

PSPICE Tutorial





Outline

- Introduction
- Installation
- Prepare a circuit for simulation
- Simulation using PSPICE
- A typical example





Before we begin

• Reference: <u>https://engineering.purdue.edu/~ee255d3/readings.html</u>

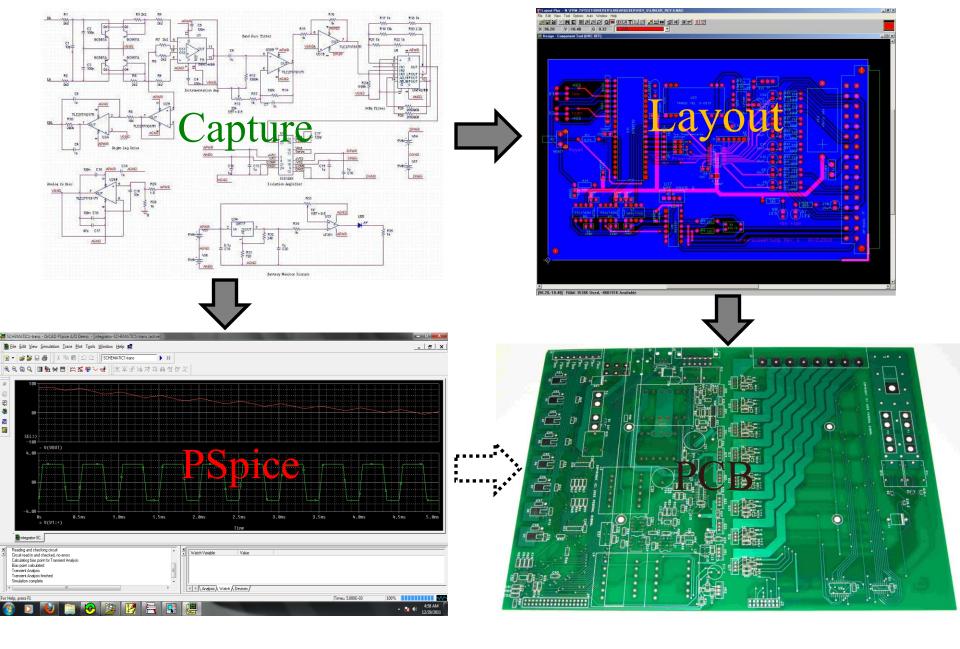




Introduction

- SPICE Simulation Program with Integrated Circuits Emphasis
- Developed by University of California at Berkeley in 1970s.
- A simulation program that models the behavior of a circuit containing analog or mixed A/D devices, used to test and refine your design before implementing on hardware (PCB).
- PSPICE is the most prominent commercial version of SPICE, initially developed by MicroSim (1984), but now owned by Cadence Design System. Pspice is now a component of the OrCAD® Product Family (including Capture CIS, PCB Editor, Pspice, Layout Plus ...)







Installation

- Almost every computers in ECN labs are equipped with the standard version of PSPICE, a product of Cadence.
- You can also download the PSPICE student version <u>here</u>.
- To install the student version: Unzip the downloaded file, run setup.exe and follow the instructions on the screen. *Note: you should close all other Windows programs (especially web browsers) before installing PSPICE*.
- To run PSPICE on ECN computers: go to Start > Programs > ECN Software > Cadence SPB 16.01 > Design Entry CIS.





. 🤌 - 🧏			
My Documents _kwFWARe			
My Computer DjView (MSEE180PC1)			
S 🗧 🦉			
My Network url Places			
Internet 91pspstu			
Explorer			
ECN Network			
	AMS Simulator		
	PCB Editor Utilities PCB Editor Utilities Tutorials		
SopCast	What's New in Release 16.01		
	Cadence Help		
	📑 Cadence Switch Release		
	Design Entry CIS		
SopCast-3.2.4	Design Entry HDL		
	% Design Entry HDL Rules Checker		
	Marcayout Plus SmartRoute Calibrate		
Google	🛄 Library Explorer		
Chrome	Model Integrity		
	🧱 Package Designer		
Dissertation	RCB Editor		
	PCB Router		
	💥 PCB SI		
	README PCR		
Programs , ECE Software Mini Lattice Semiconductor	SigXplorer		
ECN Software Field Cadence SPB 16.01	SiP		
Games • Games •			
E Settings			
Search Microsoft Silverlight Search Microsoft Windows SDK v6.0A			
Michola Michol			
Help and Support in Windows Explorer			
😤 🖅 Run			
g Log Off ptran			2
Settings Search Microsoft Silverlight Microsoft Windows SDK v6.0A Microsoft Windows SDK v6.0A Microsoft Windows Explorer Windows Explorer Windows Explorer V Log Off ptran Shut Down			Recycle Bin
BSPICE 9.1 student versi Despice-Tutorial[1].pdf	🔣 Allegro Design Entry CIS 🛛 🏠 PSPI	ICE	🕐 루 < 🕅 🔍 5:00 PM



Prepare a circuit for simulation

- To simulation your design, you need to provide Pspice with the following information:
- 1. the parts in your circuit and how they are connected \rightarrow schematic
- 2. what analyses you want to run \rightarrow simulation profile
- and the simulation models that correspond to the parts in your circuits → part library.
- 4. The stimulus definitions to test with \rightarrow stimulus editor
- Two ways to describe your circuit:
- By scripts (write an input file *.cir)
- By drawing schematic
- (Today I mostly focus on the latter).





