

Electronic Design for Windows
EDWIN XP



VISIONICS

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the EDSpice Simulator document can be reproduced in any form or by any means without the prior written permission of Visionics. EDSpice Simulator document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, DocOne, EDComX, SIMWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

Contents

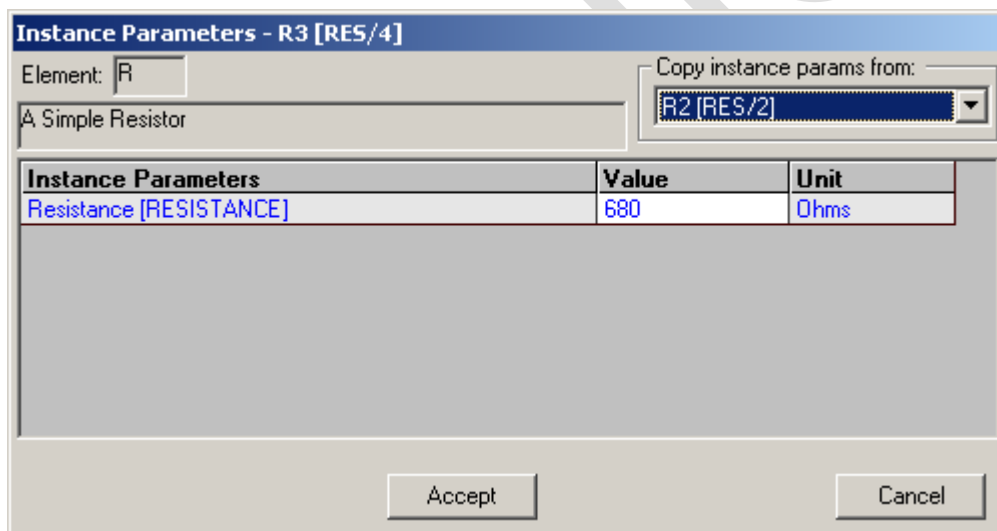
EDSPICE SIMULATOR	4
STEPS FOR EDSPIICE SIMULATION	5
<i>Preprocess</i>	5
ANALYSIS	5
TRANSIENT ANALYSIS	6
SMALL SIGNAL AC ANALYSIS	8
DC TRANSFER FUNCTION ANALYSIS	10
DISTORTION ANALYSIS	12
OPERATING POINT ANALYSIS	14
NOISE ANALYSIS.....	16
DC / AC SENSITIVITY ANALYSIS.....	18
TRANSFER FUNCTION ANALYSIS	21
POLE- ZERO ANALYSIS.....	23

EDSpice Simulator

EDSpice is a SPICE-like simulator giving improved performance over SPICE2 based simulators. It is solely based on Berkeley SPICE3 with a number of extensions and improvements. Featured are ten types of analyses with improved convergence algorithms.

Assign Component Parameter Values

1. Select **Component Properties** (function tool) → **Change Simulation Parameters** (option tool) and click on the component. The Instance Parameter window opens.



2. Change the parameter values.
3. Click **Accept**.

Viewing the output

1. Select **Tools** → **Instruments** → **Set wave form Contents** (Third function tool) → **Voltage Waveform** (First option tool).
2. Click on the nets where the voltage waveforms to be displayed.

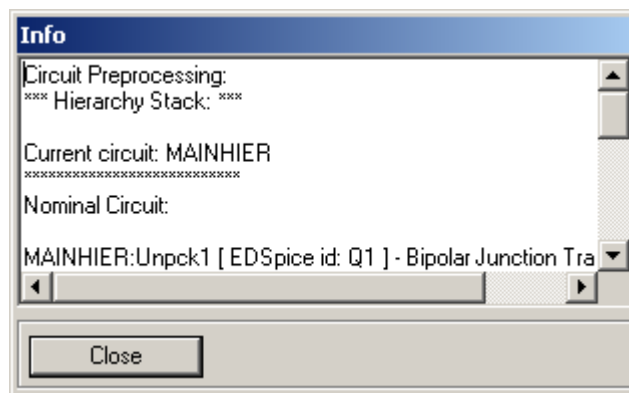
Steps for EDSpice Simulation

Preprocess

Preprocessing confirms whether the circuit is ready for simulation. Preprocessing must be performed at all times, when elements have been added or deleted from the circuit or connectivity between them is changed.

In order to preprocess the circuit

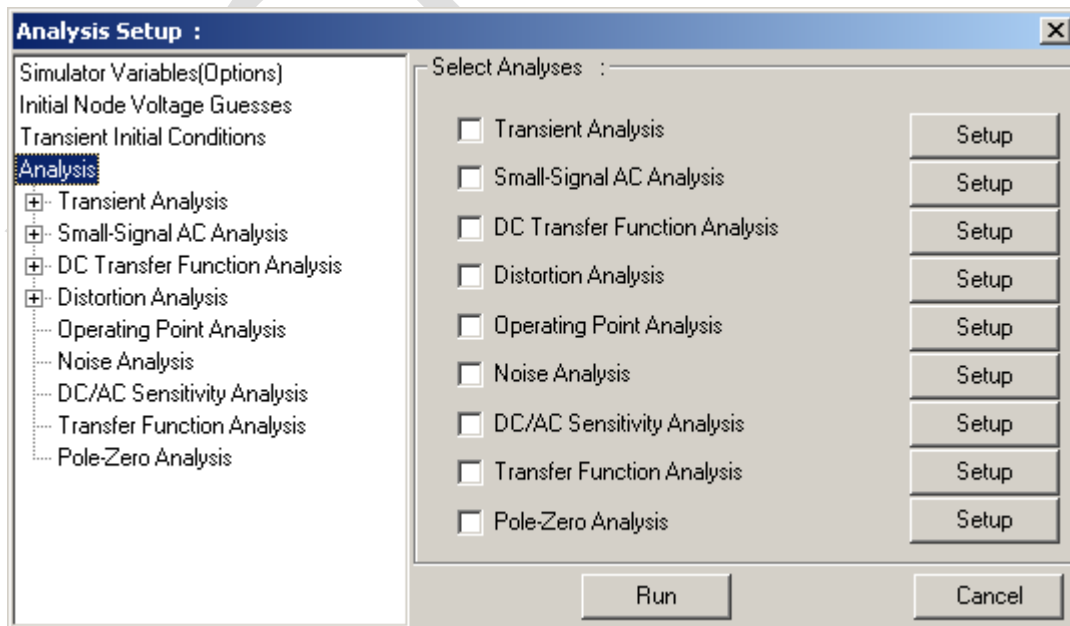
1. Select **Simulation** → **Preprocess**



2. Click **Close**.

Analysis

1. Select **Simulation** → **Analysis**



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. In order to run a Transient Analysis, specify the analysis parameters by selecting Analysis/ Transient analysis from the tree view. Specify the simulation parameters such as start time, time step etc.

