

# Multi-Board Design AltiumLive 2018 University Day



**David Haboud**

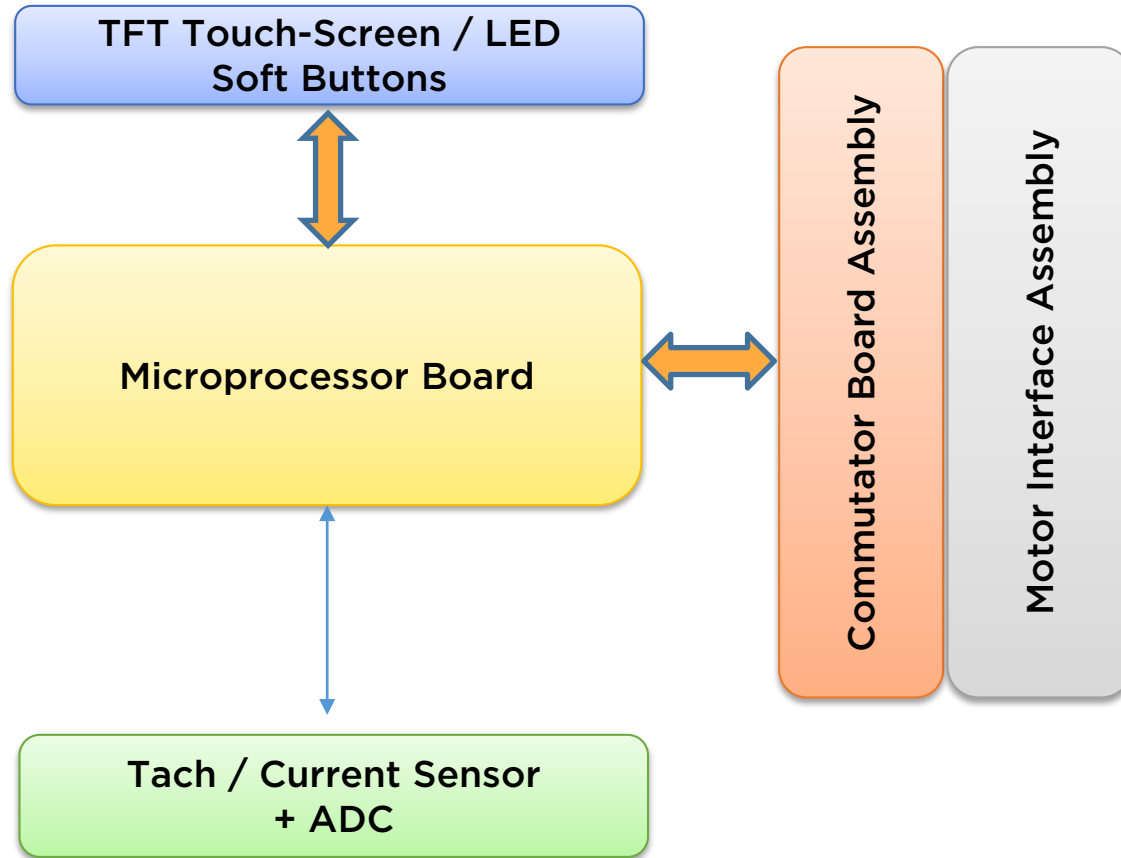
*Product Marketing Engineer*

**Altium**



- 1 **Design Methodology**
- 2 **Schematic Workflow**
- 3 **Assembly Workflow**
- 4 **Group Discussion**

