Introduction to EDSPICE Simulation using OPUSER

LAB MANUAL FOR ANALOG CIRCUITS LAB - EDSPICE

Albi Easo
Principal Application Engineer
Visionics India Pvt Ltd, Technopark, Trivandrum

Introduction to EDSPICE Simulation using
OPUSER First Edition

Albi Easo

Preface

This eBook is compiled for the easier illustration of EDSPICE simulation of Analog Simulation lab to cater the B.Tech syllabus (**EC010 307 ANALOG CIRCUITS LAB**) of M.G University, Kerala. The experiments are described in a step by step manner.

The EDSpice simulation experiments describes the Aim, Components required for the simulation, the circuit diagram, procedure, steps for the simulation and finally the result of the simulation. The Components required for the experiment is given as Type of Component – OPUSER Component Name (Number of Components). The procedure section describes the method of simulating the circuit with the values to be assigned to each component in the circuit and the steps for simulating the circuit for the specified analysis. The result is obtained in the Waveform viewer.

Contents

Introduction to SPICE

SPICE is an acronym for **Simulation Program with Integrated Circuit Emphasis**. This is a sort of node description language which is used to describe the circuits, especially analog circuits.

Simulation is the process by which the intended output is analyzing without actually fabricating the circuit.

Why it is required ?

- Any circuit can be analyzed by means of any type of available analysis.
- There is no need of real life components and instruments.
- There is no risk of damaging components and instruments etc.

SPICE can perform various analysis for:

- Evaluating effects of variations in elements
- Assessment & performance/degradation
- Evaluating effects of noise
- Fourier analysis
- DC analysis
- Transient analysis
- Operating point analysis
- AC analysis
- Noise analysis

Input for simulation

1. Through .CIR file

A circuit is described by statements that are stored in a file called a circuit file. The circuit file is read by the SPICE simulator .Each statement in circuit file is self contained and independent.

2. By Schematic capturing of the circuit diagram

Capture the components available in the library and interconnect to make the circuit.

The circuit must contain the descriptions for following:

- Circuit elements
- Sources

- Nodes
- Element values
- Element models
- Types of Analysis
- Output variables
- Output Commands
- .END

Format of a circuit file

- .Title
- Circuit description
- .model
- .Analysis description
- output description
- .End

```
*RCCOUPLED AMPLIFIER
vcc  5 0 dc 10v
rc   5 3 2.2k
r1   5 2 33k
r2   2 0 18k
re   4 0 1k
c1   1 2 10u
ce   4 0 100u
c2   3 6 10u
rl   6 0 470
v1   1 0 sin(0,20mv,1k)
Q1   3 2 4 BC107
.model BC107 NPN(BF=400)
.tran 0.1ms 10ms
.plot tran v(6) v(1)
.end
```

About OPUSER

OPUSER (Electronic Design for Windows) is a PCB design CAD/CAE package. It can be also used for circuit simulation purposes. An electronic design engineer can use the computerized tools provided in OPUSER to create an electronic circuit, design the PCB and fabricate PCB. Due to the integrated structure of OPUSER database, a design may be started from either the schematic diagram or the PCB layout and design information is annotated automatically to other modules. This front and back annotation makes OPUSER a fully integrated package.

OPUSER may be visualized as a package comprising of five task oriented modules such as *Schematic Editor, Layout Editor, Fabrication, Simulation and Library modules*.

Introduction to Simulation

Analysis is an important part of any design process. The soundness of a design should be tested to ensure proper functioning of the final PCB. Circuit Analysis involves testing the performance of the circuit for different DC operating points and with signal applied, over a range of output levels, supply voltages and temperatures.

Simulators allow testing of the circuits without having to build them thereby reducing the development time and cost. Further simulators allow testing of the circuits over a wide parameter range easily.

OPUSER provides two types of simulators one is OPUSER Mixed Mode Simulator and other is EDSpice Simulator which adheres to the SPICE standards. Here we emphasis more about EDSpice Simulator. The user must specify the source and the mode of analysis prior to running a simulator.

EDSpice Simulator:

EDSpice is a SPICE-like simulator. It is solely based on Berkeley SPICE3 with a number of extensions and improvements. It provides users with the facility to analyze and validate the functionality and behavior of circuits captured in the form of Schematic diagrams.

The interactive module of EDSpice, provides seamless integration between the simulator and the rest of the system. The core of the package is the EDSpice Simulation Engine, which performs the simulations and outputs the results. The user communicates with the engine through the interactive module. This module includes a set of function tools to prepare circuits for simulation, initiate the execution of analyses, and review the results. *Waveform Viewer* is used to display and post process the results of analyses in graphical form. If required, the results may be also presented textually.

To Setup Simulator Variables (options)

Certain global parameters, preset options and conditions for analyses are passed to the simulation engine in the netlist. Dialog windows for defining the contents such lines as .OPTIONS (Simulator variables), .IC (Initial Condition) and .NODESET (Initial Node Voltage Guesses) may be accessed from the *Analysis* menu of *Simulation*

Setup of the .OPTIONS line involves either including keywords in the line (clicking the parameter in the table toggles the selection "Yes/ No") or defining non-zero values of the

variables (clicking the parameter in the table opens an edit box for typing in the values). Only variables not equal to zero will be included in the .OPTIONS line. Click ACCEPT to accept these values.

To Setup Initial Node Voltage Guesses

Certain global parameters, preset options and conditions for analyses are passed to the simulation engine in the netlist. Dialog windows for defining the contents such lines as .OPTIONS (Simulator variables), .IC (Initial Condition) and .NODESET (Initial Node Voltage Guesses) may be accessed from the *Analysis* menu of *Simulation*.

The procedure of setting up the contents of the .IC and .NODESET lines involves selection of nets and specifying their voltage values. Nets may be selected from the check boxes against each net name. To specify the voltage value, click the Voltage column in the table. The voltage value for the selected net may be typed in the edit box. The operation of specifying a voltage value is confirmed by pressing <Enter> key. Click ACCEPT to accept these values.

To Setup Transient Initial Conditions

Certain global parameters, preset options and conditions for analyses are passed to the simulation engine in the netlist. Dialog windows for defining the contents such lines as .OPTIONS (Simulator variables), .IC (Initial Condition) and .NODESET (Initial Node Voltage Guesses) may be accessed from the *Analysis* menu of *Simulation*.

The procedure of setting up the contents of the .IC and .NODESET lines involves selection of nets and specifying their voltage values. Nets may be selected from the check boxes against each net name. To specify the voltage value, click the Voltage column in the table. The voltage value for the selected net may be typed in the edit box. The operation of specifying a voltage value is confirmed by pressing <Enter> key. Click ACCEPT to accept these values.

Types of Analysis supported by EDSpice Simulator.

1. Transient Analysis

A Transient Analysis examines the time domain response of a circuit within a time frame specified by the user, and reports the variation of any voltage or current over this time.

SYNTAX

General Form

.TRAN TSTEP TSTOP <TSTART <TMAX>>

Examples

.TRAN 1NS 100NS

.TRAN 1NS 1000NS 500NS

.TRAN 10NS 1US

TSTEP is the printing or plotting increment for line-printer output. For use with the post-processor, TSTEP is the suggested computing increment. TSTOP is the final time, and TSTART is the initial time. If TSTART is omitted, it is assumed to be zero. The transient analysis always begins at time zero. In the interval <zero, TSTART>, the circuit is analyzed (to reach a steady state), but no outputs are stored. In the interval <TSTART, TSTOP>, the circuit is analyzed and outputs are stored. TMAX is the maximum step-size that SPICE uses; for default, the program chooses either TSTEP or (TSTOP-TSTART)/50.0, whichever is smaller. TMAX is useful when one wishes to guarantee a computing interval which is smaller than the printer increment, TSTEP.

2. Fourier Analysis

Fourier Analysis generates information used to expresses any complex, periodic waveform as a Fourier Series, provided it satisfies the Dirichlet Conditions. In other words, a Fourier Analysis gathers information about the harmonic contents of the waveform.

Fourier Analysis can also be used to characterize the Distortion in linear circuits that occur due to part non-linearities, provided the distortions are not too small.

SYNTAX

General Form

FOUR FREQ OV1 <OV2 OV3 ...>

Examples

.FOUR 100K V(5)

The Four (or Fourier) line controls whether SPICE performs a Fourier analysis as a part of the transient analysis. FREQ is the fundamental frequency, and OV1, ..., are the output variables for which the analysis is desired. The Fourier analysis is performed over the interval <TSTOP-period, TSTOP>, where TSTOP is the final time specified for the transient analysis, and period is one period of the fundamental frequency. The dc component and the first nine harmonics are determined. For maximum accuracy, TMAX (see the .TRAN line) should be set to period/100.0 (or less for very high-Q circuits).

3. Small Signal AC Analysis

Small Signal AC Analysis is a linear analysis, over a user defined frequency range, based on the linearized small signal AC model values of all circuit components at the Operating point. It is used to obtain the small signal AC behavior of the circuit. Both AC currents and voltages at various nodes can be obtained.

SYNTAX

General Form

.AC DEC ND FSTART FSTOP

.AC OCT NO FSTART FSTOP

.AC LIN NP FSTART FSTOP

Examples

.AC DEC 10 1 10K

.AC DEC 10 1K 100MEG

.AC LIN 100 1 100HZ

DEC stands for decade variation, and ND is the number of points per decade. OCT stands for octave variation, and NO is the number of points per octave. LIN stands for linear variation, and NP is the number of points. FSTART is the starting frequency, and FSTOP is the final frequency. If this line is included in the input file, SPICE performs an AC analysis of the circuit over the specified frequency range. Note that in order for this analysis to be meaningful, at least one independent source must have been specified with an ac value.

4. DC Transfer Function

DC Transfer Function Analysis is used to obtain the variation in circuit voltages and/or currents, with respect to variations in either one or two, independent source/s, at DC Bias conditions. It is used for obtaining the small signal DC bias solution of a circuit, as one

(or two) independent sources sweep over a range of values. It is also often used for obtaining the characteristic output curves of semiconductor devices.

SYNTAX

General Form

.DC SRCNAM VSTART VSTOP VINCR [SRC2 START2 STOP2 INCR2]

Examples`

.DC VIN 0.25 5.0 0.25

.DC VDS 0 10 .5 VGS 0 5 1

.DC VCE 0 10 .25 IB 0 10U 1U

The DC line defines the dc transfer curve source and sweep limits (again with capacitors open and inductors shorted). SRCNAM is the name of an independent voltage or current source. VSTART, VSTOP, and VINCR are the starting, final, and incrementing values respectively. The first example causes the value of the voltage source VIN to be swept from 0.25 Volts to 5.0 Volts in increments of 0.25 Volts. A second source (SRC2) may optionally be specified with associated sweep parameters. In this case, the first source is swept over its range for each value of the second source. This option can be useful for obtaining semiconductor part output characteristics.

5. **Distortion Analysis**

Signal distortions like Crossover distortion, Failure-to-follow distortion, Frequency distortion, Phase distortion can be viewed and analysed by running an appropriate AC small signal analysis.

However, Harmonic distortion and Inter modulation distortion can be studied in SPICE only by running a Distortion analysis. Harmonic Distortion is the change in a signal at the output, compared to the input, due to the frequency components that get introduced into the output as a result of part non-linearity. The additional frequencies are integer multiples of the input frequency.

SYNTAX

General Form

.DISTO DEC ND FSTART FSTOP <F2OVERF1>

.DISTO OCT NO FSTART FSTOP <F2OVERF1>

.DISTO LIN NP FSTART FSTOP <F2OVERF1>

Examples

.DISTO DEC 10 1kHz 100Mhz

.DISTO DEC 10 1kHz 100Mhz 0.9

The Disto line does a small-signal distortion analysis of the circuit. A multi-dimensional Volterra series analysis is done using multi-dimensional Taylor series to represent the nonlinearities at the operating point. Terms of up to third order are used in the series expansions.

6. Operating Point Analysis

Operating Point Analysis is used to determine the DC behavior of a circuit. It is performed using DC values, at which inductors are effectively short circuits and capacitors are open circuits.

The program must be informed about the results you wish to know. The default selection is "As Marked". This means that results will be displayed at all Voltage, Current and Logic State Markers placed on the circuit schematic. After analysis, the results will be presented by updating the markers. If you wish to know the values at all nodes and branches of the circuit, select "All Points". The results will be available in the default .raw file RAWSPICE.RAW located at \..\EDS_WRK. You may view it by selecting "*File → View EDSpice Files → Rawfile*"

SYNTAX

General Form

 .OP

The inclusion of this line in an input file directs EDSpice to determine the dc operating point of the circuit with inductors shorted and capacitors opened. Note: a DC analysis is automatically performed prior to a transient analysis to determine the transient initial conditions, and prior to an AC small-signal, Distortion, and Pole-Zero analysis to determine the linearized, small-signal models for nonlinear parts (see the KEEPOPINFO variable on the .OPTIONS control line).

7. Noise Analysis

SPICE is capable of modeling the following kinds of electronic noise:

* Thermal noise

* Shot noise, and

* Flicker noise

Noise analysis is used to analyze the noise existing at any point in a circuit, due to the combined effect of all noise sources in the circuit.

SYNTAX

General Form

.NOISE V(OUTPUT <,REF>) SRC (DEC | LIN | OCT) PTS FSTART FSTOP
+ <PTS_PER_SUMMARY>

Examples

.NOISE V(5) VIN DEC 10 1kHZ 100Mhz

.NOISE V(5,3) V1 OCT 8 1.0 1.0e6 1

The Noise line does a noise analysis of the circuit. OUTPUT is the node at which the total output noise is desired; if REF is specified, then the noise voltage V(OUTPUT) - V(REF) is calculated. By default, REF is assumed to be ground. SRC is the name of an independent source to which input noise is referred. PTS, FSTART and FSTOP are .AC type parameters that specify the frequency range over which plots are desired. PTS_PER_SUMMARY is an optional integer; if specified, the noise contributions of each noise generator is produced every PTS_PER_SUMMARY frequency points.

The .NOISE control line produces two plots - one for the Noise Spectral Density curves and one for the total Integrated Noise over the specified frequency range. All noise voltages/currents are in squared units (V /Hz and A /Hz for spectral density, V and A for integrated noise).

8. DC/ AC Sensitivity Analysis

Signals at any point of a circuit are affected by the rest of the circuit. The dependence of any circuit current or voltage, on parameters of all other parts in the circuit can be measured by means of a Sensitivity Analysis. Sensitivity Analysis provides information about the circuit parameter/s which is mostly affected by the specified current or voltage. This information can be used to decide the maximum variations in the values of various parts and parameters. In other words, acceptable tolerances for part values can be determined.

In order to run DC or Small-Signal AC Sensitivity Analysis, specify the analysis parameters by selecting Analysis/ DC or Small-Signal AC Sensitivity Analysis from the tree view. Specify the simulation parameters. If you require only a DC Sensitivity Analysis, make sure the "AC Sensitivity" check box is not selected. (However, make sure it is checked if you want an AC Sensitivity Analysis.). If 'As Setup' is checked, the analysis is performed at the

temperature set from the menu Setup/ Temperatures. If this option is not checked, the operating temperature is taken as 27° C. Select the type of presenting results from 'Results' dropdown and set the parameters as per the selection made for Waveform Viewer. Click ACCEPT button to accept these values and on RUN button to execute the analysis.

After analysis, the results will be presented as follows:

* The results of a DC Sensitivity Analysis will be presented in the .raw file RAWSPICE.RAW.

* The results of AC Sensitivity Analysis will be presented in the Waveform Viewer. The full results can be examined by selecting "*File → View EDSpice Files → Rawfile*".

SYNTAX

General Form

      .SENS OUTVAR

      .SENS OUTVAR AC DEC ND FSTART FSTOP

      .SENS OUTVAR AC OCT NO FSTART FSTOP

      .SENS OUTVAR AC LIN NP FSTART FSTOP

Examples

      .SENS V(1,OUT)

      .SENS V(OUT) AC DEC 10 100 100k

      .SENS I(VTEST)

      The sensitivity of OUTVAR to all non-zero part parameters is calculated when the SENS analysis is specified. OUTVAR is a circuit variable (node voltage or voltage-source branch current). The first form calculates sensitivity of the DC operating-point value of OUTVAR. The second form calculates sensitivity of the AC values of OUTVAR. The parameters listed for AC sensitivity are the same as in an AC analysis. The output values are in dimensions of change in output per unit change of input (as opposed to percent change in output or per percent change of input).

9. Transfer Function Analysis

      The magnitudes of the following quantities can be obtained by means of a Transfer Function Analysis:

* Voltage Gain

* Current Gain

* Transfer Impedance

* Transfer Admittance

In order to run Transfer Function Analysis, specify the analysis parameters by selecting Analysis/ Transfer Function Analysis from the tree view. Specify the simulation parameters. Click ACCEPT button to accept these values and on RUN button to execute the analysis. After analysis, the results will be presented in the .raw file RAWSPICE.RAW.

SYNTAX

General Form

.TF OUTVAR INSRC

Examples

.TF V(5, 3) VIN

.TF I(VLOAD) VIN

The TF line defines the small-signal output and input for the dc small-signal analysis. OUTVAR is the small signal output variable and INSRC is the small-signal input source. If this line is included, SPICE computes the dc small-signal value of the transfer function (output/input), input resistance, and output resistance. For the first example, SPICE would compute the ratio of V(5,3) to VIN, the small-signal input resistance at VIN, and the small-signal output resistance measured across nodes 5 and 3.

10. Pole-Zero Analysis

A Pole-Zero analysis is used to compute poles and zeroes of a small signal AC transfer function of the circuit. Pole-zero analysis is most commonly used for determining the stability of control circuits.

In order to run Pole-Zero Analysis, specify the analysis parameters by selecting *Analysis/ Pole-Zero Analysis* from the tree view. Specify the simulation parameters. Click ACCEPT button to accept these values and on RUN button to execute the analysis. After analysis, the results will be presented in the **.raw** file RAWSPICE.RAW.

