## **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.

Edit Object Properties ×						
Apply To Only current	: 🔽 instance 🔽					
Show system	Show					
Browse	Reset Instance Labels Display					
Property	Display					
Library Name ar	NalogLib					
Cell Name 🗸 🗸	in	off 🔽				
View Name S3	mibol.	off 🔽				
Instance Name 🛛 👽	)	off 🔽				
	Add Delete Modify					
User Property	Master Value Local Value	Display				
Ivsignore TF	RUE	off 🔽				
CDF Parameter	Value	Display				
First frequency name		off 🔽				
Second frequency name		off 🔽				
Noise file name		off 🔽				
Number of noise/freq pairs	0	off 🔽				
DC voltage	800.0m ¥	off 🔽				
AC magnitude	100.0m V	off 🔽				
AC phase		off 🔽				
XF magnitude		off 🔽				
PAC magnitude		off 🔽				
PAC phase		off 🔽				
Delay time		off 🔽				
Offset voltage		off 🔽				
Amplitude		off 🔽				
Initial phase for Sinusoid		off 🔽				
Frequency	1M Hz	off 🔽				
Amalituda 2	Annely Defaults Dominant					

- 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**

	Choosin	g Analyses	ADE L (1	L)	×	
Analysis	<ul> <li>tran</li> <li>xf</li> <li>stb</li> <li>pss</li> <li>pxf</li> <li>qpnoise</li> <li>hbac</li> </ul>	<ul> <li>dc</li> <li>sens</li> <li>pz</li> <li>pac</li> <li>psp</li> <li>qpxf</li> <li>hbnoise</li> </ul>	<ul> <li>ac</li> <li>dcmatch</li> <li>sp</li> <li>pstb</li> <li>qpss</li> <li>qpsp</li> <li>hbsp</li> </ul>	<ul> <li>noise</li> <li>acmatch</li> <li>envlp</li> <li>pnoise</li> <li>qpac</li> <li>hb</li> </ul>		
		AC Analys	is			
Sweep Va	riable					
<ul> <li>Freque</li> <li>Design</li> <li>Temper</li> <li>Compo</li> <li>Model</li> <li>None</li> </ul>	ncy Variable rature nent Paramet Parameter	er				
Sweep Rau Start-S Center- Sweep Typ Automatic	nge top S -Span De	tart 1	Stop	106		
Add Specific Points						
Specialize	d Analyses					
None		<b>•</b>				
Enabled ⊻	<u>0</u> K	<u>C</u> ancel	Defaults	Options	.) elp	

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

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

ADE	_ = ×				
Launch Session Setup <u>A</u> nalyses <u>V</u> ar	riables <u>O</u> utputs <u>S</u> imulation	<u>R</u> esults <u>T</u> ools Calibre	e <u>H</u> elp	cādence	
Image: Second state sta	Analyses Type Enable 1 ac 1 10G 1 ac 1 10G 1 mm Outputs Name/Signal/Expr	Plot Qutputs Direct Plot Print Annotate Vector Circuit Conditions Violations Display Reliability Data EM/IR Data Save Sglect Delete Printing/Plotting Option Plotting mode: Status: Readuly T-27	nns	Main Form Transient Signal Transient Minus DC Transient Sum Transient Diffgrence AC Magnitude AC dB10 AC dB20 AC Phase <u>AC Magnitude &amp; Phase</u> <u>AC Gain &amp; Phase</u> Equivalent Output Noise Equivalent Output Noise Squared Output Noise Squared Input Noise Squared Input Noise Squared Input Noise Squared Input Noise Squared Input Noise Noise Figure <u>D</u> C	
3(5)     AC Gain & Phase       Status: Ready     T=27       C     Simulator: spectre					

- 8. 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**)
- 9. 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.