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
1. Multi-Board Challenges
2. Multi-Board in Altium Designer
3. Resolving Challenges
How do you manage…

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

Image courtesy of Google Ara project.
In any system level solution, we look for
1. Definition (what),
2. Collaboration (who),
3. and Tools (how).
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.
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

Time intensive model rendering and file transfers!
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.
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 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.
Connectors and Connections

Connectivity Management

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

Commonly managed with XLS or DOC files and Emails!
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):

\[
\text{abbreviation for Manufacturer's Name (e.g. 3M, DEGSON, HARWIN...)} + \ _\ \ \ \ \ (underscore) + \text{Manufacturer's Part Number (Manufacturer's Code)}
\]

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
Multi-Board in Altium Designer

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

1. Multi-Board Challenges
2. Multi-Board in Altium Designer
3. Resolving Challenges
DEMO
I. System Level Design Strategies

✓ Logical System-Level Design
✓ ECO Driven Design Synchronization
✓ Visualizing Your Product’s Interior
II. Form & Fit
✓ Assembly Hierarchy Navigation
✓ Board Alignment
✓ Optimized Part Placement
III. Connectors and Connections

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