

## 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 pertain to KiCad versions 7.0 or higher.

### 2. Install from website

Note: KiCad models on homepage are always the latest.

#### 2.1 Download from Würth homepage

Browse [www.we-online.com](http://www.we-online.com) >> Components >> Find the product unit according to the catalog.

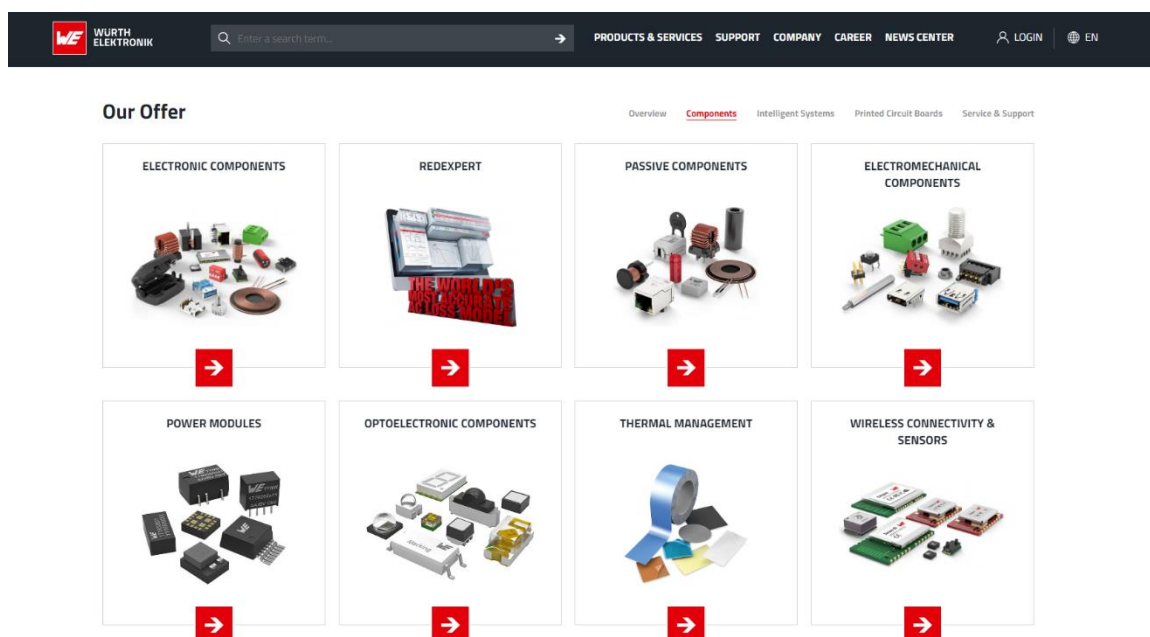


Figure 1: Würth Elektronik Homepage

# USER GUIDE

## UGXXX | WE KiCad Library

Or search for the part number or series you are interested in. For example, search part number 744910016 or series WE-CAIR, this search result shows 744910016 has an KiCad zipped library.

