Easy-PC for Windows

Number One Systems

Version 12.0.6
Software Update Details
Release Date 22-Sep-2009

Problem Fixes in Version 12.0.6

This is the final 'roll-up' patch for Version 12. No further updates will be issued for this version.

Add Shape
- o Selecting 'Free Movement' before adding first point in 'Add Single Line' would cause the application to quit.

Duplicate
- o Copper was not being connected to the required nets after saying 'No' to merge nets prompt.

Edit Track
- o When editing tracks, sometimes the track would disappear on finish.
- o With the Preference 'Edit segment towards nearest end' selected, occasionally the application would quit on finishing a track.

Gerber
- o One specific design had copper covering a via on powerplane plots.

Goto
- o Goto was not finding all packages of a component, only the components using the first package.

Import
- o No connections were being shown after importing an Orcad netlist.

Integrity Check
- o If different versions of the same component were present on different sheets in a project, no warning was given and the connectivity could be incorrect.
- o Sometimes it would report that component versions were different when both designs had been fully updated.

Library Manager
- o The footprint preview on the Components page was empty until a package was selected.
- o On Windows Vista, the Browse button to add a new library folder was not working if no folders were currently defined.

N.C.Drill
- o The minimum routing size was defaulting to zero which was not an appropriate tool size.
- o The setup dialog was not checking that the minimum routing size was greater than zero.
- o Ordering of slots and normal drills in the Excellon file was causing some CAM software to miss any drills that appeared after the first slot.

ODB++
- o Some filled shapes, particularly shapes with cutouts, were not being correctly output.
- o The angled sides of octagonal items were too short, causing the shape to be slightly distorted.
- o Tracks with multiple widths could produce shorts in Composite Powerplane layers.
- o Some shallow arcs could produce invalid ODB++ data.
- o Underlines/barring was not always being rotated with the text if the output was rotated.

Paste
- o Pasting a copper shape with a track attached was moving the track end to the start point of the copper.

Plotting
- o Deleting plots with Plot Preview enabled could cause the application to quit.

Update Component
- o If the required package for a component could not be found in the library, this component was just skipped over without issuing a warning to the report.

Value Positions
- o When adding a value position, cancelling a prompt about the component being fixed could cause the application to quit.

Program Changes in Version 12.0.5

Design Rule Check
- o New checks have been added for unplated items:
  - o Unplated vias : this check will trap out any use of an unplated pad style for a Via.
  - o Inner tracks on unplated pads : this check looks for tracks on inner layers connected to unplated pads, because without plating there will be nothing to carry the signal out to the outer layers.

- Extra detail can now be included in the report by checking the ‘Detailed Report’ checkbox.

Problem Fixes in Version 12.0.5

BoardMaker import
- Import of BoardMaker designs with invalid component names could cause the application to ‘hang’ while it attempted to determine a valid alternative name.

Change layer
- If a track ended on a via that was somehow in the same place as another via, changing layer on that track segment to remove the via would cause the application to quit.

Design Rule Check
- One specific design was producing a copper-to-via error when there was no error.

DXF import
- One specific DXF file would cause the DXF Import option to loop indefinitely, due to a slightly malformed command in the file not being intercepted by the importer.

Eagle import
- Bus terminals were not being imported correctly from Eagle schematic designs.

Easy-PC DOS import
- Attempting to open most Easy-PC DOS design files was simply producing an error message.

Easy-Router
- In some circumstances, the router would insert more than one via on powerplane fanout/stubs.

Gerber plotting
- On one specific design, a complex poured copper shape was producing a short in the Gerber file.

Libraries dialog
- On the Components tab, the preview was displaying the ‘current’ package from the Component instead of the ‘default’ package.

OrCAD netlist import
- If no package is specified in the mapping file, or the specified package does not exist, Components could be added to the design using the ‘current’ package instead of the ‘default’ package.
- The error report included unnecessary warning messages about using the default package, making it difficult to spot genuine import errors.

ODB++ export
- Some cutouts in shapes were being output as slightly the wrong size.
- On one specific design, one cutout was missing from the output file.

PDF plotting
- Tracks were being included on Pads-only plots to the PDF output device.

Properties
- Gates of a multi-gate component could not be moved to another sheet in a project by editing the gate name in Properties.

Windows plotting
- Output of a negative plot to Windows devices was always filling pad/via holes regardless of the setting on the Plotting dialog.

Problem Fixes in Version 12.0.4

Add Component
- If a component referenced a missing symbol, no warning message was being given when attempting to add that component to a design.

Change Layer
- Changing the layer of a track segment in a particular track pattern containing a loop was causing the application to quit.

Component Names
- It was not possible to set up two value positions to display the component name and reference name as separate items on a component.

Gerber Import
- Some drill files were not being correctly aligned with the Gerber data during ‘intelligent’ gerber import, resulting in some pads not having their drill holes.

Grids
- Use of keyboard arrow keys to move cursor or items was causing intermittent loss of gridding when attempting to place subsequent items.

IDF Export
- Use of package name instead of footprint name in one of the pair of IDF files was preventing data from some designs being transferred to SolidWorks.

Libraries
- After enabling or disabling a library folder on the Libraries dialog, this change was not always being detected by Add Component.
- Creating a new library with the same name as an existing one did not remove the index file, possibly resulting in incorrect listing of the new library contents.
- Creating a library by importing the contents of a design file could produce a corrupted library file.

ODB++
- Using the same layer type for all the drop-down lists could cause the resulting ODB++ data to be incorrect, generating an 'illegal matrix' error in some CAM tools.

Pro-Router
- The router was allowing some tracks to be routed within a routing area marked as a keepout.

Text Callouts
- It was sometimes possible to leave text callouts in a state where further editing of the callout could cause the application to quit.