Simulation

- Bias point details
- DC sweep analysis
- AC sweep analysis
- Transient analysis
- Frequency response
- And more ...





Examples

- Clipper.
- Transistor amplifier





Example 1: 2-diode clipper

- Bias point.
- DC sweep analysis
- Transient analysis
- AC sweep analysis





Student version

- Some features have been limited in the student version.
- There may be a little change in the user interface when creating a circuit for simulation, please refer to the slides 14-43 of this file for step-by-step simulation procedure using Pspice student version.







Question and Answer



Go to: Start: All Programs: Electrical Apps: Orcad Family... : PSpice Design Manager

	<u> </u>			 •	0
}					
upWise					
apinoo					
A	1 New Office Document				
۲	Open Office Document				
ockDown	SecureCRT 4.0				
Browser	Set Program Access and Defaults				
		_			
\mathbf{S}	Accessories	•			
ogout of	Aerospace Apps	•			
Windows	m ANSYS 11.0	•			
P	Autodesk				
	Chemical Apps				
illa Firefox	🗑 Electrical Apps	Freescale CodeWarrior			
	m EnZip	Orcad Family Release 9.2	Capture		
*	Industrial Apps	•	🛃 Layout Plus		
2	DAWS451		Layout Plus SmartRoute Calibrate		
Report a Problem	🛅 LaTex	•	Online Manuals		
	m Maple 10	•	PSpice AD		
	matlab	•	PSpice Design Manager		
	McAfee	•	PSpice Message Viewer		
	Mechanical Apps	•	PSpice Model Editor		
	Microsoft Office	•	PSpice Optimizer		
	Mozilla Firefox	•	PSpice Simulation Manager		
	MSC.Software	•	PSpice Stimulus Editor		
	Movell Groupwise		Release Notes		
	PDFCreator		Schematics		
	Polymath Software		🧐 Web Update		
Dean, Robe	Programming	•			
-	C QuickTime				
Mozilla Firefox	m Real				
	Respondus	→ ents →			
E-mail Novell GroupWi:	Roxio Creator DE				
Windows Media	SecureCRT 4.0				
	Solid Edge V20				
	SPSS Inc	nd			
Internet Explor					
	The Net				
	X-Win32 6.0				
🝌 Files and Settin	Adobe Reader 8				
Wizard	Alice Alice				
PSpice Design M	6 Internet Explorer				
PSpice AD	PowerDVD				
1400-	Remote Assistance Windows Journal Viewer				
All Programs 🕨	 Windows Journal Viewer Windows Media Player 				
					Recy
	🔑 Log Off 🛛 🧕	Shut Down			
Start					🏷 🛃 🗐 💫 💟 🛛

It will look like this:

PSpice Design Manager				 	
File Workspace View Tools Help					
For Help, press F1			li.		
😈 Scan for threats					Recycle Bin
Descention	PSpice Design Manager	Microsoft PowerPoint - [🥜 🕎 😨 🕄 📴 🔍 🖏 🕅 7:53 AM
Properties	Popice Design Manager	Microsort PowerPoint - [🖉 🛃 🛄 📅 🏷 🛃 🔧 😽 🖊 7:53 AM

