



ALTIUM
DESIGNER

Multi-Board Design Basics in AD18



David Haboud

*Product and Persona Marketing
Engineer*

July 26, 2018

1

Multi-Board Challenges

2

Electrical/ Mechanical Challenges

3

Resolving Challenges

How do you manage...

- I. board connectivity
- II. enclosure fit
- III. pin swapping
- IV. collision checking



1

Multi-Board Challenges

2

Electrical/Mechanical Challenges

3

Resolving Challenges

Board change management

System/Product connectivity definition

Signal management

Pin swapping

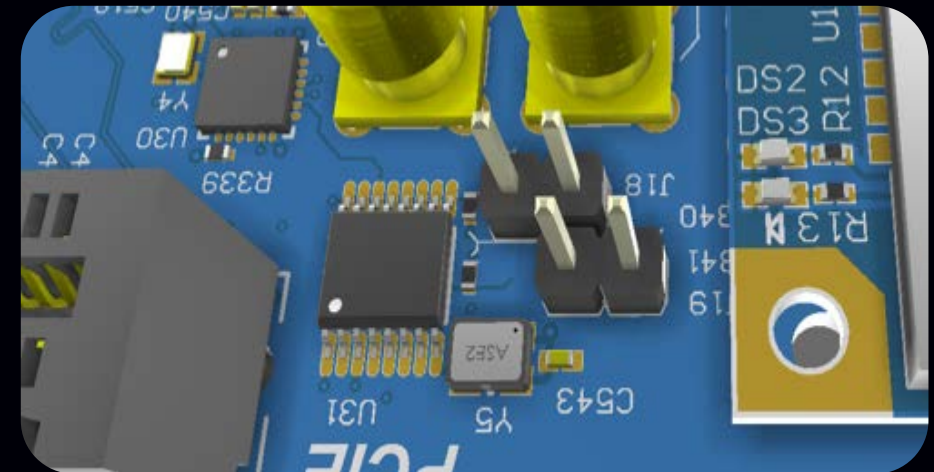


Commonly managed with XLS/DOC files and Emails!

Positioning, fit, and collision prevention

Placement optimization:

- Component
- Connector
- Board
- Enclosure



Time intensive model rendering and file transfers!

- 1 Multi-Board Challenges
- 2 Electrical/Mechanical Challenges
- 3 **Resolving Challenges**



DEMO

[Start A 15-Day Evaluation!](#)

Upcoming Webinars:

| Date | Time | Topic |
|-------------|--------------|---|
| Tue, Aug 21 | 10:00 AM PDT | <u>Easy PCB Drawings with Draftsman®</u> |
| Thu, Aug 23 | 10:00 AM PDT | <u>Altium Designer 18: High Performance Made Simple</u> |
| Tue, Aug 28 | 10:00 AM PDT | <u>Altium Designer 18: Key Benefits of Active BOM®</u> |

Thanks for your attention!

