Altium

ALTIUMLIVE:
PCB DESIGN VS PRODUCT DESIGN: UNLEASHING THE POWER OF EFFECTIVE MULTI-BOARD DESIGN

Cherie Litson CID+
Litson Consulting, Designer/Consultant/Instructor

David Haboud
Altium, Product Marketing Engineer

Munich
January 16, 2019

Agenda

1. **Multi-Board Challenges**
2. Multi-Board in Altium Designer
3. Resolving Challenges

Multi-Board Challenges

How do you manage…

I. System Level Design Strategies
II. Form & Fit
III. Connectors and Connections

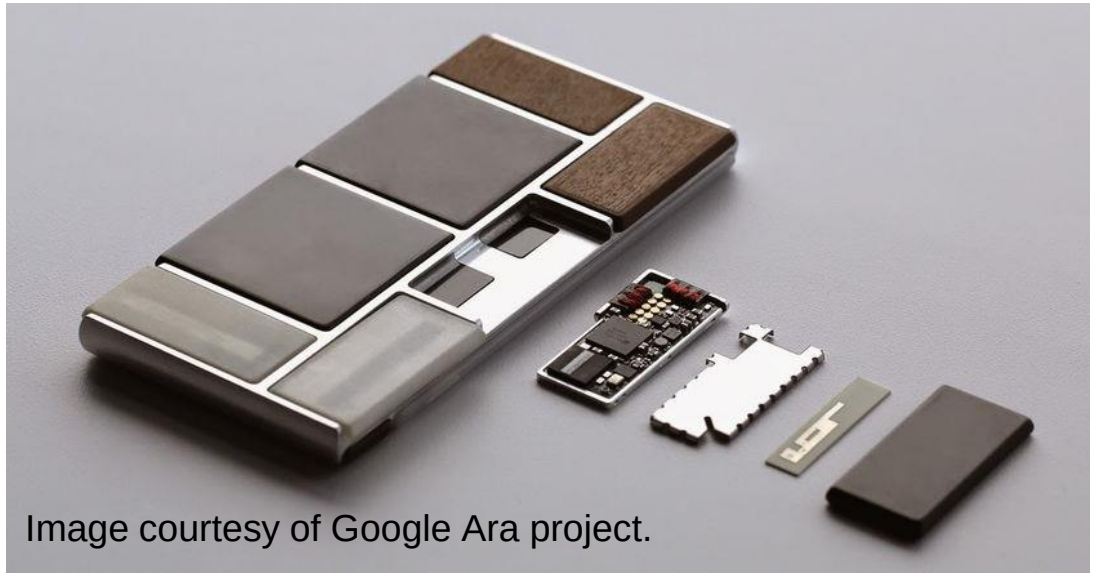


Image courtesy of Google Ara project.

System Level Design Strategies

In any system level solution, we look for

1. Definition (what),
2. Collaboration (who),
3. and Tools (how).

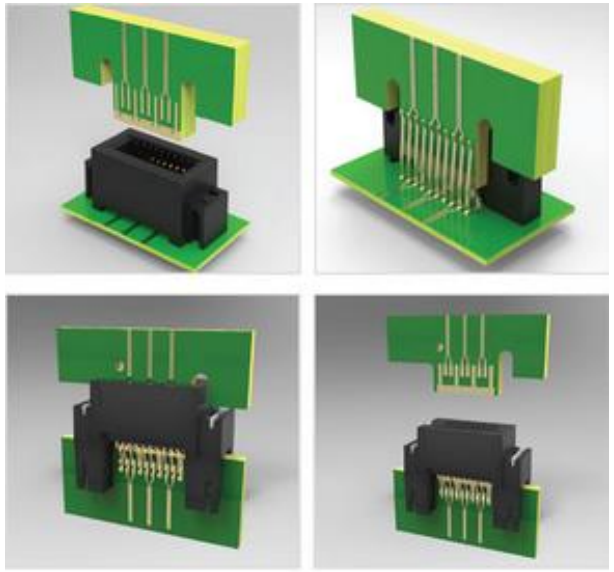
System Level Design Strategies

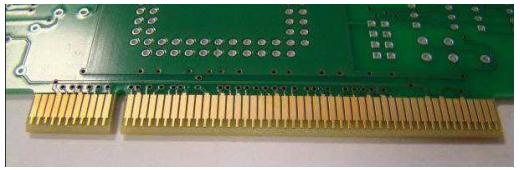
So, let’s look at WHAT first:

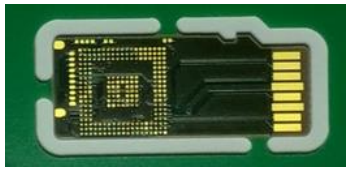
Many connectors are dual sourced. Meaning, two companies are responsible for the manufacturing of each mating part of the connection.

Examples of these are:

- Edge Connectors
- Wires
- Memory Chip Connectors (SD or PC)

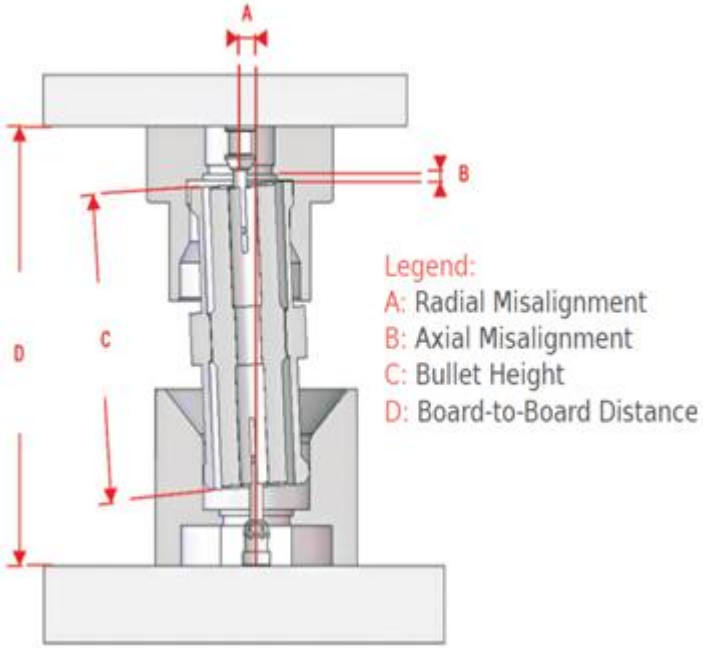


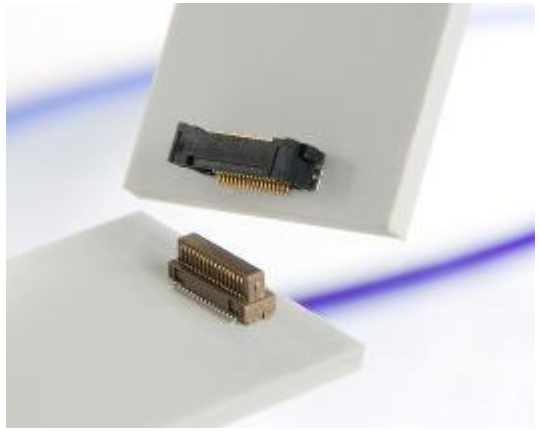




Alignment and Orientation becomes difficult to manage.

Even single source connector systems have challenges with Alignment and Orientation.

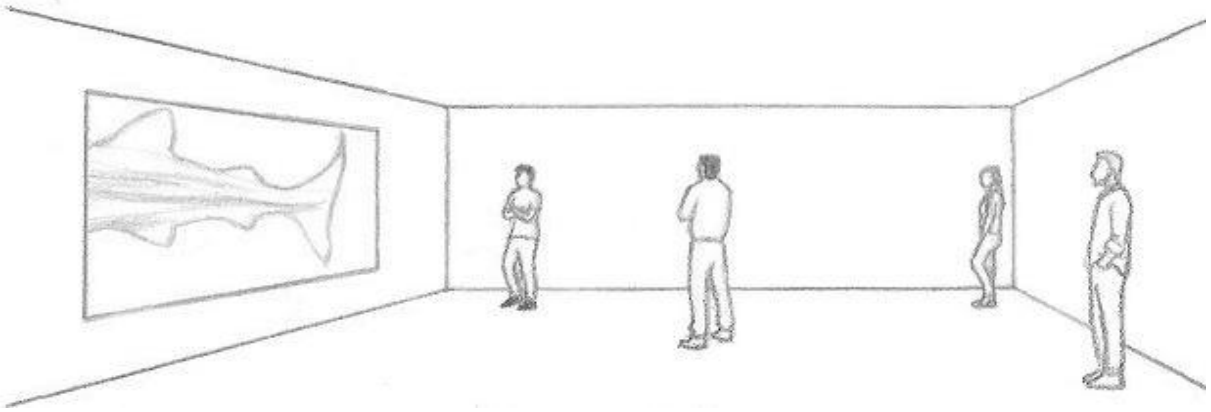




Hirose B2B connector

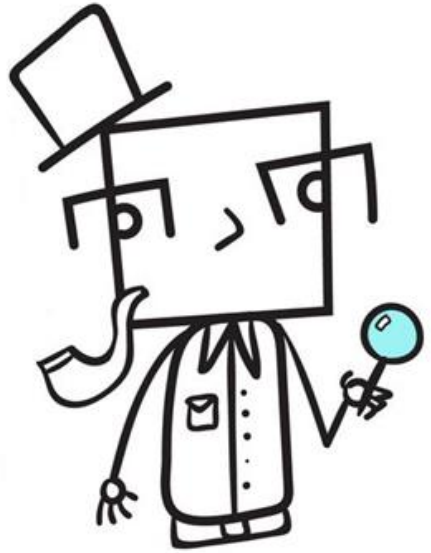
WHO? Collaboration involves many different perspectives.

The EE wants to show the signal coming in &/or going out from each source.

