CHAPTER – III

COMPUTER BASED CIRCUIT SIMULATION

CHAPTER - III
COMPUTER- BASED CIRCUIT SIMULATION

3.1)   INTRODUCTION

3.2)   FORMAT OF CIRCUIT FILES

3.3)   FORMAT OF OUTPUT FILES

3.4)   TYPES OF ANALYSIS

A] DC ANALYSIS
   i) Resistors
   ii) Modeling of elements
   iii) Operating temperature
   iv) Independent DC source
   v) Dependent source
   vi) Types of output
   vii) Types of DC analysis

B] TRANSIENT ANALYSIS
   i) Capacitor
   ii) Inductor
   iii) Modeling of transient sources
   iv) Transient sources
   v) Transient response

C] AC circuit Analysis

D] Multiple Analysis

E] Fourier Analysis

F] Noise Analysis

G] Sensitivity Analysis

H] Parametric Analysis

3.5]   OP. AMP REPRESENTATION IN PSPICE
   i) DC linear model
   ii) AC linear model
   iii) Non linear macromodel

CHAPTER - III

COMPUTER- BASED CIRCUIT SIMULATION

3.1) INTRODUCTION :-

Simulation allows design to be analyzed and modified without having to go to the effort, expense and time of building a prototype. Simulation can also be used to perform analysis that are not always possible and desirable on a physical design such as worst case component tolerance sensitivity analyses & simulating faulty circuits. Simulation packages are usually classified into four types analog-digital, mixed analog-digital (A/D) and block diagram simulator. Pspice is a mixed type of simulator.

The simulation program with integrated circuit emphasis (SPICE) is powerful computer package used to analyse electrical & electronic circuits. A circuit is described to a computer by using a file called the circuit file, which is normally typed by keyboard. The circuit file contains the circuit details of components and elements, the information about the source & the commands for calculation and output results. The circuit file is the input file to the SPICE program, which after executing the commands, produces the result in another file called output file.

3.2) FORMAT OF CIRCUIT FILE :-

A circuit must be specified in terms of element names, element values, nodes, variable parameters and sources. A circuit file that can be read by Pspice may be divided into five parts.

1. The Title which describe the type of the circuit or any comments .

2. The circuit description which defines the circuit elements and the set of model parameters.

3. The analysis description, which defines the type of analysis.

4. The output description, which defines the way the output is to be presented.

5. The end of the program (the .end command)

The format for circuit file is as follows

Title

Circuit description

Analysis description

Output description

End (end of the file statement)

i) The first line is the title line and it may contain any type of text .

ii) The last line must be .END command.

iii) The order of remaining lines is not important and does not affect the result of simulation.

iv) The circuit description, which defines the circuit elements and the set of model parameters.

v) Analysis description which defines the type of analysis.

vi) The output description, which defines the format for the output that, is to be presented.

vii) If the PSpice statement is more than one line, the statement can continue on the next line by placing + sign ,the first column of the next line.

viii) Comment line may be included anywhere, preceded by an * sign within a statement a comment is preceded by ;

ix) PSpice statements or comments can be in either upper or lower case.

3.3) FORMAT OF OUTPUT FILES :-

The results of simulation by pspice are stored in an output file. It is possible to control the type and amount of output by various commands. If there is any error in the circuit file Pspice will display a message on the screen indicating that there is an error and will suggest looking at the output file for details. The output falls into four types.

i)   A description of the circuit itself that include the net list, the device list, the model parameter list and so on.

ii)   Direct output from some of the analysis without. PLOT and PRINT commands. This includes output from OP, TF, SENS, NOISE and FOUR analysis.

iii)   Prints and plots by PLOT and PRINT commands. These include the output from DC, AC and TRAN analysis.

iv)   Run statistic, this includes the various kinds of summary information about the whole run, including time required for various analysis and the amount of memory used.

3.4) TYPES OF ANALYSIS :-

Pspice allows various types of analysis. Each analysis is performed by including its command statement. Different types of analysis used are as follows

A] DC Analysis:-

DC Analysis is used for circuits with time invariant sources. It calculates all node voltages and branch current over a range of values and their quiescent (dc) values are the outputs DC analysis is useful to study the effect of parameters variation on voltage and current in dc circuits. The simulation of dc circuits with passive elements requires the modeling of

Resistors

Model Parameters

Operating temperature

DC sources

DC output variables

Type of dc analysis

i) Resistors :- The name of a resistor must start with R and it takes the general form of

R <name> N_+ N_- R $_{NAME}$ R $_{Value}$

order of nodes does not matters, R_{NAME} is the model name that defines the parameters of the resistor.

R_{value} is the resistance in ohms and can be positive or negative but must not be zero.

