FlowCAD

Quickstart OrCAD Capture

Version 17.4

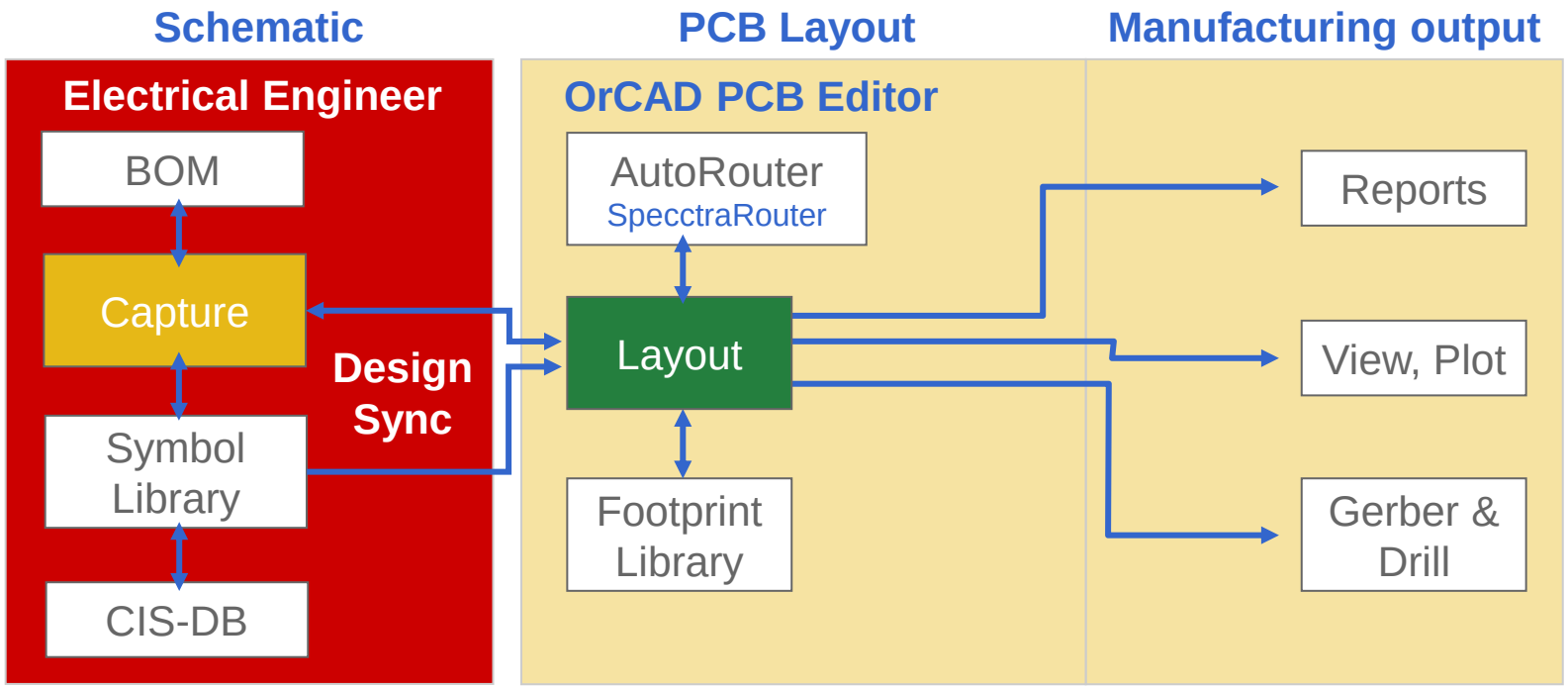
Table of Contents

Introduction

Introduction

- 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 7.
- All commands and functions used in this tutorial are available and licensed with this trail version.

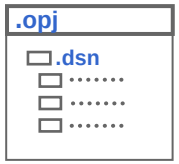
OrCAD PCB Design Flow



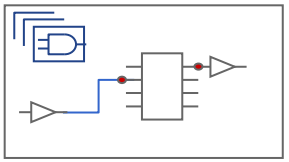
- As you can see, OrCAD PCB Designer Flow consists mainly of two parts.
  - These are the schematic capture module **Capture** and the layout module **OrCAD PCB Editor**.
- Both modules are supplemented by additional sub packages who represent in each combination an ideal tool, enabling the user to complete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

Capture Design Flow

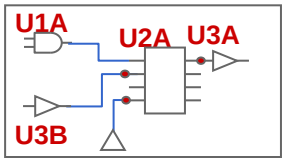
Create a new project

1 

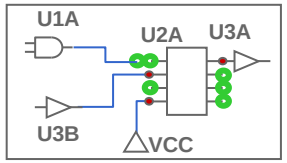
Place and connect parts

2 

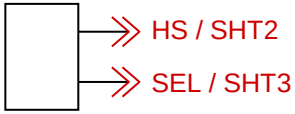
Assign part references

3 

Check design rules

4 

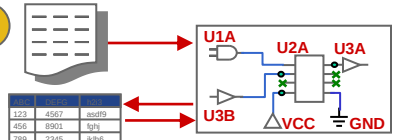
Add inter-sheet references

5 

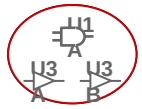
Cross reference report

6 

Edit part and net properties

7 

WIDTH = "40"



COMPGROUP = "1"

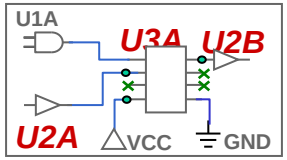
Design Sync with PCB

8 

Generate Bill of Materials

9 

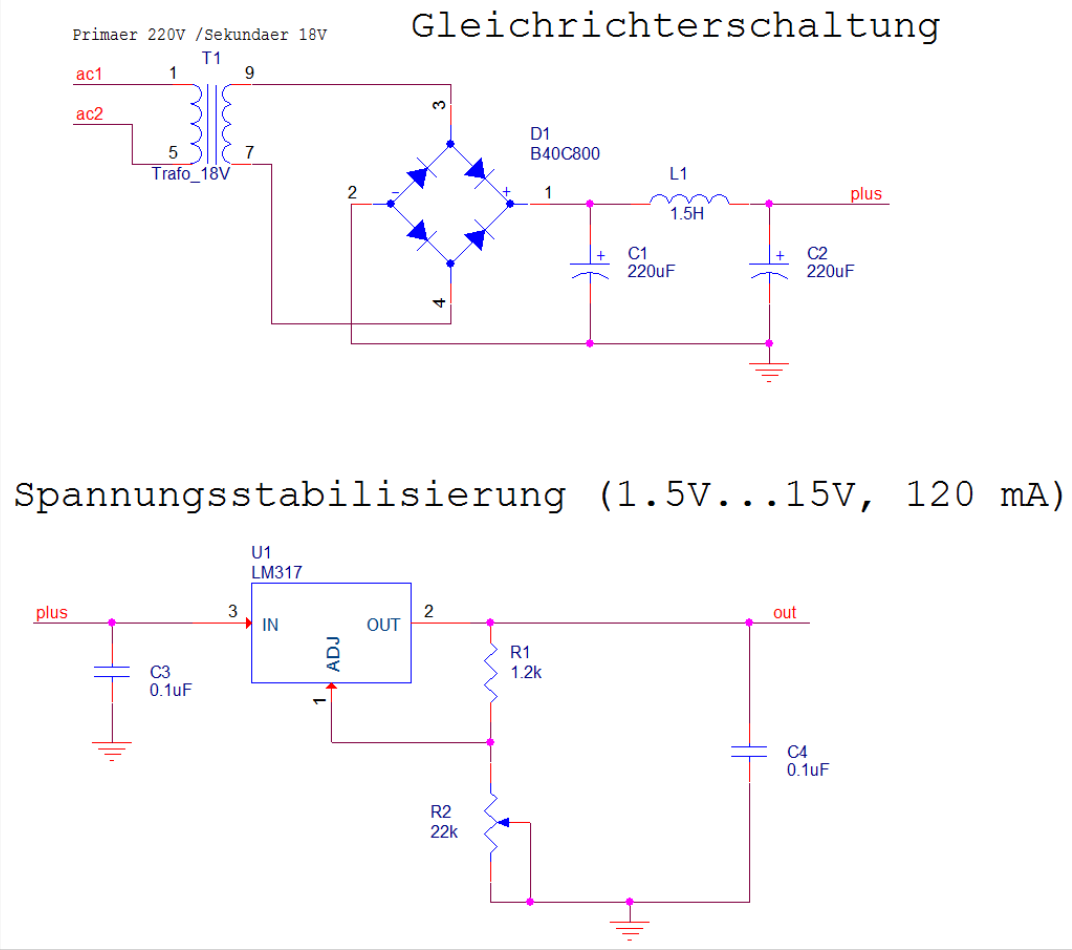
Backannotate from PCB design

10 

The graphic shows a typical flow, creating a schematic, following by PCB layout design. Numbers 1 to 10 represent the individual steps in the design process, where we will focus on next. Process starts with project creation and will end with handover to PCB layout.

Schematic Template

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



Schematic Template – Parts List

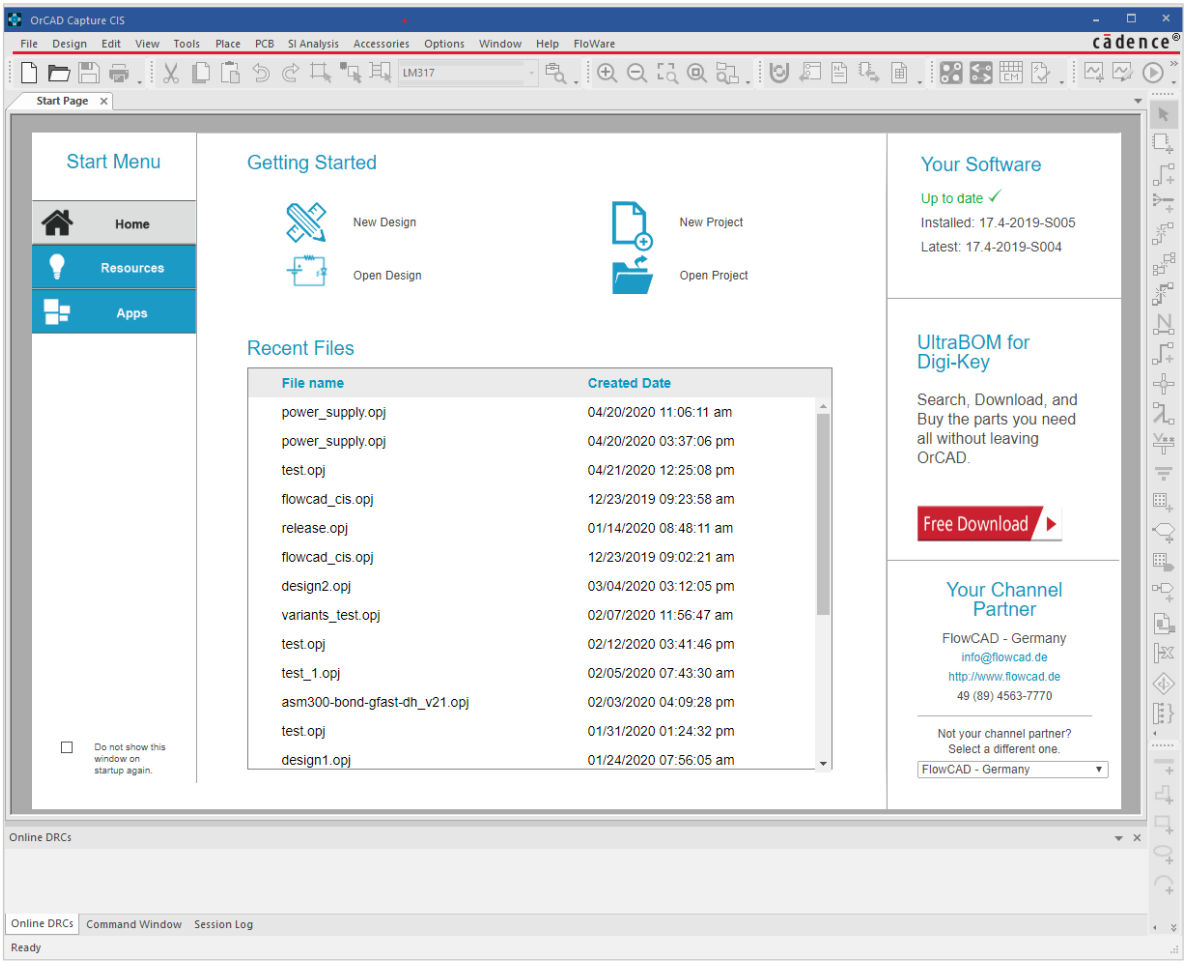
The circuit to be implemented in this quick start contains following parts:

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

Start of Capture

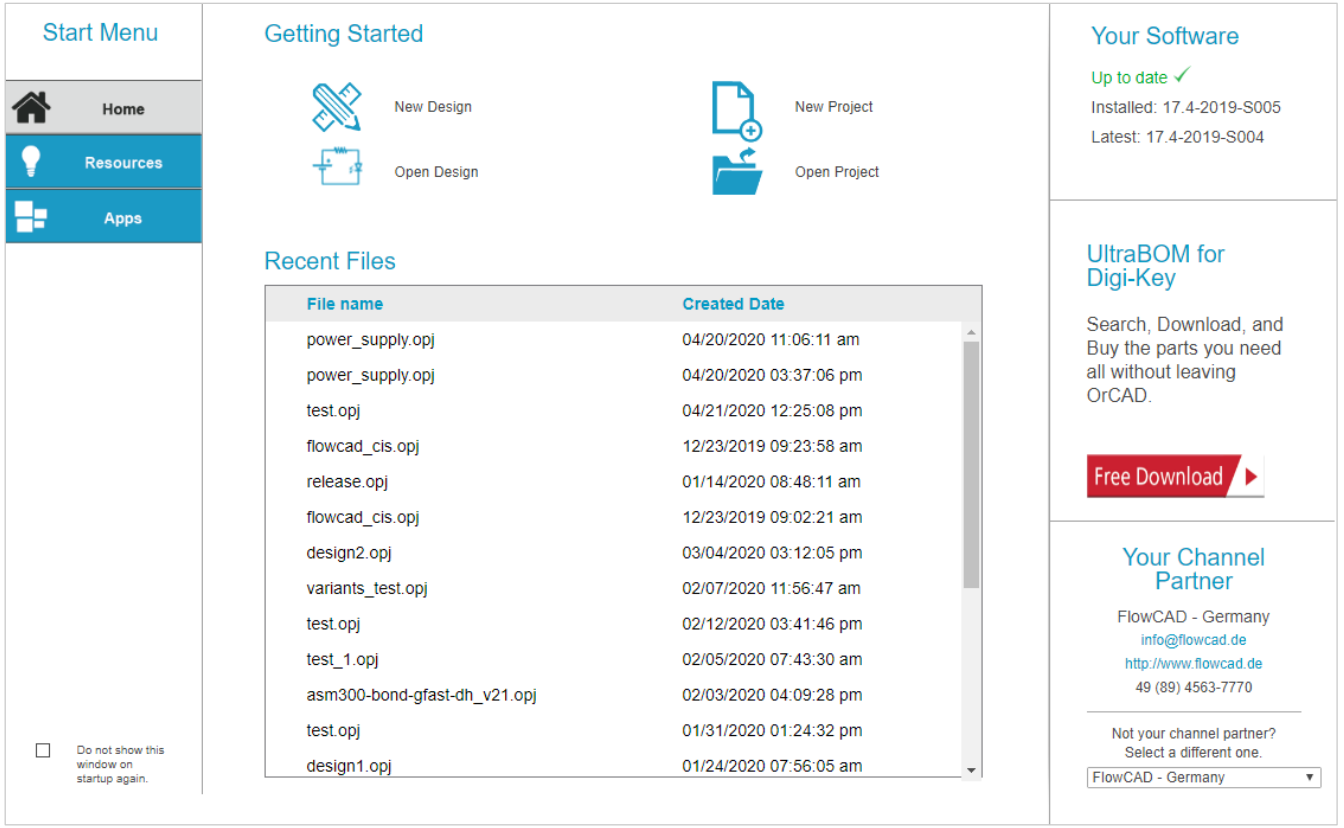
When starting Capture, Capture **Session Frame** window will open.

Start via:
Start >
Cadence PCB 17.4 2019 >
Capture CIS 17.4



Start Page

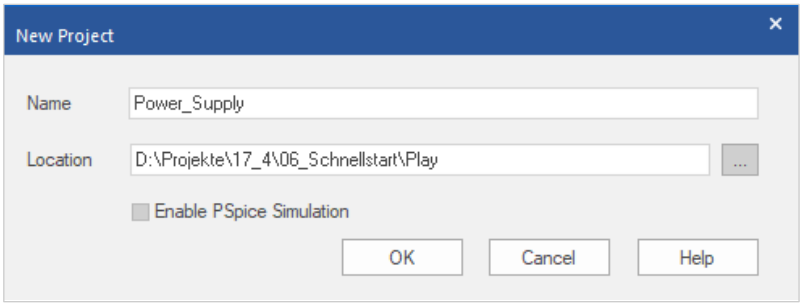
During start, interactive start page gets loaded to open existing projects or to create new projects. Furthermore, you receive additional information such as application notes, training data, etc.



New Project Setup

Lab: New Project Creation

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



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

For project type choose **PC Board Wizard**, because we also like to create a PCB design with this project.

In bottom field **Location,** please choose the folder in which new project will get 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

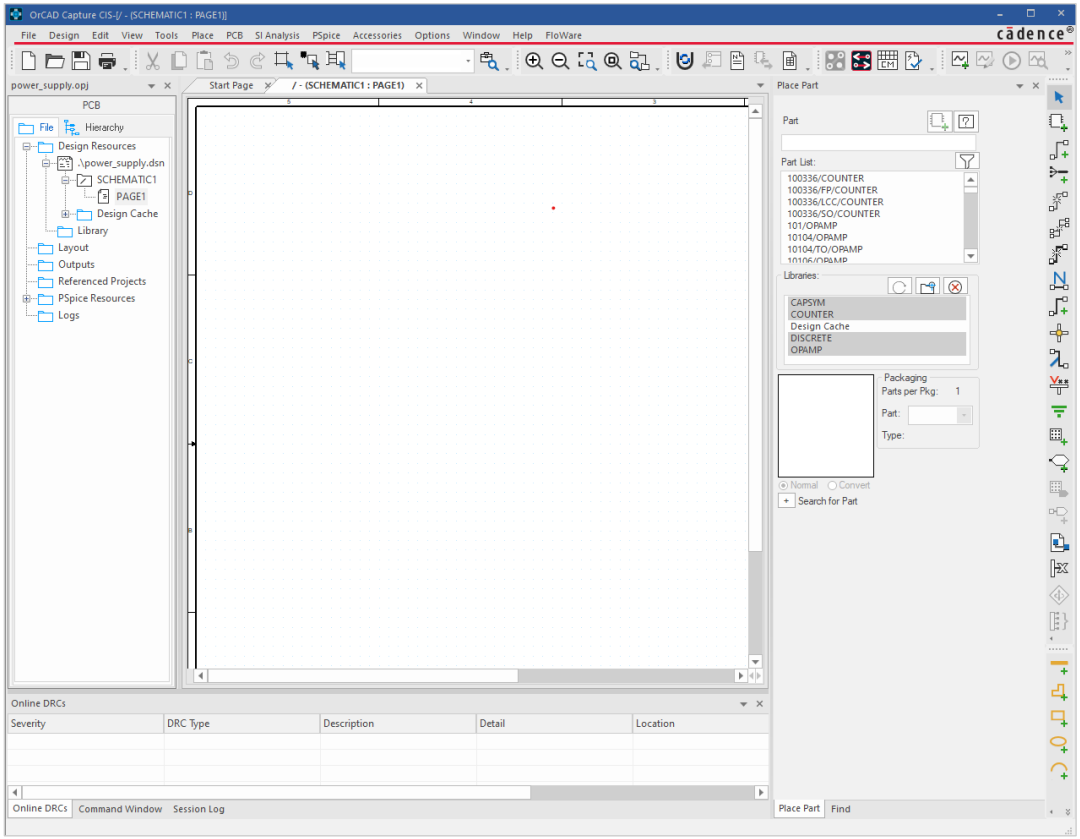
A new project with name **Power_supply** and a design with same name **power_supply.dsn** is created.

