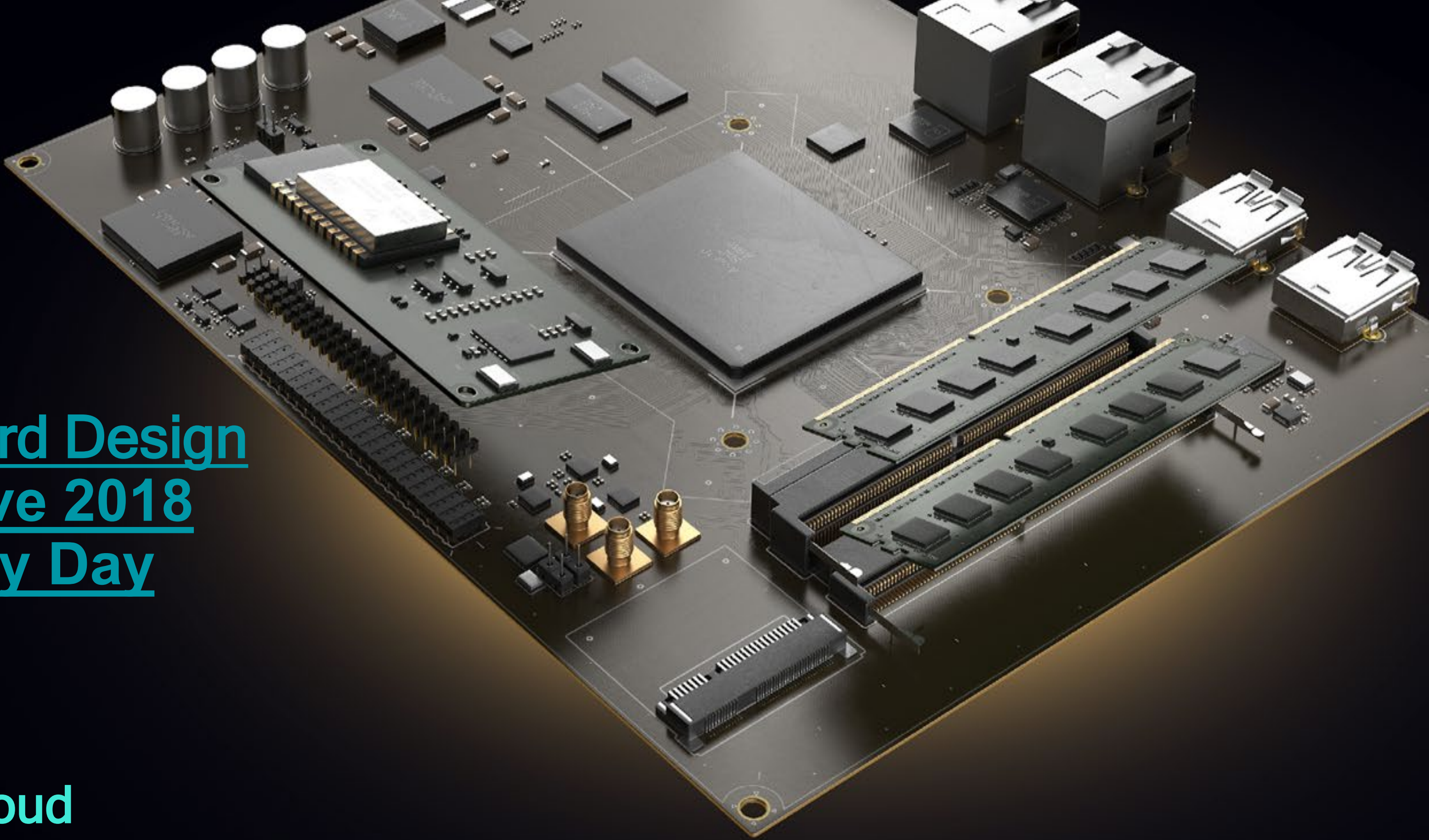
ALTIUM DESIGNER



Multi-Board Design
AltiumLive 2018
University Day

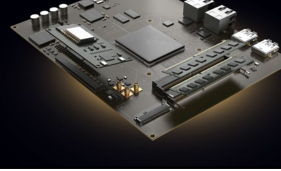
David Haboud

Product Marketing Engineer

Altium

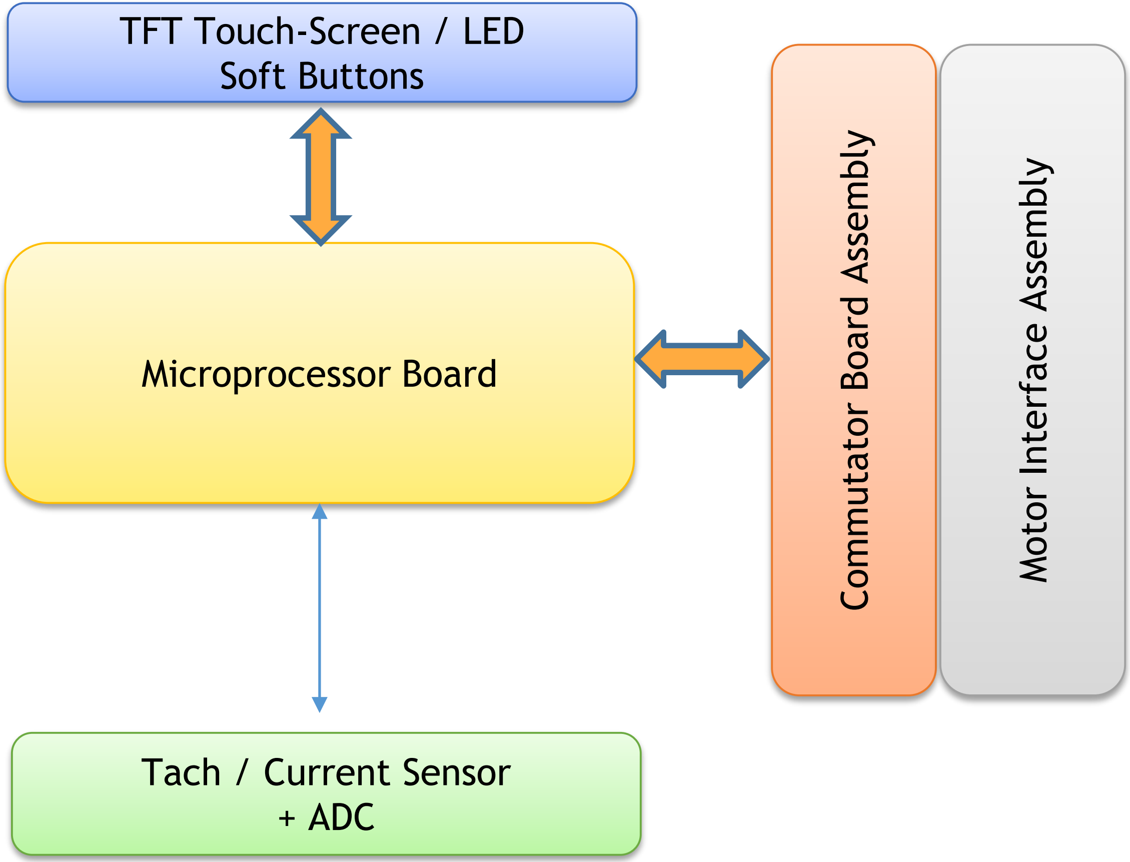
ALTIUM DESIGNER

Agenda

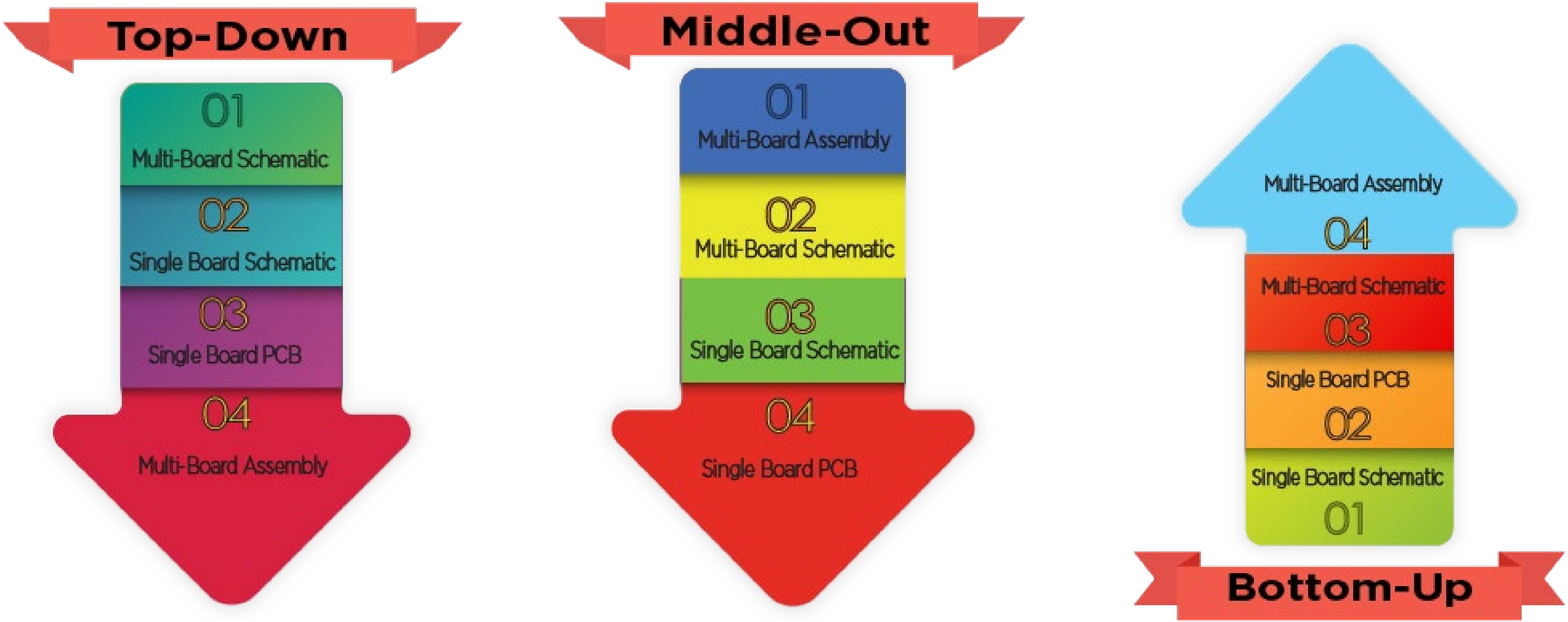


1. Design Methodology
2. Schematic Workflow
3. Assembly Workflow
4. Group Discussion