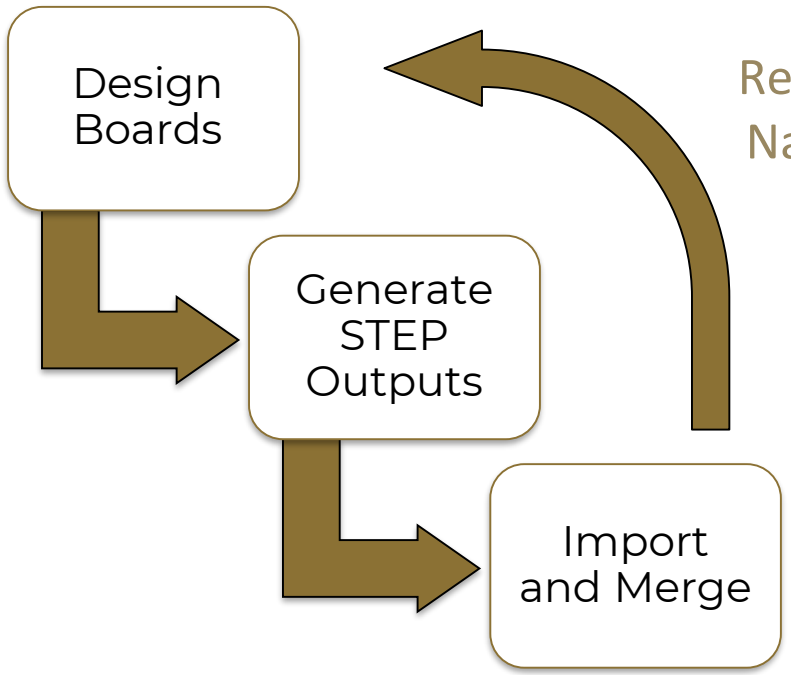


The ME wants to position the connector(s) so they are accessible.

The Designer wants to be sure the correct signal is going through the correct connector and how best to swap pins to ease routing complexity.



Form and Fit



Repeat ad Nauseam

Time intensive model rendering and file transfers!

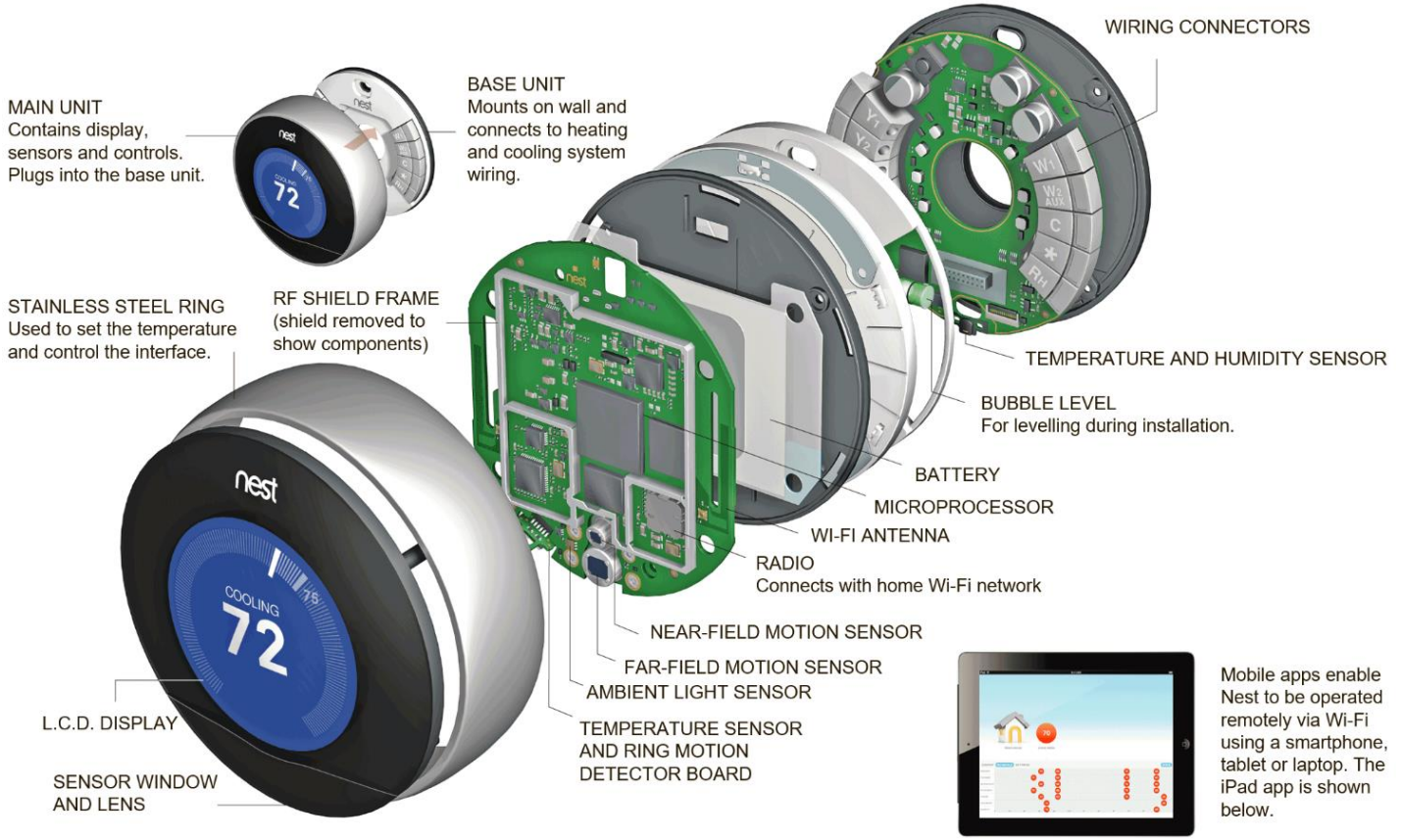
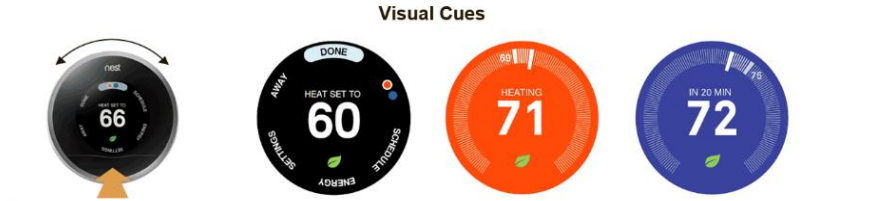


Image Courtesy of Nest Labs Installation Manual

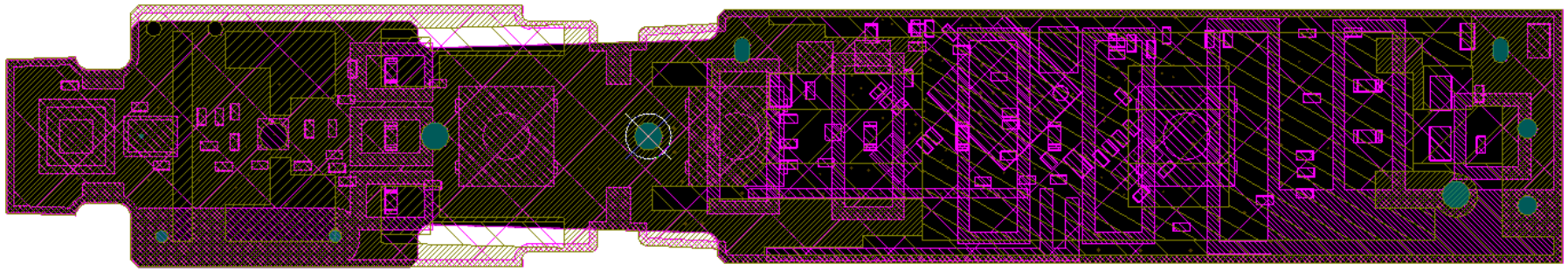


The display's background is normally black, but changes to orange when heating and blue when cooling.

Form and Fit

To track the Mechanical placement and clearances we use:

2D DFX files generated from one CAD program and imported another.



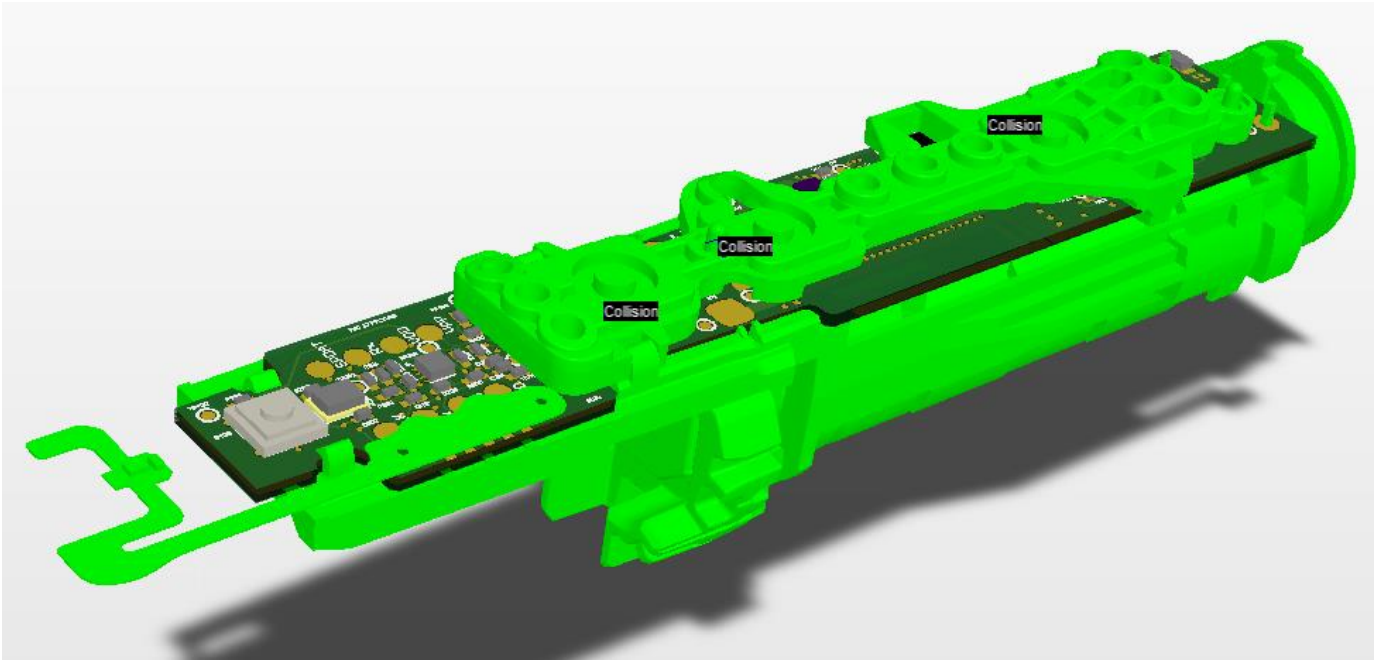
These require other additional pieces of data supplied by emails &/or pictures for heights and dimensions.

