Objective

PSPICE – Personal Simulation Program with Integrated Circuit Emphasis. It is a Cadence Electronic circuit analysis tool which helps to do the simulation of a circuit of analog and mixed-signal and eventually find out the key voltages and currents of that circuit. There are two ways to encode the inputs to the PSPICE: one is a typed "Netlist", another one is designing visual schematics which will be converted into a netlist.

Pretest – MCQ type

1. If a resistor R is connected between Nodes 1 and 2, voltage across R, v_R is given by
   a) $v_R = V(1) - V(2) = Ri$
   b) $i_R = V(1) - V(2) = Ri$
   c) $V(1) - V(2)$
2. If an inductor L is connected between nodes 3 and 4, the voltage across L,v_L and current through L,i is given by
   a) $v_L = V(3) - V(4) = R\ di/dt$
   b) $v_L = V(4) - V(3) = R\ di/dt$
   c) $i_L = V(4) - V(3) = R\ di/dt$
3. Sources in dc circuits are constant voltages or currents i.e. voltages and currents are __________
   a) Variant with time
   b) Invariant with time
   c) None of the above
4. The dc sweep is also known as ________________
   a) Transfer characteristics
   b) Transient analysis
   c) DC transfer characteristics
5. A transient analysis deals with the behavior of an electric circuit as a _____________
   a) Function of time
   b) Function of frequency
   c) Function of voltage
6. If a circuit contains an energy storage elements, a transient can also occur in a dc circuit after a sudden change due to __________________
   a) Change of time
   b) Switches opening or closing
   c) Switches opening

7. The transient sources are ____________ and can be independent or dependent.
   a) Time invariant
   b) Time variant or invariant
   c) Time variant
8. The method for calculating the transient analysis bias point differs from that of ___________
   a) Dc analysis bias point
   b) Ac analysis bias point
   c) Bias point
9. In transient bias point, the _________ values of the circuit nodes are taken into account in calculating the bias point.
   a) Temperature
   b) Voltage
   c) Initial
10. The resistance of the switch varies depending on the _______ across the switch.
   a) Voltage
   b) Current
   c) Temperature
11. The time-dependent close switch and time-dependent open switch are of the __________ type.
   a) Switch
   b) Independent switches
   c) Time-dependent switches
12. The __________ sources in a circuit are time variant.
   a) AC
   b) DC
   c) Independent
13. In the frequency response, if the frequency is increased by a factor of ____, it is called decade.
   a) 2
   b) 10
   c) Several times
14. In the frequency response, if the frequency is ________, it is called octave in frequency axis.
   a) Increased 10 times
   b) Increased several times
   c) Doubled
15. If the circuit contains ____________ devices, it is necessary to obtain the small-signal parameters of the elements before calculating the frequency response.

a) Non-linear devices
b) Linear devices
c) Storage devices

16. The value of the coefficient of coupling must be __________ than 0 and _____ than or equal to 1.
a) Less, greater
b) Greater, less
c) Greater, greater

17. Resistor and semiconductor devices generate noise and the level of the noise depends on the ___________
a) Time
b) Voltage
c) Frequency

18. Noise analysis is performed in conjunction with the _________ analysis.
a) AC
b) DC
c) Transient

19. In __________ analysis, the inductors are assumed to be short circuits and capacitors are assumed to be open circuits.
a) DC
b) Sensitivity
c) transient

20. The typical vi characteristics of a diode can be expressed by an equation known as ________
a) Diode equation
b) Characteristics equation
c) Shockley diode equation

21. The emission constant η value ranges from _________
a) 1 to 2.
b) 1 to 10
c) 1 to 100

22. The thermal voltage V_T is given by
a) $V_T = kT$
b) $V_T = kT/q$
c) $V_T = q$

23. The diode conducts fully if V_D is higher than the ___________ voltage
a) DC
b) AC
c) Threshold

24. The equation of the reverse bias region of the diode indicates that the reverse diode current is almost __________ and equal to ________

a) Constant, $-I_S$
b) Equal , $-I_S$
c) Zero , $-I_S$

25. In the breakdown region of a diode, the reverse voltage has a high magnitude, usually greater than ______

a) 10
b) 100
c) 1

26. Diodes are used in the electronic circuits for ____________

a) Image processing
b) Several applications
c) Signal processing

27. The value of diode resistance RD will be ________ if the diode is forward biased and _______ if it is reverse biased.