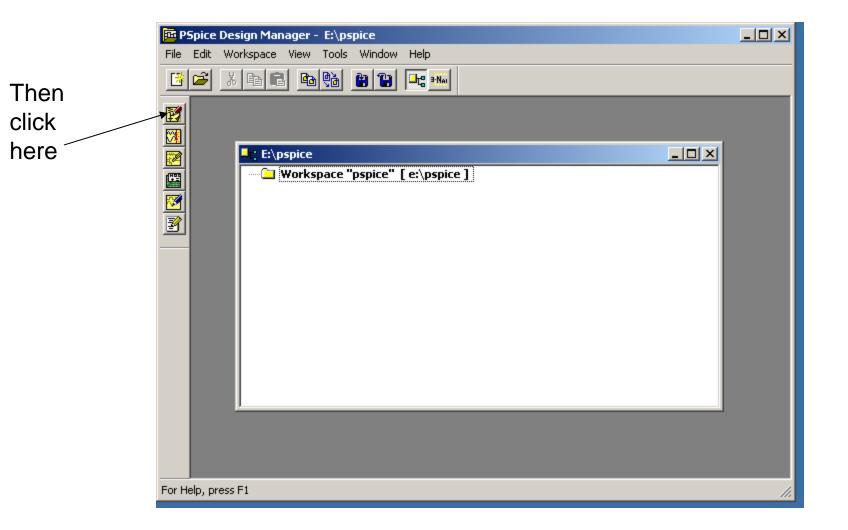
Click on: File : New Workspace

File Workspace New Workspace Open Workspace 1 E:\circuit1 Exit	
Open Workspace 1 E:\circuit1 Exit	
Exit	
Create a new design workspace	

Give your New Workspace a Names and Location

📴 PSpice Design Manager	
File Workspace View Tools Help	
New Workspace Name	
New Workspace	×
Name	
pspice	Create
Location	Cancel
E:\pspice	Help
For Help, press F1	li.

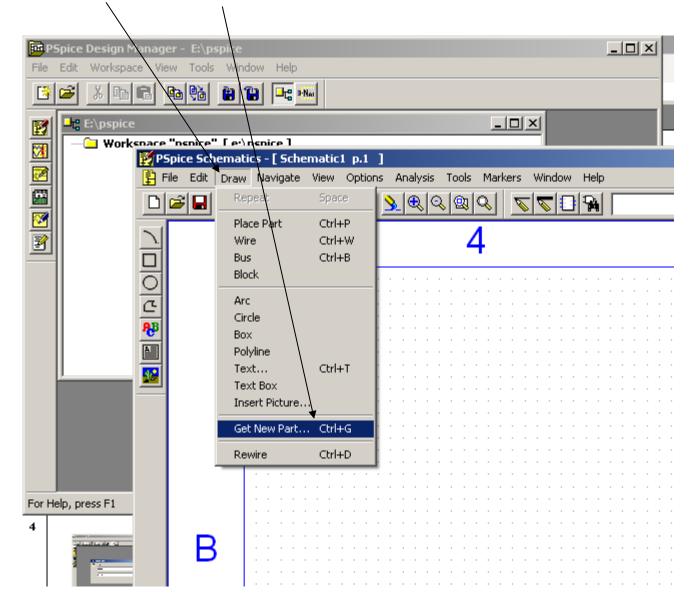
It will then look like this:



An area to draw your circuit will open up

			/	
PSpice Design Manager - E:\pspice			/	
File Edit Workspace View Tools Win		/		
🔋 🚅 🔏 🖬 🖷 🛍				
		/		
PSpice Schema	tics - [Schematic1 p.1]	/		×
File Edit Drav	v Navigate View Options Analysis Tools Markers Window Help			_ 뢴 쓰
Image: Section of the s	∦∎∎ ∽∼ <u>></u> €Q®Q ⊽⊽∎₽∦	_/	💽 🗺 💅 📄 🔀 None	• 🔊 🙉 🔽 I
		_/		
Image: Second secon	4	/		3 🗎
		/		
<u></u>				
RB				
<u>1</u>				
For Help, press F1				
B				
				Þ
6.71, 0.07			Cmd:	

