Title of Discipline:

Computer-Aided Analysis of Electronic Circuits

Laboratory Lecture 7

Bachelor : Telecommunication Technologies and Systems

Year of Study: 2

Computer-Aided Analysis of Electronic Circuits

Laboratory 7

PSpice AC analysis

Standard analyses in PSpice

PSpice command	Description
.OP (bias point)	Bias point
.DC (DC analysis)	DC sweep
.TF (transfer)	Small-signal DC transfer function
.SENS (sensitivity analysis)	DC sensitivity
.AC (AC analysis)	Frequency response
.NOISE (noise analysis)	Noise
.TRAN (transient analysis)	Transient
.FOUR (Fourier analysis)	Fourier components

.AC command

Purpose

The .AC command is used to calculate the frequency response of a circuit over a range of frequencies.

General Form

.AC <sweep type> <points value>

+ <start frequency value> <end frequency value>

Examples

.AC LIN 101 100Hz 200Hz

.AC OCT 10 1kHz 16kHz

.AC DEC 20 1MEG 100MEG

.AC command

<sweep type> The sweep type must be either LIN, OCT, or DEC.

Parameter	Description	Meaning
LIN	linear sweep	The frequency is swept linearly from the starting to the ending frequency. The <points value> is the total number of points in the sweep.
OCT	Sweep by octaves	The frequency is swept logarithmically by octaves. The <points value> is the number of points per octave.
DEC	Sweep by decades	The frequency is swept logarithmically by decades. The <points value> is the number of points per decade.

.AC command

<points value> The points value (an integer), is the number of points in the sweep.

<start frequency value> <end frequency value>

The end frequency values must not be less than the start frequency value, and both must be greater than zero. The whole sweep must include at least one point.

The simulator calculates the frequency response by linearizing the circuit around the bias point.

All independent voltage and current sources which have AC values are inputs to the circuit.

Note A .PRINT, .PLOT, or .PROBE command must be used to get the results of the AC sweep analysis

.NOISE (Noise Analysis)

Purpose

The .NOISE command causes a noise analysis of the circuit to be performed.

General Form

.NOISE V(<node> [,<node>]) <source_name> [interval value]

Example

.NOISE V(5) VIN

.NOISE V(101) VSRC 20

.NOISE V(4,5) ISRC

Note A noise analysis is performed in conjunction with AC analysis and requires a .AC command.

.NOISE (Noise Analysis)

V(<node> [,<node>])

An output voltage. It has a form such as V(5), which is the voltage at the output node five, or a form such as V(4,5), which is the output voltage between two nodes four and five.

<source_name>

The name of an independent voltage or current source where the equivalent input noise is calculated. The <source_name> is not itself a noise generator, but only a place where the equivalent input noise is calculated.

The noise-generating devices in a circuit are the resistors and the semiconductor devices. For each frequency of the AC analysis, each noise generator’s contribution is calculated and propagated to the output nodes. At that point, all the propagated noise values are RMS-summed to calculate the total output noise. The gain from the input source to the output voltage, the total output noise, and the equivalent input noise are all calculated.

.NOISE (Noise Analysis)

If <*source_name*> is a voltage source then the input noise units are Volt/Hertz$^{1/2}$

If <*source_name*> is a current source then the input noise units are Amp/Hertz$^{1/2}$

The output noise units are always Volt/Hertz$^{1/2}$

[interval value] is optional and is an integer which specifies how often the detailed noise analysis data is written to the output file.

Every *n*-th frequency, where n is the print interval, a detailed table is printed showing the individual contributions of all the circuit’s noise generators to the total noise. These values are the noise amounts propagated to the output nodes, not the noise amounts at each generator. If *[interval value]* is not present, then no detailed table is printed.

The detailed table is printed while the analysis is being performed, and does not need a .PRINT command or a .PLOT command. The output noise and equivalent input noise can be printed in the output by using a .PRINT command or a .PLOT command.

.NOISE (Noise Analysis)

For noise analysis, the output variables are predefined as follows.

