

## USER GUIDE

### UGxxx | 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.

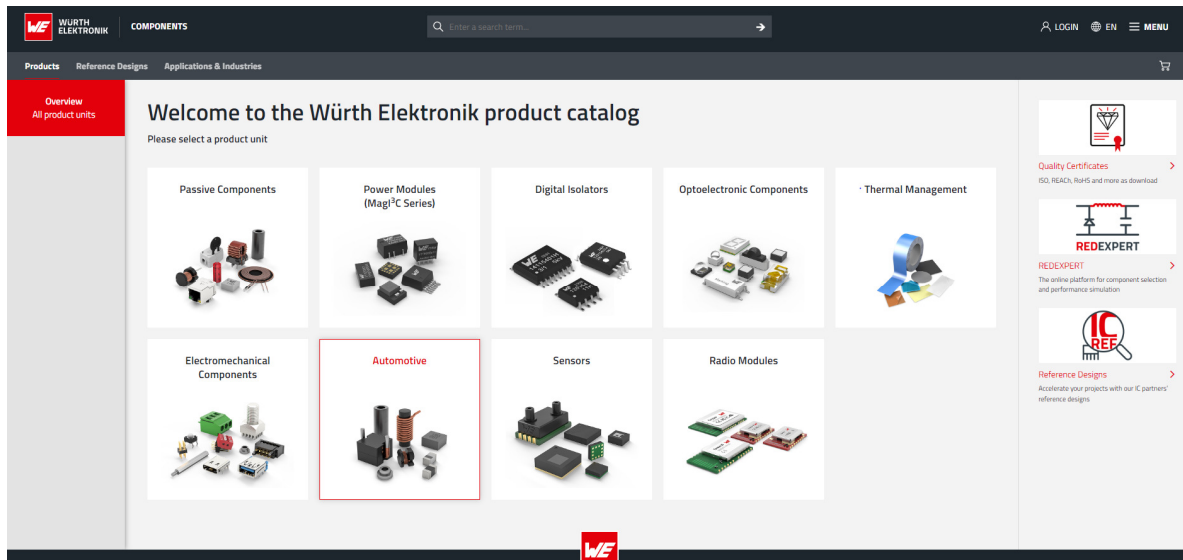


Figure 1: Würth Elektronik homepage

# USER GUIDE

## UGxxx | Using the OrCAD Libraries

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

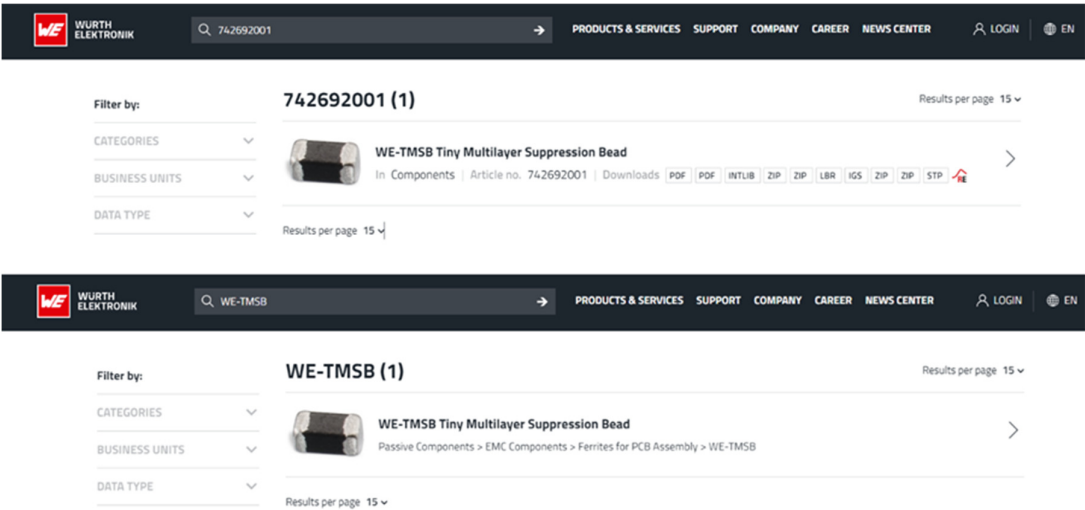


Figure 2: Search part number or series

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

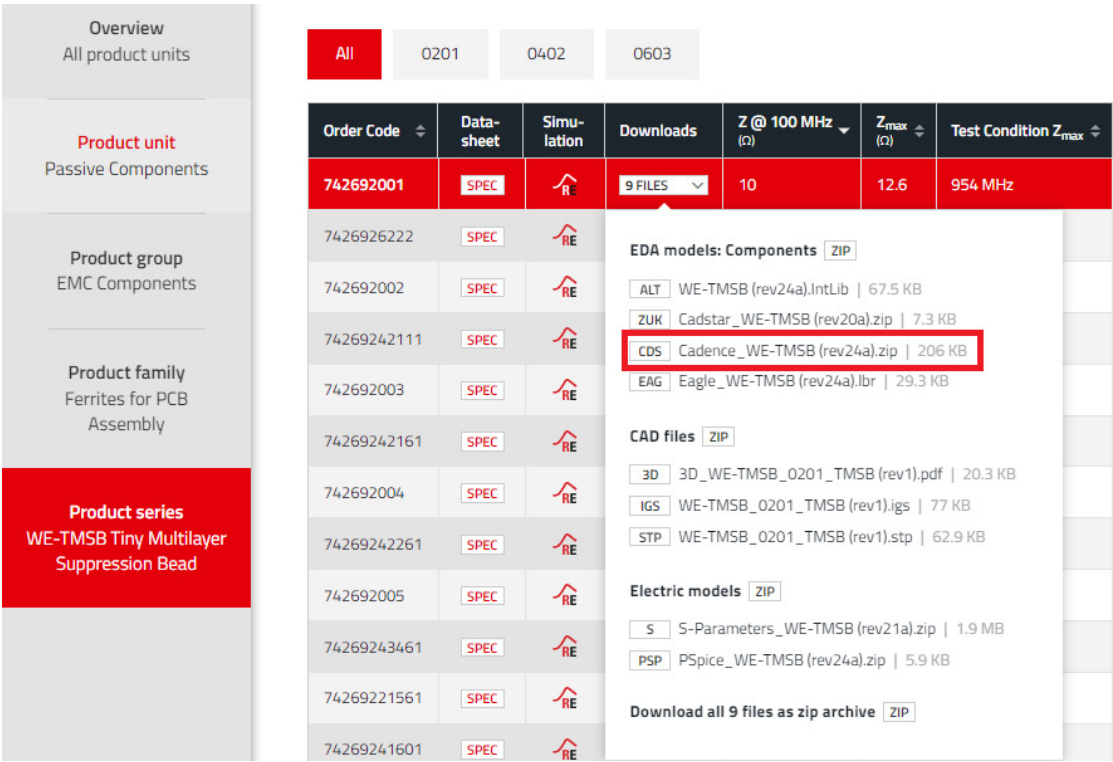


Figure 3: Download OrCAD libraries on Würth Homepage

