

AN-1516 Pspice Universal Test Circuits

Soufiane Bendaoud

ABSTRACT

This application report provides a collection of circuits that allow users to test any op amp model, in a way somewhat universal test circuits for op amp macro models.

Contents

1	Introduction	2
2	What Parameters Should Be Tested?	2
3	Open Loop Gain And Phase Margin	2
4	Slew Rate	4
5	CMRR and PSRR.....	5
6	Closed Loop Output Impedance	6
7	Voltage And Current Noise	8
8	Input Bias Current and Input Offset Voltage.....	11
9	Output Saturation Voltage.....	11
10	Supply Current vs. Supply Voltage	13
11	Overshoot And Transient Response.....	14
12	Common Mode Voltage Range.....	15
13	Phase Reversal.....	17
14	Conclusion	17

List of Figures

1	Simulated Open Loop Gain and Phase.....	3
2	Measured Open Loop Gain and Phase	3
3	Simulated CMRR Response vs. Frequency.....	5
4	Measured CMRR vs. Frequency.....	5
5	Simulated PSRR Response vs. Frequency	6
6	Measured PSRR vs. Frequency	6

1 Introduction

There is no doubt that Spice models have gained a lot of popularity over the past years. While IC manufacturers strive to provide their customers with accurate models, it is really the system designer who dictates the trend of this accuracy and the innovation in the development of Spice macro models.

Many IC companies praise and brag about how their models are the best or offer revolutionary features but what they often fail to provide is some sort of circuit that allows the user or their customer to verify the accuracy of the macro model. Op amps macro models are probably the most sought after and while they can be extremely helpful when accurate, they can also cause serious problems, especially when they are not in the hands of experts.

Most system design engineers take the time to test the op amp macro model by itself before implementing it in a more comprehensive circuit. Unfortunately, sometimes use of non-accurate models lead designers and users to think that the amplifier is faulty, when the actual problem lies either in the model itself or in not properly setting the SPICE test environment. The truth is, all models are not the same and some may not work in a particular setting.

2 What Parameters Should Be Tested?

Macro models differ in the level of complexity. Much like datasheets, the models should emulate parameters that are relevant to applications in which the op amp is thought to be appropriate. For example if a rail to rail output op amp is used, then the user should be able to test and verify the output saturation voltage versus the load current. Likewise, a low noise amplifier should have a model that emulates at least the voltage noise among other modeled parameters.

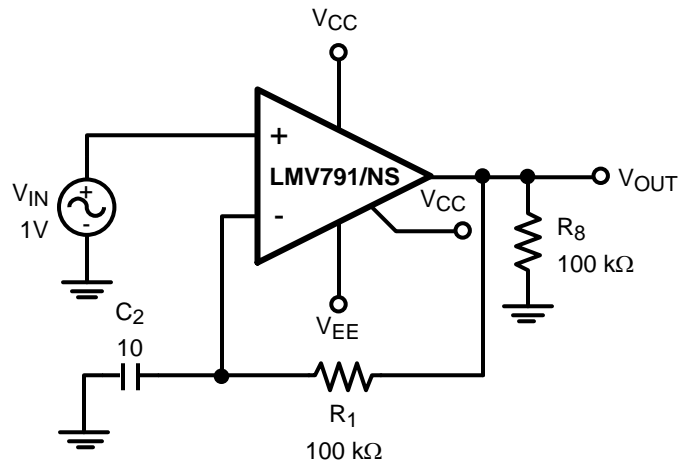
Despite their differences, amplifier macro models have a lot in common; these parameters are of the greatest interest and they are usually the starting point of the simulation. Below is a list of these parameters along with the corresponding test circuits and the simulations.

3 Open Loop Gain And Phase Margin

The open loop gain versus frequency is probably the very first test that engineers perform to evaluate the amplifier's macro model performance. This test is important because it shows the DC gain, the -3 dB frequency, the unity gain bandwidth and the phase margin. [Open Loop Gain and Phase Test Circuit](#) shows the test circuit. The RC network ensures that the output is biased at a suitable DC voltage, (mid rail in this case). At higher frequencies, the capacitor shorts the inverting input to ground which, in turn places the op amp in open loop. The capacitor is chosen to be large to provide an early roll off ($f = 1/2\pi R_1 C_2$) so that even if the op amp tested has a very low frequency dominant pole, the simulation shows a smooth transition and 20 dB per decade roll off.

When testing open loop gain and phase, the user should choose an upper frequency limit that goes beyond the unity gain bandwidth of the amplifier.

When using rail to rail output models, it is important to use the test circuit with the same load indicated in the datasheet, otherwise the result might not reflect the actual amplifier's capabilities. This is especially true about the DC gain ($A_{OL} = g_m R_L$).



