CAM350-155

The CAM350-155 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
- Work on the same data seamlessly with BluePrint-PCB
- Read and Write to BluePrint Release packages
- Work concurrently with CAD for intelligent analysis and viewing
- Translate to and from AutoCAD

With this configuration, CAM350 supplies you with all the necessary functionality to build successful PCBs in less time, with more accuracy and confidence, all while producing one single comprehensive electronic file viewable by anyone... anywhere... anytime.

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.

A Feature Rich PCB Post Processing Solution

DownStream has created this CAM350 configuration packed with all the necessary features and functionalities to get you successfully through PCB Post Processing.

CAM350-155 contains the functionality necessary to import, export, optimize, and modify your design files. Plus, with the 155 you get the additional functionality found in Design Rule Checking, the Basic NC Editor, Fast Array capabilities, and IPC-2581 import. 155 also includes the Release Package Navigator, DXF Interface, and Crossprobing.

Rule Checking

Design rules must be verified to ensure the original design intent has been meet. 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.

Crossprobing

When using analysis functionality with Crossprobing you are given the ability to ensure your CAD database is kept up to date and accurate as well. CAM350 can locate design errors (using DRC, DFF, and Streams) and quickly pinpoint the exact location of those errors in your CAD software. This feature helps you maintain the original CAD database, ensuring it is always an accurate, current source of information.

With Crossprobing, 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.

The Release Package Navigator

The Release Package Navigator option allows you to work from one single electronic file to store and distribute all the deliverables for manufacturing. For example, this one file can contain your BluePrint documents, Gerber, NC Drill/Mill, Panel Arrays, ODB++, PDF, IPC-2581, DXF, PCB CAD, MS Office and virtually any file located on your computer and/or network.

If you also have BluePrint-PCB, then you can access these files directly from the Release Package within BluePrint as well, and add them to your documentation. The Release Package Navigator also supports the importation of Gerber, and Drill and Mill data from other CAM systems.

Using the Release Package Navigator you can store, distribute, and view all the necessary data required to fabricate and assemble a PCB anywhere, anytime. Manufacturers simply receive your release package, download the free BluePrint viewer, and extract the necessary information they need to build your boards.