USER GUIDE

UG002 | Using the LTspice Model Libraries

Toby Kangas

1. INTRODUCTION

For any new users of a tool, the basic functionalities must be learned before the tool can be put to use. For LTspice, knowing how to correctly install and find the models is crucial. Although most of our models are pre-installed in the LTspice built-in libraries, sometimes it is necessary to manually install LTspice models in the library folders. Our LTspice models consist of a symbol file (*.asy) and a netlist file (*.lib or *.sub). These files usually must be saved in different locations.

Our models are available on these platforms: Würth Elektronik Homepage, LTspice built-in libraries and GitHub repository.

Note: The following instructions refer to LTspice versions 17.1 or greater. Previous versions of LTspice have some significant differences, especially the file locations of the models.

2. INSTALLING THE MODELS

2.1 Downloads from Würth Homepage

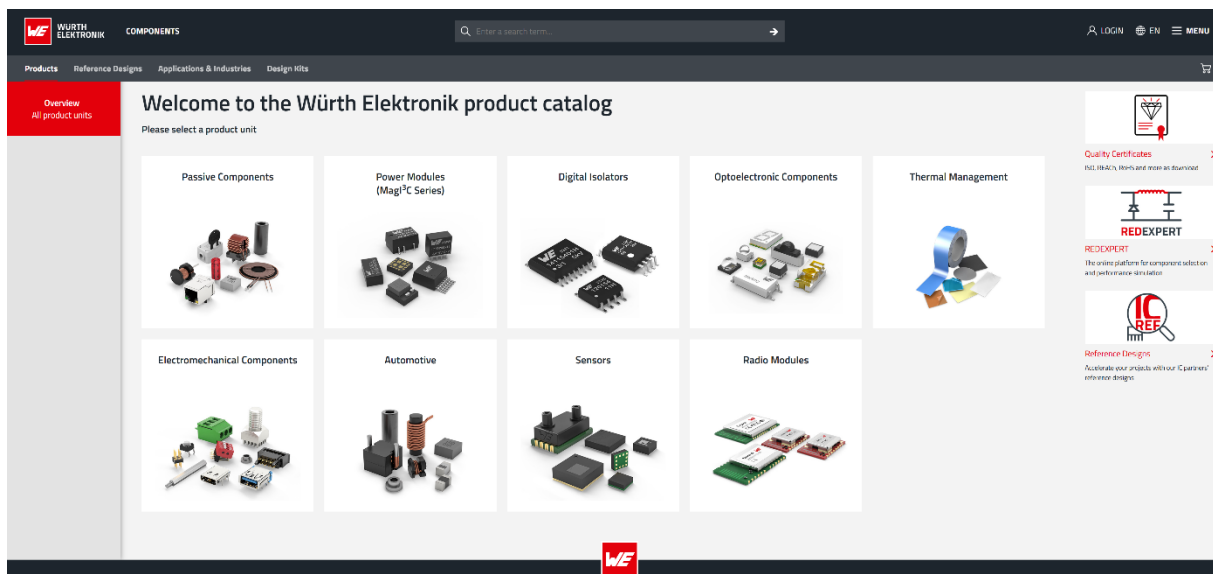Visit the WE product portfolio and navigate to the product you are interested in.

Figure 1: Würth Elektronik Homepage.

