

THE NEW AGE OF PCB DOCUMENTATION BY DANIEL FERNSEBNER

INTRODUCTION

The evolution of the PCB design process has progressed considerably since the days of light tables, tape and Mylar. The 80s introduced the first PCB design software, marking a new era in design-authoring capability and technology. Since then there has been a rise, fall, and consolidation of EDA companies; but one thing that has remained constant is the advancement of technologies that aid designers in the progressive challenges of PCB design.

The projection of Moore's Law has remained true, creating an ever-cascading set of new technologies and requirements. As designers, we have a love/hate relationship with the surging design constraints related to increasing frequencies, component densities, part sourcing and manufacturing cost. Design deadlines have forced us to unionize our EDA requests and demand new functionality for the next faster, better thing.

Based on our desires, EDA companies produced next-generation features in auto routing, high-speed routing, constraint management, component information systems, design reuse, and the list goes on. All this new functionality enabled PCB designers to capture logic and create layouts in lightning fashion, but has left some obvious, time-consuming tasks like documentation in the stone ages.

As much as we hate it, PCB design documentation is a necessary evil. It takes what we've captured and translates it into a realizable working device. We often forget the amount of time we spend documenting our designs. Just one design can include design fabrication drawing and notes, assembly drawing and process steps, PCB rework instructions, layer stackup information, drill details, Bill of Materials, and more. Every design review and re-spin warrant these documents be recreated reflecting the most recent version of your creation. For a given design, this can easily translate into double-digit documentation sets representing many lost hours that could be better spent on other tasks.

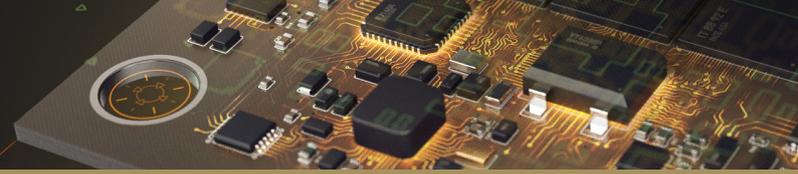
Current methodologies include lackluster drafting tools within the PCB design environment, basic text support, static automation, or external 2D Mechanical CAD solutions optimal for drawing, but lacking any type of connected intelligence.

Draftsman[®] is the new functionality being introduced in Altium Designer 16.1. It takes the effort out of creating PCB documentation by automating the creation of tables, PCB design views, layer legend, and details. The drawing document is linked to the source PCB document so they are always accurate and in synch. Let's take a look at the simplified and automated user experience that **Draftsman** provides for creating and maintaining your PCB documentation.

CREATION

We hate to admit it but we have all spent countless hours constructing beautiful documentation for our designs. I can personally admit to drawing nice layer stack legend detail views using 2D line tools only to have to redraw due to the addition/change of layers or via stacks. It seems silly to think that the technology that goes into auto routing has not translated into the documentation space - until now.

Altium Designer has a new PCB Drawing document type under **File » New**. As with all of Altium Designer's editors **Draftsman** is integrated and part of the unified design environment. Once the drawing document is created, the menus and toolbars are updated reflecting the editor change to **Draftsman** - no different than switching from a schematic to a PCB document. In the Draftsman Place menu, an array or standard drawing and dimensioning objects will be presented (Figure 1).



SYNCHRONIZATION

What makes **Draftsman** unique and where the real magic happens is the data synchronization between the PCB document and the **Draftsman** document. What does this mean? Consider the following example:

The PCB documentation has already been created for the initial design revision. Your design contains through hole vias that are constricting the BGA breakout channels. Adding blind vias will remove the constriction, allowing for maximum breakout expansion. Also, by adding buried vias, there are areas of the board you can optimize with vertical routing real-estate, allowing you to reduce layer count.

From a drill drawing perspective these changes impact the drill table, layer stack legend, and the design drill view. What was once a painful process is performed automatically since the PCB document (source) is linked to the **Draftsman** document (target). Once the PCB has been updated you can open the **Draftsman** document and synchronize the design data from **Tools » Update Board**. This automatically updates the drill drawing, eliminating the historic manual recreation process (figure 3 and 4).