a) Low, high
b) High, low
c) High, high

28. The µA741 type of op-amp consists of _____ transistors.

a) 10
b) 12
c) 24

29. The________ is used to determine the number of equivalent parallel BJTs of a specified model.

a) Area factor
b) Parameters
c) DC model

30. VTO is ___ for enhancement type n-channel MOSFETs and ______ for depletion type n-channel MOSFETs.

a) Negative, positive
b) Positive, negative
c) Zero, one

<u>Prerequisites</u>

1. Knowledge of basic analog circuits which includes the components of
   - ✓ Independent and dependent voltage and current sources,
   - ✓ Resistor
   - ✓ Capacitor
   - ✓ Inductor
   - ✓ Mutual Inductors
   - ✓ Transmission lines
   - ✓ Operational Amplifiers
   - ✓ Switches
   - ✓ Diodes
   - ✓ Bipolar transistors
   - ✓ MOS transistors
   - ✓ JFET
   - ✓ MOSFET
   - ✓ Digital gates etc.,
2. Must have an experience with Basic Schematic entry with OrCAD capture

PSPICE

Introduction

SPICE (Simulation Program for Integrated Circuit Emphasis) is a general purpose analog circuit simulator and is used to verify circuit designs and also to predict the circuit behavior.

PSPICE is a PC version of SPICE. HSPICE is a version which runs on workstations and also on larger computers.

PSPICE has standard components (such as NAND, NOR, Flip-flops and other digital gates, Op-amps etc) in Analog and Digital libraries which makes it a useful tool for a wide range of Analog and Digital Applications. PSPICE is a product of the OrCAD Corporation and we are using the student version to analyze the circuits.

The student version of PSPICE is organized to be used for the college students and it's free of cost. But it has some limitations in circuit simulation.

How to do the download and installation?

1. Go to the browser and type "pspice free student version 9.2 download"
2. Go to the specific link "pspice link"
3. One particular page will open, there will show several links.
4. In that, it is specified as "where can I get my own copy of pspice using schematics (the student version of 9.1 is free)".
5. There will be a link is given under the above mentioned title.
6. Click on "Download locally".
7. The particular file will start download and size of that file is 27.3MB.
8. Go to downloads, select the path and unzip the setup file.
9. Now go to the unzipped folder of pspice and click on "setup.exe"
10. The student version of pspice will get installed.
11. Finish the step-up-step procedure to complete the installation.

Why we need this tool?

The Electronic circuit design is very much needed with accurate methods to evaluate the circuit performance. Due to the extensive intricacy of modern integrated circuits, it is essential to have this computer-aided circuit analysis tool which provides details about circuit performance. But it is not possible to get this analysis from the laboratory prototype measurements. So this tool will perform various analyses of electronic circuits like the operating point of transistors, a time-domain response, a small-signal frequency response and so on.

PSPICE contains models for common circuit elements of both passive and active. It is an adaptable tool and is widely used in both industries and colleges.

Limitations of PSPICE student version:

There are some limitations in Pspice student version. They are,

1. The circuit can have maximum of 64 nodes, 10 transistors, 65 primitive digital devices, 10 transmission lines in total, 4 pair wise couple transmission lines.
2. The library includes 39 analog and 134 digital parts.
3. The device characterization in the Pspice model editor is limited to diodes.
4. Stimulus generation in the Pspice stimulus editor is limited to Sine wave (analog) and Clock (digital).
5. We cannot create Common Simulation Data Format (CSDF) files.
6. We can only display simulated data which is performed with the student version of the simulator.
7. We can place a maximum of 50 parts on a schematic design for the schematics.
8. It can be draw only on size A sheets.
9. We cannot save a library that contains more than 15 parts.
10. The Pspice version requires 512 kilobytes of memory (RAM) to run.
11. It does not support an iterative method of solution.

Types of Analysis:

The Pspice involved with several analysis,

1. Non-linear DC Analysis
2. Non-linear Transient Analysis
3. Linear AC Analysis
4. Noise Analysis
5. Sensitivity Analysis
6. Distortion Analysis
7. Fourier Analysis
8. Monte Carlo Analysis

Types of Circuit Components in PSPICE:

There are different circuit components are available in PSPICE analysis,

- ✓ Independent and dependent voltage and current sources,
- ✓ Resistor
- ✓ Capacitor
- ✓ Inductor

- ✓ Mutual Inductors
- ✓ Transmission lines
- ✓ Operational Amplifiers
- ✓ Switches
- ✓ Diodes
- ✓ Bipolar transistors
- ✓ MOS transistors
- ✓ JFET
- ✓ MOSFET
- ✓ Digital gates

File types:

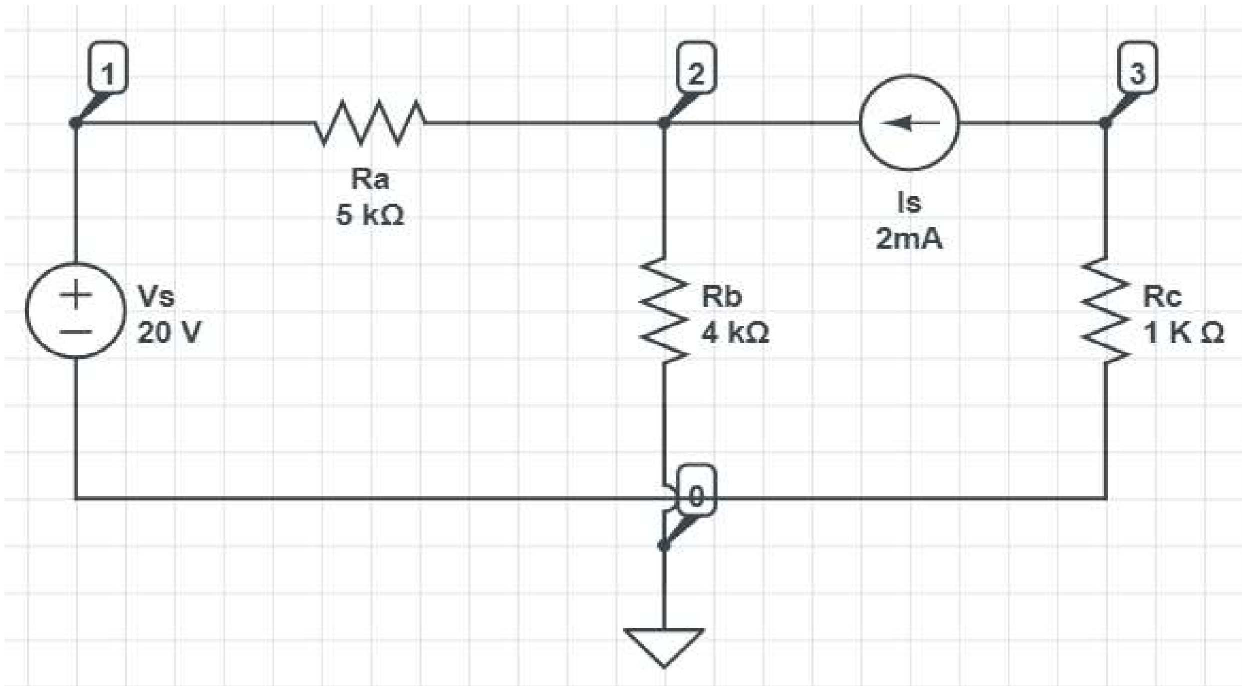
There are some files which are involved in taking the inputs and getting out with the outputs in Pspice.

In Pspice, it is two ways to encode the inputs – **Circuit file (Netlist) and Schematic file**

Type of the file	Name of the file
Input file	**.CIR file (Circuit) and .SCH file (Schematic)**
Output file	**.OUT file**
A file is by default file a binary file to see the data	**.DAT file**
A file where the details of complex parts are saved	**.LIB file**
Alias files	**.ALS file**
Network connection files	**.NET file**

Examples of files generated in a Project of PSPICE:

Example:



The above is the example circuit for a normal DC circuit which consists of a DC source and an AC source. How to write the Circuit file code for this?

Let's proceed with the circuit file code for the above circuit.

```
VS  1  0  DC   20V

IS   3  2   AC   2MA

RA   1   2   5K

RB   2   0   4K

RC   3   0    1K

.END
```

Title Line

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE

(1) 20.0000 (2) 13.3330 (3) -2.0000 --→ Results

VOLTAGE SOURCE CURRENTS

NAME CURRENT

VS -1.333E-03 --→ current entering node 1 of VS

TOTAL POWER DISSIPATION 2.67E-02 WATTS

JOB CONCLUDED

TOTAL JOB TIME 0.26

Some of the library files available in the Evaluation version of PSPICE

1. **abm.slb** for special functions like square root and multipliers
2. **analog.slb** for analog components like resistors and capacitors
3. **breakout.slb** for pots
4. **connect.slb,.plb** for connectors
5. **eval.slb, .plb** for semiconductors, digital devices and switches
6. **port.slb** for grounds, high/low digital ports
7. **special.slb** for ammeters, viewpoints
8. **source.slb** for various analog and digital sources.