Parameters: Allows setting the start and final time for which the analysis is to be performed. 'Steps' define the number of points to be included for specifying the range of time on the X-axis.

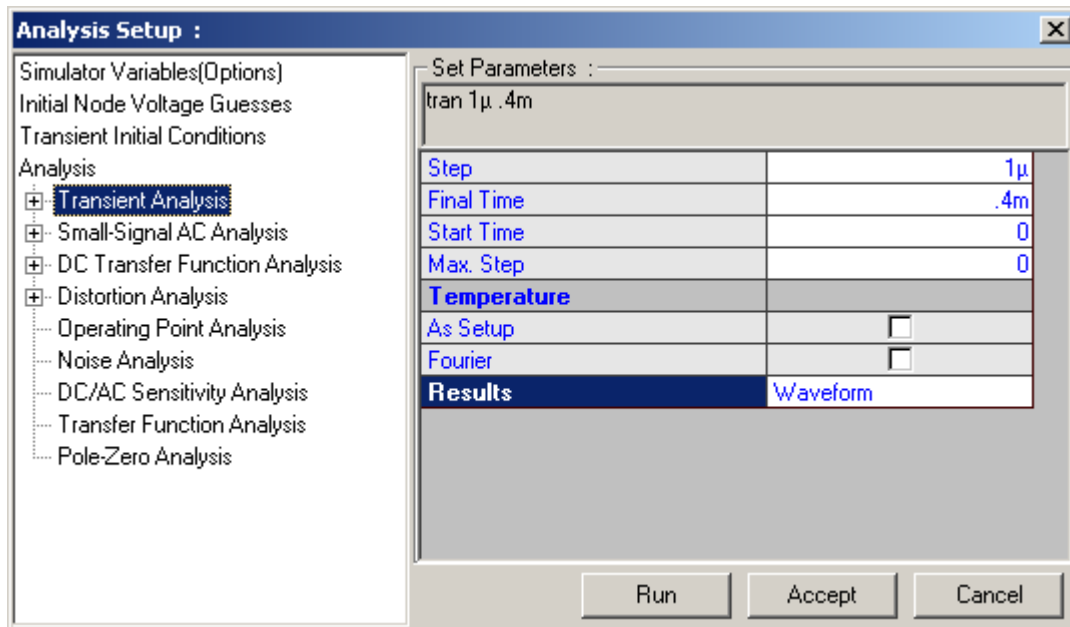
Temperatures: Considers the temperatures set prior to simulation from As Setup as the temperature level of the environment. This option is to get the results of simulation at different temperatures. If this option is not checked, then 270 Celsius (standard temperature) will be taken as the operating temperature.

Fourier: Allows setting the parameters and conduct Fourier analysis along with Transient analysis.

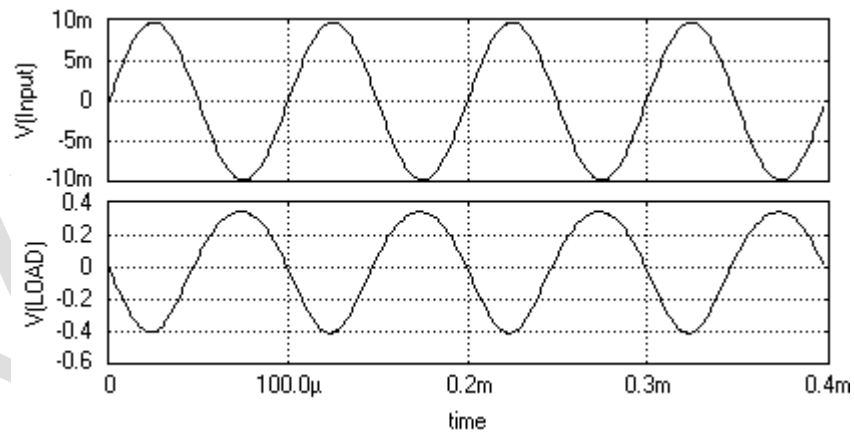
Result Allows obtaining the results of simulation to any of the selected options. A diagram may be displayed by enabling the Diagram option.

1. Select **Simulation → Analysis**.
2. Select **Transient Analysis** from the tree view.
3. Enter the values as

Step	1
Final time	. 4m
Start time	0



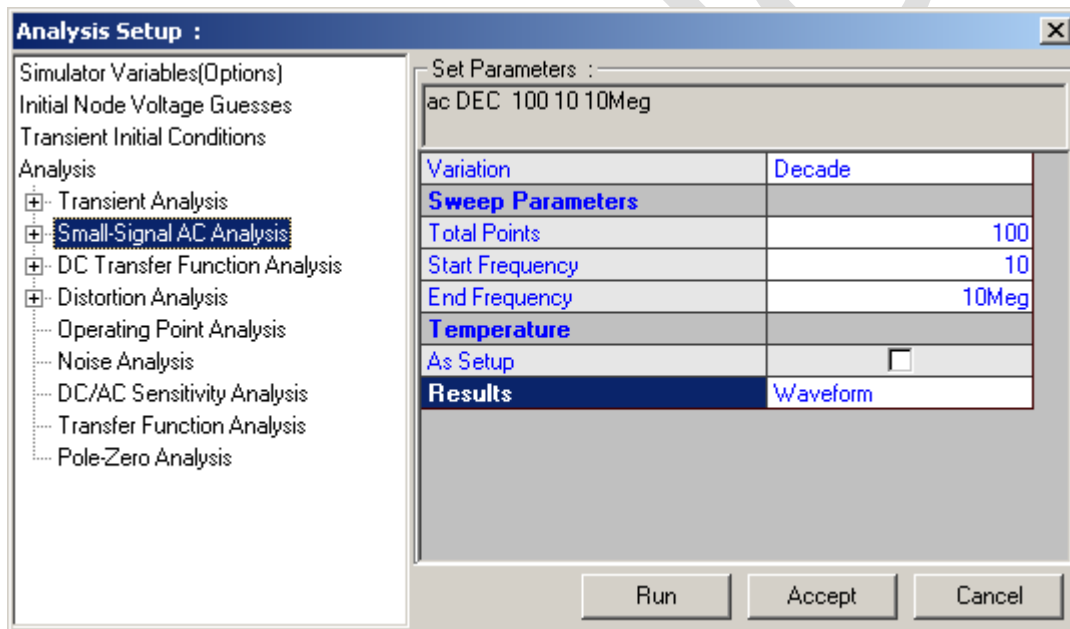
4. Click **Accept** button after entering the values to automatically switch to Analysis.
5. Click **Run** button to start simulation.



