Version 11.0.7
Software Update Details
Release Date 23-Oct-2008

Problem Fixes in 11.0.7

Add Component

- Opening the Add Component dialog, then cancelling and immediately re-entering the dialog, could in some circumstances cause the application to quit.

Auto-Route Schematic

- Attempting to auto-route some schematic designs created from PCB through the Reverse Engineer option would cause the application to quit.

Component Edit

- In some circumstances, adding and removing a number of gates from a component could cause the application to quit.

File Properties

- Attempting to access the Properties for design documents with incomplete summary information (time, date, etc) could cause the application to quit.

File Save

- Design files could not be saved in Folders marked with Windows ‘read only’ property. This property is not a valid method of making a folder read-only, and is only partially supported and maintained by Windows, leading to inconsistent results when retrieving the read-only state of the folder.

Import Gerber

- Drill sizes in some Gerber files were being imported as the wrong size.
- In some circumstances, items were being incorrectly linked together onto the same nets during import.

Move Schematic Connection

- Moving the open end of a ‘dangling’ schematic connection back over its originating terminal could cause the application to quit.

Pour Copper

- Attempting to pour on some specific designs resulted in incomplete or incorrect copper shapes.

Pro-Router

- If any layers in the design had no layer type assigned, this would cause the application to quit on attempting to launch Pro-Router.
- A specific design with a circular routing area could not be routed.

Reverse Engineer

- Positioning of components on the generated schematic drawing was not optimal, and in some situations could cause components to be positioned outside the drawing area.

Problem Fixes in 11.0.6

Back Annotation
- Changing only the case of a component name in PCB would prevent Back Annotation from running to completion.

Design Rule Check
- Minimum Annular Ring check was being applied to pads on non-electrical layers.

Disconnect From Net
- This option was not previously available for Free Pads unless they were routed.

Display
- In the Schematic View Display dialog, it was not possible to change the colour for Symbol Text. This was due to the presence of a colour control labelled Comp Names which should not have been there.

Duplicate
- If tracks on different layers were selected without selecting the vias that joined them, the resulting tracks would be joined with a junction even though they were on different layers.

DXF
- Importing one particular DXF file produced malformed shapes in the resulting design.

Footprint Wizard
- The Wizard was not detecting where pads needed to be rotated as the pad length become less than the width.
- For the CHP type, entering a new value for 'T' did not change the pad pitch.

Gerber
- Output of filled copper on one particular Gerber plot was incomplete.
- Filled shape generated for copper in one particular design was incorrect, causing a short.

Import Easy-PC DOS
- Some Easy-PC DOS schematic files would not show their component values after import.

Import Gerber
- Some filled shapes were being imported over-sized.
- When importing, styles pre-defined in the design/technology were not being used, instead it was creating new styles.

Join Nets
- The dialog that is displayed when two complete nets were joined was similar enough to that shown when joining subnets as to cause mistakes to be made by erroneously accepting the change.

Optimise Nets
- Insulated wires crossing over pads were being treated as connected when in fact no connection should be made at that point.

Pour Copper
- Attempting to pour on some specific designs resulted in incomplete or incorrect copper shapes.

Reports
- The Layers report did not include information about Layer Spans, so the details about layer stackup could not easily be passed to manufacturers.

Setup
- The example Map files for Orcad and EWB netlist import were not being installed.

Shortcuts
- The First/Next Error commands only worked in PCB, not in Schematics.

Star Points
- Pour would not create thermal spokes on star points.

Support Info
- In some circumstances, viewing the Support Info dialog was causing the application to become unstable.

Update Components
- The presence of blank lines in the 'libnames.txt' file would cause Update Components to fail, and report the wrong library name.

Problem Fixes in 11.0.5

BoardMaker
- Component names were being truncated to 11 characters, which could lead to some components using the wrong component definition if two definitions were different only in their 12th character.

Gerber
- Output of software-filled hatching was failing to produce the correct result on one specific design.

Import Orcad Netlist
- Importing Orcad netlist files was not recognising or using the Map file.