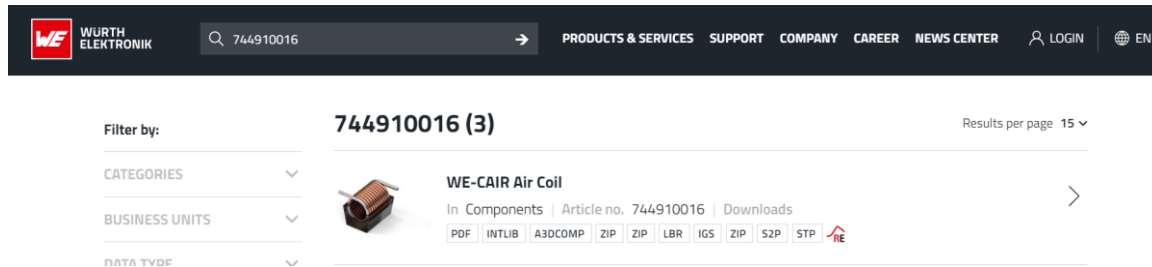


Figure 2: Search part number or series

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

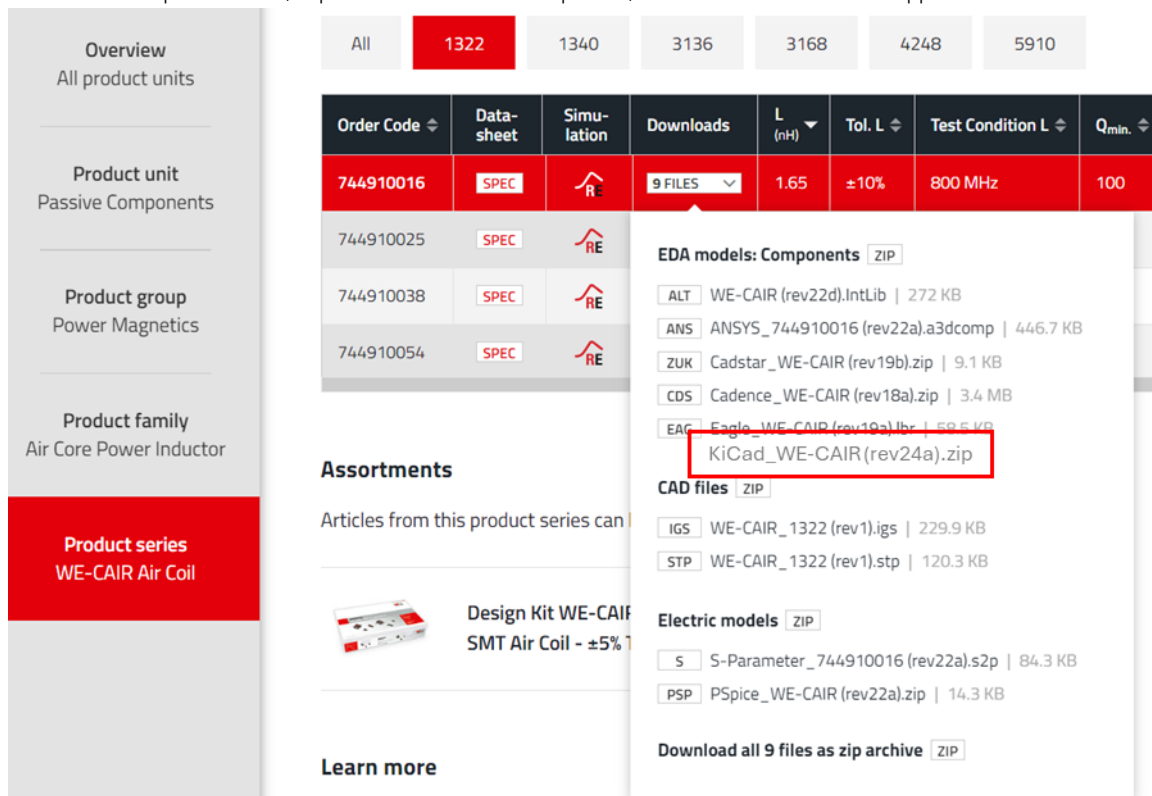


Figure 3: Download library

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

3dmodels	1/8/2024 10:25 AM	File folder
footprints	1/8/2024 10:25 AM	File folder
symbols	4/9/2024 3:47 PM	File folder

Figure 4.1: Overview

The following folders can be seen inside the footprints folder.

Inductor_SMD_Wurth.pretty	4/9/2024 3:44 PM	File folder
Inductor_THT_Wurth.pretty	1/8/2024 10:25 AM	File folder

Figure 4.2: footprints folder

# USER GUIDE

UGXXX | WE KiCad Library

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

 Inductor_SMD_Wurth.3dshapes	1/8/2024 10:25 AM	File folder
 Inductor_THT_Wurth.3dshapes	1/8/2024 10:25 AM	File folder

Figure 4.3: 3dshapes folder

The following folders can be seen inside the symbols folder.





