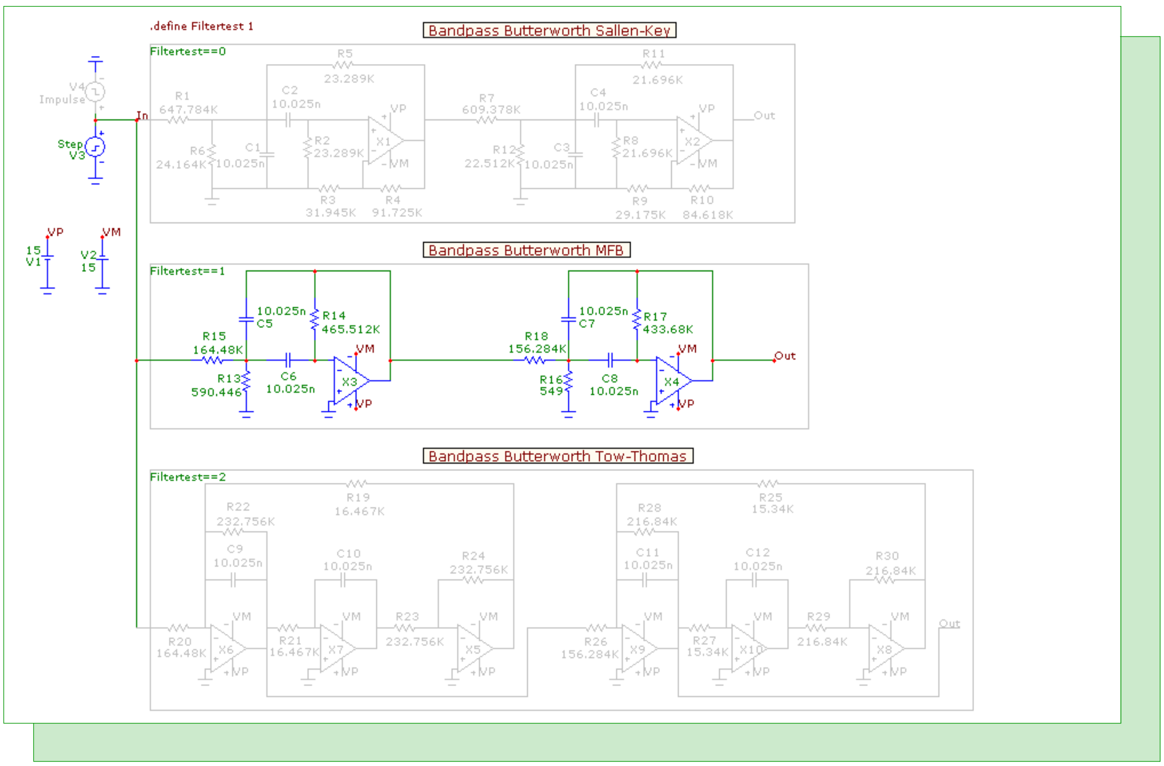
spectrum news

Applications for Micro-Cap™ Users

Spring 2009

News



Enable/Disable Capabilities

Featuring:

- Modeling a Warburg Impedance in AC Analysis
- Enabling and Disabling Schematic Objects
- Displaying Calculations in Dynamic DC

News In Preview

This newsletter's Q and A section describes how to tolerance a battery for a Monte Carlo simulation. The Easily Overlooked Feature section describes the Copy/Paste Model Information option which copies local model information along with the component being copied.

The first article describes how to model a Warburg impedance in an AC analysis using the FREQ attribute of an NFI source.

The second article describes how to use the capability available in Micro-Cap to enable or disable schematic objects. Disabled objects are ignored during simulations and when creating netlists and Bill of Materials. This capability provides a nice technique to quickly reconfigure a schematic without having to delete circuitry that may need to be used in future simulations.

The third article describes how to use both types of formula text to display calculations, such as differential voltages or the power dissipated within a subcircuit, on the schematic while in a Dynamic DC analysis.

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

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# 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: How do I define a tolerance on a battery component so I can run Monte Carlo simulations on my circuit?

