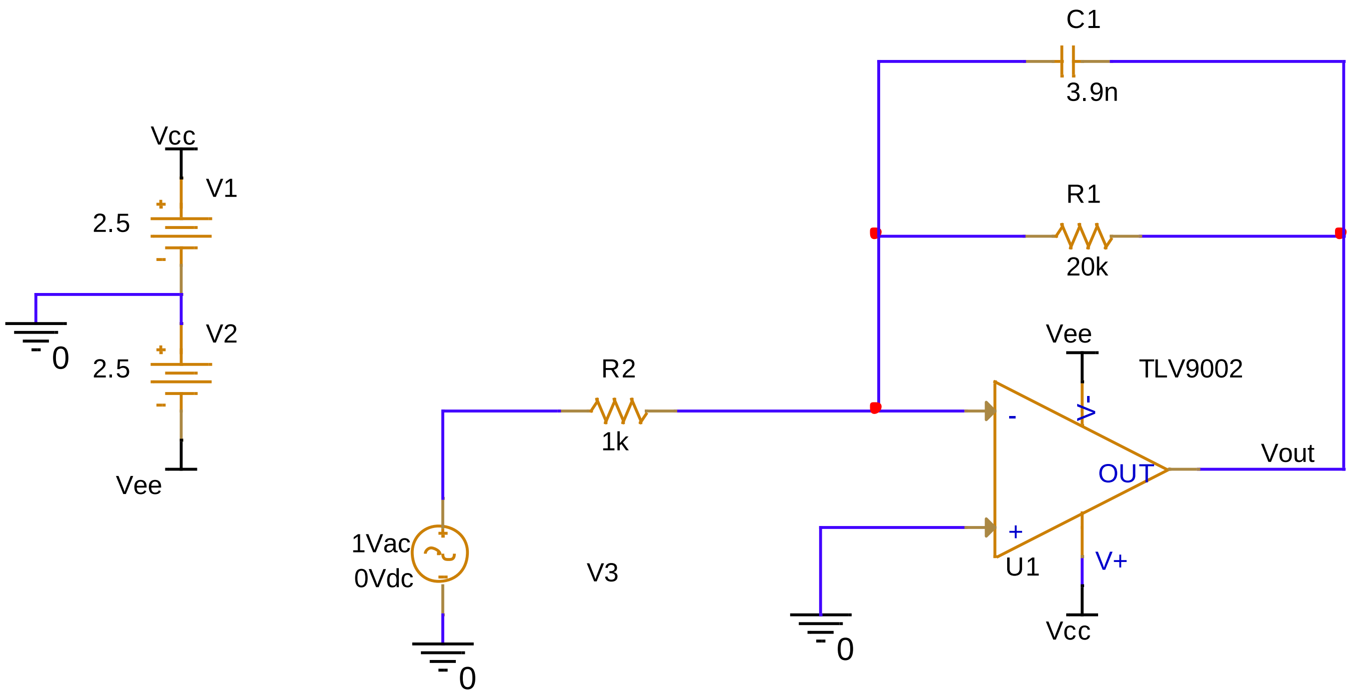
AC analysis

TI Precision Labs – PSpice® for TI

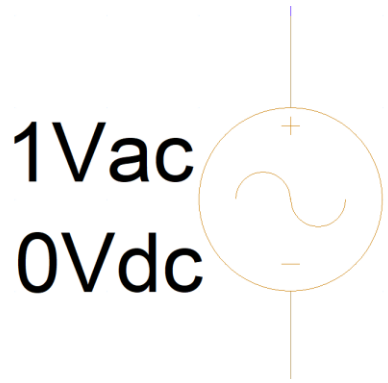
Example circuit

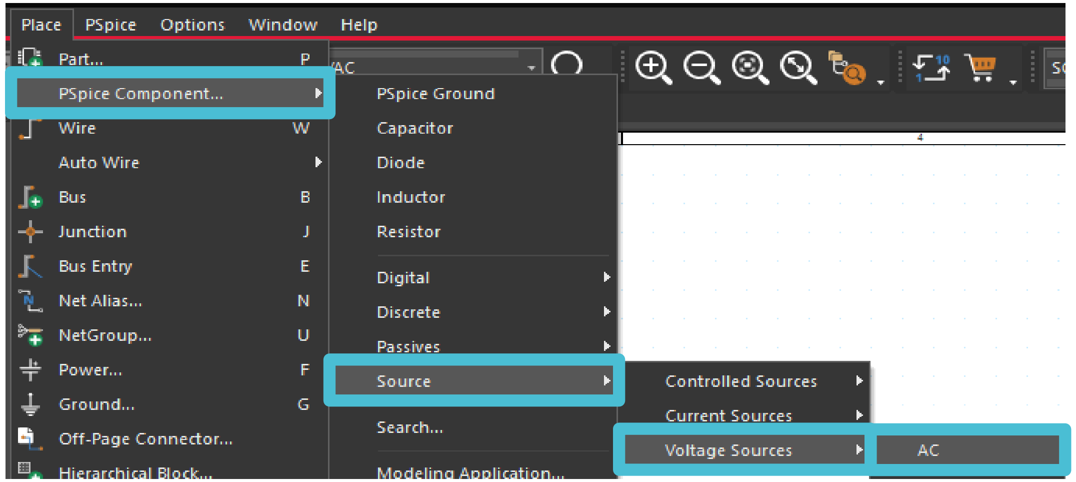
- In this example, we will be using the following circuit, which can be found in the Analog Engineer's Cookbook



Initializing your inputs

- Place an AC voltage source in your schematic by searching for “VAC” in the PSpice part search
