UMass ECE 210 – Fall 2023

Lab 10: LTSpice – Diodes

GOALS:

- Simulate and analyze diode-based circuits
- Understand how to characterize diode turn-on voltage and on/off characteristics (voltage and current)

Lab report:

1. Introduce justification for experiment.
    a. Analyze diode-based circuits via simulation
2. Properly label and document simulation schematics and simulation results
    a. Label components, interesting nodes
    b. Label simulation output plots
    c. Label specified values

You will need to RECORD all of your data independently.

The simulation data required for your lab report are listed in black boxes like this throughout the following parts.

FOLLOW ALL STEPS AND INCLUDE ALL REQUIRED DATA IN YOUR REPORT!

Introduction: **Diodes**

Diodes are useful circuit components that can act as switches (i.e., on or off) and conduct current in only one direction. Since the diode can be on or off, we need to do a few additional steps when analyzing circuits with diodes by hand: we first have to guess whether the diode is on or off, then solve the circuit based on this guess (enforce some condition on the circuit), and finally check to see if the guess was valid. Some details may change depending on how we model the diode, but they will generally follow the table below.

The main parameter of interest is the diode turn-on voltage – when the voltage across the diode equals this value, it is on and conducts current. If the voltage across the diode is less than this value, then the diode is off and does not conduct current (it acts like an open circuit).

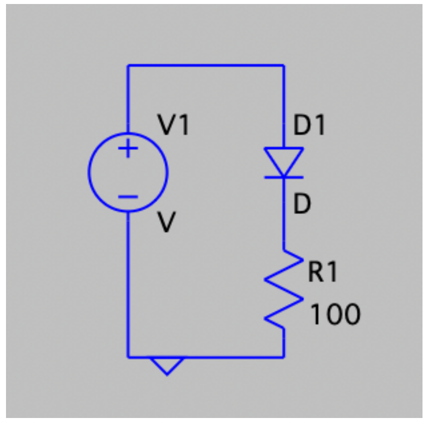
For a diode with turn-on voltage = 0.7 V…

Diode Mode	Enforce	Check
ON	$V_{diode} = 0.7$ V	$I_{diode} > 0$ A?
OFF	$I_{diode} = 0$ A	$V_{diode} < 0.7$ V?

When simulating diode-based circuits, however, we will not need to guess and check the mode. Instead, we will use the simulations to characterize the diode voltage/current in on and off modes, and look at how the mode is affected by other circuit components.

Reference Lab 3 and Lab 8 as you work to remind yourself of how to use LTSpice. Nothing new will be introduced in terms of LTspice, so use this as an opportunity to help you better understand diodes in conjunction with the homework (and practice using LTspice).

<u>Part 1</u>: Identify Diode Turn-on Voltage



Pictured aboved is a simple circuit with a DC voltage source V1, diode D1, and 100ohm resistor R1. The bottom of the circuit is grounded. The main characteristic of a diode (for our purposes) is the voltage that causes it to turn on and conduct current (turn-on voltage). We will use this circuit to identify the turn-on voltage.

→ **Simulate the <u>voltage across</u> and the <u>current through</u> the diode**

Voltage polarity: + on top;   Current direction: + to – (down)

→ **Identify the approximate turn-on voltage**

1. <u>Build the circuit schematic shown above</u>
   1.1. For the diode, type **diode** into the component search menu and place the default option that comes up. You do not need to edit any parameters.
   1.2. For the source value of V1, we will do a DC sweep.
      1.2.1. Write a SPICE statement to sweep **the value of V1 from 0 to 1.5V,** with **0.01V step**
   1.3. Label the two unlabeled nodes (i.e., between V1 and D1; between D1 and R1). Just make it so the labels are understandable for your simulation legend.

2. <u>Run the DC simulation</u>
   2.1. Add another SPICE statement **.op** in addition to your sweep statement.
   2.2. Run the simulation. Plot the diode voltage and current in the same plot.