SYNTAX

General Form

.PZ NODE1 NODE2 NODE3 NODE4 CUR POL

.PZ NODE1 NODE2 NODE3 NODE4 CUR ZER

.PZ NODE1 NODE2 NODE3 NODE4 CUR PZ

.PZ NODE1 NODE2 NODE3 NODE4 VOL POL

.PZ NODE1 NODE2 NODE3 NODE4 VOL ZER

.PZ NODE1 NODE2 NODE3 NODE4 VOL PZ

Examples

.PZ 1 0 3 0 CUR POL

.PZ 2 3 5 0 VOL ZER

.PZ 4 1 4 1 CUR PZ

CUR stands for a transfer function of the type (output voltage)/(input current) while VOL stands for a transfer function of the type (output voltage)/(input voltage). POL stands for pole analysis only, ZER for zero analysis only and PZ for both. This feature is provided mainly because if there is a non-convergence in finding poles or zeros, then, at least the other can be found. Finally, NODE1 and NODE2 are the two input nodes and NODE3 and NODE4 are the two out-put nodes. Thus, there is complete freedom regarding the output and input ports and the type of transfer function.

Model Libraries in EDSpice

The Model Parameter Editor in OPUSER eases the management and creation of model libraries. It can be used in a variety of ways including extracting **.MODEL** lines from existing SPICE netlist, loading existing models from the model libraries allowing you to modify and re-save them, and the creation of new models.

There are three kinds of model libraries used by EDSpice. The first two types facilitate the simulation of a wide range of circuit elements whose functionality is defined by software extensions to the simulation engine. The behavior of the element is encoded in a dedicated software module, which is dynamically linked to the simulation engine during runtime. This feature of the EDSpice Simulator is called code models. The program modules which contain this functionality, and currently available in EDSpice, are stored in a separate subdirectory on the hard disk. This library of code models is automatically created while the EDSpice package is installed on the user's machine.

The EDSpice Code Model Library contains program modules allowing for simulation of more than a hundred digital and analog circuit elements. Extensions of XSPICE to SPICE and its gaining popularity will bring more models available on the market from sources other than Visionics. This will allow sharing of models between the users of XSPICE based simulators without the involvement of the original vendors.

Similarly, the second type - library of User-Defined Nodes - is also available with the EDSpice Simulator. User-Defined Node types allow you to specify nodes that propagate

arbitrary data structures and data other than voltages, currents, and digital states. User-Defined Nodes communicate with the simulation engine in the same manner as the code models. The User-Defined Nodes Library is automatically installed with the rest of the EDSpice package.

According to SPICE convention, the characteristics of a circuit element are defined by part parameters, on the element line of a netlist, representing an instance of the element. However, some complex elements, such as transistors, are characterized by both part parameters, defined on the element line, and model parameters, grouped in a .MODEL statement. The model statement allows you to specify only one set of parameters common to a number of elements, for example, the parameters of all transistors with the same geometry integrated on one silicon chip. Therefore, you can have multiple instances of a model.

All part and model parameters of circuit elements should be properly set before simulation is executed. It applies without difference to elements defined by external code models and models implemented internally by SPICE. For the same type of element, each set of different parameter values describe characteristics of a specific, real life electronic component. These values are normally listed in data books and catalogs supplied by manufacturers. In many cases, associated with BJT's, MOSFETs and other more complicated circuit elements, the process of setting up these parameters may be quite tedious, very often requiring a good knowledge of the internal working of the component.

The EDSpice package allows for the maintenance of a preset model parameters library. Parameters of every model once set for simulation in one circuit may be stored on the hard disk in a selected library file and reused in other circuits. This library is called Model Parameters Library and is the third type of library of the EDSpice package.

Anticipating problems with model parameter setup, the manufacturers of electronic components supply data files in SPICE compatible format with the appropriate model parameters describing characteristics of their products. The interactive module of EDSpice includes features which can extract .MODEL lines from files compatible with the SPICE netlist format and append the set of parameters contained in one line as a single item in the Model Parameters Library. This feature is also useful for designers who already use other SPICE simulators and wish to add models from existing netlist files. The feature is called the Model Library Editor.

It is assumed that users will enhance the Model Parameters Library. During installation of the package, the subdirectory intended for this library is automatically created on the hard disk.

EDSpice Model Parameters Library may contain any number of library files, each capable of storing up to 2048 models.

The model parameters are stored in the library, using an internal format, and are accessible only for usage by a circuit elements setup features within the interactive module. For identification purposes, the models are tagged with library names, which may be up to 15 characters long. Within one library file, the names must be unique and must conform to certain naming conventions. According to this convention, aimed for easier recognition of the models, the general format of the model library name is as follows:

prefix#userID

Prefix (with a maximum length of 7 characters) is automatically added to the model library name. It is always the type of the model according to SPICE syntax. For example, all models containing parameters for NPN transistors will be assigned the prefix **NPN#**. Users may add their own ID to the model name, so that the total number of characters including the prefix# does not exceed 15. Apart from the name, each model in the library may contain a description provided as additional identification of the models.

The model library structure also supports the family approach of grouping the models. There are two purposes to group model parameters family libraries.

1. The user may find the models easier.
2. It may happen that model parameters may have the same identifier. For example BC807 from two manufacturers, Motorola and NS. In such cases they may be stored with the same identifier in different family library file and still be unique.

The grouping of models from a particular manufacturer are made to a single **.**Sparam file. For example transistors from MOTOROLA have been saved to MOTOBJTS.Sparam under the directory /../EDS_MOD. Likewise the grouping is done for models from different manufacturers.

Apart from the **.**Sparam files representing models from different manufacturers, there are two more files - MISC.Sparam and MIXMODE.Sparam. SAMPLES.Sparam contains model parameters used by samples. MIXMODE.Sparam is used by the system.

Family Prefix, may be used (optionally) to connect a part to specific model parameter library file. EDSpice Reference for models looks in following way:

MMM#IIIII

Where MMM is Model Type (ex PNP) and IIIII is identifier (ex BC807). When PNP#BC807 is coded as EDSpice Ref then the program would search through all library files in the Model Parameters Library until it finds or not finds the PNP#BC807.

The Family Prefix is optional (settable in EDSpice Symbol Editor) and is nothing else but model parameters library file name (no extension). Hence if it is decided to use Family prefix and if the parameters are stored in MOTOBJTS.Sparam then the EDSpice Reference would be as follows:

MOTOBJTS.PNP#BC807

The program will try to load this parameters from MOTOBJTS.Sparam. If unsuccessful then it would work like without family prefix - meaning that it will search through other files in this subdirectory.

The model library name may be used as the Simulation References to assign these models to parts. Such components created as parts with appropriate Simulation Reference set automatically attaches model parameters corresponding to these Simulation Parameters while preprocessing is done. This is a one time activity, i.e., model parameters pointed to by the Simulation Reference is loaded only if the component does not have any model parameters set. Once model parameters are changed by the user, the new parameter set always overrides the default ones set using Simulation Reference.

Note: EDSpice Simulation References are assigned to parts and not to symbols.
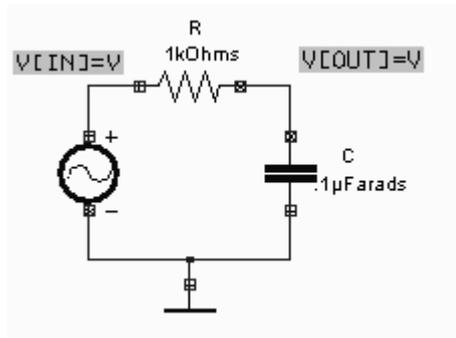
INTEGRATOR

Aim

To simulate an integrator circuit.

Components Required

Resistor- RC05 (1), Capacitor- CK05 (1), AC Voltage Source-VGEN (1), Ground- SPL0 (1)

Circuit Diagram

Theory

A circuit in which output voltage is directly proportional to the integral of the input is known as an integrating circuit. An integrating circuit is a simple RC series circuit with output taken across the Capacitor C.

In order that circuit renders good integration, the following conditions should be full-filled.

1. The time constant RC of the circuit should be very large compared to the time period of the input wave.
2. The value of R should be 10 or more times larger than Xc.

Let e_i be the input alternating voltage and let i be the resulting alternating current. Since R is very large compared to capacitive reactance Xc of the capacitor, it is reasonable to assume that the voltage across R(i.e. e_R) is equal to the input voltage.i.e.

$$e_i = e_R$$

$$Now \quad i = \frac{e_R}{R} = \frac{e_i}{R}$$

The charge on the capacitor at any instant is

$$q = \int i \, dt$$

output voltage, $\quad e_0 = \frac{q}{C} = \frac{\int i \, dt}{C}$

$$= \frac{\int \frac{e_i}{R} dt}{C} \qquad (\because i = \frac{e_i}{R})$$

$$= \frac{1}{RC} \int e_i \ dt \quad \alpha \quad \int e_i \ dt \qquad \text{(RC is constant)}$$

$$Output \ voltage \quad \alpha \quad \int input$$

Thus the output is proportional to the time integral of the input.

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation →
Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → Set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→*
Click on the required component and change its values.

The values to be provided to the components are

          Resistor   : 1k

          Capacitor   : 0.1µF

          VGEN   : Select the source function as PULSE and set its Properties as follows

                V1 (Initial value)   : 0V

                V2 (Pulsed Value)   : 5V

                Pulse width   : 0.5m

                Period   : 1m (All other values are zero)

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view.

Enter following values:

              Step   : 1µ

              Final time : 5m

              Results   : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.

In order to obtain the response as low pass filter, the small signal AC analysis has to be carried out. For performing the *Small Signal AC Analysis*, select *Simulation → Analysis*. Select *Small Signal AC Analysis* from the tree view.
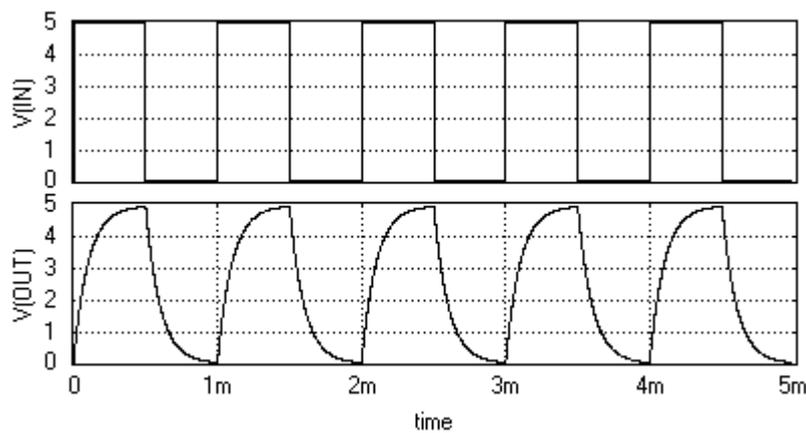
Enter the values as

Variation	: Decade
Total Points	: 100
Start Frequency	: 10Hz
End Frequency	: 10MHz
Results	: Select *Waveform* for displaying the output.

Click *Accept* button after entering the values to automatically switch to *Analysis*. Click *Run* button to start simulation. The output is obtained as given in the second waveform.

Result

Integrator Output
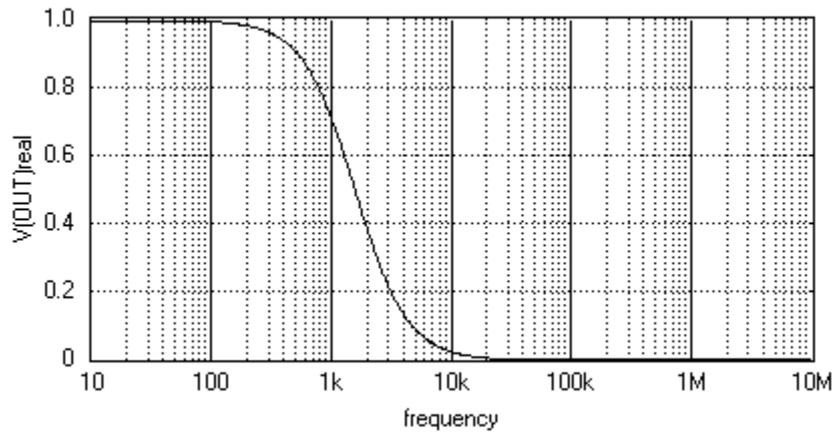
Integrator as Low Pass Filter

DIFFERENTIATOR

Aim

To simulate a Differentiator circuit

Components required

Resistor-RC05 (1), Capacitor – CK05 (1), AC voltage source – VGEN (1), Ground- SPL0 (1)

Circuit Diagram
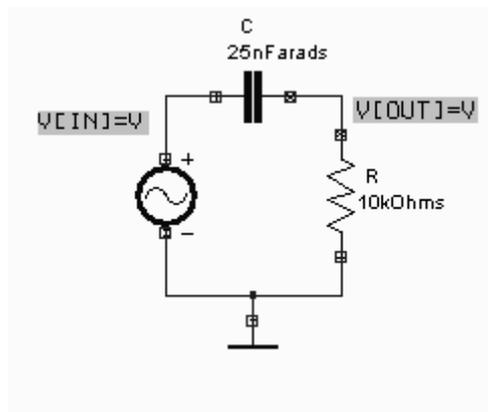
Theory

A circuit in which output voltage is directly proportional to the derivative of input is known as a differentiating circuit. A differentiating circuit is a simple RC series circuit with output taken across the resistor R.

In order to achieve good differentiation, the following conditions should be satisfied:

1. The time constant RC of the circuit should be smaller than the time period of the input wave.

2. The value of Xc should be 10 or more times larger than R at operating frequency.

Let e_i be the input alternating voltage and i be the resulting alternating current. The charge q on the capacitor at any instant is

$$q = C\, e_c$$

$$i = \frac{dq}{dt} = \frac{d}{dt}\, C\, e_c$$

$$i = C\frac{d}{dt}\, e_c$$

Since capacitive reactance is very much larger than R, the input voltage can be considered to be equal to the capacitor voltage with negligible error. i.e. $e_c = e_i$

$$i = C\frac{d}{dt}\, e_i$$

Output Voltage, $e_o = iR$

$$= RC\frac{d}{dt}\, e_i \; \alpha \; \frac{d}{dt}\, e_i$$

$$\therefore OutputVoltage = \frac{d}{dt}\, \text{(input)}$$

Hence the output is proportional to the time derivative of the input.

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation → Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

      Resistor   : 10k (1)

      Capacitor   : 25nF (1)

      VGEN   : Select the source function as PULSE and set its Properties as follows

          Initial value   : 0V

          Final Value   : 5V

Pulse width   : 0.5m

Period   : 1m (All other values are zero)

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view.

Enter following values:

Step   : 1µ

Final time   : 5m

Results   : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform* viewer, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.

For performing the *Small Signal AC Analysis*, select *Simulation* → *Analysis*. Select *Small Signal AC Analysis* from the tree view.

Enter the values as

Variation   : Decade

Total Points   : 100

Start Frequency : 10Hz

End Frequency   : 10MHz

Results   : Select *Waveform* for displaying the output.

Click *Accept* button after entering the values to automatically switch to *Analysis*.

Click *Run* button to start simulation

Result
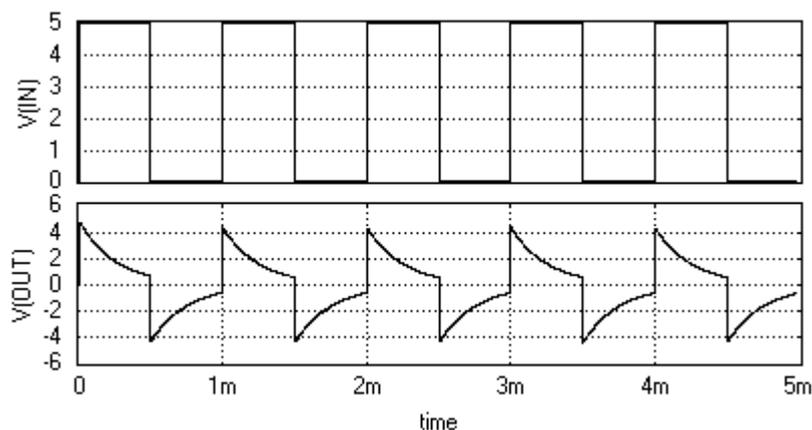
a) Transient Analysis
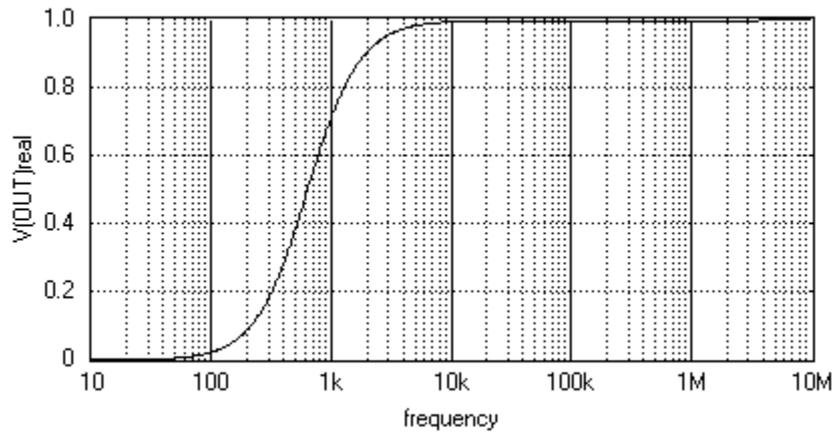
b) Small Signal AC Analysis

DIODE CHARACTERISTICS

Aim

To simulate the VI characteristics of a diode.

