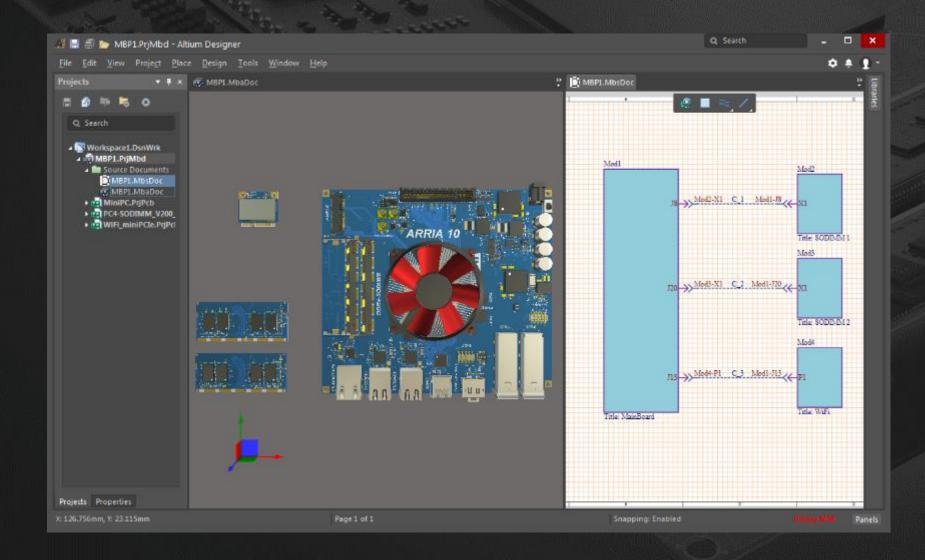


Altium. AGENDA

- Multi-PCB Design
- General
- Setup
- System Design
- Assembly
- Data Management
- Documentation
- A real case



Altium. Multi-PCB Design is complex

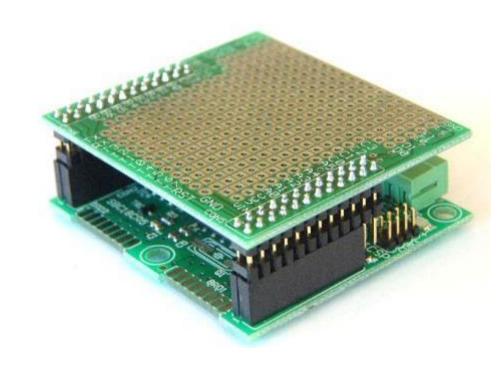
- PCB designs can be complex
- Many products include multiple, interconnected PCB's
- Multi-PCB Design can be even more complex
- Multi-PCB Design can be tedious and error-prone
- A mistake at this late stage is costly:
 - Cost of redesign
 - Delay to market



Altium. Multi-PCB Design is challenging

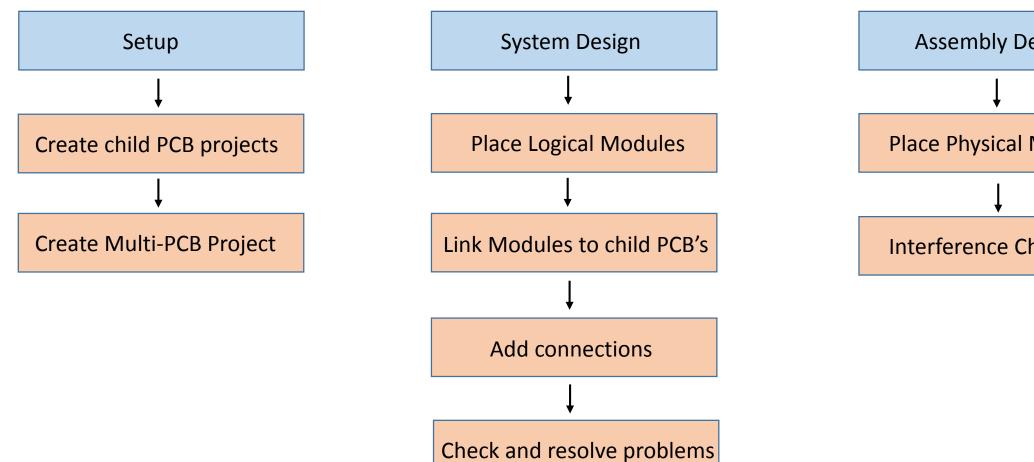
- Design challenges:
 - Design iterations between the various PCB's
 - Electrical connections
 - Mechanical connections
 - Connector Mating (pinout errors, connector mating)
 - Component Clearance
 - Individual PCB designs can be created on different locations
 - Change management and synchronization
 - Design Reuse
- Multi-PCB design is supported in Altium Designer 18

