A SPICE simulation guide for analysis of mixed analog and digital systems and circuitry in Altium Designer.

Altium Analysis Reference

ELEN90053 - Digital Systems Design

Compiled By : M J Adams [02.08.2011]

Contents

Transient Analysis

Description

A Transient analysis generates output similar to that normally shown on an oscilloscope, computing the transient output variables (voltage or current) as a function of time, over the user-specified time interval.

An Operating Point analysis is automatically performed prior to a Transient analysis to determine the DC bias of the circuit, unless the **Use InitialConditions** parameter is enabled.

Setup

Transient analysis is set up on the **Transient/Fourier Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Transient/Fourier Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Transient/Fourier Analysis Setup	
Parameter	Value
Transient Start Time	0.000
Transient Stop Time	5.000m
Transient Step Time	20.00u
Transient Max Step Time	20.00u
Use Initial Conditions	☐
Use Transient Defaults	☑
Default Cycles Displayed	5
Default Points Per Cycle	50
Enable Fourier	☐
Fourier Fundamental Frequency	1.000k
Fourier Number of Harmonics	10

Parameters

- **Transient Stat Time** - the value for the start of the required time interval for analysis (in seconds)
- **Transient Stop Time** - the value for the end of the required time interval for analysis (in seconds)
- **Transient Step Time** - the nominal time increment used in the analysis.
- **Transient Max Step Time** - the maximum variation in size of the time step that can be used by the Simulator when calculating the transient data. By default, the value used is either
- **Transient Step Time** or
- (**Transient Stop Time** - **Transient Start Time**) / 50,
- whichever is the smaller.
- **Use Initial Conditions** - when enabled, the Transient analysis begins from the initial conditions defined in the schematic, bypassing the Operating Point analysis. Use this option when you wish to perform a transient analysis starting from other than the quiescent operating point.
- **Use Transient Defaults** - when enabled, parameters are automatically calculated before each simulation run, overriding any manually set values.
- **Default Cycles Displayed** - the default number of periods of a sinusoidal waveform to display. This value is used in the automatic calculation of the **Transient Stop Time**, when the **Set Defaults** button is pressed.

- **Default Points Per Cycle** - the number of data points per sinusoidal waveform period. This value is used in the automatic calculation of the **Transient Step Time** when the **Set Defaults** button is pressed.

Notes

A Transient analysis always begins at time zero. In the time interval between zero and **Transient Start Time**, the circuit is analyzed but the results are not stored. In the time interval between **Transient Start Time** and **Transient Stop Time**, results are stored for display.

Although **Transient Step Time** is the nominal time increment used in the analysis, the actual time step is varied automatically to achieve convergence.

Typically **Transient Step Time** and **Transient Max Step Time** are set to the same value. As a starting point set both of these parameters to

(**Transient Stop Time** - **Transient Start Time**) / 1000.

If you are not sure what values to enter, press the **Set Defaults** button on the page to automatically calculate the Transient analysis parameters as follows:

- **Transient Start Time** is set to zero

- **Transient Stop Time**, **Transient Step Time** and **Transient Max Step Time** are calculated based on the values entered for the **Default Cycles Displayed** and **Default Points Per Cycle** parameters, as well as the lowest frequency source in the circuit (with frequency F_L). The formulae used for the calculations are as follows:

  Transient Stop Time = $(1/F_L)$ * **Default Cycles Displayed**

 **Transient Step Time** = $(1/F_L)$ / **Default Points Per Cycle**

  Transient Max Step Time = **Transient Step Time**

When using Initial Conditions, make sure that you first define the initial condition for each appropriate component in the circuit, or place .IC devices on the circuit. The IC value of a component overrides an .IC object attached to a net.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

The simulation results are displayed on the **Transient Analysis** tab of the Waveform Analysis window.
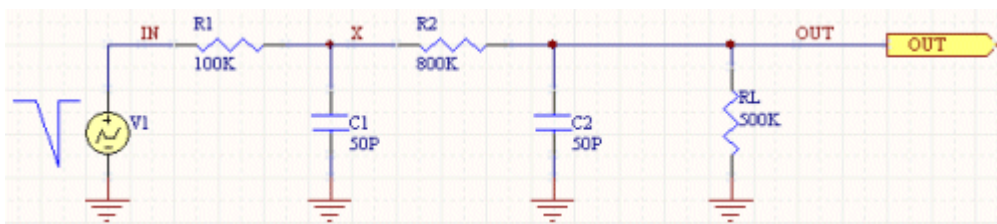
Examples

Consider the circuit in the image above, where a Transient analysis is defined with the following parameter values:

- Transient Start Time = 0.000
- Transient Stop Time = 100.0u
- Transient Step Time = 500.0n
- Transient Max Step Time = 1.000u
- Default Cycles Displayed = 5
- Default Points Per Cycle = 50

- Use Initial Conditions and Use Transient Defaults parameters are both disabled.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TRAN 5E-7 0.0001 0 1E-6
```

and running the simulation will yield the output waveforms shown in the image below:
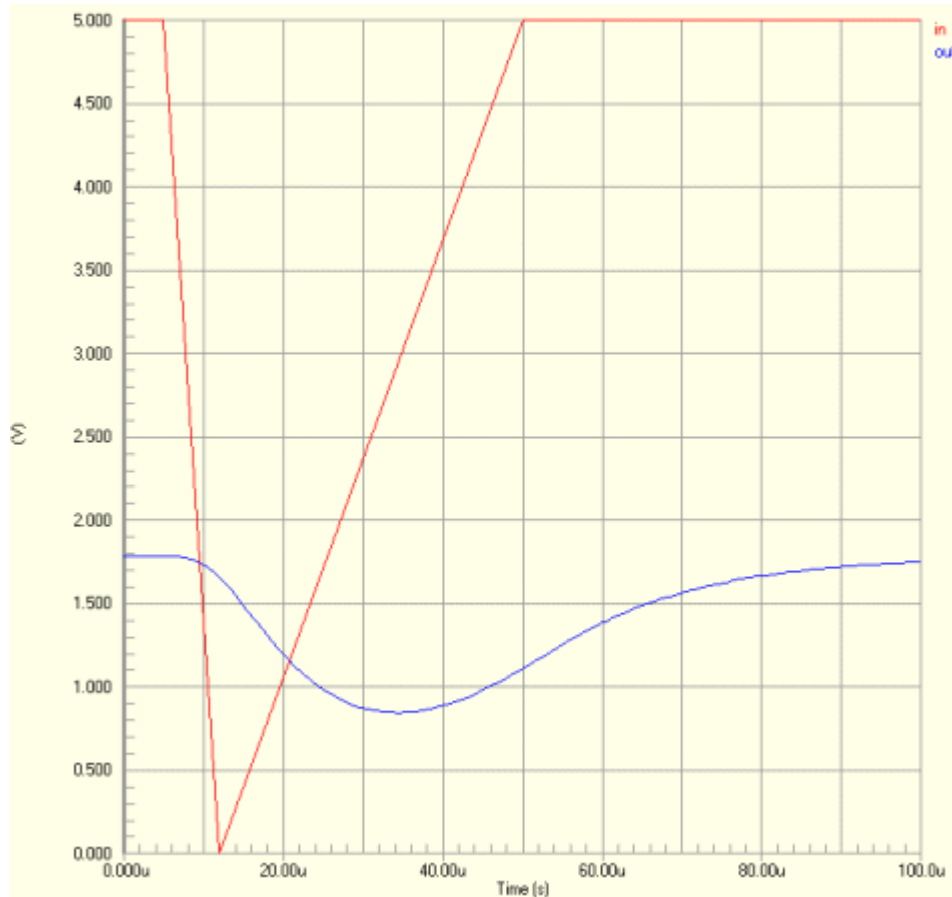
Fourier Analysis

Description

The Fourier analysis of a design is based on the last cycle of transient data captured during a Transient analysis. For example, if the fundamental frequency is 1.0kHz, then the transient data from the last 1ms cycle would be used for the Fourier analysis.

Setup

Fourier analysis is set up on the **Transient/Fourier Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Transient/Fourier Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Transient/Fourier Analysis Setup	
Parameter	Value
Transient Start Time	0.000
Transient Stop Time	5.000m
Transient Step Time	20.00u
Transient Max Step Time	20.00u
Use Initial Conditions	☐
Use Transient Defaults	☐
Default Cycles Displayed	5
Default Points Per Cycle	50
Enable Fourier	☐
Fourier Fundamental Frequency	1.000k
Fourier Number of Harmonics	10

Parameters

- **Enable Fourier** - used to include Fourier analysis in the simulation. (Default = disabled).
- **Fourier Fundamental Frequency** - the frequency of the signal that is being approximated by the sum of sinusoidal waveforms.
- **Fourier Number of Harmonics** - the number of harmonics to be considered in the analysis. Each harmonic is an integer multiple of the fundamental frequency. Together with the fundamental frequency sinusoid, the harmonics sum to form the real waveform of the signal being analyzed. The more harmonics involved in the sum, the greater the approximation to the signal's waveform (e.g. summing sinusoids to form a square wave).

Notes

Upon running the simulation, a file will be generated (`ProjectName.sim`), written to the output folder for the project and opened as the active document in the main design window. This file contains detailed information on the magnitude and phase of each harmonic in the Fourier analysis, for each of the signals in the **Available Signals** list (on the **General Setup** page of the **Analyses Setup** dialog.

You must enable the **Transient/Fourier Analysis** option in the **Analyses/Options** list of the **Analyses Setup** dialog, in order to perform a Fourier analysis.

The simulation results are displayed on the **Fourier Analysis** tab of the Waveform Analysis window.
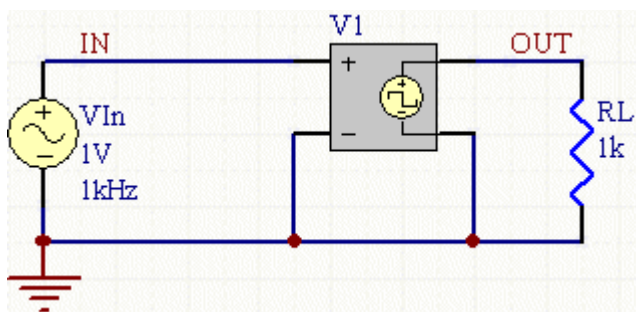
Examples

Consider the circuit in the image above, where a Transient analysis is defined with the following parameter values:

- Transient Start Time = 0.000
- Transient Stop Time = 5.000m
- Transient Step Time = 20.00u
- Transient Max Step Time = 20.00u
- Default Cycles Displayed = 5
- Default Points Per Cycle = 50
- Use Initial Conditions and Use Transient Defaults parameters are both disabled and a Fourier analysis is enabled and defined with the parameter values:
    - Fourier Fundamental Frequency = 1.000k
    - Fourier Number of Harmonics = 10

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TRAN 2E-5 0.005 0 2E-5
.SET NFREQS=10
.FOUR 1000 @VIn[p] VIn#branch @RL[p] @RL[i] OUT IN
```

