Altium.

Multi-PCB Design
The next step in 3D ECAD-MCAD

Henk de Jonge
Technical Director Altium BV

November 8th 2018, D&E Event, Eindhoven, Booth #16
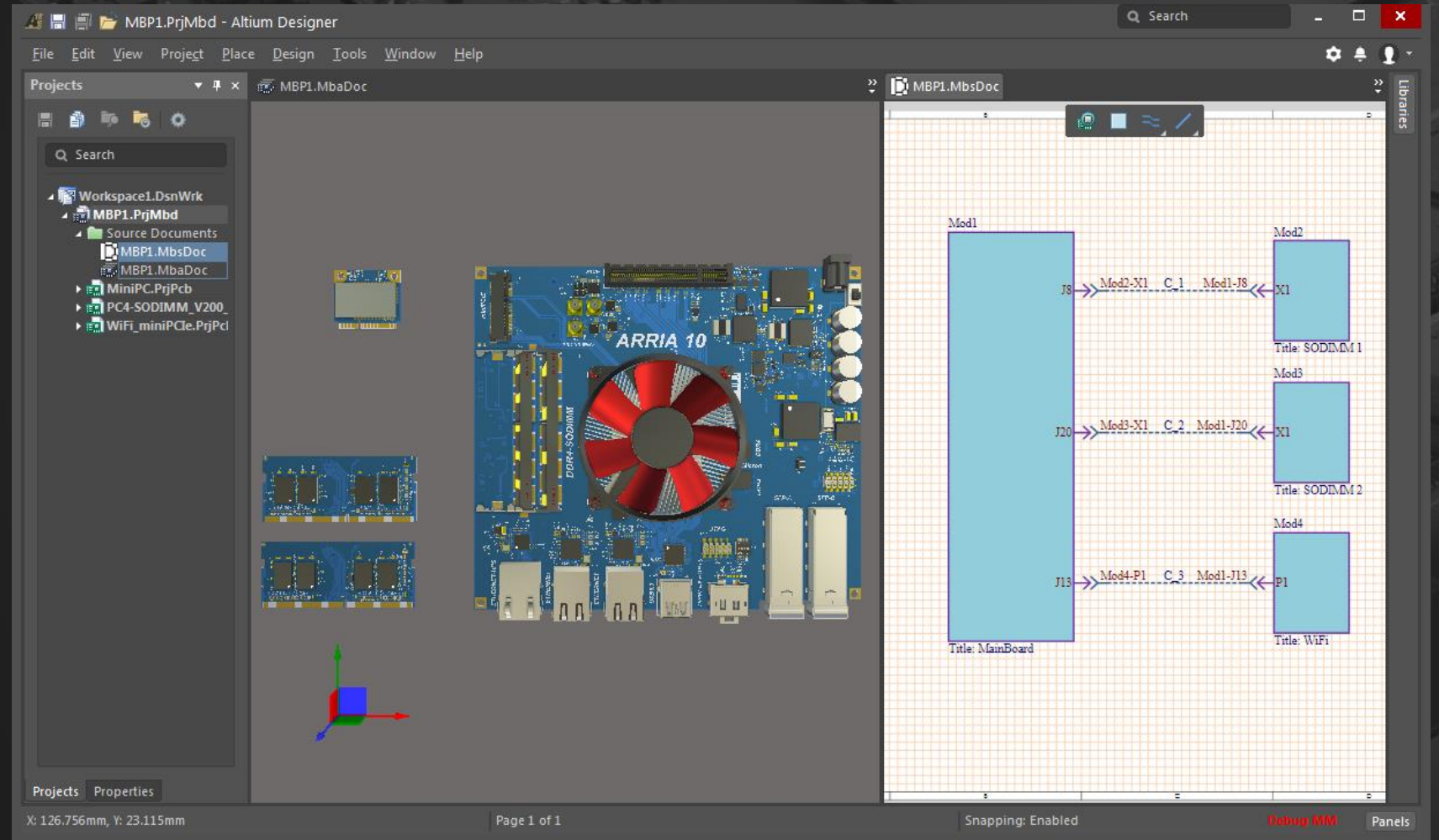
DESIGN AUTOMATION EMBEDDED SYSTEMS

8 NOV ←
VAN DER VALK HOTEL
EINDHOVEN

D&E event 2018

FPGA - SECURITY - INTERNET OF THINGS - ELECTRONIC DESIGN & PRODUCTION - EMBEDDED

- Altium LLC
- Design content
- Multi-PCB Design
- General
- Setup
- System Design
- Assembly
- New in AD 19

Automotive

Aerospace & Defence

Life Sciences

Mobile Devices & Communications

Consumer - Electronics

Electronics & High-Tech

Research & Education

Industrial Controls Automation

Computers

Semiconductors

PCB Design content

- PCB design starts with collecting content
  - Datasheets
  - Models
    - Symbols (schematic)
    - Footprints (PCB design)
    - STEP models (3D)
    - IBIS models (Signal Integrity)
    - Spice models (Analog simulation)
  - Supply chain information
    - Pricing
    - Component Life Cycles
      - Availability
      - Alternatives
  - Other content
    - PCB Templates
    - Reference Designs
- Can be a tough and time consuming process …

- Manually
  - Symbol Editor
    - Symbol Wizard
  - PCB Footprint Editor
    - IPC compliant Footprint Wizard

