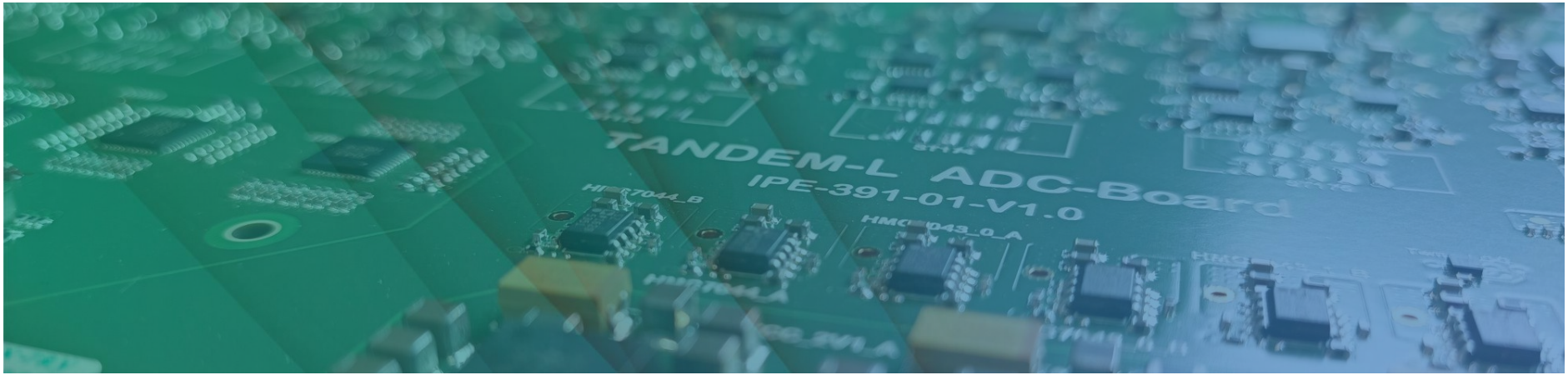
Karlsruher Institut für Technologie

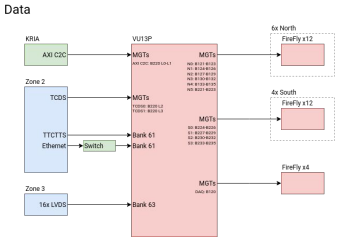
Altium Designer am IPE

Torben Mehner



25th Apr, 2024

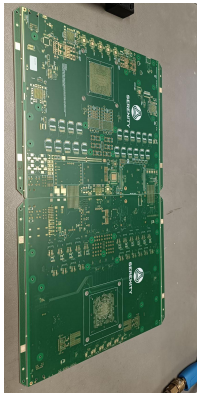
PCB Design Process



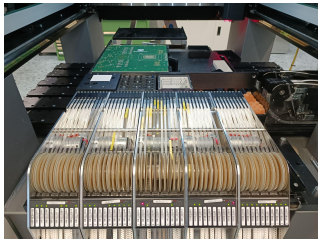
System Design

ALTIUM DESIGNER

Schematic Capture
PCB Layout



PCB Production

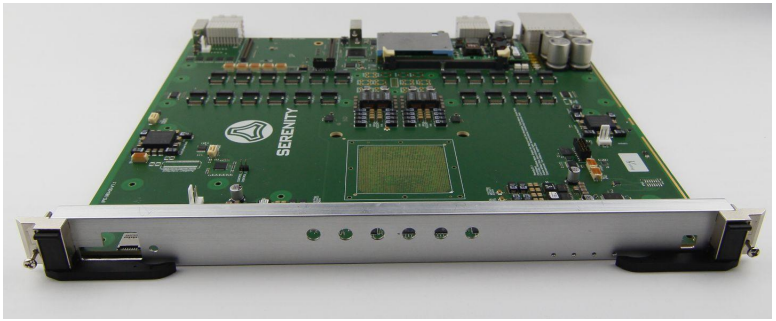


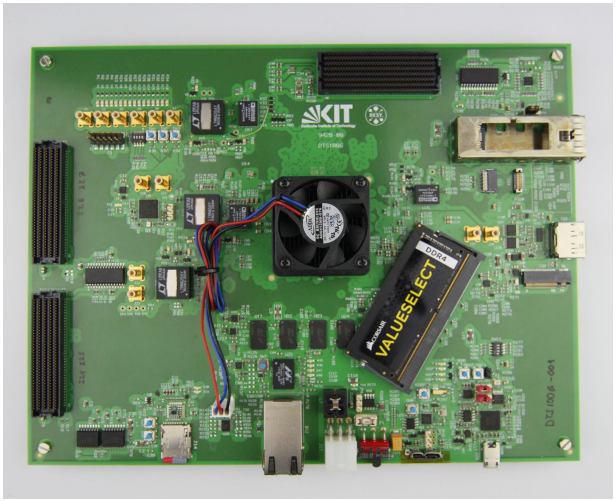
PCB Assembly

PCB Commissioning

System Design

- Off-the-shelf general purpose systems
  - Serenity-S1, DTS-100G, HiFlex
  - Little to no design work
  - Soft-/Firmware prepared once
- Common set of standardized components
  - Present in component library
  - Purchase in bulk (manufacturing/price)
  - Soft-/Firmware prepared once

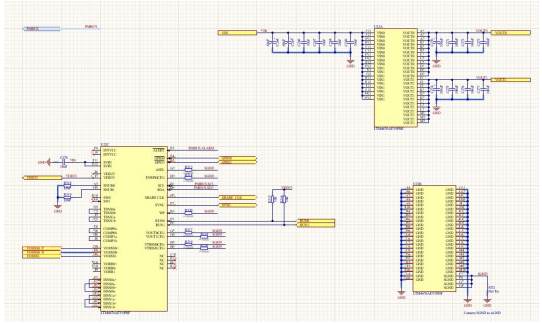




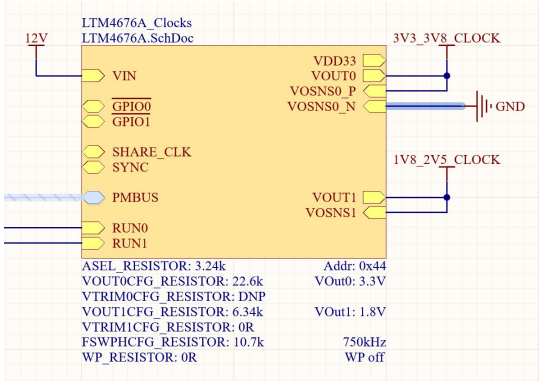
Schematic Capture and PCB Layout

- Start working in modules now
  - Modules include typical application schematics and recommended layout
  - Modules fulfil a single task (e.g. power supply, clock)
  - Reuse modules from library
- Common set of standardized components
  - Present in component library
  - Purchase in bulk (manufacturing/price)
  - Soft-/Firmware prepared once

Why do this?



When you can do this?



