

## **Release Highlights for CAM350 Product Version 9.1**

Release, May, 2006

Copyright 1994-2006, DownStream Technologies, LLC

### **Introduction to CAM350 Version 9.1**

CAM350 Version 9.1 is a support release for CAM350. The Streams Rule Check was first introduced in CAM350 Version 9.0; for CAM350 Version 9.1, Streams Rules Check is now included in what was the CAM350-265 bundle. Previously only available as add on feature, or only in the higher end packages, Streams Rule Check is now part of the Mid-level CAM 270 package for PCB Designers and CAM Engineers. Additional enhancements have been made to the interfaces with major CAD systems including Mentor, Cadence, PCAD and AutoCAD. PDF output has been updated to the latest release as well.

#### ***Feature Highlights:***

##### ***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)

##### ***Interface Updates***

- Mentor Pads Layout 2005 SP2
- Cadence Allegro15.51
- Orcad 10.51
- Altium PCAD 2004 SP2
- AutoCAD DXF updates
- Adobe PDF compatibility updates

### ***Panelization Enhancements***

- Explode Panel Frame offset panel layers while retaining step and repeat data.

### ***Soldermask Analysis Enhancements***

- Now looks for missing soldermask clearances for all pads.
- User can filter pads out if they are intended to be covered.

### ***Full Screen Mode***

- Users can turn off all Menus, allowing just the Graphics to be displayed
- Allows the crossprobing function to use the entire screen
- Users can switch in and out of this mode using hotkeys

***Defect Fixes*** – Over 35 software corrections have been made to previously existing functionality, further improving upon CAM350's quality and reliability. Please see the Read.me file or the list included below for a list of corrections.

CAM350 Version 9.0 was a major release for CAM350. CAM350 9.0 introduced Streams Rule Check, a new method of doing Design Analysis in CAM350. Run Design Rule (DRC), Design for Manufacturing (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.

CAM350 Version 8.7 made "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/DFM 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 9.1 include:

#### **IMPORT**

-----

- 29476 - For certain PADS ASCII files, the CAD Data import command does not properly mirror custom apertures with polygon voids.
- 22062 - For certain DXF files without EXTMIN and EXTMAX constructs, the DXF import command results in line segments that are not joined.
- 18456 - For DXF files without REGION and 3DFACE constructs, the DXF import command will drop unsupported graphics.
- 24262 - For PCAD 2004 files, the PCAD Data import command does not automatically generate solder mask and paste mask layers for padstacks.
- 27358 - The Auto Import command will not use the setting for UNIT preference when the FINISH button is selected.
- 30769 - For certain ODB++ files with SMT pads and coincidental micro vias, the File Import ODB++ command may drop SMT pads.
- 30548 - For certain ODB++ files with split planes, the ODB++ import command fails to complete.
- 29704 - For certain ODB++ files with SMT pads, the ODB++ import command may import extra pads.
- 29477 - For ODB++ files with parts with same named pins, the ODB++ import command may drop some of the same named pins.
- 28580 - For certain ODB++ files with metric drill sizes that were created by CAM350 ODB++ export, the ODB++ import command may drop drill hits on reimport.

#### **EXPORT**

-----

- 28841 - For certain CAM350 files with NC Origin and data extents near the world limits of CAM350, the File export Drill Data command may drop some drill hits for "0,0" origin output mode.
- 27034 - Changing the drill file extensions does not change the default extensions for the File Export Drill data command.
- 31056 - For designs with error layers generated by the Analysis > Min Gap or Analysis > Min Width commands, the File Export Gerber data command may generate oversize lines because no DCODE is specified.
- 28395 - For certain designs with complex poured polygons, the File Export Netlist command will execute slowly when exporting netlist files..

#### **PRINT**

-----

- 21989 - When CAM350 prints to the ADOBE PDF 7.0 printer driver to create PDF files, the first page in the PDF file has incorrect line widths.

#### **PANEL EDITOR**

-----

- 24803 - The Explode command in the Panel Editor should have an option to retain step and repeat data so that panel layers can be offset, while the data compression is retained.

#### **UTILITIES**

-----

- 30458 - For certain designs with lines drawn with square apertures, the Draw to Raster command may fail to complete.
- 30457, 30533 - For certain designs with lines drawn with square apertures or complex polygons, the Netlist Extract command may fail to complete.

- 27956 – For designs with saved DRC errors that refer to net names, netlist extract can invalidate the references to the net names.
- 27894, 28977, 30469 – For designs with drills and pads that are not aligned in padstacks, nets may not be extracted correctly.
- 23216, 27444, 27896, 28816 – For designs with complex polygon areas that are filled with stroked lines, the Netlist Extract command may fail to complete.
- 29393, 27829 – The Convert Composite command does not retain as many traces as the Composite to Layer command.
- 31205 – For certain designs with small, unimportant gaps between polygons, Convert Composite will not properly merge reasonably close polygons.
- 27016 – For certain designs with oblong data, the Remove Isolated Pads command will remove connected pads.

#### ANALYSIS

-----

- 28480 – For design with drills not centered on pad centers, the Plated Drill to Copper check erroneously reports zero gap errors.
- 23552 – For certain designs with complex polygons, the Compare Netlist command will fail to complete.

#### MACROS

-----

- 25367 - The pin number argument for the macro “set\_pinseq@” is ignored.

#### GUI

-----

- 27695 – Double clicking on a \*.CAM file with a long name converted the long file name to the DOS 8.3 format standard.

#### LICENSING

-----

- 27928 – The INI file setting “DisableEnv” is ignored for advanced FLEXlm network licensing settings.

FLEXlm is a registered trademark of Macrovision Corporation.