Lecture 16

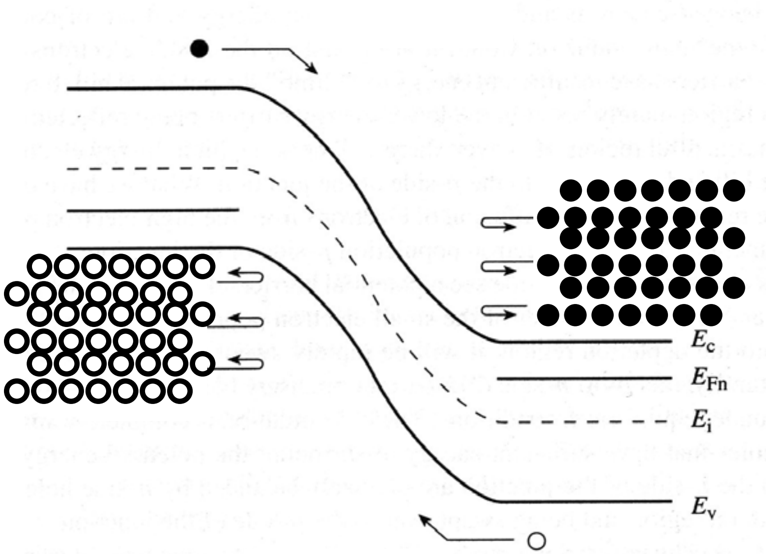
P-N Junction Diodes: Part 6

Photodiodes, Solar Cells (Photovoltaic devices), and Multiple Diode Analysis

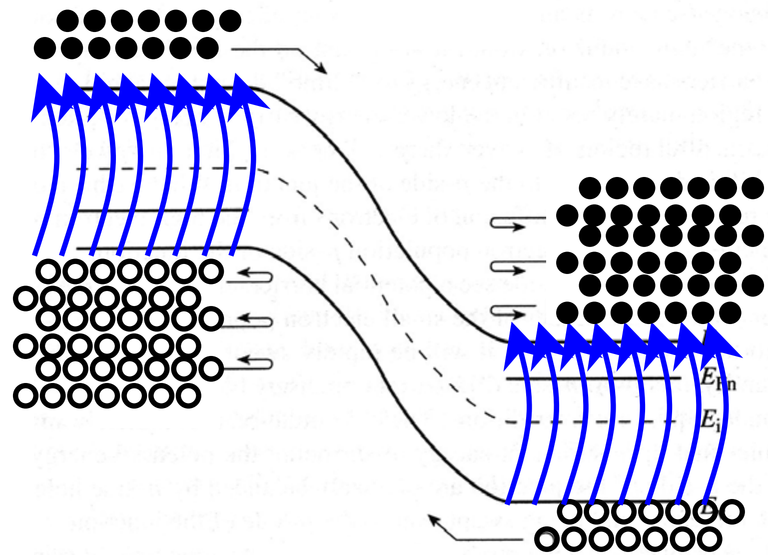
Reading:

Pierret 9.2 and Notes

Photodiode



Reversed Bias Diode with no light illumination



Reversed Bias Diode WITH light illumination results in "extra" drift current due to photogenerated ehp's that can reach the junction.

Photodiode

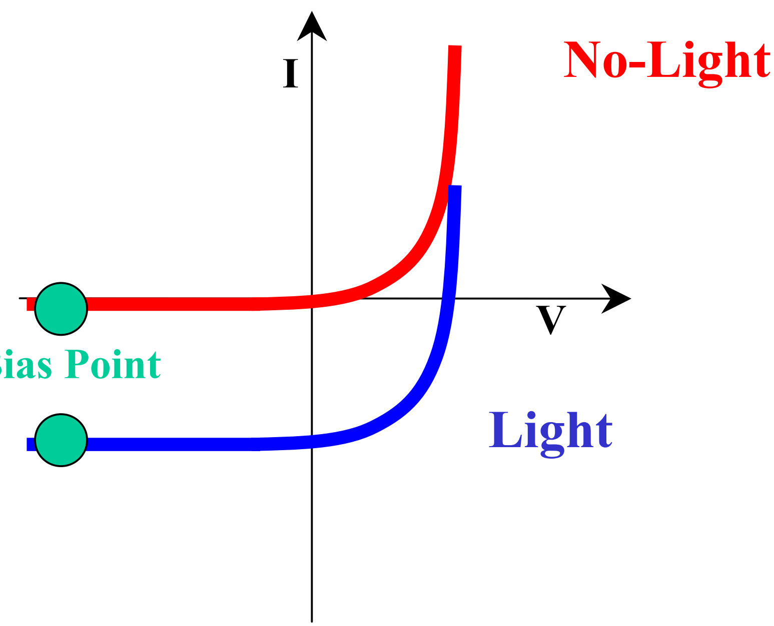
$$I_{total} = I_{dark} + I_{Due\ to\ Light}$$

