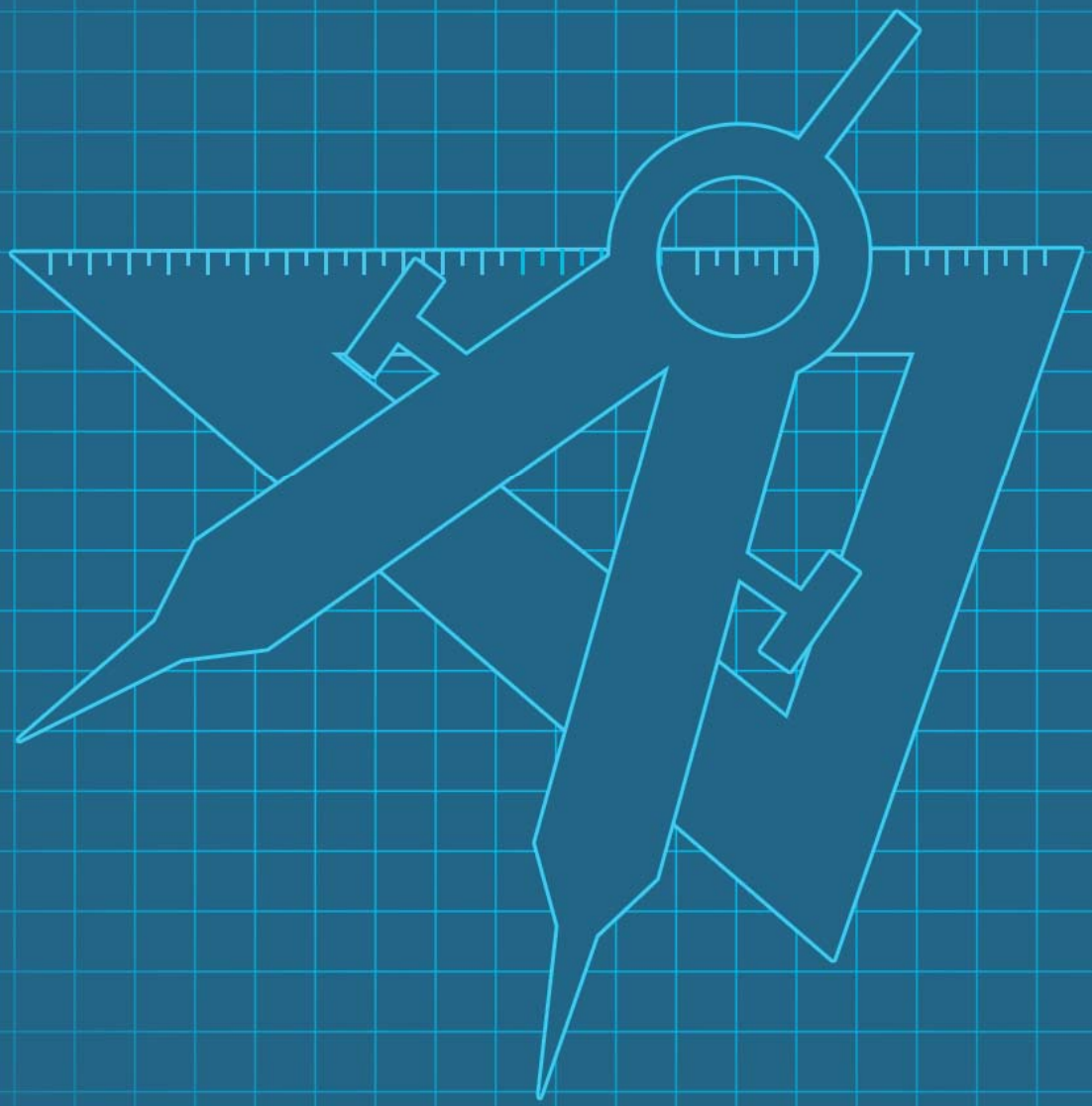
Altium®

Drawing Creation with Draftsman



Volker Brandstetter
Senior Applications Engineer

DRAWING CREATION WITH DRAFTSMAN

Technical drawings for a printed circuit board include the graphical representation of the product design along with information relevant to fabrication and assembly. In many cases, drawings have been done using Gerber for quite some time and have not been replaced by new, more intelligent output formats such as ODB ++, IPC 2581 and Gerber X2.

