Quickstart OrCAD Capture

Version 22.1

Table of Contents

Introduction – In General

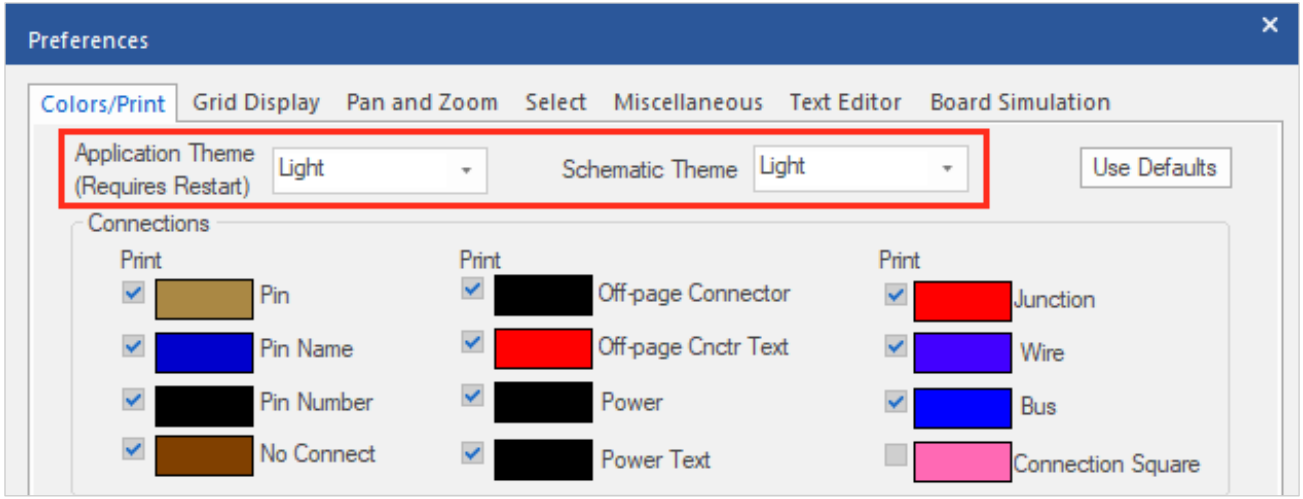
- This documentation is created for first time users of OrCAD Capture Software. It is neither a training handbook nor a complete user manual.
- Because of the compactness of this documentation, it is not possible to take up all available commands and their options. Please refer to the extensive documentation within the installation, which is available as HTML as well as PDF.
- Based on a simple schematic, we will describe most important steps of design flow. With minimum effort of training it will enable first time users to manage their first tasks by them self.
- After some preliminary information related to the software, the instructions including the circuit template will start on page 6.
- All commands and functions used in this tutorial are available and licensed with the trail version.
- Detailed training offers for the different tools can be found at https://www.flowcad.com/en/training.htm

Introduction – Color Scheme

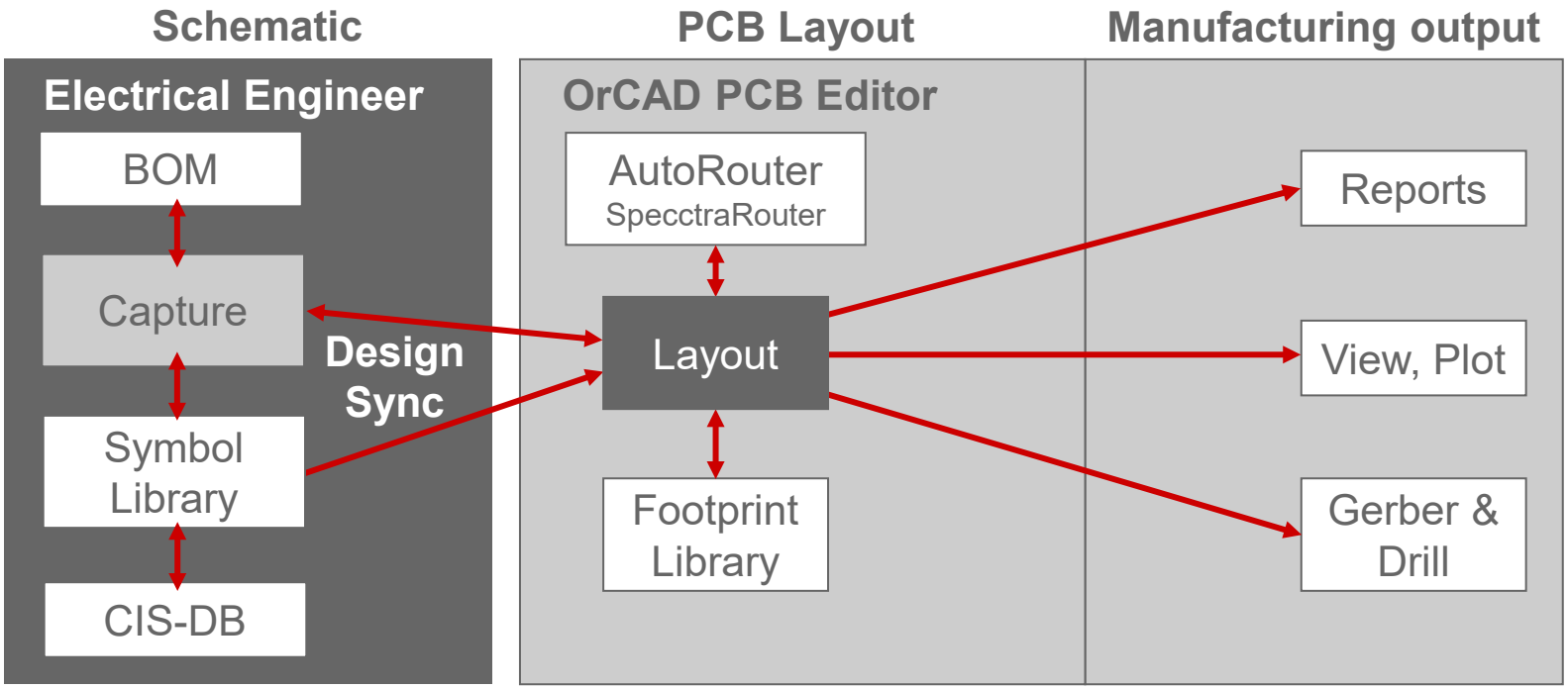
- Capture CIS as well as Schematic Canvas can be operated in both, a light and a dark theme.
- Icons will appear differently depending on which theme is selected.
- The color scheme can be switched under **Options > Preferences > Colors / Print**.
- This quick start shows light theme.

- Light Theme

- Dark Theme



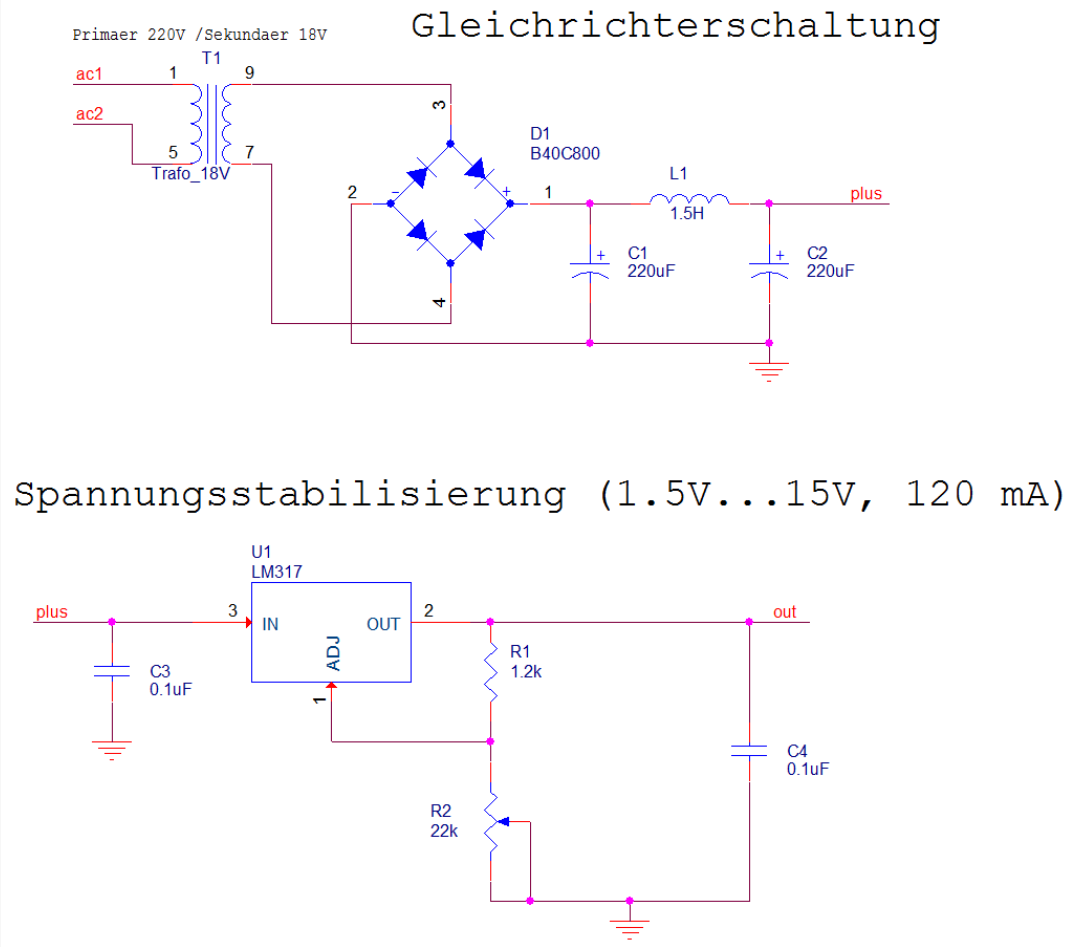
OrCAD PCB Design Flow



- As you can see, OrCAD PCB Designer Flow mainly consists of two parts.
  - These are the schematic entry module **Capture CIS** and the layout module **OrCAD PCB Editor**.
- Both modules are supplemented by additional sub packages who represent an ideal tool in each combination, enabling the user to complete all tasks with maximized efficiency.

Schematic Template

Here you can see circuit to be implemented in this quick start:



Schematic Template – Parts List

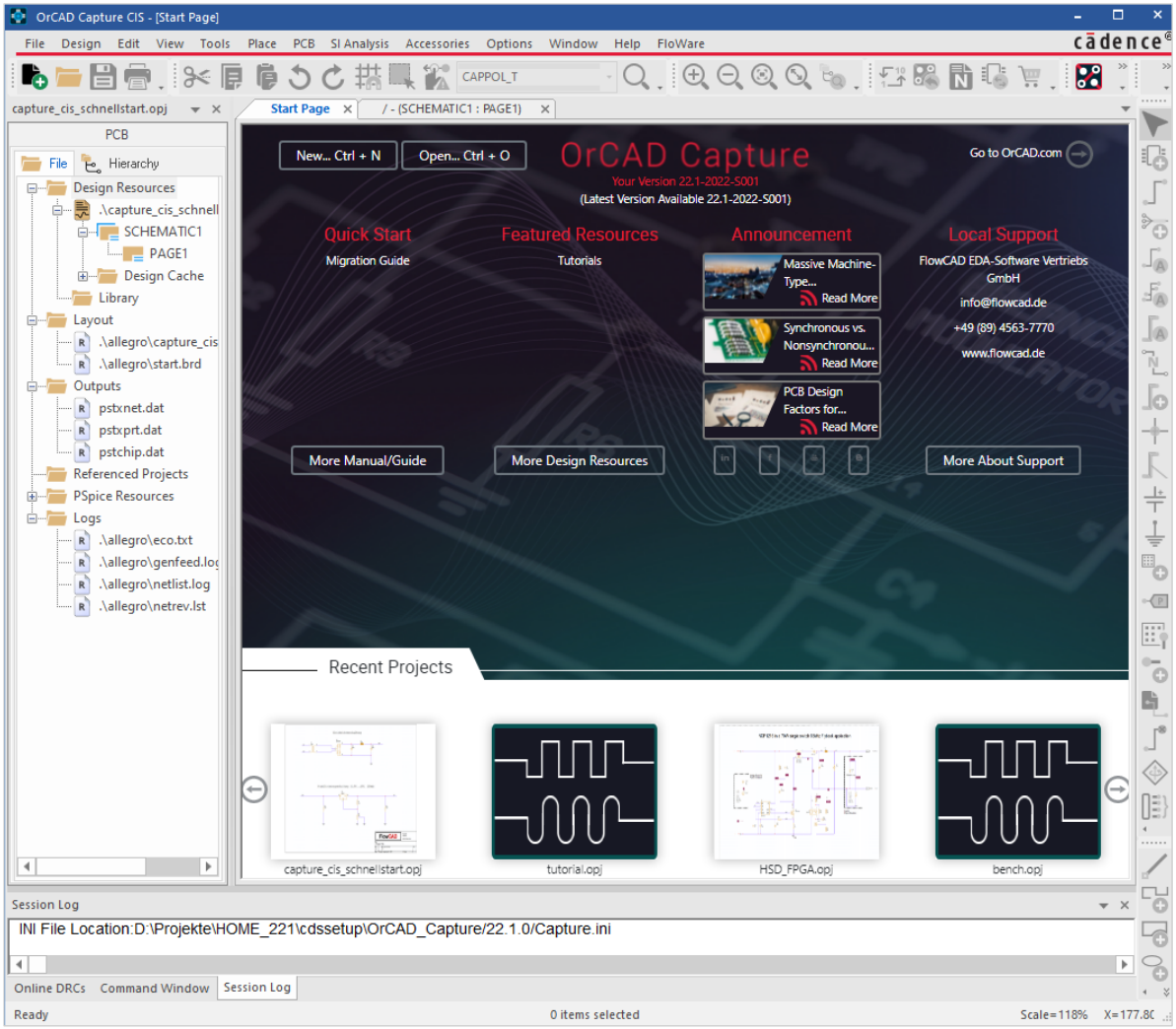
The circuit in this quick start contains following parts:

REFERENCE	VALUE	PART_NUMBER	PCB Footprint
C1	220u	FC-CAP-0057	capp_taj_e
C2	220u	FC-CAP-0057	capp_taj_e
C3	100n	FC-CAP-1019	cap_th_s_rm15x17_5x7x13_5
C4	100n	FC-CAP-1019	cap_th_s_rm15x17_5x7x13_5
L1	100n	FC-IND-0003	ind_0805
R1	1.2k	FC-RES-0356	res_th_a_rm10_16x2_54
R2	22k	FC-RES-0001	res_3269w
T1	EI30	FC-TF-0001	ei30_1579
U1	B40C800	FC-REC-0001	rec_b40_tht
U2	LM317	FC-IC-0001	sot223

Starting Capture

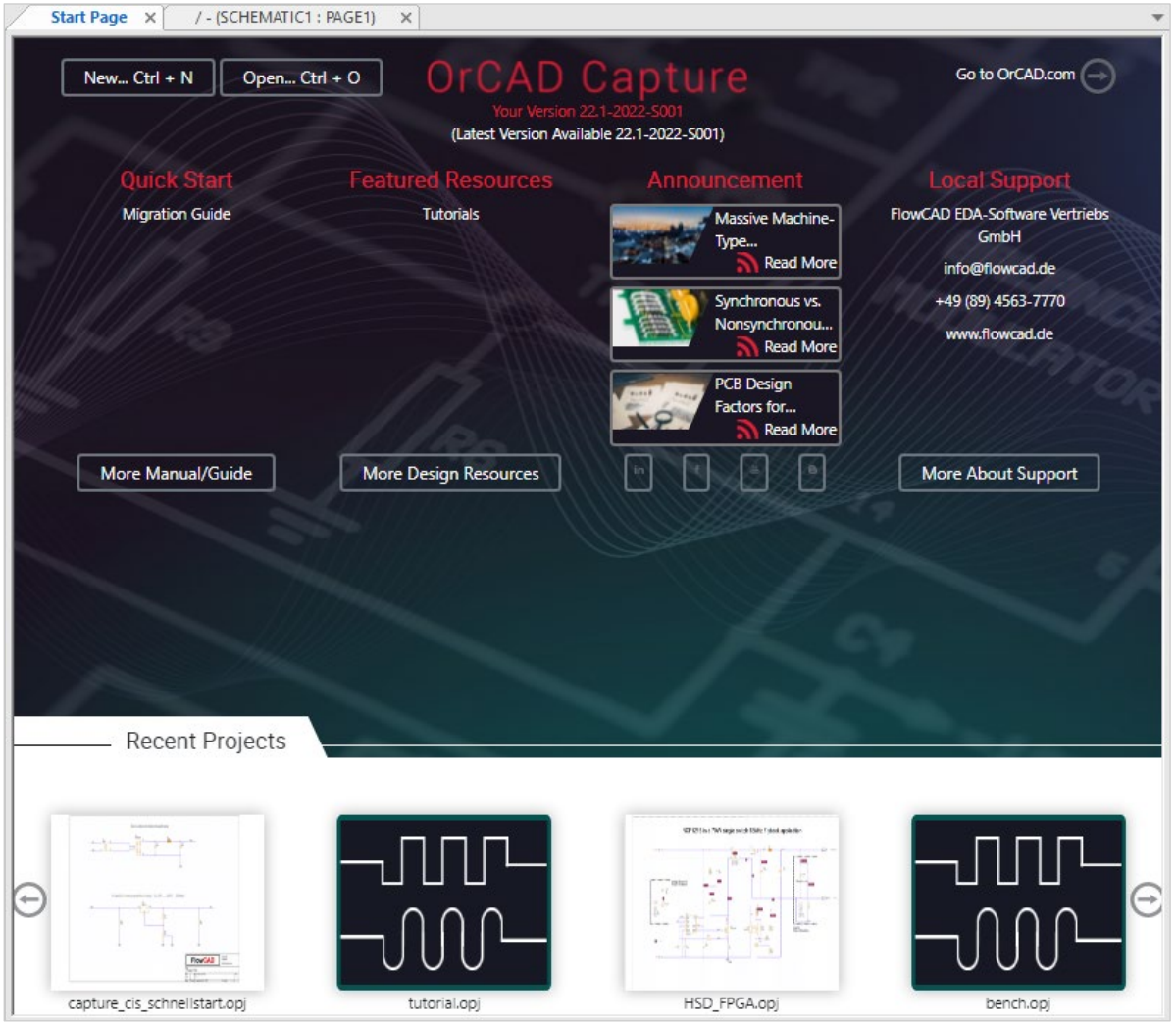
When starting Capture, the **Session Frame** window will open.

Start via:
Start >
Cadence PCB 22.1>
Capture CIS 22.1



Start Page

During start, the interactive start page gets loaded to open existing or to create new projects.
Furthermore, you receive additional information such as application notes, training data, etc.



New Project Setup

Lab: New Project Creation

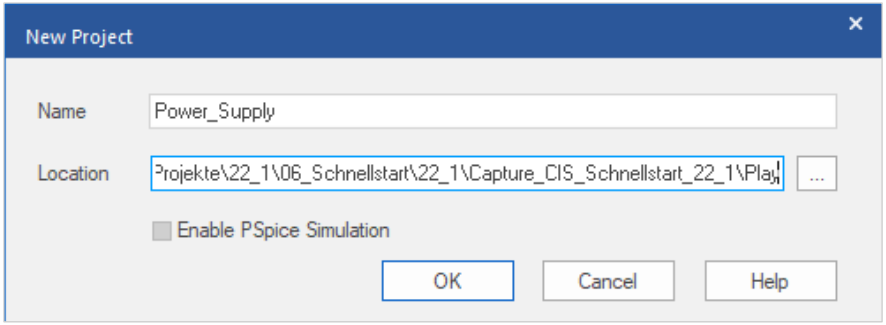
File > New > Project… creates a new project, in which the design will be stored.

In the top field under **Name**, please enter the name of your project, i.e. **Power_Supply**.

In the bottom field **Location,** please choose the folder in which the new project will be stored.

It is recommended to create a folder for each individual project.

Since we are not planning to simulate, keep **Enable project simulation** inactive.



Project Power_supply

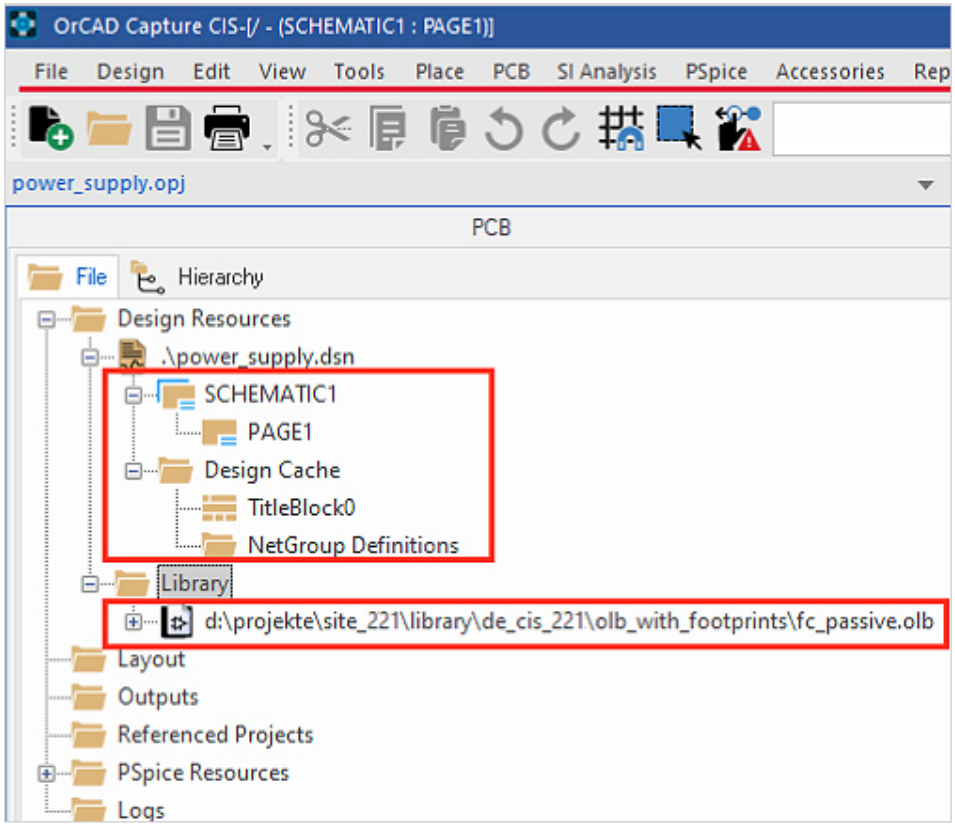
A new project with name **Power_supply** and a design with same name **power_supply.dsn** is created.

First page of your design with the name **PAGE1** is opened at the same time.

Please note the folder structure in project manager on the left side of the picture.

The folder structure is virtual, meaning it does only exist inside the project manager. **PAGE1** under **schematic** can only be found within **power_supply.dsn** file.

