



Alexandria University

Faculty of Engineering

Electrical Engineering Department

ECE 336: Semiconductor Devices

Lab#1: Reverse bias p-n Junction

Objectives

The purpose of this exercise is to get introduced to the Orcad Pspice simulator, alter the device model parameters and observe the results. In this exercise we will examine the p-n junction diode under reverse bias and examine how it behaves as a capacitor.

By the end of this exercise you should be able to:

1. Work fluently on Orcad Pspice and alter the device model parameters.
2. Run transient, AC, DC and parametric sweep analysis on Orcad Pspice.
3. Understand the behavior of p-n Junction under reverse bias, and define the shortcomings of using it as a capacitor.

Requirements and Deliverables

In this exercise you are required to use the diode in an RC circuit, tweak its model parameters, observe its behavior in the circuit and conclude how it can be used in a circuit.

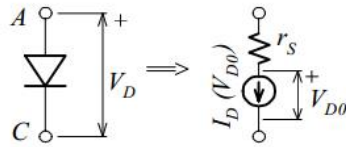
You should deliver a technical report containing the following sections:

- Introduction: A brief introduction about the device used in the lab and its model.
- Procedures: A description and snapshots of the lab procedures taken from your PC,
- Results: Numerical and graphical simulation results as requested
- Comments: Your conclusion about the results and your answers for the assignment questions.

Diode Device Model

<i>STATIC PARAMETERS</i>				
Symbol	Usual SPICE Keyword	Parameter Name	Typical Value/ Range	Unit
I_S	IS	Saturation current		A
n	N	Emission coefficient	1 – 2	
r_S	RS	Parasitic resistance		Ω
BV	BV	Breakdown voltage (positive number)		V
	IBV	Breakdown current (positive number)		A
Note: $IBV = IS \frac{BV}{V_t}$				

STATIC DIODE MODEL

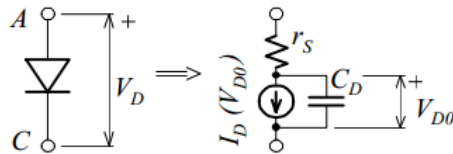


$$I_D(V_{D0}) = \begin{cases} IS (e^{V_{D0}/N V_t} - 1) + V_{D0} G_{MIN} & \text{if } V_{D0} > -BV \\ -IBV & \text{if } V_{D0} = -BV \\ -IS [e^{-(BV + V_{D0})/V_t} - 1 + \frac{BV}{V_t}] & \text{if } V_{D0} < -BV \end{cases}$$

DYNAMIC PARAMETERS

Symbol	Usual SPICE Keyword	Parameter Name	Typical Value/ Range	Unit
$C_d(0)$	CJ0	Zero-bias junction capacitance		F
V_{bi}	VJ	Built-in (junction) voltage	0.65 – 1.25	V
m	M	Grading coefficient	$\frac{1}{3} - \frac{1}{2}$	
τ_T	TT	Transit time		s

LARGE-SIGNAL DIODE MODEL



$I_D(V_{D0})$ is given in Table A.1

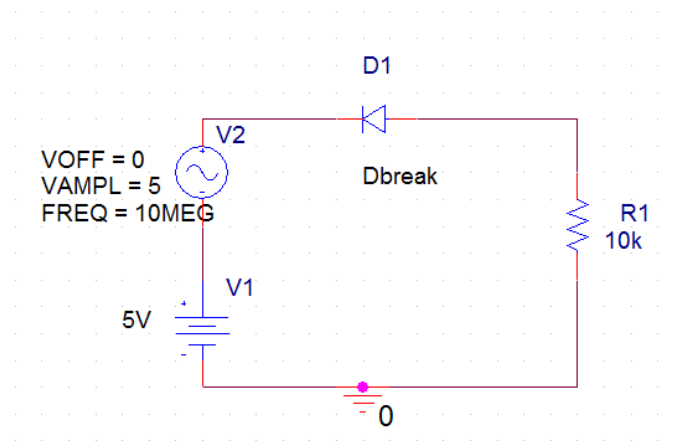
$$C_D = C_d + C_s$$

$$C_d = CJ0 \left(1 - \frac{V_{D0}}{VJ}\right)^{-M} \quad (\text{for } V_{D0} < 0.5VJ)$$

