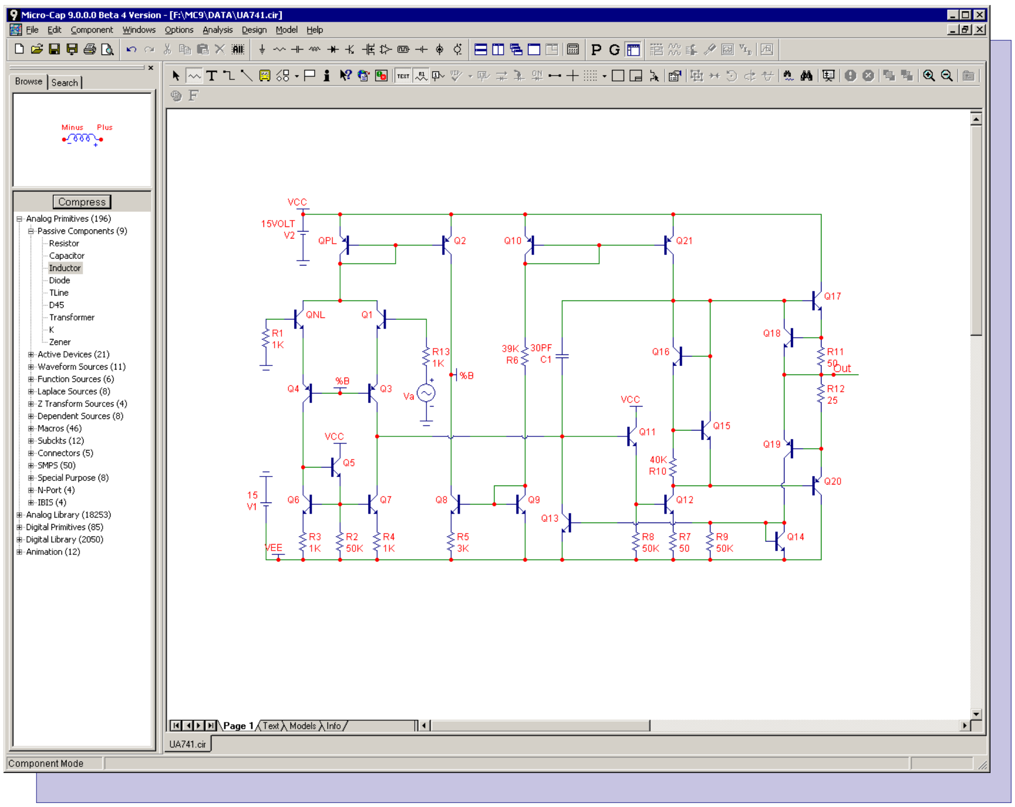
spectrum news

Applications for Micro-Cap™ Users

Winter 2007

News



Introducing Micro-Cap 9

Featuring:
- Introducing Micro-Cap 9
- Repetitive Time Switch Macro
- Measuring Insertion Loss

News In Preview

This newsletter's Q and A section describes how to assign default model statements to the resistor, capacitor, and inductor components. The Easily Overlooked Feature section describes the Protect command which can save a schematic or a macro file so that a password is necessary to view the contents of the file.

The first article gives a preview to the latest generation of the Micro-Cap simulator, Micro-Cap 9. This article highlights many of the new features available within the program.

The second article describes a macro model that simulates a repetitive time switch.

The third article describes a procedure for measuring the insertion loss for a filter.

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 - Czech

• *Resime Elektronicke Obvody,* Dalibor Biolek, 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# ISBN 951-0-25672-2

Design

• *Microelectronic Circuits High 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: Is there any way to assign a default model statement to the basic resistor component? When I place a resistor in the schematic, the MODEL attribute is blank, but I would like to have a default model assigned to that attribute so that I can define a nominal tolerance for the component. This would help me out for Monte Carlo simulations since all of my resistors would automatically have a standard tolerance defined for them.

Answer: There is a way to define a default model for the resistor, capacitor, and inductor components. The first step is to create the appropriate model statement. The model should be created in a new library file or one that the user has already created. In this case, let's call the file Default.lib and store it in the Library folder. In the Default.lib file, enter a model statement such as:

.Model Resistor Res (R=1 Lot=10% TC1=3e-3)

This will create a resistor model that assigns a 10% tolerance and a 3000ppm/degree C temperature coefficient to any resistor that references it. Any of the resistor parameters can be defined in this model. Capacitor and inductor models can also be created such as:

.Model Capacitor Cap (C=1 Lot=25%)
.Model Inductor Ind (L=1 Lot=25%)

Note that the names for these models are Resistor, Capacitor, and Inductor which match exactly with the names of the components in the Micro-Cap component library. If these models are added to a new library, the next step is to add a reference to this library into the Nom.lib file that comes with Micro-Cap. Open up the Nom.lib file from the Library folder under the main MC8 folder. In this file, add in a reference to the new library such as:

.lib "Default.lib"

Save and close the Nom.lib file. Any model statements in the Default.lib file will now be globally available within Micro-Cap. Under the Windows menu, select Component Editor. In the tree on the right, navigate to the Analog Primitives/Passive Components group and select Resistor. The information for the Resistor will appear in the editor. In the list of available options, enable the option called Assign Component Name to MODEL. When enabled, this option will assign the component name to the MODEL attribute when the part is placed in a schematic. For a resistor, the MODEL attribute would automatically be defined as Resistor which would then reference the model that was just created in the Default.lib file. Repeat for the Capacitor and Inductor components if default models are being added for these.

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.

Protect Command
Micro-Cap has the capability to encrypt proprietary schematic or macro files which can be used to protect sensitive material. To encrypt a file, go to the File menu and choose the Protect command. The Password dialog box that appears below will be invoked. The file to be encrypted must be the active file when this command is selected.

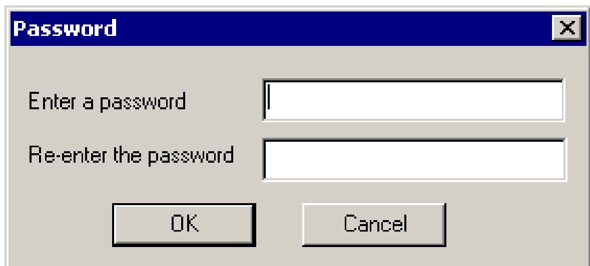


Fig. 1 - Password dialog box

The dialog box contains two fields in which the user will enter the password that they wish to use to encrypt the file. Both entries must of course match each other. The dual entry method helps shield the user from making a typo when entering the password.