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

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

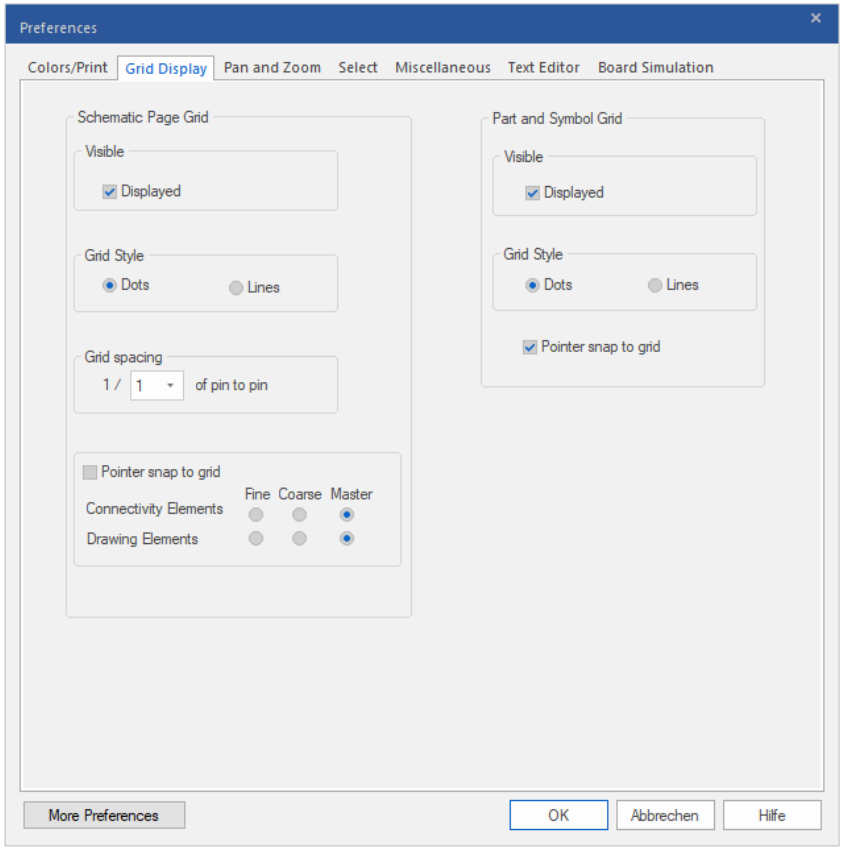
Folder structure is virtual, meaning it does exist only inside project manager. **PAGE1** under **schematic** can only be found within **power_supply.dsn** file.

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

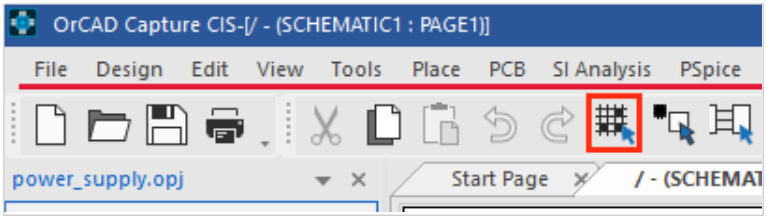


Setup – Grid Display

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

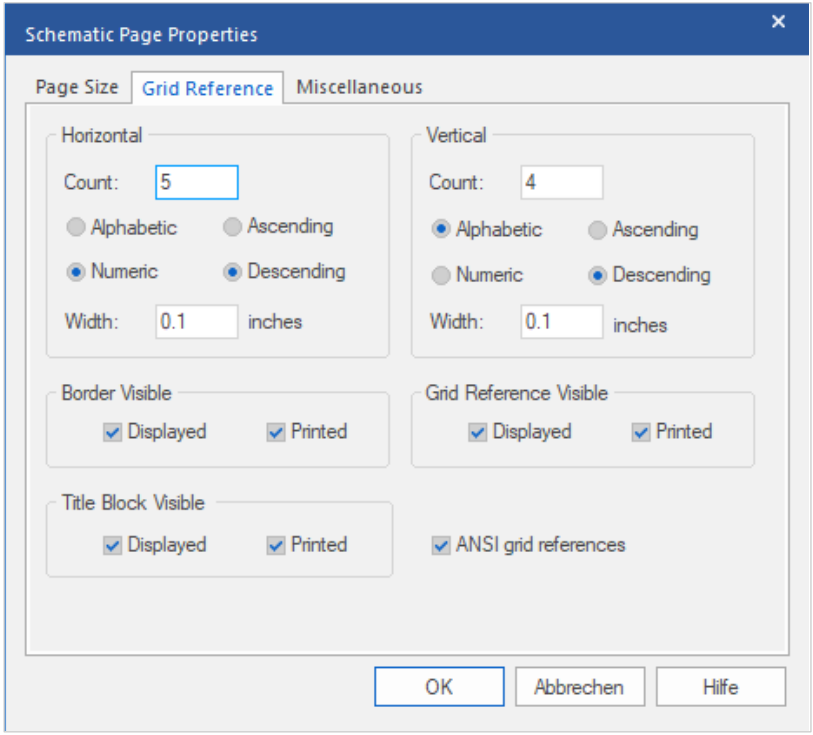


To avoid placing parts outside grid, please activate **Snap to Grid**:



Setup – Drawing Frames

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



Tip

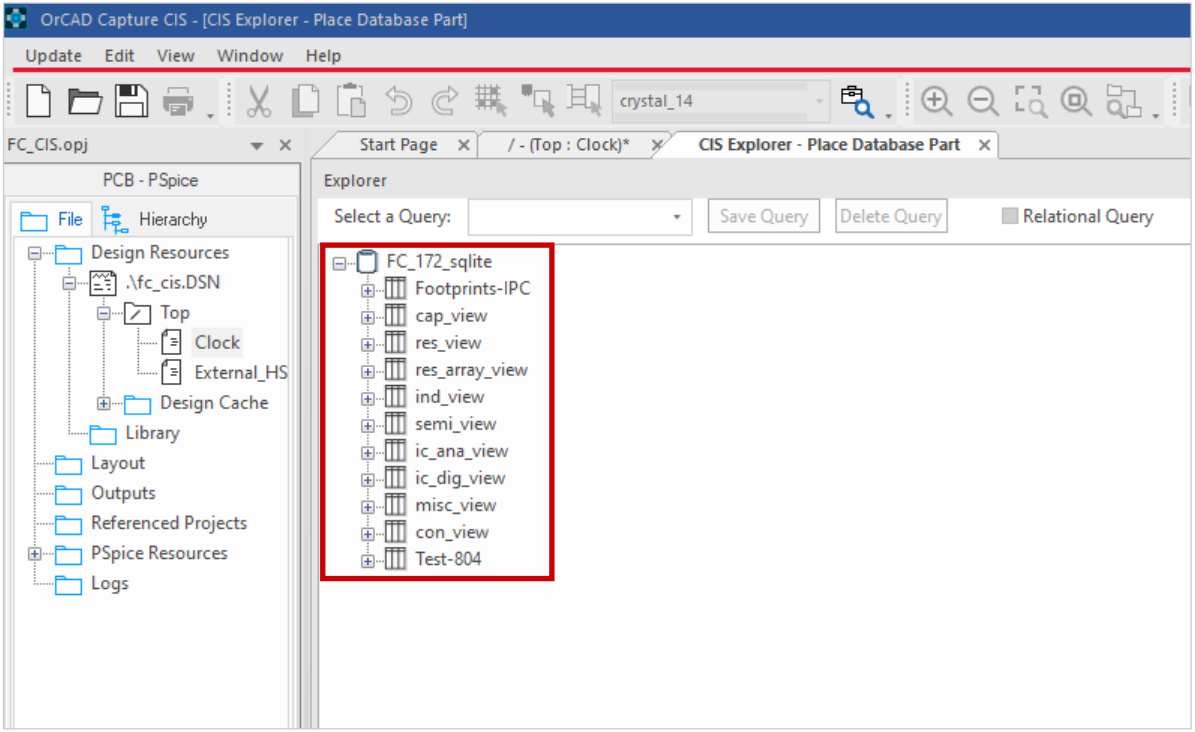
Settings can also be saved to a template that will affect all new pages and projects.
For this purpose, settings can be made under **Options > Design Template.**

More details can be found in chapter Settings and Templates starting page 82.

Part Placement

Part Search via Database (I)

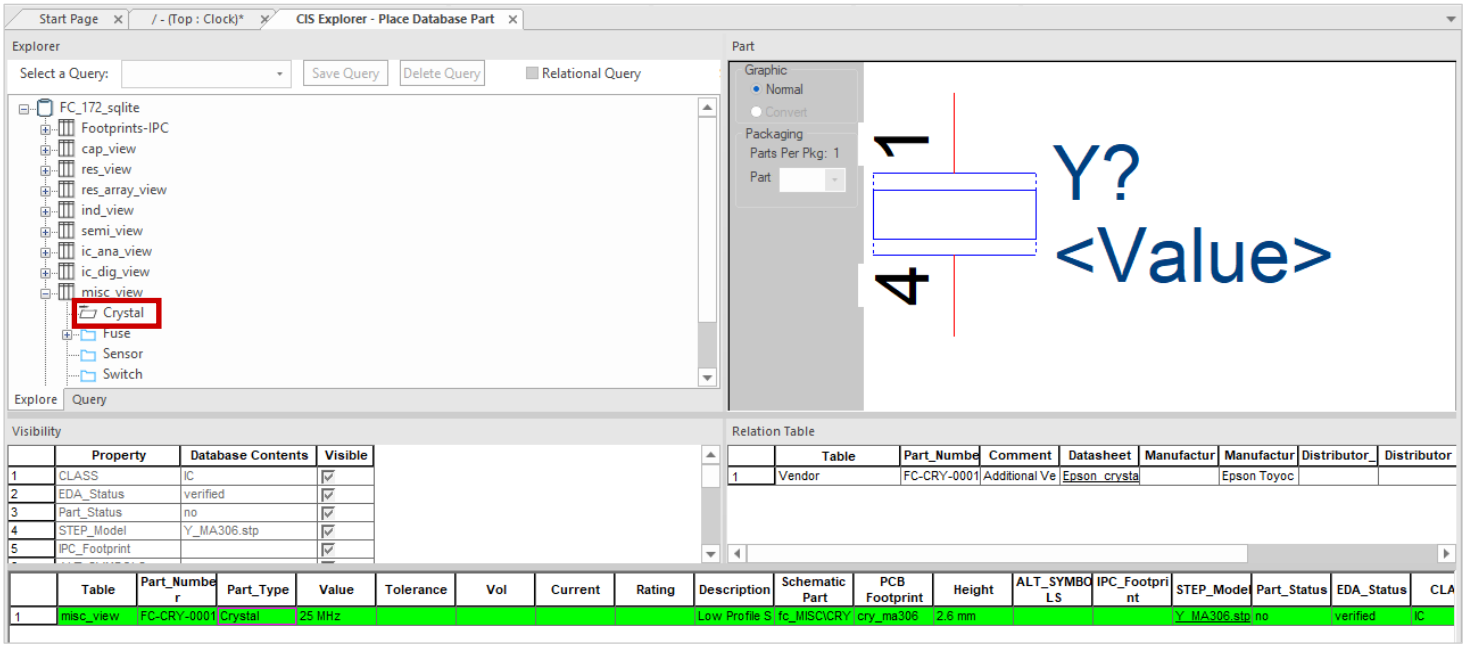
- CIS Explorer can be started via pull-down menu with **Place > Database Part** or via keyboard shortcut **Z**.
- Different database tables can be found in upper left corner.



Part Search via Database (II)

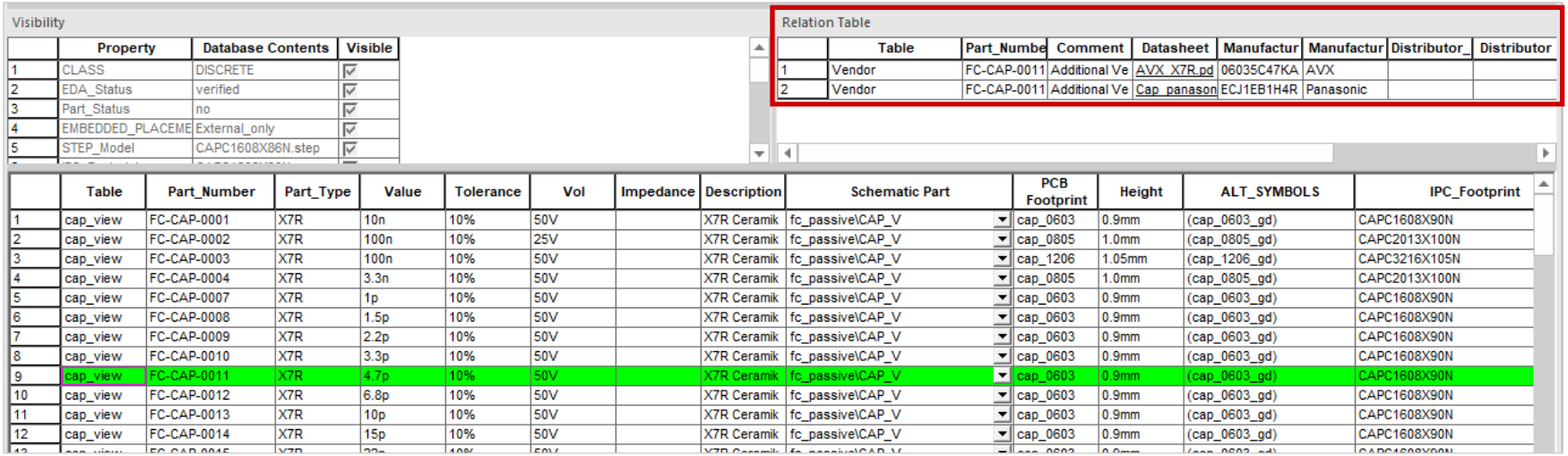
- Please open **misc_view** and select folder **Crystal**. A table entry appears in lower window. If this is selected, complete preview of part is opened (arrangement of windows can be designed individually). Entry under Datasheet is a link to data sheet of manufacturer. Double-clicking green highlighted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

- 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 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	☑

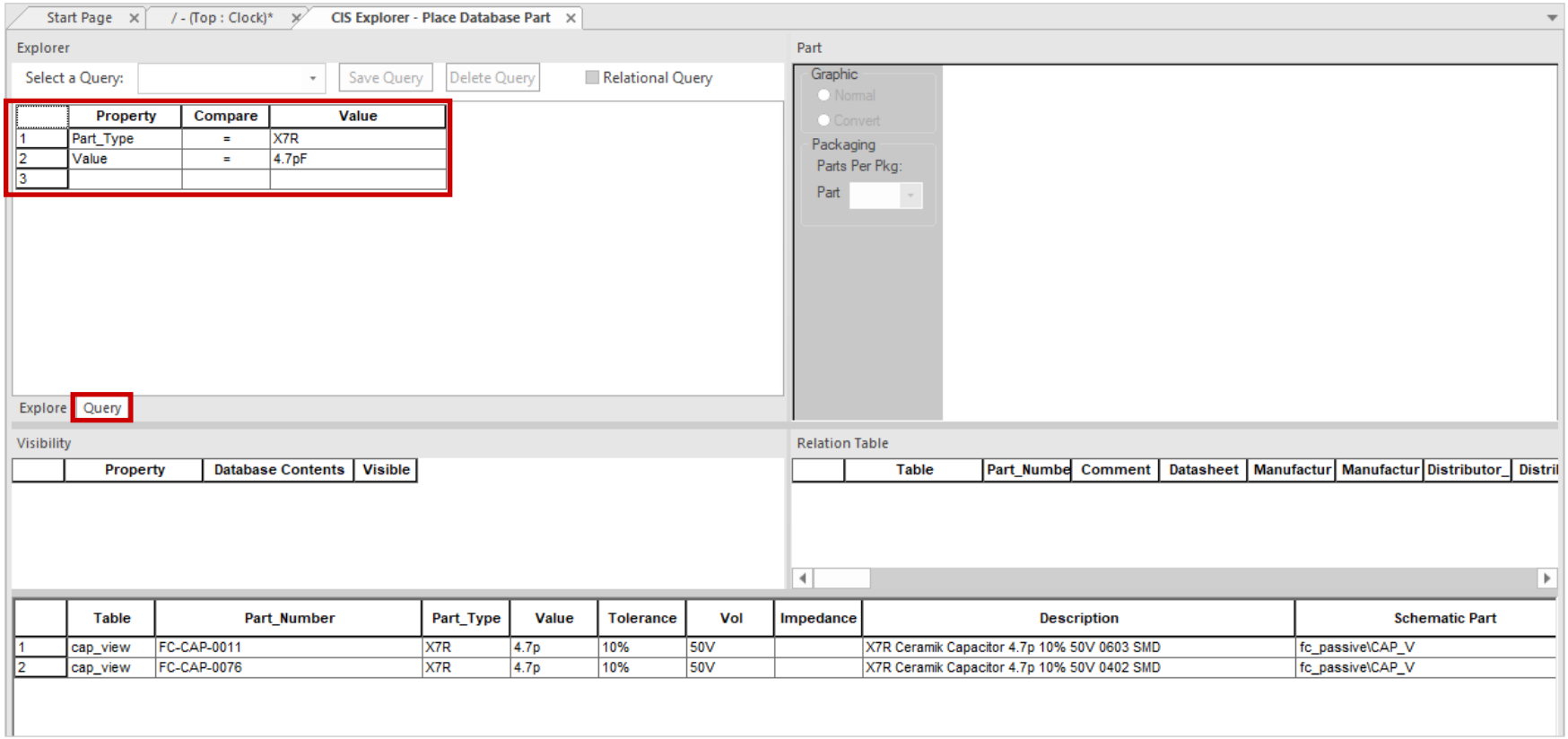
Relation Table

	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

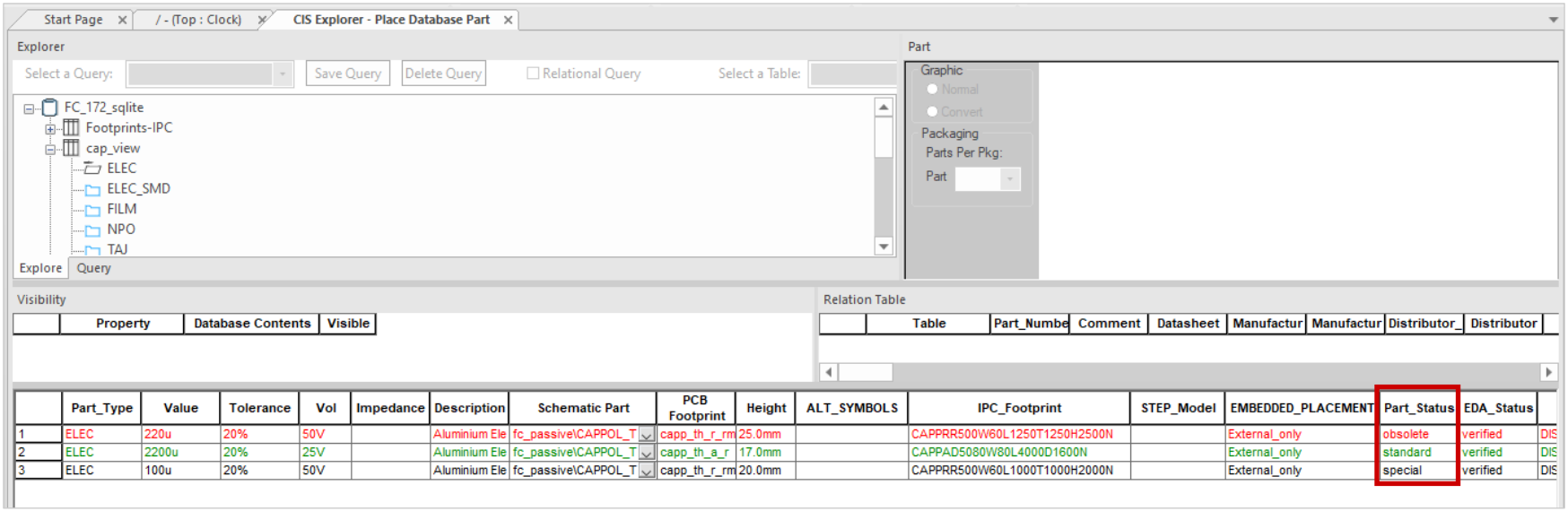
Part Search via Database (IV)

- 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
  - Locked parts are red
  - Special parts are black



Part Placement without Database

Part Placement from Library (I)

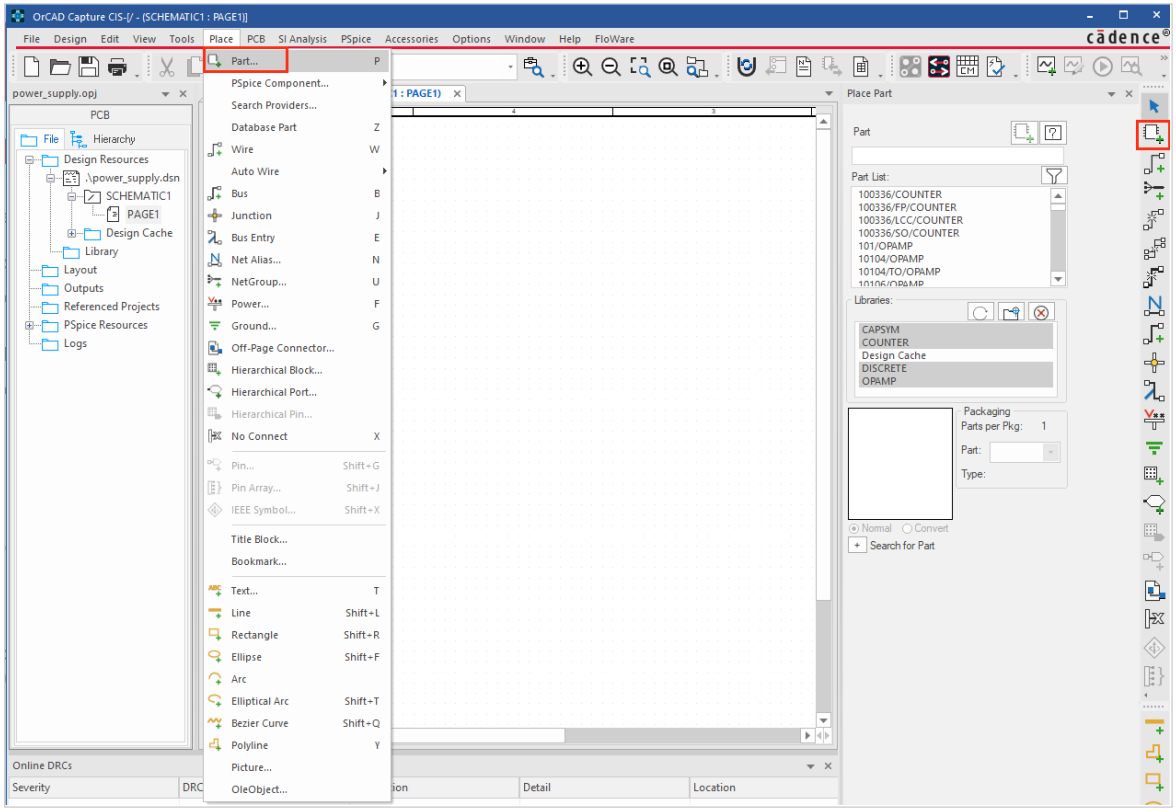
Besides the possibility to select parts from a database that provides metadata, OrCAD Capture also offers to select and place parts directly from a library (.olb).