Click on Draw: Get New Part



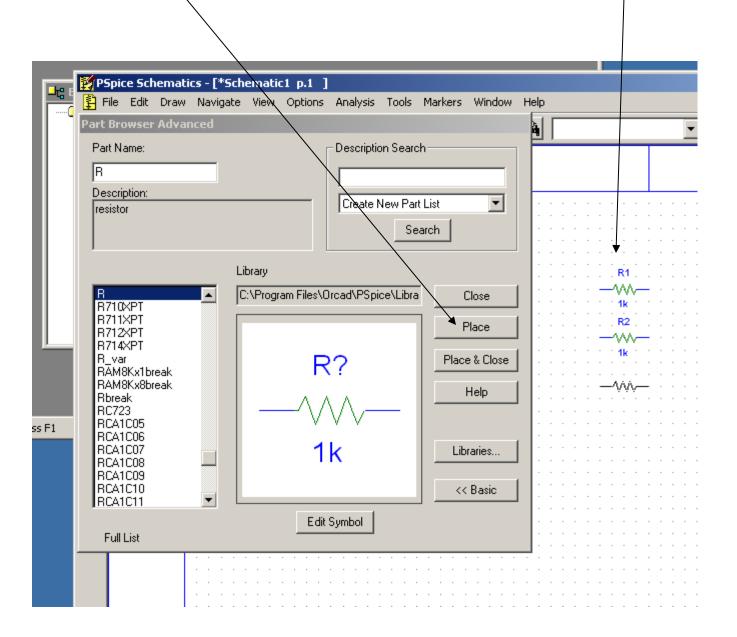
This window will then open up

PSpice Design	Manager - E:\pspice					
File Edit Workspa	ace View Tools Window	Help				
1 🖻 🔏 🖻						
	PSpice Schematics - File Edit Draw Na	[Schematic1 p.1] vigate View Options		Iarkora Window	Hala	
	Part Browser Advanced		Analysis Tools M	larkers window		•
	Part Name:	-	Description Search-			
	Description:		Create New Part Li	ist 💌		
			Searc	ch		
		Library				
	+5V	1		Close		
	-5V 100101]				
	100102 100107			Place		
	100117			Place & Close		
	100118 100122					
	100124 100125			Help		
For Help, press F1	100130					
	100131 100136			Libraries		
	100141 100150				I · · · · · · · · · · ·	
	100151			<< Basic		
	100155	· · · · · · · · · · · · · · · · · · ·	t Symbol			
	Full List					
	· · ·					

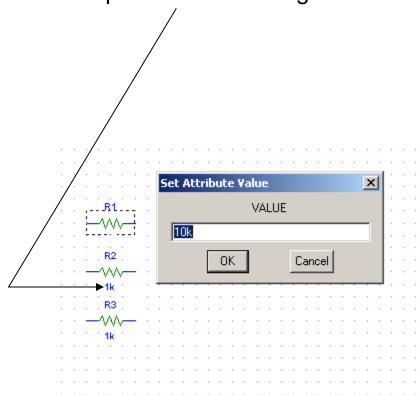
Enter a part name: R, C, L, etc...

	Manager - E:\pspice	
13 🖻 X 🖻		
	PSpice Schematics - [Schematic1 p.1] File Edit fraw Navigate View Options Analysis Tools Markers Window Help Part Brows er Advanced Part Name: Description: resistor Create New Part List	
	B C:\Program Files\Orcad\PSpice\Libra Close R710XPT C:\Program Files\Orcad\PSpice\Libra Close R711XPT Place Place R714XPT R714XPT Place R714XPT R714XPT Place R714XPT R714XPT Place R714XPT R714XPT Place R000000000000000000000000000000000000	All related parts will list here: select the one you want
For Help, press F1	RCA1C05 RCA1C06 RCA1C07 RCA1C08 RCA1C09 RCA1C10 RCA1C10 RCA1C11	lt's symbol will then appear here

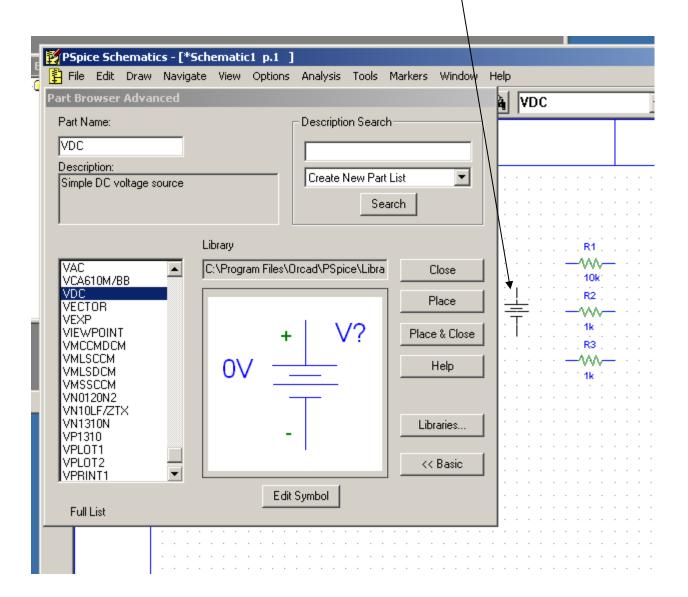
Click Place and put the part(s) where you want it or them







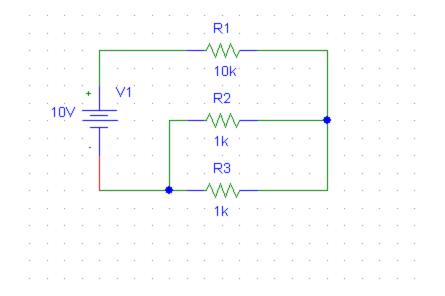
Then select a DC voltage supply and place it



