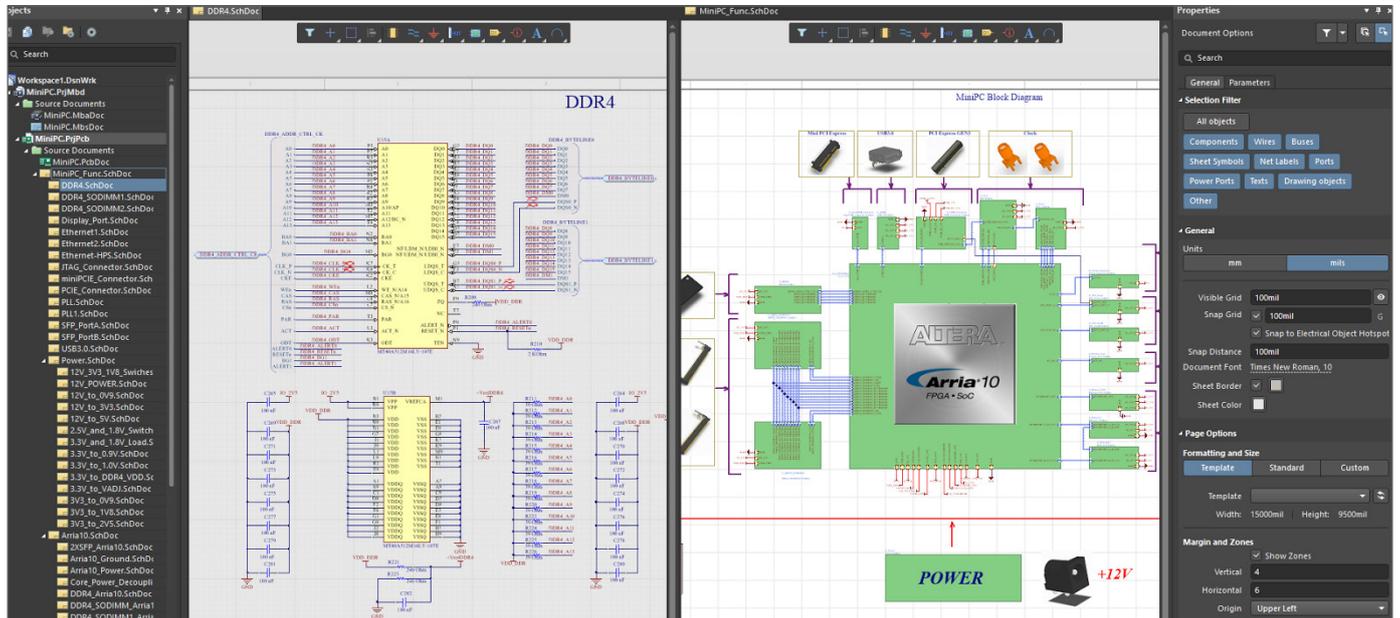


Easy, Modern and even more Powerful

FEATURES AND BENEFITS

- Natural (intuitive) UI, with modern powerful schematic capture.
- Fast Integrated Component Search and Placement.
- Flexible and simple automated annotation options.
- Connectivity and electrical rule checking verify and validate your design.
- Easier schematic design style more suitable for engineers.
- Hierarchical design to better manage large, complex projects.
- Multi-Channel and Device Sheets provide a design-for-reuse platform.
- Templates accelerate and automate schematic styles.



“STRESS FREE” SCHEMATIC CAPTURE

Every PCB design depends on accurate schematics. Whether they are starting from scratch or working on existing designs, schematic capture is fast and intuitive with our modern user interface. A two way connection is maintained between schematics and PCB through a user-approved and controlled synchronization process to provide a unified interface throughout the design process, improving productivity and enabling cross-referencing between schematic and PCB layout.



Design Importers

Users can leverage previous designs, saving time in re-creating schematics, PCB documents, and associated design data with a robust importation of project files from P-CAD[®], EAGLE[®], OrCAD[®], PADS[®], Xpedition[®] xDX Designer, Xpedition[®] Enterprise, CADSTAR[®], and Allegro[®].¹²³⁴

Users don't have to start from scratch facilitating migration, enabling them to learn about Altium Designer with a familiar project, and allowing switching tools during any stage of a project to Altium Designer.

