraction tools will use them.

Wires > Attributes > Move

Used to move an attribute associated with a connection.

Once selected, point at the attribute to be moved, then click the left button. As the cursor is moved, the selected attribute is seen attached to it with a line extending to the wire it is associated with. Position the attribute and release it with a click of the left button.

As the attribute is being moved, it can be rotated by 90 degrees by pressing the *<rotate part>* special function key. (The *Edit > Preferences, Miscellaneous, Rotated text display mode* parameter controls the orientation of rotated characters.)

Clicking the right-hand mouse button whilst the attribute is being moved, restores it to its original location.

Refer to the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it and the *Grid* menu command to find out whether grid snapping is active.

Wires > Attributes > Rotate

Used to rotate an attribute associated with a connection. Attribute strings can either be placed horizontally or vertically. The *Edit > Preferences, Miscellaneous* window, *Rotated text display mode* parameter controls the orientation of rotated characters.

Once selected, point at the attribute to be rotated, then click the left button. The attribute string rotates by 90 degrees about the centre of the string. If it is rotated again, it returns to its original orientation.

Attribute strings can also be rotated as they are being moved, by pressing the *<rotate part>* special function key. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it

Wires > Attributes > Hide

Used to hide (make invisible) an attribute associated with a connection.

Note: these hidden attributes can be difficult to select in order to make them visible/edit them in the future, so hide them with caution. (They can be restored to view using the *Wires > Attributes > Edit* command, but the segment associated with the hidden attribute has to be selected.)

Once selected, point at the attribute string to be hidden, then click the left-hand mouse button. The attribute is hidden.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Wires > Attributes > Delete

Used to delete an attribute associated with a connection.

Once selected, point at the attribute string to be deleted, then click the left button. The attribute is deleted.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Note: use the *Wires > Attributes > Add* command to restore any deleted System attributes.

Wires > Attributes > Add

Used to add a system or user-defined attribute to a connection.

There are 3 system wire attributes, which are: *signal name*, *routing width* and *minimum clearance*. The values assigned will be applied to the wire(s) in the artwork and used by the artwork checking routines.

User-defined attributes allow your own attributes to be applied to a wire, these attributes are only accessed by the user-defined output tools (*Tools > Parts/Netlist Extraction > User Defined*).

Once the command has been selected, point at the connection to which the attribute will be added, then click the left button. A window appears.

Select the *Type* of attribute required. (If the connection has already had an attribute applied, only the available attributes and *User defined* will appear in the window.)

Type the value of the attribute in the value field. If a name has previously been added, then that name will appear in the value field of the window automatically. It may be retained or modified.

Signal names – alpha-numeric characters, in any order can be used. Note: the wiring list and artwork tools have a maximum signal name limit of 16 characters, if more characters are used, the signal name will be truncated in the wiring list when the wiring list is extracted – a warning message is given.

Negated signal names can be added by preceding the signal name with the *negation bar prefix character* as defined in the *Edit > Preferences, Miscellaneous* window. A bar is drawn over the signal name on the screen and in plots, the prefix character is not displayed. Note however that the negation bar is not used in the wiring list or artwork editors, when the prefix character will appear in front of the signal name.

routing width – a default track size of 4 & 11 are used for signals and power rails respectively. If a different track code should be used, it can be applied here, specify the track code (0 to 511), not the actual size of the track.

minimum clearance – a default minimum clearance value for all tracks on the artwork is used unless a specific minimum clearance value is assigned to individual wires. (Signal and power rails each have their own default minimum clearance which is assigned when the artwork is checked.)

Enter the value required, the current units (as set when entering the schematic editor) will be used and displayed when the attribute is positioned.

Note: when the attribute is added the units will be included, for example: "*Minimum clearance = 1.25 mm*". The units should not be removed if the value is subsequently modified as the wiring list extractor requires them to ensure the correct value is passed to the artwork editor/checking routines.

user-defined – enter the name for your attribute and also the value to be assigned to it. Only the user-defined output tools (*Tools > Parts/Netlist Extraction > User Defined*) will access this information.

Change the height and visibility of the attribute as required.

Select *OK* to close the window. (Select *Cancel* if you do not want to add the attribute.) As the cursor is moved, the new attribute is seen attached to it, with a line extending to the connection it belongs to. Position the attribute and release it with a click of the left-hand mouse button.

As the attribute is being moved, it can be rotated by 90 degrees by pressing the *<rotate part>* special function key. (The *Edit > Preferences, Miscellaneous, Rotated text display mode* parameter controls the orientation of rotated characters.)

More than one version can be assigned to the attribute. Refer to the *Wires > Attributes > Edit* command for details.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it.

Clicking the right-hand mouse button cancels the new attribute.

Wires > Insert Connection

Used to insert connections.

A parameter within the *Edit > Preferences, Miscellaneous* window controls whether connections are allowed to start in free space, or whether they must start on a pin or an existing connection. One end of a connection must be attached to a pin.

Once selected, point at the starting point, which is usually a pin or an existing connection, then click the left-hand mouse button. As the cursor is moved, a connection is seen attached to it.

The connection either stretches like a piece of elastic, or horizontal and vertical lines stretch as you move the cursor - this is controlled by the *Wire 0/90 Lock* setting, accessed from the *Wires* drop-down menu or an icon in the toolbar.

As the connection is stretched, corners can be inserted with a click of the left-hand mouse button. The connection can be terminated on a connection or a pin with a click of the left-hand mouse button. To release the connection in mid-air, a double click of the left button is required.

When the connection is released, the terminal crosses representing the symbol pins disappear to indicate a connection has been made - if they remain, then the wire is not attached to the pin.

Connections left hanging in mid-air should be given a signal name to make a connection to another connection with the same name.

The *Wires > Attributes* commands are used to add signal names and other connection attributes. Refer to the details given previously under the *Wires* description for further information on named signals.

Once inserted, the connection can be modified using the other *Wires* commands.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it.

Wires > Insert Bus

Used to add bus type connections to the sheet, similar to those shown in Figure 96.

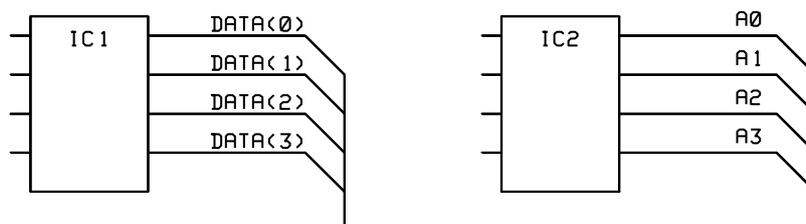


Figure 96

Once selected a window appears, similar to the one in Figure 97.

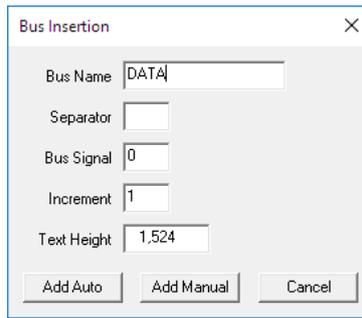


Figure 97

Fill in the parameters as follows:

Bus name The name used to identify the bus, i.e. DATA or A or D, etc. Upto 16 alpha-numeric characters can be used in any order. Note: the total maximum length for a signal name is 16 characters. Additional characters will get truncated when converted to a Ranger format wiring list (a warning is given).

(If the signal names within the bus are not similar, or numbered, perhaps you want to add CLOCK, RESET, etc. into the bus, add the bus then change the signal names individually using the *Symbol > Attributes > Edit* command.)

More than one version can be assigned to the bus name. For instance, you may require the bus name to have a value of D in UK designs, but DATA in German (GDR) designs. The version names, and which version is currently active are defined in the *Edit > Preferences, Version Control* window. The value should take the following format when specifying multiple values:

versionname!attribute_value versionname!attribute_value versionname!attribute_value

where:

versionname is the version name assigned in the *Edit > Preferences, Version Control* window.

! is the character defined for version control in the *Edit > Preferences, Miscellaneous* window.

attribute_value is the value required for that version.

Example:

UK!D GDR!DATA

This will give a busname of *D* when displayed, plotted or compiled with the *UK* version selected and *DATA* with the *GDR* version selected.

It is possible to specify a default version value using the version name *. So for example:

GDR!DATA *!D

will produce *DATA* if the *GDR* version was selected, but *D* for all other versions.

If the version control character is not used, then those attributes are unaffected by the version selection mechanism.

The version control character may be used in most part attributes (outline name, ordercode, value, etc.) in all user-defined attributes, in connection attributes (signal name) and in non-electrical text strings.

Separator Up to two characters can be entered. The first character is placed between the *bus name* and the *bus signal* name. If a second character is entered it will be placed after the *signal name*.

Bus signal The name given to the first signal in the bus. Upto 3 numeric characters can be used.

Increment The number added to the previous signal name. This number will be used as the signal name for the next connection in the bus.

Text height The height for the names used on the bus connections.

Examples:

Bus name : DATA	Bus name : DATA	Bus name : DATA
Separator : ()	Separator : :	Separator : :
Bus signal : 0	Bus signal : 0	Bus signal : 0
results in : DATA(0)	results in : DATA:0	results in : DATA0

Once the details have been entered, either *Add Auto* or *Add Manual* can be selected to add the bus.

Add Auto should be selected when a row or column of pins will be connected to the bus. The bus signal names

will increment along the row or column in numerical order as specified in the window.

Add Manual should be selected when the pins connected to the bus are not in a straight line, or the signal names do not increment in a numerical order along the row or column of pins.

Add Manual when selected, a flag appears attached to the cursor containing the first bus signal number. Select the pin that you wish to add the connection to, with a click of the left-hand mouse button. The number specified in the increment field of the window increases the number in the flag on the end of the cursor. Continue selecting the pins that are to be connected to the bus in the correct order. Click the right-hand mouse button when no more numbers are required. Move the cursor, the bus appears and stretches as the cursor is moved.

As the bus is moved up and down or from side to side, the 45 degree segments change direction. Click the left button to release the bus when you are happy with its position.

Add Auto when selected, a flag appears attached to the cursor containing the word *first*. Select the first pin in the required bus with a click of the left-hand mouse button. The word in the flag on the end of the cursor changes to *last*. Select the last pin in the required bus. Move the cursor, the bus appears and stretches as the cursor is moved.

As the bus is moved up and down or from side to side, the 45 degree segments change direction. Click the left button to release the bus when you are happy with its position.

The crosses that represent the symbol pins disappear when the bus is released to indicate a connection has been made to the pin – if they remain, then the wire is not attached to the pin.

Connections can be added to the end of the bus if required using the *Wires > Insert Connection* command. Bus connections are automatically added at the bus width, as defined in the *Edit > Preferences, Line widths* window.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Wires > Add/Move Corner

Used to add, or move existing corners in connections. This command can also be used to move a bus. Once selected:

Adding a corner Point at the connection, then click the **right-hand** mouse button. Move the cursor, the new corner is attached to it and the adjacent segments stretch with it. Release the new corner with another click of the same mouse button. (Click the opposite mouse button (left) to cancel the new corner.)

Moving an existing corner Point at the corner, then click the **left-hand** mouse button. Move the cursor to move the corner and stretch the adjacent segments.

The *Wires > 0/90 degree lock* command controls whether the adjacent segments move orthogonally or freely.

Release the corner with another click of the same mouse button. (Click the opposite mouse button (right) to cancel the move.)

Moving a bus Point at the datum point of the bus which is displayed as a small circle or cross, and select it with a click of the left-hand mouse button. Move it as required, the adjoining connections stretch, then release it with a click of the same button. Click the opposite mouse button (right) before the bus is released to cancel the move.

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* window to find out whether wire pick-up is enabled, how close you have to be to an item to select it, etc.

Wires > Delete Connection

Used to delete connections and busses. If the selected connection contains many branches, the connection between junction blobs or pins, whichever is shortest is deleted.

Once selected, point at the connection and click the left-hand mouse button. The connection is deleted.

The crosses representing symbol pins reappear when the attached wires are deleted.

If there is difficulty selecting the connection, select the connection on a corner or end point. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

To delete a bus, point at its datum, which is displayed as a small circle or cross, and select it with a click of the left-hand mouse button. A window appears, indicate what you would like deleted:

Remove bus symbol only to delete the bus and the signal names. The wires connecting the bus to the part pins remain.

Remove bus symbol and wires to delete the bus, the signal names and the wires connecting the bus to the part pins.

Wires > Delete Corner

Used to delete corners from connections.

Once selected, point at the corner to be deleted, then click the left button. The corner is removed from the connection.

Refer to the *Edit > Preferences, Miscellaneous* window to find out how close you have to be to an item to select it.

Wires > Flat Circuit Import Assist > Toggle Wire Section To Non Electrical

This command is intended to be used on schematics that were originally drawn in the old, flat-style schematic editor to allow a parts/wiring list to be extracted successfully.

This command permits a section of a connection to be switched to non-electrical mode, thereby hiding its connectivity from the parts/wiring list extractor.

Wire segments connecting to component pins, and segments which are not the first, or last in a wire link cannot be changed.

Back-ground information: There are major differences in the way that "bus" signals were handled in the flat schematic editor compared with the current hierarchical editor. The hierarchical editor uses specially created bus symbols to handle busses, whereas the flat schematic editor indicated busses by setting wires to "bus width". The old, flat style schematic parts/wiring list extractor essentially ignored wires set at "bus width" and maintained connectivity by using the signal names.

The hierarchical editor parts/wiring list extractor is unable to operate in this manner, so all imported "bus width" wires are converted to non-electrical features so they cannot be seen by the extractor.

Some schematics with busses may need some manual editing after loading if it is desired to extract a parts and net list. If parts/wiring list extraction results in errors because multiple signal names are found on a net, then it may be necessary to convert more wire sections to non-electrical mode to hide them from the compiler.

Alternatively the old-busses/non-electrical representations could be removed and the busses re-added using the *Wires > Insert Bus* command.

Alloc commands

These commands are associated with part allocation and de-allocation - adding and removing part references and pin numbers.

Individual parts, sheets, or the complete design can be allocated or de-allocated according to the selections made. A list of spare gates and unallocated gates can also be detected.

Errors cannot be made during allocation; Ranger refers back to the library parts for packaging information and checks to see which parts and gates have already been allocated on the design. For example you won't find two R1's.

Note well: if parts have been allocated and they are subsequently modified using the *Block > Edit* command, and their pin numbers changed, they will need to be de-allocated, then re-allocated to update the parts and pin numbers used on the schematic.

Before deallocating parts that have already been placed on the artwork, refer to the *Alloc > Deallocate* command for important information.

Alloc > Auto Allocate

Used to allocate all parts not already allocated on a sheet or through the hierarchy. Already allocated parts are not changed. Only unused part numbers and gates are used during allocation.

When allocation is used, the allocation offset (the starting number for parts within sheets) for each sheet is controlled from the *Structure > Set Allocation Offset* command. By default, parts are numbered from one upwards.

Once selected a window appears, giving control over the parts to be allocated. Select one of the options from the window to perform allocation, or select *Cancel* from the top of the window to cancel the command.

Allocate current sheet

Used to allocate reference designators and pin numbers to all the unallocated parts on the sheet being edited.

Allocate current sheet downwards through hierarchy

Operates in the same way as *Allocate current sheet*, except the current sheet *plus* all levels within the hierarchy *below* the current sheet are also allocated.

Allocation is performed starting from the bottom of one hierarchical branch up to the top, it then starts from the bottom of another branch up to the top and so on. Blocks on the same level within a branch are taken one after the other.

Allocate everything

Operates in the same way as *Allocate current sheet*, except *all* the sheets within the design are allocated. Allocation is performed starting from the top of the hierarchy downwards.

Alloc > Allocate Single

Used to allocate selected parts on the current sheet.

Once selected, point at the part to be allocated and click the left button. The allocation takes place.

Already allocated parts cannot be changed, they have to be de-allocated first.

Allocation looks to the allocation offset to find out which should be the first reference number used on the current sheet. If that number is in use anywhere within the design, then the next available number (or gate) is used for the part. The Allocation Offset is controlled via the *Structure > Set Allocation Offset* command.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to the part's datum to select it.

Alloc > Allocate Typed

Used to allocate a specific part number to the selected part.

Once selected, point at the part to be allocated and click the left button. A window appears requesting the number for the part. The prefix is automatically obtained from the symbol's attributes. Type in a number and press <enter>. The allocation will take place providing the part number is not already in use – a message is given in the status bar if the allocation cannot be performed.

Already allocated parts cannot be changed, they have to be de-allocated first.

When multi-gate parts are being assigned, the first available gate of the correct type within the package is used. If required, the part number followed by a pin number or gate identifier can be entered in the window. This forces a specific element within the package to be used, providing it is valid and unused.

For instance if a 7400 element is selected and **3.6** entered, the element will become IC3 gate 'b' because pin 6 was specified (pins 4, 5 and 6). Likewise, **3c** could be entered to use the third gate in the package.

Ranger will not allow gates to be allocated to the same part reference if the *Value* field of the gates is different. This also applies to split parts – all the value fields of the primitives that make up a split part must be the same. If they will be allocated the same reference number.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to the part's datum to select it.

Alloc > Show Freelist

Used to indicate free gates within allocated parts in the design, i.e. spare gates in a package.

Gates that have been added to the design, allocated but not connected to, are not regarded as free gates so will not be listed.

Unallocated parts are not included within this check. So always use the *Alloc > Unallocated Check* command first, to indicate any parts that have not been allocated.

When selected a window appears similar to that in Figure 98, listing all the free gates within allocated parts, if there are any.

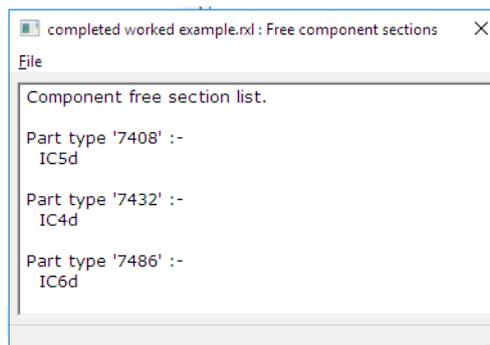


Figure 98

Alloc > Unallocated Check

Used to flag any unallocated parts or gates within the design.

All parts on the schematic should be allocated before the parts and wiring list and layout are created, otherwise parts and/or connections will be missing from the layout.

When selected a window appears listing all the unallocated parts within the design, similar to that in Figure 99.

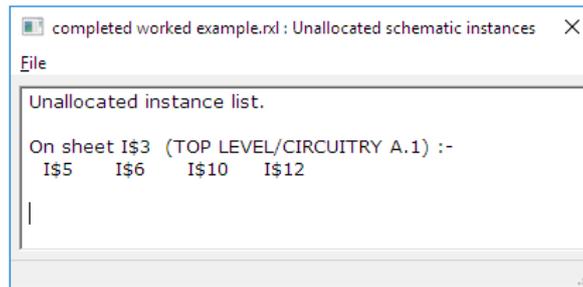


Figure 99

The part's *instance identifiers* are used to indicate the unallocated parts because they do not have reference designators. Use the *Find > Instance* command to locate them.

Alloc > Auto De-allocate

Used to de-allocate all parts on a sheet or through the hierarchy. See also *Notes on de-allocation* below. Once selected a window appears, giving control over the parts to be de-allocated.

De-allocate current sheet

Used to de-allocate reference designators and pin numbers from *all* the parts on the sheet being edited.

De-allocate current sheet downwards through hierarchy

Operates in the same way as *De-allocate current sheet*, except the current sheet *plus* all levels within the hierarchy *below* the current sheet are also de-allocated.

De-allocate everything

Operates in the same way as *De-allocate current sheet*, except *all* the sheets within the design are de-allocated.

Alloc > De-allocate Single

Used to de-allocate a selected part. See also *Notes on de-allocation* below.

Once selected, point at the part to be allocated and click the left button. The de-allocation takes place.

Notes on de-allocation

Once the parts have been placed on the artwork and tracks routed, it is generally unwise to de-allocate those parts on the schematic - unless they will be re-allocated with the same part reference/pin numbers before a parts/wiring list is re-extracted, or you have a good reason for doing so and understand what will happen as a result.

Once parts are placed on the artwork design, the parts list contains the position/orientation of all those parts. Changing part reference numbers on the schematic, then extracting a parts/wiring list has the effect of moving the parts and unrouting on the artwork, whilst the tracking, silk-screen, etc. remain in situ – effectively causing chaos to the artwork.

If a part is removed from the sheet and another part is inserted with the same reference designator and different connections, the new part's datum will appear in the same position on the artwork as the old part's datum, along with its new connections. The old part's tracks and silk screen data will remain on the artwork.

If modifications to the design have resulted in gaps in a numbering sequence that you would prefer to be used, always use the part renumbering tools in the artwork editor to renumber them. The renumbering will be back-annotated to the parts/wiring list and schematic automatically.

Nonelec commands

These commands are associated with non-electrical data. Non-electrical data is used to define drawing blanks, notes, etc. It is plotted, but Ranger otherwise ignores it.

There are two categories of non-electrical data that can be added and modified, *Drawing sheet* and *Annotation*. Each has a specific line width assigned to it which is controlled from the *Edit > Preferences, Line Widths* window.

Drawing sheet typically used to define the perimeter of the sheet and title blocks, etc. When a design sheet is created, a border made from non-electrical *drawing sheet* data is automatically added.

Annotate typically used to add general notes, tables etc.

The non-electrical commands operate on drawing sheet data and annotation data, irrespective of which mode is currently active. The only way drawing sheet data can be differentiated from annotation data by the user, is by the line widths assigned. (If both types are set to the same size, then a difference won't be visible.)

Version control also operates on non-electrical text strings, so that multiple versions of a text string may be added. The actual string used is dependent on the version setting in the *Edit > Preferences, Version Control*

window. Refer to the *Nonelect > Add Text* command for more details.

To add drawing sheet data, select *NonElec > Drawing Sheet Mode*. To add annotation data select *NonElec > Annotate Mode*.

If the sheet size is changed a prompt appears to allow the existing border to be retained, or a new auto-generated border to be defined.

NonElec > Add Line

Used to add lines that have no electrical significance, for example when creating drawing sheets, adding tables of information, etc.

Once selected, locate the cursor where the line is to start, then click the left-hand mouse key. Move the cursor, a line appears attached to it. Clicking the left button inserts corners in the line. Release the line with a click of the right button, the segment attached to the cursor is not added.

Refer to the *Grid* command to find out whether grid snapping is active.

NonElec > Corner

Used to insert or move a corner in a non-electrical line or arc.

Once selected:

Adding a corner Point at the line/arc, then click the **right-hand** mouse button. Move the cursor, the new corner is attached to it and the adjacent segments stretch with it. Release the new corner with another click of the same mouse button. (Click the opposite mouse button (left) to cancel the new corner.)

Moving a corner Point at the corner, then click the **left-hand** mouse button. Move the cursor to move the corner and stretch the adjacent segments. Release the corner with another click of the same mouse button. (Click the opposite mouse button (right) to cancel the move.)

add corner = **right** button

move corner = **left** button

(Opposite button cancels.)

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Move Point

Used to move existing points (corners) in lines or arcs, to move circles, or to change the size of circles.

Moving points in lines or arcs:

Once selected, locate the cursor on the corner to be moved, then click the left button. Move the cursor, the corner appears attached to it and the existing lines or curves stretch. Position the corner as required, then click the left button to release it. Clicking the right button before the corner is released, returns the corner to its original position.

Moving circles:

Once selected, point at the *centre* of the circle, then click the left button. Move the cursor, the circle appears attached to it. Position the circle as required, then click the left button to release it. Clicking the right button before the circle is released, returns it to its original position.

Changing the size of circles:

Once selected, point at the *circumference* of the circle, then click the left button. As the cursor is moved, the diameter of the circle changes. Once the required circle size is achieved, click the mouse button to release it. Clicking the right button before the circle is released, returns it to its original size.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Delete point

Used to delete points (corners) from lines or arcs.

Once selected, point at the corner to be deleted, then click the left-hand mouse button. The corner is removed from the line or arc.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Delete Line

Used to delete non electrical lines, arcs or circles.

Note: the complete line or curve between start and end points is deleted, not just a segment within a line or arc.

Once selected, point at the line, arc or circle to be deleted, then click the left-hand mouse button. The line, arc or circle is removed.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to

an item to select it.

NonElec > Add Text

Used to add text strings to the sheet that have no electrical significance, for example when creating drawing sheets, adding tables of information, etc.

(Use *Symbol > Attributes* to add part attributes, *Alloc* commands to define part references and pin numbers or *Wires > Attributes* to add signal names.)

Refer to the *Grid* command menu to find out whether grid snapping is active.

Special Syntax

Ranger has special text strings that it recognises and will replace them with specific names on the screen and print-outs.

- %jobname%** this text string will be replaced by the job's name. So for instance, the non-electrical text string **This job is called %jobname%**. will appear as **This job is called A12345**. if the job were called A12345. This ensures the string remains up to date if the job name is changed.
- %blockid%** this text string will be replaced by the block's name. Useful where blocks are used more than once in a design and need to be identified on print-outs. For instance if a block called **Channel** was used more than once in a hierarchy, then **This is %blockid%**. would be replaced with **This is Channel.1.**, **This is Channel.2.**, **This is Channel.3.**, etc.
- %version%** this text string will be replaced by the current version. So for instance if one sheet were used in USA and UK versions, then the string **This is the %version% version.** would be replaced with **This is the USA version.** or **This is the UK version.** depending on which was selected in the *Edit > Preferences, Version Control* window.

Versions

More than one version can be assigned to the text value field. For instance, it may be required to add a note in English or French, depending on who will be reading the schematic. The version names, and which version is currently active are defined in the *Edit > Preferences, Version Control* window. The text string should take the following format when specifying multiple values:

versionname!text_string versionname!text_string versionname!text_string

where:

- versionname** is the version name assigned in the *Edit > Preferences, Version Control* window.
- !** is the character defined for version control in the *Edit > Preferences, Miscellaneous* window.
- text_string** is the text string required for that version.

Example:

UK!Static Sensitive FRA!French Equivalent

This will give a text string of *Static Sensitive* when displayed or plotted with the UK version selected and *French Equivalent* with the FRA version selected.

It is possible to specify a default version value using the version name *.

So for example, *FRA!French Equivalent *!Static Sensitive* will produce *French Equivalent* if the FRA version was selected, but *Static Sensitive* for all other versions.

If the version control character is not used, then those strings are unaffected by the version selection mechanism.

If a text string has no value for a selected version, i.e. it has no *! value then (null) will be displayed on screen where the value would normally be. The (null) text will not appear when the sheet is plotted.

The version control character may be used in most part attributes (outline name, ordercode, value, etc.) in all user-defined attributes, in connection attributes (signal name) and in non-electrical text strings.

Once *NonElec > Add Text* has been selected a window appears.

- Text:** String to be entered. If text has already been added or modified, the previous text string appears in the window. It may be selected and modified, or a new text string entered.
- Height:** The height of the text string can be entered (in inches or millimetres depending on which units are active (*Edit > Units* command)).
- Angle:** The orientation of the text can be specified in increments of one degree, anti-clockwise. 0 degrees places text horizontally, reading from left to right. Angles can be selected (0, 90, 180 or 270) or the value selected and a new value typed which will replace the selected value.
- Mirrored:** the string can be mirrored if required.

Once all the information has been entered, select *OK*. The window closes and the text string represented by a rectangle, appears attached to the end of the cursor. Move the cursor into position and click the left button to release the text.

Clicking the right button before the text is released restores the window allowing the string to be modified or the operation cancelled.

Whilst the text is being moved it can be rotated in 90-degree increments anti-clockwise by pressing the *<rotate part>* special function key.

As the text string is released, another copy of it appears on the end of the cursor. It can be released in the same way or the right button clicked to cancel the text. The window re-appears, select *Cancel* to cancel further text addition, or the text in the window can be modified and *OK* selected again.

NonElec > Move Text

Used to move text strings that have already been added.

Once selected, point at the text string and click the left button. A rectangle representing the text string appears attached to the cursor. Move the rectangle into the required position and click the left button to release it.

Whilst the text is being moved it can be rotated in 90 degree increments anti-clockwise by pressing the *<rotate part>* special function key.

Clicking the right button whilst moving the text releases it in its original position.

Text can be selected anywhere along its length. Its datum is the lower left-hand corner of the rectangle that is displayed when it is moved.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Delete Text

Used to delete existing text strings.

Once selected, point at the text string and click the left-hand mouse button. The text is removed.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Edit Text

Used to edit existing text strings, change their height, angle of rotation or mirrored status.

Once selected, point at the text string and click the left-hand mouse button. A window appears allowing the text string to be modified.

Refer to the *NonElec > Add Text* command above for details on this window if required.

Select *OK* to implement the changes. Select *Cancel* to restore the text to its original form.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Rotate Text

Used to rotate text strings through 90 degree increments, anti-clockwise. For instance, text that has been rotated to 45 degrees will rotate to 135, 225, 315 and back to 45 degrees. Text can be rotated in increments of 1 degree using the *NonElec > Add/Edit Text* command.

Text is rotated about its datum, which is the lower left-hand corner of the rectangle that is displayed when text is moved.

Once selected, point at the text string and click the left button. The text rotates. If the text is selected on its datum, you don't have to move the cursor to rotate it again.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Add Arc

Used to add curved lines (arcs) that have no electrical significance, for example when creating drawing sheets, adding tables of information, etc. A series of arcs and straight lines can be added using this command.

Once selected, locate the cursor where the arc or line is to start, then click the left button. Move the cursor, a line appears attached to it. Move the cursor to the point where the arc or line is to end, then click the left button.

Move the cursor. A curve stretches with the cursor, its size and shape being dependent on the cursor position. Click the left button to release the curve. If the right button is clicked, a straight line segment is introduced.

The line remains attached to the cursor allowing more curves to be added, or the line released. To continue adding curves, move the line to the end point of the next arc, then click the left button, stretch the arc and release it with a click of the same button. To release the line, click the right button, the line attached to the cursor is not added.

If the right button is clicked whilst the arc is being stretched, a straight segment is introduced. This allows a mixture of curved and straight segments to be drawn using the same command.

The shape of the arcs can be adjusted using the *NonElec > Move Point* command.

Refer to the *Grid* command to find out whether grid snapping is active.

NonElec > Adjust Arc

Used to convert arcs to lines, lines to arcs and to adjust the size and shape of arcs.

Select NonElec > Adjust Arc.

Straight lines into arcs:

Point at a line segment and click the left button. Move the cursor, the segment is replaced by an arc that stretches as the cursor is moved. Click the left button to release the arc.

Arcs into a straight line:

Point at an arc and click the left button. Follow this with a click of the right button to convert the arc into a straight line segment.

Adjusting the size or shape of an arc:

Point at the arc and click the left button. Move the cursor until the arc is in the required position, then click the left button.

Clicking the right button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically, and can be used to convert 45 degree lines into quadrants of a circle.

The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

NonElec > Add Circle

Used to add circles that have no electrical significance, for example when creating drawing sheets, adding tables of information, etc.

Once selected, point at the centre of the required circle and click the left button. Move the cursor and a circle appears attached to it. The size of the circle is controlled by the cursor position. Click the left button to release the circle in its current position. Clicking the right button cancels the circle.

Use the *NonElec > Move Point* command to move existing circles, or change the size of existing circles.

Refer to the *Grid* command to find out whether grid snapping is active.

NonElec > Annotate Mode or Drawing sheet mode

Used to define which category non-electrical data, is added to. One or the other mode is ticked and therefore active.

Annotation Mode should be selected when adding general notes, tables, etc.

Drawing Sheet Mode should be selected when adding the drawing sheet borders and associated text.

The *Edit > Preferences, Line Widths* window controls the line sizes of annotation and drawing sheet lines and text.

To create user-defined borders for use on other sheets:

The supplied master schematic library contains a folder called *Sheets*. This has various sizes of design sheets with borders. These could be modified to suit your own requirements. Alternatively, they can be copied and the copies modified.

The *Copy/Paste/Rename* commands can be used to copy the sheets to the same library folder or to another design. Simply locate the design sheet to be copied, then right-click and *Copy* it. Navigate to the design sheets folder where it should be copied to, then right-click over the *Design Sheets* folder and *Paste*.

Open the design sheet and delete/edit the existing border and define the new border as required, in *Drawing Sheet Mode*.

Repeat for each different sheet size required.

Using the border on a new sheet:

If starting a new design sheet, use the right-click *Copy/Paste* commands to copy the pre-defined design sheet into the design being worked upon. Once copied, rename the sheet to something suitable, open the sheet and add parts, etc. to it.

Example:

From the navigator, navigate down through the Masters folders until the master schematic library, Design Sheets folder is open, then locate and open the folder *Sheets*. Right-click over the design sheet required and select *Copy*.

Now navigate to the Design Sheets folder of the design being worked upon, then with the cursor over the Design Sheets folder, right-click and select *Paste*. The design sheet will appear in the folder. Right-click/*Rename* the design sheet, and change its name to something suitable. Open the sheet and add the circuitry.

Using the border on an existing sheet:

If a design sheet has already been started and contains circuitry, first remove the border that's not required

using the NonElec commands.

Copy (as described in the example above) the design sheet with the border, from the master library to the design being worked upon.

Open that design sheet, then use the *Region > Copy* command to copy the border. Open the sheet being worked upon, then use *Region/Paste* to paste the border onto the sheet.

Region commands

These commands are used to move, copy and delete rectangular areas of circuitry, within a design - not between designs. (To copy areas of circuitry between designs, first copy the design sheet to the design sheets folder, then use the Region commands. Delete the copied sheet when no longer required in the design.)

Refer to the *Grid* menu command to find out whether grid snapping is active, and the *Edit > Preferences, Miscellaneous* to find out whether wire pick-up, etc. is enabled and to define the amount of data that can be moved dynamically.

Region > Move

Used to move a rectangular area of circuitry around the sheet.

Once selected, point at and select with a click of the left button, one corner of the area to be moved. As the cursor is moved, a rectangle stretches with it. Click the same mouse button again to release the diagonally opposite corner of the area. Once the area has been defined, it will move with the cursor until it is released with a click of the left button. Click the right-hand mouse button to cancel the move.

If the rectangle is blank when being moved, increase the value of the *Max window mode, dynamic nodes* setting in the *Edit > Preferences, Miscellaneous* window.

Region > Copy

Used to copy a rectangular area of circuitry to the *paste buffer*.

The contents of the paste buffer can be added to other design sheets in the same design, using the *Region > Paste* command (but not other designs).

When data is copied or cut to the design sheet paste buffer, it overwrites anything previously added to the paste buffer.

Each job has its own design sheet paste buffer, which remains intact until the job is closed, at which time it is cleared.

Part allocation information is not maintained by the contents of the paste buffer.

Once selected, point at and select with a click of the left-hand mouse button, one corner of the area to be copied. As the cursor is moved, a rectangle stretches with it. Click the same mouse button again to release the diagonally opposite corner of the area. The rectangle disappears and the content of the area has been copied to the paste buffer.

A message appears along the bottom of the screen indicating how many *nodes* were selected in the area - knowing the number of nodes in the area allows the *Max window mode, dynamic nodes* setting in the *Edit > Preferences, Miscellaneous* window to be increased for dynamic movement if required (see *Region > Move*).

Use the *Region > Paste* command to add the contents of the paste buffer to the sheet.

Region > Cut

Used to delete a rectangular area of circuitry. The area of circuitry is placed in the *paste buffer*, so it could be restored using the *Region > Paste* command.

The contents of the paste buffer can be added to other design sheets in the same design, using the *Region > Paste* command (but not other designs).

When data is copied or cut to the design sheet paste buffer, it overwrites anything previously added to the paste buffer.

Each job has its own design sheet paste buffer, which remains intact until the job is closed, at which time it is cleared.

Part allocation information is not maintained by the contents of the paste buffer.

Once selected, point at and select with a click of the left-hand mouse button, one corner of the area to be deleted. As the cursor is moved, a rectangle stretches with it. Click the same mouse button again to release the diagonally opposite corner of the area.

The area of circuitry is removed from the sheet and added to the paste buffer.

A message appears along the bottom of the screen indicating how many *nodes* were selected in the area - knowing the number of nodes in the area allows the *Max window mode, dynamic nodes* setting in the *Edit > Preferences, Miscellaneous* window to be increased for dynamic movement if required (see *Region > Move*).

Use the *Region > Paste* command to add the contents of the paste buffer to a sheet if required.

Region > Paste

Used to add the contents of the paste buffer to the sheet. (Use the *Region > Copy* or *Cut* commands to add

data to the paste buffer.)

If the circuitry in the paste buffer includes any block I/O pins, then the appropriate number of block I/O pins are added to the corresponding symbol when the circuitry is pasted into the sheet. If a symbol had not previously existed, then it will be created.

When data is copied or cut to the design sheet paste buffer, it overwrites anything previously added to the paste buffer.

Each job has its own design sheet paste buffer, which remains intact until the job is closed, at which time it is cleared.

Part allocation information is not maintained by the contents of the paste buffer.

Assuming data has been added to the paste buffer select *Region > Paste*. As the cursor is moved, the area of circuitry from the paste buffer is seen attached to it.

An empty rectangle will be seen if there are too many nodes within it, according to the *Max window mode, dynamic nodes* setting in the *Edit > Preferences, Miscellaneous* window. Click the left button to release the circuitry, or the right button to cancel the operation.

Region > Add to Artwork Tray

All parts associated with symbols in the selected region will be added to the artwork placement tray, whether or not the artwork editor is open.

In this mode, all symbols related to parts already placed on the board, or already in the placement tray will be shown with a dotted outline.

The region can include parts already shown dotted, their status is unaffected by the selection.

Select *Region > Add to artwork tray*. Any parts already placed on the artwork or in the placement tray will be shown with dotted outlines. Point at and select with a click of the left button, one corner of the area to be selected. Move the cursor, a rectangle stretches with it, position the diagonally opposite corner of the area and click the same mouse button again. The parts will be added to the placement tray and appear as dotted symbols to indicate their status (whilst the command is active).

A message in the status bar will indicate how many parts were added to the placement tray.

Note: once added to the placement tray the parts can only be removed using the artwork editor tools.

Commands available when a part, split part, primitive or block I/O is open

Tool bar commands

The toolbar contains icons that can be used as short-cuts to some commands. They are not user-definable. The icons/settings vary, depending upon which editor is open. The following settings may also appear.

Instance setting

Used to display each element of a multi-element part. Use the slider bar and/or arrows to the right of the setting to view each element, only the pin numbers between elements will be different.

If **1 of 1** is shown, then the arrows alongside will be greyed out as the part has only one element.

This setting is not present when split parts are edited.



Figure 100

Type setting

Used to select whether the *Normal* or *DeMorgan* view of the part is active.

A part can have two representations drawn for it. Its *Normal* symbol or a *DeMorgan* equivalent symbol. Everything remains the same about the part, apart from its graphical representation.

The pins and attributes on both representations are in exactly the same position. If they are moved or deleted from one representation, they are also moved or deleted from the other.

If the DeMorgan equivalent symbol is not required, then it could be used to draw the symbols to another standard. For instance the normal symbols could be drawn to the British Standard and the DeMorgan symbol to the ANSI standard.

A part is initially drawn and used on the schematic sheet showing its *normal* graphic representation. Once on the sheet it can be toggled between the representations, using the *Symbol > Logic Type* command.

Before an alternate representation of the part can be made, its original version must first be defined. When the

Type is set to *DeMorgan*, if the DeMorgan symbol has not previously been defined, all the graphics of the part will disappear, leaving behind the part attributes and pins. Use the *Outline* menu commands to draw the alternate graphic shape of the part.

To return to the original version of the part, set the *Type* to *Normal*.

I/O Pin type



Figure 101

This setting is only available when a block I/O port or a design symbol is being edited. It determines the type of pin that is added. One of two types of pin can be added, a *net-pin* or a *bus-pin*.

Net-pins are displayed as a diagonal cross, bus-pins with a circle.

If a net-pin is added only individual connections can be attached to it when it is used.

If a bus-pin is added, only bus connections can be attached to it when it is used.

Outline commands

These commands are associated with drawing the schematic shape of the part/symbol, so they appear when editing design symbols, parts, primitives and block i/o ports. They do not appear when editing a design sheet or a split part. The thickness of lines is controlled from the *Edit > Preferences, Line widths* window.

Outline > Add Line

Used to add lines to define the shape of the symbol.

Once selected, locate the cursor where the line is to start, then click the left button. Move the cursor, a line appears attached to it. Clicking the left button inserts corners in the line. Release the line with a click of the right button, the segment attached to the cursor is not added.

Refer to the *Grid* command to find out whether grid snapping is active.

Outline > Add Circle

Used to add circles to the symbol.

Once selected, point at the centre of the required circle and click the left button. Move the cursor and a circle appears attached to it. The size of the circle is controlled by the cursor position. Click the left button to release the circle in its current position. Clicking the right button cancels the circle.

Use the *Outline > Move Point* command to move existing circles, or change the size of existing circles.

Refer to the *Grid* command to find out whether grid snapping is active.

Outline > Add Rectangle

Used to add rectangular shapes to the symbol.

Once a rectangle has been added, it is treated as a line so it can be modified using the *Outline > Corner/Delete Point*, etc., commands.

Once selected, locate the cursor in the **centre** of the required rectangle, then click the left button. Move the cursor, a rectangle stretches as the cursor is moved; its size and shape is cursor dependent. Click the left button to release the rectangle in its current position. Clicking the right button before the rectangle has been released, cancels the rectangle, the *Add Rectangle* command stays active.

Refer to the *Grid* command to find out whether grid snapping is active.

Outline > Add Arc

Used to add curved lines "arcs" to the symbol. A series of arcs and straight segments can be added using this command.

Once selected, locate the cursor where the arc or line is to start, then click the left button. Move the cursor, a line appears attached to it. Move the cursor to the point where the arc or line is to end, then click the left button. Move the cursor. A curve stretches with the cursor, its size and shape being dependent on the cursor position. Click the left button to release the curve. If the right button is clicked a straight line segment is introduced.

The line remains attached to the cursor, allowing more curves to be added or the line released. To continue adding curves, move the line to the end point of the next arc/line, then click the left button, stretch the arc and release it with a click of the same button. To release the line, click the right button, the segment attached to the cursor is not added.

If the right button is clicked whilst the arc is being stretched, a straight segment is introduced. This allows a mixture of curved and straight segments to be drawn using the same command.

The shape of the arcs can be adjusted using the *Outline > Move Point* command.

Refer to the *Grid* command to find out whether grid snapping is active.

Outline > Corner

Used to move, or insert a corner in a line or arc. This command is more flexible than the *Outline > Move Point* command, as it allows corners to be moved or added. Once selected:

Adding a corner

Point at the line, then click the **right-hand** mouse button. Move the cursor, the new corner is attached to it and the adjacent segments stretch with it. Release the new corner with another click of the same mouse button. (Click the opposite mouse button (left) to cancel the new corner.)

Moving an existing corner

Point at the corner, then click the **left-hand** mouse button. Move the cursor to move the corner and stretch the adjacent segments. Release the corner with another click of the same mouse button. (Click the opposite mouse button (right) to cancel the move.)

add corner = **right** button (Opposite button cancels.)

move corner = **left** button (Opposite button cancels.)

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Outline > Move Point

Primarily used to move, or change the size of circles. Can also be used to move points (corners) in lines or arcs. Once selected:

Moving circles:

Point at the **centre** of the circle, then click the left button. Move the cursor, the circle appears attached to it. Position the circle as required, then click the left button to release it. Clicking the right button before the circle is released returns it to its original position.

Changing the size of circles:

Point at the **circumference** of the circle, then click the left button. As the cursor is moved the diameter of the circle changes. Once the required circle size is achieved, click the left button to release it. Clicking the right button before the circle is released returns it to its original size.

Moving points in lines or arcs:

Locate the cursor on the corner to be moved, then click the left button. Move the cursor, the corner appears attached to it and the existing segments or curves stretch. Position the corner as required, then click the left button to release it. Clicking the right button before the corner is released, returns the corner to its original position.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Outline > Adjust Arc

Used to convert arcs to lines, lines to arcs and to adjust the size or shape of arcs.

Once selected:

Straight lines into arcs:

Point at a line segment and click the left button. Move the cursor, the segment is replaced by an arc that stretches as the cursor is moved. Click the left button to release the arc.

Arcs into a straight line:

Point at an arc and click the left button. Follow this with a click of the right button to convert the arc into a straight line segment.

Adjusting the size or shape of an arc:

Point at the arc and click the left button. Move the cursor until the arc is in the required position, then click the left button.

Clicking the right button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically, and can be used to convert 45 degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Outline > Delete Feature

Used to delete lines, circles or arcs from the symbol.

Note: the complete line or curve between start and end points is deleted, not just a segment within a line or arc.

Once selected, point at the line, arc or circle to be deleted, then click the left button. The line, arc or circle is

removed.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Outline > Delete Point

Used to delete points (corners) from lines or arcs.

Once selected, point at the corner to be deleted, then click the left button. The corner is removed from the line or arc. If there are only two points in the line/arc (start & end points), then the line is deleted when one of the points is removed.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Outline > Symbol Datum

Used to define the datum of the part.

The datum point is the reference point used when selecting items.

Once selected, point at the position required for the datum point and click the left button. The cross that represents the datum, moves.

Refer to the *Grid* command to find out whether grid snapping is active.

Terminals commands

These commands are used to add or edit the terminals or "pins" of a part. They appear when editing parts, split parts, primitives, block i/o ports and design symbols. The commands do not appear when editing design sheets.

Terminals which are often referred to as pins ("terminals" and "pins" are used interchangeably) sometimes do not necessarily represent physical component pins, it depends on the type of schematic item the terminals are used in, as described below.

Design symbols

Terminals within the design symbol do not represent pins of a physical component. They are a means of making a connection between sheets in a hierarchical schematic. For every terminal used in the design symbol, a port is added to the associated design sheet. The terminals and ports take the same name.

Note: If the design symbol is placed in a higher level design sheet, and the pin name (terminal) attribute is edited, the new name will appear against the block i/o pin when descending into the design sheet associated with the symbol.

An iopin attribute that is overridden by an attribute on a higher level instantiation will always show the overridden value when the design sheet is displayed. The value may still be edited in this mode, but the override value will still be displayed.

Switching to the design *symbol* will show the attribute value without the effect of any higher level override. The value displayed in this mode is the value displayed against the pin for any new instantiations of the design symbol.

Parts

Terminals within a library part always represent a pin of a physical component. To ensure connectivity between the schematic and artwork, the terminal must be given a number (combinations of alpha or numeric characters), which corresponds to a pad with the same number in the outline library.

Primitives

Terminals within a library primitive represent a pin of a physical component **if** the primitive is used within a split part (or part, in the case of power symbols). The terminals should not be given numbers until the primitive is used in the split part/part.

Split parts

Terminals cannot be added to split parts, they are added to the primitives used in the split part. When the primitive is used in a split part, its terminals do represent component pins and they should be assigned pin numbers in the split part.

Block I/O ports

Terminals within a block I/O port do not represent pins of a physical component. A block I/O port is used as a means of making a connection between sheets in a hierarchical schematic. There are two types of block i/o port terminal, a net-pin and a bus-pin. When block i/o ports are used on design sheets, only signal connections can be attached to net-pins and bus connections to bus-pins.

Terminals > Add, Terminals > Add-Auto

Both these commands are used to add terminals to the symbol. The *Add* command is used to add individual terminals, whilst *Add-Auto* is used to add rows or columns of terminals. Default attributes are assigned automatically when the terminals are added. These defaults are stored in the file called *schema_attribs.txt*, which is held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL

Designer/Data by default).

IO Pin type

If a **block I/O port** or a **design symbol** is being edited, ensure the **IO Pin Type** setting in the toolbar is set to the appropriate type as one of two types of pin can be added, a *bus pin* or a *net pin*.

If a bus pin is added, only bus connections can be attached to it.

If a net pin is added only individual connections can be attached to it.

Two ports are therefore required for hierarchical schematics, one containing a bus pin, and the other a net pin.

If a bus pin port is used on the design sheet, a bus pin is automatically added to its associated symbol.

If a net pin port is used on the design sheet, a net pin is automatically added to its associated symbol.

Net pins are displayed as a diagonal cross, bus pins with a circle.

Refer to the *Grid* command to find out whether grid snapping is active.

Add

Once selected, point at the location where the terminal is required, then click the left button. The terminal appears. More terminals can be added as required.

Add-Auto

Once selected, move the cursor and a flag containing the word **first** appears attached to it. Point at the location where the first terminal is required, then click the left button. The flag is replaced by another containing the word **second**. Point at the location where the second pin is required, then click the left button. A window appears requesting the **total** number of pins required. Type in the number followed by <enter>. The total number of terminals specified is added, at the same pitch and following on in the same direction as the first two terminals.

The **nn** string indicates the position that the terminal's pin number will occupy when it is assigned.

The terminals **must** be given pin numbers (Terminals > Assign Pin Numbers) if they are used within a part or split part. The pin numbers can be moved or hidden using the *Attributes* commands.

The **Name** or **NET** string indicates the position that the terminal's name will occupy (providing it is visible). These attributes can be moved or hidden using the *Attributes* commands.

Terminals > Move

Used to move existing terminals. When terminals are moved, their associated attributes also move.

Note: if pins are moved on a part that has already been placed and connected to on the schematic sheet, the connectivity will be lost as the connections will not move.

Once selected, point at the terminal to be moved, then click the left button, the terminal disappears. Move the cursor to the location where the terminal is required and click the left button again. The terminal re-appears in its new position.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Terminals > Delete

Used to delete terminals. The terminal and all its associated attributes are deleted.

Note: if terminals are deleted from a part that has already been placed and connected to on the schematic sheet, the connections will remain on the sheet.

Once selected, point at the terminal to be deleted, then click the left button, the terminal and its attributes are deleted.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Terminals > Assign Pin Numbers

Used to assign a number to a terminal/pin. The number assigned is linked to the pad with the same number in the associated physical outline.

Up to 8 alpha-numeric characters (letters or numbers) in any order can be used in a pin number – even if letters are used, it is still a pin “number”.

Note: although it is impossible to assign the same pin number more than once with this command, it can be achieved (via an alternative method) but would not be valid. It is therefore advisable to use the *Terminals > Check* command before using the part, to ensure pin numbers have not been duplicated.

Once a number has been assigned, it cannot be used again in that part.

If pin numbers have been assigned incorrectly, they can be changed, by assigning another number to the terminal - the original number is over-written.

However, because the same number cannot be used twice, even temporarily, you might need to temporarily assign numbers that haven't/won't be used to those pins (say 50, 51, etc.), before the correct numbers can be

assigned (this saves deleting the terminal and adding a new one).

If pin numbers are being changed and the parts have already been used and allocated on the schematic, they will need to be de-allocated, then re-allocated on the schematic to implement the pin number changes.

Pin numbers **must** be assigned to pins if the pins are used in a part or split part, but they do not have to be displayed. The *Attributes > Pin* command controls pin number visibility.

Pin numbers should not be assigned to pins within library primitives; the pin numbers are assigned to the pins when they are used within the part or split part.

There is no requirement to assign pin numbers to the pins within block I/O ports or design symbols, as these pins do not represent physical pins.

If a part with more than one element/gate is being edited, each of the gates can be displayed in turn by selecting the arrows alongside the *Instance* setting in the toolbar. This allows the pin numbers from each gate to be assigned.

Select *Terminals > Assign Pin Numbers*, a window appears, similar to Figure 102.

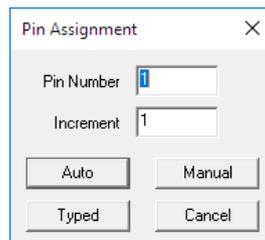


Figure 102

Three methods of numbering terminals are available, **Auto(matic)**, **Manual** and **Typed**.

When using the *Auto* or *Manual* methods, the **pin number** and **increment** values have to be supplied before the options become available.

Pin number enter the first number to be assigned to a terminal/pin. Up to 8 alpha-numeric characters in any order can be entered.

Increment the number entered here is added to the previous assigned pin number and used for the next selected pin.

For instance, if the *pin number* is set to 1 and the *increment* set to 3, pin numbers will be assigned in the following order, 1, 4, 7, 10, etc.

If the *pin number* specified does not end with a number, subsequent pin numbers have a number added, according to the *Increment* specified.

Use the **Typed** method if the last character should not be a number.

Auto

Used to automatically assign pin numbers in order, to a row or column of terminals/pins.

Enter the first *Pin Number* and *Increment* as explained above, then select **Auto**. The window closes and a flag with the word **First** appears attached to the cursor. Select, with a click of the left button the first pin in the row or column to be assigned.

The flag changes to **Last**. Select the last pin in the row or column with a click of the same mouse button. The pin numbers are assigned to each of the pins in the row or column in order, as specified.

Manual

Used to manually assign pin numbers in order to individual selected terminals/pins.

Enter the first *Pin Number* and *Increment* as explained above, then select **Manual**. The window closes and a flag with the *pin number* appears attached to the cursor. Select, with a click of the left button the pin that should be assigned that pin number.

The pin number in the flag changes according to the *Increment* specified. Select the next pin with the same mouse button, and so on. Click the right button to cancel further pin selections. The window re-appears, either select **Cancel** or continue numbering pins as before.

Typed

Used to assign pin numbers to individual selected pins. This method is typically used when the last character of a pin number is a letter, or the numbers do not follow a pattern.

Leave the *Pin number* and *Increment* boxes as they are, and select **Typed**. Select, with a click of the left button the terminal/pin to be assigned a number.

A window appears requesting the pin number. As soon as <enter> is pressed, the pin number is assigned. Another pin can be selected and a number assigned in the same way. Up to 8 alpha-numeric characters, in any combination can be used as a pin number.

Terminals > Set Pin Swap Rule

Used to define electrically equivalent pins in order to allow pin swapping on the board layout.

Pin swapping is performed on the artwork layout in order to reduce connection crossovers and thus aid routing. For instance, the connections attached to the input pins of one nand gate can be swapped with one another without affecting the electrical function of the circuit (pin swapping). If pin swapping is performed in the artwork editor, the pin number changes are back-annotated to the wiring list and schematic.

Only pins within the same symbol that are defined as equivalent to one another can be swapped with one another.

Ranger uses colours to display which pins are equivalent to one another. Pins assigned the same colour are equivalent to other pins assigned the same colour within the same symbol.

Up to six different sets of equivalent pins can be defined within one symbol. For instance if a part had 6 pins and three pins were assigned blue, and three green, then the three blue pins could be swapped one for another, as could the three green pins, but the blue and green pins could not be swapped with one another.

Pins are unswappable by default.

Note: when editing **split parts**, even though it's possible to assign all the pins within the package to the same colour, only the pins within the same primitive are defined as being equivalent to one another.

When selected, the *PinSwap Rule* dialogue bar appears on the left of the screen. Small circles replace the terminals/pins - the colour of the circles indicates which pins are equivalent to one another.

Initially all the pins are clear and **unswappable**, they are not equivalent to any other pin, so cannot be swapped on the layout.

To assign equivalent pins: select one of the colours from the *Pin Swap Rules* dialogue, then select the terminals that are electrically equivalent to one another. The terminal circles will change colour.

To make a pin unswappable if it has been made equivalent to other pins, select **U** from the *Pin Swap Rules* dialogue, then select the terminal.

Terminals > Assign Names

Used to assign a name to a pin – if required. The name does not have any link to the physical outline. It is added for user information to aid recognition of pins and their type, i.e. clock, clear, preset, etc.

The names can consist of alpha-numeric characters in any combination. The negation bar prefix character can be specified in front of the pin name if a negation bar is required over the pin name.

When a pin is added a default pin name of **Name** is added, unless a design symbol or block i/o port is being edited in which case it is **NET**.

(Defaults are defined in the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

The *Attributes > Pin* command controls pin name visibility.

Once selected, select a pin with a click of the left button. A window appears requesting the pin name. As soon as <enter> is pressed, the name is assigned and replaces *Name* or *NET*. Another pin can be selected and a name assigned in the same way.

If a design symbol is being edited, the pin name is linked to the port name on the associated sheet. Changing the pin name automatically changes the port name and vice versa.

Note: If the symbol of a sheet is placed in a higher level sheet, and the pin name (terminal) attribute is edited, the new name will appear against the block i/o pin when descending into the schematic sheet associated with the symbol.

An iopin attribute that is overridden by an attribute on a higher level instantiation will always show the overridden value when the sheet is displayed. The value may still be edited on the sheet, but the override value will still be displayed.

Switching to the symbol of the sheet will show the attribute value without the effect of any higher level override. The value displayed in this mode is the value displayed against the pin for any new instantiations of the symbol.

Over-ridden attributes can be reset to their original values, using the *Symbol > Attributes > Reset* command.

When re-setting the attributes of design symbols in a hierarchy, the attributes should be reset at the upper level.

Terminals > Check

Used to obtain a report on the pins used within the part. If warnings are given they should be investigated and rectified.

A common error is to assign the same pin number to multiple pins. If this is not detected when the part is created, this will lead to short-circuit error messages when the part is used and the artwork is checked. A connection will be listed as being shorted, but it will not indicate another net that it is shorted to. If this type of error occurs, ensure that the *Terminals > Check* command has been used on all the parts used on the schematic.

Powerpins commands

These commands appear when editing parts and split parts, they are associated with the symbols that can be used to represent a part's power pins. They do not appear, or should not be used, when editing primitives, block i/o ports, design symbols or sheets.

Power symbols are used if required, to represent the power pins of parts or split parts. For example, a 7400 is usually represented on the schematic sheet by four, two i/p nand gates and the power connections implied, i.e. automatically added to the wiring list but not seen on the schematic.

It is sometimes useful to see the power pins on the schematic, so power symbols should be used in these instances – see Figure 103.

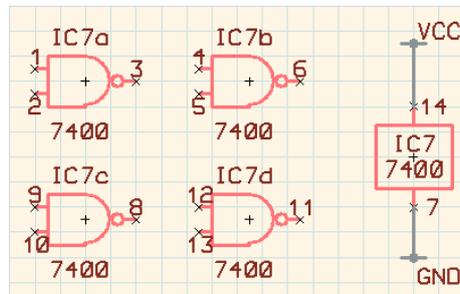


Figure 103

Power symbols can be added to parts/split parts but they do not have to be used on the schematic. If they are not used, the power connections are implied. If they appear on the schematic, connections **must** be made to the pins or they will be unconnected.

Refer to the *Attributes > Symbol, Part Power Pins* window in order to change implied power pin definitions (pin numbers & names).

Power Pins > Add

Used to add power symbols to parts or split parts. The power symbol must already have been created as a library primitive. (The worked example gives an example of creating and using a power symbol.)

Power symbols can also be added via the *Attributes > Symbol, Part Power Pins* window.

When the command is selected, if the part already has a power symbol defined, then another of the same power symbol is added, otherwise a prompt asks which power symbol is required. If the power symbol already used has to be changed, use the *Attributes > Symbol, Part Power Pins* window to change it, or delete the power symbol first and add a new one.

The position of the power symbol relative to the library symbol is not important; the symbols are added to the schematic independently of one another.

Refer to the *Grid* commands to find out whether grid snapping is active.

Select **Power Pins > Add** - a window requests the name of the power symbol required (unless a power symbol has already been defined in the part, in which case that symbol is used). Type in the name of the power symbol followed by <enter>. The power symbol appears attached to the cursor. Move the power symbol and release it with a click of the left button. Another power symbol appears, it can be released in the same way or the right button clicked to cancel it.

The power symbol primitive itself does not include pin numbers or power rail names, just text positions. These are automatically updated from the information held in the power pin definitions in the *Attributes > Symbol, Part Power Pins* window.

They can be modified using the *Terminals > Assign Pin Numbers* or *Terminals > Assign Names* commands, or using the *Attributes > Symbol, Part Power Pins* window.

Power Pins > Move

Used to move power symbols within parts and split parts.

The position of the power symbol relative to the library symbol is not important; the symbols are added to the schematic independently of one another.

The power symbol should be selected by its datum point. (This was defined in the library primitive when the symbol was made and is shown as a diagonal cross.)

Once selected, point at the datum point of the power symbol and select it with a click of the left button. Move the cursor, the power symbol appears attached to it. Position the symbol as required, then click the left button to release it. Clicking the right button before the symbol is released, returns the symbol to its original position.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Power Pins > Rotate

Used to rotate power symbols in 90 degree steps, anti-clockwise within parts and split parts. The power symbol is added to the schematic at the orientation shown in the library part. It can be rotated on the schematic as required.

The position of the power symbol relative to the library symbol is not important; the symbols are added to the schematic independently of one another.

The power symbol should be selected by its datum point.

Once selected, point at the datum point of the power symbol and select it with a click of the left button. The symbol rotates by 90 degrees anti-clockwise each time the symbol is selected.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Power Pins > Flip

Used to flip power symbols within parts and split parts.

Symbols are flipped about their original y-axis, i.e. symbols are turned left to right about their datum unless they have been rotated by 90 or 270 degrees first, in which case they are turned top to bottom about their datum.

The power symbol is added to the schematic at the orientation shown in the library part. It can be flipped on the schematic as required.

The position of the power symbol relative to the library symbol is not important; the symbols are added to the schematic independently of one another.

The power symbol should be selected by its datum point.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the power symbol (not shown) and select it with a click of the left button. The symbol flips each time the symbol is selected.

Power Pins > Delete

Used to delete power symbols from parts and split parts. The *Edit > Preferences, Part Power Pins* window is automatically updated so the parts power pin definitions are removed.

The power symbol should be selected by its datum point. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the power symbol and select it with a click of the left button. The power symbol and its associated attributes are deleted.

Power Pins > Datum

Used to position the datum of the power symbol as it will be used when the symbol is positioned on a schematic sheet.

Refer to the *Grid* command to find out whether grid snapping is active.

Once selected, point at the position where the datum is required and click the left button. The diagonal cross moves to the selected position.

(If a power symbol is moved, rotated, etc. from within a part or split part, it must be selected by its datum as defined in its library primitive.)

Attributes

The following commands are available except when editing a design sheet.

(Attributes of individual parts and connections placed on the schematic sheet are accessed via the *Symbols* and *Wires* commands.)

When selected, the commands associated with the part's attributes appear. The attributes include pin numbers, part references, gate and pin swapping rules, order codes, etc., plus any user defined attributes.

The default position and visibility of these attributes plus any user-defined attributes that are always required can be defined within the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

If parts have already been placed on the schematic and their attributes have been changed using the *Symbol > Attributes* command, those attributes will not be altered by changes made using these commands because changes made on the schematic by the user take precedence.

If parts have already been allocated on the schematic, it may be necessary to de-allocate, then allocate again those parts whose attributes have changed (for example when editing pin numbers).

If a parts/wiring list has already been extracted from the schematic, it may be necessary to re-extract the parts/wiring list to implement the changes made to the part attributes.

Attributes > Symbol

Used to display, and edit if required the attributes associated with the symbol.

Symbol attribute window

The symbol attribute window initially lists the default attributes and their values obtained from the file: *schema_attrbs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

If an asterisk (*) precedes a Name field entry (not the Value field), that attribute is a system attribute and should not be deleted (even though it could be), as it is required by the program elsewhere. i.e. all the entries in Figure 104.

Not all of the attributes have values assigned to them by default.

If an asterisk (*) precedes a value field entry (not the Name field), that entry cannot be changed from the window. i.e. the *Part ident* in Figure 104.

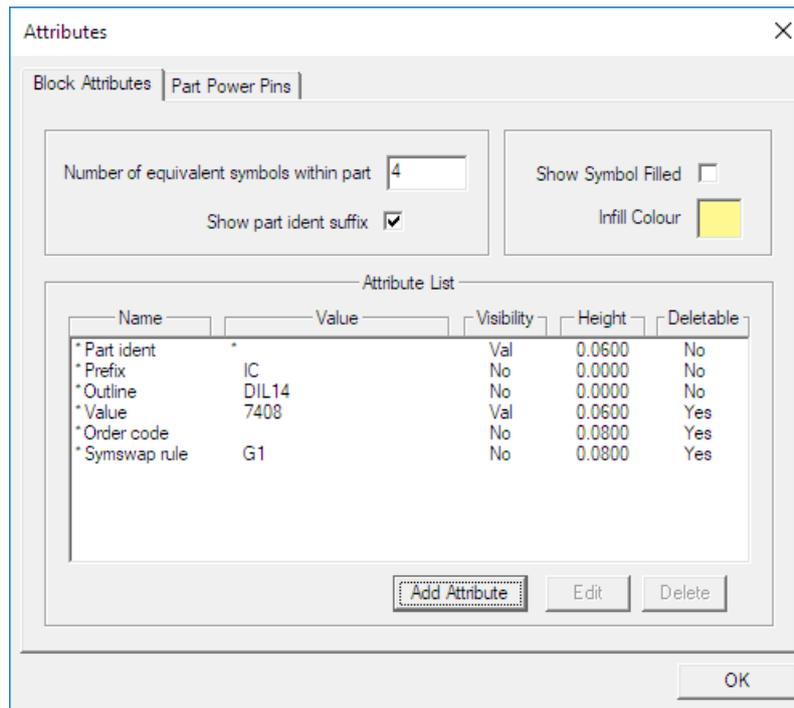


Figure 104

Number of equivalent symbols within part - if a part is being edited, the number of equivalent symbols within the part is defined here and can be edited.

Show part ident suffix - indicates whether a suffix should be added to the part's reference - applicable to parts with more than one symbol (gate) only. For instance IC1a, IC1b, IC1c, etc.

Show Symbol Filled - Symbols can be displayed with a solid infill colour to make them stand out on the sheet by ticking this box.

(If an outline does not appear filled when this mode is activated, edit the symbol outline shape to ensure that it forms a closed path.)

Infill Colour: If the symbol will be shown filled, choose the infill colour by selecting this box, then select the colour required from the colour palette window that appears.

The infill will be displayed and output using this colour, whereas the symbol's outline colour is controlled in the schematic output task configuration window.

Attribute List - lists the attributes and their settings associated with the part. When the part is added to the schematic these values are used but each occurrence of the part on the schematic may be edited individually. If an attribute change is made to the part that has been placed on the schematic, changes to that attribute in the library part will not be reflected in the part already placed on the schematic.

Attribute Columns:

- Name** contains the reference name of the attribute. If a * precedes the name, this is a system attribute and used elsewhere in the program – removing it (if permitted) would have implications for other tools, such as the extracted parts list, gate swapping in artwork, etc.
- Value** contains the text string or value assigned to the attribute. The * in this field means it cannot be edited from this window – the values for part ident's are controlled by the Allocation commands.

More than one version can be assigned to a value field. For instance, it may be required that a resistor has a value of 220k in UK designs, but 100k in USA designs. Alternatively different values may need to be assigned within each instance of a repeated sheet.

The version names, and which version is currently active are defined in the *Edit > Preferences, Version Control* window. The *Structure > Set Version Override* command is used to assign different values in repeated sheets.

The value should take the following format when specifying multiple values:

versionname!attribute_value versionname!attribute_value versionname!attribute_value

where:

versionname is the version name assigned in the *Edit > Preferences, Version Control* window.

! is the character defined for version control in the *Edit > Preferences, Miscellaneous*, window.

attribute_value is the value required for that version.

Examples:

UK!220k USA!100k

This will give a value of 220k when displayed, plotted or compiled with the UK version selected and 100k with the USA version selected.

Low!1k Mid!10k Hi!100k

This will give a value of 1k when displayed, plotted or compiled with the Low version selected, 10k with the Mid version selected and 100k with the Hi version selected.

It is possible to specify a default version value using the version name *. So for example:

UK!220k USA!100k *!500k

will produce 220k if the UK version was selected, 100k if the USA version was selected and 500k for all other versions.

If the version control character is not used, then those attributes are unaffected by the version selection mechanism.

If a normally visible attribute has no value for a selected version, i.e. it has no *! value then (null) will be displayed on screen where the value would normally be. The (null) string will not appear when the sheet is plotted.

The version control character may be used in most part attributes (outline name, ordercode, value, etc.) in all user-defined attributes, in connection attributes (signal name) and in non-electrical text strings.

Visibility this field controls whether the attribute is visible or not when the part is used on the sheet. Options are:

Not visible the attribute is not visible.

Value only the attribute's value field is visible

Name +Value the attribute's name and value field is visible

Height controls the height of the attribute when it is visible.

Deletable indicates whether the attribute is permitted to be removed from the part – this setting cannot be changed.

Note: if a system attribute (indicated by a * - value, outline, order code, etc.) is deleted, adding a user-defined attribute with the same name does not make it a system attribute, so it will not be recognised as such. For example, the value assigned to a user-defined attribute named "Order code" will not appear in the parts list. To add a system attribute, use the export/import of attributes commands (right-click on the schematic folder).

To edit any of the above fields (except *Deletable*) select the attribute then the *Edit* button. Change the details as required followed by *OK*.

To delete any of the deletable attributes, select the attribute then the *Delete* button (greyed out if a mandatory field is selected).

If changes are made within the window select **OK** to implement the changes, or **Cancel** to forget the changes.

Add attributes - Extra user-defined attributes can be added if necessary, by selecting the **Add attribute** button. A window requests information about the attribute. The extra attribute information can be extracted when the part is used on the schematic, providing a user-defined extraction program is written. Refer to the *Tools > Parts/Netlist Extraction > User Defined Extraction* command.

Extra attributes are required for the PSpice output (*Tools > Parts/Netlist Extraction > User Defined Extraction, PSpice Net Extraction*) and these are described in the Seetrix XL Designer Installation & Getting Started Guide.

If a common attribute has to be added to all parts, then the attribute could be added to the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

Default symbol attributes

The default settings for new parts are controlled from the *schema_attribs.txt* file.

Part ident this field will be updated automatically when the part is used and allocated within a schematic. The asterisk will be replaced by the prefix and its parts identifier or reference designator, i.e. IC1, IC2, etc. Normally visible.

Prefix controls the characters in front of reference designators, for example IC, R, C, etc. A prefix must be supplied. The first 4 alpha-numeric characters are used. This field is not normally visible as the part ident is more useful.

Outline controls the physical outline that is used for the part in board layout. An outline name must be given in order for a parts and wiring list to be produced in the correct format for board layout. For this reason a default value of *DIL?* is supplied. This should be changed to a suitable name.

The first 16 alpha-numeric characters (but not spaces) are inserted into the parts list.

The outline can be created just prior to artwork design.

Value this field is optional, but typically contains the part type, i.e. 74LS00, 100k, etc. When filled in, the gate and pin swapping routines use this field to determine equivalent parts.

This field is inserted into the parts list *Type* field, which has a maximum of 33 characters. However the *Type* field is made up of the part name and value fields, separated by a comma, eg 7400,74LS00 so this should be taken into account when assigning values.

Symswap this field is optional, but must be filled in if gate swapping will be required on the layout.

Equivalent symbols are usually defined within the library part, using the *Attributes > Symbol Swap Rule* command - this field will therefore be filled in automatically. However, the field is either left blank or set to:

G0 gate swapping not permitted.

G1 gate swapping permitted between equivalent symbols within the part.

G2 gate swapping permitted between equivalent symbols within the part and with other identical parts.

Order code this field is optional. The first 16 alpha-numeric characters are inserted into the parts list if present.

Any other attributes are user defined.

User-defined attributes are added by selecting *Add attribute* button, then filling in the appropriate details.

To force a user-defined attribute to appear in the part attribute window when a part is created, the file *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default) can be edited. This file also controls the default height, visibility and names for all attributes.

Part Power Pins tab

When the *Attributes > Symbol* window is opened, the *Part Power Pins* tab can be selected to open the power pins definition window for the part.

Power pins do not have to be defined in this way, they can simply be added as terminals and numbered and named in the way that other pins are added. However, defining a pin as a power pin has advantages - they don't have to be visible on the symbol or schematic, but the connection to the power rails will be made automatically. These connections are automatically assigned a different track code so that they can be assigned a different width on the artwork.

Power pins defined in this way can also be assigned a separate power symbol (if required). This symbol can be placed on the sheet as required, allowing its pins to be connected through pull up/down resistors if required.

If a pin is defined as a power pin, then a terminal should not be added to the library part. The *Seetrax XL Designer Installation & Getting Started Guide* gives examples of how to create and use parts with power pins and power pin symbols.

Attributes > Pin

Used to display, and edit if necessary the attributes associated with the pins (terminals). Most of the information can be entered graphically using the *Terminals* commands.

Select *Attribs > Pin*, point at the pin (terminal), then select it with a click of the left button.

Pin attribute window

A pin attribute window (Figure 105), lists the attributes that have been defined for the pin, the values assigned to the attribute, whether they are visible and their height. If the field is optional, it may be deleted if required.

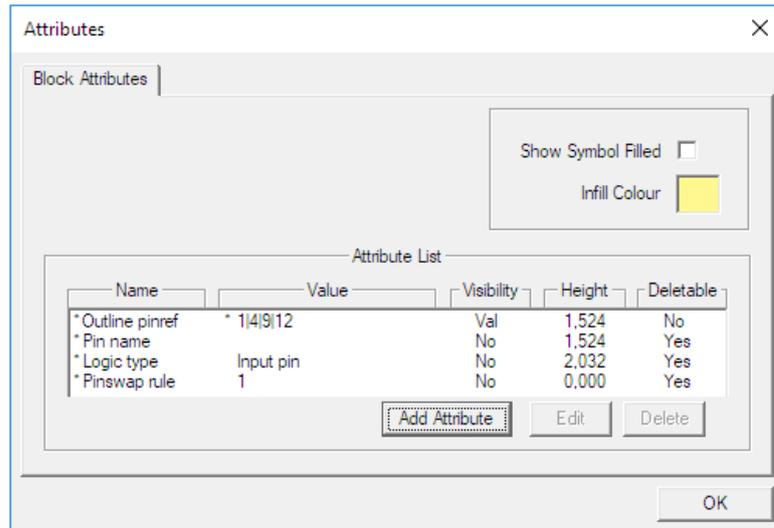


Figure 105

- Name** contains the reference name of the attribute. If a * precedes the name, this is a system attribute and used elsewhere in the program – removing it (if permitted) would have implications for other tools, such as the electrical rules checker.
- Value** contains the text string or value assigned to the attribute.
If an asterisk (*) precedes a Value field entry, that entry cannot be changed from the window. i.e. the *Outline pinref* value in Figure 105 which is controlled by the *Terminals > Assign Pin Numbers* command.
- Visibility** controls whether the attribute is visible or not when the part is used on the sheet. Options are:
Not visible - the attribute is not visible.
Value only - the attribute's value field is visible
Name +Value - the attribute's name and value field is visible
- Height** controls the height of the attribute when it is visible.
- Deletable** indicates whether the attribute is permitted to be removed from the part – this setting cannot be changed.
- Note: if a system attribute (indicated by a *, i.e. Pin Name, Logic type) is subsequently deleted, adding a user-defined attribute with the same name does not make it a system attribute, so it will not be recognised as such. For example, the value assigned to a user-defined attribute named "*Logic type*" would not be recognised by the electrical rules checker.

To edit any of the above fields (except *Deletable*) select the attribute then the *Edit* button. Change the details as required followed by *OK*.

To delete any of the deletable attributes, select the attribute then the *Delete* button.

Extra user-defined attributes can be added if necessary, by selecting the **Add attribute** button. A new attribute line is created and can be filled in as required. The extra attribute information can be extracted when the part is used on the schematic, providing a user-defined extraction program is written. Refer to the *Tools > Parts/Netlist Extraction > User Defined Extraction* command.

If a common attribute has to be added to all new part pins, then the attribute could be added to *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

If changes are made within the window select **OK** to implement the changes, or **Cancel** to forget the changes.

Default pin attributes

The default settings for new pins are controlled from the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

Outline pinref indicates the pin number of the pin. This pin number is linked automatically to the physical pad with the same pin number.

Pin numbers must be supplied for library parts and split parts.

The pin numbers can only be changed using the *Terminals > Assign Pin Numbers* command.

Where the pin selected is an element from a multi-element part, then all the pin numbers

for that pin are shown, separated with a | sign.

Upto 16 alpha-numeric characters in any order can be used.

<i>Pin name</i>	indicates the optional name assigned to the pin, for instance WE, CS, DI, etc. Upto 16 alpha-numeric characters can be used, in any order.
<i>Logic type</i>	indicates the logic type of the pin. By default it is set to <i>Unspecified port</i> . The electrical rules checker refers to this information when checking the schematic for errors. Selection is made from the arrow alongside.
<i>Pin swap rule</i>	indicates the pin group the pin belongs to. On the layout connections can be swapped between any pins within a gate or part that belong to the same group. Group numbers run from 0 to 6. Pins within group 0 are not swappable. Equivalent pins are typically defined within the library part, using the <i>Terminals > Set Pin Swap Rule</i> command.

Any other attributes are user defined. User-defined attributes are added by selecting *Add attribute* button, then filling in the appropriate details.

To force a user-defined attribute to appear in the pin attribute window when a new part is created, edit the *schema_attribs.txt* file, held in the *Configuration Data Directory* (MyDocuments/Seetrax/XL Designer/Data by default).

This file also controls the default height, visibility and names for all attributes.

Attributes > Move

Used to move a part or pin attribute.

Once selected, point at the attribute to be moved and click the left button. As the cursor is moved, the selected attribute is seen attached to it with a line extending to the item it belongs to (the part or the pin). Position the attribute and release it with a click of the left button.

As the attribute is being moved, it can be rotated by 90 degrees by pressing the **<rotate part>** special function key. (See *Attributes > Rotate* for details on attribute rotation.) Clicking the right-hand mouse button before it is released restores the attribute to its original location.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Attributes > Hide

Used to hide (make invisible) an attribute associated with the part or a pin.

Once selected, point at the attribute (text string) that you want to hide, then click the left-hand mouse button. The attribute is hidden.

It can be restored to view if the *Attributes > Symbol/Pin* command is used and the text string made visible again.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Attributes > Rotate

Used to rotate an attribute associated with the part or a pin. Only two orientations are available, 0 degrees or 90 degrees.

Once selected, point at the attribute (text string) to be rotated, then click the left button. The attribute rotates by 90 degrees. If the attribute is selected again, it rotates to its original orientation.

The *trap distance* defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it. The *rotated text display mode* in the *Edit > Preferences, Miscellaneous* window controls how/whether the characters within the string are rotated.

Attributes > Symbol Swap Rules

Used to define whether gate swapping is allowed. Gate swapping is not permitted by default. The number of gates within a part is defined in the symbol attributes window (*Attributes > Symbol*).

Symbol swapping allows all of the connections attached to a symbol to be swapped with all the connections of another identical symbol.

Gates within *parts* are always equivalent to one another, but by default swapping is not permitted. Local or global swapping can be enabled.

No swapping indicates that gate swapping is not permitted.

Global swapping indicates that gate swapping is permitted between gates within the package and also between other packages of the same type. For instance, if the gates of a 7400 were allowed to swap globally, and two 7400's were used on the schematic, gate swapping could take place between all eight gates; each gate could be swapped with seven other gates (four gates per package).

Local swapping indicates that gate swapping is permitted between gates within the same package only. For instance if the gates of a 7400 were allowed to swap locally, and two 7400's were used on the schematic, gate swapping could take place within each of the packages, but

not between them; each gate could be swapped with the other three gates within the package.

When selected a window appears from which one of the above choices should be made.

If a **split part** is being edited, the gate swapping rules are accessed from the *Subsymbol > Swap Rules* command.

Subsymbol commands

Used to add, delete, or move the primitives associated with a split part. These commands only appear when a split part is being edited.

Parts should be defined as split parts when the part is made up of symbols (or gates) that are different from one another and need to be spread around the schematic. For instance a relay is made from two different symbols (coil and contacts) that are not always positioned together.

Parts that are made up of one or more symbols which are the same, should be made as a library part, for instance a 7400 which contains 4 x 2 input nand gates, unless you want to show the power pins on one of the gates (rather than as a separate power symbol).

Subsymbol > Place

Used to select the primitives required in the split part.

Refer to the *Grid* command to find out whether grid snapping is active.

Once the command is selected, use the Navigator pane to locate the primitive required. Once located, select it, then move the cursor and attached primitive into the split part window.

The primitive can be rotated in 90-degree increments or flipped by pressing the <rotate part> or <flip part> special function keys. Position the primitive and release it with a click of the left button. Another primitive appears on the end of the cursor, release it in the same way if it is required.

The position of sub-symbols with respect to one another in the split part is unimportant. The symbols are placed onto the schematic individually.

To select and place a different primitive, move back to the navigator and select another primitive. Click the right-hand mouse button when no more primitives are required.

Subsymbol > Move

Used to move subsymbols (or primitives) within split parts.

The position of the primitives relative to one another is not important, as the primitives are added to the schematic independently of one another.

The primitive is selected by its datum point, which is shown as a small cross. This was defined in the primitive when the symbol was made.

Refer to the *Grid* command to find out whether grid snapping is active. The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the primitive, then select it with a click of the left button. Move the cursor, the primitive appears attached to it. Position the primitive as required, then click the left button to release it. Clicking the right button before the primitive is released, returns the primitive to its original position.

Subsymbol > Rotate

Used to rotate subsymbols (primitives) in 90 degree steps, anti-clockwise within split parts. The primitive is added to the schematic at the orientation shown in the split part. It can be rotated on the schematic as required.

The position of the primitive relative to the other primitives is not important, as the primitives are added to the schematic independently of one another.

The primitive is selected by its datum point, which is shown as a small cross. This was defined in the primitive when the symbol was made.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the primitive, then select it with a click of the left button. The primitive rotates by 90 degrees anti-clockwise each time it is selected.

Subsymbol > Flip - Used to flip subsymbols (or primitives) within split parts.

Primitives are flipped about their original y-axis, i.e. primitives are turned left to right about their datum unless they have been rotated by 90 or 270 degrees first, in which case they are turned top to bottom about their datum.

The primitive is added to the schematic at the orientation shown in the split part. It can be flipped on the schematic as required.

The position of the primitive relative to the other primitives is not important, as the primitives are added to the schematic independently of one another.

The primitive is selected by its datum point, which is shown by a small cross. This was defined in the library

primitive when the symbol was made.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the primitive, then select it with a click of the left button. The primitive flips each time it is selected.

Subsymbol > Delete

Used to delete subsymbols (or primitives) from split parts.

The primitive is selected by its datum point, which is shown as a small cross. This was defined in the library primitive when the symbol was made.

The trap distance defined in the *Edit > Preferences, Miscellaneous* window controls how close you have to be to an item to select it.

Once selected, point at the datum point of the primitive, then select it with a click of the left-hand mouse button. The primitive and its associated attributes are deleted.

Subsymbol > Swap Rules

Used to define whether gate swapping is allowed. Gate swapping is not permitted by default.

Symbol swapping allows all of the connections on the artwork attached to a symbol, to be swapped with all the connections of another identical symbol.

By default swapping is not permitted. Local or global swapping can be enabled.

Local swapping indicates that gate swapping is permitted between equivalent gates within the same package only.

Global swapping indicates that gate swapping is permitted between equivalent gates within the package and also between other packages of the same type.

When selected large circles appear on the datum of the individual primitives (gates) in the split part. These circles and their colour indicate which gates within the package are equivalent to one another, and whether local or global swapping is permitted.

Initially all gates are unequal (clear circles), they are not equivalent to any other gate so cannot be swapped on the layout.

Once the command is selected, use the buttons in the *Split Part Swap Rule* dialogue bar on the left of the screen, to select *Local* or *Global* swapping, then select one of the colours. Now select the gates that are to be equivalent to one another, their circle will change colour and contain the letter *L* for *Local* or *G* for *Global* swapping.

To return the gates to unequal, select *U* from the dialogue bar on the left of the screen then select the gates that are unequal.

Schema configuration file – “schema_attribs.txt”

The file *schema_attribs.txt* file exists in the *Configuration Data Directory* which, when the software was installed was set to the folder:

MyDocuments/Seetrax/XL Designer/Data

The path can be verified from the *File > System Setup* window.

(For Ranger2 XL and Ranger XL users, the *schema_attribs.txt* file replaces the *capture.cfg* file that they may have been familiar with, but it no longer holds library information.)

The *schema_attribs.txt* file defines the default attributes of parts, pins and connections, and their default visibility and position.

This file can be modified (carefully) using Notepad or a similar editor to change these defaults, to add new user-defined attributes or to change library names/descriptions.

If new attributes are added to this file, they are added to all new parts/pins.

The complete content of *schema_attribs.txt* is listed below, followed by a description of the entries.

Important!

Always close Seetrax XL Designer before modifying *schema_attribs.txt*.

Before editing it, take a backup copy of the file as a precautionary measure.

Only alter the data as described below, do not alter anything else.

Content of *schema_attribs.txt*

```
# Circuit schematic configuration and default attributes dictionary
#
#----- Reserved attribute code naming section -----
#
.dict_defaults
0 "Part ident" # Allocated part ident (placeholder only)
```

```

1  "Prefix"          # Part code prefix      (never visible)
2  "Value"           # Default component value
3  "Outline"         # Outline library entry
4  "Order code"      # Data for parts list bill-of-materials field
5  "Symswap rule"    # Swapping rule for equivalent symbol instances.
6  "Block name"      # Displayed name for instances of user blocks
7  "Pin name"        # Symbol i/o pin name
8  "Outline pinref"  # Pin reference on outline library entry.
9  "Pinswap rule"    # Swapping rule for equivalent symbol pins.
10 "Logic type"      # Pin logical type (eg.. tri-state, open-
collector)
11 "Bus subscript"   # Bus ripper signal subscript.
12 "Signal name"     # Signal name.
13 "Routing width"   # Ranger track routing width code.
14 "Minimum clearance" # Minimum permitted track clearance
15 "Datasheet Addr" # URL or filepath to online datasheet
.end
#
#----- Miscellaneous attributes -----
#
# Generic extractor configuration and tools path
# .genex_path ./genex
#
#----- Default symbol and pin attributes for each block type -----
#
# ins flag values : 'M' = mandatory, 'O' = optional.
# ved " " : 'Y' = value editable, 'N' = value non-editable
# vis " " : 'value' 'name+value' 'off'
#
.libprim_symbol
$0 "" ins=M ved=N vis=value x=0 y=0.15 height=0.08
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
.end

.libprim_pin
$8 "" ins=M ved=Y vis=value x=-0.2 y=-0.15 height=0.08
$7 "Name" ins=O ved=Y vis=value x=-0.2 y=0.15 height=0.08
$10 "" ins=O ved=Y vis=off height=0.08
$12 "" ins=O ved=Y vis=off height=0.06
.end

.libpart_symbol
$0 "" ins=M ved=N vis=value x=0 y=0.15 height=0.08
$1 "IC" ins=M ved=Y
$3 "DIL?" ins=M ved=Y
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
$4 "" ins=O ved=Y
$5 "" ins=O ved=Y
$15 "" ins=O ved=Y
.end

.libpart_pin
$8 "" ins=M vis=value x=-0.2 y=-0.15 height=0.08
$7 "Name" ins=O vis=value x=-0.2 y=0.15 height=0.08
$10 "" ins=O ved=Y vis=off height=0.08
.end

.libspart_symbol
$1 "IC" ins=M ved=Y
$3 "DIL?" ins=M
$4 "" ins=O
$15 "" ins=O ved=Y
.end

.libspart_pin
$8 "" ins=M vis=value x=-0.2 y=-0.15 height=0.08
$7 "Name" ins=O vis=value x=-0.2 y=0.15 height=0.08
$10 "" ins=O ved=Y vis=off height=0.08

```

```

.end

.userblock_symbol
$6      ""      ins=M ved=N vis=value x=0 y=0.15 height=0.08
.end

.userblock_pin
$7      "NET"   ins=M vis=value x=-0.2 y=0.15 height=0.08
.end

.blockio_symbol
.end

.blockio_pin
$7      "NET"   ins=M vis=value x=-0.2 y=0.15 height=0.08
.end

.connection_attribs
$12     ""      ins=0 vis=value      height=0.06
$13     ""      ins=0 vis=name+value height=0.04
$14     ""      ins=0 vis=name+value height=0.04
.end

```

Description of the entries within *schema_attribs.txt*

Any line that starts with a #, or text after the # on a line is ignored. These are comments.

