PSpice with Orcad 10

- 1. Creating Circuits Using PSpice Tutorial
- 2. <u>AC Analysis</u>
- <u>Dependent Sources</u>
 <u>Variable Phase VSin Source</u>

Creating Circuits using PSpice

Start \rightarrow Orcad 10.5 \rightarrow Capture CIS Demo

File \rightarrow New \rightarrow Project

New Project		×
Name example1 Create a New Project Using Image: Ima	Tip for New Create a n Mixed A/D new projec or copied f template.	OK Cancel Help Users w Users project. The trom an existing
L <u>o</u> cation H:		Browse

Type in example1 in the **Name** field. Select **Analog or Mixed A/D.** Click the **Browse...** button

Select Directory	
Directories:	ОК
h:\pspice2	Cancel
PSPICE2	<u>H</u> elp
example6-PSpicef	Cr <u>e</u> ate Dir
Dri <u>v</u> es:	
📼 h: \\winserver\Hon 💌	Net <u>w</u> ork

Select the h:\ directory and click on Create Dir...

Create Directory	×
Current Directory: h:\pspice2	ОК
Name: PSpice	Cancel
	Help

Enter PSpice in the Name: field. Click OK.

Select Directory	
<u>D</u> irectories:	OK
h:\pspice2\pspice	Cancel
PSPICE2	<u>H</u> elp
	Cr <u>e</u> ate Dir
Dri <u>v</u> es: I⊂ h: \\winserver\Hon ▼	Network

Double click on **PSpice** to select it. Click OK. That will take you back to the original **New Project** dialog box shown below.



Click **OK** to create the new project.

Create PSpice Project	×
Create based upon an existing project	ОК
empty.opj	Browse
Create a blank project	Cancel
	<u>H</u> elp

Select **Create a blank project**. Click OK. This will open the schematic editor.



Schematic Editor with all the buttons used in this tutorial labeled.

After the schematic editor opens, select the Place Part tool by clicking on the second button on the vertical toolbar on the right side shown above.

Place Part		
Part:		ОК
Part List:		Cancel
		Add Library
		Remove Library
		Part <u>S</u> earch
		<u>F</u> ilter
		Help
Libraries: Design Cache	Graphic Graphic Grant Convert Packaging Path and Plant 1	
	Paris per Fikg: T Paris Type:	

Select Add Library...

Browse File	? 🛛
Look jn: ଢ	pspice 💽 🗲 🔁 🛗 📰 -
B abm.olb analog.olb analog_p.o breakout.o eval.olb evalp.OLB	source.olb sourcstm.olb blb special.olb blb
File <u>n</u> ame:	source.olb
Files of <u>type</u> :	Capture Library(*.olb)
	C Open as read-only

On the screen select **Source.olb** and click **Open**.

Place Part			X
<u>P</u> art: VDC		-	ОК
Part List: ISFFM ISIN ISRC STIM1 STIM16 STIM4			Cancel Add Library Bemove Library Part Search
STIM8 VAC VDC VEXP VPULSE VPWL	C:\ORCAD\ORCAD_10.0_DEM		<u>S\CAPTURE\LIBRARY</u>
Libraries: Design Cache SOURCE	Graphic Normal Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous	0V	dc <u>+</u> ♥ ∨? -⊤

On this screen select a battery (VDC) then **OK**. Place the battery on the work area by clicking on the blank schematic page and click ESC to stop placing DC sources. Then double click on 0VDC to change the DC voltage. The following dialog box will be displayed:



Change value to $\overline{10VDC}$ and then click **OK**.

Click on the **Place Part** button, then the **Add Library...** button.

Browse File	? 🔀
Look jn: 隘	pspice 🔽 🗲 🗈 📸 📰 -
I abm.olb analog.olb analog_p.o breakout.o eval.olb I evalp.OLB	source.olb sourcstm.olb olb special.olb
File <u>n</u> ame:	analog.olb Dpen
Files of <u>type</u> :	Capture Library(*.olb)
	Open as read-only

Select analog.olb and press Open.