Components required

Diode – IN4001 (1), Resistor – RC05 (1), DC Voltage Source – VDC (2), Ground – SPL0 (1)
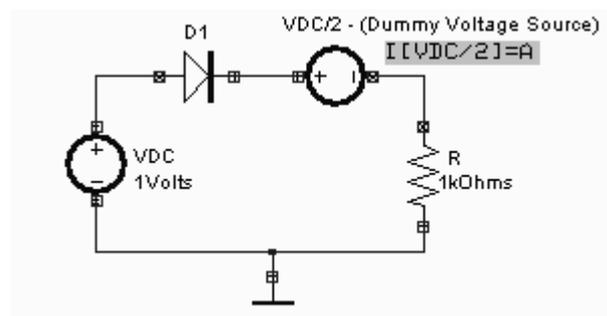
Circuit Diagram

Theory

The circuit diagram to plot the VI characteristics of a pn junction diode is shown. Applying a positive potential to the anode and a negative potential to the cathode of the pn junction diode establishes a forward bias condition on the diode. As the applied potential is increased the depletion region width decrease and conduction of electron increase. In general the characteristics of the semiconductor diode can be defined by the equation

$$I_D = I_S (e^{kV_D/T_K} - 1)$$

Where I_S = reverse saturation current, k=11600/h with h =1 for Ge and 2 for Si.

22

$T_K = T_C + 273$, T_K = Temperature in ° Kelvin, T_C = Temperature in ° Centigrade

V_D = Applied potential , I_D = Diode current

For positive values of V_D, I_D is positive and increases exponentially. At $V_D=0$, I_D is also zero (ref: equation). For negative values of V_D,

$I_D = -I_s$ which is a horizontal line. This explains the VI characteristics of pn junction diode.

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. The resistor R is assigned a proper value. Preprocess the circuit by invoking *Simulation → Preprocess*. The VI characteristics may be obtained by performing DC Transfer function Analysis. The current waveform marker is placed at the positive terminal of the dummy voltage source VDC/2. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Current waveform →* Click on the required net and place the waveform marker. The sweep parameter (voltage) for input source is set in the Analysis window. The applied voltage is swept from an initial value to final value with the steps provided. To get VI characteristics, the currents corresponding to varying input voltages are noted. The VI graph is observed in the Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Diode : 1N4001 (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistor : R = 1k

VDC/1 : 1V

VDC/2 : 0V

Note: For input/output characteristics plot of Diode, BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source   : VDC/1

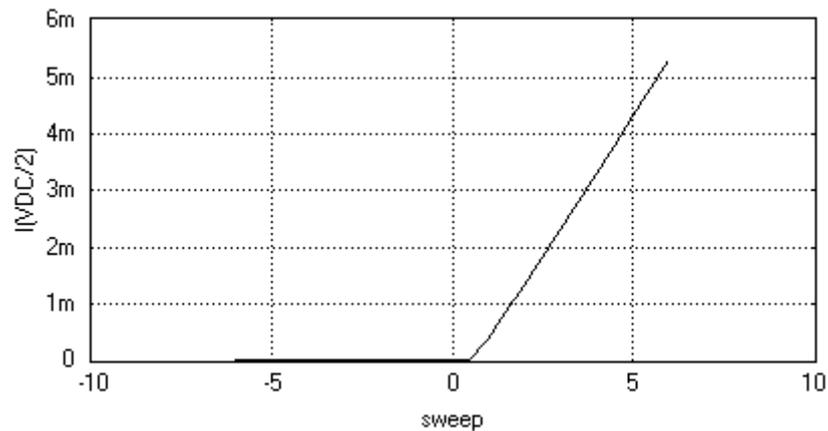Start voltage   : - 6V

Stop Voltage   : 6V

Step   : 500m

Results   : Select *waveform* from drop down menu.

Click on *Accept*. Run Analysis by clicking on *Run* button

Result

The output waveform can be observed in the waveform viewer.

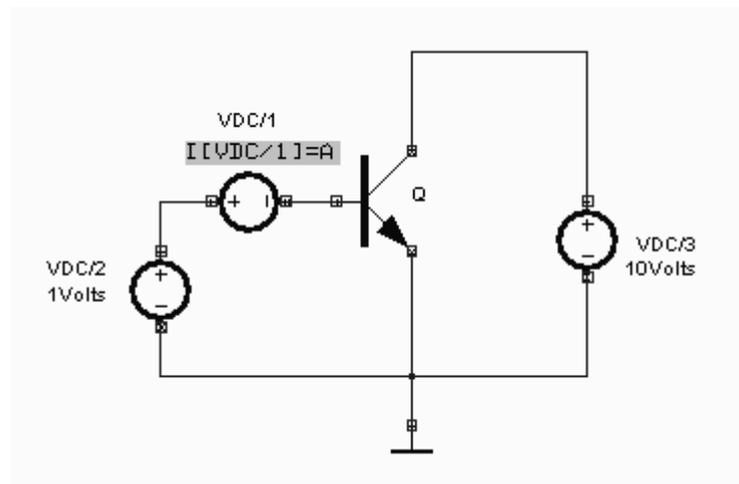# BJT INPUT CHARACTERISTICS (Common Emitter Configuration)

Aim

To simulate the BJT input characteristics in Common Emitter configuration

Components required

Transistor - BC107A (1), DC Voltage source - VDC (3), Ground – SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The circuit is preprocessed. The VI characteristics may be obtained by performing DC Transfer function Analysis. Place the current waveform marker at the positive terminal of the dummy voltage source (Voltage = 0 Volts). For placing waveform markers, select *Tools → Instruments → Set wave form Contents → Current waveform →* Click on the required net and place the waveform marker. The sweep parameter (voltage) for input source is set in the Analysis window. The applied voltage is swept from an initial value to final value with the steps provided. To get VI characteristics, the currents corresponding to varying input voltages are noted. The VI graph is observed in the Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

VDC/1 : 0V

VDC/2 : 1V

VDC/3 : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source : VDC/2

Start voltage : 0V

Stop voltage : 2V

Step : 100m

Results : Select *waveform* from drop down menu.

Click on *Accept*. Expand the *DC transfer function* and select *Waveform viewer*, Select dummy voltage source branch. Click on *Accept*. Select *DC Transfer function analysis*. Run Analysis by clicking on *Run* button

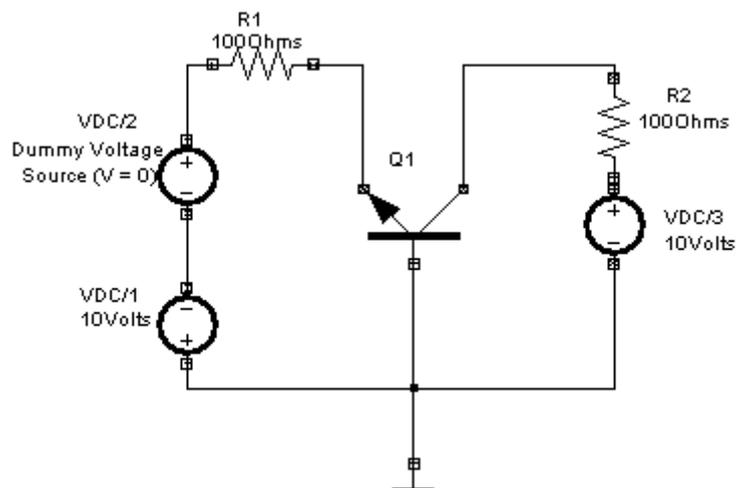**Result:**

BJT INPUT CHARACTERISTICS – CB Configuration

Aim
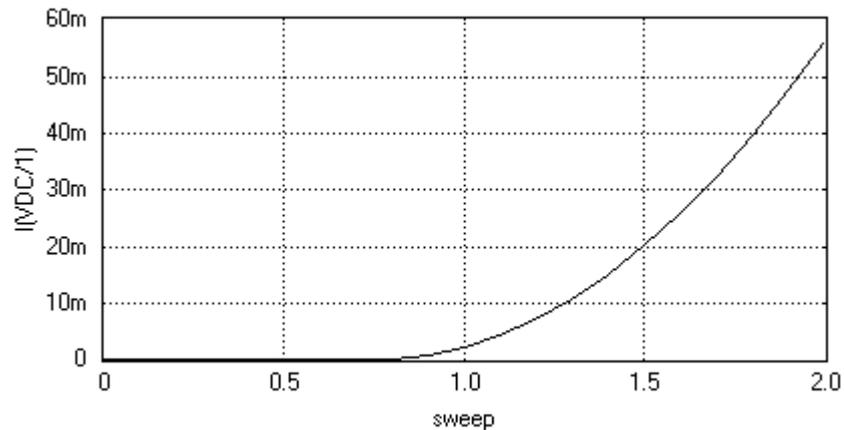
To simulate the BJT input characteristics in common base configuration

Components required

Transistor - BC107A (1), Resistor – RC05 (2), DC Voltage source - VDC (3), Ground - SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source (VDC/2) with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The circuit is preprocessed. The VI characteristics may be obtained by performing DC Transfer function Analysis. Place the current waveform marker at the positive terminal of the dummy voltage source VDC/2 (Voltage = 0 Volts). For placing waveform markers, select *Tools → Instruments → set wave form Contents → Current waveform →* Click on the required net and place the waveform marker. The sweep parameter (voltage) for input source is set in the Analysis window. The applied voltage is swept from an initial value to final value with the steps provided. To get VI characteristics, the currents corresponding to varying input voltages are noted. The VI graph is observed in the Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistors : R_1 = 100 Ohm, R_2 = 100 Ohm

VDC/1 : 10V

VDC/2 : 0V

VDC/3 : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:
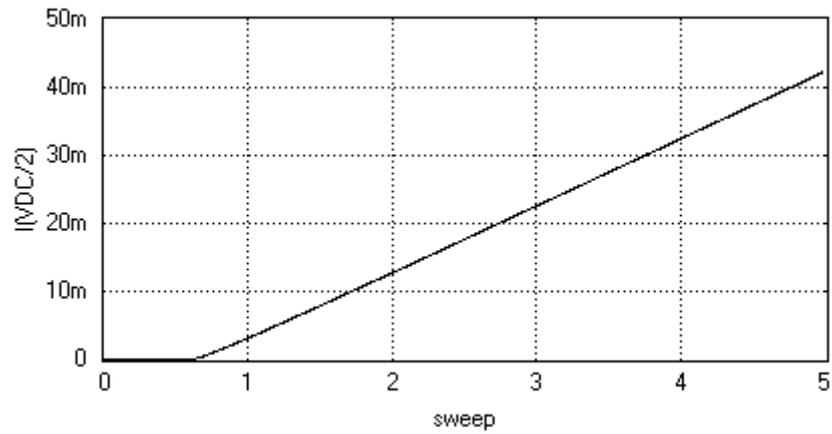
First Source : VDC/1 [V2]

Start voltage : 0V

Stop voltage : 5V

Step　　　　　: 5m

Results　　　　: Select *waveform* from drop down menu.

Click on *Accept*. Expand the *DC transfer function* and select *Waveform viewer*, Select dummy voltage source branch. Click on *Accept*. Select *DC Transfer function analysis*. Run Analysis by clicking on *Run* button.

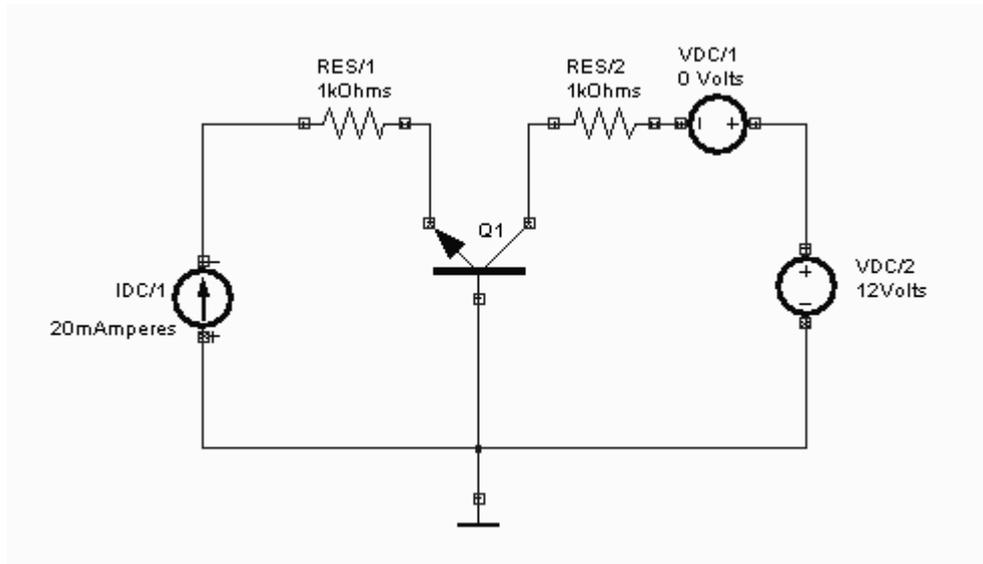**Result**

BJT OUTPUT CHARACTERISTICS – CB Configuration

Aim

To simulate the output characteristics of the BJT with common base configuration.

Components required

Transistor – BC107A (1), DC Current Source – IDC (1), Resistor – RC05 (2), DC Voltage source – VDC (2), Ground – SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The circuit is preprocessed. The VI characteristics may be obtained by performing DC Transfer function Analysis. Place the current waveform marker at the positive terminal of the dummy voltage source VDC/1 (Voltage = 0 Volts). For placing waveform markers, select *Tools → Instruments → Set Wave Form Contents → Current waveform →* Click on the required net and place the waveform marker. The sweep parameter (voltage) for input source is set in the Analysis window. The applied voltage is swept from an initial value to final value with the steps provided. To get VI characteristics, the currents corresponding to varying input voltages are noted. The VI graph is observed in the Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistors : RES/1 = 1k, RES/2 = 1k

IDC : 20mA

VDC/1 : 0V

VDC/2 : 12V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source : VDC/2 [V4]

Start voltage : -500m

Stop voltage : 10

Step : 20m

Second Source : IDC/1 [I1]

Start voltage : -1m

Stop voltage : 10m

Step : 100μ

Results : Select waveform from drop down menu.

Click on *Accept*. Expand the *DC transfer function Analysis* and select *Waveform viewer*. Select dummy voltage source branch. Click on *Accept*. Select *DC Transfer function analysis*. Run Analysis by clicking on *Run* button
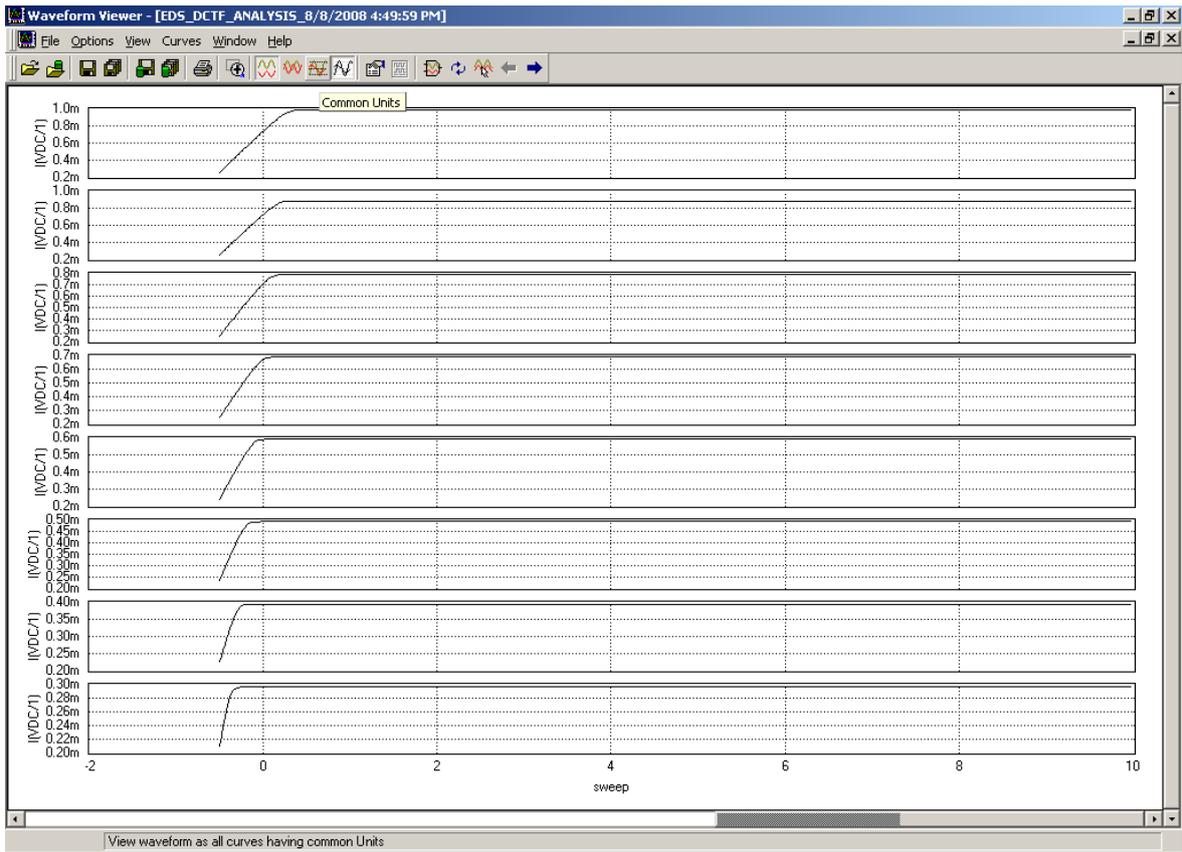
After running the analysis, Click on the *Common Units* option in the Waveform viewer.
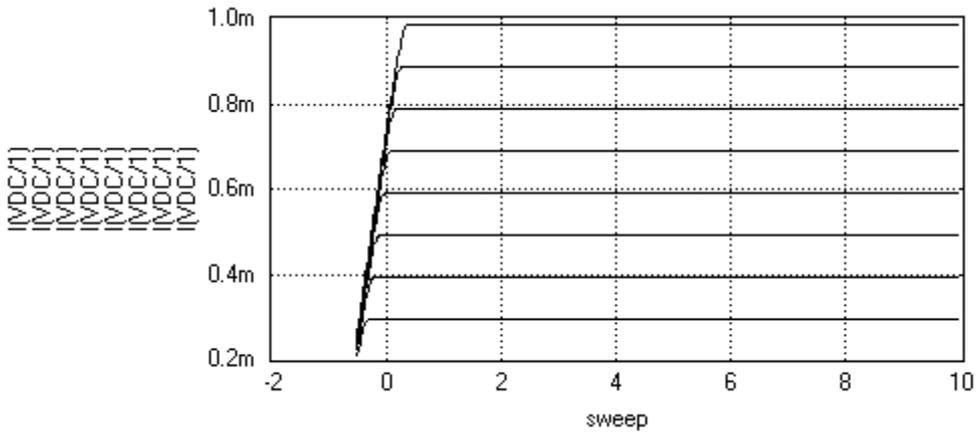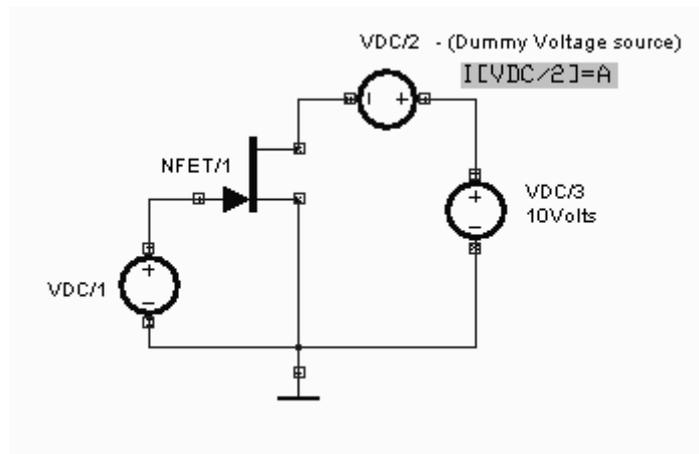
FET INPUT CHARACTERISTICS