- 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

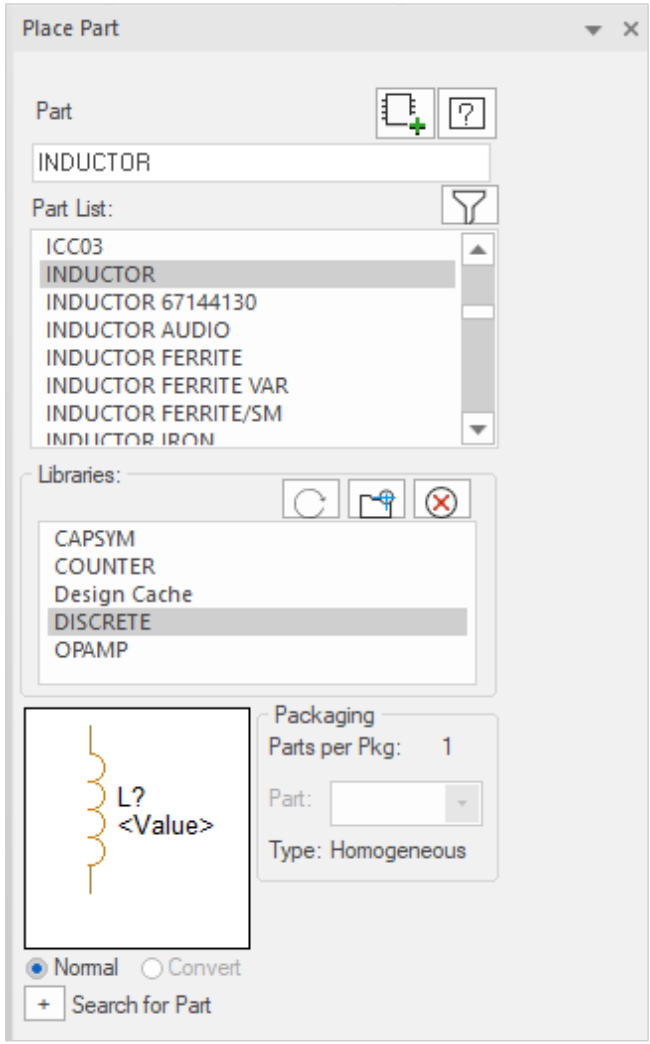


Tip

One of the schematic pages must be active to make icon panel visible (right margin).

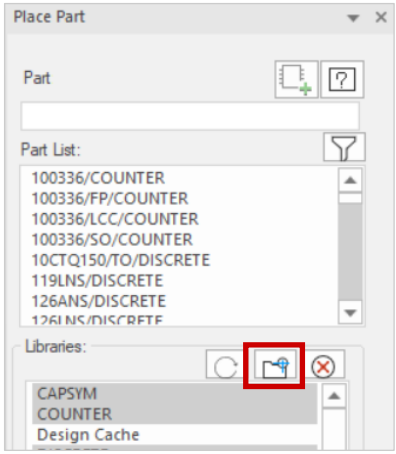
Part Placement from Library (III)

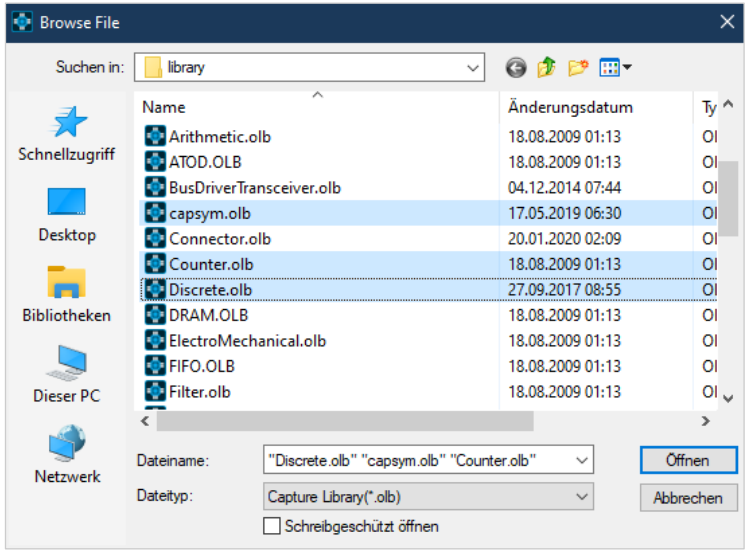
- Placement command opens menu as shown.
- In **Libraries** you can select one or more libraries in which your part can be searched.
- Enter part label in **Part**. This already behaves as a filter, but without usage of wildcards “*”.
- Search results and related library are visible in **Part List**.
- **Search for Part** allows a search including wildcards.
- **Add Library** allows to add libraries to search path.
- **Packaging** indicates if a part consists out 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 assigned.
- Left window contains all libraries installed during software installation. Additional libraries can be added at any time. Libraries assigned to project are listed in right window.
- Select the desired library and press **Add** button for assignment.
- How to define new parts that cannot be found in any existing libraries is described in chapter Libraries starting page 62.





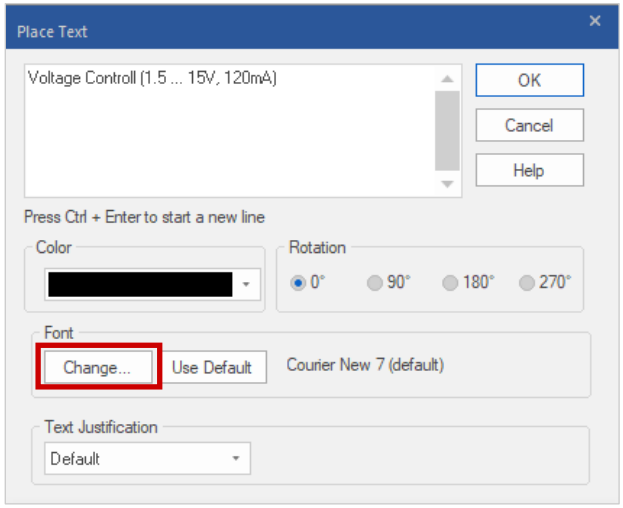
Adding / Edit Text

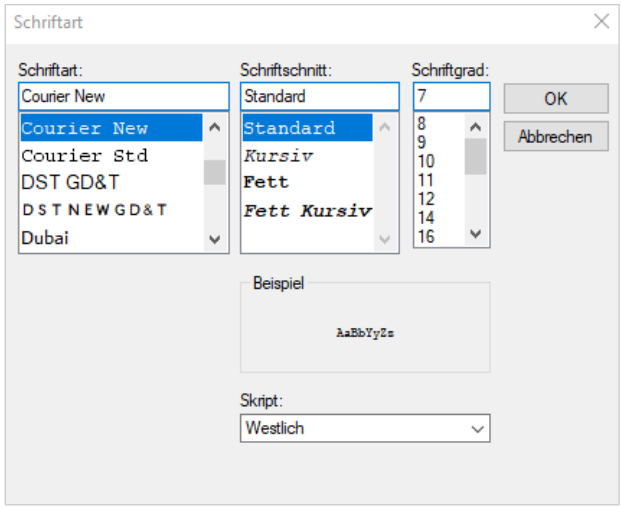
Text – Adding and Edit

Pure text notes can be added to schematic with

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

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





Connections

Connections (I) – Auto Wire

Two Points

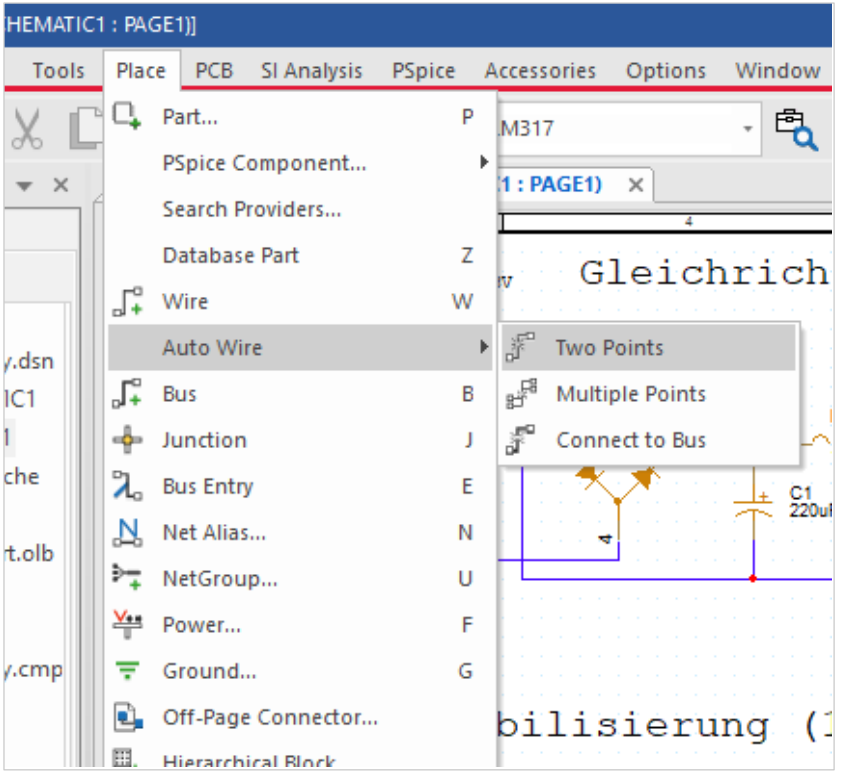
– Two selected pins are automatically connected to each other.

Multiple Points

– Multiple selected pins are automatically connected to each other.

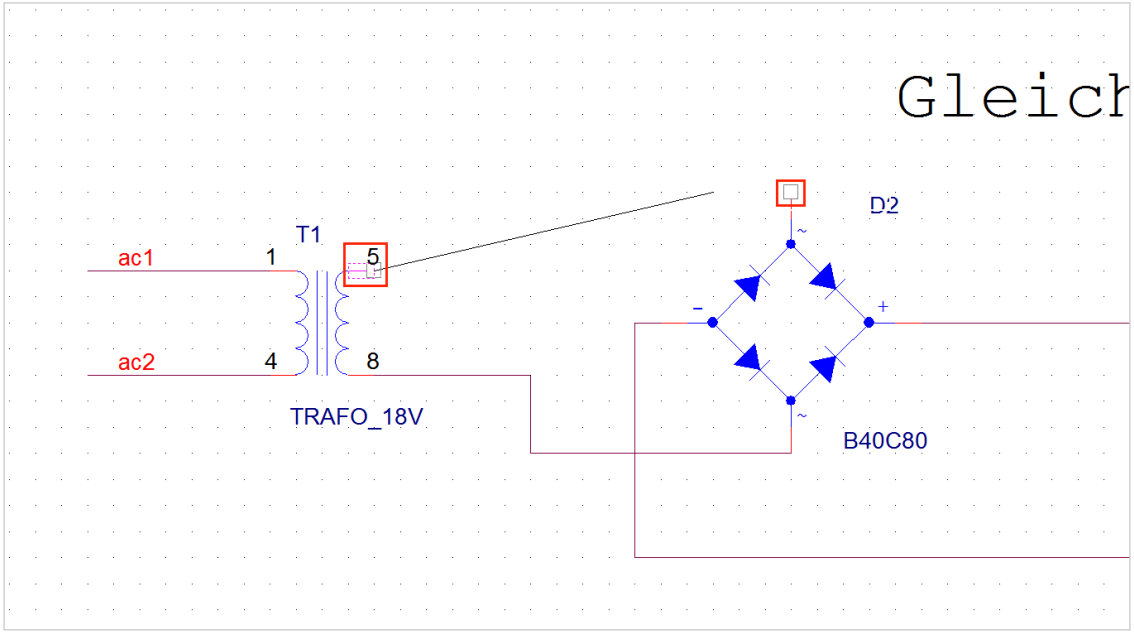
Connect to Bus

– Multiple selected lines are connected to a bus.



Connections (II) – Auto Wire

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



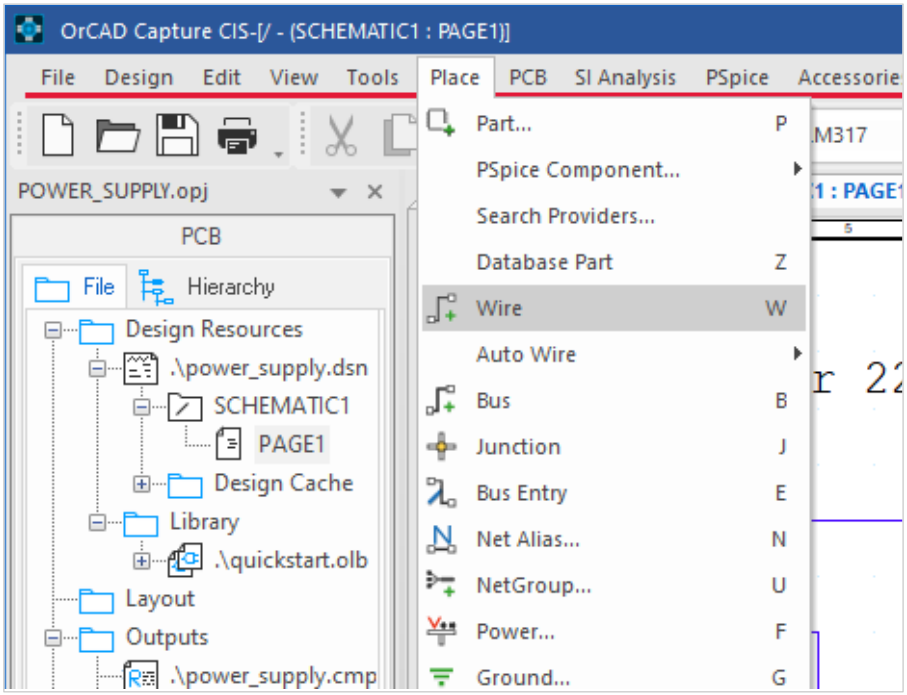
Connections (III) – Manually

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 Auto Wire command.



Connections (IV) – Method

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

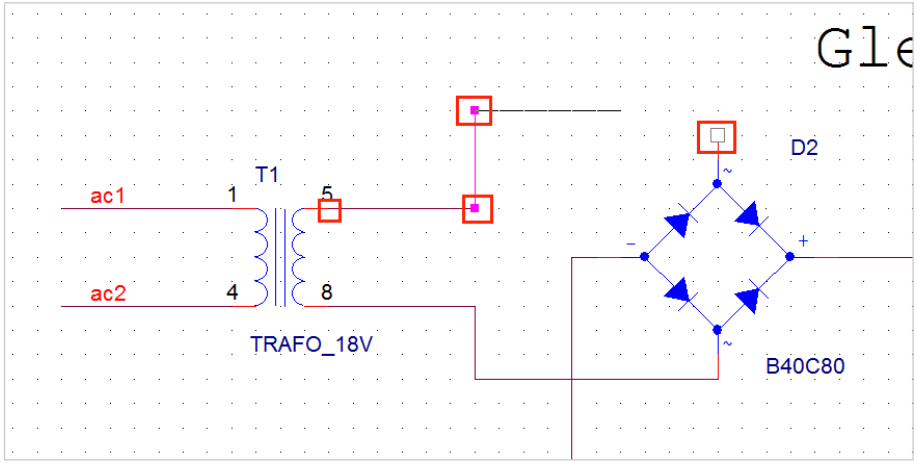
Red dot signalizes a pin which is available for a connection. **Another click** on this object completes connection. Red dot and square will disappear. A **double click** will complete the task and release all objects.

Cross sections of nets represents a connection only if a **dot** or a **junction** gets 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

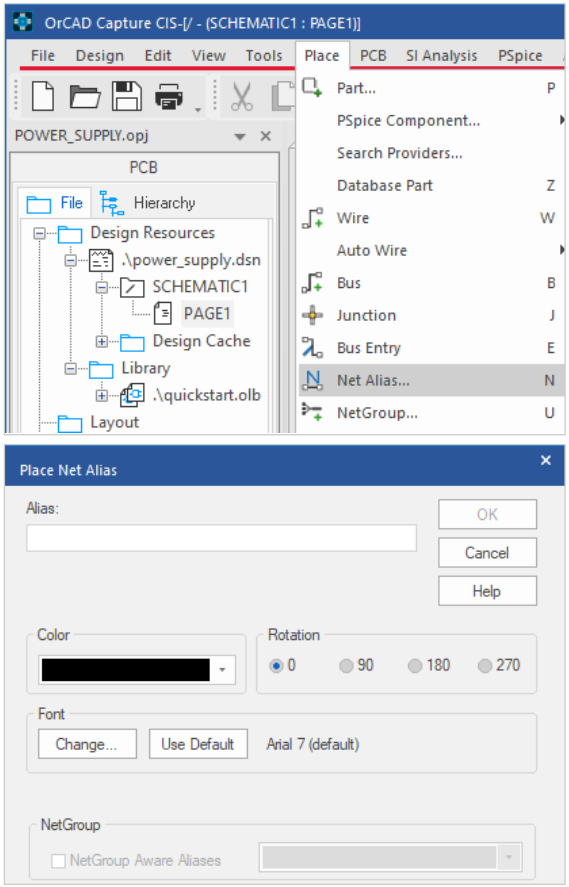
To connect parts on one schematic page, **Place > Wire** command is used to perform wiring.

Another option is usage of **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.



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.

