PSpice with Cadence

- 1. Creating Circuits
- 2. AC Analysis
- 3. <u>Step Response</u>
- 4. Dependent Sources
- 5. Variable Phase VSin Source

Creating Circuits

Select 'Start \rightarrow Engineering \rightarrow Cadence \rightarrow Capture' from the start menu.



When this dialog box appears, select Allegro PCB Design CIS XL

Select 'File \rightarrow New \rightarrow Project' in the menu bar.

New Project	
Name example1 Create a New Project Using Image: Im	OK Cancel <u>H</u> elp Tip for New Users Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.
Location H:\My Documents\PSpice	BIowse

Type **example1** in the **Name** field, select the **Analog or Mixed A/D** project type, set the location to **H:\My Documents\PSpice**, and click **Ok**.

Create PSpice Project	
O Create based upon an existing project	ОК
AnalogGNDSymbol.opj	Browse
⊙ Create a <u>b</u> lank project	Cancel <u>H</u> elp

Select Create a blank project and click Ok. The Schematic Editor will open



Once the Schematic Editor opens, select the **Place Part** tool on the right sidebar, and click the **Add Library** button. In the file dialog that appears, select the **source** library.

Allegro Design Entry CIS - [/ - (SCHEMATIC1 : PAGE1)]			
Bile Edit View Tools Place Macro PSpice Accessories Options Windo	w <u>H</u> elp		cādence – ⊲×
🕒 🗁 🖶 🕁 V 🗈 🖻 🥱 🦿	✓ ④		🔽 🛤 💌 🕘 🕟
	F (2)		
		11 0 11	
		u w tr	
🗊 example1 🔛 PAGE1		Place Part	
	abc		* Q (4)
· · · · · · · · · · · · · · · · · · ·	L	Part List	7
	1. 🔶	STIM8	~
E		VAC VDC	
D	÷ 🖪	VEXP VPULSE	
		VPWL VPWL ENH	
		VPWL_F_RE_FOREVE	ER 💌
	ц. 🗙	Lįbraries:	M X
		Design Cache	·
		SOURCE	
	∩ abc		
с	N 0		
		•	Packaging Parts per Pkg: 1
		\ <u>↓</u> ∨?	Part
× · · · · · · · · · · · · · · · · · · ·			Type. Homogeneous
		Normal <u>Convert</u>	
		- seaicritor Part	
0 items selected	1	Scale=	-100% X=2.30 Y=0.10

In the **Part List**, select the part 'VDC' and then click the **Place Part** button or hit **enter**. Place the part by clicking in the schematic, and then press escape to stop placing DC sources. Double click on the text **OVdc** to change the voltage, set it to **10Vdc**



Construct this circuit by selecting the **analog** library and placing two resistors, which appear in the part list as **R**. Rotate one of them by pressing the '**r**' key while placing it. Connect it all together by selecting the **Place Wire** tool and clicking on the points to be connected.

Place Ground		
Symbol: 0 \$D_HI \$D_LO 0 Libraries:	- <u>0</u>	OK Cancel Add Library Remove Library Help
CAPSYM Design Cache SOURCE Use 0/CAPSYM symbol to place	<u>N</u> ame: 0 ce a dc ground	

In order to simulate a circuit, PSpice needs a ground node attached to it. Select the **Place Ground** tool in the toolbar. Be warned that there are multiple libraries providing the '**0**' symbol. Select the '**Source**' library before selecting the '**0**' symbol, then click **Ok**.

						:	i,	R1	ł					
								1k	Y]		
				Ś	л ^і									
j1	ΟŅ	łdje	4									ş	Ŕ2	
			1	T :								ſ	1k	
			_											
			1	0										

Place the ground symbol in the schematic and connect it to the circuit

Display Properties	
Name: Value Value: 4k	Font Arial 7 (default)
Display Format ○ Do Not Display ○ Value Only ○ Name and Value ○ Name Only ○ Both if Value Exists	Color Default ▼ Rotation 0° 180° 270°
ОК	Cancel <u>H</u> elp

Double click the resistance value of the horizontal resistor, and set it to '4k'.

Place Net Alias		×
<u>A</u> lias: <mark>Vout</mark>		OK Cancel
Color Default	Rotation	<u>H</u> elp
Font Change Use Default	Arial 7 (default)	

It is important to name the nodes you want to plot so that you can find them easily. Select the **Place Net Alias** tool, enter the name **Vout** and click **Ok**.



Place the alias between the two resistors

New Simulation	×
Name:	Create
example1	cicate
Inherit From:	Cancel
none 💌	
Root Schematic: SCHEMATIC1	

Click the **New Simulation Profile** button to configure the simulation. Enter the name 'example1' in the dialog that appears and click **Create**.

Simulation Settings - exam	ple1 🔀
General Analysis Configuration Analysis type: Imme Domain (Transient) Imme Domain (Transient) Options: Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient) Imme Domain (Transient)<	on Files Options Data Collection Probe Window Bun to time: 1000ns seconds (TSTOP) Start saving data after: 0 seconds Iransient options Maximum step size: seconds Skip the initial transient bias point calculation (SKIPBP) Bun in resume mode Output File Options
	OK Cancel Apply Help

This will bring up the **Simulation Settings** dialog. Click **Ok** to accept the default settings.



Click on the **Voltage/Level Marker** button and place the marker on **Vout** as shown. Click the **Run PSpice** button.



The simulation results will appear as shown, with the voltages at all probes plotted. Close this window now, and return to the schematic.



