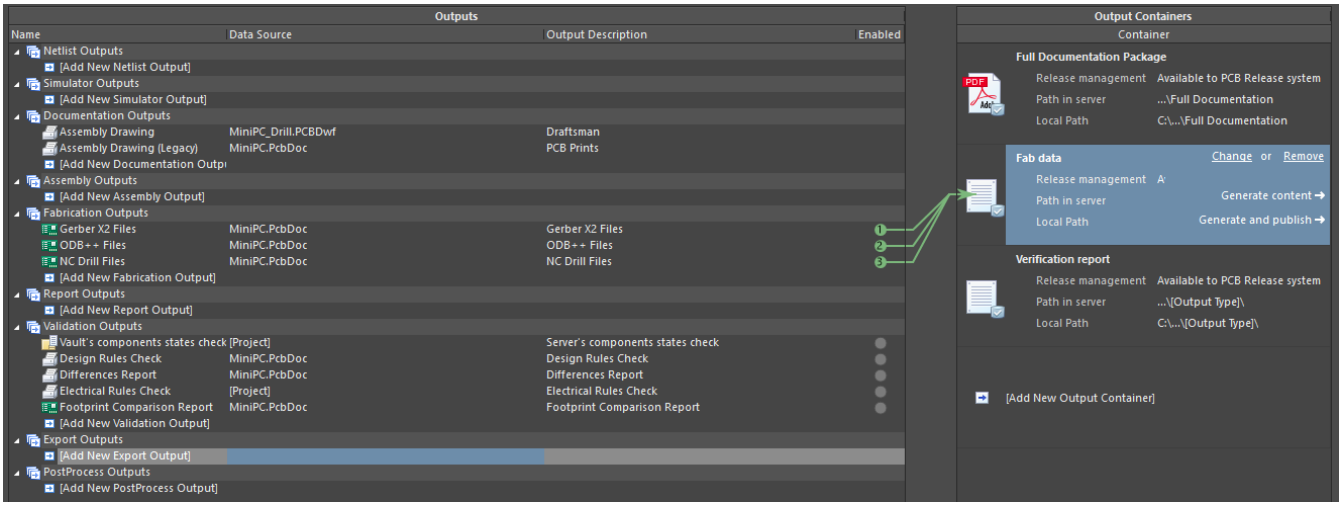
Altium Designer 18

FEATURES AND BENEFITS

- Easily generate and release complete design project documentation
- Clearly communicate design intent with your manufacturer to avoid ambiguities and errors
- Preview how your design will be viewed by your manufacturer before they receive it
- Automatically update assembly and fabrication files as changes are made to the design



