

**Department of Electrical and Electronic Engineering
Sonargaon University**

**EEE 202: Electronic Circuit Simulation
Laboratory**

Objective

Experiment # 01: Introduction to PSpice

The objective of this course is to learn the fundamentals of computer aided circuit simulation using PSpice. This experiment introduces the fundamentals of PSpice and concentrates on analysis of some DC circuits.

Introduction

SPICE is a powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behavior. This is of particular importance for integrated circuits. It was for this reason that SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies

Simulation Program for Integrated Circuits Emphasis

The EEE110 Lab uses the student version of the PSpice program extensively to simulate and predict the behavior of experimental circuits. Simulation is very important as it allows a potential circuit to be tested for errors before it is actually built, thus saving time and cost.

Some Facts and Rules about PSpice

- PSpice is not case sensitive. This means that names such as *Vbus*, *VBUS* and *vbus* are equivalent.
- All element names must be unique.
- The first line in the data file is used as a title. PSpice will ignore this line as circuit data. **Do not place any actual circuit information in the first line.**
- There must be a node designated "0" (Zero). This is the reference node (ground) against which all voltages are calculated.
- Each node must have at least two elements attached to it.
- The last line in any data file must be ".END"
- All lines that are not blank (except for the title line) must have a character in column 1, the leftmost position on the line.
 - Use "*" (an asterisk) in column 1 in order to create a comment line.
 - Use "+" (plus sign) in column 1 in order to continue the previous line (for very long lines).
 - Use "." (period) in column 1 followed by the rest of the "dot command" to pass special instructions to the program.
 - Use the designated letter for a part in column 1 followed by the rest of the name for that part (do not use spaces in the part name).
- Do not use spaces in the node name also. Use the underscore ("_") character to simulate spaces.
- Use "whitespace" (spaces or tabs) to separate data fields on a line.
- Use ";" (semicolon) to terminate data on a line if you wish to add commentary information on that same line.
- In PSpice circuits can be analyzed in two ways
 - Text based: using "Netlist" to code up circuits
 - Graphics based: using "Schematics" to draw circuits and simulate

Part A: Circuit Simulation by writing "Netlist"

A SPICE input file, called **source file ("filename.cir")**, consists of three parts.

1. *Data statements* : description of the components and the interconnections.
2. *Control statements* : tells SPICE what type of analysis to perform on the circuit.
3. *Output statements* : specifies what outputs are to be printed or plotted.

Although these statements may appear in any order, it is recommended that they be given in the above sequence. Two other statements are required, the **title statement** and the **end statement**. The title statement is the first line and can contain any information, while the end statement is always **.END**

```

TITLE STATEMENT
ELEMENT STATEMENTS
.
.
COMMAND (CONTROL) STATEMENTS
OUTPUT STATEMENTS
.END

```

1. Data Statements to Specify the Circuit Components and Topology

Here, we present the simplest circuit elements. Knowing how to model these ideal, linear circuit elements is an essential start to modeling more complex circuits.

1.1. Resistors

Although PSpice allows for sophisticated temperature-dependent resistor models, we will begin with the simple, constant-value resistor. The first letter of the name for a resistor must be "R."

Rname <+ node> <- node> value

The resistor is not an active device, so the polarity of its connection has no effect on the values of the voltages and currents reported in the solution. However, the current through a resistor is reported as that which flows from the node on the left to the node on the right in the source code line in which the resistor is entered.

Examples:

Rname	+node	-node	value	comment
Rabc	31	0	14k	; reported current from 31 to 0
Rabc	0	31	14k	; reported current changes sign
rshnt	12	15	99m	; 0.099 ohm resistor
Rbig	19	41	10MEG	; 10 meg-ohm resistor

Large and Small Numbers in PSpice

Unfortunately, PSpice cannot recognize Greek fonts or even upper vs. lower case. Thus our usual understanding and use of the standard metric prefixes has to be modified. The metric prefix designations used in PSpice are:

Number	Prefix	Common Name
10^{12}	T or t	tera
10^9	G or g	giga
10^6	MEG or meg	mega
10^3	K or k	kilo
10^{-3}	M or m	mini
10^{-6}	U or u	micro
10^{-9}	N or n	nano
10^{-12}	P or p	pico
10^{-15}	F or f	femto

An alternative to this type of notation, which is in fact, the default for PSpice output data, is "textual scientific notation." This notation is written by typing an "E" followed by a signed or unsigned integer indicating the power of ten. Some examples of this notation are shown below:

**656,000 = 6.56E5
-0.0000135 = -1.35E-5**

1.2. Ideal Independent Voltage Source

We begin with the DC version of the ideal independent voltage source. This is the default form of this class of part. The beginning letter of the part name for all versions of the ideal independent voltage source is "V."

Vname <+ node> <- node> type value

The name is followed by the positive node designation, then the negative node designation, then an optional tag: "DC" followed by the value of the voltage. The tag "DC" (or "dc") is optional because it is the default. Later, when we begin modeling AC circuits and voltage sources that produce pulses and other waveforms, we will be required to designate the type of source or it will default back to DC.