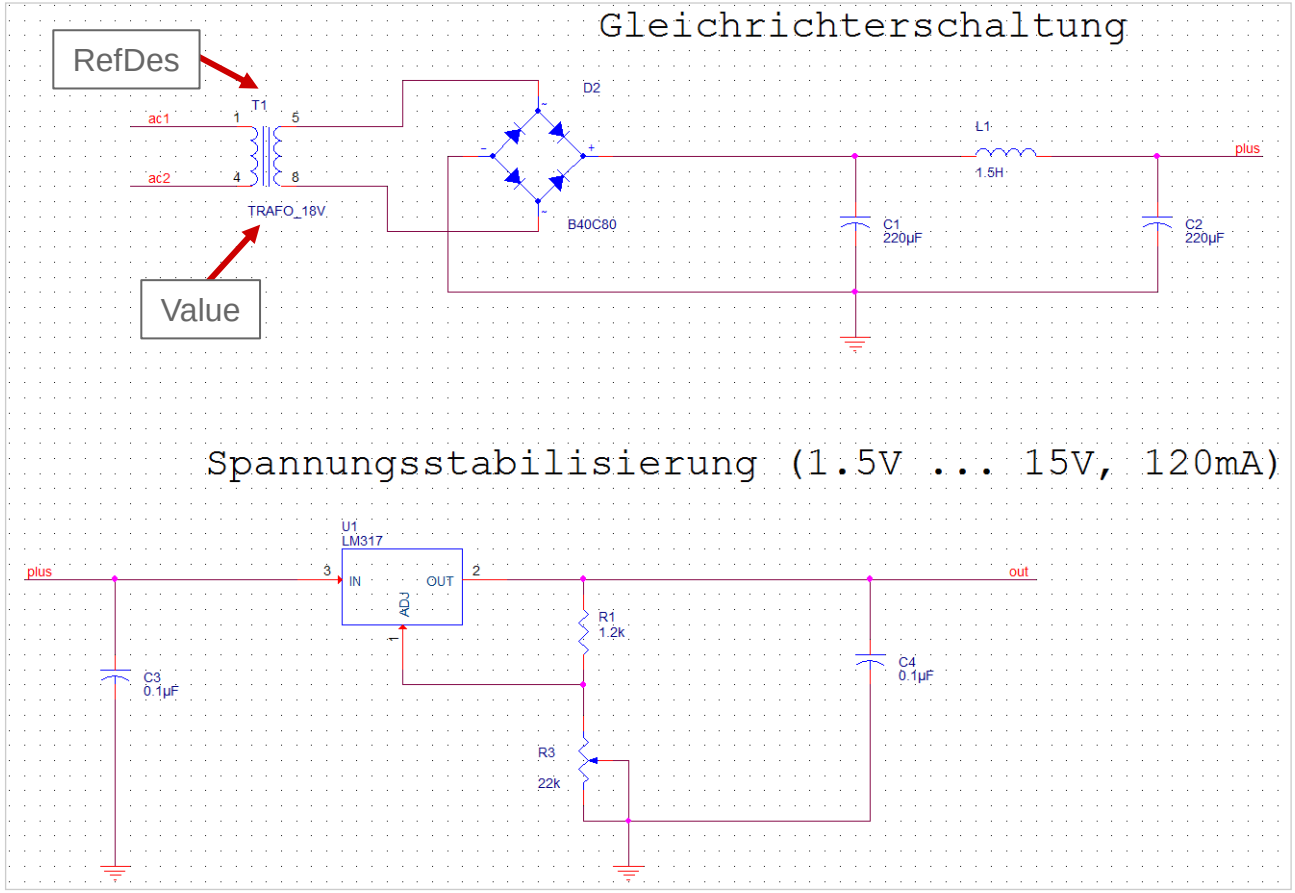
Property Editing

Edit Properties (I)

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

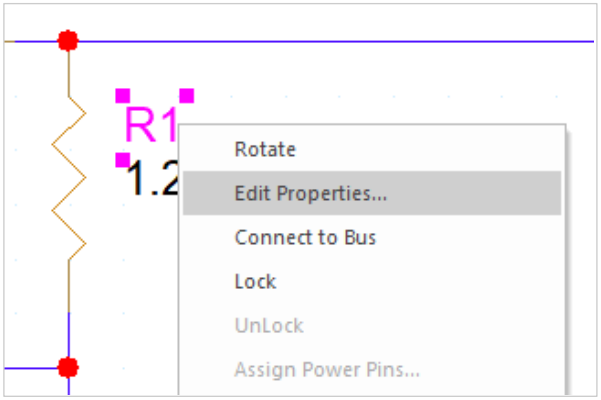
It is possible that labeling of parts (RefDes value) is not matching schematic template on page 7.

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



Edit Properties (II)

Edit REFDES and value



Property window can be opened by

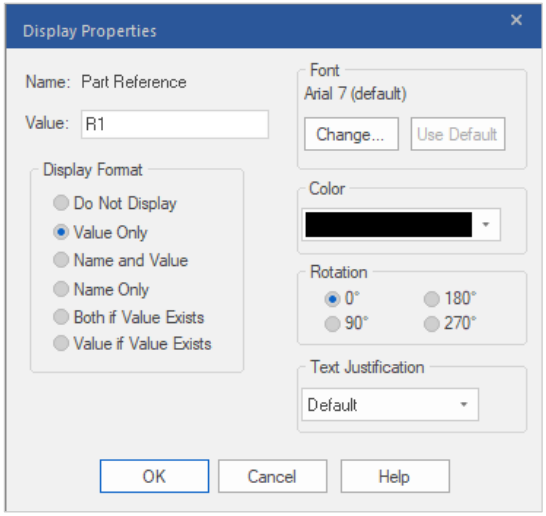
Selection (LMB) and **RMB > Edit Properties …**

or by

Double click

on desired property.

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



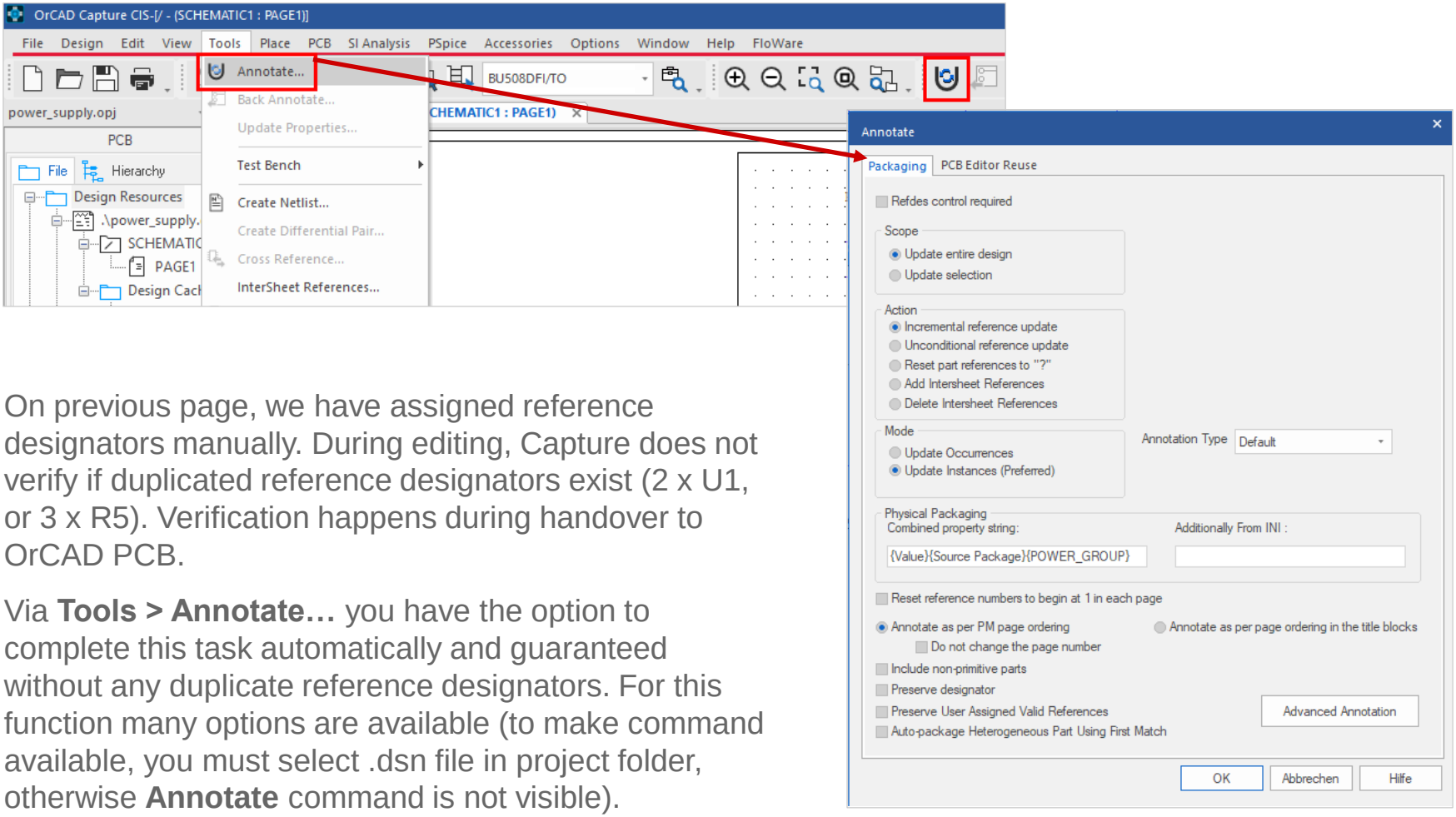
Tip

It is possible to open 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 OrCAD PCB.

Via **Tools > Annotate…** you have the option to complete this task automatically and guaranteed 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).

Complete Schematic

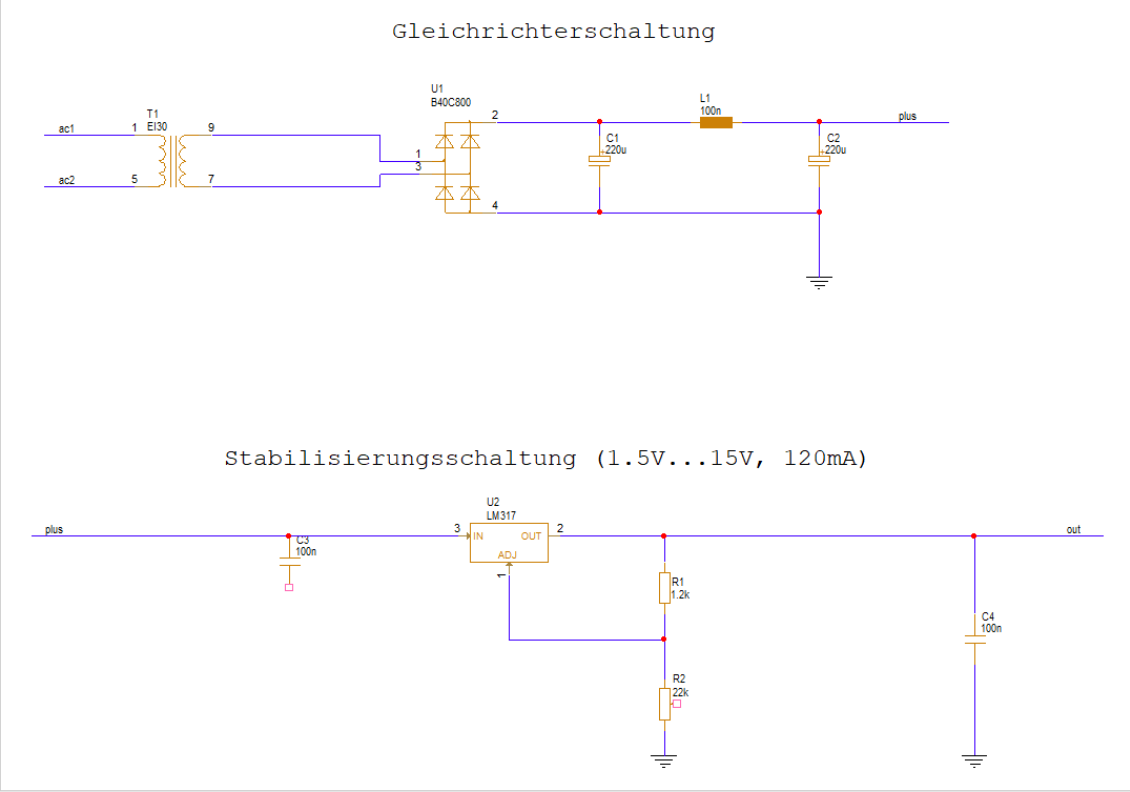
After schematic is adjusted and parts, text got aligned, completed schematic should look similar to schematic on this page.

Note

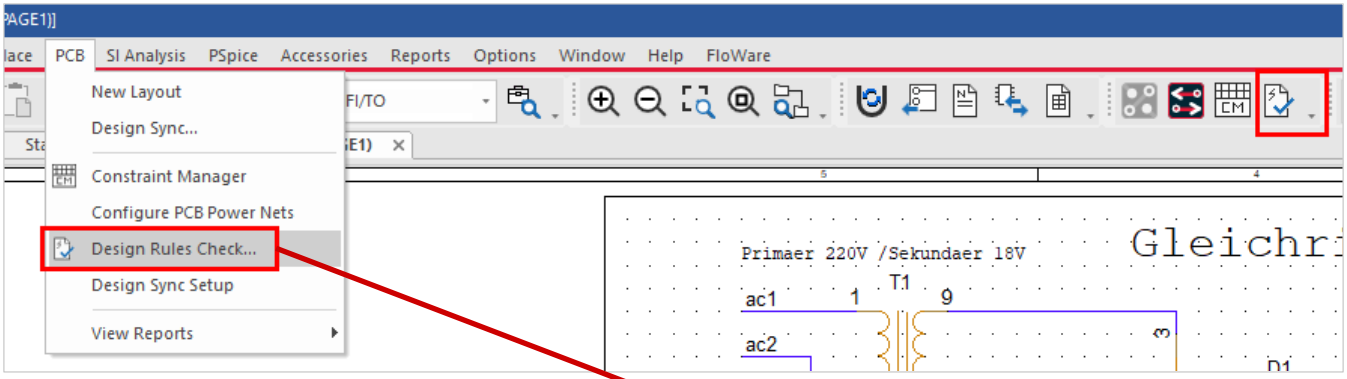
At this point we would like to mention that because of used libraries assignment of footprints (DIP14, SO14, SMD1206, etc.) is not done so far.

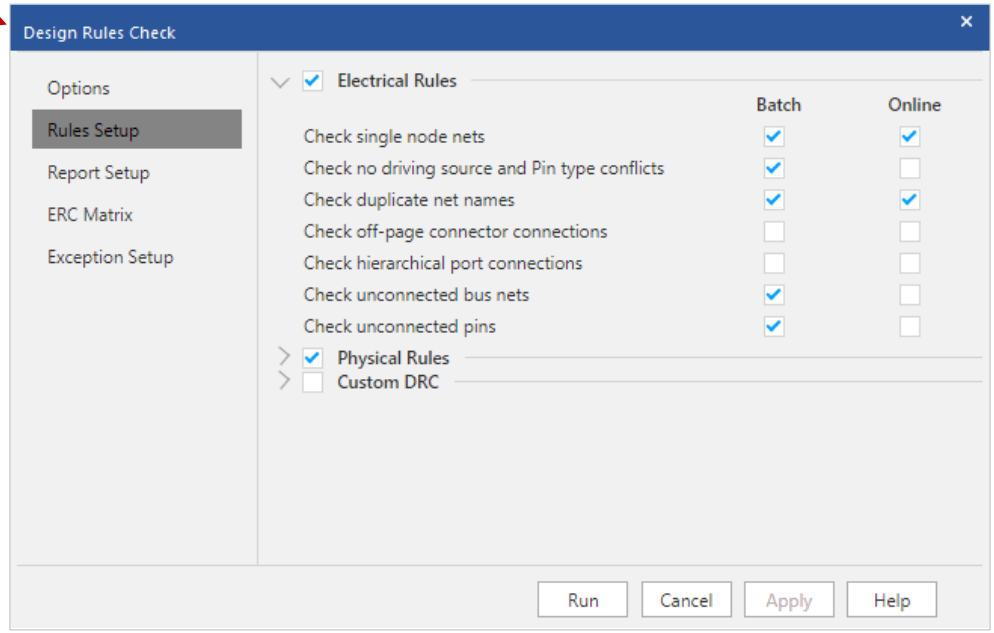
This is a subject for later when we import data into layout module.

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



Design Rule Check

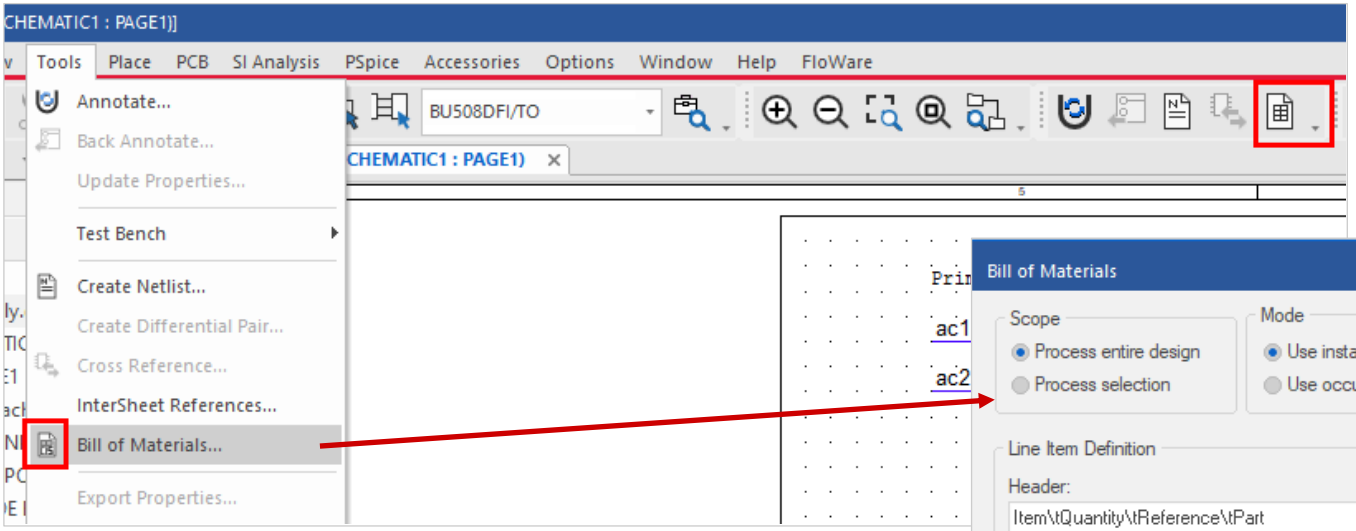


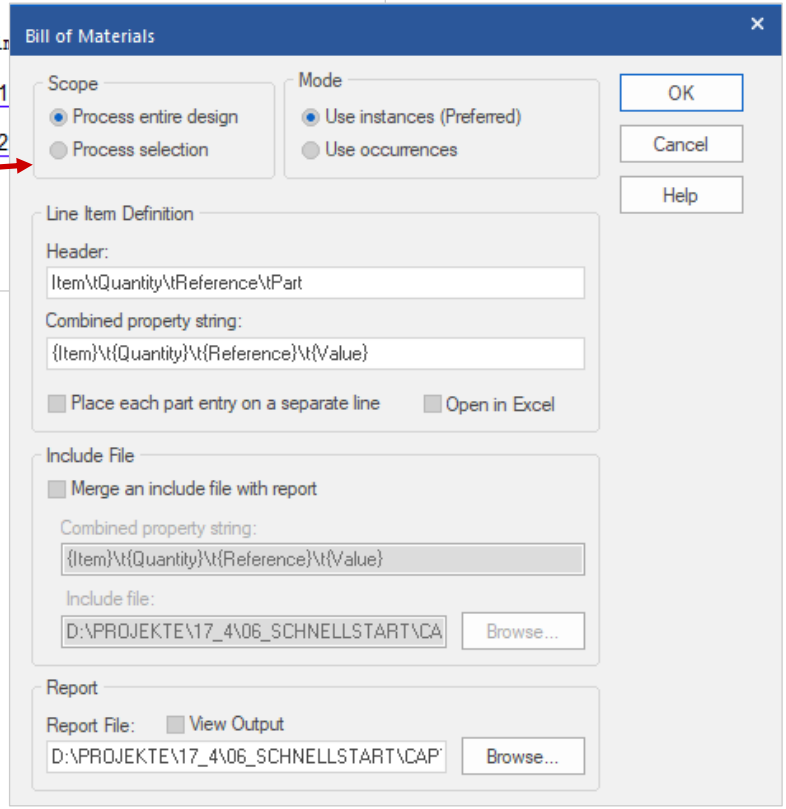


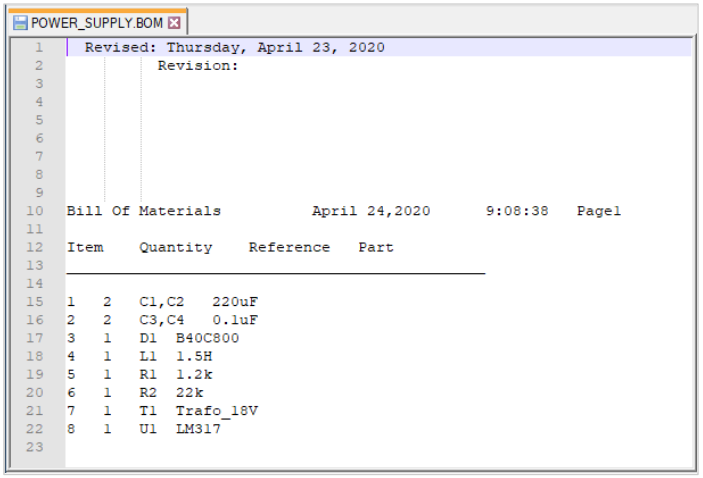
DRC is able to identify one pin nets, nets without input or output. Pre condition is a correct defined library.

