USER GUIDE

UG017 | Using the PSpice Libraries

Ella Wu

1. INTRODUCTION

PSpice is a SPICE circuit simulator application for the simulation and verification of analog and mixed-signal circuits. It integrates easily with Cadence PCB schematic entry solutions like OrCAD Capture and comes with an easy-to-use graphical user interface.

This user guide demonstrates how to locate, download and install WE component libraries for PSpice (version 2023). WE component libraries for PSpice consist of a symbol file (*.olb) and a netlist file (*.lib). 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

2.1 Downloads from Würth Homepage

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

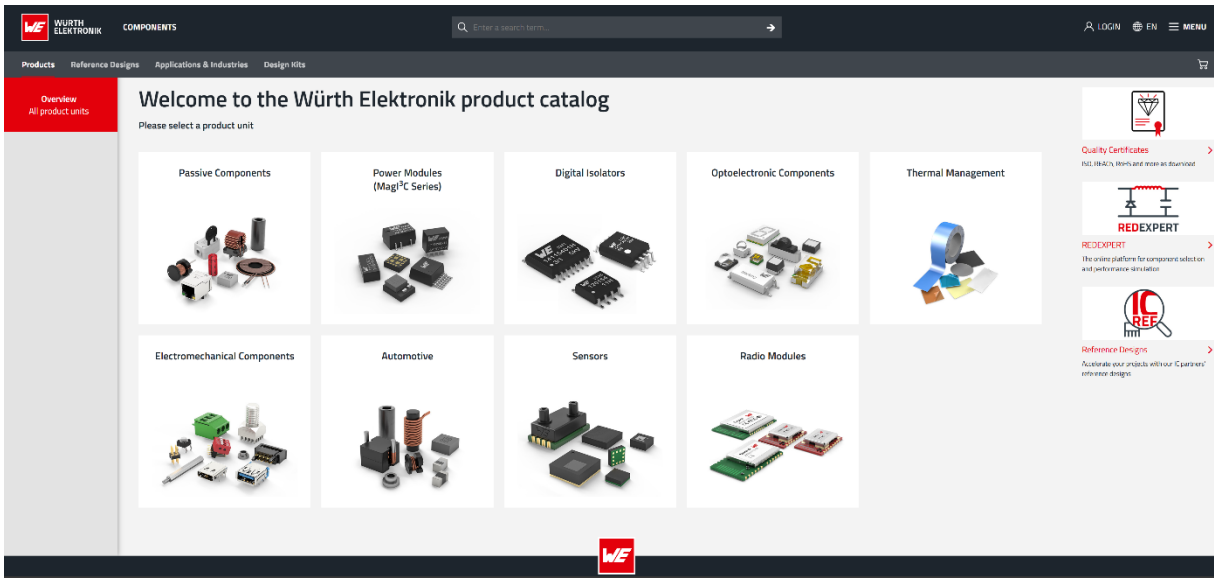

Figure 1: Würth Elektronik homepage.

