Operational Amplifier Stability
Part 2 of 15: Op Amp Networks, SPICE Analysis
by Tim Green
Strategic Development Engineer, Burr-Brown Products from Texas Instruments Incorporated

Part 2 of this series focuses on analyzing op amp circuits for stability with a special emphasis on two common op amp networks. It is important first to perform 1st-order analysis (hand analysis using mostly the thing between your ears) before moving on to SPICE simulation. Remember that GIGO (garbage-in-garbage-out) can result from circuit simulation programs if you do not know what you are looking for before simulation. The SPICE loop gain test technique will be presented, which allows easy Bode plotting of Aol curves, 1/β curves, and loop gain curves. Also, an easy-to-build ac SPICE model will be presented which allows for any op amp circuit to be quickly analyzed for ac stability.

Within this entire series we will be analyzing op amp circuits and presenting results using a versatile SPICE simulation software called "TINA." At http://www.designsoftware.com various versions of it can be previewed. Although some of the SPICE tricks presented are specific to Tina you will probably find that other SPICE simulators you use may also benefit from these shortcuts.

SPICE Loop Gain Test

The SPICE loop gain test is detailed in Fig. 2.0. LT provides a closed-loop circuit for dc since every ac SPICE analysis requires a dc SPICE analysis. During an ac SPICE analysis, as frequency increases, CT becomes a short and LT becomes an open and with one SPICE run all information for ac stability can be obtained. Op amp Aol, loop gain, and 1/β magnitude and phase plots are easily obtained from SPICE post-processing by using the equations detailed in Fig. 2.0. Although there are other techniques applicable to "break the loop" for an ac analysis in SPICE, the technique shown in Fig. 2.0 has proven to be the least error-prone and least subject to mathematical nuances inside of SPICE.
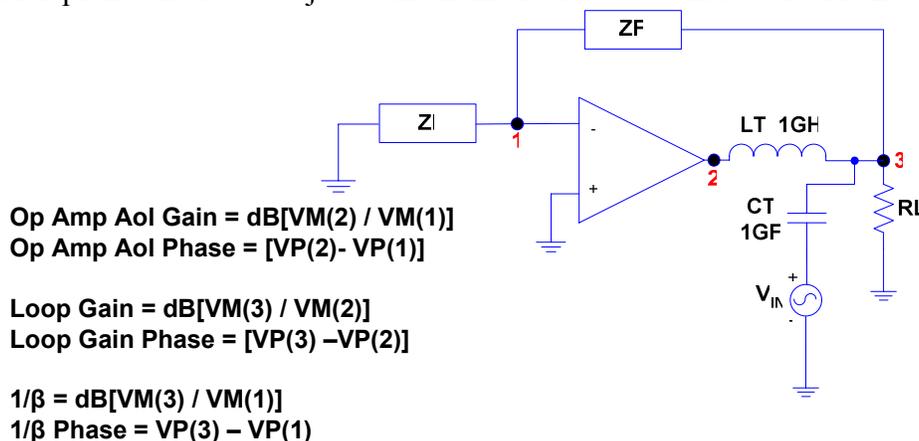

Op Amp Aol Gain = dB[VM(2) / VM(1)]
Op Amp Aol Phase = [VP(2)- VP(1)]

Loop Gain = dB[VM(3) / VM(2)]
Loop Gain Phase = [VP(3) –VP(2)]

1/β = dB[VM(3) / VM(1)]
1/β Phase = VP(3) – VP(1)

Fig. 2.0: SPICE Loop Gain Test

Op Amp Networks and 1/β

Two common op amp networks, ZI and ZF, are shown in Fig. 2.1. We will perform a 1st-order analysis on each of these networks, independently, and then use Tina SPICE to simulate the op amp circuit and check if our predicted results agree! The key to our 1st-order analysis will be to use our intuitive component models from Part 1 of this series and a little intuition.
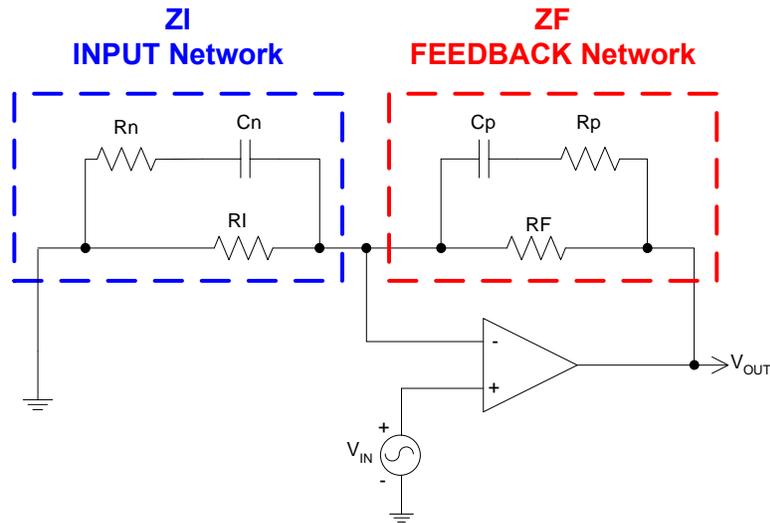

Fig. 2.1: Two Common Op Amp Networks: ZI & ZF

Op Amp Network ZF

Let's perform our 1st-order analysis for the ZF network, which is a feedback network in the circuit. Cp is open at low frequency and 1/β becomes simply RF/RI (Fig.2). At high frequencies Cp is a short and 1/β becomes (Rp//RF)/RI. However, when Cp is a short Rp<<RF and Rp dominates the feedback resistance and we approximate the hf gain to be Rp/RI. Note there is a reactive element in the feedback path, a capacitor, and there has to be poles and/or zeros somewhere in the transfer function. At the frequency where the magnitude of Cp matches that of the parallel impedance with it (dominated by RF) we anticipate a pole in the 1/β plot. Feedback resistance will be getting smaller and therefore V$_{OUT}$ must start to reduce. At the frequency where the magnitude of Cp matches that of the impedance in series with it, Rp, we expect a zero since as Cp approaches a short the net feedback resistance can become no smaller and V$_{OUT}$ must flatten out as frequency increases. By our 1st-order analysis where a pole and zero exists as well as the low-frequency and high-frequency 1/β.
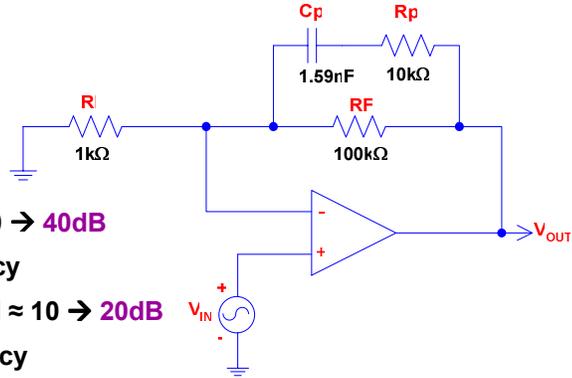
- ➤ **1/β Low Frequency = RF/RI = 100 → 40dB**
  - **Cp = Open at Low Frequency**
- ➤ **1/β High Frequency = (Rp//RF)/RI ≈ 10 → 20dB**
  - **Cp = Short at High Frequency**
- ➤ **Pole in 1/β when Magnitude of X_{Cp} = RF**
  - **Magnitude X_{Cp} = 1/(2·Π·f·Cp)**
  - **fp = 1/(2·Π·RF·Cp) = 1kHz**
- ➤ **Zero in 1/β when Magnitude of X_{Cp} = Rp**
  - **fz = 1/(2·Π·Rp·Cp) = 10kHz**

