Simulating Modelithics X-Parameter Amplifier Models in a Keysight Genesys Design Environment

by Chris DeMartino, Modelithics

Introduction

This article provides an overview of X-Parameters-based amplifier models using the Modelithics™ system-level component (SLC) Library in the Keysight Genesys circuit design environment.[1] S-parameter, noise, and harmonic-balance simulations are presented. **Table 1** lists X-Parameter models currently available for use in Keysight Genesys. In comparison to **Table 1** of **Reference 1**, the table here

for Genesys reveals fewer models. However, the same basic capabilities and theory described in **Reference 1** also apply in Genesys. An example workspace is featured to demonstrate how an X-Parameter model can be used in a Genesys project.

Frequency and Power Ranges

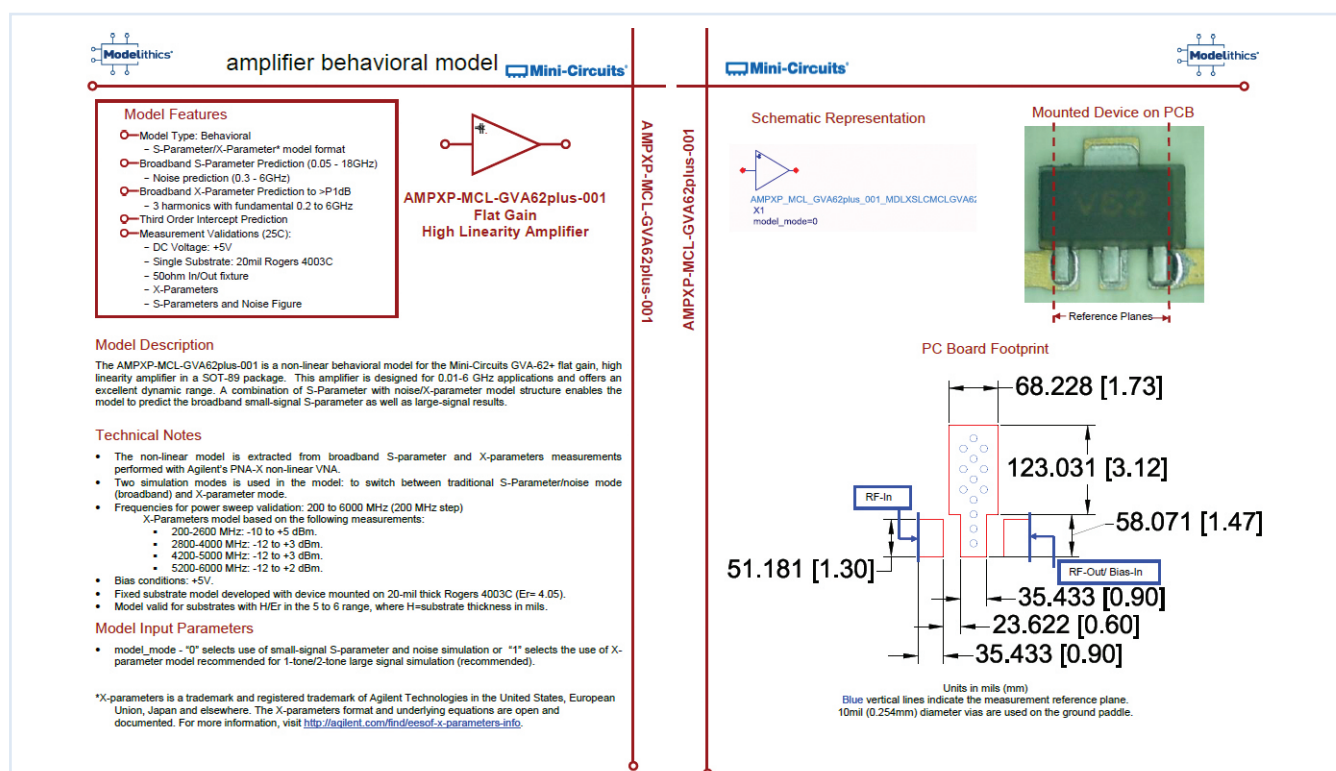When simulating Modelithics X-Parameter

Figure 1: First two pages of the 11-page model information datasheet for the GVA62+ Mini-Circuits amplifier. Frequency ranges for S-parameters, noise, and nonlinear data are 0.05 to 18 GHz, 0.3 to 6 GHz, and 0.2 to 6 GHz, respectively.