Examples:

Vname	+node	-node	type	value	comment
Va	4	2	DC	16.0V;	"V" after "16.0" is optional
vs	qe	qc	dc	24m ;	"QE" is +node & "qc" is -node
VWX	23	14		18k ;	"dc" not really needed
vwx	14	23	DC	-1.8E4 ;	same as above

One of the interesting uses of ideal independent voltage sources is that of an *ammeter*. We can take advantage of the fact that PSpice saves and reports the value of current entering the positive terminal of an independent voltage source. If we do not actually require a voltage source to be in the branch where we want to measure the current, we simply set the voltage source to a zero value. It still calculates the current in the branch.

Example:

```
vdep 15 27 DC 0V ; v-source used as ammeter
```

1.3. Ideal Independent Current Source

The name of an ideal independent current source begins with the letter "I".

Iname <+ node><- node> type value

Since the current source, is an active element, it matters greatly how it is connected. Designated current flows into the node written on the left, through the current source, out the node written on the right. As with the independent voltage source, the default type is DC.

Examples:

Iname	+node	-node	type	value	comment
Icap	11	0	DC	35m ;	35mA flows from node 11 to 0
ix	79	24		1.7 ;	"DC" not needed
I12	43	29	DC	1.5E-4 ;	
I12	29	43	dc	-150uA ;	same as above

2. Commands or Control Statements to Specify the Type of Analysis

PSpice allows various types of analysis. Each analysis is invoked by including its command statement.

2.1. .OP statement

This statement instructs Spice to compute the DC operating points:

- voltage at the nodes
- current in each voltage source
- operating point for each element

In PSpice it is usually not necessary to specify .OP as it gives you automatically the DC node voltages. However, for other types of analysis it is required to specify a certain analysis type, such as .TRAN or .AC for doing a transient or AC analysis.

2.2. .DC statement

This statement allows you to increment (sweep) an independent source over a certain range with a specified step. The format is as follows:

```
.DC SRCname START STOP STEP
```

in which SRC name is the name of the source you want to vary; START and STOP are the starting and ending value, respectively; and STEP is the size of the increment. This is a method of varying a parameter over a range of values so that we get a batch of cases solved all at once.

Example: **.DC V1 0 20 2**

Often, we do not actually want to run a sweep over many values of a parameter. We can circumvent the sweep by setting its Start and Stop values identical (and the Step is non-zero) so that it can only run one value.

Example: **.DC Vs 20 20 1**

Since the starting value equals the stopping value, the analysis will only run for one case, i.e., for Vs at 20 volts.

3. Output statements

These statements will instruct PSpice what output to generate. If you do not specify an output statement, PSpice will always calculate the DC operating points.

3.1. .Print Command

One of the many "dot commands" in PSpice is the .PRINT command. It has many uses, but we will concentrate here on using it for printing DC voltages and currents. The .PRINT command can be repeated as often as necessary in an analysis. You can list as many items on a line as you wish.

However, we must keep in mind that the .PRINT command was designed to work with a DC or an AC sweep. Usually, a DC sweep is made by changing the values of a source; although we will later learn to sweep over other circuit parameters.

Remember that in this experiment we will use DC sweep statement only to *enable* the .PRINT command. The .PRINT command will not work unless there is a sweep going on. Note: What you enter in the .DC statement *overrides* any voltage value you may have placed in the part listing for the source.

Printing DC Voltages

In addition to printing the node voltages you can print the voltage between any pair of nodes

```
.PRINT DC V(1) V(2) V(3); prints the node voltages  
.PRINT DC V(3,2) ; prints the voltage from node 3 to node 2
```

Printing DC Currents

To print currents, you type the letter "I" with the element name in parentheses. Note that the reported current is that which flows into the element from the node listed on the left in the *.CIR file, through the element, and out the node listed on the right in the *.CIR file. If you want to change the sign of the reported current in a resistor, then swap the two nodes for that resistor.

```
.PRINT DC I(Ra) ; prints the currents from + to - of Ra  
.PRINT DC I(Rb) I(Rc) ; prints the currents through Rb and Rc
```

Print Commands can be Combined

```
.PRINT DC V(1,2) I(Ra) ; voltage and current for Ra  
.PRINT DC V(2,0) I(Rb) ; V(2,0) same as V(2)
```

3.2. .Plot Command

The results from dc analysis can also be obtained in the form of plots. The maximum number of output variables is eight in any .Plot statement. More than one .Plot statement can be used to plot all the desired output variables.

Plot statements

```
.PLOT DC V(1) V(3,5) I(R1)  
.plot dc V(5) V(3,5) (0,10V) I(R1) (0, 50MA)
```

In the first statement, the y-axis is by default. In the second statement the range for voltages V(5) and V(3,5) is 0V to 10V, that for current I(R1) is 0 MA to 50 MA.

3.3. .Probe Command

Probe is a graphics post-processor/waveform analyzer for PSpice. The simulation results cannot be used directly by Probe. First, the results have to be processed by the .PROBE command, which writes the processed data on a file, PROBE.DAT, for use by Probe.

