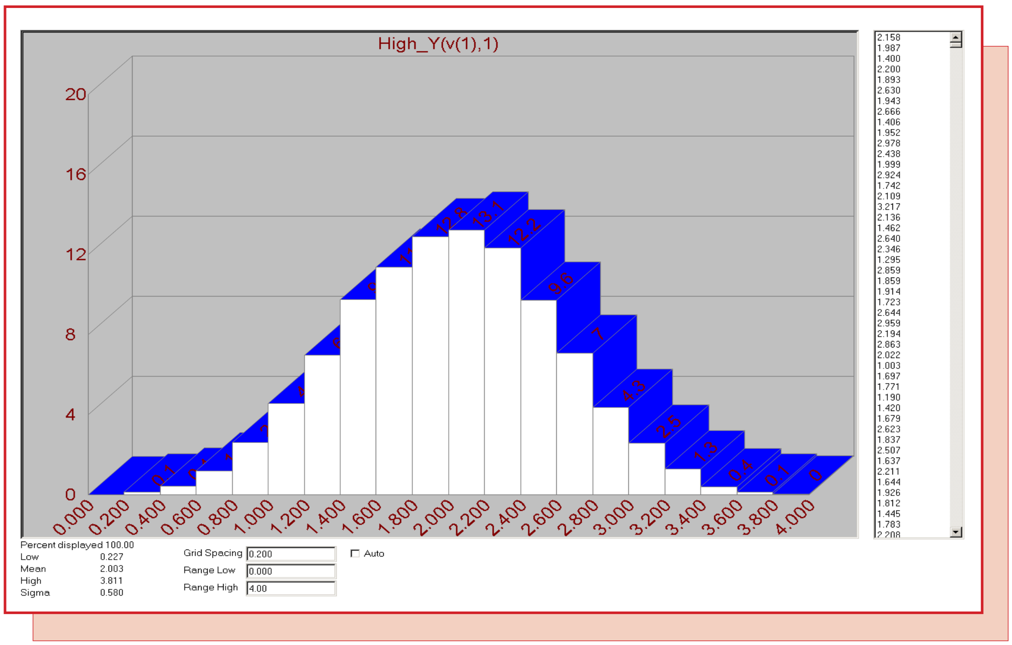
spectrum news

Applications for Micro-Cap™ Users

Fall 2005

News



Gaussian Distribution for Random Values

Featuring:

- Setting a Gaussian Distribution for Random Values
- Quasi Small Signal Analysis
- SPICE3 Operators Versus Standard Boolean Operators

News In Preview

This newsletter's Q and A section describes how to set the default path that will be used when the Open File dialog box is invoked. The Easily Overlooked Features section describes the use of the Cleanup command to delete noncritical files from the data and library directories.

The first article describes how to use the linearly distributed random functions to create a Gaussian distributed random expression with a technique derived from the Central Limit Theorem.

The second article describes how to run a quasi small signal analysis to simulate the frequency response of circuits using the Fourier and performance plot capabilities available within a transient analysis.

The third article describes the differences between the SPICE3 boolean operators and the standard boolean operators that are available within Micro-Cap and how to deal with the syntax conflicts that may arise with the use of these.

Contents

Book Recommendations

General SPICE

- *Computer-Aided Circuit Analysis Using SPICE*, Walter Banzhaf, Prentice Hall 1989. ISBN# 0-13-162579-9

- *Macromodeling with SPICE*, Connelly and Choi, Prentice Hall 1992. ISBN# 0-13-544941-3

- *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski, McGraw-Hill, First Edition, 1993. ISBN# 0-07-911525-X

- *The SPICE Book,* Andrei Vladimirescu, John Wiley & Sons, Inc., First Edition, 1994. ISBN# 0-471-60926-9

MOSFET Modeling

- *MOSFET Models for SPICE Simulation, William Liu, Including BSIM3v3 and BSIM4*, Wiley-Interscience, First Edition, ISBN# 0-471-39697-4

VLSI Design

- *Introduction to VLSI Circuits and Systems,* John P. Uyemura, John Wiley & Sons Inc, First Edition, 2002 ISBN# 0-471-12704-3

Micro-Cap - Czech

- *Resime Elektronicke Obvody,* Dalibor Biolek, BEN, First Edition, 2004. ISBN# 80-7300-125-X

Micro-Cap - German

- *Schaltungen erfolgreich simulieren mit Micro-Cap V,* Walter Gunther, Franzis', First Edition, 1997. ISBN# 3-7723-4662-6

Micro-Cap - Finnish

- *Elektroniikkasimulaattori,* Timo Haiko, Werner Soderstrom Osakeyhtio, 2002. ISBN# ISBN 951-0-25672-2

Design

- *Microelectronic Circuits High Performance Audio Power Amplifiers,* Ben Duncan, Newnes, First Edition, 1996. ISBN# 0-7506-2629-1

- *Microelectronic Circuits.,* Adel Sedra, Kenneth Smith, Fourth Edition, Oxford, 1998

High Power Electronics

- *Power Electronics,* Mohan, Undeland, Robbins, Second Edition, 1995. ISBN# 0-471-58408-8

- *Modern Power Electronics,* Trzynadlowski, 1998. ISBN# 0-471-15303-6

Switched-Mode Power Supply Simulation

- *SMPS Simulation with SPICE 3,* Steven M. Sandler, McGraw Hill, First Edition, 1997. ISBN# 0-07-913227-8

- *Switch-Mode Power Supply SPICE Simulation Cookbook*, Christophe Basso, McGraw-Hill 2001. This book describes many of the SMPS models supplied with Micro-Cap.

