EE 43

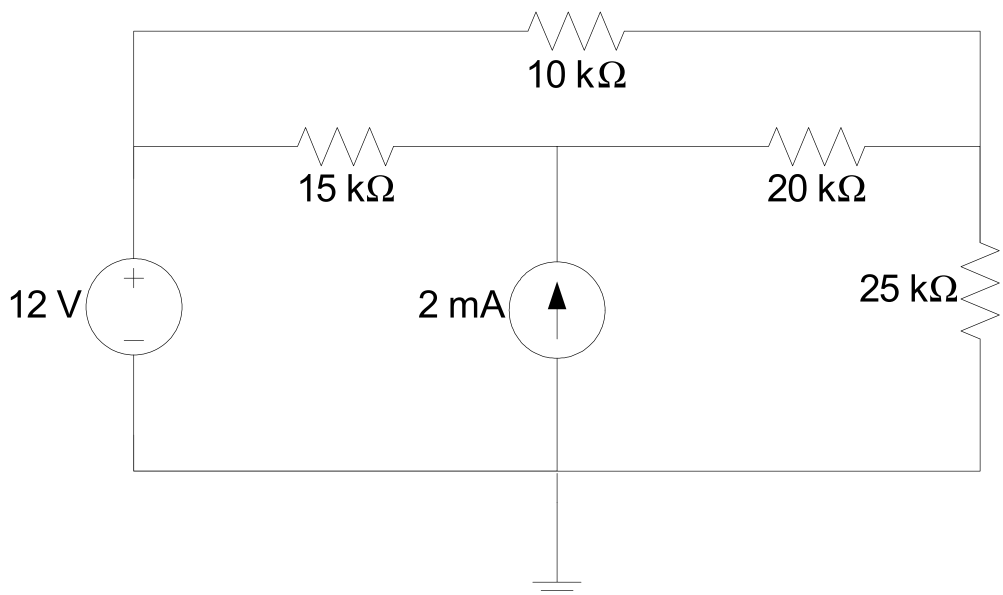
Circuit Simulators

In this lab, we will explore computer software used to simulate electric circuits. We will use the PSpice educational suite, which is available for free download at

http://www.orcad.com/downloads/demo/default.asp

1. Example Circuit

Here is the circuit we will simulate in this lab exercise.



Use nodal analysis (by hand) to find the voltage at each node with respect to ground. This goes into your lab report.

2. Creating a Circuit Graphically

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.

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.

2a. 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. If you don't see any libraries, ask your GSI.

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	10^{-12}	**t**	tera	10^{12}
n	nano	10^{-9}	**g**	giga	10^{9}
u	micro	10^{-6}	**meg**	mega	10^{6}
m	milli	10^{-3}	**k**	kilo	10^{3}

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.

2b. 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.

2c. 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 -> Simulation Profile** . Under **Analysis Type**, choose **Bias Point Analysis** 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.

The simulator uses text files as input and output. The schematic program creates a text file recording the circuit, and the simulator places the voltages and currents in a text file. To view the output file, go to **View -> Output File** in the simulator. What you see—confusing stuff. But at the bottom of the output file, you should see node voltages, the current through each voltage source, and the total power dissipated. The problem—the schematic program gave its own crazy names to the nodes (e.g., N00010) so you don't know which node is which. To give names to the nodes, go back to the schematic.

Go to **Place -> Net Alias**. Or, click the net alias button at right. This will allow you to name the nodes. Type a name for the node (e.g., Node1. Don't use whitespace.) Click OK and the label appears. Place the label so that the border intersects with some of the wire on the node you are naming.

Once you have named all your nodes, you can run the simulation again. Now, the output file will include node voltages with the nodes as you named them. **Print the output file and the schematic. This goes into your lab report.**

Back on the schematic, you can display the voltages and currents using the **V** and **I** on the horizontal toolbar. Click on them and voltages and currents appear.

There are many other ways to view voltages and currents using the other types of simulations. Numerous tutorials are available online—explore the software!

3. Creating a Circuit via Text Input

We will now create a text file describing the circuit to serve as direct input to the simulator, rather than having the schematic program create an input text file for us. When performing large-scale simulations, one often uses scripts to run the simulation, and knowledge of the text interface can be useful.

3a. <u>Anatomy of a PSpice Input File</u>

You may begin a new input text file by opening the simulator; the application is called **PSpice AD** . Go to **New -> Text File.** The following site has a good tutorial:

http://dave.uta.edu/dillon/pspice/

The first line of a PSpice input file is always a comment line. You can use it for the title. Create a comment line with an asterisk * . Anything after a semicolon ; is also ignored.

The next lines specify the circuit. In general, each element has its own line with the information

SYMBOLname begin_node end_node options value

You have to know your part symbol and its associated options to properly specify each element. You also have to name all your nodes, and pick a ground (0) node.

To specify a resistor, start with an R. Complete the name of your resistor by appending a string of your choice. Then specify the beginning and ending nodes and its value.

Example:

R_10k Node_1 Node_3 10k

To specify a constant voltage source, start with a V. Complete the name of your resistor by appending a string of your choice. Then specify the beginning (+) and ending (-) nodes. Then say that it is a DC source, and give its value.

Example:

Vsource Node_1 0 DC 12

To specify a constant voltage source, start with an I. Complete the name of your resistor by appending a string of your choice. Then specify the beginning and ending nodes. Then say that it is a DC source, and give its value.

Example:

Isource 0 Node_2 DC 2m

When you have finished specifying each element this way, end the input with the line

.end

No other options are needed for the Bias Point Analysis.

3b. Running the Simulation

First, the text file must be saved with the filename extension **.cir** . Type the extension in the dialog box and check to see that the file was really saved with this extension. The program is sort of dumb and will not accept text files with other extensions as input.

Then, close the file. Open it again, and it will be loaded as a PSpice input file. Now, you can run the simulation by pressing the blue arrow on the toolbar or through **Simulation -> Run**. Examine the output file through **View -> Output File**. Your results should be the same as they were for the circuit entered via the schematic program. **Print the input file and output file. This goes into your lab report.**

4. Lab Report

Turn in:

- By-hand nodal analysis of example circuit
- Printout of circuit schematic from schematic editor
- Printout of output text file obtained by simulating entered schematic
- Printout of input text file written from scratch
- Printout of output text file corresponding to input text file written from scratch