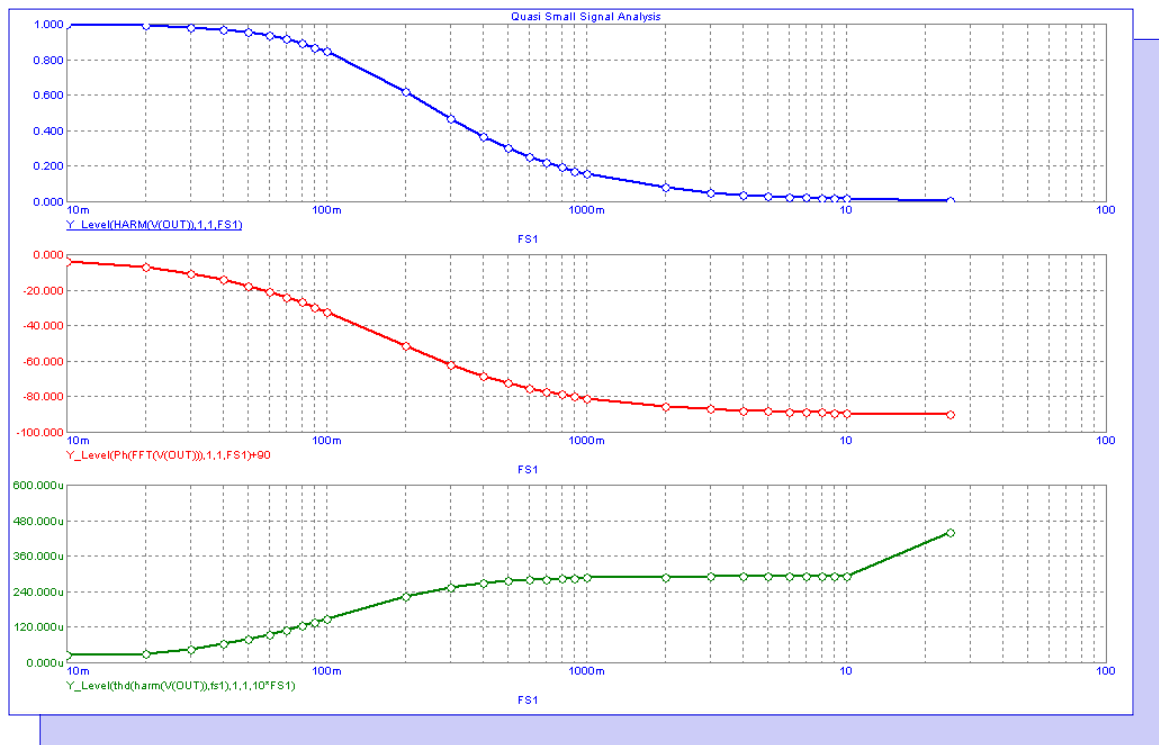


Winter 2009

News



Quasi Small Signal Analysis

Featuring:

- Quasi Small Signal Analysis
- Optimizing NTC Thermistor Coefficients
- Limiting Phase Plots to +/- 180 Degrees

News In Preview

This newsletter's Q and A section describes a conflict with a process called ASGHOST.EXE. The Easily Overlooked Feature section describes how to use the DEF and NOF switches in a batch file to set define variables and rename numeric output files.

The first 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 second article describes how to use the optimizer to calculate the coefficients of the Steinhart-Hart equation that is used to model NTC thermistors.

The third article describes how to plot phase in AC analysis and limit the plot so it stays within the range of 180 degrees to -180 degrees rather than the continuous plot that the PH() operator produces.

Contents

News In Preview	2
Book Recommendations	3
Micro-Cap Questions and Answers	4
Easily Overlooked Features	5
Quasi Small Signal Analysis (Updated for MC9)	6
Optimizing NTC Thermistor Coefficients	10
Limiting Phase Plots to +/- 180 Degrees	14
Product Sheet.....	16

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

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 Elektronické Obvody*, Dalibor Bielek, 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# 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 1: I am having problems when I try to install Micro-Cap. When I click on the Setup.exe file to launch the installation, nothing happens. I have tried rebooting the system, but that has not helped. I have downloaded the program again and have attempted running the Setup.exe file with no programs loaded, but have had no success on getting it installed. What is preventing the Setup executable file from running?

Question 2: I am running Micro-Cap on my system. I can create a circuit fine and run a simulation. However, when I close the plot window, the program crashes, and I get the Microsoft error report window. This crash occurs when I close any transient, AC, or DC simulation window. It even happens when I close the plot created from the Plot command within the Attribute dialog box. How do I prevent this from happening?

Question 3: I am having a problem whenever I try to edit a voltage source that is in my schematic. I double click on the voltage source and change the pulse waveform parameters of the component. As soon as I hit the OK button, Micro-Cap crashes. How can I get around this?

Answer: All of the above questions seem to have a similar cause. Invoke the Windows Task Manager, and click on the Processes tab. Look for the following process:

ASGHOST.EXE

Select this process and then click the End Process button to terminate it. Try running Micro-Cap to see if the problem is still present.

We have been able to trace the conflict with the above process to an error that occurs within a Windows routine that Micro-Cap calls, so the conflict is fundamental to the operating system.

We have yet to determine the validity of the ASGHOST.EXE file. There is a legitimate program that does use this file, but many websites also state that this can be malware camouflaged as ASGHOST.EXE.