MANUFACTURING DOCUMENTATION

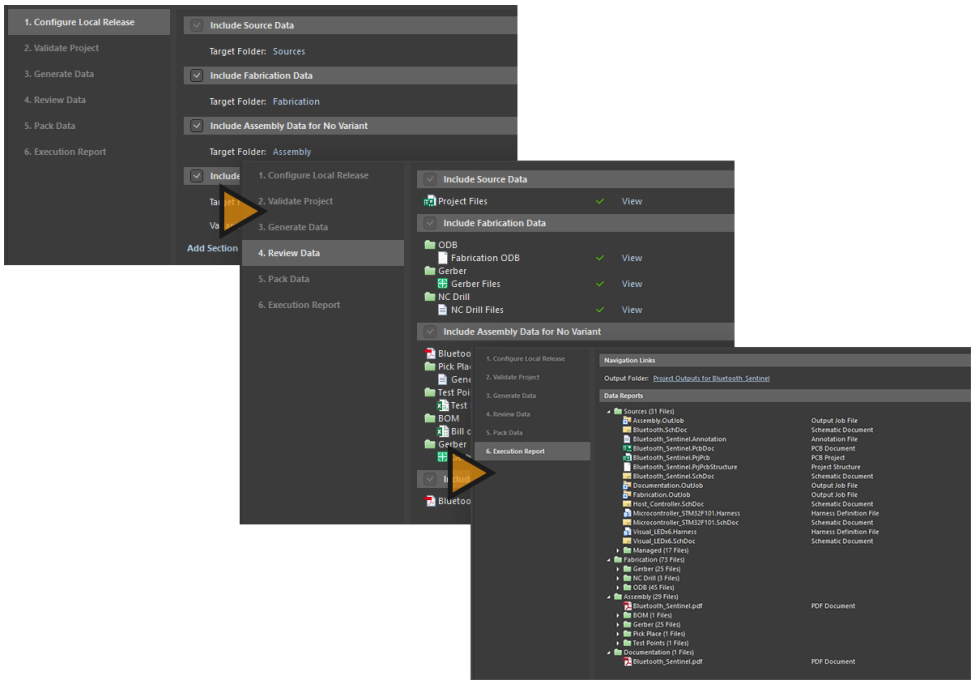
The moment that signals the final stages of the design's development cycle is a crucial one as you hand over your PCB design to manufacturing. It doesn't matter how perfect your design is on paper, or digital domain, any misstep in sending an incomplete documentation package to manufacturing can lead to delaying the release date at best, or to a non-conforming PCB at worst. With Altium Designer® output job, it's easy to generate a complete manufacturing documentation package and include configuration options for Gerber (X, X2), N.C. Drill, IPC-2581, ODB++, IPC-D-356A, 3D PDF, STEP, BOM reports, XML and more. Clear design intent communication to manufacturing is simple with powerful release management and automated documentation tools built into Altium Designer.

Reusable Batch Output Configurations

With the large variety of output formats Altium Designer can generate for a PCB design, it manages this multitude of outputs through the Output Job File. It's a preconfigured set of outputs, with each output configured with its own setting and its own output format. Output jobs act as an organized, reusable container for all necessary design outputs. Outputs can be written (where applicable) to three types of Output Container - A PDF, a specific format of output file (such as a Gerber file), or a video. You can also dynamically create customized project outputs for design variants linking fabrication and assembly outputs with the latest design source files.

Since Altium Designer is able to present output data in a number of different formats, the outputs are managed using an Output Job File. This file is a preconfigured set of outputs; each output can be configured to the particular format the user wants. The Output Job File is an organized and reusable container for all outputs. There are three types of containers: PDF containers, specific format containers (e.g. a Gerber container), and video. You can also dynamically create customized project outputs that incorporate design variants by linking fabrication and assembly outputs with the latest design source files.

With an organized release process, output generation gains consistency and accuracy while ensuring you don't use out-of-date design files. The result is conveyance of design intent to the contract manufacturer of the completed design, reduced time to market, first pass manufacturability, and increased design integrity.

