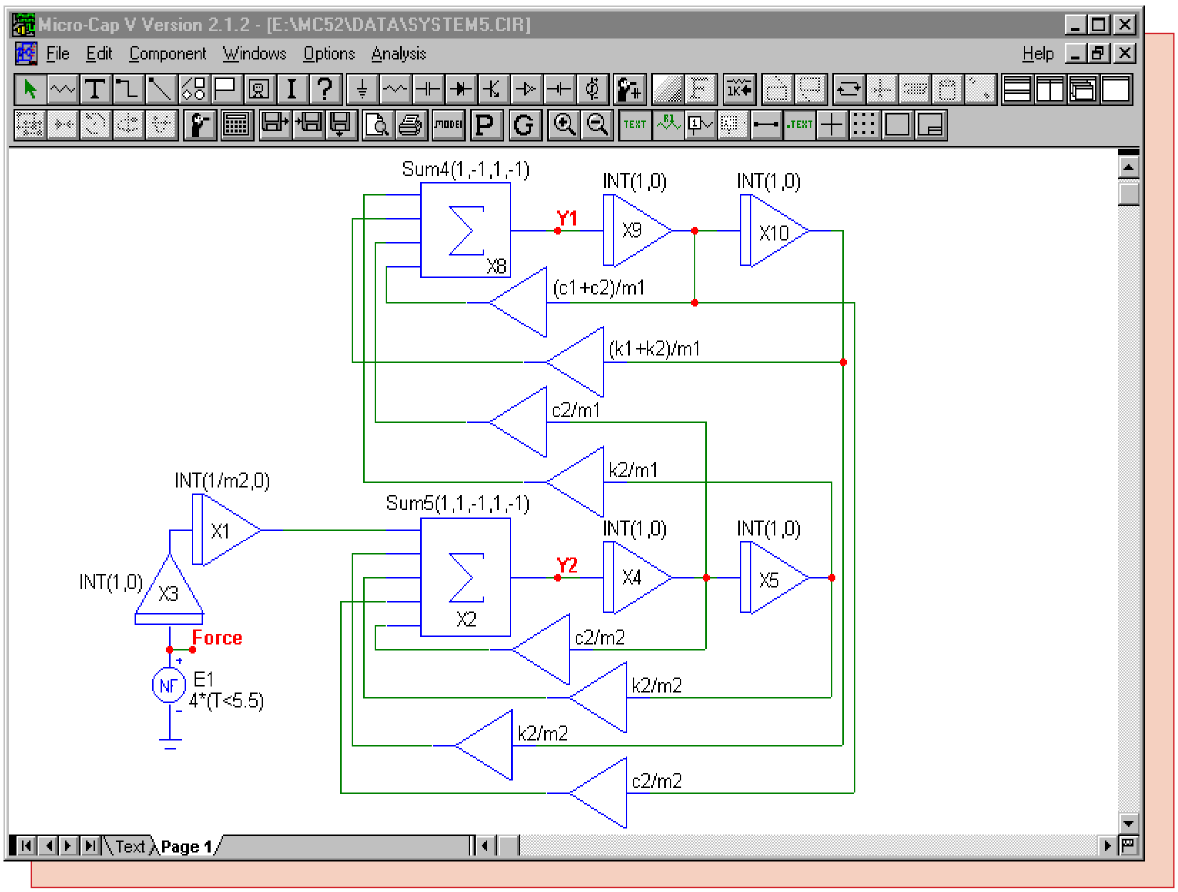
spectrum news

Applications for Micro-Cap™ Users

Fall 1998

Solving Differential Equations



Featuring:
- Revised Pink Noise Source
- Solving Differential Equations
- Thermistor Macro
- Windows NT and Service Pack 4 Incompatibilities

News In Preview

This issue features a warning about an incompatibility between Micro-Cap V Version 2's security key driver and Windows NT's Service Pack 4. This is a must read for anyone that is using Windows NT. There is another article on the pink noise source that revises the attenuation from -6dB to the correct -3dB. There is also an article that describes a method for using MC5 to solve differential equations. Finally, there is an article that describes the construction of a thermistor macro model.

Contents

Book Recommendations

Micro-Cap / 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
- *Semiconductor Device Modeling with SPICE*, Paolo Antognetti and Giuseppe Massobrio McGraw-Hill, Second Edition, 1993. ISBN# 0-07-002107-4
- *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
- *SMPS Simulation with SPICE 3,* Steven M. Sandler, McGraw Hill, First Edition, 1997. ISBN# 0-07-913227-8
- *MOSFET Modeling with SPICE Principles and Practice,* Daniel Foty, Prentice Hall, First Edition, 1997. ISBN# 0-13-227935-5

German

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

Design

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

Micro-Cap V Question and Answer

Question: I just upgraded from the Macintosh version of MC4 to MC5 Windows version. Can I import my circuits from MC4 into MC5?

Answer: Circuits can be transferred from the MC4 Macintosh version to the MC5 Windows version. This process will require two third party programs. The steps are as follows:

1) Launch a resource editor program. ResEdit from Apple is one example of such programs. This program lets you change the Creator and Type that the file is defined as. The Creator and Type fields link the file to a specific application and define it as a specific type of file. For a MC4 circuit file, the Type is 'MC4C' and the Creator is 'EXVP'. These fields need to be edited so that the Type field is defined as 'TEXT' and the Creator field is blank. Save the new resource settings. The circuit file has now been converted into a standard text file. It should appear on the hard drive now using the blank page icon.

2) Launch a Mac/PC conversion utility if your system doesn't do it automatically. Apple File Exchange, also from Apple, is an example of such a program. The utility needs to perform a text translation on the file. Essentially, what it needs to do is replace every carriage return in the Macintosh file with a carriage return / line feed. If this is not done, the circuit file will appear double spaced on a PC and will not load into MC5.

3) Copy the file over to the PC. Launch MC5. Go to the File menu and click on Open. Select the circuit file and MC5 will automatically convert it into the MC5 format. Saving the file at this point will save it in the MC5 format. Each individual circuit file would need to be processed this way.

Any user created macros or subcircuits would need to be recreated in the Component and Shape Editor of MC5. The actual macro circuit or subcircuit model may be brought over in the fashion described above, but the schematic shape and component information would need to be reentered.

Question: I just received a BSIM V3 model from my MOSFET vendor. I select an NMOS or a PMOS from the Component menu and give it a model name, but then I am unable to change the level parameter to the BSIM level 8 when I click the Edit command button. It only wants to stay with the standard SPICE levels of 1-3. How can I implement the BSIM model into MC5?

Answer: The PMOS and NMOS are going to default to a SPICE level 1-3 if the BSIM model is not available when the Attribute dialog box is opened. To make the BSIM model available, copy the model statement into the text area of the circuit file, or enter the file name that the model resides in into the NOM.LIB file. The NOM.LIB file is located in the DATA directory, and the format to enter a new library is to place the following on a new line:

.LIB "library.ext"

where library.ext is the name of the library file. Once either of the two above actions has been performed, an NMOS or PMOS component can be placed in the schematic, or a MOSFET that currently exists on the schematic can be double clicked while in select mode. When the Attribute dialog box opens, click on the Models command button and a list of all available MOSFET models will be shown in the right hand window. Choose the BSIM model from this list and it will be available for simulation.

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 an icon or a menu item.

Characteristics Dialog Box

Many of the characteristics of the active window can be changed using the Characteristics dialog box. This dialog box can be invoked by hitting the F10 hotkey or by double clicking in the window, except on an object, while in select mode. Changes made in the dialog box only affect the loaded file. The Characteristics dialog box can control the settings of each of the following windows.