Answer: Typically the only objects that can have a tolerance assigned to them are define variables and parameters within model statements. There are a couple of exceptions and tolerancing a battery is one of them. Assigning a tolerance to a battery is simple. In the VALUE attribute of the battery, enter the LOT statement after the voltage value of the battery. The syntax for defining the value of a battery is:

<value> [LOT[/<lot#>][/<distribution name>]=<value>[%]]

<lot#> optionally specifies which of one hundred random number generators, numbered 0 through 99, are used to calculate parameter values. This lets you correlate parameters among different components.

<distribution name> optionally specifies the distribution as follows:

UNIFORM = Equal Probability distribution
GAUSS = Normal or Gaussian distribution
WCASE = Worst case distribution

As an example, if you would like to enter a 10V battery that has a tolerance of 20%, the VALUE attribute for the battery component would be defined as:

10 LOT=20%

Most of the parameters in the Voltage Source and Current Source components can also be toleranced in a similar manner.

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.

Copy / Paste Model Information Option

The standard copy and paste in Micro-Cap can copy components from one schematic to another schematic. When a component is copied and pasted, any references it has to a define statement, model statement, or subcircuit statement are also copied and pasted. However, the actual define, model, or subcircuit statements themselves are not copied and pasted along with the component. A second step of copying and pasting the corresponding statement is necessary to fully transfer the component information over to the new schematic. In cases where the model or subcircuit statements are stored in the global libraries this is not a problem since the reference is global and is available to all schematics.

An alternative to the standard copy and paste can be used by enabling the Copy / Paste Model Information option. This option can be accessed within the Options - Circuit page of the Preferences dialog box. If this option is enabled, when a component is copied that uses a define, model, or subcircuit statement, Micro-Cap checks to see if the corresponding statement for that component is stored locally in the schematic file. If the statement is local, it is also sent to the clipboard. When that component is pasted in a different circuit, the statement is also pasted at the same time.

If a component uses a define statement, the define statement is pasted into the Text page of the schematic. If a component uses a model or subcircuit, then those statements are pasted into the Models page of the schematic.

If the corresponding statement is taken from a global library or external file then the statement is not copied and pasted with the component.

If a model, subcircuit, or a define statement with the same name already exists in the schematic file that is being pasted to, then the corresponding statement is not added during the paste operation. The original information in the schematic file has the precedence.

Modeling a Warburg Impedance in AC Analysis

An electrochemical cell can contain a type of impedance referred to as the Warburg impedance which is created by diffusion within the cell. The Warburg impedance represents semi-infinite linear diffusion within the solution and is represented by the following expression:

$$Z_w = \sigma/\omega^{1/2} - j\sigma/\omega^{1/2}$$

where σ is the Warburg coefficient. The value of the Warburg impedance is dependent on the frequency of the perturbation. At higher frequencies, the Warburg impedance is smaller since the reactants do not have as far to move. Conversely, at lower frequencies, since the reactants have a greater length to travel, the Warburg impedance is higher.

In order to model this impedance in an AC analysis, the component used to model it must both be able to vary with frequency and be able to handle a combination of real and imaginary values. One component that can do both of these is the NFI (Nonlinear Function Current) source. The NFI source can handle complex, frequency varying expressions through its FREQ attribute capability. The FREQ attribute is only active during an AC analysis and has no effect in any other type of simulation. An expression defined within this attribute is evaluated at each frequency point of the simuation. The following expression in the FREQ attribute would model the Warburg impedance.

V(G1)/((Aw/sqrt(2*PI*F)) - (j*Aw/sqrt(2*PI*F)))

Since the NFI output is a current, the above expression produces the equivalent current of the Warburg impedance based on the basic I = V/Z relationship. The G1 instance in the expression is the part name of the NFI source in the schematic so it is referencing the voltage across itself. The denominator of the expression is the Warburg impedance equation where Aw is the Warburg coefficient.