Unified Schematic and PCB layout

With cross-probing, when users select an object on schematic that same object is selected on the PCB and vice versa. Cross probing automatically cross-references every net, pin, and component on the PCB to give the clearest insight into the implementation of schematics.

Users can place related circuitry quickly and make better decisions on placement, making it easier to get a successful layout on the first try. Finding specific design aspects from schematic to PCB is simplified reducing time to track down errors decreasing time to market.

Mixed Simulation

Altium Designer allows users to easily create and manage multiple simulation profiles. Separate profiles allow designers to run different types of analyses with different simulation engines (SPICE3F5/XSPICE, SIMetrix, SIMPLIS). This allows multiple runs of the same simulation type (e.g., AC analysis) with different parameters and options (e.g., different frequency ranges). Active profiles can easily add, remove, edit, run, and/or generate netlists. The Profile Manager organizes profiles and uses probes or active nets to select waveforms to display.

All simulation results can be saved with other manufacturing outputs for conveyance to manufacturing. Users can convey design intent to the contact manufacturer reducing time to market and minimizing errors.

Hierarchical & Multi-channel Design

Electronic devices are generally complicated systems within systems. It's a natural desire to break up the design into pieces like blocks or modules, to "divide and conquer" the design. Also, it is often desirable to re-use specific circuit blocks in different designs, or as multiple channels within the same design. Having a Hierarchical Design Environment (Like Altium Designer's) facilitates the creation of designs at a block diagram top-level, and allowing design projects to be split into manageable logical chunks (ie. Power Supply, Analog Front-End, Processor, IO, Sensors etc.) Hierarchical design also allows engineers to instantiate multiple copies of the same block when they need multiple channels of an identical circuit (e.g. audio-visual mixing equipment).

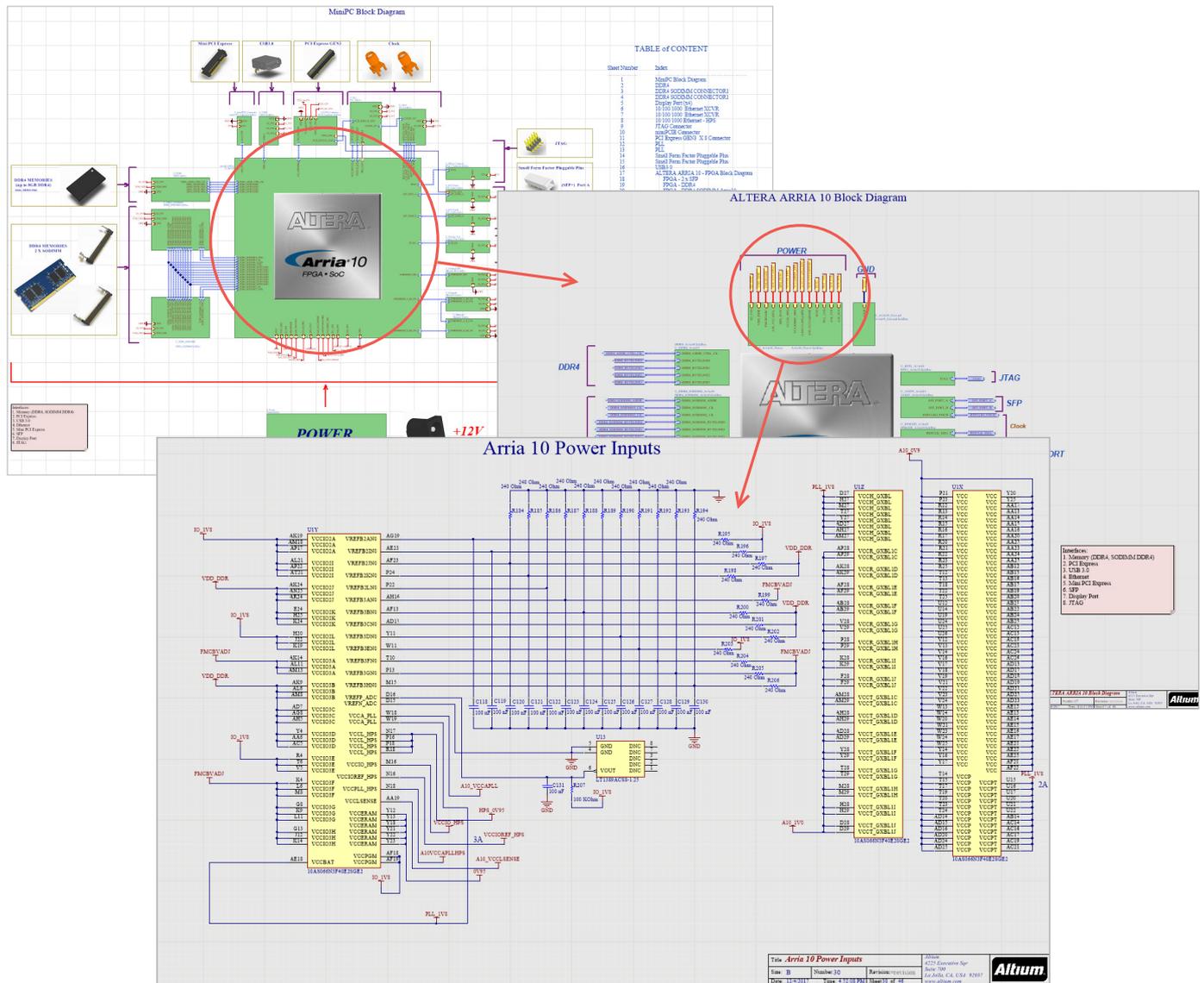
Users save time on the PCB side by allowing the circuit layout and routing to be automatically duplicated for identical circuits. When a change is made, it can be made to the base logical block and the results will propagate through the design. Overall, users minimize work and potential rework and increase design integrity by reusing blocks, reducing time to market and minimizing errors.

