

## Release Highlights for CAM350 Product Version 8.6

Release, September ,2004

Copyright @ 1994-2004, DownStream Technologies, LLC

### **Introduction to CAM350 Version 8.6**

Version 8.6 is a support CAM350 release, following the major release of Version 8.0 and the follow-on Version 8.5. CAM350 Version 8.6 CAM350 now contains specific 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 8.6 also delivers DXF interface updates.

CAM350 Version 8.5 delivers 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.

CAM350 Version 8.6 updates include the following.

*Crossprobing to PowerPCB and Allegro* - User's can click on a design error and the Crossprobing feature will zoom to the proper level and turn on the proper layers on in Power PCB or Allegro. User's can choose components, component pins, or nets in their respective CAD Software and zoom to the particular feature in CAM350, even if only Gerber files are loaded. You can also zoom to a particular feature in one system and the other system will also zoom to the same level.

*Update to AutoCAD 2004 file format* - The DXF interface has been updated to allow files generated in AutoCAD 2004 format to be translated into CAM350. CAM350 can now output this format as well. Updates include:

- /// EXTNAMES - Extended name lengths - with the introduction of extended symbol names in ACAD 2000, the character limit for strings has been removed. There is no explicit limit to the number of characters per line, although most lines should be within 2049 character range.
- /// RTEXT - Remote Text, that is, text that is read from external text file.
- /// WIPEOUT - Negative Images that can be either raster (bitmap) or vector (rectangle or polygon).
- /// MTEXT - Updates to line spacing of Multiline Text
- /// HATCH - support for so-called unassociated hatch
- /// STYLE - The STYLE definition may now be in external file

CAM350 Version 8.5 updates include the following.

**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.

**Usability Updates** – CAM350 Version 8.5 delivers control for composite conversion and netlist comparison. Gerber import optionally supports convert composite to a single layer. The Convert Composite command is optimized for speed and retention of draws and flashes. Netlist Compare errors provide additional capabilities for viewing errors. Short and open errors highlight nets in different colors. The Find Net command will find external nets in imported IPC netlists. A new user preference is added for precise arc approximation for arc and circle conversion.

### **Customer Reported Software Corrections**

Some of the customer reported issues were addressed in CAM350 8.6 include:

#### **IMPORT**

-----

- 14259,15199,15202 – Auto import of Gerber RS-274D files does not properly detect ETA 2.3 format
- 14815 – Gerber import for G04 before M02 incorrectly imports data
- 14835 – Gerber import drops void for cutline polygon
- 15315,15594,15908 – Gerber import incorrectly imports arcs for complex polygon
- 16067 – Gerber import limits aperture size in ARL files to 2 inches
- 16347 – Gerber import incorrectly fills raster
- 14635 – Mentor import fails to import properly drills associated with component definitions.
- 14631, 14678 – ODB++ import does not properly import mirrored part definitions from Zuken export.
- 14696 – ODB++ import does not extract/create net connectivity on negative planes.
- 14697 – ODB++ import creates SMD pads on unused through holes.
- 14778 – ODB++ exported of panelized data causes crash after the exported ODB++ file is imported into CAM350.
- 15426 – ODB++ import takes excessive time to complete for complex polygons.
- 15686 – ODB++ import creates incorrect apertures for imported padstacks

16616,16617 – ODB++ import may swap dcodes.  
13413 – DXF import of AutoCAD 200x DXF files produce error messages  
14471 – DXF import incorrectly approximates arc on polygon  
15722 – DXF import fails for nonstandard font directory  
16039 – DXF import imports incorrect rotation and offset for nested blocks.  
14865 – PCAD import does not flip padstacks for mirrored through hole component pins.  
15343 – PCAD import fails to complete  
15404 – PCAD import fails for padstack layers not included in design data  
15914 – PCAD import creates oversize SMD pads  
16405 – PCAD import creates oversize thermal connections  
16704 – PADS import fails with non ASCII characters are in drill chart  
15904 – GENCAD import fails for designs with padstacks that have only holes and no pads

#### EXPORT

-----  
15951 – PADS export will not properly export mask data for padstack definitions for certain designs.  
15632 – Gerber export fails to create proper output for small polygon

#### EDIT

-----  
15687 – Edit->Rotate select all causes improper rotation of Dcodes for certain designs with undefined aperture.

#### UTILITIES

-----  
14874 – The Edit->Layers->Scale command will abort if non-ASCII characters are entered for the scale units.  
15161 – Utilities->Convert Composite does not rotate Dcode properly for Create Custom option for certain designs.  
15205 – Utilities->Composite to Layer command fails to properly convert polygon voids for certain designs.  
15352 – Utilities->Build Part command fails to complete.  
15734 – Utilities->Teardrop command adds teardrops to traces with no pads.  
15819 – Raster to Vector conversion drops fill data

#### ANALYSIS

-----  
13998 – Analysis Control Panel does not support removing deleted errors; [Refresh] button needed  
14149 – DRC track to track check is slow for large poured regions  
14246 – DRC fails to display report in Notepad for Windows XP limited user accounts  
14553 – Plane Check Isolated Connections analysis finds false errors.  
14934 – The DRC command will leak excessive memory for successful faults on large polygon outlines.  
15321 – The Analysis Minimum Gap command fails to find errors for certain files with small feature definitions.  
15689 – DRC pad to pad check misses errors for certain designs with zero border line width.  
16442 – Netlist extract fails for complex polygon

#### PANEL EDITOR

-----  
14744 – After rotating panel, Gerber export causes program failure.  
15013 – File->Merge Panel Data overwrites existing Dcodes instead of defining new Dcodes for imported data.  
15160 – After File->Import->Gerber Data, the Edit commands will cause crash for certain designs.  
15193 – When deleting stepped images in the Panel Editor, the REF layer designation in the Layers Table is not properly considered.

15592 – Edit->Change->Explode->Flatten Panel will not properly flatten venting patterns for inner layers for certain designs.

#### NC EDITOR

-----

14548 – Combining tools in the Panel Editor corrupts paths.

14877 – Mill export and import results in incorrect path.

15673 – Utilities-Sort Drill Hits drops drill hits when scanning in Y direction for certain designs.

16662 – NC drill sort adds drill hit in one up path for Panel design

#### TABLES

-----

15353 – After removing a layer from a composite definition for certain designs, the View->Composite command fails to complete.

#### MACRO

-----

14640 – Minimum Gap and Minimum Width commands do not have query macros for error counts

15345 – Recorded Test point export commands do not playback properly

15686 – A recorded macro for multiple DRC passes does not find all present DRC errors for certain designs.

#### USER INTERFACE

-----

13538 – No help topic for Panel Editor NC Editor Utilities > Step and Repeat Image Order.

14677 – Double clicking for CAM350 file causes CAM350 to crash.

14824 – The NC Editor menu for the CAM Editor is empty for Edit->Change->Origin.

16403 – The negate “N” modeless command will not work on the TOP defined layer.

Allegro is a registered trademark of Cadence Design Systems, Inc.

PADS PowerPCB is a registered trademark of Mentor Graphics Corporation..

FLEXIm is a registered trademark of Macrovision Corporation.