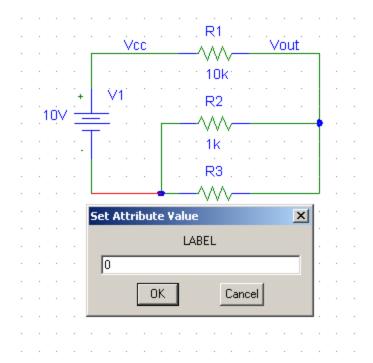
Draw the wires to connect all the parts

P P															
<u>р</u> Г	ile Edit	Draw Navigate	View Option	ns Analysis Tools Markers Window Help											
ום	2 	Repeat	Space												
		Place Part	Ctrl+P												
		Wire	Ctrl+W	4											
		Bus	Ctrl+B	•											
ਗ		Block													
<u> </u>		Arc Circle Box Polyline Text	Ctrl+T	R1	· · · ·										
		Text Box Insert Picture.		10k 10v <u>↓ V1</u> R2 10v ↓	· · ·										
		Get New Part.	Ctrl+G	[
		Rewire	Ctrl+D	R3											
			· · · · · · ·	1k 1k	· · ·										

Your circuit will look something like this:



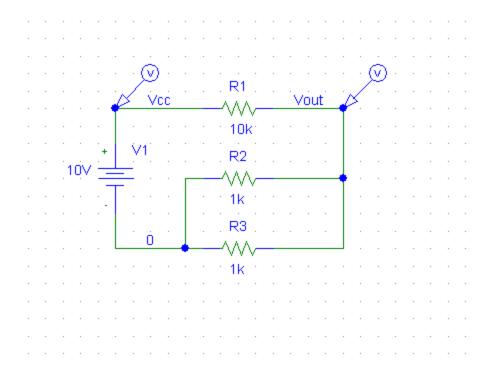
Click on the wires to label the circuit nodes: call the ground wire "0"



Now mark the circuit nodes of interest with Voltage Level Markers

	PSpic	e Scl	nema	atic	s - [*Sch	nema	atic	1 p.1	1																		
1	File	Edit	Dra	W	Navi	igate	Vie	w	Optio	าร	Ana	alysis	To	ools	ſ	Marke	ers	Wir	Idow	He	elp							
			9		¥		ß	۲O	\sim	2	6	<u></u>	2	<u>R</u>		Ma	ark V	olta	ge/Le ge Di	ffere			l+M					•
							4	L											nt in nced		Π							
0	╢																ear A ow A								L			
C	1								•					•			ow 9		ted						Ŀ			
<mark>₽</mark> ₽																												
		•	· ·				· ·	•	•	· ·		•	•		•	•			· ·			-						• •
_	1	•	· ·	•	•	•	· ·	•	•	· ·		•	•	•	•	•	•			•	F	R1			•	•	•	
	Ŀ																	Vee	:		-^	\sim			νοι	ut	٦	
							· ·	•	•	· ·			•	•			V1					10k)			•			
		•	· ·		•		· ·	•	•	· ·	•	•	<u>[</u> 1	ġν	-		_		 . г			72 7/			•			· ·
		•		•	•	•		•	•			•	•	•	•		•			•		1k		•	•	•		
																	i	j			ļ	23						
		•	· ·				· ·	•	•	· ·		•	•		•						_^	///\/ 1k						• •

Your circuit will look something like this:



Click on Analysis: Setup

	P 🔛	Spic	e Scl	hema	tic	s - ['	*Scl	nema	atic	1 p	.1]												
	1	File	Edit	Drav	N	Navi	gate	Vie	w	Opt	ions:	F	Analysis	Tools	Ma	rkers	Win	dow	Help	D				
[2		9		¥ [È	ß	n	6	×	<	Electric Create			ck				VDC			•	*
		L											Edit Sti	muli										
	T	L						4	Ŀ.			E	Setup.											
	믱	L											Library		nclude	e Files.								
	9	⊢										-	Simulat					F11	H					
	C	·		• •								-	Ducha	- hum					-1					-
	신 년 20전 10전 10<td> ·</td><td></td><td>• •</td><td></td><td></td><td></td><td>• •</td><td></td><td></td><td></td><td></td><td>Probe : Run Pr</td><td></td><td></td><td></td><td></td><td>F12</td><td></td><td></td><td></td><td></td><td>• •</td><td>-</td>	·		• •				• •					Probe : Run Pr					F12					• •	-
		·		• •				• •				_	Kulter	ope				F12	_					-
		·						• •					Examin										• •	
	<u></u>	·		• •	•			• •					Examin										1	<u> </u>
		11		• •	•			• •					Display	Result	s on	Schem	natic			R1			- X	\mathcal{O}^{\pm}
		·		• •				• •				1				K.	Vcc				Vout	1	<u>ن</u> ک	
		L :		• •				• •							1	r				~~~~		-		•
		L :		• •				• •									• •			10k1				•
		L :			•			• •							`+	- V1	• •			R2				
		· .		• •				• •						10V	·		• •			A A A				
		· .			•			• •								_	• •			- VVV-		1		
		· .						• •							1 - -		• •			1k (-
		· .		• •				• •												R3				
		· .															Ó Í			ΛΛΛ				
		<u> </u>																		-				
		· .																		1K (
		1 ·						• •							• •								• •	