Fig. 2.2: 1/β 1st-Order Analysis for ZF

To check our 1st-order analysis our ZF circuit was built in Tina SPICE as shown in Fig. 2.3. V_{IN} is set for a dc value of 0 V and an ac Source option is selected, with the ac amplitude set to 1. Our ac analysis is set up to run from 10 Hz to 10 MHz with 100 points of data, and amplitude & phase data requested to be saved for post-processing purposes. To perform our SPICE loop gain test we use L1, C1 and V_{IN} with convenient voltage probes placed at N1, N2, and N3. By inspection of this circuit we see Aol = N2/N1 and 1/β = N3/N1.
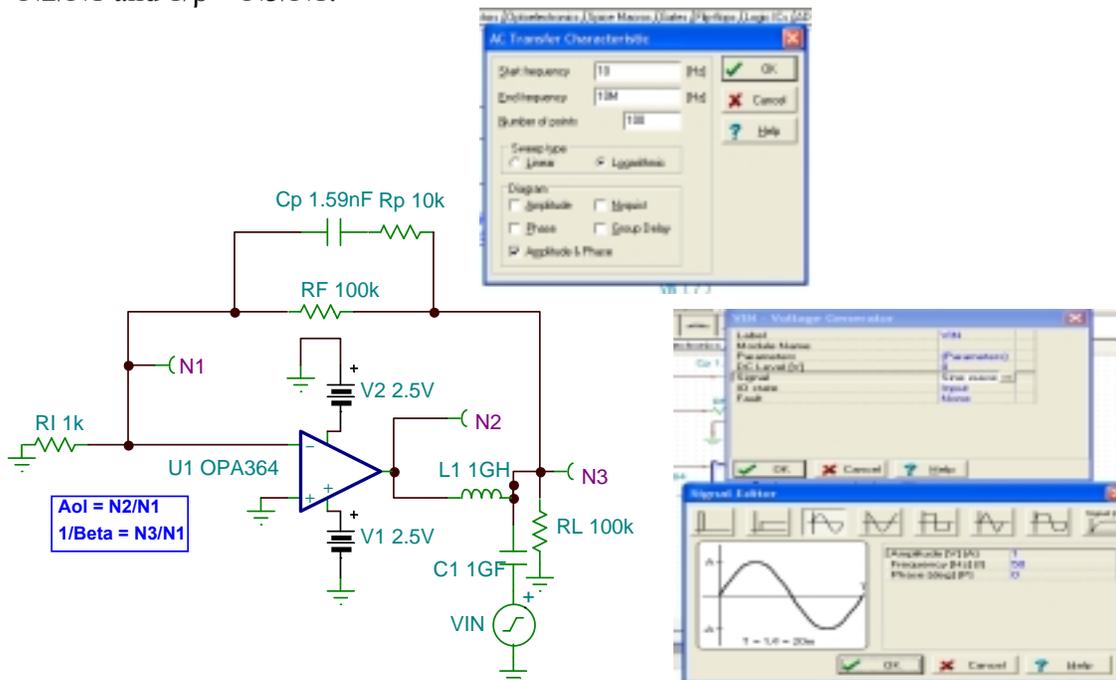

Fig. 2.3: Tina SPICE Circuit For ZF Analysis

The "Default Results" of our Tina SPICE simulation are displayed in Fig. 2.4. Not too valuable yet as we are interested in the 1/β plot for ZF and the op amp Aol Curve.
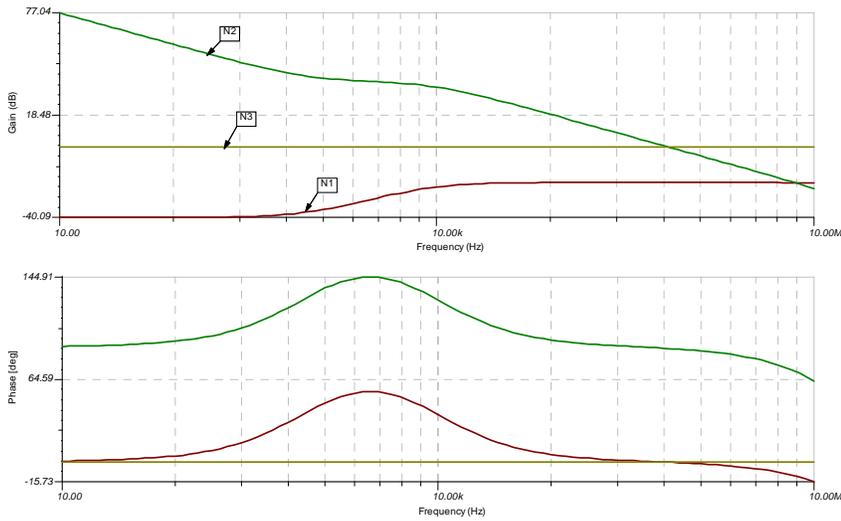

Fig. 2.4: Tina SPICE Default Results for ZF Analysis

So, to obtain the desired plots we perform "Postprocessing" as shown in Fig. 2.5. The user-defined function, Aol, has been assigned the math equation N2/N1 (for our Aol plot) and Beta1 (so named as 1/β is not an accepted name in Tina SPICE) assigned the math equation N3/N1 (for our 1/β plot).
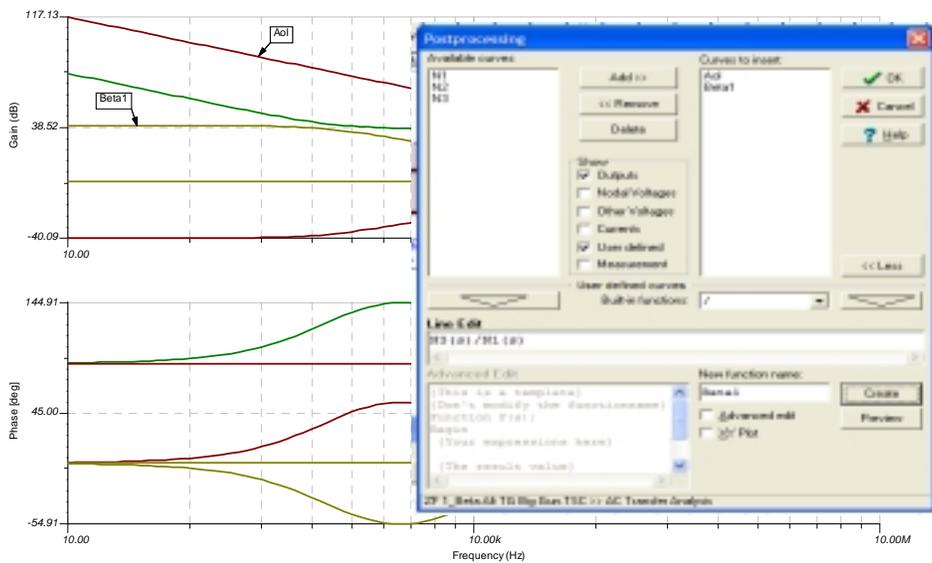

Fig. 2.5: Tina SPICE Post Processing Math For ZF Analysis

Now we have generated the math results for Aol and Beta1 (Fig. 2.6). We can clean up our resultant plot window by right-clicking on each waveform we no longer need (ie N1, N2, N3) in both the magnitude and phase plots and deleting these unneeded waveforms. After this cleanup, right-click on the y-axis for each plot and select "Default Ranges." All looks good now EXCEPT our plots are not in familiar and useful scales that make it easy to see 20-dB/decade slopes in magnitude and 45°/decade slopes in phase.
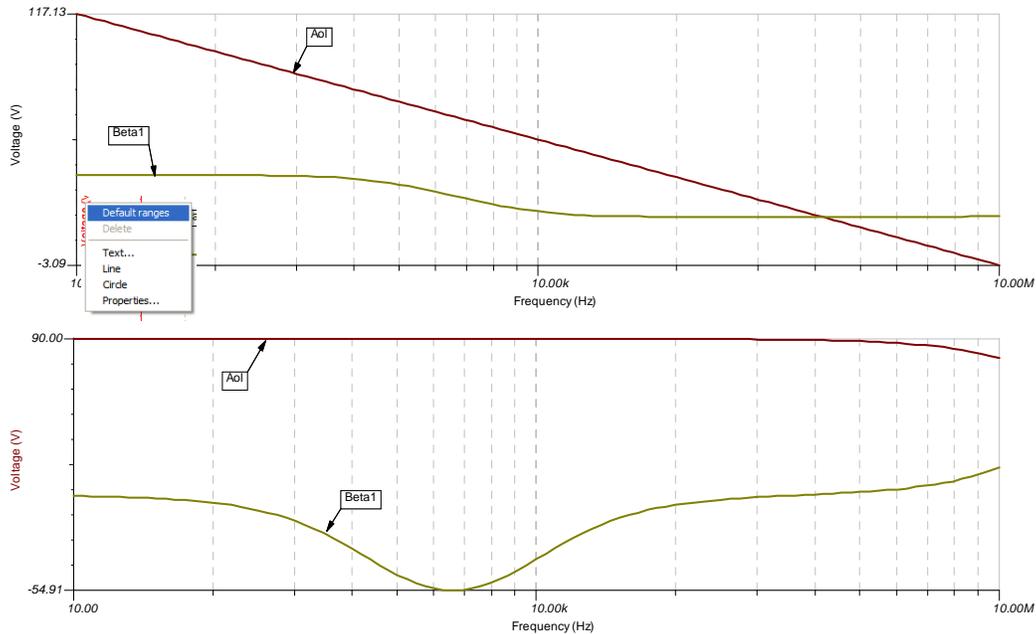
Fig. 2.6: Tina SPICE Default Scaling: Postprocessing Math For ZF Analysis