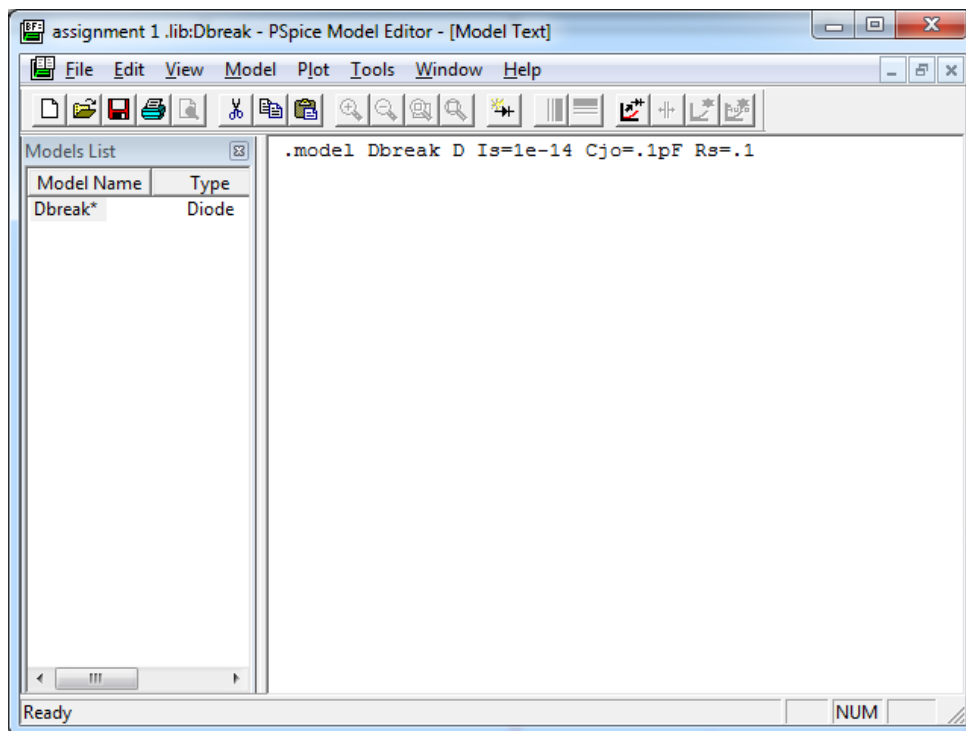
$$C_s = TT \frac{dI_D}{dV_{D0}}$$

Procedures:

1. Connect the circuit as shown in the figure, using the Dbreak diode model from Breakout library; this is a generic model library to modify the model as you want.

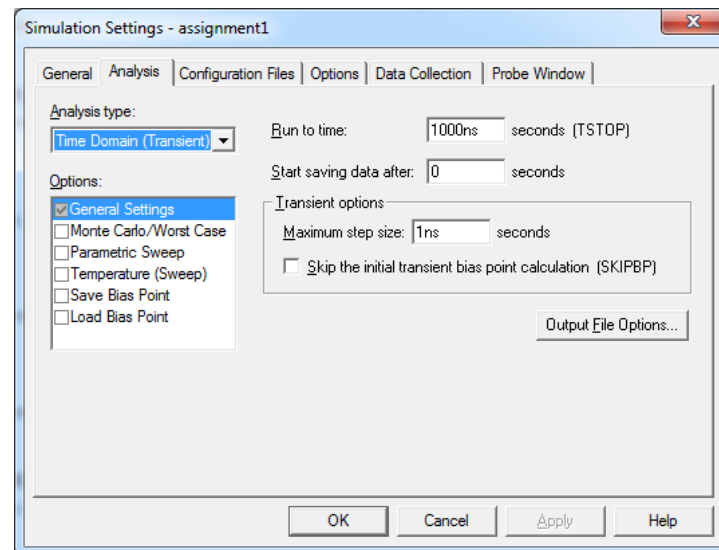


2. Right click on the diode and select “edit ps spice” model, another window will open as shown in the figure, modify the value of Cjo to be 15nF.