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.

Setting Define Parameters and Numeric Output File Names within Batch Files

The batch file capability in Micro-Cap is a useful method for running multiple circuit analyses with a single command. The batch file contains a list of circuits that Micro-Cap will run one after the other. In most cases, this will be a list of different circuits that will be simulated. However, it can be quite useful to vary one or more parameters within a single circuit file, and run a batch file that simulates each of the variations. A parameter can be varied in a circuit file through the use of the /DEF switch in the batch file whose syntax is:

```
/DEF "x value"
```

The above switch sets the variable x in the circuit to the value specified. The x variable must be set in the corresponding circuit with a define statement. Since the default numeric output file that is created when a circuit is run uses the circuit name itself, the numeric output file will be overwritten each time the same circuit is run within the batch file. In order to use the numeric output file from each variation of the circuit, the /NOF switch would need to be used to rename the numeric output file and prevent it from being overwritten. The syntax for the /NOF switch is:

```
/NOF "fn"
```

where fn is the numeric output file name without an extension. Micro-Cap will add the appropriate extension (.TNO, .ANO, or .DNO) depending on the simulation type that was specified.

A batch file can be created with any text editor. The file would need to be in standard ASCII text format. As an example for how to use the DEF and NOF switches, the following three lines were entered in a text file called TEST.BAT.

```
Prlc8 /T /DEF "Rtest 50" /NOF "Prlc8a"  
Prlc8 /T /DEF "Rtest 100" /NOF "Prlc8b"  
Prlc8 /T /DEF "Rtest 150" /NOF "Prlc8c"
```

The first line is read in. The circuit PRLC8.CIR is loaded. The Rtest define variable is set to 50, and a transient analysis (due to the /T switch) is run. The numeric output from the simulation is stored in a file called PRLC8A.TNO. The analysis is exited and the circuit file is closed. The second and third lines are processed in the same manner. For the second line, the Rtest variable is set to 100, and the numeric output is stored in the file PRLC8B.CIR. For the third line, the Rtest variable is set to 150, and the numeric output is stored in the file PRLC8C.CIR.

When the batch file is finished, there will be three numeric output files available that each contain the simulation data from one variation of the circuit file.

A batch file can be launched within Micro-Cap through the command line. At a command line, enter the following command:

```
c:\mc9\mc9 @Test.bat
```

to run the Test.bat file in MC9. In this case, the Test.bat is located in the same directory as the MC9.EXE file.

Quasi Small Signal Analysis (Updated for MC9)

This article was updated from a previous article in the Fall 2005 newsletter to reflect the improved capabilities of Micro-Cap 9 in this area.

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.

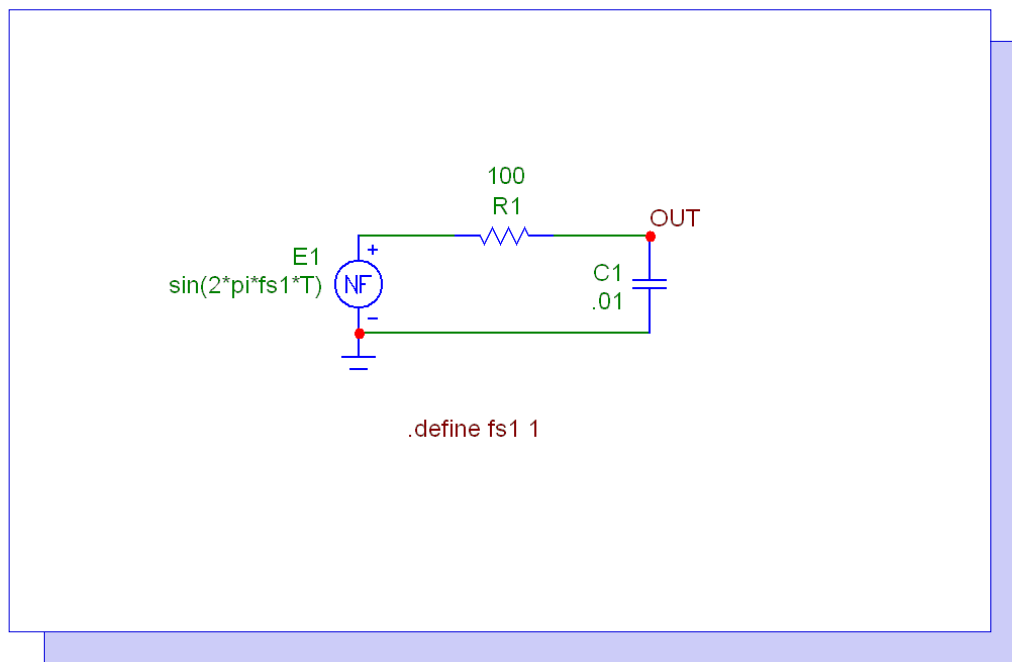


Fig. 1 - Quasi small signal example circuit

The example circuit used to demonstrate this technique is displayed 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, 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 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 below. 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.