Form and Fit

Or we can import Mechanical placement and clearances using 3D STEP files:

These files are generated first from the MCAD software and imported into the ECAD software.

Then a STEP file is generated from the ECAD software and imported to into the MCAD software.



There are many issues with this process:

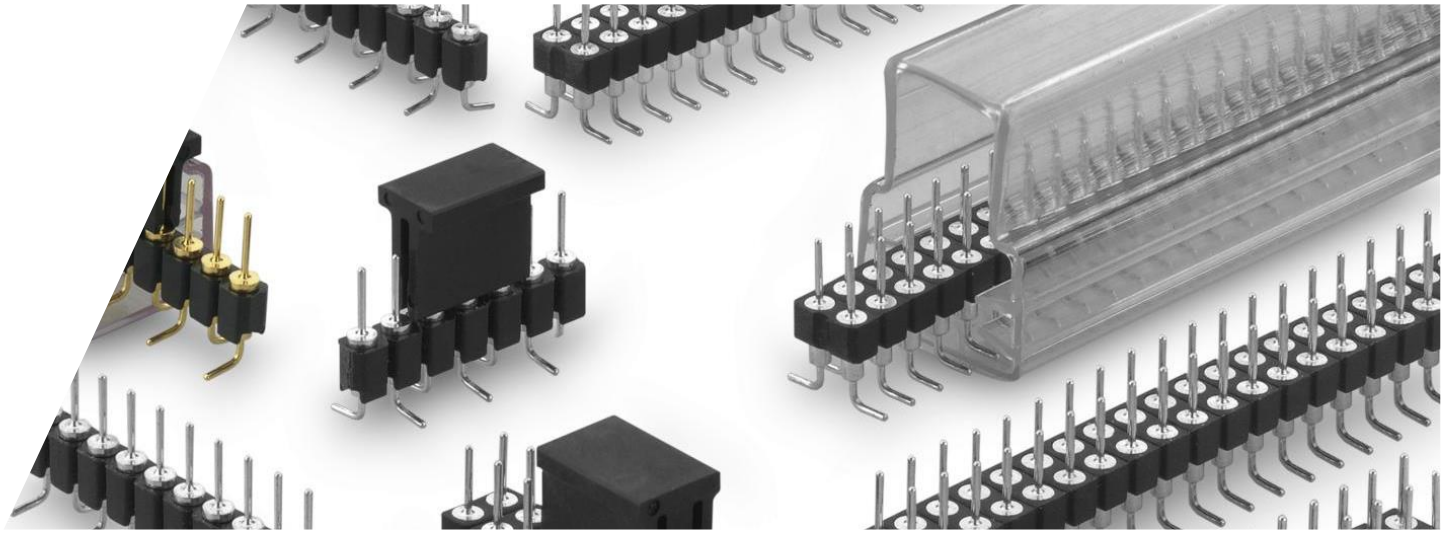
1. Alignment and Orientation are often different in each CAD package.
2. For connectors, the signals DO NOT Translate in this process.
3. Connector naming schemes are not coordinated or thought out.

Connectivity Management

- Pin Swapping
- Synchronizing Nets Across Boards
- Matching and Mirroring

Commonly managed with XLS or DOC files and Emails!

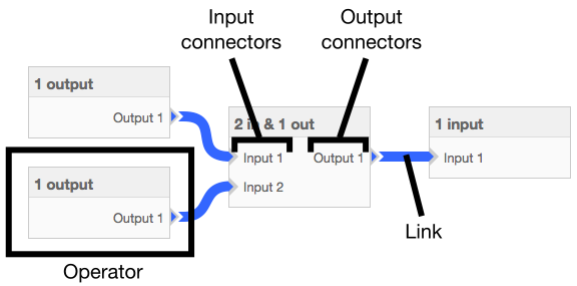
	NB2DSK-J15-3	SPK.SPK_R+	SPK01-HDR1-3	R_PLUS
	NB2DSK-J15-4	SPK.SPK_R-	SPK01-HDR1-4	R_MINUS
	NB2DSK-J15-5	GND	SPK01-HDR1-5	GND
	NB2DSK-J15-6	5V0	SPK01-HDR1-6	5V0
	NB2DSK-J15-7	SPK.SPK_L-	SPK01-HDR1-7	L_MINUS
	NB2DSK-J15-8	SPK.SPK_L+	SPK01-HDR1-8	L_PLUS
	NB2DSK-J15-9	EXTCTRL#.DIN2	SPK01-HDR1-9	DIN2
	NB2DSK-J15-10	EXTCTRL#.SCK2	SPK01-HDR1-10	SCLK2
	NB2DSK-J15-11	EXTCTRL#.CS1_N	SPK01-HDR1-11	CS1_N
	NB2DSK-J15-12	EXTCTRL#.CS2_N	SPK01-HDR1-12	CS2_N
	NB2DSK-J15-13	EXTCTRL#.DIN1	SPK01-HDR1-13	DIN1
	NB2DSK-J15-14	EXTCTRL#.1WIRE	SPK01-HDR1-14	ONE_WIRE
	NB2DSK-J15-15	EXTCTRL#.SCK1	SPK01-HDR1-15	SCLK1
	NB2DSK-J15-16	3V3	SPK01-HDR1-16	3V3
	nnection Name: Direct (C_2)			
	DaughterBoard-HDR_L1-1	EXTEND_A0	NB2DSK-HDR_L1-1	EXT_A.D0
2	DaughterBoard-HDR_L1-2	EXTEND_B0	NB2DSK-HDR_L1-2	EXT_B.D0
3	DaughterBoard-HDR_L1-3	EXTEND_A1	NB2DSK-HDR_L1-3	EXT_A.D1
4	DaughterBoard-HDR_L1-4	EXTEND_B1	NB2DSK-HDR_L1-4	EXT_B.D1



Connectors and Connections

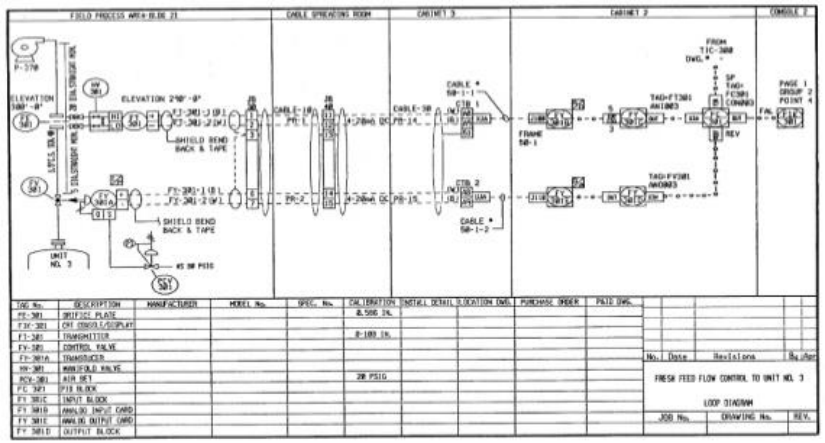
Today’s TOOLS are many and separate.
To track the Electrical properties of signal flow & logic we use:

Flow Charts

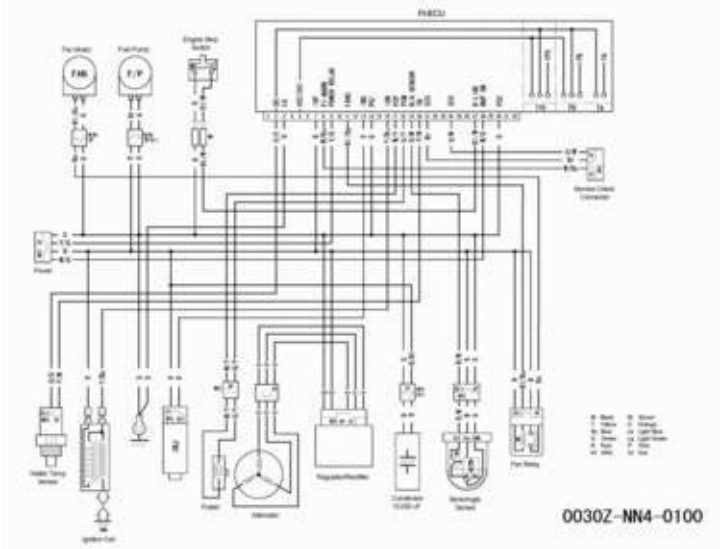


Pinout Diagrams,

Excel spread sheets



Wiring Diagrams



Connectivity Management starts with identification.

The IPC-7X51 (NEMA & MIL SPEC) have a naming conventions for Connectors and Mechanical Components. These are all very basic and specific to the Manufacturer of the connector.

Mostly they consists of (IPC shown):

abbreviation for Manufacturer's Name *(e.g. 3M, DEGSON, HARWIN…)* ***+ _ (underscore) + Manufacturer's Part Number (Manufacturer's Code)***.

These don't quite go far enough for our purposes….

We have to know which connector mates with what other connector?

To do this we have two reference designators: “J” & “P”.

These are a matching, Androgynes pair!

Their individual NUMBER then helps us to identify who goes with who:

J100 – P100
J101 – P101
J102 – P102
Etc.