  - You may also use the task bar. Place > PSpice Component… > Source > Voltage Sources > AC

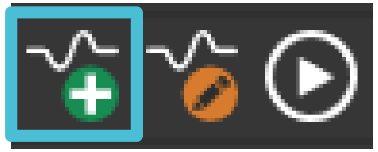




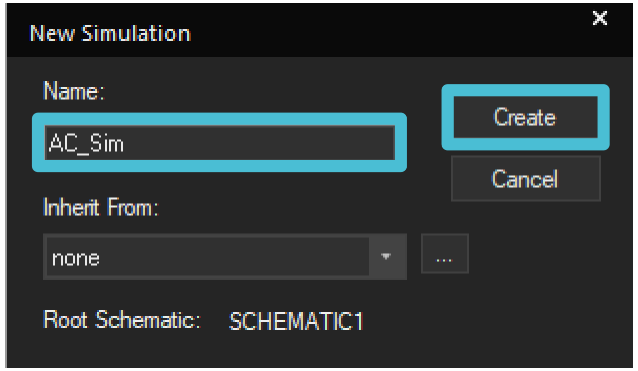
Texas Instruments

Create a new simulation

- In your toolbar, select the new simulation button

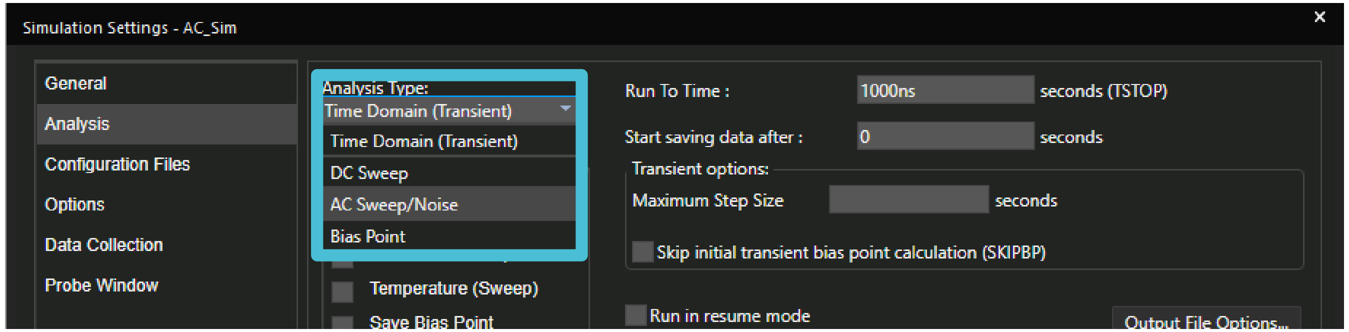


- Name your simulation, then press “Create”