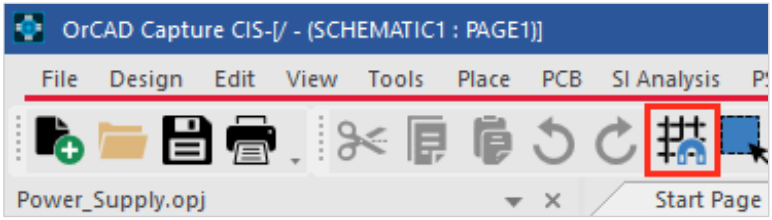
In contrast, libraries listed below are physical files under displayed paths located.

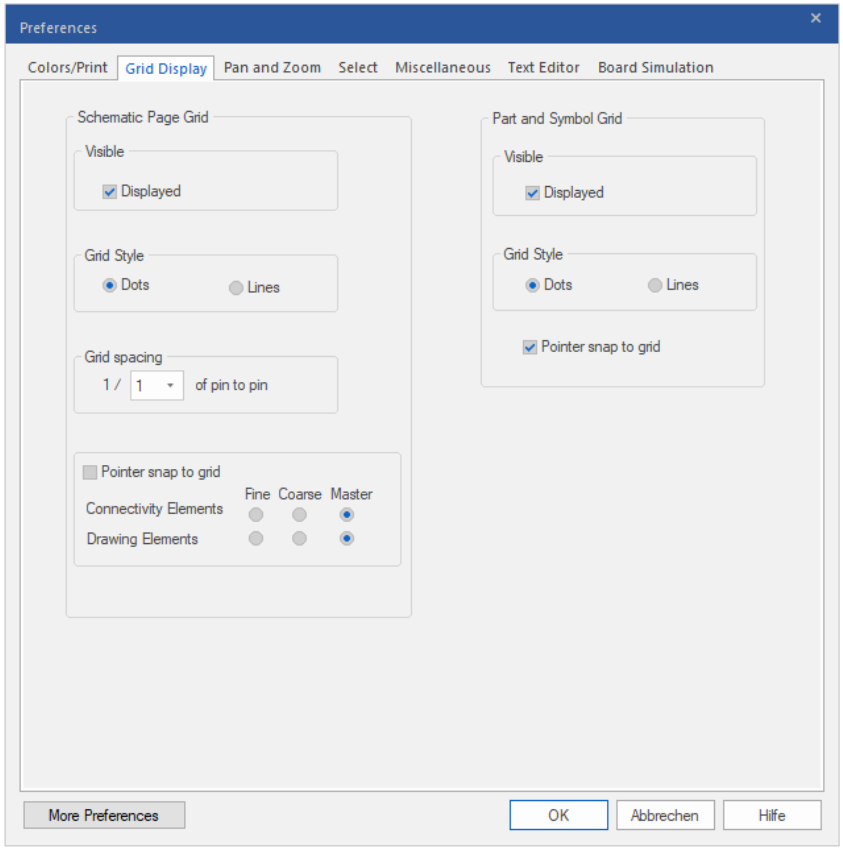


Setup – Grid Display

In menu item **Options > Preferences,** please set **Grid Display** to 1/1. This ensures that symbol pins are on grid and can be connected easily.

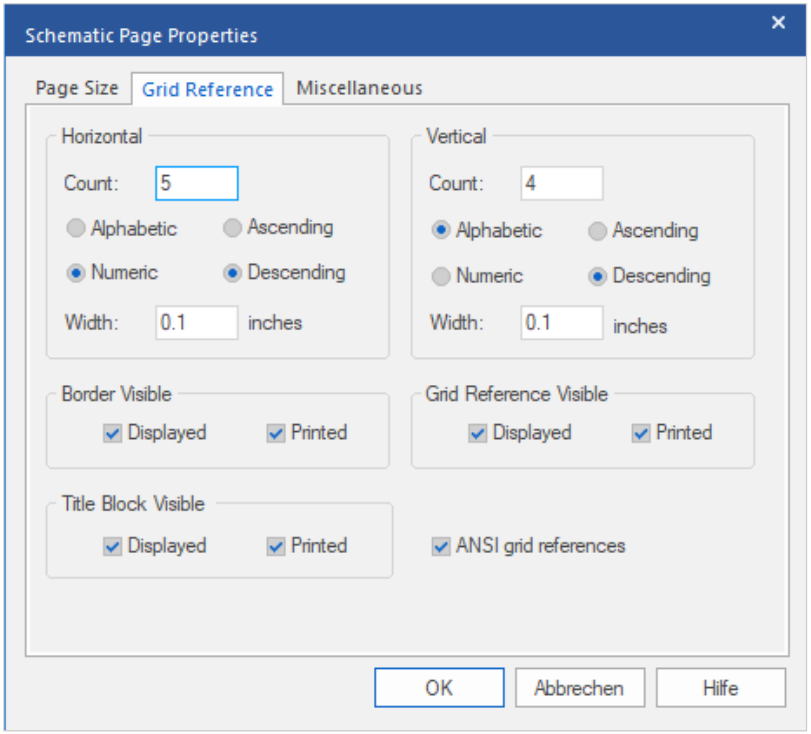
To avoid placing parts off grid, please activate **Snap to Grid**:





Setup – Drawing Frames

Appearance of the drawing frames can be configured under **Options > Schematic Page Properties**. In **Grid References** you can define subdivision of the frame in sections and its width (Capture adjusts frame width to grid settings).



Tip

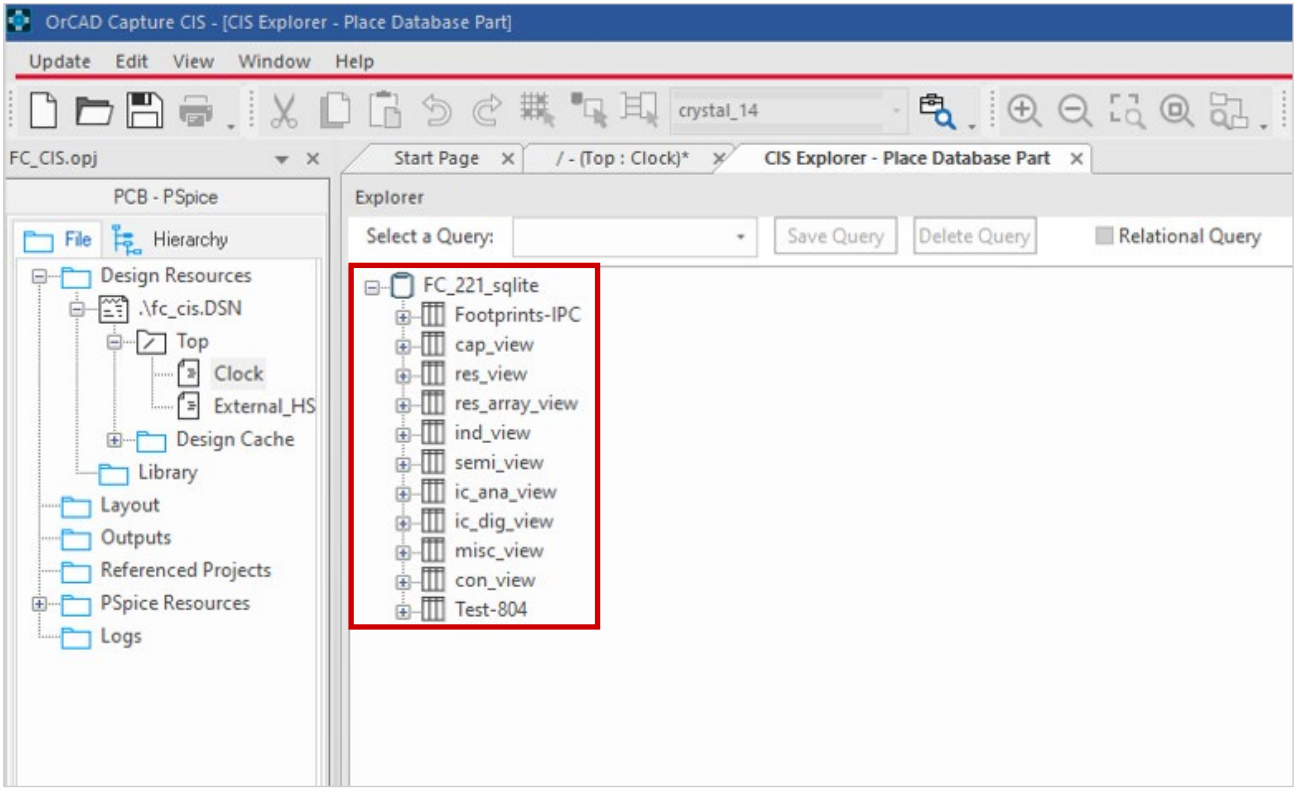
Settings can also be saved to a template that will affect all new pages and projects.
For this purpose, settings can be made under **Options > Design Template.**

More details can be found in chapter Settings and Templates starting on page 81.

Part Placement

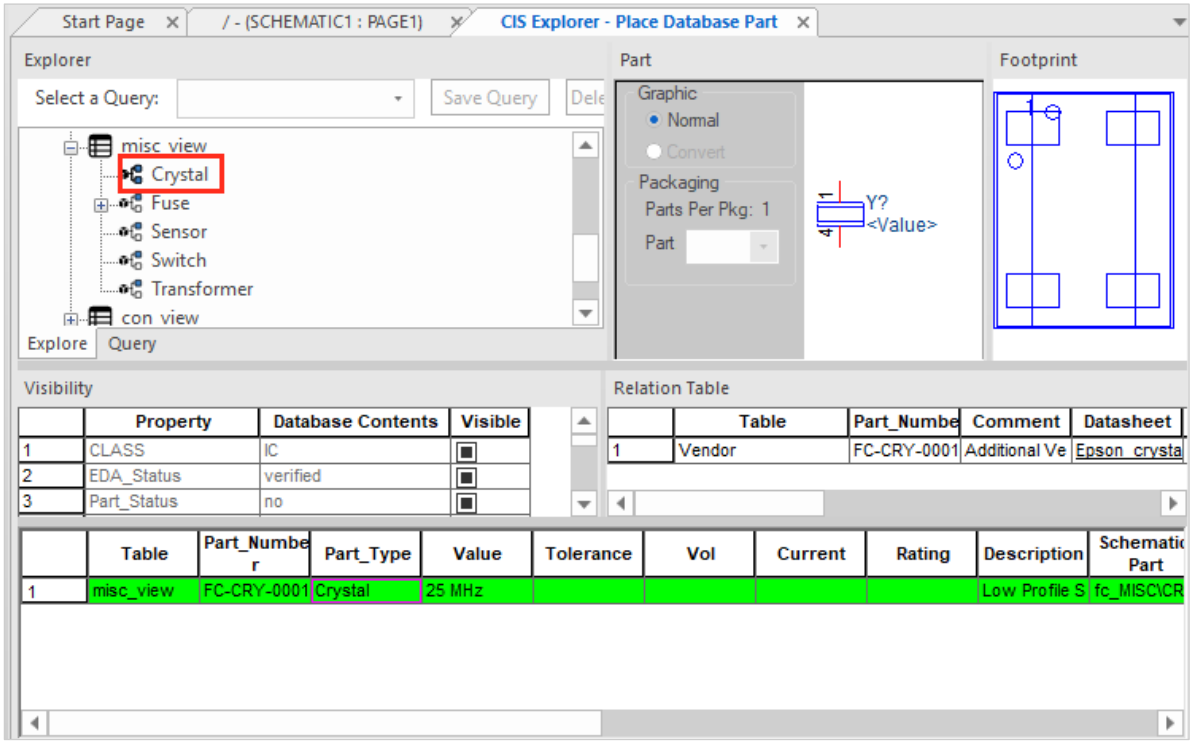
Part Search via Database (I)

- The CIS Explorer can be started via pull-down menu with **Place > Database Part** or via keyboard shortcut **Z**.
- Different database tables can be found in the upper left corner.



Part Search via Database (II)

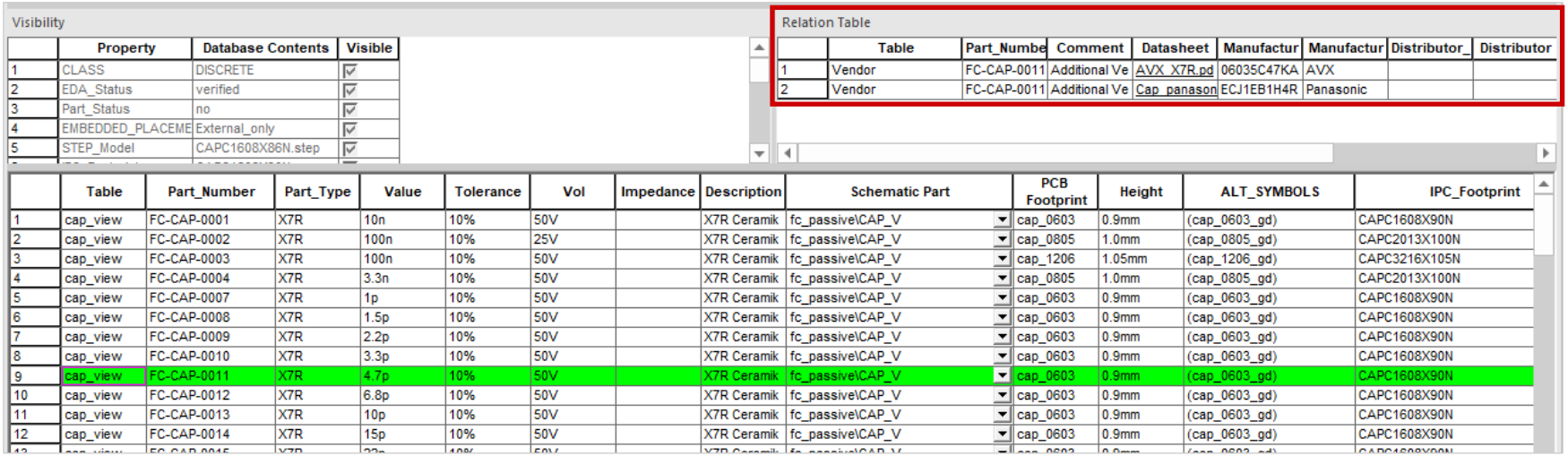
- Please open **misc_view** and select folder **Crystal**. A table entry appears in the lower window. If this is selected, the complete preview of the part is opened (arrangement of windows can be customized). The entry under Datasheet is a link to the component data sheet. Double-clicking the green highlighted entry causes the part to hang on the mouse and can be placed on schematic page.



Tip
With keyboard shortcut **Z**, CIS Explorer comes to foreground again.

Part Search via Database (III)

- Please open **cap_view** and select **X7R**. Several entries appear in lower window.
- Double-clicking on a column header sorts the list according to this value.
- If you select the capacitor with **4.7pF** in **0603** package, two data sets with manufacturer information linked to the part appear in manufacturer table.
- Graphical representation in schematic can be selected in column **Schematic Part.**



Visibility

	Property	Database Contents	Visible
1	CLASS	DISCRETE	☑
2	EDA_Status	verified	☑
3	Part_Status	no	☑
4	EMBEDDED_PLACEME	External_only	☑
5	STEP_Model	CAPC1608X86N.step	☑

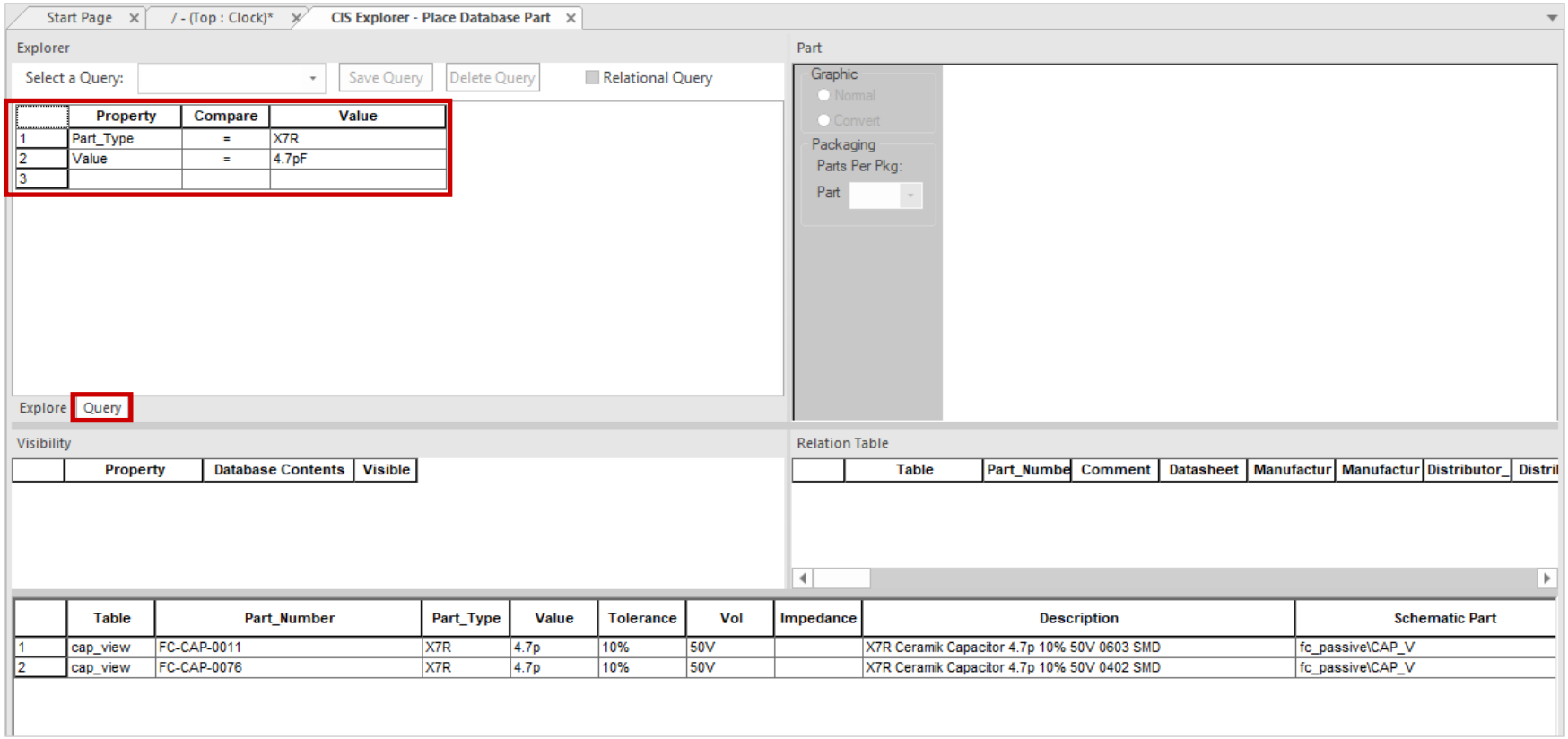
Relation Table

	Table	Part_Numbe	Comment	Datasheet	Manufactur	Manufactur	Distributor_	Distributor
1	Vendor	FC-CAP-0011	Additional Ve	AVX_X7R.pd	06035C47KA	AVX		
2	Vendor	FC-CAP-0011	Additional Ve	Cap_panason	ECJ1EB1H4R	Panasonic		

	Table	Part_Number	Part_Type	Value	Tolerance	Vol	Impedance	Description	Schematic Part	PCB Footprint	Height	ALT_SYMBOLS	IPC_Footprint
1	cap_view	FC-CAP-0001	X7R	10n	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
2	cap_view	FC-CAP-0002	X7R	100n	10%	25V		X7R Ceramik	fc_passive\CAP_V	cap_0805	1.0mm	(cap_0805_gd)	CAPC2013X100N
3	cap_view	FC-CAP-0003	X7R	100n	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_1206	1.05mm	(cap_1206_gd)	CAPC3216X105N
4	cap_view	FC-CAP-0004	X7R	3.3n	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0805	1.0mm	(cap_0805_gd)	CAPC2013X100N
5	cap_view	FC-CAP-0007	X7R	1p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
6	cap_view	FC-CAP-0008	X7R	1.5p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
7	cap_view	FC-CAP-0009	X7R	2.2p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
8	cap_view	FC-CAP-0010	X7R	3.3p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
9	cap_view	FC-CAP-0011	X7R	4.7p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
10	cap_view	FC-CAP-0012	X7R	6.8p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
11	cap_view	FC-CAP-0013	X7R	10p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
12	cap_view	FC-CAP-0014	X7R	15p	10%	50V		X7R Ceramik	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N

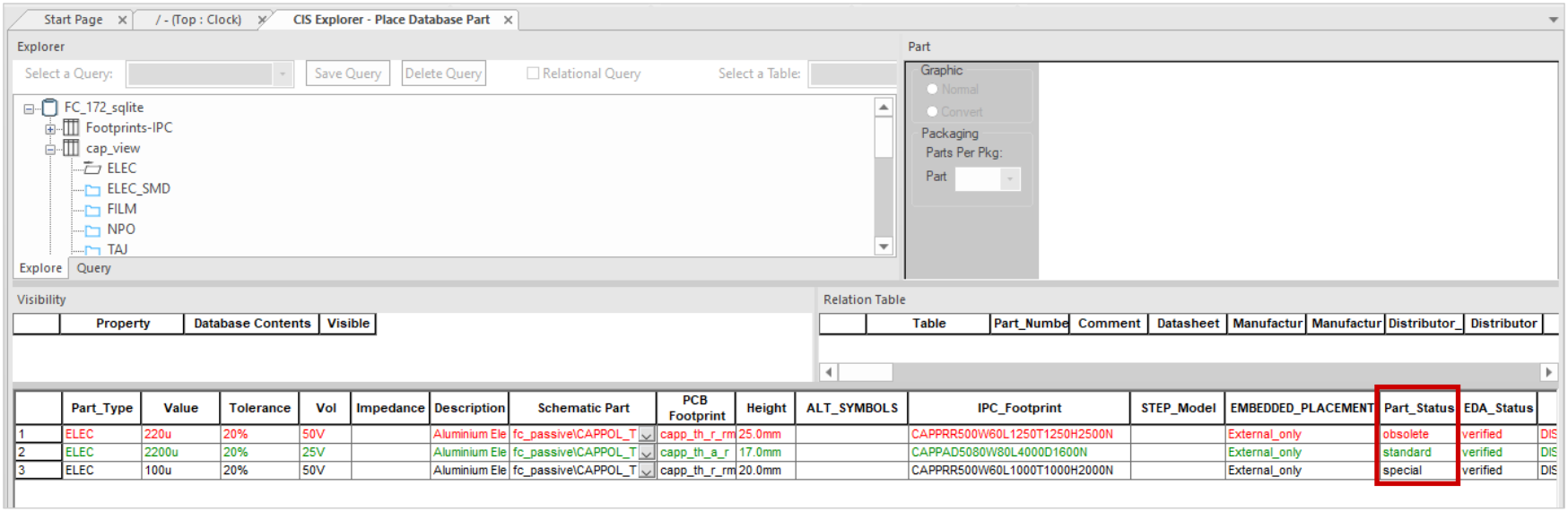
Part Search via Database (IV)

- The upper left window has two tabs. **Query** allows to search for parts using their values. For example, 4.7pF capacitor can also be found by entering values shown below. **Property** and **Compare** fields are equipped with drop-down menus that allow easy selection of search options.