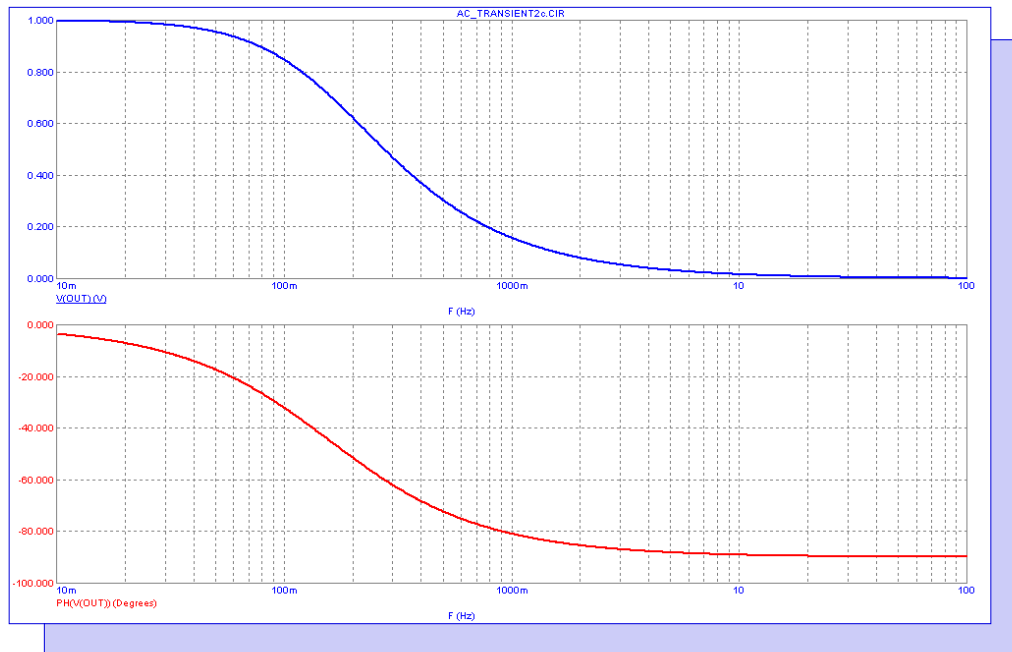


Fig. 2 - Standard AC analysis simulation

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,.03,.04,.05,.06,.07,.08,.09,.1,.2,.3,.4,.5,.6,.7,.8,.9,1,2,3,4,5,6,7,8,9,10,25

The improvements made in Micro-Cap 9 to aid quasi small signal analysis are in allowing stepped symbolic parameters to be used within the Time Range, Maximum Time Step, and the FFT Upper and Lower Time Limit fields. This allows the simulation time, timestep, and the FFT window limits to vary along with the stepped frequency which greatly speeds up the simulation. The accuracy and ease of use have also improved with these additions. The circuit example in this article takes 37s to simulate using the method specified in the older article, but the simulation time is reduced to just 6s using the updated method.

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 = $200/fs1$
 Maximum Time Step = $1/(100*fs1)$

This time range expression will simulate 200 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 100 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 = $200/fs1$

Lower Time Limit = $199/fs1$

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 hundred cycles simulated during each frequency step. Using the last cycle will exclude any initial transient that occurs from the FFT calculations. Obviously, 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. The resulting transient analysis is displayed below.

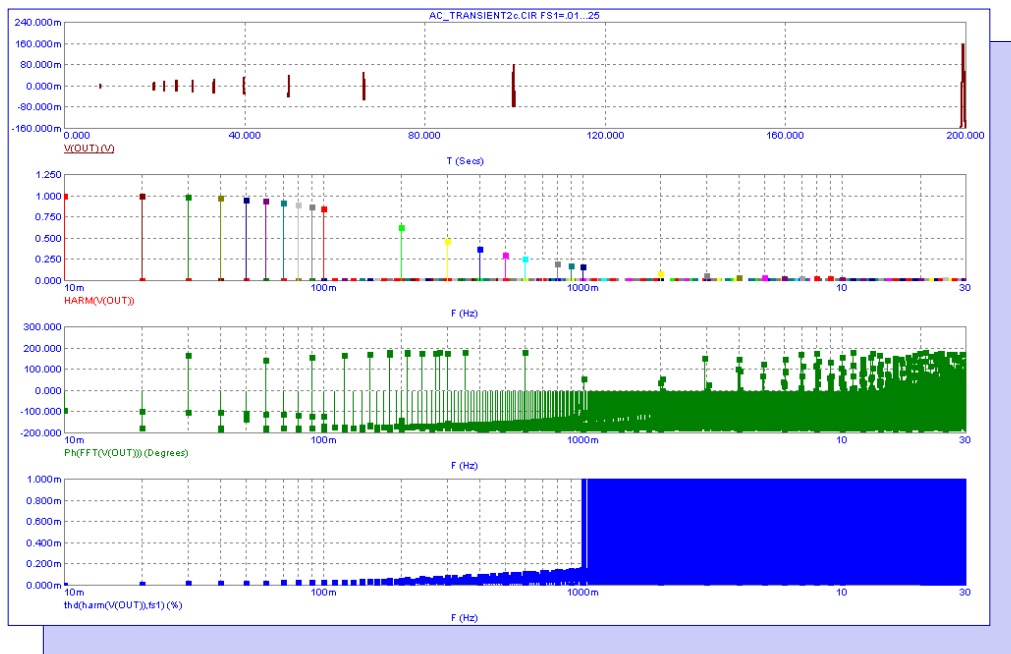


Fig. 3 - Transient analysis simulation

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, and the bottom plot shows the total harmonic distortion of the voltage at 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 below.