$$I_{total} = I_o\left(e^{\left(V_D/V_T\right)} - 1\right) + I_{Due\ to\ Light}$$

$$I_{total} = \underbrace{\left(I_o e^{\left(V_D/V_T\right)} - I_o\right)}_{\text{No-Light}} + \underbrace{\left(-qA\right)\left(L_N + W + L_P\right)G_L}_{\text{Light}}$$

No-Light **Light**

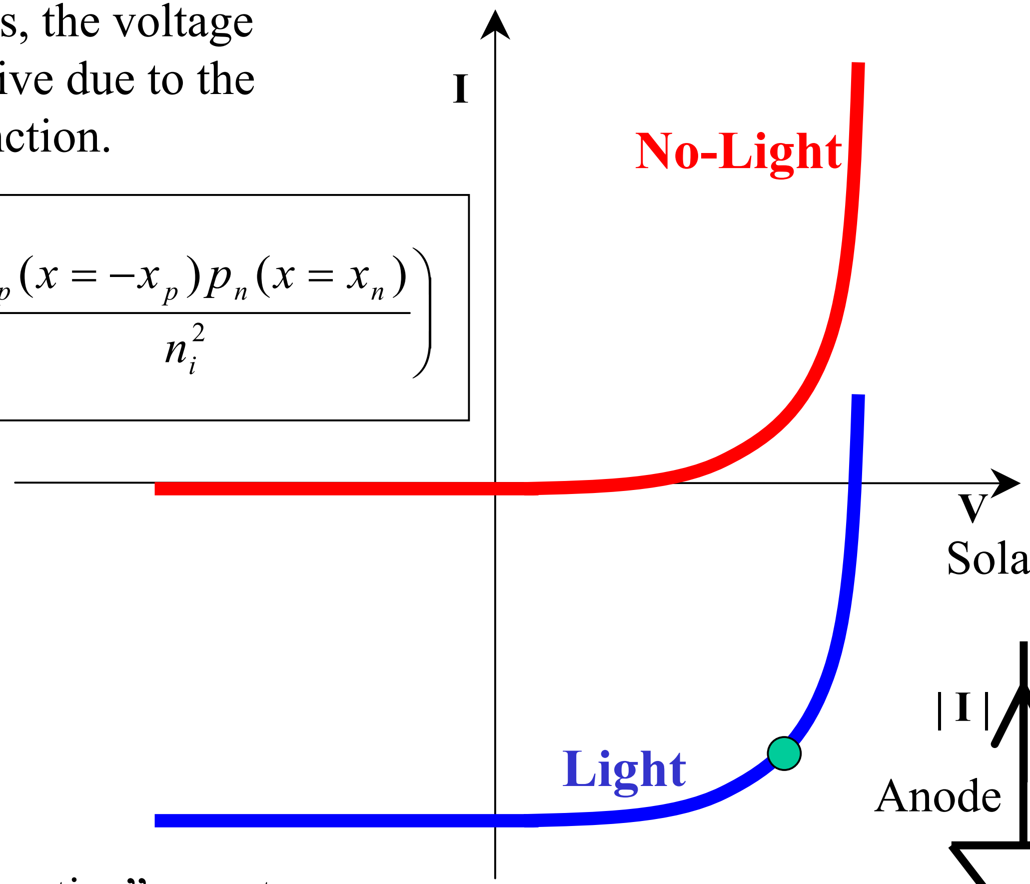
Every EHP created within the depletion region (W) and within a diffusion length away from the depletion region is collected (sweeped across the junction by the electric field) as photocurrent (current resulting from light). All other EHP's recombine before they can be collected.