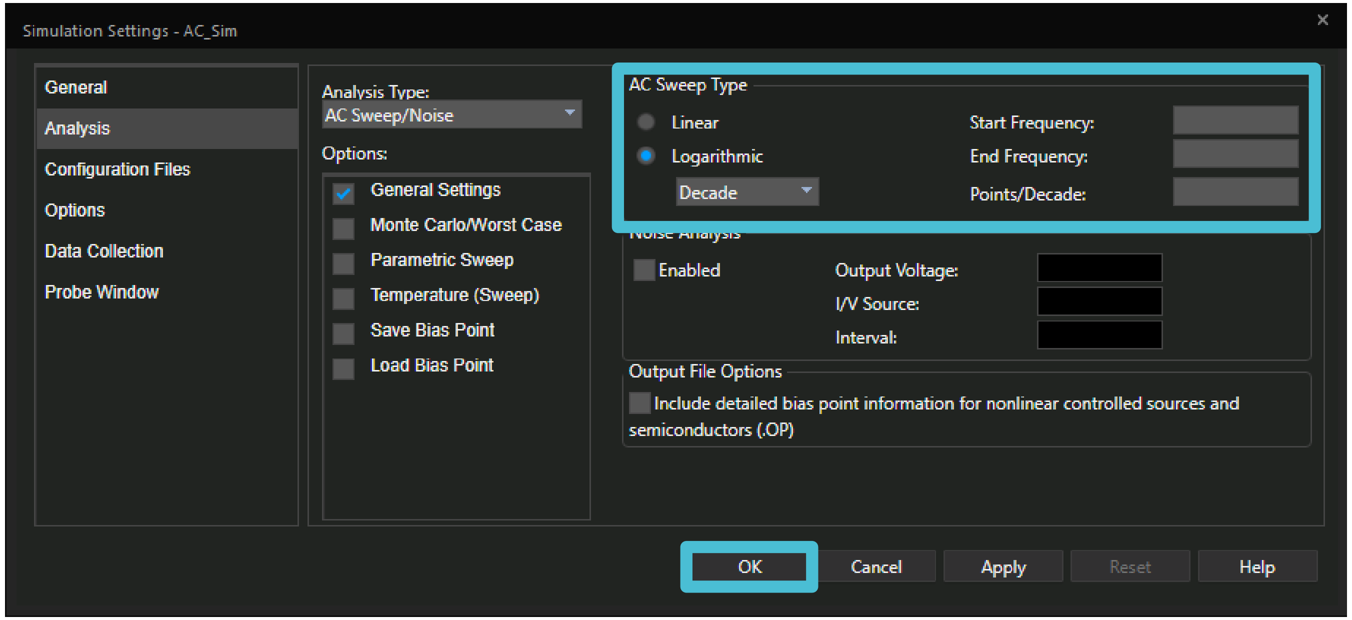


Initializing your simulation

- Under Analysis type select “AC Sweep/Noise”

