DownStream Technologies

Solutions for Post Processing Printed Circuit Board Designs

DownStream provides a comprehensive yet economical solution to PCB post processing that enables engineers and designers to quickly create the key deliverables for PCB fabrication, assembly and testing.

DownStream's tools combine powerful feature sets to handle complex designs with a very easy-to-use interface, allowing even the infrequent user the ability to generate PCB artwork, design validation, and bare board and assembly drawings and documents.

CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

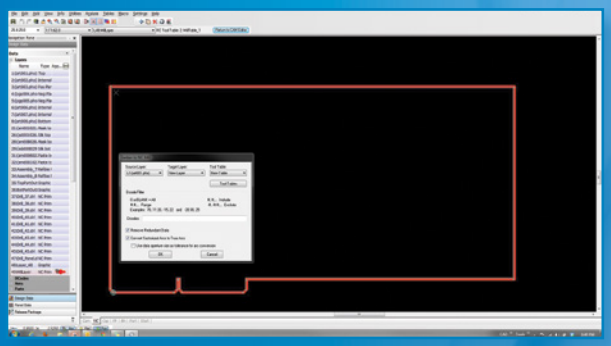
Today's complex PCB designs require comprehensive verification before they are transferred to the fabricator. This helps ensure the successful and timely manufacture of your bare boards. Problems arising during PCB fabrication can drastically impact product schedules and result in costly design re-spins. Sometimes this can require design modifications that compromise the integrity and intent of the original design. Inspecting, validating, and preparing the PCB design prior to release to manufacturing will result in a significant increase in efficiency, less risk of design re-spins, and most importantly, successful electronic products built faster and at less cost.

Features and Functionality

CAM350 offers everything necessary to import, export, optimize, modify and verify your design. With robust add-on modules, you have the option to perform comprehensive analysis quickly and easily, intelligent data transfer, enhanced engineering preparation and inspection, and much more. The Design Analyzer streamlines communication between you and your fabricator well before they have your design files. This ensures they are qualified to build your design and your design files meet their internal requirements. These additional practices will save you time and money, while allowing you to create cutting edge electronic products.

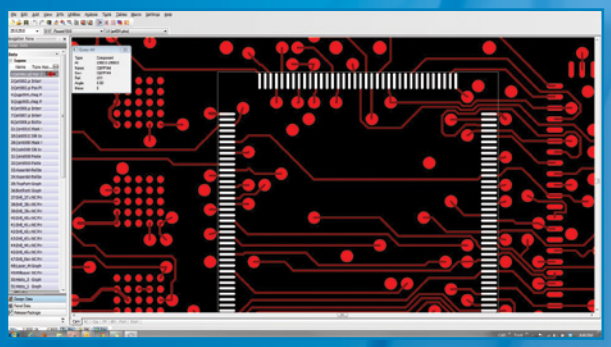
- Import and export data using intelligent data exchange options
- Seek out, identify, and repair design and manufacturing flaws
- Inspect for etching, soldermask, thermal, and spacing violations
- Avoid manufacturing complications and delays
- Optimize panelization, tooling, mill and drill, and more
- Verify and maintain design integrity and intent
- Affordable and easy to learn, use and implement

Powerful and versatile, CAM350 offers a complete solution to streamline the transition of engineering data into physical PCBs, ultimately resulting in successful electronic products.



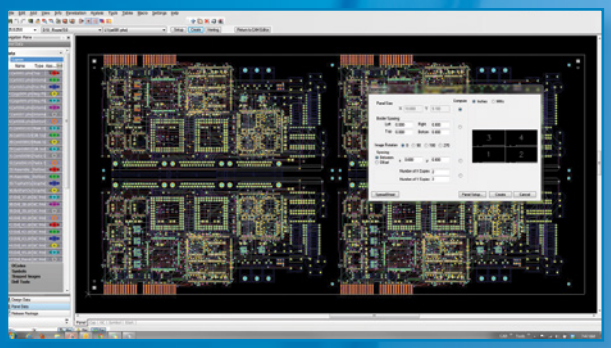
Gerber to Mill

Automatic creation of Rout (Mill) programs from Gerber files.



Information

Query intelligent data if present, not just shapes and sizes.



Fast Array

Quickly optimize board placement for assembly panels.

