USER GUIDE

UG004 | Using the OrCAD Libraries

Yingchun Shan

1. INTRODUCTION

OrCAD is a popular Electronic Design Automation (EDA) software used for PCB design. It offers a variety of products, including the popular OrCAD Capture, which is Cadence's schematic capture tool, Padstack Editor, OrCAD PCB Designer and OrCAD PSpice Designer.

This user guide demonstrates how to locate, download and install WE component libraries for OrCAD (17.2-2016 Lite Version). WE component libraries for OrCAD consist of symbol file (*.olb) and package file (*.dra) You can access our libraries through the Würth Elektronik website or our GitHub repository. For the most up-to-date version, we recommend visiting our GitHub repository.

2. INSTALLING THE LIBRARIES

a. Downloads from Würth Homepage

Visit the WE product portfolio and navigate to the product you are interested in.

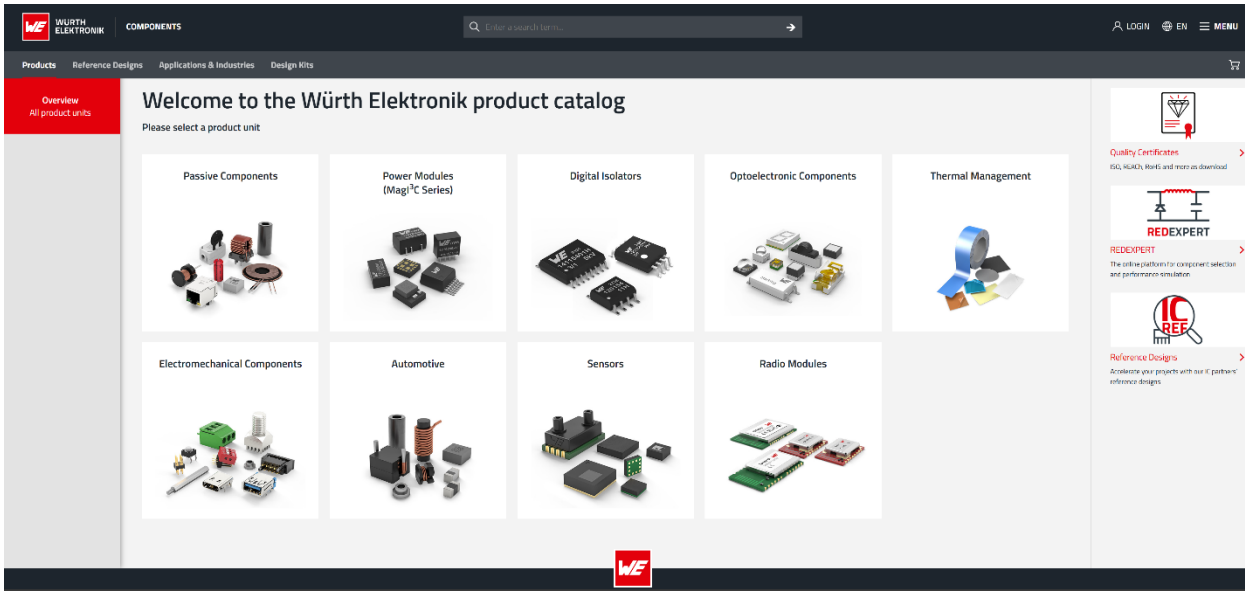


Figure 1: Würth Elektronik Homepage.

Alternatively, enter the part number or product series into the search bar located at the top of the page.

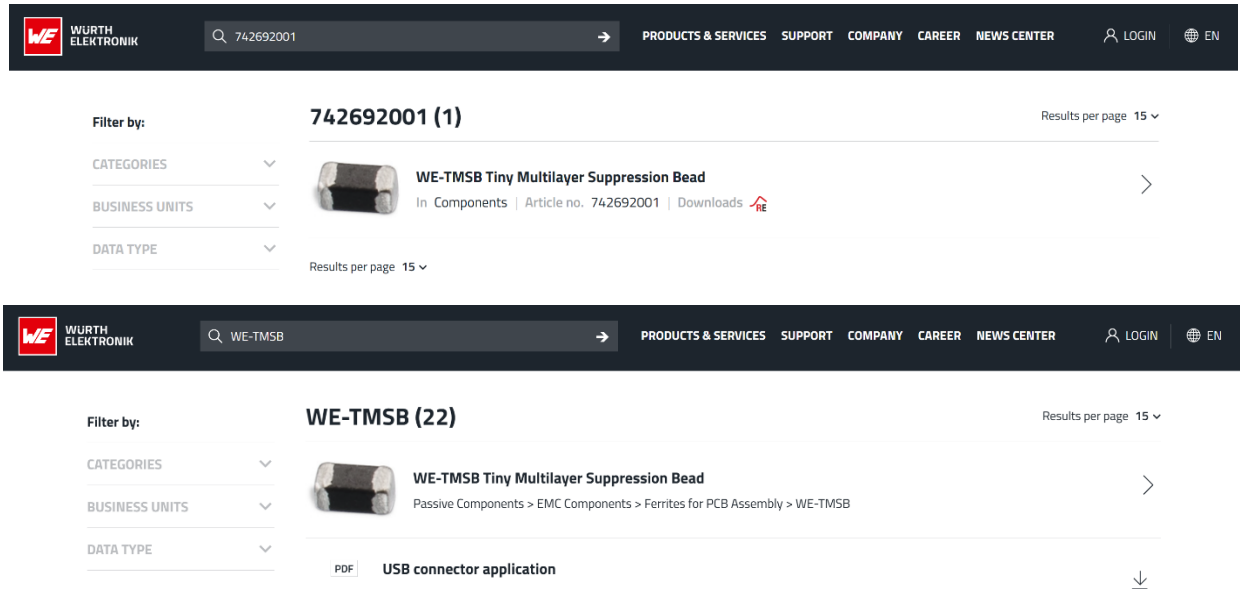


Figure 2: Search part number or series.

On each product series page, you will find the download column in the product list. Locate the OrCAD library in the dropdown list.

