WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

USER GUIDE

UGXXX | WE KiCad Library

Melon Huang

1. Introduction

KiCad is a free and open-source software suite for electronic design automation (EDA). It facilitates the design and simulation of electronic hardware. It features an integrated environment for schematic capture, PCB layout, manufacturing file viewing, SPICE simulation, and engineering calculation. Tools exist within the package to create bill of materials, artwork, Gerber files, and 3D models of the PCB and its components.

Würth Elektronik has a growing portfolio of models available for use in KiCad. These models allow KiCad users to easily design schematic and layout and view 3D model before producing PCB.

For any new users of a tool, the basic functionalities must be learned before the tool can be put to use. For KiCad, knowing how to correctly install and find the models is crucial. Our models can be installed via three ways: the Würth Electronics Homepage, the KiCad built-in library and the GitHub repository.

Note: The following instructions refer to KiCad versions 7.0 or higher.

2. Install from website

Note: KiCad models on the Würth Elektronik Github are always the latest.

2.1 Download from Würth homepage

Browse www.we-online.com >> Components >> Find the product unit according to the catalog.

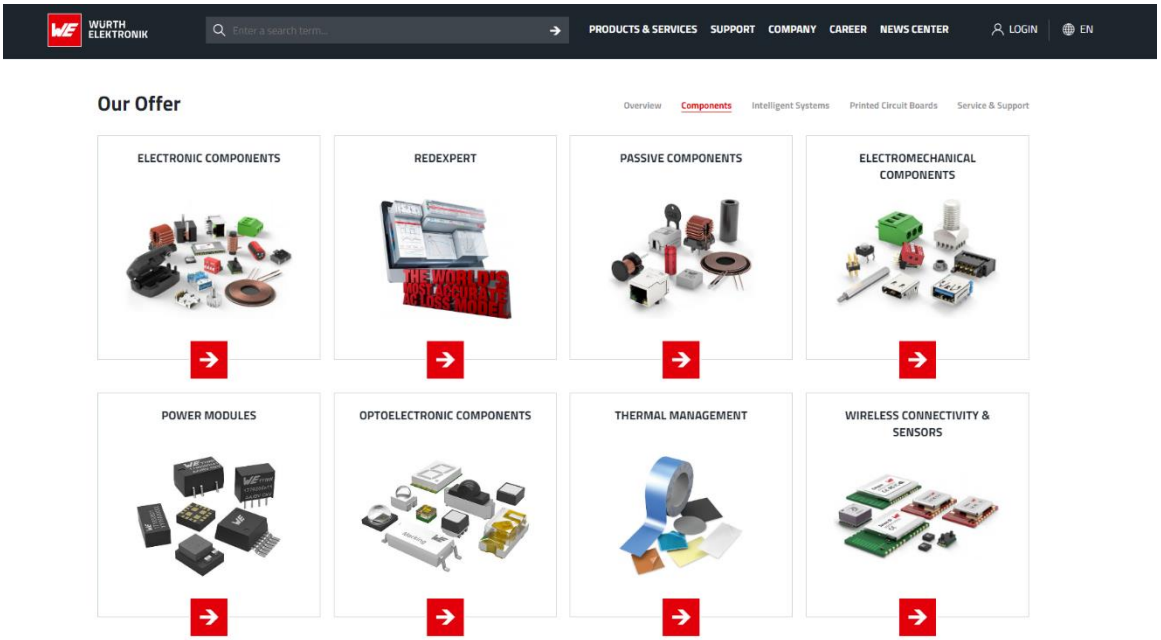


Figure 1: Würth Elektronik Homepage

Or search for the part number or series you are interested in. For example, search part number 7443835400033 or series WE-MAPI.

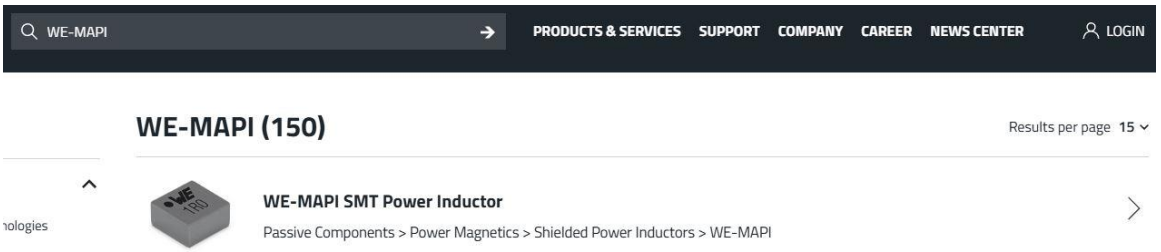


Figure 2: Search part number or series

For the desired part number, expand the Downloads dropdown, select and click the KiCad zipped file to download it.