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

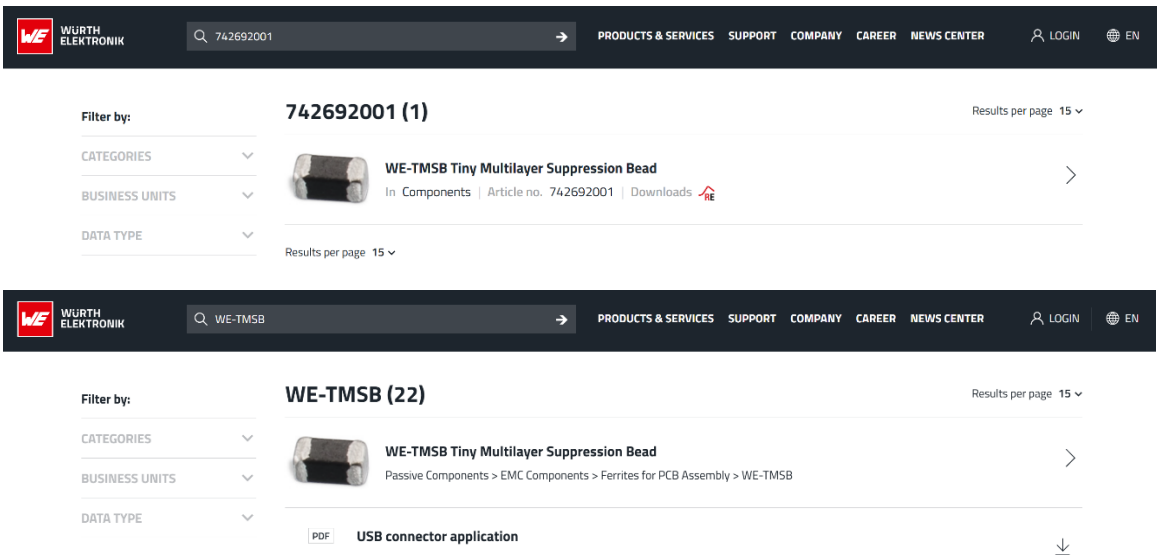

Figure 2: Search part number or series.

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

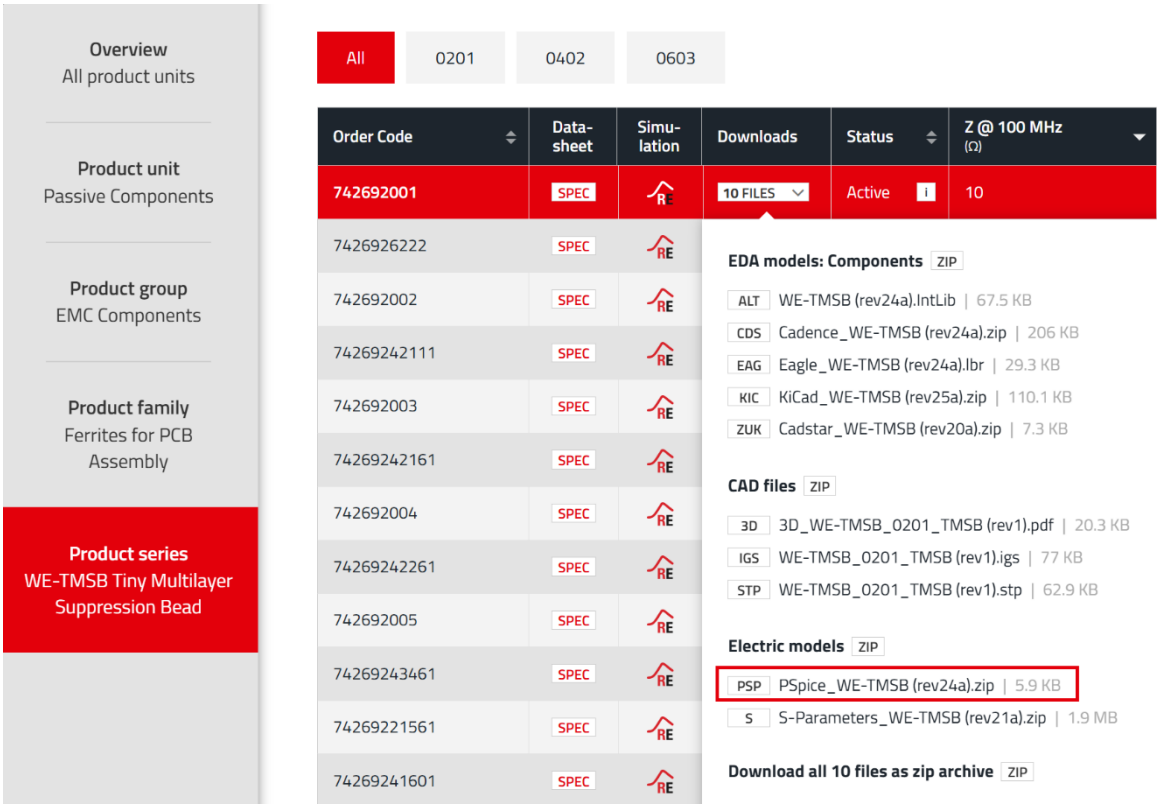

Figure 3: Download PSpice libraries on Würth Homepage.