Open Loop Gain and Phase Test Circuit

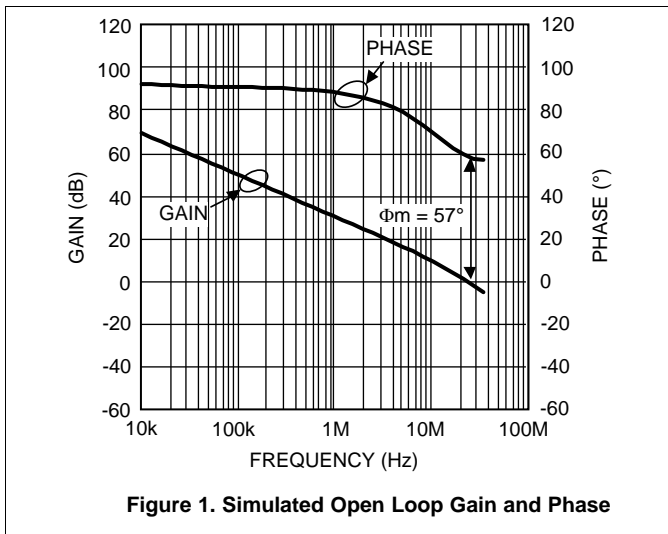


Figure 1. Simulated Open Loop Gain and Phase

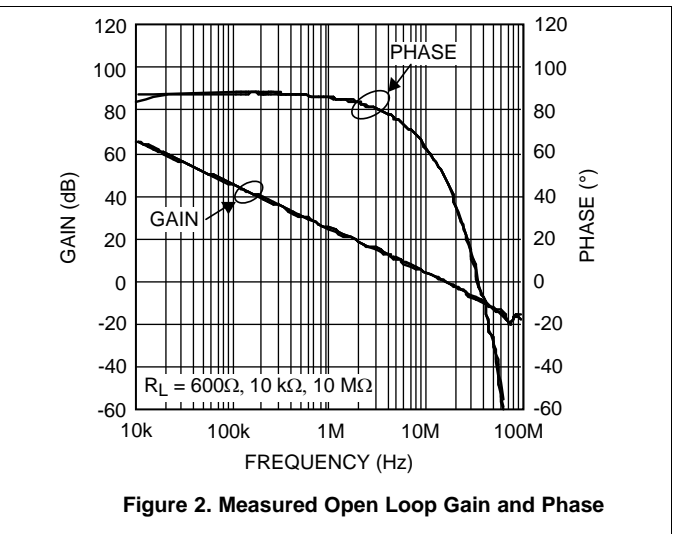


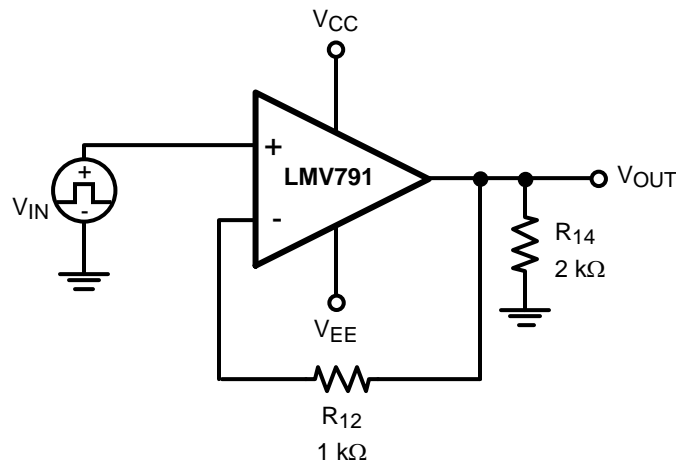
Figure 2. Measured Open Loop Gain and Phase

4 Slew Rate

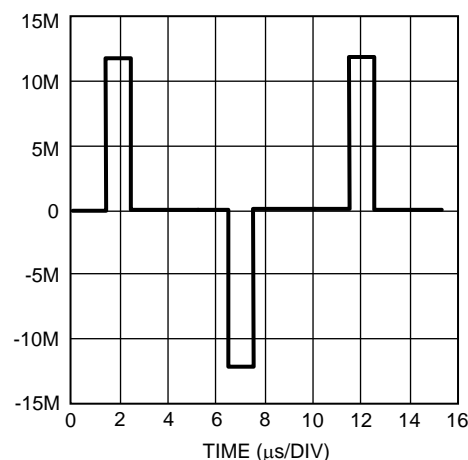
This is another element that defines the amplifier's speed and is frequently modeled. Slew rate is usually determined by the ratio of the tail current and the compensation capacitance (I/C).

Since we already know the relationship $I \cdot dt = C \cdot dv$, we can simply use the circuit in [Slew Rate Test Circuit](#) and take the derivative of the output to get the slew rate. Use the insert command in the probe screen of Pspice to add the letter "d" preceding the output voltage probe.

To assure this test circuit works properly, the input step function should have an amplitude large enough so that the effects of slew rate limitation are visible. When running the simulation for slew rate, make sure the input signal rise and fall times are shorter than the amplifier's expected slew rate. This is to ensure that the test results are dominated by the amplifier's slew rate. On the other hand, choose the input signal frequency accordingly with the op amp's speed. An input signal that's too fast will give you convergence problems.



Slew Rate Test Circuit



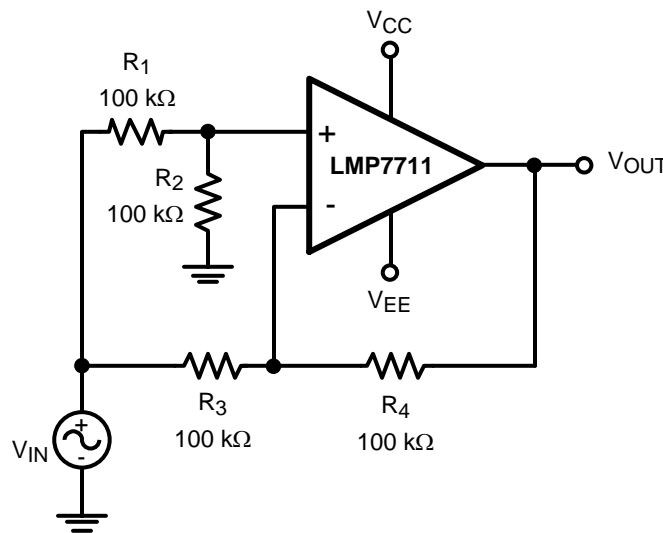
Simulated Slew Rate

5 CMRR and PSRR

These two parameters are not always modeled but they can be equally important. CMRR and PSRR are fairly easy to implement in a model as they usually consist of a simple RC network, a resistor divider and a voltage controlled voltage source.