Modelithics, Con't from pg 18

models in Genesys, it is recommended to perform simulations within the measured power levels and frequency ranges used to create the model. This information is listed in the model information datasheet, which can be easily accessed within Genesys for each model (**Figure 1**). For small-signal and noise simulations (selected with the appropriate model mode), the models can be used over the full S-parameter or noise (as applicable) frequency ranges listed in the model datasheet. For harmonic balance, simulations will be valid and accurate within the range of frequencies and power levels measured for model development. Again, this information can be found in the model datasheets. While it is acceptable to simulate below the lowest validated power level, the validated ranges allow for the most accurate representation of complete linear and nonlinear behavior across multiple conditions.

Vendor	Part Number	Modelithics Part Number	Body Style
Freescale	MWE6IC9100NR1	AMPXP-FRS-MWE6IC-9100NR1-001	TO-270
Mini-Circuits	GVA62+	AMPXP-MCL-GVA62plus-001	SOT-89
Mini-Circuits	GVA63+	AMPXP-MCL-GVA63plus-001	SOT-89
Mini-Circuits	GVA84+	AMPXP-MCL-GVA84plus-001	SOT-89
Mini-Circuits	PGA-102+	AMPXP-MCL-PGA102plus-001	SOT-89
Mini-Circuits	PGA-103+	AMPXP-MCL-PGA103plus-001	SOT-89
Mini-Circuits	PGA-105+	AMPXP-MCL-PGA105plus-001	SOT-89
Mini-Circuits	PHA-1+	AMPXP-MCL-PHA1plus-001	SOT-89
Mini-Circuits	PHA-22+	AMPXP-MCL-PHA22plus-001	DL1020
Mini-Circuits	PSA4-5043+	AMPXP-MCL-PSA4-5043plus-001	SOT-343
Qorvo	AH101	AMPXP-TQT-AH101-001	SOT-89
Qorvo	RF2878	AMPXP-RFMD-RF2878-001-S	Package

Table 1: Currently available X-Parameter models as included in the Modelithics COMPLETE Library for Genesys

Biased Models

Some of the Modelithics X-Parameter amplifier models feature multiple voltage conditions at which they have been validated. One can consult the model datasheets for information concerning available voltage biases. Models that include the voltage (Vcc) parameter allow one to select the separate bias setting.

Substrate Scalable Models

Many Modelithics X-Parameter amplifier models scale with substrate alterations. Models with substrate scaling include an H and Er user parameter set. These parameters are to set the substrate height and the average permittivity before simulation. Each model has been measured under specific substrate constraints and remains reliable within those ranges. It is not recommended to simulate with substrate settings beyond the validated range of the model, as doing so can produce unreliable results. It is recommended to consult the model datasheets for more information on substrate-scaling ranges when available.

Loading the Example Workspace

Included in the installation of the Modelithics COMPLETE Library for Keysight Genesys is a set of examples, including several that feature X-Parameter models. Modelithics example projects are installed in the C:\Modelithics\Examples – for Keysight Genesys folder.

After starting Genesys, one can open a project and navigate to the Genesys SLC examples included within the Modelithics installation. These examples are found in the C:\Modelithics\Examples - for Keysight Genesys\SLC folder. Here, the project used is AMPXP_MCL_GVA84plus_001_Ex.wsg.

Once loaded, the project displays three primary folders in the workspace tree: Designs, Analysis, and Results (**Figure 2**). Each segment of the example is contained within these folders. These folders are in place for organizational purposes only and are not required in a typical Genesys project setup.

