## 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.
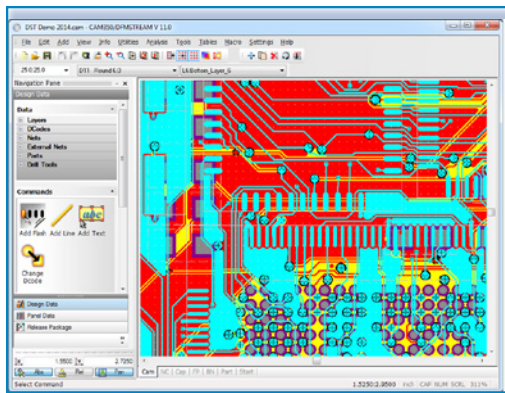
*CAM350 offers a robust CAM editing environment*

CAM350-820

CAM350-820 is a superior 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 universal file formats and direct import of PCB designs in popular CAD formats
- 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
- Crossprobing with PCB CAD systems to visualize design content in CAM350 with its native PCB design file
- Design Rule Checking (DRC) to check for minimal spacing errors, minimal annular rings, minimal feature sizes, and other anomalies
- Design for Manufacturing (DFM) analysis to analyze your design for potential flaws that may lead to fabrication delays
- Part editing to create custom footprints (or parts) required for reverse engineering Geber into intelligent CAD data
- Reverse engineering to convert unintelligent design data, such as Gerber to intelligent CAD data with parts, padstacks and nets
- Panel editor to design custom fabrication panels with mixed board images, test coupons, vent patterns, pinning holes and fiducials
- Export functions to package your completed design in standard manufacturing formats including Gerber, ODB++, IPC-2581, NC Drill and Mill and other formats
- Macro based application automation for command record and playback
- Design analysis and custom reporting to facilitate communication of design statistics to your PCB fabricator for more accurate quotes
- Flying Probe editing to define bare board test point locations
- Bed of Nails editing to produce output for manufacturing custom bed of nails fixtures including clamshell testers

**www.downstreamtech.com**   **sales@downstreamtech.com**
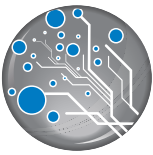
Core Features and Functionality

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 the most popular PCB CAD systems including Mentor Graphics PADS or Xpedition; Cadence Allegro or OrCAD PCB Designer; Zuken's CR5000, CR8000, or CADstar; Altium Designer and several others.* Industry standard file format support includes ODB++, IPC-2581, Gerber, DXF, Excellon, Sieb & Meyer, and HPGL.

*CAD import not included - optional purchase required.
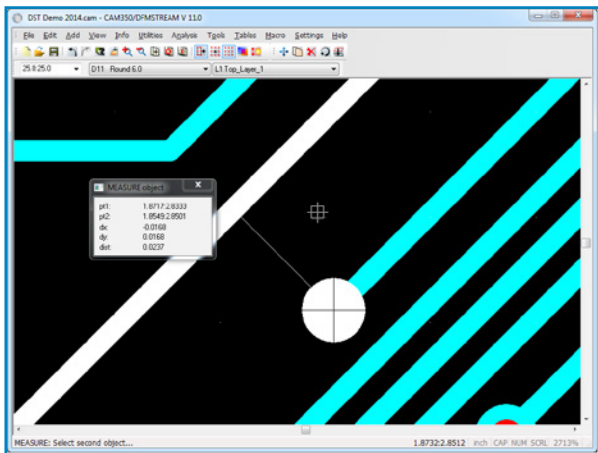
Use the measuring tool to query distance between objects.

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, slots, 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.

PCB CAD Crossprobing

Crossprobing facilitates visualization of CAM350 design data to its source PCB Design in its native PCB tool. Select elements in the PCB design tool in and the equivalent element is selected in CAM350. After a CAM350 DRC or DFM Analysis, use crossprobing to visit error locations in the PCB tool to make corrections. This expedites the process of finding and correcting errors in the source PCB design. CAM350's crossprobing is compatible with leading PCB CAD tools such as Mentor Graphics' PADS or Xpedition; Cadence OrCAD PCB Designer or Allegro.
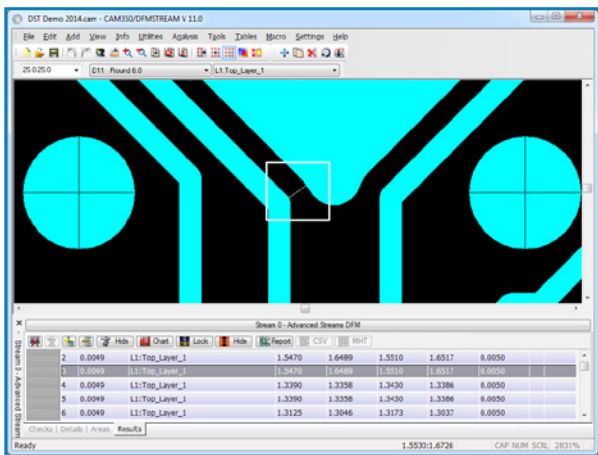
Design Analyzer

Use Design Analyzer to correlate PCB features such as trace width/spacing, number of layers, board size, and drill/via technologies to the capabilities of your preferred PCB fabricator. This guarantees submitted designs will be fabricated without hidden costs or unexpected delays. Send a Design Analyzer design report to a PCB fabricator to arrive at cost and delivery estimates for the fabricated PCB. Working
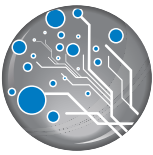
collaboratively with the report in hand, PCB fabricators can make recommendations for design changes that may result in significant cost and time savings while maintaining design intent.

