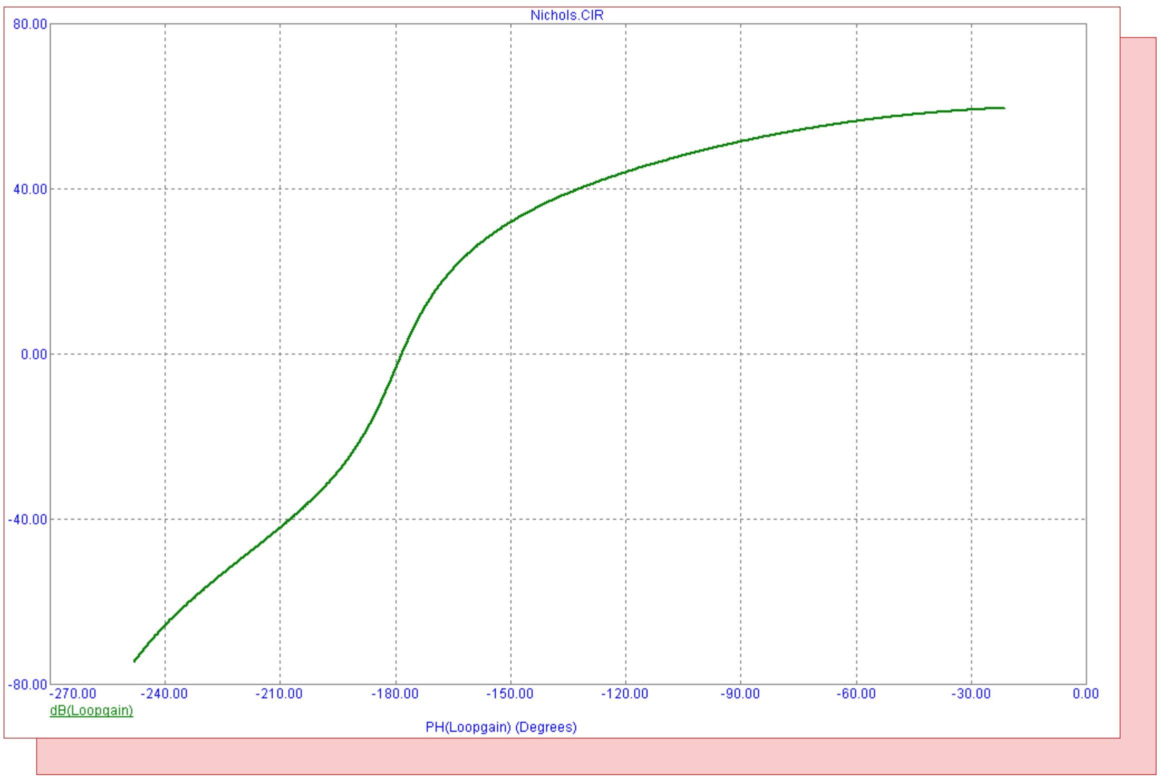
spectrum news

Applications for Micro-Cap™ Users

Fall 2010

News



Creating Nichols and Nyquist Plots

Featuring:

- Changing the Power Supplies of a Digital Library Part
- Optimizing Subcircuits in the Model Program
- Creating Nichols and Nyquist Plots

News In Preview

This newsletter's Q and A section describes how to update the information associated with the $User and $Company variables, and the purpose of the FILTER.BIN and WFB.BIN files. The Easily Overlooked Feature section describes how to edit grid text directly in the schematic.

The first article describes how to change the power supply that is assigned to a part from the digital library using global nodes and the OPTIONAL attribute.

The second article describes how to optimize user created subcircuit models within the Model program using an NTC thermistor as an example.

The third article describes how to create Nichols and Nyquist plots in AC analysis to aid in checking stability.

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 have enabled the title block in my schematic. The default settings for the title block have two variables called $User and $Company. However, no text appears in the title block where these variables are assigned. How do I get these variables to display the correct information?

Answer: The $User and $Company variables are initially set during the installation of the program. During the installation, the Company Information screen will come up that prompts for both a User Name and a Company Name. It is not mandatory that these fields be filled in, so the installation can be completed without having any entries set for these two variables.

Once the program has been installed, the information for these two variables can be modified within the Preferences dialog box using the following sequence:

1) Go to the Options menu and select Preferences.
2) In the list on the left hand side of the Preferences dialog box, select the General item that is within the Options group.
3) In the General page, there are two fields called User Name and Company Name. These fields control the information for the two variables. Edit or enter the information in these fields.
4) Click OK.

This information will now be correctly associated with the $User and $Company variables wherever they are used in the program.

Question: In the main MC10 folder, there are two files called FILTER.BIN and WFB.BIN. What are these files for?

Answer: The FILTER.BIN file stores all of the information needed for the Active and Passive Filter designers which can be found under the Design menu in Micro-Cap. This file contains all of the circuit configurations that can be used within these designers. If this file is not present, the designers will not work.

The WFB.BIN file stores all of the waveform information that the user has stored within the Waveform Buffer. If this file is not present, Micro-Cap will recreate it the next time the program runs.

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.

Editing Grid Text Directly in the Schematic

Grid text is any text that is entered into the schematic while in Text mode. Grid text can be used for entering node names, command statements, formula text, or just making comments in the schematic. Typically, to edit grid text you double click on the text object in the schematic while in Select mode, and the Grid Text dialog box will be invoked. This dialog box provides the capability to edit the content, colors, orientation, or the font of the text.

