



Uncompromised PCB Design Processing and Analysis

Before they are transitioned to the PCB fabricator, today's complex PCB designs require comprehensive verification to ensure a successful and timely fabrication process. Errors discovered during fabrication pre-processing can drastically impact product schedules and result in costly design re-spins. Fabricators can make modifications to your design data to minimize delays, but the changes may compromise the design's integrity and intent. Inspecting, preparing and validating the PCB design prior to releasing to manufacturing will result in a significant increase in efficiency. It reduces the risk of design re-spins, and most importantly, ensures successful electronic products are built faster at less cost. CAM350 offers a complete suite of tools to import your design data, modify the data if necessary and finally analyze the design for potential fabrication and assembly errors. From design through fabrication, CAM350 streamlines the transition of engineering data into successful, physical PCBs.

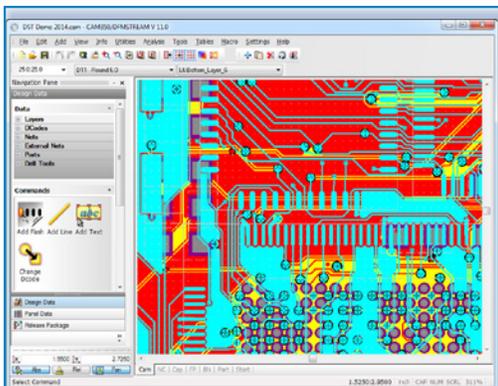
CAM350-II5

CAM350-115 is a collection of functionality to import, modify, optimize and analyze your PCB design data to ensure timely, high-quality PCB fabrication.

Features and Functionality

CAM350 functionality includes:

- Import of standard file formats including Gerber, ODB++, IPC-2581 and others
- View, query, report or measure almost any construct in the design
- CAM editing to add or modify flashes, pads, polygons, lines, text or add teardrops
- Netlist compare and export to IPC-D-356 and other formats
- Convert draws to flashes, drawn polygons to rasterized polygons
- Remove redundant data and clip silkscreen ink away from pads
- NC editing to add or modify milling, mill tabs and drills
- Custom aperture editing to create custom pad shapes or custom polygonal areas
- Design Rule Checking (DRC) to check for minimal spacing errors, minimal annular rings, minimal feature sizes, and other anomalies
- Panel editor to design custom fabrication panels
- Export functions to package your completed design in standard manufacturing formats including Gerber, NC Drill and Mill and others
- Macro based application automation for command record and playback



CAM350 offers a robust CAM editing environment

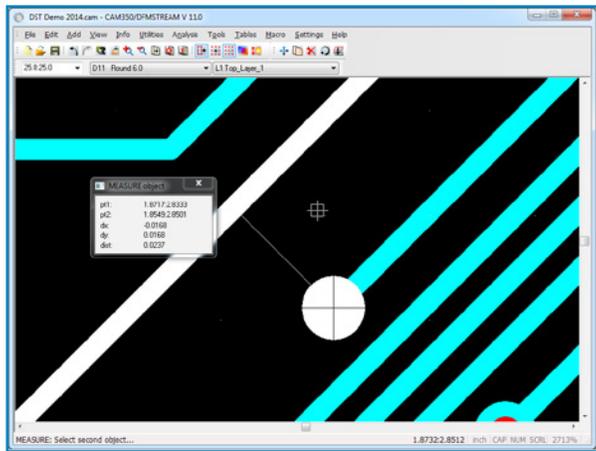


Core Features and Functionality

Successful compilation of PCB design data into usable PCB manufacturing data is critical to minimize delays in delivering new electronic products to market. CAM350 offers all the functionality you need to ensure high quality outputs to manufacturing that result in higher manufacturing yields and shorter time to market.

Import Data from Multiple Sources

Import your design directly from industry standard file formats including ODB++, IPC-2581, Gerber, Excellon, Sieb & Meyer, and HPGL.



Use the measuring tool to query distance between objects.

View, Query, Measure and Report

Filter your view of the design data by layer, Dcode, net, parts, drill or mill tools. Query any element to get pertinent details such as size, shape, area, Dcode, design layer, drill tool, and so on. Get details specific to each type of element. Measure the distance from point to point or object to object. Get reports on Dcodes, a Bill of Materials (BOM), Netlist or Centroid data. Attach custom notes to your design data for future reference.

