spectrum news

Applications for Micro-Cap™ Users

Spring 2004

News

MICRO-CAP 8

Featuring:

- Introducing Micro-Cap 8
- Repeater Macro
- Three Phase Triangle Source Macro

News In Preview

This newsletter's Q and A section describes how to define a temperature coefficient for a resistor. The Easily Overlooked Features section describes the use of the TMAX, TMIN, FMAX, and FMIN variables.

The first article introduces the latest generation of Micro-Cap simulation software - Micro-Cap 8.0.

The second article describes the creation of a repeater macro which converts a one shot waveform into a periodic waveform.

The third article describes the creation of a three phase triangle source macro.

Contents

Book Recommendations

General SPICE

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

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

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

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

MOSFET Modeling

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

VLSI Design

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

Micro-Cap - German

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

Micro-Cap - Finnish

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

Design

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

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

High Power Electronics

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

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

Switched-Mode Power Supply Simulation

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

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

Micro-Cap Questions and Answers

Question: How do I set up a temperature coefficient for a resistor? I would like to define a resistor that has a temperature coefficient of 1500ppm/degree C.

Answer: There are two methods that can be used to define temperature coefficients for the resistor. The first method is through the VALUE attribute of the resistor. The VALUE attribute has the format:

<value> [TC=<tc1>[,<tc2>]]

where the temperature coefficients are an optional entry. When these coefficients are specified, they multiply <value> by the following factor:

1 + <tc1>*(T-Tnom) + <tc2>*(T-Tnom)^2

As can be seen above, the <tc1> is a linear temperature coefficient and the <tc2> is a quadratic temperature coefficient. An example for the VALUE attribute would be:

1K TC=1.5E-3

which would produce a 1500ppm/degree C coefficient for a 1K ohm resistor.

The second method to declare a temperature coefficient is through the resistor model statement's TC1 and TC2 parameters. When in the Attribute dialog box, define the MODEL attribute of the resistor with a model name such as RESTEMP or RMOD. Any valid name will work. When the MODEL attribute has been defined, the model parameters at the bottom of the Attribute dialog box become available for editing. Among these parameters are TC1 and TC2. These act in the same manner as the <tc1> and <tc2> parameters in the VALUE attribute. Defining the TC1 parameter with the value 1.5E-3 will create the 1500ppm/degree C coefficient also. Any coefficients defined within the VALUE attribute will take precedence over model parameters.

The capacitor and inductor each have TC1 and TC2 model parameters available in order to create temperature coefficients when the MODEL attribute has been defined. These parameters operate with the same behavior as the resistor's temperature parameters. The VALUE attribute temperature coefficients are not available for these components.

The resistor model has one extra temperature coefficient parameter that the capacitor and inductor models do not have. This parameter is TCE and defines the exponential temperature coefficient in units of %/degree C as:

1.01 ^ (TCE*(T-Tnom))

In general, the TC1 parameter will be the value specified in databooks.

Easily Overlooked Features

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

TMAX, TMIN, FMAX, FMIN Analysis Variables

There are four variables available in Micro-Cap that reference the range of the simulation for either transient or AC analysis. These variables are defined as follows:

TMIN - Starting transient analysis time
TMAX - Ending transient analysis time
FMIN - Starting AC analysis frequency
FMAX - Ending AC analysis frequency

The main use for these variables is to reference them in the X Range expression field in the Analysis Limits dialog box. For example, when the X Range has been defined as:

TMAX, TMIN

for a transient analysis, the X Range will constantly be scaled to the values in the Time Range field. If the Time Range field has been set to 10m, when the simulation is run, TMAX is set to 10m and TMIN is set with the default value of 0. The advantage of this method is that the X Range will automatically update to any changes made in the Time Range field, so there isn't a need to have to manually edit the X Range fields each instance the simulation time is changed. FMAX and FMIN operate in the same manner in AC analysis by deriving their values from the Frequency Range field.

These variables are easily placed in an X Range field for either transient or AC analysis by right clicking in the X Range field when in the Analysis Limits dialog box. A left click on the X Range header will let you place them in for all available waveforms.

These variables are also available for use in the schematic. An NFV function source can have its VALUE attribute defined as:

5*sin(2*PI*(1/TMAX)*t)

This equation will produce a sine wave with an amplitude of 5 whose period will always be equivalent to the maximum time value specified in the Time Range field of the Transient Analysis Limits dialog box.

Introducing Micro-Cap 8

We are pleased to release the next generation of the Micro-Cap simulator, Micro-Cap 8.0. A number of new features, components, and models have been added to enhance the simulation power and the ease of use of the interface. A preview of some of the new features follows. Please contact our sales department for upgrade or pricing questions.

New MOSFET Models - BSIM4 and EKV

Two new MOSFET models have been incorporated into MC8. The first is the BSIM4 device which is a level 14 MOSFET. This model was created at Berkeley and is designed to address MOSFET physical effects into the sub-100nm realm. The second model is the EKV MOSFET model (level 44) from the Swiss Institute of Technology. It features a dedicated charge-based short channel dynamic model using only 18 intrinsic core parameters.