If just the text content is to be edited, it is now possible to edit it directly in the schematic and bypass the Grid Text dialog box. This requires the following sequence:

1) Enable the Select mode.

2) Hold the ALT key down on the keyboard while double clicking on the text object.

3) At this point, the text will appear within a rectangle on the schematic, and a text cursor will be available to edit the text. When the editing is complete, simply click outside of the rectangle to return to the normal view.

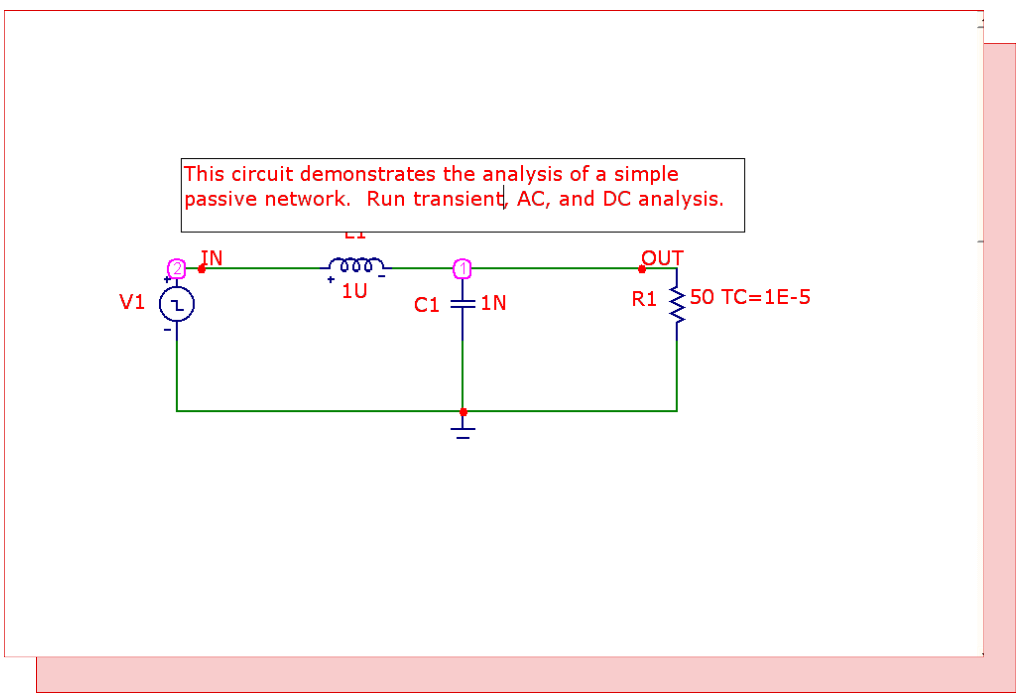


Fig. 1 - Editing grid text directly in the schematic

Changing the Power Supplies of a Digital Library Part

All of the components within the Digital Library have a default power supply defined for them. These default power supplies are defined within the I/O model that is assigned to each component. The I/O models for all of the digital parts that come with Micro-Cap can be found in the Digio.lib library file in the Library folder. For some digital families, the I/O model has been created to only handle a single power supply value such as 5V for the standard TTL devices. In other families, the I/O model can handle a range of power supply values. For example, the I/O model for the CD4000 series can handle power supplies ranging from 3V to 15V. The default power supply for the CD4000 series is 5V. One method for changing the default power supply that is assigned to a part from the digital library can be accomplished using a combination of global nodes and the OPTIONAL attribute.

The example in this article will use the CD4000 series of devices to demonstrate how to modify the power supply for a single device in the schematic. However, the CD4000 series also has a unique method of setting the default power supply for all of the CD4000 devices that are in the schematic. To change the power supply for all of these devices in the circuit, all one has to do is add the following statement either on a schematic page or in a text page.

.param CD4000_VDD=10V

All of the CD4000 components in this circuit would then use a single 10V supply. However, if the schematic contains CD4000 components that operate at different power supplies, then a different technique must be used to define the power supply for a single component. A simple example of such a circuit is shown in the figure below.