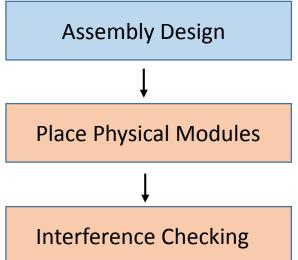


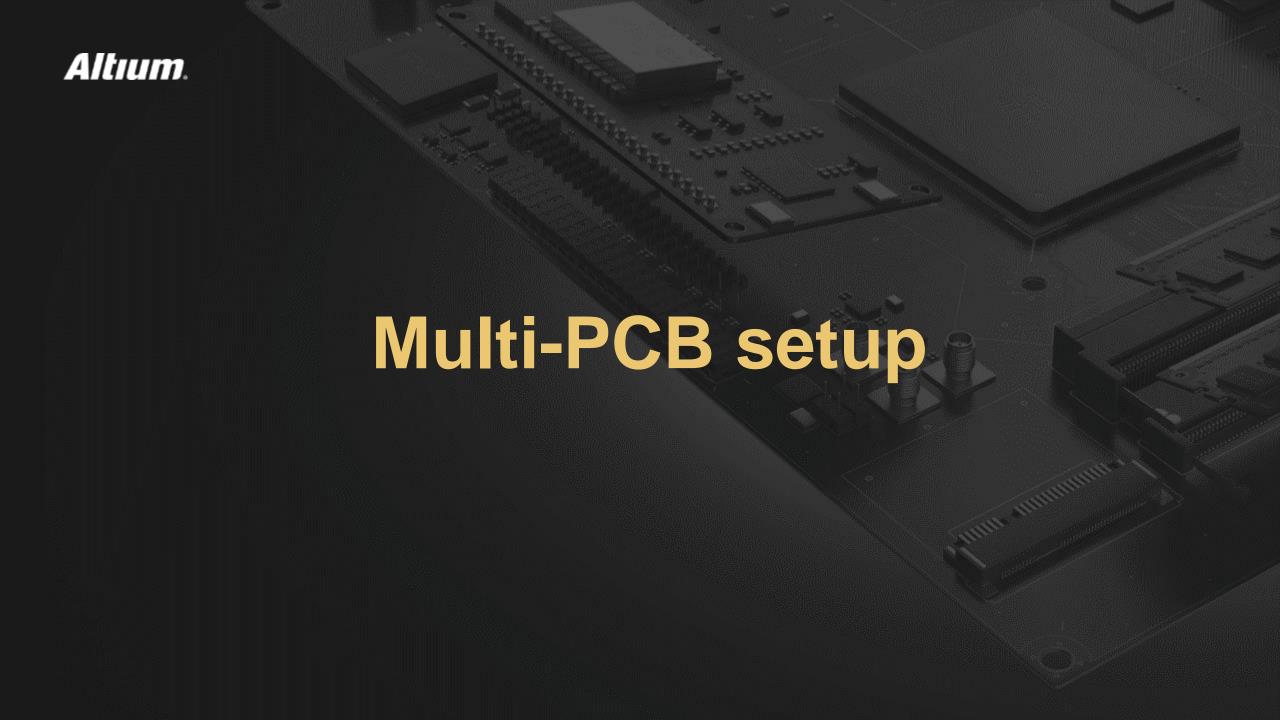


Altıum.

Multi-PCB Design – A Possible Work Flow







Altium. Setup – Create child PCB projects

Setup Create child PCB projects

- Design child PCB's (or Modules)
 - Motherboard
 - SODIMM
 - WiFi Module
- Create child PCB outlines
 - Needed for early Multi-PCB
- Place board-to-board connectors
 - Needed for early Multi-PCB
 - Use parameter: System = Connector

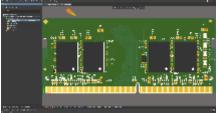
Mini-PC Mother board



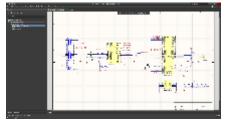


SODIMM board





WiFi Module board

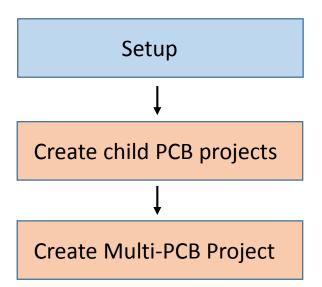




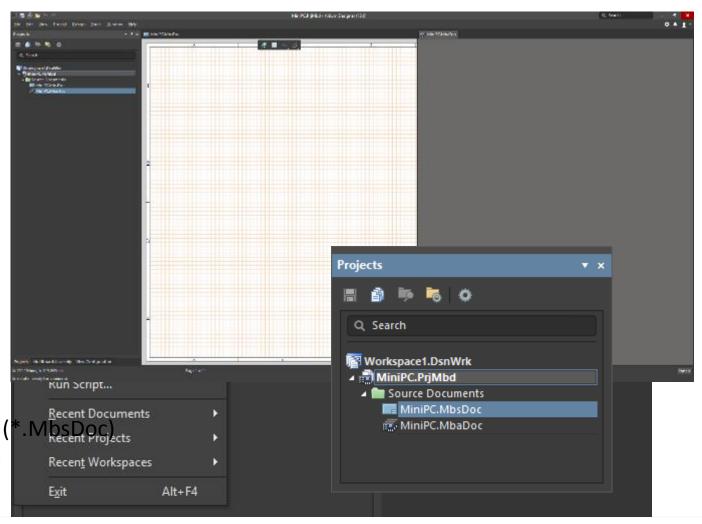
Altıum.

