TEXAS
INSTRUMENTS

Building a Simple SPICE
Model for the THS3001

*Application
Report*

IMPORTANT NOTICE

Texas Instruments and its subsidiaries (TI) reserve the right to make changes to their products or to discontinue any product or service without notice, and advise customers to obtain the latest version of relevant information to verify, before placing orders, that information being relied on is current and complete. All products are sold subject to the terms and conditions of sale supplied at the time of order acknowledgement, including those pertaining to warranty, patent infringement, and limitation of liability.

TI warrants performance of its semiconductor products to the specifications applicable at the time of sale in accordance with TI's standard warranty. Testing and other quality control techniques are utilized to the extent TI deems necessary to support this warranty. Specific testing of all parameters of each device is not necessarily performed, except those mandated by government requirements.

CERTAIN APPLICATIONS USING SEMICONDUCTOR PRODUCTS MAY INVOLVE POTENTIAL RISKS OF DEATH, PERSONAL INJURY, OR SEVERE PROPERTY OR ENVIRONMENTAL DAMAGE ("CRITICAL APPLICATIONS"). TI SEMICONDUCTOR PRODUCTS ARE NOT DESIGNED, AUTHORIZED, OR WARRANTED TO BE SUITABLE FOR USE IN LIFE-SUPPORT DEVICES OR SYSTEMS OR OTHER CRITICAL APPLICATIONS. INCLUSION OF TI PRODUCTS IN SUCH APPLICATIONS IS UNDERSTOOD TO BE FULLY AT THE CUSTOMER'S RISK.

In order to minimize risks associated with the customer's applications, adequate design and operating safeguards must be provided by the customer to minimize inherent or procedural hazards.

TI assumes no liability for applications assistance or customer product design. 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 of TI covering or relating to any combination, machine, or process in which such semiconductor products or services might be or are used. TI's publication of information regarding any third party's products or services does not constitute TI's approval, warranty or endorsement thereof.

Contents

List of Figures

Building a Simple SPICE Model for the THS3001

James Karki

ABSTRACT

The application report *Voltage Feedback Versus Current Feedback Op Amps* SLVA051 outlines the basic operation of a current feedback operational amplifier (op amp) in relation to a voltage feedback op amp. One of the basic principles of a current feedback op amp is that its transfer function is a transimpedance equal to the output voltage divided by the current from the negative input terminal, i.e., $Zt = \dfrac{Vo}{Ie}$.

This application report describes how to construct a simple model that will simulate the frequency response of the THS3001 in SPICE. The laboratory setup to measure the basic parameters of the THS3001 is illustrated.

1 Introduction

This report highlights the basic functioning of the THS3001 op amp, and helps give insight to its proper use. Limitations of the device such as input common mode range, power supply range, CMRR, PSRR, output voltage swing, etc., are purposely left out . The model is simple and easy to understand, simulates fast, and gives good results.

Figure 1 is a basic current feedback op amp model. To transform this into a SPICE circuit model, *Z*t, *Zo*, and *Re* must be determined.
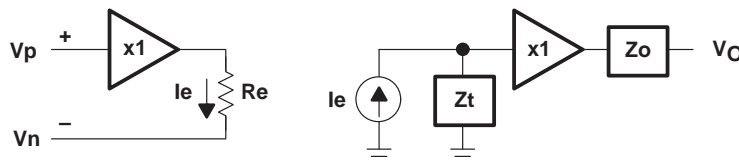