BluePrint-PCB

Create Comprehensive PCB Documentation to Drive PCB Fabrication, Assembly and Inspection

BluePrint-PCB is the premier PCB document authoring solution to create the manufacturing specifications for an electronic product. Comprehensive documentation records the engineering "intent" of a design specifying the form, fit and function of the PCB. Successful PCB documentation drives the procurement process, aids manufacturing engineering, and is used in final inspection to verify the product was built to specification.

Features and Functionality

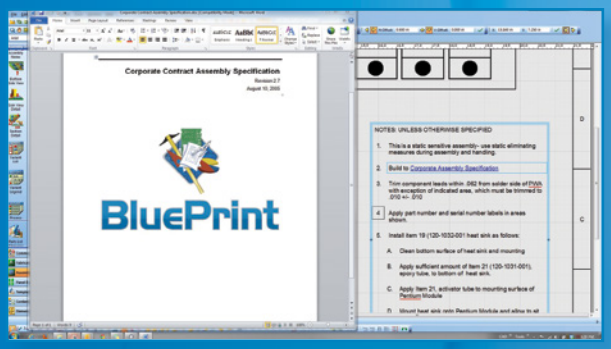
BluePrint imports the PCB CAD database to automatically create and link unlimited PCB views and details while maintaining the design intelligence. It uses a document and sheet-based approach allowing you to drag and drop PCB views, details, notes, and tables onto a drawing sheet, then arrange and format them as needed. BluePrint-PCB's patented technology allows all views, details and charts of the PCB to remain linked to the original CAD data. Whenever the original data is changed, the information in BluePrint-PCB is refreshed and updated, simplifying the ECO process.

BluePrint allows for easier navigation of documentation through the use of "active" pages. BluePrint creates live, interactive documentation that can also incorporate multi-media and hyperlinks. This structure allows you to easily view, comprehend, and use the documentation, while also simplifying the distribution across the supply chain. BluePrint offers state-of-the-art methods to view BluePrint documents, via the BluePrint application, in PDF, on the web, or with a stand-alone free viewer.

With BluePrint, PCB engineering groups can expect to reduce the documentation process to a fraction of the time it normally takes - even more when engineering change orders (ECOs) are required.

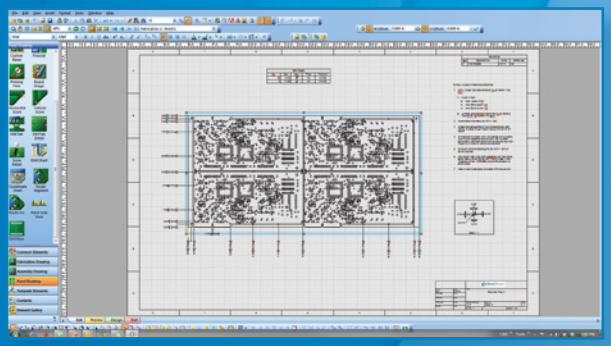
- Automates the creation of PCB documentation
- Increases documentation detail and accuracy
- Improves manufacturing instructions
- Simplifies manufacturing inspection
- Creates one electronic "release package"
- Eases PCB documentation distribution and use

BluePrint-PCB enhances and simplifies PCB documentation using automation and technology, easing the entire process from creation through distribution and usability.



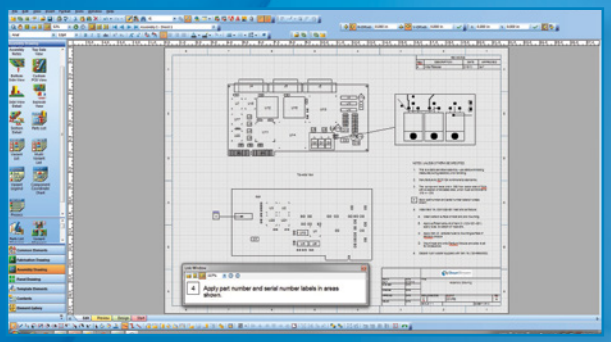
Hyperlink

Hyperlinks allow for simplified electronic documentation, conveying more information to the manufacturer than current methods.



Panel

The panel drawing pallet allows assembly panels to be created quickly and efficiently, bypassing the need for CAM tools to do step and repeat.



Top and Bottom view

BluePrint allows you to create and place unlimited views of the PCB, no longer are you confined by the limits of Electronic CAD tools.