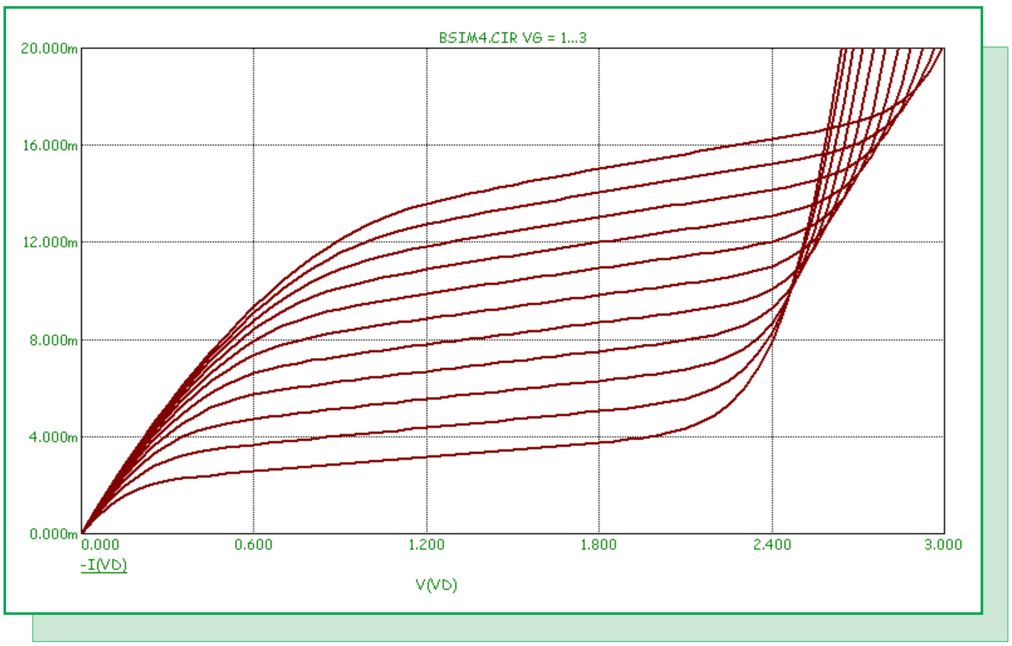

Fig. 1 - BSIM4 MOSFET Model I-V Curves

HSPICE Style Binning

Binning is the process of adjusting model parameters for different values of drawn channel length (L) and width (W). BSIM3 and BSIM4 MOSFET devices were upgraded to include the HSPICE binning option, as well as the standard Berkeley binning option.

Short Distance VT Matching for BSIM3, BSIM4, and EKV Models

The BSIM3, BSIM4, and EKV models were upgraded to include a short distance VT matching parameter, to better model short distance threshold variations.

IBIS Modeling Tool

IBIS is an acronym for Input Output Buffer Information Specification. It is a method of describing device characteristics at the input/output level without having to describe the possibly proprietary circuitry that produce the characteristics. It can be thought of as a kind of behavioral modeling specification suitable for transmission line simulation of digital systems and applicable to most digital components. New IBIS modeling tools were added to translate IBIS files to standard Micro-Cap / SPICE input and output pin models. These models accurately reproduce the IBIS golden waveforms. The IBIS editor creates both standard library SPICE models and verification models for testing and verifying golden waveforms.

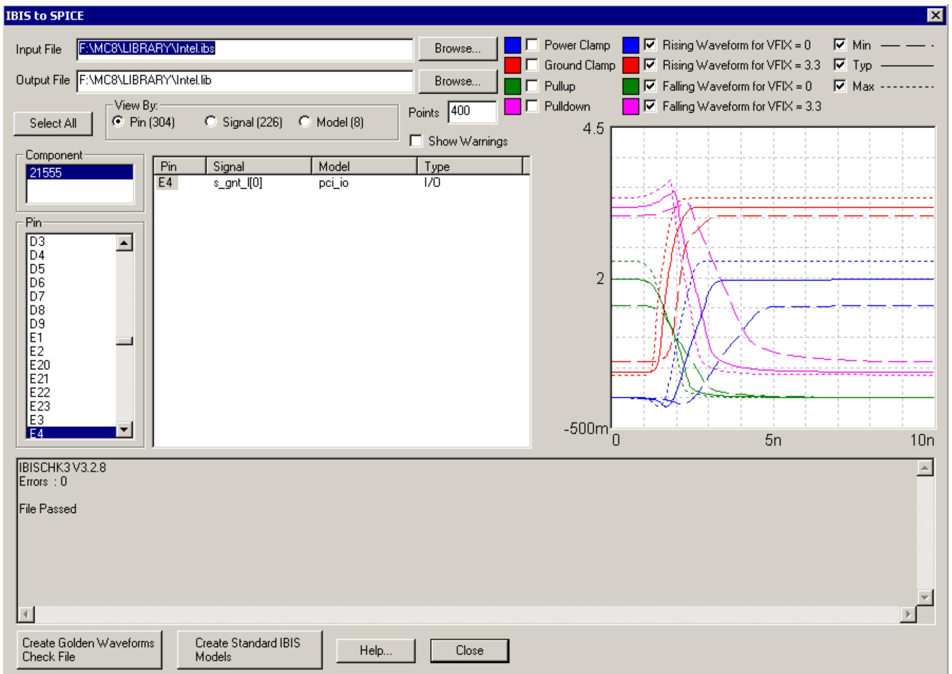

Fig. 2 - IBIS Translator

SPICE Voltage and Current Source Enhancements

New time-domain Noise and Gaussian waveforms and user definable preset tabs for common waveform settings were added to make complex waveform creation easy.