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

Bill of Material (BOM)



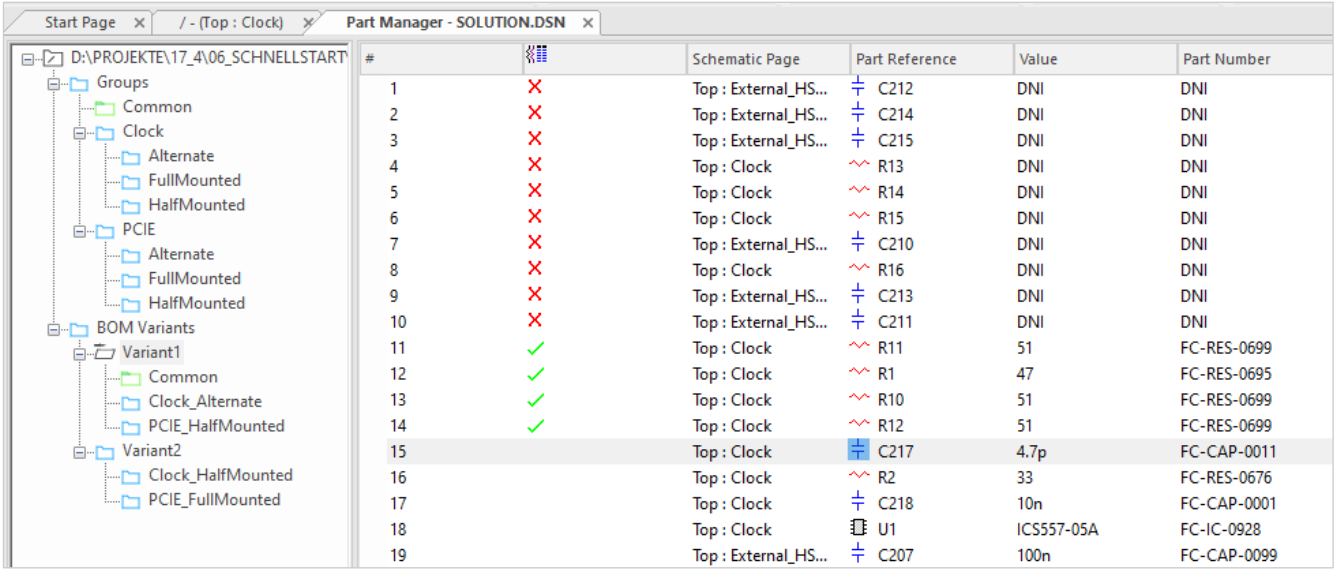




Design Variants

Variants – Overview

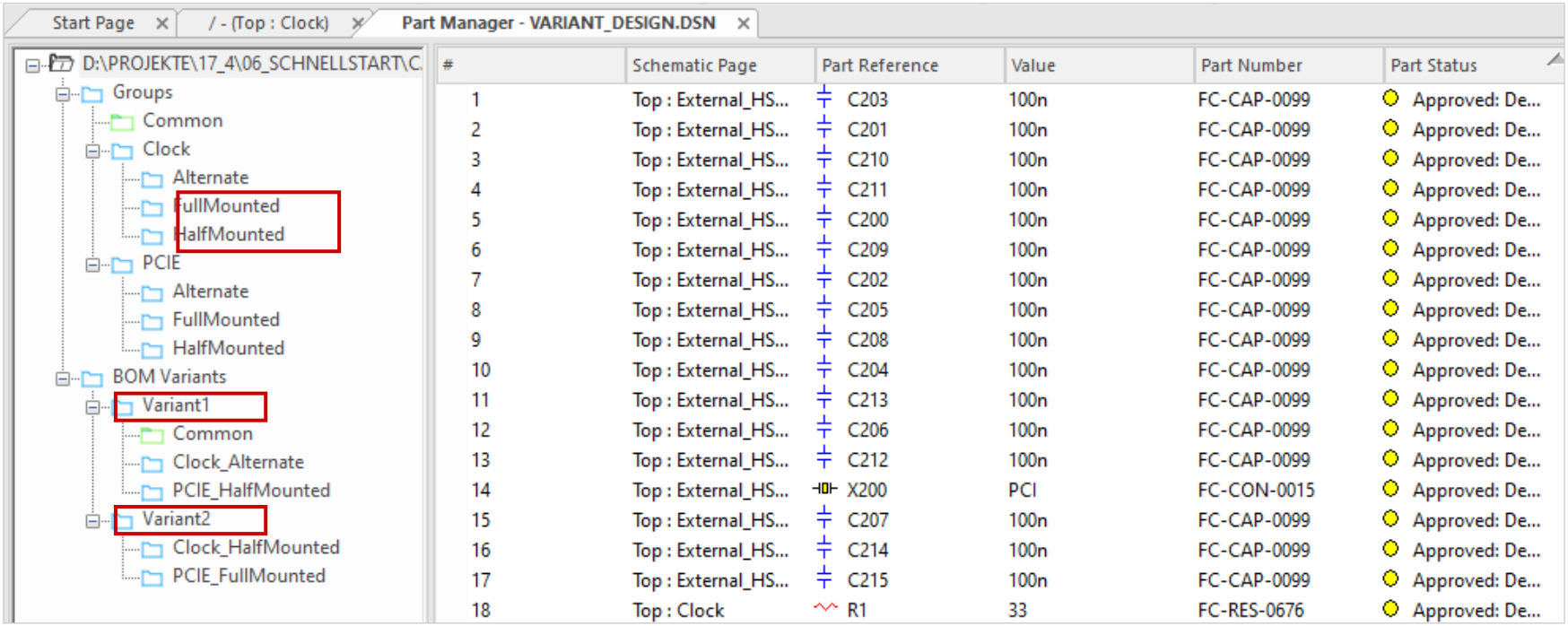
- Following steps are necessary to create design variants:
  - Dividing design into meaningful groups, for example, according to function blocks
  - Creating subgroups that reflect assembly variants of blocks
  - Creating BOM variants of entire projects (these are actual variants used by assembly house)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

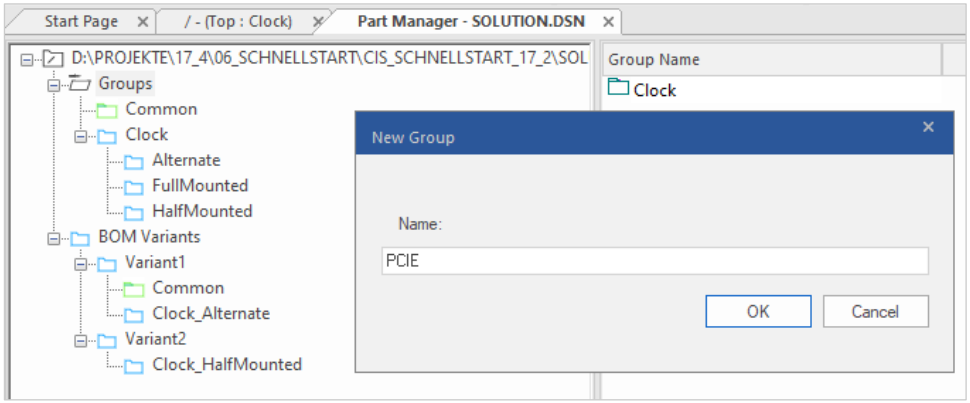
Variants – Example Design

- Capture Quick Start contains a variant sample design:
  - ~\Capture_CIS_Schnellstart_17_4\Solutions\VARIANT_DESIGN.DSN
  - Groups, subgroups and variants are already preconfigured in this example.
  - Creation of variants is described in this chapter.



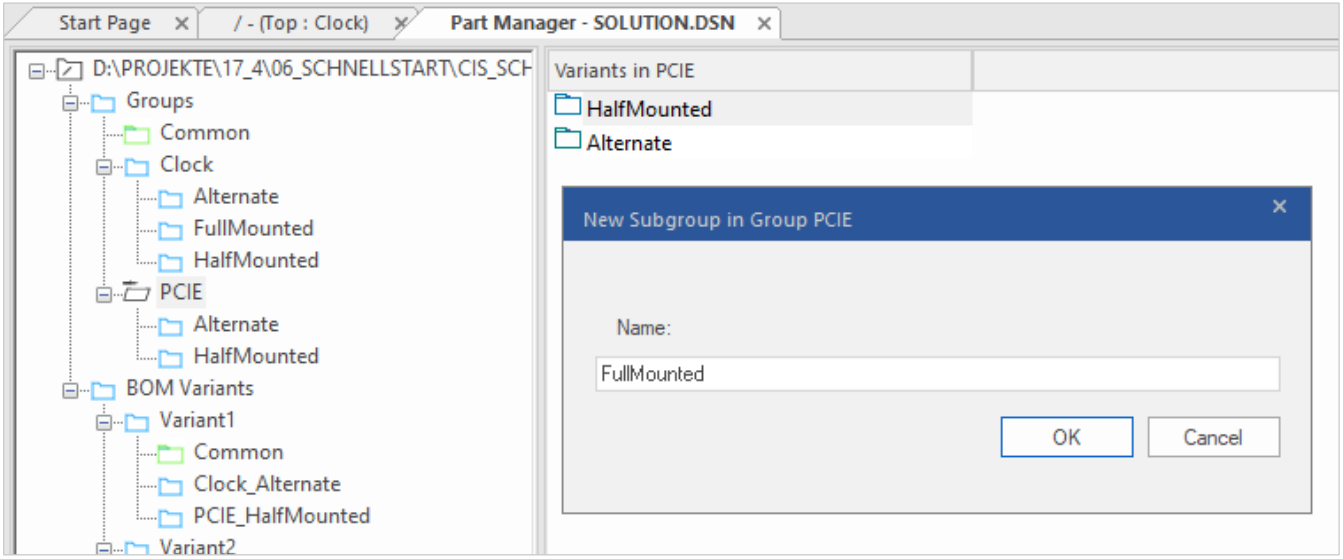
Variants – Groups

- With **Part Manager** you can manage parts in your design, carry out database updates or generate variants.
- Select .dsn file and click **RMB > Part Manager** or use menu **Tools > Part Manager > Open.**
- To describe variants, you need to divide the design into groups.
- It makes sense to store all parts of a function group in one group of variant description. 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 group **PCIE**.



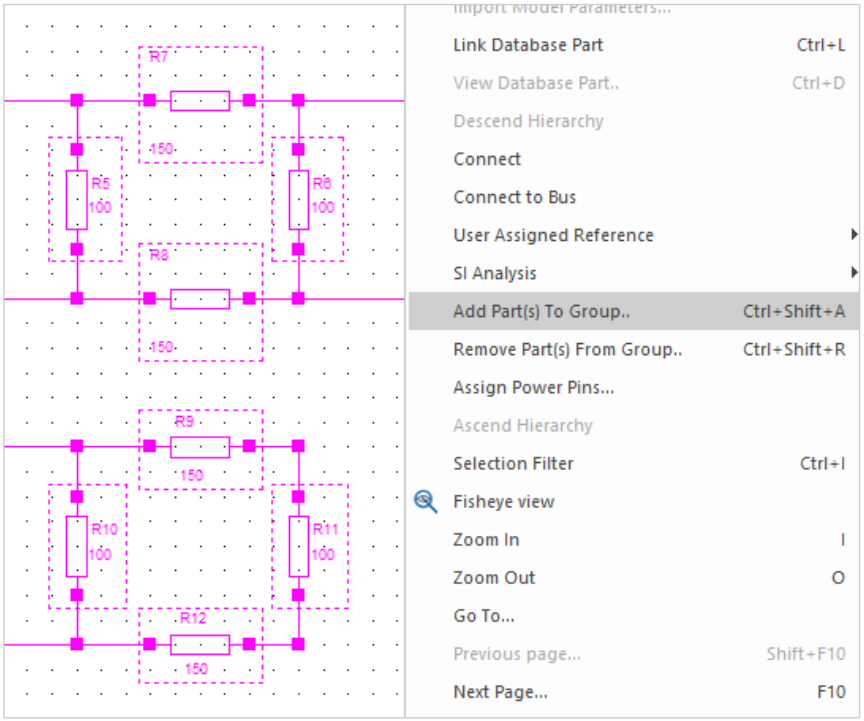
Variants – Subgroups

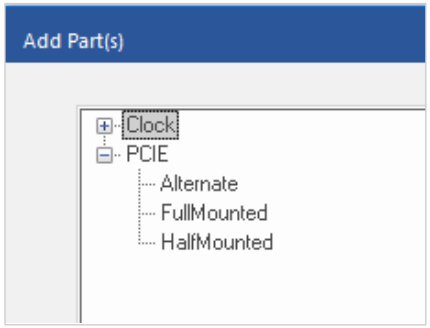
- Next step is to define subgroups that contain different assembly variants of groups.
- Select the group **Clock > RMB > New Subgroup.**
- Enter **Alternate**.
- Repeat this process, and enter **HalfMounted** and then **FullMounted**.

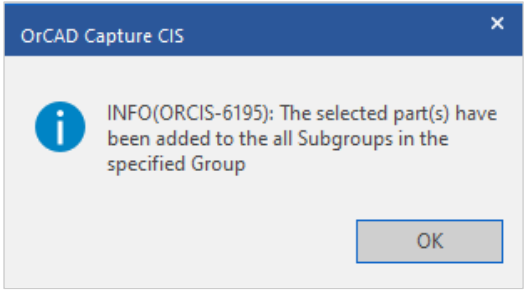


Variants – Assign Parts to Groups

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

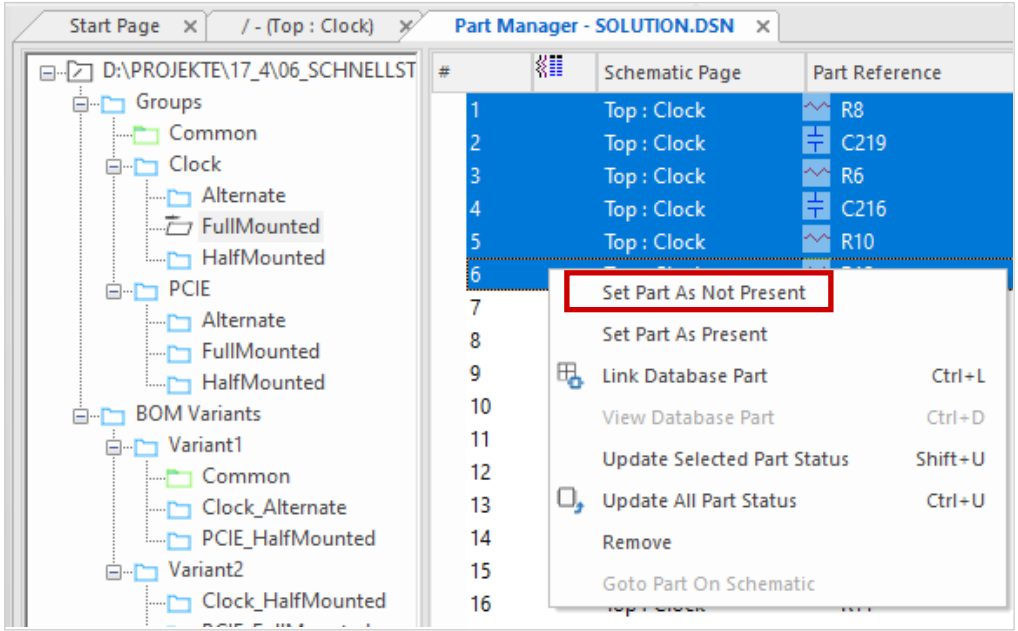






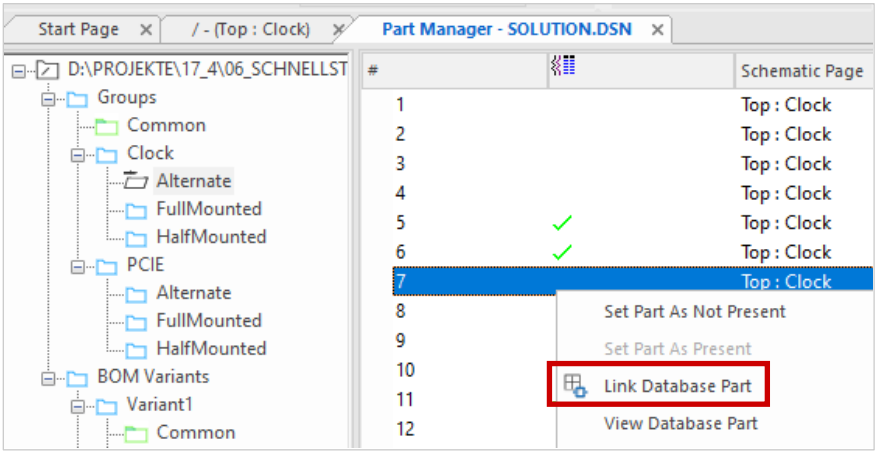
Variants – Unmounted Parts

- 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.



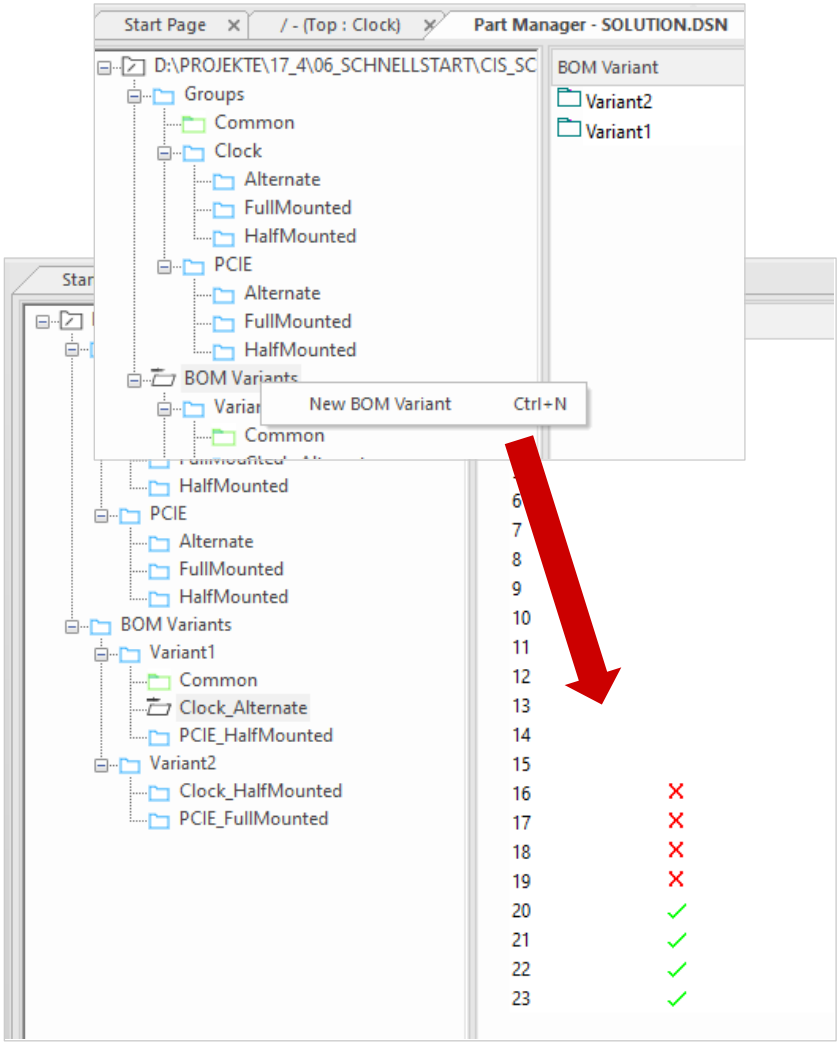
Variants – Alternative Placement

- 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.



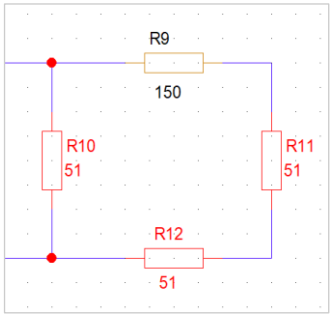
Variants – Define BOM

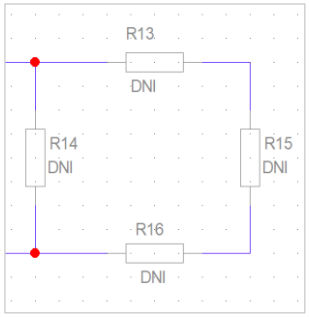
- Select **BOM Variants > RMB > New BOM Variants** and enter name **BOM1**. Repeat for **BOM2** and **BOM3**.
- Drag one subgroup per group into BOM. Finally, common group must be added to BOM, as it contains PCIE connector, that has not been assigned to any group.
- Variant BOM is shown in lower right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

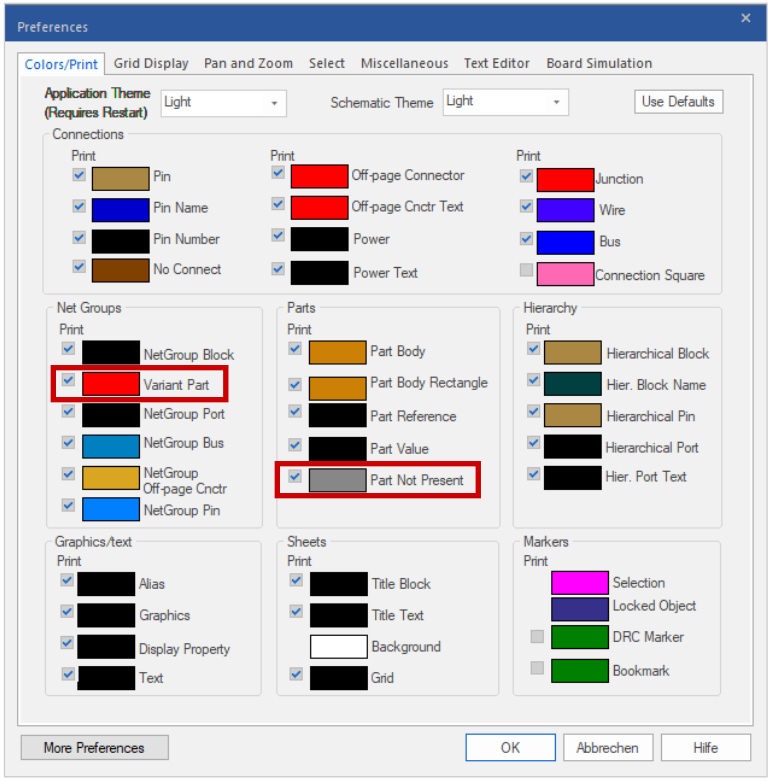


Variants – Schematic

- 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 green.
- You can set colors via **Options > Preferences**.