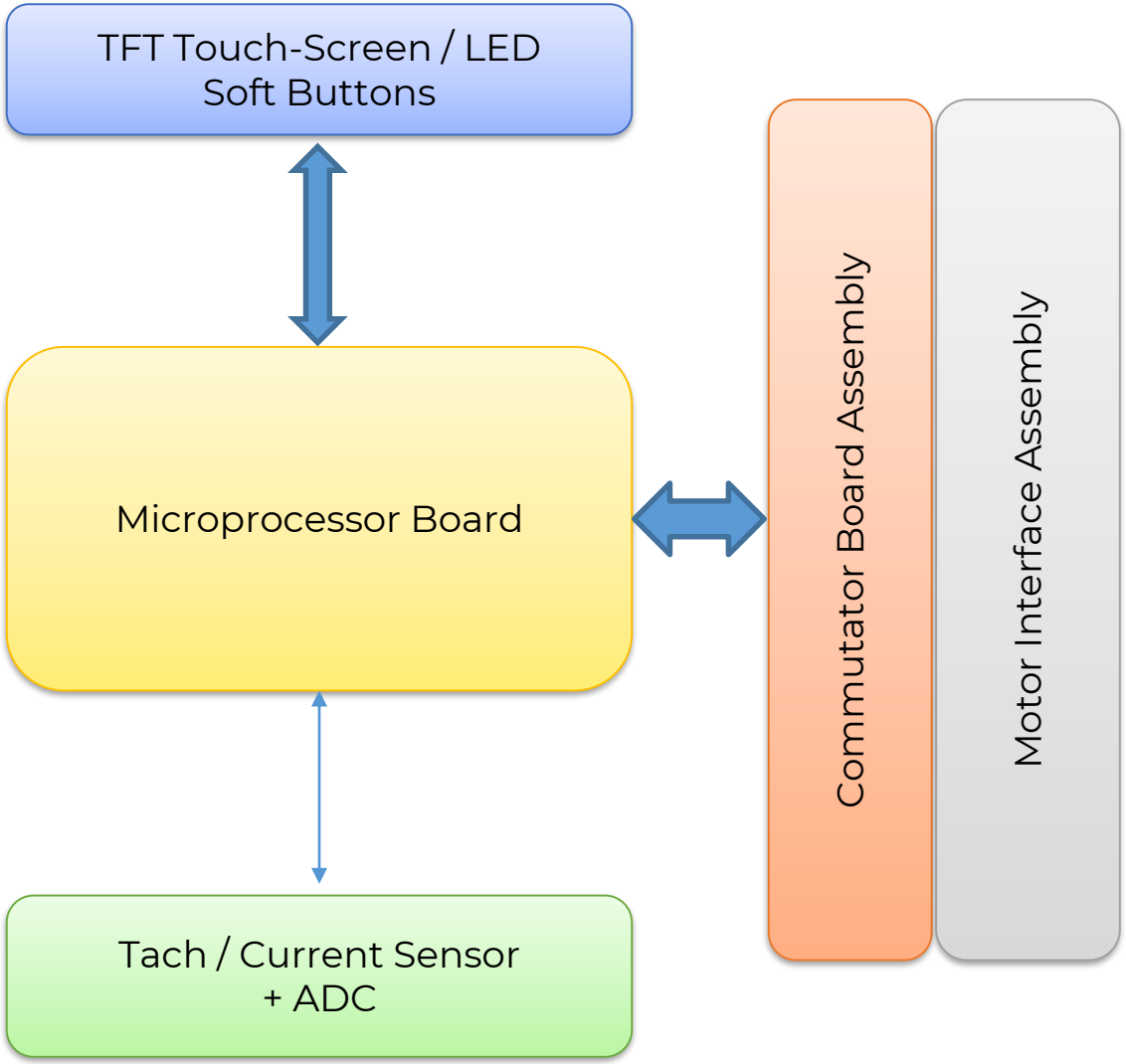
Now we are set to do some Pin Swapping and Signal (NET) management!

System Level Architecture

Edit boards in a system context

Verify system level connectivity on the logical and physical side

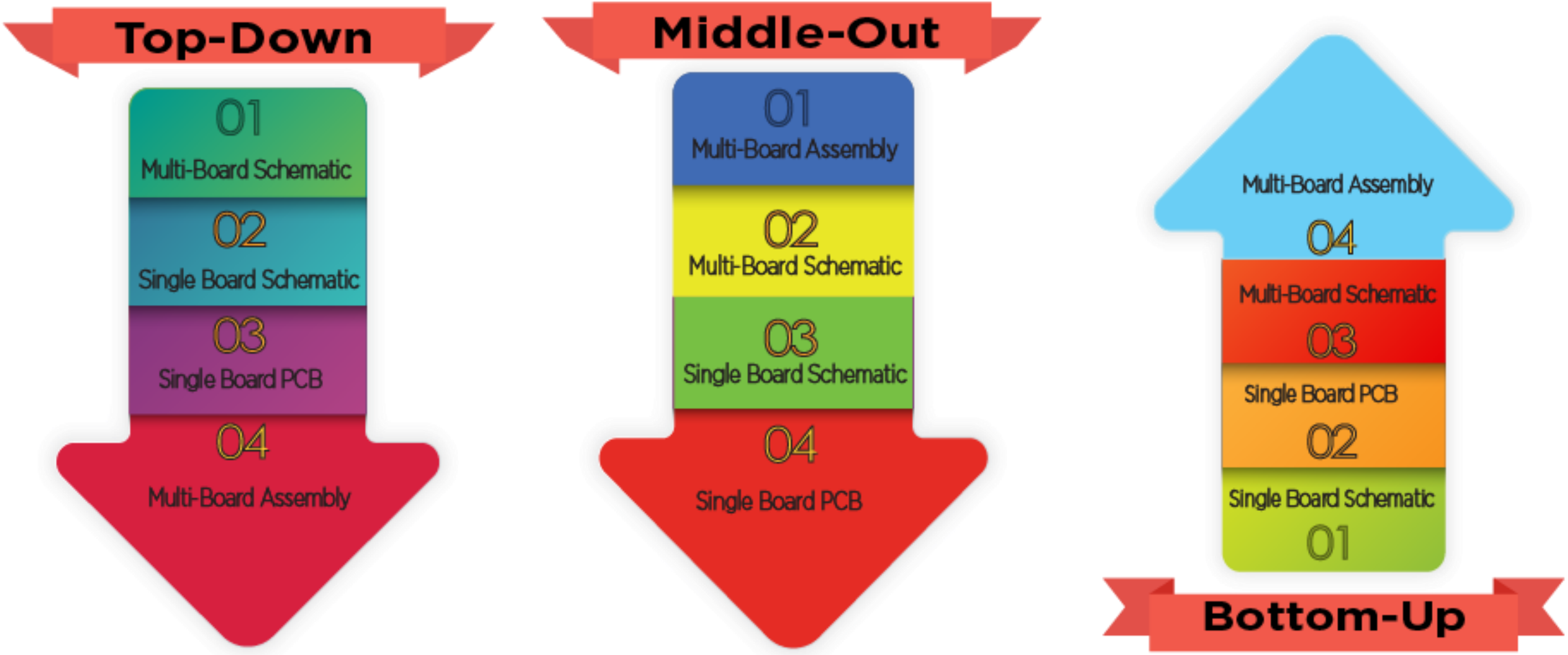
System Context Outside of Design Environment



Agenda

1. Multi-Board Challenges
2. **Multi-Board in Altium Designer**
3. Resolving Challenges

Design Methodology



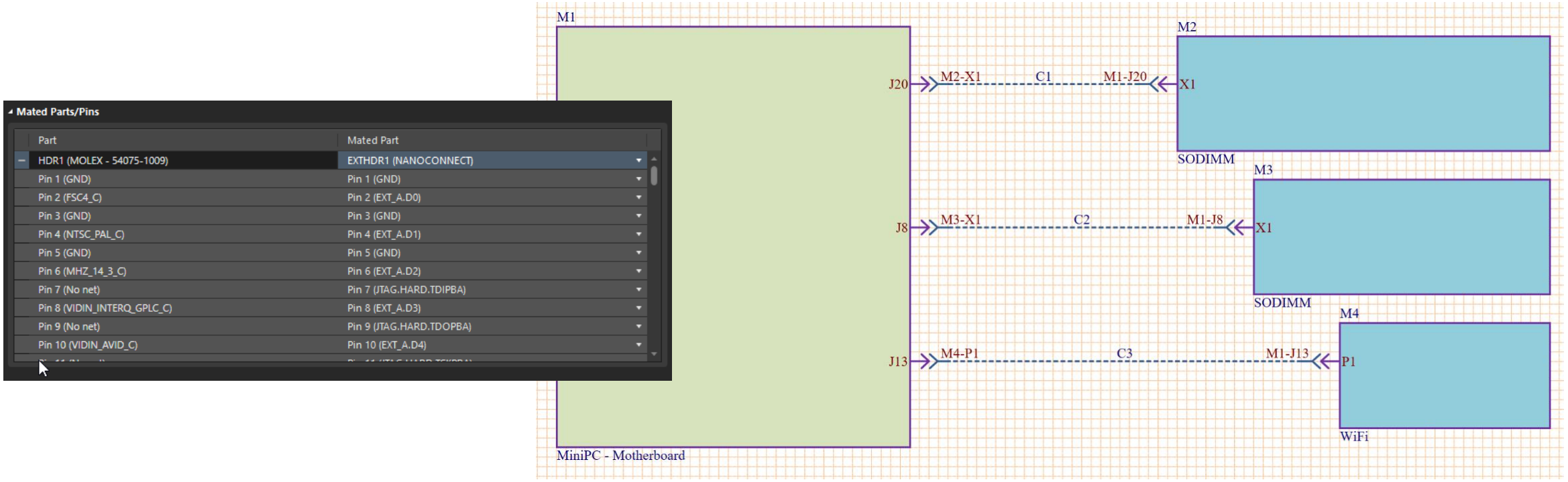
Multi-Board in Altium Designer

- Logical System-Level Design
- Electrical Rules Check
- Connection Management
- Assembly Creation
- Single Editing Environment
- Physical Assembly Optimization