Micro-Cap Questions and Answers

Question: I store all of my circuit files in a directory called:

C:\MINE

When I select the Open File command, the initial path set in the dialog box is:

C:\MC8\DATA

I then navigate to my circuit directory and load one of my circuits. The next time I enter the Open File dialog box, the initial path has reverted back to the DATA directory. Is there a way to set this initial path so that the Open File dialog box will default to my circuit directory?

Answer: In Micro-Cap, all of the path information can be accessed by selecting the Paths command under the File menu. In the Path dialog box, there is an Initial Path option that will define the method that Micro-Cap uses to set the initial path in the Open File dialog box. The following three options are available:

Use Defined Path: The initial path in the Open File dialog box will be set to the path defined in the appropriate path field within this dialog box. For circuit files, the Data field will be used. If multiple paths are listed in the field, the first path will be chosen.

Use Last Path: The initial path in the Open File dialog box will be set to the last path from which a similar file was opened. Data file, library file, and graphic file types will all have their own last path stored.

Use Current Path: The initial path in the Open File dialog box will be set to the same path as the currently active file in Micro-Cap. The currently active file is the file that is enabled under the Windows menu or whose window has been selected.

There are two methods for setting the initial path to your C:\MINE directory. The first is to enter the path into the Data field in this dialog box. Either overwrite the current path or place it in front of the current path. If multiple paths are defined in the field, they will need to be separated with a semi-colon. Then make sure that the Initial Path option is set to Use Defined Path. As long as the C:\MINE path is the first (or only) path in the Data field, every time the Open File dialog box is invoked, the C:\MINE directory will come up as the default path. This procedure is best if all of the circuit files will be stored in a single directory.

The second method is to set the Initial Path option to Use Last Path. When the Open File dialog box is invoked, the default path for opening a circuit file will be the last path that a circuit file was loaded from. If the circuit files are always loaded from the MINE directory, then the default path will consistently be C:\MINE. This procedure is best if circuit files are stored in different project directories rather than just a single directory as there would be no need to continually update the Data field in the Paths dialog box each time a new project is started.

Easily Overlooked Features

This section is designed to highlight one or two features per issue that may be overlooked because they are not made visually obvious with a toolbar button.

Cleanup Command

During the normal course of operation, Micro-Cap can create multiple files that are associated with a circuit file or a simulation. File types such as numeric output, circuit backup, index, probe, Monte Carlo statistic, and Bill of Material files are among the associated files that may be created while working with a circuit. Over time, the number of these files can start adding up. Except for the probe files, most of these files will take up relatively little hard drive space. However, they can add to the clutter of a typical directory. Rather than having to hunt down and delete each of these files through Windows Explorer, Micro-Cap provides a Cleanup command that provides a simple way to delete these files en masse.

When the Cleanup command is selected from the File menu, the dialog box in Figure 1 will appear. Micro-Cap will search through all of the paths specified in the Data and Model Library fields within the Path dialog box which is available through the Paths command under the File menu. The left side of the Clean Up dialog box will display a list of all of the file types that were found during the search. None of these file types are critical to a simulation. Clicking on one of the file types on the left will then display a list of files of that type that were found during the search. A file can be selected in the list by clicking on it. The Ctrl and Shift keys can be used when clicking to select multiple files. The number of selected files along with their corresponding size is displayed below the file list. Clicking the Delete button will then delete all of the selected files.

The Select All button will select all of the files in all of the groups. This button provides a simple method to deleting all of the files that are present in the dialog box.