Different parts which are available in PSPICE

Part is a building block which may represent

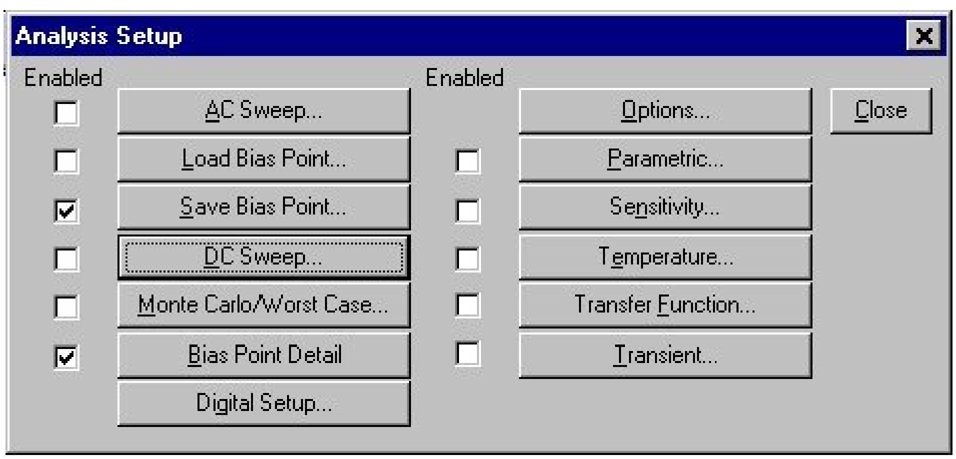
- one or more physical element
- function
- simulation model
- When a part is placed for first time its entry is done in "Design cache".

Things to remember:

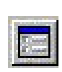
1. PSPICE is not case sensitive
2. All element names must be unique
3. There must be node designated "0" (zero). This is the reference node against where all voltages are calculated.
4. If any changes are made in circuit, make sure you create netlist again before simulating it.

Symbol	Factor
F/f	1.00E-15
P/p	1.00E-12
N/n	1.00E-09
U/u	1.00E-06
M/m	1.00E-03
K/k	1.00E+03
MEG/meg	1.00E+06
G/g	1.00E+09
T/t	1.00E+12

<u>PSPICE Analysis setup</u>



The above figure is the analysis setup window in PSPICE. Each analysis is invoked by including its command statement.

To open the analysis menu click on the  button.

A.AC Sweep

- The AC sweep allows you to plot magnitude versus frequency for different inputs in your circuit.
  In the AC sweep menu you have the choice of three types of analysis:
  - **Linear,**
  - **Octave and**
  - **Decade**.
- These three choices describe the X-axis scaling which will be produced in probe. For example, if you choose decade then a sample of your X-axis might be 10Hz, 1 kHz, 100 kHz, 10 MHz, etc.... Therefore if you want to see how your circuit reacts over a very large range of frequencies choose the decade option.
- You now have to specify at how many points you want PSpice to calculate frequencies, and what the start and end frequency will be. That is, over what range of frequencies do you want to simulate your circuit?
- In the AC sweep you also have the option of **Noise enable** in which PSpice will simulate noise for you either on the output or the input of the circuit. These noise calculations are performed at each frequency step and can be plotted in probe.
- To use input noise you need to tell PSpice where you consider the 'input' in your circuit to be, for example, if your voltage source is labeled 'V1'.
- Finally you need to specify in what interval you want the noise to be calculated (note: the default interval for spice is zero, i.e.: no noise will be calculated).

B. DC Sweep

- The DC sweep allows you to do various different sweeps of your circuit to see how it responds to various conditions.
- For all the possible sweeps,
  - voltage,
  - current,
  - temperature, and
  - parameter and global

  You need to specify a start value, an end value, and the number of points you wish to calculate.

  For example you can sweep your circuit over a voltage range from 0 to 12 volts. The main two sweeps that will be most important to us at this stage are the voltage sweep and the current sweep. For these two, you need to indicate to PSpice what component you wish to sweep, for example V1 or V2.

- Another excellent feature of the DC sweep in PSpice, is the ability to do a **nested sweep**.
- A nested sweep allows you to run two simultaneous sweeps to see how changes in two different DC sources will affect your circuit.
- Once you've filled in the main sweep menu, click on the nested sweep button and choose the second type of source to sweep and name it, also specifying the start and end values.

(Note: In some versions of PSpice you need to click on **enable nested sweep**). Again you can choose Linear, Octave or Decade, but also you can indicate your own list of values, example: 1V 10V 20V. **DO NOT** separate the values with commas.

C. Bias Point Detail

- This is a simple, but incredibly useful sweep. It will not launch Probe and so give you nothing to plot. But by clicking on **enable bias current display** or **enable bias voltage display**, this will indicate the voltage and current at certain points within the circuit.

D. Parametric

- Parametric analysis allows you to run another type of analysis (transient, sweeps) while using a range of component values using the **global parameter** setting. The best way to demonstrate this is with an example, we will use a resistor, but any other standard part would work just as well (capacitor, inductor).