Place Part		×
<u>P</u> art: R		 ОК
Part List: FPOLY G GPOLY H HPOLY K_Linear L OPAMP R R_var S T		Cancel Add Library Remove Library Part Search Eilter
L <u>i</u> braries: ANALOG Design Cache SOURCE	Graphic Normal Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous	R? ∕//∕ 1k

Select the ANALOG library and select **R** from the **Part List**. Click **OK**.

Place two resistors in the work space. To stop placing parts, press ESC or right click an item and select "End Mode". To rotate an part, select the part and press 'r' or right click and select rotate from the drop down menu. If you need to delete a part, select the part and press the Delete key.

To connect the items in the circuit, select the **Place Wire** tool by clicking on the third button from the top on the vertical toolbar on the right side. Drag the mouse between the terminals of your placed parts to connect them.



In order for PSpice to simulate your circuit, it must have a "zero" node for a ground. To add this ground, select the **Place Ground** tool by clicking on the 9th button from the top on the vertical toolbar on the right side. The following dialog box will appear.



Select the **SOURCE** Library and the **0** Symbol followed by **OK**.

Place the ground for your circuit by clicking on the schematic page and connect it with the **Place Wire** tool. Your diagram should look like this:



To change the value for an item, double click the value you want to change and enter the value you want on the dialog box that appears. Double click on the **1k** value of the horizontally placed resistor and change the **Value** to 4k as shown below.

Display Properties	
Name: Value	Font Arial 7 (default)
Value: 4k	<u>Change</u> Use Default
Display Format ○ <u>D</u> o Not Display ④ <u>V</u> alue Only ○ Name and Value ○ <u>N</u> ame Only ⓒ <u>B</u> oth if Value Exists	Colog Default Rotation ⓒ 0° ○ 180° ⓒ 90° ⓒ 270°
ОК	Cancel <u>H</u> elp

It is important to name the nodes you want to plot in PSpice so you can find them easily. To name a node, select the **Place Net Alias** tool and the following dialog box appears. Change the **Alias** to Vout and click **OK**.

Place Net Alias		×
<u>A</u> lias:	ОК	٦.
Vout	Cancel	1
	<u>H</u> elp	
Color Default	Rotation ● <u>0</u> * ● <u>9</u> 0* ● <u>1</u> 80* ● <u>2</u> 70)*
Font Change Use Default	Arial 7 (default)	

Then place the alias on the desired node for Vout (i.e. the junction of the 2 resistors).

Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** tran as shown below.

New Simulation		
<u>N</u> ame:		Create
Inherit From:		Cancel
none		
Root Schematic:	SCHEMATIC1	

Click **Create** and the following dialog box will appear.

Simulation Settings - tran		×
General Analysis Configuration Analysis type: Imme Domain (Transient) Imme Domain (Transient) Options: Imme Domain (Transient) Imme Domain (Transient) <td>tion Files Options Data Collection Probe Window <u>R</u>un to time: 1000ns seconds (TSTOP) <u>S</u>tart saving data after: 0 seconds <u>I</u>ransient options <u>Maximum step size:</u> seconds <u>Maximum step size:</u> seconds <u>Cutput File Options</u></td> <td></td>	tion Files Options Data Collection Probe Window <u>R</u> un to time: 1000ns seconds (TSTOP) <u>S</u> tart saving data after: 0 seconds <u>I</u> ransient options <u>Maximum step size:</u> seconds <u>Maximum step size:</u> seconds <u>Cutput File Options</u>	
	OK Cancel Apply Help	

Click OK to accept the default settings of the Time Domain (transient) analysis.

From the top menu select

PSpice \rightarrow **Create Netlist**. This is only necessary so that you can add the **Voltage Level** Marker.

Click on the **Voltage/Level Marker** button to add a marker to the Vout node by clicking on it as shown below.



Click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.



This automatically plots nodes with Voltage (or Current) markers on them. In this case, the voltage divider gives you 2V for Vout.