#### b. Save the Libraries

Saving the \*.zip file directly in the folder you feel convenient and extract it there. Saving the libraries in the same folder with your design if you want.

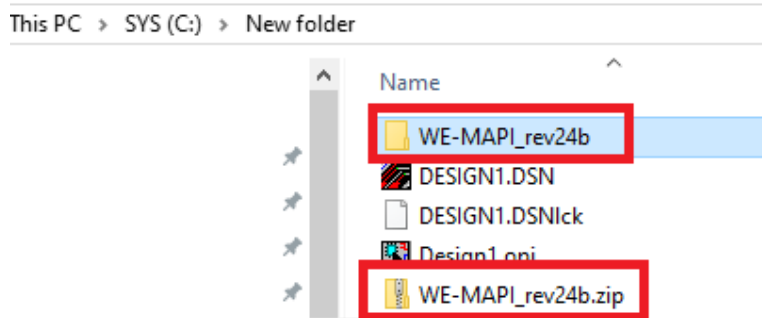


Figure 4: Library files in the same folder with schematic

#### c. Install the symbols

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

Add 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.

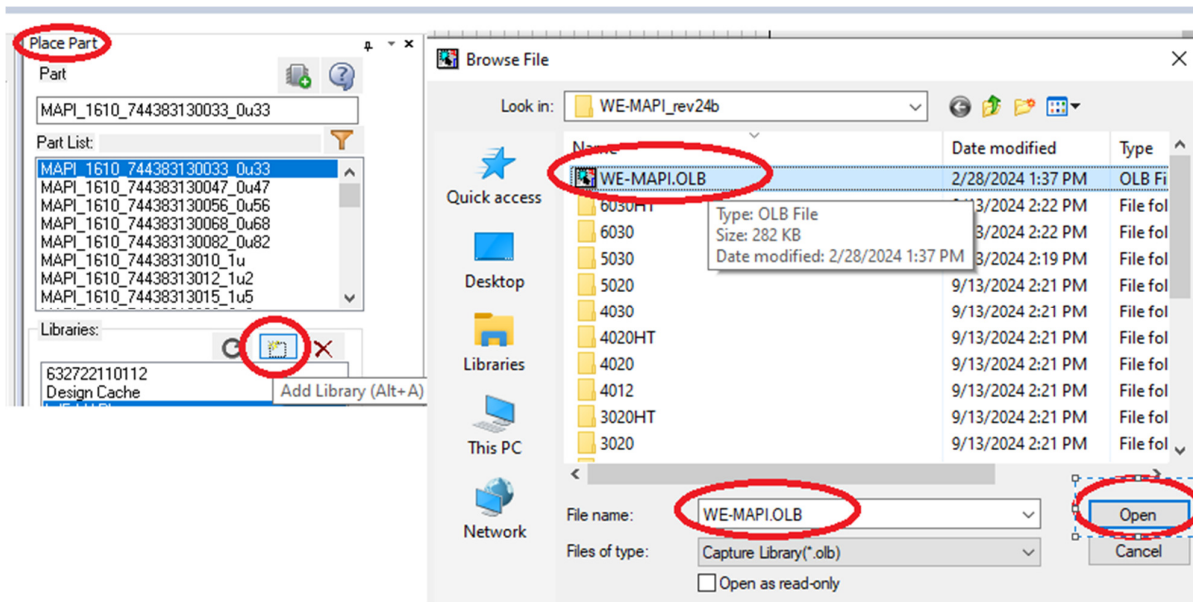


Figure 5 :Adding the library to the "Place Part" window

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.