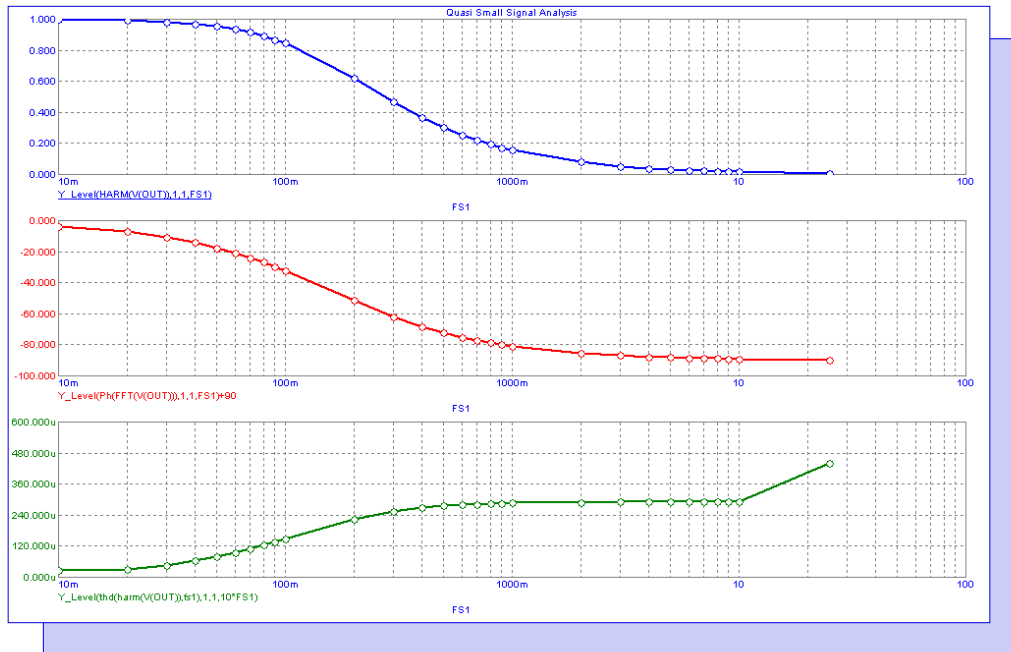


Fig. 4 - Quasi Small Signal performance plot

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
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.

Optimizing NTC Thermistor Coefficients

A thermistor is a thermally sensitive resistor that adjusts its resistance with changes in temperature in a predictable manner. Thermistors are used for such applications as temperature measurement, temperature control, power measurement, amplitude stabilization, and timing circuits. The NTC type of thermistor has a negative temperature coefficient whose resistance decreases with an increase in temperature. The behavior of an NTC thermistor can be modeled accurately through the Steinhart-Hart equation which is as follows:

$$R = e^{(\beta - \alpha/2)^{1/3} - (\beta + \alpha/2)^{1/3}}$$

where

$$\alpha = (A - 1/T)/C$$
$$\beta = \text{Sqrt}((B/3C)^3 + \alpha^2/4)$$

T is the temperature in Kelvin. A, B, and C are the three coefficients that need to be calculated in order to produce an accurate resistance versus temperature model. The optimizer available in Micro-Cap provides a great way to determine the appropriate value of these constants. The schematic below can be used with the optimizer to determine the Steinhart-Hart coefficient values.