Small Signal AC Analysis

A 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 to obtain the small signal AC behavior of the circuit. Both AC currents and voltages at various nodes can be obtained.

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



Variation: Allows setting the iteration range as linear, octave or decade.

Sweep Parameters: Allows specifying the start and end frequencies. The difference between these frequencies may be entered in `

Total Points`. `Total points` define the number of points to be included for specifying the range of frequency on the X-axis.

Results: Allows obtaining the results of simulation on any of the selected options. A diagram may be displayed by enabling the diagram option.

Temperatures: Considers the temperatures set prior to simulation from As Setup as the temperature level of the environment. This option is also used to get the results of simulation at different temperatures.

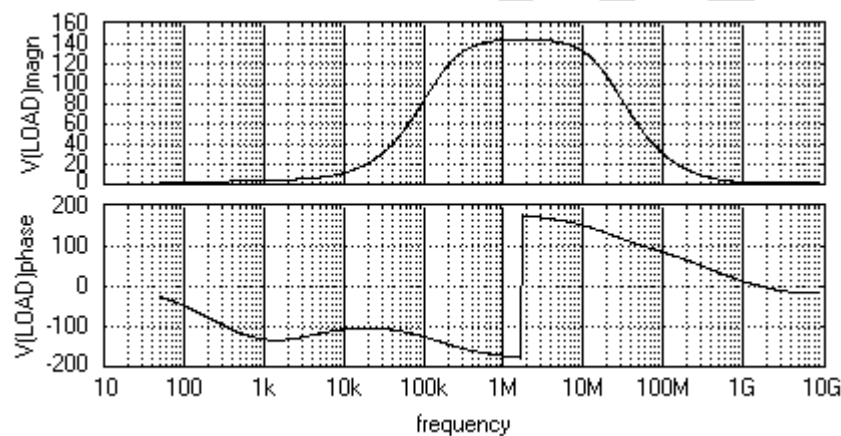
3. Enter the values as

Total points	100
Start frequency	10 Hz
End frequency	100GHz

4. Select **Waveform** for displaying the output.

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

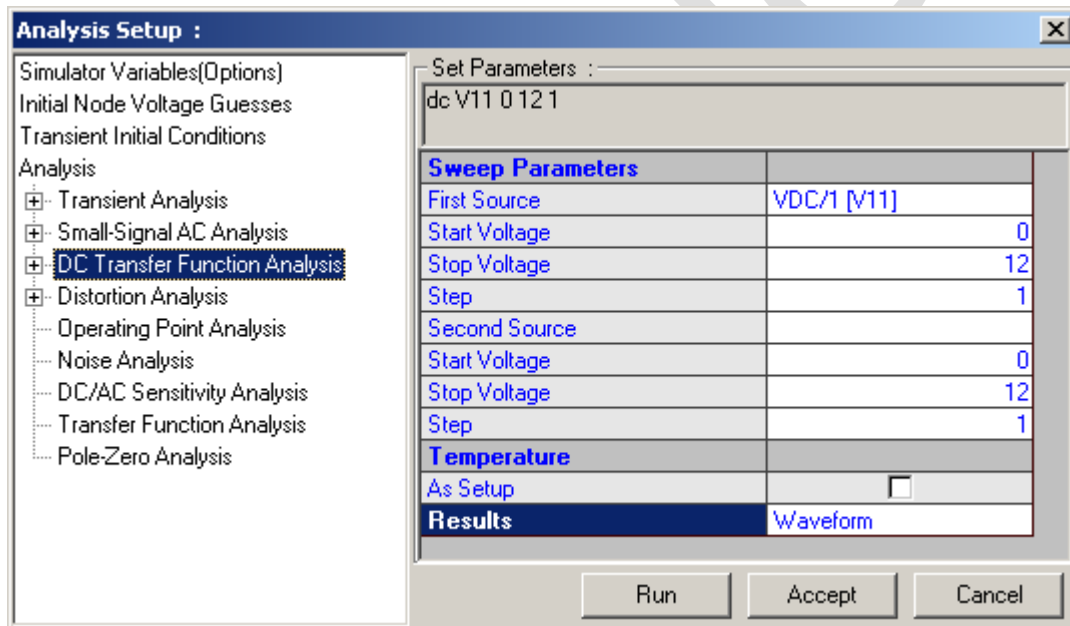
6. Click **Run** button to start simulation.



DC Transfer Function Analysis

A 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 often used for obtaining the characteristic output curves of semiconductor devices.

1. Select **Simulation → Analysis**.
2. Select **DC Transfer Function Analysis** from the tree view.



Sweep Parameters: Allows setting the start and stop voltages for first and second source for which the analysis is to be done. 'Step' defines the number of points to be included for specifying the range of voltage on the X-Axis.

Results: Allows obtaining the results of simulation to any of the selected options. A diagram may be displayed by enabling the diagram option.

Temperatures: Considers the temperatures set prior to simulation from As Setup as the temperature level of the environment. This option is also used to get the results of simulation at different temperatures. If this option is not checked,

then 270 Celsius (standard temperature) will be taken as the operating temperature.

