



# Quickstart OrCAD Capture

Version 17.4



# Table of Contents

- [Introduction](#)
- [Schematic Template](#)
- [Start of Capture](#)
- [New Project Setup](#)
- [Part Placement](#)
- [Adding / Edit Text](#)
- [Connections](#)
- [Property Editing](#)
- [Design Variants](#)
- [Constraint Manager](#)
- [Libraries](#)
- [Settings and Templates](#)
- [Appendix](#)



# Introduction

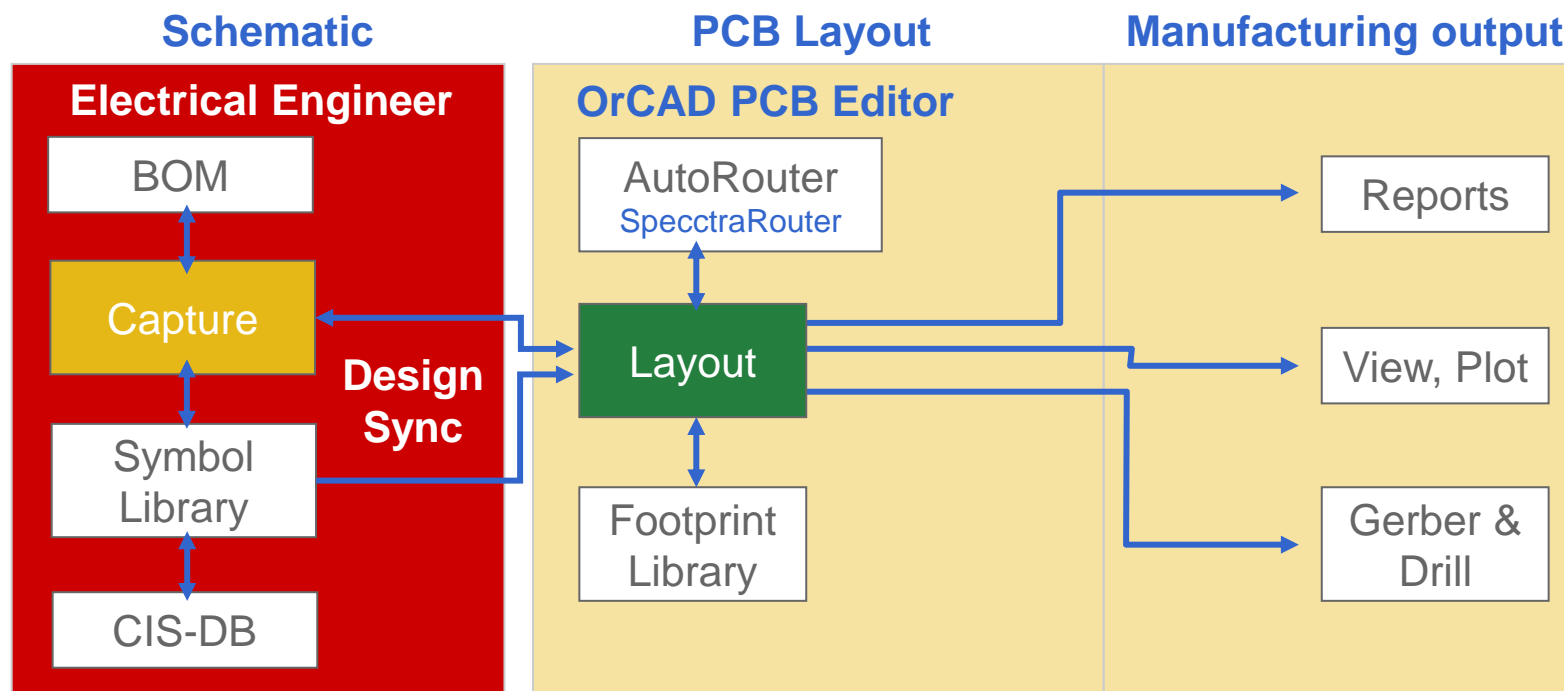


# Introduction

- 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



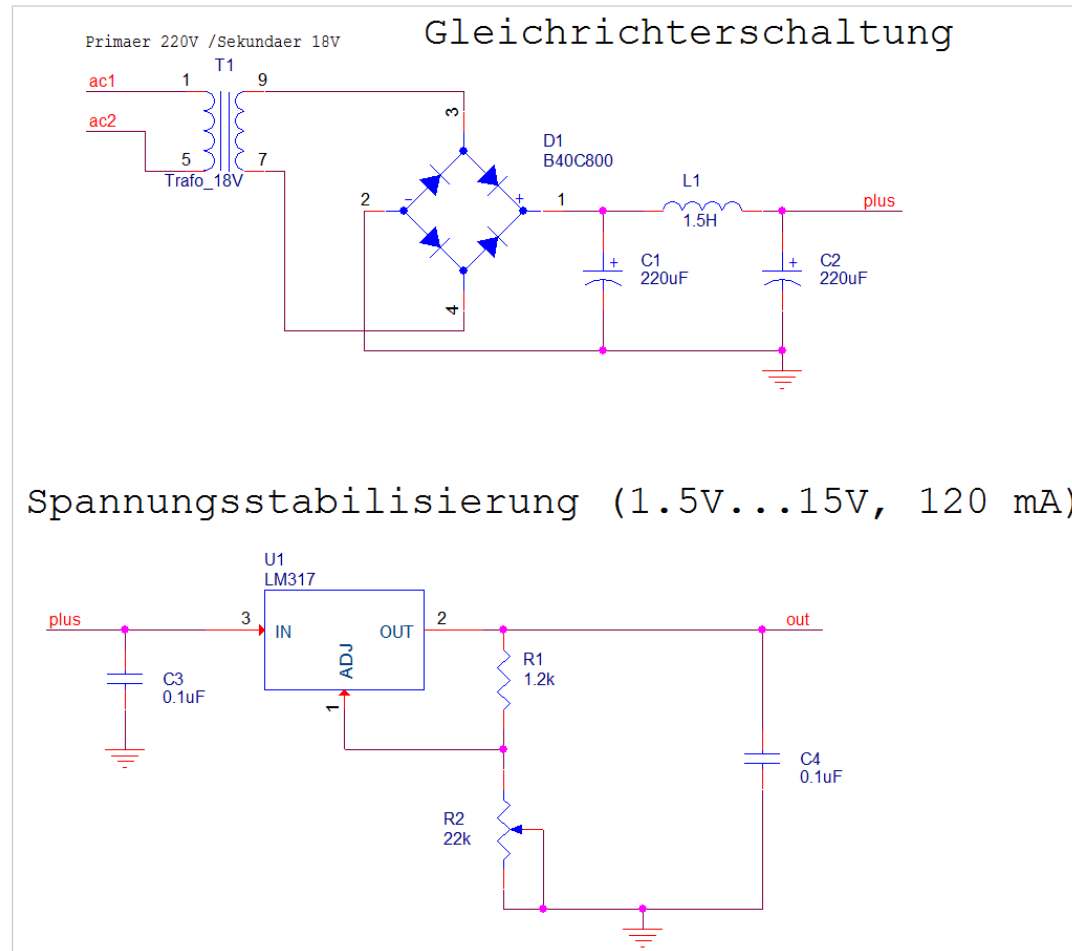
- As you can see, OrCAD PCB Designer Flow consists mainly of two parts.
  - These are the schematic capture module **Capture** and the layout module **OrCAD PCB Editor**.
- Both modules are supplemented by additional sub packages who represent in each combination an ideal tool, enabling the user to complete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

## FlowCAD Confidential | 6



# 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



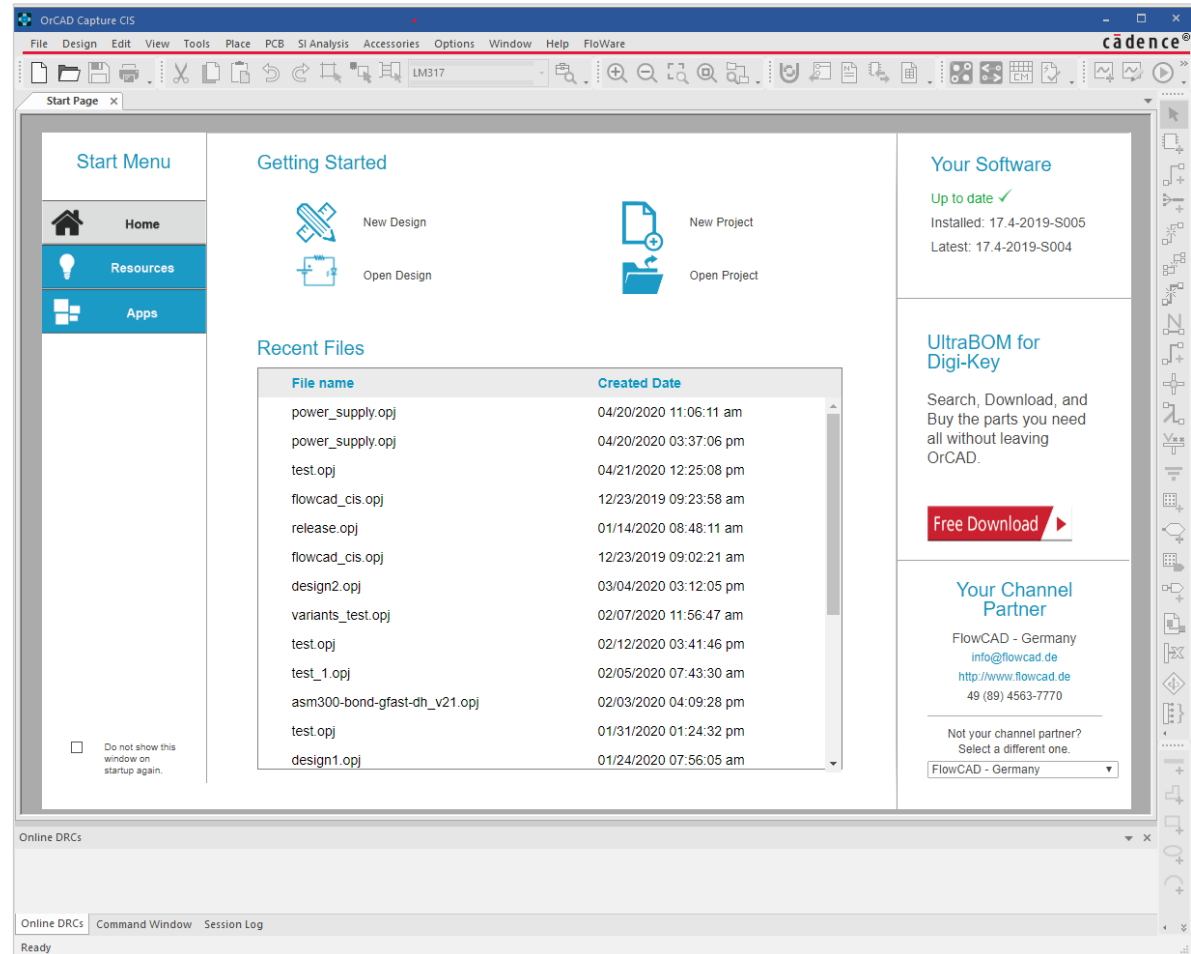


# Start of Capture

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.

### Start Menu

- Home
- Resources
- Apps

☐ Do not show this window on startup again.

### Getting Started

New Design

Open Design

New Project

Open Project

### Recent Files

File name	Created Date
power_supply.opj	04/20/2020 11:06:11 am
power_supply.opj	04/20/2020 03:37:06 pm
test.opj	04/21/2020 12:25:08 pm
flowcad_cis.opj	12/23/2019 09:23:58 am
release.opj	01/14/2020 08:48:11 am
flowcad_cis.opj	12/23/2019 09:02:21 am
design2.opj	03/04/2020 03:12:05 pm
variants_test.opj	02/07/2020 11:56:47 am
test.opj	02/12/2020 03:41:46 pm
test_1.opj	02/05/2020 07:43:30 am
asm300-bond-gfast-dh_v21.opj	02/03/2020 04:09:28 pm
test.opj	01/31/2020 01:24:32 pm
design1.opj	01/24/2020 07:56:05 am

### Your Software

Up to date ✓  
Installed: 17.4-2019-S005  
Latest: 17.4-2019-S004

### UltraBOM for Digi-Key

Search, Download, and Buy the parts you need all without leaving OrCAD.

[Free Download](#)

### Your Channel Partner

FlowCAD - Germany  
[info@flowcad.de](mailto:info@flowcad.de)  
<http://www.flowcad.de>  
49 (89) 4563-7770

Not your channel partner?  
Select a different one.

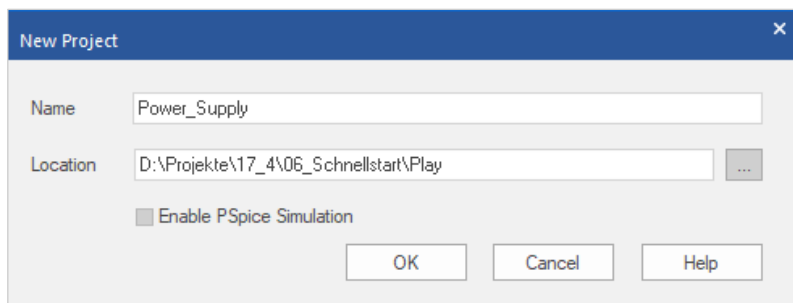
FlowCAD - Germany

# 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

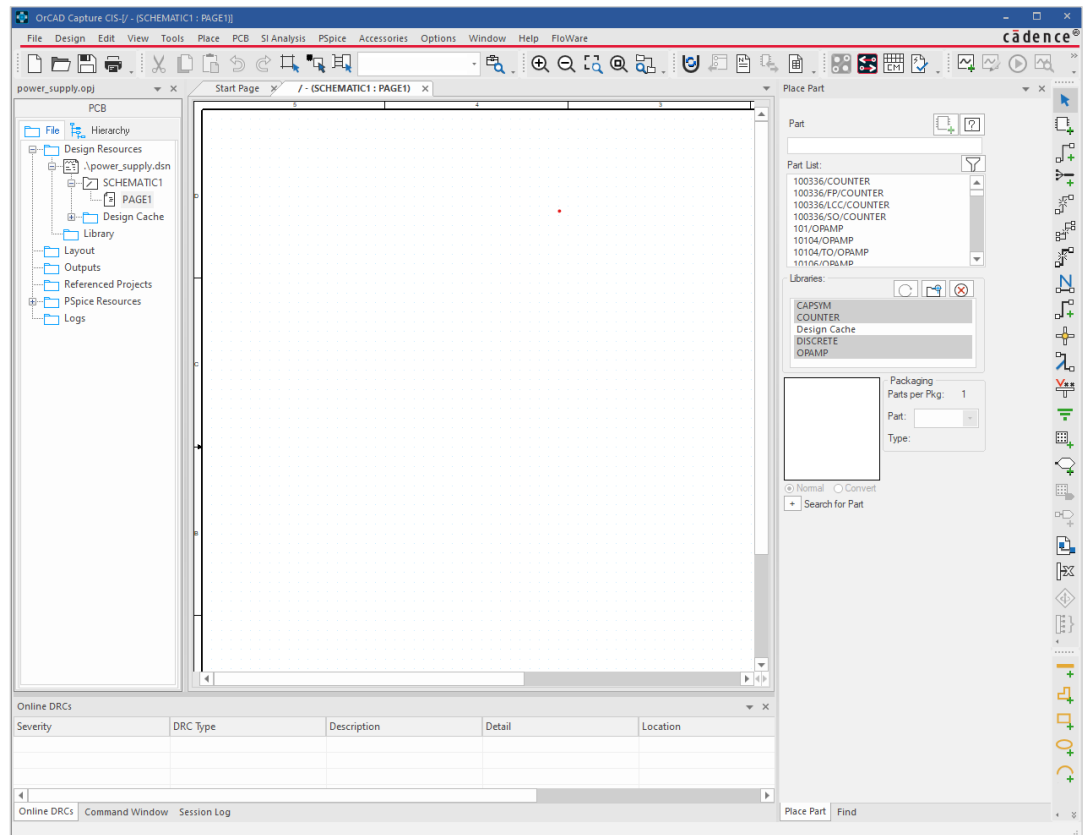
A new project with name **Power\_supply** and a design with same name **power\_supply.dsn** is created.

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.