# Design Methodology

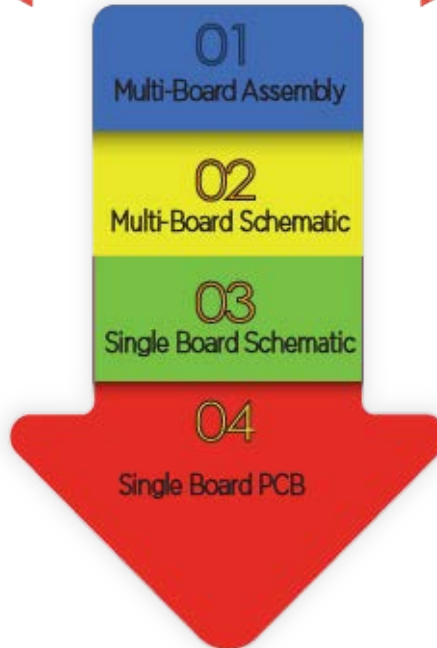




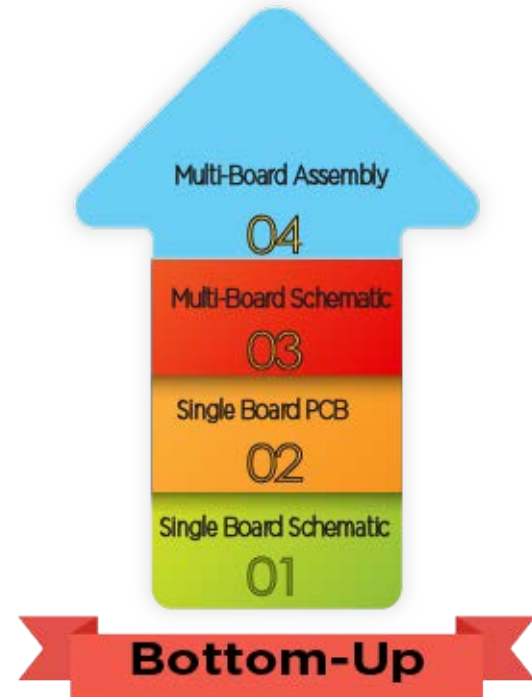
## Top-Down



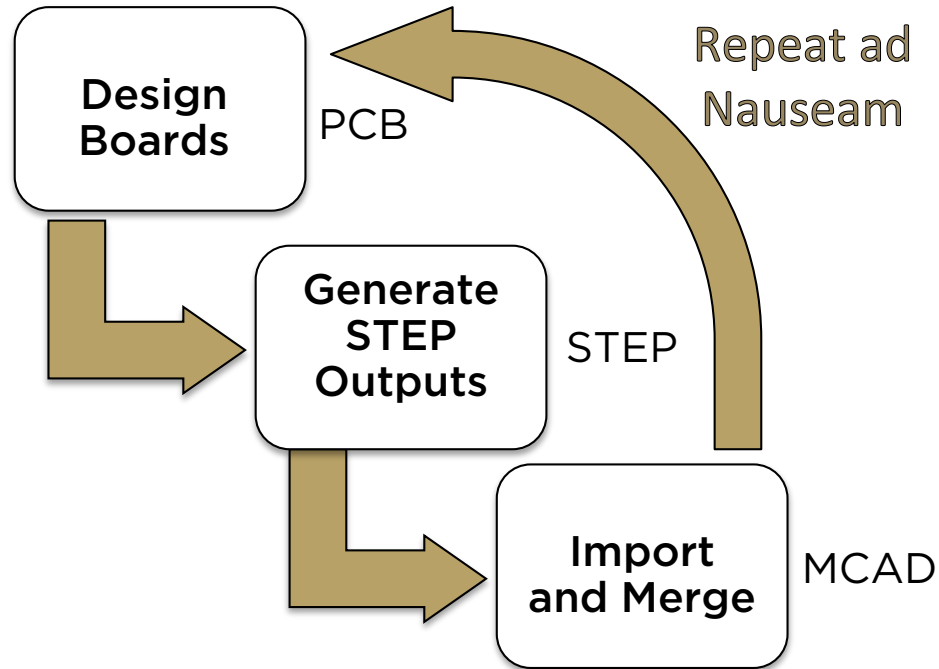
## Middle-Out



## Bottom-Up



# Design Methodology



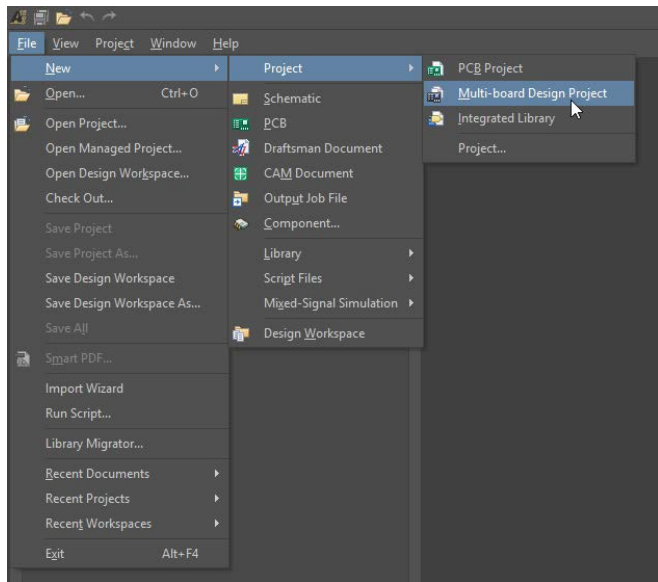


# Create New Project and Source Files



## Create a new multi-board project:

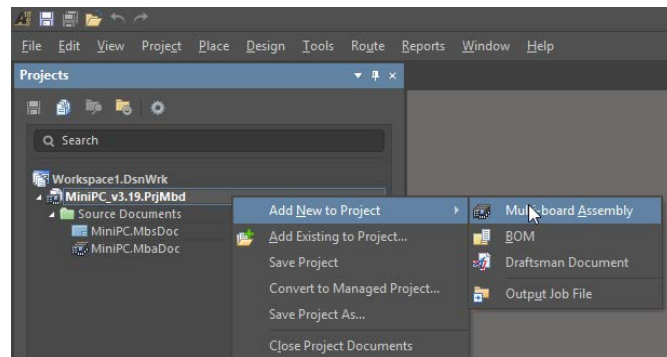
**File >> New >> Project >>  
Multi-Board Design Project (\*.PrjMbd)**



## Source files:

1. Logical design - Multi-Board Schematic document (\*.MbsDoc)
2. Physical design - Multi-Board Assembly document (\*.MbaDoc)

For adding source files into a project,  on the project name and then **Add New to Project... >> File Type.**



**Note: MBS and MBA documents should be saved before starting ECO process or linking Modules to child projects**



- 1 Design Methodology
- 2 Schematic Workflow
- 3 Assembly Workflow
- 4 Group Discussion