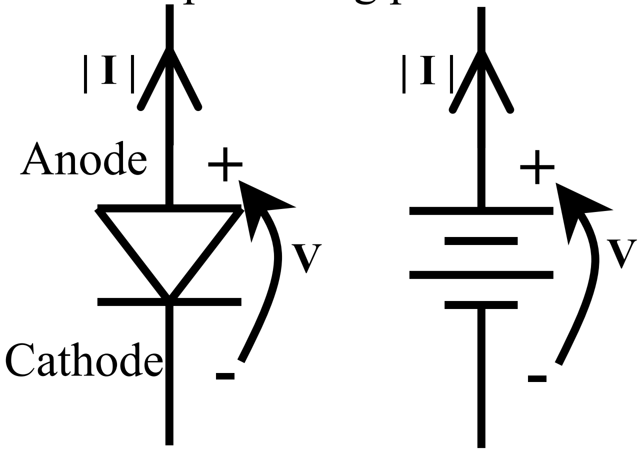
Solar Cell = Unbiased Photodiode

Since $n_p > n_o$ and $p_n > p_o$ at the junction edges, the voltage must be positive due to the law of the junction.

$$V_A = \frac{kT}{q}\ln\left(\frac{n_p(x=-x_p)\,p_n(x=x_n)}{n_i^2}\right)$$



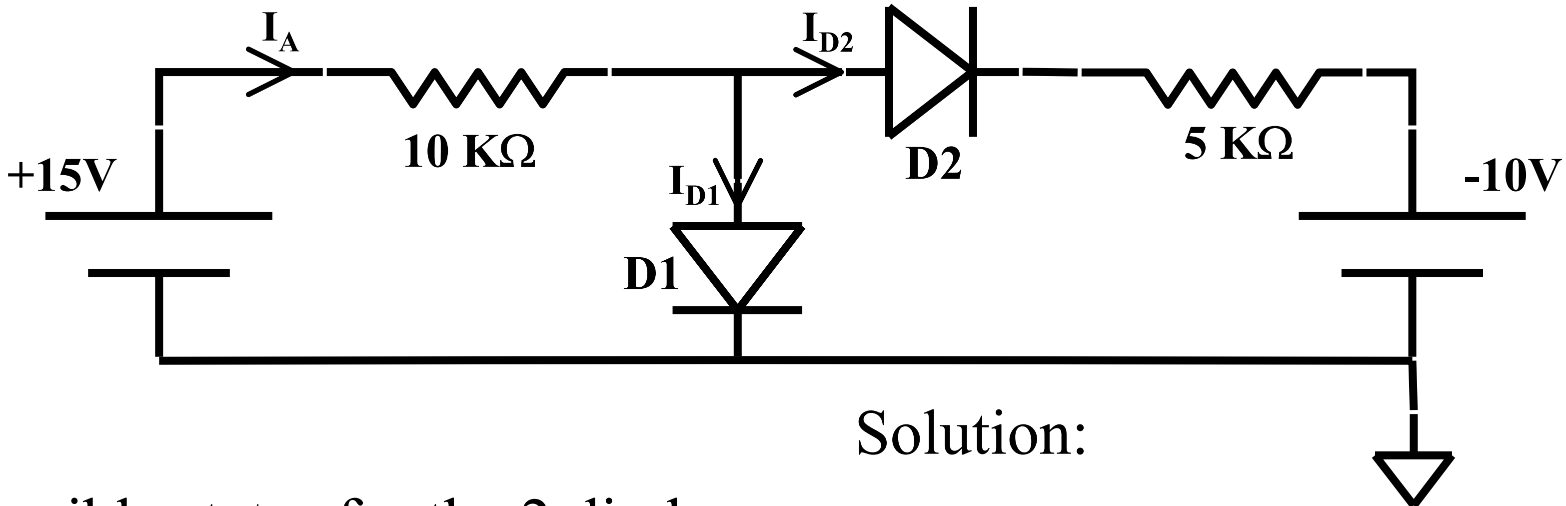
Solar Cell acts as a battery producing power



Current is "negative" or out of the p-type side (anode)

Multiple Diode Analysis

Consider the 2- diode circuit below:



Solution:

Make a guess as to one of the possible states of the circuit.

