

CAM350®

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.

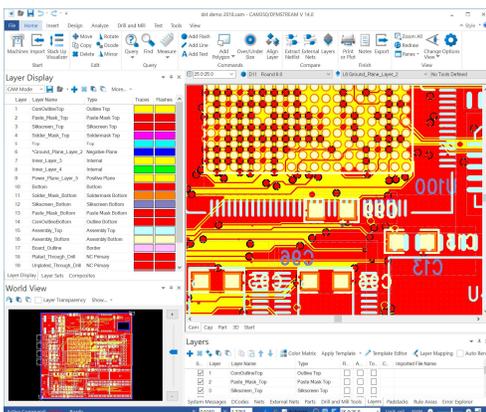
CAM350-160

CAM350-160 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

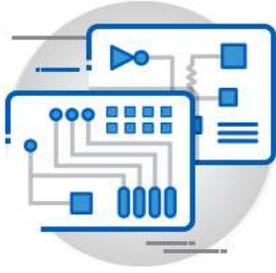
CAM350 functionality includes:

- Import of common CAM and NC 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 and teardrops
- Design Rule Checking (DRC) to check for minimal spacing errors, minimal annular rings, minimal feature sizes, and other anomalies
- 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
- 3D viewing for enhanced visualization of design data
- PCB Stackup design with user defined materials table and modeling in 3D
- Optional 3D PDF printing for sharing fully modeled PCB data
- Part editing to create custom footprints (or parts) required for reverse engineering Gerber into intelligent CAD data
- Export functions to package your completed design in standard manufacturing formats including Gerber, ODB++, IPC-2581, NC Drill and Mill and other formats
- Automation API with Record, Playback and embedded VB Editor to automate CAM350 interactions
- Quickly create fabrication panels using the PCB Panel Wizard



CAM350 offers a robust CAM editing environment

**PCB CAD data import available for an additional purchase



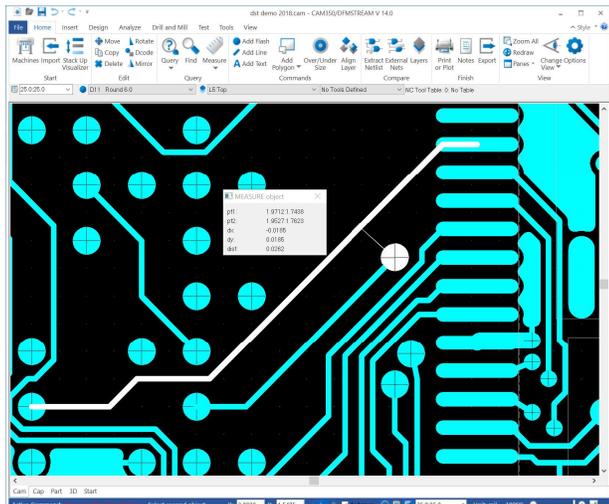
CAM350[®]

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 CAM and NC data in industry standard file formats including ODB++, IPC-2581, Gerber, Excellon, Sieb & Meyer, DXF or HPGL. Input data directly from Mentor Graphics via PADS ASCII or GenCAD**.



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, part, drill or mill tool. 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.

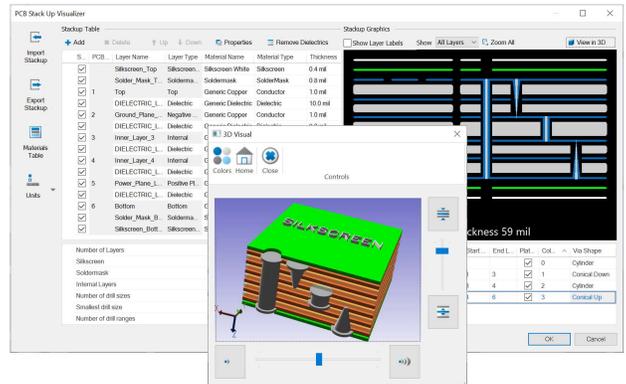
**Available for an additional purchase

Design Rule Checking

Use Design Analysis 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 graphical error charting to visualize violation thresholds.

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.

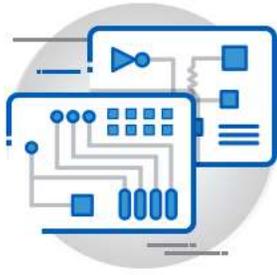


PCB Stackup Design and Visualization

Use Stackup Visualizer to design, view or modify a PCB stack up in multiple visualization modes including tablature, cross section and 3D. Drag and drop layers to interactively change layer order. Assign construction materials and set attributes such as layer thickness or dielectric constant. Interactively define and view via drill spans and via technologies including back-drills. Use export and import functions to exchange stack up design details with a manufacturer and update the stackup with material or other changes from the manufacturer.

