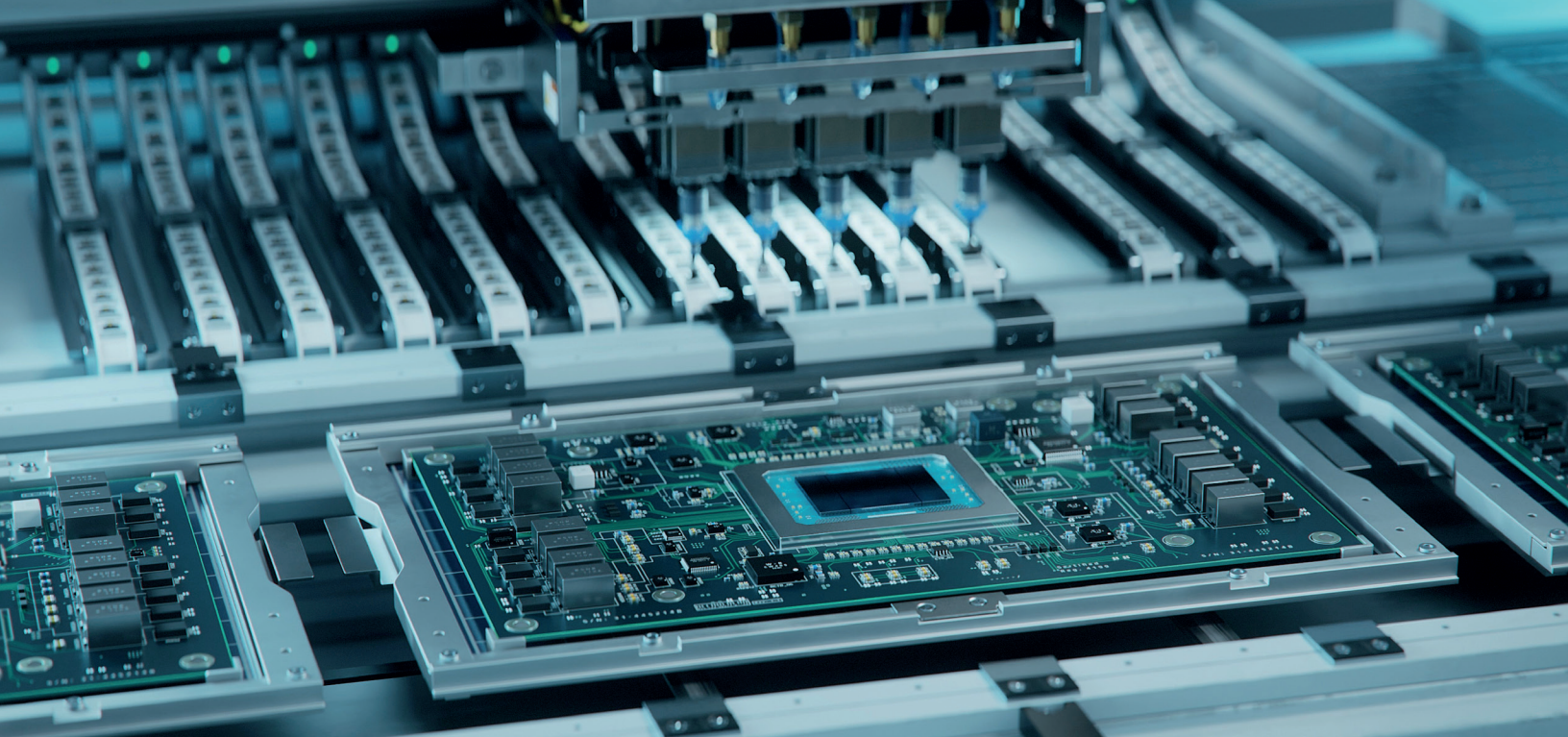




DIGITAL INDUSTRIES SOFTWARE

CAM350

Uncompromised PCB design processing and analysis

Benefits

- Streamlines the transition of engineering data into physical PCBs
- Compiles PCB design data into usable PCB manufacturing data
- 3D viewing for enhanced visualization of design data
- Analyze designs for potential flaws that may lead to fabrication delays