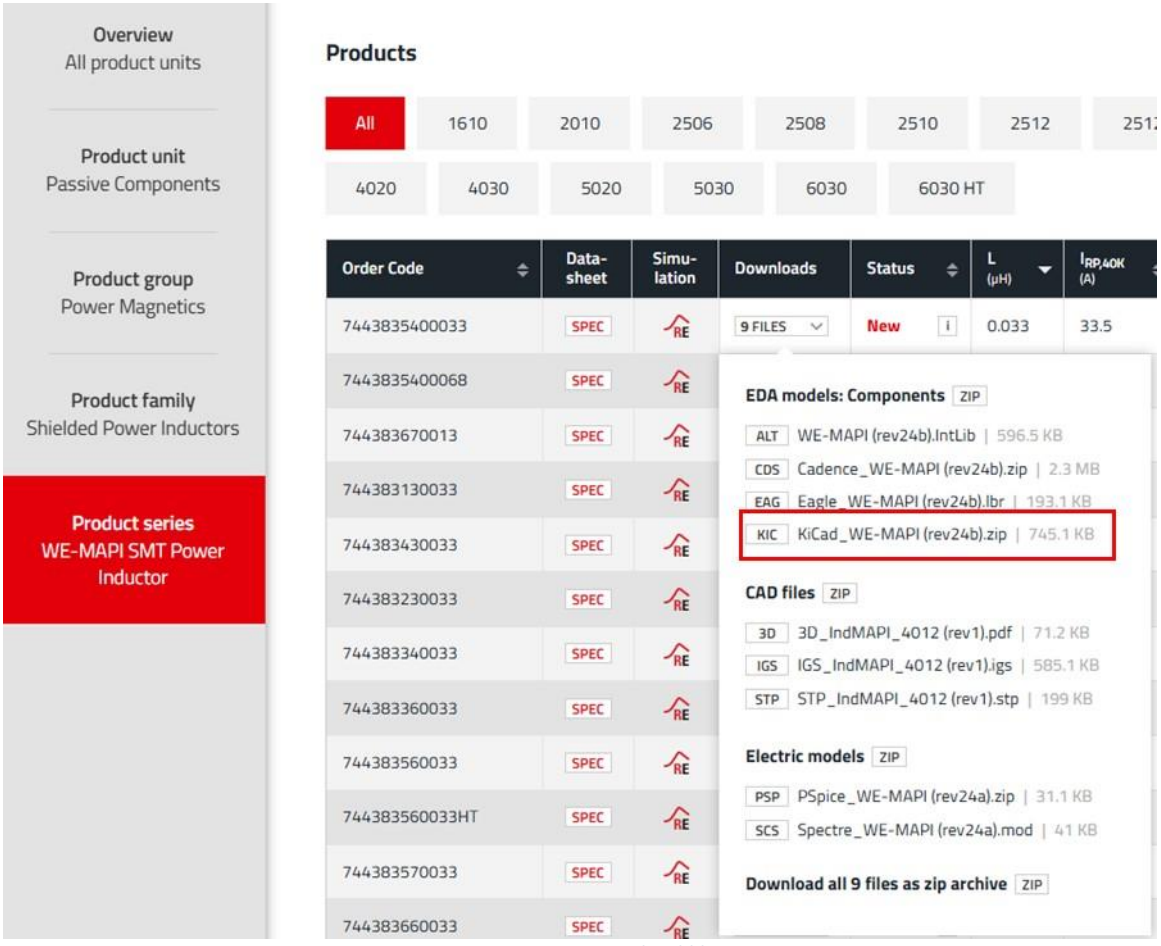


Figure 3: Download library

Unzip the downloaded file, you will see overview as below:

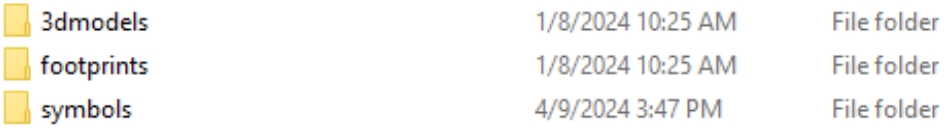


Figure 4.1: Overview

The following folders can be seen inside the footprints folder.