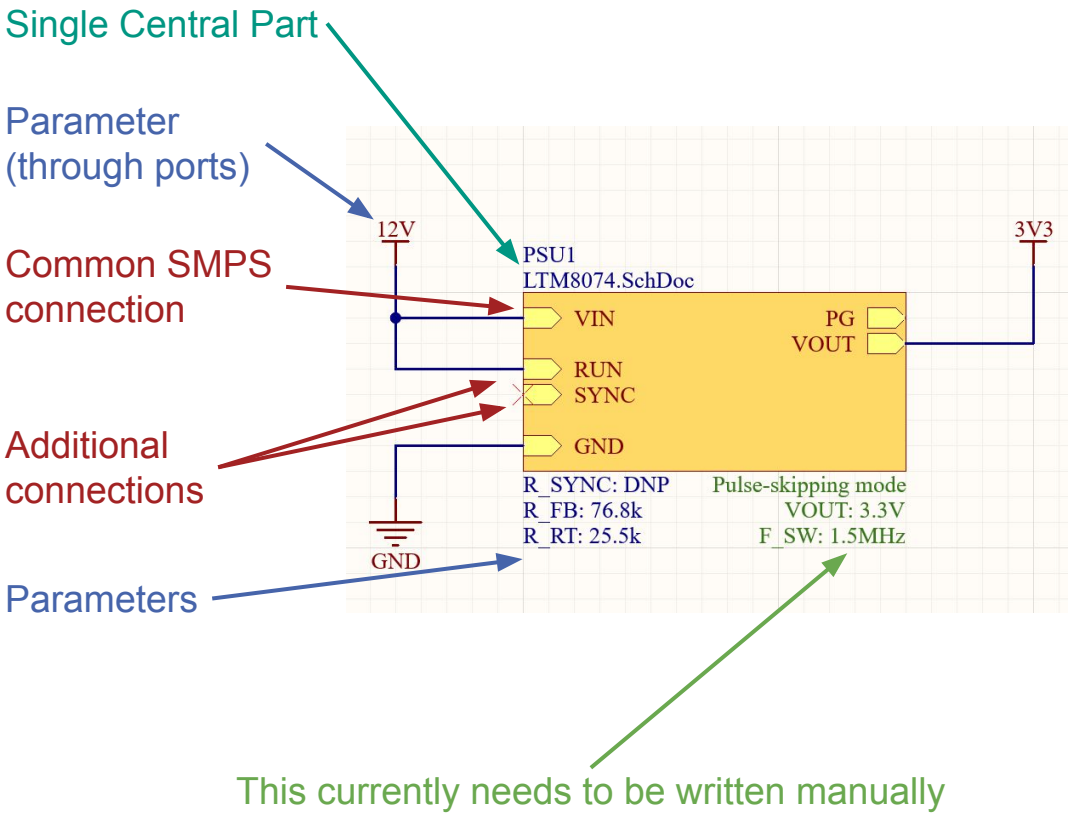
Guidelines for Modularity in EDA

- Hierarchical Design
- SOLID design principles
  - Single Responsibility
    - Single central part or
    - Single functionality by multiple parts
  - Open for additions - Closed for changes
    - Simple “Placeholder” module for e.g. LDOs, SMPS
    - Extendable with additional connections
  - Interface Segregation Principle
    - Only have absolutely needed functionality in a module
    - Use parameters for potentially changing values
- GIT integration for collaboration

SOLID design principles: https://en.wikipedia.org/wiki/SOLID

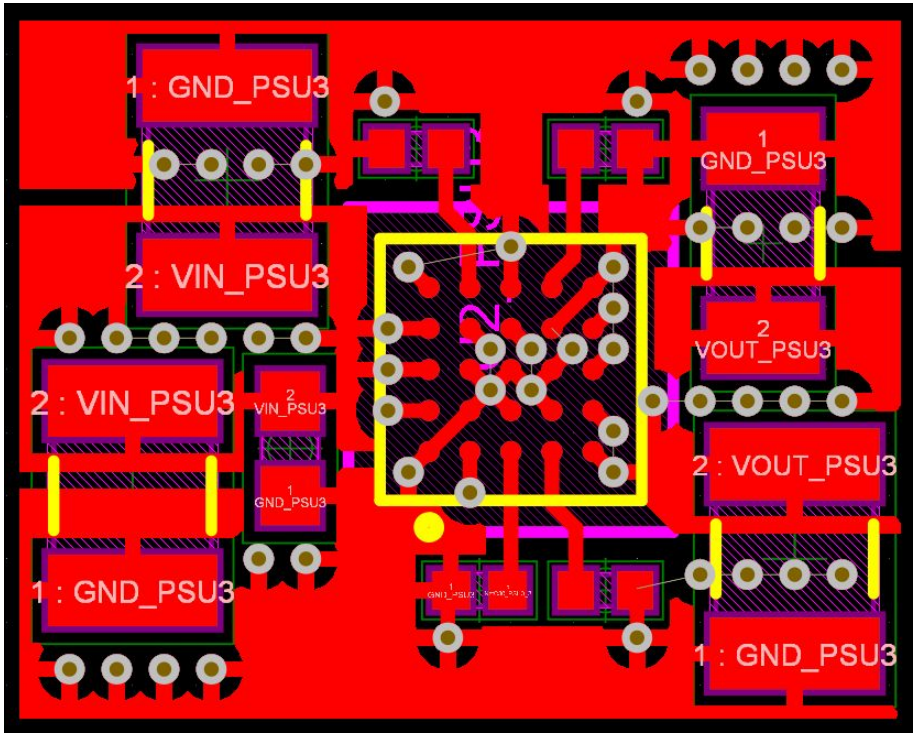
Guidelines for Modularity in Schematic Modules

- Copy recommended design diagram
- Parametrize your design
  - Through ports
  - Through parameters
- If possible, use 0402 passives
  - Larger resistors only for power
  - Larger caps are necessary



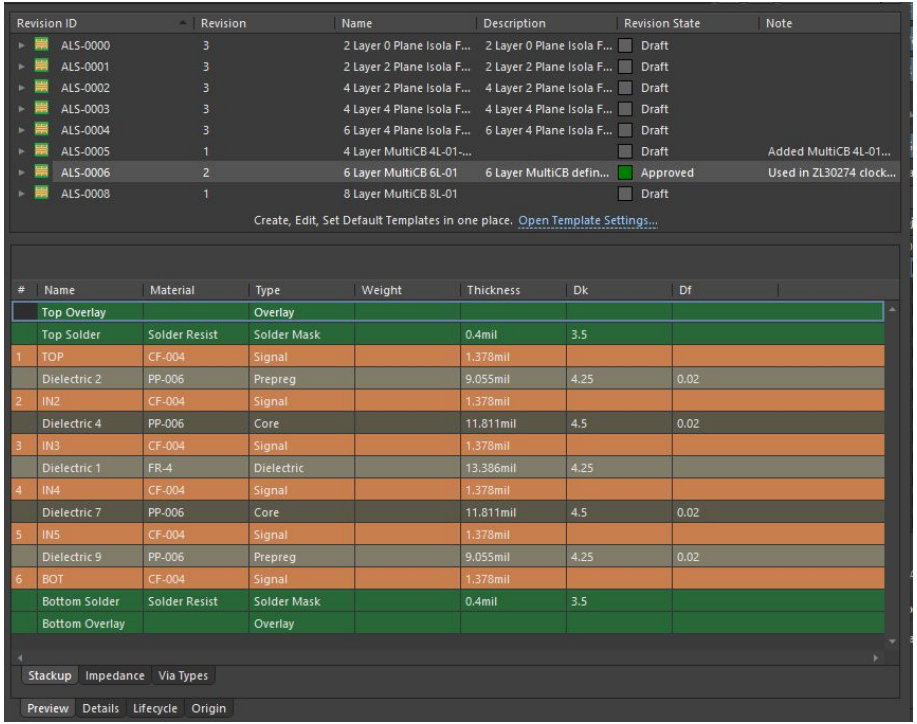
Guidelines for Modularity in PCB Modules

