

Changed in release 2.45 (15th November 2022)

Parts & Netlist editors :

- o The 'File->Save Part/Netlist as Text' dialogs now have a checkbox option to include design name and report creation date information at the top of the generated report.

General system :

- o BUGFIX: A crash was occurring when expanding master schematic library folders that contained drawing sheets.

Schematic user defined (GENEX) extraction :

- o BUGFIX: If a syntactical error in a .gex rule file caused extraction to fail, the error report was not being displayed.

- o BUGFIX: If user defined extraction was rerun without first closing the extractor and returning to the "Available Generic Extraction Tools" window, then erroneous syntactical errors were reported for the .gex rule file.

Language enhancements to the schematic user defined (GENEX) extraction :

Comments (#) may now be included within the '.format' section of the .gex files

New keywords have been added to the 'design' context and may be used in output statements to add design information to an output file :

'designname'	(string)	is the .rxl filename of the design
'filepath'	(string)	is the full filepath (directory + filename) of the design
'ninstances'	(int)	the number of symbol instances within the schematic
'nparts'	(int)	the number of components (parts) within the schematic

The set of keywords for access to attributes within the 'parts' context has been expanded.

The following keywords are now defined :

'ident'	(int)	the reference number allocated to a part (reserved attribute code 0)
'prefix'	(string)	component part code prefix (code 1)
'value'	(string)	component value attribute (code 2)
'outline'	(string)	outline library name for the part (code 3)
'ordercode'	(string)	the part order code field (code 4)
'symswaprule'	(string)	swapping rule for equivalent symbol instances (code 5)
'blockname'	(string)	the schematic library name of a part (code 6)
'partreference'	(string)	provides a concatenation of the prefix and ident

variable

User defined attributes may now be accessed within the parts context without the need

to use the 'attributes' context in conjunction with 'skipif' to pick out the required attribute.

To access user defined attributes from within the parts context, enclose the attribute name in

square brackets e.g. [Description]

(All user attributes accessed with the [] syntax are of (string) type even if they hold a number)

The 'parts' context keyword may now be followed with a 'groupby' keyword and a list of

attribute references.

When 'groupby' is specified, all parts that have identical values for all the attributes listed

after the groupbu keyword will become grouped together.

If any other attribute has a non-unique value across a grouped set of parts, that attribute value

is replaced with the text "*multiple_values*" if that attribute is output.

(for integer values associated with reserved keywords, non-unique values are returned as -1)

When 'groupby' is in use, the 'partreference' attribute becomes a comma separated list of the

part references for all parts in a grouped set.

The 'groupcount' variable becomes available in a grouped parts context, and refers to the number

of parts included in the group.

Changed in release 2.44 Patch 1 (11th August 2022)

Artwork editor :

o BUGFIX: If the connection point of a custom pad was moved, then artwork unroutes did not attach to the new location until the design had been saved, closed and reopened.

Master profile library :

o BUGFIX: A crash occurred when trying to create a new profile in the master profile library, or attempting to copy/paste a profile from a design into the master library.

Changed in release 2.44 (6th February 2022)

Artwork editor :

- o BUGFIX: A program crash was occurring when performing smooth edged copper fill if the design contained a part outline using a custom pad that contained an 'extra hole' definition but no pad shape.

- o BUGFIX: If a component outline had pins containing custom pads that had extra hole or slot features but no defined pad shape, then the extra holes/slots were not being displayed on the artwork view.

General system :

- o BUGFIX: When saving a design file, if the file name in the save dialog did not contain .rxl at the end of the name, but did contain other text at the name end which matched a file extension registered in the operating system, then the .rxl extension did not get appended to the name of the saved design.

Changed in release 2.43 (12th July 2021)

Parts list editor :

- o Added a "Make part locations relative to CNC datum" checkbox option to the parts list editor "File->Save Parts List as Text" dialog. When checked, the location of parts in the saved text file will be adjusted to be relative to the CNC datum point defined in the artwork.

N.B: This option is only available if the CNC datum is enabled in "Configuration->Gerber / CNC Output Options"

Changed in release 2.42 (10th May 2021)

Artwork editor :

- o It is possible to create a PCB by placing parts on the board, and then sketching in the required connectivity using the Amend and Mroute tools. However, it is not possible to easily create a net list from such designs, and this makes it impossible to utilise the artwork validation tool to properly check the design for clearance errors.

To assist with this use case, a new tool "Check->Build Netlist from Artwork" has been added to the artwork editor. This tool traces the artwork connectivity in a similar way to the artwork validation tool, and then creates a net list from the traced result. After the netlist

is created, it is then possible to use the artwork validation tool to check the design for clearance errors.

Also, by opening the resultant netlist in the netlist editor, it may then be exported in a variety of formats into external programs such as spreadsheet editors.

The netlist building tool cannot create a netlist where some connections are made using a power plane.

When executed, the netlist building tool DESTROYS any previous netlist before creating its own.

(Accidental destruction of your existing netlist is avoided by the tool first displaying a popup warning dialog, and requires the user to tick a confirmation checkbox before the tool can be executed.)

Netlist editor :

- o The ability to save files in .CSV format has been added to the "File->Save Nets List as Text" dialog.

- o Also added checkboxes to control the inclusion of option columns and the section start/end keywords.

Changed in release 2.41 (12th April 2021)

Artwork validation:

- o The artwork validation tool may now be accessed from a button on the artwork toolbar in addition to the normal menu entry.

- o An 'Revalidate Artwork' function has been added to the 'Check' submenu and the artwork toolbar.

- o This allows the artwork to be revalidated using the existing configuration values. The configuration and progress dialog is not displayed, and progress of the validation tool is indicated by a progress bar within the artwork toolbar.

- o Artwork validation now writes all errors and warnings into the 'artwork_validation' report file in the design's 'Log Files' folder. Whenever errors or warnings occur, the report file is automatically opened for viewing.

- o The artwork validation report has been improved to add more information to aid in the identification of detected problems.

- o The artwork validation report file now contains clickable links throughout the various report sections.
Clicking on these links will pan/zoom the artwork to the location identified in the report.

- o BUGFIX: The artwork validation tool was showing erroneous short circuits near the edge of copper filled regions.

Artwork editor :

- o BUGFIX: The 'Tools->Network->Pin Swap' command was not ripping tracks associated with the second of the two pins selected for swapping. There was also the possibility that tracks would be ripped from other nets not associated with the selected pins, or that duplicate net nodes would appear in the net list.

- o BUGFIX: When loading a design that had originated from the earlier RangerXL product, incorrect style pads were being shown on some part pins where the "Amend->Replace Pad" had been used in the RangerXL system to restyle the pin.

General system :

- o BUGFIX: A design that had internal inconsistencies in its schematic was causing a program crash at design load time. Error handling of schematic inconsistencies has now been improved and an error report will be displayed after loadup.

Schematic editor :

- o BUGFIX: If a schematic symbol contained a power pin that was also mentioned in the 'Attributes->Symbol->Part Power Pins' table for the part (blind power pins), then a duplicate net node would appear in the net list after schematic compilation. The duplicated nodes were then reported as 'unrouted' by the artwork validation tool.

- o Added display of the cursor X/Y position to the top status bar to assist in the location of errors that are reported by X/Y coordinate. Also added 'Units' selection to the Edit menu to enable the X/Y display to be switched between inch and metric modes.

Design netlist :

- o Any duplicate nodes found within a net will now be deleted at design load time.

Electra / Specetra autorouter interface :

- o A program crash was occurring when importing the routing result.

Parts list editor :

- o The ability to save files in .CSV format has been added to the "File->Save Parts List as Text" dialog.

Also added checkboxes to control the inclusion of a column headings row and the .PARTS / .ENDPARTS tags.

Changed in release 2.40 (15th February 2021)

General system :

- o BUGFIX: If a schematic sheet was renamed, and the sheet then opened using the right-click popup menu, then a crash would occur. The problem did not affect opening a renamed sheet by double-clicking on its name.

Artwork checker:

- o The artwork validation tool has been completely recoded to improve its checking speed and accuracy of short circuit location. The short-circuit checking 'cancel' button has been removed as this part of the verification process can no longer go into a deadlock state as could happen with the previous implementation. Work is still ongoing to implement checking of 'slots' and 'extra hole' data, and a warning will still be raised when validating a board that uses these features.

- o Track angle checking is now performed in the new artwork validation tool, and an option has been added to the tool dialog to allow configuration of the permitted track angles.

- o Angle errors will no longer be displayed for track segments that are completely covered by a part pad or via hole.

- o BUGFIX: Angle checking was showing track angle errors for 'partial unroute' features at the beginning and end of a track.

- o BUGFIX: The 'text in autoplacement footprint bounds' check did not work in the previous artwork checker.

Artwork editor :

- o The artwork checking tool is now named 'Validation Tool' in the 'Check'

submenu.

- o The angle check $0/90$ and $0/45/90$ commands have been removed from the 'Check' menu and track angle checking is now performed by the artwork validation tool.
- o BUGFIX: Gap clearance markers did not display correctly when the artwork display was in flipped mode.
- o CRITICAL BUGFIX: If the usage of a layer was changed from copper to powerplane after a user defined via hole had been designed with a drill range spanning the new powerplane layer, then no heat relief or antipad was being shown or generated by the powerplane tool.
- o BUGFIX: Unroutes were being shown on some boards even though the artwork checker indicated that the board routing was complete and correct.

CNC Drill output and drill sheet generation :

- o CRITICAL BUGFIX : If the drilling configuration for a user via hole definition was set as 'Include in drill data set' and "[DS01] Through-plated drill holes", then the holes were not appearing in the CNC or drill sheet outputs. (The holes appeared correctly if the via was configured as 'Include in Through-Board' drill data).
- o A 'Reset all tool codes' button has been added to the 'CNC Tool Parameters' configuration dialog to allow all tool definitions to be returned to the 'Undefined' state.

Gerber artwork output :

- o BUGFIX: If the heat relief 'Track Width, Clearance & Channel Width' configuration parameters were all set to zero for a a drill size range, then a small square was appearing in the Gerber output around the drill hole.

Changed in release 2.30 (25th March 2020)

Schematic editor:

- o BUGFIX: An 'improper argument' error was being displayed when trying to place a power symbol. (Problem inadvertently introduced by associated code fix in release 2.29)

Outline editor:

- o BUGFIX: There was a possibility of data corruption to outline definitions

while outline pin numbers
were being assigned.

Changed in release 2.29 (24th March 2020)

CNC drill outputs and Gerber artwork output alignment improvements:

XLDesigner's Gerber output facility presents the user with a preview window displaying a white media area whose size represents the media dimensions configured in the 'Output device setup->Properties' dialog.

The bottom left hand corner of the media area is represented by the location $X=0/Y=0$ in the generated

Gerber file. Because of this, the coordinates of features in the generated Gerber data will not correspond

with the coordinates of those features in the artwork editor or in generated drill output data.

However, the Gerber outputs have always operated in this way as in the past, customers needed to be able

to exercise complete control over the photoplotting of their artworks and the positioning of the plot on

the output media, which was normally film.

To fit in better with modern workflows, the default behaviour in XLDesigner is to now ensure that the

coordinates in CNC drill data and Gerber artwork data are aligned, with the user being able to specify the

location of the $X=0/Y=0$ common datum point for the drill & gerber outputs.

The artwork and profile editors now display the CNC/Gerber output common datum location as a small white

square with an X inside. The default position of this datum is at the system $X0,Y0$ location, and may be

changed using the 'Tools->Set CNC Datum' command. The CNC Datum mark may be hidden using the View menu.

Other changes:

General system :

- o BUGFIX: When opening a design created with XLDesigner version 2.21 or earlier, any CNC router paths

- that had been defined are automatically converted into 'slot' features.

However, the slots should have

been assigned to the 'DS2 Non-Plated' CNC dataset, but were being wrongly allocated to the DS1 Plated

- dataset. Also, when such a conversion is performed at design open, then

slot visibility is forced on

- in the artwork editor so that the imported router paths can be seen.

- o BUGFIX: The 'File->System Setup' dialog was allowing the same key to be assigned to more than one special

function key action. This would result in unexpected results.

- o BUGFIX: The 'Move part' special function key never worked regardless of which keyboard key was assigned to it. It will now activate artwork part move mode, schematic symbol move or schematic split part subsymbol move depending on which type of editor window it is invoked from.

Artwork editor:

- o The 'Parts->Set Datum' command (which changed the X/Y display origin and the origin when querying part locations) has been replaced by the 'Tools->Set XY Datum' command.
- o BUGFIX: The angle checker was incorrectly placing error flags on the dotted, unrouted section at the start/end of a partially routed trace.
- o BUGFIX: If the angle checker found errors, it did not switch on the display of the error flags if they were switched off.
- o BUGFIX: In certain situations, the software became unresponsive when entering the split powerplane edit mode.
- o BUGFIX: In the silk screen generator dialog, it was possible to type 'V,T or B' into the control that selects the destination layer for generated component outline or label features. If silk screen data was generated with such settings, then the V,T or B layer had its reserved layer type changed from Copper to Silk Screen, which resulted subsequent failure of many other facilities throughout the software. Also, after the reserved layer type had been changed, there was no way for the user to return the layer type to the correct setting. It is still possible to type in V,T or B as a target layer, but the situation will now be trapped and no data generated or layer assignments disturbed.

Profile editor:

- o The 'Profile->Set X/Y Display Datum' command has been replaced by the 'Tools->Set XY Datum' command.

Profile & Artwork editor common changes:

- o When the X/Y display datum has been moved away from the system origin, a *Datum moved* warning will always be shown to the right of the displayed X/Y coordinate.
- o When the X/Y display datum is at the system origin, the yellow datum mark will no longer be displayed.

Schematic editor:

- o BUGFIX: When 'Symbol->Power Place' is selected, an automatic check for duplicate part identities is performed before the part power symbol selection dialog is displayed. The log file for displaying errors detected by this test was not being initialised correctly, so if you tried to place power symbols for a design having duplicate part identities, the software would crash.

Design/Masters Configuration section:

- o A new 'Gerber / CNC Output Options' category has been added. This new configuration dialog currently contains just one setting, 'Lock CNC drill & Gerber outputs to CNC datum'. If this option is unticked, then the Tools->Set CNC Datum command will be greyed out, and the CNC drill and Gerber outputs will revert to their previous modes for choice of datum point.

CNC drill outputs:

- o BUGFIX: An invalid Y coordinate was sometimes being output for drill holes associated with used defined via holes.

Drill sheet generator:

- o User defined via holes had a symbol listed in the drill table, but the location of the user via holes was not being indicated on the drilling sheet.

Changed in release 2.28 (23rd January 2020)

Artwork editor and artwork checker:

- o BUGFIX: If a split powerplane boundary was drawn over the centre of a drill hole associated with the split pane's net, then a short circuit could result between the two nets on either side of the split powerplane boundary line, but this was not detected by the artwork checker. Also, when designing the split powerplane boundaries, the 'flashing dot' at the centre of the drill hole would stop flashing suggesting that the pin was properly within the split powerplane polygon.

To correct this situation, a new parameter 'Drill to split powerplane minimum clearance' has been added to the (Design)->Configuration->Manual Routing Parameters dialog. This new parameter specifies the minimum clearance that must exist between the edge of a drill hole and the boundary of the split powerplane.

Any drill hole that violates the configured minimum clearance will now be detected by the artwork checker, and a 'PP' error flag will be added to the artwork display. In the split powerplane designer, the 'flashing dot' will also continue flashing for drill holes that violate the clearance setting.

The initial default for this new parameter is 0.01 inches, and the default value for all new designs may also be set in the MASTERS->Configuration->Manual Routing Parameters dialog.

Changed in release 2.27 (17th January 2020)

General system :

- o BUGFIX: It was possible for a design outline to be created with a placed pin having no pad assigned to any pad layer. This resulted in incorrect operation in the artwork checker and Part->Find function.

DXF profile/keepouts importer:

- o BUGFIX: The importer was not correctly handling DXF file LWPOLYLINE entities.

- o BUGFIX: The importer was incorrectly importing graphic features defined in DXF file sections other than the ENTITIES section.

Changed in release 2.26 (2nd December 2019)

CNC drill and Rout data output changes:

When zero diameter drill holes are detected, the log file 'cnc_drill_rout' is now created/updated with location information for the affected items. Also, the zero drill diameter locations will not now be output to the data file.

When overlapping drill holes are detected, their location information will now be written to the 'cnc_drill_rout' log file instead of being displayed in the output summary window of the drill output dialog.

Added new 'Output' selector to the output dialog to allow selection of outputting drill data only, router data only, or both drilling & routing to the output file.

Added checkboxes to the CNC drill/rout data output dialog to allow control of

comment insertion into

the part program header and/or body sections.

