CAM350™-270

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 CAM350-270 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.

CAM350-270 Overview

CAM350-270 is the solution for PCB Designers looking to streamline and optimize the process of transitioning design data into successful bare printed circuit boards.

CAM350™ – A Complete PCB Fabrication Flow for PCB Designers and CAM Engineers

Built to meet the needs of both PCB Designers and CAM Engineers, 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 CAM Engineer, 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™-270

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.

A CAD Netlist is reference designator/pin number based. A Gerber extracted Netlist is XY coordinate based. IPC-D-356 contains both, allowing the comparison of the two.

Netlist Compare will find common errors such as:

- Accidental inclusion/exclusion of traces, vias, etc., on signal layers
- Gerbers with text or reference designators on a signal layer, instead of silkscreen layer
- Placement of drafting items on all layers, instead of specific document layers
- Problems with CAM Layers such as negative planes

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. Too often, fabricators alter the design to make the boards manufacturable. When done without notification, that same design can later cause scrap or faulty boards.

Design Rule Checking will:

- Reduce spacing due to linewidth alteration for impedance control
- Find insufficient annular ring due to fabrication vs. designed guidelines
- Locate linewidths and spacing below manufacturing capabilities
- Avoid insufficient clearance from drilled hole to copper on high-voltage layers

CAM350 allows easy verification of the DRC results with sortable defect lists. It also has the ability to view one or many defects at a time and create reports based on the results.

Layer Compare

Often a PCB designer is presented with Rev A and Rev B of a design and is tasked to determine the differences. The Layer Compare functionality in CAM350 allows you to compare the layers intelligently, as opposed to viewing the layers on top of each other and trying to verify the differences visually.

Differences can be filtered by size giving the designer the ability to compare the tooled artwork to their original artwork; another method to confirm fabrication has not changed the design data.

DFF Audit

DFF Audit performs over 80 essential bare-board analysis checks. DFF Audit automatically checks for acid traps, soldermask slivers, copper slivers, silk screen, power and ground, signal layers, starved thermals, soldermask coverage, and many more potential problems.

Making sure the Soldermask data is generated using proper clearances, ensuring that there are no potential Solder Bridge conditions, and fixing potential Acid Traps will eliminate bottle-necks in production.

- Analysis checks take only seconds even on complex boards
- Analysis results are graphically displayed for trouble-free viewing
- Violations can be easily inspected and potential problems quickly avoided
- Data integrity is maintained by making alterations directly to the database

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 allowing you to quickly and easily locate and resolve the issue. 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.

ODB++ Import – 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 you to bring intelligent data into the CAM system. 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 format from CAM350.

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

DownStream Technologies

For detailed descriptions and online demonstrations of the above modules please visit www.downstreamtech.com