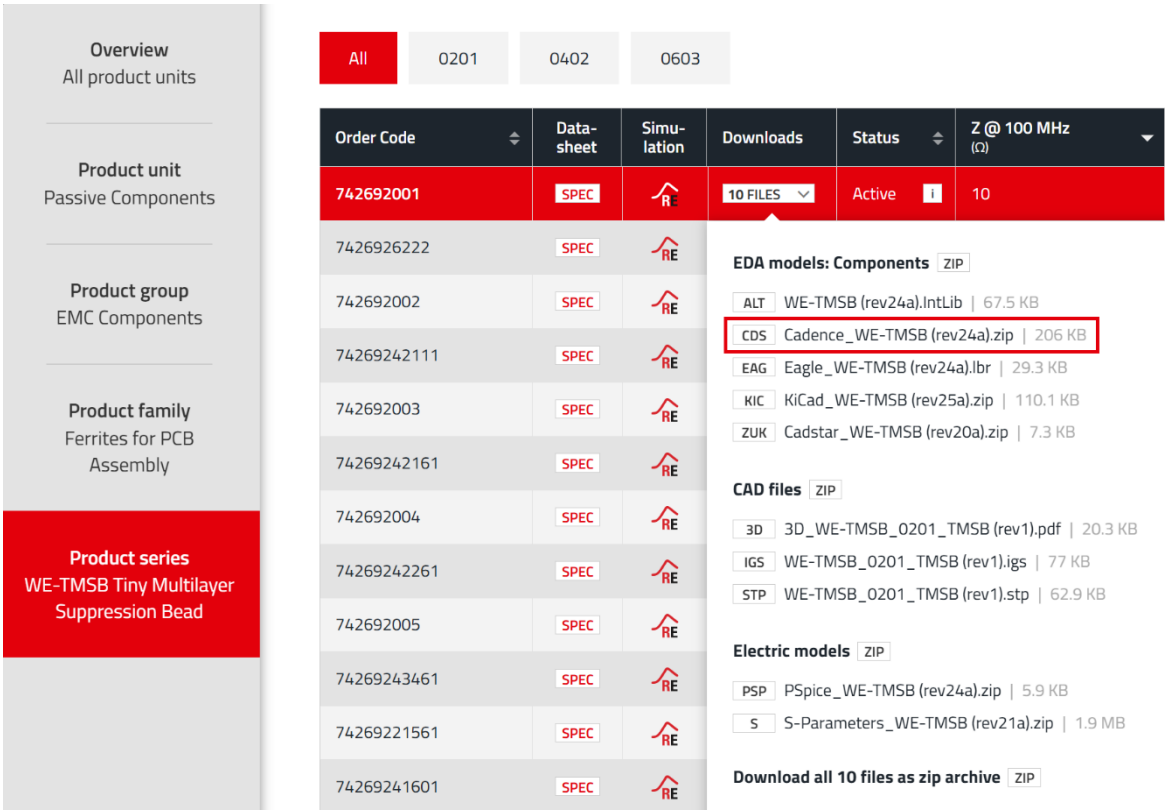


Figure 3: Download OrCAD libraries on Würth Elektronik Homepage.

b. Save the Libraries

Save and extract the *.zip file directly into your chosen directory. Saving the libraries into the same folder as your design is recommended.

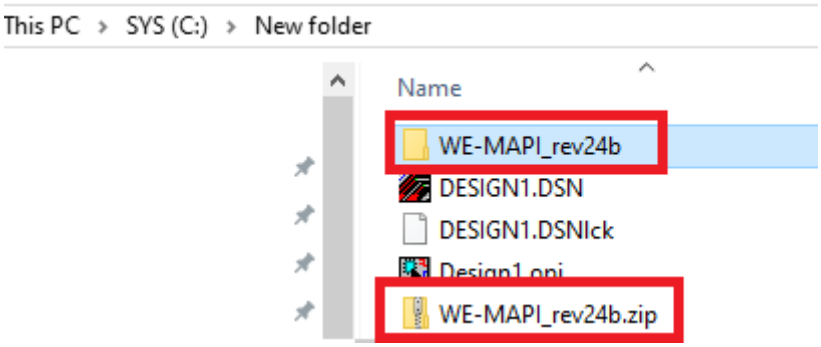


Figure 4: Model files in the same folder as schematic files.

c. Install the symbols

Run OrCAD Capture and open or create a design. Open the schematic page.

Add a symbol library to your design by clicking the "Add Library" symbol in the "Place Part" window. Open the "Browse File" window, select the desired symbol file (*.olb) and then open it.

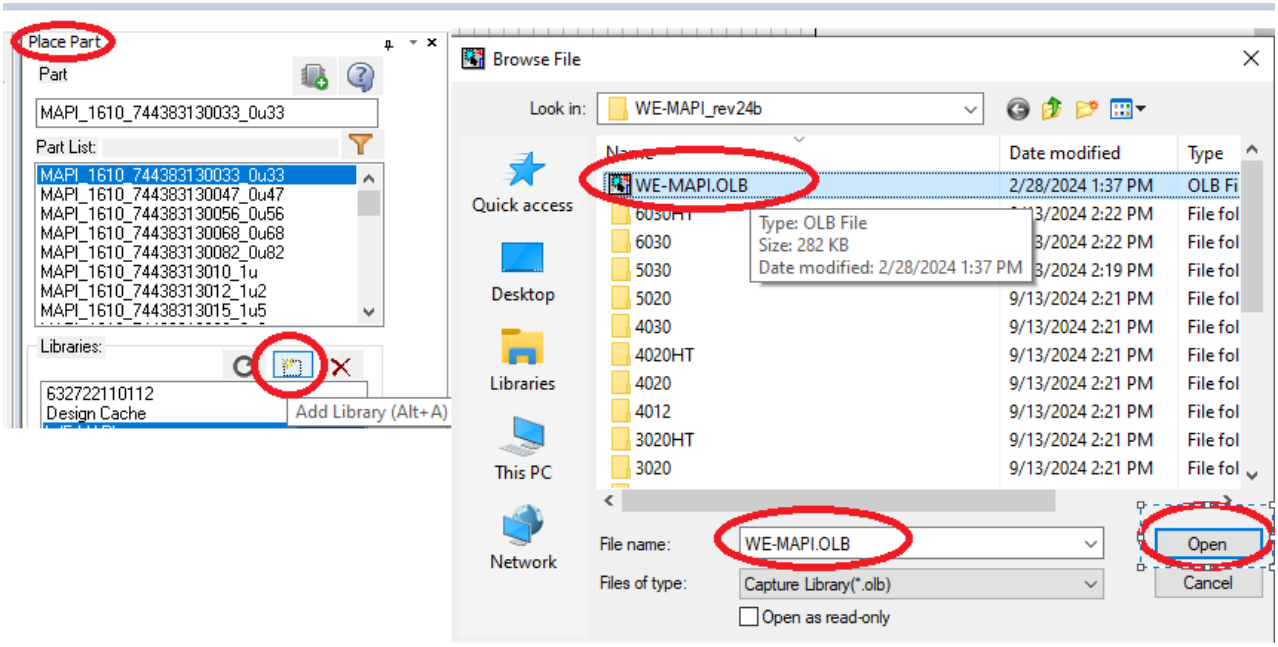


Figure 5: Adding the library to the "Place Part" window.

Place the symbols by clicking the "Place" tab and select "Part" option from the menu or pressing the "P" key directly. Alternatively, select "Place" in the action toolbar.