(Please note available Excellon specifications only mention that comments may be included in the part program

header section, and do not mention that they are permitted in the program body.

For this reason, the default setting for comments in the part program body is now 'off'

).

Added 'Tooling data section' checkbox to the output dialog. This enables/disables the insertion of

tooling data into the part program header (or at the file end for Sieb & Meyer format).

In Excellon output mode, it is now possible to select the mode of zeros suppression used in the output file, or to switch off zeros suppression entirely.

Changed the default symbol used to mark the end of the part program header in Excellon format from 'M95' to '%', as the M95 code does not seem to be well supported on CNC drilling systems.

When CNC data is output in metric mode, the data values in the output file are limited to the range +/- 999.999 mm.

(approx. 39 inches), and in inch mode to the range +/- 99.9999 inches.

The data points output to the CNC file are always relative to the system datum, so if the artwork is shifted

a long way right (or up) from the system datum point, it is possible for values to go outside the above ranges.

The CNC drill/rout output tool will now display an error dialog if an attempt is made to create an output file

when data is out of range. (It will not be possible to create the output file until data is brought within range).

Added 'Advanced Options' button to the drill tape output dialog. This button appears in Excellon output mode, and

permits more fine grained control over commands that are included in the generated data file. As 'Excellon' format

is not a precisely defined standard, and its interpretation varies amongst various CNC drilling systems, these extra

controls should give users the ability to adapt the drill data output to conform to their PCB manufacturers data requirements.

Please note that there is a checkbox called 'Append % to the end of the file'. From customer feedback, it appears

that certain Chinese manufacturers are now unable to read Excellon drill data files unless a percent mark appears

at the end of the file. No previous versions of XLDesigner (or earlier Ranger products) have ever included this mark

at the end of the data, and its requirement is not stated anywhere in available Excellon format documentation.

Nevertheless, the option has been included for those customers who need it.

To achieve Excellon drilling data output that is identical in format to that generated by pre 2.22 software releases,
please set the following configuration :

In the main drilling dialog, select Output = 'Drilling Only', Zeros Mode = 'Include All' and untick the
'Comments in Part Program Header' and 'Comments in Part Program Body'
checkboxes.

In the Advanced Options dialog, tick the 'Completely omit the Part Program header' and
'Omit G90 Absolute mode command' checkboxes.

Please note that the resultant files have poor compatibility with most CNC drilling systems as they lack the required
'M48' header start code, specification of the units used within the file, and no tooling data.

General system :

- o BUGFIX: A crash could occur when opening a design where artwork 'partial unroutes' had been eliminated
by moving the end of the routed trace onto its corresponding part pad.

Artwork editor :

- o BUGFIX: A crash was occurring with designs that contained part outlines with standard pads placed within
the pre-routed copper section of the outline.
- o BUGFIX: A crash occurred if you attempted to route an unrouted trace when no layers were enabled for routing.

Changed in release 2.25 (8th August 2019)

Profile editor:

- o BUGFIX : changes made to the profile using the numeric editor were not being saved.

General system :

- o BUGFIX : changes or additions to keepout areas were not being saved.

Changed in release 2.24 (28th June 2019)

o BUGFIX : A crash was occurring when trying to open items in the master outline library.

o BUGFIX : Gerber output was generating non-RS274X compliant macro definitions when negative numbers were required in the output. (the leading '0' was missing)

Changed in release 2.23 (26th June 2019)

2.23 is the same as 2.22, but fixes a serious artwork autorouter problem which was caused by changed code optimiser behaviour in the C++ compiler used for the system build.

Changed in release 2.22 (25th June 2019)

===== New feature =====

A new 'Slots & Extra Holes' tool has been added to the Artwork, Outline and Custom Pad editors.

It allows insertion of slots (for CNC routing) into the artwork, or into an outline or custom pad definition.

It also allows arbitrary extra plated/non-plated drill locations to be inserted into the artwork/outlines/custom pad editor.

Long machined slots are often used to allow breaking out of a board from a larger panel, and to assist this, the slots tool can insert breakout tabs into any defined slots. The tabs may be solid or perforated (a series of close spaced small holes), defining the location where the board will break. The slots can be pre-drilled (to avoid damaging the router cutter) if required.

The CNC drill and router data outputs have been updated to accommodate these changes.

The resultant CNC output files now include tool definition information embedded in the file.

A new configuration panel 'Configuration->CNC Tools & Datasets' has been added. It allows pre-definition of all drills and routing tools needed for the board manufacture. It optionally allows feed rates and RPM values to be specified, and if entered, these will appear in the CNC output file.

The 'CNC Datasets' panel allows the naming of all the various CNC output files that will be needed for the manufacture of the board. Two datasets are pre-defined, DS1 which

represents all normal through plated holes (passing all the way through the board), and DS2 which represents through board holes which should not be through plated.

Additional dataset positions may be used to represent drill strategies for boards with blind or buried via holes. For each user defined dataset, the drilling start and end layers may be set, and a graphic representation of the drill hole in the board structure will be shown.

Any drill dataset numbers that were previously defined to handle drilling of user defined via holes will be automatically imported into the new CNC Datasets table on design loadup.

Known limitations with this new feature:

With this first release of the 'Slots & Extra Holes' tool, there are some limitations which will be addressed in the next release. These are :

The artwork checker is currently blind to the presence of slotted features and extra holes added to the design, so this must be born in mind when added slots to your board.

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.

===== Other fixes =====

Artwork editor :

- o BUGFIX: When using Identify->Pin then clicking the 'Find pin in schematic' button, the pin was not located if it was on a subsymbol of a schematic split part.

Schematic editor :

- o BUGFIX: If a schematic sheet contained a comma in its name, then the Find->Part command would not find any parts located on the sheet. Also, sheets could not be opened by clicking on the associated block in the hierarchy view if the sheet name contained a comma.
- o BUGFIX: Find->Instance did not work correctly if the symbol instance was a subsymbol of a split part.
- o BUGFIX: When given a part pin reference, the Find->Part function was locating the symbol instance instead of placing the popup balloon window on the requested pin.
- o Renamed the Find->Part menu entry to 'Find Part / Part Pin' to more correctly indicate its action.

Outline editor :

- o BUGFIX: Library pad drill holes were not showing in the outline editor.

General system :

- o The license activation dialog was enabling the activate button even if no free licenses were available for activation.

Changed in release 2.21 (27th February 2018)

General system :

- o BUGFIX: A crash occurred on opening a schematic sheet that had been copied and pasted from another design.

Changed in release 2.20 (9th February 2018)

New feature :

Added an artwork 'Logo' facility to the system. This allows the insertion of company logos, compliance and warning symbols etc. into PCB artwork layers.

Photos or scanned copies of logos may be loaded into the system's new logo editor, and then converted into a vector based representation suitable for inclusion into the artwork.

Logo images may be imported in JPEG, GIF, BMP or PNG format.

A new folder 'Logos' will now appear within user designs and the 'masters'

folder in the navigation pane.

Right clicking on the logos folder permits creation of a new logo.

Double-click on the logo name and the logo editor will open to allow loading of your desired image and conversion into a vector based version.

Logos 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/Paste.

When you open the logo editor, you will see that it is divided into three sections selectable by tabs at the top.

When you initially create a logo, only the first 'Select Source Image' tab will be selectable until you have loaded an image.

Click the 'Select Image File' button and choose an image file containing a logo that you would like to use on your pcb.

After selecting an image file, your image should display in the window. Use the rotate and flip buttons

to put the image in the desired orientation. Near the edges of the image, you will see border lines with a

small selector square at their centre. Click and hold the left mouse button on these small squares to drag

the border lines inwards to select just the part of the image that you want to convert into a logo.

(The unselected parts of the image will go dim as you move the borders).

Once you have your image loaded and cropped down to the appropriate region, click on the 'Thresholding'

tab at the top of the dialog. This page is divided into two sections.

The left hand window shows a monochrome rendering of the original image. If the original image is in

colour, you can use the R,G,B sliders 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 panel of the Thresholding page is where the monochrome image is converted to black/white only

representation. The 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 you have achieved a good image in the right hand pane, select the 'Vector Generation' tab.

This page will show you how the logo will look when it is split into individual vectors.

The 'pitch' control allows you to control 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 final page of the dialog, the drab green colour represents the laminate colour of the pcb, and the white is copper. The boundary of the drab green colour represents the bounds of the logo that has been generated, and also the boundary that is used during artwork checking. (artwork checking only considers logos to be rectangular boxes, it does not check for clearance to each individual vector within the logo)

Once you have an acceptable logo in the 'Vector Generation' page, you can close the Logo Editor.

Within the artwork editor, select Logo->Insert, then choose your logo from the navigator and drop it on the board wherever you wish. During placement, the logo's orientation and size can be adjusted using the controls in the logo toolpane. When checked, the 'Constrain Proportions' checkbox will ensure that the logo does not change shape when changing its width or height.

When using Logo->Move/Copy/Edit, the destination layer may be changed, so you can move existing placed logos to different layers.

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. ** 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. **

For customers installing XL Designer for the first time, a few example logos may be found in the MASTERS/Logos library that will be installed automatically.

The example master logo library WILL NOT get installed for existing users 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)\Seetrax\XL Designer\SampleDesigns') and dragged from that design into the master logos folder if you wish to use them.

General system changes:

- o Because of internal database changes needed to implement this new facility, designs saved with this release will not load into earlier versions of XL Designer.

Changed in release 2.19 (27th February 2017)

General system :

- o Revised the layout of the File->System Setup dialog to provide a more organised layout capable of supporting future setup options.

- o Reimplemented all text report file printing throughout the system. Printed text outputs now 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 by the user in the new 'Text report formatting' section of the system setup dialog.

- o The File->System Setup dialog now contains a new section "Fonts" where the font used by printed text reports can be configured. There are also additional entries in this setup section to control the font in the schematic editor and various output tasks. (See the schematic editor and output tasks comments below.)

- o Printed text reports which have information organised into columns now 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.

- o The design autosave section of the setup dialog now contains a 'Autosave to alternate directory' checkbox.

When checked, you may now choose to have autosaved designs stored in a different directory to the one in which the design file is located. For data security in the event of filesystem corruption or hardware failure, it is advisable to store autosaved work on a different filesystem to the one that holds the design file.

- o BUGFIX: Corrected a display rendering error that could occur if a custom pad was created with no boundary shape defined.

- o BUGFIX: Fixed a crash that could occur when loading designs originally created on very early versions of the Ranger3/RangerXL product series.

Schematic editor :

- o The schematic editor now displays all text using Windows text fonts. The font used by the schematic editor is controlled by the "File->System Setup->Fonts->Editors & Output Tasks" setting. Please note that as most Windows fonts use proportional spacing, text strings will be slightly shorter in overall length than with the previous fixed pitch vector font. Text height remains the same and is still controlled by the height properties on attributes, non-electrical text items etc. Because text height is specified by attributes, the 'point size' selectable when choosing the font in System Setup is fixed at an arbitrary 16pt and has no bearing on the resultant text size.

There is a checkbox in the fonts setup category that when selected causes the schematic to revert back to using the vector text font.

Artwork editor :

- o Added a new checkbox "Trk ID" to the artwork view control pane. When checked, and when the artwork is zoomed in sufficiently close, net identities will be displayed on all tracks.
- o It is now possible to use the grid origin command to select a new grid datum point while in the process of moving/copying/rotating a region. After the new grid origin is selected, right click the mouse to return to the currently active region command. (This reinstates the former XLD V1 behaviour).
- o BUGFIX: If the outline name for a part was changed to a non-existent outline after the part had already been placed on the artwork or added to the part placement tray, then a crash would occur when trying to move or place the part.
- o BUGFIX: Fixed a problem where the wrong command was sometimes being activated after clicking a toolbar button. (problem since revision 2.11)

Parts list editor :

- o Improved the layout of the Bill of Materials "report" style output.
- o Added a new checkbox "Include part references". When unchecked, part references will not be included in the report. This reduces total page count and assists in fitting the report to the available page width.

Output tasks :

- o Output tasks that target a Windows printer now use the "Editors & Output Tasks" font configured in the system setup dialog when appropriate for text output. With outputs such as 'PDF', this results in the text being searchable in the resultant document.

The onscreen preview of an output automatically switches the display of text between real font based and vector font determined by the capabilities of the output device. (e.g. Gerber output can only render text as vectors). Please note that "output to image file" continues to output all text as vectors and this will be corrected soon. Also, artwork text is always vector based in the editor and outputs because of the strict requirements of dimensional control and clearance checking.

- o Added a 'Copy' button to the schematic plot setup dialog. When clicked, this will copy the colour configuration from the current task into all other schematic output tasks.

Changed in release 2.18 (19th December 2016)

General system :

- o BUGFIX: Fixed serious Direct2D graphic interface leak that caused resource starvation and program crash.

Changed in release 2.17 (13th December 2016)

General system :

- o BUGFIX: Fixed a serious thread handle leak related to display rendering which could result in resource starvation after many display repaints had occurred.

- o BUGFIX: Improved thread termination event wait to minimise the potential of program hang during rendering.

- o BUGFIX: Fixed a crash that occurred when trying to delete custom pads from the master library.

- o Added 'Configure Crash Dumps' configuration tool to the System Setup dialog. This new facility allows the user to enable the local storage of user mode crash dumps which contain diagnostic information which can assist the product development team in the location of program errors. To access this configuration tool, XLDesigner must be run as administrator, but once dumps are configured, the software can be restarted as a normal user and crash dumps will be stored in the user-selected directory if an application crash

occurs.

- o Added extra internal checks to ensure that temporary file names are correctly created.

- o At design load time, the settings in the track sizes table will be checked for illegal values and an error report displayed of any incorrectly defined values. (Solution for customer reported error where silk screen print was showing a large black filled area because a track width value was negative)

Artwork editor :

- o BUGFIX: SMD pads that must connect to a power plane are marked with an unroute dot. When power unroute display was enabled, the unroute dots were incorrectly being displayed on the bottom left corner of the design for parts that had not yet been placed.

- o BUGFIX: A crash could occur if the mouse was double-clicked when placing parts.

Output Tasks :

- o Reimplemented the bitmap image output mode that was temporarily removed at the 2.14 release.

- o BUGFIX: If a design had definitions for user defined via holes and none of the definitions were referenced in the artwork, then a crash could occur in Windows printer outputs.

Changed in release 2.16 (14th October 2016)

General system :

- o BUGFIX: If space characters occurred at the start or the end of a license key when it was entered into the license management dialog, then the license code appeared to be stored correctly, but online activation attempts resulted in a license server error 3 being shown.

Output Tasks :

- o BUGFIX: Output tasks that targeted a Windows printer did work correctly when run as part of a batch output.

- o BUGFIX: When a drill sheet was output to a DXF file, the drill symbols were not shown correctly.

- o It is now possible to specify an output filename for batch tasks that target a Windows printer. This is most useful

when targeting Windows printers such as 'Print to PDF'.

If the filename field is left blank, output is sent directly to the printer device.

- o BUGFIX: Changes to the settings of the 'Drilling Rules' in the User Via Hole Definitions editor did not reflect through

- to the NC drill output tasks until the design had been saved, closed and reloaded.

- o BUGFIX: When the plotting scale "Fill Sheet" option was used, the resultant printout was being clipped on the right and top edges.

- o A new "Miscellaneous Line Widths" page has been added to the Configuration->SizesTable editor. On this page it is possible to specify a width for the board profile and keepout region boundary lines when those features are selected for output in an output task. An extra setting is also provided that sets the default output width for all other features that do not have a definable line width.

- 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 currently influence the display within the artwork and board profile editor.

- o The "Width for No-width features" setting has been removed from the Gerber output setup dialog as its function has been superseded by the new width setting values described above.

- o The Artwork plot task setup dialog now includes an option to control how drill holes should be included in the resultant output. The available options are "Off", "Include at true size" (which shows the drill holes at the correct drilling diameter) & "Include at fixed size". In the fixed size mode, all drill holes are displayed at the diameter set in the "Fixed drill diameter" option value.

Artwork editor :

- o BUGFIX: Part auto renumber was not always setting part numbers correctly.

- o BUGFIX: When Amend->Replace Pad had been used to replace a partpin pad on a surface mount component, and the layer selector was set to 'V', then a pad was being incorrectly displayed on the 'V' (Vias) layer.

- o BUGFIX: When Amend->Replace Pad had been used to replace a partpin pad on a board inner layer, the displayed drill hole size was not updating to the defined drill size for the replacement pad. N.B. If 'Replace Pad' is used to replace a pad on a drilled partpin, and the pad replacement is made on the board top or bottom sides, the drill hole diameter will NOT be changed to that defined in the replacement pad. Drill hole dimensions

are always taken from the definition of the pad used on board inner layers.