Once encrypted, the file can only be opened for viewing and editing if the password is supplied. Protected files can still be used as macros and simulated in other circuits without requiring a password. This provides a useful scheme where encrypted macros may be used by another user in one of their simulations. For the other user, the macro component in their schematic would just be a black box as they would not have the capability to view the internal macro circuitry without having the password.

To remove the protection, load the protected file and use the Save or Save As commands, which will save the file in standard text format without encryption. Since the normal Save commands decrypt the file, the typical method is to protect the circuit file when it reaches its final finished form. Otherwise, the Protect command will need to be used to save the file each time it is updated.

Introducing Micro-Cap 9

We are pleased to release the next generation of the Micro-Cap simulator, Micro-Cap 9.0. 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.

Models

New BJT, MOSFET, IGBT, and Diode models

A number of new models have been incorporated into MC9.

Mextram BJT model - This is a vertical NPN or PNP bipolar model that describes velocity saturation, base widening, Kirk effect, impact ionization, and self heating effects.

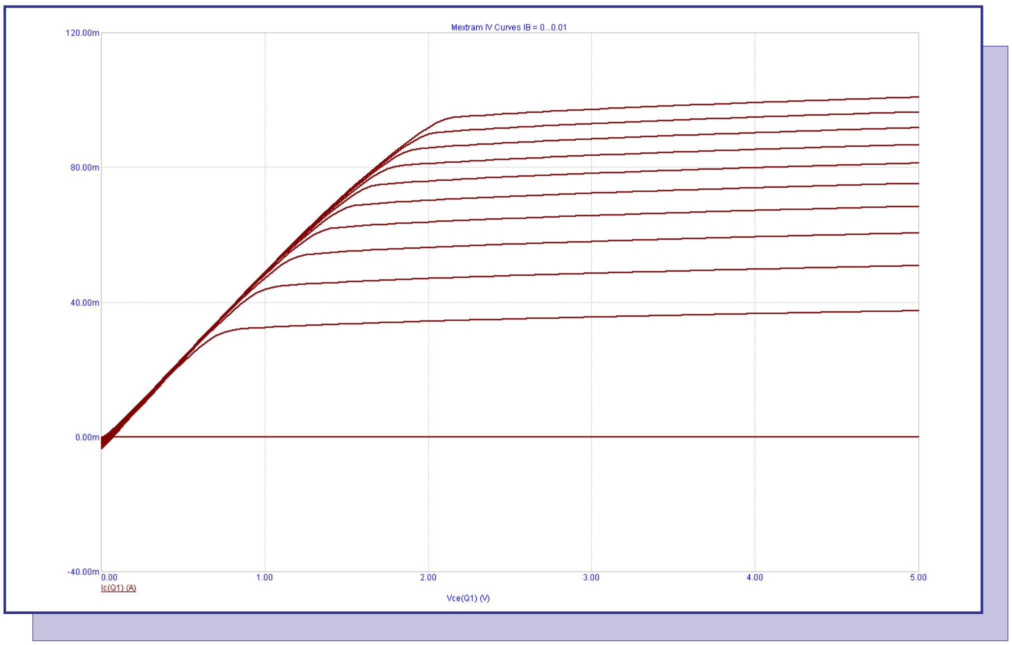


Fig. 2 - Mextram BJT model IV curves

Modella BJT model - This lateral PNP bipolar model describes current crowding, high-level injection, and bias dependent output impedance.

PSP MOSFET model - PSP is the latest compact, surface-potential based model suitable for digital, analog, and RF CMOS applications. Jointly developed by Philips and Penn State University, the model accounts for mobility reduction, velocity saturation, DIBL, gate current, and lateral doping gradient effects.

Philips MOSFET Model 11 - A popular Philips MOSFET model suitable for low voltage CMOS RF applications.

Philips MOSFET Model 20 - A high voltage compact LDMOS model.

Philips MOSFET Model 31 - A high voltage MOSFET model.

Philips MOSFET Model 40 - A high voltage Silicon-On-Insulator (SOI) MOSFET model.

Hefner IGBT model - An integrated IGBT model was added. It is based upon the implementation by Gregory Oziemkiewicz of the Hefner model.

JUNCAP and JUNCAP2 diode models - These diode models are used in conjuction with MOSFETs. JUNCAP2 is a MOSFET pn junction model that describes depletion capacitance, Shockley-Read-Hall currents, trap-assisted tunnelling and band-to-band tunnelling, avalanche breakdown, and shot noise.

New IBIS components
New IBIS components have been added that ease the importation of IBIS models into a simulation. These components can be placed in a schematic and assigned a pin from an IBIS file. It creates the appropriate model at that point and is ready for simulation.

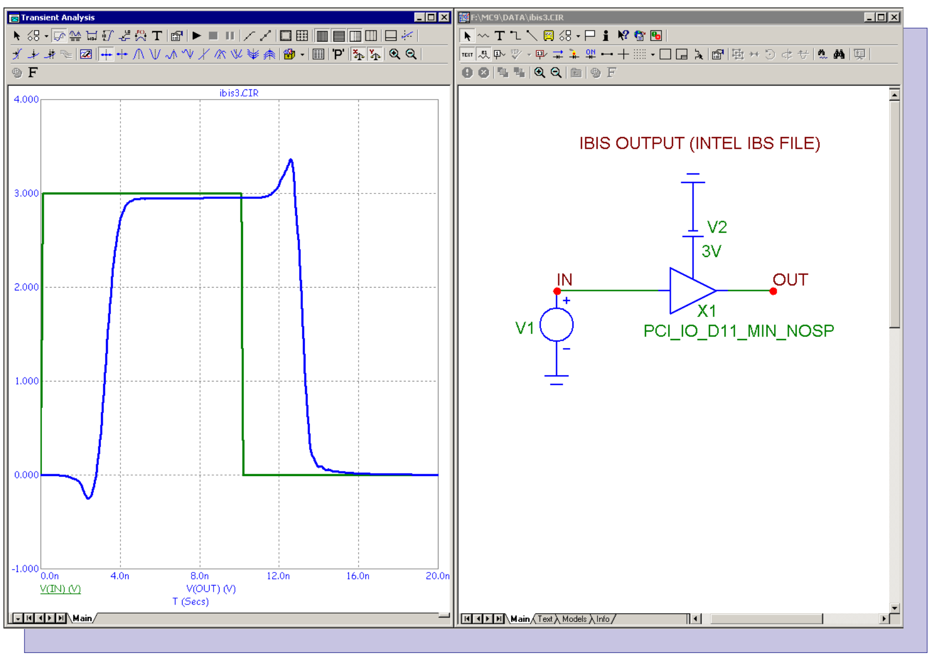


Fig. 3 - IBIS component and analysis

N-Port component
ABCD and T parameters may now be used with the N-port components.

