

USER GUIDE

UG002 | Using the LTspice Model Libraries



Toby Kangas

01. 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.

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

02. INSTALLING THE MODELS

2.1 To Install the Model

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

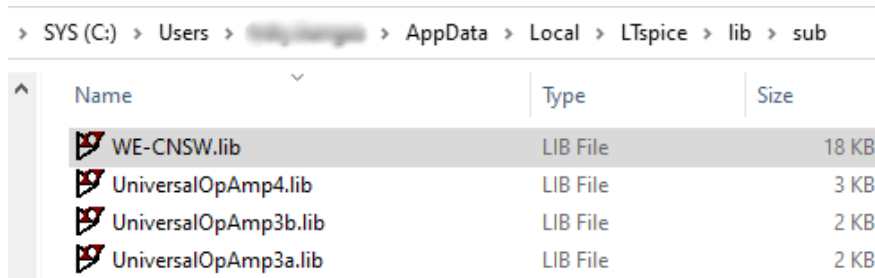


Figure 1: Save netlist files in \sub folder

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.

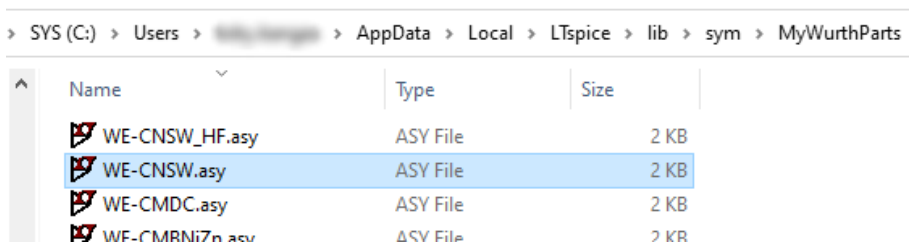


Figure 2: Save symbol files in \sym folder

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

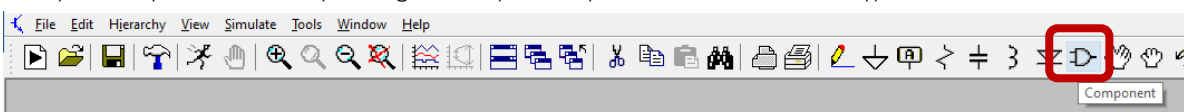


Figure 3: Insert Component

