AC analysis TI Precision Labs – PSpice® for T

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



Plac	e PSpice Options	Window He	lp						
6	Part	P (AC	- 0)	$\oplus \Theta$	\odot \odot	6	۲	<u>س</u> ا
	PSpice Component	•	PSpice Ground		~~~	~~~	. 🙂 .	1	••••
.]*	Wire	w	Capacitor					4	
	Auto Wire	•	Diode						
J.	Bus	в	Inductor						
+-	Junction		Resistor						
7	Bus Entry	E	Digital						
ړي.	Net Alias	N	Discrete						
≥∓	NetGroup	U	Passives						
÷	Power	F 🔽	Source	,	Control	led Source	< ▶		
Ť	Ground	G 📛			Current	Sources	•		
<u>-</u>	Off-Page Connector		Search		Voltage	Sources		٨C	
≡_	Hierarchical Block		Modeling Application		voltage	Joances		AC	

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



5

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!



8

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

