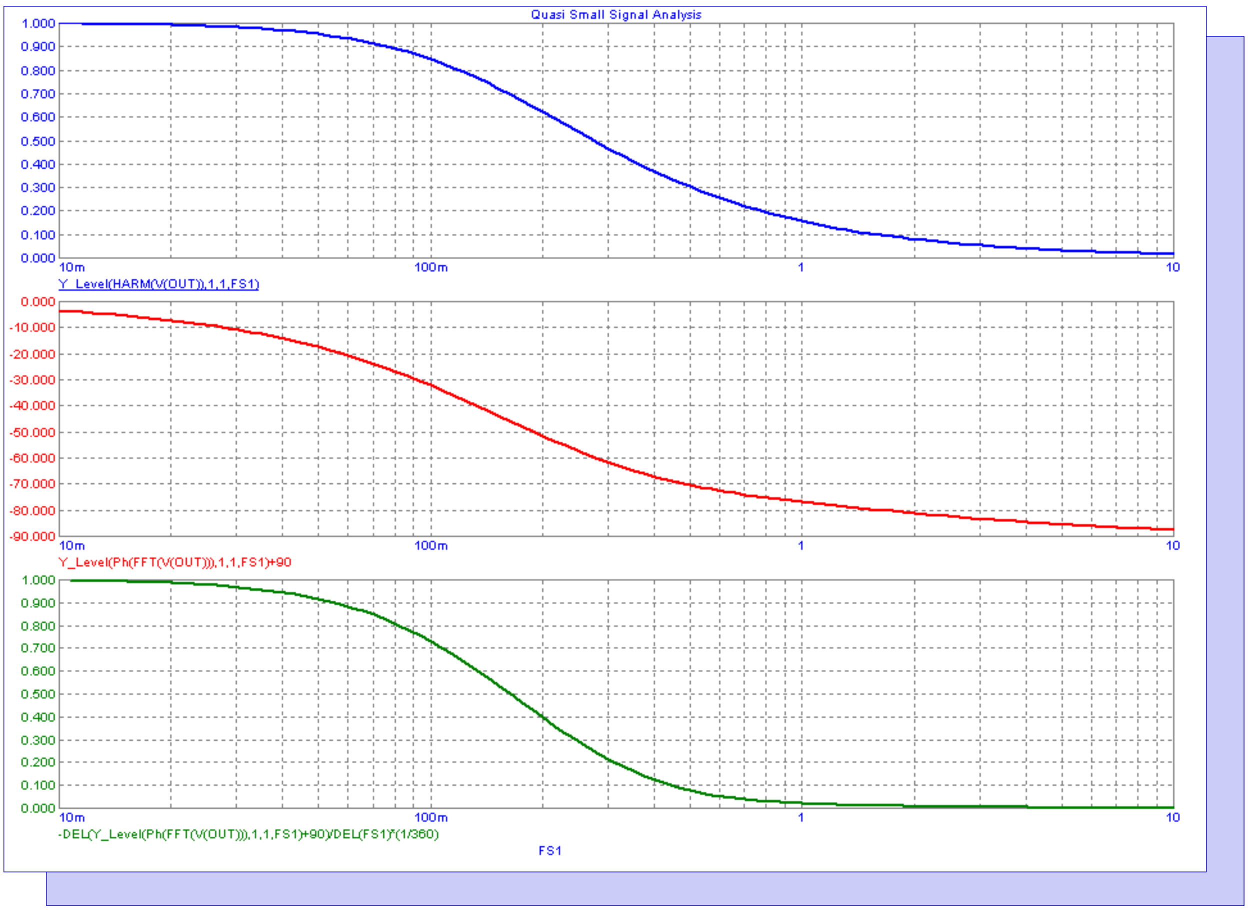
spectrum news

Applications for Micro-Cap™ Users

Winter 2015



Quasi Small Signal Analysis

Featuring:
- Creating Parts with Many Pins
- Curve Fitting
- Quasi Small Signal Analysis Updated

News In Preview

This newsletter's Q and A section describes model encryption problems and how to deal with them. It also describes problems getting the 64 bit version to work properly on systems with more than four processors. The Easily Overlooked Feature section describes the use of the Check for Updates feature that allows easy access to updates to the Micro-Cap system.

The first article describes a new capability introduced in Version 11.0.0.9 that facilitates adding parts with a huge number of pins.

The second article describes the use of Micro-Cap's curve fitting routines.

The third article describes how quasi small signal analysis capability can now compute group delay.

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, 1993. ISBN# 0-07-911525-X

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

MOSFET Modeling

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

Signal Integrity

• *Signal Integrity and Radiated Emission of High-Speed Digital Signals*, Spartaco Caniggia, Francescaromana Maradei, A John Wiley and Sons, Ltd, First Edition, 2008 ISBN# 978-0-470-51166-4

Micro-Cap - Czech

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

Micro-Cap - German

• *Simulation elektronischer Schaltungen mit MICRO-CAP,* Joachim Vester, Verlag Vieweg+Teubner, First Edition, 2010. ISBN# 978-3-8348-0402-0

Micro-Cap - Finnish

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

Design

• *High Performance Audio Power Amplifiers,* Ben Duncan, Newnes, 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, 1997. ISBN# 0-07-913227-8

• *Switch-Mode Power Supplies Spice Simulations and Practical Designs*, Christophe Basso, McGraw-Hill 2008. This book describes many of the SMPS models supplied with Micro-Cap.

Micro-Cap Questions and Answers

Question: I recently downloaded a model from a major manufacturer and I'm having trouble getting it to work. I installed it in the way your manual describes and all went well. But then I tried running a circuit with the part in it and I get the following message.

Unknown command.

$CDENSTART

Answer: The problem is that the file is encrypted. $CDENSTART defines the point at which encryption starts. Only the headers are left in readable text. The crucial part of the file, the SPICE netlist describing the model, is encrypted so that only PSPICE can read it. The purpose of the encryption is said to be protection of the proprietary content of the model, but it also prevents Micro-Cap or any other simulator but PSPICE from reading it. We recommend that you ask the manufacturer for an unencrypted model. Often they will do this if you sign a non-disclosure agreement.

Question: I am trying to use Micro-Cap 11 in a computer with Windows 8 but the 64 bit version is not able to run. I have tried with Windows 7 compatibility settings but it still does not work.