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

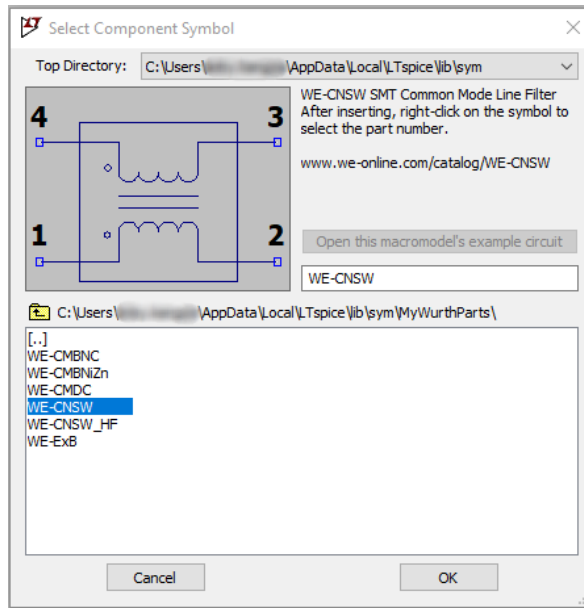


Figure 4: Select Component

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

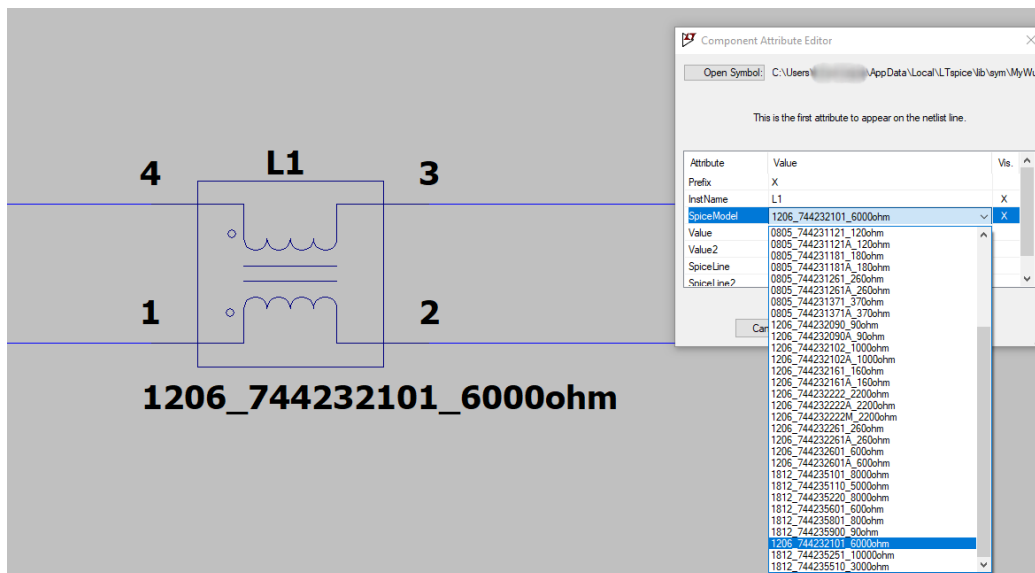


Figure 5: Choose part number

2.2 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).

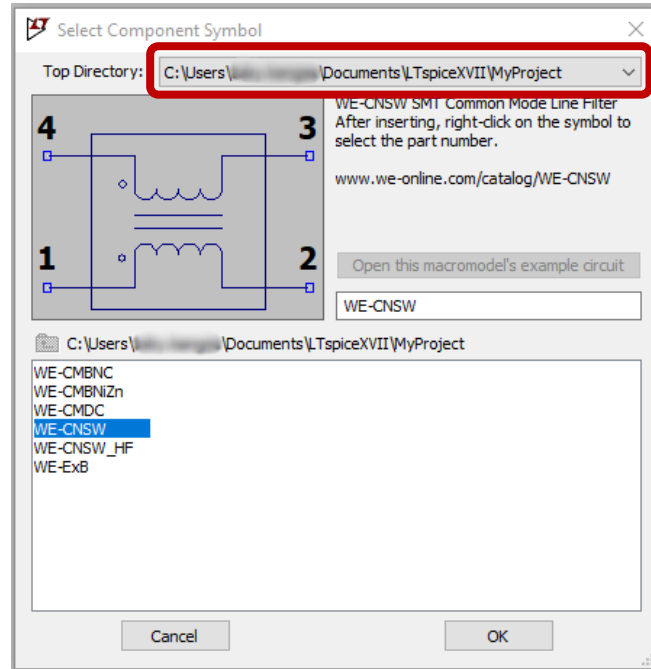


Figure 6: Model files in same folder as schematic

2.3 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.

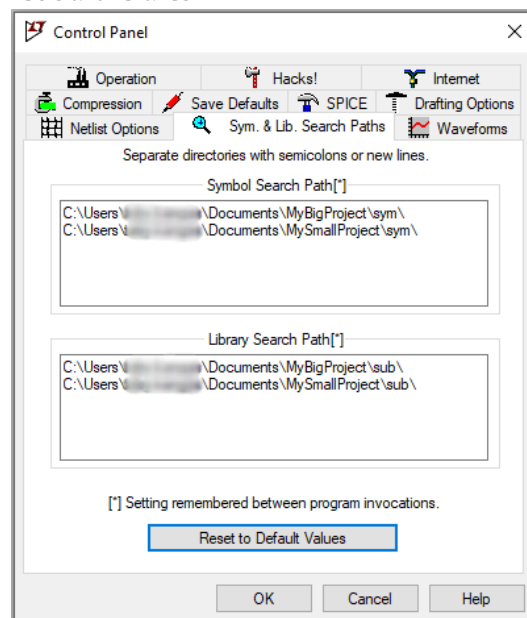


Figure 7: Define user library location

USER GUIDE

UG002 | Using the LTspice Model Libraries

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.

