Release Highlights for CAM350 Product Version 8.7

Release, February, 2005

Introduction to CAM350 Version 8.7

CAM350 Version 8.7 is a support release for CAM350. CAM350 Version 8.7 makes what was once "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

Fast Array Module – Built on our sophisticated Panel Editor technology, the purpose of this tool is to very quickly array a PCB or group of PCBs on a panel to feed fabrication and assembly processes. This module is in response to consumer demand for an easy-to-use yet highly functional module to support their needs to define PCB positions on a panel, without a lot of setup and definition, and to best optimize down stream processes more effectively.

ODB++ – The defacto standard for intelligent data exchange in EDA, ODB++ is an intelligent format that captures all the CAD/EDA, assembly and PCB fabrication knowledge in one single database. This format takes the place of individual Gerber, drill, and aperture files, and adds additional information such as components and nets. All major CAD tools, can export this format, allowing the user to bring intelligent data into the CAM system. Common issues with aperture files, drill tools, and other problematic data formats are avoided. Users of CAD tools that do not have ODB++ export capabilities, can export this format from CAM350, giving them a more accurate and intelligent method of data export.

Mentor PADS PCB Layout support – Updates to the Pads ASCII format have been incorporated into CAM350 8.7. This insures no compatibility issues once Mentor has released the next update for PADS Layout.

Altium PCAD 2004 support – The bidirectional PCAD interface has been updated to support the latest release from Altium, PCAD 2004.

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/DFF 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.

NewDFF 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 8.7 include:

IMPORT

18338 - For certain Gerber files with zero line widths in aperture macros, the import operation will not correctly create custom apertures.
18015 - For certain ODB++ files, the ODB++ import operation does properly import PLCC SMD pads.
17172 - PADS 2005 ASCII files, the PCAD import operation fails import True Type Font.
18801 – PCAD 2004 files will not import.
18881 – PCAD files with empty padstacks will not import.
17006 – Certain Mentor Boardstation files will not import.
18909,18910 – Auto import guesses the wrong unit for PADS drill files
19297 – DXF files with certain 3D extrusion settings fail to import.
16787,19762 – DXF files with certain mirrored arcs import data incorrectly.
17992 – File merge fails for files names with spaces.
17304 - For certain Allegro drill files for small boards, the Auto import operation fails to import Allegro drill data at the correct unit.

EXPORT

18041 - For certain Gerber data, the Gerber export operation will incorrectly warn and export 360 degree arcs.
16707 – Netlist export fails for designs with null pins.
18931 – ODB++ export incorrectly exports small arcs.

EDIT

17692 - For certain custom aperture definitions, the Edit>Change > Text command does not register as a database change.
17205 - For certain custom aperture definitions, the Edit>Mirror command may create incorrect data.

UTILITIES

16631 - For certain designs when converting raster polygons to vectors polygons for large raster poured polygons, the Raster to Vector Poly command may drop fill data.
17267 - Netlist extract fails to complete for certain designs with overlapping data.

ANALYSIS

18039 - For the DRC dialog, after entering text in any text edit box, if the escape key <ESC> is hit before any other action the program will abort incorrectly.
19114 – DRC errors at .025 mm reports false errors at .025 mm

MACRO

18273 - Recorded Custom Aperture macro commands referencing external CAP libraries will not properly playback in the CAM Editor.
18907 – Change text style macro commands do not playback correctly

TOOLS

18796 – Allegro 220 bundle is not supported by CAM350 8.6 installation.

FLEXlm is a registered trademark of Macrovision Corporation.