For DC circuit analysis, enable: "Bias Point Detail"

- 187	PSp	ice Sc	hema	tics -	[*Scb	emati	c1 n 1	1 1		/						
	-		Drav		vigate	View	Optic		/sis Tools Mai	rkers Windo	w Help					
··[💾	ייים הורי	alm		U U		-			1 - 1 - 1 - 1							किंग्या
			9	Φ			<u>, 1</u>		<u> </u>		3 🙀 VC)C				
~										/						
Ē	i					4			/							
	쉐															
	쉬															
1																
					-											
								/								
									· · · · ·	- · (∨) · -			· · (v)			
- 11								Enable	is Setup		Enabled				×	
							/			weep		Options.		Clos	e	
							. /			ias Point		Parametrio			- 1	
					-		/ 1	1		ias Point				1		
			• •		-	· ·/						Sensitivity				
					-	/.			DCS	weep		Temperatu	re			
					- /				Monte Carlo	/Worst Case		Transfer Fund	ction			
					\square				Bias P	oint Detail		Transient				
									Digita	Setup						
		• •	• •		-											

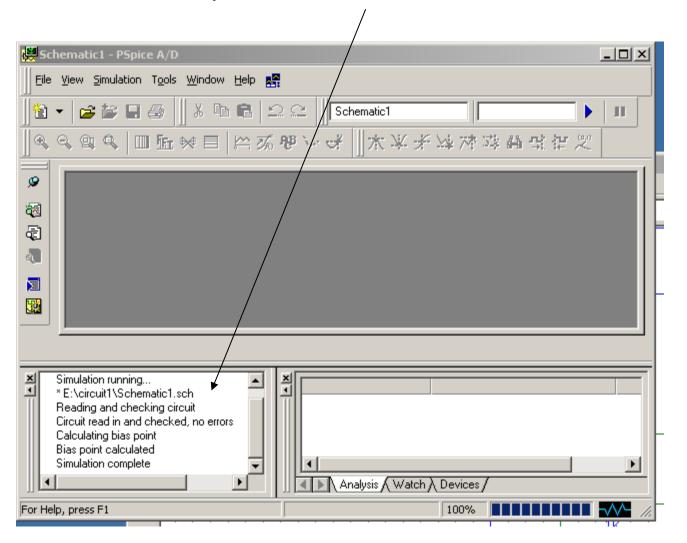
Select Analysis: Create Netlist (correct circuit layout if any errors found)

	PSpie	e Sch	nemati	ics - [*Se	chem	atic	ιр.)	1 1		
	File	Edit	Draw	Navigat			Optio	ſ	Analysis Tools Markers Window Help	
E	נ 🖴		3	メ 喧		ŝ	2		Electrical Rule Check	•
-						_	-		Create Netlist	
	X.								Edit Stimuli	
Ē	1				_	Ł			Setup	
	-					•			Library and Include Files	
\square	4⊢								Simulate F11	
C	5 ·		• •					•	Darke Celur	
8	B							•	Probe Setup	
			• •					•		
	<u> </u>		• •					•	Examine Netlist	
-	- -							•	Examine Output	÷.
_	-		• •					•	Display Results on Schematic 🔹 🕨 🔤 👘 👘 👘 👘	Ø.
	1							. '	Vcc Nout X	
1	1									
									· · · · · · · · · · · · · · · · · · ·	
									in a second provide a second sec	
1									· · · · · · · · · · · · · · · · · · ·	
									· · · · · · · · · · · · · · · · · · ·	
									· · · · · · · · · · · · · · · · · · ·	

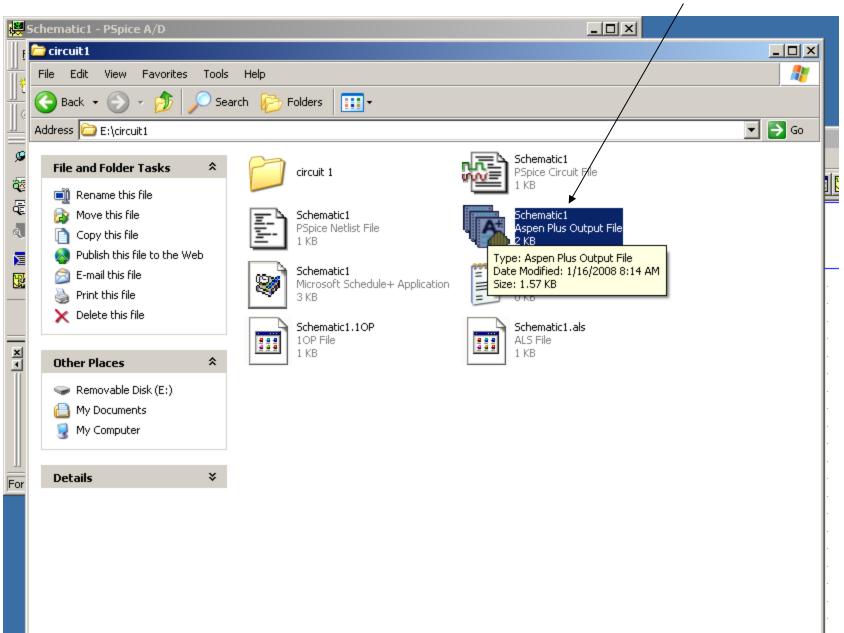
Select Analysis: Simulate to analyze the circuit