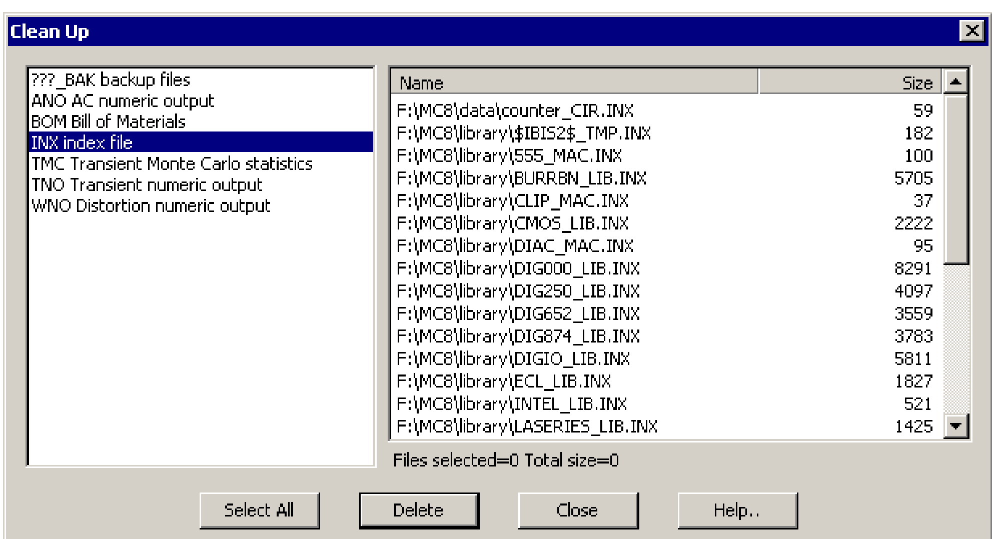


Fig. 1 - Clean Up dialog box

Setting a Gaussian Distribution for Random Values

A Gaussian distribution for a random value can be useful for many applications, particularly those in the communications arena. However, the random functions available within Micro-Cap all have a linear distribution to them. Fortunately, there is a procedure that can be used to create a Gaussian distributed random expression from the random functions that already exist in Micro-Cap. (Thanks to Hugh Stevenson for describing this procedure in the Micro-Cap Yahoo User Group. You can subscribe to the User Group by emailing the micro-cap-subscribe@yahoogroups.com address.) According to the Central Limit Theorem, linearly distributed random functions can be combined in order to produce a Gaussian distribution. The Central Limit Theorem consists of the following:

1) The mean of the sampling distribution is equal to the mean of the population from which the samples were drawn.
2) The variance of the sampling distribution is equal to the variance of the population from which the samples were drawn divided by the size of the samples.
3) If the original population has a Gaussian distribution, the sampling distribution will also be Gaussian. If the original population does not have a Gaussian distribution, the sampling distribution will increasingly approximate a Gaussian distribution as the sample size increases.

The third point is of the most interest in this case. It states that when a number of variables that have non-Gaussian distributions are combined into a single sample, the distribution of the sample will become closer to Gaussian distribution as the number of variables used increases. This provides us with the means to use the linearly distributed random functions to create a Gaussian distribution. Micro-Cap has four functions which will return a random number between zero and one. The frequency at which a new random number is calculated is dependent on the function selected as follows:

RND - Returns a new random value every timepoint.
RNDR - Returns a new random value at the start of each Run command.
RNDC - Returns a new random value at the start of each branch of a simulation.
RNDI(t) - Returns a new random value every t seconds of simulation time.

The use of a single one of these random operators produces a linear distribution. Combining two instances of them will create a triangular distribution. Combining three or more instances of these functions will begin to produce the familiar bell shape associated with a Gaussian distribution.

Testing this procedure is simple. It only requires a single nonlinear function voltage source (NFV) in the schematic. For this example, the VALUE attribute of the NFV source has been defined as:

Rndc+Rndc+Rndc+Rndc

Each instance of the RNDC function will return a new random value at the beginning of each branch of a temperature stepping, parameter stepping, or Monte Carlo run. The RNDC function was selected for the example because it provides an easy means of creating a histogram that will display the distribution associated with the function source expression since the value of the defined expression will be constant over the course of a single branch of a Monte Carlo simulation. Using the RND and RNDI functions in a similar manner would produce a similar distribution result, although it would be more difficult to display an applicable histogram. Since the RNDR returns a new random value only at the initiation of the run command, it is not really appropriate for this method.

The example circuit containing the NFV source is then run through a transient analysis. The Monte Carlo option is enabled for this simulation. The Number of Runs field in the Monte Carlo options was set to 20000 for the simulation which will provide 20000 random values created by the function source to test the distribution on. The Distribution to Use option in the Monte Carlo options was left on Gaussian. This distribution option only affects parameters that have been toleranced using either the LOT or DEV keywords. It will have no effect on the distribution of the random functions.

The resulting histogram from the Monte Carlo simulation is displayed in Figure 2. The histogram was created using the following performance function:

High_Y(V(1),1)

This performance function selects the highest value from each branch of the V(1) waveform which is the function source output. Since the function source expression will produce a constant value over a single Monte Carlo branch, the highest value will just be the value of the expression in the branch. This histogram shows the results of applying the Central Limit Theorem using the random functions available in Micro-Cap. As can be seen in the histogram, the distribution has become Gaussian even though all of the random functions in the expression have a linear distribution.

In testing this procedure, it appears that a minimum of four instances of the random function is necessary to produce a reasonable Gaussian distribution. Increasing the number of random function used in the expression should produce a closer distribution to Gaussian. The histogram in Figure 3 displays the results of summing eight RNDC functions using the same analysis specifications.