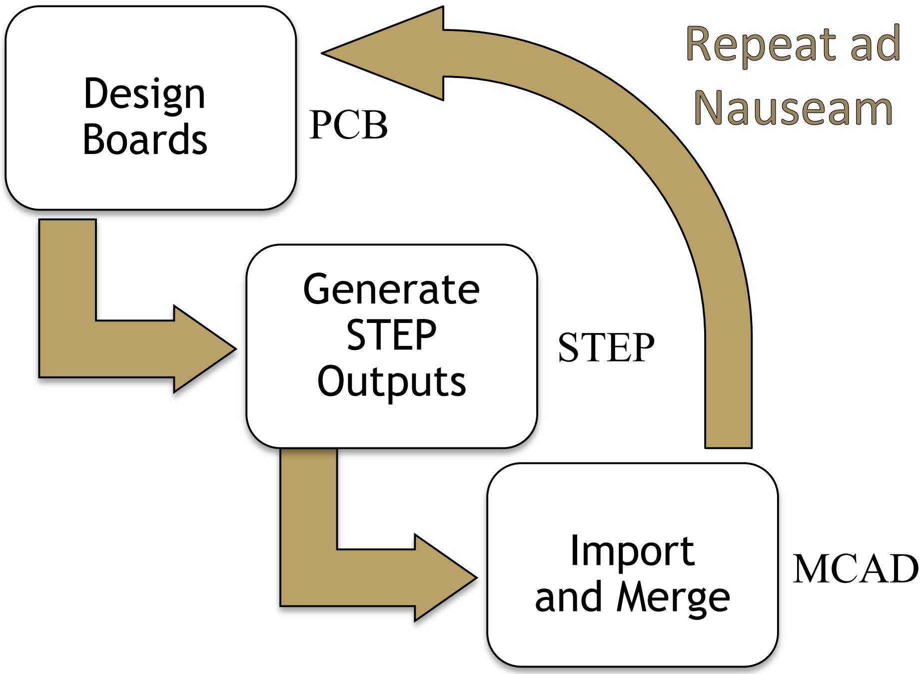
Design Methodology



Design Methodology

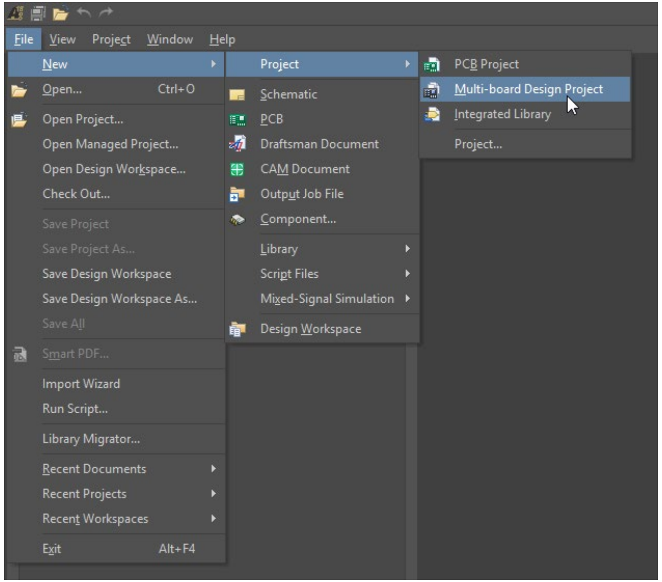


Design Methodology



Create New Project and Source Files

Create a new multi-board project:
File >> New >> Project >>
Multi-Board Design Project (*.PrjMbd)

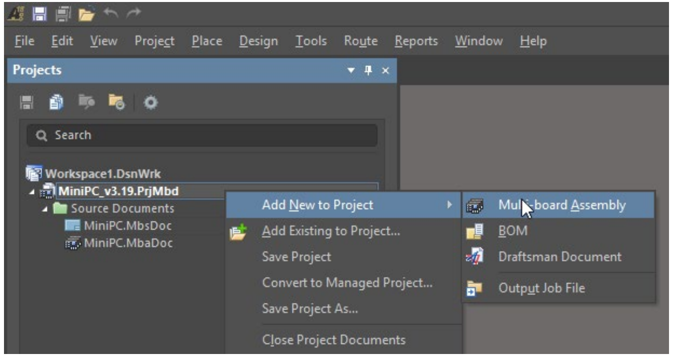


Source files:

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

For adding source files into a project, 

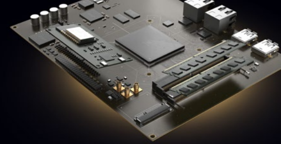
on the project name and then Add New to Project... >> File Type.



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

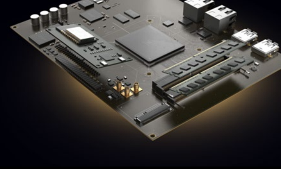
ALTIUM DESIGNER

Agenda



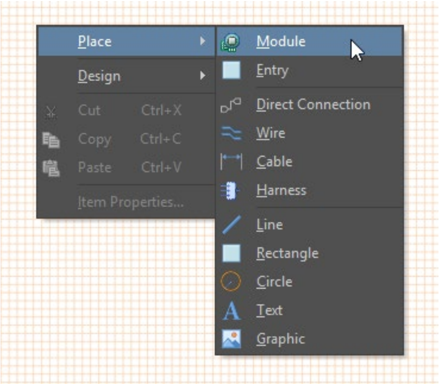
1. Design Methodology
2. Schematic Workflow
3. Assembly Workflow
4. Group Discussion