Expanding the Designs folder makes it possible to see all of the design schematics created for each section of the X-Parameter model (**Figure 3**). Opening the S-parameters schematic reveals a premade small-signal circuit with the GVA84+ situated as the device under test (DUT). In this example schematic, the model's model_mode parameter is set to Small Signal. The frequencies being swept are taken from the model datasheet and correspond to the
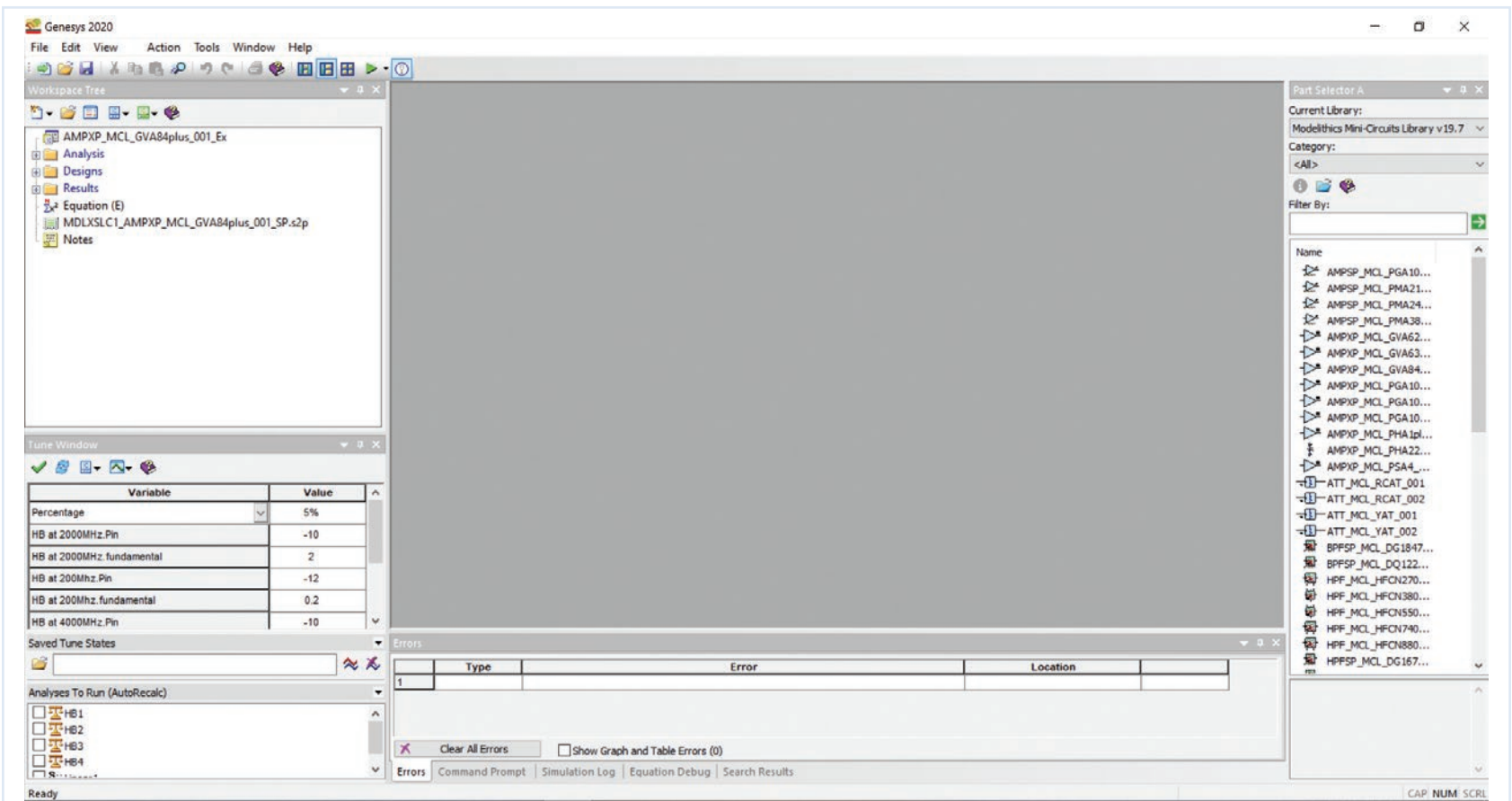
Figure 2: This is the user interface upon opening the AMPXP_MCL_GVA84plus_001_Ex.wsg project

maximum valid frequency for the substrate in which it was measured. For this example, the GVA84+ has been validated from 0.05 to 12 GHz and from 0.5 to 6 GHz for S-parameters and noise parameters, respectively.