The images below show the results of running the simulation. The first two images show waveforms obtained from the Transient analysis of the circuit, while the subsequent images show the results of the Fourier analysis. The square wave, whose fundamental frequency is 1kHz, is broken down into sinusoids with frequencies that are odd multiples of this frequency (odd harmonics), as shown in the third image (1kHz, 3kHz, 5kHz, 7kHz, etc) and with amplitudes that decrease with each subsequent harmonic.
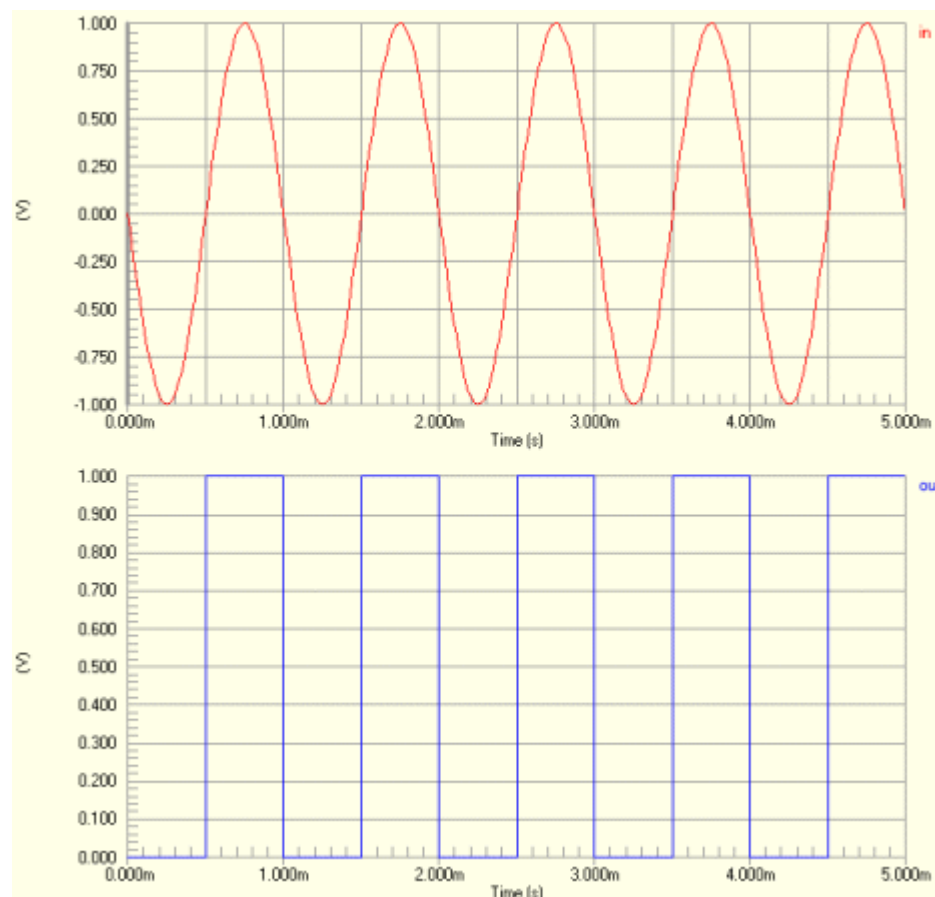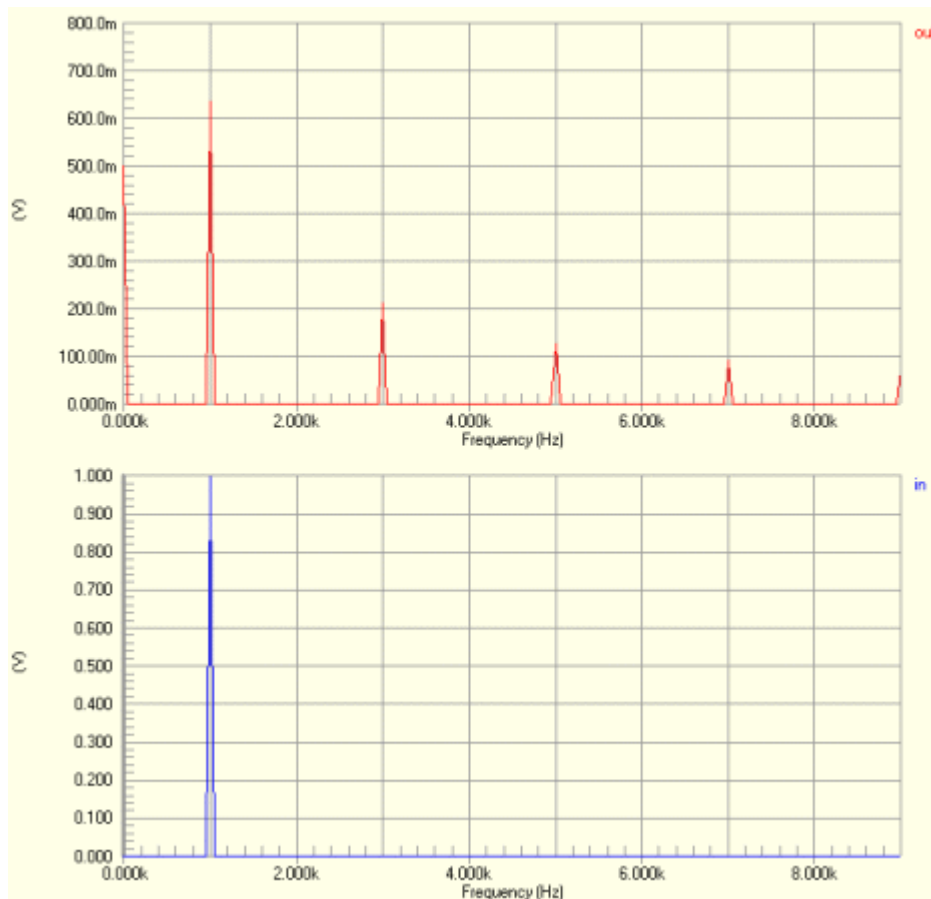
DC Sweep Analysis

Description

The DC Sweep analysis generates output like that of a curve tracer. It performs a series of Operating Point analyses, modifying the voltage of a selected source in pre-defined steps, to give a DC transfer curve. You can also specify an optional secondary source.

Setup

DC Sweep analysis is set up on the **DC Sweep Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **DC Sweep Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

DC Sweep Analysis Setup	
Parameter	Value
Primary Source	
Primary Start	0.000
Primary Stop	0.000
Primary Step	0.000
Enable Secondary	☐
Secondary Name	
Secondary Start	0.000
Secondary Stop	0.000
Secondary Step	0.000

Parameters

- **Primary Source** - the name of the independent power source in the circuit that is to be stepped.
- **Primary Start** - the starting value for the primary power source.
- **Primary Stop** - the final value for the primary power source.
- **Primary Step** - specifies the incremental value to use over the defined sweep range.
- **Enable Secondary** - allows you to sweep the primary power source over its full range of values, for each value of a specified secondary source.
- **Secondary Name** - the name of a second independent power source in the circuit.
- **Secondary Start** - the starting value for the secondary power source.
- **Secondary Stop** - the final value for the secondary power source.
- **Secondary Step** - specifies the incremental value to use over the defined sweep range.

Notes

The primary source is required and the secondary source is optional.

The **Primary Source** and **Secondary Name** parameters are chosen from drop-down lists containing all power and excitation sources in the circuit.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

The simulation results are displayed on the **DC Sweep** tab of the Waveform Analysis window.