CMRR is especially important in non inverting configurations because of the modulation of the non-inverting input with the input signal. PSRR on the other hand is important in any application where the voltage supply is susceptible to any interference or for DC PSRR where the supplies can experience significant variation.

The test circuits presented in [CMRR Test Circuit](#) and [PSRR Test Circuit](#) allow the user to simulate these two parameters. If they are modeled correctly, the pole and zero location should match the graphs in the datasheet.



CMRR Test Circuit

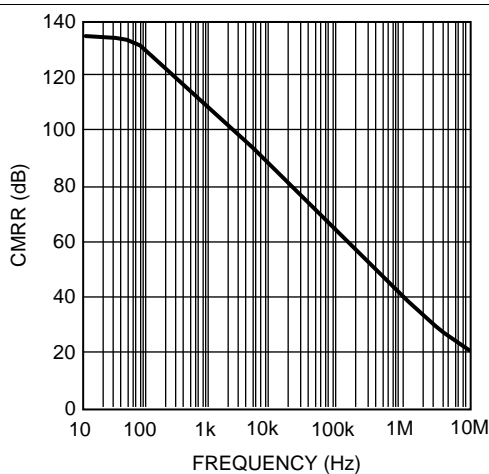


Figure 3. Simulated CMRR Response vs. Frequency

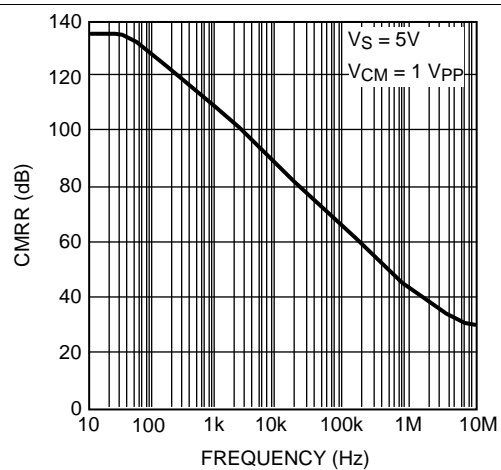
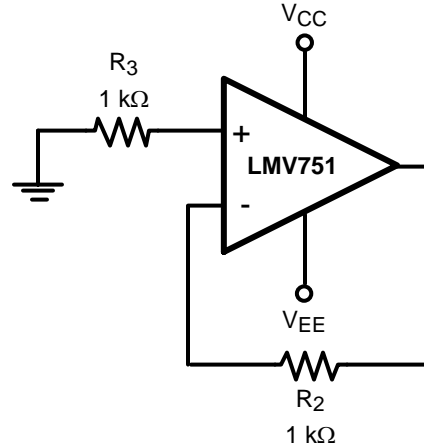
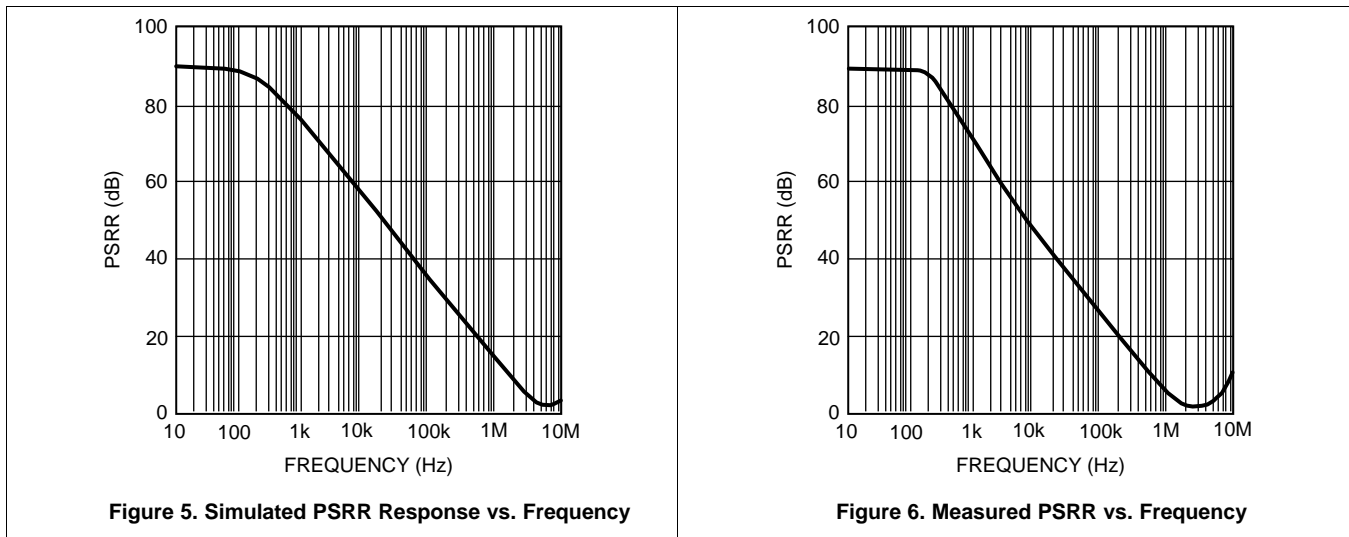


Figure 4. Measured CMRR vs. Frequency

CMRR Test Circuit (continued)