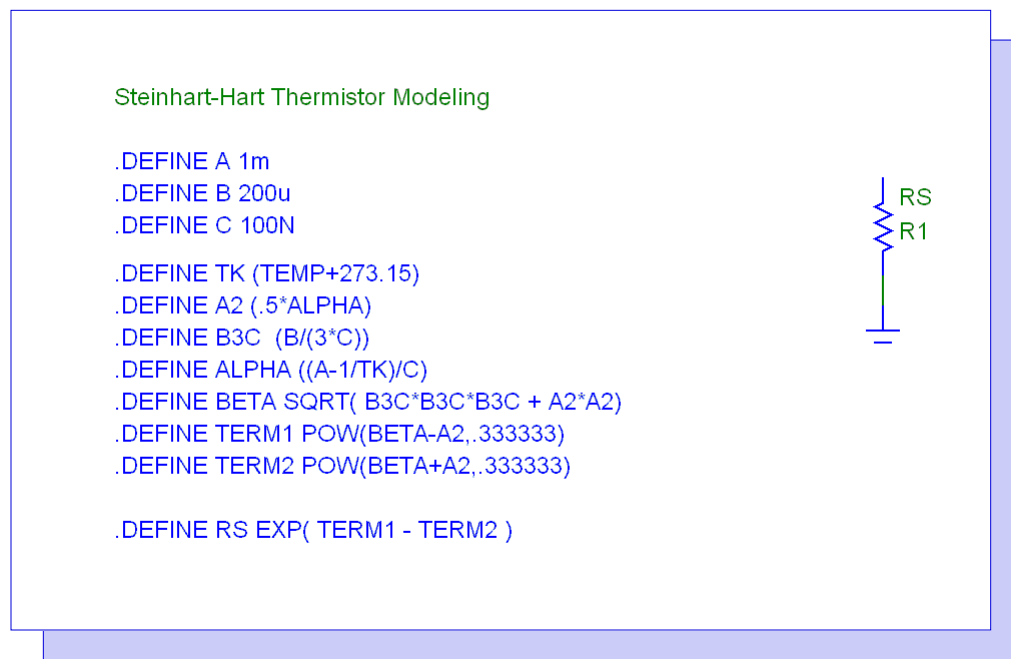


Fig. 5 - NTC Thermistor optimization circuit

The only components in the schematic are a resistor and a ground. The ground is present because it is mandatory for a ground node to be present in all schematics. The resistor has its RESISTANCE attribute defined with the symbolic variable RS. The RS variable is defined through the use of eleven define statements. These define statements reproduce the Steinhart-Hart equation shown above. The A, B, and C symbolic variables must each have their own define statement since these are the objects that will be optimized. The other define statements were used to help enhance the readability of the expression rather than having one long define statement. The define statements used are as follows:

```

.DEFINE A 1m
.DEFINE B 200u
.DEFINE C 100N
.DEFINE TK (TEMP+273.15)
.DEFINE A2 (.5*ALPHA)
.DEFINE B3C (B/(3*C))
.DEFINE ALPHA ((A-1/TK)/C)
.DEFINE BETA SQRT( B3C*B3C*B3C + A2*A2)
.DEFINE TERM1 POW(BETA-A2,.333333)
.DEFINE TERM2 POW(BETA+A2,.333333)
.DEFINE RS EXP( TERM1 - TERM2 )

```

Since the curve that is to be optimized is a resistance versus temperature curve, the best analysis to optimize this in is the DC analysis. The DC Analysis Limits dialog box is setup as below.

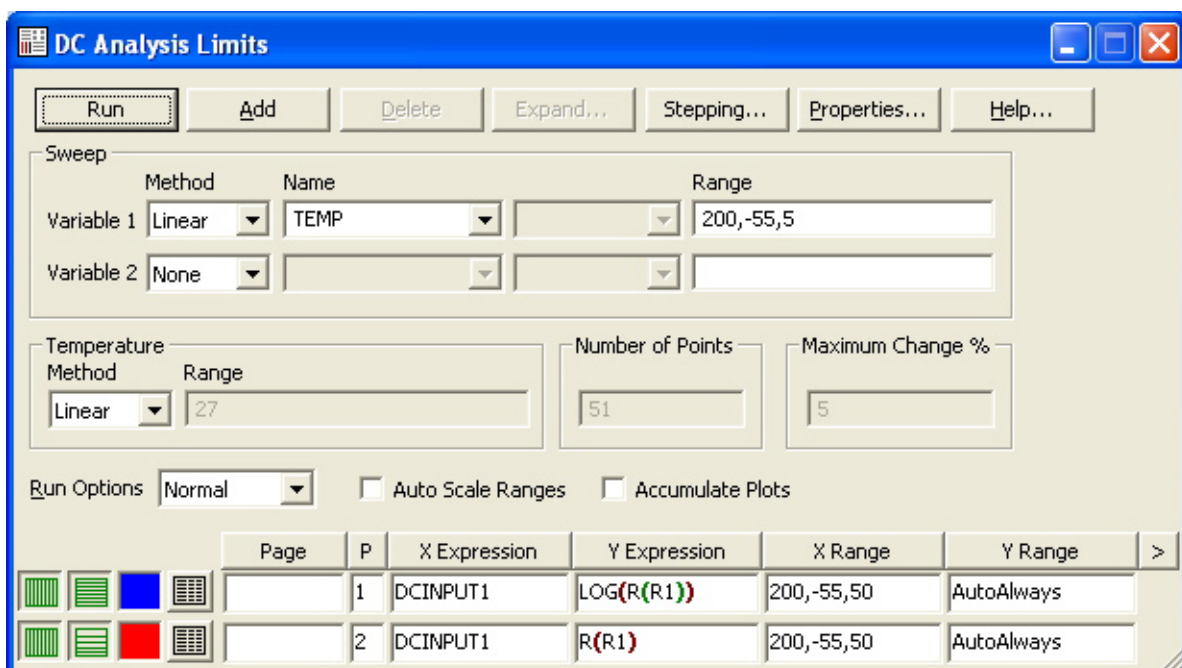


Fig. 6 - DC Analysis Limits dialog box

The Temp operator is entered in the Variable 1 field, and the temperature has been setup to be swept linearly from -55C to 200C in 5C increments. Two Y expressions for the resistance are plotted versus the swept temperature. The value of the resistance of the R1 resistor is plotted as R(R1), and the logarithmic value of the resistance is plotted as Log(R(R1)).

The optimizer can only operate on waveforms specified in the analysis limits dialog box. While NTC thermistor data sheets will describe the device in terms of the resistance versus temperature, the optimizer will produce more accurate results with its logarithmic equivalent. When matching a curve, the optimizer works better when the values it is trying to match are of similar magnitude since each data point has equal weighting. Specifying both of these expressions in the analysis limits lets the user compare the resistance versus the data sheet while also letting the optimizer work with the log expression.

The optimizer in DC analysis is accessible through the Optimize command under the DC menu. It specifies the parameters to be optimized along with the criteria to optimize to. For the NTC thermistor example, the Optimize dialog box has been set up with the following information.