Examples
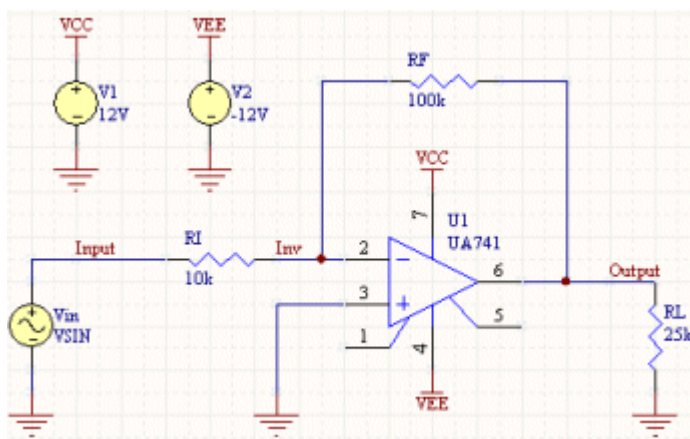
Consider the circuit in the image above, where a DC Sweep analysis is defined with the following parameter values:

- Primary Source = Vin
- Primary Start = -700.0m
- Primary Stop = -1.500
- Primary Step = -20.00m
- Secondary Name = V1
- Secondary Start = 10.00
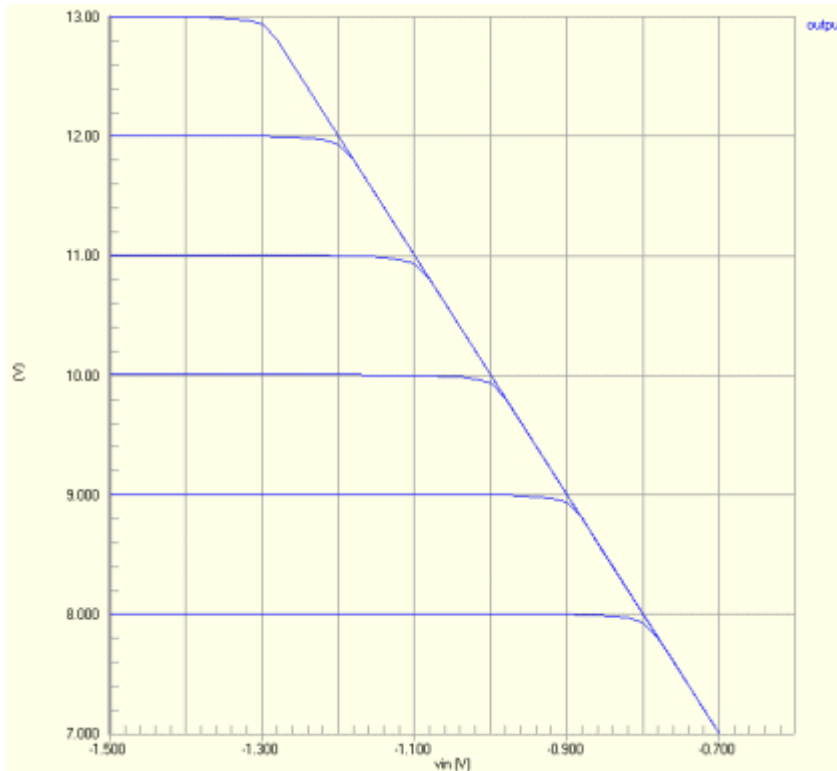- Secondary Stop = 15.00

- Secondary Step = 1.000

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.DC VIN -0.7 -1.5 -0.02 V1 10 15 1
```

and running the simulation will yield the output waveform shown in the image below:

AC Small Signal Analysis

Description

An AC Small Signal analysis generates output that shows the frequency response of the circuit, calculating the small-signal AC output variables as a function of frequency.It first performs an Operating Point analysis to determine the DC bias of the circuit, replaces the signal source with a fixed amplitude sine wave generator, then analyzes the circuit over the specified frequency range. The desired output of an AC Small Signal analysis is usually a transfer function (voltage gain, transimpedance, etc.).

Setup

AC Small Signal analysis is set up on the **AC Small Signal Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **AC Small Signal Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

AC Small Signal Analysis Setup	
Parameter	Value
Start Frequency	1.000
Stop Frequency	4.000
Sweep Type	Linear
Test Points	3
Total Test Points	3

Parameters

- **Start Frequency** - the initial frequency for the sine wave generator (in Hz).
- **Stop Frequency** - the final frequency for the sine wave generator (in Hz).
- **Sweep Type** - defines how the total number of test points is determined from the initial value assigned to the Test Points parameter. The following three types are available:
    - **Linear** - Total number of test points evenly spaced on a linear scale.
    - **Decade** - Number of evenly spaced test points per decade of a $\log_{10}$ scale.
    - **Octave** - Number of evenly spaced test points per octave of a $\log_2$ scale.
- **Test Points** - defines the incremental value for the sweep range, in conjunction with the chosen Sweep Type.
- **Total Test Points** (non-editable) - shows the total number of test points in the frequency sweep range, calculated from the initial value for Test Points and the chosen Sweep Type.

Notes

Before you can perform an AC Small Signal analysis, the circuit schematic must contain at least one signal source component with a value entered for the **AC Magnitude** parameter of its linked simulation model. It is this source that is replaced with a sine wave generator during the simulation.

The amplitude and phase of the swept sine wave are specified in the model parameters for the SIM model linked to the schematic component for the Source. To set these values, double-click on the source component in the schematic, to bring up the **Component Properties** dialog. In the **Models** region of the dialog, double-click on the entry for theassociated simulation model to launch the **Sim Model** dialog. When this dialog appears, select the **Parameters** tab to gain access to the **AC Magnitude** and **AC Phase** parameters. Enter the amplitude (in Volts) and the phase in (in Degrees). Units are not required. Set the **AC Magnitude** to 1 to have the output variables displayed relative to 0 dB.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

The simulation results are displayed on the **AC Analysis** tab of the Waveform Analysis window.

Examples
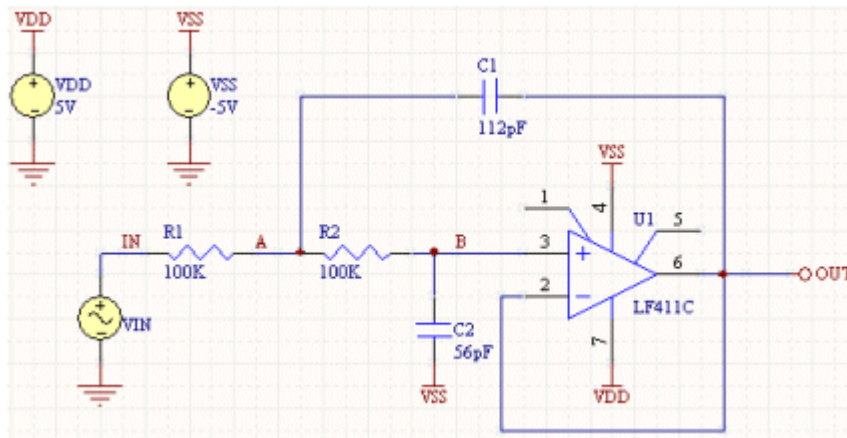
Consider the circuit in the image above, where an AC Small Signal analysis is defined with the following parameter values:

- Start Frequency = 1.000
- Stop Frequency = 1.000meg
- Sweep Type = Decade
- Test Points = 100
- Total Test Points = 601

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.AC DEC 100 1 1E6
```

and running the simulation will yield the output waveforms shown in the image below:
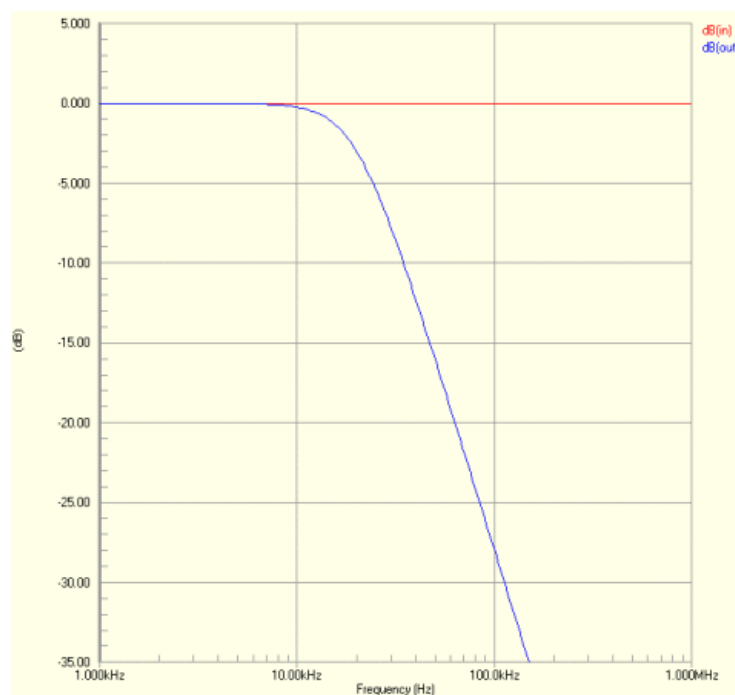
Impedance Plot Analysis

Description

An Impedance Plot analysis shows the impedance seen by any two-terminal source in the circuit.

Setup

An Impedance Plot does not have a separate setup page of its own and is normally run and plotted as part of an AC Small Signal analysis.

To include Impedance Plot analysis results in an AC Small Signal analysis, ensure that the **Collect Data For** field on the **General Setup** page of the **Analyses Setup** dialog, is set to one of the following:

- Node Voltage, Supply and Device Current
- Node Voltage, Supply Current, Device Current and Power
- Node Voltage, Supply Current and Subcircuit VARs
- Active Signals

Locate the source of interest in the **Available Signals** list and add it to the **Active Signals** list. The signal will appear with a [z] suffix, indicating that it is an impedance-based signal. For example, a source in the circuit with the designator VIn, would appear in the **Available Signals** list as VIn[z].

Notes

The impedance measurement is calculated from the voltage at the supply's positive terminal, divided by the current out of that same terminal.

To obtain an impedance plot of the circuit's output impedance, follow these steps:

1. Remove the source from the input.
2. Ground the circuit's inputs where the input supply was connected.
3. Remove any load connected to the circuit.
4. Connect a two-terminal source to the output, with the source's positive terminal connected to the output and its negative terminal connected to ground.
5. Setup the signals of interest as described previously and run the simulation.

The simulation results are displayed on the **AC Analysis** tab of the Waveform Analysis window.