Schematic - The Characteristics dialog box controls the color and font of the text area and the colors in the schematic area. In the schematic area, the dialog box controls the color of the components, wires, select mode, node numbers and voltages, pin connections, digital path, and the background.

Analysis Plot - The Characteristics dialog box controls the line width of the waveform, which plot group it is in, and whether it is showing at the time. It is also possible to set the range of a waveform, its numeric format, and the format for the scales. The colors and fonts for the window can also be edited in the dialog box. The color control is for the scale text, grid, window background, graph background, and any available waveforms.

Performance Plot - The Characteristics dialog box controls everything in this window that it does in the analysis plot window. In addition, any performance plot waveform may have its title, function, expression, X-axis, boolean, and settings edited in the dialog box.

3D Plot - The Characteristics dialog box controls the 3D plot display options, the value and scale formats, and the font and colors of the plot. The color control is for patches, background, planes, and grids. The variables of the 3D plot may also be edited here.

Monte Carlo - The Characteristics dialog box controls the fonts and the colors of the backgrounds, bars, and grids. The existing Monte Carlo plot can have its title, function, expression, boolean, and format edited here.

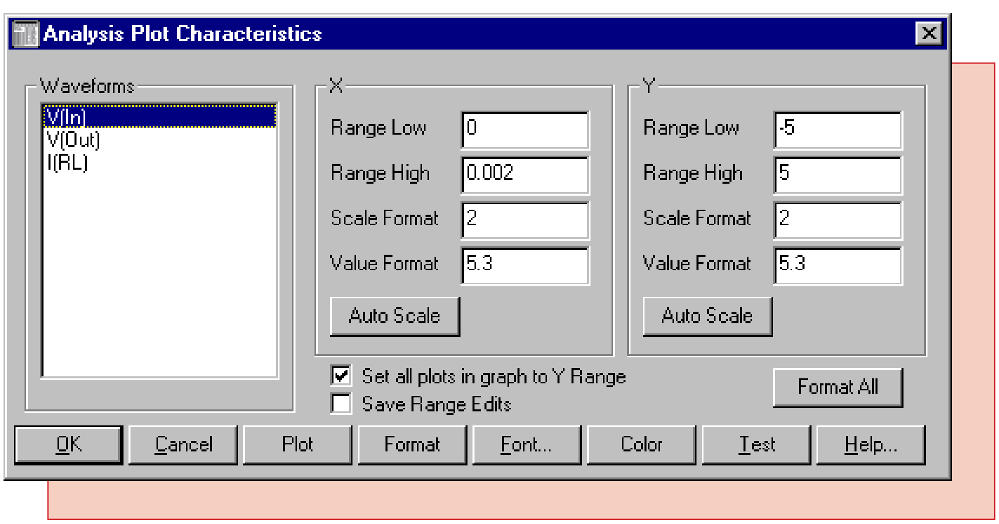


Fig. 1 - Analysis Plot Characteristics Dialog Box

Revised Pink Noise Source

In the Summer 1998 issue of the Spectrum newsletter, a pink noise source was modelled that converted white noise with a -6dB/octave bandpass filter. However, as one user pointed out, the correct attenuation level for a pink noise filter should be -3dB/octave. This article covers two ways of creating a filter to match this specification. The first method uses a Laplace source to model an ideal representation of the filter. The second method uses a passive filter that approximates the -3dB/octave characteristic.

Ideal Filter

The circuit in Figure 2 is the pink noise current source macro, and the circuit in Figure 3 is the pink noise voltage source macro. There are two parameters for each macro: NMAG and FC1. NMAG defines the magnitude of the noise current or voltage at low frequencies. FC1 defines the cutoff frequency for the filter at which frequency the attenuation band begins to take effect. The R1, V1, and E2 components produce a white noise voltage source at a magnitude of NMAG V/ $Hz^{1/2}$. For further description of the white noise source, see the article "White and Pink Noise Source Macros" in the Summer 1998 issue.

The final element in each macro is the Laplace source. The Laplace source models an ideal -3dB/ octave lowpass filter. The current noise source has a LFIofV source, and the voltage noise source has a LFVofV source. The Laplace source uses the voltage produced from the E2 source, which is the white noise voltage, as its input. The LAPLACE attribute of the Laplace source is defined as:

(freq<=fc1)+(freq>fc1)*(sqrt(fc1*2*PI)/sqrt(s))

This expression defines two different gains that are dependent on what the frequency is. When the frequency is less than or equal to the value of the parameter FC1, the gain of the source is 1. When the frequency is greater than FC1, the gain of the source is sqrt(fc1*2*PI)/sqrt(s). The 1/ sqrt(s) creates the desired -3dB attenuation, and the sqrt(fc1*2*PI) ensures that the gain is equal to 1 at frequency FC1 to provide a smooth transition from the low frequency gain. One thing to note in this equation is that the frequency is being defined with the variable freq instead of the normal variable F. This is done because the only variable that Laplace sources can handle is the S variable. Therefore, the frequency is modelled through the S variable by the following .define for freq:

.define freq (sqrt((-s*s)/(4*PI*PI)))

This equation produces the exact equivalent of frequency in the S domain.

These sources will only have an effect when a noise analysis is run under the AC analysis. The plots in Figure 4 are the noise analysis results of the pink noise voltage source. The pink noise voltage source has been defined with a magnitude of 100nV / $Hz^{1/2}$ and a cutoff frequency of 10Hz. The noise analysis has been simulated over the frequency range of 100KHz to .1Hz. The top plot displays the magnitude of the pink noise voltage source through the expression ONOISE. At low frequencies the magnitude is 100n, but as soon as the cutoff frequency of 10Hz is reached, the waveform begins its attenuation. The bottom plot displays the magnitude of the pink noise voltage source in dB through the expression dB(ONOISE). Cursors were placed at 100Hz and 200Hz in this plot. The attenuation over this octave and any other octave after the cutoff frequency is -3.01dB which is the necessary characteristic of the low pass filter.