Close the PSpice window. On the schematic, click the **Enable Bias Voltage Display** button to see all the DC voltages in the circuit as shown below.



Circuit with Bias Voltage Display Enabled

On the schematic, click the **Enable Bias Voltage Display** button to toggle it off. Then, click on the **Enable Bias Current Display** button to see all the DC current(s) in the circuit as shown below.



Circuit with Bias Current Display Enabled

AC Circuit Analysis

Transient Analysis:

For this circuit, select and wire these components: Capacitor (Place Part C in ANALOG library), Resistor (Place Part R in ANALOG library), Sinusoidal Source (Place Part VSIN in SOURCE library), and Ground (Place Ground 0 SOURCE library). Double click on the VOFF attribute of the VSIN component. The dialog box shown below should appear.



Change the value from $\langle VOFF \rangle$ to 0. Click OK. Similarly, set the VAMPL attribute to 1 and the FREQ attribute to 159000. (1/(2*PI*R*C)). This circuit is shown below:



Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** Tran and the following dialog box will appear. Set Analysis type: **Time Domain (transient)**, **Run To Time:** 20us, and **Maximum Step Size** to 100ns. Click **OK**.

Simulation Settings - Tran	
General Analysis Configural Analysis type: Time Domain (Transient) • Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	ion Files Options Data Collection Probe Window Bun to time: 20us seconds (TSTOP) Start saving data after: 0 seconds Iransient options Maximum step size: 100ns seconds Skip the initial transient bias point calculation (SKIPBP) Output <u>File Options</u>
	OK Cancel Apply Help

Click on the Voltage/Level Marker button and place a marker at the junction of the R and C and at the junction between the Source and the R. If you are unable to place the marker, you need to create the netlist first (PSpice \rightarrow Create Netlist). Give the junction between the R and C the Alias of Vout using the Net Alias tool as shown below. Give the junction between the Source and the R the alias Vin. This circuit is shown below:



Click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.



Click the **Toggle Cursor** button to display the cursors and the **Probe Curso**r window. Left mouse click on the red dot in the legend next to V(Vin). This will assign the left mouse button to the Vout trace. Drag the mouse using the left mouse button to 2^{nd} peak of Vout and note the amplitude. It should be 3dB smaller than Vin (0.707V). Click on the Mark Label button to label that point as shown below.



AC Analysis:

Close the PSpice simulation window. Then, modify the above circuit by deleting the **VSIN** component. Replace this component with the AC source (Place Part **VAC** in **SOURCE** library). The default attributes are correct. This circuit is shown below:



Next, configure the simulation by clicking the **New Simulation Profile** button on the top toolbar. Enter the **Name** AC and the following dialog box will appear. Select Analysis type: **AC Sweep/Noise**, AC Sweep Type: **Logarithmic** and enter the displayed values for **Start Frequency**, **End Frequency** and **Points/Decade**. Click **OK**.

Simulation Settings - AC			×	
General Analysis Configuration Analysis type: AC Sweep/Noise Image: Configuration AC Sweep/Noise Image: Configuration Image: Configuration Options: Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration of the configuration Image: Configuration Image: Configuration Iman	Files Options Data (AC Sweep Type C Linear C Logarithmic Decade	Collection Probe Window Start Frequency: 1 End Frequency: 1000000 Points/Decade: 10		
	Output File Options -	d bias point information for nonlinear ces and semiconductors (.OP) Cancel <u>Apply</u> Help		

Unfortunately, when you create a new simulation profile, the program deletes the voltage markers. Add a voltage marker the Vout node. Then, click the **Run PSpice** button and the PSpice Analysis results will appear as shown below.



