LABORATORY 2

CAD Tools

Guide

Practical circuit design occurs in three stages:

1. Design of an appropriate circuit diagram and calculation of all component values to meet the specifications.
2. Verification of the design with circuit simulation.
3. Building and testing the circuit in the laboratory.

The building phase usually takes considerable time and resources. Tracking down design errors in the laboratory can take hours. The turn-around time for fabricating and stuffing printed circuit boards is days to weeks, and fabricating custom integrated circuit takes several months, not to speak of the high cost.

Circuit simulation allows verifying circuit performance before committing to fabrication. Errors found at this stage are much less time consuming and costly to fix. Although most circuits designed are relatively simple and could perhaps be successfully designed without simulation, every experienced engineer can tell you about disasters where a circuit that was supposedly too simple to simulate did not work properly malfunctioned and held up a larger project. Many successful electronics companies have policies allowing circuits to be built only after they have been successfully simulated.

This lab will introduce you the simple use of two circuit simulation tools, PSpice and MultiSim.

Lab2 Report

|			
Name	_____	TA Checkoff	_____
Teammate	_____	Score	_____

PSpice

In this part, we will explore computer software used to simulate electric circuits. We will use the PSpice educational suite where already installed on the lab machine. If you want install it on your own laptops, you can ask TA for the help. If you want to download the PSpice by yourself, the website is below.

http://www.orcad.com/resources/orcad-downloads

1. Example Circuit

Here is the circuit (shown in Figure 1) we will simulate in this lab exercise. Use nodal analysis (by hand) to find the voltage at each node with respect to ground. Show your computation steps in the box below.

V_1=_____; V_2=_____; V_3=_____   __/**15pt**
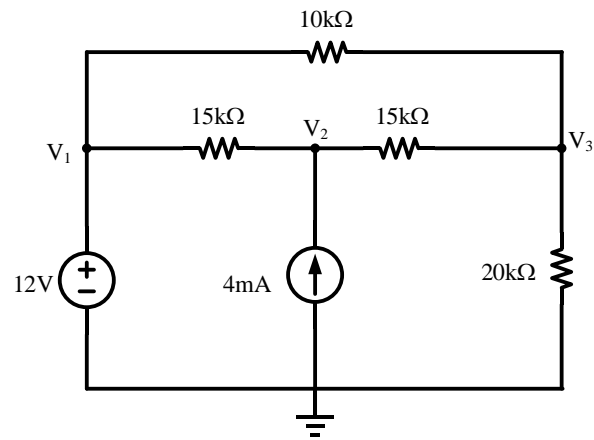
Figure 1

2. Creating a Circuit Graphically According to Figure 1 and then Simulate it

PSpice Capture allows you to enter a circuit into the PSpice simulator graphically.
- From the Start Menu, within Programs, go to the PSpice folder and select Capture.
- Click on the "New" icon or go to File -> New -> Project. Name the project whatever you like.
- From the options, choose "Analog or Mixed A/D". This will load all the part libraries you need.
- Finally, choose your working directory and click OK. Then click on the "Create a blank project" and click OK. A window with grid points will appear; this is where you will draw the circuit to be simulated.
- To make your life easier, change the preferences so that the circuit elements you place are aligned with the grid. To do this, go to Options -> Preferences -> Grid Display and make sure "Pointer Snap to Grid" is checked in both cases.

a. Placing Parts

Now you are ready to start placing the parts of your circuit. Each circuit element has a name in PSpice — and there are hundreds of them. Go to Place -> Place Part and a window appears.   (The toolbar on the right hand side also has a "Place Part" button.)   The parts are organized into libraries depending on the type of part. To see what kind of parts PSpice has, highlight a library and a part--a picture of the part will come up.

The parts we will need are:

R	in the library ANALOG	resistor
IDC	in the library SOURCE	ideal current source (constant current)
VDC	in the library SOURCE	ideal voltage source (constant voltage)

To place a part, select it, click OK, and stamp it on the schematic wherever you need it. To change the value, double click the value (if you double-click the symbol, you will get a confusing options screen). PSpice recognizes the following strings (case-insensitive) as powers-of-ten prefixes:

p	pico	t	tera
n	nano	g	giga
u	micro	meg	mega
m	milli	k	kilo

To change the name of the element, double-click the name. The parts are named automatically and you don't have to change the names.

b. Placing Wire and Ground

After you have placed all the elements and set the values, you need to wire the circuit together.   Go to Place -> Wire or click the wire icon on the toolbar at right. Click once to start a wire and click again to end it. The program will show a red circle when you connect to a wire or terminal.

Once you have connected the circuit, you must tell the simulator where the ground node is—it uses nodal analysis to find voltages and currents in the circuit. Ground is also known as "node 0" in the simulator.   To place ground, go to Place -> Ground or the ground icon on the toolbar at right. Choose 0 (zero) from the SOURCE or CAPSYM library. Place the ground and attach it with wire.

c. Running the Simulation

Now you are ready to simulate the circuit to find all of the voltages and currents. PSpice can run all sorts of simulations—it can plot currents and voltages over time (transient analysis), introduce random variations (Monte Carlo analysis), find current and voltage for a range of settings (DC and AC sweep, temperature), and more. Today, we will perform a Bias Point Analysis, which is a regular static analysis that finds voltage and current values.