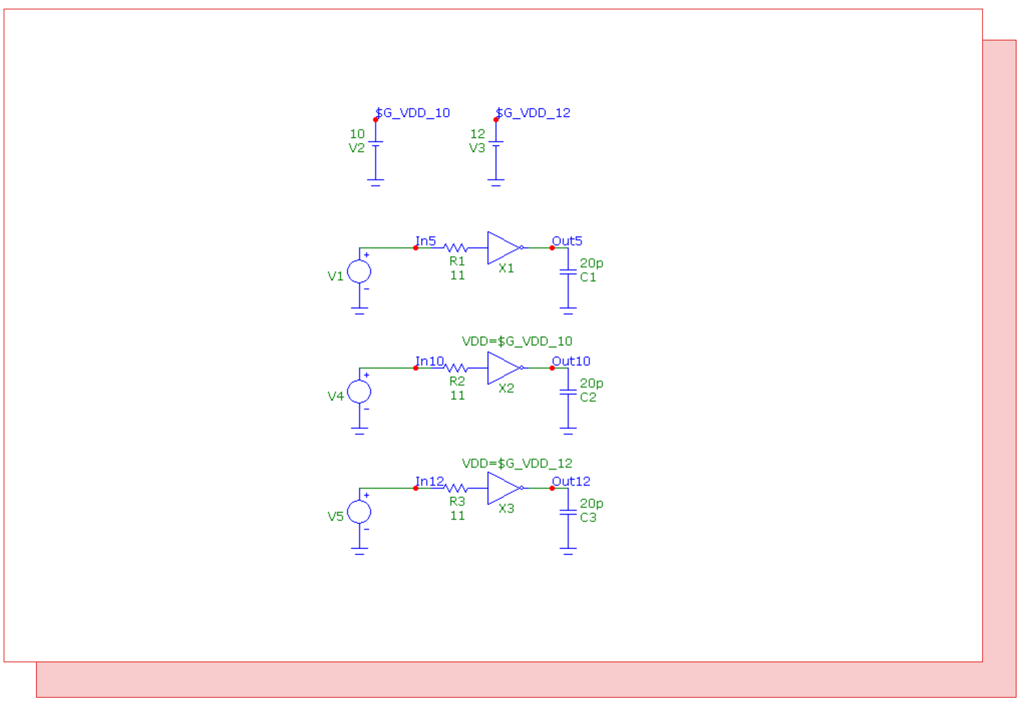


Fig. 2 - CD4000 devices using different power supplies

The circuit consist of three CD4049B inverters. Each inverter will operate on a different power supply. One will use the default 5V supply, and the other two will use a 10V and a 12V supply.

The first step is to create the power supplies on the schematic. In this case, the V2 battery has been defined to model the 10V supply, and the V3 battery has been defined to model the 12V supply. Text has been added to the output of these batteries in order to name the node. The 10V supply node has been named $G_VDD_10, and the 12V supply node has been named $G_VDD_12. Note that both of these node names begin with $G_. The use of the $G_ prefix declares that these nodes are global nodes. Global nodes are nodes whose names are globally available to all parts of the circuit, including the top level circuit and all macros and subckts used by it. A global node name must be used for the power supply node as that is the only method to connect the supply node to the underlying subcircuits that make up the corresponding I/O model for the digital device.

The next step is to look at the subcircuit header for the digital device model. The header can be viewed by double clicking on the device in the schematic to invoke the Attribute dialog box. In the text area at the bottom of the dialog box, the subcircuit model will be shown. Below is the subcircuit header for the CD4049B device.

```
.SUBCKT CD4049B INA OUTA
+  Optional: VDD=$G_CD4000_VDD VSS=$G_CD4000_VSS
+  params: MNTYMXDLY=0 IO_LEVEL=0
```

Within the header, there is an Optional keyword that declares optional nodes that may be connected externally to the subcircuit. For the digital library parts, the positive and negative power supply nodes are always declared within this Optional keyword. For the CD4049B model, the VDD and VSS nodes are the optional nodes and by default, they are connected to the global nodes $G_CD4000_VDD and $G_CD4000_VSS that are defined within the I/O model for this device.

The easiest method to connect optional nodes in a subcircuit to a supply in the schematic is to use the OPTIONAL attribute available in the Attribute dialog box. Since the X1 inverter is using the default 5V power supply, the OPTIONAL attribute for that component is left blank. The X2 inverter will use the 10V power supply, so it has its OPTIONAL attribute defined as:

VDD=$G_VDD_10

This overwrites the default node specified within the subcircuit header, so that the VDD node for the inverter is now connected to the $G_VDD_10 global node defined in the schematic. The X3 inverter will use the 12V power supply so it has its corresponding OPTIONAL attribute defined as:

VDD=$G_VDD_12

This connects its VDD node to the $G_VDD_12 global node in the schematic.

For the inverters in the schematic, the VSS node will continue to be connected to the default $G_CD4000_VSS which is just a grounded node.

When an analysis is entered, Micro-Cap will automatically connect the corresponding power supplies to any AtoD or DtoA interfaces that have been added to the circuit internally.

A 1us transient simulation is run on the circuit. Each inverter has the exact same external circuitry. The only difference lies within the power supply reference and the magnitude of the input pulse. The resulting transient simulation is shown below.