In many companies, drawing creation is often treated as an afterthought in product development. When the drawings are made, the development has already been largely completed. Despite that, the drawing are imperative for communicating the design and intent to the PCB fabricator and board assembler.

Another problem is that the ECAD systems have not generally provided the best methods for documentation. For this reason, drawing production has often been completed by the MCAD side, where they have been the benchmark when it comes to technical drawings.

Using the MCAD system requires that the data be moved from the ECAD system. The transfer often requires manual steps to get the data into the MCAD system. This method is complex, lossy and error-prone, in which data can be lost or corrupted. In addition, moving the drawing process to the MCAD system involves several people that must collaborate.

ENTER DRAFTSMAN

Draftsman is a new, modern system for creating technical drawings for a PCB, and is completely integrated into Altium Designer®. Draftsman has two different drawing views, one for the circuit board fabricator and one for the circuit board assembler.

The PCB fabricator needs the following drawings:

- The completely dimensioned PCB
- Drilling drawing with legend
- Layer structure with its materials and thicknesses
- Panelization data, including milling contours and/or scoring techniques
- Additional production-specific data

The PCB assembler requires the following drawings:

- The completely dimensioned PCB
- Drilling drawing with legend
- Critical component placement (components with tall or large dimensions)
- Parts list with cross-references for component placement
- Optionally, the detailed representation of critical areas and components
- Optionally, cutouts in the PCB
- Optionally, the data for the panelization
- Additional information relevant to the assembly

Many of the drawing contents are required by both the fabricator and assembler. Therefore, Draftsman offers the option to capture both drawing contents in a common document, as shown in Figure 1 and Figure 2.