Place Module and Link to PCB Project

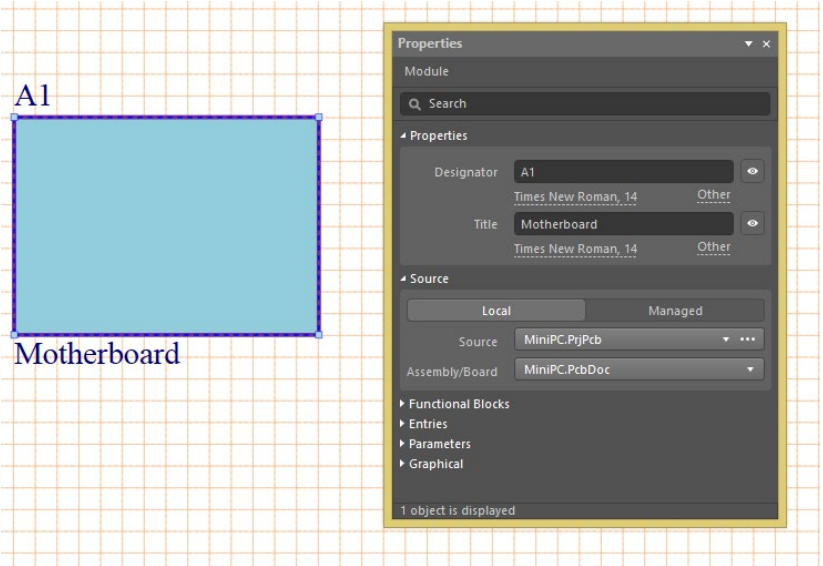


Place >> Module [P >> M]
Or

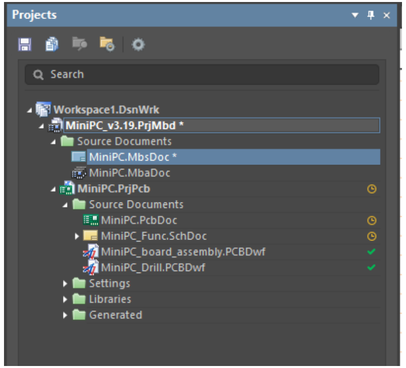
>> Place >> Module



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

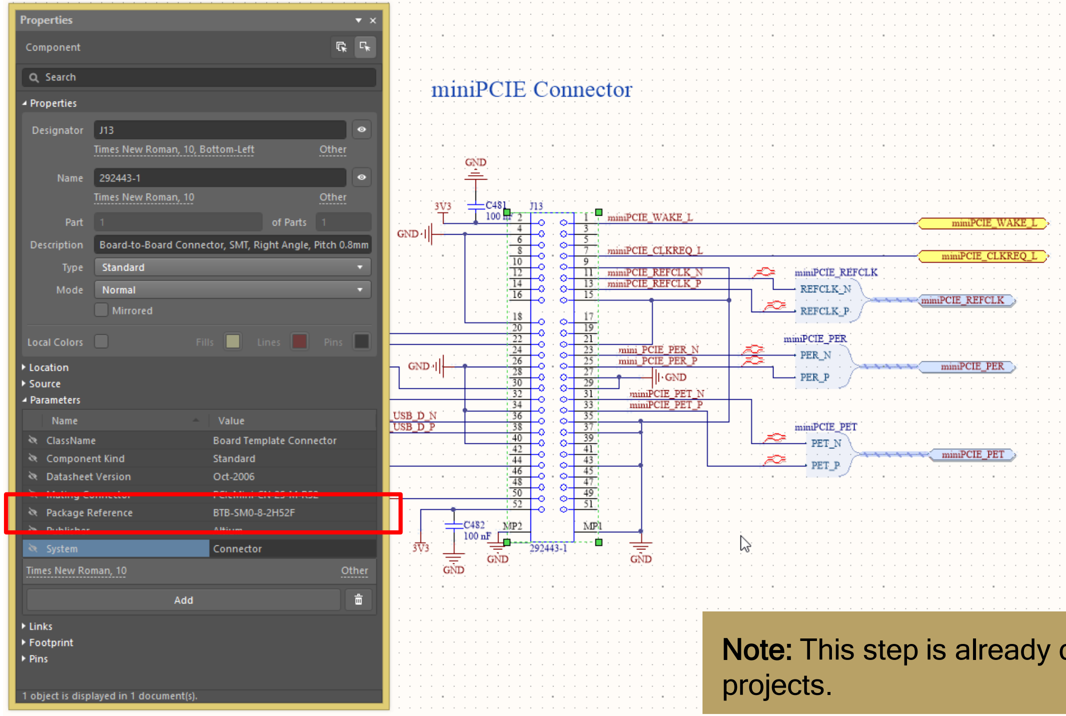


Child projects are displayed in the Multi-Board project tree



Prepare Child Project for Use in PrjMbd

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



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

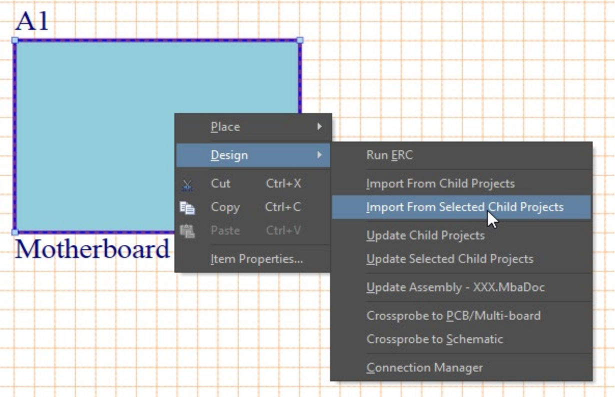
Import Module Entries

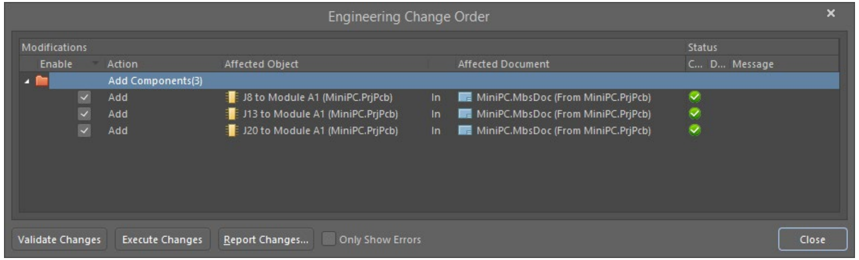
right click on Module

Select Design >> Import From Selected Child Project

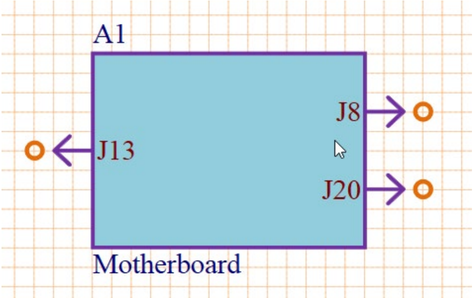
To update all modules,

Select Design >> Import From Child Projects

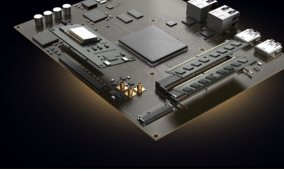




Standard ECO dialog is using for data transfer control



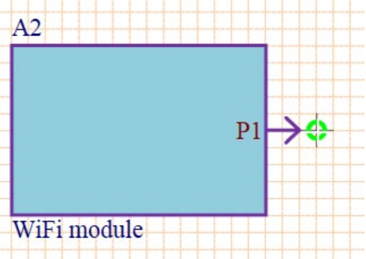
Note: Child project compilation executes on import. Using “Import From Selected Child Projects” is only recommended for quick data transfer for selected module only after small changes.