2.2 Save the Libraries

We recommend saving the libraries in a convenient folder for your project. The symbol file (*.olb) and netlist file (*.lib) can be saved in the same local folder as the schematic file (*.DSN) and the project file(*.opj).

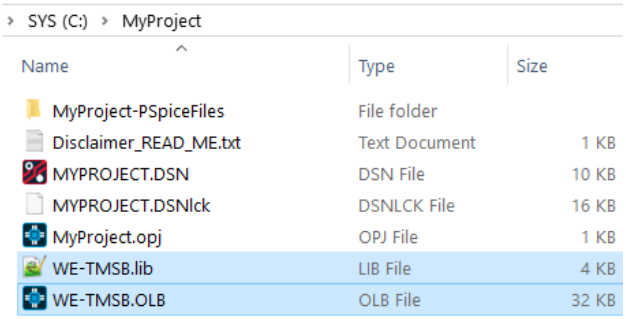

Figure 4: Model files in same folder as schematic.

2.3 Install the Local Libraries

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

Click the "Place" tab and select "Part" option from the menu. Alternatively, select "Place" in the action toolbar. You may also press the "P" key.

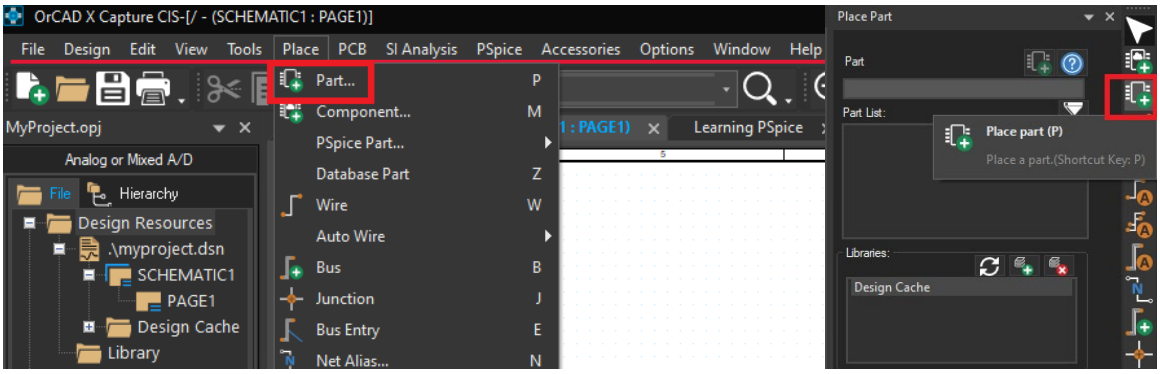

Figure 5: Navigating to the "Place Part" window using the menu (left) and using the toolbar (right).

Add the Library to your project by clicking the "Add Library" symbol in the "Place Part" window or you may also press "Alt + A". This will open the "Browse File" window where you may select the desired symbol file (*.olb) and click open.

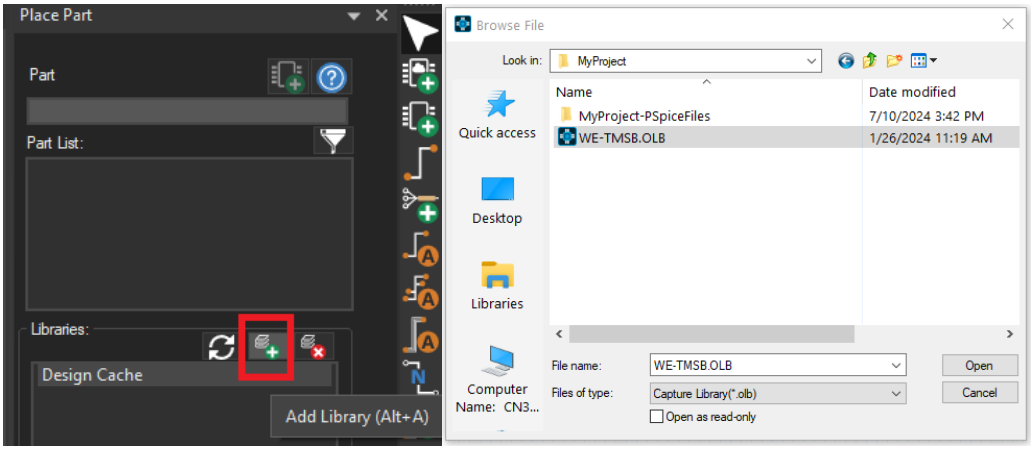

Figure 6: Adding the library to the "Place Part" window.