PSRR Test Circuit


6 Closed Loop Output Impedance

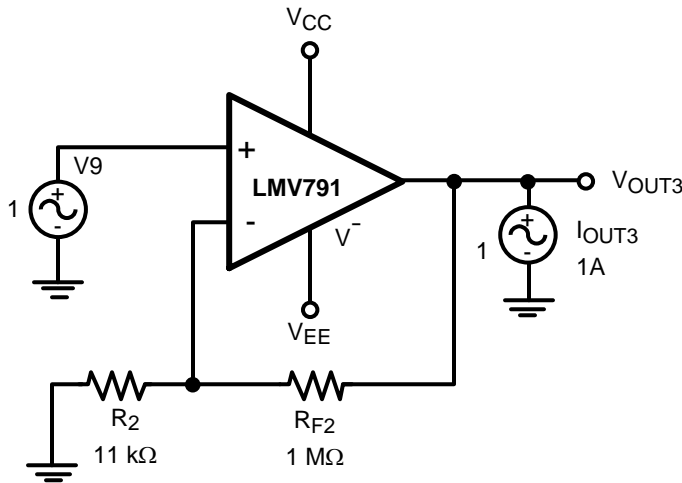
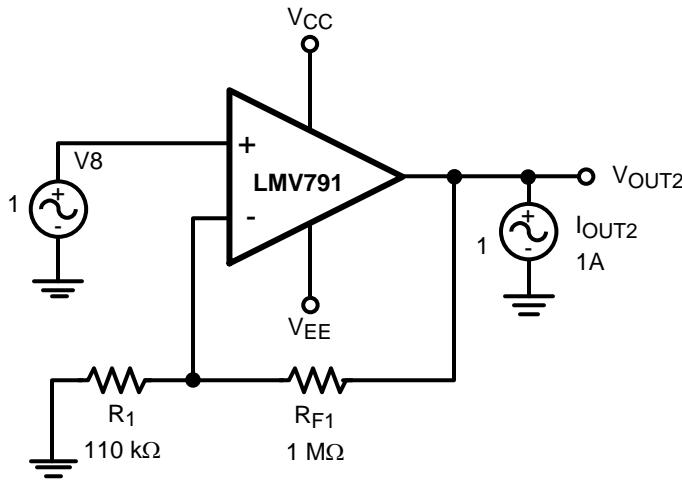
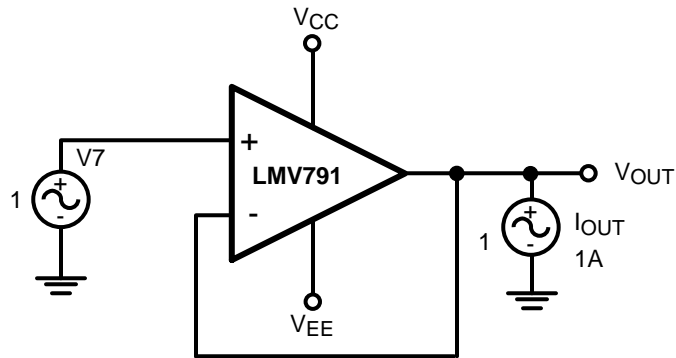
This is a specification that is often omitted from the datasheets altogether but is sometimes needed and necessary.

When modeled correctly, the output impedance helps in getting a more accurate settling time behavior under various capacitive loads.

The output impedance is also needed to calculate the proper component values when a compensation scheme is considered for stability purposes.

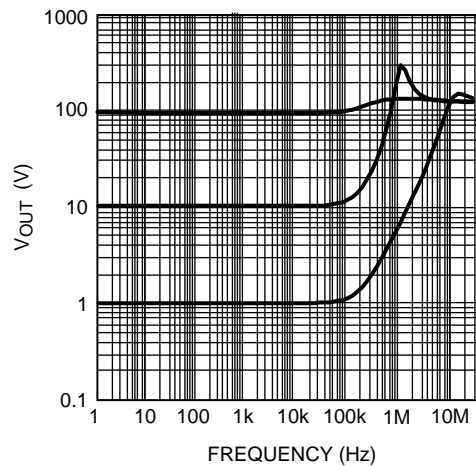
The test circuits of [Output Impedance Test Circuits for a Gain of 1, 10, and 100](#) provide the user with 3 curves for the output impedance vs. frequency at different gains, a gain of 1, 10 and 100. The output impedance is obtained by taking the ratio of the output voltage over the current (1A source at the output of the amplifier).

The graph in [Simulated Closed Loop Output Impedance](#) shows the closed loop output impedance of the LMV791. At higher frequencies (where the curves flatten) the value is about 120Ω. Make sure to plot the curves on a log log scale.



Output Impedance Test Circuits for a Gain of 1, 10, and 100

Output Impedance Test Circuits for a Gain of 1, 10, and 100 (continued)

