

Operational Amplifiers Part VI of VI Amplifiers and the SPICE of Life

by Bonnie C. Baker

Microchip Technology, Inc. bonnie.baker@microchip.com

What is operational amplifier circuit stability and how do you know when you are on the hairy edge? Typically, op amps are used with a feedback network in order to reduce the variability of the open-loop gain response from part-to-part. With this technique circuit stability is provided. But it is possible to design an amplifier circuit that does quite the opposite. You can design an amplifier circuit that is extremely unstable to the point of oscillation. In a closed-loop amplifier system stability can be determined if the phase margin of the system is known. The Bode stability analysis technique is commonly used. (See Part V at <http://www.analogZONE.com/acqt1025.htm>) With this technique the magnitude (in dB) and phase response (in °) of both the open-loop response of the amplifier and circuit feedback factor are included in a Bode plot.

They say a computer-based simulation of your analog circuit is important. This is because the use of your preferred computer SPICE (Simulation Program with Integrated Circuit Emphasis) program can reduce initial errors and development time. If you use your SPICE simulator correctly you can drum out circuit errors and nuances before you go to your breadboard. In this manner you will verify your design before you spend the time to solder your circuit. SPICE helps troubleshoot on the bench and it is a great place to try out different hypotheses. It is great at "what if" scenarios (ie exploratory design).

You can view the results from these software tools on a PC with user-friendly GUI suites. This tool will fundamentally provide dc operating (quiescent) points, small-signal (ac) gain, time domain behavior, and dc sweeps. At a more sophisticated level it will help you analyze harmonic distortion, noise power, gain sensitivity, and perform pole-zero searches. This list is not complete, but generally SPICE software manufacturers have many of these fundamental features available for the user. By finessing the Monte Carlo and worst-case-analysis tools in SPICE, you can predict the yields of your final product. If you use your breadboard for this type of investigation it could be very expensive and time consuming. All of these things will speed up your application circuit time-to-market.

But, beware. You can effectively evaluate analog products if your SPICE models or macromodels are accurate enough for your application. The key words here are "accurate enough." Such models, or macromodels, should reflect the actual performance of the component, without carrying the burden of too many circuit details. Too many details can lead to convergence problems and extremely long simulation times. Not enough details can hide some of the intricacies of your circuit's performance. Worse yet, your simulation, whether you use complete models or just macromodels, may give you a misrepresentation of what your circuit will really do. Remember that a SPICE simulation is simply a pile of mathematical equations that, hopefully, represent what your circuit will do. It is in essence a computer product that produces imaginary results.

So you might ask, "why bother?" Are SPICE simulations worth the time and effort? A pop quiz will help you clarify this question. The circuit in Fig. 1 shows a fundamental, basic circuit. Is it stable or does it oscillate? Would the output of the amplifier have an unacceptable ring? I would think that you would quickly look at this and say, "That is a silly question. Of course it is stable!" But then, again, if you are always looking for the trick question you may be suspicious. So what is the answer?