Logical System-Level Design

Create logical design interconnections between modules

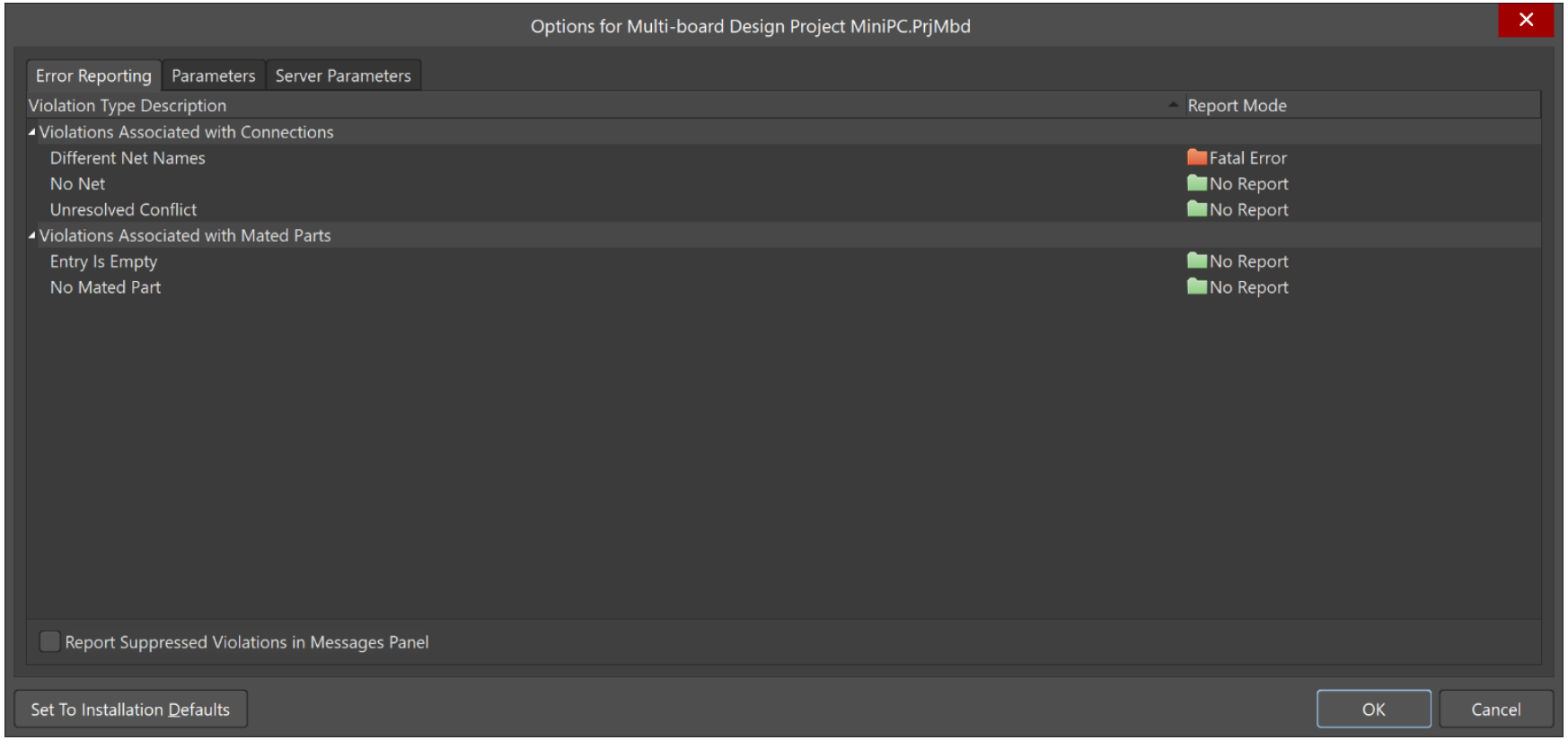
Module represents a complete printed circuit board project with all associated files



Electrical Rules Check

Connection Violations

Mated Part Violations



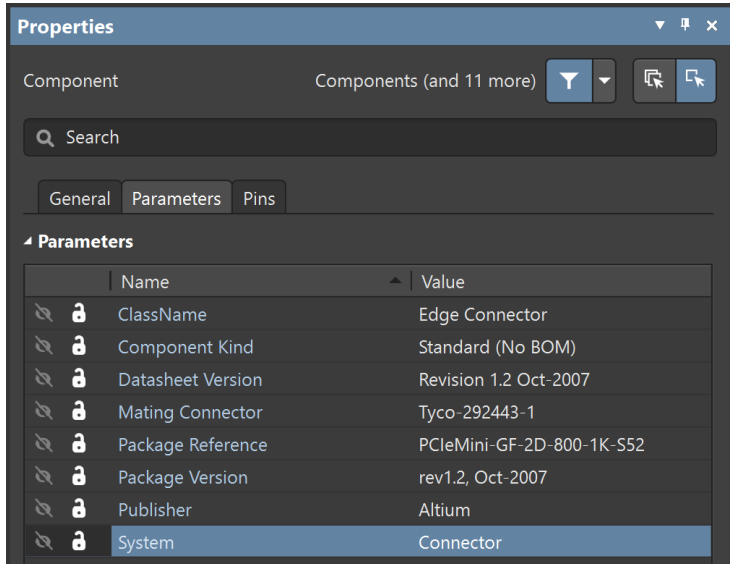
Connection Management

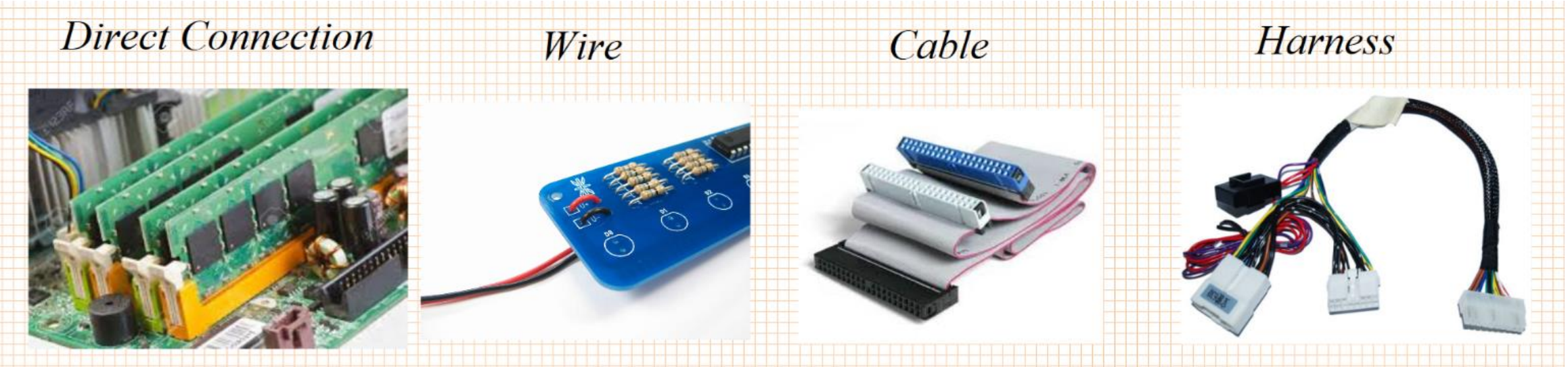
Direct Connection: Direct contact between boards.

Wire: A single wire connecting two points across boards.

Cable: An inseparable bundle of wires used to connect boards.

Harness: A collection of cables and wires connected two or more points across two or more boards.





Connection Manager

Track signals across each PCB layout

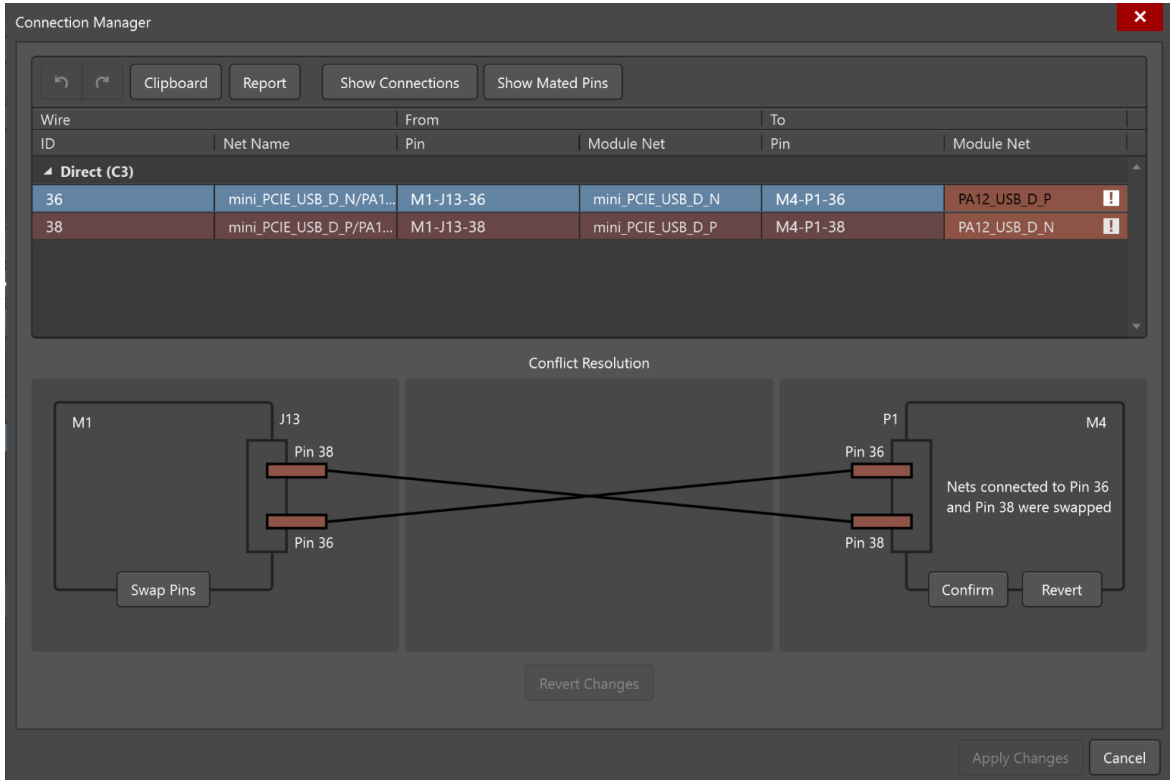
Conflict Resolution

Validate pin swaps and connectivity changes across designs to ensure acknowledgment of changes between teams

Confirm - Approves swapping without any changes

Revert - Cancels changes in first child project and requires back ECO to complete changes

Swap Pins – Replicates changes in mated part.