Figure 4.2: footprints folder

The following folders can be seen inside the 3dmodels folder.

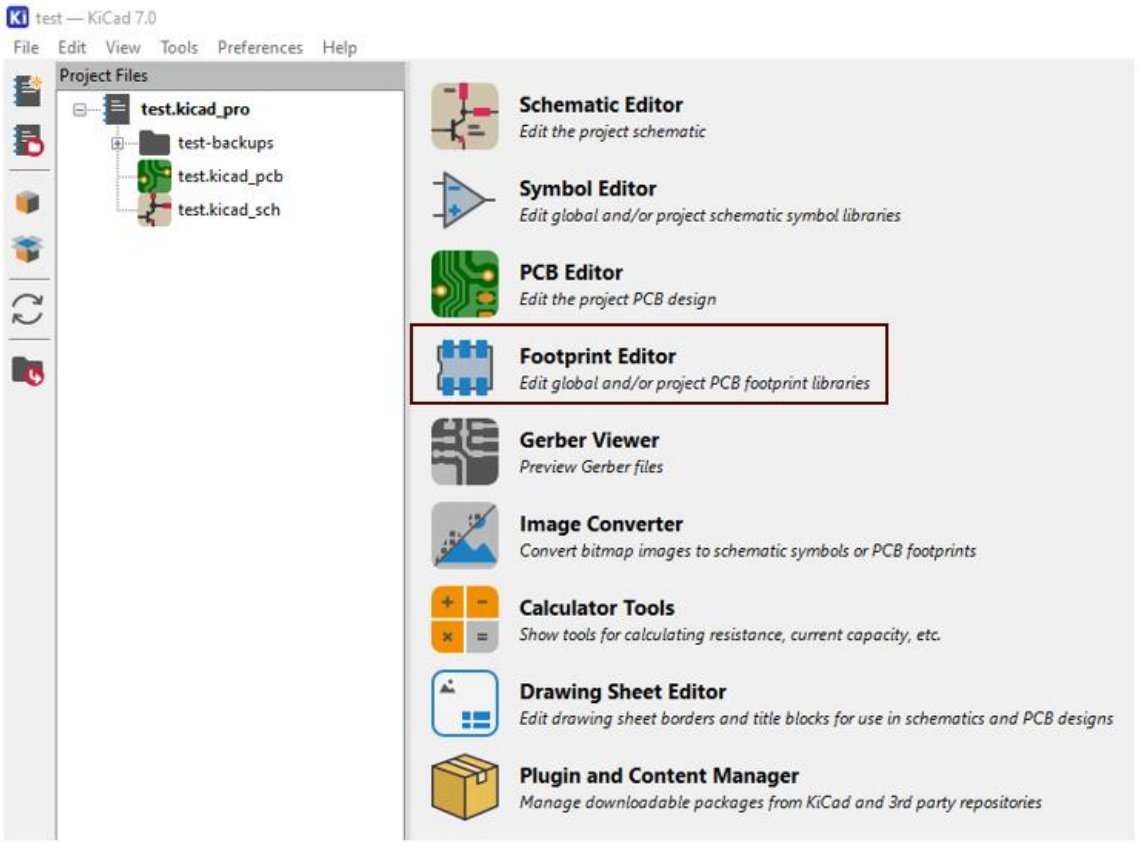
Figure 4.3: 3dshapes folder

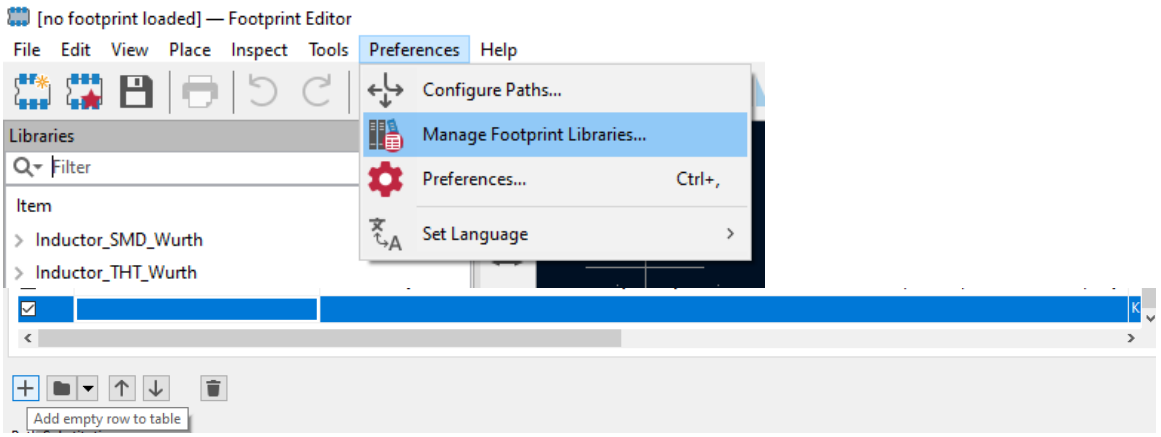
The following folders can be seen inside the symbols folder.

Figure 4.4: symbols folder

2.2 To install the model

Footprint import: Open **the Footprint Editor** in KiCad, click the **Manage Footprint Libraries** in Preference, click "**+**" to add an empty row, input the Nickname and Library Path. **Note that Nickname must be the same as .pretty prefix content**.





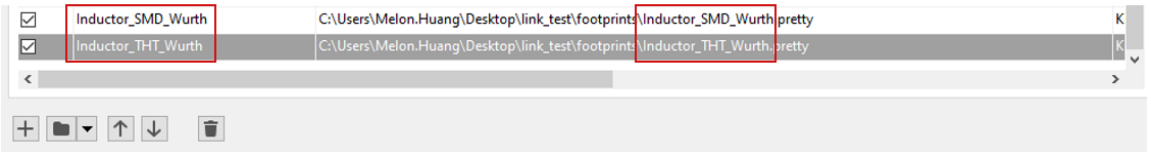


