

## Version 16 Software Update Details

---

### Problem Fixes in Version 16.0.9 (18-Sep-2013)

#### 3D View

- Editing a package in a library containing a user-defined package that uses a Prism would cause that Prism element to become corrupted.

#### BoardMaker Import

- Some tracks were being imported as shapes as their start and/or end pads were not being discovered.

#### Design Rule Check

- Component heights were not being retrieved from the Component for DRC and other uses if the value was only defined or over-ridden on a component instance.

#### Library Manager

- If a broken library item was encountered while generating a library index, or a library file was not actually a valid Easy-PC library, this could cause the application to quit.

#### Name Pins by Row/Col

- On some components with names or numbers already assigned to some or all of their pins, the Name Pins by Row/Col command would not change any pin numbering.

#### PDF Output

- Some drill holes were being covered by tracks despite the Fill Holes options being turned off in the Plotting dialog.

#### Pour Copper

- Component copper inside components mirrored to the back of the board was treated as obstacles to copper being poured on the top side.

#### Reports

- The Component Position Report was not always using the Placement Origin defined in the PCB Symbol when filling in the Place X or Place Y fields in the report.

#### Shortcuts

- Loading a saved set of Shortcuts from a file had no effect.
- 

### Problem Fixes in Version 16.0.8 (17-Apr-2013)

#### 3D View

- Height check areas on the top side of the board were always being drawn 0.5mm above their proper position.

#### Auto Rename

- The report produced when renaming components or nets in a project was not properly formatted.

#### Design Technology

- Using 'Delete Unused' would sometimes require more than one button click to remove all the unused entries, and could then cause the application to quit on closing the dialog.
- The entry for setting 'Panel To Board' spacing was only available in a Panel design, which meant that a sensible default value could not be defined in a PCB Technology file.

#### Drill Tables

- Editing some entries in a drill table (Settings, Drill Ident) could cause the application to quit.

#### Library Tools

- Using the Move button the Library dialog to move items to another library would not stop if it discovered the target library to be read-only, and would still remove the items from the source library causing them to be lost.
- The asterisk used to mark the default package on the Components page of the Library dialog was not always output next to the correct package item.

#### Reports

- Reports generated from panel designs could contain the wrong coordinates if the inserted PCB designs used system coordinate origins. The units used in the report could also be incorrect.
- Using 'Include' to include only items matching by name (e.g. producing a report listing nets that match a given net class name) did not match any items, leading to an empty report.
- Coordinates output to the text version of some reports did not take the coordinate origin into account.
- Grouping by a field that is not present as a column in the actual report was not correctly handled, sometimes causing the collation of values to fail.

## Working Area

- o Using 'Best Fit' did not shift any inserted bitmaps, causing them to be offset to from their proper position.

---

## Problem Fixes in Version 16.0.7 (08-Mar-2013)

### Auto-Router

- o Some designs had unwanted tracks added to power-plane layers.

### Design Rule Check

- o Component-to-Component checking did not correctly handle nested components.

### Goto

- o When using Goto to find named busses, the Next button was not being enabled to allow you to find the next bus of the same name.
- o If an existing unnamed bus was given a name in Properties, the Goto bar did not update to show this new name.

### N. C. Drill

- o Drill data could be generated more than once in the DRL file if the output option to 'Expand Step-and-repeat information' was checked.
- o Shapes defined within PCB symbols were not being considered for inclusion in Excellon milling output.

### Open

- o Opening some schematic design files produced in earlier versions could cause the application to quit.

### Pour Copper

- o In two specific designs, copper was being poured over items on other nets.

### Projects

- o If a named bus was pasted into a Schematic sheet in a project, this did not update the project-wide list of bus names.

### Pro-Router

- o Tracks added by the auto-router to existing copper shapes were not actually being connected to the copper, causing them to be drawn and flagged by DRC as unfinished tracks.

### Smooth Tracks

- o The option to 'Pull Tight' all tracks was sometimes leaving dangling 'stubs' where two or more tracks were brought together at a junction.

### Testpoint Output

- o The IPC-356 test land output did not include some information in the header section that is required by some third-party software packages.

---

## Program Changes in Version 16.0.6 (7-Jan-2013)

### Import Pin Data

The Import Pin Data command in Component Edit has been extended to allow schematic terminal numbers and/or gate names to be specified. This enables CSV files to be used to set up pin mappings for new components that have no pin names defined. Previously the import would only help with updating components that were already set up.

To specify terminal numbers in the CSV file, include a field with 'terminal number' as the column heading. Similarly, specify 'gate name' as the column heading if you need to enter the gate name (a, b, etc.) for multi-gate parts.