Could you give some hints about this?

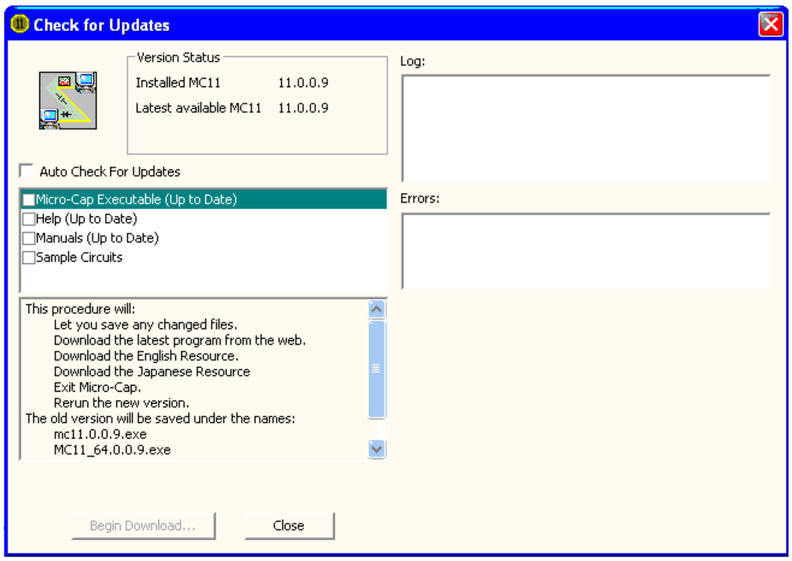
Answer: The problem is in the MC11 code that detects the number of processors in your system. If you have 16 or more processors, then MC11 versions up to and including MC11.0.0.8 will not properly run the 64 bit version, although the 32 bit version runs fine.

MC11.0.0.9 corrects the problem. You can download the updated 64 bit version (mc11_64.exe) and also the updated 32 bit version (mc11.exe) directly from the Help menu or you can download it from the web site Users Section.

Easily Overlooked Features

This section is designed to highlight one or two features per issue that may be overlooked among all the capabilities of Micro-Cap.

Spectrum makes occasional updates to the Micro-Cap elements: the Micro-Cap program, the Help system, the User Guide and Reference Manual, and the Sample Circuits. Updating these elements is easy. Just go to the Help menu and select the Check for Updates item. This dialog box will appear:



Micro-Cap checks the revision date of all items except the circuits and lets you know if they are up to date. If not, you can choose to update them. If you update the executable, then Micro-Cap will make a backup copy of it before replacing it with the newer version. Backup copies of the other elements are not done.

You can choose to update any of the four elements by clicking on the checkbox.

Sometimes a fire wall on the user side will disallow updating executables or even large files. In this case, you must contact Spectrum to obtain access to the latest elements via our web site.

Creating Parts with Many Pins

Suppose you have to create a part with a great many pins. How do you do that? The usual method is to create a suitable shape with the Shape editor using the Block shape. It looks like this:

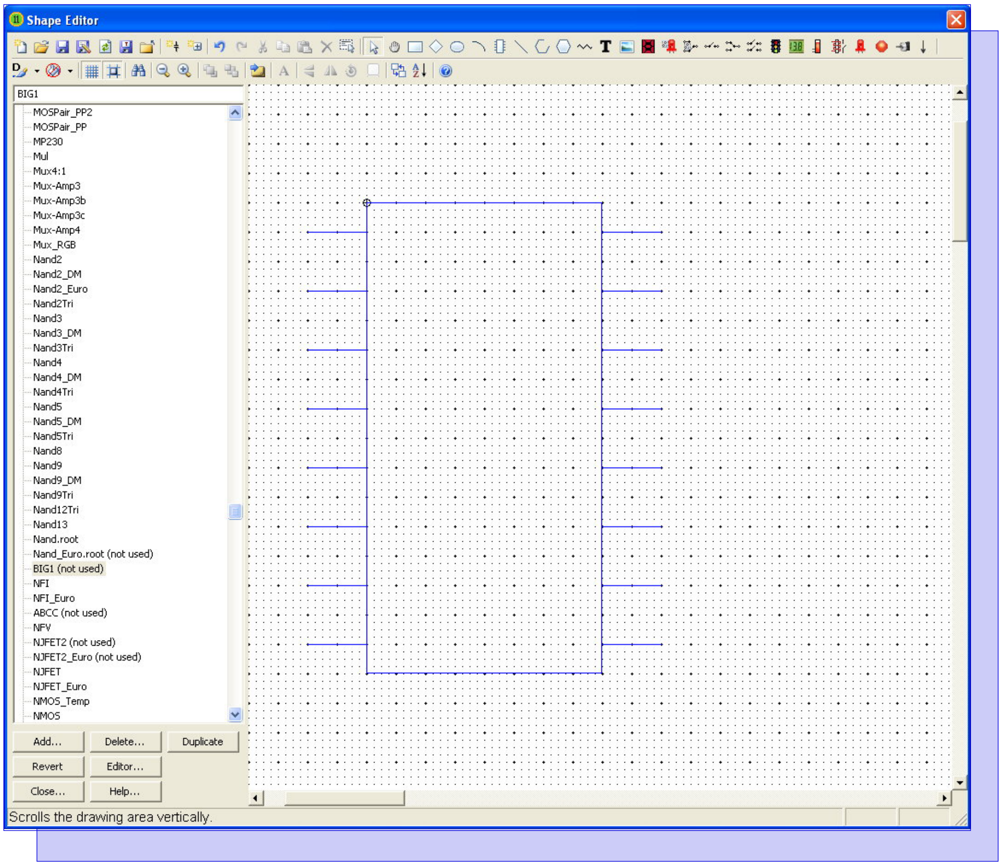


Fig. 1 - A 16 pin block shape

When the part is created in the Component editor, you use the created block shape and manually assign the pins and their names one by one. This works well up to a point, but what if you have a part with hundreds of pins? The process can be error prone with so many pins.

With the release of MC11.0.0.9, you now have a new option. The new capability is implemented with two features:

The Shape editor creates a large block shape automatically by specifying only the number of pins, The pins are placed as equally as possible on either side of the block shape.