Timer Device

A flexible new timer device was added to count events, and to measure both elapsed and last event time. Attributes set the initial, minimum, and maximum count along with the increment value. A reset pin is also available on the device.

Enhanced S and W Switches

A hysteresis option was added to the S (Voltage) and W (Current) switches.

Improved Magnetics Model

A new, more robust Jiles-Atherton magnetics model was added. Both MODEL and the mainline simulator can plot BH values in both CGS and MKS (SI) units. The convergence routines for gapped cores has been vastly improved.

N-Port Device Using Touchstone Data Format
The N-Port model in MC8 provides a way to easily include S, Y, Z, G, or H parameter data from RF suppliers in a simulation. The N-Port model imports in a Touchstone data file containing tables of data that define the specified parameter values versus frequency. While the 2-Port is the most common use of a Touchstone file, any number of ports may be imported in Micro-Cap. These text files are available from many companies such as Siemens and Infineon. With the capability to produce Smith charts and polar plots in AC analysis, creating plots of data such as S parameters is easier and more powerful than ever. Micro-Cap also contains a translator that provides the capability to convert Touchstone files from one parameter format to another.

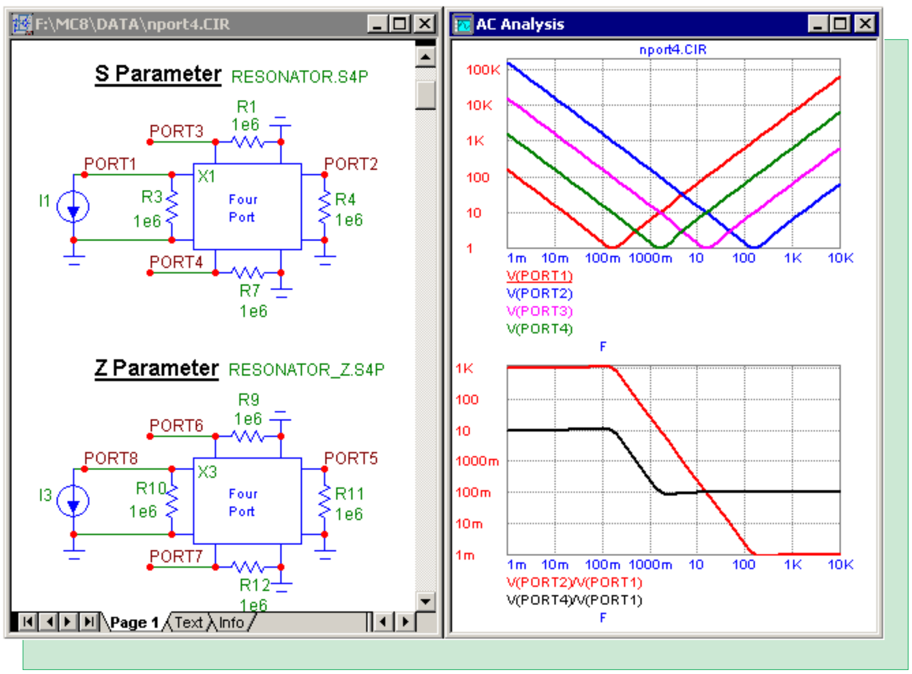

Fig. 3 - N-Port Device

Subcircuit Maker
A new Make SUBCKT command translates a schematic to a SPICE subcircuit. The subcircuit will automatically be added to the Component library of Micro-Cap for use in other schematics.

Improved Bill of Materials Report
A new Bill of Materials report features sorting on any field and a live report that responds to user format changes immediately.

New Graphic Formats
TIFF, PNG, JPG, and GIF graphics formats were added to the standard BMP, WMF, and EMF formats for saving schematic and plot images to a file. These new formats may also be used when placing graphic files into a schematic.

Component Menu Shape Display
The Component menu now shows the component shape as you browse as a preview of what the component will look like in the schematic.

Animation Components

Many new animation components have been added to MC8, and all of the animation components have been enhanced to work with the Dynamic DC analysis. The new components are:

SPST, DPDT, DPST analog switches for dynamically configuring a circuit.
Analog bar with height proportional to the input DC voltage.
Analog LED with user-defined color and on voltage.
Rotating DC motor with DC voltage dependent RPM.
Analog / digital DC voltage/current meter.
Relay with programmable resistance, inductance, and on and off currents.
Three color traffic light.