DRAWING CREATION WITH DRAFTSMAN

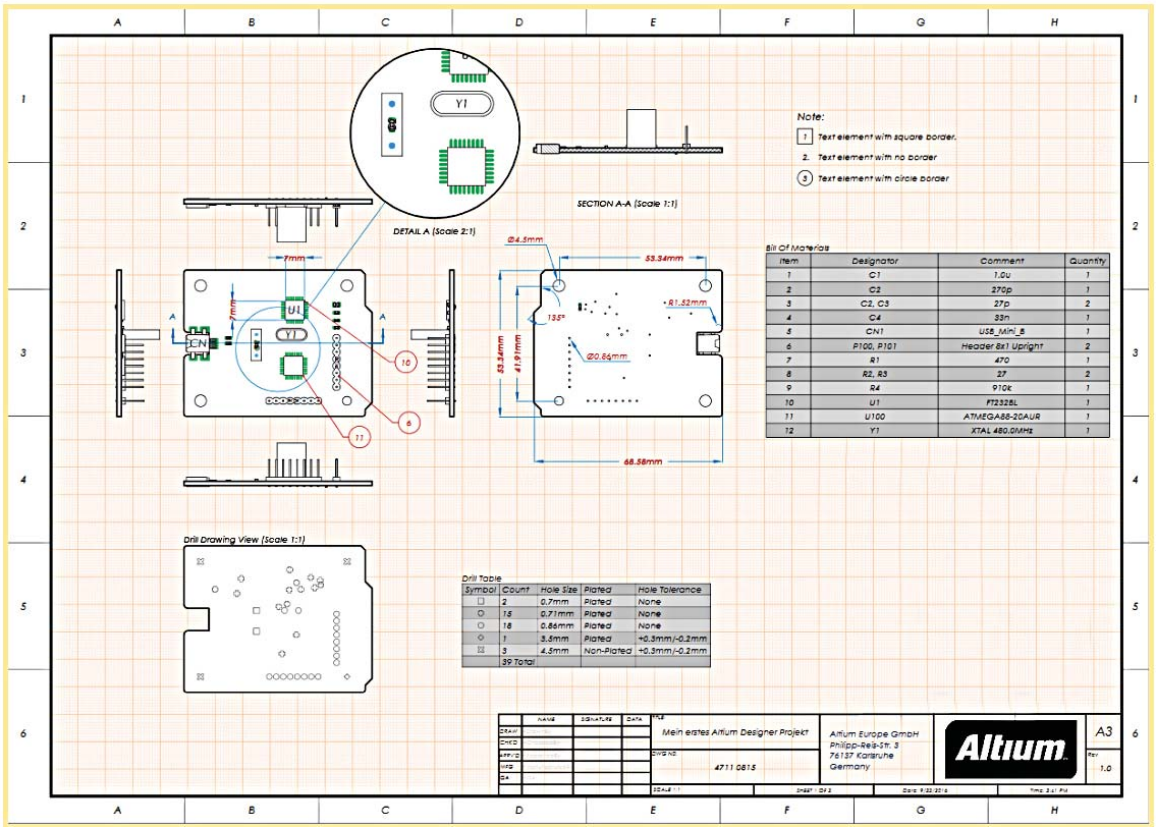


Figure 1: Example drawing from Draftsman with details for the PCB assembler.

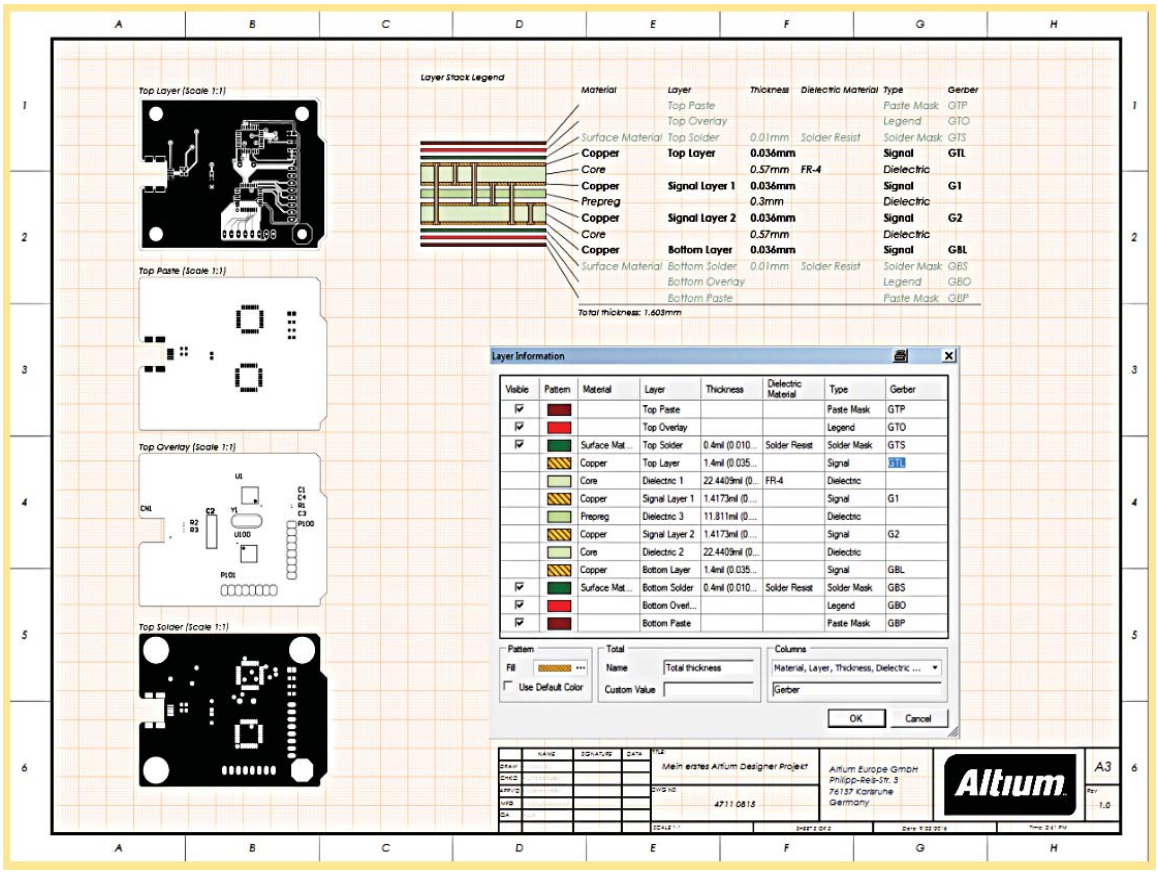


Figure 2: Example drawing from Draftsman with details for the PCB fabricator.