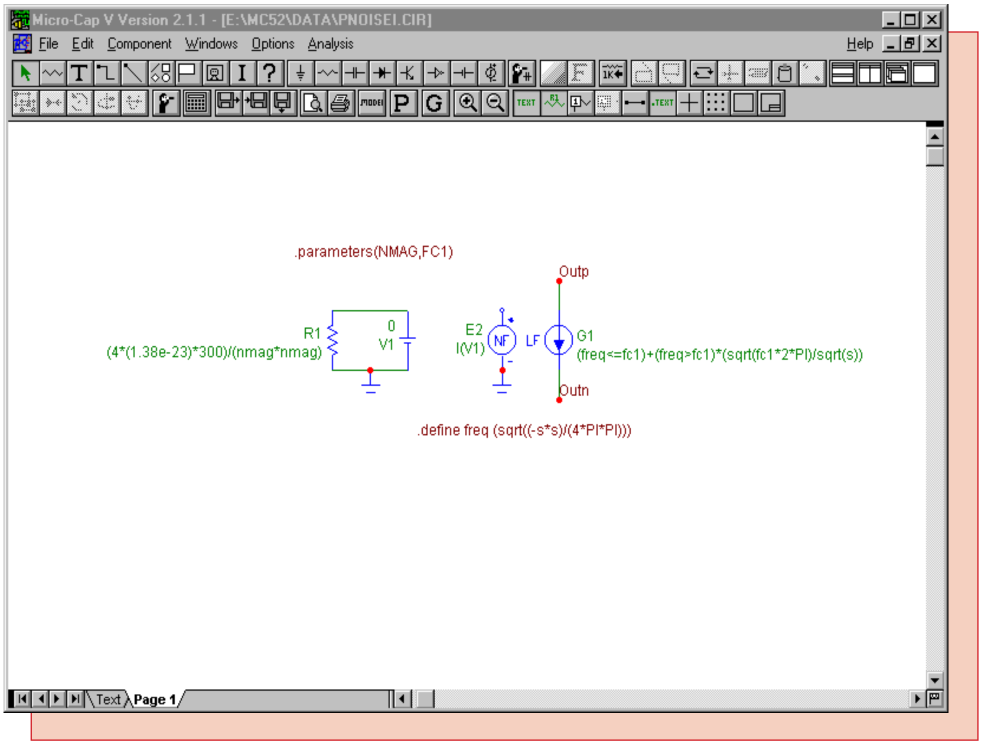


Fig. 2 - Pink Noise Current Source Macro

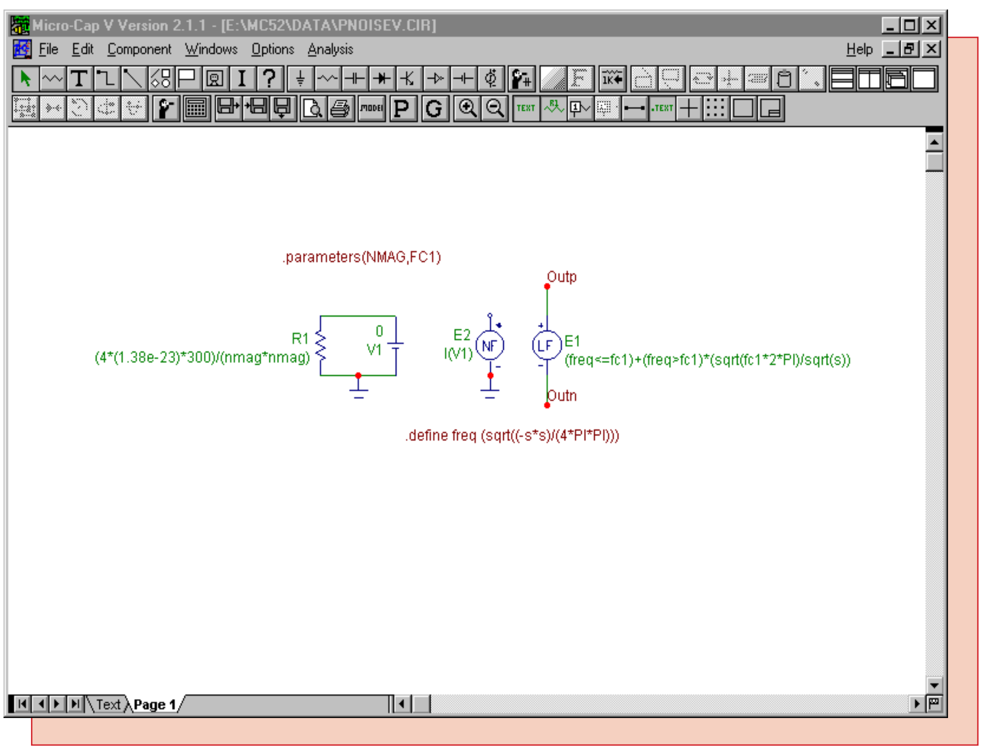


Fig. 3 - Pink Noise Voltage Source Macro

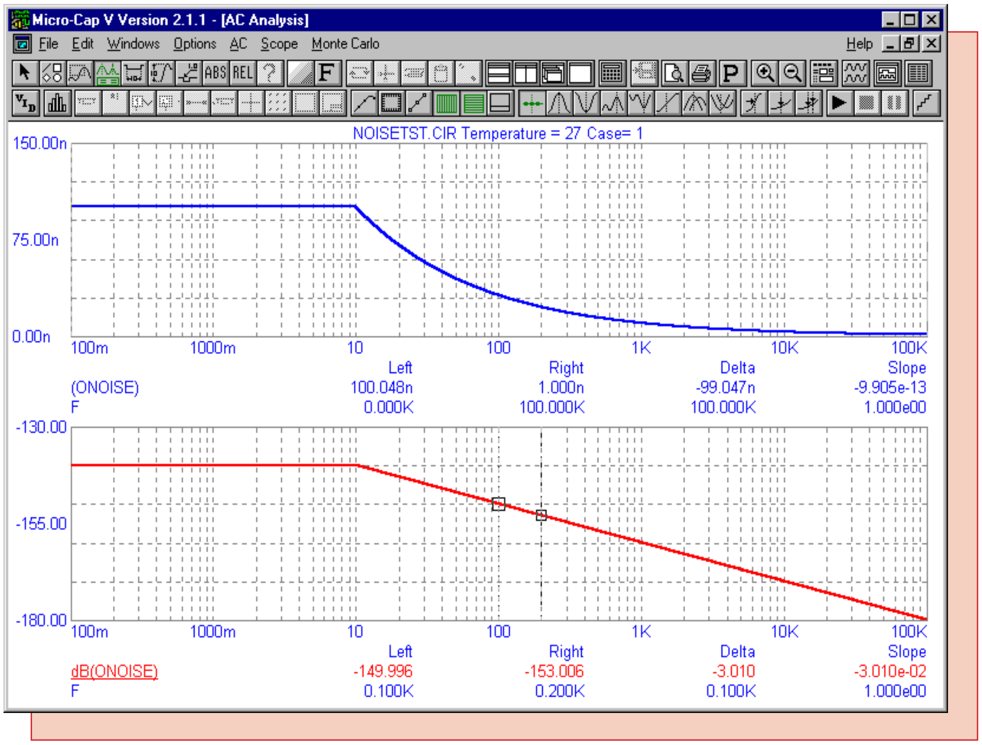


Fig. 4 - Pink Noise Output

Passive Filter

The second filter method is to create the filter out of passive elements. Figure 5 displays two such filters that will approximate a -3dB/octave attenuation from 10Hz to 20KHz. The top filter contains elements with exact values while the bottom filter contains elements with practical values. These filters can be used in any transient or AC analysis. However, to make them applicable as a noise source in a noise analysis a couple of changes would need to be made. First, the voltage source would need to be replaced with the white noise voltage source. Second, each resistor would need to be replaced with the dependent source IofV. The IofV would have to have its inputs connected across its outputs and its VALUE attribute defined as 1/R where R is the value of the resistance it is replacing. The resistors need to be replaced for a noise analysis because each resistor is in itself a noise source and would add to the noise of the pink source. For a standard AC or transient analysis, the circuits can be left as is, and the value of the voltage sources can be adjusted to meet the low frequency noise specification.

Figure 6 displays the analysis results when a standard AC analysis is run over the frequency range of 1Hz to 100KHz. The plots display the error in the pink noise filters as opposed to an ideal -3dB filter. For an ideal filter, the error would be zero, but for a passive element filter, the best that can be hoped for is an equiripple approximation. The top plot is the error of the filter with exact values, and the bottom plot is the error of the filter with practical values. As can be expected, the filter with exact values provides the better equiripple approximation although both provide a good representation over the desired frequency range.

Thanks to Rodger Rosenbaum from Trace Engineering for his help in providing one of the filters and other information for this article. The other filter comes from the 2nd edition of the textbook, Art of Electronics, by Paul Horowitz and Winfield Hill.