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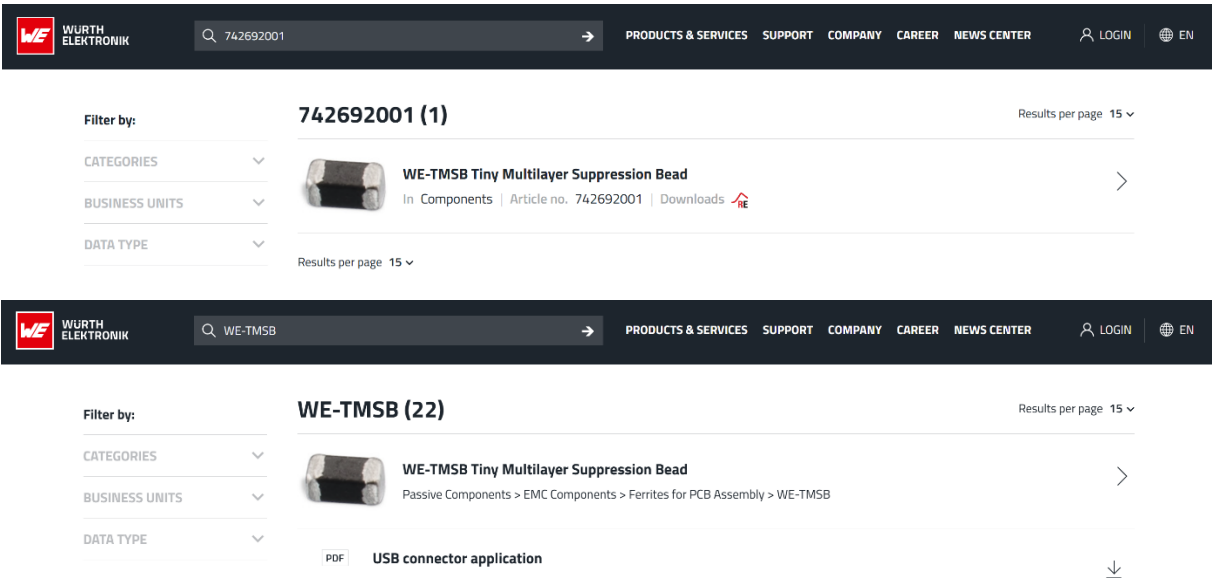

Figure 2: Search part number or series.

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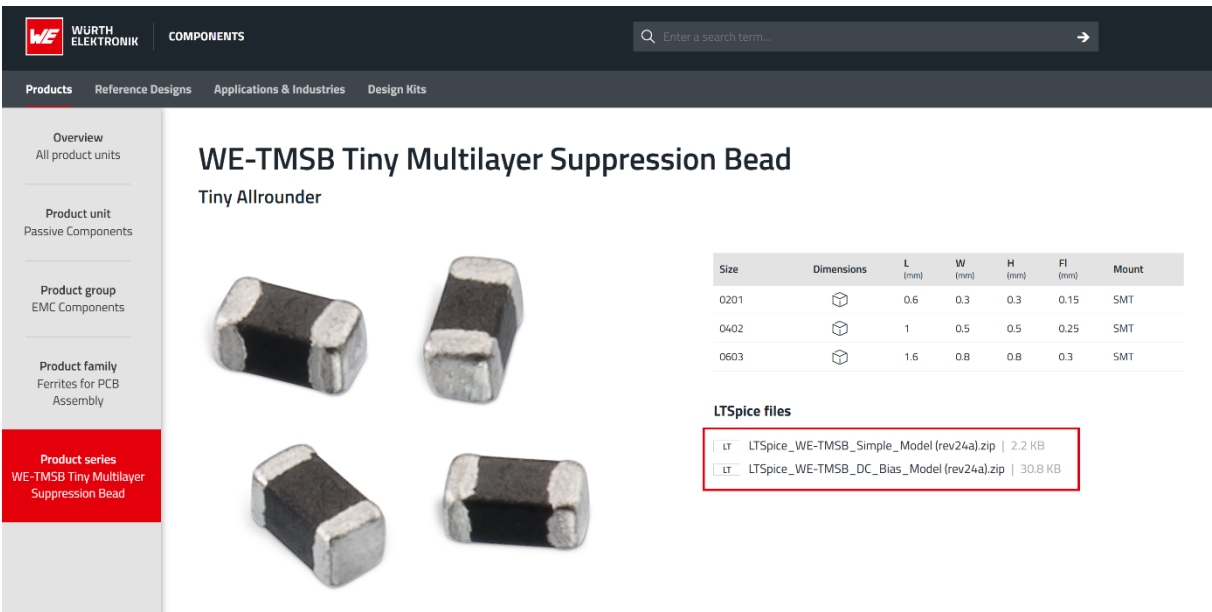

Figure 3: Download LTspice libraries on Würth Homepage.

2.2 To Install the Model

Save the *.lib or *.sub netlist files directly in the user folder ...\AppData\Local\LTspiceXVII\lib\sub.