Setup – Create a new Multi-PCB

Project

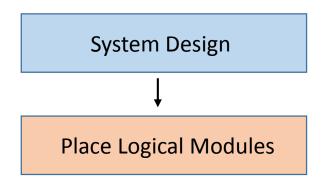


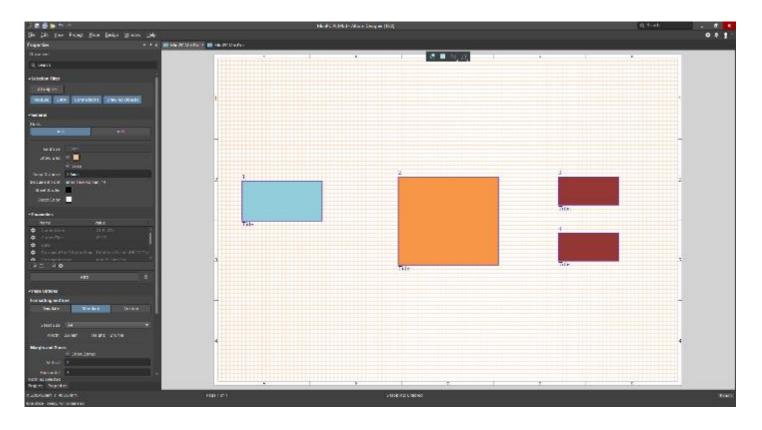
- Add Multi-PCB documents to the project:
 - Multi-Board Project document (*.PrjMbd)
 - Schematic-based logical design document (*.MbsDoc)
 - Physical PCB-based document (*.MbaDoc)
 - Project structure



Altıum. Multi-PCB system design

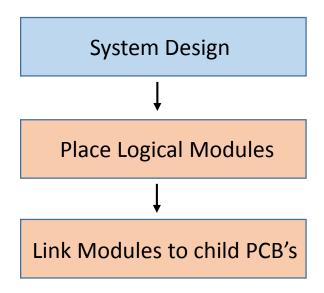
Altium. System Design – Place Logical Modules

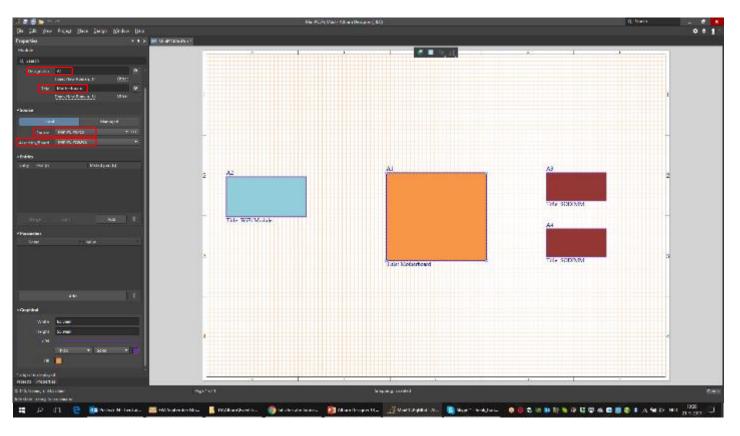




- Logical Modules refer to child projects
- Extra hierarchy level
- Must be configured
- Must be synchronized

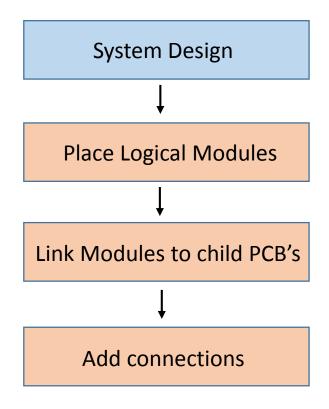
Altium. System Design – Link Modules to child PCB's

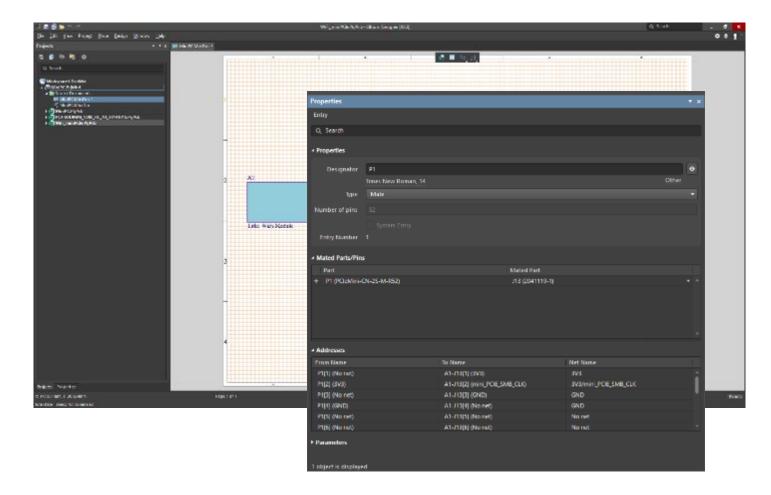




- Link Modules to child projects, using the Properties panel
 - Define Module Designator
 - Define Module Name
 - Define child project (Source)
 - Define child PCB (Assembly/Board)