- o The Mroute toolbar 'Via Autoselect' checkbox default is now 'off', and the state of the control is remembered with the design.

- o The 'Parts->Swap' command has been enabled in the menu.

- o If a User Via definition contained pads on power plane layers, then arbitrary dimensioned pads were being displayed when the power plane layers were enabled for view.

Schematic editor :

- o BUGFIX: Changes made to EMC attribute definitions were not being saved with the design.

Changed in release 2.15 (9th September 2016)

Artwork editor :

- o BUGFIX: The net highlight functions were causing a crash in some situations.

Output Tasks :

- o BUGFIX: If non-electrical text was placed over the top of a symbol having background shading, then the symbol shading obscured the text in the output and screen preview.

- o Added a 'Swell method' option selector to the solder mask task setup dialog.

When set to 'Square corners', square and rectangular pads will be swollen by a simple increase of their width and length and square corners will be maintained.

This is the method that was used in XL Designer versions prior to 2.12.

When 'Filletted (rounded) corners' is selected, swollen pads are created that maintain a constant clearance from the original pad.

For a pad with square corners, this constant clearance results in a radiused corner to the swollen pad.

The 'Swell method' selector has no effect on round/round-ended finger pads and does not affect user-defined library pads which are always swollen using the 'Filletted' method of maintaining constant clearance.

- o BUGFIX: Gerber output was creating duplicate points in AM 'Outline' primitives. Although harmless, this caused a warning to be raised by some Gerber viewers.

- o BUGFIX: Gerber output was not always returning to linear interpolation mode after circular interpolation had been used.

- o BUGFIX: Generated Gerber files contained inconsistent "0D0D0A" line terminations.

- o Added a 'File Created' date/timestamp as a comment at the top of created Gerber files.

- o Added a "RS-274X Tuning" parameter to the Gerber setup page.

With "May use any feature defined in standard" selected, XL Designer will not restrict its utilisation of 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 output devices/viewers.

With "Simplify for marginally compliant devices" selected, XL Designer will avoid known pitfalls in popular output devices/viewers that have partial or weak implementation for various areas of the RS-274X standard.

These are some of the problems we have identified in popular Gerber viewers, and this mode will attempt to work around them :

Within the implementation of the 'AM' (Aperture Macro) command :

- o Device requires an upper-case 'X' as the multiply sign in arithmetic expressions. The standard says it should be lower case.

- o Failure to rotate the x/y position of circle primitive (code 1).

- o Failure to rotate the x/y position of centre line primitive (code 21).

- o Failure to position the centre line primitive (code 21) at the current location, always placed at $x=0, y=0$

- o Failure to handle multiple arithmetic operations in one statement.

- o 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.

Changed in release 2.14 (15th August 2016)

Output Tasks :

- o Gerber output has been reimplemented to conform to the new RS274-X standard. The previous photoplotter output modes with manually editable D-Code tables are no longer available.

All pad styles regardless of shape or orientation will now be instantiated

in the output with 'D03' flash commands.

Parameterised macro definitions are included in the Gerber output to handle a design's various pad shapes.

Rasterisation is no longer used for any pad output.

The default file name extension for generated Gerber files has been changed to '.gbr' to comply with RS274-X.

- o The 'solder mask' and 'solder paste' output types are now correctly displayed on screen with the appropriate pad size expansion/shrinkage.

Pad size swell/shrinkage is now applied to all relevant pads regardless of their shape or orientation.

The solder paste output 'pad shrinkage' setting performs the same function as the previous 'pad swell' setting except

that negative numbers are no longer permitted, and previously configured negative swell settings will automatically have their sign changed to give positive 'shrinkage' values.

The solder mask output now only accepts positive values for the pad swell setting.

- o Output tasks are now displayed in the navigator with icons chosen to indicate if the task relates to artwork, schematic, ncdrill or idf output.

- o The output task viewer now always displays the full output that will be generated. It is not necessary to click a 'refresh' button to preview the output after each input setup parameter has been changed.

- o Support of pen plotters has been removed as a customer survey concluded that it was no longer being used, and its retention was impeding product improvement. If a design is loaded that has pen plotter output tasks defined, the tasks will be reconfigured to direct output to the default Windows printer. A warning will be displayed if this happens.

- o To conserve pen plotter ink, the artwork plot modes previously permitted selection of pad/track output as 'outline only'/'thin line'. All artwork modes will now always output tracks filled and at full width. Pads will always be output filled with the option of including a drill hole where appropriate.

- o For plot configuration dialogs that previously had a 'pen number' box, this has been replaced with a colour selection

box to allow direct selection of the display colour for various features.

When loading old designs, previous pen number values will be replaced with colour selection boxes preset to the colour

that was previously in the plot pen configuration table.

The pen configuration editor has been removed from the plot tool

'Configuration' menu.

- o Circuit schematics will now display and print symbols with their coloured infill. At the present time, the infill will be printed using the colour as configured in the symbol's attributes whereas the symbol outline colour can be controlled in the schematic print task configuration dialog. This anomaly will be addressed soon.

- * o The component outline editor View->Properties dialog now contains two additional checkboxes, "Omit pads from soldermask" and "Omit pads from solderpaste". When selected, pads will not appear on any instance of the outline in solder mask/paste outputs.

These new outline properties can also be edited by right-clicking on the navigator 'outlines' folder and choosing "Edit properties".

- * o The solder paste output now includes pads which are placed on the selected 'extra layer'. Pad shrinkage is applied to the pads from the extra layer.

- * o If layer 'V' is included when a powerplane layer is selected for output, via holes that are connected to the power plane with heat reliefs become isolated. A warning will now be displayed when this conflicting layer selection state exists.

- * o Added Edit->Units menu entry as it was not possible to change the working units from within the outputs task viewer.

- * o Circuit schematic output tasks were becoming disassociated from their selected sheet when instance id numbers changed within the schematic. Schematic output tasks are now linked using hierarchy paths within the schematic. Existing designs that have valid schematic outputs will be updated to use the new hierarchy path linked method.

Please note that schematic sheets will still become unlinked from their output task if the sheet is renamed or moved within the design hierarchy.

Artwork editor :

- * o Added a 'hide' selection to the net highlight colour selection panel. When hide is chosen, selected nets will disappear from view rather than being displayed in a different colour. Clearing all highlights will bring the hidden nets back into view.

- o BUGFIX: The clearance between smooth edged copper fill and the board profile/copper fill boundary polygons was incorrectly being influenced by the setting of the track sizes table size '0' setting.

- o BUGFIX: The length of the segments that make up a heat relief were being displayed too short. (Gerber and other outputs were outputting the segments at the correct length as calculated from the parameters in the heat relief configuration section of the sizes table)

* o BUGFIX: The autourouter was crashing in route by Part or Net mode.

o Geometric algorithm changes to improve the reliability of the boundary generated for smooth-edged copper fill.

Schematic part editor :

* o Edit->Preferences was greyed out.

General system :

o Because of internal database changes and improvements, designs saved with this release will not load into earlier versions of XL Designer.

Changed in release 2.12 (15th January 2016)

General system :

o The license activation dialog will now remember the value entered into the login name field.

If the 'Save password' check box is checked, the password will also be saved.

Custom pad & artwork editors :

o CRITICAL BUGFIX: When defining a custom pad shape in the pad editor, the pad becomes displayed as a filled shape

as soon as the last vertex is entered close enough to the first vertex to form a closed boundary.

When displaying custom pads in the pad editor and artwork editor, the method used to coerce the pad boundary

start and end points together was wrong and could result in the display of a pad with a shape that was different

to the pad boundary definition held within the design.

(So it was possible for a custom pad to look the correct shape in the artwork editor, but be output with the wrong

shape in the printing, gerber etc. output tools)

Artwork editor :

o BUGFIX: When Amend->Replace Pad was used to replace the top or bottom layer pad on a part pin, the amended

pad was shown on the wrong side of the board when the part was flipped.

o BUGFIX: Library pads that had been inserted from outline free-copper data were sometimes shown at the wrong

angle if their associated part was not at 0/90 degree orientation.

Changed in release 2.11 (22nd December 2015)

General system :

- o BUGFIX: Fixed problem opening Ranger3 design files when the design.idx file name was in uppercase.

Schematic editor :

- o BUGFIX: A crash could occur when a schematic had very long version dependent attribute strings.

Artwork editor :

- o BUGFIX: If layers were switched to invisible after the Mroute->Corner command had been selected, then features on the switched off layers were still being picked up by the edit operation.

Changed in release 2.10 (17th November 2015)

General system :

- o The number of "most recent used" files displayed in the file open list has been increased from 8 to 16.

- o The navigator pane and file save dialog will now preserve upper case characters in design filenames.

- o Revised communication protocol with license activation server to reduce interference from network caches and firewalls.

- o BUGFIX: Layer assignments were being all set to 'Copper' when loading a design that had been last saved with XLDesigner versions 1.72 through 1.113

- o BUGFIX: Arc segments in custom pads were being displayed as straight segments if the start and end X coordinate of the arc segment were the same.

- o BUGFIX: If the main application window was closed while a detached artwork view containing popup balloon windows was open and in a minimised state, then a crash or application hang would occur.

Parts list editor :

- o Added board side selector to the Bill of Materials report generator.

Artwork editor :

- o The Grid->Select Origin command will now snap the grid to pads and track

nodes on any visible layer. (Previously, it only snapped to pads on layer 0)

- o BUGFIX: After using "Tools->Generate Silkscreen" to insert outline free copper data, any square pads defined in an outline's free copper were not being displayed at the correct orientation if the associated part was not at 0 or 90 degree orientation.

- o BUGFIX: The net highlight functions were not working for unroutes.

- o BUGFIX: When the artwork and profile editors were both open at the same time, edits made to the profile or keepouts were not being displayed in the other editor view.

- o BUGFIX: When a part was moved, silk screen and label data associated with the part were not displayed at the new position until the display was zoomed.

- o BUGFIX: When a user defined via hole was moved, it was not shown at its new location until the displayed was zoomed.

- o BUGFIX: When a standard via hole was replaced with a user defined via hole, the new style was not shown until the displayed was zoomed.

- o BUGFIX: If you copied/pasted an outline from the master library into a design such that it replaced an existing outline, and the replaced outline was referenced by a part within the design, a crash could occur.

Schematic editor :

- o The visibility state of the 'fast parts' tray will now be remembered for each design sheet.

- o BUGFIX: If the 'Classic' colour scheme was in use, there was no rubber-banding line shown when inserting/editing non-electrical features.

Changed in release 2.9 (8th October 2015)

General system :

- o Included support for licenses requiring online activation.

Changed in release 2.8 (25th September 2015)

General system :

- o The "Display Options" values that were in the File->System Setup dialog have been moved to a new "Edit->Display Adjustments" dialog.

- o The "Autogrid long axis maxdots" setting that was in the masters grid autopitch configuration dialog has been moved to the new Display Adjustments dialog and is now named "Autogrid pitch change threshold".

- o The Display Adjustments dialog contains two new settings :

- "Minimum Line Rendering Width" - this can be adjusted to achieve the best display of thin lines with the users graphic card/monitor combination.

- "Dotted grid intensity" - adjust this to obtain your desired level of grid dot brightness.

- o All browser window contents were being displayed flipped around the X axis after you clicked on the navigator artwork icon with the artwork in 'Flipped' mode.

- o If the artwork and profile editors were both open at the same time, then redraw errors could occur when edit operations were performed on the board profile or keepouts.

Profile editor :

- o Added red shaded area to the display to indicate negative coordinate space. The board profile should not be drawn into the negative space as this can cause difficulties with artwork placement, routing and output.

- o Custom pads were not being shown when 'View->Part Pins' was selected.

Artwork editor :

- o When "display tracks at width" was selected, some tracks would appear to be missing from the display when the view was zoomed out because they were being displayed too narrow to be visible. Tracks will now never be rendered narrower than the line width controlled by the "Minimum Line Rendering Width" setting.

- o Labels for parts on the board reverse were being hidden by part pads and tracks. They are now rendered together with the topside part labels over the top of all layer features.

- o After using the Region->Delete command on a layer, features were not being rerendered on the layer during edit operations.

- o If you selected Amend->Enter Track/Arc/Circle then attempted to add a feature to a layer that was not visible, the

"layer not visible" warning would continue to be shown even after the layer had been switched on in the view control toolbar.

- o The hilite tracks command did not update its layer search list when layer visibility was changed while the command was active.

- o Added 'Extended Opacity Control' button to the side of the layers opacity control slider. This button displays a dialog that permits more fine grained control of the opacities of the various layers and features in the artwork.

- o Added red shaded area to the display to indicate negative coordinate space (as in the profile editor). This can be switch on/off with the 'View->Negative Space Shaded' command.

Schematic editor :

- o Text will no longer change to a dotted line when the display is zoomed out a long way.

- o Added display of symbol 'Flipped' status to Identify->Instance popup balloon.

Outline editor :

- o When the outline datum position is changed, the grid origin will now be adjusted to ensure that the grid remains in the same position relative to the outline.

Changed in release 2.7 (19th June 2015)

General system :

- o Fixed a display rendering issue that occurred when the Windows system display DPI value was not at the default 100% (96dpi) setting.

- o If a higher system DPI value was selected, then tool panes were not being displayed at their correct width until after a View->Restore Default layout action had been performed.

- o If XLDesigner was started by double-clicking on a .rxl design file, then the Outputs section was missing from the design navigator entry for the design.

- o When performing cross-probing between schematic and artwork (by clicking on the schematic/artwork icons in the popup balloon windows), the window that was brought to the front did not always rerender. This gave the

appearance that the information balloon windows were pointing at the wrong object.

- o Changed the "Autogrid Long Axis Maxdots" default value to 75. N.B, the new default value will only been applied when the software is run on a new user account or an account with no XLDesigner "Configuration Data Directory".

- o In File->System Setup, reduced the Artwork grid dot size setting lower limit to 10 as the minimum size appeared too large on some monitors.

Artwork editor :

- o The artwork view control toolpane has been made wider as insufficient space had been allowed for the scroll bar that appears when the window height is insufficient to display the entire pane.

N.B. You must do "View->Restore Default Layout->Editor Layout" within the artwork editor to cause the toolpane to be displayed at its new size.

Schematic editor :

- o Non-electrical version controlled text was not being displayed correctly.

- o The display was not being updated after symbol attributes were moved.

- o Made the default ruled grid colour darker to improve visibility on some monitors.

- o If a symbol was placed onto an existing wire, then in some cases the wire would vanish and not reappear again until a move operation had been performed on another part.

Changed in release 2.6 (14th June 2015)

General system :

- o CRITICAL BUGFIX: A Direct2D factory initialisation error caused the application to fail to start on some computers.

Changed in release 2.5 (14th June 2015)

General system :

- o CRITICAL BUGFIX: There was a thread timing issue that caused the application to crash on startup on some computers.

- o Fixed case sensitivity problem on the license key entry dialog.

Artwork editor :

- o When 'view part outlines' was unticked on the artwork view control toolbar, part labels were not displayed for parts that were flipped to the back of the board.

Changes between 2.4 and 1.116

General system :

- o All graphical presentation within XL Designer has been recoded to use the Microsoft Direct2D API. This now permits the product to take full speed advantage of the latest generation of graphic cards and processors.

Display rendering now uses multiple processor threads, so when working with artworks containing many complex layers, the workload will be distributed across all the available CPU cores.

Most sections of the program now use 24 bit values to represent colours, so there is no longer any limitation on the colours that you can assign to design features.

Direct2D has enabled us to apply transparency to displayed features. As an example, the artwork editor has a new 'opacity' control on the view control toolbar. This allows the user to adjust the opacity of the topmost displayed layers and gain a clearer view of those lower down.

The anti-aliasing feature of Direct2D means that the edges of displayed features are now much smoother.

Anti-aliased display is on by default, but please note that it does cause some speed penalty. If you

want more performance when working with a very high density design on a computer of limited performance,

anti-aliasing can be disabled in the File->System Setup dialog. Changes to the setting only take place

after a view is closed and reopened. A display rendering speed increase of 40% is typical when anti-aliasing is disabled.

- o If a change was made to the master configuration or master libraries, the '*' next to the MASTERS name in the browser was not being cleared on save when a design was also open in a modified state.

- o Added a 'Colours' subsection to the navigator 'Configuration' section. The MASTERS colour configuration dialog permits the configuration of a set of artwork layer and highlight colours

that are applied to all newly created designs. The colours dialog in a design's configuration section allows the user to configure layer/highlight colours within a design, replace those colours with those from the master set, or replace the master colours with those from a design.

