


CAM350™ -110

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™ – 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.

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.

CAM350-110 Overview

CAM350-110 is an entry-level solution for the PCB Designer who is looking for a basic way to start improving the process of transitioning and verifying design data for fabrication.

CAM350™ -110

Netlist Compare

Verifying that the CAD Netlist matches the one extracted from the Gerber files is crucial to ensuring that the original design intent is maintained. The Netlist Compare functionality in CAM350 minimizes the risk of translation by automatically validating the Gerber files match the original CAD 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
- Gerber files with the text or reference designators placed on a signal layer, instead of a silkscreen layer.
- Placement of drafting items on all layers, instead of specific document layers, i.e., company logo on all layer
- Problems with CAM Layers such as negative planes

Design Rule Checking

In order to meet the original intent of a PCB design, design rules must be verified and the design must meet the requirements of the fabrication process. Too often, fabricators alter design files to make the boards manufacturable. And, if that design is respun at a later date and the changes were not documented, you risk having 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 by providing sortable 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 as to size providing the designer the ability to verify the tooled artwork from the fabricator to the original artwork; another method to confirm fabrication has not changed the design data.

Fast Array Module

Built on our sophisticated Panel Editor technology, the purpose of this tool is to quickly array a PCB or group of PCBs on a panel to feed fabrication and assembly processes. This functionality is used to define PCB positions on a panel, with minimum setup and definition, optimizing downstream processes.

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.