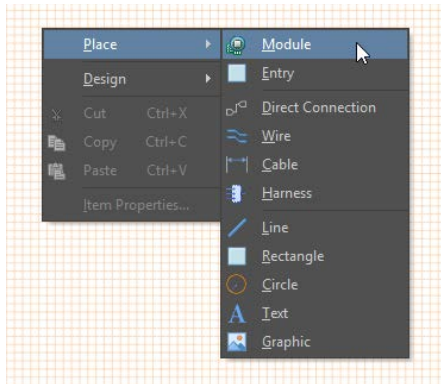
# Place Module and Link to PCB Project



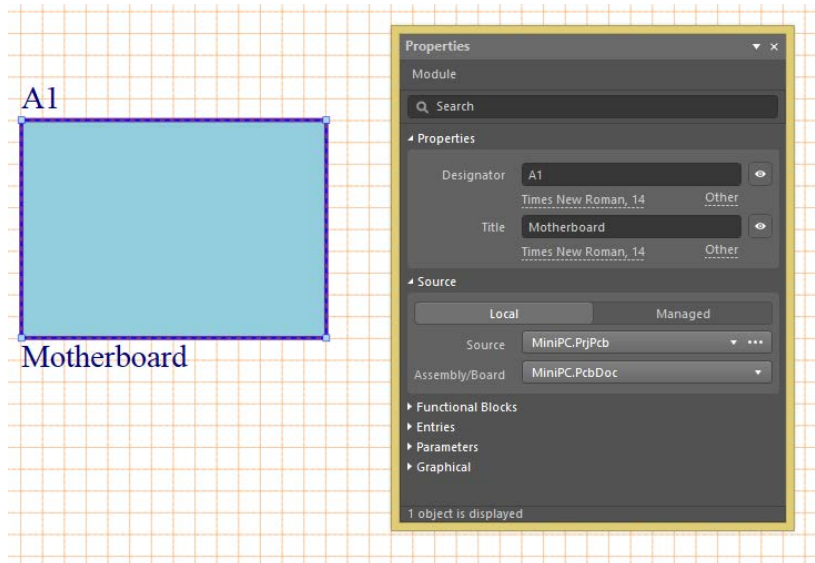
Place >> Module [P >> M]  
Or



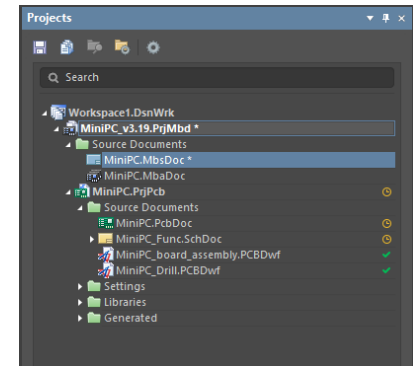
>> Place >> Module



Define Source (PCB or Multi-Board project) as  
in the Module's Properties Panel



Child projects are displayed in  
the Multi-Board project tree





# Prepare Child Project for Use in PrjMbd

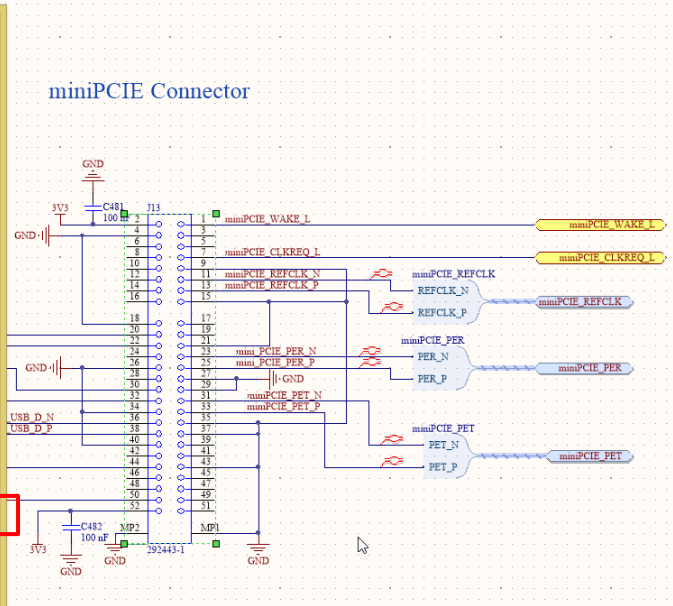


Add Parameter Name: **System**  
Set Parameter Value: **Connector** for part in single board schematics.

The Properties panel shows the following details for component J13:

- Designator: J13
- Name: 292443-1
- Part: 1
- Description: Board-to-Board Connector, SMT, Right Angle, Pitch 0.8mm
- Type: Standard
- Mode: Normal
- Local Colors: Fills, Lines, Pins
- Location: Board Template Connector
- Source: PCB Library: EN 25 00 002
- Parameters Table:
 

Name	Value
ClassName	Board Template Connector
Component Kind	Standard
Datasheet Version	Oct-2006
Manufacturer	PCB Library: EN 25 00 002
Package Reference	BTB-SM0-8-2H52F
System	Connector
- Subtypes: Add, Remove
- Links, Footprint, Pins



**Note: This step is already done for all MiniPC child projects.**

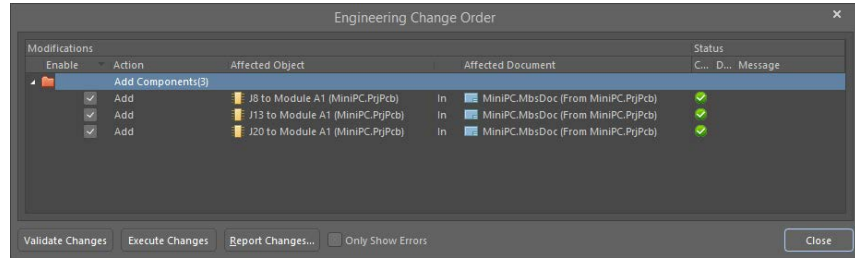
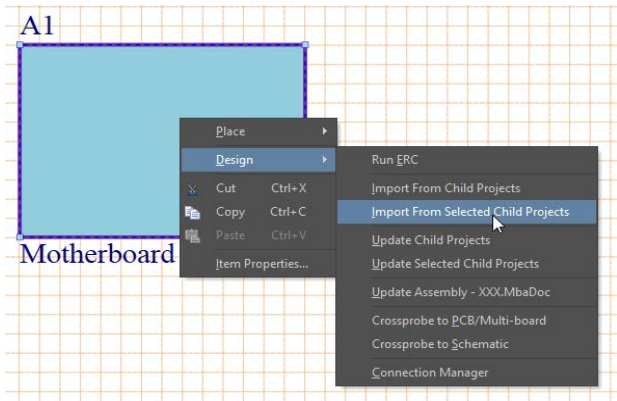
# Import Module Entries



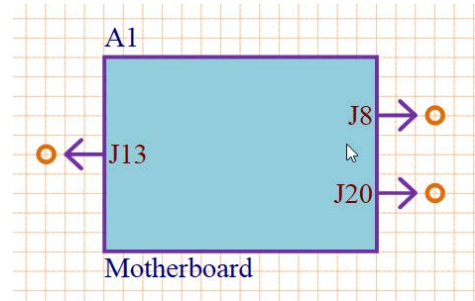
 **on Module**

Select **Design >> Import From Selected Child Project**

To update all modules,  
Select **Design >> Import From Child Projects**



Standard ECO dialog is using for data transfer control



**Note: Child project compilation executes on import. Using "Import From Selected Child Projects" is only recommended for quick data transfer for selected module only after small changes.**

# Add Connection Lines

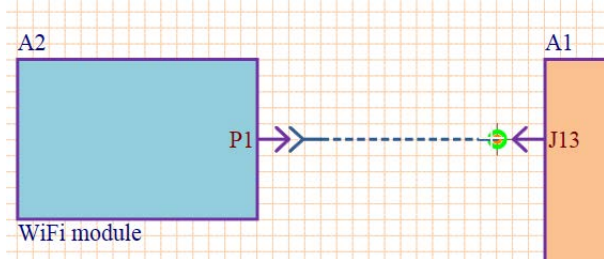
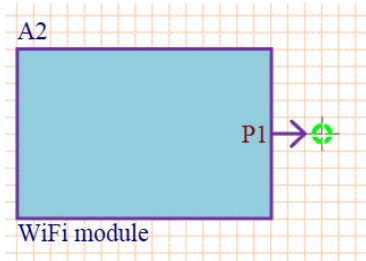


Place >> Direct Connection [Short keys: P >> D]

Or



>> Place >> Direct Connection

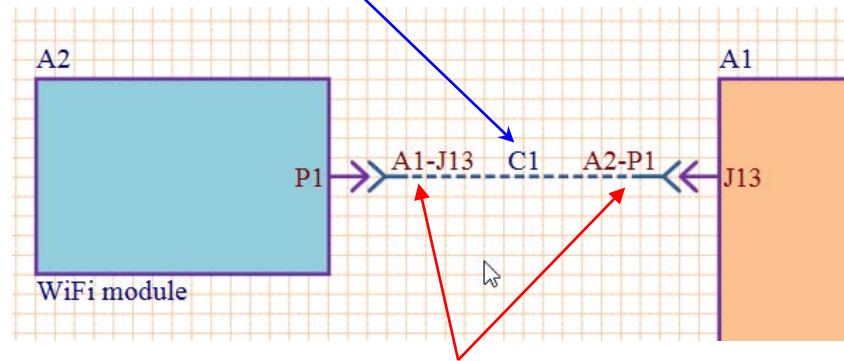


Green marker displays at valid locations to start and finish connections

**Note:** Direct Connection represents a direct board plugs into another board with direct contact. View the Glossary for summaries of all connection types.