Aim

To simulate the FET input characteristics

Components required

FET - 2N3819 (1), DC Voltage Source – VDC (3), GND – SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The current waveform marker is placed at the positive terminal of the dummy voltage source VDC/2 (Voltage = 0 Volts). The values are assigned for relevant components. The circuit is preprocessed. The DC Transfer function Analysis parameters have been set. The DC transfer function Analysis is executed and output waveform is observed in Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

      NFET   : 2N3819 (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

      VDC/1   : 1V

      VDC/2   : 0V

      VDC/3   : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.
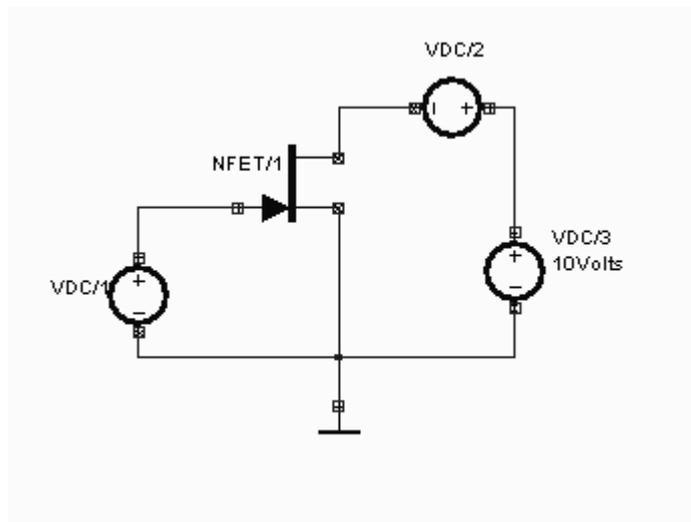
For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source	: VDC/1 [V2]
Start voltage	: -5V
Stop voltage	: 0V
Step	: 100m
Results	: Select waveform from drop down menu.

Click on *Accept*. Expand the *DC transfer function Analysis* and select *Waveform* viewer, Select dummy voltage source branch. Mark branch voltage of second voltage source (dummy voltage source) VDC/2. Click on *Accept*. Select *DC Transfer function analysis*. Run Analysis by clicking on *Run* button

Result

33

FET OUTPUT CHARACTERISTICS

Aim
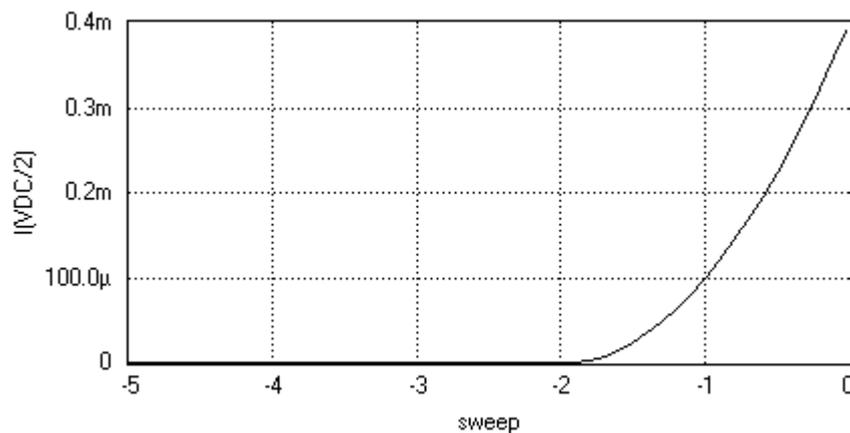
To simulate the output characteristics of an FET.

Components required

FET - 2N3819 (1), DC Voltage Source - VDC (3), Ground - SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The values are assigned for relevant components. The circuit is preprocessed. The test points and waveform markers are placed in input and output of the circuit. The DC Transfer function Analysis parameters have been set. The DC Transfer function Analysis is executed and output waveform is observed in Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

      NFET   : 2N3819 (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

      VDC/1   : 0V

      VDC/2   : 0V

      VDC/3   : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source	: VDC/3 [V4]
Start voltage	: 0V
Stop voltage	: 50V
Step	: 10V
Second Source	: VDC/1 [V2]
Start voltage	: -50V
Stop voltage	: 0V
Step	: 1V
Results	: Select *waveform* from drop down menu.

Click on *Accept*. Expand the *DC transfer function Analysis* and select *Waveform viewer*, Select dummy voltage source branch. Mark branch voltage of second voltage source (dummy voltage source) VDC/2. Click on *Accept*. Select *DC Transfer function analysis*. Run Analysis by clicking on *Run* button. Display wave form in Common Units.

Result

MOSFET INPUT CHARACTERISTICS

Aim
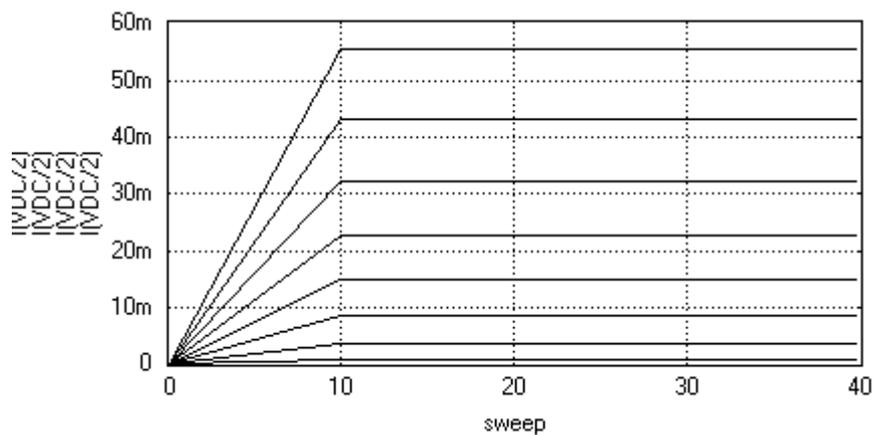
To simulate the MOSFET input characteristics.

Components Required

NMOSFET- NMESFET (1), DC Voltage Source – VDC (3), Ground - SPL0 (1)
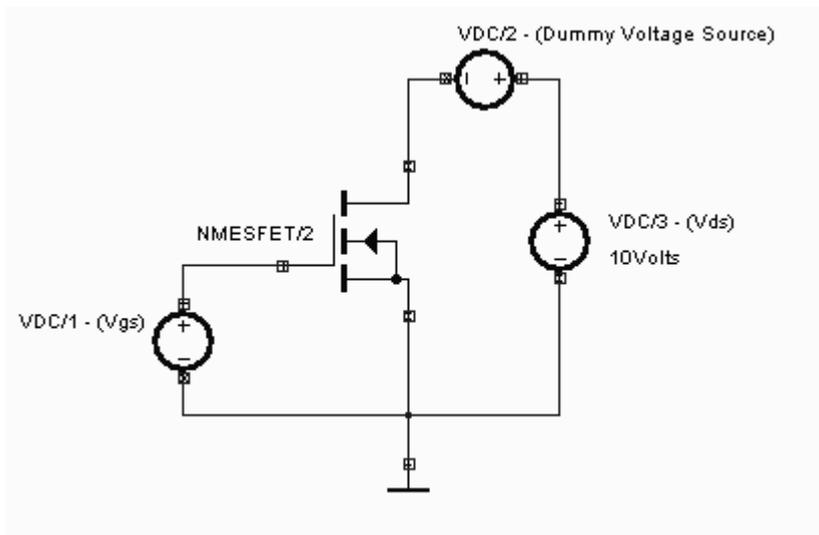
Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The values are assigned for relevant components. The circuit is preprocessed. The test points and waveform markers are placed in input and output of the circuit. DC Transfer Analysis parameters have been set. The analysis is executed and output waveform is observed in Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

NMOSFET : NMESFET (Click on the *Set up Model* of the Model Type. Click on *Accept* to get the default values assigned)

VDC/1 : 0V

VDC/2 : 0V

VDC/3 : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source	: VDC/1 [V2]
Start voltage	: -5V
Stop voltage	: 5V
Step	: 100m
Results	: Select *waveform* from drop down menu.

Click on *Accept*. Expand the *DC transfer function Analysis* and select *Waveform viewer*. Select dummy voltage source branch. Mark branch voltage of second voltage source (dummy voltage source) VDC/2. Click on Accept. Select *DC Transfer function analysis*. For plotting the *DC transfer function analysis*, the current waveform marker/plot (Vx#branch) should be taken from the dummy voltage source - VDC (additional dc voltage source included in the path of current flow having dc voltage set to 0) Run Analysis by clicking on *Run* button.
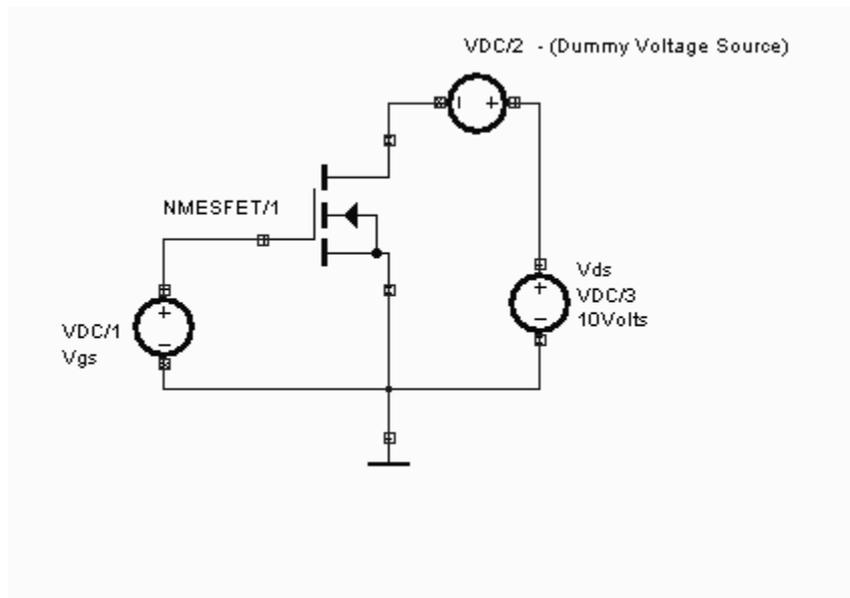
Result

MOSFET OUTPUT CHARACTERISTICS

Aim
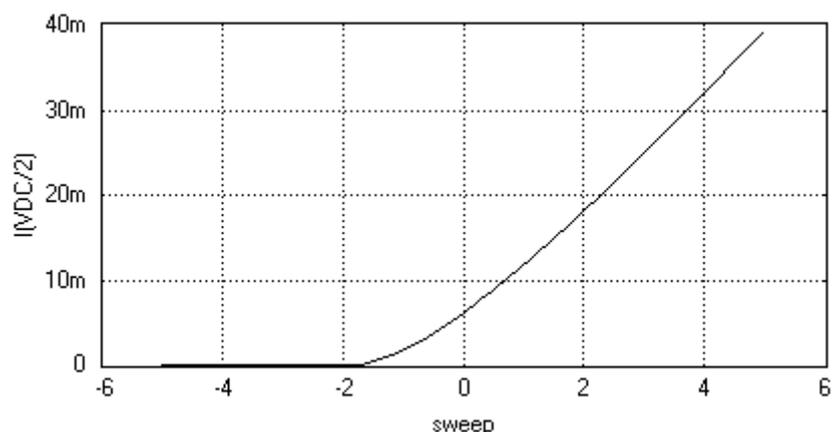
To simulate the MOSFET output characteristics.

Components required

MOSFET – NMESFET (1), DC Voltage Source – VDC (3), Ground - SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. A DC source with 0 V is placed as the dummy voltage source to obtain the current waveform. Wiring and proper net assignment has been made. The values are assigned for relevant components. The circuit is preprocessed. The test points and waveform markers are placed in input and output of the circuit. The DC Transfer function Analysis parameters have been set. The analysis is executed and output waveform is observed in Waveform Viewer.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→*
Click on the required component and change its values.

The values to be provided to the components are

   NMOSFET   : NMESFET (Click on the *Set up Model* of the *Model Type*. Click on
   Accept to get the default values assigned)

   VDC/1   : 0V

   VDC/2   : 0V

VDC/3   : 10V

Note: For input/output characteristics plot of BJT, FET, MOS etc, it is not necessary to place a waveform marker in the diagram. Its because the sweep variable of the VI parameters are being plotted. The current through a net can be obtained by enabling the Vx#branch in the waveform. Please ensure the connectivity of dummy voltage source having (0V) for obtaining the direction of current.

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

First Source	: VDC/3
Start voltage	: 0V
Stop voltage	: 15V
Step	: 100m
Second Source	: VDC/1 [V2]
Start voltage	: -1V
Stop voltage	: 1V
Step	: 1
Results	: Select *waveform* from drop down menu.

Click on *Accept*. Expand the *DC transfer function Analysis* and select *Waveform viewer*. Select dummy voltage source branch. Mark branch voltage of second voltage source (dummy voltage source) VDC/2. Click on *Accept*. Select *DC Transfer function analysis*. For plotting the DC transfer function analysis, the current waveform marker/plot (Vx#branch) should be taken from the dummy voltage source – VDC/2 (additional dc voltage source included in the path of current flow having dc voltage set to 0). Run Analysis by clicking on *Run* button.
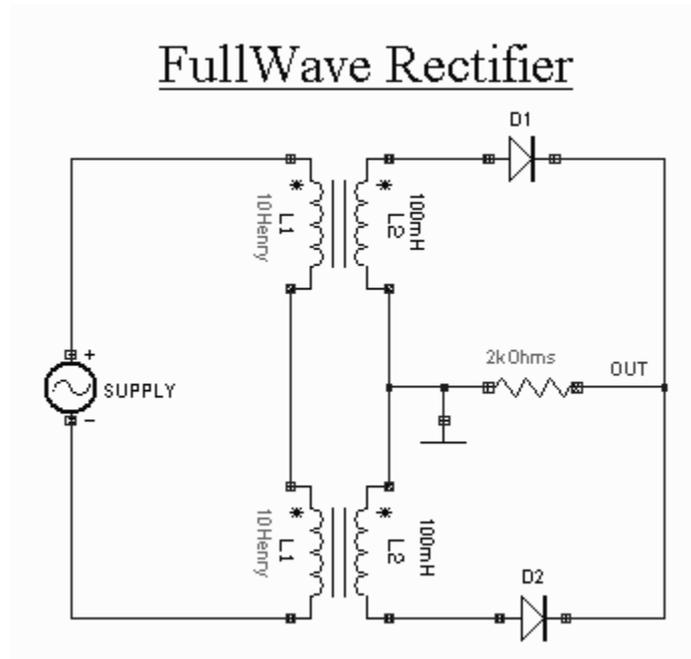
Result
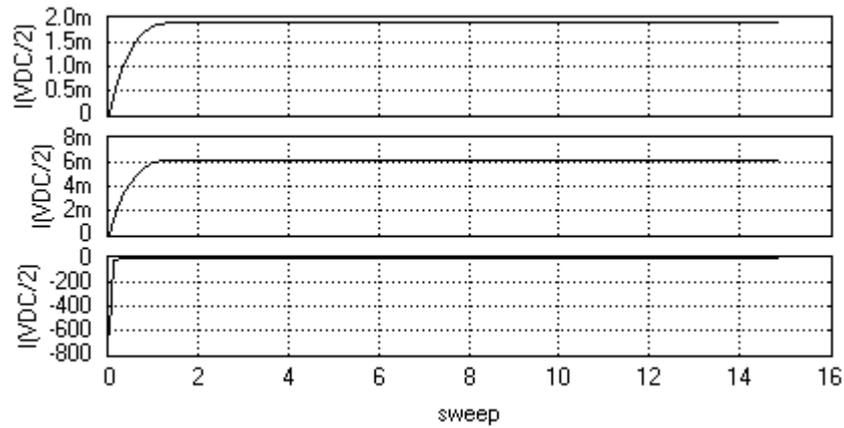
FULL WAVE RECTIFIER

Aim

To simulate a Full Wave Rectifier circuit.