David Haboud

David.Haboud@Altium.com

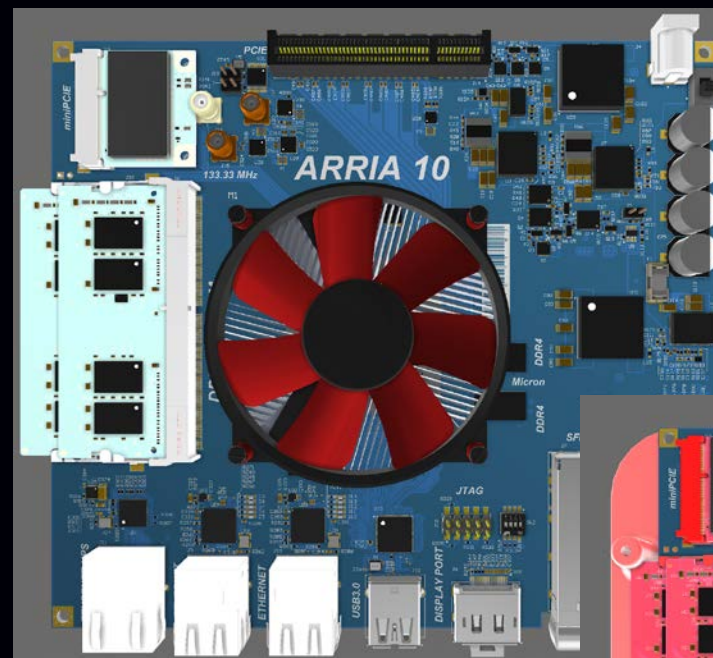
Product & Persona Marketing Engineer



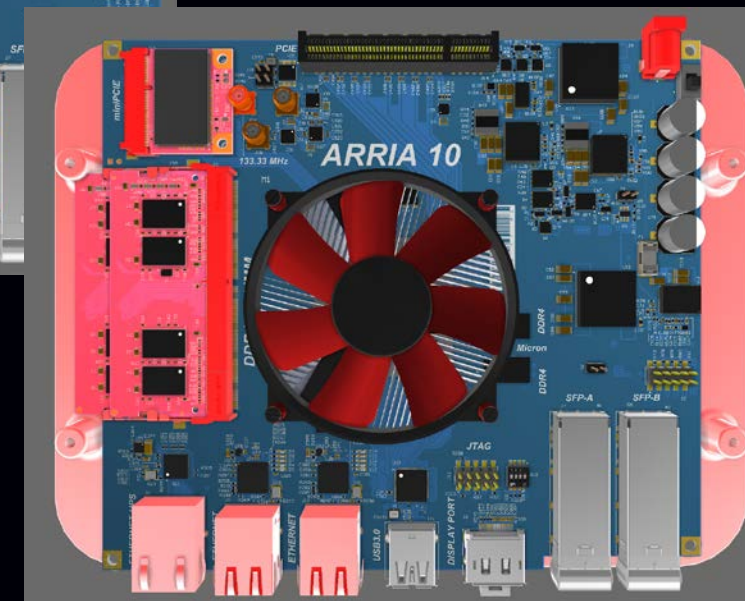
7/26/2018

Q: Is there a collision detection function?

A: Collision detection runs through the entire multi-board assembly including boards and models that have been made invisible. You can even change the color of the violation to ensure you have full visibility on all collisions. Below are two screenshots of board violations. The first screenshot shows the violations in white with the enclosure visibility disabled. In the second screenshot, you can see I have toggled the violation color to red and enabled the visibility of the bottom half of the enclosure.



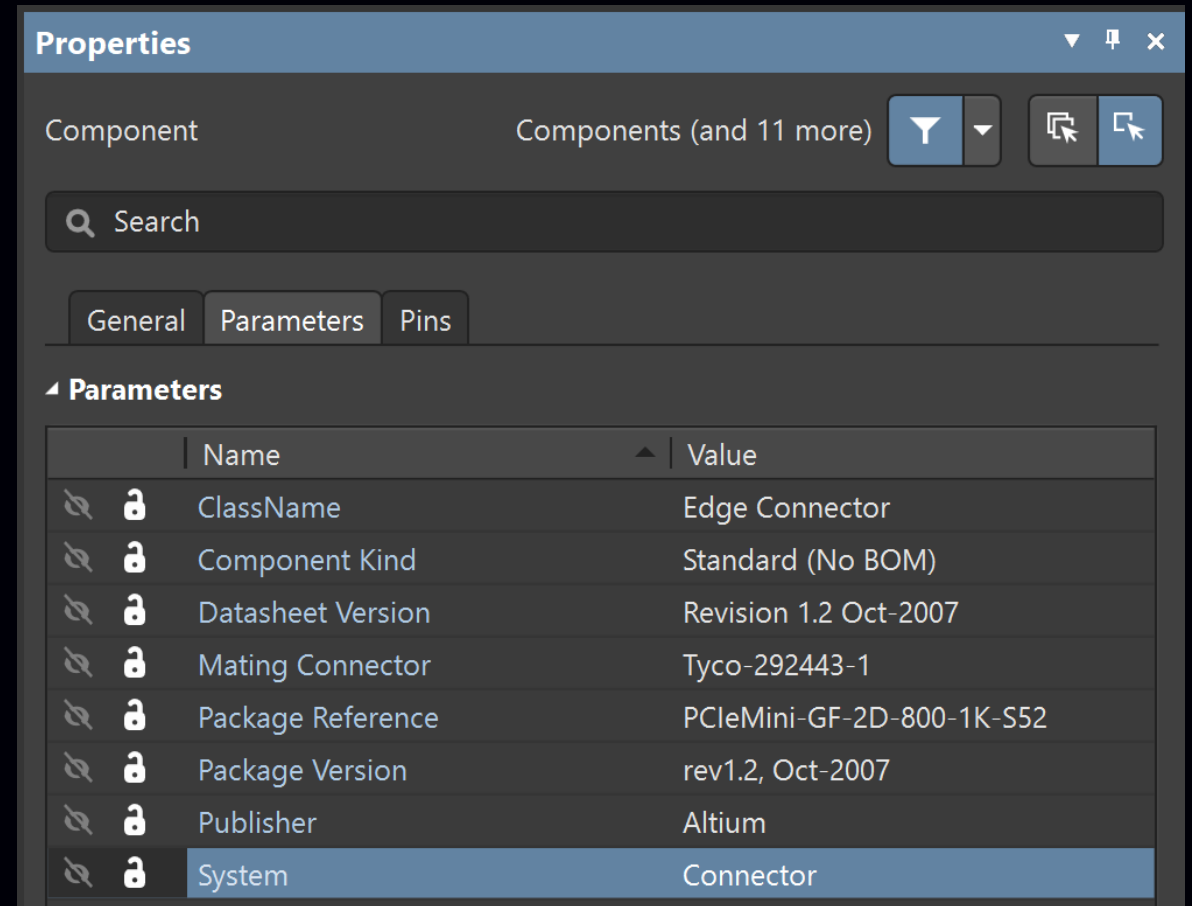
Collision Detection
with Hidden
Enclosure