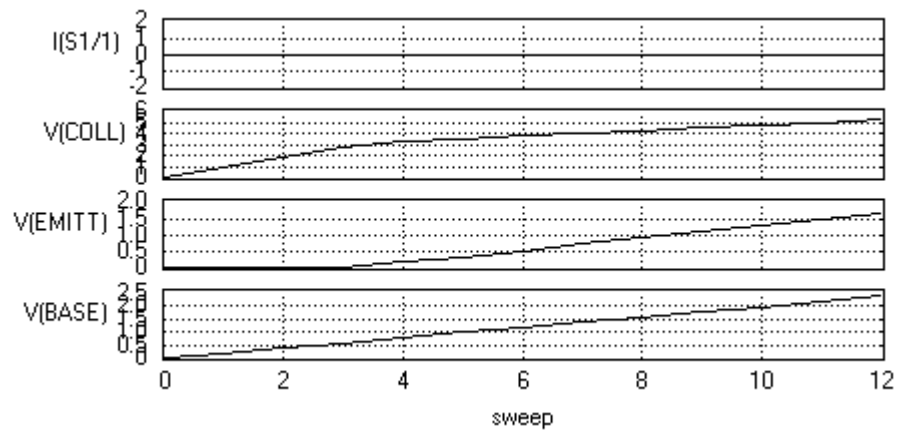
3. Set the values as

Start Voltage 0V

Stop Voltage 12V



Step 1

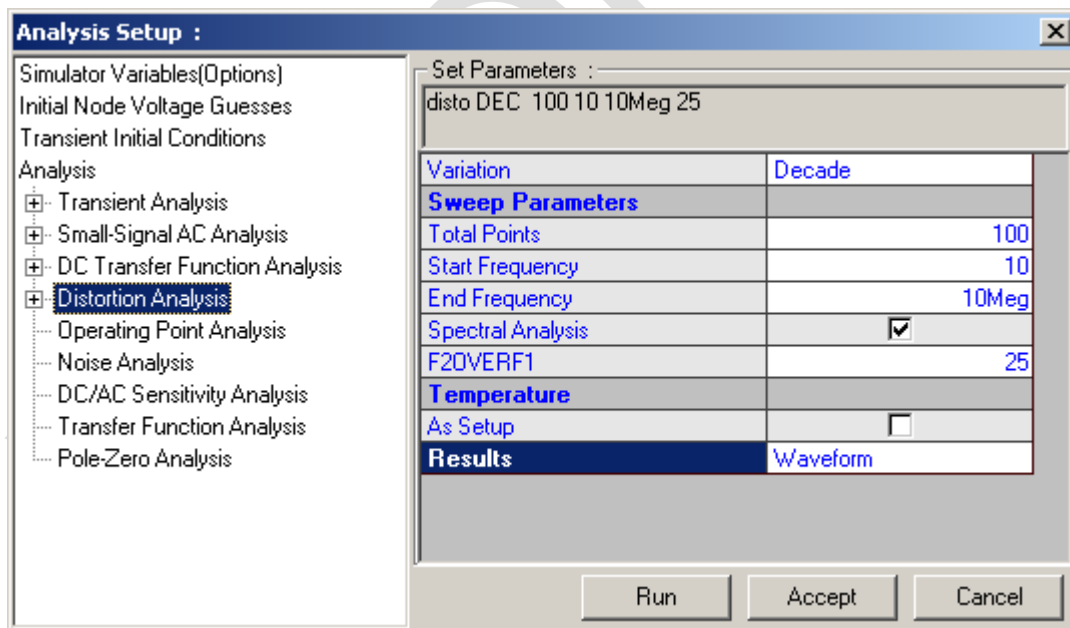
4. Select **Waveform** for displaying the output.
5. Click **Accept** button after entering the values to automatically switch to Analysis.
6. Click **Run** button to start simulation



Distortion Analysis

The distortion analysis computes steady state harmonic and intermodulation products for small input signal magnitude. If signals of a single frequency are specified as the input to the circuit, the complex values of the second and third harmonics are determined at every point in the circuit. If two frequencies are specified at the input of the circuit the analysis finds out the complex values of the circuit variables at the sum and difference of the input frequencies, and at the difference of the smaller frequency from the second harmonic of the larger frequency.

1. Set the frequency, Select  **Component parameters** (function tool) →  **Change Simulation Parameters** option tool.
2. Select **Simulation** → **Analysis**.
3. Select **Distortion analysis** from the tree view.



Variation: Allows setting the iteration range as linear, octave or decade.

Sweep Parameters: Allows specifying the start and end frequencies. The difference between these frequencies may be displayed in Total Points. Total

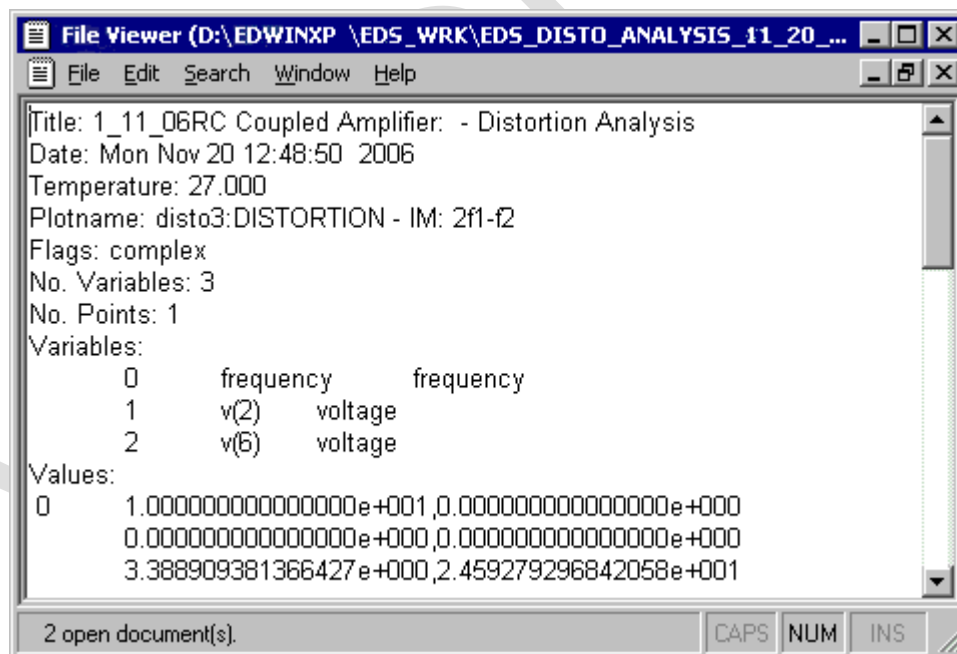
points' define the number of points to be included for specifying the range of frequency on the X-Axis.

Spectral Analysis Allows obtaining the spectral analysis, if enabled. Else Harmonic analysis will be executed.

F20VERF1 This parameter must be entered if Spectral Analysis checkbox is enabled.