The file is case sensitive, so do not replace uppercase characters with lowercase characters or vice-versa.

The file is divided into sections, and these are now described.

Reserved attribute code naming section

The first section within the file lists the internal code numbers assigned to the system attributes, as follows:

```

.dict_defaults
0 "Part ident"      # Allocated part ident (placeholder only)
1 "Prefix"         # Part code prefix      (never visible)
2 "Value"         # Default component value
3 "Outline"       # Outline library entry
4 "Order code"    # Data for parts list bill-of-materials field
5 "Symswap rule"  # Swapping rule for equivalent symbol instances.
6 "Block name"    # Displayed name for instances of user blocks
7 "Pin name"      # Symbol i/o pin name
8 "Outline pinref" # Pin reference on outline library entry.
9 "Pinswap rule"  # Swapping rule for equivalent symbol pins.
10 "Logic type"   # Pin logical type (eg.. tri-state, open-
collector)
11 "Bus subscript" # Bus ripper signal subscript.
12 "Signal name"  # Signal name.
13 "Routing width" # Ranger track routing width code
14 "Minimum clearance" # Minimum permitted track clearance
15 "Datasheet Addr" # URL or filepath to online datasheet
.end

```

These code numbers are referred to later on in the file. For instance **part idents** have an internal code of **0**, **prefixes** code **1**, **values** code **2**, etc. These entries must not be changed or added to.

Miscellaneous attributes

As supplied this section is commented out, so has no effect.

By default Seetrix XL Designer will locate the generic extractor files from the path specified for them in the *File > System Setup* window.

```

#----- Miscellaneous attributes -----
-

```

```
#
# Generic extractor configuration and tools path
# .genex_path ./genex
#
```

However the user can choose to over-ride the default path with a path of his/her own choosing and therefore store the .gex files anywhere that can be accessed.

If so required, uncomment the line as shown below and specify the path to the folder containing the generic extractor configuration files (these files all end with a file name extension of '.gex').

```
# Generic extractor configuration and tools path
.genex_path <path to *.gex files>
```

Default symbol and pin attributes for each block type

This section determines which attributes are used within the different types of "part" on the schematic. Each "part" type has its own entry starting with a . (dot) and ending with a .end statement.

Parts	look for .libpart_symbol and .libpart_pin
Split parts	look for .libparts_symbol and .libparts_pin
Primitives	look for .libprim_symbol and .libprim_pin
Design symbols	look for .userblock_symbol and userblock_pin
Block I/O ports	look for .blockio_symbol and .blockio_pin

For example, the following entries define the attributes associated with a primitive and its pins (the columns have been spread out for ease of identification).

```
.libprim_symbol
$0 "" ins=M ved=N vis=value x=0 y=0.15 height=0.08
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
.end
.libprim_pin
$8 "" ins=M ved=Y vis=value x=-0.2 y=-0.15 height=0.08
$7 "Name" ins=O ved=Y vis=value x=-0.2 y=0.15 height=0.08
$10 "" ins=O ved=Y vis=off height=0.08
$12 "" ins=O ved=Y vis=off height=0.06
.end
```

In this example, the second and third lines define the attribute codes associated with the primitive symbol. (The code numbers are preceded by a \$.) They are **0** and **2** (*part ident* and *value* as defined in the *Reserved attribute code naming section* of the file).

Lines six through to nine define the attributes associated with the pins of a primitive, which are **8** (outline pinref), **7** (pin name), **10** (logic type) and **12** (signal name).

The text string within the "" will appear everytime the symbol is created or a pin is added to the symbol.

In this example, the string *Name* will appear alongside a pin when it is added to the primitive, the string corresponds to the pin name (code7). No other values will be assigned.

The remaining parameters on these lines are now described.

- ins flag values** control whether the attribute is a mandatory one.
It can either be set to **M** for mandatory or **O** for optional - do not change the mandatory values assigned to the supplied system attributes.
If an attribute is set to mandatory, then it will always appear in the attribute window and cannot be deleted. If it is set to optional, then it will appear but can be deleted.
- ved flag values** control whether the attribute can be edited.
Can either be set to **Y**, for *yes* the value assigned to the attribute can be edited from the schematic editor or **N** for *no* the value cannot be edited.
- vis flag values** control the default visibility of the attribute. It can either be set to:
value displays the value assigned to the attribute
name+value displays the attribute name and the value assigned to it

off does not display the attribute or its value.

Looking at the attributes for a part, which are shown now.

```
.libpart_symbol
$0 "" ins=M ved=N vis=value x=0 y=0.15 height=0.08
$1 "IC" ins=M ved=Y
$3 "DIL?" ins=M ved=Y
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
$4 "" ins=O ved=Y
$5 "" ins=O ved=Y
$15 "" ins=O ved=Y
.end

.libpart_pin
$8 "" ins=M vis=value x=-0.2 y=-0.15 height=0.08
$7 "Name" ins=O vis=value x=-0.2 y=0.15 height=0.08
$10 "" ins=O ved=Y vis=off height=0.08
.end
```

As in the previous description, \$ precedes the code numbers. Each code number refers back to the internal code obtained from the first section. \$0 = part idents, \$1 = prefixes, \$2 = values, etc.

After the internal code number are a pair of "". The text string within the "" will appear by default for that attribute. For instance, the symbol's part prefix (code 1) will be set to **IC** and its outline name (code 3) to **DIL?**. When a pin is added to a library part, its pin name (code 7) is set to **Name**.

Most attributes do not have a default entry.

Each entry is followed by the three parameters described above. So for instance, looking at the entry for code 2, the *Value* attribute.

```
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
```

It doesn't have a default value assigned because the "" are empty.

It's an optional entry (**ins=O**), it can be edited (**ved=Y**), and the value assigned to it will be displayed (**vis=value**). The position of the value field with respect to the datum of the symbol, is **X=0** and **Y=-0.15"** and the attribute text will be **0.080"** tall.

When X and Y values are given for pin attributes, they are with respect to the pin itself. So for instance:

```
$7 "Name" ins=O vis=value x=-0.2 y=0.15 height=0.08
```

Code 7 the pin name, has a default value assigned of *Name* because Name is found within the "". It is an optional entry **ins=O** and its value is visible **vis = value**. It is positioned 0.2" to the left of the pin and 0.15" above the pin **x=-0.2 y=0.15**, the text height will be **0.08"**.

Adding User-Defined attributes as defaults

In the example below, new user-defined attributes **Price** and **Supplier** have been added to the part symbol.

```
.libpart_symbol
$0 "" ins=M ved=N vis=value x=0 y=0.15 height=0.08
$ "IC" ins=M ved=Y
$3 "DIL?" ins=M ved=Y
$2 "" ins=O ved=Y vis=value x=0 y=-0.15 height=0.08
$4 "" ins=O ved=Y
$5 "" ins=O ved=Y
$15 "" ins=O ved=Y
"Price" "" ins=M ved=Y vis=off
"Supplier" "" ins=M ved=Y vis=off
.end
.libpart_pin
```

```

$8      ""      ins=M          vis=value   x=-0.2   y=-0.15   height=0.08
$7      "Name"  ins=O          vis=value   x=-0.2   y=0.15    height=0.08
$10     ""      ins=O          ved=Y       vis=off   height=0.08
.end

```

User-defined attributes must not be preceded by a \$ sign and the attribute names should appear in "" as shown. Apart from that, they are treated in the same way as supplied attributes.

If these entries were added to the file, any parts edited from that point on, would contain the attribute names, *Price* and *Supplier*.

Note: if a user-defined attribute is added to this file and values entered from the schematic editor, the values will be lost if the attribute is subsequently removed from this file.

Component Outline Folder

Each design has its own component outline folder, which contains the component outlines used on that particular design, plus any others the user may choose to include.

The component outline or "footprint" represents the physical size and shape of a component that is placed on the artwork.

Component outlines that are required in the design are copied automatically from the master component outline folder (if they exist) as and when they are required. This typically happens when the parts/nets list is extracted from the schematic, the parts list is modified or component outlines are swapped on the artwork.

Component outlines can also be copied/pasted to/from other component outline folders (other designs or the master folder). The component outlines can also be created directly in the folder.

Once component outlines have been copied into the design, changes to them do not affect the original component outlines, other designs or the master component outline folder.

Changes to component outlines in the master component outline folder do not affect any other designs.

The master component outline folder is accessed from the *Masters* folder in the navigator window. It is suggested that component outlines are created within a design folder and verified before they are copied to the master folder. Once component outlines are in the master folder it is assumed they are correct.

Opening/viewing the component outline folder and its content

Locate the component outline folder required from the navigator window. The master component outline folder is held inside the *Masters* folder, whilst each design's component outline folder is held inside the design folder.

Once found, double-select select the *Component Outlines* folder (or select the + sign alongside it) to open it. If component outlines exist in the folder they will be listed below the open folder in the navigator pane.

The component outline folder of a new design will be empty.

To close the folder, either double-select the folder (or select the - sign alongside it).

If outlines are listed they can be selected in turn to browse through them. They are shown graphically in the *Browser* window and their properties are displayed in the *Properties* window (if those windows are open). An outline can be opened for editing by double-selecting it.

When the component outline folder is selected, the total number of outlines held within it are shown in the Properties pane.

Component outline information

Viewing direction (from top or bottom)

It doesn't matter whether component outlines are viewed from the top or the bottom of the board, provided they are all viewed from the same direction. In practice, most companies view outlines from the top of the board.

The supplied component outlines are viewed from the top of the board. If outlines will be added to this folder, then they should also be created as viewed from the top of the board.

Parts can be flipped from the top of the board to the bottom, once they have been placed on the artwork. Silk screen part names are automatically mirrored if parts are flipped. (Text that is added to the outline using the *Outline > Text* commands is not mirrored when parts are flipped.)

Pads

When a component outline is created, it should be designed so that it could be used on single, double or multi-layer designs without further editing. To this end, the component pads are added as a "stack" of pads.

The stack within SXLD comprises of a pad that is used on the top of the design, a pad that is used on all the inner layers of the design and a pad that is used on the bottom of the design. (These layers are described as Top, Inner & Bottom.)

For instance, pin 1 of a component could be defined as a square pad on the bottom of the board, as a round-ended rectangular pad on the top of the board and with small round pads on all the inner copper layers of the board, however many there are.

(Layers defined as power planes have their own pad definition for the layer and this is described in the description of the *standard pads folder*.)

Through-board components

When creating component outlines that require a hole through the board, pads should be added to **all** the layers in the stack, even if the outline will only be used on single or double-sided boards – pads on unused layers are ignored.

The pad used on the **inner** pad determines the hole size of the pad stack, so it must be included if a drilled pad is required. A pad stack that only has a pad on the top or bottom, or top and bottom of the board will not be drilled.

Surface mounted components

Surface mounted component outlines should have their pads added to only the top, or only the bottom of the stack, depending on which side it will be mounted. Bear in mind that outlines can be flipped from side to side once in the artwork editor, so it is only necessary to create one outline that can be used on the top and bottom of the board.

It is wise to always build outlines from one side of the board (usually top), then *flip* them if they are required on the other side of the board.

This ensures that all flipped parts are on one side of the board (usually bottom) and un-flipped parts are on the other (usually top).

When the silk-screens layers are generated, the silk-screen for flipped outlines and labels are placed on one side of the board and unflipped outlines and labels on the other. If the outlines were defined with some outlines defined on the bottom of the board and others on the top of the board, when these outlines are placed on the artwork, they will be on different sides even though they haven't been flipped and this will cause confusion.

How modifications to outlines in use on the artwork are implemented

Changes made to outlines in the design outline folder do not affect other designs or the master folder.

If pads are modified, moved or deleted using the *Pad* commands from within the design library and the outline is already used on the artwork, be aware that all occurrences of that outline in the artwork are automatically updated – but only when the outline has been closed. Whilst the outline's editing window remains open, the artwork is not updated.

Note: if individual pads in outlines have been changed in the artwork editor (using the Amend commands to change size/shape), then changes in the outline to the size/shape of those pads will not be implemented on those outlines in the artwork. Changes made on the artwork to individual pads in outlines take precedence over general changes to outlines.

If silkscreen data or free layer information (i.e. data added using the *Outline* commands or *Copper* commands on the *Free copper layers*) is modified from within the design library and the outline is used on the artwork, all occurrences of that outline in the artwork are automatically updated. However, if the silk screen outlines and free layers have been *generated* on the artwork using the *Tools > Generate silk-screen* command, then that data is not updated.

If data on the top, bottom or all layers is modified using the *Copper* commands from within the design library and the outline is used on the artwork, these changes are not updated on the artwork until the part is flipped (typically parts are flipped twice, once to update them, the second to restore them to their original side).

Component outline families - introduction

The outlines can be added to "families" (right-click on the component outline folder, then select *Manage Families* from the options that appear). Outlines within the same family can be swapped one for another in the artwork editor. For example an SO14 and DIL14 could be added to the same family. A DIL14 might be called up, but if there was insufficient space on the artwork it could be swapped for the SO14 easily and quickly (provided the part was available in both styles). There are no hard and fast rules on which outlines to add to a family, but a family of 2 pin outlines, 3 pin outlines, 14 pin outlines, 16 pin outlines, etc. might be useful.

Additional "copper"/free copper - introduction

Until now in this description, "pads" have been referred to and these have corresponded to the pads required for the part to be inserted into/soldered onto the board. Each of these pads will have a pin number that corresponds to the schematic device and "physical" part. They are added and numbered using the *Pad* commands.

Within the component outline editor there is an additional type of pad that can be added. These are added using the *Copper* commands, but they may not always be "made" from copper. For instance they can represent solder paste or glue spot pads.

An outline must always have at least one pad added from the *Pad* menu, but it does not have to contain any pads from the *Copper* menu.

Copper pads on the top, inner or bottom layers are only used for particular applications such as stringers and dual outlines (both described below).

"Copper pads" can also be placed on "free layers" in order to create the user-defined solder paste pads, solder mask pads, glue spots, etc. There are eight "free layers". They are called free layers because the data that is added to them can appear on any user-specified layer once the outline has been placed on the artwork. (The layers used can vary between designs.)

The free layers can be given user defined names, for instance, top paste, bottom paste, glue spot, etc. to assist in their recognition. These layers will be defined as silk screen layers; they do not form part of the electrical connectivity and are not checked during artwork checking.

If a "copper pad" is added, it is never given a number. If it is added to the top, bottom or inner layers, it must be placed on the end of a piece of copper track on the same layer, which in turn has to start from a pad added using the *Pad* commands.

The pads and tracks on the free layers do not appear automatically when the outlines are used in a layout. The layer is added as required via the artwork editor *Tools > Generate silk-screen* command.

Note: Ranger will produce solder paste and solder mask output files automatically, which output the used pins with a uniform user-defined pad reduction/swell. It is therefore unnecessary to create these free layers unless you have specific requirements.

Pads and tracks that are added to free layers can be placed anywhere, they do not have to be attached to a track or pin.

Points to bear in mind when using outlines containing "copper"

- * If family swapping is used within the artwork (*Parts > Change Outline* command) and the new outline contains copper, it will need to be flipped twice to introduce the copper (the first flip introduces the copper as well as flipping the part, the second flips it back onto the correct layer). Alternatively, the part could be removed to the tray and placed again.
- * The stringers can be ripped up using the *Mroute > Ripup* command. The connections are displayed from the point where the stringer was. Use the *Parts > Reconnect Power* command to restore the connections to the pins.
- * Gate and pin swapping cannot be used on gates or pins whose connections have been pre-routed. This means that gate and pin swapping cannot be used on pins with stringers or dual outlines. (If gate and pin swapping has to be used, rip-up the copper using the *Mroute > Ripup* command. Use the *Parts > Reconnect Power* command to reconnect the connections back to the pins. Gate and pin swapping can then be used. Use the *Parts > Flip* command (twice) to restore the copper to the part.)
- * None of the auto-routers can accept stringers, so do not submit boards containing stringers to the auto-routers.

Stringers

These are tracks that lead from the pad of a surface mounted device to a via.

Under normal circumstances, when routing to a surface mounted pad, the track has to terminate on the same layer as the pad, which can lead to congestion around the component.

If stringers are used in a suitable pattern, the track to the via on the end of the stringer can appear on any layer, so there is less congestion. Stringers should therefore assist in the routing of the layout but the down-side can be the increase in the number of vias on the board that may not be required.

The stringers can be built into the component outline. The stringer is treated as a via and track once the outline appears in the artwork editor so it can be modified or ripped up if it is not required.

Note: it is not recommended that stringers are added to outlines within Ranger. The auto-routers add stringers as required and will not accept those defined in the outline. When manually routing, it is usual just to add the stringers as required.

Dual outlines

A dual outline consists of an outline with an extra set of pads linked to the original pads, with copper tracks.

Dual outlines are typically used when a part can be supplied in different packages and it is unknown which type will be available when the board is assembled, for instance a dual-in-line or surface mount package.

Saving an outline

Individual outlines cannot be saved, all the changes within the design are held in memory until the design is saved.

Creating an outline, general guidelines

The commands referred to are described in detail later in this chapter.

From the navigator, right-click over the component outline folder where the component outline will be created, then select *New* from the options that appear, supply an appropriate name. Double-select the outline to open it for editing.

Check the *Grid* is set to suit the pad pitch requirements.

Check the pad and line sizes available - double-select the *Standard pads* folder then double-select any of the pad shape folders to open the complete list of pad shapes/sizes & track sizes. Change them as necessary, but bear in mind that vias and existing outlines may be using the size codes with values assigned in the table (i.e.

those set to 0 are unused).

If copper will be added to the free layers, define the free layer names and colours via the *Copper > Free Copper Setup* command.

Add the pads using the *Pad > Add/Auto Add* command.

Assign the pin numbers using the *Pad > Assign/Auto-assign* command.

Position the datum of the outline, *Outline > Set Outline Datum*.

Define the bounds of the outline using the *Autoplace > Adjust Footprint* command.

Add the silk screen outline using the *Outline* commands, you may need to change the *Grid*.

Add the stringers, dual outlines or free pads using the *Copper* commands.

Define the component's height details if the DXF 3D Solid model output tool will be used *View > Properties*.

If the Specctra or Elecctra autorouter will be used, decide whether vias will be allowed in the smd pads in the outline (if applicable) *View > Properties*.

The outline is ready to be used on the artwork once the outline editing window has been closed.

If more outlines need to be created or modified, repeat the procedure.

Individual commands in the component outline editor

An overview of the outline editor can be found in the *Installation & Getting Started Guide*. Here we describe each of the outline editor commands in detail.

Right-click on component outline folder

Right-clicking on the Component Outline folder in the navigator window introduces the following commands:

New Paste Manage Families Edit Properties Purge Unused Outlines

These commands are described below.

New - Creating a new component outline

A new outline can be created by right-clicking the *component outline folder* in the navigator window and selecting *New* from the options that appear. A new outline is added at the bottom of the list of outlines in the folder, called *Unnamed*. It can be renamed at that point as it is highlighted ready for editing. However, if <enter> is pressed or the cursor is selected elsewhere, the original name (unnamed) will remain. It can be renamed by right-clicking on the outline and selecting *Rename*.

Paste

A component outline that has been copied to the paste buffer can be pasted (copied) to the component outline folder. Right-click the *component outline folder* in the navigator window, then select *Paste* from the options that appear. The outline is added to the bottom of the list of outlines in the folder, it will take its original name unless an outline with that name already exists in the folder, in which case it will be named "*Copy of <original name>*". (Refer to "*Right-click on a component outline*" for details on copying and dragging/dropping outlines to the paste buffer.)

It can be renamed by right-clicking on the outline and selecting *Rename*.

When an outline is copy/pasted between designs or to/from the master library, the pad and track codes used in the imported outline may change, but the actual sizes will remain the same.

For example, take an outline that was created with code 4 pads where code 4 was set to 0.060" diameter with a 0.025" drill. If the outline were pasted into a library where code 4 pads were set to 0.080" diameter with a 0.040" drill, then the pads in the pasted outline would be wrong. To ensure this doesn't happen, Ranger ignores the pad code from the pasted outline and only takes into account the actual sizes/shapes assigned during the paste. It looks for an exact match in the destination sizes table and uses that code in the outline. If an exact match cannot be found, then it looks for an empty value in the table (set to 0) and updates that value to the size required (including drill diameter, pad diameter/width/length), then uses that code in the outline.

Manage Families

This command is used manage the family groups in the component outline folder.

A family contains a group of outlines that can be easily substituted one for another within the artwork editor using the *Parts > Change Outline* command.

Typically, outlines with the same number of pins are added to one family. For instance, the outlines DIL14, DILS14 and SO14 could be added to one family called 14PINS. At the layout stage, the DIL14 could be replaced easily for an SO14 because of space considerations, etc.

An outline can only belong to one family. Note: an outline could however belong to different families within the design and master libraries. If this outline were used on an artwork and the *Parts > Change Outline* command used, then the outlines from both families are listed. The letter M is placed alongside the outlines from the master family.

Once selected, the family management window appears, similar to the one shown in Figure 106.

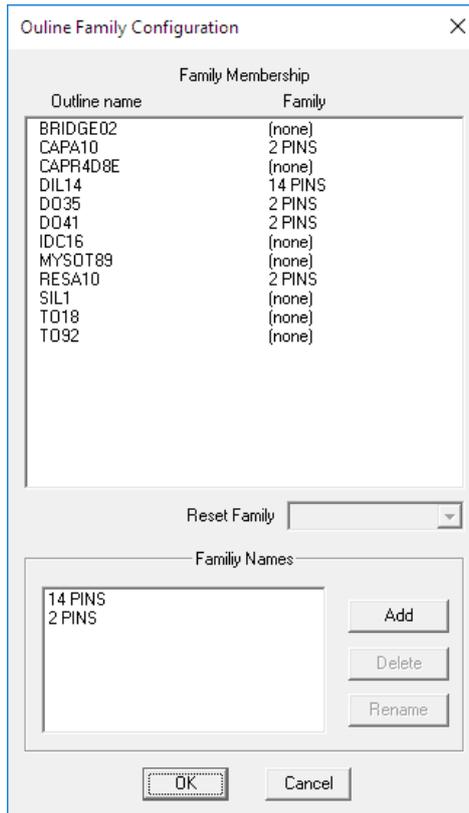


Figure 106

The top pane shows the *Family Membership* list, which lists all the outlines in the library and the family group to which they belong. The bottom pane shows the *Family Names*.

When all the changes required have been made, select *OK* to close the window and keep the changes or *Cancel* to discard the changes.

Creating a family

To create a family name, select the *Add* button from the window. Supply a name for the family when prompted, followed by <enter> or select *Proceed*. The name appears in the lower pane in the window. Repeat as required.

Deleting a family

To delete a family name, select the family name from the lower pane, then select the *Delete* button from the window. Confirm or *Cancel* the selection as required. Deleting the family does not delete the outlines themselves from the library.

Renaming a family

To rename a family name, select the family name from the lower pane, then select the *Rename* button from the window. Provide a new name as required followed by <enter> or select *Proceed*.

Adding an outline to a family

(A family name must have been created and appear in the Family Names list.) Select the outline, then select the arrow from alongside the “*Reset Family*” attribute. A drop down list appears showing all the family names available, plus *(none)* similar to the one shown in Figure 107.

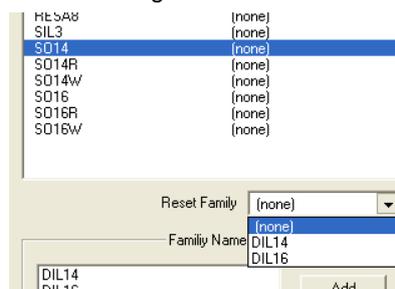


Figure 107

Select the family name and the Family Membership list is updated.

Removing/Changing an outline from its family

Select the outline, then select the arrow from alongside the “Reset Family” attribute. A drop down list appears showing all the family names available, plus (*none*). None should be selected if the outline does/should not belong to a family. Select the family or (*none*) as required. The Family Membership list is updated.

Edit Properties

This command can be used to allow the bulk editing of component outline properties. It saves time when many outlines have to be updated as the individual outlines do not need to be opened.

Some/all of the outlines can be selected and some/all of the properties edited.

(If an outline is opened for editing and the *View > Properties* command is selected, then only the properties of that particular outline can be changed.)

When selected, the window shown in Figure 108 appears.

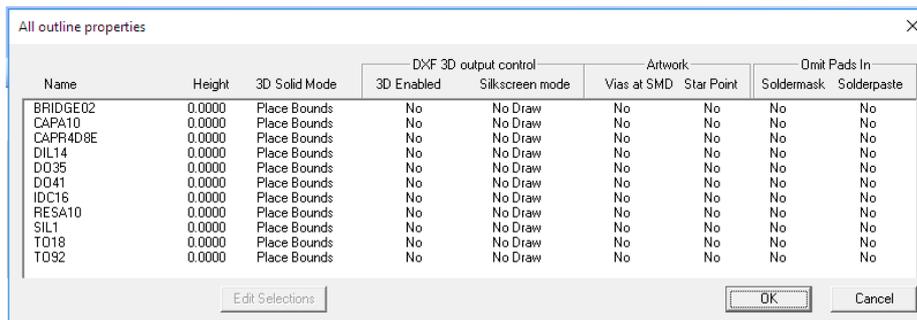


Figure 108

The actual fields (or attributes) are described under the *View > Properties* heading. Here we describe how to select the outlines.

Selecting an individual outline

select the outline with a click of the left button, it is highlighted.

Selecting a continuous group of outlines

position the cursor over the first outline name, then hold down the left mouse button, move the mouse over the outlines to be selected, then release the left button, they are highlighted.

Alternatively, select an outline, hold down the <shift> key, then select the last in the group of outlines, release the <shift> key, they are highlighted.

Selecting randomly spread outlines

hold down the <Ctrl> key on the keyboard whilst selecting individual outlines with a click of the left mouse button. Release the <Ctrl> key when all the outlines required are highlighted.

Editing the selections

Once the outline(s) have been selected, select the *Edit Selections* button from the window (greyed out until outlines have been selected). The *Outline Properties* window appears. Any changes to this window affects all the selected outlines. Select *OK* to update the properties list or *Cancel* to discard the changes.

Repeat as required.

Once all the outlines have been updated as required, select *OK* to implement the changes or *Cancel* to discard the changes.

Purge Unused Outlines

Used to delete all the unused outlines from the design's outline library.

Outlines are automatically added to the job as they are used. They can become redundant due to a design modification, or new outlines could have been created or copied by the user, which may not be required.

The parts list is used to determine which outlines are used in a design.

Once selected, a window appears listing all the unused outlines in the design. Select the outlines that should be deleted by ticking the box alongside them, then select *Delete*.

The *Select All & Deselect All* buttons can be used to quickly select/deselect all the outlines in the list.

Right-click on a component outline

Right-clicking on a Component Outline in the navigator window introduces the following commands:

Open Cut Copy Delete Rename

These commands are described below.

Open

When selected, the component outline is opened for editing. Changes are not implemented in the artwork until the outline is closed.

Cut

Currently (version 1.68) operates in the same way as Copy, though this may change in a future release – refer to the readme.txt in the ..\Seetrax\XL Designer folder.

Copy

The component outline(s) is/are copied to the paste buffer, overwriting anything previously added to the paste buffer. Multiple outlines can be selected if the *Shift/Ctrl* keys are used in unison with the right-click.

(To paste the outline from the paste buffer to any component outline folder, right-click on the appropriate component outline folder (which might be the same design, another design or the master component outline folder), then select *Paste* from the options that appear.)

Drag-drop functionality

In addition to the *Copy/Paste* command, “drag & drop” is available to *copy* component outlines between the master library and designs and/or from design to design. Select the outline(s) required, then drag them by holding down the left mouse key to the other component outline folder, releasing the button whilst hovering over the target component outline folder.

Delete

When selected, the component outline is deleted from the folder/library. It cannot be deleted if referred to in the parts list of the design.

Rename

Used to alter the name of a component outline. Once selected, the name is highlighted and the usual editing keys (backspace, delete, left/right arrows, etc.) can be used to rename it.

It is not possible to rename an outline that is referred to in the parts list editor.

View commands, outline editor

View > Show Outline Filled

When selected, the display toggles between filled pads and wide lines and unfilled pads and lines shown as centre lines. The editor starts with unfilled pads and centre lines displayed.

View > Show Pin Numbers

When selected, the pin number assigned to each pad is visible at the datum position of the pad. If the pin numbers have not been assigned, a “?” (questionmark) is displayed. The editor starts with pin numbers displayed.

View > Show Connection Points

When selected, a small mark appears for each pad indicating where tracks will be attached to the pad in the artwork editor. (This is always in the centre of the pad for standard pad shapes.)

View > Properties

Used to view and edit the properties of an individual component outline. The properties are used by the 3D output tools (DXF and IDF) and also the artwork editing and artwork solder mask/paste output tools. Each of the entries is described below.

(It is also possible to open this window for multiple outline editing, by right-clicking on the component outline folder, then selecting *Edit Properties*.)

When selected the window shown in Figure 109 appears.

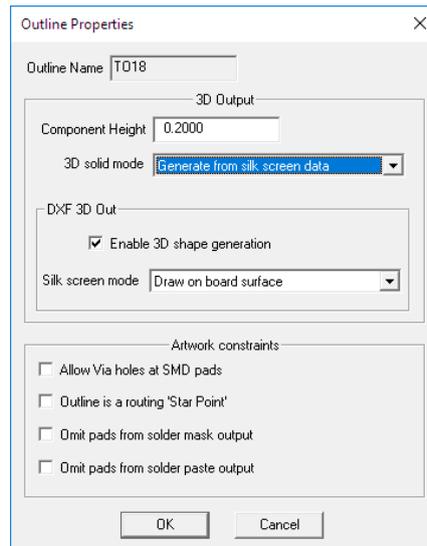


Figure 109

Outline name if just one outline is selected, then its name is indicated. (The name can only be changed by right-clicking on the component outline in the navigator and selecting *Rename*.
If more than one outline has been selected, then *Multiple selections* is shown.

3D Output

Component height: the height specified here will be used as the height for the outline if any of the 3D output tools are used.

It is suggested that the maximum height including tolerances is specified to ensure accuracy if the 3D solid models will be used to show clearances in enclosures, etc.

3D solid mode: This setting controls how the solid model is generated. It can be set as follows:

Generate from placement bounds box

if selected, then a simple rectangular block is shown at the part location in the 3D solid model. The rectangle is defined by the the *Autoplace > Adjust Footprint* command in the outline editor.

Generate from silk screen data

if selected, an attempt is made to construct a solid shape from the silk screen data. Not all silk screens will successfully convert to a solid figure – for example silk-screens that are not made from a continuous polygon - like that in the supplied SOT23. In these cases, it would be advisable to use the placement bounds box for the solid model as nothing would be output in the DXF file for that part if the silk screen data was selected.

In order for the silk screen mode to work, the silk-screen must have been generated in the artwork editor prior to the 3D file being output and the silk-screen layer must also be included in the 3D output.

DXF 3D Out

Enable 3D shape generation: if this setting is enabled (ticked) and an artwork output is made to a DXF file, then the DXF output task will attempt to create data for 3D solid figures representing the placed components.

The silk screen mode is greyed out until this setting is enabled (ticked),

Silk screen mode: this setting controls how the silk screen labels (IC1, R1, D1, etc.) are drawn for the part in the DXF output files when a 3D solid shape is also drawn for the corresponding part.

Draw raised to component height

the silk screen part labels are drawn on the top of the 3D solid model.

Draw on Board Surface

the silk screen part labels are drawn on the board surface.

Do not draw

the silk screen part labels and outline shapes are never drawn for parts that have a 3D solid figure shown.

Artwork constraints:

Allow via holes at SMD pads: this setting indicates whether via holes are permitted within an smd pad in the outline. This setting is only used by the Specctra & Electra auto-router and can be over-ruled when submitting the design for auto-routing.

Outline is a routing star point: when this checkbox is ticked, the outline takes on a special mode of behaviour that allows different nets to be connected at a single point with free copper, effectively causing a short-circuit or star-point.

The star point outline defines the track geometry for the location where the nets meet.

The check-box should be selected BEFORE editing the star point outline.

One component pin should be added to the star point outline for each net that will be connected at the star point.

The pads should then be connected together by inserting *free copper* on the top, bottom or all layers.

The free copper features that connect the star outline pins is never seen by the artwork autorouter, the copper fill routines or the artwork checker. The only time it is present is during screen display and when creating artwork outputs.

To prevent the autorouter and copper fill routines from placing copper in the area of the star outline occupied by the interconnecting free copper traces, a keepout area must be defined.

The keepout areas must not enclose any of the component pins on the outline.

Because the artwork checker does not verify the copper connections between the star outline pins, you must manually verify that your outline does actually perform the desired interconnectivity between its various pads.

An example job is supplied showing the use of a starpoint outline. A step-by-step procedure is also given at the end of this chapter.

Omit pads from solder mask output: if ticked, then the pads from this component outline will not be included when the automatic solder mask output task is used.

Omit pads from solder paste output: if ticked, then the pads from this component outline will not be included when the automatic solder paste output task is used.

Fill out the fields as required.

Note: if multiple selections have been made, then the change made to each field will apply to all the selected outlines. If a field is untouched, then no changes will be made to that particular field and the settings for each outline will remain as they were.

So for example, if all the outlines were selected, and only the *Silk-screen mode* were changed, then the various settings for height, 3D solid mode, etc. would remain unchanged, but all the silk-screen modes would be updated. Likewise if all the outlines were selected, and the height was altered, this height would apply to all the selected outlines.

Select *OK* to continue.

Dialogue bar settings, outline editor

Before creating or modifying outlines, notice should be taken of the settings in the dialogue bar, as shown in Figure 110.



Figure 110

These control the size, orientation and layers of the items being added or modified. The dialogue bar is divided into areas as follows:

Pad Style

This setting allows the pad shape to be selected. Selecting the arrow alongside produces a list of standard pad shapes plus any user-defined pads in the current pad library, from which one should be selected. If a rectangular shaped pad style is chosen the *Angle* setting should be set as required. If a standard pad shape is selected then the *Size* setting should also be updated as required.

Pad Angle

This setting controls the orientation of any pads that are added or updated. The angle is selected from the list that appears (0, 90, 180 or 270) when the arrow alongside the setting is selected, or a value may be

typed in to obtain non 90 degree angles. Resolution is 0.01 degrees.

When typing a value, ensure the cursor remains in the dialogue bar area.

Note: standard pad shapes added using the *Copper* commands can only be added at the standard 0/90/180/270 degree angles, but custom pads may be rotated freely.

Pad Size

The *Size* setting corresponds to the size codes from the Pad Sizes table so it controls the size of the pad. The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside. The Browser window displays the sizes assigned to each pad code as they are selected.

Pin Scope

used to indicate the currently active layer(s). Pads can only be added to, or selected from, the currently active layer(s).

Pads on each layer are displayed with different colours.

Top pads	red
Inner pads	green
Bottom pads	blue

A complete pad "stack", where all the pads are the same size and shape, will result in the pads being displayed white.

When pads are added to the outline, they are either added to the top, inner, bottom or all layers. This allows differently sized or shaped pads to be defined on different layers of the board. For example, pads on the inner layers of a board could be smaller than those on the outer layers, or pin 1 of a component could have a square pad on the solder side and round pads on the inner layers and component side of the board.

It is quicker to add the pads to all layers, and then modify the ones that need to be different, rather than to add pads on top of one another to the individual layers.

Only pad stacks with a pad on the **inner** layers are drilled, even though a drill size may be specified within the pads used on the top and bottom layers. This means that when defining component outlines with drilled pads, the pads must be added to **all** layers, even if the outline will only be used on single sided boards. The **inner** pad defines the drill size.

When defining surface mounted devices, add the pads to either the **top** or **bottom** layer, depending on which side the component is to be mounted on.

Tip: always define outlines from the same side, usually the top, then flip them if required in the artwork editor. (Ranger stores outlines as flipped or unflipped on the artwork, not whether they are on the top or bottom of the board.)

Pads only on the top, only on the bottom or only on the top and bottom of the board will not be drilled.

Top refers to the top, component side of the board.

Bottom refers to the solder side of the board, which is also the bottom component side if components are being mounted on both sides.

Inner refers to all the other copper layers in the design.

All includes all the copper layers (top, inner and bottom layers).

Next Pin Number

used to define the next pin number to be assigned to a pad. It is user definable. Up to eight alpha-numeric characters in any order can be used in a pin number.

The pin number can be changed by typing it in (the cursor must remain in the dialogue bar whilst typing).

Once the pin number has been assigned, the number in the box increments by one, unless 0 was entered, in which case it does not change. (This allows more than one pad to be re-numbered to 0, ready for subsequent re-numbering if incorrect pin numbers were assigned.)

If the pin number ends in an alpha character, then a number is added to the end of the string after the first pin has been assigned, and increments by one each time another pin is selected unless the string is already eight characters long.

If alpha-only strings are required, they have to be typed in individually.

Outline/Copper Width

This setting corresponds to the size codes from the Track Sizes table and it controls the width of any lines/characters/copper that are added/updated. The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside.

The actual size assigned to the code is shown above the code.

Free copper layer

Used to indicate the currently active layer(s) when adding/modifying pads/tracks/lines using the *Copper* commands.

Items added using the copper commands can only be selected if they are on the currently active free copper layer(s).

Items on these layers can be displayed with different colours and this is controlled by the *Copper > Free Copper Setup* command within this editor.

The layer is chosen from a list that appears if the arrow alongside the setting is selected.

If only the Top, Inner and Bottom layers are listed, then no free layers have been defined (*Copper > Free Copper Setup*).

The coloured box below the layer setting indicates the colour assigned to the chosen layer to assist with recognition.

Region Operations

This indicates which categories of data will be included in the *Region* commands.

- | | |
|--------------------|--|
| <i>Outlines</i> | when ticked will include any items added using the <i>Outline</i> commands (silk screen lines or text) in the window operations. |
| <i>Pins</i> | when ticked will include any items added using the <i>Pads</i> commands in the window operations. |
| <i>Free Copper</i> | when ticked will include any items added using the <i>Copper</i> commands in the window operations. |

Tools > Outline Definition

Once Tools > Slots & Extra Holes has been selected, use this command to revert back to the main Outline editor commands.

Tools > Slots & Extra Holes

This command provides access to the *Slots and Extra Holes* commands which allow routed slots and/or additional holes to be added to an individual outline.

Their use is identical to the same commands in the Custom Pad and Artwork editors. The commands are therefore described under the heading *Slots & Extra Holes (Tools > Slots & Extra Holes commands)* on page 241 of this manual.

Outline commands, outline editor

These commands are used to add the silk-screen definition to the outline. Once the silk-screen has been generated on the artwork (added to a layer), changes to the silk-screen outline in the outline library have no effect on the silk-screen layer - unless the layer is re-generated.

Outline > Add Line

Used to add lines that will become the silk-screen shape of the outline.

The thickness of the line is defined by the *Outline/Copper Width* setting in the dialogue bar on the left of the window. The number represents a line thickness taken from the track sizes table. Use the spin-controls to change the code number. The current size for the code number is shown above the code.

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected, locate the cursor where the line is to start then click the left button. Move the cursor, a line appears attached to it. Clicking the left button inserts corners in the line. Release the line with a click of the right button, the segment attached to the cursor is not added.

Outline > Add Circle

Used to add circles that will become the silk-screen shape of the outline.

The thickness of the line used to draw the circle is defined by the *Outline/Copper Width* setting in the dialogue bar on the left of the window. The number represents a line thickness taken from the track sizes table. Use the spin-controls to change the code number. The current size for the code number is shown above the code.

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected, point at the centre of the required circle and click the left button. Move the cursor and a circle appears attached to it. The size of the circle is controlled by the cursor position. Click the left button to release the circle in its current position. Clicking the right button cancels the circle.

Use the *Outline > Move Point* command to move or change the size of existing circles.

Outline > Add Arc

Used to add curved lines "arcs" which will become the silk-screen shape of the outline. A series of arcs and straight lines can be added using this command.

The thickness of the line used to draw the arcs is defined by the *Outline/Copper Width* setting in the dialogue bar on the left of the window. The number represents a line thickness taken from the track sizes table. Use the spin-controls to change the code number. The current size for the code number is shown above the code.

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected, locate the cursor where the arc or line is to start, then click the left button. Move the cursor, a line appears attached to it. Move the cursor to the point where the arc or line is to end, then click the left button. Move the cursor. A curve stretches with the cursor, its size and shape being dependent on the cursor position. Click the left button to release the curve. If the right button is clicked a straight line segment is introduced. The line remains attached to the cursor, allowing more curves to be added or the line released. To continue adding curves, move the line to the end point of the next arc, then click the left button, stretch the arc and release it with a click of the same button. To release the line, click the right button, the line attached to the cursor is not added.

If the right button is clicked whilst the arc is being stretched, a straight segment is introduced. This allows a mixture of curved and straight segments to be drawn using the same command.

The shape of the arcs can be adjusted using the *Outline > Move Point* command.

Outline > Corner

Used to move or insert a corner in a line or arc that has been added using the *Outline* commands (not those added with the *Copper* commands).

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected:

Adding a corner

Point at the line, then click the **right-hand** mouse button. Move the cursor, the new corner is attached to it and the adjacent segments stretch with it. Release the new corner with another click of the same mouse button. (Click the opposite mouse button (left) to cancel the new corner.)

Moving an existing corner

Point at the corner, then click the **left-hand** mouse button. Move the cursor to move the corner and stretch the adjacent segments. Release the corner with another click of the same mouse button. (Click the opposite mouse button (right) to cancel the move.)

add corner = **right** button

move corner = **left** button

(Opposite button cancels.)

Outline > Move Point

Used to move existing points (corners) in lines or arcs, to move circles, or to change the size of circles that have been added using the *Outline* commands (not those added with the *Copper* commands).

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected:

Moving points in lines or arcs

Locate the cursor on the corner to be moved, then click the left button. Move the cursor, the corner appears attached to it and the existing lines or curves stretch. Position the corner as required, and then click the left button to release it. Clicking the right button before the corner is released returns the corner to its original position.

Moving circles

Point at the **centre** of the circle, then click the left button. Move the cursor, the circle appears attached to it. Position the circle as required, and then click the left button to release it. Clicking the right button before the circle is released returns it to its original position.

Changing the size of circles

Point at the **circumference** of the circle, and then click the left button. As the cursor is moved, the diameter of the circle changes. Once the required circle size is achieved, click the left button to release it. Clicking the right button before the circle is released returns it to its original size.

Outline > Adjust Arc

Used to convert arcs to lines, lines to arcs and to adjust the size or shape of arcs that have been added using the *Outline* commands (not those added with the *Copper* commands).

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the

information bar.

Once selected:

Straight lines into arcs	Point at a line segment and click the left button. Move the cursor, the segment is replaced by an arc that stretches as the cursor is moved. Click the left button to release the arc.
Arcs into a straight line	Point at an arc and click the left button. Follow this with a click of the right button to convert the arc into a straight-line segment.
Adjusting size/shape of arcs	Point at the arc and click the left button. Move the cursor until the arc is in the required position, and then click the left button. Clicking the right button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically, and can be used to convert 45-degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Outline > Delete Feature

Used to delete lines, circles or arcs that have been added using the *Outline* commands (not those added with the *Copper* commands).

Note: the complete line or curve between start and end points is deleted, not just a segment within a line or arc.

Once selected, point at the line, arc or circle to be deleted, and then click the left button. The line, arc or circle is removed.

Outline > Delete Point

Used to delete points (corners) from lines or a series of arcs that have been added using the *Outline* commands (not those added with the *Copper* commands).

Once selected, point at the corner to be deleted, and then click the left button. The corner is removed from the line or arc.

Outline > Change Width

Used to change the line thickness of an existing line, arc, circle or text string that has been added using the *Outline* commands (not those added with the *Copper* commands).

Ensure the *Outline/Copper Width* setting in the dialogue bar on the left of the screen is set to the size code required. The number represents a line thickness taken from the track sizes table. Use the spin-controls to change the code number - the current size for the code is shown above the code.

Once selected, move the cursor over the line, arc, circle or text string then click the left-hand mouse button. The line width is changed to the size determined by the *Outline/Copper Width* setting in the dialogue bar.

Outline > Text

The text commands should only be used for standard details, such as pin numbers at each end of a connector.

Silk screen component references (R1, R2, etc.) are produced automatically for each part so should not be added, those references are placed on the datum of the outline. The references can be moved individually in the artwork editor.

Once the silk-screen has been generated on the artwork (added to a layer), changes to the silk-screen outline text in the outline library have no effect on the silk-screen layer - unless the layer is re-generated.

Outline > Text > Add

Used to add text strings to the outline that will appear as silk-screen text on the artwork. Text that is added to the outline will appear every time it is used, so do not add text that is unique to one particular part, for instance its name R1, R2, etc.

Text that is added to an outline is not mirrored when a part is flipped in the artwork editor.

The thickness of line used to "draw" the characters is defined by the *Outline/Copper Width* setting in the dialogue bar on the left of the screen, so ensure it is set to the size code required. (The number represents a line thickness taken from the track sizes table - use the spin-controls to change the code number, the current size for the code number is shown above the code.)

When adding small text characters, a thin line size should be used, otherwise the characters appear as blobs. Taller characters can be drawn with thicker lines.

Once selected a window appears. If text has already been added or modified, the previous text string appears in the window. It may be selected and modified, or a new text string entered.

The height of the text string is shown in the currently selected units (a decimal point is shown as a dot in imperial values and a comma in metric values) and can be changed as required.

The angle of text is specified in degrees, anti-clockwise, 0 degrees places text horizontally, reading from left to right. The angle can be typed in (0 to 359), or a selection made from the list that appears when the arrow

alongside the setting is selected.

Once all the settings have been filled in as required, select *OK*. The window closes and the text string, represented by a rectangle, appears attached to the end of the cursor.

Whilst the text is being moved it can be rotated in 90 degree increments by pressing the user defined special function key for <part rotate>.

Move the cursor into position and click the left-hand mouse button to release the text. Clicking the right-hand button before the text is released restores the window, allowing the string to be modified or the operation cancelled.

As the text string is released, another copy of it appears on the end of the cursor. It can be released in the same way. To cancel the text, select another command or click the right-hand mouse button to restore the window. *Cancel* can be selected to cancel further text addition, or the values can be modified and *OK* selected again.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Outline > Text > Move

Used to move text strings that have already been added.

Whilst the text is being moved, it can be rotated in 90 degree increments by pressing the user defined special function key for <part rotate>.

The text's datum is the lower left-hand corner of the rectangle that is displayed when it is moved, but it can be selected anywhere along its length.

Once selected, point at the text string and click the left-hand mouse button. A rectangle appears attached to the cursor, representing the text string. Move the rectangle into the required position and click the left-hand mouse button to release it.

Clicking the right-hand mouse button whilst moving the text returns it to its original position.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Outline > Text > Edit

Used to edit existing text strings or change their height or angle of rotation.

Once selected, point at the text string and click the left-hand mouse button. A window appears containing the text string, its height and angle of rotation. Refer to *Outline > Text > Add* for details of these entries.

Select *OK* when you are ready to implement the changes. Selecting *Cancel* restores the text to its original form.

Outline > Text > Mirror

Used to mirror text strings about their centre.

Text that is added to an outline is not mirrored when a part is flipped in the artwork editor.

Once selected, point at the text string to be mirrored and click the left-hand mouse button.

Outline > Text > Rotate

Used to rotate text strings through 90 degree increments, anti-clockwise. For instance, text that was added at 45 degrees rotates to 135, 225, 315 and back to 45 degrees.

Text can be rotated in increments of 1 degree using the *Outline > Text > Edit* command.

Text is selected anywhere along its length, but it rotates about its datum which is the lower left-hand corner of the rectangle that is displayed when the text is moved.

Once selected, point at the text string and click the left-hand mouse button to rotate it.

Outline > Text > Delete

Used to delete text strings from the outline.

Once selected, point at the text string you wish to delete and click the left-hand mouse button.

Outline > Set Outline Datum

Used to define the datum of the outline. The datum is displayed as a small yellow cross with a circle around it. In the outline editor the X-Y readout is given with respect to the datum's position.

When a part is moved or selected in the artwork editor, it must be selected by its datum.

Note: if the datum of an outline is moved in the design library and the outline is already in use on the artwork, the part will be "moved" on the artwork. The datum retains its position on the artwork, but the pins etc. will adjust their position so they are in the correct position with respect to the outline's datum. Silk screen and free layer data already generated will be unaffected.

Items that are being moved, added, etc. do not have to be released to allow the datum command to be selected.

Once selected, point at the position required for the datum, then click the left-hand mouse button. The datum moves. The datum command remains active until the right-hand mouse button is clicked when control returns to

the previously active command, or another command is selected.

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Pad commands, outline editor

The *Pad* commands are used to add pads to the outline and assign their pin numbers. They can be manipulated as required. Pads can be added in any order.

Information about the pads in an outline is given across the top of the open outline editor, as shown in Figure 111. This indicates how many pin numbers have been assigned (unique pin idents), the number of pads on the top, inner and bottom layers and if any of those are unassigned.

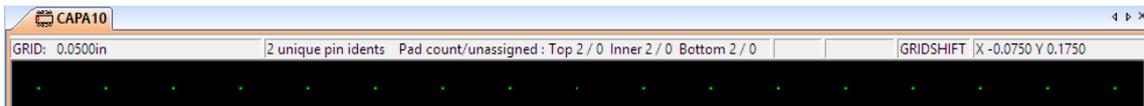


Figure 111

Default pin numbers are not given to pads. Pin numbers **must** be assigned to ensure the connections defined on the circuit diagram are connected to the correct pads on the outline. If pin numbers are not assigned to pads, the pads cannot be referred to.

Pin numbers can be omitted from an outline. As an example, one type of reed relay uses a 14-pin DIL package, but pins 3, 4, 5, 10, 11 and 12 do not exist. The outline can be made with 8 pads using pin numbers 1, 2, 6 to 9, 13 and 14.

Ensure the pin numbers are assigned to ALL the layers of each pad stack.

These commands cannot be used to modify pads added using the *Copper* commands, which should be modified with the *Copper* commands.

Pad > Add

Used to add individual pads to an outline. When pads are added, they do not have pin numbers, so the order in which they are added is unimportant.

The pad size/shape/layer can be controlled by the dialogue bar, so it should be set as required. As the pad shape/size/style is selected in the dialogue bar, the associated folder opens in the navigator.

Once selected, move the cursor into the main drawing area and a pad appears attached to it. Move the cursor and the pad into position and click the left-hand mouse button to release it. A second pad appears attached to the cursor. Move it into position and release it. As one pad is released, another appears and can be released. To stop adding pads click the right-hand button.

The datum point of a standard pad is its centre. The datum point of a library pad is indicated by a small pink circle and cross.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Alternative pad selection option

The pad shape and size can also be selected from the navigator window - from the navigator, open the standard pads folder of the current job, then select the + sign of the pad shape folder, select the pad size required from the navigator (the dialogue bar is updated to correspond, automatically).

The angle of the pad & layers it's added to, is controlled from the dialogue bar which should be set as required.

Pad > Auto Add

Used to add a series of pads. The position of the first and second pad is indicated, the additional pads are added automatically on the same pitch and continuing in the same direction. The total number of pads is user defined. Automatic pad addition is very useful when creating outlines with a lot of pads in a line, as in a pin grid array.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar box.

Refer to the *Pad > Add* command for details on selecting the pad size/shape.

Once selected, move the cursor into the main drawing area, the first pad appears attached to it. Move the pad into position and click the left-hand mouse button to release it. A second pad appears attached to the cursor. Move it into the position of the second pad in the line and release it. A window appears requesting the total number of pads required. Type in the *total* number of pads required followed by <enter> or select OK. The pads appear one after the other at the same pitch and following on in the same direction as the first two pads were added.

Pad > Move

Used to move existing pads interactively. This command cannot be used on pads added using the *Copper* commands.

Pads can also be moved by typing in their new co-ordinates if required using the *Pad > Key Move* command.

Only pads on the currently active layer as defined by the dialogue bar on the left of the screen are moved. *All* should be selected to move the complete pad stack.

If pin numbers have been assigned to a pad stack and the pads from individual layers are moved, the pads still retain the pin numbers assigned. This situation should be avoided because of the confusion that will entail.

Ensure the active layer is set as required. Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

The datum point of a standard pad is its centre. A small pink circle and cross indicate the datum point of a library pad.

Once selected, point at the pad's datum and click the left-hand mouse button, the pad moves with the cursor. Move the pad into position and click the same button again to release it.

Clicking the right-hand mouse button whilst the pad is attached to the cursor returns the pad to its original position.

Pad > Key Move

Used to move a pad/pad stack to a specific X and Y co-ordinate, or to find out the position of a pad stack.

This command cannot be used on pads added using the *Copper* commands.

The X and Y co-ordinates are given with respect to the datum of the outline, which can be moved using the *Outline > Set Outline Datum* command.

The complete padstack is moved, irrespective of the layer setting in the dialogue bar.

Once selected, point at the datum of the pad stack and select it with a click of the left-hand mouse button. A window appears containing the current X and Y position of the pad. The title box of the window also displays the size code or name of the pad. The co-ordinates are shown in the current units. A dot is used as a decimal point in imperial values and a comma in metric values. Enter the new values required for the pad, then select *OK* to implement the move. Select *Cancel* to cancel the move.

Pad > Rotate

Used to rotate existing pads/pad stacks in 90-degree increments, anti-clockwise.

This command cannot be used on pads added using the *Copper* commands.

Only pads on the currently active layer as defined by the dialogue bar on the left of the screen can be rotated. For example, with the *Pin Scope* set to *Inner*, only the inner pads of the pad stack are rotated.

Ensure the active layer is set as required. Once selected, point at the datum of the pad to be rotated and click the left-hand mouse button. The pad rotates 90 degrees anti-clockwise each time the pad is selected.

The datum point of a standard pad is its centre. The datum point of a library pad is indicated by a small pink circle and cross.

Pad > Change Style/Size

Used to replace an existing pad for the currently active pad. (Refer to *Pad > Add* for how to select the pad size/shape.)

This command cannot be used on pads added using the *Copper* commands.

The datum of the new pad is positioned directly over the datum of the pad it is replacing.

Only pads on the currently active layer as defined by the dialogue bar on the left of the screen are replaced. For example, with the *Pin Scope* set to *Inner*, only the inner pads of the pad stack are replaced.

Pin numbers are maintained if pads are replaced.

Ensure the active layer is set as required. Select *Pad > Change Style/Size*, as the cursor is moved into the main drawing area, the active pad appears attached to it. Move the cursor over the datum of the pad to be replaced and click the left-hand mouse. The pad is replaced. More pads can be replaced until another command, for instance *Pad > Move/Rotate*, etc. is selected.

The datum point of a standard pad is its centre. The datum point of a library pad is indicated by a small pink circle and cross.

Pad > Delete

Used to delete pads from the outline.

This command cannot be used on pads added using the *Copper* commands.

Any pad from the currently active layer as defined by the dialogue bar on the left of the screen can be deleted, even if it has been assigned a pin number. The pin numbers of remaining pads are unaffected. For instance, with the *Pin scope* set to *Bottom*, only the bottom pad of the pad stack is deleted.

Ensure the active layer is set as required. Once selected, point at the datum of the pad to be deleted and click the left-hand mouse button. Confirmation is not required.

The datum point of a standard pad is its centre. The datum point of a library pad is indicated by a small pink circle and cross.

Pad > Assign

Used to assign pin numbers individually to the pads. The pin number is controlled by the *Next Pin Number* setting from the dialogue bar on the left of the screen, which increments by one each time a number is assigned. This command cannot be used on pads added using the *Copper* commands.

If the *Next Pin Number* is set to 0, there is no auto-increment of the pin number. This allows multiple pins to have their pin numbers removed if required.

The same pin number cannot be used twice on the same layer. If a mistake is made and it is necessary to renumber the pins, the existing pin numbers will need to be reset to 0, using either this command or auto assign. Once selected, move the cursor and a flag appears attached to it, displaying the *Next Pin Number* as defined in the dialogue bar. Select a pad to assign that pin number to the pad. The flag increments by one to indicate the number has been assigned. (If the pin number doesn't change, the status bar may be displaying an error message or the pad may not be on the selected layer.

More pads can be selected as required. If a pad is selected again, then its original number is updated.

If the next pin number is not correct, type in the correct number in the *Next Pin Number* field and continue. Select another command to terminate manual assignment of pin numbers.

Use the *Pad > Describe* command to obtain pin numbering information on a particular pad.

Pad > Auto-Assign

Used to assign sequential pin numbers automatically to a line of pads. This command cannot be used on pads added using the *Copper* commands.

The first pin number is controlled by the *Next Pin Number* setting from the dialogue bar on the left of the screen.

If the *Next Pin Number* is set to 0, there is no auto-increment of the pin number. This allows multiple pins to have their pin numbers removed if required.

The first and last pads in a line are selected. Pin numbers are assigned sequentially from the first to the last pad.

Only pads on the active layer(s) as defined by the dialogue bar on the left of the screen are assigned.

Once selected, a "first" flag appears attached to the cursor. Select the first pad in the line to be assigned. A "last" flag appears attached to the cursor. Select the last pad in the line to be assigned. As soon as the first and last pads have been selected, the pin numbers are assigned - a flag indicates which pads and pin numbers have been assigned.

Use the *Pad > Describe* command to obtain pin numbering information on a particular pad.

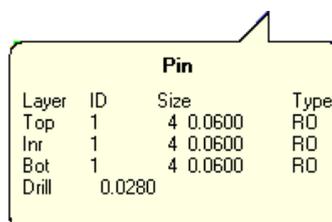
Pad > Describe

Used to obtain information about a pad stack. This command cannot be used on pads added using the *Copper* commands (use the *Copper > Describe Pad* command on those instead).

Use the *Pad > Key Move* command to find out the X and Y co-ordinates of pads.

Once selected, point at a pad and select it with a click of the left-hand mouse button. An information window appears. A line extends from one corner of the information window to the pad stack it is describing.

A sample window is shown in Figure 112.



Pin			
Layer	ID	Size	Type
Top	1	4 0.0600	RO
Inr	1	4 0.0600	RO
Bot	1	4 0.0600	RO
Drill		0.0280	

Figure 112

LYR: indicates the layers in the pad stack.

ID: indicates the pin number of the pad on that layer.

Size: indicates the size code and actual size of the pad on that layer.

TYPE: indicates and shape of the pad on that layer. Pad shapes are listed as:

RO = round, RS = rectangular, square-ended

SQ = square, RR = rectangular, round-ended

DRILL: indicates the drill size used in the pad stack. If the pad stack is made from pads with different drill sizes, a warning "*multiple drill sizes*" is given. If the outline is used in the artwork editor, the drill size is defined by the drill size of the **inner** pad.

Pads can be selected until the right-hand mouse button is clicked, which cancels the describe command.

Copper commands, outline editor

These commands are used to add additional copper to the outline in the form of tracks or pads. This allows stringers to be added to SMD's, dual outlines to be created, or specialist solder mask/solder paste layers to be defined. The dialogue bar on the left of the screen as described previously controls the size/shape of pads/tracks that are added.

More information also appears at the beginning of this chapter under the heading "**Additional "copper"/free copper - introduction**".

Notes:

An outline must contain one set of "pads" that were added and numbered using the *Pad* commands, which form the basic outline.

If a pad is added using the *Copper* commands, it is never given a pin number.

Copper pads (and tracks) can be placed on any of the following layers, Top, Bottom, All or on up to eight *free layers*.

The "*free layers*" are generally used in order to create specialist solder paste pad layers, solder mask pad layers, glue spot layers, etc. (Standard solder paste and solder mask output files can be produced automatically, with a uniform, user-defined pad swell. It is therefore not necessary to create these free layers unless there are specific requirements.)

They are called *free layers* because the data that is added to them can be added to any user-defined inner layer once the outline has been placed on the artwork, using the *Tools > Generate Silkscreen* command in the artwork editor. These layers are treated as silk-screen layers, they do not form part of the electrical connectivity and are not checked during artwork checking.

Tracks and pads that are added to "free layers" can be placed anywhere.

The pads and tracks on the free layers do not appear automatically when the outlines are used in a layout.

Tracks added to the top, bottom or "all" layers can only be added if the track starts from a pad on the same layer, which was added using the *Pad* commands.

Pads being added to the top, bottom or all layers can only be added on to the end of a piece of track that is also on the same layer.

Only pads or tracks added to the top, bottom or all layers have electrical significance, i.e. they form part of the copper on the layout and are checked during artwork checking.

Creating "dual" outlines:

Dual outlines are created when components are available in different packages, and it is unknown at the layout stage which package will be available when the board is assembled.

Here we describe how to add a surface mounted outline (SO14) to a conventional outline (DIL14). (Refer to individual commands for more details.) Copy the supplied DIL14. With the copied outline open, decide which layer the secondary pads have to be added to (typically top) and add a track to that layer, from one of the original numbered pads to the position where the secondary SMD pad will be added (*Copper > Add Line*). These tracks will form the link between the DIL14 pads and the SO14 pads. Repeat for each pad. Choose the size and shape of the secondary pad and place a pad on the end of each of the tracks (*Copper > Add Pad*) on the appropriate layer. The dual outline is complete.

Note, rip-up, layer swap, etc. can be used on the tracks in the artwork editor, but the pad remains, as it is not regarded as a via, unless it was added as a size code 0 round or square pad.

Adding "copper" to the "free layers"

Select *Copper > Free Copper Setup*. Assign names and different colours to the layers if this has not already been done.

"Tracks" or "pads" can be added to the "free layers". If adding a pad, select the pad to be added, set the dialogue bar to suit the pad/track width to be added, the *Free Copper Layer* selection will list all the layers available. If a name was not entered in the *Free Copper Setup* window, then no extra layers will appear. Select the layer required and add the pads or tracks required. Remember that "copper" added to the free layers is not treated as copper by Ranger.

When the outlines appear on the layout, their free layer copper is not automatically used or displayed. It can be added using the *Tools > Generate Silkscreen* command.

Copper > Add Line

Used to add lines to the outline (not silk-screen lines which are usually added using the *Outline* commands).

The line may or may not be copper, depending upon which layer it is added to.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen controls the layer the line is added to. Top refers to the top, component side of the board (T), bottom refers to the solder side of the board (B). ALL includes all the copper layers, i.e. top, inner copper and bottom layers. The other layers are "free" layers and are not defined as copper layers.

Lines added to the free layers can be placed anywhere.

If the line/track is added to the top, bottom or all layers, the line must start on a pad added using the *Pad* commands, which is also on the same layer. For instance, if the pad stack is on all layers, then the line can be added to any layer, but if the pad is on the top layer only, then the line must also be on the top layer.

The *Outline/Copper Width* setting in the dialogue window on the left of the screen, controls the line size, which should be set as required.

Ensure the line size and layer is set as required, then select *Copper > Add Line*. Point at:

the pad where the track is to start if it is being added to the top, bottom or all layers, or

the position required if it is being added to a free layer

and click the left-hand mouse button.

Move the cursor, and a track appears attached to it. Click the left-hand mouse button to release the track from the cursor. It is not possible to add corners to this track whilst it is being added. Use the *Copper > Add/Move Corner* command to modify the track.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

It is not possible to start more than one track from a pad.

Copper > Add/Move Corner

Used to add and move existing corners in lines added using the *Copper* commands.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the line is on.

It is possible to disconnect tracks on the top, bottom or all layers from their pads, but this should be avoided. If the track is moved, the pin should also be moved.

Corners are **added** and released with clicks of the **right-hand** mouse button. Corners are **moved** and released with clicks of the **left-hand** mouse button. Once the corner has been selected, clicking the opposite mouse button cancels the operation.

right button = **add** corner

left button = **move** corner

Opposite button cancels

Adding a corner to a track:

Select *Copper > Add/Move Corner*, then point at the track and click the right-hand mouse button. Move the new corner and release it with another click of the same (right) button. Clicking the opposite (left) hand button before the corner is released, cancels the new corner.

Moving an existing corner in a track:

Select *Copper > Add/Move Corner*, then point at the corner (or end of the track) and click the left-hand mouse button. Move the corner and release it with another click of the same (left) button. Clicking the opposite (right) hand button before the corner is released, cancels the move.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Copper > Add Keepout Line

Used to add keepout areas to the outline. The copper fill and auto-routers will avoid keepout areas. The checking routines will flag violations of the keepout lines/areas if requested.

A closed shape should be drawn defining the required keepout area.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen controls which layer the keepout line operates on, but keepout lines/areas can only be added to the Top, Bottom or All (copper) layers.

The command is greyed out if one of the free copper layers is selected.

Ensure the *Free Copper Layers* setting is set to Top, Bottom or All as required, then select *Copper > Add Keepout Line*. Locate the cursor where the line is to start then click the left button. Move the cursor, a line appears attached to it. Clicking the left button inserts corners in the line, which is displayed dotted when released. Release the line so that an enclosed shape is formed with a click of the right button, the segment attached to the cursor is not added.

Use the *Copper > Add/Move Corner* command to modify the line.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Copper > Add Keepout Circle

Used to add circular keepout areas to the outline. The copper fill and auto-routers will avoid keepout areas. The checking routines will flag violations of the keepout lines/areas if requested.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen controls which layer the keepout circle operates on, but keepout lines/areas can only be added to the Top, Bottom or All (copper) layers.

The command is greyed out if one of the free copper layers is selected.

Ensure the *Free Copper Layers* setting is set to Top, Bottom or All as required, then select *Copper > Add Keepout Circle*. Locate the cursor on the centre of the new circle then click the left button. Move the cursor, a circle appears attached to it. Clicking the left button releases the circle, which is displayed dotted when released.

Use the *Copper > Adjust Arc/Circle* command to move or resize the circle.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Copper > Delete Point

Used to remove points (corners) from lines added using the *Copper* commands.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the line is on.

Select *Copper > Delete Point*, move the cursor over a point (corner) in the track and click the left-hand mouse button. The point is removed. If the start or end point of a track is deleted, the segment is removed.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Copper > Delete Line

Used to delete a line added using the *Copper* commands.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the line is on.

A complete line is deleted between its start and end points, not just a segment between points.

Pads on the top, bottom or all layers should not be left isolated if the track attached to them is removed. Either a new track should be added or the pad removed.

Select *Copper > Delete Line*, point at the line to be deleted and click the left-hand mouse button.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Copper > Adjust Arc/Circle

Used to convert keepout arcs to straight segments, keepout line segments to arcs, to adjust the size or shape of keepout arcs and to move or resize a keepout circle that have been added using the *Outline* commands (not those added with the *Copper* commands).

Refer to the *Grid* command to find out whether grid snapping is active. The current grid is shown in the information bar.

Once selected:

- | | |
|---------------------------------|--|
| Straight keepout lines to arcs | Point at a keepout line segment and click the left button. Move the cursor, the segment is replaced by an arc that stretches as the cursor is moved. Click the left button to release the arc. |
| Keepout Arcs to straight lines | Point at a keepout arc and click the left button. Follow this with a click of the right button to convert the arc into a straight-line segment. |
| Adjusting size/shape of ko arcs | Point at the arc and click the left button. Move the cursor until the arc is in the required position, and then click the left button.
Clicking the right button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically, and can be used to convert 45-degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed. |
| Moving/adjusting ko circles | To move a keepout circle, select the centre of the circle with a click of the left mouse button, move the circle and release with a further click of the left button.
To resize the keepout circle, select the circumference of the circle with a click of the left mouse button, move the cursor and release with a further click of the left button. |

Once the circle is selected, clicking the right button cancels the change.

Copper > Add Pad

Used to add pads to an outline.

The pad shape/size/angle is selected from the dialogue bar which should be set as required. (Refer to the *Pad > Add* command for more details on pad selection.)

The *Free Copper Layers* setting in the dialogue bar controls the layer the pads are added to. Top refers to the top, component side of the board (T), bottom refers to the solder side of the board (B). ALL includes all the copper layers, i.e. top, inner copper and bottom layers. Other layers listed are user-defined free layers.

Pads added to the top, bottom or all layers must be attached to a track on the same layer - the track has to be added first. If an empty space is selected, or a track from a different layer is selected, the pad is not released. Pads added to the free layers can be placed anywhere.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Ensure the pad shape/size and layer(s) are set as required.

Select *Copper > Add Pad*. Move the cursor into the main drawing area and a pad appears attached to it. Move the cursor and the pad into position and click the left-hand mouse button to release it.

A second pad appears attached to the cursor. Move it into position and release it. As one pad is released, another appears and can be released. Click the right-hand button to remove the pad from the cursor. A further click of the left button will restore the pad.

Copper > Move Pad

Used to move pads that were added using the *Copper* commands.

Although it is possible to move pads away from their copper tracks on the layers top, all and bottom, they should become re-attached to the track before the outline is used.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the pad is on.

Refer to the GRID command for details on whether grid snapping is active. The current grid is shown in the information bar.

Ensure the layer is set as required.

Select *Copper > Move Pad*. Point at the pad's datum and click the left-hand mouse button, the pad moves with the cursor. Move the pad into position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the pad is attached to the cursor returns the pad to its original position.

Copper > Key Move Pad

Used to move a pad that was added using the *Copper* commands to a specific X and Y co-ordinate, or to find out the position of a pad.

The X and Y co-ordinates are given with respect to the datum of the outline, which can be moved using the *Outline > Set Outline Datum* command.

Once selected, point at the datum of the pad and select it with a click of the left-hand mouse button. A window appears containing the current X and Y position of the pad. The co-ordinates are shown in the current units. A dot is used as a decimal point in imperial values and a comma in metric values. Enter the new values required for the pad, then select *OK* to implement the move. Select *Cancel* to cancel the move.

Copper > Rotate Pad

Used to rotate existing pads added using the *Copper* commands in 90 degree increments, anti-clockwise.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the pad is on.

Ensure the layer is set as required. Select *Copper > Rotate Pad*, point at the datum of the pad to be rotated and click the left-hand mouse button. The pad rotates 90 degrees anti-clockwise each time it is selected.

User-defined pads can be rotated in increments of one degree if required. Select a user-defined pad with a click of the right-hand mouse button. A window appears and the angle of rotation can be entered. Subsequent selections of that pad with the left-hand mouse button rotates it in 90 degree increments.

Copper > Change Pad Style/Size

Used to replace an existing pad added using the *Copper* commands, for the currently active pad. (Refer to *Pad > Add* for how to select the pad.)

The datum of the new pad is positioned directly over the datum of the pad it is replacing.

Only pads on the currently active layer as defined by the *Free Copper Layer* in the dialogue bar are replaced.

Ensure the active layer is set as required. Select *Copper > Change Pad Style/Size*, as the cursor is moved into

the main drawing area, the active pad appears attached to it. Move the cursor over the datum of the pad to be replaced and click the left-hand mouse. The pad is replaced. More pads can be replaced until another command, for instance *Pad > Move/Rotate*, etc. is selected.

The datum point of a standard pad is its centre. The datum point of a library pad is indicated by a small pink circle and cross.

Copper > Describe Pad

Used to identify the size/shape of free copper pads on the currently selected free copper layer.

First, select the layer required from the *Pads* dialogue bar (circled in Figure 113), then select the *Copper > Pad Describe* command, then select the *copper* pad with a click of the left-hand mouse button. An information balloon appears pointing to the selected pad.

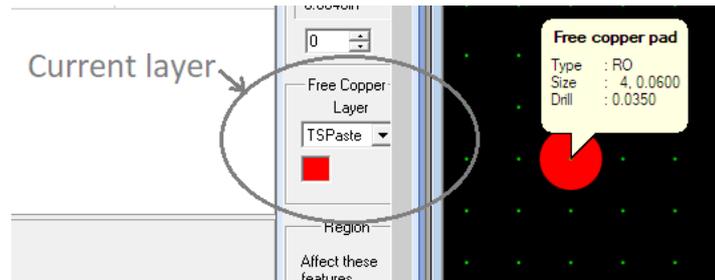


Figure 113

Type: indicates and shape of the pad on that layer. Pad shapes are listed as:

RO = round, RS = rectangular, square-ended

SQ = square, RR = rectangular, round-ended

Size: indicates the size code and actual size of the pad on that layer.

Drill: indicates the drill size defined in the pad (only copper pads on "All" layers actually get drilled)

This command cannot be used on pads/padstacks added using the *Pad* commands (use the *Pad > Describe* command on those instead).

Use the *Copper > Key Move Pad* command to find out the X and Y co-ordinates of *copper* pads.

Copper > Delete Pad

Used to delete pads from the outline that were added using the *Copper* commands.

The *Free Copper Layers* setting in the dialogue bar on the left of the screen should be set to the layer the pad is on.

Ensure the layer is set as required. Select *Copper > Delete Pad*, point at the datum of the pad to be deleted and click the left-hand mouse button. Confirmation is not required.

Copper > Free Copper Setup

Used to assign names and colours to the eight free layers that are available within the outline editor. The free layers are used to create user-defined solder paste footprints, solder mask footprints, glue spot footprints, etc., that can then be used in the artwork editor if required.

(Ranger produces solder paste and solder mask output files automatically, which output the used pads with a uniform user-defined pad swell. It is therefore unnecessary to create these layers unless you have specific requirements.)

Pads and/or tracks can be added to any of the eight free layers within the outline editor, using the *Copper* commands. The data is not automatically used in the layout. Once the outlines have been used on the layout, the data on the free layers can replace the inner pads on selected inner layers of the layout. These layers automatically become assigned as silkscreen layers, so they have no electrical significance to the layout.

The eight "free layers" do not have names assigned to them automatically and they are all set to the same colour. It is therefore sensible to assign a name and colour so that they can be differentiated from one another.

Names can be typed in the vacant name fields or existing names changed.

To select a colour for the layer, select the arrow alongside the existing colour and choose from the list. Select *OK* to close the window and implement the changes.

Region commands, outline editor

These commands are used to define a rectangular area around a group of items in order to move, copy, rotate or delete them.

The *Outlines*, *Pins* & *Free Copper* settings in the *Outline* dialogue bar (shown in Figure 114) determine which categories of data are included in the area.

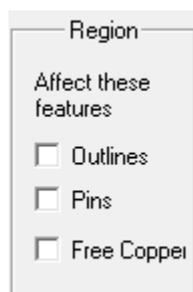


Figure 114

If no features are included, then an error message will be displayed in the status bar if an attempt is made to move/rotate/copy/delete an area using the Region commands.

When the rectangular area is defined, the first two left-mouse button clicks will define the area to be operated on. The second click determines the datum point of the area.

Pads that are partially within the rectangle are only treated as being inside the rectangle, if their datum is within the rectangle.

Lines (copper or silkscreen) that are not wholly within the rectangle are excluded from the *Copy* and *Delete* commands, but included in the *Move* and *Rotate* commands.

Text that is not wholly within the rectangle is excluded from all the *Region* commands.

The usual Grid snapping rules apply.

Region > Move/Rotate

Used to move and/or rotate data within a defined rectangular area.

Ensure the required features are enabled as required, Figure 114 – a message will appear in the status bar if nothing is enabled within the defined area when the move/rotate is attempted.

Select *Region > Move*. Click the left mouse button to locate one corner of the rectangular area to be moved/rotated. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and move with the cursor.

Each press of the rotate function key will rotate the rectangle anti-clockwise by 90 degrees.

Click the left mouse button again to release the area in its new position. A right-click will cancel the move/rotate.

Region > Copy

Used to copy data within a defined rectangular area.

The area can be rotated whilst being copied using the rotate function key.

If pads with pin numbers are copied, then the copied pads have their pin number allocation re-set to 0.

Ensure the required features are enabled as required, Figure 114 - a message will appear in the status bar if nothing is enabled within the defined area when the copy is attempted.

Select *Region > Copy*. Click the left mouse button to locate one corner of the rectangular area to be copied. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and move with the cursor.

Each press of the rotate function key will rotate the rectangle anti-clockwise by 90 degrees.

Click the left mouse button again to copy and release the area in its new position. A right-click will cancel the move/rotate.

Region > Delete

Used to delete data within a defined rectangular area.

Ensure the required features are enabled as required, Figure 114 – a message will appear in the status bar if nothing is enabled within the defined area when the delete is attempted.

Select *Region > Delete*, click the left mouse button to locate one corner of the rectangular area to be deleted. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and a prompt will indicate that a left click will delete the area or a right-click will cancel the operation.

Autoplace commands, outline editor

Autoplace > Adjust Footprint

Used to define the space required by an outline when placed on the artwork.

The automatic placement routine uses this boundary when determining how close to one another parts can be

placed. It will not overlap boundaries.

The artwork checking routines can also be used to indicate any silk-screen text located within this boundary.

The boundary can also be a useful manual placement aid to ensure parts are not positioned too close to one another, as quite often silk-screen outlines are smaller than the actual parts.

When part EMC info is displayed in the artwork editor, the area defined by this command is shaded in with the colours assigned to that particular part type (noisy/quiet/susceptible).

The boundary area is rectangular. Its minimum size is pad dependent and always encloses all the pads, but it can be made larger.

The footprint can be displayed in the artwork editor, but it is never plotted.

When pads are added to the outline, this boundary is automatically extended to enclose the pads being added.

The corners of the bounds can be moved by selecting them with a click of the left-hand mouse button, moving the cursor, and then releasing them with a second click of the left-hand mouse button. The rectangle always encloses all the pads, even if the corners are released inside the pad area.

Clicking the right-hand mouse button or selecting another command cancels the command.

Creating and using Starpoints

A sample design called *Starpoints Example.rxl* is included with the software and can be found in the ..\XLDesigner\SampleDesigns\PCB folder.

It's recommended that the example be studied before/whilst creating a starpoint.

It is assumed the user is familiar with the general use of XL Designer – refer to the Getting Started Guide for a worked example if required.

What is a Star point?

A star point is a common term used to indicate a location where multiple nets (usually power supplies) meet at a known location on the artwork.

In order to allow this "short-circuit" to occur without errors being flagged, a special component outline has to be used.

The special star-point outline has one component pin for each different net that will be joined together. These pins are then joined together within the outline, using *free copper*. In a standard outline this would be identified as a short-circuit but the "starpoint" outlines are handled differently. Because of this, the star-point outline has to be created carefully because it by-passes the usual automated checks.

To ensure the auto-routers and copper fill routines do not violate the added free copper, a keepout area should surround it (but not the component pads).

A typical star point component outline for a 3 net star-point is shown in Figure 115. (Silk-screen and autoplacement footprint are not shown to aid clarity.

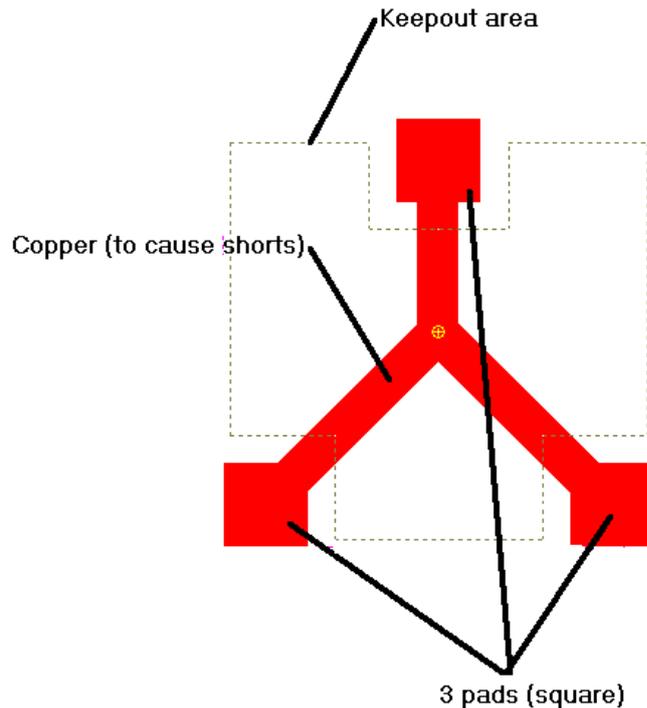


Figure 115

Creating a starpoint outline

Create a new outline.

Select *View > Properties* and tick the *Outline is a Routing Star Point* checkbox.

One component pad should be added to the star point outline for each different net that will be connected at the star point (*Pad > Add*).

The pads may be placed on the *Top*, *Bottom* or *All* layers and any pad style may be used.

The pads should be numbered (*Pad > Assign*).

The pads should be interconnected by inserting *free copper* on the *Top*, *Bottom* or *All* layers (*Copper > Add Line*).

Because the artwork checker does not verify the copper connections between the star outline pins, you **must** manually verify that your outline does actually perform the desired interconnectivity between the pads.

The autorouter, copper fill routine and the artwork checker never see the free copper features that interconnect the star point outline pads. To prevent the autorouter and copper fill routines from placing copper in the area of the star outline occupied by the free copper, a keepout area must be defined within the starpoint outline.

A keepout area should be added to the outline using the *Copper > Add Keepout Line* command. The keepout area must not enclose the pads in the outline - see Figure 115.

By using a keepout to prevent nets from violating the star interconnections, there is no need for the artwork checker to process the star interconnections and shorts will not be detected between those copper lines in the outline.

Using the starpoint outline

A schematic part that references the star point outline should be created, and one terminal placed for each pin on the star point outline. The terminal/pins should be numbered as for any normal part and correspond to the numbers in the starpoint outline.

On the schematic, each net that must meet at the starpoint must be added separately, and if named, each must have a unique name. Each net must then be connected to one of the pins on the starpoint part symbol.

Within the artwork, the star point part should be placed at the desired location, and the board may be routed, copper filled and checked as normal.

Logos Folder

This folder provides access to the logos editor which facilitates the insertion of symbols such as company logos, compliance and warning symbols etc. into PCB artwork layers.

The symbol can be generated from a photo or scanned image. The image has to be loaded into the system's logo editor, and then converted into a vector based representation suitable for inclusion into the artwork.

Images may be imported in JPEG, JPG, GIF, BMP or PNG format.

Note: For customers installing XL Designer for the first time, a few example logos may be found in the MASTERS/Logos library that was installed automatically.

The example master logo library WILL NOT get installed for existing users who have updated their system from earlier versions as the installer will never interfere with the content of user library directories after first-time setup. For those users, some example logos may be found in the "Logo Examples.rxl" design in the samples directory (normally 'C:\Program Files (x86)\Seetrix\XL Designer\SampleDesigns') and dragged from that design into the master logos folder if you wish to use them.

Important Notes:

The boundary of each symbol is represented by a rectangular area. This rectangular area is used during artwork checking to identify the extents of the symbol. Artwork checking only considers logos to be rectangular boxes, it does not check for clearance to each individual vector within the logo.

Logos may be placed on powerplane layers, and will be visible within the powerplane edit mode. When a power plane is generated/regenerated, the logos will be preserved within the layer.

Logos that are placed in powerplane layers are currently invisible to the artwork checker. No checks for violation with other powerplane features will be performed.

Creating the logo

Right-click on the *Logos* folder, then select *New* from the menu that appears.

Copying logos

Once a logo has been created, it/they may be copied between designs and to/from the master logo folder by dragging them within the navigator pane or right clicking and choosing *Copy*.

To **Paste** the copied logo to another folder, right-click on the destination *Logos* folder, then select *Paste*.

Deleting logos

Right click the logo(s) and choose *Delete*.

Renaming logos

Right click the logo(s) and choose *Rename*.

Selecting multiple Logos

Multiple logos can be selected using the Shift or Ctrl keys, followed by a right-click/copy/delete/etc.

Editing/opening a Logo

Once a logo has been created, it should be opened for editing. Once the editor is open, the desired image can be imported and converted into a vector based version, ready for use on the artwork.

Double-click on the logo name from the Logo folder and the logo editor will open. The editor is divided into three editing areas, each selectable by tabs at the top, as shown in Figure 116, where the *Select Source Image* tab is selected.

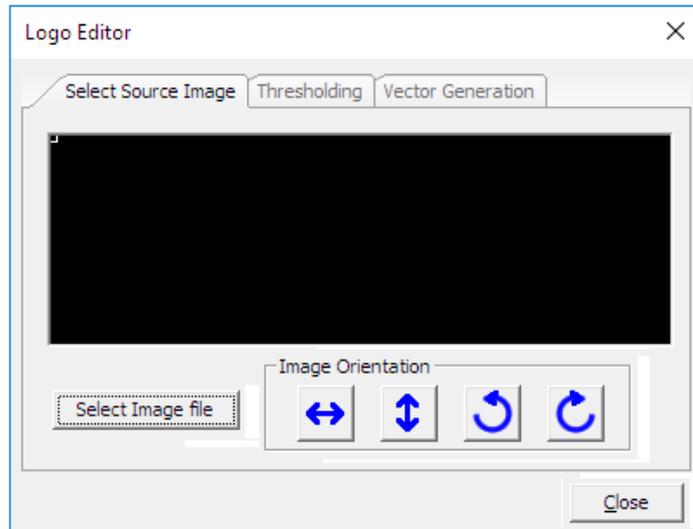


Figure 116

When the logo is initially created, only the *Select Source Image* tab is selectable until an image has been imported.

Use the *Select Image File* button to choose and import an image file containing a logo that will be used on a PCB artwork.

After selecting an image file, the image should display in the window, as shown in Figure 117.

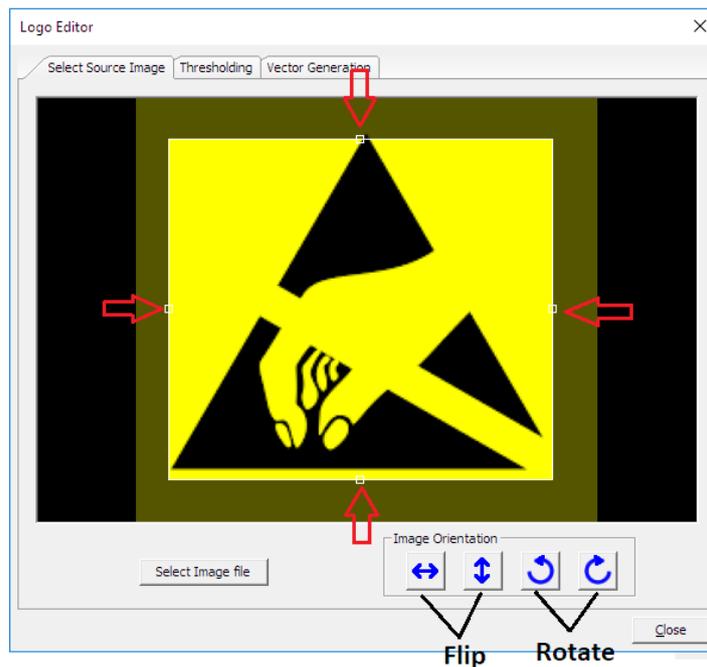


Figure 117

Use the *Flip* and *Rotate* buttons (indicated in Figure 117) to obtain the desired orientation of the image.