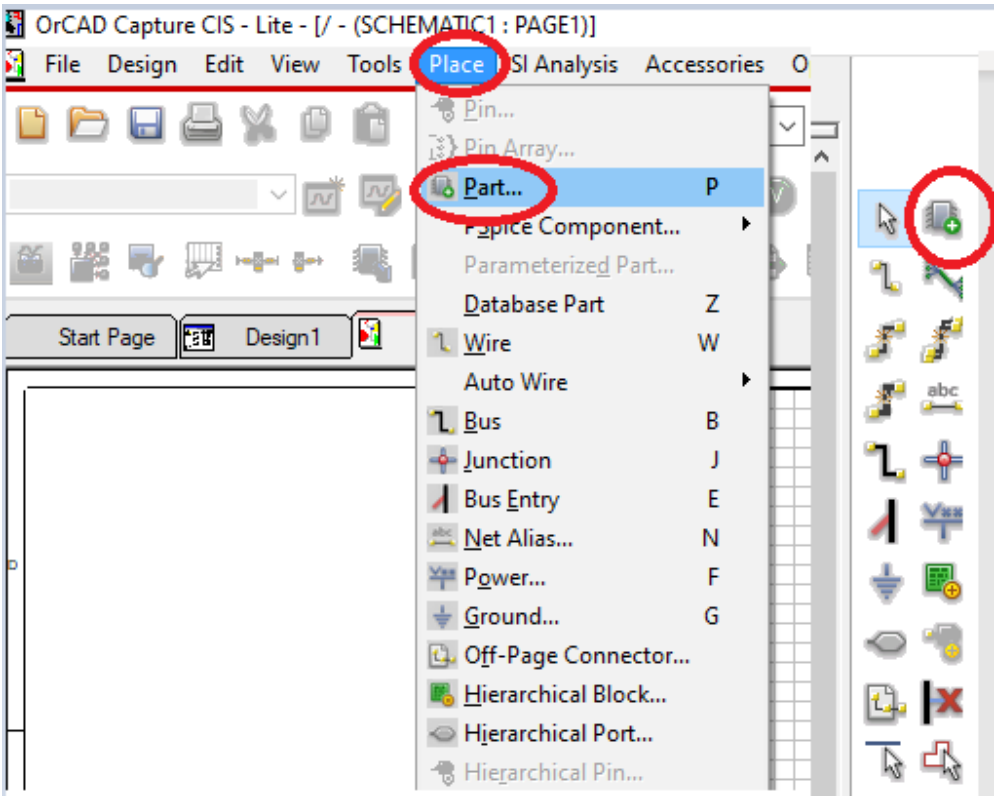


Figure 6: Navigating to the "Place Part" window using the menu (left) and using the toolbar (right).

Double click on the desired part number and place the symbol in the schematic.

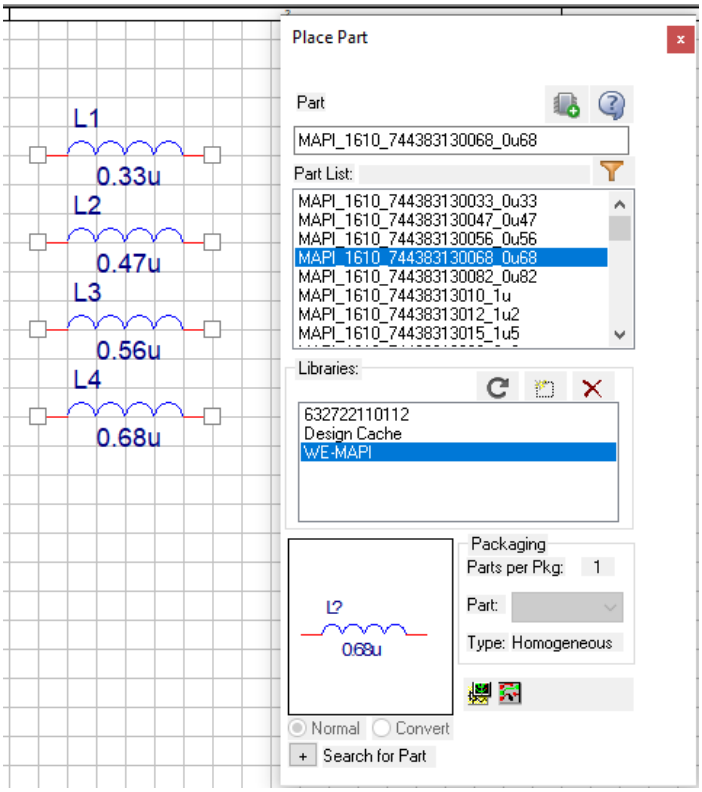
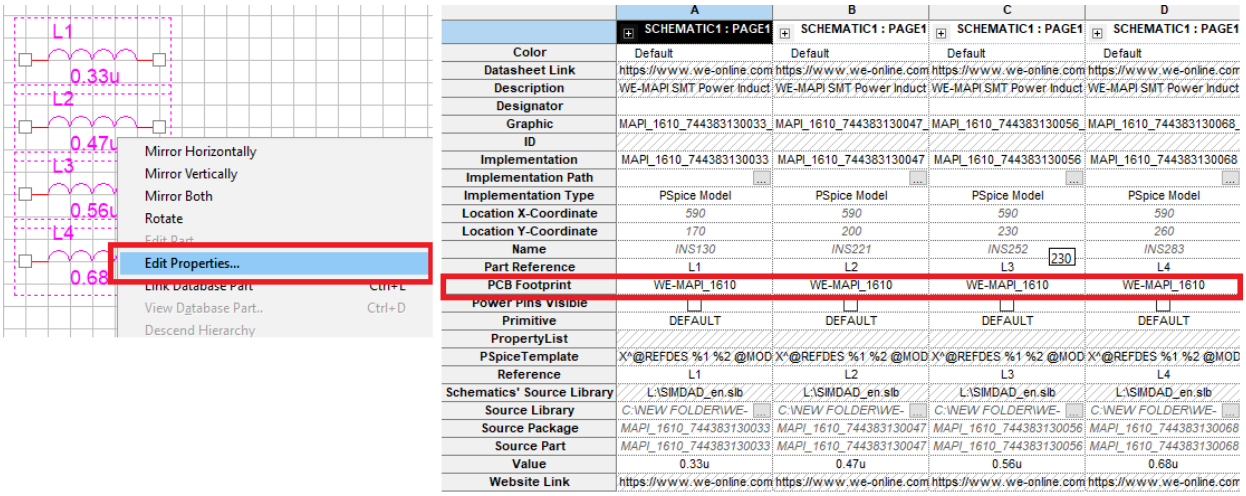


Figure 7: Place symbols in the schematic from the "Place Part" window.

Edit the properties of the symbols. selecting the symbols, double click them or click the right mouse, select "Edit Properties" to open the properties window and check the footprint information.



