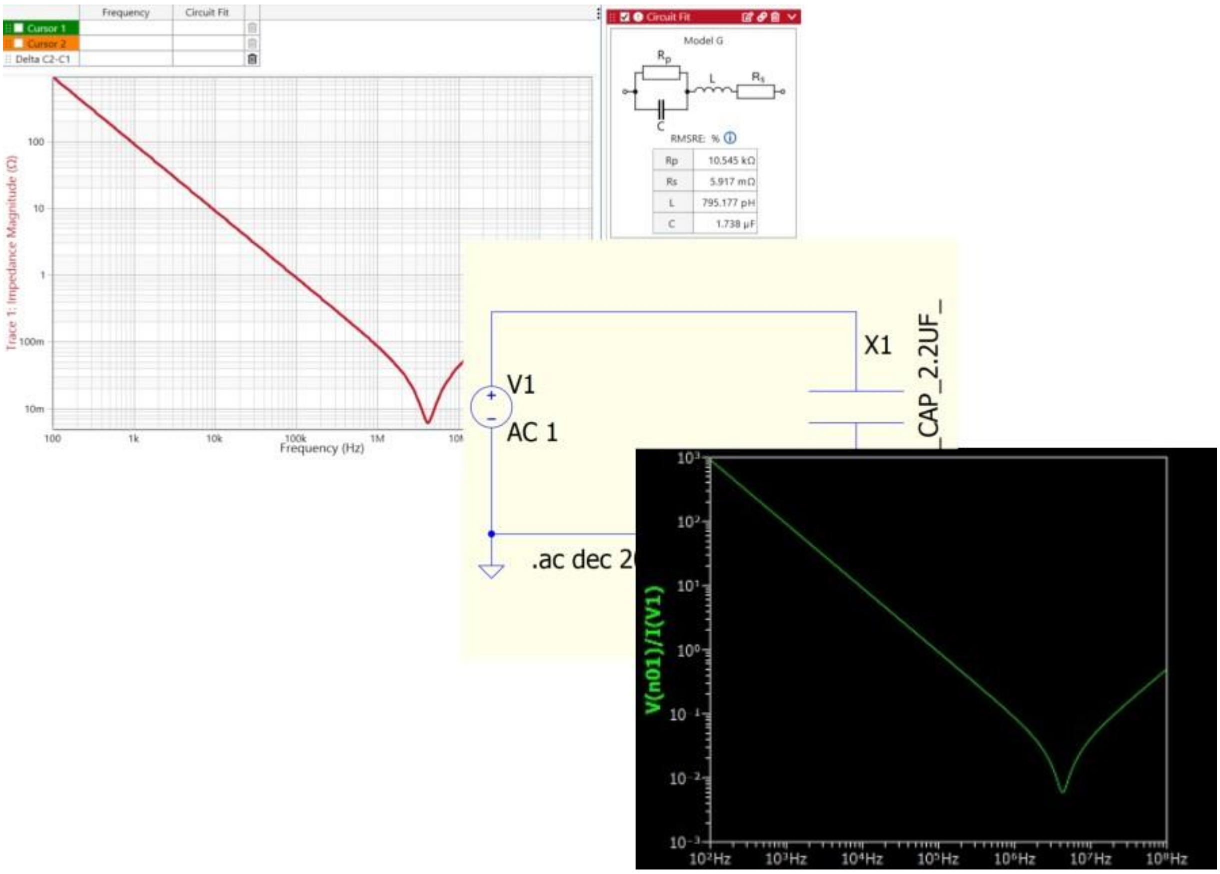
Bode Analyzer Suite - Information Note

Circuit Fit –SPICE Import Guide



By Sahil Ghate

Visit www.omicron-lab.com for more information.
Contact support@omicron-lab.com for technical support.