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

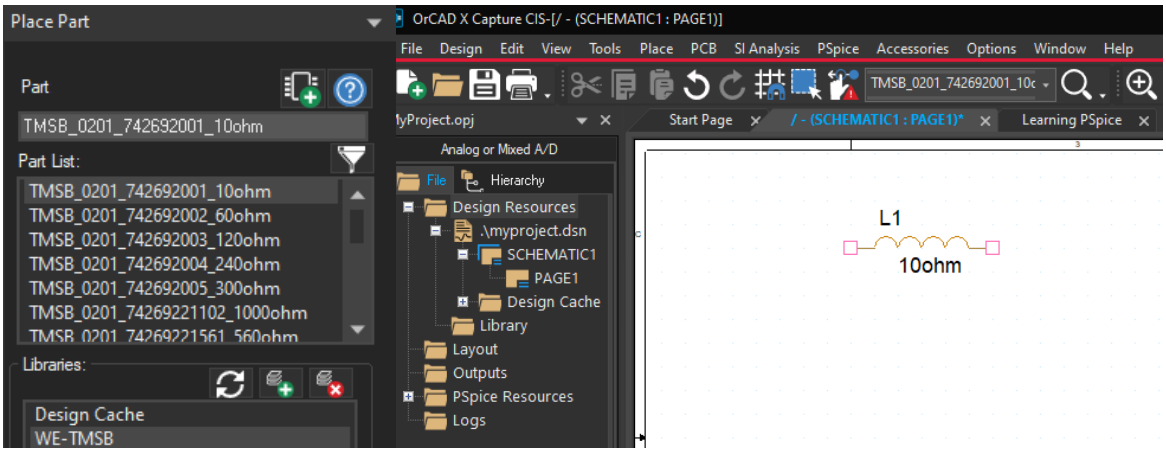

Figure 7: Place component in the schematic from the "Place Part" window.

2.4 Configure Netlist File

At this stage, it is critical to correctly configure the library to the appropriate netlist file.

Select "PSpice" in the menu and select "New Simulation Profile" from the menu. Alternatively, select "Edit Simulation Profile" if you have already created a Simulation Profile.

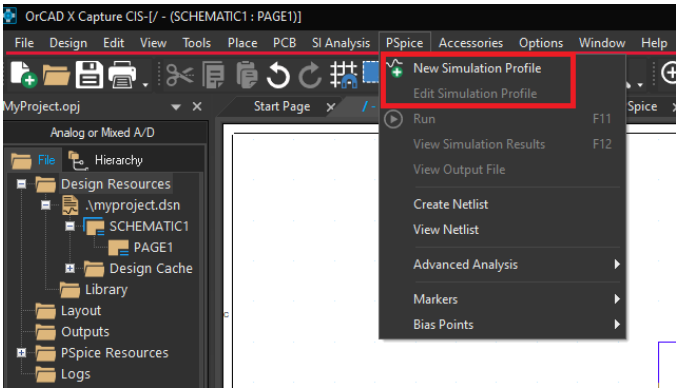

Figure 8: Navigating to the "Simulation Profile" menu options.

In the "Simulation Settings" window, click "Configuration Files", "Library" and then "Browse…". Once you have chosen the folder containing the relevant library, you can select the desired netlist file (*.lib) and click open.