Now that the simulation design has been reviewed, the analysis portion of the project can be described. Located under the Analysis folder in the workspace tree is the SPsim subfolder, which represents the linear analysis. The information here includes the frequency range, temperature, and step size (**Figure 4**). Noise parameters can be turned on or off. If turned on, they will be calculated within the same analysis as the S-parameters.

If desired, the project can allow for additional linear analysis using the same schematic. Each simulation is separate from the other and can be run in conjunction with the original. To add an additional sweep, right-click on the desired destination folder in the workspace tree. From the subsequent menu, select Add>Analyses>Linear Analysis.

After simulating, the results for this example can be found in the Small Signal subfolder within the Results folder. Included here are premade plots to display the results of both S-parameter and noise simulations.

Nonlinear Simulations

Included in the example are harmonic-balance simulations for a range of frequencies in which the model was tested. The corresponding plots for these sche-
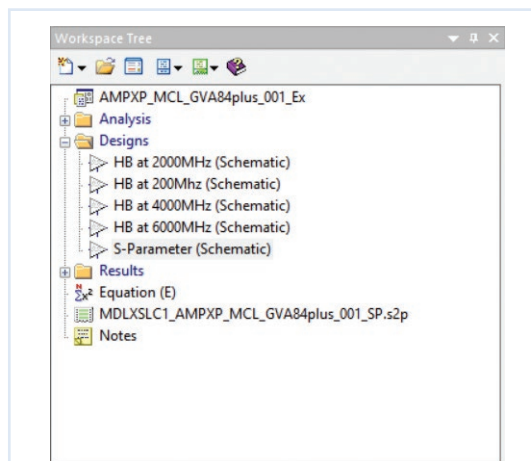

Figure 3: Genesys workspace tree with Designs folder expanded to show the available schematics in the example project
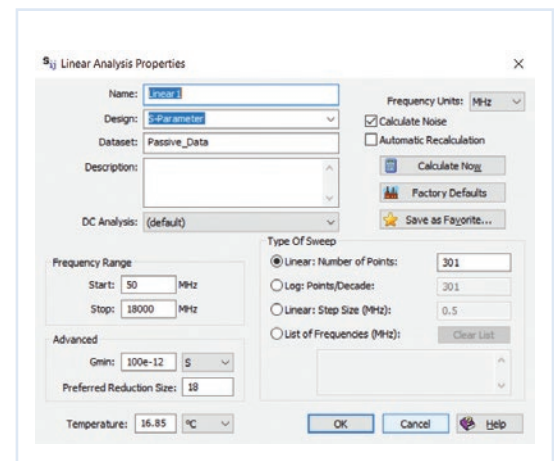

Figure 4: The Linear Analysis Properties window with pre-populated model information
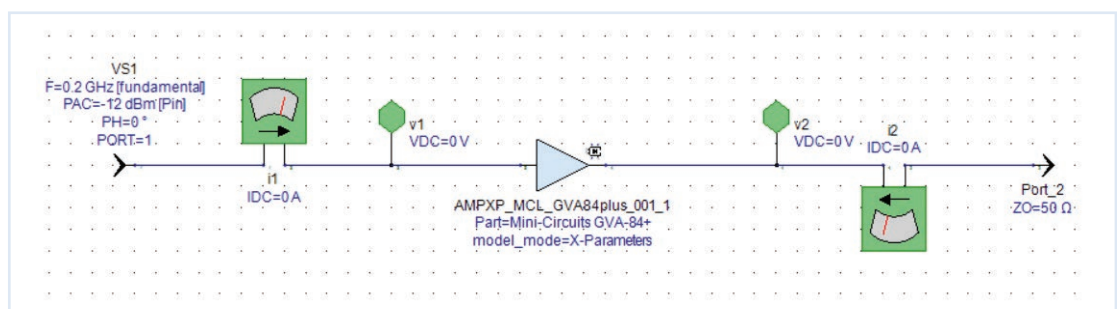

Figure 5: Harmonic Balance simulation schematic using the Modelithics AMPXP-MCL-GVA84plus-001 model

matics can be found in the model datasheets.

