

USER GUIDE

UG019 | 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

Visit the [WE product portfolio](#) and navigate to the product you are interested in.

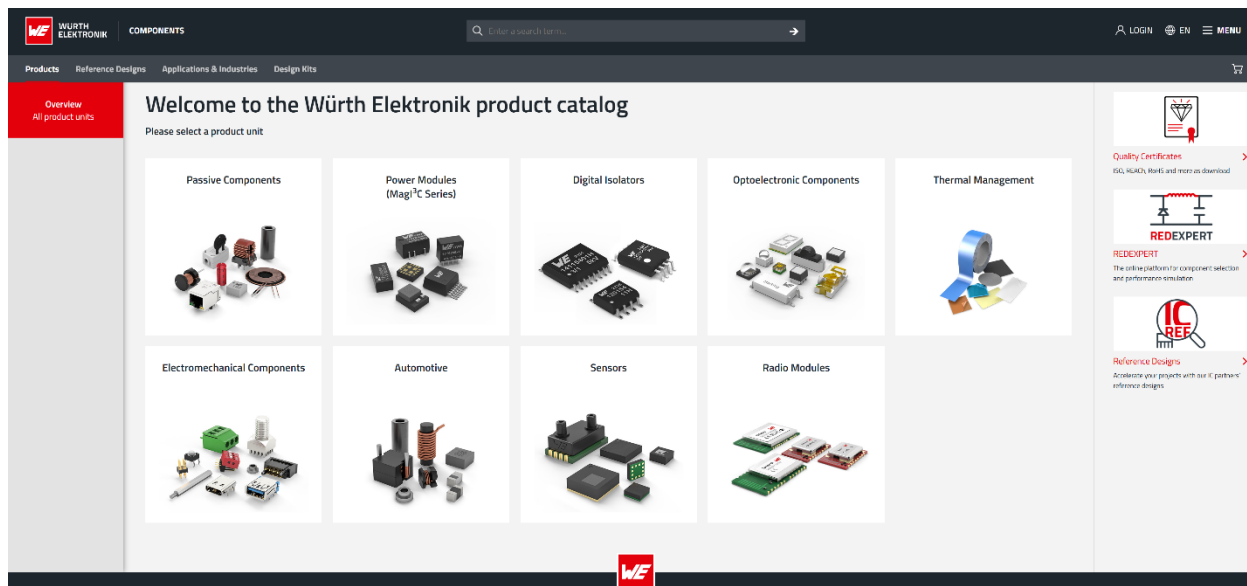


Figure 1: [Würth Elektronik Homepage](#).

USER GUIDE

UG019 | WE KiCad Library

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

The first screenshot shows the search results for part number 7488960245. The search bar at the top contains '7488960245'. The results show 'WE-MCA Antennas' with a list of download options: PDF, ZIP, INTLIB, A3DCOMP, ZIP, ZIP, LBR, IGS, ZIP, S1P, STP, and a red 'RE' icon. The second screenshot shows the search results for the product series 'WE-MCA'. The search bar contains 'WE-MCA'. The results show 'WE-MCA Antennas' with a breadcrumb trail: 'Passive Components > Signal & Communications > Antennas > WE-MCA Antennas'.

Figure 2: Search part number or series.

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

The screenshot shows the 'WE-MCA Antennas' product series page. On the left, there is a sidebar with navigation links: 'Overview All product units', 'Product unit Passive Components', 'Product group Signal & Communications', and 'Product family WE-MCA Antennas'. The main content area displays a table of products. The table has columns: 'Order Code', 'Data-sheet', 'Simulation', 'Downloads', 'Status', 'Frequency Range', and 'G_{peak} (dBi)'. The 'Downloads' column shows a dropdown menu with 10 files. The 'KiCad' file is highlighted in red. The table lists 10 products, each with a 'SPEC' link and a 'RE' icon. To the right of the table, there are sections for 'EDA models: Components', 'CAD files', and 'Electric models', each with a list of files and their sizes.