Even though the FREQ attribute of the NFI source takes precedence during an AC analysis, the VALUE attribute of the NFI source must also be defined. The value specified for the VALUE attribute will be used during the DC operating point calculation that is performed at the beginning of the AC analysis simulation. The VALUE attribute may need to be set to an appropriate value for some circuits. While the NFI will still work fine no matter what value is used, the rest of the circuit is linearized based on the results of the DC operating point calculation. Therefore, the VALUE attribute can have an effect on what region of operation the circuit is linearized to for the resulting AC analysis run. For this example, the VALUE attribute has been defined as 0 which will create an open circuit during the DC operating point calculation.

In order to plot some of the basic Warburg impedance curves, an NFI source has been defined with the above FREQ expression. The Aw coefficient for the expression is set to 300. The source is in series with a 20 ohm resistor that represents the charge transfer resistance.

The first Warburg impedance curve is shown in Figure 1. The Y expression is defined as Log(V(Out)/I(G1)). V(Out) measures the voltage across both the NFI source and the series charge transfer resistance. I(G1) is the current through this branch. This expression plots the log value of the impedance of the combination of the Warburg impedance and the series charge transfer resistance. The impedance is plotted versus the log value of the frequency, Log(f). At lower frequencies, where the Warburg impedance dominates, the slope of the impedance plot is -1/2. The -1/2 slope,

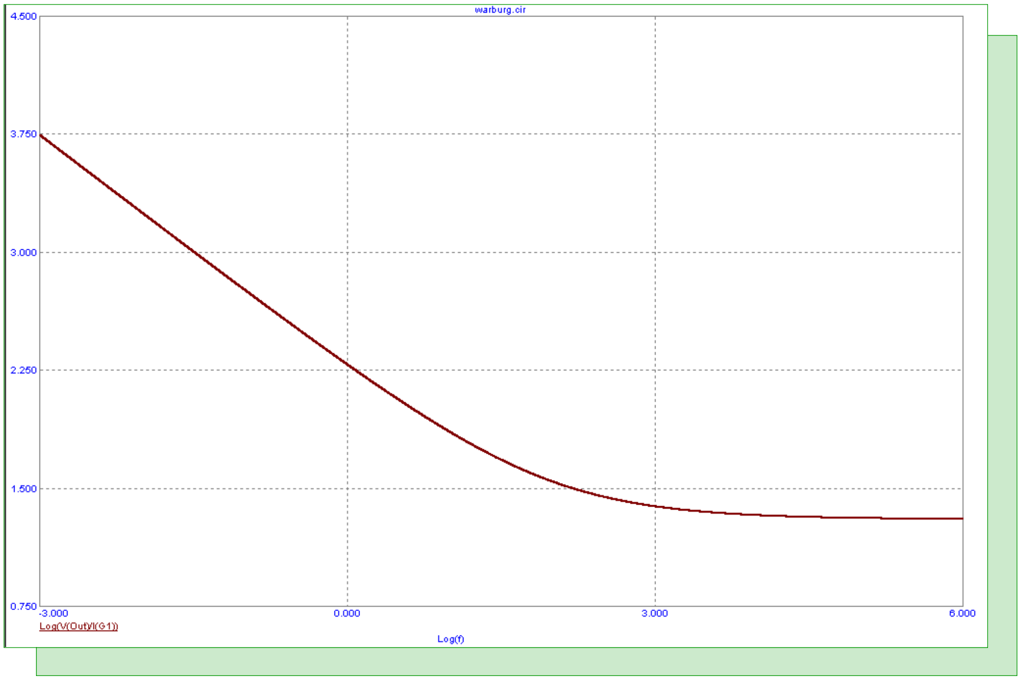


Fig. 1 - Impedance Magnitude Plot of the Warburg Impedance

which also appears as a 45 degree phase shift in phase plots, is characteristic of diffusion impedances.

The other two basic Warburg impedance curves are shown in Figure 2. In this case, the real and imaginary parts of the Warburg impedance are plotted on the Y axis. These impedances are plotted versus the expression 1/sqrt(2*PI*F) which is equivalent to $1/\omega^{1/2}$. Once again, the series charge transfer resistance is also factored into these curves. The real and imaginary parts of the impedance are linear and parallel to each other as can be expected by looking at the standard expression for the Warburg impedance. The imaginary part intersects 0 when the frequency is 0. The real part intersects the value of the series charge transfer resistance when the frequency is 0. In this case, that is 20 ohms. Note that the slopes of both of these curves, which are displayed in the cursor tables below the plot, are equivalent to the Warburg coefficient which has been set to 300 for this AC analysis simulation.