Components required

Transformer – TRANSFORMER (2), Resistor – RC05 (1), Diode – 1N4007 (1), AC Voltage Source – VGEN (1), Ground - SPL0 (1)

Circuit Diagram

Theory

A Full Wave Rectifier is a circuit, which converts an ac voltage into a pulsating dc voltage using both half cycles of the applied ac voltage. It uses two diodes of which one conducts during one half cycle while the other conducts during the other half cycle of the applied ac voltage.

40

During the positive half cycle of the input voltage, diode D1 becomes forward biased and D2 becomes reverse biased. Hence D1 conducts and D2 remains OFF. The load current flows through D1 and the voltage drop across RL will be equal to the input voltage.

During the negative half cycle of the input voltage, diode D1 becomes reverse biased and D2 becomes forward biased. Hence D1 remains OFF and D2 conducts. The load current flows through D2 and the voltage drop across RL will be equal to the input voltage.

Rectifier with Filter

The output of the Full Wave Rectifier contains both ac and dc components. A majority of the applications, which cannot tolerate a high value ripple, necessitates further processing of the rectified output. The undesirable ac components i.e. the ripple, can be minimized using filters.

FullWave Rectifier with Filter
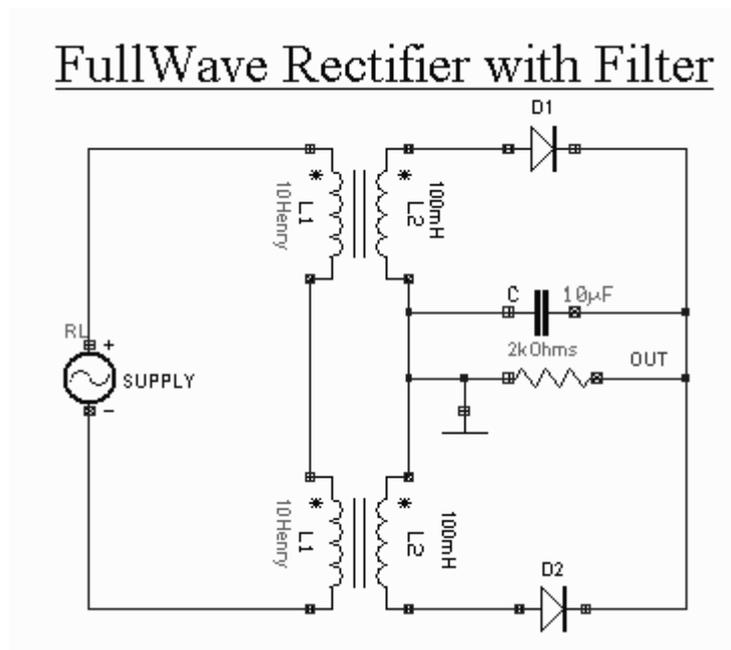
The output of the rectifier is fed as input to the filter. The output of the filter is not a perfect dc, but it also contains small ac components.

Some important filters are

1. Inductor Filter
2. Capacitor Filter
3. LC Filter
4. CLC or ☐ Filter

Inductor Filter

The figure shows an inductor filter. When the output of the rectifier passes through an inductor, it blocks the ac component and allows only the dc component to reach the load.
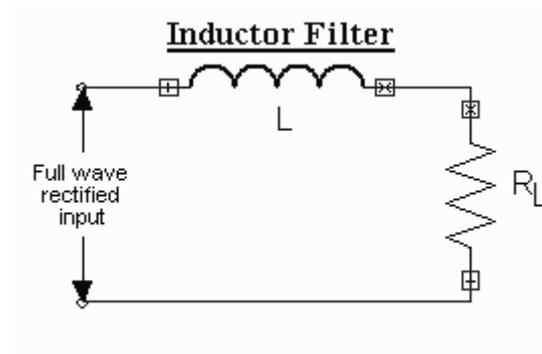
Inductor Filter

Ripple factor of the inductor filter is given by $\gamma = \dfrac{R_L}{3\sqrt{2}\ \omega L}$.

The above equation shows that ripple will decrease when L is increased and R_L is decreased. Thus the inductor filter is more effective only when the load current is high (small R_L). The larger value of the inductor can reduce the ripple and at the same time the output dc voltage will be lowered as the inductor has a higher dc resistance.

The operation of the inductor filter depends on its property to oppose any change of current passing through it. To analyze this filter for full wave, the Fourier series can be written as

$$V_o = \frac{2V_m}{\pi} - \frac{4V_m}{\pi}\left[\frac{1}{3}\cos 2\,\omega t + \frac{1}{15}\cos 4\,\omega t + \frac{1}{35}\cos 6\,\omega t + \cdots\right]$$

The dc component is $\dfrac{2V_m}{\pi}$.

Assuming the third and higher terms contribute little output, the output voltage is

$$V_o = \frac{2V_m}{\pi} - \frac{4V_m}{3\pi}\cos 2\,\omega t$$

The diode, choke and transformer resistances can be neglected since they are very small compared with R_L. Therefore the dc component of current $I_m = \dfrac{V_m}{R_L}$

The impedance of series combination of L and R_L at 2□ is

$$Z = \sqrt{R_L^2 + (2\,\omega L)^2} = \sqrt{R_L^2 + 4\,\omega^2 L^2}$$

Therefore for the ac component,

$$I_m = \frac{V_m}{\sqrt{R_L^2 + 4\,\omega^2 L^2}}$$

Therefore, the resulting current i is given by,

$$i = \frac{2V_m}{\pi R_L} - \frac{4V_m}{3\pi} \frac{\cos(2\omega t - \varphi)}{\sqrt{R_L^2 + 4\omega^2 L^2}} \quad where \quad \varphi = \tan^{-1}\left(\frac{2\omega L}{R_L}\right)$$

The ripple factor which can be defined as the ratio of the rms value of the ripple to the dc value of the wave, is

$$\gamma = \frac{\dfrac{4V_m}{3\pi\sqrt{2}\sqrt{R_L^2 + 4\omega^2 L^2}}}{\dfrac{2V_m}{\pi R_L}} = \frac{2}{3\sqrt{2}} \frac{1}{\sqrt{1 + \dfrac{4\omega^2 L^2}{R_L^2}}}$$

If $\dfrac{4\omega^2 L^2}{R_L^2} \gg 1$, then a simplified expression for γ is

$$\gamma = \frac{R_L}{3\sqrt{2}\,\omega L}$$

In case, the load resistance is infinity i.e., the output is an open circuit, then the ripple factor is

$$\gamma = \frac{2}{3\sqrt{2}} = 0.471$$

.

This is slightly less than the value of 0.482. The difference being attributable to the omission of higher harmonics as mentioned. It is clear that the inductor filter should only be used where R_L is consistently small.

 Capacitor Filter

-
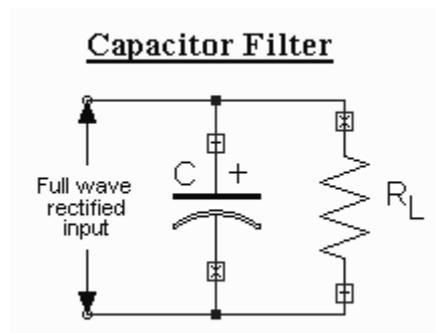
Capacitor Filter

 A capacitor filter connected directly across the load is shown above. The property of a capacitor is that it allows ac component and blocks dc component. The operation of the capacitor filter is to short the ripple to ground but leave the dc to appear at output when it is connected across the pulsating dc voltage.

 During the positive half cycle, the capacitor charges upto the peak vale of the transformer secondary voltage, V_m and will try to maintain this value as the full wave input drops to zero.

Capacitor will discharge through R_L slowly until the transformer secondary voltage again increase to a value greater than the capacitor voltage. The diode conducts for a period, which depends on the capacitor voltage. The diode will conduct when the transformer secondary voltage becomes more than the diode voltage. This is called the cut in voltage. The diode stops conducting when the transformer voltage becomes less than the diode voltage. This is called cut out voltage.

Referring to the figure below, with slight approximation the ripple voltage can be assumed as triangular. From the cut-in point to the cut-out point, whatever charge the capacitor acquires is equal to the charge the capacitor has lost during the period of non-conduction, i.e., from cut-out point to the next cut-in point.

$$\text{The charge it has acquired} = V_{rp-p} \times C$$

$$\text{The charge it has lost} = I_{d.c.} \times T_2$$

Ripple Voltage Triangular Waveform
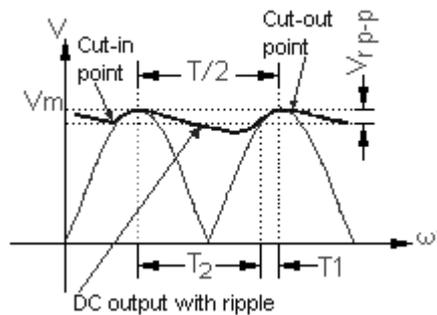
$$\therefore V_{rp-p} \times C = I_{d.c.} \times T_2$$

If the value of the capacitor is fairly large, or the value of the load resistance is very large, then it can be assumed that the time T_2 is equal to half the periodic time of the waveform.

$$T_2 = \frac{T}{2} = \frac{1}{2f}, \quad then \quad V_{rp-p} = \frac{I_{d.c.}}{2fC}$$

From the above assumptions, the ripple waveform will be triangular and its rms value is given by

$$V_{r\,rms} = \frac{V_{rp-p}}{2\sqrt{3}}$$

$$V_{r\,rms} = \frac{I_{d.c.}}{4\sqrt{3}f\,C}$$

$$= \frac{V_{d.c.}}{4\sqrt{3}f\,C\,R_L}, \quad \because I_{d.c.} = \frac{V_{d.c.}}{R_L}$$

$$\therefore Ripple, \gamma = \frac{V_{r\,ms}}{V_{d.c.}} = \frac{1}{4\sqrt{3}f\,C\,R_L}, \quad \because I_{d.c.} = \frac{V_{d.c.}}{R_L}$$

The ripple may be decreased by increasing C or R_L (both) with a resulting increase in the dc. output voltage.

 LC Filter: - The ripple factor is directly proportional to the load resistance R_L in the inductor filter and inversely proportional to R_L in the capacitor filter. Therefore if these two filters are combined as LC filter or L section filter as shown in figure the ripple factor will be independent of R_L.

LC Filter
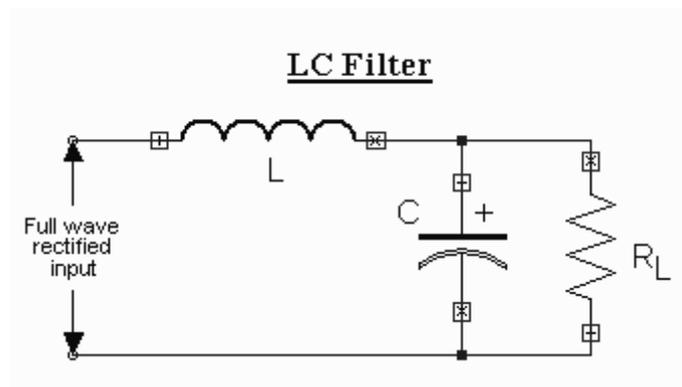
If the value of inductance is increased it will increase the time of conduction. At some critical value of inductance, one diode, either D1 or D2 will always conducting.

From Fourier series, the output voltage can be expressed as

$$V_o = \frac{2V_m}{\pi} - \frac{4V_m}{3\pi}\cos 2\,\omega t$$

The dc output voltage, $\quad V_{dc} = \frac{2V_m}{\pi}$

$$\therefore V_{r\,ms} = I_{rms} \cdot X_C = \frac{\sqrt{2}}{3} \cdot V_{dc} \cdot \frac{X_C}{X_L}$$

The ripple factor $\quad \gamma = \frac{V_{r\,ms}}{V_{dc}} = \frac{\sqrt{2}}{3} \cdot \frac{X_C}{X_L}$

$$= \frac{\sqrt{2}}{3} \cdot \frac{1}{4\,\omega^2 CL} \quad \because X_C = \frac{1}{2\,\omega C} \quad \& \quad X_L = 2\,\omega L$$
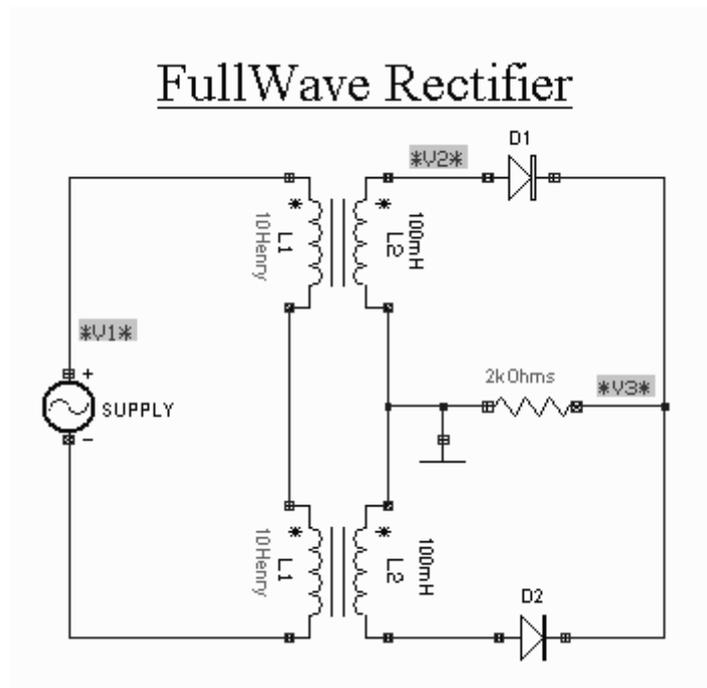
<u>CLC or π Filter</u>
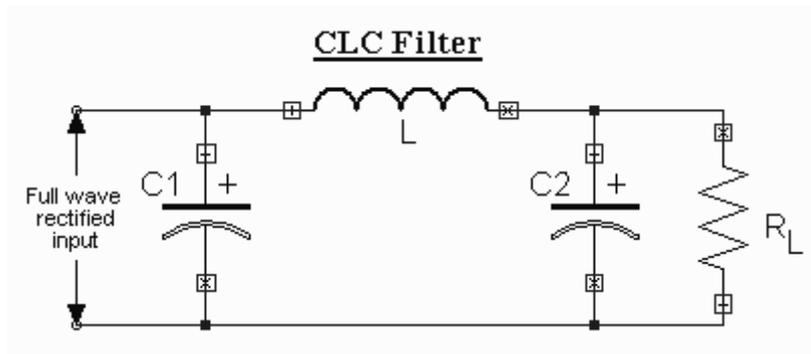
The above figure shows CLC or □ type filter, which basically consists of a capacitor filter, followed by LC section. This filter offers a fairly smooth output and is characterized by highly peaked diode currents and poor regulation. As in L section filter the analysis is obtained as follows.

$$\gamma = \sqrt{2}\frac{X_{C1}}{R_L}\cdot\frac{X_{C2}}{X_L}$$

Procedure

The circuit diagram is drawn by loading components from the library. Wiring and proper net assignment has been made. The values are assigned for relevant components.

The circuit is preprocessed. The test points and waveform markers are placed in input and output of the circuit. GND net is set as reference net. The Transient Analysis parameters have been set. The Transient Analysis is executed and output waveform is observed in Waveform Viewer.

Result

The output waveform for Full Wave Rectifier with filter and without filter may be observed in the waveform viewer.

BRIDGE RECTIFIER

Aim

To simulate a Bridge Rectifier circuit.

Components Required

Resistor - RC05 (1), Transformer – TRANSFORMER (1), Diode - 1N4007 (4), Ac voltage source - VGEN (1), Ground – SPL0 (1)

Theory

The Bridge rectifier is a circuit, which converts an ac voltage to dc voltage using both half cycles of the input ac voltage. The Bridge rectifier circuit is shown in the figure. The circuit has four diodes connected to form a bridge. The ac input voltage is applied to the diagonally opposite ends of the bridge. The load resistance is connected between the other two ends of the bridge.

For the positive half cycle of the input ac voltage, diodes D1 and D3 conduct, whereas diodes D2 and D4 remain in the OFF state. The conducting diodes will be in series with the load resistance R_L and hence the load current flows through R_L.