Order Code	Data-sheet	Simulation	Downloads	Status	Frequency Range	G _{peak} (dBi)
7488910043	SPEC	RE	10 FILES	Active	423-443 MHz	-4
7488918022	SPEC	RE				
7488910092	SPEC	RE				
7488910915	SPEC	RE				
7488920157	SPEC	RE				
7488915724	SPEC	RE				
7488912455	SPEC	RE				
7488920245	SPEC	RE				
7488960245	SPEC	RE				
7488922455	SPEC	RE				
74889102450	SPEC	RE				
74889302450	SPEC	RE				

Figure 3: Download KiCad libraries on Würth Elektronik Homepage.

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.**

USER GUIDE

UG019 | WE KiCad Library

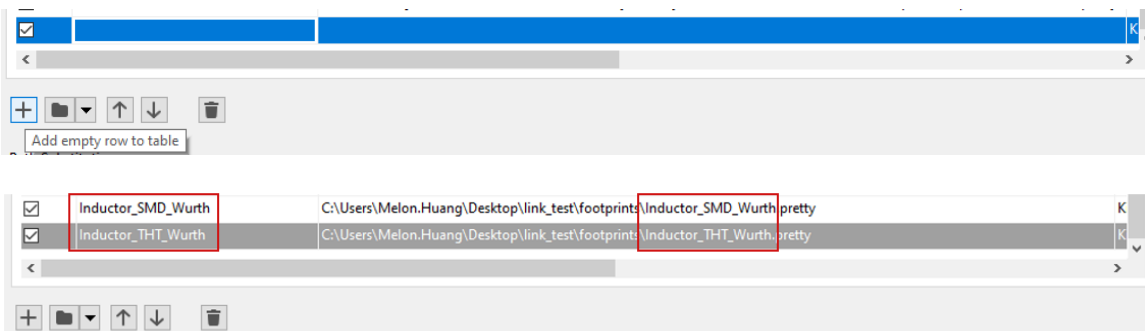
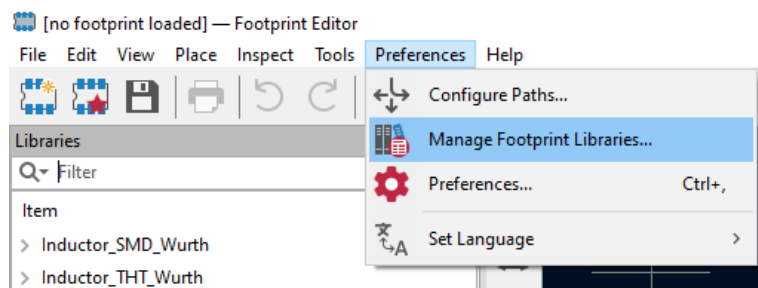
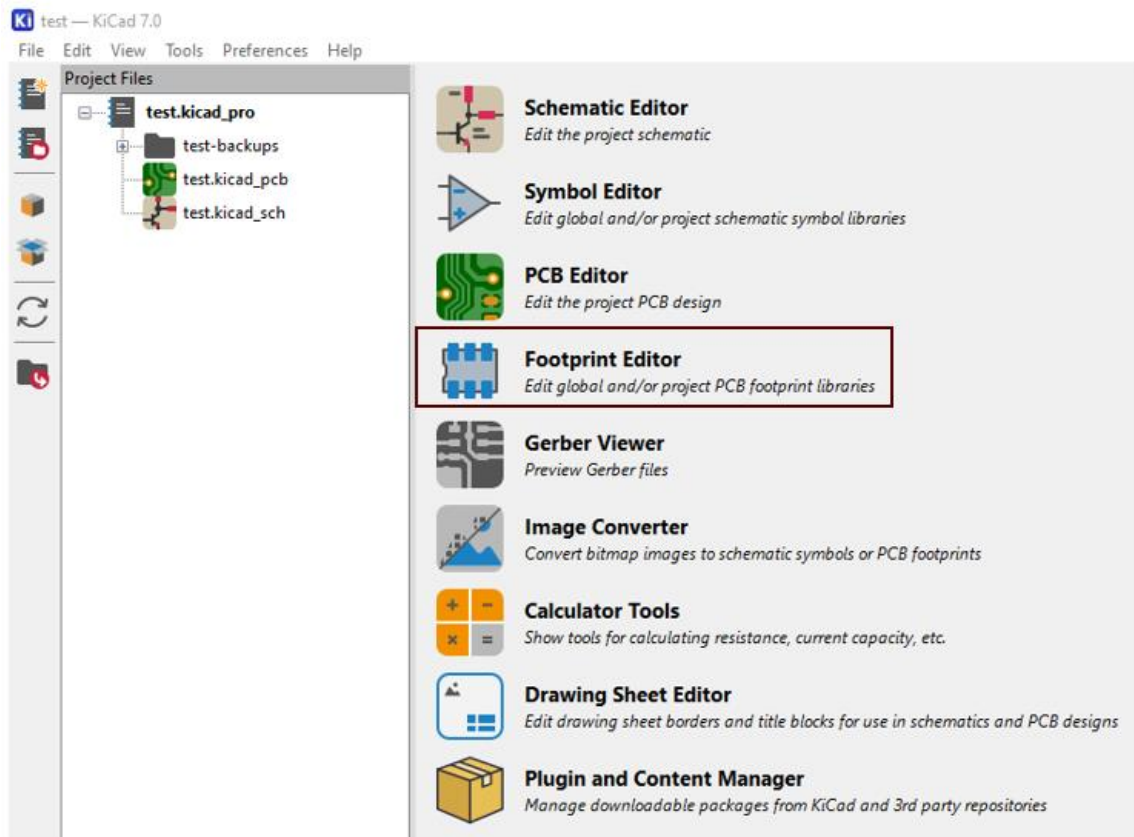