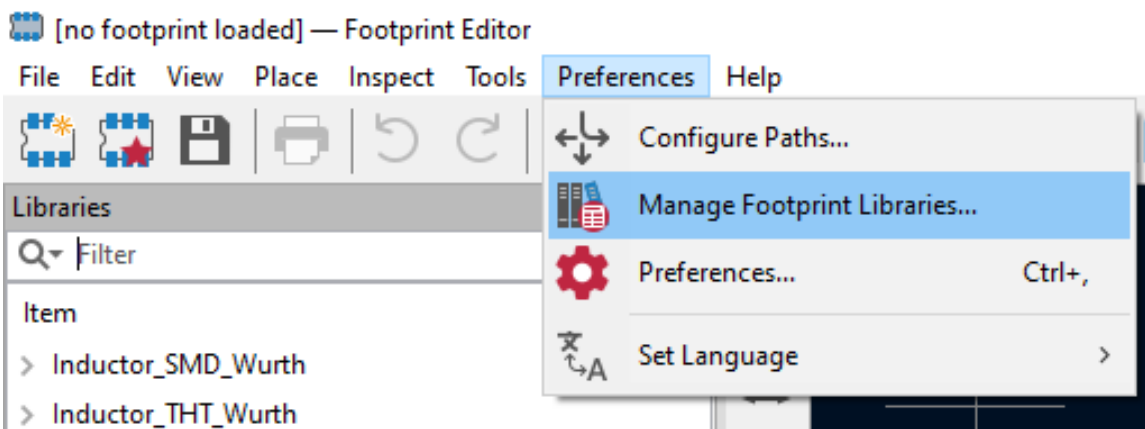
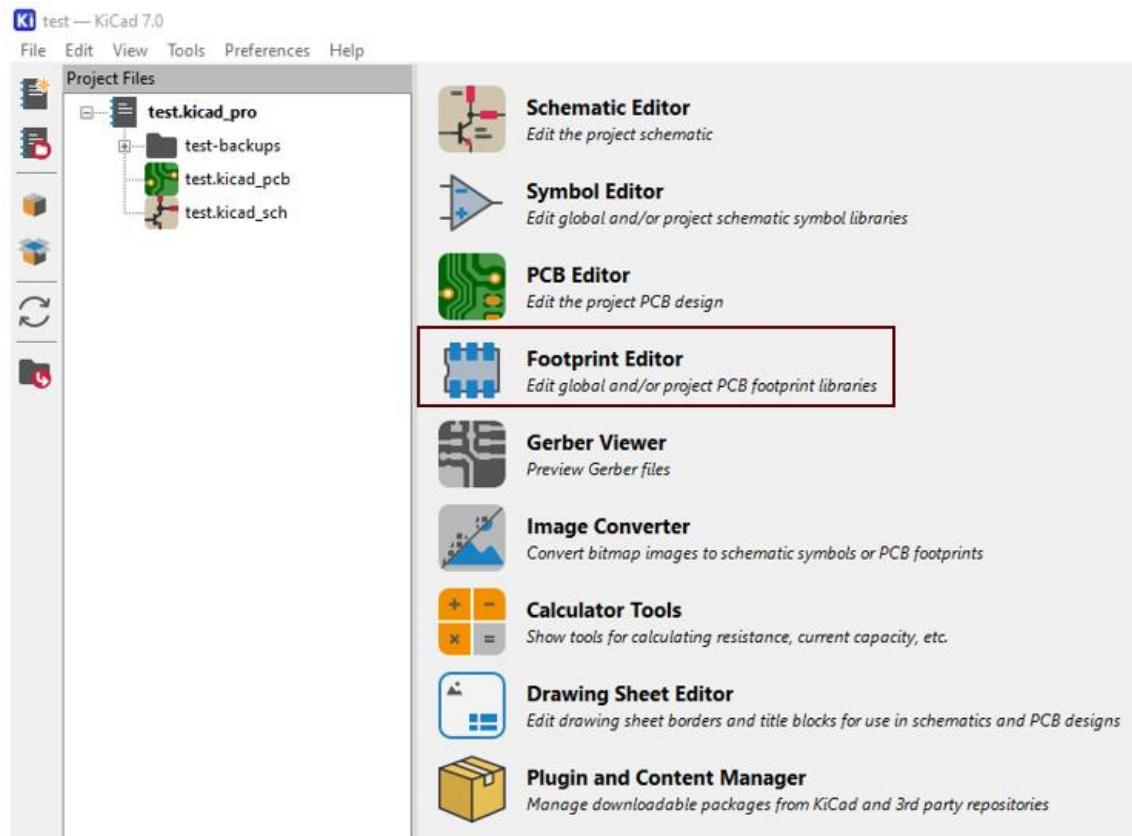
 Inductor_Wurth_WE-HCFT.kicad_sym	4/9/2024 3:47 PM	KICAD_SYM File	116 KB
 Inductor_Wurth_WE-LQS.kicad_sym	4/9/2024 3:46 PM	KICAD_SYM File	916 KB
 Inductor_Wurth_WE-LQSH.kicad_sym	11/16/2022 4:57 PM	KICAD_SYM File	139 KB
 Inductor_Wurth_WE-TI.kicad_sym	4/9/2024 3:45 PM	KICAD_SYM File	642 KB

