

CREATING BODE PLOTS USING PSpICE AND PSpICE A/D WITH SCHEMATICS

In Schematics:

1. Draw your circuit schematic, including DC power supplies.
2. Use a Vac symbol as the input (stimulus) source, and edit its attributes to set ACMAG=1. (A VSRC source will also work)
3. From the Analysis menu, choose Setup, then click AC Sweep. Define the sweep's type, range, and resolution. (Usually logarithmic(Decade) selected. Range is from a low value (e.g. 1 or 0.01) to a high value (e.g. 1Meg or 100Meg)
4. Simulate.

When the simulation ends, Probe starts and displays any probes placed in the circuit. In Probe:

Magnitude Plot:

1. From the Trace menu, choose Add Trace.
2. Select DB() and put in the output node/input node. {DB stands for Decibels} (e.g. if V(R1:1) is my output node and V(V1:+) is my input node => I would put $DB(V(R1:1)/V(V1:+))$ in the Trace Expression Box.)

Phase Plot:

3. From the Plot menu, choose Add Plot to Window.
4. Select the empty plot by clicking on it to move the SEL>> label.
5. From the Trace menu, choose Add Trace.
6. Select P() and put in the output node/input node. {P stands for Phase} (e.g. if V(R1:1) is my output node and V(V1:+) is my input node => I would put $P(V(R1:1)/V(V1:+))$ in the Trace Expression Box.)