Figure 4: Importing the footprint.

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.**

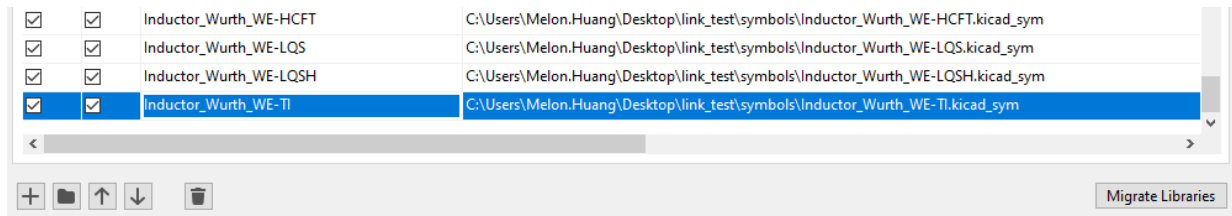


Figure 5: Importing the symbol.

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:

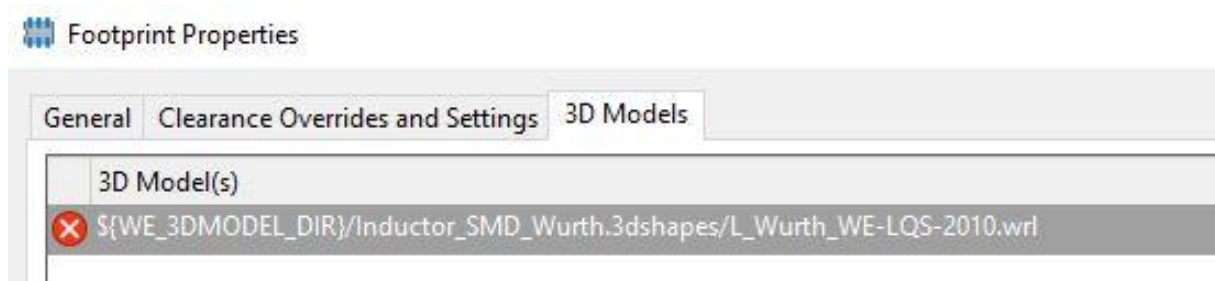


Figure 6: The 3D modes window when the model is missing.

Click ‘Configure Paths’ in Footprint Editor.