NC Editing

Use NC editing features to import or export NC drill and mill data. Add drills, mill paths and mill tabs for panel separation. Modify drill and mill tool definitions. Create custom NC drill files for specific drilling operations. Convert Gerber content to NC content and NC to Gerber.

Data Conversion and Optimization

Convert draws to flashes, drawn polygons to rasterized polygons, polygons to board outlines and other conversions. Optimize your design data by removing isolated or redundant pads or remove pads covered by copper. Add teardrops, oversize pads for a solder mask or undersize pads for a paste mask. Remove silkscreen ink from pads.

CAM Editing

Use editing commands to cut, copy, paste, move, rotate or mirror any element. Add flashes, draws, lines, polygons, and many other design elements. Remove or re-order layers. Change attributes, reference designators, text, via or pad properties, Dcodes, text font or style. Add design elements such as vias, wires, parts, or board outline. These are just a sample of the features available to edit your design data.

Custom Aperture Editing

Create custom apertures for use in your design. Custom apertures are effectively shapes that include both positive (additive) and negative (subtractive) data. Create custom thermals, pads shapes or polygonal areas and add them to your design.

Design Rule Checking

Use Design Rule Checking (DRC) to analyze your design for violations of minimal spacing, minimal copper or mask annular rings, minimal pad, gap and track sizes and redundant pads. Locate plated drills without pads, pads without drills or drill to copper violations. Compare layers for differences. Calculate copper area per layer. Use the graphical histogram to visualize minimal spacing thresholds.



Macros

Use Macros to create custom automation in CAM350. For example, record a sequence of often used commands and operations and save time by recalling (reusing) the recorded sequence repeatedly. A CAM350 macro can interact with the user, make conditional decisions, and parse information from the database. Several sample macros are provided along with a web based user driven macro exchange program.

Netlist Compare

Use the Netlist compare features to verify the integrity of design data extracted from the PCB CAD tool. Generate a netlist derived from the data in CAM350 and compare it to an IPC-D-356 netlist exported from the source PCB CAD tool. This comparison ensures the design data was not compromised by the PCB CAD data extraction process.

Export to Multiple Formats

Design data in CAM350 can be exported to Industry standard file formats including IPC-2581, Gerber, Excellon, Sieb & Meyer, and an IPC-D-356 netlist.

Worldwide Sales, Technical Support and Training

All DownStream Technologies products are sold and supported by a worldwide network of channel partners. For sales, technical support or training, contact your local channel partner or visit:

<http://downstreamtech.com/worldwide-distribution.php>

DownStream Technologies, LLC

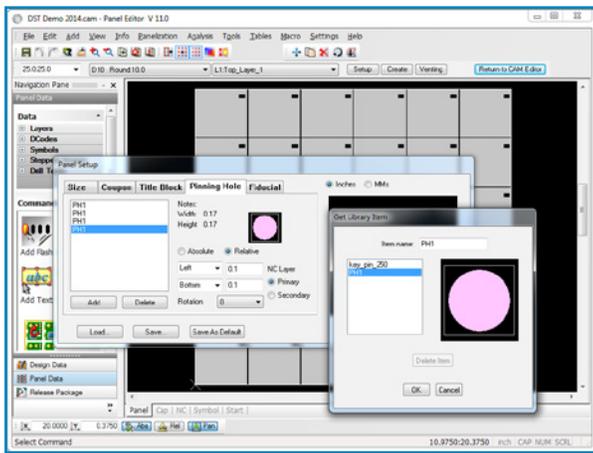
225 Cedar Hill St.

Suite 333

Marlborough, MA 01752

US: (800) 535-3226

Outside the US: (508) 970-0670



Use the Panel Editor to create custom PCB panels

PCB Panel Design

Create a multi-PCB image panel quickly using the panel design features. Array PCB images individually, or enter a few basic parameters and the panel is created for you. The original one up image of the PCB design is preserved separately from the panel design. Add mill paths and mill tabs to create a custom routing design for the panel. When completed, export finished panel contents to Gerber including mill, and drill data.