There is a "frequency rescale" trick (Fig. 2.7) which will allow us to conveniently get the best decade resolution of frequency. Right-click on the x-axis and select "Properties." A "Set Axis" window will appear. The secret to selecting the right number of "Ticks" for scaling is to add 1 to the number of decades plotted. For 10 Hz - 10 MHz. Now our axes looks like a familiar semi-log plot.

➢ **Right click on X-Axis. Select "Properties"**

➢ **# Ticks = # Decades + 1**

    i.e.   **10Hz-10MHz → 6 decades**
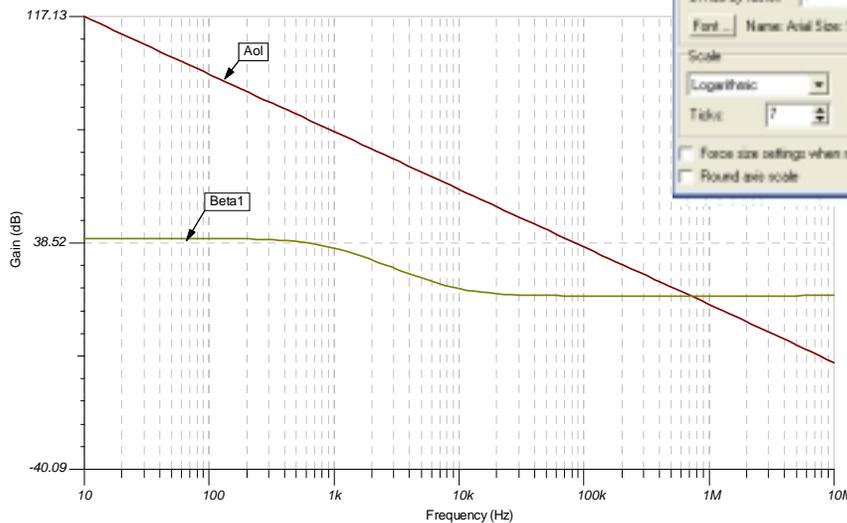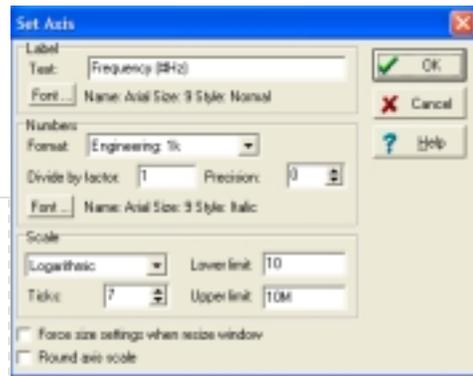
           # Ticks = 6 + 1 = 7

Fig. 2.7: Tina SPICE Frequency Rescale For ZF Analysis

Now we wish to re-scale the y-axis on the magnitude plot to a more familiar 20 dB/division. Our "Gain Re-Scale" trick is (Fig. 2.8) to right-click on the y-axis and select "Properties" and a window will appear. The secret to selecting the right number of "Ticks" for scaling is to first set the "lower limit" to the nearest even increment of 20 dB less than the default lower limit shown. Now set the "upper limit" to the nearest even increment of 20 dB more than the default upper limit shown. Subtract the new upper limit from the new lower limit and divide the result by 20. To this number add 1 and we have calculated the correct number of ticks to set to get a familiar y-axis scaling of 20 dB/division.

> **Right click on Y-Axis again and select "Properties"**
> **Lower Limit = Nearest 20dB < Min Gain (i.e. -20dB < Min Gain)**
> **Upper Limit = Nearest 20dB > Max Gain (i.e 120dB > Max Gain)**
> **# Ticks = [Upper Limit – Lower Limit]/20 + 1**
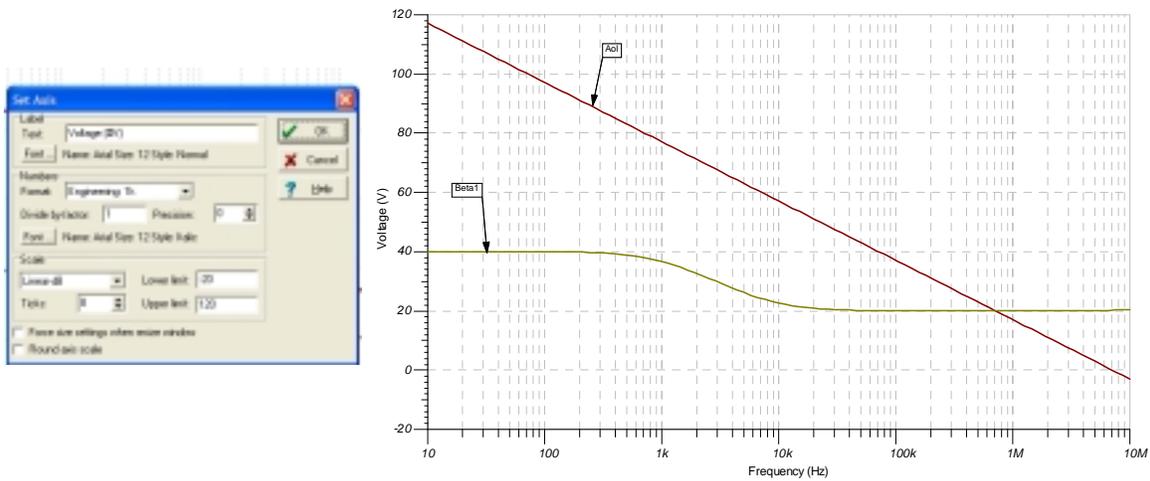>   **# Ticks = [120- (-20)]/20 + 1 = 8**

Fig. 2.8: Tina SPICE Gain Rescale For ZF Analysis

Also, for ease of reading the phase plot, we will rescale the y-axis to a more familiar 45°/division. Our phase rescale trick (Fig. 2.9) is to right-click on the y-axis and select "Properties" and a window will appear. The secret to selecting the right number of "Ticks" for scaling is to first set the "lower limit" to the nearest even increment of 45° less than the default lower limit shown. Now set the "upper limit" to the nearest even increment of 45° more than the default upper limit shown. Subtract the new upper limit from the new lower limit and divide the result by 45. To this number add 1 and we have calculated the correct number of ticks to set to get a familiar y-axis scaling of 45°/division.