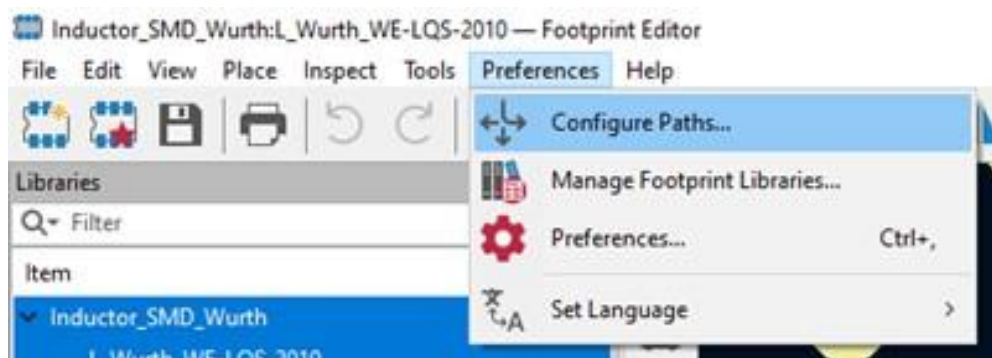


Figure 7: Open Configure Paths.

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.

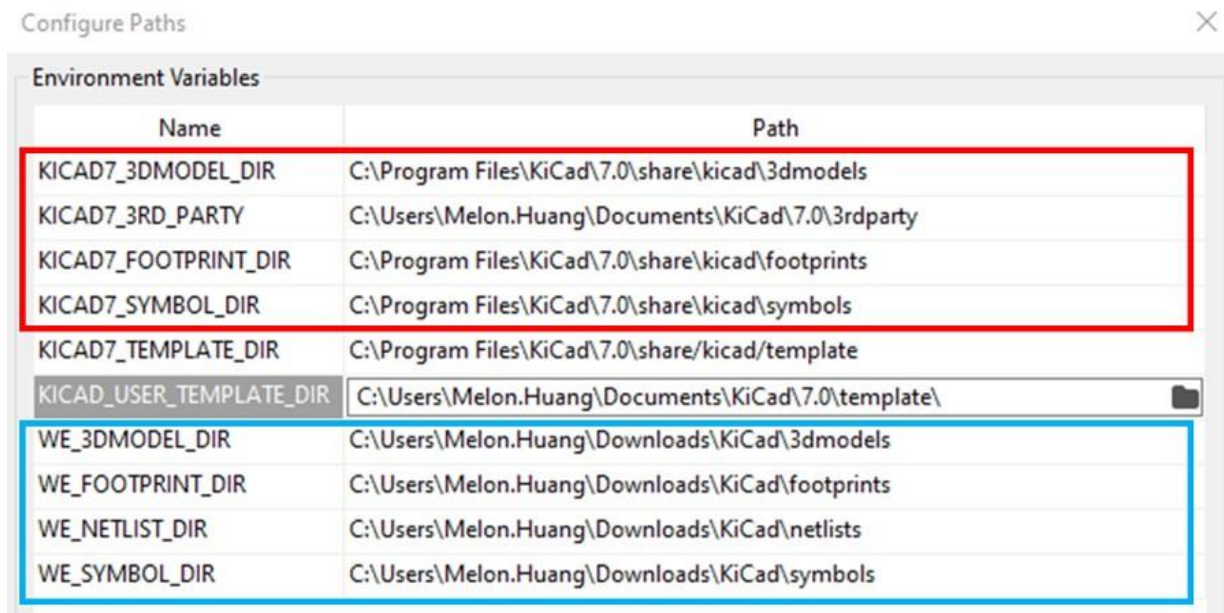


Figure 8: Add new variable paths.

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

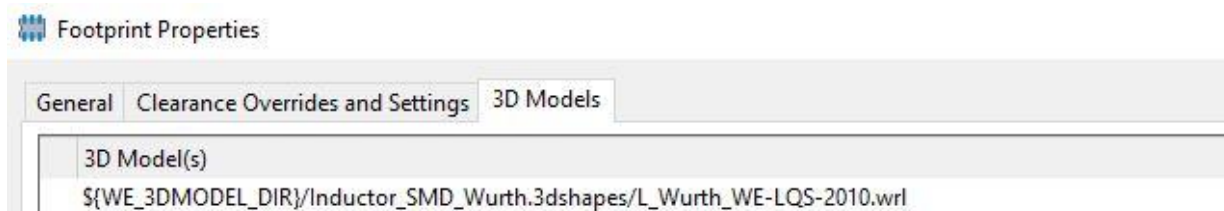


Figure 9: A correctly linked 3D model.