- First, double-click the value label of the resistor that is to be varied. This will open a "Set Attribute Value" dialog box. Enter the name **{RVAL}** (including the curly braces) in place of the component value. This indicates to PSpice that the value of the resistor is a global parameter called RVAL. In order to define the RVAL parameter in is necessary to place a global parameter list somewhere on the schematic page. To do this, choose "**Get New Part**" from the menu and select the part named **param.**

- Place the box anywhere on the schematic page. Now double-click on the word **PARAMETERS** in the box title to bring up the parameter dialog box. Set the NAME1= value to **RVAL** (no curly braces) and the VALUE1= value to the nominal resistance value. This nominal value is required, but it is only used if the DC bias point detail is computed. Otherwise, the value is ignored by PSpice.

- Finally, go to the "Analysis Setup" menu and enable "Parametric" analysis. Open the Parametric setup dialog box and enter the sweep parameters: Name: **RVAL** Swept variable type: Global Parameter. Make sure the other analysis type(s) are selected in the analysis setup menu (transient, sweeps). PSpice will now automatically perform the simulation over and over, using a new value for **RVAL** during each run.

E. Sensitivity

- Sensitivity causes a DC sensitivity analysis to be performed in which one or more output variables may be specified.
- Device sensitivities are provided for the following device types only:
  - resistors,
  - independent voltage and current sources,
  - voltage and current-controlled switches,
  - diodes, and
  - bipolar transistors.

- You would use the sensitivity setting for discovering the maximum range of circuit performance and the causes of extreme operation. These techniques are used to identify effective changes to improve the quality of circuit operation (for example, which components need to have tight tolerance and which can be lower quality and less expensive).
- This isn't as important for us in the lab, but some day when you are constructing real circuits that need to function under various conditions this will be useful.

F. Temperature

- The temperature option allows you to specify a temperature, or a list of temperatures (do not include commas between temperature values) for which PSpice will simulate your circuit.
- For a list of temperatures that simulation is done for each specified temperature.

G. Digital Setup

- This paragraph will only indicate the features of the digital setup on the analysis menu, see below for a more complete description.
- In addition to letting you simulate analog circuits, PSpice provides a number of digital parts that can be used in a homogeneous digital circuit, or a heterogeneous analog/digital combination. The digital analysis option allows you to specify the timing of your circuit, by running the gates at their minimum, maximum and typical values. A superb feature allows you to test the worst case timing of your circuit to see how it will operate under these extreme conditions. You also have the option of setting the value of any flip flops you have in your circuit to predefined states which is good to simulate any startup conditions for finite state machines that you are simulating.

H. Transient

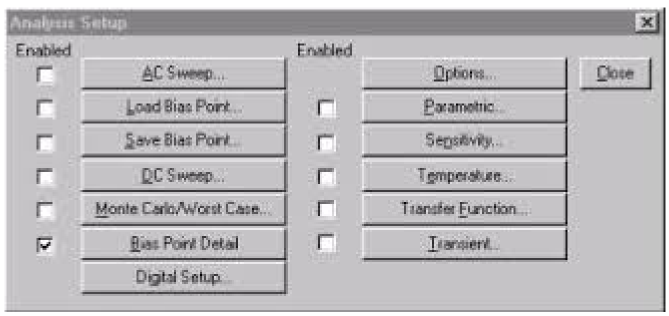
- The transient analysis is probably the most important analysis you can run in PSpice, and it computes various values of your circuit over time. Two very important parameters in the transient analysis are:
    - **print step**
    - **final time**.
- The ratio of **final time: print step** determines how many calculations PSpice must make to plot a wave form. PSpice always defaults the start time to zero seconds and going until it reaches the user defined final time. It is incredibly important that you think about what print step you should use before running the simulation, if you make the print step too small the probe screen will be cluttered with unnecessary points making it hard to read, and taking extreme amounts of time for PSpice to calculate. However, at the opposite side of that coin is the problem that if you set the print step too high you might miss important phenomenon that are occurring over very short periods of time in the circuit. Therefore play with step time to see what works best for your circuit.

- You can set a step ceiling which will limit the size of each interval, thus increasing calculation speed. Another handy feature is the Fourier analysis, which allows you to specify your fundamental frequency and the number of harmonics you wish to see on the plot. PSpice defaults to the 9th harmonic unless you specify otherwise, but this still will allow you to decompose a square wave to see it's components with sufficient detail.

DC Circuit Analysis

Draw the specified DC circuit in the Schematic window by dragging the components which are available in the components icon. Pick out the component one by one which are really needed to draw the circuit and connect the components using the wire option and save the file as .SCH. Finally the circuit has to go the analysis setup to complete the analysis of the DC circuit.