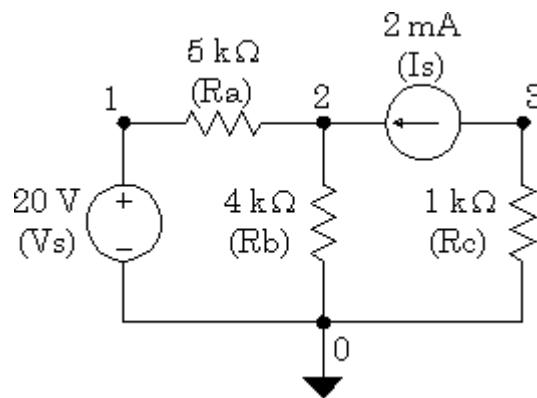
Probe statements

.PROBE

.PROBE V(5) V(4,3) I(R2)

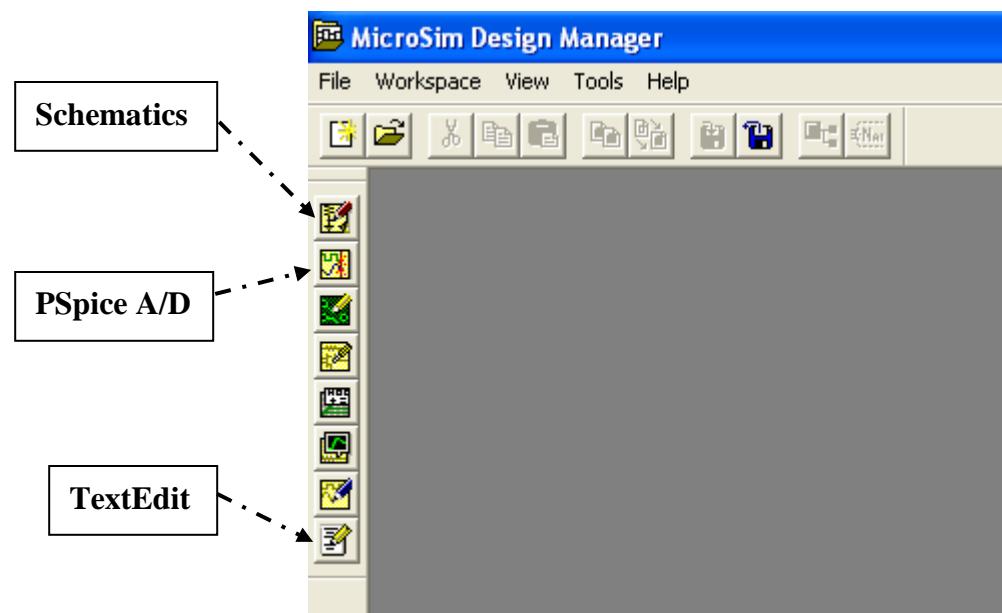
In the first form, where no output variable is specified, the .PROBE command writes all the node voltages and all the element currents into the PROBE.DAT file. In the second form, where the output variables are specified, PSpice writes only the specified output variables to the PROBE.DAT file. Once the results of the simulation are processed by the .PROBE command, the results are available for graphical displays and can be further manipulated through expressions.

Circuit Example 1



Steps:

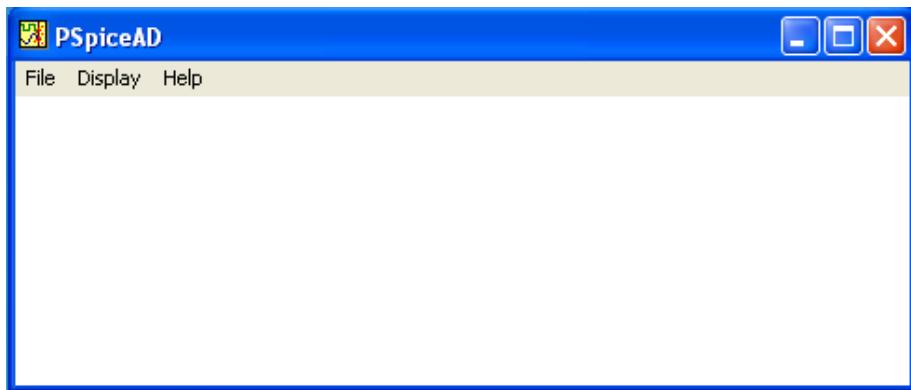
1. Open MicroSim Design Manager by following the path start>programs>MicroSim Eval 8 or as appropriate in your PC.



