cadence®

Allegro PCB Designer Manufacturing Option

Powerful DFM checker, efficient documentation process, and intelligent panel design

Cadence® Allegro® PCB Designer Manufacturing Option is a comprehensive, powerful, easy-to-use suite of tools that makes it efficient and cost effective for PCB designers to streamline the development of a release-to-manufacturing package for their products. The Manufacturing Option includes three modules: Design for Manufacturing (DFM) Checker, Documentation Editor, and Panel Editor.

Finalizing the PCB design data for release-to-manufacturing is a critical and often fragile step within the new product introduction (NPI) process. The Allegro PCB Designer Manufacturing Option and its three key modules—DFM Checker, Documentation Editor, and Panel Editor—assist designers with an efficient and successful handoff to manufacturing by ensuring that the design adheres to all manufacturer's rules, and that the fabrication assembly intent is clearly specified in the PCB manufacturing documentation package.

DFM Checker

Manufacturing flaws discovered by manufacturers prior to production result in costly time-to-market delays as designs need to be updated and reprocessed to address these issues. While manufacturers are fully capable of addressing minor issues, their resolutions are rarely fed back into the source CAD data, resulting in same set of iterations of modifications on future design revisions. In worst-case scenarios, design intent can unknowingly be sacrificed when the manufacturer alters the source design files prior to production.

The Manufacturing Option's DFM Checker module is a powerful, yet easy-to-use suite of manufacturing analysis toolsets. It is designed for engineers and designers who appreciate the benefits of manufacturing analysis and want to conduct it in a robust environment, with ease and sensibility at all phases of the PCB design process. DFM Checker offers comprehensive analysis for all major PCB design tools, Gerber files, intelligent manufacturing files, and NC data to ensure the content supplied to the manufacturer will minimize costly delays.

DFM Checker identifies design content with the potential to result in low manufacturing or assembly yields, or costly scrap, including:

- Insufficient spacing between design objects including pads, tracks, copper, drills, and vias of all types including blind, buried, laser, and back drilled
- Insufficient annular rings of pad, copper, or mask
- Insufficient spacing between SMD or through hole pads or parts
- Copper, mask slivers, and pin holes
- Acid traps, solder bridge potential, isolated or starved thermal reliefs, and trace antennas
- Insufficient mask spacing, missing paste, missing solder mask, extra mask areas, or poor mask-to-pad ratios
- Overlapping, coincidental, or redundant drills, mill path errors, and poor drill-to-board thickness ratios

Design Analyzer

Design Analyzer correlates features within a PCB design—such as minimum trace width/spacing, number of layers, board size, and drill/via technologies—to the requirements of your preferred PCB fabricator. This correlation guarantees submitted designs will be fabricated without hidden costs or unexpected delays. Design Analyzer's design reports contain all of the information required by a PCB fabricator for cost and delivery estimates of the fabricated PCB. Working collaboratively with the

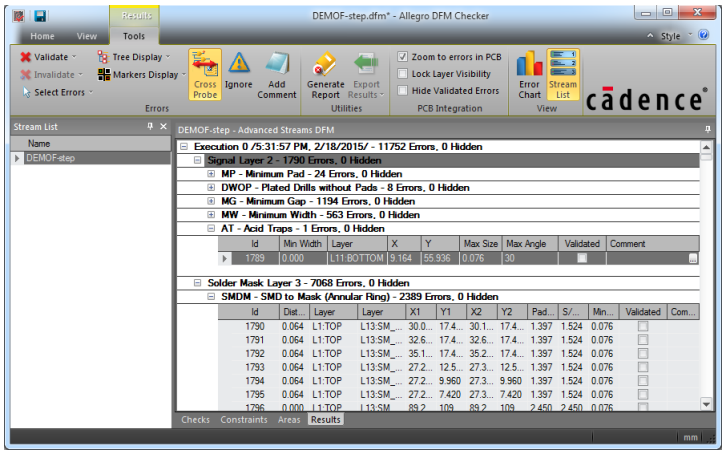

Figure 1. Use of hierarchical rule sets helps identify defects, avoiding costly bare board scrap

report in hand, PCB fabricators can make recommendations for design changes that result in significant cost and time savings while maintaining design intent. Design Analyzer bridges the gap between engineer and PCB fabricator by extracting key information about the PCB and presenting it in an easy-to-read format.

Hierarchical Rule-Set-Driven Analysis

Manage the myriad of checks and the analysis process by creating rules sets. Checks are organized into layer types and sub-categories to simplify the selection of checks to perform and the setting of corresponding parameters. Define the type and order of a group of checks (rule set) to be performed. 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 rule sets can be saved and recalled for use on any design. Rule sets can be defined for a specific PCB technology, vendor capability, or unique design requirement.