Enhanced RLC models
Resistors, capacitors, and inductors now have optional model parameters for parasitic resistance, capacitance, and inductance.

User source
A trigger expression was added. The source is idle until the expression becomes true.

Digital file stimulus source
Added a repeat capability to the source.

Analysis

Enhanced convergence methods

The Junction GMIN Stepping and Pseudo Transient methods were added to the arsenal of operating point methods. There are now five methods:

Standard Newton-Raphson
Source Stepping
Diagonal Gmin Stepping
Junction Gmin Stepping
Pseudo Transient

Means were added to select the preferred order of usage and to automatically search for the fastest of the methods.

RSHUNT, CSHUNT, and RP_FOR_ISOURCE global parameters were added to provide augmented convergence. These parameters add an RSHUNT resistance or CSHUNT capacitance from every node to ground, or an RP_FOR_ISOURCE resistance across any current source.

Dynamic analysis mode

A new interactive analysis mode allows dynamic plot updates as changes are made. This is an extension of Dynamic DC to transient analysis, AC analysis, and DC analysis. Plots can either accumulate or replace showing the effect of successive changes in component parameters, model parameters, or user variables. Parameter changes can be controlled by sliders, cursor keys, or edits of any kind.

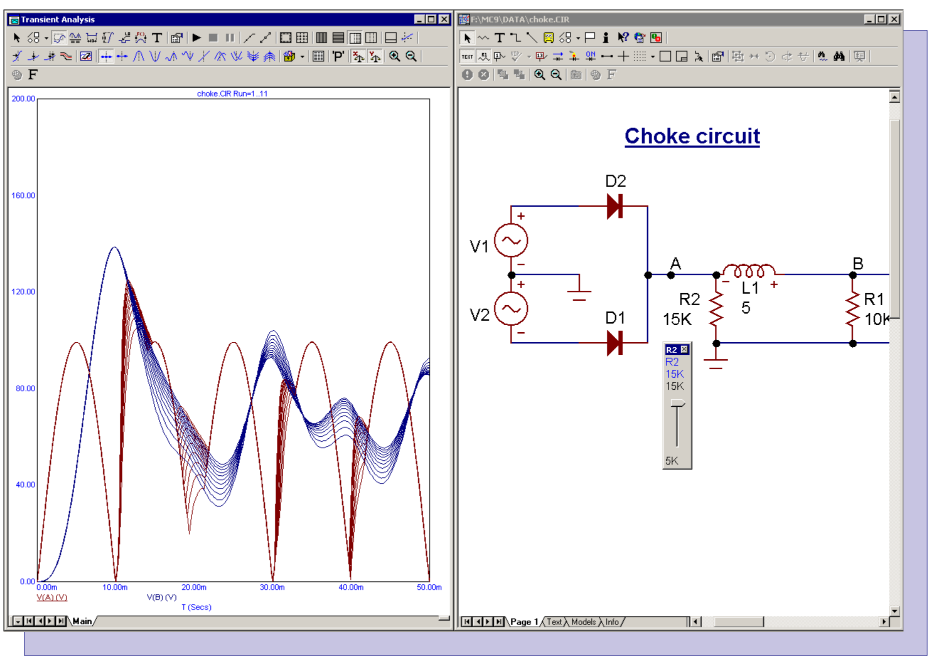


Fig. 4 - Dynamic analysis mode

Dynamic performance tags

These new tags perform curve measurements showing the new values each time an analysis is run or a new branch of a stepped curve is selected.

Plot pages

Analysis plot pages with assignable names allow plot grouping in different windows.

Waveform buffer

A new Waveform Buffer stores curves / waveforms automatically or under user control for later plotting and comparison. The buffer provides a convenient method for importing/exporting waveforms into any analysis.

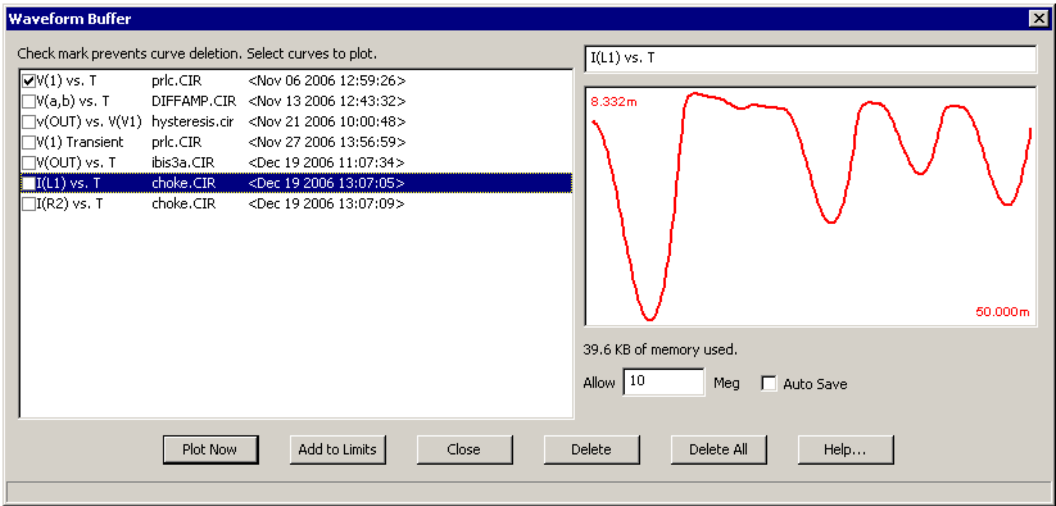


Fig. 5 - Waveform buffer dialog box

Numeric output improvements

Variable engineering notation within a column of numbers, selectable row/column tabular format, and curve aliases were added to numeric output files. Printing can be limited to a subset of the simulation range. Numeric output is now available for Dynamic AC and Dynamic DC.