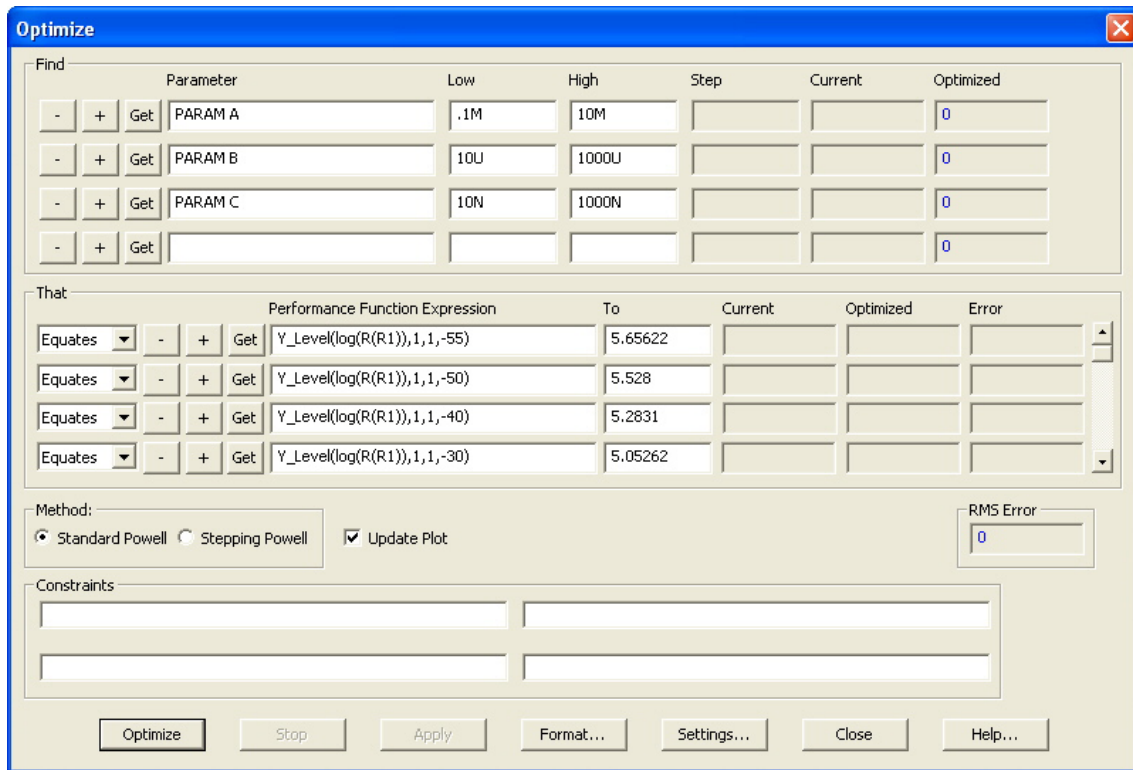


Fig. 7 - Optimize dialog box

The Find section specifies the A, B, and C parameters as the objects to be optimized along with a low and high value that defines the range that the value of these parameters may be optimized within. The That section defines the curve that the optimizer is trying to match. In order to match a curve, the Equates option should be used in conjunction with the Y_Level performance function. Each data point in the curve would be defined using an Equates - Y_Level entry. For this example, the thermistor data that will be used is for the B57885S0103F002 thermistor from Epcos which has a nominal resistance of 10kohms at 25C. This particular data point in the resistance versus temperature curve would be defined with the following performance function expression:

$$Y_Level(\log(R(R1)),1,1,25)$$

where the corresponding To field would be set to 4 since that is the log equivalent of 10k. This entry tells the optimizer to try to match the $\log(R(R1))$ expression to a value of 4 at the specified X axis value (temperature) of 25. Other resistance/temperature data pairs for the device from a range of -55C to 155C have also been entered in the That section to produce a good sampling of the thermistor curve. Clicking the Optimize button initiates the optimization process. The final results for this particular optimization are shown in Figure 8.

Hitting Apply in the Optimize dialog box modifies the parameters in the schematic with the optimized values that were calculated. Running DC analysis plots the optimized curve which is shown in Figure 9. The left cursor has been set to show that the resistance value is now 10kohm at 25C.