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

PCB Fabrication Analysis

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

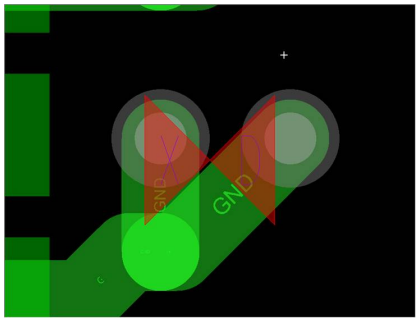

Figure 2. Acid trap violation

PCB Assembly Analysis

Assembly analysis will detect specific PCB content that might have an adverse effect on PCB assembly. Solder and paste mask features—such as less-than-minimal mask spacing, missing masks, extra mask, minimal mask annular rings, and mask-size-to-pad-size ratios—are examples of mask analysis available.

Silkscreen-related analysis includes ink over pads, ink over mask exposures, and other checks. Less-than-minimal part spacing, pad spacing between adjacent parts, and minimal spacing between component pads and adjacent vias, drills, and copper objects are examples of component-related analysis.

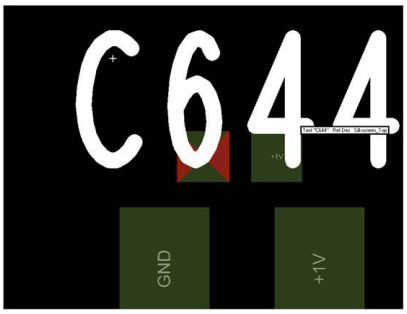

Figure 3. Silkscreen-to-solder-mask spacing violation

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. DFM Checker'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.

Crossprobing with Allegro PCB Editor

The Manufacturing Option's DFM Checker module is tightly integrated with Allegro PCB Editor to walk through errors. Select errors within DFM Checker and it zooms into the error location within Allegro PCB Editor. This expedites the process of finding and correcting errors in the source PCB design. DFM Checker adds DRC markers in Allegro Constraint Manager with details of the violation.

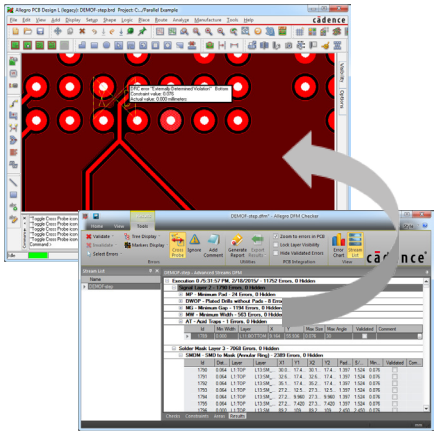

Figure 4: Crossprobing between errors and Allegro PCB Editor

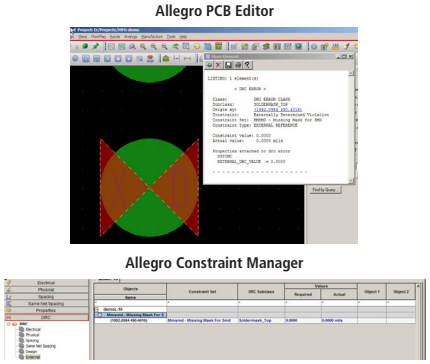

Figure 5. Missing solder mask

Documentation Editor

PCB documentation creates the manufacturing specification for an electronic product. Comprehensive documentation records the engineering "intent" of a design specifying the form, fit, and function of the PCB. Documentation drives the procurement process, aids manufacturing engineering, and is used in final inspection to verify that the product was built to engineering's specification. It also gets archived to ensure later production runs can be repeated with the same level of consistent quality. PCB documentation must capture all of the information necessary to not only build the product today, but also any repeat builds in the future—eliminating any guesswork.

The Manufacturing Option's Documentation Editor module is a PCB documentation-authoring tool that intelligently automates your documentation creation process to produce complex PCB documentation in a fraction of the time versus traditional methods. Documentation Editor enables you to quickly create the manufacturing drawings that drive PCB fabrication and assembly.

Utilizing the Allegro PCB CAD data, Documentation Editor creates intelligent linked PCB views, drawing details, document notes, drill charts, parts lists, and other crucial documentation details. The result is documentation that more accurately articulates instructions for the successful fabrication, assembly, and inspection of PCBs. The completed documentation release package contains all the data necessary to build, view, and archive the final product.