For this example, the HB at 200 MHz schematic in the Designs folder is examined (**Figure 5**). While this schematic is similar to the S-parameters schematic, an AC power port component is placed at the input rather than a Standard version. Note that the frequency and power level being used can be found in the Technical Notes section of the model's datasheet. It is important to note that these parameters are tied to equations, which allows

the parameter to be swept over a range of values rather than being set to a static value. For our purposes, the input port is preset to a fundamental frequency of 200 MHz with an initial power level of −12 dBm.

The first step in analyzing this schematic involves a Harmonic Balance analysis (**Figure 6**). A Harmonic Balance analysis can be added by selecting Add>Analyses>Harmonic Balance. For this example, this analysis is already configured. The analysis should be based on the schematic that will be simulated (in this case HB at 200 MHz).

To correctly simulate a harmonic balance schematic, an evaluation sweep must be implemented. This sweep must correlate with the harmonic evaluation just described (**Figure 7**). For this workspace, the input power (Pin) parameter for the valid range of the model at 200 MHz is already specified as the parameter to be swept. This parameter, displayed as Parameter to Sweep in the user interface, can be set to any equation in the schematic. When configuring a custom sweep, ensure the parameter being swept is part of the specific schematic being referenced in the Harmonic Balance analysis. To set up additional evaluations, right-click on the folder in the workspace tree and select Add>Evaluations>Sweep.

For the 200 MHz simulation, the sweep range is preset to a range of −12 to −1 dBm with a 0.5 dB step size. After running an analysis, the results of the evaluation sweep can be plotted. Gain and output power spectrums are created to match the plots found in the datasheet. This example includes premade plots to display forward transmission gains, forward transmission phase, and the first three orders of output power and phase (**Figure 8**). Additional plots can be created from the swept analysis.
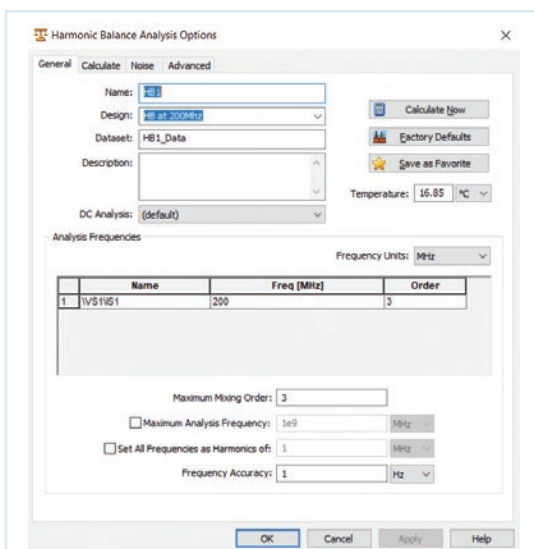

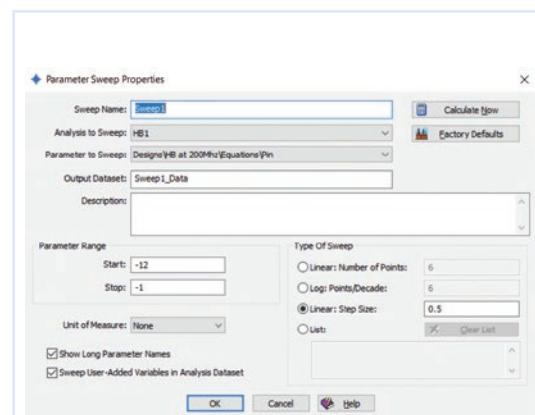**Figure 6: Harmonic Balance analysis window for the HB at 200 MHz schematic**