Examples
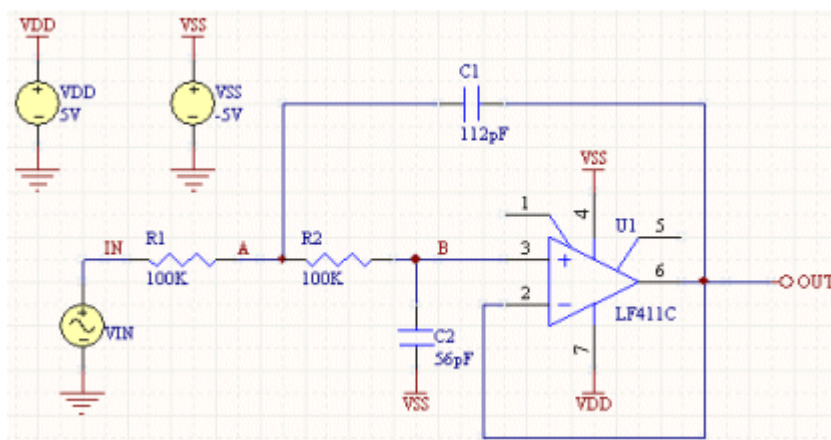
Consider the circuit in the above image, where an AC Small Signal analysis is to be run. To obtain an Impedance Plot analysis of the source VIN, the signal VIN[z] is taken across into the **Active Signals** list on the **General Setup** page of the **Analyses Setup** dialog.

Running the simulation yields the impedance plot shown in the image below:
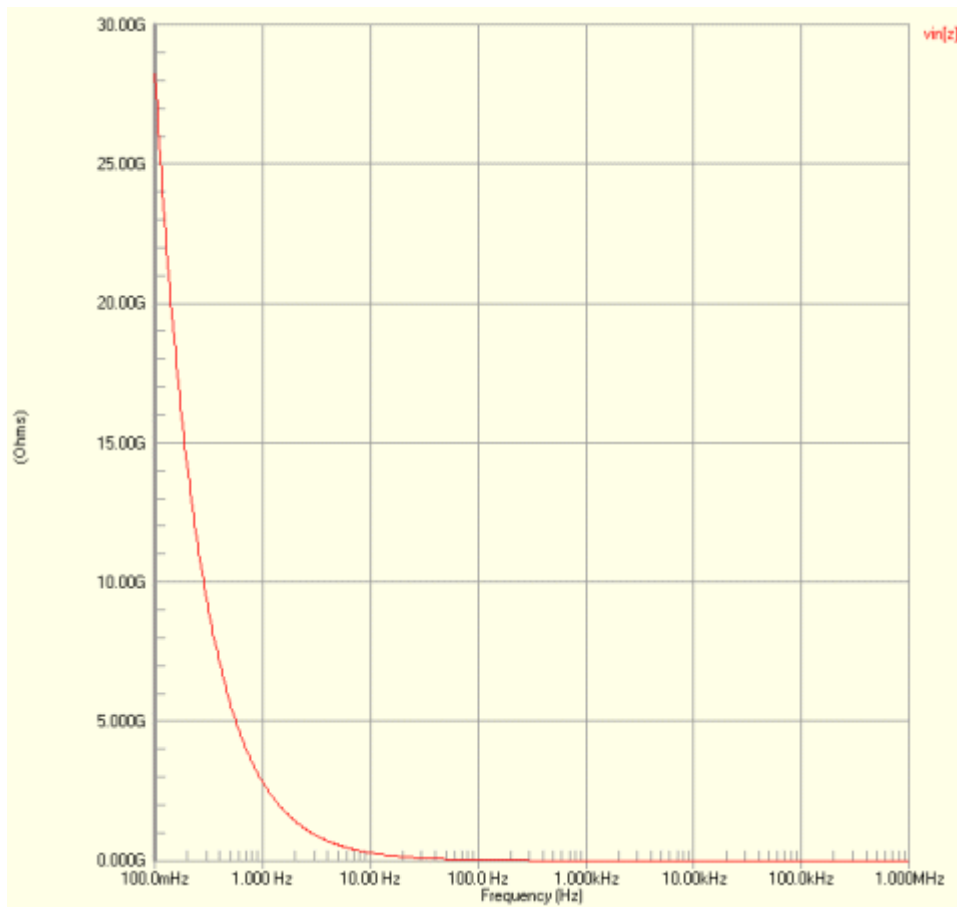
Noise Analysis

Description

Noise analysis lets you measure the noise contributions of resistors and semiconductor devices by plotting the Noise Spectral Density, which is the noise measured in Volts squared per Hertz (V^2/Hz).
Capacitors, inductors and controlled sources are treated as noise free.

The following noise measurements can be made:

Output Noise - the noise measured at a specified output node.

Input Noise - the amount of noise that, if injected at the input, would cause the calculated noise at the output. For example, if the output noise is 10p, and the circuit has a gain of 10, then it would take 1p of noise at the input to measure 10p of noise at the output. Thus the equivalent input noise is 1p.

Component Noise - the output noise contribution of each component in the circuit. The total output noise is the sum of individual noise contributions of resistors and semiconductor devices. Each of these components contributes a certain amount of noise, which is multiplied by the gain from that component's position to the circuit's output. Thus the same component can contribute different amounts of noise to the output, depending on its location in the circuit.

Setup

Noise analysis is set up on the **Noise Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Noise Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Noise Analysis Setup	
Parameter	Value
Noise Source	
Start Frequency	0.000
Stop Frequency	0.000
Sweep Type	Linear
Test Points	0
Points Per Summary	0
Output Node	
Reference Node	0
Total Test Points	0

Parameters

- **Noise Source** - an independent voltage source in the circuit which is to be used as an input reference for the noise calculations.
- **Start Frequency** - the initial frequency for the range over which to perform the noise calculations (in Hz).
- **Stop Frequency** - the final frequency for the range over which to perform the noise calculations (in Hz).
- **Sweep Type** - defines how the test points are distributed over the defined frequency range. The following three types are available:
- **Linear** - evenly spaced test points on a linear scale.
- **Decade** - evenly spaced test points per decade of a $\log_{10}$ scale.
- **Octave** - evenly spaced test points per octave of a $\log_2$ scale.
- **Test Points** - defines the number of points over the defined frequency range at which noise calculations will be performed.
- **Points Per Summary** - allows you to control which noise measurement is performed. Setting this parameter to 0 will cause input and output noise to be measured only. Setting to a 1 will measure the noise contribution of each component in the circuit.
- **Output Node** - the node in the circuit at which the total output noise is to be measured.
- **Reference Node** - the node in the circuit used as a reference for calculating the total output noise at the desired Output Node. By default, this parameter is set to 0 (GND). If set to any other node, the total output noise is calculated as:
  - **V(Output Node) -V(Reference Node)**
- **Total Test Points** (non-editable) - shows the total number of test points in the frequency sweep range, calculated from the initial value for Test Points and the chosen Sweep Type.

Notes

The **Start Frequency** must be greater than zero.

The independent voltage source specified in the **Noise Source** parameter must be an ac source in order for the simulation to proceed.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

The simulation results are displayed on the **Noise Spectral Density** tab of the Waveform Analysis window.

Examples
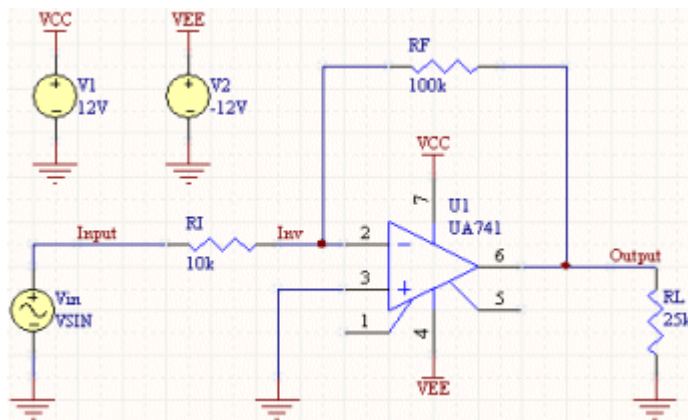
Consider the circuit in the image above, where a Noise analysis is defined with the following parameter values:

- Noise Source = Vin
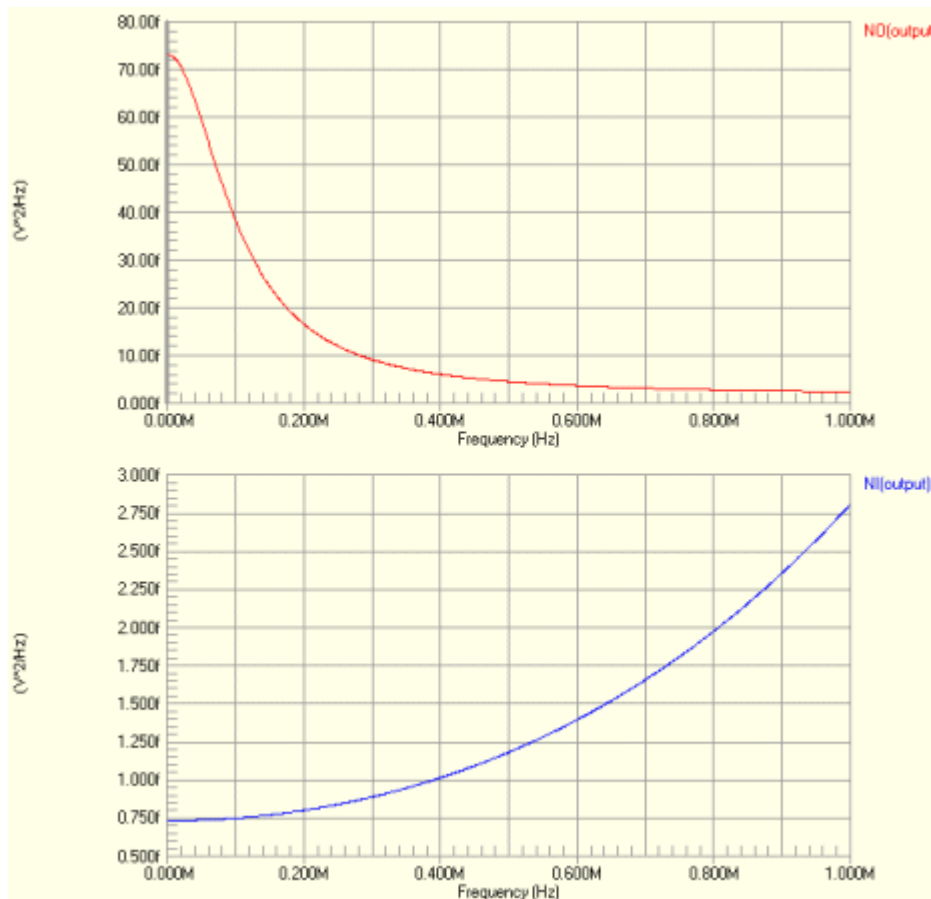- Start Frequency = 1.000k
- Stop Frequency = 1.000meg
- Sweep Type = Linear
- Test Points = 1000
- Points Per Summary = 0
- Output Node = Output
- Reference Node = 0 (GND)
- Total Test Points = 1000
-