Part Status

- If you go to **cap_view** category in CIS Explorer, you will see that listed parts are colored differently:
  - Standard parts are green
  - Obsolete parts are red
  - Special parts are black



Part Placement without Database

Part Placement from Library (I)

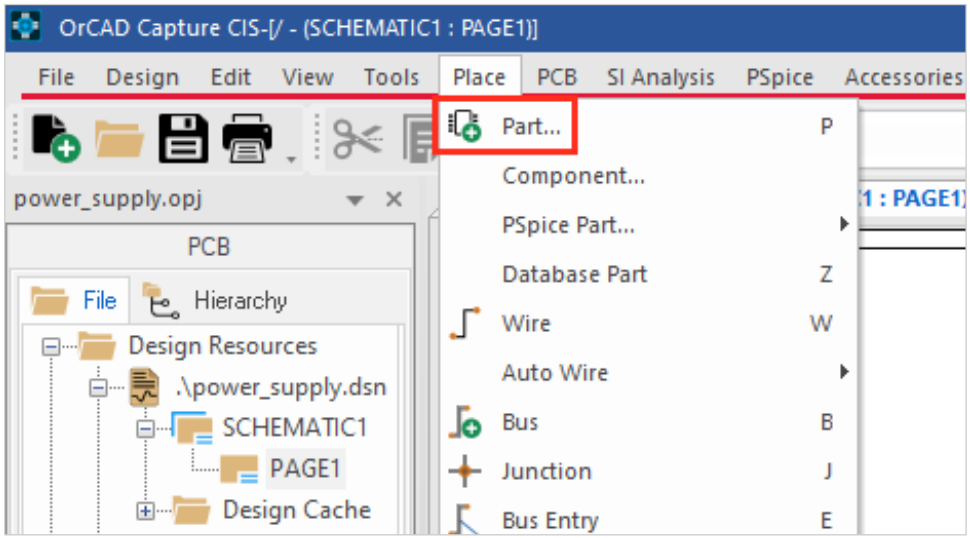
Besides the possibility to select parts from a database that provides metadata, OrCAD Capture also offers to select and place parts directly from a library (.olb).

- Advantage
  - No library database is needed.
- Disadvantage
  - All part information must be manually defined as properties (effort, error-prone).
  - Variant management is not possible.

Part Placement from Library (II)

You can use one of three methods to place new parts:

- **Place > Part...** via pull-down menu
- **P** (keyboard)
- **Place Icon** on right side 

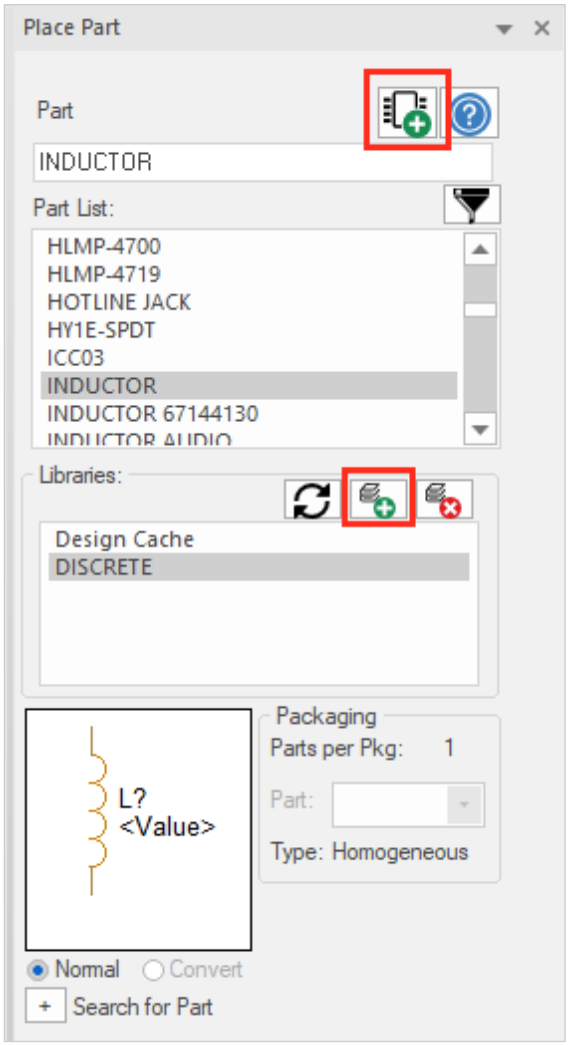


Tip

One of the schematic pages must be active to make icon panel visible (right edge of OrCAD Capture window).

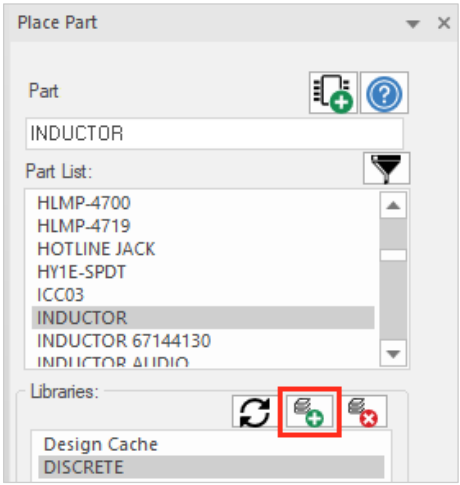
Part Placement from Library (III)

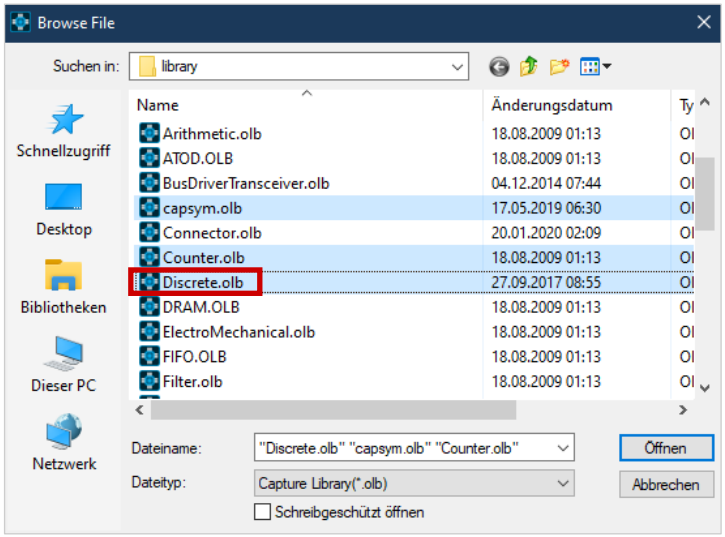
- Placement command opens the menu as shown.
- In **Libraries** you can select one or more libraries in which your part can be searched.
- Enter part name in **Part**. This already behaves as a filter, but without usage of wildcards “*”.
- Search results and related library are visible in **Part List**.
- **Search for Part** allows a search including wildcards.
- **Add Library**  allows to add libraries to search path.
- **Packaging** indicates if a part consists of multiple gates or slots like a resistor pack or an IC with multiple gates.
- With a **double click** in part list you get back into schematic and can place the part with LMB (left mouse button).



Library Assignments

- Choose the necessary libraries for the project.
- Select the desired library and press **Add** button to add the library to the project.
- How to define new parts that cannot be found in any existing libraries is described in chapter Libraries on page 62.





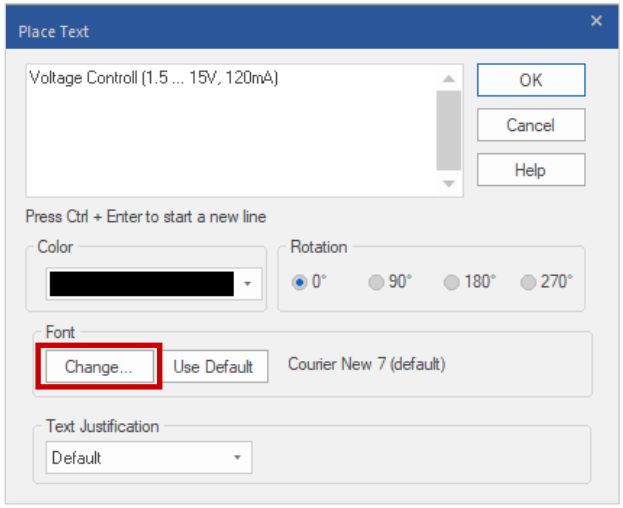
Adding / Edit Text

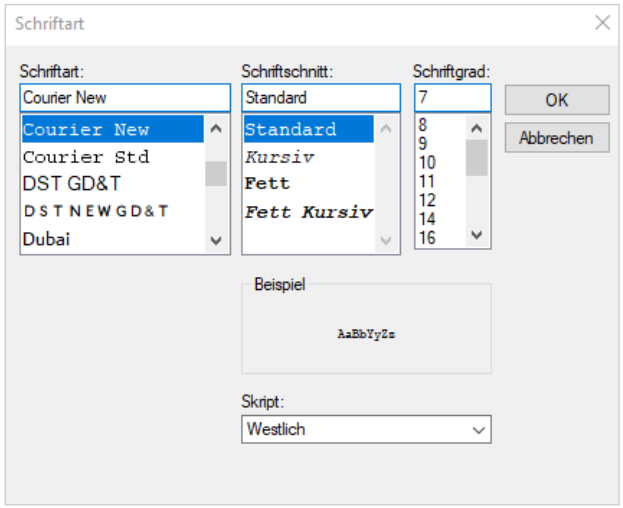
Text – Adding and Edit

Pure text notes can be added to the schematic with

- **Place > Text**
- **T** (keyboard)
- **Place Text Icon** 

Font and size can be modified during text placement with the **Change** command.





Connections

Connections (I) – Auto Wire

Two Points

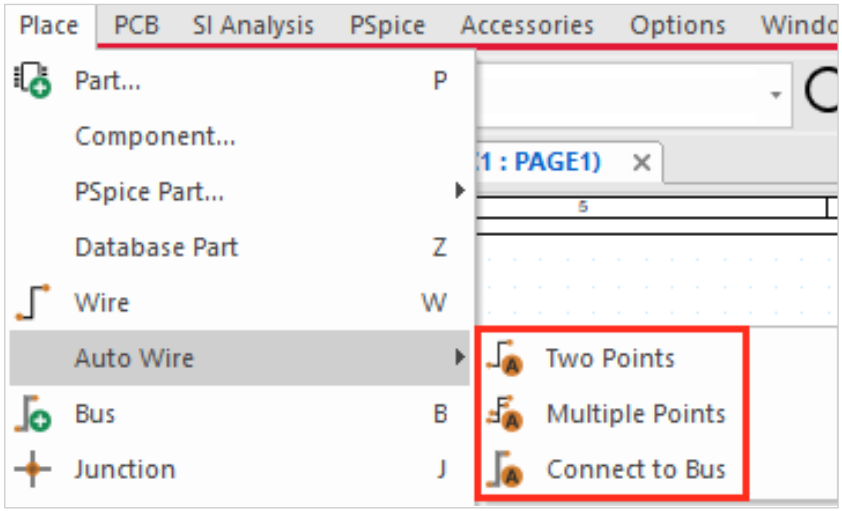
– Two selected pins are automatically connected to each other.

Multiple Points

– Multiple selected pins are automatically connected to each other.

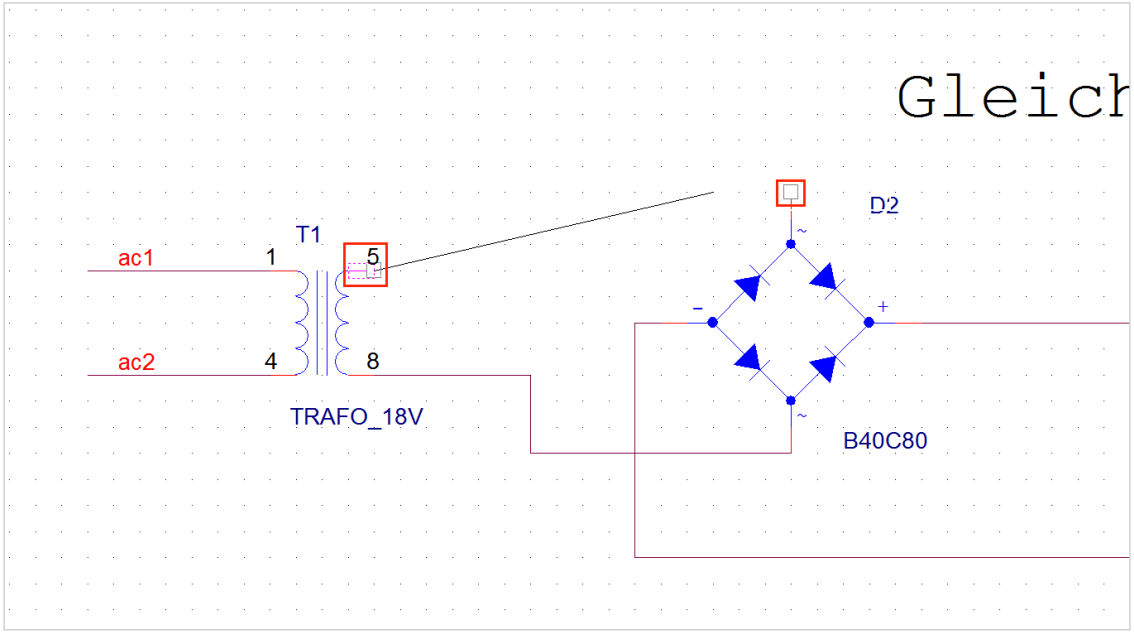
Connect to Bus

– Multiple selected lines are connected to a bus.



Connections (II) – Auto Wire

To use the function **Two Points,** you only need to select two pins or nets.
Connection get automatically wired.



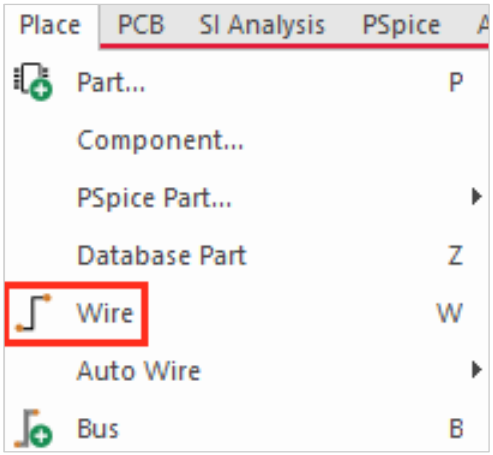
Connections (III) – Manually

Connection can also be done manually:

- **Place > Wire**
- **W** (keyboard)
- **Place Wire Icon** 

For manual connection, user must click for each direction change with left mouse button to fix a vertex of connection.

Otherwise this is identical to the Auto Wire command.



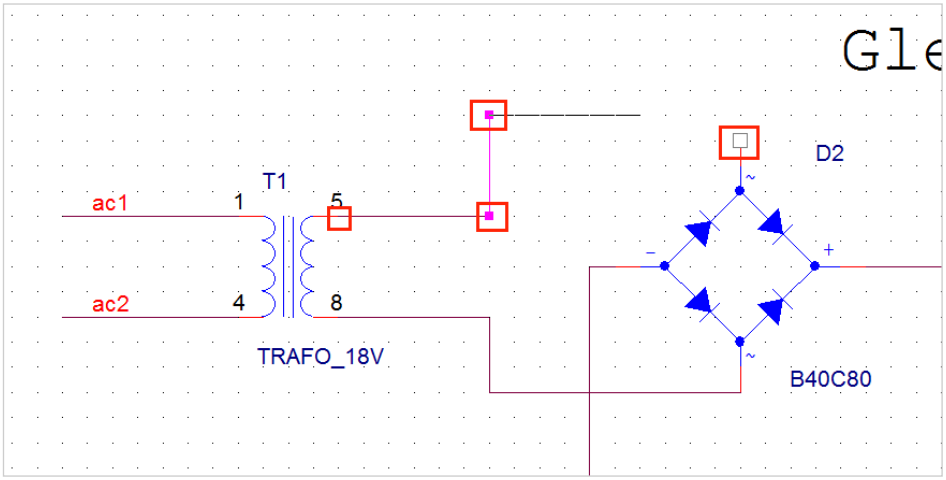
Connections (IV) – Procedure

Wiring is performed by a **click** on an open end, marked by a red square on a pin. This marker disappears after the connection is done.

Red dot signalizes a pin which is available for a connection. **Another click** on this object completes connection. Red dot and square will disappear. A **double click** will complete the task and release all objects.

Crossroads of connect lines only represent a connection between the two lines if a **dot** or a **junction** is placed or a t-junction is used as a interim step.

By simply selecting a connection and pressing **Del key,** an existing connection can be deleted again.



Tip

Ctrl key allows selection of multiple objects at same time.

Connections (V) – Net Names

Net name assignment

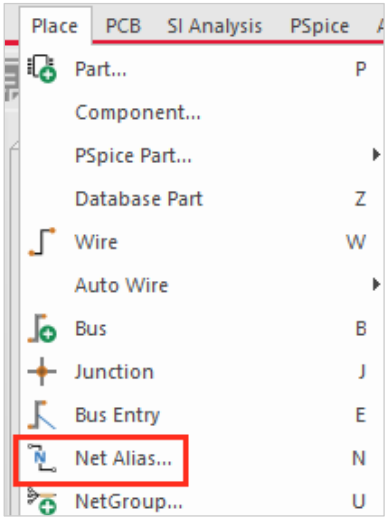
To connect parts on one schematic page, the **Place > Wire** command is used to perform wiring.

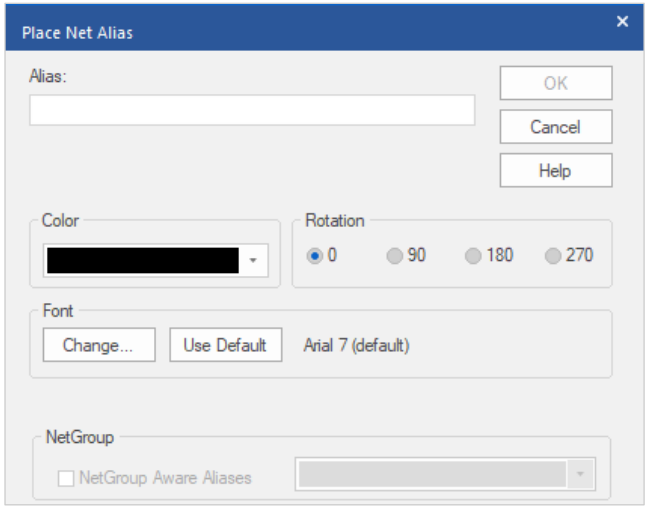
Another option is using **Net Aliases**.

Place > Net Alias...

Place Net Alias Icon 

A net name is assigned to nets and used to connect two parts. Net with name **plus** is used as an example here (see schematic on page 6).





Tip

To connect nets via net names across multiple sheets or even across designs, **Offpage Connectors** or **Port Connectors** are used. More details can be found in documentation.

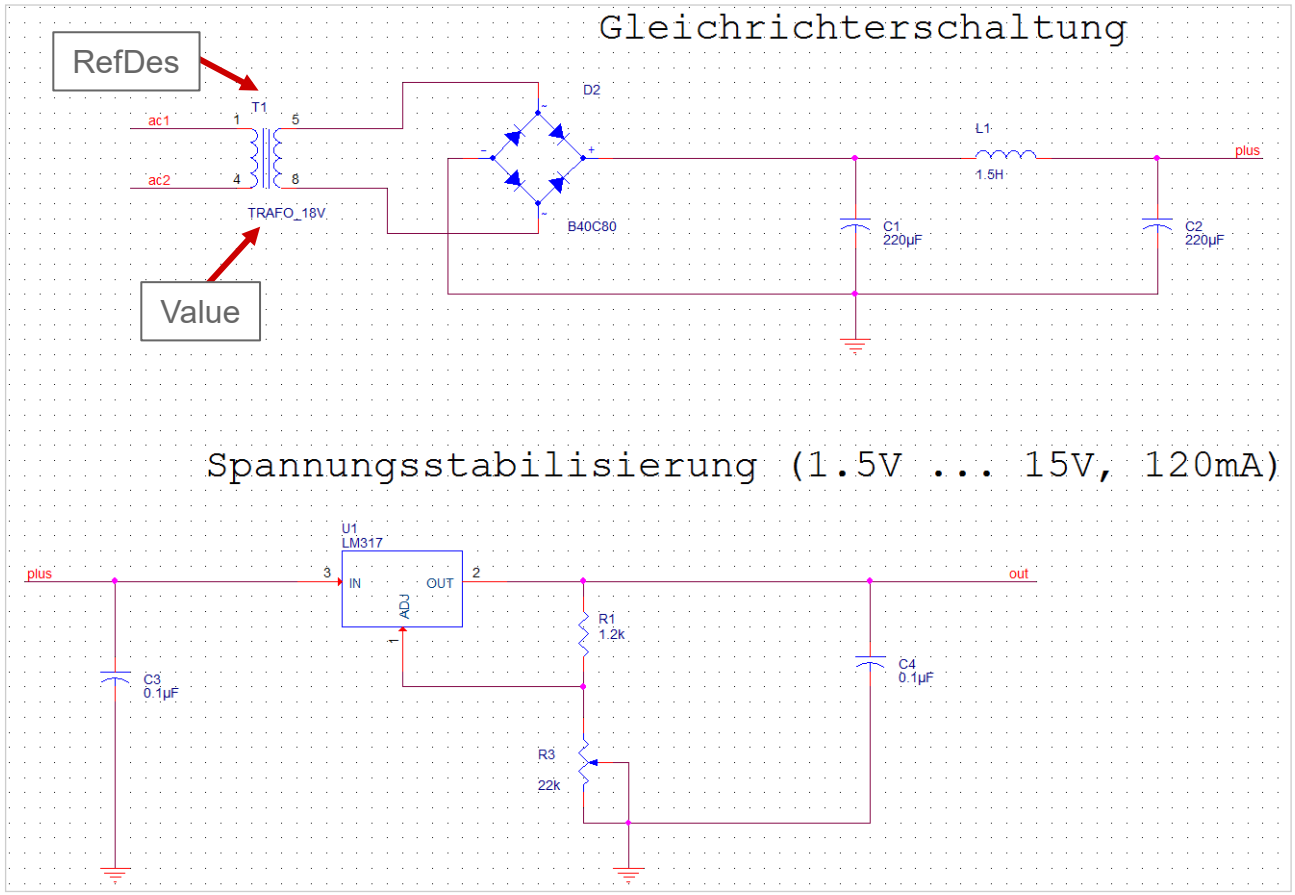
Edit Properties

Edit Properties (I)

After placement and wiring, schematic should look something like this:

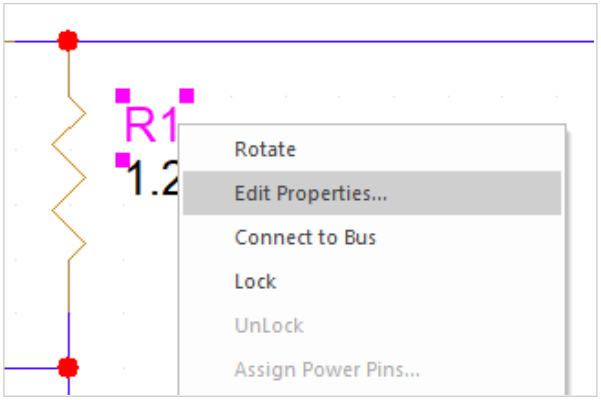
It is possible that labeling of parts (RefDes value) is not matching schematic template on page 6.