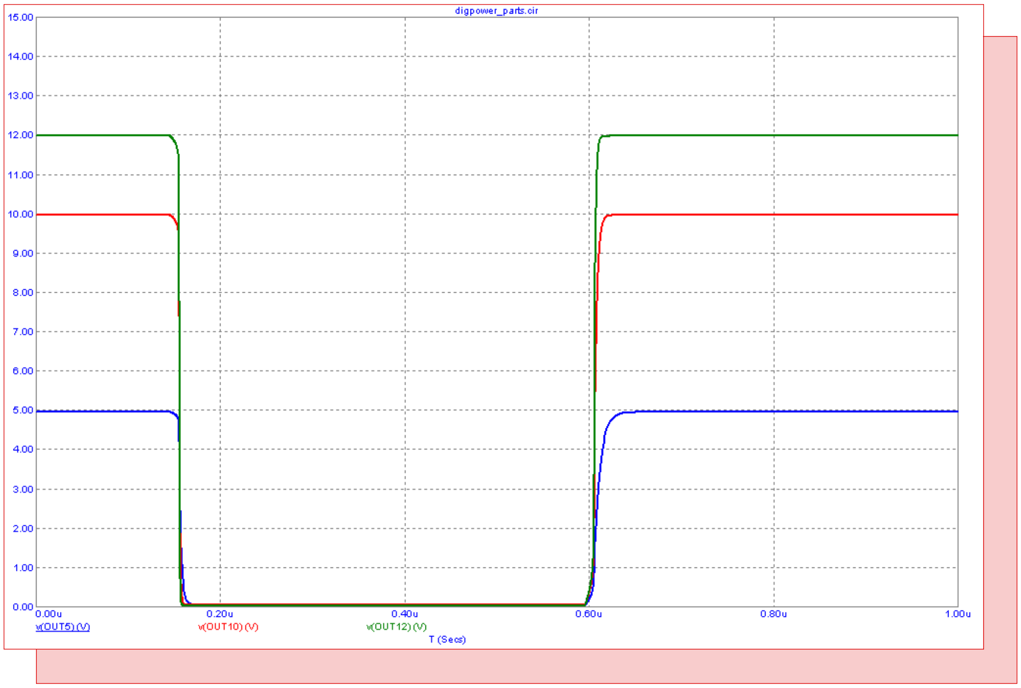


Fig. 3 - Transient output using different power supplies

The plot shows the output at each of the inverters. When the output of the inverter is at a one state, the analog voltage is dependent on the power supply assigned to the device. As expected, these inverters produce analog output voltages that correspond with the power supply voltage that they are using.

For the CD4000 devices, the .Param statement can also be used in conjunction with the OPTIONAL attributes. For example, in the schematic, if a couple of CD4000 devices were to operate at 10V, but the rest of the CD4000 devices were operating at 12V, simply place the following .param statement in the schematic:

.param CD4000_VDD=12V

Then add a 10V power supply to the schematic that uses a global node name, and define the OPTIONAL attributes for those devices that are to operate at 10V. The .Param statement overwrites the default setting for all of the CD4000 devices, but then the OPTIONAL attribute overwrites the .Param setting for those specific instances.

Optimizing Subcircuits in the Model Program

The Model program now has the capability to create subcircuits, macros, and other models based on test circuits provided by the user. This capability provides a method to optimize new libraries of parts using a template model for any type of component. This article will describe the process of optimizing a subcircuit model for an NTC thermistor in the Model program. The test circuit in this example is distributed with the professional version of Micro-Cap 10 as SteinHart4.cir.

The first step is to create a subcircuit model that can be used as a template. An NTC thermistor is a thermally sensitive resistor that adjusts its resistance with changes in temperature in a predictable manner. The behavior of an NTC thermistor can be modeled accurately through the Steinhart-Hart equation which is as follows:

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

where

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

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 following subcircuit uses .Param statements to define the Steinhart-Hart equation directly in a resistor component.

```
.SUBCKT NTC Plus Minus
.PARAM B3C={(B/(3*C))}
.PARAM TK={(TEMP+273.15)}
.PARAM A2={(.5*ALPHA)}
.PARAM ALPHA={((A-1/TK)/C)}
.PARAM TERM1={POW(BETA-A2,.333333)}
.PARAM BETA={SQRT( B3C*B3C*B3C + A2*A2)}
.PARAM A=1M
.PARAM B=100U
.PARAM C=100N
.PARAM RS={EXP( TERM1 - TERM2 )}
.PARAM TERM2={POW(BETA+A2,.33333)}
R1 Plus Minus {RS}
.ENDS NTC
```

The next step is to create the test circuit that can be used to optimize the circuit parameters. For the Steinhart-Hart equations, the values that need to be optimized are the A, B, and C parameters. These parameter values can be determined by optimizing the resistance versus temperature plot of the thermistor.

The circuit shown in Figure 4 can be used as the test circuit for the resistance versus temperature curve of the thermistor. The circuit consists of just a single component that uses the NTC subcircuit model described above. The ground is present because it is mandatory for a ground node to be present in all schematics.

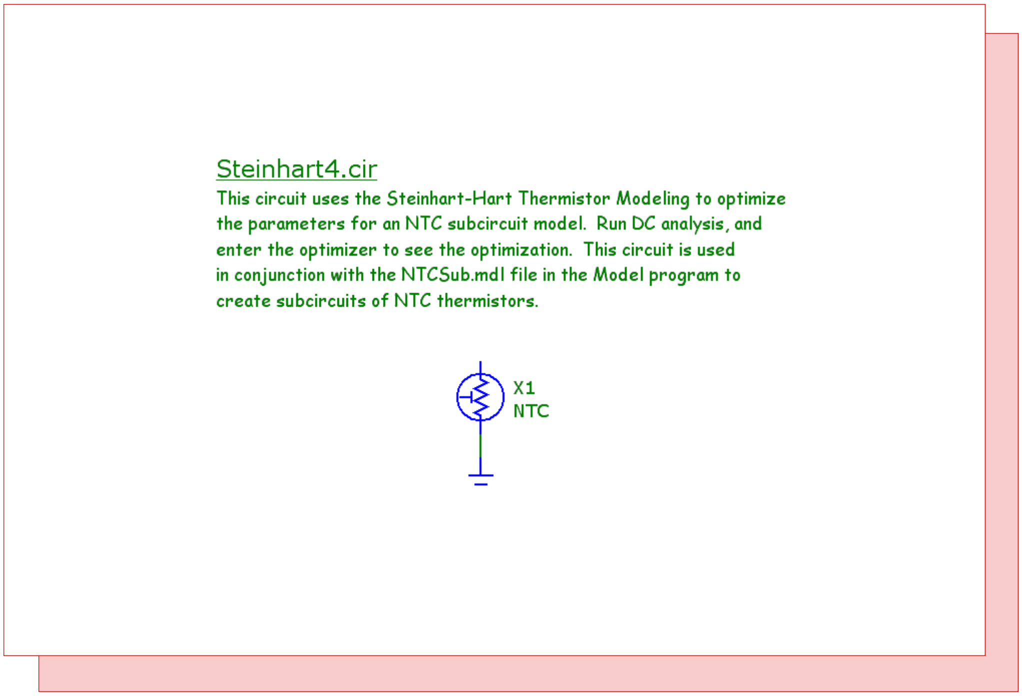


Fig. 4 - NTC thermistor optimization test circuit

Once the test circuit has been created, the corresponding analysis needs to be setup. For a resistance versus temperature curve, the best analysis to optimize this in is the DC analysis. The DC Analysis Limits dialog box for the test circuit is specified as below.

