Version 14.0.6
Software Update Details

Problem Fixes in Version 14.0.6 : 15-Aug-2011

Add Block Port
o   Cancelling Add Block Port in a Schematic design which did not belong to a Project could cause the application to quit.

Colours
o   Design data for custom colours was not being rationalised, causing the resulting design files to grow in size each time the file was saved.

Eagle Import
o   Logic names were not being imported in designs coming from Eagle.

Gerber Output
o   Slots were being output as round holes instead of the correct shape for the slot as defined in the Padstyle.
o   The macro for finger (oval) pads positioned at odd angles was not rotating the rectangle accurately.

Move
o   Moving a section of schematic circuit added using Paste could cause bus connections to be lost.
o   There was a delay of up to 20 seconds in some designs after dragging a component out of the Component Bin into the design.

ODB++
o   Plated Slots were not being included in the ODB++ output.
o   Slots were being output as open shapes instead of filled.

Plotting
o   Powerplane copper was not being backed away to leave the required gap around drilled-out pads.

Pour Copper
o   Copper was being poured outside a circular board outline in a particular design.

TraceRouter
o   The presence of copper in PCB symbols could cause the TraceRouter to abandon routing without reporting any reason for the failure.

Problem Fixes in Version 14.0.5 : 30-Mar-2011

Design Rule Check
o   Several instances of spurious errors were being reported (copper shape verification, net connectivity, and track to area).

Drawing
o   The picture was not being refreshed after changing the oversize value on a layer class.

Gerber Output
o   The isolation gap used on power planes was incorrect for some nets that used Guard Spacing.
o   Part of the hardware-filled copper was missing on one specific design.
o   Generation of hardware-filled copper was hanging the application on one specific design.

Integrity Check
o   The component setting 'Suppress from Scm to/from Pcb' was not being used by Integrity Check, resulting in design changes being forwarded to the Pcb when they were not required.

Measure
o   The Area of a filled shape was displaying the value for Height instead.
o   The spacing for a board outline was not being displayed.

ODB++ Output
o   Two specific designs had problems with cutouts in copper shapes.

PDF Output
o   In some cases the PDF Output would erroneously report 'Error: Plot Failure'.
o   A schematic design containing a sheet outline implemented as schematic-only component could result in that component 'covering' all other components in the PDF output, preventing the selection of those other components to see the popup showing their properties.

Plotting
o   Some textual elements (net names, block names, etc) were missing from plots.
o   Connected block ports were not being suppressed in the same way as component pins.

Pour Copper
o   Some specific issues with pouring copper in particular designs.

Reports
o   Using both 'collated' columns and column totals could result in the totals being output in the wrong column.

Styles
o   Various properties of the Hatch Style of a shape were changing or reverting to their previous values when the shape itself was edited.

Problem Fixes in Version 14.0.4 : 08-Mar-2011

Component Edit
o   Deleting a gate in a multi-gate package could corrupt the pin assignments on other gates.

Design Rule Check
o   Copper shape verification on one specific design file was causing the application to quit.
o   Using the 'selection only' option to check only between a selected track segment and a selected pad was not reporting a spacing error.

DXF
o   Presence of brackets in padstyle names could cause the DXF file to be rejected by AutoCAD.
o   If more than one layer is output, pads were being output on silkscreen layers.
o   Oversized layers such as solder masks were being output without over-sizing the pads.
o   Vias were not being included on solder mask layers.
o   If outputting both paste and solder masks, oversizing of pads on the paste mask was too big.
o   Pads with 'by layer' exceptions were not being output on the layer with the exception.
o   Drill drawing layers could be empty.

Easy-PC Import
o   Through-hole pads in Easy-PC DOS design files were being placed on [Top] layer during import.

IDF Export
o   Nested board cutouts were not being output as cutouts in the IDF data file.

Library Manager
o   If no library files exist in any library folder, clicking New to create the first new library would cause the application to quit.
o   Attempting to edit a package definition when the 3D Package library was read-only would cause the application to quit.

ODB++
o   Some footprints with pins numbered non-sequentially could cause some CAM software to mis-report the pin number for some component pads.

Optimise Nets
o   The calculated position for connecting to copper shapes was not always optimal.

Plotting
o   Some newly created plot entries had their plot 'type' set to Drill Ident instead of Artwork.
o   Two specific design files caused the application to hang while generating Gerber data.

Properties
o   PCB-only components were being reported as Schematic-only instead.

