Green-Williams-Lis: Improved op amp spice model

Tamara Alani, Ian Williams

What is a SPICE Model?

Simulation program with integrated circuit emphasis (SPICE) is a powerful circuit design and simulation tool that is used for designing, testing, and troubleshooting a broad variety of basic and advanced circuits. A device model is a SPICE circuit component that resembles a real-world electronic device. The model characteristics and specifications are uniquely programmed to mimic the lab-tested results of the physical device.

Engineers use device models with a SPICE simulator to evaluate performance of new devices, sometimes even before they are released to the market, from the comfort of any computer. After the devices are selected, customers may then create their board- and system-level designs with the confidence that their simulation results will match closely to their real-world circuit measurements. Finally, applications engineers also use these simulations to test and solve issues in their customer's designs.

A primary benefit of simulations includes not waiting for ordered parts, PCBs, or lab gear in order to quickly examine an issue. This benefit makes verifying circuit connections and functions easy.

All SPICE Simulators are not Created Equal

TI's preferred simulator, TINA-TI, uses advanced algorithms for quick simulation convergence. Other SPICE simulators may use older algorithms and can struggle to achieve convergence in complex circuits. Semiconductor manufacturers must provide SPICE models that perform reliably, no matter the simulator.

TI SPICE Models in the Past

Before 2007, TI op amp models were either transistor-based (complex to design and converge), or hybrid (a combination of transistor and behavioral architecture). Some of these older models had inaccuracies compared to real-world devices.

In 2007, Tim Green and Marek Lis, applications engineers at TI, released their new SPICE model architecture, the Green-Lis (GL) model. This architecture was behavioral and modular, allowing for fast and accurate modeling of key op amp data sheet parameters. Most GL models worked well with TINA-TI, but some struggled in other simulators.

TI SPICE Models in the Present

The next generation SPICE model, introduced in 2016, is the Green-Williams-Lis (GWL) architecture. Developed by Ian Williams, an applications engineer at TI, this new model improves upon the GL architecture across three main categories:

1. Compatibility across simulators
2. Speed of convergence
3. accuracy of modeling datasheet parameters

GWL models are fully tested and verified for electrical accuracy and convergence performance in all PSpice-based simulators. This verification allows customers and designers to get identical results and great performance with any SPICE software. New features are added on a regular basis to continually improve accuracy versus the real world.

Which Specifications Does TI Model?

Amplifier SPICE models are designed to target typical electrical characteristics at room temperature, in order to give a good overall representation of the device performance. Typical specifications cover 68.3% of all devices produced, or ±1-sigma over a normal Gaussian distribution.

GWL models allow a designer to simulate the following parameters and many more:

- Input offset voltage (V_{OS})
- Input bias current (I_B)
- Input voltage noise density (e_n)
- Open-loop gain and phase (A_{OL})
- Output impedance (Z_O)
- Common-mode rejection (CMR)
- Power-supply rejection (PSR)

GWL model enhancements are continually in progress to help designers pass first-PCB through simulation alone. The following section shows how GWL models compare to tested datasheet curves, as well as competitor models.

GWL Example and Comparison

Figure 1 shows the curve for open-loop output impedance (Z_O) vs frequency of the OPA2189, the industry's highest bandwidth (14 MHz) Zero-Drift, MUX-Friendly op amp with only 1.5 µV of input offset voltage, and 0.005 µV/°C of drift. With the new GWL model, you can simulate this characteristic, as shown in Figure 2, and prove that the model matches the real device.