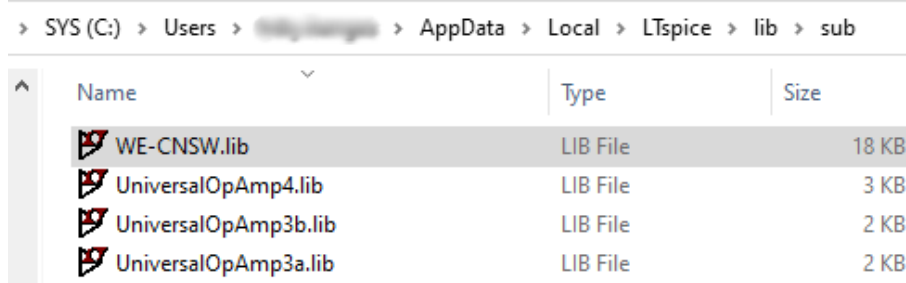*Figure 4: Save netlist files in \sub folder.*

Save the *.asy files in the user folder ...\AppData\Local\LTspiceXVII\lib\sym or a subfolder thereof. If LTspice is open, it must be closed and re-opened to view the new models in the component directory.
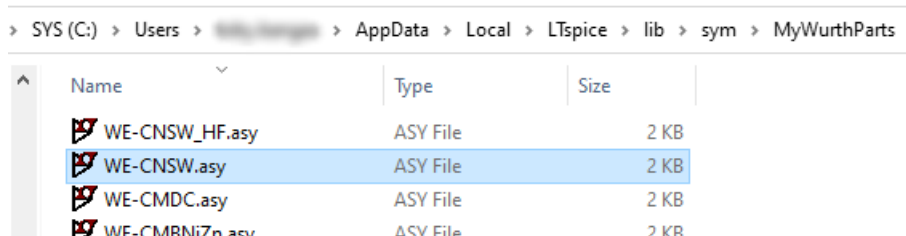

Figure 5: Save symbol files in \sym folder.

Add the symbol to your schematic by clicking the Component symbol from the toolbar (or type 'F2').

Figure 6: Insert Component.

Navigate to the location where the symbol file was saved and select the file.
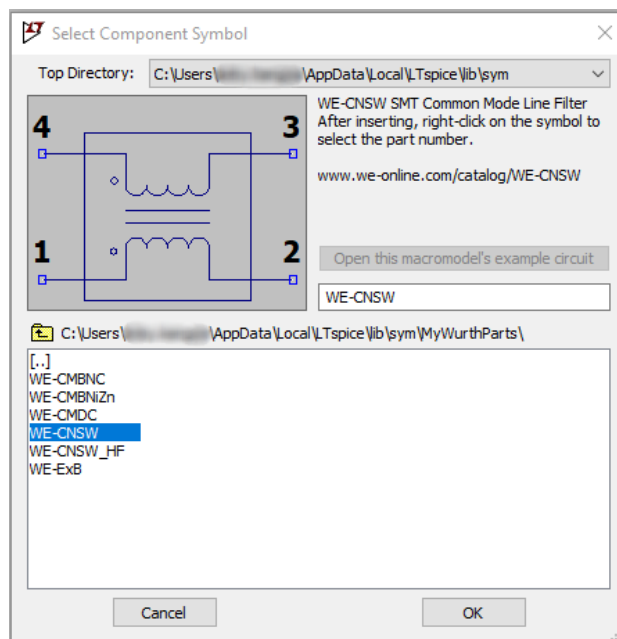

Figure 7: Select Component.