If R_{NAME} is included and TCE is specified, the resistance as a function of temperature is calculated from

$$TCE * (T - T0) \; R_{RES} = R_{VALUE} * R * 1.01$$

If R_{NAME} is included & TCE is not specified, the resistance as a function of temperature is calculated from

$$RES = R_{VALUE} * R * [1 + TC1*(T-T_0) + TC2*(T-T_0)^2]$$

T and T_0 are the operating and room temperatures respectively in ^0C

ii) Modeling of Elements:-

To consider the effect of parameter variations the models are necessary. Models specifies a set of parameters for an element is specified in Pspice by the MODEL command. The same model can be used by one or more elements in the same circuit. The general form of the model statement is

MODEL MNAME TYPE ($P_1 = A_1$, $P_2 = A_2$, $P_3 = A_3$ ..... PN=AN)

MNAME ---> Name of the model

P_1, P_2 ..... P_N are the element parameters and A_1, A_2.... A_N are their values. TYPE is the type name of the elements

TYPE NAME OF ELEMENTS

Type Name	Element
RES	Resistor
CAP	Capacitor
D	Diode
IND	Inductor
NPN	NPN bipolar junction transistor
PNP	PNP bipolar junction transistor
NJF	N- Channel junction FET
PJF	P – Channel junction FET
NMOS	N – Channel MOSFET
PMOS	P – Channel MOSFET
GASFET	N – Channel Ga AS MOSFET
CORE	Non linear magnetic core

iii) Operating Temperature:-

Operating temperature of an analysis can be set to any desired value by TEMP command. The general form of the statement is-

.TEMP < (One or more temperature) values>

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

vi) Independent DC sources :-

The independent sources can be time-invariant or time variant they can be current or voltage.

Voltage source:- The general form of independent voltage source is

V<name> N^+ N^- [DC <Value>]

N^+ is positive node and N^- is the negative node

The source is set to the dc value in dc analysis. A voltage source may be used as an ammeter in Pspice by inserting a zero.valued voltage source into the circuit.

Current source - The symbol of an independent current source is I and the general form is-
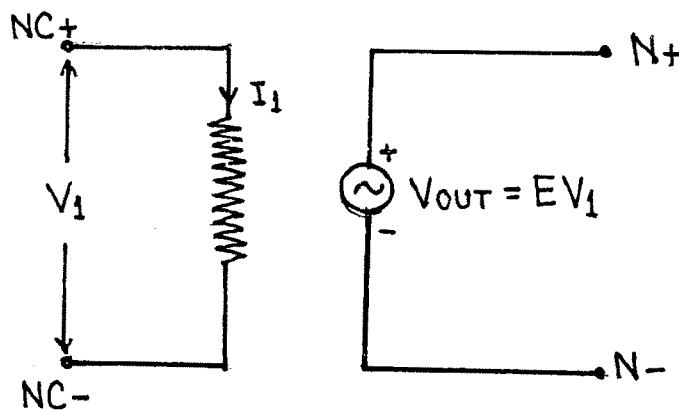
I<name> N^+ N^- [DC <value>]

N^+ is positive node and N^- is the negative node

V)   Dependent sources :-

There are four types of dependent sources

a)Voltage- controlled voltage source

b)Voltage – controlled current source

c)Current – controlled current source

d)Current - controlled voltage source

a) Voltage controlled voltage source :-

Fig 3.1 (Voltage controlled voltage source)
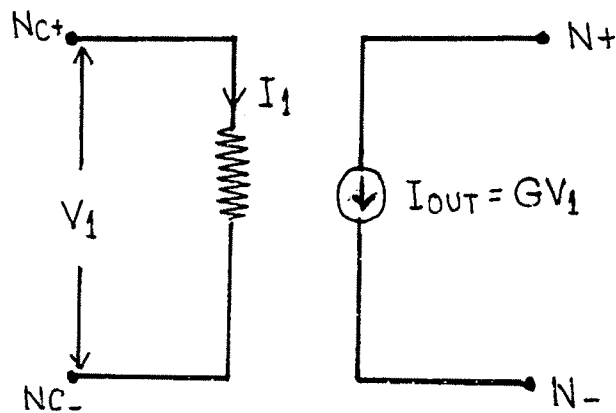
The symbol of voltage controlled voltage source is shown in fig.3.1. The general form of it is

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

N^+ and N^- are the positive and negative output nodes and NC^+ and NC^- are positive and negative nodes of the controlling voltage.

b) Voltage- Controlled current sources

Fig 3.2 (Voltage controlled current source)

The symbol of voltage controlled current source is shown in fig.3.2. The general form of it is-

G <name> N^+ N^- Nc^+ Nc^- <(transconductance)value>

N^+ and N^- are the positive and negative output nodes respectively. NC^+ and NC^- are the positive and negative nodes respectively of the controlling voltages.
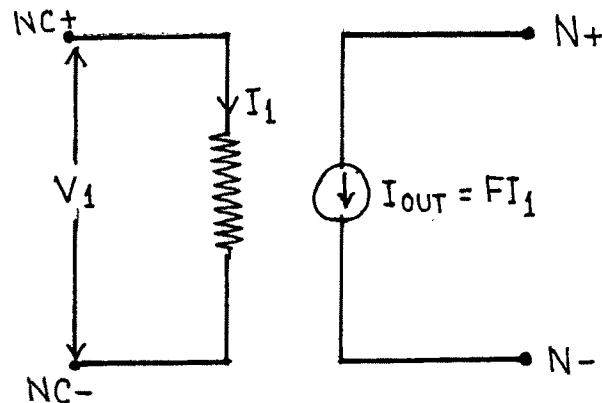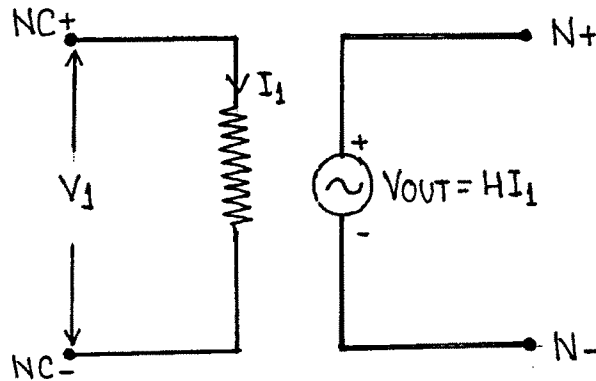
c) Current controlled current source