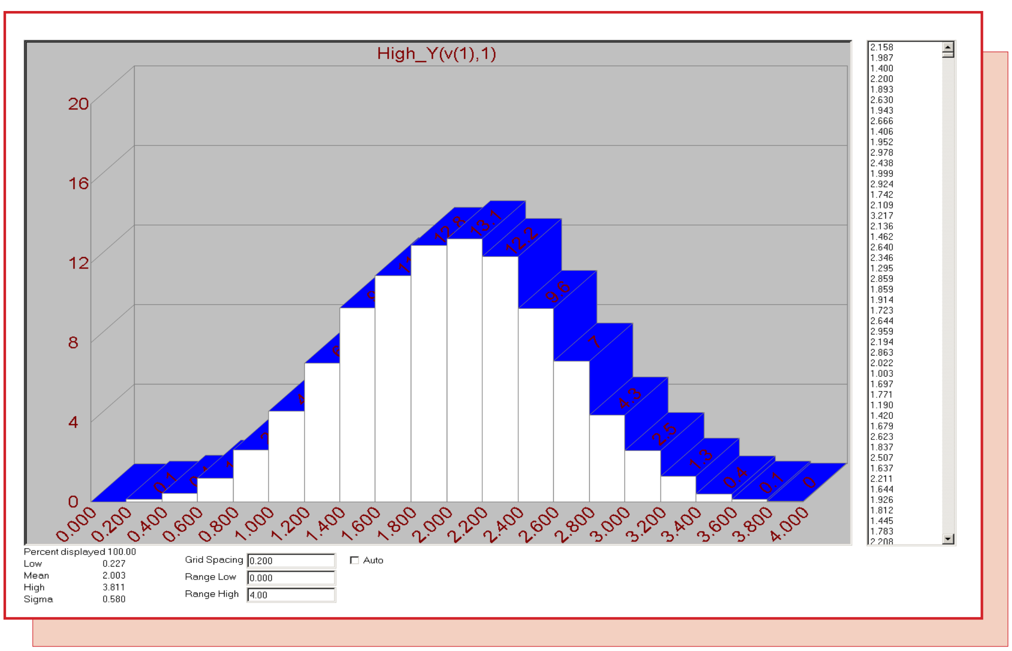


Fig. 2 - Four RNDC example histogram

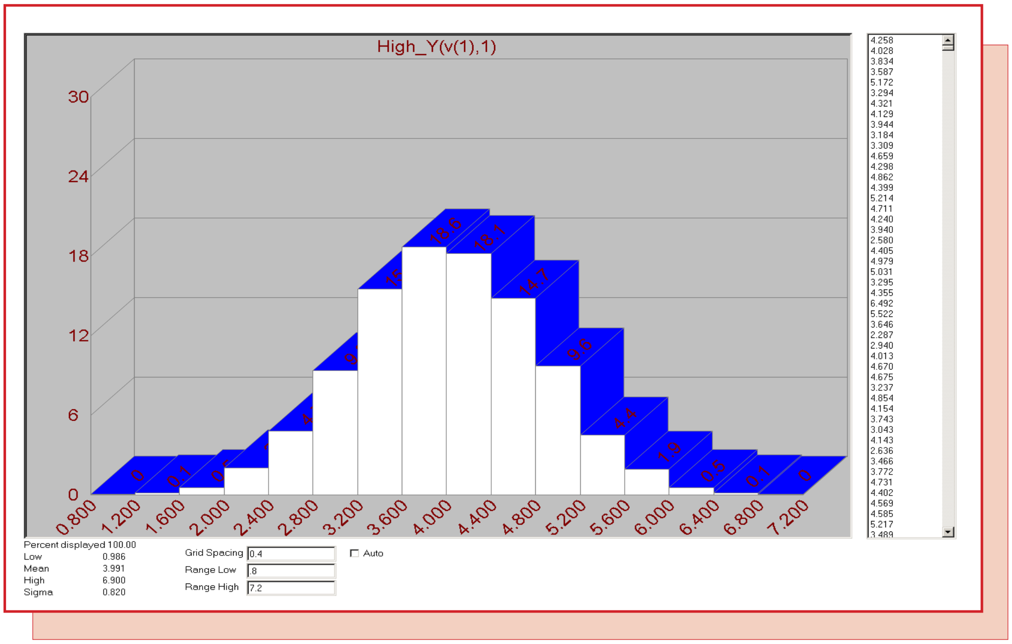


Fig. 3 - Eight RNDC example histogram

While the range of a single random function is between zero and one, note that the range of the function source expression in the above examples varies between zero to four and zero to eight depending on the number of random functions used. In most applications, a scale factor and/or an offset would need to be utilized to compensate for the use of multiple random functions in the expression.

Quasi Small Signal Analysis

The standard AC analysis is a small signal analysis that calculates the DC operating point of a circuit and then linearizes the devices about the operating point values. For AC analysis to produce reasonable results, the operating point values should be characteristic of the circuit's standard mode of operation such as the linear mode of operation for an opamp circuit. With switching circuits, there are commonly two modes of operation that an AC response would have to take into account. Since a standard AC analysis can only take into account a single mode, different methods must be used to obtain a frequency response.

One method is to use average models of all the switching components in the schematic. Average models average the state equations of the two switch positions over a switching cycle, but they are not very common and can be difficult to create. The method that will be described in this article is a quasi small signal analysis that uses the Fourier capabilities of transient analysis to convert a nonlinear simulation into its frequency domain equivalent.

The example circuit used to demonstrate this technique is displayed in Figure 4. The circuit is a simple low pass RC filter. A linear circuit is used in this example instead of a switching circuit in order to be able to compare the quasi small signal analysis to the standard AC analysis. In addition to the RC filter, the only other component in the schematic is a nonlinear function voltage source whose main attributes are defined as:

VALUE = sin(2*PI*FS1*T)
FREQ = 1

The Freq attribute, if defined, has priority over the Value attribute during an AC analysis, and in this case, it defines a 1 volt small signal source for an AC simulation run. The Freq attribute will be ignored during a transient simulation. The Value attribute defines a one volt peak amplitude sine wave signal

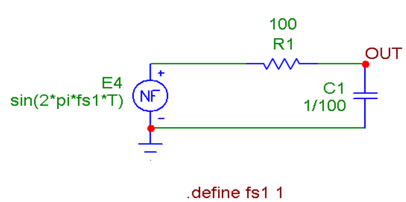


Fig. 4 - Quasi Small Signal example circuit

during a transient run whose frequency is set by the symbolic variable FS1. FS1 has its value set through a define statement present in the schematic. The standard AC analysis is displayed in Figure 5. The magnitude of the output voltage is the top plot, and the phase of the output voltage is the bottom plot. This plot will be the benchmark for the quasi small signal results.