Add Traces					
Simulation Output Variables			Eunctions or N	dacros	
×			Analog Opera	tors and Functions	-
Frequenci			tt.		
I(C1)	M.	<u>A</u> nalog	Ö		
l(C1:1) l(R1)	Γ	<u>D</u> igital	+		
I(B1:1) I(V1)	•	<u>V</u> oltages	;		
l(V1:+) V(0)	•	Cu <u>r</u> rents	@ ABS()		
V(C1:1) V(C1:2)	$\overline{\mathbf{v}}$	<u>P</u> ower	ARCTAN()		
V(N00093) V(N000100)		Nojse (V²/Hz)	AVG() AVGX(_)		
V(R1:1) V(R1:2)	•	Alias <u>N</u> ames	COS()		
V(V1:+) V(V1:-)	Γ	<u>S</u> ubcircuit Nodes	DB() ENVMAX(,)		
V1(C1) V1(B1)			ENVMIN(,) EXP()		
			G()		
V2(01) V2(R1)			LOG()		
V2(V1)	25	variables listed	LOG10()		
W(C1) W(B1)			MAX()		~
Full List					
Trace Europeanian				OK Cancel	Help
Trace Exhiession:					Teih

Click on the Add Trace button shown above and the following diaglog box will appear.

Now select **Plot Window Templates** from the **Functions or Macros** drop down menu and the following dialog box will appear.

Add Traces			
Simulation Output Variables			Eunctions or Macros
×			Plot Window Templates 🗨
Frequency		nalog	3dB Bandwidth - Band pass [multi-run][1 3dB cut-off frequency - High pass [multi
		igital	3dB cut-off frequency - Low pass [multi-
(B1:1) (V1)		oltages	Average(1) Bode Plot - dual Y axes(1)
1(V1:+) V(0)		lurrents	Bode Plot - separate(1) Bode Plot dB - dual Y axes(1)
V(C1:1) V(C1:2)	₽ P	ower	Bode Plot dB - separate(1) Conductance(1,2)
V(N00100)	ΠN	ojse (V²/Hz)	Current Gain(1,2)
V(R1:2) V(V1:+)	▼ A	lias <u>N</u> ames	DC Voltage Gain(1,2) DC Voltage Gain(1,2) Derivative(1)
V(V1:-)		ubcircuit Nodes	Falltime of Step Response [multi-run](1)
V1(C1) V1(R1) V1(V1) V2(C1) V2(R1) V2(V1) W(C1) W(R1) Full List	25 va	ariables listed	First Peak (multi-fun)(1) Fourier Transform(1) Impedance(1,2) Integral(1) Log-Linear(1) Log-Log(1) Nyquist Plot(1)
Irace Expression: Bode Plot dB - dual Y axes(V(Vout))			<u>Q</u> K <u>C</u> ancel <u>H</u> elp

Select Bode Plot dB - dual Y axes(1) on the right side. The cursor in the Trace Expression field at the bottom indicates the place to insert the signal name to plot. Select V(R1:2) (or V(C1:2) which is the same node) from the list of nodes on the left side and click OK.

This should be your result. The Phase is indicated by P(V(Vout)) in green (left Y-axis in degrees) and the Magnitude is indicated by DB(V(Vout)) in red (right Y-axis in dB).



Click the Toggle Cursor button to display the cursors and the Probe Cursor window. Left mouse click on the green dot in the legend next to P(V(Vout)). This will assign the left mouse button to the phase trace. Drag the mouse using the left mouse button to find the frequency where the phase is -45 degrees. Click on the Mark Label button to label that point as shown below.



Right mouse click on the red dot in the legend next to DB(V(Vout)). This will assign the right mouse button to the magnitude trace. Drag the mouse using the right mouse button to find the frequency where the magnitude is -3 dB. Click on the Mark Label button to label that point as shown below.