New wildcard syntax for plot expressions

A new wildcard syntax is available for use in the Y Expression field of the Analysis Limits dialog boxes. This syntax allows the user to compactly specify the plotting of many similar expressions.

Define variable usage

.Define variables can now be used in the X and Y Range fields, the Maximum Time Step, and in the FFT Upper and Lower Time Limit fields.

Envelope command

This new command creates a polygon encompassing the curves of a Monte Carlo or stepped run. It is useful for graphically depicting the plot variation.

Visible region auto-scale

A new auto scale command is available that operates only over the X portion being currently displayed on the screen.

Additional analysis features

Added RMS and Average performance functions.
Added a data point only waveform style.
Cursors can now be placed on separate curves.
The numeric format can now be set individually for each tag.
The Save Curve command can now save in CSV format.
Stepping tabs are now check-marked if they are active.

Probe

Edit while probing

The interactive analysis feature of MC9 lets you edit the schematic while in probe. Each edit updates the simulation and shows the effect of the change. CTRL + SPACEBAR cycles through the available probe modes, while SPACEBAR now toggles between probe mode and schematic select mode.

Visual probing of macros and subcircuits

Drill down probing lets you visually probe macro schematics or subcircuits to any level. Instead of selecting items from a list, you can probe macro objects directly. With subcircuits, you can probe directly on netlist nodes and names.

Expanded Probe displays

FFT, Performance, and 3D Windows are now available in Probe.

Schematic Editor

Window tabs

Window tabs allow easy access to open shematics, plots, histograms, performance windows, etc.

Circuitry region enable / disable

Circuitry can be temporarily enabled or disabled with a simple click, allowing multiple versions of a circuit to be contained in a single schematic. Disabled regions are ignored during an analysis. Whole pages or regions within pages can be controlled.

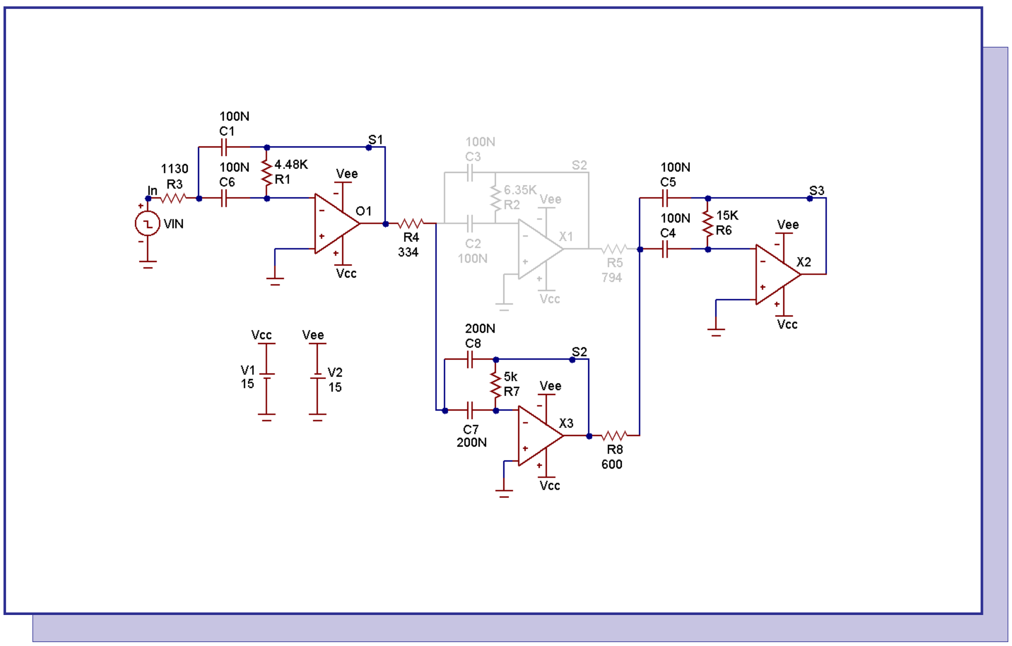


Fig. 6 - Disabled circuitry in the schematic

Region enable expressions

A new Region Box with an enable expression allows conditional inclusion of circuitry.

Schematic display

On-schematic display of last, RMS, average, or peak values.

Macro embedding

The Localize command now embeds external macro files into the circuit, for better file portablity.

Component panel

The component panel provides easy access to the component library for placing parts in a schematic. It has the capability to browse through the component tree or to quickly search for specific parts.

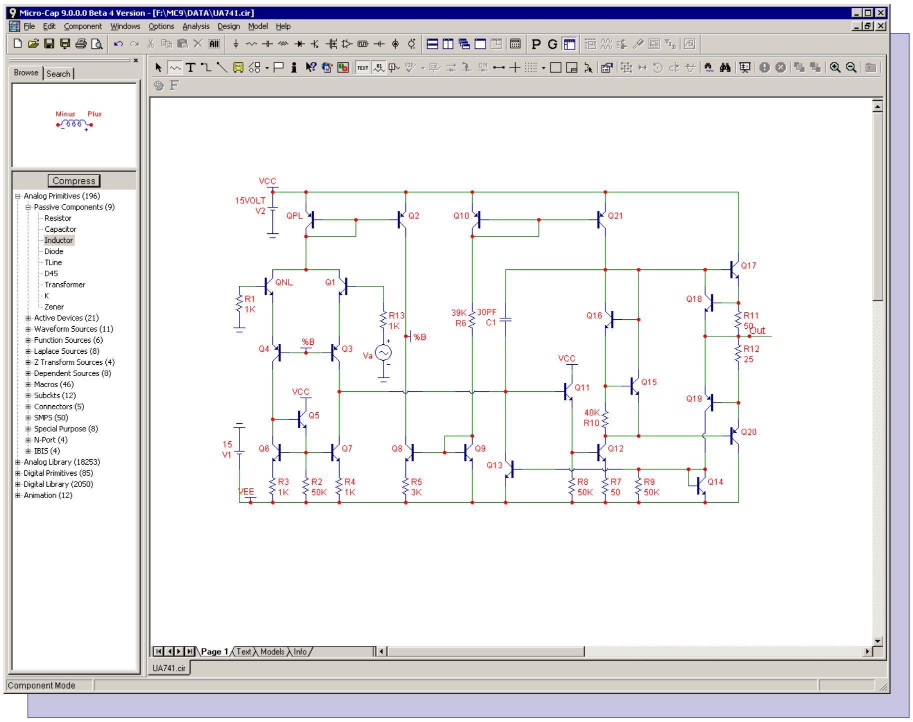


Fig. 7 - Component panel

User-assignable hotkeys

Users can set their own hotkeys for any command.