Figure 5:Importing the footprint

The process is the same for symbols import: open **Symbol Editor** in kicad, click **Manage Symbol Libraries** in the Preferences toolbar, click "**+**" to add empty row, input the Nickname and Library Path. **Note that Nickname must be the same as .kicad_sym prefix content**.

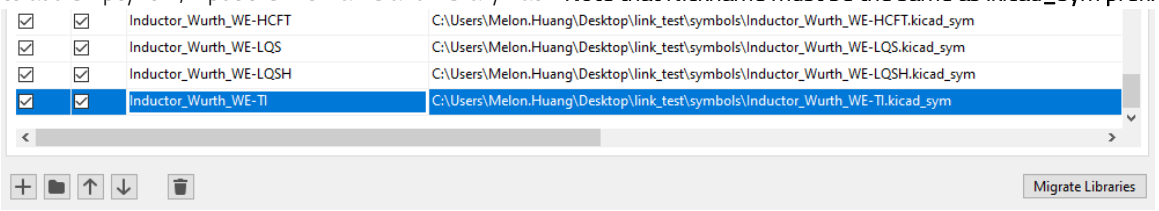


Figure 6: Importing the symbol

The symbols and footprints are now linked. However, the 3D Models in the footprint are still missing. You can link the 3D models as following:

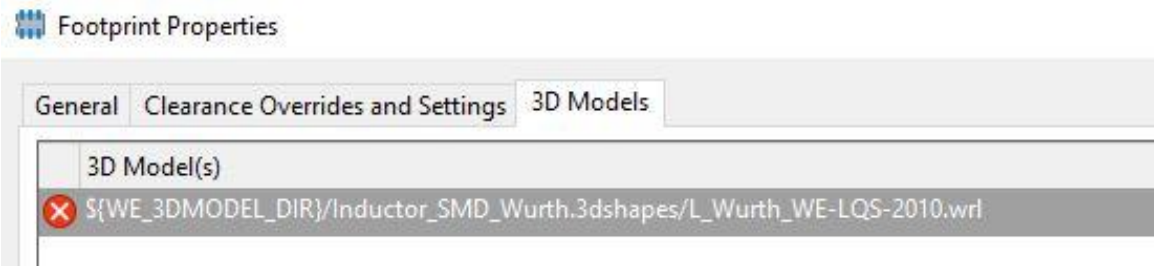


Figure 7: The 3D modes window when the model is missing.