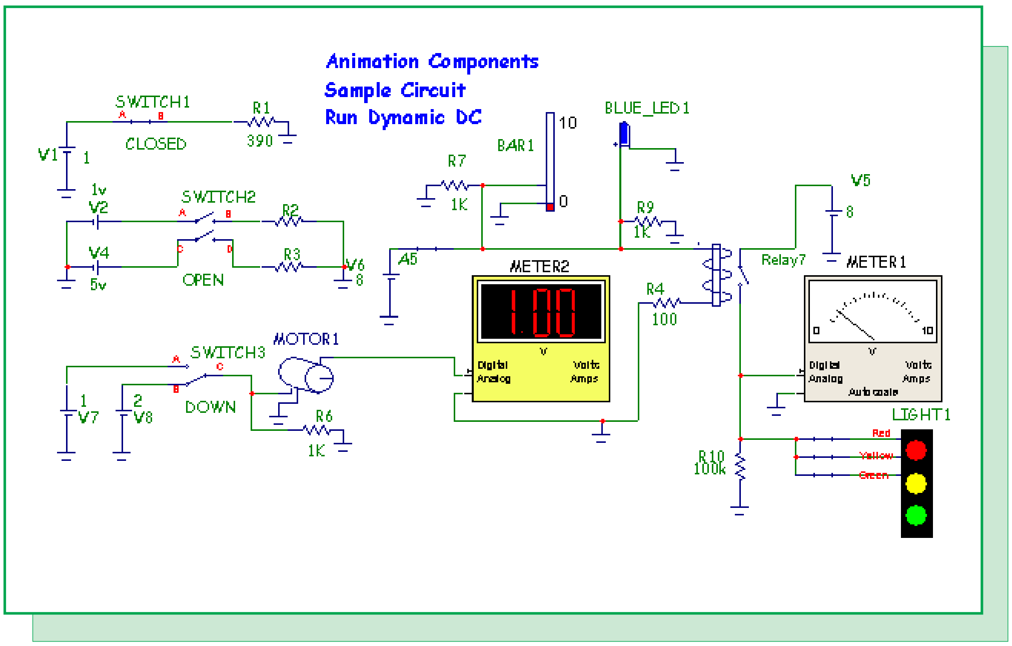

Fig. 4 - New Animation Components

Password Protection

A new protection feature added password protection to circuit and macro files. Another user may simulate a circuit with a macro that has been password protected, but they will be unable to view the inner workings of the macro.

Find in Files Command

A new Find in Files command was added to search circuit or other files for text, components, or other content. This is particularly useful if you remember a specific component or text string used in a circuit, but are unable to recall the actual circuit name.

Info Page

An Info text page was added to show the location of models, macros, subcircuits, or variables used in the circuit, resolving uncertainty over the source of the device model or variable.

Cleanup Command
A new Cleanup command for deleting miscellaneous extraneous files was added. This feature provides an easy way to delete files such as numeric output or probe files that may not be needed anymore from the hard drive.

Local Path Commands
New local .PATH commands let a circuit specify DATA and LIBRARY paths, overriding the global paths. This feature lets you store the path information locally within a circuit.

New Dynamic AC Analysis
A new analysis mode was added that features dynamic on-schematic display of AC voltage, current, and power as frequency is stepped, with the display dynamically responding to schematic edits.

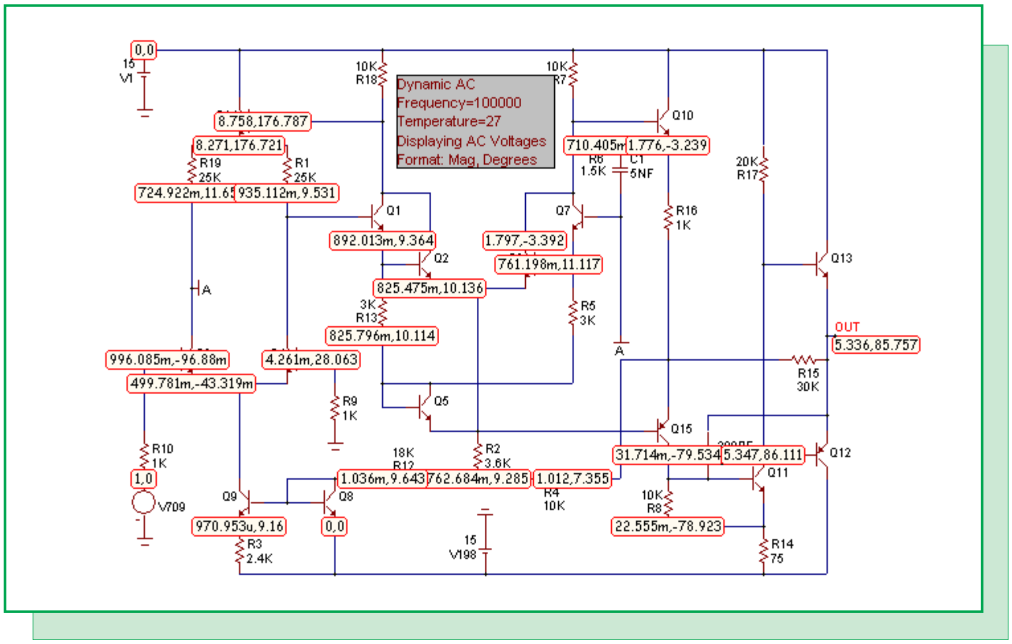

Fig. 5 - Dynamic AC Analysis

Enhanced Dynamic DC Analysis
Stepping, optimization, and an Analysis Limits dialog box were added to Dynamic DC.