Near the edges of the image, border lines will be seen, with small selector squares at their centre, indicated by the red arrows in Figure 117.

Use these small squares to adjust the image size/shape, by clicking and holding the left mouse button on them, then dragging the border lines to select just the part of the image that will be converted into a logo.

The unselected parts of the image will go dim as you move the borders.

Once the image has been flipped/rotated/cropped as required, click on the *Thresholding* tab at the top of the window.

Thresholding

This page is divided into two areas as shown in Figure 118.

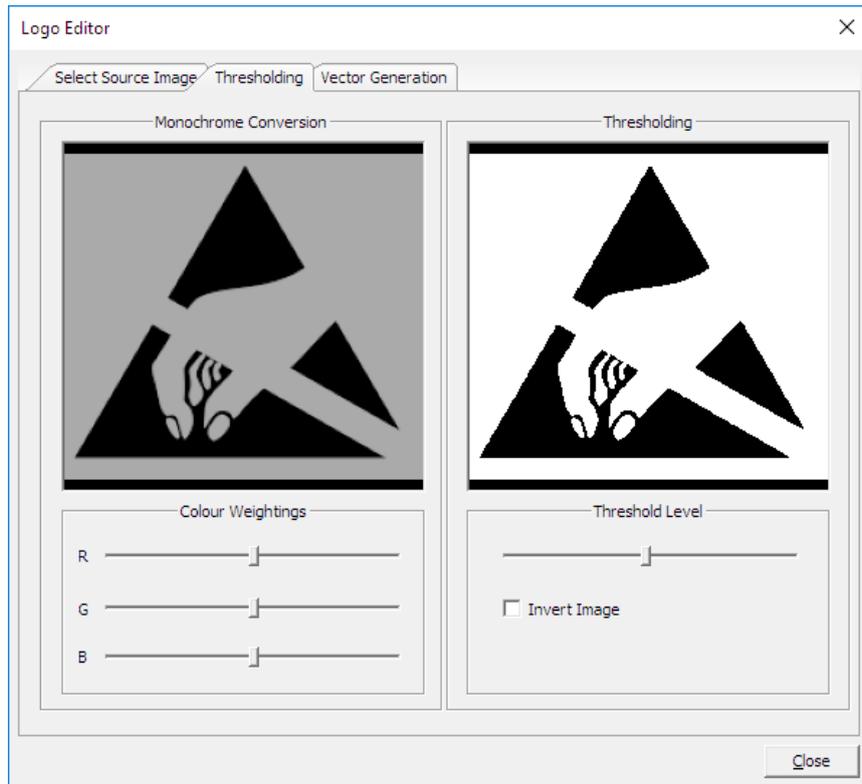


Figure 118

The left hand area shows a monochrome rendering of the original image.

If the original image is in colour, the *Colour Weightings* slider bars for Red, Green and Blue can be used to adjust the priority that each colour has in the conversion of the colour image to monochrome. With some images, this can be helpful to obtain a suitably contrasty image to threshold into pure black/white.

The right hand area is where the monochrome image is converted to black & white only representation.

The *Threshold Level* slider controls the brightness point in the image where the changeover takes place.

The *Invert Image* checkbox switches the sense of the resultant image. White in the right hand image gets converted into copper on the PCB.

Once a good image has been achieved in the right hand pane, select the *Vector Generation* tab which is shown in Figure 119.

Vector Generation

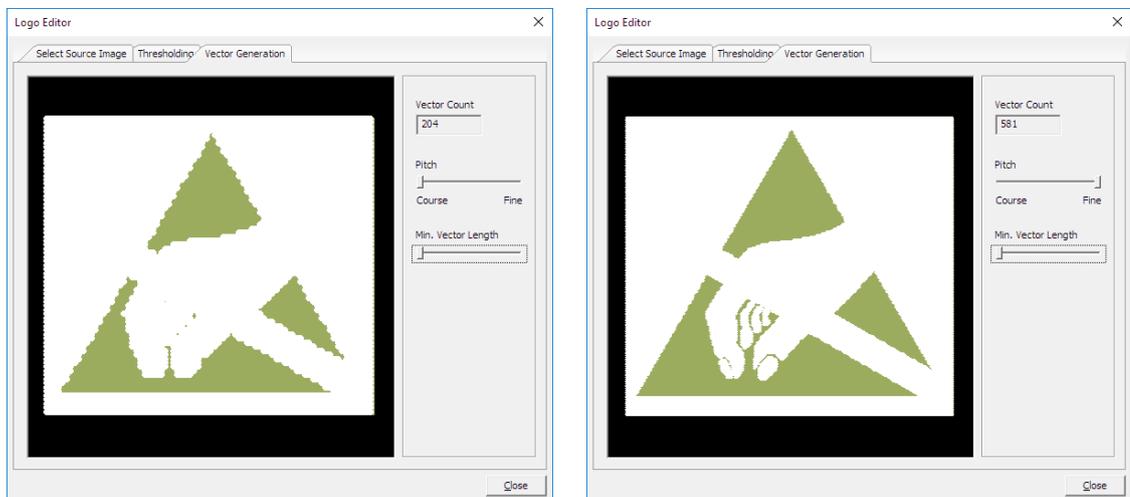


Figure 119

This window shows how the logo will look when it is split into individual vectors, the before and after adjustment views are shown side by side.

The *Pitch* control provides control of the fineness of the resultant logo.

The *Minimum Vector Length* control can be used to remove small unwanted spots that sometimes occur as a result of not being able to find a perfect setting for the thresholding level.

In this window, the drab green colour represents the laminate colour of the pcb, and the white is copper.

The boundary of the symbol is represented by the rectangular area shown. This rectangular area is used during artwork checking to identify the extents of the symbol. Artwork checking only considers logos to be rectangular boxes, it does not check for clearance to each individual vector within the logo.

Once an acceptable logo is displayed in the *Vector Generation* page, the Logo editor can be closed and the Logo is ready for use within the artwork editor. Refer to the Logo commands in the artwork editor (page 286) for details on how to use the Logos.

Parts & Nets List Folders

In order to design a PCB artwork, XL Designer requires a parts list and a net list. These lists can be generated automatically from a schematic diagram drawn in XLD, typed in using XLD's own parts and nets list editor or imported from text files.

It is also possible to create a parts/net list that has been back-annotated from the artwork editor using the Tools > Network commands.

Once the artwork is complete, checking routines are supplied to ensure that the artwork connectivity matches the parts/nets list, however it was generated.

Wherever they have come from, the parts/nets lists can be viewed, and edited if required, using the parts/nets list editing facilities. These facilities can also be used to access the cross-probing tools.

In addition to being able to type in a parts/nets list, the parts/nets list editors are also a useful tool for checking parts and nets lists that have been extracted from a schematic or imported, prior to board design

The editors will highlight outlines that do not exist or invalid nodes, supply a list of unconnected pins or identify "single-node nets" (connections that start but do not connect to anything else).

Note: if the parts/nets list has been extracted from the schematic editor and it needs to be changed, then the changes should be made to the schematic and the parts/nets list re-extracted, otherwise the two will become out of step with one another.

Any changes made to the parts/nets list using the parts/wiring list editors will be lost when the parts/nets list is extracted from the schematic.

Viewing the list of parts or nets

Expand a design's folder in the navigator window, then expand the **Parts** or **Nets** folders to list the parts and nets for the design as shown in Figure 120.

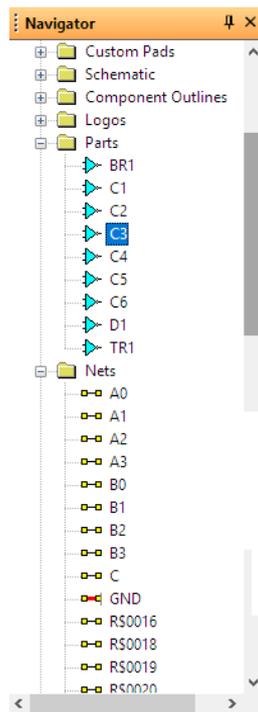


Figure 120

When the Parts or Nets folders are selected in the navigator pane, the Properties pane shows a count of the placed/unplaced parts in the design, or the number of power and signal nets in the design, depending on which folder is selected.

Double-clicking on any part or net from the navigator list opens the editing window – this displays the complete parts or net list – both windows are shown in Figure 121. These windows can be used to edit any of the entries if appropriate.

(If the parts or net list is empty, then the selected editor is opened so that entries may be typed in if required.)

Part	Type	Outline	Code	MCAD Model	X	Y	Rot
BR1	BRIDGE02,W01M	BRIDGE02			78,740	62,230	0.00
C1	CAPA10,0.1uF	CAPA10			85,090	80,010	270.00
C2	CAPA10,4.7nF	CAPA10			92,710	80,010	270.00
C3	CAPA10,1.8uF	CAPA10			106,680	62,230	90.00
C4	CAPA10,4.7nF	CAPA10			88,900	62,230	270.00
C5	CAPR4D8E,220uF	CAPR4D8E			101,600	46,990	180.00
C6	CAPR4D8E,47uF	CAPR4D8E			85,090	49,530	0.00
D1	BZX83C,BZX83C	DO35			101,600	78,740	270.00
D2	1N4002,1N4002	D041			106,680	78,740	270.00

Name	UID Code	MinClear	Size								
	R0016			IC3.3	IC2.5	IC3.5					
	R0017			PL1.5	TP5.1	IC4.8					
	R0018			C1.1	PL1.4	R1.1	C3.1	C2.1			
	R0019			C2.2	R2.1	R1.2	C4.1				
VCC	R0051			PL1.1	IC3.14	IC2.14	IC4.14	IC5.14	IC6.14		
	R0020			PL1.6	R3.2	BR1.2					
	R0021			R4.2	TR1.3						
	R0022			C3.2	R3.1						
GND	R0050			PL1.16	IC1.1	C6.2	IC2.7	IC3.7	IC4.7	IC5.7	IC6.7
	R0023			BR1.4	C4.2	C1.2	R2.2	PL1.7			
	R0024			TR1.2	D2.1						

Figure 121

If the windows contain invalid entries, they are highlighted in the window so they are easy to identify. Figure 122 shows an outline that is highlighted in the parts list, caused by a typing mistake - the outline in the library is called DO41 whilst D041 has been called up. This results in problems in the associated net list – because the outline D041 doesn't exist, neither do the pin numbers that it refers to.

Part	Type	Outline	Code	MCAD Model	X	Y	Rot
BR1	BRIDGE02,W01M	BRIDGE02			78,740	62,230	0.00
C1	CAPA10,0.1uF	CAPA10			85,090	80,010	270.00
C2	CAPA10,4.7nF	CAPA10			92,710	80,010	270.00
C3	CAPA10,1.8uF	CAPA10			106,680	62,230	90.00
C4	CAPA10,4.7nF	CAPA10			88,900	62,230	270.00
C5	CAPR4D8E,220uF	CAPR4D8E			101,600	46,990	180.00
C6	CAPR4D8E,47uF	CAPR4D8E			85,090	49,530	0.00
D1	BZX83C,BZX83C	DO35			101,600	78,740	270.00
D2	1N4002,1N4002	D041			106,680	78,740	270.00

Name	UID Code	MinClear	Size								
	R0020			PL1.6	R3.2	BR1.2					
	R0021			R4.2	TR1.3						
	R0022			C3.2	R3.1						
GND	R0050			PL1.16	IC1.1	C6.2	IC2.7	IC3.7	IC4.7	IC5.7	IC6.7
	R0023			BR1.4	C4.2	C1.2	R2.2	PL1.7			
	R0024			TR1.2	D2.1						
	R0025			D2.2	IC1.3	D1.2	BR1.3	C5.2			
V+	R0026			IC1.2	TR1.1	BR1.1	PL1.3	C5.1	D1.1	R4.1	C6.1
B3	R0027			PL1.15	IC6.9	IC5.9					
B2	R0028			PL1.14	IC6.1	IC5.1					
B1	R0029			PL1.13	IC3.9	IC2.9					

Figure 122

Simply scroll/page through the parts and net list windows looking for invalid (highlighted) entries. Invalid entries should always be investigated and corrected or problems will occur later in the design. Typical invalid entries and solutions to them are given later in this chapter. The parts list, lists all the parts that have been added to the design, along with associated information in

columns. Each part appears on one line.

The columns can be sorted in ascending/descending order by clicking on the column headers.

The net list lists all the connections or “nets” that have been added between the parts in the design, along with associated information in columns.

A net will have a minimum of two nodes, the start and end pins. There is no maximum number of nodes in a net, so nets may occupy more than one line, in which case the word *Contd->* appears in the *Name* column to indicate the line has been continued from the previous line.

Power rails can be treated differently by XLD on the artwork design - if a signal name has been defined as a power rail (*Configuration > Power Names*), the name is shown in a different colour in the net list to indicate it is a power rail – VCC & GND in Figure 122.

Deleting the parts list

Cross-probing tools

Once the list of parts or nets is displayed in the navigator window, right-clicking the mouse over any of the parts or nets will introduce the *Find in schematic* or *Find in artwork* commands as shown in Figure 123.

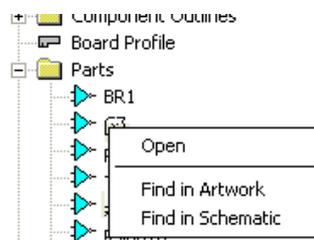


Figure 123

When selected, the appropriate editor (schematic or artwork) is opened/popped forward and the item indicated by a balloon containing information about it., Figure 124.

Icons within the balloon allow further selections to be made, for example if a multi-element schematic part is located, “arrow” icons in the balloon allow the next occurrence of the part to be found on the schematic, or the item to be located in the other (artwork/schematic) editor.

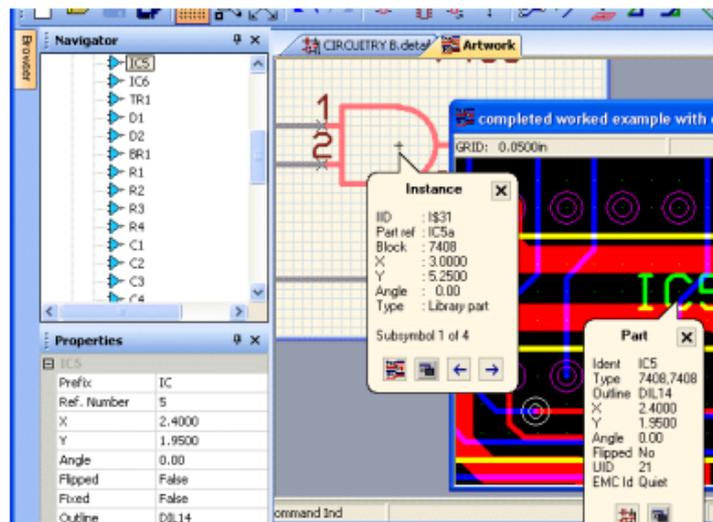


Figure 124

Making Nets visible/invisible

Whilst working in the artwork editor, it's possible to make all signal and all power unrouted (connections) visible/invisible independently of one another (View Control dialogue bar). However, it can also be useful to hide or view specific nets within each of those general categories.

Displaying particular nets (unroutes)

The View Control dialogue bar in the artwork editor has a setting that allows specific nets to be made visible (Figure 145),

Which nets are specified is controlled from the expanded Nets list folder (Figure 125) - right-clicking any of the nets will introduce the commands, also shown in Figure 125.

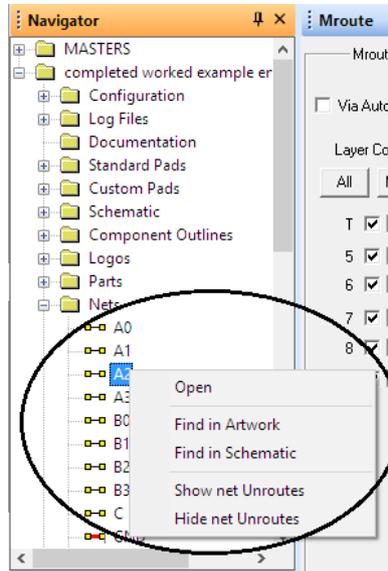


Figure 125

Once *Show net unroutes* is selected, the net will have "(s)" inserted alongside it in the expanded Nets folder (Figure 126) to indicate it will be "shown" when the Nets unroutes are selected for display in the artwork editor.

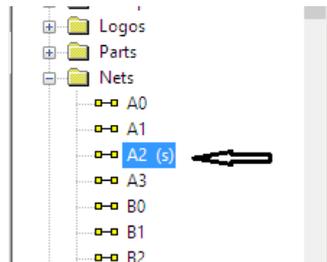


Figure 126

To restore its status, right-click the net again and select "Show Net Unroutes" to toggle the setting off. Multiple nets can be selected using the Shift and Ctrl keys before right-clicking.

Hiding nets (unroutes)

Which nets are hidden is controlled from the expanded Nets list in the navigator window. Right-clicking any of the nets will introduce the commands shown in Figure 125.

Multiple nets can be selected using the Shift and Ctrl keys before right-clicking.

Select *Hide Net Unroutes* to hide the selected net when the general category of unroutes (signal or power) are made visible in the artwork editor (View Control dialogue bar).

The net will have "(h)" inserted alongside it in the expanded Nets folder to indicate it is hidden - Figure 127.

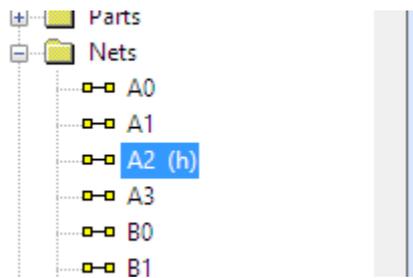


Figure 127

The hidden status is restored to visible when the design is closed so the net(s) are not forgotten!

To restore its non-hidden status whilst working, right-click the net again and select “Hide Net Unroutes” to toggle the setting off.

Typing in a parts list

With the job folder open in the navigator, either double-select the *Parts* folder or select it with a right-click of the mouse, then select *Open* (Figure 128).

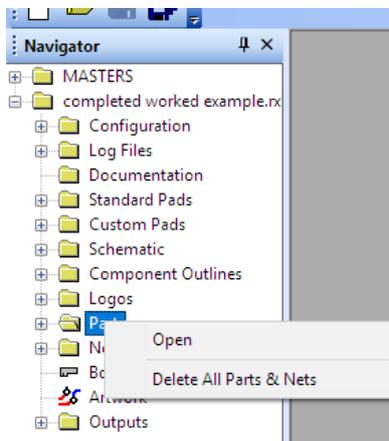


Figure 128

A window similar to the one shown in Figure 129 appears. The parts that are required on the design must appear in this window. The window is divided into fields consisting of *UID*, *Part*, *Type*, *Outline*, *Code*, *X*, *Y* and *Rotation* fields. They are described later under the heading *Parts list fields*.

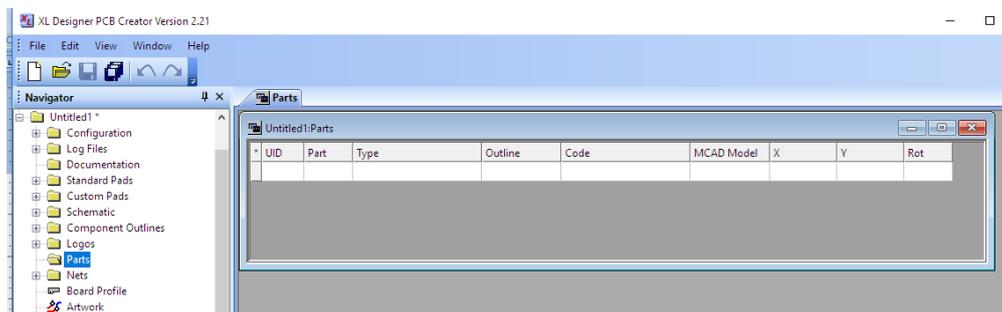


Figure 129

The **Part** and **Outline** fields must have an entry, the other fields are optional so can be left blank or at their default settings.

Double-select a field to introduce the text-entry cursor. Once a value has been typed, use the <tab> key or the arrow keys to move between fields.

Start by typing in a **Part**, for example **IC1**, then press <tab> to move to the next field. The X, Y and Rot fields are inserted automatically.

The **Type** field is optional but it makes the link to the schematic parts that will be required if gate and pin swapping will be used in the artwork editor.

If a *type* is not required, press <tab> to skip to the *Outline* field.

The **Outline** field must contain the name of the component outline that defines the physical representation of the part. Press <tab> once the name has been typed. If the outline doesn't exist in the design's component outline library it is automatically copied into it from the master component outline library.

If the outline cannot be found in either location, the field will get highlighted to indicate it must be changed or an outline with that name supplied in the design's outline library.

Dragging/dropping outline names

Outline names can be dragged & dropped into the outline field to save typing. Ensure the *Component Outline* folder is open in the navigator (either the design's own or the master outline folder), then scroll through the list until the outline required is located – each one can be displayed in the Properties & Browser windows (if visible) if selected with a click of the left-mouse button. Once the outline has been located, position the cursor over the outline in the folder, hold-down the left mouse button and drag the cursor over to the Outline field in the parts list. The name will be inserted.

The **Code** and **MCAD Model** fields are optional (see below for details).

The **X, Y** and **Rotation** fields are updated as parts are moved on the board. (The co-ordinates can only be entered or modified after selecting *Edit > Allow Position Edit* command from the command bar.)

Continue to <tab> or use the arrow keys until the cursor appears in the parts field of the next line and type in the next part.

Modifying the parts list

If a spelling-type mistake is made, double-select the string to be edited, the text entry cursor appears and the changes required made.

A field can be deleted by selecting the string so that it is highlighted, then pressing the <Delete> key.

To delete the complete part (line), select the *Part* field and delete it, the complete entry (line) is removed.

Multiple entries in columns can be selected by holding down the left mouse button and sliding the cursor over them - once highlighted they can be deleted using the <Delete> key..

Repeating parts

Select the part name field or complete line of an existing part, then select **Edit > Repeat part** from the command bar. Enter the details required in the window that appears. The new parts are added.

Parts list fields

- | | |
|-------------|--|
| UID | UID stands for <i>unique identification number</i> . This field is automatically assigned by SXLD when a part is added and cannot be edited.
This column can be hidden/revealed using the <i>View > Show Part UIDs</i> command. |
| Part | This is the name the user allocates as a unique reference for each component in the design. It must comprise of a prefix and number, for instance IC1, R1, etc.
The prefix used must be present in the part prefix table, if it isn't, then a window will appear requesting that the prefix be added to the table. If it is not added, the part will be removed from the parts list.
Maximum of 4 + 4 characters (four for the prefix and four for the ident). i.e. PLUG1 through to PLUG9999). |
| Type | This field is accessed by the gate and pin swapping routines. It is used as the link between the physical package (the outline), and its schematic library part.
If gate/pin swapping will not be performed on the part in the artwork editor, this field could contain anything that would individually identify it, i.e. its stock number, a catalogue reference, or it could be left blank.
Even if a schematic has not been drawn (the parts/net list has been typed in or imported), if gate or pin swapping will be performed on the part in the artwork, then the design must contain all the schematic parts representing each of the parts that will be swapped. The schematic part(s) must have the appropriate equivalent gates &/or pins defined.
Gate or pin swapping cannot be performed in the artwork if the associated schematic parts do not exist.
For gate swapping purposes the <i>Type</i> field must comprise the schematic library part name (maximum 16 characters) and the schematic library part value field (maximum 16 characters), separated by a comma. Maximum of 33 characters in total.
For example the schematic part might be called: RPK100K whilst the value of the resistors might be 100k. The <i>Type</i> field in the parts list would be: RPK100K, 100k
Gate swapping between packages is only permitted if the <i>Type</i> fields of the packages are |

identical.

If the parts list has been extracted from a schematic, then the type field is made up from the schematic library part name and the value attribute field from the schematic part's symbol attributes, separated by a comma.

Outline

This field identifies the physical package from the component outline library to be used for the part on the artwork. Maximum 16 characters.

If the outline does not exist in the design's component outline library, the outline is automatically copied from the master component outline library into the design's component outline library.

If the outline does not exist in either place, it will be highlighted to indicate the outline does not exist. The highlighting remains until a valid outline name is entered, or the outline is added to the design's component outline library.

If the parts list has been extracted from a schematic, then the outline name is initially obtained from the *Outline* attribute field in the schematic part's symbol attributes. It is possible to change the outline used on the artwork from the parts list editor or from the artwork editor, so the outline name in the parts list and that defined in the schematic part may be different. (When the parts/net list is extracted from the schematic again, it depends on the extractor settings as to whether the outline is updated or not.)

If an outline is changed in the parts list and the part has already been placed on the board, the change to the outline is immediate, but it does not affect routed tracks, silk-screen data, power-planes etc. which would have to be updated in the artwork editor.

Code

This field can be used as required, for a company's own stock record number, a specification reference, a catalogue reference, or anything which identifies it. Maximum 16 characters.

The information is not currently used by SXLD, but may be accessed along with the rest of the parts list by external application programs, if it is sent to a file using the *File > Save As Text* command.

If the parts list has been extracted from a schematic, then the code is obtained from the *Order code* attribute field in the schematic part's symbol attributes.

MCAD Model

Values entered in this column can be used as the 3D model/geometry library name by the IDF Output Task. It can be left blank.

If not used, this column can be hidden/revealed using the *View > Show MCAD Model Name* command.

X and Y

These columns contain the position of the part's datum on the artwork with respect to absolute 0,0 (lower left corner of working area) in the artwork editor (which is not necessarily the position of the movable datum point).

The units are shown in inches or millimetres, depending on the units selected (*Edit > Units*). Decimal points are shown as dots if inches are in use, and commas if mm's are in use. The special function keys for Inch/Metric mode can be used to toggle the units.

It is usual to leave these columns set to 0 and position the parts interactively. These columns are updated automatically when parts are moved in the artwork editor.

Some components such as connectors, switches, etc. have to be positioned precisely, in which case their co-ordinates could be entered.

To edit these fields, select *Edit > Allow Position Edit* from the command bar.

If the parts and net lists were created from a schematic, re-extracting the schematic will not change these fields, unless the part has been removed. Do not delete the parts and net list before extracting the parts/net list, as all the X/Y values will be re-set to 0,0 which means all the parts will become unplaced on the artwork.

Rot

This column indicates the orientation and layer of the parts on the board. It is usual to leave this column set to 0 and rotate or flip the parts interactively. The column is updated automatically when parts are rotated or flipped in the artwork editor.

To edit this field, *Edit > Allow Position Edit* should be enabled from the command bar.

An entry of 0 represents the part as it was created in the component outline library. Rotation is specified in degrees, anti-clockwise.

If **F** (for flipped) precedes the entry, the part is flipped (mirrored) about its original Y axis.

When entering the data for flipped parts, enter it as one string, i.e. type **F45** if the part is required flipped and rotated through 45 degrees; the entry will appear as **F 45**.

If the parts and net lists were created from a schematic, re-extracting the schematic will not change these fields, unless the part has been removed. Do not delete the parts and net list before extracting the parts/net list, as all the Rot values will be re-set to 0.

Outlines are highlighted - typical causes of invalid entries in parts list editor

Highlighted outlines will often result in highlighted entries in the net list as well - correcting the invalid outline may also clear up the net list, so always start with correcting the invalid outlines!

This can be caused by a spelling mistake or because the outline does not yet exist in the component outline library.

Correcting spelling mistakes

Look at the outline that's highlighted - is it a spelling/typing mistake? The name must correspond to an outline held in the design's outline library.

If it's simply that the outline hasn't been created or copied to the design's outline library yet, as soon as the outline is created/copied then the highlighting will disappear.

If the spelling is incorrect, how it is corrected depends on where the parts list originated from.

Parts list compiled from a schematic:

For example IC's 1 to 6 which are all 7400's, may call up DIL1, instead of DIL14.

If so, return to the 7400 schematic part, and edit the part's symbol attributes and correct the outline name. This will affect all of the parts (IC's 1 to 6) on the schematic sheet. To alter it so that only IC4 were affected, then open the schematic sheet containing IC4 and edit its attributes.

In both cases, extract the parts/net list again, ensuring the extraction setup window is set to update outline fields.

It is also possible to correct the name in the parts list editor by selecting it, then re-typing it, or dragging/dropping the outline from the outline folder. Be aware that the next time the parts list is extracted from the schematic, this outline may revert back to that called up by the part – depending on the settings in the parts/netlist extractor.

Typed in parts list::

Correct the name by selecting the entry and re-typing it, or drag an outline from the design or master outline library.

Imported parts list:

How this is corrected depends on what may happen in the future. The easiest and quickest way where only a couple of outlines are involved is to edit the entry as per the instructions for a typed in parts list.

However, if many outlines are involved, or it's likely that modifications will be performed on the design in the future by importing an updated parts list, it would be sensible to return to the source of the parts list and correct it there, then import the corrected parts list. This means that if modifications are made to the "updated original" parts list in the future and the new file imported, the outline changes won't be lost.

Once the spelling has been corrected (and the outline exists) the highlighting will disappear.

Correcting non-existent outlines

If it is believed that the outline name is spelt correctly in the parts list, then it's likely the outline doesn't exist in the component outline library (or the outline name itself has been mis-spelt).

The outline will need to be created/copied/renamed in the design outline library at which time the highlighting in the parts list will disappear.

Typing in a net list

The net list cannot be typed in until a parts list exists. The inter-connections between the parts in the parts list should be entered in the net list.

With the job folder open in the navigator, either double-select the *Nets* folder or select it with a right-click of the mouse, then select *Open*. The window shown in Figure 130 appears. The window is divided into columns entitled *Name*, *UID code*, *Min clear*, *Size* and some untitled columns - they are described later under the heading *Net list fields*.

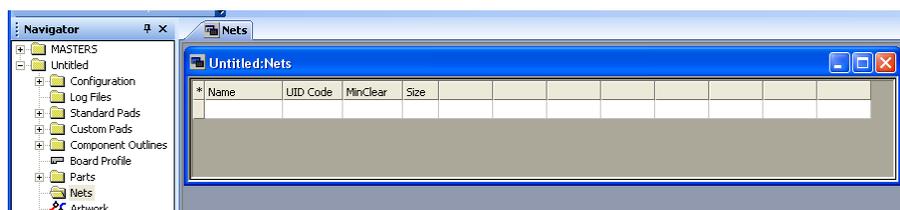


Figure 130

The columns will increase their width automatically as required.

The *Name*, *UID code*, *Min clear* and *Size* fields may be left blank and a default value will be inserted automatically if one is required. The untitled columns are where the nodes in a net should be typed/appear.

All the nodes in one row are connected together. If the net has more nodes than can be fitted in the row, they are continued on the next line and the word *Contd->* appears in the *Name* field to indicate the net has been continued.

Double-select a field to introduce the text-entry cursor. If a signal name, minimum clearance or size is not being specified, select the field in the first untitled column as shown in Figure 131.

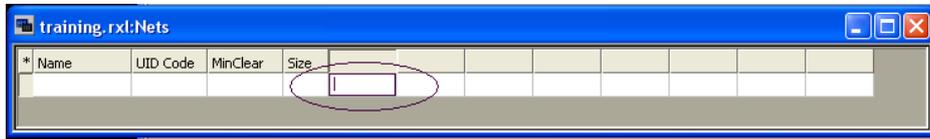


Figure 131

Once a value has been typed, use the <tab> or the arrow keys to move between fields.

If the net is not complete once the last column is reached, continue to use the <tab> key, this will create a new line with the word *Contd->* in the *Name* column.

The same pin number cannot be used more than once - the entry will have to be re-typed, but it will allow invalid entries, for instance referring to parts or pins that don't exist. The invalid entry will remain highlighted until the entry is changed or the appropriate changes are made elsewhere.

Modifying the wiring list

If mistakes are made the entry can be edited or removed.

To delete an entry, select the entry once, then press the <delete> key. If a mid-node is deleted, the remaining nodes move back so that an empty space does not remain.

To edit an entry, double-select the entry, the text entry cursor appears and the string can be modified.

If an extra node is required, select the next empty node field on the end of the net it will be added to. If there are no empty fields on the end of an existing row, double-select the last occupied node, then press the <tab> key so that a *continued* line is inserted.

When entering signal names, if they are defined as power rails they will highlight so that they can be differentiated from the signal connections (see below).

Net list fields

- Name** This field is reserved for optional signal names, or continuation characters. (Do not enter the first node of a signal in this column, since it will be used as a signal name.) Maximum of 16 characters.
- When a name is entered, a comparison is made against the names held in the power names table (case insensitive). If a match is detected, then the case of the entered name is changed to suit the case of the name in the power names table, and the net becomes a power rail.
- (Once a power name has been defined it is not possible to have a signal name with the same characters, but different case. This is to avoid confusion, for instance having a power connection called **GND** and a signal connection called **gnd**.)
- When signal names defined as power rails are entered, the signal name highlights. If the name does not highlight, it is not being recognised as a power rail.
- UID code** UID stands for *unique identification number*. This field displays the unique identity number that is automatically assigned to the net - each net has its own code so that unnamed nets can be more easily identified in the reports that SXLD may produce, such as artwork checking results. This value cannot be edited.
- The .dsn file that is created by the Spectra interface uses the UID codes for identification of unnamed nets.
- This column can be hidden/revealed using the *View > Show Net UID Codes* command.
- MinClear** This field can be used to assign a specific minimum clearance for a net. If a value is not entered, then the minimum clearance requirement for the net is as specified by the on-line DRC and artwork checking routines.
- The units are shown in inches or millimetres, depending on the units selected (*Edit > Units*). Decimal points are shown as dots if inches are in use, and commas if mm's are in use. The special function keys for Inch/Metric mode can be used to toggle the units.
- If the net list has been extracted from a schematic, the default minimum clearance sizes as specified by the on-line DRC and artwork checking routines is used unless a specific minimum

clearance was assigned to the connection on the schematic.

This column can be hidden/revealed using the *View > Show Minimum Clearances* command.

Size This field can be used to assign a specific track width code to a signal. Valid codes are 0 to 511 and . (dot). If a size is not entered, then "." is inserted automatically.

The . (dot) indicates the default track codes should be used for signal and power connections. The default codes are defined in the design's *Configuration* folder in the *Manual Routing Parameters* window. The actual sizes are defined in the same folder in the *Sizes table* window.

If the net list has been extracted from a schematic, the default track width codes are used unless a specific code was assigned to the connection on the schematic.

Nodes The unnamed fields are used for *nodes* in a connection. Nodes should be typed in as:

part reference.pin number

For example: IC3.3 R1.2 TR3.E

The part reference must appear in the parts list.

The pin number can have a maximum of 8 alpha-numeric characters in any order. They should correspond to the pin numbers in the associated component outline as defined in the parts list.

When typing in part references that use the first entry in the Part Prefix Code table, it is not necessary to type in the prefix. For example, if **IC** is the first entry in the part prefix table, and IC4.2 has to be typed, simply type **4.2**, the IC is added automatically. (The order of the part prefix code table should not be adjusted to take advantage of this facility once the parts list has been started or the existing part prefixes in the parts/net list will be altered as well.)

Nodes on the same line are connected to one another, but not to nodes on the next line unless the word **Contd->** appears in the signal name column of the next line.

Nodes are highlighted - typical causes of invalid entries in net list editor

Invalid entries are typically produced because the component outline has not yet been defined, is named incorrectly, pin numbers on the schematic part have not been assigned correctly, or some pins in an outline are missing or have not been numbered.

To identify the cause:

Look at the pin numbers that are highlighted. For example: IC7.16 Open the parts list and check that the outline for IC7 is not highlighted – if it is, correct this first as that may be what is causing the pin number(s) to highlight.

If the outline isn't highlighted, then the outline does exist so it's a problem related to the pin number.

Correcting Pin 0's

For example IC1.0, R12.0, etc., should never appear in the net list. How this is corrected depends upon where the net list has come from, as follows:

Net list extracted from a schematic:

If the pin numbers are set to 0, then it's likely that the schematic part's pins will not have had pin numbers assigned.

Take a note of the part name, for instance IC1, R12, etc., then open the parts list. Locate the part and take a note of the *type* field - the name before the comma in the type is the schematic part name, which is what you require.

Open the schematic part, use the *Terminals > Check* command to help indicate any problems. Assign the pin numbers correctly. Use the *Terminals > Check* command again to ensure no further problems were introduced.

The part(s) on the schematic will need to be de-allocated, then re-allocated for the change to be implemented. Extract the parts/net list again. Check the parts/net list for highlighted entries.

Typed in wiring list:

Correct the entry by selecting the entry and re-typing it.

Imported wiring list:

How this is corrected depends on what may happen in the future. The easiest and quickest way where only a couple of pins are involved is to edit the entry as per the instructions for a typed in wiring list.

However, if many pins are involved, or it's likely that modifications will be performed on the design in the future by importing an updated net list, it would be wise to return to the source of the net list and correct it there, then import the corrected net list. This means that if modifications are made to the "updated original" net list in the future and the new file imported, the pin number changes won't be lost.

Correcting non-pin 0's

If the pin numbers appear to be correct, then it's likely that the pads in the component outline have not had pin numbers assigned, are numbered incorrectly or there are not enough numbered pads in the outline for the pins used on the schematic part.

If the associated outline in the parts list is also highlighted, then the outline doesn't correspond to one in the outline library.

Take a note of the part name, for instance IC43, R56, etc., then open the parts list. Locate the part and take a note of its *Outline* name.

Open the *Component Outlines* folder and open the outline noted above. Use the *Pad > Describe* command to identify the pin numbers assigned to pads.

Are they correct? Does the outline have the correct number of pads? If the wiring list calls up IC1.20, but a DIL16 has been called up for the part, then either the incorrect outline was used (perhaps a DIL20 should have been called up) or the part should not have pin 20 assigned. Make the appropriate corrections depending on the reason for the error.

If alpha-numeric pin numbers have been used, ensure the pin numbers in the outline correspond exactly.

(The outlines in the supplied library only use numeric pin numbers.)

Single node nets

What is a single -node net? It's a connection that has been started but is not connected to anything else.

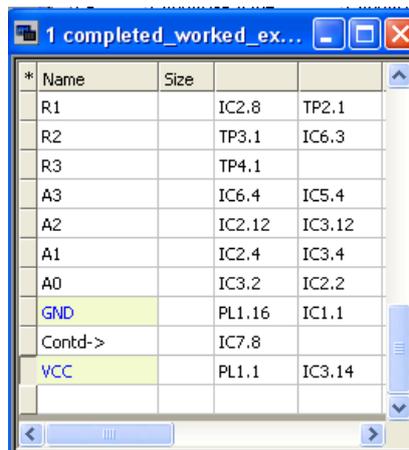
Single-node nets are reported when a parts/net list is extracted from a schematic and when the artwork checks are run. However if the net list has been typed/imported, or extracted from the "old, flat" schematic editor, having them reported when the design is complete may be a little late in the day; especially if the connection should be attached to a pin on the other side of a tightly packed board.

Single node nets occur because a connection has been attached to a part pin, then released in mid-air - probably with the intention of adding a signal name to it, to "make" a connection to another same-named connection, or a spelling mistake has been made. For example AO typed in instead of A0. Or, if a net list has been typed in, the phone may have rung and diverted attention from the net being typed.

Single node nets can be located as follows:

With the net list editor open, scroll through it, watching the second node column for an empty space.

This can be made easier if the window is displayed & resized as shown in Figure 132, so that only the first two node columns visible. The UID and Minimum clearance columns have also been hidden using the View commands.



* Name	Size		
R1		IC2.8	TP2.1
R2		TP3.1	IC6.3
R3		TP4.1	
A3		IC6.4	IC5.4
A2		IC2.12	IC3.12
A1		IC2.4	IC3.4
A0		IC3.2	IC2.2
GND		PL1.16	IC1.1
Contd->		IC7.8	
VCC		PL1.1	IC3.14

Figure 132

The only time the second node column could legitimately be empty is if the line has been continued from the previous line.

In Figure 132, the connection with the signal name "R3" is a single node net – it starts at TP4.1 but isn't connected to anything else.

The node IC7.8 is not a single node net because it has been continued from the previous line.

When single node nets are located, they should be investigated and a correction made by either making the correct connection on the schematic or removing the offending connection and re-extracting the parts/net list. If a schematic was not drawn, edit the net list.

Unused pins list

Refer to the heading *Edit > Find Unconnected Pins* later in this chapter.

BOM (Bill of Materials) output

Refer to the heading *View > Bill of Materials* later in this chapter.

Individual commands in parts/net list editor

The following commands are available when either the parts or net list editors are active. If the commands are greyed out, these commands are not applicable to the active window.

File commands

File > Save Parts (or Nets) List As Text

The information held in the parts and net list editors can be output to text files. These files can be used as a source for pick and place machines, wire wrap machines, etc. and can be modified using a standard text editor if required. When the output file is created, it contains the data from whichever list is active.

When selected, the Windows browser appears, use it to locate the folder where you would like the file to be created in, and then supply a name for the file, including the .txt extension if required.

The "Save as Type" setting, indicated in Figure 133, can be used to output the file with space or tab separated columns. The tab separated output makes it easier to import the resultant file into spreadsheets and other programs.

When saving the net list with tab separation, the signal name column for unnamed nets will be blank, for space separated columns a full stop is output.

Nets nodes will not be continued onto multiple lines. All net nodes will be output on a single line.

The net UID values is included.

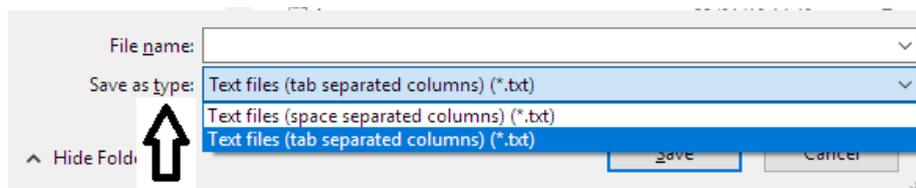


Figure 133

Select **Save** to create the file.

Note: A GenCAD ASCII format file containing more comprehensive information suitable for pick & place machines is also available. (Right-click on the design name in the navigator, then *Export > Gencad Format ASCII File Generation.*)

File > Print

The parts and/or net list can be printed to the Windows printer as required using this command. When the printout is selected, it contains the data from whichever list (parts or wires) is active.

File > Close Parts (or Nets) List

When selected, the parts (or Nets list) editor window is closed.

Edit commands

Edit > Units

Used to change the units in use between metric or inches. The units in use can be recognised by the appearance of the decimal point - a dot (.) is used in inch values and a comma (,) in metric values.

Edit > Find Part

Used to find a specific part in the parts list. When selected, the part reference should be typed in the window that appears, i.e. **IC3** to locate IC3. Not case sensitive.

Edit > Repeat Part

Used to repeat the selected part in the parts list, whilst automatically incrementing the part reference number. Useful when typing in a parts list with multiple copies of the same part.

Select the part reference to be repeated, then *Edit > Repeat Part*, fill in the details in the window. The new parts will be added to the end of the existing entries.

Edit > Allow Position Edit

Activate this command (by selecting it) if the X, Y or Rotation fields in the parts list have to be edited. Until this command is activated, these fields are not selectable.

Edit > Find Node

Used to find a particular node in the net list. When selected, the node should be typed in the window that appears, as they would appear in the net list, i.e. **IC3.6** to locate IC3, pin 6.

Edit > Find Signal Name

Used to find a particular signal name in the net list. The search is not case-sensitive unless there are multiple occurrences of a signal name in use when the first name will be located. For example, if *clock* and *CLOCK* occur in the list, then searching for *Clock* will find whichever one occurs first in the list.

Edit > Find Unconnected Pins

A list of the unused/unconnected pins in a design can be obtained from the net list editor.

When using this method to identify unused pins, they are defined as those pins that exist in **component outlines**, but are not referred to in the net list.

A list of unconnected pins can also be obtained from the schematic editor. This may produce a different result, as the check is based on schematic parts (for instance a connector may have 9 pins on the schematic, but the component outline could have 11 pins that includes two mounting pins/holes).

Obtaining the list

With the net list editor open and active, select **Edit > Find Unconnected Pins**. The list is displayed on the screen automatically. It can be printed (*File > Print*), sent to a file (*File > Save as*) and closed (*File > Close*).

If the *Tools > Find Unconnected Pins* command is not available, ensure the net list is popped forward/active.

View commands

View > Show Part UID's

Used to toggle the display of the part uid (unique identification) field on/off. (The part uid value cannot be edited.)

View > Show Net UID Codes

Used to toggle the display of the net's uid (unique identification) code field on/off. (The net uid value cannot be edited.)

View > Minimum Clearance

Used to toggle the display of the minimum clearance field on/off.

View > Bill of Materials

A bill of materials (BOM) can be obtained from the parts list editor. This is a fixed format file and can include parts from both sides of the board, or either side of the board..

More flexible, user-definable bill of materials files can be obtained from the schematic editor when it is open (*Tools > Parts/Netlist Extraction > User Defined Extraction*).

Producing the Seetrix format BOM file

With the Parts list window open/active, select *View > Bill of Materials*.

The window shown in Figure 134 opens.

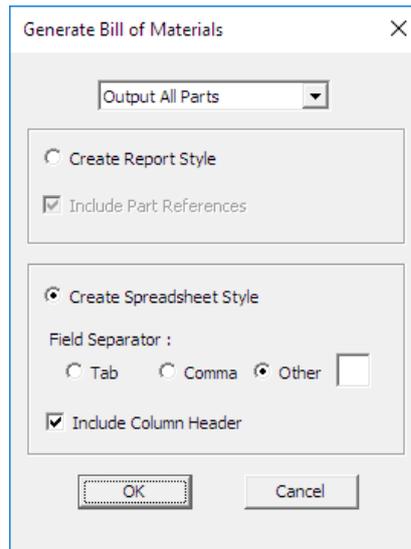


Figure 134

Part selection: All the parts can be output or just the parts from one side of the board.

If “*Create Report Style*” is selected, this will produce a standard output using spaces between fields.

An option is given so that the part references (R1, R2, R3, etc.) can be included or excluded as required. Excluding the part references reduces total page count and assists in fitting the report to the available page width.

If “*Create Spreadsheet Style*” is selected, this will produce an output using either Tabs, Commas or a user-defined character between fields.

The column headers can be included/excluded as required.

When OK is selected, the output is produced and appears in a window.

It can be printed (*File > Print*), sent to a file (*File > Save as*) and closed (*File > Close*).

Right-clicking the Parts or Nets Folders

Delete All Parts and Nets

By right-clicking on the Parts folder, it is possible to delete the entire parts and net lists. This command should be used with extreme caution. (Always save a backup copy of the design prior to its use so that you have something to return to.) The parts list stores all the placement information for the artwork, so deleting it will remove all the parts from the artwork (the artwork layers will remain).

If a schematic has been created/modified it is never necessary to use this command. The parts/wiring list should be re-extracted and the parts/wiring list will be updated.

However there are times when performing modifications to a design where you might want to re-start the artwork design, based upon an existing schematic, so this command does have its uses.

Delete All nets

Same as the *Delete All Parts and Nets* command, but only removes the nets list. Use with extreme caution and copy the design first!

Board Profile

Before laying out the artwork, a board profile is required.

The board profile is used as a guide for yourself when placing parts, and as a basic working area for the automatic routines such as copper fill, power plane generation and auto-routing.

The *View > Full* command also uses the board profile as its guide when filling the screen with the layout.

The board profile editor is used to define the board profile and can also be used to define keepout areas for the auto-router and copper fill routines.

Data in the board profile editor can be imported from an AutoCAD DXF file if required/preferred. Once imported, the data can be modified as though it were originally defined using the board profile editor.

The board profile and keepout definitions can also be modified or added from within the artwork editor, using either the *Amend* or *Region* command menus. Any changes made in the artwork editor are back-annotated to the board profile editor immediately. Changes made in the profile editor are also implemented in the artwork editor immediately.

Changes made in the board profile editor do not affect the position of parts in the artwork or power planes, so they may need to be updated as a consequence of the change.

Only one profile is held in the design.

Board profiles can be copied to the master board profile library and used on other designs. Once in the master library, the profiles can be copied and modified.

Opening the Board Profile Editor (design or master)

To open the design's board profile editor, with the design's folder open in the navigator window, double select the *Board Profile* (or right-click/Open). The editor opens.

The master board profiles are held in a folder inside the *Masters* folder, so first open the *Masters* folder by selecting the + sign alongside it, then open the *Profile Library* folder in the same way. If board profiles exist in the folder they will be listed below the open folder in the navigator pane. They can be selected in turn to browse through them. As the board profiles are selected, they are shown graphically in the *Browser* window and their properties displayed in the *Properties* window (if those windows are open).

A board profile can be opened for editing by double-selecting it. New designs will not have a profile defined.

Once the editor has been opened, if a profile has been defined it will appear.

A grid may or may not be visible.

A red shaded area around the bottom and left hand side of the working area indicates negative coordinate space. The board profile should **not** be drawn touching or within the negative space as this can cause difficulties with artwork placement, routing and output. This red area cannot be switched off in the profile editor (though it can be made invisible in the artwork editor).

Creating a board profile – general guidelines (incl. keepouts)

Once the board profile editor has opened, the board profile and keepouts can be created/edited.

The tool bar shows the current *Edit Mode* category, which is initially set to *Profile*. This indicates that any lines drawn will become part of the board profile. It can be changed to *Keepouts* by selecting the arrow alongside, then choosing from the list that appears.

It is often convenient to have the grid switched on when defining the profile. If the grid is switched off or not set to a suitable pitch, it should be switched on/changed using the *Grid* commands.

Once defined, the co-ordinates of the profile/keepout/router data can be viewed and edited using a text-style editor (*Profile > Numeric Editor*).

Adding the board profile

The board profile **MUST** be a completely enclosed polygon. It can be drawn with combinations of straight lines and arcs or a circle.

Do **NOT** define more than one board profile polygon, as the auto-router, split power plane generator, copper fill, etc. routines use the board profile to define their working area and will become confused if there are two or more board profiles. (Tooling holes can be added as extra holes or created as outlines and placed on the board as parts. If you want to define a position for them in the profile editor, add the locations using *keepouts*)

The profile is typically drawn from the starting point indicated by the pink circle/cross, although it's not essential. This is because it is useful to have space around the sides of the board to allow parts to be placed off the board temporarily.

The starting point is 1.0" in and up from the lower left corner of the working area. It cannot be moved. The co-ordinate readout is given with respect to the datum of the editor, which can be moved.

Do not draw the profile along the edges of the working area - either touching or in the red-shaded negative area in order to avoid problems later on.

Use the commands from the *Profile* pull-down menu as required to draw/modify the board profile.

In the profile editor, the profile line is displayed in blue so that it can be differentiated from the yellow keepout lines.

The board profile lines can also be modified from within the artwork editor, using the *Amend* commands.

Adding the keepout outline(s)

The keepout outline is drawn using the same commands as the board profile, except that the *Edit Mode* setting should be set to *Keepouts*.

The keepout line should completely enclose an area if that area has to be kept free of tracks, etc. It can be made up of combinations of straight lines and arcs or a circle. As many keepout areas as required can be added.

Enclosed keepout areas are shown cross-hatched in the board profile and artwork editors.

Do not draw the keepout lines along the edges of the working area - either touching or in the red-shaded negative area in order to avoid problems later on.

The auto-routers and copper fill routines stay inside or outside of the areas enclosed by keepout lines. If a single line is added, then the automated routines will go around the line, but will use the space either side of the line.

Sometimes, keepout areas are related to the positions of certain parts. If this is the case, add the keepout areas after the parts have been placed on the board. The position of part pins can be seen by selecting the *View > Drillholes* command.

Keepout lines can also be modified from within the artwork editor, using the *Amend* commands.

In the profile editor, the keepout line is displayed in yellow so that it can be differentiated from the blue profile.

Individual commands - board profile editor

An overview of the board profile editor can be found in the *Installation & Getting Started Guide*. Here we describe each of the board profile editor commands in detail.

Right-click on board profile/profile library in the navigator

Right-clicking on the board profile or the master profile library in the navigator window introduces the following commands and these are described below:

New Open Copy Paste Delete Rename

New

Only available when the *Profile Library* folder in the *Masters* folder is right-clicked. When selected, a new board profile is created and added at the bottom of the list of profiles already in the folder, called *Unnamed*. It can be renamed at that point as it is highlighted ready for editing. However, if <enter> is pressed or the cursor is selected elsewhere, the original name (unnamed) will remain. It can be renamed by right-clicking on the outline and selecting *Rename*.

Open

When selected, the board profile is opened for viewing/editing.

Copy

When selected, the board profile is copied to the paste buffer, overwriting anything previously added to the paste buffer. (See Paste command for using the contents of the Paste buffer.)

Paste

A board profile that has been copied to the paste buffer can be pasted (copied) to the design profile editor (overwriting an existing profile) or master profile folder.

To paste to the design: in the navigator window right-click the *Board Profile* in the design's folder, then select *Paste* from the options that appear. If a profile already exists an opportunity to proceed or cancel is given.

To paste to the master profile folder: right-click the *Profile Library* folder from the *Masters* folder in the navigator window, then select *Paste* from the options that appear. The profile is added to the bottom of the list of profiles in the folder, it takes the name of the job it was copied from. If a profile with the same name already exists, the option to overwrite or cancel the paste command is given.

Delete

Only available when a profile from the *Profile Library* folder in the *Masters* folder is right-clicked. When selected, the board profile is deleted – there is no confirmation window, the profile is deleted immediately.

Rename

Only available when a profile from the *Profile Library* folder in the *Masters* folder is right-clicked. When selected, the board profile can be renamed.

View Commands – specific to Board profile editor

View > Part Pins

Used to toggle the display of component pins on and off. The pins can only be seen once parts have been placed on the board.

The pin positions can be used as a guide when defining keepout areas. Pads and vias added to the artwork editor or outlines using the *Copper* commands are not displayed.

Profile commands

Using the *Profile* commands, lines, arcs, etc. can be added to form part of the board profile, router outline or keepout lines/areas. The data is added to the category or “layer” indicated by the *Edit Mode* setting in the tool bar, which can be set to *Profile*, *Keepout* or *Router*.

Data can only be modified if it belongs to the active category. For instance, if a line was added as part of the *Router* outline, it can only be modified if the *Edit Mode* is set to *Router*.

The colour of the drawn lines indicate which category they belong to:

blue	=	board profile
yellow	=	keepout
pink	=	router

If lines of different categories are placed over one another, the colours merge and a white line is displayed.

Note: in the artwork editor, the profile is displayed in white and keepouts in purple.

Toolbar “Edit Mode” setting

Edit Mode, Profile

Should be selected to define the external edges of the board. It's important that only one *Profile* polygon should be defined. Ranger's automatic routines (split power planes, copper fill, auto-placement/routing, etc.) operate within the area defined by the profile polygon and will become confused by additional board profile definitions.

Mounting/tooling holes, cut-outs should not have their positions defined using *profile mode*. Use the *Router* mode and *Keepout* modes to identify the position of holes or cut-outs within the profile.

Edit Mode, Keepout

Should be selected to define areas that the auto-router and copper fill routines should not cross. For instance cut-outs within the profile.

Profile > Add Line

Used to add a line, the line is added to the current *Edit Mode* category.

Once selected, point at the location where the line will start and click the left button. Move the cursor, a line appears attached to it. Click the left button to insert corners in the line. Release the line from the cursor by clicking the right-hand mouse button.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

The line can be modified using the *Profile > Corner/Adjust Arc/Delete point*, etc. commands.

Profile > Add Arc

Used to add an arc(s), the arc is added to the current *Edit Mode* category.

Once selected, point at the location where the curved line is to start and click the left button. Move the cursor and its attached line to the position where the curve should end, and click the left button again. A straight segment is produced that bends as the cursor is moved. The shape of the arc is dependent on the position of the cursor. Click the left button to release the curve in its current position. The line remains attached to the cursor, allowing a series of curves to be added.

Once the arc has been released, clicking the right-hand mouse button releases the line from the end of the cursor.

If the right-hand mouse button is clicked whilst the curve is being stretched, a straight-line segment is introduced, thus allowing a mixture of curved and straight lines to be added.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Arcs can be modified with the *Outline > Corner/Adjust Arc* commands.

Profile > Add Circle

Used to add circles, the circle is added to the current *Edit Mode* category.

Once selected, point at the centre of the required circle and click the left-hand mouse button. As the cursor is moved, a circle appears attached to it. The size of the circle is controlled by the cursor position. Click the left-hand mouse button to release the circle in its current position. Clicking the right-hand mouse button whilst adding the circle cancels the circle.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The circle can subsequently be moved or modified with the *Outline > Move Point* command.

Profile > Add Rectangle

Used to add a rectangle, the rectangle is added to the current *Edit Mode* category.

Once added it can be modified using the *Outline > Corner/Adjust Arc* commands.

The size of the rectangle can be specified in either inches or millimetres. The units in use are controlled by the *Edit > Units* command.

Once selected, a window appears allowing the *Width* and *Height* of the rectangle to be typed in. Select **OK** to add the rectangle.

The rectangle appears attached to the cursor (the screen magnification may change in order to display the complete rectangle), move it into position and click the left button to release it. Another rectangle appears on the end of the cursor until either another command is selected, or the right-hand mouse button clicked when the window reappears, and *Cancel* should be selected.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Once in position, the rectangle is treated as a line, which can have corners added, deleted or moved or converted to arcs.

Profile > Corner

Used to add and move existing corners in lines or arcs.

Lines and arcs can only be selected if they belong to the current *Edit Mode* category.

Corners are **added** and released with clicks of the **right-hand** mouse button. Corners are **moved** and released with clicks of the **left-hand** mouse button. Once the corner has been selected, clicking the opposite mouse button cancels the operation.

right button	=	add corner
left button	=	move corner
Opposite button cancels		

Adding a corner to a line or arc:

Once selected, point at the line or arc and click the right-hand mouse button. Move the new corner and release it with another click of the same (right) button. Clicking the opposite (left) hand button before the corner is released, cancels the new corner.

Moving an existing corner in a line or arc:

Once selected, point at the corner in the line or arc and click the left-hand mouse button. Move the corner and release it with another click of the same (left) button. Clicking the opposite (right) hand button before the corner is released, cancels the move.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Profile > Move Point

Used to move circles and to change the size of circles. (It can also be used to move an existing point (corner) in a line or arc, but the *Profile > Corner* command is more flexible.)

Circles and points can only be selected if they belong to the current *Edit Mode* category.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Moving a circle:

Once selected, move the cursor over the centre of the circle and click the left-hand mouse key. Move the cursor with the circle attached, into the required position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the circle is attached to the cursor returns it to its original

position.

Changing the size of a circle:

Once selected, point at the circumference of the circle and click the left-hand mouse key. As the cursor is moved, the circle's size changes. Click the left-hand mouse button again to release the circle at its current size.

Clicking the right-hand mouse button whilst circle is attached to the cursor returns it to its original size.

Moving points on lines or arcs:

Once selected, move the cursor over the point (corner) you want to move and click the left-hand mouse key. Move the cursor with the corner attached into the required position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the corner is attached to the cursor returns it to its original position.

Profile > Adjust Arc

Used to convert straight lines into curved lines, curved lines into straight lines, and to change the shape of existing curved lines.

Lines and arcs can only be selected if they belong to the current *Edit Mode* category.

Straight lines into arcs:

Once selected, point at a line segment and click the left-hand mouse button. Move the cursor, the segment is replaced with an arc on the end of the cursor. Once the arc is in the required position, click the left-hand mouse button to release it.

Arcs into straight line segments:

Once selected, point at the arc and select it with a click of the left-hand mouse button. Follow this with a click on the right-hand mouse button to convert it to a straight line segment.

Changing the shape of an arc:

Once selected, point at the arc and select it with a click the left-hand mouse button. Move the cursor with the arc attached, then click the left-hand mouse button when the arc is in the desired position.

Clicking the right-hand mouse button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically and can be used to convert 45 degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Profile > Transfer to current layer

Used to transfer line/arc/circle features from their current layer onto the layer selected in the *Edit Mode* category.

This command can be useful to separate profile and keepout data that has been imported in to a single layer, or added to the wrong layer.

Set the *Edit Mode* category to the layer the data should be transferred to.

Once selected, point at the line, arc or circle to be transferred and click the left-hand mouse button. The item will change colour to indicate the transfer has worked.

Refer to the Grid command for details on whether grid snapping is active.

Profile > Delete Point

Used to remove corners (points) from lines or arcs.

Points can only be selected if they belong to the current *Edit Mode* category.

Once selected, move the cursor over a point (corner) in a line or arc and click the left-hand mouse button. The point is removed.

If a start or end point of a line is deleted, the segment is removed.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Profile > Delete Feature

Used to delete a line, an arc or circle.

Items can only be selected if they belong to the current *Edit Mode* category.

A complete line is deleted, between its start and end points, not just a segment between points.

Once selected, point at the line, arc or circle to be deleted and click the left-hand mouse button.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Profile > Numeric Editor

Used to edit or view the co-ordinates of the lines drawn in the profile editor. This facility is useful to check the size/accuracy of the drawn lines or to locate lines that are difficult to see.

Although possible, it is not recommended that the numeric editor be used to define a shape from scratch. Instead use the graphical tools to draw the basic shape of the profile, then use the numeric editor to fine-tune the co-ordinates if necessary. This is easier than trying to define the board numerically from nothing.

Once selected, a window appears as shown in Figure 135 divided into three main areas.

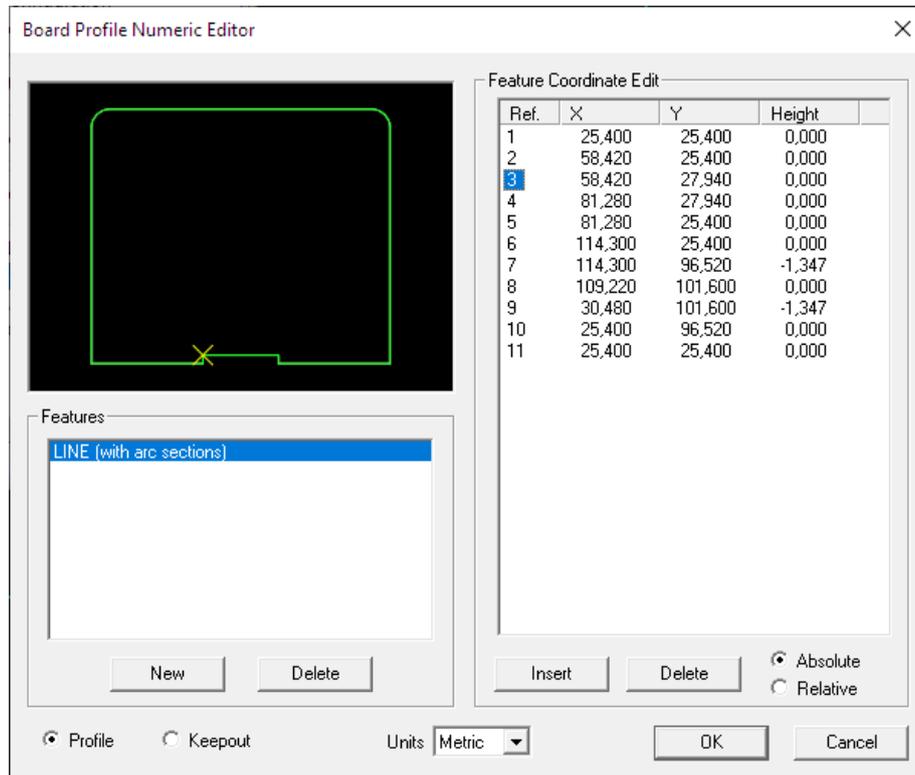


Figure 135

The top left shows the overall *picture* of the profile, keep-out or router lines.

The radio buttons along the bottom of the window control which category of data is visible and therefore can be edited, *Profile* is selected in this example.

The bottom left area lists the *Features* (lines, lines with arc sections or circles) that make up the profile, keep-out or router lines. Each one can be selected with a click of the left button. This will turn the corresponding line/arc/circle in the upper left picture, green and show the X/Y co-ordinates that make up the line/arc/circle in the *Feature Co-ordinate Edit* area on the right hand side of the window. This is shown in Figure 135.

Each feature that is added as a continuous entity is listed in the *Feature* list. For instance if a polygon shape were added graphically (Profile > Add Line) with one continuous line/arc, as shown in Figure 135, it would be one "feature". When the feature is selected, the complete polygon turns green as shown. However if the same polygon shape were made from *separate* lines, then each line would be listed separately as a *feature* and individual segments would turn green when selected.

Co-ordinate definitions

The co-ordinates in the *Feature Co-ordinate Edit* area can be selected and changed. The change will be reflected in the picture.

Each pair of co-ordinates is given a reference number and if this number is selected (in Figure 135, ref no. 3 is selected), a cross appears on the picture of the profile at the location of the co-ordinate to assist with identification.

Lines have an X and Y co-ordinate for their starting position and an X/Y co-ordinate for their end position (two sets of X/Y co-ordinates minimum).

If there are corners between the start and end points then these X/Y co-ordinates will appear between the start and end points in order, as shown in Figure 135.

Arcs/Circles

The Height field is only present if the line contains arc sections, it is used to define the height of the arc within the line. Arcs have two sets of X/Y co-ordinates (start and end points) plus a height for the arc. The height is entered alongside the starting X/Y co-ordinates of the arc. A height value will not be accepted alongside the end X/Y co-ordinates of the arc.

A positive height value draws the arc in a clockwise direction from its starting point (first pair of co-ordinates). A negative height draws the arc in an anti-clockwise direction from its starting point - see diagram below.

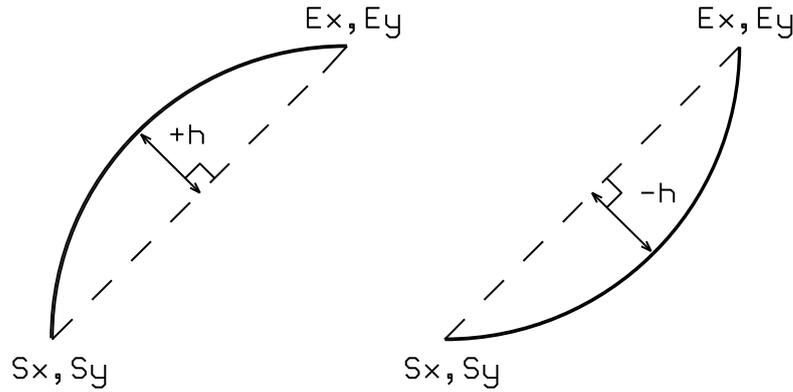


Figure 136

Typical co-ordinates for the arc on the left would be:

X	Y	Height
2.5	1.5	0.16
3.0	2.0	0.0

The co-ordinates for the arc on the right (if it were in the same position) would be identical except the height would be -0.16.

Circles are defined by the X/Y position of their centre and a radius.

Units

Co-ordinates can be entered in inches or millimetres after choosing the appropriate units from the bottom of the window.

Absolute/Incremental format

By default, co-ordinates are entered/displayed in absolute mode, each co-ordinate is given with respect to 0,0 in the profile editor.

If the radio button alongside *Relative* is selected, then the first pair of co-ordinates is given with respect to 0,0 in the profile editor, but subsequent co-ordinates are given with respect to the previous pair of co-ordinates.

Co-ordinates for a rectangle with absolute selected:

1.0	1.0
4.0	1.0
4.0	3.0
1.0	3.0
1.0	1.0

Co-ordinates for the same rectangle with relative selected:

1.0	1.0
3.0	0.0
0.0	2.0
-3.0	0.0
0.0	-2.0

The position of the rectangle would be in the same location at 1.0,1.0 (first set of co-ordinates) in from the lower left corner (0,0) of the profile editor.

Deleting features

Features can be deleted if they are selected from the *Features* list, then the **Delete** button selected.

Adding features

New lines, lines with arcs or circles can be added by selecting the **New** button from underneath the *Features* list. (It is usually easier to draw the new feature, then edit the co-ordinates afterwards.) A window appears requesting the type of feature that should be added and how many points (corners) it should contain (minimum of 2, the start and end points). (If *Circle* is selected, the points setting is greyed out.)

When *Create* is selected, the feature is added to the end of the list and the co-ordinates for the feature (all set to 0) appear in the *Feature Co-ordinate Edit* area of the window. The co-ordinates must be updated or there will be lines/circle/arcs at 0,0 with no size assigned and they may cause problems.

Modifying features

Additional corners can be added to existing lines/arcs if required. (It is usually easier to draw the new feature, then edit the co-ordinates afterwards.) Select the line/arc to be modified from the *Features* list, select the reference number where the co-ordinate should be inserted, from the *Feature Co-ordinate Edit* area, then select the *Insert* button. An additional pair of co-ordinates is inserted with the same X/Y positions as the selected pair. One set of co-ordinates should be modified.

Corners can be deleted from existing lines/arcs if required. Select the line/arc to be modified from the *Features* list, select the reference number of the co-ordinate that will be removed from the *Feature Co-ordinate Edit* area, then select the *Delete* button.

Leaving the numeric profile editor

If **OK** is selected from the numeric editor window, the numeric editor window is closed, the changes are kept and the graphical editor updated. If **Cancel** is selected, the numeric editor window is closed and any changes made to the window are lost. The graphical editor is not changed.

Profile > Set XY Display Datum

Used to change the co-ordinate readout so that it is with respect to a selected point, rather than its default position which is located at the lower left corner of the working area.

When the editor is closed the datum is returned to its default position.

The display datum is displayed as a small yellow circle with an upright cross inside it.

Once selected, point at the position required for the datum point. (Refer to the *Grid* command for details on whether grid snapping is active. The current grid setting is displayed in the information bar.)

This command remains active until another command is selected.

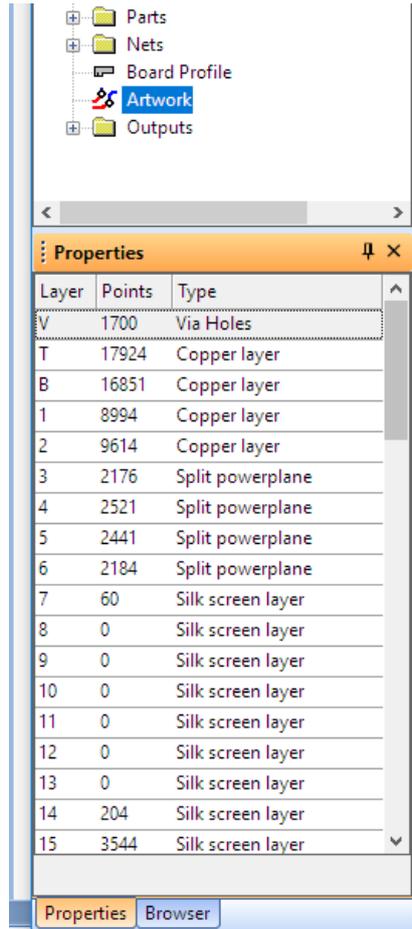
To restore the datum point to its default position, click the right-hand mouse button whilst the *Profile > Set XY Display Datum* command is active.

Artwork Folder

Artwork

An overview of the artwork editor can be found in the *Installation & Getting Started Guide*. Here we describe each of the artwork editor editor commands in detail.

When *Artwork* is selected from the navigator pane, the *Properties* pane displays a layer type and point count summary for all the artwork layers as shown in Figure 137.



Layer	Points	Type
V	1700	Via Holes
T	17924	Copper layer
B	16851	Copper layer
1	8994	Copper layer
2	9614	Copper layer
3	2176	Split powerplane
4	2521	Split powerplane
5	2441	Split powerplane
6	2184	Split powerplane
7	60	Silk screen layer
8	0	Silk screen layer
9	0	Silk screen layer
10	0	Silk screen layer
11	0	Silk screen layer
12	0	Silk screen layer
13	0	Silk screen layer
14	204	Silk screen layer
15	3544	Silk screen layer

Figure 137

When the artwork editor is opened, tooltips appear (when appropriate) when the cursor is hovered over boxes, as shown in the View Control dialogue bar in Figure 138.

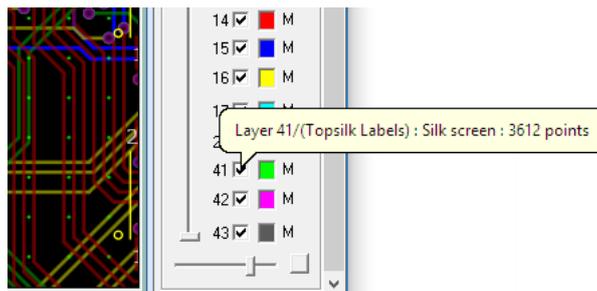


Figure 138

Layer colour boxes which have an associated edit/spin control to change layer selection, as shown in Figure 139, can be selected to display the layer assignment list, to quickly aid identification and selection of layers.

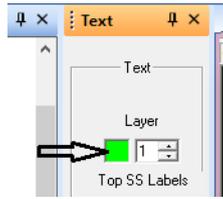


Figure 139

Right-click on Artwork

Right-clicking on *Artwork* in the navigator window introduces the following commands:

Open Open Detached Delete Tool Optional Autorouters

Open

When selected, the artwork editor is opened for use.

Open Detached

When selected, the artwork editor is opened for use in a separate window so that it can be moved outside of the main Seetrix XL Designer program. This allows it to be viewed on a separate display if available. The window can be moved & resized using the standard Windows tools.

Delete Tool

Used to delete the complete artwork, i.e. tracks, vias, silk screen, power plane pads, text, etc., or individual layers with the option to retain fixed tracks. The parts remain in position irrespective of the choices made.

When selected the window shown in Figure 140 appears.

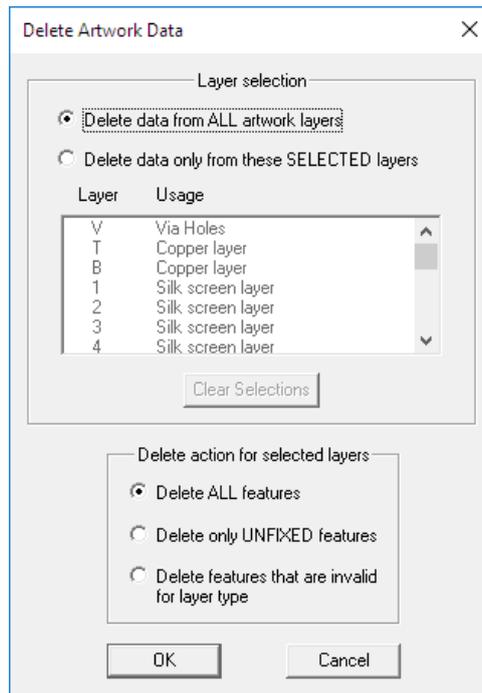


Figure 140

This window has two areas of control, *Layer Selection* and *Delete action for selected layers*. Ensure both are set as required before proceeding - there is no undo!

Layer Selection

Delete data from ALL artwork layers

when selected all the data added to all layers in the artwork editor is removed. This includes tracks, vias, copper fill, silk screen, power plane pads, text, etc.

Delete data only from SELECTED layers

Parts (and therefore part pins) remain placed on the design. when selected all the data added to specific layers in the

artwork editor is removed. This includes tracks, vias, copper fill, silk screen, power plane pads, text, etc. on the selected layers.

When selected, the lower part of the *Layer Selection* pane becomes active and this is used to control which layers are included in the delete action – the selected (highlighted) layers are included in the deletion.

Repeated clicks of the left mouse button over the layer, selects/deselects the layer. As many layers as required can be included.

Delete action for selected layers

Delete ALL features

when selected all the data added to the selected layers (*ALL* or *Selected*) in the artwork editor is removed.

Delete only UNFIXED features

when selected only the unfixed data added to the selected layers (*ALL* or *Selected*) in the artwork editor is removed.

This setting is useful as it allows pre-routing to be retained whilst removing everything else.

Delete features that are invalid for layer type

Heat-relief and anti pads should not be used on copper layers.

The artwork checking routine checks all *Copper layers* to determine if any are present and they should be removed as they could cause short-circuits and/or gap errors. This utility provides a quick and easy method of removing them.

When selected all antipads and heat reliefs pads on any layers defined as copper layers will be removed.

Optional autorouters

Seetrix XL Designer has interfaces to some third-party supplied auto-routers. The interface can be enabled from the *File > System Setup* window. When enabled, the auto-router appears in a pull-down list when this command is selected. Note: this is an interface, not the auto-router that has to be purchased and installed separately.

When the interface is selected a window appears and this is described in a separate chapter, later in this guide.

View commands

These commands control what is seen on the screen in the artwork editor and how it is displayed. The zoom in/out, pan, etc. commands which are common to all editors have been described in the chapter *Commands Common to All Editors*.

View > Artwork Flipped

When selected, the artwork is displayed “flipped” or “mirrored” as if looking at the back of the board.

When the artwork is displayed in “flipped” mode, flipped parts will have their outline displayed in solid (cyan) colour and unflipped parts will have a dotted (yellow) outline.

Many part related operations in the artwork editor require clicking the left mouse button to select unflipped parts and the right button for flipped parts. When the artwork is displayed in “flipped” mode, the sense of the buttons becomes reversed.

When using the *Text > Get Label* command to add a label for a flipped part, the label is now inserted in mirrored mode (so if viewing the back of the board with the display “flipped”, the label will appear the right way around).

View > Part Labels

Alternative (longer) way of toggling the *Part labels* setting. Refer to the *View > View Control* dialogue bar for quicker method.

View > Lines At Width

Alternative (longer) way of toggling the *Wide tracks* setting. Refer to the *View > View Control* dialogue bar for quicker method.

View > Pads Filled

Alternative (longer) way of toggling the *Pads Filled* setting. Refer to the *View > View Control* dialogue bar for quicker method.

View > Drill holes

Alternative (longer) way of toggling the *Drill* setting. Refer to the *View > View Control* dialogue bar for quicker method.

View > Isolated Copper

Used to toggle the visibility of isolated copper filled areas that were identified by the artwork checking routines. Only visible layers are highlighted, in white. (Isolated copper fill can be deleted using the *Tools > Copper Fill, Filled Copper > Delete All Isolated Copper* command.)

View > Net Clearances

Alternative (longer) way of toggling the visibility of the minimum clearance requirement of tracks, on and off. Refer to the *View > View Control* dialogue bar for quicker method.

View > Negative Space Shaded

Used to toggle the display of the red, negative space that surrounds the working area, on and off.

Although it can be made invisible (in response to user requests), under no circumstances should anything be placed touching or within the negative area.

View > View Control Toolbar

Used to toggle the visibility of the *View Control* dialogue bar, on and off. This bar provides control over what is displayed in the artwork editor at any one time. It is shown in Figure 141.

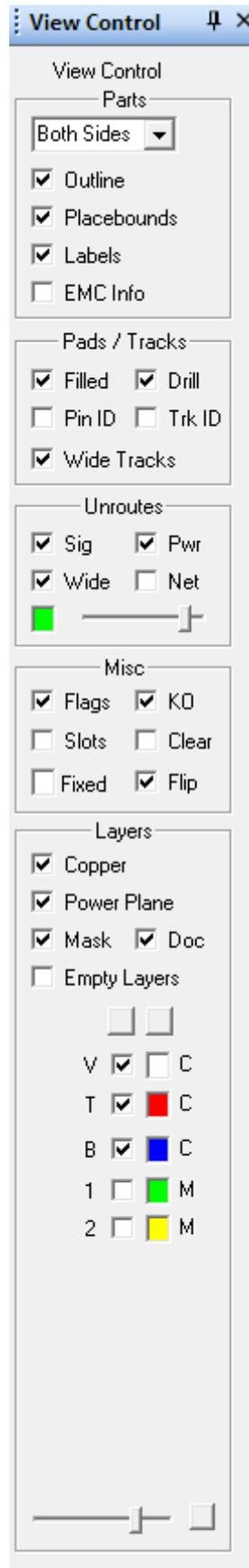


Figure 141

All the buttons on the bar have “tooltips” enabled, so hover over any button to get a description of the button, as shown in Figure 142.

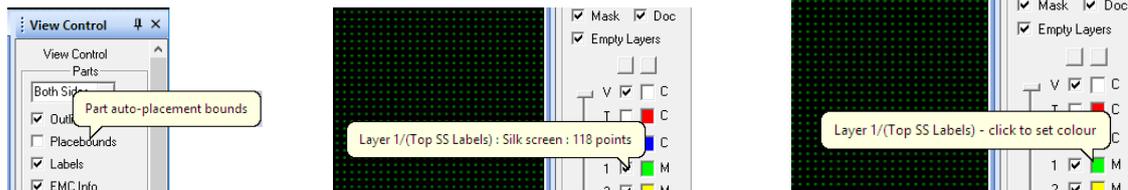


Figure 142

The bar is divided into sections and these are described below.

Parts Section (Figure 143)

When placing and moving parts, it is helpful if the *silkscreen outline* and/or *autoplacement footprints, labels* and *EMC Information* are visible. But this will clutter up the screen when routing for example. Control over this visibility is gained from this section.

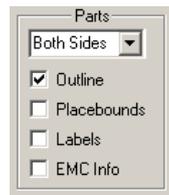


Figure 143

Both sides/Top/Bottom This controls whether only the parts on the top of the board, only the parts on the bottom of the board, or parts on the top and bottom of the board are displayed.

The silkscreen outlines/placebounds are shown in yellow for parts that are unflipped and in dashed cyan for flipped parts.

Select the arrow alongside to obtain the list to choose from. What is displayed is selected by the checkboxes underneath.

Outlines When selected (ticked) displays the silk screen outlines of the parts, as defined in the outline editor. Note: these are representative of the silkscreens only - to act as a guide when placing/moving parts. The actual silk-screen data has to be added to the artwork layers using the *Tools > Generate Silkscreen* command.

Unflipped parts are shown in solid yellow, flipped parts in dotted cyan.

If the artwork is displayed in "flipped" mode, then the colours remain the same but the flipped parts are shown with a solid (cyan) line and the unflipped ones with a dotted (yellow) line.

Placebounds When selected (ticked) displays the autoplacement footprint of each part, as defined in the outline editor. The bounds are not shown when a part is moved.

Labels When selected (ticked) displays the part labels (i.e. IC1, IC2, R1, etc.).

Part labels are shown in yellow for parts that are mounted on the top of the board, and in cyan for parts that are mounted on the bottom of the board.

(Use the *Edit > Display settings* to control the height of the labels.)

Part labels should not be confused with the silk screen ident that is added to an artwork layer.

The part labels can be used as a guide when moving and locating parts, etc. as they indicate the position of the part's datum. Parts that have been flipped, have their labels slightly offset and a tail extends to the datum point. This allows the labels to be read more easily when parts are mounted directly over the top of one another.

Unlike the silk-screen ident, part labels are always read from left to right and they do not rotate or flip with the part.

EMC info When selected (ticked) displays the EMC information of parts. Any parts that have had a part placement hi-lite colour defined in the schematic editor (*Tools > EMC Attribute Tags (Parts)* command), will be displayed cross-hatched in that colour. The cross-hatching extends to the area occupied by the placement bounds of the outline.

Typically, the hi-lite colours indicate whether the part is susceptible to noise, produces noise or is quiet. This allows more attention to be given to the positioning of parts, depending on their noise considerations.

When parts are moved or placed, their emc group is displayed in the status bar. The *Identify > Part* window also gives this information.

Pads & Tracks Section (Figure 144)

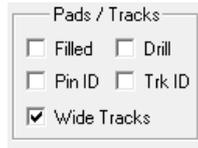


Figure 144

- Filled** When selected (ticked), pads are displayed in solid colour. Otherwise, only the perimeter of the pads are visible. Re-draw speeds are faster when only the outlines are displayed. If *Pin numbers* is ticked, this setting is automatically switched off as the white pin numbers would not be seen.
- Drill** When selected (ticked), the drilled holes in pads can be seen. Re-draw speeds are slightly faster when the drilled holes are not displayed.
- Pin ID** When selected (ticked), at higher zoom levels pin ID's (numbers) are displayed in the centre of component pins. When selected the *Pads filled* setting is automatically switched off as the white pin numbers would not be seen.
- Trk ID** When selected (ticked), and when the artwork is zoomed in sufficiently close, net identities will be displayed on all tracks and the perimeter of copper filled areas.
- Wide Tracks** When selected (ticked), tracks and lines (silk screen lines, text, etc.) are displayed at their true size. Otherwise only the centre lines of tracks and lines are displayed. Re-draw speeds are faster when only the centre lines are displayed.
Note: tracks and lines will never be rendered narrower than the line width controlled by the "Minimum Line Rendering Width" setting in the *Edit > Display adjustments* window.

Unroutes Section (Figure 145)

This section controls which unrouted tracks (connections) are visible and their opacity.

At various stages in a layout, different combinations of connections are important. This setting allows the user to concentrate on particular groups of connections, without having the others cluttering up the screen. For instance, on some boards the power supplies do not have a great effect on part placement, so just the signals can be viewed at the placement stage. Alternatively some critical connections might need to be routed first, so it helps not having the other connections as a distraction.

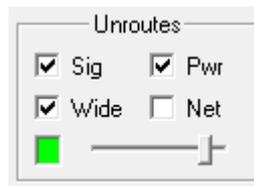


Figure 145

- Sig** When selected (ticked), all the unroutes that have not been defined as power rails will be visible, unless they have been "hidden" (right click a net from the the Nets folder of the navigator, then select *Hide Net Unroutes* - the net will be marked in the navigator with "(h)" to indicate it is hidden Figure 146).

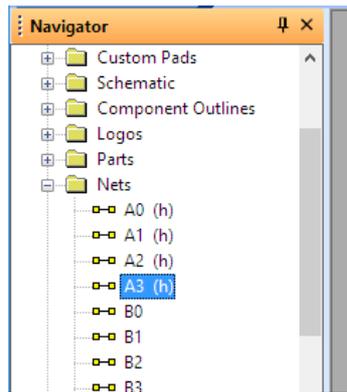


Figure 146

Pwr When selected (ticked), all the unroutes that have been defined as power rails will be visible, unless they have been “hidden” (right click a net from the the Nets folder of the navigator, then select *Hide Net Unroutes* - the net will be marked in the navigator with “(h)” to indicate it is hidden see Figure 146).

Note: The unroutes for power nets that have been defined as a power plane, will not appear unless they are attached to the pin of a surface mounted component. In this case, a "connection blob" will appear over the pin - it can be moved using the *Mroute > Corner* command. The blobs are easier to see if the unroutes are being displayed at their full width.

Wide When selected (ticked), all the unroutes will be displayed at their true size. Otherwise only the centre lines are displayed.

Re-draw speeds are faster when only the centre lines are displayed.

Unroutes will never be rendered narrower than the line width controlled by the "Minimum Line Rendering Width" setting in the *Edit > Display adjustments* window.

Net When selected (ticked), only unroutes from selected nets are visible – the *Sig* and *Pwr* checkboxes are automatically deselected.

To select the nets to be displayed, expand the *Nets* folder from the navigator, then right click on a net and select *Show net unroutes*. The status will be indicated by (S) appearing alongside the net name of the selected nets.