Figure 1. Basic CF Op Amp Model

2 Measuring *Zt*

A network analyzer is designed to measure the transfer function of a circuit. Therefore an HP8753E network analyzer is used to measure the transfer function of the THS3001. It has a lower frequency limit of 30 kHz. Below 30 kHz a servo-loop is used. Figure 2 shows the laboratory test setups. The servo-loop is on the left and the network analyzer is on the right.
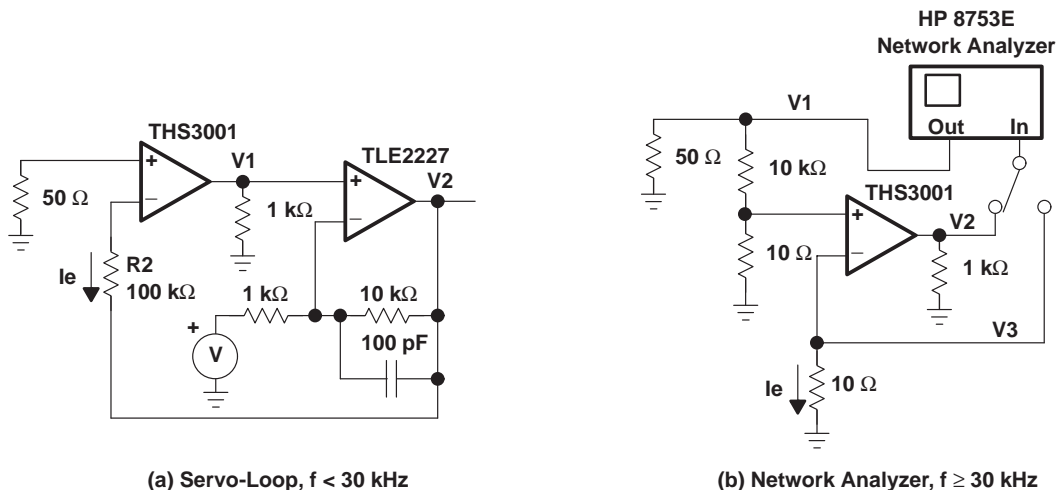
(a) Servo-Loop, f < 30 kHz   (b) Network Analyzer, f $\geq$ 30 kHz

Figure 2. Test Setup

Solving the servo-loop circuit shown in Figure 2 (a), the transimpedance of the THS3001 is $Zt = -\dfrac{\Delta V1}{\Delta V2} \times R2$. The 100-pF capacitor limits the bandwidth to 159 kHz and the 1 kΩ provides a load.

Using the network analyzer as shown in Figure 2 (b), the transfer functions $\dfrac{V2}{V1}$ and $\dfrac{V3}{V1}$ are measured by switching the input between *V2* and *V3*. To compute $Zt = \dfrac{V2}{Ie}$, divide $\dfrac{V2}{V1}$ by $\dfrac{V3}{V1}$ to get $\dfrac{V2}{V3}$, then multiply by 10 Ω to get the transimpedance of the THS3001. By default the network analyzer displays the results in dB. Therefore, the mechanics of the transimpedance calculation are: $Zt\ (dB) = \dfrac{V2}{V1}(dB) - \dfrac{V3}{V1}(dB) + 20\ \ dB$.