- Import from other CAD designs
  - Altium
  - Other EDA software

- Copy → Paste → Modify → Save
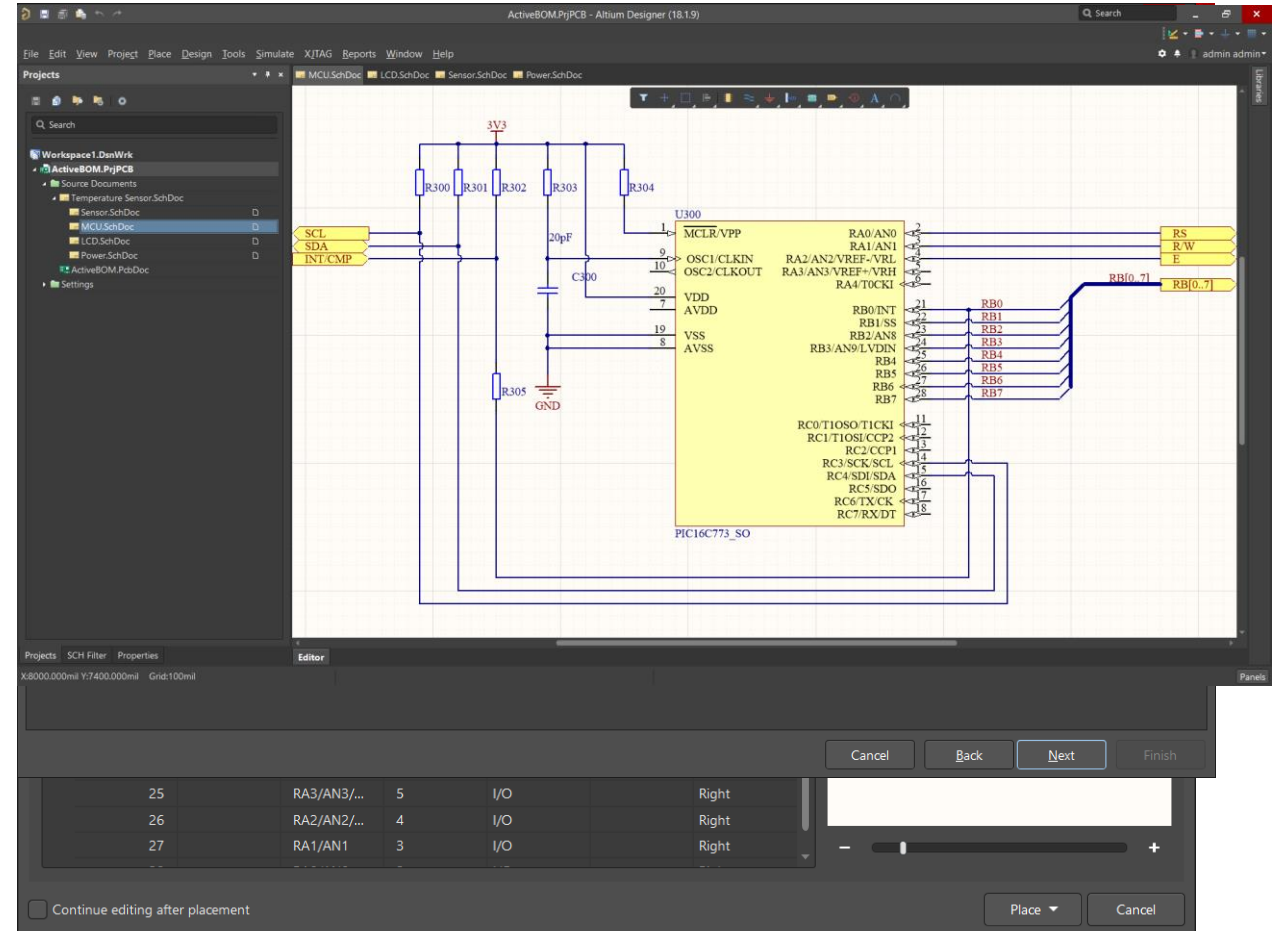
- Altium Content Vault
  - Free for all Altium Designer and Altium Nexus users
  - Place direct from Explorer panel
  - Download via the Explorer panel
  - More than 500,000 models
- Altium website
  - https://designcontent.live.altium.com/
- Octopart
  - Wikipedia for components
  - Free to use for every engineer
  - www.octopart.com
- Ciiva
  - Component Lifecycle Management
  - www.ciiva.com
- EE Concierge
  - Buy On Demand content
  - https://eeconcierge.com/
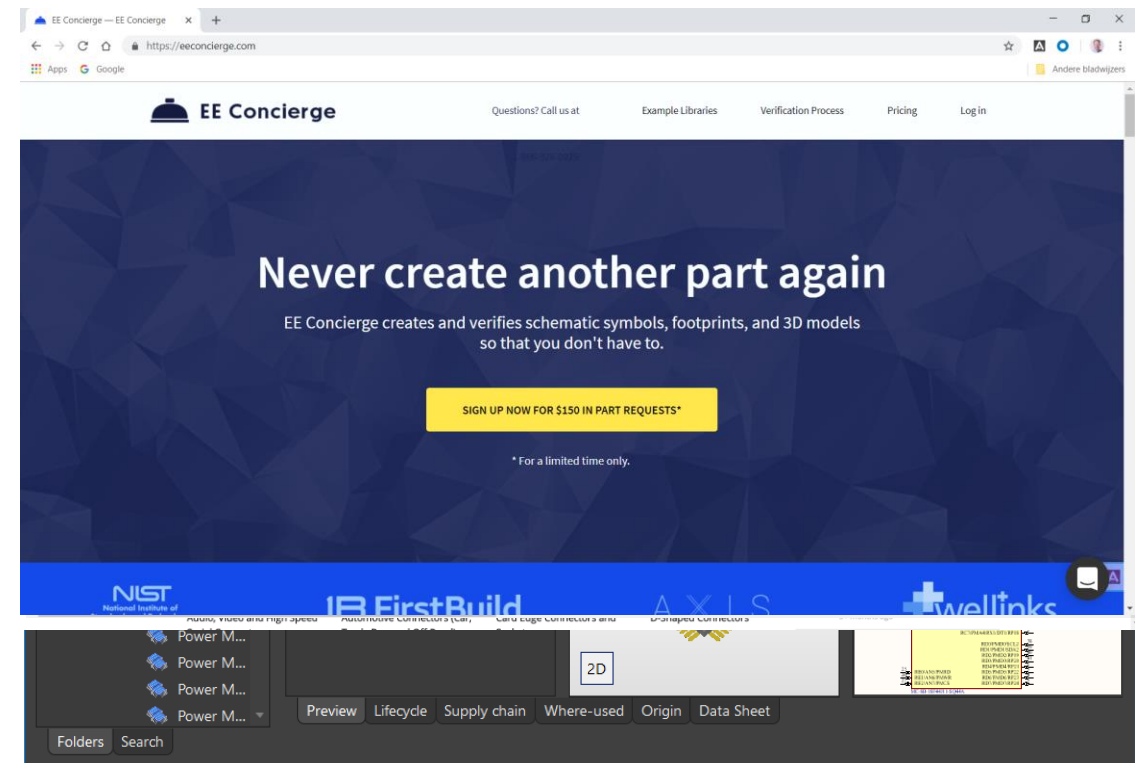
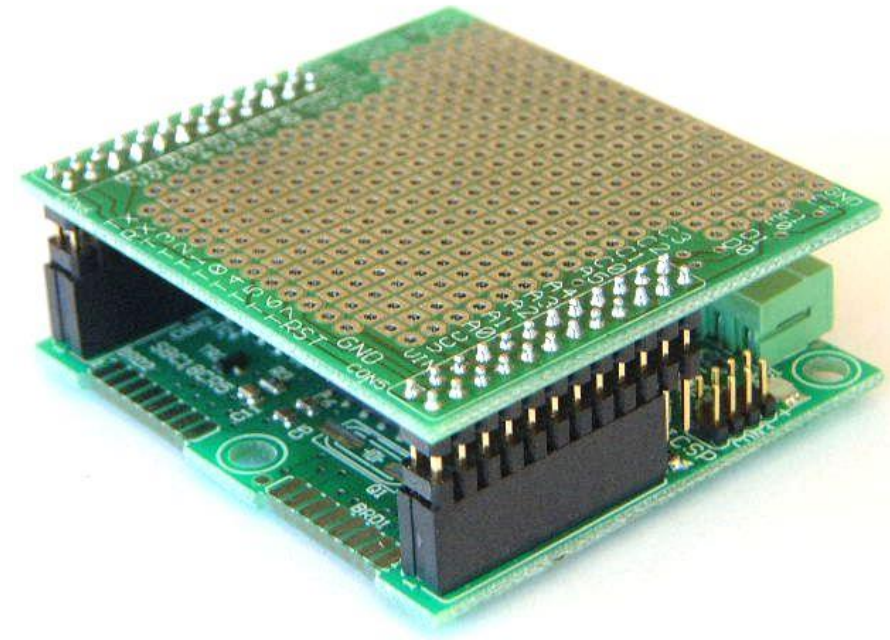# **Multi-PCB design**

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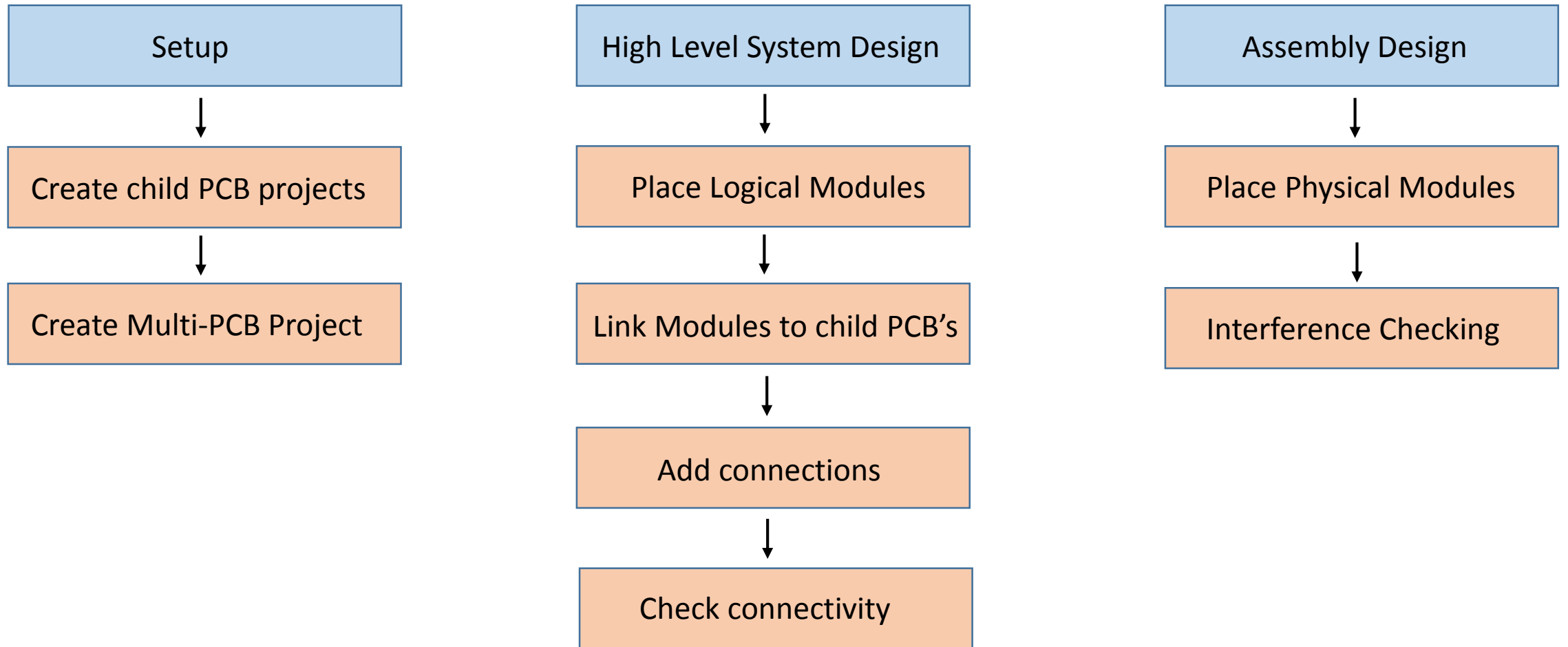   - Cost of redesign
    - Delay to market

- Design challenges:
  - Design iterations between the various PCB's
  - Manage Electrical connections
  - Manage 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 started in Altium Designer 18**

Multi-PCB Design – A Possible Work Flow

Setup
↓
Create child PCB projects
↓
Create Multi-PCB Project

High Level System Design
↓
Place Logical Modules
↓
Link Modules to child PCB's
↓
Add connections
↓
Check connectivity

Assembly Design
↓
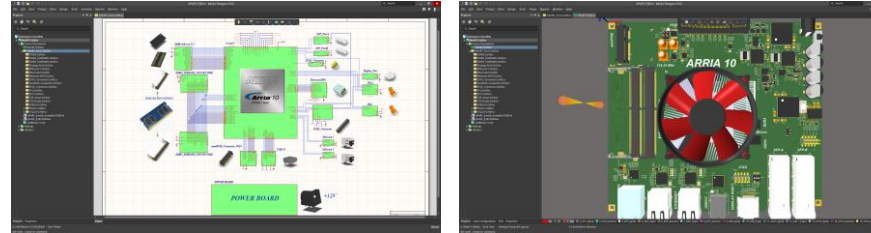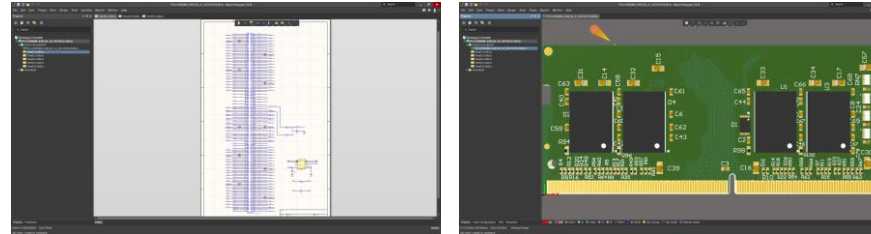Place Physical Modules
↓
Interference Checking