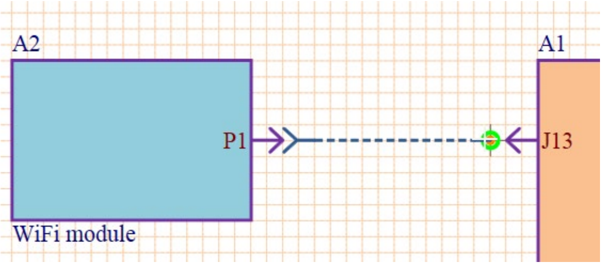


Add Connection Lines

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

right click >> Place >> Direct Connection



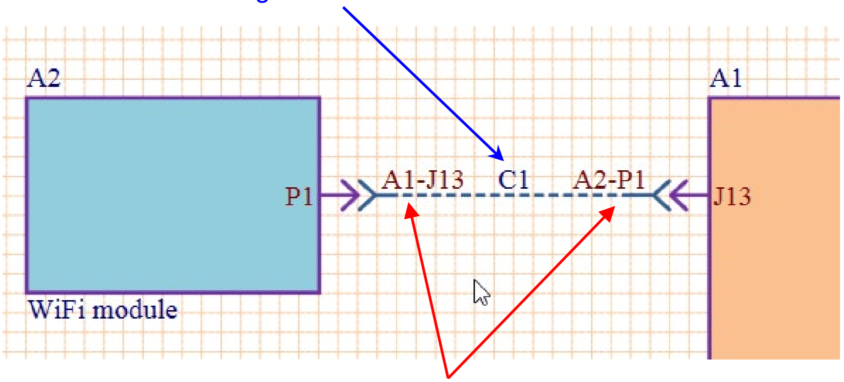


Green marker displays at valid locations to start and finish connections

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

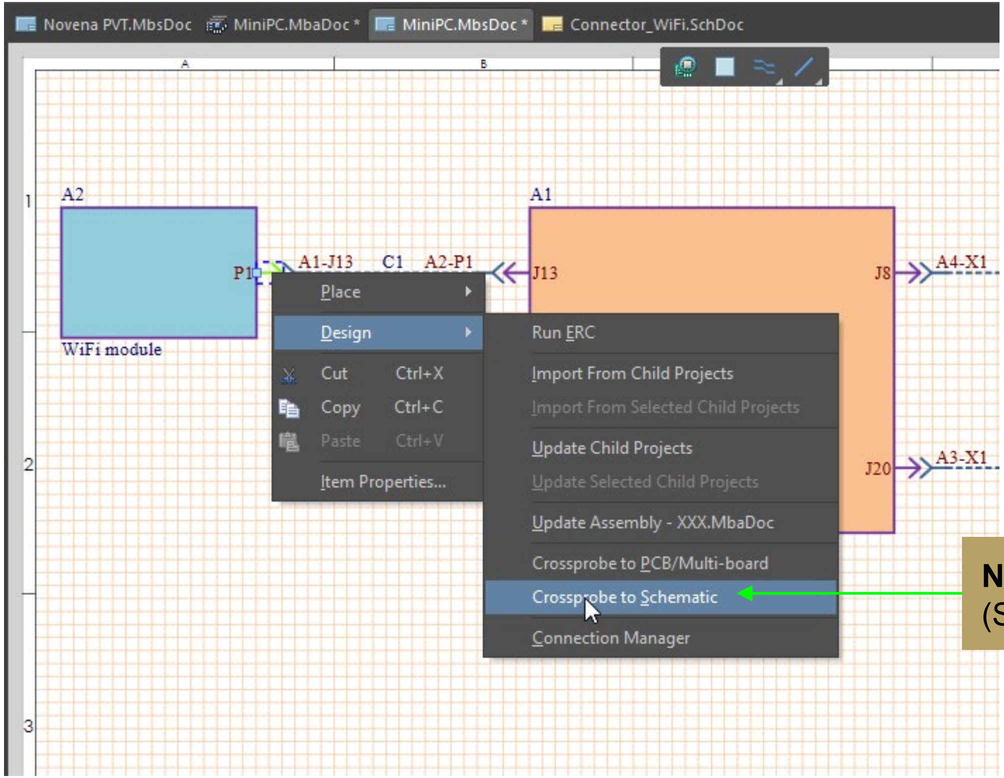
Following data is displayed for Direct Connection by default:

C1 - Connection Designator



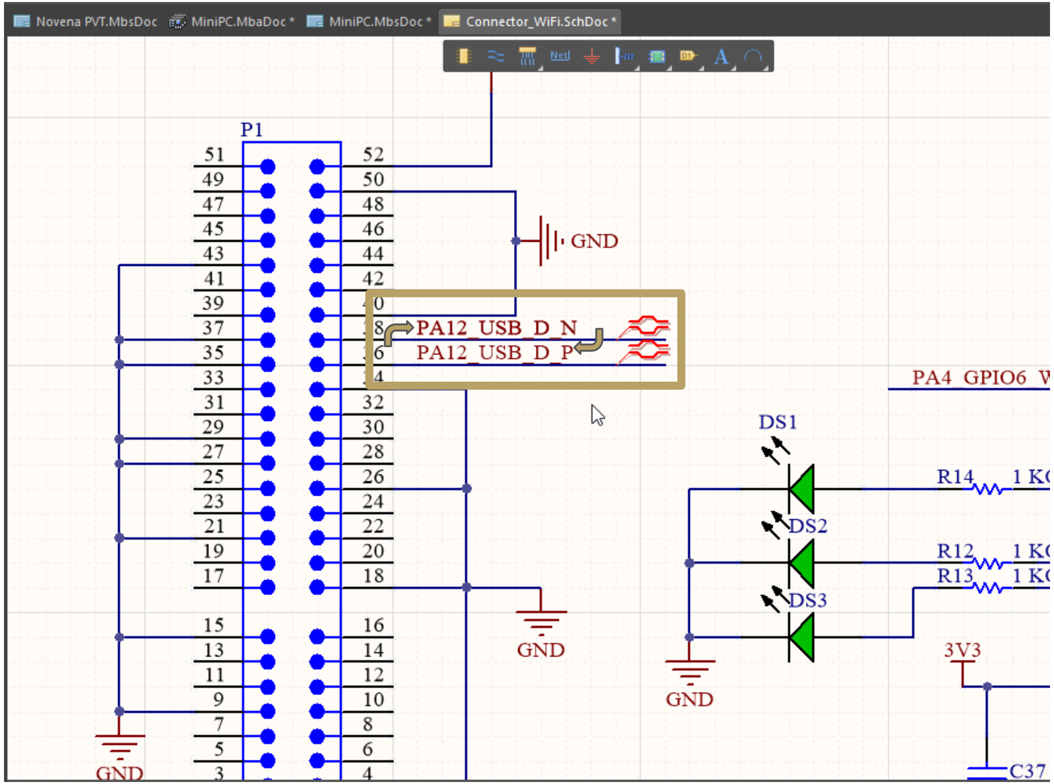
A1-J13 and A2-P1 - Mated Part Designator

Cross-Probe to Child Project Schematic



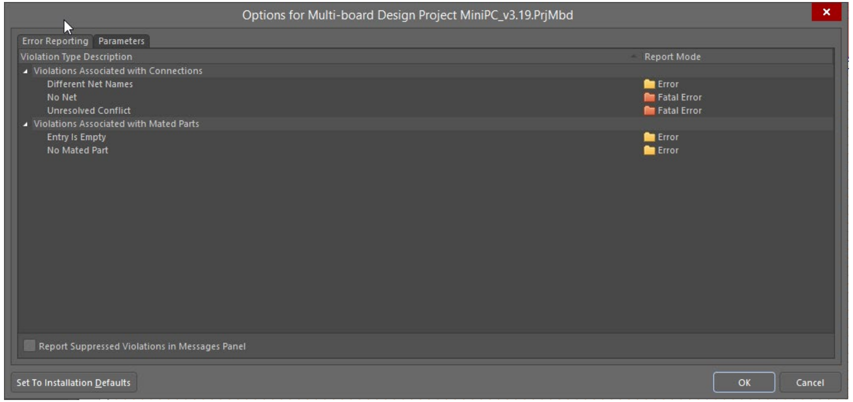
Note: Use Crossprobe for quick navigation to child projects (Schematic or PCB)