Figure 3 is a plot of the THS3001 transimpedance magnitude and phase versus frequency from data collected using the test setups and methods described above. The simulation result of the SPICE model, constructed later, is also plotted.
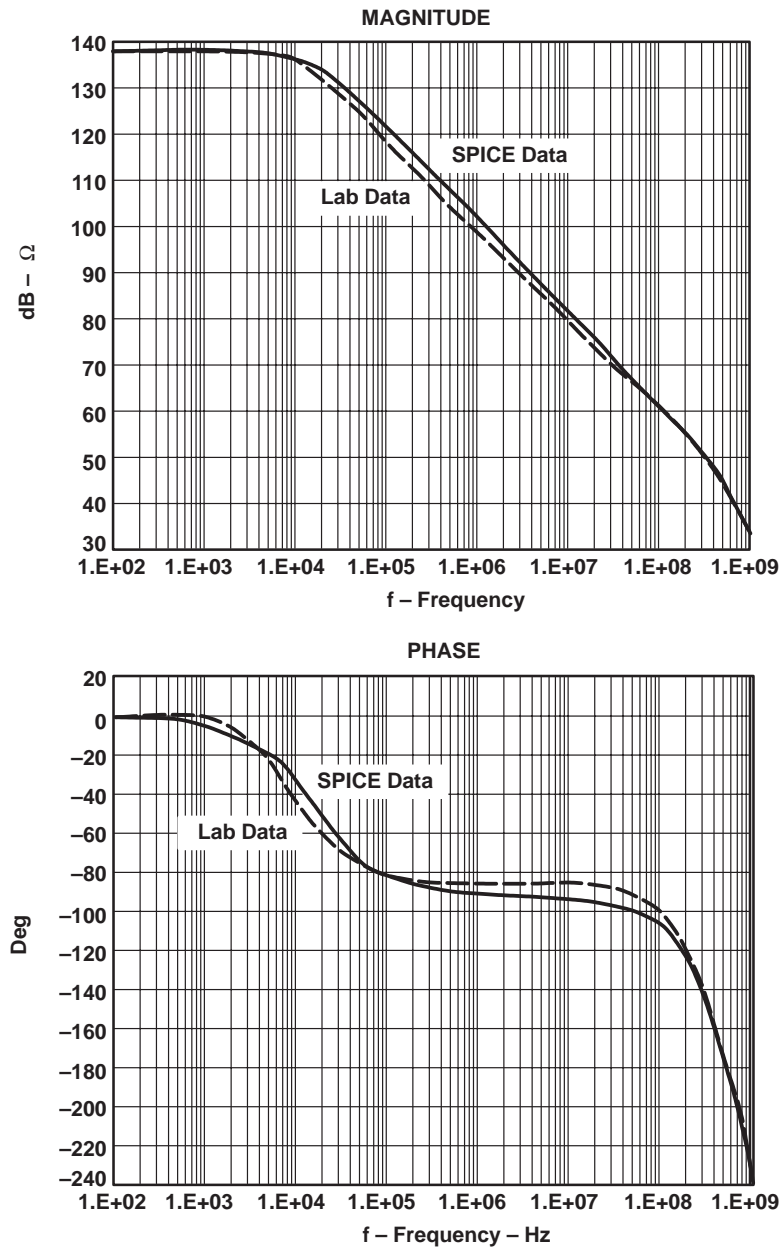
MAGNITUDE

PHASE

Figure 3. THS3001 Open Loop Transimpedance

Analyzing this information:

dc gain = 138.5 dB Ω = 8.5 MΩ
dominant pole near 10 kHz,
multiple upper frequency poles above 200 MHz

3   Measuring *Zo*

With the circuit shown in Figure 4, the amplifier tries to maintain 0 V at its output terminal. Because of the finite output impedance *zo*, there is a finite voltage at V2. By varying the voltage V1 and recording V2, the output impedance, Zo_{CL}, is calculated by the formula shown.
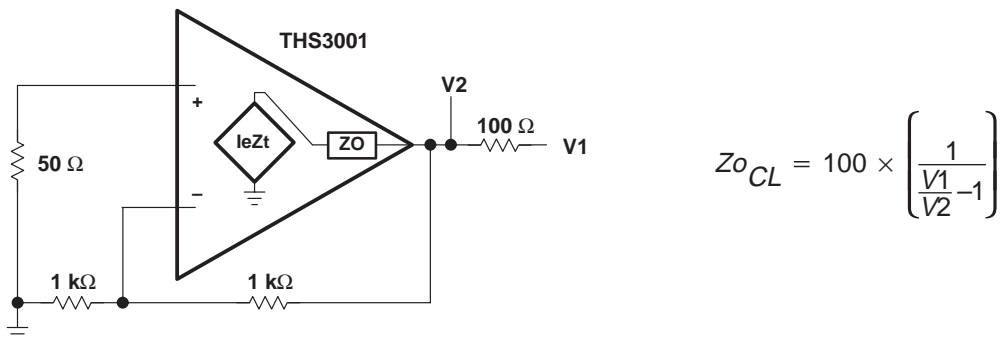
Figure 4. Measuring *Zo*

In the closed loop situation shown, $Zo_{CL} \neq zo$, but it is related by the loop transmission as follows:

$$Zo_{CL} = zo \times \left[\frac{1}{1 + \frac{Zt}{1k}} \right]$$

At low frequencies Zo_{CL} is very low because Zt is very high, and decreases as frequency increases. The graph shown in Figure 5 shows the measured Zo_{CL} from 100 kHz to 1 GHz. The SPICE simulation of the output impedance model (developed below) configured the same as the test circuit in Figure 4 is also plotted.