## USER GUIDE

UGxxx | Using the OrCAD Libraries

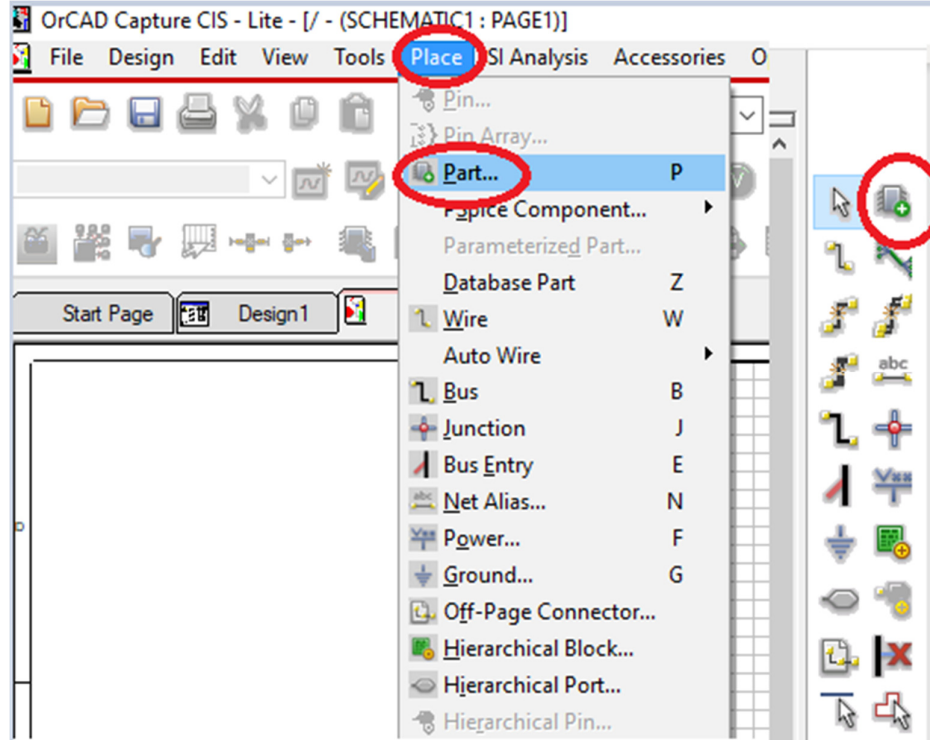


Figure 6: Navigating to the "Place Part" window using the menu (left) and using the toolbar (right)

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

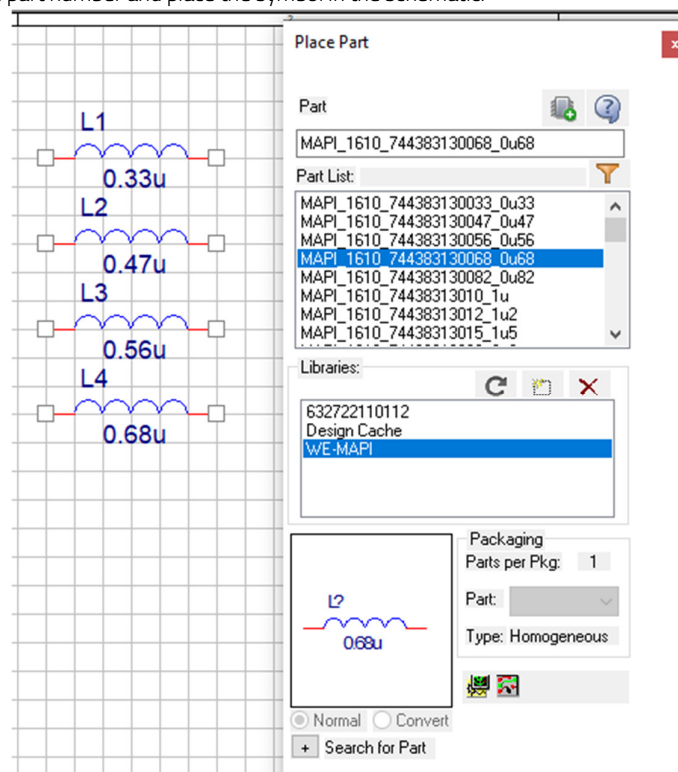


Figure 7: Place symbols in the schematic from the "Place Part" window

# USER GUIDE

## UGxxx | Using the OrCAD Libraries

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.