For the negative half cycle of the input ac voltage, diodes D2 and D4 conduct whereas, D1 and D3 remain OFF. The conducting diodes D2 and D4 will be in series with the load resistance R_L and hence the current flows through R_L in the same direction as in the previous half cycle. Thus a bi-directional wave is converted into a unidirectional wave.
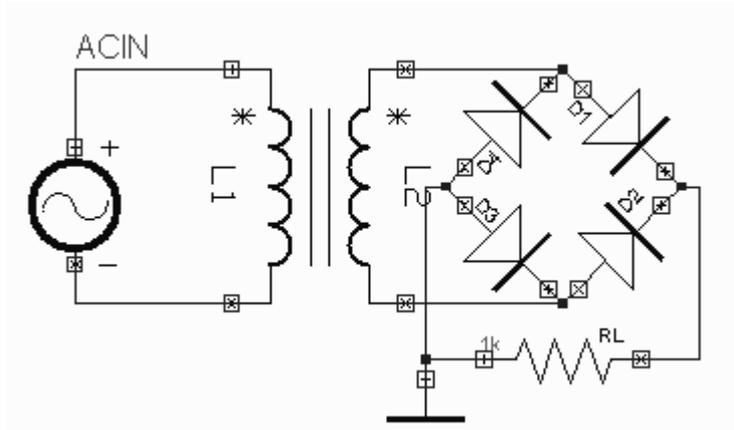
Peak Inverse Voltage

Peak inverse voltage represents the maximum voltage that the non- conducting diode must withstand. At the instance the secondary voltage reaches its positive peak value, Vm the diodes D1 and D3 are conducting, where as D2 and D4 are reverse biased and are non-conducting. The conducting diodes D1 and D3 have almost zero resistance. Thus the entire voltage V_m appears across the load resistor R_L. The reverse voltage across the non-conducting diodes D2 (D4) is also V_m. Thus for a Bridge rectifier the peak inverse voltage is given by

$$PIV = V_m$$

Ripple Factor

The ripple factor for a Full Wave Rectifier is given by

$$\gamma = \sqrt{\left(\frac{V_{rms}}{V_{dc}}\right)^2 - 1}$$

The average voltage or the dc voltage available across the load resistance is

$$V_{dc} = \frac{1}{\pi} \int_0^\pi V_m \sin \omega t \, d(\omega t)$$

$$= \frac{V_m}{\pi} \left[- \cos \omega t\right]_0^\pi = \frac{2V_m}{\pi}$$

$$I_{dc} = \frac{V_{dc}}{R_L} = \frac{2V_m}{\pi R_L} = \frac{2I_m}{\pi} \quad and \quad I_{rms} = \frac{I_m}{\sqrt{2}}$$

RMS value of the voltage at the load resistance is

$$V_{rms} = \left[\frac{1}{\pi} \int_0^\pi V_m^2 \sin^2 \omega t \, d(\omega t)\right]^{1/2} = \frac{V_m}{\sqrt{2}}$$

$$\therefore \gamma = \sqrt{\left(\frac{V_m/2}{2V_m/\pi}\right)^2 - 1} = \sqrt{\left(\frac{\pi}{8}\right)^2 - 1} = \underline{0.482}$$

Efficiency

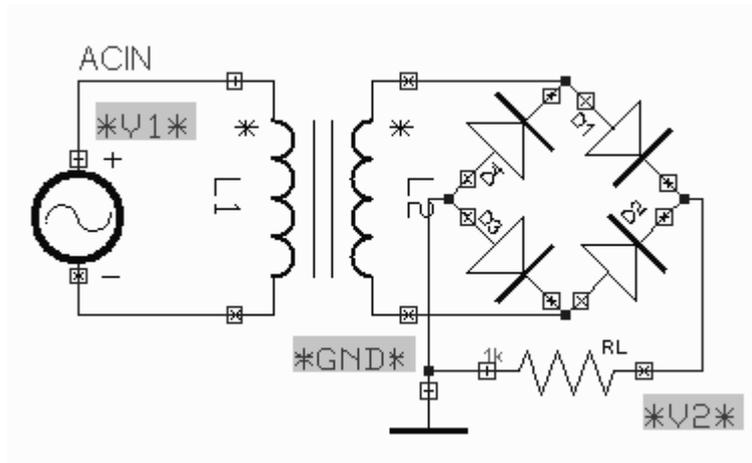Efficiency, η is the ratio of the dc output power to ac input power

$$\eta = \frac{dc\,output\,power}{ac\,input\,power} = \frac{P_{dc}}{P_{ac}}$$

$$\frac{V_{dc}^2/R_L}{V_{rms}^2/R_L} = \frac{\left[2V_m/\pi\right]^2}{\left[V_m/\sqrt{2}\right]^2} = \frac{8}{\pi^2} = 0.812 = \underline{81.2\%}$$

The maximum efficiency of a Full Wave Rectifier is 81.2%.

Procedure

OPUSER -> *Schematic Editor:* The circuit diagram is drawn by loading components from the library. Wiring and proper net assignment has been made. The values are assigned for relevant components.

The circuit is preprocessed. The test points and waveform markers are placed in input and output of the circuit. GND net is set as reference net. The Transient Analysis parameters have been set. The Transient Analysis is executed and output waveform is observed in Waveform Viewer.

Result

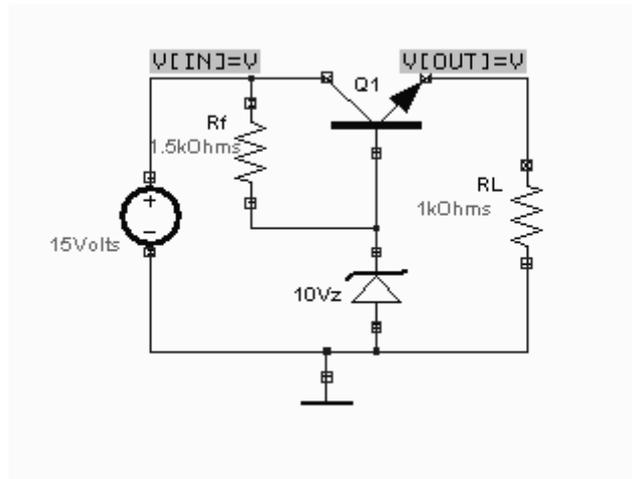The output waveform may be observed in the waveform viewer.

SERIES VOLTAGE REGULATOR

Aim

To simulate a series voltage regulator circuit.

Components Required

Transistor- BC107A (1), Resistor- RC05 (2), Zener-1N3016A (1), DC voltage source- VDC (1), Ground- SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking Simulation → *Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Zener diode : Breakdown Voltage = 10V

Resistors : 1k (1), 1.5k (1)

VDC/1 : 15V

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view. Enter following values:

  Step   : 1μ

  Final time   : 10m

  Results   : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.
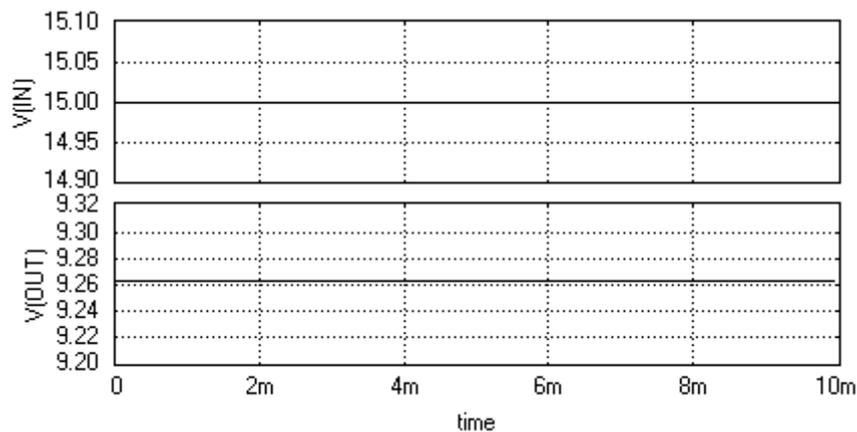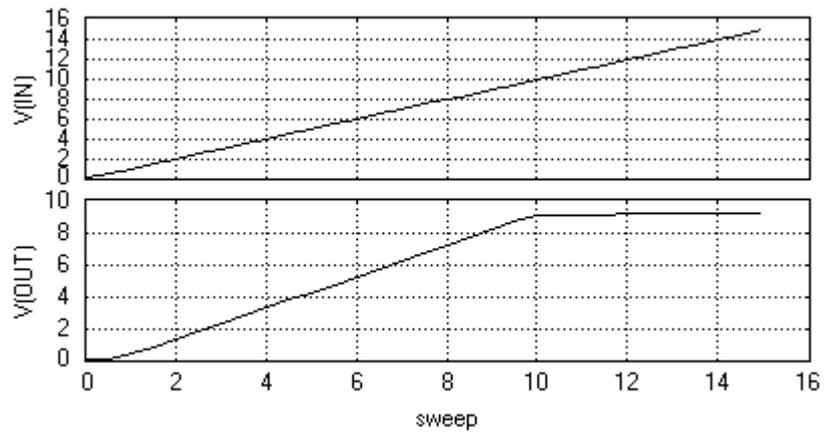
For setting up *DC Transfer function Analysis*, select *Simulation* menu, choose *Analysis*. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values:

  First Source   : VDC/1 [V5]

  Start Voltage   : 0V

  Stop Voltage   : 15V

  Step   : 500m

  Results   : Select waveform from drop down menu.

Click on *Accept*. Expand the transient tree view and select waveform viewer, click on *As marked*. Click on *Accept*. Run Analysis by clicking on *Run* button

Result

a) Transient Analysis

b) DC Transfer function Analysis
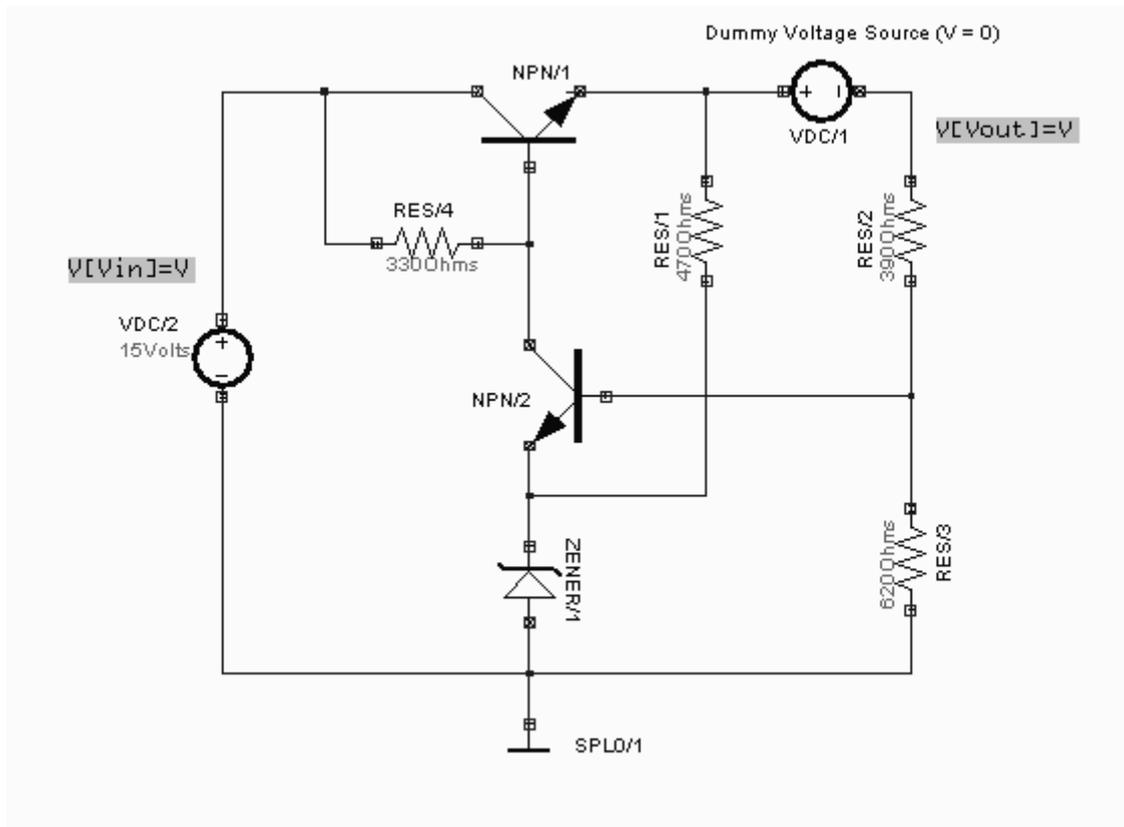
SERIES VOLTAGE REGULATOR

Aim

To simulate the series voltage regulator.

Components required

Resistor - RC05 (4), Transistor – BC107A (1), 2N3055 (1), Zener diode- 1N3016A (1), DC

voltage source – VDC (2), Ground – SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation → Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

      Transistors   : 2N3055, BC107A (Click on the *Set up Model* of the Model Type.

      Click on Accept to get the default values assigned)

      Resistors   : 330 Ohm, 470 Ohm, 620 Ohm, 390 Ohm

      Zener diode   : 1N3016A, Break down voltage = 5.6V

      VDC/2   : 15V

      VDC/1   : 0V

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select Transient analysis from tree view. Enter following values:

Step   : 10μ

Final time   : 10m

Results   : Select Waveform from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.

For setting up *DC Transfer function Analysis*, select *Simulation* menu, choose Analysis. In the window pops up, select *DC Transfer function analysis* from tree view. Enter following values

First Source   : VDC/2

Start Voltage   : 5V

Stop Voltage   : 25V

Step   : 500m

Results   : Select *waveform* from drop down menu.

Click on *Accept*. Expand the transient tree view and select *waveform viewer*, click on *As marked*. Click on *Accept*. Run Analysis by clicking on *Run* button

Result
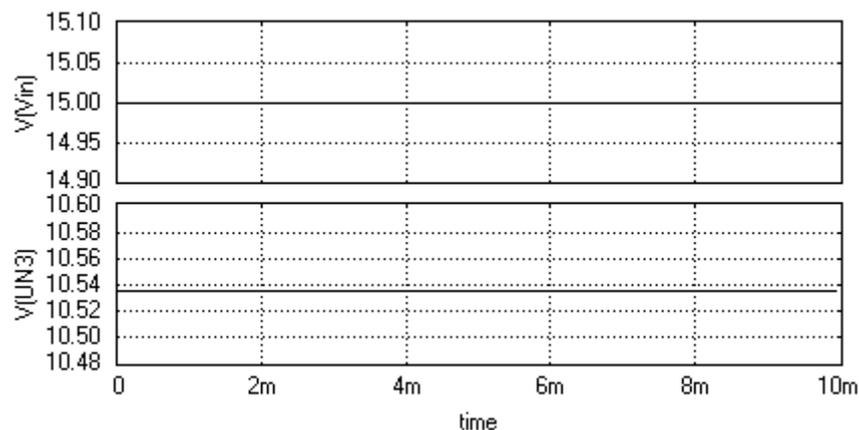
Transient Analysis
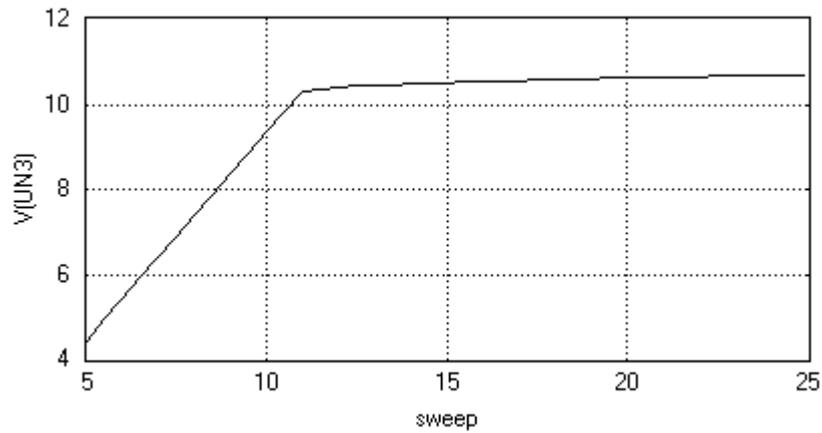
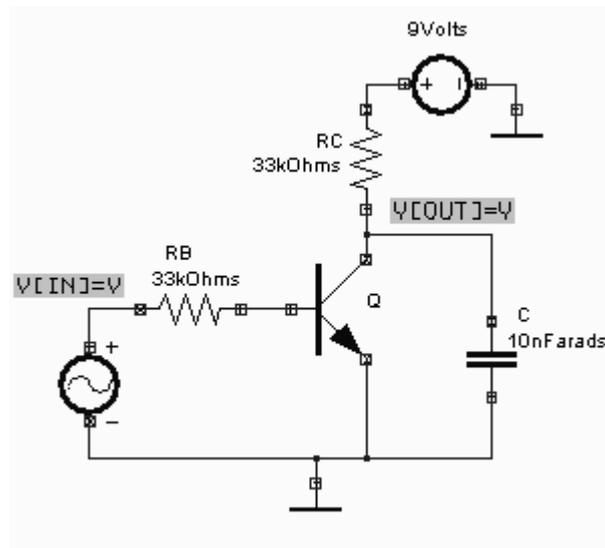DC Transfer Characteristics

SWEEP GENERATOR

Aim

To simulate a transistor sweep generator circuit.