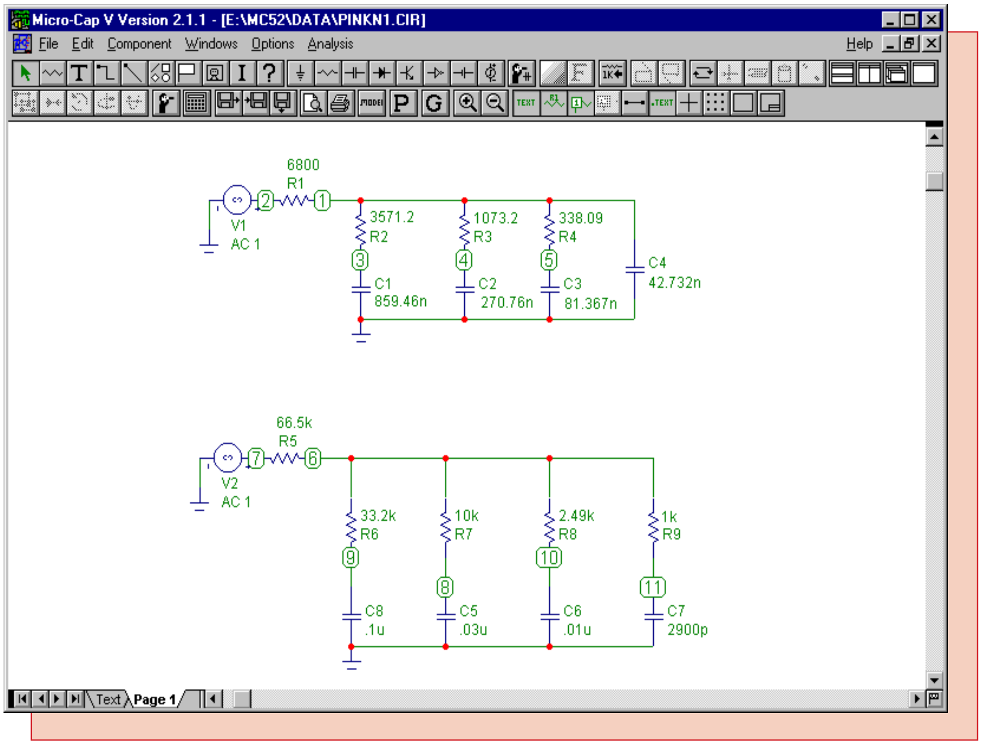


Fig. 5 - Passive Filter Representations

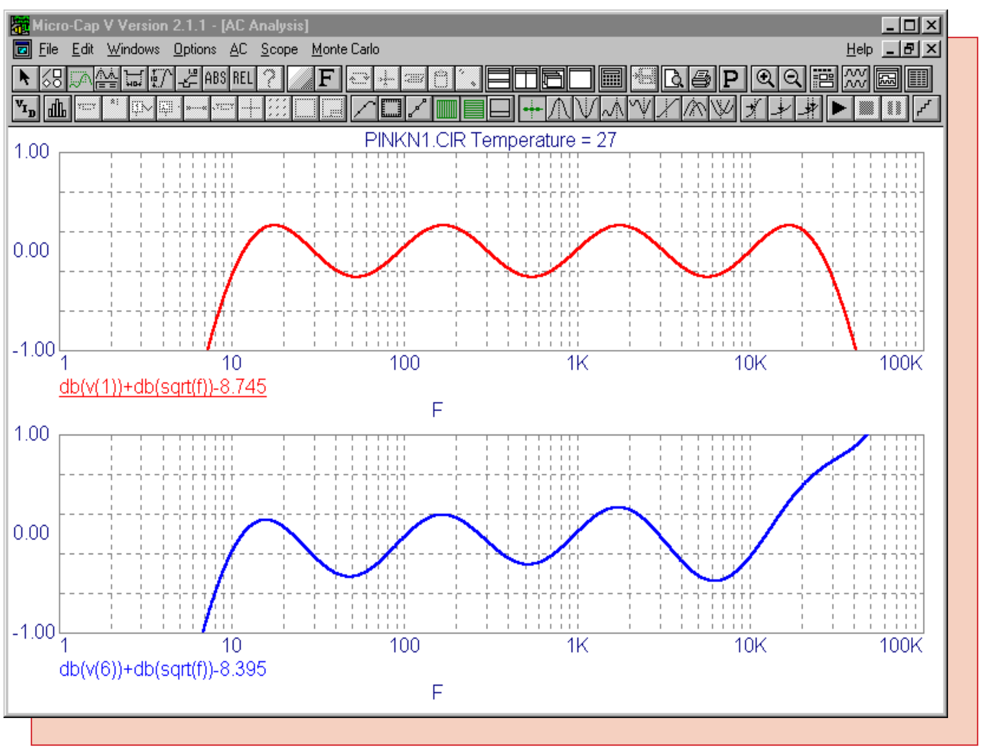


Fig. 6 - -3dB Filter Error Plot

Solving Differential Equations

Micro-Cap is of course known for its circuit simulation capabilities. However, the nature of SPICE's iterative process allows it to simulate many types of systems that can be modelled through standard equations and differential equations. This article covers the method of simulating spring systems that can be modelled with a single differential equation or coupled differential equations.

Single Differential Equation

The system in Figure 7 consists of a mass, m, connected to a spring with a spring constant of k. A force P is applied to the system. A dashpot in the system produces a counteracting force of friction with a coefficient of friction c. The simulation will determine the displacement, y, of the mass due to the application of the force P. This entire system is governed by the differential equation:

m*y'' + c*y' + k*y = P

Both the displacement and the power will be considered in relation to time. After integrating the above equation twice and solving for y, the new integral equation produced is:

y(t) = (1/m)*S2(P(t)) - (c/m)*S(y(t)) - (k/m)*S2(y(t))

S is the integral operator and S2 is the double integral operator. The circuit in Figure 8 implements this equation.

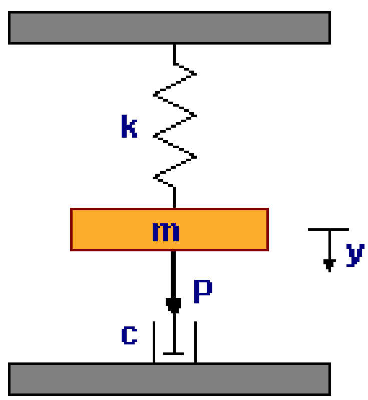


Fig. 7 - Mass on a Spring

The entire circuit consists of 4 INT macros, 1 SUM3 macro, 1 AMP macro, and a nonlinear function voltage source. The (1/m)*S2(P(t)) product is produced by the X5 and X1 INT macros. The X5 macro integrates the force the first time and then the X1 macro integrates the force the second time and supplies it with its 1/m coefficient. The output of X1 is then fed into the SUM3 macro.

The (c/m)*S(y(t)) product is produced by the X3 INT macro and the AMP macro. The X3 macro integrates the output of the SUM3 macro which is the displacement y. This signal is then fed into the AMP macro which multiplies it by the coefficient c/m. The INT macro doesn't supply the coefficient in this case, as it does in the other two products, because the output of the integrator is subsequently used to produce S2(y(t)). The resultant product is then fed into the SUM3 macro.