¹ Xpedition[®] and PADS[®] are registered trademarks of Mentor Graphics Corporation and Altium claims no rights therein.

² EAGLE[®] a registered trademark of Autodesk Inc. and Altium claims no rights therein.

³ OrCAD[®] and Allegro[®] are registered trademarks of Cadence Design Systems, Inc. and Altium claims no rights therein.

⁴ CADSTAR[®] is a registered trademark of Zuken and Altium claims no rights therein.



Hierarchical Design - Break Your Design Into any Depth Logical Modules, to Design More Efficiently

Design Designer Annotation

Every component in any design must be uniquely identified in order to purchase, assemble and test the product. Traditional approaches lack unification between schematic and PCB, and rely solely on each component reference designer for component identification. Altium Designer on the other hand is truly unified, in that each and every component instance is known by a unique identifier and the link between the schematic symbol, PCB footprint, and simulation models.

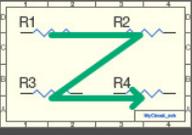
As a result, annotation and re-annotation in Altium Designer is able to be driven automatically from schematic, with auto-increment during parts placement, or can be driven from the PCB side, and back-annotated to schematic, to allow geometry-based ordering of reference designers.

Annotation can be left automatic, or can be user controlled, to allow for annotation based on schematic page, location, and sub-part, or channels and any combination, giving ultimate flexibility when needed.



Schematic Annotation Configuration

Order of Processing
Across Then Down



Process Location of
Designator

Matching Options
Complete Existing Packages: None

Component Parameter (Strictly)

- Capacitance
- Case Code (Imperial)
- Case Code (Metric)
- Case/Package
- Case/Package
- Category
- ClassName
- Code_IPC
- Code_JEDEC
- Color
- Comment

Schematic Sheets To Annotate

Schematic Sheet	Annotation Scope	Order	Start Index	Suffix
2.5V_and_1.8V_Switches.SchDoc	All	0	1	
2XSFP_Arria10.SchDoc	All	1	1	
3.3V_and_1.8V_Load.SchDoc	All	2	1	
3.3V_to_0.9V.SchDoc	All	3	1	
3.3V_to_1.0V.SchDoc	All	4	1	
3.3V_to_DDR4_VDD.SchDoc	All	5	1	
3.3V_to_VADJ.SchDoc	All	6	1	
3V3_to_0V9.SchDoc	All	7	1	
3V3_to_1V8.SchDoc	All	8	1	
3V3_to_2V5.SchDoc	All	9	1	
12V_3V3_1V8_Switches.SchDoc	All	10	1	
12V_POWER.SchDoc	All	11	1	
12V_to_0V9.SchDoc	All	12	1	
12V_to_3V3.SchDoc	All	13	1	