- *If possible*, use low-spec prerequisites
  - Low-spec design rules
  - All components on top
  - Al traces on top and bottom
  - No Via-In-Pad
  - Only through-hole vias
- Copy recommended layout
- Fanout signals and terminate with vias



PCB Production

- Use shared production information
  - Common PCB stack-ups
  - Common design rules
  - Predefined board outlines
- Automatic Gerber/ODB+ generation



Revision ID	Revision	Name	Description	Revision State	Note
ALS-0000	3	2 Layer 0 Plane Isola F...	2 Layer 0 Plane Isola F...	Draft	
ALS-0001	3	2 Layer 2 Plane Isola F...	2 Layer 2 Plane Isola F...	Draft	
ALS-0002	3	4 Layer 2 Plane Isola F...	4 Layer 2 Plane Isola F...	Draft	
ALS-0003	3	4 Layer 4 Plane Isola F...	4 Layer 4 Plane Isola F...	Draft	
ALS-0004	3	6 Layer 4 Plane Isola F...	6 Layer 4 Plane Isola F...	Draft	
ALS-0005	1	4 Layer MultiCB 4L-01-...		Draft	Added MultiCB 4L-01...
ALS-0006	2	6 Layer MultiCB 6L-01	6 Layer MultiCB defin...	Approved	Used in ZL30274 clock...
ALS-0008	1	8 Layer MultiCB 8L-01		Draft	

Create, Edit, Set Default Templates in one place. Open Template Settings...

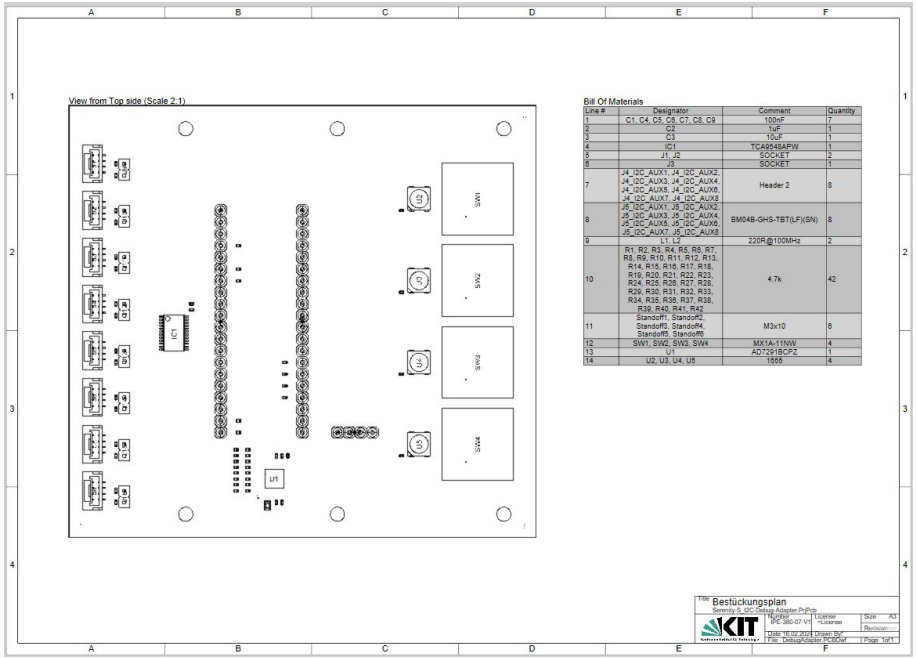
#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.4mil	3.5	
1	TOP	CF-004	Signal		1.378mil		
	Dielectric 2	PP-006	Prepreg		9.055mil	4.25	0.02
2	IN2	CF-004	Signal		1.378mil		
	Dielectric 4	PP-006	Core		11.811mil	4.5	0.02
3	IN3	CF-004	Signal		1.378mil		
	Dielectric 1	FR-4	Dielectric		13.386mil	4.25	
4	IN4	CF-004	Signal		1.378mil		
	Dielectric 7	PP-006	Core		11.811mil	4.5	0.02
5	IN5	CF-004	Signal		1.378mil		
	Dielectric 9	PP-006	Prepreg		9.055mil	4.25	0.02
6	BOT	CF-004	Signal		1.378mil		
	Bottom Solder	Solder Resist	Solder Mask		0.4mil	3.5	
	Bottom Overlay		Overlay				

Stackup | Impedance | Via Types

Preview | Details | Lifecycle | Origin

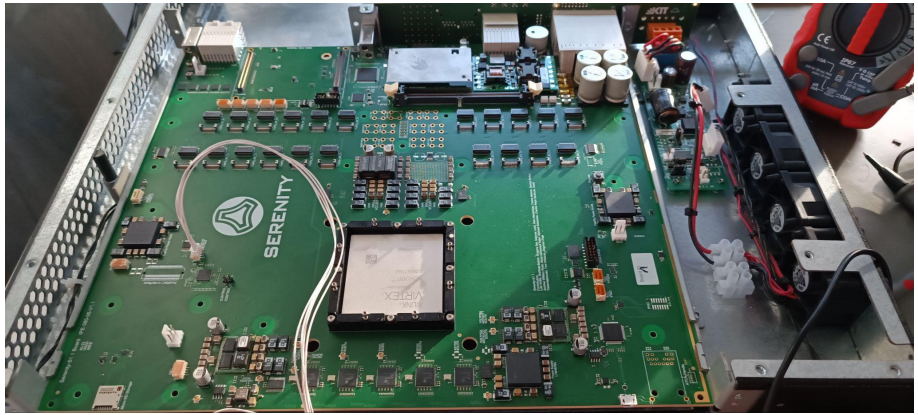
PCB Assembly

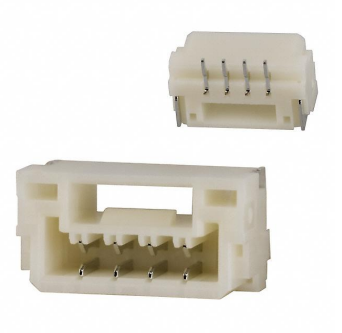
- Templates for assembly file generation
  - Draftsman for assembly plan
  - Pick and place files
  - Bill of materials
- *TODO*: scripts to convert Altium output directly to AVT-usable files
- *TODO*: central AVT parts database (as supplier for Octopart)