Click 'Configure Paths' in Footprint Editor.

Figure 8.1: Open Configure Paths

Add new variable paths for directory where you have saved the downloaded files, as blue-framed shows. (Note: netlists folder is optional because it's for our future update plan to attach LTspice netlist with symbol in our library.

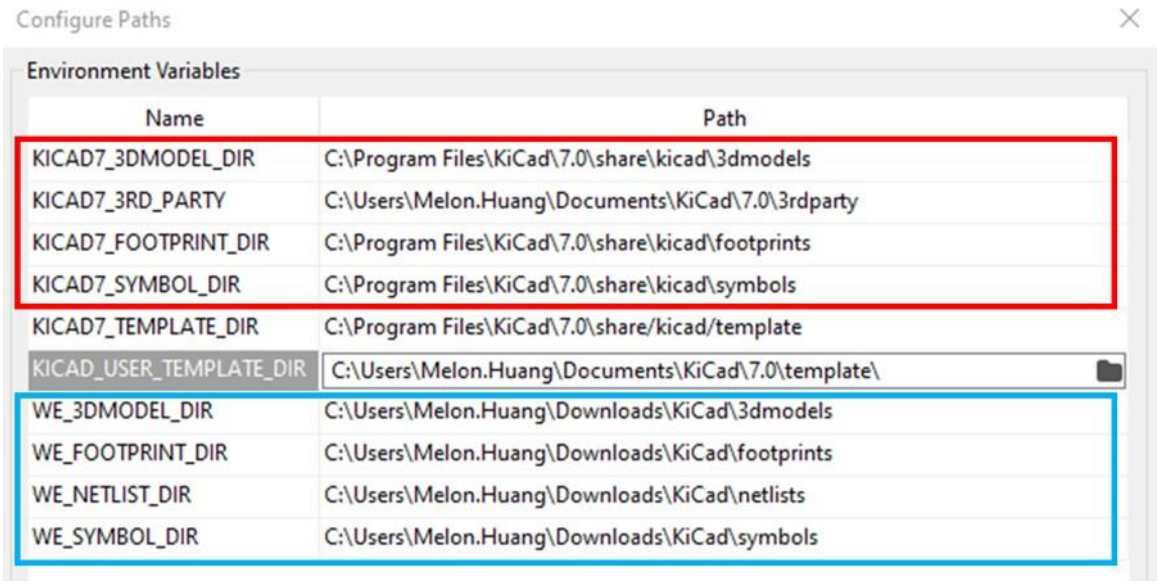


Figure 8.2: Add new variable paths

The footprints, symbols and 3dmodels you download are linked together. You can now design schematic and PCB with them now.

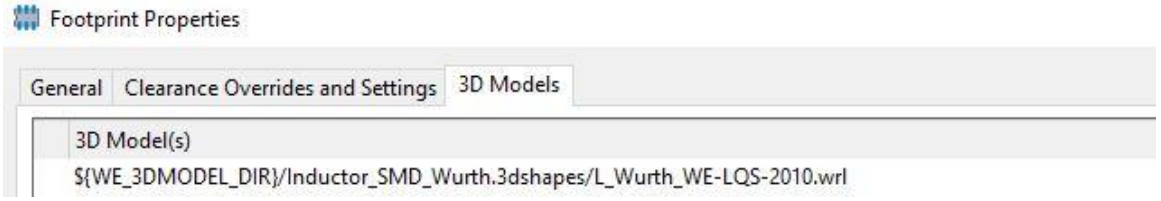


Figure 9: A correctly linked 3D model

3. Install from KiCad built-in library

Note: KiCad built-in library is reviewed and updated periodically. We therefore recommend downloading the Würth Elektronik libraries from the GitHub download page.