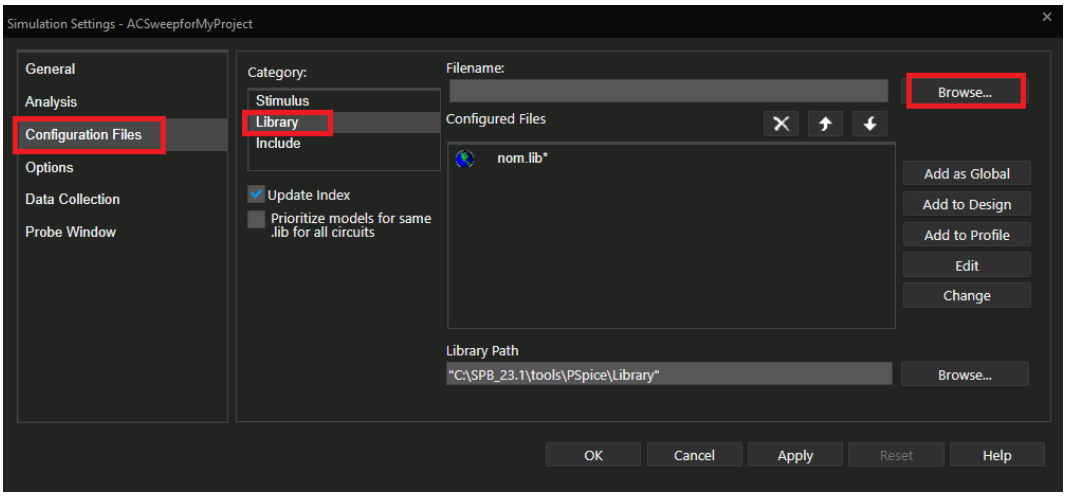

Figure 9: Navigating the "Simulation Settings" window to add the relevant netlist file.

You may now either select "Add as Global" if you wish the library to be available for all designs, or "Add to Design" if you wish to only use the library for the current design. Click "OK" and save the simulation profile.

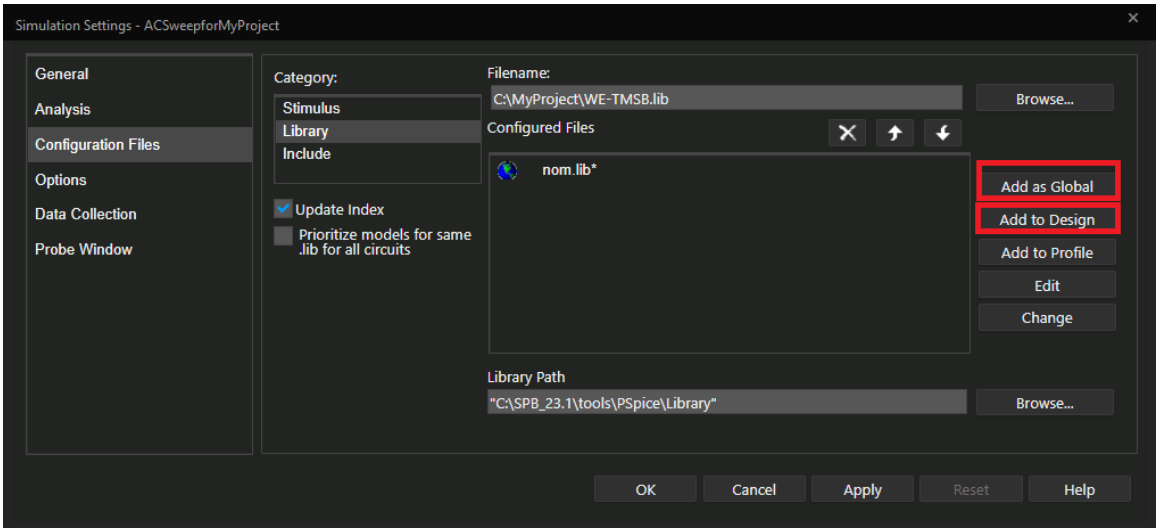

Figure 10: Adding the library to be either available for all designs or limited to the current design.

3. INSTALL FROM GITHUB REPOSITORY

3.1 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.