Table of Contents

1 Introduction **3**
2 Generate Model from Measurement **3**
3 Exporting Model as SPICE Netlist **4**
4 Use the Circuit Fit Model in QSPICE **4**
4.1 Importing the Model in QSPICE 4
4.2 QSPICE Simulation Results 5
5 Use the Circuit Fit Model in LTspice **6**
5.1 Importing the Model in LTspice 6
5.2 LTspice Simulation Results 7
6 Conclusion **8**

Note: Basic procedures such as setting-up, adjusting and calibrating the Bode 100 are described in the Bode 100 user manual available at:
[Bode 100, Bode 500 Vector Network Analyzer User Manual (omicron-lab.com)]

Note: All measurements in this application note have been performed using the Bode Analyzer Suite V3.51 Use this version or a newer version to perform the measurements shown in this document. You can download the latest version at
[Bode Analyzer Suite - Download Area - OMICRON Lab (omicron-lab.com)]

1 Introduction

This **Circuit Fit** tool is a convenient way to generate component models from the measured DUT impedance data. It enables users to simulate components based on real-live measured data. This data-driven modeling approach reduces the uncertainty between simulation and live measurements.

To achieve this goal, this document will provide a quick guide on how to export a fitted component model and how to import it in SPICE software. The **Circuit Fit** model created in **Bode Analyzer Suite (BAS)** is compatible with most SPICE simulators; only the import mechanism varies based on the SPICE software. In this document, we show how to import the model in QSPICE as well as LTspice.

2 Generate Model from Measurement

Before we export and simulate a DUT, we first need to have a model for it. Check out the documents “Circuit Fit Feature – Simple Model Guide” and “Circuit Fit Feature – Network Model Guide” to learn how to fit a model to measured impedance data.

In the following, we use the model from the document “Circuit Fit Feature – Simple Model Guide”. The fitted model is shown in the figure below.