ProRouter
o   Some pre-routing was being removed even though the switches on the dialog were set to prevent this.

Status Bar
o   The size of a rectangular board was not always being shown or updated on the status bar.

Teardrops
o   Some teardrops were not being added despite there being sufficient space to do so.

Problem Fixes in Version 14.0.3 : 25-Nov-2010

DXF Import
o   Block entries with spaces in their block name could cause the import process to hang.

Eagle Import
o   Some Eagle designs did not have the correct layer stack set up on import into Easy-PC.

Pro-Router
o   Installation of 14.0.2 patch caused ProRouter to stop working.

Program Changes in Version 14.0.2 : 16-Nov-2010

Edit Track
When attempting to use 'Finish Here' to end a track when it was over another item on a different layer, it was not possible to actually Finish the track. This has been changed in this update as follows:

1)   When doing 'Finish Here' the 'Confirm Add Via' dialog has 'Finish Here' button instead of 'Add Corner'.

2)   When adding a corner, the Shift button can be held down to force the corner to be added without attempting to select or connect to anything that might be at or near the cursor.

3)   Shift+DoubleClick will always finish the track (first click does Corner, second does Finish).

4)   The PCB Tracks page of the Preferences dialog has a setting for when the 'Confirm Add Via' dialog is displayed. These settings are 'Always' (as it was behaving before), 'Never', and 'If not on different net'.

Problem Fixes in Version 14.0.2 : 16-Nov-2010

Component Edit
o   Duplicate copies of the Component Name etc could be displayed in the previews when editing a Component.
o   It was not possible to add Values to a schematic-only Component during Component Edit or when the Component was used in a design.
o   If both Schematic or PCB checkboxes were accidentally unchecked on the New Component dialog, the program would hang on attempting to OK the dialog.

Dual Screen
o   If a design file had been saved at the same time as it was being used by the 'other' instance in a Dual Screen configuration, the design could be marked as locked which then prevented it from being opened correctly next time.

Eagle Import
o   Mounting holes were not being correctly imported from Eagle PCB designs.
o   One specific Eagle design caused the application to quit on attempting to import the file.

IDF Export
o   All possible Component values were being output, instead of only those which were actually used on each Component.

ODB++
o   The ODB++ output data for some designs did not include the drill information.

PDF Output
o   Mirrored TrueType text was not being included in the PDF plot.
o   Bitmaps could be misplaced when generating a rotated plot.
o   Some pad shapes such as Octagon were always being drawn filled regardless of dialog settings.

Plotting
o   Thermal pads for bullet-shaped pads were not being generated correctly on powerplane outputs.

Pour Copper
o   Two specific PCB design files had areas that would not pour correctly.

TraceRouter
o   It was not possible to selectively route by chosen Net Class(es).
o   Tracks would sometimes be routed on a power/ground plane even though there was space available on a normal electrical layer.

Problem Fixes in Version 14.0.1 : 29-Jul-2010

Cut to Block
o   Some connections were not being 'cut' in the correct place, leading to the wrong number of block ports being created.

Databook
o   The Library Databook did not 'track' the current selected library so it proved difficult to make it display information about libraries from another folder. It should now track the currently selected Component library when the Databook is opened or refreshed.

Design Rule Check
o   Progress bar not being updated when running 'Copper Verification' check.

Display
o   The display dialog in a Schematic design still included the 'old' colour settings for Screen Grid.

Excellon (N.C.Drill) Output
o   Most drill files were being generated with no drill data in them.

Forward Design Changes
o   When a component in PCB was being updated due to a change in Package, the component name was sometimes being shifted a long way from the original component position.

Grids
o   If all grids were marked as 'draw first', then display of grids could not be switched off.
o   Ruler Stops were always being drawn first (behind design items).
o   Changing grid while moving an item was not snapping the item onto the new grid.

Plotting & Printing
o   The settings for plated/unplated round/slot holes were not being saved to Plot Job files for Drill Ident plots.
o   The checkboxes for selecting inclusion of plated/unplated board outlines on normal plots were no longer visible.

Pro-Router
o   The Pro-Router was not displaying it's Report on the screen at the end of the routing process.

Save To Library
o   If a new library was created from the Save Items To Library dialog, this new library did not appear in the list of available libraries unless the folder containing that library was disabled and re-enabled, or the application restarted.

Setup
o   Installing V14 over an existing previous version was not putting the new Report Templates into the template folder used by the existing application. As a result those new reports were not appearing in the list on the Reports dialog.