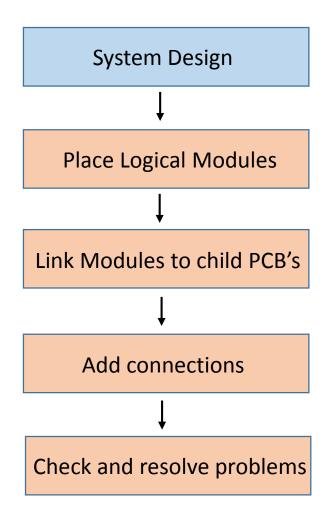
Altium. Multi-PCB Design – Add connections

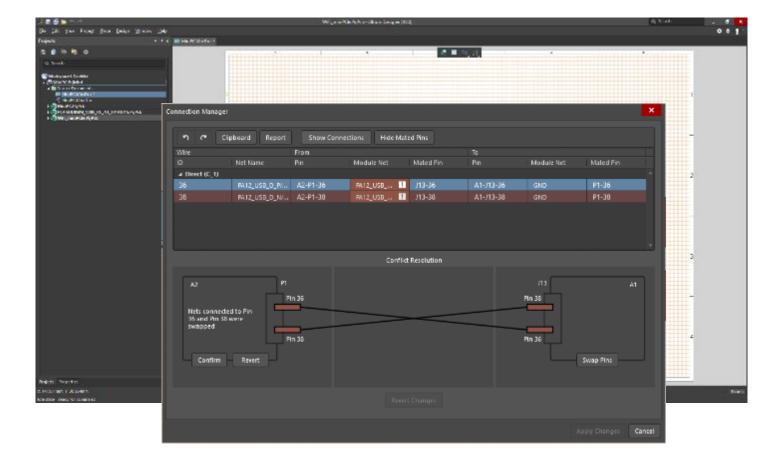




- Add Connections between Modules:
 - Import from Child Projects
 - Module connectors will be added automatically to the Modules
 - Reposition the connector symbols and add connections between them
 - Connections can be configured via the Properties panel

Altium. Multi-PCB Design – Check and resolve problems

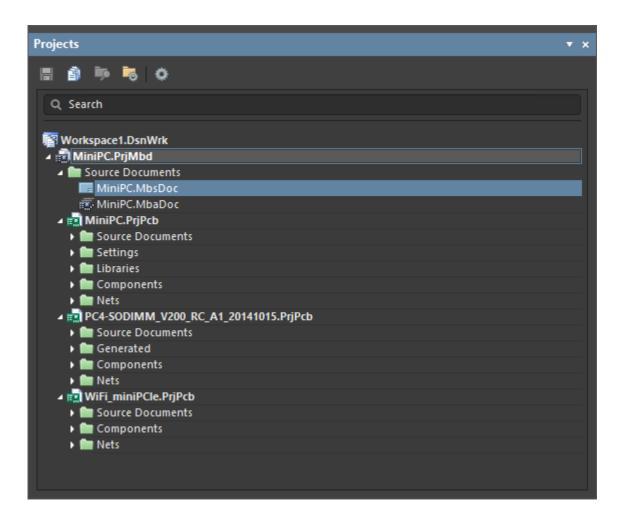




- Use the Connection Manager for:
 - Overview of the overall connectivity
 - Checking and resolving conflicts

Altium. System Design – Project Structure

- After setup and system design, the project structure should look like this:
 - Multi-Board Project (*.PrjMbd)
 - Schematic Multi-Board (*.MbsDoc)
 - Physical Multi-Board (*.MbaDoc)
 - Child PCB projects