Documentation Editor includes the following key features:

- Creation and placement of unlimited views of the PCB. Each view can have its own display settings and can be formatted independently of the source data.
- Tight integration with Allegro PCB Editor enables simple click-through passing of design data to expedite the drawing creation process
- Intelligent automation saves hours, if not days, creating PCB drawings for fabrication, assembly, and inspection
- Automatic creation of component assembly views, drill pattern view, details, and parts lists derived from the Allegro PCB CAD data, eliminates manual drawing of tedious document elements
- Design changes made in Allegro PCB Editor are automatically propagated to all effected drawing elements, dramatically reducing time spent updating documentation due to the design changes

Drawing Creation

Standard or custom drawing elements are easily added by dragging and dropping from a palette of drawing-specific elements, such as sheet borders, title blocks, PCB views, drill charts, detail views, text boxes, and note blocks. Elements added to your drawings can be easily relocated, resized, and customized. Each element can be scaled, formatted, and transformed (rotated, mirrored, flipped, etc.) independent of each

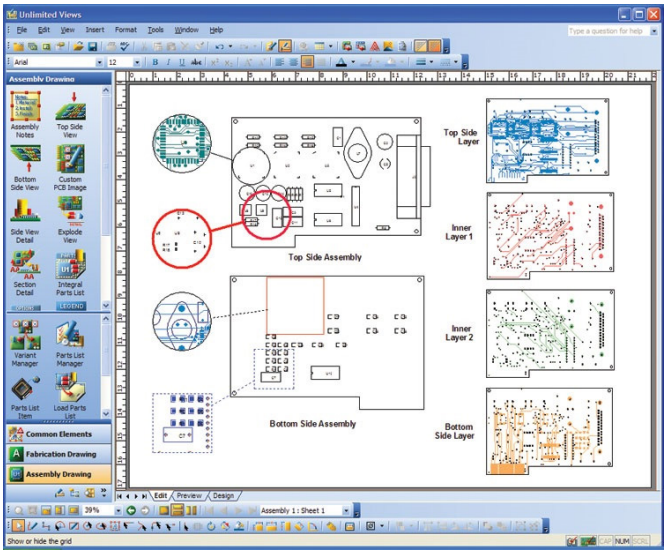

Figure 6. Produce complex PCB documentation in a fraction of the time

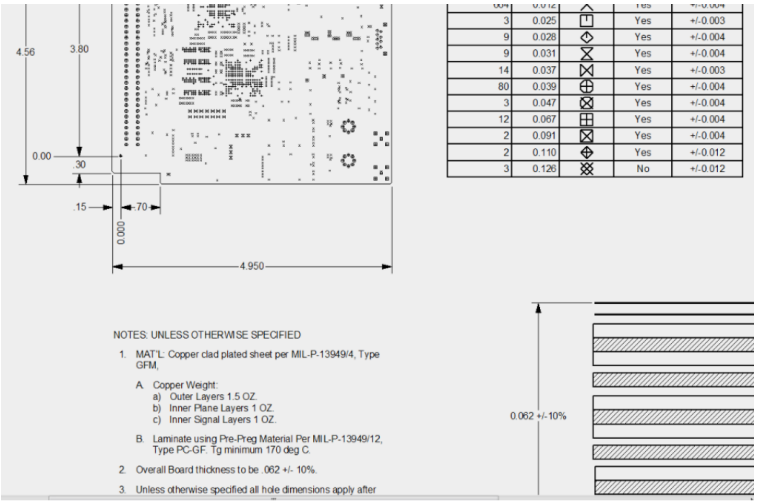

Figure 7. Drawing-detail wizards automate the layer stack-up, finger chamfer, and V-score drawing

other. There is no limit on the number of elements that can be placed on a drawing.

Fabrication and Assembly Drawing Tools

The drawing tools include support for adding PCB views to fabrication and assembly drawings. You can drag and drop views of the board outline, top/bottom side of the PCB, drill pattern, drill chart, integral parts list, component coordinate charts, and many others. Drawing detail templates automate the creation of detail views of via stack-ups, layer stack-ups, finger chamfers, V-score, and several other common drawing element types. Component-assembly PCB views can be filtered by variant, or can apply variant status in parts lists. You can also create assembly-process documentation for process-driven PCB assembly, and easily create color-coded drawings for each assembly process, process step charts, and process-based parts lists. Advanced capabilities also include drawing functionality for geometric dimensioning and tolerance (GD&T), coordinate dimensioning, blind/buried via support in via stack ups, PCB views, and drill charts.