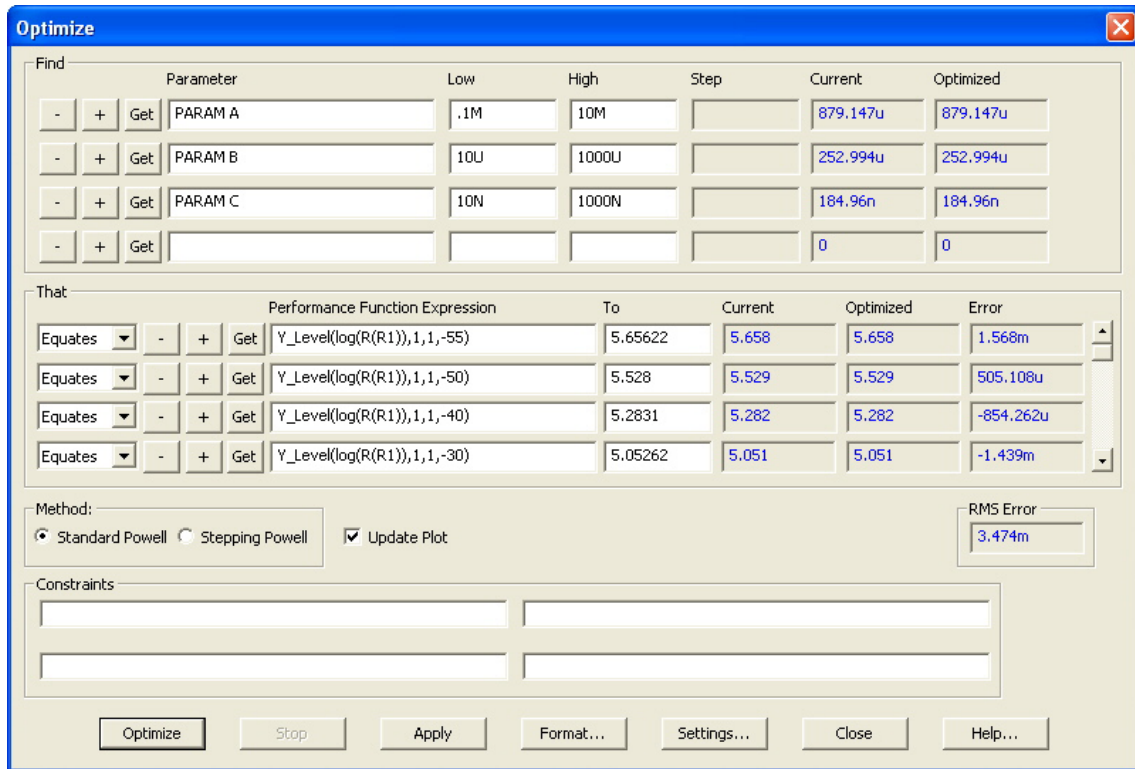


Fig. 8 - Optimized thermistor results in the Optimize dialog box

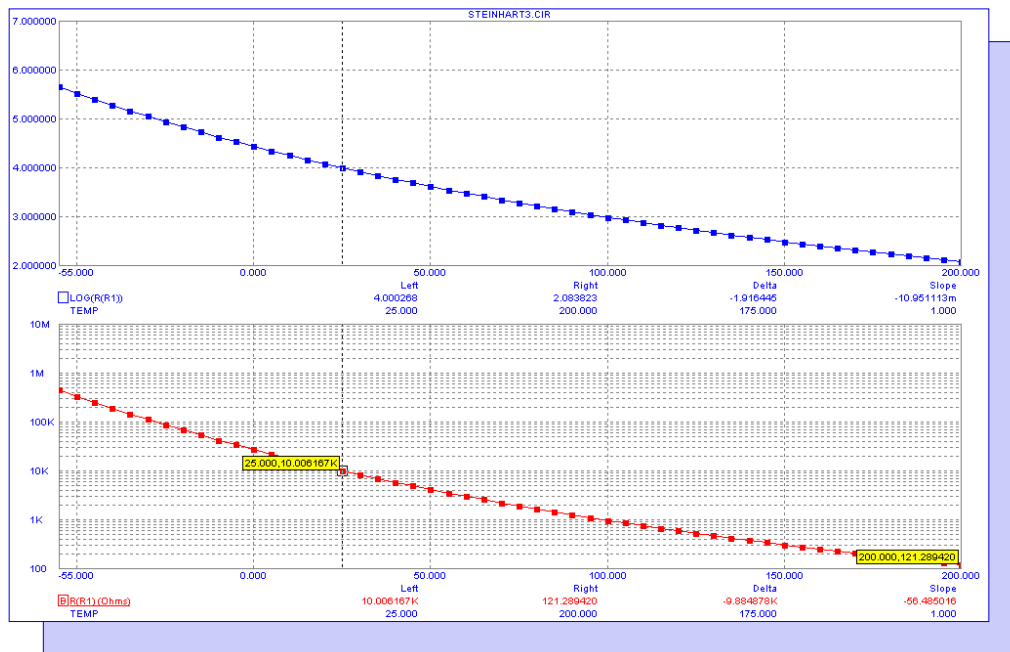


Fig. 9 - Optimized DC analysis results

Limiting Phase Plots to +/- 180 Degrees

The PH(z) function in Micro-Cap is used in AC analysis to plot the phase in degrees of the specified complex expression z. When plotting, this function checks the history of the phase plot in order to make the waveform continuous. For example, when the phase is at -179 degrees and then crosses back into the second quadrant, the phase value may be specified as either -181 degrees or +179 degrees. The PH(z) function selects -181 degrees as the value to plot to prevent large discontinuities in the waveform. However, since both values are valid, some users may want to display the phase with the values limited to +/-180 degrees. This function can be easily plotted by using the macro capabilities of the define statement. In order to demonstrate this, the simple passive network below will be simulated in AC analysis.