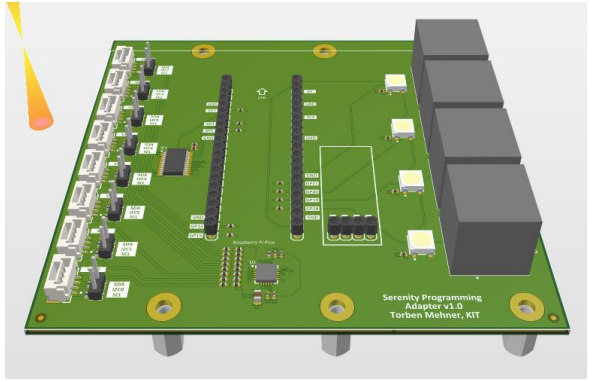


PCB Commissioning

- Bridging back to system design
  - If reused design is proven to work, no commissioning is necessary
  - Soft- and Firmware can be reused
- Using common PCB commissioning tools
  - Serenity Debug Adapter accesses I2C through very small SMT connectors

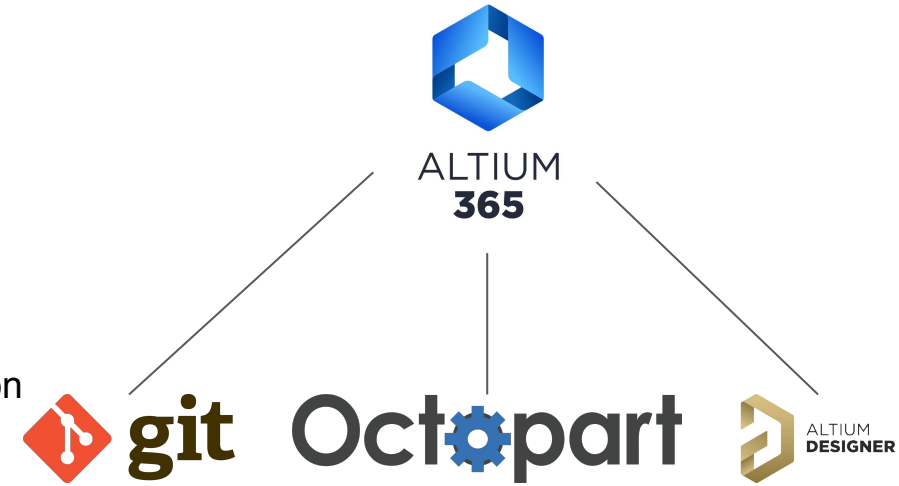






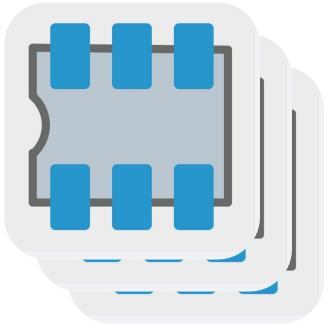
But why Altium?

- Full featured, widely used PCB software
- Altium 365 brings cloud integration
  - Ease of sharing design
  - Unprecedented reuse capabilities
  - Git integration, allows great collaboration
- Octopart integration
  - Better BOM management
  - Manufacturer part search