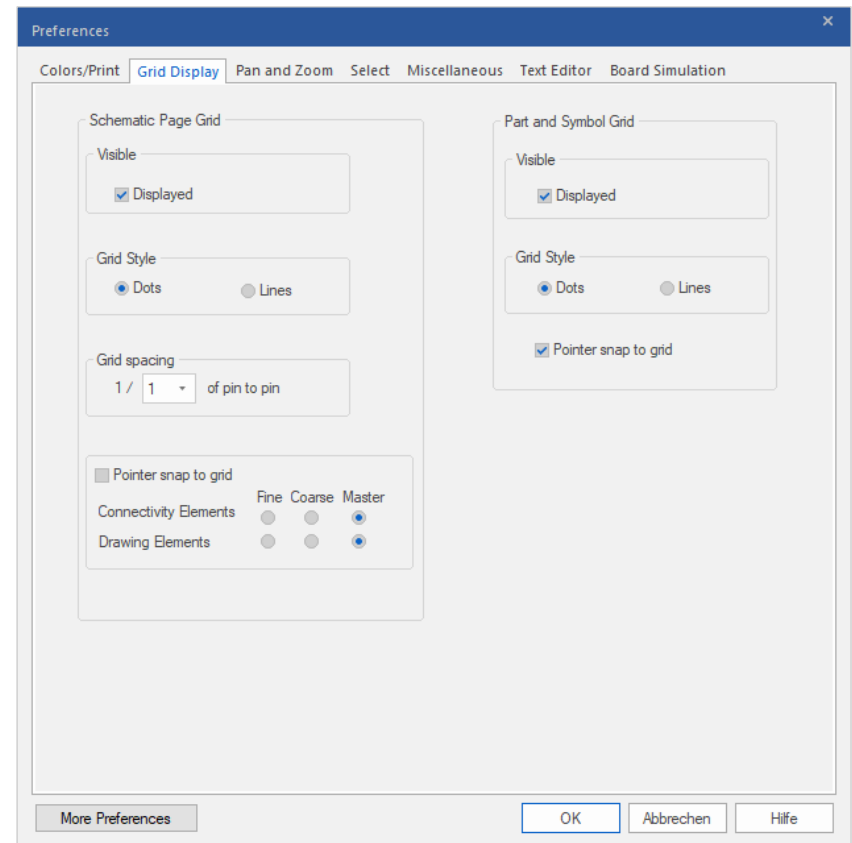
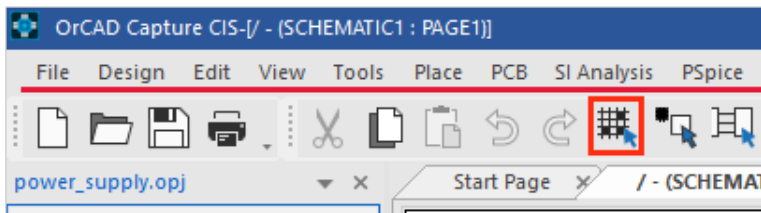




# Setup – Grid Display

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).

The screenshot shows the 'Schematic Page Properties' dialog box with the 'Grid Reference' tab selected. The dialog is divided into three sections: 'Horizontal', 'Vertical', and 'Miscellaneous'. The 'Horizontal' section has a 'Count' of 5, 'Alphabetic' and 'Ascending' radio buttons selected, and a 'Width' of 0.1 inches. The 'Vertical' section has a 'Count' of 4, 'Alphabetic' and 'Descending' radio buttons selected, and a 'Width' of 0.1 inches. The 'Miscellaneous' section has 'Border Visible' (Displayed and Printed checked), 'Grid Reference Visible' (Displayed and Printed checked), 'Title Block Visible' (Displayed and Printed checked), and 'ANSI grid references' checked. At the bottom are 'OK', 'Abbrechen', and 'Hilfe' buttons.

Section	Property	Value
Horizontal	Count	5
	Alphabetic	Selected
	Ascending	Selected
	Width	0.1 inches
Vertical	Count	4
	Alphabetic	Selected
	Descending	Selected
	Width	0.1 inches
Miscellaneous	Border Visible - Displayed	Checked
	Border Visible - Printed	Checked
	Grid Reference Visible - Displayed	Checked
	Grid Reference Visible - Printed	Checked
	ANSI grid references	Checked

## 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.



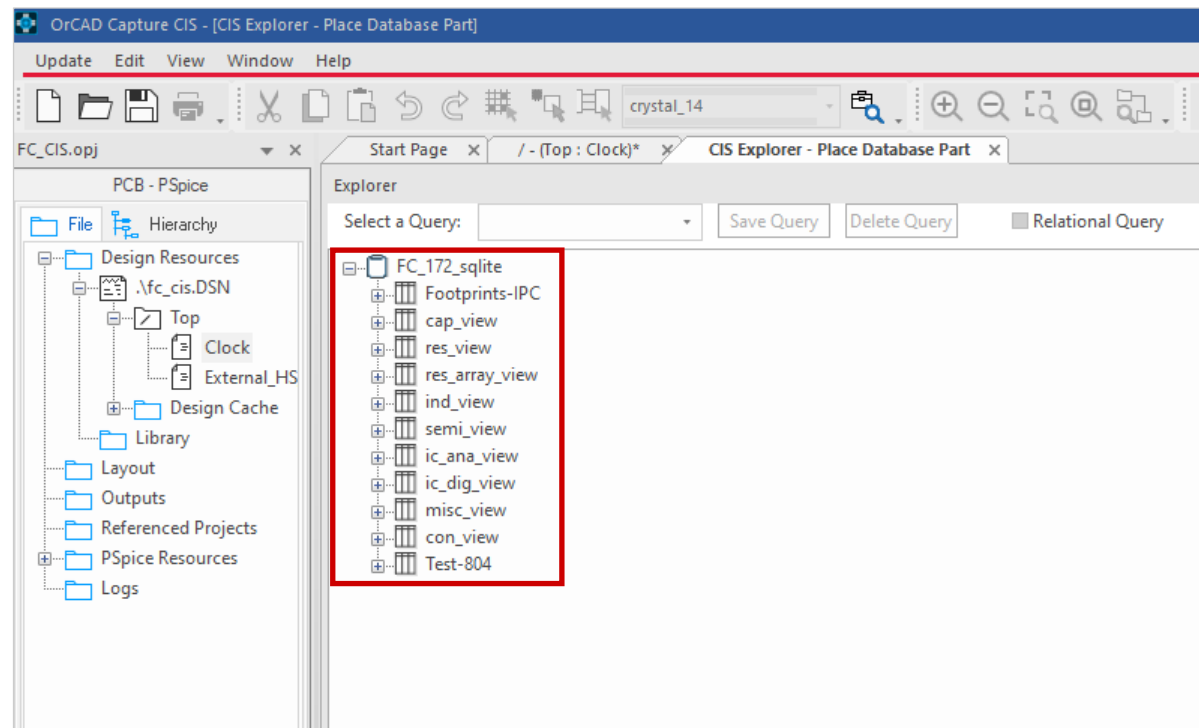
# Part Placement





# 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.

Visibility

	Property	Database Contents	Visible
1	CLASS	IC	✓
2	EDA_Status	verified	✓
3	Part_Status	no	✓
4	STEP_Model	Y_MA306.stp	✓
5	IPC_Footprint		✓

Relation Table

	Table	Part_Number	Comment	Datasheet	Manufactur	Manufactur	Distributor	Distributor
1	Vendor	FC-CRY-0001	Additional Ve	Epson_crysta	Epson	Toyoc		

	Table	Part_Number	Part_Type	Value	Tolerance	Vol	Current	Rating	Description	Schematic Part	PCB Footprint	Height	ALT_SYMBOL	IPC_Footprint	STEP_Model	Part_Status	EDA_Status	CLA
1	misc_view	FC-CRY-0001	Crystal	25 MHz					Low Profile S	fe_MISCICRY	cry_ma306	2.6 mm			Y_MA306.stp	no	verified	IC



# 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				Relation Table								
	Property	Database Contents	Visible		Table	Part Number	Comment	Datasheet	Manufacturer	Manufacturer	Distributor	Distributor
1	CLASS	DISCRETE	<input checked="" type="checkbox"/>		1	Vendor	FC-CAP-0011	Additional Ve	AVX_X7R.pdf	06035C47KA	AVX	
2	EDA_Status	verified	<input checked="" type="checkbox"/>		2	Vendor	FC-CAP-0011	Additional Ve	Cap_panaso	ECJ1EB1H4R	Panasonic	
3	Part_Status	no	<input checked="" type="checkbox"/>									
4	EMBEDDED_PLACEMENT	External_only	<input checked="" type="checkbox"/>									
5	STEP_Model	CAPC1608X86N.step	<input checked="" type="checkbox"/>									

	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 Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
2	cap_view	FC-CAP-0002	X7R	100n	10%	25V		X7R Ceramic	fc_passive\CAP_V	cap_0805	1.0mm	(cap_0805_gd)	CAPC2013X100N
3	cap_view	FC-CAP-0003	X7R	100n	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_1206	1.05mm	(cap_1206_gd)	CAPC3216X105N
4	cap_view	FC-CAP-0004	X7R	3.3n	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0805	1.0mm	(cap_0805_gd)	CAPC2013X100N
5	cap_view	FC-CAP-0007	X7R	1p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
6	cap_view	FC-CAP-0008	X7R	1.5p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
7	cap_view	FC-CAP-0009	X7R	2.2p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
8	cap_view	FC-CAP-0010	X7R	3.3p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
9	cap_view	FC-CAP-0011	X7R	4.7p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
10	cap_view	FC-CAP-0012	X7R	6.8p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
11	cap_view	FC-CAP-0013	X7R	10p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
12	cap_view	FC-CAP-0014	X7R	15p	10%	50V		X7R Ceramic	fc_passive\CAP_V	cap_0603	0.9mm	(cap_0603_gd)	CAPC1608X90N
13	cap_view	FC-CAP-0015	X7R	22p	10%	50V		X7R Ceramic	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.

Start Page x / - (Top : Clock)\* x CIS Explorer - Place Database Part x

Explorer

Select a Query:  Save Query Delete Query ☐ Relational Query

	Property	Compare	Value
1	Part_Type	=	X7R
2	Value	=	4.7pF
3			

Part

Graphic

☐ Normal

☐ Convert

Packaging

Parts Per Pkg:

Part:

Explore **Query**

Visibility

	Property	Database Contents	Visible
--	----------	-------------------	---------

Relation Table

	Table	Part_Numbe	Comment	Datasheet	Manufactur	Manufactur	Distributor	Distri
--	-------	------------	---------	-----------	------------	------------	-------------	--------

	Table	Part_Number	Part_Type	Value	Tolerance	Vol	Impedance	Description	Schematic Part
1	cap_view	FC-CAP-0011	X7R	4.7p	10%	50V		X7R Ceramik Capacitor 4.7p 10% 50V 0603 SMD	fc_passiveICAP_V
2	cap_view	FC-CAP-0076	X7R	4.7p	10%	50V		X7R Ceramik Capacitor 4.7p 10% 50V 0402 SMD	fc_passiveICAP_V



# 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

The screenshot shows the FlowCAD CIS Explorer interface. The 'Explorer' pane on the left displays a tree structure with 'FC\_172\_sqlite' as the root, containing 'Footprints-IPC' and 'cap\_view'. The 'cap\_view' category is selected, showing a list of parts. The 'Part' pane on the right shows the 'Graphic' section with 'Normal' and 'Convert' options, and the 'Packaging' section with 'Parts Per Pkg' and 'Part' dropdowns. The 'Relation Table' pane at the bottom displays a table of parts with their status.

Part_Type	Value	Tolerance	Vol	Impedance	Description	Schematic Part	PCB Footprint	Height	ALT_SYMBOLS	IPC_Footprint	STEP_Model	EMBEDDED_PLACEMENT	Part_Status	EDA_Status	
1	ELEC	220u	20%	50V	Aluminium Ele	fc_passive(CAPPOL_T	capp_th_r_rm	25.0mm		CAPPRR500W60L1250T1250H2500N		External_only	obsolete	verified	DIS
2	ELEC	2200u	20%	25V	Aluminium Ele	fc_passive(CAPPOL_T	capp_th_s_r	17.0mm		CAPPAD5080W80L4000D1600N		External_only	standard	verified	DIS
3	ELEC	100u	20%	50V	Aluminium Ele	fc_passive(CAPPOL_T	capp_th_r_rm	20.0mm		CAPPRR500W60L1000T1000H2000N		External_only	special	verified	DIS



# 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.



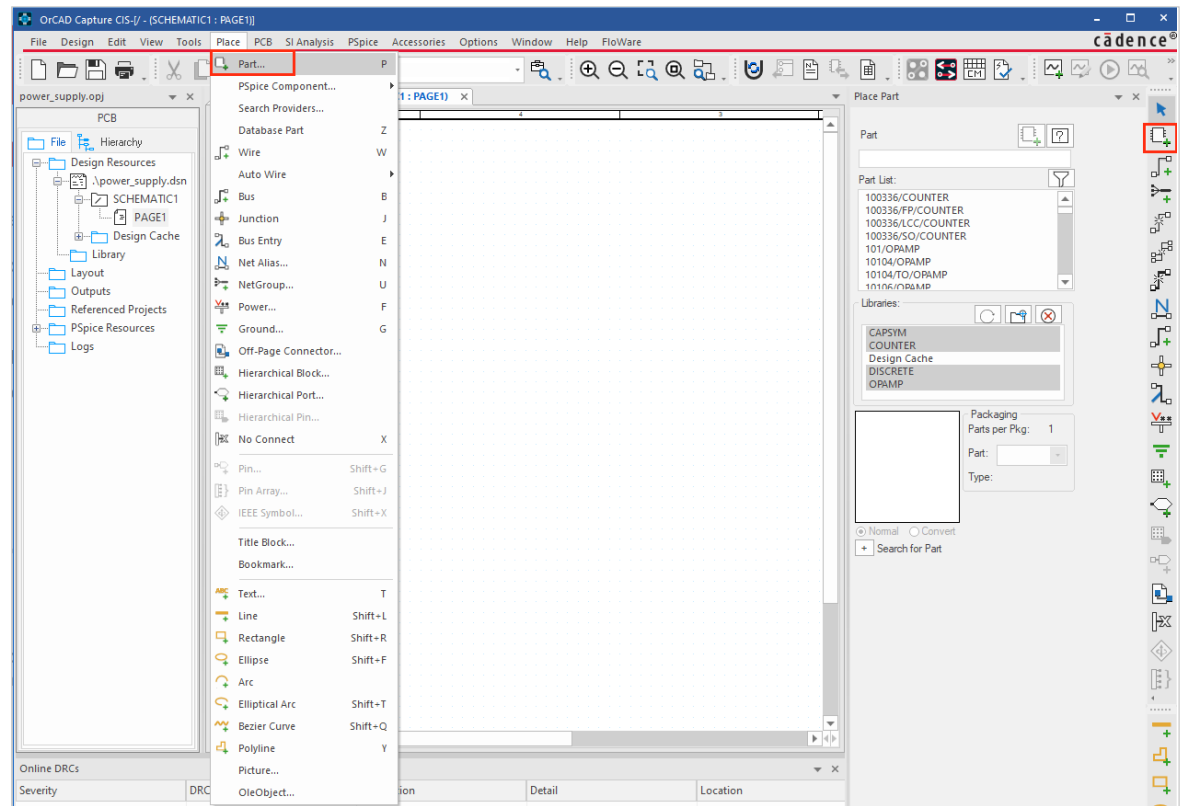
# Part Placement from Library (II)

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).

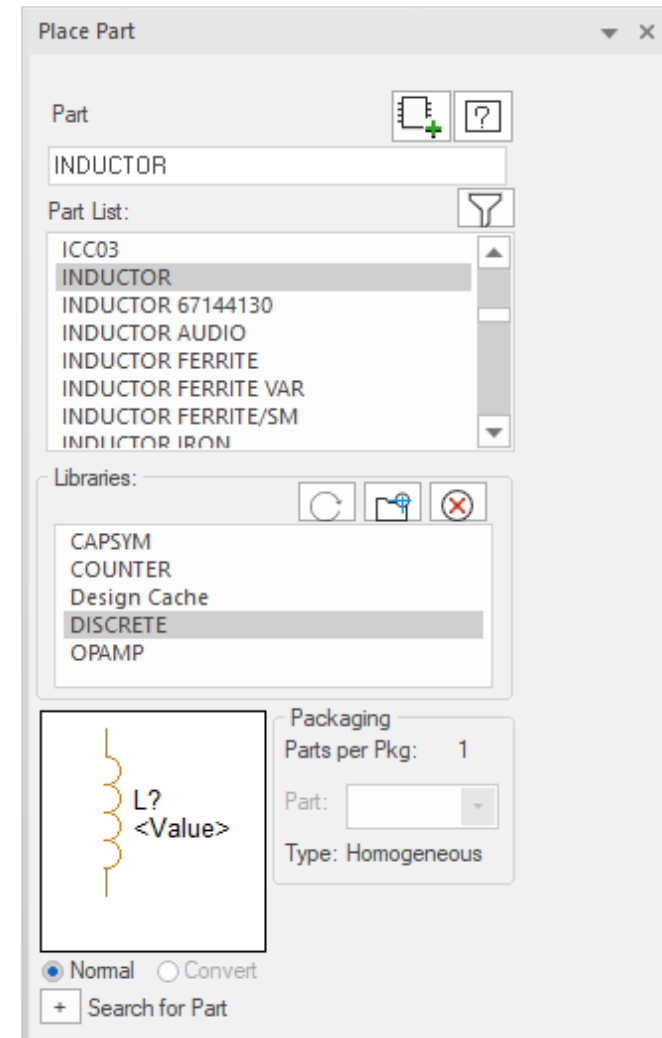







# Part Placement from Library (III)

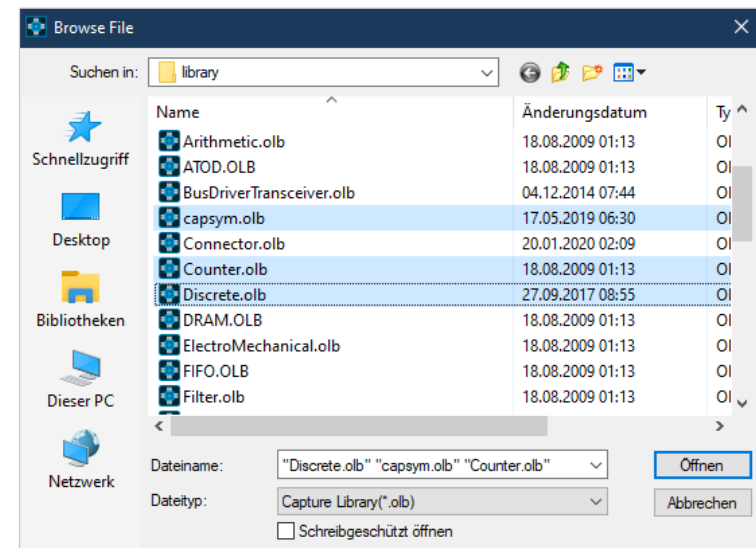
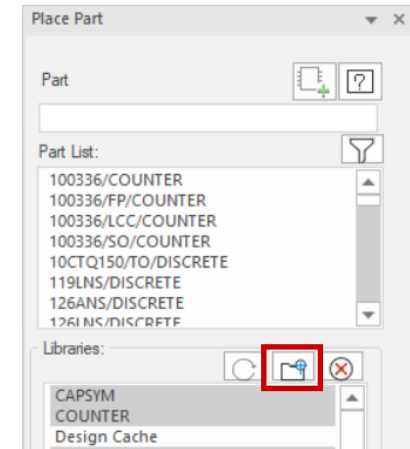
- Placement command opens menu as shown.
- In **Libraries** you can select one or more libraries in which your part can be searched.
- Enter part label in **Part**. This already behaves as a filter, but without usage of wildcards “\*”.
- Search results and related library are visible in **Part List**.
- **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