4. In order to run distortion analysis, specify the analysis parameters by selecting Distortion Analysis from the tree view on the left side of the **Analysis Setup**.
5. Set the values as

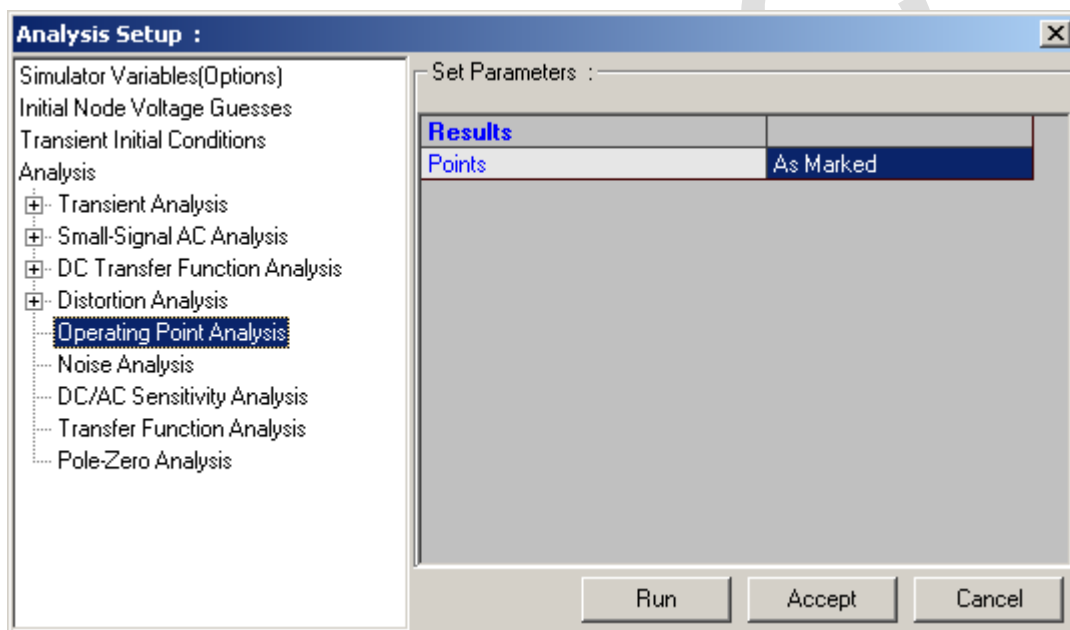
Total points	100
Start frequency	10Hz
End frequency	10Meg
6. Click **Accept** button after entering the values to automatically switch to Analysis.
7. Click **Run** button to start simulation. The output obtained is a text file.



Operating Point Analysis

An Operating Point Analysis is used to determine the DC behavior of a circuit; in other words, the DC bias conditions and static power consumption. When the analysis is performed for the entire circuit, all capacitors are open circuited and inductors are short-circuited for calculating the values. This is because an operating point analysis is performed using DC values, at which inductors are effectively short circuits and capacitors are open circuits.

1. Select **Simulation → Analysis**.
2. Select **Operating point Analysis** from the tree view.



As Marked

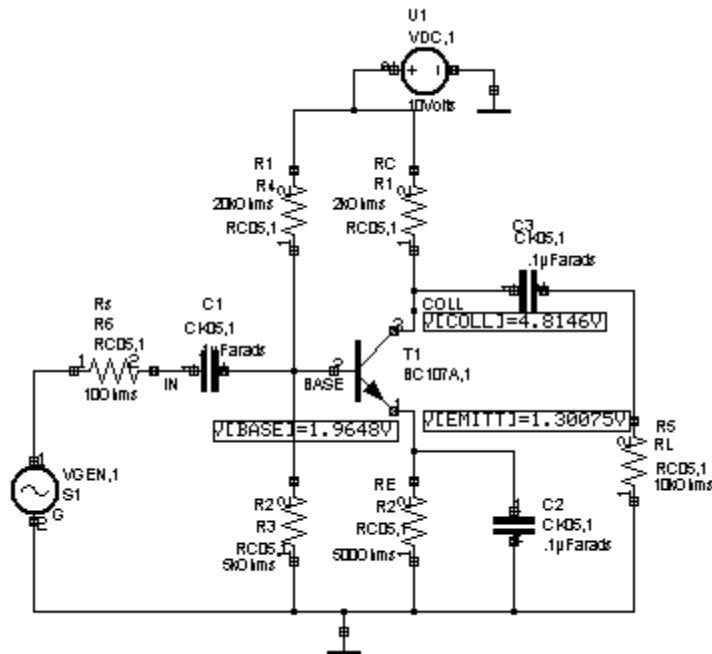
Allows obtaining the output of the analysis at points marked in the schematic circuit.

All Points

Allows obtaining the output at all points (output at all points may be viewed from File | View EDSpice Files/Rawfile)

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.

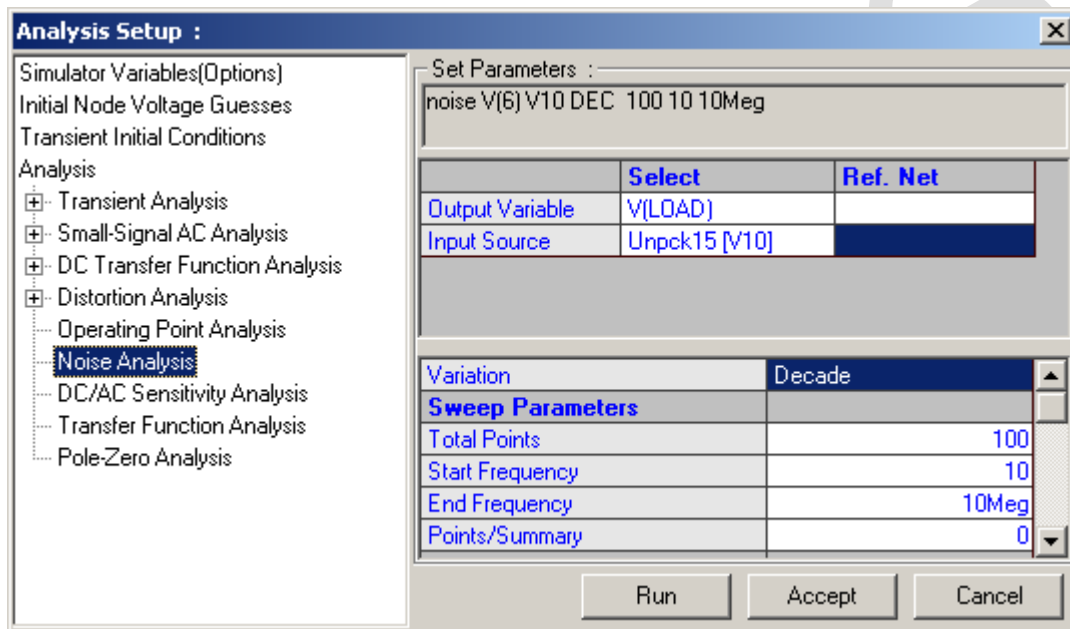
4. If we want to know the values at all nodes and branches of the circuit, select '**All Points**' the results can be viewed in the **RAWSPICE.RAW** file, For that Select **Options** → **View EDSpice Files** → **Raw file**, from that Open **rawspice.raw** file.