Output variable	Meaning
INOISE	Total RMS summed noise at input node
ONOISE	INOISE equivalent at output node
DB(INOISE)	INOISE in decibels
DB(ONOISE)	ONOISE in decibels

Applications

Application 1

Activities: 1. Describe into the SPICE circuit file (.cir) the circuit shown in following figure:

The model type for the Q1 and Q2 transistors is BC107A. The PSpice description of these transistors is as below:

Q1 2 1 4 BC107A

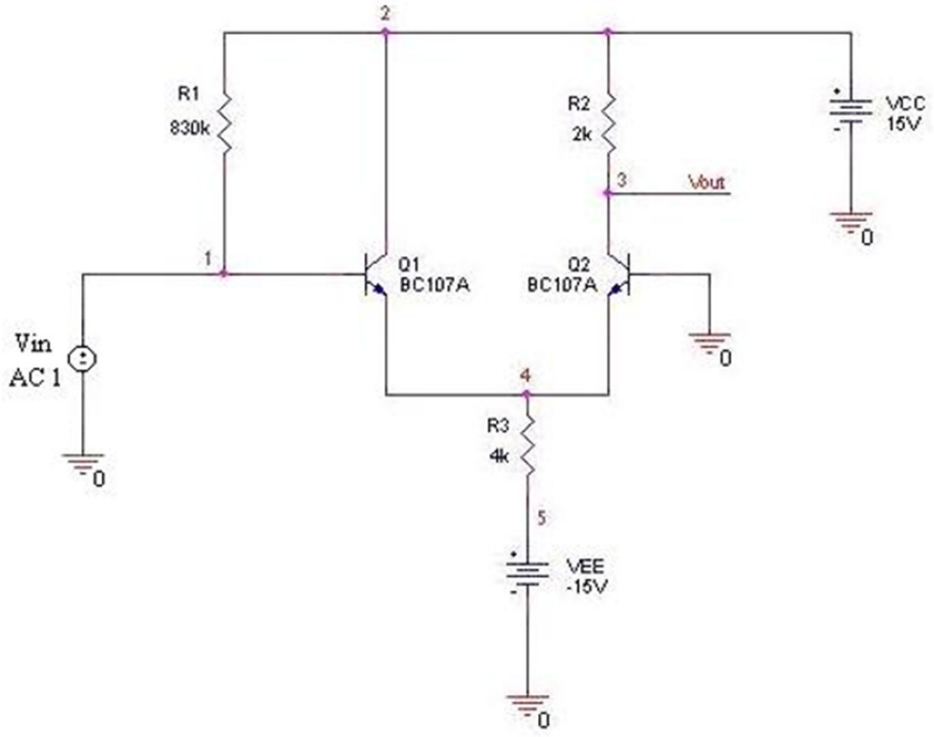
Q2 3 0 4 BC107A

.LIB NOM.LIB

The BC107A model is defined in the library as:

.MODEL BC107A NPN(model parameters).

The .LIB command defines the name of the library file.



Applications

Application 1

2. For the mentioned circuit do the following:

Apply a voltage source with AC 1 specification at the input of the circuit, perform the frequency domain analysis along with the noise analysis in the (10Hz, 100MHz) range and visualize the following:

a) Magnitude transfer characteristic. Determine and write down the maximum transfer value. Compare this value with DC and low signal (.TF);

b) Transfer function in dB. Determine the frequency band to -3dB;

c) Phase transfer characteristic;

d) Input impedance. Determine the maximum input impedance value and compare it to the DC input value determined from the .TF analysis.

Applications

Application 1

e) Noise characteristics:

- Input noise and equivalent input noise;
- Determine total noise. The following expression will be represented: SQRT (s (V (ONOISE) * V (ONOISE))
- Determine the signal / noise ratio when the signal is 2mV.

3. Determine the frequency dependence of the output impedance of the circuit in the frequency range (10Hz, 100MHz). To do this, perform AC analysis as at point 2., but an independent current source with AC 1 specification will be applied at the output of the circuit and passivate the signal source at the input. Determine the maximum output impedance value and compare it with the output resistance value determined from the .TF analysis.