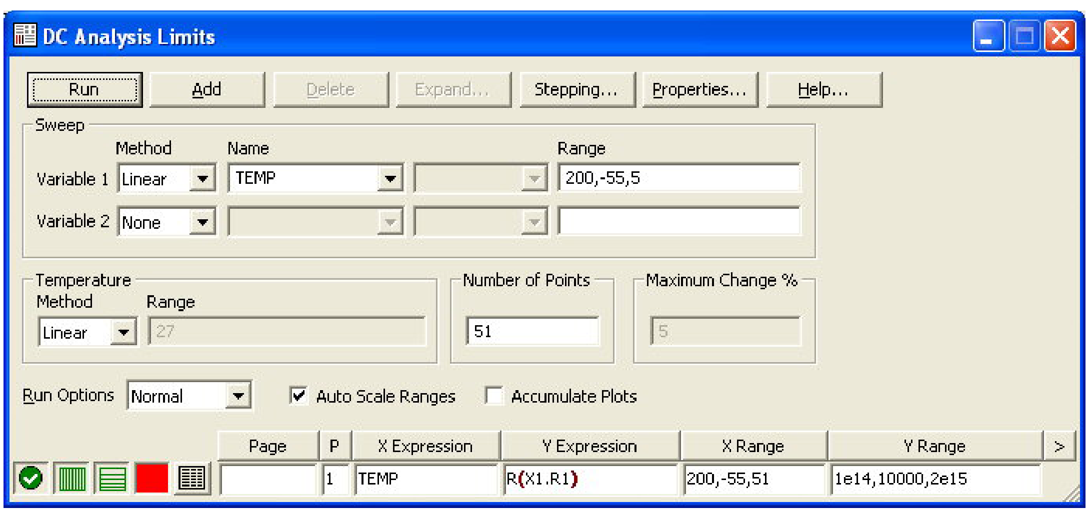


Fig. 5 - DC analysis limits for the test circuit

Temperature is selected as the Variable 1 option and is swept linearly from -55C to 200C. The resistance of the thermistor is plotted through the Y Expression R(X1.R1). This expression plots the resistance of the R1 resistor that is inside of the X1 component in the schematic. With the X Expression on that plot line set to the Temp variable, the resistance versus temperature curve of the thermistor has now been defined.

Once the test circuit has been saved, the NTC subcircuit can be optimized in the Model program. A new model file is created by selecting New from the Model menu. The New Part dialog box will be invoked automatically which prompts for the device type to specify for the first part in the model file. For a user created part, the part type selected should be User. At this point, the Analysis Type, Circuit, Waveform, and Graph Title can be specified. The Circuit field should be set to the test circuit that was created. Next, the Analysis Type field is specified as DC Analysis since that was the analysis type used in the test circuit. Once the circuit and analysis type have been set, a list of available waveforms will be shown in the Waveform field. Only waveforms that have been entered in the analysis limits of the test circuit will be shown. In this case, the expression TEMP vs. R(X1.R1) is selected. Finally, the Graph Title is defined as Temperature vs. Resistance. This title simply labels the plot in the Model program. The final settings for the dialog box are shown below.

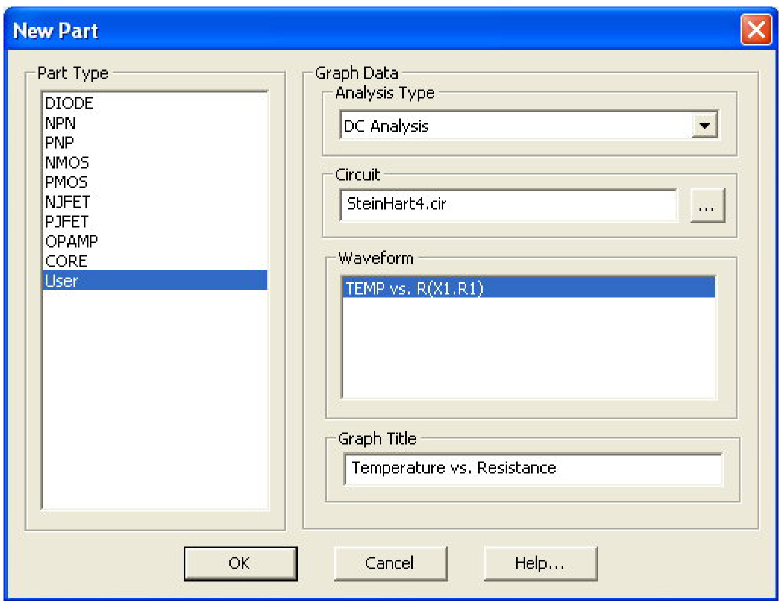


Fig. 6 - New Part dialog box settings for the NTC subcircuit

When the OK button is clicked, the model file will be displayed. All of the graph information that was specified in the New Part dialog box can also be edited in the main model file. The Analysis Type and Circuit fields are available to modify. The waveform can be changed by double clicking on the header above the graph data, and the graph title can be changed by double clicking on the title above the plot.