3.2 Clone the Library

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

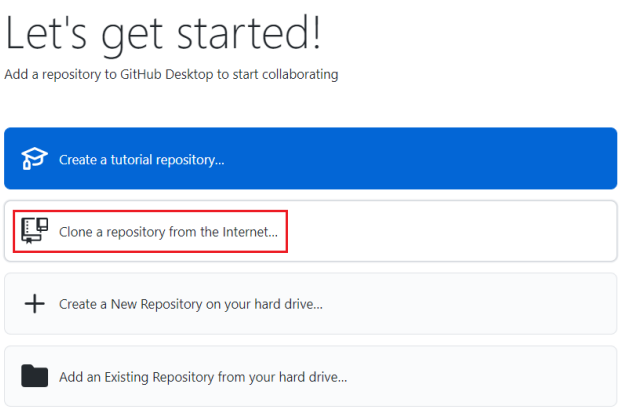

Figure 11: Clone a repository from the Internet.

Enter the URL of Würth Elektronik PSpice Library repository https://github.com/WurthElektronik/Pspice-Library.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.

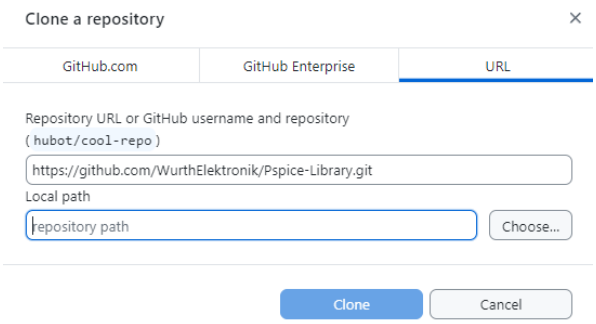

Figure 12: Cloning the Würth Elektronik PSpice Library repository to your local directory.

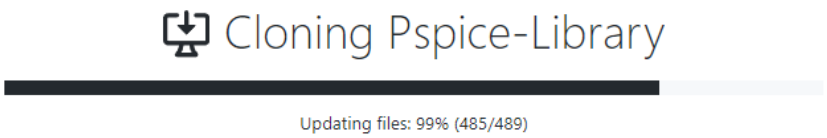

Figure 13: Cloning in progress.

3.3 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.

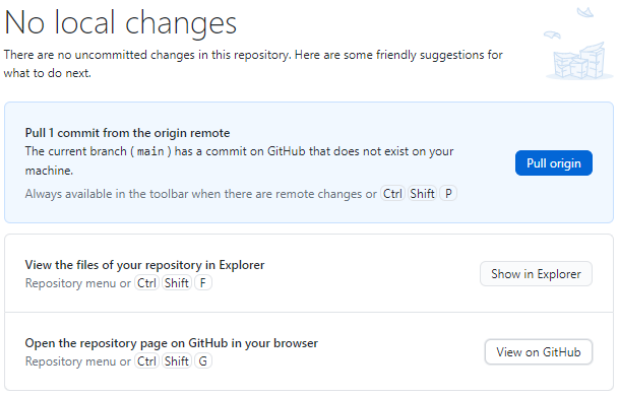

Figure 14: Pulling the repository to your local directory.

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

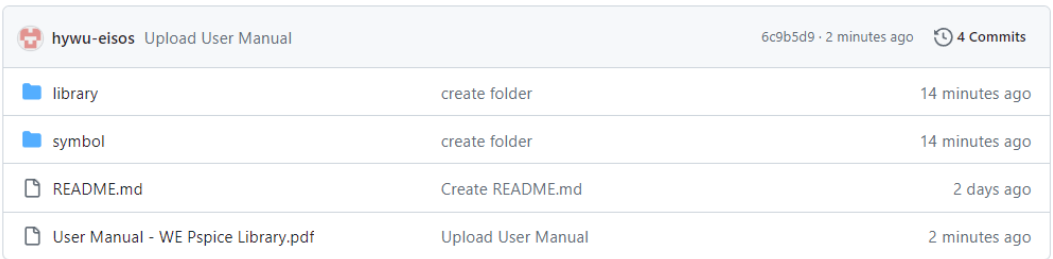

Figure 15: View the updates on GitHub.

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