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.

Use DFM Analysis to locate potential acid traps

Design Rule Checking

Use Design Rule Checking (or 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. Compare a design to an externally generated IPC-D-356 netlist. 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 (or reusing) the recorded sequence repeatedly. A CAM350 macro can interact with the

user, make conditional decisions, and parse information from the database. A macro debugging tool and macro encryption are also provided. 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.

Part Editing

Use part editing features to modify or manage part definitions created during CAD data import. Part definitions are most useful when reverse engineering design data from unintelligent sources such as Gerber, into a more CAD compliant database with part definitions, padstacks, nets, vias and so on.
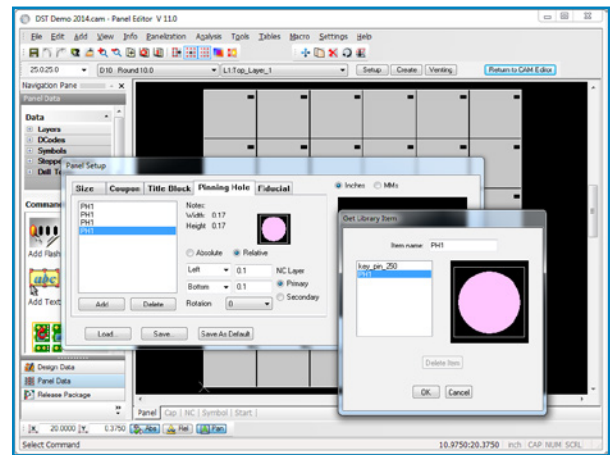
Use the Panel Editor to create custom PCB panels

PCB Panel Design

Create a multi-PCB image panel quickly using the panel design features. Choose the automated panel wizard, enter a few basic parameters, and the panel layout with minimal material waste is designed for you. Use the design merge features and create custom panels with images from different PCB designs. Create panels with test coupons, pinning holes,
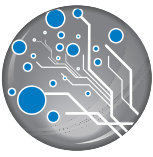
fiducials, robber bars, negative venting, and other panel constructs. The original one up image of the PCB design is preserved separately from the panel design. Add mill paths, mill tabs to create a custom routing design for the panel. Export the completed panel design to Gerber, DXF, Mill and Drill, IPC-2581 or ODB++.

Export to Multiple Formats

Design data in CAM350 can be exported to most popular PCB CAD systems including Mentor Graphics PADS or Xpedition; Cadence Allegro or OrCAD PCB Designer; Zuken's CR5000, CR8000, or CADstar; Altium Designerand several others. Industry standard file format support includes ODB++, IPC-2581, Gerber, DXF, Excellon, and Sieb & Meyer.

Reverse Engineering

Use Reverse Engineering features to convert non-intelligent design data such as Gerber back into an intelligent CAD format. Basic draws and flashes are converted to traces, vias, padstacks, footprints and nets. Use the new, more intelligent database to revise, process analyze and process the design for fabrication of the new revision.
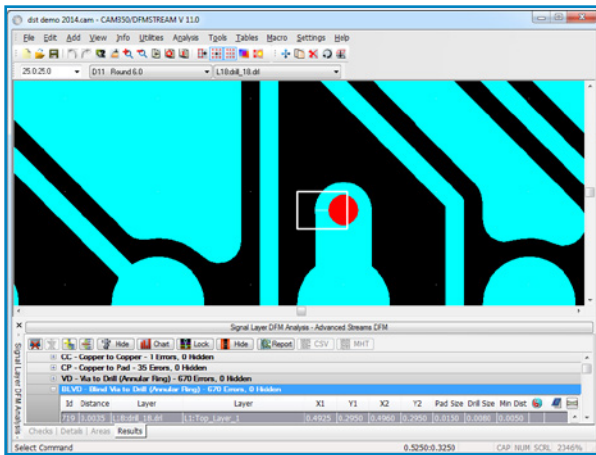
Use DFM Analysis to locate blind via related errors

DFM Analysis

Analyze your design for critical design flaws that commonly lead to fabrication and assembly delays. Choose basic analysis for acid traps, copper slivers, mask slivers, solder bridging, antennas, pin holes, thermal relief errors and other basic checks. Expand to advanced analysis and add greatly expand the breadth and depth of analysis to include plated or unplated holes, copper, blind and buried vias, back drills, more mask analysis, and a multitude of advanced analysis on intelligent design data. Explore our DFM Stream product from more details.

Flying Probe Editing

Use Flying probe features to locate bare board test point and fixture alignment pin locations. Use automated features to assign test points by pad style (SMD or through hole), by Dcode, by net, or other criteria. Use the interactive features to selectively choose the test point locations. Export the test point data to popular file formats including Probot, ATG, Integri-test, PROBOTECH, Microcraft, and IPC-D-356 as well as our own ACT neutral format.

Bed of Nails Editing

Use Bed of Nails features to locate test points and export the data to facilitate test fixture fabrication and testing. Assign test points for single sided or double-sided (a.k.a clamshell testing) fixtures. Define fixture size, probe sizes, fixture drill sizes, drill deflection, and other fixture properties. Use automated features to assign test points by pad style (SMD or through hole), by Dcode, by net, or other criteria. Use the interactive features to selectively assign test point locations. Export the test point data to popular file formats including TTI (Test Technologies International), Circuit-Line, IPC-D-356, IPC-D-356A, as well as generic plate drill files and netlists.

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

www.downstreamtech.com   sales@downstreamtech.com