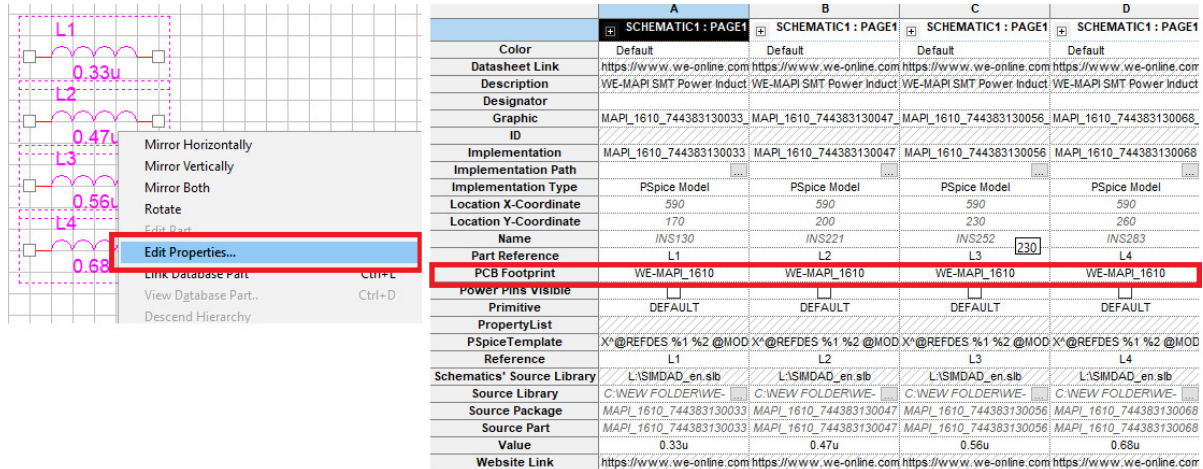


Figure 8 Information in the properties

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

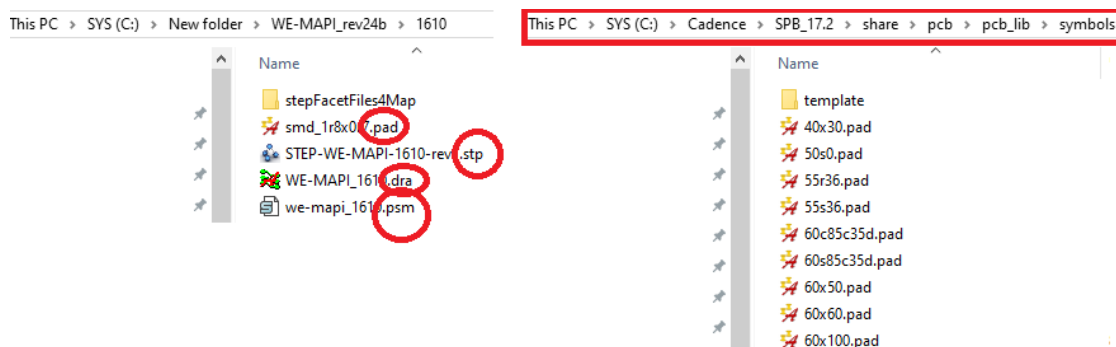


Figure 9 Copy footprint files to Cadence

Then we can start to export the netlist.

### d. Export Netlist File

Back to your design page, selecting your design, clicking "Tools" tale and select "Create Netlist" option from the menu, then the "Create Netlist" window is open.

## USER GUIDE

### UGxxx | Using the OrCAD Libraries

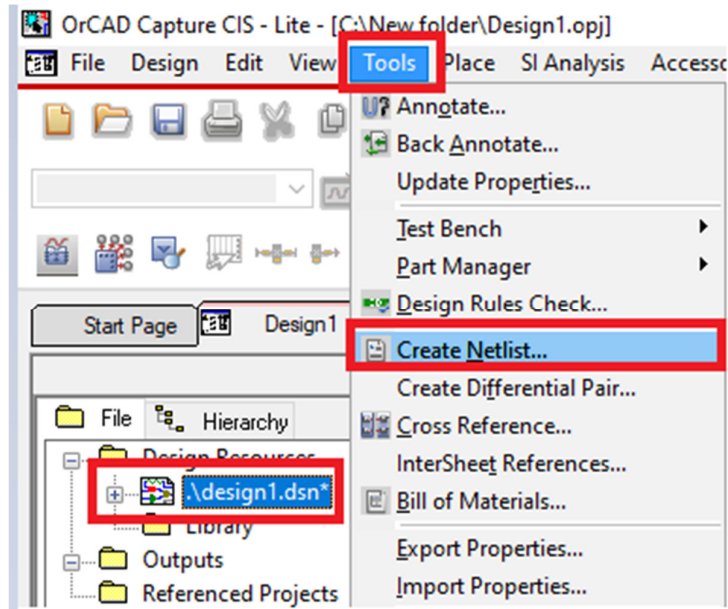


Figure 10: Navigating to the "Create List" menu options

In the "Create Netlist" window, make sure "Create PCB Editor Netlist" is selected, click "Create or update PCB Editor Board (Netrev)", change 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