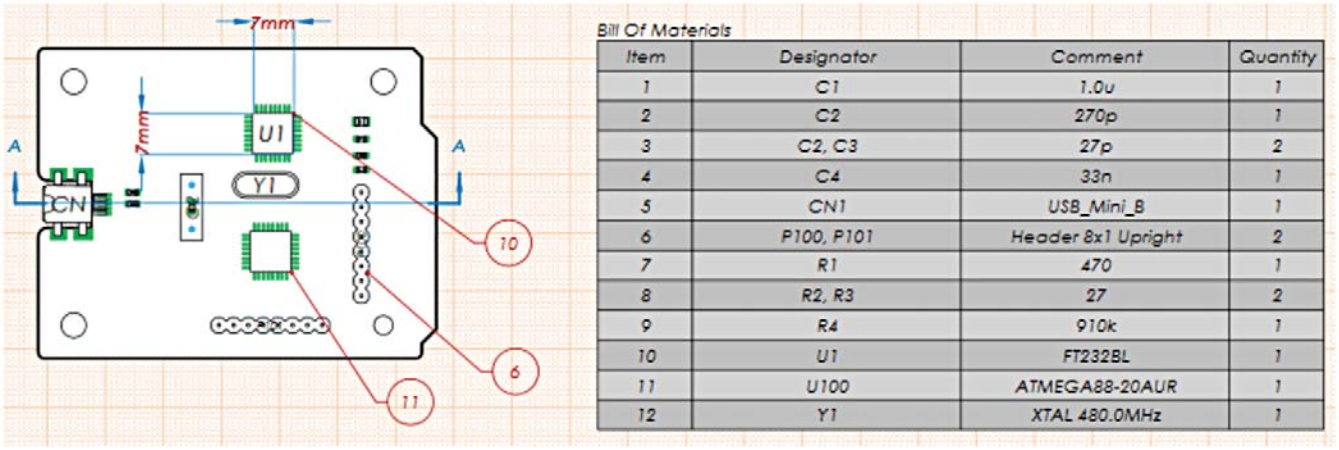
DRAWING CREATION WITH DRAFTSMAN

Since Draftsman is fully integrated, drawing data can be accessed directly from the parameters defined in Altium Designer. That eliminates exporting and importing data between different tools. The drawing frame, with drawing number, company, company logo and other parameters, can be set up as a template. The necessary views of the printed circuit board (top, bottom, left and right view) can then also be defined in a template and the necessary drawing standards taken into account. This is especially useful when working with recurring, identical or similar printed circuit board geometries. An update function ensures the automatic transfer of changes to the drawing.

A SUPERIOR PROCESS

With Draftsman, drawing creation takes place in parallel to the development not just at the end. The diagrams for the PCB design can thus be qualified and discussed with the participating partners in advance. After completion of the layout, you only have to update the trace data in Draftsman.

In Draftsman, the individual objects that belong together are linked together. For example, in the drilling table, if the drilling symbol is changed, the representation of the hole in the drilling drawing automatically changes. In the same way, a symbol with position information can be added to the assembly drawing for documentation of critical components. These callouts are automatically updated in the parts list (Figure 3). Manual errors that result from duplicate entries are eliminated.



Bill Of Materials

Item	Designator	Comment	Quantity
1	C1	1.0u	1
2	C2	270p	1
3	C2, C3	27p	2
4	C4	33n	1
5	CN1	USB_Mini_B	1
6	P100, P101	Header 8x1 Upright	2
7	R1	470	1
8	R2, R3	27	2
9	R4	910k	1
10	U1	FT232BL	1
11	U100	ATMEGA88-20AUR	1
12	Y1	XTAL 480.0MHz	1

Figure 3: Callouts with direct reference to the parts list.