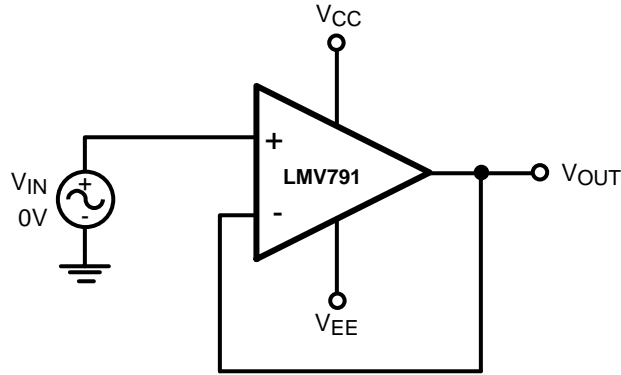


Simulated Closed Loop Output Impedance

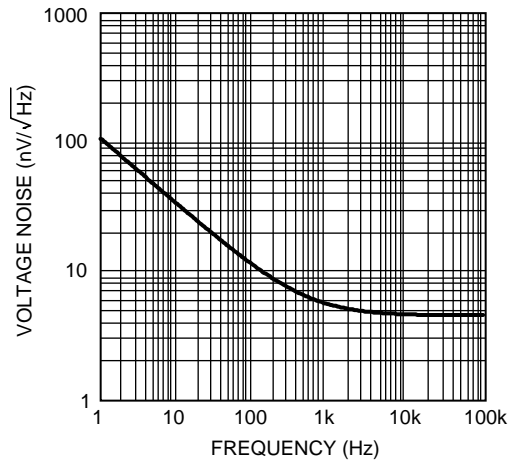
7 Voltage And Current Noise

If there is an area where the folks creating amplifier macro models have made progress, this is one of them. Some of today's models allow users to simulate voltage noise with its flicker noise component and current noise with excellent accuracy. Modeling noise into the macro model doesn't take much more computing or simulation time but it is somewhat a difficult task, at least until you figure out the right equations that make the voltage noise density curve mimic the datasheet graph with the $1/f$ corner as well. One can easily test the voltage noise density by taking the output of a voltage follower (with a voltage source of 0 volts) on a log log scale. To simulate the current noise density, the same circuit can be used with a 100 k Ω resistor placed in series with the non inverting terminal. In the probe window, make sure to divide the result by the value of the resistor chosen, 100 k Ω in this case.

Using a large resistor value makes the current noise dominate since it is coupled into the resistance and thus voltage noise and thermal noise become negligible compared to the current noise. Of course, there are some exceptions where the current noise is very low and a higher source resistance may be needed. It's always good practice to evaluate the noise sources as a function of the source impedance. It is important to specify the output voltage in the analysis setup window of Pspice. In [Voltage Noise Density Test Circuit](#), the output voltage is specified as $V(V_{OUT})$ and the input voltage as V_{IN} and box "noise enabled" is checked.

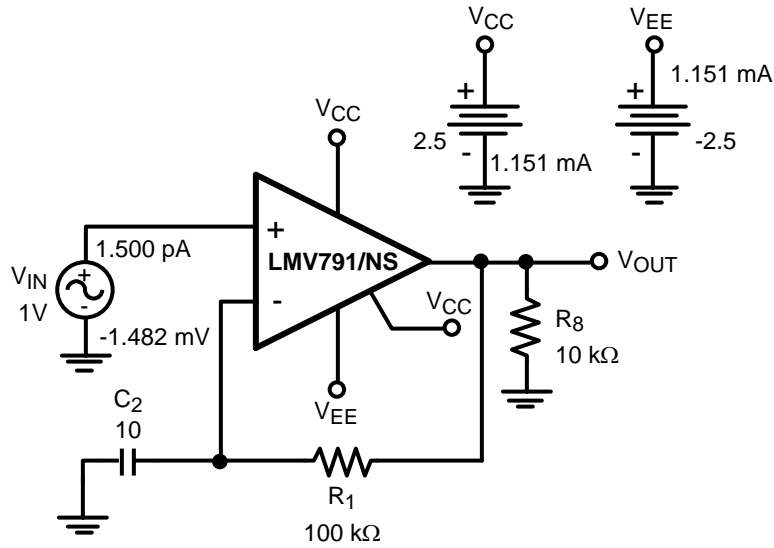


Voltage Noise Density Test Circuit

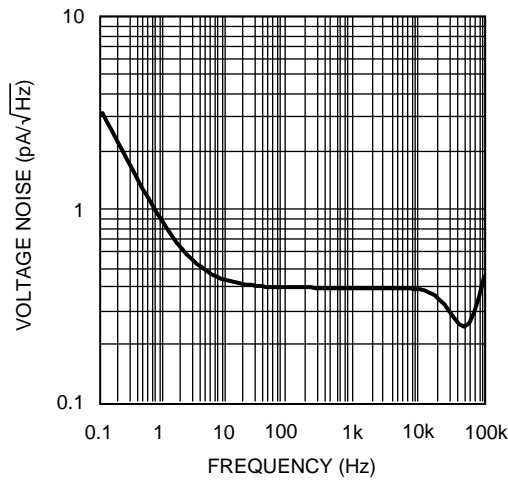


Simulated Voltage Noise Density

Simulated Voltage Noise Density (continued)



Current Noise Density Test Circuit



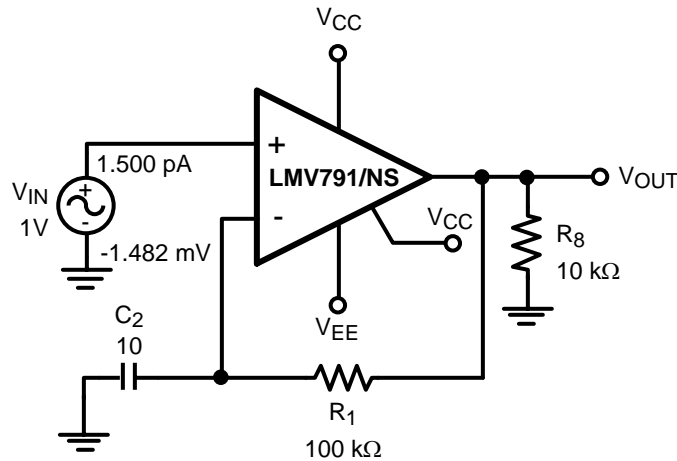
Simulated Current Noise Density

8 Input Bias Current and Input Offset Voltage

These parameters are probably the easiest ones to model. Input offset voltage can easily be implemented as a voltage controlled voltage source at the input whose value is taken from the datasheet.