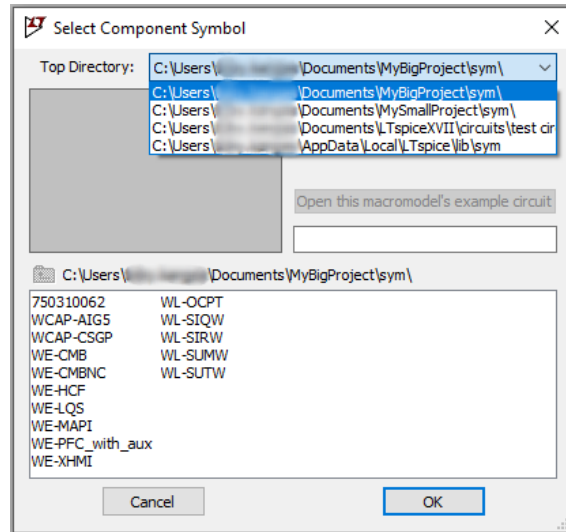


Figure 8: Select library directory

03. 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.

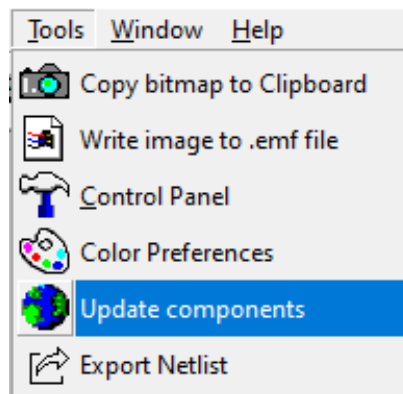


Figure 9: Update Components

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').