DFMStream™

Verify Design and Manufacturing Rules Anytime during the PCB Design Cycle

Preparing PCB design data to be released to manufacturing is a critical yet often fragile step in the new product introduction process. Designs that work in a virtual PCB CAD system may unknowingly break critical manufacturing rules. When these rules are overlooked, manufacturing halts, delivery times get delayed, and work arounds are performed.

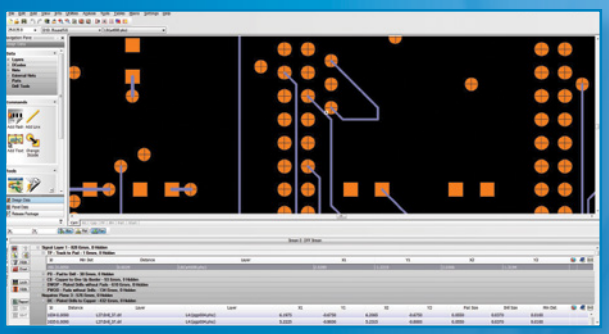
While manufacturers are fully capable of addressing these critical issues, resolutions are rarely fed back into the source CAD data resulting in repeated DFM violations on later runs. In worse case scenarios, design intent may also be unknowingly sacrificed when a third party alters a design. With DFMStream you can avoid all this, saving precious time and resources, simply by locating and amending manufacturing issues and barriers in advance.

Features and Functionality

DFMStream offers full verification of design and manufacturing rules on PCB design databases, Gerber, and NC Data to ensure the data complies with design intent, as well as the manufacturers requirements. With DFMStream, rule files may be saved and re-used in the future for specific vendor capabilities, greatly reducing set up time for DFM verification. DFMStream was designed for the user who understands the importance of analysis and wants to conduct it in a robust environment, easily and affordably. DFMStream offers intelligent data exchange options and many powerful features.

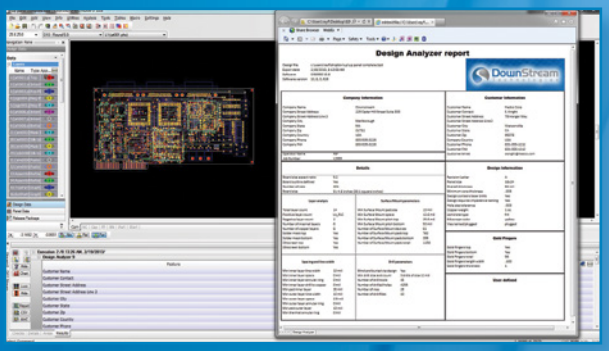
- Comprehensive, fast, design data analysis and rule checking
- Clean reporting and graphical displays to simplify analysis results
- Unique features to streamline quoting, manufacturing, and communication
- Quick to learn, install, and use, for even the most novice users

DFMStream is the only design for manufacturing solution that offers affordable, thorough, easy, fast analysis, in a clean robust environment.



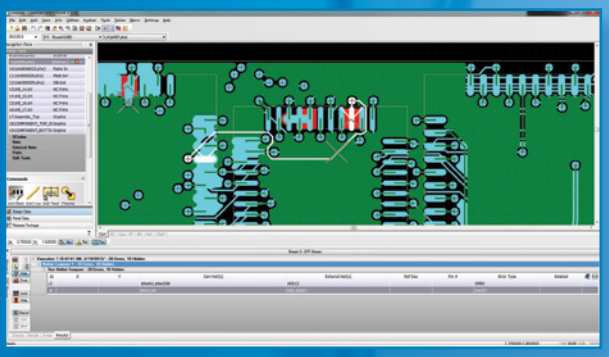
Streams

DFMStream includes Streams RC for robust rule checking for common design rule errors, as well as potential manufacturing problems.



Design Analyzer

The Design Analyzer extracts the data necessary to properly quote and manufacture a PCB, streamlining communication with fabrication.



Streams Netlist Compare

Graphical netlist compare with color coding helps you find the location of the error in the design data, not just on a report.

DownStream Technologies

Tel: 800.535.3226 • 508.970.0670 • www.downstreamtech.com

©2013 DownStream Technologies, LLC. All rights reserved. CAM350®, BluePrint PCB® and DFMStream® are registered trademarks of DownStream Technologies, LLC. All other trademarks are the property of their respective owners.