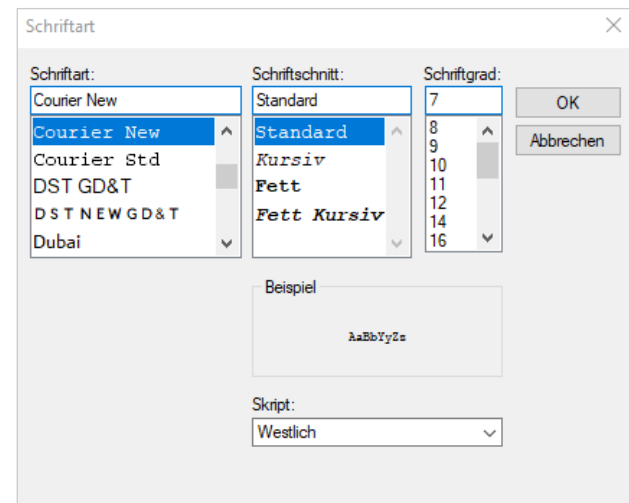
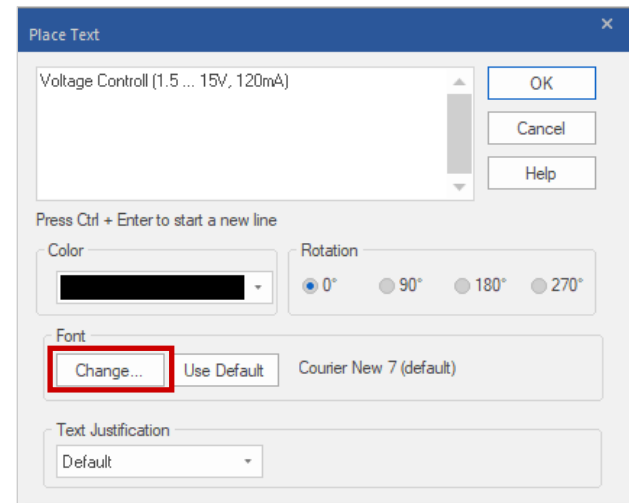


# Text – Adding and Edit

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.



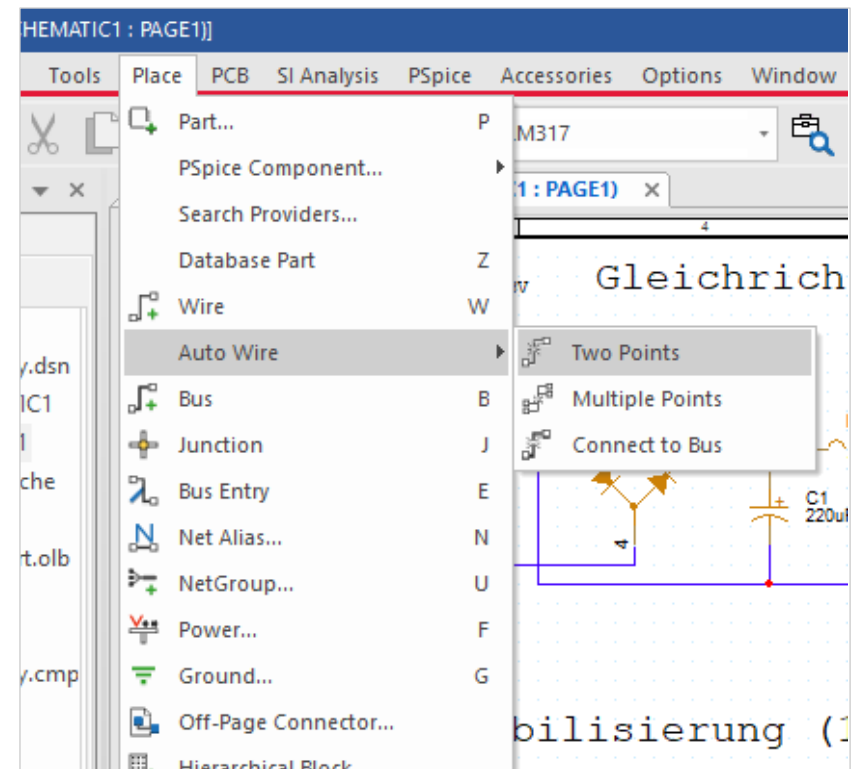
## Multiple Points

- Multiple selected pins are automatically connected to each other.



## Connect to Bus

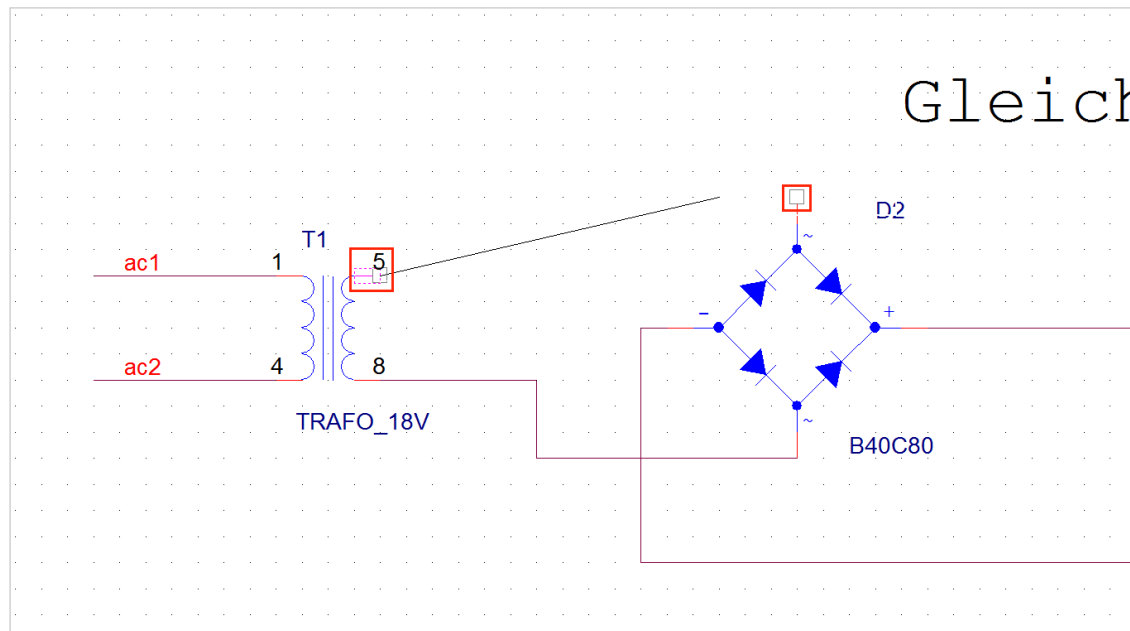
- Multiple selected lines are connected to a bus.





# Connections (II) – Auto Wire


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





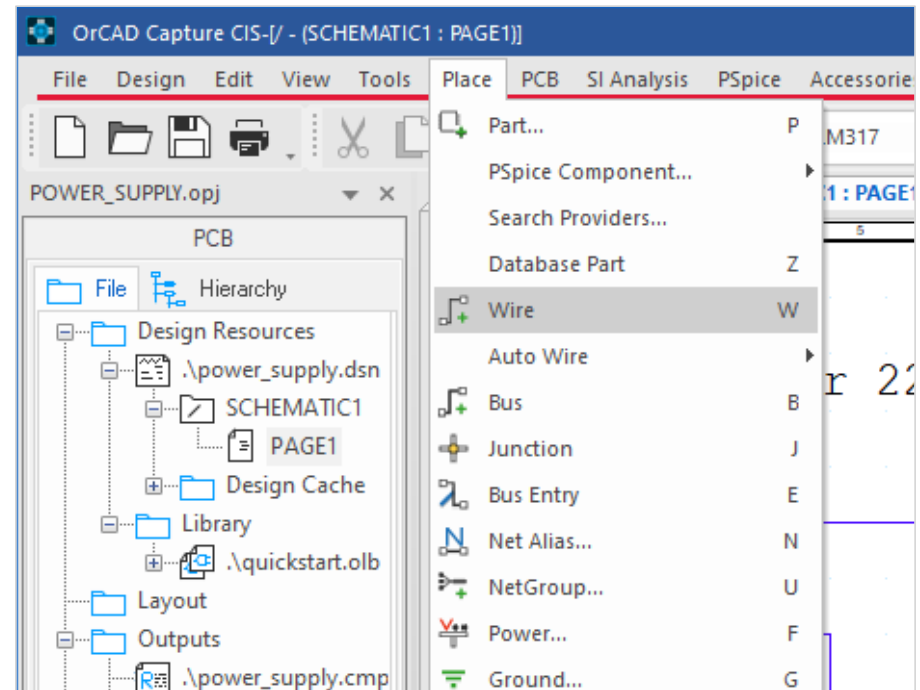
# Connections (III) – Manually

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.

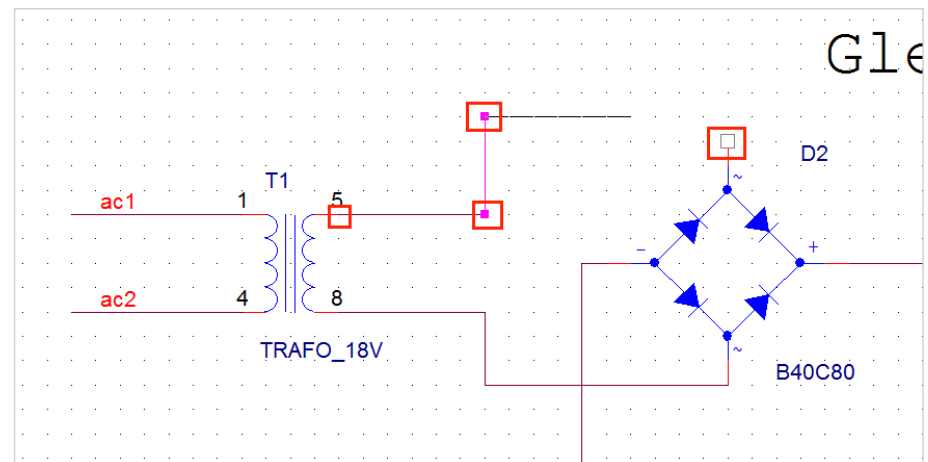
Red dot signalizes a pin which is available for a connection. **Another click** on this object completes connection. Red dot and square will disappear. A **double click** will complete the task and release all objects.

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

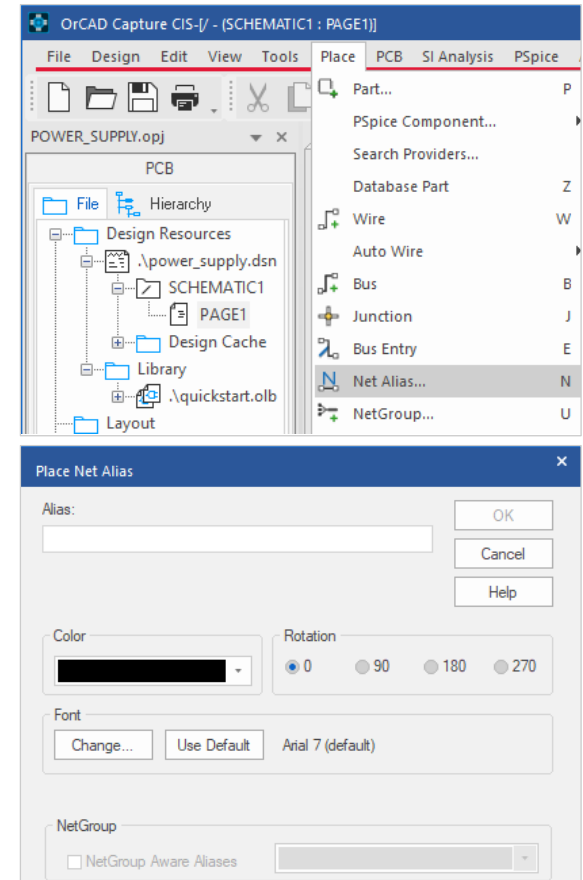
To connect parts on one schematic page, **Place > Wire** command is used to perform wiring.

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

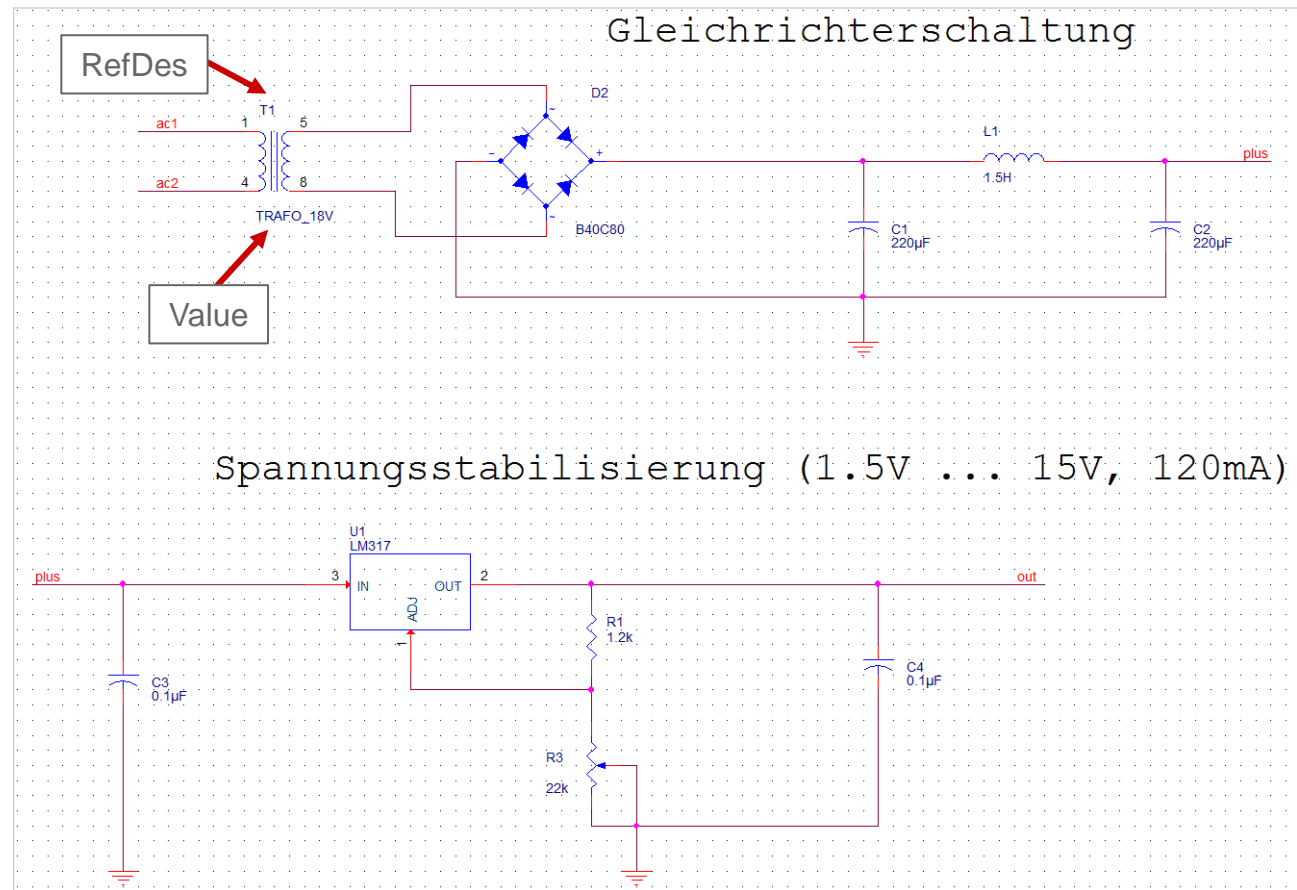


# Edit Properties (I)

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.