- ➢ **Right click on Y-Axis again and select "Properties"**
- ➢ **Lower Limit = Nearest 45 degrees < Min Phase (i.e. -90 degrees < Min Phase)**
- ➢ **Upper Limit = Nearest 45 degrees > Max Phase (i.e +180 degrees > Max Phase)**
- ➢ **# Ticks = [Upper Limit – Lower Limit]/45 + 1**

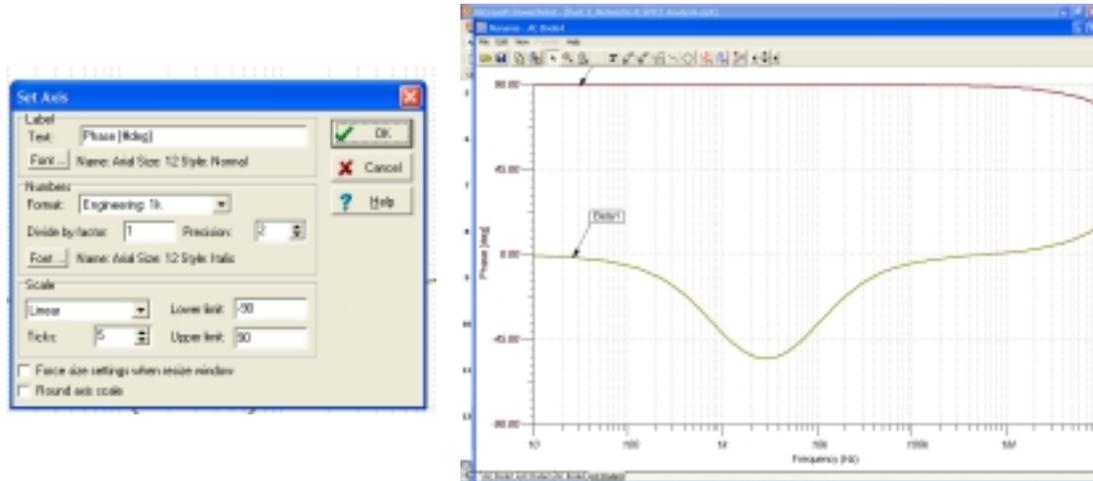 **# Ticks = [90- (-90)]/45 + 1 = 5**

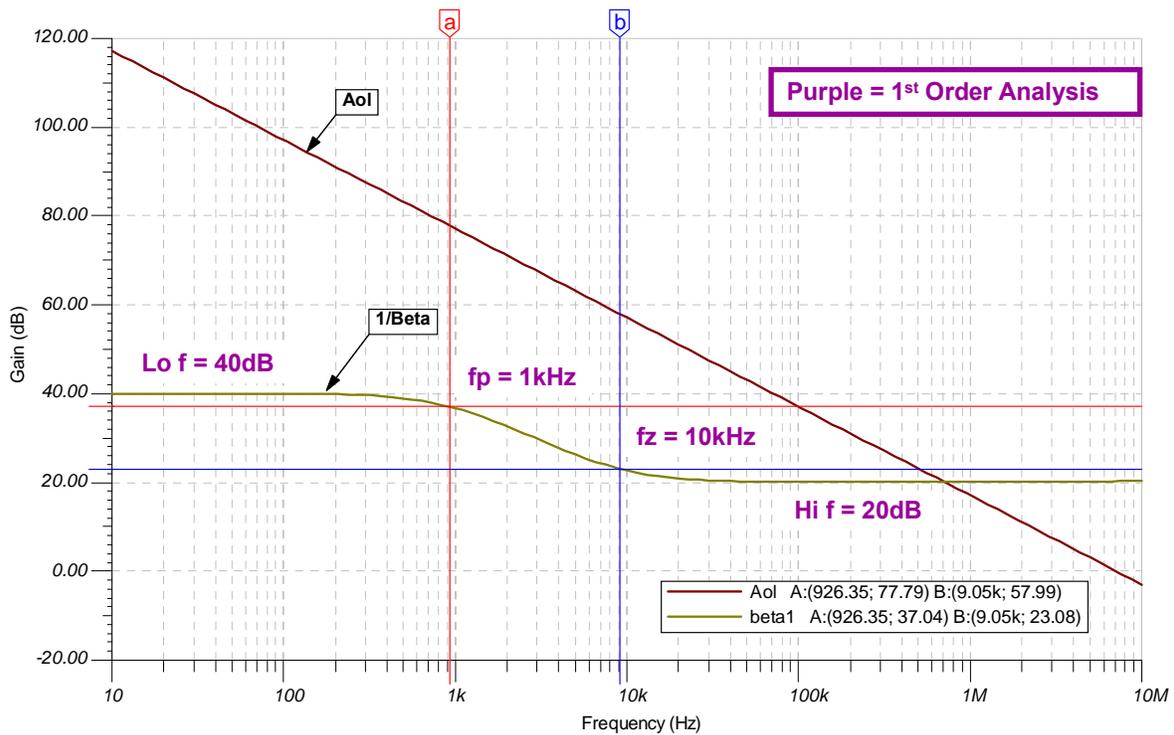**Fig. 2.9: Tina SPICE Phase Rescale For ZF Analysis**

Fig. 2.10: Tina SPICE Optimally-Scaled Results For ZF Analysis

In our optimally-scaled Tina SPICE simulation results (Fig. 2.10) the purple colored text denotes our 1st-order analysis predictions. The cursors are set for an exact amplitude difference of -3 dB from the low-frequency $1/\beta$ and +3 dB from the high-frequency $1/\beta$. The predictions and results are not exact but are certainly more than acceptable for powerful and intuitive ac stability analysis.

Op Amp Network ZI

Let's perform our 1st-order analysis for the ZI network (Fig. 2.11) which is an input network in the circuit. Cn is open at low frequency and $1/\beta$ becomes RF/RI. At high frequencies Cn is a short and $1/\beta$ becomes RF/(RI//Rn). However, when Cn is a short Rn<<RI and Rn should dominate the input resistance and so we approximate high frequency gain to be RF/Rn. There is a reactive element in the input path, a capacitor, and we know there has to be poles and/or zeros somewhere in the transfer function. At the frequency where the magnitude of Cn matches that of the parallel impedance with it (dominated here by RI) we anticipate a zero in the $1/\beta$ plot. Input resistance will be getting smaller and therefore V_{OUT} must start to increase. At the frequency where the magnitude of Cn matches that of the impedance in series with it, Rn, we expect a pole since as Cn approaches a short the net input resistance can become no smaller and V_{OUT} must flatten out as frequency increases. So we have predicted by our 1st-order analysis where a pole and zero exist, and low- and high-frequency $1/\beta$.
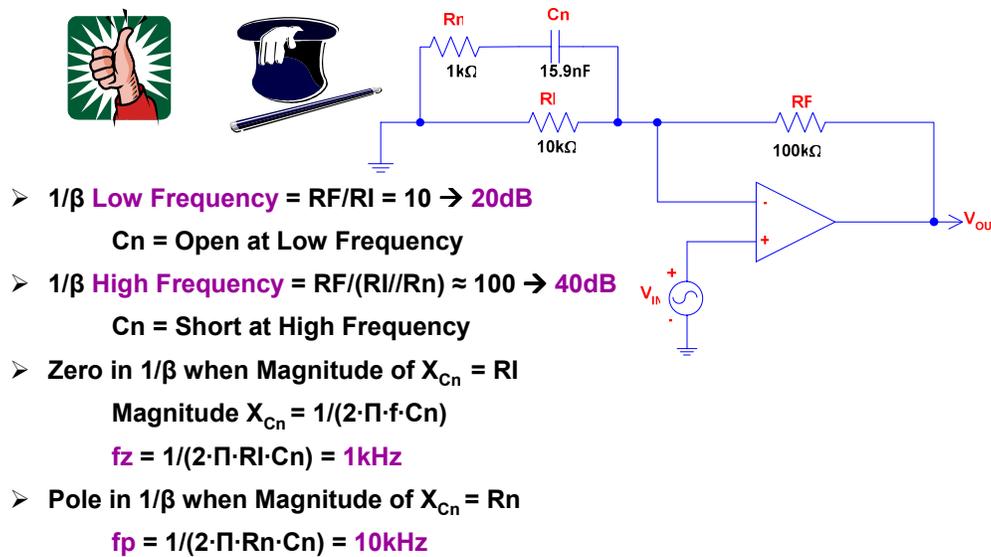

- $1/\beta$ **Low Frequency = RF/RI = 10 → 20dB**
  - **Cn = Open at Low Frequency**
- $1/\beta$ **High Frequency = RF/(RI//Rn) ≈ 100 → 40dB**
  - **Cn = Short at High Frequency**
- **Zero in $1/\beta$ when Magnitude of X_{Cn} = RI**
  - **Magnitude X_{Cn} = 1/(2·Π·f·Cn)**
  - **fz = 1/(2·Π·RI·Cn) = 1kHz**
- **Pole in $1/\beta$ when Magnitude of X_{Cn} = Rn**
  - **fp = 1/(2·Π·Rn·Cn) = 10kHz**

