Powerful filter simulation with SPICE

Today, a successful circuit design is (almost) always preceded by a simulation. However, any simulation is just as good as the data available for it. SCHURTER therefore offers sophisticated simulation models for a large number of its single- and three-phase common mode chokes.

SCHURTER DKIV-1: Common mode choke for vertical PCB mounting

Circuit simulation is a key element for electronics design and can be performed using various computer-based tools. One that has been proven for this purpose for many decades is SPICE (Simulation Program with Integrated Circuit Emphasis), which was first introduced at the University of California in 1973. This program has been further developed over decades and is now an established and widely used program that calculates algorithmic approximations for analog, digital and mixed electrical circuits.

Close to reality

A SPICE simulation model is a virtual description of the physical behavior of an electrical component or assembly. The goal of modeling is to provide a picture as accurate as possible of the actual functional behavior of the element under testing. To achieve this goal, the simulation must account for various parasitic effects in addition to the nominal behavior. This requires advanced models and thus more accurate simulation results than those obtained with an oversimplified " image" of the component.

Choke models

SPICE models, now available as libraries, refer to some of the latest SCHURTER single-phase and three-phase common mode choke families. These include for example the ferrite chokes of the types DKIH-1, DKIV-1 and DKIH-3.
They can be easily imported into a standard SPICE simulation program, and their simulated common-mode (CM) impedance can be compared to the choke's real CM impedance, which is specified on the product data sheet. This is done over the entire specified frequency range (from 10 kHz to 10 MHz) and under standard conditions (line/load impedance of 50 Ω).

Model and application

The following paragraphs provide details on choke model development and model validation, as well as an example of how they can be used in a real circuit of a more complex system simulated with LTSpice [1].

The choke

The following figure shows the equivalent circuit of an inductor, the basic element of a choke (Fig. 1), which is identified by the international abbreviation L:

Fig. 1: Wiring diagram for inductance

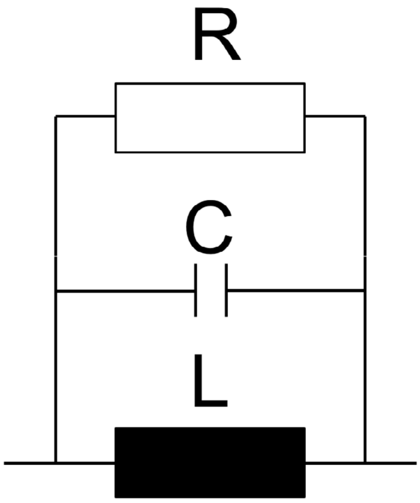

Fig. 2: Equivalent circuit diagram of the first approximation of an inductance

In addition to the nominal inductance (L), two other elements are added in parallel to better characterize it: the stray capacitance of the winding (marked C) and a resistance contribution (R) proportional to the core losses. By adding new parameters, it is possible to take into account parasitic effects associated with the choke.
This allows fine tuning of the simulated impedance curve and improves its match with the measured impedance curve, as will be shown in the next section.
In the case of a single-phase choke (e.g. DKIH-1, DKIV-1), the complete model consists of two equal inductance models, one for each winding of the choke. In the case of a three-phase choke (e.g. DKIH-3), the three windings are modeled with three equal impedance models.
Models for all current ratings have been modeled (ferrite core version), and these are now available on the SCHURTER website upon request.

Model validation

The impedance curves for all chokes of the DKIH-1, DKIV-1 and DKIH-3 families were measured with a Vector Network Analyzer. The measurements were performed in the frequency range from 10 kHz to 10 MHz, with input and output impedances of 50 Ω / 50 Ω.
The parameter values used in the simulation model were previously extrapolated by direct measurements and subsequently modified to improve the results, i.e., the matching between simulated and measured impedance curves was increased.
In particular, L is proportional to the impedance slope at low frequency (ΔL), i.e., before resonance, while C and R are proportional to the resonant frequency (fr) and the impedance magnitude at the resonant frequency (Z(fr)), respectively (Fig. 3).
The following diagram shows simulated (in green) and measured (in red) impedance curves for the 16 A single-phase ferrite choke of the DKIH family.

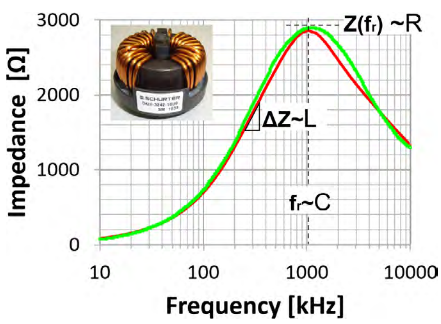

Fig.3: DKIH-1 16 A impedance comparison

The following graph shows the same comparison for a three-phase choke of the DKIH-3 family (simulation in green and measurement in red):

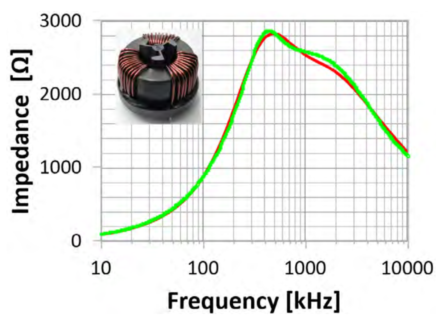

Fig. 4: DKIH-3 10 A impedance comparison

Using the models

During the design phase of an electronic device, the customer is usually interested in evaluating the performance of a choke before even testing a sample. Access to the SCHURTER choke SPICE library helps him to make the best choice for his own development.
SCHURTER chokes are developed to offer chokes for a wide range of end-use applications in various electrical fields: from frequency converters to charging stations and switching power supplies (SMPS). This, combined with the widely used SPICE software, makes SCHURTER SPICE models a versatile and useful tool for development purposes.
The schematic below is a simplified representation of a buck converter simulated with LTSpice at an input voltage of 10 V containing an artificial mains network (LISN) and a MOS transistor.