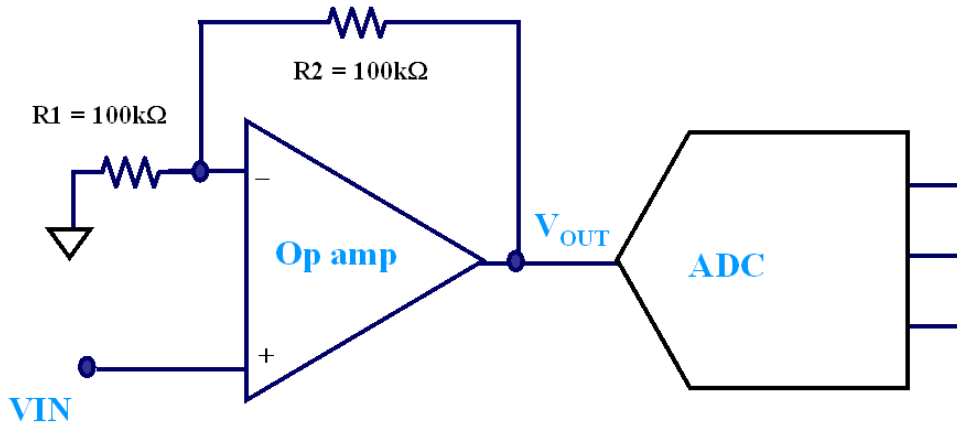


Fig. 1: Many Applications Use This Simple Embedded Sub-Circuit

This simple amplifier circuit takes an analog input and gains it +2 V/V (so a dc signal input of +0.5 V to +1.0 V would output +1.0 V to +2.0 V dc). Would it oscillate? And with a bandwidth of 2.8 MHz would a 50-kHz signal oscillate or ring? The amplifier has a 100-k Ω resistor connected to its inverting-input to ground and a 100-k Ω resistor in the feedback loop. It would be easy to assume that this is stable but tedious calculations will verify that this circuit will ring. This is due to the parasitic capacitances around the resistors and the high differential/common-mode capacitance (3 pF and 6 pF) of the input stage (Fig. 2): they interact with the feedback resistor causing a semi-unstable condition, easily seen on a scope in a bench test. Breadboard parasitics will aggravate the instability.

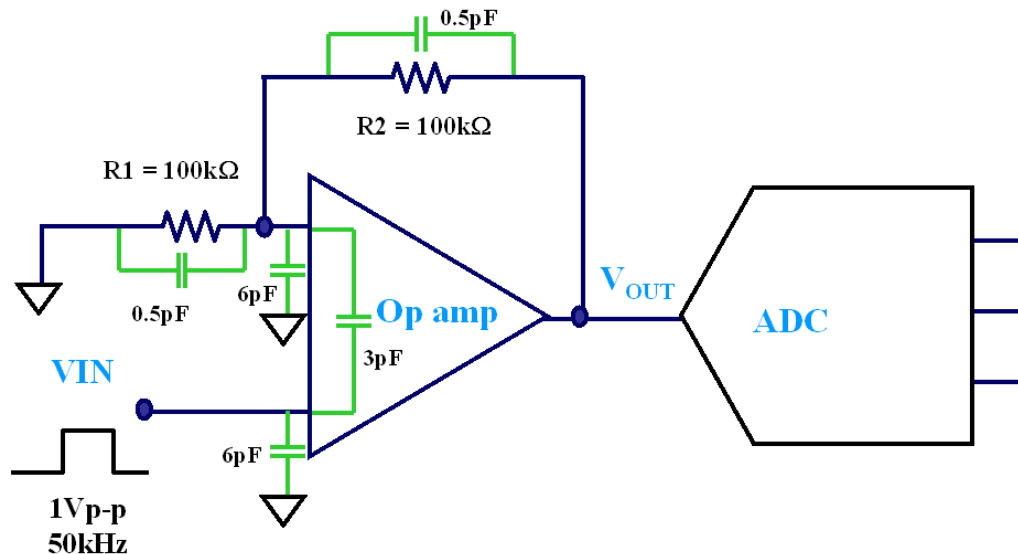


Fig. 2: Enhancing the Circuit in Fig.1 -- Not So Simple, Now

Enhancing the circuit in Fig. 1 with the parasitic capacitances of the resistors and amplifier, things aren't as simple. The 100 k Ω resistor in the feedback loop will have a parallel 0.5 pF (approximately) and the parasitic to ground could be as high as 2 pF or 3 pF. In the dc domain these capacitors will operate as open circuits. In the ac domain the capacitors will affect the perfect square wave from input to output and there will be quite a ring at the V_{OUT} node. If you use the amplifier's SPICE macromodel, with input impedances in the model and board parasitics, you will see this problem immediately in your simulation. If you bread board the circuit, you most certainly will see this ringing.