	A	B	C	D
	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1	SCHEMATIC1 : PAGE1
Color	Default	Default	Default	Default
Datasheet Link	https://www.we-online.com	https://www.we-online.com	https://www.we-online.com	https://www.we-online.com
Description	WE-MAPI SMT Power Induct	WE-MAPI SMT Power Induct	WE-MAPI SMT Power Induct	WE-MAPI SMT Power Induct
Designator				
Graphic	MAPI_1610_744383130033_	MAPI_1610_744383130047_	MAPI_1610_744383130056_	MAPI_1610_744383130068_
ID				
Implementation	MAPI_1610_744383130033	MAPI_1610_744383130047	MAPI_1610_744383130056	MAPI_1610_744383130068
Implementation Path				
Implementation Type	PSpice Model	PSpice Model	PSpice Model	PSpice Model
Location X-Coordinate	590	590	590	590
Location Y-Coordinate	170	200	230	260
Name	INS130	INS221	INS252	INS283
Part Reference	L1	L2	L3	L4
PCB Footprint	WE-MAPI_1610	WE-MAPI_1610	WE-MAPI_1610	WE-MAPI_1610
Power Pins Visible				
Primitive	DEFAULT	DEFAULT	DEFAULT	DEFAULT
PropertyList				
PSpiceTemplate	X^@REFDES %1 %2 @MOD	X^@REFDES %1 %2 @MOD	X^@REFDES %1 %2 @MOD	X^@REFDES %1 %2 @MOD
Reference	L1	L2	L3	L4
Schematics' Source Library	L:\SIMDAD_en.slb	L:\SIMDAD_en.slb	L:\SIMDAD_en.slb	L:\SIMDAD_en.slb
Source Library	C:\NEW FOLDER\WE-	C:\NEW FOLDER\WE-	C:\NEW FOLDER\WE-	C:\NEW FOLDER\WE-
Source Package	MAPI_1610_744383130033	MAPI_1610_744383130047	MAPI_1610_744383130056	MAPI_1610_744383130068
Source Part	MAPI_1610_744383130033	MAPI_1610_744383130047	MAPI_1610_744383130056	MAPI_1610_744383130068
Value	0.33u	0.47u	0.56u	0.68u
Website Link	https://www.we-online.com	https://www.we-online.com	https://www.we-online.com	https://www.we-online.com

Figure 8: Information in the properties.

Copy the footprint files (*.dra,*.psm,*.pad,*.step) for the symbols from the library to below folder:
C:\Cadence\SPB_17.2\share\pcb\pcb_lib\symbols

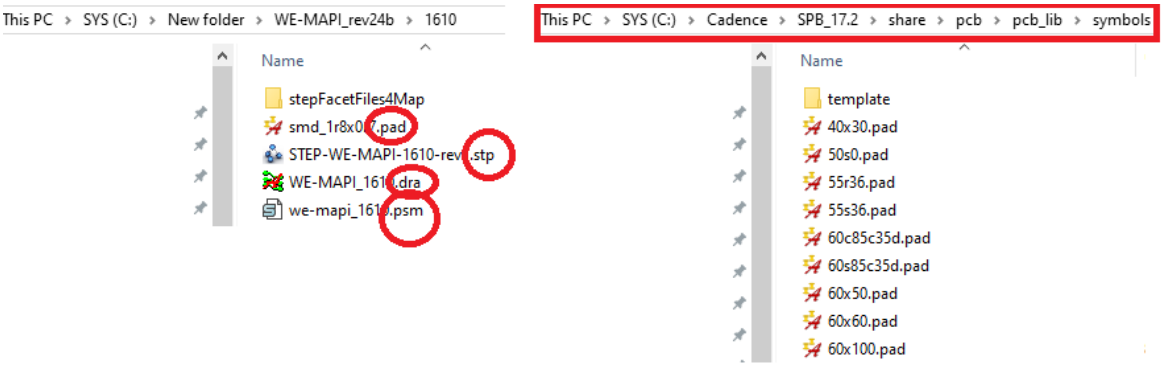


Figure 9: Copy the footprint files to Cadence.

Next, the netlist can be exported.

d. Export Netlist File

Go back to your design page, select your design, click "Tools" and select the "Create Netlist" option from the menu. The "Create Netlist" window will then open.

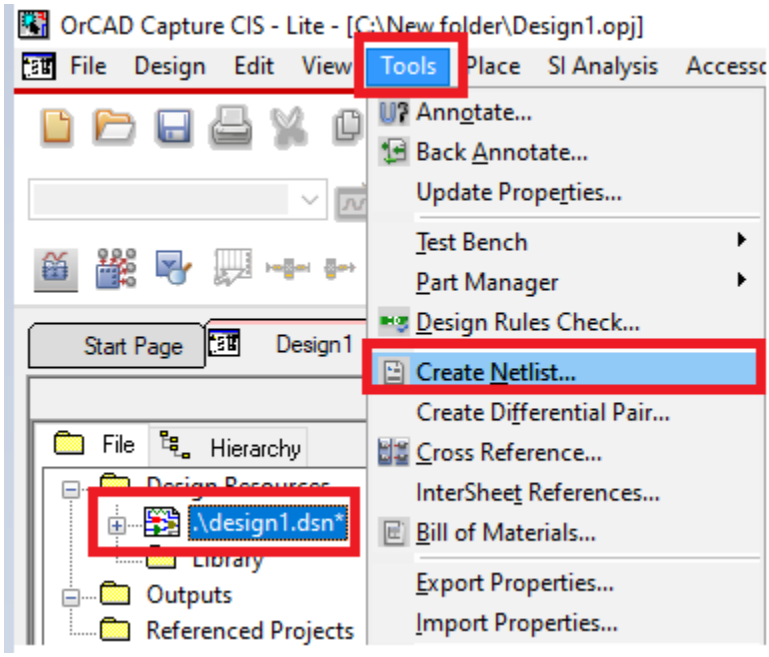


Figure 10: Navigating to the "Create List" menu options.

In the "Create Netlist" window, make sure "Create PCB Editor Netlist" is selected, click "Create or update PCB Editor Board (Netrev)", change the board name if you want, click "Open Board in OrCAD PCB Editor", then click "OK" button. Click "Yes" in the new Pop-up window and the Click "OK" for the second Pop-up window.