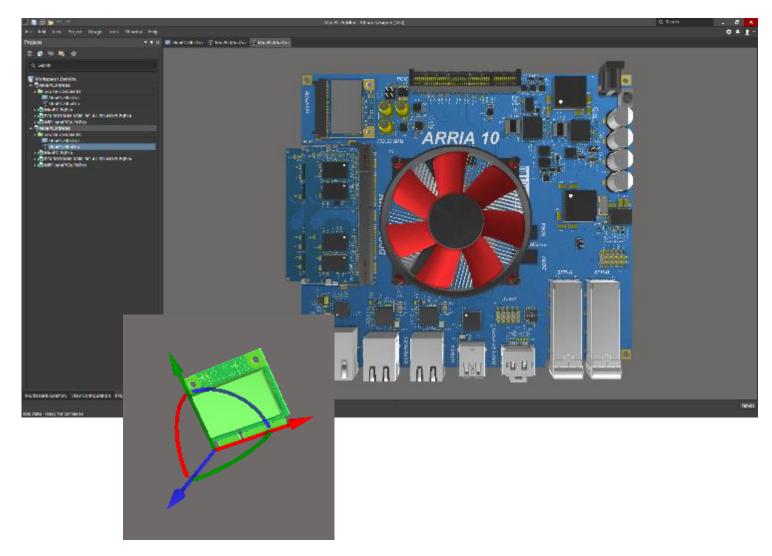
Altıum. Multi-PCB assembly

Altium. System Design – Place Physical Modules

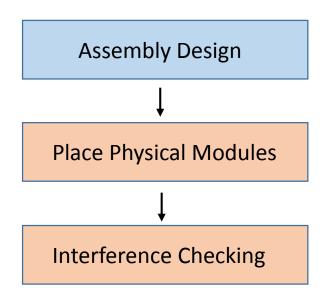
Assembly Design

Place Physical Modules

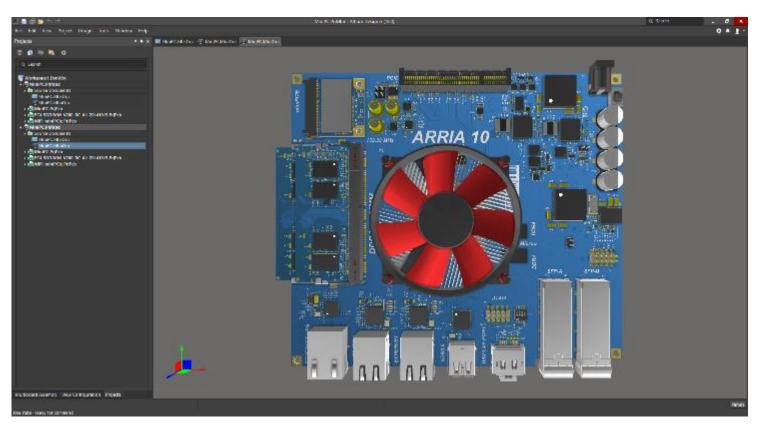
- Create the Multi-Board Assembly
 - Import the changes from the Logical System Design
- All child PCB's become visible
- Position the Physical Modules
 - Manual
 - GIZMO
 - Alignment functions
 - Plane-to-Plane
 - Axis-to-Axis



Altium. System Design – Interference checking

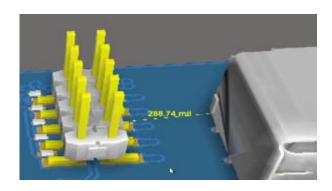


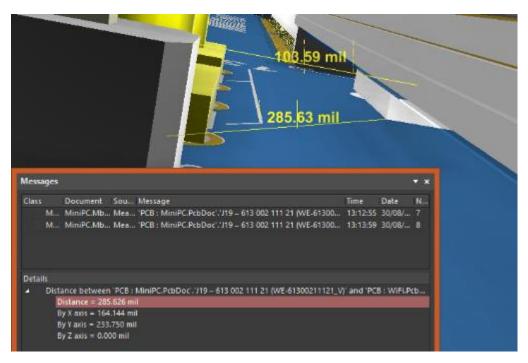
- Check collisions between:
 - Components on Modules
 - Components to other Modules
 - Modules to Modules
 - Components/Modules to Enclosure
- Resolve problems:
 - Reposition Modules
 - Reposition components on Module
 - Update child PCB Project