Multi-PCB setup

Setup

Create child PCB projects

- Create child PCB projects (Modules)
  - Motherboard
  - SODIMM
  - WiFi Module
- Create child PCB outlines
  - Needed for early Multi-PCB design
- Place board-to-board connectors
  - Needed for early Multi-PCB design
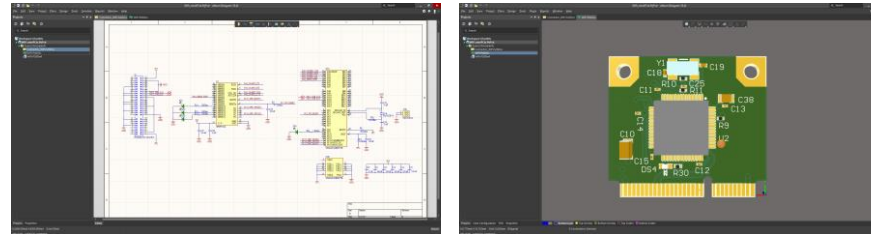 - Use parameter: System = Connector
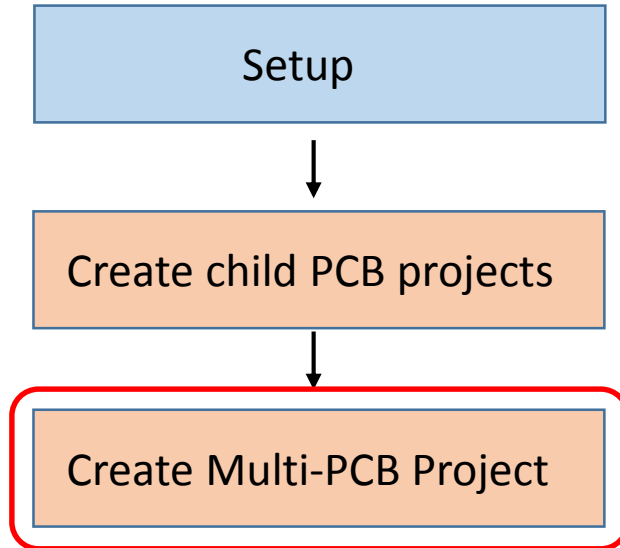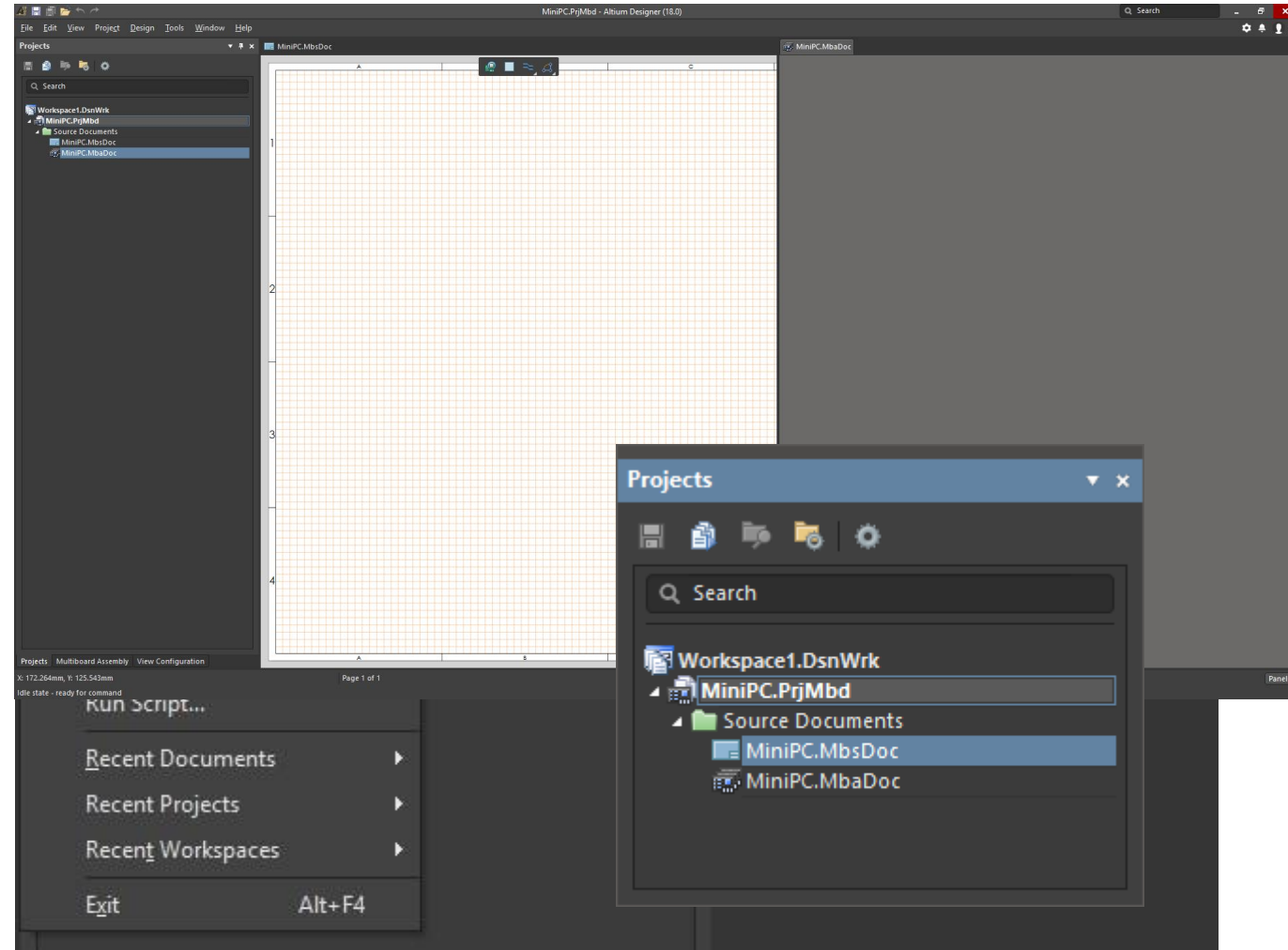
Mini-PC Mother board

SODIMM board

WiFi Module board

Setup

↓

Create child PCB projects
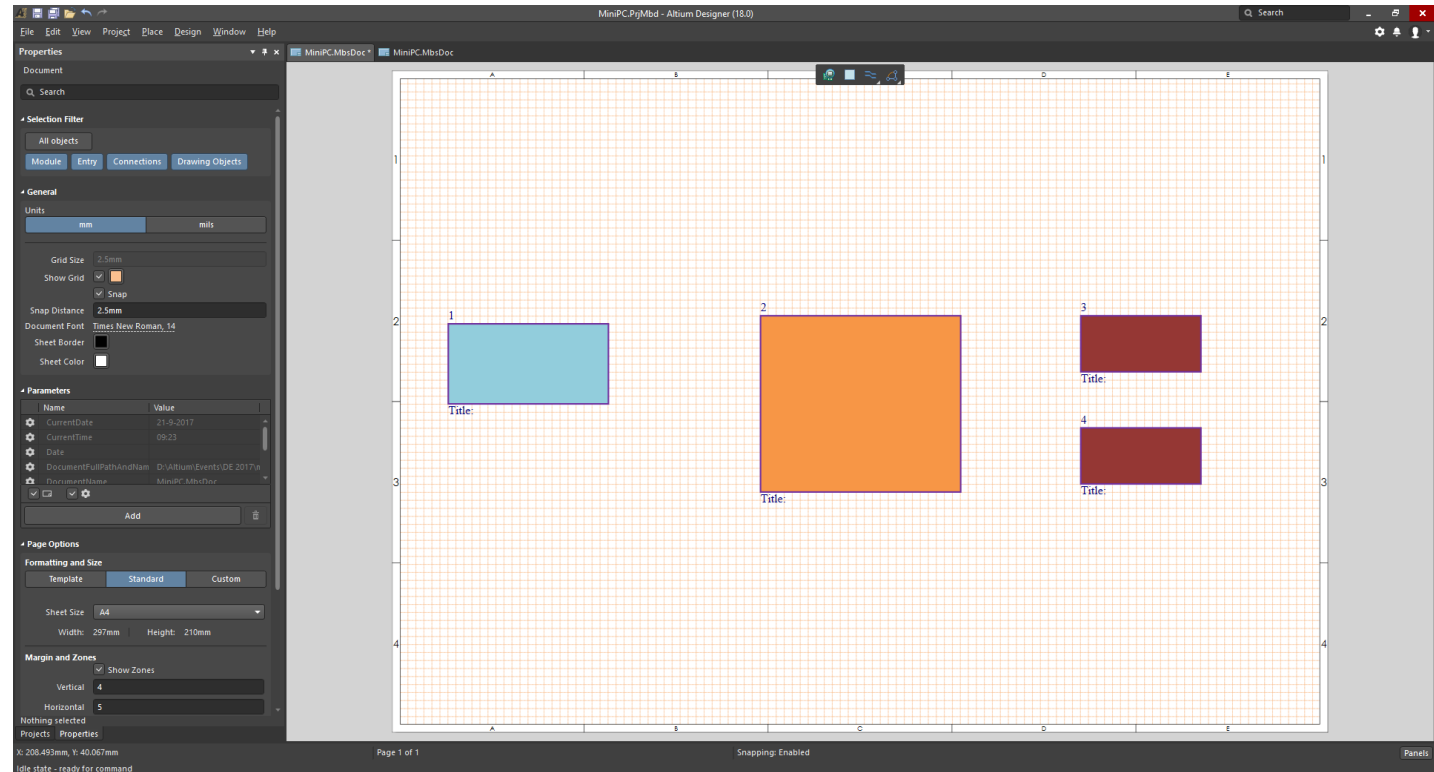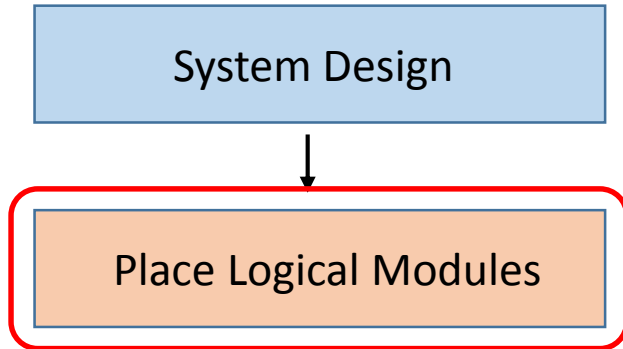
↓