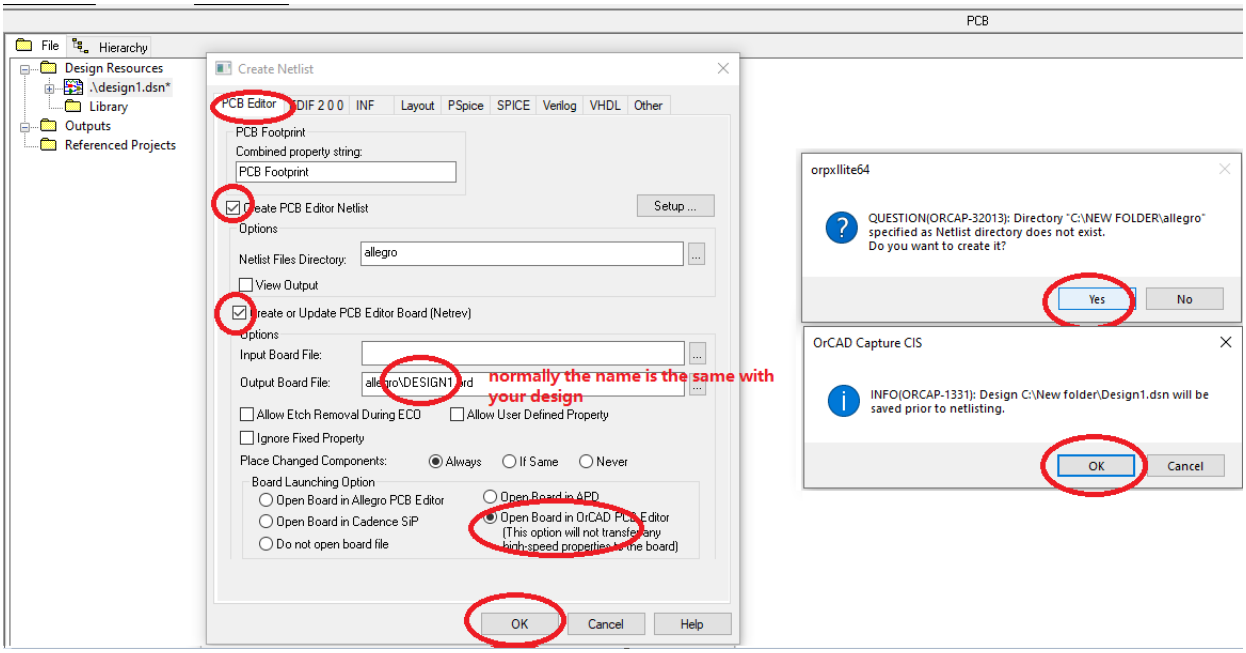


Figure 11: Navigating the "Create List" window.

In this way, you can create and open a new board directly.

If the process goes well, we can get three files "pstxnet.dat","pstxprt.dat","pstchip.dat" , and two folders named "allegro" and "signoise.run" in the same folder as your design is saved. The files, "pstxnet.dat","pstxprt.dat","pstchip.dat" are all saved automatically in the "allegro" folder.

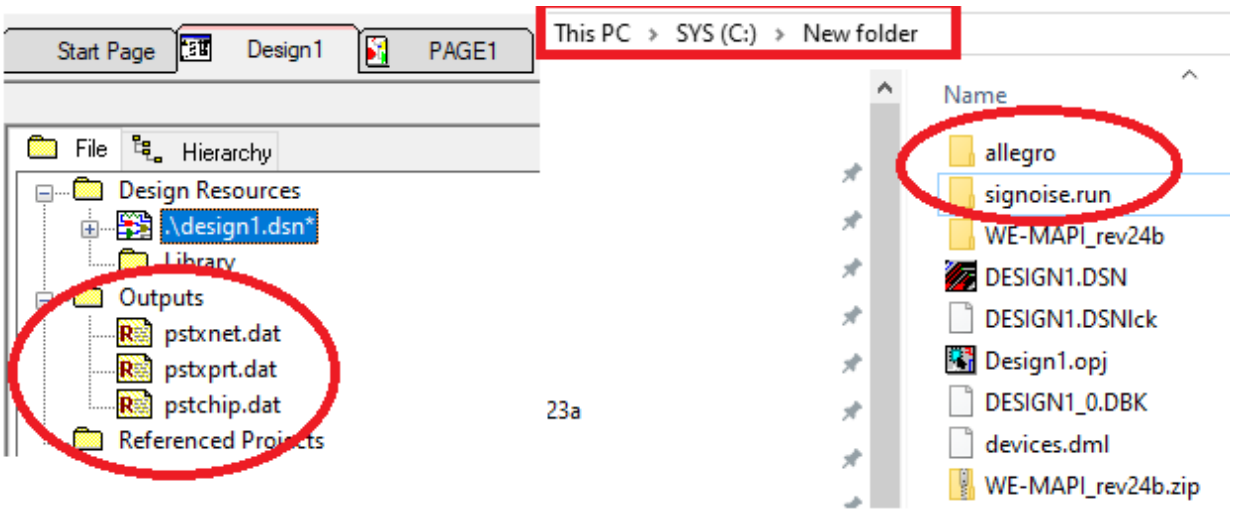


Figure 12: Netlist files and saved location.

e. Import Netlist File

Open the board directly or run the" OrCAD PCB Editor" to create a new board", the*.brd" file. Click "Import" tab and select "Netlist" option, then "Import Logic" window will be open. Click "import logic type" according to your need, you can choose "Design entry CIS (Capture)". The most important point is to change the "Import directory" to the directory where your netlist files are saved.