Apply Display Properties Command
A new command was added to copy a selected part's display properties (part color, attribute font, size, color, and print location) to other similarly named and rotated parts.

Change Attribute Command
A new Change Attribute command was added to add, delete, edit, show, or hide attributes of any subset of the parts in a schematic, making mass changes easy.

New Color Options
Parts and wires can now have individual colors in a schematic.

New Attribute Dialog Box Plot
Diodes acquired a new Ir vs. Vr plot to show reverse characteristics within the Attribute dialog box.

New Distortion Analysis
A new analysis mode was added that employs the DSP routines to calculate and plot measures of distortion, including raw harmonics, as well as total and individual harmonic distortion. Distortion analysis is a type of transient analysis that applies a single frequency sinusoidal signal to the named input source and measures the resulting distortion in the specified output expression using the IHD (Individual Harmonic Distortion) function.

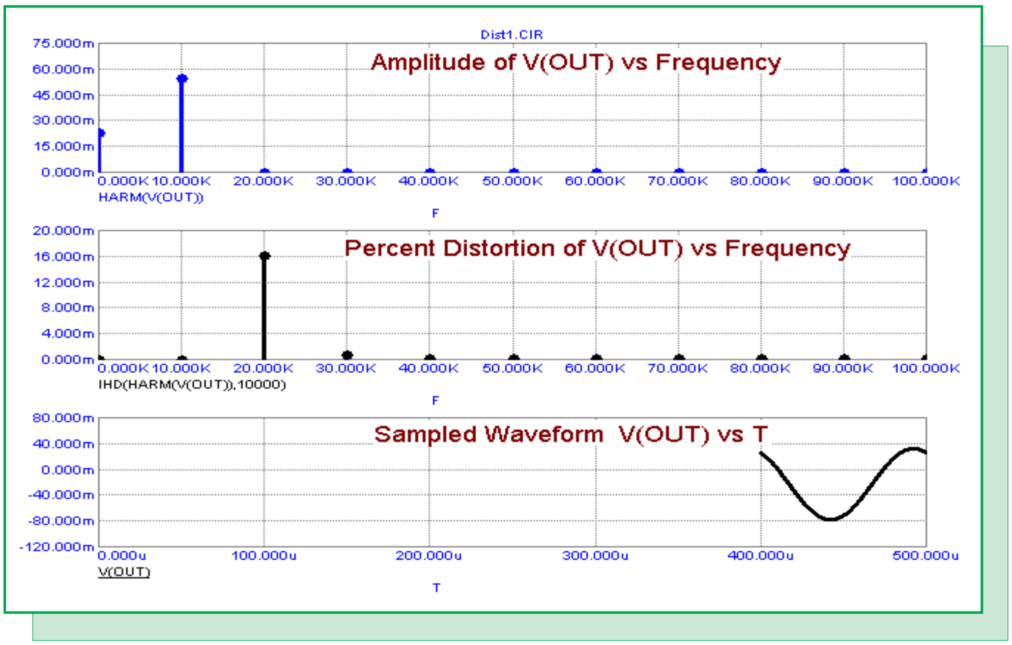

Fig. 6 - Distortion Analysis

Enhanced Fourier Analysis
A new FFT window was added to automatically convert transient plots to Fourier plots. This conversion is performed after the simulation is finished running so there is no need to specify the FFT function before the run to view harmonic information. A new FS(expr,n1,n2) function was added that creates a waveform from the n1 to n2 terms of the Fourier series of (expr). A new RES(expr,n1,n2) function was added to create a residual waveform by subtracting the n1 to n2 terms of the original waveform. FFT auto-scale commands were upgraded to allow ignoring the DC component and auto scaling over only a specified number of harmonics.

Analysis Limits Dialog Box
The dialog box size and X and Y fields are user-adjustable.

Optimizer Enhancements
The new optimizer supports dynamic plots to show optimization progress and works with both Dynamic DC and Dynamic AC.

Analysis Plot Improvements
Scale factors (Meg, K,...p, f) for X/Y axes. e.g. "Time (nS)"
Units (Amps, Volts, Watts,...) for plots. e.g. "V(26) (Volts)"
Open and closed circles and squares data point markers
Normal and popsicle data point styles
X, Y, or both auto scaling options
New option to force all X scales to be the same
Retrace mode for rolling displays
Right clicking on expressions to invoke FFT, performance, and 3D plots
New minor log grid option to show and label 1-2-5-10 sequence

New Expressions and Functions
Integration and differentiation operators for use in function sources
Array variables for indexed selection of variable values from a predefined array of values
Transmission line power and energy terms
AC power and energy terms
New random functions, RNDR, RNDC, and RNDI(interval)
New DELAY(x,d) function returns expression x delayed by d seconds
S and W switch resistance, voltage, current, power, and energy terms
SPICE3 analog Boolean operators, &, |, ~, and ^ with analog parameters
LAST(expr,n) function which returns a value from n data points before
MAXR and MINR function to track an expression's highest and lowest value during a run
NORM, NORMMAX and NORMMIN functions to normalize at specified, maximum, and minimum points
International engineering notation (e.g. 3R3 = 3.3)
Global settings variables
Model parameter use of variables like TEMP that are constant during a run
Nested series expressions
Product series function (e.g. PROD(I,1,3,I) = 1*2*3)

MODEL Import
MODEL provides one step exporting of its parts directly to the Component library.

