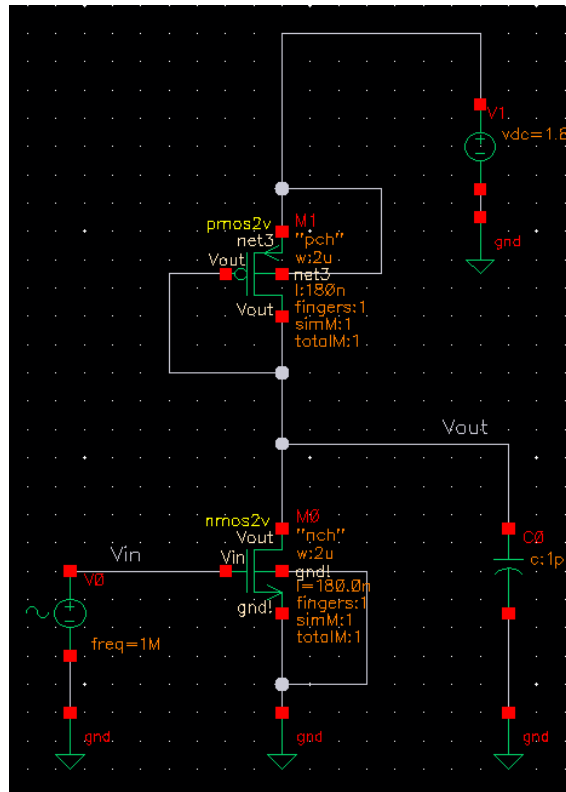


AC Simulation

Author: Jinhua Wang

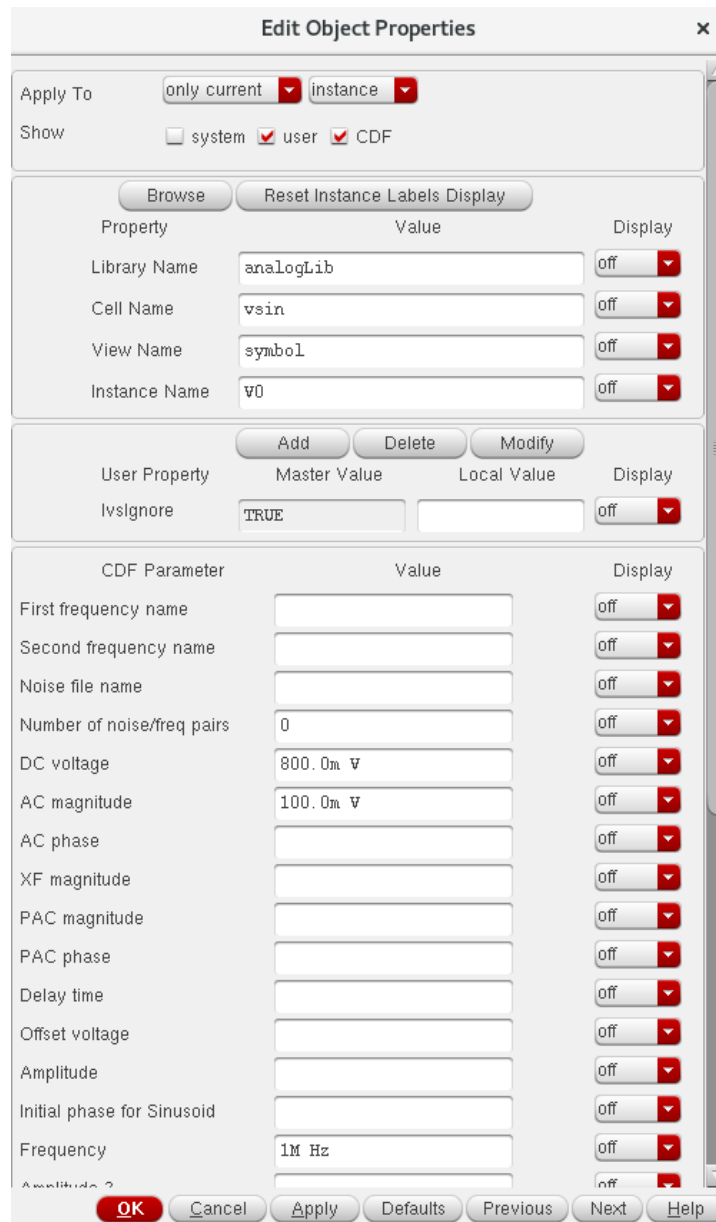
1. Complete the schematic as shown below
 - a. Use **vsin** from the library of **analogLib** as the input signal
 - b. Use **vdc** from the library of **analogLib** for VDD
 - c. Use **cap** from the library of **analogLib** as the load