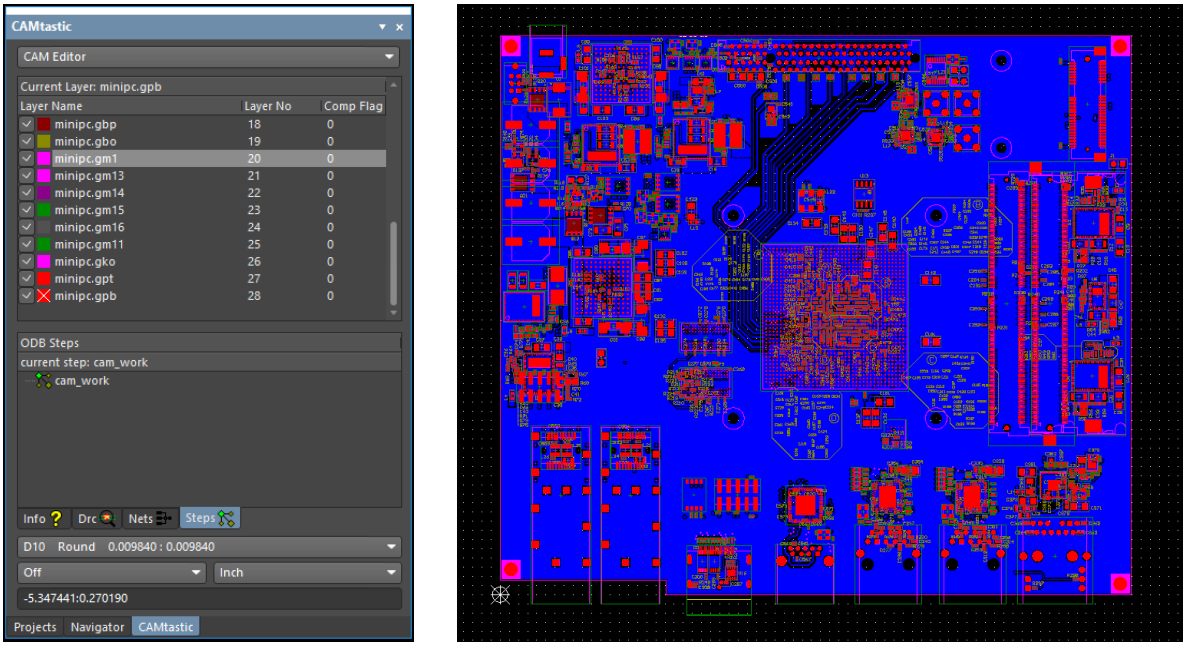


Easy and Simple to Generate Documentation Output in Multiple Formats

Visual Manufacturing Outputs

The Altium Designer CAM editor (CAMtastic) offers a variety of tools, the most basic of which are for viewing and editing CAM data. Once the image and drill files have been imported, the CAM editor can receive instructions determining layer types and stackup, at which point a netlist can be extracted and compared with an IPC netlist generated from the original PCB design software. These netlists will handle not just through-hole vias, but blind and buried vias as well. The CAM editor also offers design rule checking, panelization and NC-Routing (plus milling) tools.

You can view the manufacturing outputs of the design to gain insight into what the contract manufacturer will receive. This insight allows you to make changes at identified problem areas. It also allows you rudimentary reverse engineering capabilities to create a skeleton PCB from manufacturing files. You'll be able to increase design integrity, design intent communication, and first pass manufacturability.