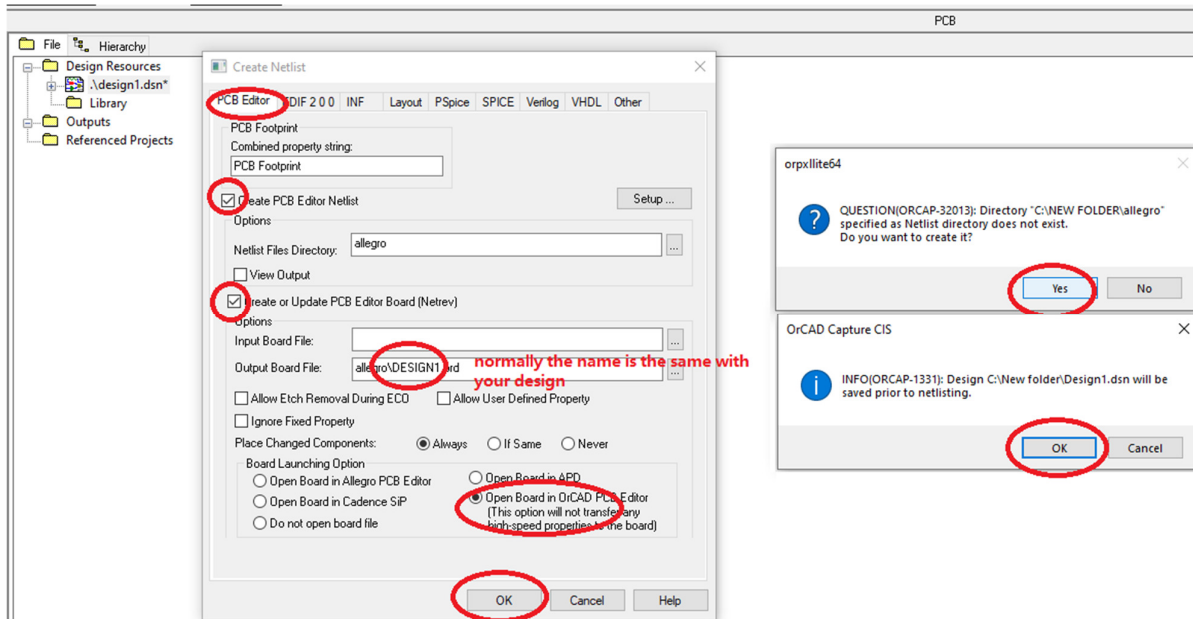


Figure 11: Navigating the "Create List" window

This way 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. And the next list files, "pstxnet.dat", "pstxprt.dat", "pstchip.dat" are all saved automatically in the "allegro" folder.



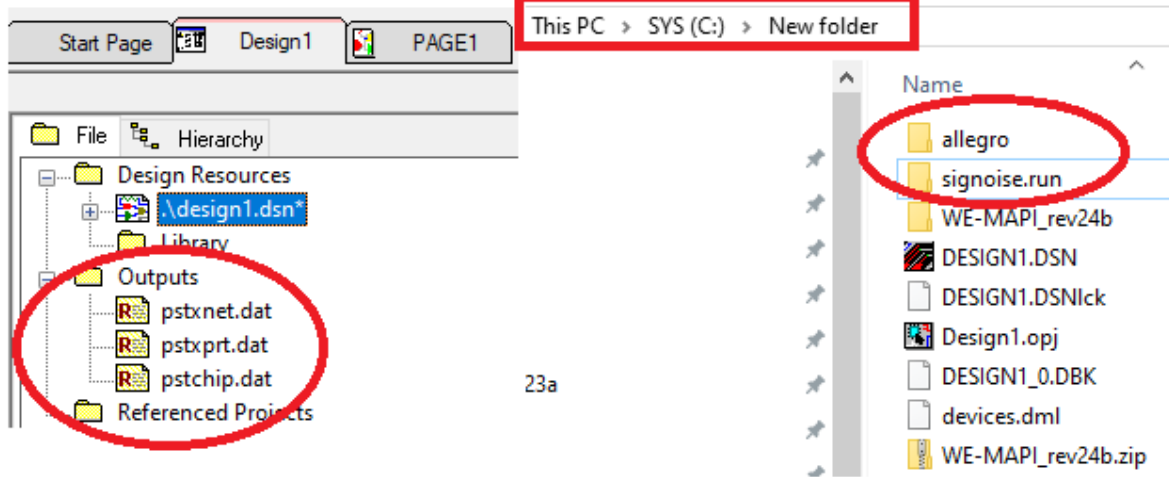


Figure 12: Netlist files and saved location.

### 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 your need, here will choose "Design entry CIS (Capture)", the most important is to change the "Import directory" to the location of your netlist files are saved.