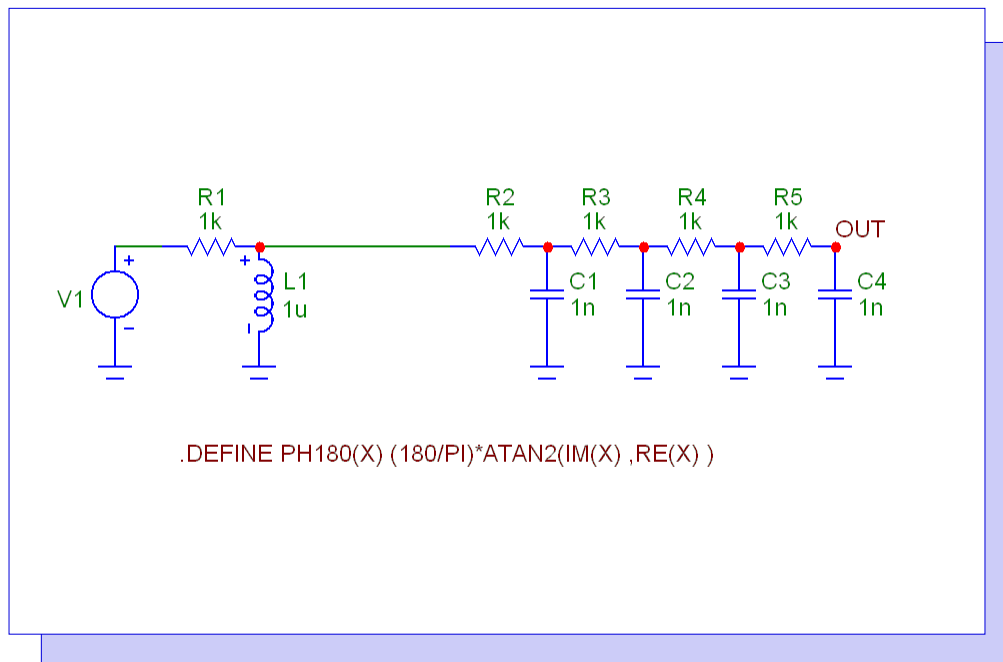


Fig. 10 - PH180 filter circuit example

The object of interest in this schematic is the define statement. This define statement creates a function to calculate the phase as follows:

```
.DEFINE PH180(X) (180/PI)*ATAN2(IM(X),RE(X))
```

The ATAN2 function calculates the inverse tangent value of the ratio of the imaginary value of the expression X versus the real value of the expression X. The ATAN2 function output is limited to values between PI and -PI. The inverse tangent value calculated is equivalent to the phase value of the expression X in units of radians. The radian value is then multiplied by the constant 180/PI to convert it into units of degrees. Since the ATAN2 function is limited to +/-PI, the equivalent limit in degrees will be +/-180.

Once this define statement is entered in the schematic file, the specified function PH180(X) is now available to be used in the AC Analysis Limits dialog box. This function would be used in the same manner as a function that is hardcoded in Micro-Cap. For example if you wanted to plot the phase of the voltage at node Out, the following expression would be entered in the Y Expression field:

PH180(V(Out))

An AC analysis is run on this schematic in which both the PH180(V(Out)) expression and the standard PH(V(Out)) expression are plotted. The resulting simulation is shown below.

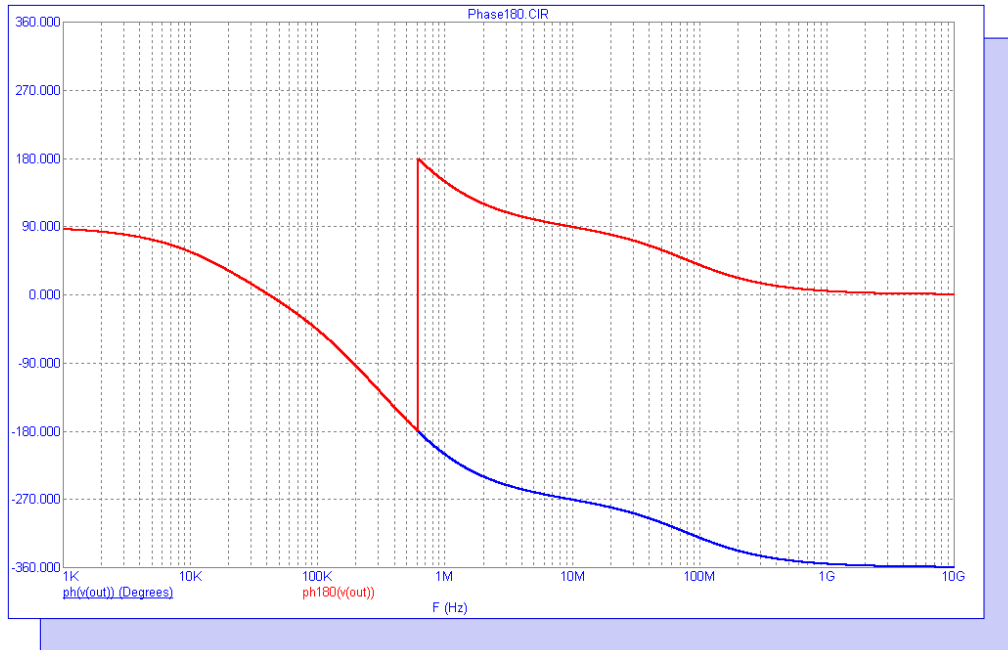


Fig. 11 - PH180 AC analysis plot

The two phase plots track each other exactly up until the point at which the phase crosses the -180 degree level. At that point the PH180 function transitions back up to the +180 degree level. The two plots then continue 360 degrees apart for the rest of the simulation.

The define statement for the PH180 function has to be accessible by the schematic. Rather than copying it into every schematic though, it can instead be placed into the Mcap.inc file. The Mcap.inc file is a file in the main Micro-Cap folder that contains a list of statements that are automatically included with every schematic. This file can be accessed within Micro-Cap by selecting the User Definitions option under the Options menu.

Product Sheet

Latest Version numbers

Micro-Cap 9Version 9.0.6.1
Micro-Cap 8Version 8.1.3
Micro-Cap 7Version 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 supportsupport@spectrum-soft.com
Web Site.....http://www.spectrum-soft.com
User Groupmicro-cap-subscribe@yahoogroups.com