

Release Highlights for CAM350 Product Version 9.0

Release, October, 2005
Copyright 1994-2005, DownStream Technologies, LLC

Introduction to CAM350 Version 9.0

CAM350 Version 9.0 is a major release for CAM350. CAM350 9.0 introduces Streams Rule Check, a new method of doing Design Analysis in CAM350. Run Design Rule (DRC), Design for Manufacturing (DFF,DFM), and Netlist Comparison together in one checklist. Create, save and recall multiple “Streams” based on design technology, and/or manufacturing capabilities. Run differing analysis in High Technology areas such as BGA or Wire Bond, and run standard analysis on the rest of the design, all from one “Stream”. Users can define as many analysis areas as desired. Errors can be quickly verified by charting the results. Users are no longer required to view each individual error. Charting of results allow the user to quickly determine what, if any, changes need to be made to the design database. Users of Mentor Graphics PADS Layout or Cadence Design Systems Allegro can then crossprobe the results back into the design database, and repair the original (or actual) design rather than the Gerber layers. This allows the user to maintain database integrity, instead of depending on Fabrication to fix any errors found.

Stream Rule Check

- One setup for DRC, DFF and Netlist Comparison.
- Save multiple streams based on technology, manufacturing capabilities, or customer requirements.
- Run differing analysis on discrete areas of the design, or on different layers.
- Define Areas as keep out and run analysis on all area except those.
- View errors individually or in groups.
- Chart groups of errors to quickly identify common issues
- Annotate any error individually with comments and/or instructions

New DFF Analysis checks

- New Negative Plane analysis
 - Plated Drill to Copper
 - Unplated Drill to copper
 - Thermal Annular ring
 - Copper to true board edge
 - Thermal Tie Width
 - Plane Separator Minimum Width
- Copper to True board Edge (positive and negative layers)
- Find Antennas (Non-terminating traces)
- Missing Soldermask clearance
- Silkscreen minimum line width
- Enhanced Silk to Solder analysis (10x performance increase)

Panelization Enhancements

- No Partial in Negative Venting
- User defined venting keep out areas
- Scaling on export from Panel editor

Crossprobing included in 265 bundle and higher

- First added in the 8.6 release a separate module
- Crossprobing to Mentor PADS layout

- Crossprobing to Cadence Allegro

Defect Fixes – Over 35 software corrections have been made to previously existing functionality, further improving upon CAM350's quality and reliability.

Please see the Release Notes included with the update for a complete list of corrections.

CAM350 Version 8.7 makes "optional" functionality "standard" in the entry-level Gerber analysis and editing configurations. Now included in those systems are the Fast Array module and the ODB++ Import module. In addition, all mid-range and high-end configurations - for both PCB Designers and CAM Engineers - include the ODB++ Export module. CAM350 Version 8.7 also includes support for Altium's PCAD 2004 and the next product update of Mentor's PADS PCB layout software, as well as a number of other software corrections.

CAM350-110 - Now with Fast Array Module and ODB++ Import modules.

CAM350-260 and 460 - Now with ODB++ Import and Export CAM350-260 and 460 now supports a bidirectional ODB++ interface

CAM350 Version 8.6 delivered cross probing functionality to Mentor Graphic's PADS PowerPCB® and Cadence's Allegro® PCB design solutions. Design errors can be highlighted in CAM350 and its location will also be shown in the CAD software allowing the designer to quickly and easily correct the problem. CAM350 can also be used to view intelligent data in the CAD software (components, pins, nets) while at the same time, viewing the corresponding locations as Gerber data in CAM350. Key features include:

- Bi-directional zoom/view window location between CAM350 and PowerPCB and Allegro databases
- Window/View synchronization
- Automatic layer displaying and synchronization
- Bi-directional selection of components, pins and nets
- DRC/DFE error identification contained in CAM350 is identified in PCB layout system
- Cross Probing between intelligent CAD file and unintelligent Gerber data

CAM350 Version 8.5 delivered important Panel Editor enhancements that give fabrication users more control over both stepped image definitions and panel fabrication. CAM350 Version 8.5 also includes important customer requested enhancements and critical customer reported defect resolutions.

CAM350 Version 8.0 delivered significant usability and performance updates for the mainstream CAM marketplace. Version 8.0 updates included a new Graphical User Interface (GUI) with many customizable features, enhanced error verification and identification, negative plane verification updates, and performance improvements.

Panel Editor - Merge Panel Data – The Panel Editor for Version 8.5 incorporates commands that permit import and merging of data into the Panelized database. Gerber data, drill data and mill data can be imported directly into an existing Panelized database. The Panel Editor for Version 8.5 also incorporates a file merge capability that allows previously defined panelization definitions to be merged with new image definitions. Merging panelized data allows base panel templates to be defined and then reused for successive panel designs.

Panel Editor - Stepped Image Manipulation - The Panel Editor for Version 8.5 incorporates edit commands that permit rotation, mirroring and copying of panel elements, independent of the Panelization setup process. Stepped images and panel symbols, including coupons, title blocks, pinning holes and fiducials, can be independently rotated, mirrored, and copied and in the Panelized database. In addition, the entire Panel definition can be rotated at multiples of 90 degrees.