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.**



# USER GUIDE

UGXXX | WE KiCad Library

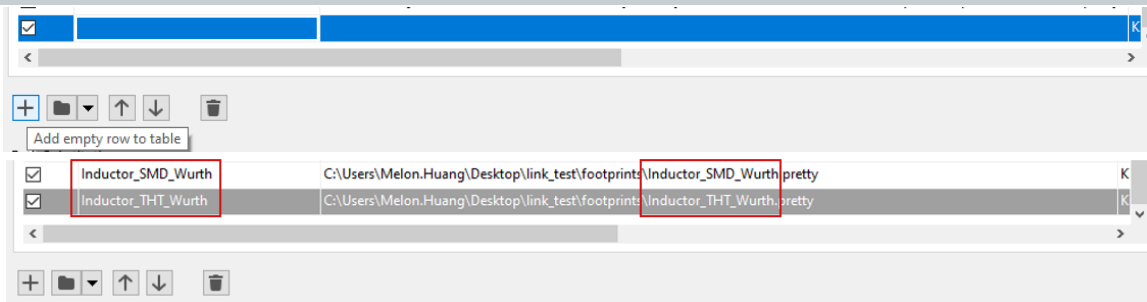


Figure 5: footprint import

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

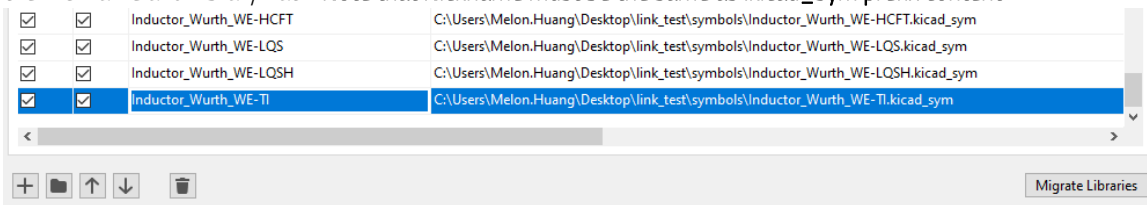


Figure 6: symbol import

Now the symbols and footprints are linked. But 3D Models in the footprint are still missing. You can link 3D models via two ways:

## Footprint Properties

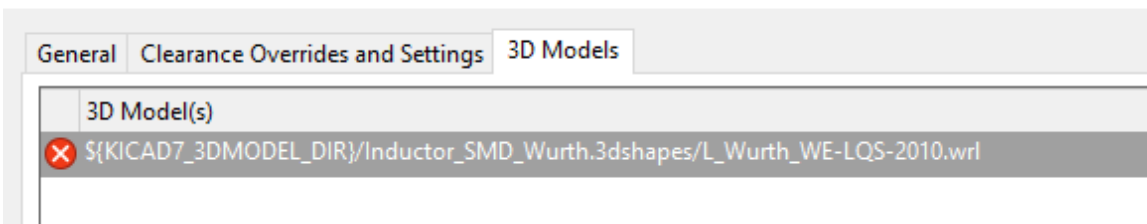


Figure 7: missing 3D model

Firstly, you can move .3dshapes folder that you download to C:\Program Files\KiCad\7.0\share\kicad\3dmodels, which is the KiCad default directory.

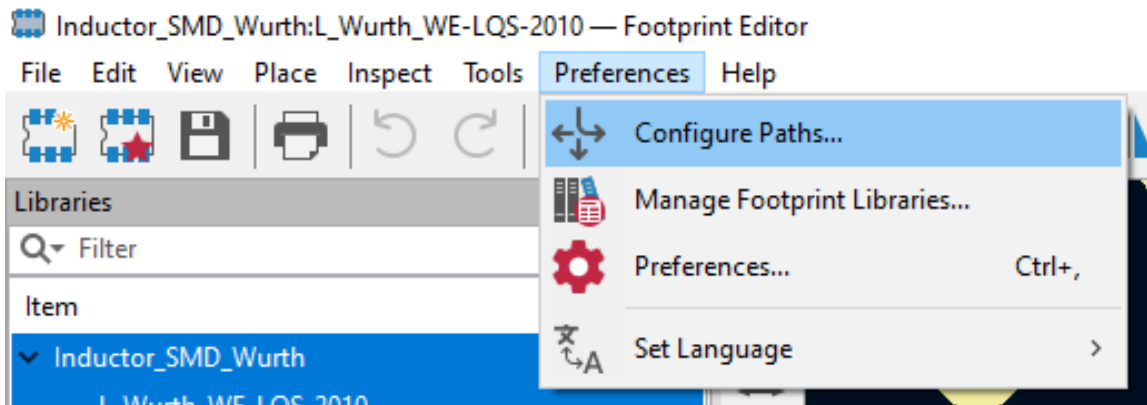
# USER GUIDE

UGXXX | WE KiCad Library

SYS (C:) > Program Files > KiCad > 7.0 > share > kicad > 3dmodels >			
	Name	Date modified	Type
	Battery.3dshapes	4/25/2023 5:41 PM	File folder
	Button_Switch_SMD.3dshapes	4/25/2023 5:41 PM	File folder
	Button_Switch_THT.3dshapes	4/25/2023 5:41 PM	File folder
	Buzzer_Beeper.3dshapes	4/25/2023 5:41 PM	File folder
	Capacitor_SMD.3dshapes	4/25/2023 5:41 PM	File folder
	Capacitor_Tantalum_SMD.3dshapes	4/25/2023 5:41 PM	File folder
	Capacitor_THT.3dshapes	4/25/2023 5:41 PM	File folder
	Connector.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_AMASS.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_BarrelJack.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_Card.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_Coaxial.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_Dsub.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_FFC-FPC.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_HDMI.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_IDC.3dshapes	4/25/2023 5:41 PM	File folder
	Connector_JST.3dshapes	4/25/2023 5:41 PM	File folder

Figure 8.1: 3D models import way1

Secondly, you can directly modify the default 3dmodel directory. Click Configure Paths in Footprint Editor, change the KICAD7\_3DMODEL\_DIR path to the location where you put the files you previously download.