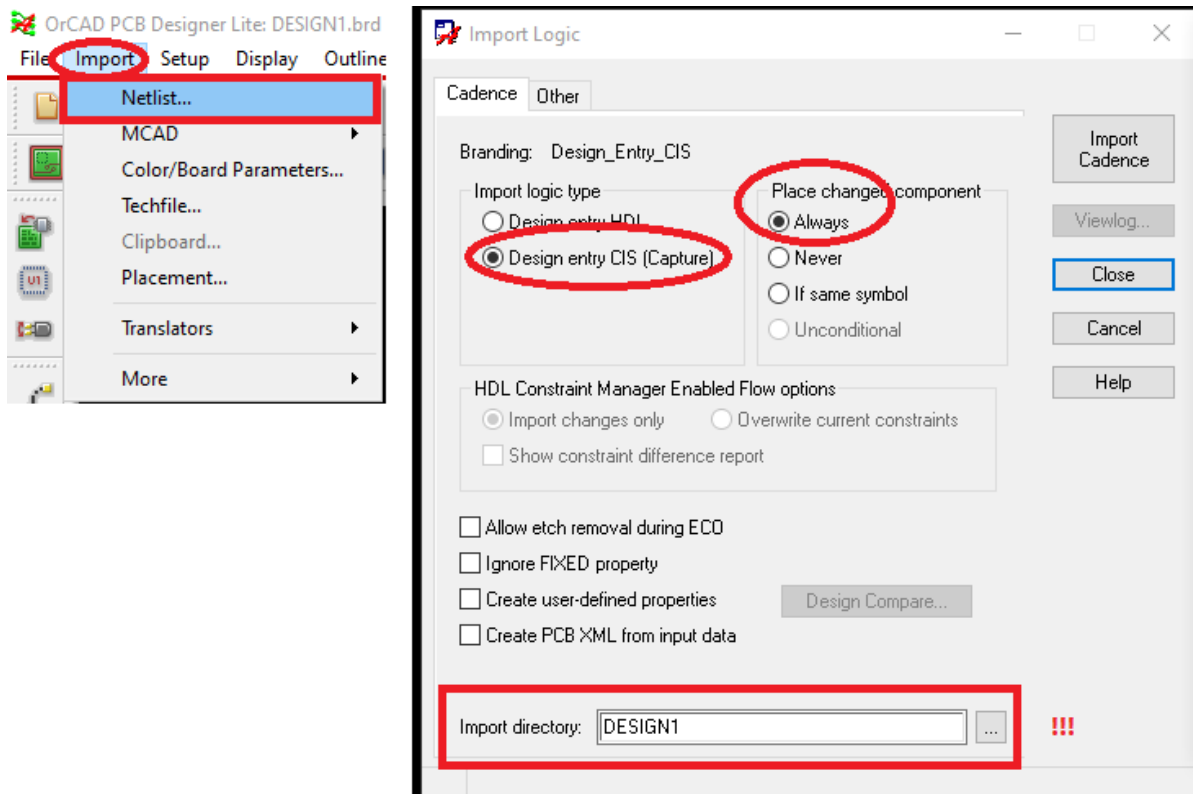


Figure 13: Navigating to the "Import Logic" menu options

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".

