

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 3.1 Build 2273 : 18-Jul-2005

Functional changes

None.

Problems Fixed

CAM/Plot

Gerber - output file was missing some cutouts. Program hung generating Silkscreen secondary plot. Hung during output of design's ground layer.

Powerplane Plot - was not adding thermal pads to star points split over two templates. Was backing off tracks on powerplane layers.

Display

View Powerplane - was not working for non-isolated pads. Some non isolated GND vias were being connected to all plane layers.

Export

Cadstar - attribute owners were not correctly output.

ODB++ - had problems with multi-line net attributes and certain pad style names. Could not output positive power-planes. User defined pad styles were coming out wrong at certain angles. Steps\profile should only hold board outline not any cut-outs.

Import

DXF - was ignoring embedded font instructions in text strings when importing.

Eagle - surface mount pads were imported at wrong rotation.

Easy-PC - problems importing some V8.0 PCB designs. Did not read a particular schematic. Had problems reading in a documentation symbol library.

Pads - Pin Names had wrong alignment in V4 Pads schematic import. Problems reading specific PADS PCB V5 design. Would not read in a design with mixed net names on busses.

Protel - had problems reading a DXP ascii file.

Ultiboard - created a part with no name, which caused the program to exit.

Interaction

Change Block Symbol - new name was not stored on block instance after change symbol.

Copy/Paste - was not duplicating starpoint connectivity correctly. Copying components with symbol alternatives could cause the program to exit.

Hierarchy - a schematic had two components with same id number. Premature exit whilst loading block symbol.

Insert Connector - was not incrementing the pin numbers and was using PL1.1 for all pins.

Library Manager

Part Editor - some parts were being saved to more than one group.

Properties and Technology

Properties - program exited when entering properties on two connector pins from different parts. Could not change a block symbol to one with less pins.

Pad Styles - allowed deletion of pad styles being used by symbols where the symbol instance used an alternate pad style.

Tools

Copper Pour - would not pour some specific templates. Did not handle dangling tracks. Failed to pour correctly around doc symbol copper on the bottom layer.

Router - was ignoring some net class track layer restrictions. Route selected option sometimes hung the program when using network licensing. Route Selected altered routing in other nets.

Version 3.1 Build 2272 : 14-Apr-2005

Functional changes

None.

Problems Fixed

CAM/Plot

Schematics - using the preview window caused the program to exit for a particular design.

Import

DXF - some arcs on closed shapes were imported as straight lines. Did not handle blank lines correctly.

Protel - would not read in protel schematics hierarchy files if only 1 font was used. Hierarchy block symbols were wrong if their terminals were on the top edge. Did not correctly deal with case sensitive part names.

Interaction

Grid - using "Grid Step" option on older Footprint design files could cause the program to exit.

Move - shifting a particular framed design area caused the program to fail.

Library Manager

Part Edit - additional carriage return put in by another product caused the part index creation to fail. Did not save the "Part Family" value with all the parts in a group.

Technology

Styles - deleting track styles in a specific design caused the design to become corrupt.

Tools

Copper Pour - poured slightly outside the board outline all the way round.

Electrical Rules Check - was producing spurious ERC errors for a particular design.

Load Technology - could corrupt designs if two layer classes of type "Electrical" were involved.

Reload From Library - did not match styles according to the value in Design Settings.

Router - Mounting Holes were not set up as obstacles if a component had pad exceptions. Routed over zero-sized mounting holes that had drill holes.

Version 3.1 Build 2271 : 08-Mar-2005

Problems Fixed

CAM/Plot

PCB - Doc Symbols were not being included on plots.

Version 3.1 Build 2270 : 04-Mar-2005

Functional changes

Import

Now reads in PCAD 2004 PCB designs, Schematic designs and Libraries.

Problems Fixed

CAM/Plot

DXF Output - Pads were sometimes being output on the wrong layers.

Gerber - Program failed processing a dangling track. User defined pad shapes on specific layers were always output as unfilled.

ODB++ - Program failed when trying to output a design with starpoints. Did not handle user defined pads and multi-line attribute values correctly.

Windows Plot - When plotting a schematic block page the outline was included on the top level sheets.

Display

Changing the style values of a blind via made the drill hole disappear.