Outline editor :

- o Keepout lines, circles and free copper items did not get repositioned if the outline's datum position was moved.
- o When an attempt was made to drop a pad onto a location where a pad already existed, the resultant pop warning message became stuck on screen until a different command was selected.
- o Changed the pad insertion layer selector from 'radio' boxes to checkboxes, so it is now possible to select any combination of layers. Previously, selection was limited to a single layer or all layers.
- o Added statistics line to the top of the window. This displays the number of unique pin assignments on the outline together with counts of the number of pads and number of unassigned pads on each pinstack layer.
- o The region commands have been simplified down to three basic menu actions : Move/Rotate, Copy & Delete.
For all region functions, the first two left-mouse button clicks will define the area to be operated on.
For move/copy, a third mouse click will execute the action at the chosen position. While positioning the target frame for move/copy, the 'Rotate' assigned keyboard key is enabled. Each press of the rotate key will rotate the target rectangle an extra 90 degrees.
For region delete, a third mouse click confirms the delete action or right click cancels the action.
- o The region commands will display a warning message in the status bar if no features are enabled in the region section of the tool configuration pane.

Artwork editor :

- o In the powerplane setup dialog, the selection of the display colour for subnet polygons was not working correctly.
- o When adding or moving user-defined via holes, no animated pad was shown attached to the cursor.
- o For overall layer rendering efficiency, URL text markers are now always displayed in the colour of the layer

on which they are inserted. The alternate display colour option has been removed from the text URL entry/edit dialog.

- o The region commands have been reimplemented in a manner consistent with the changes to the outline editor region commands described above.

- o Reduced the number of menu entries in the 'Partfix' menu section. Single parts will now toggle their fix/unfixed state on each click. Likewise, each selection of the fix/unfix 'all parts' action will toggle the fix state of all parts on the enable sides.

- o Added layer opacity control slider to the view control toolbar.

- o Added unroutes opacity control slider to the view control toolbar. At the left of the sliders travel, unroutes are displayed behind the copper layers, and at the right of its travel, they are displayed 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.

- o All the colours in the highlight toolbar are now user configurable.

- o If you right-click a net in the navigator 'Nets' section, you can now choose to 'hide' the net's unroutes. The net will be marked in the navigator with '(h)' to indicate it is hidden. Nets that are selected for exclusive viewing using the 'Net' option in the artwork view control toolbar now are indicated with a '(s)' marker in the navigator (was '(u)' formerly).

- o Configuration settings have been added to the File->System Setup dialog to enable the user to control the size of text displayed for part labels and pin numbers and also the diameter of grid dots. The controls take a value in the range 20-100 with larger numbers increasing the size of the corresponding feature.

Artwork checker :

- o A numeric overflow was causing incorrect reporting of shorts and clearance errors after smooth-edged copper fill had been used.

- o When a check had been performed on a board with multiple clearance values, the gap error flags were not being displayed with their clearance group information until the design was saved and reloaded.

Schematic editor :

- o When assigning pin numbers to split parts, the pin number dialog was sometimes displaying arbitrary values in the pin number and pin increment fields.

- o By default, the ruled grid is now displayed with much finer lines than were possible with the V1 series of XL Designer.

If the ruled schematic grid is too faint, or not visible, the width of the grid lines can be controlled from within

the File->System Setup dialog. The "Schematic ruled grid line width control" value can be set in the range 10-100

where larger numbers result in thicker grid lines. The default setting is '30'. Changes to this setting only become

visible in a view after the view is closed and reopened.

- o Connection crosses now disappear from symbol pins when the pins are connected. Please note that the crosses will not

currently be removed from symbol pins if two symbols are placed such that their pins overlap (i.e. no wire segment

drawn between them). A connection will however be formed in the netlist.

- o Symbols may now be displayed with a solid infill colour. To show a symbol filled, open the associated part or

design symbol definition, select Attributes->Symbol and check the 'Show Symbol Filled' box. Choose the colour for

the infill using the 'Infill Colour' box. If an outline does not appear filled when this mode is activated, please

edit the symbol outline shape to ensure that it forms a closed path.

Please note that colour infill will not currently be shown in a schematic output task. This will be implemented in

the next release.

Custom pad editor :

- o Undo/Redo was not working for any of the drill hole commands.

- o Pad will now be shown infilled when the outline forms a closed shape.

Output tasks :

- o Artwork plot previews now show pads as filled shapes.

- o If artwork pads filled/drilled output mode was selected and the artwork contained a custom pad where no drill hole was

defined, then a crash could occur.

===== XL Designer Version 1 release notes
=====

Changed in release 116 (1st September 2014)

Circuit schematic editor :

- o A crash was occurring when loading or saving a design where more than 254 pins had been assigned on a schematic symbol.

Changed in release 115 (25th July 2014)

Artwork checker :

- o Circular keepout regions defined in the profile and outlines were not being included in the checking.

Drill sheet output :

- o The drill symbol key (normally a letter) on the left of each drill summary line was being drawn at the same size as the symbol was drawn on the board display. For better appearance, it is now be drawn at the same size as the drill summary text.

- o Metric drill sizes had 'MM.' (megametres) in the column header. This has been changed to 'mm.'

Artwork editor :

- o When the artwork is displayed in 'flipped' mode, parts that are on the bottom of the board will now have their outline displayed in solid cyan colour and parts on the top will have a dotted yellow outline.

- o 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 now becomes reversed making it easier to position parts when working on the back of the board.

- o When using 'Text->Get Label' 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' function, the label will appear the right way around).

Changed in release 114 (19th March 2014)

General system :

- o Added "Title" field to the 'Configuration->Layer Assignments' dialog. This extra field may be used to store arbitrary text to help describe the usage of a layer.
- o The 1.112 release inadvertently removed the 'thin line printwidth' and 'reverse printed arc' settings in the system configuration dialog. These have been reinstated.
- o Design was not being marked as modified after free copper setup had been performed in the outline editor.
- o Design was not being marked as modified after using the navigator options to delete all parts/nets.
- o Design was not being marked as modified after a text parts/wiring list had been imported.

Artwork editor :

- o All boxes which display the colour associated with an artwork layer now have a 'tooltip' associated with them which will be displayed when you hover over the box. The displayed tooltip will include the content of the new layer "Title" field which can be defined in the Layer Assignments dialog.
- o Layer colour boxes which have an associated edit/spin control to change layer selection are now sensitive to clicking. Clicking on the layer colour box will now bring up a selection list where the required layer can be quickly selected.
- o Extra space has been added underneath the layer selection controls in the AddPad, AddTrack, Text and CopperFill toolbars. When a layer is selected that has the extra "Title" field defined in the layer assignments dialog, the title text will be displayed below the layer selection control.
- o The Grid->Origin command now also includes component datums in the locations where the origin may be snapped to.
- o The Parts->Align X/Y command will now snap the reference line to part datums. If you do not click near a part, the reference line will be snapped onto grid.
- o When a region move or rotate operation was performed, copper fill boundary definitions were not being moved. A new "Fill Bounds" checkbox has been added to the region toolbar permitting fill boundary definitions to be moved/rotated with other artwork features.
- o When silk screen text part id labels are being moved, a line will now be drawn from the cursor location to the

datum point of the component that the text label is associated with.

Artwork and Solder Mask output setup pages :

- o An extra 'Description' column has been added to the layer setup pages for these two output modes. This column displays the usage, point count and optional title for each layer.

Circuit schematic editor :

- o If you added a hidden signal name attribute to a connection, then other signal name attributes that were attached to other nodes on the connection were not having their names synchronised correctly to the newly added value.

As the netlist extractor only ever utilises the first encountered signal name value on a connection segment, this could have resulted in unexpected interconnections within the extracted netlist.

- o The summary count of the number of errors and warnings found during compilation did not always match the number of actual errors/warnings recorded in the compilation logfile.

- o It is possible to hide signal name attributes attached to connections. If no other visible name attribute is attached elsewhere on the connection, then unexpected net grouping will occur in the extracted netlist.

To reduce this risk, the schematic compiler will now log a warning message for all hidden signal name attributes encountered during netlist extraction.

Changed in release 113 (7th October 2013)

Artwork editor :

- o A crash was occurring if you tried to open the artwork editor in 'detached' mode.

Changed in release 112 (7th October 2013)

Artwork editor :

- o Added 'View Artwork Flipped' entry to artwork View menu. Also added 'Flip' checkbox to the view toolbar 'Misc' section. When selected, the artwork is displayed as if looking at the back of the board.

- o If custom pads were used on free copper layers within an outline, then the pads would sometimes not be inserted correctly into the artwork by the Tools->Generate SilkScreen

utility.

Circuit schematic editor :

- o Added 'Symbol->Add to Artwork Tray' function. 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.

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.

- o Added 'Region->Add to Artwork Tray' function. All parts associated with symbols in the selected region will be added to the artwork placement tray.

- o The Symbol->Copy operation could cause a crash if more than one copy of a part was inserted.

- o Fixed problem where non-electrical text was sometimes being displayed dotted.

Changed in release 111 (22nd May 2013)

(.exe only issued for emergency fix to power names table case sensitivity problem)

Artwork editor :

- o Smooth edged copper fill was not handling user defined via holes correctly.

Changed in release 110 (20th May 2013)

General system :

- o Because of new features recently introduced, designs saved with this release will not load into versions of XL Designer earlier than 1.109.

- o Added IDF 3.0 file exporter.

If is now possible to export an IDF version 3 file for use with MCAD modelling tools.

Please note that this is a chargeable add on feature and not included with normal system licenses.

Please contact Seetrax sales office for pricing and availability information.

Note that licenses that are supplied with the IDF export feature enabled will not work with versions of XL Designer earlier than 1.110. Non IDF-enabled licenses will continue to work as normal.

To access IDF file export, right click on Outputs->Tasks in the navigator and create a "New IDF Output" task.

===== IDF export usage notes
=====

IDF configuration options

Part library outline creation mode :

"Generate from outline placement bounds box" mode will create a simple rectangular block to represent each part in 3D. The dimensions of the box are taken from the placement footprint definition in the outline library entry associated with the part.

"Generate from outline silk screen data" mode will attempt to create the footprint shape used for the 3d figure by processing the data in the outline silk screen. If so suitable polygon can be created from the outline silk screen, then the placement bounds is used as for the above option.

"Use 3D solid mode setting on outlines" mode will generate the part outline according to the 3D solid mode setting stored within each outline.

IDF package name / part numbering method :

"Outline name / (blank)" 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.

To define the height of components, right click on 'Component Outlines' in the navigator and select 'Edit Properties'. (Please note that the "3d Enabled", and "Silk Screen Mode" outline properties are used by the Autocad DXF 3d exporter and not utilised by the IDF exporter). If a height is not specified (left at zero) for an outline, then the default component height as set in the IDF exporter configuration dialog will be used.

"Outline name / component value" 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.

If the "Use parts list MCAD model name for package" check box is ticked, then any model name specified in the "MCAD Model" field in the parts list editor 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.

When run, the exporter 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, 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.

When opening the .emn file in Pro Engineer, it should be opened as an 'Assembly'. You will then be prompted to also open 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.

===== (end of IDF notes)

- o Changed outline properties edit dialog to permit editing of component height and 3d generation mode attributes without switching on DXF 3d output mode.
- o Rearranged outline global properties edit dialog to separate the component height attribute and 3d generation mode configuration values from the DXF 3d output settings.
- o Drag/drop may now be used in the navigator pane to copy output tasks between designs.
- o A crash could occur if you attempted a drag operation on a non-library related member in the navigator.
(e.g. trying to drag a part or a net)

Parts list editor :

- o Added "View->Show MCAD Modelname" option. When switched on, values entered in this column can be used as the 3D model/geometry library name when exporting designs to MCAD systems.

Changed in release 109 (2nd May 2013)

- o Preliminary implementation of IDF export facility. Not made available as a general release.

Changed in release 108 (9th April 2013)

General system :

- o The navigator in releases prior to 1.107 permitted the creation of library/design member objects which had leading spaces in their names. Other work carried out in the 1.107 release prevented the correct handling of these illegally named entries. These illegal leading spaces will now be automatically removed at design load time.

Artwork editor :

- o Unroutes were not being drawn correctly when a part was being moved.

Changed in release 107 (4th April 2013)

General system :

- o If you right-click on any schematic library folder in the navigator pane, you now have the option to create a new library folder at that level. This new feature has required changes to the file layout of the master schematic library. On program startup, if no restructured master library is found, a newly structured master library will be created from the contents of your old format library. The contents of your original master schematic library will not be altered in any way, and remains in place should you ever need to revert to using an earlier product version.
- o Added 'Rename' and 'Delete' options to the right-click menu associated with schematic

library folders.

- o Drag-drop functionality is now available for moving schematic library parts between the various folders in the master library and between designs. Please note that parts that have dependencies on other library entries cannot be moved or copied by drag/drop. When dragging parts between the various 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.
- o Added drag-drop functionality to copy outlines between designs and the master outline library.
- o Removed the inoperative 'Schematic library manager' option from the right-click menu associated with the master schematic library folder.
- o Removed the redundant library container folder that was located under the top level master schematic library folder. Clicking on the top level schematic library folder now shows the schematic library folder path in the properties pane.
- o Clicking on schematic and outline library folders now displays the number of library members in the property pane.
- o If you performed a cross-probing action to locate a net in the artwork view, and the artwork view was not currently open, then a crash could occur.
- o Popup message boxes could sometimes go behind the XLDesigner main application window giving the impressions that the application had locked up.
- o Popup 'balloon' windows could sometimes be shown over the top of the wrong window.
- o If you repeatedly clicked on the cross-probing buttons in a popup balloon window, then multiple overlapping ident balloons could be created in the target view.

Artwork editor :

- o The Identify->Pin popup balloon window now displays buttons which will locate the pin in the schematic and netlist editor views. On initial display and subsequent clicks on the balloon window, the properties of any associated net will be displayed in the properties pane.
- o The Identify->Track popup balloon window now displays buttons which will locate the associated net in the schematic and netlist editor views. Also, clicking on the balloon window will cause the net properties to be displayed in the properties pane.
- o The highlight colour palette was incorrectly displaying the user-configured layer colours instead of a fixed system default set of colours.
- o When you minimised an artwork view that had been open 'detached', popup balloon windows were not always hidden.
- o When you maximised an artwork view that was already displaying popup balloon windows, the balloon windows were not redrawn in the correct location.
- o When panning and zooming the artwork view, highlighted and fixed nets did not display correctly if 'View->Net Clearances' was switched on.

Circuit schematic editor :

- o Added "Identify->Part Pin" command. Clicking on a symbol pin will display the associated part pin identify for the symbol if the part is allocated. If the part pin can be located in the net list, then the popup balloon window will contain buttons that will locate the part pin in the artwork and netlist editor views.
- o Added "Identify->Connection" command. If you click on a connection segment that connects to an allocated symbol that can be located in the netlist, then a popup balloon window will display the net name. Clicking on the balloon window will cause the net properties to be displayed in the properties pane. The balloon window also contains buttons which will locate the associated net in the artwork and netlist editor views.
- o If a terminal pin was added in the symbol view of a schematic sheet, and the design was saved while still in the symbol view editor, then the resultant design

file was

saved with invalid terminal data and could not be reloaded.

In addition to correcting the source of this corruption, a fix has been incorporated

which will allow loading of corrupted designs. A warning will be displayed showing

the name of affected design symbols and the number of invalid terminal pins.

The invalid terminal pins will be removed automatically, and will need to be redefined

by the user.

o When a split part was copy/pasted into a design, the various part elements did not appear in

the split part's folder until the design had been saved and reopened.

o The "Tools->EMC Attribute Tags" parts mode did not correctly locate split part symbols.

o If a power symbol was added to the definition of a part, then the power symbols could not

be placed on the schematic until the design had been saved, closed and reopened.

o If Region->Copy was used to copy and paste a section of schematic containing split part elements,

then the pasted split parts could not be allocated until the design had been saved, closed and reopened.

Outline editor :

o Added "Copper->Describe Pad" command for identifying the dimensions of free copper pads on

the currently selected free copper layer.

DXF profile import :

o Circles were sometimes imported from the DXF file with the wrong diameter.

o If the last segment of a closed polygon had curvature, then it was imported with the

wrong radius.

Gerber Output :

o When the 'Output used DCodes as' setting is 'RS274X (Extended Gerber)', then the image

polarity 'IPPOS' and 'IPNEG' directives will be output in accordance with

the output

mode setting.

Changed in release 106 (1st October 2012)

Circuit schematic editor :

- o Corrected serious problem with automatic conversion of old flat circuit schematics to hierarchical editor format. Flat schematics with > 1 page would sometimes have connectivity errors.