Altium. System Design – Measuring

Measuring

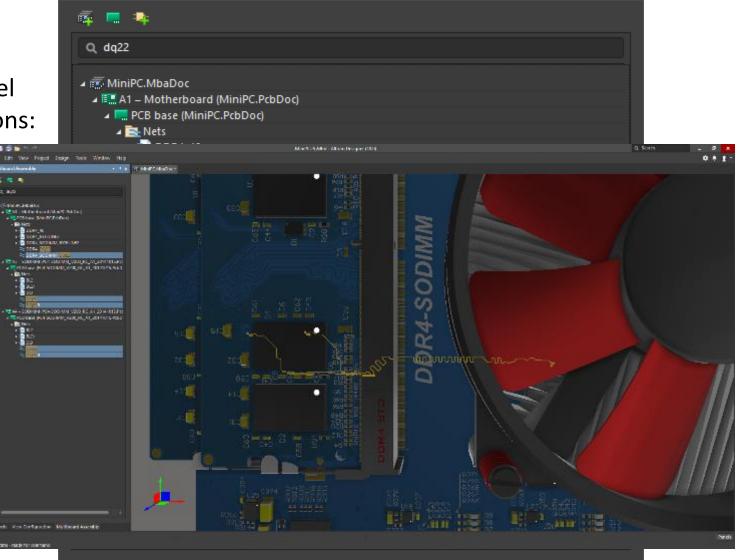




Altium. System Design – Multiboard Assembly panel

Multiboard Assembly

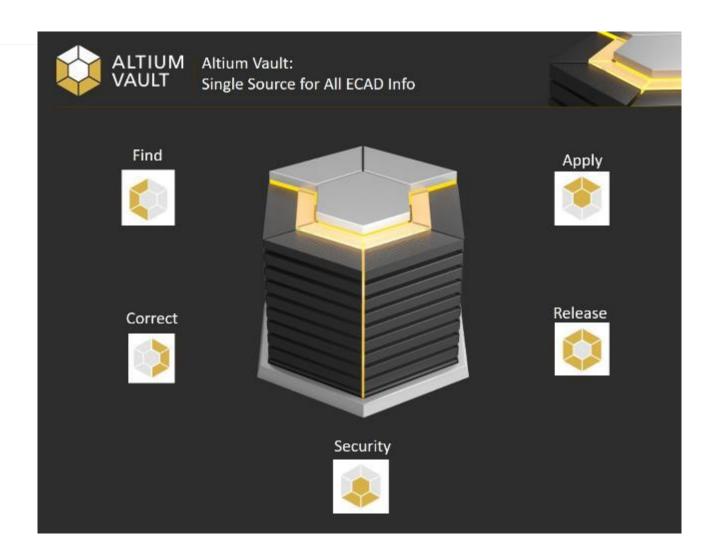
- Multiboard Assembly Panel
 - Miscellaneous functions:
 - Searching
 - Net highlighting

