

LTSpice

To measure the loop gain ¹⁾

¹⁾

$T = A\beta$) of an op-amp in LTspice, insert an AC source in series with the feedback loop, usually using an LC circuit to maintain DC bias. The loop gain is the ratio of the feedback signal $V(fb)$ to the injected stimulus $V(in)$. LTspice Loop Gain Simulation Steps

1. Open the Loop: Insert a 0V voltage source in the feedback path. A good technique is to insert it between the output and the feedback network to avoid changing the DC operating point.
2. Add Stimulus: Configure the inserted voltage source with an AC amplitude of 1 in the "Advanced" tab of the component properties.
3. Ensure Proper Biasing: Use a large capacitor (e.g., $1\text{ text{kF}}$)

in series with a large inductor (e.g., $1\text{ text{kH}}$) to inject the AC signal while allowing DC feedback,

1. Run Simulation: Add a `.ac dec 10 1 10meg` command to perform frequency analysis. Plot Loop Gain: After running, plot $V(fb)/V(in)$. Magnitude: $db(V(fb)/V(in))$ Phase: $ph(V(fb)/V(in))$
2. Analyze Phase Margin: The point where the gain is 0dB (magnitude is 1) reveals the phase margin ($180^\circ - |phase|$)

Key Considerations
DC Operating Point: The DC loop must remain closed. If the feedback is opened, the high open-loop gain will cause the output to rail.
Frequency Range: Sweep from low (e.g., 1Hz) to high (e.g., 100MHz) frequencies to capture the full gain bandwidth.

From:

<https://wiki.oscardegroot.nl/> - HomeWiki

Permanent link:

<https://wiki.oscardegroot.nl/doku.php?id=computing:ltspice&rev=1777911980>

Last update: **2026/05/04 16:26**