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 “Würth” in Footprint Editor filter, the results are as below:

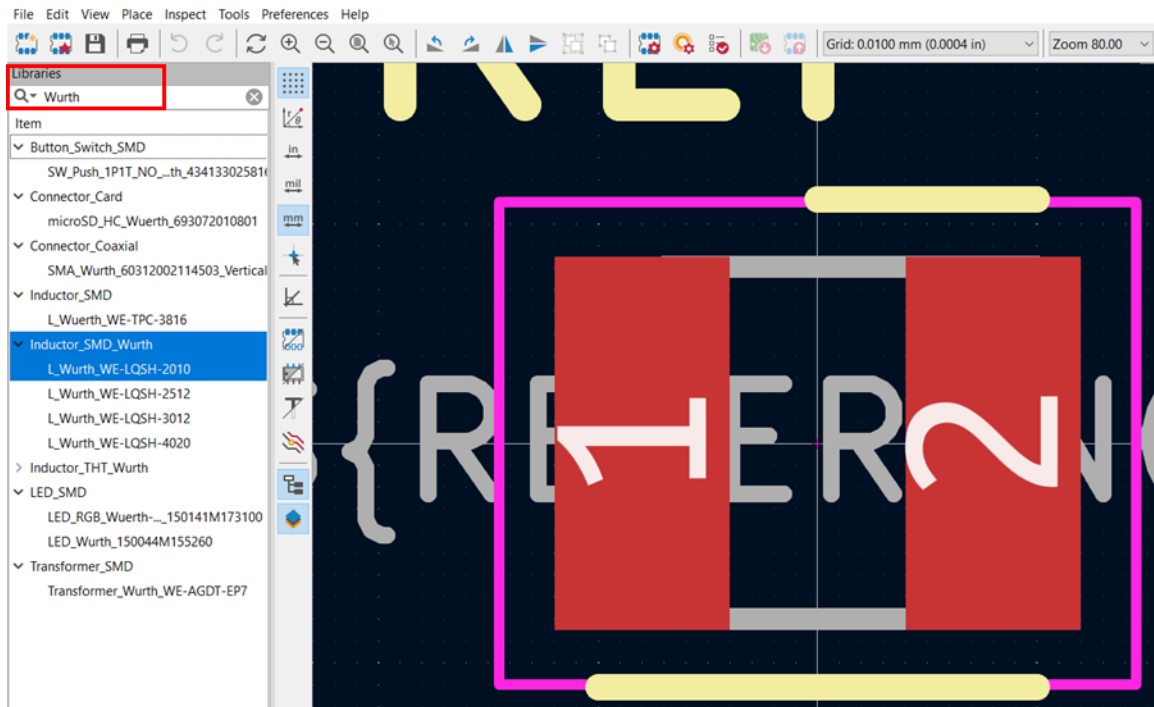


Figure 10: Example of a component in the KiCad built-in library.

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/WurthElektronik/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.


Let's get started!

Add a repository to GitHub Desktop to start collaborating

 Create a tutorial repository...

 Clone a repository from the Internet...

 Create a New Repository on your hard drive...

 Add an Existing Repository from your hard drive...

Clone a repository

GitHub.com

GitHub Enterprise

URL

Repository URL or GitHub username and repository
(hubot/cool-repo)

Local path
 Choose...

Clone Cancel

Cloning KiCad-Library

Receiving objects: 7% (13/177), 252.00 KiB | 72.00 KiB/s






File folder (4)			
	.git	6/27/2024 4:04 PM	File folder
	3dmodels	6/6/2024 3:32 PM	File folder
	footprints	6/6/2024 3:32 PM	File folder
	symbols	6/6/2024 3:32 PM	File folder
MD File (1)			
	README.md	6/27/2024 4:04 PM	MD File 1 KB

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.