If a diode is assumed on, verify the current calculated flows in the correct direction consistent with the diode being on

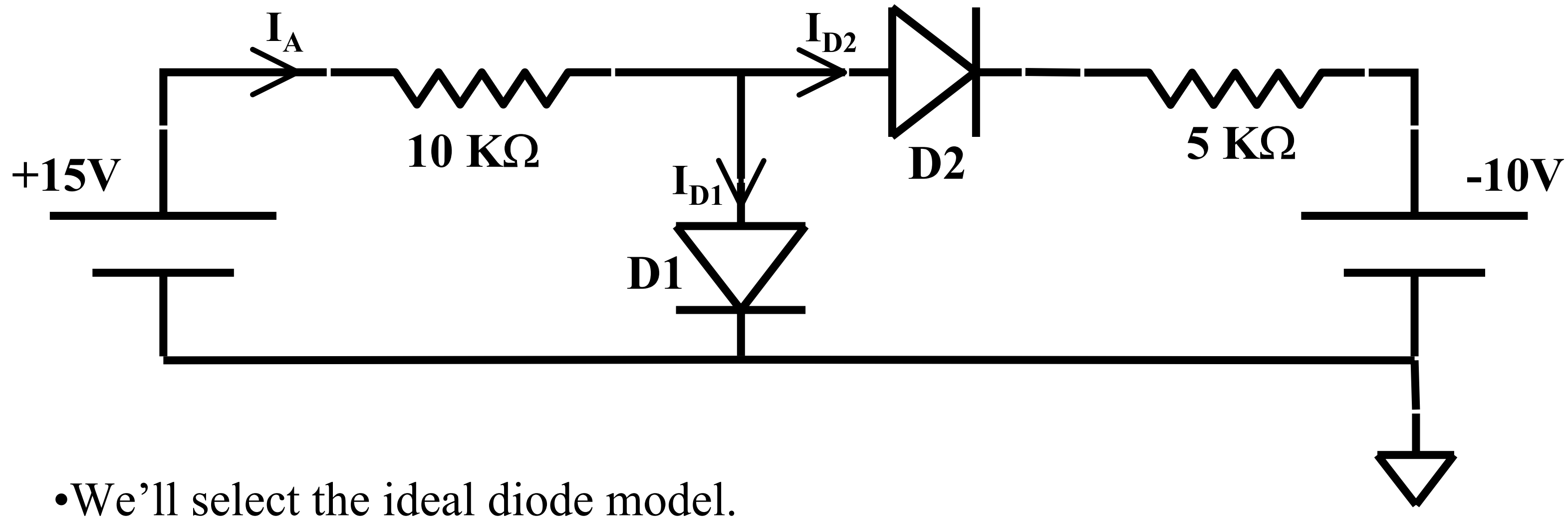
If a diode is assumed off, verify the voltage across the diode calculated has a polarity consistent with the diode being off

Only one solution exists for a given model (ideal or CVD)

Possible states for the 2 diodes:

D1	D2
Off	Off
Off	On
On	Off
On	On

Multiple Diode Analysis



- We'll select the ideal diode model.
- Let's first assume both diodes are on, then:

$$I_A = \left(\frac{(15-0)V}{10k\Omega} \right) = 1.5\ mA$$

$$I_{D2} = \left(\frac{(0-(-10))V}{5k\Omega} \right) = 2.0\ mA$$

but…

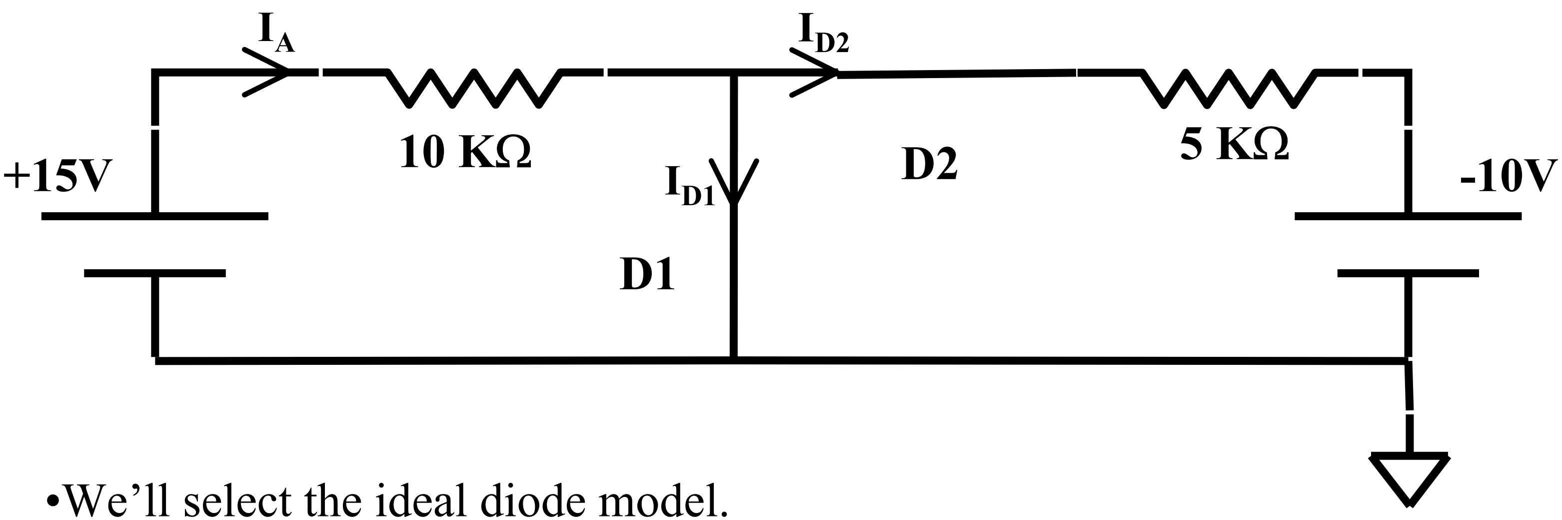
$$I_A = I_{D1} + I_{D2} \Rightarrow I_{D1} = -0.5\ mA$$

- **<u>This contradicts the assumption that D1 is on!</u>**

Possible states for the 2 diodes:

D1	**D2**
Off	Off
Off	On
On	Off
On	On

Multiple Diode Analysis



- We'll select the ideal diode model.
- Let's first assume both diodes are on, then:

$$I_A = \left(\frac{(15-0)V}{10k\Omega} \right) = 1.5\ mA$$

$$I_{D2} = \left(\frac{(0-(-10))V}{5k\Omega} \right) = 2.0\ mA$$

but...

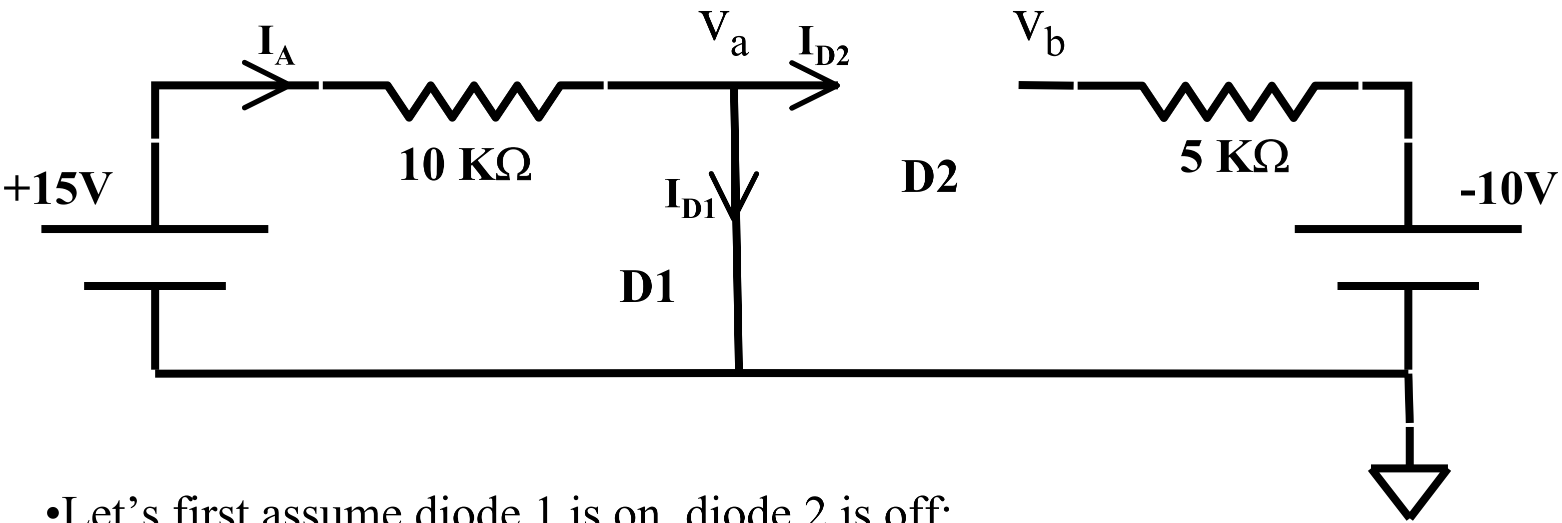
$$I_A = I_{D1} + I_{D2} \Rightarrow I_{D1} = -0.5\ mA$$