Dependent Sources



The 4 dependents sources available in the Analog library are shown below:

Voltage Controlled Voltage Source

Place Part			×
<u>P</u> art: F			ОК
Part List:			Cancel
C C shat		^	Add Library
C_elect C_var F			<u>R</u> emove Library
EPOLY			Part <u>S</u> earch
FPOLY G GPOLY H			<u>F</u> ilter
HPOLY K Linear		~	Help
Libraries: ANALOG BREAKOUT Design Cache EVAL SOURCE SOURCE SOURCSTM SPECIAL	Graphic © Normal © Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous	F	-? ↓ ↓ -

Current Controlled Current Source



Voltage Controlled Current Source

Place Part			X
Part: H		_	ОК
Part List:			Cancel
C C elect		^	Add Library
C_var E			<u>Remove</u> Library
EPOLY F			Part <u>S</u> earch
FPOLY G	C:\ORCAD\ORCAD_10.5_D	EMO\TO	OLS\CAPTURE\LIBRAR
GPOLY			
HPOLY K Linear		~	<u>H</u> elp
Libraries: ANALOG BREAKOUT Design Cache EVAL SOURCE SOURCE SOURCSTM SPECIAL	Graphic © Normal © Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous	P P	1 ? ∲ ♥ 1

Current Controlled Voltage Source

After placing the part, the Gain needs to be set. Double click on the part to bring up the Property Editor shown below:

ſ	Property Editor									
	New Column Apply Display	Delete Property	Filter by:	Current	properties >			✓ Help		
		Color	Designator	GAIN	Graphic	ID	Implementation	Implementation Path	Implementation Type	Location X-Co 🔺
	1 SCHEMATIC1 : PAGE1 : E1	Default		1/1/	E.Normal	////			<none></none>	760
										-
ľ	Parts & Schematic Nets	🖌 Flat Nets 🖌 Pins 🖌	Title Blocks	Glo	bals 🖌 Port	ৱ ৰ	1			►
l	Parts / Schematic Nets	KFlat Nets KPins K	Title Blocks	Glo	bals 🖌 Port	s 📢				×

Click on the 1 in the **Gain** field and change it to the appropriate value. Then click on **Display...** The dialog box below will be displayed.



Click **Yes**. The dialog box below will be displayed:

Display Properties	
Name:	Font Arial 7
Value: Display Format Do Not Display Value Only Name and Value	Color Default
 <u>Name Only</u> <u>B</u>oth if Value Exists 	Rotation
OK	Cancel <u>H</u> elp

Select **Name and Value** and press **OK**. Click the X in the upper right hand corner of the Property Editor to close it. You can select the Phase property on the schematic and move it to a more convenient spot.

Variable Phase VSin Source

The Phase isn't displayed as a property on the VSin component in the Source library. However, you can set it and display it. To do this, place the VSin component on your schematic. Double click on it to bring up the Property Editor as shown below:

ų	PAGE1																
٩	Property Editor																
1	New Column Apply Di	splay	Delete Property	Filter by:	Current propert	es >			▼ Help								
1		AC	Color	DC	Designator	DF FREG	Graphic	ID	Implementation	Implementation Path	Implementation Type	Location X-Coordinate	Location Y-Coordinate	Name	Part Reference	PCB Footprint	PHASE 🔺
1	1 SCHEMATIC1 : PAGE1 :	V4	Default		8	0	VSIN.Normal	1///	8		PSpice Model	840	140	JNS1031	∨4		0
ſ																	and the state
I																	
L																	-
L	Parts & Schematic N	ets 🖌 Fla	it Nets 🖌 Pins 🖌	Title Blocks	s 🖌 Globals 🖊	Ports ()	Aliases /			4							•

Click on the 0 in the **Phase** field and change it to the appropriate value. Then click on **Display...** The dialog box below will be displayed.



Click **Yes**. The dialog box below will be displayed:

Display Properties	
Name: PHASE Value: 90 Display Format © Do Not Display © Yalue Only © Name and Value © Name Only © Both if Value Exists	Font Arial 7 Change Use Default Color Default ■ Rotation ● 0° ● 180° ● 90° ● 270°
ОК	Cancel <u>H</u> elp

Select **Name and Value** and press **OK**. Click the X in the upper right hand corner of the Property Editor to close it. You can select the Gain property on the schematic and move it to a more convenient spot.