After placing the symbol in the schematic, right-click on the symbol to open the Component Attribute Editor. Choose the part desired number by double-clicking or triple-clicking on the Spice model value. Note that some parts have a symbol which is unique to that part number, and the part number cannot be edited.
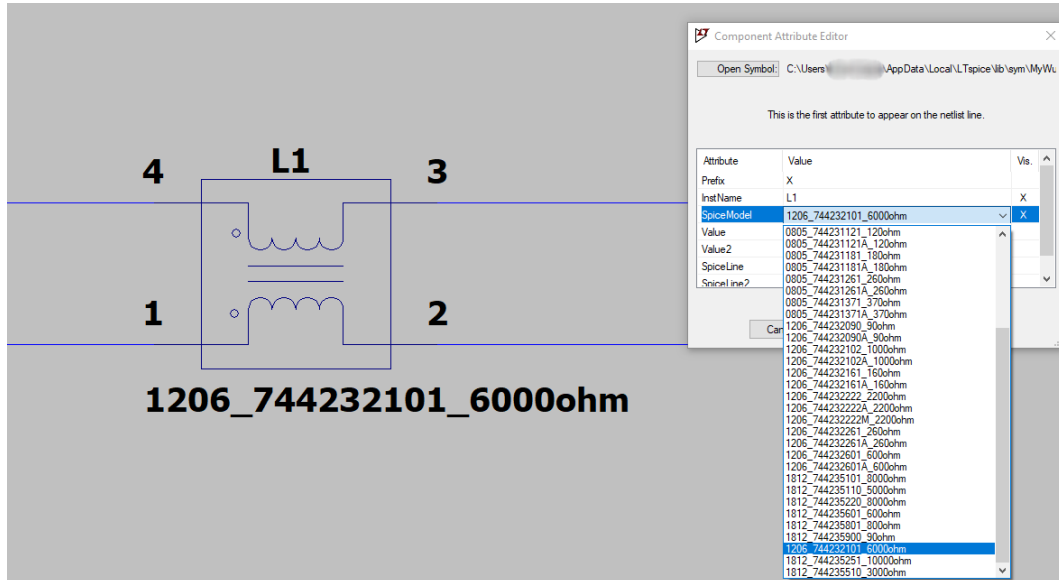

Figure 8: Choose part number.

2.3 Alternate Option 1

If the LTspice model is only being used in a certain schematic, the *.lib or *.sub netlist file and the *.asy symbol file can be saved in the same local folder as the schematic file (*.asc).
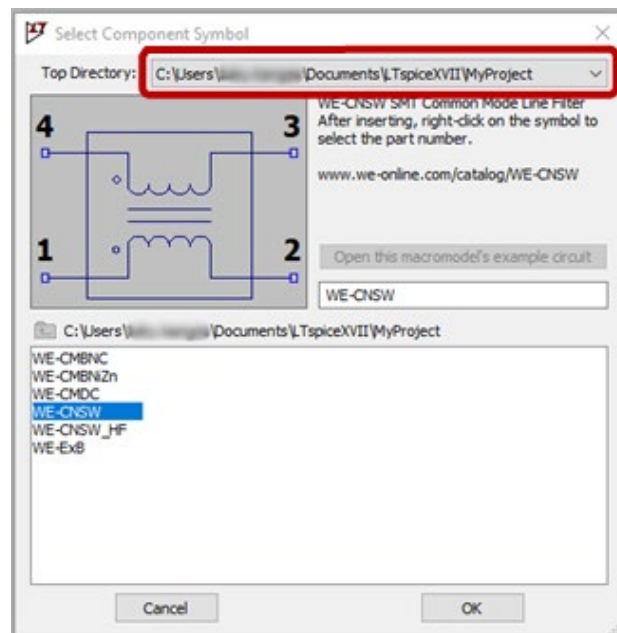

Figure 9: Model files in same folder as schematic.

2.4 Alternate Option 2

Library locations can also be defined directly in the Control Panel. (Tools >> Control Panel). Under the "Sym. & Lib. Search Paths" tab, define the location for symbols and libraries.
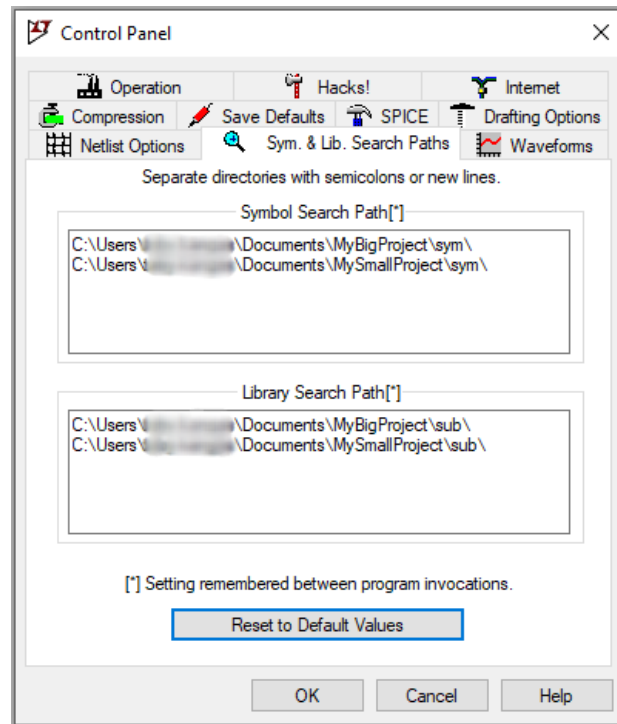

Figure 10: Define user library location.

