

Pulsonix Change Notes

These notes list the main functional changes and problem fixes in each release of the software. They are listed in order, latest first.

Version 4.5 Build 2804 : 14-Aug-2007

Functional changes

Protel Import

New option to add "No Net" tracks as copper.

Problems Fixed

CAM/Plot

LPKF Output - Used wrong names for inner electrical layer plots.

Windows Plot - Bitmap was not correctly printed to a PDF file.

Windows Plot - Rotated bitmaps were sometimes plotted in the wrong position.

Export

ODB++ - Poured copper was sometimes missing from the output.

Import

Cadstar - Was not importing spice values from the parts *SPI line.

Eagle - Did not handle PCB tracks with arcs correctly.

Eagle - Some schematic signal references were imported as block ports.

Integra - Schematic only parts did not import correctly.

Integra - A particular file failed to import.

Orcad - Particular schematics design using multiple instance hierarchy failed to import.

Interaction

Delete - Deleting a bitmap when more of the same bitmap exist could cause problems.

Properties

Templates - Could not remove a net class override switch from a template.

Settings

Folders - Report dialog was not correct when the folder did not exist.

Technology

Grids - Polar grids were artificially limited to 500 steps.

Layers - Layers were not being marked as used when pad style exceptions existed.

Tools

Copper Pour - Failed to pour a specific template correctly.

Copper Pour - Poured over some pads it should have avoided.

Copper Pour - Failed to pour correctly for a particular design.

Copper Pour - Did not pour a specific template on the bottom layer.

Design Rule Check - Area Height checks sometimes caused the program to quit.

Design Rule Check - Performing check within just the window did not report Component to Board errors.

Electrical Rules Check - Did not always handle nets connected to blocks correctly, sometimes gave split net errors.

Reload From Library - Pin networks sometimes became unconnected.

Router - Pad style exceptions could cause the router not to run.

Synchronise Designs - Testpoint pads sometimes lost their testpoint flag after synchronise.

Teardrops - Did not add teardrops on some vias if online DRC was enabled.

View

3D View - Area cutouts that overlapped the board outline were ignored.

Version 4.5 Build 2803 : 30-May-2007

Functional changes

None.

Problems Fixed

CAM/Plot

Gerber - Barcode bars were too close together.

Gerber - Program stopped whilst outputting bottom layer with hardware fill on.

Gerber - Break Silkscreen removed the 'orientation' mark from the plot.

Gerber - Was plotting copper shapes with dashed lines.

Export

Gencad - Was outputting spaces in padstack names.

IDF - Got incorrect results importing resultant file into CATIA.

ODB++ - Layers were not output correctly.

ODB++ - Program sometimes quit when using the "Use CAM Plot Settings" option.

Pads - Failed to output if the design contained connected mounting holes.

Import

Gerber - Import was intolerant of spaces in the aperture definition.

Gerber - Some circles failed to import.

Integra - Did not import the pin and gate swap information from a particular library.

Integra - Did not handle unnamed pad styles correctly

Integra - Some parts ended up with -1 as a pin count.

Integra - Self intersecting templates were not rejected, which caused problems in copper pour.

Integra - Did not deal with TopCad files that had items with no layer name.

Integra - Did not understand an alternative method of defining board cutouts.

Intelligent Gerber - Some flashed items were missing after import of specific data.

Intelligent Gerber - Was getting the component pin counts wrong.

Intelligent Gerber - Lines sometimes did not end up in their correct place.

Orcad - Failed to import duplicate multiple gate part.

Orcad - Program quit after importing specific EDIF file.

Protel - Program quit reading a record that spanned multiple lines.

Protel - Did not handle with unnamed pads.

Interaction

Add Component - Blind vias in a buried footprint became through-hole.

Add Via - Could cause the program to exit when adding a via whose span did not reach the picked item.

Delete - Design got problems after deleting a block instance which had nets that were wrongly marked as local.

Delete - Performing a frame select followed by delete could cause the program to exit later.

Generate Component - Components were being generated on the wrong side.

Generate Component - There were various problems generating components.

Generate Component - Components could end up the wrong way round.

Generate Component - Mirrored non-symmetrical pads were not rotated correctly and symmetrical components did not always get the correct orientation.

Generate Component - Sometimes placed pads on the wrong side for mirrored components.

Generate Component - Parts were being generated with the number of pins set to -1.