Numeric Formats
A new decimal numeric format uses only numbers, commas, and decimal points. 1.234e6 can be formatted as 1,234,000.00. Support added for international engineering notation (e.g. 3R3 = 3.3).

Filter Designer Improvements
Step and Impulse response plots were added.

Monte Carlo Improvements
A new feature lets users reproduce statistical distributions by specifying the random number seed. New features included user-specified histogram range and intervals.

Expanded LAN Operation
Several features were added to better support LAN installations. The model library index is now written locally, avoiding the need to write to possibly write-protected LAN disks. Component, Shape, and Package library files could also be located on write-protected LAN disks.

Miscellaneous

New .WARNING message, condition command to let users create custom warnings such as .WARNING “Capacitor voltage exceeded.” V(C1) > 50
SPACEBAR toggle between voltage and current in Probe.
DEFINE statement for use in SPICE files.
All On and All Off buttons to enable/disable all stepped variables.
Rename command to provide left to right and top to bottom options.
Translator to convert file from S, Y, Z, G, or H format to any other format.
New numeric output options let the user decide what to include.

Repeater Macro

The repeater macro provides the capability to take a one shot waveform and give it a periodic characteristic. For some waveform sources, such as the User source, creating a periodic waveform requires a lot of data to be generated in order to produce the waveform over the time range that is being simulated. Rather than creating multiple cycles in the User source file, only one cycle needs to be created when the source is used in conjuction with the repeater macro. The repeater macro is shown in Figure 7.

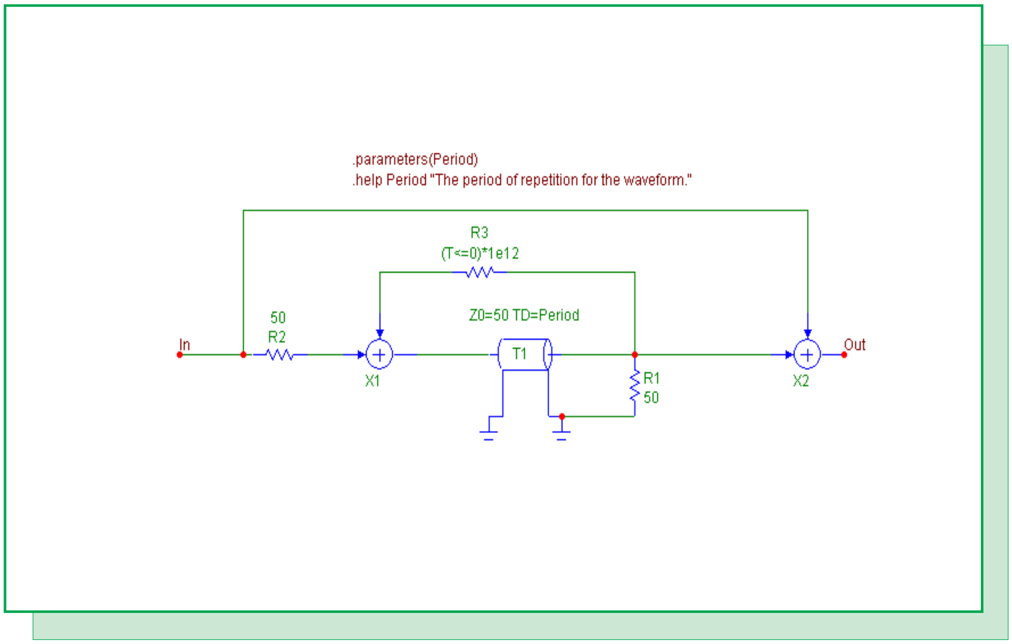

Fig. 7 - Repeater Macro

The repeater macro takes in one parameter: Period. This defines the period of the input waveform that is to be repeated. The repeater macro consists of three resistors, two Sum macros, and a lossless transmission line. The R1 and R2 resistors along with the T1 transmission line produce a delay element whose delay is defined by the parameter Period. The input signal is fed into the two Sum macros. The original waveform will be passed directly to the Sum macro at the output and produce the expected waveform during the first Period seconds. The input is also passed into the delay element through the Sum macro at the input. After Period seconds have passed, the original waveform now starts to appear at the output of the transmission line and is reproduced at the output of the macro. This waveform is also fed back into the input of the delay element so that the cycle will repeat continuously. The R3 resistor in the feedback loop has had its VALUE attribute defined as:

(T<=0)*1e12

which will produce a 1E12 resistor when the simulation time is zero or negative, and a RMIN (set in the Global Settings) valued resistor when the simulation time is greater than 0. The high resistance at time=0 helps prevent a positive feedback loop from occurring during an operating point calculation.

In order for the repeater macro to work correctly, the input waveform must be at zero once the time of the original period has passed. Any nonzero voltages after the original period has expired will be passed to the output and be summed with the original waveform. If a DC voltage is passed through, a ladder effect will occur where each subsequent period is greater than the previous by the value of the DC voltage. Also, this macro does not work in DC so a DC or Dynamic DC analysis will not converge when this macro is used in a circuit. For transient analysis, the operating point should generally be disabled.

