CAM350™ - 465

CAM350 Modules	Design Family Configurations			
	C350-765	C350-465	C350-270	C350-110
Import	✓	✔	✓	✓
Information	✓	✔	✓	✓
Export	✓	✔	✓	✓
Modification	✓	✔	✓	✓
Optimization	✓	✔	✓	✓
Design Rule Check (DRC)	✓	✔	✓	✓
Basic NC Editor	✓	✔	✓	✓
Quote Agent	✓	✔	✓	✓
Fast Array Module	✓	✔	✓	✓
ODB++ Import	✓	✔	✓	✓
ODB++ Export	✓	✔	✓	Option
DXF Interface	✓	✔	✓	Option
DFF Audit	✓	✔	✓	Option
Macro Debugger	✓	✔	✓	Option
Crossprobing - PADS	✓	✔	✓	Option
Crossprobing - Allegro	✓	✔	✓	Option
Streams Rule Check	✓	✔	✓	Option
Panel Editor	✓	✔	Option	Option
Advanced NC Editor	✓	✔	Option	Option
DirectCAD Interface (Out Only)	✓	Option	Option	Option
Reverse Engineering	✓	Option	Option	Option
Flying Probe Editor	✓	Option	Option	Option
Bed of Nails Editor	✓	Option	Option	Option
DirectCAD Interface (In Only)	Option	Option	Option	Option
Camtek AOI	Option	Option	Option	Option
	Fabrication			Engineering

CAM350's modularity allows for the configuration of a solution to meet the exact needs of any organization. CAM350-110 and CAM350-265 are pre-configured solutions based on the typical needs of a PCB Designer. CAM350-465 and CAM350-765 are pre-configured solutions based on the typical needs of a CAM Engineer. These solutions can be purchased "as-is" or you may build a CAM350 solution based on your specific requirements.

CAM350-465 Overview

CAM350-465 module is designed for the CAM Engineer who needs to automate and optimize the process to efficiently produce bare printed circuit boards. CAM350-465 manages data input and preparation, analysis, test, mill and drill and final bare-board production while increasing productivity, quality and throughput.

CAM350™ – A Complete PCB Fabrication Flow for PCB Designers and PCB Fabricators

Built to meet the needs of both PCB Designers and PCB Fabricators, CAM350 is a complete PCB fabrication flow that streamlines the transition of engineering design data into physical PCBs. This powerful solution provides superior price/performance value in an easy-to-use, modular product suite.

For the PCB Designer, CAM350 can detect and correct PCB fabrication errors early in the design process. Design for Fabrication (DFF) verification drastically reduces costly design re-spins and increases PCB manufacturability.

For the PCB Fabricator, CAM350 accurately prepares and optimizes PCB design data to dramatically increase yield, quality and turnaround time of bare PCBs.

800-535-3226 www.downstreamtech.com sales@downstreamtech.com

DownStream Technologies

CAM350™ - 465

Netlist Compare

Ensure that the original design intent is maintained by verifying the CAD Netlist matches the extracted Gerbers. The Netlist Compare functionality minimizes the risk of translation error by automatically validating the Gerber files match the original data.

Design Rule Checking

Design rules must be continually verified to meet the original intent of a PCB design, while simulateously ensuring compliance with the fabrication process. DRC ensures design files meet original specifications and still comply with manufacturing. Checks include spacing, annular ring, histogram, copper area calculation, layer compare, and more.

Layer Compare

Allows you to compare different revisions intelligently, as opposed to viewing the layers on top of each other and trying to verify differences visually.

DFF Audit

This feature performs over 80 essential bare-board analysis checks prior to manufacturing. Searching the PCB design for such issues as acid traps, copper and mask slivers, solder bridges, starved thermals and more. It uses raster-based processing that is both fast and independent of layout technique.

Streams Rule Check

Now you can define a series of various verification steps (Streams) for DRC, DFF and netlist comparisons, execute all of them with a touch of button and re-use them over again on other designs.

Crossprobing

Dynamic Crossprobing functionality to PADS Layout® and Allegro® Windows-based PCB design solutions. Design errors will be highlighted in CAM350 and its exact location shown in the CAD software. With Crossprobing,CAM350 can also view intelligent data in the CAD software (components, pins, nets) while at the same time, viewing the corresponding locations as Gerber data in CAM350.

Additional Modules Included

Import & Export – Offers a range of import/export capabilities, plus view, query, report and measure.

Modification – Edit and change design files to your specific requirements.

Optimization – Optimize the design files using draw-to-custom, draw-to-flash, and draw-to-raster polygon conversion, netlist extraction, silkscreen clipping, redundant pad and data removal, and teardropping.

Basic NC Editor – For NC-Mill and Drill Data – Import, export, and creation capabilities are included, as well as some editing capabilities to change drill tool definitions, add basic mill paths to assembly panels, and to change break tabs.

Quote Agent – Designed to extract the necessary information from the PCB design to accurately quote manufacturing costs.

Macro Debugger – Macro Script development allows the user to set watch points and intelligent break points, analyze variables, and more.

Bi-Directional DXF Support – This feature offers bi-directional AutoCAD and DXF support. It handles filled solids, true text, blocks, nested blocks, and builds aperture tables automatically..

Fast Array Module – Automates the sub-panel process to quickly array a PCB or group of PCBs on a panel to feed fabrication and assembly processes.

Panel Editor — Automates the panelization process and allows for the creation of panel templates, intelligent coupons, pinning holes, fiducials, and title blocks. Populate panels in either an automatic stepping mode or use a spreadsheet or total control. Venting and thieving are fully automated and it allows for multiple layers to be processed in either a positive or negative polarity.

Advanced NC-Editor — The Advanced NC Editor allows for the manipulation of NC drill and mill data in PCB designs. Add drill hits and mill paths, as well as advanced canned NC routines such as drill text, mill circles, operator messages, pilot holes and more.

ODB++ Import — 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. Common issues with aperture files, drill tools, and other problematic data formats are avoided with ODB++. Users of CAD tools that do not have ODB++ export capabilities, can export this form at from CAM350.

800-535-3226 www.downstreamtech.com sales@downstreamtech.com

DownStream Technologies