Changing the values of the two resistors in this circuit solves this problem. Hand calculations will help you find the correct values. A SPICE simulation will facilitate the process. This is a little easier than swapping out resistors on the breadboard until your find the right values. In SPICE, you can also look at the response of the amplifier using various resistors. This will help you find the "corner" of this oscillation. If you go back and change both values to 10 k Ω you will have great success in SPICE and on the bench.

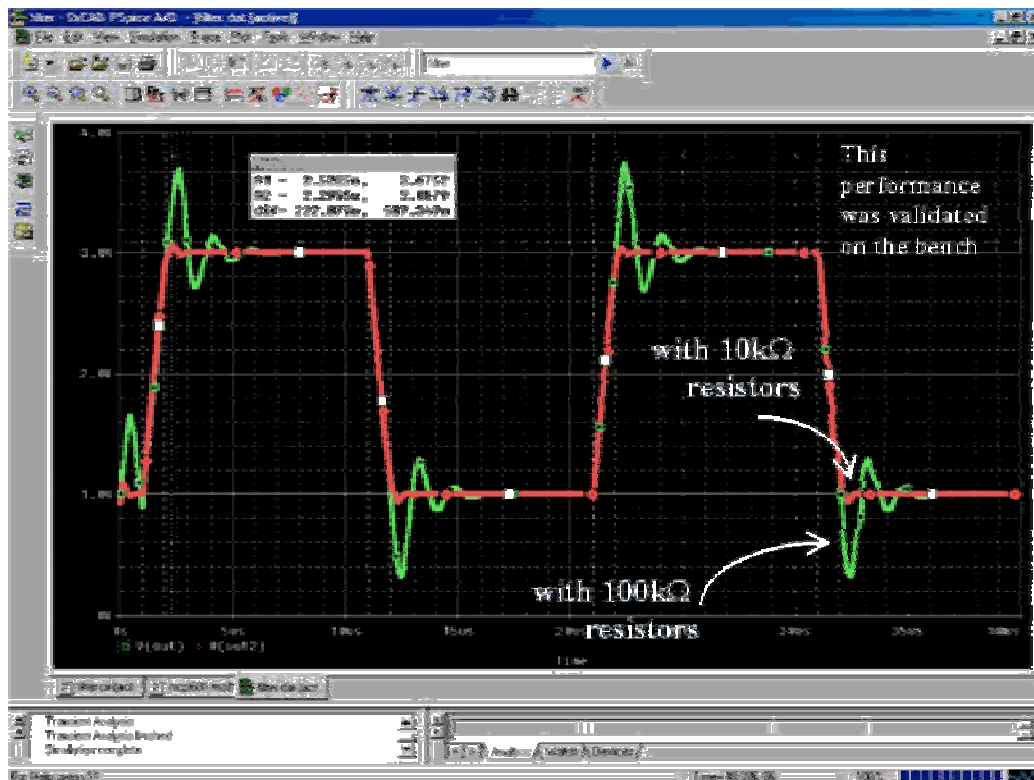


Fig. 3: Result of Simulation For Both Resistor Values

You do need to understand where the problem came from before you continue with your circuit design. But the simulation caught a significant stability problem which was an easy one to miss by just an inspection of the schematic.

The naysayer in the industry will tell you your computer-based simulation tools will not work and using them will be a waste of time. These people are a bit misguided and, in my

opinion, they have a superficial view of what this tool can really do. Sure, SPICE tools can lead you astray. But like any tool, it is only as good as the user. Any insight that you gain from simulations emerges if you understand and use you SPICE tools properly. Better yet, SPICE simulations will point out problems that you had never anticipated. In most cases, they use double precision calculations. This makes it easier to detect low-level problems that are impossible to find on the bench. SPICE simulations of analog circuits is just one more way to avoid the design of a "singing" circuit when you actually want stability.

This TechNote is excerpted from Bonnie Baker's book, "A BAKER'S DOZEN: Real World Solutions to Real World Analog Design Problems," to be published in the Spring of 2005 by Elsevier. ISBN 0-7506-7819-4