Swap Net Labels



Electrical Rules Check (ERC)

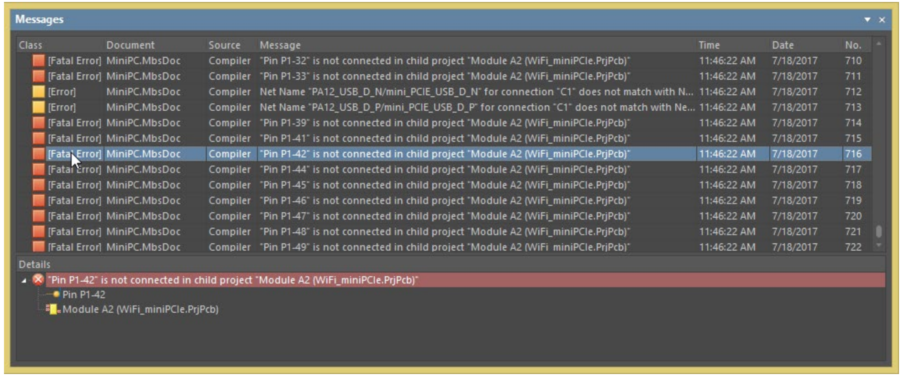
Review violations level for ERC:
Project >> Project Options >> Error Reporting



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



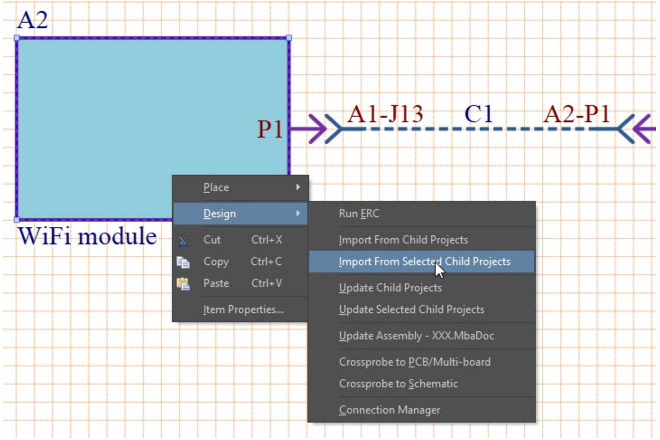
Violations are listed in the Messages panel

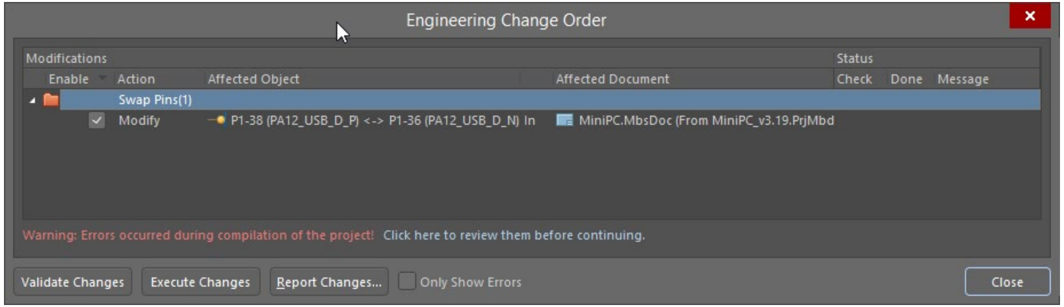
Restriction: There are currently five violations checks in AD18, but more electrical error checking will be introduced in future releases.

Synchronizing Changes Child Projects

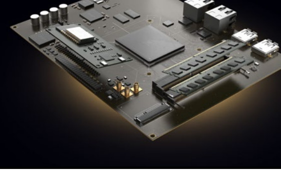
The Multi-Board ECO process is controlled in the Multi-Board Schematic Editor to push and pull changes to and from Child projects:

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





Standard ECO dialog is using for data transfer control



Conflict Resolution

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)

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

For Conflict Resolution:

Step 1 - Open Design >> Connection Manager

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

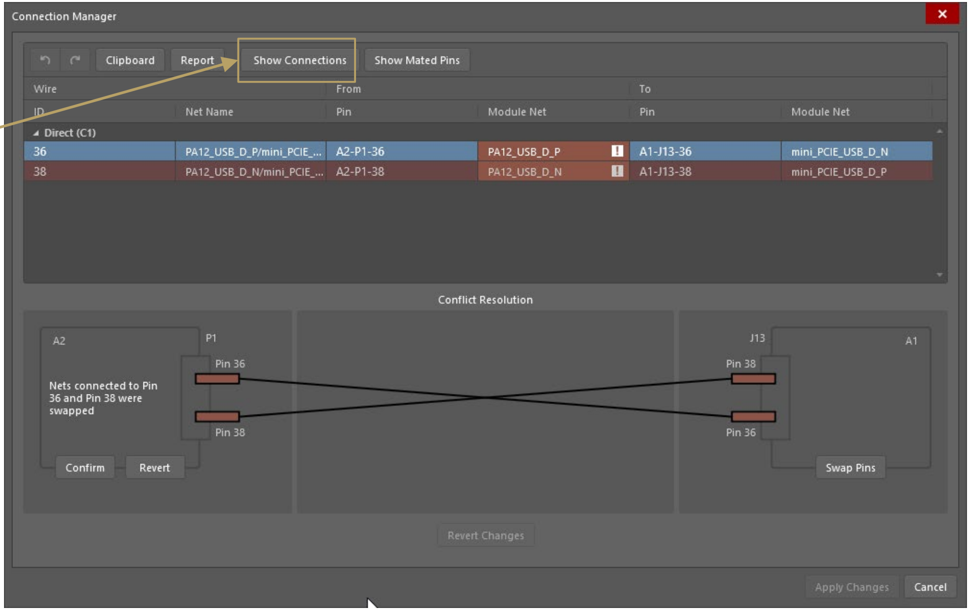
Step 3 - Select cell with exclamation mark and choose one of the options in bottom section:

Confirm - Approves swapping without any changes

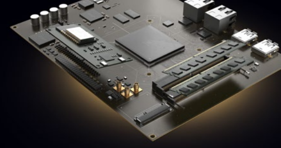
Revert - Cancels changes in first child project and requires back ECO to complete changes

Swap Pins - Replicates changes in mated part.