Proposed Change List

Current Designator	Sub	Proposed Designator	Sub	Location of Part
C1		C1		2.5V_and_1.8V_Switches.SchDoc
C2		C2		2.5V_and_1.8V_Switches.SchDoc
C3		C3		2.5V_and_1.8V_Switches.SchDoc
Q1		Q1		2.5V_and_1.8V_Switches.SchDoc
Q2		Q2		2.5V_and_1.8V_Switches.SchDoc
Q3		Q3		2.5V_and_1.8V_Switches.SchDoc
Q4		Q4		2.5V_and_1.8V_Switches.SchDoc
Q5		Q5		2.5V_and_1.8V_Switches.SchDoc
Q6		Q6		2.5V_and_1.8V_Switches.SchDoc
Q7		Q7		2.5V_and_1.8V_Switches.SchDoc
R1		R1		2.5V_and_1.8V_Switches.SchDoc
R2		R2		2.5V_and_1.8V_Switches.SchDoc
R3		R3		2.5V_and_1.8V_Switches.SchDoc
R4		R4		2.5V_and_1.8V_Switches.SchDoc
R5		R5		2.5V_and_1.8V_Switches.SchDoc
R6		R6		2.5V_and_1.8V_Switches.SchDoc
R7		R7		2.5V_and_1.8V_Switches.SchDoc
R8		R8		2.5V_and_1.8V_Switches.SchDoc
R9		R9		2.5V_and_1.8V_Switches.SchDoc
R10		R10		2.5V_and_1.8V_Switches.SchDoc
R11		R11		2.5V_and_1.8V_Switches.SchDoc
U1	15	U1	15	2XSFP_Arria10.SchDoc

Annotation Summary

Annotation is enabled for all schematic documents. Parts will be matched using 2 parameters, all of which will be strictly matched. (Under strict matching, parts will only be matched together if they all have the same parameters and parameter values, with respect to the matching criteria. Disabling this will extend the semantics slightly by allowing parts which do not have the specified parameters to be matched together.) Existing packages will not be completed. All new parts will be put into new packages.

Buttons: Update Changes List, Reset All, Back Annotate, Accept Changes (Create ECO)

Simple Design Annotation

Aesthetic, Natural, and WYSIWYG

Most schematic editors give users a headache from squinting all the time. Concentration for engineers is a key part of the job, but having to stare at a screen of pixel-width poorly colored lines on a pure white or black background just hurts. Altium Designer uses higher contrast colors and line scaling so you don't suffer from eye fatigue when engineering your circuits, unlike most other schematic capture tools.

Couple that with Altium Designer's elegant and intuitive contextual UI. Unlike our "big" competitors, Altium Designer doesn't distract you with dozens of buttons and toolbars of cryptic glyphs which needlessly take up screen real-estate. What you need is right there by your cursor, when you need it.

Schematic Design Rules

Engineers can add design rule "Directives" to the schematic - nets, wires, busses, harnesses, any component or sheet or document parameter. These are used to drive rules for correctly laying out the PCB to help the PCB designer get the board design right the first time.