Changed in release 105 (11th July 2012)

Outline editor :

- o Added 'View->Show Outline Filled' command. This permits outline display to be toggled between normal and 'pads unfilled / tracks not at width'.
- o The outline editor now starts with pin number display switched on by default, and pads displayed unfilled / tracks not at width.
- o The 'Outline->Change Width' function now operates on outline text as well as lines.
- o Added 'Copper->Add Keepout Circle' function.
- o Added 'Copper->Adjust Arc / Circle' function. This permits straight line sections to be converted to arcs and also the moving and resizing of circle features.
- o In addition to selecting standard pad angles of 0/90/180/270 degrees, it is now possible to type other angle values into the pad angle selector box. Angle values will be represented to a resolution of 0.01 degrees. Please note that it is not currently possible to enter square, rounded ended finger or square ended finger pads into the free copper section of an outline at angles other than 0/90/180/270 degrees. This is an internal database restriction that will be lifted in a future release. Custom pads may be entered into free copper at any angle. Pads on pin padstacks are not affected by this limitation.

Artwork editor :

- o Smooth edged copper fill was sometimes omitting the edging line around a circular keepout area.

o When pulling out via-stubs from SMD pins with 'via autoselect' active, it was possible to drop via holes on top of holes belonging to other nets. Also, if a stub was pulled out and dropped on top of a compatible user via from the same net, then a duplicate user via hole was inserted.

Changed in release 104 (5th May 2012)

General system :

o If a user's MyDocuments folder was located on a Windows server, and that user attempted to use a host computer that had been detached from the network, then a message saying 'Cannot locate MYDOCUMENTS' could be displayed and you would then be locked out of the software.

This message has now been changed to a warning, and the software will proceed to run as normal. However, the configuration paths in 'File->System Setup' should be verified to ensure that they point to locations that do exist on the network detached computer.

Changed in release 103 (2nd April 2012)

Netlist editor :

o The netlist can now be sorted by clicking on the column headers.

Power name table editor :

o The power names can now be sorted by clicking on the column headers.

Output tasks :

o When an output task is configured to target a Windows printer, the printer selection and any selected printer options will now be 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 raised when the output task is opened, and printer selection will revert to the Windows default printer and settings.

It is possible to return 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' to restore the default Windows printer on all tasks.

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.

General system :

- o It is now possible to restore a design's part prefixes, pad/track sizes and power plane feature dimensions to master default values. To access this function, right click on the design's 'Configuration' folder in the navigator tree and select the 'Restore Design Defaults' option.

- o When right clicking on a design's 'Documentation' folder, the 'New' submenu was blank.

It now correctly shows blank document templates as configured in the <INSTALL_DIR>\UserTemplate\DocTemplates\index.txt file.

- o Because of internal database changes and improvements, designs saved with this release will not load into earlier versions of XL Designer.

Changed in release 102 (14th February 2012)

General system :

- o The pad and track sizes editor has been re-coded to permit sorting of sizes by clicking on the column headers. The editor now also uses "in place" editing.

- o The power names editor has been re-coded to use in place editing.

Circuit schematic editor :

- o If a symbol was placed on a schematic sheet, and the source library of the symbol was another design, then a crash could occur after the other design was closed.

- o When schematic parts are copied between designs (either by copying/pasting directly using the navigator pane, or by using symbol->place and using another design's schematic as the symbol source) the outlines required for the copied schematic

parts will now be imported into the target designs outline library.

(When symbols are placed from the schematic master libraries, the previous behaviour of not importing required outlines until parts/netlist extraction still applies).

- o Added a slider control to the parts edit view to enable rapidly moving between part equivalent symbols.

Changed in release 101 (10th January 2012)

Artwork editor :

- o If any layer is selected as visible in the view control toolbar, then clicking the 'all layers' visibility toggle button will cause all layers to be switched off.

(Previously, they were all being switched on)

- o If the artwork top layer was visible, 'Identify->Pad' and 'Identify->Pin' would not work on visible bottom layer pads.

Parts and netlist editors :

- o When saving parts and netlists to a file, the File->Save Parts/Nets dialog now allows selection of saving the file with spaces or tab separated columns. The tab separated output makes it easy to import into spreadsheets and other programs.

When saving the net list with tab separation, the output is restructured slightly :

The signal name column for unnamed nets will be blank. (For space separated output, a full stop is shown)

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.

Foreign system import :

- o The Calay parts/net list importer that shipped with the previous Ranger XL product has now been included in XL Designer.

- o The text parts/net list importer could crash when re-importing the text file if any part description or outline values had been changed.

General system :

- o A new entry 'Select Grid Origin' has been added to the 'File->System Setup->Special Function Keys' dialog. The initial key definition for this item is blank, but the user may define a key that can be used to activate the Grid->Select Origin action in any of the editors.

o Updated version of reference guide included.

Changed in release 99 (28th November 2011)

General system :

- o If the "Save Designs with File Compression" checkbox was unchecked in the File->System Setup dialog, then designs were still being saved with data compression.

Circuit schematic editor :

- o When a schematic was being compiled to parts/net list, artwork connectivity tracing and unroute generation was inadvertently being performed for each processed net. When an artwork had a large number of tracks/nodes, this process can take a long time and the repeated artwork scanning resulted in the schematic compiler appearing to hang. The unroute update is now performed only once at the end of parts/nets extraction.

DXF profile import :

- o When a profile was imported from a .dxf file, the design modified flag was not set.

Artwork editor :

- o At the beginning of split power plane and copper fill operations, the board profile must be processed into a continuous unbroken outline. When the profile had been built from multiple line/arc sections where the endpoints did not always exactly meet, then this profile analysis was failing because it had zero tolerance for endpoint mismatch.

The problem was particularly noticeable with profiles imported from .dxf files.

The profile analyser now permits slight coordinate mismatch in line/arc coordinates.

The maximum permitted coordinate misalignment is now 0.005".

Profile editor :

- o Added 'Profile->Transfer to Current Layer' function.

When this mode is active, selected line/arc/circle features will be moved from their

current layer onto the layer selected in the 'Mode' selector.

This function can be useful to separate profile/keepout data has been imported from a

.dxf file where all the data is on a single layer.

Outline editor :

- o When the grid origin had been snapped to an outline pad, it became impossible to escape

from the Outline->Move point function when moving line endpoints.

Changed in release 98 (18th October 2011)

Artwork editor smooth edged copper fill:

- o If keepout line/arc features did not form a closed polygon, they were being ignored.

- o If a keepout area was drawn with parallel edges that were closer together than twice

the profile clearance value, then the keepout area was being ignored.

- o Holes were appearing in copper filled areas and in power planes at the location of

user defined via holes even when the user via hole style did not define any pad or

drilling on the layer.

- o Copper fill did not take account of clearance values applied to individual nets.

- o Added a 'Via Holes' configuration option to the copper fill toolbar. 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

'Heat Relief' - heat relief stubs are inserted to connect the via to the fill region.

'Cover' - the copper fill floods over the top of the via hole.

o Added 'Component Pins' configuration option to the copper fill toolbar. 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 pins are connected with heat relief stubs. SMD part pins (undrilled pins on layers T/B) are covered over by the copper fill.

'Cover All' - copper fill covers all part pins.

o Copper fill toolbar was not remembering the Fill Layer and Fill Size selections.

Changed in release 97 (14th September 2011)

Artwork editor :

o Re-coded the smooth edged copper fill function because of severe accuracy and performance problems with the previous implementation.

o Added a new control to the copper fill toolbar to control the minimum number of part pins that must be connectable to a copper fill region for the region to appear in the fill result.

NB: setting this value to zero will result in the creation of isolated copper areas.

Artwork checker :

o When a clearance violation occurred to a keepout area, the error flags could sometimes be displayed at an arbitrary position or not displayed at all.

Changed in release 96 (limited distribution release for new copper fill testing only)

Changed in release 95 (24th May 2011)

Artwork editor :

o If a layer swap operation was performed on a track feature that had been entered

with the 'Amend->Enter Tracks' mode, then a crash could occur.

- o If layers 'T' or 'B' are selected as invisible, the Identify->Pin and Identify->Pad functions will ignore pin/pad features on those layers.

Changed in release 94 (14th March 2011)

General system :

- o When XL Designer is installed on a computer with a 64 bit version of the Windows operating system, the software will now correctly see Sentinel Super Pro security keys (dongle) attached to the USB or parallel printer port.
IMPORTANT: This change requires the Sentinel System Driver to be upgraded to version 7.5.2 or later. The version 7.5.2 driver is available for download from the XL Designer downloads web page.

- o The software installer creates an association of the .rxl file extension with an open action within XL Designer, and stores it in the HKEY_CLASSES_ROOT registry hive under the key 'XLDesigner.Document' (and also removes it on product uninstall)

However, a much older file association registration was built into the software.

This caused an unwanted duplicate association to be created in the registry under the name 'xl_designert'. This association was not removed on product uninstall, and could cause confusion if the product was reinstalled to a different install path.

The 'xl_designert' association will not now be created.

- o In the File->System Setup dialog, the two browse buttons for 'Text I/O Dir' and 'Generic Extractors' were the wrong way around/crossed over.
- o When the main application window was moved or resized, information 'balloon' windows did not reposition to the correct screen location.
- o When an artwork or circuit schematic view was minimised, a cross probing locate action did not cause the editor window to come out of its minimised state.
- o If a design containing schematic split-parts was opened, then closed, and

another design

then opened, subsequent schematic/artwork cross probing actions could result in a crash.

Circuit schematic :

- o A crash could occur when using any of the non-electrical edit commands when undo-redo was disabled.

- o Fixed crash when compiling parts/net list after a component outline name change.

(Customer report ref: DV CL599_A_temp1)

Artwork editor :

- o On low resolution displays, it was not possible to see all of the view control bar.

A vertical scrollbar will now appear on tool bars when all of the bar cannot be seen.

NB: The artwork view control toolbar has been made slightly wider to allow space for the

scrollbar when displayed. To ensure that the toolbar is displayed at its new width,

it will be necessary to perform a 'View->Restore Default Layout->Editor Layout' action

within the artwork editor.

- o When the boundary lines of a split powerplane polygon formed a closed shape, the size code of the boundary line was being set to zero. (Bug since release 1.91)

- o If a new user defined via hole style was created while the artwork editor amend/enter

pads toolbar was displayed, the toolbar did not update to include the new style in the pulldown list of available user via styles.

- o A crash could occur if a new user defined via hole style was created while the artwork

editor was open.

NC Drill output :

- o If a new user defined via hole style was created, and pads of that style added to the artwork,

and then an NC drill output generated without first saving, closing and re-opening the design,

the newly added via holes did not appear in the NC drill output.

Changed in release 93 (15th February 2011)

General system :

- o The system was not handling UNC filepaths (server shares) correctly during the startup validation needed before seeding user directories from the UserTemplate files.

This caused the system to abort with an error dialog referring to _mkdir failure.

- o There was a possibility of complete lockout from the software requiring manual editing of the registry in situations where an incorrect configuration path existed.

Any configuration path that now fails validation will cause the software to drop you into the System Setup dialog where any errors can be corrected.

Changed in release 92 (14th February 2011)

Artwork editor :

- o A crash was occurring when performing a track ripup operation.

Artwork checker :

- o The checker was not honouring the 'Include Keepouts' flag correctly, so whether keepouts were checked or not was indeterminate.

Changed in release 91 (14th February 2011)

Product installer :

- o GDI+ library was not being installed on systems where it was absent (Windows 2000).

- o The software was packaged with an incorrect set of WinSXS assemblies resulting in an oversized installation image file.

- o The installer will now correctly register the .rxl file extension and create a default open action, so double clicking on a .rxl design file will start XL Designer and open the design. (There is however still an issue where clicking on additional designs will result in an attempt to launch another copy of XL Designer rather than open the extra designs in the same program instance. This will

be corrected
in a future release)

o There has been a rearrangement of how the installer lays out the product files within the installation target directory.

Before performing an upgrade to this product version, existing users are advised to make a backup of their configuration data, PCB and Circuit schematic master library directories. Although we have taken great care to ensure that this upgrade occurs without incident, a backup is good insurance should something unexpected occur.

Previously, at install time, the user was given the choice of where to place the 'Configuration Data' directory, 'PCB Masters' directory, and 'Hierarchical Circuit' library directory. The default was to place them in the following locations :

```
<TARGETDIR>/Data  
<TARGETDIR>/Masters/Pcb  
<TARGETDIR>/Masters/HierCir
```

(where <TARGETDIR> is the product installation root directory, normally 'c:\ProgramFiles\Seetrax\XL Designer').

This install time choice of library and configuration directory has been removed from the installer, and the supplied default libraries and configuration will be installed into the following structure :

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

Also, the 'DocTemplates' directory has been moved to the new <TARGETDIR>/UserTemplate directory.

The configuration files and libraries in the new UserTemplate structure never get written to by XL Designer. At startup of XL Designer, a personal configuration and libraries structure will be created in the user's 'MyDocuments' directory, and this will be seeded with files from the UserTemplate structure. The new personal files structure is layed out as follows :

```
MyDocuments/Seetrax/XL Designer/Artmacros
```

MyDocuments/Seetrax/XL Designer/Data
MyDocuments/Seetrax/XL Designer/Libraries/PCB
MyDocuments/Seetrax/XL Designer/Libraries/Schematic

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 'Files->System Setup' dialog may be used to choose different locations for the various directories within the user's personal data and libraries.

This file layout change overcomes the permissions problems that were associated when running the application without administrative privileges on Microsoft Vista or Windows 7 systems. When computers are shared between multiple users, it also allows individuals to edit/pollute their personal configurations and libraries without affecting other users of the system. 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.

As in previous releases, it is possible to configure the system to continue using master libraries/configurations shared amongst multiple users. Just access the 'File->System Setup' dialog and ensure that the Configuration Data directory, PCB Masters and Schematic Masters directory paths point to your shared content. As long as these directories are not empty, files from the UserTemplate area will never be copied over the top of your shared libraries and configuration. Please note that when running on Vista/Windows 7, you will still need to take administrative action to ensure that all users have appropriate access permissions to the shared content.

Apart from making the recommended backups, no special action needs to be taken before upgrading to this version of XL Designer. The new UserTemplate structure will be created and populated with the supplied libraries and configuration. Previously configured paths to user's configuration and library directories will be honoured, and never overwritten.

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.

Product license :

- o Prior to this release, the product license key was stored in the system registry in the <HKEY_LOCAL_USER> hive. In a shared user environment, each individual user had to install the product license key.

The license key is now stored in the <HKEY_LOCAL_MACHINE> hive, so the key now needs to be installed just once, and will be available to all system users.

On startup, if a key is found in the old location, but not in the new location, the key will be copied into the new registry location, so no special action need be taken during upgrade.

Profile editor :

- o When keepout areas are entered which form a properly closed polygon, then the area is displayed cross-hatched.

Artwork editor :

- o NEW FEATURE: A new 'View Control' dialog bar has been added to the artwork editor. This brings together all the visibility controls that were previously available from the artwork 'View->Set Colours' and 'View->Setup Visibility' dialogs (which have been removed). All buttons on the view bar have 'tooltips' enabled, so hover over any button to get a description of the button's action. The layer on/off buttons have layer usage and point count information in their tooltips.

The unroutes 'Net' button allows manual selection of which unroutes to display. When this button is checked, right click on a net in the navigator, and choose 'Show net unroutes'.

Multiple nets may be selected and switched on/off.

- o NEW FEATURE: When the artwork icon is selected in the navigator, the 'Properties' pane will

now display a layer type and point count summary for all the artwork layers.

- o After routing a board with the artwork built in autorouter, the 'design modified' flag was not set.

- o The 'Amend->Corner' mode was incorrectly allowing segments of copper fill to be selected for new node insertion. This resulted in artwork corruption. Corner insertion is not currently supported on copper fill segments. Only point move and segment delete is available.

- o When entering a boundary line for a split powerplane polygon, a crash could occur if the line was terminated without the polygon being closed, and a new separate line segment then entered to complete the polygon.

- o When a split powerplane polygon line was entered without forming a closed polygon, and another edit action was selected, the 'Add Line' command became greyed out so no further line segments could be entered.

- o Keepout areas are now displayed cross-hatched.

Artwork checker :

- o NEW FEATURE: The artwork checker now tests for violation of keepout areas. These are flagged on the artwork with a letter 'K' marker. This extra check can be enabled/disabled on the art checker run dialog.

Electra/Specctra optional autorouter interface :

- o Keepout areas defined within component outlines were not being included in the exported router design file.

Changed in release 90 (17th January 2011)

General system :

- o Because of the new features introduced with this release, designs saved with this release will not load into earlier versions of XL Designer.

Outline editor :

- o It is now possible to define keepout areas within a part outline definition.

Keepout zones should be defined using the 'Copper->Add Keepout Line' function.

A closed shape should be drawn defining the required keepout area. Keepout areas

may be defined for the board Top, Bottom or 'All' layers. NB: to see these new outline

specific keepouts within the artwork editor, the 'Keepouts' checkbox must be ticked

in the "Artwork View Properties" dialog.