To correct this, **values** and **REFDES** of parts can be edited. This also makes sense for bill of materials generated later in the process.



Edit Properties (II)

Edit REFDES and value



The property window can be opened by

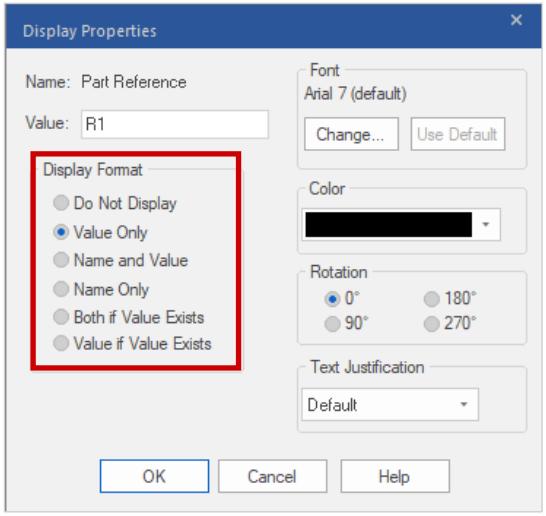
Selection (LMB) and **RMB > Edit Properties …**

or by

Double click

on desired property.

By **Display Format** different settings related to visibility in schematic are possible.



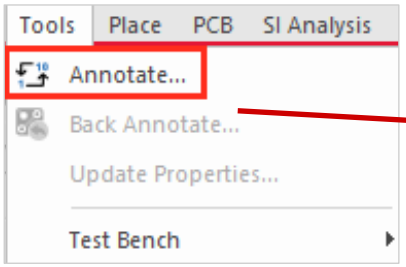
Tip

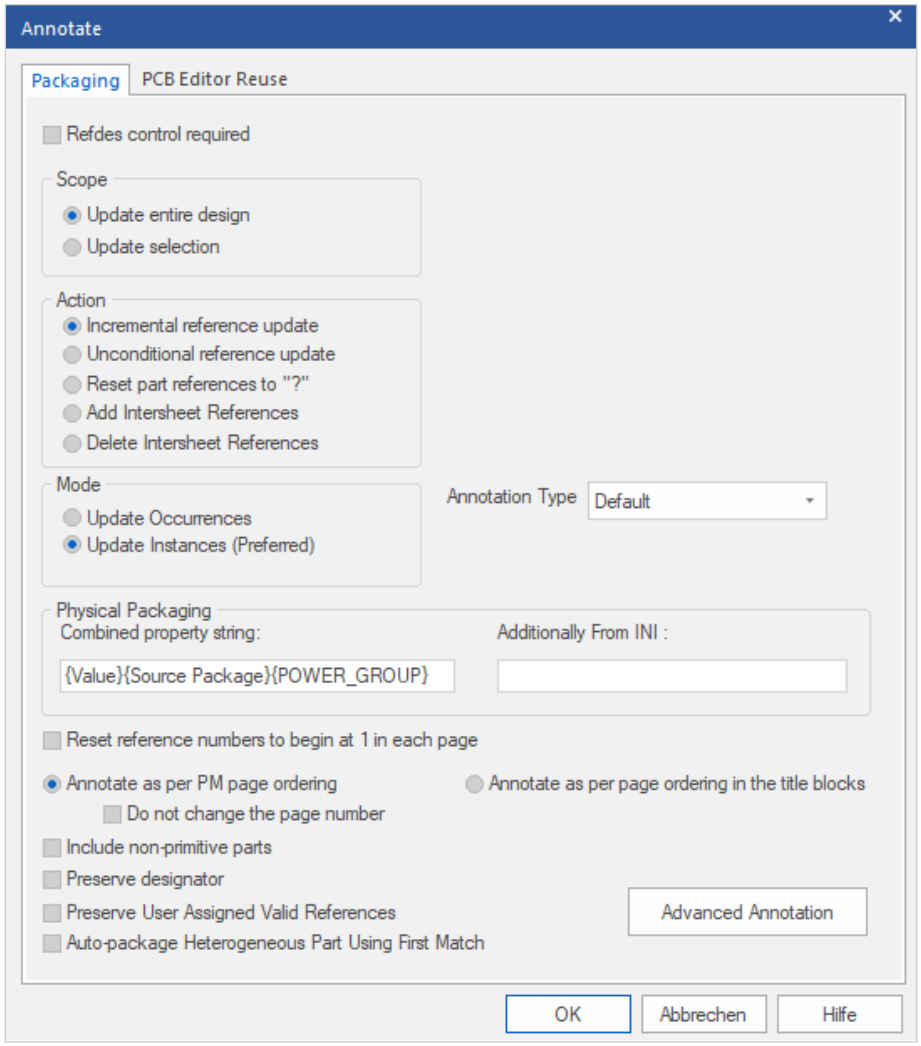
It is possible to open the property editor for multiple symbols, all symbols on one sheet or even for entire design.

- **Ctrl + LMB Click** or **Ctrl + A** followed by **RMB > Properties...**
- Select Page or Design (.dsn) in Project Manager, **Edit Object Properties** from pull-down menu.

Packaging – Annotation

Packaging and annotation of parts





On previous page, we have assigned reference designators manually. During editing, Capture does not verify if duplicated reference designators exist (2 x U1, or 3 x R5). Verification happens during handover to PCB Editor.

By **Tools > Annotate…** you have the option to complete this task automatically without any duplicate reference designators. For this function many options are available (to make command available, you must select .dsn file in project folder, otherwise **Annotate** command is not visible).

Completed Schematic

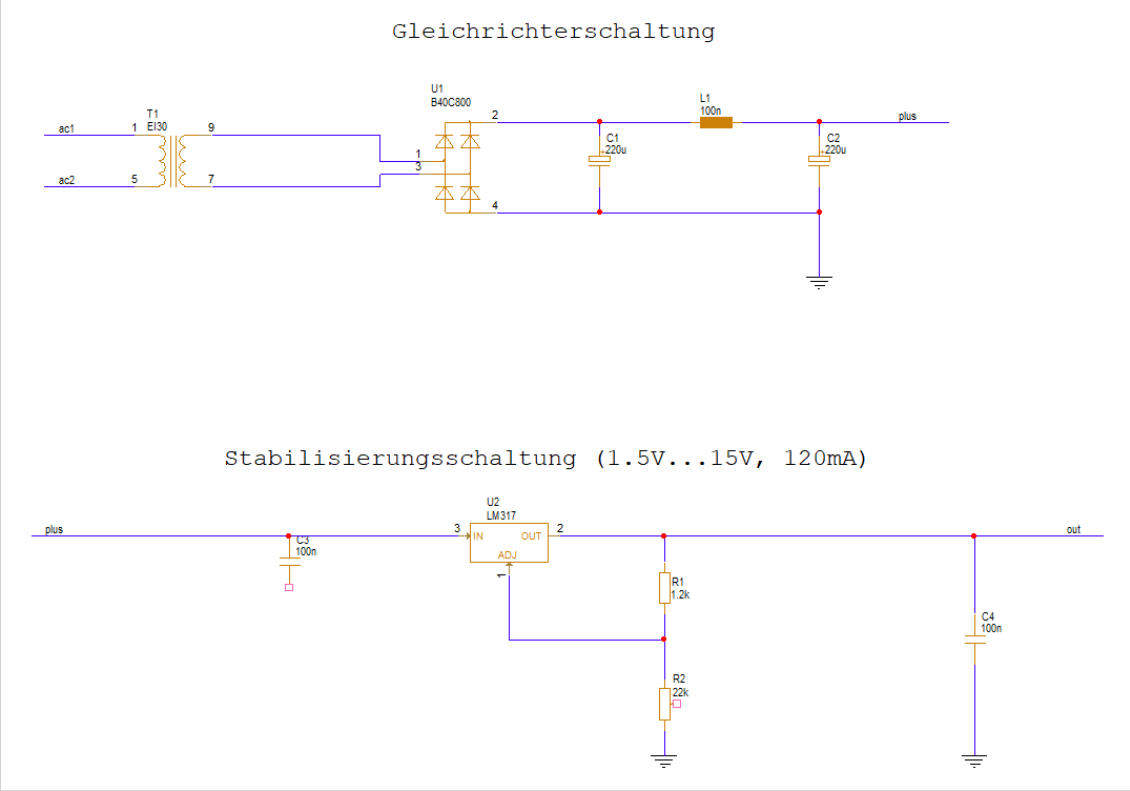
After the schematic is adjusted and parts and text got aligned, the completed schematic should look similar to the schematic on this page.

Note

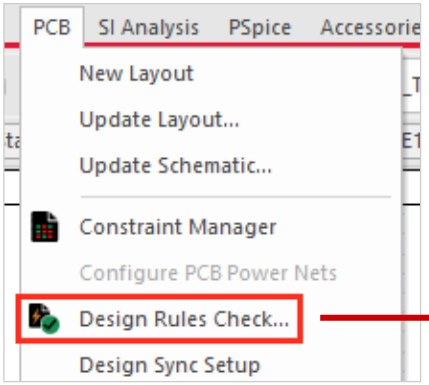
At this point we would like to mention that because of the used libraries assignment of footprints (DIP14, SO14, SMD1206, etc.) is not done so far.

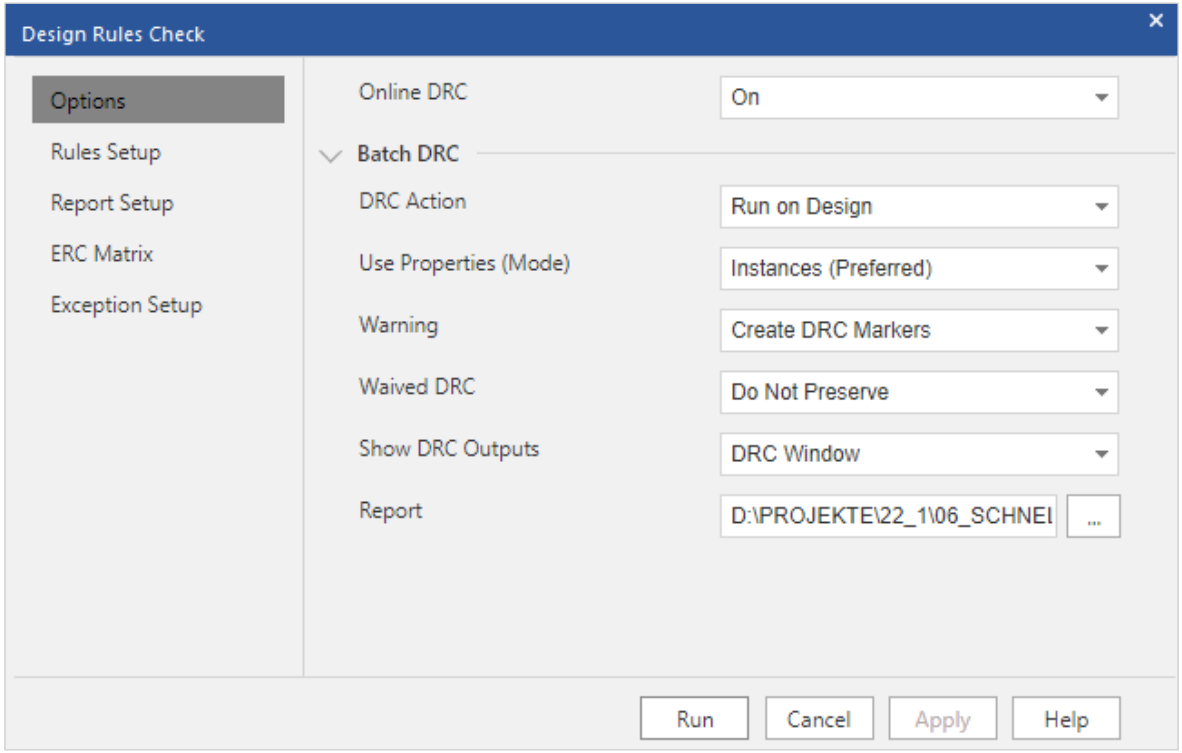
This is a later subject, when we import the data into layout module.

Just an upfront information:
Footprint assignment has to be done in schematic. A missing footprint entry will cause an error message during data synchronization. If the name is not finalized, it is possible to use a dummy text string as a placeholder.
Footprints can be predefined in the library or by the property editor later.



Design Rule Check

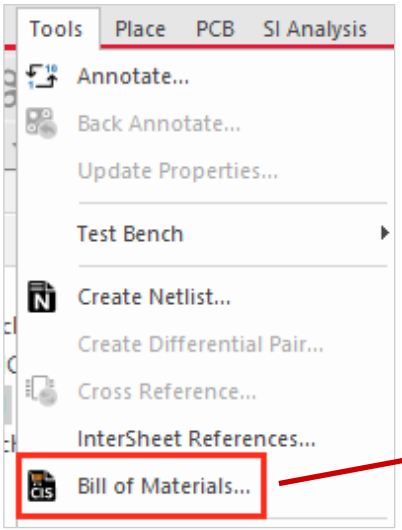


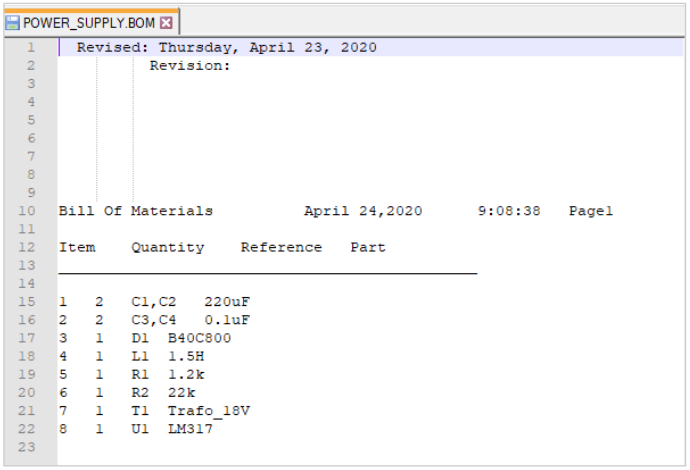


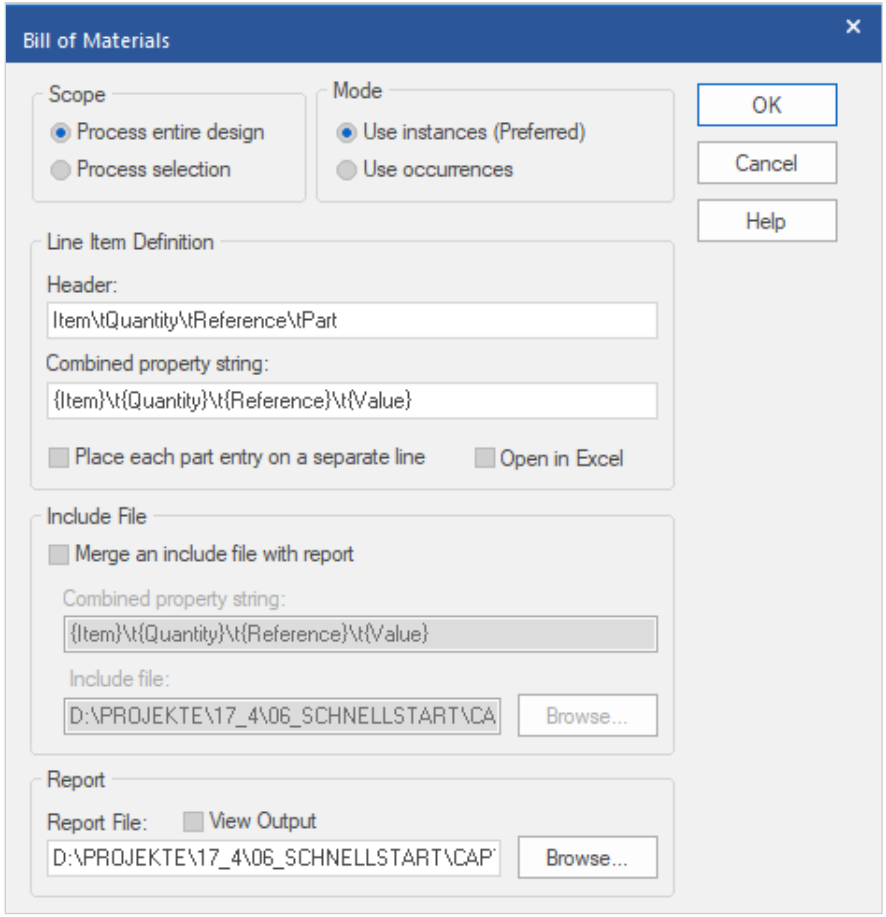
The DRC is able to identify one pin nets, nets without input or output. Pre condition is a correct defined library.

Please select the .dsn file in project folder, otherwise the **Design Rule Check** will not be available.

Bill of Material (BOM)



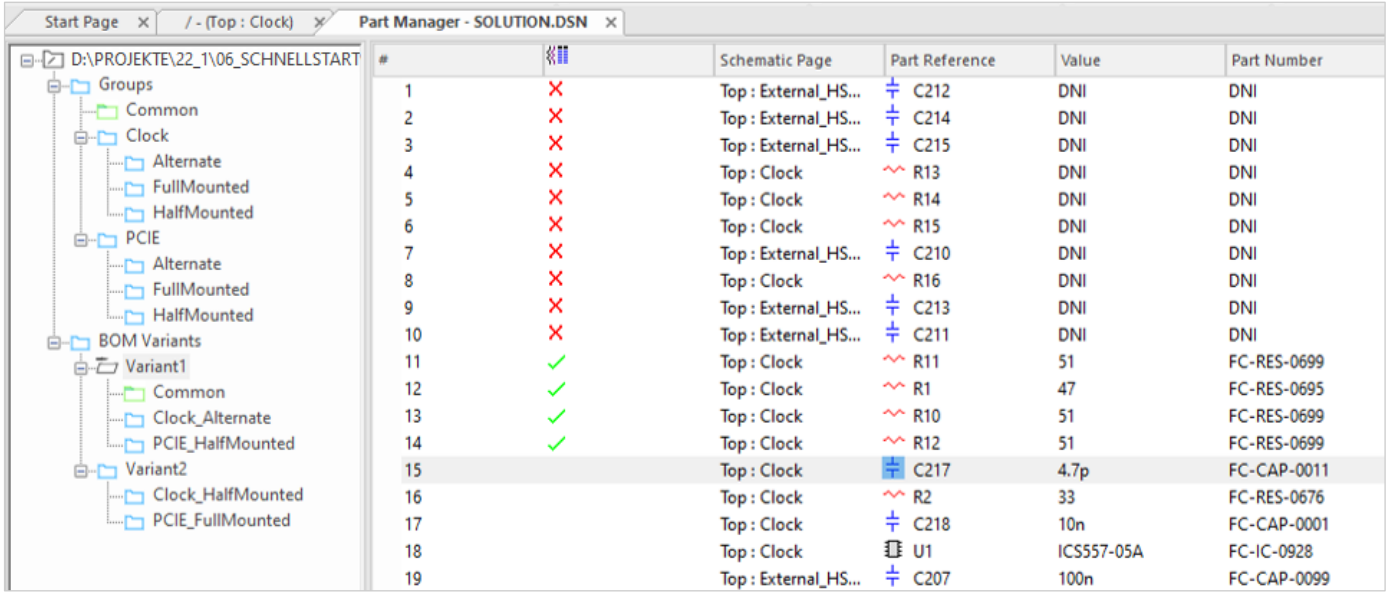




Design Variants

Variants – Overview

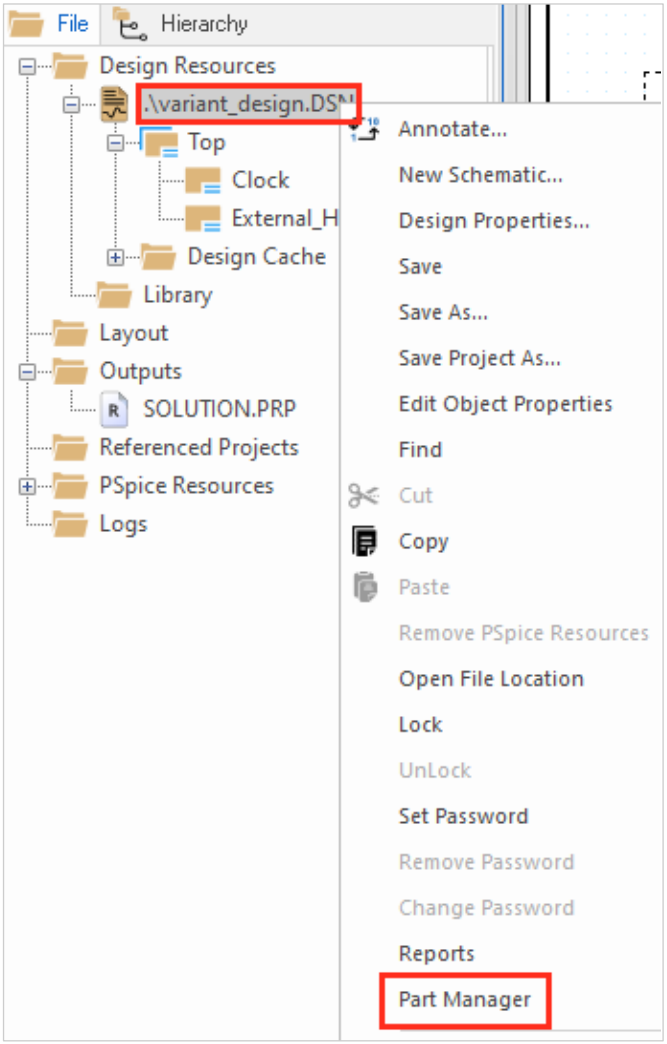
- Following steps are necessary to create design variants:
  - Dividing the design into meaningful groups, for example, according to function blocks
  - Creating subgroups that reflect assembly variants of blocks
  - Creating BOM variants of entire projects (these are actual variants used by the assembly house)
  - Adding subgroups to BOM variants to create the desired assembly
- These steps are explained in detail on the following pages.