If the **Enable Bias Voltage Display** button is not already selected, click it. Voltage markers should appear at every node of the schematic as shown.

Now turn off the **Bias Voltage Display** and enable the **Bias Current Display**. Cadence will display the DC current through the circuit as shown.

AC Circuit Analysis

Transient Analysis



Create a new project and assemble the circuit shown. The voltage source is part '**VSIN**' in the '**SOURCE**' library.

Simulation Settings - tran	
General Analysis Configuration Analysis type: Time Domain (Transient) Image: Configuration Options: Image: Configuration Image: Configuration Image: Configuration Image: Configuration Image: Configuration <td< td=""><td>on Files Options Data Collection Probe Window <u>B</u>un to time: 20us seconds (TSTOP) Start saving data after: 0 seconds Iransient options </td></td<>	on Files Options Data Collection Probe Window <u>B</u> un to time: 20us seconds (TSTOP) Start saving data after: 0 seconds Iransient options
	OK Cancel Apply Help

Configure the simulation with the **New Simulation Profile** button and enter the name '**tran**'. Set the **Analysis type** to '**Time Domain (Transient)**', the run time to **20us** and the maximum step size to **100ns.**



Place voltage markers on each side of the resistor. Name the nodes **Vin** and **Vout**. Run the simulation and the results window should appear. Click the **Toggle Cursor** button and left click the colored dot for **Vout** in the legend. Use the mouse to drag the cursor over to the second peak of **Vout** and note the amplitude. It should be 3dB smaller than the peak voltage of **Vin** (0.707V). Click on the **Mark Label** button to label the point.



AC Analysis

Close the simulation and modify the circuit, replacing the **VSIN** source with a **VAC** source. Leave the new source with the default attributes. Configure a new simulation profile with name **AC**. Set the analysis

type to **AC Sweep/Noise**, the sweep type to **Logarithmic**, the frequency range from **1-1000000**, and the points/decade to **10**.

Simulation Settings - AC			
Simulation Settings - AC	Files Options Data Col AC Sweep Type Linear Logarithmic Decade Noise Analysis Enabled Out I/V Inte	lection Probe Window Start Frequency: 1 End Frequency: 1000000 Points/Decade: 10 put Voltage:	
	Controlled sources	as point information for nonlinear and semiconductors (.OP)	
	ок с	Cancel <u>Apply</u> H	elp

Creating a new simulation profile deletes the preexisting voltage markers, so re-add one on **Vout** and run the simulation.



Click the Add Trace button and select Plot Window Templates from the Functions or Macros dropdown menu. Select V(Vout) on the left and Bode Plot dB – dual Y axes(1) on the right, then click Ok.

Add Traces		
Simulation Output Variables		Eunctions or Macros
×		Plot Window Templates 🛛 🗸
Frequency I(C1) I(C1:1)	Analog	3dB Bandwidth - Band pass [multi-run][1 3dB cut-off frequency - High pass [multi 3dB cut-off frequency - Low pass [multi-
I(B1) I(B1:1) I(V1) I(V1)+)		Admittance(1,2) Average(1) Bode Plot - dual Y axes(1) Bode Plot - separate(1)
V(0) V(C1:1) V(C1:2)	✓ Currents ✓ Power	Bode Plot dB - dual Y axes(1) Bode Plot dB - separate(1) Conductance(1,2)
V(R1:1) V(R1:2) V(V1:+)	Nojse (V²/Hz)	Current Gain(1,2) DC Current Gain(1,2) DC Voltage Gain(1,2)
V(V1:-) V(Vin) V(Vout)	Subcircuit Nodes	Derivative(1) Falltime of Step Response [multi-run](1) First Peak [multi-run](1)
V1(C1) V1(R1) V1(V1) V2(C1) V2(R1) V2(V1) V2(V1) V(C1)	25 variables listed	Fourier Transform[1] Impedance(1,2) Integral(1) Log-Linear(1) Log-Log(1) Nyquist Plot(1)
Full List		
Irace Expression: Bode Plot dB - dual Y axes(V(Vout))		<u>OK</u> <u>C</u> ancel <u>H</u> elp



You should see a plot similar to this one. The green trace shows phase and the red trace shows magnitude. Use the cursor to find the point where the phase hits -45, and label it.



Find and mark the point where the magnitude is -3dB.

Step Response

To simulate a step response, create a new project and perform a transient analysis using the same steps as above, but with a **VPULSE** voltage source. The **VPULSE** source has 7 parameters affecting the waveform.



V1 = First Voltage V2 = Second Voltage TD = Initial Delay TR = Absolute Rise Time TF = Absolute Fall Time PW = Pulse Width PER = Period

To simulate a step response, we use a **VPULSE** source set to V1=0V, V2=1V, TD=0, TR=1ps, TF=1ps, PW=1s, and PER=2s. A 1 picosecond rise/fall time is extremely small with regard to the simulation time, so it closely approximates the ideal step function.



Dependent Sources

The ANALOG library provides four dependent sources:

• E – Voltage Controlled Voltage Source



• F – Current Controlled Current Source



• G – Voltage Controlled Current Source



• H – Current Controlled Voltage Source



Variable Phase VSin Source

The phase isn't displayed on the schematic for the **VSIN** source, but it can be added. Double-click on it to bring up the property editor. Find the '**PHASE**' field and select it, then click the '**Display...**' button. Select **Name and Value** from the **Display Format** list, then click **Ok**.

The '**PHASE'** property is now displayed on the schematic, and can be moved and edited the same as the default properties.