Multiple shape capability

Components can now have multiple shapes assigned to them. Each shape has a shape group as well as a shape name, allowing easy switching between shape families.

Warning messages

Parts and nodes involved in a triggered warning are now colored. The command has been improved with the addition of global names like R@ to refer to any and all resistors. Warning messages are now also sent to the numeric output file.

Text control blocks

New text control blocks (.IF boolean_expression, .ELSE, .ELIF boolean_expression, and .ENDIF) allow conditional inclusion of text, which may control .commands, model statements, and SPICE netlist circuitry.

.SPICE / .ENDSPICE commands

These new commands let you add SPICE netlist-defined circuitry to the text pages of schematics without using a subcircuit.

Intelligent Paste command
A paste between circuits optionally adds the appropriate model statements, subcircuits, and macros to the new circuit.

Bus
A bus connector and associated wiring routines were added.

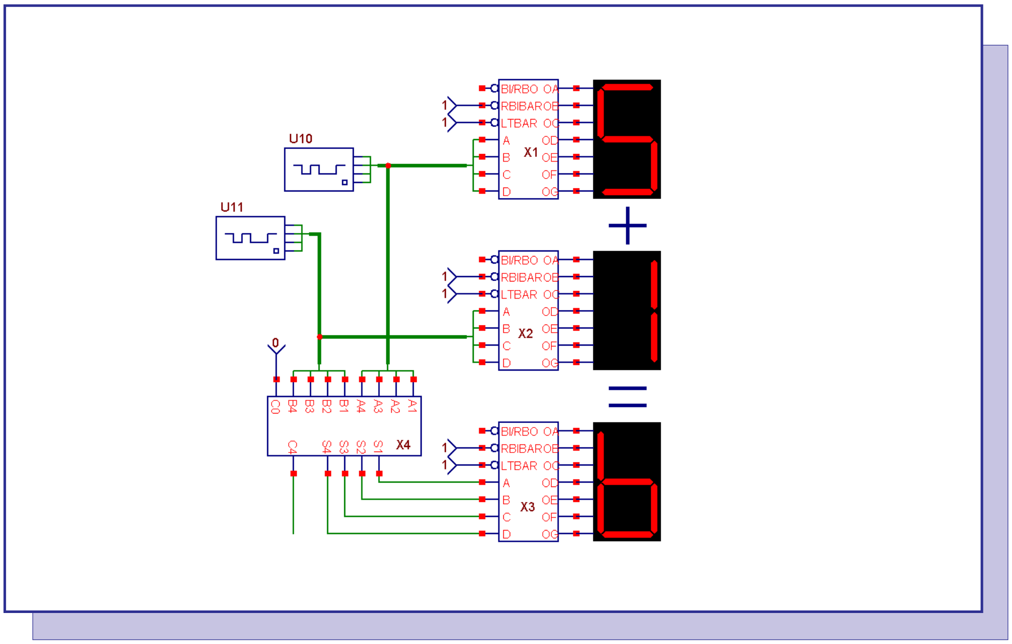


Fig. 8 - Bus connectors and wiring

Assignable component links
Components now have a user-assignable link for quick access to data sheets or other technical documents from an internet URL or a disk location.

Live text expressions
Expressions can now be buried in grid text and are updated as the constituent variables change.

Fit to Standard Values command
This finds the closest single or series/parallel combination of standard part values that match a given R, L, or C value.

RGB color stored in file
RGB color information (rather than a palette number) is now stored in the circuit file so when the circuit is given to someone else, the colors stay the same.

Additional schematic improvements
Improved Component Find searches as you type.
Parameter Find command now available in the Attribute dialog box.
Common patterns are now button selectable for the digital stimulus source.
Rubberbanding now eliminates all diagonal lines.
Multiple attributes of a component can be moved together.
A delimiter can now be specified for the Bill of Materials export.
Multiple circuits can be assigned a password simultaneously.

Next Object command lets you select overlapping objects.
Parameters can now be passed to a circuit through a batch file.
Node Snap and Text Increment have been assigned hotkeys.
Element information is displayed at the mouse cursor in the schematic.

Monte Carlo

New Tolerance dialog box
This provides a rapid way to apply LOT and DEV tolerances. Tolerance templates can be saved and applied to future models.

Improved Load MC File command
A Load MC file dialog box lets you choose which of the cases you want to load for review.

Performance measurement display
Selecting a case from the Histogram display list while the plot is in Cursor mode causes the corresponding performance function measurement to be shown.

Filter Designer

Passive elliptic filters
Passive elliptic filter capability was added.

Fit to Standard Values command
A new command was added to find the optimal series or parallel arrangement of standard resistors, capacitors, and inductors.

Expressions

Arrays were extended to handle multiple dimensions and complex values.
FFTS and HARM functions now have an optional bandwidth parameter.
Complex impedance and conductance expressions are available in AC for passive elements.
Legendre polynomials were added.
The Lambert W function was added.
The limiting exponential functions EXPL and EXPLP were added.

Component Editor

Right click menu lets you add, copy, delete, paste, replace, and find information for parts.
Can now delete groups that contain parts.
Move Parts command can now select the From group as well as the To group.
Add Part Wizard now lists the available models in the selected files.
Added an option to initially display the macro parameters when the part is placed in a schematic.

Miscellaneous Improvements

An enhanced migrate command now handles macro and library files.
Recently used component list size is now user adjustable.
Find in Files command now searches text in analysis limits and plots.
File dialog boxes retain the view settings. Windows XP or later.