Create 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)
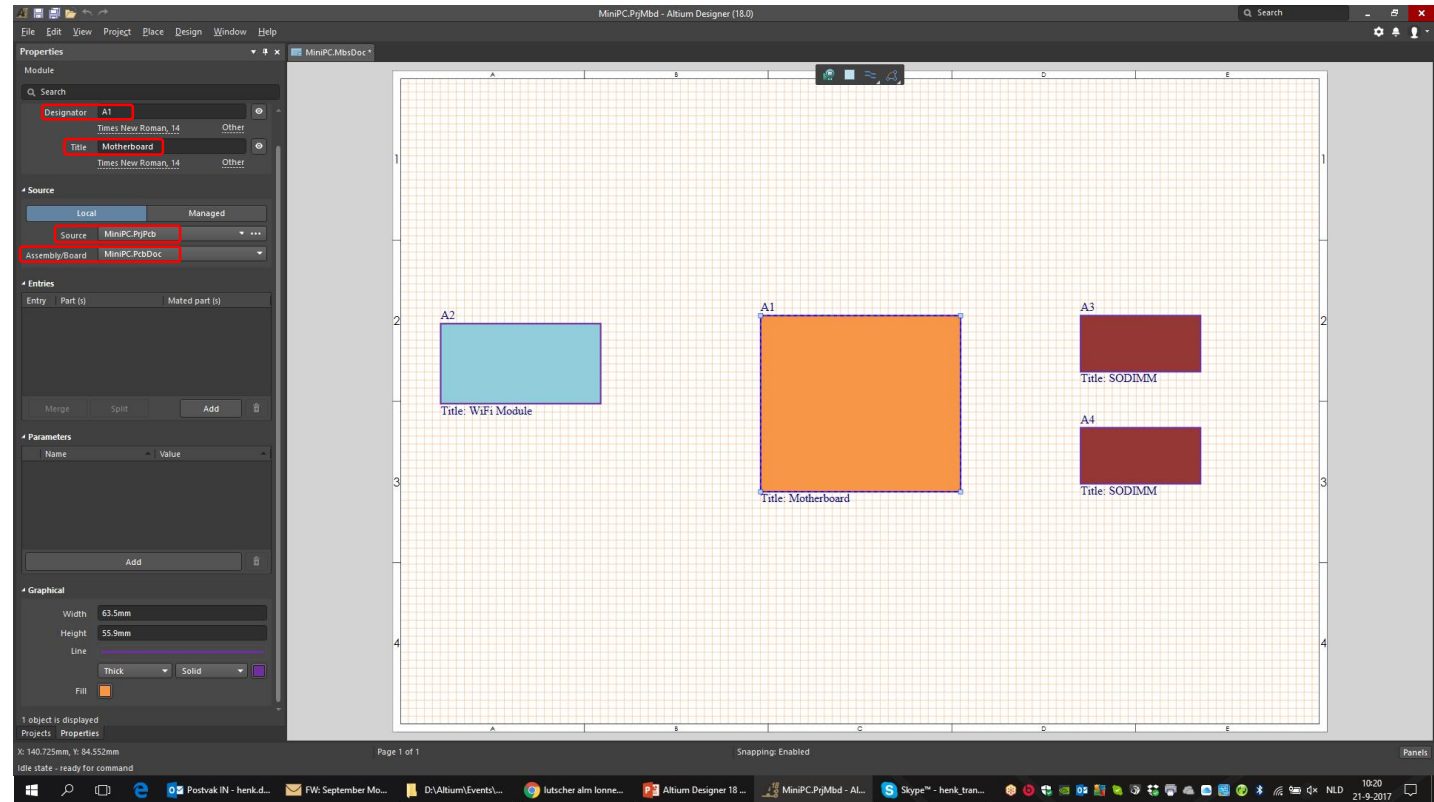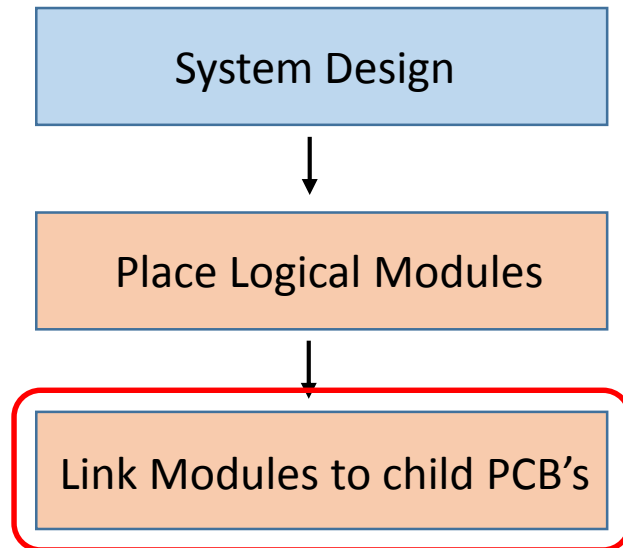 - Project structure

Multi-PCB system design

System Design

↓

Place Logical Modules

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

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

Multi-PCB Design – Add connections

System Design

Place Logical Modules

Link Modules to child PCB's
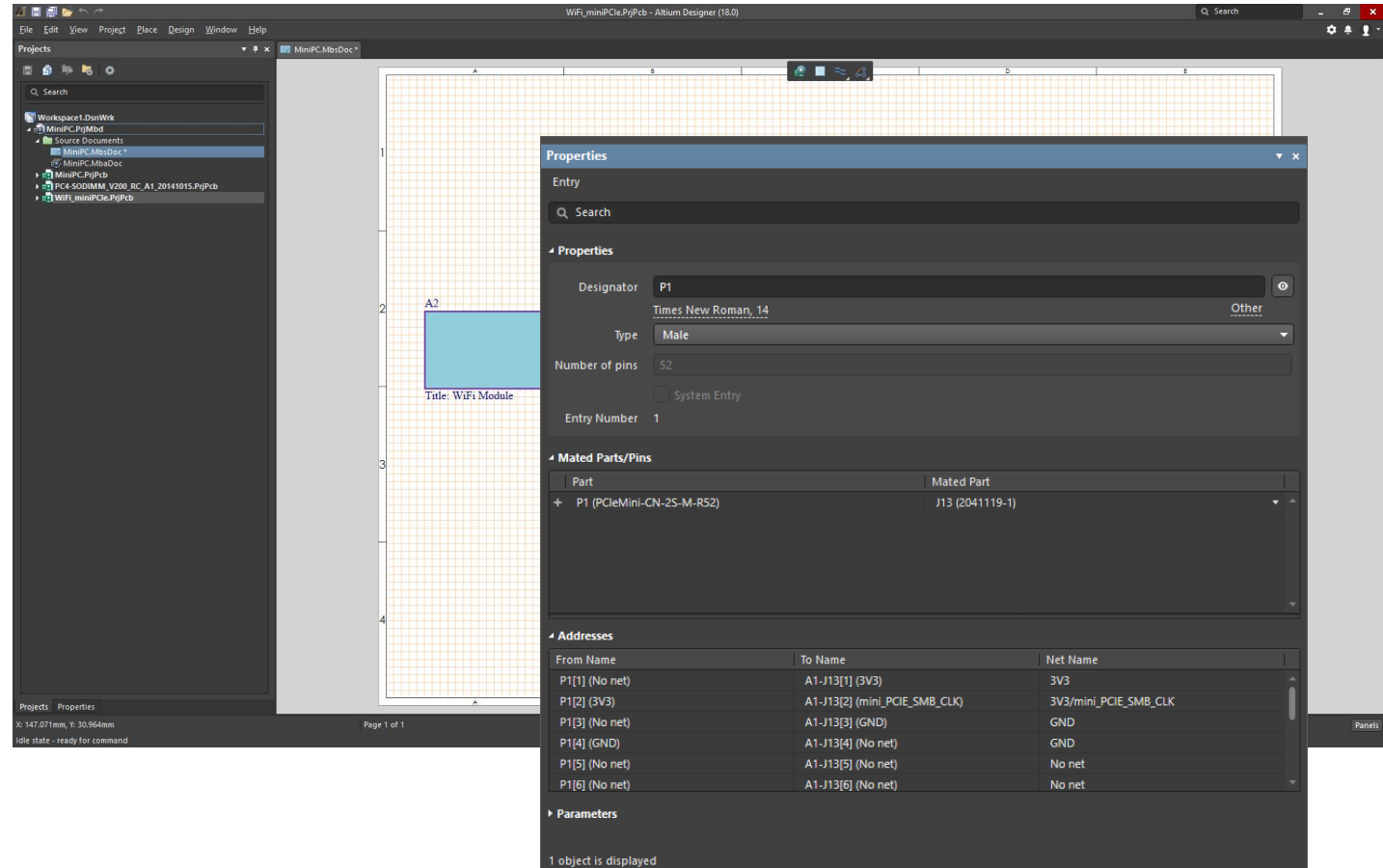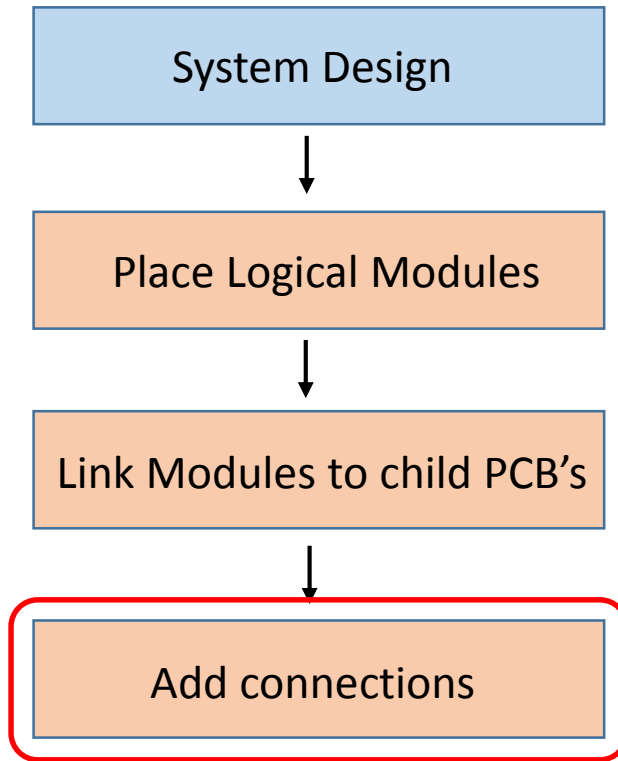
Add connections

