CAM350-115

The CAM350-115 allows you to:

- Translate Gerber, NC, and HPGL files with confidence
- Measure and View data in a user friendly environment
- Create optimized Solder Masks, Solder Pastes, and Stencils
- Analyze data for design and output errors
- Edit and generate legacy Gerber data
- Easily work with NC data for creating optimized manufacturing outputs
- Buried and Blind via support for Analysis and manufacturing outputs
- Fast generation of subpanels for assembly
- Import intelligent data using ODB++ or IPC-2581

This configuration gives you the power and functionality necessary to start building more successful PCBs in less time, with more accuracy and ease.

An Entry Level Solution to Meet Your Basic Requirements

DownStream offers this core CAM350 configuration to assist you with getting your basic PCB Post Processing needs met. It gives you enough functionality to start improving your process of verifying and delivering your design data for fabrication.

CAM350-115 contains the functionality necessary to import, export, optimize, and modify your design files. Plus, with the 115 you get the additional functionality found in Design Rule Checking, the Basic NC Editor, Fast Array capabilities, and IPC-2581 import.

Core Features and Functionality

CAM350 was designed to optimize and streamline the transition of engineering data into physical PCBs. It is the price-performance leader in easy-to-install, learn, and use PCB Post Processing software solutions.

Import, Export, Modify, Optimize
CAM350 offers a wide range of import /export options, including Direct-CAD, ODB++ and IPC-2581, in order to view, query, report and measure the PCB design data. Once imported into CAM350 you have options to optimize the design data using draw-to-custom, draw-to-flash, draw-to-raster polygon conversion, netlist extraction, silkscreen clipping, redundant pad and data removal, and teardropping.

Netlist Compare
Verifying that the CAD Netlist matches the one extracted from the Gerber file is crucial to ensuring that the original design intent is maintained. The Netlist Compare functionality in CAM350 minimizes the risk of translation errors by automatically validating the Gerber files match the original CAD data.

Rule Checking
Design rules must be verified to ensure the original design intent has been met. Just as crucial is defining and setting up manufacturing rules and requirements, prior to packaging the design up for fabrication. Rule Checking will perform spacing checks, annular ring checks, spacing histogram, copper area calculations, layer compare, net checks, and more.

Layer Compare

The Layer Compare functionality in CAM350 allows you to graphically compare two layers for differences. You can compare Rev A to Rev B of a particular design, or compare the original artwork to the tooled artwork from the fabricator. This way you can find problems introduced into the design by changes made by the fabricator.

Fast Array

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 is used to define PCB positions on a panel, with minimum setup and definition, to optimize downstream processes.

Basic NC Editor

CAM350 comes with a NC Editor for NC Mill and Drill capabilities. You have the ability to import, export, and create mills and drill. Some editing tools are also included to change drill tool definitions, add basic mill paths to assembly panels and change break tabs. NC Editing capabilities can be expanded by upgrading to the Advanced NC Editor, available in higher configurations of CAM350.