Pour Copper
- Three specific designs exhibited problems with their Pour results.

ProRouter
- When using ProRouter to interactively route selected nets, vias were being set as Protected.

Problem Fixes in 11.0.4

Auto Rename
- Components outside the board outline could be renamed regardless of the settings on the dialog.
- No components were being renamed if the option to rename selected components was chosen.

Component Wizard
- If the Preview checkbox is turned off while using the Component Wizard, next time the Wizard is used the application would quit on reaching the Symbol page.

Delete
- In some circumstances, deleting poured copper or other items on a net could cause other poured copper to be removed from the net. This problem was generally showing up when pouring one copper area would cause another copper area to 'lose' its net.

Duplicate
- Duplicating a via would cause the new via to be marked as Fixed instead of Protected.

DXF

- If the design coordinate origin was non-zero, some items in the DXF output were misplaced from the rest of the design.
- Junction points in schematic designs were always being output in yellow.

Easy-Router

- In a particular set of circumstances one calculation could result in a divide-by-zero error and cause the application to quit.

Import Easy-PC DOS

- Easy-PC DOS design files with text items longer than 127 characters would fail to load.

Import Gerber

- Attempting to import a specific Gerber file from another system was causing the application to quit.
- When using Intelligent Gerber Import, no warning was given when ‘drawn’ items were found to be using an aperture (shape) other than Round.

Integrity Check

- Comparing a specific pair of designs that compared with no differences in earlier versions of the program was giving spurious differences.

Library

- A report listing the contents of any symbol library could be missing up to five items from the end of the report.

Load Technology

- Using Load Technology to update an existing net with new net class information could result in the design file containing invalid settings, which may later cause the application to quit.

ODB++

- Multi-line names in PCB (names displaying more than one field) could cause the ODB++ output file to be invalid.

Open

- No warning was given when attempting to open a backup or security copy file.

PCB Symbol Wizard

- The controls for the ‘chip’ style footprints were confusing the ‘b’ and ‘L’ values.

Plotting

- The settings for Board Outline (Board, Plated, Unplated) were not being consistently saved and retrieved in Plot Job files.

Pour Copper

- One particular design would cause the application to lock up during ‘merge shapes’ while pouring.
- Two odd-shape boards with many irregular shapes were not producing the correct result.

Print

- When printing a PCB, some items were not being printed in the correct layer order as displayed on the screen. This could for example cause some silkscreen shapes to be printed underneath tracks.
- A message about printing masks in colour was displayed every time a schematic was printed to a colour printer.

Properties

- Selecting multiple track segments on different layers and using Properties to change the width would cause the application to quit.
- Selecting multiple text items and using Properties to change the text style would cause the application to quit.

Rotate

- Selecting several component names and using Rotate was rotating them as a 'set' instead of simply rotating each name in place as in previous releases.

Sketch Connection

- Attempting to use Sketch Connection in point-to-point mode could cause the application to quit.

Track Fatten

- In some PCB designs, using Fatten All Tracks could cause the application to quit.

Track Hugging

- After changing style partway through Track Hugging, the style would revert to the original style after adding the next corner.

Program Changes in 11.0.3

Change Shape Type

- A new option has been added to the shortcut (right-mouse) menu when selecting different types of shape. Select a shape and right-click, choose Change Shape Type option from the shortcut menu. A dialog will be shown with a drop-down list to use to choose the new shape type. This could be used for example to change a documentation shape to a board outline.

Problem Fixes in 11.0.3

Add Reference Origin

- Attempting to add a Reference Origin to a component that already had Values was giving the wrong results.

Email

- Depending on the type and availability of the mail software installed, on some systems it was possible that attempting to use the in-built 'Email Sales' or 'Email Support' commands would cause the application to quit.

Layers

- The Layers dialog was preventing non-electrical layers being set as side 'Inner'.

ODB++

- Some octagonal pads were being output 'stretched' (length greater than their width).

Paste

- Attempting to paste a component into an existing design where the component was already used in that design, and where both source and target components had fewer pins mapped than exist on the footprint, and the mapping was different in each case, could cause the application to quit if the source component had tracks connected to the pin that was not mapped in the target design.