#		Schematic Page	Part Reference	Value	Part Number
1	✗	Top : External_HS...	C212	DNI	DNI
2	✗	Top : External_HS...	C214	DNI	DNI
3	✗	Top : External_HS...	C215	DNI	DNI
4	✗	Top : Clock	R13	DNI	DNI
5	✗	Top : Clock	R14	DNI	DNI
6	✗	Top : Clock	R15	DNI	DNI
7	✗	Top : External_HS...	C210	DNI	DNI
8	✗	Top : Clock	R16	DNI	DNI
9	✗	Top : External_HS...	C213	DNI	DNI
10	✗	Top : External_HS...	C211	DNI	DNI
11	✓	Top : Clock	R11	51	FC-RES-0699
12	✓	Top : Clock	R1	47	FC-RES-0695
13	✓	Top : Clock	R10	51	FC-RES-0699
14	✓	Top : Clock	R12	51	FC-RES-0699
15		Top : Clock	C217	4.7p	FC-CAP-0011
16		Top : Clock	R2	33	FC-RES-0676
17		Top : Clock	C218	10n	FC-CAP-0001
18		Top : Clock	U1	ICS557-05A	FC-IC-0928
19		Top : External_HS...	C207	100n	FC-CAP-0099

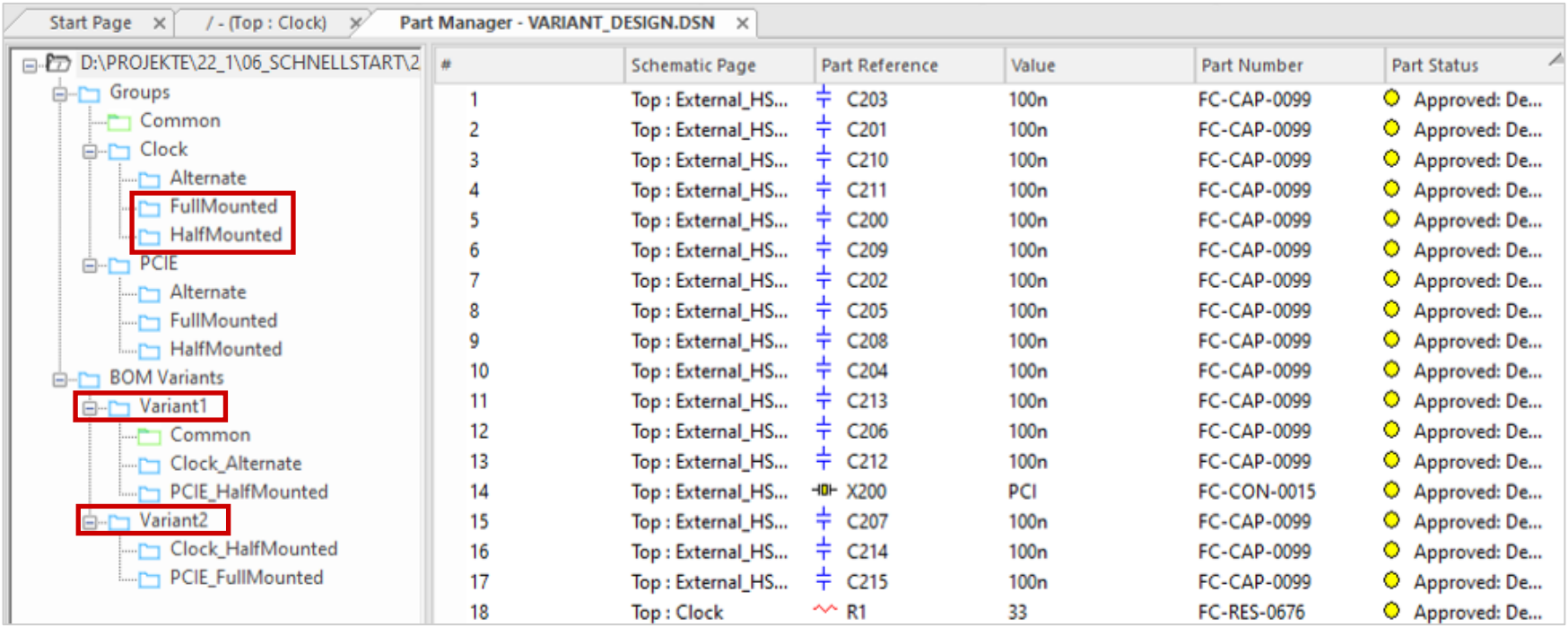
Variants – Part Manager

- **Part Manager** is used to manage the parts in the design, perform database updates or generate groups, subgroups and finally the variants.
- You can open **Part Manager** by the right mouse button menu on the dsn file.



Variants – Example Design

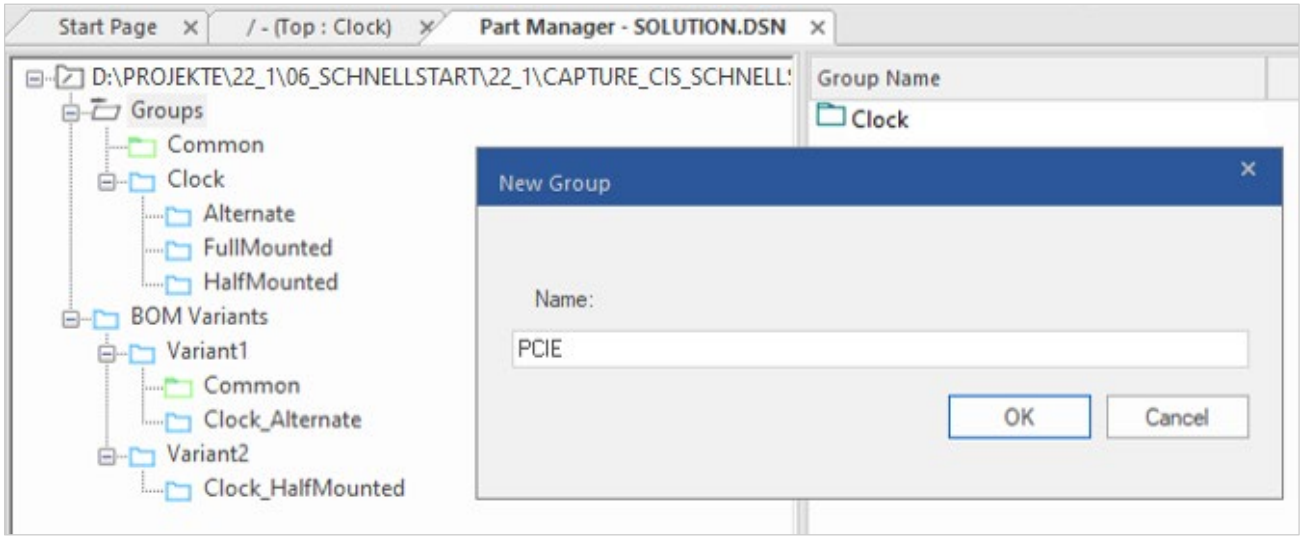
- The Capture Quick Start contains a variant sample design:
  - ~\Capture_CIS_Schnellstart_22_1\Solutions\VARIANT_DESIGN.DSN
  - Groups, subgroups and variants are already preconfigured in this example.
  - Creation of variants is described in this chapter.



#	Schematic Page	Part Reference	Value	Part Number	Part Status
1	Top : External_HS...	C203	100n	FC-CAP-0099	Approved: De...
2	Top : External_HS...	C201	100n	FC-CAP-0099	Approved: De...
3	Top : External_HS...	C210	100n	FC-CAP-0099	Approved: De...
4	Top : External_HS...	C211	100n	FC-CAP-0099	Approved: De...
5	Top : External_HS...	C200	100n	FC-CAP-0099	Approved: De...
6	Top : External_HS...	C209	100n	FC-CAP-0099	Approved: De...
7	Top : External_HS...	C202	100n	FC-CAP-0099	Approved: De...
8	Top : External_HS...	C205	100n	FC-CAP-0099	Approved: De...
9	Top : External_HS...	C208	100n	FC-CAP-0099	Approved: De...
10	Top : External_HS...	C204	100n	FC-CAP-0099	Approved: De...
11	Top : External_HS...	C213	100n	FC-CAP-0099	Approved: De...
12	Top : External_HS...	C206	100n	FC-CAP-0099	Approved: De...
13	Top : External_HS...	C212	100n	FC-CAP-0099	Approved: De...
14	Top : External_HS...	X200	PCI	FC-CON-0015	Approved: De...
15	Top : External_HS...	C207	100n	FC-CAP-0099	Approved: De...
16	Top : External_HS...	C214	100n	FC-CAP-0099	Approved: De...
17	Top : External_HS...	C215	100n	FC-CAP-0099	Approved: De...
18	Top : Clock	R1	33	FC-RES-0676	Approved: De...

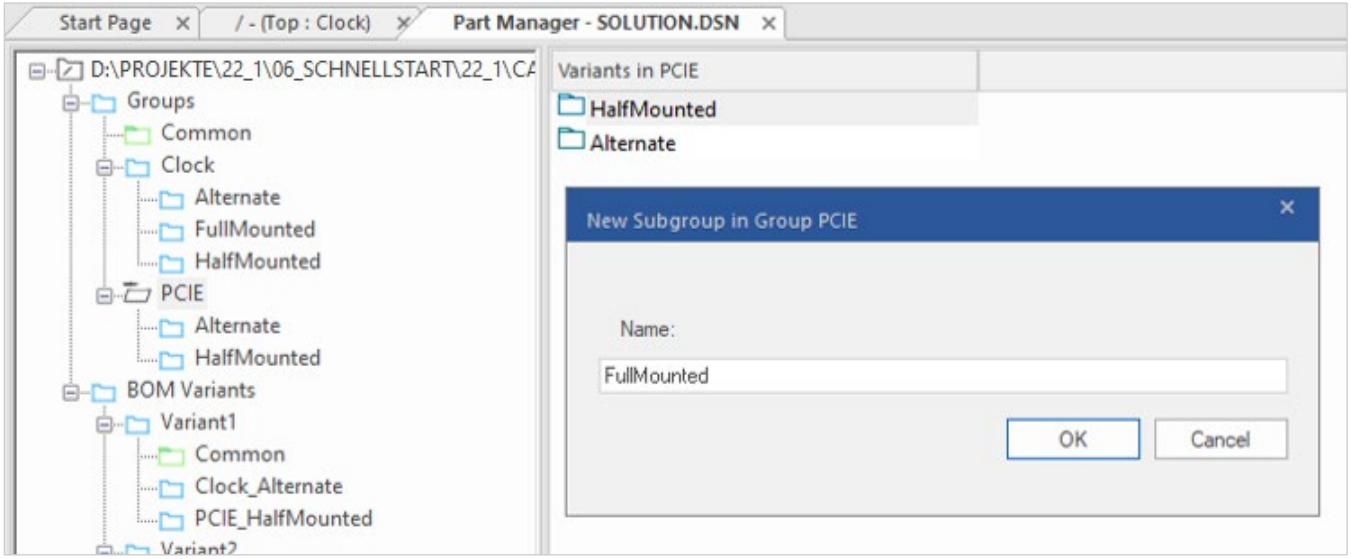
Variants – Groups

- To create variants, you need to divide the design into groups.
- It makes sense to store all parts of a function group in one group for variant definition. Function groups that do not have any assembly variants remain in common group.
- Select **Groups > RMB > New Group.** Create a group named **Clock**. Repeat this process and generate a group **PCIE**.



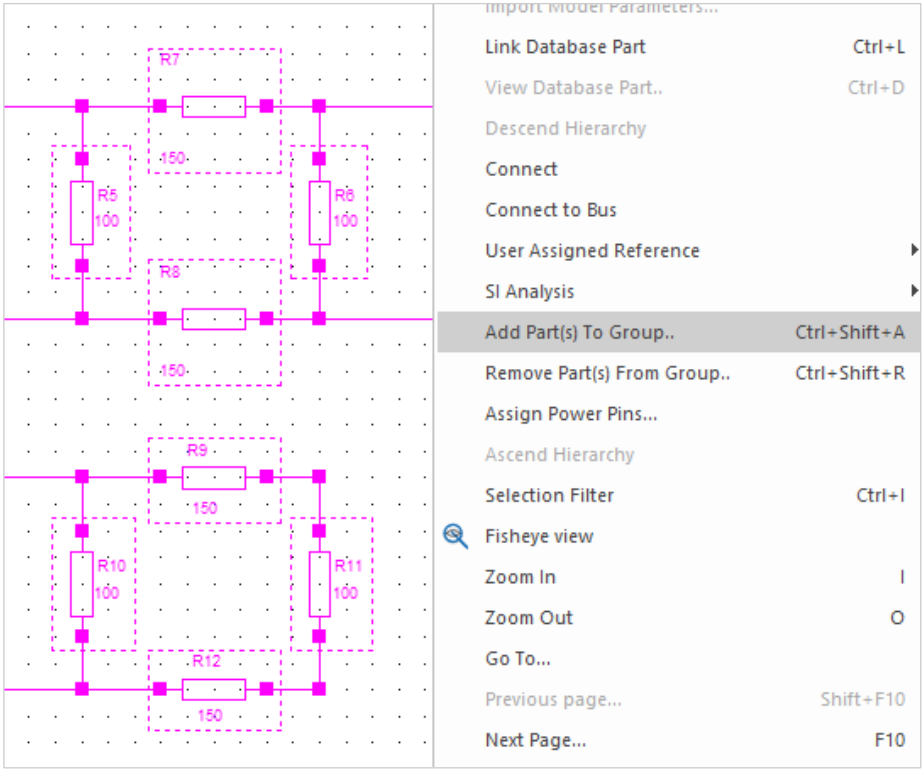
Variants – Subgroups

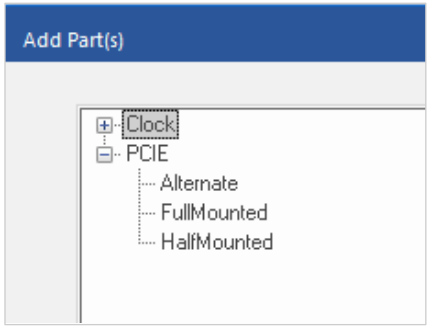
- Next step is to define subgroups that contain different assembly variants of the groups.
- Select the group **Clock > RMB > New Subgroup.**
- Enter **Alternate**.
- Repeat this process, and enter **HalfMounted** and then **FullMounted**.

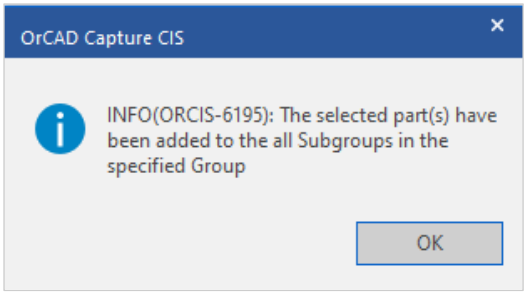


Variants – Assign Parts to Groups

- Open **Clock** page and select all parts.
- **RMB > Add Part to Group,** select the group **Clock > Add.**
- Repeat this action with the second page and move all capacitors (not connectors) into group **PCIE**.

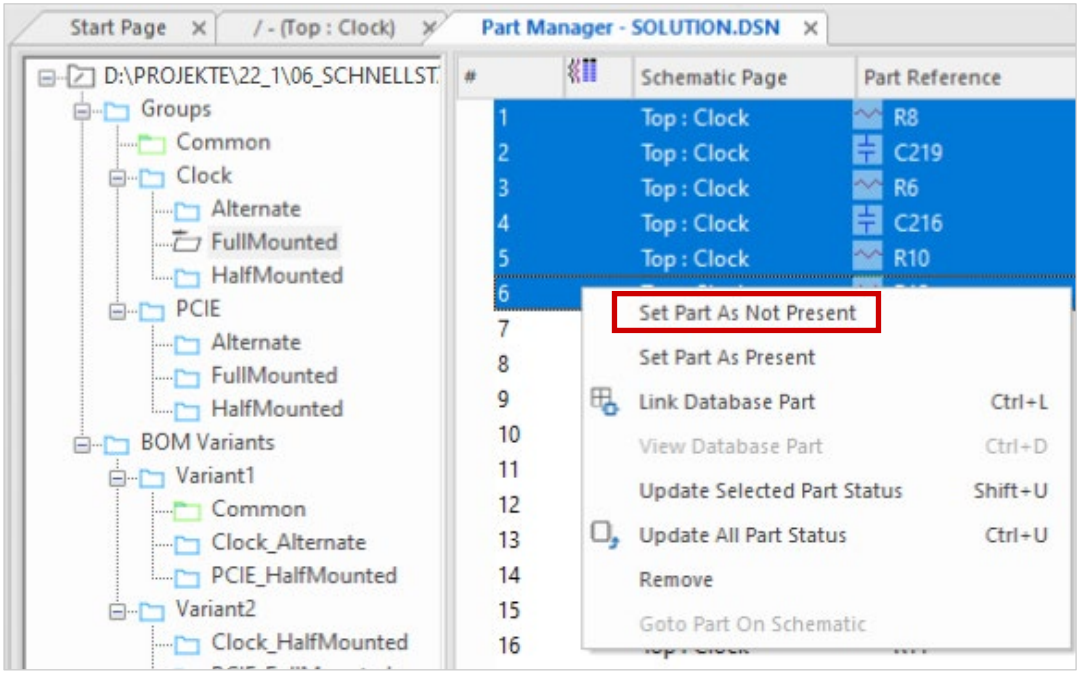






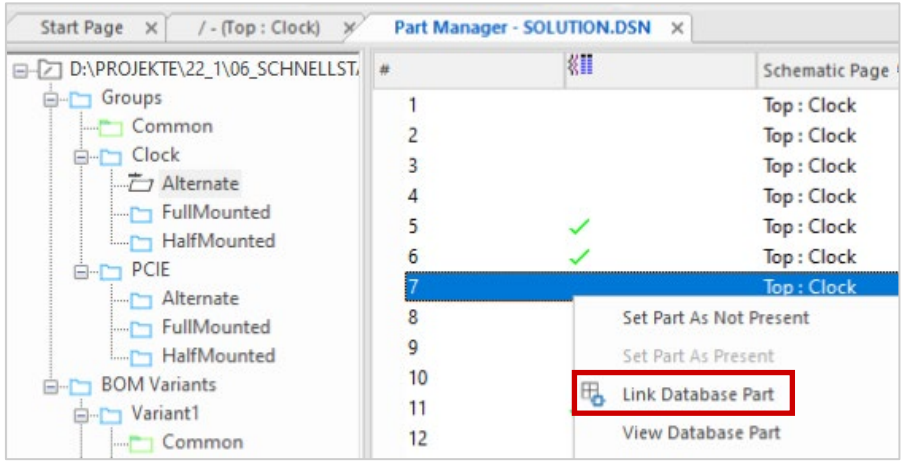
Variants – Unmounted Parts

- Go to Part Manager and select a subgroup **HalfMounted.**
- Select some parts and set them as not present with **RMB > Set Part as not present.**
- All unmounted parts are marked with a red cross.
- Repeat this action in another subgroup.



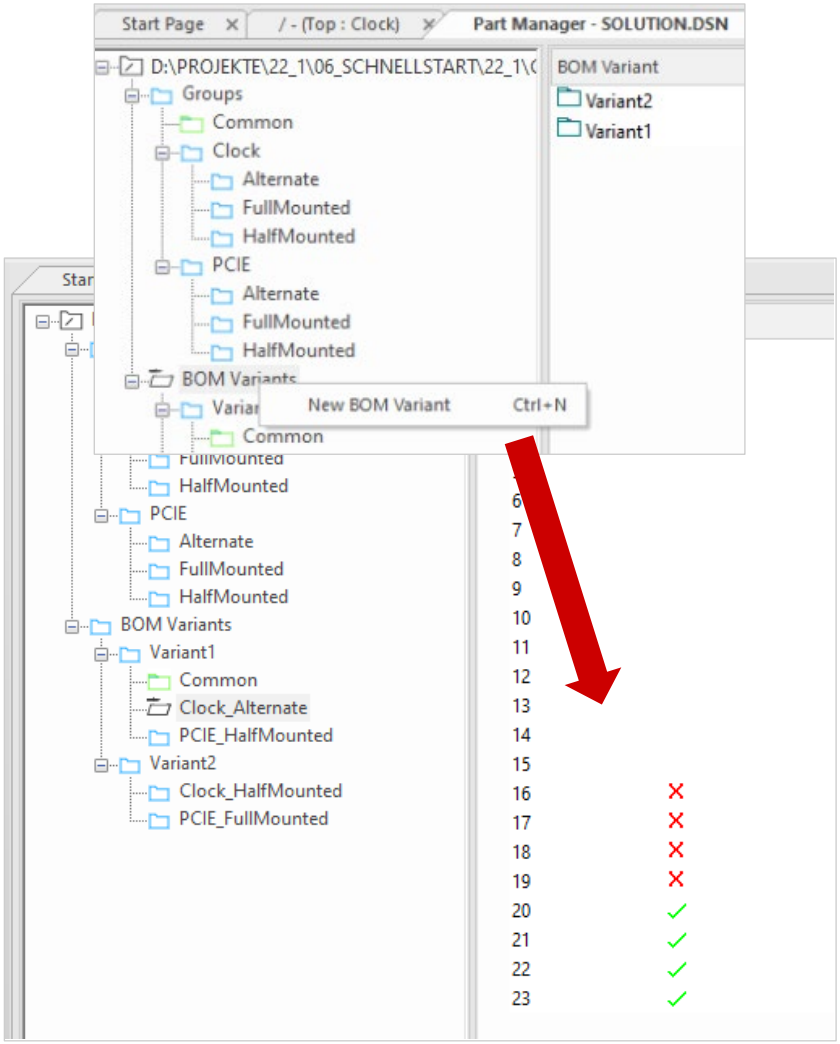
Variants – Alternative Placement

- Go to **Alternate** subgroup and select a part, preferably a resistor or capacitor **RMB > Link Database Part.**
- CIS Explorer opens, and an alternative part can be selected. Make sure that parts have identical symbols and footprints. This can be ensured, for example, by a query search in which footprint is specified.
- CIS Explorer supports you in your search. Properties with different contents are highlighted in red.
- **Link Database Part** also works if you select several identical parts in Part Manager. All parts with alternative placement are marked with a green tick.



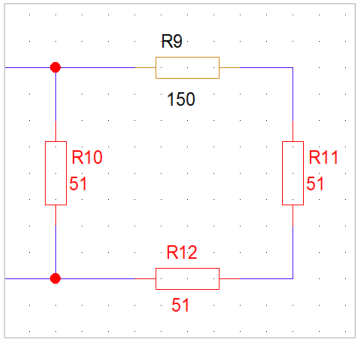
Variants – Define BOM

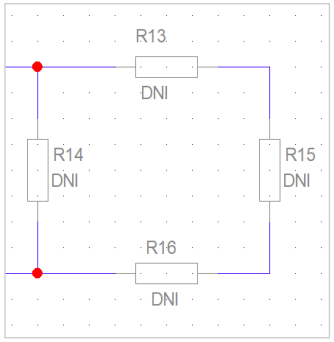
- Select **BOM Variants > RMB > New BOM Variants** and enter name **BOM1**. Repeat for **BOM2** and **BOM3**.
- Drag one subgroup per group into BOM. Finally, the common group must be added to BOM, as it contains the PCIE connector, that has not been assigned to any group.
- Variant BOM is shown in the lower right corner.

