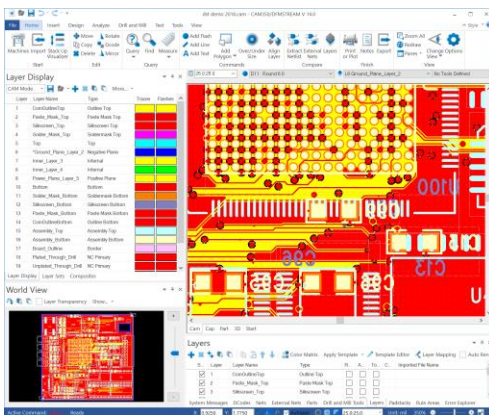


Uncompromised PCB Design Processing and Analysis

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. PCB Verifier 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, PCB Verifier streamlines the transition of engineering data into successful, physical PCBs.



PCB Verifier offers a robust CAM editing environment

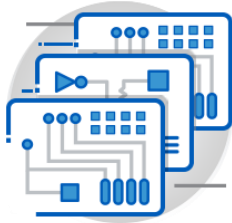
DFMStream PCB Verifier

DFMStream PCB Verifier 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

DFMStream PCB Verifier functionality includes:

- Import of Gerber and NC file formats and intelligent formats of ODB++ and IPC-2581
- 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
- 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
- Custom aperture editing to create custom pad shapes or custom polygonal areas
- 3D viewing for enhanced visualization of design data
- PCB Stackup design with user defined materials table and modeling in 3D
- Export functions to package your completed design in standard manufacturing formats including Gerber, NC Drill and Mill and other formats
- Less than minimal spacing between design objects including pads, tracks, copper, drills and vias
- Less than minimal annular rings of pad, copper, or mask
- Less than minimal spacing between SMD or Through hole pads or parts
- Copper and mask slivers and pin holes
- Isolated thermal, starved thermals and minimum tie width on plane layers
- Acid traps, solder bridge potential, isolated or starved thermal reliefs or trace antennas
- Minimal mask spacing, missing paste, missing solder mask, extra mask areas
- Overlapping, coincidental or redundant drills, mill path errors and poor drill to board thickness ratio
- Design differences between design layers
- And many other error types

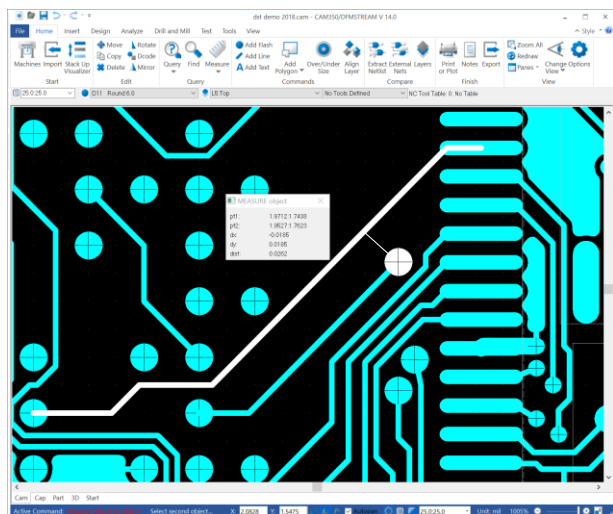


Core Features and Functionality

Successful compilation of PCB design data into usable PCB manufacturing data is critical to minimize delays in delivering new electronic products to market. PCB Verifier 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 and Sieb & Meyer.



Use the measuring tool to query distance between objects

View, Query, Measure and Report

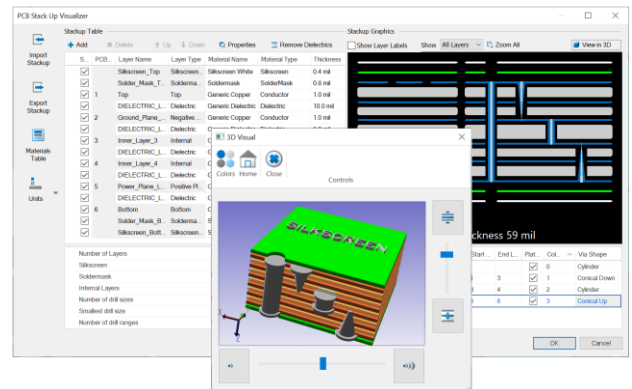
Filter your view of the design data by layer, Dcode, net, part, drill or mill tool. Query any element to get pertinent details such as size, shape, area, Dcode, design layer, drill tool, and so on. Get details specific to each type of element. Measure the distance from point to point or object to object.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

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 Stackup Design and Visualization

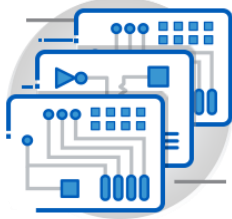
Use Stackup Visualizer to design, view or modify a PCB stack up in multiple visualization modes including tabulation, cross section and 3D. Drag and drop layers to interactively change layer order. Assign construction materials and set attributes such as layer thickness or dielectric constant. Interactively 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.

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.

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 PCB Verifier 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.