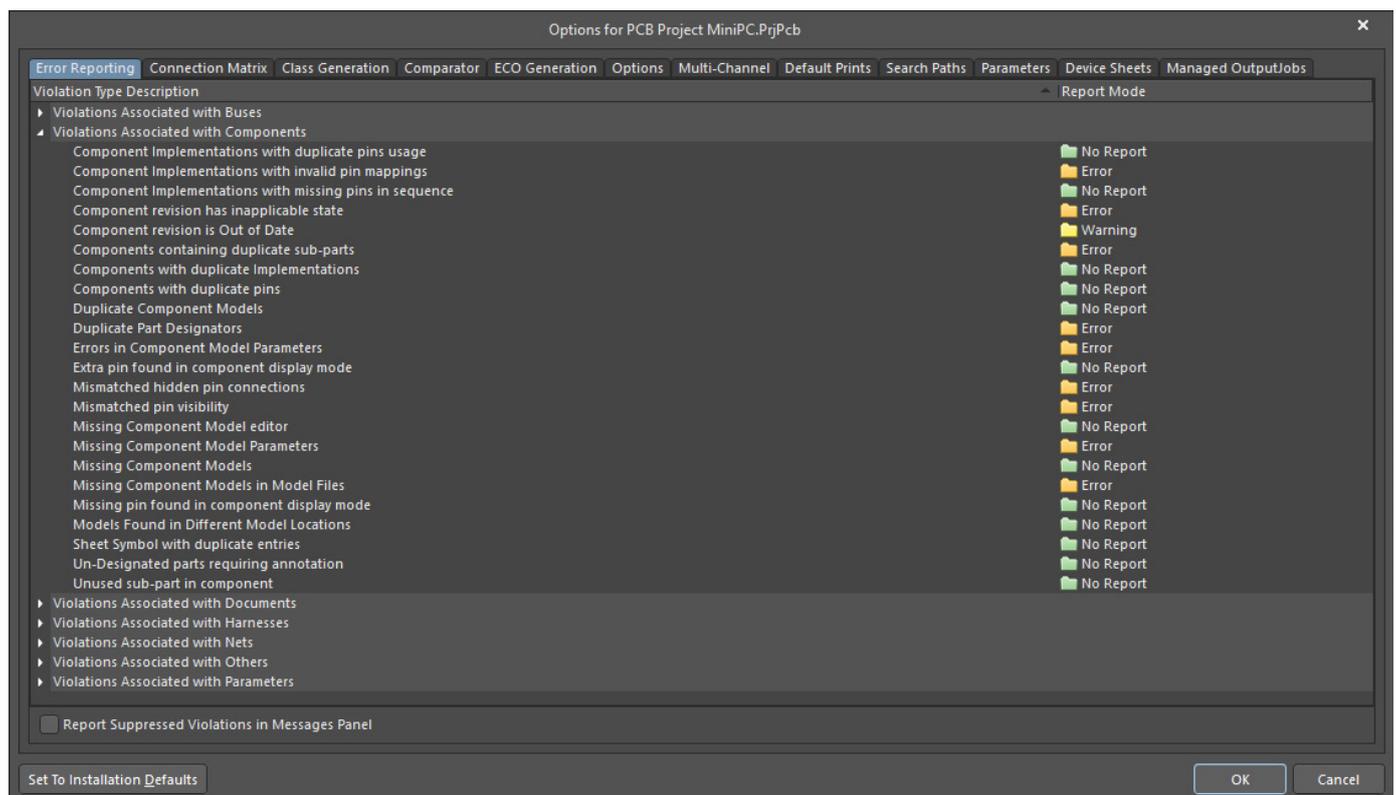
Examples of this are differential pair definitions and length matching rules for DDR memory routing. Created design rules drive the routing and layout, saving time and also providing guidance for the PCB designer from the schematic (ie. the engineer). Users benefit by reducing the number of possible errors and helping identify existing errors, for example, collision errors with an enclosure. Users will experience fewer errors and find them quicker leading to a reduction in time to market and reducing manufacturing and respin costs.



Electrical Rule Checks

The Electrical Rule Checks (ERC) in schematic alert the engineer to problems in the design. While Design Rule Checks help the PCB designer correctly layout the board and meet manufacturing requirements, the Electrical Rule Check helps prevent the engineer from making design mistakes at engineering level.

For instance, connecting two output driving sources together will cause a rule violation and associated error message in the Messages Panel, so the engineer knows they've made an error that would cause an electrical fault in the finally assembled circuit. Users will experience fewer electrical errors and find them quicker leading to a reduction in time to market and reducing manufacturing and respin costs. ERC also reinforce the design will function correctly once manufactured.



Electrical Check Validation Report

Design Reuse with Uniform, Defined Design Materials

Templates create uniform design units to keep design information organized. The design units range from primitives as small as pads, to full project types to act as a common baseline for all new design materials.

Snippets are saved pieces of circuitry on a schematic and PCB level that can be used on any design to leverage common circuitry.

Device sheets allow users to create known-good circuitry blocks for reuse across designs. They differ from snippets by increased complexity and predefined interconnection to other parts of the design. For example, a power supply system that has a defined output of 5 Volts to power another circuit on the design.

Users save time on the Schematic and PCB side by allowing the circuit layout and routing to be reused. When a change is made, it can be made to the base logical chunk and the results will propagate through the design. Overall users minimize work and potential rework and increase design integrity by reusing blocks reducing time to market and minimizing errors.