# USER GUIDE

UGXXX | WE KiCad Library

Configure Paths		✕
Environment Variables		
Name	Path	
KICAD7_3DMODEL_DIR	C:\Program Files\KiCad\7.0\share\kicad\3dmodels	default
KICAD7_3RD_PARTY	C:\Users\Melon.Huang\Documents\KiCad\7.0\3rdparty	
KICAD7_FOOTPRINT_DIR	D:\Melon\Project\KiCad\KiCad_Libraries\00_Released\footprints	
KICAD7_SYMBOL_DIR	D:\Melon\Project\KiCad\KiCad_Libraries\00_Released\symbols	
KICAD7_TEMPLATE_DIR	C:\Program Files\KiCad\7.0\share\kicad\template	
KICAD_USER_TEMPLATE_DIR	C:\Users\Melon.Huang\Documents\KiCad\7.0\template\	

Configure Paths		✕
Environment Variables		
Name	Path	
KICAD7_3DMODEL_DIR	C:\Users\Melon.Huang\Desktop\link_test\3dmodels	modified
KICAD7_3RD_PARTY	C:\Users\Melon.Huang\Documents\KiCad\7.0\3rdparty	
KICAD7_FOOTPRINT_DIR	D:\Melon\Project\KiCad\KiCad_Libraries\00_Released\footprints	
KICAD7_SYMBOL_DIR	D:\Melon\Project\KiCad\KiCad_Libraries\00_Released\symbols	
KICAD7_TEMPLATE_DIR	C:\Program Files\KiCad\7.0\share\kicad\template	
KICAD_USER_TEMPLATE_DIR	C:\Users\Melon.Huang\Documents\KiCad\7.0\template\	

Figure 8.2: 3D models import way2

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



## Footprint Properties

General	Clearance Overrides and Settings	3D Models
3D Model(s)		
{KICAD7_3DMODEL_DIR}/Inductor_SMD_Wurth.3dshapes/L_Wurth_WE-LQS-2010.wrl		

Figure 9: linked 3D model

## 3. Install from KiCad built-in library

Note: KiCad built-in library is reviewed and updated by KiCad, which takes some time. So the library may not be the latest.