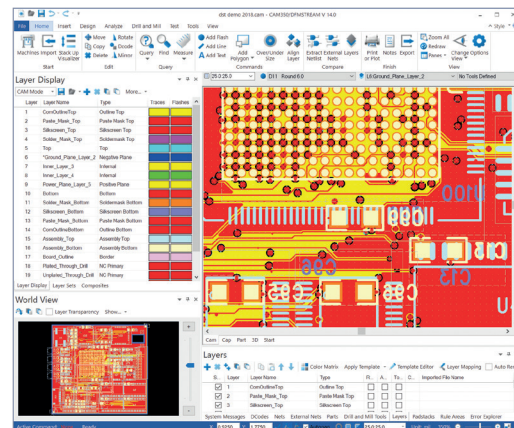
Features

- 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

Summary

CAM350 manufacturing data preparation solution is a superior collection of functionalities to import, modify, optimize and analyze your PCB design data to ensure timely, high-quality PCB fabrication.

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.



CAM350 offers a robust CAM editing environment.

SIEMENS

siemens.com/eda

Features *continued*

- CAM editing to add or modify flashes, pads, polygons, lines, text or and 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 silk-screen 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
- 3D PDF printing for sharing fully modeled PCB data
- Design for Manufacturing (DFM) analysis to analyze your design for potential flaws that may lead to fabrication delays
- DFM analysis for Rigid-Flex designs including trace fracture potential, coverlay anomalies and inter-layer dependencies
- Part editing to create custom footprints (or parts) required for reverse engineering Gerber into intelligent CAD data
- Reverse engineering to convert unintelligent design data, such as Gerber to intelligent CAD data with parts, padstacks and nets
- Design custom paste mask stencils

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.

Core features and functionality

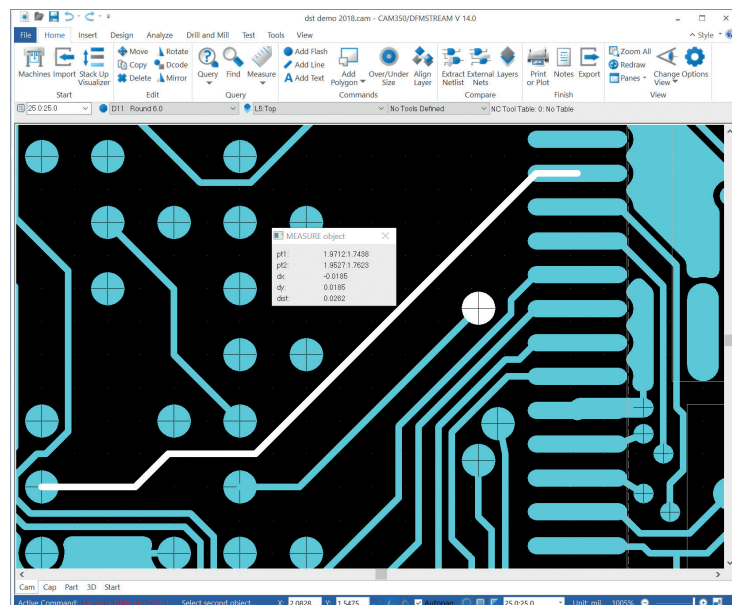
Successful compilation of PCB design data into usable PCB manufacturing data is critical to minimize delays in delivering new electronic products to market. CAM350 offers all the functionality you need to ensure high quality outputs to manufacturing that result in higher manufacturing yields and shorter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

Import data from multiple sources

Import CAM and NC data in industry standard formats, including ODB++, IPC-2581, Gerber, Excellon, Sieb & Meyyer, PADS STD ASCII, GenCAD, DXF, and HPGL.

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.



Use the measuring tool to query distance between objects.