These locations can be accessed from the Top Directory dropdown in the Insert Component window. Note that this method does not support subfolders of the defined location. Every folder must be explicitly defined.
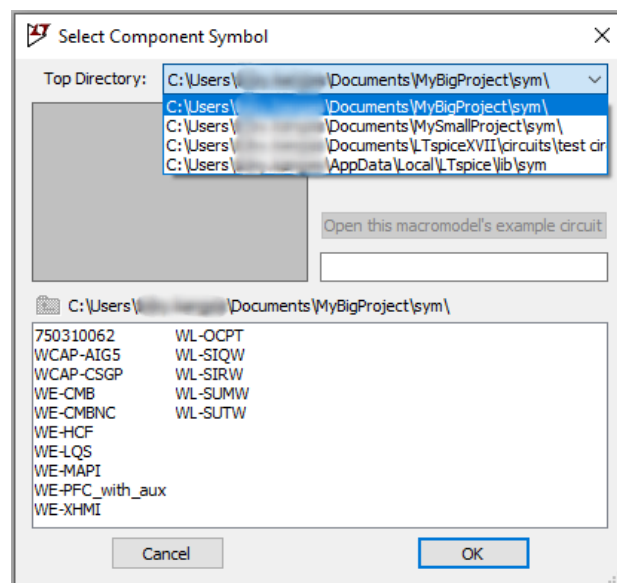

Figure 11: Select library directory.

3. BUILT-IN LIBRARIES

Did you know that the model may already be in LTspice? Most Würth Elektronik components can be found in the standard inductor, capacitor or ferrite bead libraries or in the Contributors folder.

3.1 Update Libraries

To update the libraries in LTspice to the most recent version, select Tools >> Update components. Note that library updates and software updates are now separate features in LTspice.
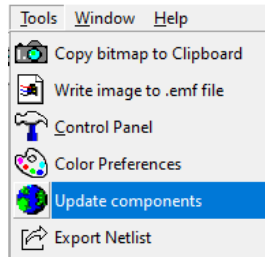

Figure 12: Update Components.

3.2 Standard Inductor, Capacitor or Ferrite Bead Libraries

Basic inductor, capacitor and ferrite bead simulation models are included in LTspice's standard libraries. To choose an inductor or capacitor model, insert the inductor or capacitor symbol from the toolbar (or type 'L' or 'C').
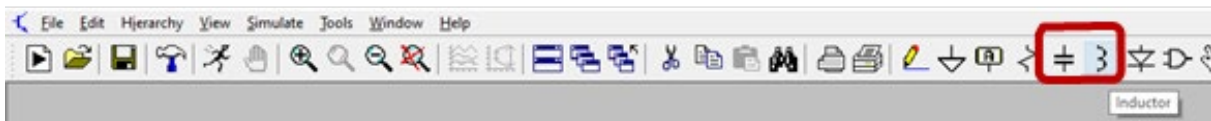

Figure 13: Insert Inductor or Capacitor.

For ferrite beads, click the Component symbol (or type 'F2') and select the FerriteBead component.
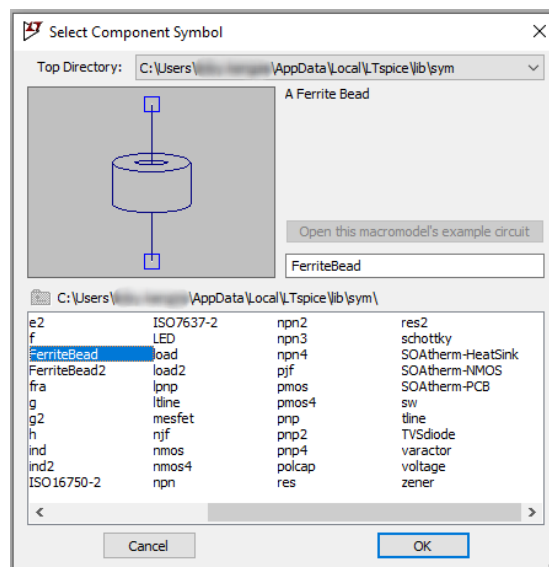

Figure 14: Insert Component.