Multiple nets may be selected using the shift or Ctrl keys with the mouse.

Nets are de-selected in the same way, the (S) will disappear from alongside the net name.

There is no need to de-select nets in order to display all the signal, or all the power unroutes as those controls will switch off the *Net* selection when they are checked.

The show status of the nets is removed when the job is closed/reopened.

Coloured box Controls the colour of unroutes when visible. The colour can be altered by selecting the box, then choosing an alternative colour from the palette that appears.

Opacity Slider A the left of the sliders travel, unroutes are displayed behind the copper layers, and at the right of its travel, they are display over the top of the copper layers.

Please be aware that if you choose to display unroutes behind the copper layers, SMD pad powerplane unroutes (which are shown as a small blob) may not be visible.

Miscellaneous Section Figure 147

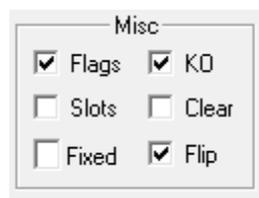


Figure 147

Flags When ticked, error flags are displayed if they exist. Error flags are produced when errors are found by the *Check* commands.

M = silkscreen mask error

K = keepout error

U = unroute

G = gap errors

If multiple minimum clearance values have been defined, the gap error flags are given a numeric suffix (G1, G2, G3, etc.) that indicates the minimum gap required in that location. The suffix is cross-referenced to a minimum gap size in a table at the bottom left of the screen.

- KO* When selected (ticked), keepout lines/areas are made visible. Keepout areas are shown cross-hatched.
Whether visible or not, they are always active (unless a specific control is available, like for example in the copper fill dialogue bar).
- Slots* Slots & Extra Holes have 3 visibility states:
- | | |
|-----------------------|---|
| Off (unticked) | Slots & Extra Holes are set as invisible |
| On (ticked) | Slots & Extra Holes are visible with the routing direction and pre-drilling details of slots visible. |
| Half On (greyed tick) | Slots & Extra Holes are shown, but without the routing direction and pre-drilling marks in slots. |
- Clear* When switched on, all tracks (not unroutes) will be drawn with a shadow, the size of which represents the required clearance for the net. Note: this function will not correctly display on monitors configured in the Windows setup for 256 colours or less.
- Fixed* Controls the colour of fixed tracks. The colour can be altered by selecting the box, then choosing an alternative colour from the palette that appears.
- Flipped* When selected, the artwork is displayed "flipped" or "mirrored" as if looking at the back of the board.
When the artwork is displayed in "flipped" mode, flipped parts will have their outline displayed in solid (cyan) colour and unflipped parts will have a dotted (yellow) outline.
Many part related operations in the artwork editor require clicking the left mouse button to select parts on the board top and the right button for flipped parts. When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.

Layers Section (Figure 148)

This section controls which layers are visible in the artwork and the colours assigned. Tooltips (hover mouse over item) also provide information relating to each layer, this information is described under the *View > Properties* command.

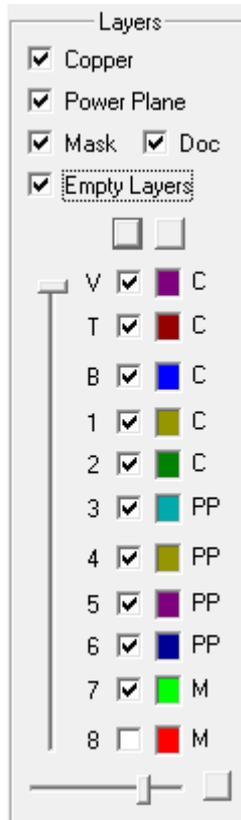


Figure 148

- Copper* Toggle switch used to add/remove the *Copper* layers to the list of layers below.
- Power Plane* Toggle switch used to add/remove the *Power* and *Split Power Plane* layers to the list of layers below.
- Mask* Toggle switch used to add/remove the *Silkscreen mask* layers to the list of layers below.
- Doc* Toggle switch used to add/remove the *Documentation* layers to the list of layers below.

Note: Layer definitions are controlled from the *Configuration* folder, *Layer Assignments & Ordering* window.

Empty Layers This toggle switch controls whether empty layers from the selected categories above are included in the list. This will help reduce the length of the layer list and provide easier access to the used layers.

Layer list The layers of the board are listed down the left hand side. The slider bar, which appears when required, provides access to all the layers.

The checkboxes alongside each layer allow that layer's visibility to be toggled on/off. If the cursor is hovered over these checkboxes, a balloon appears providing layer usage and point count information for the layer. (This information is described under the *View > Properties* command.)

All layers can be quickly made invisible or visible by selecting the check-box at the top of the check-box column. It's quicker to turn all the layers off and then one or two on, than it is to selectively turn most of the layers off.

Layers are made invisible to aid the design process. For instance, when modifying a completed board the silk screen layers can be switched off, or when working on a particular layer all other layers can be switched off.

The colour box alongside each layer shows the colour assigned to the layer. A different colour can be chosen if the coloured box is selected - a palette of colours appears from which one can

be selected.

The default set of colours is restored if the *Restore Default Colours* button (above the column of colour boxes) is selected.

The letters alongside the layers indicate the *layer type* of the layer.

- “V” - *Via* layer
- “T” - *Top* component side
- “B” - *Bottom* component side (or solder side) of the board
- “C” - copper layers
- “M” - Mask (silk-screen)
- “PP” - power plane layers including split power plane layers
- “D” - documentation layers

Opacity Slider

Figure 149. This setting alters the opacity and therefore the transparency of all copper layers. At the left of its travel all the layers will be less opaque and more transparent. At the right of its travel they will be more opaque and less transparent. Finer tuning is available for individual layers using the *Opacity Setup* button indicated in Figure 149.



Figure 149

The slider toggles between a slider and an “Edit Opacity” button as shown in Figure 150, if the fine tuning has been selected.

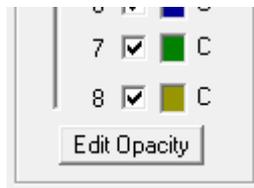


Figure 150

Use the “Revert to single slider” button in the Extended Opacity Control window (Figure 151) to revert to the single slider control.

Opacity Setup Button

This button introduces the Extended Opacity Control window as shown in Figure 151.

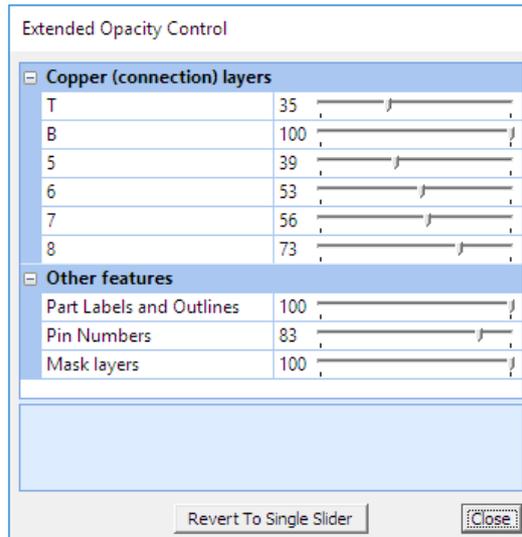


Figure 151

It allows the adjustment of the opacity and therefore the transparency, of the individual layers (unused layers are not included), along with Part Outlines and Labels, Pin Numbers and Mask (silk screen) Layers.

Only one adjustment is available for all silkscreen layers.

The range is from 0 (transparent or invisible) to 100 (completely opaque or solid).

By sliding the opacity of each layer/category, somewhere in between 0 and 100, it is possible to overlay multiple layers on top of each other with the ability to “see through” uppermost layers.

The effect of the changes can be seen whilst the window is open, so it is helpful if it is moved to one side.

View > Properties

When the artwork editor is active this command is used to toggle the visibility of the artwork Properties window, on and off. A sample Properties window is shown in Figure 152. It contains information about the artwork that is useful at various times throughout the life of the design.

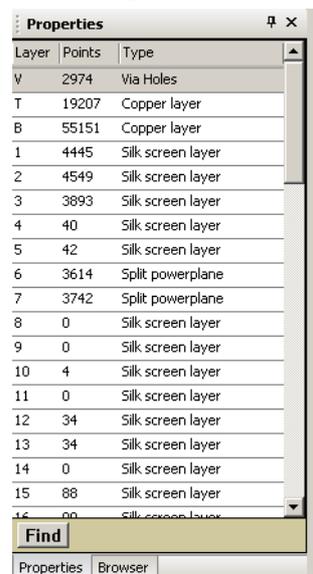


Figure 152

The window is divided into rows and columns, each row being a layer within the artwork. These columns are described now.

Layer The layers of the board are listed. Use the scroll bar to view the complete list. Some layers have names, others numbers. “V” is short for *Via*, “T” for the *top* component side

and “B” for the *bottom* component side (or solder side) of the board. The other layers are numbered.

Points This column indicates the number of points used on a particular layer. Every via, heat-relief pad, anti-pad or added pad is counted as one point. A track has a minimum of two points (start and end point), and each corner in the track is also added to the point count. The point count of text strings varies, depending on the number of characters used.

Component pads are not included in the point count. The number of vias is only counted on layer V.

If a layer has 0 points, then that layer is not in use. (If set as a copper layer, then inner pads from the component pad stacks will be present on that layer.)

Type this column indicates the layer type, i.e. whether the layer is defined as a copper layer, silk screen layer, powerplane layer, etc. The layer type cannot be changed from this window.

Some routines like the silk-screen generation tool automatically update the layer type setting, but generally speaking the layer type is controlled from the *Configuration* folder, *Layer Assignments and Ordering* window.

View > Restore Default Layout > Main frame layout

This will restore the default visibility and positioning of the navigator, browser and status panes. Typically used when these panes have been accidentally closed or moved off screen.

View > Restore Default Layout > Editor layout

This will restore the default visibility and positioning of the dialogue bars of the active editor. Typically used when these bars have been accidentally closed or moved off screen.

Identify commands

Identify > Part

Used to obtain information about a part and also locate it in either the parts list or schematic editors.

Select *Identify > Part*. Select a part on its datum point, which is indicated by the position of the part label (*View > Part Labels*). Once a part has been selected, an information window appears similar to that shown in Figure 153. A tail extends to the selected part to indicate which part was selected.

The icons representing the schematic and parts list editors can be selected to open the appropriate editor and locate the selected part.

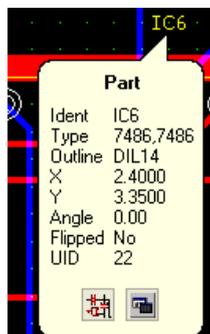


Figure 153

<i>Ident</i>	Part's reference designator.
<i>Type</i>	Obtained from the type column in the parts list. If the parts list was extracted from the schematic, it is a combination of the schematic part name and its value field separated by a comma.
<i>Outline</i>	Part's current outline name.
<i>X and Y</i>	Position of the part's datum in the X and Y axes, with respect to the artwork editor's current datum position.
<i>Angle</i>	Part's angle of rotation, anti-clockwise from its original orientation, as defined in the outline library.
<i>Flipped</i>	Whether the part has been flipped.
<i>UID</i>	Ranger's internal reference number for that part. Parts are numbered from 16 onwards, in the order they were originally entered in the Parts list.
<i>EMC Id</i>	If the part has been assigned to a part EMC group on the schematic, then the group to which it

belongs is specified here.

Parts may be selected until the right hand mouse button is clicked, when the previously active command is restored.

Identify > Pin

Used to obtain information about a visible component pin and also locate it in either the parts list or schematic editors using the buttons in the balloon window.

Select *Identify > Pin*. Select a component pin on its datum point. Once a pin has been selected, an information window appears similar to that shown in Figure 154, pointing to the selected pin. The properties of any associated net will also be displayed in the properties pane.

If layers 'T' or 'B' are set as invisible, pin/pad features on those layers are ignored.

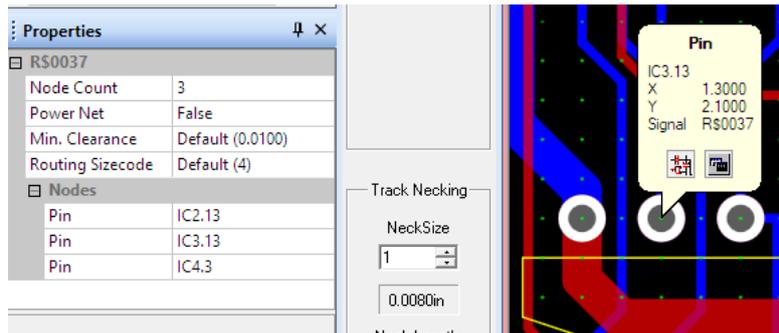


Figure 154

Pin no Such as IC1.4, R1.2, etc.

X and Y Position of the pin's datum in the X and Y axes, with respect to the artwork editor's datum.

Signal name The name of the connection to which the pin is connected.

Pins may be selected until the right hand mouse button is clicked, when the previously active command is restored.

Identify > Pad

Used to obtain information about any pad or via.

Select *Identify > Pad*. Select a pad on its datum point. Once a pad has been selected, an information window appears similar to that shown in Figure 155. A tail extends to the selected pad to indicate which pad was selected.

If layers 'T' or 'B' are set as invisible, pin/pad features on those layers are ignored.



Figure 155

Pinstack

Top indicates which pad shape/size is used on the top layer of the artwork.

Inner indicates which pad shape/size is used on the inner layers of the artwork. The inner pad only appears on layers defined as copper layers. The pad shape/size is the same on all inner copper layers.

Bottom - indicates which pad shape/size is used on the bottom layer of the artwork.

Standard pads are described using the following format:

shape, size code, size of the pad, drill size as defined in the sizes table.

For example:

SQ 4, 0.0600, 0.0350

The following shorthand is used for the shape:

- RO = round
- RS = rectangular, square ended
- SQ = square
- RR = rectangular, round ended

If a user defined pad is selected, then its name is given.

- Drilled* the inner pad determines the size of the stack's drilled hole. If the inner pad is not used, or does not have a drilled hole defined then the stack will not be drilled.
- X/Y* Position of the pad's datum in the X and Y axes, with respect to the artwork editor's datum.

Pins may be selected until the right hand mouse button is clicked, when the previously active command is restored.

Identify > Track

Used to obtain information about a routed track or an unrouted track (connection) and also locate it in either the parts list or schematic editors, using the buttons in the balloon window.

Tracks that are added using the *Amend* menu commands will not be recognised as belonging to a particular signal until they have been validated by the artwork checking routine.

Select *Identify > Track*. Select a track or connection on its centre line.

Once a track/unroute has been selected, an information window appears similar to the ones shown in Figure 156. A tail extends to the selected track/unroute to indicate which track/unroute was selected and the Properties pane provides information about the net..

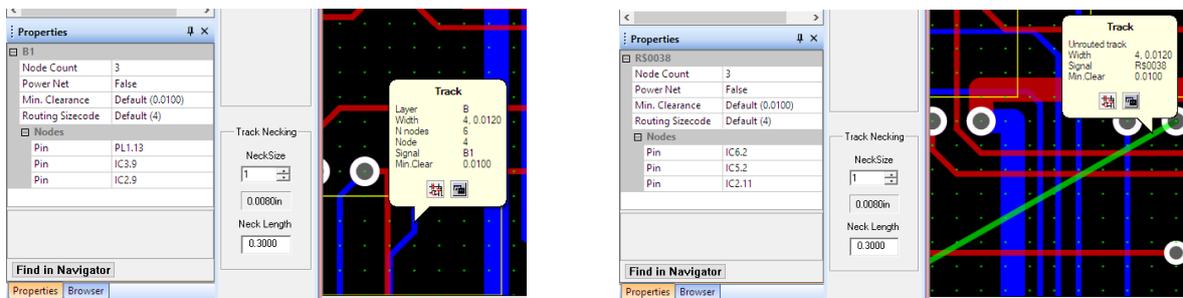


Figure 156

- Unrouted track* Indicates whether the selected connection is either partially unrouted, or completely unrouted.
- Fixed item* Indicates the selected track has been fixed in position, using the *NetFix* commands.
- Layer* Indicates which layer the selected track segment is on.
- Width* Indicates the size code and real size of the selected track segment or unroute.
- N nodes* Indicates the number of nodes (points) along the complete length of track, including the start and end points.
- Node* Indicates the segment number of this particular track segment.
- Signal* Indicates the signal name of the selected track. Unnamed tracks have their internal id number given, i.e. R\$0052. These numbers can be viewed from within the net list editor under the *Net UID* column.
- Min Clear* Indicates the minimum clearance requirement of this particular track/unroute.

Tracks may be selected until the right hand mouse button is clicked, when the previously active command is restored.

Identify > Feature

Used to display information about a selected item. It is typically used to identify text strings and data that has been imported onto a documentation layer, but tracks/unroutes can also be selected.

Select *Identify > Feature*. Select an item. Once it has been selected, an information window appears similar to that shown in Figure 157. A tail extends to the selected item to indicate which item was selected.

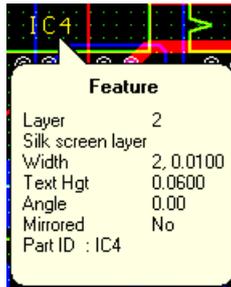


Figure 157

<i>Layer</i>	Indicates which layer and the type of layer (i.e. silkscreen) the selected item is on.
<i>Width</i>	Indicates the size code and real size of the selected item. If text is selected, then this is the width of the line used to draw the characters.
<i>Text hgt</i>	Indicates the height of selected text.
<i>Angle</i>	Indicates whether the item has been rotated in degrees anti-clockwise.
<i>Mirrored</i>	Indicates whether the item has been mirrored.
<i>Part ID</i>	indicates whether the item is associated to a part.

Items may be selected until the right hand mouse button is clicked, when the previously active command is restored.

Identify > Track Measure

Used to display information about track length in the status bar, similar to that shown in Figure 158.



Figure 158

Select *Identify > Track Measure*. Select a track for measurement using the left-hand mouse button. The selected track has a dotted centre-line added to show it is selected.

Each time a new track is selected, its length is added to the displayed value. Selecting a track a second time causes it to be deselected and its length subtracted from the displayed value.

Clicking the right hand mouse button causes all selected tracks to become deselected. Clicking again terminates the measurement mode.

The measurement tool will not measure arc tracks.

Tools > Placement & Routing command

This command can be used to return to the main manual placement and routing commands after selecting one of the commands from the *Tools* menu which introduces a feature specific set of commands. Often a left-pointing arrow icon is available to perform the same command.

Autoplacement (Tools > Autoplace)

The auto-placement commands are used to transfer parts automatically from the tray onto the board, using the interconnections and user defined settings as guides for optimum part positions.

Before parts can be placed on the board, they must appear in the *Tray*. It is therefore possible to concentrate on areas of the layout by adding specific parts to the tray.

Each outline must have its auto-placement "footprint" defined in the outline editor, as it is this outline that is used by the automatic placement routine. The silk screen outlines are not taken into account.

Due to external requirements, some parts such as connectors, must be positioned in specific locations. These parts must be manually placed and fixed in position, before using the auto placement routine to ensure they are in the correct location.

Parts are automatically positioned on the layout according to the "pull" of already placed parts. It is therefore a requirement to have at least one part placed on the board before commencing auto-placement. The pre-placed parts must be connected to parts in the tray.

Parts can be added to the layout in the order in which they appear in the tray (sequential) or the most heavily

connected parts in the tray can be added first (costed). Both methods position the parts according to the "pull" of existing placed parts. The choice between the two methods is made in the Autoplace dialogue bar.

During autoplacement, the part datums are placed on user-defined placement grid intersections. It is suggested that a coarse placement grid should be used when placing the larger parts so that they are spread uniformly across the layout, followed by a finer grid when placing the smaller parts. Placing the larger parts, such as ICs, resistor networks, etc. first is often a good idea, to ensure there is sufficient space for them.

The parts can be placed at a specific user defined orientation or the automatic placement routine can be allowed to choose the orientation of each part from its list of permitted orientations, these are defined in the *Autoplace > Control Orientation* window. Giving the placement routine the freedom to rotate parts may improve the routability of the board, however some design rules insist that all parts lie in the same direction, so this would not be possible.

The parts are placed on the board on a user defined grid or "placement cell". The cell size is defined in the Autoplace dialogue bar. The required clearance between the bounds of parts is also defined there.

Keep-out areas are not recognised by the autoplacement tools.

Once the parts have been placed on the board, the user can then use his or her skill and judgement to "fine tune" the part positions if necessary using the *Parts* commands, to achieve a layout that can be routed with confidence.

Autoplacement dialogue bar

When the Autoplacement tools are selected, a dialogue bar similar to the one shown in Figure 159 appears.

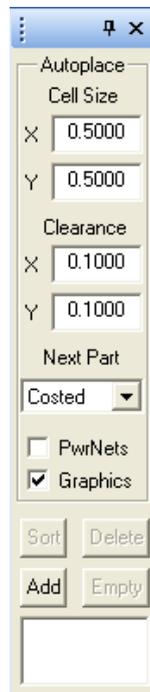


Figure 159

Cell size, X and Y

The values entered here define a placement grid over the board. The placement cell size being the space between the grid lines.

Part datums may be placed on any of the grid intersection points provided the autoplacement footprint of the part does not violate the clearance requirements also specified.

The placement grid radiates from the grid origin, as does the routing grid. By default, this is the lower left hand corner of the working area, however it can be moved using the *Grid* commands.

If a small placement cell size is selected, perhaps 0.05" by 0.05", the automatic placement routine has more placement positions to choose from, which increases the time taken to place all the parts. If a very large cell size is selected, the process speeds up, but there may be insufficient cell positions to place all the parts.

If larger parts such as ICs are being placed, select a larger cell size in order to help align the parts. Routing (whether automatic or manual) will be made easier if the part pads are in rows and columns. For instance, when placing 14 and 16 pin DIL's horizontally, a placement grid of x=1" and y=0.5" could be used. If larger DIL's were being placed, the grid could be increased. If a mixture of DIL's were being placed, it might be sensible to

decrease the X grid to permit smaller DIL's to fit between the spaces left by the larger DIL's.

When placing discrete parts, a smaller cell size should be used to allow the parts to fit into the spaces left by the larger parts.

If a large clearance is required between parts, then a smaller cell size may be required in order to position all the parts on the layout.

Values may be entered in inches or millimetres, depending on the current *Edit > Units* setting. The appearance of the decimal point indicates which units are being used. A dot is used in imperial values and a comma in metric values.

Clearance, X and Y The values entered here define the minimum space that must be left between the auto-placement footprints of adjacent parts.

Values may be entered in inches or millimetres, depending on the current *Edit > Units* setting. The appearance of the decimal point indicates which units are being used.

Next Part Choices are *Sequenced* and *Costed*. Parts are added to the layout from the tray in the order selected here – typically *Costed* is used.

Sequenced selected when the parts must be placed on the board in the order or "sequence" that they appear in the tray. The automatic placement routine assesses the connections of parts that are already on the board, and places the next part from the tray in the best position possible, according to its connectivity and the rules laid down in the dialogue bar. Often used with a fine cell size to determine whether all the parts can physically fit on the board.

Costed selected when the automatic placement routine can choose which part from the tray it will place next. The automatic placement routine assesses the connections of parts that are already on the board and those in the tray. The part from the tray that is most heavily connected to those already on the board is chosen next and placed in the best position possible, according to its connectivity and the rules laid down in the dialogue bar.

Power nets Parts are placed on the board by the "pull" from signal connections of parts already placed on the board. This setting allows power connections to be included in the pull if required. Tick the box to include the power connections.

For example, when placing digital type boards, generally speaking, the power supplies do not affect the positions chosen for parts. Power nets can therefore be excluded when the automatic placement routine is assessing the pull of connections on a part from the tray.

When placing analogue type boards, the power supplies do affect the positions chosen for parts. It is therefore necessary to include the pull of power connections when assessing the pull of connections on a part in the tray.

Graphics The automatic placement routine assesses every possible placement cell to determine the best position for each part. This setting allows the operator to see the cells which makes it easier to choose placement cell sizes.

The placement cells are displayed with blue lines. As the cells are assessed during auto-placement a red box representing the footprint of the part is displayed.

Displaying the graphics slows down the placement process. However, it is useful to see the cells being assessed, as it may help your choice of cell sizes. It also shows that something is happening. The scanning process can take some time if parts are permitted at any angle, and a very small placement cell size has been selected.

Tick the box to see the placement cells.

Add, Sort, Delete, Empty these commands control the parts that appear in the tray and therefore assessed for auto-placement. Refer to the *Parts > Place* command for details on the tray.

Autoplace > Control Orientation

This command is used to control whether the autoplacement tools may rotate parts. When selected, a window similar to the one shown in Figure 160 appears.

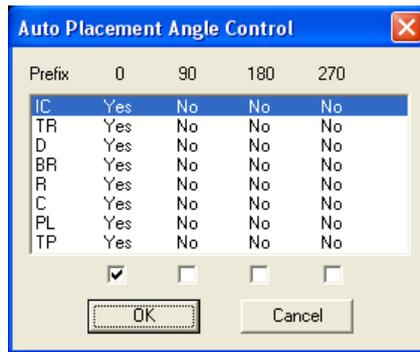


Figure 160

Parts are separated into groups by their prefix names. If Yes appears in the 0, 90, 180, 270 degree columns for a particular prefix, then the auto-placement routine is allowed to rotate parts with that prefix to those angles, to achieve a better placement result.

Yes must be selected for at least one of the angles for each prefix, otherwise the parts cannot be added to the layout.

No stops the auto-placement routine placing parts with that prefix at that angle.

Outlines are created at 0 degrees. Rotation is anti-clockwise.

To change a setting, select the prefix and the line is selected. Select the tick-boxes underneath the columns to toggle the settings between Yes and No.

Autoplace > Replace Part

This command is used to assess the position of a selected part on the layout. If a better position can be found for it, the part is moved. Existing parts are not moved out of the way.

Parts that have been fixed in position with the *PartsFix* commands, or have routed tracks attached to them cannot be selected.

Parts on the top of the board are selected with a click of the left hand mouse button, parts on the bottom with a click of the right hand mouse button.

If the parts were placed automatically and nothing has changed on the board, this command will have no effect, as its decisions are based on the same strategies as the placement routine. However, if some parts have been moved manually, the placement cell size or clearance requirements changed, then the part may well move to a more appropriate position.

Select *Autoplace > Replace Part*. Select the datum of the part with the appropriate mouse button. The board is scanned, and if a more appropriate position can be found for the part, the part moves. Displaying the graphics (Autoplace dialogue bar) indicates what is happening.

Autoplace > Place Parts

When selected, automatic placement starts. Parts from the tray are placed onto the board in the order controlled by the Autoplace dialogue bar.

Each part is placed on the board according to the pull of connections from parts that have already been placed. It is therefore essential to pre-place at least one part before selecting this command. That part must of course be connected to parts in the tray.

Typically, parts such as connectors are manually placed first as their placement is controlled by external hardware requirements. They are also usually heavily connected to other parts, so will influence part placement to a greater extent than for instance, discrete parts.

Other parts whose positioning is critical should also be manually placed. For instance tooling holes, pots, switches, LEDs, etc.

Once critical parts have been placed, the required parts added to the tray and the Autoplace dialogue bar defined, auto-placement can commence. Select *Autoplace > Place Parts* to start auto-placement.

Once auto-placement has been started, the routine scans every placement cell on the board as a possible final position for the part it is placing. As parts are being placed, their name is displayed along the bottom of the screen. If more than one orientation is allowed for the part, the board is scanned again to determine whether a better position could be obtained if the part were rotated. This process is repeated for every permitted orientation of the part. The part is then positioned in the most appropriate position according to the rules laid down and connectivity. The next part is chosen from the tray and the process repeated.

Autoplace > Replace All

This command is used to assess the position of all the unfixed, unrouted parts on the layout. If better positions can be found for them, the parts are moved.

Existing unfixed, unrouted parts are moved out of the way if necessary, and new positions found for those parts.

Parts that have been fixed in position, or have routed tracks attached to them are not moved out of the way.

If the parts were placed automatically and nothing has changed on the board, this command will have no effect, as its decisions are based on the same strategies as the placement routine. However, if some parts have been moved manually, the placement cell size or clearance requirements changed, then some parts may well move to more appropriate positions.

The parts are evaluated, starting with the most heavily connected, unfixed, unrouted parts on the board.

Select *Autoplace > Replace All*. Each part is evaluated in turn and if more appropriate positions can be found for them, the parts are moved. Displaying the graphics (Autoplace dialogue bar) indicates what is happening.

Auto-routing (Tools > Autorouter)

The integrated auto-router is a "rip-up and re-try" auto-router. This means that if there are any connections remaining after it has attempted every connection, it will remove or "rip-up" tracks it has already routed in order to find space for those that are left. The tracks that have been ripped up are then subsequently routed elsewhere "re-tried". In theory this process continues until there are no connections left, however it is possible to design a board that is impossible to route completely.

The auto-router works on a user-definable grid. However very fine routing grids increase the amount of time and memory required to route the board because more routing channels have to be assessed and held in memory. If possible, it is better to use a coarser routing grid, especially if the tracks will be manually adjusted at some later stage.

Optimising passes can be performed which "tidy" up tracks that have already been routed.

Automatic track necking can be used when permitted by the user, to improve routing success.

Critical connections can be routed manually and fixed in position if required before running the auto-router.

The router can be used to route all the connections on the board, one particular connection net or all the connections radiating from a selected part.

The router has different strategies for power connections, memory type signal connections and orthogonal signal connections. These are described later.

Preparations for auto-routing

Before attempting to route a board, a great deal of thought and care should be exercised over the placement of parts. A good part placement is critical in order to achieve success when either manually or automatically routing a board. The sizes used for pads, tracks, gaps and routing grid will also affect the success rate of the auto-router.

Placement

The auto-router can be setup to work on any routing grid, though very fine grids will make subsequent manual adjustments less easy.

Before the placement is finished, try to assess the auto-routing grid that will be used. The final position of parts, and consequently their pads, can then be adjusted to fall in line with this grid. If care is taken over the positioning of parts then the need for very fine routing grids can be avoided.

If parts are placed so that their pads line up with the routing grid, more routing channels will be available for routing. Connections to pads that are off the routing grid will be routed, but the tracks may block adjacent routing channels. It is therefore sensible to minimise "off the routing grid" pads where ever possible.

Pad, via, track, clearance and grid sizes

The choices of pad, track, clearance requirements and routing grid also affects the success rate of the router. It is possible to choose sizes that will not allow for instance a single track between the legs of IC pins. Or you may have calculated the pad, track and clearance sizes correctly, but the minimum routing grid combined with the position of parts will not permit these things to happen.

For instance, take a DIL package whose pins are on a 0.1" pitch with the following design rules:

Pad diameter	= 0.062"
Track size	= 0.015"
Min. clearance	= 0.012"

In this case, irrespective of the routing grid chosen and the position of the part, a track cannot pass between adjacent pads of the part because of insufficient space (a minimum pad pitch of 0.101" would be required).

If the pad size were decreased to 0.060" (more typical than 0.061") a track could pass between the centre of adjacent pads. In practice this could be achieved provided the part were placed on a 0.1" grid and a 0.050" routing grid were used.

Whether the router can route the track between the pads also depends on the position of the part in conjunction with the routing grid.

The router can be allowed to neck tracks in order to travel between adjacent pads or obstacles. These functions should be used to assist the auto-router in congested areas of the layout. They should not be used as a reason

for not calculating sensible pad, track and clearance sizes.

Pre-routing

Because of some design requirements it can be necessary to pre-route some critical power or signal connections and then fix them in position so that the router cannot move them.

If this is the case, manually route the connections using the *Mroute* commands and then fix them in position using the *Netfix* commands. If the tracks are not fixed in position, the router will work around the existing tracks to start with, but it may move them in order to complete other tracks during the rip-up stage.

When manually routing the tracks, try to route them so that they do not lie on a finer grid than the auto-router will be setup to use. Also try to route them so that they run horizontally and vertically on the same sides that the router will be setup to use.

Bearing these points in mind will help increase routing success. There will be less manual routing to complete after the router has finished.

Too many fixed tracks can affect the success rate of the router as they act as obstacles to the router.

Memory requirements

A "large" board as far as memory requirements are concerned, is not only determined by its physical size. The size of the minimum routing grid chosen, the number of routing layers, the number of component pads, fixed tracks, obstacles, etc. all have to be stored in memory. Therefore halving the routing grid or increasing the number of layers makes the board "larger" in terms of memory requirements, even though the physical size has not changed.

Keepout areas/lines

Keepout areas/lines can be defined in the board profile or artwork editor to define areas in the board that the auto-router is not allowed to use or to act as obstacles that the auto-router has to go around.

Too many keepout areas can affect the success rate of the router as they act as obstacles to the router.

Summary for Routing Success

- Spend time on achieving a good placement
- Ensure as many component pads as possible lie on a coarse routing grid
- Choose optimum sizes for pads, tracks, vias, clearance and minimum routing grid
- Ensure any pre-routing matches the auto-router's rules
- Minimise fixed tracks wherever possible

Pre-routing checklist

- Place all parts within the board profile
- Check sizes of pads, vias, tracks and clearance
- Fix all pre-routed tracks where required
- Add keep-out areas as required

Accessing the auto-router

With the artwork editor open, select *Tools > Autoroute*. A dialogue bar appears which contains all the auto-routing commands - these command are also located in the *Autoroute* pull down menu.

Router dialogue bar

Setup button.

When selected, a window appears similar to the one in Figure 161, listing the 8 different routing strategies available for use.

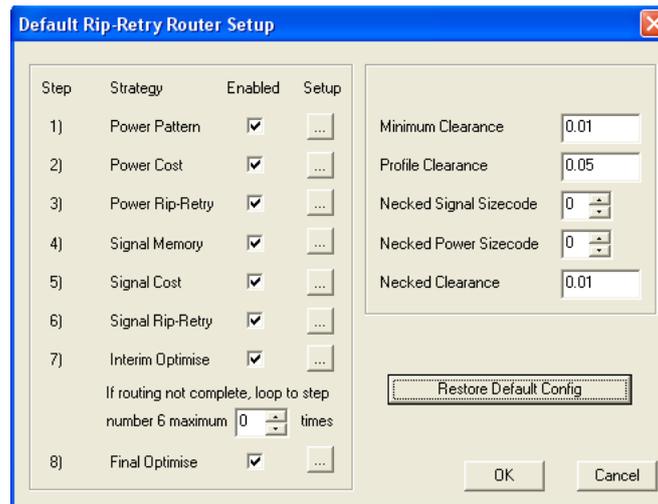


Figure 161

Each routing strategy has its own user definable setup. These are all described in the chapter titled *Configuration Folder* and the heading *Configuration – Rip Retry Autorouter costs*.

Different types of layout may use different combinations of the strategies with different settings. The setup for each strategy can be viewed by selecting the Setup button from alongside the strategy.

Typically power/thicker/critical connections are routed first and completed prior to routing the other connections. This can be accomplished by just enabling the power strategies, or selecting named nets for routing. Once completed, the power strategies can be switched off and the other strategies enabled.

Enable/disable the routing strategies required and check all the other settings in the window are set as required, then close the window.

The routing settings are stored with the design. They were originally copied from the master router settings. The routing setup can be changed within the design without affecting other designs or the master set.

If the master routing setup is modified, only new designs will be affected. Existing designs maintain their routing setup.

Layers button

Select the *Layers* button from the dialogue bar. The auto-router can route on up to six layers simultaneously. This window is used to control which layers are available for auto-routing and the preferred routing direction for each layer. The options being Horiz(ontal), Vert(ical) or Any.

Select the routing layers to be used and the direction of routing on each layer. (Only copper layers can be selected for routing.) For best results, take care that the chosen direction corresponds to any pre-routed fixed tracks on the layer. Typically horizontal tracks are placed on one layer, vertical tracks on the other, this is repeated for each pair of routing layers. (The final optimisation routine will remove excess vias that are introduced.) Close the window.

Starting the router

The router is now ready to be started. Select one of the following buttons to start the router:

- All* when selected, the auto-router will attempt every connection on the board providing the appropriate strategies have been enabled from the Router setup window. The auto-router starts straight away.
- Signame* when selected, the auto-router will attempt just the connections of a named signal. Type in the name of the net to be routed when requested. The auto-router starts as soon as the signal has been identified.
If the selected signal is a power net, then the enabled power strategies are used, if it is a signal net then the enabled signal strategies are used.
- Part* when selected, the auto-router will attempt just the connections radiating from a selected part. This is achieved by selecting the PART command followed by the part to be routed. The auto-router starts as soon as the part has been identified, using the appropriate enabled strategies for each net.
- Net* when selected, the auto-router will attempt just the connections of a selected signal. The auto-router starts as soon as the signal has been identified.
If the selected signal is a power signal, then the enabled power strategies are used, if it is a signal connection then the enabled signal strategies are used.

Whilst the router is running

The router uses the enabled routing strategies from the setup window, in the order listed. It repeats each strategy until all the connections within that category have been completed or the specified number of passes has been completed even if all the connections have not been completed.

As the router is running, information appears along the bottom of the screen to indicate what is currently happening.

<i>Mode</i>	Indicates which strategy is currently running.
	Power-P Power pattern routing strategy
	Power-C Power cost routing strategy
	Memory Memory routing strategy
	Signal-C Signal cost routing strategy
	Optim Optimising routing strategy
<i>Pass</i>	Indicates which pass of the current strategy is being attempted.
<i>Nroutes</i>	Indicates the total number of routes that must be attempted.
<i>Ndone</i>	Indicates the total number of routes completed by the auto-router.
<i>Nvias</i>	Indicates the total number of vias currently used on the board.
<i>Failed</i>	Indicates the number of connections that have not been routed.
<i>Vio</i>	Indicates the number of violations currently present on the board. A violation is either a short circuit or clearance error. The router is permitted to make violations whilst it is attempting to find a path for a connection. These violations will be rectified before routing is completed.

Interrupting the auto-router

The auto-router can be interrupted at any time by pressing the *STOP* button from the dialogue bar. After the router has completed the connection or route it is currently attempting, a window appears. This could take a few minutes, depending on the size of the net.

The window contains choices to, stop routing, skip to the next pass or continue routing.

Copper Fill (Tools > Copper Fill)

The copper fill commands are used to add areas of copper to a copper (tracking layer) layer of the board. The areas of copper are attached to the tracks and component pads of a selected signal, whilst avoiding all other pads, vias and tracks. Copper fill is typically used for screening purposes. (Copper fill should not be confused with power planes which are discussed separately.)

The filled area can either be a solid area of copper or a cross-hatched area of copper. It is made up of parallel horizontal and/or vertical lines that cross one another according to the settings in the *Fill* dialogue bar.

Two types of filled area can be added, one with a smooth edge or one with a non-smooth edge. The smooth edge fill gives far superior results. The non-smooth edge option is present for historic reasons only.

In "*smooth edge*" mode, a track is drawn around existing tracks and pads that are not part of the selected net, leaving a space (minimum clearance) away from those tracks and pads. This track will not necessarily be "on grid". The area inside this edge is filled with copper tracks which are placed on the grid as specified in the fill dialogue bar, but the end points of those tracks will butt up against the edge track, so will not necessarily be "on-grid" either.

Note: the smooth edge assumes that user-defined pads are rectangular in shape, so it will not trace their outline precisely.

If the "non smooth edge" mode is selected, the copper fill radiates from a point on the current grid, at the pitch specified. It is therefore possible for some pads on the selected net to be "missed" by the copper fill. It is therefore necessary to ensure that the current grid origin is in the position required and a suitable fill pitch is selected. The non-smooth edge will give a jagged edge around angled/curved tracks, and a jagged edge if the fill is added in only one direction.

Whether a solid or cross hatched filled area is added, pads on the selected net are joined to the filled area according to the settings in the copper *Fill* dialogue bar.

Copper fill should be added after all routing has been completed and checked, otherwise there will be no space to route the connections, or correct errors.

The area to be filled is defined by the user, and can include keep-out areas if the smooth edged option is chosen. The copper forming the filled area can be modified using the *Amend > Movepoint/Delete Track* commands if required.

The user first defines the settings that the filled area has to conform to, in the *Fill* dialogue bar. The perimeter of the required filled area should be defined using the *Boundary > Add Line* command, and modified if necessary with the *Boundary > Corner/Delete Point*, etc. commands. The *StartFill* commands are used to add the copper area.

The copper area connects to all the component pads and tracks that are connected to the selected/named signal. Vias on the selected/named net can be connected or exposed as defined in the *Fill* dialogue bar. Clearance is provided around all other pieces of copper.

Notes: (i) any tracks that have been entered using the *Amend* menu commands will not be recognised as part of a specific net until they have been validated by the artwork checking routines. This means that the fill routine will leave a clearance around them until they have been validated. If the fill was supposed to be attached to the new tracks, the existing fill would have to be deleted and added again once the tracks had been validated. Short circuits can also cause the copper areas to avoid tracks – ensure there are no short-circuits before adding the copper fill to avoid this happening.

(ii) if more than one area of fill is to be added to a layer, improved performance can be achieved if the existing fill is saved using the *Filled Copper > Save Copper* command, then deleted (*Filled Copper > Delete All*), before adding the next area of copper fill. Once the additional fill has been added, the saved fill can be merged back into the artwork, *Filled Copper > Merge Copper*.

(iii) adding copper filled areas with the smooth-edge option enabled can result in areas of isolated copper fill, depending on the setting in the *Fill* dialogue bar. These isolated areas will be identified by the artwork checking routines and can subsequently be deleted automatically if required (using the *Filled Copper > Delete All Isolated Copper* command).

The speed of the actual fill process varies depending on the fill pitch and type, the size of the filled area and the speed of the machine. Smooth edged fill, finer grids and/or larger areas take longer. The amount of memory installed on the machine governs the amount of copper fill that can be added in two ways. Finer fill pitches mean that more memory is required to map the data. If it isn't available the polygon areas have to be reduced in size. The amount of memory also governs the amount of data that can be added to a layout. The Artwork Properties tab indicates the number of points used on each layer.

Fill dialogue bar

When copper fill is selected, the copper fill dialogue bar, similar to that shown in Figure 162, appears. It is used to set the rules for the copper fill.



Figure 162

Layer

this determines which layer the copper fill will be added to, or deleted from. It is the "active" layer. All layers are available for selection.

If a pad is selected as the datum, it must be on this layer.

If the active layer has been assigned a title (Configuration > Layer Assignments window), this is also displayed.

The colour of the box helps to identify and also select layers; it can be selected to produce the layer assignments list which aids layer selection when using the spin-controls to change layers might be cumbersome.

Size

This setting defines the width of the copper lines that make up the filled area. The size code corresponds to the size codes in the Track Sizes table. The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside.

The current size assigned to the size code number is shown underneath. The units in use (inches/mm) are those set when the dialogue bar was opened.

The filled area can either be a solid or cross hatched area of copper. Solid areas are created if the pitch and the thickness of the lines in the filled area are the same size. (Typically the track size is set to a few thou larger than the pitch to allow for inaccuracies during plotting.) Cross hatched areas are created if the fill pitch is larger than the track thickness.

<i>Boundary</i>	A boundary must be defined that defines the area in which the copper fill will be added. When the boundary is added, it is assigned the number set here (from 1 to 99). The boundary is not saved with the copper fill. If it is required to delete then re-add the copper fill at some later stage, selecting the appropriate boundary number will restore the boundary.
<i>Edge clearance</i>	Specifies the minimum distance required between edges of the board profile and keepouts and the copper forming the filled area. If the non-smooth edged fill is used, the actual distance can sometimes be a little larger depending on the pitch and the thickness of the copper lines.
<i>Minimum clearance</i>	Specifies the minimum distance required between existing copper pads & tracks and the copper forming the filled area. The smooth-edged copper fill gives precedence to specific clearance requirements defined on the schematic or in the net list if they are greater than the minimum size specified here. If the non-smooth edged fill is used, no attention is paid to specific clearance requirements for individual nets and the actual distance can sometimes be a little larger depending on the fill pitch and the thickness of the copper lines.
<i>Pitch</i>	Specifies the distance required between centres of each parallel line of copper that forms the filled area. To achieve a cross-hatched fill the fill pitch should be larger than the selected fill track size (fill pitch > track width). To achieve a solid fill, the fill pitch should be equal or smaller than the selected fill track size (fill pitch ≤ track width). In order for a solid fill to obtain entry to areas within the board with a narrow entry channel, it may be necessary to use a fine fill pitch and track size.
<i>Direction</i>	The copper fill can be made up of parallel horizontal and vertical lines, or just horizontal, or just vertical parallel lines. Select the arrow alongside the setting and choose from the list that appears. <i>Hatched</i> the fill is made up of horizontal and vertical strips of copper. Only choose this option if a cross-hatched (not solid) effect is required, or an un-smooth edge is selected. Selecting hatched unnecessarily will increase the point count on the layer and increase the design size for no benefit. It will also increase the size of the output files. If Hatched is selected, the filled area is made up of horizontal and vertical copper lines. Jagged edges can be left around angled tracks, or perimeter lines, unless smooth edge is selected. <i>Horizontal</i> the filled area is made up of only horizontal strips. Use this option if a solid filled area with a smooth edge is required. <i>Vertical</i> the filled area is made up of only vertical strips. Use this option if a solid filled area with a smooth edge is required.
<i>Use keepouts</i>	This setting is only operational if the "Smooth edge" setting is also selected. When selected (ticked), the copper fill will not cross any keepout lines. If it's not selected then keepout lines are ignored.
<i>Smooth edge</i>	If smooth edge is selected (ticked) the copper fill is surrounded by a track that forms a smooth edge around the copper fill. This line clears other pads, tracks and vias by the distance specified as "minimum clearance". If unselected, the copper fill will just be made from horizontal and/or vertical lines, on the selected fill pitch. This will result in a ragged edge around angled/curved tracks and in some cases a slightly larger than specified minimum clearance.
<i>Min Connections/region</i>	Controls the minimum number of part pins that must be connectable to a copper fill region for the region to appear in the fill result. Note: setting this value to zero will result in the creation of isolated copper areas.
<i>Via Holes</i>	This option provides control over how copper fill connects to via holes that are members of the fill net. Available options are:

No Connection	No heat relief stubs are inserted so the vias remain exposed.
Heat Relief	Heat relief stubs are inserted to connect the via to the fill region. Providing the fill track size is smaller than the pads in use, the pads will not be completely covered.
Cover	The copper fill floods over the top of the via.

Heat-relief stubs are made from up to four track segments that connect to the via, using the track size assigned for the fill.

Component Pins This option provides control over how copper fill connects to component pins that are members of the fill net. Available options are:

Heat Relief	Heat relief stubs are inserted to connect the part pin to the fill region.
Cover SMD	Drilled component pins are connected with heat relief stubs. SMD part pins (undrilled pins on layers T or B) are covered over by the copper fill.
Cover All	Copper fill covers all part pins.

Heat-relief stubs are made from up to four track segments that connect to the component pin, using the track size assigned for the fill.

Boundary > Add Line

Used to define the perimeter of the copper filled area.

If an un-smooth edged fill will be added, any exclusion areas within the filled area can also be defined using this command.

If smooth edged fill is being added, only one boundary can be defined - use keepout lines to stop the fill entering specific areas if required.

Any enclosed straight-line polygon shape may be defined. Polygons within the outer polygon will act as keep-out areas for the copper fill if the un-smooth edged fill is being added, providing the datum point is selected within the main polygon area.

The *Boundary > Corner/Delete Point*, etc. commands can be used to modify the line that is added.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Choose a boundary number from the dialogue bar on the left-side of the screen. This number will be stored with the boundary. To restore the boundary at some later stage (as it is not saved with the copper fill), select the boundary number that was current when the boundary was added.

Select *Boundary > Add Line*. Position the cursor at a point on the desired perimeter outline and click the left hand mouse button. Move the cursor to stretch the line and insert corners in the line by clicking the left hand mouse button. Finish by inserting a corner at the starting point of the line to enclose the area, and then click the right hand mouse button to release the line.

Keep-out areas within the main polygon outline are added in the same way if the unsmooth edge is selected.

The area is then ready to be filled.

Boundary > Delete Line

Used to delete the perimeter lines added using the *Boundary > Add Line* command.

A complete line, from its start and end points is deleted, not just one segment.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Boundary > Delete Line*, then select the perimeter outline to delete it - select on a corner/endpoint if the line is difficult to select.

Boundary > Corner

Used to modify the copper fill perimeter outline by adding or moving existing corners in the line.

Corners are *added* and released with clicks of the *right* hand mouse button. Corners are *moved* and released with clicks of the *left* hand mouse button. Once the corner has been selected, clicking the opposite mouse button cancels the operation.

right button = *add* corner
left button = *move* corner
Opposite button cancels

Adding a corner to a line

Select *Boundary > Corner*, then point at the line and click the right hand mouse button. Move the new

corner and release it with another click of the same (right) button. Clicking the opposite (left) hand button before the corner is released, cancels the new corner.

Moving an existing corner in a line

Select *Boundary > Corner*, then point at the corner in the line and click the left hand mouse button. Move the corner and release it with another click of the same (left) button. Clicking the opposite (right) hand button before the corner is released, cancels the move.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Boundary > Delete Point

Used to delete points in a copper fill perimeter outline.

Select *Boundary > Delete Point*, then select a corner in the perimeter outline. The corner disappears.

StartFill > Set Datum

Used to select a *component pin* on the signal that the copper fill will be joined to. The component's pad must be on the same layer as the copper fill.

If you cannot locate a pad easily and it is connected to a named signal, use the *StartFill > Connect to Net* command instead.

Whether a solid or cross hatched filled area is added, pads on the selected net are joined to the filled area with a simple cross where possible. The cross is made using the track size assigned for the fill. Providing the fill track size is smaller than the pads in use, the pads will not be completely covered.

Note: if tracks have been added using the *Amend* menu and have not been validated using the artwork checking routines, the tracks will be avoided even if they are connected to the same net as the selected pad.

When the filled area is added, the fill may not fill the entire polygon area. For instance, if a track cuts the polygon area in two on the same side as the copper fill, the fill would fill the area from the datum pad up to the track, but it would not cross the track. This can lead to unfilled areas. It may be better to assign a signal name to the net (in the wiring list editor) and use the *StartFill > Connect to Net* command.

To add multiple areas of copper fill to a layer, refer to the *Filled Copper > Save Copper* command.

Check that a perimeter outline has been defined and the dialogue bar is set as required, then select *StartFill > Set Datum*. Select a component pad to which the copper fill will be connected. A message should appear in the status bar indicating that a valid datum point has been selected. Select the *Start Fill > Connect To Datum* command to start the copper fill. Messages appear along the bottom of the screen to indicate what is happening.

Start Fill > Connect To Datum

Used to start the filling process after a datum pad has been selected (*StartFill > Set Datum*).

Copper fill should be added after all routing has been completed and checked, otherwise there will be no space to route the connections or correct errors.

A perimeter outline must be defined and a valid datum point selected before this command can be actioned.

Check that a perimeter outline has been defined, the dialogue bar is set as required and a valid datum point selected (*StartFill > Set Datum*).

Select *Start Fill > Connect To Datum*. Messages appear along the bottom of the screen to indicate what is happening.

StartFill > Connect to Net

Used to add copper fill to the layout by entering the signal name of the tracks and pads that the copper should be attached to.

This command can be used instead of the *Start Fill > Set Datum/Connect To Datum* commands to start the fill process, providing the pads are attached to a named signal.

Pads on the selected net are joined to the filled area with a simple cross where possible. The cross is made using the track size assigned for the fill. Providing the fill track size is smaller than the pads in use, the pads will not be completely covered.

When the filled area is added, the fill is added to the entire polygon area. For instance, if a track cuts the polygon area in two on the same side as the copper fill, the fill would fill the areas on either side of the track, providing there were tracks or pads on either side of the track belonging to the specified net. This can lead to many filled areas being added.

Note: if tracks have been entered using the *Amend* menu, and have not been validated by the Artwork Checking routines, the tracks will be avoided even if they are connected to the specified net.

Check that a perimeter outline has been defined, the dialogue bar is set as required and then select *StartFill > Connect to Net*. A window appears, requesting the name of the net that the copper fill should be connected to. Enter the signal name and the filled area is added.

FilledCopper > Delete All

Used to delete all the copper fill from a particular layer, providing it was added using the copper fill commands. Ensure the layer setting in the dialogue bar is set to the layer containing the filled areas to be removed. Select *FilledCopper > Delete All*, the entire copper fill from the layer is removed.

FilledCopper > Delete Area

Used to delete one area of copper fill from a particular layer, providing it was added using the copper fill commands.

Ensure the layer setting in the dialogue bar is set to the layer containing the filled area to be removed. Select *FilledCopper > Delete Area*, point at and select, with a click of the left hand mouse button, a point in the copper filled area that is to be deleted. The filled area is removed.

Note: If the area has a smooth edge, the inner area and edges have to be deleted separately. Select the inner area and once deleted, select a point in the edge, bearing in mind that there may be many edges.

FilledCopper > Delete Node

Used to delete an individual segment from an area of copper fill from a particular layer, providing it was added using the copper fill commands. It is easier to identify the segments if the tracks are being shown as a centreline rather than filled (*View > Lines at Width*).

Ensure the layer setting in the dialogue bar is set to the layer containing the segment of copper fill to be removed. Select *FilledCopper > Delete Node*, point at and select, with a click of the left hand mouse button, an end point of the segment to be deleted. The segment is removed.

FilledCopper > Delete All Isolated Copper

Used to delete all areas of isolated copper fill detected by the artwork checking routine, from all layers, whether visible or not. (Their visibility is controlled from the *View > Isolated Copper* command.)

FilledCopper > Save Copper

Used to improve the performance of the copper fill routines when adding more than one area of fill to a layer of the board. For instance, if there is to be more than one filled area of VCC on a layer, or many areas of fill connected to different signals on a layer.

When adding more than one area of copper fill to a layer, after the first area has been filled, save the fill using this command, then delete the fill (using the *FilledCopper > Delete All* command). Then add the extra area of fill. If more fill is to be added to this layer, repeat this process until all the areas of fill have been added, saved and deleted. When all the filled areas have been added and deleted, restore all the deleted areas of fill using the *FilledCopper > Merge Copper* command.

Ensure the layer setting in the dialogue bar is set to the layer containing the filled area to be saved. Select *FilledCopper > Save Copper*. A window appears requesting a name (without an extension) for the file that is to be created. When <enter> is pressed, the fill is saved within the design. (These files can be removed if required using the *FilledCopper > Delete Saved Copper* command.)

The fill can be restored as required using the *FilledCopper > Merge Copper* command.

Note: all the copper fill from the artwork on the selected layer is saved.

FilledCopper > Merge Copper

Used to restore an area of copper fill to the *selected layer*, from a previously saved fill file.

Ensure the layer setting in the dialogue bar is set to the layer that the fill will be merged to. Select *FilledCopper > Merge Copper*, a window appears listing all the copper files held in the design.

Select the fill file required, followed by the *Merge* button to restore the area. Repeat for all the fill files required.

Take care to only restore the fill file once, or there will be copper fill on top of copper fill (unless the fill was deleted before the second merge).

FilledCopper > Delete Saved Copper

Used to delete copper fill files that were created using the *FilledCopper > Save Copper* command. These files should be deleted once they are no longer required.

Select *FilledCopper > Delete Saved Copper*, the list of fill files within the design appears. Select the file to be deleted, followed by the *Delete* button. Repeat for all the files to be deleted.

Power planes (Tools > Powerplane)

What is a power plane?

A power plane is an internal layer of a multi-layer board. A power plane is shown in Figure 163.

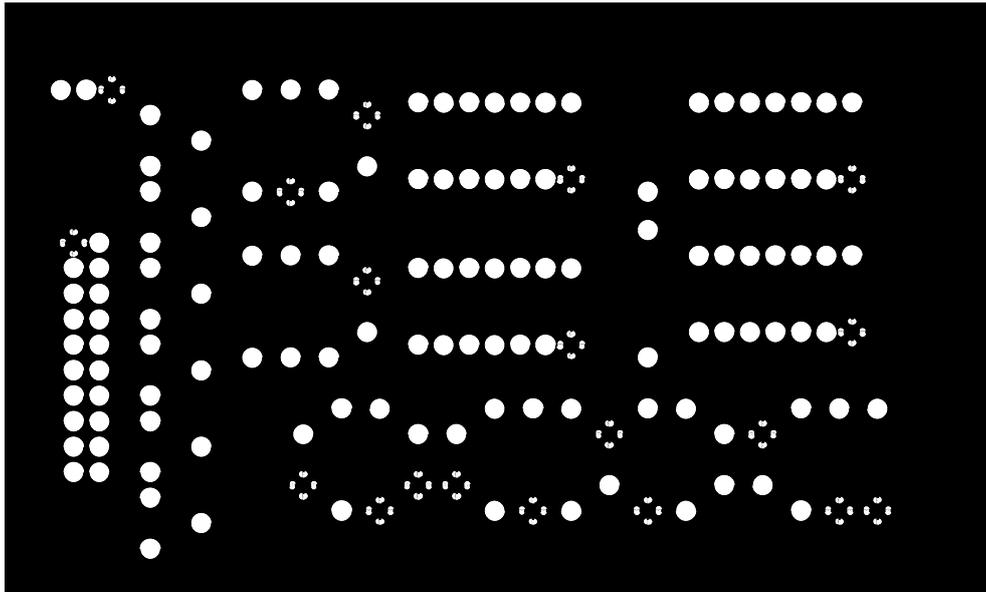


Figure 163

It is made from a solid sheet of copper (a plane) which connects all the plated through holes on the board together, unless they have a clearance area or "anti-pad" around them - the round areas in Figure 163.

Heat-relief pads are added around holes that should be connected to the plane; they are added to aid the manufacturing process, they are not required as far as electrical connectivity is concerned. They are the "dotted" areas in Figure 163.

Vias that are connected to the plane can have a heat-relief pad added or not as required – because the vias are not generally soldered to, the heat-relief pad is not essential – the heat-reliefs around vias can also cause problems by isolating other holes that should be connected to the plane.

Generally speaking, the plane of copper is usually connected to a power rail, hence the name "power" plane, but any connection could be routed in this way.

Power planes can be used when it is impossible to complete a layout on two layers. Removing the power tracks from the outer layers of the board can leave enough room to complete the signal tracking. Power planes are also used for various design reasons as, for instance they can reduce cross-talk between signals.

A reverse image of the power plane is displayed **except** when the power plane tools are being used in the artwork editor.

When producing output files, the pads are usually plotted and a reversed image subsequently produced by photographic techniques. However, some outputs, (Gerber photoplot and Postscript) can be produced as they are finally required, providing the plotter can accept the appropriate commands.

What is a Split Power Plane?

A split power plane is similar to the power plane described above, except that more than one net is connected using the copper plane, but they are isolated from each other as shown in Figure 164.

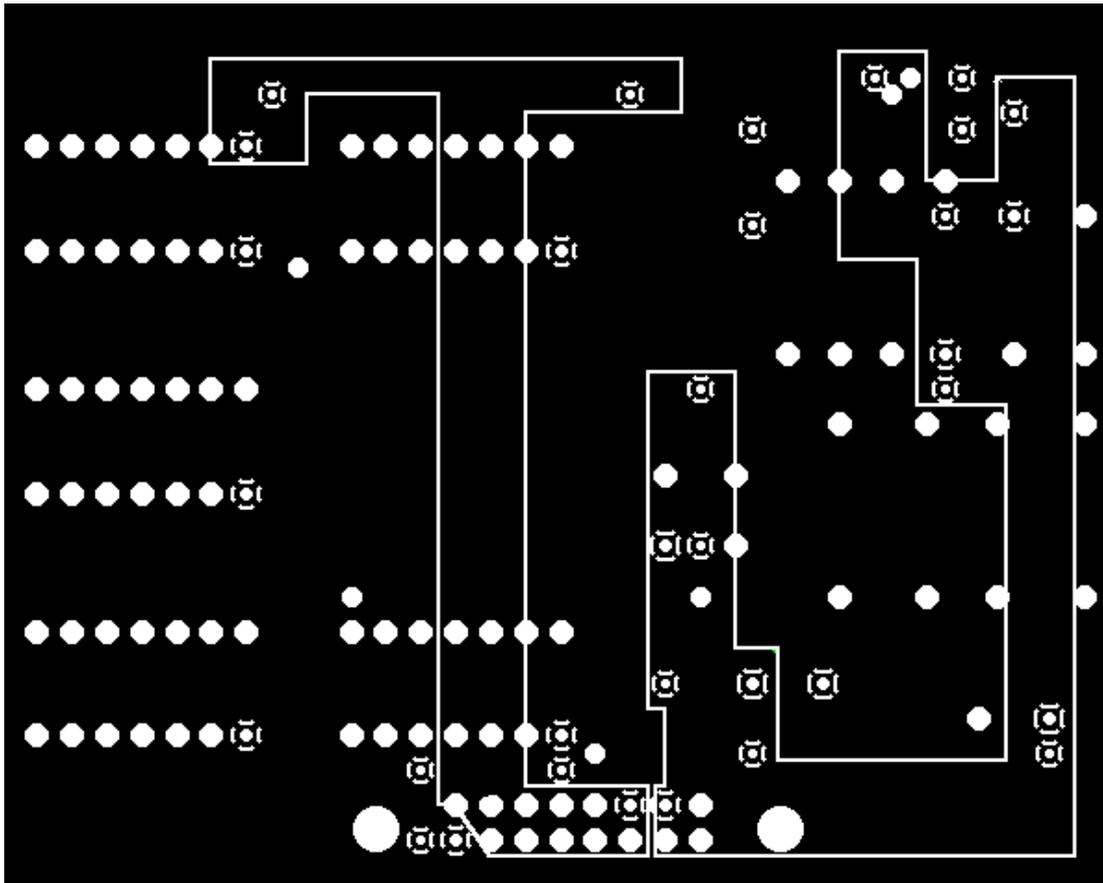


Figure 164

Power plane layers

When a layer is defined as a power plane (or split-powerplane) it is initially a solid sheet of copper (this is not visible in normal viewing mode) and every hole through the layer is connected to it. Anything that is added to that layer actually removes copper. So any tracks or pads or text that are added to the layer, are “not copper”.

Once the power plane tool has been exited, the reverse image of the plane is displayed, so pads/tracks and text will be seen, but they are removing copper from the plane.

Once a power plane has been generated, the Amend commands can be used to add/modify/delete the pads/tracks on the plane, but bear in mind they are “not copper”.

Creating a power plane, or split power plane - basic procedure

These notes assume the design is open, a parts/net list has been created, and the part placement is complete. Before power planes can be generated, the following items have to be defined and/or checked.

The connection to be routed on the plane has to have a signal name (in the schematic or wiring list editor) and should also be defined as a power rail (*Configuration* folder, *Power names*).

Ensure the heat-relief and anti-pad sizes have been set as required (*Configuration* folder, *Sizes Table*, *Heat-relief/Anti-pads* tab).

Ensure the layer assignments have been set as required (*Configuration* folder, *Layer Assignments and Ordering*).

Routing the board

Generally speaking, the power planes should always be generated AFTER all the other connections have been completely routed, otherwise errors will be introduced to the layout if vias are added, parts moved, etc. However, the planes can be created at any stage, provided the planes are created again at the end of further changes.

If modifications are made to the layout, the power planes should always be re-created. Always run the artwork checks as a final check on the artwork.

Surface mounted components cannot be connected to the power plane until a connected via is added to make the connection through to the plane. Once the plane has been defined in the layer assignments table, a connection “blob” is provided from these sm pads (see Figure 165, left) and it has to be moved into an appropriate location. (If the “blob” can’t be seen, ensure the power unroutes are visible and at width (controlled from the *View Control* dialogue bar.)

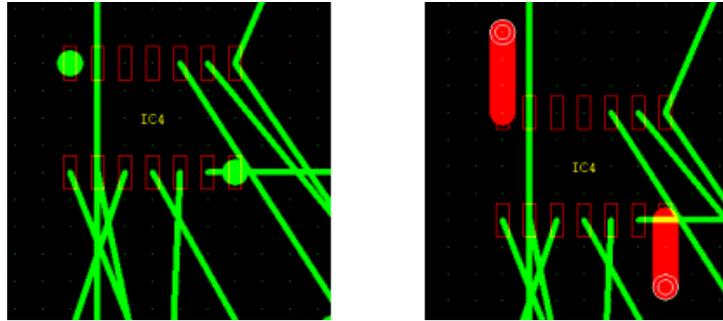


Figure 165

This “blob” should be pulled out using the *Mroute > Corner* command (right-button) and a track and via will appear to connect the smd pin to the inner layers Figure 165, right.

The auto-router’s *power pattern* strategy or Spectra/Electra auto-routers will do this automatically.

Once all routing is complete, create the power planes.

If the Spectra/Electra auto-router will be used and the board has split power planes, the split planes have to be created prior to submission to the router.

After any routing (manual/auto), the planes should be re-generated (split-polygons modified if necessary).

Power Plane Tools

Once all the routing on the board is complete, or the board will be submitted to the Spectra/Electra auto-router, the power planes are ready to be generated.

Note: when a power plane is generated, everything from the power plane layer(s) is automatically deleted first, including anything added to that layer by the user. This ensures the power plane is correct.

Select the power plane tool (*Tools > Powerplane*). A new *Powerplane* dialogue bar appears and the display shows one plane layer similar to that shown in Figure 166. The heat-relief and anti-pads are shown in the plane.

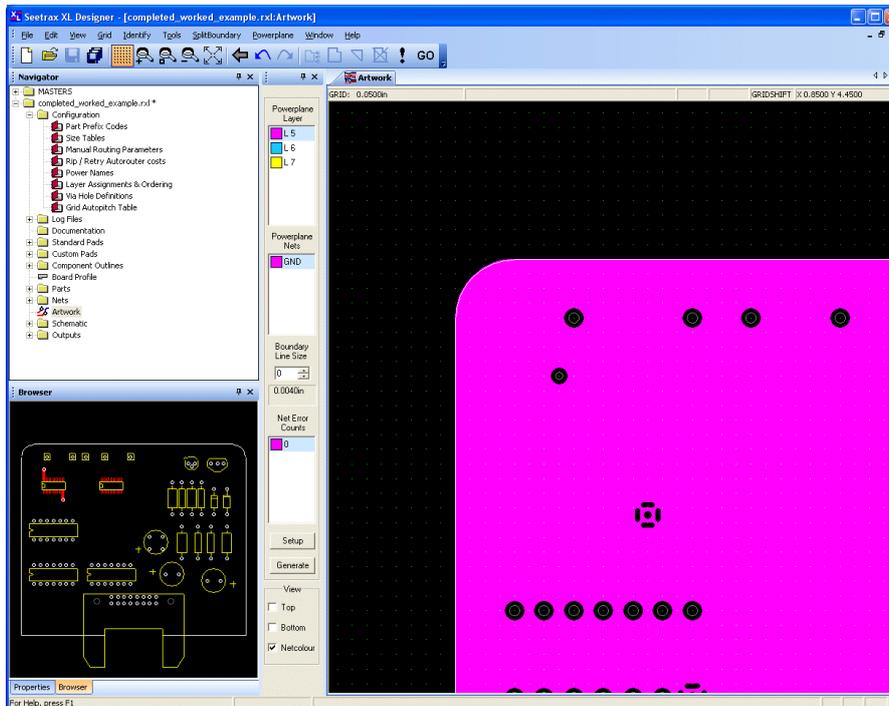


Figure 166

All the layers configured as power planes/split planes will appear in the top selection of the *Powerplane* dialogue bar, under the heading "*Powerplane Layer*". Clicking on each layer will change the display to show that selected layer. (The colour for the layer is as defined by the *View Control* dialogue bar and can be altered as required.)

In the mid-section of the dialogue, under the heading "*Powerplane Nets*", the power signal names to be used on the selected layer are shown.

If the layer is a 'split' plane, the primary rail for that layer is always shown at the top of the "*Powerplane Nets*" list, followed by the secondary rails - see Figure 167 where layer 7 has been selected as the *powerplane layer* and the *powerplane nets* are 0V (at the top, therefore the primary rail) and +12V, -12V and V+.

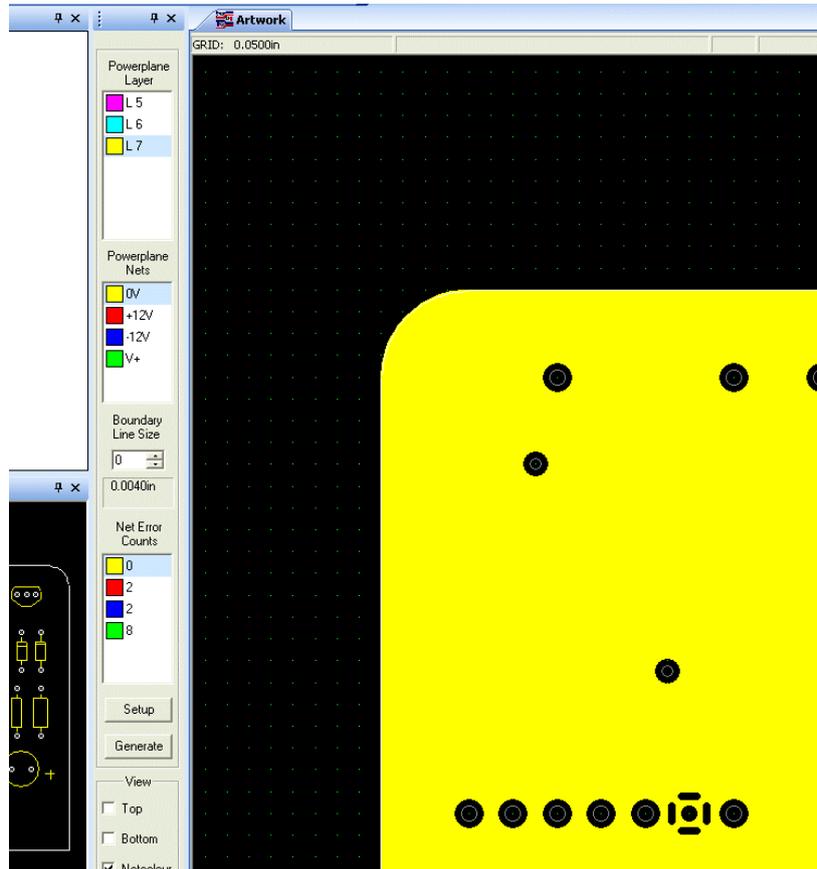


Figure 167

When a split plane layer is chosen, the display shows the primary plane with its heat-relief/antipads, and flashing coloured dots on some pads - Figure 168

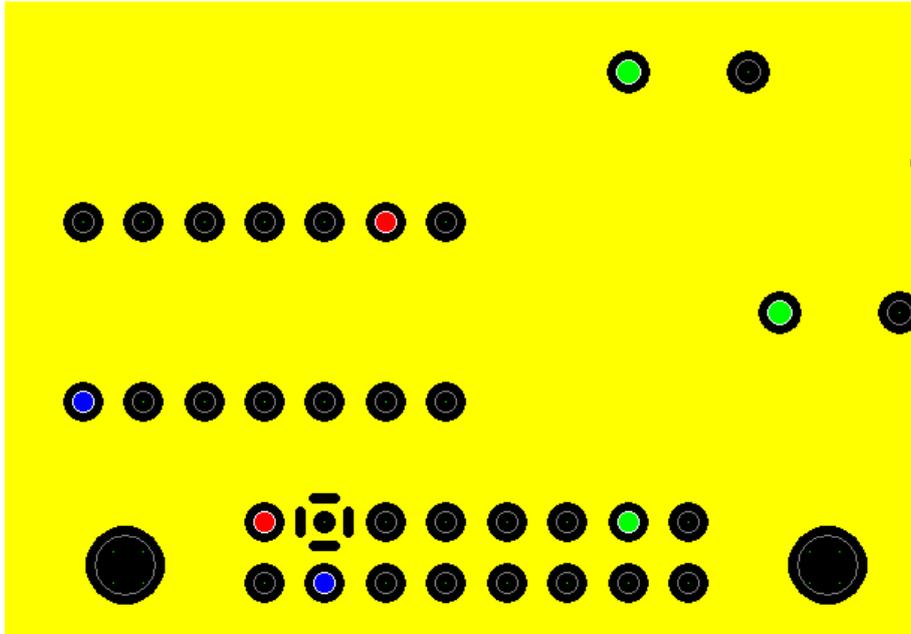


Figure 168

The coloured flashing dots represent the pins on the secondary power nets.

Coloured boxes alongside the powerplane net names in the dialogue bar show the colour assigned to each power net for recognition purposes. (The colour can be changed from the Setup button which is described later.)

Each of the secondary power net pins (flashing dots) have to be isolated from the other powerpin nets by a line that should enclose each of the same-coloured pins, to create an island of copper that forms the connection.

Drawing the split-plane boundaries

From the dialogue bar and with the split-plane layer selected, select one of the secondary powername nets.

The *SplitBoundary* commands become active and the polygon can be drawn (*SplitBoundary > Add Line*) to isolate the pins associated with that net. The line has to start/stop in the same place to form an enclosed polygon shape, at which time the area will change colour to reflect the power rail and the heat-relief pads for that rail will appear.

The line must not touch the edge of the board profile, cross itself or other lines.

Tip: add an enclosed polygon shape around a/some of the same-coloured dots, then use the *SplitBoundary > Corner* command to stretch it out - this is usually easier than trying to draw the polygon "right first time".

Pins that are within the wrong polygon region will flash on and off.

The flashing colour indicates which net the pin belongs to.

The "Net Error Count" area in the dialogue also indicates the number of pins on that net that are in error (ie outside of the polygon or in another polygon). There should be 0 errors for each power net.

Select each of the secondary power names and define their polygons.

As polygons are edited, the display will dynamically update changing pin powerplane features from antipad to heatrelief state as appropriate.

Modifying the split-plane boundaries

Once the polygon has been added, it can be modified with the *SplitBoundary* commands. Select the secondary power name from the dialogue (for example +12V) then the *SplitBoundary* command required.

Generating the planes

The planes are not created until the *Generate* button in the dialogue bar has been used.

Each layer is generated individually unless the "All Powerplanes" checkbox in the dialogue bar is ticked, when all the planes will be generated simultaneously when *Generate* is selected.

A message appears in the status bar to indicate that the planes were successfully generated.

Designs with split planes from earlier versions:

The power plane tool was altered in XLD version 1.72. Designs with split power planes saved prior to that version remain unaffected by the changes until it is required to regenerate the split planes or run the artwork checks.

When it is necessary to regenerate the planes, or run the artwork checks, for example following a modification, it is necessary to identify which split-plane polygon is associated with each power rail before the planes can be

updated and re-generated (the first time only). To do this:

- i. Open the artwork editor. Select the power plane tool, one power plane layer will be visible. If it's a split powerplane layer, the embedded power rail pins will be flashing.
- ii. From the Powerplane dialogue bar, under the "Powerplane Nets" heading, select one of the secondary power rail names.
- iii. The first powername in this list is always the primary power rail (so don't select that one). The coloured boxes indicate the colour assigned to the power rails for identification purposes.
- iv. Once the power rail name has been selected, this will enable the *SplitBoundary > Import Boundary* command, which should now be selected (an icon is available for this command).
- v. The split plane polygons will now appear, and the one associated with the selected powername should be selected.
- vi. If the correct polygon is chosen, the same-coloured dots will stop flashing and be replaced with heat-relief pads in that area.
- vii. If the dots continue to flash, the incorrect polygon was chosen - use *Undo*, then repeat until the correct polygon is selected.
- viii. Once all the polygons for that layer have been imported, use the *Generate* button (or *Powerplane > Generate* command) to create the plane.
- ix. Repeat for the other split-plane layers.

Once completed, it is not necessary to import the polygons again.

Note: all plane layers must be generated – the *Generate* button creates the selected plane only, unless the *All Powerplanes* checkbox is ticked.

Adding text, logos, extra clearance areas, etc. to the power planes

Any tracks, pads or text that are added to a power plane layer (using the *Amend* or *Text* commands) are "not" copper, they are clearances in the plane, they are therefore not included in the checking routines. It is possible to isolate heat-relief pads with this additional data. These instances will not be flagged by the artwork checking routines.

Please note that logos placed in powerplane layers are currently invisible to the artwork checker. No checks for violation with other powerplane features will be performed.

When a power plane is generated/regenerated, the logos will be preserved within the layer.

Notes:

Tracks, pads and text that are added to the power plane layers will be deleted if the power planes are ever generated again. For this reason, the data could be added to one currently unused silk-screen layer of the board to save re-adding it if the planes are ever re-generated. That layer would need to be output with the power plane layer. (A silk screen layer should be chosen, as an inner copper layer would contain the inner layer pads of the components, these could lead to isolated pins in the resultant output file.) The data on the silk screen layer would not be checked by the artwork checking routines.

When the planes are created a clearance line is added around the outside of the board to stop the plane reaching the edges of the board and possibly causing shorts once manufactured and in use. (The line is centred on the board profile and the power plane dialogue bar determines its thickness (boundary line size). Once added it can be altered using the *Amend* commands.)

Due to the position of holes and or isolation lines, it is possible for heat-relief pads to become isolated from the power plane, or only connected by one or two channels. It is also possible for a heat-relief pad to be connected to more than one power net if the isolation line crosses through a heat-relief pad. Neither of these situations is identified by the checking routines.

The powerplane isolation lines can be altered using the *Amend* commands but these changes are not back-annotated to the powerplane tool - refer to the *SplitBoundary > Import Boundary* command to do this.

Power plane commands

When the *Tools > Powerplane* command is selected, a dialogue bar appears, similar to that shown in Figure 169, which controls the plane generation, along with a new set of commands across the top of the window.

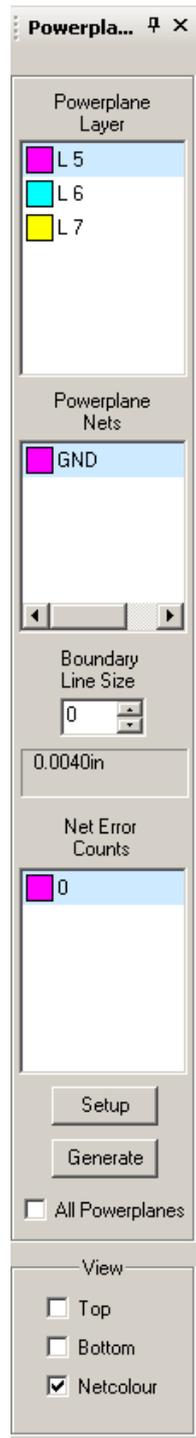


Figure 169

Power Plane Dialogue Bar

This should be set up as required, using the following information for guidance. The layers & nets that appear are controlled from the Configuration folder > Layer Assignments & Ordering.

Powerplane Layer

All the layers configured as power planes/split planes will appear in the top selection of the dialogue bar, under this heading.

Clicking on each layer will change the display to show that selected layer. (The colour for the layer is as defined by the *View Control* dialogue bar and can be altered as required.)

Powerplane Nets

The power signal names used on the selected layer are listed here.

If the layer is a 'split' plane, the primary rail for that layer is always shown at the top of this list, followed by the secondary rails - see Figure 167 where layer 7 has been selected as the *powerplane layer* and the *powerplane nets* are 0V (at the top, therefore the primary rail) and +12V, -12V and V+.

When the *SplitBoundary* commands are used to add or modify the isolation polygon, they will only operate on the isolation line of the selected power rail in this list.

Boundary Line Size

This setting corresponds to the size codes from the Track Sizes table and it controls the width of the isolation lines that are added between power rails in a split plane and the line that is added around the outside of the profile.

The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside. The actual size is shown below the setting.

New lines are added at the size selected.

Existing lines can be altered by selecting the powerplane net from the dialogue, then changing the size control.

Net Error Counts

The numbers here correspond to the number of component pins that are outside of the polygon. The coloured box identifies which power rail they belong to.

Setup button

The powerplane setup window Figure 170, contains information about each layer and power net(s) used.

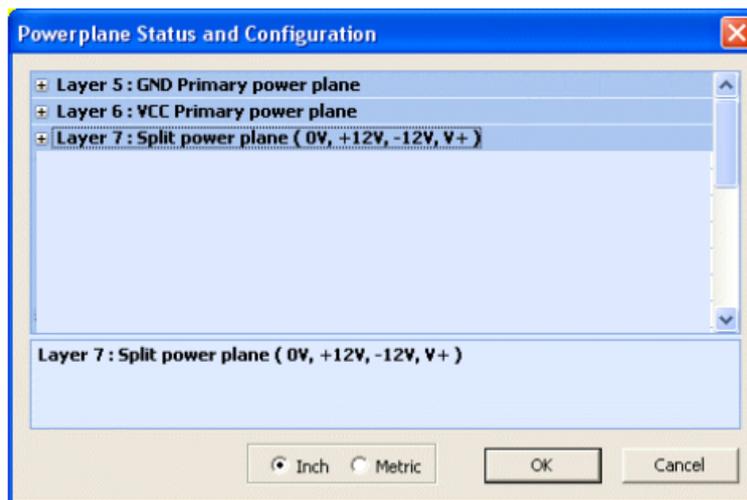


Figure 170

Use the "+" button to expand the information on a particular layer or power rail. Figure 171. shows Layer 7 expanded, along with the power rails on that layer.

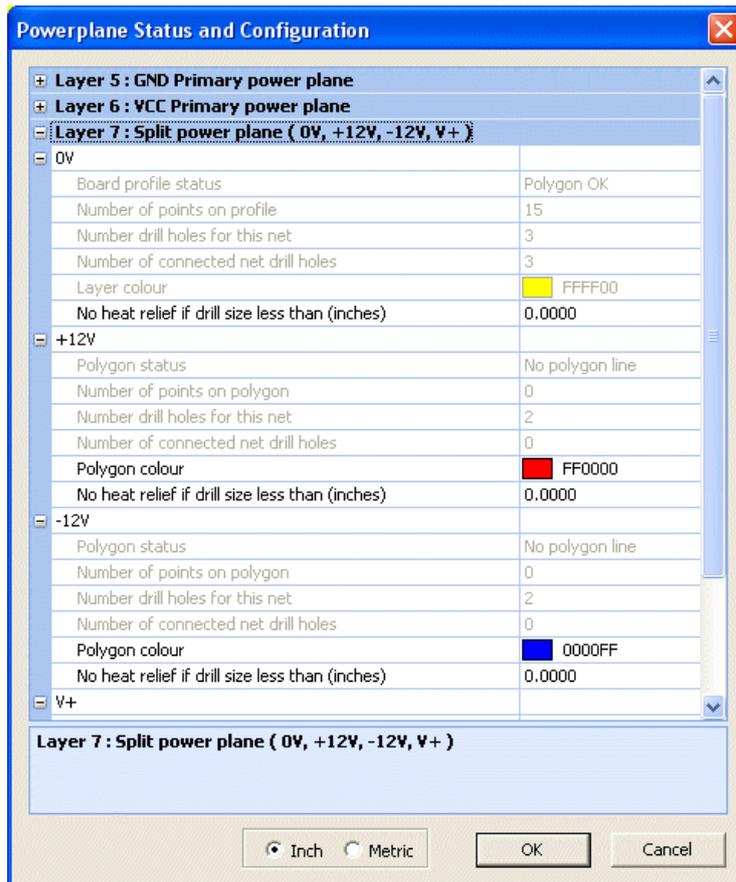


Figure 171

Items in the "Setup" window that are not 'greyed' can be edited.

Board Profile/Polygon Status: the board profile and each secondary polygon line must form an enclosed area. This indicates whether the line is OK, not defined or not closed.

Note: to allow for non-coincident points in the board profile (typically occurs when the profile has been imported), a tolerance of 0.005" (0.127mm) is permitted in board profile line end-points.

No. of points on profile/polygon: the number of points (typically corners, though a point can exist without a change of direction) in the polygon/profile.

No. drill holes for this net: the number of drilled holes connected to the power rail.

No. of connected net drill holes: the number of through plated component pins in this power rail that have been connected.

Layer colour: the colour assigned to the layer by the *View Control* dialogue bar. The primary power rail will use this colour.

Polygon colour: the colour assigned to the secondary power rail.

Selecting the coloured box allows the colour to be changed. Once selected, select the arrow that appears alongside. A colour palette appears from which the colour can be selected.

No heat-relief if drill size less than: any drilled pads for this net, that are smaller than the size specified will not have a heat-relief pad added around the hole.

The default is zero, so heat reliefs are always created. The size of heat relief pads (and anti-pads) is defined in the Sizes Table.

A separate threshold may be specified for each net section of a split powerplane.

Heat-relief pads are generally added to holes that are connected to the plane to avoid heat-dissipation when soldering to the hole. Excluding the heat-relief pads on smaller holes can free up channels that may otherwise cause pads that should be connected to the plane from becoming isolated.

Once the settings in this window have been defined, select *OK* to continue.

Generate button

Used to create the power planes.

If the *All Powerplanes* check box is ticked, then all the powerplane layers are generated.

If the check box is unticked, only the selected powerplane layer is generated.

Everything previously added to the power plane layers, except for logos will be removed when *Generate* is selected. This ensures the planes are correct – a warning window will indicate that this is about to happen and the command can be cancelled if required.

Heat-relief pads will be added around holes that are connected to the plane (depending on the threshold set in the setup window) and anti-pads around all other pads.

An isolation line is also added around the board profile, its size is determined by the *Boundary Line Size* setting. This stops the plane shorting into metal objects that might touch the edge of the manufactured board.

View Buttons

Top/Bottom: when ticked, the tracking/text/vias/component pads on the selected layer are displayed. (Only the component pads/vias are displayed if the layer is set to invisible in the *View Control* dialogue bar.)

Net colour: when unticked, the split power plane layer is shown in the primary power colour only, so the secondary power rails cannot be identified by their colour.

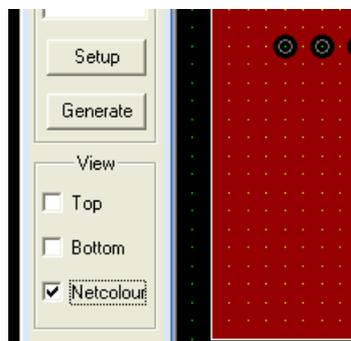


Figure 172

SplitBoundary commands

These commands are used to add or modify the isolation barrier between power rails on the same split power plane layer. One isolation line for each secondary power rail, should form a polygon shape to enclose all the pads of that power rail. One isolation line/area is required for each secondary power rail. Multiple areas are not permitted.

- The isolation line for each power rail must be defined from one continuous line that doesn't overlap or cross itself or other isolation lines or the board profile.
- The line must form an enclosed polygon shape around all the pins connected to the same power rail (those of the same colour). Pins of other colours must not be enclosed within that polygon.
- The checking routines do not identify isolated power pins caused by isolation lines or heat-relief and anti-pads.

Do not:

- lay the line across or touch any coloured power dots. The coloured dots must be completely inside or completely outside the isolation area – a heat-relief pad that has an isolation line through it will be shorted between two power nets.
- cross isolation lines, either of the same colour or different colours, otherwise areas will become isolated.
- allow the isolation lines to touch the edge of the board profile or one another – this may lead to isolated areas and pins

The additions/changes to the polygons are not updated on the powerplane layer until the planes are generated.