The technique of using an NFI source can be used with other frequency varying complex impedances also. As long as one has an expression that models the impedance, the NFI source can be used to model it in an AC analysis.

References:
1) http://www.consultrsr.com/resources/eis/warburg1.htm - Research Solutions and Resources LLC.
2) http://www.gamry.com/App_Notes/EIS_Primer/EIS_Primer.htm#About_The_EIS_Primer - Gamry Instruments

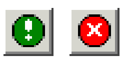
Fig. 2 - Real and Imaginary Warburg Impedance Plots

Enabling and Disabling Schematic Objects

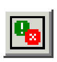
Micro-Cap has the capability to disable components, grid text, and wires within a schematic. Disabled objects are not included in any analyses, SPICE netlist translations, Bill of Materials reports, or PCB netlists. This capability provides a nice technique to quickly reconfigure a schematic without having to delete circuitry that may need to be used in future simulations. There are three methods to enabling or disabling an object.

1) Double click on the object while in Select mode. In the dialog box that comes up, there will be an Enable checkbox. When checked, the object is enabled, and when not checked, it is disabled. This method lets you enable or disable a single object.

2) Select one or more objects. Under the Edit menu, there are two commands called Enable and Disable. These commands will appropriately enable or disable all of the selected objects. These commands are also available in the toolbar with the following icons:



3) Use the Region Enable mode which can be activated through the Options / Mode menu or by clicking on the following icon:



Once the Region Enable mode is activated, the mouse can be used to box a section of circuitry. The Region Enable dialog box will appear so that a boolean expression involving symbolic variables (those created with a .Define or a .Param statement) can be defined for the region. When the boolean expression evaluates to true, all objects originating within the region will be enabled. When the boolean expression evaluates to false, all objects originating within the region are disabled. This lets you switch blocks of circuitry back and forth between being enabled and disabled by modifying just a single symbolic variable.

An example circuit that uses the enable/disable capability is shown in Figure 3. The circuit contains three different bandpass Butterworth filters. Each of these filters was created in the Active Filters design tool in Micro-Cap. The specifications used for designing each of these filters were:

Gain = 0dB
Center Frequency = 1kHz
Order = 2
Q = 10

The only design specification that varied between the three filters was the type of circuitry used to implement the stages of the filter. The top filter used the Sallen-Key circuit type. The middle filter used the MFB (multiple feedback) circuit type. The bottom filter used the Tow-Thomas circuit type.

When the active filter designer is set to implement the filter with ideal opamps and exact resistor and capacitor values, the frequency response of each of these three filters would be the same. To provide a little real world element to each of these filter configurations, the Implementation page in the active filter designer has the Resistor Values and Capacitor Values entries set to Combination. The Combination option finds the best series / parallel combination of standard part values that approximates the exact values that were originally calculated by the filter designer.

