HOW2POWER TODAY
Your Power Design Newsletter

Exclusive Technology Feature

ISSUE: November 2024

Using VNA Software To Build Measurement-Based Models

by Charles E. Hymowitz, Paul Ho and Michael LaNovara, AEi Systems, Centennial, Colo.

In today's power integrity (PI) simulations picohenries matter. Timely availability, fidelity, and model accuracy matter. However, vendor-supplied component models, including those available for passive and other components, are often inaccurate or not available. For system designers performing PI simulations, the answer to this problem is either to obtain better models from third-party sources or to build their own measurement-based models.

This second option has been made easier with the introduction of the new Curve-Fitting–Simulation Model generation feature in the Bode Analyzer software Suite (BAS)[1] (Fig. 1). The BAS software drives the popular OMICRON Lab Bode 100 and 500 vector network analyzers (VNAs) and with the new model generation feature, this software can curve fit the data from impedance measurements to create lumped element models (SPICE).

This capability is also enabled by the overall usability of the Bode 100 and 500 VNAs. In general, VNAs are the primary instrument used to measure impedance, s-parameters, and loop-gain/phase over frequency. But in the past, these instruments were not widely used because many designers lacked the training to use them. Cost and usability were also factors. However, the Bode 100 and Bode 500 have simplified test setup and measurement for VNAs, moving the once daunting network analyzer class of tools into the ease-of-use category.

This in turn has made frequency domain testing accessible for educators and advanced PI gurus alike. While most power electronics engineers are typically not PI gurus, numerous articles and videos have discussed why PI issues are relevant to power designers.[2]

The Bode 100 and Bode 500 take frequency domain measurement further with the ability to post process the test data into a simulation model. This capability can be used to generate models for passive components, sensors, filters, VRMs, black boxes, and basically any item whose impedance you can measure. Best of all, your model accuracy will be known and in your control.

In this article, impedance measurements are performed on a set of capacitors, a power inductor, and a ferrite bead. The Bode Analyzer software suite is then used to curve fit the data to SPICE models. This article does not discuss how to make the impedance measurement and only covers the steps to create the models as there are already many published articles related to performing the measurements.

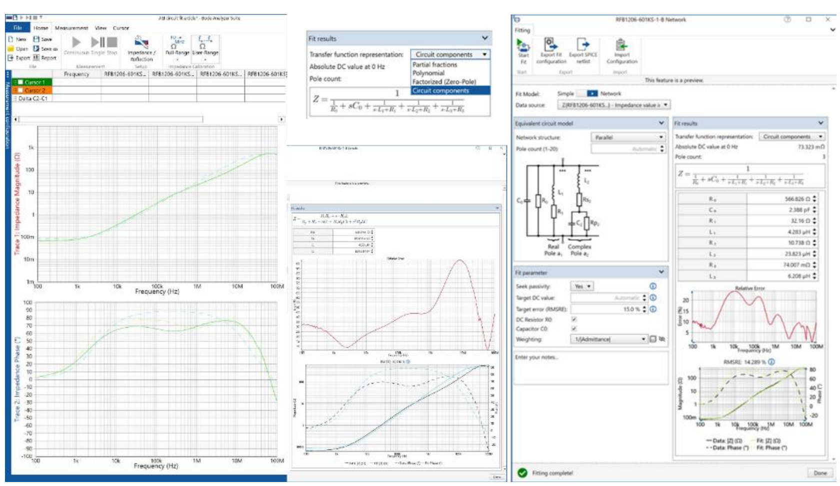


Fig. 1. In BAS v3.5 of the software suite for the OMICRON Labs' Bode 100 and Bode 500 VNAs, a new curve fitting and SPICE modeling feature has been added to the advanced graphics and menu driven S-parameter and impedance tests.

The Need For Measurement-Based Models

There are new analog and EM simulators being created over time. Witness the recent emergence of QSPICE into the analog/mixed AD realm, trying to further increase the reach of SPICE-based tools. But in the end, they ALL bow to the significance of the justifiably maligned simulation model. It really is a wonder that so much emphasis is placed on simulation speed (a nebulous and dubious factor given today's computer horsepower) over model fidelity.

Two factors invariably override claims of "superior" performance. First, most convergence problems are due to poor models.[3,4, 5] And second, if the models aren't sufficient, do we really care about getting results faster. The lack of focus on higher quality models by EDA tool vendors is challenging to those of us in search of the *right* answer and not just any old answer faster.

For power integrity simulations, especially those using EM and including the PCB, the model accuracy is often suspect, or bandwidth limited. Given that most vendor models don't come with any documentation, lacking correlation data and what characteristics they do or do not have; the need for known-good models is a critical factor.

For many vendors, the measurement methods and test setups employed by them are suspect. For instance, it can be unclear whether the capacitor mount is included, what instrument was used, the test setup accuracy, and many other factors.[6]

It may come as a surprise to many that the models contained in online software tools are often based on heritage data that may not be accurate for recent lots of parts. We once found a capacitor nominal curve that was greater than the specification maximum. When questioned about the discrepancy, the vendor stated that lots of newer parts had a lower mean, and the software hadn't been updated to reflect the newer data.