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

System Design

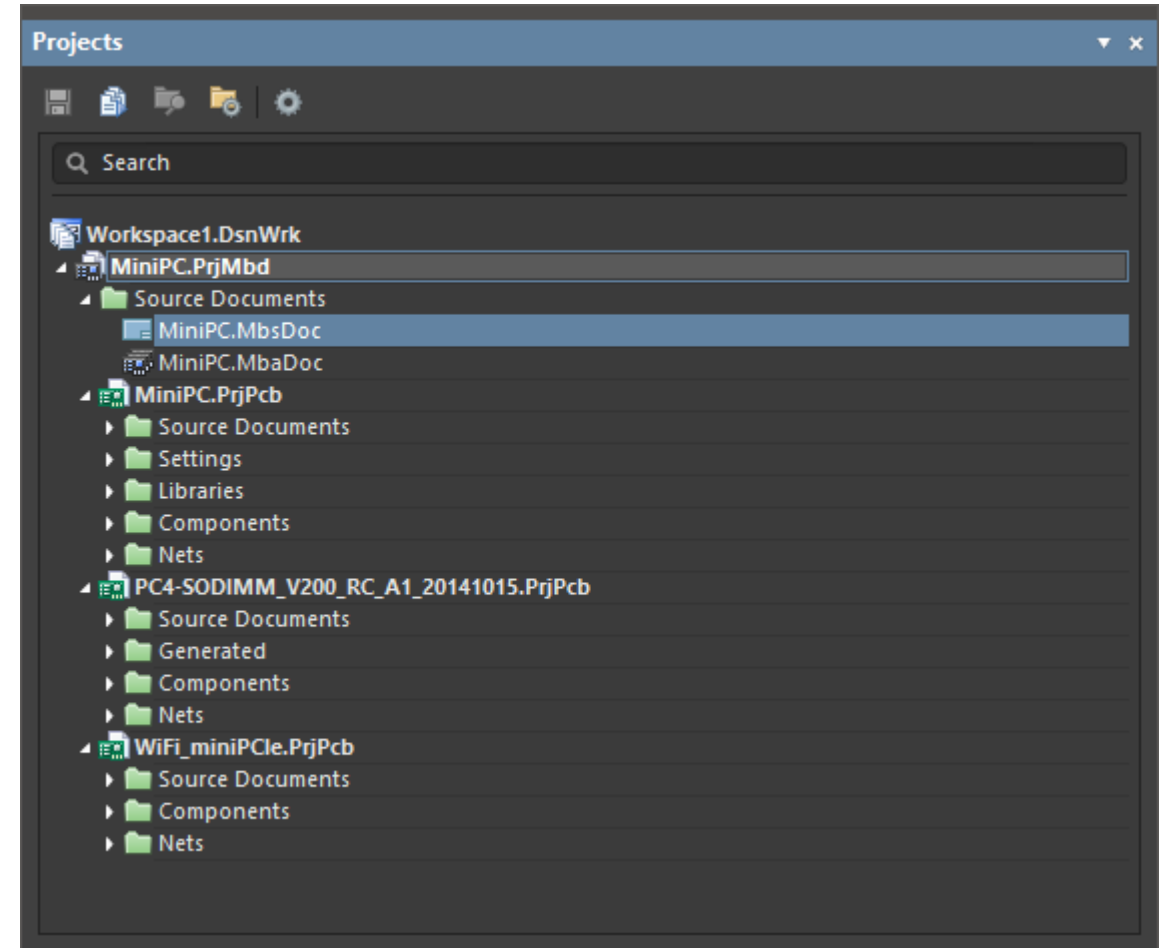Place Logical Modules

Link Modules to child PCB's

Add connections
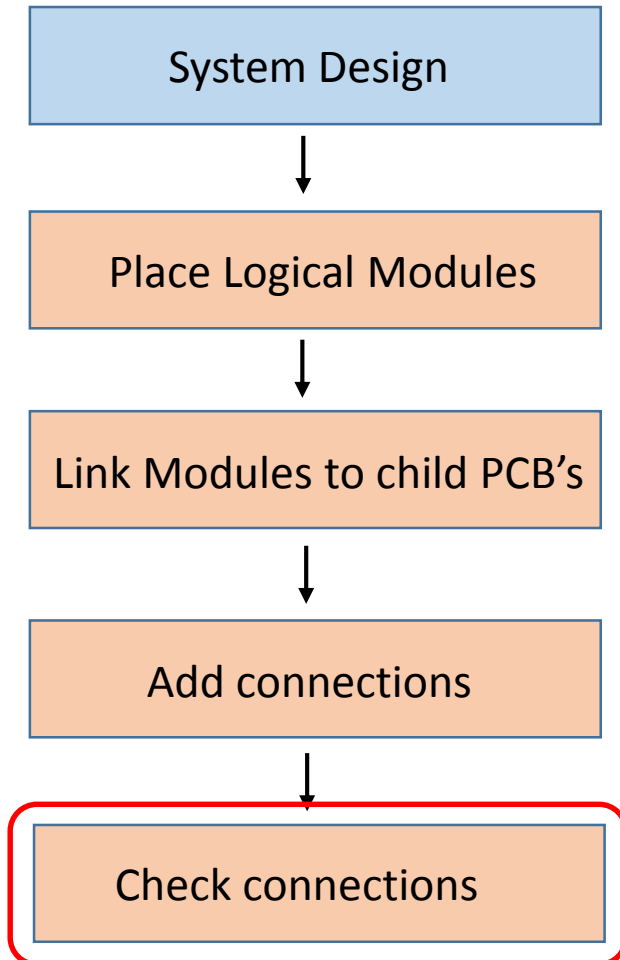
Check connections

- Use the Connection Manager for:
  - Overview of the overall connectivity
  - Checking and resolving conflicts