Noise Analysis

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.

1. Select **Simulation → Analysis**.
2. Select **Noise Analysis** from the tree view.



Select Net/ Source: Provides a combo box listing all the nets present in the current circuit. The required net may be selected from this. A net is selected for the output variable while a source is selected for the input source, in the next row.

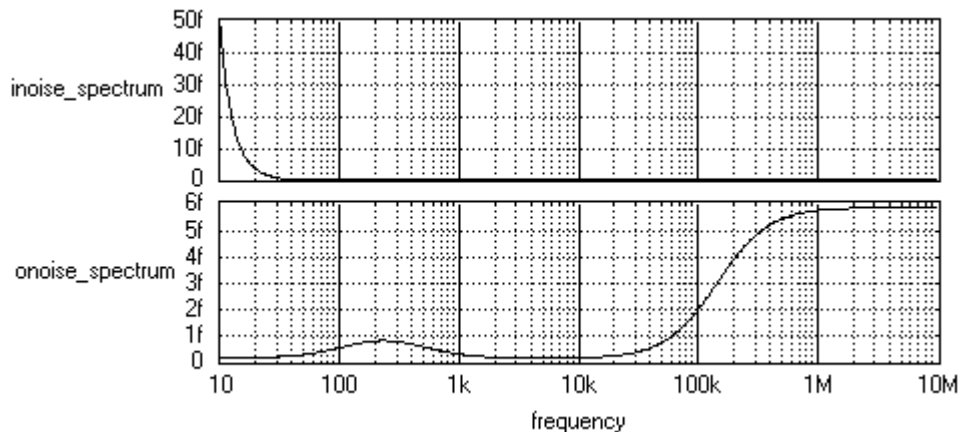
Reference Net: Allows selecting the reference net for which the difference between the node voltages may be specified. Select a reference net from the drop down list. Click a particular cell to enable the drop down list.

Variation: Allows setting the iteration range as linear, octave or decade.

Sweep Parameters: Allows specifying the start and end frequencies. The difference between these frequencies may be displayed in Total Points. Total points' define the number of points to be included for specifying the range of frequency on the X-Axis.

Points/ Summary: Allows setting the total number of points to be plotted in the output diagrammatic representation.

3. Set the simulation parameters such as start frequency, end frequency, total points etc.
4. After analysis the **Noise Spectral Density Curves** will be presented in the Waveform viewer as shown below.



5. The total integrated Noise will be presented in the rawspice.raw file. It can be viewed from **Options → View EDSpice Files → Raw file**, this .raw file is pasted below.

```

File Viewer (D:\EDWINXP\EDS_WRK\RAWSPICE.RAW) - [D:\EDWINXP..
File Edit Search Window Help
Title: 1_11_06RC Coupled Amplifier: - Noise Analysis
Date: Mon Nov 20 13:10:43 2006
Temperature: 27.000
Plotname: noise2:Integrated Noise - V^2 or A^2
Flags: real
No. Variables: 2
No. Points: 1
Variables:
    0      onoise_total    voltage
    1      inoise_total    voltage
Values:
0      5.754142702638551e-008
      1.957670426641511e-013

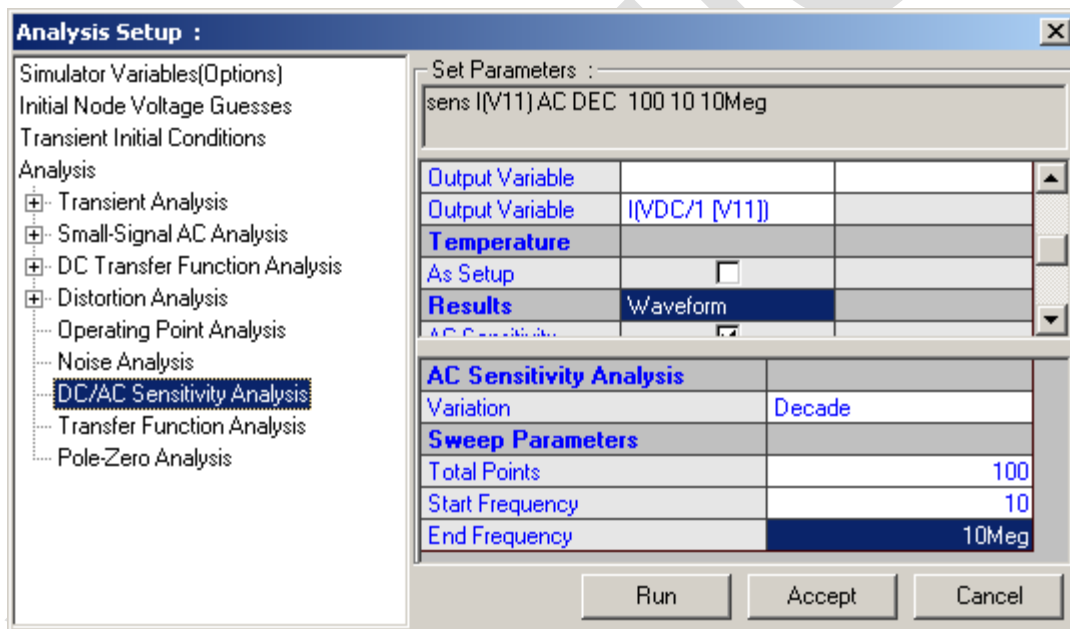
Title: 1_11_06RC Coupled Amplifier: - Noise Analysis
Date: Mon Nov 20 13:10:43 2006
1 open document(s).
CAPS NUM INS

```

DC / AC Sensitivity Analysis

In a circuit signals at any point affects 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 which circuit parameter/s most affect 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.

1. Select **Simulation → Analysis**.
2. Select **DC / AC Sensitivity Analysis** from the tree view.



Select Net/ source Provides a list of all the nets present in the loaded project in the combo box, from which the required net may be selected for one of the output variables.

