

OrCAD Documentation Editor

Automating the PCB documentation process

Overview

OrCAD® Documentation Editor is a PCB documentation authoring tool that intelligently automates the documentation creation process to produce complex PCB documentation in a fraction of the time versus traditional methods. Developed in the style and use-model of Windows desktop applications (shown in Figure 1), OrCAD Documentation Editor enables you to very quickly and easily create documentation that drives PCB fabrication and assembly.

OrCAD Documentation Editor “knows” it is creating PCB documentation. Utilizing the OrCAD PCB CAD data, it 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 Editor release package contains all the data necessary to build, view, and archive the final product.

Highlights

- Intuitive Windows desktop application-style use-models and methods increase productivity and ease of creation for PCB documentation
- Tight integration with OrCAD 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
- Eliminates manual drawing of tedious document elements by automatically creating component assembly views, drill pattern view, details, and parts lists derived from the OrCAD PCB CAD data
- Design changes made in OrCAD PCB Editor are automatically propagated to all effected drawing elements, dramatically reducing time spent updating documentation due to the design changes

Documentation Features

Drawing creation

Standard or custom drawing elements are easily added by dragging and dropping from a palette of drawing-specific elements. You can drag and drop into place drawing 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 may be scaled, formatted, transformed (rotated, mirrored, flipped, etc.) independent of each other. There is no limit on the number of elements that can be placed on a drawing.

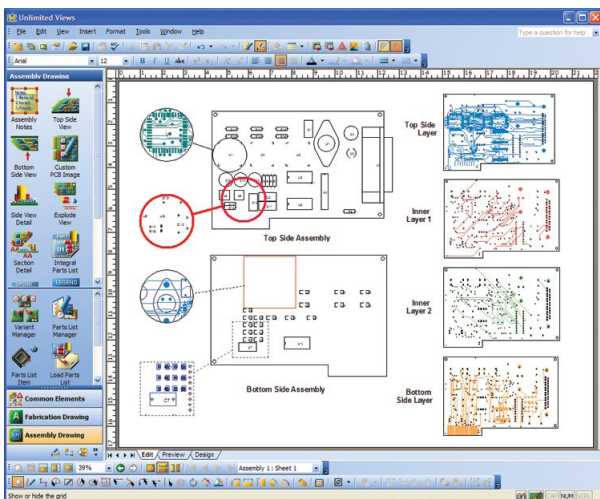


Figure 1. Produce complex PCB documentation in a fraction of the time with OrCAD Documentation Editor

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.

Design-driven documentation

OrCAD Documentation Editor drawing content is derived directly from the OrCAD 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 may 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.

ECOs

All drawing elements in the PCB Documentation Editor are derived from the source OrCAD PCB CAD data. Opening the revised design in OrCAD Documentation Editor allows you to perform lightning-fast engineering change order (ECO) updates by simply refreshing the source 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.

For the latest product or release information, visit us at www.orcad.com or contact your local Cadence Channel Partner.

Sales, Technical Support, and Training

The OrCAD product line is owned by Cadence Design Systems, Inc., and is supported by a worldwide network of Cadence Channel Partners (VARs). For sales, technical support, or training, contact your local channel partner. For a complete list of authorized channel partners, visit www.orcad.com/CCP-Listing.