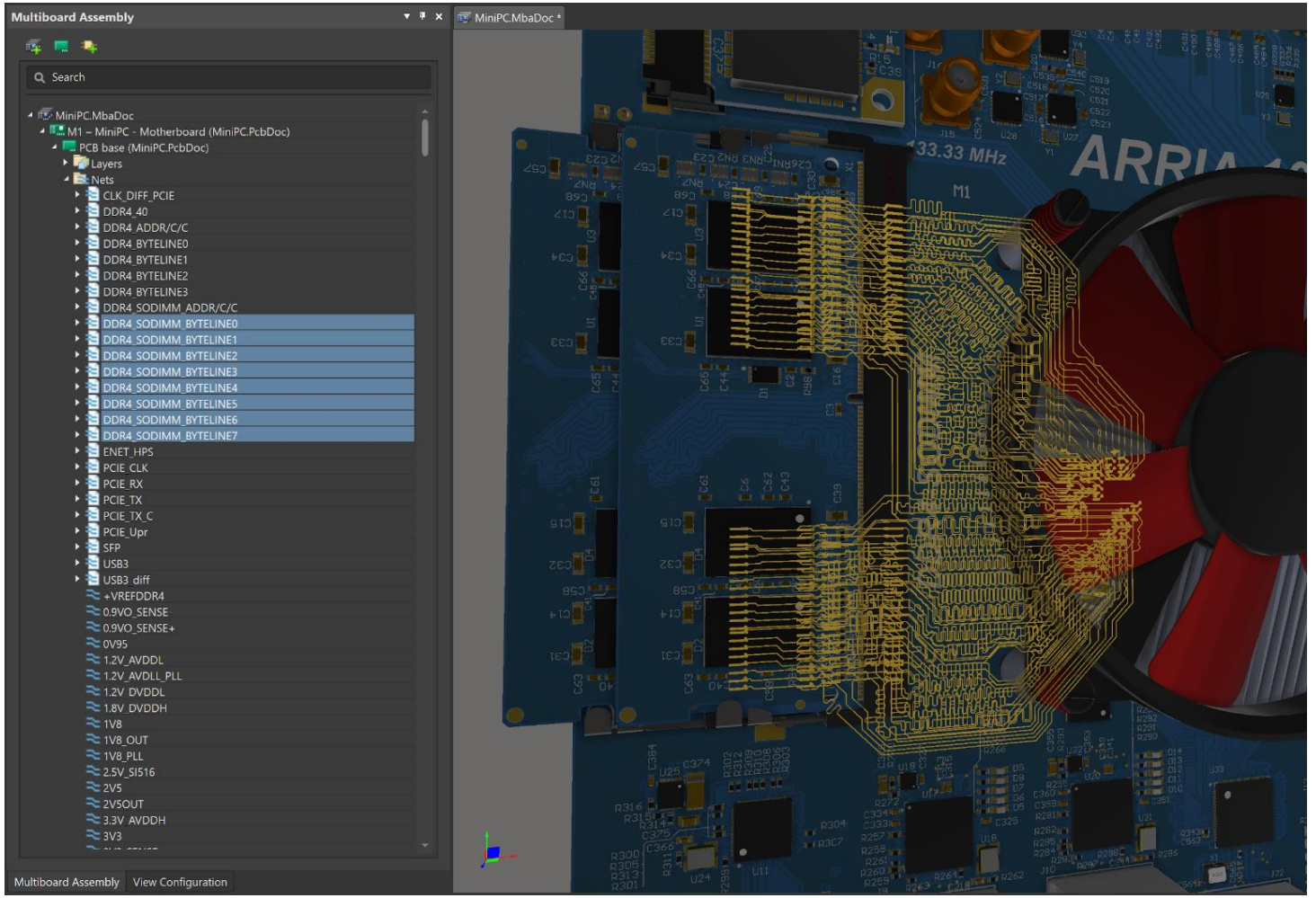


Assembly Creation

Physical connections between individual designs and enclosures

Navigate all assembly aspects

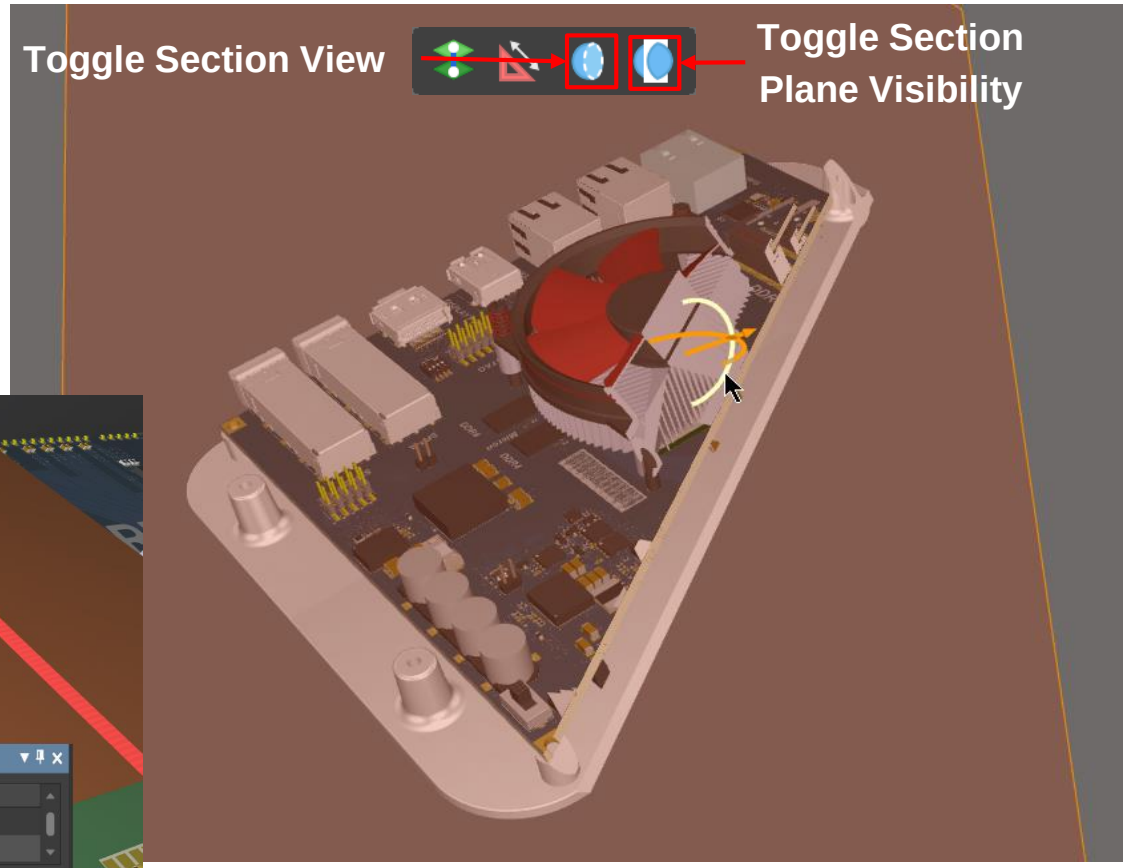
Track signal connectivity on a physical and logical level.