- o A new 'Star Points' checkbox has been added to the Routing Constraints section of the

outline properties dialog. When this checkbox is set for a component outline, the

outline takes on a special mode of behaviour used for star-connecting multiple nets

at a single point. Please read Star Points section below for a detailed overview of

this new facility.

Artwork editor :

- o Gridded copper fill did not honour keepout areas.

Starpoints :

- o It is now possible to use star-connected nets within a design.

A sample design (Starpoints Example.rxl) using this new facility is included with this

product release and can be found in the

<ProductInstallationDir>/SampleDesigns/PCB folder.

The example also demonstrates the use of a keepout area placed within an outline definition.

The first step in using star points is to create a special component outline that when

placed on the artwork defines the track geometry for the location where the nets meet.

To begin creating a star point outline, create a new blank outline as normal, then before

adding any other features to it, select View->Properties (within the outline editor)

and check the 'Outline is a Routing Star Point' checkbox.

Pads should now be added to the star point outline. One pad is required for each net that

will be connected at the star. The pads may be placed on the Top, Bottom, or 'All' layers.

Any pad style may be used.

The pads should then be interconnected by inserting 'free copper' on the top, bottom or 'Any' layer.

Keepout areas should then be added to the outline using the Copper->Add Keepout Line function.

The keepout areas limit the scope of autorouters and the copper fill routines from placing copper through the centre of the star point outline. The areas must not enclose any of the connection points on the outline. Please read the implementation section below for more detail on this.

A schematic part symbol that references the star point outline should be created, and one terminal placed for each pin on the star point outline. The pins should be numbered as for any normal part.

At the schematic level, each net that must meet at the star must be created 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.

Starpoint implementation method :

- o The method chosen to provide star point capability is simple, but hopefully effective.

The free copper features that interconnect the various star outline pads 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 within the star outline. 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 the nets.

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.

Starpoint implementation features not yet complete :

- o Outline based keepout areas do not get included in router files created for the optional Electra/Specetra autorouters.

- o The artwork checker does not yet check for clearance errors or violation of the keepout areas.

- o Both the above two ommisions will be corrected in the next release. (approx 7 days time)

Changed in release 89 (7th January 2011)

Artwork editor :

- o CRITICAL BUGFIX:

Every time that the artwork was repainted, one 'Graphics GDI' object was leaked.

This resulted in redraw performance degrading each time that the artwork display was refreshed, and the process eventually running out of graphics resources causing the program to crash.

General system :

- o Behaviour change for 'partial unroutes'.

If a trace is only partly routed from source to destination, then 'partial' unroutes

occur at the start and end of the trace. These are shown dotted to discriminate them from normal unroutes which are shown solid.

Internally, partial unroutes are handled very differently from normal unroutes, a partial unroute being nothing more than a continuation of the part routed track with a special dotted visibility mode for the start/end segment, and also made invisible to output tools and the artwork checker.

Using 'Amend->Move Point', it is possible for a user to pick up mid points of a partially routed track, and attach them to part pins or via hole locations, thereby forming a connection, the netlist then being patched up later to match the manually contrived

connection. However, the connected trace does not finish at the part pin as it was only a mid point node that was moved. The trace still continues on past the pin and finishes at the end of the partial unrout which has been abandoned.

These abandoned partial unroutes cannot then be easily removed, and unrout regeneration and artwork checking will not remove them.

To assist in cases where such routing action has been performed, the following changes have been made :

When a design is loaded, the artwork is scanned for all tracks having partial unroutes.

If any partial unrout is found where the 'inboard' node (ie. the partial unrout node farthest from the end of the trace) is connected to a part pin or a via hole, then that partial unrout segment will be removed.

When the inboard node of a partial unrout is moved with 'Amend->Move Point' and attached to a part pin or via hole, the partial unrout will be removed.

Changed in release 88 (30th December 2010)

Artwork checker :

o If the type of a layer was switched from power plane type to copper type, and the layer had existing powerplane antipads/heat reliefs, then these unwanted pad features were not included during an artwork check, and undetected short circuits and gap errors could occur.

The artwork checker will now test 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. (For an easy way to clean out these unwanted pads, please read next note)

General system :

o The artwork delete tool (accessible by right-clicking the artwork icon in the navigator) has been updated. A new delete option 'delete features that are invalid for layer type'

has been added to the tool. The new mode will remove antipads and heat reliefs which are found on a 'copper' type layer.

o When the type of a non-empty artwork layer is switched between power plane and copper type, a warning will be displayed advising the need to delete existing layer contents.

o The design modified '*' flag was not being set in the following situations :

During any changes to the schematic when schematic 'undo/redo' was disabled as a temporary performance workaround.

When an artwork power plane was generated.

After making a board profile edit with the numeric editor.

o The design modified '*' flag was incorrectly being set in the following situations :

After opening and closing a plot batch, even if no changes were made.

After highlighting tracks in the artwork editor.

Changed in release 87 (8th December 2010)

Circuit schematic :

o Added attributes import/export facility. Import/export commands may be accessed by right clicking on the top 'Schematic' folder for a design, or the top folder of a master library (eg. schematic folder '74xx')

Exported attributes may be saved into tab separated (.txt) or comma separated (.csv) files as selected in the save dialog.

The first row in the created file will always contain the column names for the data in the following rows. The first column is always the name of a block/part/symbol in the schematic or library. Each additional column represents an attribute setting.

To export a file suitable for editing in Microsoft Excel spreadsheet, it is recommended to export attributes as a tab separated (.txt) file. On loading, Excel

will display

an import Wizard where you may change the type of all imported columns to 'text' type.

If you do not do this, and try to load from a .csv format, Excel always treats strings of digits as numbers, which is a problem if some of your symbols/attributes have all numeric names with leading zeros.

Artwork editor :

- o If the artwork did not contain any placed parts, then pads entered with 'Amend' mode were always drawn unfilled even if filled drawing mode was selected.

Changed in release 86 (26th November 2010)

Design outputs :

- o Bitmap image output was not creating arc features correctly.

Changed in release 85 (24th November 2010)

Design outputs :

- o Added bitmap image output capability. This filter may be found under the new output group category "Image file". Images may be saved in 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.

- o Removed redundant "2D Drawing" category from the list of plot tasks that can be created.

- o Schematic non-electrical "Annotate" mode features were not being included in the calculation of the "Fill Sheet" view.

Artwork editor :

- o Added "All powerplanes" checkbox to the powerplane generator toolbar. When this is ticked, the "Generate" button will create all powerplanes in one operation.

Changed in release 84 (25th October 2010)

Artwork editor :

- o When a component outline containing standard pad shapes was placed in the free copper section, and the part was rotated on the board to angles other than 0/90/180/270 degrees, then the copper insertion action of the mask generator tool was placing pads on the artwork layers at the wrong angle.

Changed in release 83 (28th September 2010)

Circuit schematic editor :

- o Identify->Instance did not work correctly for block I/O symbols.
- o Symbol->Copy could sometimes crash when copying block I/O symbols.

Artwork editor :

- o Powerplane generation could crash on split power planes if no polygon was defined for a secondary signal.
- o Track ripup could crash if you ripped up a track that was not correctly associated with a net.
- o When ripping up tracks a segment at a time (with the LH mouse button), you must rip the tracks segment by segment starting at a component pad. When there were very tiny track segments at the start of a route to force a track onto the routing grid, it was not always possible to see that you were not clicking the first/last segment connecting the pad, and the ripup operation was ignored. During segment by segment ripup, all very short track segments (<0.01" long) that are adjacent to the selected segment will no longer inhibit ripup, and they will be removed together with the selected segment.
- o Mroute->Move Track command was causing a crash.
- o The power plane generator was not honouring the "Use Heat-Relief when connecting to power planes" flag in user via hole definitions.

Changed in release 82 (8th September 2010)

Circuit schematic editor :

- o If a schematic view was opened before any circuit view had been shown in the browser window, then display colours were not configured correctly. This was normally seen as a wrong coloured grid after the schematic had been opened via an artwork 'Find' command.

Artwork editor :

- o Unroutes were not always correctly removed when user defined via holes were in a route.

Artwork checker :

- o The artwork checker was not always revalidating unroutes on completion.

General system :

- o Reference guide updated.

Changed in release 81 (6th September 2010)

Outline Editor :

- o Pad->KeyMove command could cause a crash when moving a custom pad where the pad's name was longer than 11 characters.

Artwork editor :

- o If a user defined via hole did not have 'Allow in power nets' checked in the via hole definition, the hole style was still being selected automatically when manually routing power nets.
- o The powerplane generator was inserting an antipad on a powerplane layer for user defined via holes even if the hole definition defined the stack as not passing through that layer.
- o When dragging out power unroute stubs for smd power pins, user defined via holes were not being selected when the autoselect option was on.
- o Equalised the width of all toolpanes to reduce unnecessary screen redraws when moving between various edit modes.

Gerber file generation :

- o Gerber output tasks were crashing if a used pad had zero dimensions in the sizes table.

General system :

- o When toggling between inch/metric units in the user defined via hole definition editor, the drill hole size value did not change.

Changed in release 80 (18th August 2010)

General system :

- o Corrected incorrect display of lines that were partly out of the view window.
(Bug only occurred in release 1.79)
- o Corrected a configuration problem where version 1.79 was compiled without code optimisation.

Artwork editor :

- o When layer colours were redefined, the layer colour indicator boxes in the manual routing, pad/track/text editing, region rules and copper fill toolbars did not update to show the new colours.
- o Changed the manual routing tool pane to display layer selection checkboxes for all copper layers. To help with understanding the topology of multilayer boards, layers are presented in the tool pane in the assembly order defined for the board. Powerplane layers are also shown in the appropriate places.
- o Added a 'Via Autoselect' option to the manual routing tool pane. When this is checked and the system needs to insert a via hole while performing a layer swap or point insertion operation, then the via hole will be automatically selected from the user defined via hole set. The system will attempt to locate a via hole that traverses the minimum number of board layers while still providing the correct connectivity.
If 'Via Autoselect' is not checked, or auto mode can find no suitable via, then a normal through-board via is inserted.

Changed in release 79 (4th August 2010)

General system :

- o When running on Windows 7 or Windows Vista with Data Execute Protection (DEP) enabled, a crash would occur when using any action that caused a popup 'balloon' information window to appear.
- o A 'Colour table overflow' message would sometimes occur when browsing items in the navigator window.

Circuit schematic editor :

- o Internal roundoff errors caused connections to sometimes not be correctly associated with symbol pins when metric grids were in use for symbol creation and schematic drawing.
- o Added a 'Disable Schematic Undo/Redo' checkbox to the 'Temporary fixes' area of the 'File->System Setup' dialog. The schematic editor Undo/Redo functionality currently has performance problems when working on large schematics. As a temporary measure to help customers speed up work on large schematics, it is now possible to switch off the undo/redo code.

Changed in release 78 (14th June 2010)

Artwork editor :

- o CRITICAL BUGFIX:
When Amend->ReplacePad was used to change the style of a pad, the net identity of the modified pad was not always correctly retained. The problem showed up most often when swapping standard via holes to a user defined via hole style. Once a pad's identity became lost or corrupted, this caused erratic behaviour in the correct selection of heat-reliefs/anti-pads when creating power planes, and also caused unroutes to be displayed where none were expected.

Performing an artwork check will often correct the invalid net identity on the affected pads. However, no connectivity problems must exist for the checker to be able to clean up incorrect net identities on artwork features.

- o Every time the amend pads mode was activated, custom library pad names and user defined via hole styles would be repeatedly added to the pad style and user via pulldown lists in the amend pads toolbar.

General system :

- o The Via Hole Definitions editor was not remembering drill hole diameters correctly.

Changed in release 77 (8th June 2010)

Artwork editor :

- o The state of the 'show filled pads' selection was not being preserved with the design.

- o When a track was ripped up from an SMD part pad that connected to a power plane via a stub and power via, the unroute 'blob' did not reappear on the part pad.

Circuit schematic editor :

- o Colour configuration was not being saved correctly with the design.

- o Colour configuration changed erratically when moving between designs and views with different colour setup.

Changed in release 76 (17th May 2010)

General system :

- o When a circuit schematic was recompiled while the artwork editor was open, and the recompilation caused a part outline to be replaced, and the replacement outline had a different pre-routed copper arrangement, then a crash could occur.

- o When a circuit schematic was recompiled while the parts list editor was open,

and the recompilation caused extra parts to be inserted, then a crash could occur.

- o If autosave was enabled and an unsaved job was open, the "Design Autosave reminder" dialog was posted to the screen every 10 seconds.

- o When performing a design "Save As", the original name is now displayed in the file save dialog.

- o Reordered some design icons in the navigator window to better reflect typical design flow.

- o Identify commands which displayed popup 'balloon' windows with no close button became stuck on the screen if their associated identify command was cancelled while the balloon was panned offscreen.

- o The navigator was automatically scrolling and expanding folders in order to show currently selected entities when working inside the various editors. This behaviour has now been changed. Selected entities will now be shown in the browser/properties window but the navigator will not scroll and expand folders to show the item. The browser and properties window now contain a "Find" button. Pressing this will cause the navigator to locate the item shown in the browser/property window.

Artwork editor :

- o Added a horizontal scrollbar to the net selector within the powerplane toolbar.

The powerplane toolbar has also been increased in width to accommodate the display of longer powerplane subnet signal names.

- o The part placement tray was sometimes showing parts that were already placed.

- o Large time delays occurred when using undo/redo with a large number of parts in the part placement tray.

Circuit schematic editor :

- o When a block I/O port was duplicated using the Region->Copy and Region->Paste

commands, then the resultant I/O port was forgetting its attribute values.

- o When multiple instances of a sheet occurred in the design hierarchy, the ident (suffix number) of the instance being edited was not displayed in the window title bar (or window tab when in maximised view mode).

Changed in release 75 (30th March 2010)

Artwork checker :

- o When a power net was used simultaneously on more than one power plane layer, the artwork checker was reporting 'interplane shorts' for drill holes that correctly interconnected the layers.
- o The checker will now report parts that are unplaced or are placed outside of the board profile.

Changed in release 74 (1st March 2010)

Artwork editor :

- o When a split powerplane was generated, the subnet boundary polygons were not being tagged with the correct net identity. This caused them not to be recognised by the Specctra/Electra autorouter export function.
(To correct this problem on affected designs, use the powerplane 'Generate' button to recreate each power plane layer)

Changed in release 73 (22nd February 2010)

Artwork editor :

- o A crash occurred on entering the powerplane generation mode if any net contained a reference to a non existent part pin.

Circuit schematic editor :

- o Fixup crash at schematic load time when allocation data was bad.

Changed in release 72 (17th February 2010)

Artwork editor :

- o Reimplementation and simplification of the powerplane generation mode.

It is now possible for a power net to be used simultaneously on more than one power plane layer. However, the artwork checker currently reports such usage as 'interplane shorts'. This problem will be fixed in the next release.

Split power plane net selection is now performed in the Configuration->Layer Assignments and Ordering dialog in the Navigator pane.

Split power plane subnet polygons are now retained with the design, so it is no longer necessary to import them from the powerplane layer to regenerate a powerplane. The import function is retained for compatibility with existing design files.

The user guide included with this release still refers to the previous powerplanes implementation. An updated guide will be included in the next software release.

NB: Designs saved with this version of XLDesigner are not compatible with, and cannot be loaded into earlier releases of the software.

Changed in release 71 (3rd December 2009)

Circuit schematic editor :

- o When old 'flat schematic editor' circuits were converted for use in the hierarchical schematic editor, some connections were not correctly processed. The hierarchical editor has a strict rule that only 'ednpoints' of connection wires are permitted to be attached to symbol terminals and connection 'blobs'. Some converted schematics were not complying with this requirement, and after subsequent editing of the schematic, internal inconsistencies occurred which resulted in a program crash or incorrect netlist compilation results.

This bugfix corrects the incorrect initial conversion, and also fixes up the inconsistencies that have resulted from editing of the affected schematics.

Changed in release 70 (9th November 2009)

Artwork editor :

- o After a region delete operation had been performed, the Mroute->RipUp and Mroute->Corner operations would cause a crash.
- o A crash occurred after using the silk screen generator to add outline free copper to the artwork.
- o Part renumbering and gate/pin swap history will now be logged into the 'renumber' file in the design log files folder.
- o Network changes made with the Tools->Network commands will now be logged into the 'netmods' file in the design log files folder.
- o A crash could occur when you moved or flipped a part having pre-routed copper features defined in the outline.

Outline editor :

- o Added Copper->Key Move Pad command.
- o Added Copper->Change Pad Style/Size command.
- o When alternating between outline mask editing and free copper editing, the free copper functions became confused as to which layer to enter features into.
- o Pad style and size controls have been added to the toolbar. When pad style/size is changed in the toolbar, the navigator and browser windows will update to display the selected pad. Pads may also be selected directly from the navigator (as before) and the outline editor toolbar controls will update accordingly.