Import

Easy-PC - Reading schematic produced incorrect connection paths. Would not read some Easy-PC designs. Pad Style Exceptions were coming out 90 degrees out.

Pcad - PDIF files with part names containing spaces did not read in.

Library Manager

Parts - Library index file did not list some parts correctly as connector parts.

Tools

Align - Could cause the program to exit if aligning a framed selection containing tracks and components. Aligning a framed selection containing a dimension could cause the program to exit.

Copper Pour - Did not pour correctly in areas enclosed by copper.

Design Rule Check - Program failed when checking a starpoint.

Reload Block - Added extra plots to the schematic.

Router - There was a problem setting up the router for a single pin net breakout with a via. Spaces in component names prevented the router from starting. Reported 0% completion due to breakout vias not being output correctly.

Version 3.1 Build 2269 : 24-Dec-2004

Functional changes

None.

Problems Fixed

CAM/Plot

Gerber - Plot of filled shapes sometimes goes wrong. Was sometimes losing tracks when track did not go to pad centre. Some silkscreen outlines were not output.

Pen Plot - Did not take pen width into account for drawing text.

Display

View Powerplane - Was not correct with more than one board outline in the design.

Import

DXF - Would not import a particular dxf file which contained a closed shape.

Eagle - PCB Spacings in Eagle Schematic net classes could cause program failure. Mounting Holes were not being dealt with. Did not convert circular keepouts. Intermediate file generation did not deal with the % character in strings.

Orcad - Did not deal with odd name for a gate in a heterogenous part. Orcad - Certain way of labelling and naming parts caused the import to fail.

Pads - Importing design resulted in corrupt Layers which gave a "error setting up router" message.

Pcad - Import PDIF hanged if using a technology file.

Protel - Footprints were being translated with alphanumeric pad names.

Ulticap - Some pin logic labels were missing off some parts .

View Logic - Did not deal with continuation lines in the package file.

Install / Licensing

Network Licensing - Was not reliable with some users system configurations.

Interaction

Mounting Holes - Converting a pad to a mounting hole did not remove the pad from the net.

Board - There was a problem assigning some line styles, such that the board had no style and could not have one assigned to it.

Change Pin Number - created duplicate pin number in a displayed footprint.

Libraries

trans.pal - Some pinouts, footprints and schematic symbols were wrong.

microchip.pal - 28 pin PIC's used the wrong footprint and PIC16C73B had the wrong pinouts.

footprints.pfl - Some connectors had lost their mounting holes. 3M & 2 row Molex connectors had incorrect footprint sequences.

Library Manager

Part Editor - Incrementing pin names in selected rows was not working correctly.

Index Files - A problem in index files could end up with illegal names in the library editor dialog.

Index Files - Sometimes did not regenerating library index correctly, leaving the number of pins wrong.

Tools

Design Rule Check - Caused program to fail after doing backoff/padland checks on Starpoints.

Reload From Library - Allowed reloading a PCB only part into a schematic design.

Router - There was a problem restoring routes if a net had tracks but no nodes. Problem existed between implied copper and physical copper using the same signal. Designs with SMD footprints containing pad exceptions could cause problems for the router. Vias were being tidied out when they should not be.

Version 3.1 Build 2268 : 12-Oct-2004

Functional changes

Variants

You can now show Not-Fitted Components in the Cam/Plot output using the new 'Plot' checkbox in the Variant Manager dialog.

Problems Fixed

CAM/Plot

Gerber - Program failed trying to generate Gerber plots from a particular design.

LPKF - Program failed when trying to open the LPKF dialog for a specific design.

Import

Eagle - Oval pads from Eagle V4.11 were not correct.

Orcad - Zero sized circles in Orcad PCB designs were importing as huge circles. The hierarchy was not connected up properly after import of a Orcad design.

Protel - Elliptical arcs were not handled very well.

Interaction

Star Points - Editing a connection into a track from a bottom-side Star Point put the track on the top side.

Add Track - You could use 'Complete' to add a PCB track to a pad on a component in bin.

Move To Bin - Many problems after moving block instances or block terminals to the component bin. This has been disabled.

Library Manager

Import - Importing a footprint into the library lost breakouts on the bottom layer.

Tools

Copper Pour - Pouring a particular template on the bottom layer covered a pad on a different net. Pouring a particular copper shape caused the program to exit.