### 3.1 KiCad built-in library

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

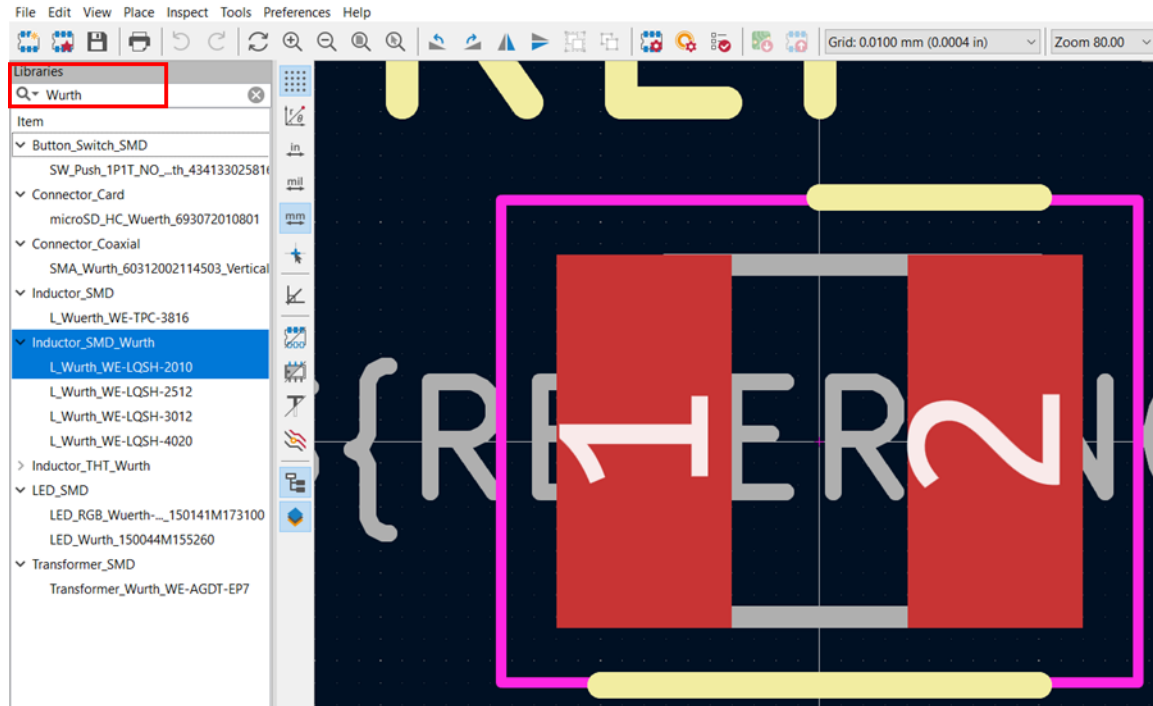


Figure 10: KiCad built-in library

You can use built-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.