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



Load Multi-Board Assembly Editor Data

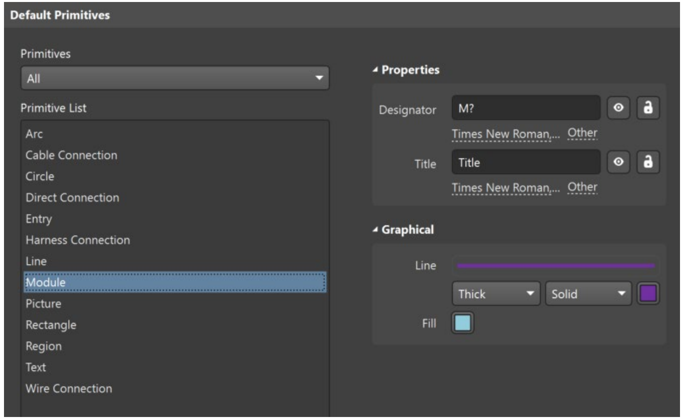


In Multi-Board Assembly Editor:
Design >> Import Changes from ...

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

ALTIUM
DESIGNER

Agenda

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

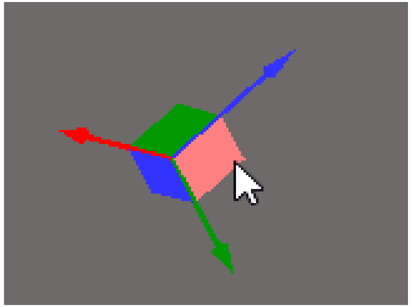
Zoom/Pan Control

Standard AD shortkeys can be used for zooming and panning:

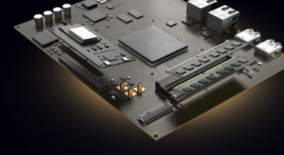
CTRL + Scroll	= Zoom In/Zoom Out
Right Mouse + Drag	= Panning
SHIFT + Scroll	= Left/Right Panning
Scroll	= Up/Down Scrolling
CTRL + PgDown	= Fit All Objects

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

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



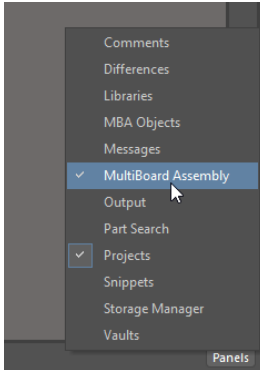
Navigation in Assembly Hierarchy

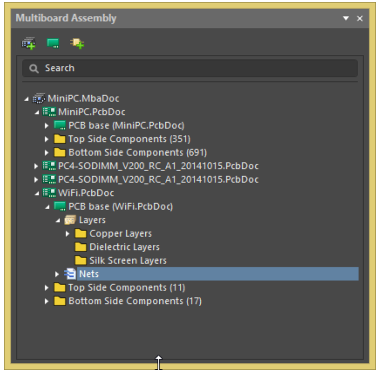


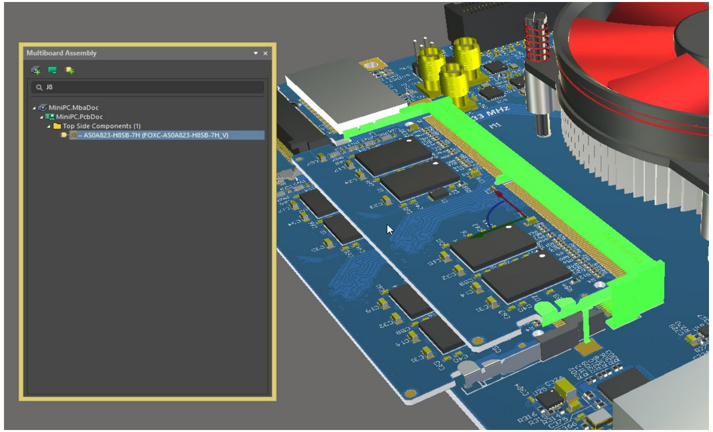
Multiboard Assembly Panel could be opened with quick button **Panels** or from menu
View >> Panels >> Multiboard Assembly

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**

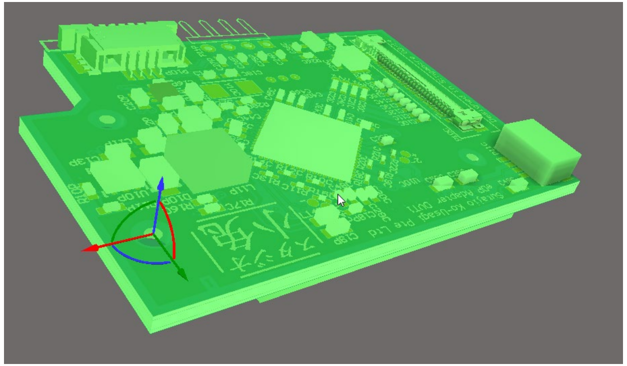
Following key actions are accessible through **Multiboard Assembly Panel**:

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:

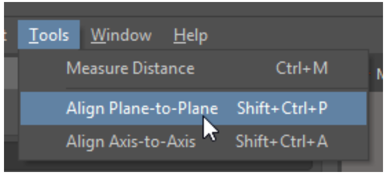
1. Manual placement



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

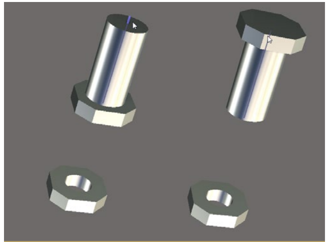


Tools >> Align Plane-to-Plane

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



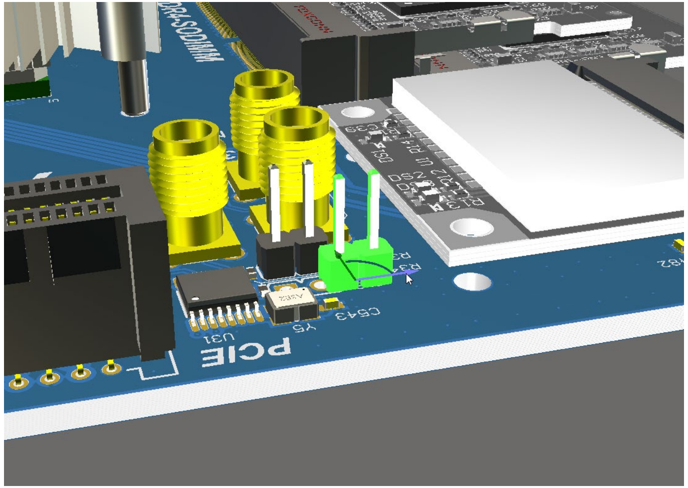
Note: Use TAB for switch alignment direction

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).