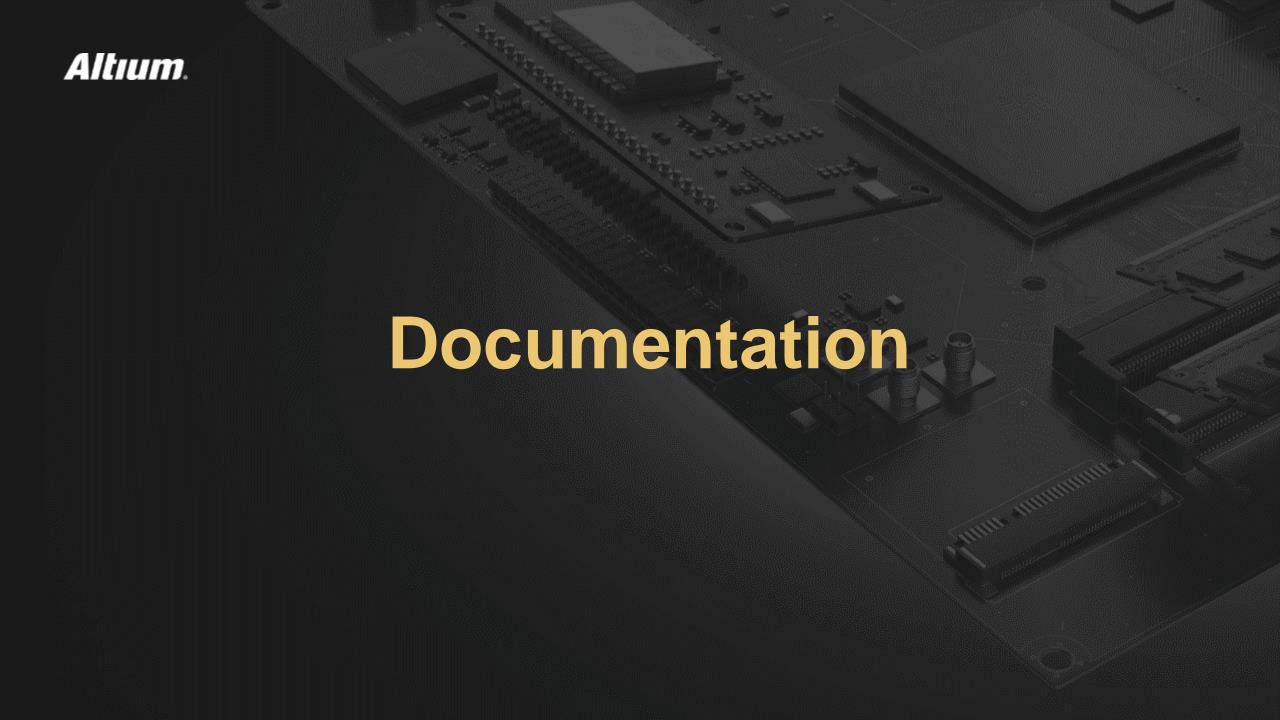




Altium. Data Management

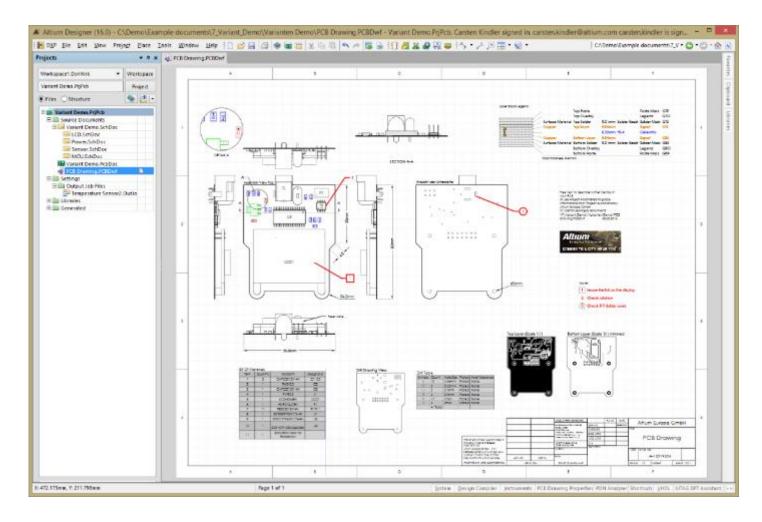
- Altium Vault is Altium's PDM system
 - Components
 - Footprints
 - Symbols
 - Managed Sheets
 - Managed Projects
 - Managed production data
 - Infrastructure Management
 - Licenses
 - User and Role Management
 - Dedicated install repositories
 - NIS
 - Multi-Board Modules can refer to Managed Projects in the Vault





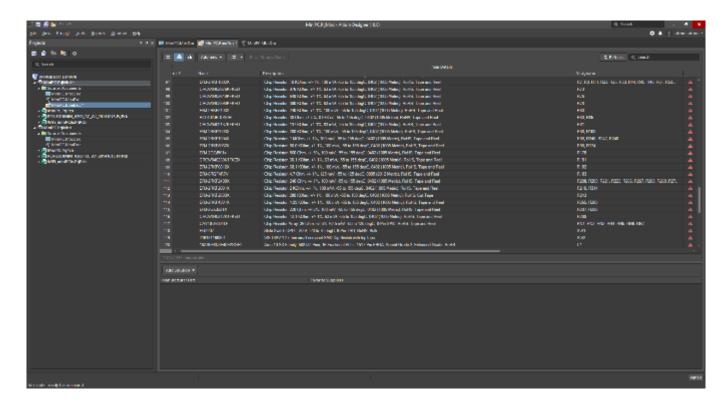
Altium. Documentation - Draftsman

- Draftsman is an extension for <u>template based</u> Technical Product Documentation:
 - Assembly
 - Assembly drawings
 - Simple BOM
 - Detailed views
 - Dimensioning
 - Assembly notes
 - Bare Board Production
 - Layer drawing
 - Layer Stack Drawing
 - Production Notes
 - Dimensioning
 - Drill table/Drill drawing
 - Update after PCB changes
 - PDF output



Altium. Documentation – Multi-PCB

- Technical Production has been produced per child PCB Project
- A .BOM document can be added to the Multiboard project
 - List of all components in the Multi-PCB Design





A real example of Multi-PCB

DOME microserver for exascale computing (IBM/Astron/DsignWorx)



A real example of System Design – DOME Project (IBM/Astron/DsignWorx)

- https://www.astron.nl/r-d-laboratory/dome/dome
- Square Kilometer Array project (SKA)
- BIG DATA (2014: 15 Peta flops; 2024: 15 Exa flops)
- Exascale Technology





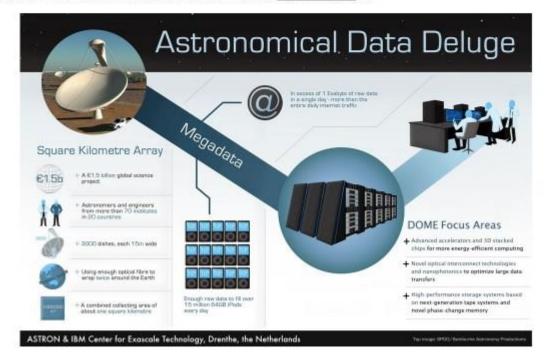


DOME

DOME is a five-year collaboration between IBM and ASTRON, The Netherlands Institute for Radio Astronomy, to research extremely fast, but low-power exascale computer systems targeted for the international Square Kilometre Array (SKA). The SKA is an international consortium to build the world's largest and most sensitive radio telescope. Scientists estimate that the processing power required to operate the telescope will be equal to several millions of today's fastest computers.

The project will investigate emerging technologies for large-scale and efficient exascale computing, data transport and storage processes, and streaming analytics that will be required to read, store and analyze all the raw data that will be collected daily. Scientists from both organizations collaborate at the newly established ASTRON & IBM Center for Exascale Technology in Dwingeloo, the Netherlands.

Further information on DOME can be found at the DOME website .



Altıum.

A real example of System Design – DOME Project (IBM/Astron/DsignWorx)

- Development of:
 - Processor Board
 - BringUp Base Board
 - MiniBase Board
 - Connector Board
 - Switch Board
 - Power Board



133mmx55mm









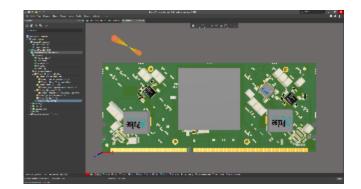




Altıum.