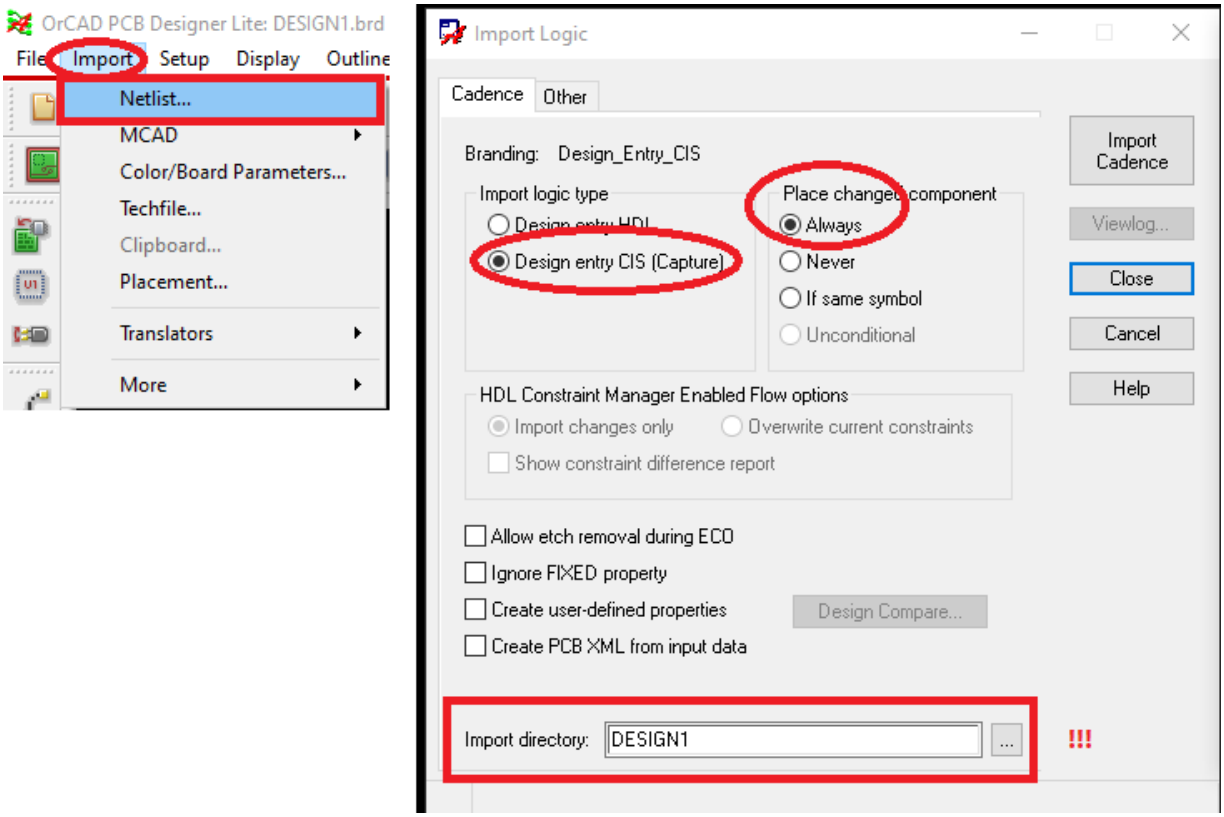


Figure 13: Navigating to the "Import Logic" menu options.

Click the "..." button on the right of the "Import directory" to change the path of the netlist files saved, open the browser window, choose the folder named "allegro" you've just created and then click "OK".

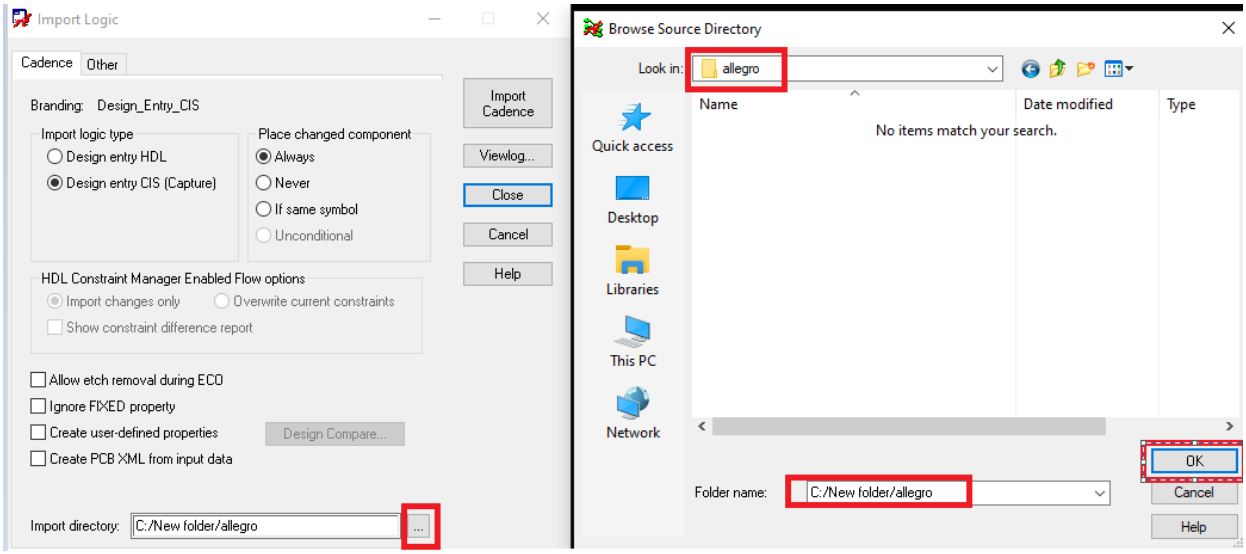


Figure 14: Navigating to find the netlist files.

Then click "Import Cadence". If there's no error information informed, it means the netlist files are imported successfully.

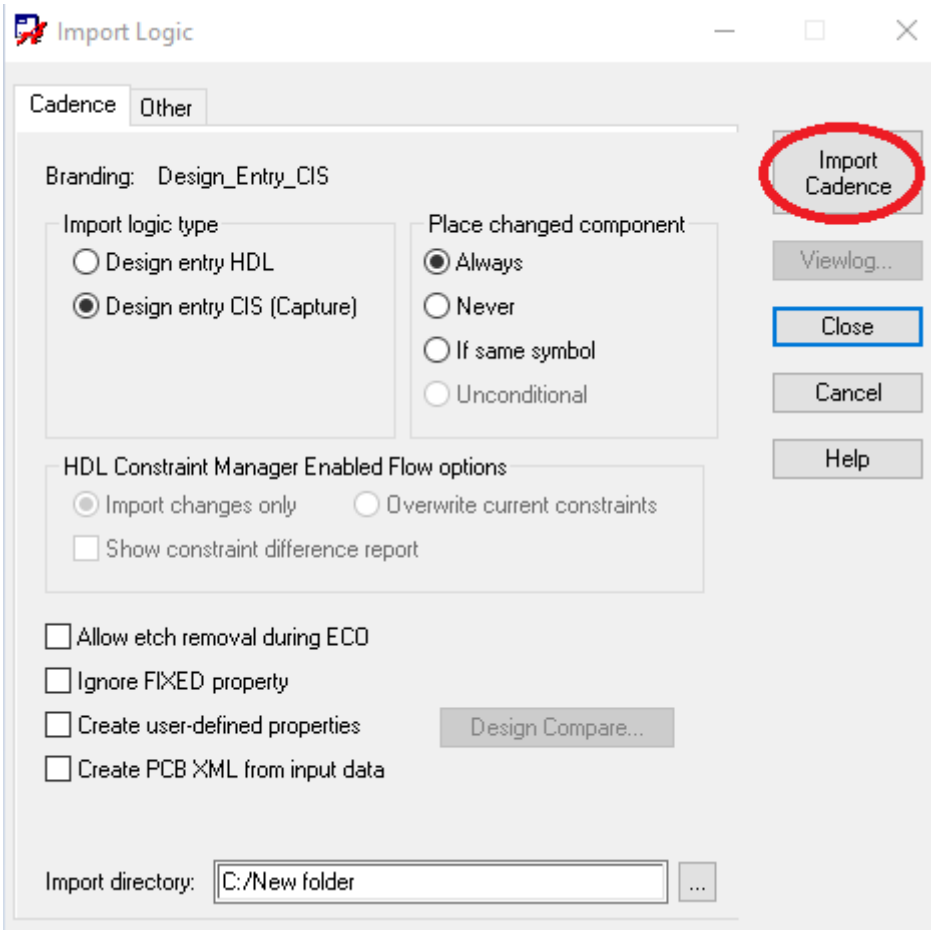


Figure 15: Import Cadence Option.