The Component editor's Add Part wizard now automatically assigns the pin names from the .subckt statement. The name assignment starts at the shape's origin point and assigns names sequentially in a counter-clockwise order. The first pin name in the .subckt statement is assigned to the pin nearest the shape origin. Then the next pin name in the subckt list is assigned to the second pin and so on until the last pin name is assigned. Even if you want the pins to be assigned in a different way, the process at least places the pins systematically with the correct names, so you can drag the pins to different locations.

Consider this subcircuit:

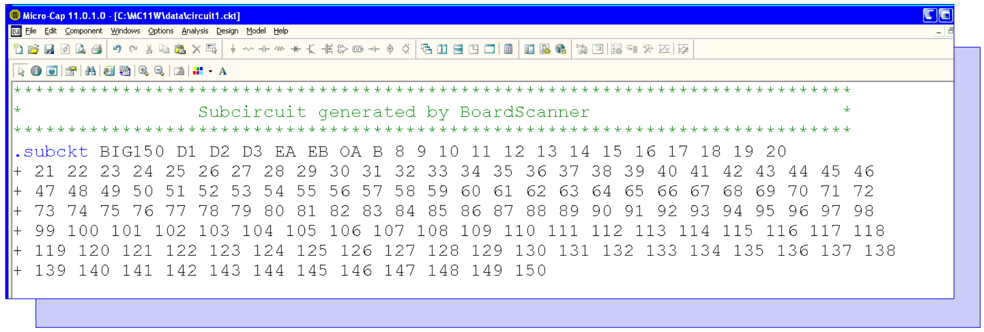


Fig. 2 - The Big150 subcircuit

This subcircuit, called BIG150, uses 150 pins as can be seen from its .subckt statement:

.subckt BIG150 D1 D2 D3 EA EB OA B 8 9 10 11 12 13 14 15 16 17 18 19 20
+ 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40 41 42 43 44 45 46
+...
+ 139 140 141 142 143 144 145 146 147 148 149 150

In this case some of the pins have names with letters, but most have numeric names. To illustrate, we'll create a suitable shape and then add the part in the Component editor. If you wish to follow along, you must copy the Big150.LIB from this newsletter's zip file to the LIBRARY folder.

Select Windows Menu / Shape Editor... Click on the Add button. A dialog box will come up and ask for the shape name. Enter Big150. Select the Block icon. Click on the Origin symbol in the shape area. A dialog box will come up and ask for the number of pins. Enter 150. The screen should now look like this:

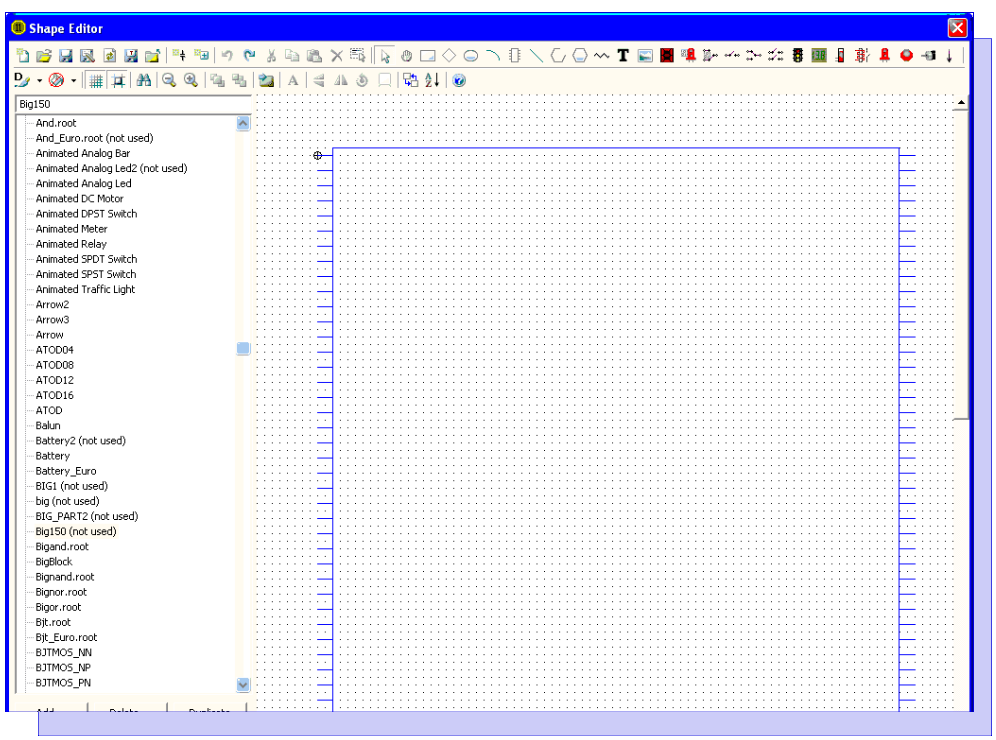


Fig. 3 - The Big150 shape

Exit the Shape editor and select Windows menu / Component Editor. Select a place in the part hierarchy by clicking on one of the groups, like Analog Library. Click on the Add Part wizard.

Select Subckt from the first panel. Click on the Next button or press Enter.
Enter Big150.lib. Click on the Next button or press Enter.
Click on the Big150 part name. Click on the Next button or press Enter.

Click in the Shape list and press 'B' until you see the 'Big150' shape created earlier.
Press Enter five more times to complete the wizard. Your screen should now look like this:

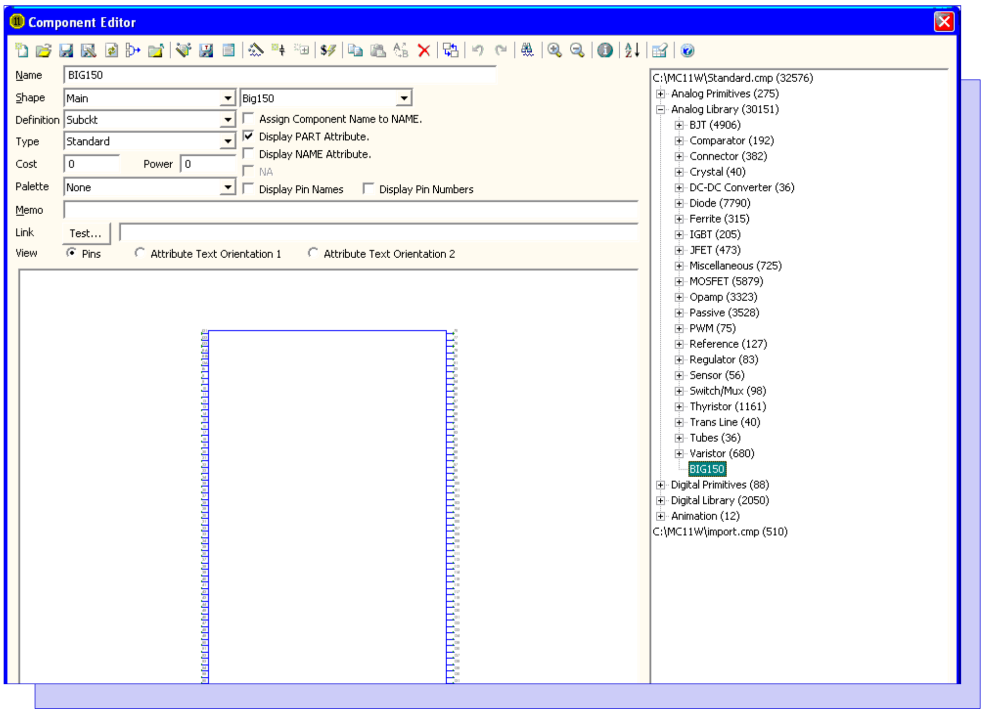


Fig. 4 - The Big150 part

At this scale it is hard to see the pin names. Click on the Magnifying icon several times to the enlarge the shape and you see this:

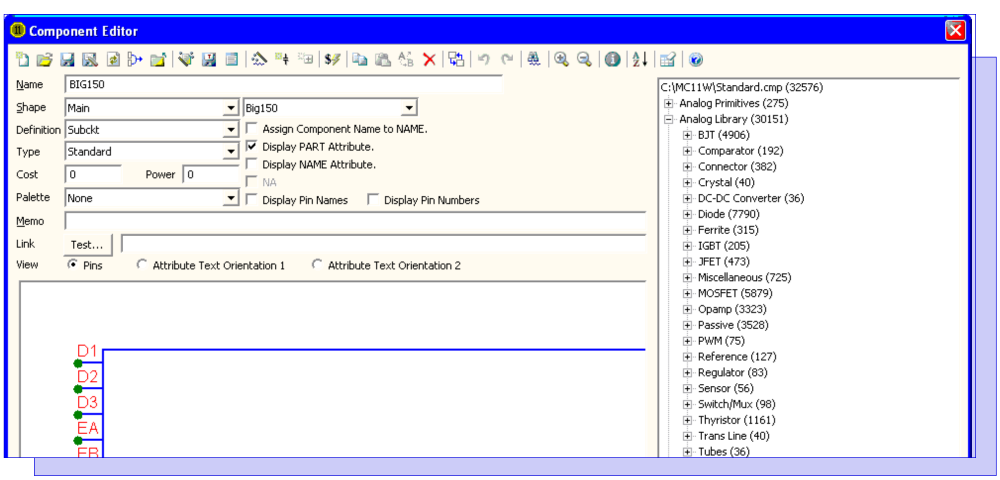


Fig. 5 - The Big150 part enlarged

You can see the first few pin names imported from the subckt statement in the Big150.lib file. Exit from the Component editor by clicking on the X in the upper right corner of the dialog box.

Now we can create a circuit using the Big150 part. Press CTRL + N to create a new circuit file. Click on the Component menu / Find Component and type in Big150 in the name field and press Enter. This selects the Big150 part. Click in the upper left of the circuit. Click the Show check box next to the Big150 part name. Click on the Pin Names check box. Click OK in the Attribute dialog box or just press Enter. This will place the Big150 part. Press the Spacebar key to toggle from Add Part mode to Select mode, and the drag the 'X1' name and the 'Big150' name roughly center then in the block shape.

Your circuit should now look like this:

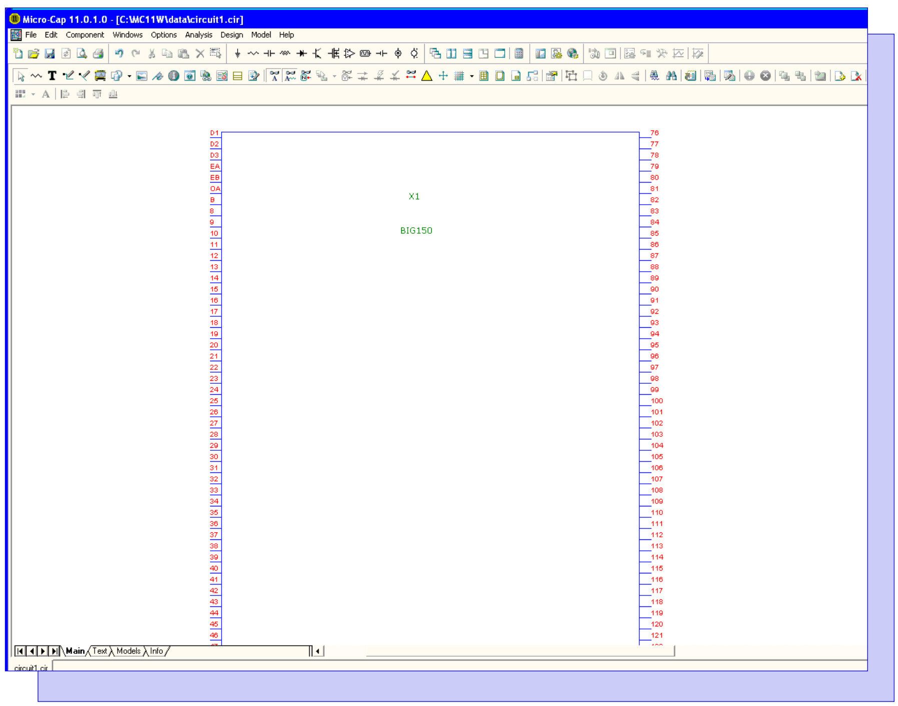


Fig. 6 - The Big150 part placed in a circuit.

When you request an analysis, the program reads the Big150.lib to fill in the content of the Big150 part. Of course, there is still much to do in the circuit; setting up sources to drive the inputs and circuitry to load the outputs, but a great deal of the work is done.