Fig 3.3 (Current controlled current source)

The symbol of the current controlled current source is shown in fig.3.3 and its linear form is

F<name> N^+ N^- VN   <(Current gain> value>

N^+ and N^- are the positive and negative nodes respectively of the current source. V_N is the voltage source through which the controlling current flows. The current through the controlling voltage source I (V_N) determines the output current.

d) Current Controlled voltage source:-

Fig. 3.4 (Current controlled ~~current~~ voltage source)

The symbol of a voltage controlled current source is shown in fig.3.4 and its linear form is

H <name> N^+ N^- VN <(transconductance)value>

N^+ and N^- are positive and negative nodes of voltages source VN is the voltage source through which controlling current flows.

vi)   Types of Output:-

The command that are available to get output from the results of simulation are-

    a) . PRINT

    b) . PLOT

    c) . PROBE

    d) . WIDTH

a)   .PRINT (Print Statement)

The results from dc analysis can be obtained in the form of tables. The print statement of dc output takes form

.PRINT DC   [Output variables]

The maximum number of output variable is eight in any . PRINT statement.   More than one statement can be used to print all the desired output variables.

b)   .PLOT (Plot statement)

The results from dc analysis can also be obtained in the form of line printer plots. The plot statement for dc output takes the following form

.PLOT DC <Output variables>

+ [<Lower limit> value>, <(Upper limit)value>]

The maximum number of output variables is eight in any PLOT statement. The range and increment of the x-axis is fixed by the dc analysis command. The range of Y-axis is set by adding lower limit value and upper limit value at the middle of a set of output variables.

c)   .PROBE (Probe statement)

Probe is a graphic post processor/waveform analyzer for Pspice. The simulation results cannot be used directly by probe. First the results have to be processed by .PROBE command which write the processed data on a file, PROBE. DAT, for use of probe. The command takes the form.

.PROBE

.PROBE <one or more output variables>

.PROBE command writes all the node voltages and the entire element current into the PROBE. DAT file.

Once the results of the simulations are processed by .PROBE command. The results are available for graphical displays and can be further mainipulated through expressions.

d)   .WIDTH (Width statement)

The width of the output in columns can be set by the .WIDTH statement, which has the general form

.WIDTH OUT = <Value>

The <Value> is in columns and must be either 80 or 132

vii)   Types of DC Analysis:-

The commands that are commonly used for dc analysis are

   a) .OP   DC operating point

   b) .TF   Small signal transfer function

   c) .DC   DC sweep

a)   .OP (Operating Point)

Electronic and Electrical circuit contain nonlinear devices like, diodes, transistor, whose parameters depends on the operating point. The

operating point is always calculated by Pspice in calculating the small signal parameters of non-linear devices during the dc sweep and transfer function analysis. The command takes the form .OP

The .OP command control the output of the bias point if the .OP command is present Pspice prints the current and power dissipation of all the voltage sources.

b)   .TF (Small-signal Transfer Function)

The small signal transfer function capability of pspice can be used to computer the small signal dc gain, the input resistance and the output resistance of a circuit.

The statement for the transfer function has the following forms

.TF V_{OUT} V_{IN}

.TF I_{OUT} I_{IN}

The .TF command calculates the parameters of Thevenins or Norton's equivalent circuit for the circuit file. It automatically prints the output and does not requires .PRINT or .PROBE statements.

c)   .DC (DC sweep)

The dc sweep is also known as the dc transfer characteristics. The input variable is varied over a range of values. For each value of input variable, the dc operating point and the small signal dc gain are computed by calling the small signal transfer function capability of pspice. The dc

sweep is obtained by repeating the calculation of the small signal transfer function for a set of values. The statement for performing the dc sweep takes one of the following general forms.

.DC LIN SWNAME SSTART SEND SINC [nested sweep specification]

.DC OCT SWNAME SSTART SEND NP [nested sweep specification]

.DC DEC SWNAME SSTART SEND NP [nested sweep specification]

.DC SWNAM LIST <Value> [(nested sweep SWNAME is the sweep variable name and could be either a voltage or current source. SSTART, SEND and SINC are the start value, the end value and the increment value of the sweep variable respectively.