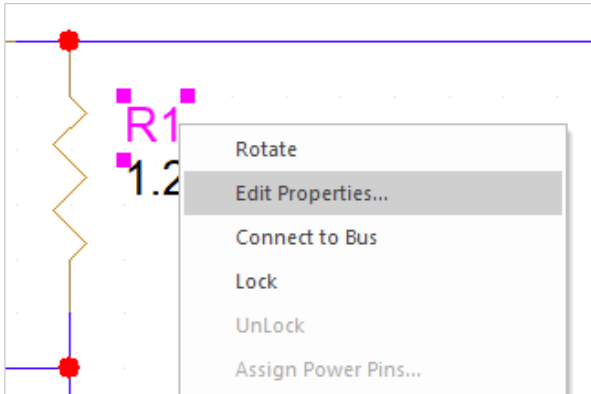
To correct this, **values** and **REFDES** of parts can be edited. This makes also sense for a bill of materials generated later in the process.





# Edit Properties (II)

Edit REFDES and value



Property window can be opened by

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

or by

**Double click**

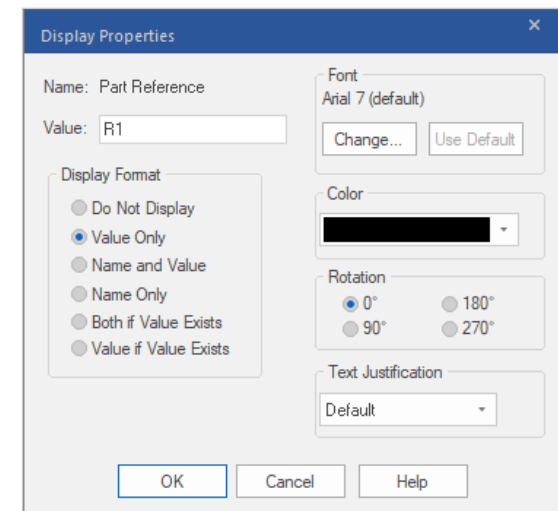
on desired property.

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.

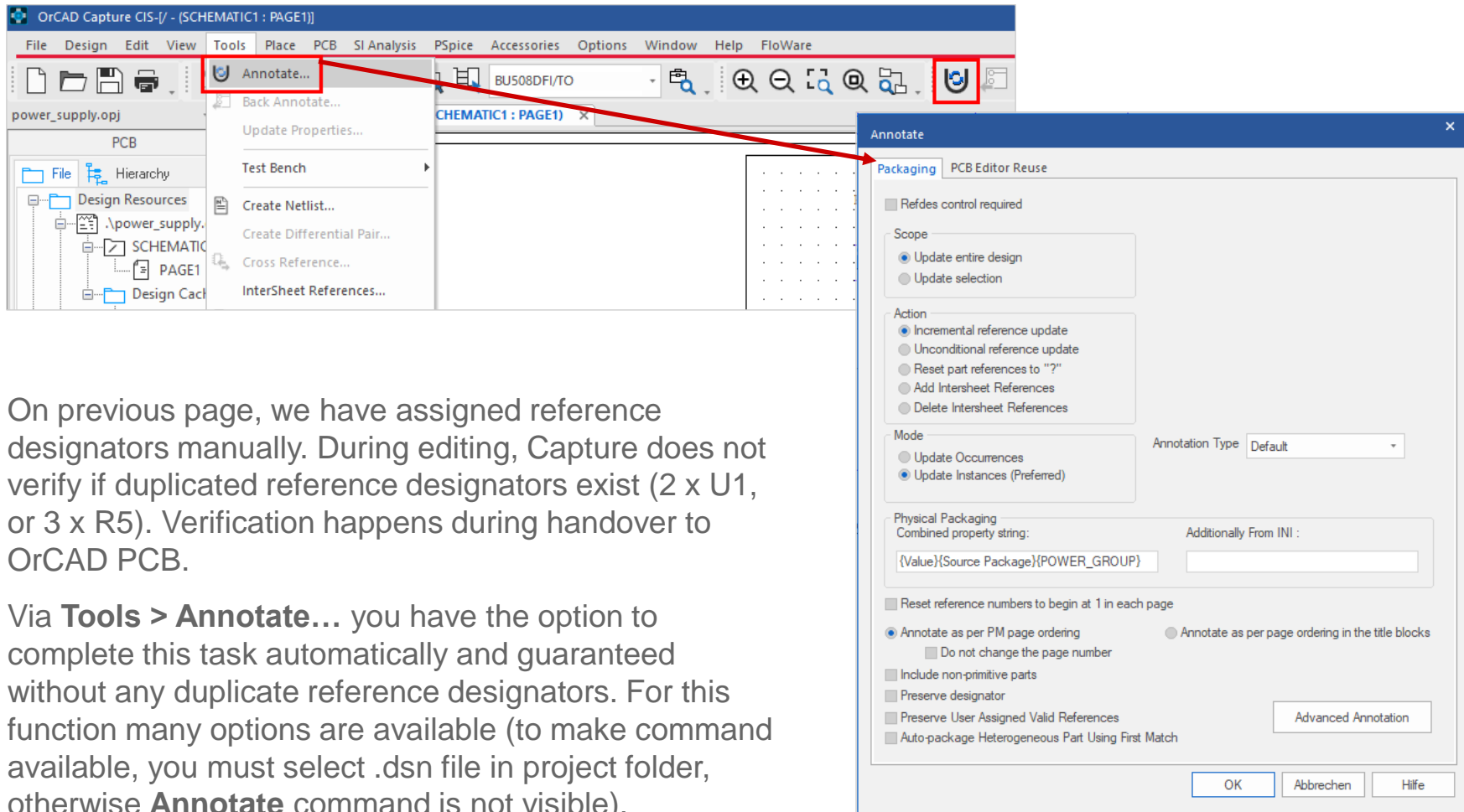
- **Ctrl + LMB Click** or **Ctrl + A** followed by **RMB > Properties...**
- Select Page or Design (.dsn) in Project Manager, **Edit Object Properties** from pull-down menu.





# Packaging – Annotation

Packaging and annotation of parts



The screenshot shows the OrCAD Capture CIS interface. The 'Tools' menu is open, and the 'Annotate...' option is highlighted. A red arrow points from this menu item to the 'Annotate' dialog box. The dialog box has two tabs: 'Packaging' and 'PCB Editor Reuse'. The 'Packaging' tab is active, showing options for 'Refdes control required', 'Scope' (Update entire design, Update selection), 'Action' (Incremental reference update, Unconditional reference update, Reset part references to "?", Add Intersheet References, Delete Intersheet References), 'Mode' (Update Occurrences, Update Instances (Preferred)), 'Annotation Type' (Default), 'Physical Packaging' (Combined property string: {Value}{Source Package}{POWER\_GROUP}, Additionally From INI:), 'Reset reference numbers to begin at 1 in each page', 'Annotate as per PM page ordering' (Do not change the page number, Annotate as per page ordering in the title blocks), 'Include non-primitive parts', 'Preserve designator', 'Preserve User Assigned Valid References', and 'Auto-package Heterogeneous Part Using First Match'. There are buttons for 'OK', 'Abbrechen', and 'Hilfe' at the bottom.

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

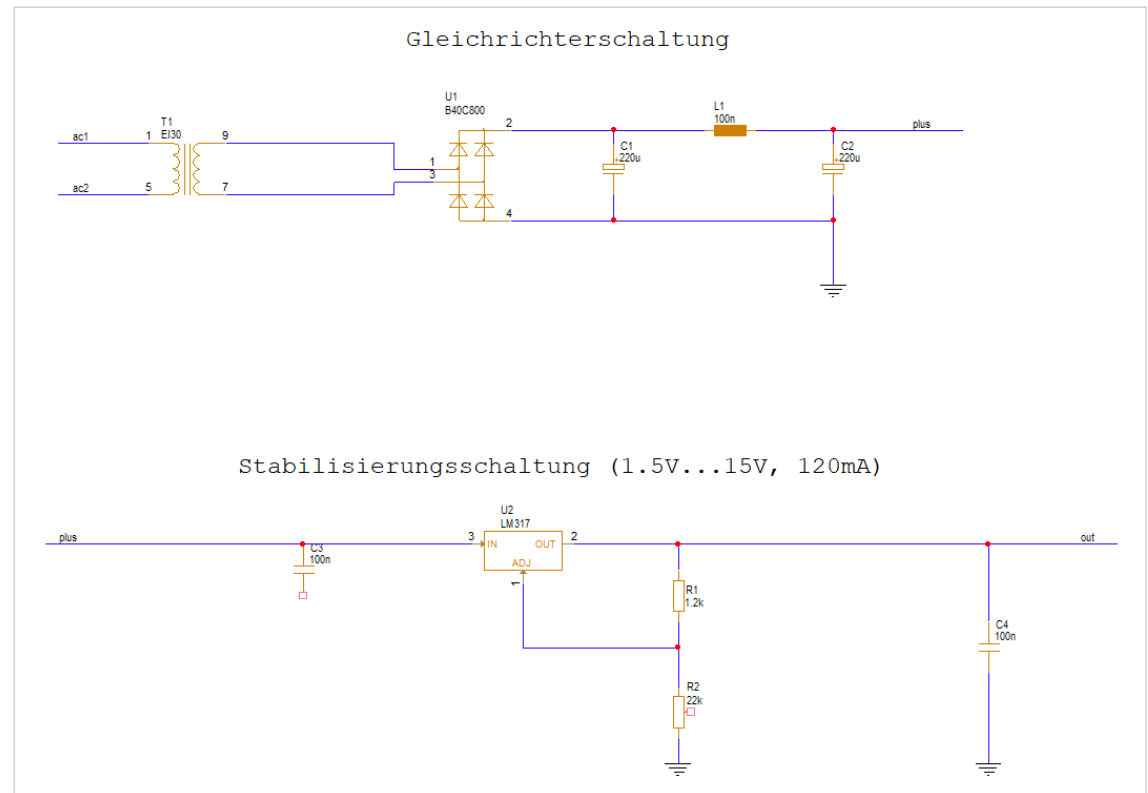
At this point we would like to mention that because of used libraries assignment of footprints (DIP14, SO14, SMD1206, etc.) is not done so far.

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

Just a upfront information:

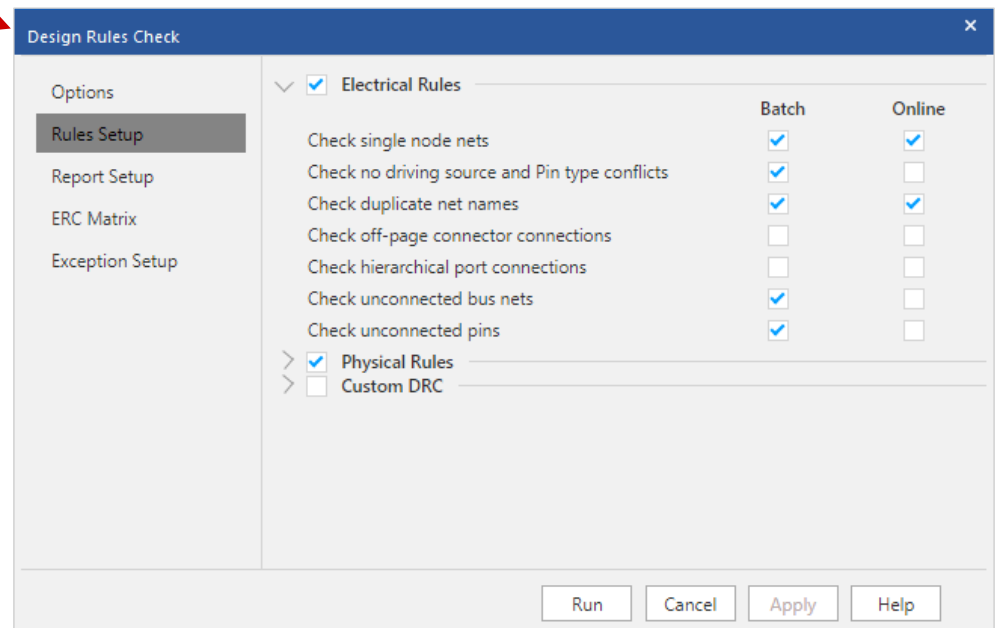
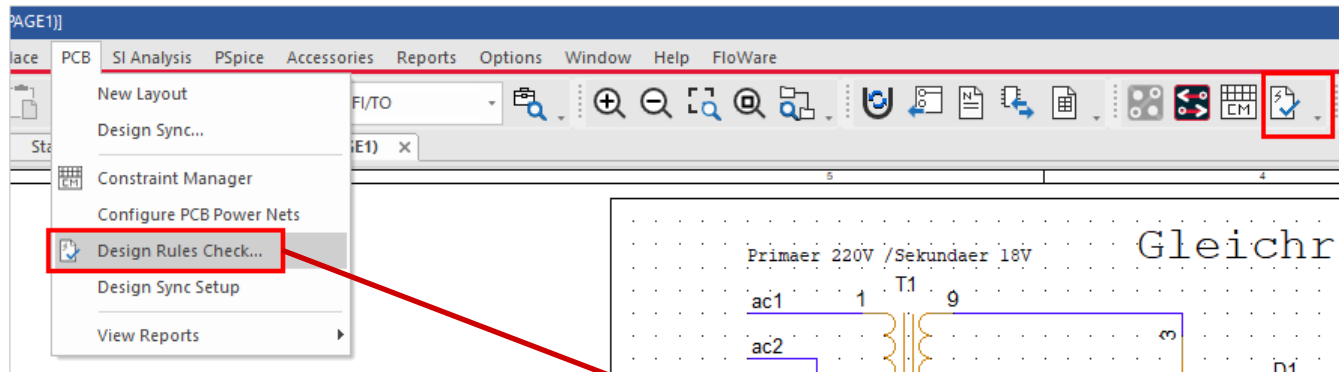
Footprint assignment has to be done in schematic. A missing footprint entry will cause an error message during data synchronization. If name is not finalized, it is possible to use a dummy text string as a placeholder.

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)

**Bill of Materials**

Scope: ☒ Process entire design, ☐ Process selection

Mode: ☒ Use instances (Preferred), ☐ Use occurrences

Line Item Definition

Header: Item\Quantity\Reference\Part

Combined property string: {Item}\{Quantity}\{Reference}\{Value}

☐ Place each part entry on a separate line ☐ Open in Excel

Include File

☐ Merge an include file with report

Combined property string: {Item}\{Quantity}\{Reference}\{Value}

Include file: D:\PROJEKTE\17\_4\06\_SCHNELLSTART\CA Browse...

Report

Report File: D:\PROJEKTE\17\_4\06\_SCHNELLSTART\CAP View Output Browse...

**POWER\_SUPPLY.BOM**

Revised: Thursday, April 23, 2020

Revision:

Item	Quantity	Reference	Part
1	2	C1,C2	220uF
2	2	C3,C4	0.1uF
3	1	D1	B40C800
4	1	L1	1.5H
5	1	R1	1.2k
6	1	R2	22k
7	1	T1	Trafo_18V
8	1	U1	LM317