SplitBoundary > Add Line

Before adding the isolation line, ensure the appropriate layer & secondary power rail is selected from the dialogue bar. Position the cursor where you intend starting the isolation outline and click the left-hand mouse button. Move the cursor to stretch the line and insert corners in the line by clicking the left-hand mouse button. Enclose all the pins of the selected secondary power rail, excluding the other coloured pins. Complete the isolation line by inserting a corner on top of the starting point of the line to enclose the area, then click the right hand mouse button to release the line. Once the polygon forms an enclosed shape, the area inside the isolation line changes colour (unless the Powerplane dialogue bar *View – NetColour* check box is unticked).

If the polygon is not completely enclosed, or pins are incorrectly included/excluded, use the *SplitBoundary > Corner* command to adjust the line.

Although multiple lines can be added, they should all be joined together and form an enclosed shape – whilst multiple disconnected lines exist, the area will not be valid or change colour appropriately.

SplitBoundary > Delete Line

Used to delete an isolation line that has been added. A complete isolation line is deleted, not just a segment in the line.

Select *SplitBoundary > Delete Line*, select the appropriate layer & secondary power rail from the dialogue bar. Point at the isolation line to be deleted and click the left-hand mouse button. The line is deleted.

(If the planes have already been generated, this will not remove the isolation line from the plane itself until the planes are re-generated.)

SplitBoundary > Corner

Used to add and move corners in isolation lines.

(If the planes have already been generated, this will not update the plane layer until the planes are re-generated.)

Corners are *added* and then released with a click of the *right* hand mouse button. Corners are *moved* and then released with a click of the *left* hand mouse button. Clicking the opposite mouse button once a track has been selected cancels the operation.

right button = *add* corner

left button = *move* corner

(Opposite button cancels.)

Only lines from the selected power rail can be modified.

Select *SplitBoundary > Corner*, then select the appropriate layer & secondary power rail from the dialogue bar.

Adding a corner to an isolation line

Point at an isolation line, and click the right hand mouse button. Move the cursor and the new corner. Locate the cursor in the position for the new corner and click the right hand mouse button again to release the corner.

Clicking the left hand mouse button before releasing the line, releases the line without the new corner.

Moving an existing corner in an isolation line

Point at an existing corner in the isolation line, and click the left hand mouse button. Move the cursor and the corner stretches with it. Locate the cursor in the new position for the corner and click the left hand mouse button again to release the corner.

Clicking the right hand button before releasing the line, returns the corner to its original position.

SplitBoundary > Delete Point

Used to remove points (or corners) from isolation lines.

Delete Point will not remove the end points of lines.

Select *SplitBoundary > Delete Point*, then select the appropriate layer & secondary power rail from the dialogue bar. Point at a point (corner) in the isolation line and click the left hand mouse button. The point is removed.

(If the planes have already been generated, this will not update the plane layer until the planes are re-generated.)

SplitBoundary > Import Line

There are two reasons why this command is used:

- i) the power plane tool was altered in version 1.72 and when it is necessary to regenerate the split planes or run the artwork checks in designs saved in v1.71 or earlier, it is necessary to identify which split-plane polygon is associated with each secondary power rail, before the planes can be re-generated (the first time only) or the artwork checks run.
- ii) If the isolation lines on the layer have been altered using the Amend commands, these changes are not back-annotated to the powerplane tool.

In each case, it is necessary to re-link the isolation lines to the appropriate power rail as follows:

Select the appropriate layer & one of the secondary power rails from the dialogue bar. Select *SplitBoundary > Import Line* (this command remains greyed out until the secondary rail is selected). The display changes and the boundary polygons appear as white lines.

Select the polygon associated (or is believed to be associated) with the selected secondary power rail. If the correct polygon is chosen, the same-coloured dots will stop flashing and be replaced with heat-relief pads in that area. Move onto the next power rail. However, if the dots continue to flash, the incorrect polygon was chosen -

use *Undo*, then repeat until the correct polygon is selected.

Once all the polygons for the layer have been imported, use the *Generate* button (or *Powerplane* > *Generate* command) to create the plane.

Repeat for the other split-plane layers.

Once completed, it is not necessary to import the polygons again.

Powerplane commands (Setup & Generate)

These commands (*Setup* and *Generate*) replicate the commands with the same name in the dialogue bar on the left side of the screen. They are described above under the heading *Power Plane Dialogue Bar*.

Slots & Extra Holes (Tools > Slots & Extra Holes commands)

These commands allow the insertion of slots (for CNC routing). They also allow arbitrary extra plated or non-plated drill locations to be inserted.

(The same commands are available within the custom pad and outline editors.)

Figure 173 shows slots and extra plated/non-plated holes added to the artwork editor.

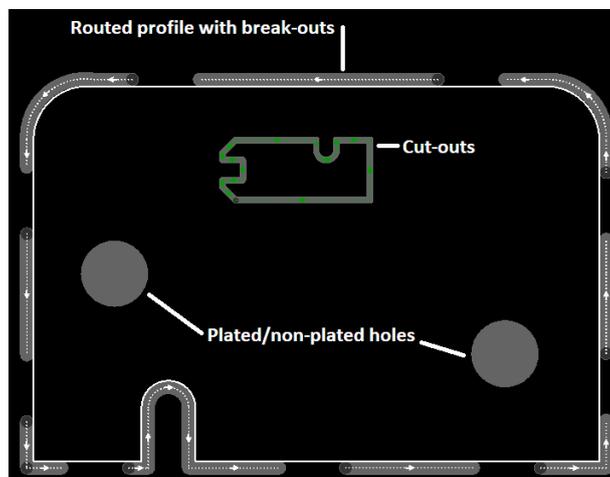


Figure 173

Long machined slots are often used to allow breaking out of a board from a larger panel or to produce cut-outs within the board..

Break-out tabs may be inserted in the slots to assist “breaking out of a board” from a larger panel. The tabs may be solid or perforated (a series of close spaced small holes). Figure 174

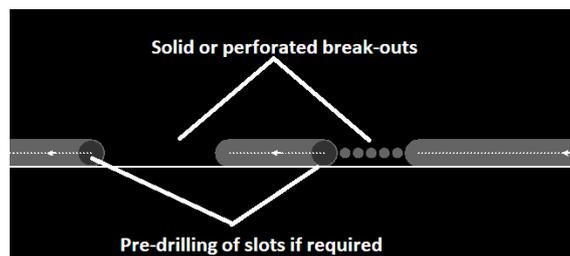


Figure 174

The slots can be pre-drilled (to avoid damaging the router cutter) if required.

A configuration panel **CNC Tools & Datasets** (found within the Configuration folder) allows pre-definition of all drills and routing tools needed for the board manufacture.

When the board is manufactured, it is likely that the drilling and/or routing will be done in stages:

- Drill holes/rout slots that need plating. Board is then plated.
- Drill holes/rout slots that are unplated.
- If slots need to be routed separately, an extra stage(s) would be required.
- Multilayer designs with blind/buried vias require extra stages, depending on the number of layers in use.

Each of these stages needs the drilling/routing data in a separate file. To achieve this, the drilling/routing data for each stage is added to its own “dataset” which can be output on its own.

Two datasets are pre-defined:

DS1 - plated holes (through the board)

DS2 - non-plated holes (through the board)

Additional user-defined datasets may be defined, from DS3 onwards.

Known limitations with this feature (in v2.25)

- The artwork checker is currently “blind” to the presence of slotted features and extra holes added to the design, so this must be borne in mind when slots or extra holes are added – a warning is given when checking is selected.
- Copper fill tool does not “see” slots or extra holes – temporarily add keepout areas around them.
- Powerplane tool does not “see” slots or extra holes – add clearance/anti-pads manually using Amend.
- Once a breakout is added to a slot, it is not yet possible to edit its parameters without deleting and recreating it.
- Once an extra hole is added, it is not yet possible to alter its size or drill dataset without deleting and recreating it.
- The 'Identify Feature' tool will not provide any information about slots and extra drill holes.
- Undo/Redo does not work within the Slots & Extra Holes tool.
- If a slot is defined in a component outline, then the tool cutting direction will be reversed if the part is flipped to the other side of the board.

Opening older jobs (prior to v2.23)

All router lines in v2.23 onwards must have a thickness assigned.

Any router lines defined in versions prior to v2.22 would not have had a thickness assigned to them.

For this reason, all router lines already in existence will be assigned a thickness of 0.1”/2.54mm when the job is opened. This will impact upon the edge of the profile or any cutouts defined, so remedial action will be required.

Either reduce the size of the cutter, move the lines appropriately, delete the lines and re-define them or a combination of all the above – the option chosen will depend on the circumstances of each job.

Slots & Extra Holes Dialogue bar

When the *Tools > Slots & Extra Holes* command is selected, a dialogue bar appears, similar to that shown in Figure 175, which contains controls and settings for slots and extra holes, along with a new set of commands across the top of the window.

If no slots/holes exist, then the bar indicates that no slots are defined, arrowed in Figure 175 and most settings will be greyed out.

This manual describes each control/setting and command; a step-by-step procedure for adding/modifying slots and extra holes is given in a separate guide available from: http://www.seetrax.com/xld_docs.htm

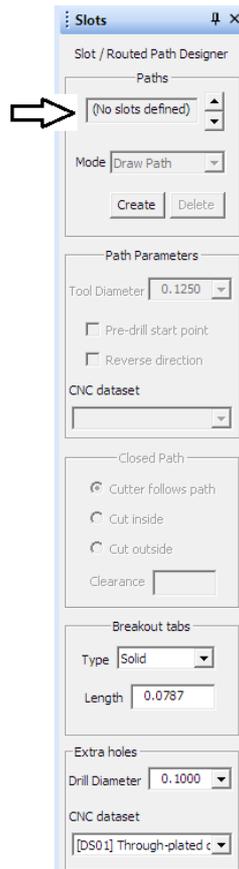


Figure 175

Paths

This indicates if any paths (or slots) have been *created* - a path cannot be defined/drawn until it has been *created*.

Figure 176 shows a portion of the dialogue bar prior to a path/slot being created, then with one created. Once a path has been created, the appropriate greyed out settings in the bar become editable.

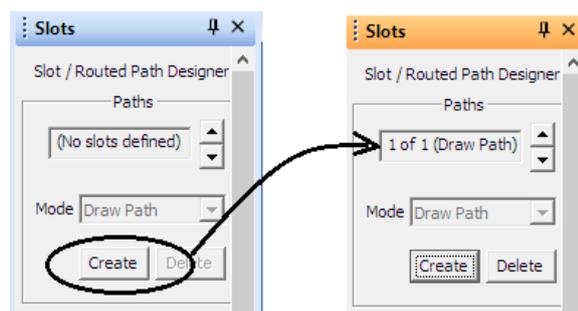


Figure 176

The slots/paths are numbered for reference purposes.

Use the **Create** button to add a new slot

Use the up/down arrows alongside the slot/path to make any of the paths, the “*active*” slot.

Once drawn, the *active* path/slot is shown with a dotted white centre line. The other inactive paths/slots will be shown with a dotted green line.

The *Active* slot/path has its settings displayed in the dialogue bar. Set these according to the path/slot being added. They can be modified later and the active path/slot will be updated correspondingly.

The path/slot is actually drawn/modified using the **Slot Definition** commands (in the command menu).

Use the **Delete** button to delete a path/slot that is no longer required. Ensure the path to be removed is the

active path as described above before pressing the Delete button. The remaining path numbers are updated accordingly.

There is no need to remove the drawn path before using this command as the drawn path will be removed automatically.

Path Parameters

The *Path Parameters* show the settings associated with the active path/slot as shown in Figure 177.

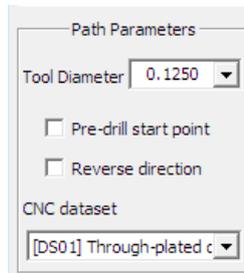


Figure 177

All of these settings can be changed whilst the slot is being drawn, or after the slot has been added. Changes affect the currently selected or "active" slot - the dotted line/arrows in the active slot will be white.

- Tool diameter:** the width of the active slot (Edit > Units to change between inches/mm)
Type in a value (keep the cursor within the area) or choose from the drop-down list, which is populated from the *CNC Tools & Datasets* table (Configuration folder).
- Pre-drill Start Point:** Pre-drilling can help reduce tool breakage - a hole is drilled before the router tool plunges into the board to cut the slotted path.
The pre-drill size is specified in the *CNC Tools & Datasets* table (Configuration folder).
To see the pre-drills in a slot, the size of the pre-drill in the *CNC Tools & Datasets* table must be defined.
If a pre-drill size is not specified in the *CNC Tools & Datasets* table, then it will not be pre-drilled even if this box is ticked.
The pre-drill will appear at the start of each slot, and after each break-out within the slot if added.
- Reverse direction:** The slot is routed in the direction in which the line is drawn unless the box is checked. The direction is indicated by the dotted line/arrows in the slot. (It's easier to draw the path first, then change its direction if required.)
The direction can be changed as it is important to obtain the best edge finish on the correct side of the slot.
- CNC Dataset:** Each slot can be added to one dataset and this controls which drill/router output file it is included in.
When the slots are output, they are output with other drilled holes and/or slots in the same dataset. Which dataset is chosen depends on the stage in the manufacturing process that they will be used.
There are two pre-defined datasets, one for plated (DS1) and one for non-plated holes/slots (DS2).
Other user-defined datasets are available and they are defined in the *CNC Tools & Dataset* table (Configuration folder).
Speak to the manufacturer if you do not know which slots to include in which dataset.

Closed Path

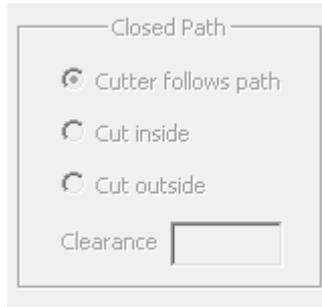


Figure 178

This feature is not currently coded so remains greyed out.

Break-out tabs

Break-out tabs stop the board falling out of a panel once routed. Either solid or perforated tabs can be inserted - Figure 179.

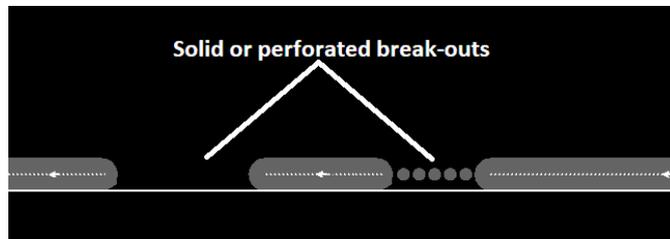


Figure 179

A perforated tab is made up from a small row of holes that form the weak point where the board will be broken out.

Figure 180 shows the settings available for solid and perforated break-out tabs.

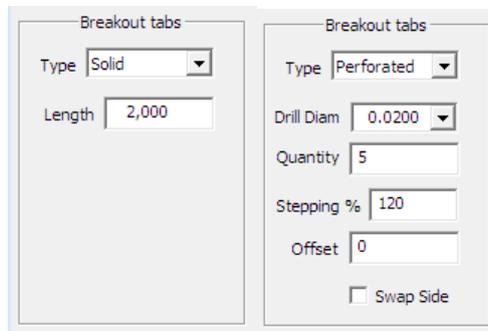


Figure 180

Note: currently (v2.25) these settings cannot be altered once the tab has been added, so if changes are required, the tab will have to be deleted and a new one added.

- Type: The breakout can be solid or perforated (as shown in Figure 179).
- Length: If a solid break-out tab is chosen, then a length for the tab has to be specified.
- Drill Diam: If a perforated break-out tab is chosen, then a size for the drill that will make the perforations has to be specified.
Type in a value (keep the cursor within the area) or choose from the drop-down list, which is populated from the *CNC Tools & Datasets* table (Configuration folder).
- Quantity: The number of drilled holes within the perforated break-out tab.
- Stepping %: Is the distance between the perforation holes as a percentage of drill diameter, see Figure 181.

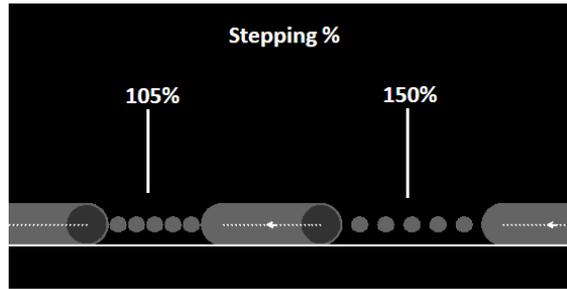


Figure 181

150% would mean that the holes are spaced apart by one and a half times the drill's diameter. Note: 105% is the minimum value permitted to avoid tool breakage.

Offset: The holes can be offset away from the centre line of the slot, see Figure 182.

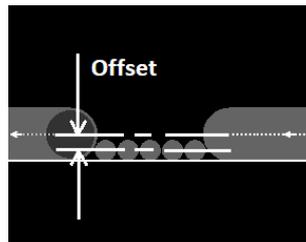


Figure 182

0 the perforations are along the centre line of the slot

100 the perforations have their centre line aligned with the edge of the slot (i.e they are displaced by half the slot width from its centre line).

Think about positioning the perforations so they do not violate the edge of the board, whilst leaving the board with minimum cleanup required after it has been broken out.

Swap Side: Makes the holes appear on the opposite side of the path, Figure 183.

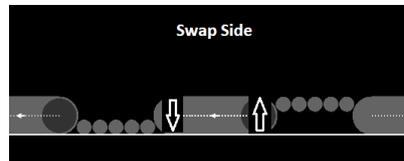


Figure 183

Extra Holes

These holes are typically added for tooling purposes, they can be plated or unplated but will not have copper surrounding them.

In v2.25 it is not possible to edit *extra holes*, so to change their diameter or dataset, they must be deleted and reinserted – this will be rectified in a future release.

Depending on the screen size, it may be necessary to scroll down to the bottom of the dialogue bar to see the *Extra Hole* settings, as shown in Figure 184.

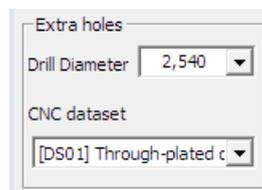


Figure 184

Set the parameters as required:

Drill Diameter: Size of hole required. Type in a value (keep the cursor within the area) or choose from the drop-down list, which is populated from the *CNC Tools & Datasets* table (Configuration folder).

CNC Dataset Which CNC Dataset the hole should be included in.

Slot Definition > Add Line

Used to add a line to define the active path/slot which is selected from the dialogue bar.

Only one continuous line can be added per path. Multiple paths can be added.

Ensure a path is selected in the dialogue bar (if 0 paths are shown, then use the *Create* button in the bar first).

If a path is selected/active and the *Add Line* command remains greyed out, then there is already a path drawn for that path.

Alter the settings in the Slots dialogue bar as required for the new path/slot.

Select Slot Definition > Add Line, point at the location where the path will start and click the left button. Move the cursor, a line appears attached to it, representing the path of the cutter, from start to finish - ignore the position of break-outs as they are added later.

Click the left button to insert corners in the line.

Once the first segment has been drawn, the thickness of the slot, the pre-drill hole (if selected) and a dotted line with an arrow in the centre of the slot, which shows the cutting tool direction, will appear.

These settings can be changed in the dialogue bar whilst the line is attached to the cursor if required.

Draw the shape required; if necessary make the two ends coincident to form a closed shape.

When complete, release the line from the cursor with a click of the right-hand mouse button.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The shape of the line can be modified using the Slot Definition > Corner/Adjust Arc/Delete point, etc. commands.

Changes to the dialogue bar are reflected in the active path immediately.

Slot Definition > Add Arc

Used to add an arc(s) or a combination of arcs/lines to define the active path/slot which is selected from the dialogue bar.

Only one continuous arc/line can be added per path. Multiple paths can be added.

Ensure a path is selected in the dialogue bar (if 0 paths are shown, then use the *Create* button in the bar first).

If a path is selected/active and the *Add Arc* command remains greyed out, then there is already a path drawn for that path.

Alter the settings in the Slots dialogue bar as required for the new path/slot,

Select Slot Definition > Add Arc, point at the location where the arc/line is to start and click the left button. Move the cursor and its attached line to the position where the curve should end, and click the left button again. A straight segment is produced that bends as the cursor is moved. The shape of the arc is dependent on the position of the cursor. This represents the path of the cutter, from start to finish

Once the first segment has been drawn, the thickness of the slot, the pre-drill hole (if selected) and a dotted line with an arrow in the centre of the slot, which shows the cutting tool direction, will appear.

These settings can be changed in the dialogue bar whilst the line/arc is attached to the cursor if required.

Draw the shape required; if necessary make the two ends coincident to form a closed shape.

Click the left button to release the curve in its current position. The line remains attached to the cursor, allowing a

series of curves to be added.

Once the arc has been released, clicking the right-hand mouse button releases the line from the end of the cursor.

If the right-hand mouse button is clicked whilst the curve is being stretched, a straight-line segment is introduced,

thus allowing a mixture of curved and straight lines to be added.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Arcs can be modified with the Slot Definition > Corner/Adjust Arc commands.

Slot Definition > Add Circle

Used to add a circle to define the active path/slot which is selected from the dialogue bar.

Only one circle (or arc/line) can be added per path. Multiple paths can be added.

Ensure a path is selected in the dialogue bar (if 0 paths are shown, then use the *Create* button in the bar first).

If a path is selected/active and the *Add Circle* command remains greyed out, then there is already a path drawn for that path.

Alter the settings in the Slots dialogue bar as required for the new path/slot,

Select Slot Definition > Add Circle, point at the centre of the required circle and click the left-hand mouse button. As the cursor is moved, a circle appears attached to it. The size of the circle is controlled by the cursor position. Click the left hand mouse button to release the circle in its current position. Clicking the right-hand mouse button whilst adding the circle cancels the circle.

The thickness of the slot, the pre-drill hole (if selected) and a dotted line with an arrow in the centre of the slot, which shows the cutting tool direction, will appear.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The circle can be subsequently moved or modified with the *Outline > Move Point* command.

Slot Definition > Corner

Used to add or move existing corners in the active line/arc.

Lines and arcs can only be selected if they belong to the *active* path as selected in the Slots dialogue bar.

The active path is displayed with a white, dotted centre line along its length.

Corners are **added** and released with clicks of the **right-hand** mouse button.

Corners are **moved** and released with clicks of the **left-hand** mouse button. Once the corner has been selected, clicking the opposite mouse button cancels the operation.

right button = **add** corner

left button = **move** corner

Opposite button cancels

Adding a corner to a line or arc:

Once selected, point at the line or arc and click the right-hand mouse button. Move the new corner and release it with another click of the same (right) button. Clicking the opposite (left) hand button before the corner is released, cancels the new corner.

Moving an existing corner in a line or arc:

Once selected, point at the corner in the line or arc and click the left-hand mouse button. Move the corner and release it with another click of the same (left) button. Clicking the opposite (right) hand button before the corner is released, cancels the move.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Slot Definition > Move Point

Used to move the active circle and/or to change its size.

Circles can only be selected if they belong to the *active* path as selected in the Slots dialogue bar.

The active path is displayed with a white, dotted centre line along its length/circumference.

(the command can also be used to move an existing point (corner) in a line or arc, but the *Profile > Corner* command is more flexible.)

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Moving a circle:

Once selected, move the cursor over the centre of the circle and click the left-hand mouse key. Move the cursor with the circle attached, into the required position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the circle is attached to the cursor returns it to its original position.

Changing the size of a circle:

Once selected, point at the circumference of the circle and click the left-hand mouse key. As the cursor is moved, the circle's size changes. Click the left-hand mouse button again to release the circle at its current size.

Clicking the right-hand mouse button whilst circle is attached to the cursor returns it to its original size.

Moving points on lines or arcs:

Once selected, move the cursor over the point (corner) you want to move and click the left-hand mouse key.

Move the cursor with the corner attached into the required position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the corner is attached to the cursor returns it to its original position.

Slot Definition > Adjust Arc/Circle

Used to convert straight lines into curved lines, curved lines into straight lines, and to change the shape of existing curved lines; also to move a circle and/or to change its size.

Lines/arcs/circles can only be selected if they belong to the *active* path as selected in the Slots dialogue bar.

The active path is displayed with a white, dotted centre line along its length/circumference.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Straight lines into arcs:

Once selected, point at an active line segment and click the left-hand mouse button. Move the cursor, the segment is replaced with an arc on the end of the cursor. Once the arc is in the required position, click the left-hand mouse button to release it.

Arcs into straight line segments:

Once selected, point at an active arc and select it with a click of the left-hand mouse button. Follow this with a click on the right-hand mouse button to convert it to a straight line segment.

Changing the shape of an arc:

Once selected, point at an active arc and select it with a click the left-hand mouse button. Move the cursor with the arc attached, then click the left-hand mouse button when the arc is in the desired position.

Clicking the right-hand mouse button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically and can be used to convert 45 degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Moving a circle:

Once selected, move the cursor over the centre of the circle and click the left-hand mouse key. Move the cursor with the circle attached, into the required position and click the left-hand mouse button again to release it.

Clicking the right-hand mouse button whilst the circle is attached to the cursor returns it to its original position.

Changing the size of a circle:

Once selected, point at the circumference of the circle and click the left-hand mouse key. As the cursor is moved, the circle's size changes. Click the left-hand mouse button again to release the circle at its current size.

Clicking the right-hand mouse button whilst circle is attached to the cursor returns it to its original size.

Slot Definition > Delete Feature

Used to delete path of the active line/arc/circle. This does not remove the path from the dialogue bar, so a new line/arc/circle could be added to that path after the feature has been deleted.

To remove the path and its entry in the dialogue bar, use the *Delete* button from the dialogue bar.

The complete path is deleted, between its start and end points, not just a segment between points.

Ensure the correct path is selected/active in the dialogue bar. If a path is selected/active and the *Delete Feature* command remains greyed out, then a path has not be drawn for that path.

Select Slot Definition > Delete Feature, point at the line/arc/circle and click the left button, it will be deleted.

Select the item on a corner, or the centre of the circle if there is difficulty selecting the item, also zoom in if necessary.

Slot Definition > Delete Point

Used to remove corners (points) from the path of the active line or arc.

Once selected, move the cursor over a point (corner) in a line or arc and click the left-hand mouse button. The point is removed.

A start or end point of a line cannot be deleted.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

Breakout commands

Break-out tabs may be inserted in the path/slot to assist "breaking out of a board" from a larger panel. The tabs may be solid or perforated (a series of close spaced small holes) and this is controlled from the *Slots* dialogue bar as described previously.

Slot Definition > Add Breakout

Select Slot Definition > Add Breakout position the cursor over the centre of the intended breakout position and click the left button. Repeat as required.

Slot Definition > Move Breakout

Used to move a break-out tab.

Break-out tabs, are selected about their centre, along the axis of the slot - see Figure 185, not on the breakout itself if it has been offset..

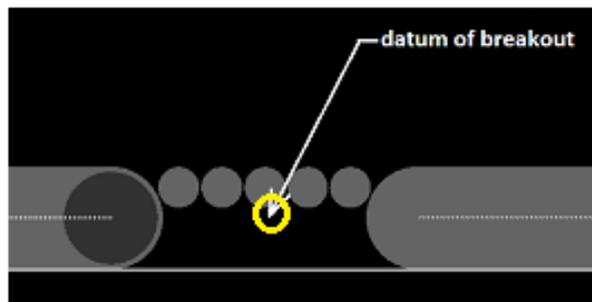


Figure 185

Slot Definition > Delete Breakout

Used to delete a break-out tab.

Break-out tabs, are selected about their centre, along the axis of the slot - see Figure 185, not on the breakout itself if it has been offset.

Extra Holes commands

Standard pads of any shape or custom pads with a hole, have a copper area with a hole inside the area. When drilled, they are always output with Dataset 1 (plated).

Extra Holes added using the *Slots* command menu are simply a hole – they do not have a copper area of any shape around them, though the hole itself can be plated through. They can be added to any drill dataset.

The *Extra Holes* section of the *Slots/Routed Path Designer* dialogue bar (Figure 184) controls the settings of the hole being added.

Extra Holes > Add

Select Extra Holes > Add Hole, the hole appears attached to the cursor, position as required with a left click.

Refer to the Grid command for details on whether grid snapping is active. The current grid is shown in the information bar.

The size of the hole and which drill dataset it is included in, is controlled from the *Extra Holes* section of the *Slots/Routed Path Designer* dialogue bar Figure 184.

(In version 2.25 of the software, once the hole has been added it cannot be changed, so delete the hole and re-add if a change of size/drilldataset is required.)

Extra Holes > Move

Used to move a hole that's already been added using the Extra Holes > Add command (not pads).

When the hole is selected, the dialogue bar is updated to reflect the size/dataset of the selected hole (which cannot be changed whilst the hole is being moved in v2.25).

Select Extra Holes > Move Hole, point at the centre of the hole and click the left hand mouse button. The hole moves with the cursor. Move the hole into position and click the left hand mouse button again to release it.

Clicking the right hand mouse button whilst the hole is attached to the cursor returns it to its original position.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Extra Holes > Delete

Used to delete a hole that's already been added using the Extra Holes > Add Hole command (not pads).

Select Extra Holes > Delete Hole, point at the centre of the hole and click the left hand mouse button. The hole is removed.

Network commands

The following Information applies to all the *Network* commands:

These commands are used to add parts or connections to the artwork, or change connections on the artwork

without editing the wiring list.

The changes are automatically back annotated to the parts and wiring lists.

The artwork checks take these changes into account when checking the artwork.

The circuit schematic is NOT updated.

A log report is generated to allow the circuit schematic to be manually back annotated. The report is saved in the *Log Files* folder. The report is called *netmods*.

When the circuit schematic is viewed, providing the netmods log report has not been deleted, a warning message is given to indicate that it may not match the parts/wiring list, and to refer to the netmods file for details.

It is **IMPORTANT** that the changes are added to the circuit schematic before the parts/wiring list is extracted from the circuit again. Failing to do this, means the changes made via the *Network* commands will be lost from the new parts/wiring list.

If that happens, the changes made via the *Network* commands will be lost and any tracks that remain on the board will be treated as shorts (unless they don't touch anything) by the artwork checking routines.

Once the changes have been implemented on the circuit, the log report should be saved if required and the original deleted, otherwise the warning message will keep re-appearing when entering the circuit schematic. Once the changes have been added satisfactorily, the circuit can be recompiled.

Tools > Network > Add Part

Used to add a new part to the layout and the parts list. A part prefix and number plus an outline has to be specified for the new part, the part is then listed in the tray ready to be placed on the artwork.

To aid selection, the prefix can be selected from those already defined in the design and the first available number for that prefix appears automatically (can be changed). The outline for the part is selected from a list.

The list of outlines is made up from the outlines held in the design outline library and any others from the master outline library that are not already held in the design. If an outline from the master library is selected, a copy is automatically transferred to the design library when the part is added.

To add a part:

Select *Tools > Network > Add Part*, a window appears similar to that shown in Figure 186.

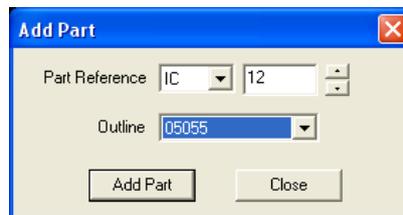


Figure 186

Select the down-pointing arrow alongside the *Part reference* setting and select the part prefix required from the list that appears. The number alongside is automatically set to the first unused number available - this can be changed if required using the spin controls alongside or by selecting the number and typing in the number required (duplicate references will not be accepted). Use the down-pointing arrow alongside the *Outline* setting to choose the outline for the part from the list available.

Select the *Add Part* button when the window is set as required. The part is added to the placement tray. More parts can be added as required until the window is closed, by selecting the *Close* button.

Tools > Network > Test Points Setup

Used to control the part prefix and component outline used when test points are automatically added to the design.

When selected, a window appears similar to that shown in Figure 187.

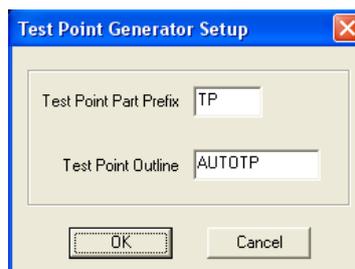


Figure 187

There are two settings:

Test Point Part Prefix

Specifies the prefix that will be used for the auto-generated test points.

It can be a prefix in use already or a new one.

If a prefix that is not already defined in the part prefix table is specified, then the new prefix will be added to the part prefix table when the test points are added to the design.

The testpoints will be added starting from reference 1. For example if TP is specified as the part prefix, then the test points will become TP1, TP2, TP3, etc.

If a used prefix is specified, then the new test point references will fill in any gaps in the existing references, then be added to the end. For example, if TP is specified as the part prefix and TP's 1, 3, 5 and 7 are already on the design, then the new test points will become TP2, TP4, TP6, TP7, TP8, etc.

Test Point Outline

Specifies the component outline that will be used for the auto-generated test points.

An outline that doesn't exist can be specified in this setup window, but the test points cannot be added to the artwork until the outline exists within the design's component outline folder.

Once the required prefix and outline have been specified, the test points can be added to the design, using the *Tools > Network > Test Points* command.

Tools > Network > Test Points

Used to automatically add test points to the design. This command will add a test point to each user-selected net, which can be placed as required and routed. Unused (unplaced) test points can also be removed automatically using this command.

The test points appear in the parts and wiring list automatically and a report is provided so that the schematic can be updated manually.

Before using this command, ensure the *Tools > Network > Test Points Setup* window is set as required.

Using the test point generator

Typically test points are added once the layout is complete, as it is often unnecessary to add a test point to every net.

Note: if a circuit diagram has been drawn, it is easier to update the circuit and know that it is correct, if the artwork has been completed and checked with no errors reported before you add the test points.

Assuming the artwork is complete and correct (although this is not necessary, see note above), select *Tools > Network > Test Point Setup*. Specify the part prefix and outline to be used for the test points.

Select *Tools > Network > Test Points*. A window appears similar to that shown in Figure 188.

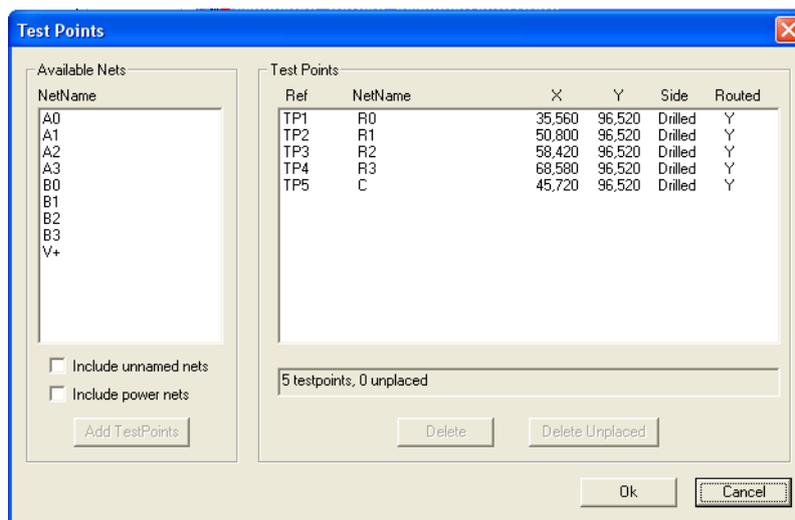


Figure 188

The right hand side lists all the test points currently in the design (if any) and their locations.

The left-hand side lists all the named nets in the design. Tick boxes below the list allow the unnamed nets and power nets to be added to the list if required.

To add a test point to one of the nets in the list, select the net, then select the *Add TestPoints* button. A test point appears in the list in the right-hand side of the window and the net is removed from the left.

Repeat for any other nets that require test points.

Groups of, or all the nets can be selected by holding down the left hand button over a net and moving the mouse up/down, to highlight them, let go of the button, then select *Add TestPoints*.

Select *OK* once all the test points required have been added.

The placement tray appears in the artwork editor allowing the new test points to be placed as required and then the connections routed.

Removing unwanted test points

Test points that have been added to the tray but un-required can be left in the tray (or returned to the tray if they were placed) and removed.

Open the test point generation window (*Tools > Network > Test Points*), the right hand side of the window indicates which test points have been placed along with details about position, side of the board and whether the net has been routed, it also lists the unplaced test points.

Select the *Delete Unplaced* button to remove all the unplaced test points from the design - the nets that the unplaced test points were connected to will re-appear in the left hand side of the window and test points can be added to those nets in the future if required.

Back Annotation and Other Information

The *Undo/Redo* commands can be used to undo/redo these actions if required (once the test point window is closed).

The parts/wiring list is automatically updated.

If the artwork checking routines are run after the test points have been added, and provided all the required test points have been placed and routed and unrequired test points removed, there should be no errors reported (provided there were none to start with).

Upon entry to the hierarchical schematic editor a warning will appear to indicate that the circuit and parts/wiring list may not match. Note: if a p/w list is extracted from the circuit at this stage, all the added test points will be removed from the parts/wiring list and removed from the artwork (the tracks will remain) as they are not present on the schematic.

Open the *Log Files* folder from the navigator and open the *netmods* file. A report appears listing all the parts and connections that have been added.

This report should be used to update the schematic.

If the p/w list is extracted once the changes to the schematic are complete, the artwork checking routines should not report any additional errors. If there are errors, then the schematic hasn't been updated correctly.

Tools > Network > Add Link

Used to add a connection (link) to the layout and the wiring list. Refer to the notes under the *Network commands* heading for details on updating the schematic.

If the connection is added to an existing signal, the new connection uses the same track size code. If the new connection is being added between unconnected pins, a prompt appears requesting the track size code for the new connection.

Select *Tools > Network > Add Link*, then select the component pin where the connection will start from. As the cursor is moved, a connection stretches with it. Once the connection has been started but not released, clicking the right hand mouse button cancels it. Select the component pin where the connection is to end. If either of the pins is already connected, the connection is inserted with the same size code. If both pins are unconnected, then a window appears requesting the size code for the new connection. If <enter> is pressed before changing the size code, the default signal track size code as displayed in the window is used.

The connection should be routed in the usual way.

A warning message is posted in the status bar if an attempt is made to add a connection to an unassigned pin. For instance to an unassigned mounting hole on a connector. The connection will not be made.

Tools > Network > Delete Pin

Used to delete all the connections from a pin and update the wiring list. Refer to the notes under the *Network commands* heading for details on updating the schematic.

If the selected pin is connected to just one other pin, the connection is removed and both nodes are removed from the wiring list.

If the selected pin is part of a larger net, the net is reconnected so that it is not split in two. This can cause routed tracks to be returned to connections. For instance, take a situation where pins 1 and 2 and 3 are connected and routed in that order. If pin 2 is selected, the tracks will be removed and a connection inserted between pins 1 and 3. The wiring list will show pins 1 and 3 as connected.

Select *Tools > Network > Delete Pin*, then select the pin that the connection(s) will be removed from.

Tools > Network > Move Node

Used to move all the connections from one pin to another pin and update the wiring list. Refer to the notes under

the *Network commands* heading for details on updating the schematic.

Connections are moved from one pin to another, they are not swapped. For instance, if a connection is moved to a connected pin, there will be two connections attached to that pin and the first pin will be unconnected.

Connections may be moved to either unconnected or connected pins. If a connection is moved to a connected pin, it becomes part of that net.

Once the move has taken place, the signal connections are automatically reconnected to show the shortest connection path, as defined by the *Manual Routing Parameters* window. Refer to the *Parts > Reconnect Power* command for details on reconnecting power connections.

Routed tracks are converted back to connections when they are moved.

Select *Tools > Network > Move Node*, then select a connected pin. As the cursor is moved, a line stretches from the selected pin to indicate which pin was selected. Select the pin that the connection(s) will be moved to. The move takes place.

Tools > Network > Pin Swap

Used to swap connections between pins and update the wiring list. All the connections attached to one pin are swapped with all the connections on another pin.

Refer to the notes under the *Network commands* heading for details on updating the schematic.

The pins do not have to be equivalent to allow the swap to take place - this is not pin swapping between equivalent pins - this command will change the electrical functioning of the circuit (refer to *Parts > Pin Swap* for pin swapping equivalent pins).

Routed tracks are converted back to connections when they are swapped. The signal connections are automatically reconnected according to the settings in the *Manual Routing Parameters* window after the swap has taken place. Refer to the *Parts > Reconnect Power* command for details on reconnecting power connections.

Select *Tools > Network > Pin Swap*, then select a connected pin. As the cursor is moved, a line stretches from the selected pin to indicate which pin was selected. Select the connected pin that it will be swapped with. The swap takes place.

Tools > Network > Set Net Name

Used to rename a net on the artwork and in the wiring list. Refer to the notes under the *Network commands* heading for details on updating the schematic.

Either a connection or track can be selected, the change affects the complete net.

When the new name is selected, if the name is already in use by another net, then a warning window appears. The rename can be confirmed or cancelled. If the rename is confirmed, the selected net is NOT linked to the other net with the same name.

It is possible to "join" the two nets together if required and this is achieved using the *Tools > Group Signal Names* command whilst the net list editor is open. Until that time, the two nets are treated separately and if they are shorted together on the artwork, this will cause a short-circuit error when the artwork is checked.

Every net on the artwork has a name. Because signal names assigned by the user are optional, Ranger assigns its own id to each net. Ranger's assigned names start at R0016 and proceed numerically, R0017, R0018, etc.

(The auto-assigned names can be viewed from the net list editor if required.)

To change a net name:

Select *Tools > Network > Set Net Name*, then select the track to be renamed. A window appears showing the *Nets Current Name* and with a space for the *New Net Name* to be typed.

The *current name* setting is set to the net's user-assigned signal name, whilst unnamed nets show their Ranger-assigned name. Once the new name has been typed, select *OK* to proceed.

Note: If *OK* is selected and a new name has not been typed, the net retains its Ranger-assigned name but its user-assigned signal name is removed (if it had one).

Take care when using UNDO after this command has been used - the undo will take place but there is no visual indication of what has been "undone".

Silk-screen

The silk-screen outline definitions are added to the component outline using the outline editor.

When the parts are placed on the artwork, a visual representation of the silk-screen outline and the labels can be displayed (*View Control* dialogue bar), but these are not the actual silk-screen outlines or labels that will appear on the finished board. They are used for identification purposes only and cannot be modified.

The silk-screen outlines and labels have to be added to a silk-screen layer of the board. This can be achieved automatically from within the artwork editor using the *Tools > Generate silk-screen* command.

Typically, the silk-screen layers are not created until all the placement, routing and part renumbering is complete, but they can be generated at any time.

Data added to the silk-screen layers can be checked to ensure the text or lines do not cover pads/holes on the outer copper layers and that the text is outside the area occupied by the auto-placement footprint.

Silk screen information

The silk data can be generated at any time, but it is typically created once the layout is complete.

When the silk-screen layers are generated, the silk screen labels and outlines are added to user-specified layers of the board. This allows the labels to be moved using the *Text* commands, and the outlines to be modified individually using the *Amend* commands if required. It does not matter which layers are used, providing they are not used for anything else.

The labels are positioned automatically on the datum of each part. (Do not be tempted to move the position of the datum points within each outline, because this will have the affect of moving the parts on the board.) The silk screen labels can subsequently be moved using the *Text > Move* command.

When the silk-screen data is generated, the selected layer(s) is reassigned to a silk-screen layer if it is not already defined as one.

it is assumed the silk screen data is made from a non-conducting material so no connectivity checks are made on silk-screen layers. However, checking routines are available which indicate whether silk-screen data covers pads/holes on the outer copper layers and whether the text is inside the area occupied by the auto-placement footprint.

Tip: Modifications to the board in the future will be made easier if the silk screen outlines and labels are added to **different** layers. (The layers would obviously need to be output together.) This is because when the outlines (not labels) are generated, all the data from the specified layer is removed first. If the labels are on the same layer, the time spent moving them into position will have been wasted. Adding the labels to a different layer means that the outlines can be re-generated if a modification has been performed, without affecting the silk screen labels.

When the silk-screen labels are generated, they are added to existing data on the chosen layer. This means that once the silk screen labels have been generated, they should not be auto-generated again even if new parts have been added to the artwork, otherwise duplicate labels will be produced for the original parts. Instead add extra labels using the *Text > Get Label* command. Redundant labels can be deleted using the *Text > Delete* command. This method is less time consuming than regenerating the labels and having to re-position them again.

Free layers

The *Tools > Generate silk-screen* command is also used to add data from the "free layers" as defined in the component outline editor, to user selected layers in the artwork.

Free layer information

Using the component outline editor, outlines can have additional *pads* and *tracks* added on upto 8 *free layers*, which are given user-defined names. When the outlines are placed on the artwork, the free layer information is not seen or present, until it is generated via the *Tools > Generate silk-screen* command, at which time it is placed on user-selected layers within the artwork editor.

It does not matter which layer numbers are selected for the "free layer" data, providing they are not used for anything else, as the data is *added* to existing data on the layer.

When the free layer data is generated, the selected layer(s) is converted into a silk-screen layer if it is not already defined as one.

Although the free layer data was added using the *Copper* commands in the outline editor, it is defined as silk screen data when it is added to the artwork.

Generating the silk screen data and/or "free layer" data.

From within the artwork editor, select *Tools > Generate silk-screen*. A window similar to the one in Figure 189 appears. The *Outline Free Copper* section of the window is not active unless there are some outlines on the artwork that have free copper in them.

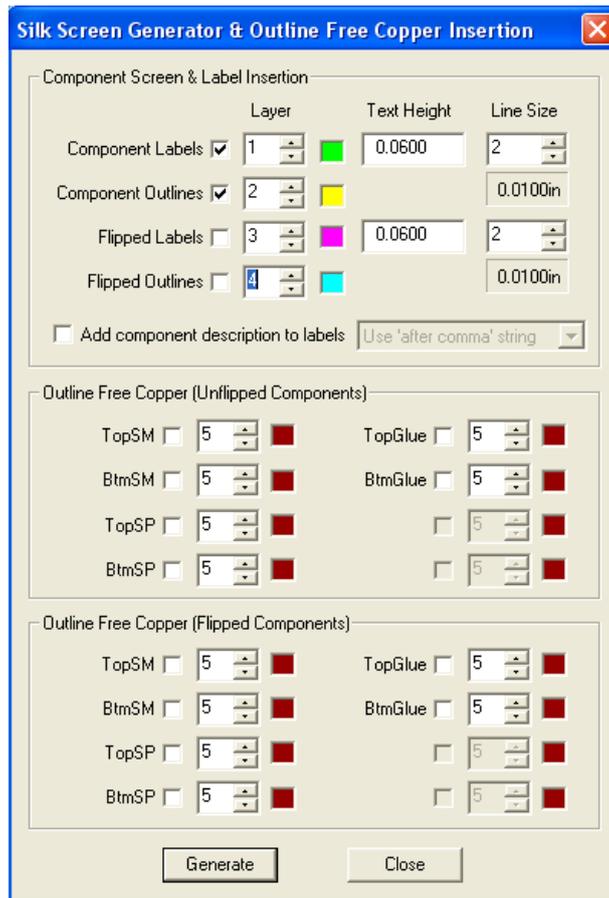


Figure 189

This window should be setup as required, using the following information for guidance.

Component screen and Label Insertion

Component labels silk screen labels for all parts that are un-flipped in the artwork editor.

Component outlines silk screen outlines for all parts that are un-flipped. The outline shapes and the size of the lines are defined via the component outline editor.

Flipped labels silk screen labels for all parts that have been flipped.

Flipped outlines silk screen outlines for all parts that have been flipped. The outline shapes and the size of the lines are defined via the component outline editor.

Important note: all the un-flipped parts are not necessarily on the top of the board and all the flipped parts are not necessarily on the bottom of the board.

Un-flipped parts are placed on the side of the board as defined in the outline library. In the case of surface mounted parts, they can be defined on the top or bottom of the board. If the pads are placed on the bottom of the board in the outline, then this part when it's un-flipped in the artwork will be on the *bottom* of the board. It's therefore important to define all outlines from the same side, usually the top to ensure that all flipped parts are on the same side of the board.

Layer Indicates which layer the silk-screen data for that category will be added to, providing the category is selected/ticked.

Bear in mind that the *component outlines* and *flipped outlines* overwrite data already on the selected layer, so ensure you choose an appropriate layer.

Use the spin controls to select the layer required. The coloured boxes indicate the colour of the selected layer, which may help you.

Layers V, T and B cannot be used.

Check-box Between each of the above settings is a check-box. To indicate that the labels or outlines for parts and/or un-flipped parts should be generated, the check-box should be ticked. If the box is not ticked, then that category of data will not be generated.

Text Height Specifies the height of the characters that form the silk screen labels.

Line size Specifies the track code which will be used to form the characters in the labels. The actual

size related to this code is displayed underneath the code. (The line size for the outline is defined within the component outline definition.)

Add component description to labels

If this option is selected/ticked, then the value assigned in the "Type" field of the parts list is also added to the layer that the silk-screen labels are added to, for each part. If the labels are not included, then this setting has no effect.

The setting alongside allows selection of the whole type field, part of the field up to the comma, or part of the field after the comma. (In an auto-generated parts list the *Type* field is made up of: "partname,value" for example R0.25W,10k. So either R0.25W,10k will appear, R0.25W or 10k depending on the setting chosen.)

The value is added as a separate text string that can be moved independently of the labels.

Outline Free Copper

This part of the window only becomes active if at least one outline on the artwork has some free copper defined within it.

Each named free copper layer is listed, with a section for unflipped parts and one for flipped parts. This allows layers to be created with reference to the side the parts are placed on. For example, when generating the top solder paste layer (if defined in the outline), then you would not want the bottom mounted parts included.

To add the "free" data from the outlines onto the artwork, choose an appropriate layer (typically one that's not used by anything else) and tick the box alongside for each layer to be generated.

Generating the data

Once the window has been filled out as required, select *Generate* to create the data on the selected layers. A window indicates the generation is complete, select *OK* to continue.

Create more silk-screen/free layers as required or *Close* the window.

If selecting the *Generate* button more than once, ensure that label layers or free layers are not selected again (remove the tick) or duplicate labels and "free copper" will be created on the layer.

When the window is closed, the layout is re-drawn and the data appears. If you cannot see it, ensure the layers are visible using the *View Control* dialogue bar.

Modifying the silk-screen and/or free copper

The silk screen labels can be moved using the *Text > Move* command. Generally, the labels are moved so that they can be seen on a populated board, and are not covering through plated holes or component pads.

The text (labels and descriptions) can be edited/resized using the *Text* commands.

The silk-screen outlines and data added to the "free layers" can also be modified if required using the *Amend* commands.

To remove all the silk-screen data from a layer, use the *Region > Delete* command with only that layer selected.

Additional text can be added to the layer as required using the *Text* commands.

If a label is accidentally deleted, use the *Text > Get Label* command to re-insert it.

Modifications to completed designs

At some stage in the future it's likely that a completed artwork will have parts added or removed which will mean the silk-screen outlines and labels have to be updated.

If the labels and outlines have been added to separate layers as advised, this can be accomplished quite easily. Simply re-generate the silk-screen **outlines** as described previously (on to the same layer), the existing outlines will be removed (along with anything else on that layer) and the "new" set added.

Do not re-generate the labels – unless duplicate labels for all the existing parts are required. Use the *Text > Get Label* command to add the new part labels and the *Text > Delete* command to remove unwanted labels. This method is usually less time consuming than starting from scratch, for example by *Window* deleting the existing labels and re-generating them, as all the labels will have to be moved into position again.

If the labels and outlines were added to the same layer originally, then this will prove more time consuming. When the outlines are regenerated on the same layer, all the existing data is removed so you will lose all the labels. When regenerating the labels, put them on a different unused layer.

The alternative is to manually amend the existing silk-screen outlines by deleting the unwanted outlines and drawing in the new ones (using the *Amend* commands) and removing and adding the labels using *Text > Delete* and *Get Label* commands. Note: if the manually drawn outlines do not exactly match the original outline defined in the outline library, the silk-screen outline will not move if the outline is subsequently moved.

Tear drop pads

A "teardrop" widens a track as it enters a via pad, as shown in Figure 190. This facility can be quite useful if vias with very small copper areas are used, since the extra copper attached to the via can stop them peeling off the board.



Figure 190

Teardrops should be added once the tracking is complete. If modifications have to be performed to the board, it is best to delete the teardrops first.

Once teardrops have been added to the board, the artwork checking routines should be run to ensure errors have not been introduced.

The teardrops can be modified using the *Amend* commands.

Creating the teardrops

From within the artwork editor, select *Tools > Teardrop Pads*. A window appears similar to that shown in Figure 191.

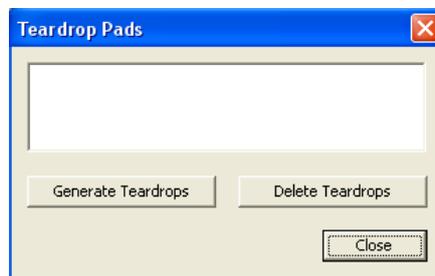


Figure 191

Select *Generate Teardrops* to insert the teardrop pads. A report appears in the window to indicate how many tear drop pads have been added to each layer.

Once the teardrops have been generated, close the window. The teardrops will be seen on tracks leading to vias.

Run the artwork checking routines to ensure errors were not introduced.

Deleting teardrops

From within the artwork editor, select *Tools > Teardrop Pads*. A window appears similar to that shown in Figure 191. Select *Delete Teardrops*. A report appears in the window to indicate how many tear drop pads have been removed from each layer.

Once the teardrops have been generated, close the window.

Only the teardrops are deleted, the vias remain.

Parts commands

Before parts can be placed on the board, they must appear in the tray.

Parts are always selected by their datum. This is indicated by the position of the part's label. Part labels are shown in yellow for parts that are unflipped (typically mounted on the top of the board), and in cyan for parts that are flipped (typically mounted on the bottom of the board). The labels of flipped parts have their labels slightly offset, a tail extends to the datum point. This allows the labels to be read more easily when parts are mounted directly over the top of one another. Part labels can be switched on and off using the *View > Part Labels/Setup Visibility* command.

The left mouse button is used to select parts that are unflipped. The right mouse button is used to select flipped parts. However, when the artwork is displayed in "flipped/mirrored" mode, the sense of the buttons becomes reversed making it easier to position parts when working on the back of the board.

Each command is described individually as follows:

Parts > Place

Used to manually transfer parts listed in the tray onto the board. The tray is visible when the *Parts > Place* command is selected (though it can be hidden - refer to the *View > Navigator/Browser/Properties* command for details).

Before parts may be manually or automatically placed on the board, they must appear in the tray. When the tray is open parts can be added to, and removed from it.

During manual placement, parts appear on the end of the cursor in the order that they appear in the tray, but this can be overridden. This allows the user to control which parts are placed, and in what order. During auto-placement, parts are added according to the settings in the *Autoplace* dialogue bar.

Controlling the tray

Select *Parts > Place*, the tray appears as shown in Figure 192 - it's empty to start with. It has four commands that are used to control the parts that are added to the tray, *Add*, *Delete*, *Sort* and *Empty*.



Figure 192

Tray commands - Add

Used to add parts to the tray in preparation for placement. When selected a window similar to the one in Figure 193 appears.

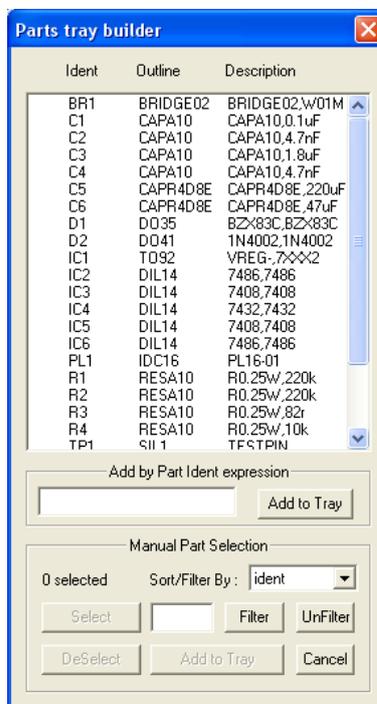


Figure 193

Parts may be added to the tray by typing in their part reference (IC1, R1, etc.) in the *Add by Part Ident expression* dialogue, then selecting *Add to Tray* from alongside. The wildcards * and ? can be used to represent a group/range of parts. For example * represents all parts, *IC** all IC's. *IC?* will select IC's with one numeric value (IC1-9), *IC??* will select IC's with two numeric values (IC10-99), etc. All parts with ident's matching the entered expression will be placed into the tray.

Alternatively, the parts that should be added to the tray can be selected from the list, then the buttons in the *Manual Part Selection* area of the window used as follows:

To select a part either, select the part so that it highlights, then select the *Select* button, or double-select the part quickly. A double asterisk (**) alongside the part indicates it's selected. A group of parts in the list can be selected by pointing at a part, then holding down the left button, then moving the cursor up or down - the parts high-light - then select the *Select* button.

Parts can be deselected in the same way with a double-click or using the *DeSelect* button.

Once a/some parts are selected, select the *Add to Tray* button to transfer them to the tray.

The order that the parts appear in the list is controlled by the *Sort/Filter by* parameter as follows:

Ident lists the parts in alphabetical/numeric order using the *Ident* field (A1, A2, A3, B1, B2, C1,

C2, etc.) to sort by.

Outline lists the parts in alphabetical/numeric order using the *Outline* field to sort by.

Description lists the parts in alphabetical/numeric order using the *Description* field to sort by.

The *Filter* button can be used to further filter the list. For example, set the *Sort/Filter by* parameter to *Outline*, then type *CAP* in the box between the *Select* and *Filter* buttons. Select the *Filter* button to action the filtering. Any outlines with *CAP* in their name will appear in the list. Select the *Unfilter* button to restore the complete list.

This facility is useful on larger designs.

Tray commands - Delete

Used to delete parts from the tray - the parts are not deleted from the design and can be re-added to the tray as required. The part must be selected, then the *Delete* button selected.

Tray commands - Empty

Used to delete all the parts from the tray. The parts are not deleted from the design and can be re-added to the tray as required.

Tray commands - Sort

Used to sort the parts in the tray into alpha-numeric order.

Placing the parts

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Only parts that appear in the tray can be placed.

Select *Parts > Place*, as the cursor is moved, the first part listed in the tray appears attached to it. Its ident is displayed in the status bar. If the part has been assigned to an EMC group on the schematic, then the group to which it belongs is also shown in the status bar.

Clicking the left mouse button releases the part at the current mouse position, the next part in the tray attaches itself to the cursor. As the part is moved, its name is highlighted in the tray. The name is removed from the tray once the part has been released on the board.

Parts appear attached to the cursor in the order they appear in the tray. The placement order can be altered by selecting any of the parts listed in the tray, placement continues from that part onwards.

As the parts are moved, their signal connections are automatically reconnected, as defined by the *Manual Routing Parameters* in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting.

Clicking the right hand mouse button stops the placement process. The part that was attached to the cursor remains in the tray.

If the part labels are visible (*View Control* dialogue bar) the label will appear at the parts datum position – this is not the silk-screen label and is shown purely for identification purposes.

Parts > Unplace

Used to remove parts from the board. The parts are returned to the tray for subsequent re-placement. The parts are not removed from the design.

Only unrouted parts can be removed. Use the *Mroute > Ripup* command to convert tracks back to connections.

Unflipped parts (typically mounted on the top of the board) should be selected with a click of the left mouse button. Flipped parts (typically mounted on the bottom of the board) should be selected with a click of the right mouse button.

(When the artwork is displayed in “flipped” mode, the sense of the buttons should become reversed, but this is not the case in versions upto v2.22 and possibly beyond, check the readme.txt file for details of changes made after the date of this guide.)

Select *Parts > Unplace*. Select the datum of the part to be removed - the part label if visible (*View Control* dialogue bar) indicates the part's datum position.

Parts > Move

Used to move parts that have already been placed on the board.

Unflipped parts (typically mounted on the top of the board) should be selected with a click of the left mouse button. Flipped parts (typically mounted on the bottom of the board) should be selected with a click of the right mouse button. (When the artwork is displayed in “flipped” mode, the sense of the buttons becomes reversed.)

Once a part is being moved, the opposite mouse button can be clicked to cancel the move and restore the part to its original location.

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

As the part is moved, its ident is displayed in the status bar and If the part has been assigned to an EMC group

on the schematic, then the group to which it belongs is also shown in the status bar.

Connectivity is maintained when parts are moved, whether the connections are routed or unrouted.

As the parts are moved, their signal connections are automatically reconnected, as defined by the *Manual Routing Parameters* in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting. Reconnection does not take place on partially routed connections or tracks.

When a routed part is moved, the routed tracks will probably cause clearance violations when the part is released. In these situations, it is probably easier to unrout the last track segment going to each pin before moving the part to make re-routing easier, using the *Mroute > Ripup* command.

Select *Parts > Move*. Select the datum of the part to be moved with the appropriate mouse button, then move the cursor and the part to its new position and click the same mouse button to release it.

Clicking the opposite mouse button as the part is being moved, returns it to its original position.

The part label if visible (*View Control* dialogue bar) indicates the part's datum position.

Parts > Key Move

Used to move and/or rotate parts to specific positions on the board, by entering co-ordinates.

Co-ordinates can be entered in imperial or metric units providing the required units are selected prior to selecting the part (*Edit > Units*).

Connectivity is maintained when parts are moved and/or rotated, whether the connections are routed or unrouted.

When the part is moved and or rotated, unrouted signal connections to it are automatically reconnected according to the settings in the *Manual Routing Parameters* window in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting. Reconnection does not take place on partially routed connections or tracks.

When a routed part is moved and or rotated, the routed tracks will probably cause clearance violations. In these situations, it is probably easier to unrout the last track segment going to each pin before moving or rotating the part to make re-routing easier, using the *Mroute > Ripup* command.

Unflipped parts (typically mounted on the top of the board) should be selected with a click of the left mouse button. Flipped parts (typically mounted on the bottom of the board) should be selected with a click of the right mouse button. (When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.)

Select *Parts > Key Move*. Point at and select the datum of the part to be moved. A window appears containing the part's name, its current X, Y position and its angle of rotation.

The part label if visible (*View Control* dialogue bar) indicates the part's datum position.

The angle of rotation is specified in degrees anti-clockwise. 0 degrees is the orientation of the outline in the library.

To change any of the values, select the value to be changed and enter the new one. Once the appropriate values have been changed, select *OK* to implement the change.

Select *Cancel* to leave the part in its original position.

Parts > Rotate

Used to rotate parts in 90 degree increments anti-clockwise. Use *Parts > Key Move* to rotate parts in increments of 1 degree.

Unflipped parts (typically mounted on the top of the board) should be selected with a click of the left mouse button. Flipped parts (typically mounted on the bottom of the board) should be selected with a click of the right mouse button.

Connectivity is maintained when parts are rotated, whether the connections are routed or unrouted.

Unrouted signal connections to the part are automatically reconnected according to the parameters in the *Edit > Manual Routing Parameters* window when the part is rotated.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting. Reconnection does not take place on partially routed connections or tracks.

When a routed part is rotated, the routed tracks will probably cause clearance violations. In these situations, it is probably easier to unrout the last track segment going to each pin before rotating the part to make re-routing easier, using the *Mroute > Ripup* command.

Select *Parts > Rotate*, then point at and select the part's datum. Each click of the mouse button rotates the selected part anti-clockwise by 90 degrees.

The part label if visible (*View Control* dialogue bar) indicates the part's datum position.

Parts > Flip

Used to flip surface mounted parts from one side of the board to the other.

Parts are flipped about their original Y axis, as defined in the outline library. For instance, parts are turned left to right about their datum, unless they have been rotated by 90 or 270 degrees first, in which case they are turned from top to bottom about their datum.

Unflipped parts are selected with a click of the left mouse button. Flipped parts are selected with a click of the right mouse button. (When the artwork is displayed in “flipped” mode, the sense of the buttons becomes reversed.)

Connectivity is maintained when parts are flipped, whether the connections are routed or unrouted.

When the part is flipped, unrouted signal connections to it are automatically reconnected according to the settings in the *Manual Routing Parameters* window in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting.

Reconnection does not take place on partially routed connections or tracks.

If the part has routed connections going to it, the last segment of the track is ripped up and becomes a partial unrouted when the part is flipped, otherwise the track would become disconnected from the pad.

Select *Parts > Flip*, then point at and select the datum of the part to be flipped. The part must now be selected with the opposite mouse button as it is mounted from the opposite side of the board.

The part label if visible (*View Control* dialogue bar) indicates the part’s datum position.

This command can also be used to introduce additional copper on parts that have had their outlines swapped using the *Parts > Change Outline* command, or to restore stringers that have been ripped up. (Use *flip* twice to restore the part to its original layer.)

Parts > Swap

Used to swap the position of two parts, one with the other. The parts do not have to be identical, be on the same side of the board, or have the same number of pins.

If either part has routed tracks connected to it, the tracks must be completely unrouted before the parts can be swapped, using the *Mroute > Ripup* command.

Outlines that have additional copper attached are regarded as partially routed. The copper will have to be ripped up using the *Mroute > Ripup* command before using the swap routine. Once the parts have been swapped, use the *Parts > Flip* command twice to restore the copper and the part to the correct side of the board.

Connectivity is maintained when parts are swapped.

When the parts are swapped, unrouted signal connections to them are automatically reconnected according to the settings in the *Manual Routing Parameters* window in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting.

Unflipped parts are selected with a click of the left mouse button. Flipped parts are selected with a click of the right mouse button. (When the artwork is displayed in “flipped” mode, the sense of the buttons becomes reversed.)

Select *Parts > Swap*, select the datum of the first part with the appropriate mouse button, it highlights. Now select the datum of the part it is to be swapped with, with the appropriate mouse button. The parts swap their positions over.

The part label if visible (*View Control* dialogue bar) indicates the part’s datum position.

Parts > Align X/Y

Used to align parts in either the X or Y axis by their datum. The X axis is the vertical axis, and the Y horizontal.

When positioning the reference axis (line), it will snap to part datums unless the cursor is located away from a part, in which case it will snap to grid/half grid if grid snapping is switched on.

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Connectivity is maintained whether the parts are routed or unrouted. Typically, parts are aligned before any routing is performed.

When the parts are aligned, unrouted signal connections to them are automatically reconnected according to the settings in the *Manual Routing Parameters* window in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting.

Reconnection does not take place on partially routed connections or tracks.

When a routed part is moved, the routed tracks will probably cause clearance violations. In these situations, it is probably easier to unrout the last track segment going to each pin before moving the part to make re-routing easier, using the *Mroute > Ripup* command.

Select *Parts > AlignX/Y*. Locate the cursor in a position suitable for aligning the required parts, then click the *left* hand mouse button until the reference line is in the required location.

A blue dashed line appears that represents the axis of alignment. Select each part to be aligned using the *right* hand mouse button (irrespective of the side of the board the parts are positioned on, or whether the artwork is

displayed in flipped/unflipped mode). The parts move so that their datum lies over the axis line. The alignment axis can be moved by selecting another position with a click of the *left* hand mouse button.

Parts > Highlight

Used to highlight the unrouted connections radiating from a particular part.

When a part is selected for highlighting, a message appears in the status bar indicating the length (in inches) of totally unrouted connections radiating from it. Partially unrouted connections are not included in the total.

Select *Parts > Highlight*, then select the part whose connections you wish to highlight, with a click of the left button (irrespective of the side of the board the parts are positioned on or whether the artwork is displayed in flipped/unflipped mode). The highlighting disappears when another part or command is selected.

Parts > Find

Used to find a particular part by virtue of its reference name. Once selected, a window appears where the part reference can be typed, or select the down-pointing arrow to the right of the text entry area to produce a list of all the parts in the design, from which one can be selected. Once the name has been entered/selected, the screen is panned if necessary to identify the required part. A balloon points to the part's datum and supplies details about the part. The schematic and parts list icons can be selected from the balloon to locate the part in either the schematic or parts list.

Parts > Change Outline

Used to replace a part's outline shape for another within the same family group. (Family groups are defined by right-clicking on the *Component Outline* folder.)

The datum of the new footprint is positioned in the same location as the one it is replacing.

If the new outline has additional copper defined, it does not appear automatically. Use the *Parts > Flip* command (twice) on the part to introduce the copper (the second flip returns the part to the correct side of the board).

Connectivity is maintained when part outlines are replaced, whether the connections are routed or unrouted. Connections or tracks remain connected to pins with the same pin number. For instance, a connection stays with pin 1, irrespective of its new position.

When the part is changed, unrouted signal connections to it are automatically reconnected according to the settings in the *Manual Routing Parameters* window in the *Configuration* folder.

Power connections may/maynot automatically reconnect, depending on the *Parts > Dynamic Powernets* setting. Reconnection does not take place on partially routed connections or tracks.

When the outline of a routed part is replaced, the routed tracks may cause clearance violations. In these situations, it is probably easier to unrout the last track segment going to each pin before replacing the outline to make re-routing easier, using the *Mroute > Ripup* command.

Select *Parts > Change Outline*, then point at and select the part whose outline shape will be replaced. A window appears listing the outlines in the same family group. Because an outline can belong to a family in the design library and the master library, the letter M alongside an outline means that it is from the master family. Select the outline required, followed by *Apply* to implement the change. If an outline from the master library is selected, a copy is transferred to the design library.

Unflipped parts should be selected with a click of the left mouse button. Flipped parts should be selected with a click of the right mouse button. (When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.)

If *Cancel* is selected, the outline is not replaced.

The outline change is back-annotated to the parts list. If the schematic is ever re-compiled, ensure the schematic extraction rules (*Tools > Parts/Netlist Extraction > Setup*) are set as required as they control whether the outline change remains, or reverts back to the original setting. A warning message will indicate that the outline called up by the schematic part is different to the one currently used on the artwork.

There are a number of ways to restore the original outline, the easiest is to change the outline again in the artwork editor (*Parts > Change Outline*), alternatively, edit the parts list manually and re-type the outline name.

Parts > Gate Swap

Used to swap equivalent gates within the same part or between parts to reduce connection crossovers on the board.

Only unrouted gates are available for swapping. Gates that have additional copper attached are regarded as partially routed. The additional copper will have to be ripped up using the *Mroute > Ripup* command before using the swap routine. Once the gates have been swapped, use the *Parts > Flip* command twice to restore the copper and the part to the correct side of the board.

The wiring list and circuit schematic is automatically back annotated.

A log report is generated and saved in the *Log Files* folder. The report is called *renumber*.

Parts must contain swappable gates before they can be selected for gate swapping. Swappable gates are defined within the schematic parts. This applies even if a circuit has not been drawn, i.e. if entry has been made via a parts and wiring list. The part name and value (Type field) in the parts list is the link between the parts on the board and the schematic parts. Gate swapping between parts is only permitted if the parts have identical type fields in the parts list.

Once a part has been selected for gate swapping, all the equivalent gates on the board that are available for swapping highlight. The gates selected for swapping do not have to belong to the part originally selected. For instance, you may know that IC3 is a 74LS00. If it is selected, all the available gates, as permitted from the schematic part definition, belonging to 74LS00's highlight. Selection of the gates for swapping can be made from any of the gates, not necessarily from those belonging to IC3.

It is easier to distinguish the gates if the part outlines are switched off.

Select *Parts > Gate Swap*, then select a part. All the available equivalent gates highlight. Select the gate to be swapped, followed by the gate it is to be swapped with. The connections move as the gates swap.

Unflipped parts should be selected with a click of the left mouse button, flipped parts with a click of the right mouse button. (When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.)

The gates are always selected with a click of the left button.

The right button, cancels the last action, so for instance if a gate within a part was selected, one click of the right button cancels the gate selection, the second the part.

Parts > Pin Swap

Used to swap equivalent pins within the same package to reduce connection crossovers on the board.

Only unrouted pins within the same group are available for swapping. To swap routed pins, use the *Mroute > Ripup* command to restore the routes to connections.

Pins that have additional copper attached, are regarded as partially routed. Any additional copper will have to be ripped up using the *Mroute > Ripup* command before using the swap routine. Once the pins have been swapped, use the *Parts > Flip* command twice to restore the additional copper and the part to the correct side of the board.

The wiring list and circuit schematic are automatically back annotated.

A log report is generated and saved in the *Log Files* folder. The report is called *renumber*.

Parts must contain equivalent pins before they can be selected for pin swapping. Equivalent pins are defined within the schematic parts. This applies even if a circuit has not been drawn, i.e. if entry has been made via a parts and wiring list. The part name and value field (type column) in the parts list is the link between the parts on the board and the schematic parts.

It is easier to distinguish the pins if the part outlines are switched off.

Select *Parts > Pin Swap*, then select a part. All the available equivalent pins within the part highlight. Select the pin to be swapped, followed by a pin within the same group. The connections move as the pins swap.

Unflipped parts should be selected with a click of the left mouse button, flipped parts with a click of the right mouse button. (When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.)

The pins are always selected with a click of the left button.

The right button, cancels the last action, so for instance if a pin within a part was selected, one click of the right button cancels the pin selection, the second the part.

Parts > Dynamic Pownets

Used to toggle the automatic reconnection of power connections on/off.

It can be undesirable for the power connections to be reconnected automatically based upon shortest net length. Also, on slow machines and depending on the size of the power nets, automatic reconnection can slow the response. So for both these reasons, the automatic reconnection of power nets can be switched on (ticked) or off (unticked) as required.

Power nets are reconnected in the direction defined in the *Configuration* folder, *Manual Routing Parameters* window.

If this command is off (unticked), then the *Power > Reconnect Power* command can be used to reconnect the power nets if required.

Select *Parts > Dynamic Pownets* to toggle on/off.

Parts > Reconnect Power

Used to reconnect the power connections. Only unrouted power connections are evaluated for reconnection. They are reconnected in the direction defined in the *Configuration* folder, *Manual Routing Parameters* window.

(This command is greyed out/unselectable if the *Parts > Dynamic Pownets* command is enabled (ticked).)

This command also restores signal connections to their part pins if their additional copper has been ripped-up and they were still hanging in mid-air.

Power connections, unlike signal connections, are not automatically reconnected as parts are moved, rotated, etc. unless the *Parts > Dynamic Powernets* command is enabled (on).

It is also possible to manually reconnect unrouted and routed tracks in a user-preferred order, using the *Mroute* commands.

Select *Parts > Reconnect Power*. The power connections reconnect. Any signal connections that have been left hanging in mid-air because of ripped-up copper are also re-attached to their pins.

Parts > Auto Renumber

Used to renumber the parts on the board in a specified logical order automatically. Manual part renumbering is also available (*Parts > Manual Renumber*). Parts on the board are often renumbered to make it easier for field engineers to find them.

Note: once parts have been placed on the board, never de-allocate and re-allocate part numbers from the schematic editor as this will have the effect of shuffling the parts on the board.

If the silk screen labels have been generated, they are automatically updated as the renumbering is performed.

During the renumbering process, parts on both sides of the board are scanned, in user defined directions. When parts within the range specified are located, they are renumbered sequentially starting from a user defined number.

The parts and wiring lists and circuit schematic are automatically back annotated.

Duplicate names and numbers are not permitted.

Note: if blocks were excluded from the schematic when the parts/wiring list was extracted, then it is possible that renumbering will duplicate part references on the schematic diagram. This will not cause a problem, unless the excluded block is subsequently included and a new parts/wiring list extracted. However a warning will be posted when the extraction is performed, indicating the duplicated references.

A log report is also generated which lists the old and new part numbers. The report can be accessed from the *Logfiles* folder. The report is called *renumber*.

Select *Parts > Auto Renumber*. A window appears as shown in Figure 194.

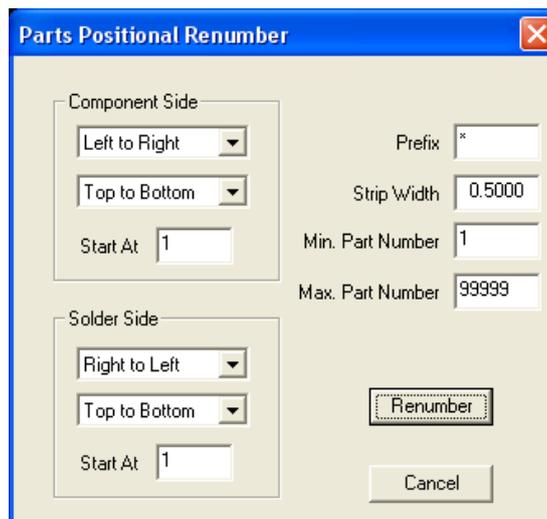


Figure 194

Choose the settings required then select *Renumber* to renumber the parts. Select *Cancel* to close the window without performing the renumber.

Component Side, Directions (Left to Right, Top to Bottom)

This setting controls the direction in which part datums on the component side of the board are scanned and renumbered.

For example, consider the following settings:

left to right

top to bottom

The board would be divided up into horizontal strips (left to right) as shown in Figure 195.

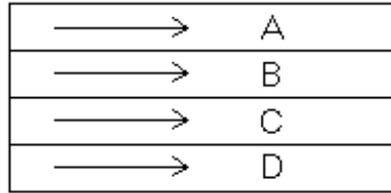


Figure 195

Each strip would be scanned from *left to right* as shown by the arrows, starting from the *top* of the board (A strip), working downwards to the D strip. (The strip widths are user-defined.)

If the settings were changed to:

right to left

top to bottom

Each strip would be scanned from *right to left* in the opposite direction of the arrows, starting from the *top* of the board (A strip), working downwards to the D strip. (The strip widths are user-defined.)

If the settings were changed to:

left to right

bottom to top

The board would be divided up into horizontal strips in the same way, and the strips would be scanned from *left to right*. However, the strips would be scanned from the *bottom* of the board (strip D), working upwards (to the A strip).

If the settings were changed to:

top to bottom

left to right

The board would be divided up into vertical strips (top to bottom) as shown in Figure 196.

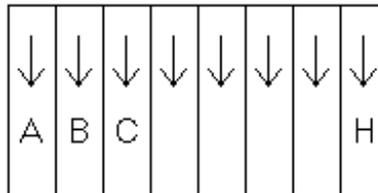


Figure 196

The strips would be scanned from top to bottom as shown by the arrows, starting from the left hand side (column A) across to the right (column H).

Start at

Specifies that the new part numbers should start at this number on this side of the board. Note: if this number is already in use, the next available number is used.

This setting also determines whether the component side or solder side of the board is scanned and renumbered first. If the same numbers are entered for *Component side*, *Start at* and *Solder side*, *Start at*, the component side of the board is always scanned first. If the numbers are different, the lower number controls which side is renumbered first.

Solder side, Directions (Left to Right, Top to Bottom)

These settings operate in exactly the same way as the *Component side* settings, except that part datum's on the solder side of the board are scanned.

Prefix

Indicates the parts to be renumbered, according to their prefix, i.e. IC, or R, or C, etc. The * wild card is used to represent ALL prefixes. If specifying individual prefixes, only one prefix can be specified at a time.

Strip width

Specifies the width of the strip (row or column, see above) that is scanned for part datums.

Parts *within* a strip are always scanned from bottom to top, right to left. This can cause unexpected results if a wide strip width is specified. For example, if left to right, top to bottom is selected, you would expect a vertical column of parts to be numbered 1, 2, 3, 4, etc. from the top down. However, if there were 3 parts in each strip, they would be numbered from the bottom of the strip up. The strip underneath would be numbered in the same way - see Figure 197. If this happens, select a narrower strip width.



Figure 197

Min Part No.

Specifies that parts from this number should be renumbered.

Max Part No.

Specifies that parts up to this number should be renumbered.

Examples:

Prefix:	IC
Component side start at:	1
Solder side start at:	1
Min Part No:	10
Max Part No:	25

Using these settings, only ICs from IC10 to IC25 would be renumbered. IC's on the Component side will be renumbered first (because the *Start at* setting for Component side and Solder side were both set to 1). If IC's 1 to 9 existed, then the first available part number would be IC10 even though the *Start at* setting was set to 1.

Prefix:	R
Component side start at:	2
Solder side start at:	1
Min Part No:	30
Max Part No:	55

Using these settings, the resistors between R30 and R55 would be renumbered. Resistors on the solder side of the board will be renumbered first, starting from 1, followed by resistors on the component side of the board, starting from 2.

Assuming that resistors from R1 to R100 exist on the layout, the following renumbering would take place. The solder side of the board is scanned in the directions defined at the top of the window for any resistors between R30 and R55. If any are found, an attempt is made to renumber them starting from R1. However, R1 to 29 exist, but are not being renumbered, so the first "free number" will be R30 (because the existing resistors R30 to R55 are being renumbered). Once the resistors on the solder side of the board have been renumbered, the component side of the board is scanned. An attempt will be made to renumber the first resistor it finds to R2, but this will cause a duplication, so the next "free resistor" number will be used. If R30 to R43 had been used on the solder side of the board the next free number would be R44, renumbering would continue from there.

A perhaps more typical application would be to renumber all parts on the component side of the board starting from 1000 and those on the solder side of the board starting from 2000. This would assist in finding parts on a populated board.

Parts > Manual Renumber

Used to renumber selected parts on the board. Automatic part renumbering is also available using the *Parts > Auto Renumber* command.

If the silk screen labels have been generated, they are automatically updated as the renumbering is performed.

The parts/wiring lists and circuit schematic are automatically back annotated.

Duplicate names and numbers are not permitted on the artwork layout.

Note: if blocks were excluded from the schematic when the parts/wiring list was extracted, then it is possible that renumbering will duplicate part references on the schematic diagram. This will not cause a problem, unless the excluded block is subsequently included and a new parts/wiring list extracted. However a warning will be posted when the extraction is performed, indicating the duplicated references.

A log report is also generated which lists the old and new part numbers. The report can be accessed from the *Logfiles* folder. The report is called *renumber*.

Select *Parts > Manual Renumber* and then select the part to be renumbered.

Unflipped parts should be selected with a click of the left mouse button and flipped parts with a click of the right mouse button. When the artwork is displayed in "flipped" mode, the sense of the buttons becomes reversed.

(When the artwork is displayed in "flipped" mode, the sense of the buttons should become reversed, but this is not the case in versions upto v2.22 and possibly beyond, check the readme.txt file for details of changes made after the date of this guide.)

A window appears indicating the parts number and requesting a new number for the part (only the number can

be changed, not the prefix).

Once the new number is typed and *OK* selected, the part is renumbered. If the new part number is already in use, an error message appears in the status bar to indicate the part already exists, and the renumber does not take place.

Parts > Status

Used to report the number of parts placed on the board and the number of parts waiting to be placed on the board.

Routing should not normally be started until all the parts have been placed on the board.

Select *Parts > Status*, and the report appears. Select *OK* to continue.

Parts > Set Datum

Used to move the datum of the artwork editor.

Unless the datum has been moved, it is positioned in the lower left corner of the working area.

The datum remains in the selected position until it is moved or restored to its original location.

Note: the new datum position is local to the artwork editor. All other editors use the original datum position as a reference point. For instance, within the artwork editor, part positions are referenced to the datum. If the parts list is viewed, it will be found that the part's positions are referenced to the original datum position.

The X-Y readout is given with respect to the datum position.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Parts > Set Datum*. Position the cursor in the required datum location and click the left mouse button. The datum is shown as a small yellow cross surrounded by a circle.

To restore the datum to its original position, select *Parts > Set Datum*, then click the **right** hand mouse button. The datum is restored to the lower left corner of the working area.

PartFix commands

These commands are used to fix parts in position so that they're not accidentally moved. Once fixed, they can be unfixed as required.

The part toggles between fixed and unfixed status each time it is selected.

If the *PartFix > Show Fixed Parts* toggle is enabled, then the area occupied by the component outline's autoplacement footprint changes colour (Figure 198) which shows the unfixed and fixed status of the same part..

Note: fixed parts can still be moved via the parts list editor but their fixed status is retained (in the new position).

When using the part fixing commands, unflipped parts should be selected with a click of the left mouse button and flipped parts with a click of the right mouse button. (This does not change whether the artwork is displayed in flipped/unflipped mode – check the readme.txt file for changes in future updates after v2.21.)

PartFix > Fix/Unfix Selected Parts

Used to fix or unfix individually selected parts.

The *PartFix > Enable Flipped Parts* and *PartFix > Enable Unflipped Parts* settings control whether flipped and/or unflipped parts can be selected.

Select *PartFix > Fix/Unfix Selected Parts*, then select a part, it toggles between a fixed and unfixed status.

Use the *PartFix > Show Fixed Parts* command to indicate all the fixed parts.

PartFix > Fix/Unfix All Parts

Used to fix/unfix all the parts.

The *PartFix > Enable Flipped Parts* and *PartFix > Enable Unflipped Parts* settings control whether flipped and/or unflipped parts are included.

Select *PartFix > Fix/Unfix All Parts*. All the parts change their status between all fixed or all unfixed.

Use the *PartFix > Show Fixed Parts* command to indicate all the fixed parts.

PartFix > Fix By Partcode

Used to fix parts by entering their part names.

The asterisk (*) can be used as a wild card to fix a group of parts. For instance: * operates on all parts, IC* operates on all the parts with IC as a prefix, R3* operates on R3, R30 to 39, R300 to 399, etc. The questionmark (?) can be used to represent one number, for example R2? represents resistors R2, R20 - 29.

The *PartFix > Enable Flipped Parts* and *PartFix > Enable Unflipped Parts* settings control which parts can be included and take precedence.

Select *PartFix > Fix By Partcode*, a window appears in which the name of the part (or parts) to be fixed should

be entered. Select *OK* or press <enter>.

The parts toggle between their fixed and unfixed status.

Use the *PartFix > Show Fixed Parts* command to indicate all the fixed parts.

PartFix > Unfix By Partcode

Used to unfix parts by entering their part names.

The asterisk (*) can be used as a wild card to unfix a group of parts. For instance: * operates on all parts, IC* operates on all the parts with IC as a prefix, R3* operates on R3, R30 to 39, R300 to 399, etc. The questionmark (?) can be used to represent one number, for example R2? represents resistors R2, R20 - 29.

The *PartFix > Enable Flipped Parts* and *PartFix > Enable Unflipped Parts* settings control which parts can be included and take precedence.

Select *PartFix > Fix By Partcode*, a window appears in which the name of the part (or parts) to be unfixed should be entered. Select *OK* or press <enter>.

Use the *PartFix > Show Fixed Parts* command to indicate all the fixed parts.

PartFix > Enable Unflipped Parts

This setting can be active (ticked) or inactive (unticked). When active, all unflipped parts will be included in the *PartFix/Unfix* operations.

PartFix > Enable Flipped Parts

This setting can be active (ticked) or inactive (unticked). When ticked, all flipped parts will be included in the *PartFix/Unfix* operations.

PartFix > Show Fixed Parts

Used to highlight the fixed parts.

This command shows all fixed parts irrespective of the *PartFix > Enable Unflipped/Flipped Parts* settings.

Figure 198 shows how a part looks when unfixed on the left and fixed on the right - the area occupied by the component outline's autoplacement footprint changes colour.