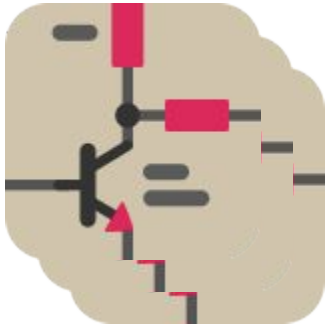


Current PCB Design Process

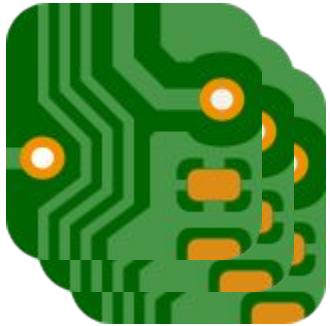
Create all symbols and footprints

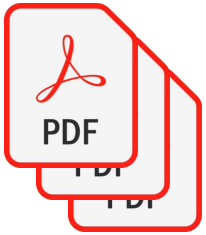


Draw all schematics

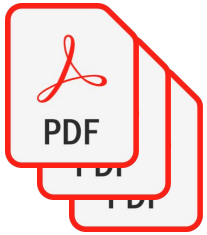


Layout PCB

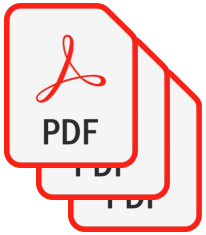




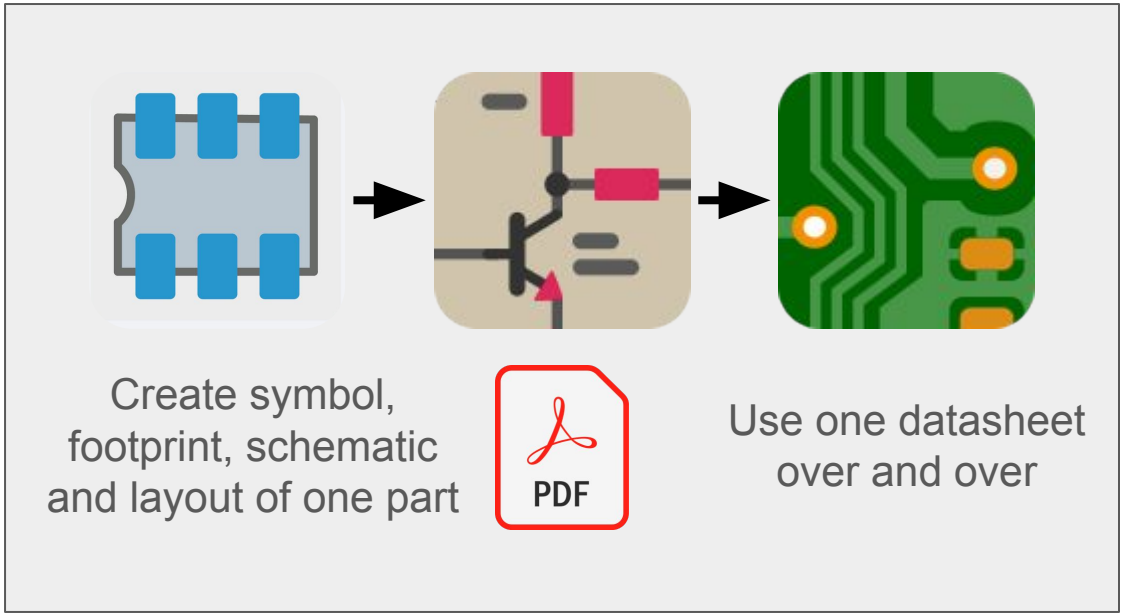
Forget datasheet



Forget datasheet



Proposed PCB Design Process



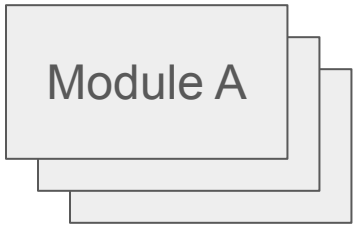
Module A

Proposed PCB Design Process

Design all modules

Connect modules in schematic

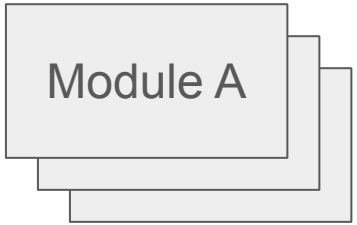
Connect modules in layout



Proposed *Reverse* PCB Design Process

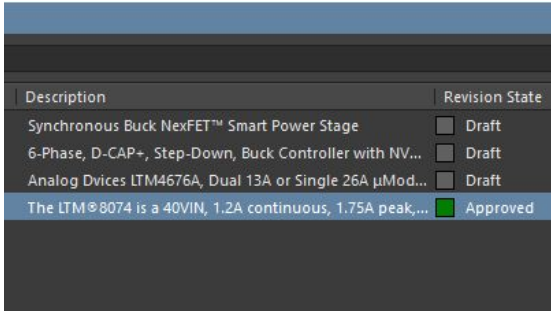
Finished & Tested PCB

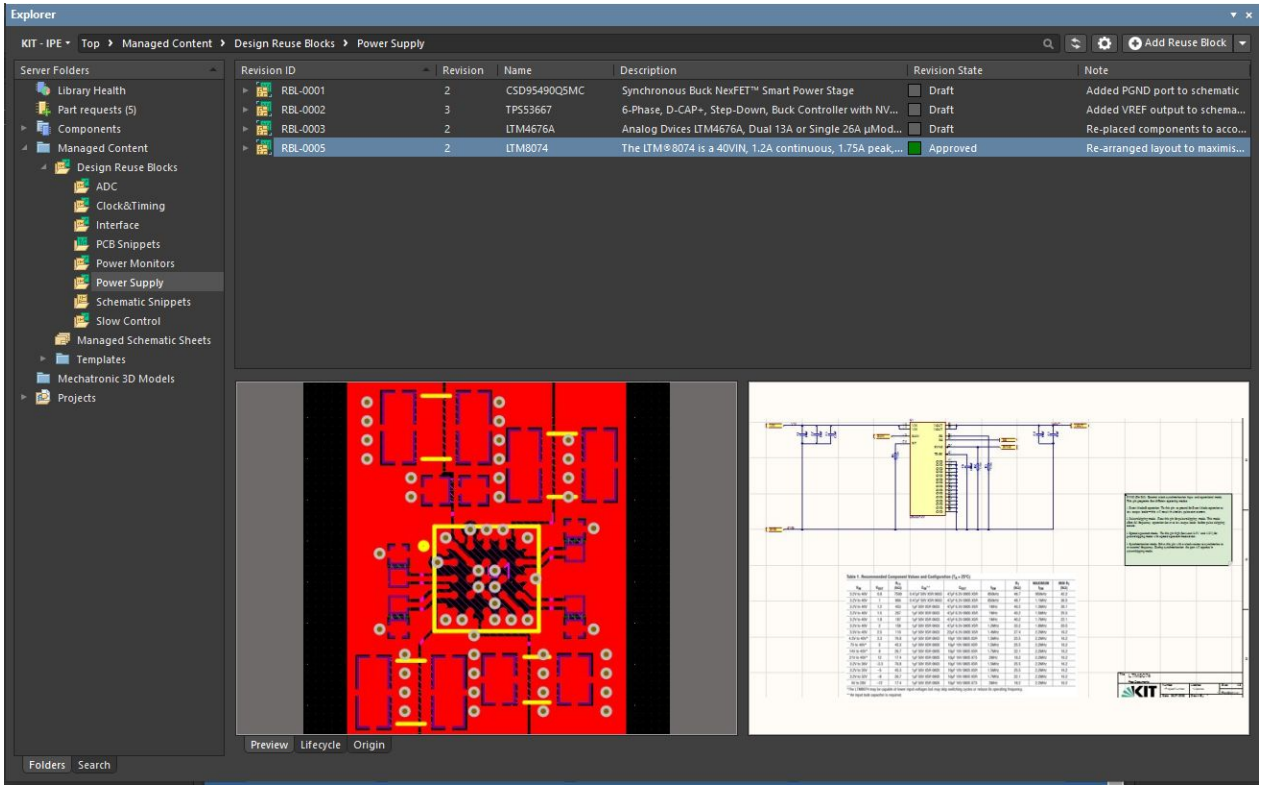
Extract working modules



Module Library

- Library to store modules
- Adaptable categories through sub-folders
- Revisions allow for improving modules
- Revision status shows if module is already tested