For all these reasons, the ability to quickly generate your own models directly from test measurements is extremely beneficial.

What Is the Circuit Fit Feature?

This new feature, at least as far as VNAs are concerned, allows you to fit an equivalent circuit model to measured impedance or admittance data and then generate a simulation (SPICE) model. For this article, the Coilcraft AE598PTA472KSZ power inductor, Vanguard Electronics RFB1206-601KS-1-B ferrite bead, and a simple PCB with five 1-µF capacitors in parallel on it, were measured and modeled.

The two-port shunt-thru method is used to gather the impedance data for the examples in this article since we are measuring small package components. The Bode 500 has a higher frequency measurement capability compared to the Bode 100 (450 MHz vs. 50 MHz). However, only the ferrite beads required higher frequency capability as the self-resonant frequency was greater than 50 MHz.

The Circuit Fit/SPICE Modeling feature requires that the impedance data be measured. There are many ways to do this, though the two-port shunt-through test is the gold standard, and the one used here.[7] Fig. 2 shows the two-port impedance test setup using a VNA (OMICRON Lab Bode 500), a two-port probe (Picotest P2102A) and a common-mode transformer (Picotest J2102B) to minimize the effects of the ground loop offsets in the measurement.

The two-port shunt through setup can be used to measure low impedance from ohms down to micro-ohms. Any impedance test setup, one-port, three-port, etc. can be used to measure impedance that can be fit to a model.

Fig. 2. Two-port shunt through impedance test setup showing the VNA, two-port probe and the common-mode transformer to break the ground loop.

Using The Circuit Fit Feature

The following describes the procedure after testing the impedance of a DUT. In the Measurement tab, you click the Add Circuit Fit icon to start the process (Fig. 3).

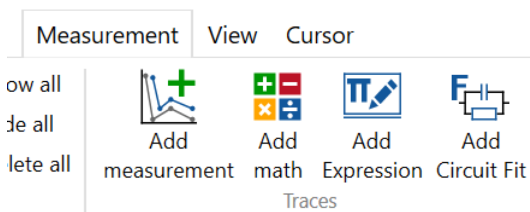


Fig. 3. From the Measurement tab of the Bode Analyzer Suite, users can select the "Add Circuit Fit" feature.

There are two types of "Fit Model" structures that the BAS software tries to fit to your impedance data. The Simple Fit Model (Fig. 4) uses a group of eight pre-defined model structures whereas the Network Fit Model uses an arbitrary series or parallel structure with a user-definable number of real and/or complex poles.

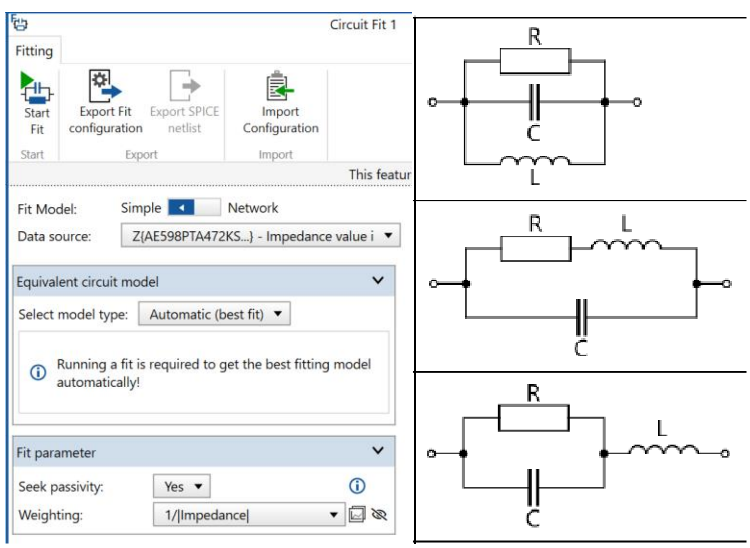


Fig. 4. The Circuit Fit menu (left) and some of the preset Simple model topologies (right).

To fit an impedance curve to a SPICE Model:

1. Choose the impedance or admittance measurement data you want to fit from the Data source menu.

2. The best pre-defined model with the smallest RMSRE (root mean square relative error) will be automatically selected.

3. The "Weighting" default $\frac{1}{|Impedance|}$ can be used in this case since the resulting model contains only a few poles and zeros without much complexity. This field also allows you to add weight to certain frequency ranges to increase or decrease the accuracy of that frequency range.

4. "Seek Passivity" forces the solver try to use positive component values in the resulting models. Negative component values can potentially cause simulation convergence problems in certain SPICE and S-parameter software.