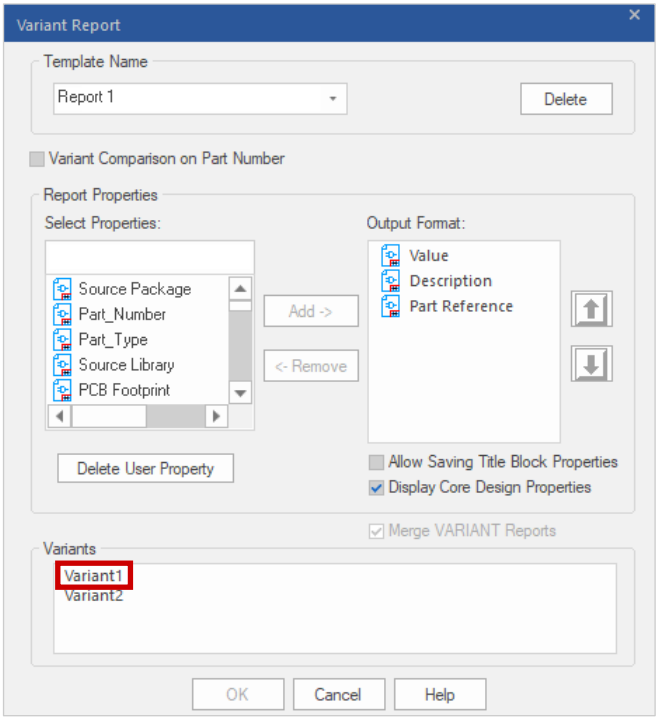






Variants – Report

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



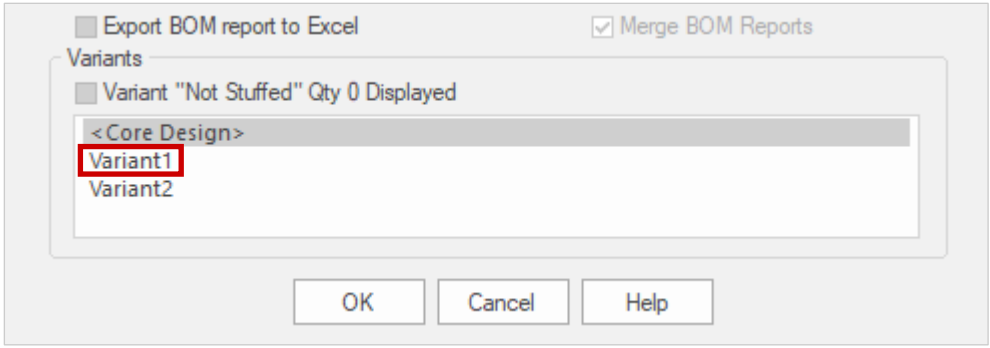
Start Page | / - (Top : Clock) Variant1 | SOLUTION.VRT*

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 Resisto 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 41 via **Report > CIS Bill of Material > Standard**.
- Use an existing template and select one of variants below.
- You will find **Variants Not Stuffed Qty 0 Displayed** via list of variants. If you check this box, all unstuffed parts with number 0 in parts list are also displayed.
- Click **OK** to start parts list output. Variant BOM is displayed.



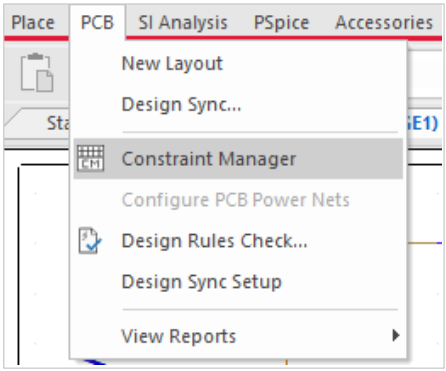
Constraint Manager

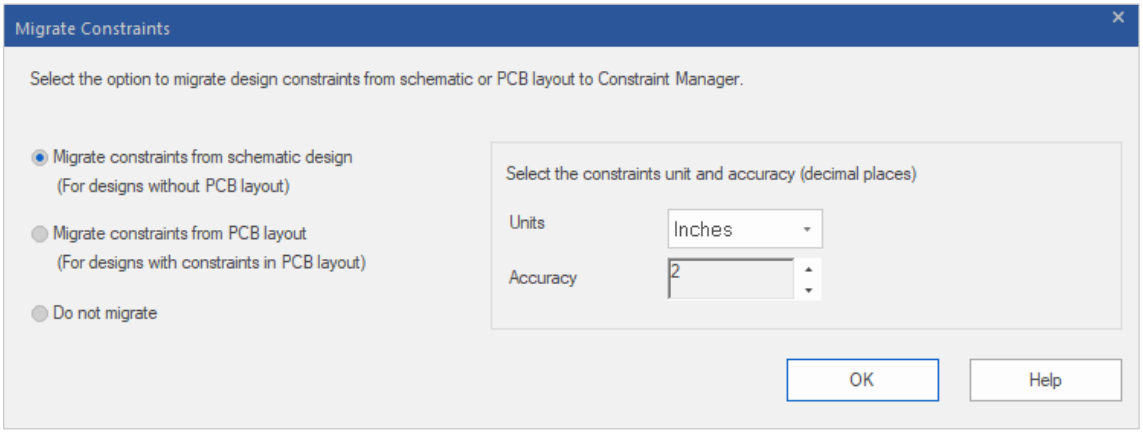
Constraint Manager in Capture

- This chapter in Capture Quick Start gives a brief Constraint Manager overview.
- You can find more detailed information in **PCB Editor Quick Start** starting on page 73.

Constraint Manager in Capture

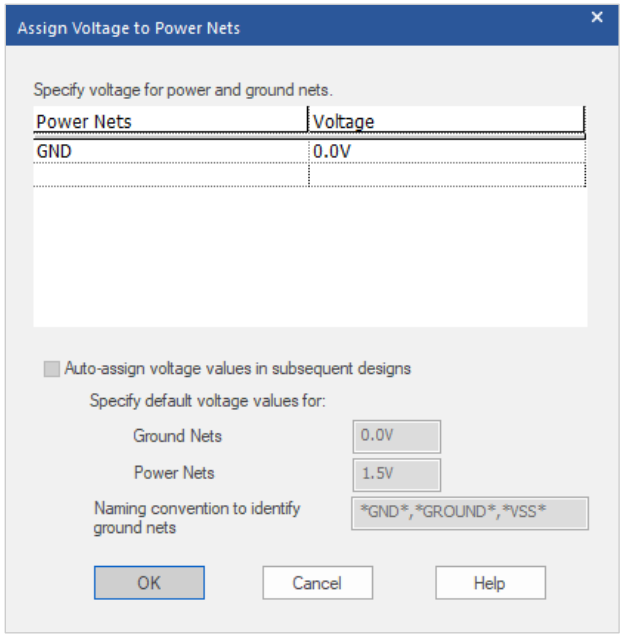
- To start Constraint Manager from Capture, please use command **PCB > Constraint Manager** or icon .
- Confirm query with **Migrate constraints from schematic design.**





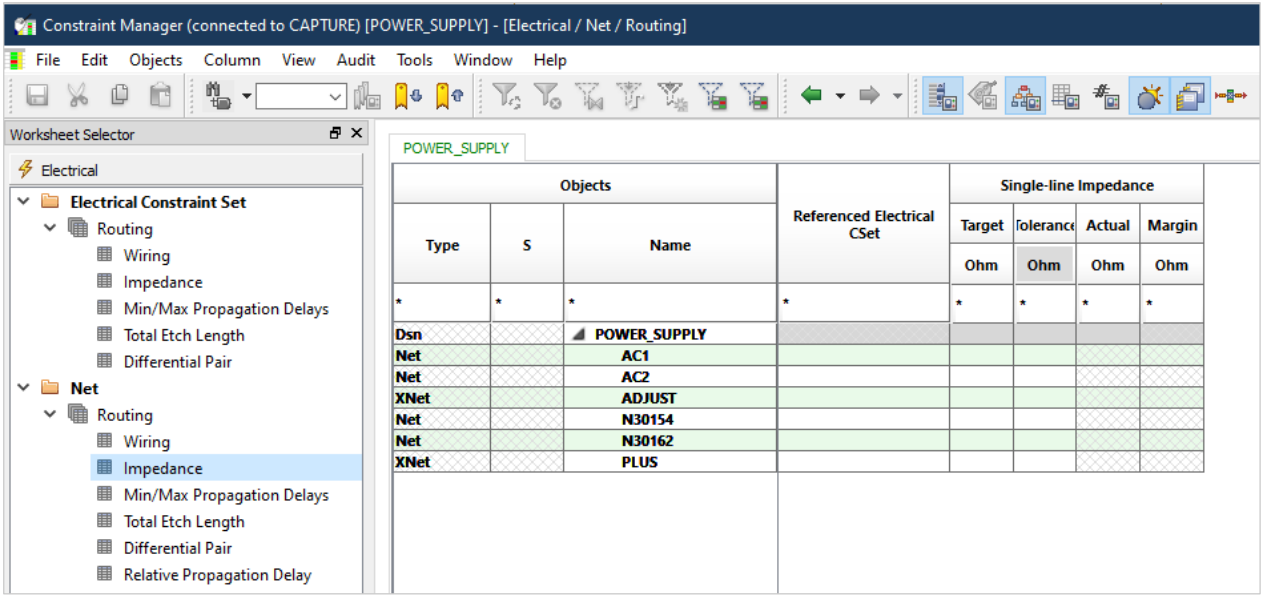
Assign Voltages to Power Nets

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



Constraint Manager in Capture

- From a schematic point of view only definition of electrical constraints makes sense.
- Many Constraints require data from layout. This data is available after a back annotation from layout.

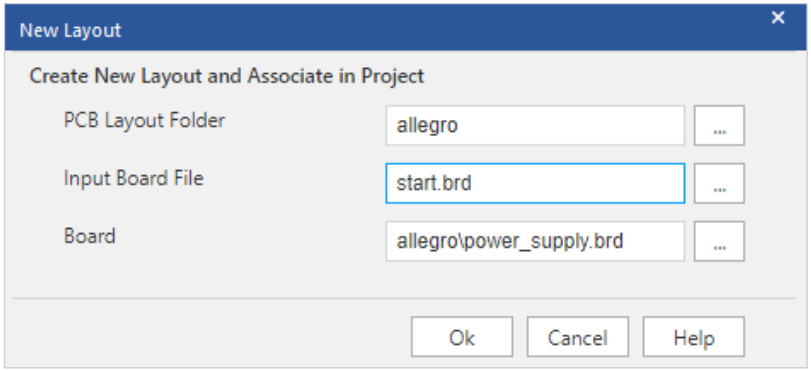


Design Sync

Generate Layout (I)

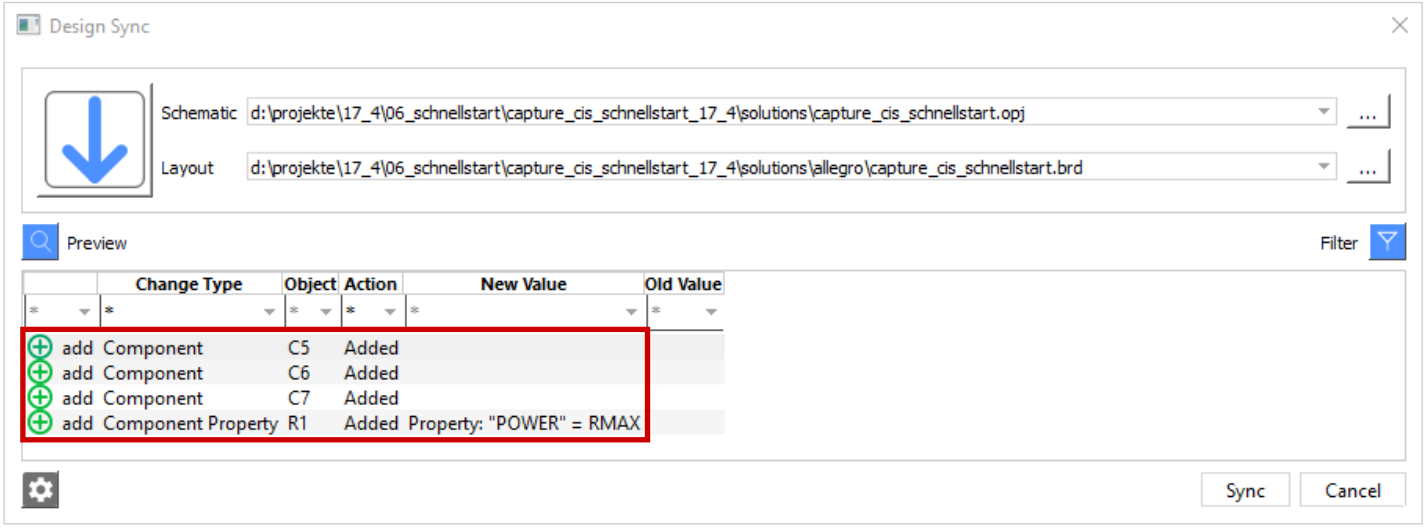
Via **PCB > New Layout** logic information can be transferred to a new PCB.

- 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 via **PCB > Design Sync**.
- Changes are displayed in form of an ECO list.



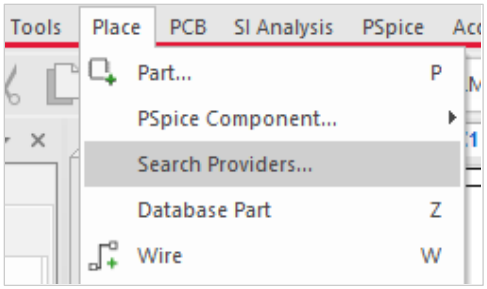
Libraries

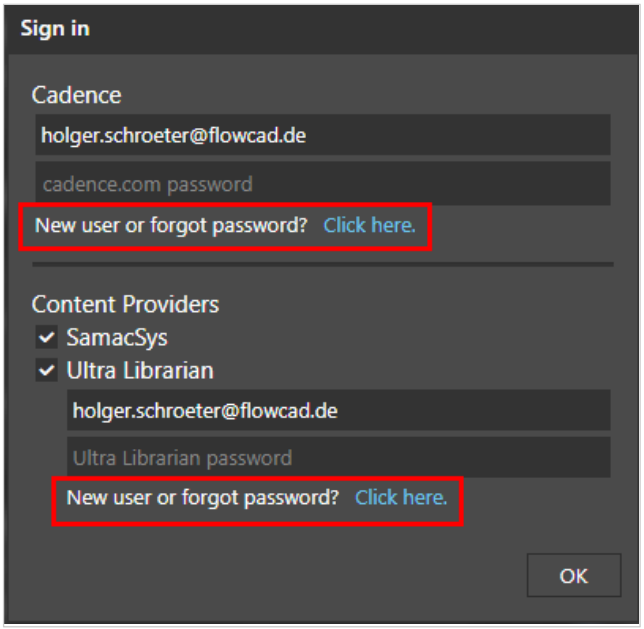
Missing Parts

In this chapter we describe how to search parts in web portals via function **Search Providers**. This function can be used to create new parts which are not part of OrCAD provided libraries.

Part Search (I)

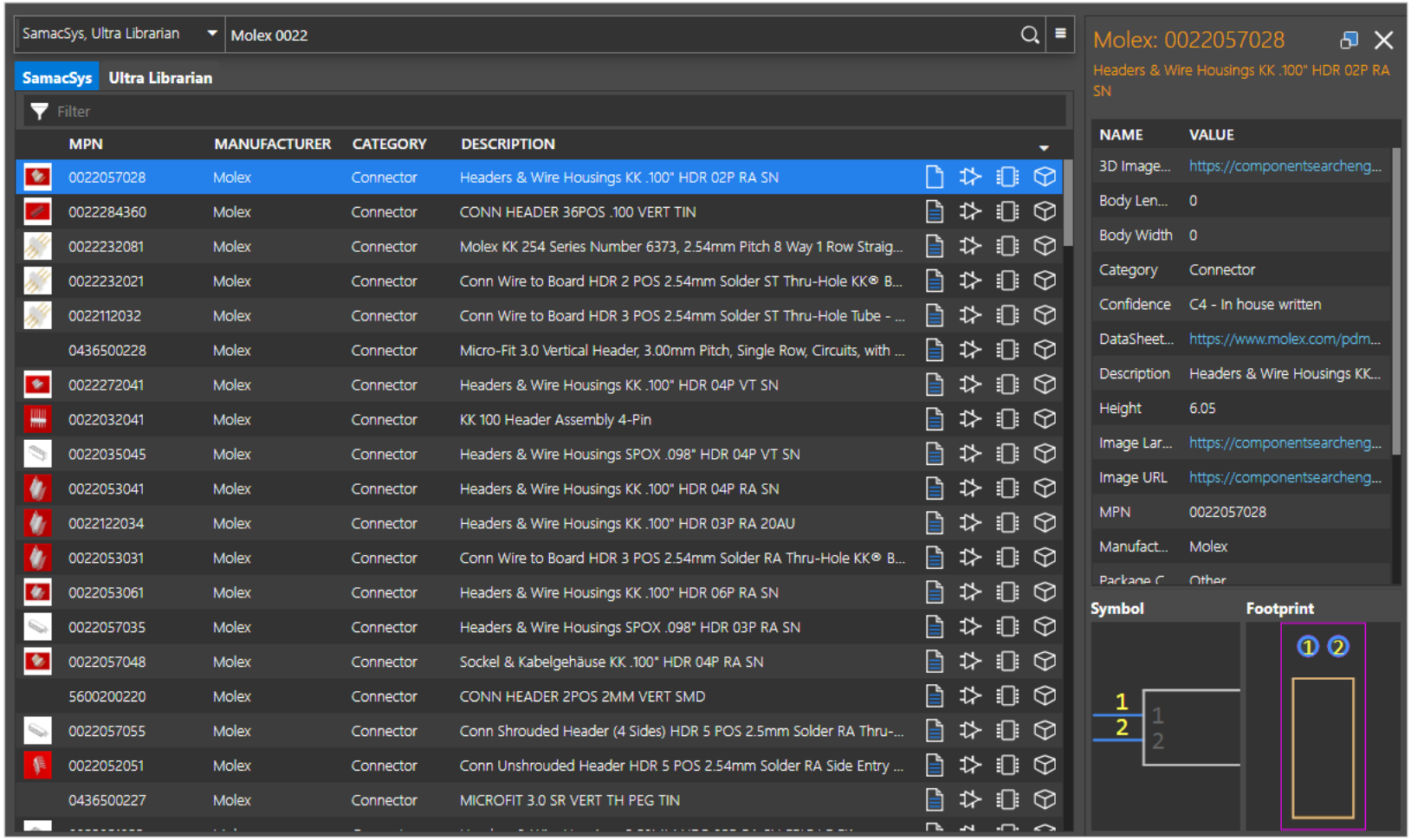
- If required parts are not available in assigned libraries, you have the option to search and download parts from Ultra Librarian and SamacSys web portal.
- Portals provide datasheets, schematic symbols, footprints and step models.
- Select **Place > Search Providers** to start web search.
- To use web portals, a free of charge Cadence and Ultra Librarian account is required.
- Please click on New user or forgot password? **Click here** to create free of charge accounts.