In your lab report, include the following…

- ☐ Your circuit schematic (with labeled nodes!)
- ☐ Simulation plot with <u>diode voltage</u> and <u>diode current</u>
- ☐ Using the table on page 1 to help define on/off, what is the turn-on voltage for this diode?
  - You can be approximate and read from the plot, or use a .meas statement!

<u>Part 2</u>: Rectifier Circuits

A useful application of diodes is to convert an AC signal to DC – these are called rectifiers. A simple AC signal (e.g., a sine wave) will alternate between positive and negative voltages (and thus forward/reverse-going current). We can use diodes to "rectify" the AC signal into an entirely positive voltage signal with forward-going current. Depending on the amount and configuration of diodes, we can build different types of rectifier circuits.

→ **Simulate a half-wave rectifier (using the same circuit schematic from part 1)**

→ **Identify the approximate turn-on voltage**

1. <u>Change some values</u>
   1.1. For the source value of V1, change the value to **SINE(0 5 100)**
      1.1.1. V1 is now a 100Hz sine wave with V_{peak} = 5 V
   1.2. Delete your SPICE directives (**.op, DC sweep, etc.**)
   1.3. Add a SPICE directive to measure the transient response for 40ms: **.tran 0.04s**
      1.3.1. This will include 4 periods of V1

2. <u>Run the transient simulation</u>
   2.1. Separate the simulation window into two plot panes.
      2.1.1. In one pane, plot the <u>diode current</u> and <u>diode voltage</u>.
      2.1.2. In the other pane, plot the <u>source voltage V1</u> and the <u>voltage across resistor R1</u>.

In your V1/resistor plot, you should see the entire V1 sine wave, but only the positive halves of this sine wave for the resistor voltage. When V1 goes negative, the resistor voltage goes to 0V.

If we treat the resistor voltage as our output and V1 as our input, then we are only outputting the positive parts of V1, i.e., we are rectifying *half* the input signal! The slight mismatch in voltage is coming from the voltage drop across R1. More diodes (in a specific configuration) could be used to rectify the negative halves into positive halfs – this would be a full-wave rectifier.

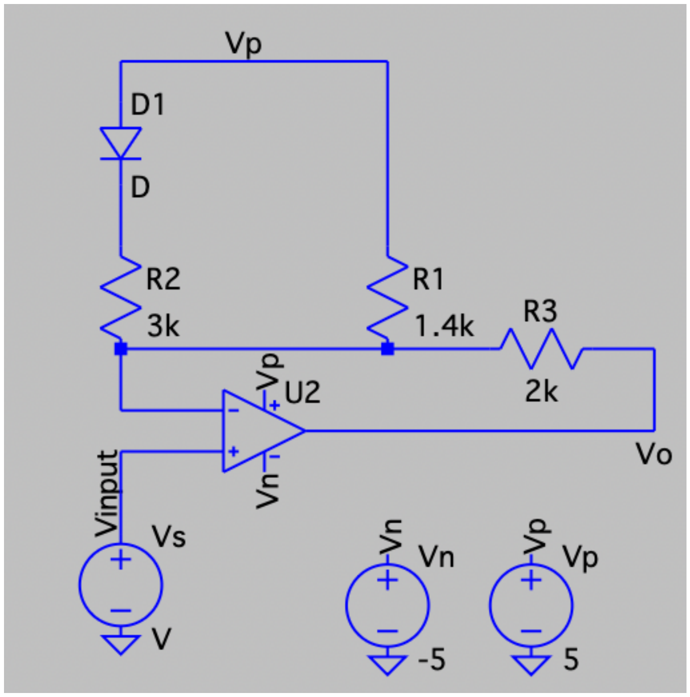
From your diode voltage/current plot, you can see when the diode is off (current = 0A; voltage < turn-on value). You can also see when the diode is on (voltage ~= turn-on value; current > 0A).