Circuit schematic editor :

- o When using the region commands, the region rectangle corners and move/copy destination will now always snap to grid/half grid even if "Snap to Grid" is disabled in the Grid menu.
- o When a flat schematic was converted, the associated parts did not have their pin swap rules imported.
- o When a flat schematic was converted, instantiated part symbols that were members of a part with >1 available instance were not tagged with the correct

sub-instance reference if 'gate' (part element) swapping had been performed within the old flat circuit editor.

- o If two wires were butted end-to-end on a flat schematic with no junction blob or part terminal at their join point, then connectivity was not preserved through the wire in the converted schematic. (Lloyd research 'Nets not being joined by signal name' bug report)

General system :

- o Added "Edit Datasheet Addresses" to the navigator top level schematic folder right-click menu. This permits bulk assignment of datasheet addresses to each schematic part/split-part.
- o Enabled the "Show Datasheet" option in the right-click navigator menu associated with schematic parts. Please note the command will only be active for parts that have the datasheet address attribute defined.
- o Added "Purge Unused Outlines" commands to the right-click navigator menu associated with outlines.
- o Enabled the "Purge Unused Blocks" commands in the navigator top level schematic folder right-click menu.
- o When trying to delete custom pads from the pad library, a "dependency checks not coded" error was displayed.
- o Reference guide updated.

Changed in release 69 (11th September 2009)

Artwork editor :

- o If you generated a power plane, then left the power plane tools menu and selected 'Undo', a crash would occur.
- o After using 'Redo', all entries vanished from the part placement tray and you had to exit and restart the program to recover them.
- o When an 'Undo' operation caused a net or part of a net to return to an unrouted status, then all signal names vanished from the net list.

Circuit Schematic :

- o Added Undo/Redo capability.

Design Outputs :

- o Added HP-GL pen plotter output filters.

General system :

- o Reference guide updated.

Changed in release 68 (29th June 2009)

Circuit Schematic :

- o If a block I/O symbol was deleted from a schematic sheet, and the corresponding symbol for the schematic was instantiated on other schematics in the design, and the pin being deleted was connected on those other schematics, then an internal inconsistency occurred which resulted in compilation errors.

When you delete an I/O pin, the system will now warn you if that pin is connected elsewhere in the design hierarchy. If you continue to delete the pin, the attached connections will now become correctly deassociated with the pin.

- o When a schematic compilation caused part outlines to be fetched from the master outline library, the outline pad dimensions were not being correctly merged from the master library into the design library.

- o It is now possible to have block names with / in them. eg. I/O
Please note that designs created with such names will not load into earlier versions of XL Designer.

Changed in release 67 (12th June 2009)

Circuit Schematic :

- o When entries were deleted from the schematic part folders within a design, those deleted part names would appear as empty user blocks if the design were opened in RangerXL. XL Designer was not affected by this problem.

- o When a schematic part or userblock was fetched into a design from a master library

or another design, the type of the block in the source library was inadvertently set as '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 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 that is displayed when you right click on a schematic split part in the navigator. Choose the appropriate submenu entry to alter the selected split part to be a userblock, 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.

NB: The "Change Block Type" menu is not normally visible, as it is only intended for recovery of corrupted designs. To enable the menu, enter the File->System Setup dialog and check the "Enable Schematic Block Type Change" box. Please note that XL Designer always starts with this feature disabled. (ie. the enable checkbox state is not remembered)

o Back compatibility note: When a design containing schematic split parts is opened in Ranger XL, the split parts will appear in the 'parts' section of the parts selector together with all the normal parts. Ranger XL does not differentiate between split/non-split parts in its parts browser. XL Designer correctly separates them out into their appropriate folders. This has caused some users to believe that XL Designer is wrongly categorising normal parts as split parts.

Changed in release 66

Artwork editor :

- o If you right-click on the artwork icon in the navigator, and choose 'open detached', then the artwork will open in a separate window that may be dragged to a second monitor.

General system :

- o 'View->Restore Default Layout' command has been added. This will restore the default visibility and positioning of the navigator, browser and status panes.
- o Reference manual updated.
- o IMPORTANT : This product update requires a version 2 license file. Previously issued licenses will require upgrading. (Not relevant for XL Designer Lite version)

Changed in release 65 (26th January 2009)

Bath Scientific test point generator :

- o A crash was occurring when generating test data for some designs.

Artwork editor :

- o The autoplacement, autoroute, copper fill and powerplane modes now contain a back arrow on the toolbar which will return you to manual place/route mode.

Netlist editor :

- o The displayed network 'MinClear' value was not being updated when you changed between inch/metric units.

General system :

- o When the inch/metric hotkeys (normally F9/F10) were used, values were shown incorrectly in certain system dialogs (eg. Artwork editor Parts->Keymove, Outline editor Text->Add, Manual routing parameters dialog, Schematic attribute edit height values)
- o A crash could occur if you had the artwork and parts list editors both open, and you deleted a part from the parts list when that same part was also in the artwork part placement tray.
- o Corrected a roundoff error when points were being snapped to a metric grid. (This error had been observed when checking the coordinates of profile lines using the profile numeric editor)
- o The system should not have allowed an outline to be renamed when the outline is referenced in the parts list.
- o Added 'Delete all Parts & Nets' action to the right-click menu for the navigator Parts folder.

- o Added 'Delete all Nets' action to the right-click menu for the navigator Nets folder.

Changed in release 64 (11th December 2008)

Artwork editor :

- o When a design uses library pads or buried vias, an internal hidden table is created to map drill sizes required for these pads into a size code that can be applied to power plane heat reliefs and anti pads. An internal error in the management of this table resulted in power plane features being created with the wrong drill size associated with their size code. This resulted in anti pads and heat reliefs around buried via and library pad drill holes begin generated with the wrong dimensions.

Designs affected by this problem can be recovered by regenerating the power planes.

Changed in release 63 (5th December 2008)

Circuit Schematic :

- o Find->Signal Name was displaying an 'Internal Error' dialog if a search was attempted on a signal name that did not exist on the schematics.
- o If you began entry of a connection, and terminated the insert mode by selecting another menu operation (instead of clicking on a symbol terminal, or dropping the connection with the right mouse button), then a crash would occur.

Outline Editor :

- o While a component outline was open for editing, every design save operation could cause a shift of the component outline and autoplacer footprint box positions if the 'Outline->Set Outline Datum' command had been used.

General system :

- o A crash could occur when a design was closed.

Changed in release 62 (24th November 2008)

Circuit Schematic :

- o Circuit schematic undo/redo operations have been temporarily

disabled. They will be re-enabled as soon as their defective operation has been corrected.

- o Crash occurred if you used 'Terminals->Assign Names' and edited a terminal name that was > 16 characters long.

Outline Editor & Artwork Editor :

- o The Outline->Corner and SplitBoundary->Corner commands could cause a situation where the line endpoint could not be released.

Parts list editor :

- o When units were toggled between inch/metric, the displayed component position values were displayed incorrectly.

Artwork editor :

- o The artwork checker 'Include copper fill' box should have been checked automatically if the artwork contains copper fill features. Also, if copper fill had been created and then deleted, the artwork checker was warning that the 'Include copper fill' mode was not selected even though the artwork no longer contained filled areas.

Changed in release 61 (10th November 2008)

Circuit Schematic :

- o When a schematic sheet containing placed power block symbols was open for editing, and the 'paste' operation was used to add extra parts to the schematic parts folder, then a crash could occur.
- o The 'Find->Signal Name' command did not work correctly. It should now function in a similar manner to the find part and find net commands which present 'previous/next' buttons in the popup balloon window to enable moving through all instances of the signal name.

Changed in release 60 (3rd November 2008)

General system :

- o Includes updated version of the XL Designer Reference Guide.
- o Added 'Documentation' folder to design tree. It is now possible to create documentation files which are stored internally within a .rxl design file.

To create a documentation file, right click on the 'Documentation' folder within a design, and choose the appropriate file type from the 'New' menu.

It is possible for the user to customise the file types that are presented in the 'Documentation->New' menu :-

In the \Program Files\Seetrax\XL Designer\ product installation directory, there is now a subdirectory named 'DocTemplates'. The DocTemplates\index.txt file defines all the file extensions and file descriptions that are to appear in the Documentation->New menu. When editing this file, the extension and description fields must be separated by a single tab character. For each file type defined in the index.txt file, if a corresponding file named 'templates.{extension}' is created in the DocTemplates directory, then that file will be used as the initial file content for newly created documentation files. If no template file is created for a file type, then the 'New' file action will create an empty (0 bytes long) file.

The documentation file 'Import' action allows any existing file to be added to the documentation folder of the design.

Changed in release 59 (13th October 2008)

Artwork editor :

- o If Active DRC was enabled, then a crash could occur when trying to insert a point into an unroute.

Circuit Schematic :

- o When Ranger XL designs containing flat schematics were opened, the converter was re-converting the flat schematic to hierarchical mode every time the design was saved and reopened, thereby destroying any subsequent editing of the schematic.
- o When a flat schematic is automatically converted to hierarchical mode on design opening, a report of the converted schematic pages will be displayed, and a 'flatcircuit_conversion' entry will be created in the design logfiles.

General system :

- o If you used keyboard up/down arrow keys to move through entries in the navigator, then the properties and browser panes were not updated.

Changed in release 58 (7th October 2008)

Artwork editor :

- o If an unroute was picked up with the Mroute->Move Unroute command, and the operation then cancelled with the right hand mouse button, the unroute was not being redisplayed. An artwork check, job close and reload was needed to make it come back.

Outline editor :

- o It was not possible to use auto pad assign on a bga outline having pads on a 0.5mm pitch as it was incorrectly assigning pads on adjacent rows/columns.

General system :

- o The navigator did not restrict the name length for custom pads, outlines, schematic parts, output tasks and batches, but when the design was saved, the names were truncated to the various internal system limits. (Custom pad and outline names were truncated at 16 characters. Schematic parts, output tasks and batches were truncated at 20 characters). The navigator will now prevent the entry of element names longer than the internal system limits.
- o The maximum permitted name length for output tasks and output batches has been raised from 20 characters to 80 characters.
NB: If designs are created with output task/batch names > 20 characters, and then loaded into earlier versions of XL Designer, the names will again be truncated to 20 characters.
- o The File->Print command in the sizes table editor did not work.

Changed in release 57 (2nd October 2008)

Artwork editor :

- o If a design contained nets with single nodes, and any one of those nodes were shorted to another net, then the artwork checker was reporting all the single node nets as being shorted together.

General system :

- o Added preliminary reference manual (schematic editor only at the moment) to the help menu.

Electra router compatibility :

- o The electra router interface was incompatible with the command syntax requirements of the current series 2 routers.
An option has been added to the Electra router installation configuration dialog to allow select of version 1 or version 2 routers.

Changed in release 56 (8th September 2008)

General system :

- o After the 'ALT' key had been pressed, defined function keys stopped

working in the graphical editors. Please note that if ALT is pressed and no subsequent Windows menu action is selected by keystroke, then it will be necessary to hit ALT a second time to cancel its action and re-enable operation of the function keys in the editors.

- o If a design .rxl file was opened directly from a Windows Explorer window, then the Outputs folder did not appear.

Parts list editor :

- o Parts that were added to the list with "Repeat Part" could not be added to the artwork part placement tray.
- o Clicking on column headers will now sort the parts list by the selected column.

Artwork editor :

- o Pin numbers display did not take into account whether the associated component pad was visible.

Circuit Schematic :

- o The datum point of a bus ripper symbol could not be seen once a connection had been made to it.
- o If a part contained no terminals, then part allocation data was lost after the design had been saved and reloaded.

Design navigator :

- o Duplicate outline names were being created when using the 'Paste' operation.

Specctra & Electra optional autorouters :

- o Added browse buttons to the router installation configuration dialogs to assist in selection of the work directory and router executable file.
- o Work directory and router executable file paths were being limited to 55 characters by the configuration dialog.
- o Circular board profiles were being incorrectly output as octagons. A circular profile will now be interpolated into 128 segments (2 degree steps)

Changed in release 55 (1st September 2008)

General system :

- o It is now possible to load designs that were created with any previous Ranger product.

In the file open dialog, select the appropriate "Files of type" value, and navigate to the appropriate design file or design index file.

Circuit Schematic :

- o If a design is loaded that has an old style flat schematic, then the schematic and its associated device library entries will be converted into design blocks within the hierarchical schematic editor. Please note that there are major differences in the way that "bus" signals were handled in the flat schematic editor and the hierarchical editor. The hierarchical editor uses specially created bus ripper symbols to handle busses, whereas the flat schematic editor indicated busses by setting wires to "bus width". The flat schematic compiler essentially ignored wires set at "bus width", leaving net node grouping by equivalent signal name tags to perform the required connections. The hierarchical editor compiler 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 hierarchical schematic compiler.

Some schematics with busses will need some manual editing after loading if it is desired to recompile the schematic to a parts and net list. If recompilation 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.

A new command :

Wires->Flat Circuit Import Assist->Toggle Wire Section To Non Electrical

has been added to the schematic editor. This permits a section of a connection to be switched to non electrical mode thereby hiding its connectivity from the compiler. Please note that this function only permits certain wire sections to be converted. Wire sections connecting to component pins and sections which are not the first or last in a wire link cannot be changed.

- o The system crashed if you deleted a powerpin symbol from a part, or a subsymbol from a split part.

Design Outputs :

- o It was possible to delete an output task from the navigator while the task was open. This resulted in an immediate crash.
- o The right and bottom print margin was incorrect for outputs to the MS Windows printer.
- o When an artwork layer was output with pads set as filled/drilled, tracks were being pulled back from the pad datum point. This resulted in tracks not being picked up when parts were moved.

Changed in release 54 (18th August 2008)

Gerber Output :

- o When the system output arc tracks to a gerber file, they are approximated by a series of straight line segments. The individual segments were calculated to each subtend a fixed 6 degrees of arc around the line being plotted. This worked well in most cases. However, where an arc track is very shallow (huge radius of curvature) this resulted in only one straight line segment being output. The arc approximation algorithm will now generate as many segments as necessary in order that no point on a segment may be more than 0.001" away from the true arc position.
- o In the sizes table heat relief parameters page, it is possible to enter channel width values that cannot be achieved within the stated drill range for the entered track width and clearance values. When a channel width is too large, it will be observed in the artwork (with pads displayed unfilled) that the heat relief segments appear to turn inside out (they look a bit like bow ties). This does not affect artwork editing but is serious if the job is output to Gerber with this misconfiguration. Artwork output to Gerber (and other targets) will now check for this misconfiguration and raise a warning if detected.

General system :

- o If you created an outline with a comma in its name, the system was crashing when you tried to open the outline.

Changed in release 53 (11th August 2008)

- o The 'Print to File' option for Windows printer outputs did not work.
- o Copy & Paste were not available in the navigator for output tasks.
- o Implemented output batch functionality.

Artwork editor :

- o The artwork checker was giving incorrect results for designs containing custom pads shapes. (Customer supplied job "3004-07_11-03.rxl" was wrongly reporting short circuits)

Circuit Schematic :

- o Find->Part was crashing if you entered a part ident that did not exist.
- o Allocate->ShowFreelist was crashing.
- o When performing a region copy, bus connections were not being copied with the correct width applied.

- o Region->Paste command was shown as active even if nothing had been cut/copied.
- o Region Cut/Copy now works correctly between sheets within the same design. (Features cut/copied in one design cannot be pasted into another design as each design has its own private schematic cut/copy buffer)
- o Added 'Show Dependencies' action to navigator right-click menu for schematic items.

General system :

- o If a design having an old style flat schematic was saved from within XL Designer, then access to the flat schematic was lost if you attempted to reload the design in Ranger XL. As long as no hierarchical schematic blocks are created in a loaded design, any flat schematic will now remain accessible if the design is saved and reloaded in Ranger XL. If heirarchical blocks are created, or schematic parts copied into the design, then deleting all of them will reinstate access to the flat schematic in Ranger XL.
- o Navigator Copy/Paste operations involving schematic design sheets or split parts did not work correctly.

Changed in release 52 (17th July 2008)

General system :

- o The licensing implementation has been improved to allow for the provision of 'schematic only' licenses. Schematic only licenses will only work with this and newer releases of XL Designer.
- o Newly issued full system XL Designer licenses which are locked with a security key (dongle) will now allow operation of all program sections except artwork when the key is unplugged. With the key unplugged, the artwork editor works in view-only mode. Newly issued licenses having this feature will only work with this and newer releases of XL Designer. Previously issued licenses will continue to work with this version, but will not benefit from the ability to detach the key while retaining partial operation.