A simple circuit was set up to test the repeater macro. It consisted of a User source connected to the input of the repeater macro. The User source imports in the SAMPLE.USR file that is distributed with Micro-Cap. The SAMPLE.USR file has 1us worth of data stored in the file and simulates a few cycles of a damped sine wave. The repeater macro has its Period parameter set to 1u. The resulting transient analysis for a 5us simulation is displayed in Figure 8.

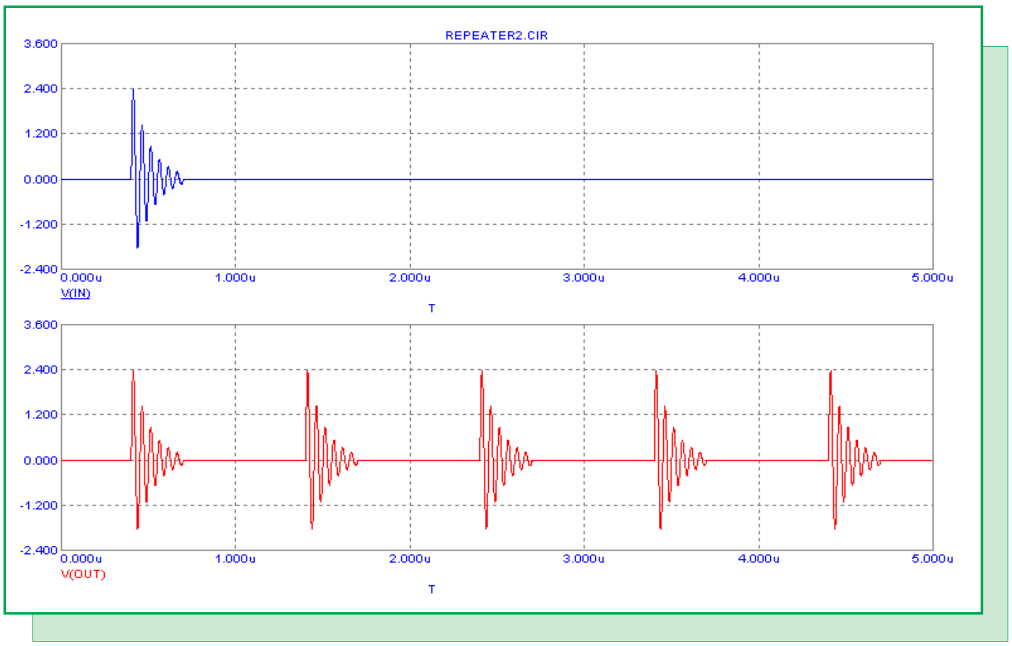

Fig. 8 - Repeater Macro Analysis

The top waveform is the waveform directly from the User source. It simulates the 1us of data within the file and then stays at zero volts for the rest of the simulation. The bottom waveform is the output of the repeater macro. Note that it has taken the 1us of data from the User source and has turned it into a periodic waveform with a period of 1us.

Three Phase Triangle Source Macro

Many power system simulations need a three phase voltage input source available. For a three phase sine wave, it is a simple task to place three sine sources in a schematic and set their phases at 0, 120, and 240 degrees. For other input waveforms, such as a three phase triangle wave, it is a little more difficult since there is not a phase parameter that is easily set. Figure 9 displays a three phase triangle source macro created in a standard Y configuration.