Another important feature is that the output drawings are integrated into the **OutputJob** file in Altium Designer. With the **OutputJob** file, all production data are generated at the push of a button. In addition to the drawings, the OutJob file also includes Gerber, NC drill, pick and place data and much more, as shown in FIgure 4. This ensures that the production data output always takes place completely and with the same quality.

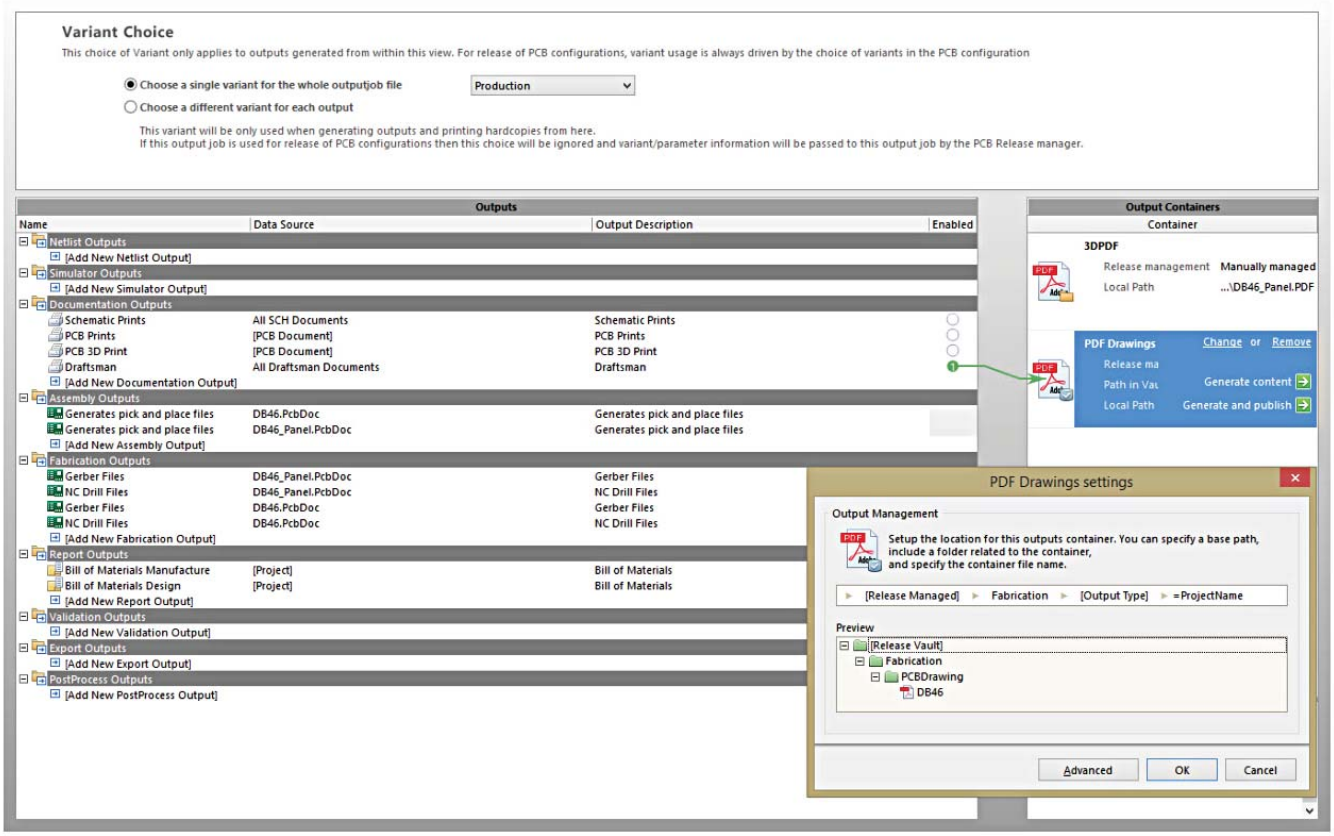


*Figure 4: Example of an **OutputJob** file with definition and configuration of the design.*

SUMMARY

Draftsman can significantly improve the quality of drawing production. Manual errors and additional steps in the process chain are eliminated. With Draftsman and the complete integration into the Altium Designer, Altium offers a complete development system from concept to the creation of the production data.