Problem Fixes in Version 12.0.3

Add Component
- Typing into the library name drop-down list to change the selected library did not update the list of components to match.
- No warning was given if the component that would actually be added was from 'earlier' in the library list than the one being previewed on the Add Component dialog.

Auto Rename Components
- In some cases, Auto Rename was giving unpredictable results not relating to the actual positions of components on the board.

Dimensions
- Editing dimensions (including moving the dimension text) after resizing the Working Area would cause the dimension to be stretched or shifted, possibly ending up outside the Working Area.

Display
- Pad Exceptions with 'drawn' shapes (e.g. Oval, Rectangle) were not being drawn unless their width was greater than the default pad width, regardless of their length.

DXF
- Importing a particular DXF file was causing the application to quit.

Fix/Unfix Item
- Selecting a component pad and doing Fix/Unfix had no effect.

Footprint Wizard
- Using any of the 'spin' controls when the adjacent edit-box was empty was causing the application to quit.

Forward Design Changes
- If any components referenced non-existent symbols, this could cause the application to quit when forwarding design changes, instead of simply writing an error to the report.

Import Netlist
- Importing a .PNT netlist could cause duplicate component names to be created.

Libraries
- The 'contents' of libraries were being re-read each time Add Component or other dialogs was used, which was impacting performance if many libraries were installed.
- The symbol "5P" in the standard Plug.ssl was the wrong symbol (not consistent with 1P, 2P, etc).
- A component required for the Evaluation Guide was missing from the standard Opamps library.
- The Find dialog was failing to find items if it had to generate an index file before looking for the required item.
- Some duplicate library items were not being shown 'disabled' in the Find dialog.

PCB Symbol Edit
- The various menu options for 'Add Copper' were not enabled when editing PCB Symbols.

PDF Output

- The line drawn for barring or underlining text was sometimes drawn in the wrong direction.
- Designs using TrueType fonts not installed on the current machine were causing PDF output to be abandoned with a ‘plot failure’ message.

Plotting

- Saving Plot Job files did not treat Wire layertype as a separate type, thus on re-reading the Job the corresponding layer was not being reselected for plotting.
- On the Position page of the dialog, the Plot From and Plot To controls were disabled for Windows output and enabled for other types, when it should have been the other way round.
- Slots were being output as round holes when generating Windows plots.

Properties

- Attempting to view the properties of two shapes on different layers could cause the application to quit.

Replace Component

- If <All Libraries> is selected, no components are shown in the list until the <All Libraries> entry is selected a second time.

Update Component

- Attempting to update a component after one of its Values had been deleted from the library Component was causing the application to quit.

Values

- The order in which Values were displayed on Components did not follow the order defined on the component in the library.

Problem Fixes in Version 12.0.2

Add Connection

- Using double-click to add a connection to a schematic pin with multiple pin names (e.g. 1,2,3,4) was also adding other pins on the component to the same net.

Bus Nets

- The list of nets on a bus as displayed on the ‘Choose Bus Net’ dialog would only ever contain a single net regardless of how many were defined on the bus.

Clear Copper

- Attempting to Clear Copper from some designs could cause the application to quit.

Design Rule Check

- The application could ‘hang’ when attempting to do a ‘Selected Items’ check on a component.
- Schematic DRC was not clearing errors from the screen as they were deleted.

Gerber Output

- Filling for one particular copper shape was incomplete in the Gerber file.

Layers

- If the layers had somehow been placed in the wrong order with [Bottom] not at the end of the list, layerspans for blind/buried via would no longer work correctly.

Library Manager

- Renaming a library item was moving it to the end of the ‘contents’ list instead of putting it in its proper place in the sort order.

Measure

- Repeated use of the Measure tool could cause the application to quit.

ODB++

- Output did not include dashed/dotted line styles.
- Very short bullet-shaped pads were not coming out correctly when undersized for solder paste.

PDF Output

- Some text items were being drawn at the wrong size and could clipped from the output.
- No margin was included on the PDF output.
- Some of the terminology used on the ‘popups’ in Schematic PDF files was incorrect.

Picking

- It was sometimes impossible to set Netnames in a Schematic design to be selectable.

Plotting & Printing

- Attempting to display the Preview for a drill output plot would cause the application to quit if no layer spans were selected to be included in the plot.

o Lack of a newline after the M15 command in Excellon routing output was causing the wrong results when the file was used on some manufacturing systems.

Pour Copper

o When pouring multiple areas, for example when using the Pour option from the main menu to pour multiple, this could cause individual settings for thermals and solid/hatched styles on different areas to all be set to the same values.

Toolbars

o The toolbar icon for 'Save to Library' was still enabled in PCB and Schematic designs, causing the application to quit if this button was used.

Ulticap import

o A particular Ulticap schematic design file would not import.

Problem Fixes in Version 12.0.1

Change Net

o The Change Net dialog was confusingly titled 'Ungroup'.

Component Edit

o Using 'Apply to Other Gates' in a multi-gate component where the last gate had fewer pins than the others could cause the application to quit.

Cross Probe

o Cross-probing in either direction between Schematic and PCB when 'target' design was not already open could cause the application to quit.

Gerber Output

o Thermal relief shapes on powerplanes were incorrect for pad shapes other than round.

Integrity Check

o After adding Connections to the schematic, the corresponding Pins were not being connected in the PCB on the first attempt.

Pour Copper

o The ability to Pour a 'free' copper shape was not enabled.

Reverse Engineer

o Attempting to Reverse Engineer a PCB design with many components was sometimes placing some of those components outside the valid design working area.

Save

o When saving a design file which was from a version of Easy-PC prior to V12, the warning about over-writing old format files was not being displayed.