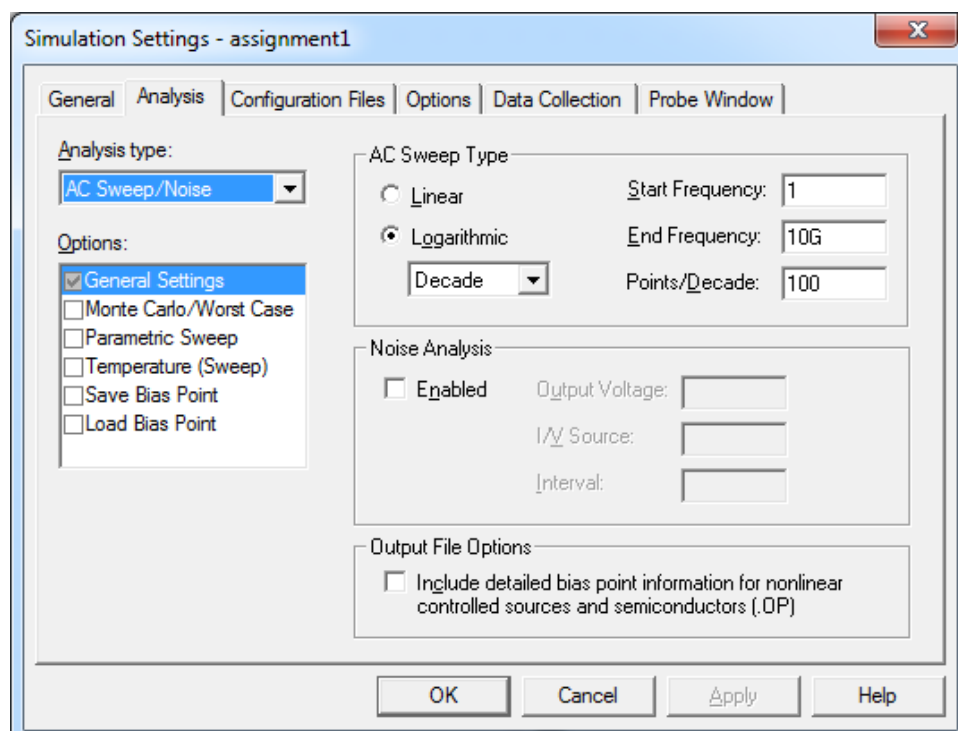


3. Place probes on the output node of the circuit and the input (source node).

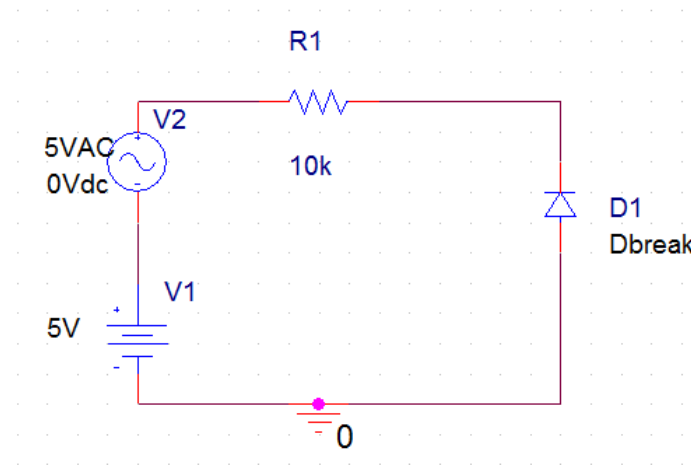
4. Create a new simulation profile, edit it → select transient analysis and set the numbers as shown in the figure.



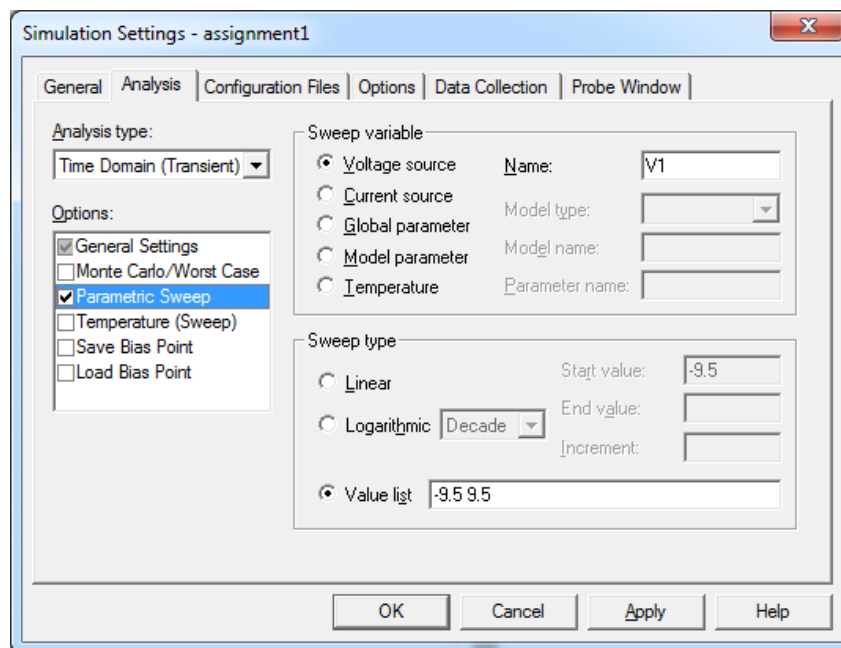
5. Run simulation and observe the plotted results, what do you observe ?
6. Repeat the steps (2 – 4) for $C_{jo} = 5\text{pF}$ and $C_{jo} = 1\text{e-}20\text{F}$, what do you observe ? why ?
7. Replace the sine source with an AC source (not necessary step for new versions of Orcad).
8. Edit the simulation profile as shown in the figure (select AC sweep).