Mechanical Component Support

Documentation Editor understands that PCBs are comprised of both electrical and mechanical components. Users can define any drawing item as a mechanical component and store it in a "gallery" for re-use. Mechanical components can also be associated to electrical components to help automate correct parts list quantity calculations, particularly when assembly variants are also used. Documentation Editor can construct a mechanical component in a number of different ways: Manually using Documentation Editor's drafting tools, by importing DXF objects from a mechanical system, or by importing any digital image file (JPEG, BMP, etc.) that the user wishes to depict as a mechanical component.

Design-Driven Documentation

Documentation Editor drawing content is derived directly from the Allegro PCB CAD data. Components, pins, assembly mounting sides, layers, vias, nets, assembly variants, part attributes, drill holes and symbols, and other critical design data are imported to ensure document accuracy. External content such as JPEGs, BMPs, GIFs, and TIFFs can be imported and incorporated in the drawings. OLE objects, audio, and video files can also be imported and linked within the documentation and stored with the documentation release package. In addition, Gerber and DXF files can be imported and exported.

ECO Updates Minimize Documentation Rework

All drawing elements in Documentation Editor are derived from the source Allegro PCB CAD data. Opening the revised design in Documentation Editor allows you to perform lightning-fast engineering change order (ECO) updates by simply refreshing the source computer-aided design (CAD) data. When refreshed, all drawing elements, such as PCB views, tables, details, parts lists, drill charts, etc. are updated with the new design data. All user customization for each element is maintained during the refresh, further minimizing documentation rework.

Pack and Release

Users can select drawings and files and compress them with an integrated viewer for use with users who do not have a viewer or do not wish to install one.

Template Customization Toolkit

The Template Customization Toolkit allows custom template creation and enables template tool pallet and format template dialogs.

Automation Toolkit

The Automation Tool Kit enables VB scripting tools to automate the process of creating PCB documentation. You can reduce the process of creating documentation to a few clicks for a PCB.

Panel Editor

Electronics manufacturers often panelize single-board designs into arrays or sub-panels to facilitate their PCB assembly process. PCB CAD tools are very good at designing the single-board design or "one-up" PCB, but lack the necessary functionality to design the varied complexities and details that are required for panels. Documenting the finished panel and adding features such as pinning holes, fiducials, score lines, and breakaway tabs can be difficult and time consuming in today's CAD and computer-aided manufacturing (CAM) tools.

The Manufacturing Option's Panel Editor module intelligently automates the complex process of panel definition and documentation, simplifying the design

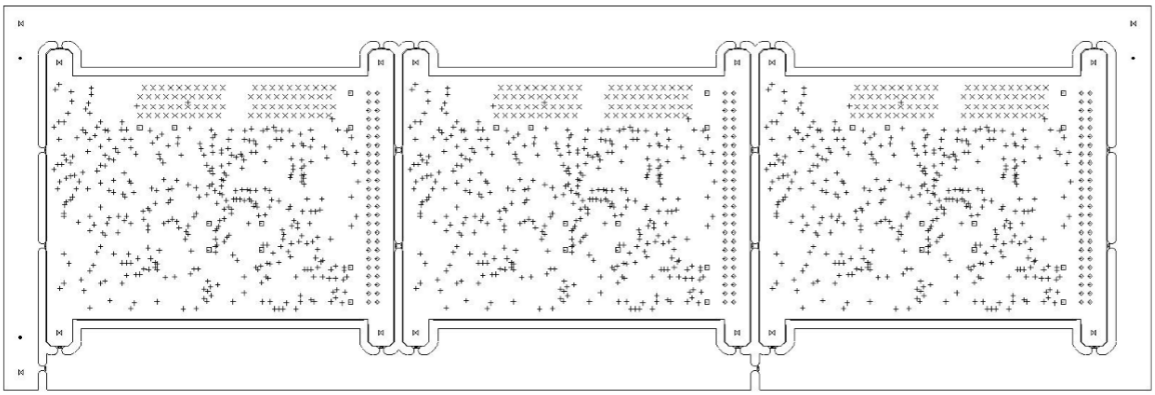

Figure 8. Automatically create panel arrays of PCBs on a panel documentation drawing

process. This solution enables designers to quickly create electronic manufacturing documents that clearly articulate the panel specification and instructions for successful fabrication, assembly, and inspection of their designs.