3. <u>Check the turn-on voltage</u>
   3.1. In the diode voltage/current plot, what is the turn-on voltage, approximately?
      3.1.1. You can indicate this on the plot, or with a .meas statement. It should be similar to the turn-on voltage from part 1 (no comparison needed).
   3.2. In the V1/resistor voltage plot, you should be able to see a mismatch between the two voltages. In particular, the resistor voltage starts rising after V1. Why?
      3.2.1. This should be a simple answer – think about the diode switching between on/off. What must happen for the resistor voltage to start rising?
      3.2.2. <u>You should indicate a value on the diode voltage/current plot or use a .meas statement to prove this</u> (e.g., what is the value of V1 once resistor voltage starts changing?)

In your lab report, include the following…

- ☐ Your circuit schematic (nodes labeled!)
- ☐ Simulation plot with <u>diode voltage</u> and <u>diode current</u>
- ☐ Simulation plot with <u>V1 voltage</u> and <u>resistor voltage</u>
- ☐ Diode turn-on voltage – shown by indicating on voltage/current plot OR .meas statement
- ☐ Answer to question – explained using either diode voltage/current plot OR .meas statement

<u>Part 3</u>: Diode + Op-amp Circuit



Pictured above is an op-amp circuit with a diode and some resistors. Two DC voltage sources, Vp and Vn, are built "separate" from the main circuit, but they are just used to supply voltage to the three nodes labeled Vp or Vn. This is simply a cleaner way of making the schematic – we essentially have a +5V or -5V source going to each node labeled Vp or Vn, respectively.

In this circuit, it turns out that the value of the input voltage source Vs will determine the diode mode (on or off). We will see what value of Vs causes the diode to switch modes.

→ Simulate the <u>diode voltage</u>, <u>diode current</u>, and <u>input voltage Vs</u>
→ Identify the Vs value that causes the diode mode to switch

1. Build the circuit schematic shown above
   1.1. For the source value of Vs, we will do a DC sweep.
      1.1.1. Write a SPICE statement to sweep **the value of Vs from 1 to 6V,** with **0.1V step**
   1.2. Make sure the nodes are labeled accordingly.

2. Run the transient simulation
   2.1. Add a SPICE directive to measure DC values: **.op**
   2.2. Run the simulation.
   2.3. Plot the diode voltage, diode current, and input voltage Vs.

3. Identify the diode modes relative to Vs
   3.1. Based on parts 1 and 2, you should be able to identify when the diode is on/off in the plot.
      3.1.1. Indicate on your plot when the diode is on, and when it is off.
   3.2. You should also be able to identify what Vs is when the diode switches modes.
      3.2.1. Use a .meas statement to find this value of Vs. You can base it off of either the diode voltage or the diode current (or use two statements and do both!)

In your lab report, include the following…

- ☐ Your circuit schematic (nodes labeled!)
- ☐ Simulation plot with diode voltage, diode current, and Vs voltage
- ☐ Diode modes shown on the simulation plot
- ☐ Vs value that causes diode to switch modes (exact value shown by a .meas statement)

LAB REPORT DUE NEXT WEEK

LAB REPORT 10 – RUBRIC

2,000-word limit 1 report per person

Submission contents listed briefly below, but double check the black box for each section!

Part	Submission Material	Points
	Introduce and define concept (transient analysis)	2.5
	Motivation for experiment	2.5
1	Diode turn-on voltage	15
	Schematic	5
	Simulation plot (with specified voltages/currents)	5
	Turn-on voltage value	5
2	Half-wave Rectifier	20
	Schematic	5
	Simulation plots (with specified voltages/currents)	5
	Turn-on voltage value	5
	Plot analysis (answer the question as specified)	5
3	Diode + Op-amp	20
	Schematics	5
	Simulation plot (with specified voltages/currents)	5
	Diode modes (indicated on plot)	5
	Vs value	5

Appendix

Helpful list of LTspice syntax and shortcuts

COMMANDS

SPICE Analysis

.OP	find the DC operating point
.TRAN	perform nonlinear transient analysis
.AC	perform small signal AC analysis
.DC	perform DC source sweep analysis
.TF	find the DC small-signal transfer function
.NOISE	perform noise analysis