Note: if you do not know how to create a schematic, please refer to “01_Schematic_Creation”



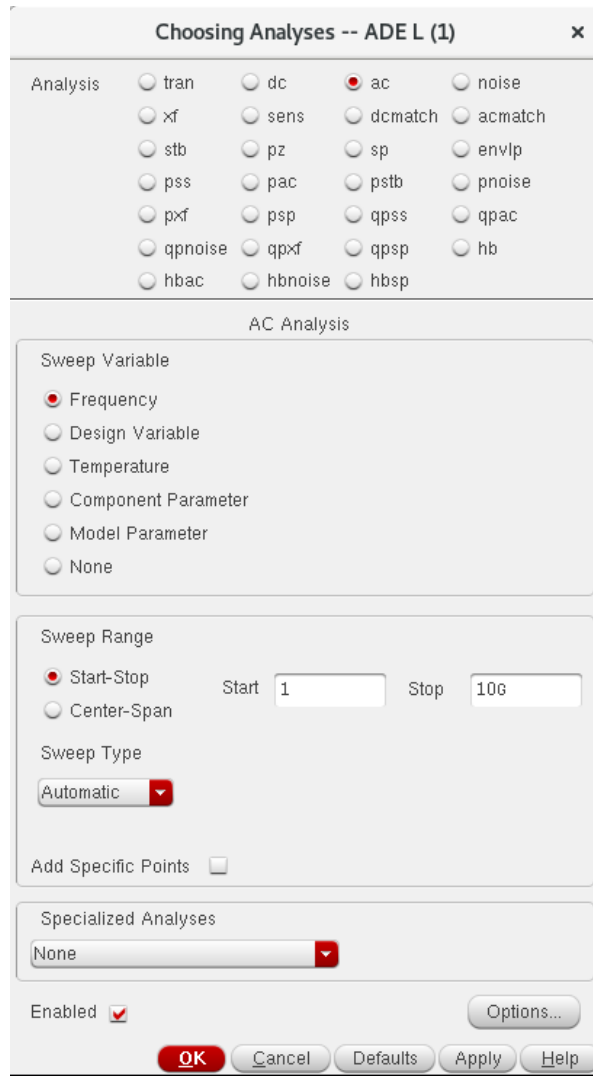
2. Component setup

- a. Set **vsin** offset voltage of **0.8 V**, amplitude of **0.1 V**, and frequency of **1MHz**.