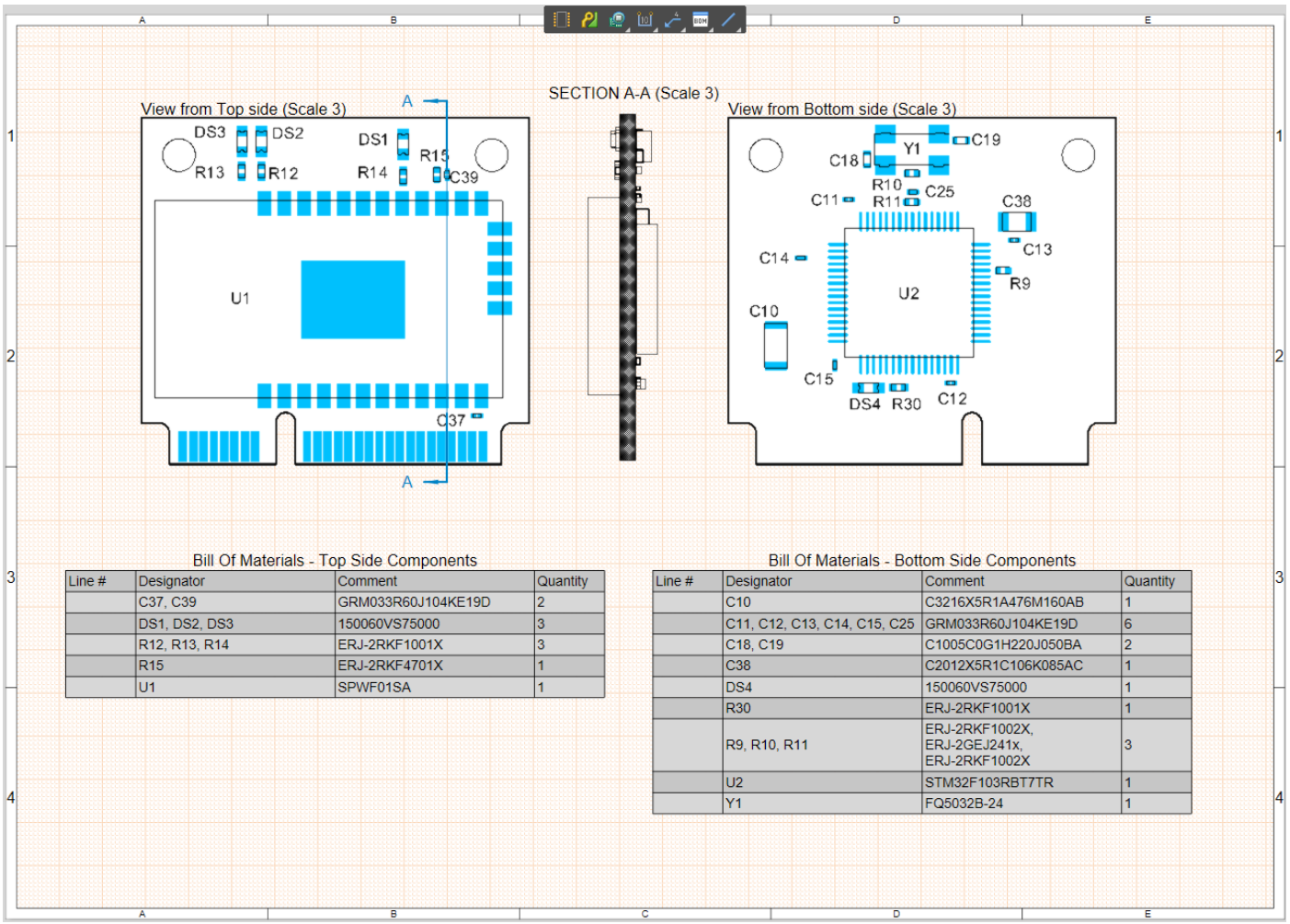


The CAM Editor lets you View Manufacturing Outputs

Seamless PCB Documentation Process

You can create assembly and fabrication documentation directly linked to source designs with Draftsman® and update the entire documentation at the click of a button. You can create templates for documentation that only require minimal customization across designs. In addition, you can add PCB dimensions, measurements, notes, and callouts between points of interest (datums) and design objects to customize documentation workflow.

The set of powerful and easy-to-use features integrated into Altium Designer automate documentation to ensure consistency. Assembly and fabrication documentation automatically sync and update as you make changes to your design since they are directly linked to the design source files. The possibility for data mismatch is reduced to nearly zero, which leads to reduced time to market, increased design integrity, improved design intent communication, and first pass manufacturability.



Bill Of Materials - Top Side Components

Line #	Designator	Comment	Quantity
	C37, C39	GRM033R60J104KE19D	2
	DS1, DS2, DS3	150060VS75000	3
	R12, R13, R14	ERJ-2RKF1001X	3
	R15	ERJ-2RKF4701X	1
	U1	SPWF01SA	1

Bill Of Materials - Bottom Side Components

Line #	Designator	Comment	Quantity
	C10	C3216X5R1A476M160AB	1
	C11, C12, C13, C14, C15, C25	GRM033R60J104KE19D	6
	C18, C19	C1005C0G1H220J050BA	2
	C38	C2012X5R1C106K085AC	1
	DS4	150060VS75000	1
	R30	ERJ-2RKF1001X	1
	R9, R10, R11	ERJ-2RKF1002X, ERJ-2GEJ241x, ERJ-2RKF1002X	3
	U2	STM32F103RBT7TR	1
	Y1	FQ5032B-24	1

Automated Fabrication and Assembly Documentation