In general, no specific test circuit is needed for testing bias current or offset voltage. Any of the circuits already discussed in this document can be used. In order to view the values of offset voltage and bias current, the user must activate voltage and current labels in SPICE. This is shown in [Offset Voltage and Input Bias Current](#) where the input bias current is 1.5 pA and the input offset voltage is 1.48 mV.

The current shown through the voltage supplies of 1.15 mA is the quiescent current.

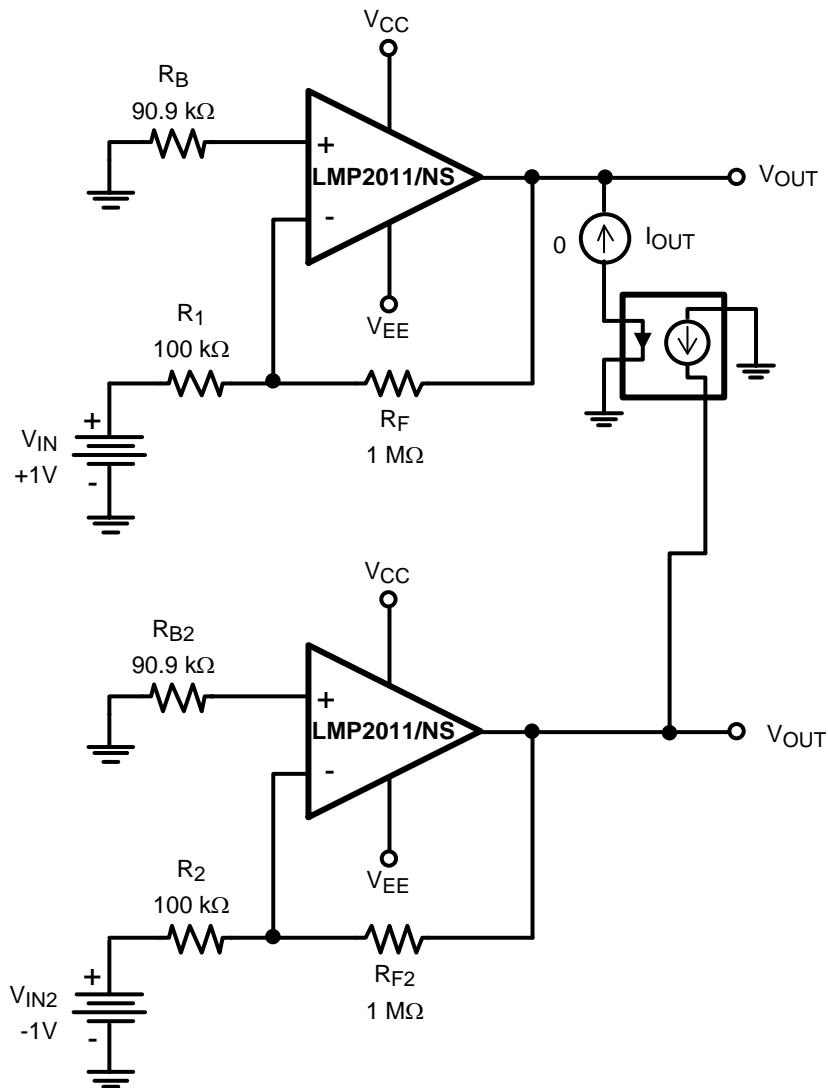


Offset Voltage and Input Bias Current

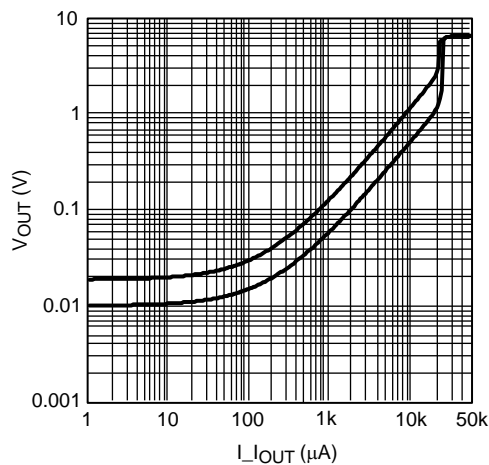
9 Output Saturation Voltage

This parameter is also known as the dropout voltage. It is particularly important in rail to rail output amplifier models as it represents the output voltage swing as a function of the load current and can help the system designer choose the appropriate op amp especially when driving heavy loads or when dynamic range is a concern.

The test circuit, [Output Saturation Voltage vs. Load Current Test Circuit](#), uses a simple DC sweep with 2 equal input voltages of opposite magnitude to replicate the sourcing and sinking of the load current.