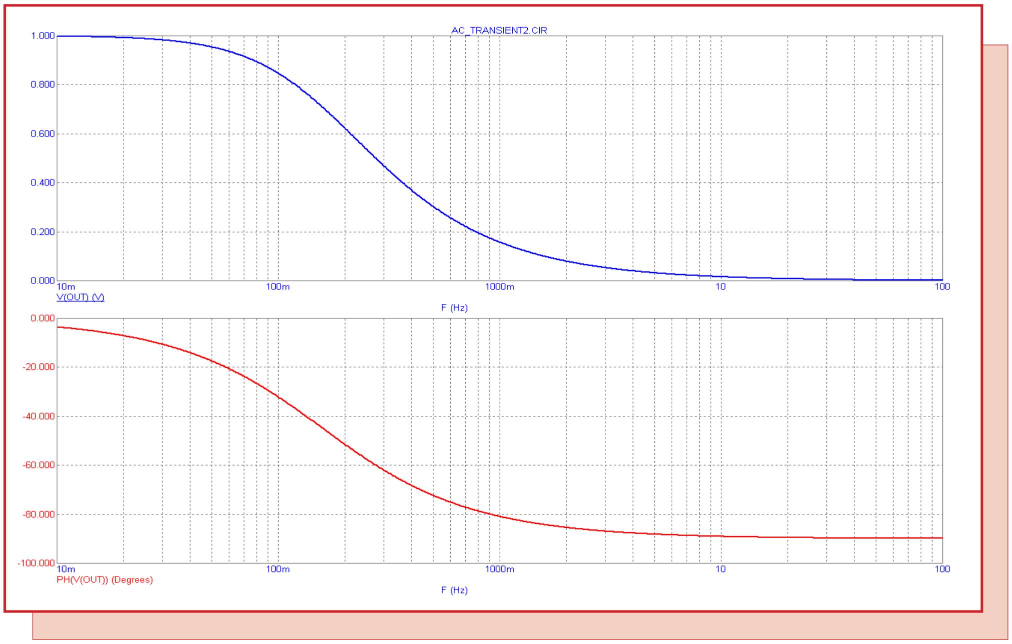


Fig. 5 - Standard AC analysis simulation

The quasi small signal analysis is simulated through the transient analysis capabilities of Micro-Cap. For transient analysis, the symbolic variable FS1 will be stepped in order to obtain the circuit's output waveform at different frequencies of operation. In the Stepping dialog box, the List method has been selected for FS1, and the list of values to be stepped through has been defined as:

.01,.02,.04,.08,.1,.2,.4,1,2,5,10,25

One limitation of the quasi small signal analysis method is that it has to run a full transient simulation for each frequency, so the run time can get lengthy. Therefore, both the frequency range and the number of frequency samples will tend to be limited when compared to the standard AC analysis. For this example, three to four frequency samples will be taken for each decade as the range varies from .01Hz to 25Hz.

The next step is to determine the appropriate Time Range and Maximum Time Step values that will provide a good sample across all of the different frequencies. Since the smallest frequency being simulated is .01Hz, the minimum time range for a single cycle at that frequency must be 100s. All of the other frequencies specified also share 100s as a common period multiple so this can be the sampling range that will be used. However, since there is an initial startup transient that will need to be ignored for a proper Fourier analysis, the Time Range field will be set to 200s so that the waveforms are given time to settle. In the FFT page of the Analysis Properties dialog box, the Upper Time Limit value is set to 200 and the Lower Time Limit value is set to 100. These settings will exclude the initial transient from the Fourier analysis and set the range that the Fourier analysis will sample to the last hundred seconds of the simulation. With this range, the

Fourier will produce a frequency sample every .01Hz. Therefore, the Number of Points field in the FFT page has been set to 65536 to ensure that the Fourier samples will be adequate for the higher frequencies being simulated.

For the Maximum Time Step, the value will be set according to the highest frequency being simulated. In this case, the highest frequency has been specified as 25Hz which translates into a period of 40ms. Setting the Maximum Time Step to 1ms will produce a minimum of forty data points per cycle at 25Hz and will provide a good sampling for all of the frequencies being simulated. The resultant transient analysis is displayed in Figure 6.

The top plot is the actual voltage of node Out at each frequency step. The first 100s of simulation data has been discarded due to the settings in the FFT page of the Analysis Properties dialog box.