NP is the number of steps

LIN, OCT or DEC specifies the type of sweep

B] Transient Analysis

To study the behaviour of an electric circuit as a function of time, transient analysis is used. The simulation of transients in circuits with linear elements requires modeling of

i) Capacitors
ii) Inductors
iii) Modeling of transient sources
iv) Transient sources
v) Transient output variables
vi) Output commands
vii) Transient analysis

i) **Capacitor :-**

The symbol for a capacitor is C. The name of a capacitor must start with C and it takes the general form

C <name> N+ N – CNAME CVALUE IC = V0

N+ and N- are the positive and negative node respectively.

C NAME is the model name and CVALUE is the nominal value of the capacitor, IC defines initial voltage of the capacitor.

If C NAME is omitted. CVALUE is the capacitance in farads. If CNAME is included, the capacitance that depends on voltage and temperature is calculated from

$$CAP = CVALUE * C * (1+VC_1 * V +VC_2 * V^2)$$

$$* [1 + TC_1 * (T-T_0) +TC_2 * (T - T_0)^2]$$

TC_1 and TC_2 are linear and quadratic temperature coefficient.

VC_1 and VC_2 are linear and quadratic voltage coefficient.

T is operating temp in ^0C & T^0 is room temp. in ^0C

ii) **Inductor:-**

The name of inductor must start with L and it takes the following general form.

L <name> N+ N- L_{NAME} L_{VALUE} IC = I0

LNAME is the model name and LVALUE is the nominal value of the inductor. IC defines the initial current of the inductor I0 IF LNAME is omitted LVALUE is the inductance in henrys and LVALUE can be positive or negative but must not be zero. IF LNAME is included the inductance that depends on current and temperature is calculated from

$$IND = LVALUE * L * (1 + IL_1 * I + IL_2 * I^2)$$

$$*(I + TC_1 * (T-T_0) + TC_2 * (T-T_0)^2]$$

T   --> Operating temp. in ^0C

T_0 --> Room temp. in ^0C

L   --> inductance multiplier

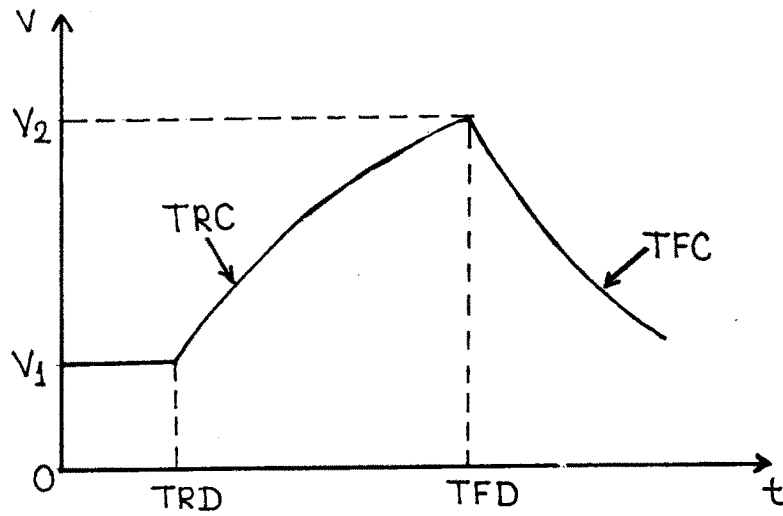**iii)   Modeling of Transient Sources:-**

The independent voltage or current sources that can be modeled by pspice as follow

    a) Exponential

    b) Pulse

    c) Piecewise linear

    d) Sinusoidal

    e) Single frequency frequency modulation

a)   Exponential Source :-

The symbol of exponential source is EXP, and the general form is
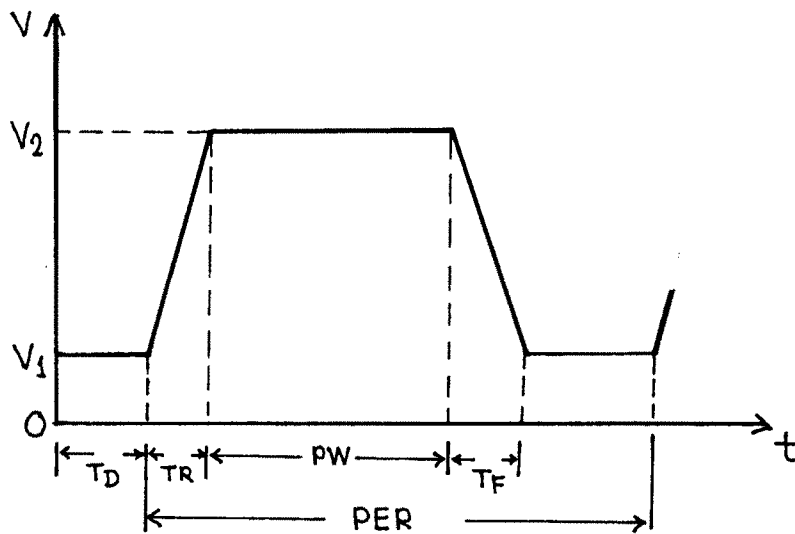
EXP (V_1 V_2 TRD TRC TFD TFC)

Fig. 3.5 (Exponential waveform)