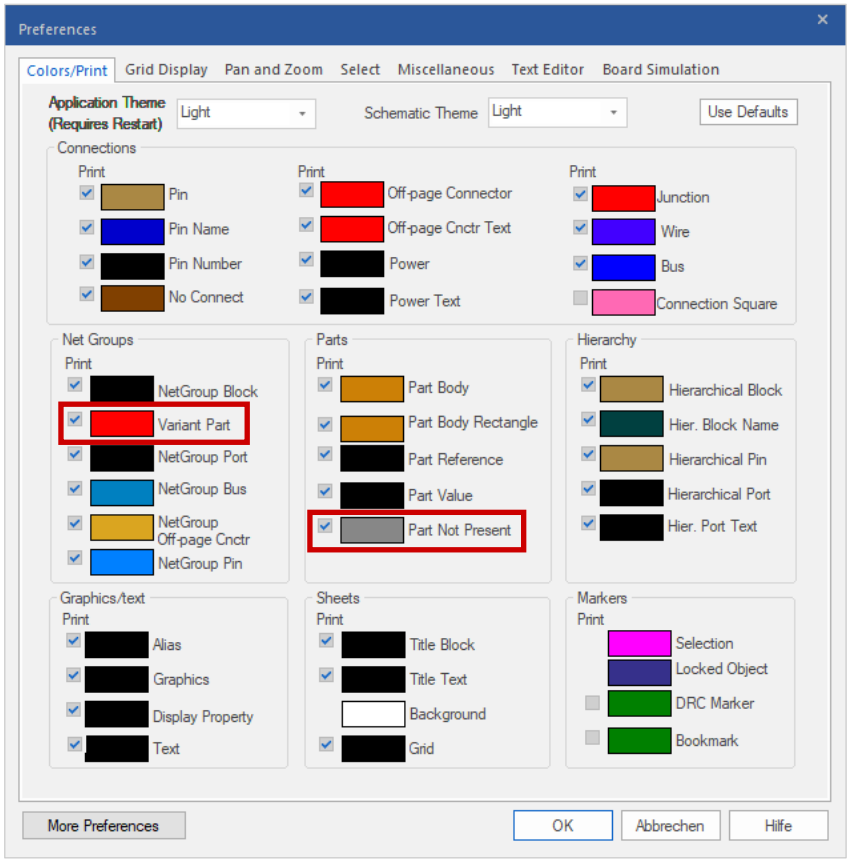


Variants – Schematic

- Close Part Manager, go to a schematic page with variant description and select **View > Variant View Mode.** Select one of the variants. Unmounted parts are displayed in grey with lettering DNI, parts with an alternative placement are displayed in red.
- You can set colors via **Options > Preferences**.

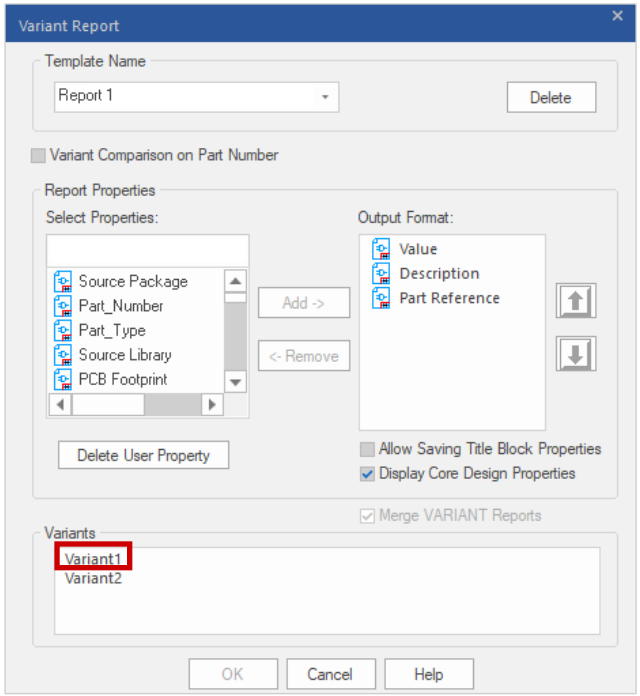






Variants – Report

- You can use Variant Report to check variant definitions or compare them with the core design. Open Part Manager and select **Report > Variant Report.** Make settings as shown in the picture. Select one of the variants > **OK**.
- The report is created and displayed. You get a direct comparison of core design and variants.



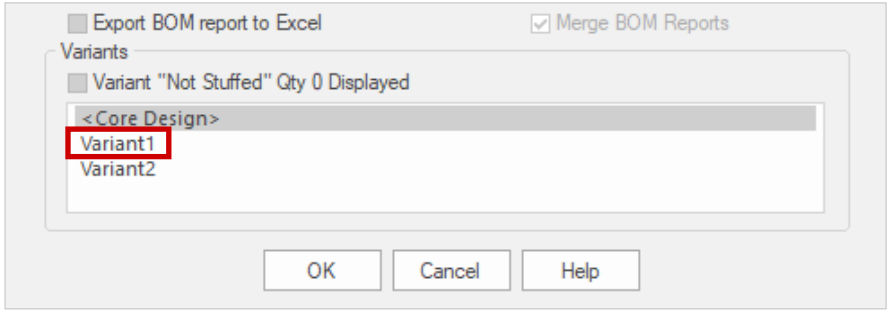
Start Page | / - (Top : Clock) Variant1 | SOLUTION.VRT*

Capture CIS -
Report Created on Fr

Item Number	Part Reference	<Core Design>	Variant1	Value	Description
1	R12	FC-RES-0750	FC-RES-0699	[150] 51	[Thick Film Resistor 150r 1% 0.063W 0402 SMD] Thick Film Resisto 51r % 0.063W 0402 SMD
2	R11	FC-RES-0732	FC-RES-0699	[100] 51	[Thick Film Resistor 100r 1% 0.063W 0402 SMD] Thick Film Resistor 51r 1% 0.063W 0402 SMD
3	R10	FC-RES-0732	FC-RES-0699	[100] 51	[Thick Film Resistor 100r 1% 0.063W 0402 SMD] Thick Film Resistor 51r 1% 0.063W 0402 SMD
4	R1	FC-RES-0676	FC-RES-0695	[33] 47	[Thick Film Resistor 33r 1% 0.063W 0402 SMD] Thick Film Resistor 47r 1% 0.063W 0402 SMD
5	R16	FC-RES-0750	DNI	DNI	DNI
6	R15	FC-RES-0732	DNI	DNI	DNI

Variants – Generate BOM

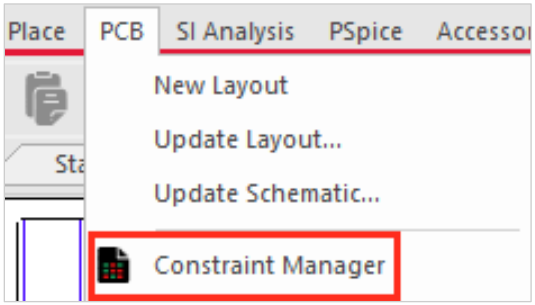
- BOM variants can be generated as shown on page 40 by **Report > CIS Bill of Material > Standard**.
- Use an existing template and select one of the variants below.
- You will find **Variants Not Stuffed Qty 0 Displayed** above list of variants. If you check this box, all unstuffed parts are also displayed in the parts list with count 0.
- Click **OK** to start parts list output. The Variant BOM is displayed.

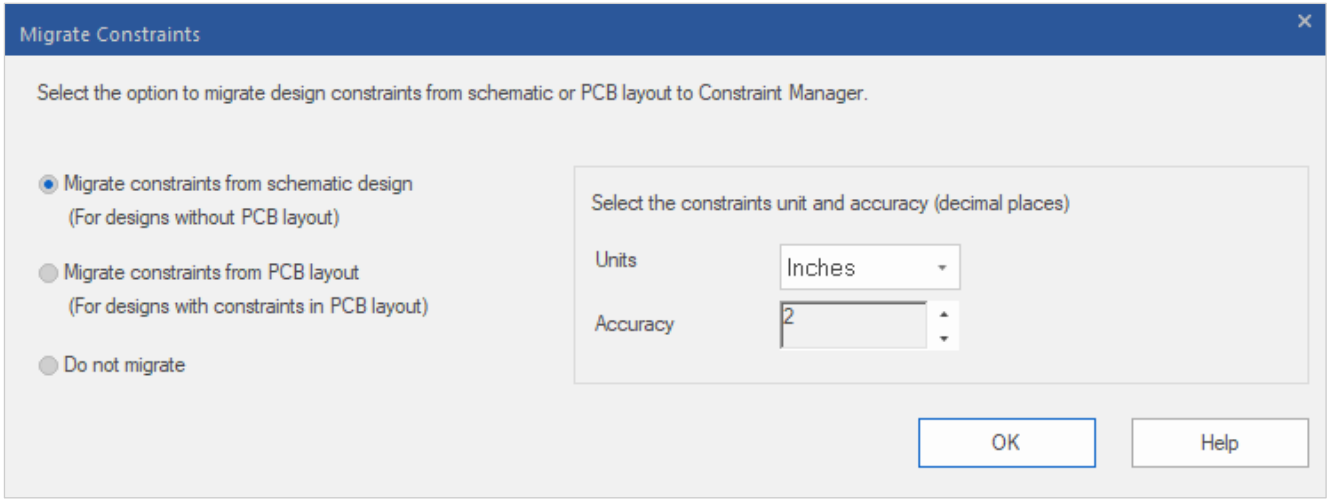


Constraint Manager

Constraint Manager in Capture

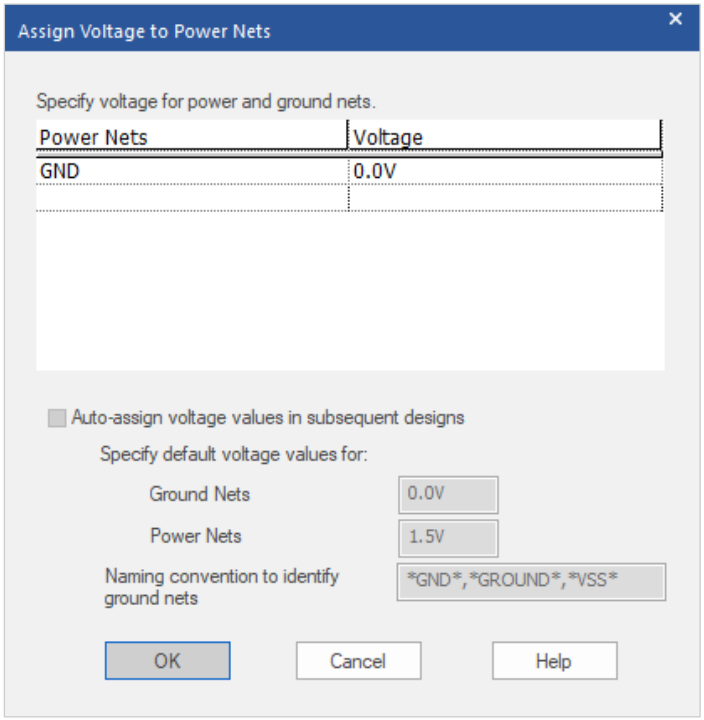
- To start Constraint Manager from Capture, please use command **PCB > Constraint Manager** or the icon .
- Confirm query **Migrate constraints from schematic design** with **OK.**





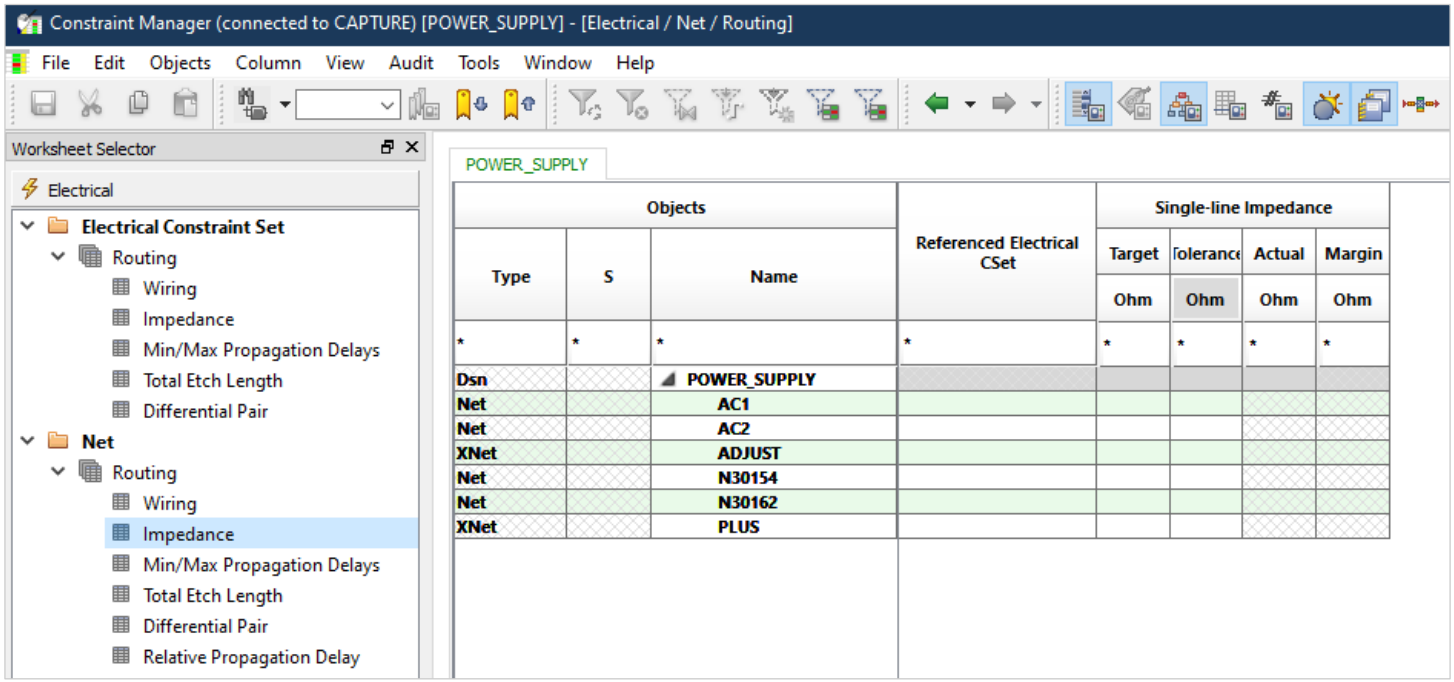
Assign Voltages to Power Nets

- In next step, after first start of Constraint Manager, you can assign Voltages to Power Nets.



Constraint Manager in Capture

- From schematic point of view, mostly definition of electrical constraints makes sense.
- Many constraints require data from layout. This data is available after back annotation from layout.
- You can find more detailed information in the PCB Editor Quick Start starting on page 77.

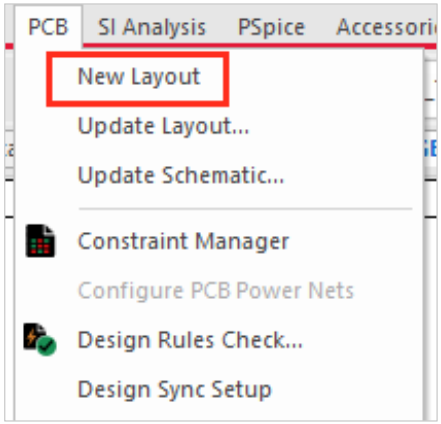


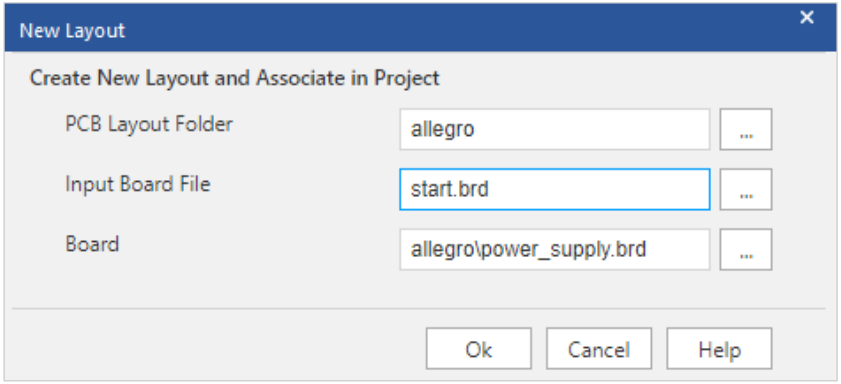
Design Sync

Generate Layout (I)

By **PCB > New Layout** logic information is transferred to a new PCB.

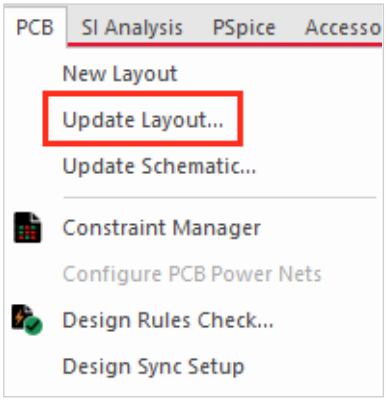
- PCB Layout Folder: Directory in which design sync information is written
- Input Board File: A base or board template
- Board: Newly generated board file

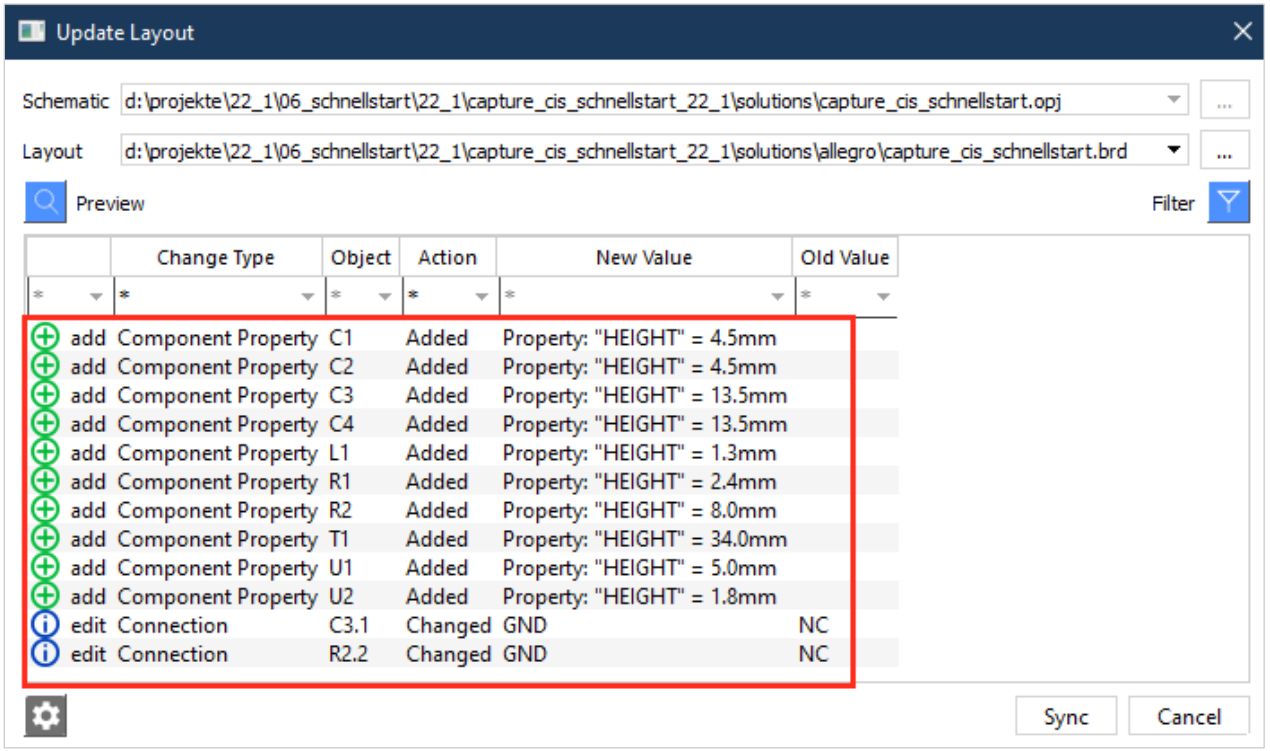




Generate Layout (II)

- An existing PCB can be updated by **PCB > Design Sync**.
- Changes are displayed in an ECO list.





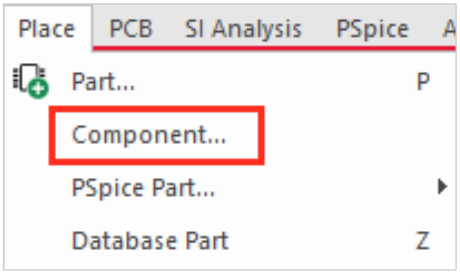
Libraries

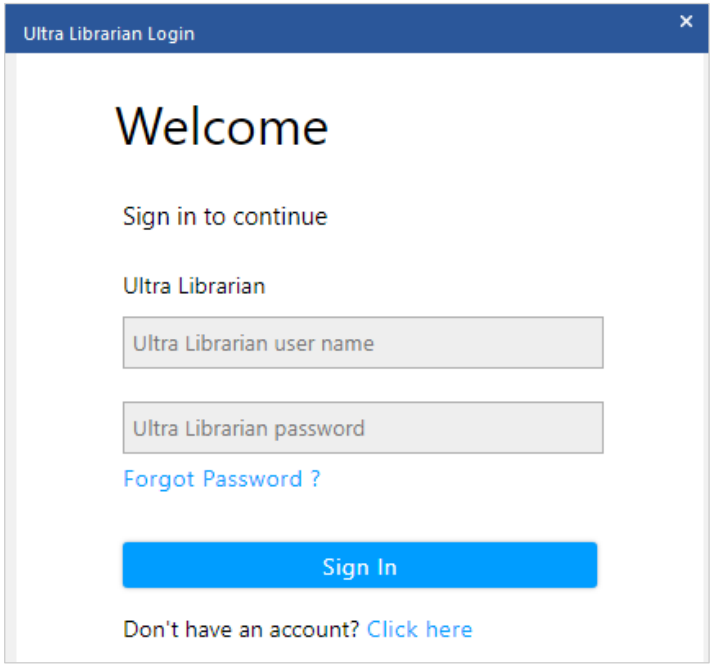
Missing Parts

In this chapter we describe how to search parts in web portals by the **Place > Component** function and also how to create new parts which are not in the OrCAD provided libraries.