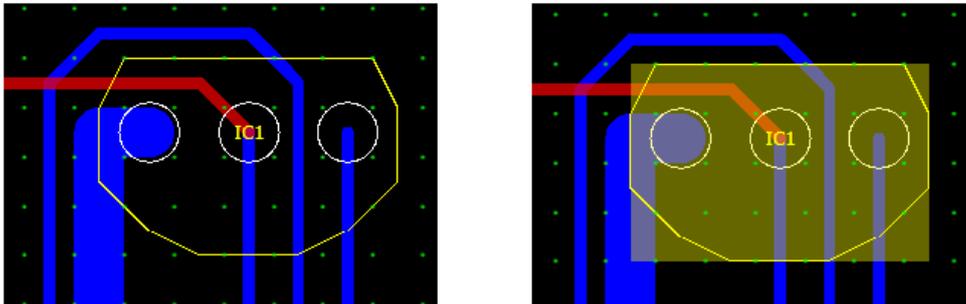


Figure 198

Whenever any of the *PartFix* commands are selected, this setting is enabled (ticked).

Mroute commands

Once all the parts have been placed on the board, the inter-connections are ready to be routed. They can either be auto-routed or manually routed. It is not recommended that tracks be routed until all the parts have been placed on the board.

The *Mroute* (manual routing) commands are used to manually route the connections. They are used to stretch the point to point connections between pads to form routed copper tracks on the board and to modify existing routed tracks. They do not allow pads to become disconnected.

(The *Amend* commands provide the flexibility for the user to "do what he/she wants" but should be used with caution by inexperienced users.)

Critical connections are usually manually routed first, fixed in position using the *NetFix* commands, then the remaining connections routed, probably by the auto-router.

The *Seetrix* auto-router is accessed via the *Tools* command in the artwork editor.

The interfaces to other available auto-routers are accessed whilst the artwork editor is closed, then by right-clicking on *Artwork* from the navigator and selecting *Optional Auto-routers*. (These other auto-routers have to be purchased separately and have to be enabled from the *File > System Setup* window before they become available for selection.)

The *Edit > Manual Routing Parameters* window (accessed from the *Configuration* folder) controls the default

direction/ordering for unroutes. But it is sometimes a requirement to change the order in which connections or routed tracks are routed. To change the order manually, use the *Mroute > Move Unroute/Track* commands. This window also controls the way in which connections can be manipulated, whether on-line DRC is switched on and its clearance requirements.

To start manually routing, the *Mroute > Corner* command is used, followed by *Mroute > Layer Swap/ Move Segment*, etc. as required.

When any of the Mroute commands are selected, the *Mroute* dialogue bar appears. It controls how the Mroute commands operate.

Mroute Dialogue Bar

When the Mroute commands are selected, the *Mroute* dialogue bar appears. Its settings are described now and they control the tracks that can be selected and how they are operated upon.

The dialogue bar will vary slightly depending on the number of layers in the design - two variants are shown in Figure 199. The (a) is from a double-sided design and (b) is from a design with 10 layers. The only difference is the number of layers listed.

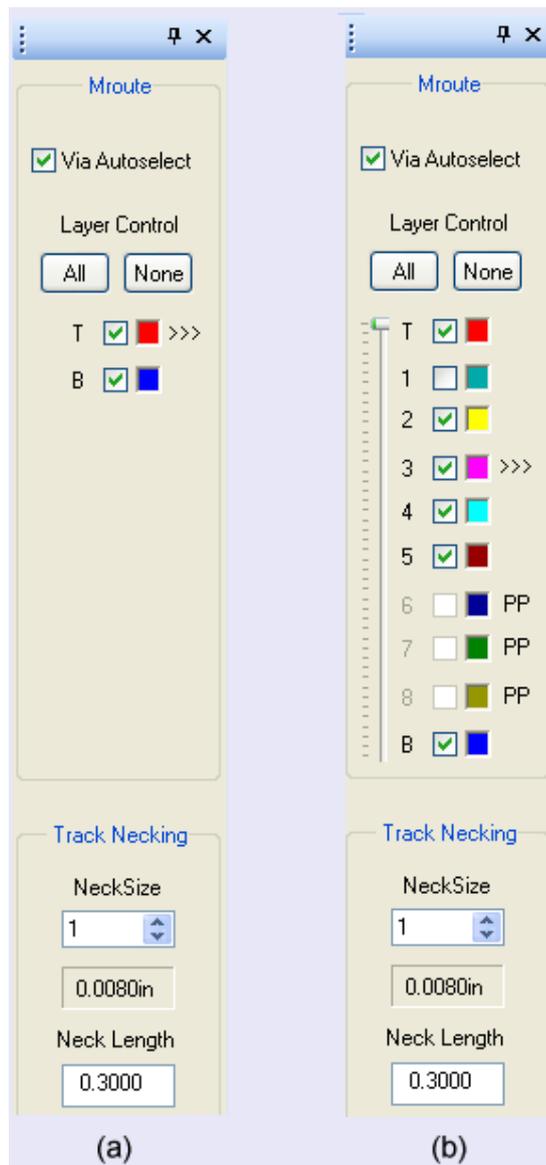


Figure 199

Via Autoselect When enabled (ticked) and a via has to be inserted (for example whilst performing a layer swap or point insertion operation) the most appropriate via from those available is used. The via selection is made from the user-defined via list (defined in the *Configuration* folder, *Via Hole Definitions* window).
The “most appropriate via” is one that traverses the minimum number of board layers whilst providing the correct connectivity.

	If 'Via Autoselect' is not checked, or autoselect cannot find a suitable user-defined via, then a standard via is inserted.
Layer Control	<p>This section of the dialogue lists all the copper and power/splitpower plane layers in the design. The layers are listed in their layer assembly order to help visualisation of the layer stack (as defined in the <i>Configuration</i> folder, <i>Layer Assignments and Ordering</i> window).</p> <p>When working on single or double designs, it's usual for all the other layers to be assigned as silk-screen layers, so only layers T & B will appear (Figure 199 (a)).</p> <p>When the list is too long to be displayed, the slider on the left of the layer list should be used to scroll up and down through the list when required.</p> <p>The checkboxes indicate the active tracking layers – if the box isn't ticked, then tracks can't be added to, or selected from, that layer.</p> <p>Power/split power plane layers are listed and identified by the letters <i>PP</i>. They are displayed for information purposes only, so they are "greyed out" and cannot be selected.</p> <p>When working on a double-sided board the T and B (the outer layers of the board) should be active.</p> <p>When working on single-sided boards, only T or B should be active, to avoid the use of the other layer.</p> <p>When working on multi-layer designs, the active layers can be varied depending on the layers being worked upon - disabling layers stops the inadvertent selection of tracks from those layers.</p> <p>The coloured boxes along-side the layers help identify track layers.</p> <p>Note: The <i>Mroute</i> commands will only select <i>visible</i> tracks (<i>View Control</i> dialogue bar) from the active layers.</p>
>>>	Whilst manually routing connections using <i>Mroute > Corner</i> , the new track can be added to any of the active layers, after pressing the user-defined layer swap key, providing the connectivity will be maintained. The chevrons (>>>) alongside the layer box indicates the layer the track will be added to.
All/None	These boxes can be selected to quickly select/deselect all the tick boxes.
Neck Size	<p>This setting defines the track thickness to be used when the <i>Mroute > Neck</i> command is used. The number represents a line thickness taken from the track sizes table. Use the spin-controls to change the size code.</p> <p>The current size assigned to the size code is shown underneath in inches or millimetres depending on the units in use (<i>Edit > Units</i>).</p>
Neck Length	<p>This setting defines the length of necked track introduced when using the <i>Mroute > Neck</i> command.</p> <p>The units in use (inches/mm) are controlled by the <i>Edit > Units</i> command.</p>

Mroute > Corner

This command is used to:

add or move corners in tracks

add corners to connections - as the corner is added, the connection is converted to a track or partially routed (one end becomes a track, the other remains as a connection, shown with a dotted line to differentiate it from connections that haven't been partially routed).

"lift off" the end-points of connections and move them to a pin, via or track corner on the same net, provided the *Mroute > Enable Unroute Reconnect* setting is checked.

This command will not allow tracks to become disconnected from component pins or vias.

Use the *Mroute > Move Track* command to change the order of routed tracks.

Use the *Mroute > Move Unroute* command to change the order of unroutes.

Use the *Mroute > Convert to Track* command to convert connections into tracks without adding corners.

Corners are *added* and then released with a click of the *right* hand mouse button. Corners are *moved* and then released with a click of the *left* hand mouse button. Clicking the opposite mouse button once a track has been selected cancels the operation.

right button = *add* corner

left button = *move* corner

(Opposite button cancels.)

Only tracks from a currently active layer in the *Mroute* dialogue bar can be selected.

Refer to the *GRID* command for details on whether grid snapping is active. The current grid is shown in the information line above the graphical editor.

If the *Mroute > Enable Unroute Reconnect* setting is checked, the end of a connection can be selected with a click of the left hand button, then the connection can be moved to a component pin, via or track corner on the same net with a further click of the left button.

To assist in finding alternate locations, especially when zoomed in on a large board, a small arrow points in a direction towards the next-nearest available attachment location. Click the right button to cancel the operation.

If the connection is moved to a corner or via, then subsequent selections of that point with *Mroute > Corner* will move the connection and the corner/via. To lift the connection off separately, hold down the <shift> key whilst clicking on the end of the connection with the left mouse button.

Adding a corner to a connection or a track

Parameters within the *Edit > Manual Routing Parameters* window (*Configuration* folder) affect whether the connection remains attached to the cursor (multi-node), whether it is routed completely or partially, etc. when adding corners to connections.

Select *Mroute > Corner*, ensure one of the active layer settings in the Mroute dialogue bar is set to the layer the track is on, or will be added to. Point at a connection, or a track from one of the active layers, and click the right hand mouse button. Move the cursor and the new corner.

If a connection was selected, the chevrons (>>>) in the Mroute dialogue bar indicate which layer the track segment will appear on when the corner is released.

If a connection was selected and the routing parameters are configured so that the connection is partially routed, then it is the shortest segment that gets converted to a track.

Locate the cursor in the position for the new corner and click the right hand mouse button again to release the corner.

Clicking the left hand mouse button before clicking the right hand button releases the connection or track without the new corner.

If the connection remains attached to the cursor (multi-node is selected), further corners can be introduced until the left hand mouse button is clicked, which releases the connection.

If the corner is being inserted into a track segment that is connected to a component pad and multi-node is selected, the track also remains attached to the cursor allowing more corners to be inserted until the left hand mouse button is clicked. This does not happen if a corner is inserted into a mid-segment.

Whilst the connection is attached to the cursor the <layer swap> special function key can be pressed to change the layer the track segment will appear on (toggles between the active layers as defined in the Mroute dialogue bar). This is provided the layer swap maintains connectivity, for example, a layer swap would not be permitted on the track segment entering a surface mounted pad, the track has to be on the appropriate layer.

Orthogonal (0/90) and 45 degree segment locking can be switched on and off using the <Toggle track node lock mode> user-defined special function key (defined in the *File > System Setup* window, *Special Function Keys, Setup* button). Each time the key is pressed, the mode toggles between free movement and 0, 45 & 90 degree fixed movement.

The locking facility when activated comes into effect after the first corner has been inserted in a connection. If adding corners to an existing track, the facility is only available when the track segment closest to a pad or via is selected.

Moving an existing corner in a track

Select *Mroute > Corner*, ensure one of the active layers in the Mroute dialogue bar is set to the layer the track is on. Point at an existing corner from one of the active layers, and click the left hand mouse button. Move the cursor and the corner stretches with it. Locate the cursor in the new position for the corner and click the left hand mouse button again to release the corner.

Clicking the right hand mouse button before clicking the left hand button returns the corner to its original position.

Connections to power planes from SMD pins (assuming the power planes have been defined)

Surface mounted pads that are connected to power planes have a connection complete with via attached to them automatically. The connection sits on top of the SMD pad so looks like a "blob" when the power unroutes/connections are visible at width (*View Control* dialogue bar), until it is positioned.

It can be moved into an appropriate position using the *Mroute > Corner* (adding) command. (If the power plane layers have not been defined, the connections will appear as normal connections.)

If the auto-router will not be used to insert the stubs required to make the connection between the SMD pad and plane, then proceed as follows.

Zoom into a surface mounted pad that is connected to a power plane. Ensure the power connections (unroutes) are visible and displayed at width. A connection "blob" will be seen over the SMD pad. Select it with a click of the right hand mouse button. Move the cursor with the attached connection and via into a space and click the right hand mouse button again. The connection is converted into a track with a via on the end of it. This track and via will make the connection between the SMD pad and the power plane.

Where there are multiple blobs in close proximity, the vias can be released directly over the top of one another in order to save on the number of vias - the subsequent vias are automatically removed as the blob/stub is released so that there is only one via at the location.

Via selection (standard or user-defined) is controlled from the Mroute dialogue bar (*Via Autoselect*).

Implications of the routing parameters when adding corners (Mroute > Corner) to connections

The routing parameters are found in the *Edit > Manual Routing Parameters* window, accessed from the *Configuration* folder.

If the *Corner Insertion* setting is set to *Multinode*, the connection remains attached to the cursor until the left hand mouse button is clicked. Whether the last segment is left as a partial unroute is determined by the *Unroute Convert* and *Unroute Minimum Length* settings.

Whilst the connection remains attached to the cursor, the layer swap special function key can be used to select the layer the next track segment will be released on. The layers used are controlled by the Mroute dialogue bar.

If the *Corner Insertion* setting is set to *Select/Drop* the layer that the track appears on is controlled as follows:

If the corner is inserted into a connection between two surface mounted devices that are on the same side of the board, the track will appear on that layer.

If a corner is inserted into a connection between two surface mounted devices that are on opposite sides of the board, a prompt will appear in the status bar requesting a position for the via hole that will maintain the connectivity between the two parts. Move the cursor into the position where the via hole is required and click the right hand mouse button.

If the corner is added to a connection between through plated pads and the connection is more horizontal than vertical, then a scan is done from the top of the board working downwards through the copper layers in assembly order until a layer is found that is enabled for routing.

If the unroute is more vertical than horizontal, then a scan is done from the bottom of the board working upwards through the copper layers in assembly order until a layer is found that is enabled for routing.

If the *Corner Insertion* setting is set to *Multinode* only the shortest segment of the connection is converted into a track, the rest of the connection remains attached to the cursor, and the layer it is added to is controlled by pressing the user defined layer swap key - the chevrons (>>>) in the dialogue bar indicate which layer the track segment will be added to.

The track can be added to any of the active layers from the Mroute dialogue bar providing the connectivity will be maintained. Vias are inserted as required when layer swapping is activated.

Mroute > Enable Unroute Reconnect

This setting determines whether connection ends can be selected for movement when *Mroute > Corner* is in use.

With this setting enabled it can sometimes be difficult to move a corner that is very close to the end of a connection, as the end point of the connection takes precedence.

Disabling this setting will not allow connection end-points to be selected/moved when moving corners using *Mroute > Corner*.

Mroute > Move Segment

Used to move a complete track segment whilst stretching the adjoining track segments.

The movement of horizontal and vertical track segments is controlled by the *Segmove Mode* setting in the *Manual Routing Parameters* window (*Configuration* folder).

Track segments that end on a component pad cannot be moved, as one end of the segment is fixed.

If on-line DRC is switched on, errors will be flagged if a violation occurs when the segment is moved.

Select *Mroute > Move Segment*. Ensure one of the layer settings in the Mroute dialogue bar is set to the layer that the segment is on. Select the track segment. Move the cursor. The track segment moves and stretches the adjoining segments. Click the left hand mouse button again to release the segment in its new position.

Clicking the right hand mouse button after the segment has been selected returns the segment to its original position.

Mroute > Layer Swap

Used to transfer a track segment from one layer of the board to another, inserting and/or removing vias as necessary.

The track segment to be selected must lie on one of the active layers as defined in the Mroute dialogue bar. It will be swapped to the next active layer, in the Mroute dialogue bar, from top to bottom, then back to top. (The layer order is defined in the *Configuration* folder, *Layer Assignments and Ordering* window.)

For example, if layers T, 1, 2, 5 and B were active and listed in that order, then a track segment that lies on layer 2 will be swapped to layer 5, then B, then T, then 1 and back to 2 and so on, each time it is selected.

A track segment is a length of track between two points, not necessarily the complete run of a straight line. The last track segment attached to a SMD pin cannot be swapped otherwise connectivity would be lost. On-line DRC does not flag violations caused by this command.

Select *Mroute > Layer Swap*. Ensure the layer that the track segment is on, along with the layer the segment is to be swapped to, are active in the Mroute dialogue bar. Select the track segment, the segment swaps between two layers adding or removing vias as necessary. It may be necessary to swap the segment again until it lies on the layer required if more than two layers are active.

Mroute > Delete Point

Used to remove points (or corners) from tracks.

This command will not remove component pads, vias, track ends that meet at component pads or vias, or the end points of tracks. Use *Mroute > Ripup* to delete a track and restore the connection.

If on-line DRC is switched on, errors will be flagged if a violation occurs when the point is deleted.

Select *Mroute > Delete Point*. Ensure the layer is active in the Mroute dialogue bar. Select the point and it is removed.

Mroute > Convert Arc

Used to convert straight line track segments into curved tracks, curved tracks into straight line segments, or to change the shape of existing arcs.

On-line DRC does not flag violations caused by this command.

Ensure the layer the track is on, is active in the Mroute dialogue bar.

Straight lines into arcs

Select *Mroute > Convert Arc*, then select a track segment. As the cursor is moved, the segment is replaced with an arc that stretches with the cursor. Click the left hand mouse button to release the arc in its current position.

Arcs into straight lines

Select *Mroute > Convert Arc*, then select a curved track segment with a click of the left hand mouse button. Follow this by a click of the right hand mouse button, and the arc is replaced with a straight-line track segment.

Changing the shape of existing arcs

Select *Mroute > Convert Arc*, then select a curved track segment. Move the cursor with the arc attached and click the left hand mouse button to release it in its new position.

Note: once an arc or segment has been selected and released, clicking the right hand mouse button converts that segment between an arc and a straight line until a new segment is selected with the left hand mouse button. The arc will always be the same size, but its direction is cursor position dependent. This facility is useful to convert 45 degree track segments into quadrants of a circle.

Mroute > Fix 45

Used to force non-45 degree angled track segments entering component pads to enter at 45 degrees.

This command is often used when trying to enter pads at 45 degrees that are not on a multiple of the current snap grid.

The point\corner before the pad is moved so that the track segment enters the pad at 45 degrees.

Fix 45 will not work if the other end of the track segment is attached to a non-horizontal or non-vertical track segment, a curved track segment or a via.

Select *Mroute > Fix 45*. Ensure the layer the track segment is on, is active in the Mroute dialogue bar. Select a non-45 degree track segment entering a pad. The segment forms a 45 degree angle.

Mroute > Fix 90

Used to force non-90 degree angled track segments entering component pads to enter at 90 degrees.

This command is often used when trying to enter pads at 90 degrees that are not on a multiple of the current snap grid.

The point\corner before the pad is moved so that the track segment enters the pad at 90 degrees.

Fix 90 will not work if the other end of the track segment is attached to a non-horizontal or non-vertical track segment, a curved track segment or a via.

Select *Mroute > Fix 90*. Ensure the layer the track segment is on, is active in the Mroute dialogue bar. Select a non-90 degree track segment entering a pad. The segment forms a 90 degree angle.

Mroute > Neck

Used to change the thickness of individual sections of tracks. This is typically used to route a thick track

between the pins of ICs.

A neck length and neck size has to be specified in the Mroute dialogue bar. The neck length defines the length of the section of track that has to be changed. The neck size defines the thickness of the necked section of track.

On-line DRC does not flag violations caused by this command.

Ensure the layer the track segment is on, is active in the Mroute dialogue bar, and the neck size and length is set as required.

Select *Mroute > Neck*. Select the centre of the required necked section of track. The necked track is introduced centrally about the cursor position.

Removing necked sections of track

- * Use "Undo" when appropriate
- * Neck the segment back to its original size.
- * Layer swap the necked track segment until it returns to the original layer. This method only works if there is a track segment attached to the necked section that remains at its original size.

The last two methods leave points in the tracks at the locations where the neck started and stopped. If layer swap was used, they can be removed using the *Mroute > Delete Point* command.

Mroute > Convert to Track

Used to convert a connection into a track segment on a copper layer of the board, without introducing corners. (Use *Mroute > Corner* instead if corners are also required.)

The layer chosen for the track is selected as follows:

If the corner is added to a connection between through plated pads and the connection is more horizontal than vertical, then a scan is done from the top of the board working downwards through the copper layers in assembly order until a layer is found that is enabled for routing.

If the unroute is more vertical than horizontal, then a scan is done from the bottom of the board working upwards through the copper layers in assembly order until a layer is found that is enabled for routing.

If a connection between two surface mounted devices that are on the same side of the board is converted, the track appears on that layer.

If a corner is inserted into a connection between two surface mounted devices that are on opposite sides of the board, a prompt appears in the status bar, requesting a position for the via hole that will maintain the connectivity between the two parts.

Select *Mroute > Convert to Track*, then select a connection. It appears as a track.

On-line DRC does not flag violations caused by this command.

Refer to the *Mroute > Ripup* command for details on converting tracks back into connections.

Mroute > Ripup

Used to convert a track segment attached to a pad, or the complete track, back to a connection or "unroute". (Only unfixed track segments can be selected – use the *NetFix > Unfix Mode* and associated commands to unfix, fixed tracks.)

An unrouted connection is shown as a solid line between two pads. A partially unrouted connection is shown as a dashed line between a pad and the end of the routed section.

Only track segments attached to pads or partial unroutes can be partially ripped-up. Track segments between track segments cannot be partially ripped up.

Once tracks have been ripped-up, they can be manually routed or auto-routed (complete unroutes).

If a stringer is ripped up, the connection remains "attached" to the position where the stringer's via was, and not to the pin of the component. Either use the *Parts > Flip* command twice on the part to restore the stringers, or use the *Parts > Reconnect Power* command to reconnect the connections to the part pins.

Ensure the layer the track/track segment is on, is active in the Mroute dialogue bar. When ripping up a complete track, only the selected track segment layer needs to be active, the attached segments will be ripped up even if the layers they are on, are not active or visible.

Ripping up complete tracks

Select *Mroute > Ripup*, then select any (active layer) track segment with a click of the *right* hand mouse button. The complete track is replaced by a connection.

Ripping up track segments

Select *Mroute > Ripup*, then point at a track segment attached to a component pad or a partial unroute and click the *left* hand mouse button. The track segment is replaced by a partially unrouted connection.

Selecting the next track segment in the same way replaces it with a partially unrouted connection line. The layer the segment is on, has to be active.

Note: if a track is deleted using the *Amend* or *Region* commands, its connection will not re-appear. It will be necessary to run the artwork checking routines to restore the "lost" connections.

Mroute > Move Track

Used to change the order of routed tracks belonging to one net, whilst maintaining connectivity. Often used to route power supplies in the order required. This command does not operate on unroutes.

For example, take a net that joins pins A, B and C together, in that order (Figure 200, before). It may be preferable for the tracks to be attached in the order A to C and A to B (Figure 200, after).

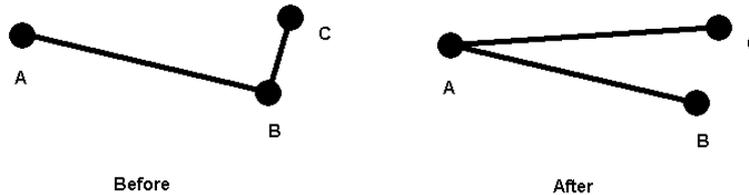


Figure 200

The connectivity order has changed, not change the connectivity.

On-line DRC does not flag violations caused by this command, though it is not possible to move the track to a pin on a different net.

Ensure the layer the track is on, is active in the Mroute dialogue bar.

Select *Mroute > Move Track*. Select a *component pad* belonging to the net you wish to modify, with one or more tracks attached to it. Component pads on all layers, or the top of the board should be selected with the left hand mouse button. Component pads on the bottom of the board should be selected with the right hand mouse button. As the cursor is moved, the track is disconnected from the selected pad and moves with the cursor. All the pads on the same net highlight.

If more than one track is attached to the selected pad, and the wrong track is attached to the cursor, move the cursor back over the original component pad and select it again. Keep selecting the pad until the correct track is attached to the cursor. (Only tracks from active layers are selected.)

Now move the cursor and the attached track, over the highlighted component pad that you would like it to be attached to, and click the same mouse button. The track moves. If a pad would become disconnected by this operation, a connection appears to maintain the complete net.

The track is reconnected, providing the new component pad is on all layers, or on the same side of the board as the selected track. For example, moving a track from a top mounted SMD to a bottom mounted SMD would not be allowed as the track would become disconnected.

To cancel the operation whilst the track is attached to the cursor, click the opposite mouse button.

Mroute > Move Unroute

Used to move the end of a connection, to a component pin, via or corner of a track on the same net in order to reduce the length of connection to be routed, or alter the order of connectivity (circuit connectivity cannot be altered with this command).

Select *Mroute > Move Unroute*, select the end of a connection and if connectivity permits, the connection will be detached from the current location. Whilst moving the connection, a dotted line extends from the cursor to the nearest available node where the connection end may be attached. To assist in finding alternate locations, especially when zoomed in on a large board, a small arrow points in a direction towards the next-nearest available attachment location. Once an alternative connection point has been located (another part pin, via hole or track corner on the same net), select it with a click of the left-hand mouse button. Click the right button to cancel the operation.

Where multiple connections are attached to the initial point being selected, select the required connection slightly away from its end point to be sure of selecting the connection required.

Amend commands

These commands provide the freedom to change and add items to the layout as required. Because of this freedom, it is possible to completely change the layout, for better or worse. It is therefore unlike the *Mroute* commands, which maintain connectivity at all times. If connectivity is lost, or short circuits or clearance errors are added whilst using the *Amend* commands, these problems will be highlighted when the *artwork checking* routine is performed.

Typically, things like crop marks, plating bars, etc. are added using these commands. Copper filled areas can also be modified using the *Amend > Move Point/Delete Track* commands.

The board profile and keepout lines can also be modified or added using the *Amend* commands. Any changes to these two categories of data are automatically back annotated to the board profile editor. This saves having to

return to the profile editor in order to re-define keep-out areas between auto-routing strategies, etc.

The *Amend* commands can also be used to modify data on the silk screen layers, and data that has been added or imported onto a documentation layer

Any errors that are introduced to the copper layers of the layout will be flagged when the artwork checking routines are performed.

When any of the *Amend* commands are selected, the *Enter/Amend* dialogue bar appears and it controls how the Amend commands operate.

Amend Dialogue Bar

When the Amend commands are selected, depending on which command is selected, one of the *Amend* dialogue bars appears as shown in Figure 201. The bar is either configured for pads or tracks. The settings are described below.

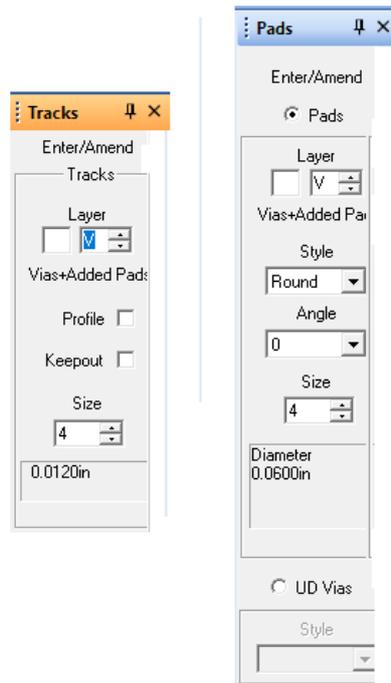


Figure 201

Settings common to both dialogue bars:

Layer selection this determines which layer tracks or pads are added to, or selected from. It is the "active" layer. All layers are available for selection.

The *Amend* commands will only select tracks/pads from the layer indicated by the active layer setting.

If the active layer has been assigned a title (Configuration > Layer Assignments window), this is also displayed.

The colour of the box along-side (white is a colour) helps to identify and also select layers; it can be selected to produce the layer assignments list which aids layer selection when using the spin-controls to change layers might be cumbersome.

Pads that are added to layer V will be drilled (if they have a drill hole defined). Pads that are added to any other layer, i.e. layer T, B, 1, 2, etc. will not be drilled.

Size selection this setting corresponds to the size codes from the Track/Pad Sizes table and it controls the size of any tracks/lines or pads that are added/updated. The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside.

The box underneath shows the actual size assigned to each code as it is selected, in whichever units are active.

(Profile and keepout lines have no thickness so this setting is not used when adding/modifying these line types.)

Track dialogue bar

Profile tick this box to edit or add the board profile. Only one polygon line should form the profile - refer to the board profile editor for more details regarding the board profile definition.

Keepout tick this box when you wish to edit or add to the keepout definitions - refer to the board profile editor for more details regarding keepouts.

Pad dialogue bar

When this dialogue bar appears, either "standard pads" (including user-defined pads and heat-relief/anti-pads) or user-defined (UD) via stacks can be added or updated. The radio buttons alongside *Pads* or *UD Vias* determines which type of pad are active. Depending on which types are selected, parts of the dialogue bar will be greyed out.

<i>Pads</i>	if this button is selected, then the following settings are enabled and can be used to add/modify or delete pads. (Component pads cannot be deleted from within the artwork editor.)
<i>Style</i>	allows the pad shape to be selected - select the arrow alongside to produce a list of pad styles from which one should be selected. Included are the standard pad shapes, user-defined pads from the design library, heat-relief and anti-pads. If a standard pad shape or heat-relief/anti-pad is selected, then the <i>Size</i> setting should also be updated as required. Whichever pad style is chosen the <i>Angle</i> setting should also be set as required.
<i>Angle</i>	setting determines the orientation (angle) of the selected pad. The angle can be typed in (0, 90, 180 or 270), or a selection made from the list that appears when the arrow alongside the setting is selected.
<i>UD Vias</i>	If the user-defined vias (UD Vias) are selected, then the size of the vias that are added and the layers they appear on are controlled by the via style selected, the other settings in the dialogue bar are greyed out.
<i>Style</i>	allows the user-defined via to be selected - select the arrow alongside to produce a list of via styles from which one should be selected. Included are all the via stacks defined in the <i>Configuration</i> folder, <i>Via Hole Definitions</i> .

Amend > Enter Pads

Used to add a pad to the board. These pads are not associated with components.

The pad's size, shape, layer and whether drilled is defined in the Amend Pads dialogue bar.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Enter Pads*, ensure the Amend Pads dialogue bar is set as required, then move the cursor and the attached pad into position and click the left hand mouse button to release it. Another pad appears attached to the cursor until the right-hand button is clicked.

Amend > Enter Tracks

Used to add a track or a silk-screen line to the board, on the layer and at the size specified by the Amend dialogue bar, or to add a line to the board profile or keepout definition.

Tracks that are added using this command, will not be recognised as belonging to a connection until they have been validated by the artwork checking routines. This means that routines such as *Identify*, *Highlight* and *copper Fill* will not recognise them as valid nets or tracks until that time.

Note: if a track is accidentally deleted, the checking routines will re-introduce the connection again and flag an error unless it has been successfully reinserted using the *Amend* commands. Provided the new track completes a correct connection, the checking routines will validate it and it will be recognised as a valid track.

Adding a piece of track in this way does not add a connection to the wiring list. Refer to the *Tools > Network > Add Link* command to achieve this, or add a connection to the schematic and recompile the parts and wiring list.

If the piece of track causes a short or clearance errors on a copper layer, these will be reported when artwork checking is performed.

If the line is added to the profile or keepout categories, the profile editor is automatically updated.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Enter Tracks*, ensure the Amend Tracks dialogue bar is set as required, then point at the location where the track/line is to start and click the left hand mouse button. The line stretches as the cursor is moved. Click the left-hand mouse button to insert corners in the track. Release the track by clicking the right hand mouse button.

Amend > Enter Arc Tracks

Used to add a piece of curved track or line to the board, on the layer and at the size specified by the Amend dialogue bar, or to add a curved line to the board profile or keepout definition.

The *Mroute > Convert Arc* command can be used to convert existing track segments to arcs.

Tracks that are added using this command will not be recognised as belonging to a connection until they have

been validated by the artwork checking routines. This means that routines such as *Identify*, *Highlight* and *Copper Fill* will not recognise them as valid nets or tracks until that time.

Note: if a track is accidentally deleted, the checking routines will re-introduce the connection again and flag an error unless it has been successfully reinserted using the *Amend* commands. Provided the new track completes a correct connection, the checking routines will validate it and it will be recognised as a valid track.

Adding a piece of track in this way does not add a connection to the wiring list. Refer to the *Tools > Network > Add Link* command to achieve this, or add a connection to the schematic and recompile the parts and wiring list.

If the piece of track causes a short or clearance errors on a copper layer, these will be reported when artwork checking is performed.

If the profile or keepout categories are modified, the profile editor is automatically updated.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Enter Arc Tracks*, ensure the Amend Tracks dialogue bar is set as required, then point at the location where the track/line is to start and click the left hand mouse button. Move the cursor and the line that is attached to it, into the position where the arc should finish and click the left hand mouse button again. A straight segment is produced that bends as the cursor is moved. The size and shape of the arc is dependent on the position of the cursor. Click the left hand mouse button to release the curve in the position chosen. The track remains attached to the cursor allowing a series of arcs to be added. Once the arc has been released, clicking the right hand mouse button releases the track from the end of the cursor.

If the right hand mouse button is clicked whilst the curve is being stretched, a straight line segment is introduced, thus allowing a mixture of curved and straight tracks to be added. The arcs can be subsequently modified with the *Amend > Corner/Adjust Arc* commands.

Amend > Enter Circle

Used to add a circle on to the profile or keepout layers of the artwork. This command will not add a circle on any other layer.

Select *Amend > Enter Circle*, tick either the *Profile* or *Keepout* category in the *Amend Tracks* dialogue bar (the layer and size settings have no effect with this command), then point at the centre of the required circle and click the left hand mouse button. As the cursor is moved a circle stretches, click the left-hand mouse button to release the circle.

Use *Amend > Move Point* to move or adjust the size of the circle.

Amend > Corner

Used to add and move existing corners in tracks, lines or arcs.

This command is typically used to modify individual silk screen outlines, documentation data, keepout lines, etc. Note, that tracks and pads can be disconnected using this command. Any errors will be highlighted by the artwork checking routine.

(Use the *Mroute > Move Track* to change the connection order of tracks as it will not let you make mistakes.)

Corners are *added* and then released with a click of the *right* hand mouse button. Corners are *moved* and then released with a click of the *left* hand mouse button. Clicking the opposite mouse button once a track or line has been selected cancels the operation.

right button = add corner

left button = move corner

(Opposite button cancels.)

Only tracks or lines from the currently active layer as specified by the Amend dialogue bar can be selected.

If the profile or keepout categories are modified, the profile editor is automatically updated.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Adding a corner to a track, line or arc

Select *Amend > Corner*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the track/line is on, then point at the line and click the right hand mouse button. Move the cursor and the new corner. Locate the cursor in the position for the new corner and click the right hand mouse button again to release the corner.

Clicking the left hand mouse button before clicking the right hand button releases the track or line without the new corner.

Moving an existing corner in a track, line or arc

Note: this move corner command does not perform in the same way as the *Mroute > Corner* command, as it does allow the ends of tracks to be disconnected from pads or vias.

Select *Amend > Corner*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the track/line is on, then point at an existing corner from the layer, and click the left hand mouse button. Move the cursor and the corner

stretches with it. Locate the cursor in the new position for the corner and click the left hand mouse button again to release the corner.

Note: if a track end is being moved and it is released within the capture distance of a layer 0 pad, a pad on the selected layer or any part pins, then it will snap to the centre of that pad. If the end-point is not near a pad or part pin, then it will snap to the nearest half or whole grid point if grid snapping is switched on. (Track mid-points do not snap to pads or part pins when released.)

Clicking the right hand mouse button before clicking the left hand button returns the corner to its original position.

Amend > Move Point

Used to change the size or position of circles. This command can also be used to move an existing point (corner) in a piece of track or line, or to disconnect a piece of track from a pad or via, but the *Corner* command is more flexible for these purposes.

If the "circle" has been defined using arcs, then it cannot be moved, the endpoints of the arcs can be moved to adjust its shape though.

(Use the *Mroute > Move Track* to change the connection order of tracks as it will not let you make mistakes.)

Any errors introduced will be flagged during the checking routines.

If the profile or keepout categories are modified, the profile editor is automatically updated.

Only tracks or lines from the currently active layer as specified by the Amend dialogue bar can be selected.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Moving a circle

Select *Amend > Move Point*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the circle is on, then move the cursor over the centre of the circle and click the left hand mouse key. Move the cursor with the circle attached into the required position and click the left hand mouse button again to release it.

Clicking the right hand mouse button whilst the circle is attached to the cursor will return it to its original position.

Changing the size of a circle

Select *Amend > Move Point*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the circle is on, then point at the circumference of the circle and click the left hand mouse key. As the cursor is moved, the circle's size changes. Click the left hand mouse button again to release the circle at its current size.

Clicking the right hand mouse button whilst circle is attached to the cursor returns it to its original size.

Moving points in tracks, lines or arcs

Select *Amend > Move Point*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the line is on, then move the cursor over the point you want to move and click the left hand mouse key. Move the cursor with the corner attached into the required position and click the left hand mouse button again to release it.

Note: if a track end is being moved and it is released within the capture distance of a layer 0 pad, a pad on the selected layer or any part pins, then it will snap to the centre of that pad. If the end-point is not near a pad or part pin, then it will snap to the nearest half or whole grid point. (Track mid-points do not snap to pads or part pins when released.)

Clicking the right hand mouse button whilst the corner is attached to the cursor returns it to its original position.

Amend > Delete Point

Used to remove points (or corners) from tracks, lines or arcs.

This command will remove the end points of tracks, unlike the *Mroute > Delete Point* command.

If a piece of track consists simply of two points, the start and end points, and one of the points is deleted, the track is removed. The artwork checking routines will flag an error if this means a valid track has been removed.

If the profile or keepout categories are modified, the profile editor is automatically updated.

Only tracks or lines from the currently active layer as specified by the Enter/Amend dialogue bar can be selected.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Delete Point*, ensure the Amend Tracks dialogue bar is set to the layer the track/line is on, then point at a point (corner) in a track and click the left hand mouse button. The point is removed.

Amend > Adjust Arc

Used to convert straight segments into curves, curves into straight segments, and to change the shape of existing curves.

If the profile or keepout categories are modified, the profile editor is automatically updated.

Only tracks or lines from the currently active layer as specified by the Enter/Amend dialogue bar can be

selected.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Straight lines into arcs

Select *Amend > Adjust Arc*, point at a track segment and click the left hand mouse button. Move the cursor, the segment is replaced with an arc on the end of the cursor. Once the arc is in the required position, click the left hand mouse button to release it.

Arcs into straight line segments

Select *Amend > Adjust Arc*, point at the arc and select it with a click of the left hand mouse button. Follow this with a click on the right hand mouse button to convert it to a straight line segment.

Changing the shape of an arc

Select *Amend > Adjust Arc*, point at the arc and select it with a click the left hand mouse button. Move the cursor with the arc attached and click the left hand mouse button when the arc is in the desired position.

Clicking the right hand mouse button toggles the last selected line or arc between a line and an arc. The shape of the arc is defined automatically, and can be used to convert 45 degree lines into quadrants of a circle. The direction of the arc is determined by the position of the cursor about the line when the button is pressed.

Amend > Delete Pad

Used to delete pads from the layout.

Component pads cannot be deleted in the artwork editor, as they are defined as part of the component outline. (They can be replaced with a different pad, using the *Amend > Replace Pad* command.)

Only pads from the currently active layer as specified by the Enter/Amend dialogue bar can be selected.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Delete Pad*, ensure the Enter/Amend Pads dialogue bar is set to the layer the pad is on, then point at the datum of the pad to be deleted and click the left hand mouse button. The pad is removed.

Amend > Delete Track

Used to delete a track, line, arc or circle from the board (tracks/silkscreen), or the profile or keepout lines. A complete length of track is deleted, i.e. between pads or vias.

To restore the track to a connection, use the *Mroute > Ripup* command.

Deleting a piece of track in this way does not remove the connection from the wiring list. Refer to the *Tools > Network > Delete Pin* command, or delete the connection from the schematic and recompile the parts and wiring list.

If a track is deleted, the point to point connection is not restored until the artwork checking routines are performed.

If the profile or keepout categories are deleted, the profile editor is automatically updated.

Only tracks/lines from the currently active layer as specified by the Amend dialogue bar can be selected.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Delete Track*, ensure the Enter/Amend Tracks dialogue bar is set to the layer the track/line is on, then point at the track, line, arc or circle to be deleted and click the left hand mouse button. The track is removed.

Amend > Move Pad

Used to move existing pads. Component pads cannot be moved in the artwork editor, as they are defined as part of the component outline.

If via pads are moved, the tracks do not move with the pad so become disconnected. Use the *Mroute > Corner* command to maintain connectivity.

Only pads from the currently active layer as specified by the Enter/Amend dialogue bar can be moved.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Move Pad*, ensure the Enter/Amend Pads dialogue bar is set to the layer the pad is on. Point at the pad's datum and click the left hand mouse button. The pad moves with the cursor. Move the pad into position and click the left hand mouse button again to release it.

Clicking the right hand mouse button whilst the pad is attached to the cursor returns it to its original position.

Amend > Rotate Pad

Used to rotate existing pads in 90 degree increments anti-clockwise.

Component pads cannot be rotated – instead use the *Amend > Replace Pad* command and replace the pad with a rotated pad of the required size and shape.

Only pads from the currently active layer as specified by the Enter/Amend dialogue bar can be moved.

Select *Amend > Rotate Pad*, ensure the Enter/Amend Pads dialogue bar is set to the layer the pad is on. Point at the datum of the pad to be rotated and click the left hand mouse button. The pad rotates 90 degrees anti-clockwise each time the mouse button is clicked.

Amend > Replace Pad

Used to replace an existing pad for the currently active pad.

Pin numbers are maintained if component pads are replaced.

The datum of the new pad is positioned directly over the datum of the pad it is replacing.

If the layer setting in the pads dialogue bar is set to V and a component pad is selected, the complete pad stack is replaced. If it is set to layer T or B, just the pads on those layers will be replaced (these are the top and bottom pads of the pad stack). If any of the other layers is active, then all the inner pads are replaced.

Via pads are on layer V and the complete stack is replaced. Note: the *Mroute > Corner* command recognises vias as being a code zero round or square pad. If a different size or shape via pad is used, the *Corner* command will not operate on that via.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Amend > Replace Pad*, ensure the Enter/Amend Pads dialogue bar is set to the layer the pad is on and the new angle required if appropriate. Point at the datum of the pad to be replaced and click the left hand mouse button. The pad is replaced with the active pad. More pads can be replaced as required.

Note: if pads within outlines are changed using this command in the artwork editor, any subsequent changes to the pad size/shape in the outline library part, will not be implemented on the outline in the artwork editor which has had its pads amended.

Amend > Set Size

Used to change the size code of a track. A complete length of track is changed, i.e. between pads or vias or end points.

Note: this command cannot be used on *Profile* or *Keepout* lines which have no size.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Only tracks/lines from the currently active layer as specified by the Amend dialogue bar can be selected.

Select *Amend > Set Size*, ensure the Amend Tracks dialogue bar is set as required. Point at the piece of track to be changed and click the left-hand mouse button.

Note: changing the size code may not change the width of the track as the width is defined in the track sizes table.

Text commands

Text can be added at any time to the board. If it is added to a copper layer, it is included in the artwork checking routine (copper) to ensure it does not cause clearance errors or short circuits.

If it is added to a power plane layer, it is "not" copper, so ensure it does not overlap heat-relief pads to isolate them from the plane. If it is added to a silk screen or documentation layer, it is not checked in the artwork copper checking routine, as it is assumed that the layer will be manufactured with a non-conducting material.

Separate silk-screen checking is available within the artwork checking routine to identify text over holes, underneath parts, etc.

Silk screen names (labels) can be generated automatically using the *Tools > Generate SilkScreen* command. Once generated, the names can be manipulated as required using the *Text* commands.

When the Text commands are selected, the *Text* dialogue bar appears down the left-hand side of the screen as shown in Figure 202.

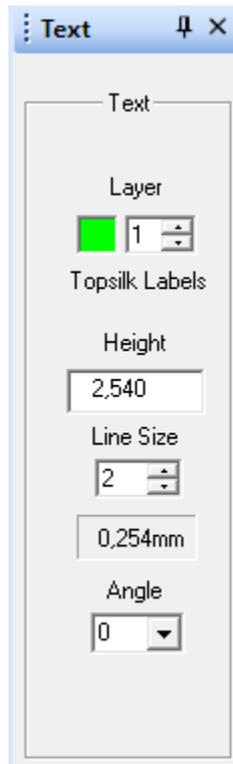


Figure 202

This dialogue bar should be adjusted to suit the text being added or modified as follows:

Layer This determines which layer text is added to, and which layer text can be selected from. It is the "active" layer. All layers are available for selection.

The *Text* commands will only select text from the active layer.

If the active layer has been assigned a title (Configuration > Layer Assignments window), this is also displayed.

The colour of the box helps to identify and also select layers; it can be selected to produce the layer assignments list which aids layer selection when using the spin-controls to change layers might be cumbersome.

Height this setting controls the height of new text strings or those being modified.

Line Size this setting corresponds to the size codes from the Track Sizes table and it controls the thickness of line used to draw text characters when adding or changing text strings. The code can be changed by typing in a number (the cursor must remain in the dialogue bar whilst typing) or by selecting the spin controls alongside.

The box underneath shows the actual size assigned to each code as it is selected.

When adding tiny characters, a thin line size should be used, otherwise the characters will appear as blobs. Larger characters can be drawn with thicker line sizes.

Rotation this setting determines the orientation of text strings that are added. If a text string is moved the setting is updated to correspond to the orientation of the selected text string.

Text > Add

Used to add text strings to the artwork. Text strings may be added to any layer - see the introduction above for details.

Text may be added outside the board profile for informational purposes if required.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Ensure the Text dialogue bar is set correctly for the text string you wish to add.

Select *Text > Add*, a window appears.

If text has already been added or edited, the old text string appears in the window and may be selected and modified.

If not, type in the text string required and tick the "Insert mirrored" checkbox if mirrored text is required.

Select *Place*, the window closes and the text string appears on the end of the cursor, represented by a rectangle. As the string is moved, it may be rotated by pressing the user defined special function key for <rotate

a part>.

Move the cursor into position and click the left hand mouse button to release it. Clicking the right hand button before the text is released restores the window allowing the string to be changed or cancelled.

As the text is released, another copy of the text string appears on the end of the cursor and can be released in the same way or the right hand mouse button clicked to restore the window allowing the string to be changed or text addition cancelled.

Text > Move

Used to move text strings, including silk screen labels and URL anchors that have been placed on the board. The lower left corner of the text string is its datum point, though it can be selected anywhere along its length.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string/URL symbol resides. Select *Text > Move*, then select the text string/URL symbol you wish to move. If a silkscreen label is selected, a line extends to its associated part. Text strings may be rotated by pressing the user defined special function key for <rotate a part>. Click the left hand mouse button to release the text/symbol in its new position. Clicking the right hand mouse button returns the text to its original position.

When the text string is selected, the rotation setting in the text dialogue bar is updated to show the current angle of the text string.

Text > Rotate

Used to rotate text strings through 90 degree increments, anti-clockwise about the lower left hand corner of the string. For instance, text that has been rotated to 45 degrees will rotate to 135, 225, 315 and back to 45 degrees. (Text can be rotated in increments of 1 degree once it is has been placed on the board using the *Text > Edit* command.)

Text at 0 degrees is placed horizontally and read from left to right.

Select *Text > Rotate*. Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string resides. Select the text string, it rotates anti-clockwise by 90 degrees each time it is selected.

Text > Mirror

Used to mirror text strings about their centre.

The layout is usually viewed looking down onto the top layer. Text placed the bottom layer should be mirrored when it is added so that it can be read correctly when the board is manufactured.

Select *Text > Mirror*. Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string resides, then select the text string you wish to mirror. Selecting the text string again mirrors it again.

Text may also be mirrored/unmirrored whilst being added or edited.

Text > Edit

Used to edit text strings, including silkscreen labels, to change their height, orientation, mirrored status or the line size used to form the characters, once they've been added to the board. Also used to edit URL anchors added using the *Text > Add URL Anchor* command.

Select *Text > Edit*. Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string or URL Anchor symbol resides, and the line size is set as required if editing text strings. (If the text's line size should not be changed, ensure the line size setting corresponds to the text's current line size - use *Identify > Feature* to find this out if it is unknown.) Select the text string or URL Anchor Symbol to be edited.

Text strings: a window appears containing the text string, its height, orientation and mirrored status - these can be changed as required. The angle setting has a choice of 4 "standard" angles to choose from, or the angle can be selected and re-typed to specify a "non-standard" angle, such as 27 degrees.

URL Anchor symbol: a window appears allowing the referenced file/address, colour or symbol to be edited. (To view the actual URL/file, use the *Text > Display URL* command.)

Select *OK* to implement the changes. Selecting *Cancel* leaves the text/URL anchor symbol as it was.

Text > Add URL Anchor

Used to attach design notes held in a file or a web-address, to the artwork layers in the shape of a symbol that is associated to the URL or file.

A component outline is used as the symbol for the URL anchor - only the silk screen data part of the outline library entry is used. It is suggested that an outline be specially made for the URL anchor, to avoid any future confusion with "real" parts.

When selected, a window appears, similar to that shown in Figure 203.

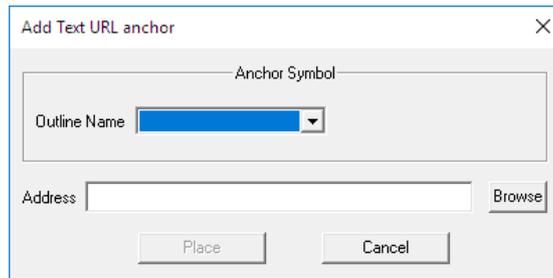


Figure 203

It requires the following information:

- Outline name:** selecting the down-pointing arrow from the right hand side of the setting produces the list of outline names from the design and master libraries, from which one should be selected. This will be the symbol used for the URL anchor on the artwork. Characters can be typed in the field if preferred, but only valid outline names will appear – for example if DI is typed, then DIL14 will appear (if it is the first outline commencing with DI, if DIL2 is typed then DIL20 will appear).
- Address:** the full path to the URL or file must appear here. A path can be typed, files “dragged and dropped” or the Browse button used to locate a file.

Once this window has been set appropriately, the *Place* button will become active and can be selected. The symbol will appear attached to the cursor and can be positioned with a click of the left button.

The *Text > Move/Edit/Delete* commands can be used on the symbol.

Text > Display URL

Used to display the file, web page etc. associated with a symbol that was added using the *Text > Add URL Anchor* command.

This command will work on all displayed URL symbols regardless of the active layer selected in the Text dialogue bar. Once the command has been selected, select the symbol, the associated web-page or file will open on the screen.

Note: if the associated file does not exist, no warning messages or indications are given when the symbol is selected. Use the *Text > Edit* command to check the path associated with the symbol.

Text > Delete

Used to delete text strings including part labels or URL anchors from the board.

Select *Text > Delete*. Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string or URL anchor resides and then select the text string/symbol. The string/symbol is deleted.

Text > Change Width

Used to change the line size used to form the characters in text strings.

Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string resides and the line size is set to the size code required.

Select *Text > Change Width*, then select the text string to be changed with a click of the left-hand button. The line size is changed.

Note: Line size can also be changed using the *Text > Edit* command.

Text > Change Height

Used to change the height of text strings.

Select *Text > Change Height*. Ensure the layer setting in the Text dialogue bar is set to the layer on which the text string resides and the *Height* is set as required. Select the text string with a click of the left-hand button. The height changes.

Note: Height can also be changed using the *Text > Edit* command.

Text > Get Label

Used to add individual labels (silk screen names/part references) to parts.

Silk screen labels are usually added automatically using the *Tools > Generate Silkscreen* command. However, if a modification has been made to a completed board, selecting auto-generation will duplicate existing labels as well as adding the new ones. The *Text > Get Label* command should therefore be used.

The *Text > Get Label* command should also be used if a silk screen label is deleted accidentally.

Refer to the *Grid* command for details on whether grid snapping is active. The current grid is shown in the information bar.

Ensure the layer, height and line size settings in the Text dialogue bar are set as required.

Select *Text > Get Label*, then select the datum of a part with the left button for unflipped parts and the right button for flipped parts. (When the artwork is displayed in “flipped” mode, the sense of the buttons becomes reversed.)

A rectangle appears on the end of the cursor representing the silk screen name. It may be rotated by pressing the user defined special function key for <rotate a part>. Move it into position and click the left-hand mouse button to release it.

Clicking the right hand mouse button before the label is released cancels that label.

When adding a label for a flipped part, the label is inserted in mirrored mode (so if the artwork is displayed in “flipped” mode, the label will appear the correct way around).

Logo commands

These commands are used to add, resize or delete symbols that have already been imported to the Logos Folder (details on page 175).

The Logo dialogue bar shown in Figure 204 controls which layer the logo is placed upon, its size, orientation and whether it is flipped.

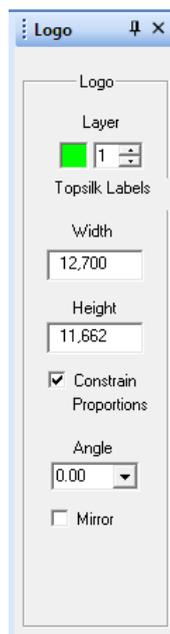


Figure 204

Once a logo is attached to the cursor (during placement or moving), the settings can be changed to update the attributes of the attached logo.

Logos may be placed on powerplane layers, and will be visible within the powerplane edit mode.

- Layer: indicates which layer the Logo can be selected from. As each layer is selected, the datum positions of any placed logos will be highlighted on the artwork.
- Width: indicates the width of the rectangular area that represents the attached logo
- Height: indicates the height of the rectangular area that represents the attached logo
- Constrain Proportions: when checked, the logo will not change shape when changing its width or height setting.
- Angle: indicates the angle at which the logo will be positioned, pre-set angles (0/90/180/270) can be selected using the drop-down arrow, or the setting can be manually changed to increments of 1 degree – select the current value, then type the new angle required – take care not to move the cursor away from the dialogue bar when making the change.
- Mirror: when checked, the logo will be mirrored (flipped).

Logo > Place

Used to add a pre-defined logo/symbol from the Logos folder, onto the artwork.

The *Logo* dialogue bar indicates the layer/size/orientation of the logo as it is positioned; these settings can be altered whilst the logo is attached to the cursor.

Once selected, expand the *Logos* folder in the navigator, then select the desired logo with a click of the left button. Move the cursor across into the artwork editor at which time the logo will be seen attached to the cursor. Release the logo on the artwork in the desired location with a click of the left mouse button, the right button will cancel the attached logo.

Logo > Move

Used to move a logo that has already been positioned on the artwork.

The settings within the *Logo* dialogue bar can be altered whilst the logo is attached to the cursor to change its size/layer/orientation.

Select *Logo > Move*, ensure the *Layer* setting in the dialogue bar is set to the layer the logo is on, the datum points of all logos on the current layer are shown by small white crosses. Select the logo to be moved with a click of the left mouse button, it attaches to the cursor.

Use the <rotate> user defined special function key to rotate the logo as it is moved.

Click the left mouse button to release the logo in its new position. Clicking the right hand mouse button returns the logo to its original position.

When the logo is selected, the *Logo* dialogue bar is updated to show the current settings of the logo, which can be changed as required.

Logo > Rotate

Used to rotate a logo that has already been positioned on the artwork.

Select *Logo > Rotate*, ensure the *Layer* setting in the dialogue bar is set to the layer the logo is on, the datum points of all logos on the current layer are shown by small white crosses. Select the logo to be rotated with a click of the left mouse button, it rotates anti-clockwise by 90 degrees from its current angle each time it is selected.

It is often easier to rotate a logo whilst it's being moved using the <rotate> special function key.

When the logo is selected, the angle setting in the *Logo* dialogue bar is updated to show its current value (none of the other settings change).

Logo > Mirror

Used to mirror a logo that has already been positioned on the artwork.

Select *Logo > Mirror*, ensure the *Layer* setting in the dialogue bar is set to the layer the logo is on, the datum points of all logos on the current layer are shown by small white crosses. Select the logo to be mirrored with a click of the left mouse button, it mirrors about its original vertical axis each time it is selected. (So if the symbol has been rotated by 90/270 degrees on the artwork, it appears to mirror about the horizontal axis.)

When the logo is selected, the *Mirror* setting in the *Logo* dialogue bar is updated to show its current value (none of the other settings change).

Logo > Edit

Used to edit any of the settings shown in the *Logo* dialogue bar for a selected logo.

Select *Logo > Edit*, ensure the *Layer* setting in the dialogue bar is set to the layer the logo is on, the datum points of all logos on the current layer are shown by small white crosses.

Select the logo to be edited with a click of the left mouse button, a dotted line will be shown around the edges of the selected logo; a new "Apply changes" button will appear in the *Logo* dialogue bar which is greyed out until a change is made in the dialogue bar.

Make the changes required to the settings in the dialogue bar (when adjusting Width or Height settings, use the <tab> key to move between settings to ensure the new value is accepted in the box).

Once a change has been made, select the "Apply changes" button to make the change. To cancel the command select another logo or a new command.

Logo > Copy

Used to copy a logo that has already been positioned on the artwork.

Select *Logo > Copy* and ensure the *Layer* setting in the dialogue bar is set to the layer the source logo is on, the datum points of all logos on the current layer are shown by small white crosses.

Select the logo to be copied with a click of the left mouse button, a copy of the logo appears attached to the cursor. The settings in the *Logo* dialogue bar can be altered to adjust the new logo if required.

Logo > Delete

Used to delete a logo from the artwork.

Select *Logo > Delete* and ensure the *Layer* setting in the dialogue bar is set to the layer the logo is on, the datum points of all logos on the current layer are shown by small white crosses.

Select the logo to be removed with a click of the left mouse button on its datum, it will be removed.

Region commands

The *Region* commands are used to define a rectangular area around a group of items on selected layers in order to move, copy, rotate, delete or store them.

The *Show Rules* command determines which layers/categories of data are included in the rectangular area.

Notes: when tracks are deleted using the *Region* commands, their unroutes are not restored until the artwork checking routine is run.

Deleting individual layers with an option to retain fixed tracks is also available separately - right-click on the Artwork from the navigator pane and select the Delete Tool – refer to page 202 for details of this command.

Region > Show Rules

Used to display the rules dialogue that controls which layers/items are included/excluded when using the Region commands.

When *Region > Show Rules* is selected, the *Rules* dialogue appears as shown in Figure 205.

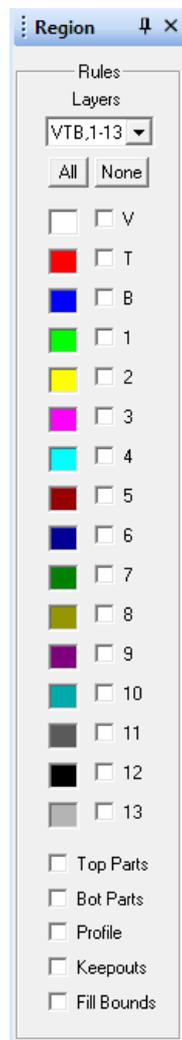


Figure 205

The layers of the board are listed, along with coloured boxes representing those layers for ease of recognition. A check box alongside each layer allows it to be selected/deselected for use with the Region operations. By default, none of the layers are selected (ticked).

Only 16 layers can be displayed at one time, so a layer selection setting will be found at the top of the dialogue, allowing layers V,T,B,1-13, 14-29, 30-45 or 46-61 to be displayed in the dialogue.

When individual layers are selected/deselected they maintain their status even though they may not be visible in the dialogue bar.

To assist with the selection/deselection of all layers (V,T,B,1-63), *None* and *All* buttons are provided.

In addition to the data on the layers, parts on the top and/or bottom of the board, the board profile, keepout areas and copper fill boundaries can also be selected/deselected for use with the Region commands.

Depending on what is required, a combination of layers/items will probably be selected.

Region > Move/Rotate

Used to move and/or rotate data from selected layers/categories within a rectangular area.

Refer to the *Region > Show Rules* command for details on item selection within the area.

The datum of a part has to be inside the area for it to be included.

If parts are not included in the rule selection, but there are tracks on the selected layers connected to their pins, the tracks will move and become disconnected from the pads. If tracks become disconnected in this way, connectivity (not the tracks) will be restored the next time the artwork is checked.

Tracks, profile lines or keepout lines that straddle the region are excluded from the move/rotate.

The datum of the region is the corner that is attached to the cursor.

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Region > Move/Rotate*. Click the left mouse button to locate one corner of the rectangular area to be moved/rotated. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and move with the cursor.

Each press of the rotate function key will rotate the rectangle anti-clockwise by 90 degrees.

Click the left mouse button again to release the area in its new position. A right-click will cancel the move/rotate.

Region > Copy

Used to copy or copy/rotate data from selected layers within a rectangular area, excluding parts and component pads.

(Use *Region > Macro Generate/Add* to include and copy component pads.)

Tracks, profile or keepout lines that straddle the area are excluded from the copy.

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Refer to the *Region > Show Rules* command for details on item selection within the area.

Select *Region > Copy*. Click the left mouse button to locate one corner of the rectangular area to be copied. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and move with the cursor.

Each press of the rotate function key will rotate the rectangle anti-clockwise by 90 degrees.

Click the left mouse button again to copy and release the area in its new position. A right-click will cancel the move/rotate.

Region > Delete

Used to delete data from selected layers within a rectangular area, excluding parts.

Use *Region > Unplace* to remove parts from the artwork.

Tracks, profile or keepout lines that straddle the region are excluded from the delete.

If tracks are deleted using this command, the unrouted are not restored until after an artwork check is performed. To maintain the unrouted when deleting tracks, use the *Mroute > Ripup* command.

Refer to the *Region > Show Rules* command for details on item selection within the area.

Select *Region > Delete*, click the left mouse button to locate one corner of the rectangular area to be deleted. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and a prompt will indicate that a left click will delete the area or a right-click will cancel the operation.

Region > Unplace

Used to return unrouted, unfixed parts within a rectangular area to the placement tray.

The datum of a part has to be inside the area for it to be included in the area.

Any parts that have routed or partially routed tracks connected to them will not be removed - a message appears in the information bar to that effect.

Parts that have been fixed with the *PartFix* commands are not removed, they should be unfixed first.

Region > Unplace operates on parts from the selected layers in the rules dialogue, refer to the *Region > Show Rules* command for details.

Select *Region > Unplace*, click the left mouse button to locate one corner of the rectangular area. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area.

The area will change colour and a prompt will indicate that a left click will unplace the parts within the area or a right-click will cancel the operation.

The unrouted, unfixed parts within the area are returned to the part placement tray.

Region > Macro Generate

Used to create a "macro" of the selected layers within a rectangular area and save it to disk.

A "macro" is a piece of artwork that can be saved and then re-introduced into the same or another artwork. It has the advantage over *Region > Copy* as it will also include component pads (but not the parts themselves), however part and connectivity information is not saved with the macro.

Note: the pads in the macro are not identified as component pads with pin numbers, etc. This means that routines that operate on component pads will ignore them, such as the automatic solder paste output.

The datum of a part has to be inside the area for its pads to be included.

The board profile and keepout data are not included when using the *Region > Macro Generate* command, even if they are selected via the *Region > Show Rules* command.

Tracks that straddle the region are excluded from the macro.

The layers the items reside on are maintained in the macro, but the *layer usage* is not. For instance, layer 3 could have been a silk screen layer when the macro was generated. This macro can be added to any layout without affecting the layer usage of the layout.

It is not possible to transfer data between layers using macros. (Depending on the requirement, either Gerber input, or input to a documentation layer from the output tools could be used.)

The macro file is saved in a folder on the hard disk - by default in the folder defined by the *File > System Setup* window however, any other folder can be chosen.

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Refer to the *Region > Show Rules* command for details on item selection.

Select *Region > Macro Generate*, click the left mouse button to locate one corner of the rectangular area. Move the cursor, stretching the attached rectangle to its desired shape, then click the left mouse button again to define the area. The area will change colour and a prompt will indicate that a left click will store the macro or a right-click will cancel the operation.

If the left-click is used, the browser appears, allowing you to navigate around to locate the folder in which you'd like the macro stored. Type in a name for the macro (the default extension of .amf is added automatically).

Once the name has been typed, press <enter> or select *OK*. The information bar should indicate the operation was successful.

Region > Macro Place

Used to add previously created macros to the layout.

When the macro is added to the current layout, the actual pad and track sizes used in the macro (not the code numbers) are matched to the layout's existing pad and track sizes. As an example, a 0.065" round code 8 pad in the macro might become a 0.065" round code 12 pad in the layout.

If a pad or track size does not appear in the sizes table of the layout but it is used in the macro, that size is added to the layout, providing an unused code is available. An error message is given if there are no unused codes available.

Layer usage is not stored with the macro. For instance, layer 3 could have been a silk screen layer when the macro was generated. This macro can be added to any layout without affecting the layer usage of the layout.

It is not possible to transfer data between layers using macros. (Depending on the requirement, either Gerber input, or input to a documentation layer from the output tools could be used.)

Refer to the *Grid* commands for details on whether grid snapping is active. The current grid is shown in the information bar.

Select *Region > Macro Add*. A window appears listing all the artwork macros that exist in the artwork macro folder, as defined by the *File > System Setup* window. Use the browser to locate a different folder if required. Select the macro required, followed by *Open*. The window closes, and a rectangle representing the macro appears attached to the cursor.

Select the position required for the macro and the contents of the macro are added to the layout. Another copy of the macro appears attached to the cursor, which can be added in the same way. Click the right mouse button to cancel the next macro.

Deleting/Renaming macros

Artwork macro files can be deleted or renamed using the Windows browser, outside of XL Designer.

Use the browser to locate the macro file, then select the file with a click of the right mouse button. Select *Delete* or *Rename* from the options that appear.

The browser window appears when *Region > Macro Generate/Add* is selected – at that time any of the files can be selected and then deleted/renamed.

Check commands

Check > Angles 45 OK

Used to indicate any track segments that are at an angle other than 0, 45 or 90 degrees. Arced track segments are not included in this check.

The check is run for user information only. No action is required if angle errors are flagged.

When the check is selected, all existing angle error flags are removed.

Select *Check > Angles 45 OK*. The checking is activated. The number of track segments that do not conform to the angles specified is given in the information bar. The segments are indicated with error flags showing the letter A, for *Angle error*, inside them.

If angle errors were reported but the flags cannot be seen, they are probably invisible. The *View Control* dialogue bar controls their visibility, and the *Check > Next ErrorFlag* command can be used to locate them.

The error flags are not removed until another angle check is performed, or they are deleted manually using the *Check > Delete/Delete All ErrorFlag* commands.

Check > Angles No 45

Used to indicate any track segments that are not at 0 or 90 degree angles. Arced track segments are not included in this check.

The check can be run for user information only. No action is required if angle errors are flagged.

When the check is selected, all existing angle error flags are removed.

Select *Check > Angles No 45*. The checking is activated. The number of track segments that do not conform to the angles specified is given in the information bar. The segments are indicated with error flags showing the letter A, for *Angle error*, inside them.

If angle errors were reported but the flags cannot be seen, they are probably invisible. The *View Control* dialogue bar controls their visibility, and the *Check > Next ErrorFlag* command can be used to locate them.

The error flags are not removed until another angle check is performed or they are deleted manually using the *Check > Delete/Delete All ErrorFlag* commands.

Check > Connectivity

Before the artwork can be regarded as "finished" this command should always be used.

It comprises two separate checks *Copper checks* and *Silk-screen checks*. It is **essential** that the *Copper checks* are performed and any errors investigated and corrected before proceeding to output/manufacture the design, otherwise an incorrect board may be produced. The *Silk-screen checks* are optional and used to produce a more manufacturable board.

Only one of the checks can be selected and performed at a time.

The *copper checking* routine is used to indicate any clearance violations between copper items, or differences between the artwork and the parts and wiring list (such as missing or shorted tracks).

Providing the parts and wiring list was compiled from a circuit schematic and the schematic has not been changed, the checks will also indicate differences between the artwork and the schematic.

The *silk-screen checks* are optional and indicate whether silk-screen data is touching pads/vias on the outer layers of the board, an undesirable feature from assembly/soldering purposes. They can also indicate whether the silk-screen text lies within the area occupied by the auto-placement footprint, something that is undesirable when trying to locate part references on a populated board.

The *Parts > Status* command indicates whether all the parts have been placed.

Other reasons to use the connectivity checker

- Checking can also be performed to validate tracks that have been added using the Amend commands – these are not automatically recognised as belonging to a particular net. Provided they create a valid connection the artwork checker will validate them, they will then highlight, join to copper fill, etc..
- Schematic/parts/wiring list changes can sometimes result in unroutes appearing for routed tracks after the modification, in which case the checker will remove unroutes provided the associated tracking is correct (there are no short circuits).
- Unroutes sometimes remain for routed tracks after the auto-router has been used. Running the check will remove unrequired unroutes.

Copper checks

Layers defined as silk-screen or documentation layers are **not** included in the artwork copper checking routines - they are ignored as it is assumed they will be produced from non-conducting materials or are not part of the

board.

When the checking routine is selected, if any copper filled areas have been added to the artwork then the copper fill is automatically included in the copper checks, but this can slow down the checking process. The copper fill checks can be switched off if required using the switch provided, but ensure the implications of switching them off are understood – see below.

The checking routine can also identify copper (tracks/pads/text) within a keepout area. This check can also be excluded if required.

The copper checks require the minimum default clearance required between copper items to be specified. Any specific clearance requirements defined on the schematic or in the wiring list for selected connections will be applied to the appropriate connections.

Note: when defining minimum clearance values on the schematic it is possible to define two different minimum clearance values for one net (using the *Wires > Attributes* and *Tools > EMC Attribute Tags* commands), if this occurs, the larger of the two minimum clearances specified is used.

When started, the copper check routine checks every space between copper items (pads, tracks, text, etc.) to ensure the space or "gap" is at least the size specified. If gaps are found that are less than the size specified, a gap error is reported, and a gap error flag placed on the artwork next to the offending gap.

Because it is possible to have more than one minimum gap requirement, each gap error flag is given a numeric suffix that appears inside the error flag. When error flags are visible, a table is displayed in the bottom left corner of the artwork editor, showing the numeric suffixes and the minimum clearance requirement for each one.

Once the gap checking sequence has finished, a connectivity check is performed. The copper (not the copper fill unless it has been included in the check) on the board is scanned and a "wiring list" generated from the scan. This "wiring list" is compared against the original parts and wiring list for differences.

Typical errors are connections that have not been routed or tracks that have been shorted to one another.

If missing connections are found, the connections are re-inserted, and will need routing.

If short circuits are found, the checking routine tries to determine where the actual short has occurred and will place a short circuit error flag there. This can take a considerable amount of time, so the user can move on to the next stage of the checks by selecting the *Skip Short Search* button when it becomes active. As a guide, if the short circuit check has been running for more than 30 seconds, it is unlikely to be able to find the exact location of the short. If skipped, the final error report indicates that short circuits were found between the nets identified, but doesn't flag the actual locations.

Unplaced parts and parts outside the board profile are also flagged as possible errors.

Once the checking routine has finished, the error report can be printed or directed to a file.

When the checks are complete, the artwork should be viewed and the reported errors corrected.

Always run the artwork checks again once the errors have been corrected, as it is very easy to introduce more errors when clearing reported errors. The board should not be manufactured until **all** errors have been cleared and no errors are reported (it is possible to produce the output files with the errors uncorrected, but not recommended).

Important Notes:

- On power plane layers, the artwork checking routine does not detect isolated pins which can be caused by the proximity of anti-pads, heat-relief pads or isolation lines to one another.
- On power plane layers, anything added to the plane layers is removing copper and could cause isolated areas/pins and this is not identified by the checking routine.
- On power plane layers, short circuits caused by split plane boundaries crossing through heat-relief pads are not flagged.
- Free copper added within a component outline is NOT included in the artwork checking routines.
- Logos may be placed on powerplane layers, and will be visible within the powerplane edit mode. However, logos that are placed in powerplane layers are currently invisible to the artwork checker. No checks for violation with other powerplane features will be performed
- If a single node net has been assigned an EMC rules clearance parameter in the schematic, the default clearance value is used to check violations of the part pin on that single node net. (A single node net is a component pin that is not connected to any other component pin in the wiring list.)
- If a net has been assigned an EMC rules clearance parameter in the schematic, and the checking routine detects that the net has been incorrectly routed, then the default clearance value is used to check that net until the net has been correctly routed.
- Unroutes for routed tracks will disappear provided each track segment of the net is attached to the centres of connected pads/vias. By manually adding tracks that aren't connected to the pad/via centres, it is possible to make a connection that would not be permanent if the part/via were to be moved in the future. In this event, an error is not reported because the connection has been made, however to indicate this may lead to a problem in future, the unroute will remain even though the track

is “joined”. To ensure the unroutable is removed upon checking, ensure the tracks end on the centres of pads – this can best be identified with the tracks and unroutes set to centrelines and the pads unfilled. Use Amend > MovePoint to release the off-centre track-ends onto the centres of the pads, then re-check.

- Free copper features that interconnect component pins in an outline defined as a “star point” outline is never seen by the artwork checker. It is essential that the star point outline is manually verified to ensure it does actually perform the desired interconnectivity between its pins.
- Parts outside the board profile may be flagged even though they appear to be within the profile - this is due to an approximation being made of the area occupied by the outline and the board profile shape – the check is designed to identify parts that have been temporarily placed outside the profile that may have been forgotten about, rather than a precise check.
- The artwork checker tests all copper type layers to determine if any antipads or heat-reliefs are present, and will abort artwork checking if any are found. A report of the unwanted pad locations will be displayed - they can be cleared automatically by right-clicking *Artwork* from the navigator pane, then selecting the *Delete Tool*.
- Star points: the checking routine does not verify the copper connections between the star point outline pins, these must be manually verified to ensure the outline does actually perform the desired interconnectivity between its various pads..
- Logos: Artwork checking does not check for clearance to each individual vector within the logo. The boundary of each logo is represented by a rectangular area and it is this rectangular area that is used during artwork checking to identify the extents of the symbol.

Running the copper checks

Select *Check > Connectivity*. The window shown in Figure 206 appears and should be set up as required. A description of each setting follows.

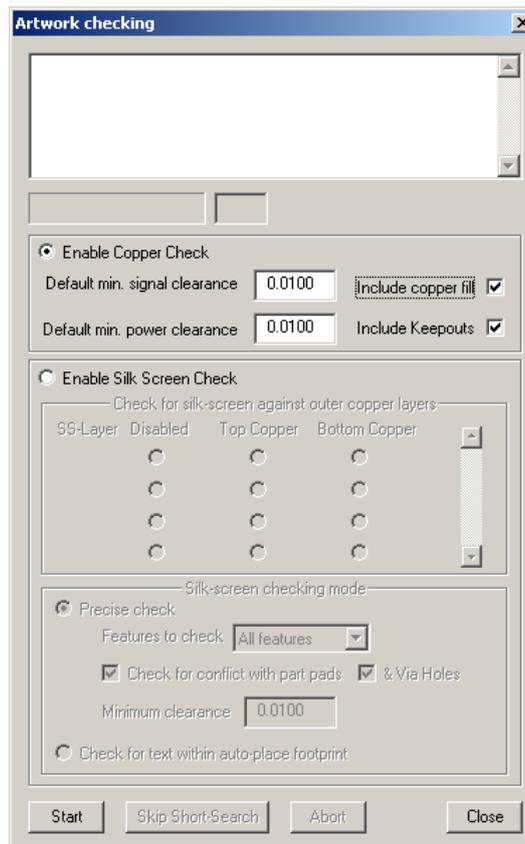


Figure 206

Tick the button alongside *Enable Copper Check* if it isn't already ticked to enable the copper checks (this will automatically disable the Silk-screen Checks).

Default Minimum Signal Clearance edit this value to correspond to the smallest clearance acceptable between separate (non-connected) signal items (pads/tracks/copper) on the board that do not have a minimum clearance specified from the schematic or wiring list.

Default Minimum Power Clearance

edit this value to correspond to the smallest clearance acceptable between separate (non-connected) power items (pads/tracks/copper) on the board that do not have a minimum clearance specified from the schematic or wiring list.

Ranger always checks using the largest minimum clearance specified if there is a conflict – for example if a connection on the schematic has a minimum clearance value of 0.005" assigned, but the default minimum clearance for the design is set to 0.012", then the 0.012" value is used when checking.

These settings are automatically updated to correspond to the values assigned for online-DRC in the artwork editor when manually routing tracks. If the values are changed in this window, then the change is automatically updated in the manual routing parameters window in the artwork editor.

Include Copper Fill

This is automatically selected if copper fill has been detected on the board - only switch it off if the implications of not including it are understood. If unsure, keep it selected.

The copper fill checks do slow down the running time of the checking routine and increase the memory requirements of the design which is why a switch is provided.

The copper fill checks can safely be switched off if the copper fill was added using the non-smooth edge mode **and** it has not been modified manually (using the Amend commands).

The copper fill checks should be included if:

- the copper fill has been modified manually (using the Amend commands)
- it is suspected that the connectivity provided by the copper fill has been broken.
- the smooth edge mode was used to add the fill - this can result in isolated areas of copper occurring
- the copper fill is being relied upon to provide the connectivity of a net . (Typically the copper fill is added after all the connections have been routed. However it is possible to leave a net unrouted and use the copper fill routine to form the connectivity of that net. In this case the fill routine cannot be relied upon to connect to every node in the net and this check should be performed to identify unconnected nodes.

If the copper fill is included in the checking, then each segment of the copper filled area is checked:

- to ensure it is not too close to any other piece of copper
- it is not isolated from a component pin
- not shorting different tracks together
- that it completes the net to which it is attached

Include Keepouts

This check will indicate any copper items (tracks/pads/text) inside a keepout area, or that cross a keepout line. This may/maynot be a problem so the check can be switched off if required.

Violations are indicated by an error flag on the artwork with a letter K (for Keepout).

The *Close* button can be selected to abandon the checks, or select *Start* to run the checks.

Whilst the check is running, messages appear at the top of the checking window to indicate what is happening as the checking routine runs through its various "phases".

If short circuits are detected it can take a long time to identify exactly where the short exists. The *Skip Short Search* button can be selected to speed up the checks, but if selected, error flags will not be placed at the locations of the short circuits - but the nets which are shorted will be identified in the error report.

When the checking routine is complete, the report will either indicate *Artwork check complete, No errors found* or a list of errors will be reported.

If no errors were found, the artwork is complete.

If errors were found they must be investigated and corrections made as appropriate.

The error report can be printed or directed to a file by selecting the *File > Print* or *File > Save As* command from the top of the report window. If *File > Save As* is selected, the file can be placed wherever required using the

browser window that appears.

When *connectivity errors* are reported, the missing connections are added to the artwork, and will be seen when the artwork is next viewed. They should of course be routed.

The artwork should be viewed and the reported errors corrected. Ensure the error flags are visible using the *View Control* dialogue bar. The *Check > Next Errorflag* command can also be used to find the error flags. The flags, shown as a small circle with a tail, contain the letter G for a gap (clearance) error or the letter S for a short circuit error.

Artwork checking Aborted

Under certain conditions the checking routine cannot perform so the check will be aborted. A message is given and the reason for the failure to complete the checks will be given. This problem should always be rectified so that the checking can be performed.

Typical error messages

The following messages are given as examples from an error report:

Example 1

```
-----Shorted Signals-----  
Signal  
C4.1          R2.1          IC1.4  
  --- shorted to signal ---  
C3.1          C2.1          PL1.5  
Short circuit(s) detected at:-  
X 3606 Y 2798 (flagged)
```

This message indicates that the two signals listed are shorted to one another at the location X 3.606", Y 2.798", and a short circuit flag has been added.

Example 2

If more than two signals were shorted together, the message would include all the signals in the report. For instance, the following message indicates that three signals are shorted to one another:

```
-----Shorted Signals-----  
Signal  
C4.1          R2.1          IC1.4  
  --- shorted to signal ---  
C3.1          C2.1          PL1.5  
  --- shorted to signal ---  
IC3.10 C12.1          IC2.5  
Short circuit(s) detected at:-  
X 3606 Y 2798 (flagged)  
X 4500 Y 3609 (flagged)
```

Typically flags identify the offending short-circuit so they are easy to locate and correct. However, where very large nets are involved the shorts may not be located. In these cases, the shorted signals are identified, but not the actual location of the short-circuit. These have to be located by eye.

If it seems as though every net on the board is shorted to every other net on the board, does the board contain power planes? If it does, have they been generated, or have some extra pads or vias been added since the power planes were generated? If the answer is Yes to any of these questions, the power planes should be generated again and the checks re-run.

Example 3

If the report indicates a short circuit, but does not specify which other net the net is shorted to, then this indicates the net includes duplicate pins (which should be rectified). For example:

```
-----Shorted Signals-----  
Signal : GND  
C14.2          D19.2          D11.2          IC5.16  
D17.3          C5.1          D4.2          D5.2  
IC5.16         LK1.2          IC1.8          IC1.19  
IC2.10         IC3.10         IC4.8
```

If this net is examined, you'll notice that **IC5.16** appears twice. To rectify this problem, return to the source of the wiring list – either the schematic diagram or the imported netlist and remove the duplicate pin

reference.

On a schematic diagram a typical mistake involves defining one pin twice in a part, as a terminal and also an implied power pin. Edit the appropriate part and use the Terminals > Check command to identify multiple use of pins. Correct (by removing one of the definitions) as appropriate. Extract the parts/wiring list, then run the artwork check again.

In an imported netlist there is no checking performed on the netlist when it is imported, so either edit the wiring list from the wiring list editor (remember to correct the change at the source as well) or correct the netlist at source and re-import.

Example 4

The following message is given when connections have not been routed for whatever reason:

```
-----Connectivity Errors-----
Error in net
R2.2          C4.2          BR1.4          C1.2          PL1.7
Subnet R2.2          C4.2
not connected to
Subnet BR1.4          C1.2          PL1.7
```

The above message indicates that there is an error in the net (signal) which is listed. It then indicates that the error is caused because the net has been split into two halves or "subnets", i.e. R2.2 and C4.2 are connected together, as are BR1.4, C1.2 and PL1.7, but there is a track missing between the two "subnets". When viewing the artwork, the missing connection will re-appear.

Example 5

The following message is similar, but this time PL1.6 is not connected to anything.

```
-----Connectivity Errors-----
Error in net
R3.2          BR1.2          PL1.6
No connection to node(s):-
PL1.6
```

Example 6

The following message is given when the complete net has not been routed:

```
Error in net:-
PL1.11          IC6.4          IC10.11
Net has not been routed
```

The same message which lists only one part pin as follows, indicates that a "single node net" exists.

```
Error in net:-
IC6.3
Net has not been routed
```

A single node net is an entry in the wiring list that is not connected to anything else. They can be caused on the circuit by a connection being added to a pin and not terminating on another pin, or a signal name being left off. Find out whether the pin should be connected, then rectify the schematic accordingly and recompile the parts and wiring list. Take appropriate action on the artwork.

Example 7

```
-----The following partpins should not be connected-----
IC6.2
```

This message indicates that a track has been connected to the pin listed (IC6.2), which should in fact be unconnected:

Example 8

```
Warning: 4 isolated copper fill regions found
```

This message indicates that the copper fill has areas that are unconnected. It is undesirable to complete an artwork with isolated copper areas, so these should be removed. To view the isolated copper areas, ensure all the copper layers are visible (*View Control* dialogue bar) and isolated copper areas are visible (*View > Isolated Copper*). The isolated copper areas will be displayed in white. They can be deleted automatically if *Tools > Copper Fill* followed by *Filled Copper > Delete All Isolated Copper* is selected. A message will appear asking for confirmation of the deletion.

Example 9

```
----- Parts check -----  
  
Warning : Part MP5 is not placed  
Warning : Part MP6 is not placed  
Warning : Part MP8 is not placed  
Warning : Part SK11 is outside the board profile  
Warning : Part SK12 is outside the board profile
```

This message indicates that the parts MP5, 6 & 8 have not been placed on the artwork and parts SK11 & 12 have been placed outside the board profile. This is a warning and should be investigated but can be ignored if appropriate.

Example 10

```
----- Artwork checker initialisation problems -----  
  
Split powerplane layer 1 : Subnet polygon error for power signal '12V  
  No polygon defined  
  
Split powerplane layer 1 : Subnet polygon error for power signal  
'EARTH  
  No polygon defined  
  
Copper_check aborted
```

This message indicates that the copper checking has been aborted because the split-powerplane layer 1 has not been defined or generated. The split-plane must be created and the check run again.

Example 11

```
----- Artwork checker initialisation problems -----  
  
Split powerplane layer 6 : Subnet polygon error for power signal '0VA1  
  Polygon is defined in powerplane generator, but cannot locate the matching  
  polygon on the artwork layer  
  
Split powerplane layer 6 : Subnet polygon error for power signal '0VA2  
  Polygon is defined in powerplane generator, but cannot locate the matching  
  polygon on the artwork layer  
  
Split powerplane layer 6 : Subnet polygon error for power signal '0VA3  
  Polygon is defined in powerplane generator, but cannot locate the matching  
  polygon on the artwork layer  
  
Split powerplane layer 6 : Subnet polygon error for power signal '0VA4  
  Polygon is defined in powerplane generator, but cannot locate the matching  
  polygon on the artwork layer  
Copper_check aborted
```

This message indicates that the copper checking has been aborted because the split-powerplane polygons for the secondary power rails (0VA1, 2, 3 & 4) on layer 6 have not been correctly identified. Use the *SplitBoundary > Import Boundary* command to identify the polygons and generate the planes again. Run the check once the planes have been re-generated.

This message typically appears on designs that have been created with Ranger XL or SXL D prior to v1.72. Once the polygons have been imported and planes re-generated, the problem should be resolved.

Example 12

```

Heat relief pad found on non-powerplane layer 1 at X 3.4500 Y 1.9500
Heat relief pad found on non-powerplane layer 1 at X 1.8000 Y 2.5000
Heat relief pad found on non-powerplane layer 1 at X 2.7000 Y 3.2000
Antipad found on non-powerplane layer 1 at X 4.0500 Y 3.7000
Antipad found on non-powerplane layer 1 at X 4.1500 Y 3.7000
Antipad found on non-powerplane layer 1 at X 1.2000 Y 2.5000
Copper_check aborted

```

This message indicates that the copper checking has been aborted because heat-relief and/or anti-pads have been found on a copper layer and this is invalid. These pads MUST be deleted before the checks can be run. Use the Delete Artwork tool (right-click on Artwork in the navigator) to quickly remove unwanted heat-relief/Antipads from copper layers or use the Amend command in the artwork editor to remove them individually

Note: X/Y co-ordinates are given with respect to absolute 0/0 and not the datum, which can be moved.

This message typically appears on designs that have once had a powerplane and this has been changed to a copper layer.

All errors should be investigated and corrected, then the checks re-run. Repeat until no errors are reported.

Artwork checking, silk-screen checks

This routine identifies any layers defined as silk-screen layers and indicates data (lines/text) on those layers that is close to or touching pads and/or vias on either of the two outer copper layers (top and bottom) of the board.

It can also be used to indicate text within the area of a component outline's auto-placement footprint.

Running the Silk-screen Checks

Select *Check > Connectivity*. The window shown in Figure 207 appears.