V_1 and V_2 must be specified by the user and can be either voltages or currents

TRC ---> Rise time constant   TFC ----> Fall time constant

TFD ----> Fall delay time

b) Pulse source

Fig. 3.6 (Pulse waveform)

The waveform and parameter of a pulse waveform are shown in fig.3.6.The general form of pulse waveform is

$$PULSE\ (V_1\quad V_2\quad T_D\quad T_R\quad T_F\quad P_W\quad PER)$$

Model Parameters of PULSE sources

Name	Meaning	Units	Default
V_1	Initial voltage	Volts	None
V_2	Pulsed voltage	Volts	None
TD	Delay time	Seconds	0
TR	Rise time	Seconds	TSTEP
TF	Fall time	Seconds	TSTEP
PW	Pulse Width	Seconds	TSTOP
PER	Period	Seconds	TSTOP

C)   Piecewise Linear Source:-

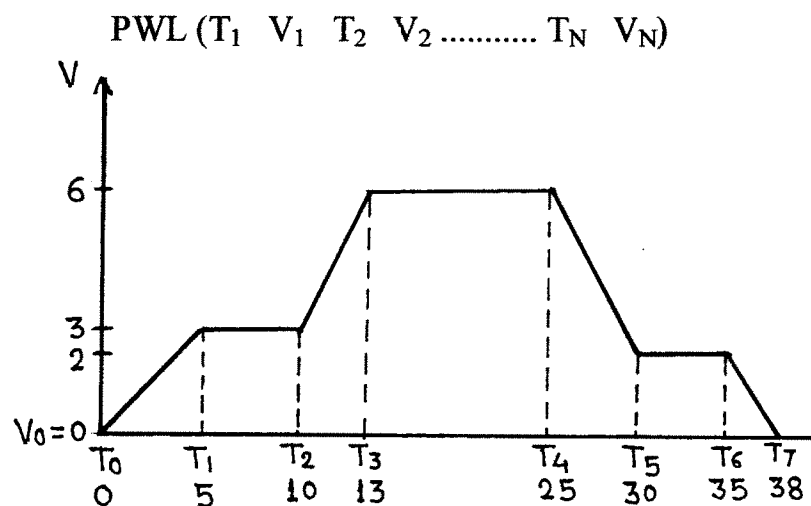The symbol of piecewise linear source is PWL and the general form is

$$PWL\ (T_1\quad V_1\quad T_2\quad V_2 \ldots\ldots\ldots T_N\quad V_N)$$

Fig. 3.7 (Piecewise linear waveform)

Model Parameter of PWL sources are

Name	Meaning	Units	Default
Ti	Time at a point	Seconds	None
Vi	Voltage at a point	Volts	None

d)   Single Frequency Frequency Modulation:-

The general form of source with single frequency frequency modulation is

$$\text{SFFM} (V_0 \quad V_A \quad F_C \quad \text{MOD} \quad \text{FS})$$

Model parameters of SFFM waveform are

Name	Meaning	Units	Default
V_0	Offset voltage	Volts	None
V_A	Amplitude of voltage	Volts	None
FC	Carrier frequency	Hz	1/ T STOP
MOD	Modulation index		0
Fs	Signal frequency	Hz	1/STOP

e)   Sinusoidal Source:-

The general form of sinusoidal source is

$$\text{SIN} (V_0 \quad V_A \quad F_{REQ} \quad TD \quad ALP \quad THETA)$$

Model parameters of sinusoidal waveform are

Name	Meaning	Units	Default
V0	Offset voltage	volts	None
VA	Peak Voltage	Volts	None
FREQ	Frequency	Hz	1/TSTOP
TD	Delay Time	Seconds	0
ALPHA	Damping factor	1/seconds	0
THETA	Phase delay	Degree	0

iv)   **Transient sources**

The transient sources are time-variant and can be either independent or dependent.

a)   Independent voltage source

The general form of independent voltage source is

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

+ [transient value]

+ [PULSE] [SIN] [EXP] [PWL] [SFFFM] [Source argument]

A source can be assigned either a dc value or a transient value. The source is set to the dc value in dc analysis. The time dependent source is assigned for transient analysis.

b)   Independent current source:-

The symbol of an independent current source is I and the general form for assigning dc and transient value is

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

+   [(transient value)]

+   [PULSE] [SIN] [EXP] [PWL] [SFFM] [Source arguments]

V]   Transient Response:-

A transient response determines the output in the time domain in response to an input signal in the time domain. The determination of the transient analysis requires statements involving

.IC   Initial transient condition

. TRAN Transient analysis

The transient analysis can be performed by the .TRAN command. Which has one of the general forms.

.TRAN TSTEP TSTOP [TSTART TMAX] [UIC]

. TRAN [/OP] TSTEP TSTOP [TSTART TMAX] [UIC]

TSTEP is the printing increment. TSTOP is the final time and TMAX is the maximum size of internal time step.

An . OP command with a .TRAN command . TRAN/OP will print the small signal parameters during transient analysis.

If UIC is not specified as an option at the end of the .TRAN statement, pspice calculates the transient]_analysis bias point before the beginning of transient analysis.