Micro-Cap 12 will have a more sophisticated version of this, allowing for 1- 4 sides with varying pin spacing, pin length, and pin alignment, but the basic capability will be similar.

Curve Fitting

Did you know that Micro-Cap 11 has a built-in curve fitting function? Any plotted curve can be fitted to an N-degree polynomial. Consider this circuit:

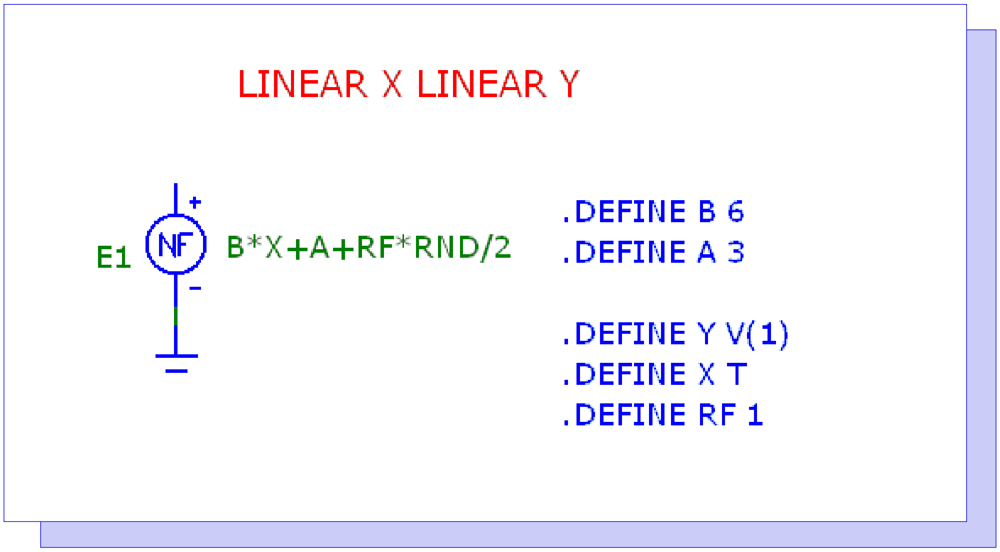


Fig. 7 - A sample circuit for curve fitting

This circuit has a single voltage source whose output is:

B*X+A+RF*RND/2

where A, B, and RF are 3 , 6, and 1 respectively and X is set to the T (Time) variable. RND is a random number between 0 and 1. The E1 source then generates a line whose equation is

6*T+3 + RND/2. If you run transient analysis on it you get this plot.

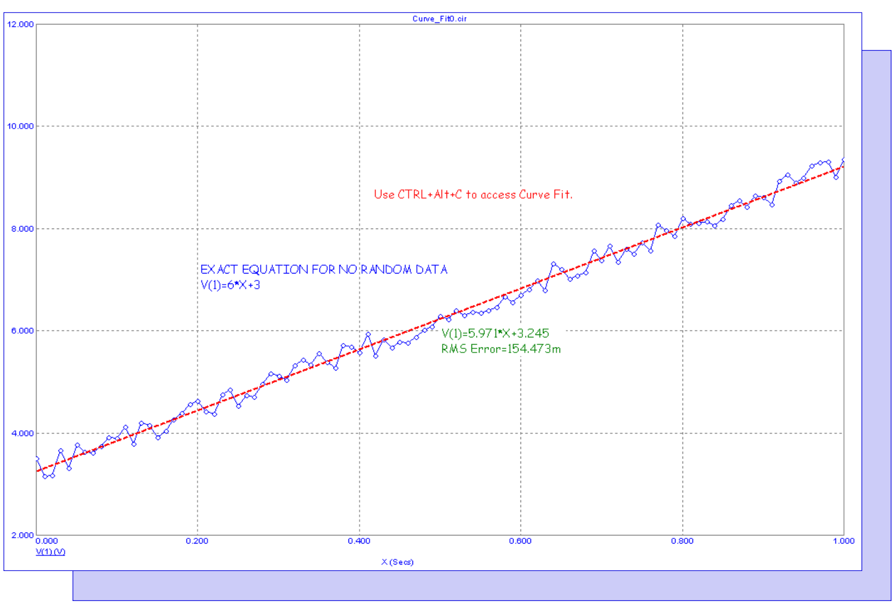


Fig. 8 - The transient analysis run

Here the line is drawn with its random element. The curve has been fitted to a linear equation per the instruction from the Curve Fit dialog box. It looks like this:

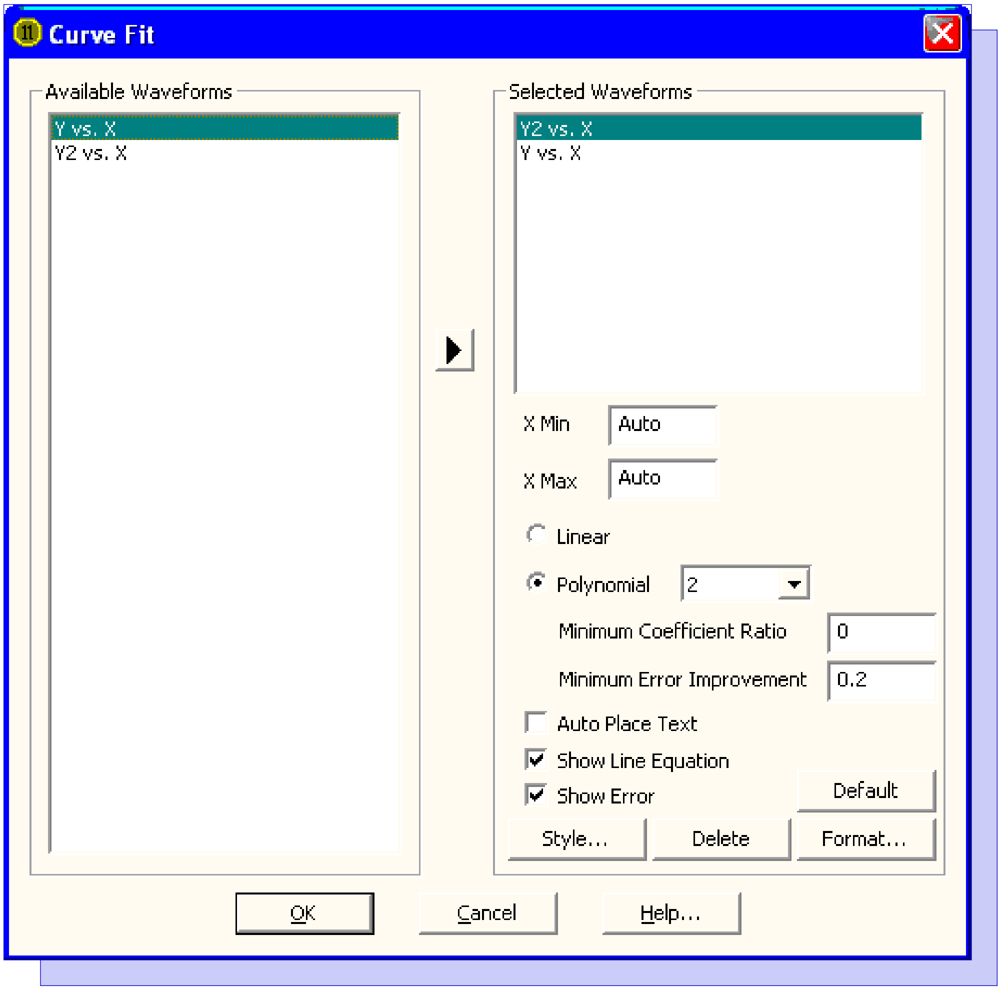


Fig. 9 - The Curve Fit dialog box

This dialog box lets you select the curve(s) to be fitted, the range over which the fit will occur, and the polynomial degree for the fit. You can also choose options for placing the equation and error text as well as options for numeric format, and text style.

The dialog box is selected from the Scope menu / Curve Fit or by using CTRL+ALT+C, or by right-clicking on any expression text in the analysis plot.

This example is a simple one. Next we'll try a more complex curve to fit.

Consider this circuit.

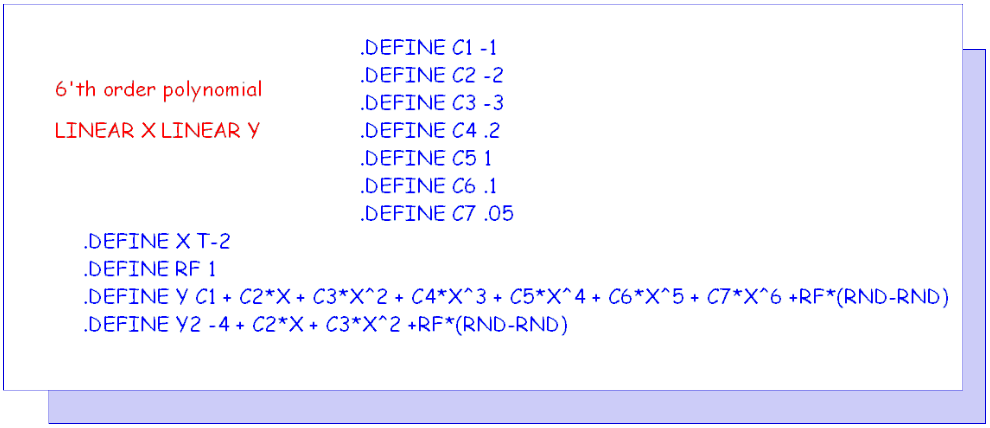


Fig. 10 - A sample circuit for curve fitting

There are no sources, just .define statements specifying a quadratic (Y2) and a 6'th order polynomial (Y). Run transient analysis and you get this plot:

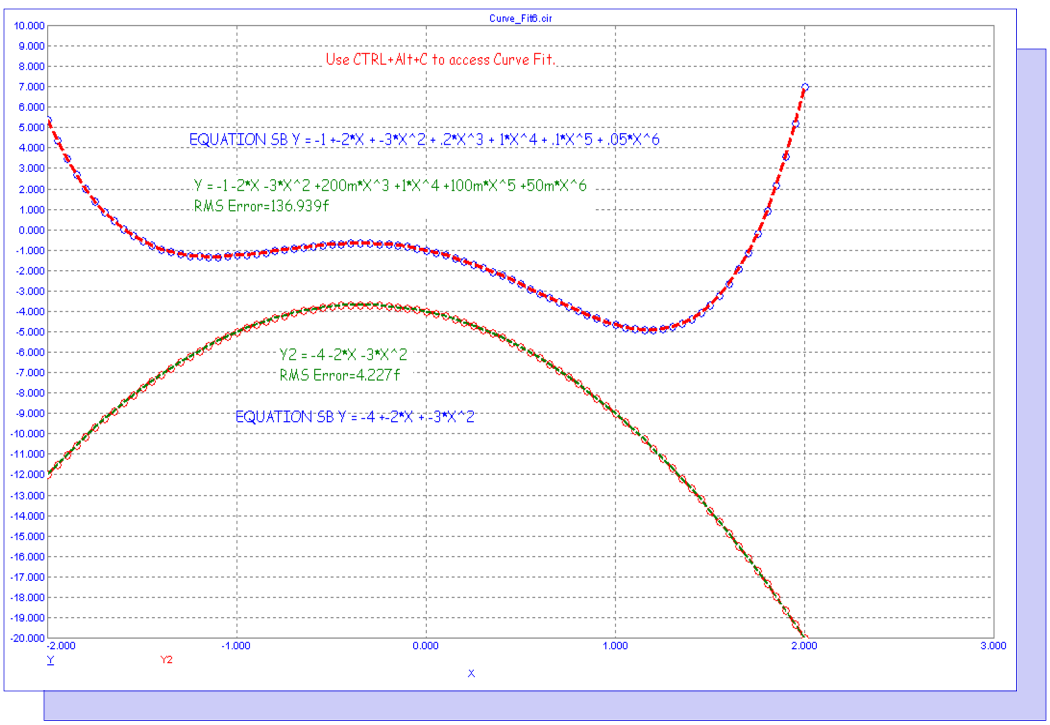


Fig. 11 - The transient analysis run

The routine has fitted the quadratic curve as Y = -4 +-2*X + -3*X^2 and the 6'th order curve as Y = -1 -2*X -3*X^2 +200m*X^3 +1*X^4 +100m*X^5 +50m*X^6. Both fits have RMS errors of less than 1e-12.