5. Clicking "Start Fit" extracts the model topology and shows the matching accuracy which can be refined.

Note: the fit calculation is performed in the cloud and requires an active internet connection as well as a connected *Bode 100* or *Bode 500* instrument. However, a useful feature of the BAS is that measurements can be saved, and you do not need an "active" measurement being recorded to use the Circuit Fit feature. It can be from stored data.

6. After a few moments the fit results are displayed. If the fit results are satisfactory, you can click on the Export SPICE netlist button to save a SPICE .subckt netlist.

You can save iterations of the curve fit waveforms for comparison and choose the one(s) you want to keep. The following are a few specific examples using the Curve Fit feature.

Power Inductor Model

The AE598PTA472KSZ power inductor, whose impedance is shown in Fig. 5, has a relatively low RMSRE (3.726%) using the Model F model type.

The Model F model type was automatically selected by the solver. It's also possible to click on the "Select model type" drop down menu to see the other available pre-defined Simple Fit Models and their RMSRE values. The pre-defined impedance transfer function is shown on the top right corner along with the generated component values, calculated relative error vs. frequency. The model impedance magnitude and phase are plotted to show the measured vs. resulting models' frequency response along with the error.

A SPICE netlist is exported by clicking on the Export SPICE netlist button. A dialog window will pop up asking for the file name and save location. The Save as type (ASCII) defaults to "Spice netlists (*.cir, *.txt, *.net, *.sp)". A netlist for the example inductor is shown below.

```
.SUBCKT AE598PTA472KSZ_Simple Pin1 Pin2
C Pin1 Pin2 6.0507313239495896E-012
Rs Pin1 N1 6.1949613647142932E-003
L N1 Pin2 4.7981894537851418E-006
Rp Pin1 Pin2 2.1485337387142699E+003
.ENDS AE598PTA472KSZ_Simple
```

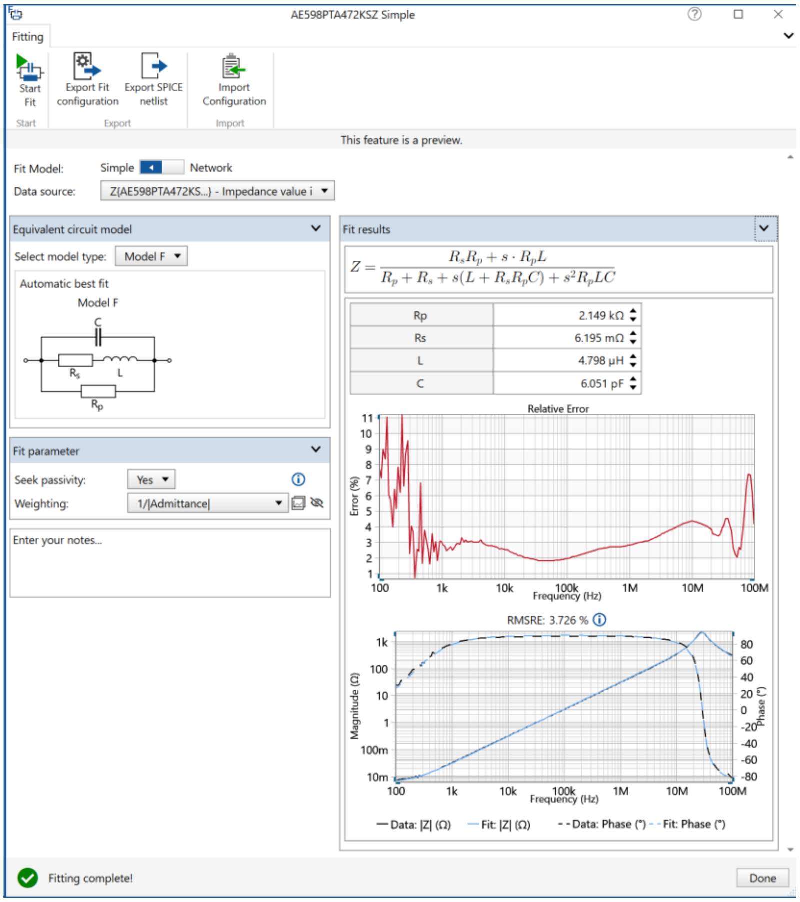


Fig. 5. The resulting match and model after fitting the power inductor impedance data.

The overlay between the measurement and model of the AE598PTA472KSZ shows a very good match between the two (Fig. 6).