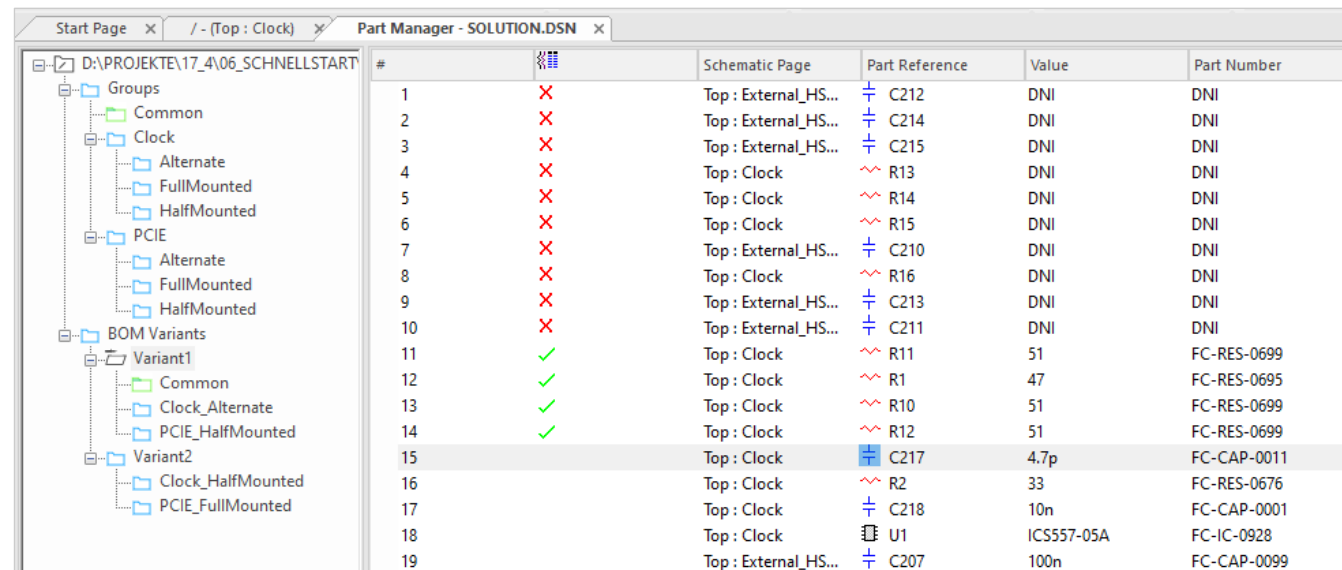


# Design Variants



# 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.



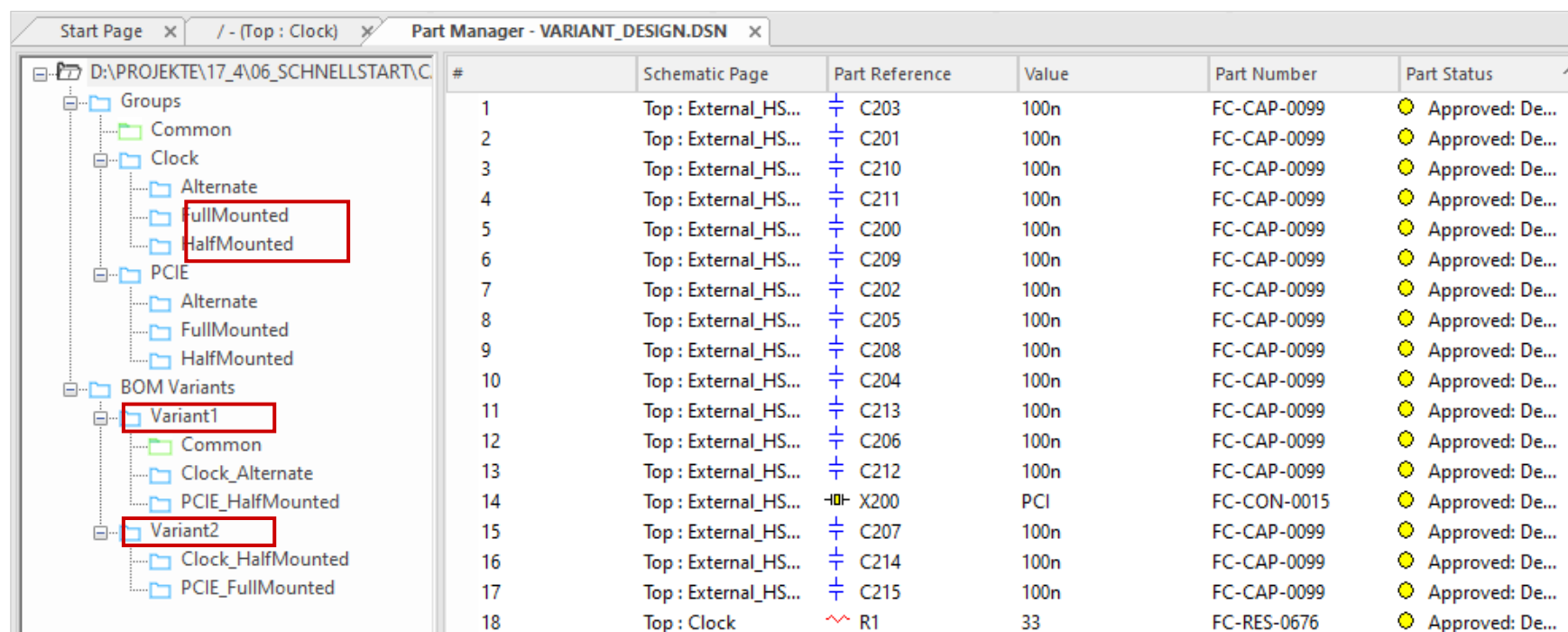
The screenshot shows the 'Part Manager - SOLUTION.DSN' window. On the left is a tree view of the project structure, and on the right is a table listing 19 design variants.

#		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.

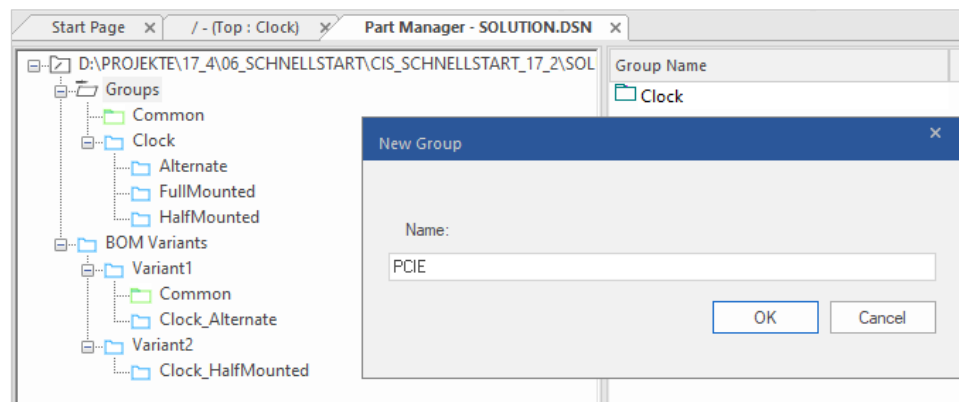


#	Schematic Page	Part Reference	Value	Part Number	Part Status
1	Top : External_HS...	⚡ C203	100n	FC-CAP-0099	⚡ Approved: De...
2	Top : External_HS...	⚡ C201	100n	FC-CAP-0099	⚡ Approved: De...
3	Top : External_HS...	⚡ C210	100n	FC-CAP-0099	⚡ Approved: De...
4	Top : External_HS...	⚡ C211	100n	FC-CAP-0099	⚡ 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

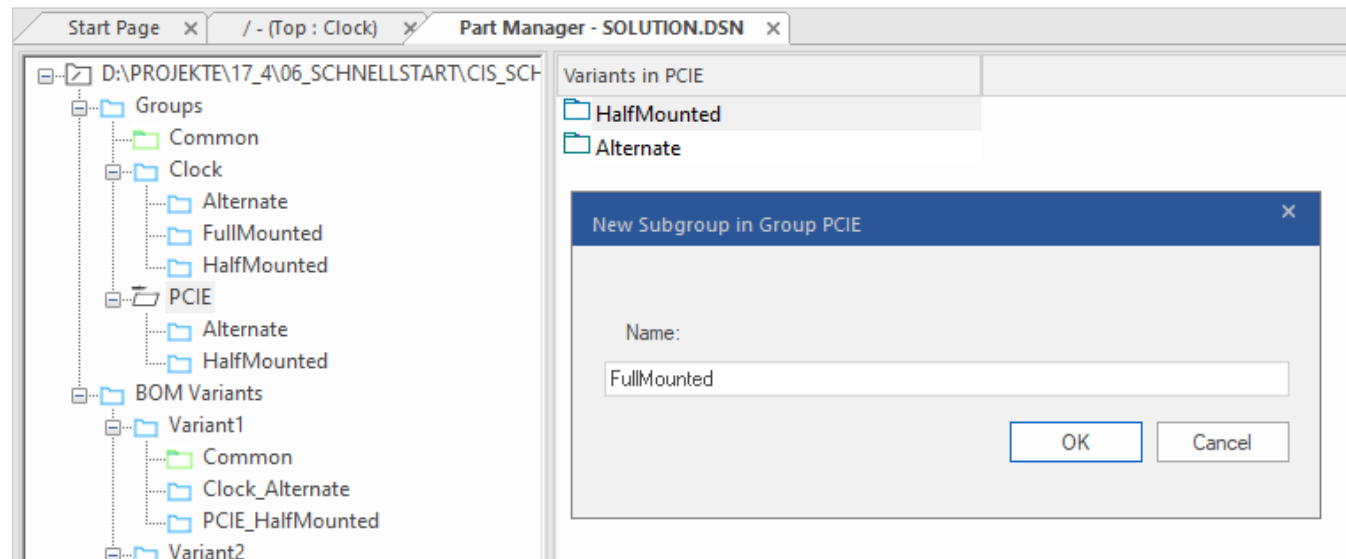
- With **Part Manager** you can manage parts in your design, carry out database updates or generate variants.
- Select .dsn file and click **RMB > Part Manager** or use menu **Tools > Part Manager > Open**.
- To describe variants, you need to divide the design into groups.
- It makes sense to store all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

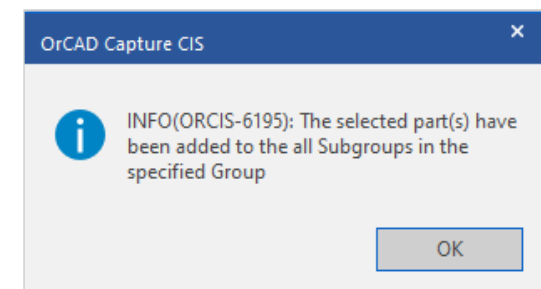
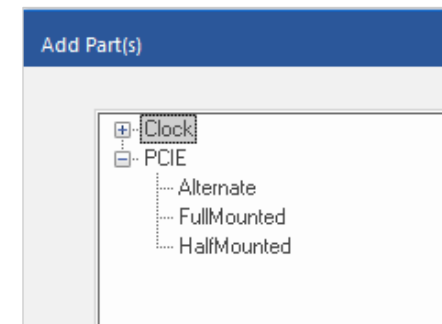
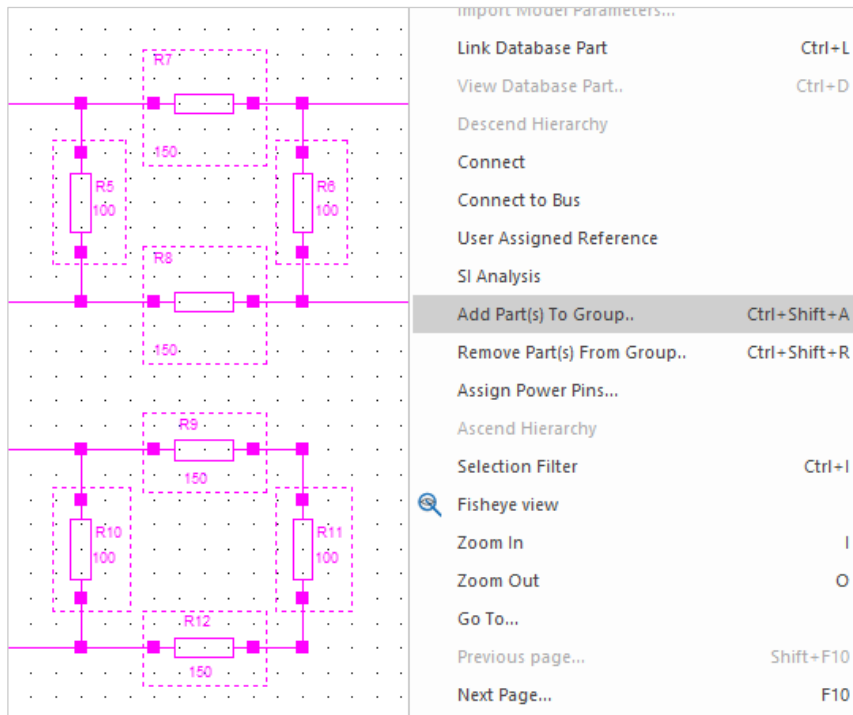
- Next step is to define subgroups that contain different assembly variants of groups.
- Select the group **Clock > RMB > New Subgroup**.
- Enter **Alternate**.
- Repeat this process, and enter **HalfMounted** and then **FullMounted**.





# Variants – Assign Parts to Groups

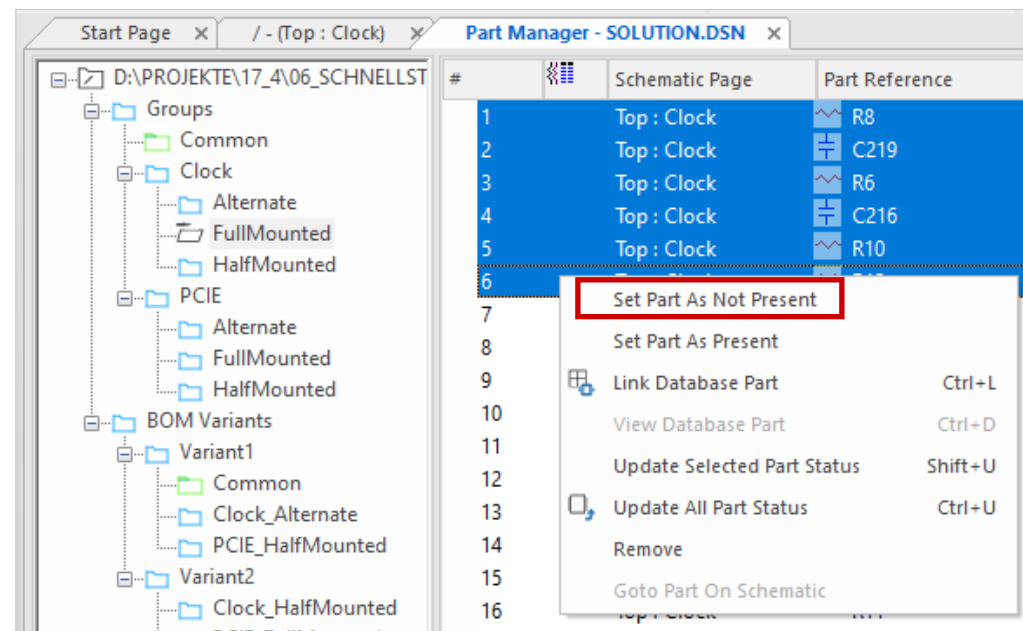
- Open **Clock** page and select all parts.
- **RMB > Add Part to Group**, select the group **Clock > Add**.
- Repeat this action with second page and move all capacitors (not connector) into group **PCIE**.





# 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.

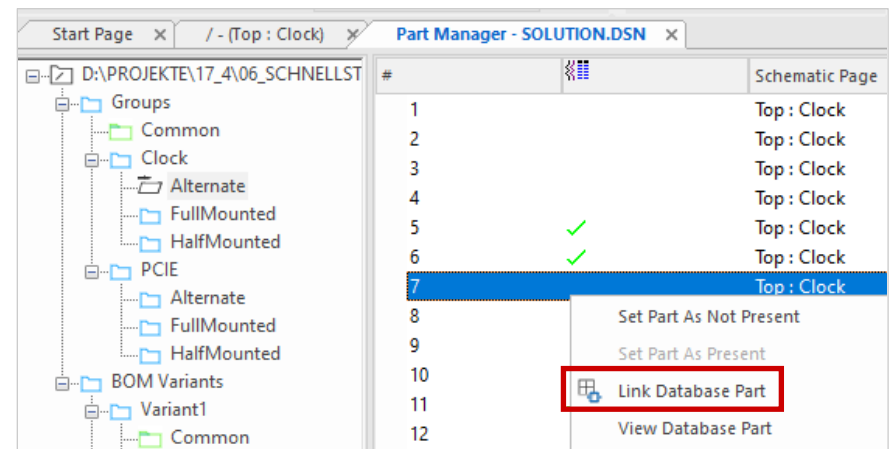






# Variants – Alternative Placement

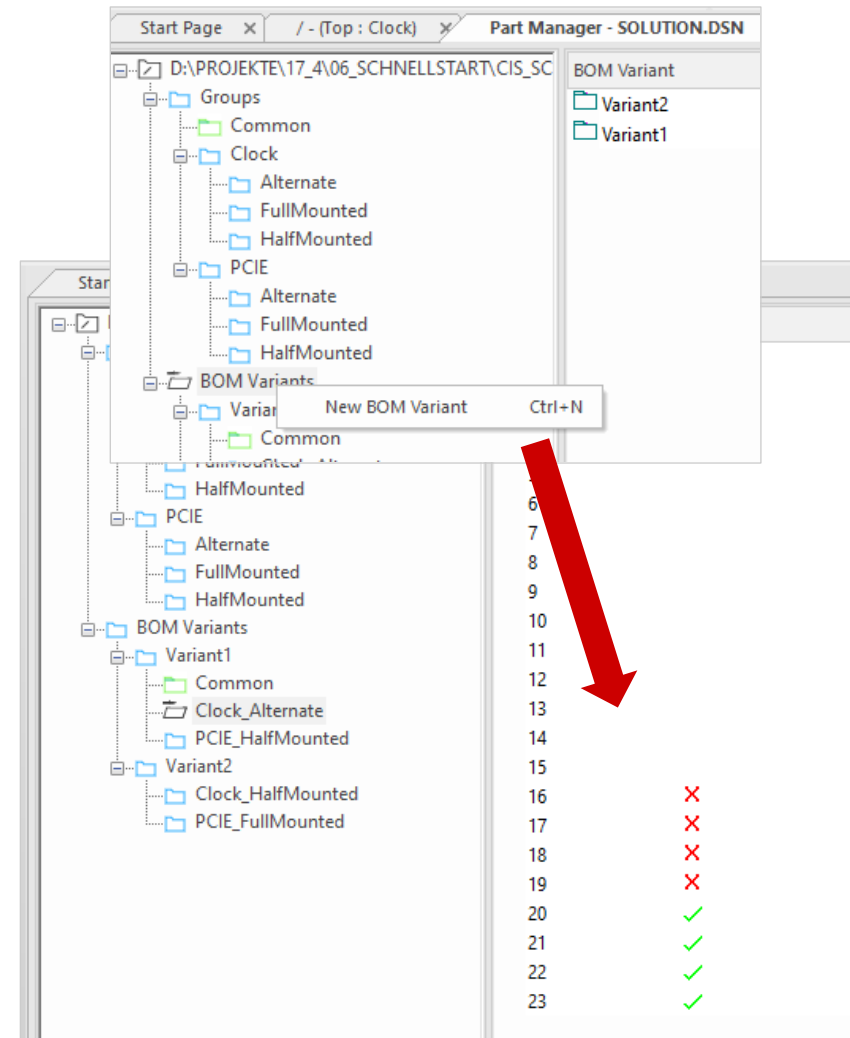
- Go to **Alternate** subgroup and select a part, preferably a resistor or capacitor **RMB > Link Database Part**.
- CIS Explorer opens, and an alternative part can be selected. Make sure that parts have identical symbols and footprints. This can be ensured, for example, by a query search in which footprint 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