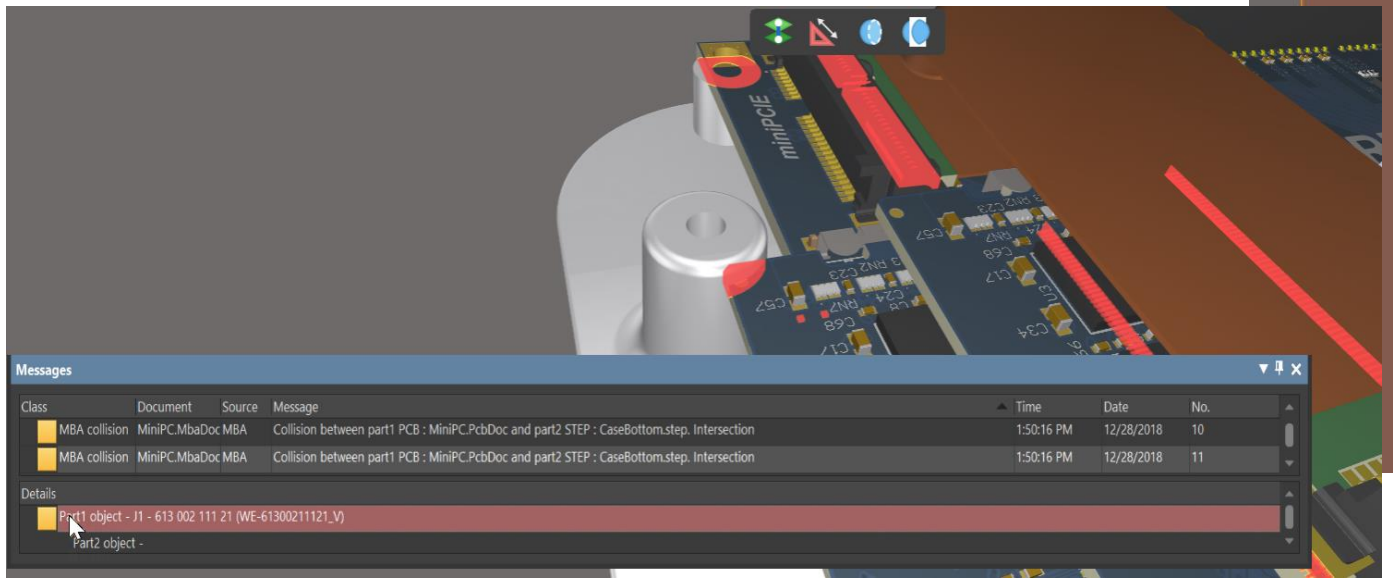


Assembly Creation

Visual verification of position and enclosure fit

Adjustable and X/Y/Z plane section cutout

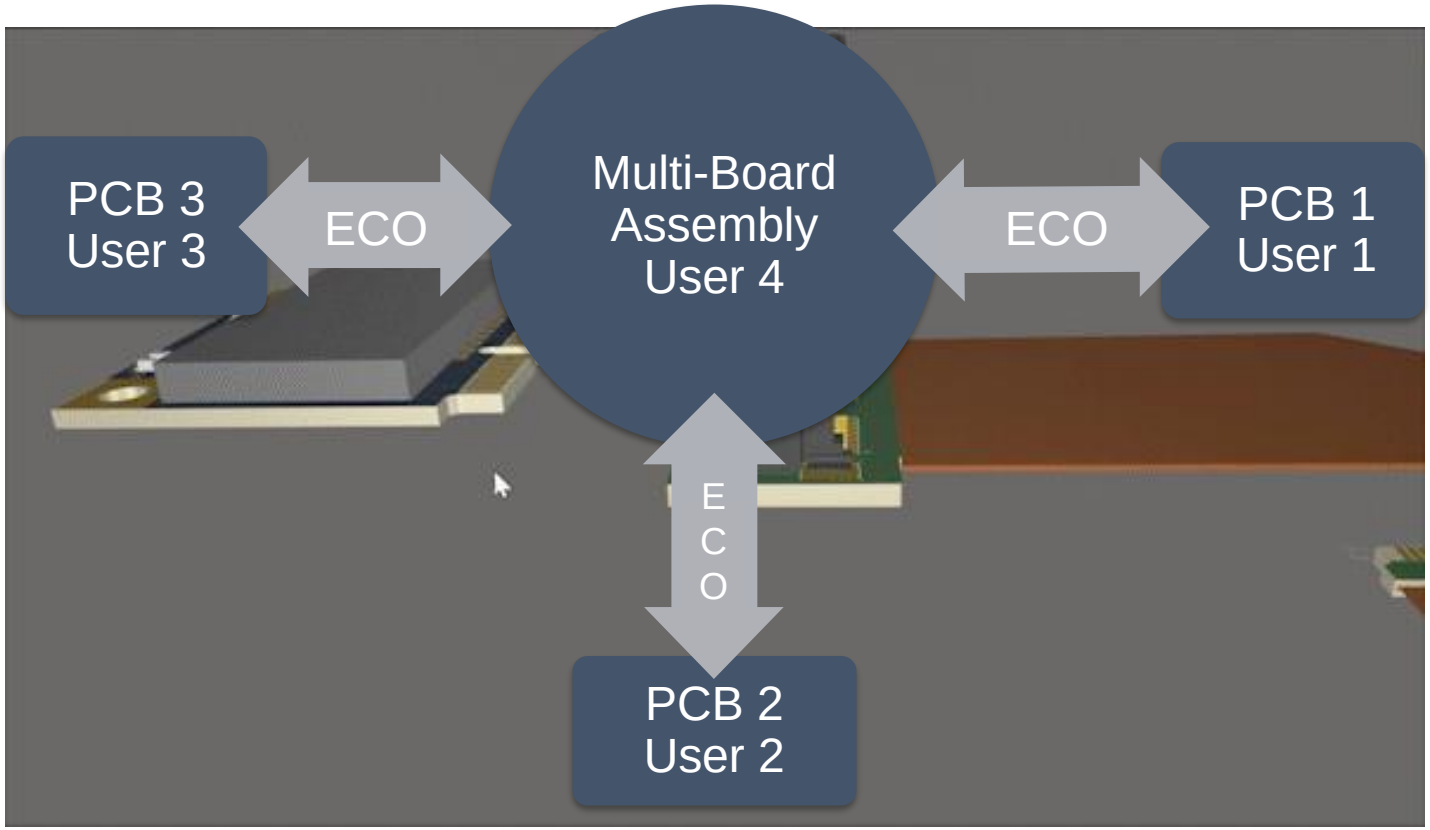




Single Editing Environment

Precise board alignment

Two point, plane-to-plane, and axis-to-axis alignment



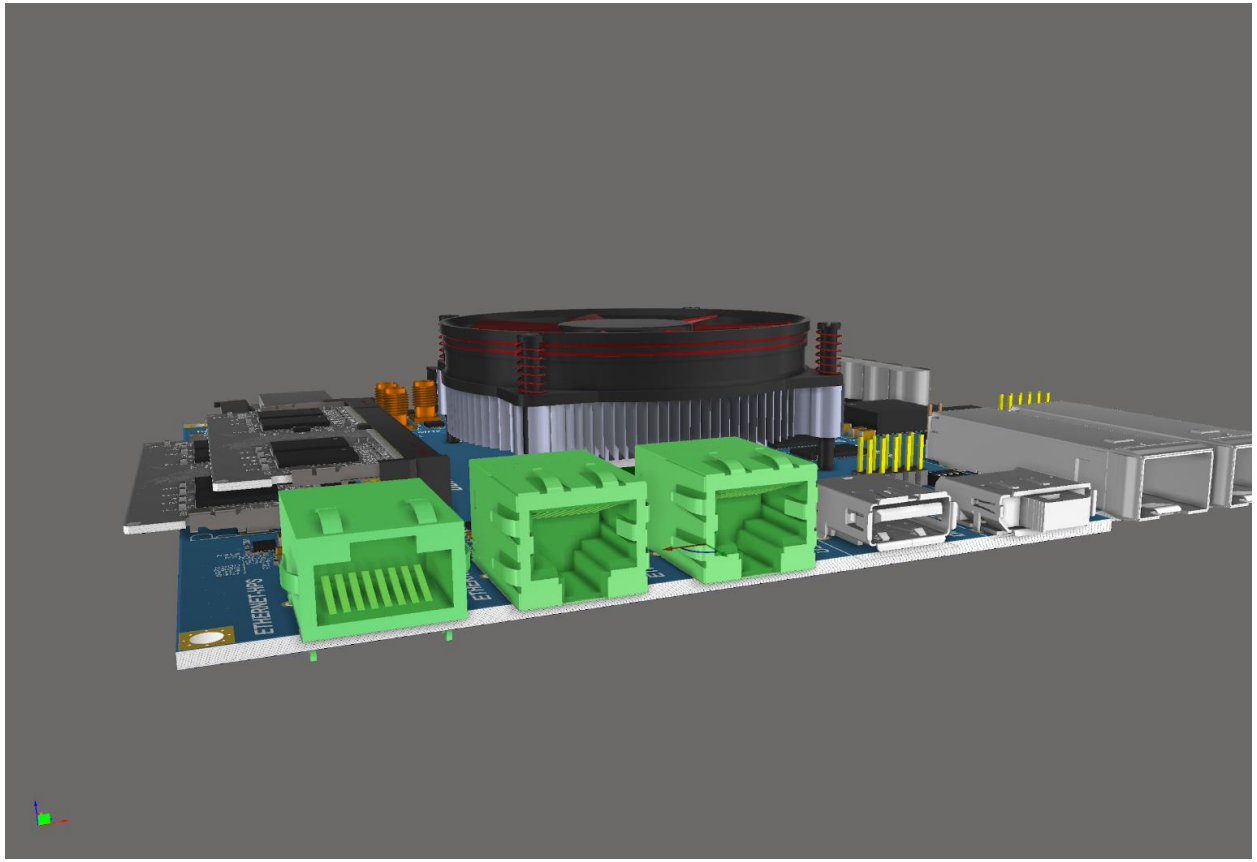
Physical Assembly Optimization

Move components on any selected board in the assembly

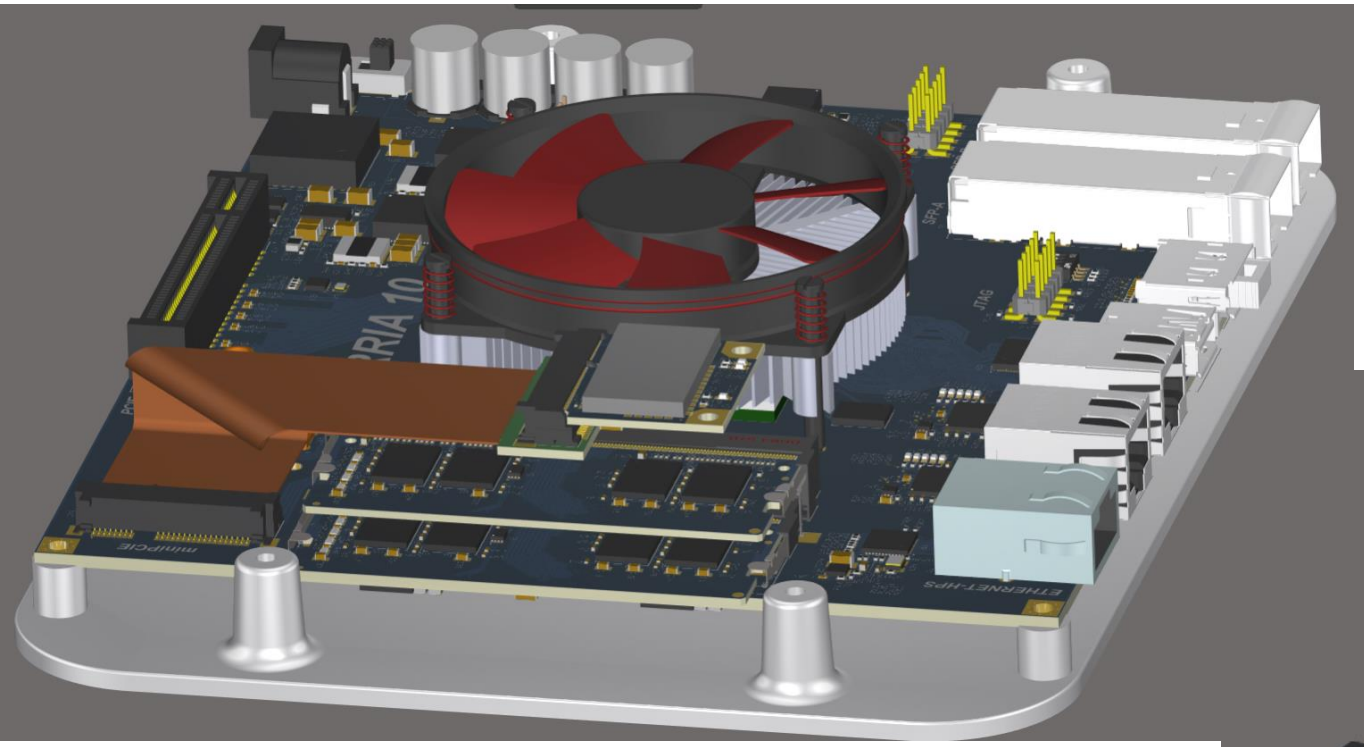
Changes sent to the original PCB design

Ensure relative position while allowing placement optimization

Measure distance between design aspects

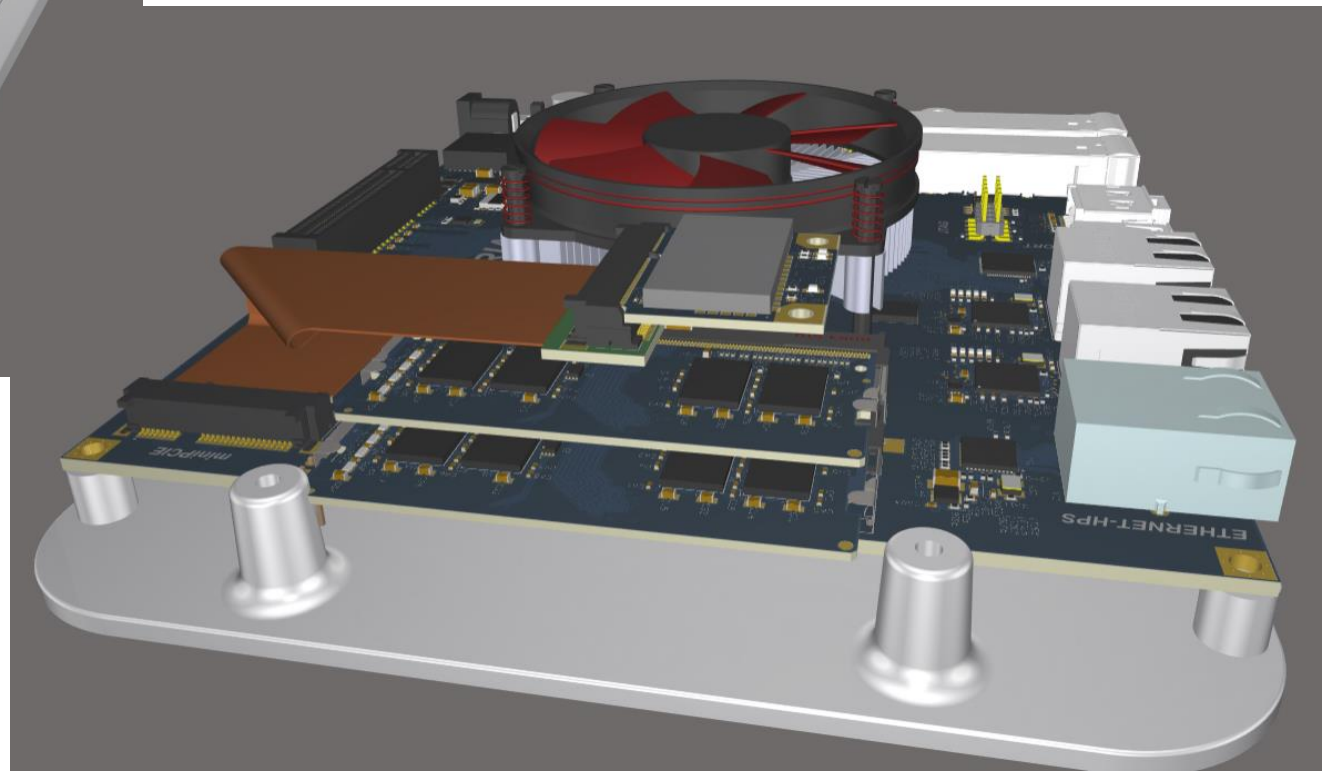


Physical Assembly Optimization



Orthographic View

Perspective View



Agenda

1. Multi-Board Challenges