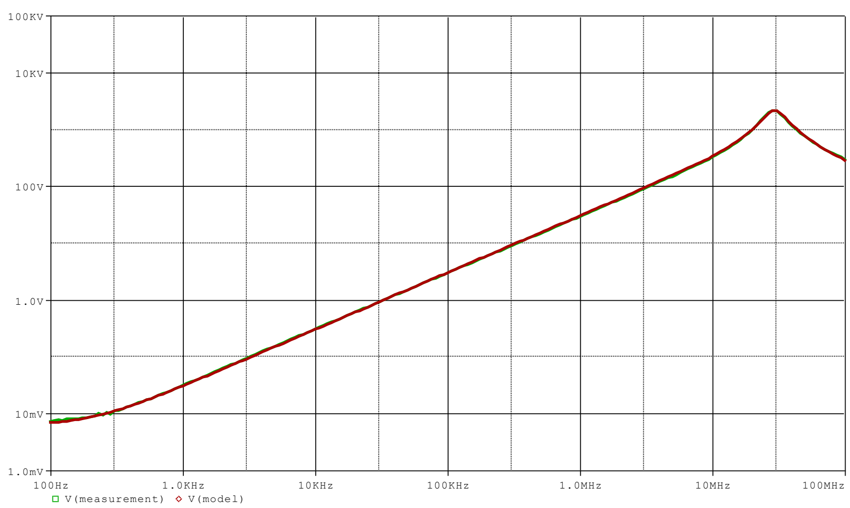
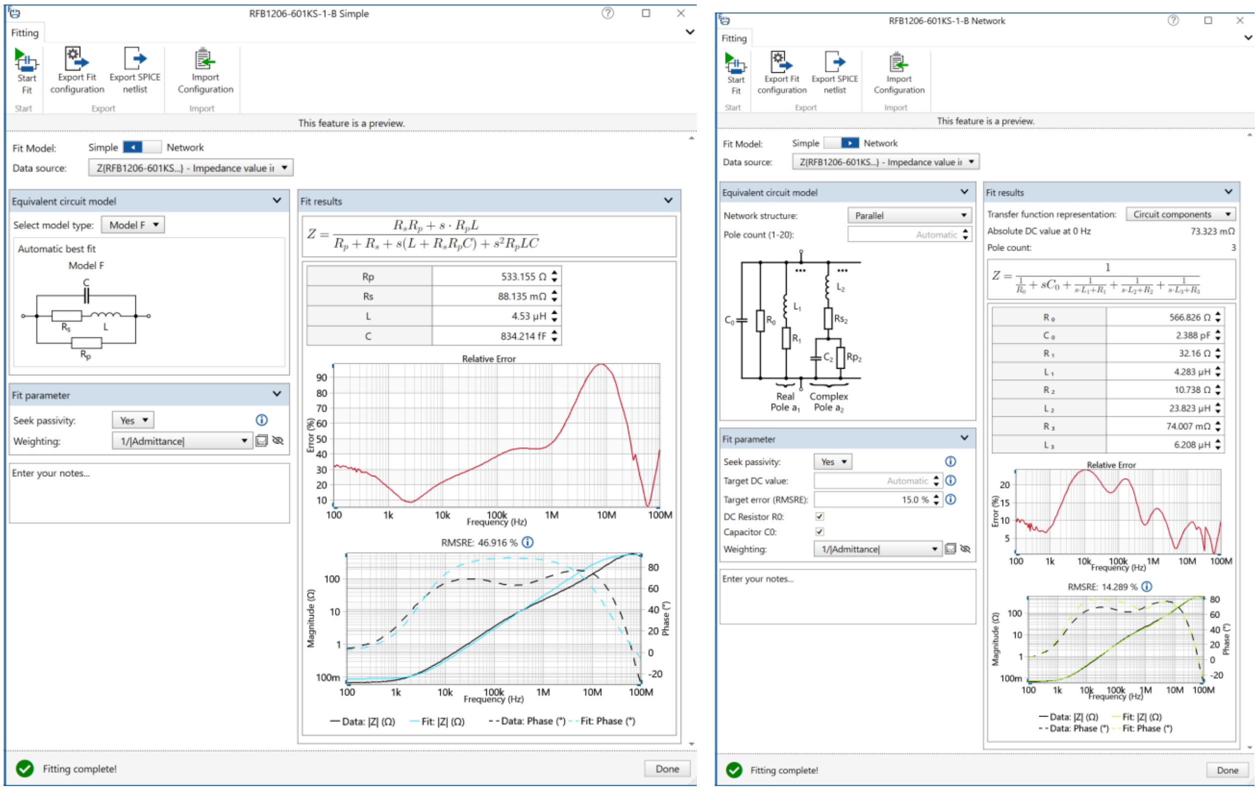


Fig. 6. AE598PTA472KSZ test data and simulated overlay in PSpice.

Ferrite Bead Model

Here's a Simple Fit Model of a ferrite bead, the RFB1206-601KS-1-B shown in Fig. 7a. The RMSRE is quite high (46.916%), so we'll try using the Network Fit Model to improve accuracy. Shown next is the Network Fit Model which has as an RMSRE of 14.289% (Fig. 7b). This is a factor of 3x improvement. Note the Absolute DC value at 0 Hz which is 73.323 mΩ, which is reasonable for this ferrite bead.



(a) (b)

Fig. 7. The Simple Fit Model of the RFB1206-601KS-1-B ferrite bead needs improvement (a). The Bead Network Fit Model shows significant improvement in accuracy (b).

Also note that the Fit parameter section (bottom left of Fig. 6b) has two additional checkbox options DC Resistor R0 and Capacitor. If the checkbox is not checked, the corresponding component is replaced with an open circuit which may be necessary depending on your part.