Place a footprint by clicking the “Placement” bar to open the “Placement” window and click a component to add it to the board.

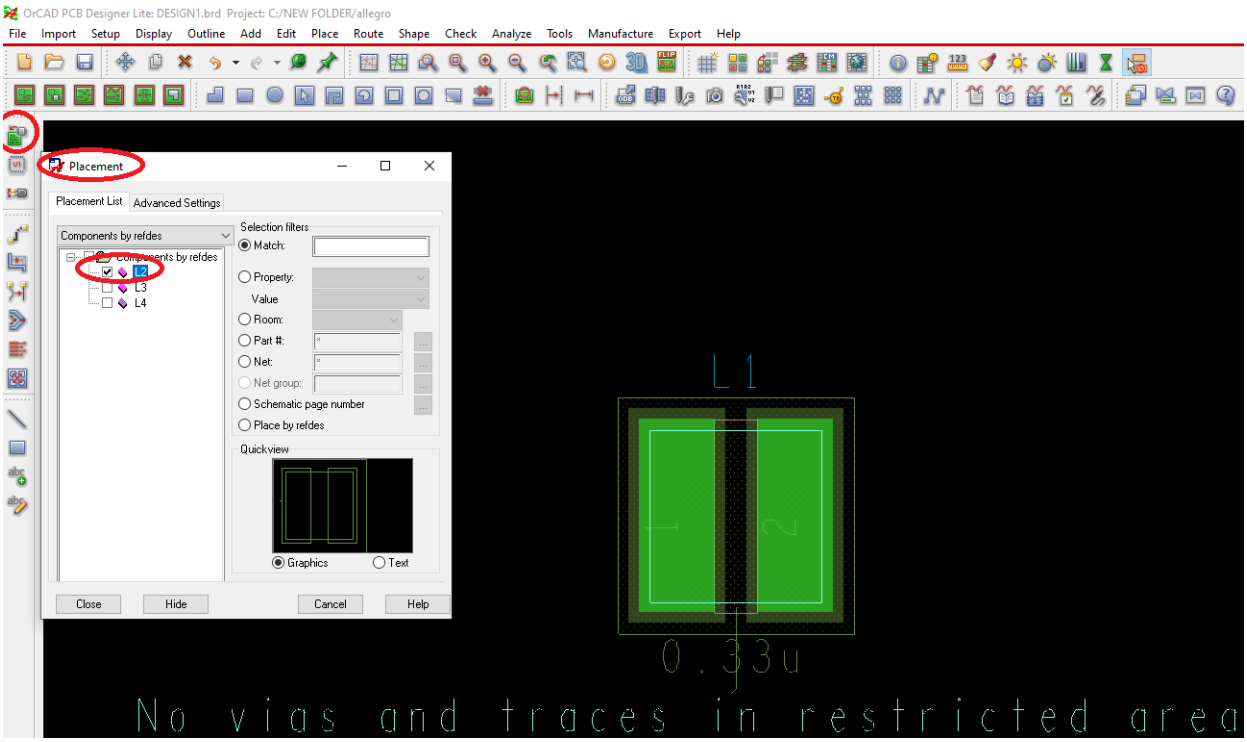


Figure 16: Add Components to board.

3. INSTALL FROM GITHUB REPOSITORY

a. Install GitHub Desktop

GitHub Desktop is the most user-friendly tool for working with GitHub projects, and we recommend you use it for keeping your library files up to date.

Go to https://desktop.github.com/to download the appropriate package for your operating system and install it on your computer.

During the Desktop installation, register or sign in with your GitHub Account and click next. On opening the GitHub Browser webpage, authenticate yourself and give permission to the GitHub desktop application. The process will then return you to the desktop application.

b. Clone the Library

From GitHub Desktop, click the button "Clone a repository from the Internet...".

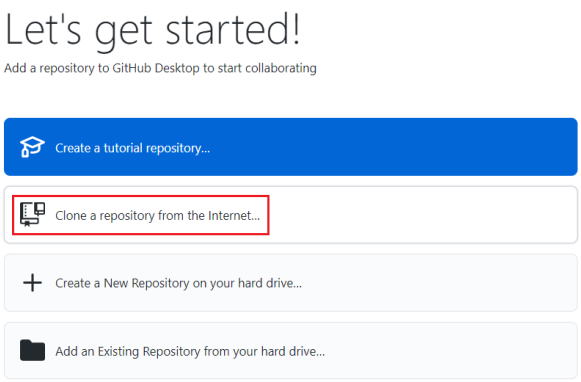


Figure 17: Clone a repository from the Internet.

Enter the URL of Würth Elektronik OrCAD Library repository: https://github.com/WurthElektronik/CadenceLibrary.git and define a local directory to which to clone the repository.

Then click the "Clone" button. A window will then open, synchronizing the libraries into your local directory from the online repository. Cloning repository may take some time.

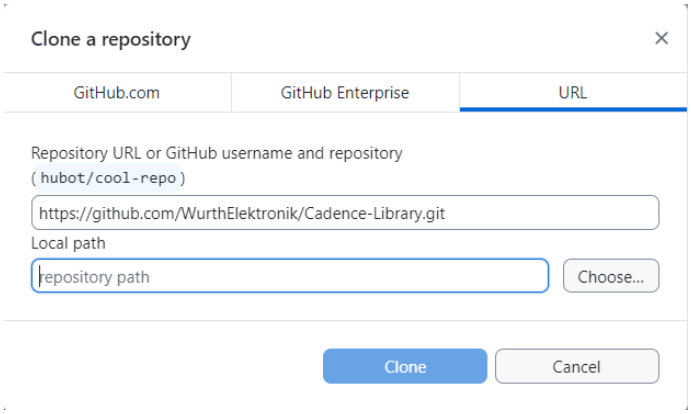


Figure 18: Cloning the Würth Elektronik OrCAD Library repository to your local directory.