Output Saturation Voltage vs. Load Current Test Circuit

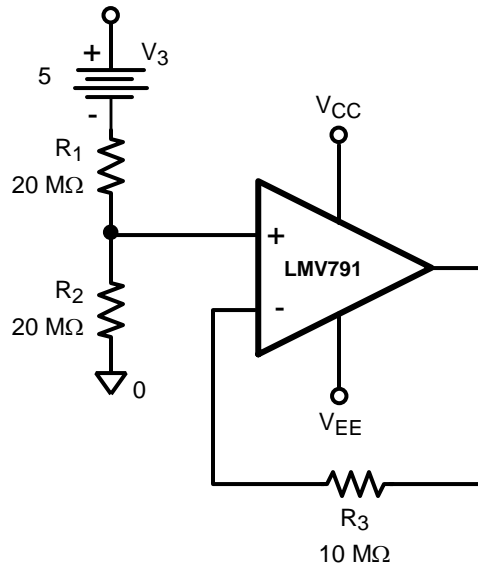


Simulated Output Saturation Voltage

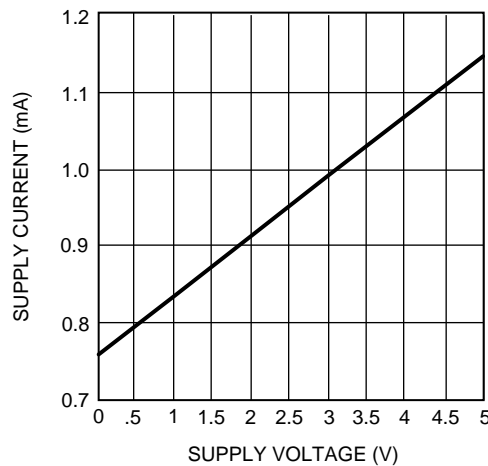
10 Supply Current vs. Supply Voltage

The test circuit, [Supply Current vs. Supply Voltage](#), sweeps the current across the supply and allows the user to determine how much current is drawn by the amplifier at different supply voltages. This test is particularly helpful for power conscious applications.

The slope of the supply current curve can easily be added into the model.



Supply Current vs. Supply Voltage



Simulated Supply Current vs. Supply Voltage

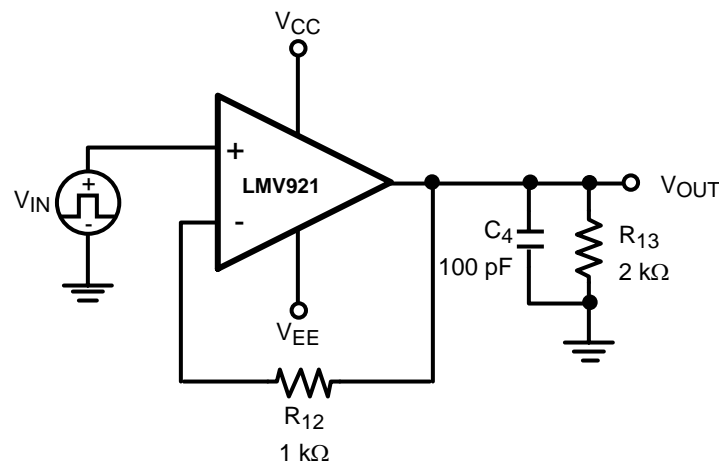
11 Overshoot And Transient Response

This test circuit, [Overshoot Test Circuit](#), serves two purposes: testing the transient response (whether small signal or large signal) and the overshoot.

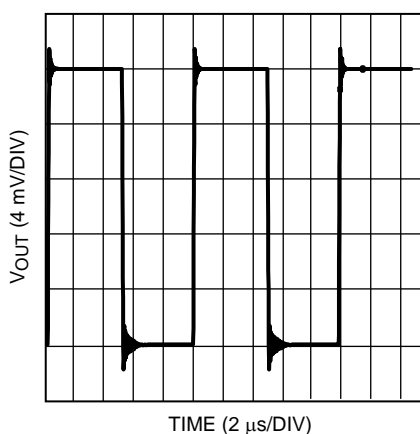
Overshoot is important because it indicates how much ringing an amplifier has in the presence of a capacitive load. Overshoot is a measure of stability in time domain; it is the equivalent of what peaking is in the frequency domain.

Some macro models use extra passive components to mimic the overshoot accurately but generally, if the phase margin is accurate, the overshoot should come pretty close to what it should be.

The transient response can be tested using the test circuit used for the overshoot test, without the 100 pF capacitor. Some datasheets indicate whether a small capacitance is used as a load when measuring the small signal transient, in that case simply use the same value of capacitance.



Overshoot Test Circuit

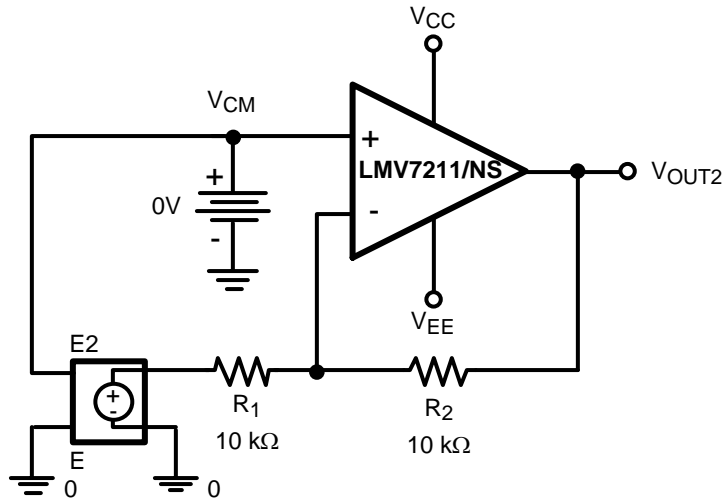


Simulated Overshoot

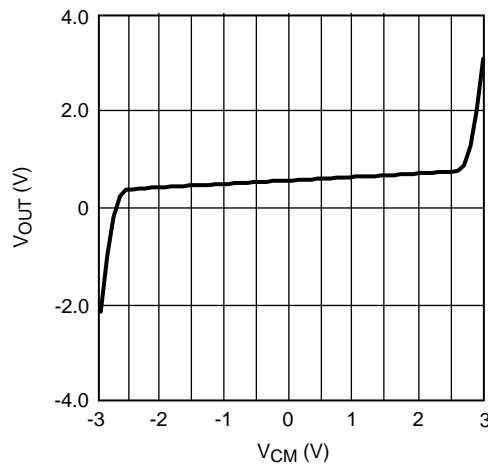
12 Common Mode Voltage Range

This parameter is important as it allows the user to see the head room or how far away the input signal needs to be from the supply.