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.NOISE V(OUTPUT) Vin LIN 1000 1000 1E6
```

and running the simulation will yield the output waveforms shown in the image below:

The top waveform shows the total output noise (NO) measured at the specified output node, in this case Output. The bottom waveform shows the amount of noise that would have to be injected at the input (NI) to obtain the measured output noise at this node.

If the **Points Per Summary** parameter had been set to 1 instead of 0, the output noise contribution of each applicable component in the circuit would have been measured and the corresponding waveforms for each made available in the **Sim Data** panel, ready for use in the Waveform Analysis window.

Pole-Zero Analysis

Description

Pole-Zero analysis enables you to determine the stability of a single input, single output linear system, by calculating the poles and/or zeros in the small-signal ac transfer function for the circuit. The dc operating point of the circuit is found and then linearized, small-signal models for all non-linear devices in the circuit are determined. This circuit is then used to find the poles and zeros that satisfy the nominated transfer function.

The transfer function can either be Voltage Gain (output voltage/input voltage) or Impedance (output voltage/input current).

Setup

Pole-Zero analysis is set up on the **Pole-Zero Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Pole-Zero Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Pole-Zero Analysis Setup	
Parameter	**Value**
Input Node	
Input Reference Node	0
Output Node	
Output Reference Node	0
Transfer Function Type	V(output)/V(input)
Analysis Type	Poles and Zeros

Parameters

- Input Node - the positive input node for the circuit.
- Input Reference Node - the reference node for the input of the circuit (Default = 0 (GND)).
- Output Node - the positive output node for the circuit.
- Output Reference Node - the reference node for the output of the circuit (Default = 0 (GND)).
- Transfer Function Type - defines the type of ac small-signal transfer function to be used for the circuit when calculating the poles and/or zeros. There are two types available:
- **V(output)/V(input) -** Voltage Gain Transfer Function.
- **V(output)/I(input) -** Impedance Transfer Function.
- Analysis Type - allows you to further refine the role of the analysis. Choose to find all poles that satisfy the transfer function for the circuit (**Poles Only**), all zeros (**Zeros Only**), or both **Poles and Zeros**.

Notes

Pole-Zero analysis works with resistors, capacitors, inductors, linear-controlled sources, independent sources, diodes, BJTs, MOSFETs and JFETs. Transmission lines are not supported.

The method used in the analysis is a sub-optimal numerical search. For large circuits it may take a considerable time or fail to find all poles and zeros. For some circuits, the method becomes "lost" and finds an excessive number of poles or zeros. If there is non-convergence in finding both poles and zeros, refine the analysis to calculate only poles or only zeros.

The simulation results are displayed on the **Pole-Zero Analysis** tab of the Waveform Analysis window.

Examples
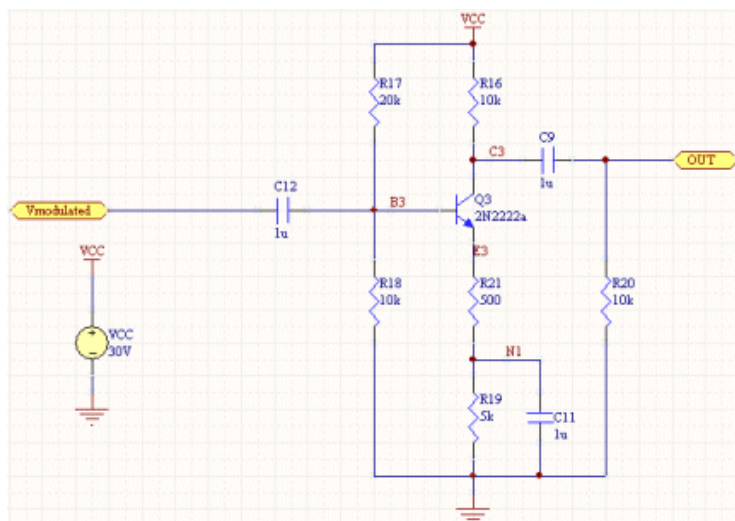
Consider the circuit in the image above, where a Pole-Zero analysis is defined with the following parameter values:

- Input Node = IN
- Input Reference Node = 0
- Output Node = OUT
- Output Reference Node = 0
- Transfer Function Type = V(output)/V(input)
- Analysis Type = Poles and Zeros

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.PZ IN 0 OUT 0 VOL PZ
```

and running the simulation will yield the output wave plot shown in the image below:
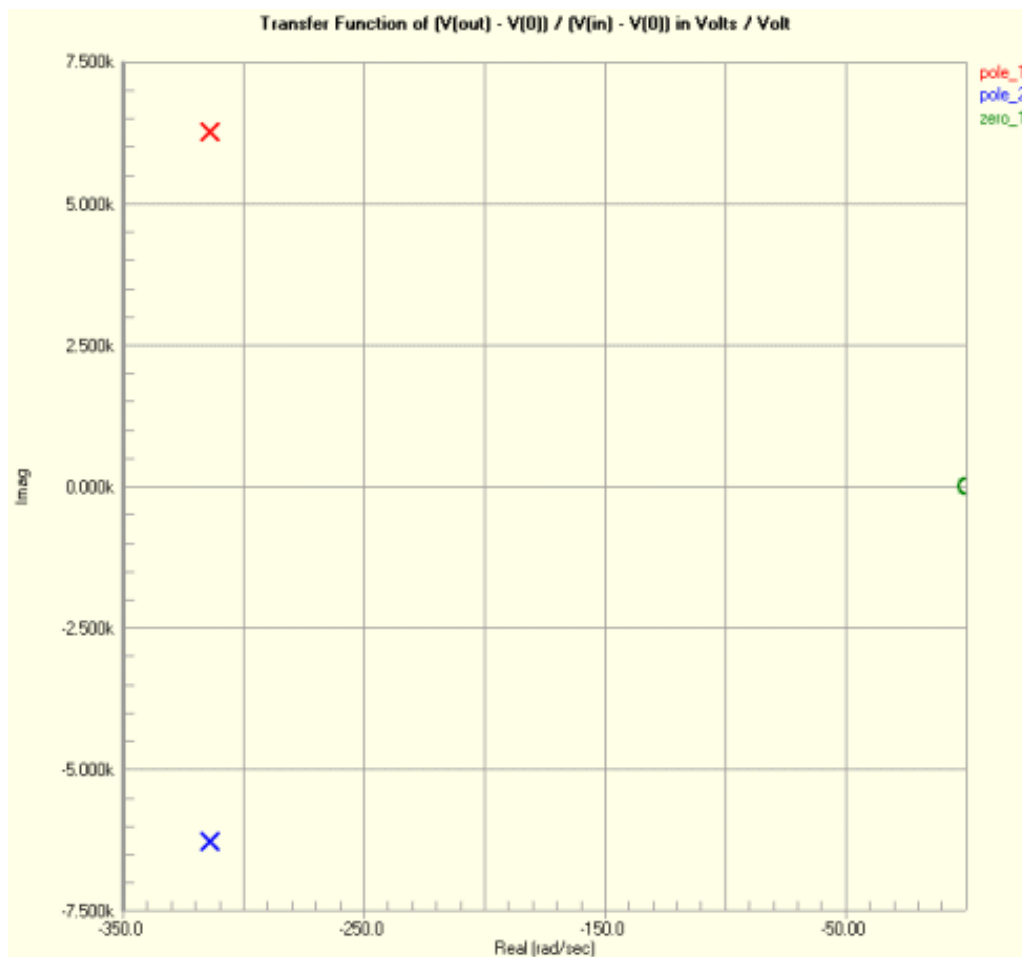
Transfer Function Analysis

Description

The Transfer Function analysis (DC small signal analysis) calculates the DC input resistance, DC output resistance and DC gain, at each voltage node in the circuit.

Setup

Transfer Function analysis is set up on the **Transfer Function Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Transfer Function Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Transfer Function Analysis Setup	
Parameter	Value
Source Name	
Reference Node	0

Parameters

- **Source Name** - the small signal input source used as the input reference for the calculations.
- **Reference Node** - the node in the circuit used as a reference for the calculations at each specified voltage node. By default, this parameter is set to 0 (GND).

Notes

Data is saved for all voltage nodes in the circuit only.

The three small signal calculations are:

- Gain (Transfer Function) - the ration of the voltage at the specific Voltage Node in the circuit to the input source defined by the **Source Name** parameter.
- Input resistance - measured at the input source defined by the **Source Name** parameter.
- Output resistance - measured across the specific Voltage Node in the circuit and the node defined by the **Reference Node** parameter.

The simulation results are displayed on the **Transfer Function** tab of the Waveform Analysis window.