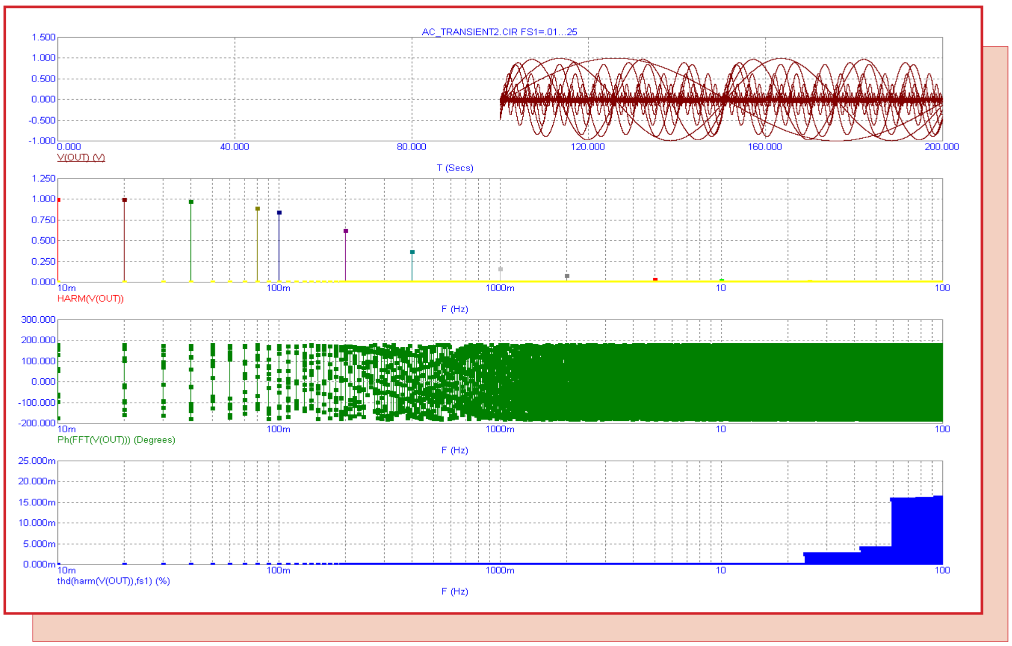


Fig. 6 - Transient analysis simulation

The second plot displays the harmonic content of each of the stepped output waveforms. The third plot displays the phase output of the Fourier plots for each of the stepped output waveforms, and the bottom plot shows the total harmonic distortion of node Out at each frequency step. Note that the Thd operator has had its optional reference frequency parameter defined as FS1 so that the distortion will be calculated versus the frequency of operation for each step. Although the distortion in this analysis is due to timestep aliasing, there is one slight difference in the following technique for the Thd operator that makes it a worthwhile addition to this example.

Though all of the important data is displayed on the screen, it is not in an easily readable format in determining the quasi small signal response. A performance plot can be used to extract the appropriate data and provide a better visualization of the frequency information. Performance plots can be generated by right clicking on the waveform name in the plot or by selecting the menu option Transient/Performance Windows/Add Performance Window. For this example, the performance plot created is displayed in Figure 7. The plot has been defined to display the following three waveforms:

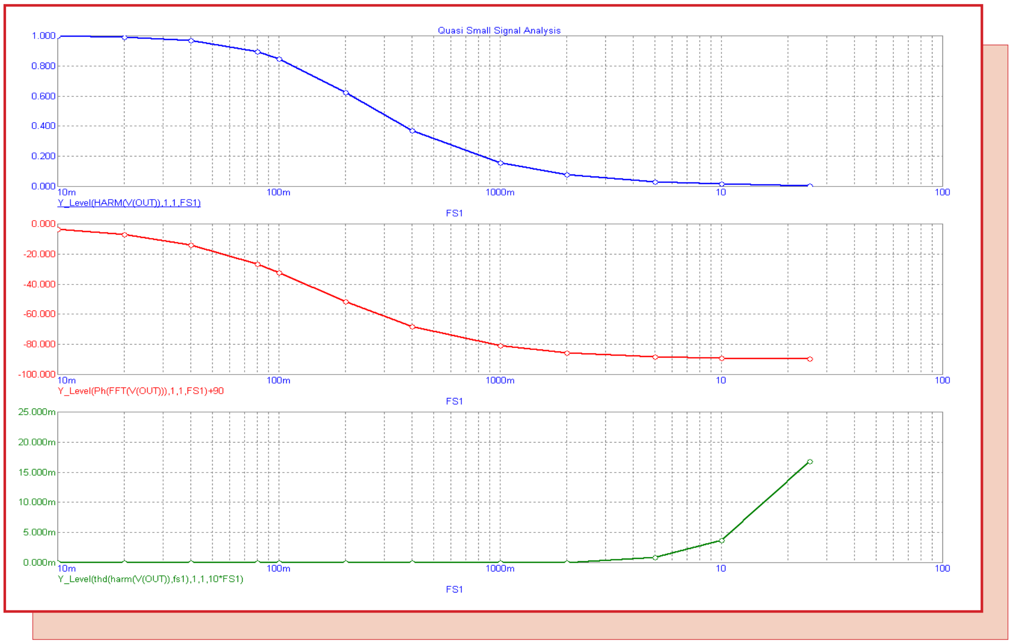


Fig. 7 - Quasi Small Signal performance plot

Y_Level(HARM(V(OUT)),1,1,FS1)
Y_Level(PH(FFT(V(OUT))),1,1,FS1)+90
Y_Level(THD(HARM(V(OUT)),FS1),1,1,10*FS1)

The Y_Level operators will return the Y value of each stepped waveform at the X value that is specified within the expression. The top waveform plots the value of the harmonic of V(Out) at frequency FS1 for each stepped frequency. The plot reproduces the AC gain response and matches the V(Out) waveform from the standard AC analysis simulation.