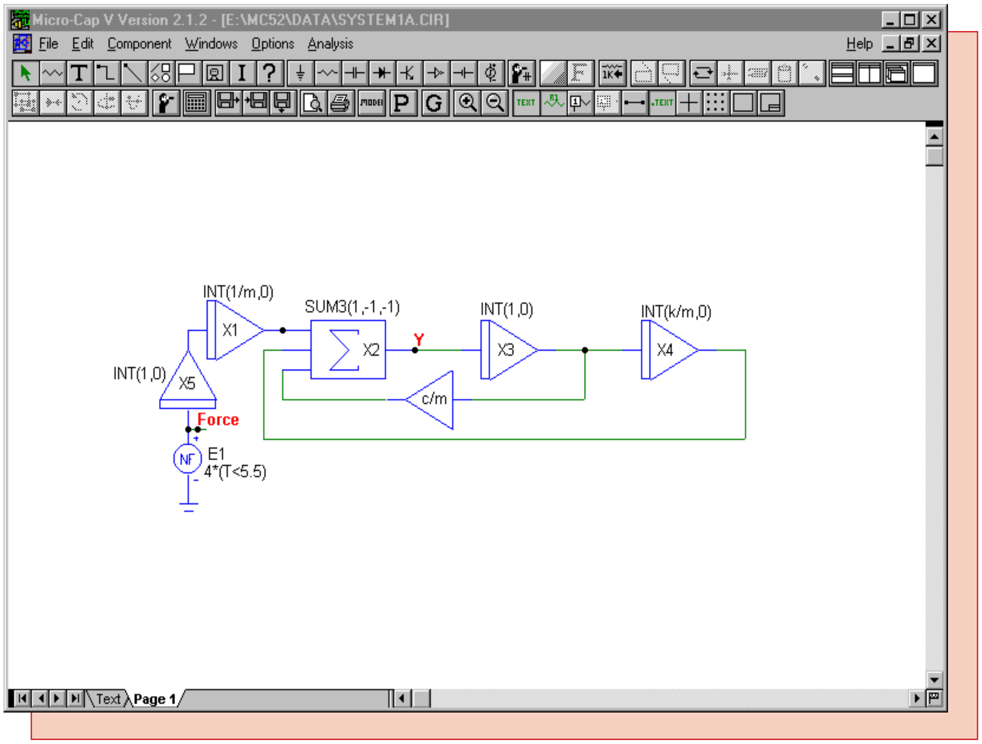


Fig. 8 - Mass on a Spring Equivalent Circuit

The (k/m)*S2(y(t)) product is produced by the X3 and X4 INT macros. The X3 macro integrates the displacement the first time and then the X4 macro integrates the displacement the second time and multiplies it with its k/m coefficient. The output of X4 is then fed into the SUM3 macro which along with the other two products produce the value of the displacement y of mass m at the output of the SUM3 macro. Whether the product is added or subtracted in the SUM3 macro is dependent on the gain parameters passed to the macro. In this case, (1,-1,-1) subtracts the second and third product.

The force applied to the system is produced by the E1 NFV source. This NFV source has its VALUE attribute defined as:

4*(T<5.5)

which will produce a 4V pulse for the first 5.5s of the simulation. This 4V pulse is the equivalent of a 4N force. The analysis results for this circuit appear in Figure 9. For the simulation results, a 20s transient analysis was run. In this case, the system variables m, c, and k were defined as:

m = 4 kg
c = 5 N*s/m
k = 20 N/m

Both the force waveform, V(Force), and the displacement waveform, V(Y), were plotted. The value of zero for the displacement waveform is considered the equilibrium position when the mass is hanging with no external force acting on it. As can be seen in the plot, the displacement hits a maximum of .28 meters and eventually settles back to its equilibrium position once the external force is shut off.

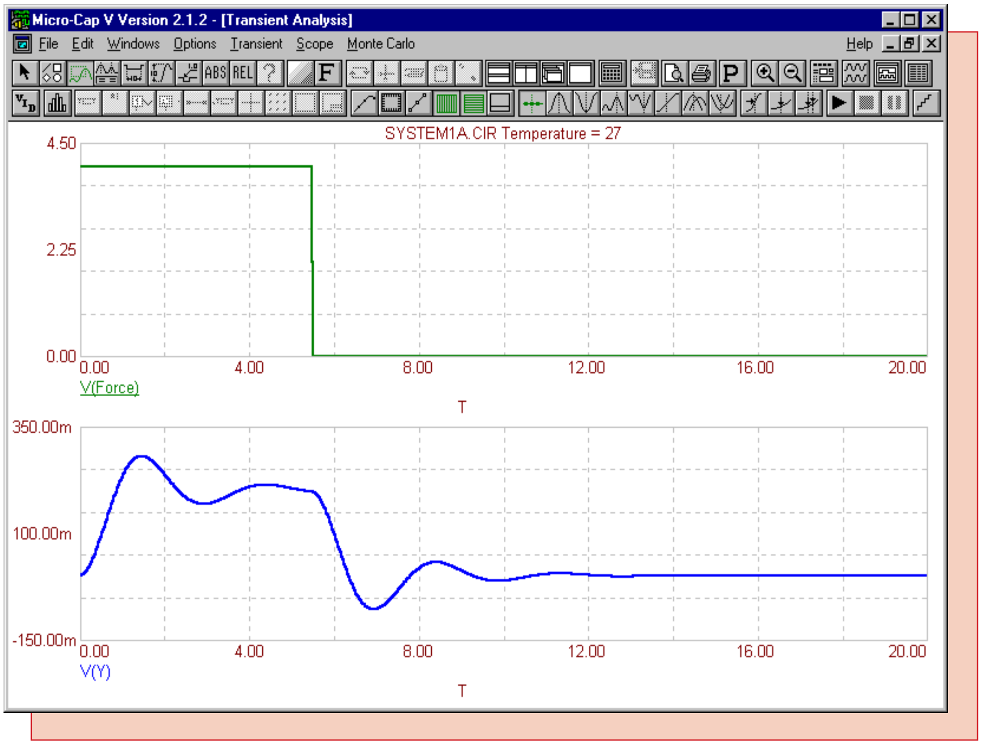


Fig. 9 - Mass on a Spring Analysis Results

Coupled Differential Equations

More sophisticated mechanical systems require the calculation of coupled differential equations. Modelling coupled differential equations uses the same procedure as modelling a single differential equation. The system in Figure 10 consists of two springs, two dashpots, two masses, and one external force.

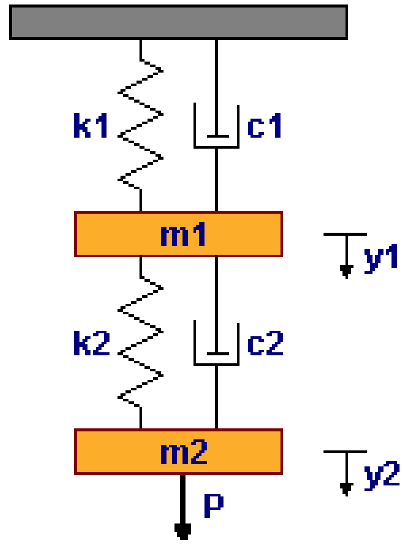


Fig. 10 - Coupled Differential Mechanical System

The mass m1 is acted upon by two springs with spring constants k1 and k2 and two dashpots that provide the damping coefficients c1 and c2. The mass m2 is acted upon by the spring with constant k2 and the dashpot c2 along with the external force P. The differential equations that describe this system are as follows:

$$m1*y1'' + c1*y1' + c2*(y1'-y2') + k1*y1 + k2*(y1-y2) = 0$$
$$m2*y2'' + c2*(y2'-y1') + k2*(y2-y1) = P$$

After integrating the above equations twice and solving for y1 and y2, the new integral equations produced are:

$$y1 = (k2/m1)*S2(y2(t)) - ((k1+k2)/m1)*S2(y1(t)) + (c2/m1)*S(y2(t)) - ((c1+c2)/m1)*S(y1(t))$$
$$y2 = (1/m2)*S2(P(t)) + (k2/m2)*S2(y1(t)) - (k2/m2)*S2(y2(t)) + (c2/m2)*S(y1(t)) - (c2/m2)*S(y2(t))$$

The circuit in Figure 11 implements these equations. The circuit consists of 6 INT macros, 8 AMP macros, an NFV source, and two macros, SUM4 and SUM5, that were created for this circuit. The SUM4 and SUM5 are just extended versions of the SUM3 macro that enable four products and five products, respectively, to be summed together.