Repetitive Time Switch Macro

The time switch capability available within the Switch component in Micro-Cap provides a method to define a time range window where the switch will either be open or closed. Only a single time range may be specified. To specify more than one time window, multiple switches would need to be placed in a series or a parallel configuration depending on the application. If the switch opens and closes with a constant duty cycle and period, the switch macro in the figure below can be used instead. This repetitive time switch macro can be useful in oscillators, timers, or any other circuit that has a predictable switching time. This macro circuit has many advantages over using multiple instances of the Switch component. Not only does this macro improve on the ease of use over using multiple switches, but it also provides smooth transitions during its switching periods which greatly aids convergence.

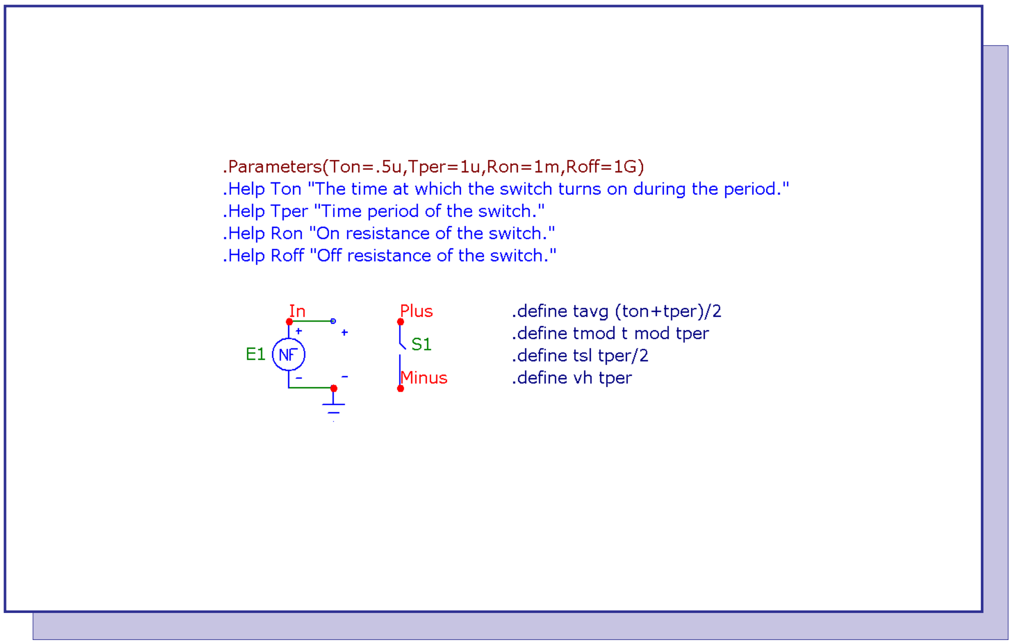


Fig. 9 - Repetitive time switch macro

The time switch macro has four parameters that are passed to it: Tper, Ton, Ron, and Roff. The Tper parameter defines the period of repetition in units of seconds. The Ton parameter defines the transition point within each time period. For example, if Ton is set to .5u and Tper is set to 1u, the switch will close at .5u, open at 1u, close at 1.5u, open at 2u, etc. Ton must be less than Tper. The Ron parameter defines the resistance when the switch is closed. The Roff parameter defines the resistance when the switch is open.

The macro has two pins: Plus and Minus. An S (V-Switch) component models the switch portion of the macro with its output being between these two pins. This switch has its model defined as:

.MODEL TSW VSWITCH (RON=Ron ROFF=Roff VON=Ton VOFF={Ton-Ton/100})

The RON and ROFF parameters of the switch model are defined using the Ron and Roff parameters that are passed through to the macro. The VON parameter is defined with the value of the transition point parameter Ton. The VOFF parameter uses the following equation to define its value:

Ton-Ton/100

Since the S (V-Switch) has smooth transitions, a transition region must be defined for the model. When the input voltage to the switch is greater than VON, the switch resistance will be equal to RON. When the input voltage is less than VOFF, the switch resistance will be equal to ROFF. The transition region where the resistance of the switch goes between ROFF and RON is defined by the difference between the VON and VOFF parameters. For convergence reasons, the transition region should be large enough where at least a few data points will be sampled in this region during the simulation. For this macro, the transition region has been arbitrarily set so that it is 1/100 of the Ton value.

Because the S (V-Switch) is actually a voltage dependent switch, the E1 nonlinear function voltage source has been placed at the inputs to the switch. This NFV source creates a voltage waveform with the following expression:

(t > ton/2) * (((tmod >= ton/2) AND (tmod < tavg)) * ((vh/tsl)*(tmod-ton/2)) +
(tmod >= tavg) * (vh+(-vh/tsl)*(tmod-tavg)) + (tmod < ton/2) * (ton-2*tmod))

where the tavg, tmod, tsl, and vh parameters are set using define statements as follows:

.define tavg (Ton+Tper)/2
.define tmod t mod Tper
.define tsl Tper/2
.define vh Tper

While it looks complicated, this expression produces a basic triangle waveform whose period is equal to Tper. The tmod variable which uses the Mod operator sets the periodicity of the waveform. Its valley voltage is set to 0, and its peak voltage is set to Tper volts. The rise and fall times of this waveform are set by the variable tsl which is the Tper value divided by 2. During each period, on the rising edge of the waveform, the source will produce a value of Ton volts at Ton seconds, with respect to the current period, thus closing the switch. On the falling edge of the waveform, the source will produce a value of Ton volts every Tper seconds thus reopening the switch. Some users may wonder why we didn't incorporate a simpler voltage expression to control the switch such as a sawtooth waveform that can be defined with the expression:

T Mod Tper

While this expression would create a waveform that would toggle the switch at the correct transition times, it also creates an instantaneous voltage change every Tper seconds. This voltage change would cause the switch to instantaneously change resistance every time it reopened which can lead to potential convergence issues. Using the triangle waveform defined above assures a smooth transition when the switch both opens and closes.