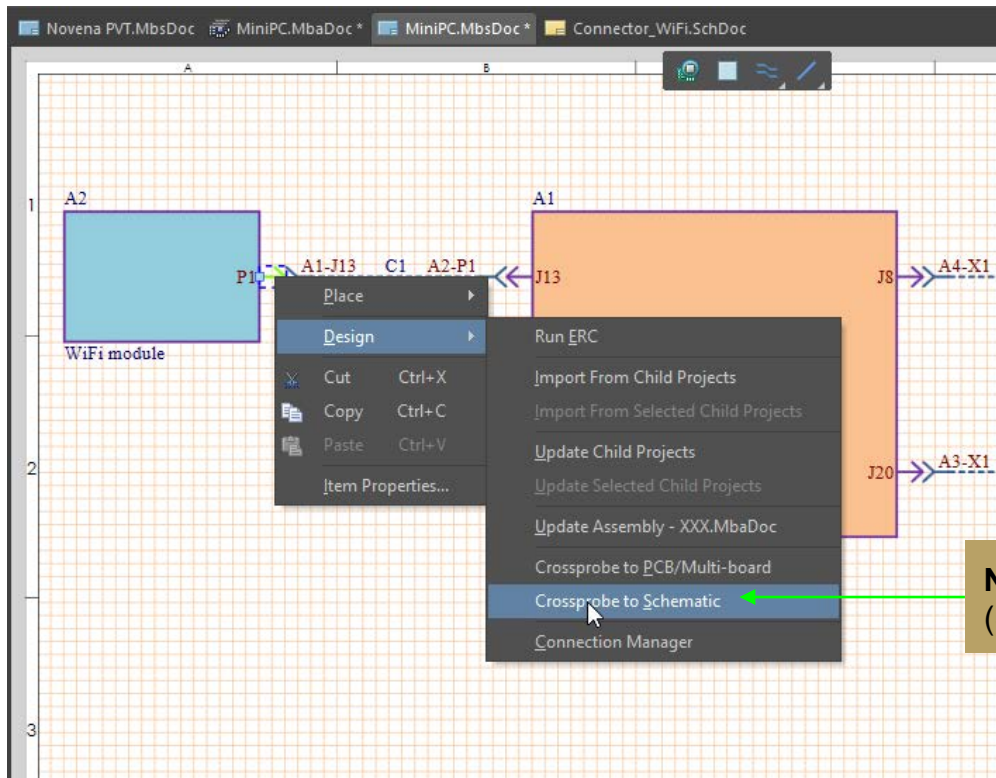
Following data is displayed for Direct Connection by default:

C1 - Connection Designator



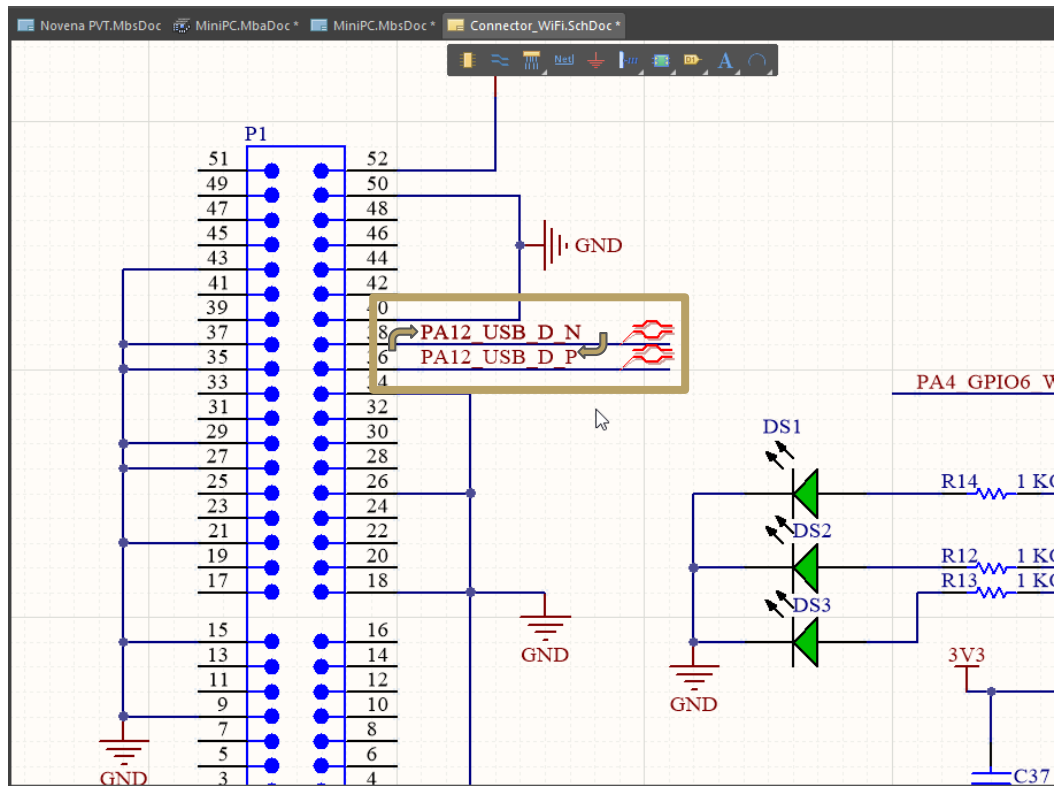
A1-J13 and A2-P1 - Mated Part Designator

