Experiment 4: Computer Aided Design and Analysis

1 OBJECTIVES.

This experiment will provide exposure to PSpice software. A primary purpose of this lab course is for you to become familiar with the use of the software and to learn to use it to assist you in the analysis of circuits. The software is already installed in the computer of every station. This is just an introduction to PSpice, you will be getting more directions on how to do some particular tasks as soon as you will be needing to do so. Hopefully by the end of the semester you will be proficient with the software.

2 INTRODUCTION AND TEST CIRCUITS.

For those who are familiar with older versions of PSpice please note that the program we now need to run in order to do schematics is called CAPTURE. The program that will let us do the simulations and see the graphics is called PSPICE. You still can run the simulation from the program where your schematic is. There are a lot of things we can do with this software but the most important for you to learn, so you can successfully use the software, are:

- Design and draw circuits.
- Simulate circuits using PSpice.
- Analyze simulation results using PSpice (Probe for older versions).

For this course you will not need the full capability of Capture. The devices that we will use will be resistors, inductors, capacitors and various sources. However it is good to know that Capture has an extensive symbol libraries and includes a fully integrated symbol editor for creating your own symbols or modifying existing symbols.

The main tasks in Capture are:

- Creating and editing designs.
- Creating and editing symbols.
- Creating and editing hierarchical designs.
- Preparing your design for simulation.

3 PREPARATION.

Be sure to read over the appropriate appendices in your lab manual so you remember how to use the breadboard, the equipment and how to read component values off of the components themselves.

We will start by creating a simulation and analysis of the circuit shown in figure 4-1. The objective of the analysis will be to find the current in the 10 Ω resistor.
Proceed as follows, to get the answer using PSpice:

1. Run the CAPTURE program.
2. Select File/New/Project from the File menu.
3. On the New Project window select Analog or Mixed A/D, and give a name to your project, then click Ok.
4. You will get the Create PSpice Project window, select create a blank project, then click Ok.
5. now you will be in the schematic environment where you are to build your circuit.
6. Select Place/Part from the Place menu.
7. Click ANALOG from the box called Libraries:, then look for the part called R. You can do it either by scrolling down on the Part List: box or by typing R on the Part box. Then click OK.
8. Use the mouse to place the resistor where you want and then click to leave the resistor there, you can continue placing as much resistors as you need and once you have finished placing the resistors Right-click your mouse and select end mode.
9. To rotate the components there are two options:
   - Rotate a component once is placed: Select the component by clicking on it and then use Ctrl-R.
   - Rotate the component before it is placed: Just use Ctrl-R.
10. Select Place/Part from the Place menu.
11. Click SOURCE from the box called Libraries:, then look for the part called VDC. You can do it either by scrolling down on the Part List: box or by typing VDC on the Part box. Then click OK.
12. Place the Source. Repeat steps 10 - 12 to get and place a current source named IDC.
13. Select Place/Wire and start wiring the circuit. To start a wire click on the component terminal where you want it to begin. Then click on the component terminal where you want it to finish. You can continue placing wires untill all components are wired. Then Right-click and select end wire.
14. Select Place/Ground from the Place menu, click on GND/CAPSYM. Now you are able to see the ground symbol.
15. Type 0 on the Name: box and then click Ok. Then place the ground. Wire it if necessary.
16. Now change the component values to the required ones. To do this you just ned to double-click on the parameter you want to change. A window will appear where you will be able to set a new value for that parameter.
17. Once you have finished building your circuit, the next step is to prepare it for simulation.
18. Select PSpice/New Simulation Profile type a name, this can be the same name as your project, and click Ok.
19. The Simulations Settings window will now appear. You can set up the type of analysis you want PSpice to do. In this case it will be Bias Point. Click Apply then Ok.
20. Now you are ready to simulate the circuit. Select PSpice/Run and wait untill the PSpice finishes. Go back to Capture and see the voltages and currents on all the nodes.
21. If you are not seeing any readout of the voltages and currents then select PSpice/Bias Point/Enable Bias Voltage Display and PSpice/Bias Point/Enable Bias Current Display. Make sure that PSpice/Bias Point/Enable is checked.
4  PROCEDURE.


1. Simulate and analyze problems 11 and 15 from chapter 2.
2. Simulate and analyze problems 10 and 46 from chapter 3.

Hint: To leave a pin floating you have to do the following:

- Double-click on the pin you will leave with no connection.
- Make sure you are on the tab called pins.
- Click on New Column..., type FLOAT in the name: box.
- Then type RtoGND in the value: box. Click Apply then Cancel.
- Close the Propert Editor window.

5  Analysis

Be sure to answer any question in the procedure section. In your write up, include a printout of the output. Also, solve the problems analytically to verify the PSpice solution.

- To find the output file select PSpice/View Output File.
Figure 4-1: Simple DC circuit.
4.1 Find $v_g$ and $i_g$. Verify that the total power developed equals the total power dissipated.

![Figure 4.1 (P2.11)](image1)

4.2

a) Find $i_g$ and $i_c$. 
b) Find the power dissipated in each resistor. 
   Find $v_g$ 
c) Show that the power delivered by the current source is equal to the power absorbed by all the other components

![Figure 4.2 (P2.15)](image2)

4.3 Find $v_0$.

![Figure 4.3 (P3.10)](image3)

4.4 Find the power dissipated in the 3kohm resistor in the circuit.

![Figure 4.4 (P3.46)](image4)