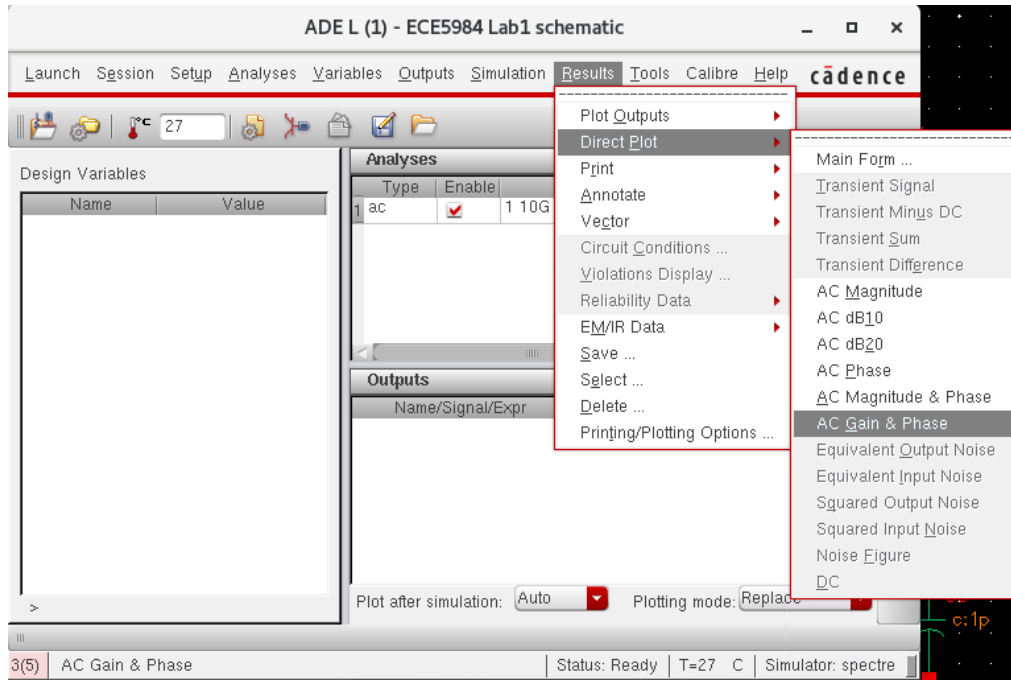
- b. Set **1.8 V** for **vdc**

3. In the **Schematic Editor**, click **Launch**, and then click **ADE L**
4. In **ADE L**, click **Choose Analyses**
5. Select **ac** as the type of **Analysis**, and set the frequency range from **1** to **10G**, and click **OK**

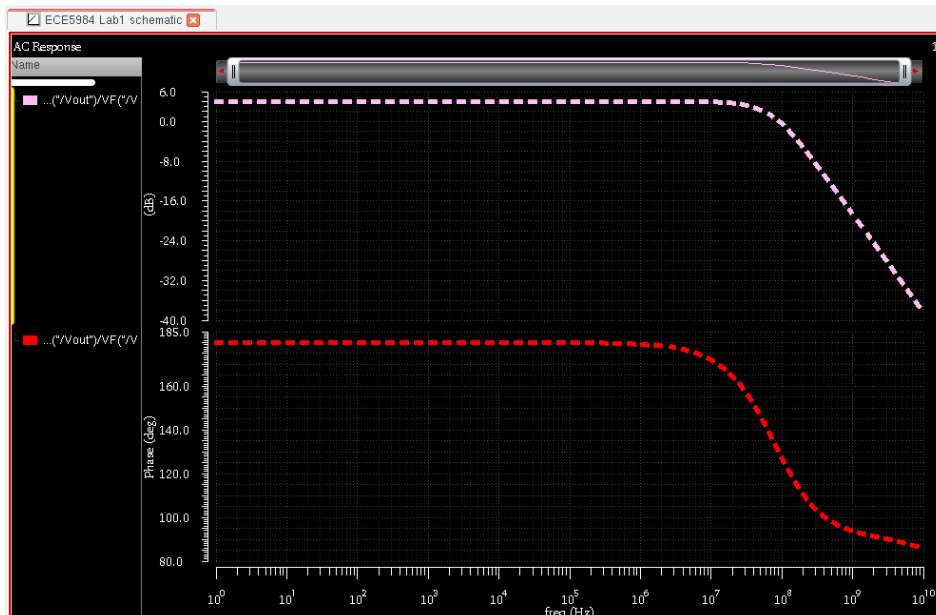


6. In ADE L, click **Netlist and Run ("Play" Button)**

- Once the simulation is finished, in **ADE L**, click **Results**, and then click **Direct Plot**, and then **AC Gain & Phase**



- The Schematic Editor will pop up, and select the signal path (wire) of “Vout” and “Vin” orderly (the order is important, which means Vout/Vin)
- The AC simulation plot should pop up



Note: This is not the only way to generate an AC simulation plot. There are many good online resources including documents and videos. I suggest you take a look at them.