2. Click the TextEdit button to open MicroSim Text Editor. Here write down the following Netlist

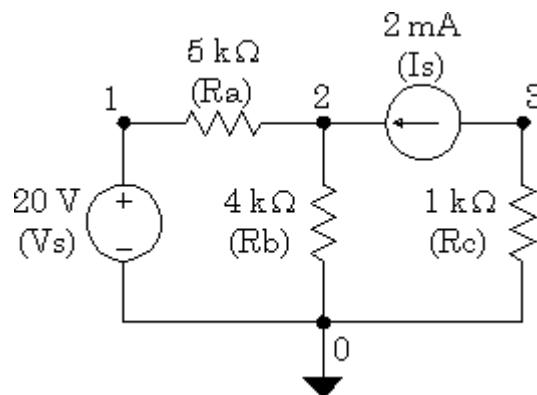
```
Example_1a Exmpl01a.CIR
Vs    1    0    DC    20.0V ; note the node placements
Ra    1    2    5.0k
Rb    2    0    4.0k
Rc    3    0    1.0k
Is    3    2    DC    2.0mA ; note the node placements
.END
```

3. Save the file with a name **Exmpl01a.cir**
4. Click PSpice A/D button to open the PSpice Analog/Digital Simulator



5. Open your **Exmpl01a.cir** file by clicking file and then open. If there is no error in the netlist then it will automatically complete the simulation and will give the message “**simulation completed successfully**”.
6. To examine the result click file and then **Examine Output**.

Now use .PRINT with Previous Example



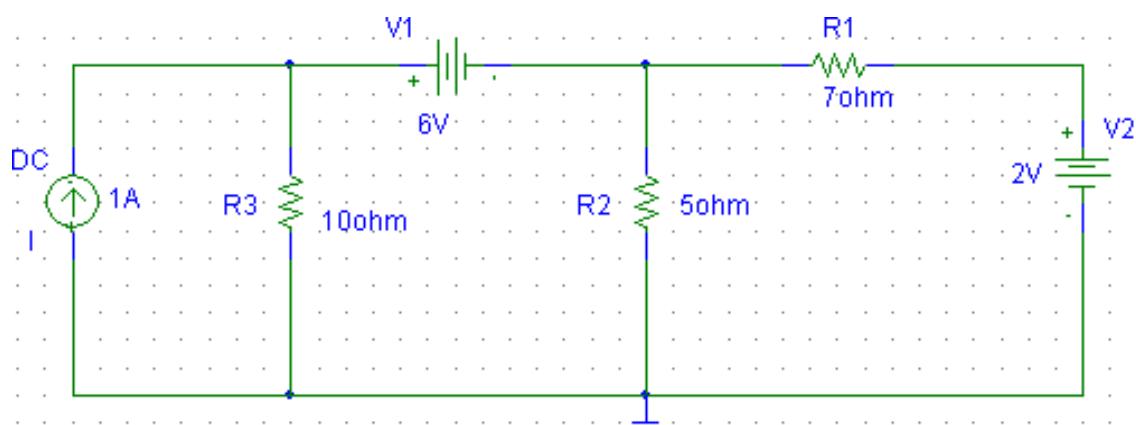
```

Example_1b Exmpl01b.CIR
Vs    1    0    DC    20.0V ; note the node placements
Ra    1    2    5.0k
Rb    2    0    4.0k
Rc    3    0    1.0k
Is    3    2    DC    2.0mA ; note the node placements
.DC Vs 20 20 1           ; this enables the .print commands
.PRINT DC V(1,2) I(Ra)
.PRINT DC V(2) I(Rb)
.PRINT DC V(3) I(Rc)
.END

```

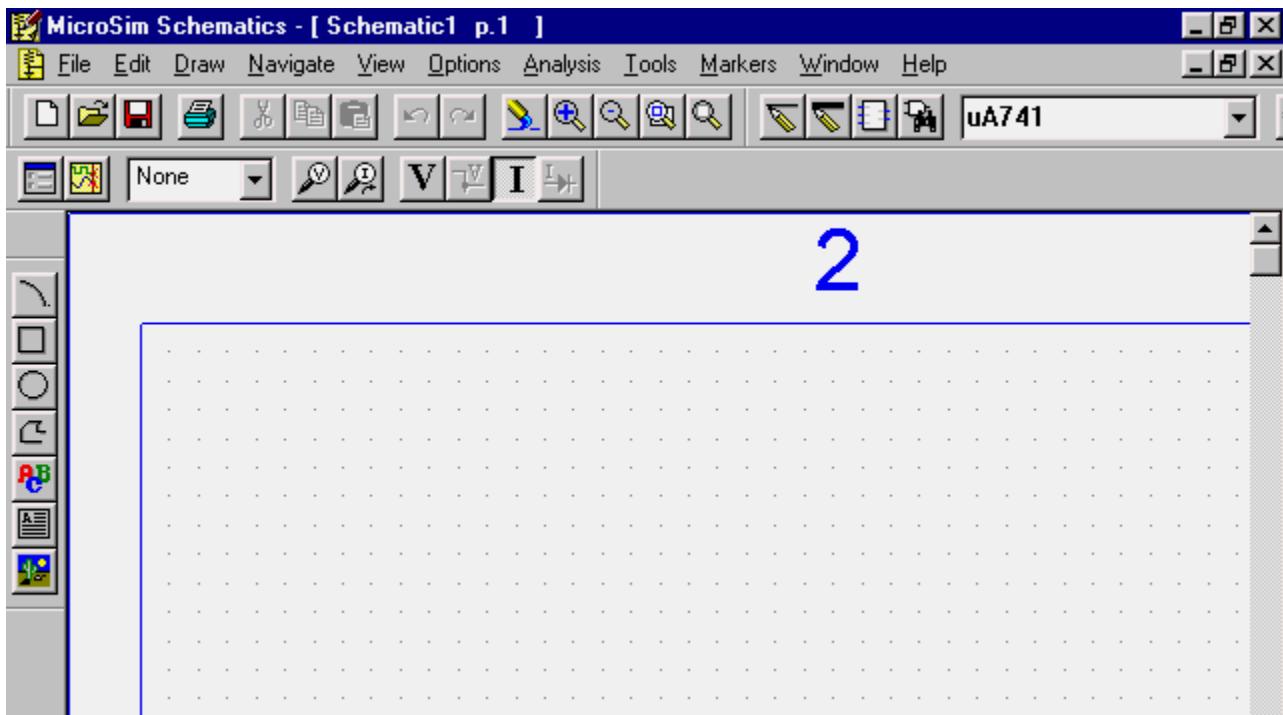
With a little bit of effort, we can get PSpice to do most of the work, most of the time. Note that using .PRINT has suppressed the default printing of all the node voltages. Be sure that you include everything you need in the .PRINT statements