Analyzing the graph in Figure 5: Zo_{CL} increases at approximately 20 dB/dec from 100 kHz to over 100 MHz; it peaks at 600 MHz, and falls at about 20 dB/dec to 1 GHz. Correlating Zo_{CL} to Zt, and the above equation: zo appears resistive up to about 100 MHz where $Zt = 1000$. After that zo appears reactive in nature.
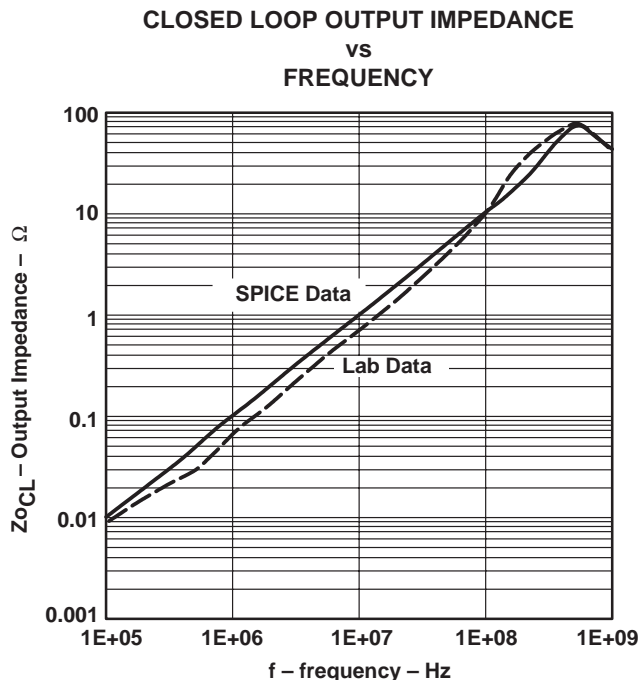
CLOSED LOOP OUTPUT IMPEDANCE
VS
FREQUENCY

Figure 5. *Zo~CL~* vs Frequency

Figure caption should be: **Figure 5. Zo_{CL} vs Frequency**

4   Measuring *Re*

By measuring the transfer function $\frac{V2}{V1}$ in Figure 6, *Re* is approximated by the formula shown. At higher frequencies, parasitic inductance and capacitance modify the impedance, but for the most part, these effects are included in the measurement of *Zt*. So, for the purposes of this report, take *Re* = 25 Ω resistive.
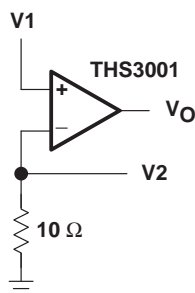
$$Re = 10 \times \left(\frac{V1}{V2} - 1 \right) = 25\,\Omega$$

Figure 6. Measuring *Re*

5   Constructing the Model

To build a SPICE model for the THS3001, use Figure 1 as a template, and use the data collected to compute component values. E and F devices are used to construct the SPICE model. An E device is a voltage-controlled voltage source and an F device is a current-controlled current source. Unity gain is used for these devices.

The input is modeled by an E device driving an F device. The current in the F device is set by *Re* between the negative input terminal and ground. The output of the F device emulates the error current *Ie*.

The output of the F device drives $Rt = 8.5\ \text{M}\Omega$ and $Cc = 1.25\ \text{pF}$. The values are chosen to set the dc gain and dominant pole of *Zt*.

Trial and error reveals that an RC single pole at 1 GHz, and an LRC double pole at 800 MHz (Q = 1) shapes the upper frequency response close to the measured values. E1 – E3 provide buffering. R1 and C1 form the RC single pole, and L2, R2, and C2 form the LRC double pole.

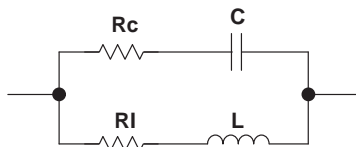The model shown in Figure 7 is proposed for *zo*.