Profile editor :

- o "Grid->Select Origin" did not work.

Custom pad editor :

- o "Grid->Select Origin" did not work.

- o "Grid->Measure Distances" did not work.

Changed in release 51

General system :

- o If you selected a net in the navigator pane, and the net had a large number of nodes, then the property pane would take a very long time to draw. (A net with > 500 nodes was taking > 30 seconds)
- o Navigator "Find in Schematic" for nets was not finding net nodes that had alphanumeric pin references.
- o Right clicking on navigator standard pad folders now shows an "Open Sizes Table" action. Previously, it was incorrectly showing a "New" which created a custom pad.
- o Properties pane will now display attributes and power pin information when browsing schematic parts in the navigator.
- o Clicking on a navigator design parts folder will now cause the properties pane to show a count of the number of placed and unplaced parts in the design.
- o Clicking on a navigator design nets folder will now cause the properties pane to show a count of the number of power and signal nets in the design.
- o The navigator Nets folder now uses a different icon for power nets.
- o Implemented design auto-save. This must be switched on by ticking the option box in the 'File->System Setup' dialog. Please note that auto-save does not save changes that are in progress on master libraries.
- o The selected system inch/metric units mode now gets preserved across program sessions.
- o Grid snapping state now gets saved with the design. Also, the state is now stored separately for the artwork/profile, outlines, custom pads and schematic views within the design.

Design outputs :

- o Plot preview was not being shown correctly for some plots (black screen)
- o When the N/C drill output encountered co-incident drill holes, invalid data was being placed in the drill data file.

Artwork editor:

- o If a directory named c:\temp existed, the artwork checker was creating a file called c:\temp\xldebug.txt

Outline editor :

- o 'Outline->Set Outline Datum' command did not work.
- o If a design was saved while a component outline was opened for editing, then changes/additions to the outline pins were not saved into the design file.

Parts list editor :

- o Enabled bill of materials output function in the 'View' menu. Bill of materials may now be output in the old "RangerXL" report style, or in a spreadsheet compatible style with selectable field separator.
- o Enabled 'File' menu 'Save Parts List as Text' action.
- o Enabled 'File' menu 'Print' action.
- o 'View->Show Part UIDs' setting will now get remembered between program sessions.
- o 'Edit->Find Part' has been made case insensitive.

Netlist editor :

- o Enabled 'File' menu 'Save Nets List as Text' action.
- o Enabled 'File' menu 'Print' action.
- o Enabled 'Edit->Find Signal Name' action.
- o Enabled 'Edit->Find Node' action.
- o Enabled 'Edit->Find Unconnected Pins' action.
- o Enabled 'View->Show NETUID Codes' and 'View->Show Minimum Clearances' actions.

Demonstration version only :

- o When you load a design that has too many parts or nets for the design size limits of the demonstration version, a warning dialog will be displayed if you attempt to save the truncated design over the original loaded design.

Changed in release 50

General system :

- o Clicking parts and nets in the navigator pane did not show the correct properties information in the properties pane.
- o System can now display multiple popup windows for the navigator "Find in Artwork" and "Find in Schematic" operations.

- o Popup windows for the "Find in Artwork" and "Find in Schematic" operations for parts/nets now contain buttons that allow you to rapidly go to the identified feature in other views. Also, when an information popup is placed on a circuit schematic symbol, and the symbol is allocated to a part containing multiple elements, the popup will contain buttons to assist in finding other members of the same part.

Changed in release 49

Circuit Schematic :

- o System was crashing when you dropped parts into the fast parts tray.

General system :

- o If you created a new circuit schematic, then compiled it, then opened the parts or net list editor, there was no correlation between the parts and nets displayed in the navigator pane and their entries in the parts/nets editors. Also, if an artwork was produced, then the navigator "Find in artwork" options did not work. Normal operation was only obtained after saving, closing and reopening the design.
- o If you added a part to the parts list by using the artwork "Tools->Network->Add Part" function, and then tried to delete the part from the parts list editor, then the system would crash.
- o Added software reference manual to the Help menu.
- o Added online check for product updates to the Help menu.
- o A customer supplied design (BZ821_defekt.rxl) was failing to load because a net had a signal name tag that was missing from the signal name table. Invalid signal name tags now get cancelled on net loadup. The source of the invalid name has yet to be determined.

Changed in release 48

General system :

- o Popup information windows (eg. identify part, x/y etc) have been restyled to be "Balloon" type windows.
- o Navigator "Find in Artwork" is now operational for parts and nets.
- o Initial implementation of navigator "Find in Schematic". This release only contains limited functionality for finding parts in the schematic, and does not yet work correctly for parts that have multiple elements.

Changed in release 47

General system :

- o Added Cadence/CCT Specctra and Konekt Electra optional autorouter interface.
(To access optional routers, right click on your artwork icon in the navigator.
Optional routers must be enabled in the File->System Setup dialog in order to appear in the artwork icon context menu)

Outline Editor :

- o X/Y readout was not working.

Circuit Schematic :

- o The system was crashing if you tried to place a symbol from a split part that was located in a master library.

Changed in release 46

General system :

- o Added profile import from AutoCAD DXF file.
- o Added artwork import from Gerber file.
- o Added parts/netlist import from text file.
- o GenCAD export tool was crashing on supplied customer designs.
(The supplied designs had some pad table definitions where the dimensions displayed in the standard pad size editor appeared to be zero, but internally, roundoff error had resulted in a very small non-zero value, causing the pad output algorithm to fail)

Changed in release 45

Outline Editor :

- o System was crashing when you closed an outline that had been opened from the master outlines collection.

General system :

- o Added GenCAD 1.4 export tool.
- o Added Bath Scientific Limited testpoint generator.

Changed in release 44

Outputs Editor :

- o "RS 274X (Extended gerber)" text change was missing from the Gerber output non-auto job setup dialog.
- o Added DXF output filter.
- o Added output to artwork layer filter.

Changed in release 43

Circuit Schematic :

- o Added wires 0/90 degree lock button to the toolbar, and entry to the wires submenu.
- o A horizontal scroll bar will now appear in the fast parts tray if the contained text is too wide for the window.
- o It should have only been possible to select "Bus Pin" type on the toolbar when editing "User Block" and "Block I/O" symbols.

Artwork editor :

- o When entering powerplane mode, the powerplane toolbar would display all powernames that were defined in the power names configuration table, even if the power name did not occur in the design netlist. After entering the powerplane setup dialog, and closing it again, the displayed powername table would reduce to just those power names that appeared in the netlist. This was confusing.
- o In powerplane mode, if you selected a powername in the toolbar, then entered powerplane setup mode and switched the toolbar selected power name if 'Inactive', then the system would crash after the setup dialog was closed.
- o The powerplane preview mode did not repaint correctly. (After panning/zooming, or using the setup dialog, it was necessary to switch preview mode off and on again to restore the correct preview display)
- o When using 'Text > Move' mode to move a 'URL' text item, the item was not undraw correctly when picked up.

Outputs Editor :

- o In the gerber output configuration dialogs, "Extended gerber" output mode has been renamed to "RS 274X (Extended gerber)"

Outline Editor :

- o If you added, deleted, or moved pads on an outline, then the change was not visible in the artwork until the design had been saved and reloaded.

Profile Editor :

- o The "View > Drill Holes" mode was only switching on/off the display of drill holes, and the part pins were always being displayed. This was different to the behaviour in Ranger XL. Also, the menu entry has been renamed to "View > Part Pins" to agree with its actual functionality.

General system :

- o When new standard pad sizes were defined with the pad sizes editor, the newly entered sizes did not appear in the navigator until the software had been terminated and restarted.
- o When windows were resized to extreme aspect ratios, full use was not being made of the available drawing area. (eg. Window > Tile Horizontal with several windows open would result in windows that only had partial grid coverage)

Changed in release 42

Circuit Schematic :

- o Implementation of sheet editor Symbol->Power Place mode.
- o Implementation of 'Region' commands.
- o Implementation of user defined design extractor
- o Schematic compiler was causing an "Invalid Argument" error popup to appear. Trying to continue would cause the system to crash.
- o Power symbols were vanishing from schematic sheets after a part had been placed from a master library.
- o If a userblock symbol was deleted from a schematic sheet while a view of the design hierarchy was open, the displayed hierarchy was not updated and could cause a crash.
- o It was not possible to delete parts and user blocks because of warning about "dependency checks not coded"

- o When a new part was added to the schematic, allocated, then the circuit compiled, artwork silk screen features that had been manually added were being deleted.
- o EMC attribute tags were not visible, and the system crashed if you tried to apply them.
- o F4(full) and View > Full was not working
- o The split part subsymbol swap rules dialog bar global/local selector had no default selection.
- o The RangerXL product uses a separate data section in the design database to hold gate and pin swapping rules extracted from library part and split parts. XLDesigner derives its swap rules directly from the library members, and does not require and was not generating this separate swap rules database.
This resulted in a backward compatibility problem when loading XLDesigner created designs in RangerXL.
XLDesigner now generates this missing database.
- o Symbol and pin swap rules setting modes incorrectly showed rule 'circles' on placeable power blocks and power block pins.
- o Compiler did not correctly insert EMC attributes into extracted parts and nets data.
- o Reinstated the compilation result viewer.
- o When connections were inserted onto a blockio symbol pin, the connection would not attach correctly to the pin. (if you picked up the symbol to move it, the connection would stay behind.
- o Symbol and pin swapping rules should only have been visible for part and split-part type symbols.
- o It was not possible to specify a typed pin reference to force allocation of a split part symbol to a specific subsymbol within the component.
- o Pin number assignment did not place a default value in the pin number box.

Artwork editor :

- o Corrected memory corruption in gate/pin swap rules handling.
- o Gate and pin swap rules were not being handled for schematic split parts.
- o Parts EMC highlighting mode was not working.

- o Region > Move/Rotate was not updating unroutes.
- o Region > Macro Generate and Region > Macro Add were disabled.
- o User defined via holes were not being displayed.
- o Amend > Enter Pads did not permit entry of user defined via holes.

Custom Pad Editor :

- o DrillHoles > Delete command did not work.
- o The drill size value in the toolbar did not work.

General system :

- o Schematic symbols, outlines etc. now get loaded into the navigator window sorted into alphanumeric order.
- o Component outline names in the navigator now get forced to upper case to be consistent with the parts list editor.

Changed in release 41

(Release not distributed)

Changed in release 40

(Release not distributed)

Changed in release 39

Circuit Schematic :

- o Implementation of the fast-parts tray.

Artwork editor :

- o A design that had custom pads in the free copper section of an outline was crashing if the design was saved, then reopened, then attempted to open the artwork editor.

Changed in release 38

Circuit Schematic :

- o Fixed compiler problem with signal names.

- o Fixed display problem with attribute positions on split part subsymbols.
- o When a design sheet was renamed, the corresponding symbol entry was not renamed.
- o Paste buffer was not being cleared at start of a new copy/cut operation.
- o When editing a symbol outline, line endpoints could not be released after the attributes move command had been used.

Changed in release 37

Circuit Schematic :

- o Implementation of schematic parts & split parts editors.
- o Implementation of hierarchy navigator functions.

Changed in release 36

Design Outputs :

- o Added the hierarchical schematic plot filter.

Circuit schematic :

- o Non electrical text strings containing version control markup was not being handled correctly.

Changed in release 35

General system :

- o When outlines are imported into a design, and pad/track size code translation is required on import, size code 0 will no longer be allocated to translated features.
- o If you renamed a circuit schematic library entry, and then moved the main application window without first clicking again on a navigator entry, a crash occurred.
- o It was not possible to save changes made to the design master libraries.

Outline editor :

- o It was possible to place a pad on an outline when no pad style had been selected from the navigator. A default round pad was being displayed, and it was possible to allocate a pin number to it. However, the resultant outline was unusable, and could cause a crash when trying to add it to the parts list.
(System will now beep, and display a prompt on the status bar asking for the selection of a pad style)
- o When using Region > Copy with pads selected, the system would become unresponsive, and consume memory until a crash occurred.
- o Copper > Add/Move Corner would not move corners that had been added in copper lines on copper layers.
- o Copper > Add/Move Corner would not move copper line end points that were not attached to a part pin.

Artwork editor :

- o Tools > Network > Add Part was causing a crash. (Problem introduced at version 1.34)
- o Tools > Generate Silk Screen caused a crash if you selected insertion of free copper features.

Changed in release 34

Profile editor:

- o The system was crashing when trying to add a line feature to the profile in the master profile library.

General system :

- o It was possible to use "Paste" in the navigator window to overwrite a design custom pad, outline, schematic library entry or profile object when that object was open for editing. This resulted in a crash.

Component outlines :

- o When using Copy/Paste to copy outlines between designs and the master outline library, required custom pads were not transferred with the copied outlines.
- o When outlines were imported to a design from the master library or another design, the size codes of imported pads and line features were not translated if there was a mismatch in the size table configurations between the source and destination.
- o Outline "families" are now available. To configure families for the outlines in a design, or the master outlines, right click on the

top level outline folder in the navigator, then choose "Manage Families" from the context menu.

- o Bulk editing of outline properties is now available. To invoke the editor, right click on the top level outline folder in the navigator, then choose "Edit Properties"

Artwork editor :

- o "Parts -> Change Outline" command is now available.

Parts list editor :

- o Editing the outline name field for a part that is already placed on the artwork will now have the same editing effect on the artwork as using the artwork editor "Parts -> Change Outline" command.

Changed in release 33

Outline editor :

- o System could crash when creating a new outline.
- o When an outline was opened, the autoplacement footprint was not being checked to ensure that it encircled the pin bounds.
- o Problems with outline corner/move point and free copper corner commands.
- o Pad angle control was not working when adding pads.

Changed in release 32

Outline editor :

- o Initial release of component outline editor.

Changed in preview release 31

General system :

- o Circuit schematic, design output tasks, and unreferenced outlines and custom pads would be lost from a design if a design was saved without any of the above data being first explored in the design navigator.

Changed in preview release 30

Artwork editor :

- o Back annotation from gate/pin swap and renumbering operations will now be propagated to the circuit schematic.
- o The artwork checker did not see size changes made to the sizes table in the current artwork editing session.

Circuit schematic :

- o System was crashing if you tried to open a part that had a comma in its name.

Sizes table editor :

- o If you used the function keys to toggle between inch/metric units, the currently selected value got corrupted.

Changed in preview release 29

Artwork editor :

- o System was crashing when you closed the artwork editor for a design where Amend->Replace Pad had been used on component pads.
- o Pads disappeared if you selected "filled" display mode.
- o "Named" unroutes view mode was not working.

Design Outputs :

- o NC Drill and Rout data output implemented.
- o System will now issue an audible beep and display "Plot Complete" on the status bar on completion of plot file creation.
- o Configure->Move Plot and Keymove Plot were not working.
- o Solder Paste setup dialog was failing.
- o Drill drawing output was failing.

Changed in preview release 28

- o First release of Gerber plot implementation.

Changed in preview release 27

Circuit schematic :

- o Symbol->Attribs->Move/Edit etc. was not picking up the correct attribute.
- o Wires->Attribs->Add was behaving in an erratic manner.
- o Allocate mode often required a deallocate action before it could be made to work.

Changed in preview release 26

Circuit schematic :

- o It was not possible to make a connection to a block io pin.
(This error was reported as "p/w list compiler is not joining nets up through the hierarchy". A test job had been created, but because the connections would not attach to the block io pins, they had been released over the screen location where the io pin appeared, visually making a connection. However, because the io pin locator had not correctly located the pin as a valid connection target, the connection node was not tagged as connecting to the pin, with the resultant apparent compilation error)

Changed in preview release 25

Circuit schematic :

- o Circuit schematic was not being saved.
- o Edit -> Preferences dialog now available.
- o Schematic compilation now correctly generates unroutes in the artwork.

Changed in preview release 24

Artwork editor

- o Artwork editor now picks up gate and pin swap rules from the schematic parts. Please note that 'split' parts do not yet generate gate and pin swap information

Initial preview of circuit schematic capture.

Known problems :

- o Symbol editing mode not yet available.
- o Design navigation using the hierarchy tree not possible.
- o When 'find' operations require the opening of another view, the find marker is not placed on the correct sheet.
- o Many problems with 'split' parts and placeable power blocks.
- o Gate and pin swapping in the artwork does not yet back annotate to the schematic.
- o 'Region' commands not yet available.
- o Undo/Redo not yet available.
- o Allocation sometimes does not work unless you first do a deallocation operation.
- o Design compilation produces parts and net lists, but unrouted may not appear in the artwork editor.
- o System will crash if you use 'Copy' and 'Paste' on the design navigator to copy a schematic design sheet from another design into the currently open design. However, using part place mode to insert a design sheet from another open design will cause the design sheet and all required dependencies to be correctly imported.