3.1 KiCad built-in library

Type "Wurth" in Footprint Editor filter, the results are as below:

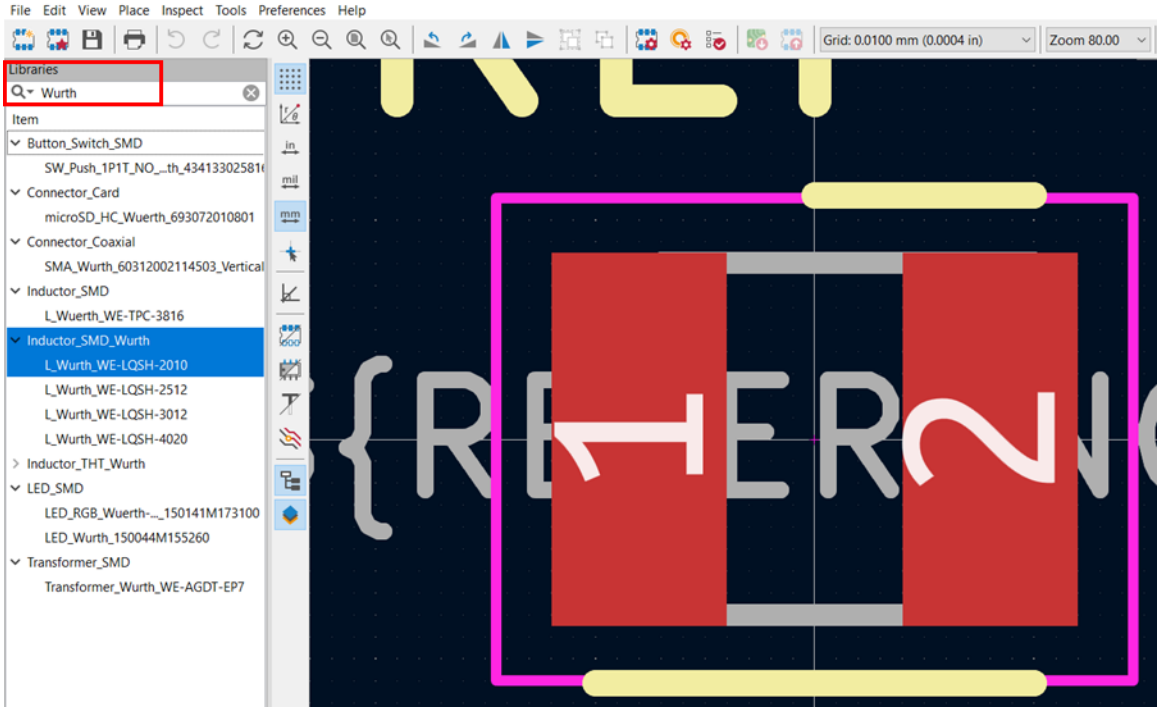


Figure 10: Example of a component in the KiCad built-in library

You can use the built-in library directly for schematic and layout design.

4. Install from GitHub repository

Note: KiCad library in GitHub repository is always the latest.

4.1 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 the opening GitHub Browser webpage authenticate yourself and give permission to the GitHub desktop application. Then the process jumps back to the Desktop tool/application.

4.2 Clone the library

From GitHub Desktop, click the button Clone a repository from the Internet. Enter the URL of Würth Elektronik KiCad Library repository https://github.com/WürthElektronik/KiCad-Library.git and define the local directory to clone the repository.

Then click the Clone button, all the files from the online repository will begin to synchronize into local.