- **<u>This contradicts the assumption that D1 is on!</u>**

Possible states for the 2 diodes:

D1	**D2**
Off	Off
Off	On
On	Off
On	On

Multiple Diode Analysis



- Let's first assume diode 1 is on, diode 2 is off:

$$I_A = \left(\frac{(15 - v_a)V}{10k\Omega}\right) = \left(\frac{(15 - 0)V}{10k\Omega}\right) = 1.5\ mA$$

$$I_{D2} = 0\ mA$$

$$v_b = -10v$$

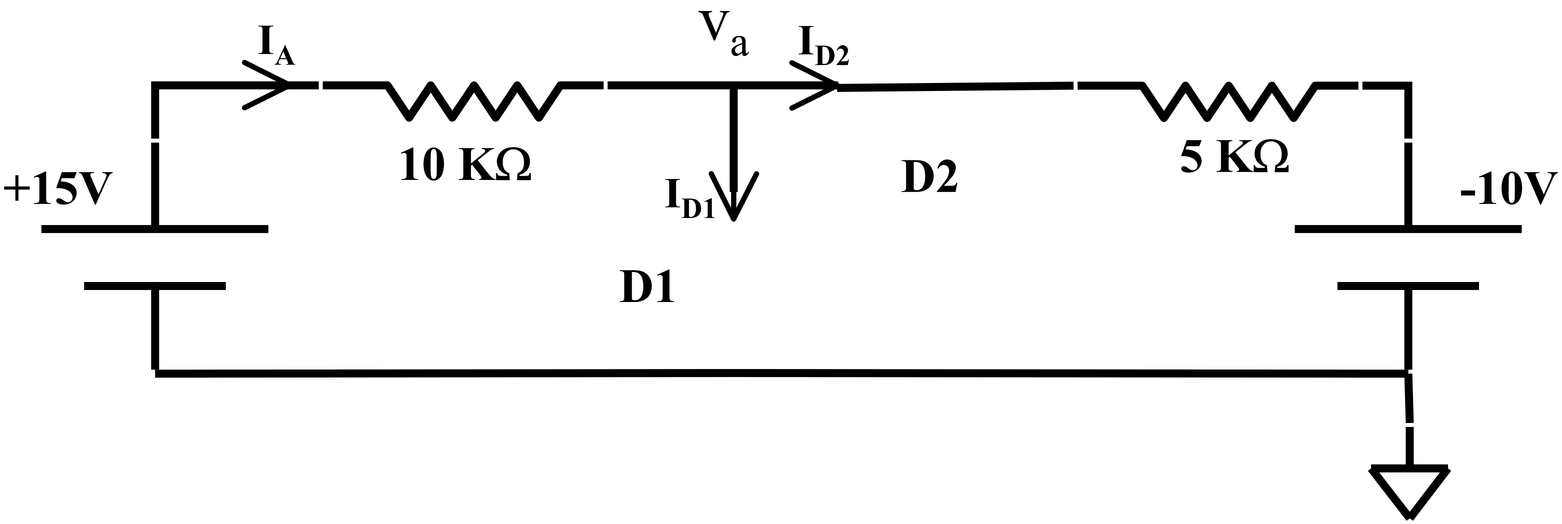
$$V_{Diode1} = v_a - v_b = 0 - (-10)V = 10V$$

- **<u>This contradicts the assumption that D2 is off!</u>**

Possible states for the 2 diodes:

D1	D2
Off	Off
Off	On
On X	Off X
On X	On X

Multiple Diode Analysis



Let's now assume D1 is off and D2 is on:

$$I_A = I_{D2}$$

$$15V - 10{,}000 I_A - 10{,}000 I_A - \left(-10V\right) = 0$$

$$I_A = 1.7\ mA$$

$$v_a - 0 = V_{Diode1} = 15 - 10{,}000 I_A = -1.7V$$

<u>Assumptions: D1 off and D2 on verified!</u>

Possible states for the 2 diodes:

D1	**D2**
Off (X)	Off (X)
Off	On
On (X)	Off (X)
On (X)	On (X)

It is left as an exercise to disprove the last combination.