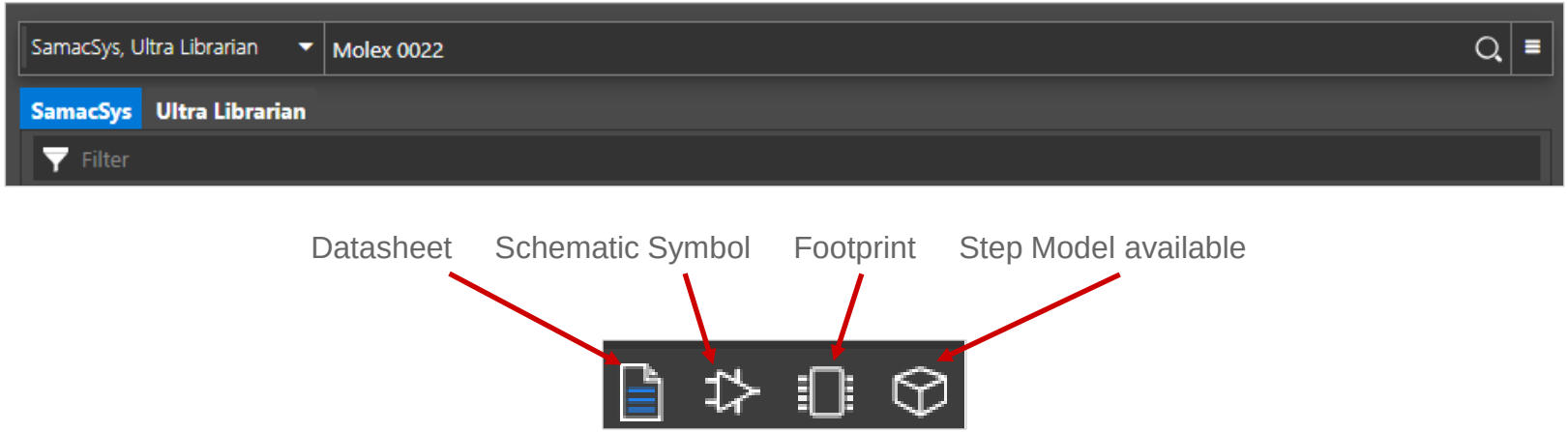
Part Search (II)

- **Search Providers** user interface provides a Google like search.

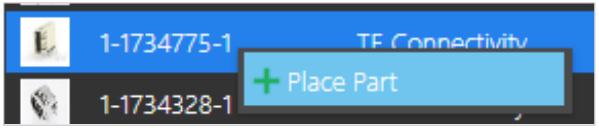


Part Search (III)

If required parts are not available in existing libraries, they can be downloaded from web portals.



During placement, parts will automatically be downloaded with command **RMB > Place Part** and saved in home folder: **~/Site/Library/downloaded_parts**



Part Creation

New parts can also be created with OrCAD Capture provided capabilities.

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

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

Completed part can be found in

- **Quickstart.olb**

New Library (I)

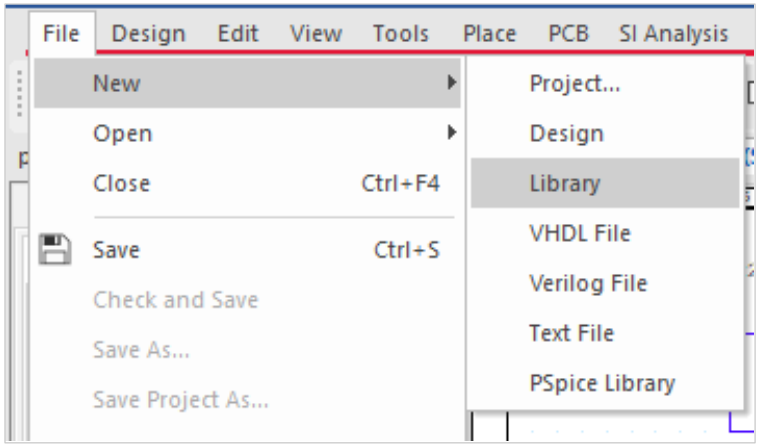
You can create new parts in a new library or open an existing library and add new parts.

Both actions happens via

- **File > New > Library**

or

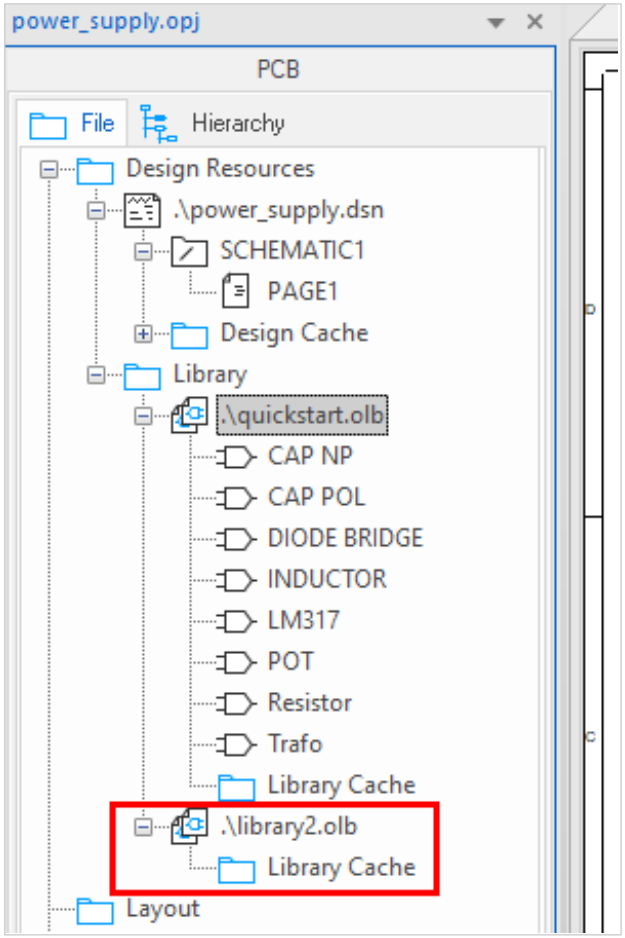
- **File > Open > Library**



New Library (II)

New library can be saved in project manager **RMB > Save As...** at new chosen path.

Afterwards this library can be assigned to your existing project.



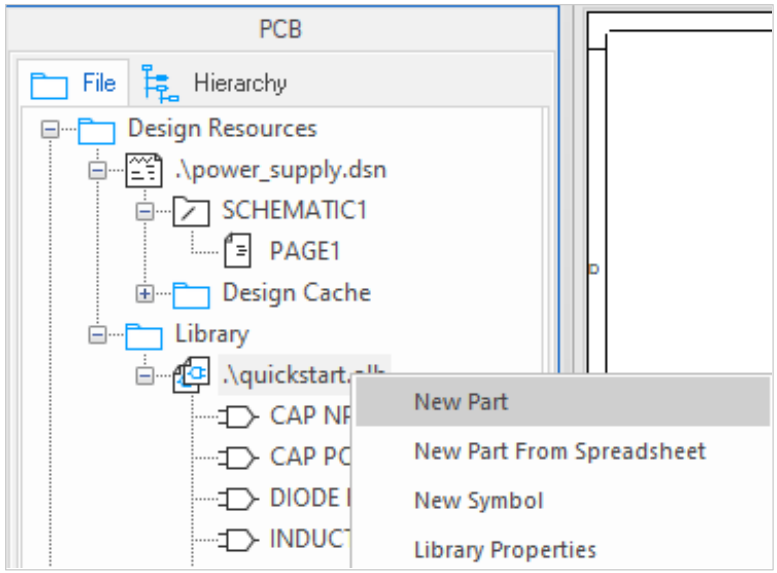
New Part Creation (I)

Via **Design > New Part...**

or

RMB > New Part

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



New Part Creation (II)

Please enter displayed values into the form.

New Part Name:

Part Reference Prefix:

PCB Footprint:

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

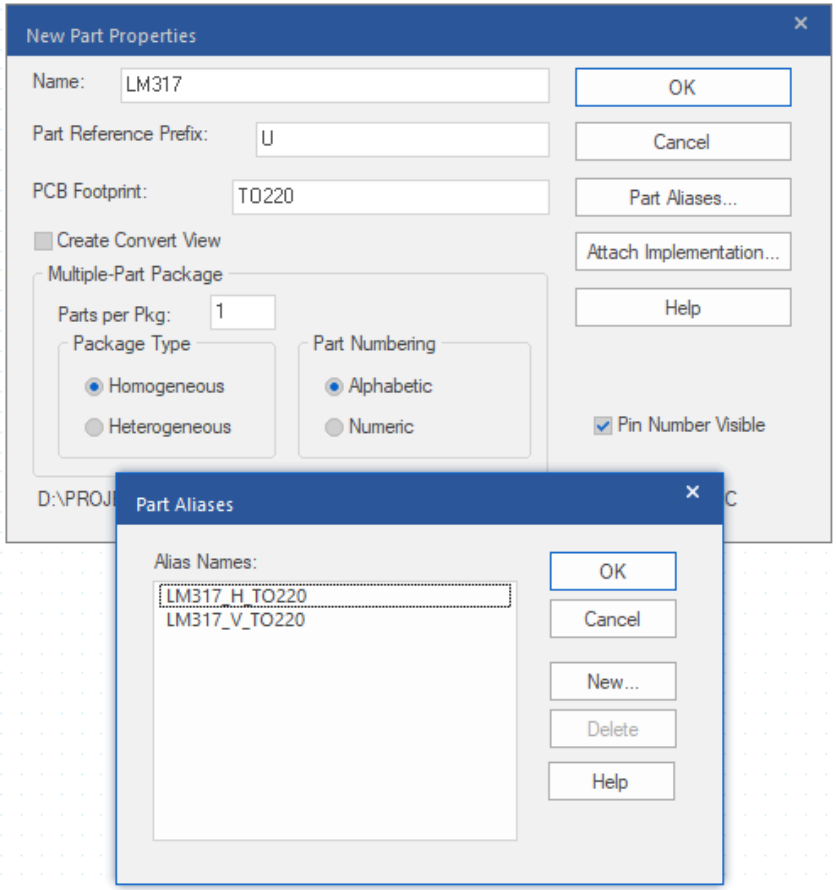
Parts per package:
i.e. 4 gates in 74LS00

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

Heterogenous:
i.e. relay with coil and switch

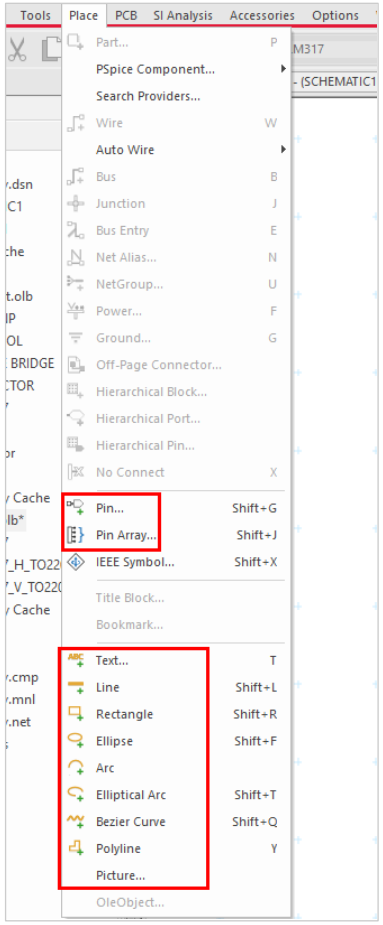
Part Numbering:
U?A, U?B ... oder U?1, U?2 ...

Click **OK**.



Commands in Capture

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



Hovering over icons shows functions as text.

On next pages we will explain most important functions.

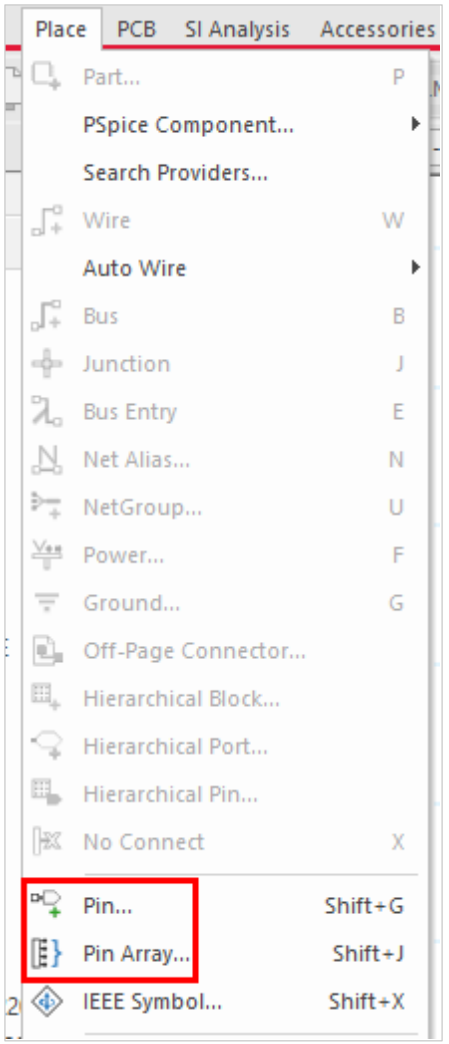
Adding Pins (I)

Place pins by:

- **Place > Pin...**

or

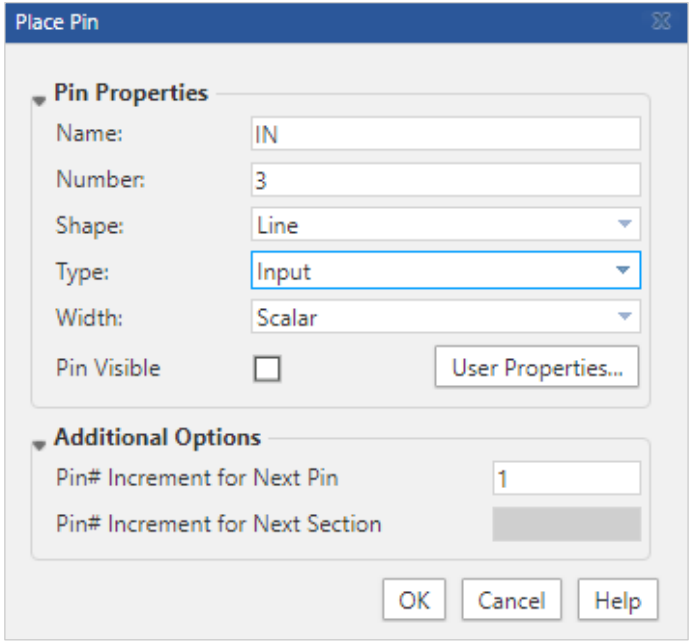
- **Place Pin Icon** 



Adding Pins (II)

Place Pin panel appears during pin property definition.

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

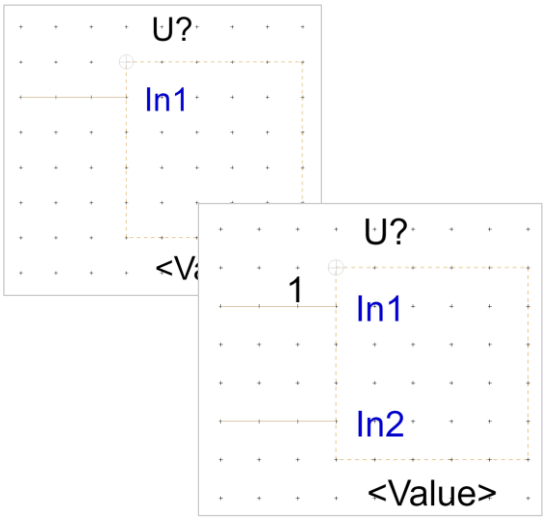


Tip

Using pin type Power allows visible and invisible pins.

Adding Pins (III)

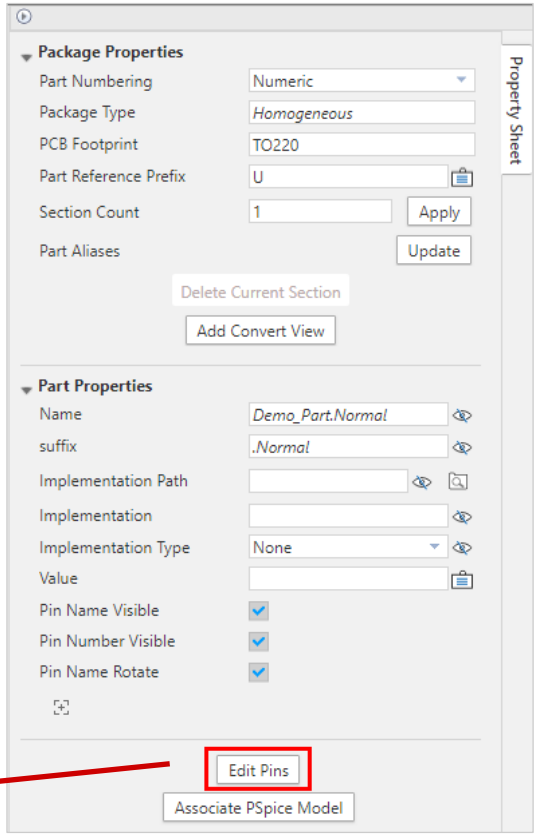
Individual pins get placed.

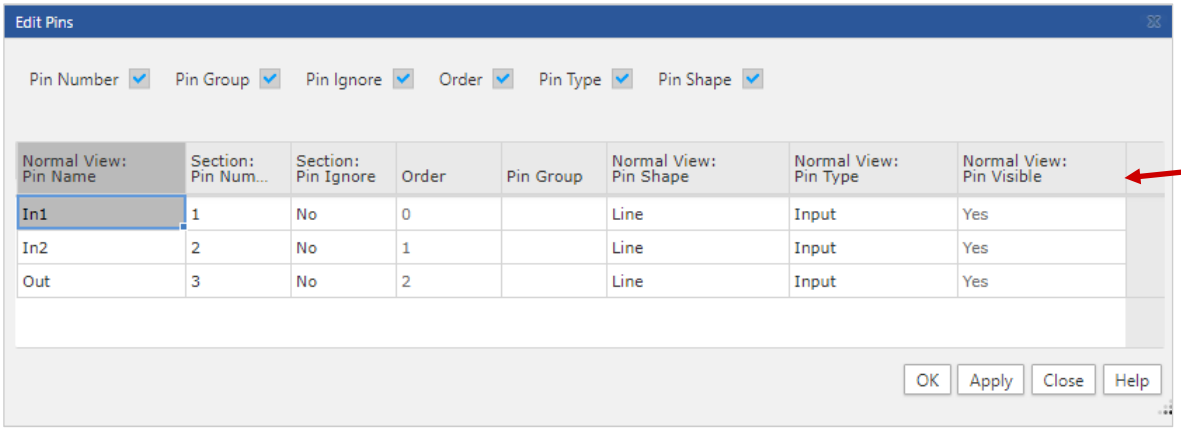


Additional pins can be placed via previous menu or via copy / paste **(Ctrl C, Ctrl V)**.

Placement of Pins is always adjusted to dashed line.

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





Symbol Graphics

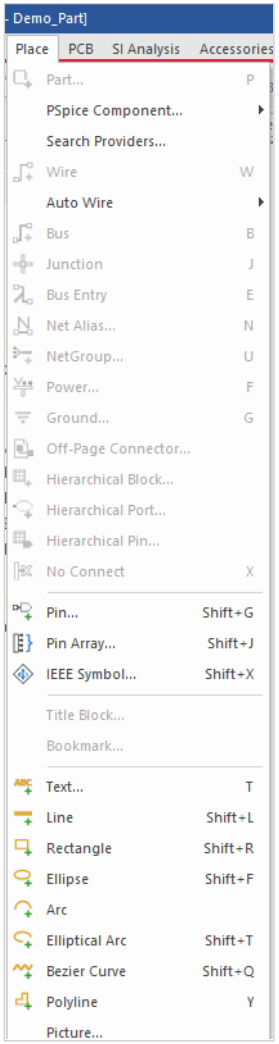
Completion of symbols

Symbol outline gets assigned finally. Commands are:

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

etc.

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



Tip

Dashed line is a frame to show real occupied area of 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, rectangle will be adjusted automatically.