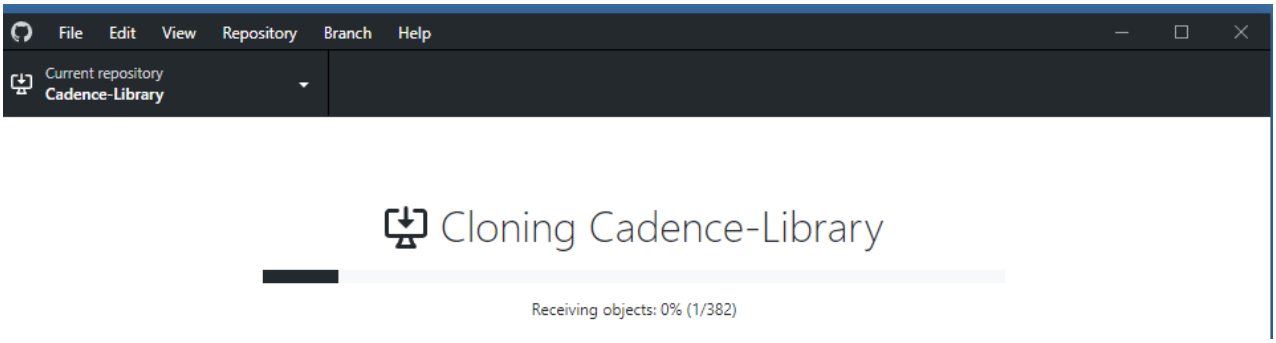


Figure 19: Cloning in progress.

c. Synchronize Local Library from GitHub

If there are updates in GitHub repository, GitHub Desktop will detect it. You can "Pull" the update to your local directory. If there are any new commits on the online master repository, from GitHub Desktop you'll receive the update information automatically.

Click "Pull origin" button to fetch the updates to your local directory immediately.

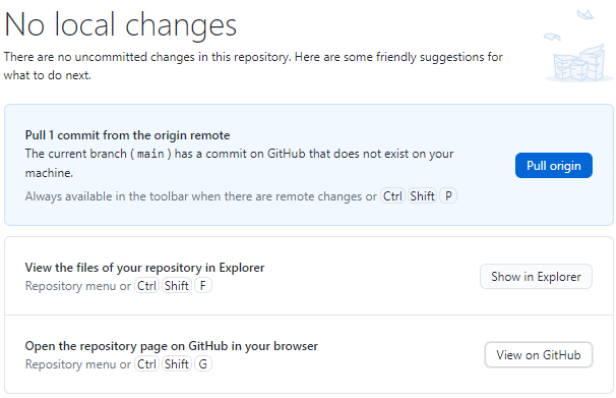


Figure 20: Pulling the repository to your local directory.

Click "View on GitHub" to explore more details of the latest updates.

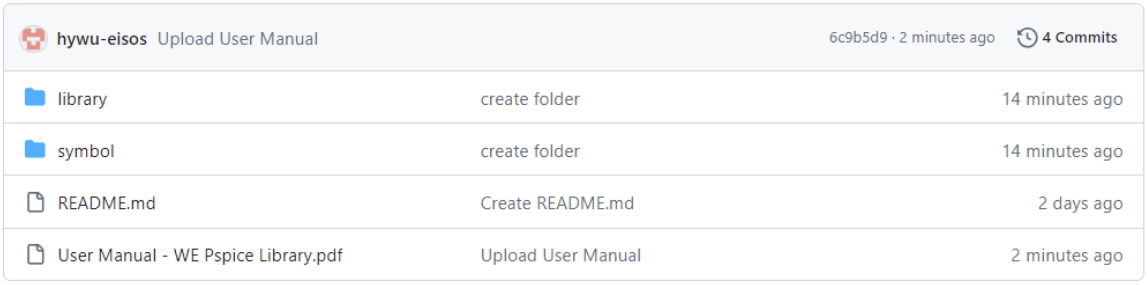


Figure 21: View the updates on GitHub.

IMPORTANT NOTICE

The Application Note is based on our knowledge and experience of typical requirements concerning these areas. It serves as general guidance and should not be construed as a commitment for the suitability for customer applications by Würth Elektronik eiSos GmbH & Co. KG. The information in the Application Note is subject to change without notice. This document and parts thereof must not be reproduced or copied without written permission, and contents thereof must not be imparted to a third party nor be used for any unauthorized purpose.

Würth Elektronik eiSos GmbH & Co. KG and its subsidiaries and affiliates (WE) are not liable for application assistance of any kind. Customers may use WE's assistance and product recommendations for their applications and design. The responsibility for the applicability and use of WE Products in a particular customer design is always solely within the authority of the customer. Due to this fact it is up to the customer to evaluate and investigate, where appropriate, and decide whether the device with the specific product characteristics described in the product specification is valid and suitable for the respective customer application or not.

The technical specifications are stated in the current data sheet of the products. Therefore the customers shall use the data sheets and are cautioned to verify that data sheets are current. The current data sheets can be downloaded at www.we-online.com. Customers shall strictly observe any product-specific notes, cautions and warnings. WE reserves the right to make corrections, modifications, enhancements, improvements, and other changes to its products and services.

WE DOES NOT WARRANT OR REPRESENT THAT ANY LICENSE, EITHER EXPRESS OR IMPLIED, IS GRANTED UNDER ANY PATENT RIGHT, COPYRIGHT, MASK WORK RIGHT, OR OTHER INTELLECTUAL PROPERTY RIGHT RELATING TO ANY COMBINATION, MACHINE, OR PROCESS IN WHICH WE PRODUCTS OR SERVICES ARE USED. INFORMATION PUBLISHED BY WE REGARDING THIRD-PARTY PRODUCTS OR SERVICES DOES NOT CONSTITUTE A LICENSE FROM WE TO USE SUCH PRODUCTS OR SERVICES OR A WARRANTY OR ENDORSEMENT THEREOF.

WE products are not authorized for use in safety-critical applications, or where a failure of the product is reasonably expected to cause severe personal injury or death. Moreover, WE products are neither designed nor intended for use in areas such as military, aerospace, aviation, nuclear control, submarine, transportation (automotive control, train control, ship control), transportation signal, disaster prevention, medical, public information network etc. Customers shall inform WE about the intent of such usage before design-in stage. In certain customer applications requiring a very high level of safety and in which the malfunction or failure of an electronic component could endanger human life or health, customers must ensure that they have all necessary expertise in the safety and regulatory ramifications of their applications. Customers acknowledge and agree that they are solely responsible for all legal, regulatory and safety-related requirements concerning their products and any use of WE products in such safety-critical applications, notwithstanding any applications-related information or support that may be provided by WE.

CUSTOMERS SHALL INDEMNIFY WE AGAINST ANY DAMAGES ARISING OUT OF THE USE OF WE PRODUCTS IN SUCH SAFETY-CRITICAL APPLICATION.

USEFUL LINKS

Application Notes
www.we-online.com/appnotes

REDEXPERT Design Platform
www.we-online.com/redexpert

Toolbox
www.we-online.com/toolbox

Product Catalog
www.we-online.com/products

CONTACT INFORMATION

appnotes@we-online.com
Tel. +49 7942 945 - 0

Würth Elektronik eiSos GmbH & Co. KG
Max-Eyth-Str. 1 74638 Waldenburg Germany
www.we-online.com