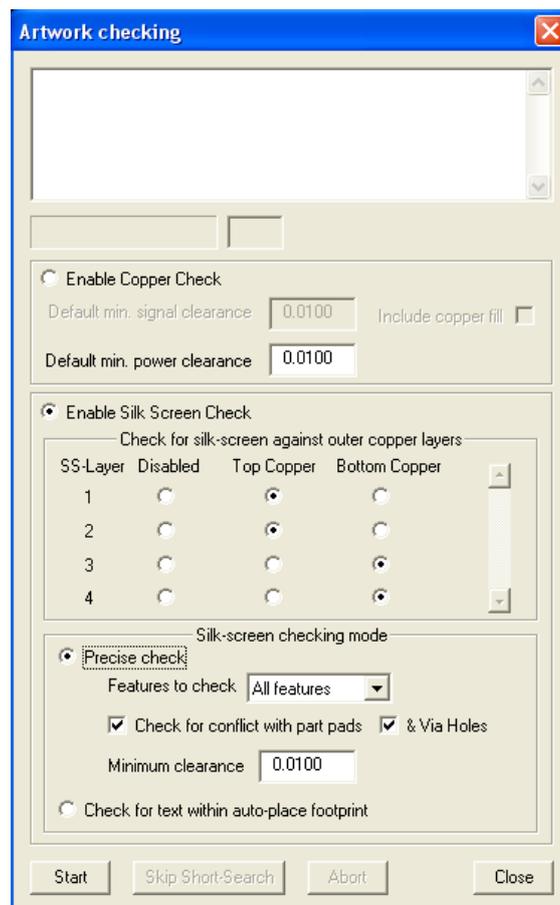


Figure 207

Tick the button alongside *Enable Silk-screen Check* to enable the silkscreen checks (this will automatically disable the Copper Checks). (This button is not selectable if no silk-screen layers have been detected in the

artwork.)

Set the window as required, a description of each setting follows.

Check for silk-screen against outer copper layers

This section is used to specify which silk-screen layers should be checked with which of the outer layers.

A list of the silk-screen layers detected appears, and alongside them, three columns from which one setting can be selected:

<i>Disabled</i>	if selected this silk-screen layer is not included in the checks. If all the silk-screen layers are disabled, due to there being nothing to check, the <i>Start</i> button will remain greyed out.
<i>Top Copper</i>	if selected, this silk-screen layer is checked against the top layer.
<i>Bottom copper</i>	if selected, this silk-screen layer is checked against the bottom layer.

Silkscreen checking mode

The buttons alongside *Precise check* or *Check for text within auto-place footprint* can be selected, depending on what is required. Both checks can be run, one after the other.

Precise Check

Features to check when the check is performed both silkscreen lines and text can be identified and checked or just the text. Select from:

All features Lines and text on the selected silk-screen layers is included in the check.

Text only Only the text on the selected silk-screen layers is included in the check.

Check for conflict with Part pads when selected, the checking routine will identify any of the silk-screen features being checked that are closer than the minimum clearance requirement to component pads.

Check for conflict with Via holes when selected, the checking routine will identify any of the silk-screen features being checked that are closer than the minimum clearance requirement to via holes. Pads added using the *Amend > Enter Pad* command are also included in this check.

Minimum Clearance this defines the minimum distance required between silk-screen data included in the check and the pads/vias on the selected layer(s).

Check for text within autoplacement footprint

This check if enabled will identify any text that is within the autoplacement footprint (as defined within the component outline). This check can be used to ensure the part references can be viewed on a populated board.

The *Close* button can be selected at this stage to abandon the checks, or select *Start* to run the checks.

Messages appear at the top of the checking window to indicate what is happening, the checking routine runs through various "phases".

When the checking routine is complete, the report will either indicate *Silkscreen check complete, No errors found* or a number of errors will be reported.

If errors were found they should be investigated and corrections made if required. When the artwork is viewed, ensure the error flags are visible using the *View Control* dialogue bar. The *Check > Next Errorflag* command can also be used to locate the error flags. The flags, shown as a small circle with a tail, contain the letter M for a silk-screen *Mask* error.

Re-run the check if changes were made.

Check > Count Unroutes

Used to indicate the number of unrouted tracks on the artwork.

It is often easy to miss unrouted tracks that are between pins of ICs, etc. and think the board is completely routed. This check can be run to ensure that all the tracks have been routed before running the final artwork checks. (The final artwork checks also indicate unrouted tracks.)

Select *Check > Count Unroutes*. A report appears in the information bar listing the number of normal unroutes and the number of partial unroutes - a "normal" unroute is a pad to pad connection that has not been routed, it is shown as a solid line - a partial unroute is a pad to pad connection that has been routed but not completed, the unrouted part is shown as a dashed line.

Partially unrouted connections are highlighted with an error flag containing the letter U for *Unroute*, complete unroutes are not flagged. If partial unroutes were reported but the flags cannot be seen, they are probably invisible. The *View Control* dialogue bar controls their visibility, and the *Check > Next ErrorFlag* command can be used to locate them.

If the dashed partial unroutes cannot be seen, they will appear when the copper layer that they are attached to is displayed.

If the unroutes cannot be seen, ensure that all signal and power connections are visible (*View Control* dialogue bar).

If they still cannot be seen, make all the layers temporarily invisible (*View Control* dialogue bar) except for the unroutes.

The error flags remain until they are deleted using the *Check > Delete/Delete All ErrorFlag* commands or the checks performed again.

Check > Next ErrorFlag

Used to indicate the "next" error flag. It momentarily flashes the next occurrence of an error flag to aid its location.

Ensure that the error flags are visible in order to see them, using the *View Control* dialogue bar.

This facility is useful when clearing reported errors, since time is saved when trying to locate them.

If the next error flag is outside of the current screen area, the screen is re-drawn to centre the flag.

Each error flag is highlighted in turn until each one has been shown. If next is selected again, the errors are highlighted again. This happens even if the errors have been corrected. Use the *Check > Delete ErrorFlag* command to remove the individual error flags so they are no longer located.

Select *Check > Next ErrorFlag*, the next error flag momentarily flashes on the screen.

Check > Delete ErrorFlag

Used to delete individual error flags. Deleting the error flag does not remove or correct the error.

As reported errors are cleared, it is beneficial to remove the error flags, as it is easier to see which errors are left. The *Check > Next ErrorFlag* command highlights all error flags in turn, even if the cause of the error has been removed.

Once errors have been corrected, it is good practice to run the checks again. This ensures that the error flags were not deleted accidentally, or that new errors have not been introduced.

Select *Check > Delete ErrorFlag*. Point at the end of the tail extending from an error flag, and click the left hand mouse button to delete it.

This command stays active until the right hand mouse button is clicked.

Check > Delete All ErrorFlags

Used to delete all types of error flags (angle errors, gap errors and short circuit errors). Deleting the error flags does not remove or correct the errors.

When the error checking routines are selected, all existing error flags for that type of check are deleted automatically. For instance, existing angle error flags are deleted when angle checking is performed, but not when *Check >Connectivity* is selected.

Once all the error flags have been deleted using this method, it is good practice to run the checks again. This ensures the error flags were not deleted accidentally, or that new errors have not been introduced.

Select *Check > Delete All ErrorFlags*. All the error flags are deleted.

Netfix commands

These commands are used to fix and unfix tracks in the artwork editor.

There is no requirement to fix tracks unless the board is being submitted to a rip-up auto-router. However, the *Delete All Artwork* command does allow individual layers to be deleted whilst retaining fixed tracks, which is useful if you want to start the artwork tracking again, but retain some existing routed features.

The Seetrix rip-up router or Specctra/Elecctra routers will not rip-up fixed tracks. Before boards are submitted to the auto-routers, it is often desirable to pre-route some tracks that have specific routing requirements. Examples of this are power tracks and high speed clock signals which are best routed by the designer to ensure they meet the design criteria.

Power and any hand-routed signal tracks are automatically fixed as they are added.

Whilst in the Netfix menu, fixed tracks are highlighted to indicate they are fixed. The highlighting is removed as soon as they are unfix.

Note: before tracks that have been added using the Amend commands can be fixed, they must be identified and recognised as valid tracks by running the artwork checking routines. A valid track is one that makes a connection between pins in a net from the wiring list.

The Netfix menu allows tracks to be fixed and unfix in a variety of ways using the following commands.

NetFix > Selected Nets

Used to select tracks for fixing or unfixing, depending on whether the *Fix* or *Unfix Mode* is selected from the

NetFix pull-down menu. All the tracks connected to the selected track will become fixed or unfixed.

Ensure the *NetFix > Fix Mode/Unfix Mode* is selected as required. Select *NetFix > Selected Nets*, then select a track. The highlighting switches on or off depending whether *Fix Mode* or *Unfix Mode* is selected to indicate the track is fixed or unfixed.

NetFix > By Pin Reference

Used to fix or unfix tracks, depending on whether the *Fix* or *Unfix Mode* is selected from the *NetFix* pull-down menu, that are connected to a pin of a specified part.

Ensure the *NetFix > Fix Mode/Unfix Mode* is selected as required. Select *NetFix > By Pin Reference*. A window appears requesting a pin reference. Type in the pin reference as a part name, followed by a dot and the pin number, e.g. *IC4.3* or *R1.2*, etc. Press <enter> or select *OK* and all the tracks attached to that pin become fixed or unfixed. The highlighting switches on or off depending whether *Fix Mode* or *Unfix Mode* is selected to indicate the track is fixed or unfixed.

If the net has a name, its name is shown in the information bar.

NetFix > By Signal Name

Used to fix or unfix tracks, depending on whether the *Fix* or *Unfix Mode* is selected from the *NetFix* pull-down menu, by their signal name.

Ensure the *NetFix > Fix Mode/Unfix Mode* is selected as required. Select *NetFix > By Signal Name*. A window appears requesting a signal name to be entered. Type in the signal name then press <enter> or select *OK* and all the tracks connected to that signal become fixed or unfixed. The highlighting switches on or off depending whether *Fix Mode* or *Unfix Mode* is selected to indicate the track is fixed or unfixed.

NetFix > All Nets

Used to fix or unfix, depending on whether the *Fix* or *Unfix Mode* is selected from the *NetFix* pull-down menu, all of the routed tracks.

Ensure the *NetFix > Fix Mode/Unfix Mode* is selected as required. Select *NetFix > All Nets*. All of the routed tracks become fixed or unfixed. The highlighting switches on or off depending whether *Fix Mode* or *Unfix Mode* is selected to indicate the tracks are fixed or unfixed.

NetFix > Fix Mode

When selected/active the *Netfix* commands will fix any tracks that are specified from that point onwards, until the selection is changed.

NetFix > UnFix Mode

When selected/active the *Netfix* commands will unfix any tracks that are specified from that point onwards, until the selection is changed.

Hilight commands

These commands are used to high-light or hide specific connection nets and, or routed tracks with a chosen colour for identification purposes.

Note: before tracks that have been added using the Amend commands can be highlighted, they must be identified and recognised as valid tracks by running the artwork checking routines. A valid track is one that makes a connection between pins in a net from the wiring list.

When any if the Hilight commands is selected, the Hilight dialogue bar appears as shown in Figure 208.

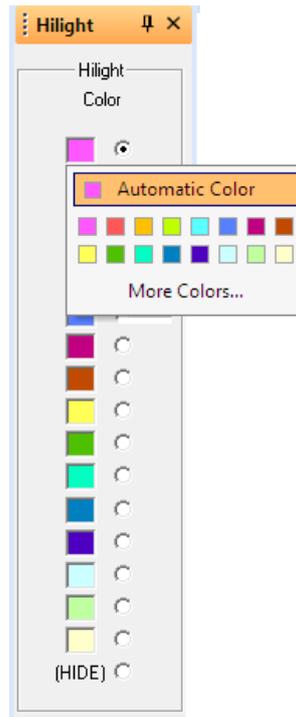


Figure 208

The coloured boxes can be selected to produce the colour palette (as shown in Figure 208) and the colour of choice selected.

The radio button alongside one of the colours indicates which colour will be used to highlight the selected/named nets.

If the colour is changed whilst a net is highlighted with that selection, the highlighting is changed to reflect the new colour chosen.

The (Hide) setting allows the selected nets to be hidden from view by making them black. Clearing all highlights will bring the hidden nets back into view.

Highlight > Selected Nets

Used to high-light connections and/or tracks that are attached to the selected connection or track.

Connection or tracks are highlighted in the colour selected from the *Highlight Colour* dialogue that appears when any of the *Highlight* commands are selected.

Select the colour required for the highlight before high-lighting the tracks/nets.

The highlighting remains on until it is switched off by either highlighting an already highlighted net or by selecting the *Highlight > Clear All* command.

Select *Highlight > Selected Nets*, then select a track or a connection. All the tracks and connections belonging to that net are highlighted. If the net has a name, its name is displayed in the information bar.

Selecting the track or connection again removes the highlighting.

Highlight > By Pin Reference

Used to high-light the position of connections and/or tracks that are attached to a pin of a specified part.

Connection or tracks are highlighted in the colour selected from the *Highlight Colour* dialogue that appears when any of the *Highlight* commands are selected.

The highlighting remains on until it is switched off by either highlighting an already highlighted net or by selecting the *Highlight > Clear All* command.

Select *Highlight > By Pin Reference*. A window appears requesting a pin reference. Type in the pin reference as a part name, followed by a dot and the pin number, i.e. **IC4.3** or **R1.2**, etc. Press <enter> or select *OK* and all the tracks and connections attached to that pin are highlighted. If the net has a name, its name is displayed in the information bar.

Selecting the track or connection again removes the highlighting.

Highlight > By Signal Name

Used to highlight connections and tracks by their signal name.

If a signal exists in both upper and lower case versions (i.e. **CLOCK** and **clock**), then only the one matching the

case entered will be highlighted.

If a signal name is unique (i.e. only CLOCK exists) then the search name may be specified in upper\lower or mixed case.

The wild card * can be used to represent all characters to highlight all named signals.

Connection or tracks are highlighted in the colour selected from the *Highlight Colour* dialogue that appears when any of the *Highlight* commands are selected.

The highlighting remains on until it is switched off by either highlighting an already highlighted net or by selecting the *Highlight > Clear All* command.

Select *Highlight > By Signal Name*. A window appears requesting a signal name to be entered. Type in the signal name pin. Press <enter> or select *OK* and all the tracks and connections attached to that signal are highlighted.

Selecting the track or connection again removes the highlighting.

Highlight > Clear All

Used to remove all highlighting from connections and tracks.

If you wish to remove the highlighting from selected tracks only, highlight those nets again individually to toggle the highlighting off.

Select *Highlight > Clear All*. All the highlighting is removed.

Specctra auto-router interface

XL Designer comes with a two-way interface to the Specctra auto-router. The Specctra auto-router is an optional extra that can be purchased if required.

The interface allows an artwork layout to be output from XL Designer in a format that the Specctra auto-router software can accept. Once the Specctra software has auto-routed the design, its output file can be converted back into XL Designer for further design work and checking as required.

The artwork design that is submitted to Specctra can include keep-out data (Specctra will not cross a keepout line) and pre-routed fixed or un-fixed tracks.

The Specctra auto-router will use user-defined via stacks if they are defined and enabled for use.

To access the Specctra interface, it should first of all be enabled (*File > System setup* window).

Then, open the design to be submitted to the Specctra auto-router. Right-click on *Artwork* from the navigator, then select *Optional Autorouters > Cadence/ CCT Specctra router*. A window similar to the one in Figure 209 appears.

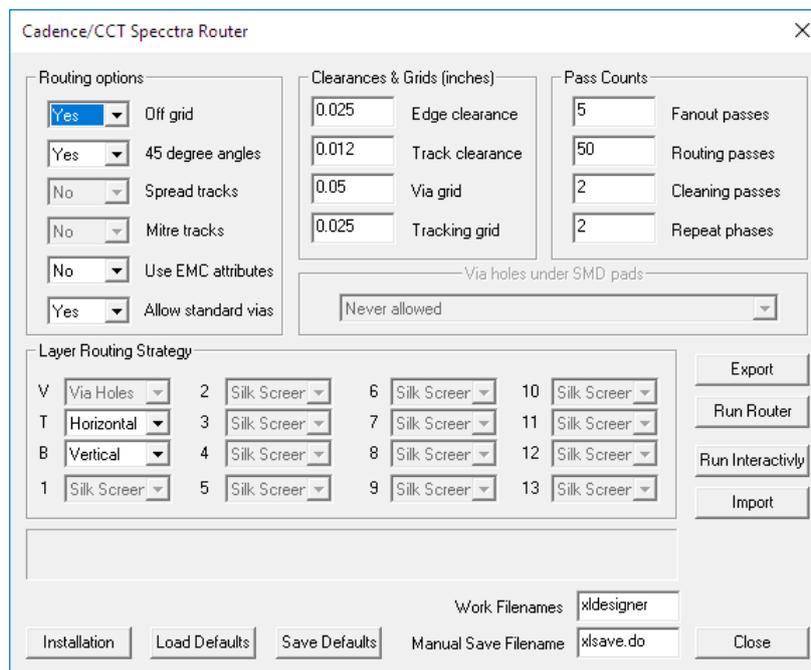


Figure 209

Before submitting the design to the auto-router, the settings in this window should be set as required. The window is divided into areas that control the options that are included in the file that is output for the Specctra auto-router. Use the following information to decide how it should be set.

If the interface has not been used before, select the *Installation* button and ensure the window is set to match the Specctra auto-router to be used (described below).

Routing options

Off grid selections are *Yes* or *No*. With *Yes* selected, tracks can be routed off the chosen tracking grid where necessary. Typically set to *No*.

45 degree angles selections are *Yes* or *No*. With *Yes* selected, the following command is added to the script file (rangerxl.do) that controls the routing of the design.

bus diagonal

Refer to the Specctra Reference Guide for details on this command, but basically if enabled, 45 degree tracks are permitted in tracks. Typically set to *Yes*.

Spread tracks - selections are *Yes* or *No*. With *Yes* selected, the following command is added to the script file (rangerxl.do) that controls the routing of the design.

spread (extra 5 25)

Refer to the Specctra Reference Guide for full details on this command, but basically when routing is complete, tracks are spread apart.

Mitre tracks selections are *Yes* or *No*. With *Yes* selected, the following command is added to the script file (rangerxl.do) that controls the routing of the design.

miter (slant 50) (bend 50 25)

Refer to the Specctra Reference Guide for full details on this command, but basically when routing is complete, right-angled tracks have their corners replaced with chamfers.

Use EMC attributes selections are *Yes* or *No*. With *Yes* selected, any attributes added to connections or pins on the schematic using the *Tools > EMC Attribute Tags* will be included in the Specctra design file. The router will then attempt to comply with these commands.

With *No* selected, none of the EMC Net or Pin attributes will be output in the design file.

Allow standard vias selections are *Yes* or *No*. With *Yes* selected, code 0 round pads will be included in the output file produced, for use by the Specctra auto-router as vias. If *No* is selected, then code 0 round pads will not be included in the output file, so therefore these vias could not be used.

(Vias defined in the Configuration folder (*Via Hole Definitions*) that have the *Enable for routing* box ticked will always be included in the output file produced.)

Clearances and Grids

- Edge clearance* enter the default minimum clearance required between copper items on the artwork and the board profile.
- Track clearance* enter the default minimum clearance required between adjacent copper items on the artwork. Specific clearance requirements added to the schematic will over-ride this default.
- Via grid* Defines the minimum pitch that vias can be placed on. Unless there are specific reasons to choose a different via grid, use the same grid as defined for the tracking grid. Refer to the Specctra Reference Guide for details.
- Tracking grid* Defines the minimum pitch that tracks can be placed on. Because Specctra is a shape-based router, it would be unwise to restrict it with a course tracking grid. Refer to the Specctra Reference Guide for details on the size of tracking grid to use.

Pass Counts

- Fanout passes* supply the number of fanout passes required. Refer to the Specctra Reference Guide for details. As a rule of thumb, 0 fanout passes should be specified for boards with less than 6 layers and 5 passes for boards with 6 layers and above.
- Routing passes* supply the number of routing passes required. Refer to the Specctra Reference Guide for details. Typically 50 are specified.
- Cleaning passes* supply the number of cleaning passes required. Refer to the Specctra Reference Guide for details. Typically 2 are specified.
- Number of re-routes* supply the number of re-route passes required. The number entered, multiplies the number of routing passes and cleaning passes performed in sequence. Refer to the Specctra Reference Guide for details. Try 3 to start with, but this may need to be increased depending on board complexity.

Via Holes Under SMD Pads

The following settings can be selected:

- Never allowed* vias are not allowed within any SMD pads.
- Allowed on all outlines* vias are allowed in all SMD pads.
- Only on outlines with "Allow vias at SMD" property*
vias are only allowed in SMD pads who have the "Allow vias at SMD pads" setting ticked in the outline's properties window (*Component Outline editor, View > Properties*).

Layer routing strategy

Each layer is listed, with its current layer assignment shown. Only layers defined as copper layers can be altered. The preferred routing direction for each copper layer should be selected.

The layer can be set to *Disabled* if it should not be used.

Starting the router

Once the setup window has been set as required, the router can be run automatically from within XL Designer with no further intervention by the user, or the output files can be produced for manual submission to the Specctra autorouter.

Automatic method

This is the easiest way to run the router and requires little knowledge of the Specctra software.

Once the settings in the window have been set as required, select the *Run Router* button.

When this option is chosen, a design file (.dsn) and script (.do) file are extracted from XL Designer, the Specctra

software is run and both files submitted to the router automatically. When routing is complete, the tracking is automatically fed back into the artwork design in XL Designer.

If *Run Router* is selected, messages appear in the window to indicate what is happening, then the auto-router starts.

If errors occur and the router doesn't start, close the Specetra interface window, then open the *Logfiles* folder from the navigator and select the file *specetra_export.txt*. This file supplies information that will be helpful in working out why the router didn't start, which can be for a variety of reasons.

When the auto-router has finished, the results are saved, Specetra is closed down and the results fed back into the XL Designer design.

When the artwork is being converted back into XL Designer, messages appear on the screen to indicate what is happening. If any errors or warnings are reported, they should be investigated and action taken where necessary. The errors and warnings are described in a file, which can be viewed by opening the *Logfiles* folder, then selecting the file *specetra_import.txt*.

The *Run Interactively* button starts the Specetra auto-router in the same way, but when routing has finished, Specetra remains open and the design is not fed back into XL Designer automatically.

Note: If the router is stopped manually, for instance **stop** is entered at the router prompt, the results are not saved. To force a save, type in the following command in response to the router prompt:

```
do $xlsave.do
```

The results are saved. Use the *Import* button to bring the routed tracks back into Ranger.

Manual Method

This method should be used if you wish to modify the design file or script file that Specetra will use, or the Specetra auto-router software is on a different machine.

The layout has to be extracted in the correct format and submitted to the router manually.

The layout is extracted using the *Export* button, at which time three files are produced. The design file (*xldesigner.dsn*) and two script files (*xldesigner.do* and *xlsave.do*) are placed in the folder specified by the *Installation* button.

These files should be submitted to the Specetra auto-router for routing. Refer to the Specetra documentation for details on how to do this.

The file *xldesigner.do* contains the commands as defined by the Specetra setup window within XL Designer. The file can be modified as required.

Sample xldesigner.do file:

```
# Script file for file c:\specetra\rxl\designs\rangerxl.dsn
bestsave on $best.w
status_file $rangerxl.sts
bus diagonal
route 50
clean 2
route 50 16
clean 2
route 50 16
clean 2
center
spread (extra 5 25)
miter (slant 50) (bend 50 25)
write wire $rangerxl.w
write routes $rangerxl.rte
report status $rangerxl.sts
```

If any warnings or errors are reported when the file is being exported from XL Designer, the problems should be investigated. Close the Specetra interface window, then open the *Logfiles* folder, select the file *specetra_export.txt*. Take any action required, then export the data again if required.

Once the Specetra router has completed routing the design and saved the results, a file called *xldesigner.rte* will have been created. This should be moved if necessary to the same folder that the files were originally exported to. This file has to be imported back into XL Designer; so within XL Designer, open the design from which the file was exported and open the Specetra auto-router interface. Select the *Import* button.

When the artwork is being imported, messages appear on the screen to indicate what is happening. If any errors or warnings are reported, they should be investigated and action taken where necessary. The errors and warnings are described in a file called *specetra_import.txt*, which can be viewed from the *Logfiles* folder.

Specetra configuration window

Before submitting a design to the Specetra auto-router, this window has to be set to match the Specetra

installation to be used.

From the Specctra interface window, select *Installation*, the window shown in Figure 210 appears.

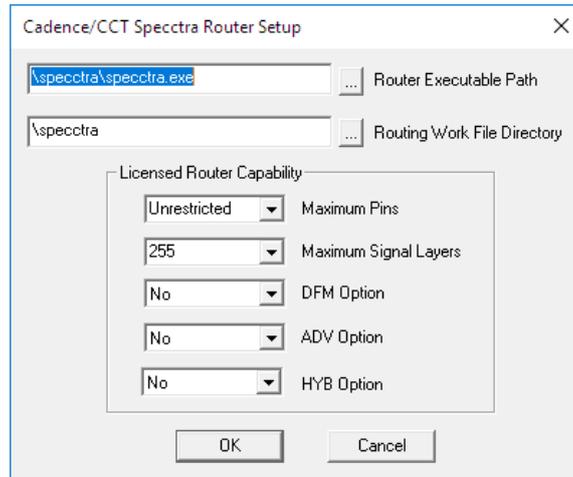


Figure 210

<i>Router executable path</i>	this path should be set to the location where the Specctra executable file is located. Specctra versions 6 and 7, path is typically: \specctra\bin\specctra.exe Specctra version 8, path is typically: \cct_cds\tools\SPECCTRA\bin\specctra.exe If this path is incorrect, then Ranger will be unable to start the Specctra auto-router when <i>Run Router/Run Interactively</i> is selected.
<i>Routing Work File Directory</i>	this path can be set to any folder. The folder MUST exist prior to selecting <i>Run/Export</i> . XL Designer will place the files it creates for the Specctra router in this folder. It also expects to find the Specctra files that should be imported, in this folder.
Licensed router capability	The Specctra auto-router comes with different specifications and options enabled. In order that XL Designer only creates files that are within the specification of the auto-router, the following settings should be set correctly. Refer to the Specctra license file for details on your router's specification/options.
<i>Maximum pins</i>	the maximum number of component pins that this Specctra can handle.
<i>Maximum signal layers</i>	the maximum number of signal layers that this Specctra can handle - power plane/split power planes are not included in this count.
<i>DFM Option</i>	whether or not the DFM (Design for Manufacture) option has been enabled.
<i>ADV option</i>	whether or not the ADV (Advanced) option has been enabled.
<i>HYB option</i>	whether or not the HYB (Hybrid) option has been enabled.

Electra auto-router interface

XL Designer comes with a two-way interface to the Electra auto-router. The Electra auto-router is an optional extra that can be purchased if required.

The interface allows an artwork layout to be output from XL Designer in a format that the Electra auto-router software can accept. Once the Electra software has auto-routed the design, its output file can be converted back into XL Designer for further design work and checking as required.

The artwork design that is submitted to Electra can include keep-out data (Electra will not cross a keepout area) and pre-routed fixed or un-fixed tracks.

The Electra auto-router will use user-defined via stacks if they are defined and enabled for use.

Circular board profiles are converted into a polygon with 128 segments (2 degree steps).

To access the Electra interface, it should first of all be enabled (File > System setup window).

Then, open the design to be submitted to the Electra auto-router. Right-click on *Artwork* from the navigator, then select *Optional Autorouters > Konekt Electra router*. A window similar to the one in Figure 211 appears.

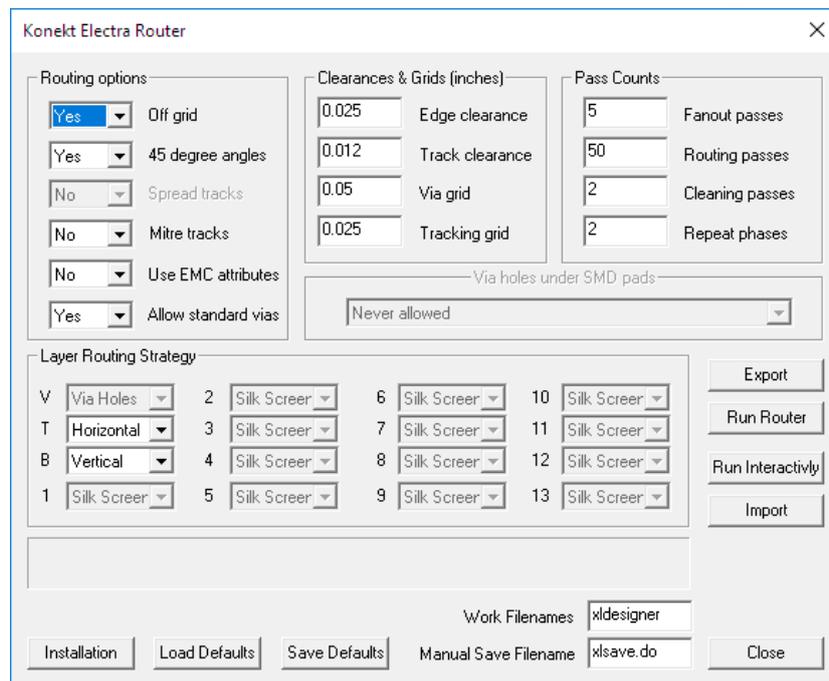


Figure 211

Electra router setup

Before submitting the design to the auto-router, the settings in this window should be set as required. The window is divided into areas that control the options that are included in the file that is output for the Electra auto-router. Use the following information to decide how it should be set.

If the interface has not been used before, select the *Installation* button and ensure the window is set to match the Electra auto-router to be used.

Routing options

- Off grid* selections are *Yes* or *No*. With *Yes* selected, tracks can be routed off the chosen tracking grid where necessary. Typically set to *No*.
- 45 degree angles* selections are *Yes* or *No*. With *Yes* selected, the following command is added to the script file (rangerxl.do) that controls the routing of the design.
bus diagonal
If enabled, 45 degree tracks are permitted in tracks. Typically set to *Yes*.
- Spread tracks* currently not supported by Electra.
- Mitre tracks* selections are *Yes* or *No*. With *Yes* selected, the following command is added to the script file (rangerxl.do) that controls the routing of the design.
recorner pin 200
recorner slant 500
recorner bend 400

When routing is complete, right-angled tracks have their corners replaced with chamfers.

Use EMC attributes selections are *Yes* or *No*. With *Yes* selected, any attributes added to connections or pins on the schematic using the *Tools > EMC Attribute Tags* will be included in the Electra design file. The router will then attempt to comply with these commands.

With *No* selected, none of the EMC Net or Pin attributes will be output in the design file.

Allow standard vias selections are *Yes* or *No*. With *Yes* selected, code 0 round pads will be included in the output file produced, for use by the Electra auto-router as vias. If *No* is selected, then code 0 round pads will not be included in the output file, so therefore these vias could not be used.

(Vias defined in the Configuration folder (*Via Hole Definitions*) that have the *Enable for routing* box ticked will always be included in the output file produced.)

Clearances and Grids

Edge clearance enter the default minimum clearance required between copper items on the artwork and the board profile.

Track clearance enter the default minimum clearance required between adjacent copper items on the artwork. Specific clearance requirements added to the schematic will over-ride this default.

Via grid Defines the minimum pitch that vias can be placed on. Unless there are specific reasons to choose a different via grid, use the same grid as defined for the tracking grid.

Tracking grid Defines the minimum pitch that tracks can be placed on. Because Electra is a shape-based router, it would be unwise to restrict it with a coarse tracking grid.

Pass Counts

Fanout passes supply the number of fanout passes required. As a rule of thumb, 0 fanout passes should be specified for boards with less than 6 layers and 5 passes for boards with 6 layers and above.

Routing passes supply the number of routing passes required. Typically 50 are specified.

Cleaning passes supply the number of cleaning passes required. Typically 2 are specified.

Number of re-routes supply the number of re-route passes required. The number entered, multiplies the number of routing passes and cleaning passes performed in sequence. Try 3 to start with, but this may need to be increased depending on board complexity.

Via Holes Under SMD Pads

The following settings can be selected:

Never allowed vias are not allowed within any SMD pads.

Allowed on all outlines vias are allowed in all SMD pads.

Only on outlines with "Allow vias at SMD" property
vias are only allowed in SMD pads who have the "Allow vias at SMD pads" setting ticked in the outline's properties window (*Component Outline editor, View > Properties*).

Layer routing strategy

Each layer is listed, with its current layer assignment shown. Only layers defined as copper layers can be altered. The preferred routing direction for each copper layer should be selected.

The layer can be set to *Disabled* if it should not be used.

Starting the router

Once the setup window has been set as required, the router can be run automatically from within XL Designer with no further intervention by the user, or the output files can be produced for manual submission to the Electra autorouter.

Automatic method

This is the easiest way to run the router and requires little knowledge of the Electra software.

Once the settings in the window have been set as required, select the *Run Router* button.

When this option is chosen, a design file (.dsn) and script (.do) file are extracted from XL Designer, the Electra software is run and both files submitted to the router automatically. When routing is complete, the tracking is automatically fed back into the artwork design in XL Designer.

If *Run Router* is selected, messages appear in the window to indicate what is happening, then the auto-router

starts.

If errors occur and the router doesn't start, close the Electra interface window, then open the *Logfiles* folder and select the file *electra_export.txt*. This file supplies information that will be helpful in working out why the router didn't start, which can be for a variety of reasons.

When the auto-router has finished, the results are saved, Electra is closed down and the results fed back into the design.

When the artwork is being converted back into XL Designer, messages appear on the screen to indicate what is happening. If any errors or warnings are reported, they should be investigated and action taken where necessary. The errors and warnings are described in a file, which can be viewed by opening the *Logfiles* folder and viewing the file *electra_export.txt*.

The *Run Interactively* button starts the Electra auto-router in the same way, but when routing has finished, Electra remains open and the design is not fed back into XL Designer automatically.

Note: If the router is stopped manually, the results are not saved. To force a save, execute the *xlsave.do* command from within Electra. Use the *Import* button to bring the routed tracks back into Ranger.

Manual Method

This method should be used if you wish to modify the design file or script file that Electra will use, or the Electra auto-router software is on a different machine.

The layout has to be extracted in the correct format and submitted to the router manually.

The layout is extracted using the *Export* button, at which time three files are produced. The design file (*xldesigner.dsn*) and two script files (*xldesigner.do* and *xlsave.do*) are placed in the folder specified by the *Installation* button.

These files should be submitted to the Electra auto-router for routing. Refer to the Electra documentation for details on how to do this.

The file *xldesigner.do* contains the commands as defined by the Electra setup window within XL Designer. The file can be modified as required.

Sample xldesigner.do file:

```
# Script file for file c:\Program Files\electra\rxldesigns\rangerxl.dsn
bestsave on
bus diagonal
route 50
clean 2
route 50 16
clean 2
route 50 16
clean 2
recorner diagonal
write routes c:\Program Files\electra\rxldesigns\rangerxl.rte
write session c:\Program Files\electra\rxldesigns\rangerxl.ses
status_file
```

If any warnings or errors are reported when the file is being exported from XL Designer, the problems should be investigated. Close the Electra interface window, then open the *Logfiles* folder and view the file *electra_export.txt*. Take any action required, then export the data again if required.

Once the Electra router has completed routing the design and saved the results, a file called *xldesigner.rte* will have been created. This should be moved if necessary to the same folder that the files were originally exported to. This file has to be imported back into XL Designer; within XL Designer, open the design from which the file was exported and open the Electra auto-router interface. Select the *Import* button.

When the artwork is being imported, messages appear on the screen to indicate what is happening. If any errors or warnings are reported, they should be investigated and action taken where necessary. The errors and warnings are described in a file called *electra_import.txt*, which can be viewed via the *Logfiles* folder.

Electra configuration window

Before submitting a design to the Electra auto-router, this window has to be set to match the Electra installation to be used.

From the Electra interface window, select *Installation*, the window shown in Figure 210 appears.

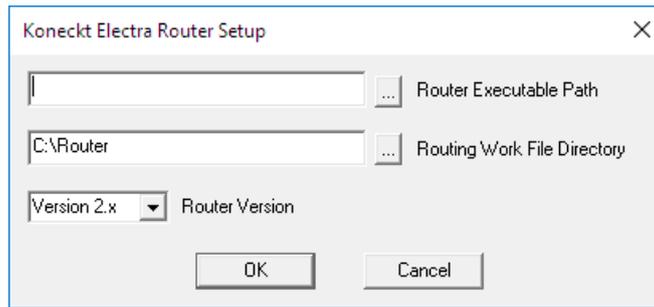


Figure 212

- Router executable path* this path should be set to the location where the Electra executable file is located. Select the browser button alongside the path name to assist in the selection of the correct path.
Path is typically: c:\Program Files\electra\electra.exe
If this path is incorrect, then Ranger will be unable to start the Electra auto-router when *Run Router/Run Interactively* is selected.
- Routing Work File Directory* this path can be set to any folder. The folder MUST exist prior to selecting *Run/Export*. XL Designer will place the files it creates for the Electra router in this folder. It also expects to find the Electra files that should be imported back, in this folder.
Select the browser button alongside the path name to assist in the selection of the correct path.
- Router version* There are two versions (1 and 2) of the Electra autorouter. Select the version you will be using.

Importing Files

The following importers are available within Seetrax XL Designer:

Parts & Wiring List Import	to import parts/wiring lists for artwork design
Gerber Format File Import	to import gerber files for checking/design regeneration
AutoCAD DXF File Input	to import board profile definitions

The importers are each described separately in the following chapters. They are accessed once a design has been opened, by right-clicking the design name from the Navigator pane, then selecting *Import* followed by the import routine required.

Gerber Format File Import

The Gerber file import routine is used to re-create Gerber files in the artwork editor. Once imported, the data can be:

- visually checked to ensure the Gerber file(s) contain the correct type of information.
- modified and new Gerber files produced
- used to re-create a complete design (from another CAD system) for further design work

Information:

A file in the Gerber format is required. XL Designer can accept various Gerber formats and the following details will be required by XL Designer before the files can be imported accurately:

- the number of decimal places in the co-ordinates
- whether the co-ordinates are in incremental or absolute format
- the X and Y shifts used when outputting the data
- which DCodes have been used, the size and shape assigned to each DCode

If this information is not available the resultant input may be incorrect. It may take 3 or 4 attempts at importing the file before success is achieved - if the size/shape assigned to the dcodes is unknown, then the pad and track sizes on the imported artwork are highly likely to be incorrect.

An understanding of the Gerber file format can explain why some problems occur when importing the gerber files - sometimes the importer does not produce the results expected. (Refer to the chapter on gerber output for brief details on the gerber file format.)

If the gerber file contains a header, the importer ignores the information within the header – this means the dcode table in the importer setup window has to be defined manually.

Procedure:

Open the design that the Gerber files will be imported into.

Note: when working on an existing design, it is suggested that a backup copy of the design is taken before importing the file - the results may not be as expected at the first attempt.

Right-click the design name from the Navigator pane, then select *Import > Gerber photoplot data*. A window similar to the one in Figure 213 appears:

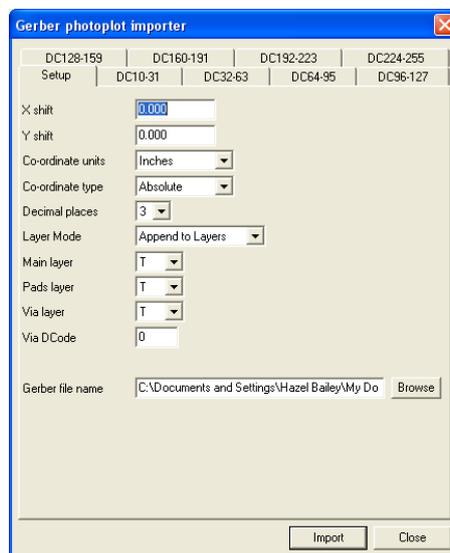


Figure 213

This window controls the Gerber import and should always be checked before proceeding with each Gerber file as it is unlikely that any two Gerber files will use the same setup parameters. Change it as required, using the following information to assist you:

Gerber Photoplot Importer window, Setup tab window settings:

X and Y shift: the Gerber files contain X/Y co-ordinates. The co-ordinates can be positive or negative numbers. In order to see the regenerated artwork in XL Designer, the co-ordinates must be positive and within XL Designer's working area (0 to 99" in the X & Y axes). The values

entered here are added to all the co-ordinates in the gerber file as they are imported (the gerber file is not modified).

Inspect the co-ordinates within the gerber file and calculate the values required for the shifts. Alternatively experiment with different values until the data appears in the artwork editor.

- Co-ordinate units:** Gerber files can be produced in either Imperial or Metric units (usually Imperial). The importer needs to know which units are being used so that the data is regenerated at the correct scale.
If the data is imported, but it is too big or small by a factor of 2.54, then it's likely that the co-ordinate units was set incorrectly.
- Co-ordinate type:** Gerber files can be produced in either absolute or incremental format. The importer needs to know which format to expect. (Absolute format gives co-ordinates with respect to a specific datum whilst incremental format gives co-ordinates with respect to the last co-ordinate given.)
- Decimal Places:** the Gerber file typically contains co-ordinates containing five figures without decimal points. The importer needs to know how many decimal places there are in each co-ordinate to ensure the artwork appears at the correct scale.
If your data is imported, but it is too big or small by a factor of 10, then it is likely that the number of decimal places was set incorrectly.
- Layer mode:** the imported data is placed on a user-specified layer in the artwork editor. The data can be added to the existing data on the layer (select *Append to Layers*) or it can replace the data (select *Replace layers*).
- Main Layer:** the layer number specified here specifies which layer the Gerber data will be placed on, with the exception of flashed apertures ("*pads*" and "*vias*").
- Pads layer:** the flashed apertures ("*pads*") within the Gerber file will be added to the layer specified here. Seetrax XL Designer recognises "*pads*" in a Gerber file as follows:
any shaped DCode that is flashed
any rectangular shaped DCode that is drawn and makes a one segment orthogonal line
- If the Gerber files are being imported for a visual check prior to plotting, add the contents of each Gerber file to one artwork layer (*Main layer*, *Pads layer* and *Via Layer* set to the same layer).
When the artwork is viewed, the layer will correspond to the contents of the Gerber file. Each separate Gerber file should be added to a different layer.
If the Gerber files are being imported to regenerate a design, the data within the file would probably be split on to different layers – refer to *Regenerating a design from Gerber files* for more details.
- Via Layer:** any pads produced by a flashed aperture using the dcode assigned for vias will be added to the layer specified here. If they are added to layer V, XL Designer will recognise them as vias.
"*Vias*" within a Gerber file are recognised purely by their DCode number, indicated by the next parameter.
If the Gerber files are being imported for a visual check prior to plotting, add the contents of each Gerber file to one artwork layer (*Main layer*, *Pads layer* and *Via Layer* set to the same layer).
When the artwork is viewed, the layer will correspond to the contents of the Gerber file. Each separate Gerber file should be added to a different layer.
If the Gerber files are being imported to regenerate a design, the data within the file would probably be split on to different layers - refer to *Regenerating a design from Gerber files* for more details.
- Via DCode:** any flashed apertures in the Gerber file with the DCode number specified here, will be assigned to round, code 0 in the pad sizes table. XL Designer will then regard them as vias. This facility is useful when re-generating designs.
If a design is being imported for checking purposes only, this can be left at 0.
- Gerber filename:** specify the name of the Gerber file that will be imported. The browse button can be used to locate folders and files.

Gerber Photoplot Importer window, Dcode tab window settings:

When the **DC 10-31** tab is selected from the top of the Gerber Photoplot Importer window, a window similar to the one shown in Figure 214 appears. This is the Dcode table for the incoming gerber file and should be set to suit the incoming gerber file.

Dcodes from 10 to 255 can be defined by selecting the appropriate tab from the top of the window.

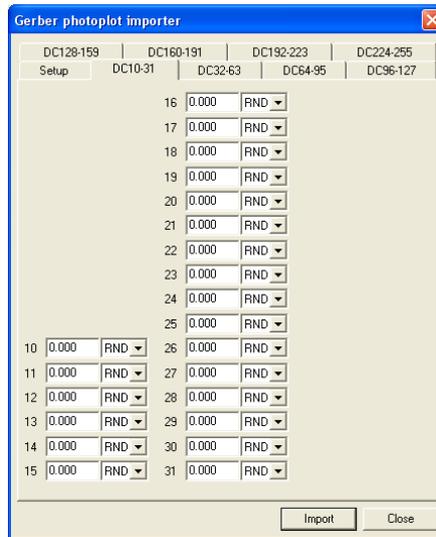


Figure 214

The Gerber file contains DCodes which were defined when the Gerber file was created. Each DCode represents a specific size and shape of aperture, which in turn represents pads and tracks on the imported artwork. It is essential that the import DCode table is set up to exactly match the original DCode table used when creating the Gerber files. If it isn't, the regenerated pads and tracks will probably be the incorrect size or shape, although they will be in the correct relative places.

The design's pad and track size tables are updated automatically to include the sizes required when the file is imported.

Note: even if the Dcode table is held within the gerber file being imported, XL Designer cannot use it and the importer's Dcode table has to be filled in.

Dcode nos: DCode numbers are listed, 10 to 31 in the example shown above. They represent valid DCode numbers and cannot be changed. The correct size and shape should be assigned to each DCode number in this table to match those used in the Gerber file. Dcodes 32 to 255 are available by selecting the appropriate tab from the top of the window.

Size: enter the size that should be assigned to the Dcode.

Sizes can be entered in inches or millimetres, the decimal point indicates which units are in current use (a dot (.) for Imperial values and a comma (,) for Metric values).

Shape: select the shape that should be assigned to the Dcode. Only Round or Square is available. (If other shapes were used in the gerber file, they can only be imported as round or square - it is possible to replace pads from within the artwork editor.)

Once the Setup and DCode windows have been setup as required, select the *Import* button to start the import process.

Information appears on the screen to indicate what is happening, as shown in Figure 215. This information will be stored in the report log.



Figure 215

Problems encountered during the import are separated into two types, Warnings and Errors - errors are generally more serious than warnings.

Each problem encountered is reported in detail and each one must be investigated and corrected where necessary - this might involve editing the Dcode table or changing the parameters in the import setup window, then importing the file again.

The first part of the report indicates when the importer was started and which file was used as input.

The report continues and two warnings are given in this example.

The first warning will always appear if the *Via Dcode* in the “Import setup” window is set to 0. No action is required unless it is required that a particular Dcode should be converted to a via, in which case the import would have to be started again with the correct Dcode assigned for vias.

A second warning is given which indicates that Dcode 12 has been used in the gerber file but has not been assigned a size in the Dcode table.

Note: although the message indicates the dcode was not assigned correctly, XL Designer does not know what the correct sizes are, simply that no size has been assigned.

Any used Dcodes that have not been assigned a size in the Dcode table will be imported as a code 4 round pad or track.

The report finishes with a statement indicating the import is complete and the artwork has been saved.

All the errors/warnings should be investigated and appropriate action taken.

Close the report window when ready to continue. The import process is complete.

The report can be re-viewed by opening the *Logfiles* folder from the navigator and viewing the report named “import_gerber”.

Viewing/editing the imported artwork

Once the file has been imported and the report acted upon as necessary, the imported data can be displayed from the artwork editor (or another Gerber file could be imported).

It's a good idea to view the artwork after the first file has been imported, this will ensure the correct shifts, layers, etc. were used.

If the import was unsuccessful investigate what went wrong using the following information to assist you.

Typical problems when importing gerber files

The data cannot be seen when the artwork editor is opened:

- use *View > Zoom out* until the whole working area can be seen, the grid (*Grid > On, Grid > Inch or Metric*) will indicate the working area. (*View > Full* does not operate without a board profile.)
- check the appropriate layers are visible (*View Control* dialogue bar)
- check the number of points being used on the appropriate layer(s) (*View Control* dialogue bar or the *Artwork Properties* pane). If the layer contains some points but the data is not visible after trying the above, it is likely that the data is outside of the working area. Import the data again with different shifts until the data is found. Zero points indicate there is nothing on the layer.
- open the gerber file (using Notepad/Wordpad or similar) and verify it is a valid Gerber file (a sample file is shown below)

All the pads appear to be too big, and too close together:

- check the number of decimal places in the Gerber file. Ensure that the Gerber input Setup window has been set to the same number of decimal places.
- check whether the Gerber co-ordinates are in Imperial or Metric units. Ensure that the Gerber input Setup window has been set to the correct units.

All the pads appear to be too small, and too far apart:

- check the number of decimal places in the Gerber file. Ensure that the Gerber input Setup window has been set to the same number of decimal places.
- check whether the Gerber co-ordinates are in Imperial or Metric units. Ensure that the Gerber input Setup window has been set to the correct units.

Some pads are too small, or too big, or the wrong shape:

- Check the importer's Setup, DCode table. Ensure the DCode numbers correspond to the correct size and shape as originally defined in the gerber file.

Note: Dcodes (pads) that are flashed in the gerber file can only be set to round or square when the file is imported, so other shaped, flashed Dcodes (pads) can only be imported as round or square.

Sample Gerber file

Command	Brief description
G01X00000Y00000D02*	initialise

G54D37*	select aperture D37
G01X1000Y5000D02*D03*	go to x=1", y=5" and flash
G01X5000Y5000D02*D03*	go to x=5", y=5" and flash
G01X5000Y1000D02*D03*	go to x=5", y=1" and flash
G54D58*	select aperture D58
G01X2000Y2000D02*	go to x=2", y=2" and draw, continue
G01X4000Y2000D01*	to x=4", y=2" continue to
G01X4000Y4000D01*	x=4", y=4" continue to
G01X2000Y4000D01*	x=2", y=4" continue to
G01X2000Y2000D01*	x=2", y=2", turn light off
G54D12*	select aperture D12
G01X2400Y3200D02*	go to x=2.4", y=3.2" draw, continue
G01X2400Y2800D01*	to x=2.4", y=2.8", turn light off
G01X2700Y2800D02*	go to x=2.7", y=2.8", draw, continue
G01X2700Y2600D01*	to x=2.7", y=2.6", turn light off
G01X2800Y3200D02*	go to x=2.8", y=3.2", draw, continue
G01X3300Y3200D01*	to x=3.3", y=3.2", continue to
G01X3300Y2200D01*	x=3.3", y=2.2", continue to
G01X2800Y2200D01*	x=2.8", y=2.2", continue to
G01X2800Y2300D01*	x=2.8", y=2.3", turn light off
G54D60*	select aperture D60
G01X2400Y2300D02*D03*	go to x=2.4", y=2.3" flash
G01X2500Y2300D02*D03*	go to x=2.5", y=2.3" flash
G01X2600Y2300D02*D03*	go to x=2.6", y=2.3" flash
G01X2700Y2300D02*D03*	etc., etc.
G01X2800Y2300D02*D03*	
G01X2900Y2300D02*D03*	
G01X2500Y2600D02*D03*	
G01X2400Y2600D02*D03*	
G54D65*	go to x=2.4", y=2.6" flash
G01X3100Y2350D02*D03*	select aperture D65
G01X3100Y2550D02*D03*	go to x=3.1", y=2.35" flash
G01X2400Y3200D02*D03*	go to x=3.1", y=2.55" flash
G01X2800Y3200D02*D03*	go to x=2.4", y=3.2" flash
G54D55*	go to x=2.8", y=3.2" flash
G01X2400Y2800D02*D03*	select aperture D55
G01X2700Y2800D02*D03*	go to x=2.4", y=2.8" flash
M02*	go to x=2.7", y=2.8" flash
	stop

Regenerating a design from Gerber files

Please read the following information carefully as it describes assumptions made during Gerber import, which will affect the regenerated design in the artwork editor.

Import Assumptions:

- flashed DCodes produce pads
- any round, drawn DCodes produce tracks (round ended finger pads that were created with a draw in the Gerber file will be imported as tracks)
- any square, drawn Dcodes that produce a one segment, orthogonal line, produce rectangular pads
- vias were assigned a unique DCode, i.e. the DCode assigned to vias was not used for any other component pads. If this is not the case it will be impossible for XL Designer to differentiate between component pads and vias, which will cause problems when modifying tracks.

Gerber file content and implications for importing:

The Gerber files for a double sided board would typically have been produced to include the following data:

- File 1** copper (tracks and text) from top component side
component pads from top component side
vias
alignment markers
- File 2** copper (tracks and text) from bottom component side
component pads from bottom component side
vias
alignment markers
- File 3** silk screen outlines and text
alignment markers
- File 4** solder mask
alignment markers

To allow the re-created design to be worked upon and checked in the normal way, it is essential the imported data appears correctly within XL Designer. Some of the Gerber data will become redundant.

For example, in the Gerber files listed above, the same vias appear in files 1 and 2. If both sets of vias are imported onto the XL Designer via layer, duplicate via information will be produced. Similarly, XL Designer must obtain component pad information from the outline library so that it has pin numbering information, hole sizes, etc. This will make the pad data in the Gerber files redundant.

If the data is imported without thought, much manual editing may need to be performed to reduce the errors produced when the board is checked.

The following parameters within the Setup window allow the imported data to be added in a way that allows duplicated or unrequired data to be separated onto different layers and then removed using the Window delete command.

Main layer: Indicates which layer drawn items (tracks, text, round-ended-finger pads) will appear on.

Pads layer: Indicates which layer flashed items will appear on, with the exception of "vias".

Vias layer: Indicates which layer "vias" will appear on.

Via Dcode: Indicates the unique Dcode used for vias. If a unique DCode was not used for vias, every pad with the DCode specified will appear as a via in the regenerated design. These "extra" vias would obviously need to be removed from the artwork.

If a DCode is not specified for vias, then the vias will be imported, but XL Designer will not recognise them as vias.

If importing the example Gerber files listed above, the parameters would typically be set as follows for each file:

File 1:	Main layer	T
	Pads layer	11
	Via layer	V
	Via DCode	23 (assuming vias were output using D23)
File 2:	Main layer	B
	Pads layer	12
	Via layer	12 (because the vias from file 1 will be used)
	Via DCode	23 (assuming vias were output using D23)
File 3:	Main layer	1
	Pads layer	13
	Via layer	13
	Via DCode	23 (assuming vias were output using D23)
File 4:	Main layer	2
	Pads layer	14
	Via layer	14
	Via DCode	23 (assuming vias were output using D23)

The pads in every case were placed on a different unused layer ready for removal later. They are not required as the component pads will appear automatically when a parts list is added to the data. As these pads will be removed, they could have all been placed on the same unused layer, but this approach will help the operator determine which pads belong to which Gerber file in the event of a query.

One set of vias was added to layer V, the other sets were added to an unused layer for removal later.

If power plane layers are included in the design it would be better if they were re-generated in XL Designer.

However, if being re-generated from the gerber file, it is likely that the "pad" and "track" data should be imported

on to the same layer within XL Designer. The layer should be assigned as a power plane layer either before or after the Gerber files are imported.

A board profile was not included in the Gerber files. If it was, it would appear as a TRACK on the *main layer* used when the file was imported. This "track" could be used as a guide to define the real board profile.

Additional Data required to regenerate a design:

Once the Gerber files have been imported into XL Designer, the following data has to be added:

- Parts list
- Net list
- Physical outlines

If the parts and net list data exists as a text file(s), it should be possible for the operator to manipulate the text so that it is in a format suitable for text importer. If not, the data will have to be entered by hand using the XL Designer parts/net list text editor, or extracted from a schematic.

Parts List

The parts list should include for each part, its position, orientation and reference to a physical outline.

The positional information in the parts list is with respect to 0,0 in the artwork editor. The Gerber data may have been created, or imported with a shift. It is essential that the parts and Gerber data line up. Use the *Identify > Pad* command in the artwork editor to obtain the position of a known pad on the imported artwork, for instance IC1, pin 1. This can then be cross referenced to the parts list (the pin's offset from the component datum will have to be taken into account).

Net List

The net list should contain every inter-connection required on the design.

Physical outlines

The outline library should contain an outline for all the outlines referred to in the parts list. The outline should include pad/hole position and size, pin numbers, part size, part datum, etc.

The co-ordinates given in the parts list refer to the datum of each outline. Note: different systems use different datum points within outlines. If the positional information was extracted automatically from a layout, it is essential that the datum point of each outline be known.

Verifying the design:

Once the parts/net lists and outlines have been added view the artwork. Check that the artwork "looks" correct. For instance the parts aren't offset from the tracks, that the component pads are the same size/shape as the pads added to the unused layers. If the data doesn't line up, take the appropriate action. For example the artwork could be deleted, leaving the parts and net list behind, and the Gerber files re-imported using different shifts.

The point-point connections between the parts will appear even though the routed tracks are there. As yet XL Designer will not recognise the "tracks" as being valid. Running the artwork checks will rectify this providing the data is correct.

If the artwork appears to be correct, delete the data that was added to unused layers during the Gerber import (layers 11 to 14 in the example given above) using the *Region > Delete* command in the artwork editor.

All the layers within the design will be defined as copper layers. If silkscreen or power plane layers have been imported assign the layers appropriately (*Configuration* folder, *Layer assignments & Ordering*) otherwise the checking routines will flag problems that do not exist.

Once the above have been checked, run the artwork checking routine from the artwork editor (*Check > Connectivity*). Investigate and rectify any problems reported. The design is then ready for further work.

AutoCAD DXF File Input

DXF is an ASCII file format that allows data to be transferred between CAD systems.

When imported to XL Designer, DXF data appears in the profile editor.

This allows complex profiles saved in a .DXF file format to be used and edited in XL Designer's board profile editor.

DXF data files can also be extracted from XL Designer, refer to the chapter describing the Outputs for details.

Information:

A file in the DXF ascii format is required. It should be output from AutoCAD in the "entity" mode.

The DXF data can only be imported into the profile editor. However once imported, it could be transferred to a documentation layer within the artwork editor using the output tools.

Procedure:

Open the design that the DXF file will be imported into.

Note: when working on an existing design, it is suggested that a backup copy of the design is taken before importing the file - the results may not be as expected at the first attempt.

Right-click the design name from the Navigator pane, then select *Import > Profile from AutoCAD DXF file*. A window similar to the one in Figure 216 appears:

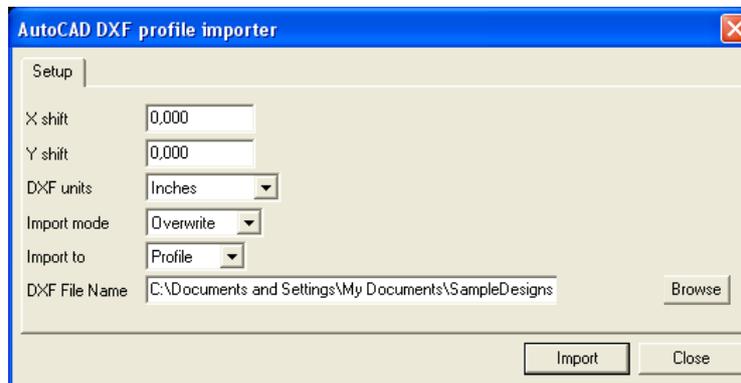


Figure 216

X shift and Y shift these settings allow the data being imported, to be positioned wherever required in the profile editor.

If the shifts required are unknown, import the data into a copy of the design with the shifts set to 0, and view the board profile. Calculate whether or not shifts are required and what they are, and then import the data into the original design, with the appropriate shifts.

DXF Units: XL Designer can import DXF files containing co-ordinates in inches, thousandths of inches, centimetres or millimetres. The importer needs to know which units are in use.

If the units specified are incorrect, the scale of the imported data will be incorrect.

Import Mode: the DXF data can either be added (select *Append*) to the existing data or it can overwrite (select *Overwrite*) the existing data.

Select the option required. (Only data within the same category is overwritten - see *Import to*.)

Import to: the DXF data being imported can be imported as either *Profile*, *Keepout* or *Router* data. Select the category the data should be imported to.

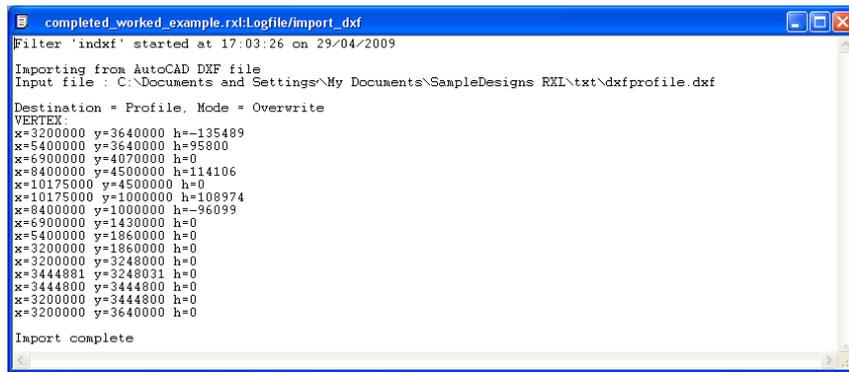
DXF Filename: specify the name of the DXF file that will be imported. The browse button can be used to locate folders and files.

Set the parameters as required, then select the *Import* button to start the import process.

A window indicates when the import is complete. Select *OK* to continue.

A conversion report will appear. It includes details of the conversion and details of any problems encountered. Once closed, the report can be re-viewed by opening the *Logfiles* folder from the navigator and viewing the report named "import_dxf".

A sample indxf.txt report is shown in Figure 217.



```
completed_worked_example.rxl:logfile/import_dxf
Filter 'indxf' started at 17:03:26 on 29/04/2009
Importing from AutoCAD DXF file
Input file : C:\Documents and Settings\My Documents\SampleDesigns RXI\txt\dxfprofile.dxf
Destination = Profile, Mode = Overwrite
VERTEX:
x=3200000 y=3640000 h=-135489
x=5400000 y=3640000 h=95800
x=6900000 y=4070000 h=0
x=8400000 y=4500000 h=114106
x=10175000 y=4500000 h=0
x=10175000 y=1000000 h=108974
x=8400000 y=1000000 h=-36099
x=6900000 y=1430000 h=0
x=5400000 y=1860000 h=0
x=3200000 y=1860000 h=0
x=3200000 y=3248000 h=0
x=3444981 y=3248031 h=0
x=3444800 y=3444800 h=0
x=3200000 y=3444800 h=0
x=3200000 y=3640000 h=0
Import complete
```

Figure 217

The first part of the report indicates when the importer started and the file used as input.

The report then indicates the destination of the data (profile, keepout or router) and whether the data was added to any existing data in the editor or whether any existing data was overwritten.

The details of the DXF file then follow, followed by *Import Complete* to indicate the report is complete.

If any errors or warnings are listed, these should be investigated and the appropriate action taken.

Once the file has been imported, the board profile may be viewed or modified using the profile editor – from the design's open folder in the navigator, select *Board Profile*.

Parts & Wiring List Import

This routine is used to import parts and/or wiring list files that exist in a specific format, into XL Designer. The following data will also be required, either before or after the parts/wiring list has been imported, in order to design the artwork:

- A board profile
- Details of which connections are power rails (unless specified in the file to be imported)
- The physical footprints (in the component outline library)
- Pad, track, etc, sizes
- If gate and pin swapping will need to be performed, schematic parts will also need to be defined in the design's schematic folder and the parts list updated with the correct *Type* field.

Parts/wiring list files can also be extracted from XL Designer, refer to the chapter describing the Parts & Net List folders for details.

Expected file format:

The text file can exist as one complete file containing three sections, or up to three separate files each containing one or more sections. The sections are:

Parts list
Wiring list
Power rails list

Whether each section is held in its own file, or all three sections combined in one file, they must be imported in the following order:

Parts section 1st or 2nd
Power rails section 1st or 2nd
Wiring section 3rd

Each section must start and end with a unique delimiter or keyword which can be chosen by the user. Data within a section must follow the format rules, with fields separated by a space or spaces.

The importer ignores lines added outside of a section.

Parts list file/section

This file/section should take the format described below. Fields cannot be omitted and there should be one line for each part.

Keyword1
Reference Type Outline X-Position Y-Position Rotation
Keyword2

where:

- Keyword1:** is replaced with a unique keyword to allow the start of the parts list section to be recognised within the file.
User defined keywords can be used, but the keywords are case sensitive.
The default parts list keyword is: **.PARTS**
- Reference:** is replaced with the name allocated as a unique reference for each component in the design, for instance IC1, R1, etc. It can contain up to 4 letters followed by a number, e.g. IC1, PLUG1, etc.
- Type:** this field will be accessed by the gate and pin swapping routines in the artwork editor, maximum of 32 characters.
If gate or pin swapping will not be performed on the package, this field could contain a stock number, catalogue reference, etc. An entry must be given, though one character will suffice.
If gate and pin swapping will be used, the field should take the form of a component name and value separated by a comma. For example:
PARTNAME,VALUE
R0.25W,10k
Both items should correspond to a schematic part. For instance, in the above example there must be a part in the design's schematic parts folder called R0.25W with a value field set to 10k.
Gate swapping between packages is only permitted if the type fields of the packages are identical.

- Outline:* is replaced with the name of the component outline (footprint) from the component outline library that will be used for the part. Maximum of 16 characters.
The outline does not have to exist in the component outline folder before importing the parts list, but should exist before designing the artwork.
- X & Y-Position:* are replaced with the position of the part's datum on the board, with respect to 0,0 in the artwork editor (lower left corner of working area, even if the datum in the artwork editor has been moved). Units can be entered in inches or millimetres, but not both.
It is usual to leave these fields set to 0 and position the parts interactively.
- Rotation:* This field indicates the orientation and layer of the parts on the board. It is usual to leave this column set to 0 and rotate or flip the parts interactively.
An entry of 0 represents the part as it was created in the outline library. Rotation is specified in degrees, anti-clockwise (0 to 359 degrees).
If the part has to be flipped (or mirrored), as well as rotated, F (for flipped) should precede the entry. The part is flipped (mirrored) about its original Y-axis, as defined in the outline library. For example, F 45 will flip and then rotate the part through 45 degrees.
- Keyword2:* is replaced with a unique keyword to allow the end of the parts list section to be recognised. User defined keywords can be used, but the keywords are case sensitive.
The default parts list keyword is: **.ENDPARTS**

Sample parts list file/section:

```
.PARTS
IC1 7400,74LS00 DIL14 0 0 0
IC2 7400,74LS00 DIL14 0 0 0
IC3 7402,74LS02 DIL14 0 0 0
IC4 74244,74LS244 DIL20 0 0 0
R1 RES,100k 1206R 0 0 0
R2 RES,100k 1206R 0 0 0
C1 CAP,100uF 0805R 6.3 4.2 F 90
C2 CAP,100uF 0805R 0 0 0
.ENDPARTS
```

Power Rails file/section

This file/section does not have to be included. However, if it is, it must be imported before the wiring list.

```
Keyword3
Powername
Keyword4
```

where:

- Keyword3:* is replaced with a unique keyword to allow the start of the power rails section to be recognised within the file. User defined keywords can be used, but the keywords are case sensitive.
The default parts list keyword is: **.POWERNAMES**
- Powername:* is replaced with the name of a connection in the wiring list that has to be treated as a power rail. Each separate powername should occupy one line in the file.
- Keyword4:* is replaced with a unique keyword to allow the end of the power rails section to be recognised within the file. User defined keywords can be used, but the keywords are case sensitive.
The default parts list keyword is: **.ENDNAMES**

Sample Power Rails file/section:

```
.POWERNAMES
VCC
GND
VEE
0V
+12V
.ENDNAMES
```

Wiring list file/section

This file or section should take the following format:

```
Keyword5
Signalname Node Node Node Node
Keyword6
```

where:

Keyword5: is replaced with a unique keyword to allow the start of the wiring list section to be recognised within the file. User defined keywords can be used, but the keywords are case sensitive.

The default parts list keyword is: **.NETS**

Signalname: is the signal name of the net and must be specified. The name can consist of up to 10 characters.

If more than 10 characters are used, the name will be truncated when the file is imported. Warnings are given, but this could result in two or more nets being joined together within XL Designer.

For example, if the signal names 12345678901 and 123456789012 are used, they will both get truncated to 1234567890. This could result in XL Designer seeing them as one net. (Both nets will remain separate on the artwork, providing the group function is not used from within the wiring list editor.)

& (ampersand) should not be used as a signal name.

If & (ampersand) is used in the signalname field, this indicates the nodes on the previous line are connected to the nodes on the line the ampersand appears on.

Node: is replaced with each node in the net. Nodes should be typed in as part reference dot pin number, e.g. IC3.3 or R1.2, etc. (The part reference must also appear in the associated parts list.

Nodes on the same line are connected to one another, but not to nodes on the next line, unless the & (ampersand) sign appears in the signal name field of the next line.

Keyword6: is replaced with a unique keyword to allow the end of the wiring list section to be recognised within the file. User defined keywords can be used, but the keywords are case sensitive.

The default parts list keyword is: **.ENDNETS**

Sample wiring list file/section:

```
.NETS
A1 R2.1 IC3.2 IC1.3 IC4.12 IC2.11 R1.2
N0001 C2.2 IC1.4
N0002 IC1.5 R2.2
VCC IC1.14 IC2.14 IC3.14 IC4.20 IC5.14 IC6.14
& IC7.14 IC8.14 C1.1 C2.1 C3.3 C4.4
& C5.1 C6.1
.ENDNETS
```

Example file containing all sections

```
.PARTS
IC1 74LS00,74LS00 DIL14 4.25 3.6 F 45
IC2 74LS00,74LS00 DIL14 0 0 0
IC3 74LS02,74LS02 DIL14 0 0 0
IC4 74LS244,74LS244 DIL20 0 0 0
IC5 74LS244,74LS244 DIL20 0 0 0
R1 100k RESA8 0 0 0
R2 - RESA8 0 0 0
C1 - CAPA10 0 0 0
C2 - CAPA12 0 0 0
.ENDPARTS
This line will be ignored because it is not within a section.
.POWERNAMES
VCC
GND
.ENDNAMES
.NETS
A0 IC1.1 R1.1 IC2.13
A1 R2.1 IC3.2 IC1.3 IC4.12 IC2.11 R1.2
N0001 C2.1 IC1.4
```

N0002	IC1.5	R2.2		
VCC	IC1.14	IC2.14	IC3.14	IC4.20
&	IC5.14			
GND	IC1.7	IC2.7	IC3.7	IC4.10
&	IC5.7			
.ENDNETS				

Information:

When working on an existing design, it is suggested that a backup copy of the design is taken before importing the file - the results may not be as expected at the first attempt.

If positional information is supplied within the file it will be used to place the parts on the layout.

The board layout can be designed once the parts/wiring list data has been imported, but the circuit diagram will not be re-created.

If gate and pin swapping is performed on the design, the imported parts and wiring list is automatically updated (back-annotated). A back-annotation file is generated within XL Designer, and can be used by the operator to update the original circuit diagram or parts and wiring lists. The back annotation file can be viewed/printed from the *Logfiles* folder.

Sometimes the importer does not produce the results expected. Errors and warnings are produced which may indicate the original file is not in the format expected or silly mistakes have been made, like the same pin used twice in different connections, or the wrong outline name has been specified. In this way, the importer highlights errors in the original file.

In many cases the original file has to be modified, then the importer used again, or sometimes the corrections can be made from within XL Designer after the file has been imported. It all depends on the problem and which method will be most effective. If the changes are made within XL Designer, it is important for future modifications, to modify the original parts/wiring list (and circuit if that's where the parts/wiring list originated from).

If the first import isn't successful and the files have to be imported again, it's easiest to delete the original design and start again with another new design. However, if modifications are being made to an existing design this wouldn't be an option, unless a copy of the original design was being worked on - so always work on a copy of the design.

The importer overwrites any existing parts/wiring list information held in the design, except for certain items, which are user controlled via the importer setup parameters.

Procedure:

Open the design that the parts/wiring list file(s) will be imported into.

Note: when working on an existing design, it is suggested that a backup copy of the design is taken before importing the file - the results may not be as expected at the first attempt.

Right-click the design name from the Navigator pane, then select *Import > Parts/Wiring Lists from Text File*. A window similar to the one in Figure 218 appears.

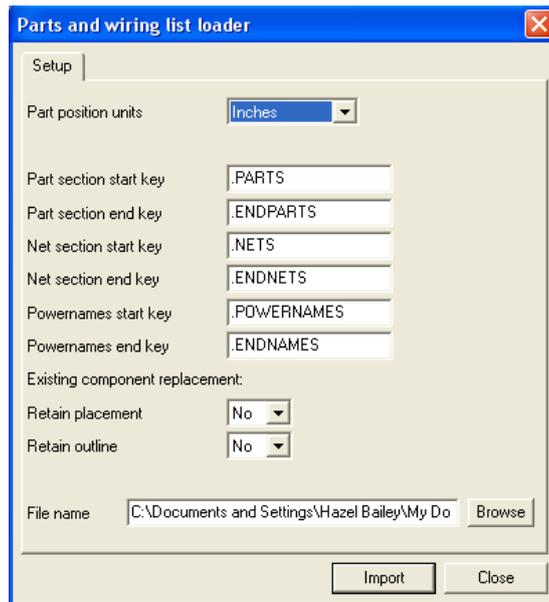


Figure 218

Set the parameters as required, using the following information to assist you:

- Part position units: set this parameter to suit the units defined in the parts list file.
- Part section start key: see description for “Power names end key” below.
- Part section end key: see description for “Power names end key” below.
- Net section start key: see description for “Power names end key” below.
- Net section end key: see description for “Power names end key” below.
- Power names start key: see description for “Power names end key” below.
- Power names end key: the text file can consist of up to three sections, a parts list, a wiring list and a power names list. Each section is recognised by keywords or delimiters at its beginning and end. If the default keywords are used, the section key fields can be left blank. If different keywords are used, they should be specified (case sensitive).

Existing component replacement:

- Retain placement: If the text file is being imported into a design that already has parts placed on the layout (typically when modifications are being performed) there may be a conflict between part positions in the design and in the new parts list. This setting indicates whether the existing part placement should be retained or the part positions from the new file used.
Typically the existing placement should be retained, in which case, Yes should be selected. Any part placement information found in the text file will be ignored. If No is selected, all the part positional information found in the file will be used. Parts that had previously been placed on the layout will take their positions from the imported text file.

- Retain Outline: If the text file is being imported into a design that already has a parts list (typically when modifications are being performed) there may be a conflict between the outlines defined in the existing parts list and in the new parts list. This setting indicates whether the existing part outline should be retained or the part outline from the new file used.
Care should be exercised with this setting if changes have been made to the outlines in the existing parts list that have not been back-annotated to the original circuit diagram and parts/wiring list. If you are unsure which option to select, take copies of the design and try both options.

- If the outlines are not retained, the report that appears after the import has taken place lists which parts have changed.

Filename: specify the name of the file that will be imported. The browse button can be used to locate folders and files.

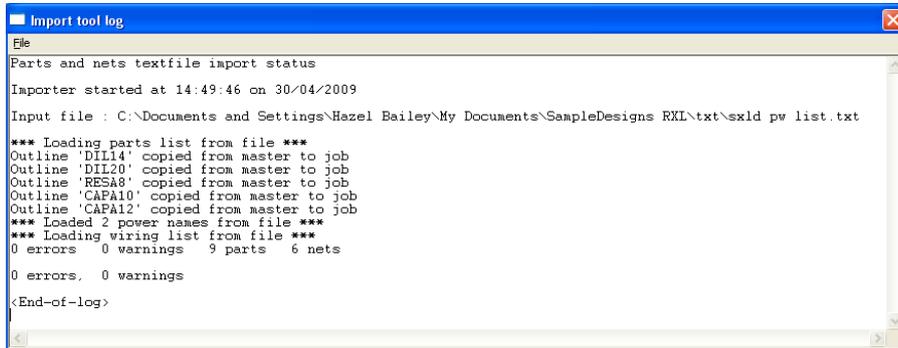
If the parts list, power names list and wiring list are held in separate files, each file has to be imported separately. The parts list must be imported before the

wiring list, and the powernames list before the wiring list.

Once this window has been setup as required, select the *Import* button to start the import process.

Information appears on the screen to indicate what is happening. This information will be stored in the report log.

The report that is generated will look similar to the one shown in Figure 219.



```
Import tool log
File
Parts and nets textfile import status
Importer started at 14:49:46 on 30/04/2009
Input file : C:\Documents and Settings\Hazel Bailey\My Documents\SampleDesigns RXI\txt\sxld pw list.txt
*** Loading parts list from file ***
Outline 'DIL14' copied from master to job
Outline 'DIL20' copied from master to job
Outline 'RES8' copied from master to job
Outline 'CAPA10' copied from master to job
Outline 'CAPA12' copied from master to job
*** Loaded 2 power names from file ***
*** Loading wiring list from file ***
0 errors 0 warnings 9 parts 6 nets
<End-of-log>
```

Figure 219

Problems are separated into two types, Warnings and Errors - errors are generally more serious than warnings. Each problem encountered is reported in detail and each one must be investigated and corrected where necessary - this might involve editing the original files or the imported parts/wiring list from within XL Designer.

The first part of the report indicates when the importer was started and which file was used as input:

The report then indicates what has happened during the part list import, for example, which outlines have been copied from the master library into the design.

If power rails were specified in the file, then the number of power rails imported is indicated and finally the wiring list is imported.

Typical Warnings/Errors

If warnings/errors are given they should be investigated and changes made as and when appropriate

- Warning : *Outline '0805' not available* - this warning is given because the outline name specified (0805) is not a valid name in the component outline library. (Either edit the imported parts list and correct the name, create the outline in the outline library or edit the original parts list and re-import.)
- Warning : *Net node 'IC2.15' - Pin not found on outline* - this warning may be caused because an outline was specified that doesn't exist or maybe the outline does exist but does not have enough pins.
- Warning : *Wiring list requires 'group' operation* - this indicates that the net list has nets with the same name that have been separated. This can be rectified (if appropriate) from XL Designer's own parts/wiring list editor, using the *Edit > Group signal names* command.

Close the report window when you are ready to continue. The import process is complete.

The report can be re-viewed by opening the *Logfiles* folder from the navigator and viewing the report named "import_textpw".

If the import was unsuccessful, investigate what went wrong using the error/warning messages to guide you. It is essential that appropriate action is taken as the resultant artwork design may be incorrect and much time could be wasted.

Viewing/editing the imported parts/wiring list

By viewing and then scrolling through the imported parts and net lists it's possible to identify any component outlines and/or pin numbers that are invalid – these should be rectified before the artwork is designed.

To view or modify the imported parts/wiring list, from the open design folder in the navigator window, double-select the *Parts* and/or *Nets* folders. Scroll through the windows and look for highlighted outline names (parts list) or pin numbers (net list).

These problems must be rectified - in the case of highlighted outline names, by using a valid outline name or creating/copying the specified outline in the design's component outline library. Once an outline name is corrected, this may also resolve invalid pin numbers, if they were associated with that part. If the pin numbers remain highlighted, check that the outline called up for the part has the appropriate pin numbers within it.

Calay parts/wiring lists import

This routine is used to import Calay format parts/wiring lists. The board layout can be designed once this data has been imported, but the circuit diagram will not be re-created.

The following data will be required either before or after the parts/wiring list has been imported in order to design the artwork:

- A board profile
- Details of which connections are power rails
- The physical footprints (in the component outline library)
- Pad, track, etc, sizes
- If gate and pin swapping will need to be performed, schematic parts will also need to be defined in the schematic library and the parts list updated with the correct *Type* field that matches the schematic part's value field accordingly.

Expected Calay file format:

A Calay format parts/wiring list should be supplied as two separate files, one containing the parts list, the other the wiring list.

The Calay parts list

This file requires the following fields:

Type	Reference	Outline	X-Position	Y-Position	Rotation
------	-----------	---------	------------	------------	----------

Fields cannot be omitted. The X-Position, Y-Position and Rotation columns are typically set to zero.

Type maximum of 23 characters. This field will be accessed by the Ranger gate and pin swapping routines if used.

If gate and pin swapping will be used, the field should take the form of a component name and value separated by a comma. For example:

TYPE,VALUE

R0.25W,10k

Both items should correspond to a circuit part. For instance, in the above example there must be a part in the schematic library called R0.25W with a value field set to 10k.

Reference maximum of 4 letters followed by a number. For example: IC1, R1, PLA1, XXXX1, etc.

Outline maximum of 11 characters. The outline (footprint) does not have to exist in the outline library before the import takes place, but must exist before laying out the design.

X & YPosition these fields are usually set to zero although co-ordinates can be entered to pre-place parts on the board. Units can be entered in inches or millimetres, but not both.

Rotation this field is usually set to zero although parts may be pre-rotated as required. Units are degrees, from 0 to 359. The part is rotated anti-clockwise about its datum as defined in the outline library.

Example:

74LS00	IC1	DIL14	0	0	0
74LS00	IC2	DIL14	0	0	0
74LS02	IC3	DIL14	0	0	0
74LS244	IC4	DIL20	0	0	0
100k	R1	RESA8	0	0	0
100k	R2	RESA8	0	0	0
100uF	C1	CAPA10	0	0	0
100uF	C2	CAPA12	0	0	0

The Calay Wiring list

This file requires the following fields:

/NetID	Node	Node;					
/NetID	Node	Node	Node	Node	Node	Node	Node,
	Node	Node	Node	Node	Node	Node;	

Each line represents a separate connection and is terminated by a semi-colon (;).

If the connection extends beyond one line, a comma (,) should be used as a continuation character on the end

of each line that is not complete.

- /NetID** The unique signal name of the connection or 'net'. Maximum of 10 ascii characters, excluding / at the beginning of the name which must be present but is not included as part of the signal name.
If more than 10 characters are used the name will get truncated. Warnings are given if this happens.
A netid must be supplied.
The ampersand sign, (&) should not be used as a signal name as it is used as a continuation character elsewhere.
- Node** Consists of the reference that must appear in the parts list and a pin number in brackets. Each node must be separated by at least one space or tab.
A semi-colon (;) indicates the end of a connection.
A comma (,) indicates the connection is continued on the next line.

Example:

```
/N00001 IC1(1) R1(1) IC2(13);
/N00002 C1(1) R2(1) IC3(2) IC1(3) IC4(12) IC2(11),
        R1(2) C2(1) IC1(4);
/N00003 IC1(5) R2(2);
/VCC IC1(14) IC2(14) IC3(14) IC4(20);
/GND IC1(7) IC2(7) IC3(7) IC4(10);
```

Useful Information:

Sometimes the importer does not produce the results expected. Errors and warnings are produced which may indicate the original file is not in the format expected or silly mistakes have been made, like the same pin used twice in different connections, or the wrong outline name has been specified. In this way, the importer highlights errors in the original file.

In many cases the original file has to be modified, then the importer used again, or sometimes the corrections can be made after the file has been imported. It all depends on the problem and which method will be most effective. If the changes are made after the file has been imported, it is important for future modifications to modify the original parts/wiring list (and circuit if that's where the parts/wiring list came from).

If modifications are being made to an existing job, make sure a backup copy of the design has been taken PRIOR to importing the files – sometimes things don't go according to plan.

The importer overwrites any existing parts/wiring list information held in the job, except for certain items, which are user controlled via the Setup Parameters (described below).

If the wiring list contains nets with the same name that have been separated, then these are grouped together during the import process.

The files to be imported should be text files in the format shown above.

Procedure:

1. Create or open the job that the files will be imported into – when working on an existing design, always take a backup copy before importing the file - the results may not be as expected!
2. Right-click on the design's name in the navigator, select *Import > Calay parts and wiring lists from text file*. A window similar to the one in Figure 220 appears:

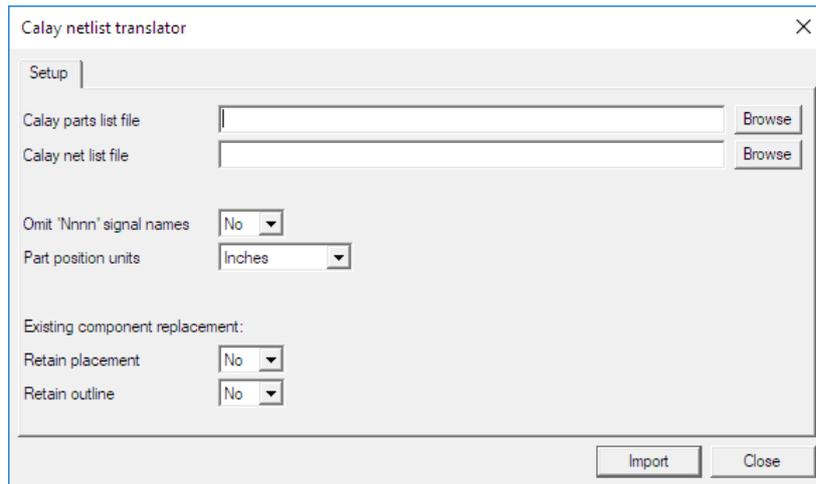


Figure 220

Set the parameters as required, using the following information to assist you:

Calay parts list file: specify the name of the file containing the Calay parts list that will be imported. The browse button can be used to locate folders and files.

Calay net list file: specify the name of the file containing the Calay net list that will be imported. The browse button can be used to locate folders and files.

Omit 'Nnnn' signal names: unnamed nets from the original circuit diagram are each given a unique name in the Calay wiring list, for example N00001, N00002, etc. These names are not required and the user can choose to omit them. Signal names that do not take this form are always imported.

Part position units: Set this parameter to suit the units used in the Calay parts list file.

Existing component replacement:

Retain placement: If the text file is being imported into a design that already has parts placed on the layout (typically when modifications are being performed) there may be a conflict between part positions in the design and in the new parts list. This setting indicates whether the existing part placement should be retained or the part positions from the new file used.

Typically the existing placement should be retained, in which case, Yes should be selected. Any part placement information found in the text file will be ignored.

If No is selected, all the part positional information found in the file will be used. Parts that had previously been placed on the layout will take their positions from the imported text file.

Retain Outline: If the text file is being imported into a design that already has a parts list (typically when modifications are being performed) there may be a conflict between the outlines defined in the existing parts list and in the new parts list. This setting indicates whether the existing part outline should be retained or the part outline from the new file used.

Care should be exercised with this setting if changes have been made to the outlines in the existing parts list that have not been back-annotated to the original circuit diagram and parts/wiring list. If you are unsure which option to select, take copies of the job and try both options.

If the outlines are not retained, the report file lists which parts have changed.

Once the parameters are set as required, select the *Import* button to start the import process.

A report log will appear on the screen to indicate what happened during the import process. This information will be stored in the report log (import_calay) that can be accessed from the *Logfiles* folder in the design's navigator window.