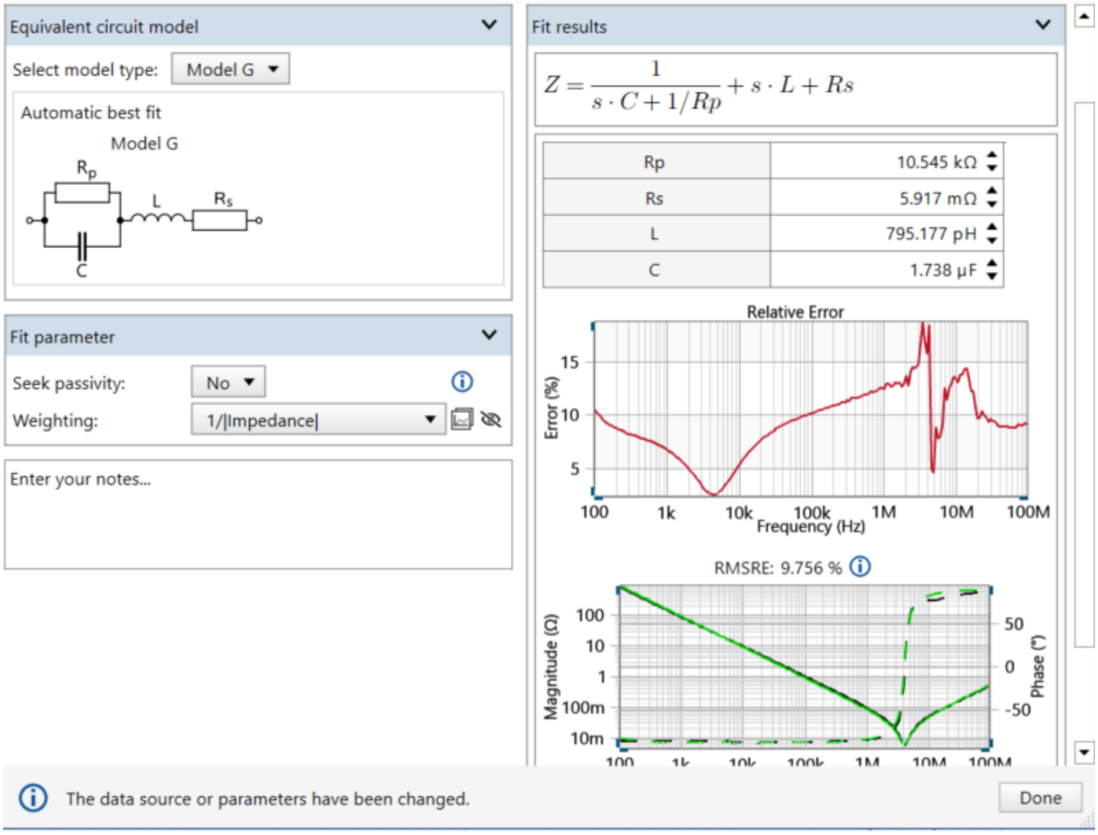


Figure 1: Fitted capacitor model based on measured DUT.

3 Exporting Model as SPICE Netlist

Once an acceptable model of the measured DUT is created, click on the **Export SPICE netlist** at the top of the **Circuit Fit** window. This will open a dialog where you can define where to save the exported netlist file. Depending on the requirements of your simulation software, the netlist can be saved as a .txt, .net,.sp, or .cir file. For simplicity, in this example, it will be saved as a .txt file.

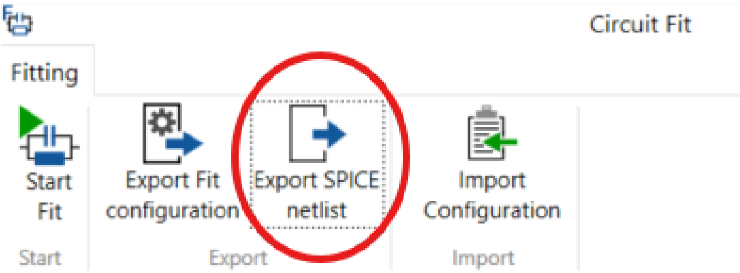


Figure 2: Export SPICE netlist button.

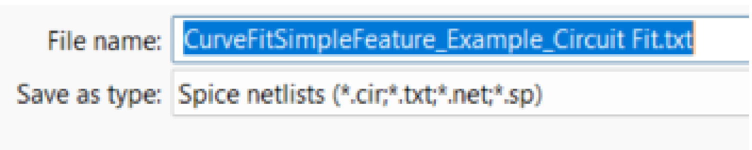


Figure 3: Save File options.

4 Use the Circuit Fit Model in QSPICE

QSPICE is a popular SPICE simulator from Qorvo. It can be downloaded from www.qorvo.com.

4.1 Importing the Model in QSPICE

Open the saved text file exported from BAS, copy the entire content (see Figure 4), and paste it into QSPICE. This will create a default symbol with the model's attributes as shown in Figure 5 a) and b)

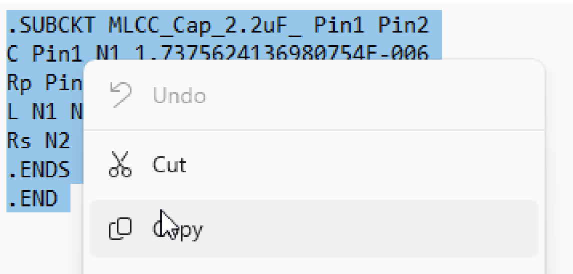
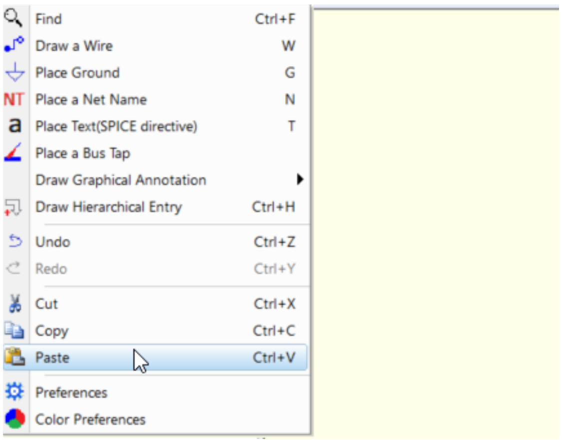
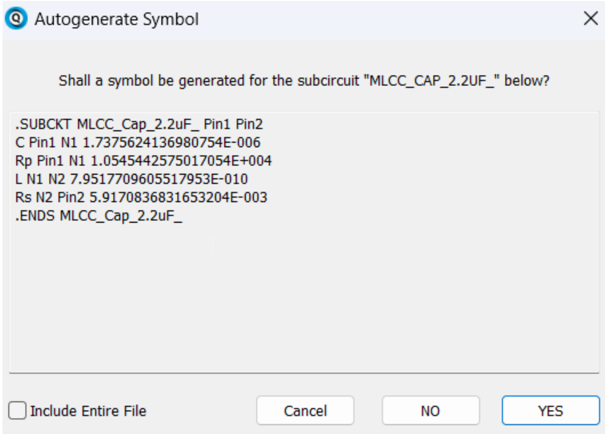