Quasi Small Signal Analysis Updated

This article was updated from a previous article in the Winter 2009 newsletter to reflect the improved capabilities of Micro-Cap 11 in this area. Specifically MC11.0.0.9 and later versions can now use the DEL operator on Performance functions to compute group delay.

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.

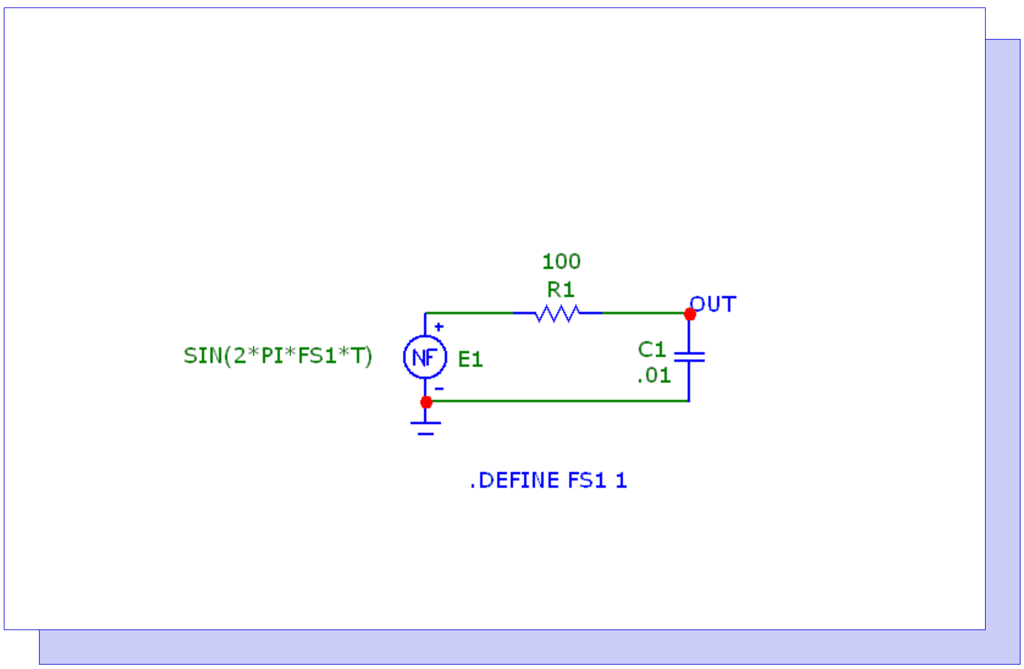


Fig. 12 - Quasi small signal example circuit

The method to 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 circuit used to demonstrate this technique is shown above. 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. In this case, it specifies a 1 volt small signal source for an AC simulation run. The Freq attribute is ignored during a transient simulation. The Value attribute defines a one volt peak amplitude sine wave signal during a transient run whose frequency is set by the symbolic variable FS1. FS1's initial value is set through a define statement present in the schematic. The AC analysis is shown below. The magnitude of the output voltage is the top plot, the phase of the output voltage is the middle plot and the group delay is the bottom plot. This plot will be the benchmark for the quasi small signal results.

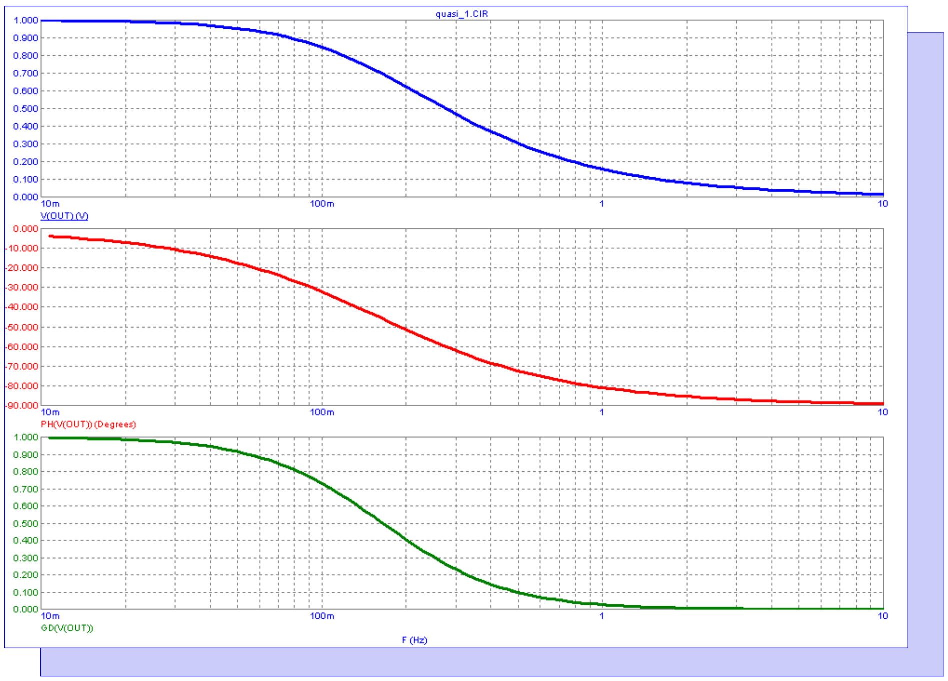


Fig. 13- Standard AC analysis simulation

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 Log method has been selected for FS1, and the parameters are as follows:

From .01
To 10
Step Value (10/.01)^(1/99)

These settings will produce exactly 100 points over the frequency range of .01 to 10 Hz.

For this example, the Time Range and Maximum Time Step fields in the Transient Analysis Limits dialog box have been set to the following:

Time Range = 2/fs1
Maximum Time Step = 1/(50*fs1)

This time range expression will simulate 2 cycles of the input sine source at each stepped frequency. In addition, the maximum time step dynamically adjusts at each step to produce a minimum of 50 data points per cycle.