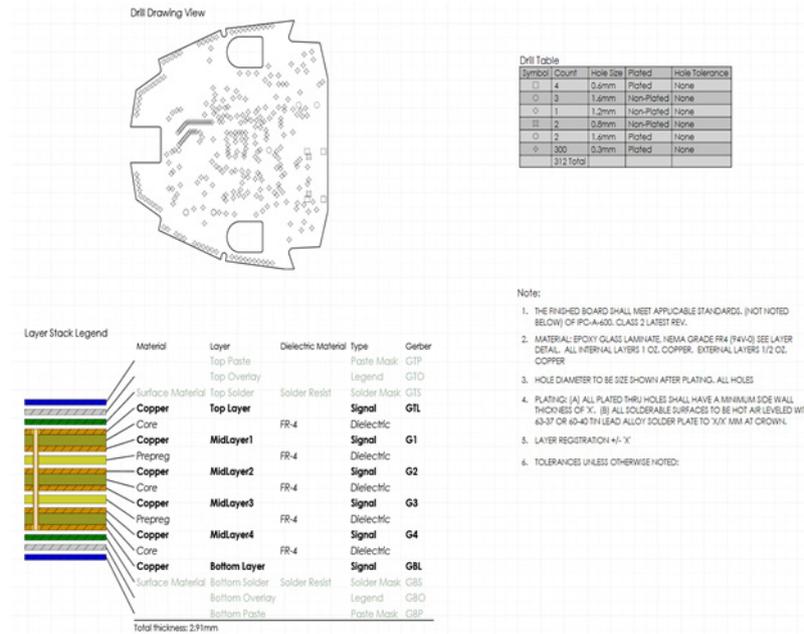


Figure 3: Drill drawing before the design change

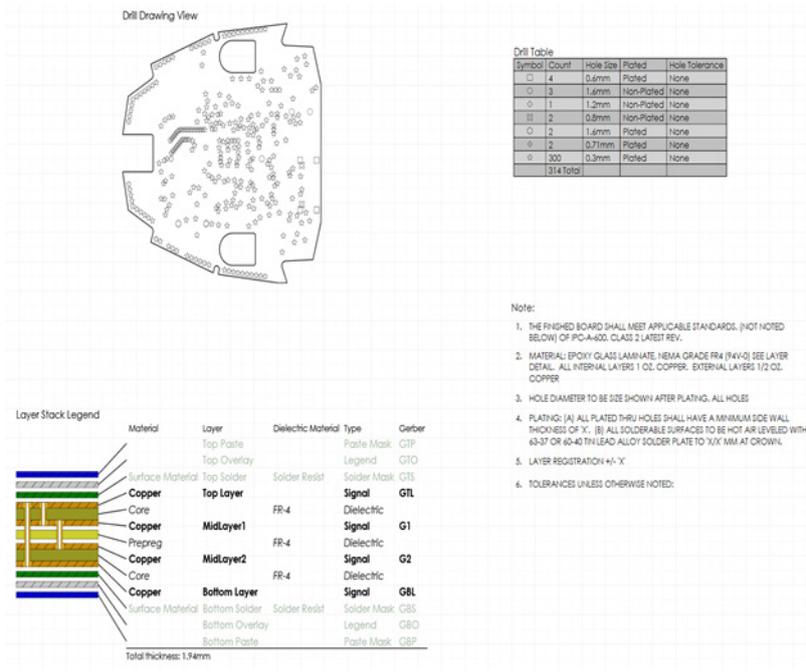
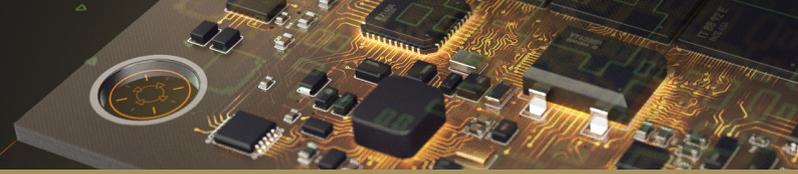


Figure 4: Drill drawing after the blind & buried via additions and layer stack change

Because the documents are linked it introduces another benefit of create intelligent design templates. Think about the beauty of creating PCB assembly, drill, and fabrication templates once. Opening a template allows the **Draftsman** document to be reprinted to a different PCB source. The object views you've placed are fixed and automatically updated based on the new source information. No need to redraw, replace, or update anything! You could literally get to a point where you have a library of these PCB documentation templates to account for various product families, board shapes, and required fabrication/assembly specifications. This not only automates the creation of PCB documentation, but ensures consistency and enforces necessary standards.

CONCLUSION

My father always told me, "You can't use a wrench as a hammer, a screwdriver as a chisel, or a shovel as a spade." These words have always resonated, yet as a designer I often find myself using the wrong tool for the job. How many of us have used Visio to capture schematics or AutoCad to do PCB layout and/or documentation? Both tools can get the job done, but because of the lack of electrical intelligence downstream, the side effects that occur when changes arise far outweigh any temporary convenience.

Draftsman brings PCB documentation to the next level, as push and shove did for routing. Its ease of use and intelligent data synchronization makes the headaches of PCB documentation creation and maintenance a thing of the past.

ABOUT ALTIUM

Altium Limited (ASX: ALU) is a multinational software corporation headquartered in San Diego, California, that focuses on electronics design systems for 3D PCB design and embedded system development. Altium products are found everywhere from world leading electronic design teams to the grassroots electronic design community.

With a unique range of technologies Altium helps organisations and design communities to innovate, collaborate and create connected products while remaining on-time and on-budget. Products provided are Altium Designer®, Altium Vault®, CircuitStudio®, PCBWorks®, CircuitMaker®, Octopart® and the TASKING® range of embedded software compilers.

Founded in 1985, Altium has offices worldwide, with US locations in San Diego, Boston and New York City, European locations in Karlsruhe, Amersfoort, Kiev and Zug and Asia-Pacific locations in Shanghai, Tokyo and Sydney. For more information, visit www.altium.com. You can also follow and engage with Altium via Facebook, Twitter and YouTube.