Figure 15: Select ferrite bead.

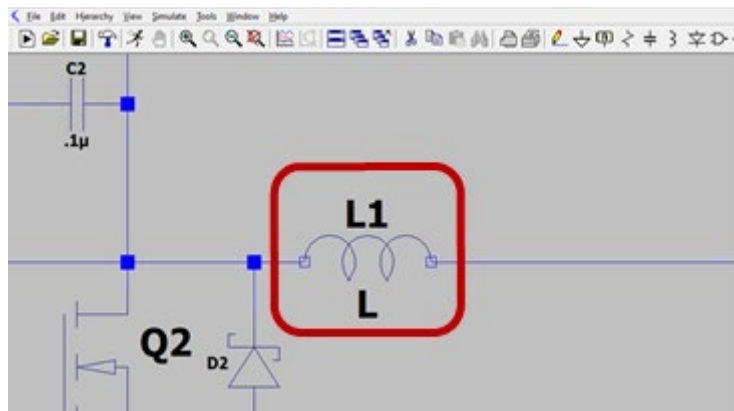Place the inductor, capacitor, or ferrite bead model in the schematic.

Figure 16: Place component.

Right-click on the model and click Select Inductor (or Select Capacitor or Select Ferrite Bead).
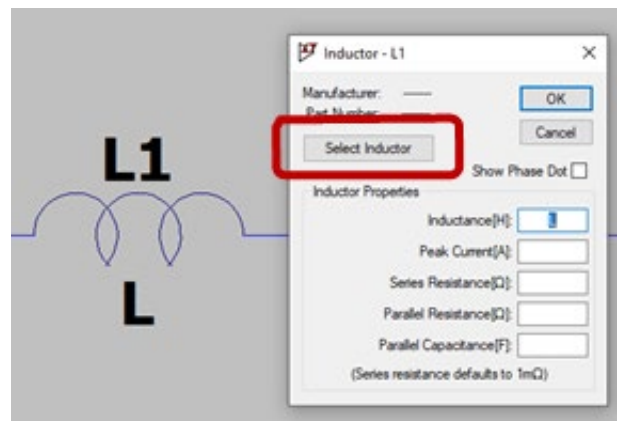

Figure 17: Select inductor.

Scroll or use the Part No. search to find the desired Würth Elektronik part number. (The "Find" button must be in focus if using the Enter key to search after typing the part number; otherwise the Enter key will select the highlighted part.) Parts can also be sorted by clicking on the column headings. Note that if changing a part selection, the list is already pre-filtered for similar parts (similar inductance or capacitance). Click the All button to remove this filter.
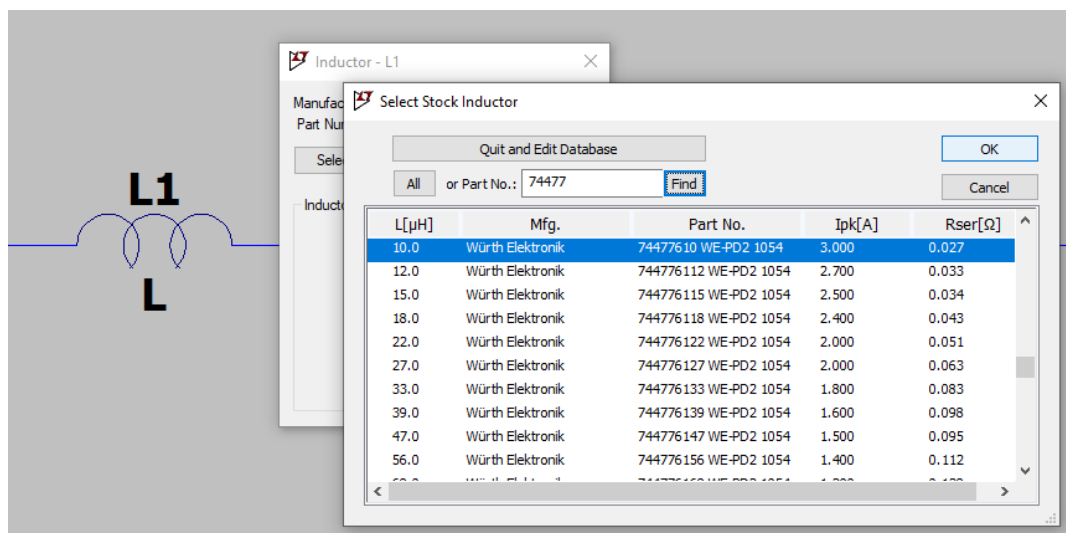

Figure 18: Select part number.