Features *continued*

- 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
- Automation API with Record, Playback and embedded VB Editor to automate CAM350 interactions
- 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

NC editing

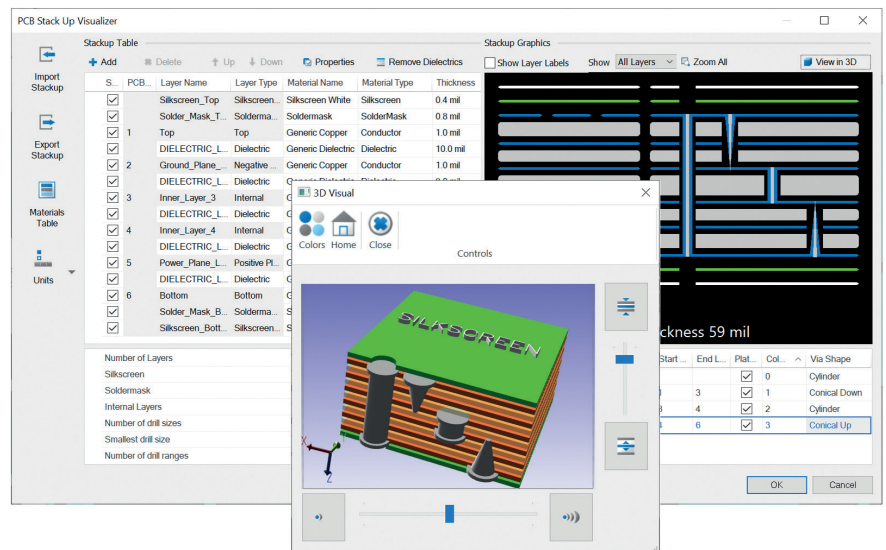
Use NC Editing features to import or export NC drill and mill data. Add drills, slots, mill paths and mill tabs for panel separation. Modify drill and mill tool definitions. Create custom NC drill files for specific drilling operations. Convert Gerber content to NC content and NC to Gerber.

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.

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 backdrills. 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.



Use Stackup Visualizer to design, view or modify a PCB stack up.

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.

PCB CAD cross-probing

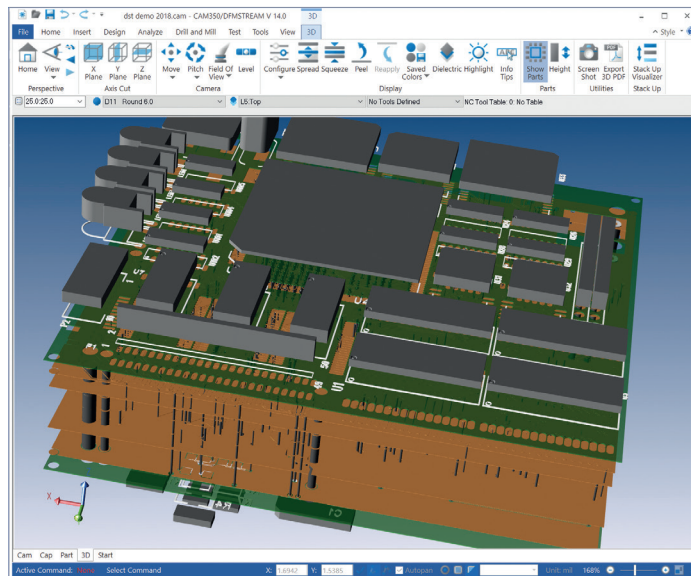
Cross-probing facilitates visualization of CAM350 design data to its source PCB Design in its native PCB tool. Select elements in the PCB design tool and the equivalent element is selected in CAM350. After a CAM350 DFM Analysis, use cross-probing to visit error locations in the PCB tool to make corrections. This expedites the process of finding and correcting errors in the source PCB design. CAM350's cross-probing is compatible with leading PCB CAD tools, such as Xpediton, Cadence Allegro, and OrCAD PCB Designer.

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 one or many preferred PCB fabricators. These guarantee 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 significantly reduce cost and fabrication time.

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.



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

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.

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++.