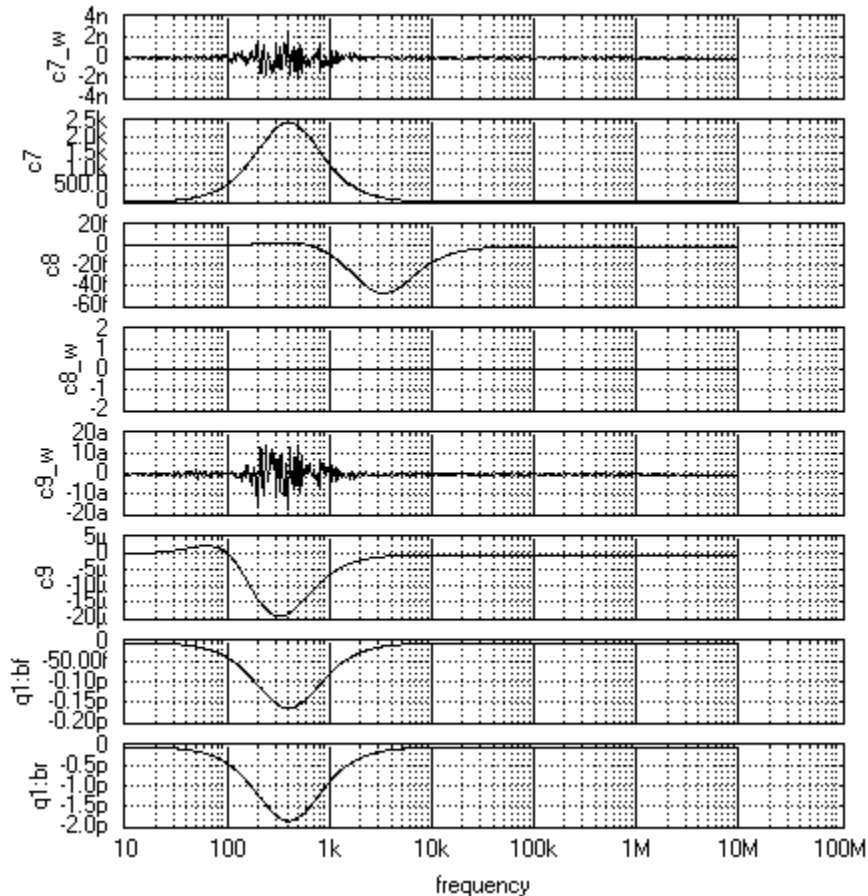
Reference net Allows selecting the reference net for which the difference between the node voltages may be specified. Select a reference net from the drop down list. Click a particular cell to enable the drop down list.

AC Sensitivity Allows conducting AC sensitivity analysis, if enabled. The variation and sweep parameters may be activated only if this option is enabled.

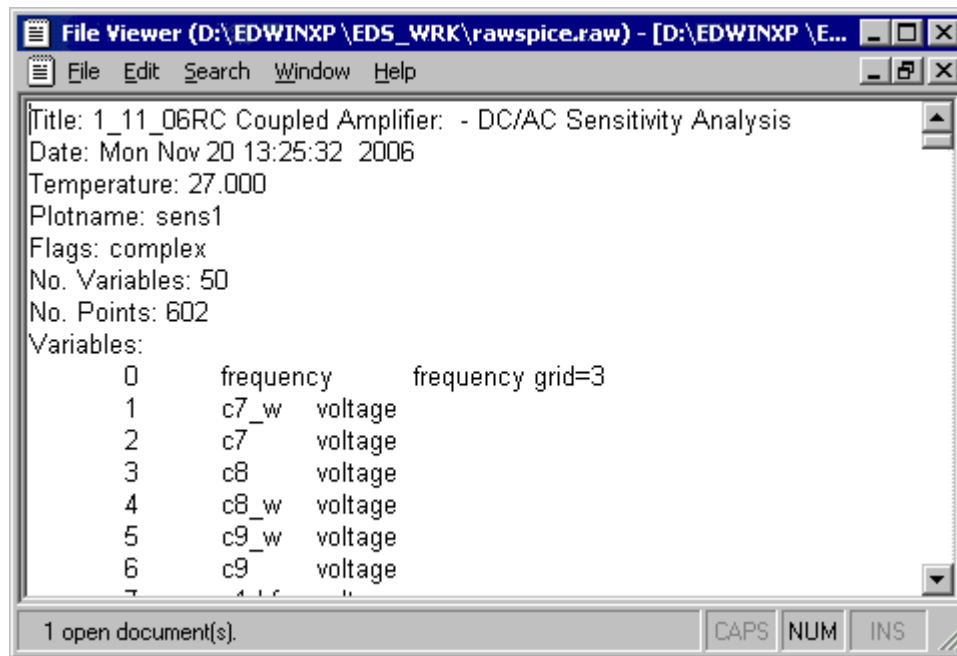
Variation Allows setting the iteration range as linear, octave or decade.

Sweep Parameters Allows specifying the start and end frequencies. The difference between these frequencies may be displayed in 'Total Points'. 'Total points' define the number of points to be included for specifying the range of frequency on the X-Axis.

3. Click **Accept** button after entering the values to automatic switch to Analysis.
4. Click **Run** button to start simulation



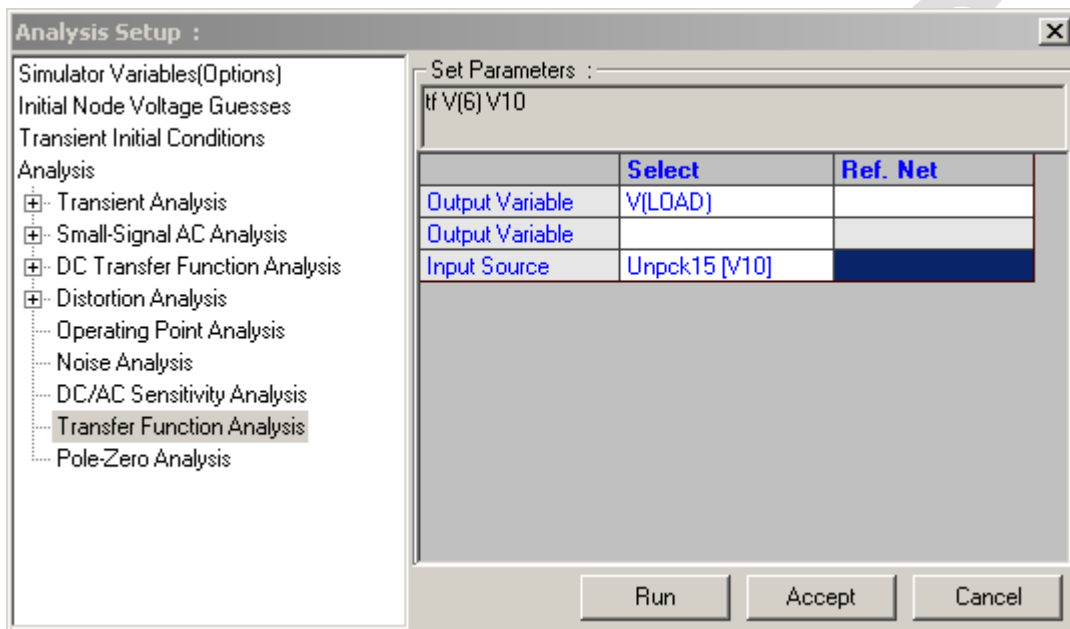
5. After analysis the results of the **DC Sensitivity Analysis** will be presented in the rawspice.raw file, open this file from **Options View EDSpice Files Rawfile**.
6. The results of **AC Sensitivity Analysis** will be presented in the Wave form viewer.