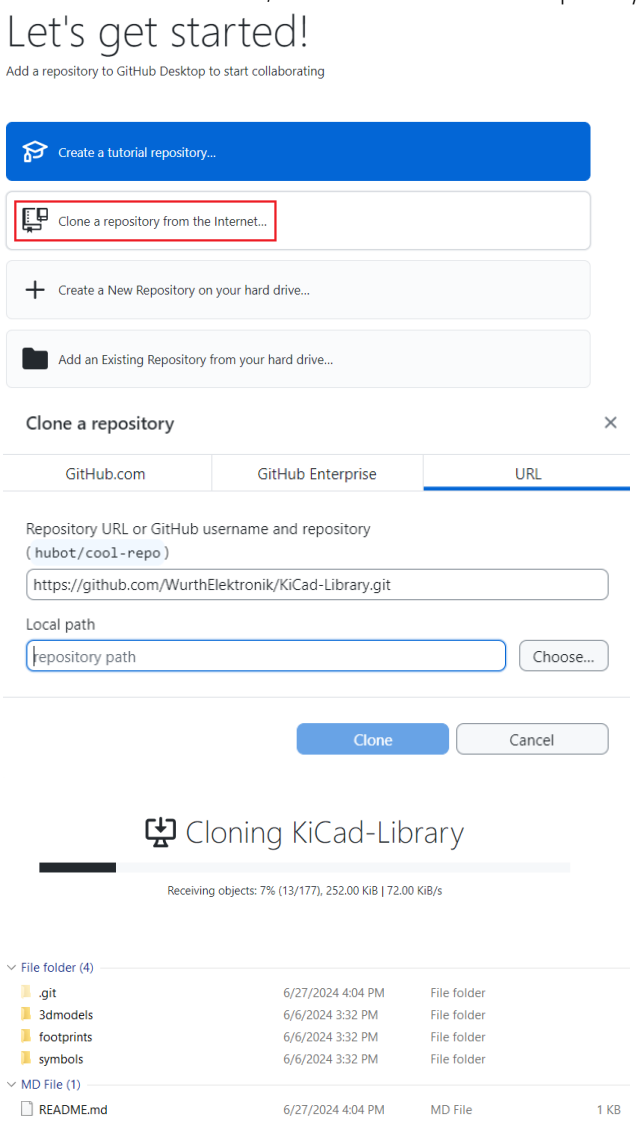


Figure 11: Clone the library

4.3 Synchronize local library from GitHub

If there are any update in GitHub repository, GitHub Desktop will detect it and you can "Pull" the update to your local.