Fig. 2.11: $1/\beta$ 1st Order Analysis For ZI

To check our 1st-order analysis our circuit for ZI analysis was built in Tina SPICE (Fig. 2.12). V_{IN} is set for a dc value of 0 V and an ac source option is selected with the ac amplitude set to 1. Our ac analysis is set to run from 10 Hz to 10 MHz with 100 points of data and amplitude & phase data requested to be saved for Postprocessing purposes. To perform our SPICE loop gain test we use L1, C1 and V_{IN} with convenient voltage probes placed at N1, N2, and N3. By inspection of this circuit we see Aol = N2/N1 and $1/\beta$ = N3/N1.
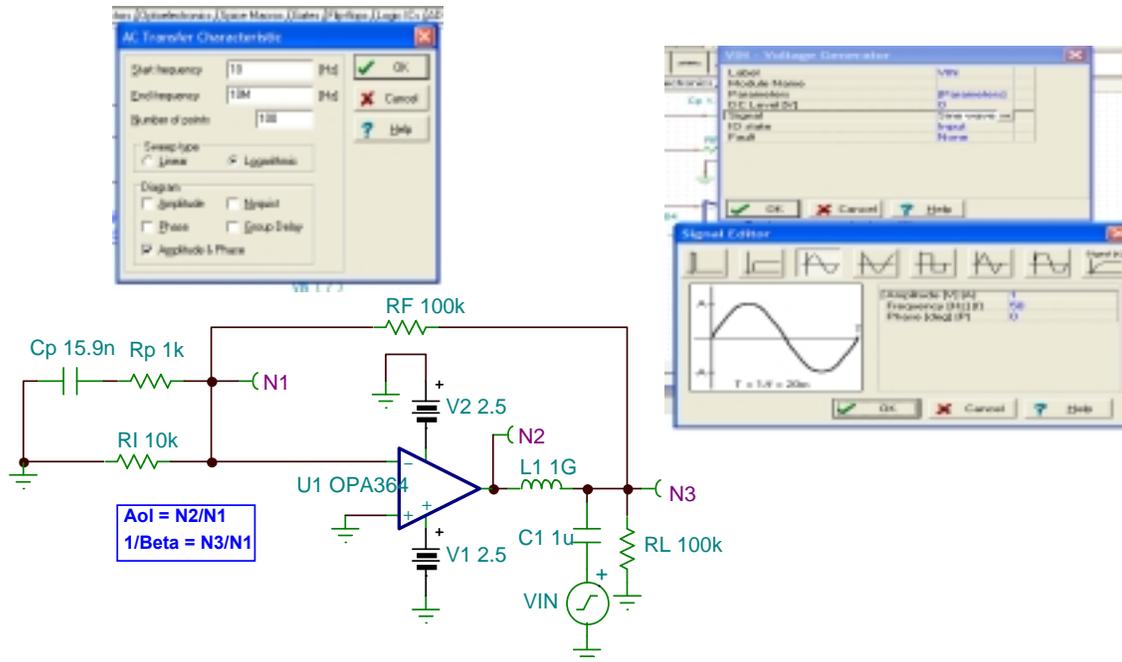
Fig. 2.12: Tina SPICE Circuit For ZI Analysis

In our optimally-scaled Tina SPICE simulation results (Fig. 2.13) the purple colored text denotes our 1st-order analysis predictions. The cursors are set for an exact amplitude difference of +3dB from the low-frequency 1/β and -3dB from the high-frequency 1/β. The predictions and results are not exact but certainly more than acceptable for powerful and intuitive ac stability analysis.
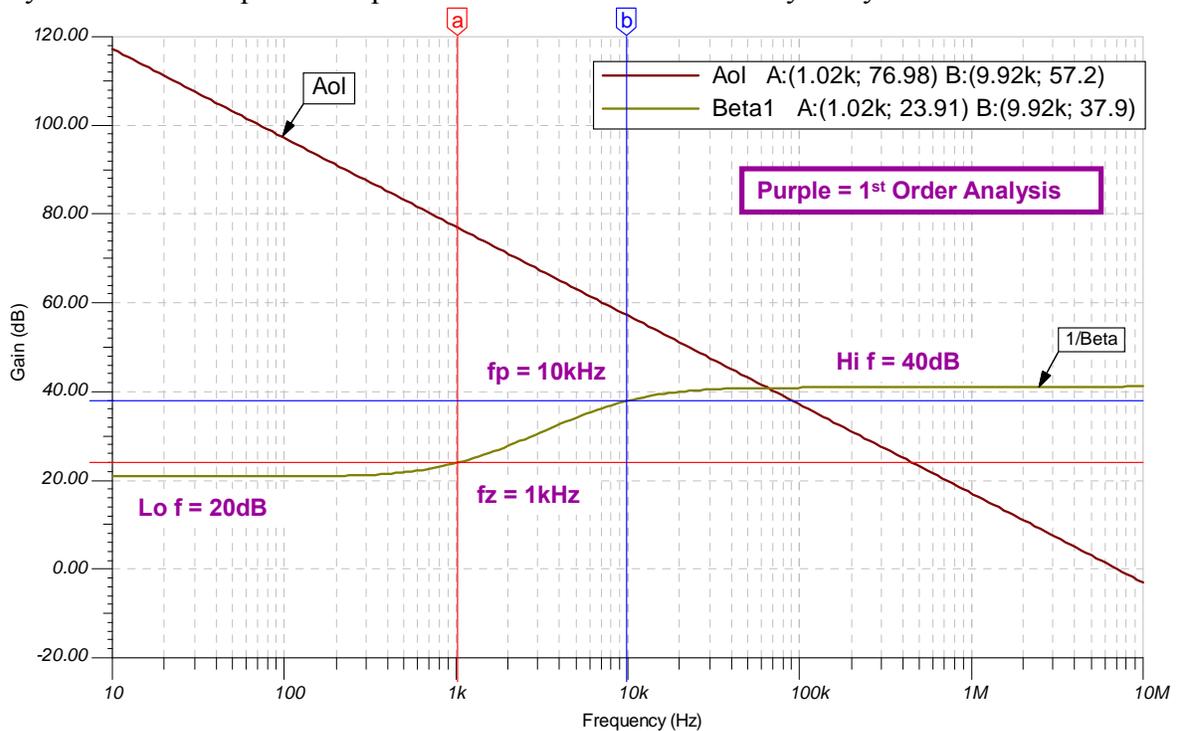

Fig. 2.13: Tina SPICE Optimally-Scaled Results For ZI Analysis

Simple Op Amp Ac SPICE Model

As we have seen, SPICE can be a powerful tool to check our 1st-order analysis. However, for ac stability analysis it requires that we have an op amp model to build our circuit with. Perhaps we do not have a SPICE model available but do have the data sheet for the op amp being used. For this example we will pretend we do not have an op amp model for the OPA364, a single-supply, RRIO, CMOS op amp but we have the open-loop gain/phase plot from the data sheet (Fig. 2.14). A common characteristic of CMOS op amps is exhibited: that the low-frequency open-loop gain is load dependent (the default 10 kΩ load is shown as well as a 100kΩ load). From the phase portion of the plot we use our log scaling technique (see Part 1) to determine that at -45° the frequency is 29 Hz. The unity-gain bandwidth of the OPA364 is 7.4 MHz. We first develop a simple op amp ac SPICE model using a two-pole approach with the second pole, fp1, set at the frequency where the phase dips to -135°.
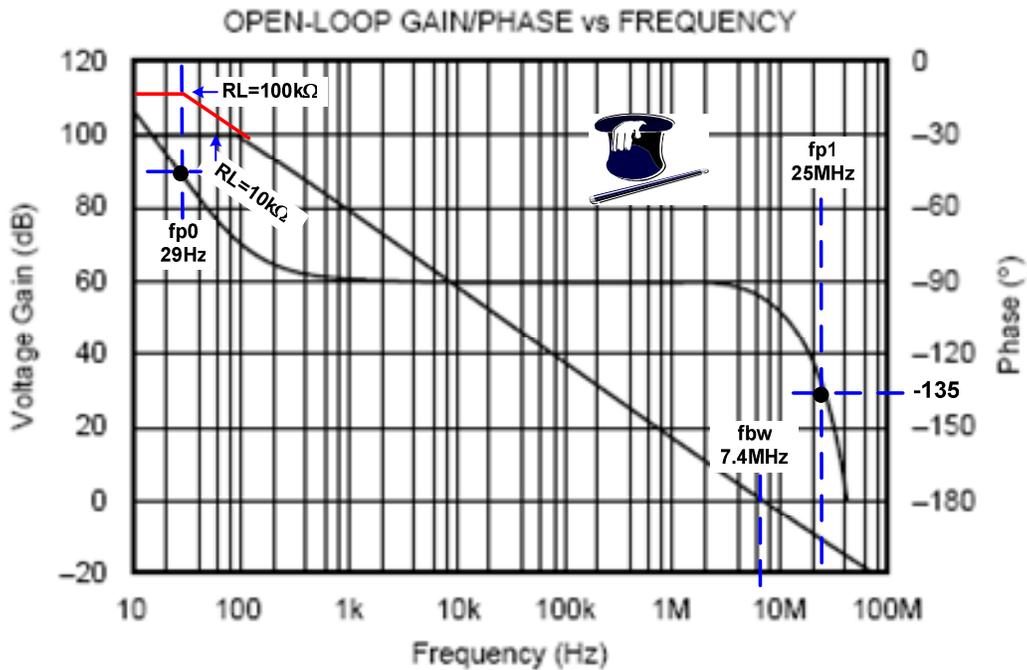