In the analysis setup window, there is an option called DC bias point which gives the voltage values of all the important points in the DC circuit.



Circuit File

In the circuit file option, we have to go along with DC circuit which may consists of Resistor, Capacitor, Inductor and also the various DC and AC sources.

How to model a Resistor?

The symbol for a resistor is R and it start with R. The resistor’s name and its nominal value can be changed. Also, a tolerance value can be assigned to it.

The netlist takes the general form of

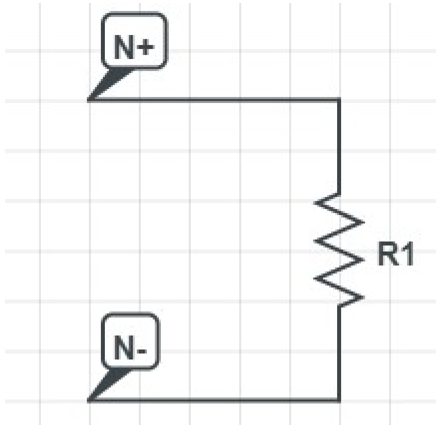
R <name> N+ N- RNAME RVALUE

N+ --→ Positive Node

N- --→ Negative Node

RNAME --→ model name that defines the parameters of the resistor

RVALUE --→ nominal value of the resistance.



The resistance parameters options are given by

Example:

R1 6 5 10K

This example shows that the resistance name is R1, which is connected between the nodes 6 and 5 and has the value of 10KOHM.

Modeling of Elements

The models are necessary to take into account the parameter variations, ie. the value of the resistor depends on the operating temperature. A model that specifies a set of parameters for an element is specified by the .MODEL command.

The general form of the model statement is

.MODEL MNAME TYPE (P1=A1 P2=A2 P3=A3 . . . PN=AN)

MNAME -→ name of the model

P1, P2 → element parameters

A1, A2 → element parameter values

TYPE → type name of the elements

Example:

.MODEL RMOD RES (R=1.1 TCE=0.001)

Operating temperature

The operating temperature of an analysis can be set to any desired value by the .TEMP command.

The general form of the statement is

.TEMP <(one or more temperature) values>

The temperature is in degrees Celsius. If more than one temperature is specified, then the analysis is performed for each temperature.

Example

.TEMP 50

Independent DC Sources

The independent sources can be time invariant or time variant and can be currents or voltages.

1) Independent DC voltage source

V<name> N+ N- [DC <value>]

Example

V1 15 0 6V

2) Independent DC current source

I<name> N+ N- [DC <value>]

Example

I1 15 0 2.5MA

Schematic Independent sources

The PSPICE source library source.lib is available in the PSPICE version. DC voltage and current sources are also available. The user can change the values of the sources.

Dependent sources

There are four types of dependent sources

Voltage-controlled voltage source

Voltage-controlled current source

Current-controlled voltage source

Current-controlled current source

Voltage-controlled voltage source

E<name> N+ N- NC+ NC- <(voltage gain) value>

Voltage-controlled current source

G<name> N+ N- NC+ NC- <(transconductance) value>

Current-controlled voltage source

H<name> N+ N- NC+ NC- <(transresistance) value>

Current-controlled current source

F<name> N+ N- NC+ NC- <(current gain) value>

Schematic Dependent Sources

The PSPICE analog library analog.slb is available in the PSPICE version.

DC Output variables

PSPICE has some unique features for printing or plotting output voltages or currents. The output variables can be divided into two types: voltage output and current output.

Types of Output:

The commands that are available to get output from the results of simulations are as follows:

.PRINT → print

.PLOT → plot

.PROBE → probe

.WIDTH → width

Types of DC Analysis

The commands that are commonly used for dc analysis are

.OP → DC operating point

.TF → Small-signal transfer function

.DC → DC Sweep

.PARAM → DC Parametric Sweep

.OP(Operating point)

Electrical and electronic circuits contain nonlinear devices whose parameters are depends on the Operating point. The operating point is also known as a bias point or quiescent point. The .OP command controls the output of the bias point. If the .OP command is omitted, PSPICE prints only a list of the node voltages. If the .OP command is present, PSPICE prints the currents and power dissipations of all the voltage sources.

