

### The Ultimate CAM350 Configuration

This CAM350 contains it all. If you want to be involved and control every aspect of your design throughout PCB Post Processing, this is the configuration for you. It is packed with everything you'll need to get your designs through manufacturing successfully.

### Features and Functionality

The base CAM350 comes equipped with the functionality necessary to import, export, optimize, and modify your design files. Then to make the system more robust, we add Design Rule Checking, NC Editor functionality, Fast Array capabilities, ODB++ Import, the Release Package Navigator, DXF, Crossprobing, Streams Rule Checking, DFF, Macro Debugger, ODB++ Export, and the all new Design Analyzer... but that's not all, with CAM350-810 you also receive Panel Editing, Advanced NC Editor functionality, DirectCAD Output, Reverse Engineering, Flying Probe Editor, and Bed-of-Nails Editor.

## CAM350-810

The CAM350-810 configuration 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 from most major CAD systems via ODB++
- Work on the same data seamlessly with Blueprint-PCB
- Read and Write to Blueprint Release packages
- Work concurrently with popular CAD programs for intelligent analysis and viewing
- Translate to and from AutoCAD
- Complete checklist based Analysis environment
- Find manufacturing defects as well as design problems
- Compare original vs. manufactured designs for differences
- Insure the ECO's are reflected in the Gerber data
- Macro debug tools for creating automation
- Generate intelligent manufacturing outputs via ODB++
- Automate the creation of Fabrication panels
- Electrical test fixturing and test program generation
- Automated Flying Probe test program generation
- Load intelligent data directly from many CAD programs
- Add net and component information to legacy data for Centroid or CAD file creation
- Export CAD files to PADS, PCAD, GENCAD and more...
- Analyze the contents and complexity of a design to create a design report for fabrication quoting and design qualification

*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

### Import, Export, Modify, Optimize

CAM350 offers a wide range of import/export options, including Direct-CAD and ODB++, 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 drills. 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.

## Advanced Features

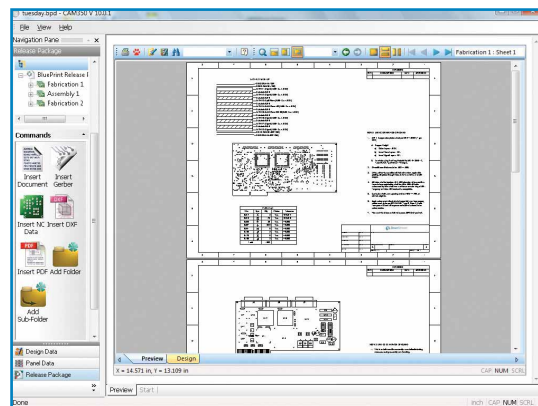
### The Release Package Navigator

The Release Package Navigator found in CAM350-150 and above 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, 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.

With the new compatibility found in CAM350 10.5 and BluePrint-PCB 3.0, you now have the power and ease to control all our PCB Post Processing requirements with one integrated solution.





StreamsRC. Create, save, and recall multiple "Streams" based on design technology, and/or manufacturing capabilities. Run differing analysis on High Technology areas such as BGA or Wire Bond, and run standard analysis on the rest of the design, all from one "Stream".

You can define as many analysis areas as desired. Errors can be quickly verified by charting the results. You are no longer required to view each individual error. Charting of results allow you to quickly determine what, if any, changes need to be made to the design database.

## CAM Engineering Tools

### Panel Editor

Automates the panelization process. Allows for creation of panel templates, intelligent coupons, pinning holes, fiducials, and title blocks. Populate panels in either an automatic stepping mode or use a spreadsheet for total control. Venting and thieving are fully automated as well, processing multiple layers in either a positive or negative polarity with user-defined patterns.

### Advanced NC Editor

The Advanced NC Editor gives you powerful capabilities to manipulate NC drill and mill data in your designs. You can 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. Allows checking and optimization of NC data in preparation.

### Flying Probe

Flying Probe test for bare-board PCB's has never been easier! A powerful graphical editor and filtering options allow you to extract all the necessary data like nets, test-points, and adjacency information in a snap. The Flying Probe Interface presently writes the Probot, ATG, Integri-test, PROBOTECH, IPC-D-356, IPC-D356A file formats, as well as a special ACT Neutral format.

### Bed-of-Nails Editor

The Bed-of-Nails Editor produces all files necessary to build a single- or double-sided clamshell test fixture. A graphical editor and filtering options give you complete interactive control of the test point information. Supported formats include TTI (Test Technologies International), Circuit-Line, IPC-D-356, IPC-D-356A, as well as generic plate drill files and netlists.

## Reverse Engineering

DownStream can "reverse engineer" legacy Gerber files allowing you to bring them back into any CAD system. First, a fully automated netlist extraction will be performed. This process supports even the most complex blind & buried vias, MCMs, and hybrid designs. Once complete, a new data file will be created and then imported into your CAD system.