Panel Editor - Extended origin definition - The Panel Editor for Version 8.5 incorporates commands that permit in specification of new data origins. Data origins can now be independently specified in Version 8.5 for the Panel Space, the Filmbox and the Panel NC data. The Panel Space origin affects Gerber data export from the panelized database. The NC origin affects Drill and Mill data export from the panelized database. You can specify whether the Filmbox origin is specified independent from the Panel Space origin or not.

PADS Export enhancements – The CAD Export interface for Version 8.5 is enhanced for PADS PowerPCB ASCII file export. For Version 8.5, you can now export PADS PowerPCB ASCII version 3.0 and version 4.0. Support for PowerPCB version 4.0 includes support for increased layer counts, including a maximum of 64 electrical layers and 250 total layers.

New DFF commands for Gaps and Feature Sizes – CAM350 Version 8.5 delivers new DFF commands to find Minimum Gaps and Minimum Widths for features on artwork layers. The Minimum Gap command detects all possible gaps smaller than the threshold, including gaps created by pads, traces and/or polygons. Further, violations are detected for gaps created by a polygon or trace against itself, and in polygon void regions. The Minimum Width command detects feature sizes smaller than the threshold, including feature sizes for pads, traces and/or polygons.

Fire 9xxx Export Enhancements – CAM350 Version 8.5 delivers Fire 9xxx export enhancements. The Gerber Export command for Version 8.5 gives you advanced control over the plotter commands in the Fire 9XXX. For Fire 9XXX headers for Gerber output, the Gerber Export command allows you to fully specify general mirror settings, emulsion side and axis swap settings.

Add Polygon Stencil Enhancements – CAM350 Version 8.5 delivers new advanced controls for adding polygons based on existing artwork. For Version 8.5, Add Polygon permits you to create customized outlines of all artwork features for a given layer. These advanced controls support stencil generation and other applications that require outline rather than filled polygon representation.

Customer Reported Software Corrections

Some of the customer reported issues were addressed in CAM350 9.0 include:

IMPORT

- 22108 - For certain ODB++ files with cutline polygons, the ODB++ import command may not properly import polygon voids.
- 22068 - For certain PADS ASCII files, the CAD Data import command fails to complete.
- 21862 - For certain ODB++ files with composite negative plans, the ODB++ import command fails to complete.
- 21837 - For certain PADS 2005 ASCII files with copper pour priorities defined, the import command will not complete.

EXPORT

- 22201 - For certain ODB++ files with cutline polygons, the ODB++ export command may not properly export polygon voids.
- 22089 - For certain panel designs, the Export Panel Drill data command for Seib & Meyer 3000 fails to create proper drill file format.
- 22081 - For certain designs, venting must be reset using the Venting dialog in the Panel Editor, before correct Gerber data can be exported.
- 22063 - For certain panel designs, the Export Panel Drill data command fails when double hits are present in the Panel data.
- 22007 - For certain designs with loop nets that have no defined endpoints, the Bed of Nails Editor export command for Circuitline will drop loop nets.
- 21816 - For certain CAM350 files with long names, the File Export IPC350 command will not complete.
- 16632 - For certain CAM350 files jagged mill paths, the File Export Mill Data command will export incorrect data.
- 12130 - For panel data, the Export Panel Gerber command should support scaling data by type.

EDIT

- 21764 - For certain designs, when attempting to explode custom apertures the Edit Change Explode Custom will fail to properly add the exploded data to the CAM350 database.

PANEL EDITOR

- 22083 - The Info Report NC Drill command does not report drill count for stepped images on the panel.
- 22082 - Polygon conversion for vector polygons fails for certain designs.
- 19109 - Negative vents are not properly exploded when panel data is flattened.
- 15784 - Symbol clearance is only rectangular. Symbol clearance should be general polygon shapes.
- 14741 - Exploding Dot and Hatch venting in general does not match the original venting in the Panel Editor.
- 14740 - Dot venting with for negative vents does not have support for prohibiting partial dots.
- 14214 - When using create to update an existing panel or loading a redundant setup file (*.PAN), redundant data is defined or added to the panel setup and recreated panel.

UTILITIES

- 22870 - Polygon conversion for vector polygons fails for certain designs.
- 14565, 17698, 19042, 22188 - Polygon conversion from Draw to Raster fails for certain designs.

ANALYSIS

- 22572 - For certain designs, the current window changes the number of errors that are reported.
- 22559 – The Minimum Gap DFF check generates false errors at teardrop segments for certain designs.
- 22008 – Allegro IPC netlist soldermask access information needs to be specially handled for File > Import > IPC netlist command.
- 20850 – Terminology change from "No CAM Net at Point" to "Extra" and from "No Hit on CAM net" to "Missing".
- 20849 – The option should be supported to create the reference External netlist from the extracted CAM netlist.
- 19048 – Customer would like to mark particular DRC failures as ignore, so that they don't show again in successive runs.

SETTINGS

- 19050 - The View External Netlist options should be saved with the design when setting color and shapes for external net points.

QUERY

- 15201 - Selected padstacks are not highlighted when they are selected in the Query Padstack command.

INSTALLATION

- 17078 - The default Symbol library path should be set to ...\\Symbol.

GUI

- 16743 - Commands cannot be added to a new menu.
- 19047 – Coordinate bar should be dockable and popup.

FLEXIm is a registered trademark of Macrovision Corporation.