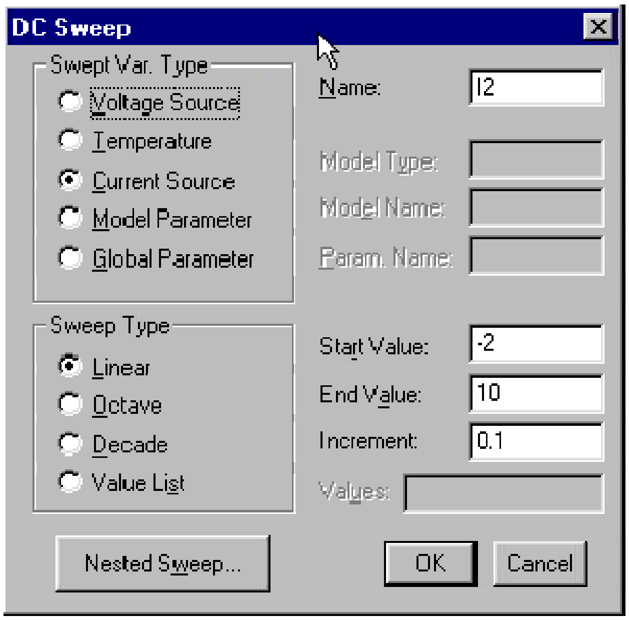
.TF(Small-signal transfer function)

The small signal transfer function capability of PSPICE can be used to compute the small-signal DC gain, the input resistance and the output resistance of the circuit. PSPICE calculates the small-signal dc transfer function by linearizing the circuit around the operating point. The .TF command calculates the parameters of Thevenin's (Or Norton's) equivalent circuit for the circuit file. It automatically prints the output and does not require .PRINT,.PLOT or .PROBE statements.

.DC(DC Sweep)

The dc sweep is also known as the dc transfer characteristics. In the analysis setup, there is a very important option which is used to vary the voltage along with time is called the DC Sweep.

If you click on the DC sweep in the analysis setup window, there is a specific DC Sweep window will open.



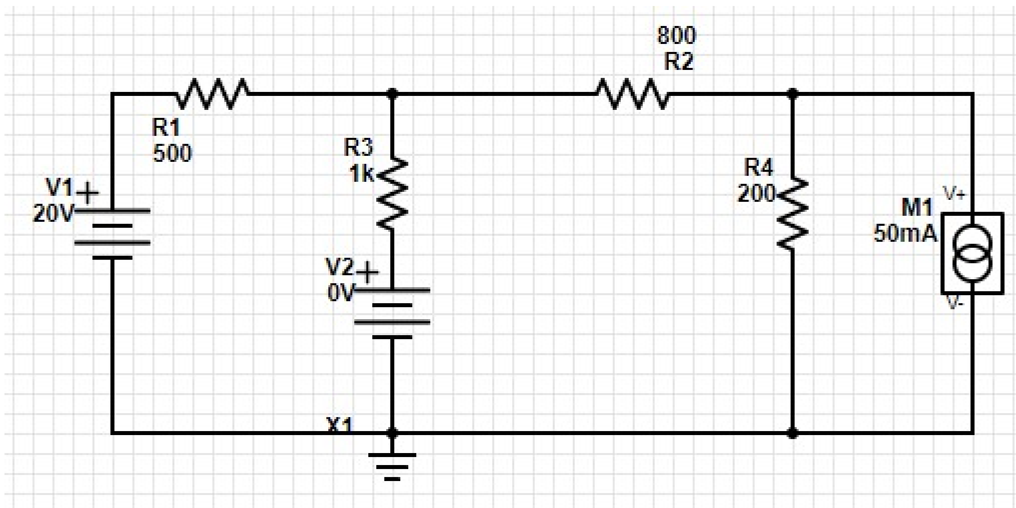
In the above window, there is an option called **Swept Var**. **Type**. In that, there are multiple options available this specifies Voltage source, Temperature, Current Source, Model Parameter, Global Parameter. Then we have the option called **Sweep type** this specifies the multiple types of sweeps which are available in PSPICE in such a way that Linear, Octave, Decade and Value List.

The next option which is available in the above window is **Name** which specifies the Name of the Source. We have also the Start Value, End Value and Incremented Value of the DC. Finally we have the Nested Sweep option is also available in DC Sweep.

.PARAM (DC Parametric sweep)

PSPICE allows one to vary a parameter and to evaluate its effects on the dc analysis. The parameters could be a circuit element such as R, L, C and their model parameters.

Example



For simplicity use the M1 as Is.