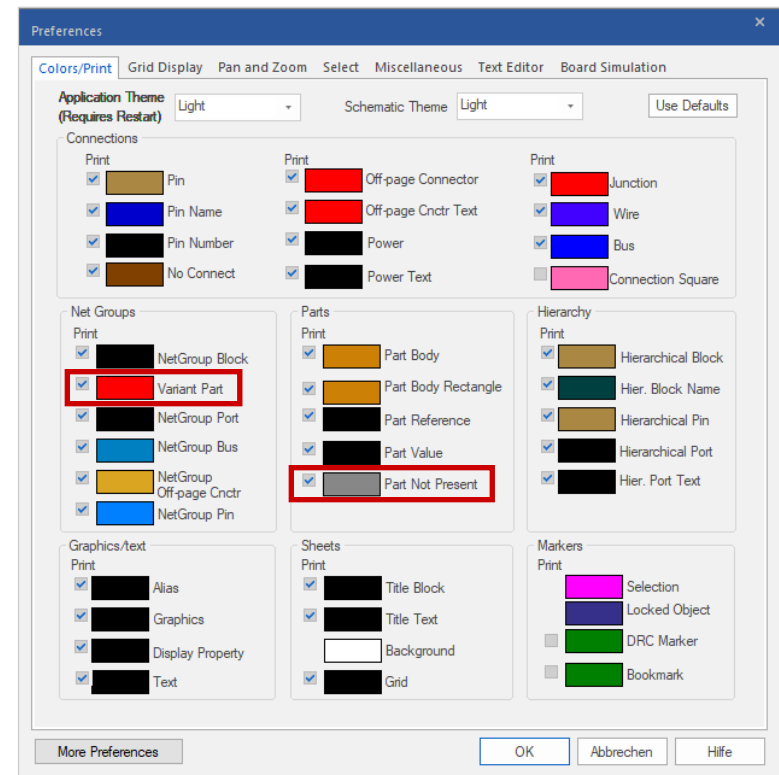
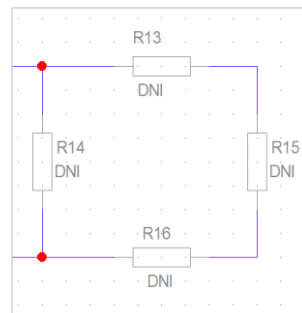
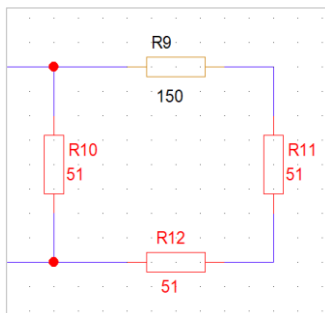
- Select **BOM Variants > RMB > New BOM Variants** and enter name **BOM1**. Repeat for **BOM2** and **BOM3**.
- Drag one subgroup per group into BOM. Finally, common group must be added to BOM, as it contains PCIE connector, that has not been assigned to any group.
- Variant BOM is shown in lower right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





# Variants – Schematic

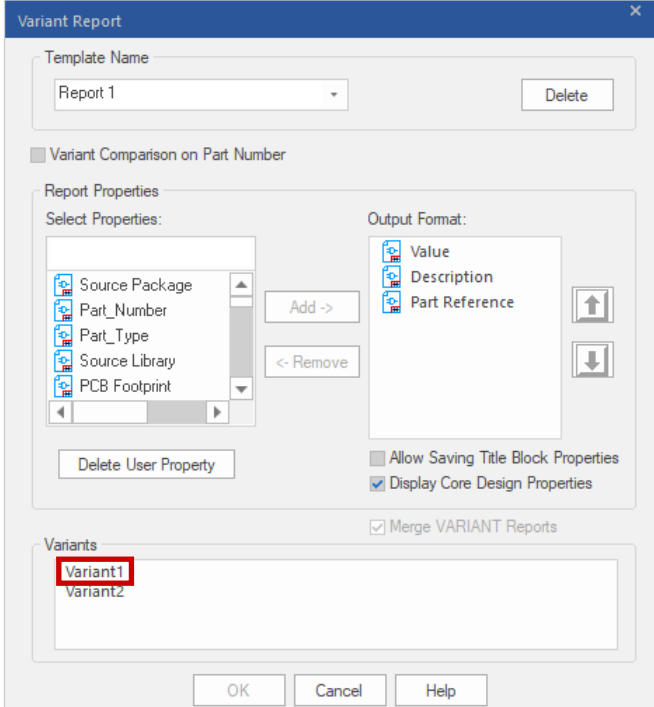
- Close Part Manager, go to a schematic page with variant description and select **View > Variant View Mode**. Select one of the variants. Unmounted parts are displayed in grey with lettering DNI, parts with an alternative placement are displayed in green.
- You can set colors via **Options > Preferences**.





# 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.



The Variant Report dialog box is shown with the following settings:

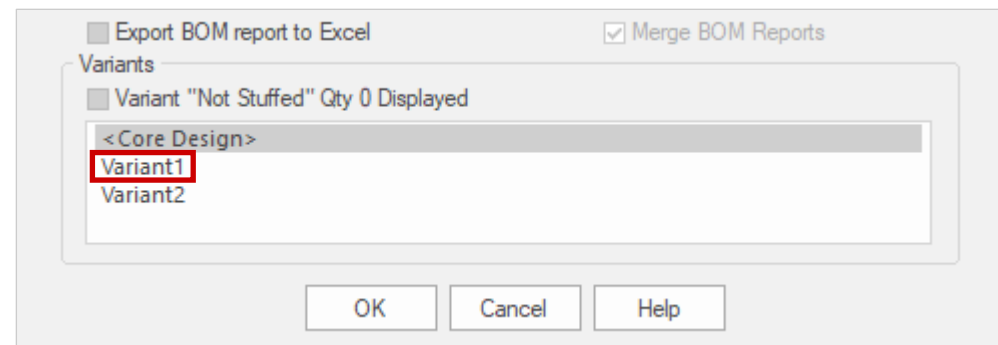
- Template Name: Report 1
- Variant Comparison on Part Number: ☐
- Report Properties:
  - Select Properties: Source Package, Part\_Number, Part\_Type, Source Library, PCB Footprint
  - Output Format: Value, Description, Part Reference
  - Allow Saving Title Block Properties: ☐
  - Display Core Design Properties: ☒
- Merge VARIANT Reports: ☒
- Variant: Variant1

Start Page x / - (Top : Clock) Variant1 x SOLUTION.VRT* x					
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 Resistor 51r 1% 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




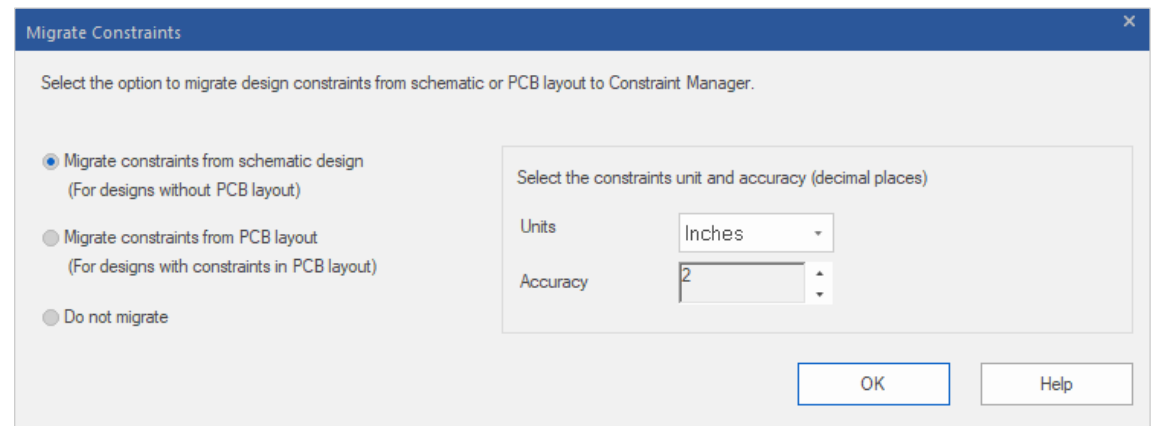
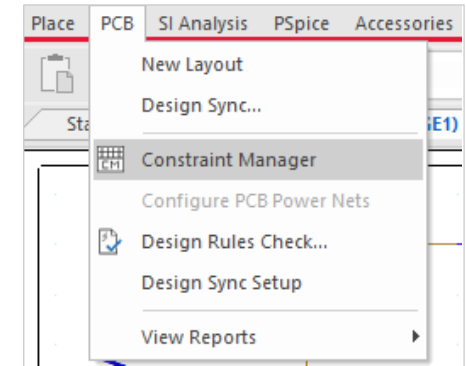
# Constraint Manager in Capture

- 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**.

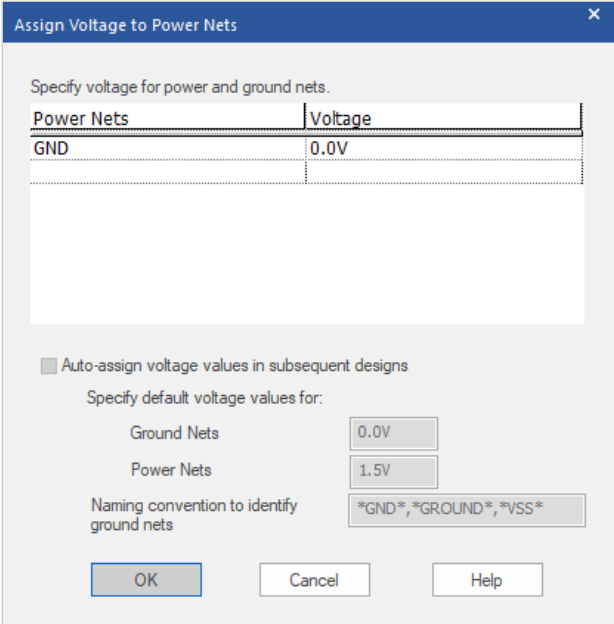






# Assign Voltages to Power Nets

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



Assign Voltage to Power Nets

Specify voltage for power and ground nets.

Power Nets	Voltage
GND	0.0V

☐ Auto-assign voltage values in subsequent designs

Specify default voltage values for:

Ground Nets

Power Nets

Naming convention to identify ground nets

OK Cancel Help



# 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.

Constraint Manager (connected to CAPTURE) [POWER\_SUPPLY] - [Electrical / Net / Routing]

File Edit Objects Column View Audit Tools Window Help

Worksheet Selector

Electrical

Electrical Constraint Set

- Routing
  - Wiring
  - Impedance
  - Min/Max Propagation Delays
  - Total Etch Length
  - Differential Pair
- Net
  - Routing
    - Wiring
    - Impedance
    - Min/Max Propagation Delays
    - Total Etch Length
    - Differential Pair
    - Relative Propagation Delay

POWER\_SUPPLY

Objects			Referenced Electrical CSet	Single-line Impedance				
Type	S	Name		Target	Tolerance	Actual	Margin	
				Ohm	Ohm	Ohm	Ohm	
*	*	*	*	*	*	*	*	
Dsn		POWER_SUPPLY						
Net		AC1						
Net		AC2						
XNet		ADJUST						
Net		N30154						
Net		N30162						
XNet		PLUS						



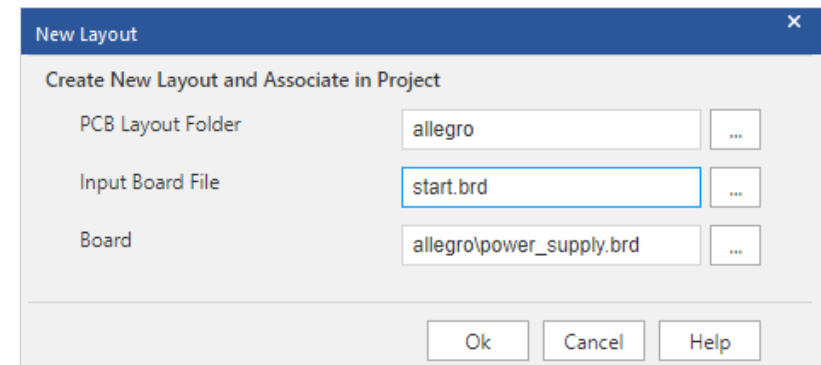
# Design Sync



# Generate Layout (I)

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

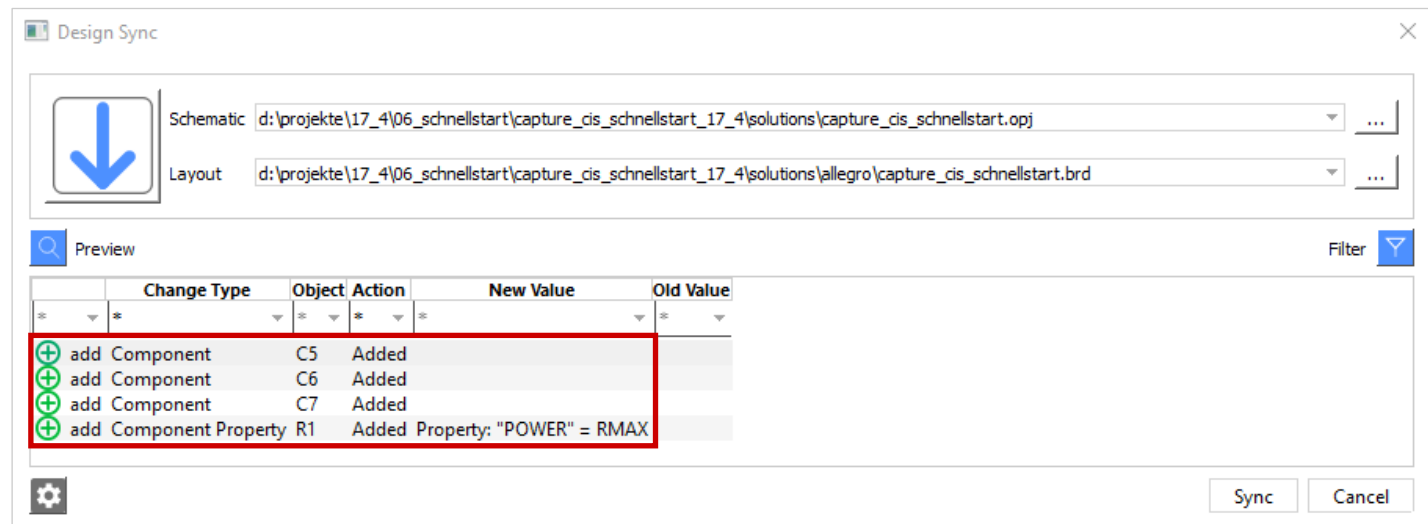
- PCB Layout Folder: Directory in which design sync information is written
- Input Board File: A base or board template
- Board: Newly generated board file





# Generate Layout (II)

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



# Libraries



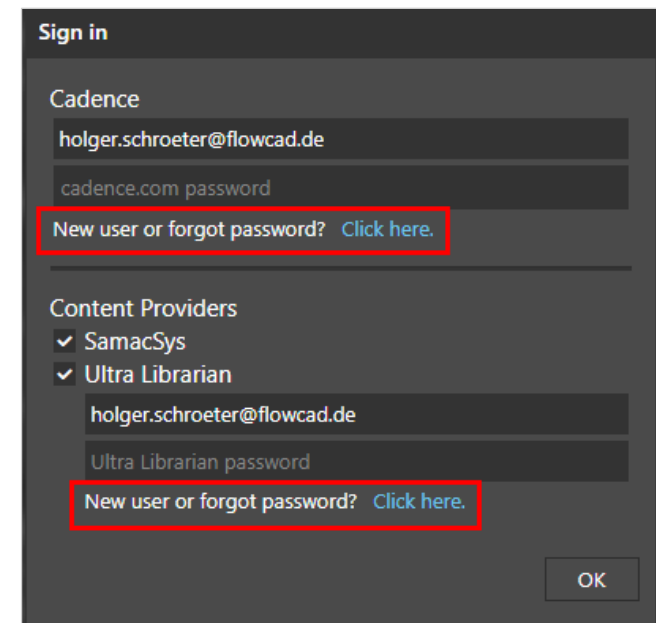
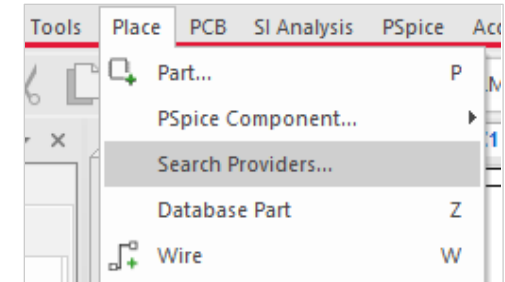
# Missing Parts

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.