SPICE Directives

.BACKANNO	annotate subcircuit pin names on port currents
.END	end of netlist
.ENDS	end of subcircuit definition
.FOUR	compute fourier component
.FUNC	user defined functions
.FERRET	download a file from URL
.GLOBAL	declare global nodes
.IC	set initial conditions
.INCLUDE	include file
.LIB	include library
.LOADBIAS	load a previously solved DC solution
.MACHINE	arbitrary state machine
.MEASURE	evaluate user-defined electrical quantities
.MODEL	define a SPICE model
.NET	compute network parameters in .AC analysis
.NODESET	supply hints for initial DC solution
.OPTIONS	set simulator options
.PARAM	user-defined parameters
.SAVE	limit the quantity of saved data
.SAVEBIAS	save operating point to disk
.STEP	parameter sweeps
.SUBCKT	define a subcircuit
.TEMP	temperature sweeps
.TEXT	user-defined string
.WAVE	write selected nodes to a .WAV file

ANALOG DEVICES

LTspice
Fast • Free • Unlimited

SHORTCUTS

Schematic and Symbol Editing Modes

Windows	Choose Mode then select component Exit mode: Press Esc or right-click	Mac
F5 or Delete or Ctrl X	cut/delete	F5
F6 or Ctrl C	copy/duplicate*	F6
F7	move* *unselected wires remain*	F7
F8	drag* *connected wires adjust*	F8
Esc	exit current mode *or right-click*	Esc

Zoom and Grid

Windows	Zoom in and out with scroll wheel or track pad pinch	Mac
Ctrl Z	**Schematic** zoom area (drag over area) zoom in (click on scheme) **Waveform** zoom area is default mode F9 for previous zoom **Symbol** zoom in	
Ctrl B	zoom out	
Space	zoom to fit (schematic viewer)	Space
Ctrl E	zoom extents (waveform viewer)	
Ctrl G	toggle grid	

Place Component Modes*

Windows	Press Esc or right-click to exit place component mode	Mac
R	resistor	R
C	capacitor	C
L	inductor	L
D	diode	D
G	ground	G
V	voltage	V
S	spice directive *right-click text field to open "Help me Edit" dialog*	S
T	text/comment	T
F2	component	F2
F3	draw wire	F3
F4	label net	F4
	bus tap	B

*Rotate and Mirror

Windows	*enabled in place modes	Mac
Ctrl R	rotate	⌘ R
Ctrl E	mirror	⌘ E

Undo/Redo

Windows	### Levels of Undo	Mac
F9	undo	F9 or ⌘ Z
⇧ F9 or Ctrl ⇧ Z	redo	⇧ F9 or ⌘ ⇧ Z

TRICKS

Waveforms

Windows	when clicking waveform label	Mac
click	add cursor and see measure	click
Alt click	highlight corresponding net in schematic	⌘ click
Ctrl click	integrate waveform	Ctrl click

Schematics

Windows		Mac
Alt click	**component:** plot instantaneous power **wire:** plot current	⌘ click
hold Ctrl	draw wires at an angle	hold Shift
Ctrl Alt Shift H	show hidden component values/text, e.g. parallel or series resistance and capacitance	

any text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, active low, signal

NUMBERS

Prefixes (Case Insensitive)

LTspice	Means	Value
T or t	tera	10^{12}
G or g	giga	10^{9}
meg	mega	10^{6}
K or k	kilo	10^{3}
M or m	milli	10^{-3}
U or u	micro	10^{-6}
N or n	nano	10^{-9}
P or p	pico	10^{-12}
F or f	femto	10^{-15}

Constants

LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0
mil	25.4×10^{-6}m

©2022 Analog Devices, Inc. All rights reserved. Trademarks and registered trademarks are the property of their respective owners. Ahead of What's Possible is a trademark of Analog Devices. LTspice-4/22(A) analog.com