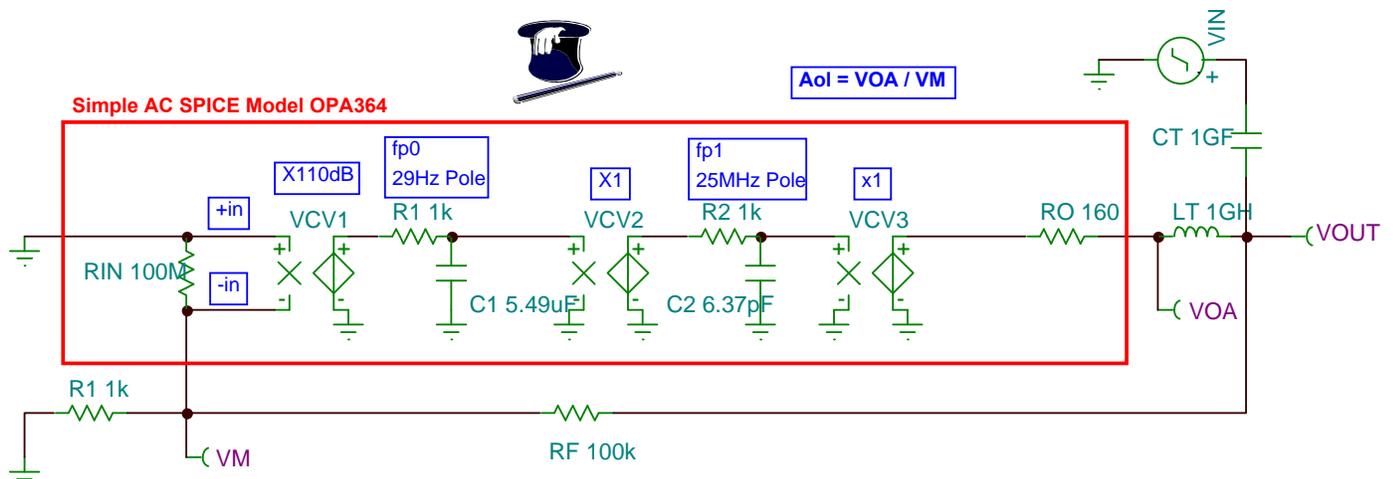Fig. 2.14: Simple Op Amp Model: OPA364 Data Sheet Curve

Fig. 2.15: Simple Op Amp Ac SPICE Model For The OPA364

The key frequency components (Fig. 2.15) are the elements used to form fp0 and fp1. Note that the voltage-controlled voltage sources, VCV1, VCV2, and VCV,3 provide perfect buffering between our frequency elements and prevent them from interacting with, or loading, down one another. The other important element is RO, the op amp's ac, small-signal, open-loop output impedance. We will study this in detail in Part 3 of this series, discussing how to obtain RO from either the manufacturer's data sheet or by measurement. Here we will assign a value of 160 Ω to RO for this OPA364 ac model which will run very quickly in SPICE. If our main concern is a good stable design then this will be about all we need. We also show our use of SPICE loop gain testing (Fig. 2.15, again) with LT, CT and V_{IN} with voltage probes placed at VM, VOA, and V_{OUT}. By circuit inspection, Aol = VOA/VM.
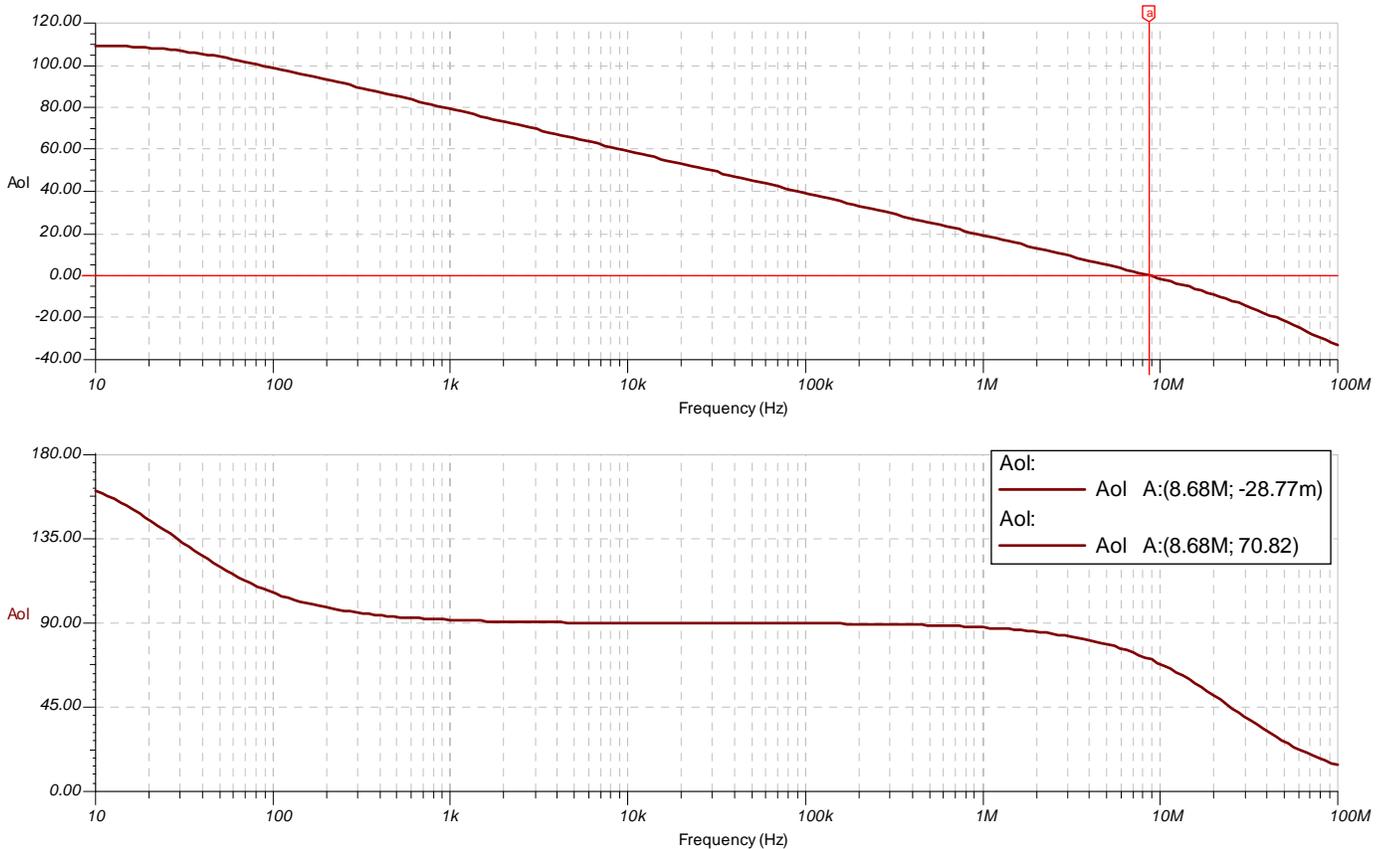

Fig. 2.16: Simple Op Amp Model: Ac SPICE Results

Our optimally-scaled Tina SPICE simulation (Fig. 2.16) shows phase results in SPICE start at 180° and go down to 0°. Typical data sheet curves show phase starting at 0° and going down to -180°. This is because most of these plots are viewed from passing a signal through the non-inverting input of the op amp to the output. The postprocessing math performed by SPICE to yield our desired results ends up with a 180° phase factor since we are computing VOA (voltage out of the op amp) divided by VM (the inverting input of the op amp which implies a -1 factor or 180° phase shift). To view this in direct comparison with the data sheet subtract 180° from each value on the y-axis. In the phase plot above we see a 70.82° reading at the unity-gain bandwidth frequency of 8.68 MHz which would equate to -109.18 degrees (70.82 – 180) on a data sheet open-loop gain/phase plot. This looks close to the phase shift on the data sheet plot at fbw = 7.4 MHz (Fig. 2.14). If we wanted our model to exactly match fbw = 7.4Mhz we could reduce the low frequency-Aol amplitude slightly.