The Network Fit model has multiple transfer function representations that can be viewed (Fig. 8). The overlay between the measurement and model of the RFB1206-601KS-1-B shows a very good match between the two (Fig. 9).

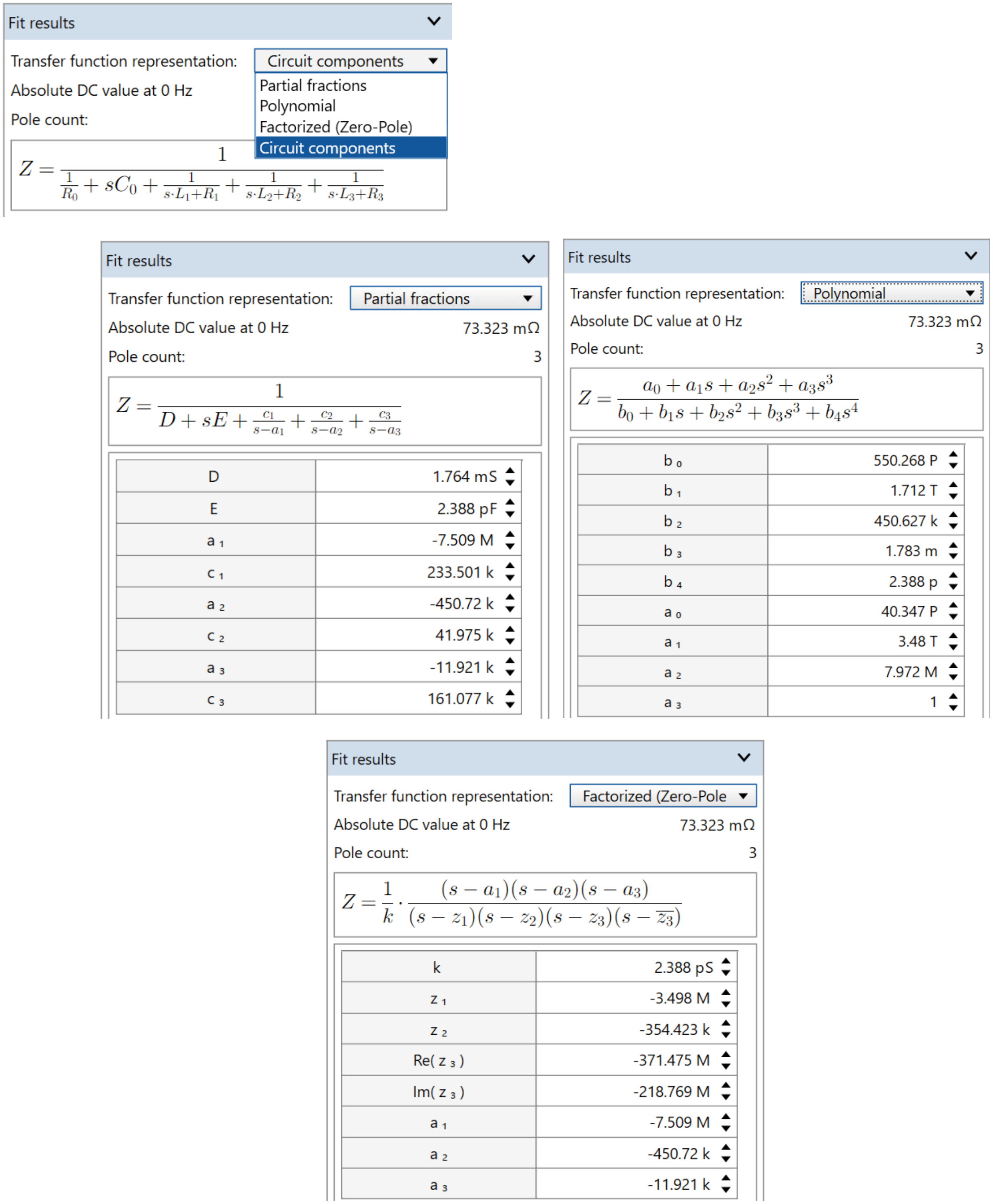


Fig. 8. The Fit results Transfer function dialog can be changed to either Partial fraction, Polynomial, or Factorized (Zero-Pole), to show those representations. No model is exported but the coefficients are available.