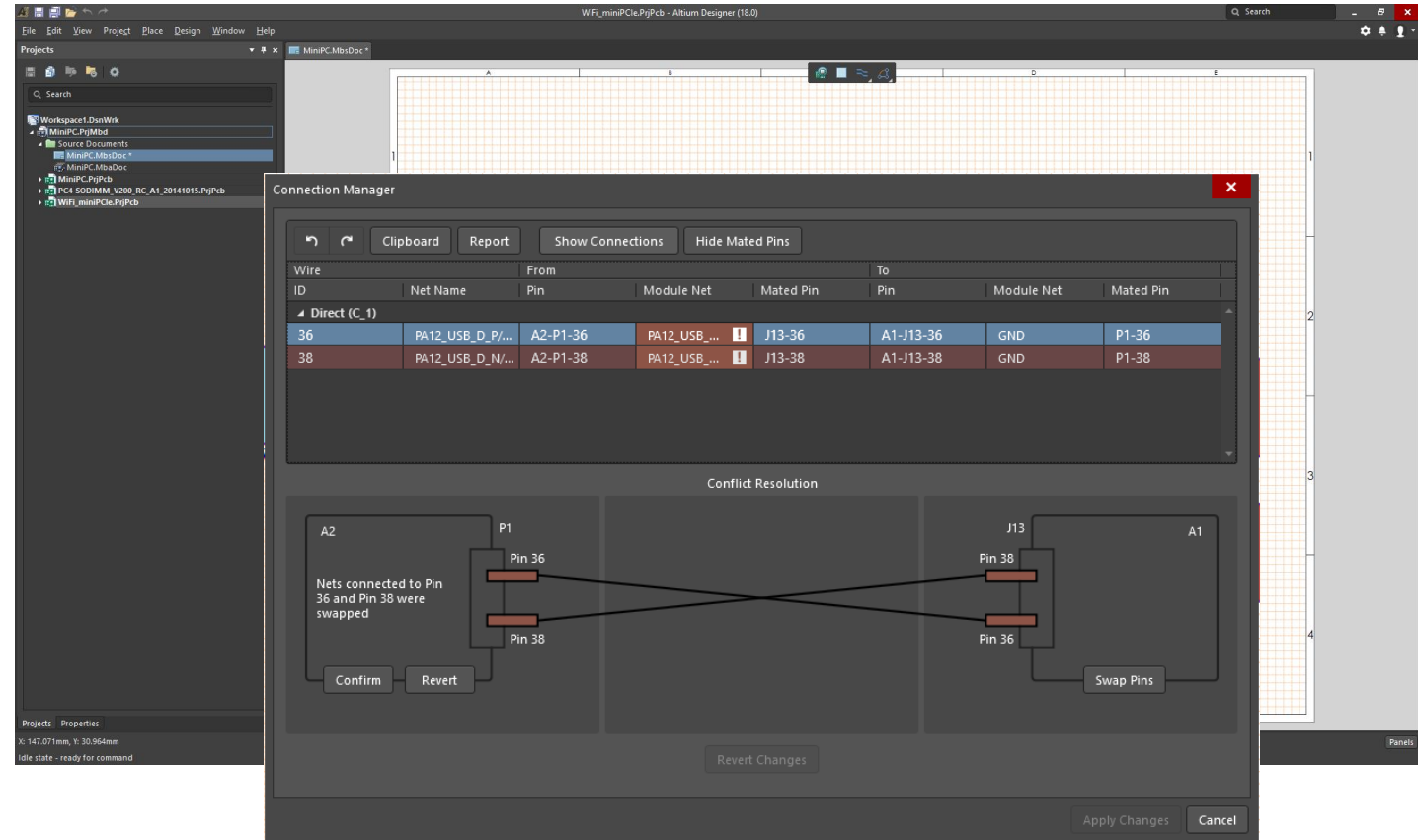
- After setup and system design, the project structure should look like this:
  - Multi-Board Project (*.PrjMbd)
    - Schematic Multi-Board (*.MbsDoc)
    - Physical Multi-Board (*.MbaDoc)
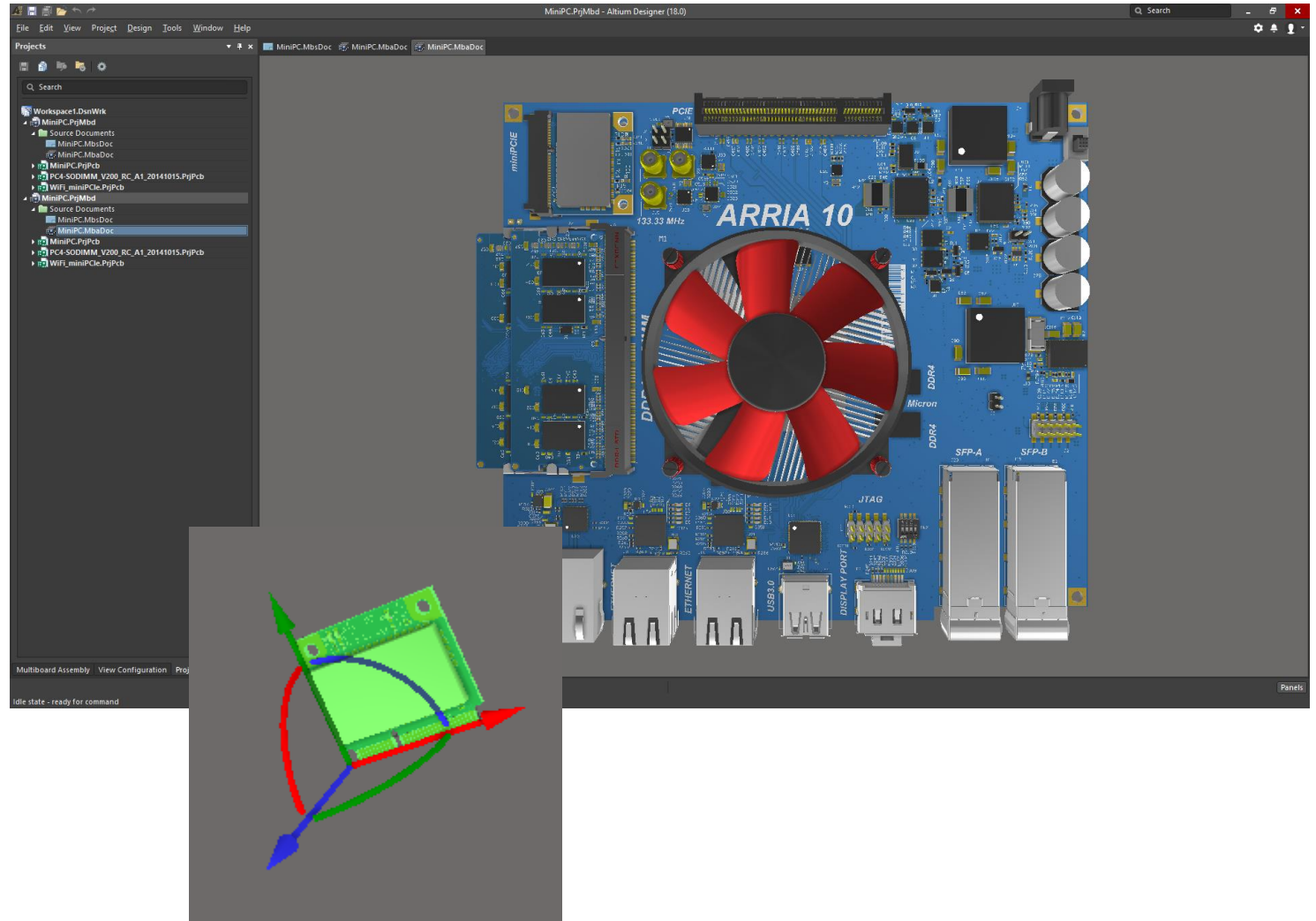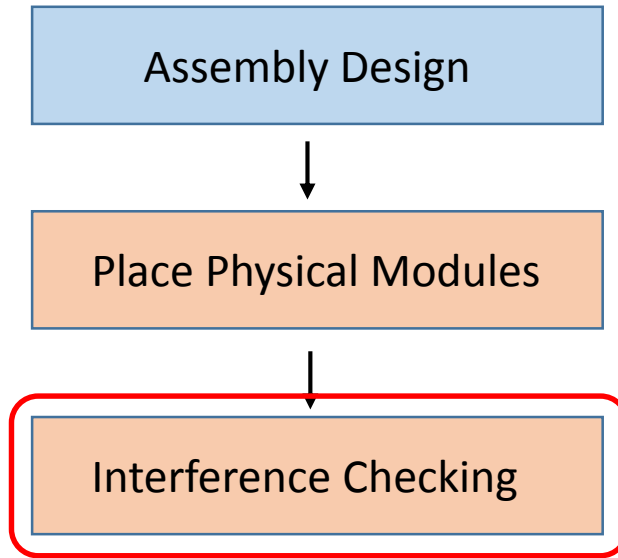   - Child PCB projects