An example circuit using the repetitive time switch macro is displayed in Figure 10. When the switch is closed, the 10V battery charges the C1 capacitor to a value of 5V due to the resistor divider. When the switch is open, the capacitor is discharged through the R1 resistor. The parameters of the switch macro have been defined as follows:

Ton = .5u
Tper = 1u
Ron = 1G
Roff = 1m

For each period of 1us, the switch resistance will be set to Roff for the first .5us and set to Ron for the second .5us. Note that the Ron parameter has been set to 1Gohm and the Roff parameter has been set to 1mohm. This has the effect of swapping the open and close states for the switch. The simple rule for this is that when Ron << Roff, the switch will close at the specified Ton time. When Roff >> Ron, the switch will open at the specified Ton time.

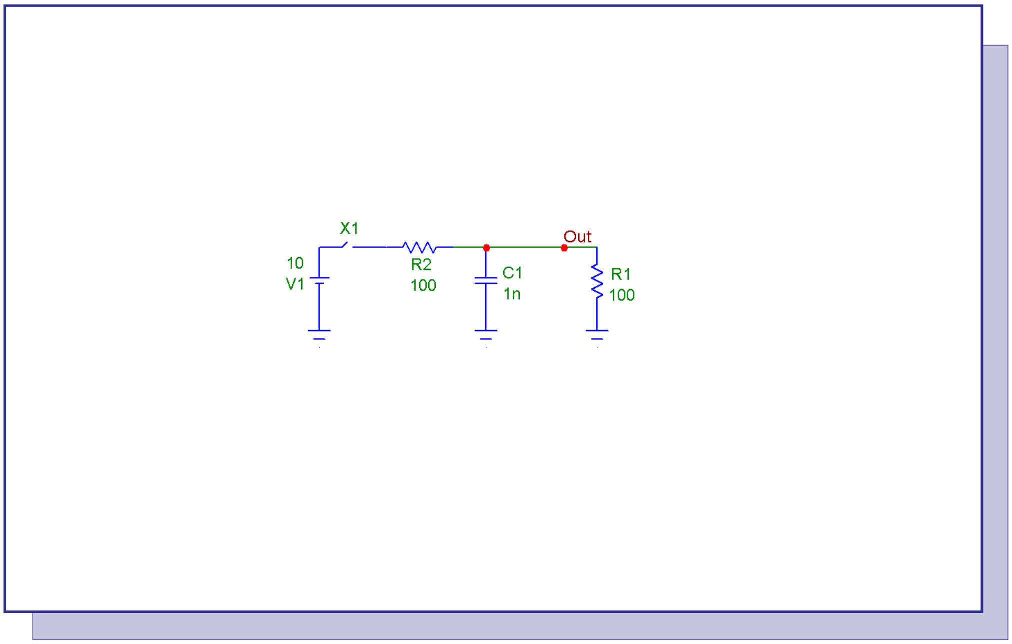


Fig. 10 - Repetitive time switch example circuit

The transient simulation for this circuit is displayed in Figure 11. The top waveform shows the voltage at node Out. As can be seen, the Out voltage is a periodic exponential waveform due to the charging and discharging of the capacitor. The bottom plot shows the resistance of the switch in the macro. It is plotted using the expression:

R(X1.S1)

This expression plots the resistance of the S1 switch within the X1 macro. If the View / Data Points option from the Scope menu was enabled for the plot, it would show that a good sampling of data points was taken during each of the transition regions of the switch resistance.

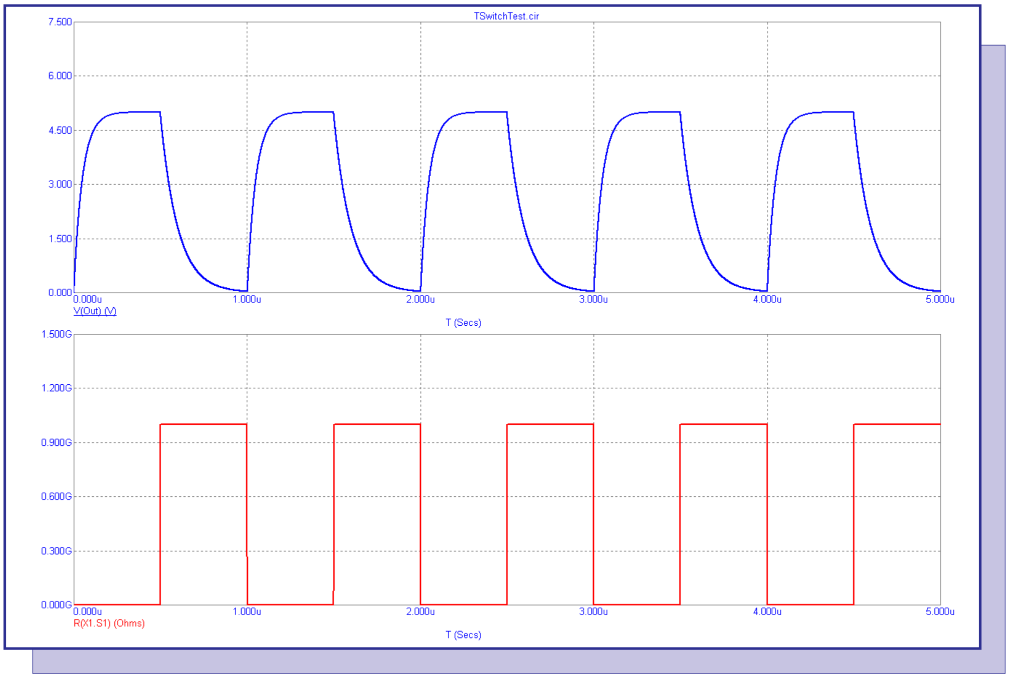


Fig. 11 - Repetitive time switch analysis

Measuring Insertion Loss

Insertion loss, often used within telecommunications design, is a means of determining the terminal attenuation that results from inserting a device into the signal path. It is measured as the ratio of the signal power delivered to the terminal prior to insertion versus the signal power delivered to the terminal after insertion. The following example consists of a two port filter that is operated between a resistive source and a resistive load. In this case, the insertion loss is calculated by comparing the power dissipated in the load resistor before the filter is inserted to the power dissipated in the load resistor after the filter is inserted. The example schematic is displayed in the figure below.