The middle waveform plots the value of the phase of the Fourier response of V(Out) at frequency FS1 for each stepped frequency. This plot reproduces the AC phase response and matches the Ph(V(Out)) waveform from the standard AC analysis simulation. Note that an offset of 90 degrees was added to the waveform expression. This offset compensates for the use of the cosine expressions within the Fourier mathematical routines and aligns the phase plot with the typical AC results.

The bottom waveform plots the total harmonic distortion level at frequency 10*FS1 for each stepped frequency. Since FS1 is the reference frequency at which the distortion is measured against, the data must be extracted from a frequency greater than FS1. Setting the X value to 10*FS1 will include the second through tenth harmonics in the distortion calculation. One caveat to this procedure is that enough points must be calculated for the Fourier plots. For example, if the Number of Points field was set to 8192 in the FFT page, at the .01Hz sampling used in this example, the frequency information would only extend out to 40.96Hz. There would not be data available at 10*FS1 for the 5Hz, 10Hz, and 25Hz steps, and the Y_Value expression would return a value of zero for those points.

SPICE3 Boolean Operators Versus Standard Boolean Operators

Micro-Cap contains a group of boolean operators available for use in analog expressions that provides a means to simulate basic digital functions. There are two sets of these boolean operators: the SPICE3 set and the standard set. The difference between the two sets is in their output and threshold values. The two sets appear in the table below:

Function	SPICE3	Standard
AND	&	&, AND
OR	|	|, OR
NOT	~	~, NOT
NAND		NAND
NOR		NOR
XOR		XOR

As can be seen in the table, the &, |, and ~ operators are present in both sets. Since Micro-Cap supports SPICE2G, SPICE3, PSPICE, and even some HSPICE syntax along with its own extensions, occasional conflicts do arise between the different types of SPICE. Due to these conflicts, there is a Spice Type option available in the Schematic Properties dialog box. Multiple types of SPICE can be simulated within the same circuit without a problem no matter what the Spice Type option is set to. However, this option determines which syntax type will have priority when a conflict does arise. The syntaxes available in the Spice Type page are SPICE2G, PSPICE, SPICE3, HSPICE, Ambiguous, and Not Spice. The default setting is Ambiguous in which Micro-Cap will try to determine the best syntax to prioritize based on the syntax present in the circuit.

An example circuit that demonstrates how the different sets of boolean operators work in conjunction with the Spice Type option is displayed in Figure 8. The V1 and V2 pulse sources create two five volt pulses that represent basic digital waveforms. These two pulse sources are then referenced by the six nonlinear function (NFV) sources. The NFV sources use both available operators to simulate the And, Or, and Not digital functions using the following expressions:

Function	SPICE3/Standard	Standard
AND	V(IN1) & V(IN2)	V(IN1) AND V(IN2)
OR	V(IN1) | V(IN2)	V(IN1) OR V(IN2)
NOT	~V(IN1)	NOT V(IN1)

Each of the NFV sources then has an RC combination connected to its output to provide a slight time delay in the output waveform.

For the first simulation, the Spice Type option in the Properties dialog box will be set to Ambiguous. For this circuit, Micro-Cap will give priority to the standard method of operation for the boolean operators. This means that the output of all six boolean expressions will be either one if true or zero if false. The transient analysis result of this simulation is displayed in Figure 9. For the expressions referenced with the boolean operators, any nonzero value is considered true and only an exact zero value is considered false. Even a very small input value such as 1e-15 would be considered true. This example works well because it is referencing independent voltage sources, but when referencing nodes elsewhere in a circuit, exact zero values may be rare. Due to this, in many cases the boolean operators should be combined with the relational operators to provide a threshold such as in the following expression:

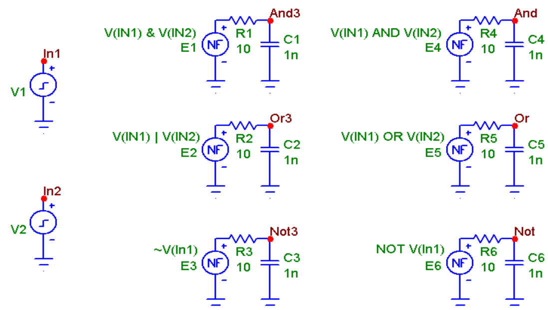


Fig. 8 - Boolean operator example circuit

(V(In1) > 1) And (V(In2) > 1)

where both of the voltages at nodes In1 and In2 must be greater than one volt before the expression will return a true value.

For the second simulation of the example circuit, the Spice Type option in the Properties dialog box is set to SPICE3. This gives priority to the SPICE3 method of operation for the &, !, and ~ operators. This method uses the LONE, LZERO, and LTHRESH variables available in the Global Settings. LONE determines the output level when the boolean expression is true. LZERO sets the output level when the boolean expression is false, and LTHRESH sets the threshold value for the boolean logic. In the Text page of the schematic, the following options statement has been defined:

.options lone=3.5 lzero=.4 lthresh=1.5

This statement sets the true output level to 3.5, the false output level to .4, and the transition point to 1.5. The resultant transient analysis is shown in Figure 10. For the SPICE3 operators, the output levels have now shifted to the values set by the Global Settings variables, and the transition point has slightly shifted over in time when compared to the first simulation.

