**LT Spice Common Mode Choke**

Please download the LT SPICE Model from the table at the bottom of this page. Then, click the ZIP file to save it to your desktop or desired folder.  
  
**LT Spice Library models**  
Design engineers and product development engineers typically start with schematic design and do simulations to optimize their designs. This inductor SPICE model can help engineers model inductors in their circuit.  
CWS provides this LT Spice simulation model so that design engineers can select the most optimal inductive component for the circuit design.  
  
**LT Spice – how it works**

* Analog devices provide the LT Spice Schematic and simulation platform (www.Analog.com) for easy circuit modeling. The LT SPICE platform has become an industry standard for many other simulation software and tools. This platform saves time, cost, and effort by minimizing re-spins of actual circuit hardware.
* Many design engineers and educational institutions use LT Spice to design and simulate the circuits by importing the library components from various manufacturers.  
  Various components manufacturers such as inductors, transformers, coils, capacitors, resistors, and other active and passive components have made SPICE Library of their components that users can download and link within their LT Spice circuit design to do the simulation.
* For more details on the LT Spice, please visit <http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
* We suggest that designers download the latest version from the above link. Analog devises Inc. updates its software regularly to improve its design and simulation tools.

**How to use the LT Spice is provided in the above link from Analog Devices.**

===================================================================

* Coil Winding Specialist Inc www.CoilWS.com CWS Coils - Coil manufacturer.
* LTspice Library for CWS Coils Common Mode Choke Version 1.1.
* Assembled on (April 2022) By Coil Winding Specialist Inc [www.coilws.com](http://www.coilws.com/)
* This LTspice library contains C-Series Common mode Choke Part Numbers: Please keep in mind that real components can vary in the range given in the datasheets.
* The values are based on the data provided in the datasheet.

 ==================================================================

Installation of COILWS Model Library

==================================================================

Please make sure that LTspice software is not running and follow the below steps

1. The zipped model library folder of the CWS\_Spce\_Library\_CMM for LTspice comprises the following two folders:
2. - Sub: This folder contains the ASCII data file (C\_CMM.lib) that represents the actual library
   * Copy and paste this file to C:Users"USER NAME"DocumentsLTspiceXVIIlibSub .
3. - Sym: This contains the subfolder "CWS Comp.Chokes" which includes the symbol data files for the graphics information required by the graphic user interface.
   * Copy and Paste this file to C: Users"USER NAME"DocumentsLTspiceXVIIlibsym .
4. On the next start of LTspiceXVII, open a New Schematic, Click on the Component Library (or press on F2), and find the [CWS Comp.Chokes] folder appears in the LTspice Component library.
5. Double Click on [CWS Comp.Chokes] folder and choose your Part number.

 NOTE:

* PLEASE DON'T RENAME ANY OF THE ABOVE 2 FOLDERS.
* ' "USER NAME" - this varies for each individual user's laptop/PC
* In the DEMO circuit attached below - in case the LTSpice shows any error/missing component - please add the 'DEMO Circuit Supplement Library file in the LTSpice folder as explained in the "Readme-Demo-Ckt-Simulation."

DISCLAIMER

* The specific LT Spice models given in this file/folder are the property of Coil Winding Specilist Inc. [COILWS.COM](http://www.coilws.com/).
* LT SPICE MODEL is an effective tool for testing product performance by simulation; however, it does not simulate product performance in all test environments and is not intended to be a replacement for testing the actual device by means of a test board or otherwise.
* Please refer to the datasheet for all electrical parameters.
* Simulation results are for reference purposes only; the CUSTOMER shall perform thorough testing using the actual device.
* The LT Spice models will be updated as needed as new products are added - always download the latest version from the http://www.coilws.com website.
* The LT Spice software also will be updated without notice - refer to the software developer website for the latest versions.
* ALL PRODUCT, PRODUCT SPECIFICATIONS, AND DATA ARE SUBJECT TO CHANGE WITHOUT NOTICE TO IMPROVE RELIABILITY, FUNCTION OR DESIGN, OR OTHERWISE.