Part Search (I)

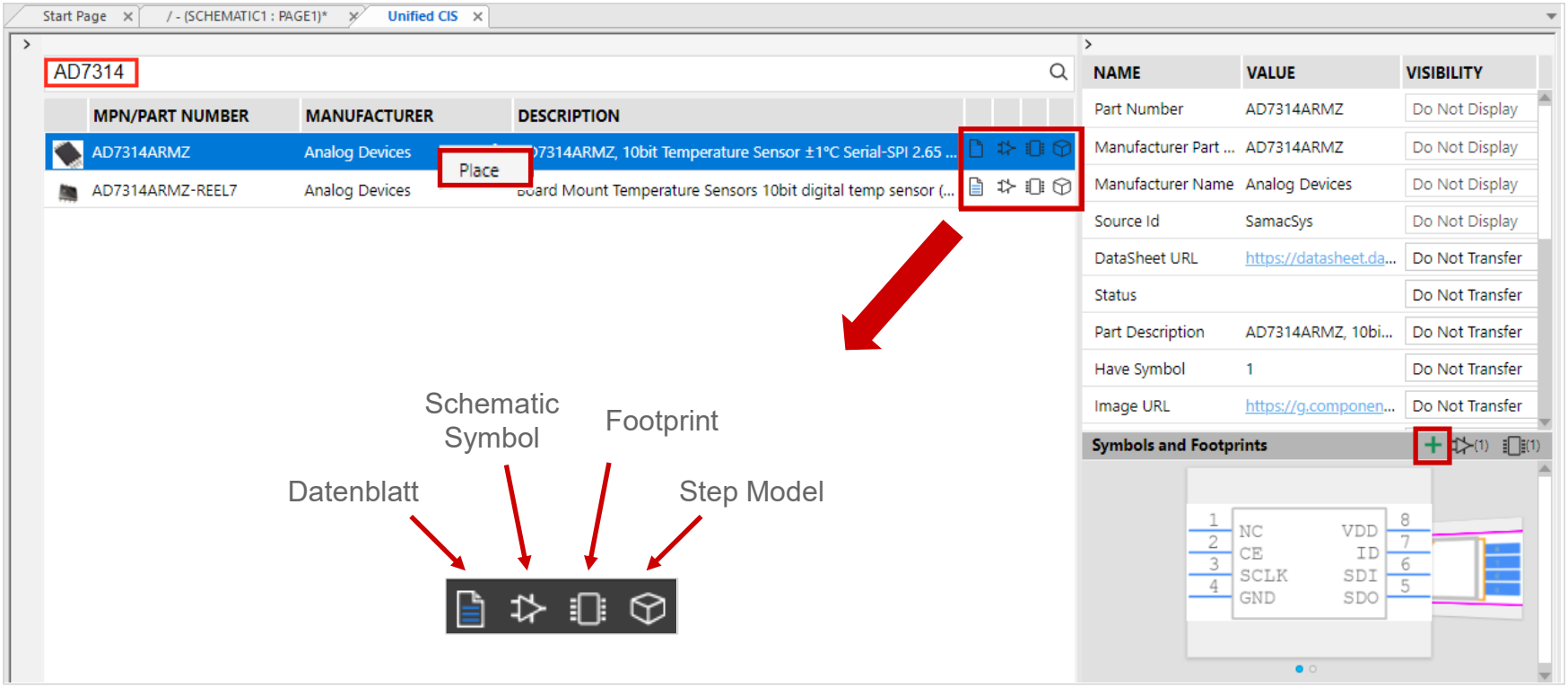
- If required parts are not available in present libraries, you have the option to search and download parts from Ultra Librarian and SamacSys web portals.
- Portals provide datasheets, schematic symbols, footprints and step models.
- Select **Place > Component** to start web search.
- To use web portals, a free of charge Cadence and Ultra Librarian account is required.
- Don't have an account? **Click here** to create a new account.





Part Search (II)

- **Unified CIS** user interface provides a Google like search.
- The icons on the right side of the result list indicate whether data sheet, schematic symbol, footprint and step model are already available.
- By right mouse button and **Place** or with a click on + the components can be placed.



Part Creation

New parts can also be created with OrCAD Capture capabilities.

Procedure will be demonstrated using LM317 as an example. This part is not available in the already connected libraries.

- **Counter.olb**
- **Discrete.olb**
- **OPAMP.olb**

Completed part can be found in:

- **Quickstart.olb**

New Library (I)

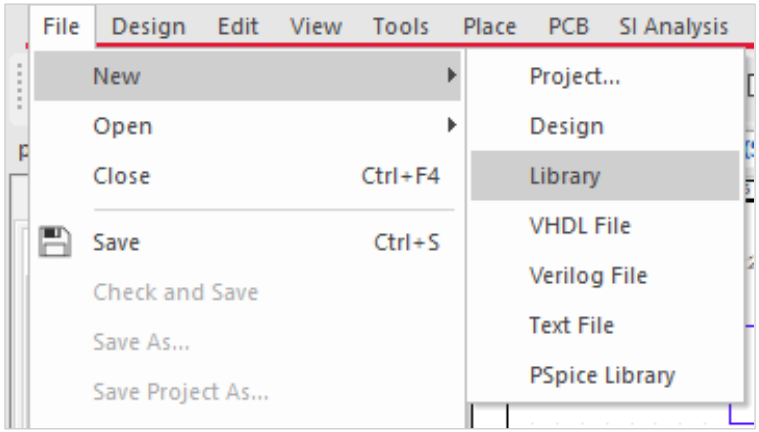
You can create new parts in a new library or open an existing library and add new parts.

Both actions happens by

- **File > New > Library**

or

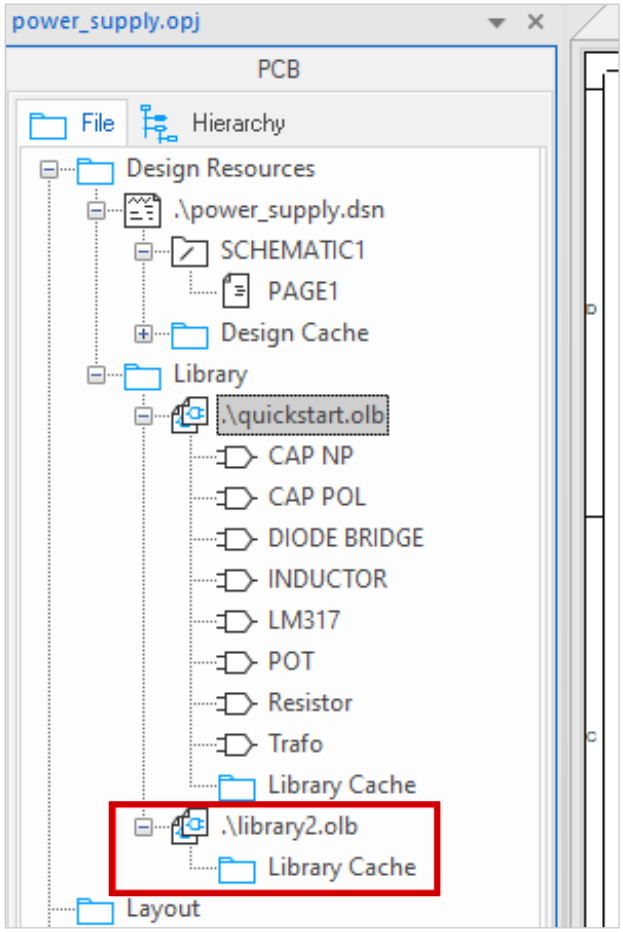
- **File > Open > Library**



New Library (II)

The new library can be saved in project manager **RMB > Save As...** at a desired location.

Afterwards this library can be assigned to your existing project.



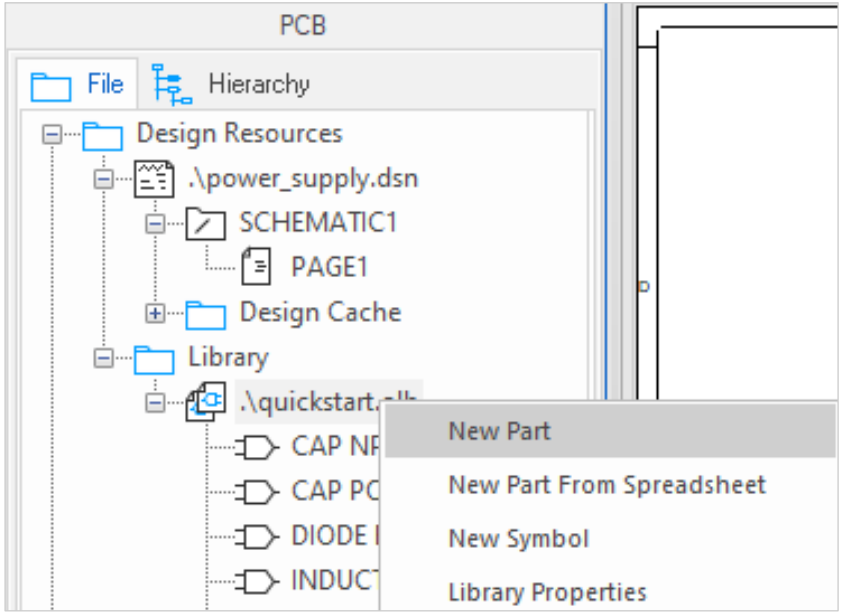
New Part Creation (I)

By **Design > New Part...**

or

RMB > New Part

you can start creation of a new part in a library.



New Part Creation (II)

Please enter the displayed values into the form.

New Part Name:

Part Reference Prefix:

PCB Footprint:

Part Aliases:
Same symbol, same function, but different package

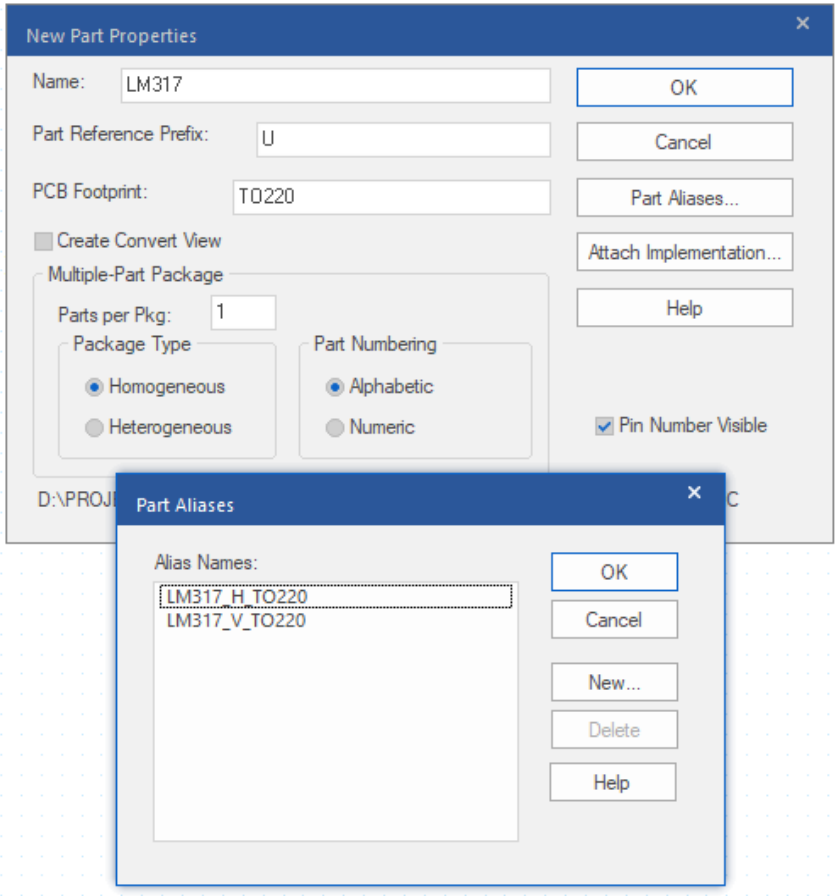
Parts per package:
i.e. 4 gates in 74LS00

Homogeneous:
Multiple identical circuits, i.e. 74LS00

Heterogenous:
i.e. relay with coil and switch

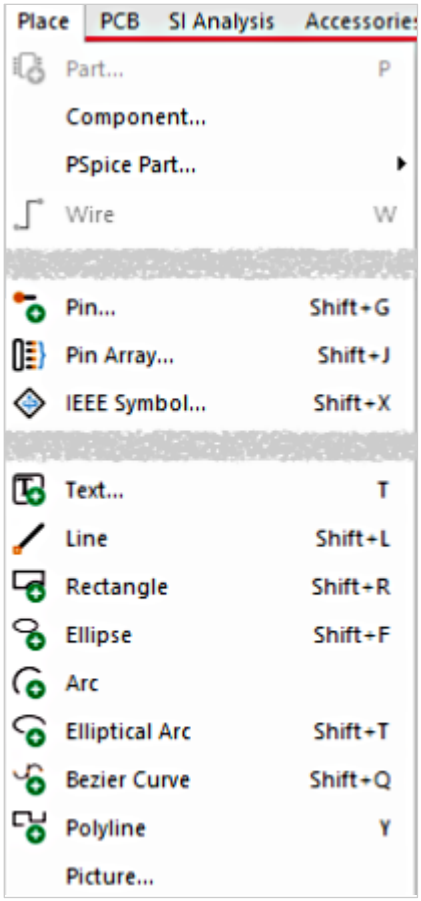
Part Numbering:
U?A, U?B ... oder U?1, U?2 ...

Click **OK**.



Commands in Capture

Commands for part editing are available in the Capture user interface. You can find them in the panel on the right boarder of Capture or in the **Place** pull-down menu.



Hovering over the icons shows function as Mouse Over.

On next pages we will explain most important functions.

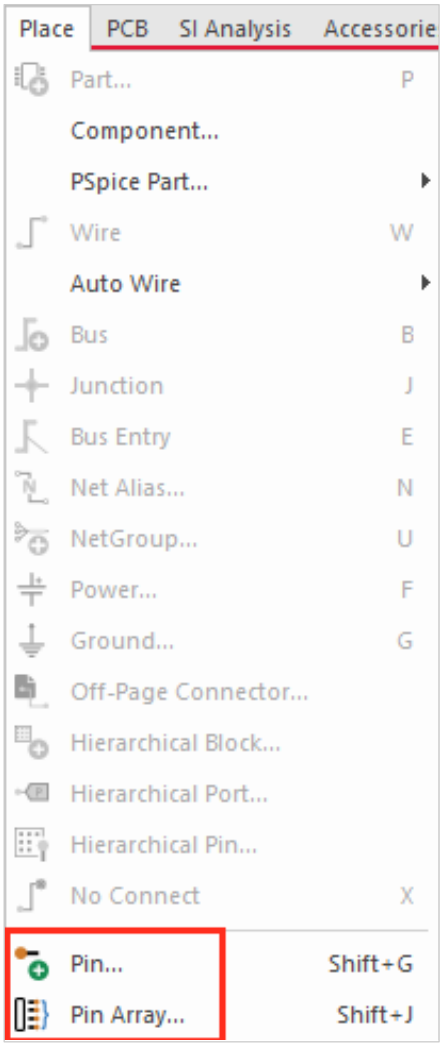
Adding Pins (I)

Place pins by:

- **Place > Pin...**

or

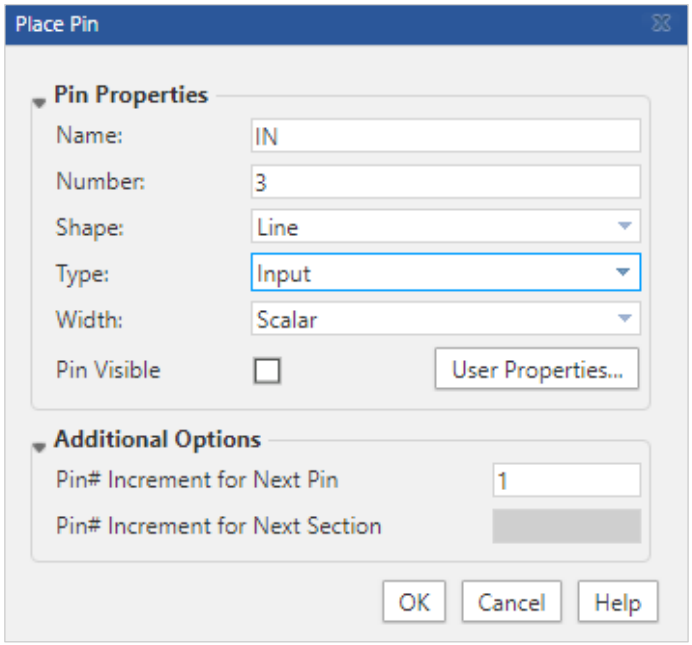
- **Place Pin Icon** 



Adding Pins (II)

The Place Pin panel appears, in which pin property definition can be done.

After clicking **OK**, the pin is attached to the curser and ready for placement.

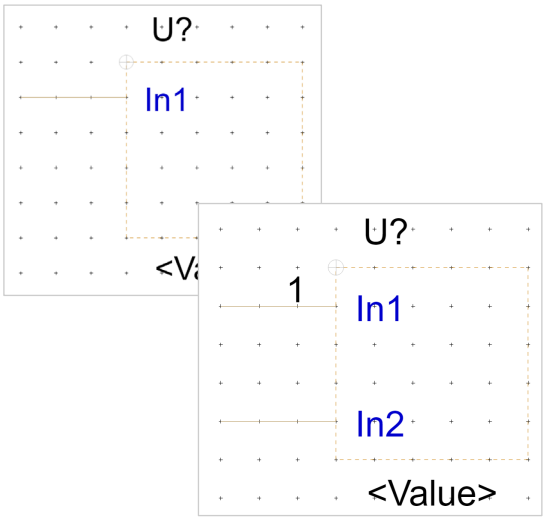


Tip

Using pin type Power allows visible and invisible pins.

Adding Pins (III)

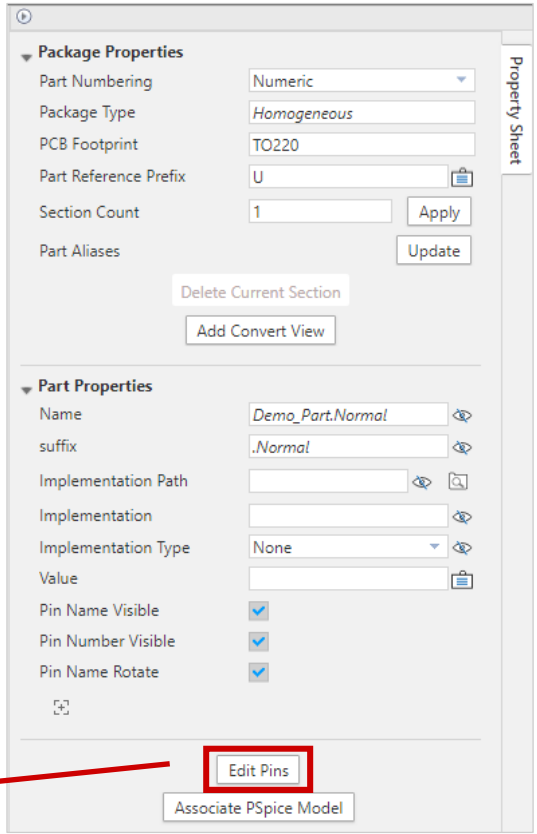
Individual pins get placed.

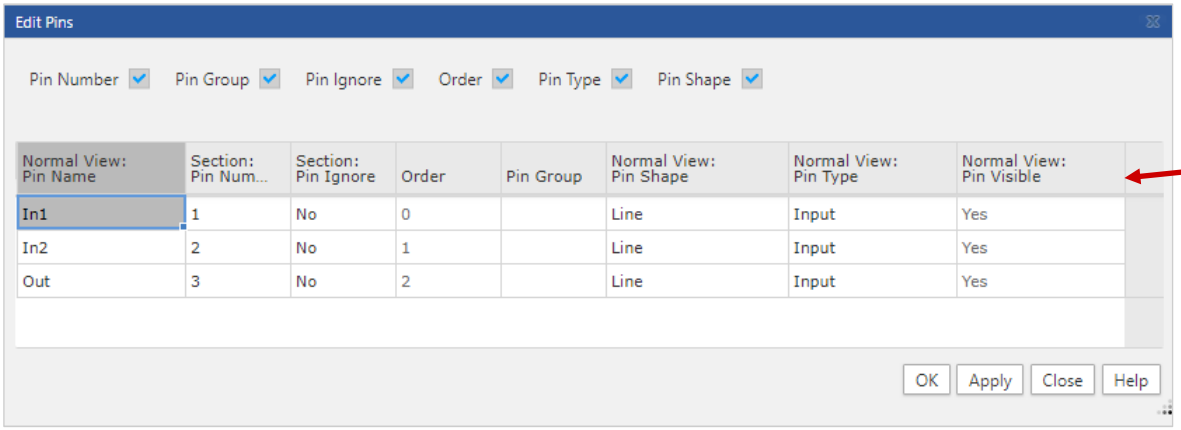


Additional pins can be placed by previous menu or by copy / paste **(Ctrl C, Ctrl V)**.

Placement of Pins is always adjusted to dashed line.

Package und pin properties can be modified with property sheet shown on the right.





Symbol Graphics

Completion of symbols

Symbol outline is drawn finally. Commands are:

- **Place > Rectangle**
- **Place > Polyline**

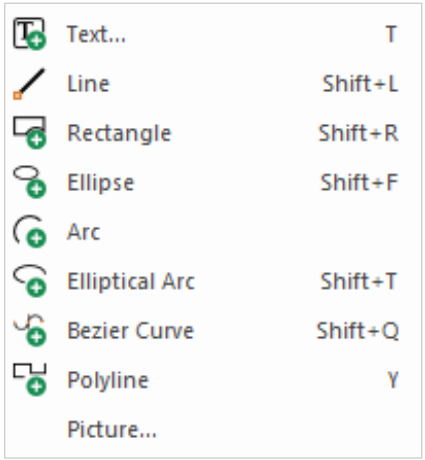
etc.

You can also use icons on right side of the Part Editor window.

Tip

The dashed line is a frame to show real occupied area of the symbol on schematic. This frame is invisible in schematic.

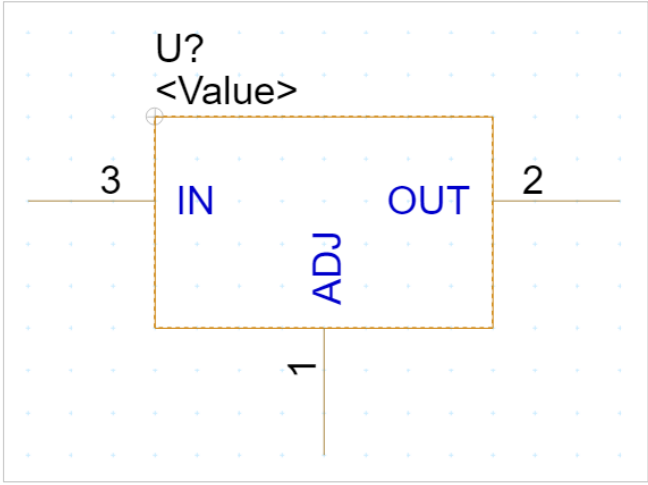
Rectangle can be selected and stretched or compressed over corners. Dashed frame is always at least as large as real symbol body. **Only during symbol body expansion**, the rectangle will be adjusted automatically.



Completed Symbol

Completed symbol LM317

Please do not forget to save!



Tip

Additional text or graphics for this part can be added by **Place > Text** and **Place > Line** commands..

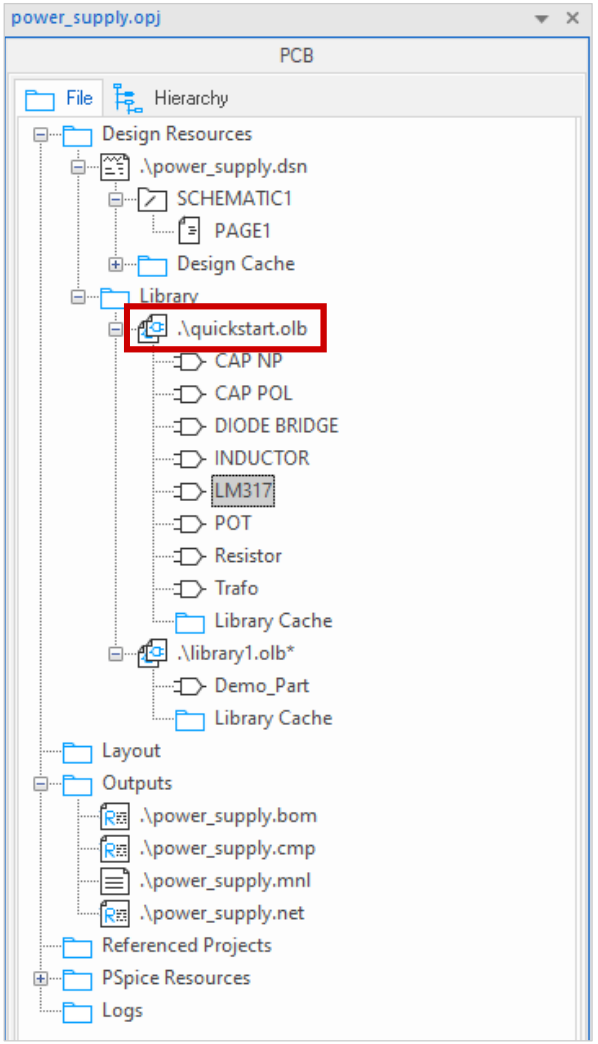
Library Management

Now you can save **library1.olb** as **Power_supply.olb**. Voltage regulator LM317 is now stored in library **Power_supply.olb**.