For example, here are the first few lines of a CSV file for a 144-pin STM32F103:

```
Terminal Number,Pin Name,Pad Number,Terminal Name
1,34,34,PA0
2,35,35,PA1
3,36,36,PA2
4,37,37,PA3
5,40,40,PA4
6,41,41,PA5
7,42,42,PA6
8,43,43,PA7
9,100,100,PA8
10,101,101,PA9
11,102,102,PA10
12,103,103,PA11
13,104,104,PA12
14,105,105,PA13
15,109,109,PA14
16,110,110,PA15
17,46,46,PB0
```

---

## Problem Fixes in Version 16.0.6 (7-Jan-2013)

### 3D View

- The 3D View windows did not take into account the Preferences setting for 'reverse mouse zoom'.

### Change Layer

- In a PCB design with no bottom electrical layer defined, changing the layer of a track segment could cause the application to quit as it attempted to find a 'best fit' layer span for vias.

### DXF

- One particular DXF file would not place imported arcs in the correct location.
- If the current default line style was an unnamed style, shapes in the DXF could not be imported as Board Outline but would instead always revert to being Shapes.

### Import Eagle

- Power and ground plane layers were not being transferred from Eagle PCB designs.
- Eagle files could not be imported on some Windows 7 or Windows 8 systems.

### Import Gerber

- During import of shapes as tracks, any nets created were not being assigned to a net class.

### Integrity Check

- Sometimes an existing component in the PCB would be renamed and replaced instead of simply adding a new component.

### N.C. Drill

- The milling commands for some slotted pads were not taking the size of the milling tool into account, and were thus causing the milled slot to be slightly over-sized.

### Net Names

- Predefined net names from the component were not using value positions defined in the Schematic symbol when choosing the position for those net names when the component was added to the Schematic design.

### ODB++

- Board cutouts were not being exported to the ODB++ file.
- In some circumstances, a copper cutout in a poured copper area could be omitted from the output file.

### Open

- Some designs from earlier versions of the application would not read into version 16.

### Output PDF

- A design imported from Easy-PC ProXM would not plot to PDF without causing the application to quit.

### Save All

- Invoking the Save All command from the PCB screen in a dual-screen configuration would cause the application to go into an infinite loop.

---

## Problem Fixes in Version 16.0.5 (16-Oct-2012)

### 3D Libraries

- Adding an unnamed 'prism' element to a user-defined 3D package was causing the library file to become unreadable.

### Change Net

- After allowing Change Net to merge nets across the whole project, Integrity Check would fail with an error 'Cannot find net ""'.

### Component Edit

- Attempting to map multiple pins in a multi-gate component was producing incorrect results.

### Delete

- After deleting a block port, the application could unexpectedly quit when attempting to save the block file.

### Dimensions

- Dimensions created on the bottom side of the board could incorrectly alter the position of their text.

### Drawing

- When adding or moving free pads in a PCB design containing vias, some or all of the other free pads would disappear.

#### Help

- Several options were not linked to the correct help page when pressing F1 for context-sensitive help.

#### Import BoardMaker

- Some designs were imported without creating sufficient Net Class information, which could then cause the application to quit when running Design Rule Check.

#### Import Easy-PC

- Easy-PC DOS design files would not import.

#### Import OrCAD Netlist

- Importing an OrCAD netlist was creating nets with no net class.

#### Layers

- Layer data could get mixed up after using the Up or Down buttons to re-order layers.

#### Move

- Moving some shape segments adjacent to arcs could cause the arc to reverse direction.

#### Nudge

- The application would quit after attempting (and failing) to nudge a fixed component or un-routed connection.

#### Open

- The application was failing to open a PCB design from an earlier version of Easy-PC when the design contained unprotected Vias with no tracks attached.

#### Panel Wizard

- Attempting to start the Panel Wizard without using a Technology file could cause the application to quit.

#### Plotting

- When plotting multiple layers to the same Windows plot, the output was not in layer order.
- Implied (automatic) junction points on Schematic diagrams were usually printing as hollow instead of filled.

#### Reports

- The text file version of some HTML-based reports included elements of HTML code.

#### Shape Bar

- Attempting to change the direction of an arc using the Shape Bar could cause the application to quit.

---

## Problem Fixes in Version 16.0.4 (05-Sep-2012)

#### Auto Weld

- Attempting to move a selected item after auto-welding some 'free standing' connection stubs in Schematic could cause the application to quit.