Practice Problem 1



Part 2: Introduction to Schematic

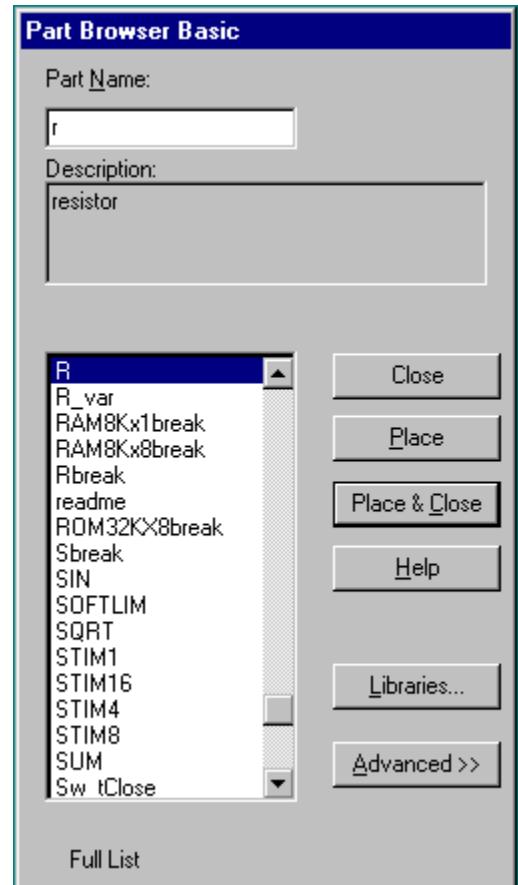
In this part you will learn to use the PSpice circuit simulation with the schematic capture front end, Schematics. Click **Schematic** button in the MicroSim Design Manager window to get the following schematic window.



A. Drawing the Circuit

1. Getting the Parts

- The first thing that you have to do is get some or all of the parts you need to simulate your circuit.
- This can be done by
 - Clicking on the 'get new parts' button , or
 - Pressing "Control+G", or
 - Going to "Draw" and selecting "Get New Part..."
- Once this box is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it
- Some common parts are:
 - r - resistor
 - GND_ANALOG or GND_EARTH -- this is very important, **you MUST have a ground in your circuit**
 - VAC and VDC – voltage sources
 - IAC and IDC – current sources
- Upon selecting your parts, click on the place button then click where you want it to be placed.
- Once you have all the parts you think you need, close that box. You can always open it again if you need more or different parts.



2. Placing the Parts

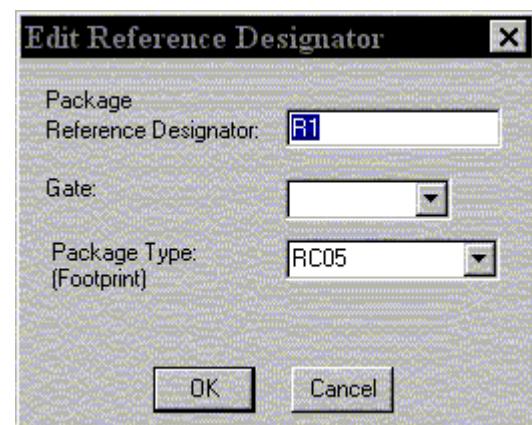
- You should have most of the parts that you need at this point.
- Now put them in the places that make the most sense (usually a rectangle works well for simple circuits). Just select the part and drag it where you want it.
- To rotate parts so that they will fit in your circuit nicely, click on the part and press "Ctrl+R" (or Edit "Rotate"). To flip them, press "Ctrl+F" (or Edit "Flip").
- If you have any parts left over, just select them and press "Delete".