# Cross-Probe to Child Project Schematic



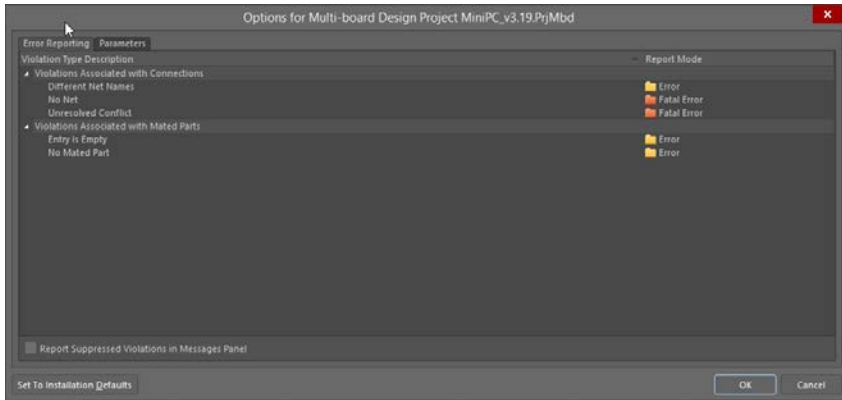
**Note:** Use Crossprobe for quick navigation to child projects (Schematic or PCB)

# Swap Net Labels





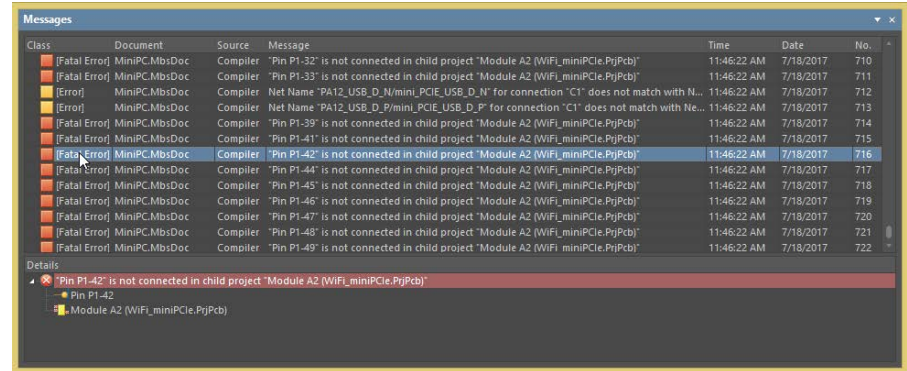
## Review violations level for ERC: Project >> Project Options >> Error Reporting



### Violation Description

- Different Net Names - Net names in Multi-Board Schematic and in child project does not match
- No Net - One of the pins in Multiboard Connection does not have Net in Child project
- Unresolved conflict - Changes in one child project affect other connected project (user defined connectivity was wrong)
- Entry is Empty - Module or Harness Entry do not have any assigned parts
- No Mated Part - Part in module or harness do not have any assigned pair

## Run verification process: Design >> Run ERC



### Violations are listed in the Messages panel

**Restriction:** There are currently five violations checks in AD18, but more electrical error checking will be introduced in future releases.





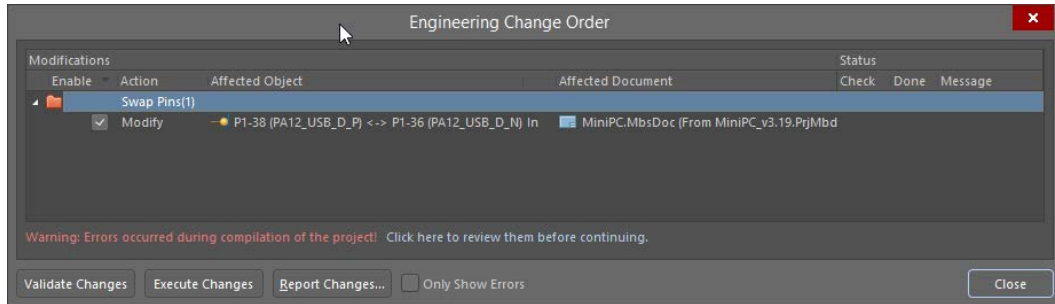
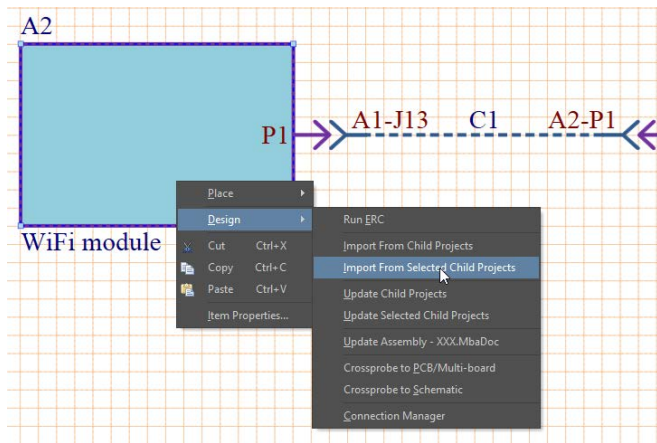
# Synchronizing Changes Child Projects



The Multi-Board ECO process is controlled in the Multi-Board Schematic Editor to push and pull changes to and from Child projects:



on the WiFi Module,  
Design >> Import From Selected Child Projects



Standard ECO dialog is using for data transfer control



Three steps are required for right workflow of changes applying between child projects and Multi-Board Schematic:

1. Import changes from child projects to Multi-Board Schematic
2. Conflict Resolution
3. Export changes to child projects (Update child projects)

For Conflict Resolution:

Step 1 - Open **Design >> Connection Manager**

Step 2 - Press button **Show Changes Only** to filter by conflicts

Step 3 - Select cell with exclamation mark and choose one of the options in bottom section:

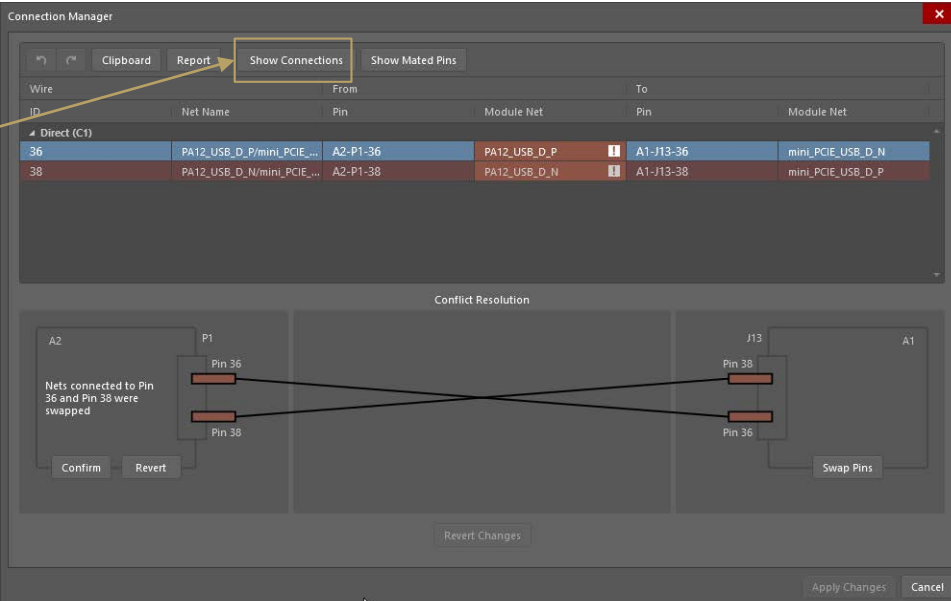
**Confirm** - Approves swapping without any changes

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

**Swap Pins** - Replicates changes in mated part

Note: The available conflict resolutions displayed depend on connection type so some options will not display.

Note: A conflict occurs when two pins or nets are swapped in a child project and the change breaks the user-created connectivity.



Connection Manager

Clipboard Report **Show Connections** Show Mated Pins

Wire ID	Net Name	From Pin	Module Net	To Pin	Module Net
36	PA12_USB_D_P/mini_PCIE...	A2-P1-36	PA12_USB_D_P	A1-J13-36	mini_PCIE_USB_D_N
38	PA12_USB_D_N/mini_PCIE...	A2-P1-38	PA12_USB_D_N	A1-J13-38	mini_PCIE_USB_D_P

Conflict Resolution

A2: Nets connected to Pin 36 and Pin 38 were swapped

A1: J13

Buttons: Confirm, Revert, Swap Pins, Revert Changes, Apply Changes, Cancel





In Multi-Board Assembly Editor:

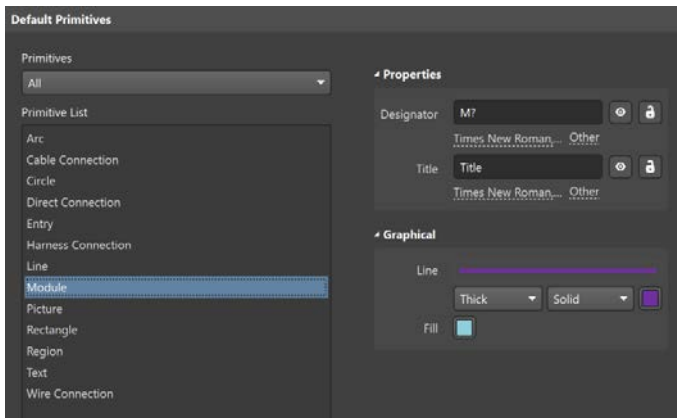
**Design >> Import Changes from ...**

The following content could be loaded from the Multi-Board Schematic:

1. List of modules (PCBs or Multiboards)
2. List of connections (each harness, cable, wire, etc)
3. List of physical connections (single pin to pin connections)

Configure import options:

**Preferences >> Multi-board Schematic >> Defaults**



The following details can be configured for import from the PCB into the Multi-Board Assembly:

- Import Copper
- Import Overlay Layers
- Import Polygons
- Import Free 3D Bodies
- Restrict by Minimum Component Height

Importing all details requires a lot of resources and can slow performance down.

Configure import options:

**Preferences >> Multi-board Assembly >> Defaults**



**PCB Import Preferences**



- 1 **Design Methodology**
- 2 **Schematic Workflow**
- 3 **Assembly Workflow**
- 4 **Group Discussion**

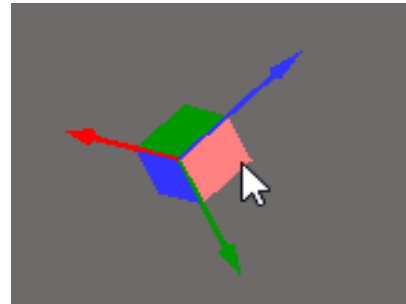


Standard AD shortcuts can be used for zooming and panning:

<b>CTRL + Scroll</b>	= Zoom In/Zoom Out
<b>Right Mouse + Drag</b>	= Panning
<b>SHIFT + Scroll</b>	= Left/Right Panning
<b>Scroll</b>	= Up/Down Scrolling
<b>CTRL + PgDown</b>	= Fit All Objects

Gizmo and short keys can be used for alignment view with standard plane (X, Y, Z):

<b>Red Gizmo Square</b>	- X Plane (Short Keys <b>X</b> And <b>Shift + X</b> to flip)
<b>Green Gizmo Square</b>	- Y Plane (Short Keys <b>Y</b> And <b>Shift + Y</b> to flip)
<b>Blue Gizmo Square</b>	- Z Plane (Short Keys <b>Z</b> And <b>Shift + Z</b> to flip)





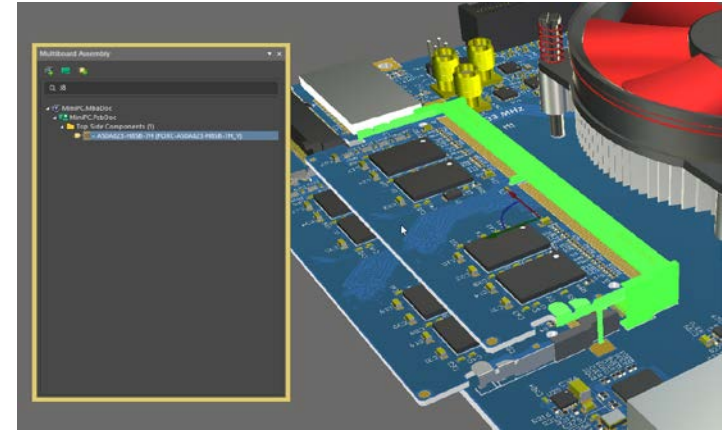
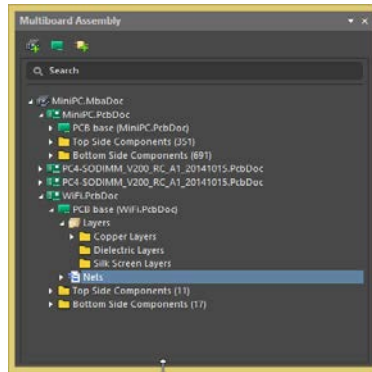
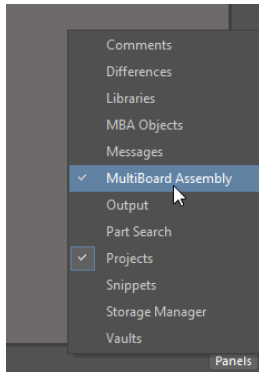
# Navigation in Assembly Hierarchy



Multiboard Assembly Panel could be opened with quick button Panels or from menu **View >> Panels >> Multiboard Assembly**

All boards, board layers, components, net classes, and other design aspects are found in this panel.