Multi-PCB assembly
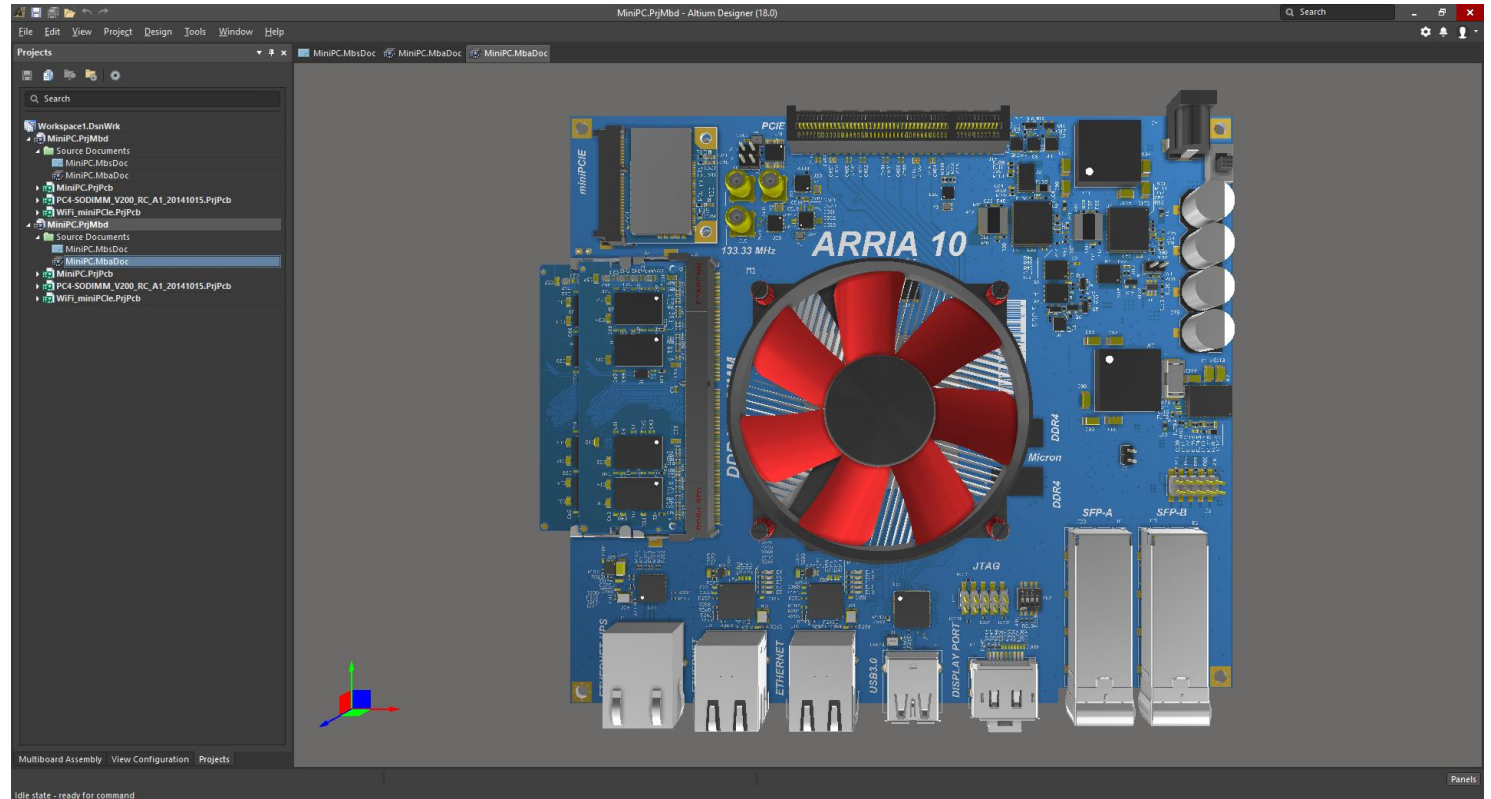
Assembly Design

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

Assembly Design

↓

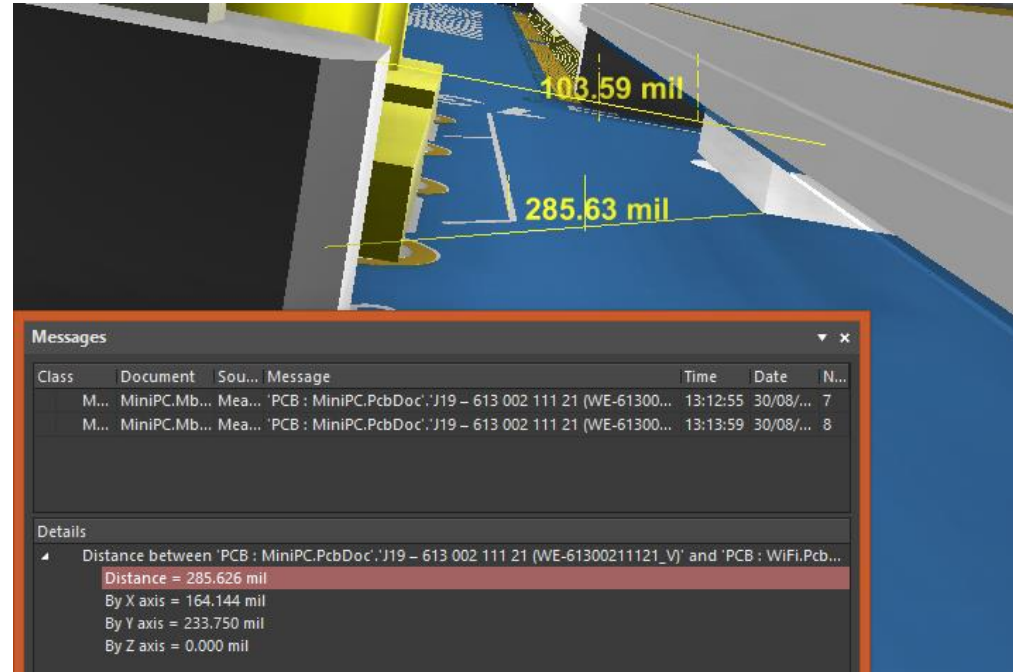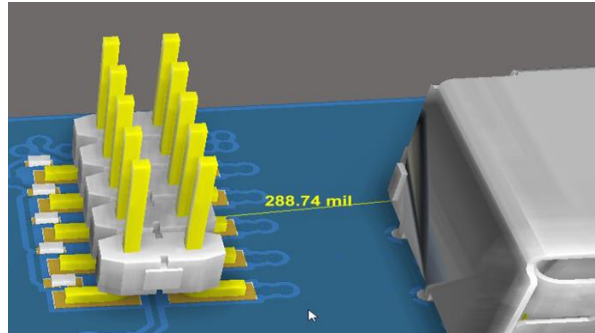Place Physical Modules

↓

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
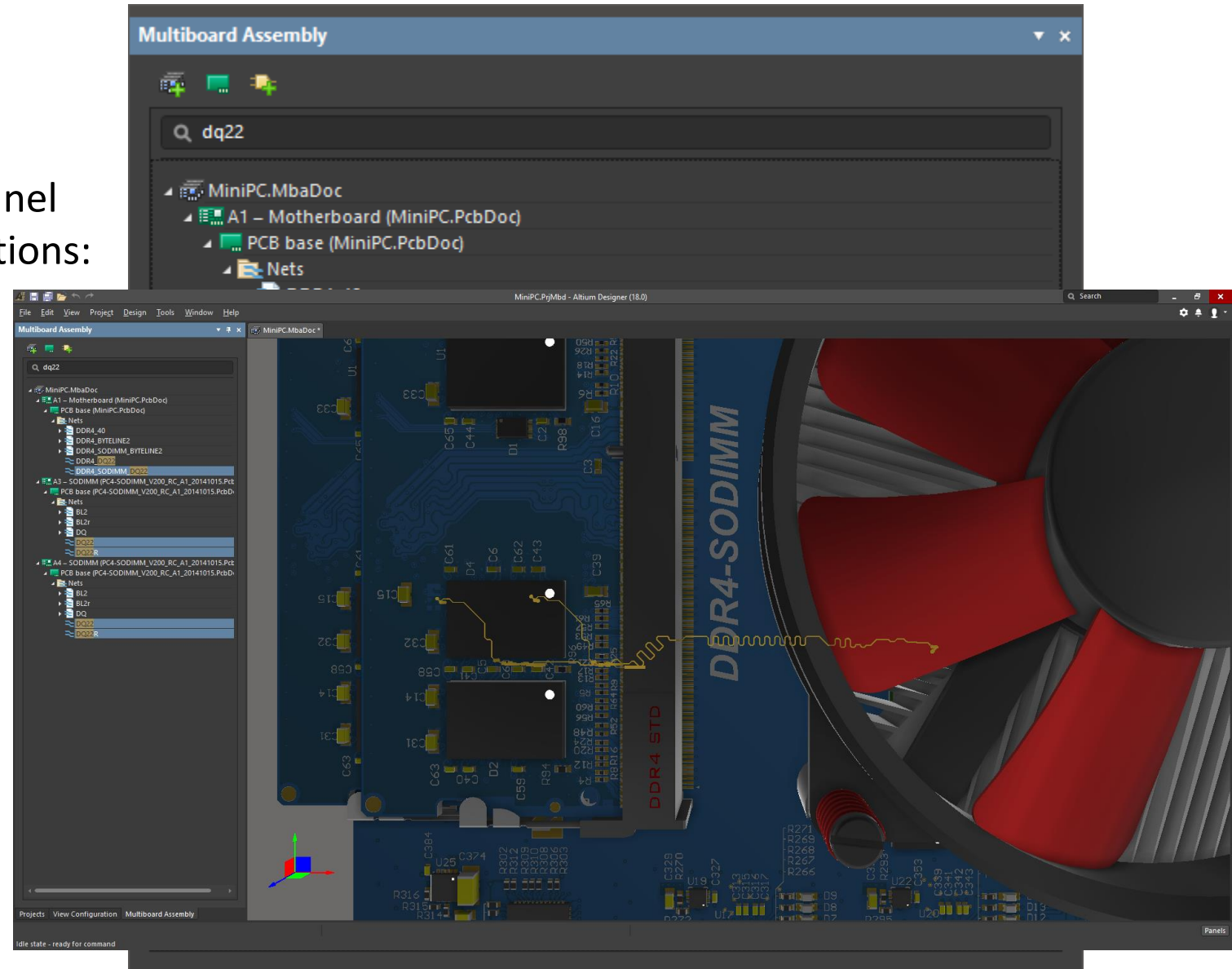   - Update child PCB Project
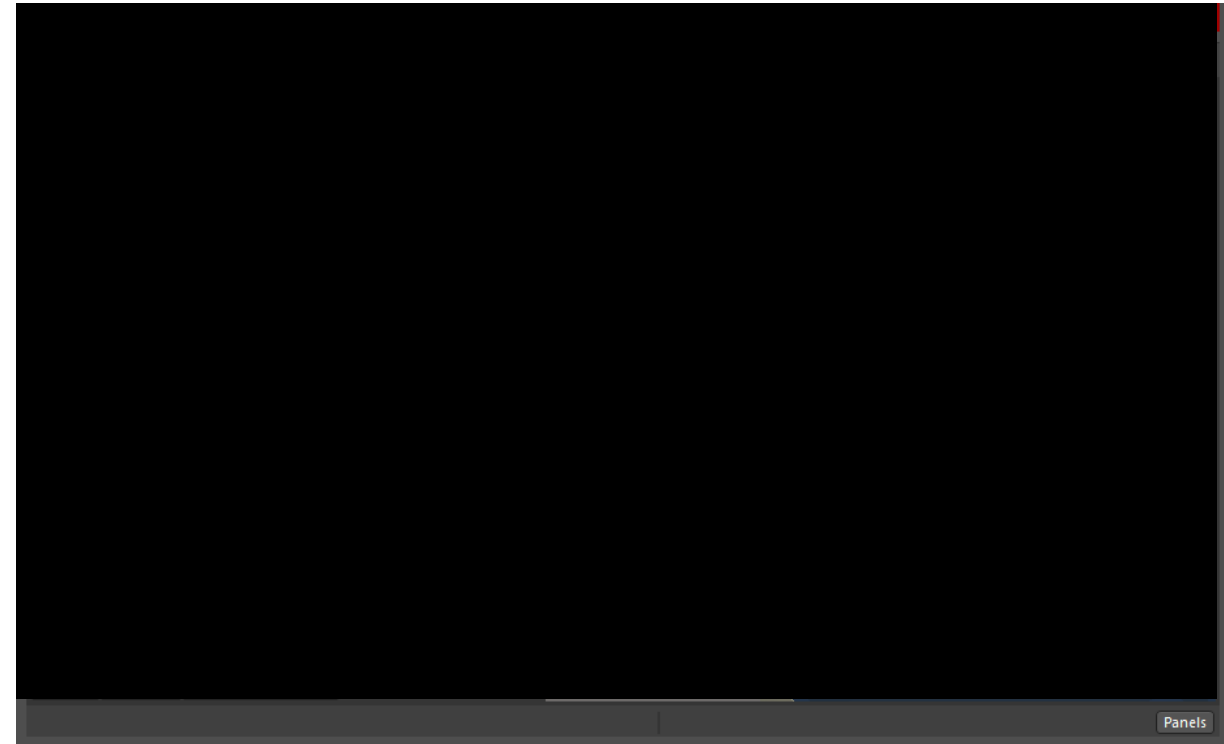
- Measuring