Figure 7: Parameter Sweep Properties window. Used in conjunction with Harmonic Balance analysis to enable sweeping of various parameters like input power and frequency.
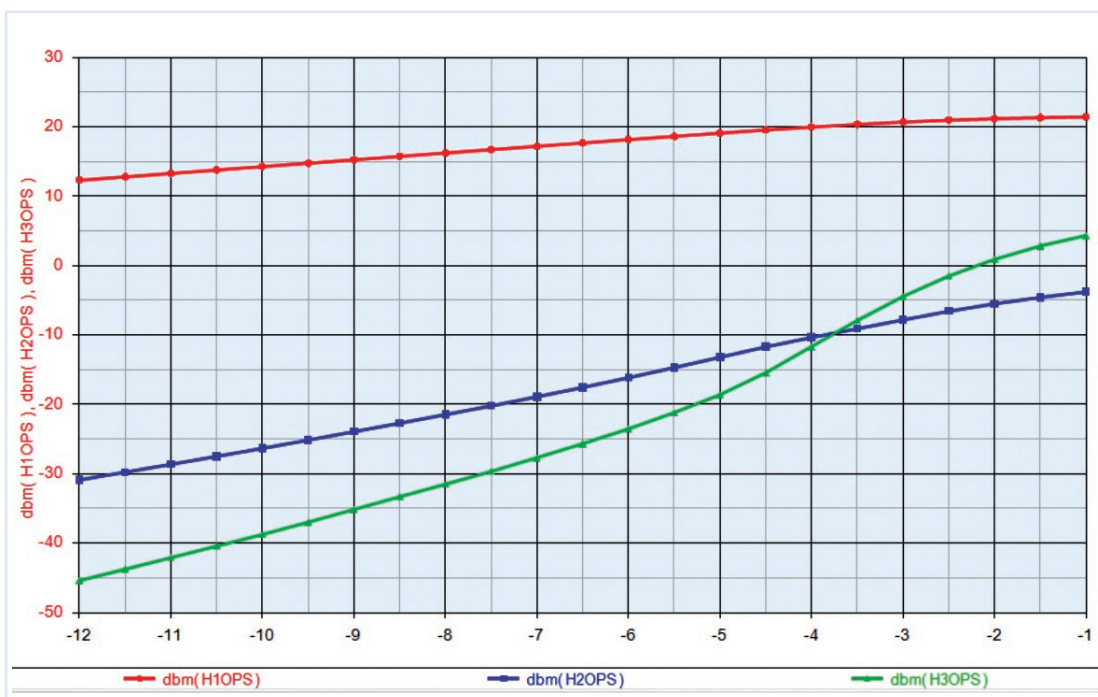

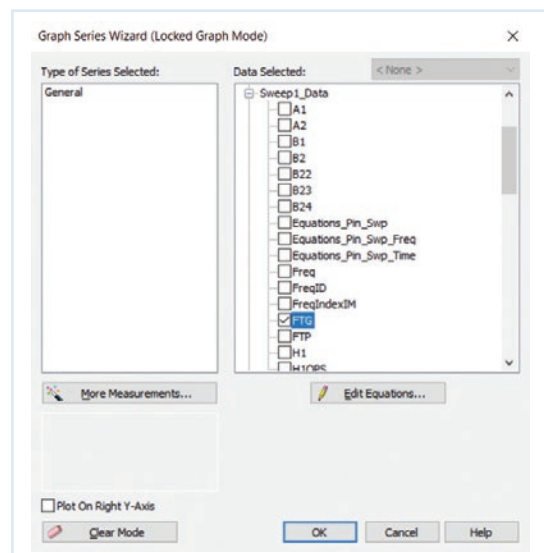**Figure 8: Output power spectrum for three orders. Values are in dBm.**

Figure 9 - Selection of forward transmission gain (FTG) from the Sweep1_Data file in the Graph Series Wizard

A results graph can be added by right-clicking in the workspace tree and selecting Add>Graph. Clicking the Edit or Add buttons on the left side of the subsequent Graph Properties window will open the Graph Series Wizard window (**Figure 9**). The appropriate dataset in the right menu area can then be expanded to view various parameters that can be plotted.

Summary

Modelithics X-Parameter models provide Keysight Genesys users with advanced features that can help facilitate design projects. S-parameter, noise, and harmonic balance simulations are all possible with a single model. In addition, every model comes with its own detailed datasheet to provide designers with the necessary documentation. Lastly, several Genesys workspace examples are offered by Modelithics to further aid those who wish to employ X-Parameter models in their designs.

Acknowledgements

References

[1] "Advanced Microwave Amplifier Models and X-Parameter Simulation Setup Examples for Advanced Design System Simulation," Modelithics Application Note 059.

• MODELITHICS •