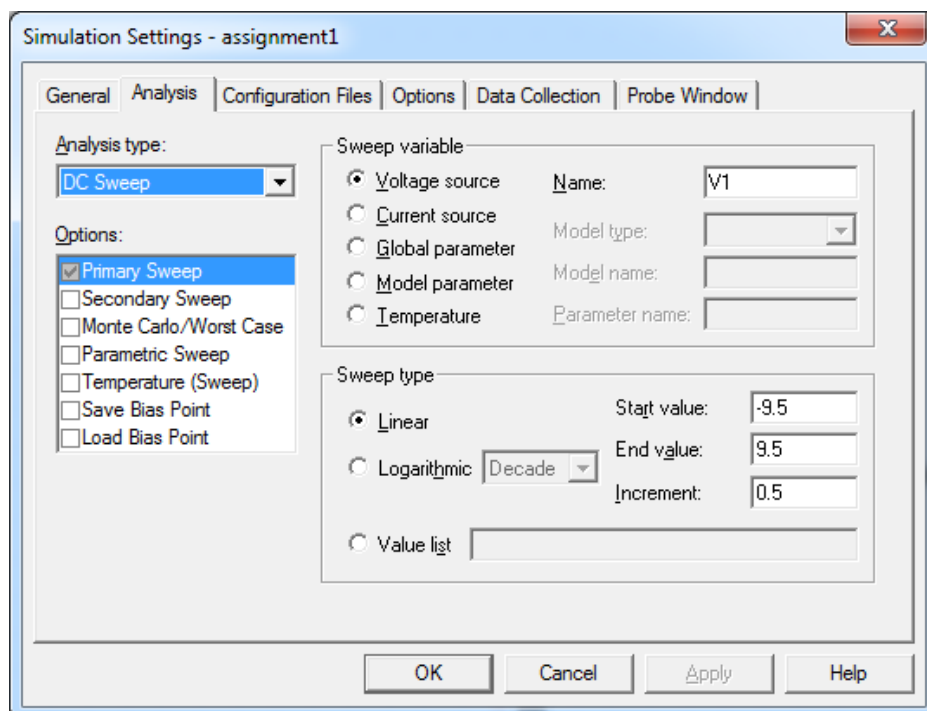
9. Run simulations for the three values of C_{jo} , explain.
10. Connect the circuit as shown in the figure and repeat the steps (2-8) for the shown circuit (save the first circuit).



11. Return to the first circuit (save the second), edit the diode parameter **M** to be equal to 0.42 and **VJ = 0.75V with a 5pF Cjo**.
12. Run transient simulation and comment on the results. (Hint: refer to the device model in this document)
13. Change the Sine amplitude to 0.2V and and adjust the VDC = 0.5V and 9.5V and run transient simulation for both values, explain?
14. Run AC simulations for the previous settings.
15. For the 2nd circuit use the AC source with amplitude = 0.5V and set the simulation setting to **parametric sweep** and set the values of the DC source to **9.5V and -9.5V**, run the simulation and comment on the results.



16. Run a DC sweep on the voltage source from -9.5V to 9.5V, comment on the results.



Remarks to avoid errors:

1. When you create new project make sure it is *Analog or Mixed Signal A/D*
2. The next step, select *Create blank project*.
3. Use the ground named *0* from the library named *source*.