Use the Search Bar to search by designator to filter for desired design aspects.



Finding Connector With Component Designator with the Search Bar

Following key actions are accessible through Multiboard Assembly Panel:

1. Add new item in Assembly (assembly, board, body)
2. Show/Hide any item (board, component, body)
3. Show/Hide layers in PCB
4. Highlight nets and net classes in PCB

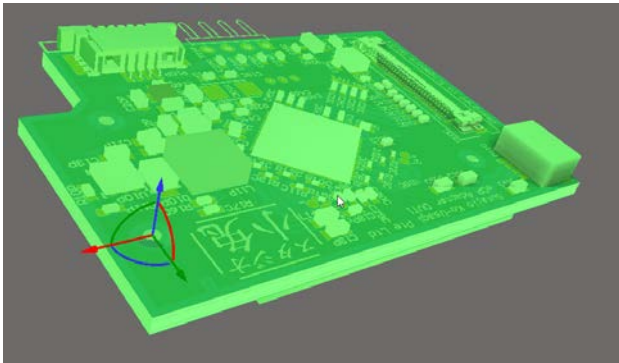


# Detail Placement and Alignment



There are two ways for component placement in the Multi-Board Assembly Editor:

## 1. Manual placement

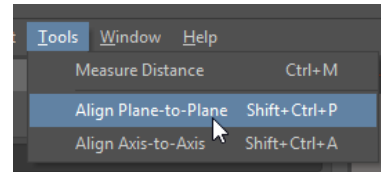


Use the gizmo for manual placement:

Drag Gizmo's arrow for move selected item along arrow axis

Drag Gizmo's arc for rotate selected item around same color axis

## 2. Alignment



### Tools >> Align Plane-to-Plane

Step 1 - Select first surface (based surface - will not moved)

Step 2 - Select second surface (will align with first one)

Step 3 - Press TAB for switch alignment direction

Step 4 - Press ESC for exit from alignment mode

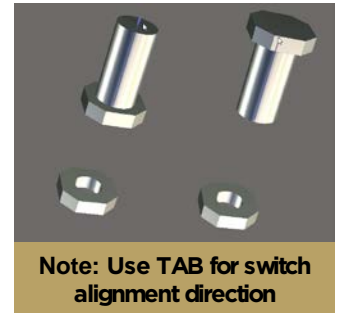
### Tools >> Align Axis-to-Axis

Step 1 - Select first axis (based axis - will not moved)

Step 2 - Select second axis (will align with first one)

Step 3 - Press TAB for switch alignment direction

Step 4 - Press ESC for exit from alignment mode





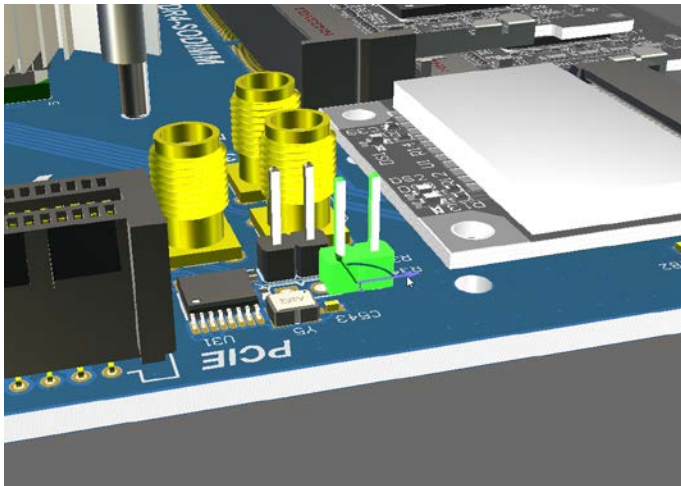
# Edit selected PCB in Assembly



Any PCB file can be edited in Multi-Board Assembly document (only component placement possible)

For editing of particular PCB, select the target PCB and enable 'Edit Part Mode':

Edit >> Edit Selected Part (CTRL+E)



Only active PCB will displayed with colors and other PCB are grayed out (read-only mode).

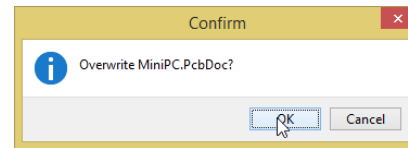
Exit 'Edit Part Mode' using the same short key or menu

Edit >> Finish Part Editing (CTRL+E)

Cancel last changes with the menu

Edit >> Cancel Part Editing

All changes made in 'Edit Part Mode' will transferred and save to the original PCB file after the confirmation dialog.



# Section View



The Section View allows you to use the X/Y/Z planes to visualize the interior of your multi-board assembly.

