

Facts about PCB Documentation Today

In an industry wide survey we conducted, nearly every PCB Designer cited that current PCB documentation methods are time-consuming, make it difficult to implement any design revisions, and prevent them from moving on to the next project. In a response to this industry-wide dilemma and concern, we developed BluePrint for Printed Circuit Boards.

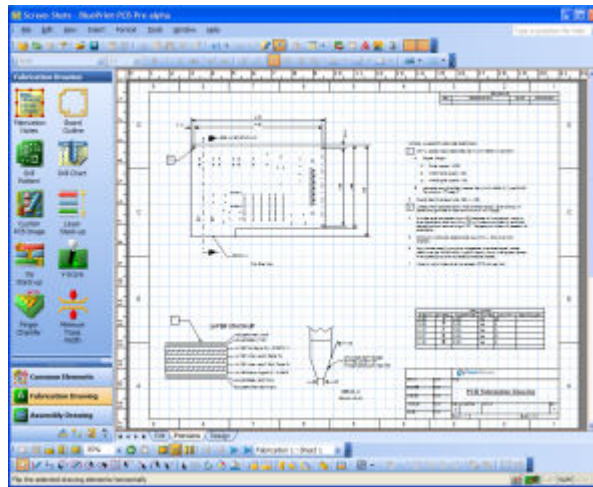
Importance of Complete Documentation

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 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.

- To specify form, fit, and finish, as well as drive procurement and scheduling
- To build and inspect finished good to the original engineering specifications
- To archive intellectual property, retaining instructions to ensure consistent quality
- To produce release milestones for manufacturing

The Challenge

PCB documentation is a critical part of the design chain as it defines the specifications on how an electronic product is to be built. The more detailed the documentation the less likely manufacturing mistakes will occur. It also ensures that repeat builds of the same product are consistent and makes final inspection easier with less chance of unit rejection or shipment delay while waiting for assembly deviations to be approved.

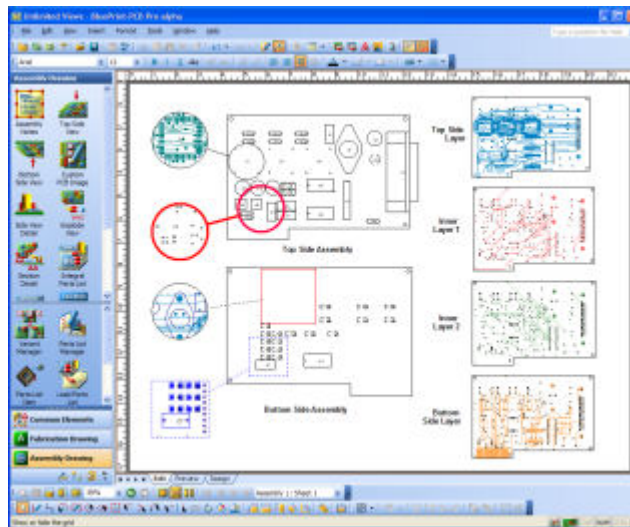


The Solutions

BluePrint is a dynamic new solution, developed in the style of Microsoft Office that quickly creates electronic drawings to drive PCB fabrication and assembly. BluePrint "knows" it is creating a PCB document and imports PCB CAD data to automatically create and intelligently link PCB Views, details, document notes, external documents, URLs and multi-media content. The result is an electronic document which better articulates the instructions for successful fabrication, assembly and inspection of printed circuit boards and which contains all the data necessary to build, view and archive the final product.

- BluePrint-PCB creates PCB drawings for fabrication, assembly, and inspection
- BluePrint-PCB uses MS Office methods to increase productivity and ease-of-use

- BluePrint-PCB creates and displays documents as they would appear to document control and manufacturing
- BluePrint-PCB is the most revolutionary breakthrough in the PCB market since the inception of PCB CAD



This new, state-of-the-art solution creates PCB documentation quickly and easily and replaces the more error-prone, manual approach currently employed. Documentation created with BluePrint is more detailed, can be changed instantly and will lead to faster and more accurate new product introduction.

Problem with PCB Documentation Today

- Tools today are NOT designed for PCB documentation
- Tool constraints create documentation constraints
- Tools used require extensive translation of data
- Documentation process is long, as high as 40% of design cycle
- ECOs, even small, require same documentation time
- Process is unintelligent and manual, prone to error
- Solutions and output vary from company to company
- Critical link in the design chain is made of paper

BluePrint's Value

Product success requires solid execution. Documentation provides zero end product value, but requires skilled PCB Designers to produce it. Poor documentation increases manufacturing complexity.

- Minimize time spent on low-value task
- Move onto the next design sooner
- Increase documentation quality eliminates manufacturing guesswork
- Lower risk of shipment delay owing to deviations and MCO approvals