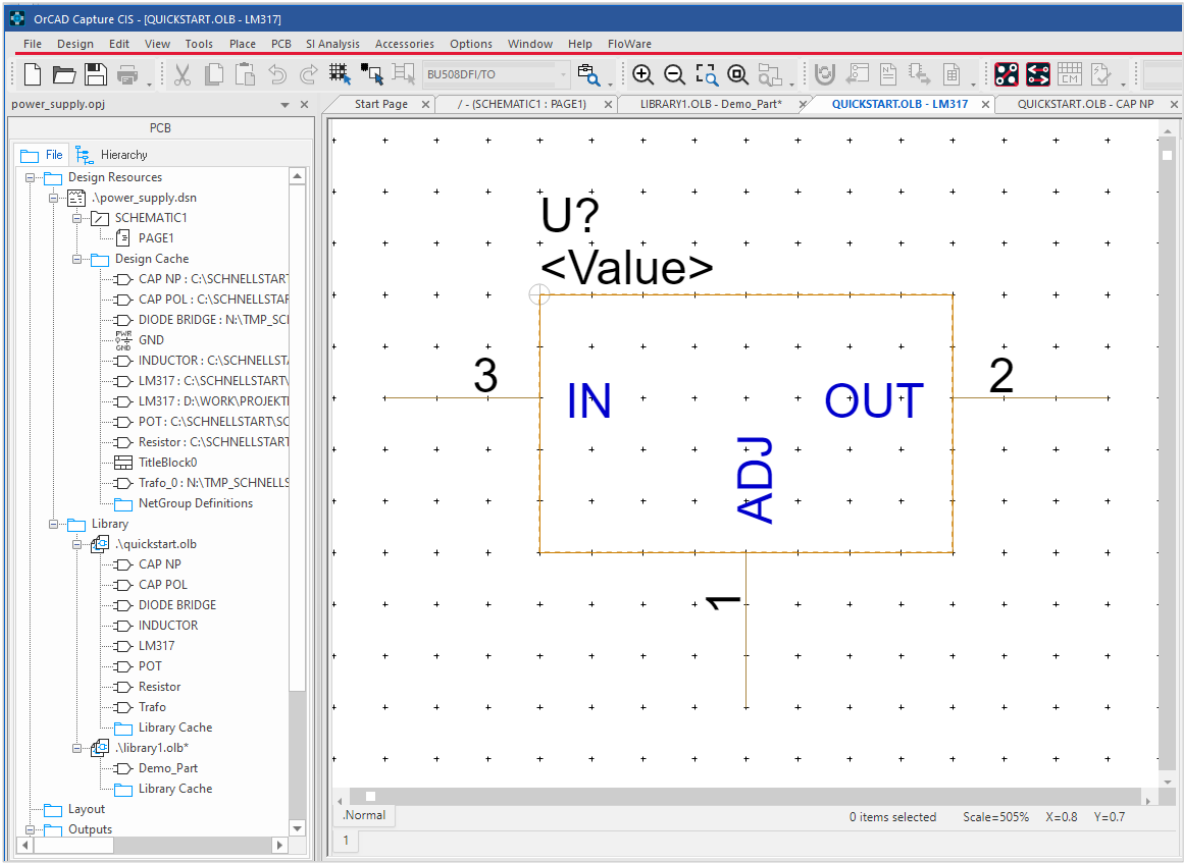
Completed Symbol

Completed symbol LM317

Please do not forget to save!

Tip

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



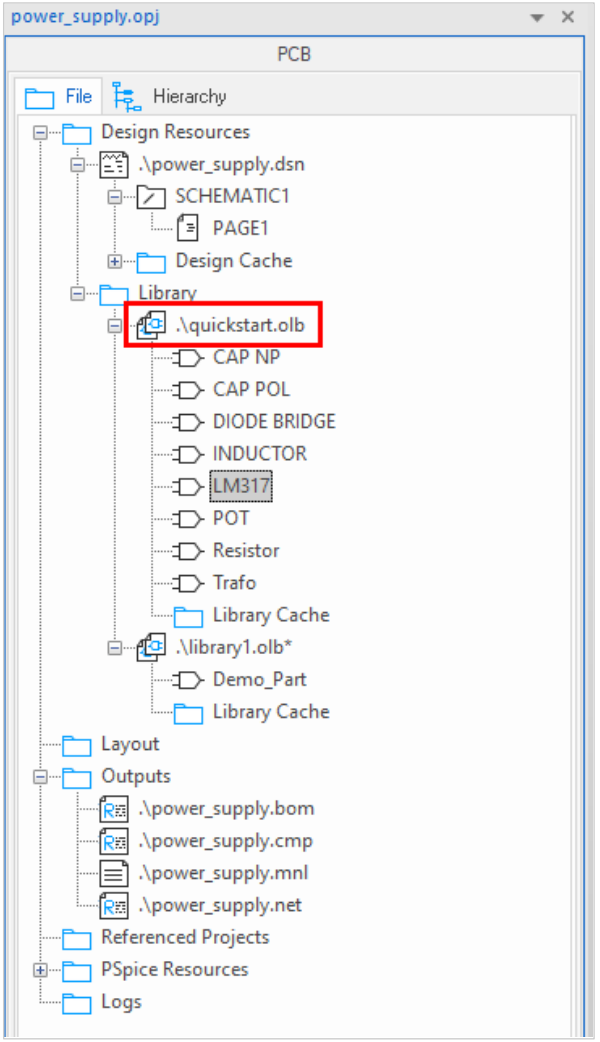
Library Management

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 26.

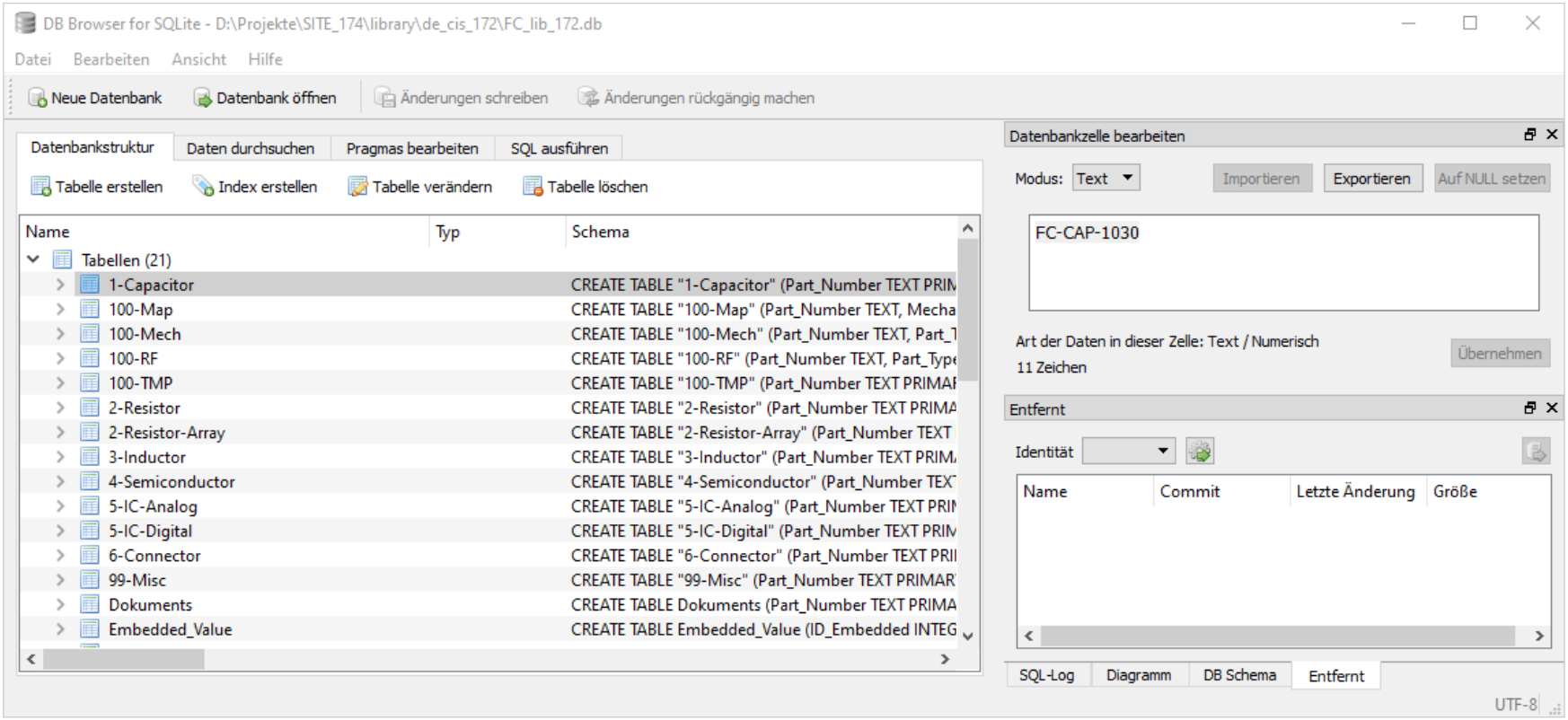
All used parts of this quick start are available in library **Quickstart.olb**.



Edit Database

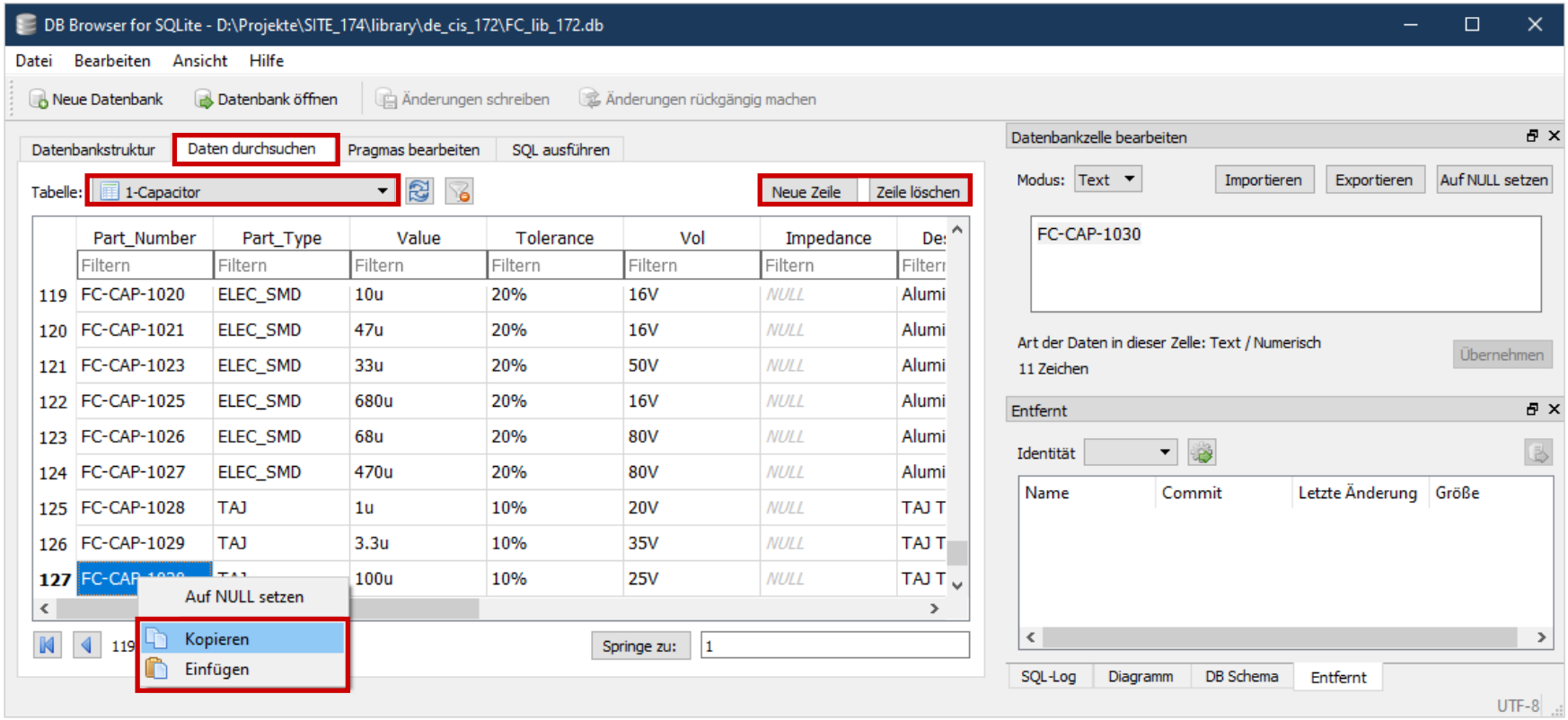
Edit Database (I)

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



Edit Database (II)

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

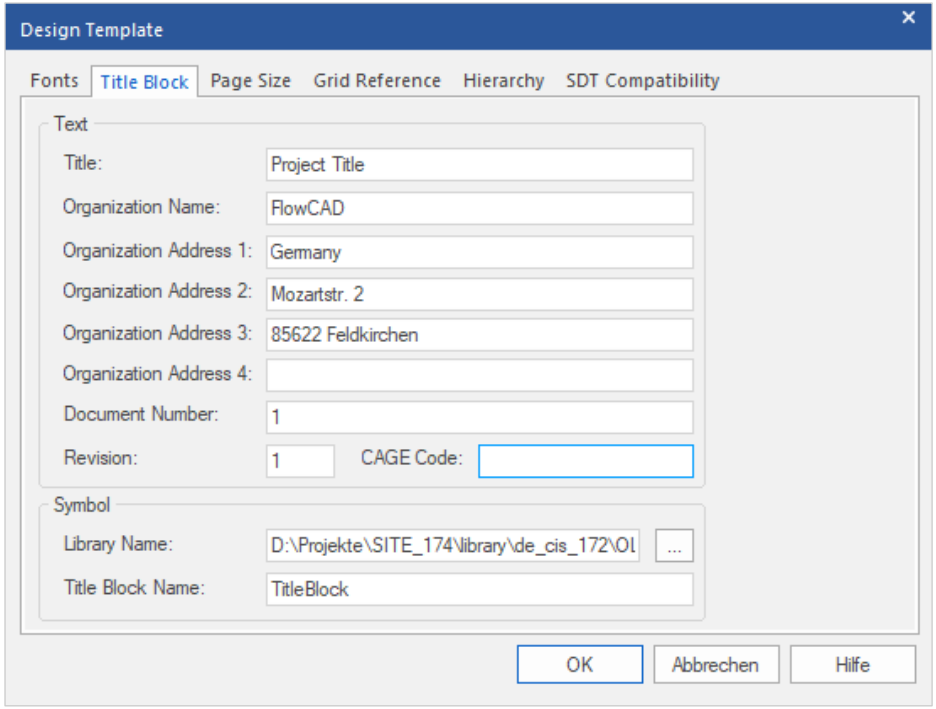


Settings and Templates

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 title block.

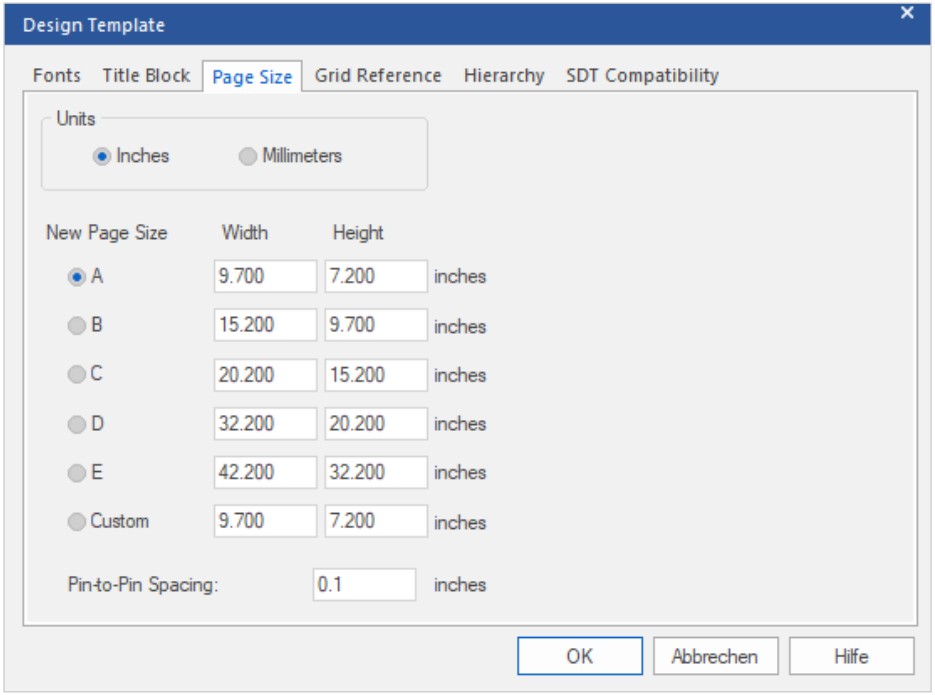


Design Template – Page Size

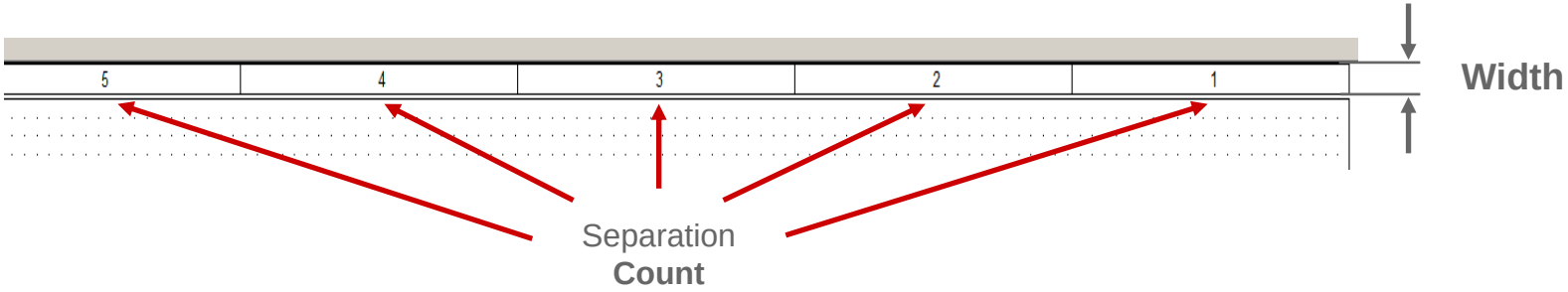
Size of schematic sheet can be defined at tab **Page Size.**

Attention

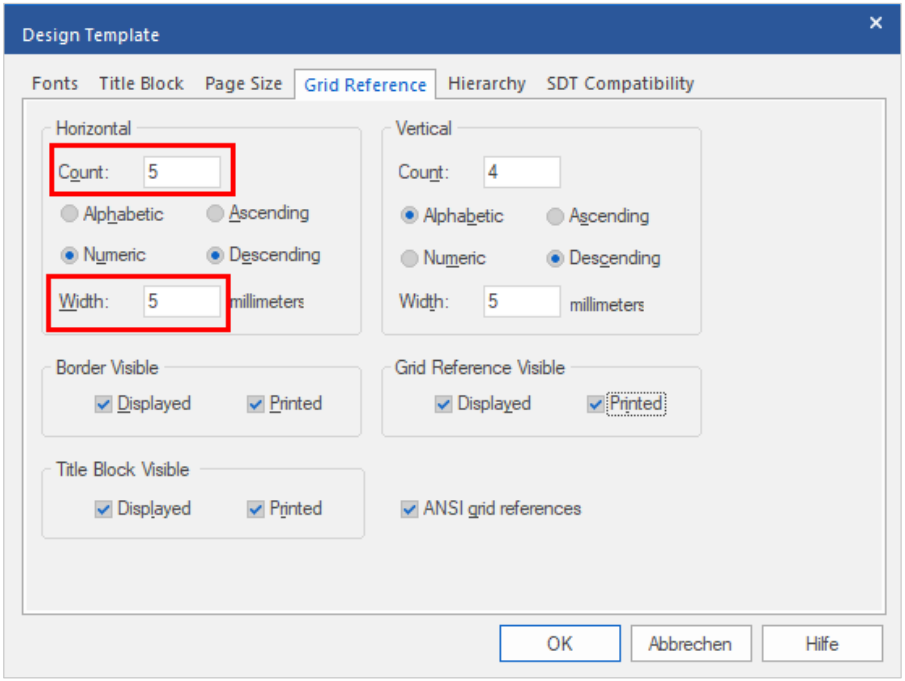
Settings under Pin-to-Pin Spacing must match pin-to-pin spacing used in library. This prevent later off grid connection problems.



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 17.4)

Operating Systems	Windows 10 (64-bit) Professional, Windows 2012 Server (All Service Packs); Windows Server 2012 R2; Windows 2016 Server
Hardware	Intel® Core™ i7 4.30 GHz or AMD Ryzen™ 7 4.30 GHz with at least 4 cores 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 services

Properties of Full Version

In General

- Grid imperial or metric
- Easy to create or edit libraries
- Forward-backward annotation
- Cross-probing between design entry (Capture) and layout (PCB Editor)

Schematic Editor Capture

- Maximum workspace up to 11.430 x 11.430 mm
- Multiple designs as part of one project
- Hierarchical structures with automated synchronization
- Automated reference designator assignment
- Electrical Design Rules Check (configurable)
- Configurable automated drawing border and title block
- Export of different netlist formats
- TCL interface

Usability Concept

- OrCAD Capture is menu driven.
- Menus are context sensitive. This means that panels or commands and resulting pop-up windows or pull-down menus get automatically adjusted to fit for selected objects.
- All commands or entries can get executed via following options:
  - Pull-down menus
  - Icons
  - Short keys
  - Pop-up windows
- There is no traditional command line. TCL script can be executed via command window.

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

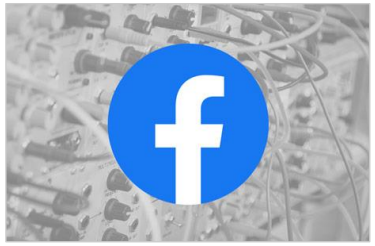
Follow Us

FlowCAD.com/newsletter

The FlowCAD newsletter for PCB designers appears about every two months. It is free of cost and will be sent by e-mail.

youtube.com/FlowCAD

On our YouTube channel you can find 100+ tutorials to learn more about electronic circuits. Our playlists also offer product news and webinars.



twitter.com/FlowCAD

On FlowCAD's Twitter we provide press releases, news articles, films and images as well as reports from events.

facebook.com/FlowCAD

Join our Facebook page. You will find selected news, events, success stories and insights.

Don't forget to subscribe, share and like!

FlowCAD