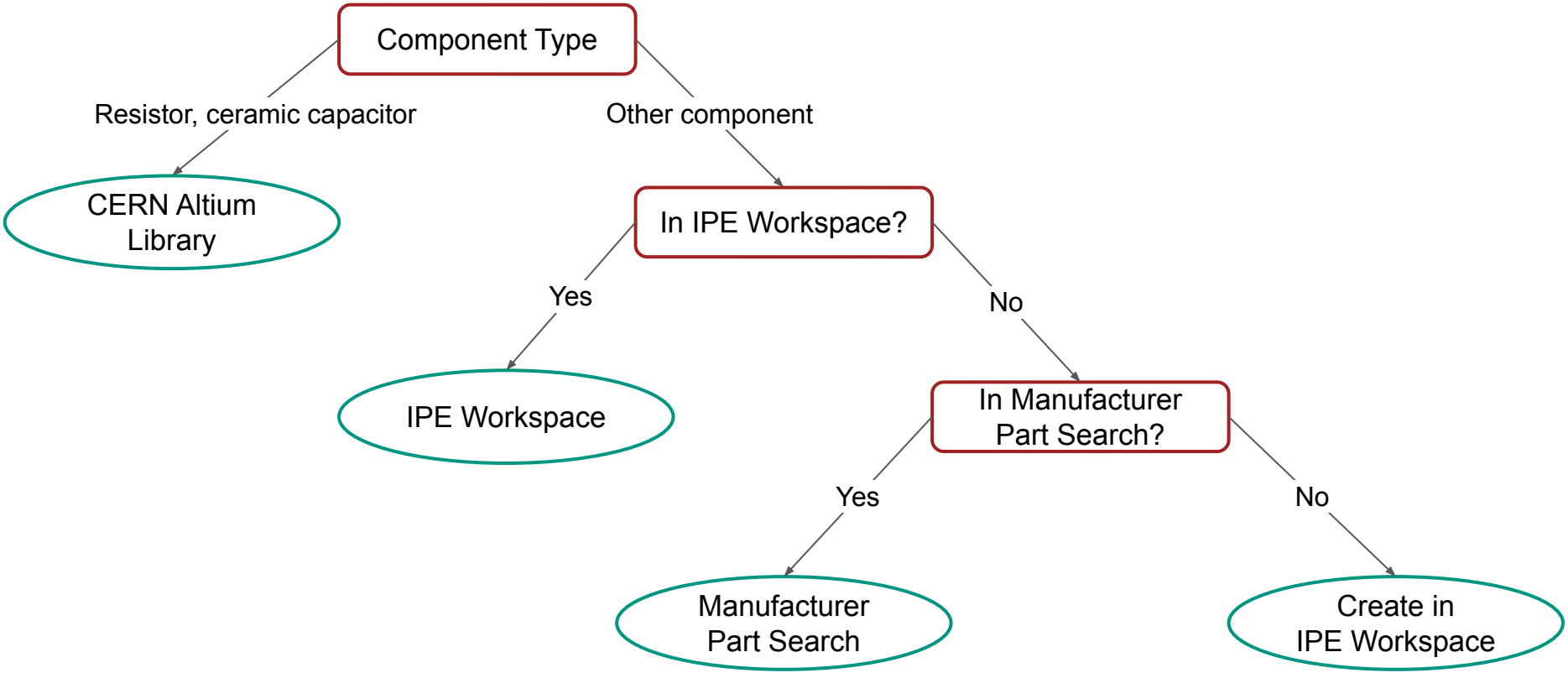




Where do I find…?

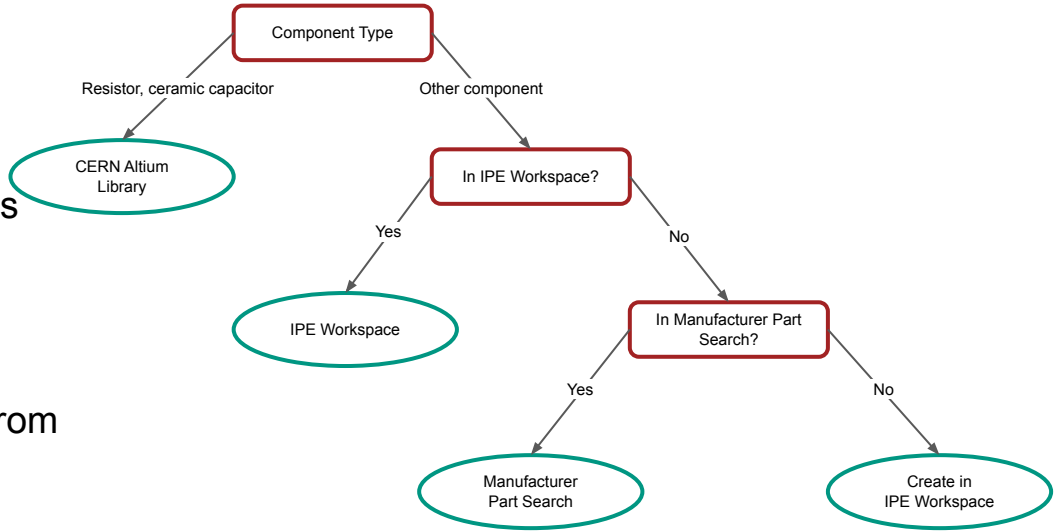
- Components
- PCB Modules
- KIT Project Template
- KIT Schematic Template
- PCB Layer Stacks
- Design rules

Where do I find components?



Where do I find components?

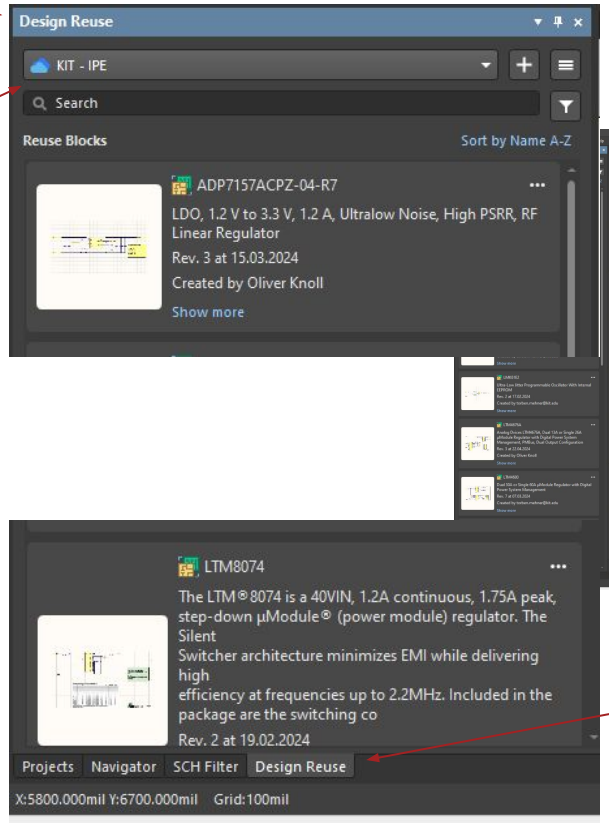
- CERN Altium Library
  - Good passive footprints
  - Use specific types for ceramic caps
  - Use “no-value”-types for resistors
- IPE Workspace
  - Components that were improved from manufacturer part search
  - Specific components
  - Video on how to create a component
- Manufacturer part search
  - Wide variety of available components

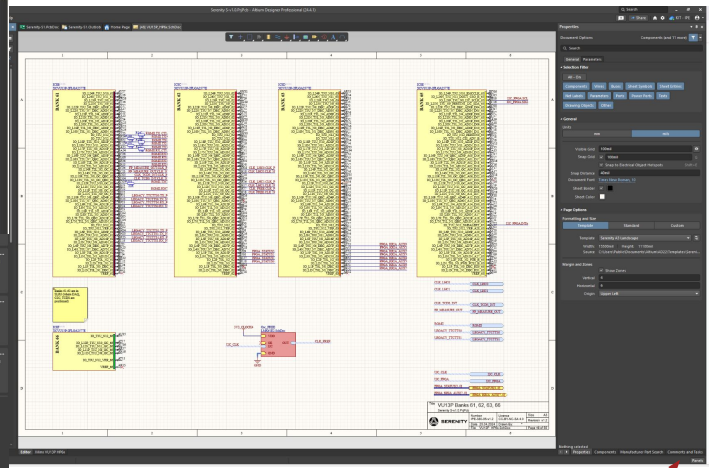


Where do I find PCB Modules?