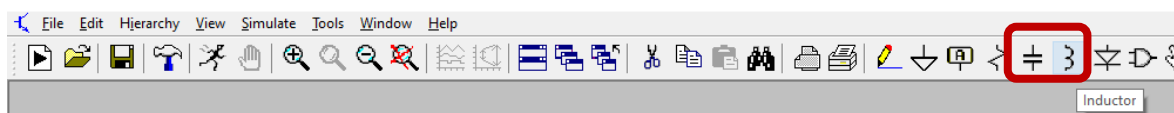


Figure 10: Insert Inductor or Capacitor

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

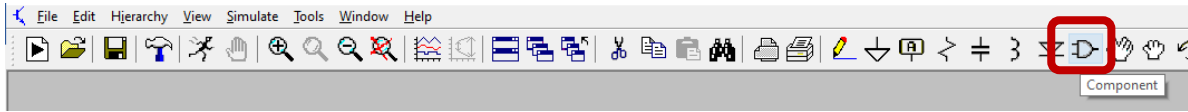


Figure 11: *Insert Component*

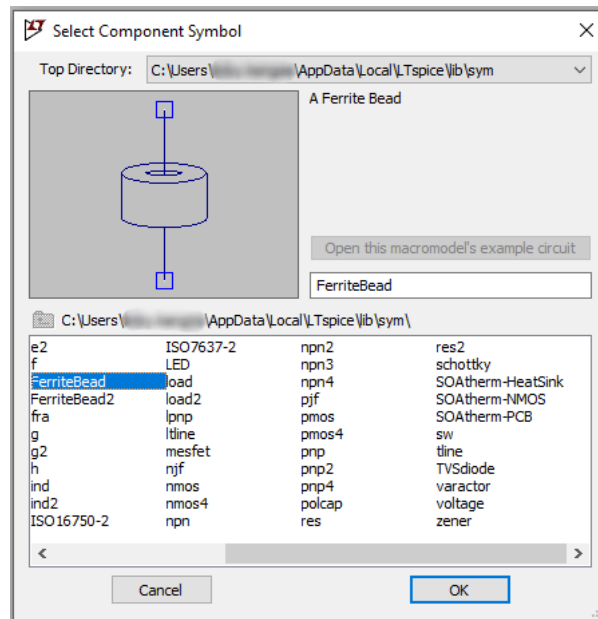


Figure 12: *Select ferrite bead symbol*

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

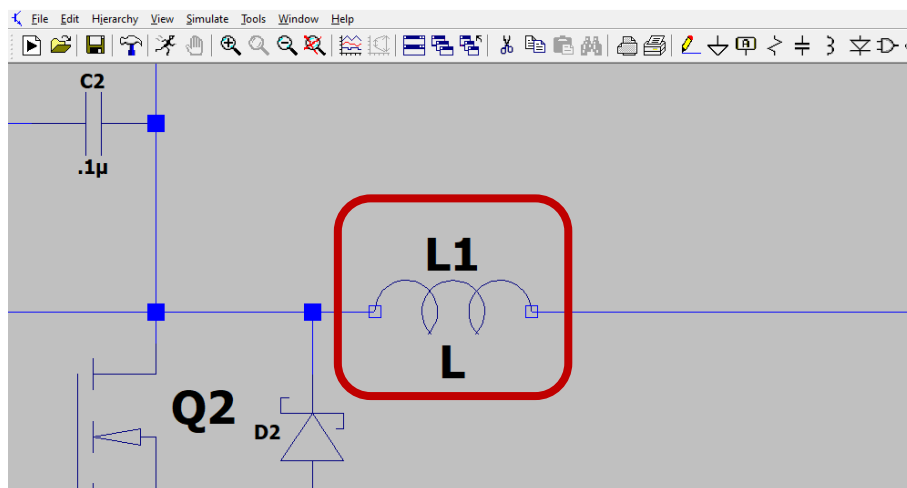


Figure 13: *Place component*

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

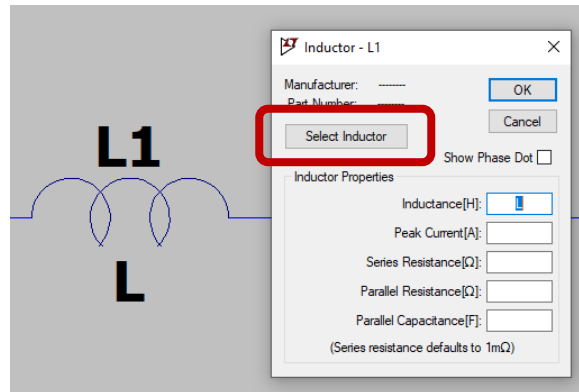


Figure 14: Select inductor

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.

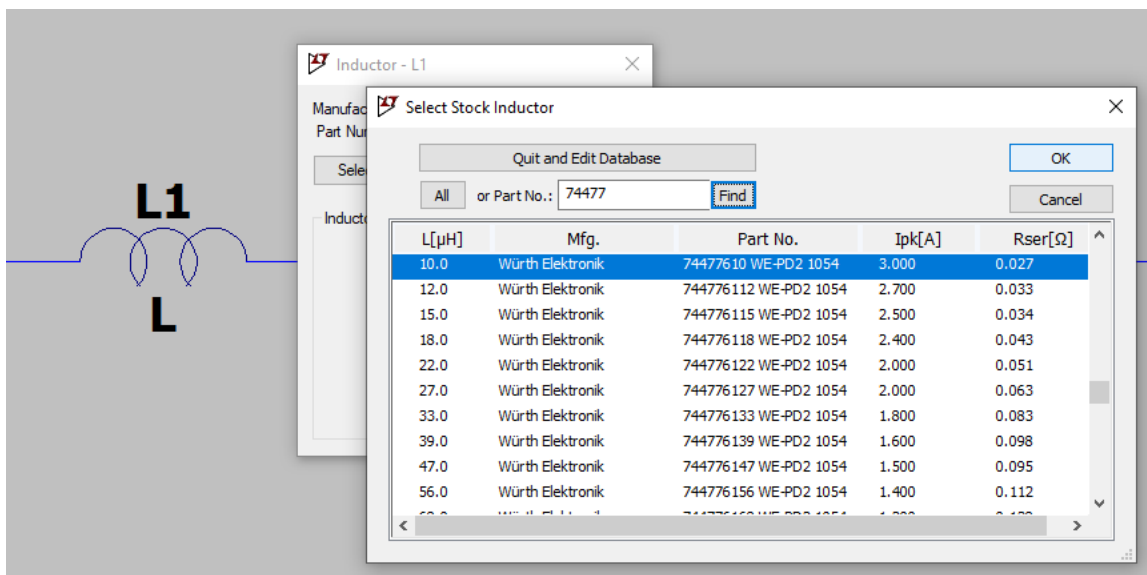


Figure 15: Select part number

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').