Examples
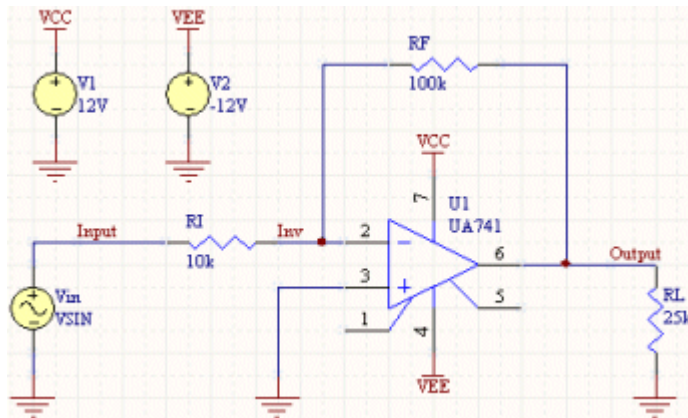
Consider the circuit in the image above, where a Transfer Function analysis is defined with the following parameter values:

- Source Name = Vin
- Reference Node = 0 (GND)

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.TF V(INPUT) Vin
.TF V(INV) Vin
.TF V(OUTPUT) Vin
.TF V(VCC) Vin
.TF V(VEE) Vin
```

If the nodes INPUT and OUTPUT are taken into the **Active Signals** list on the **General Setup** page of the **Analyses Setup** dialog, then the following data will be obtained upon running the simulation:

TF_V(OUTPUT)/VIN	-9.999	: Transfer Function for V(OUTPUT)/VIN
IN(OUTPUT)_VIN	10.00k	: Input resistance at VIN
OUT_V(OUTPUT)	15.38m	: Output resistance at OUTPUT
TF_V(INPUT)/VIN	1.000	: Transfer Function for V(INPUT)/VIN
IN(INPUT)_VIN	10.00k	: Input resistance at VIN
OUT_V(INPUT)	0.000	: Output resistance at INPUT

Temperature Sweep

Description

The Temperature Sweep feature is used to analyze the circuit at each temperature in a specified range, producing a series a curves, one for each temperature setting. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

Setup

Temperature Sweep is set up on the **Temperature Sweep Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Temperature Sweep** entry in the **Analyses/Options** list). The default setup for this feature is shown in the image below:

Temperature Sweep Setup	
Parameter	Value
Start Temparature	0.000
Stop Temparature	0.000
Step Temparature	0.000

Parameters

- **Start Temperature** - the initial temperature of the required sweep range (in Degrees C).
- **Stop Temperature** - the final temperature of the required sweep range (in Degrees C).
- **Step Temperature** - the incremental step to be used in determining the sweep values across the defined sweep range.

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Temperature Sweep analysis.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

Running a Temperature Sweep can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the **Analyses Setup** dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Temperature can also be varied using a Parameter Sweep. This is useful if you want to vary the temperature as either the primary or secondary parameter in a two-parameter sweep.

As running a Temperature Sweep actually performs multiple passes of the analysis (using a different value for the temperature with each pass), there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Temperature Sweep, the letter used is **t** and the number used identifies which pass the waveform relates to (e.g. **Output_t1**, **Output_t2**, etc)..

Examples
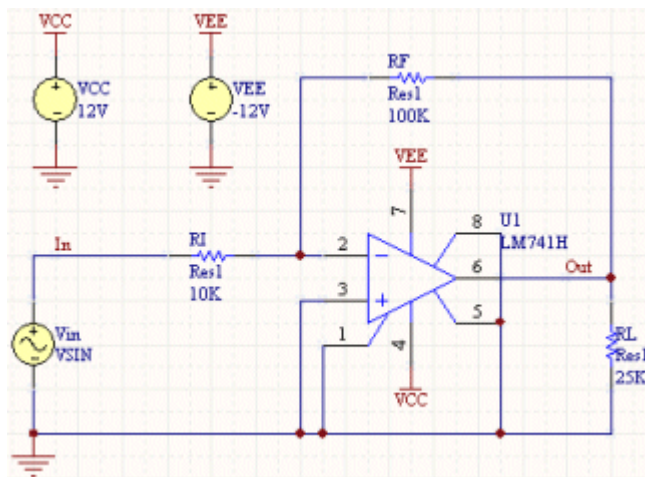
Consider the circuit in the image above, where an AC Small Signal analysis is to be performed in conjunction with the use of the Temperature Sweep feature. The Temperature Sweep is defined with the following parameter values:

- Start Temperature = 0.000
- Stop Temperature = 100.0
- Step Temperature = 25.00

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
```
.CONTROL
```
SWEEP OPTION[TEMP] 0 100 25
```
.ENDC

There will be five waveforms in all generated by the sweep (five different values for temperature across the defined sweep range, resulting in five separate simulation passes). The default value waveform will also be generated for comparison. Hence, running the simulation will yield the following waveforms with respect to the Out node:
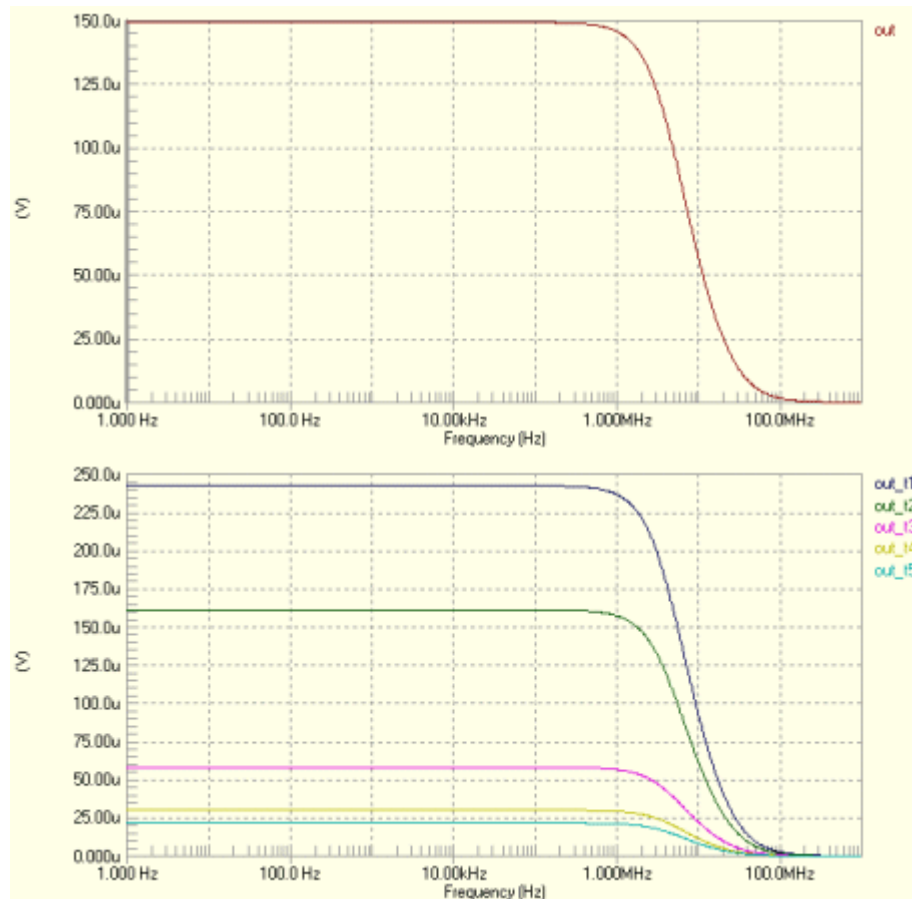
Parameter Sweep

Description

The Parameter Sweep feature allows you to sweep the value of a device in defined increments, over a specified range. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

The Parameter Sweep can vary basic components and models - subcircuit data is not varied during the analysis. You can also define a Secondary parameter to be swept. When a Secondary parameter
is defined the Primary parameter is swept for each value of the Secondary parameter.

Setup

Parameter Sweep is set up on the **Parameter Sweep Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Parameter Sweep** entry in the **Analyses/Options** list). The default setup for this feature is shown in the image below:

Parameter Sweep Setup	
Parameter	Value
Primary Sweep Variable	
Primary Start Value	0.000
Primary Stop Value	0.000
Primary Step Value	0.000
Primary Sweep Type	Absolute Values
Enable Secondary	☐
Secondary Sweep Variable	
Secondary Start Value	0.000
Secondary Stop Value	0.000
Secondary Step Value	0.000
Secondary Sweep Type	Absolute Values

Parameters

- **Primary Sweep Variable** - the device or parameter in the circuit whose value you wish to have swept. All possible variables that can be swept in the circuit are automatically loaded into a convenient drop-down list, from which to choose.
- **Primary Start Value** - the initial value for the Primary Sweep Variable.
- **Primary Stop Value** - the final value in the required sweep range for the Primary Sweep Variable.
- **Primary Step Value** - the incremental step to be used in determining the sweep values across the defined sweep range.
- **Primary Sweep Type** - set to Absolute Values to step through the defined sweep range exactly as entered (from Primary Start Value to Primary Stop Value) and thereby obtain a set of progressive, absolute values for the parameter. Set to Relative Values to add the values of the sweep range to the default value of the device or parameter, thereby creating a relative set of values for the parameter. For example, consider a parameter sweep defined a s follows:
  - Primary Sweep Variable = 10k resistor
  - Primary Start, Stop and Step values are 5k, 15k an 5k respectively
- With the Primary Sweep Type set to Absolute Values, the resulting resistor values would be used in the simulation passes:
  - **5k, 10k, 15k**
- If Relative Values is chosen instead, the resulting values used would be:
  - **15k, 20k, 25k**
- **Enable Secondary** - enables the use of a secondary parameter variable in the sweep. In this case, the Primary Sweep Variable is swept for each value of the secondary.
- **Secondary Sweep Variable** - the device or parameter in the circuit whose value you wish to have swept and used as a control to sweeping the Primary Sweep Variable. All possible variables that can be swept in the circuit are automatically loaded into a convenient drop-down list, from which to choose.
- **Secondary Start Value** - the initial value for the Secondary Sweep Variable.
- **Secondary Stop Value** - the final value in the required sweep range for the Secondary Sweep Variable.
- **Secondary Step Value** - the incremental step to be used in determining the sweep values across the defined sweep range.
- **Secondary Sweep Type** - as per Primary Sweep Type above, but applied to the generation of values to be used for the Secondary Sweep Variable.
-

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Parameter Sweep analysis.