Figure 4 Copy the model from the text file.



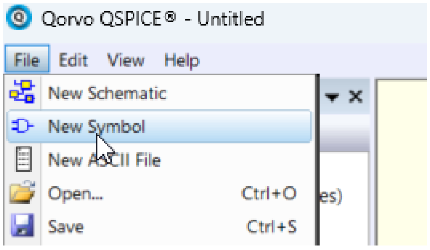
a) Use Edit → Paste.



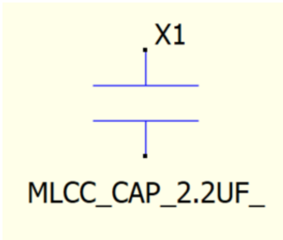
b) Autogenerate Symbol.

Figure 5: Import the model in QSPICE.

The model can be used with the autogenerated symbol. Alternatively, you can create a new symbol by clicking on the File menu and select New Symbol. Click on the model in QSPICE and copy it into the new symbol window. Modify the symbol drawing if necessary and save the file to a suitable location.



a) Use File → New Symbol.



b) Capacitor symbol created.

Figure 6: Create a new component symbol in QSPICE.

4.2 QSPICE Simulation Results

To verify that the model has been imported correctly, a simple impedance curve is calculated in QSPICE. The setup for the simulation consists of an AC frequency sweep with the impedance calculated through dividing the voltage across the capacitor by the current through the component. The circuit is shown in Figure 7.

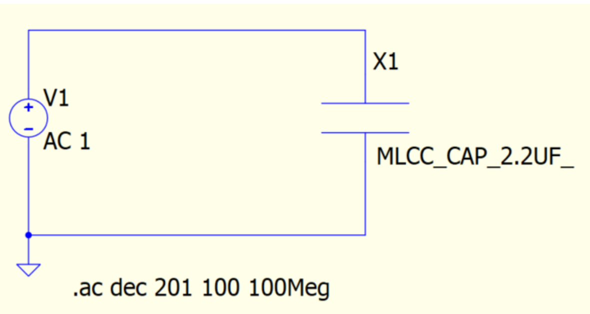


Figure 7: QSPICE simulation setup.

The simulation results from the impedance calculation in QSPICE for the DUT are displayed in Figure 8 below. The simulation results match the real measurement data acquired by Bode 500 with good accuracy.