Panel Editor includes the following key features:

- Automatic array creation in spreadsheet or auto-calculate mode using imported, intelligent Allegro PCB design data results in an optimized panel that includes as many PCBs as possible
- Allows quick placement of top, bottom, drill pattern, or custom panel arrays on a drawing, as well as drill charts, note blocks, mill, and V-score details
- Panel-level drill and coordinate charts allow the user to place a drill and coordinate charts reflective of all the PCBs contained in a panel to define all components, hole sizes, locations, symbols, and quantities for the entire panel. All of the formatting capabilities of the single-instance PCB drill and coordinate chart are also supported for panels. In addition, any pinning holes added to the panel are automatically included in the drill chart. Coordinate charts or the panel can be exported in CSV format to drive pick-and-place machinery.
- Automatic NC milling definition allows the route path around a PCB to be defined within a panel as well as the breakaway tabs, then the NC milling can be automatically applied to all the PCBs within the panel as well as merged route information for one unified NC milling path
- Panel Drawing Detail wizards use the existing PCB and NC milling data to automatically depict mill tab and V-score details
- Web Publisher allows for the creation of PCB release packages in HTML format
- Flip panel support

Ease of Drawing Creation

PCB designers often turn to CAM tools for their PCB assembly panel design, tools created specifically for the CAM engineer at the PCB fabricator that most PCB designers find difficult to use. The Manufacturing Option's Panel Editor module is designed for ease of use, with drag-and-drop drawing elements—including PCB views, charts, details, text boxes, and note blocks—that are treated as drawing elements and can be quickly added and modified. Each element can be independently scaled, formatted, and transformed (e.g., rotated, mirrored, flipped). There is no limit on the number of elements that can be placed on a drawing.

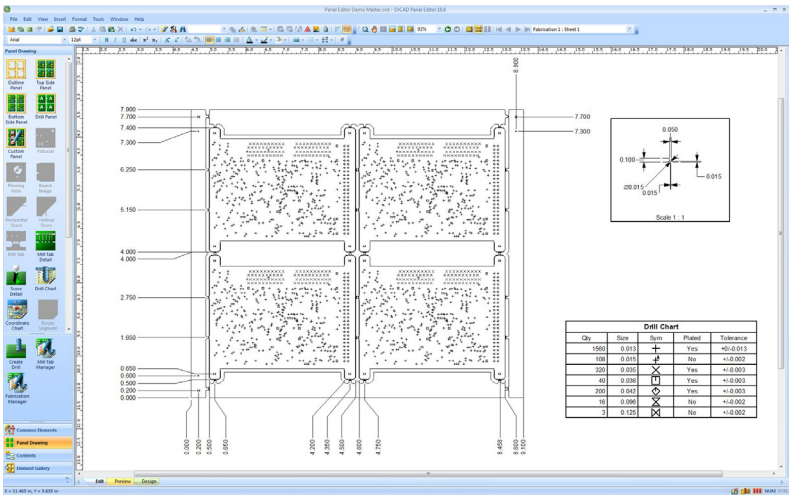

Figure 9. Panel Drawing Detail wizards use the existing PCB and NC milling data to automatically depict mill tab and V-score details

Data Importation

The Manufacturing Option's Panel Editor module uses the Allegro PCB CAD data in the IPC-2581 format to drive the panel definition and documentation creation process. External content such as JPEGs, BMPs, GIFs, and TIFFs can be imported and incorporated in the drawings. OLE objects, audio, and video files can also be imported and linked within the documentation drawings and stored with the documentation release package. In addition, RS274X files can be imported and DXF can be imported and exported.

Always Up-to-Date ECOs

The panel definition and all drawing elements are derived from the source Allegro PCB CAD data and always remain linked to that source data. These links allow you to perform lightning-fast ECOs by simply refreshing only the source CAD data. When refreshed, all instances of the original Allegro PCB CAD data (array instances, PCB views, tables, details, drill charts, etc.) are refreshed to display the new design data changes. All unique user settings defined for each instance are maintained during the refresh, further minimizing ECO documentation rework.

Cadence Services and Support

- Cadence application engineers can answer your technical questions by telephone, email, or Internet—they can also provide technical assistance and custom training
- Cadence certified instructors teach more than 70 courses and bring their real-world experience into the classroom
- More than 25 Internet Learning Series (iLS) online courses allow you the flexibility of training at your own computer via the Internet
- Cadence Online Support gives you 24x7 online access to a knowledgebase of the latest solutions, technical documentation, software downloads, and more

cadence®

Cadence Design Systems enables global electronic design innovation and plays an essential role in the creation of today's electronics. Customers use Cadence software, hardware, IP, and expertise to design and verify today's mobile, cloud, and connectivity applications. **www.cadence.com**

4455 04/15 SA/DM/PDF