Į.	🛐 P	Spic	e Sch	iemati	ics - [Sche	ematio	:1 p	.1]			_										
	1000	File	Edit	Draw			View		tions	A	nalysis	Tools	Mark	ers W	/indow	Help	p					
L		2		9	\$	B		n (×		Electr	ical Rule e Netlis	e Check				VDC			-		1
	<u> 10000</u> 1000 1000						4				Edit S Setup Librar Simula) y and Ir	nclude I	Files	F11							
	근 원			· ·	· ·		· ·	· ·	•		Probe Run P	: Setup. 'robe			F12		· · ·		· · ·	· ·	· ·	· ·
				· ·	· ·			 	•			ine Netli ine Outp					· · ·					· ·
	_	1									Displa	iy Resul	ts on S	chemat	ic	•	R1			· · · {	v) ∙	
÷.													1	x v	cć i		-^^/		Vout	K		
I		·															10k					
I		11											· +	V1			R2					
ł					• •	•		• •			• •	10V			· ·			-			• •	
		L .											· T				· 1k · ·			Ι.		
		1.											1									
		Ŀ												Ó			. R3 .					
		· ·						• •					. L	- 0	•		-^///-					
		1								•							1 k 1					
		1 ·				•		• •					• •	• •								

This window tells you how the simulation went



Go to your workspace directory and click on the Output File



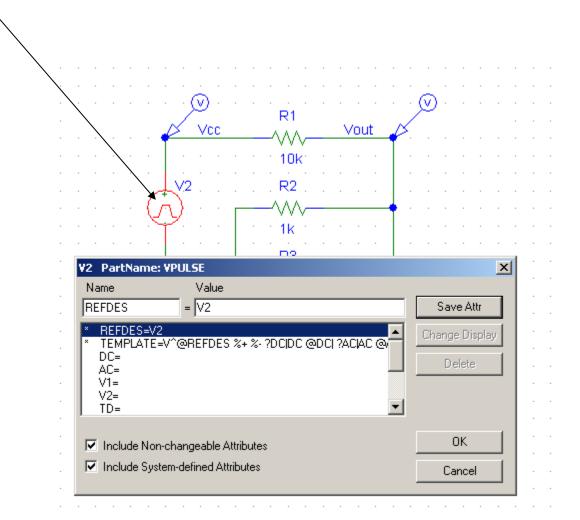
Spice\PSpice.ini:
Spice\PSpice.ini:
spice\Pspice.ini:
Spice\PSpice.ini:
Scroll down to you find th voltages for the nodes yo marked in the circuit. The
voltages are referenced t node "0."
/**** ID# 1108003738
27.000 DEG C

IODE VOLTAGE

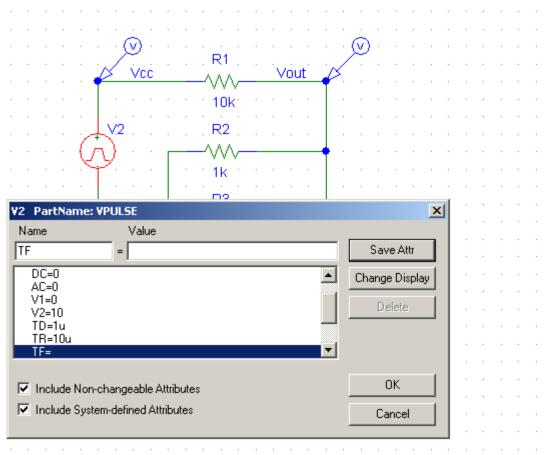
This time, replace the DC voltage source with a time varying voltage source, such as VPULSE

Part Browser Advanced Part Name: VPULSE Description: Pulse voltage source		Description Search	·	à VI
		Create New Part		
Librar VDC VECTOR VEXP VIEWPOINT VMCCMDCM VMLSDCM VMLSDCM VMLSDCM VMSSCCM VMSSCCM VN0120N2 VN10LF/ZTX VN1310N VP1310 VPL0T1 VPL0T2 VPINT1 VPLNT2 VPULSE	rogram Files\C	Drcad\PSpice\Libra	Close Place Place & Close Help Libraries	