Components Required

Transistor- BC107A (1), Resistor - RC05 (2), Capacitor - CC0805 (1), Voltage Generator - VGEN (1), Voltage source - VDC (1), Ground-SPL0 (1)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation → Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →*

55

Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select Tools → Components → Component properties→ Change simulation parameters→

Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistors : 33k (2)

Capacitors : 10nF (1)

VDC/1 : 9V

VGEN : Select the source function as PULSE and set its Properties as follows

V1 (initial value) : 0V

V2 (final value) : 5V

PW (Pulse width) : 0.5m

PER (Period) : 1m (All other values are zero)

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view.
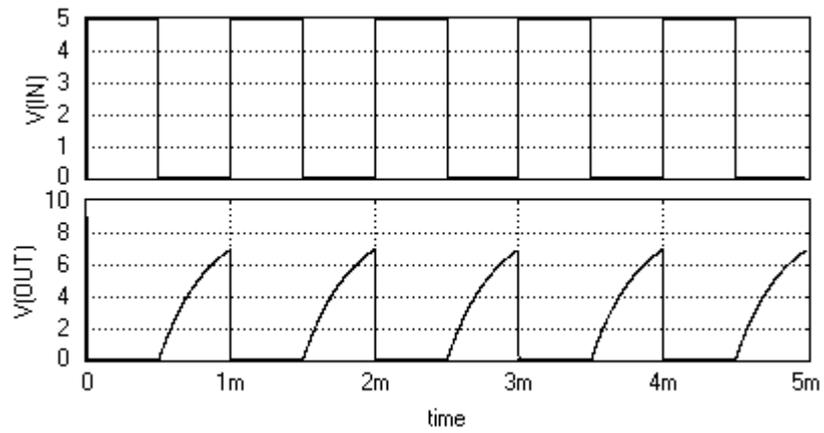
Enter following values:

Step : 1μ

Final time : 5m

Max. Step : 1μ

Results : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.
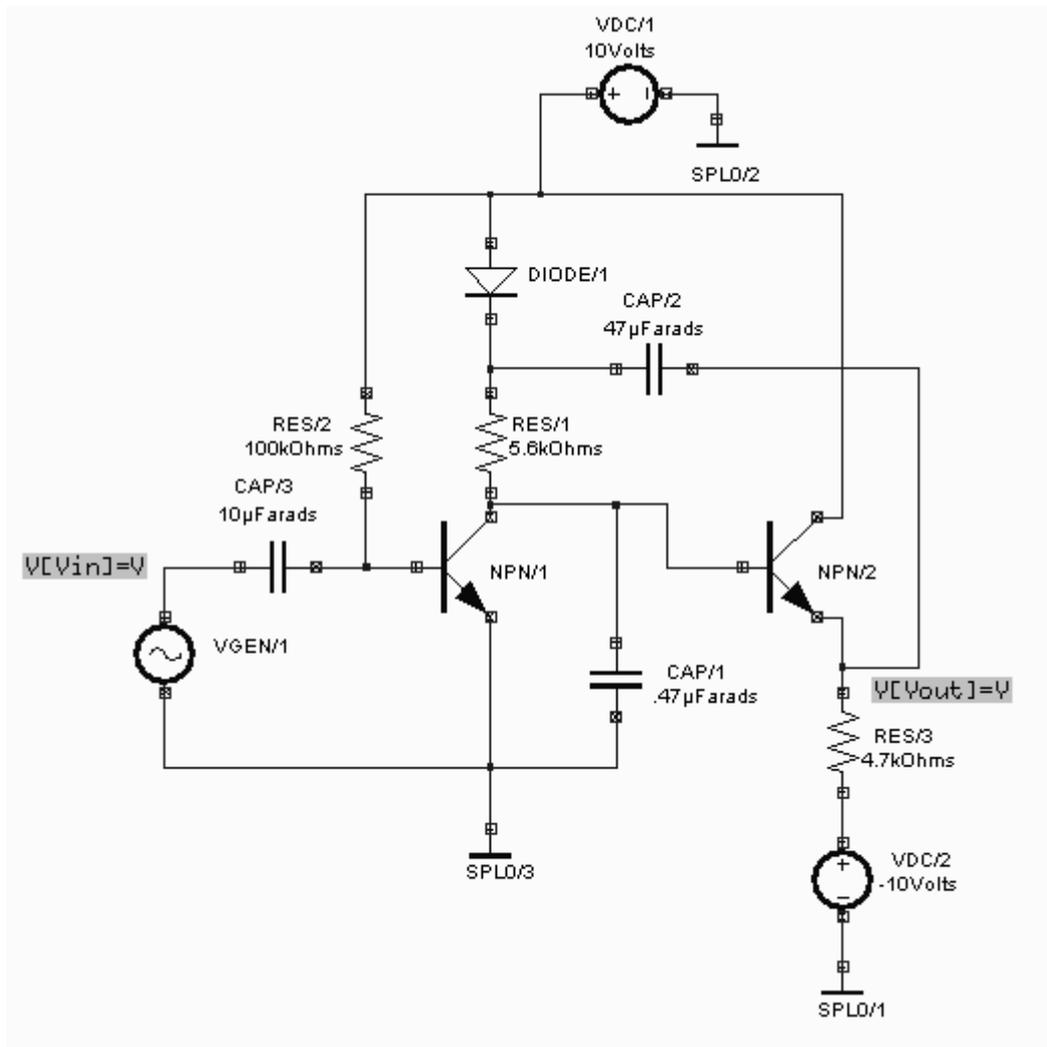
Result

BOOTSTRAP SWEEP CIRCUIT

Aim

To simulate a bootstrap sweep circuit.

Components Required

Transistor- BC107A (2), Diode – 1N4001 (1), Resistor - RC05 (3), Capacitor - CC0805 (3),

AC Voltage Generator - VGEN (1), DC Voltage source - VDC (2), Ground-SPL0 (3)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation →Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

Transistor   : BC107A (2) (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Diode   : 1N4001 (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistors   : 100k, 5.6k, 4.7k

Capacitors   : 10μF, 47μF, 0.47μF

VDC/1   : 10V

VDC/2   : -10V

VGEN   : Select the *source function* as PULSE and set its *Properties* as follows

        V1 (initial value)   : -5V

        V2 (final value)   : 5V

        PW (Pulse width)   : 0.5m

        PER (Period)   : 1m (All other values are zero)

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view. Enter following values:
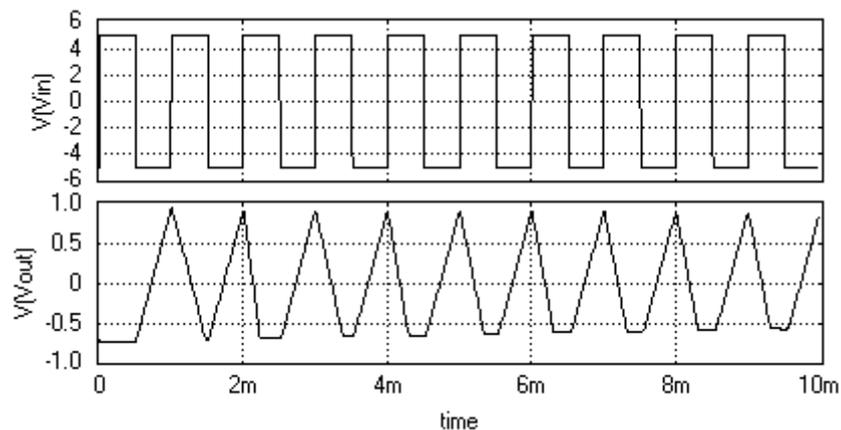
        Step   : 10μ

        Final time   : 10m

        Results   : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.
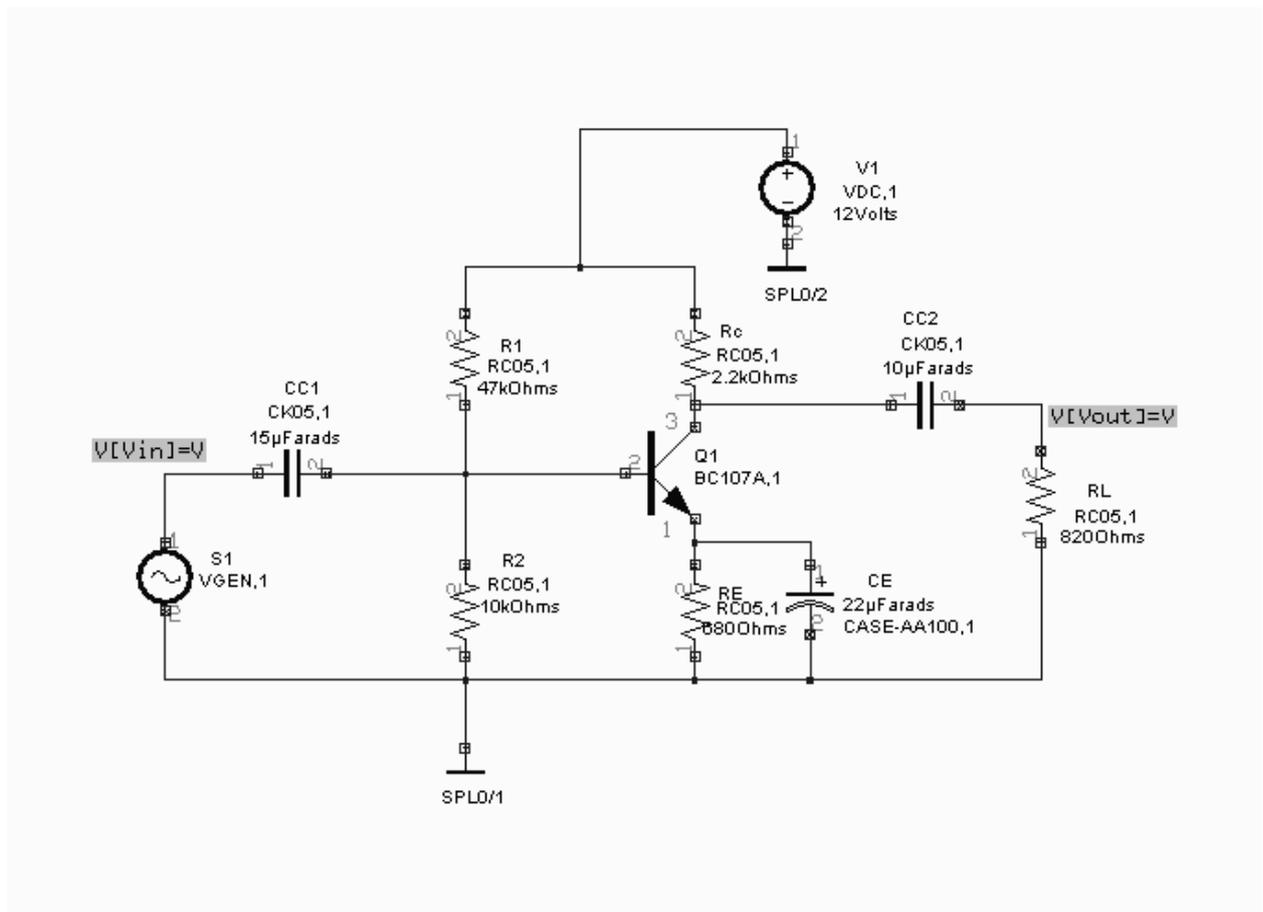
Result

RC COUPLED AMPLIFIER

Aim

To simulate an RC coupled amplifier.

Components required

Transistor – BC107A (1), Resistor – RC05 (5), Capacitor – CK05 (2), CASE-AA100 (1), AC Voltage Source – VGEN (1), DC Voltage Source – VDC (1), Ground – SPL0 (2)

Circuit Diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → Set Wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker to view the waveform.

Preprocess

Preprocess allows the simulator to recognize components and their simulation models used in the circuit. Preprocess must be done before performing analysis, or after any update to the circuit schematic. To preprocess, select *Simulation → Preprocess*, check and close the Info window that pops up.
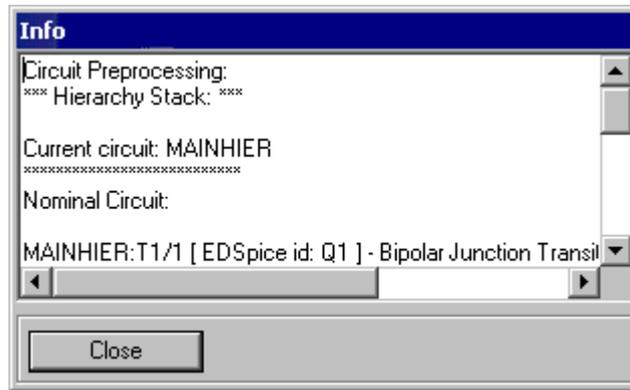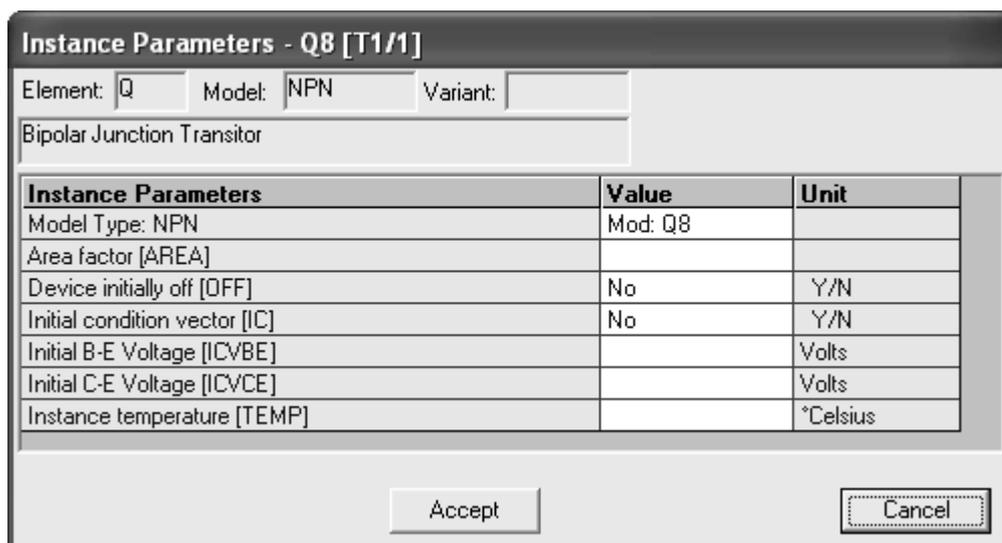
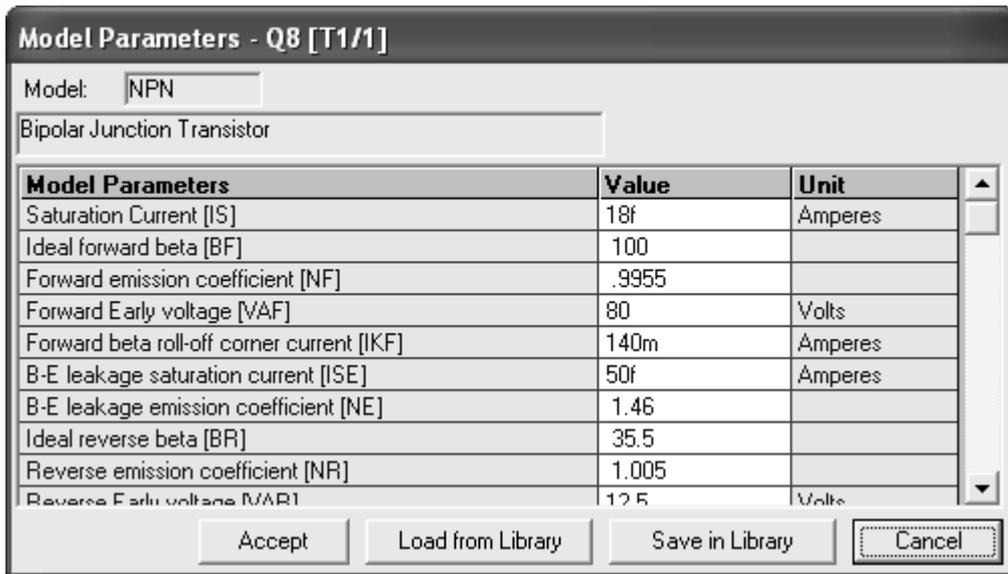

To proceed with the simulation, the steps are as follows.

Select *Tools → Components → Component properties → Change simulation parameters →* Click on the required component and change its values.

The values to be provided to the components are

Transistor : BC107A (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Model Parameters - Q8 [T1/1]

Model: NPN

Bipolar Junction Transistor

Model Parameters	Value	Unit
Saturation Current [IS]	18f	Amperes
Ideal forward beta [BF]	100	
Forward emission coefficient [NF]	.9955	
Forward Early voltage [VAF]	80	Volts
Forward beta roll-off corner current [IKF]	140m	Amperes
B-E leakage saturation current [ISE]	50f	Amperes
B-E leakage emission coefficient [NE]	1.46	
Ideal reverse beta [BR]	35.5	
Reverse emission coefficient [NR]	1.005	
Reverse Early voltage [VAR]	12.5	Volts

Accept   Load from Library   Save in Library   Cancel

Resistors : $R_1 = 47k$, $R_2 = 10k$, $R_C = 2.2k$, $R_E = 680$ Ohm, $R_L = 820$ Ohm

Capacitors : $C_{C1} = 15\mu F$, $C_{C2} = 10\mu F$, $C_E = 22\mu F$

VDC/1 (+Vcc) : 10V

VGEN : Select the *source function* as SIN and set its *Properties* as follows

VA (Amplitude) : 10mV

FREQ (Frequency) : 10kHz

Analysis

Various analysis can be performed on the circuit to study the circuit behavior

Transient Analysis

Transient analysis is used to view input and output with respect to time.

1. Select *Simulation → Analysis*.

2. Select *Transient Analysis* from the tree view.

3. Expand *Transient Analysis* and select *Waveform Viewer*.

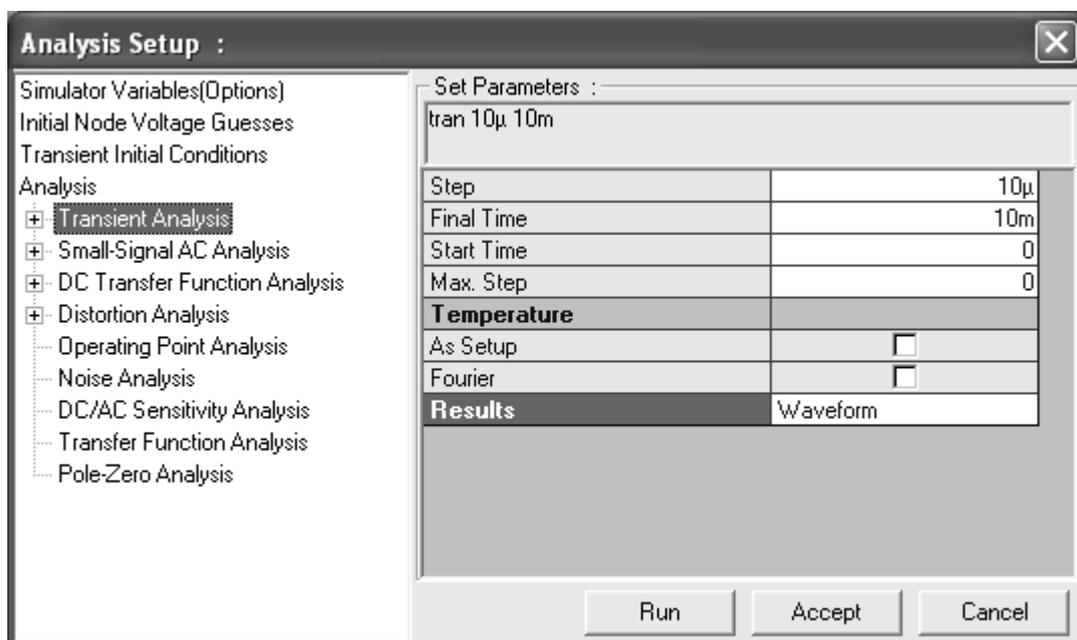4. Click on the *As Marked* button and then on *Accept*.
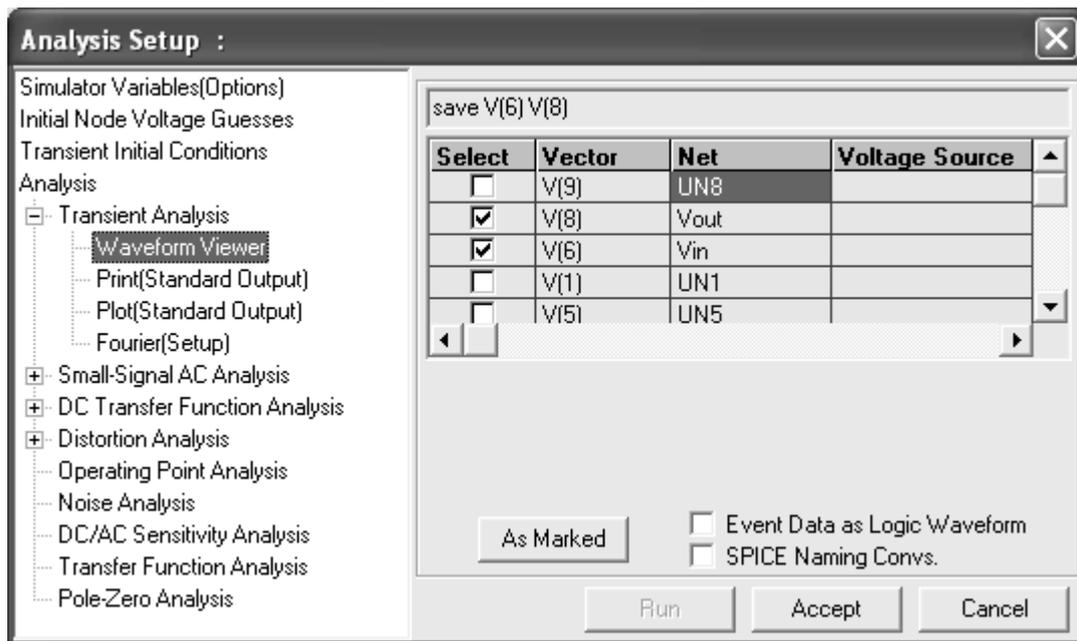
5. Click on *Transient Analysis*