Note that the expressions using the And, Or, and Not operators continue to work as they did in the first simulation with outputs of zero and one. Since there is no conflict with these operators, the Spice Type option has no effect on how they work.

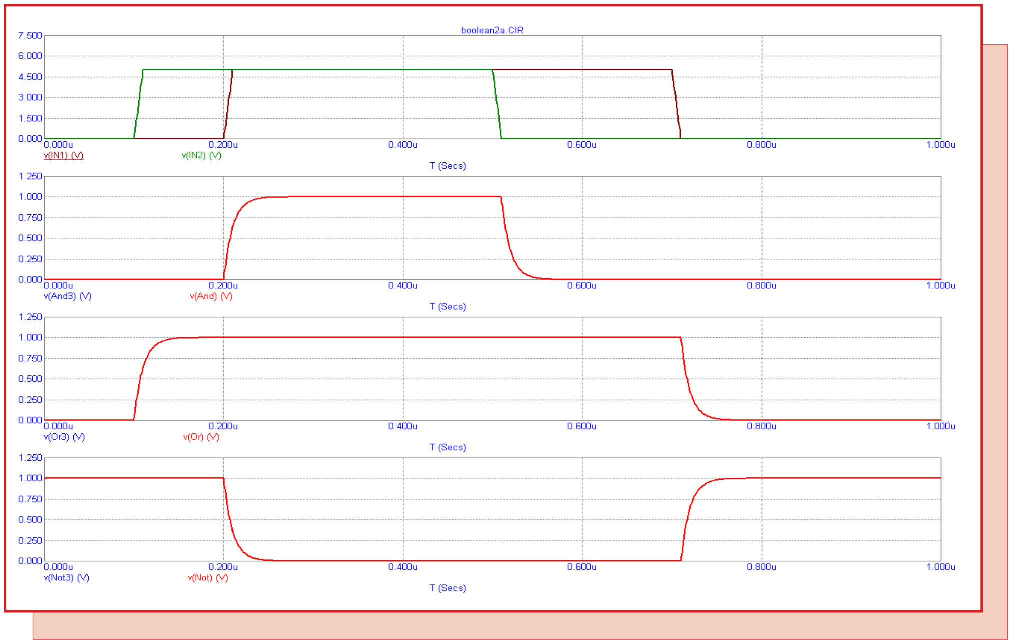


Fig. 9 - Ambiguous Spice Type analysis run

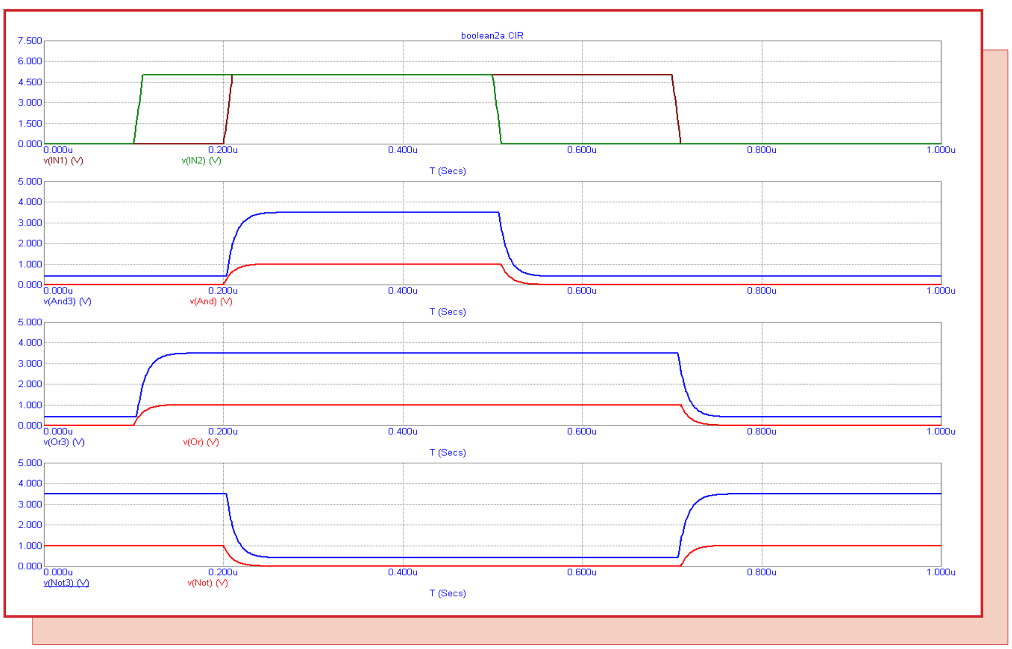


Fig. 10 - SPICE3 Spice Type analysis run

Latest Version numbers

Micro-Cap 8 .... Version 8.1.1
Micro-Cap 7 .... Version 7.2.4
Micro-Cap 6 .... Version 6.3.3
Micro-Cap V .... Version 2.1.2

Spectrum's numbers

Sales .... (408) 738-4387
Technical Support .... (408) 738-4389
FAX .... (408) 738-4702
Email sales .... sales@spectrum-soft.com
Email support .... support@spectrum-soft.com
Web Site .... http://www.spectrum-soft.com
User Group .... micro-cap-subscribe@yahoogroups.com