Figure 7. Model for *zo*

Select component values as follows:

1. $Rl = Zo_{CL}$ divided by Zt/1000 at 100 kHz $\rightarrow Rl \approx 7.7\ \Omega$

2. $sL = Rl$ at 100 MHz $\rightarrow L \approx 11\ \text{nH}$

3. $1/sC = sL$ at 600 MHz $\rightarrow C \approx 6.3\ \text{pF}$

4. $Rc = \sqrt{\dfrac{\left(Zo_{CL}Rl - \dfrac{L}{C}\right)^2 + \left(\dfrac{Rl}{sC}\right)^2}{\left(Rl - Zo_{CL}\right)^2 + (sL)^2}}$, with $s = 2\pi * 600\ MHz$, $\rightarrow Rc \approx 18\Omega$

Figure 8 shows the SPICE model created. The simulated open loop transimpedance and closed loop output impedance are included in Figure 3 and Figure 5 to see the correlation between simulating this model and the lab data collected on the THS3001.
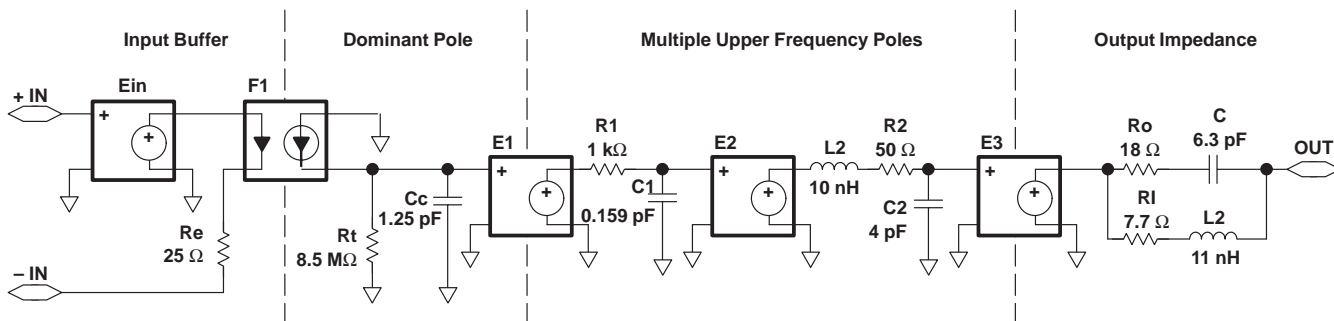

Figure 8. Simple THS3001 SPICE Model

To test the model further, SPICE simulations of circuits from the THS3001 data sheet, literature number SLOS217, is performed and compared to the data sheet graphs. The circuits and the results are shown in Figure 9 through Figure 11. Note that 1-pF capacitors are added to the output and negative input to simulate parasitic board capacitance. The data from the data sheet is prefaced by DS, followed by the circuit configuration. The data from the SPICE simulation is prefaced by SPICE, followed by the circuit configuration.
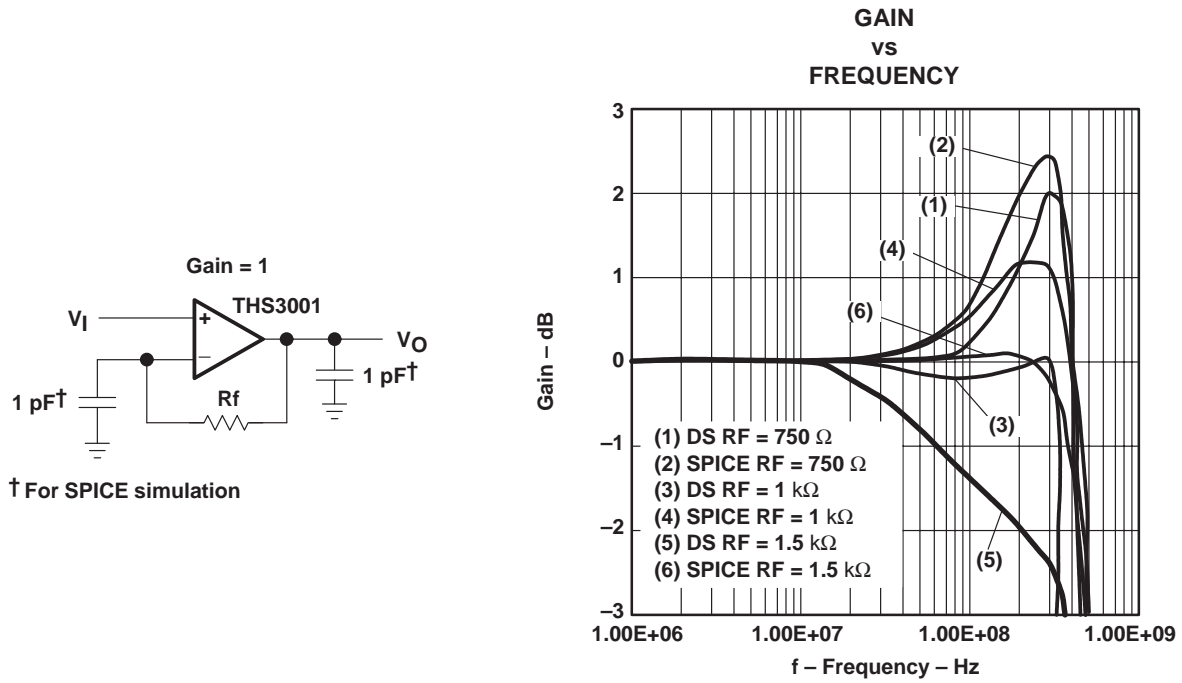
GAIN
vs
FREQUENCY