DFMStream™

Design Rule Checking

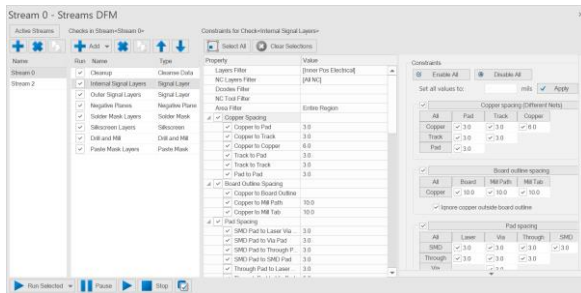
Use Design Analysis to analyze your design for violations of minimal spacing, minimal copper or mask annular rings, minimal pad, gap and track sizes and redundant pads. Locate plated drills without pads, pads without drills or drill to copper violations. Compare layers for differences. Calculate copper area per layer. Use graphical error charting to visualize violation thresholds.

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.

PCB CAD Crossprobing

Crossprobing facilitates visualization of 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 PCB Verifier. After a 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. PCB Verifier's crossprobing is compatible with leading PCB CAD tools such as Mentor Graphics' PADS or Xpedition; Cadence Allegro or OrCAD PCB Designer.



"Streams" Driven Analysis

Manage the myriad of checks and the analysis process using the Streams methodology. Checks are organized into layer types and sub categories to simplify selection of checks to perform and setting the corresponding parameters. Use the Streams methodology to define the type and order or "Stream" of checks to be performed. A stream of analysis can include netlist or layer comparison, design rule verification, fabrication and assembly checks on the entire design, a specific layer or a region of the design. This dramatically reduces set up and execution of the analysis. Analysis streams can be saved and recalled for use on any design. Streams definitions can be defined for a specific PCB technology, vendor capability, or unique design requirement.

Hierarchical Analysis

For many designs, different regions or layers of a PCB have unique constraints and subsequently require a custom analysis. Use PCB Verifier's unique rule hierarchy to tailor an analysis to bare board construction, board density, or component technology.

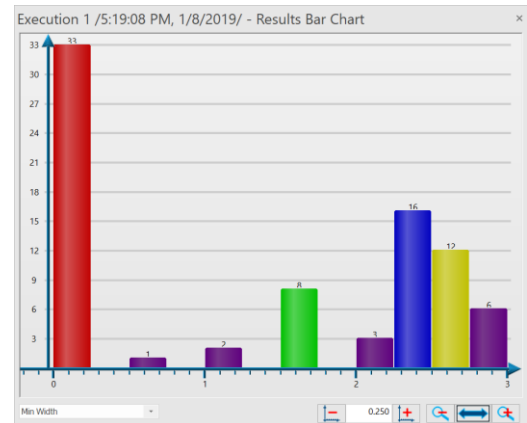
PCB Fabrication Analysis

PCB Verifier's analysis will detect specific design content that may have an adverse effect on PCB fabrication. Features less than minimal spacing, acid traps, minimal annular rings, minimum features sizes, copper and mask slivers, are just a few examples of the fabrication analysis available. Drill related analysis include minimal distances between drills, pads without drills, mill path errors, coincident or overlapping drills and others.



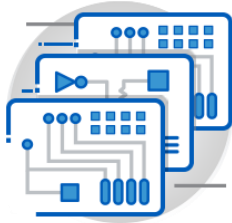
Selective Analysis

Use Analysis features individually outside of a Stream to focus on specific DFM concerns. Run an annular ring analysis on the entire design or just a region of the design. Run a Negative plane analysis on a one or more plane layers.

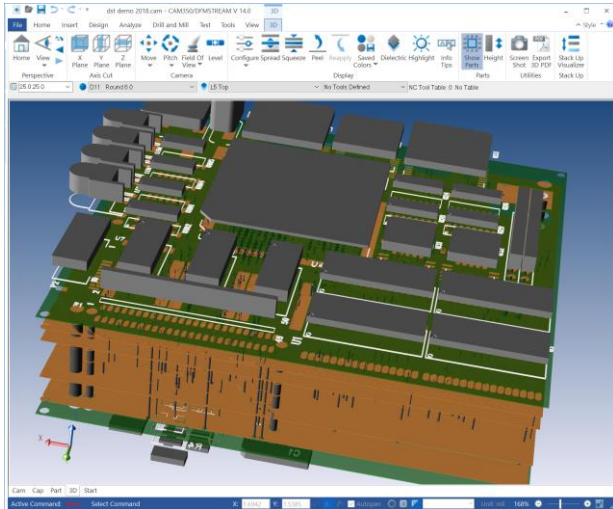


Error Charting

Large scale analysis can often result in a large number of reported failures. Viewing the results of large scale analysis in chart form allows you to get to the root of the failure and quickly ascertain a remedy. Charting allows you to review specifics of the failures to identify trends or unexpected results. PCB Verifier's charting feature reports the exact nature of the error and the PCB features related to that error. The charting function groups common errors so they can be quickly identified and resolved in PCB Verifier or the source PCB CAD tool.



DFMStream™



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 Gerber, Excellon, and Sieb & Meyer.

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 08/16/2023