Detailed Op Amp Ac SPICE Model

If we want to duplicate the high-frequency phase effects of the OPA364 we can create a detailed op amp SPICE model. On the data sheet open-loop gain/phase plot (Fig. 2.17) we draw phase slopes in ever-increasing multiples of -45°/decade. This information will allow us to compute where we want to place higher order poles to get the shown response.
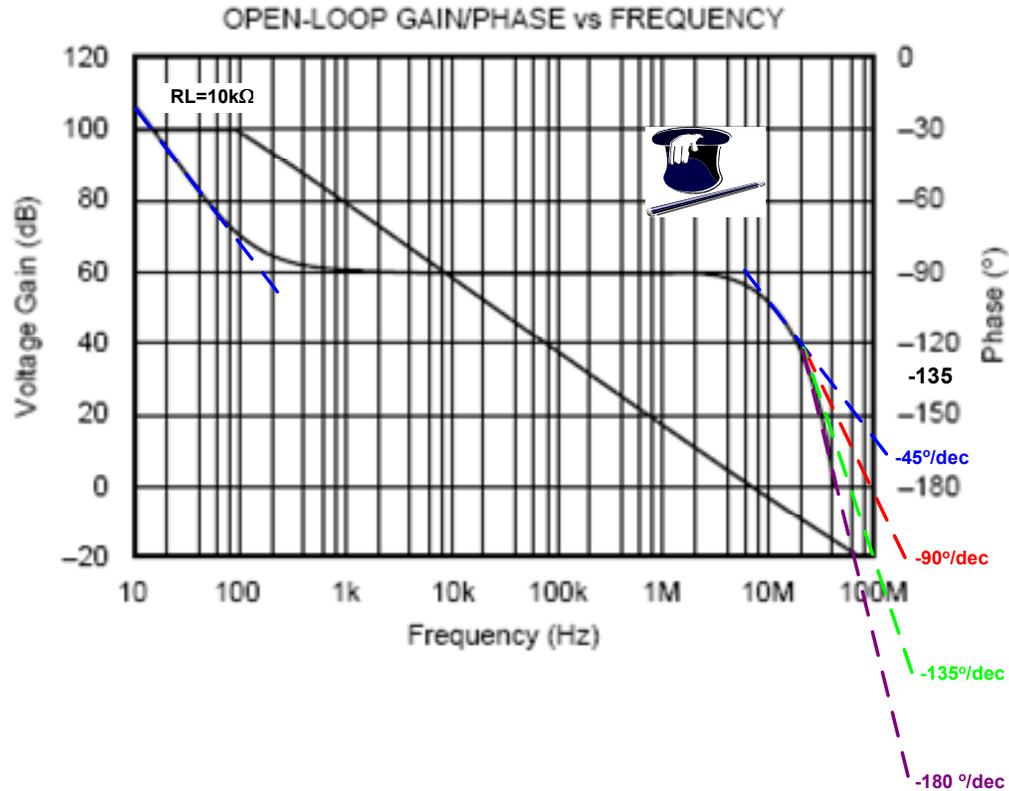

Fig. 2.17: Detailed Op Amp Model: OPA364 Data Sheet Curve
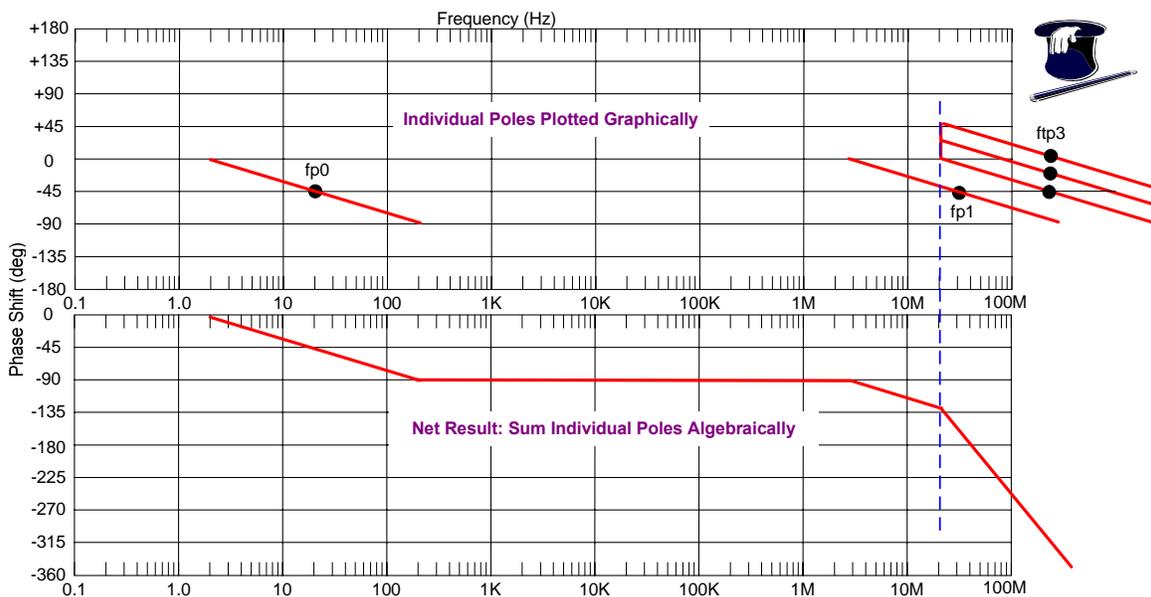

Fig. 2.18: Detailed Op Amp Model: Graphing Aol Phase Response

From Fig. 2.17 we can transfer the phase slope information into components which can create such a response. In Fig. 2.18 we place fp0 at the frequency where the phase is -45° on the data sheet plot in Fig.2.17. We place fp1 place at the frequency where open-loop phase is -135°. From Fig. 2.17 we observe that starting at 20 MHz there must be a -180°/decade slope. -45°/decade of this will come from fp1. Therefore, since a pole has phase impact a decade in frequency below and a decade in frequency above its actual location we know a decade above 20 MHz we must have 3 additional poles to get the desired slope. Graphically this is shown above as ftp3 (triple pole at fp3). The slope starting at 20 MHz must be -45°/decade and a decade away we will find the actual location of ftp3 (200 MHz). This graphical technique allows us an easy way to synthesize the desired phase response and also plot the resultant summation of each pole and/or zero.