SamacSys Ultra Librarian

Filter

MPN	MANUFACTURER	CATEGORY	DESCRIPTION
0022057028	Molex	Connector	Headers & Wire Housings KK .100" HDR 02P RA SN
0022284360	Molex	Connector	CONN HEADER 36POS .100 VERT TIN
0022232081	Molex	Connector	Molex KK 254 Series Number 6373, 2.54mm Pitch 8 Way 1 Row Straig...
0022232021	Molex	Connector	Conn Wire to Board HDR 2 POS 2.54mm Solder ST Thru-Hole KK® B...
0022112032	Molex	Connector	Conn Wire to Board HDR 3 POS 2.54mm Solder ST Thru-Hole Tube - ...
0436500228	Molex	Connector	Micro-Fit 3.0 Vertical Header, 3.00mm Pitch, Single Row, Circuits, with ...
0022272041	Molex	Connector	Headers & Wire Housings KK .100" HDR 04P VT SN
0022032041	Molex	Connector	KK 100 Header Assembly 4-Pin
0022035045	Molex	Connector	Headers & Wire Housings SPOX .098" HDR 04P VT SN
0022053041	Molex	Connector	Headers & Wire Housings KK .100" HDR 04P RA SN
0022122034	Molex	Connector	Headers & Wire Housings KK .100" HDR 03P RA 20AU
0022053031	Molex	Connector	Conn Wire to Board HDR 3 POS 2.54mm Solder RA Thru-Hole KK® B...
0022053061	Molex	Connector	Headers & Wire Housings KK .100" HDR 06P RA SN
0022057035	Molex	Connector	Headers & Wire Housings SPOX .098" HDR 03P RA SN
0022057048	Molex	Connector	Socket & Kabelgehäuse KK .100" HDR 04P RA SN
5600200220	Molex	Connector	CONN HEADER 2POS 2MM VERT SMD
0022057055	Molex	Connector	Conn Shrouded Header (4 Sides) HDR 5 POS 2.5mm Solder RA Thru-...
0022052051	Molex	Connector	Conn Unshrouded Header HDR 5 POS 2.54mm Solder RA Side Entry ...
0436500227	Molex	Connector	MICROFIT 3.0 SR VERT TH PEG TIN

Molex: 0022057028

Headers & Wire Housings KK .100" HDR 02P RA SN

NAME	VALUE
3D Image...	<a href="https://componentsearcheng...">https://componentsearcheng...</a>
Body Len...	0
Body Width	0
Category	Connector
Confidence	C4 - In house written
DataSheet...	<a href="https://www.molex.com/pdm...">https://www.molex.com/pdm...</a>
Description	Headers & Wire Housings KK...
Height	6.05
Image Lar...	<a href="https://componentsearcheng...">https://componentsearcheng...</a>
Image URL	<a href="https://componentsearcheng...">https://componentsearcheng...</a>
MPN	0022057028
Manufact...	Molex
Parklane C	Other

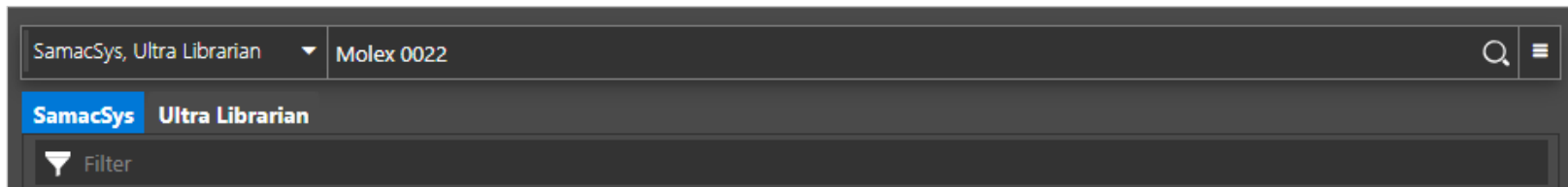
Symbol

Footprint



# Part Search (III)

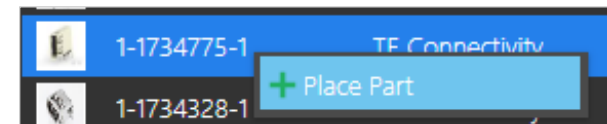
If required parts are not available in existing libraries, they can be downloaded from web portals.



Datasheet   Schematic Symbol   Footprint   Step Model available



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)

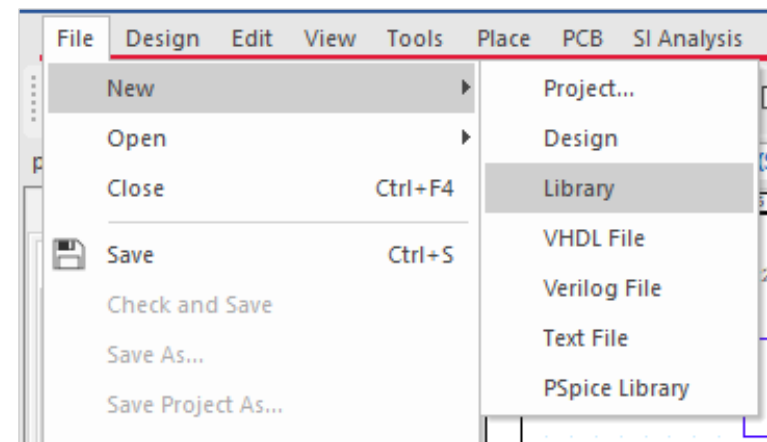
You can create new parts in a new library or open an existing library and add new parts.

Both actions happens via

- **File > New > Library**

or

- **File > Open > Library**

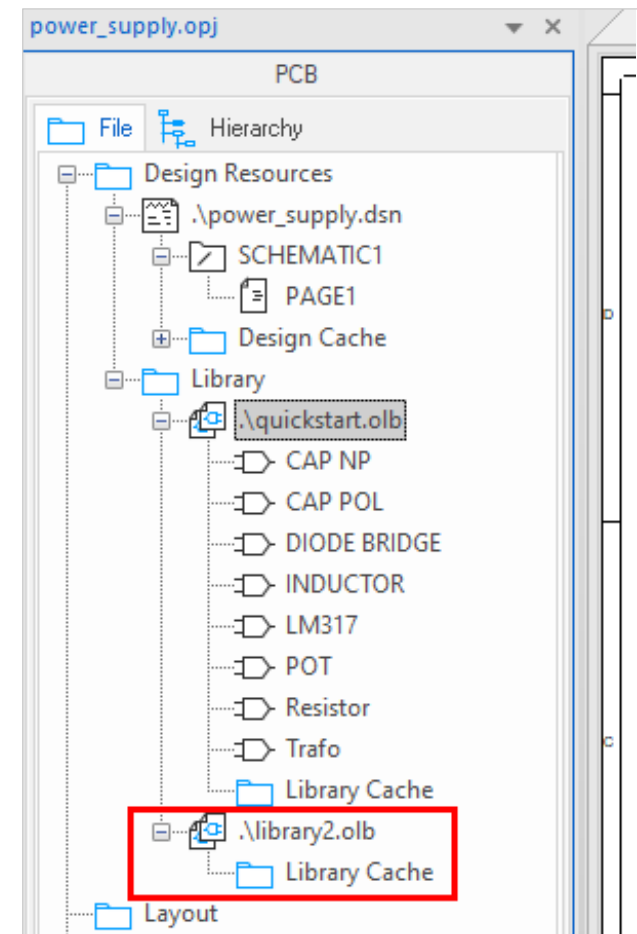




# New Library (II)

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.





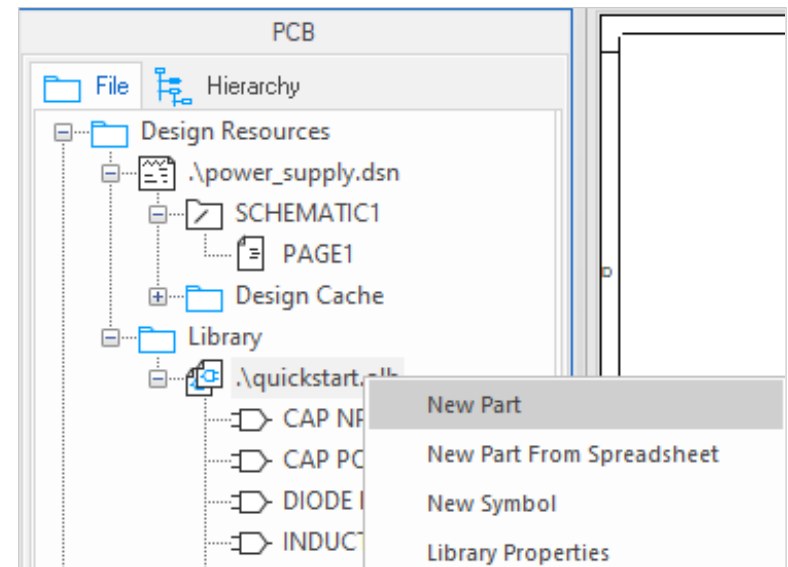
# New Part Creation (I)

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**.

**New Part Properties**

Name: LM317

Part Reference Prefix: U

PCB Footprint: TO220

☐ Create Convert View

Multiple-Part Package

Parts per Pkg: 1

Package Type

☒ Homogeneous

☐ Heterogeneous

Part Numbering

☒ Alphabetic

☐ Numeric

☒ Pin Number Visible

**Part Aliases**

Alias Names:

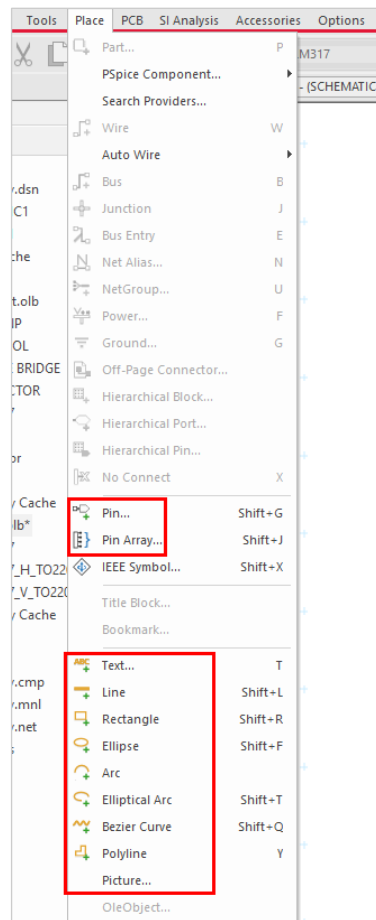
LM317\_H\_TO220

LM317\_V\_TO220



# 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.





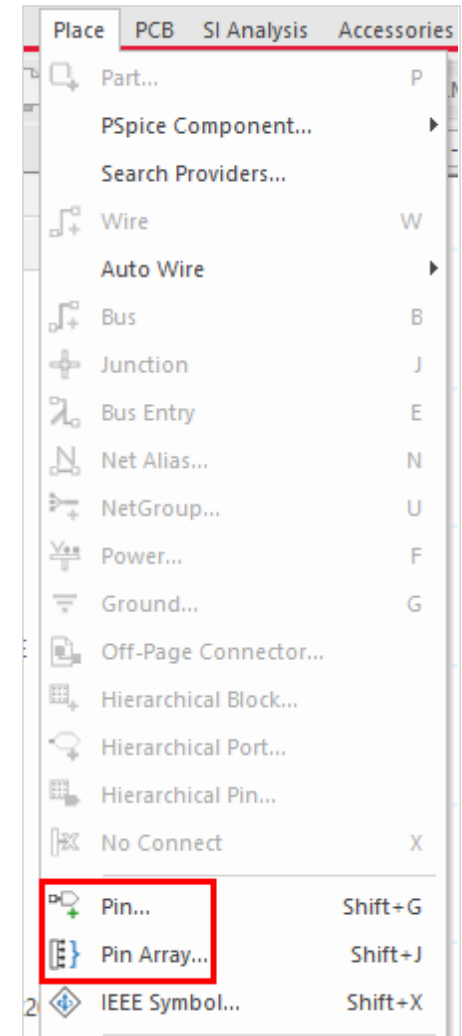
# Adding Pins (I)

Place pins by:

- **Place > Pin...**

or

- **Place Pin Icon** 





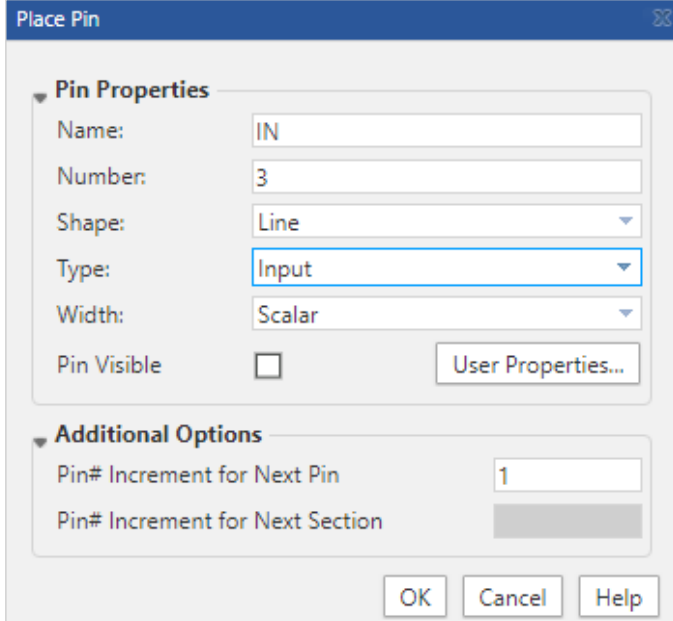
# Adding Pins (II)

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.



The 'Place Pin' dialog box is shown with the following settings:

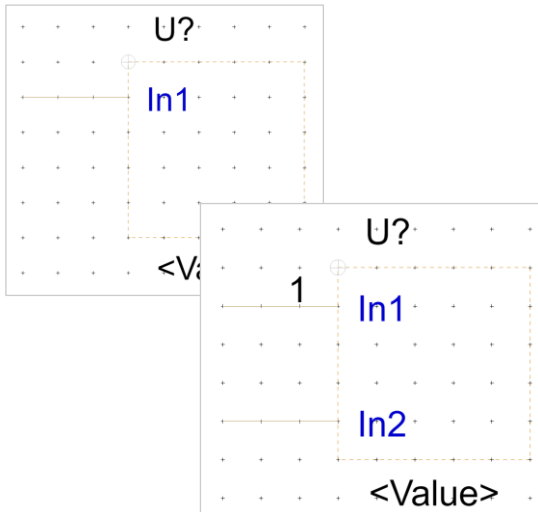
- Pin Properties**
  - Name: IN
  - Number: 3
  - Shape: Line
  - Type: Input
  - Width: Scalar
  - Pin Visible: ☐
  - User Properties... button
- Additional Options**
  - Pin# Increment for Next Pin: 1
  - Pin# Increment for Next Section: (empty)

Buttons: OK, Cancel, Help



# 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.

Edit Pins

Pin Number ☒ Pin Group ☒ Pin Ignore ☒ Order ☒ Pin Type ☒ Pin Shape ☒

Normal View: Pin Name	Section: Pin Num...	Section: Pin Ignore	Order	Pin Group	Normal View: Pin Shape	Normal View: Pin Type	Normal View: Pin Visible
In1	1	No	0		Line	Input	Yes
In2	2	No	1		Line	Input	Yes
Out	3	No	2		Line	Input	Yes

OK Apply Close Help

Property Sheet

**Package Properties**

Part Numbering: Numeric

Package Type: Homogeneous

PCB Footprint: TO220

Part Reference Prefix: U

Section Count: 1

Part Aliases:

Delete Current Section

Add Convert View

**Part Properties**

Name: Demo\_Part.Normal

suffix: .Normal

Implementation Path:

Implementation:

Implementation Type: None

Value:

Pin Name Visible: ☒

Pin Number Visible: ☒

Pin Name Rotate: ☒

Edit Pins

Associate PSpice Model



# Symbol Graphics

Completion of symbols

Symbol outline gets assigned finally. Commands are:

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

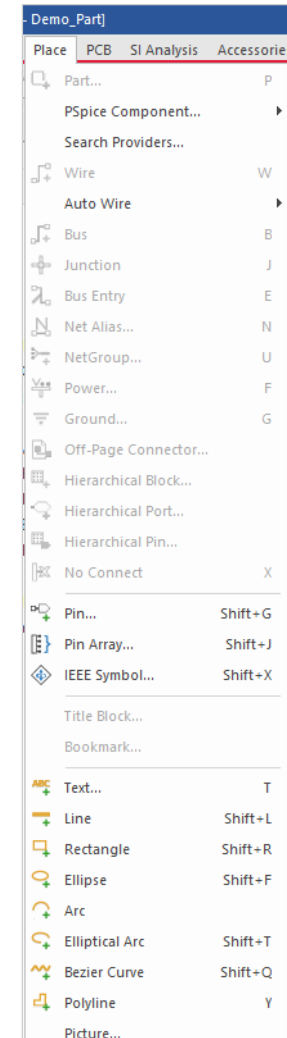
etc.

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.