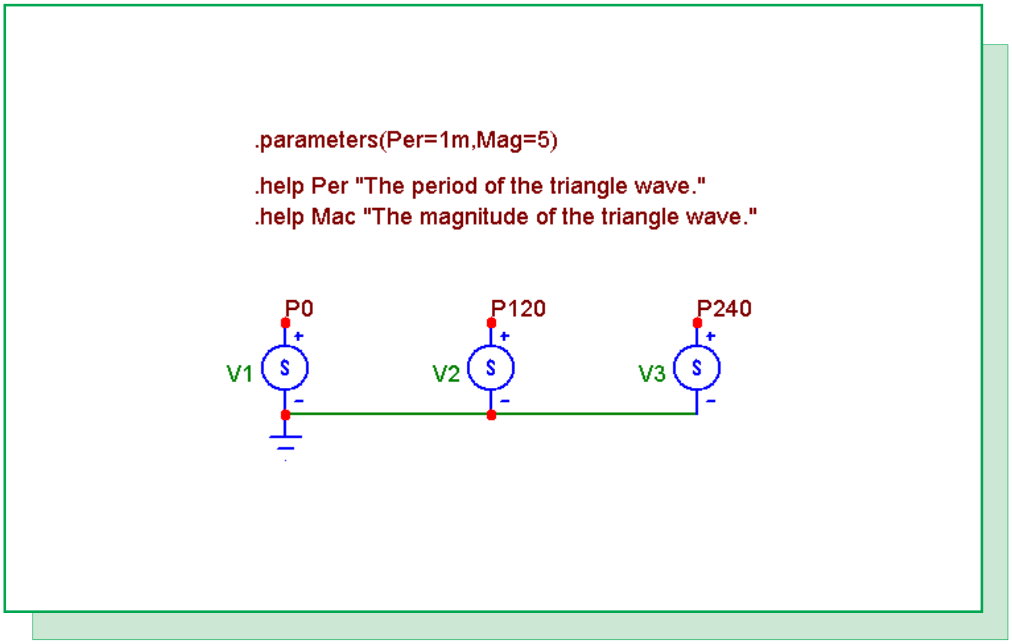

Fig. 9 - Three Phase Triangle Source Macro

The macro has two parameters available for passing: Per and Mag. Per defines the period of a single triangle wave, and Mag defines the magnitude of the triangle wave. The macro circuit consists of three independent SPICE voltage sources. These sources are located under the Analog Primitives/Waveform Sources section of the Component menu with the name 'V'. The sources have their VALUE attributes defined using the PWL, piece-wise linear, capability of the source as follows:

V1 (0 degrees):
PWL TIME_SCALE_FACTOR=per VALUE_SCALE_FACTOR=mag REPEAT FOREVER
0,0 .5,1 1,0 ENDREPEAT

V2 (120 degrees):
PWL TIME_SCALE_FACTOR=per VALUE_SCALE_FACTOR=mag REPEAT FOREVER
0,.6667 .3333,0 .8333,1 1,.6667 ENDREPEAT

V3 (240 degrees)
PWL TIME_SCALE_FACTOR=per VALUE_SCALE_FACTOR=mag REPEAT FOREVER
0,.6667 .1667,1 .6667,0 1,.6667 ENDREPEAT

Each of the sources have their time and voltage pairs defined in relation to one cycle of the triangle wave. For example, the V1 source has its time/voltage pairs defined as:

0,0 .5,1 1,0

which will produce a voltage of 0 at time=0, a voltage of 1 at time=.5 and a voltage of 0 at time=1. The V2 and V3 sources have their time and voltage data adjusted to reflect the phase shift that each one of them needs to produce in a three phase system. The REPEAT FOREVER and ENDREPEAT keywords will repeat the data for the entire simulation time so that the waveform is now periodic.

The TIME_SCALE_FACTOR uses the Per parameter to scale all of the defined time values. The VALUE_SCALE_FACTOR uses the Mag parameter to scale all of the defined voltage values. These two scale factors give the macro great flexibility in that the period or magnitude of the waveforms can be easily scaled with the factor rather than having to edit each and every data point within the PWL data. This is also the reason why the time and voltage in the PWL definitions were based on a scale of 1 so that the factors can scale to the appropriate period or magnitude using the parameter values passed into the macro.

The results of a 5ms transient analysis with the three phase triangle source is shown in Figure 10. In this case, the Per parameter was defined as 1m, and the Mag parameter was defined as 5. Note that the period of the triangle waves is 1ms with a magnitude of 5V just as the parameters specified, and that the three sources have produced a nice 120 degree phase shift between each other.

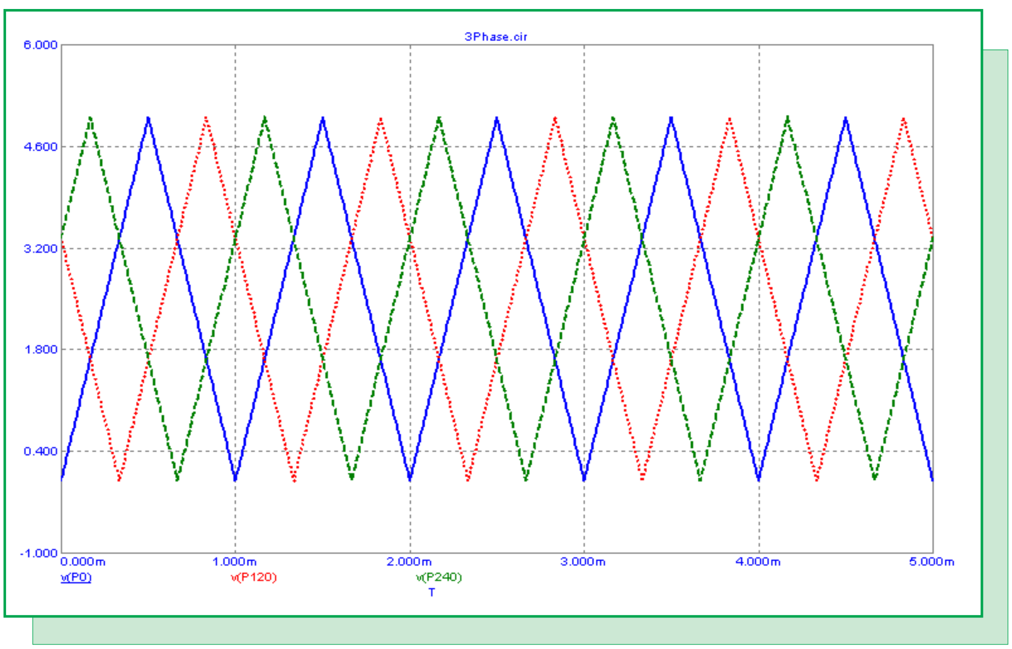

Fig. 10 - Three Phase Triangle Source Analysis

Product Sheet

Latest Version numbers

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

Spectrum's numbers

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