Move - Was sometimes really slow dragging components from the bin.

Library Manager

Footprints - Footprint library appeared to be corrupt.

Tools

Apply Layout Pattern - Failed to correctly apply a pattern in a specific design.

Copper Pour - Failed to pour template in a particular design.

Copper Pour - Poured too close to adjacent copper at one point, giving a copper-to-copper error.

Copper Pour - Did not always pour correctly around items which have a large net class match spacing.

Copper Pour - Failed to pour a template in a specific design.

Copper Pour - A ground template was pouring over items in a default net.

Design Rule Check - Did not handle checking a design with "Layer Name" attribute positions that had no layer.

Electrical Rules Check - Reported incorrect errors for star points and busses connected to block ports.

Reload From Library - Replacing multiple pin schematics star points caused problems when saving the design.

Reload From Library - Incorrectly got essential layer error message when there was copper on the top and bottom layers in the footprint.

Synchronise Designs - Synchronise sometimes introduced unnecessary vias.

Synchronise Designs - Was incorrectly reporting designs different because of group difference.

Unroute Nets - Using "Unroute Track Segments" could cause the program to exit.

View

3D View - Used to extract the height attribute only from the component, not from the part.

Version 4.5 Build 2802 : 27-Mar-2007

Functional changes

None.

Problems Fixed

CAM/Plot

Plot Positioning - If plot position was set to XY, values entered for XY changed after save and load of the file.

Export

DXF - Program quit trying to output DXF for a specific design.

IDF - Did not use package heights if height was not specified elsewhere.

IDF - Only used precision of 1 regardless of design units, which could lead to rounding errors.

IDF - Component was in the wrong position when imported into Solidworks.

IDF - Mounting holes were not visible in Solidworks.

IDF - Did not strip extra lines from multi-line attributes.

Pads - Program quit whilst saving as PADS ASCII file.

Import

Eagle - Used to output layers that were not used.

Edwin - Attempting to import a particular Edwin file caused the program to exit.

Easy-PC - Some tracks were on the wrong layer after importing an Easy-PC PCB design.

Integra - Did not deal with board cutouts which were defined as shapes on "BoardCutout" layers.

Integra - Failed to import some Integra TXF files.

Orcad - Sometimes schematic circles were translated as zero size.

Protel - Failed to import a specific Protel PCB file.

Protel - Did not deal with 16 signal layer older Protel designs.

Ultiboard - Got incorrect pin mappings in some parts after importing to a library.

Visula - Some polygon shapes were not being closed.

Visula - Tracks with null ends were being lost.

Visula - Non poured templates ended up with no layer.

Visula - Did not translate Visula attribute positions onto a sensible layer.

Visula - Did not set the relative origin correctly in CADIF import.

Interaction

Add Callout - Repeatedly cancelling the add text callout dialog box caused the program to exit.

Add Callout - Program could exit if adding a callout with straight open arrow style.

Add Component - Using "Insert Multiple" in a schematic design only added one component.

Add Track - Pushing tracks produced spurious track segments.

Cut/Copy/Paste - Program quit when pasting a component with essential layer classes into a four layer board.

Cut/Copy/Paste - Added the wrong layers when you pasted a mirrored component with wires.

Security Copy - Was overwriting the previous good copy when output was not successful.

Library Manager

Footprint Wizard - Type BGA sometimes added through-hole pads.

Properties

Inserted Reports - Program entered an endless loop if you edited two inserted user reports at once.

Simulation

Setup - In demo mode the "Associate Models to Parts" button did not work.

Tools

Copper Pour - Poured copper violated design rules to pads and tracks nearby.

Copper Pour - A specific template failed to pour.

Copper Pour - Pouring a particular ground template shorted across other pads and tracks.

Copper Pour - Failed to pour most of the bottom layer for a specific design.

Design Rule Check - Did not use the 3D Package height, it only used the Height attribute.

Optimise Nets - Incorrectly calculated that specific pads were touching.

Router - A particular design got the message "Error setting up router".

Router - Nets with backslashes in names were ignored by the router.

Router - Did not deal with component names with a ")" character in them.

Router - Did not handle cutouts in templates correctly.

Synchronise Designs - A specific design was corrupted after running Synchronise.

View

3D View - Program appeared to hang after attempting to view 3D for a particular design.

3D View - Did not show silkscreen for unfitted components when the silkscreen layer class had the "Comp. Items visible In All Variants" option set.