The detailed op amp ac SPICE model adds 3 more high-frequency poles to match the data sheet open-loop gain/phase plot (Fig. 2.19).
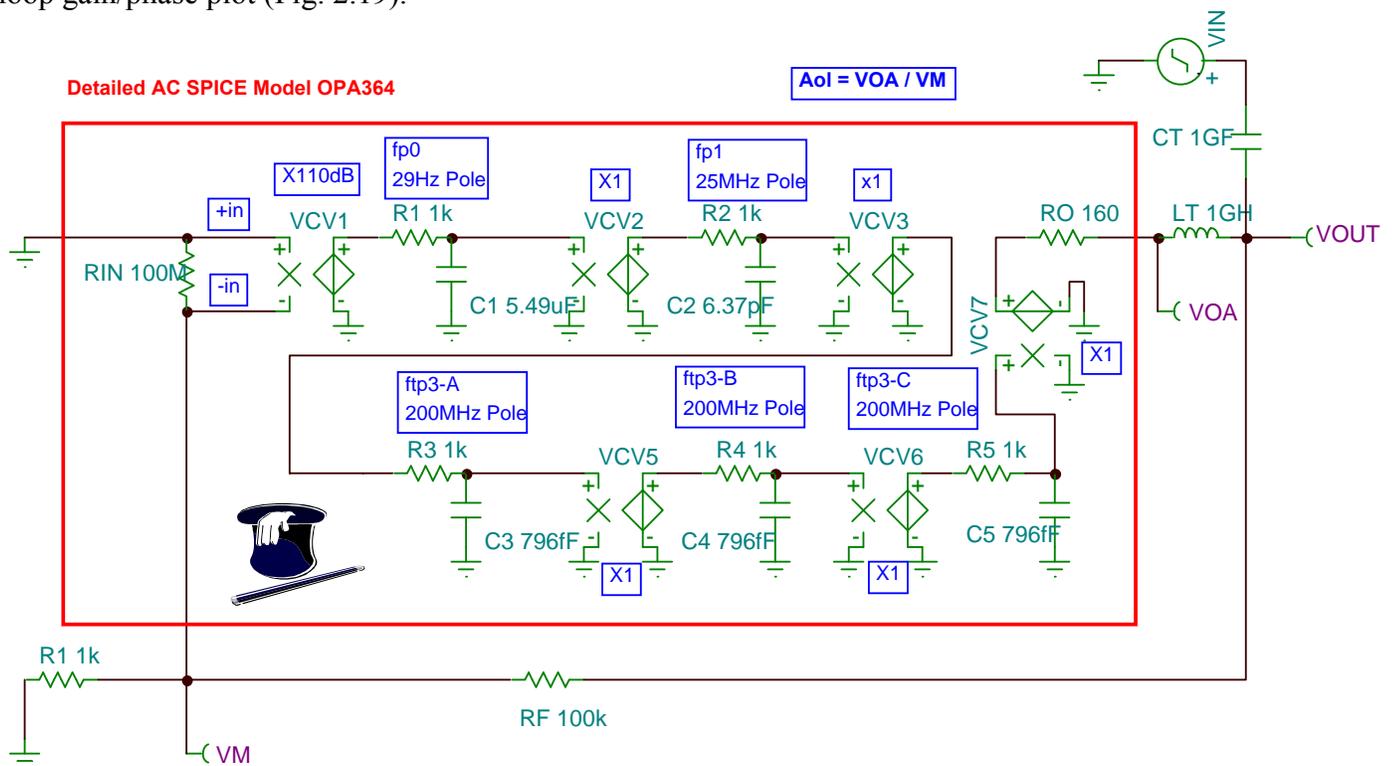

Fig. 2.19: Detailed Op Amp Model: Ac SPICE Model

Our optimally-scaled Tina SPICE simulation results for the detailed op amp ac SPICE model are shown in Fig. 2.20. A close look at these results versus those of the data sheet open-loop gain/phase plot reveals a very accurate replication created in our detailed op amp ac SPICE model. For most op amp stability analyses the simple op amp ac SPICE model will suffice. However, for cases where performance and bandwidth are being pushed as wide as possible we have a way to rather accurately model the high-frequency phase shifts of the op amp.
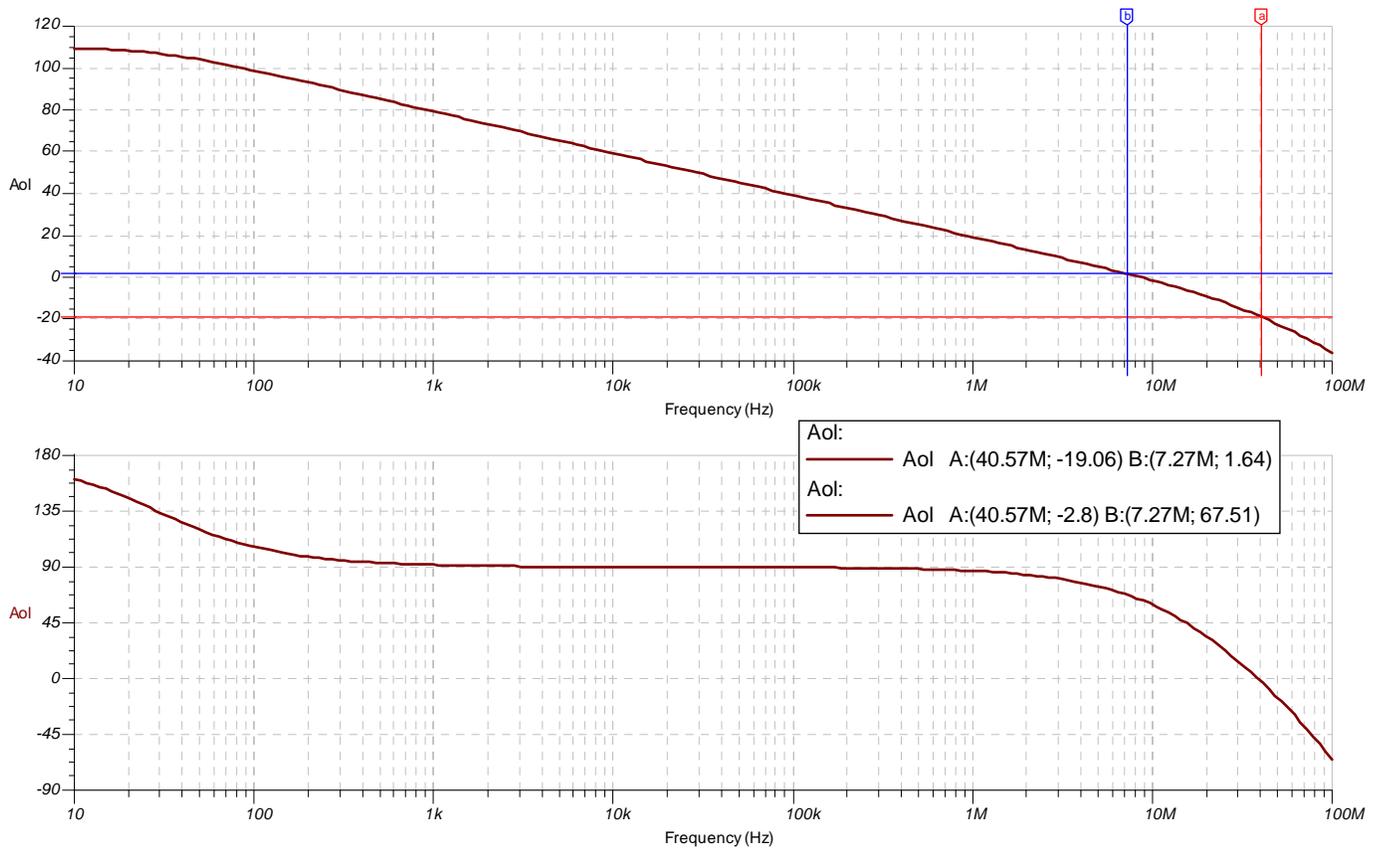
Fig. 2.20: Detailed Op Amp Model: Ac SPICE Results

Appendix: Blank Magnitude and Phase Plots

For ease of 1ˢᵗ-order analysis the last two pages of this part contain blank magnitude and phase plots

.
About The Author

After earning a BSEE from the University of Arizona, Tim Green has worked as an analog and mixed-signal board/system level design engineer for over 23 years, including brushless motor control, aircraft jet engine control, missile systems, power op amps, data acquisition systems, and CCD cameras. Tim's recent experience includes analog & mixed-signal semiconductor strategic marketing. He is currently a Strategic Development Engineer at Burr-Brown, a division of Texas Instruments, in Tucson, AZ and focuses on instrumentation amplifiers and digitally-programmable analog conditioning ICs.

Gain (dB)

Frequency (Hz)

0.1　1.0　10　100　1K　10K　100K　1M　10M　100M

Phase Shift (deg)

Frequency (Hz)

+180
+135
+90
+45
C
-45
-90
-135
-180

+180
+135
+90
+45
C
-45
-90
-135
-180

0.1 1.0 10 100 1K 10K 100K 1M 10M 100M