The (1/m2)*S2(P(t)) product is produced by the X3 and X1 INT macros with the X1 macro providing the needed coefficient. The X9 INT macro creates the first integral of Y1, and the X10 INT macro creates the second integral of Y1. The X4 INT macro creates the first integral of Y2, and the X5 INT macro creates the second integral of Y2. The eight AMP macros provide the coefficients specified from the differential equations. The Y1 variable is created at the output of the SUM4 macro, and the Y2 variable is created at the output of the SUM5 macro. The gain parameters passed to the sum macros specify whether the product will be added (1) or subtracted (-1).

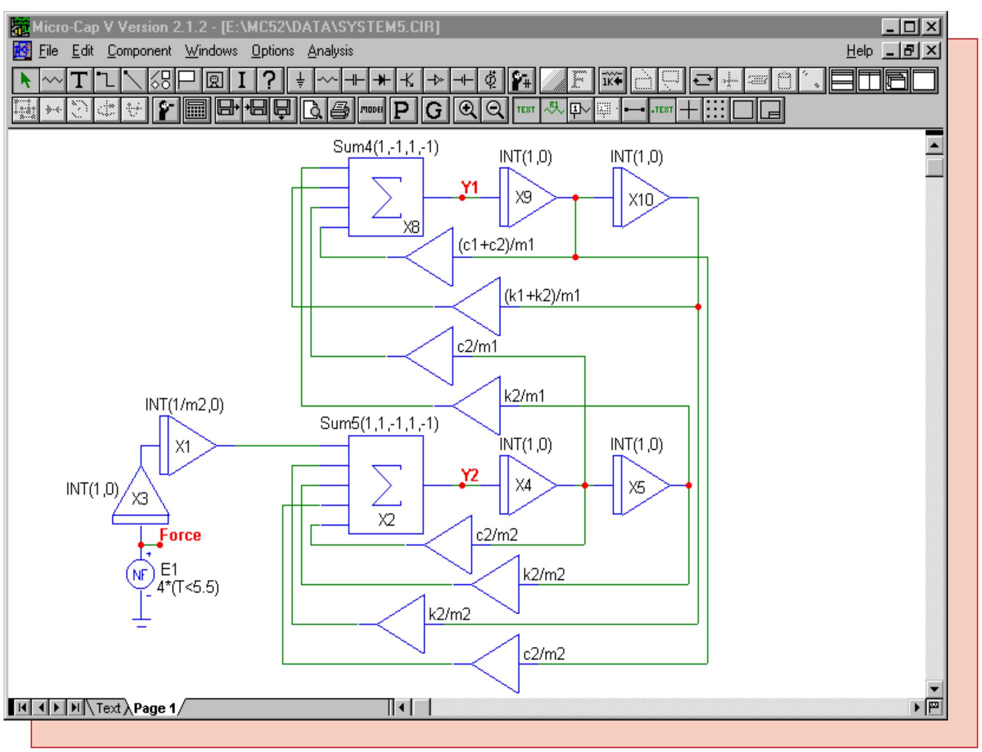


Fig. 11 - Coupled Differential Equations Circuit

Figure 12 displays the transient analysis results for a 20s simulation. Once again, the force for the system was created by an NFV source modelling a 4N force for 5.5s. The system variables for this mechanical system were set at:

m1 = 4 kg
m2 = 6 kg
c1 = 5 N*s/m
c2 = 7 N*s/m
k1 = 20 N/m
k2 = 25 N/m

The three waveforms plotted were the force, V(Force), the displacement of m1, V(Y1), and the displacement of m2, V(Y2). The equilibrium, with no external force P acting on the system for both displacement waveforms, is at zero. With the 4N force and the above variables, the maximum displacement for the mass m1 is .336 meters, and the maximum displacement for the mass m2 is .571 meters. Once the force goes back to zero, both displacement waveforms settle back into their equilibrium state.

The iterative process of SPICE is well adapted to handle simulations such as these. This technique can be used to plot any type of differential equations.

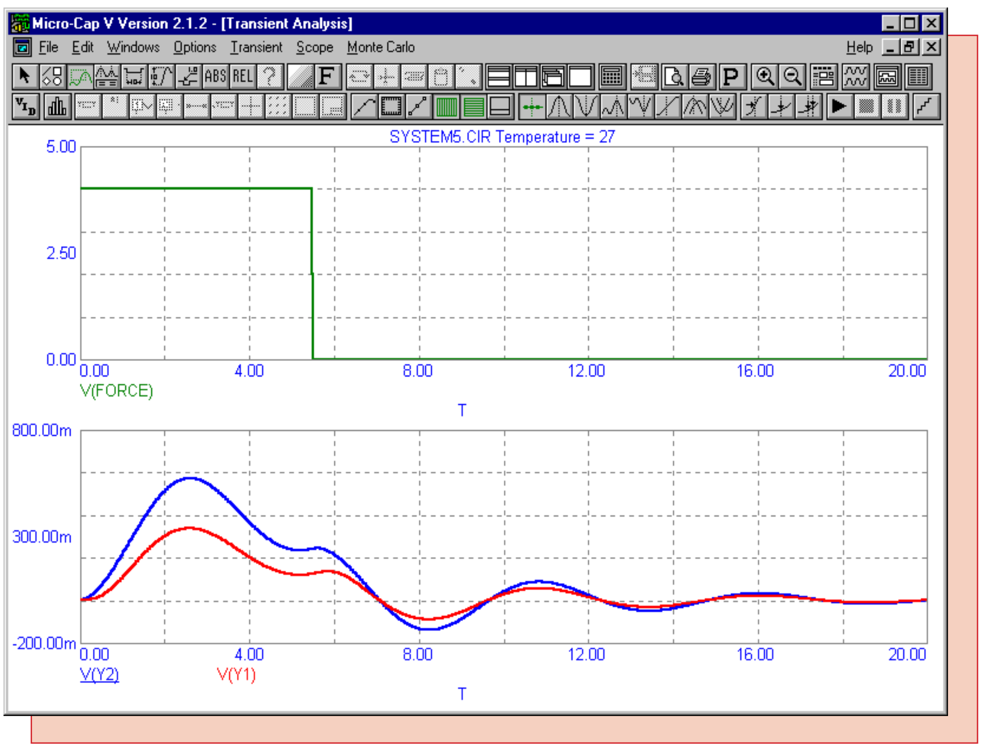


Fig. 12 - Coupled Differential Equations Analysis Results

Thermistor Macro

A thermistor is a thermally sensitive resistor that changes 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 following macro model was derived from a design idea by Lutz Wangenheim titled "SPICE Subcircuit Models Thermistors" in the July 3, 1997 issue of EDN.