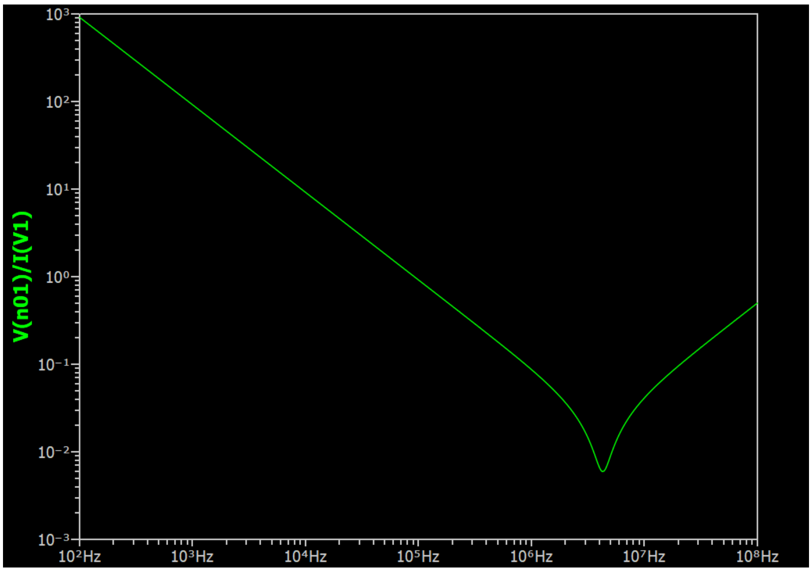


Figure 8: QSPICE simulation result.

5 Use the Circuit Fit Model in LTspice

Another popular SPICE simulator is LTspice which is available from www.analog.com.

5.1 Importing the Model in LTspice

Open the exported text file from BAS in LTspice. Right-click on the first line containing ".SUBCKT". In the drop-down menu, select "Create Symbol". A pop-up window will open requesting to save the file as .asy extension. Select an appropriate location to save the symbol file.

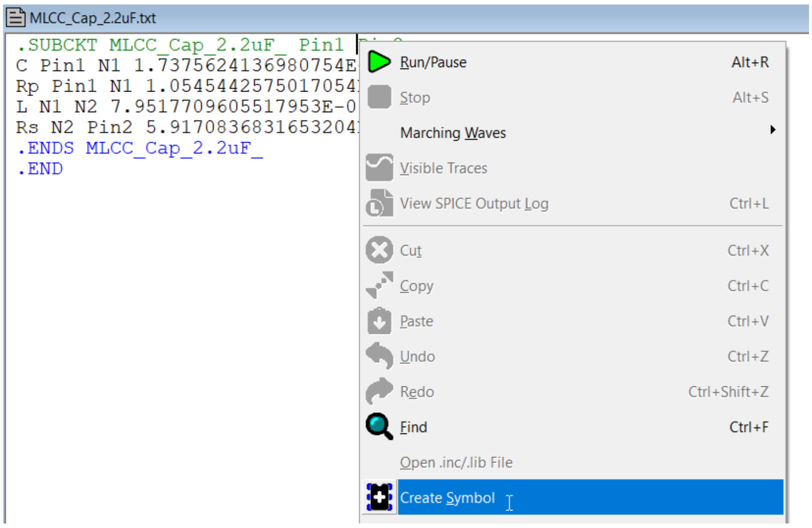


Figure 9: Open the model in LTspice.

Once saved, another window will open with a default symbol and the appropriate pins. Change the symbol according to your needs. The symbol created can be inserted into the simulation as any other component symbol; however, the directory may need to be changed when searching for the symbol if you have saved in a location other than the default LTspice directory.

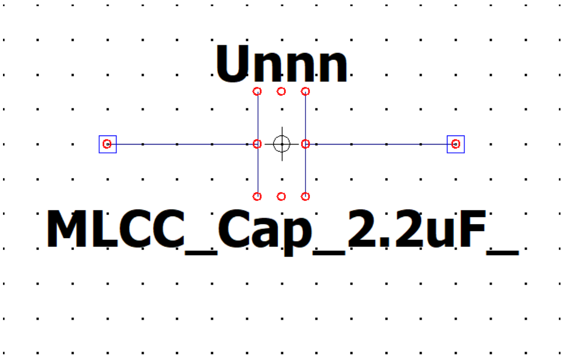


Figure 10: Capacitor symbol created in LTspice.

5.2 LTspice Simulation Results

The simulation in LTspice was set up as shown Figure 11. The ".net" parameter command was used to acquire the impedance of the component.