(1) DS RF = 750 Ω
(2) SPICE RF = 750 Ω
(3) DS RF = 1 kΩ
(4) SPICE RF = 1 kΩ
(5) DS RF = 1.5 kΩ
(6) SPICE RF = 1.5 kΩ

Gain = 1

THS3001

V_I

V_O

Rf

1 pF†

1 pF†

† For SPICE simulation

Figure 9. Comparison of Data Sheet and SPICE Model with Gain = 1

GAIN
vs
FREQUENCY

(1) DS RF = 560 Ω
(2) SPICE RF = 560 Ω
(3) DS RF = 680 Ω
(4) SPICE RF = 680 Ω
(5) DS RF = 1 kΩ
(6) SPICE RF = 1 kΩ

Gain = 2

THS3001

V_I

V_O

Rg

Rf

1 pF†

1 pF†

† For SPICE simulation
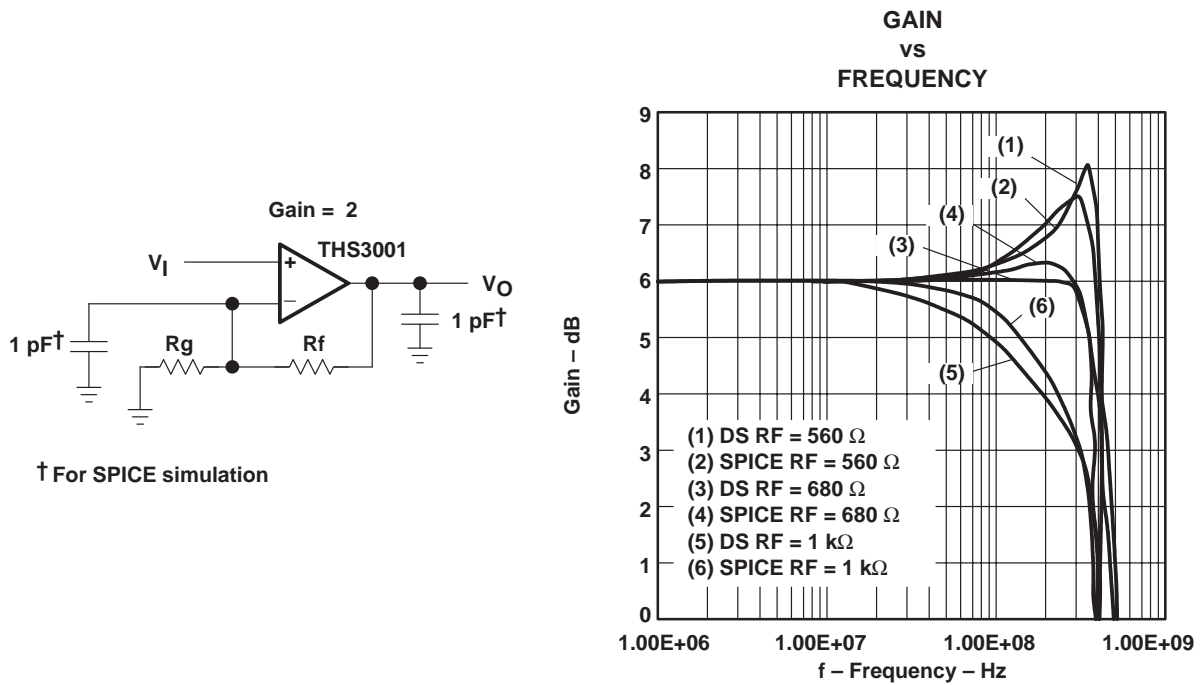
Figure 10. Comparison of Data Sheet and SPICE Model with Gain = 2
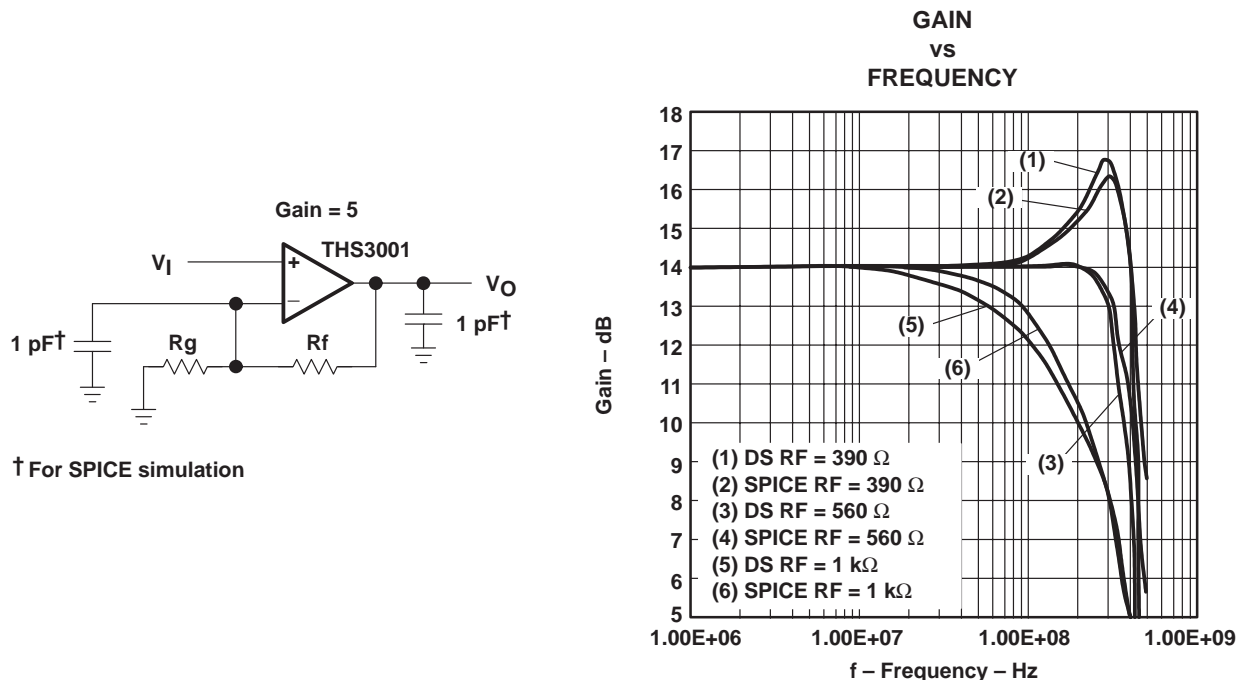
GAIN
vs
FREQUENCY

Figure 11. Comparison of Data Sheet and SPICE Model with Gain = 5

6 Summary

The simple model developed here is useful for simulating and understanding the basic operation of the THS3001. Hopefully this model will prove practical to you. Use it in good health.