3D View - Unfitted comps on the back of the board appeared on the 3d view.

Version 4.5 Build 2801 : 09-Feb-2007

Functional changes

Footprint Wizard

The Footprint Wizard has been improved to make it work more in-line with the way in which footprint data is presented in the manufacturers datasheets.

New footprint types have also been added.

The footprint preview window has moved to the right side of the wizard and it is now bigger to make the more detailed footprints clearer to see.

Symbol Wizard

New symbol types have been added and the preview window has been improved.

Part Editor

New "Name Pins by Row/Col" feature allows pins to be renamed based on their row and column positions in the footprint. This is useful for grid based components such as Ball Grid Arrays.

Problems Fixed

CAM/Plot

LPKF - Could not get a specific layer output.

Print – Bitmaps on a rotated drawing were not being printed in the right position on the page.

Windows Plot - Bottom-side Truetype text was not searchable in a PDF output.

Export

Pads - Exporting Pads designs was not working.

UNIDAT - Did not handle star points correctly.

Hierarchy

Update Block - In a specific design, updating a multi-instance block changed a net name to a component name.

Import

DXF - Could not move some imported shapes.

DXF - Data was offset such that shapes from different files did not line up.

Eagle - Imported PCB design has pads on the wrong layer and loses some components.

Edwin - Reading a project containing a track ring caused the program to exit.

Edwin - Some components ended up with incorrect and missing pins.

Gerber - Was losing some shapes on text items.

Gerber - One shape in a file was broken after import.

Integra - Imported TXF data to a parts library did not produce the correct part names.

Intelligent Gerber - Importing a ground plot produced incorrect shapes.

Intelligent Gerber - Program hung attempting to load a particular set of files.

Intelligent Gerber - Removed top side pads that had vias on them.

Intelligent Gerber - Did not create user-defined pads.

Pads - A schematic design that imported in 4.0 did not read in.

PCAD - Did not import the whole of a specific PDIF design.

Protel - Using a mapping file with a specific Target netlist did not produce any connections.

Install / Licensing

Network Licensing - Users could not be prevented from seeing some licenses using the Permissions dialog in the Network License Manager.

Interaction

Auto-Weld - Unfinished schematic connections disappeared when moving ends over pins.

Cut/Copy/Paste - Pasting a copy of a particular design lost connectivity on two pads.

Find - When finding a via, switching on the layer visibility could cause the program to exit.

Move - With "Push Tracks" enabled an incorrect message was given when trying to move a via surrounded by copper.

Move - With "Push Tracks" and "Push Vias" enabled moving a via could move other vias into invalid positions.

Split Net - Did not tidy up unwanted junction points.

Split Net - Dropping a 2 pin component onto an existing connection renamed other separate parts of the net.

Library Manager

Copy - Copying or moving a footprint to another library did not prompt to overwrite existing item.

Performance - Access to libraries on a network device was much slower than in 4.0.

Generate Index - Failed for a particular parts library.

Part Editor

Insert Attribute – attempting to Insert Attribute into a newly-created (unsaved) part with the rows and columns “reversed” would cause the program to exit.

Reports

Report Maker - Performing "If Attribute Exists" on a library item caused the program to exit.

Report Maker - Did not use design units, like it did in version 4.0.

Settings

General Options - Could not change the default units back to imperial after changing to metric.

Technology

Nets - There was sometimes a problem closing the technology dialog after renaming nets.

Colours - The program sometimes quit whilst closing a design after changing the colour of a net.

Colours - Loading PCB colour files when editing a footprint could make attributes invisible.

Layers - Check boxes on the layers bar overlapped on a particular machine configuration.

Tools

Copper Pour - Failed to pour most of a template on the bottom layer of a design.

Copper Pour - A Specific pour produced a design corruption.

Reload From Library - Sometimes incorrectly said that a symbol was in error due to a missing essential layer.

Reload From Library - Put some pads the wrong way round when they had offset holes.

Router - No routing and no report produced for a particular design.

Synchronise Designs - Program exited performing synchronise after adding net class rules to the schematic.

Synchronise Designs - Failed to rename testpoints.

View

3D View - Was slow generating the initial picture.

3D View - Closing the application when a 3D view was open onto an unsaved design could cause an “unrecoverable application error” as the program closed down.

3D View - The picture displayed did not leave out “not fitted” components when using Variants.