## USER GUIDE

### UGxxx | Using the OrCAD Libraries

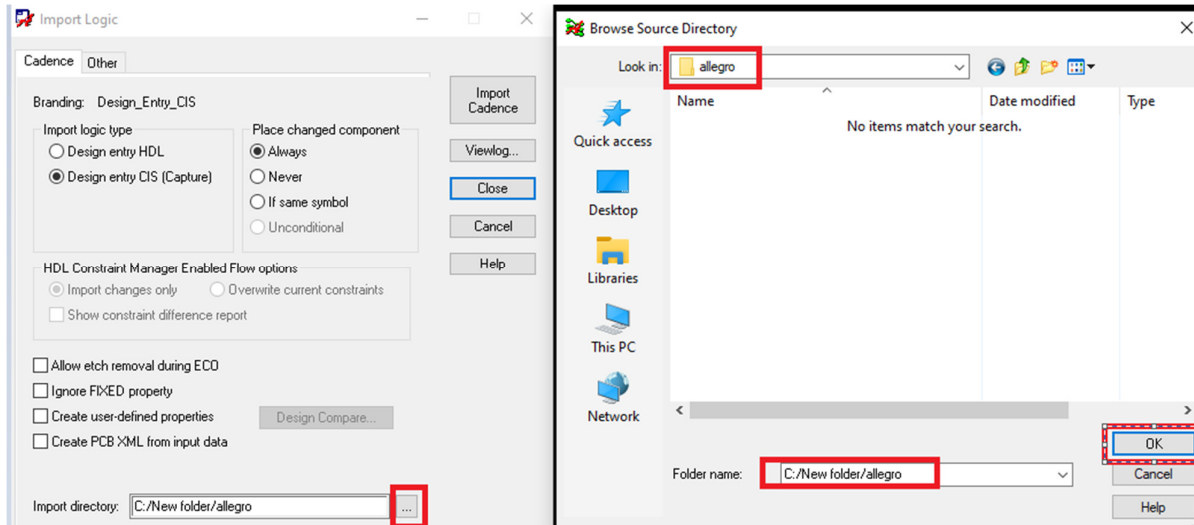


Figure 14 : Navigating to find the netlist files

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

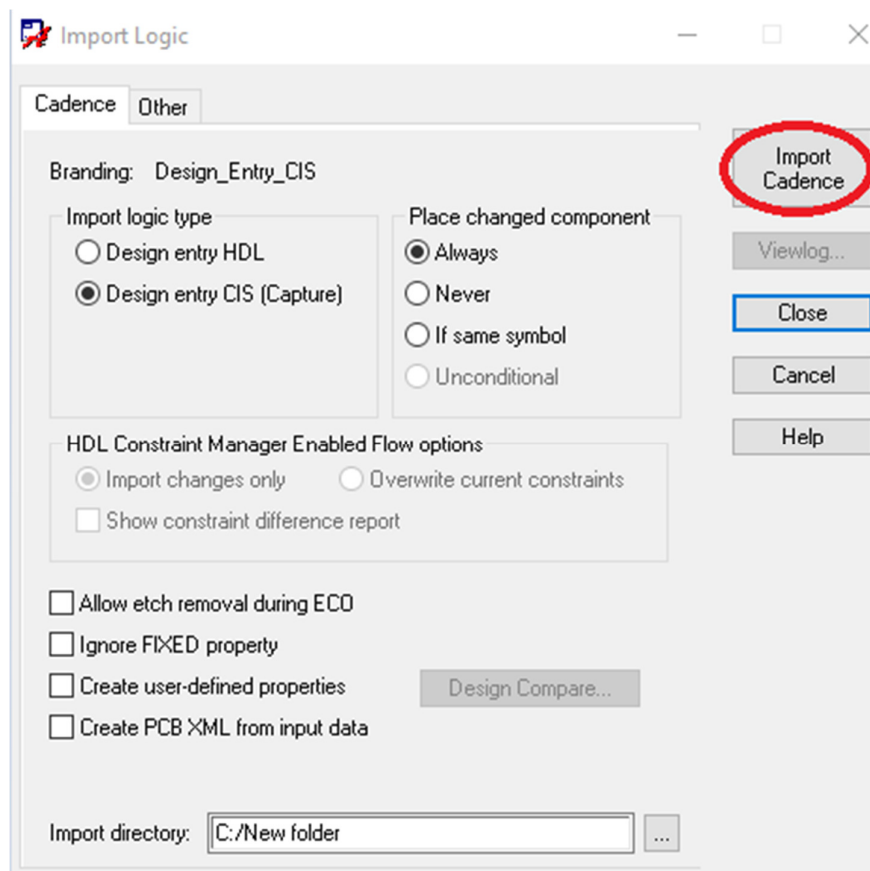


Figure 15: Import Cadence Option



# USER GUIDE

## UGxxx | Using the OrCAD Libraries

Place footprint by click the "Placement" bar to open the "Placement" window and click the components to add them to the board.

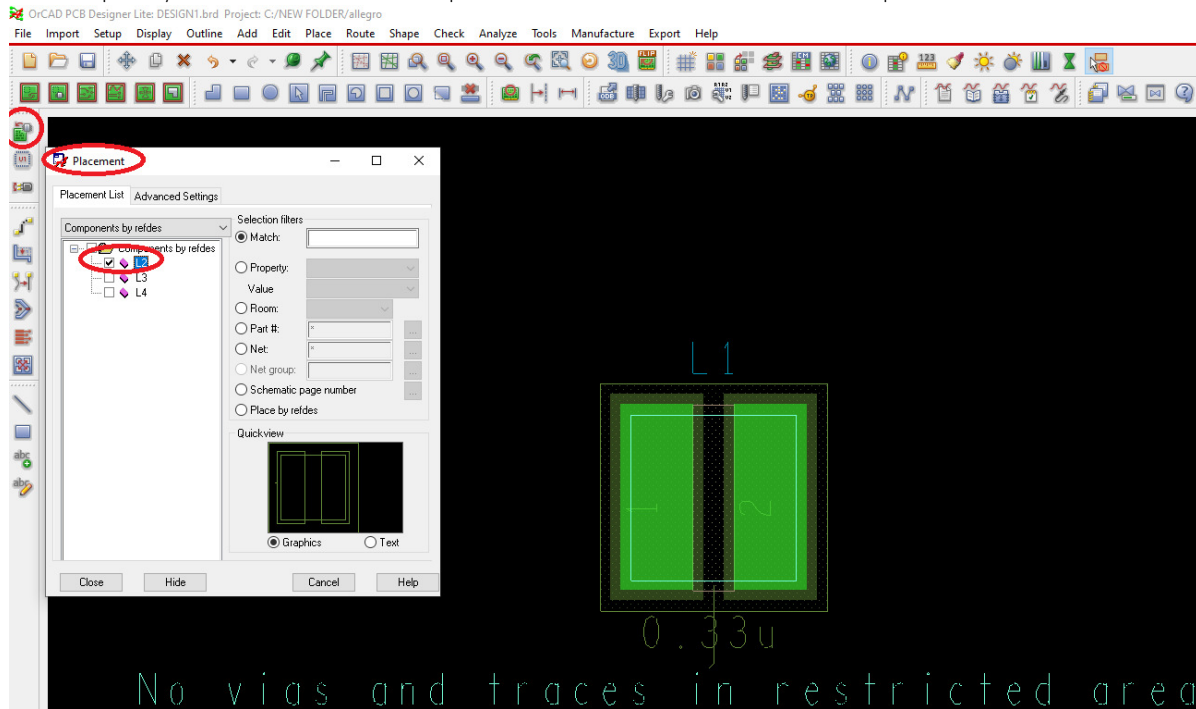


Figure 16: Add Components to board

### 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..."

Let's get started!