The macro circuit for the thermistor appears in Figure 13. The macro was derived from the basic resistance-temperature equation used to describe thermistors which is as follows:

R = Rnom*exp(B1/T - B1/Tnom)

where R is the resistance of the thermistor, Rnom is the nominal resistance, B1 is the material constant, T is the thermistor body temperature, and Tnom is the nominal temperature.

The macro has four parameters: RNOM, B1, D1, and TAU. RNOM defines the nominal resistance at the nominal temperature. B1 defines the material constant. D1 defines the thermistor's dissipation factor. TAU defines the thermal time constant of the thermistor body. The R1 resistor and the E1 NFV source model the resistance of the thermistor. The R1 resistor has its VALUE attribute defined as RNOM and models the nominal resistance. The E1 source takes into account the ambient temperature and the power dependent portion of the temperature to adjust the equivalent resistance of the thermistor. The thermistor's equivalent resistance is equal to:

R = RNOM + V(E1)/I(R1)

replacing R with the first equation and solving for V(E1) returns:

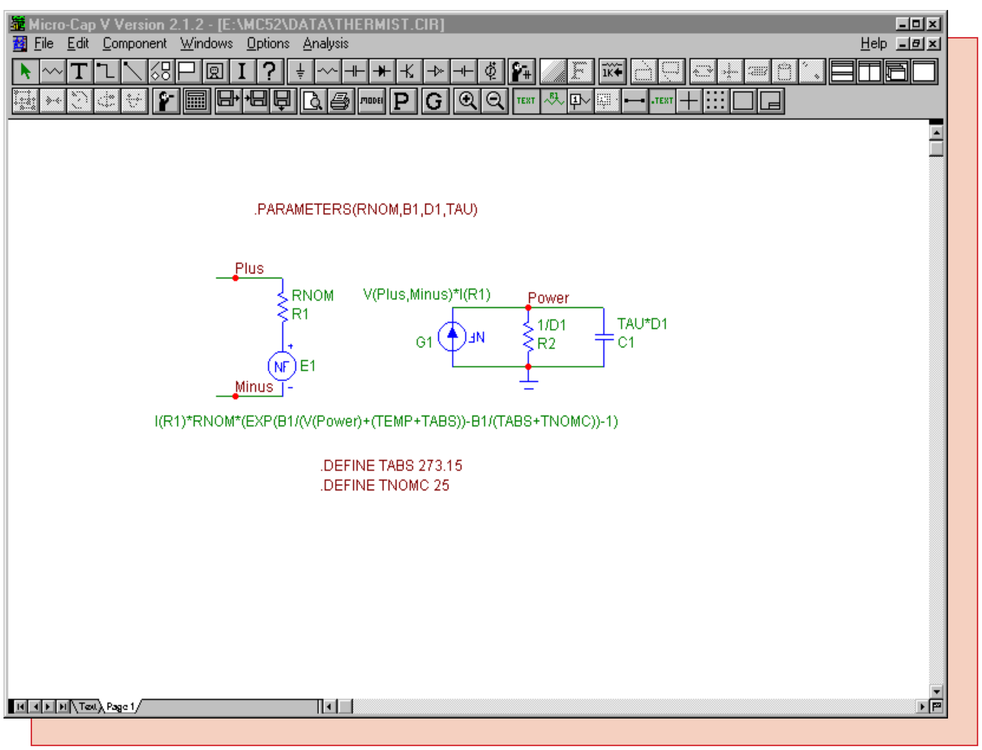


Fig. 13 - Thermistor Macro Model

V(E1) = I(R1)*RNOM*(exp(B1/T - B1/Tnom) - 1)

To model this equation, the E1 source has its VALUE attribute defined as:

I(R1)*RNOM*(EXP(B1/(V(Power)+(TEMP+TABS))-B1/(TABS+TNOMC))-1)

The nominal temperature is represented by the TABS + TNOMC equation. This equation uses the two .define statements in the macro to produce an equivalent Kelvin temperature value from the specified measurement temperature TNOMC in Celsius. To edit the nominal temperature of the thermistor macro, simply edit the .define statement for TNOMC.

The thermistor body temperature is represented by the V(Power)+(TEMP+TABS) equation. V(Power) models the power dependent portion of the temperature due to internal heating. It is generated from the G1 NFI source, the R2 resistor, and the C1 capacitor. The G1 NFI source calculates the power between the thermistor's Plus and Minus pins and produces an equivalent current. The R2 resistor is defined with a value equal to the reciprocal of the thermistor's dissipation factor, and the C1 capacitor models the thermal time constant in conjunction with the R2 resistor. (TEMP+TABS) models the ambient temperature. TEMP is the temperature variable that is defined in the Temperature text field in the Analysis limits dialog box. TABS is defined as 273.15 and converts the specified TEMP variable from Celsius to Kelvin.

The Component Editor settings for the thermistor macro appear in Figure 14. The Name has been defined as Thermist to match the macro circuit file name which was Thermist.cir. The name of the macro must match the macro circuit file name without the extension. The Shape chosen was Thermistor which is an existing shape in the Shape Editor. The Definition chosen was Macro to define this component as a macro. Two pins have been defined for the Thermist macro. These

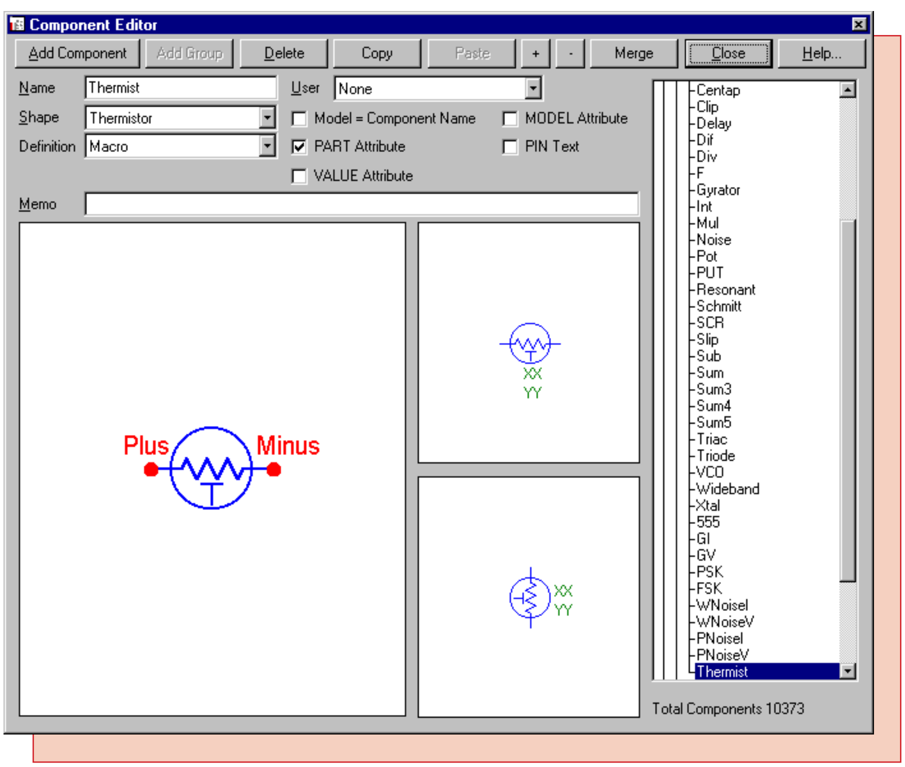


Fig. 14 - Component Editor Setting for the Thermistor Macro

pins are Plus and Minus which match the node names assigned within the macro circuit.