6. Enter the values as

    Step   : 10µ

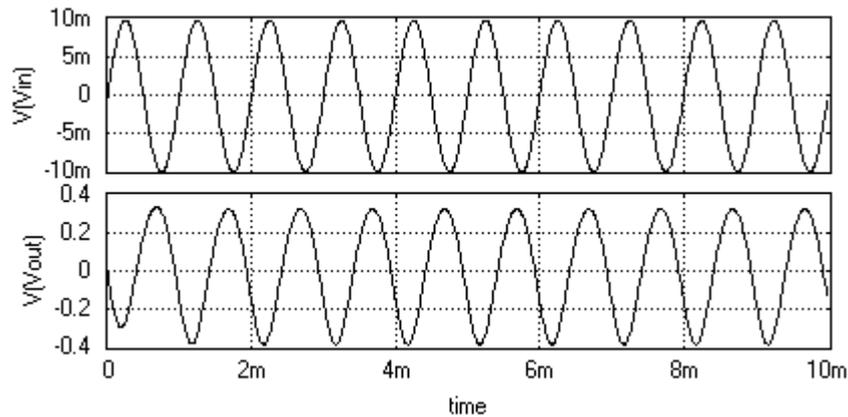   Final time   : 10m

    Start time   : 0

7. Select *Waveform* from the *Results* field for displaying the output
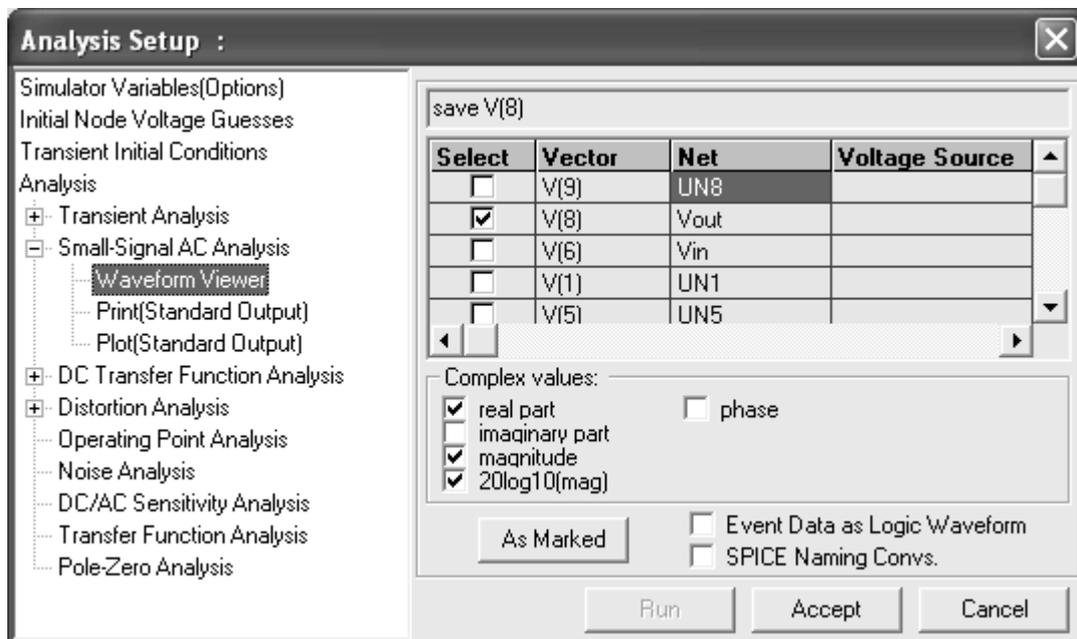
8. Click *Accept* button after entering the values.

9. Click *Run* button to start simulation.

Small Signal AC Analysis

This analysis is used to obtain the small signal AC behaviour of the circuit.

1. Select *Simulation → Analysis*.

2. Select *Small Signal AC Analysis* from the tree view.

3. Expand *Small Signal AC Analysis* and select *Waveform Viewer*.

4. Click on *As Marked* button.

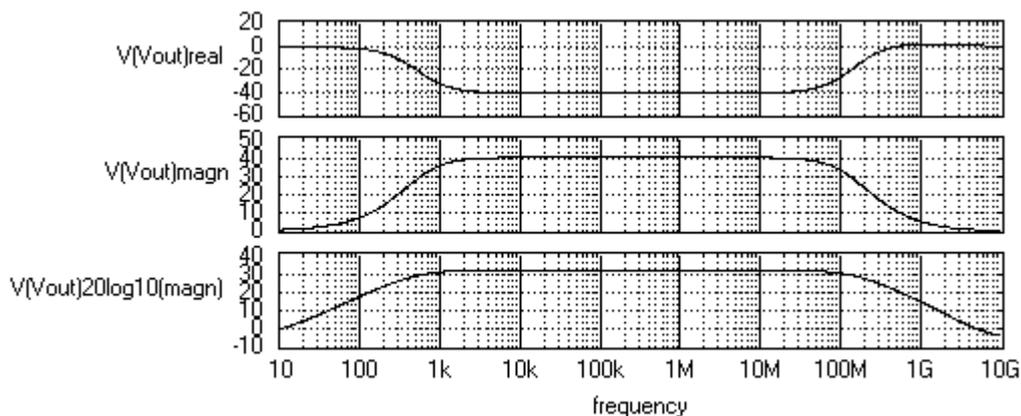5. Select the required Complex values and then click on *Accept*.

6. Enter the values for the *Small-Signal AC Analysis* as

        Variation   : Decade
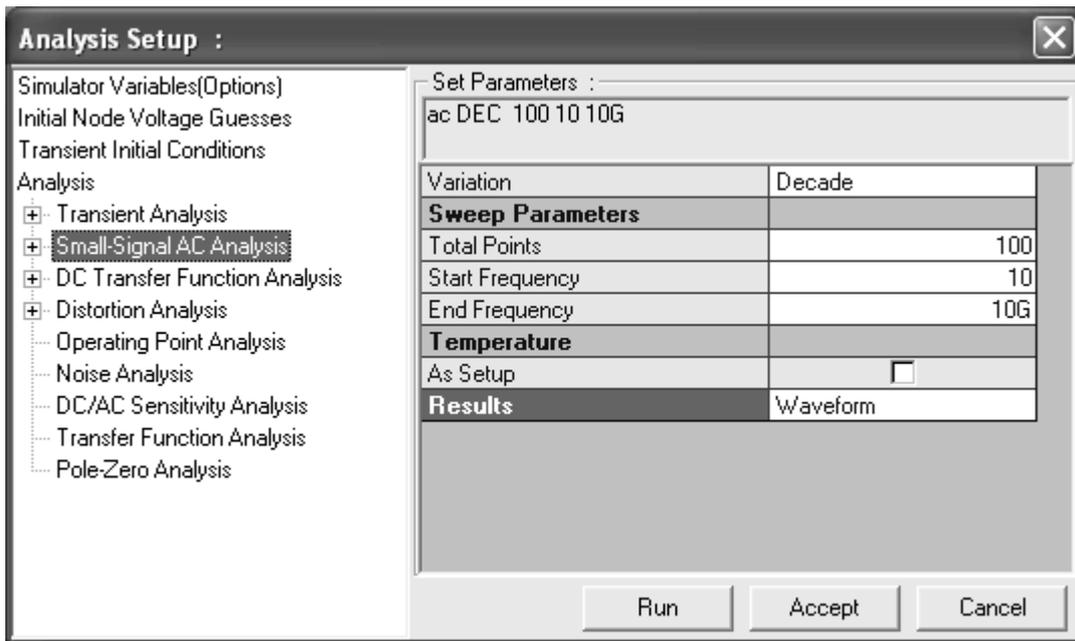
        Total points   : 100

        Start frequency   : 10 Hz

7. Select *Waveform* for displaying the output.

8. Click *Accept* button after entering the values to automatically switch to Analysis.

9. Click *Run* button to start simulation

Operating Point Analysis

Operating Point Analysis is used to determine the DC behaviour of a circuit. This is done automatically when an AC analysis and Transient Analysis is done.

1. Select *Simulation → Analysis*.

2. Select *Operating point Analysis* from the tree view.

3. The default selection is 'As marked' means that results will be displayed at all markers placed on the circuit schematic. After analysis, the values will be presented by updating the markers.

4. If we want to know the values at all nodes and branches of the circuit, select 'All Points' the results can be viewed in the RAWSPICE.RAW file, For that Select *Options → View EDSpice Files → Raw file*, from that Open rawspice.raw file.
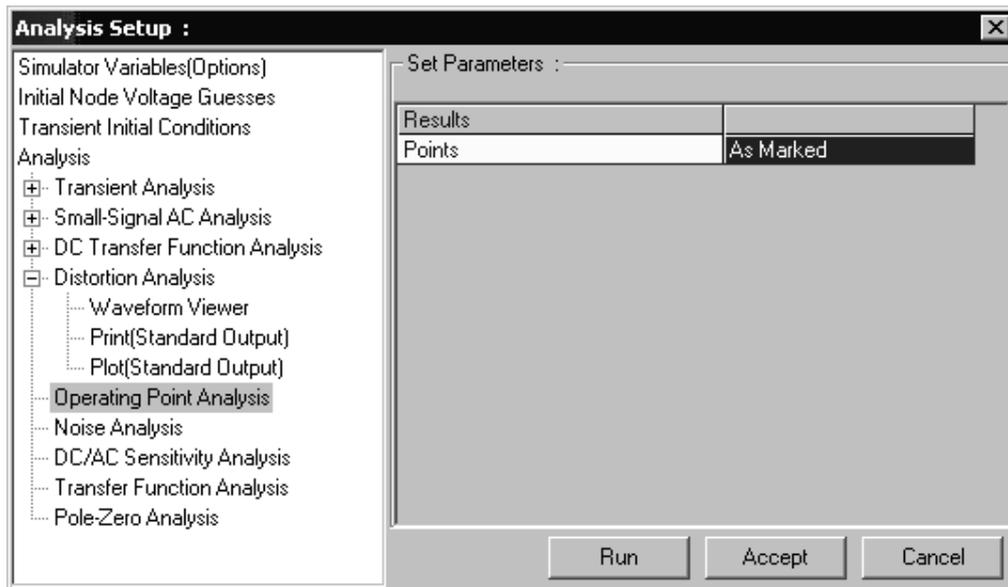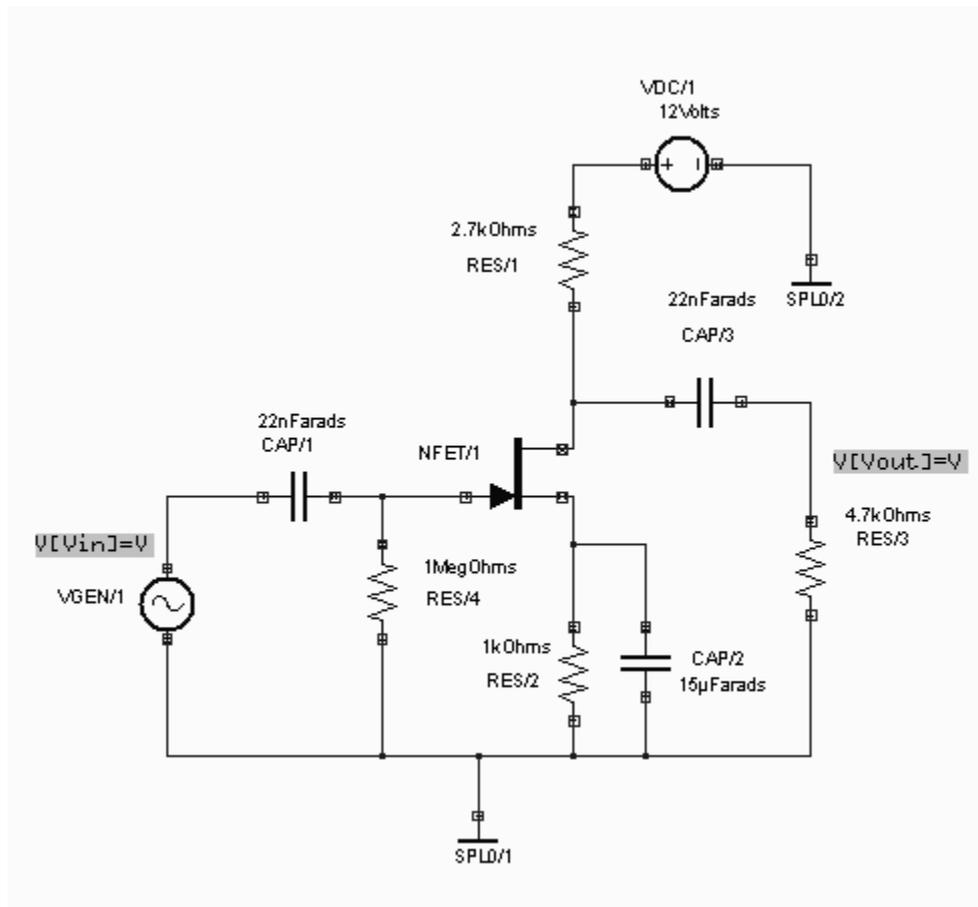

FET AMPLIFER

Aim

To simulate a JFET Amplifier

Components required

JFET - 2N3819 (1), Resistors - RC05 (4), Capacitors – CK05 (3), AC Voltage Source - VGEN (1), DC Voltage Source – VDC (1), Ground – SPL0 (2)

Circuit diagram

Procedure

Draw the circuit diagram after loading components from library. Assign net names for connections wherever necessary. Preprocess the circuit by invoking *Simulation →* *Preprocess*. Place waveform markers on input and output nodes. For placing waveform markers, select *Tools → Instruments → set wave form Contents → Voltage waveform →* Click on the required net and place the waveform marker. Run transient analysis after setting up required parameters.

To proceed with the simulation, the steps are as follows

Select *Tools → Components → Component properties→ Change simulation parameters→* Click on the required component and change its values.

The values to be provided to the components are

JFET   : 2N3819 (Click on the *Set up Model* of the *Model Type*. Click on *Accept* to get the default values assigned)

Resistors   : 2.7k (1), 4.7k (1), 1 M (1), 1k (1)

Capacitors   : 22nF (2), 15µF (1)

VDC/1   : 12V

VGEN   : Select the *source function* as SIN and set its Properties as follows

    VA (Amplitude)   : 100mV

    FREQ (Frequency)   : 1kHz

For setting up simulation time and analysis types, select *Simulation* menu, choose *Analysis*. In the window pops up, select *Transient analysis* from tree view. Enter following values:
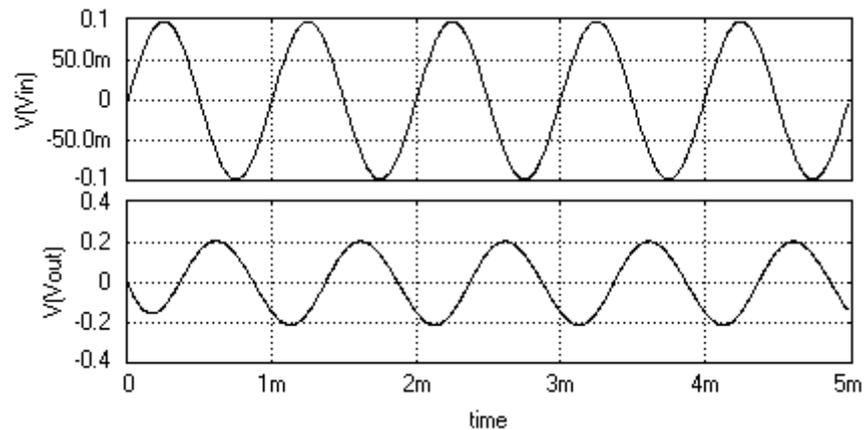
    Step   : 5μ

    Final time   : 5m

    Results   : Select *Waveform* from drop down menu.

Click on *Accept*. Expand the *Transient Analysis* and select *Waveform viewer*, Select *As Marked*. Click on *Accept*. Select *Transient Analysis*. Run Analysis by clicking on *Run* button.

Result

References

1. OPUSER Help File

2. Electronics Lab Manual – K. A. Navas

3. Wikipedia