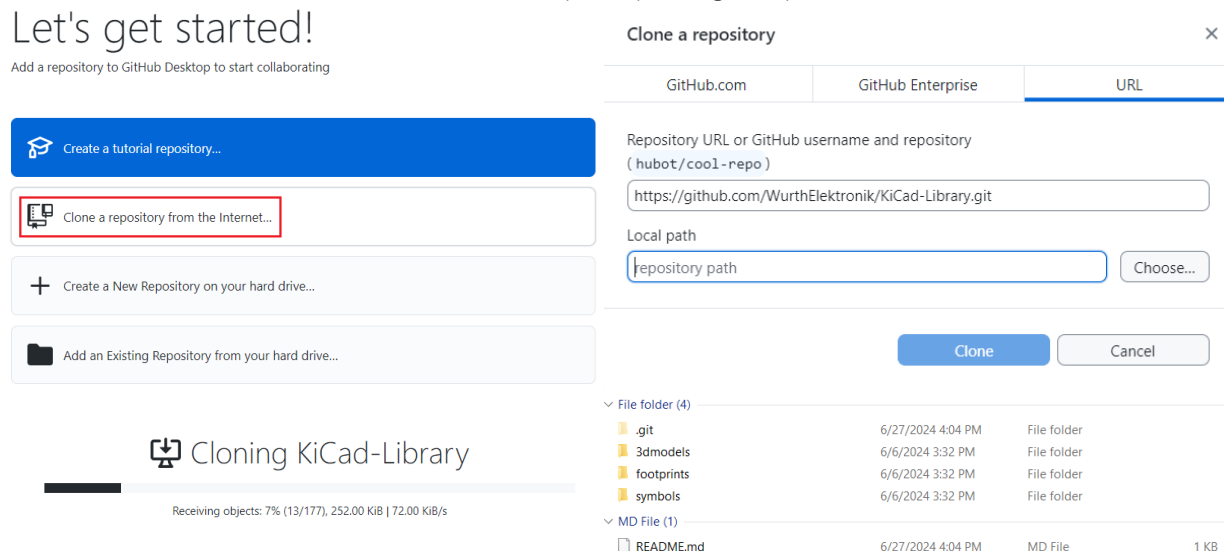


Figure 11: Clone the library

### 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.



# USER GUIDE

UGXXX | WE KiCad Library

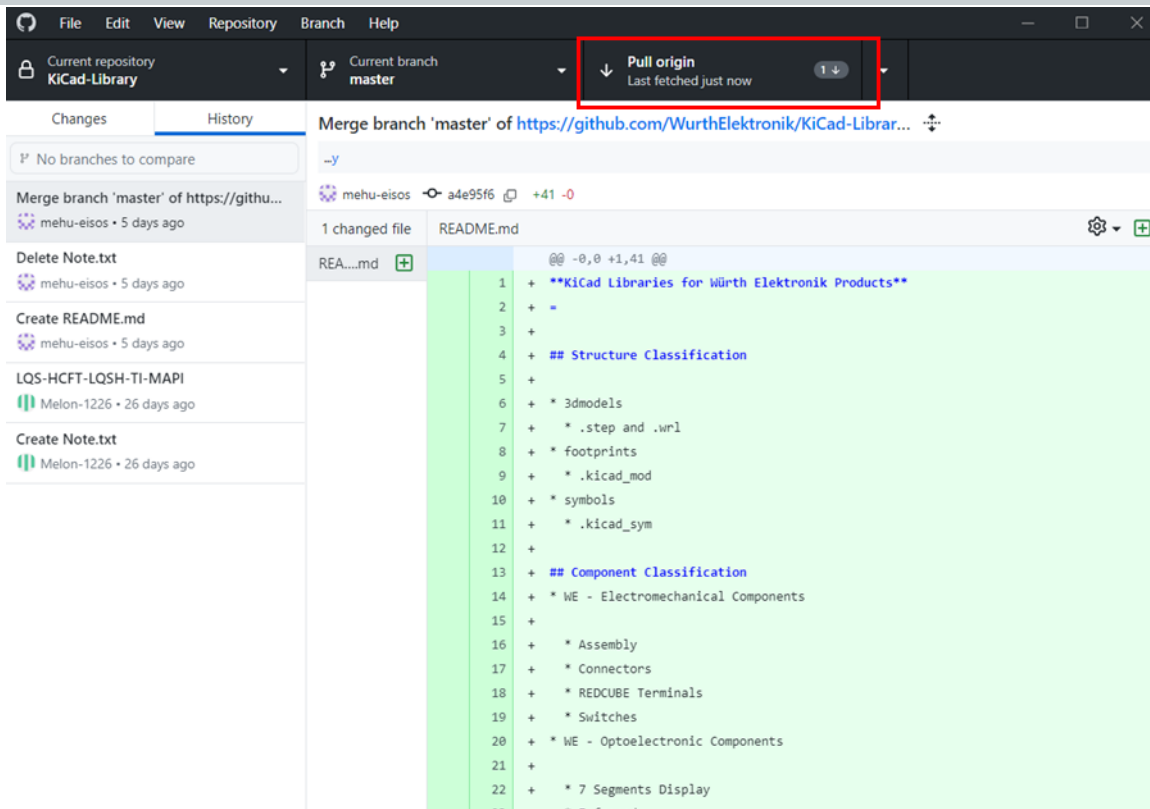


Figure 12: Synchronize from GitHub

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			
L_Wurth_WE-LQS-2010.kicad_mod	4/10/2024 2:08 PM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-2512.kicad_mod	11/22/2023 9:23 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-3015.kicad_mod	5/8/2023 9:29 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-4012.kicad_mod	4/28/2023 1:46 PM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-4018.kicad_mod	4/28/2023 10:26 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-4025.kicad_mod	11/22/2023 9:38 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-5020.kicad_mod	11/22/2023 9:36 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-5040.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-6028.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-6045.kicad_mod	4/28/2023 10:27 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-8040.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQS-8065.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQSH-2010.kicad_mod	11/22/2023 9:38 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQSH-2512.kicad_mod	4/28/2023 10:28 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQSH-3012.kicad_mod	4/28/2023 10:29 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-LQSH-4020.kicad_mod	4/28/2023 10:29 AM	KICAD_MOD File	3 KB			
L_Wurth_WE-MAPI-1610.kicad_mod	10/19/2023 5:33 PM	KICAD_MOD File	3 KB			
L_Wurth_WE-MAPI-2010.kicad_mod	10/19/2023 5:07 PM	KICAD_MOD File	3 KB			
L_Wurth_WE-MAPI-2506.kicad_mod	10/19/2023 5:09 PM	KICAD_MOD File	3 KB			
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](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**



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)