To set up the simulation, go to PSpice -> New Simulation Profile->Name it-> Create . Under Analysis Type, choose Bias Point and click OK.

To run the simulation, go to PSpice -> Run . The schematic program will launch the simulator.   Look in the lower-left hand space for messages as the simulation runs. If there are errors reported, check to see that all of your connections are intact and that ground is connected.

Print out your circuit diagram and paste it into the form shown on the next page.

__/20pt

MultiSim

In lab2 we are using a simulator called MultiSim to verify our circuits. We will use the MultiSim which is already installed on the lab machine. If you want install it on your own laptops, you can ask TA for the help. If you want to download the MultiSim by yourself, the website is below.

http://www.ni.com/multisim/student-edition/zhs/

1. Component characteristics

Use MultiSim to produce a plot of the I/V characteristics of a <u>Diode virtual</u> (Figure 2) for V = 0 … 1V.

Now you are ready to start placing the parts of your circuit. Go to Place -> Component-> Diodes -> Diodes_virtual. Place a diode.

Suggestion: click the Simulate\Instruments\IV analyzer. It is specially used to measure the I-V characteristic curve of the diode and MOS tube.
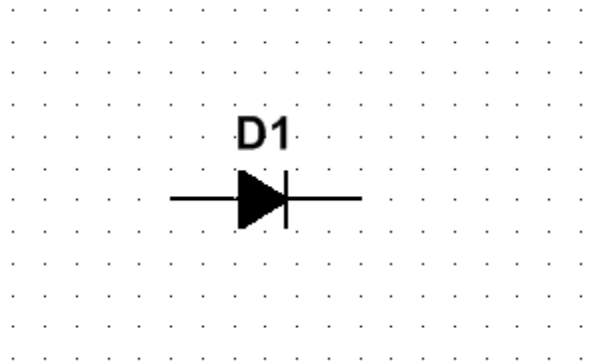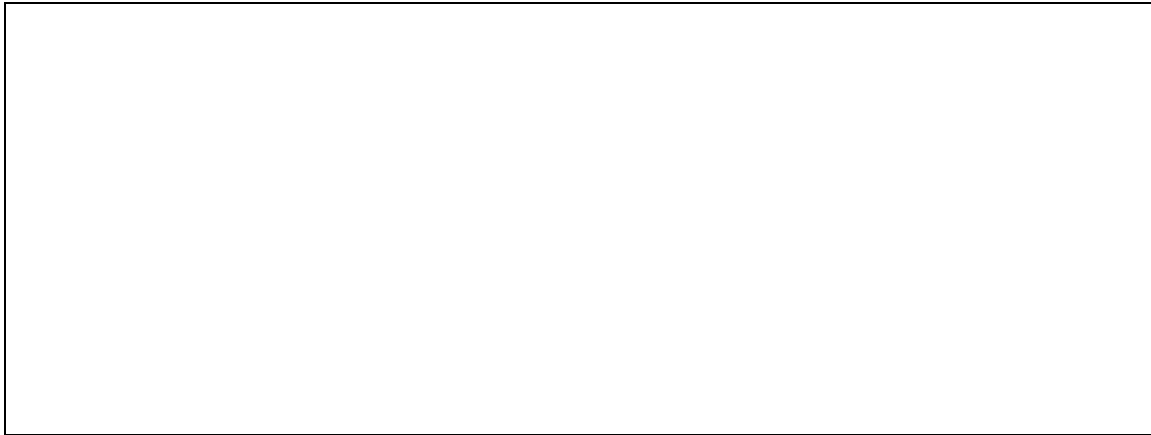

Figure 2: Diode Symbol

Print out your circuit diagram and paste it into the form.

__/20pt

Use your plot to determine by what amount the voltage across the diode must be increased to increase the current from 20µA to 100µA?

__/10pt

Simulated Value: ____to_____mV

It turns out that this value (usually referred to by the term "subthreshold slope") has great importance for the power dissipation of electronic circuits: it tells how well circuits can be turned off to conserve power. The value obtained above from simulation is characteristic for transistors used today. Finding new types of devices for which this value is lower is an area of intense research and opportunity for great fame and wealth!

2. Node voltage analysis

Calculate the value of voltage V_x (Figure 3) in the circuit below and use MultiSim to verify your result. In sources, there are Ground and AC_power and DC_power.

__/10pt

Calculated value for V_x: _____V
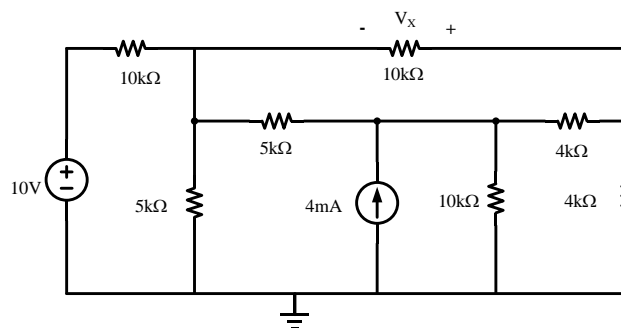Simulated value for V_x :   _____V

Figure 3

Print out your circuit diagram and paste it into the form shown on the page.

__/25pt

Reference

[1] UC Berkeley, course EE43-100, Spring 2012.

TA: _____

Total: _____**of 100Pt.**