Netlist compare

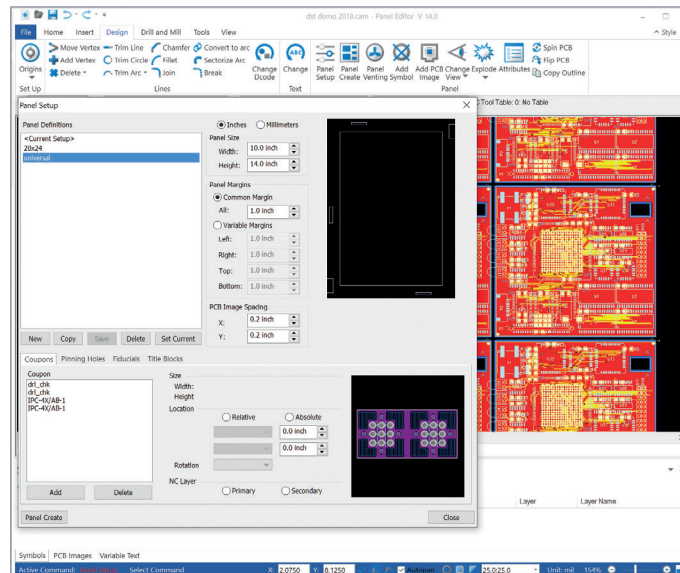
Use the Netlist compare features to verify the integrity of design data extracted from the PCB CAD tool. Generate a CAM netlist derived from parsing the data in CAM350 and compare it to an IPC-D-356 netlist exported from the source PCB CAD tool. This comparison ensures the design data was not compromised 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, pad stacks, nets, vias and so on.

Export to Multiple Formats

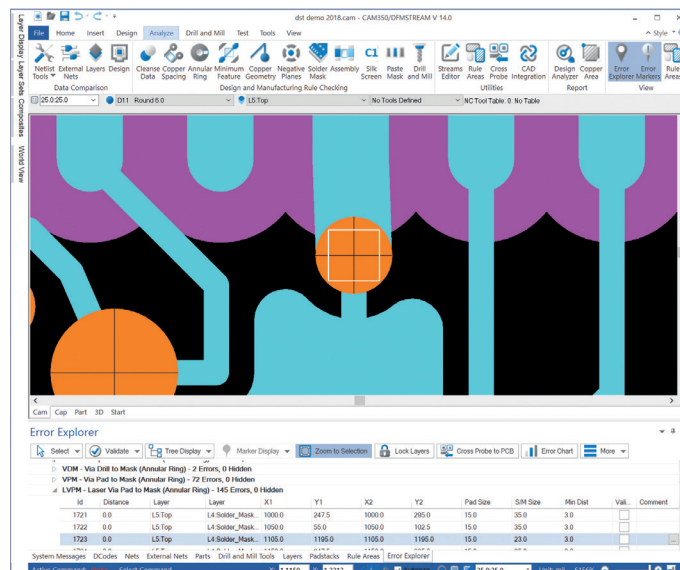
Design data can be exported to industry standard formats, including ODB++, IPC-2581, Gerber, Excellon, Sieb & Meyer, PADS STD ASCII, GenCAD, DXF, and HPGL.



Use Panel Editor to create custom PCB panels.

DFM analysis

Analyze your design for critical design flaws that commonly lead to fabrication and assembly delays. Perform basic analysis for acid traps, copper slivers, mask slivers, solder bridging, antennas, pin holes, thermal relief errors and other basic checks. Perform advanced analysis on intelligent design data to include plated or unplated holes, copper, blind and buried vias, back drills, flex and rigid-flex and a multitude of other analysis.



Use DFM Analysis to locate laser via related errors.

DFM analysis for rigid-flex designs

Analyze a rigid-flex design for trace fracture potential including trace corners, solid copper or trace width transitions in bend areas. Additionally, analyze for coverlay anomalies including missing exposures for button plating, coverlay minimum spacing and inter-layer dependencies.

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.

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, pad stacks, footprints and nets. Use the new, more intelligent database to revise, process analyze and process the design for fabrication of the new revision.

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.

Stencil designer

Create custom paste stencils from Gerber, PCB footprints or a stencil library. Store stencil configurations in a user defined stencil library and reuse stencil designs across multiple PCB designs. Create paste mask shapes such as homeplate, bullet, DPack, snubnose and many other commonly used stencil shapes.

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.

**Siemens Digital
Industries Software**
[siemens.com/software](https://www.siemens.com/software)

Americas (USA)
1-800-498-5351

EMEA (United Kingdom)
0800-279-0464

APAC (India)
1-800-202-6796

For additional numbers, click [here](#).

© 2026 Siemens. A list of relevant Siemens trademarks can be found [here](#). Other trademarks belong to their respective owners.

87250-D2 3/26 H