The report window that is generated will look similar to the one shown in Figure 221.

```

Untitled:/logs/import_calay.txt : Logfile
File
Filter 'incalay' started at 12:46:12 on 05/04/2018
Importing from Calay parts and wiring list files
Parts input file : C:\Documents and Settings\SEETRAX\My Documents\SampleDesigns RXL\txt\Calayparts.txt
Nets input file : C:\Documents and Settings\SEETRAX\My Documents\SampleDesigns RXL\txt\Calaynets.txt

*** Processing input parts list ***
*** Processing input wiring list ***
Warning : C1 value changed
Warning : C1 outline changed
Warning : C2 value changed
Warning : C2 outline changed
Warning : C3 value changed
Warning : R2 outline changed
Warning : R3 value changed
Warning : R3 outline changed
Warning : Outline '????' not available
Warning : Outline 'SOD80RA' not available
Warning : Outline '0805RN' not available
Loading wiring list from file
Warning : Net node 'IC1.3' - Pin not found on outline
Warning : Net node 'C3.2' - Pin not found on outline
Warning : Net node 'B1.1' - Pin not found on outline
Warning : Net node 'R3.1' - Pin not found on outline
Warning : Net node 'C2.2' - Pin not found on outline
Warning : Net node 'C1.2' - Pin not found on outline
Warning : Net node 'D1.2' - Pin not found on outline
Warning : Net node 'CN1.2' - Pin not found on outline
Warning : Net node 'R2.1' - Pin not found on outline
Warning : Net node 'D1.1' - Pin not found on outline

0 errors 21 warnings 14 parts 10 nets
Import complete

```

Figure 221

Problems are separated into two types, Warnings and Errors - errors are generally more serious than warnings. Each problem encountered is reported in detail and each one must be investigated and corrected where necessary - this might involve editing the original Calay files, the imported parts/wiring list or introducing the required outlines. The total number of errors and warnings is then given, followed by a statement to indicate the report is complete. Close the report window when you are ready to continue. The import process is complete. The report can be re-viewed by opening the *Logfiles* folder from the navigator - the report is called *import_calay* and can be printed or saved as a text file as required by selecting the *File* command whilst the window is active. If the import was unsuccessful, then you need to investigate what went wrong using the report to guide - what do the error messages indicate - look at the original Calay file - is it in the expected format?

Typical warnings/errors

Warning: <part ref> value changed

Means that the value (type) of this part has been altered (will only occur if an existing design is being modified).

Warning: <part ref> outline changed

Means that the component outline of this part has been altered (will only occur if an existing design is being modified).

Warning: Outline <name> not available

Means that the outline specified has not been found in the outline library. It will need to be copied to/created in the design's component outline library, or the original parts list file or design modified to suit what is available.

Warning: Single node net <name> omitted

Means that a connection was started in the Calay netlist file but it did not terminate on another pin so was not included in the imported file. The original Calay file/circuit should be checked and appropriate action taken.

Warning: Net node <part ref.pin number> - Pin not found on outline

Means that the pin specified does not exist in the outline specified - either the wrong outline was specified or the pin numbers are incorrect.

Viewing/editing the imported parts/wiring list

By viewing and then scrolling through the imported parts and net lists it's possible to identify any component

outlines and/or pin numbers that are invalid – these should be rectified before the artwork is designed.

To view or modify the imported parts/wiring list: from the open design folder in the navigator window, double-select the *Parts* and/or *Nets* folders. Scroll through the windows and look for highlighted outline names (parts list) or pin numbers (net list).

These problems must be rectified - in the case of highlighted outline names, by using a valid outline name or creating/copying the specified outline in the design's component outline library. Once an outline name is corrected, this may also resolve invalid pin numbers, if they were associated with that part.

If the pin numbers remain highlighted, check that the outline called up for the part has the appropriate quantity of pin numbers within it and they are numbered appropriately.

Outputs Folder - Tasks

With a design open, a variety of output data can be produced from the *Outputs* folder. This folder contains two sub-folders: *Tasks* & *Batches*, both are empty initially.

The *Tasks* folder will contain specific output tasks, for example tasks might be created to produce gerber files for the top and bottom artwork layers, top & bottom solder masks, top & bottom silk-screens, etc., other tasks for the schematic sheets to a printer, artwork layers to a printer, nc drill output files and so on.

Once output tasks have been created, they can be added to a batch process to speed up the output or to ensure the same tasks are output together in the future – the batch processes are defined in the *Batches* folder and this is described later in this chapter.

What can be output

Typically during the design, printouts are produced of the schematic and artwork for checking purposes.

Once a design is complete and checked without errors being reported, it's ready to be output for manufacturing.

Manufacturing files can be produced even if the checking routines report errors, but this is not recommended.

Typically gerber files are required to produce the artwork films (artwork layers, silk-screens, solder masks and solder pastes) and either Excellon or Sieb & Meyer files to drive NC drilling and routing machines.

The *Outputs* folder gives access to all of these output tools.

Note: **GenCAD** and **BSL** (Bath Scientific Ltd) files can also be produced for automatic test equipment, pick & place machines, etc. These are produced by right-clicking on the design name in the navigator and choosing the *Export* command.

BOM's can be produced from the schematic design sheet editor (*Tools > Extract*) or from the Parts list editor (*View > BOM*) - these two options produce different types of BOM.

Creating a New Output Task

With a design folder open in the navigator pane, open its *Outputs* folder. Right-click on the *Tasks* folder and create the new task required from the list that appears. The choices are to create a new *Task* for:

Artwork Layers	Choose this Task to output artwork data, such as the top, inner & bottom copper layers, power plane layers, silk screen layers and documentation layers. <i>User-defined</i> solder mask and solder paste layers (those created from the artwork editor, <i>Tools > Generate silkscreen</i> command) should also be output using this type of Task.
Solder Mask	Choose this Task to output an auto-generated solder mask, where all the pads and/or vias have a uniform pad swell (if required) applied. User-defined solder masks that have been generated using the <i>Tools > Generate silkscreen</i> command in the artwork editor should be output with an <i>Artwork layers</i> Task.
Solder Paste	Choose this Task to output an auto-generated solder paste where all the surface mounted component pads have a uniform pad size decrease (if required) applied. User-defined solder pastes that have been generated using the <i>Tools > Generate silkscreen</i> command in the artwork editor should be output with an <i>Artwork layers</i> Task.
Drill sheet	Choose this Task to output a drill sheet (drill drawing). Typically, a drill sheet (drawing) contains the positions of all the holes in the design, each hole being represented by a different symbol, and a table listing the symbols used, their sizes and number off of each.
Schematic	Choose this Task to output any of the schematic design sheets.
NC Drill & Rout Data	Choose this Task to output files that will be used to drive an NC drilling and/or routing machine.
IDF Output	Choose this Task to output files in an IDF version 3 format file, for use with mechanical CAD modelling tools.

Once the new Task has been selected, supply an appropriate name – the names should help identify the type of output (sheet1, top, bottom, inner, etc.) and the output format (gerber, printer, documentation layer, etc.).

Examples: "top copper - gerber", "top copper - printer", "top soldermask - gerber", "sheet 3 - printer", etc.

Move on to the heading, *Opening a task* to define the settings within the Task.

Renaming a Task

Once a task has been created, it can be renamed by selecting it with a right-click of the mouse, then selecting *Rename*. Type in the name required followed by <enter>.

Copying an Output Task

Once a task has been created and its content defined, it can be used as a template for other similar tasks, within the same design or other designs, by copying it and modifying the copy. This will be quicker than creating another Task from scratch.

With a design folder open in the navigator pane, open its *Outputs* and *Tasks* folder. Right-click on the task to be copied, select *Copy*. Locate the destination *Tasks* folder (which might be in the same or a different design) then right-click on the *Tasks* folder and select *Paste*. The Task is copied. It can be opened (double-select it) and modified.

Drag/Copy/Drop functionality

In addition to the *Copy/Paste* command, "drag & drop" is available to *copy* tasks within or between designs. Select the task(s) required, then drag them by holding down the left mouse key to the target Task folder, releasing the button whilst hovering over the target folder.

Deleting a task

A task can be deleted by selecting it with a right-click of the mouse, then selecting *Delete*.

Opening a task

With a design folder open in the navigator pane, open its *Outputs* folder, then double-select the Task to open it. Either a self-contained window will open (for example when the NC Drill & Rout Data or the IDF Outputs are selected) or an editor will open.

If an editor opens, use the *Configure* commands to control what is included in the output (symbols, connections, pads, tracks, etc.) and its format (gerber, dxf, printer, etc.).

Layer selections

Data types/categories

For output purposes the artwork data is divided into two categories, *Pads* and *Copper*. If something is not a pad, then it is *copper*. (This may seem odd, for instance silk-screen labels and outlines being regarded as *copper*, but this only applies to the output settings.)

<i>Vias</i>	standard vias only exist on layer V, so if they should be included in the output, select layer V pads.
<i>Blind/buried vias</i>	are only included if the layer they appear on has the <i>Pads</i> box ticked AND layer V pads is included as well.
<i>Component pads</i>	component pads are made as a "stack" of pads, so if they should be included in the output then <i>pads</i> from the appropriate layer should be selected. Component pads do not exist on the via layer, V. Surface mounted component pads only appear on layer T (top component side) or layer B (solder side). Note: component pads do not appear on layers defined as <i>Power planes</i> , <i>Silk-screen</i> or <i>Documentation</i> layers.
<i>Other pads</i>	pads that have been added using the <i>Amend</i> menu are either added to particular layers or layer V. Include <i>pads</i> from the appropriate layer.
<i>Tracks</i>	tracks on a layer are included if <i>copper</i> from the appropriate layer is selected from the setup window.
<i>Copper fill</i>	is included if <i>copper</i> from the appropriate layer is selected from the setup window.
<i>Text</i>	on a layer is included if <i>copper</i> is selected from the appropriate layer (even if the layer is set to a silk-screen/documentation layer).
<i>Silk-screen (labels and outlines)</i>	are included if <i>copper</i> is selected from the appropriate layer. There are no component pads on a silk-screen layer. The only pads that would exist on a silk-screen layer are pads that have been added by user-actions such as <i>Amend > Enter Pad</i> .
<i>Power planes</i>	heat-relief and anti-pads are output if <i>pads</i> are selected from the appropriate layer. <i>Copper</i> should also be included from that layer to include the split-plane isolation lines and the clearance track around the edge of the board. Note: when outputting power planes, layer V (vias) should not be included.

Typical layer selections:

Artwork, top side: Layer V and T Pads, plus Layer T copper

Artwork, bottom side:	Layer V and B pads, plus Layer B copper
Inner copper layers:	Layer V and pads from the inner layer, plus copper from the inner layer
Top silk-screen:	Layer 1 and 2 copper (assuming labels on layer 1 and outlines on layer 2)
Bottom silk-screen:	Layer 3 and 4 copper (assuming labels on layer 3 and outlines on layer 4)
Power plane/split plane:	Layer 5 pads and copper (assuming it's on layer 5) Note: do not include layer V pads)
Bottom solder mask:	Layer B pads If vias should be exposed through the solder mask, then also <i>Include vias</i> from the <i>Setup</i> window and include layer V pads. Copper & pads from other layers can be included as/if required, in order to expose other areas of the board through the solder mask.
Top solder mask:	Layer T pads If vias should be exposed through the solder mask, then also <i>Include vias</i> from the <i>Setup</i> window and include layer V pads. Copper & pads from other layers can be included as/if required, in order to expose other areas of the board through the solder mask.
Bottom solder paste:	Bottom pads
Top solder paste:	Top pads

Configure Commands

These commands control what is output, where the output is directed, the position and scale of the output data. Some commands are context sensitive, so for example the *Configure > Plot Source* and *Configure > Output Filter* commands will produce different windows depending on which type of task is open and the type of output format selected. All the variants are described below.

Configure > Plot Source (with an Artwork Layers Task open)

The Artwork Layers Task is used to output artwork data, such as the top, inner & bottom copper layers, power plane layers, silk screen layers and documentation layers.

User-defined solder mask and solder paste layers (those created from the artwork editor, *Tools > Generate silkscreen* command) should also be output using this type of Task.

This command should be selected to control exactly what layers, etc., will be included in the artwork output.

The *Configure > Plot Source* command can also be activated by selecting the *Setup* button alongside the Artwork Layers Source setting in the Toolbar, arrowed in Figure 222.



Figure 222

The *Source* setting (Artwork Layers) determines what type of data will be output and this can't be changed as it's controlled by the type of *Task* that's open.

When the command is selected, a window appears as shown in Figure 223.

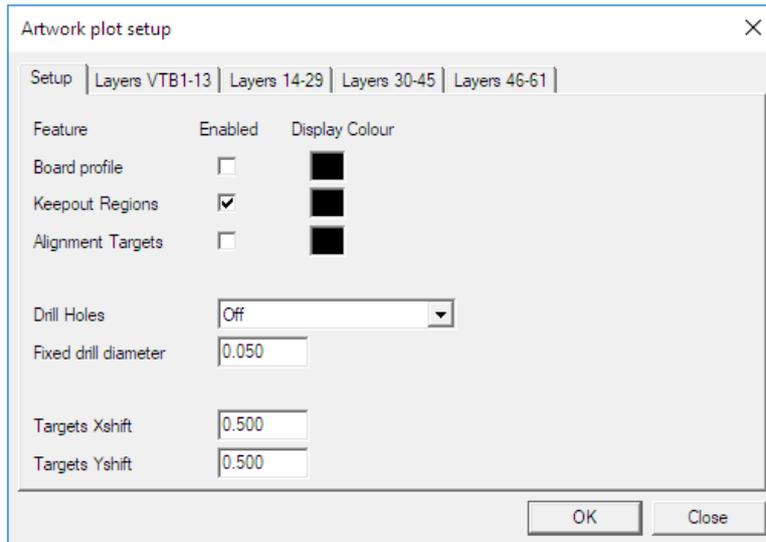


Figure 223

This window, made up of 5 pages that are accessed from the *Setup* and *Layers* tabs at the top of the window, controls what will be included from the artwork when it is output. Different combinations of layers, pads and copper are output, depending on what is required – typical layer selections are described above under the heading: Layer selections.

Output files are typically produced for the top copper layer, bottom copper layer, silk screen layer(s), documentation layer(s) and inner layers (copper & powerplanes) if used.

User-defined solder mask and solder paste layers (those created from the artwork editor, *Tools > Generate silkscreen* window) should also be output using this task.

Set the pages in the window as required using the following information to guide you.

- Board profile** if enabled, the board profile (as defined in the profile editor) is output. Typically the profile is not included in gerber files, but it may be useful when producing check plots/prints.
- Keepout regions** if enabled, the keep-out lines/regions as defined in the profile/artwork editors are output. (Not those defined within component outlines). Typically keepout lines are not included in output files unless there is a specific requirement, perhaps for a check plot/print in order to assess their positions.
- Alignment Targets** if enabled, three alignment target markers are output. *Target markers* or "alignment markers" are used to align the layers of the board together when manufacturing the board. They would normally be included with all output files, and in the same relative positions. The alignment target markers are produced automatically by the output tool so are not visible in the artwork editor though they are seen in the output tool when selected. They are positioned automatically on the top corners and bottom right corner of the profile if it is rectangular. Irregularly shaped boards have an imaginary rectangle placed around them that is used for target positions - the rectangle is drawn as small as possible without overlapping the real profile. The *Target X & Y Shifts* setting is used to move the alignment targets away from the corners of the profile. Unless output to gerber files, the target markers appear as 4 unfilled quadrants of a circle, with 2 diagonally opposite quadrants missing. Target markers within gerber files are defined by the Dcode that is used to produce them.
- Display Colour** the data will be displayed and output in the colour indicated on the screen. Note: there may not be a close colour match between the display colour and that output to the colour printer/plotter. Select the coloured box and choose a colour from those in the colour palette window that appears.
- Drill Holes** This setting is useful when producing plots for prototype boards that will be spot drilled – a small sighting spot can be output to assist in the manual drilling process. The available options are:

Off	Drill holes are not output. (Typically used when producing gerber photoplot files.)
Include at true size	Drill holes are displayed and output at the drill size specified within the pad
Include at fixed size	All drill holes are displayed and output at the diameter set in the <i>Fixed drill diameter</i> parameter. (Typically used when producing prototype boards for spot drilling.)

Fixed Drill Diameter Specifies the drill size to be displayed and output if the *Drill Holes* parameter is set to *Include at fixed size*

Target X and Y shift without a shift (0 specified) the target markers are positioned on 3 corners of the rectangular area occupied by the profile. The target markers do not need to be part of the finished artwork so a shift is usually entered to move the targets away from the edge of the profile.

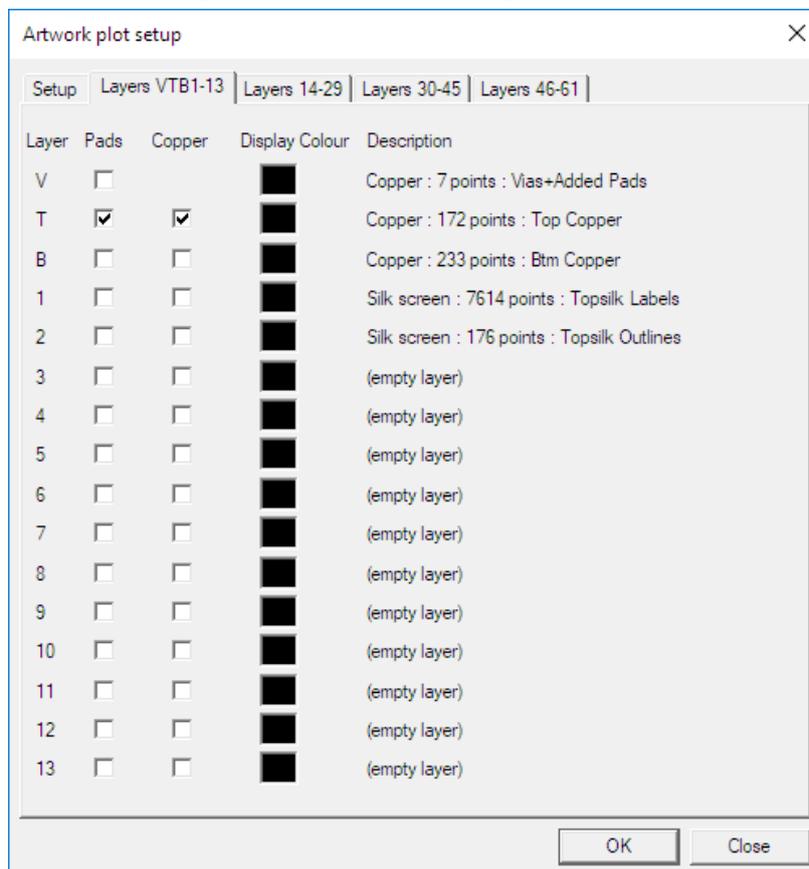
A positive X and Y shift is used to offset the targets away from the board. Typically, the targets are offset by 0.5"/10mm or 1"/20mm in the X and Y axis.

The same shifts should be used for all the output files of a particular design.

“Layer n1-n2” pages

Because of the number of layers, the layer settings are sub-divided across pages. Use the appropriate *Layers* tabs to access the layers required. The page showing *Layers V, T, B, 1-13* is shown here.

Note: always check all the layers pages to ensure there is nothing selected/unselected that shouldn't be.



Whichever layers page is visible, it shows the selected range of layers and the following settings, which should be set as required.

Layer The layers listed represent the layers of the board as displayed in the artwork editor. Whether the data on the layers is output, is determined by the selections made in the *Pads* and *Copper* columns alongside.

Pads If ticked, **all** the pads from that layer are output. This includes any pads that have been added using the AMEND menu.

Copper if ticked, anything from that layer that is not a pad will be output, for example, tracks, text, copper fill, silk-screen lines, irrespective of the layer type - for example a silkscreen layer does not contain any copper, but any lines, text, etc on that layer will be output when selected.

Display Colour the data will be displayed and output in the colour indicated on the screen. Note: there may

not be a close colour match between the display colour and that output to the colour device.

Select the coloured box and choose a colour from those in the colour palette window that appears.

Description if the layer is in use (i.e. not empty) useful information about it is displayed. The layer type, the number of points on the layer and its Title (if one was assigned) is displayed. (Refer to *Configuration > Layer Assignments & Ordering* for layer types and titles).

Once the pages in the window have been set as required, move on to the heading: Configure > Output Filter

Configure > Plot Source (with a Solder Mask Task open)

This command should be selected to control exactly what will be included in an auto-generated solder mask when it is output.

User-defined solder masks that have been generated using the *Tools > Generate silkscreen* command in the artwork editor should be output with an *Artwork layers Task*.

Typically, a solder resist mask is created for the bottom of the board only, but some companies create a solder mask for the top of the board as well. One output file is required for each side of the board. Each output could also contain the profile, target markers and data from other layers if required.

The *Configure > Plot Source* command can also be activated by selecting the *Setup* button alongside the Solder Mask Source setting in the Toolbar, arrowed in Figure 224



Figure 224

The *Source* setting (Solder Mask) determines what type of data will be output and this can't be changed as it's controlled by the type of *Task* that's open.

When the command is selected, a window appears as shown in Figure 225.

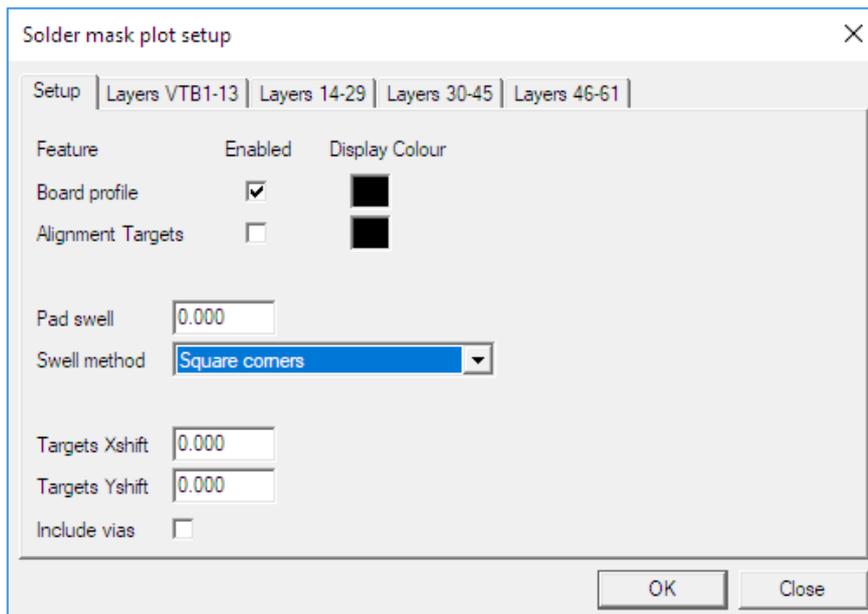


Figure 225

This window, made up of 5 pages that are accessed from the *Setup* and *Layers* tabs at the top of the window, controls what will be included in an auto-generated solder mask when it is output.

Different combinations of pads/vias, sometimes copper, are output, depending on what is required.

Data that is output, results in areas free of solder mask.

It is usual to increase the size of pads that are output so that any mis-alignment of the actual solder mask doesn't cover the pads on the board and hinder soldering.

Set the pages in the window as required using the following information to guide you.

Board Profile if enabled, the board profile (as defined in the profile editor) is output. Typically the profile would not be output, but can be included if required.

Alignment Targets if enabled, three alignment target markers are output.

Target markers or "alignment markers" are used to align the layers of the board together when manufacturing the board. They would normally be included with all output files, and in the same relative positions.

The alignment target markers are produced automatically by the output tool so are not visible in the artwork editor, though they are seen in the output tool when selected.

They are positioned automatically on the top corners and bottom right corner of the profile if it is rectangular. Irregularly shaped boards have an imaginary rectangle placed around them that is used for target positions - the rectangle is drawn as small as possible without overlapping the real profile.

The *Target X & Y Shifts* setting is used to move the alignment targets away from the corners of the profile.

Unless output to gerber files, the target markers appear as 4 unfilled quadrants of a circle, with 2 diagonally opposite quadrants missing.

Target markers within gerber files are defined by the Dcode that is used to produce them.

Display Colour the output data will be displayed and output in the colour indicated on the screen. Note: there may not be a close colour match between the display colour and that output to the colour printer/plotter.

Select the coloured box and choose a colour from those in the colour palette window that appears.

Pad swell the value entered here increases the size of all the pads that are output.

Typically, a pad swell is specified to make the pads that are output, larger than the actual pads and therefore suitable for a solder resist mask.

For instance a pad swell of 0.015" increases all the pad sizes by 0.015". i.e. a 0.060" pad is output at 0.075", and a 0.1" x 0.050" pad is output at 0.115" x 0.065".

(In metric a 0.4mm swell would increase a 1.5mm pad to 1.9mm, and a 2.5mm x 1.2mm pad to 2.9mm x 1.6mm.)

Swell method Options are *Square corners* or *Filleted (rounded) corners*. Typically *Square Corners* are output.

Square corners: square and rectangular pads will be swollen by a simple increase of their width and length so right-angled corners will be maintained.

Filleted (rounded) corners: swollen pads are created that maintain a constant clearance from the original pad. For a pad with right-angled corners, this constant clearance results in a radiused corner to the swollen pad.

This setting has no effect on round/round-ended finger pads and does not affect custom pads which are always swollen using the filleted method of maintaining constant clearance.

Target X and Y shift without a shift (0 specified) the target markers are positioned on 3 corners of the rectangular area occupied by the profile. The target markers do not need to be part of the finished artwork so a shift is usually entered to move the targets away from the edge of the profile.

A positive X and Y shift is used to offset the targets away from the board. Typically, the targets are offset by 0.5" or 1" (10mm or 20mm) in the X and Y axis.

The same shifts should be used for all the output files of a particular design.

Include vias Via pads can be included or excluded from the output. Some companies prefer to cover via holes with the solder mask, whilst others do not.

Including the vias (ticked) will result in the via pads being exposed through the solder mask, **provided** the pads from layer V (in the *Layer V, T, B -13* window) are **also** included.

If layer V pads are not included, then the vias will not be included, even with this setting ticked – check the display before producing the output file if you are unsure.

(If Vias are not included, but layer V pads are included, only added pads on the V layer will be output.)

Once the *Setup* window has been set as required, the layers from the artwork have to be selected.

Layer pages

Because of the number of layers, the layer settings are sub-divided across pages. Use the appropriate *Layers* tabs to access the layers required. The page showing *Layers V, T, B, 1 -13* is shown in Figure 226.

Note: always check all the layers pages to ensure there is nothing selected/unselected that shouldn't be.

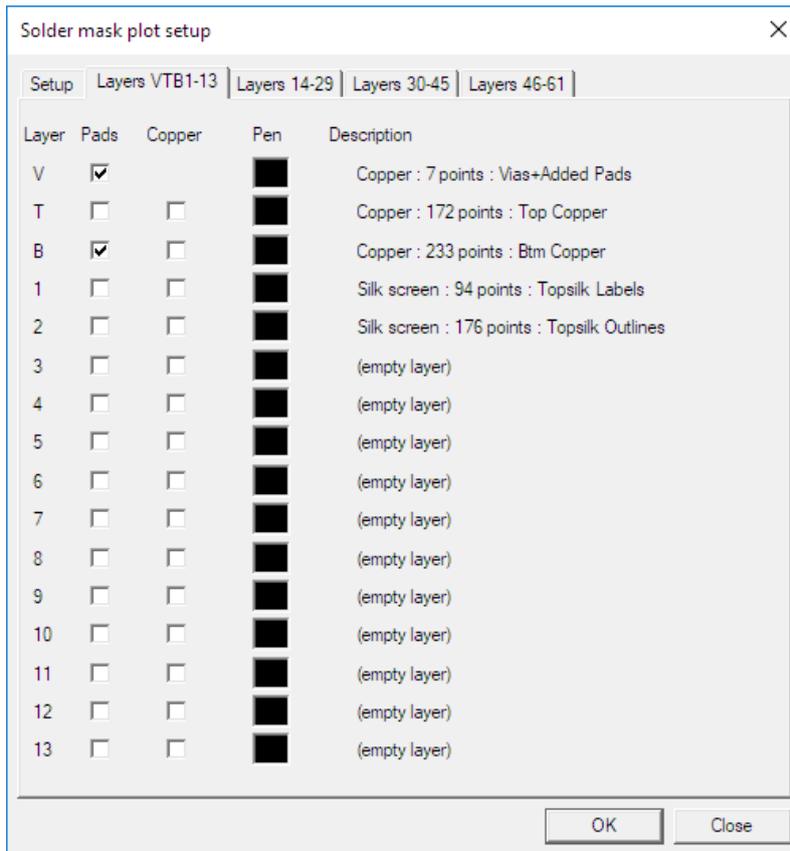


Figure 226

Whichever layers page is visible, it shows the selected range of layers and the following settings, which should be set as required.

Layer The layers listed represent the layers of the board as displayed in the artwork editor. Whether the data on the layers is output, is determined by the selections made in the Pads and Copper columns alongside.

Just layer B (bottom) pads would be output for a bottom solder mask which requires vias to be covered by the solder mask.

Pads If ticked, **all** the pads from that layer are output. This includes any pads that have been added using the AMEND menu.

Including the V layer (ticked) will result in all the added pads on the V layer being exposed through the solder mask.

If the vias have to be exposed through the mask as well, ensure the “*Include Vias*” setting on the Setup page is also included.

If layer V pads are not included, then no pads from the V layer will be output - check the display before producing the output file if you are unsure.

Copper if ticked, anything from that layer that is not a pad will be output, for example, tracks, text, copper fill, silk-screen lines.

For example, it may be a requirement to expose a drawing number, layer identification, issue number, etc. through the solder mask.

Display Colour

The data will be displayed and output in the colour indicated on the screen. Note: there may not be a close colour match between the display colour and that output to the colour printer/plotter.

Select the coloured box and choose a colour from those in the colour palette window that appears.

Description

If the layer is in use (i.e. not empty) useful information about it is displayed. The layer type, the number of points on the layer and its Title if one was assigned is displayed. (Refer to *Configuration > Layer Assignments & Ordering* for layer types and titles).

Once the pages in the window have been set as required, select **Close** to continue, then move on to the

heading: Configure > Output Filter

Configure > Plot Source (with a Solder Paste Task open)

This command should be selected to control exactly what will be included in an auto-generated solder paste when it is output.

User-defined solder pastes that have been generated using the *Tools > Generate silkscreen* command in the artwork editor should be output with an *Artwork layers Task*.

If surface mounted components exist on both sides of the board, typically two output files need to be generated. One containing pads from SMD's on the top of the board, the other containing pads from SMD's on the bottom of the board.

Each output could also contain the profile and target markers, if they are required.

Copper and/or pads from one other user-selected layer can also be included in the output. This allows text or special features to be included.

The *Configure > Plot Source* command can also be activated by selecting the *Setup* button alongside the *Source* setting in the Toolbar, arrowed in Figure 227.



Figure 227

The *Source* setting (Solder Paste) determines what type of data will be output and this can't be changed as it's controlled by the type of *Task* that's open.

When the command is selected, a window appears as shown in Figure 228.

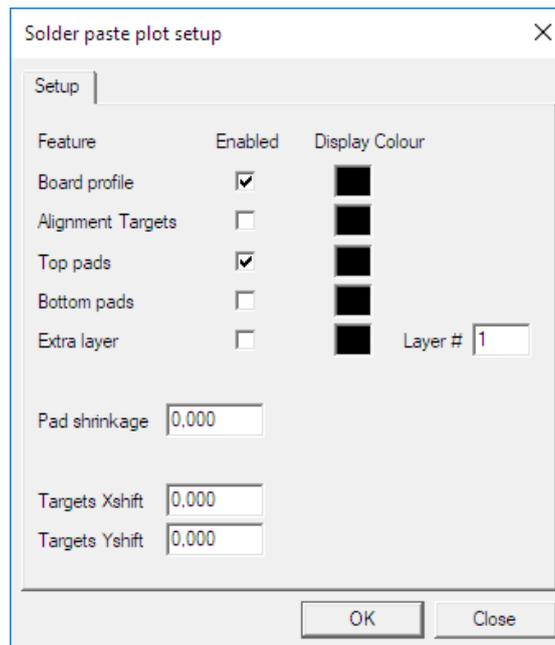


Figure 228

This window controls what will be included in an auto-generated solder paste when it is output.

Set the pages in the window as required using the following information to guide you.

Board profile If ticked, the board profile (as defined in the profile editor) is output. Typically the profile would not be output, but can be included if required.

Alignment Targets if ticked, three target markers are output.

Target markers or "alignment markers" are used to align the layers of the board together when manufacturing the board. They would normally be included with all output files, and in the same relative positions.

The target markers are produced automatically by the output tool so are not visible in the artwork editor though they are seen in the output tool when selected.

The target markers are positioned automatically on the top corners and bottom right corner of the profile if it is rectangular. Irregularly shaped boards have an imaginary rectangle placed around them that is used for target positions - the rectangle is drawn as small as possible without

overlapping the real profile.

The *Target X & Y Shifts* setting is used to move the targets away from the corners of the profile. Unless output to gerber files, the target markers appear as 4 unfilled quadrants of a circle, with 2 diagonally opposite quadrants missing.

Target markers within gerber files are defined by the Dcode that is used to flash them.

Top pads if ticked, the pads of surface mounted components mounted on the top of the board (layer T) are output.

Any pads that have been added to layer T using the Amend commands, or those belonging to conventional components or vias are not included.

Bottom pads if ticked, the pads of surface mounted components mounted on the bottom of the board (layer B) are output.

Any pads that have been added to layer B using the Amend commands, or those belonging to conventional components or vias are not included.

Extra layer this setting allows copper (tracks, text and pads) from one specified layer to be included in the solder paste output, if required.

Pad shrinkage is applied to the pads from the extra layer.

For example, it may be a requirement to include a drawing number, layer identification, issue number, etc. in an output file. Tick the box to include the extra layer and type in the required layer number in the - *number* dialogue.

Layer # type in the number of the layer that should be included in the output if the *Extra layer* setting has been ticked.

Pad shrinkage the value entered here reduces the size of all the pads that are displayed and output.

This makes the pads that are output, smaller than the actual pads and therefore suitable for a solder paste mask.

For instance a pad shrinkage of 0.015" reduces all the pad sizes by 0.015". i.e. a 0.060" pad is output at 0.045", and a 0.1" x 0.050" pad is output at 0.085" x 0.035".

Target X/Y shift without a shift (0 specified) the target markers are positioned on 3 corners of the rectangular area occupied by the profile. The target markers do not need to be part of the finished artwork so a shift is usually entered to move the targets away from the edge of the profile.

A positive X and Y shift is used to offset the targets away from the board. Typically, the targets are offset by 0.5" or 1" (10mm or 20mm) in the X and Y axis.

The same shifts should be used for all the output files of a particular design.

Once the *Setup* window has been set as required, select **Close** to continue, then move on to the heading: Configure > Output Filter

Configure > Plot Source (with a Drill Sheet Task open)

This command should be selected to control exactly what is included in a drill sheet (drill drawing) when it is output. A sample drill drawing is shown in Figure 229.

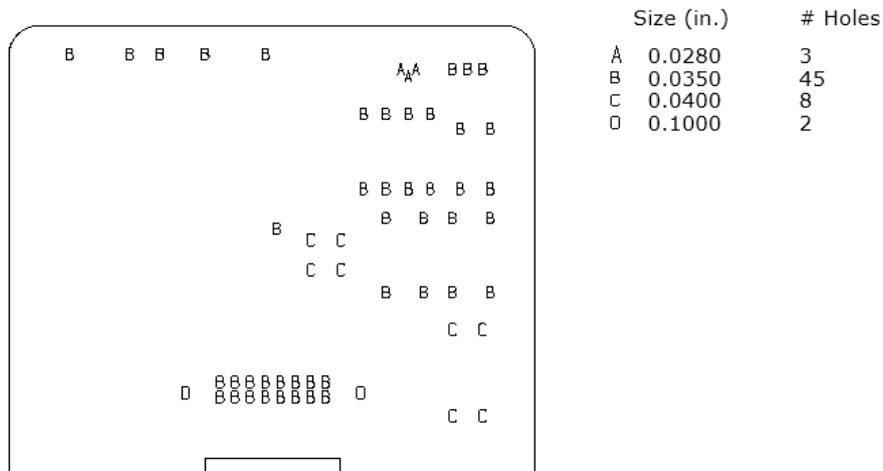


Figure 229

It shows the board profile, the positions of all the holes in the design using a symbol to represent each hole size, and a table listing the symbols, their sizes and number off of each.

The output could also contain the keep-out areas and target markers, if required.

If blind/buried vias have been used, then multiple drill sheets will be required for the different drill data sets. Refer to using blind/buried vias for more information.

Many companies like to add extra information to their drill drawings, so it is possible to direct the output to a documentation layer in the artwork editor so that the additional information can be added. (That modified layer would then be output using an Artwork Layers Task, not a drill sheet task.)

The drill table can be output in imperial or metric units (Edit > Units or the the special function keys can be used to toggle between the two units, however the Task will have to be closed/opened to see the effect of the change).

The *Configure > Plot Source* command can also be activated by selecting the *Setup* button alongside the Source setting in the Toolbar, arrowed in Figure 230

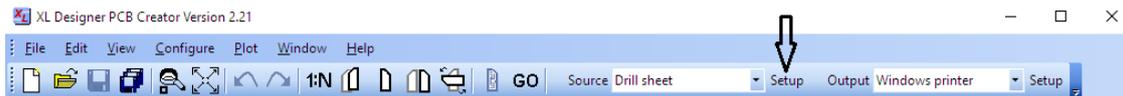


Figure 230

The *Source* setting (*Drill Sheet*) determines what type of data will be output and this can't be changed as it's controlled by the type of *Task* that's open.

When the command is selected, a window appears as shown in Figure 231.

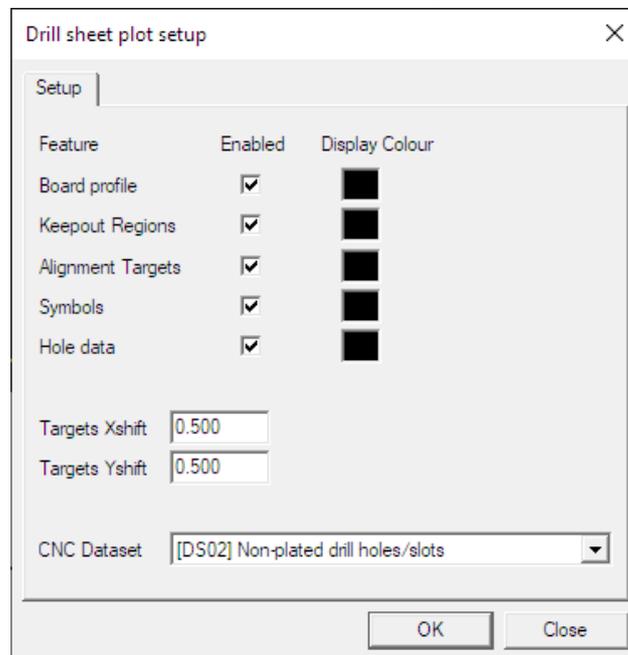


Figure 231

This controls what is included in the drill sheet output. Set the parameters as required, using the following information to assist you:

Board Profile if ticked, the board profile (as defined in the profile editor) is output. Typically the profile is included in a drill drawing.

Keepouts if ticked, the keep-out lines (as defined in the profile editor) are output.

Typically keepout lines are not included in output files unless there is a specific requirement, perhaps for a check plot/print in order to assess their positions.

Alignment Targets if ticked, three alignment target markers are output.

Target markers or "alignment markers" are used to align the layers of the board together when manufacturing the board. They would normally be included with all output files, and in the same relative positions.

The target markers are produced automatically by the output tool so are not visible in the artwork editor though they are seen in the output tool when selected.

The target markers are positioned automatically on the top corners and bottom right corner of the profile if it is rectangular. Irregularly shaped boards have an imaginary rectangle placed around them that is used for target positions - the rectangle is drawn as small as possible without overlapping the real profile.

The *Target X & Y Shifts* setting is used to move the targets away from the corners of the profile. Unless output to gerber files, the target markers appear as 4 unfilled quadrants of a circle, with 2 diagonally opposite quadrants missing.

Target markers within gerber files are defined by the Dcode that is used to flash them.

Symbols if ticked, the symbols that are used to represent each different hole size are output.

Hole data if ticked, the table of information that cross-references the symbols used, to the hole sizes and quantities, is output. Sizes in the table can be displayed and output in imperial or metric units - changing the units will require the task to be closed/reopened for the change to be made.

Display Colour

The data will be displayed and output in the colour indicated on the screen. Note: there may not be a close colour match between the display colour and that output to the colour printer/plotter.

Select the coloured box and choose a colour from those in the colour palette window that appears.

Target X/Y shift without a shift (0 specified) the target markers are positioned on 3 corners of the rectangular area occupied by the profile. The target markers do not need to be part of the finished artwork so a shift is usually entered to move the targets away from the edge of the profile.

A positive X and Y shift is used to offset the targets away from the board. Typically, the targets are offset by 0.5" or 1" (10mm or 20mm) in the X and Y axis.

The same shifts should be used for all the output files of a particular design.

CNC Dataset:

Choose from the list, by selecting the arrow alongside.

Typically an output is required for each drill data set.

Once the *Setup* window has been set as required, select **Close** to continue, then move on to the heading: **Configure > Output Filter**

Configure > Plot Source (with a Schematic Task open)

This command controls what will be included in a schematic when it is output.

The *Configure > Plot Source* command can also be activated by selecting the *Setup* button alongside the *Source* setting in the Toolbar, arrowed in Figure 232.

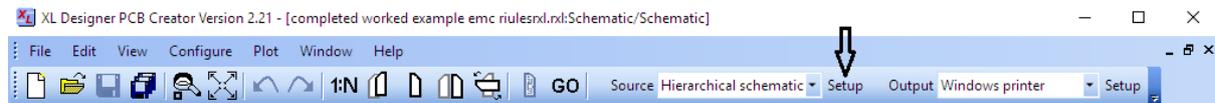


Figure 232

The *Source* setting (*Hierarchical schematic*) shows what type of data will be output and this can't be changed as it's determined by the type of *Task* that's open.

When the command is selected, a window appears as shown in Figure 233.

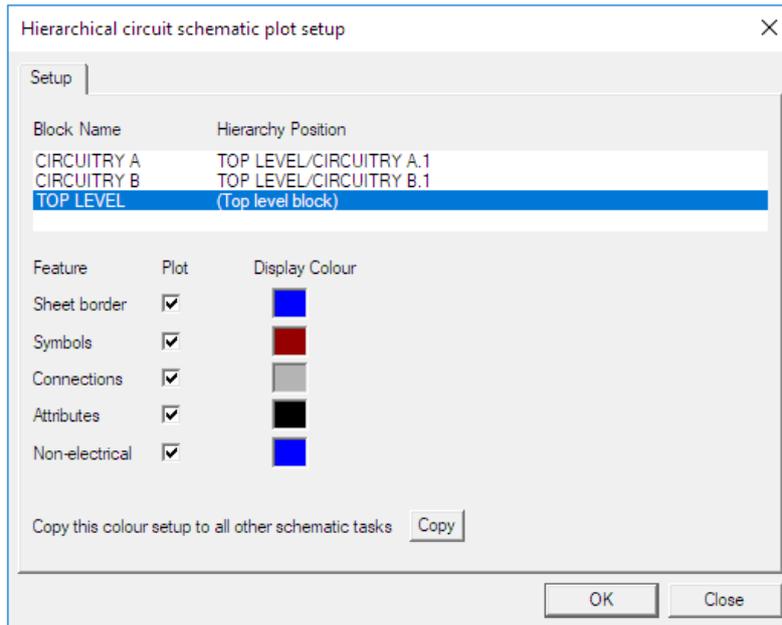


Figure 233

This shows which sheets are available for output, their position in the hierarchy and what is included in the output. Set the parameters as required, using the following information to assist you:

Block Name & Hierarchy Position

Select the sheet to be output from the list provided.

Each sheet has its position in the hierarchy shown, along with a suffix (.1, .2, .3, etc.) useful for identification when the sheet is used multiple times in the hierarchy.

Sheet border the default drawing border (lines and text) that appears when a schematic sheet is created, plus any non-electrical data added in *Drawing sheet* mode.

Symbols any thing added using the outline commands within a part.

Connections includes signal and bus connections and junction blobs.

Attributes the attribute text associated with symbols and connections. The attributes are only included if the associated symbol or connections are also included.

Non-electrical any non-electrical data added in *Annotate* mode in the schematic editor.

Display Colour the data will be displayed and output in the colour indicated on the screen. Note: there may not be a close colour match between the display colour and that output to the colour printer/plotter.

Select the coloured box and choose a colour from those in the colour palette window that appears.

Copy this colour setup to all other schematic tasks

Use this button to copy the Display Colours assigned to this sheet, to all the schematic tasks in this design – useful when consistent colour choices are required for all sheets within the design.

Once the *Setup* window has been set as required, select **Close** to continue, then move on to the heading: **Configure > Output Filter**

Configure > Output Filter

This command is used to control the type of output that is produced (Gerber, DXF, Bitmap, JPEG, Windows printer, etc.).

The output format is controlled by making a selection from the **Output** setting in the toolbar shown in Figure 234, select the arrow alongside the current setting and make a choice from the drop-down list, the choices are:

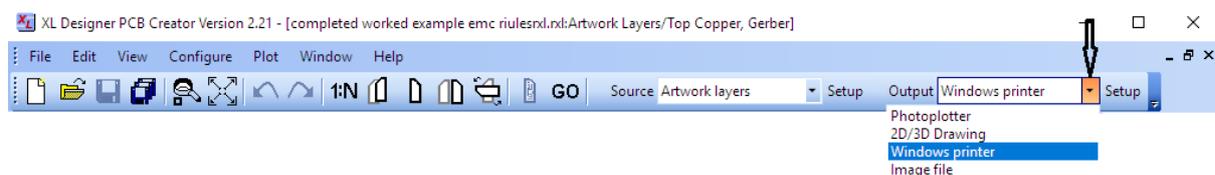


Figure 234

Photoplotter - for gerber files

2D/3D Drawing - for DXF files and re-directing data into the artwork editor

Windows Printer - for devices (printers/plotters/pdf) with a Windows printer driver

Image file - for bitmap, .png, .gif or .jpeg formats

Once the Output format has been selected, select the *Setup* button from alongside it, the windows that appear will vary depending on which type of output format is selected. All the variants are described below.

Configure > Output Filter - Photoplotter selected

Selecting gerber file output

Assuming the output task is open, ensure the *Output* dialogue is set to *Photoplotter* as shown in Figure 235.



Figure 235

Select the *Setup* button alongside it, or select the *Configure > Output Filter* command, to produce a window similar to the one shown in Figure 236 which indicates Gerber format RS274X is being output.

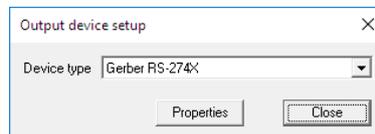


Figure 236

Select the *Properties* button to control the plotter variables which are shown below in Figure 237.

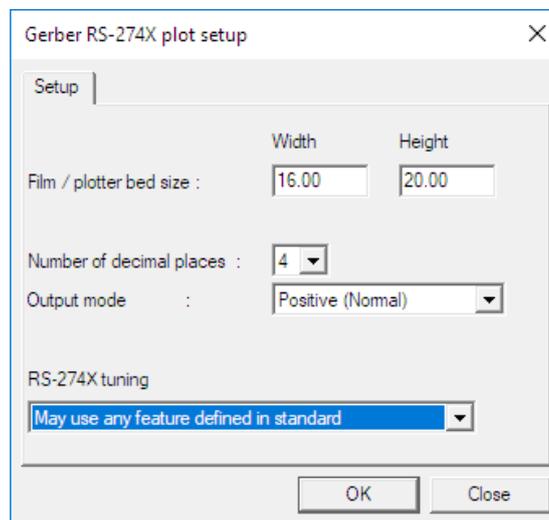


Figure 237

Set the parameters as required, using the following information to assist you:

Film/Plotter bed size: Select a larger size than the data you are outputting requires.

When the window is closed, the film is displayed on the screen as a white background. The dashed lines around the edge indicate the plottable area on the film, which is usually slightly smaller than the actual film. (Note: this setting does not control the size of film actually used in the plotter, it is shown purely for reference purposes.)

Number of decimal places: The co-ordinates in the Gerber file can be output with either 4, 5, or 6 decimal places. Depending upon the accuracy required in the output file, choose the appropriate number - 4 decimal places is adequate for most designs.

Output mode: Most outputs should be produced in *positive* mode.
However, when power plane, solder mask and solder paste layers are created in positive mode, they have to be photo-reversed in order to create the correct image for board manufacture.

A command can be inserted at the front of the gerber file to instruct the plotter to reverse the images of the supplied file. It is achieved by setting the output mode to *Negative (inverted)*.

This will save having to perform the photo-reversal procedure, but not all plotters support the command so you may need to check with the plotting bureau.

RS-274X Tuning: With "*May use any feature defined in standard*" selected, XL Designer will utilise the features described in the latest RS-274X specification document. This will result in the most efficient Gerber output but may result in incompatibilities with some older output devices/viewers.

With "*Simplify for marginally compliant devices*" selected, XL Designer will avoid known pitfalls in older, popular output devices/viewers that have partial or weak implementation of the RS-274X standard.

These are some of the problems we have identified within the implementation of the 'AM' (Aperture Macro) command in popular Gerber viewers, and this mode will attempt to work around them:

- Device requires an upper-case 'X' as the multiply sign in arithmetic expressions. The standard says it should be lower case.
- Failure to rotate the x/y position of circle primitive (code 1).
- Failure to rotate the x/y position of centre line primitive (code 21).
- Failure to position the centre line primitive (code 21) at the correct location, always placed at x=0,y=0
- Failure to handle multiple arithmetic operations in one statement.
- Failure to handle bracketing to control arithmetic precedence.

Please note that the 'simplified' mode does not imply that the generated Gerber will be shorter. It will in fact be longer and the AM definitions more difficult to read as various tactics are used to work around the above problem areas.

Once the window has been setup as required, select *OK*, then *Close*. The white background now represents the film that the data will be placed on. You may be able to see some of the data but don't worry if you can't.

Gerber output has now been selected you are ready to continue. Move on to the heading: Positioning and scaling the output data

Information on gerber files & Dcodes

A Gerber file does not contain pads, holes or tracks, or any electrical information.

It contains commands in a Gerber format to expose areas on a piece of photographic film.

The size/shape of the area is controlled by an aperture

If the film is moved whilst the area is being exposed, elongated areas are exposed.

These exposed areas become the "pads" and tracks" that you see on the film.

If the film was stationary when the light was switched on/off, the exposed area was "*flushed*".

If the film was moved whilst the light was switched on, the exposed area was "*drawn*".

Each aperture is given a unique number for identification, known as its "*DCode*".

The list of apertures and their associated DCodes is held within the header of the gerber file.

The Gerber file simply contains commands to select apertures, move to the required positions and either "flash" or "draw" on to the film. It does not contain pads, tracks, holes, text, etc.

Configure > Output Filter – 2D/3D Drawing selected

This Output type provides access to the *DXF* output **and** *Redirection of data to the artwork editor documentation layers* output.

Redirection of data to artwork documentation layers

Any data that can be output from the graphical tasks can be re-directed back into the artwork editor. This facility can be used to accommodate tasks that would not otherwise be possible, for example, it allows the drill sheet output to be modified, or a border defined in a schematic to be added to the artwork for documentation purposes.

When the output takes place, the selected layer automatically becomes a *Documentation* layer.

Assuming the output task is open, ensure the *Output* dialogue is set *2D/3D Drawing* as shown in Figure 238.



Figure 238

Once selected, select the *Setup* button alongside or select the *Configure > Output Filter* command, to produce a window similar to the one shown in Figure 239.

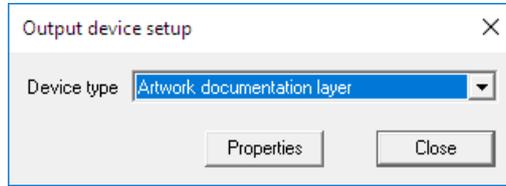


Figure 239

Ensure the *Device Type* is set to *Artwork documentation layer* as shown in Figure 239 (select the arrow alongside the setting to list the options available and make your choice if it's set differently), then select the *Properties* button to produce a window similar to that shown in Figure 240.

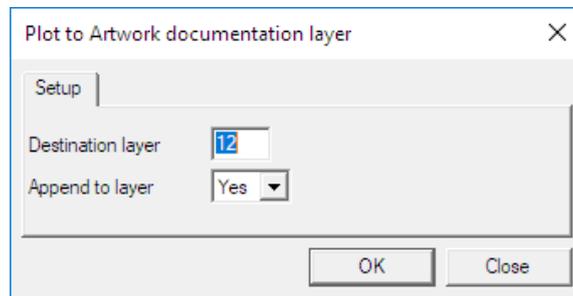


Figure 240

Set the window as required using the following information to assist you.

- Destination layer* the data that is output will be placed on the layer specified, in the artwork editor of the current design. Select the entry then type the layer number the data should be placed on.
Once the data has been added to the layer it can be modified in any way required. For example additional information could be added to a drill drawing, or the drill table moved/edited.
- Append to layer* the data being output may be added to, or replace any data that is already on the layer specified. With *No* selected, the data being output replaces any data that is already on the layer specified.

Once the window has been setup as required, select *OK* to continue.

The white background now represents the artwork editor and the layer the data will be placed on. You may be able to see some of the data but don't worry if you can't. You are now ready to continue. Move on to the heading: Positioning and scaling the output data

Note: if attempting to align the output (maybe the drill drawing) with the existing artwork, output the data with the X and Y shifts set to 0 (*Configure > Keymove Plot*).

Output to AutoCAD DXF file

Any data that can be output from the graphical outputs editor can be output to an AutoCAD DXF (.dxf) file.

Assuming the output task is open, ensure the *Output* dialogue is set *2D/3D Drawing* as shown in Figure 238.

Once set, either select the *Setup* button alongside (to the right) or select the *Configure > Output Filter* command to produce a window similar to the one shown in Figure 241.

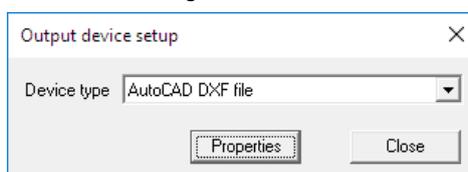


Figure 241

The *Device type* setting should be set to *AutoCAD DXF File*. If it isn't, then it must be changed by selecting the arrow alongside and selecting it from the list that appears.

Once selected, select the *Properties* button to control the output variables. An example of the properties window is shown in Figure 242.

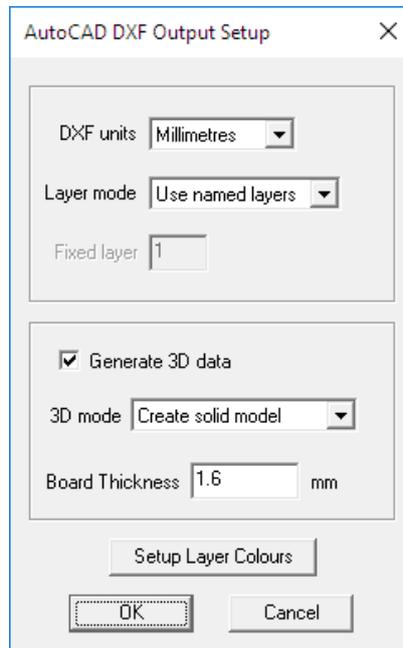


Figure 242

Set the parameters as required, using the following information to assist in your decisions:

- DXF units* the output data is produced in the units selected. Select the arrow alongside the setting and select the units required.
- Layer mode* this setting controls the conversion of the Ranger layers to corresponding DXF layer identities.
 - Use Named Layers* if this is selected, the .dxf output will be placed into layers that have symbolic names relating to the source that created the plot information. e.g. via holes will be placed into a DXF layer called VIAHOLES. This is the recommended setting for clarity in AutoCAD when used by Ranger users.
 - Fixed Layer Number* if this is selected, the .dxf data will be generated in the layer number specified in the *Fixed Layer* setting, which will become activated.
- Fixed Layer* this field becomes active when the *Layer Mode* is set to *Fixed Layer Number*. The fixed layer number may be in the range 1-16.
- Generate 3D data* when enabled (ticked) the .dxf output file that is subsequently created will contain a 3D representation of the board layout. The *3D Mode* & *Board Thickness* settings will become activated when this is enabled.
- 3D mode* if 3D data has been selected for output, two types of output file can be generated. One is a larger file that allows the resultant file to be viewed with shading, the other smaller file cannot be viewed with shading.
 - Create Solid Model* possible to display the resultant file with shading, larger file
 - Create Wireframe only* not possible to display the resultant file with shading, smaller file
- Board Thickness* if 3D data has been selected for output, this setting specifies the thickness of the board material.

Please note that as the board thickness is specified in the output filter, it has no knowledge of any plot scale factor that might have been used in the plot source setup dialog. So if plots are created at a scale other than 1:1, then the thickness value should be adjusted by a corresponding amount.

Setup Layer Colours this button permits the configuration of the colours used for the various layers. Three

settings are available by selecting the appropriate tab at the top of the window.

<i>Artwork Named Layers</i>	permits assigning colours to various artwork features when <i>Layer Mode</i> is set to <i>Use Named Layers</i>
<i>Schematic Named Layers</i>	permits assigning colours to various circuit schematic features when <i>Layer Mode</i> is set to <i>Use Named Layers</i>
<i>Numeric Layer Colours</i>	tab permits assigning colours to the numeric layers that are created when the layer mode is set to <i>Layer = Pen Number</i> or <i>Layer = Fixed Layer Number</i> .

Select **OK** to return to the DXF Output Setup window.

The white background in the output task window represents the paper that the data will be placed on. You may be able to see some of the data but don't worry if you can't.

The *DXF* output has now been selected. You are now ready to continue. Move on to the heading: Positioning and scaling the output data

Configure > Output Filter – Windows Printer selected

This output setting provides access to the Windows Printer drivers installed on the system.

Assuming the output task is open, ensure the *Output* dialogue is set *Windows Printer* as shown in Figure 243.

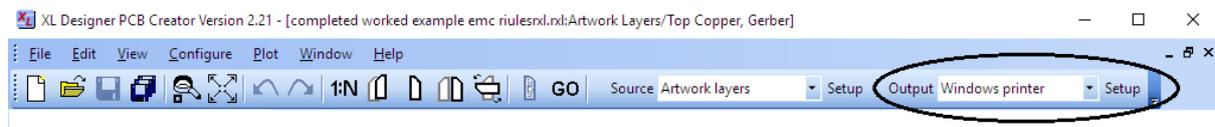


Figure 243

Once selected, either select the *Setup* button alongside (to the right) or select the *Configure > Output Filter* command to produce a window similar to the one shown in Figure 244. The actual window and printers available will vary depending on the printer drivers loaded on your particular machine.

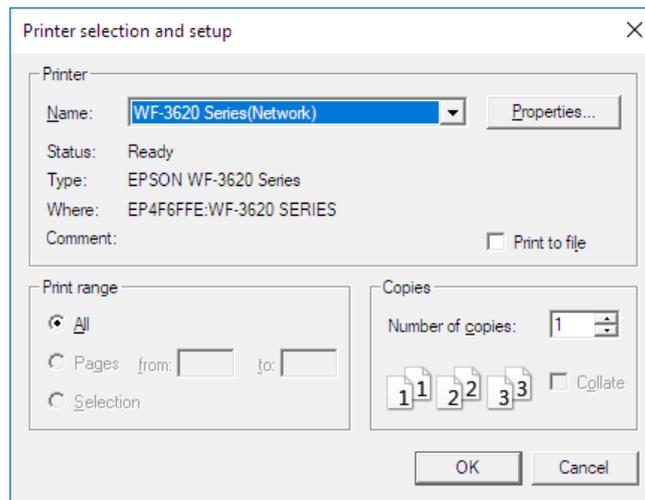


Figure 244

This *Printer* section of this window should be changed as required to suit the particular printer/plotter that is attached to your pc or network. Refer to the documentation supplied with the printer for help on the settings.

Printer selection

The printer selection and any selected printer options are retained with the task. If the design is later loaded on a computer not configured with the originally selected printer, then a warning will be displayed when the output task is opened and the printer selection will revert to the Windows default printer and settings.

It is possible to return the output task Windows printer selections to the default printer at any time - right-click on the output tasks folder and select "*Restore Default Windows Printer*", this will affect the Windows printer setting on all the tasks within the design.

Right-click on individual tasks and select the "*Restore Default Windows Printer*" option to restore the default Windows printer on the selected task only.

NB: Output task windows must be closed in order for the restore command to operate.

The *Print range* and *Copies* sections are not used by Ranger.

Select **OK** to continue.

The white background in the output task window represents the paper that the data will be placed on. You may be able to see some of the data but don't worry if you can't.

The *Windows Printer* output has now been selected. You are now ready to continue. Move on to the heading: Positioning and scaling the output data

Configure > Output Filter – Image File selected

This output setting provides access to the Image file output to produce files in a PNG, BMP, GIF or JPEG format.

When adding image output tasks to batch outputs, you must append the appropriate .png, .bmp, .gif, or .jpg file name extension to each output file name in order for the appropriate output encoder to be selected.

Assuming the output task is open, ensure the *Output* dialogue is set *Image File* as shown in Figure 245.



Figure 245

Once selected, either select the *Setup* button alongside (to the right) or select the *Configure > Output Filter* command to produce a window similar to the one shown in Figure 246.

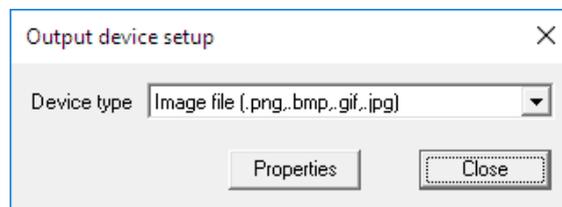


Figure 246

Select the *Properties* button to produce the output setup window as shown in Figure 247.

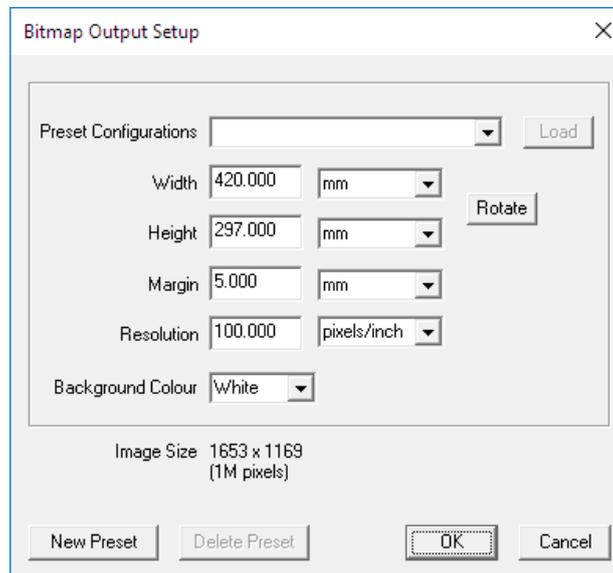


Figure 247

Set the parameters as required, using the following information to assist in your decision:

Positioning and scaling the output data

Once the source (circuit, drill drawing, solder mask, solder paste or artwork) and output format (gerber photo-plotter, printer, plotter, DXF, etc.) and have been selected, the output data has to be positioned on the screen representation of the paper/film, at the scale required. This ensures that it will appear on the physical medium when it is output.

Depending on the source data and paper/film size, you may be able to see part of the board profile or schematic

border on the screen at this stage, but don't worry if you can't. The following Configure commands and procedure will reveal it.

Making the output visible

If the schematic sheet outline or board profile cannot be seen, ensure you are viewing the complete output area (View > Full or press <F4>).

What is visible on screen is what will be output. The solder mask and solder paste output types are displayed on screen with the appropriate pad size expansion/shrinkage.

Select **Configure > Auto Centre** to bring the data to the centre of the white background.

If the data still cannot be seen, you may need to zoom in and pan (View commands).

If the data still can't be seen, select **Configure > Scale Plot, Fill sheet** to ensure the data is likely to be visible, for instance, an A1 schematic scaled at 10:1 is unlikely to be seen on an A4 back-ground.

These commands may not necessarily put the data where you want it to appear, but it's a good place to start.

Use some or all of the following commands to position & scale the data as required. Most of these commands have an icon shortcut in the toolbar.

Configure > Move Plot

Used to dynamically move the data into position on the white background. This command can only be used if the schematic sheet outline or board profile can be seen. If they can't, use the *Configure > Keymove Plot* command.

Once active, select a corner of the sheet outline/board profile, or the centre of a circular profile. Move the cursor and attached data then click again to release the data. Clicking the right-hand mouse button whilst the data is being moved, returns the data to its original position.

Note: wherever the data is positioned, it is located at a specific X and Y location. This location can be viewed using the *Configure > Keymove Plot* command.

When plotting artwork layers, solder masks/pastes it's wise to output all the files at the same X and Y location.

If the files will be imported back into the artwork editor, or aligned with the NC Drill files via a 3rd party viewer, then it's wise to output the data without "a shift", i.e. with the X and Y shift set to 0.

Configure > Keymove Plot

Used to position the data at an exact location by typing or "keying-in" co-ordinates.

This command can also be used if the data cannot be selected using the *Configure > Move Plot* command because the sheet outline or board profile are not visible and therefore not selectable.

Once selected, a window appears into which the offset values for the X and Y-axes should be typed. Select **OK** to implement the changes.

When plotting artwork layers, solder masks/pastes it's wise to output all the files at the same X and Y location.

If the files will be imported back into the artwork editor, or aligned with the NC Drill files via a 3rd party viewer, then it's wise to output the data without "a shift", i.e. with the X and Y shift set to 0.

Configure > Rotate Plot

Used to rotate the data, typically to make more use of the available paper/film. Each time *Configure > Rotate Plot* is selected, the data rotates through 90 degrees anti-clockwise.

Schematic sheets and artworks are rotated about their centre, i.e. the centre of the smallest rectangle that completely encloses the sheet outline or profile.

Configure > Mirror Plot

Used to mirror the data. The icon alongside the command (and in the toolbar) is highlighted when the data is mirrored.

Schematic sheets and artworks are mirrored about their original centre, Y-axis. For instance, they are mirrored left to right about their centre, unless they have been rotated by 90 or 270 degrees, in which case they are mirrored top to bottom about their centre.

Typically one outer artwork layer is mirrored. (To allow both outer board layers to be produced with the emulsion side up or down.)

Configure > Scale Plot

Used to increase or decrease the size of the output data.

Once selected, a window appears into which the required scale should be entered. For example, entering 2 produces output data at twice full size, entering 0.5 produces output data at half full size, etc. To save typing when a 1:1 scale is required, a button *Set 1:1 scale* is provided.

If the *Fill sheet* button is selected, the data is scaled up or down automatically to fill the plottable area of the sheet, which is indicated by the dashed lines on the white background. If the *Configure > Scale Plot* box is re-selected, the scaling factor used is displayed in the Plot scale dialogue.

Fill sheet is typically used when plotting circuit diagrams, because for instance, an A3 schematic will not fit in the plottable area of an A3 piece of paper.

Gerber files are generally produced at 1:1 and a warning will appear if a different scale is selected - the warning can be ignored if required.

Configure > Auto Centre

Used to position the data centrally on the sheet.

Plot commands

These commands are used to preview what will be output and to start the output.

Plot > Execute

Used to produce the output selected.

Once the appropriate source and output type has been selected, the data positioned and scaled as required, it is ready to be output.

Select **Plot > Execute** (or the **Go** icon).

If the data is being output to a file, use the Windows browser that appears to locate the folder where the file will be saved in, then supply a name for the file in the *Filename* dialogue. Select **Save** to create the file.

Plot complete will appear in the status bar to indicate that the output was successful.

If the data is being output to a device, the data will be spooled to the Windows device queue.

Plot complete will appear in the information bar at the bottom of the window to indicate that the data has been sent to the queue.

NC Drill & Rout Data Task

Drilling and routing output files can be produced in either an Excellon or Sieb & Meyer format. The file(s) contain information that describes the drilling data for the holes in the board, or the routing data for the board.

A separate NC drill output file (and Task) is required for each of the datasets:

- conventional through hole vias/component holes
- non-plated holes
- each set of blind vias
- each set of buried vias

Some drilling machines can also rout short slots, so the slots might be included with the appropriate drilling datasets.

Longer slots, particularly those that define the outer edge of the board may need their own separate output file. This is something that should be checked with the individual manufacturer.

Producing the NC Drill/Router output files

From the navigator pane and with the design open, open the *Outputs* folder. Right-click on the *Tasks* folder, select *New NC Drill & Rout Data Output*. Rename it to reflect the type of output file that will be produced.

If the design has plated/non-plated holes, blind/buried vias, etc., then more than one file will be required, so supply a name that reflects the type of output file it is. One task will be required for each CNC Dataset.

Open the Task by double-clicking it. A window similar to the one in Figure 248 appears. Each of the settings is described below.

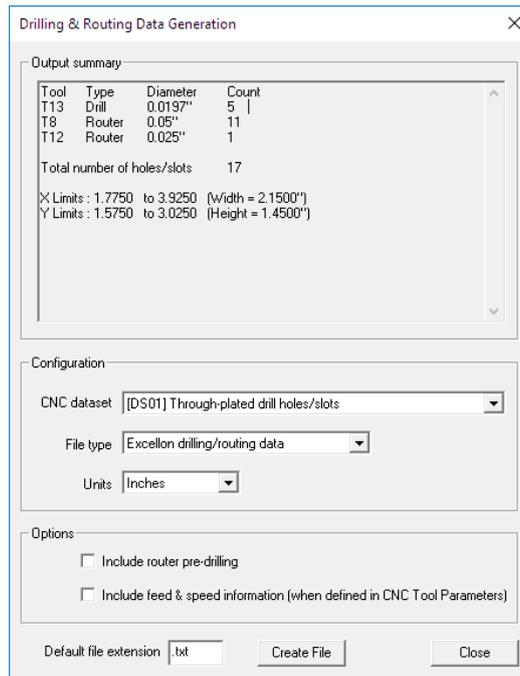


Figure 248

The window provides a summary of the output data, followed by the parameters that can be changed. The tool numbers, type, diameters and number of items obviously vary from design to design and this part of the window is updated automatically and cannot be changed.

Set the parameters as required, some are greyed out if they aren't applicable to the output type selected. Use the following information as a guide:

Configuration:

CNC Dataset: all the drills and slots included in the selected dataset will be included in the output file.

File Type: Either Excellon or Sieb & Meyer format files can be output. Both file types have the tooling information embedded in the file.

A separate file with just the tooling information can also be output for reference.

Select the format required - discuss this with the board manufacturer if you are unsure which to choose.

Units: An imperial (inches) or metric format file can be chosen. Select the format required - discuss this with the board manufacturer if you are unsure which to choose.

Options:

Include router pre-drilling: **all** of the pre-drilling information within slots can be included or excluded as required. If unchecked, pre-drilling details included in the slot will be excluded.

Include feed & speed info: if defined in the CNC Tool Parameters, the feed & speed information can be included or excluded as required.

Note: most modern drilling/routing machines have pre-set speeds and feeds for different sized drills/cutters, therefore consult your manufacturer before over-riding these values.

Default file extension: the output file will be given the specified file extension when created.

If an extension is specified in this field, but a different extension is typed in when the file is being created, then the extension specified at the file creation stage is used in preference to the default setting.

If no extension is specified in this field, then an extension is not added when the output file is created, unless one is specified at file output stage.

Once these parameters have been set as required, the output files can be produced.

When the drill datasets are used for drilling *blind* vias, it is essential that the holes are drilled from the correct

side of the board (top or bottom) and to the correct depth. Bear in mind that there could be more than one set of blind holes (drill datasets) drilled from the same side of the board, but to different depths. There is no drill depth information held within Ranger.

Use the *Create File* button to create the file. Use the Windows browser that appears, to locate the folder where you would like the file to be created in, then supply a name for the file in the *Filename* dialogue.

Select *Save* to create the file.

Over-lapped drill holes

When the NC Drill output task is run, a check is performed to identify any concentric drilled holes on the artwork. If any are found, a report appears as shown in Figure 249, before the NC Drill/Rout window appears.

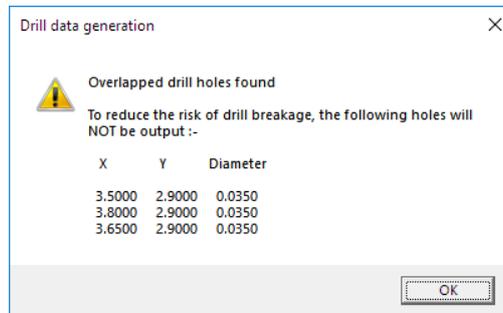


Figure 249

This report is given for information, it does not stop the output files from being produced - though it would be wise to investigate the cause of the overlapped holes, by viewing the artwork and locating the holes at the co-ordinates reported, then making any changes as appropriate. Typically vias may have been moved on top of other vias or component pads.

You can choose to ignore the report and continue with the output.

Although the report specifies "overlapped" drill holes, only concentric drill holes are identified, so there may be overlapped drill holes present that are not reported.

Where concentric holes are reported, only the largest hole at the specified co-ordinate is included in the subsequent output file.

The co-ordinate readout is given with respect to absolute 0,0 in the artwork editor. If the datum of the artwork editor has been moved, the co-ordinate readout of the mouse will not correspond to the co-ordinates in the report. The datum would have to be reset to 0,0 in the artwork editor to locate the position reported.

Select *OK* to close the report, the output routine continues.

IDF Output Task

This output task will create two output files. The ".emn" file contains the definition of the board outline and the placement information for all parts. By selecting the appropriate options in the configuration dialog window, it can also optionally contain board drilling information.

The ".emp" file is the part library which contains the outline definitions that the MCAD application "extrudes" by the component height along the Z axis to form the 3D representation of the part.

Producing the IDF Version 3 output files

From the navigator pane and with the design open, open the *Outputs* folder. Right-click on the *Tasks* folder, select *New IDF Output*. Rename it to reflect the type of output file that will be produced.

Open the Task by double-clicking it. A window similar to the one in Figure 250 appears. Each of the settings is described below.

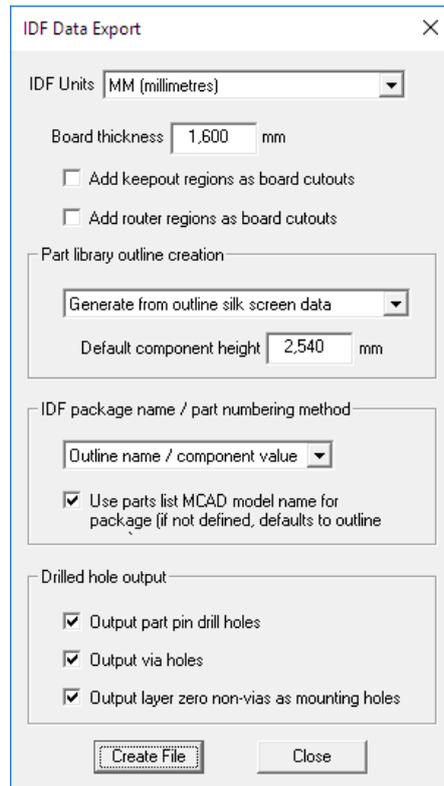


Figure 250

IDF Units: the output file will contain values in the units specified here. The MCAD application will take these units into account when importing the file to ensure the scale of the object is correct. The setting does not alter the units used in the MCAD application.

Board Thickness: this setting specifies the thickness of the board material. The units are always in inches for this setting.

Add Keepout regions as board cutouts:

When enabled (ticked) any closed polygon shapes that have been added to the “Keepout” layer (in the profile or artwork editors) will be output as cutouts within the IDF file.

Add Router regions as board cutouts:

When enabled (ticked) any closed polygon shapes that have been added to the “Router” layer (in the profile editor) will be output as cutouts within the IDF file.

Part library outline creation:

When the output files are created, there are 3 ways in which the part outlines can be defined. Choose the most appropriate to suit your requirements.

Generate from outline placement bounds box – this mode will create a simple rectangular block to represent each part in 3D. The dimensions of the box are taken from the auto-placement footprint definition in the component outline library entry associated with the part.

Generate from outline silk screen data – this mode will attempt to create the footprint shape used for the 3D figure by processing the data in the outline silk screen. If no suitable polygon can be created from the outline’s silkscreen definition (for example because the silkscreen is just a single straight line), then the auto-placement footprint is used as described above.

Use 3D solid mode setting on outlines – this mode will generate the part outline according to the 3D *solid mode* setting stored within each individual component outline. This allows for a combination of the above two settings to be used.

Default component height:

If the height of any component outlines has not been specified (or left at 0) within the component outline in the library, then the height specified here will be used for those outlines.

IDF package name/part numbering method:

Outline name / (blank) – this mode will create one part library entry for each component outline shape used in the design. If several parts of different types share the same outline, then they will all be mapped to the same part library name.

Outline name / component value – this mode will create one part library entry for each unique combination of outline name and part description (component value) field. This permits more fine-grained control of the mapping of part references to 3D part models when using mapping files - such as the "ecad_hint.map" file in Pro Engineer.

Use parts list MCAD model name for package:

When enabled (ticked), then any model name specified in the "MCAD Model" field in the parts list editor of XL Designer will be used as the IDF model name for the part instead of the outline name. If no MCAD model value is defined for a part, the component outline name is used.

Drilled Hole Output:

Drilled holes can be included in the output file if required. To allow fine-tuning of the types of hole to be included in the output file, the following selections are available:

Output part pin drill holes – if ticked will include all holes from component outlines.

Output via holes – if ticked will output all via holes (holes in standard code 0 round/square pads on the via layer "V")

Output layer zero non-vias as mounting holes – if ticked will output holes from pads on the via layer "V", excluding vias.

Creating the output files

To produce the files, ensure the appropriate selections have been made in the window using the descriptions above, then select *Create file*. Use the Windows browser that appears, to locate the folder where you would like the .emp (electro-mechanical neutral board) file to be created in, then supply a name for the file in the *Filename* dialogue. The job name will be used by default with a .emn file extension. Select *Save* to create the file.

Repeat to provide a name for the .emp (electro-mechanical part) file. The job name will be used by default with a .emp file extension. Select *Save* to create the file.

Two output files will be created, the ".emn" file contains the definition of the board outline and the placement information for all parts and board drilling information if selected.

The ".emp" file is the part library which contains the outline definitions that the MCAD application 'extrudes' by the component height along the Z axis to form the 3D representation of the part.

When opening the .emn file in Pro Engineer, it should be opened as an 'Assembly'. Prompts will appear requesting the location of the corresponding .emp file.

IDF output tasks may also be added to an output batch definition. The output filename should be specified without a trailing extension. When the batch is executed, the IDF exported will automatically append .emn and .emp to the output filename for the two files that it creates.

Outputs Folder - Batches

Once the output tasks have been created, they can be used in a *batch process* to save selecting each output task individually. One batch process might contain all the output tasks to create the gerber files for the job. Another might contain all the schematic sheets to be output to a printer. This ensures that the same settings are used each time the files are created and also saves time.

A batch process cannot be defined until the tasks to be included in it have been created/copied within the design's Outputs/Tasks folder.

In addition to running the batch process to output all the tasks within it, it is also possible to select and run an individual task from the batch process.

Batch files created in Ranger XL jobs cannot be converted to Seetrix XL Designer format, so they have to be re-created.

Creating a batch process

With a design folder open in the navigator pane, open its *Outputs* folder. Right-click on the *Batch* folder and select *New*, supply an appropriate name.

Names should help identify the type of batch process. Examples: "artwork layers, gerber", "schematic sheets, printing" etc.

Renaming a batch process

Once a batch process has been created/copied, it can be renamed by selecting it with a right-click of the mouse, then selecting *Rename*. Type in the name required followed by <enter>.

Copying a batch process

Once a batch process has been created and its content defined, it can be used as a template for other similar batch processes, within the same design or other designs, by copying it and modifying the copy. This will be quicker than creating another batch process from scratch.

With a design folder open in the navigator pane, open its *Outputs* and *Batch* folder. Right-click on the batch process to be copied, select *Copy*. Locate the destination *Batch* folder (which might be in the same or a different design) then right-click on the *Batch* folder and select *Paste*. The batch process is copied. It can be renamed or opened (double-select it) and modified.

When copying tasks/batches between designs, be aware that schematic output task/batches may need to be edited as the design sheet names are likely to be different

Deleting a batch process

A batch process can be deleted by selecting it with a right-click of the mouse, then selecting *Delete*.

Opening a batch process

With a design folder open in the navigator pane, open its *Outputs* folder, then double-select the batch process to open it.

A window appears similar to the one in Figure 251.

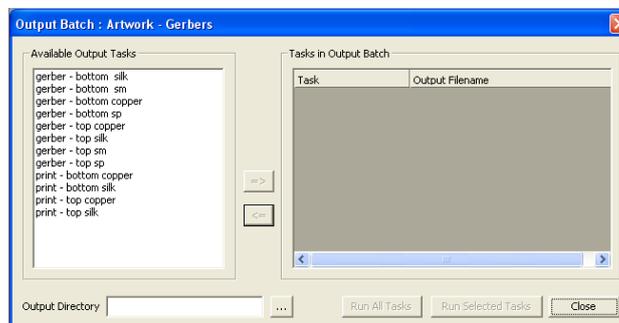


Figure 251

Adding/removing Tasks from the batch process

Each of the Tasks that have been created in the design appears in the left pane of the window. Each Task that should be included in the batch process has to be transferred to the right pane. This is achieved by selecting the Task, then selecting the Transfer arrow as shown in Figure 252.

More than one task can be selected at a time - to select multiple tasks, hold down the left mouse button whilst moving over them, or hold down the <ctrl> key whilst selecting individual tasks.

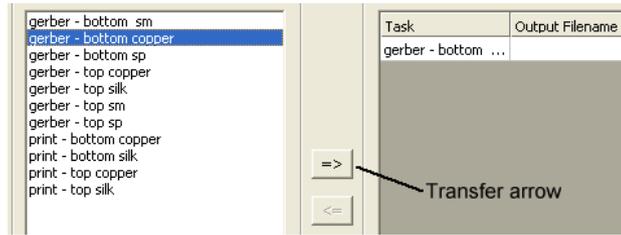


Figure 252

The task is transferred to the right window pane. Repeat for all the tasks required.

The tasks can be transferred between panes by selecting the task, then the appropriate (left/right) arrow.

Specifying filenames for the resultant output files

If the task will direct the output to a file (rather than directly to a device like a printer), then a filename has to be supplied for the new file – select the *Output filename* dialogue alongside the task in the right-hand pane and type in the filename required. A path should not be specified. Repeat for each task.

If a Windows printer is targeted and the field is left blank, the output is sent directly to the printer device. Adding a filename can be useful when targeting Windows printer devices such as *Print to PDF*.

NOTE: When adding image output tasks to batch outputs, you must append the appropriate .png, .bmp, .gif, or .jpg file name extension to each output file name in order for the appropriate output encoder to be selected.

Specifying the folder for the resultant output files

In the *Output Directory* dialogue, either type in the path, or use the browser button alongside to use the Windows navigator to select the folder required, Figure 253.



Figure 253

Running the batch process or an individual task from the batch process

When ready to run the batch process or a selected task within the batch process, select the *Run all tasks* or *Run Selected Tasks* button. (To select multiple tasks, hold down the left mouse button whilst moving over them, or hold down the <ctrl> key whilst selecting individual tasks.)

Saving the batch process

The batch process is part of the design, so is saved when the design is saved.

BSL (Bath Scientific Ltd) output files

An output file can be produced which can be used to drive a Bath Scientific bare board connectivity tester.

Creating the Bath Scientific output file

With a design folder open in the navigator pane, right-click on the design name, then select *Export > Bath Scientific Test Data*. A window similar to the one in Figure 254 appears:



Figure 254

Output Filename: type in a name for the output file that will be created. The browse button can be used to locate folders and/or files.

Select **Export** to produce the output file. Information appears on the screen to indicate what is happening. This information will be stored in the report log, which will be found in the design's *Logfiles* folder and named "*export_bsl*". A typical report is shown in Figure 255.



Figure 255

If problems are encountered they are reported in detail. Each one should be investigated and appropriate action taken.

Close the report window when you are ready to continue.

The export process is complete and the exported BSL file will be found in the location specified.

Output file format

The BSL file takes the following format:

Part	Pin	X	Y	Surface	Signal
Part	Pin	X	Y	Surface	Signal
Part	Pin	X	Y	Surface	Signal

Where:

Part is replaced by component reference i.e. IC1, R1, etc.

Pin is replaced by a pin number/name

X & Y is replaced by the X & Y position of the pin, with respect to absolute 0, 0 in the artwork editor.

Surface is replaced with the layer the pin appears on:

U = upper

L = lower

B = both

N = untestable

Signal is replaced by the name assigned to the track that is attached to the pin, i.e. VCC, GND, CLOCK, etc. If a name has not been assigned, then a unique name is assigned automatically. i.e. RNET0015, RNET0016, etc. Unconnected pins are given the name *_NC_1*, *_NC_2*, *_NC_3*, etc.

Lines preceded by ! are comments.

GenCAD output

An output file can be produced in a GenCAD ASCII format. The file contains information that describes everything in the artwork: board profile, pads, tracks, outlines, parts, connections and tracks.

This information can be used by Generic CAD Software (or any other software that can understand it) to drive various board manufacturing processes, such as automatic or manual component insertion machines, automatic test equipment (ate), etc.

Creating the GenCAD text file

With a design folder open in the navigator pane, right-click on the design name, then select *Export > GenCAD-format ASCII File Generation*. A window similar to the one in Figure 256 appears.

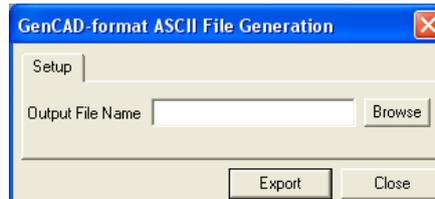


Figure 256

Output Filename: type in a name for the output file that will be created. The browse button can be used to locate folders and/or files.

Select **Export** to produce the output file. Information appears on the screen to indicate what is happening. This information will be stored in the report log, which will be found in the design's *Logfiles* folder and named "export_gencad". A typical report is shown in Figure 257.

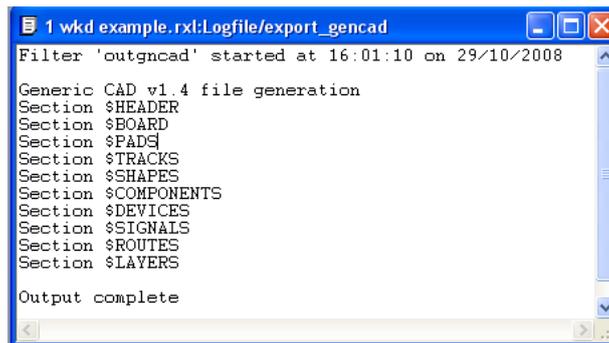


Figure 257

If problems are encountered they are reported in detail. Each one should be investigated and appropriate action taken.

Close the report window when you are ready to continue.

The export process is complete and the exported BSL file will be found in the location specified.

Output file format

The specification of the GenCAD version 1.4 format is available as a .pdf file upon request to: support@seetrax.com

Seetrax CAE Ltd, Woodstock, Hangersley Hill, Hangersley, Ringwood, Dorset, BH24 3JP, England

Telephone: 01425 489666 (overseas callers: + 44 1425 489666)

Fax: 01425 461641 (overseas callers: + 44 1425 461641)

Web-site: www.seetrax.com

Email: sales@seetrax.com
support@seetrax.com