3. Connecting the Circuit

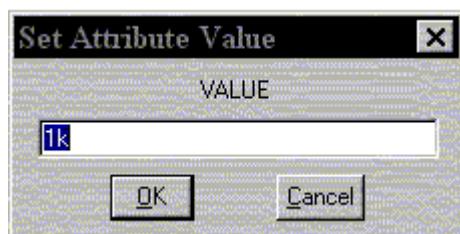
- Now you'll have to connect the parts with wires.
- Go up to the tool bar and
 - select "Draw Wire"  or
 - "Ctrl+W" or
 - go to "Draw" and select "Wire".
- With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.
- Repeat this until your circuit is completely wired.
- If you want to make a node (to make a wire go more than one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.
- To get rid of the pencil, right click or press "Esc".
- If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".
- If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).

4. Changing the Name of the Part

- To change the name, double click on the present name (C1, or R1 or whatever your part is), then a box will pop up (Edit Reference Designator). In the top window, you can type in the name you want the part to have.
- Please note that if you double click on the part or its value, a different box will appear.

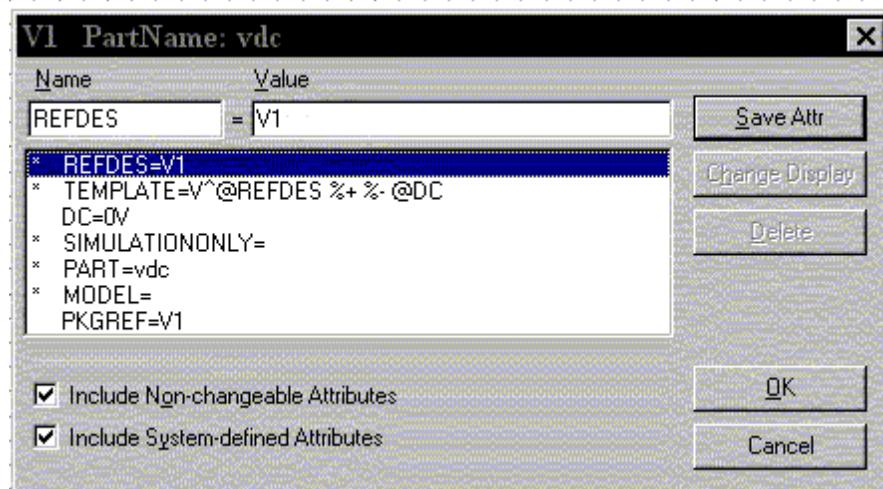


5. Changing the Value of the Part



- If you only want to change the value of the part, you can double click on the present value and a box called "Set Attribute Value" will appear. Type in the new value and press OK.

- If you double click on the part itself, you can select VALUE and change it in this box.



6. Making Sure You Have a GND

This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.

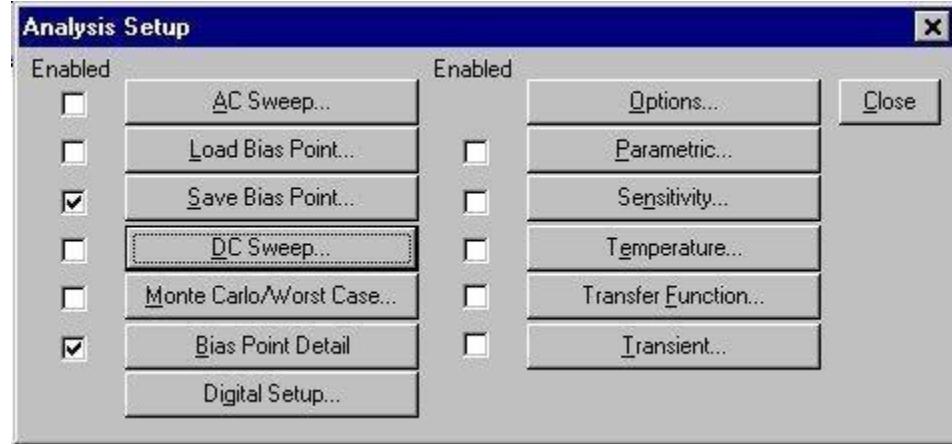
7. Voltage and Current Bubbles

- These are important if you want to measure the voltage at a point or the current going through that point.

- To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M)  or "Current Marker" . To get either of these, go to "Markers" and either "Voltage/Level Marker" or "Current Marker".

B. Analysis

- Open the analysis menu by clicking the  button. Enable the appropriate analysis options and then press close.



- Click on the Simulate button on the tool bar  (or Analysis, Simulate, or F11).

C. Probe

1. Before you do the Probe

- You have to have your circuit properly drawn and saved.
- There must not be any floating parts on your page (i.e. unattached devices).
- You should make sure that all parts have the values that you want.
- There are no extra wires.
- It is very important that you have a ground on your circuit.
- Make sure that you have done the Analysis Setup and that only the things you want are enabled.

2. To Start the Probe

- Click on the Simulate button on the tool bar  (or Analysis, Simulate, or F11).
- It will check to make sure you don't have any errors. If you do have errors, correct them.
- Then a new window will pop up. Here is where you can do your graphs.