C) AC circuit Analysis:-

The ac analysis calculates the frequency response of a circuit over a rang of frequencies. If the circuit contains non linear devices or elements, it is necessary to obtain the small signal parameter of the elements before calculating the frequency response prior to ac analysis, pspice determines the small signal parameters of the element. The method of calculating the bias point for ac analysis is identical to that for dc analysis.

The general form to perform ac analysis is a follows.

.AC LIN NP FSTART   FSTOP

.AC OCT NP FSTART FSTOP

.AC DEC NP FSTART FSTOP

NP is the number of points in a frequency sweep. FSTART is the starting frequency and FSTOP is the ending frequency

LIN, OCT DEC specify the type of sweep.

FSTART must be less than FSTOP and cannot be zero. The results of the frequency response that are obtained on the display by the .PROBE command.

D] Multiple Analysis:-

A source may be assigned the specifications of dc, ac and transient analysis for different analysis. The statements for voltage and current source have the following general forms

V<name>   N+   N-

+   [DC <value>]

+   [AC <(magnitude)value> <(phase) value>]

+   [Transient specifications)]

I <name>   N+   N-

+   [DC <value>]

+   [AC < (magnitude) value> <(phase) value>]

+   [(transient specifications)]

The source is set to a dc value in dc analysis. It is set to an ac values in ac analysis. The time dependent source is assigned for transient analysis. This allows source specification for different analysis in the same statement.

E] Fourier Analysis:-

The results of transient analysis are in discrete forms. These sampled data can be used to calculate the coefficients of a Fourier series.

Pspice uses the result of the transient analysis to perform the Fourier analysis up to n^{th} harmonics. The statement takes the general form

.FOUR FREQ N V1   V2   V3 .... VN

.FOUR FREQ N I1 I2 I3 ... IN

FREQ is the fundamental frequency V1, V2 ...VN or I1, I2 .... IN

are output voltages or current for which the Fourier analysis is desired. A. FOUR statement must be used with a. TRAN statement . If the number of harmonics N is not specified, the dc component, fundamental and second through ninth harmonic are calculated by default.

Pspice does print a table, automatically, showing the results of Fourier analysis and does not require. PRINT, .PLOT or .PROBE statements.

F] Noise Analysis

Semiconductor devices and resistors generate noise. The level of the noise depends on the frequency. Noise analysis is done in combination with ac analysis. For each frequency of ac analysis the noise level of each generator in a circuit is calculated and there contributions to output nodes are computed by summing the rms noise values. The gain from the input source to the output voltage is calculated. From this gain, the equivalent input noise level at the specified source is calculated by pspice.

The statement for performing noise analysis has the following form.

.NOISE V (N+, N-) SOURCE M

Where V (N+ , N-) is the output voltage across nodes N+ and N-.

Source is the name of an independent voltage or current source at which the equivalent input noise will be generated.

M is the print interval that permits pspice to print a table for the individual contributions of all generators to the output nodes for every m^{th} frequency. It M is not specified then Pspice does not print a table of individual contributions.

G] Sensitivity Analysis:-

The sensitivity, of output voltages or current with respect to every circuit and device parameter can be calculated by .SENS statement. Which has the following general form.

.SENS <(one or more output) variables>

The .SENS statement calculates the bias point and the linearized parameters around the bias point. In this analysis inductors are assumed to be short circuit and capacitors are assumed to be open circuit. The sensitivity of each output variable with respect to all the device values and model parameters is calculated and . SENS statement may generate a huge amount of data if many output variables are specified.

H] **Parametric Analysis:-**

.STEP command is used to evaluate the effects of parameter variations. The general form of it is.

.STEP LIN SWNAME SSTART SEND SINC

. STEP OCT SWNAME SSTART SEDN NP

. STEP DEC SWNAME SSTART SEND NP

. STEP SWNAME LIST <Value>

SWNAME is the sweep variable name. SSTART, SEND and SINC are the start value, the end value and the increment value of the sweep variables. NP is the number of steps LIN, OCT or DEC specifies the type of sweep.

The sweeps start value START may be greater than or less than the sweep end value SEND.

The sweep increment SINC must be greater than zero.

The number of points NP must be greater than zero.

If. .STEP command is included in a circuit file, all specified analysis are done for each step.

3.5] **OP . AMP. REPRESENTATION IN PSPICE :-**

Linear model of operational Amplifier gives best result for determining the approximate design value of Op.Amp. Circuit for accurate response of the circuits in some application the simulation of actual behaviour of Op.Amp is required. Pspice does not have any model for

Op.Amp. The Op.Amp can be simulated from the circuit arrangement of particular type of OP.Amp. like µA741. IC 741 consists 24 transistors and it is beyond the capacity of the student version of Pspice. Therefore the macromodel, which is simplified version of the OP. Amp and requires only two transistors, is quite accurate for many applications and can be simulated as a subcircuit or library file.

In absence of a complex Op.Amp. model the characteristics of Op.Amp. circuit may be determined approximately by one of the following model.