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.



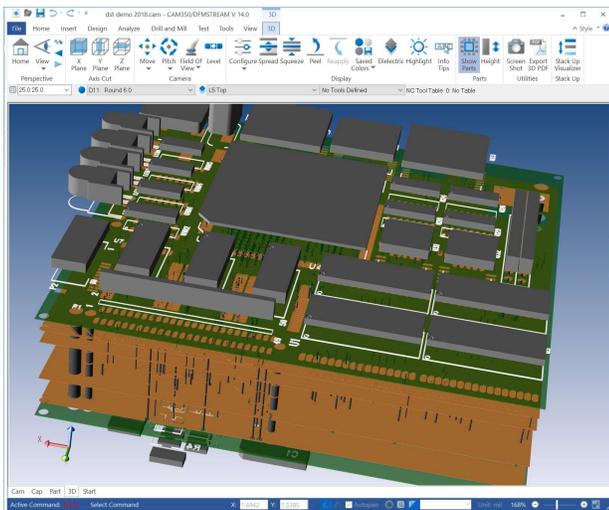
CAM350[®]

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.

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 and the equivalent element is selected in CAM350. After a CAM350 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 Allegro or OrCAD PCB Designer.



Use 3D Visualization to view a virtual model of a PCB

3D Visualization

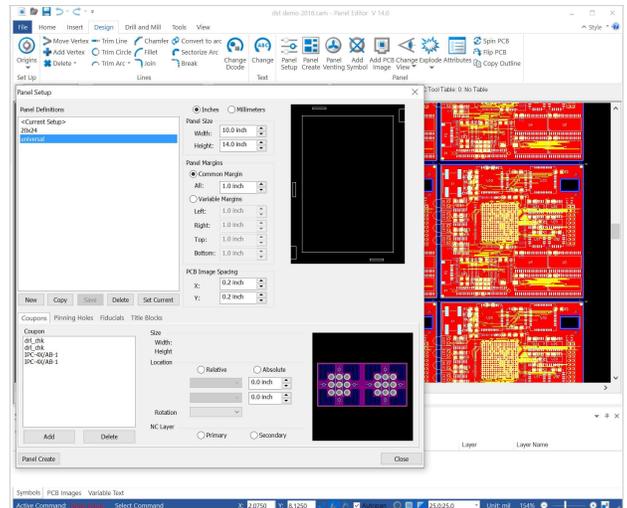
3D manufacturing data visualization provides a virtual model of the PCB to predict how a designers manufacturing data will be integrated and constructed into a finished PCB. The use of 3D provides a more robust environment to interrogate PCB manufacturing data as users can explore what the finished PCB will look like at any angle including between layers when compared to traditional 2D technology.

Export to Multiple Formats

Design data can be exported to industry standard file formats including ODB++, IPC-2581, Gerber, DXF, Excellon, Sieb & Meyer or HPGL. Additionally, export to Mentor Graphics PADS ASCII or GenCAD**.

Automation API with Record, Playback and Editor

Use the COM based API of CAM350 to record a sequence of often used commands and other operations as a visual basic script. Use the embedded VB editor to view, playback or customize the recorded script. Use Visual Basic or other common languages like C or C++ to create custom applications with dialogs, make conditional decisions, or to parse information from the CAM350 database.



Use Panel Editor to create custom PCB panels

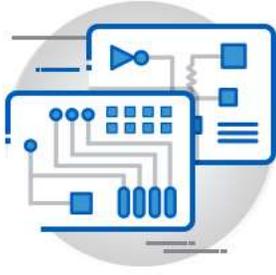
Netlist Compare

Use the Netlist compare features to verify the integrity of design data extracted from the PCB CAD tool. Generate a CAM netlist derived from parsing 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.

**Available for an additional purchase



CAM350[®]

Print to 3D PDF

Users can export a 3D PCB model into standard Adobe 3D PDF. Once exported, the 3D document can be shared with design team members or manufacturers for feedback. Adobe PDF viewers offer the ability to interact with the 3D PCB model through camera changes, pan and zoom, and navigating the model hierarchy**.

PCB Panel Wizard

While focused primarily on analysis, DFMSStream also offers multi-image PCB panel design to quickly create a multi-image PCB panel. Use the automated panel wizard, enter a few basic parameters to have the panel layout designed for you with minimal material waste. Use the design merge features and create custom panels with images from multiple PCB designs.

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:

www.downstreamtech.com/worldwide-distribution.php

DownStream Technologies, LLC

290 Donald Lynch Blvd
Suite 301
Marlborough, MA 01752
(+1) 508 970-0670

sales@downstreamtech.com
www.downstreamtech.com

Revised 01/05/2022