3. Graphing

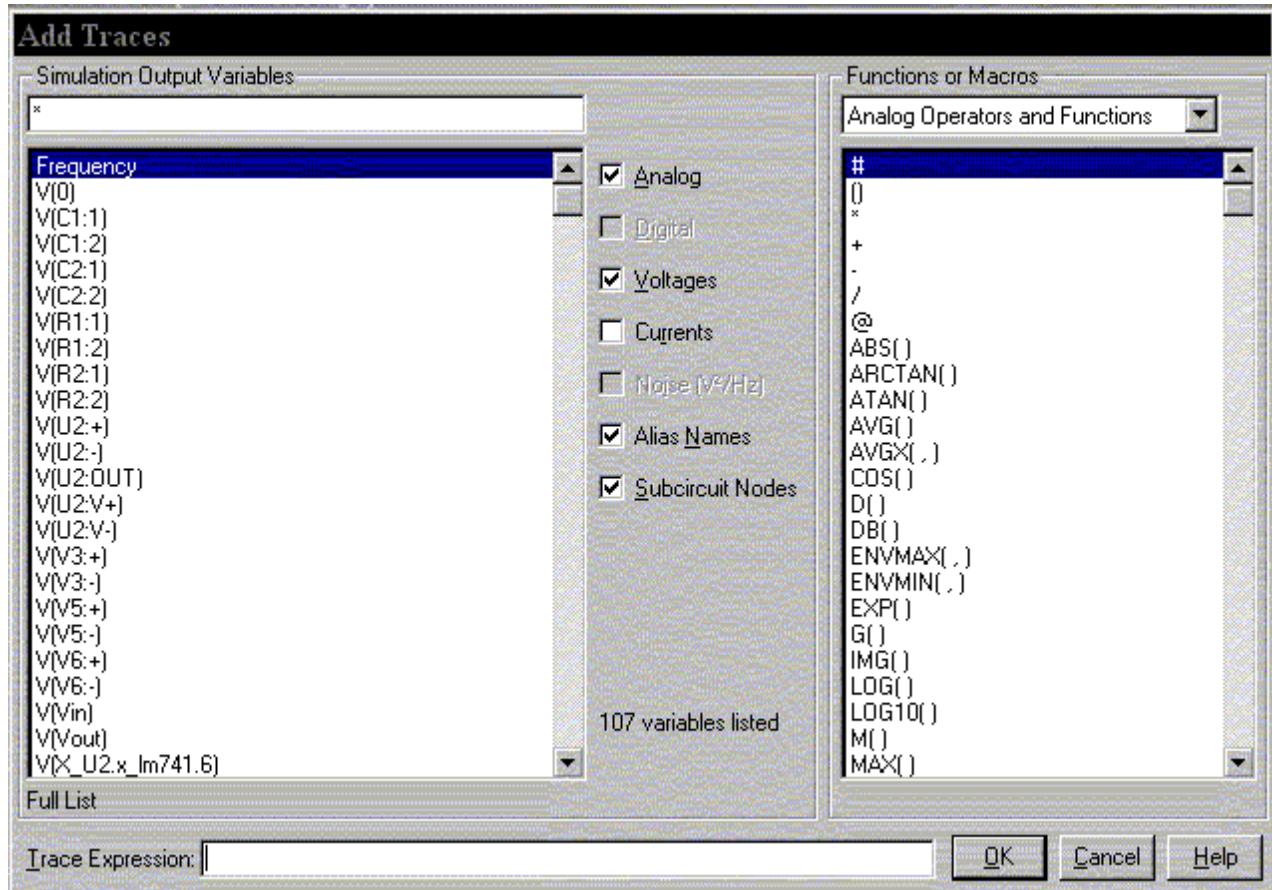
- If you don't have any errors, you should get a window with a black background to pop up
- If you did have errors, go To "View - Output File" to check the errors.

4. Adding/Deleting Traces

- PSpice will automatically put some traces in.
- To change them go to Trace - Add Trace or  on the toolbar. Then select all the traces you want.
- To delete traces, select them on the bottom of the graph and press "Delete".

5. Doing Math

- In Add Traces, there are functions that can be performed, these will add/subtract (or whatever you chose) the lines together.
- Select the first output then either on your keyboard or on the right side, click the function that you wish to perform.
- It is interesting to note that you can plot the phase of a value by using IP(xx), where xx is the name of the source you wish to see the phase for.