A simple circuit was set up to test the resistance that consisted of a 10V battery, a 1K resistor, and the thermistor macro all in series. The macro was defined with the following VALUE attribute:

Thermist(1k,3k,1m,1)

In this case, a transient analysis was simulated for 1us in which the circuit was stepped from -10C to 90C in 1C increments. The waveform plotted was v(plus)/i(r4) in which Plus was defined as the node at one end of the thermistor with the other end grounded, and R4 is a resistor in series with the thermistor. This waveform is equal to the thermistor resistance. The transient analysis was simulated with the Operating Point off so that the internal heating would not have an effect in the duration simulated. Once these runs were finished, a performance plot was created that plotted the peak of each of these runs versus the temperature that the run was simulated at. This produced the thermistor resistance vs temperature plot that appears in Figure 15. Note that at the nominal temperature of 25C, the resistance is at its nominal value of 1Kohm.

For an AC analysis run, the thermistor's resistance must be constant. Therefore, there can be no ac effects from the internal heating. To avoid this, the TAU parameter must be set high so that the C1 capacitance will not cause the resistance to change during a simulation. This limits the resistance to being a function of RNOM, the ambient temperature, and the DC bias power portion of the temperature. A good rule of thumb is to set the TAU parameter to a value greater than 100/fmin where fmin is the minimum frequency being simulated.

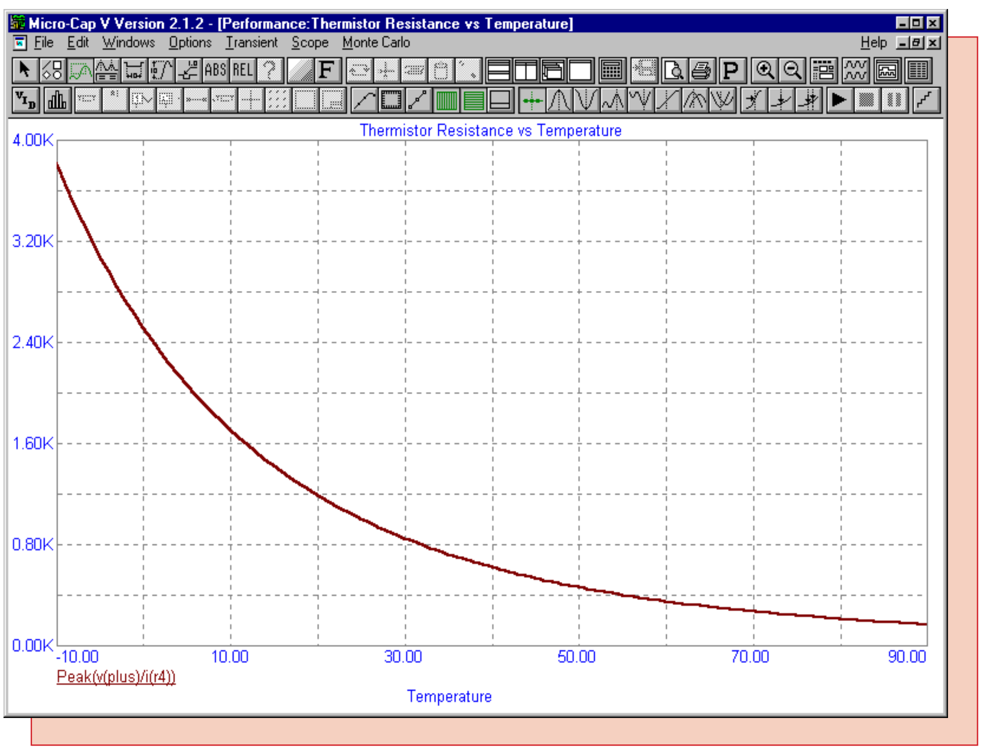


Fig. 15 - Thermistor Resistance vs Temperature

Windows NT and Service Pack 4 Incompatibilities

It has just come to our attention that there is a serious problem between the security key driver we have distributed and Windows NT 4.0 Service Pack 4. The latest security driver for MC5 must be installed before installing Service Pack 4 otherwise on some systems the system will not be able to reboot into NT again. Below is the application note from Alladin about the problem.

HASP Technical Note
Windows NT 4.0 Service Pack 4 and the HASP Device Driver
Date: November, 1998

Summary:
To support the technological upgrade in Service Pack 4 for Windows NT 4.0, Aladdin released HASP Device Driver 3.72. Version 3.72 of the HASP Device Driver solves the following problem:

In many but not all cases if you upgrade to SP4 while an older version of the HASP Device Driver is installed you will receive a blue screen error upon reboot. This Windows NT error message will contain the following text "Kmode Exception Error". Versions of the HASP Device Driver that are affected include version 3.1 through 3.64 inclusive. Any version of the HASP Device Driver that is later than 3.7 will not trigger this behavior.

Prevention:
To prevent this problem we recommend downloading the latest security key driver at:

http://www.spectrum-soft.com/down/hinstall.zip

Unzip this file and then run "hinstall /i" to install the latest driver.

Treatment:
If a customer installed SP4 on top of an installation that already had one of the affected HASP Device Driver versions the following treatment is recommended.

We recommend the following steps:

1. If you have a dual boot machine, enter through the alternative operating system and remove the file "haspnt.sys", located at:

<windows>\system32\drivers\

and then reboot.

2. During startup, when it says "Press spacebar now to invoke..." during the loading of NT, press the spacebar. Then chooses the option of restoring the "Last Known Good Configuration" and start NT. Note that this option will work best if the user followed instructions laid out in the Microsoft readme for SP4:

"...it's recommended that you do the following before installing the Service Pack:

1. Update the system Emergency Repair Disk using the Rdisk.exe command with the /s switch.
2. Perform a full backup of the system, including the system registry files.
3. Disable any nonessential third-party drivers and services (that is, drivers and services that aren't required to boot the system).

4. Contact the original equipment manufacturer (OEM) that provided the driver or service for the updated versions of the file(s).
5. Restart the computer and check Event Viewer to ensure there are no system problems that could interfere with the installation of SP4.

NTFS Cases
NTFS poses special problems. If you want to access a NTFS partition from DOS you need special tools. There is a product called NTFSDOS that may allow you to fix this.
Information on this product can be found at

http://www.sysinternals.com/ntfs20.htm

This product is not in any way related to Aladdin Knowledge Systems and we cannot guarantee its functionality but it may provide an alternative to a complete reinstall.

Product Sheet

Latest Version numbers

Micro-Cap V	Version 2.1.2
Micro-Cap IV IBM/NEC/MAC	Version 3.04

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

Spectrum's Products

• Micro-Cap V	$3495.00
• Micro-Cap V LAN (single seat)	$3595.00
• Upgrade from MC5 Ver 2 to MC5 Ver 2 LAN	$100.00
• Upgrade from MC5 Ver 1 to MC5 Ver 2	$500.00
• Upgrade from MC5 Ver 1 to MC5 Ver 2 LAN	$600.00
• Upgrade from MC4 to MC5	$1000.00
• Upgrade from MC4 to MC5 LAN	$1100.00

You may order by phone or mail using VISA, MASTERCARD, or American Express. Purchase orders accepted from recognized companies in the U.S. and Canada. California residents please add sales tax.