2. Multi-Board in Altium Designer
3. **Resolving Challenges**

DEMO

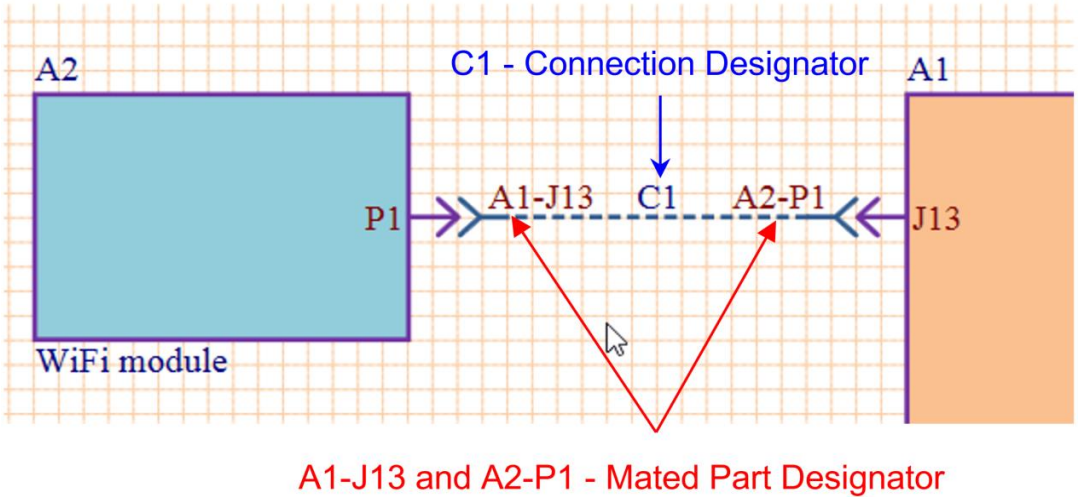
Demo Recap Highlights

I. System Level Design Strategies

✓ Logical System-Level Design

✓ ECO Driven Design Synchronization

✓ Visualizing Your Product's Interior



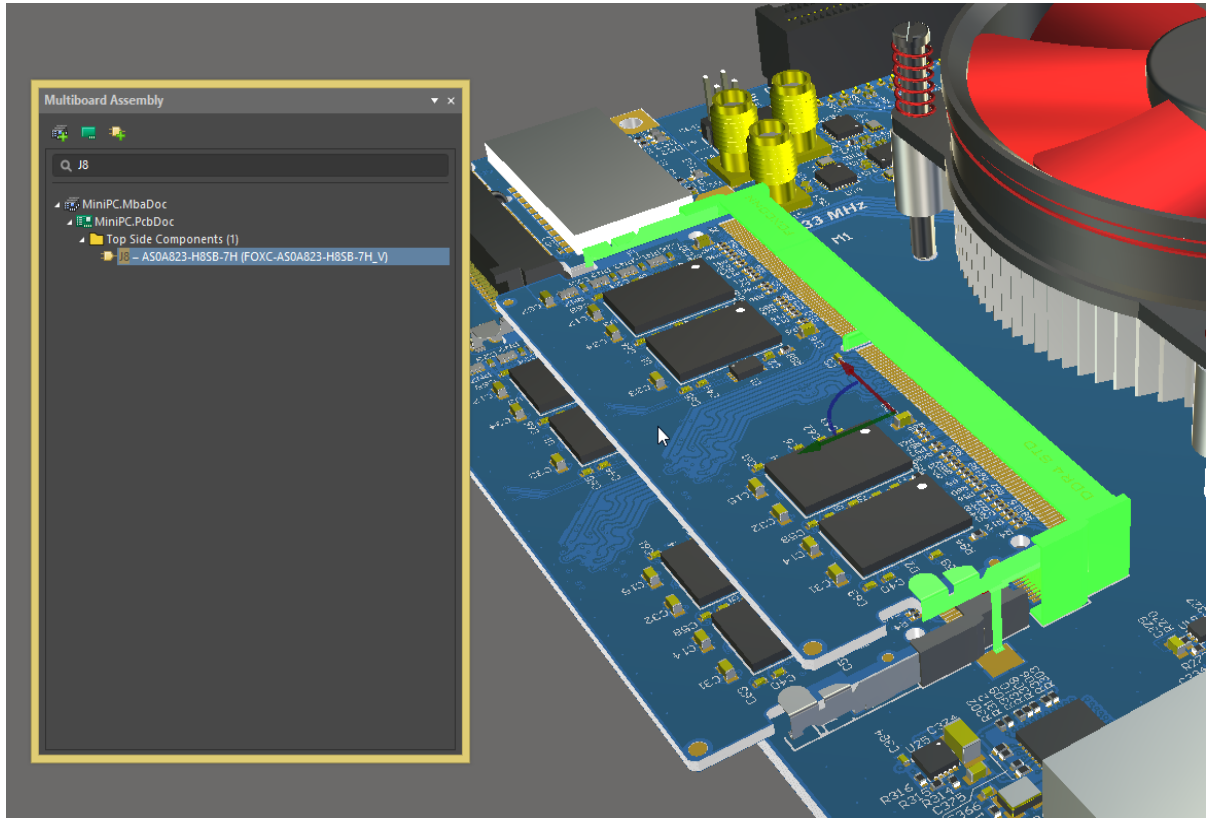
Demo Recap Highlights

II. Form & Fit

✓Assembly Hierarchy Navigation

✓Board Alignment

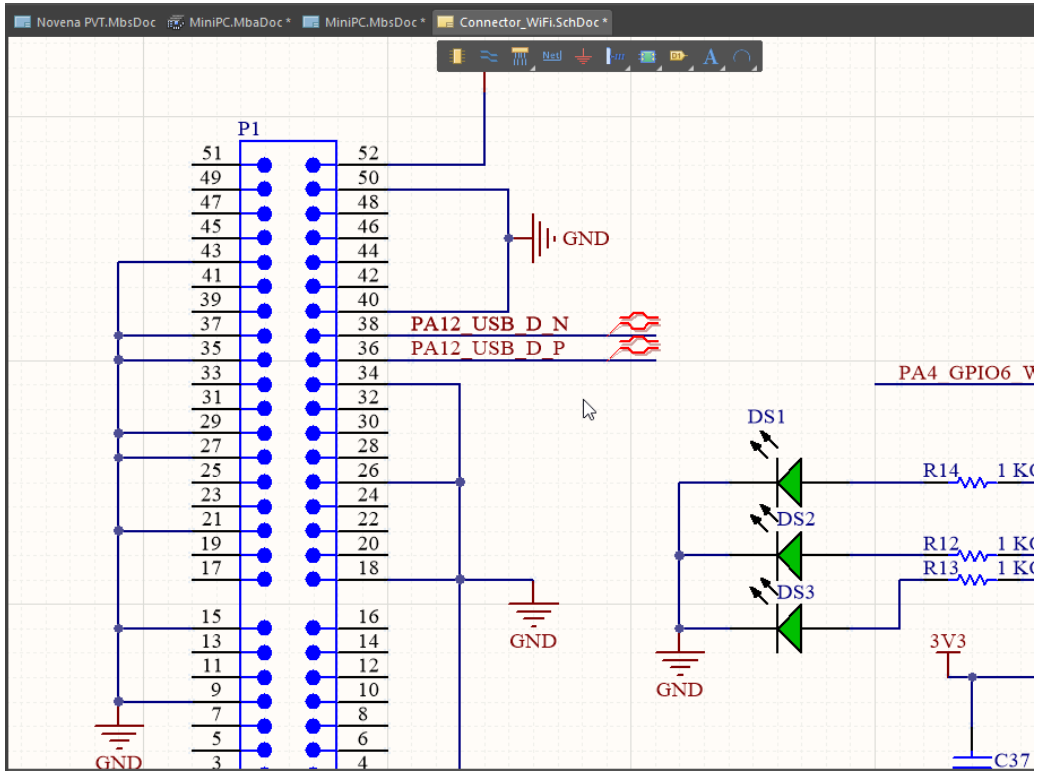
✓Optimized Part Placement



Demo Recap Highlights

III. Connectors and Connections

- ✓ Connection Definitions
- ✓ Electrical Rule Check
- ✓ Resolving Board Connectivity Conflicts



Altium

AltiumLive Questions?

Cherie Litson CID+
Litson Consulting,
Designer/Consultant/Instructor

David Haboud
David.Haboud@Altium.com
Product Marketing Engineer