Design reuse panel

KIT-IPE Workspace

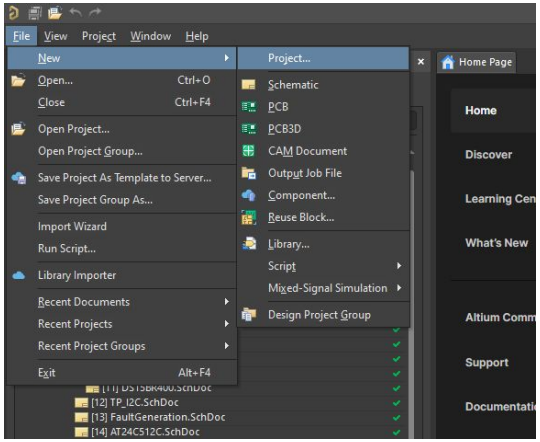


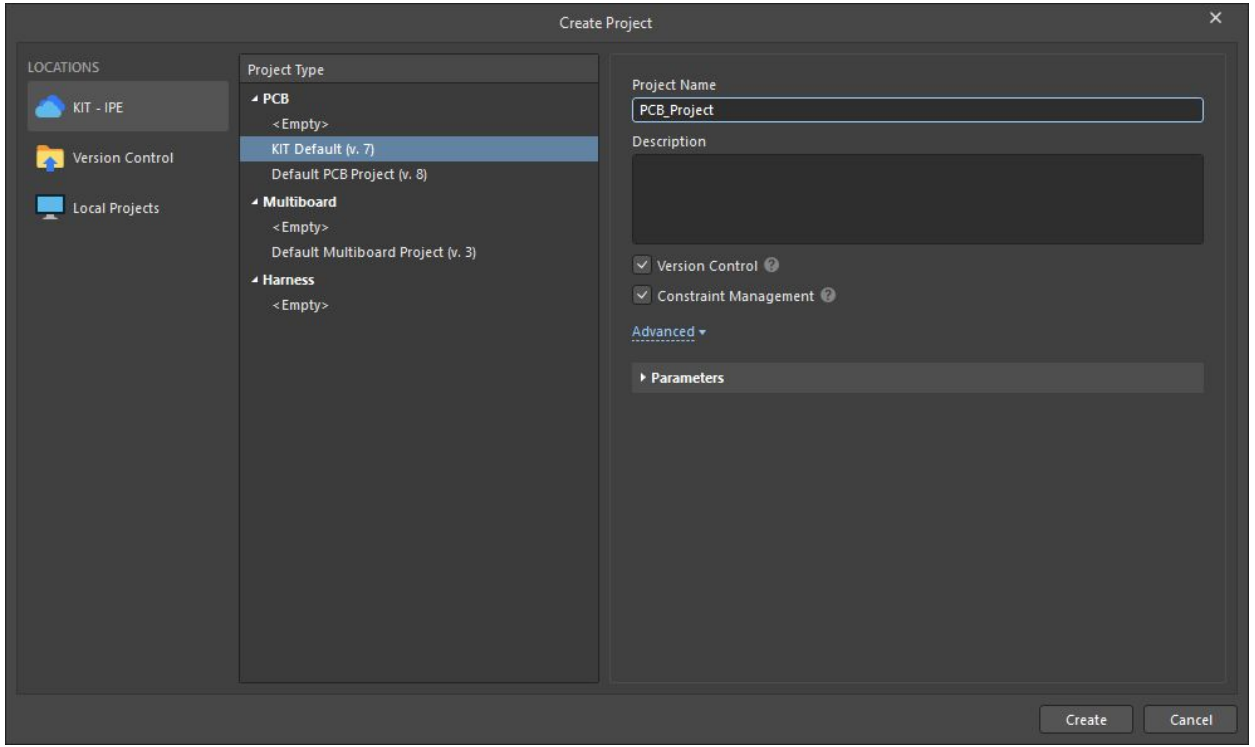


Design reuse panel

If not found, you must enable it in the bottom right

Where do I find the KIT Project Template?

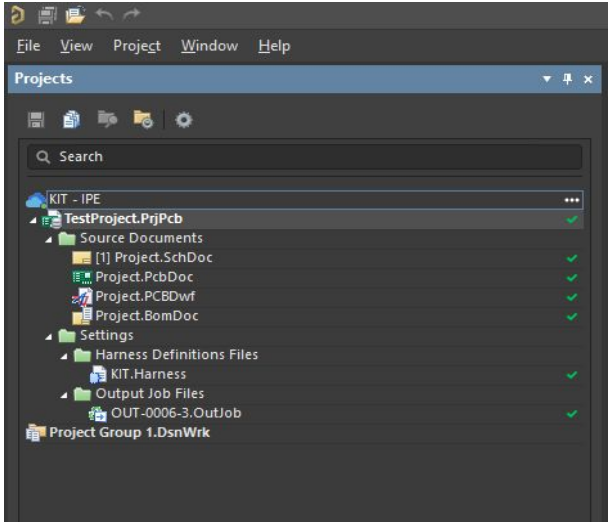


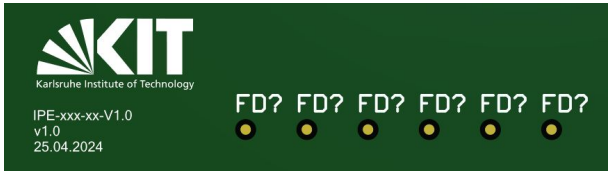


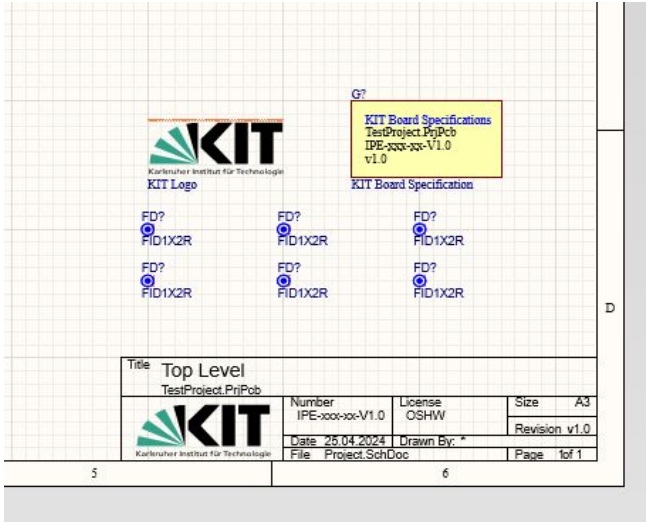
- New - Project…
- Select KIT Default

What does the KIT Project Template include?