Add a repository to GitHub Desktop to start collaborating

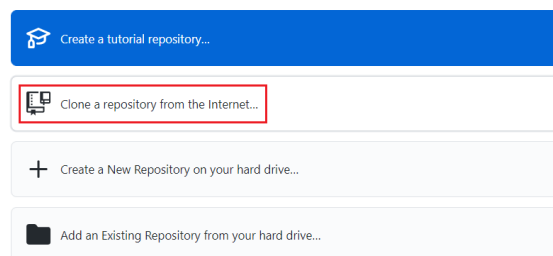


Figure 17: Clone a repository from the Internet

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.

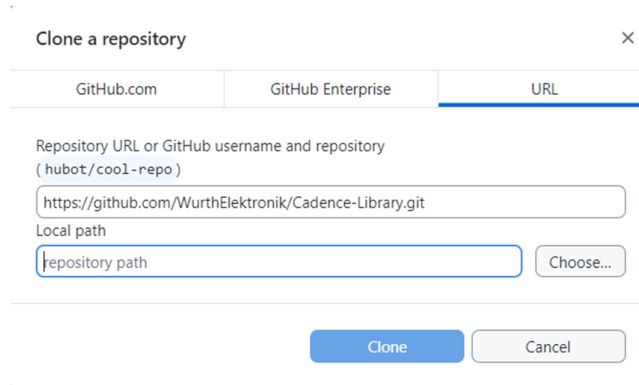


Figure 18: Cloning the Würth Elektronik OrCAD Library repository to your local directory.

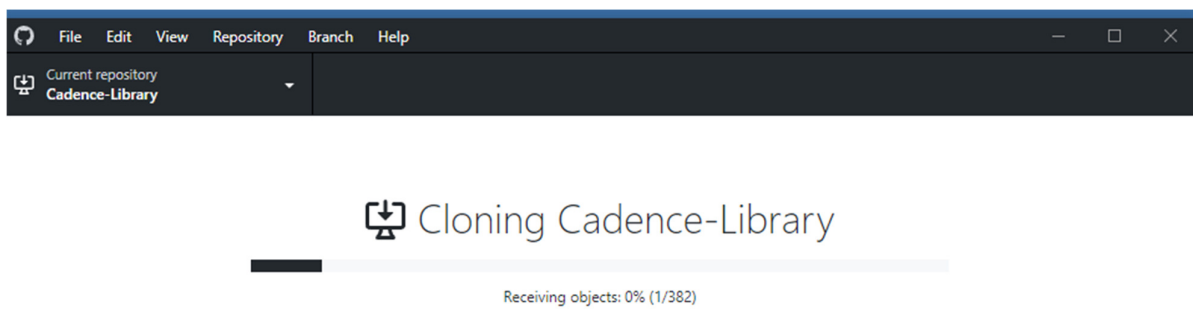


Figure 19: Cloning in progress

#### 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.

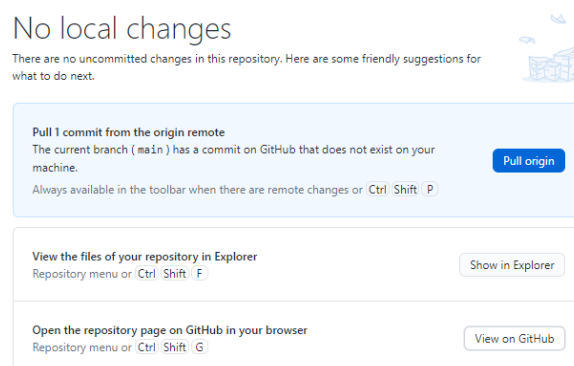


Figure 20: Pulling the repository to your local directory

# USER GUIDE

## UGxxx | Using the OrCAD Libraries

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







 hywu-eisos Upload User Manual		6c9b5d9 · 2 minutes ago	 4 Commits
 library	create folder	14 minutes ago	
 symbol	create folder	14 minutes ago	
 README.md	Create README.md	2 days ago	
 User Manual - WE Pspice Library.pdf	Upload User Manual	2 minutes ago	

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](http://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 APPLICATIONS

### USEFUL LINKS



Application Notes

[www.we-online.com/appnotes](http://www.we-online.com/appnotes)



REDEXPERT Design Platform

[www.we-online.com/redexpert](http://www.we-online.com/redexpert)



Toolbox

[www.we-online.com/toolbox](http://www.we-online.com/toolbox)



Product Catalog

[www.we-online.com/products](http://www.we-online.com/products)

### CONTACT INFORMATION

[appnotes@we-online.com](mailto: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](http://www.we-online.com)