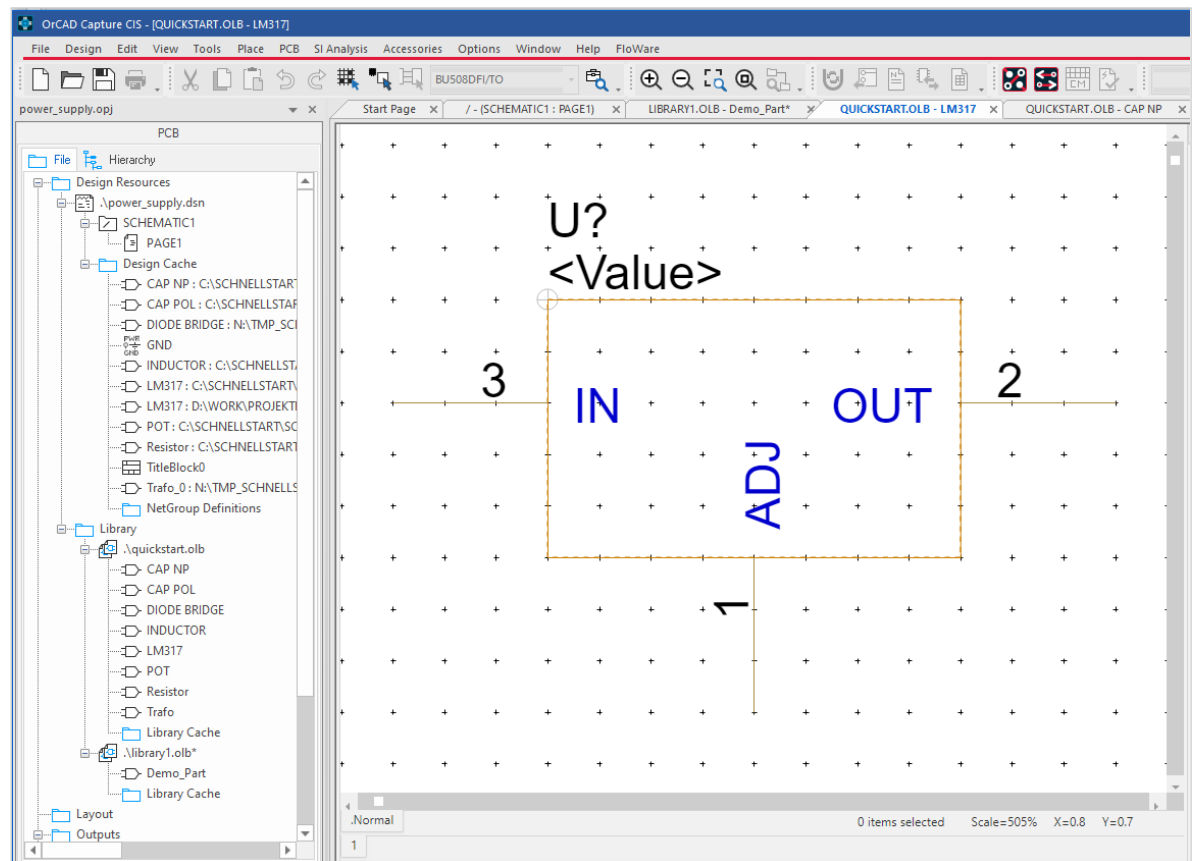
# Completed Symbol

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.





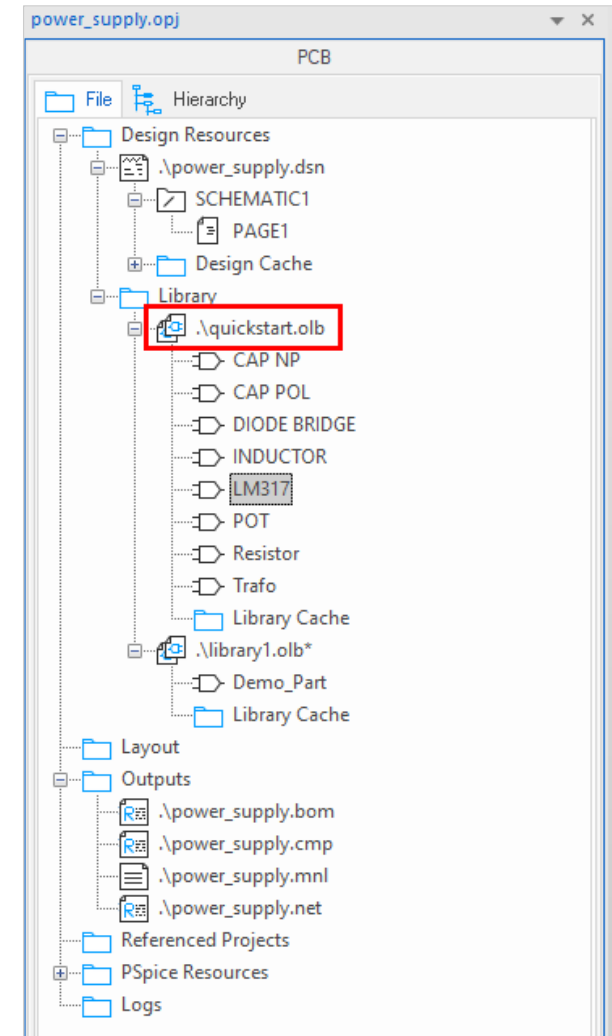
# Library Management

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**.

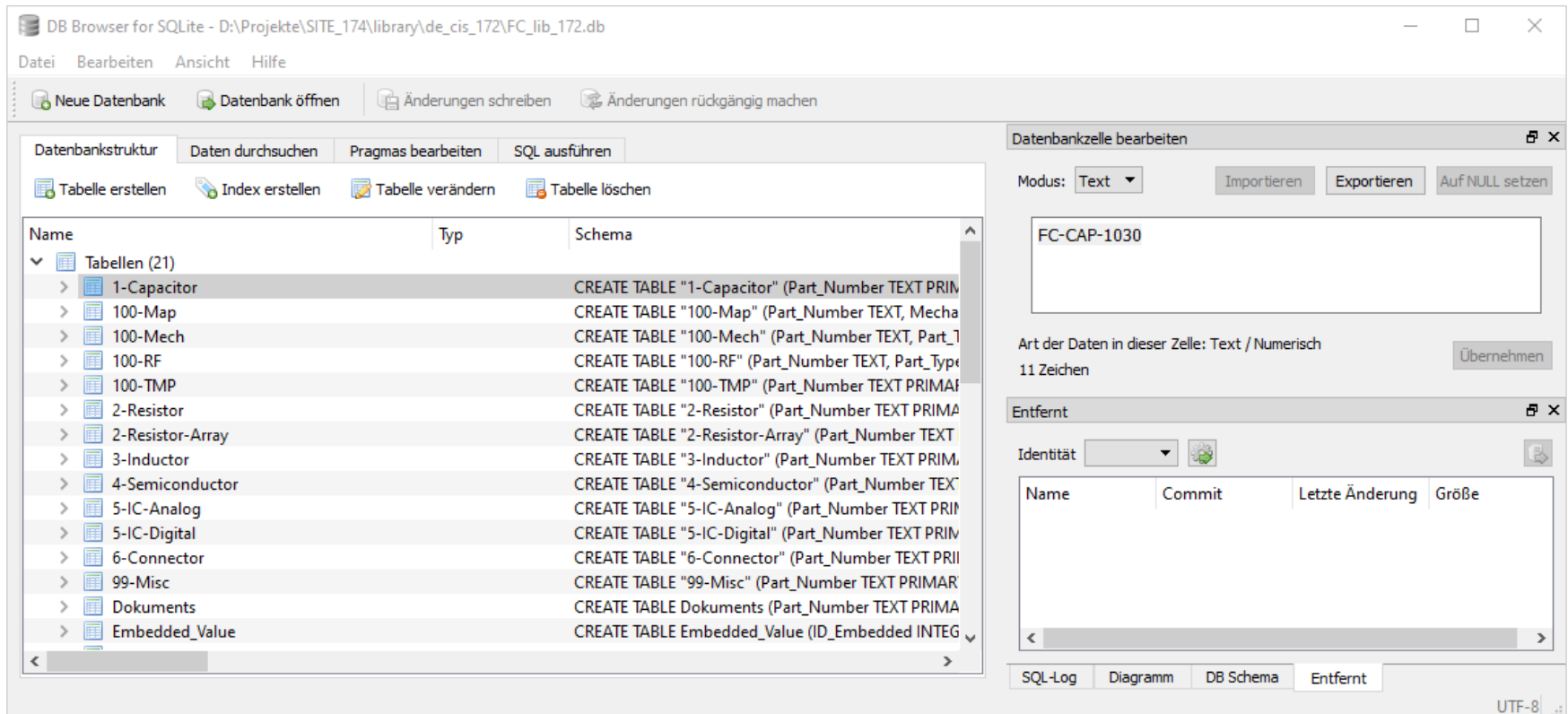


# Edit Database



# 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.

The screenshot shows the 'DB Browser for SQLite' application. The main window displays a table named '1-Capacitor' with columns: Part\_Number, Part\_Type, Value, Tolerance, Vol, Impedance, and De:. The table contains 11 rows of capacitor data. A context menu is open over the first row (FC-CAP-1020), showing options like 'Auf NULL setzen', 'Kopieren', and 'Einfügen'. A secondary dialog, 'Datenbankzelle bearbeiten', is open on the right, showing the selected cell's content 'FC-CAP-1030' and options for importing, exporting, and setting to NULL.

DB Browser for SQLite - D:\Projekte\SITE\_174\library\de\_cis\_172\FC\_lib\_172.db

Datei Bearbeiten Ansicht Hilfe

Neue Datenbank Datenbank öffnen Änderungen schreiben Änderungen rückgängig machen

Datenbankstruktur **Daten durchsuchen** Pragmas bearbeiten SQL ausführen

Tabelle: 1-Capacitor **Neue Zeile** **Zeile löschen**

	Part_Number	Part_Type	Value	Tolerance	Vol	Impedance	De:
	Filtern	Filtern	Filtern	Filtern	Filtern	Filtern	Filtern
119	FC-CAP-1020	ELEC_SMD	10u	20%	16V	NULL	Alumi
120	FC-CAP-1021	ELEC_SMD	47u	20%	16V	NULL	Alumi
121	FC-CAP-1023	ELEC_SMD	33u	20%	50V	NULL	Alumi
122	FC-CAP-1025	ELEC_SMD	680u	20%	16V	NULL	Alumi
123	FC-CAP-1026	ELEC_SMD	68u	20%	80V	NULL	Alumi
124	FC-CAP-1027	ELEC_SMD	470u	20%	80V	NULL	Alumi
125	FC-CAP-1028	TAJ	1u	10%	20V	NULL	TAJ T
126	FC-CAP-1029	TAJ	3.3u	10%	35V	NULL	TAJ T
127	FC-CAP-1030	TAJ	100u	10%	25V	NULL	TAJ T

Auf NULL setzen

Kopieren

Einfügen

Springe zu: 1

Datenbankzelle bearbeiten

Modus: Text Importieren Exportieren Auf NULL setzen

FC-CAP-1030

Art der Daten in dieser Zelle: Text / Numerisch 11 Zeichen Übernehmen

Entfernt

Identität

Name	Commit	Letzte Änderung	Größe
------	--------	-----------------	-------

SQL-Log Diagramm DB Schema Entfernt

UTF-8



# 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.

The screenshot shows the 'Design Template' dialog box with the 'Title Block' tab selected. The dialog has a blue title bar with a close button. Below the title bar are tabs: 'Fonts', 'Title Block' (selected), 'Page Size', 'Grid Reference', 'Hierarchy', and 'SDT Compatibility'. The 'Text' section contains the following fields:

- Title: Project Title
- Organization Name: FlowCAD
- Organization Address 1: Germany
- Organization Address 2: Mozartstr. 2
- Organization Address 3: 85622 Feldkirchen
- Organization Address 4: (empty)
- Document Number: 1
- Revision: 1
- CAGE Code: (empty)

The 'Symbol' section contains the following fields:

- Library Name: D:\Projekte\SITE\_174\library\de\_cis\_172\OI ...
- Title Block Name: TitleBlock

At the bottom right are three buttons: 'OK', 'Abbrechen', and 'Hilfe'.

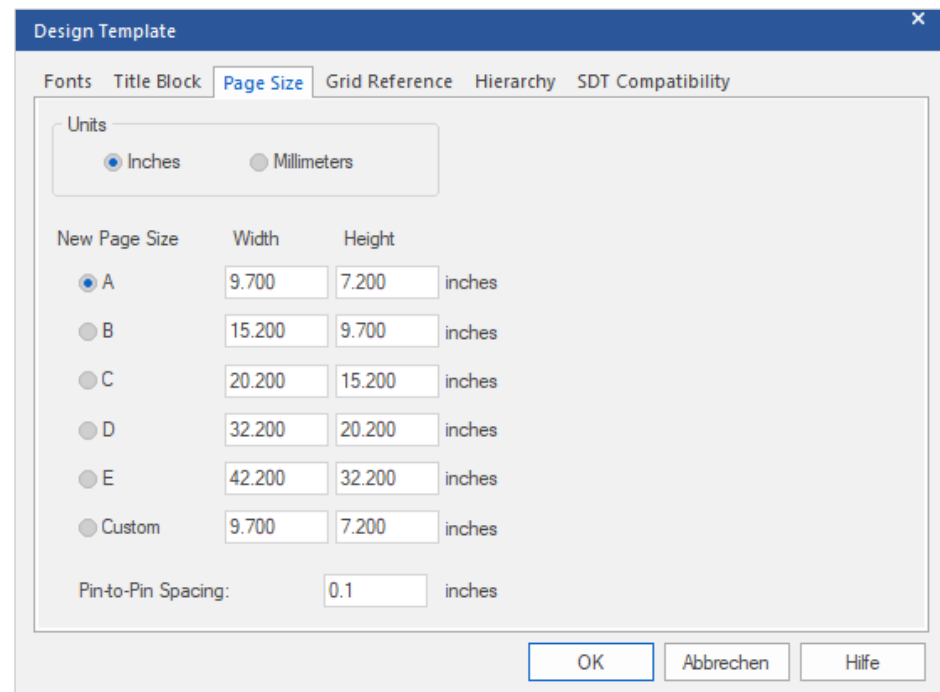


# Design Template – Page Size

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.



The image shows the 'Design Template' dialog box with the 'Page Size' tab selected. The 'Units' section has 'Inches' selected. The 'New Page Size' section lists options A through E and Custom, each with corresponding Width and Height values in inches. The 'Pin-to-Pin Spacing' is set to 0.1 inches.

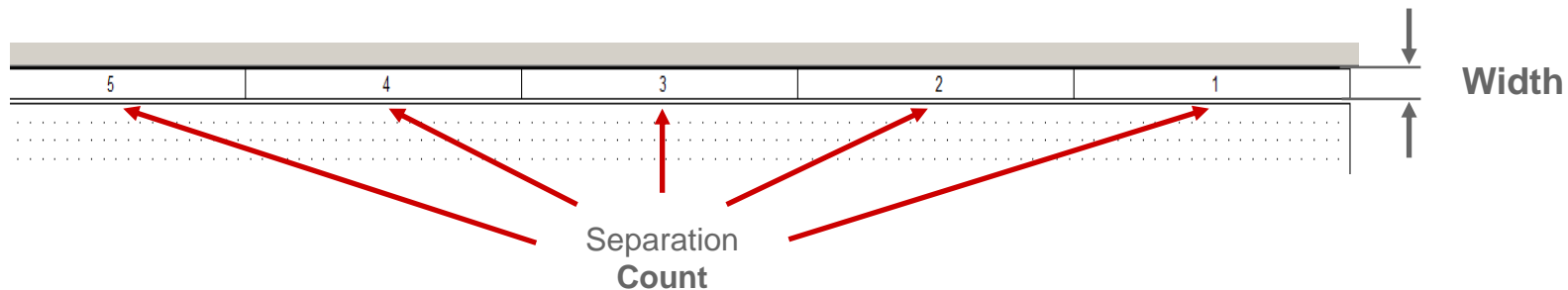
New Page Size	Width	Height	Unit
<input checked="" type="radio"/> A	9.700	7.200	inches
<input type="radio"/> B	15.200	9.700	inches
<input type="radio"/> C	20.200	15.200	inches
<input type="radio"/> D	32.200	20.200	inches
<input type="radio"/> E	42.200	32.200	inches
<input type="radio"/> Custom	9.700	7.200	inches

Pin-to-Pin Spacing: 0.1 inches

Buttons: OK, Abbrechen, Hilfe



# Design Template – Grid Reference



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

The screenshot shows the 'Design Template' dialog box with the 'Grid Reference' tab selected. The 'Horizontal' section has 'Count' set to 5 and 'Width' set to 5 millimeters. The 'Vertical' section has 'Count' set to 4 and 'Width' set to 5 millimeters. Both sections have 'Alphabetic' and 'Ascending' selected. The 'Border Visible' section has 'Displayed' and 'Printed' checked. The 'Title Block Visible' section has 'Displayed' and 'Printed' checked. The 'Grid Reference Visible' section has 'Displayed' and 'Printed' checked, and 'ANSI grid references' is checked. The 'OK' button is highlighted.

# 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](mailto:info@FlowCAD.de)



## FlowCAD Schweiz

Hintermättlistr. 1  
5506 Mägenwil  
T +41 56 485 91 91  
[info@FlowCAD.ch](mailto:info@FlowCAD.ch)



## FlowCAD Polska

ul. Sąsiedzka 2A  
80-298 Gdańsk  
T +48 58 727 90 90  
[info@FlowCAD.pl](mailto:info@FlowCAD.pl)





# Follow Us



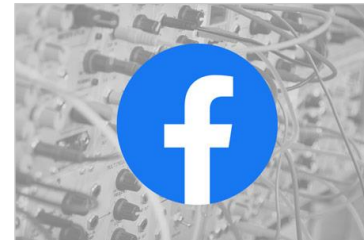
## [FlowCAD.com/newsletter](#)

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