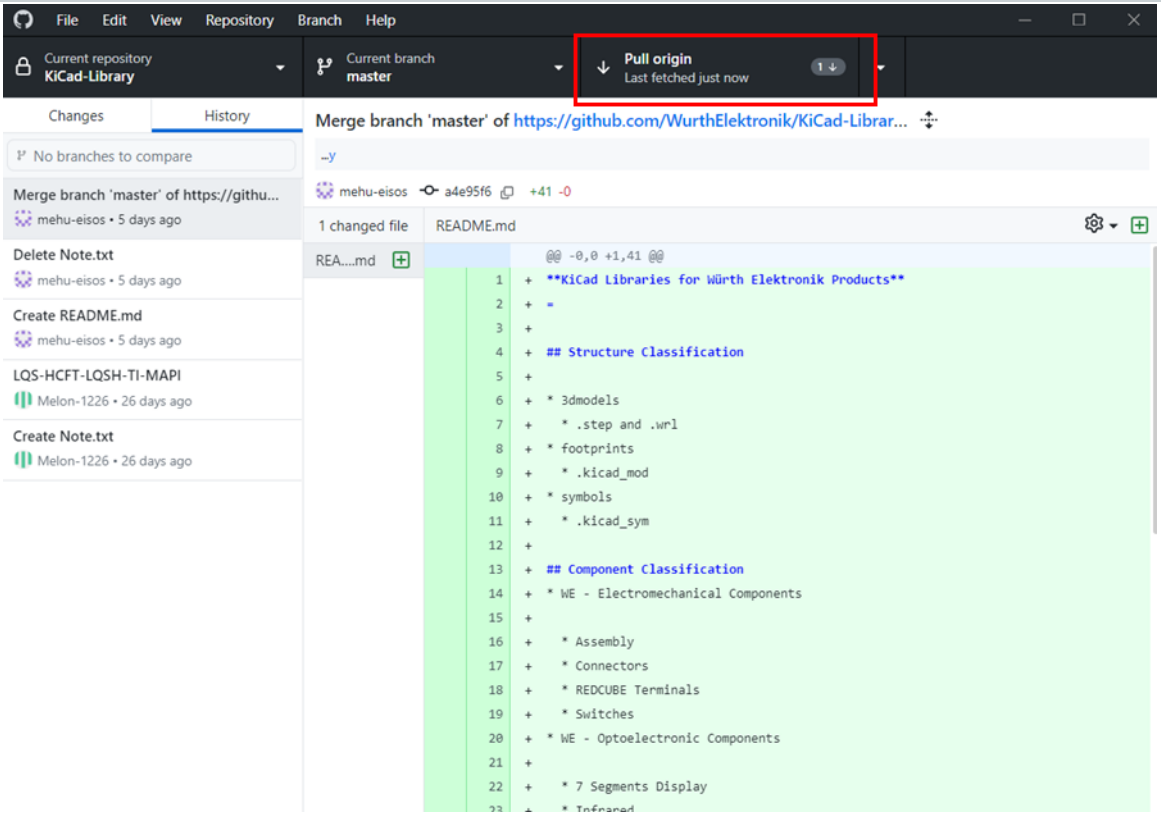


Figure 12: Synchronize from GitHub

Use the models in your local folder as introduced in Chapter 2.

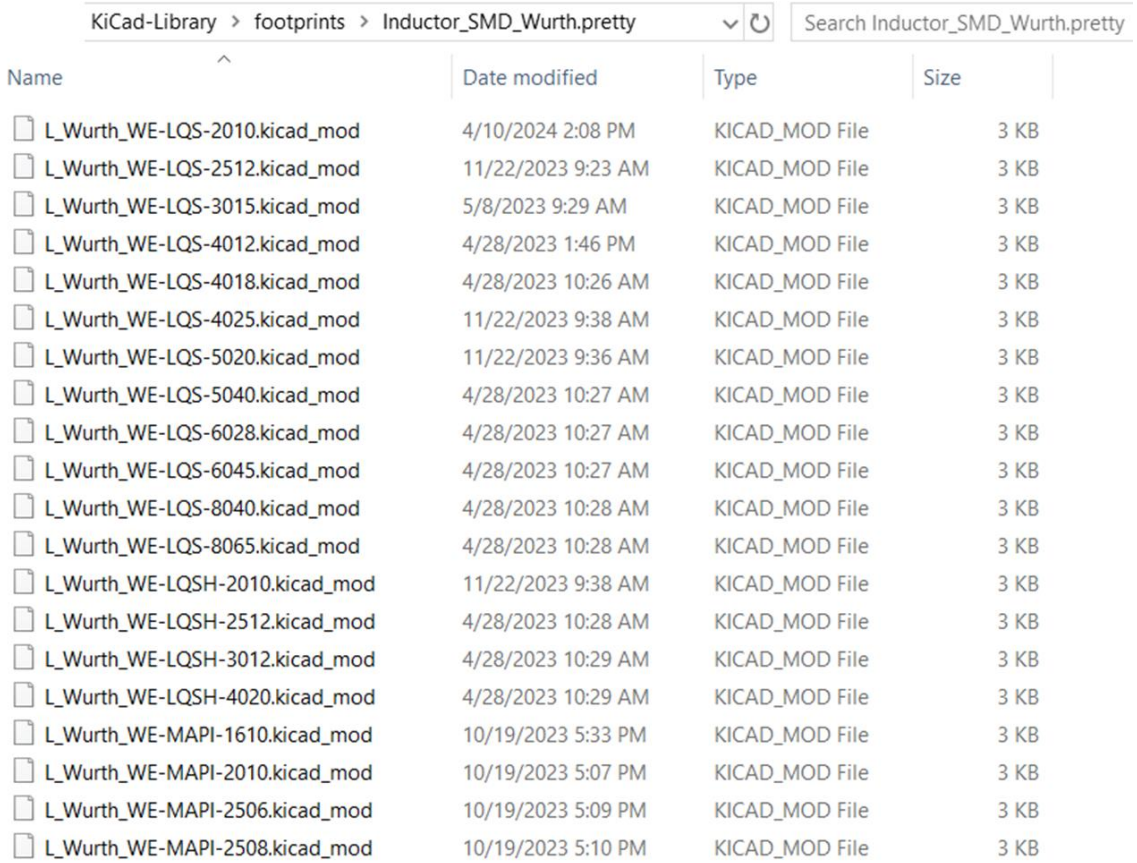


Figure 13: Use for layout

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 APPLICATIONS

USEFUL LINKS

Application Notes
www.we-online.com/appnotes

REDEXPERT Design Platform

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