In the FFT page of the Analysis Properties dialog box, the Upper and Lower Time Limit fields have been set to the following:

Upper Time Limit = 2/fs1
Lower Time Limit = 1/fs1
This produces a frequency step for the FFT of 1/(2/FS1 - 1/FS1) = FS1

These two fields set the portion of the simulation that the FFT operators will perform their calculations on. With these settings, the FFT operators will only work with the last cycle out of the two cycles simulated during each frequency step. Using the last cycle will exclude much of the initial transient that occurs from the FFT calculations. If a circuit takes a different time to reach its steady state operation, then both the time range and the FFT limits can be adjusted accordingly, or you can use PSS to compute the steady state solution. The resulting transient analysis is displayed below.

The top plot is the actual voltage of node Out at each frequency step. Only the last cycle in each step is displayed due to the settings in the FFT page of the Analysis Properties dialog box. 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.

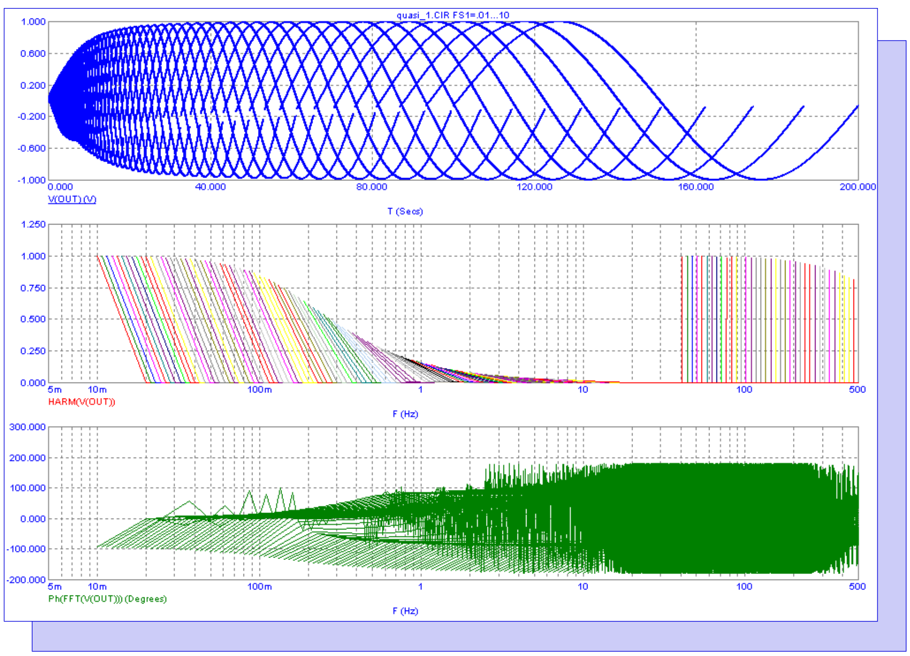


Fig. 14 - Transient analysis simulation

Though all of the data is shown plotted, it's not in an easily readable format to show the 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 plot created is displayed below.

The plot has been defined to display the following three waveforms:

Y_Level(HARM(V(OUT)),1,1,FS1)
Y_Level(PH(FFT(V(OUT))),1,1,FS1)+90
-DEL(Y_Level(Ph(FFT(V(OUT))),1,1,FS1)+90)/DEL(FS1)*(1/360)

The Y_Level operators will return the Y value of each stepped waveform at the X value (FS1) 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

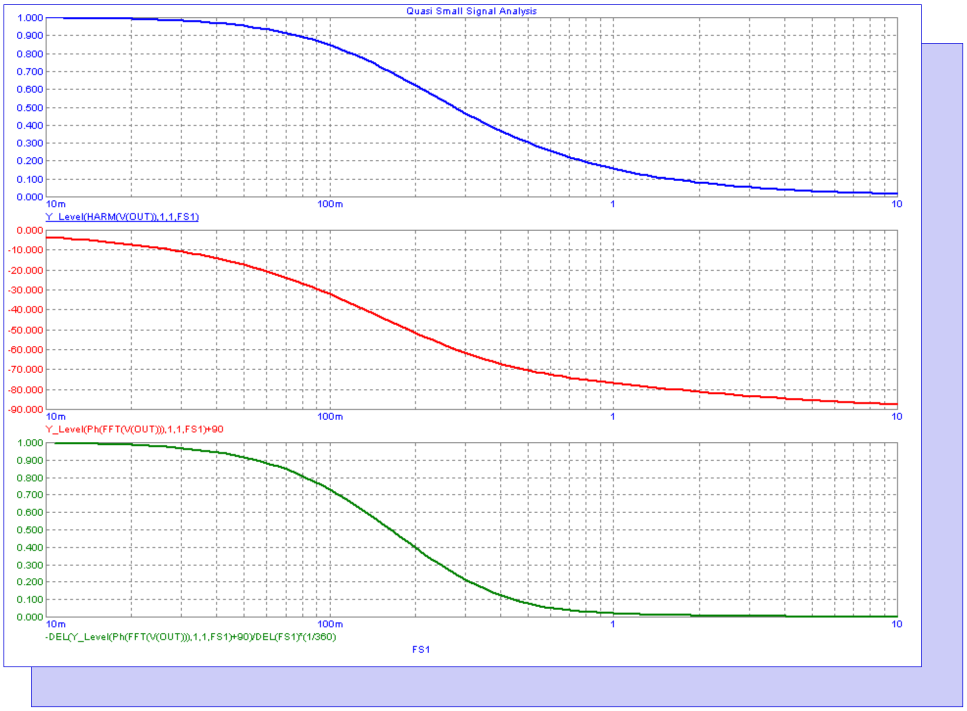


Fig. 15 - Quasi small signal plot

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 value of this expression:

-DEL(Y_Level(Ph(FFT(V(OUT))),1,1,FS1)+90)/DEL(FS1)*(1/360)

Since group delay is defined as

GD = DELTA(PHASE(DEGREES))/DELTA F /360,

this formula yields group delay directly. The new thing in Micro-Cap 11.0.0.9 is the ability to calculate and plot performance functions using the DEL operator. The DEL operator calculates the change in its argument from one value of FS1 to the next, so the ratio calculates the numeric derivative to arrive at the GD value.

You can see by comparing the AC and transient plots that the results for gain, phase, and group delay are the same.

Product Sheet

Latest Version numbers

Micro-Cap 10..Version 11.0.0.9
Micro-Cap 10..Version 10.1.0.4
Micro-Cap 9..Version 9.0.9.1
Micro-Cap 8..Version 8.1.4.0
Micro-Cap 7..Version 7.2.4

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