The circuit file follows as:

```
VS  1   0  DC  20V
 IS   0   4  DC  50MA
 R1  1   2   RMOD1    500
 R2   2   5  RMOD1    800
.MODEL RMOD1 RES(R = 1.05)
R3   2   3    RMOD2   1KOHM
R4   4   0    RMOD2    200
.MODEL   RMOD2  RES(R = .9)
 V2    3    0    DC   0V
.OP
.END
```

The results that are obtained by printing the contents of output file:

NODE VOLTAGE	NODE VOLTAGE	NODE VOLTAGE	NODE VOLTAGE
(1)20.000	(2) 11.7410	(3) 0.0000	(4) 9.4836
(5) 9.4836			

VOLTAGE SOURCE CURRENTS

Name	Current	Calculated
VS	-1.573E-02	-15.73 mA
V2	1.305E-02	13.05 mA

TOTAL POWER DISSIPATION 3.15E-01 WATTS

POST MCQ

1. The Acronym of SPICE stands for ________

a) Simulation Program with Integrated Circuit Emphasis

b) Simulation Program with International Circuit Emphasis

c) Synthesis program with International circuit Emphasis.

2. The PC version required ___________ of memory (RAM) to run.

a) 1KB

b) 512KB

c)2KB

3. The student version of PSPICE is restricted to circuits with maximum of ___________ transistors.

a) 20

b) 100

c) 10

4. ____________ analysis is not available in PSPICE.

a) Sensitivity analysis

b) Distortion analysis

c) Parametric analysis

5. The scale suffices are all ___________ but pspice allows _________

a) lowercase, uppercase

b) uppercase, lowercase

c) lowercase, lowercase

6. The details of all node voltages as well as current and power dissipation of all voltage sources can be sent to the output file by the ______ command.

a) .OP

b) .TRAN

c) .DC

7. The tool which is used for the graphical postprocessor for viewing the simulation results are ______________

a) Capture

b) PSPICE A/D

c) Probe

8. The DC Sweep is also known as _______

a) Small-signal function

b) DC Operating point

c) DC transfer characteristics

9. PSPICE allows one to vary a parameter and to evaluate its effects on the dc analysis _____

a) .OP

b) .PARAM

c) .TRAN

10. The PSPICE design center has a __________ for circuit drawings that uses part symbols to represent devices and wire symbols for connections.

a) Schematic editor

b) Circuit editor

c) Schematic input file

11. The maximum number of output variables is _________ in any .PRINT statement.

a) Five

b) Three

c) Eight

12. If the .OP command is omitted, PSPICE prints only a list of the __________

a) node voltages

b) node currents

c) voltages & currents

13. An .OP command with a .TRAN command namely, .TRAN/.OP will print the __________ during transient analysis.

a) Small signal analysis

b) DC analysis

c) AC analysis

14. The library file ________ contains the list of device library files and the device model statements that are available with the student's version.

a) DVIL.LIB

b) AVAL.LIB

c) EVAL.LIB

15. General purpose open source analog electronic simulator is known as

a) MATLAB

b) SPICE

c) C++

16. In SPICE, non-linear quiescent point calculation is termed as

a) DC analysis

b) AC analysis

c) Noise analysis

17. First commercial version of SPICE is

a) PSPICE

b) ISPICE

c) TSPICE

18. Components which can often be estimated more occurring using SPICE simulator is

a) Parasitic components

b) Non-parasitic components

c) Resistive components

19. PSPICE does not support an iterative method of solution

a) Yes

b) No

20. The statement that R1 has a value of 500Ohm and is connected between nodes 1 and 2 is

a) R1 500

b) R1   2   1   500

c) R1   1   2   500

21. _______ defines the circuit elements and the set of model parameters.

a) Output description

b) Circuit description

c) Analysis description

22. A comment line may be included anywhere, preceded by an ________

a) ;

b) +

c) *

23. The statement which is to calculate and print the voltage for VS = 10V, 20V & 30V

a) .DC VS 10V 30V 10V

b) .AC VS 10V 30V 10V

c) .DC IS   10V   30V   10V

24. The .PROBE command requires a math coprocessor for the ________ of PSPICE.

a) Student version

b) Professional version

c) A/D version

25. Electrical and electronic circuits contain nonlinear devices whose parameters depend on the

_____________ .

a) Temperature

b) Operating Point

c) DC parametric sweep.

Conclusion

Finally it is concluded that the PSPICE DC simulation is used for the extensive research and design of the DC simulated circuits for the analysis of transient, parameter scanning and optimization of the circuit. It is very easy and simple software for simulations and analysis of the circuits.

References

1. Introduction to PSPICE using OrCAD for circuits and Electronics by Muhammad H. Rashid.
2. http://tuttle.merc.iastate.edu/ee201/spice/pspice_DC.pdf
3. https://www.engr.colostate.edu/ECE562/Pspicetutorial.pdf
4. https://www.seas.upenn.edu/~jan/spice/PSpice_UserguideOrCAD.pdf
5. https://www.csun.edu/~skatz/pspice_tutorials/pspice_tutorial_1.pdf
6. https://engineering.purdue.edu/~ee255d3/files/PSPICEtutorial.pdf
7. https://www.circuitlab.com/