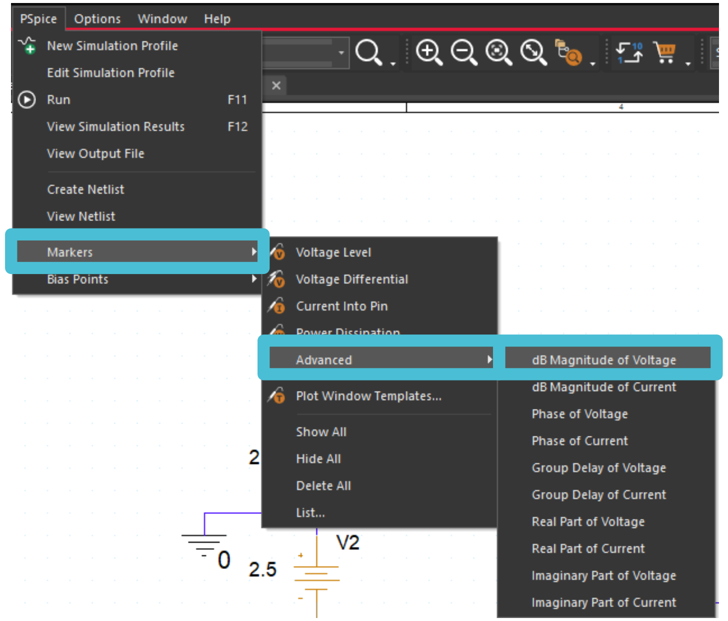


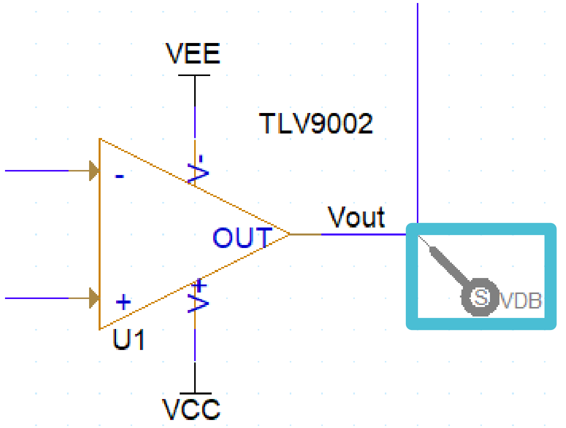
- Under “AC Sweep Type” enter the Start Frequency as 10Hz, the End Frequency as 100kHz, and Points/Decade as 100
- Click OK



Initializing your outputs

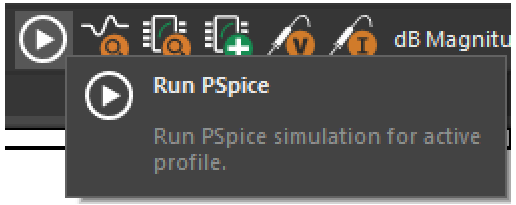
- To measure the output voltage in decibels navigate to PSpice > Markers > Advanced > dB Magnitude of Voltage
- Use left-click to place the marker on the wire

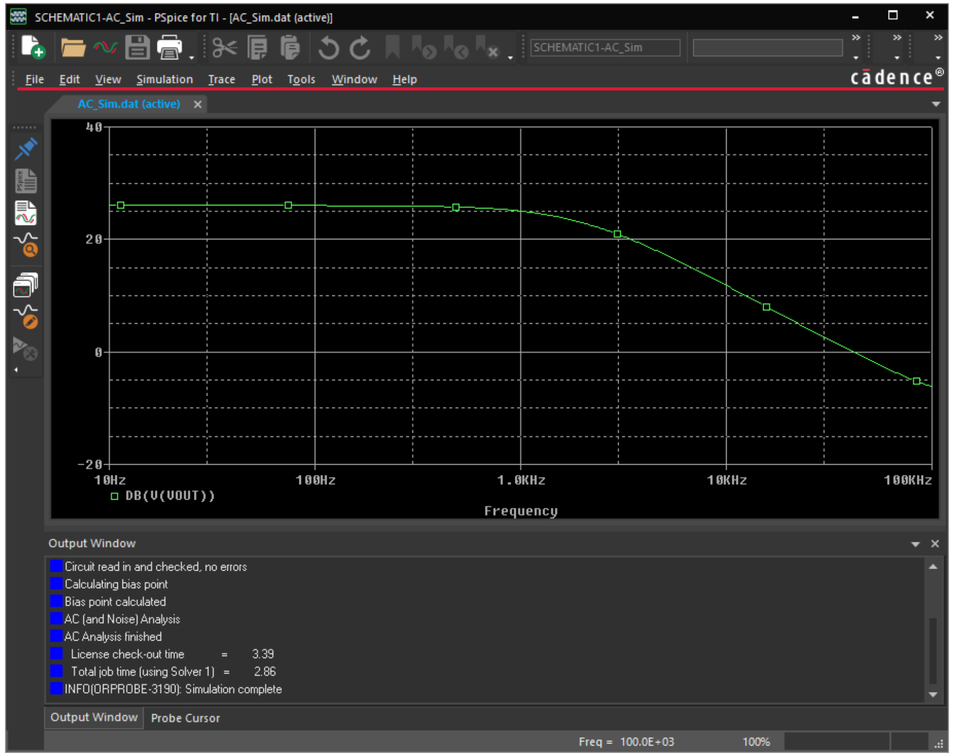




Running your simulation

- Run your simulation by pressing the play button or by pressing F11
- A new window will open with your simulation results





Thanks for your time!

To find more **PSpice® for TI** technical resources and search products, visit **ti.com/tool/PSPICE-FOR-TI**.