Pour Copper

- Pouring one particular design caused the copper to flood across the whole board.

- On components with pads close together, some thermal spokes were being added even though these would cause Design Rule Check errors.

Projects
- Some actions on designs within Projects (including right-click on Components, saving design files) could repeatedly cause the display of the spurious message “Failed to open document”.

ProRouter
- Pre-routed vias placed very close to pads on the same net were being flagged as errors, and preventing those nets from being fully routed.

Select Mode
- The menu option “Disconnect From Net” was not available for pads if they had no connections. So pads could be put onto a net using “Add To Net” but could not be removed from the net.

Shape Information Bar
- The ‘Closed’ checkbox was being left enabled even for items which are not allowed to change their Closed state (Board, Bus, etc).

Problem Fixes in 11.0.2

Design Revision Analysis
- Some changes of Component were not being reported.

Design Rule Check
- The Drill check in DRC has been extended to report all coincident drill holes, regardless of net and size.
- Some checks were occasionally using the wrong co-ordinates for the location of the error marker, causing those markers to be placed outside the working area.
- The check for Min Annular Ring was not checking Vias, only Pads, and did not handle pad style exceptions.

DXF Output
- The set of colours used for items in DXF output was incomplete, which could result in some colours being mapped to Yellow. The DXF output option will now map all Easy-PC colours to their nearest AutoCAD equivalent colour.
- Most track and shape segments had square ends instead of being rounded-off as they are on-screen.

Footprint Wizard
- On some types of footprint, ticking the checkbox for ‘Dot at pad 1’ would cause the footprint picture to become extremely small as it placed some items a long distance from the main contents of the symbol.

Intelligent Gerber Import
- Some silkscreen imports were producing unreadable results.

Optimise Nets
- Some vias were being removed during Optimise Nets even if they were set as Protected.

Pour Copper
- The wrong clearance was being used when pouring against closed, unfilled shapes.
- Some poured copper shapes were being treated as isolated even though they overlapping component copper on the pour net.

Print
- Text using the in-built stroke font had a yellow dot on the end of each stroke when printed to Postscript and to some PDF drivers.

ProRouter
- Some track segments were being added marked as ‘Fixed’ even without setting the ‘Fix New’ option.

Reports
- The Design Status report included ‘duplicate’ drill holes in its count of holes, which could result in it showing different counts to the N.C.Drill output and Drill Tables. The report now includes positions of duplicate holes.

Support
- The Support report was not being written to a properly-defined path, causing a ‘common dialog error’ on attempting to use Save As from the resulting Notepad window.

Problem Fixes in 11.0.1

Add Via
- If the default Net Class had no pad style assigned to use for Vias, the Add Via command was not trapping this out which was then causing the application to quit.

Change Style
- Changing the style of a track while interactively adding or editing the track could cause the application to quit, if the positions of the various track corners resulted in one or more track segments being tidied out as the style was changed.

Edit Track
- Tidying up tracks after editing was not always retaining vias marked as Protected.

Integrity Check
- After removing connections from the Schematic, Integrity Check would report the corresponding connections in the PCB design as PCB-only, and not remove them.

ODB++
- One particular design produced an ODB++ file that would not read into CAM software.

Pour Copper
- One particular design with a track around the board outline would result in isolated copper being left outside the board.
- One particular design with slightly rotated components containing component copper would result in copper being poured across the whole component.

Properties
- Having selected multiple free pads, attempting to changing one the coordinates of those pads using Properties would mistakenly report that the value was outside the design area.

Pro-Router
- Vias marked as ‘protected’ were not always being treated as such by the router.

Select
- Using Ctrl+Click to toggle items out of the current selection was not redrawing those items, giving the appearance of them still being selected.

Styles

- Using the Delete Unused button to remove unused pad styles could remove pad style exceptions that were actually still being used.

Windows Output

- Using Number of Copies to produce multiple copies of Windows outputs was not working with some Windows printer devices.