* DC linear model
* AC linear model
* Non linear macromodel

i)   DC Linear Model:-

For IC 741 the input impedance is 2 Mega ohm and the output impedance is 75 ohm. By considering these parameters the operational amplifier is reduced to that of Fig. 3.8
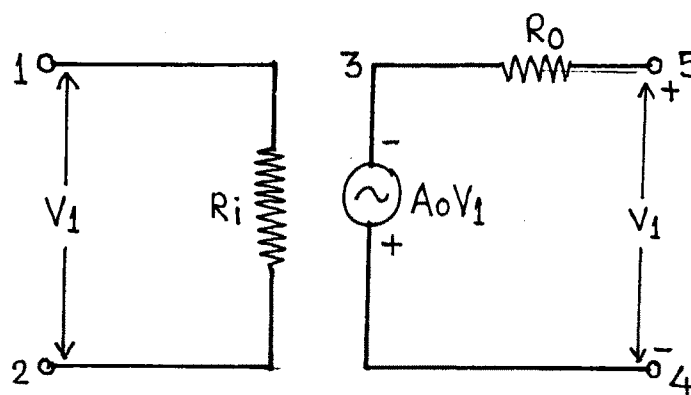

Fig. 3.8 (DC linear model of Operational Amplifier)

Ri represents input impedance of Op. Amp, R_0 represents o/p impedance of op.Amp. and A_0 is the open loop gain of the operational Amplifier.

In this model the effect of slew rate and saturation effect is not considered and the gain is also frequency dependent. In actual op.Amp gain falls with frequency. This model is suitable for dc and low frequency application.

ii) **AC Linear Model :-**

The frequency response of an op-amp can be approximated by a single break frequency as shown in Fig.3.9
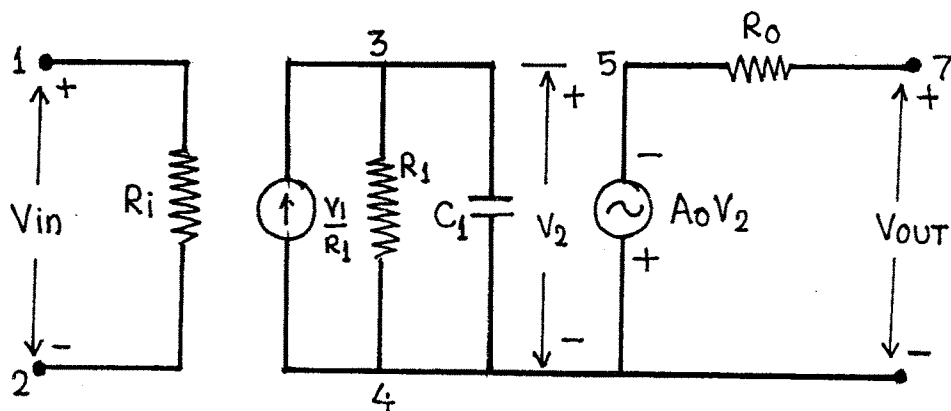
Fig. 3.9 (AC Linear model with a single break frequency)

The number of capacitor determines the number of break frequency. It is a high frequency model of op-amp with single break frequency. This model does not take into account the saturation effect and is suitable only if the op-amp operates within the linear region.

The output voltage $V_{out} = -A_0 V_2 = \dfrac{-A_0 V_{in}}{1 + j\, f/f_b}$

fb is the break frequency

$$f_b = \dfrac{1}{2\pi R_1 C_1}$$

A_0 is the large signal gain of Op.Amp.