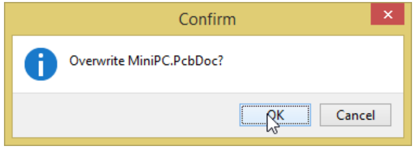
Exit “Edit Part Mode” using the same short key or menu

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.



ALTIUM DESIGNER

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

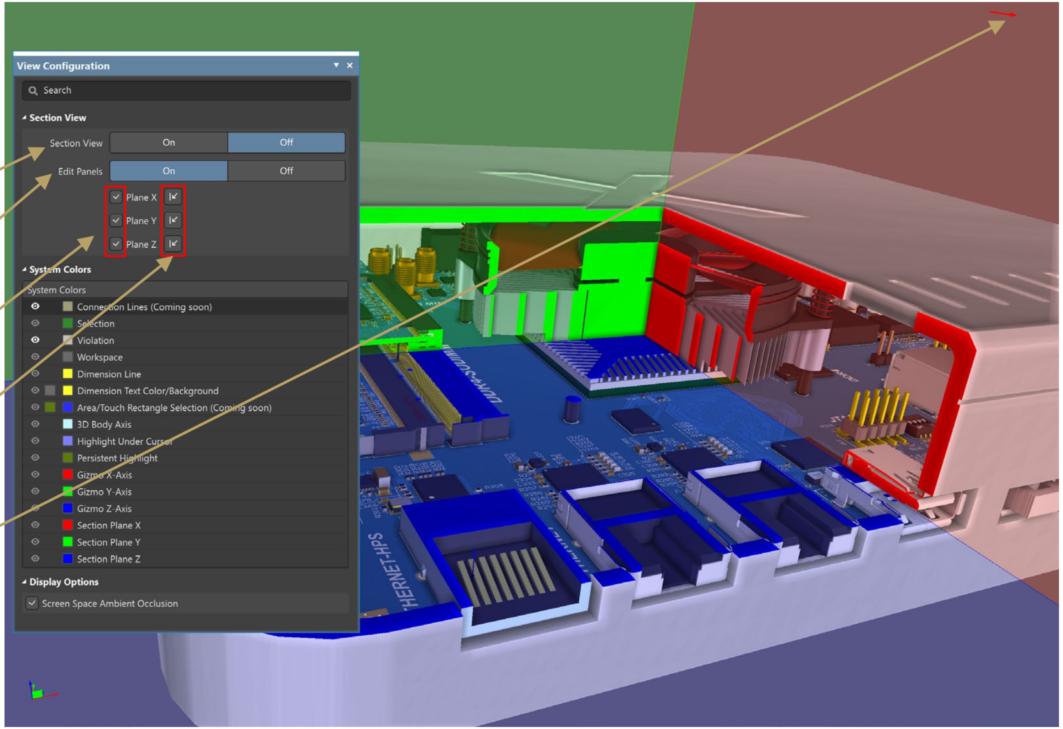
Step 1 - Enable Section View: Use buttons in panel or CTRL+SHIFT+V

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

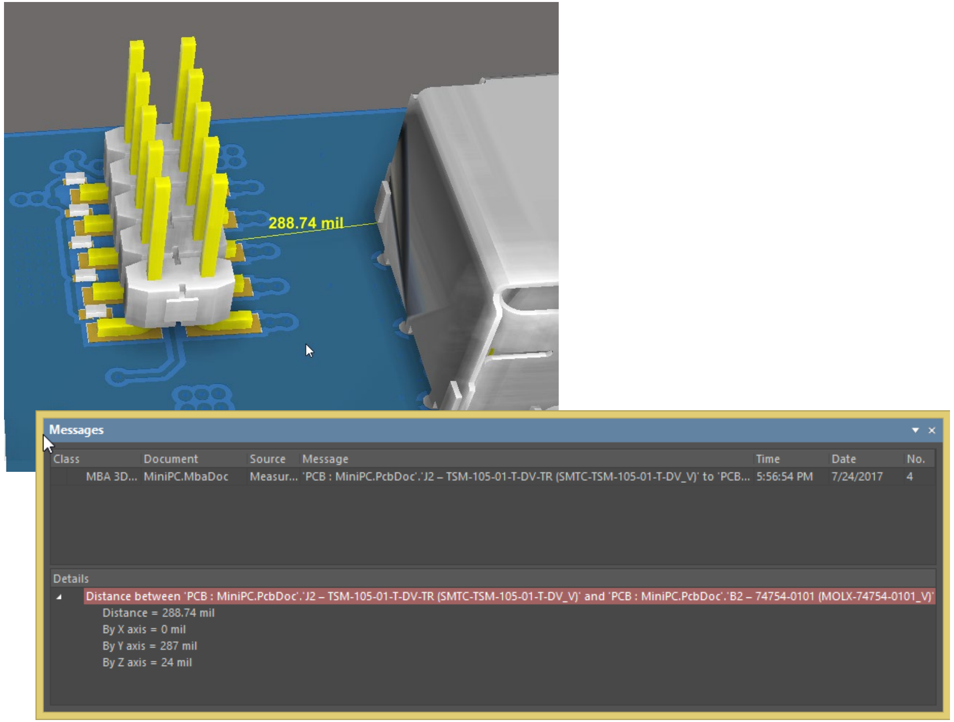
Step 1 - Start Measurement with:
Tools >> Measure Distance or CTRL+M

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

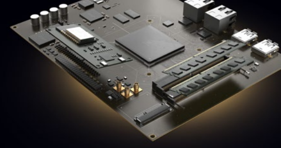
Step 4 - Stop measurement with ESC
(all results will clear)

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



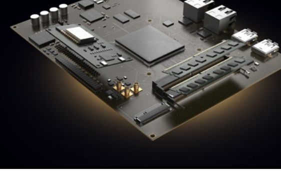
ALTIUM DESIGNER

Agenda



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

Group Discussion



AltiumLive 2018
Questions?

David Haboud

David.Haboud@Altium.com

Product & Persona Marketing Engineer

Glossary

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.