Transfer Function Analysis

The transfer function analysis calculates the small signal ratio of the output node to the input source, and also the input and output impedance of the circuit.

1. Select **Simulation → Analysis**.
2. Select **Transfer function Analysis** from the tree view.



Select Net/ Source: Provides a list of all the nets present in the loaded project in the combo box, from which the required net may be selected for one of the output variables.

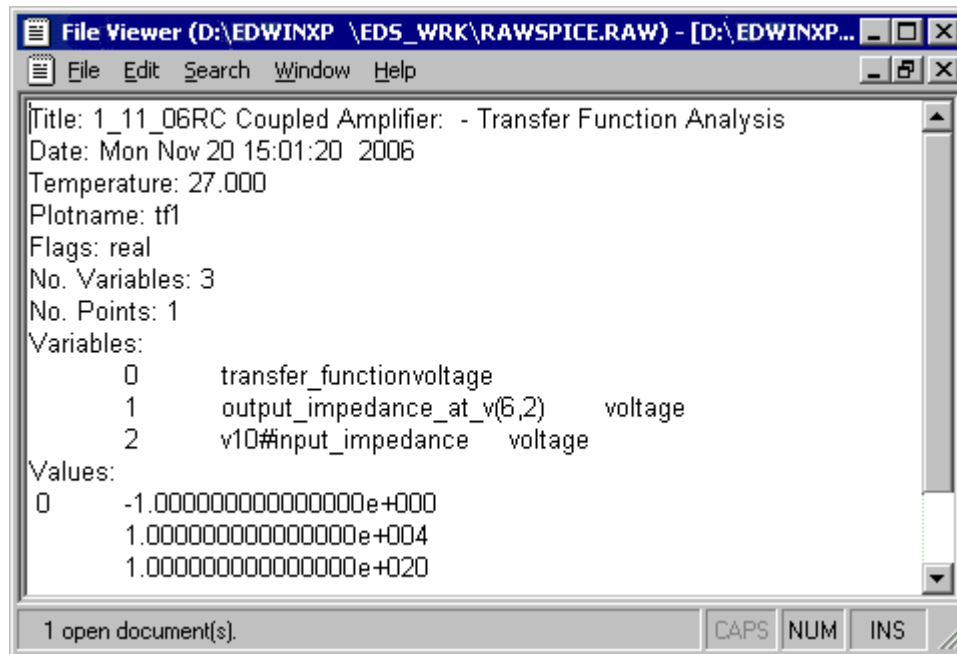
Reference net: Allows selecting the reference net for which the difference between the node voltages may be specified. Select a reference net from the drop down list. Click a particular cell to enable the drop down list.

Output Variable: Displays the selected net names set as the output.

Input Source: Allows setting the input node source. The parameters for this may be selected by clicking the cell under Select net| Source column.

3. Set parameters and click **Accept** to accept these values.
4. Click **Run** button to execute the analysis.

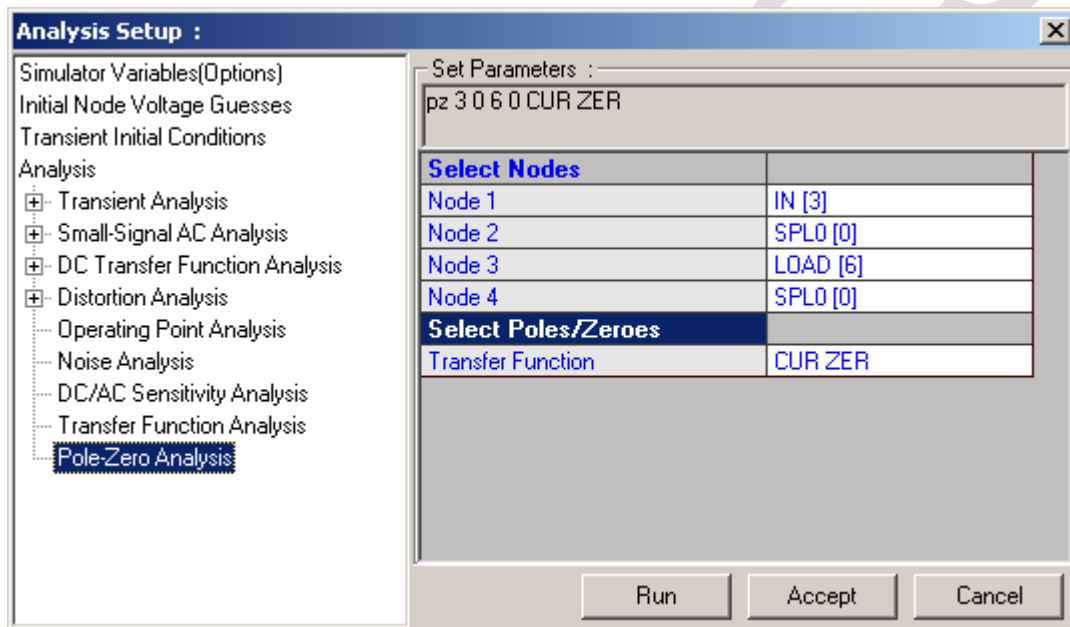
5. After the analysis, the results will be displayed in the .raw file. Open this file from **Options** → **View EDSpice Files** → **Rawfile**.



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.

1. Select **Simulation → Analysis**.
2. Select the **Pole Zero Analysis** from the tree view.



3. Click **Accept** button to accept these values.
4. Click **Run** button to execute the analysis.
5. After analysis, the results will be displayed in the .raw file .For opening this file select **Options →View EDSpice Files →Rawfile**.