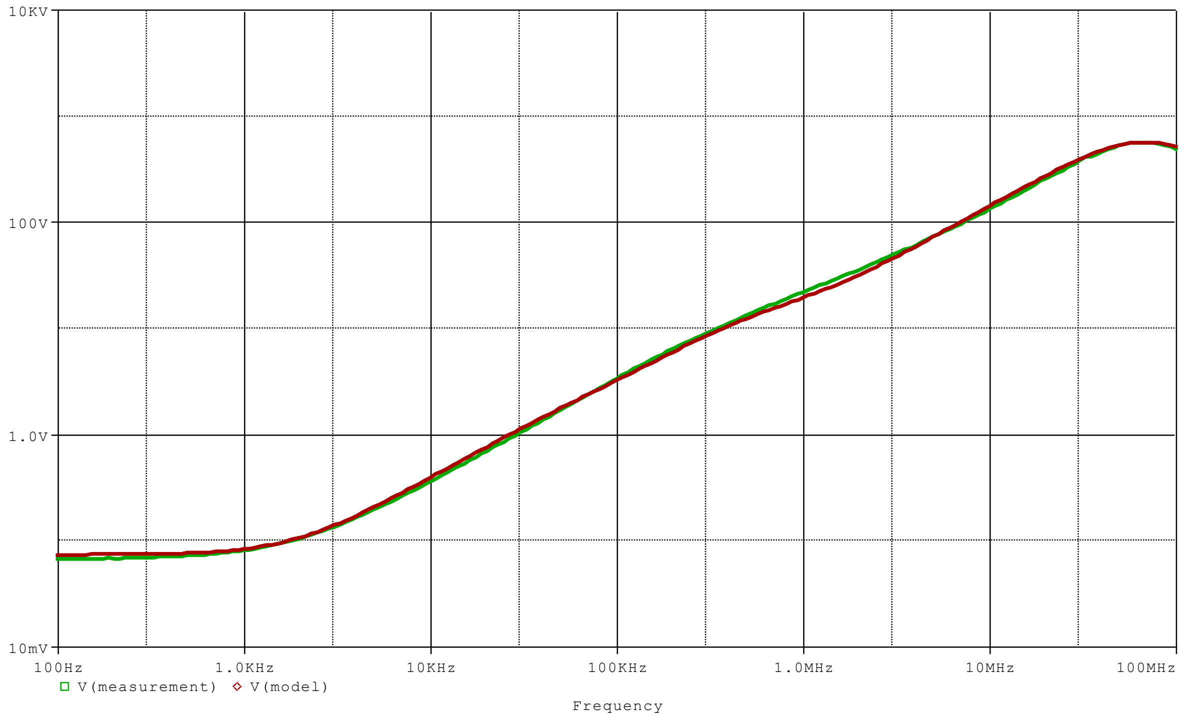


Fig. 9. The overlay between the measurement and model of the RFB1206-601KS-1-B ferrite bead now shows a good match between the two when using the Network Fit option. A SPICE netlist for the model is shown below.

```
.SUBCKT RFB1206-601KS-1-B_Network Pin1 Pin2
C0 Pin1 Pin2 2.3878383198655065E-012
R0 Pin1 Pin2 5.6682615470598114E+002
L1 Pin1 N1 4.2826285172432402E-006
R1 N1 Pin2 3.2159762445906701E+001
L2 Pin1 N2 2.3823451989151539E-005
R2 N2 Pin2 1.0737710864729273E+001
L3 Pin1 N3 6.2081969557272381E-006
R3 N3 Pin2 7.4006685040412065E-002
.ENDS RFB1206-601KS-1-B_Network
```

Parallel Capacitors Model

For parallel elements, SPICE does not represent impedance correctly given the lack of the PCB parasitics interconnecting the parts.[8] The impedance measured reveals a very different result as demonstrated by measurement of a test board with five parallel capacitors (Fig. 10). The resulting model fit also has quite high RMSRE (51.901%) and has multiple resonances above 3 MHz that are not modeled at all. We'll explore using the Network Fit Model instead.

The Network Fit Model shown below has a RMSRE of 14.289% which is improvement by a factor of 3 (Figs. 11 and 12). Take note of the Absolute DC value at 0 Hz which is 29.3362 kΩ, which is reasonable for set of five parallel capacitors. The dc value can be tweaked by changing the Target DC value. A SPICE netlist for the model is shown below.

Paralleling same value caps

	C	ESR	ESL
C1-C5	1 uF	7 mΩ	300 pH

Fig. 10. The five-parallel-capacitors test PCB.