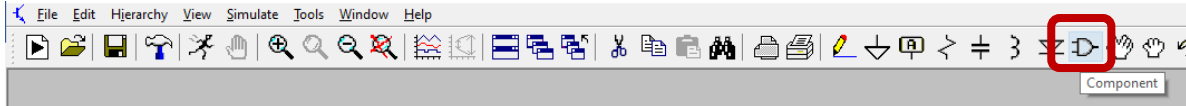


Figure 16: Insert component

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

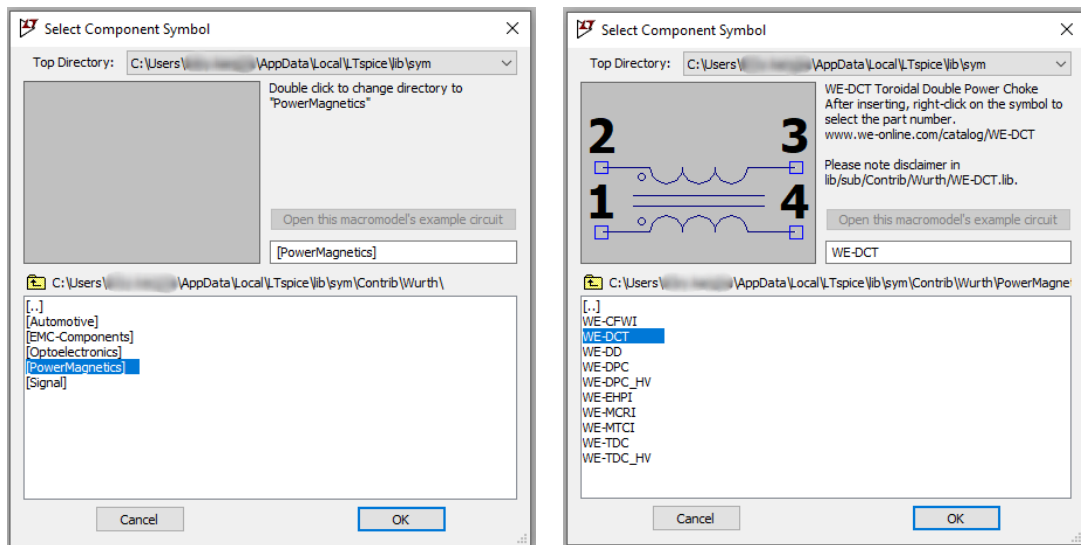


Figure 17: Navigate to component

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 SpiceModel value in the Component Attribute Editor, select the desired part number, and click OK.

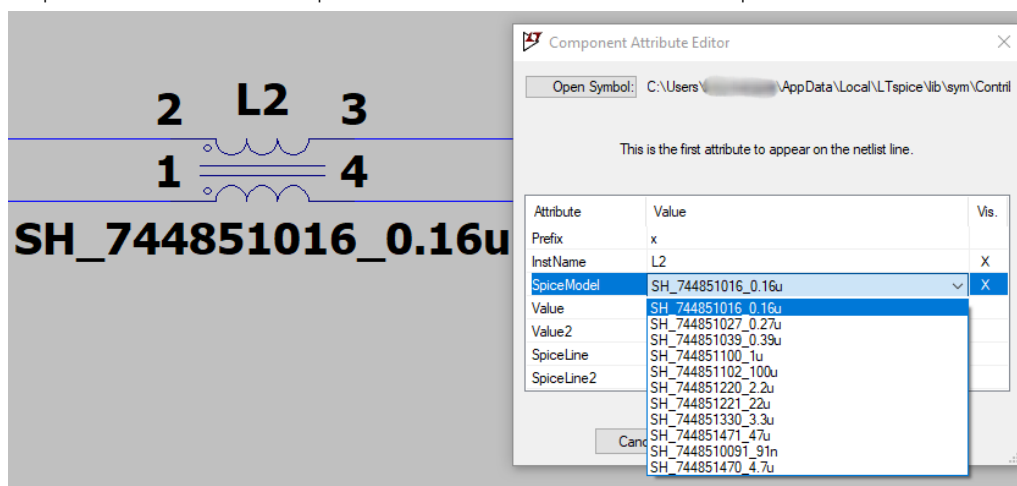


Figure 18: Choose part number

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 Plattform
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