Changing Collision Violation
Color and Enabling
Enclosure Visibility

Q: How do you define connectors between the boards?

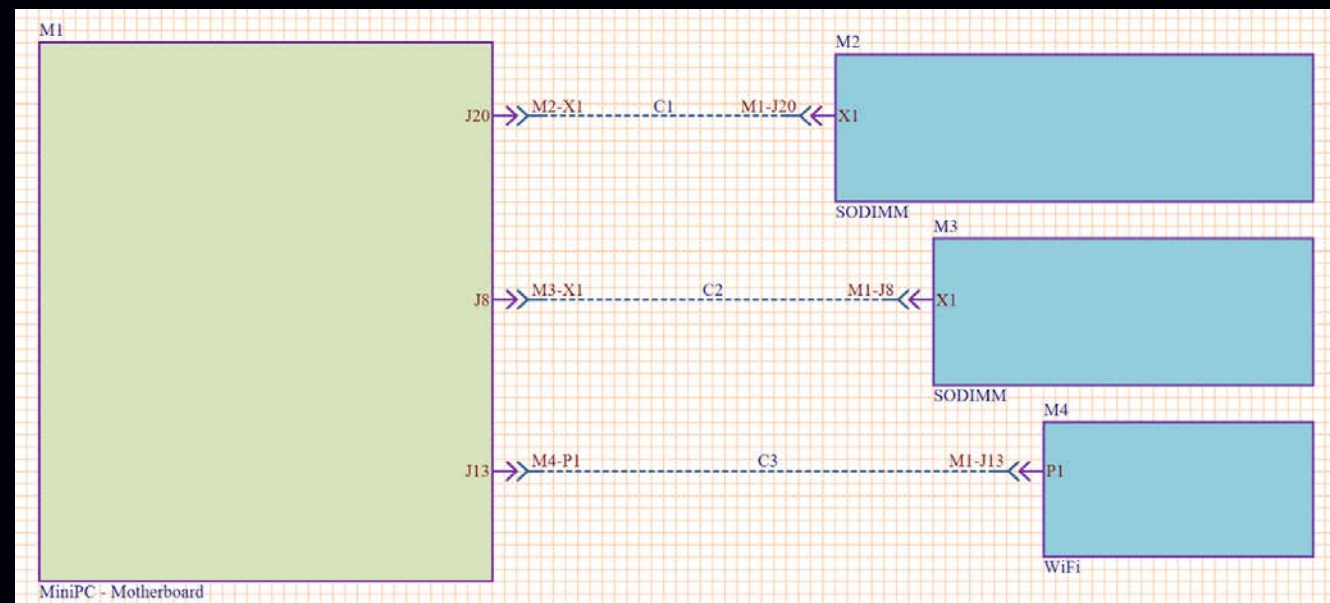
A: You define connectors by setting the “System” component parameter to "Connector."



Properties Panel

Do you have to individually import separate PCB project into a multi-board project?

A: There are two ways to add multiple individual designs to the multi-board schematic. You can add existing projects in the Projects Panel and you can define modules and connect them to existing designs. Connecting modules to a design will automatically add the design to the multi-board project. Defined modules can be associated to any existing design with the Properties Panel.



Adding Modules to the Multi-board Schematic

Q: How do you aligned connectors?

A: Altium Designer uses plane-to-plane and axis to axis alignment to get the perfect fit of all of your boards. The SODIMM and WiFi boards have been aligned to using a combination of the two methods with the connectors on the MiniPC board.

Q: Can a multi-board design output a parts list for the connectors/pins of cable and harness connections?

A: You can define a “manual solution” in the multi-board or in any design ActiveBOM. This allows you define connectors, pins, and harness in your BOM.

Q: What happened on the original PCB file when you move a component on the multi-board assembly?

A: Moving a component in the multi-board assembly, moves the footprint on the original pcb design file. Although you must currently update the tracks and polygon in the original design file, we are looking at developing the feature further to facilitate placement optimization.