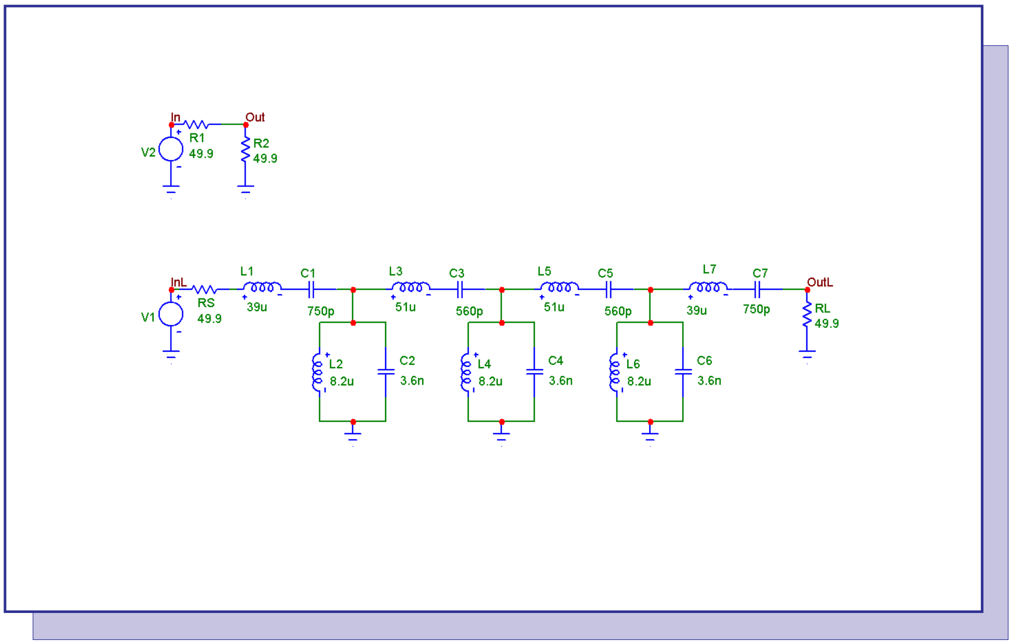


Fig. 12 - Insertion loss example circuit

This schematic actually consists of two circuits. The circuit on top represents the state before the insertion of the filter, and the circuit on the bottom represents the state after the insertion of the filter. Both circuits use a Voltage Source component as their input. These two sources need to be defined with the same values. In this case, the AC magnitude value has been set to 1, and the AC phase and DC values have been set to 0.

The filter inserted into the signal path was created from the Passive Filter Designer in Micro-Cap. The filter is a passive bandpass Chebyshev dual implementation. It was created with the following characteristics:

Center Frequency = 1MHz
Passband Frequency = .7MHz
Stopband Frequency = .8MHz
Passband Gain = 0dB
Passband Ripple = 3.01dB
Stopband Attenuation = 20dB

The power dissipated in the load resistors must be compared to each other. A way to simplify the expression is shown as follows:

$P_B = V(Out)^2 / 49.9$ = Power dissipated in the load resistor before insertion
$P_A = V(OutL)^2 / 49.9$ = Power dissipated in the load resistor after insertion
$A_{i,Np} = .5*\ln(P_B/P_A) = \ln(\ |V(Out)/V(OutL)|\ )$
$A_{i,dB} = (20/\ln 10) * A_{i,Np}$
$A_{i,dB} = (10*\log_{10}(P_B/P_A))$
$A_{i,dB} = 20*\log_{10}(\ |V(Out)/V(OutL)|\ )$
$A_{i,dB} = dB(V(Out)/V(OutL))$

For a resistive load, insertion loss reduces to the simple expression dB(V(Out)/V(OutL)), which calculates the ratio of the voltages across the two load resistors in dB. The insertion loss plot for this filter is shown in the figure below.

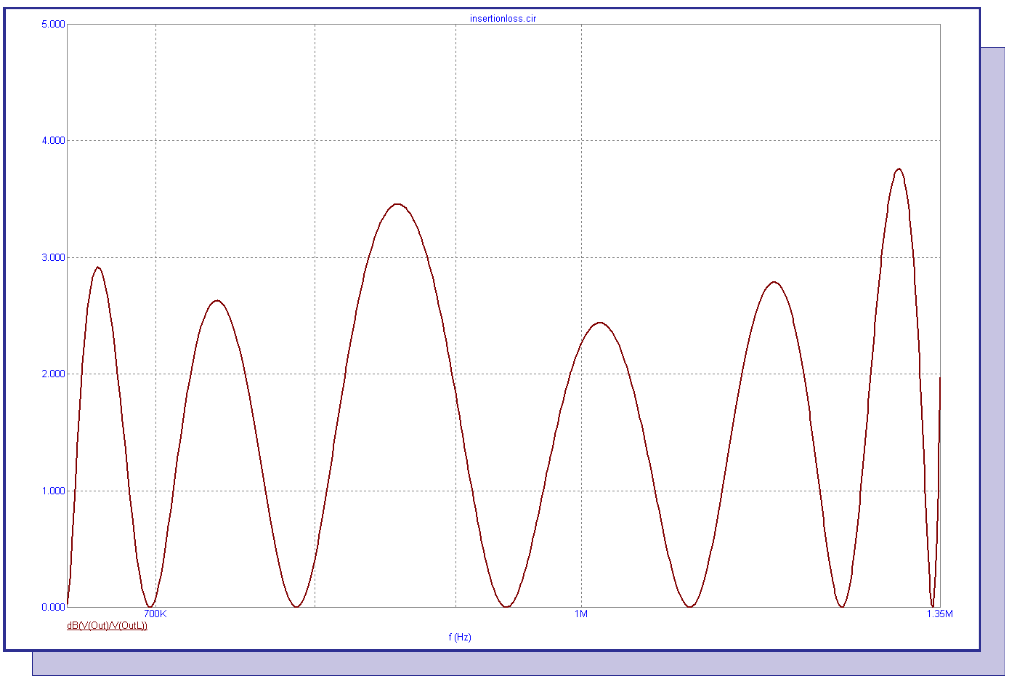


Fig. 13 - Insertion loss measurement

The plot was created through an AC analysis that simulates the frequency range across the passband region (650kHz to 1.35MHz) of the filter. For this example, since the passband gain of the filter was 0dB, the insertion loss measurement across the passband region is due to the passband ripple of the filter.

Product Sheet

Latest Version numbers

Micro-Cap 9 .... Version 9.0.0
Micro-Cap 8 .... Version 8.1.2
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