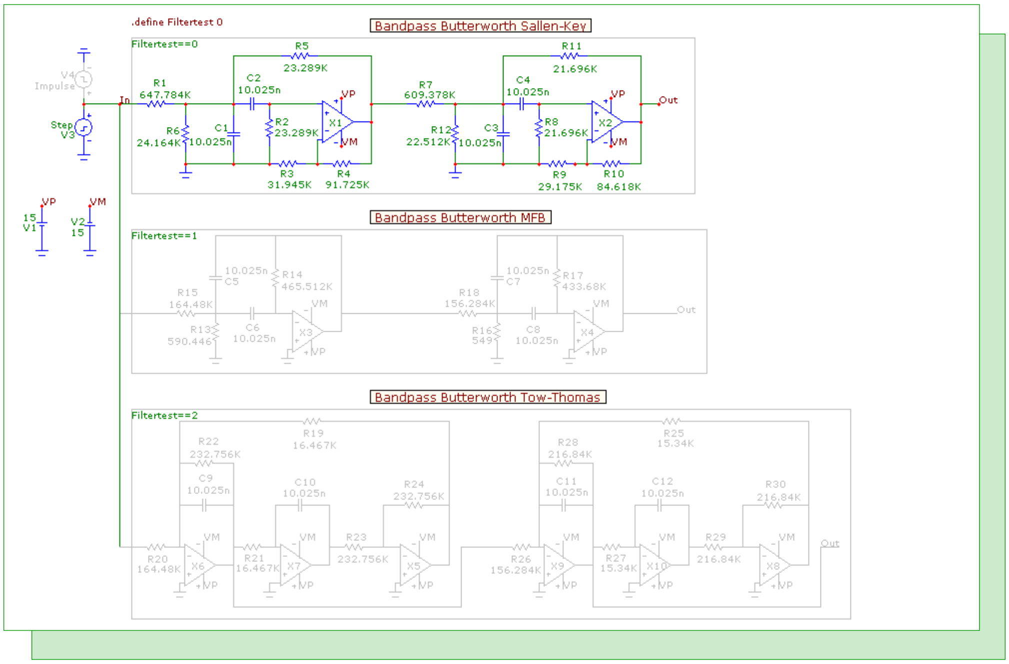


Fig. 3 - Sallen-Key bandpass filter enabled

The combination values are not calculated by the filter designer until the filter is actually created in the schematic. Once the filter is created, each resistor and capacitor will have an attribute called Combination available in the Attribute dialog box that displays the results of the combination calculation. For example, the 21.696K resistor in the Sallen-Key filter has the following defined for the Combination attribute.

21.696K = 21.5K+196 (e=-0.0073% n=2)

This means that the closest combination is achieved by a 21.5kohm resistor in series with a 196ohm resistor. This calculation produces a result that is within -.0073% of the exact value that the filter designer calculated. There will also be a page called Standard Values that is added to the schematic file which provides a general report of the combination calculations for all of the corresponding components. Even though the impedance values vary by a small percentage with the Combinations option, the effect on the frequency response is quite noticeable.

In the schematic, each of the filters share the same input and power supplies. There are actually two voltage sources at the input. The V3 voltage source defines a step signal, and the V4 voltage source defines an impulse signal. Having both of these sources present in the schematic provides a quick way to switch between a step response and an impulse response in transient analysis. Of course, one of these sources must be disabled when an analysis is run. At this point, the V4 source has been disabled through the use of the Disable command under the Edit menu. This source appears in the schematic with a grey color. The color used for disabled components can be controlled in the Color/Font page of the Properties dialog box.

The Region Enable mode has been used to define the region around each of the filters with a boolean expression that determines whether the region will be enabled or disabled. The regions have been defined with the following boolean expressions:

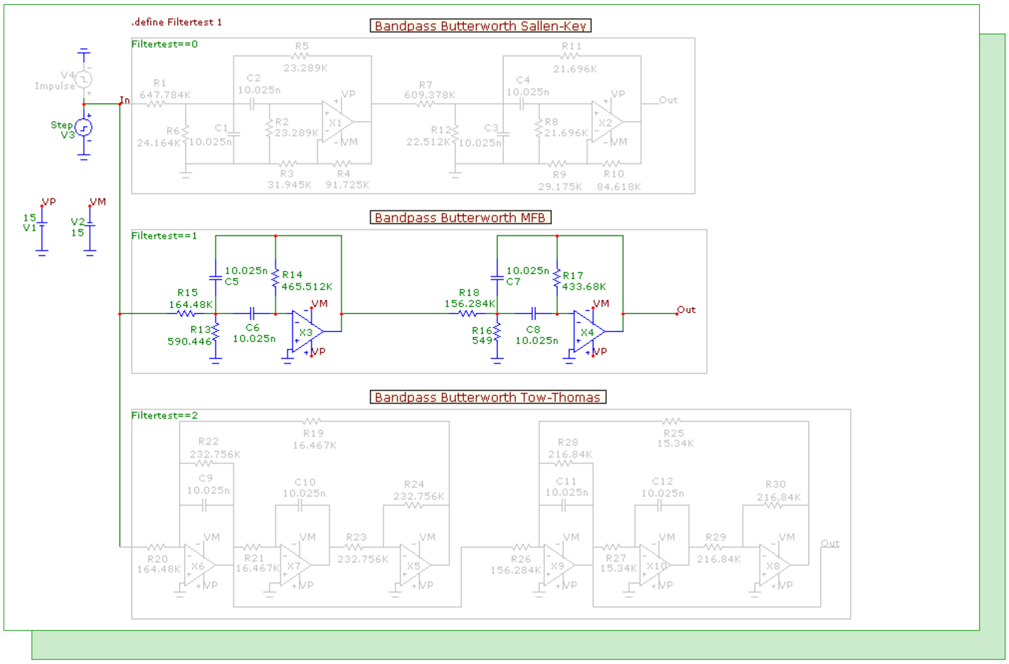


Fig. 4 - MFB bandpass filter enabled

Sallen-Key:
Filtertest == 0

MFB:
Filtertest == 1

Tow-Thomas:
Filtertest == 2

The Filtertest variable has been set using the following define statement:

.define Filtertest 0

Using a common variable in the above boolean expressions ensures that only one region at a time will be active. When the Filtertest variable is set to 0, the Sallen-Key filter will be enabled. When the Filtertest variable is set to 1, the MFB filter will be enabled, and when the Filtertest variable is set to 2, the Tow-Thomas filter will be enabled. If the Filtertest variable is set to any other value, all three of the filters will be disabled. The boolean expressions for regions can be more complex than the previous examples, but any variables used must be static variables that have been set through define or param statements. Dynamic variables such as voltages and currents are not allowed in the expressions.

When a variable used in a boolean expression is modified in the schematic file, the schematic will update accordingly. Initially, the Filtertest variable is set to 0 so that the Sallen-Key filter is enabled. If the Filtertest define statement is modified so that the variable is now set to 1, the schematic will appear as in Figure 4. The MFB filter is now shown as enabled while the Sallen-Key filter is now drawn with the specified disabled color.

Symbolic variables used within the region enable boolean expressions can also be stepped in an analysis. This provides the capability to easily analyze different schematic configurations in the same analysis. In this example, the Filtertest variable is stepped in AC analysis. Filtertest is stepped through the values of 0, 1, and 2. The analysis will produce the frequency responses of the different circuit types. The resulting simulation is shown below.