3.3 Contributor Directory

Products not found in the standard libraries may be found in the Contributors directory. To choose a model, click the Component symbol from the toolbar (or type 'F2').

Figure 19: Insert component.

Navigate to [Contrib] >> [Würth] >> and then to the respective product folder. Select the symbol for the desired series. Click OK.
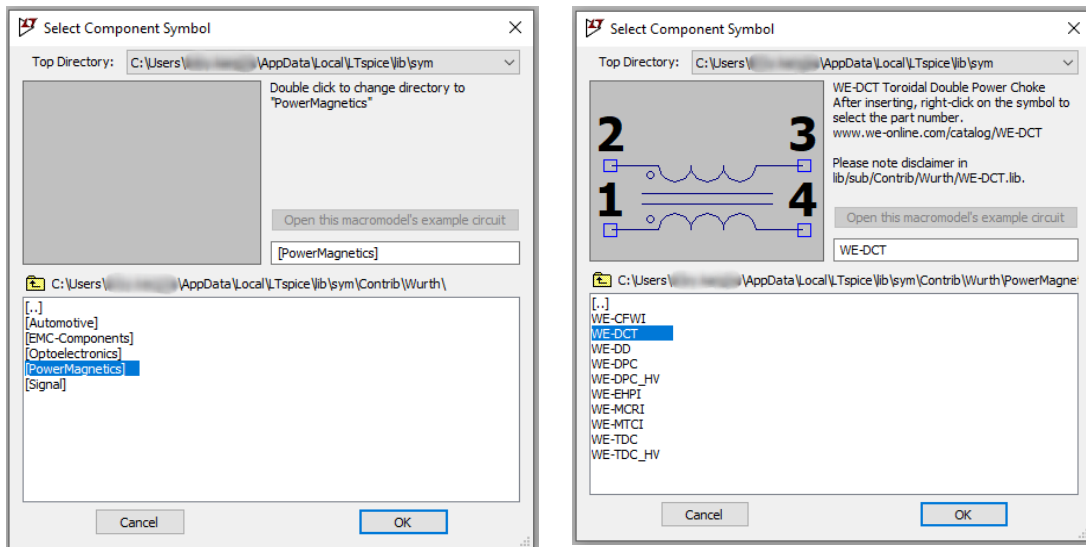

Figure 20: Navigate to component.

After placing the symbol in the schematic, the part number can be changed by right-clicking the symbol. Then double-click or triple-click the Spice model value in the Component Attribute Editor, select the desired part number, and click OK.
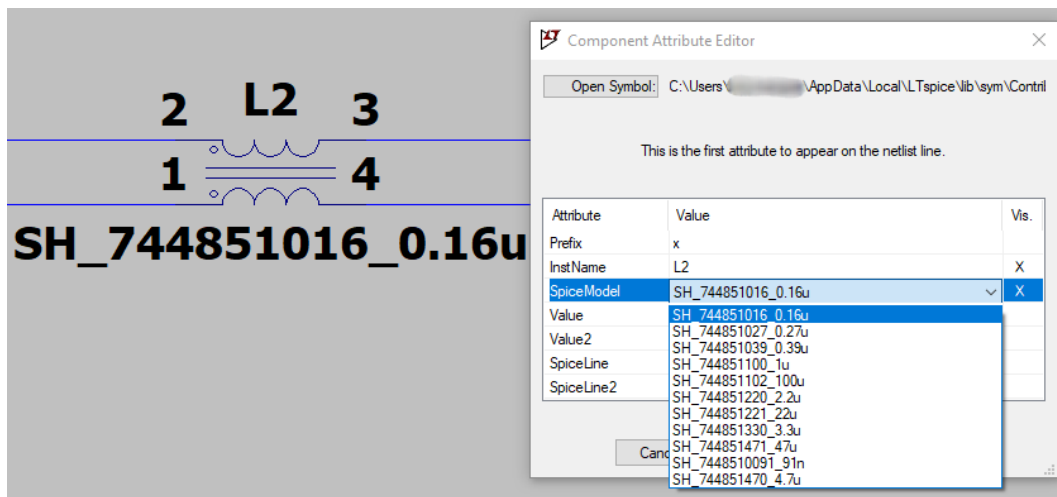

Figure 21: Choose part number.