- Schematic with
  - KIT Logo
  - Board specifications
  - 6 Fiducials
- Empty Layout
- Assembly Drawing
- Interactive BOM
- KIT Harness Definition (I2C, UART, JTAG, ETH, …)
- Output Job File

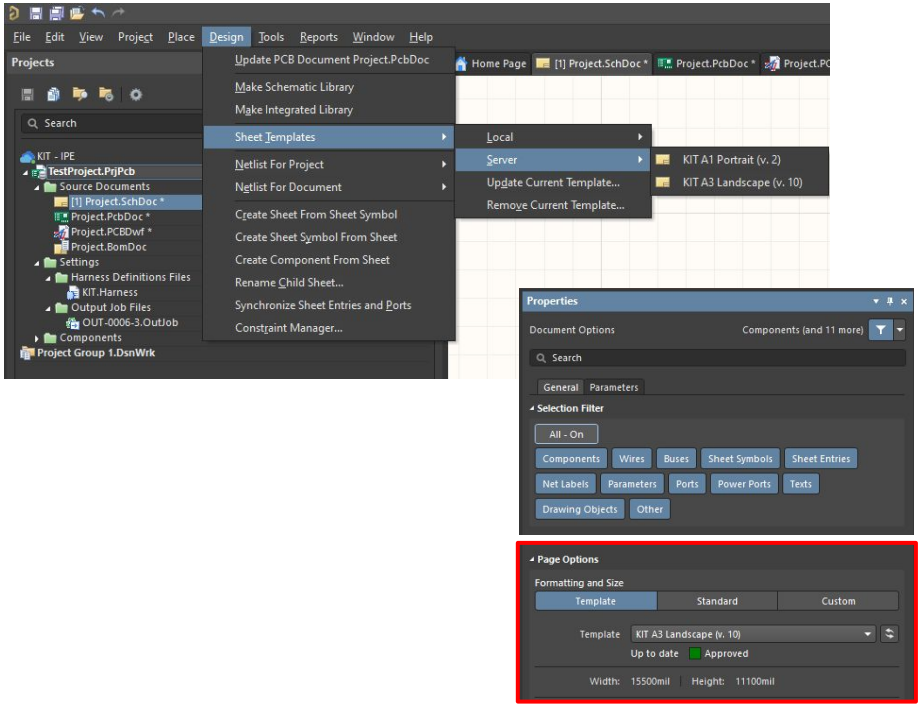






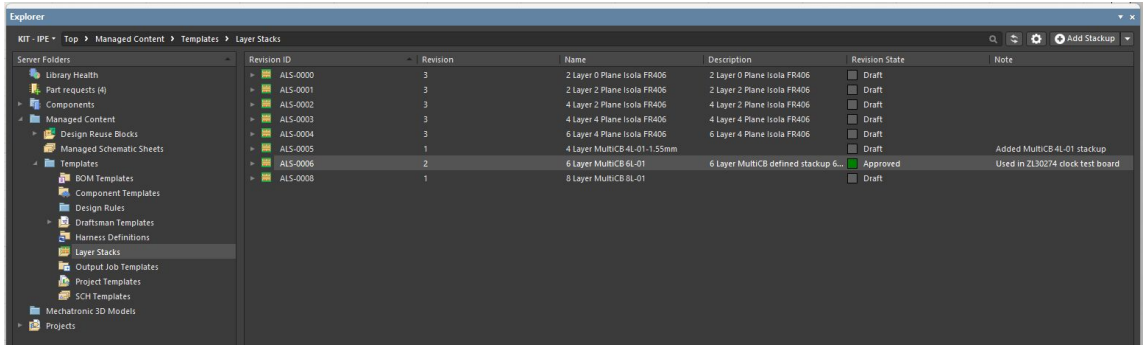
Where do I find the KIT Schematic Template?

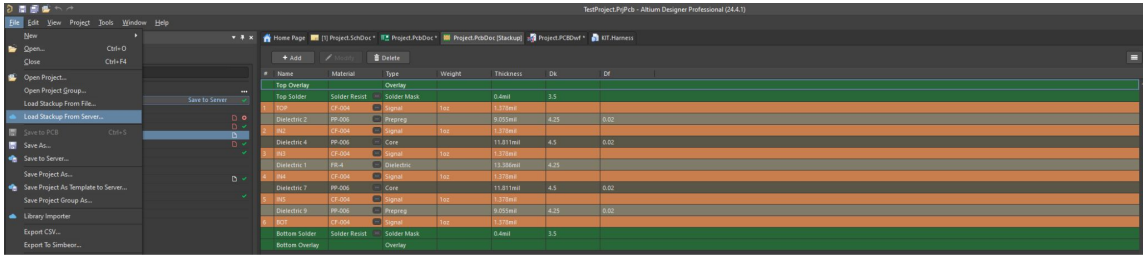
- Either in the properties panel
- Or Design - Sheet Templates - Server
- Project parameters
  - License
  - ProjectName
  - ProjectNumber
  - Project Revision
- Sheet parameters
  - Title
  - DrawnBy



Where do I find PCB Layer Stacks?

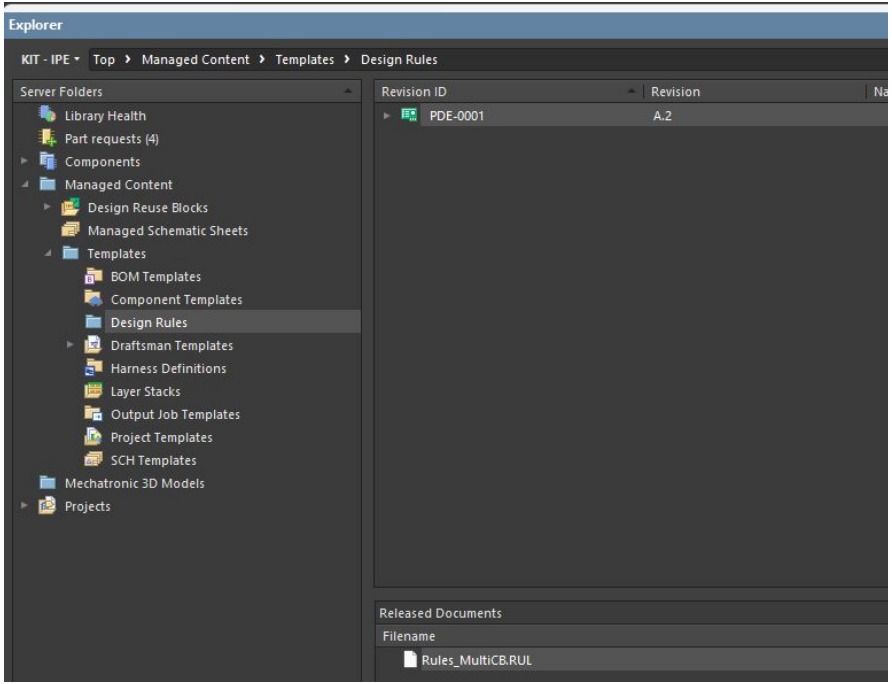
- Templates found in the “Explorer”-panel under “Managed Content - Templates”
- Loaded by opening the current PCB Stackup and then clicking “File - Load Stackup from Server”





Where do I find design rules?

- *Rule set for MultiCB is accessible in “Explorer Panel - Managed Content - Templates - Design Rules”*
  Deprecated since AD24
- Constraints management has changed in Altium Designer 24
- *TODO*: Update to Constraints Manager



Discussion

Karlsruher Institut für Technologie

Questions?