#### Component Values

- If the entries in the Component Values dialog were sorted by clicking on column headers, any subsequent edits to entries in this dialog could have been applied to the wrong components.

#### Defaults

- Attempting to view the Text page of the Defaults dialog could either produce an error message or cause the application to quit.

#### DXF

- Dotted line styles were being output to DXF as solid lines.

#### IDF Output

- When retrieving height information to output to the IDF file, the height value from the Component was being used even if this was over-ridden on the instance of that component in the design.

#### Mitre Corner

- The Name item for a bus could be left 'dynamic' (drawn in XOR mode) after using Mitre Corner on a bus item.

#### N.C. Drill

- Generating Excellon data with the option to 'Remove Duplicate Holes' enabled was causing the application to quit.

#### Online DRC

- The prompt that 'this may take some time' has been suppressed when placing PCB Design items in a Panel design.
- Several interactive routing operations were reporting errors when crossing the edges of any type of Area.

#### Open

- Free Text items in some schematic designs from earlier versions were being drawn as Mirrored and/or incorrectly positioned.

#### Rename Component

- The radio buttons at the top of the Rename Component dialog were not working, meaning that the only available option was to rename 'All Components'.

#### Reports

- When editing a custom report, the setting 'Ignore Parts List Suppression' was not being saved to the report file.
- When displaying reports as HTML in the Report Browser, the plain text version of some report files could contain some fragments of HTML mark-up.

#### Shape Bar

- The selected segments were not being drawn in bold to indicate the selected item as they used to appear in V15.
- Editing an existing shape with the Shape Bar displayed could cause the application to quit.

#### Technology Files

- Saving a technology file from a design containing Notes could cause those Notes to display, or the design extents to be incorrectly calculated, in symbol previews during Component Edit.

#### Trunk Routing

- Attempting to start trunk routing when net classes had no via and/or track style defined was causing the application to quit.

---

### Problem Fixes in Version 16.0.3 (14-Aug-2012)

#### 3D View

- After editing 3D packages changes were sometimes not being saved.
- You can now choose between using three or four bands to represent the value for an Axial 3D package.
- Was not always showing a preview, or showing Axial bands.
- Going to "User" mode on an existing 3D package did not always take the position of the symbol origin into account, so the body parts could be a long way from the preview shown.
- "Capped" end of a Pipe did not work.

#### Design Rule Check

- Reported incorrect errors due to ignoring the net match spacings.

#### DXF Output

- Did not handle duplicate block names.

#### File New

- Selecting the Technology tab on the File New dialog caused the program to exit.

#### Properties

- Adding a pad exception to a component pad could cause the program to exit..

#### Select

- Using Frame Select with the Control key pressed down could sometimes cause the program to exit.

#### Setup

- The product key code was not being recorded into the correct system registry, causing the application to only operate in 'demo' mode until the product code was re-entered.

---

### Problem Fixes in Version 16.0.2 (02-Aug-2012)

#### Design

- The settings for Maximising copper spokes in Pour Copper and for positioning a Bus Name were not being saved to the design file. Note that the change for this means that design files saved from V16.0.2 will not be readable with V16.0.1 or earlier.

#### Design Rule Check

- Error markers which are now added to the design to indicate Components in the Bin were not being added to the DRC Report.

#### Pour Copper

- A minimum number of two thermal spokes was being enforced during Pour, rather than allowing a single spoke to be enough as in previous releases.

#### Reports

- Any report generated as CSV was being displayed in the Report Browser instead of being handed off to the appropriate external application to be viewed.

---

### Problem Fixes in Version 16.0.1 (01-Aug-2012)

### 3D View

- o Creating a new 3D package using the new 'User' package style when an existing package was currently selected in the list box on the Library dialog was over-writing that selected package with the newly created one.

### Add Bus

- o It was possible to create a closed bus as two separate bus items with different nets contained in each item.

### Defaults

- o The Text page on the Defaults dialog could not be accessed without displaying an 'unsupported operation' error.

### Design Technology

- o Right-click on the Nets page (or on the 'grid' in some other dialogs too) was sometimes failing to display the popup menu.

### Drawing

- o Attempting to draw a tiny arc with radius less than half the line style thickness could cause the application to hang.

### Report Browser

- o Hyperlinks from some DRC error report items did not select the corresponding item in the design.

### Setup

- o On some Windows 7 systems the product key code was not being recorded into the system registry, causing the application to only operate in 'demo' mode until the product code was re-entered.

### TraceRouter

- o Some complex shapes were blocking off more of the routable space than necessary, in some cases preventing designs from being fully routed.