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

Future developments

- Multi-PCB design started in Altium Designer 18

- More to come in Altium Designer 19 (MCAD like editing functionality):
  - Export to STEP for complete assembly
  - Export to Parasolid (*.x_t)
  - Ability to Mate objects with:
    - Object Mating based on a chosen surface location
    - Locking of Mates
    - Manipulation of Mates as a single object
    - Separating Mates by a specific distance
  - Enhanced and faster collision checking
  - Enhanced and faster section view
  - Flex-rigid designs, viewed in the folded state

- These new Multi-board features are delivered by a new 3D engine. This requires an update to the Multi-board file format, so existing Multi-board designs must re-import the child PCB assemblies (**Design » Import Changes**).

- Mate:
    - Connection formed between 2 separate objects.
    - The connection is at a user-selected point on a surface on each object
    - Once mated the objects will re-orient so their surface planes and their perpendicular axes are aligned
    - Mate site:
        - A location nominated on the surface of an object, to be used as the point where the Mating occurs.
    - Mate axis:
        - The perpendicular axis passing through the center of the Mate
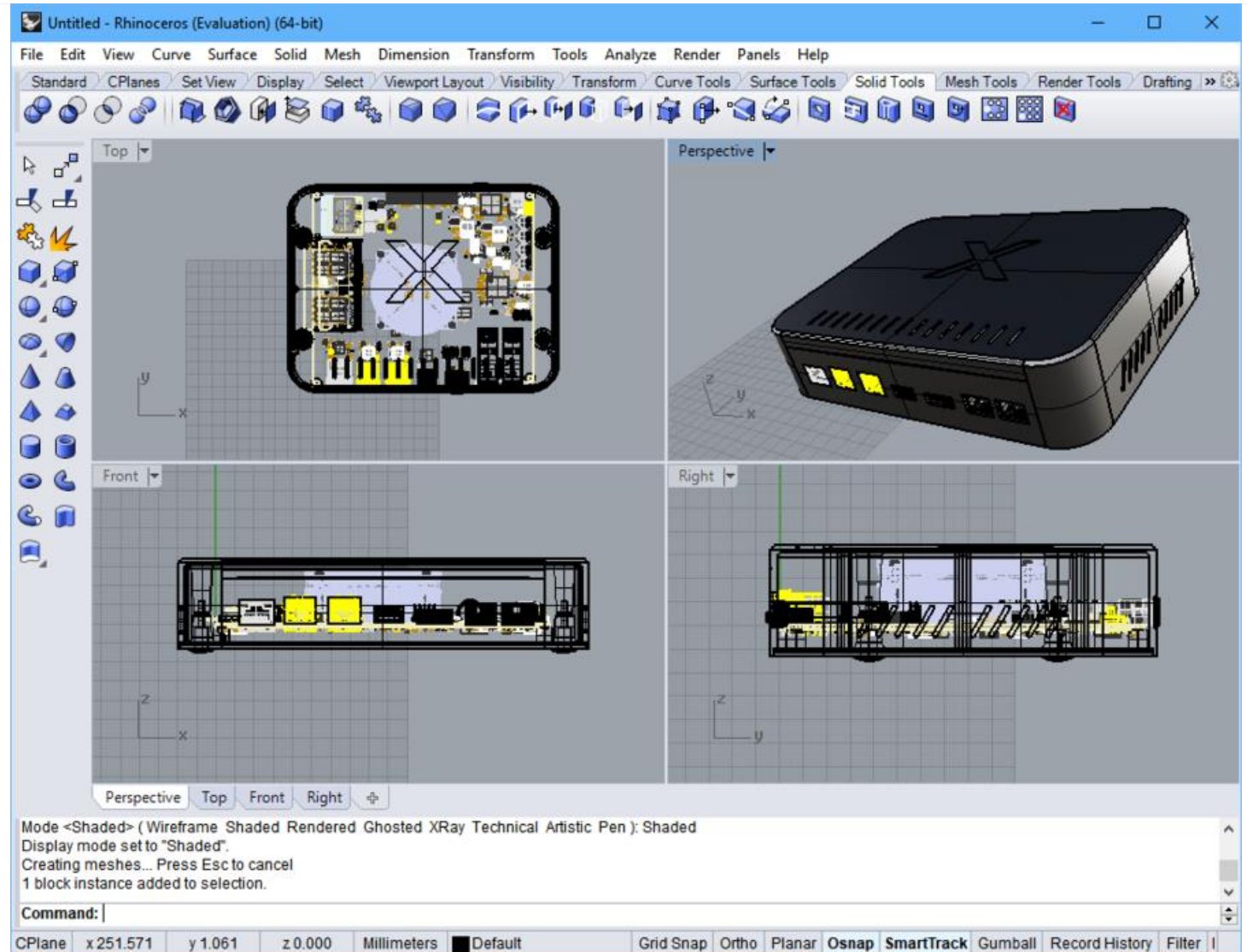   - Mating example:

- Use the **Section Plane Gizmo** to move or re-orient the plane by clicking and dragging on the required Gizmo line or arc, as shown in the image.
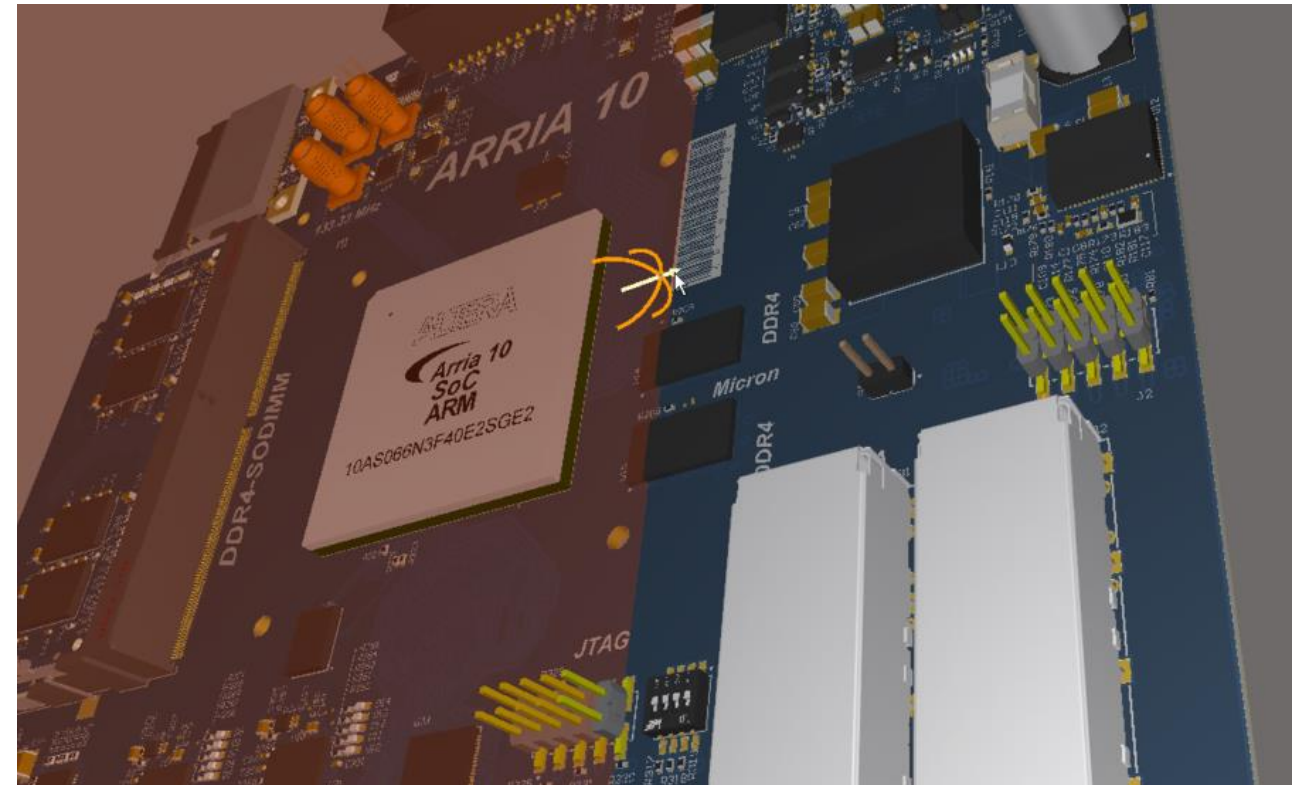
Multi-PCB Design improvements – Export to STEP

- File » Export » STEP

Thanks for your attention!

Henk de Jonge
Technical Director Altium BV

November 8th 2018, D&E Event,
Eindhoven
Visit us at Booth #16