If you need more parts, you can create them as well in this library.

You can assign this library to design projects as described on page 25.

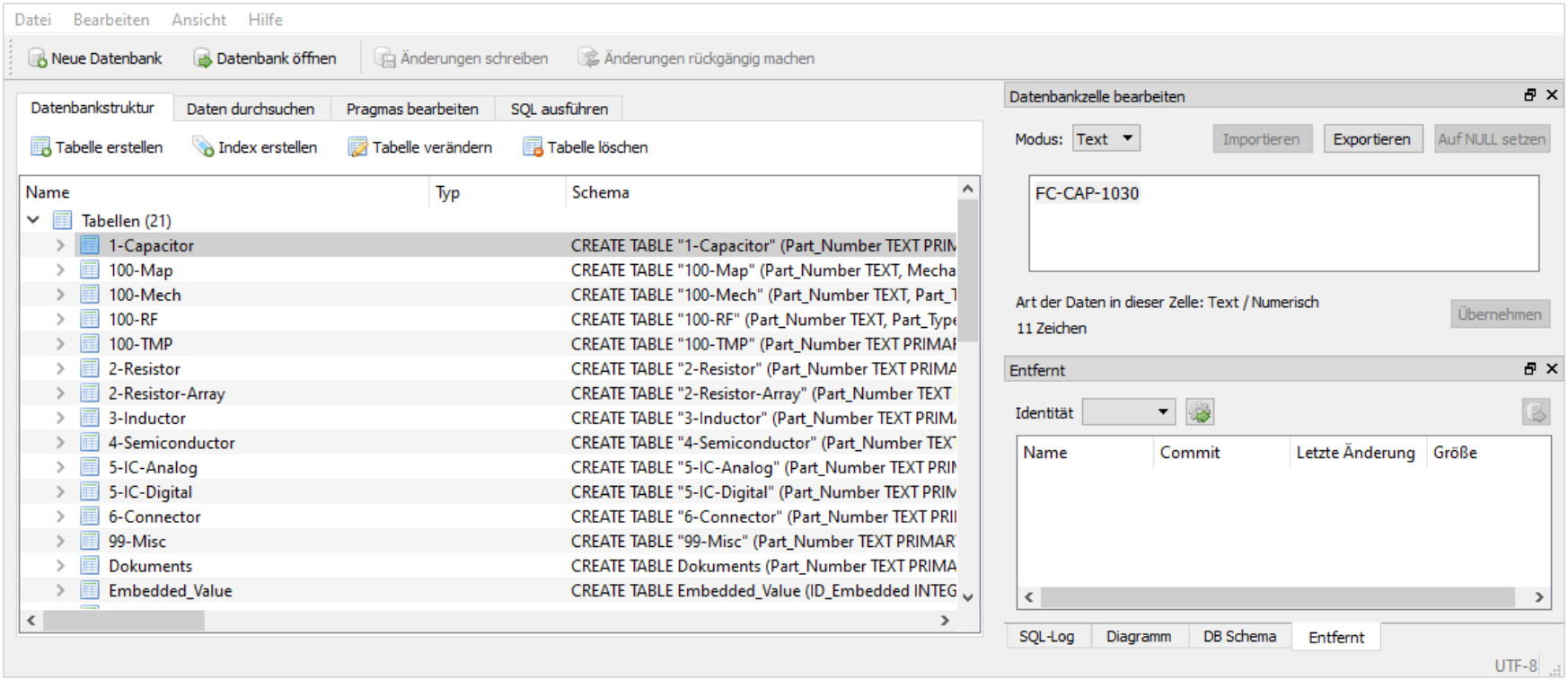
All parts used in this quick start are available in library **Quickstart.olb**.



Edit Database

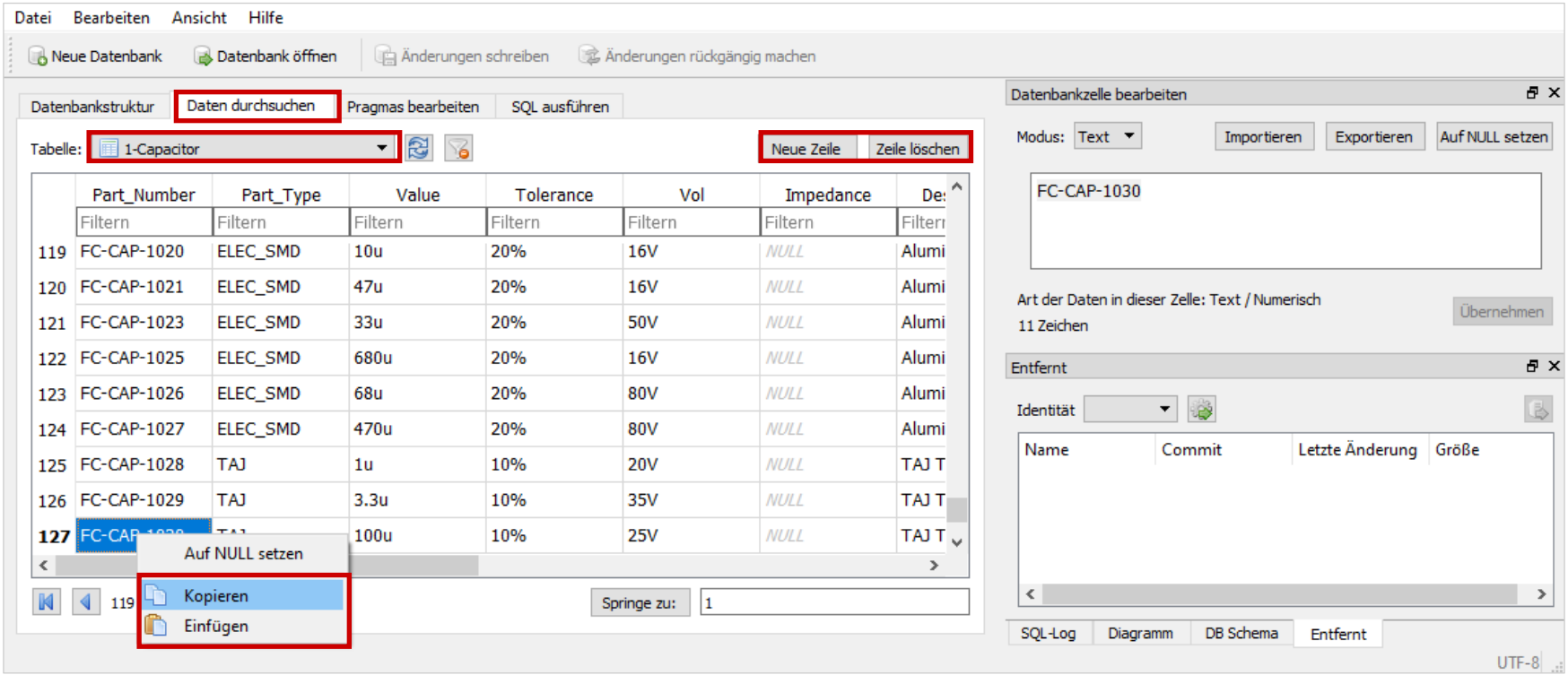
Edit Database (I)

- An SQLite editor is required to edit database entries.
- You can download it here, for example: https://sqlitebrowser.org/dl/



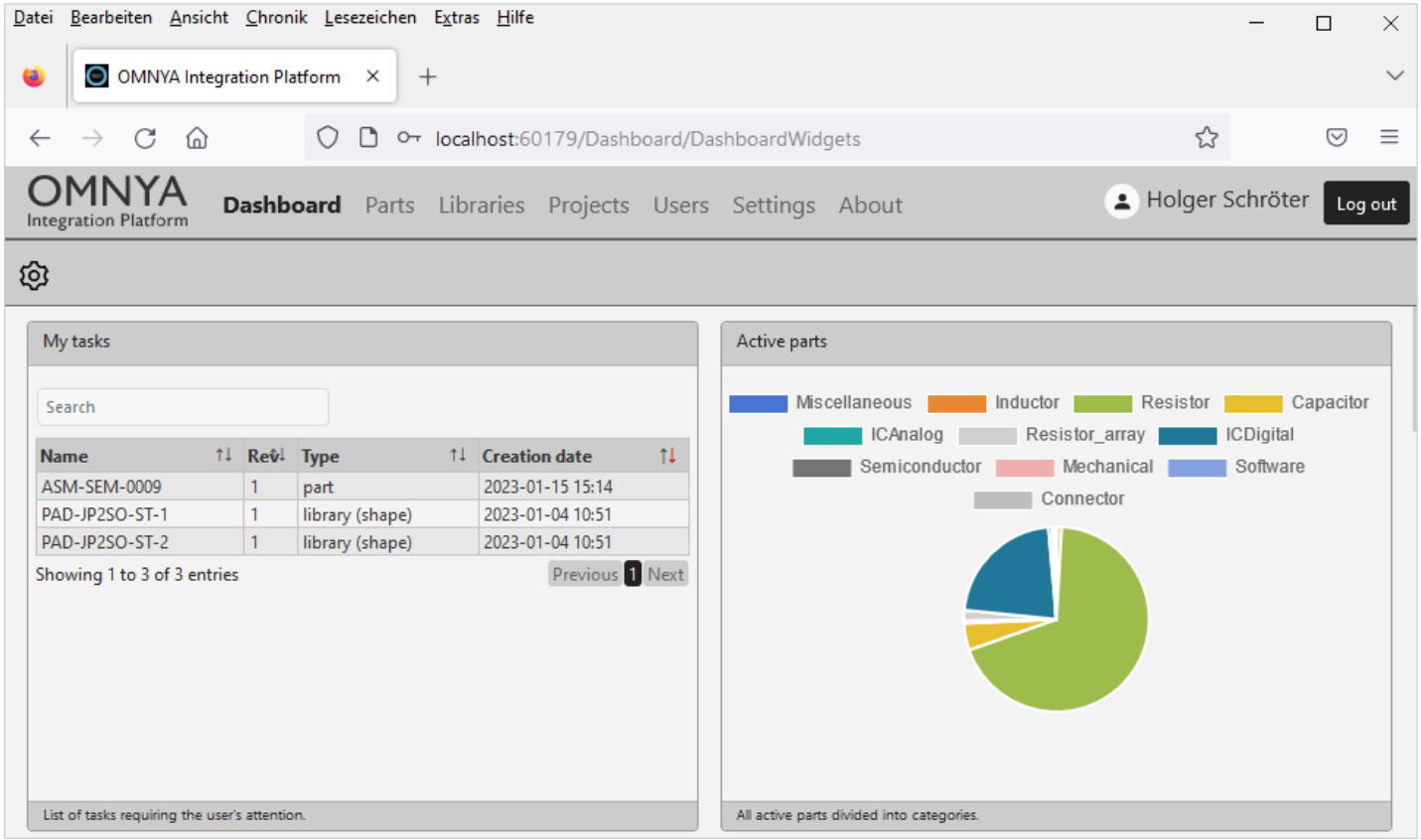
Edit Database (II)

- The desired part table must be opened for editing.
- Commands such as **Copy / Paste** and **New Line** are available there.



FlowCAD Database Solution OMNYA

- FlowCAD offers the database solution OMNYA to manage component date and library elements, as well as project data.



OMNYA dashboard in webinterface

FlowCAD Database Solution OMNYA

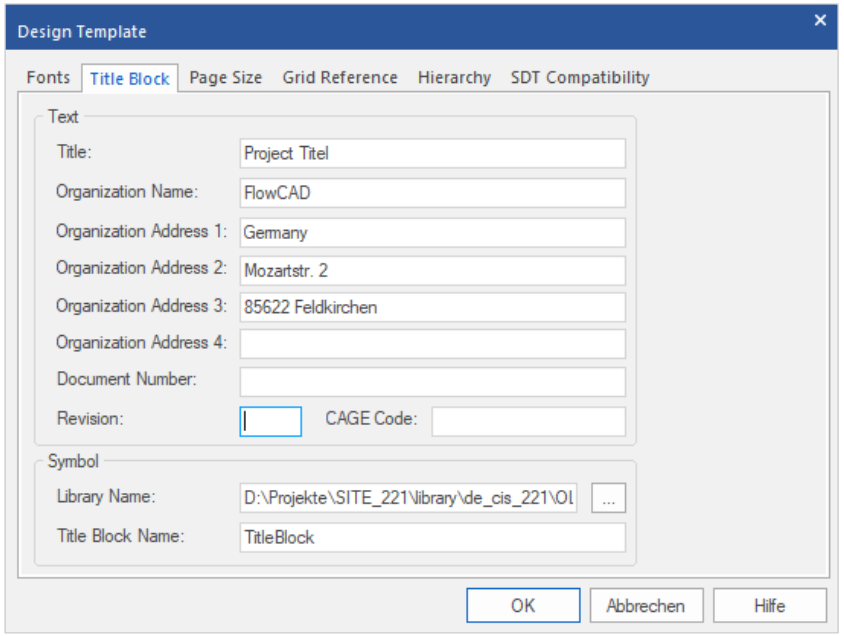
- Among others OMNYA offers these possibilities for component management:
  - Release of new components / changes by four-eye principle
  - Version control and “Where Used” feature for components
  - Generic parts, several MPN per part, alternate parts and part assembly
  - Integration of component distributors
  - Integration in OrCAD Capture
- In addition OMNYA offer:
  - Project und BOM Management
    - Repository for project data
    - Version control of data and Check-OUT, Check-IN
  - Library Management
    - Version control of schematic symbols and PCB footprints
    - “Where Used” functionality for library elements including hierarchical objects (e.g. padstacks, shapes)
- More details about OMNYA at https://www.flowcad.com/de/omnya-integration-platform.htm

Settings and Templates

Design Template – Title Block

You can define pre settings for new projects and new pages in an existing project under **Options > Design Template.**

In tab **Title Block** you are able to define text modules of the title block.

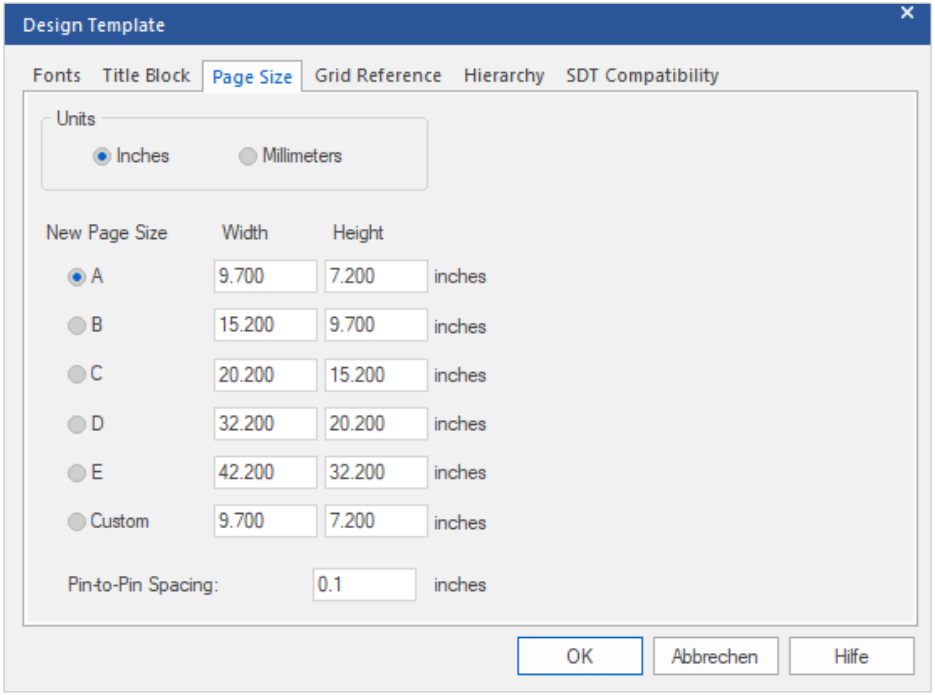


Design Template – Page Size

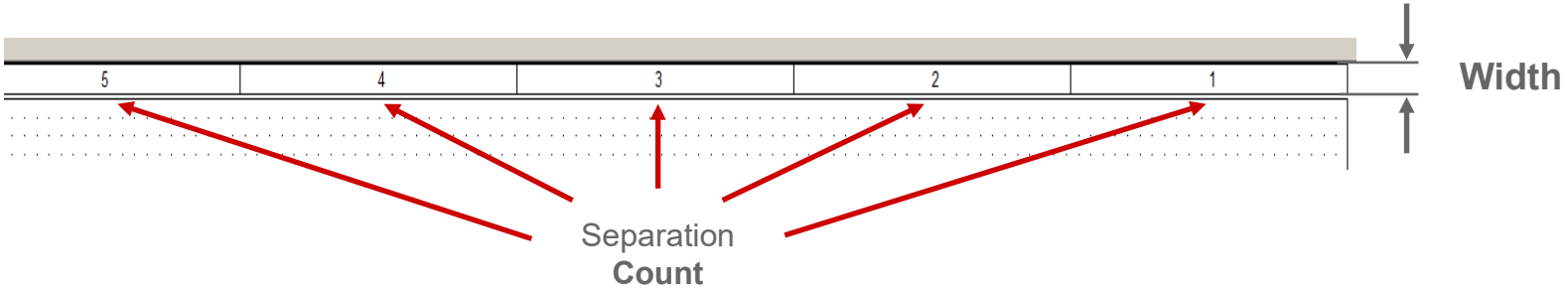
Size of schematic sheet can be defined at tab **Page Size.**

Attention

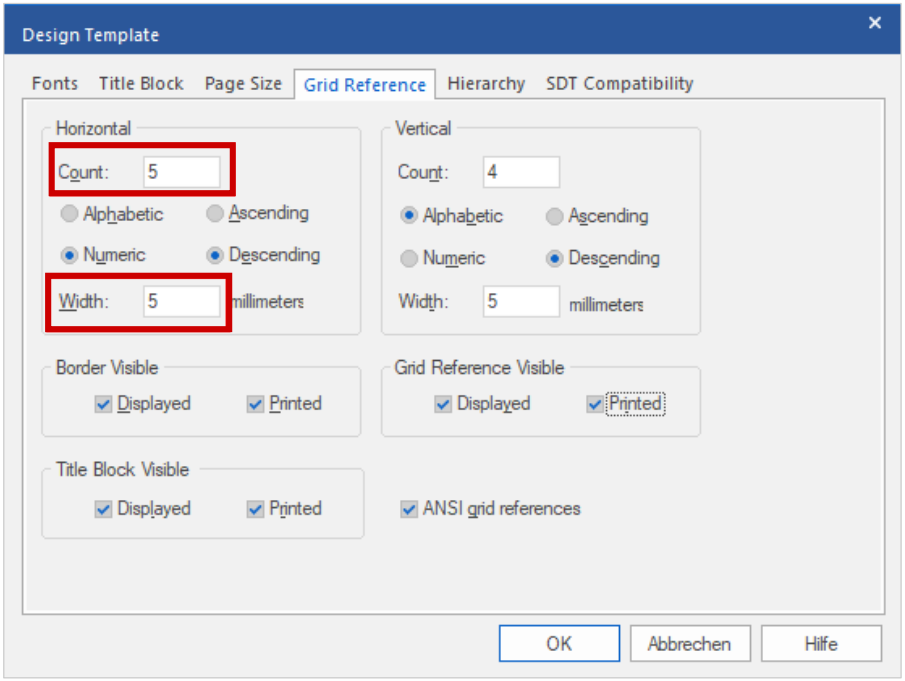
Settings under Pin-to-Pin Spacing must match pin-to-pin spacing used in library. This prevents off grid connection problems later.



Design Template – Grid Reference



Grid Reference defines width of drawing frame as well as count of horizontal and vertical segments.



Appendix

System Requirements (Full Version 22.1)

Operating Systems	Windows 11 Professional and Enterprise Windows 10 (64-bit) Professional and Enterprise, including Dark Theme mode; Windows Server 2016 (All Service Packs) Windows Server 2019
Hardware	Intel® Core™ i7 4.30 GHz or AMD Ryzen™ 7 4.30 GHz with at least 4 cores Note: Faster processors are preferred. 16 GB RAM 50 GB free disk space (SSD drive is recommended) 1920 x 1200 display resolution with true color (at least 32 bit color) A dedicated graphics card supporting OpenGL, minimum 2 GB (with additional support for DX11 for 3D Canvas) Dual monitors (for physical design) Broadband Internet connection for some service Ethernet port / card (for network communications and security HostID) Three-button Microsoft-compatible mouse

Files in OrCAD Capture

Most important files used by OrCAD Capture:

.OPJ	Project
.DSN	Design
.DBK	Backup
.OLB	Symbol Library
.UPD	Property Update File
.DRC	Design Rules Check
.BOM	Bill of Materials
.EXP	Property Export File
.MNL	Layout Netlist
.SWP	Layout Backannotation
.VHD / .VHO VHDL	Source
.EDF / .EDN EDIF	Netlist or Backannotation
.XRF	Cross Reference
.NET	Other Netlist

Note

- Only design file ***.dsn** is necessary for sharing or external editing of designs. All design data is included in this file.
- Project-File ***.opj** is meaningful, but not necessary. It contains pre-settings of project as well as used libraries, output files, folder settings, etc.

Contact us / Kontakt zu FlowCAD

Please do not hesitate to contact us.
Für weitere Fragen und Informationen stehen wir gerne zur Verfügung.

FlowCAD Deutschland
Mozartstr. 2
85622 Feldkirchen bei München
T +49 89 45637-770
info@FlowCAD.de

FlowCAD Schweiz
Hintermättlistr. 1
5506 Mägenwil
T +41 56 485 91 91
info@FlowCAD.ch

FlowCAD Polska
ul. Sąsiedzka 2A
80-298 Gdańsk
T +48 58 727 90 90
info@FlowCAD.pl

Follow Us

» FlowCAD.com

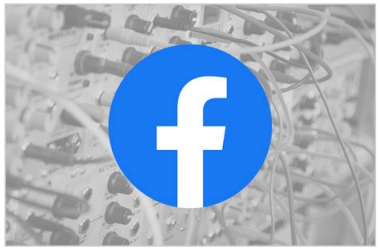
» FlowCAD.com/newsletter

» youtube.com/FlowCAD

» linkedin.com/FlowCAD

» twitter.com/FlowCAD

» instagram.com/FlowCAD



» facebook.com/FlowCAD

» xing.com/FlowCAD

Don’t forget to subscribe, share and like!

FlowCAD