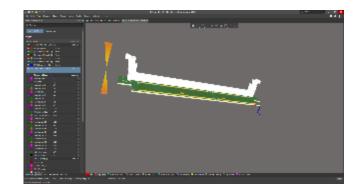
System Design – DOME Project (IBM/Astron/DsignWorx)

Processor Board

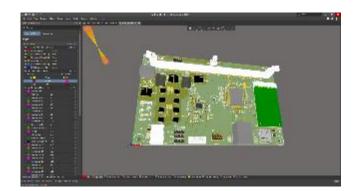


STEP based mechanical interface only

Connector Board



Mini-Base Board



No electrical interface!!!

Altium. System Design – DOME Project (IBM/Astron/DsignWorx)



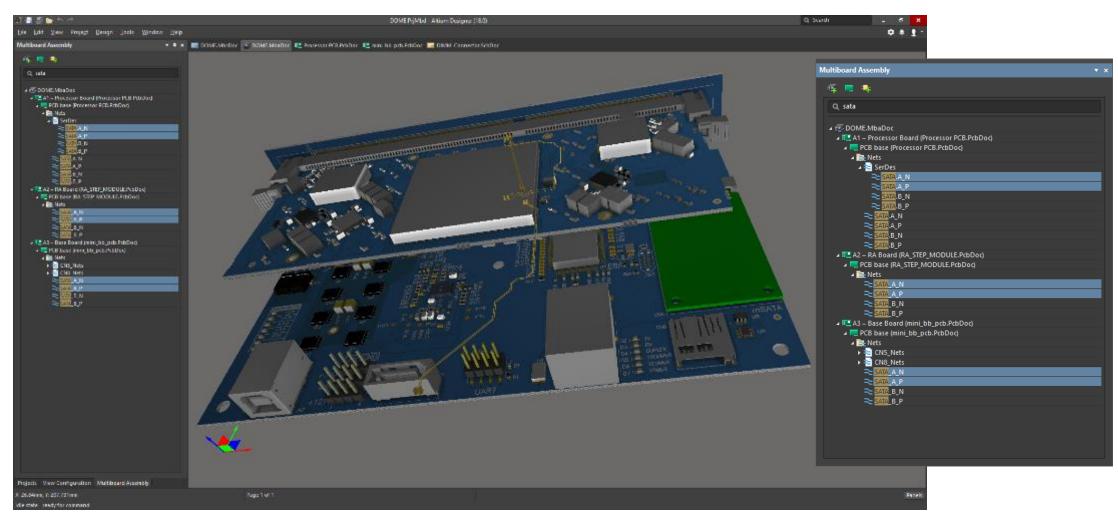
- Processor Board
- **Connector Board**
- Mini-Base Board

Multi-PCB !!!



Altıum. System Design – DOME Project (IBM/Astron/DsignWorx)





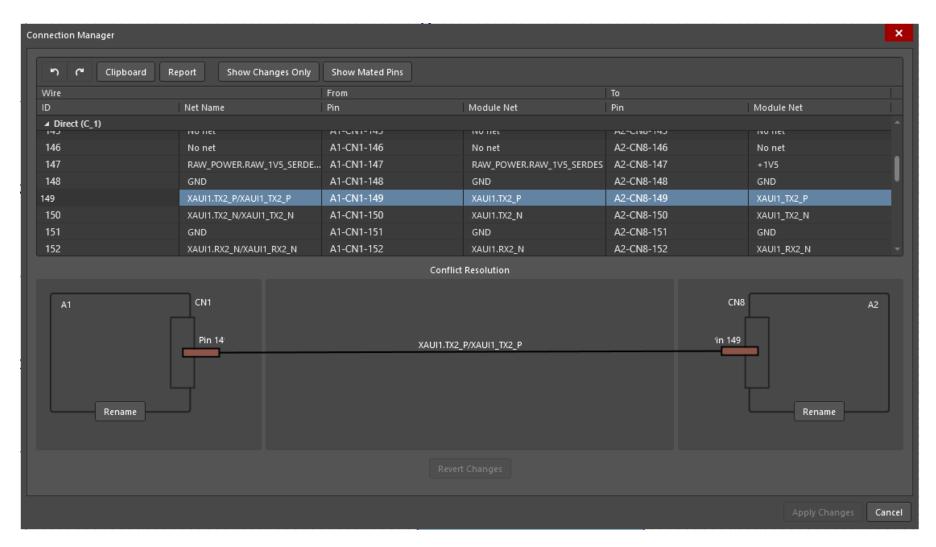
- Net Highlighting
 - SATA_A_N and SATA_A_P



Altıum. System Design – DOME Project (IBM/Astron/DsignWorx)



Connection Manager







Thanks for your attention!

Henk de Jonge Technical Director Altium BV

October 11/12th 2017, D&E Event, Mechelen/'s-Hertogenbosch