Put it in the circuit and click on it to select it's parameters



Here I selected the constant DC and AC values to be zero, the initial voltage level (V1) to be zero, the final voltage level (V2) to be 10V, the time delay (TD) to be 1us and the rise time (TR) to be 10us. I left the other options blank



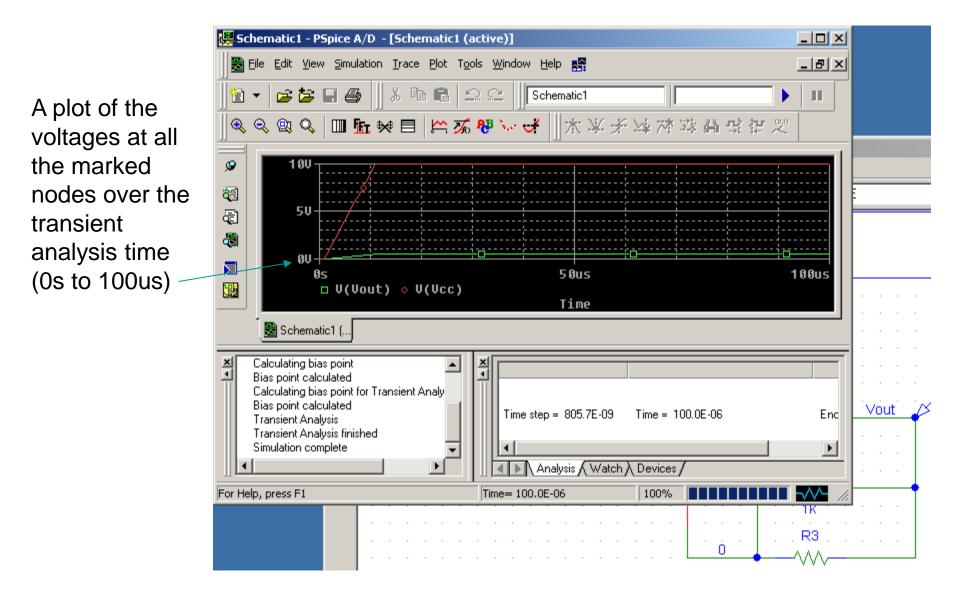
Under Analysis Setup, enable "Transient..." and click on it

	_			<i>.</i> .																							
				(sta		in T		Ma					le le														
	Viev	V C	Optio	ns A	Analys	is i	ools	Ма	irkers	5 VV	/indov		lelp								_						
		ŝ	<u>റ</u> ч	<u>}</u>	. 🔍	्	<u>Q</u>	Q			<u> </u>	3 1	À	٧P	UL	SE				-	ŀ	•	Ø		Ξ	X	
		-		· · ·		· · ·	-	- - -	· · ·		•	 			- - - -	-	 	-		•		-		-	-	-	-
						🕅												(\mathbf{v})									
Keep the		•		_	a <mark>lysis</mark> abled		up						Enabl	امط											×Ц		
"Bias Point								AC 9	Gwee	р		1	Enabi	lea			0	ptior	ns			1	Clo	ose	1		
Detail"	•	•	•				Lo	oad B	lias F	Point.		1	F	1			Pa	ame	tric			1			-	•	
enabled too <							Sa	ave B	lias F	Point.			F	1			Se	nsitiv	rity								2
	<u> </u>							DCS	Gwee	:р				1			Tem	ipera	iture.						\rightarrow	/	
						м	onte	Carlo	o/Wo	orst C	ase		Ļ]		٦T	ansf	er Fu	inctio	on			/				
					•		В	ias P	'oint l	Detai	il		V	1			Tr	ansie	ent			1					
	•	•					[Digita	al Set	up																•	

Set the Transient Analysis Final Time to 100us

[*] Schematic1 p.1 (stale)]		
gate View Options Analysis Tools M	arkers Window Help	
16 <u> 3</u> 8 8 8 8	🛛 🔽 🕄 🙀 VPULSE	
4		
	Fransient	× · · · · · · · · ·
	Transient Analysis	
	Print Step: Ons	
· · · · · · · · · · · · Analysis Setup	Final Time: 100us 💌	X · ·
Enabled	No-Print Delay:	
· · · · · · 🗖 🗛		Close
	Step Ceiling:	
· · · · · · · · · · · · · · · · · · ·	🔲 Detailed Bias Pt.	
	Skip initial transient solution	
	- Fourier Analysis	
🔲 Monte Ca	Enable Fourier	- h
🔽 🛛 🛛 🛛 🖓		
	Center Frequency:	
· · · · · · ·	Number of harmonics:	
· · · · · · · · · · · · · · · · · · ·	Output Vars.:	· · · · · · · · · · · · · · · · · · ·
		· · · · · · · · ·
	OK Cancel	

Run the Simulation and these result will appear:



Play with this window to adjust display time and what signals are displayed