The circuit parameters to be optimized now need to be specified. In the Parameters section, click on the ... button to the right of the top text field. The Parameter to Optimize dialog box will appear. For the NTC, the A, B, and C parameters set in the .Param statements within the subcircuit are the objects that need to be optimized. The Parameter Type is set to Symbolic. The Parameter field will then contain a list of all of the symbolic variables available within the test circuit. The X1.A parameter is selected from this list. The Optimize option should be set to Yes. The final settings for the dialog box are shown in Figure 7. When the OK button is clicked, an entry called PARAM X1.A will now be present in the Parameters section. This same sequence is then repeated for the X1.B and the X1.C parameters on subsequent lines in the Parameters section.

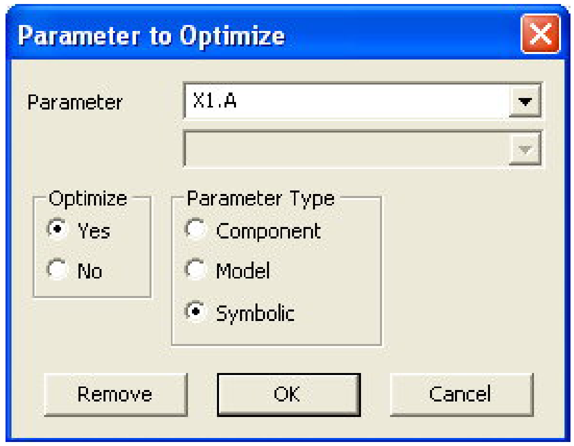


Fig. 7 - Parameter to Optimize dialog box settings for the A parameter

Data to optimize the curve to can now be entered. An NTC thermistor from Vishay with an R25 of 2k ohms is used for this example. The temperature and resistance pairs are entered in the graph data fields. Since the resistance data spans multiple decades, the Use Log Error option will be enabled since this typically provides a better fit for logarithmic data. The Optimize command is selected from the Model menu. The final results of the optimization is shown in the figure below.

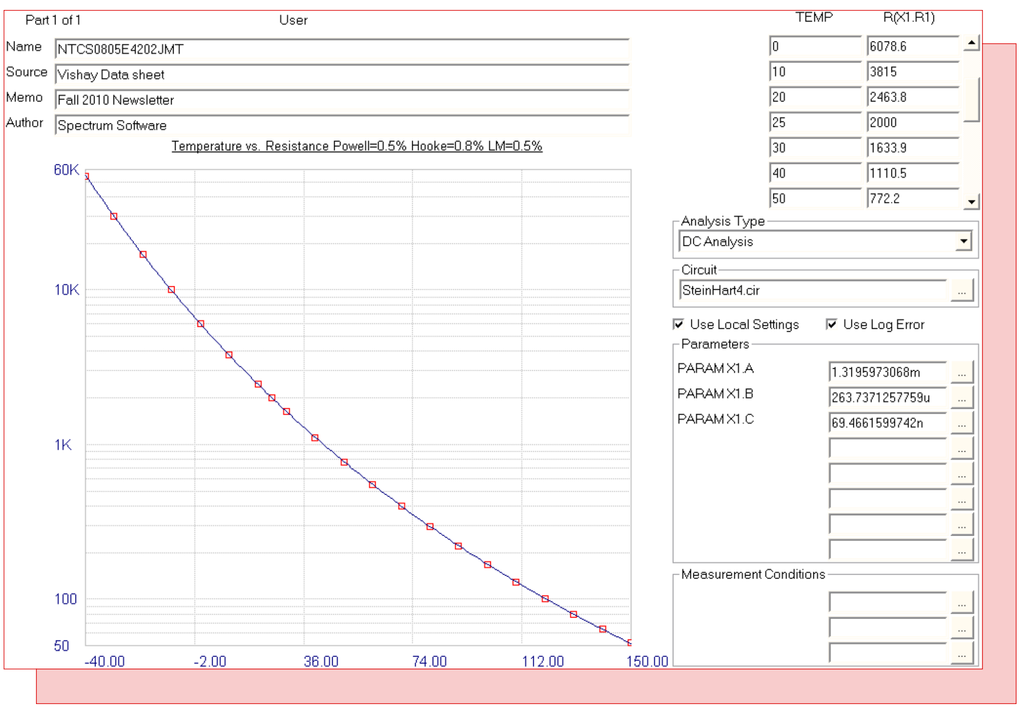


Fig. 8 - Final optimization results for the 2k ohm thermistor

As can be be seen in the plot, the Steinhart-Hart equation provides a very realistic match to the actual datasheet curve. All three optimization methods used were able to match the curve with an error margin below 1%. When multiple optimization methods are used, the best match will be shown as the final result. In this case, the best match has an error of 0.5%.