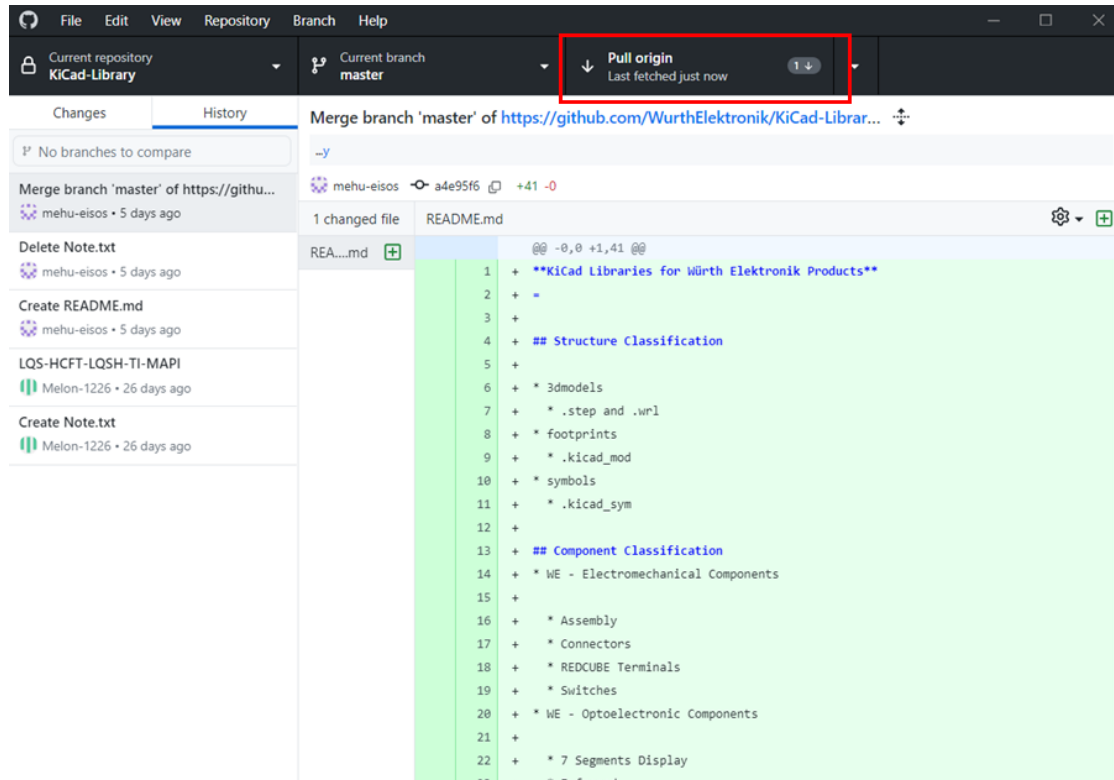


Figure 12: Synchronize from GitHub.

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

KiCad-Library > footprints > Inductor_SMD_Wurth.pretty				Search Inductor_SMD_Wurth.pretty
Name	Date modified	Type	Size	
<input type="checkbox"/> L_Wurth_WE-LQS-2010.kicad_mod	4/10/2024 2:08 PM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-2512.kicad_mod	11/22/2023 9:23 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-3015.kicad_mod	5/8/2023 9:29 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-4012.kicad_mod	4/28/2023 1:46 PM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-4018.kicad_mod	4/28/2023 10:26 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-4025.kicad_mod	11/22/2023 9:38 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-5020.kicad_mod	11/22/2023 9:36 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-5040.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-6028.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-6045.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-8040.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQS-8065.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQSH-2010.kicad_mod	11/22/2023 9:38 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQSH-2512.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQSH-3012.kicad_mod	4/28/2023 10:29 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-LQSH-4020.kicad_mod	4/28/2023 10:29 AM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-MAPI-1610.kicad_mod	10/19/2023 5:33 PM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-MAPI-2010.kicad_mod	10/19/2023 5:07 PM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-MAPI-2506.kicad_mod	10/19/2023 5:09 PM	KICAD_MOD File	3 KB	
<input type="checkbox"/> L_Wurth_WE-MAPI-2508.kicad_mod	10/19/2023 5:10 PM	KICAD_MOD File	3 KB	

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 APPLICATION.

USEFUL LINKS



Application Notes

www.we-online.com/apnotes



REDEXPERT Design Platform

www.we-online.com/redexpert



Toolbox

www.we-online.com/toolbox



Product Catalog

www.we-online.com/products

CONTACT INFORMATION



apnotes@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