6. Finding Points

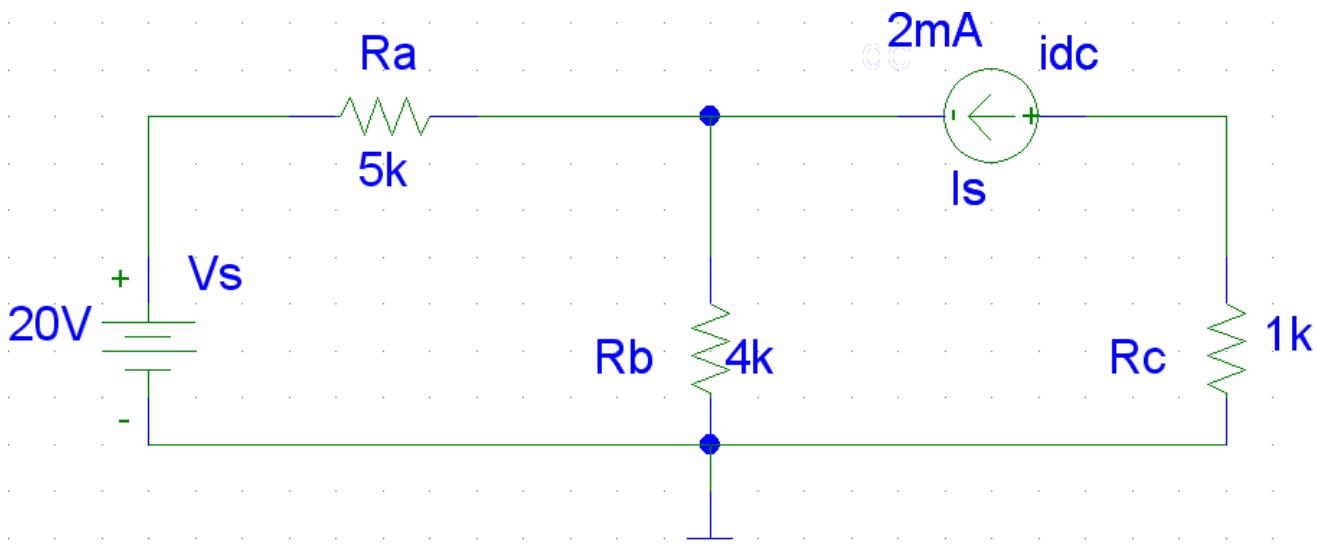
- There are Cursor buttons that allow you to find the maximum or minimum or just a point on the line. These are located on the toolbar (to the right).
- Select which curve you want to look at and then select "Toggle Cursor"
- Then you can find the max, min, the slope, or the relative max or min (is find relative max).

7. Saving

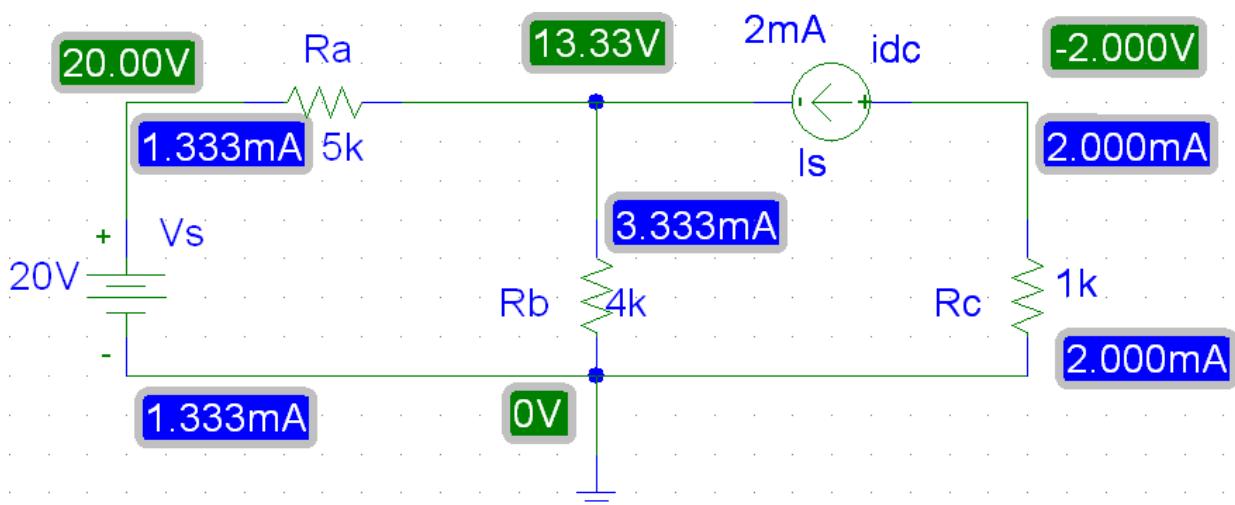
- To save your probe you need to go into the tools menu and click display, this will open up a menu which will allow you to name the probe file and choose where to save it. You can also open previously saved plots from here as well.

Circuit Example 2

- Using the steps explained above draw and simulate the following circuit. This is the same circuit that we used for our example 1.
- In Analysis Setup window only enable “Bias Point Detail” option.



- To examine the node voltages and current through each part go to Analysis menu in the Schematic window and then Display results on Schematic.



- You can also generate the netlist from the schematic by using the Create Netlist option from Analysis menu. To see the created netlist use Examine Netlist option from Analysis menu.

* Schematics Netlist *

```

R_Ra      $N_0002 $N_0001      5k
V_Vs      $N_0002 0            20V
R_Rb      0                 $N_0001      4k
I_Is      $N_0003 $N_0001 DC  2mA
R_Rc      0                 $N_0003      1k

```

Practice Problem 2