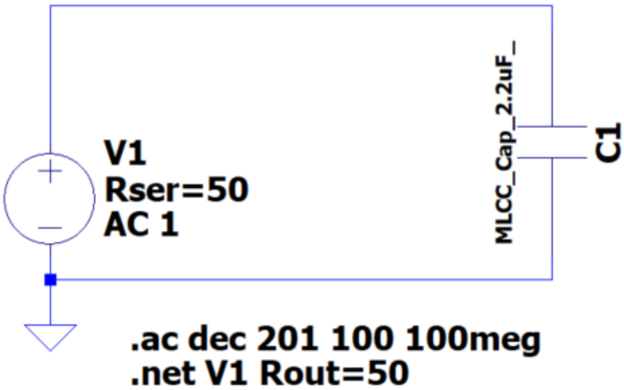


Figure 11: Simulation setup for the component

The simulation results of the component are displayed in Figure 12. The results are identical to those obtained in the real-world measurement performed at the beginning with the Bode 500.

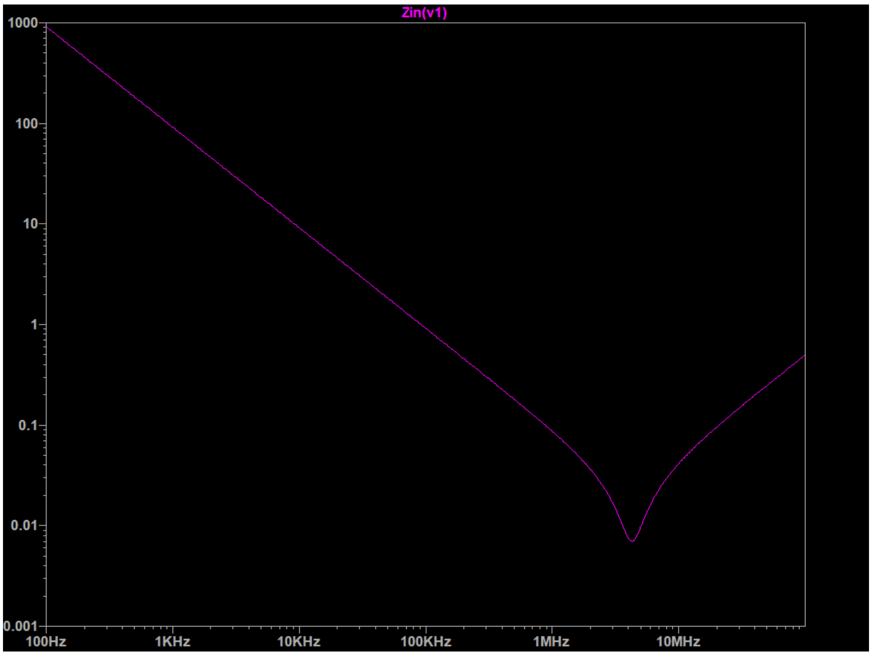


Figure 12: LTspice simulation result of the imported component

6 Conclusion

The **Circuit Fit** feature is a user-friendly tool that enables the integration of real-world measurements into the simulation process. This feature allows to easily use data-driven models that accurately capture the component impedance. The data-driven approach encompasses parasitic, manufacturing defects, and external influences, which increases the accuracy of simulations in relation to their real-world counterparts. The feature streamlines the process from measurement to simulation, reducing the user's workload.

OMICRON Lab is a division of OMICRON electronics GmbH, specialized in providing Smart Measurement Solutions to professionals such as scientists, engineers and teachers engaged in the field of electronics. It simplifies measurement tasks and provides its customers with more time to focus on their real business.

OMICRON Lab was established in 2006 and is meanwhile serving customers in more than 60 countries. Offices in America, Europe, East Asia and an international network of distributors enable a fast and extraordinary customer support.

OMICRON Lab products stand for high quality offered at the best price/value ratio on the market. The products' reliability and ease of use guarantee trouble-free operation. Close customer relationship and more than 30 years in-house experience enable the development of innovative products close to the field.

Europe, Middle East, Africa
OMICRON electronics GmbH
Phone: +43 59495
Fax: +43 59495 9999

Asia Pacific
OMICRON electronics Asia Limited
Phone: +852 3767 5500

Americas
OMICRON electronics Corp. USA
Phone: +1 713 830-4660

info@omicron-lab.com • www.omicron-lab.com