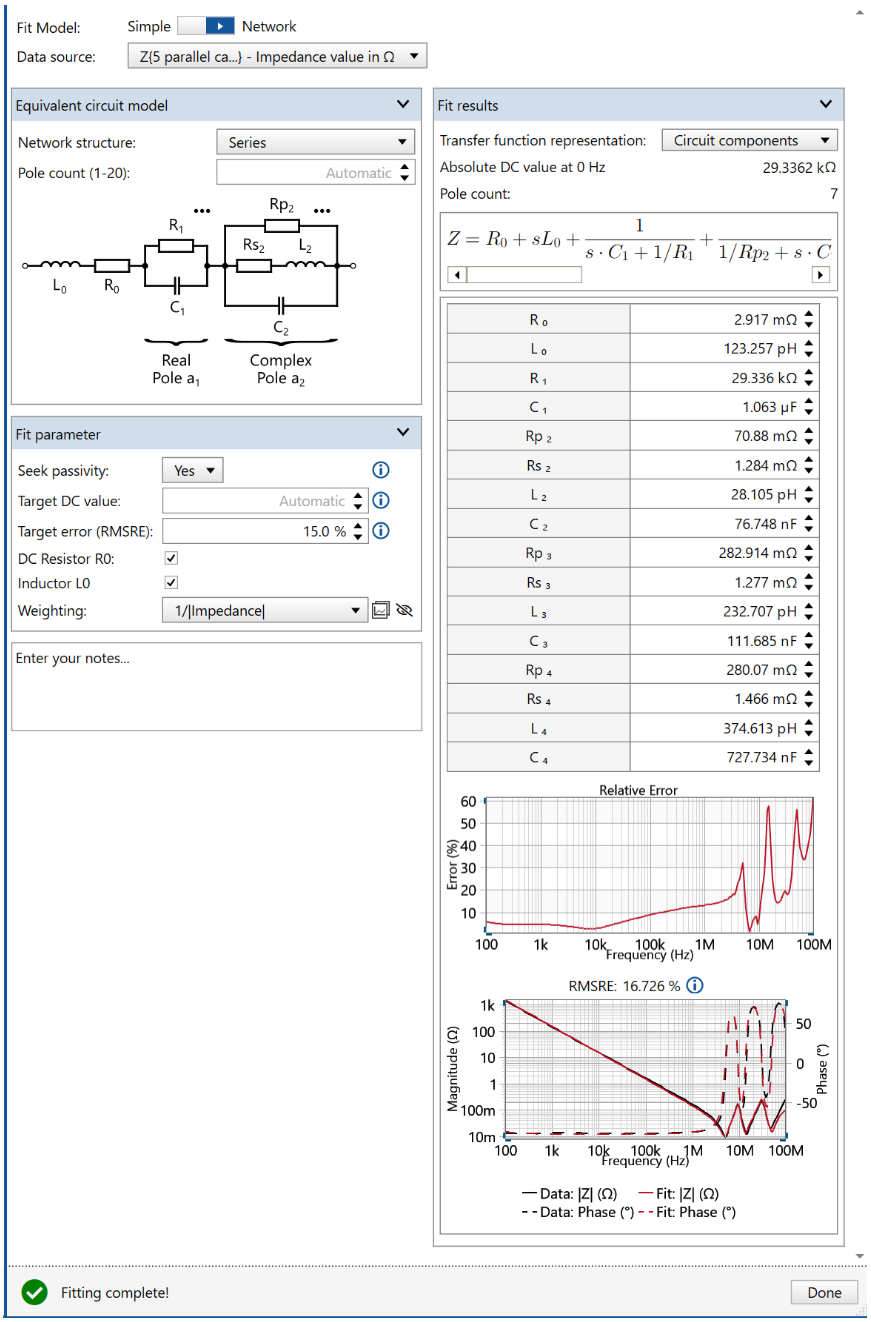


Fig. 11. The Network Fit dialog and series structure model for the five parallel capacitors.