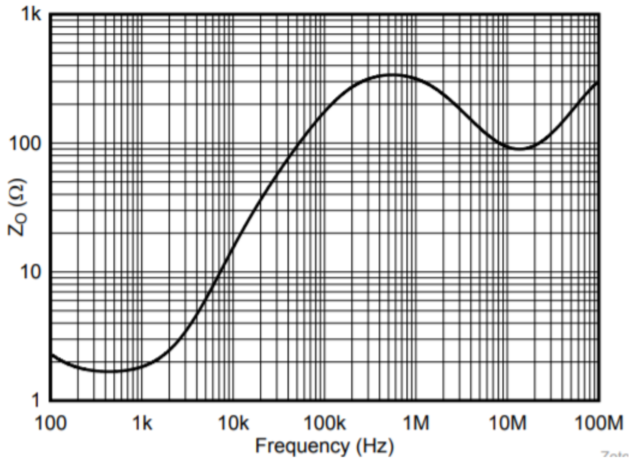

Figure 1. OPA2189 Z_O Curve From the Data Sheet

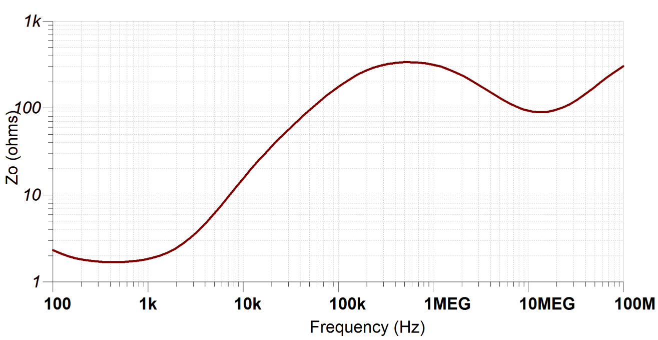

Figure 2. OPA2189 Z_O GWL Simulation Results

Other semiconductor companies also release their devices with models. However, in many cases, the model does not accurately represent the real device. When a datasheet curve is compared to the model simulation, you can see noticeably different results. Figure 3 shows the data sheet curve of closed-loop output impedance from a competitor's op amp. Figure 4 shows the model simulation results for the same device. The curves do not align, so the model is ultimately not useful for small-signal stability analysis or driving dynamic loads. For crucial designs, it is important to verify that your models match the specs promised by the data sheets. To read more about designing with a complete simulation test bench for op amps, the following link goes to a multi-part article series written by Ian Williams: https://www.edn.com/design/analog/4460430/The-complete-simulation-test-bench-for-op-amps--Part-1--Output-impedance.

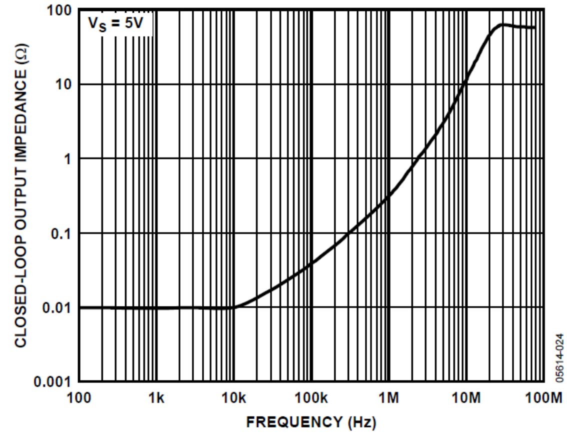

Figure 3. Competitor Closed-Loop Output Impedance Curve From Data Sheet

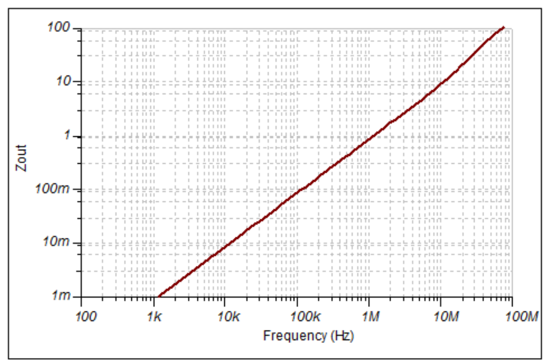

Figure 4. Competitor Model Simulation of Closed-Loop Output Impedance

Which Devices Have This New Model?

All new precision amplifier models released after June 2016 use GWL architecture. Older op amp models are regularly updated to GWL. To check if a model is GWL, download the model from TI's op amp product folder, and check the netlist for the characters *GWL*.

Follow Table 1 for a list of our newest GWL op amps. Find more TI op amps at ti.com/amplifier-circuit/op-amps/products

Table 1. TI's Newest GWL Op Amps

Device	Description
OPA2210	36-V, 18-MHz, 5-µV offset, 2.2-nV/$\sqrt{\text{Hz}}$ noise, RRO
OPA2156	36-V, 25-MHz, 25-µV offset, 4.3-nV/$\sqrt{\text{Hz}}$ noise, RRIO
OPA828	36-V, 50-MHz, 50-µV offset, 4-nV/$\sqrt{\text{Hz}}$ noise, RRO
OPA2189	36-V, 14-MHz, 0.4-µV offset, 5.2-nV/$\sqrt{\text{Hz}}$ noise, RRO
OPA388	5-V, 10-MHz, 0.25-µV offset, 7-nV/$\sqrt{\text{Hz}}$ noise, RRIO

IMPORTANT NOTICE AND DISCLAIMER

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265