The optimized subcircuit can now be created by selecting the Create Model for this Part command under the Model menu. This command uses the name specified in the Name field as the name of the subcircuit. There is also an option to add this part to the component library, so you can immediately place it in a schematic. The subcircuit for the 2k ohm NTC thermistor is as follows:

```
.SUBCKT NTCS0805E4202JMT  Plus Minus
.PARAM B3C={(B/(3*C))}
.PARAM TK={(TEMP+273.15)}
.PARAM A2={(.5*ALPHA)}
.PARAM ALPHA={((A-1/TK)/C)}
.PARAM TERM1={POW(BETA-A2,.333333)}
.PARAM BETA={SQRT( B3C*B3C*B3C + A2*A2)}
.PARAM A=1.3195973068m
.PARAM B=263.7371257759u
.PARAM C=69.4661599742n
.PARAM RS={EXP( TERM1 - TERM2 )}
.PARAM TERM2={POW(BETA+A2,.33333)}
R1 Plus Minus {RS}
.ENDS NTCS0805E4202JMT
```

If multiple NTC parts were to be created, the current configuration can be saved as a template for easy reuse in optimizing other NTC models. First, the Name field should be changed to something that describes the generic component. In this case, the Name field could be set to NTC. Next, select the Save Part to Template Library command under the Model menu. Now when a new part is added to any model file, NTC will be one of the part type options. It is recommended that the template be stored in its own model file. A new model file could then be created that would store all the individual instances of the optimized models.

The NTC thermistor is a relatively simple model to setup for optimization. More complex models can also be optimized in the Model program using multiple test circuits. The Add Graph command under the Model menu, provides a new screen for the model that can use a different test circuit and optimize other circuit parameters within the model.

Creating Nichols and Nyquist Plots

The typical polar plot used in AC analysis is the well known Bode plot. However, there are numerous other polar plot types that can be useful in analyzing a circuit. Two of the more popular ones are the Nichols and Nyquist plots. Both of these plots can be easily created within Micro-Cap by setting up the appropriate expressions in the AC analysis limits.

Nichols plot
The Nichols plot is frequently used with signal processing and closed loop control systems. This plot provides a means to quickly read specifications such as phase margin and gain margin to help determine the stability of the system. In this plot, the magnitude of the output in decibels is plotted versus the output phase of the frequency response. In order to do this in Micro-Cap, in the AC Analysis limits, set the Y Expression to:

dB(Output)

and the corresponding X Expression to:

ph(Output)

where Output is the desired output expression of the circuit. That can be any expression such as V(Out), I(RLoad), or something more complex. The schematic below uses the open loop gain technique described in the Winter 2001 newsletter issue. An AC analysis is run on this schematic that produces the Nichols plot shown in Figure 10.

In this analysis, the expression dB(Loopgain) is plotted versus ph(Loopgain). Loopgain is a variable defined in the schematic to measure the open loop gain of the circuit.