4. INSTALL FROM GITHUB REPOSITORY

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 as shown in the following screen in Figure 13.
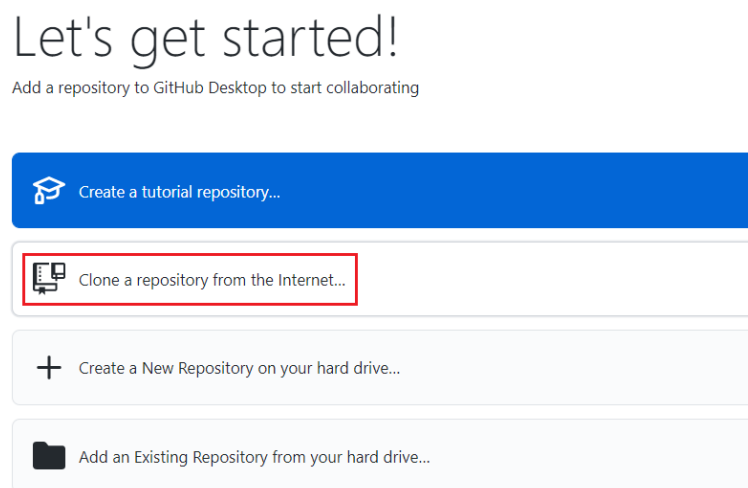

Figure 22: Clone a repository from the Internet.

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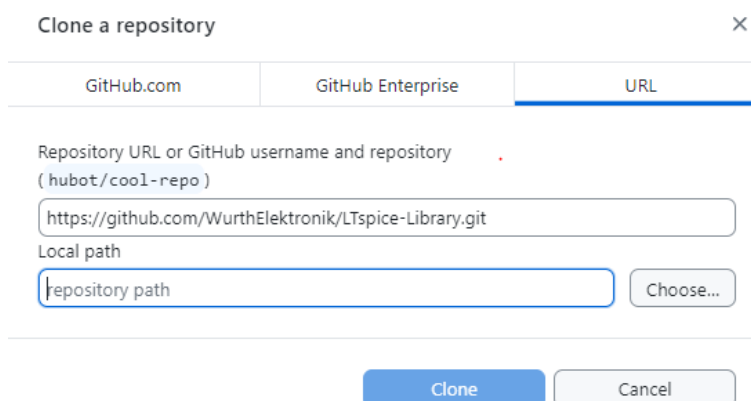

Figure 23: Clone Setup.

Cloning repository will take some time.

Figure 24: Clone in progress.

4.3 Synchronize Local Library from GitHub

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

If there is update in GitHub repository, GitHub Desktop will detect it and you can "Pull" the update to your local. Once there's any new commits on the online master repository, from GitHub Desktop you'll receive the update information automatically.

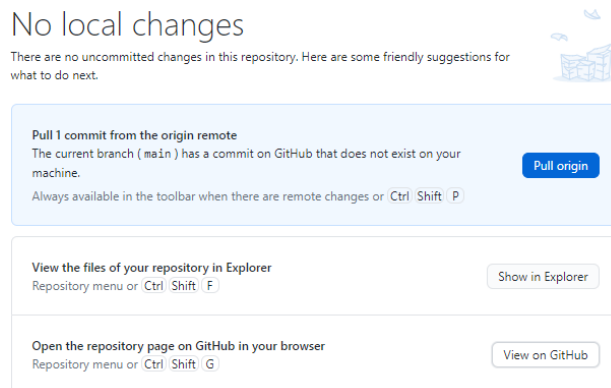Click **Pull origin** button to fetch the updates to local folder immediately.

Figure 25: Clone Setup.

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

Figure 26: 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