The open loop gain $A(f) = \dfrac{-A_0}{1 + k\, f/fb}$

For µA741 IC

$f_b = 10$ Hz $\qquad A_0 = 2 \times 10^5 \qquad\qquad R_i = 2$ Mega ohm

$R_0 = 75$ ohm $\qquad C_i = 1.15619\ \mu f$

iii)   Non linear macromodel for operational Amplifier :-

The macromodel of Op.Amp is simulated at room temperature. The macromodel can be used as a subcircuit with the SUBCKT command. However if an OP.Amp is used in various circuits it is convenient to have
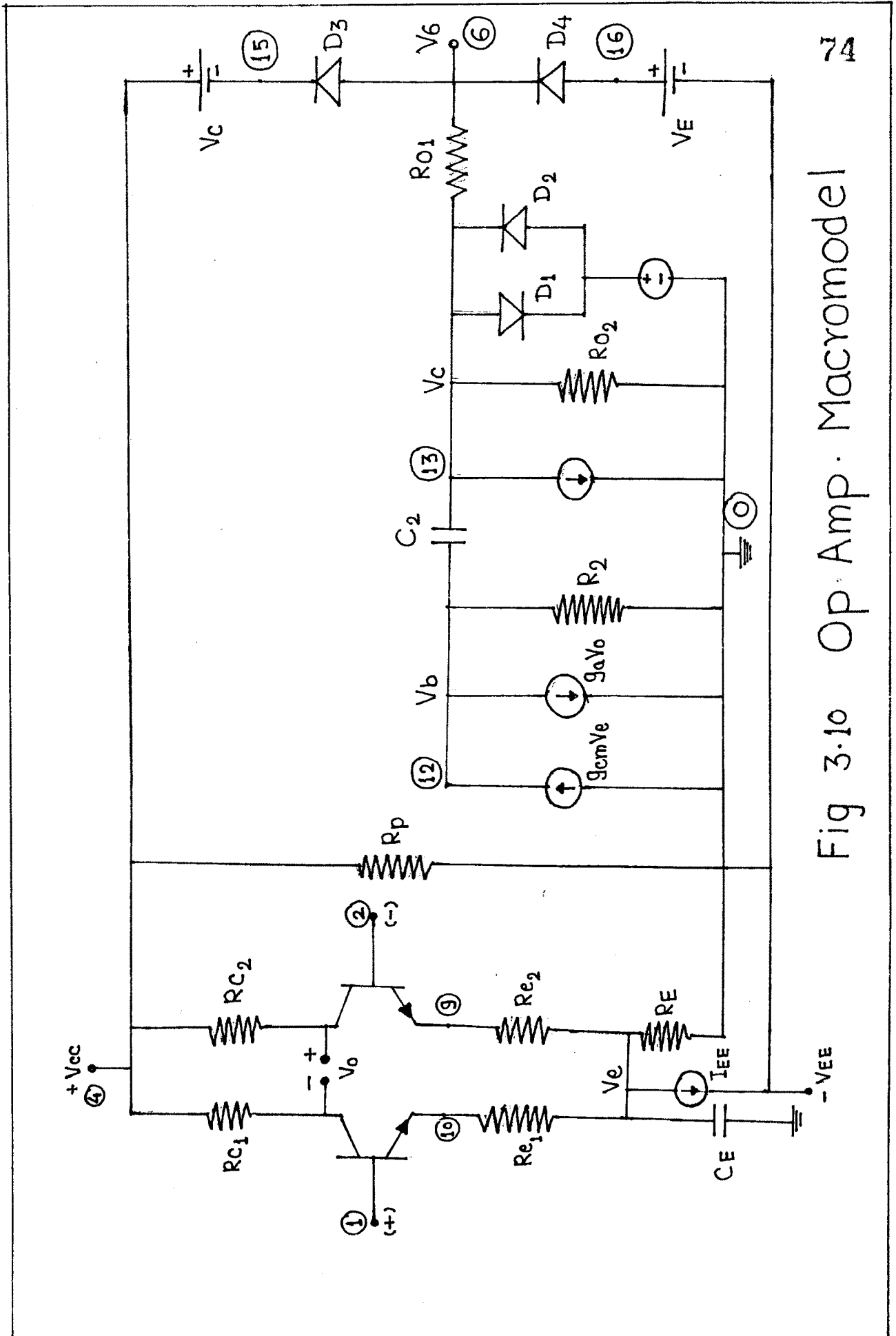
Fig 3.10 Op. Amp. Macromodel

74

the macromodel as a library files namely EVAL.LIB and it is not required to type statements of the macromodel in every circuit where the macromodel is employed. These models contain nominal, not worst case devices and do not consider the effect of temperature.

The student version of Pspice consist of the macromodel of

OP.Amp. LM324, μA741 and LM311

The circuit diagram of Op.Amp. macromodel is shown in fig.3.10

The listing of macromodel of μA741 in library file EVAL.LIB is as follows.

*   **Library file of OP.Amp. UA 741**

*   Library file "EVAL. LIB" for UA741 Op.Amp.

*   Connections : noninverting input

*   : Inverting input

*   : : Positive power supply

*   : : : Negative power supply

*   : : : : Output

*   : : : : :

*   SUBCKT UA741   1   2   4   5   6

*   Vi+   vi-   vp+   vp-   Vout

    Q_1 7   1   10   UA741 QA

    Q_2 8   2   9   UA741 QB

    RC_1 4   7   5.305165 D + 03

RC$_2$	4	8	5.30165 D + 03
C$_1$	4	8	5.459535 D − 12
RE$_1$	10	11	2.5151297 D + 03
RE2	9	11	2.151297 D + 03
IEE	11	05	1.666000 D − 05
CE	11	0	3.00000 D − 12
RE	11	0	1.200480 D + 07
GCM	0	12	11 3 5.960753 D − 09
GA		12	0 8 7 1.884955 D −04
R2	12	0	1.00000D + 05
C2	12	13	3.0000 D-11
GB		13	0 12 0 2.357851 D +02
RO2	13	0	4.50000 D + 01
D1	13	14	UA741 DA
D2	14	13	UA741 DA
EC	14	0	6 3 1.0000
R01	13	0	4.5000 D + 01
D3	6	15	UA741 DB
VC		4	15 2.803238 D + 00
D4	16	06	UA741DB
VE	16	05	2.803238D +00
RP	4	05	18.16D+03

* Models for diodes & transistor

. MODEL UA741 DA D (Is = 9.752287 D-11)

. MODEL UA741 DB   D (Is = 8.0000 D – 16)

. MODEL UA 741 QA NPN (Is = 8.000 D – 16 BF = 9.16667 D + 01)

. MODEL UA 741 NPN (IS = 8.000 D-16 BF = 1.178571D + 02)

* END OF Library File

* END of subcircuit definition

. ENDS