Open the View Configuration Panel:  
**View >> Panels >> View Configuration**

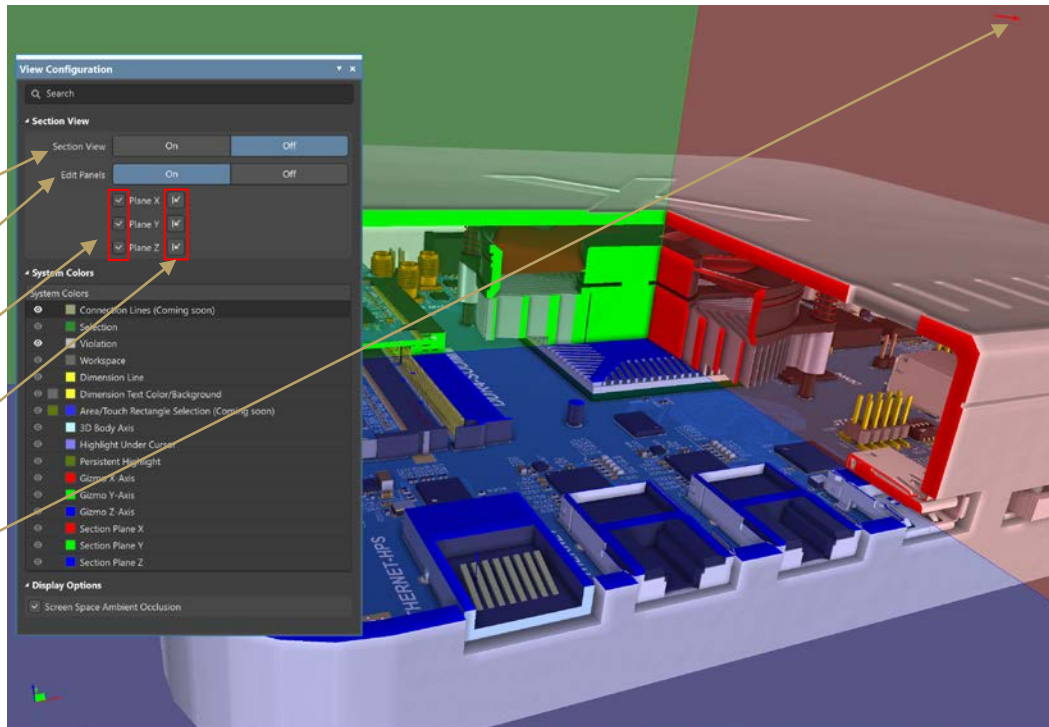
**Step 1 - Enable Section View:** Use buttons in panel or **CTRL+SHIFT+V**

**Step 2 - Enable/Disable edit mode**

**Step 3 - Enable/Disable Plane**

**Step 4 - Flip the side of hidden scene part**

**Step 5 - Use arrows in workspace to position Assembly plane sections**



**Note:** Turn off “Edit Panels” to enable Assembly editing and selection of individual design aspects.



# Measure Distances Between Bodies



**Step 1 - Start Measurement with:**

**Tools >> Measure Distance** or **CTRL+M**

**Step 2 - Select 3D body by left mouse click**

or

**Select Edge by left mouse click with CTRL**

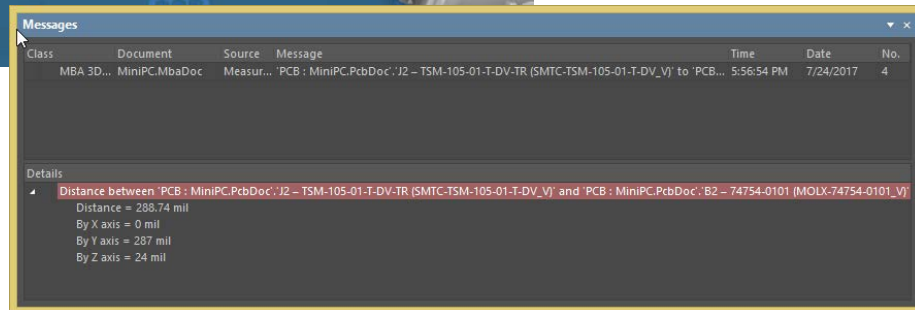
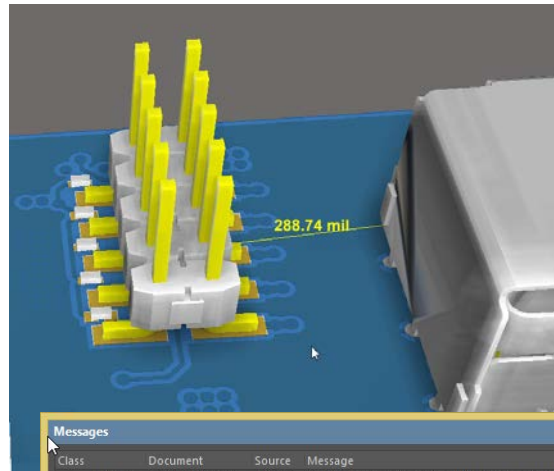
**Step 3 - See Measurements details in**

**Messages Panel**

**Step 4 - Stop measurement with ESC**

**(all results will clear)**

**Note:** Complicated 3D bodies can cause performance issues during measurements.







- 1 **Design Methodology**
- 2 **Schematic Workflow**
- 3 **Assembly Workflow**
- 4 **Group Discussion**



# AltiumLive 2018 Questions?

**David Haboud**

**David.Haboud@Altium.com**

**Product & Persona Marketing Engineer**





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

**Connection Manager:** This dialog lists all Net/Pin assignments, grouped under their parent Connection Designators and Connection Type (Wire, Direct, etc), and includes their system design ID and Net Name, along with their From and To Pin/Net connections.

**Connection Type:** One of four methods to connect Module Entries- Direct Connection, Wire, Cable, and Harness.

**Connections:** The connectivity between Child Project connectors, connector pins and Nets in the overall system design.

**Child Project:** A project associated with the high level system Multi-Board Schematic Document.

**Cross-Section View:** A view that you can toggle and move X/Y/Z plane sections to see internal assembly positioning.

**Direct Connection:** Direct contact between boards.

**Entry:** A logical representation on a module of a physical connector.

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

**Mated Part:** Two parts connected logically that will connect physically in the Multi-Board Assembly.

**Multi-Board Assembly (MBA):** The physical design with design models to create the full system level assembly.

**Multi-Board Assembly Document (MbaDoc):** – A document containing a Multi-Board Assembly.

**Multi-Board Design Project (PrjMbd):** Contains Multi-Board Schematic and Assembly documents and all child projects.

**Multi-Board Schematic (MBS):** The logical design with Modules and Entries to create the full system level Connections.

**Multi-Board Schematic document (MbsDoc):** – A document containing a Multi-Board Schematic.

**Module:** A logical representation in the Multi-Board Schematic document of a physical PCB used to define interconnections.

**Object Gizmo:** The red/green/blue (X/Y/Z) axis marker at the origin corner of an object.

**Split:** Logically divide, in terms of Pins/Nets, a Module Entry to create Connections to other modules.

**Workspace Gizmo:** The red/green/blue (X/Y/Z) axis marker at the bottom left of the Assembly editor workspace.

**Wire:** A single wire connecting two points across boards.