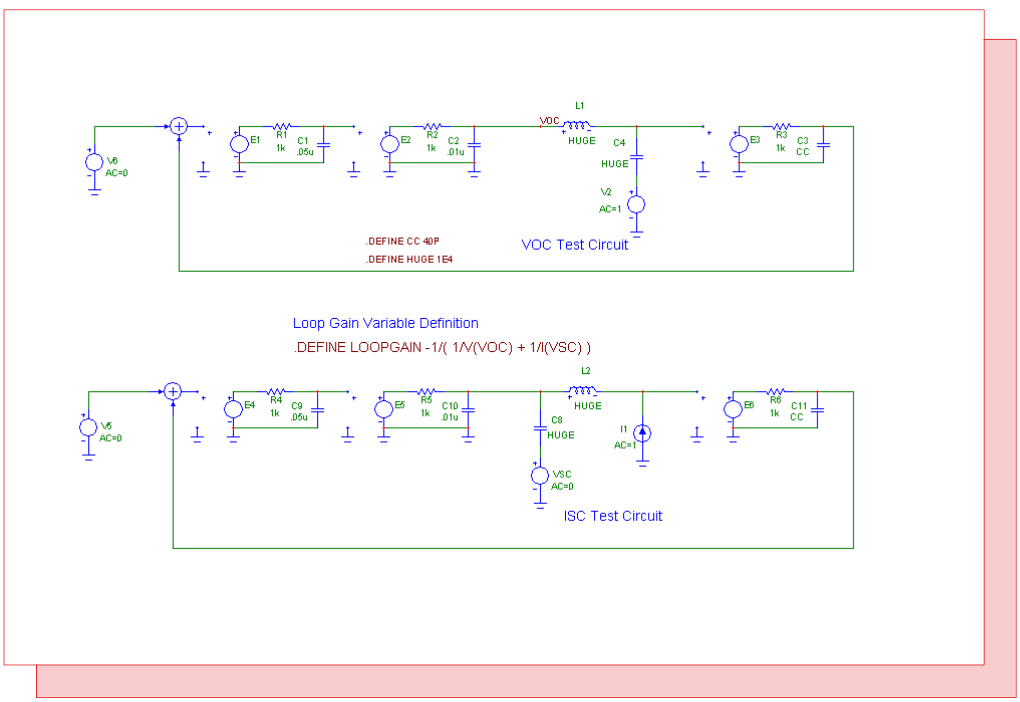


Fig. 9 - Open loop gain measurement circuit

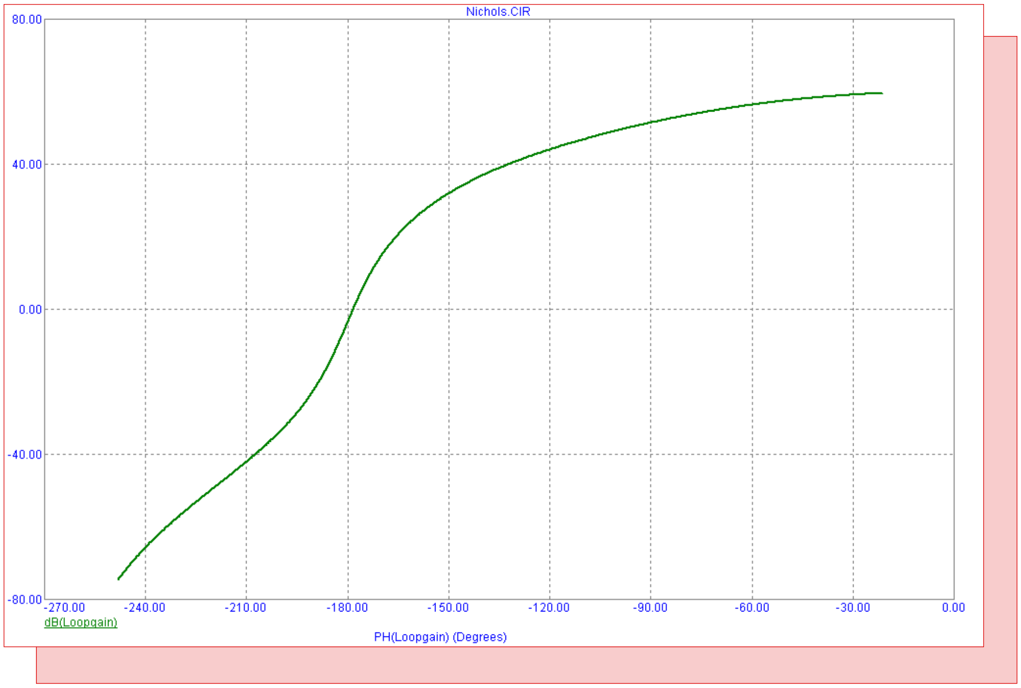


Fig. 10 - Nichols plot

Nyquist plot
The Nyquist plot is also frequently used with signal processing and control systems to help determine the stability of the system. It can be used to help measure the phase margin and gain margin along with the poles and zeros of the transfer function. In this plot, the imaginary quantity of the output is plotted versus the real quantity of the output. In order to do this in Micro-Cap, in the AC Analysis limits, set the Y Expression to:

Im(Output)

and the corresponding X expression to:

Re(Output)

where Output is again any type of expression that the user would like plotted. An AC analysis is run on the previous open loop gain schematic which produces the Nyquist plot shown in Figure 11.

In this analysis, Im(Loopgain) is plotted versus Re(Loopgain).

Any variety of polar plots can be created in this manner by just setting the X Expression and Y Expression fields on a plot line to the approprate expressions.

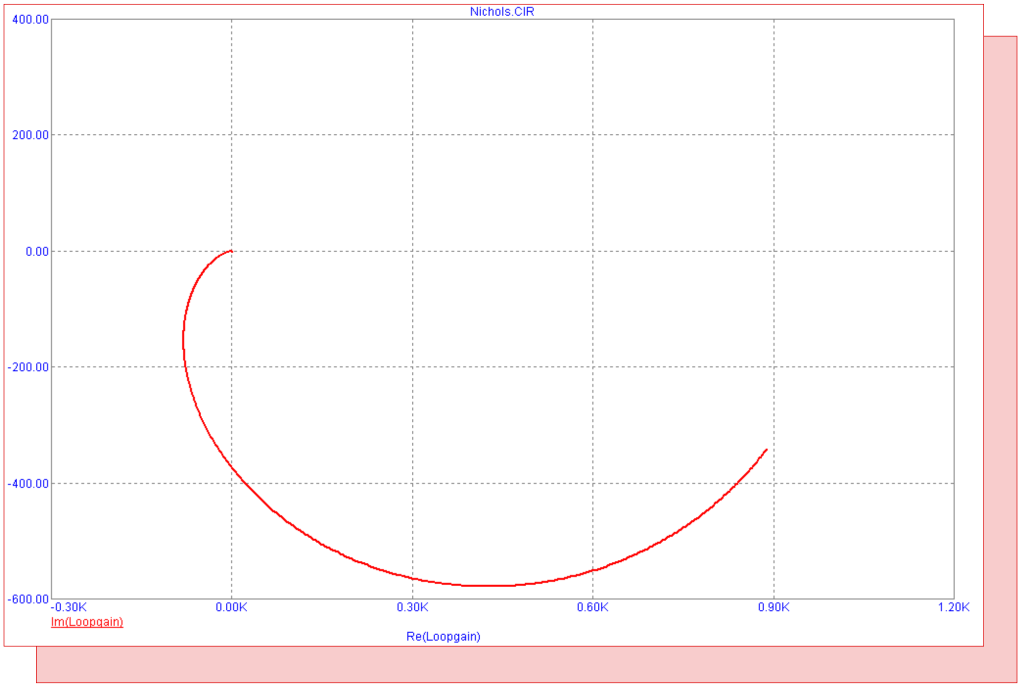


Fig. 11 - Nyquist plot

Product Sheet

Latest Version numbers

Micro-Cap 10	Version 10.0.2
Micro-Cap 9	Version 9.0.7
Micro-Cap 8	Version 8.1.3
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
User Group	micro-cap-subscribe@yahoogroups.com