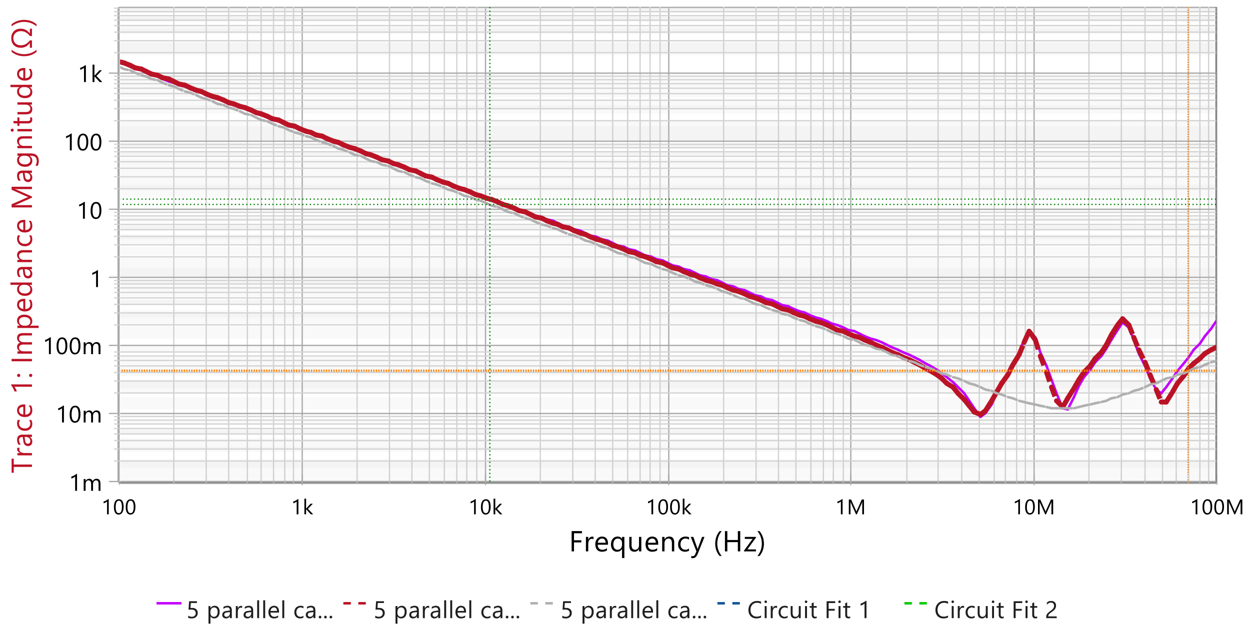


Fig. 12. The overlay between the impedance measurement and model of the five parallel capacitors shows a good match between the two (red –test data, dashed – fit, grey – simple fit).

```
.SUBCKT 5_parallel_caps_Network Pin1 Pin2
L0 Pin1 N1 1.2325681059396860E-010
R0 N1 N2 2.9170182241176398E-003
R1 N2 N3 2.9336241925574446E+004
C1 N2 N3 1.0625804898478540E-006
Rp2 N3 N4 7.0879834541606554E-002
Rs2 N3 N5 1.2844132415094219E-003
L2 N5 N4 2.8104799006930449E-011
C2 N3 N4 7.6747512256935477E-008
Rp3 N4 N6 2.8291432160441432E-001
Rs3 N4 N7 1.2770424086801365E-003
L3 N7 N6 2.3270650934333502E-010
C3 N4 N6 1.1168478213527071E-007
Rp4 N6 Pin2 2.8006996548898161E-001
Rs4 N6 N8 1.4664789153314704E-003
L4 N8 Pin2 3.7461334476868791E-010
C4 N6 Pin2 7.2773366497405673E-007
.ENDS 5_parallel_caps_Network
```

Summary

The Bode Analyzer Suite's new Curve-Fitting Simulation Model Generation feature marks a significant advancement in power integrity simulations by offering users the ability to generate precise, measurement-based models. With the Bode 100 and Bode 500 VNAs, users can now easily fit measured impedance data to SPICE models for various components such as capacitors, inductors, and ferrite beads. This article details how this feature addresses the critical issue of unreliable or outdated vendor models by enabling users to create more accurate simulations based on real-time measurements.

By using the software's Circuit Fit tool, even those with limited simulation experience can produce high-fidelity models that can enhance design accuracy and minimize convergence issues in simulation environments. Whether using simple or network fit models, this capability empowers users with greater control over model quality, which is essential for tasks like power integrity testing, component characterization, and ensuring accurate simulations across a wide frequency range. The ease of use and flexibility of the Bode Analyzer Suite in generating reliable, real-world models makes it an invaluable tool for engineers working in the field of electronics and power systems.

References

1. Bode Analyzer Suite 3.x, https://www.picotest.com/support/bode-100/
2. "This Misconception About Power Integrity Can Cost You Big" by Steve M. Sandler, How2Power Today, March 2016.
3. "SPICE Models Need Correlation to Measurements," AEi Systems website.
4. AEi Systems Component Test and Model Summary, AEi Systems website.
5. Solving SPICE Convergence Problems, AEi Systems website.
6. "How to Design for Power Integrity: Measuring, Modeling, Simulating Capacitors and Inductors" by Steve Sandler, YouTube.
7. 2-Port Shunt-Through Impedance Measurement, Picotest website.
8. "Why EM Is Replacing SPICE For Simulation Of Board-Level Power Delivery" by Heidi Barnes and Steve Sandler, How2Power Today, November 2021.

About The Authors

Charles E. Hymowitz serves as managing director of AEi Systems. Hymowitz is a technologist, marketer, and business executive with over 30 years of experience in the electrical engineering services and EDA software markets. He has been chairman and CEO of AEi Systems since 2002. He currently guides all aspects of the company's operations.

In 2012, Hymowitz was recognized as an SME (subject matter expert) on worst-case circuit analysis ("WCCA"). He was a key contributor to Aerospace Corporation's industry guidelines ("TOR") for WCCA. In 1985, Hymowitz co-founded Intusoft, a leading CAE/EDA SPICE software vendor. Additionally, he has co-authored several books on SPICE including, SPICE Circuit Handbook, Simulating with SPICE, The SPICE Cookbook, Power Specialist's App Note Book, and The SPICE Applications Handbook, plus the Intusoft EDA Newsletter. Charles has a bachelor's degree in electrical engineering from Rutgers University.

Paul Ho is a senior engineering scientist at AEi Systems where he helps to guide analog and power worst-case circuit analysis. Paul has a bachelor's degree in electrical engineering from UCLA.

Michael LaNovara is an electrical engineer at AEi Systems where he performs test and measurements for SPICE modeling and worst-case circuit analysis. Michael has a bachelor's degree in electronic systems engineering technology from Cal Poly Pomona.

For further reading on modeling of power electronics, see the How2Power Design Guide, locate the "Design Area" category and select "Modeling and Simulation".