The parameter to be swept can be a single designation or a designation with a device parameter in brackets. The following are some valid examples:

- RF - Resistor with designation RF
- Q3[bf] - Beta forward on transistor Q3

- R3[r] - Resistance of potentiometer R3
- option[temp] - Temperature
- U5[tp_val] - Propagation delays of digital device U5

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

Running a Parameter Sweep can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the **Analyses Setup** dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Normally you would use a Temperature Sweep to vary the temperature for simulation. However, temperature can also be varied in the Parameter Sweep. This is useful if you want to vary the temperature as either the primary or secondary parameter in a two-parameter sweep.

As running a Parameter Sweep actually performs multiple passes of the analysis (varying one or more circuit parameters with each pass), there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Parameter Sweep, the letter used is **p** and the number used identifies which pass the waveform relates to (e.g. **Output_p1**, **Output_p2**, etc). When you click on a waveform name in the Waveform Analysis window, the values used for the parameters in that pass of the sweep are displayed.

Examples
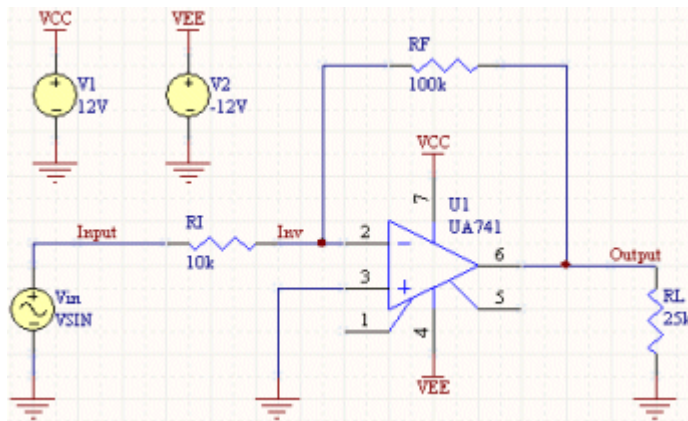
Consider the circuit in the image above, where AC Small Signal and Transient analyses are to be performed in conjunction with the use of the Parameter Sweep feature. The Parameter Sweep is defined with the following parameter values:

- Primary Sweep Variable = RF[resistance]
- Primary Start Value = 50.00k
- Primary Stop Value = 150.0k
- Primary Step Value = 50.00k
- Primary Sweep Type = Absolute Values
- Secondary Sweep Variable = RI[resistance]
- Secondary Start Value = 5.000k
- Secondary Stop Value = 15.00k
- Secondary Step Value = 5.000k
- Secondary Sweep Type = Absolute Values

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.CONTROL
SWEEP RF[resistance] 5E4 1.5E5 5E4 RI[resistance] 5000 1.5E4 5000
.ENDC
```

There are three values of the primary parameter that will be swept for each of the three values of the secondary. Therefore there will be nine waveforms in all generated by the sweep. The default value waveform will also be generated for comparison. Hence, running the simulation will yield the output waveforms shown in the images below:

AC Small Signal analysis
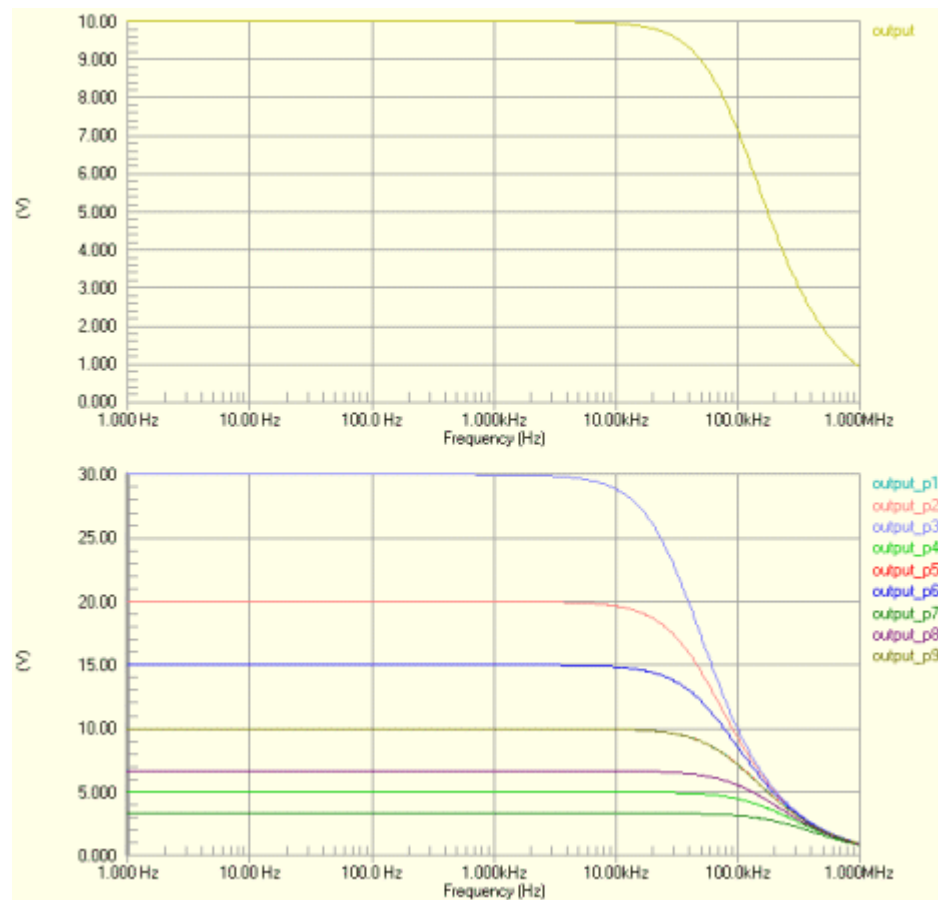
Transient analysis
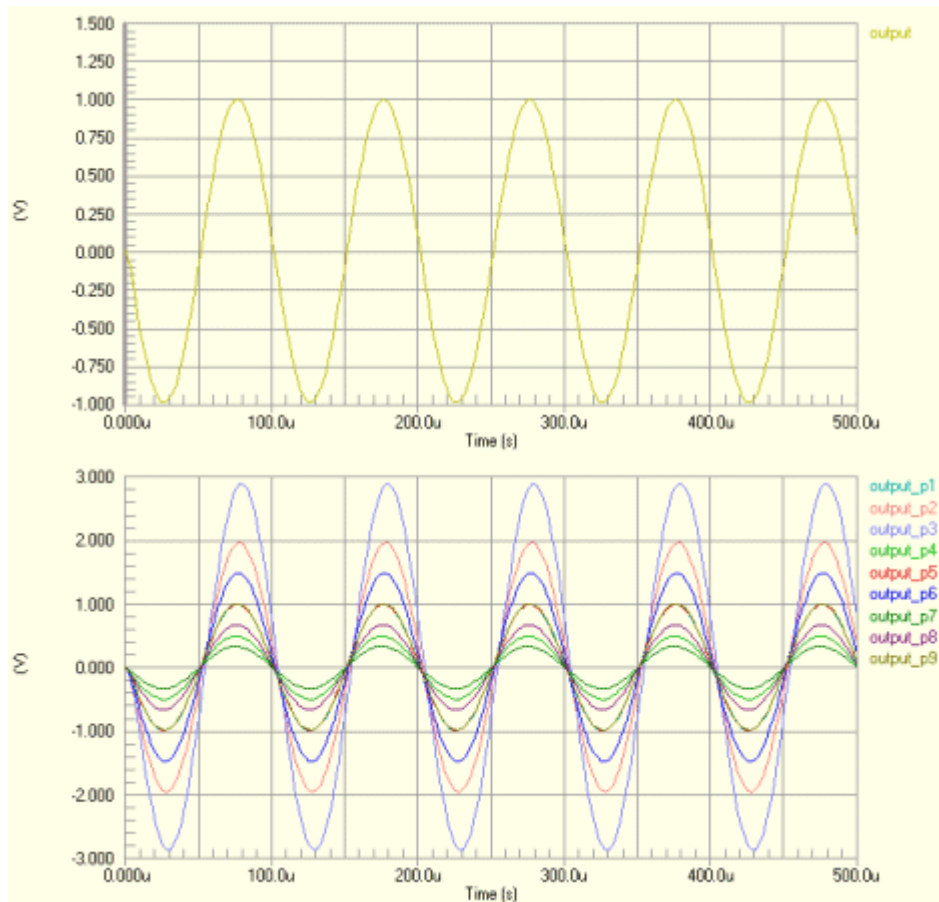
Monte Carlo Analysis

Description

Monte Carlo analysis allows you to perform multiple simulation runs with component values randomly varied across specified tolerances. The Simulator performs multiple passes of any of the standard analyses that are enabled (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise).

The Monte Carlo analysis can vary basic components and models - subcircuit data is not varied during the analysis.

Setup

Monte Carlo analysis is set up on the **Monte Carlo Analysis Setup** page of the **Analyses Setup** dialog (after the dialog appears, simply click the **Monte Carlo Analysis** entry in the **Analyses/Options** list). The default setup for this analysis type is shown in the image below:

Monte Carlo Analysis Setup	
Parameter	Value
Seed	-1
Distribution	Uniform
Number of Runs	5
Default Resistor Tolerance	10%
Default Capacitor Tolerance	10%
Default Inductor Tolerance	10%
Default Transistor Tolerance	10%
Default DC Source Tolerance	10%
Default Digital Tp Tolerance	10%
Specific Tolerances	0 defined...

Parameters

- **Seed** - this value is used by the Simulator to generate random numbers for the various runs of the analysis. If you want to run a simulation with a different series of random numbers, this value must be changed to another number. (Default = -1).

- **Distribution** - this parameter defines the distribution of values obtained during random number generation. Three distribution types are available:

- **Uniform** (Default)

- This is a flat distribution. Values are uniformly distributed over the specified tolerance range. For example, for a 1K resistor with a tolerance of 10%, there is an equal chance of the randomly generated value being anywhere between 900 Ω and 1100 Ω.

- **Gaussian**

- Values are distributed according to a Gaussian (bell-shaped) curve, with the center at the nominal value and the specified tolerance at +/- 3 standard deviations. For a resistor with a value of 1K +/- 10%, the center of the distribution would be at 1000 Ω, +3 standard deviations is 1100 Ω and -3 standard deviations is 990 Ω.

- With this type of distribution, there is a higher probability that the randomly generated value will be closer to the specified value.

- **Worst Case**

- This is the same as the Uniform distribution, but only the end points (worst case) of the range are used. For a 1K +/- 10% resistor, the value used would be randomly chosen from the two worst case values of 990 Ω and 1100 Ω. On any one simulation run there is an equal chance that the high-end worst case value (1100 Ω) or low-end worst case value (990 Ω) will be used.

- **Number of Runs** - the number of simulation runs you want the Simulator to perform. Different device values will be used for each run, within the specified tolerance range. (Default = 5).

- **Default Resistor Tolerance** - the default tolerance to be observed for resistors. The value is entered as a percentage (Default = 10%).

- **Default Capacitor Tolerance** - the default tolerance to be observed for capacitors. The value is entered as a percentage (Default = 10%).

- **Default Inductor Tolerance** - the default tolerance to be observed for inductors. The value is entered as a percentage (Default = 10%).

- **Default Transistor Tolerance** - the default tolerance to be observed for transistors (beta forward). The value is entered as a percentage (Default = 10%).

- **Default DC Source Tolerance** - the default tolerance to be observed for DC Sources. The value is entered as a percentage (Default = 10%).

- **Default Digital Tp Tolerance** - the default tolerance to be observed for Digital Tp (propagation delay for digital devices). The value is entered as a percentage (Default = 10%).

- The tolerance is used to determine the allowable range of values that can be generated by the random number generator for a device. For a device with nominal value Val_{Nom}, the range can be expressed as:
    - **Val_{Nom} - (Tolerance * Val_{Nom}) ≤ RANGE ≥ Val_{Nom} + (Tolerance * Val_{Nom})**

- **Specific Tolerances** - this parameter shows how many specific tolerances are currently defined. These are user-defined tolerances that are applied to specific components in the circuit. You can set up your own specific tolerances as required, by clicking the **...** button to the right of the field. The **Monte Carlo -**

Specific Tolerances dialog will appear. Specific tolerances that are defined will override the default tolerance settings. (Default = 0 defined).

Notes

At least one of the standard analysis types (AC, DC Sweep, Operating Point, Transient, Transfer Function, Noise) must be enabled in order to perform a Monte Carlo analysis.

The Monte Carlo analysis can vary basic components and models - subcircuit data is not varied during the analysis.

Data is saved for all signals in the **Available Signals** list, on the **General Setup** page of the **Analyses Setup** dialog.

Running a Monte Carlo analysis can result in a large amount of data being calculated. To limit the amount of data calculated, you can set the **Collect Data For** option on the **General Setup** page of the **Analyses Setup** dialog to **Active Signals**. With this option, data is only calculated for variables currently listed in the **Active Signals** list.

Each component is randomly varied independent of other components. For example, if a circuit has two 10 K resistors, and the default tolerance is set to 10%, then during the first pass of the simulation, one resistor might have a value of 953 Ω, and the other one could be 1022 Ω. The program uses a separate and independent random number to generate the value for each component.

As running a Monte Carlo analysis actually performs multiple passes of the enabled standard analyses, there is a special identifier used when displaying the waveforms in the Sim Data Editor's Waveform Analysis window. Each pass is identified by adding a letter and number as a suffix to the waveform name. For a Monte Carlo analysis, the letter used is **m** and the number used identifies which pass the waveform relates to (e.g. **Output_m1**, **Output_m2**, etc).

Examples
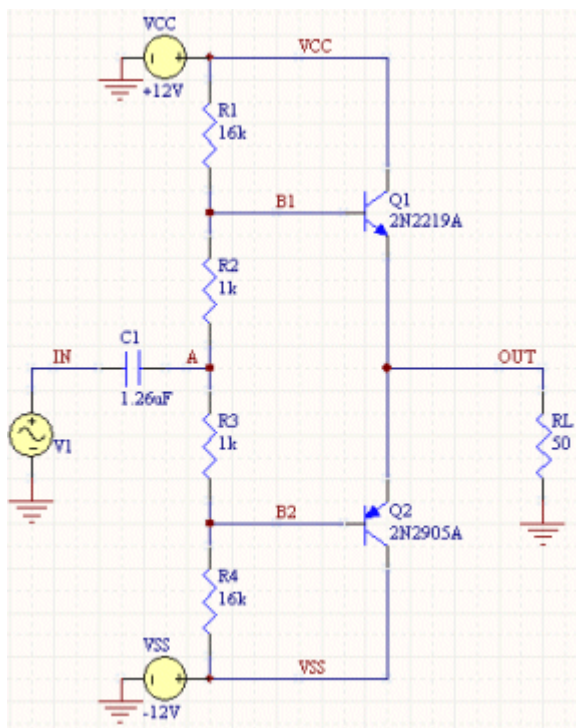
Consider the circuit in the image above, where a Transient analysis is to be performed in conjunction with the use of the Monte Carlo analysis feature. The Monte Carlo analysis is defined with the following parameter values:

- Seed = -1
- Distribution = Uniform
- Number of Runs = 5

- Default Resistor Tolerance = 15%
- Default Capacitor Tolerance = 15%
- All other parameters are left at their default values.

The entry in the SPICE netlist will be:

```
*Selected Circuit Analyses:
.CONTROL
TOL C1 DEV=15% Uniform
TOL Q1[BF] DEV=10% Uniform
TOL Q2[BF] DEV=10% Uniform
TOL R1 DEV=15% Uniform
TOL R2 DEV=15% Uniform
TOL R3 DEV=15% Uniform
TOL R4 DEV=15% Uniform
TOL RL DEV=15% Uniform
TOL V1 DEV=10% Uniform
TOL VCC DEV=10% Uniform
TOL VSS DEV=10% Uniform
MC 5 SEED=-1
.ENDC
```

and running the simulation will yield waveforms for the OUT signal as shown in the image below:
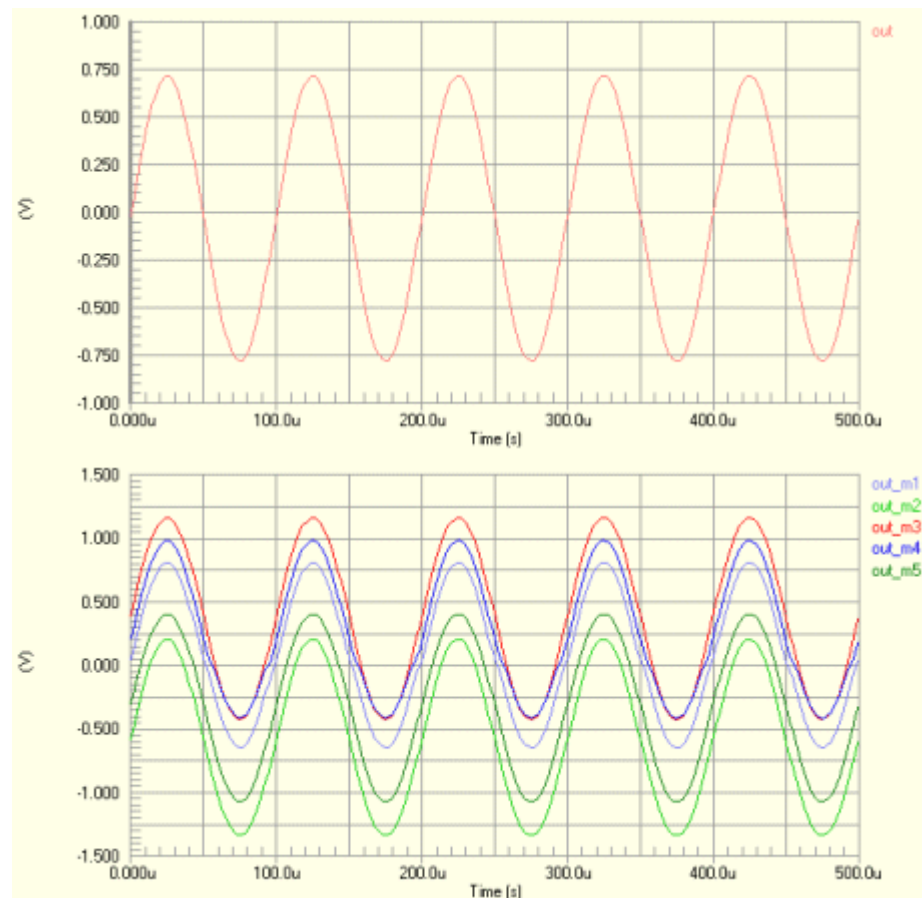
References

1. www.kxcad.net (2011)
2. Altium Designer help files (2008-2009)