The first test circuit in [CMVR Test Circuit](#) uses a voltage controlled voltage source. In the second test circuit, [CMVR Test Circuit \(optional\)](#), the voltage is swept from -2.5V to 2.5V.

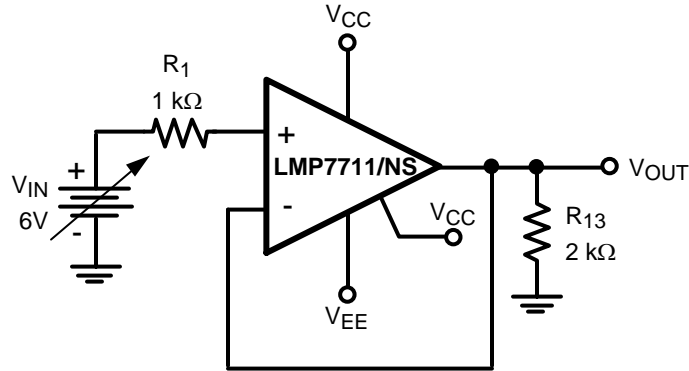


CMVR Test Circuit

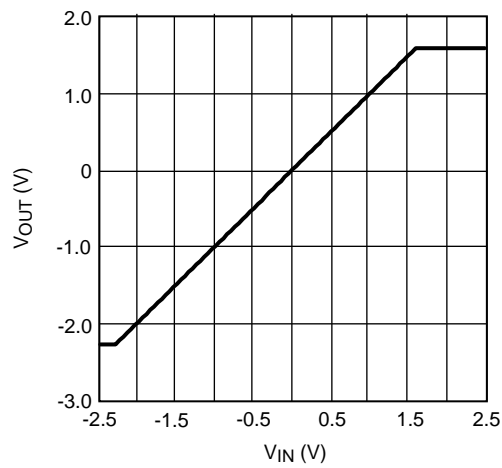


Simulated CMVR

Simulated CMVR (continued)



CMVR Test Circuit (optional)

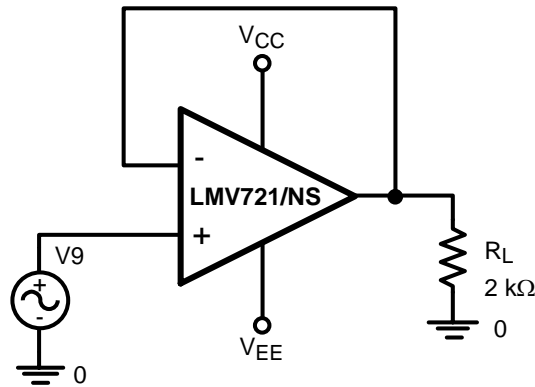


Simulated CMVR

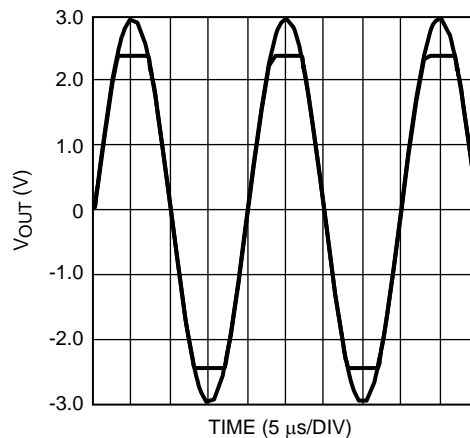
13 Phase Reversal

Phase reversal occurs in some amplifiers when the input signal exceeds the input common mode voltage range. During a phase reversal, the output changes polarity and may cause damage to the op amp resulting in system lockups.

The test circuit, [No Phase Reversal Test Circuit](#), is a simple voltage follower with a sine wave input which goes beyond the common mode voltage range of the amplifier, 6V in this example. The output waveform shown in [No Phase Reversal](#) indicates that the macro model just like the op amp doesn't exhibit any phase reversal, it is clipped at $\pm 2.5V$.



No Phase Reversal Test Circuit



No Phase Reversal

14 Conclusion

The test circuits described above are not meant to replace the evaluation of the device on the bench. Rather, they provide the user with the flexibility of making quick assessments with respect to the accuracy of the macro model.

Special thanks to the applications group and the design community at Texas Instruments for their thoughtful insights.

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as "components") are sold subject to TI's terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI's terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers' products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers' products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI's goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or "enhanced plastic" are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have **not** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products

Audio	www.ti.com/audio
Amplifiers	amplifier.ti.com
Data Converters	dataconverter.ti.com
DLP® Products	www.dlp.com
DSP	dsp.ti.com
Clocks and Timers	www.ti.com/clocks
Interface	interface.ti.com
Logic	logic.ti.com
Power Mgmt	power.ti.com
Microcontrollers	microcontroller.ti.com
RFID	www.ti-rfid.com
OMAP Applications Processors	www.ti.com/omap
Wireless Connectivity	www.ti.com/wirelessconnectivity

Applications

Automotive and Transportation	www.ti.com/automotive
Communications and Telecom	www.ti.com/communications
Computers and Peripherals	www.ti.com/computers
Consumer Electronics	www.ti.com/consumer-apps
Energy and Lighting	www.ti.com/energy
Industrial	www.ti.com/industrial
Medical	www.ti.com/medical
Security	www.ti.com/security
Space, Avionics and Defense	www.ti.com/space-avionics-defense
Video and Imaging	www.ti.com/video

TI E2E Community

e2e.ti.com