SPICE Computer simulations of Circuits

•SPICE = "Simulation Program with Integrated Circuit Emphasis"

•A general purpose program that simulates electronic circuits.

•*PSPICE* = PC version of SPICE (from MicroSim Corporation which was recently purchased by Cadence Corporation)

•References:

- •Program (Version 9) is available in a limited use (small number of components) from http://www.orcad.com/Product/Simulation/PSpice/eval.asp
- •Also available (Version 8) with detailed reference and instructions in the required text for ECE3041: "Schematic Capture Using MicroSim Pspice for Windows 95/98/NT" by Herniter. This is the version I will use in class.

Other good "older" references useful for "device model" descriptions are:

[1] P. Tuinenga, *A Guide to Circuit Simulation and Analysis Using PSpice* , 2nd Ed., Englewood, NJ: Prentice Hall, 1992.

[2] S. Reidel and J. Nilsson, Introduction to PSpice, Menlo Park, CA: Addison Wesley, 1997.

[3] M. Rashid, SPICE for Circuits and Electronics Using PSpice, Englewood, NJ: Prentice Hall, 1995.

Types of analysis supported by SPICE:

- Quiescent operating point determination (.OP)
- DC sweeps of current/ voltage sources (.DC)
- Time-domain (transient) response (.TRAN)
- Small-signal frequency response (.AC)
- Fourier analysis (.FOUR)
- Noise analysis (.NOISE)
- Sensitivity analysis (.SENS)
- Thevenin equivalents (.TF)
- Others

SPICE Computer simulations of Circuits

List of most Common SPICE diode model parameters:

Parameter	Symbol	SPICE Name	Units	Default
Saturation current	I_0 or I_S	IS	A	10e-14
Emission coefficient	n or η	N	-	1
Series resistance	R_S	RS	Ω	0
Built- in voltage	V_{bi} or ϕ_j	VJ	V	1
Junction Capacitance	C_{j0}	C_{J0}	F	0
Grading coefficient	m	M	-	0.5
Transit time	τ_t	TT	s	0
Breakdown voltage	V_{BD}	BV	V	∞
Reverse current at breakdown	I_{BD}	IBV	A	10e-10

SPICE Computer simulations of Circuits: In class example

Steps to Perform in SPICE	***Comment***
File-new	
Draw-Get New Part-Dbreak-Place & Close	"Breakout" parts are meant to have the user modify there parameters
Escape	Returns cursor
Repeat above steps for a resistor (R), DC source (VDC) and Ground reference (Gnd_Analog)	In SPICE, a ground reference is *ALWAYS* required.
Connect components with wires	
Edit-model-Edit Instance Model Change model name, save to a library	Change diode model parameters such as IS, BV, etc… and change name and library for which the part is stored in for future use.
Analysis-Display results on schematic-Enable Voltage	Change settings to display bias voltages on schematic
Simulate	Quiescent Bias point is calculated and displayed
Marker-Mark Current into Pin	Used to measure current into diode
Analysis-Setup-DC sweep-Name V1, Start –15V, End 2V, Increment 0.1V	Will enable a sweep of the chosen source, V1
Simulate	A new program, called "Probe" will display the results
Delete VDC (V1)	
Draw-Get New Part-VSIN-Place & Close	We will do a different type of analysis, letting the source vary sinusoidally with time.
DC=-5V, VAMPL=10V, VOFF=0, FREQ=60	60 Hz, 20 V peak-peak sine wave centered around zero, that has a value of –5V for the DC operating point determination.
Analysis-Display results on schematic-Disable Voltage	
Analysis-Setup-Uncheck DC Sweep-Add transient with Print Step=0.1 mS, Final Time=60 mS, Step Ceiling=0.01 mS	Limit step size to 0.01 mS, which is small enough to result in smooth curvatures in signals, stop analyzing after a couple of cycles (60 mS), and print results to the output file (*.out) in larger 0.1mS steps.
Simulate	
Trace-Add-V(R1:1)-V(R1:2)	Add voltage across the diode
Plot-Add-Y-axis-Trace-Add-I(D1)	Add a different y-axis so 2 variables of grossly different magnitude can be compared.