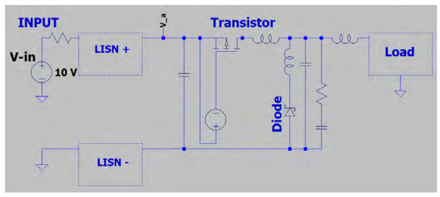

Fig. 5: Switching power supply in LTSpice

The components are modeled taking into account real parasitic effects, which lead to noise contributions superimposed on the expected signals. In particular, the input voltage is affected by some noise, as can be seen when examining the node V_a:

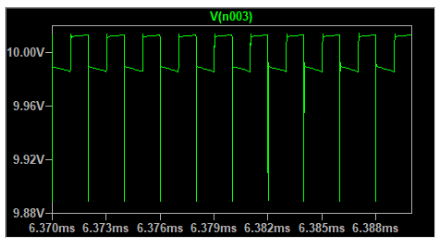

Fig. 6: Voltage at node V_a

In the given example, the customer may choose to install a choke between the LISN and the switching power supply to reduce the amplitude of the input noise. In this case, the 10 A choke from the DKIH-1 family may be the best choice because it is compatible with the available space.
When you download the DKIH-1 SPICE library from the SCHURTER Website, two files are shared:

In both cases, it is the DKIH-1 family, which contains models for all elements (ferrite version only), and the symbol of the product family, i.e. the circuit element itself, which represents the choke to be

Fig 7: SPICE Simulation Files

uploaded to the circuit. After downloading the LTSpice software, a special folder is created within the "Documents" folder in Windows. It is saved under the following path, which may vary depending on the operating system and the downloaded software version:

C:\user*username*\Documents\LTspiceXVII

After transferring the files shown in Fig. 7 to the lib\sub and lib\sym subfolders, the customer can import the choke symbol into his own circuit (Fig. 8), properly link it to other circuit elements, and select a specific element of the family via a drag-and-drop menu (Fig. 8).

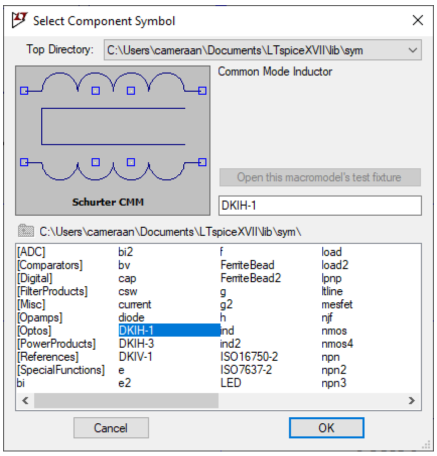

Fig. 8: Import of SCHURTER SPICE Models

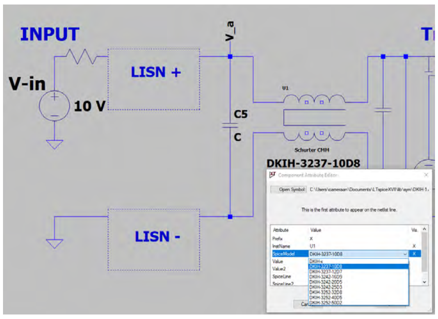

Fig. 9: Choice of choke

In the example shown above, the input noise is significantly reduced by adding the DKIH-1 10 A ferrite choke model, as shown in Fig. 10.

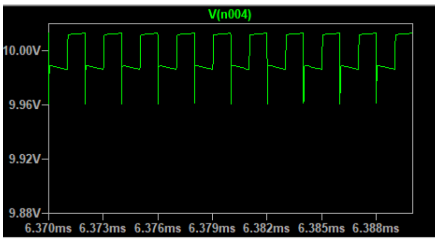

Fig. 10: Voltage signal at node V_a with implemented choke

Conclusions

With these advanced simulation models it is possible to simulate very precisely the effect of a common mode choke used in an electrical circuit with electrical noise problems. SCHURTER's service is designed to help the customer to find the best solution for his problem before testing a real choke prototype.

About SCHURTER

The SCHURTER Group is a globally successful Swiss family business. With our components ensuring the clean and safe supply of power, input systems for ease of use and sophisticated overall solutions, we impress our customers with agility and excellent product and service quality.

SCHURTER AG
Werkhofstrasse 8-12
6002 Lucerne
CH-Switzerland
+41 41 369 31 11
contact.ch@schurter.com
schurter.com

SCHURTER Headquarters in Lucerne/Switzerland

EMC-EMS Competence Center in Mendrisio/Switzerland

References / Downloads

[1] https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

[2] Download SCHURTER SPICE library