Router - Failed attempting to route a specific design containing dangling tracks. Was not dealing correctly with surface mount free pads. 'Route Selected Net' option could cause track crossovers. Did not route Power and Ground correctly if there was poured copper using the same signal as the planes. Sometimes ripping up good tracks during the clean phase.

Synchronise Designs - Was not matching internally connected pcb pads properly.

Version 3.1 Build 2267 : 13-Aug-2004

Functional changes

None.

Problems Fixed

CAM/Plot

Gerber – Gerber output would get stuck on one particular shape.

ODB++ - Not outputting teardrops properly.

Import

Data Transfer Wizard - Using this sometimes did not import all the parts.

DXF - Did not translate open DXF shapes into pads correctly.

Easy-PC - Did not deal properly with component names that had no number. Translation of visibility of part names and value positions was incorrect. Did not support Easy-PC Version 8.

Orcad - Another change to the Orcad V10 edif format.

Pads - Was not dealing with connectors properly in PERFORM format.

Pcad - Failed to import particular Pdif file.

Protel - Importing non through-hole vias caused the program to exit.

Ultiboard - Failed to read a specific design file.

Interaction

Place - Needed to be able to retain the "free angle adjoining segs" state for next time.

Duplicate - Using Duplicate and Autoweld to copy breakouts in footprints put pads onto the same net.

Library Manager

Part Wizard - Sometimes locked up opening the part wizard from library manager dialog.

Properties

Design Properties - When you deleted an associated part, the part was not completely removed from the design.

Tools

Copper Pour - Problem pouring template surrounded by obstacles

Reload From Library - Did not allow documentation symbols to be reloaded as signal references. Associated part was losing some attributes.

Router - Was giving error message during routing the selection. Was not dealing with double quotes in names. Did not handle split planes on a layer with no net name assigned. Got spurious board-track/pad errors. After stub routing and stubs being locked the optimise phase was removing vias. Locked up restoring routing results to design.

Version 3.1 Build 2266 : 26-May-2004

Functional changes

Report Maker

Can now report the name and path of the library from which each part in a design was added from. This is new information in the design, and so existing designs will have to have their parts reloaded from their libraries for the report to work.

The Report Maker can now report each page that a net is on in a schematic design. Use "List Of Nets" command followed by "List Of Pages" command.

New PCB Automatic Tools

Pulsonix includes a new suite of automatic tools for new customers to use for placing and routing PCB designs.

Existing customers who wish to evaluate this suite should contact Pulsonix Sales or their local distributor.

Problems Fixed

CAM/Plot

Gerber - Application failed whilst writing top copper layer to a Gerber file using RS274X hardware fill. When powerplane spoke widths were too large and overlapped, the thermal pads were incorrectly output.

Display

User defined pads were being drawn as rectangular or zero sized shapes.

Import

Protel - A design imported from Protel caused routing errors.

Ultiboard - Pulsonix could not read some Ultimate schematic designs. Ultiboard PCB importer had unrecoverable error for a particular design.

Install / Licensing

Network licensing - the install of the Network License Server program did not include the "System" component.

License Manager - did not work on some windows 95 or 98 machines. A separate "licenceManager95.exe" program is now supplied for users that experience this problem.

Interaction

Delete - Got unrecoverable error shortly after deleting a bus.

Hierarchy - Hierarchy problems after using Regenerate Block Symbol.

Move - Program exited after moving a via to push adjacent tracks.

Nets - Problems attempting to join to a net that only existed in the component bin.

Online DRC - With continuous Online DRC on, could not start or end a PCB track on a Star Point.

Properties

Component - There was a problem changing a component's footprint with a connected pin selected.

Tools

Autoplace - autoplace of selected components could cause the program to exit.

Back Annotation - report of component swaps of single gate components added a ' character to the end of the name. Did not perform component name swaps if the gate swap had no gate id.

Pour - On a particular design Copper Pour failed to pour the bottom-most template on Top layer. Copper Pour did not always add thermal spokes.

Router - could put tracks across, or too near to, pads on other nets if design contained wires.

Specctra - unrecoverable error when restoring Specctra results where "via" was a single sided testpoint. Problems exporting a design containing Star Points to Specctra format. Specctra output was only using one via size per design.

Synchronise Designs - did not handle Star Points very well.