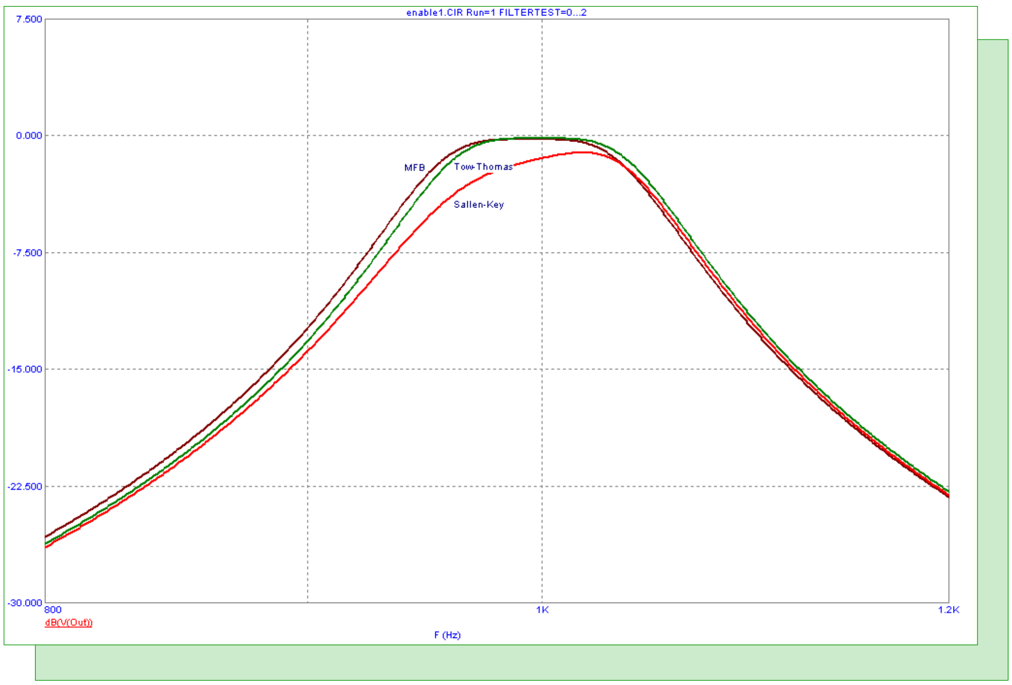


Fig. 5 - Stepping Filtertest in AC analysis

Since the regions have been defined so that only one at a time can be active, the output nodes for each of the filters share the node name Out as there is no danger in creating an unintended circuit connection. This makes plotting the filter output easier since a single expression can be used in the analysis limits no matter what filter region is enabled. In this case, specifying dB(V(Out)) in the analysis limits plots the output voltage in decibels of all three filter configurations.

For a transient analysis, the step response source is currently enabled. To quickly switch to the impulse response source, the V3 source is first selected. The Disable command is selected under the Edit menu. The V3 source is now shown in the schematic as disabled. Next, the V4 source is selected. The Enable command is selected under the Edit menu. The V4 source is now enabled for operation and the schematic appears as in Figure 6. If these Enable and Disable commands are going to be used often, it could be worthwhile to create shortcut keys for the commands in the Shortcuts section of the Preferences dialog box.

Running transient analysis on the schematic produces the results in Figure 7. This plot displays the impulse response of the bandpass Butterworth filter. Since the Filtertest variable is set to 1 in the schematic, the MFB circuit type is the one that is analyzed.

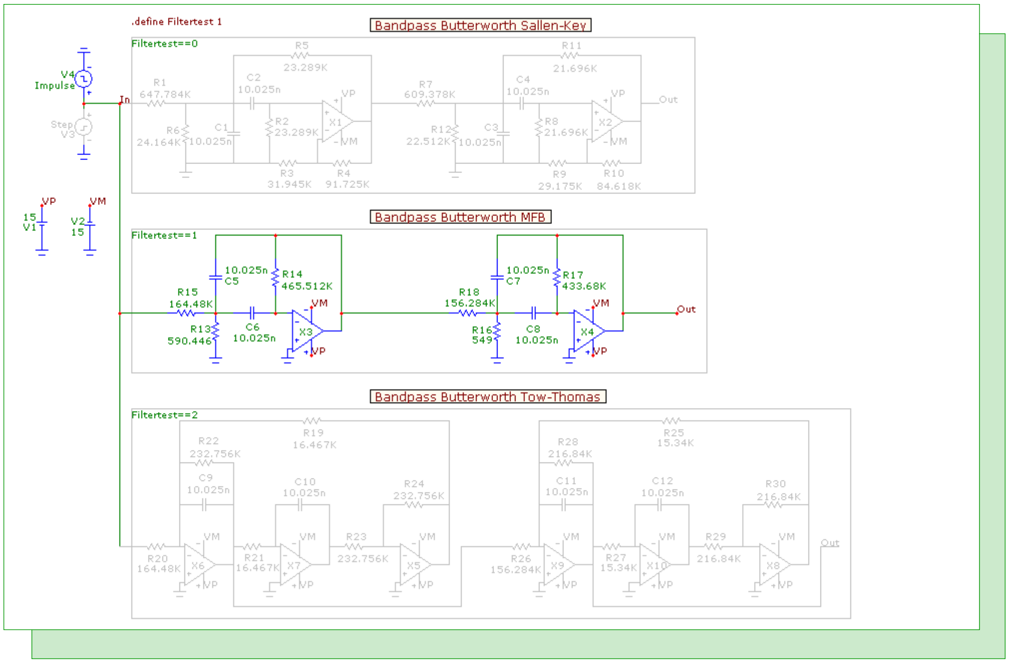


Fig. 6 - Impulse response source enabled

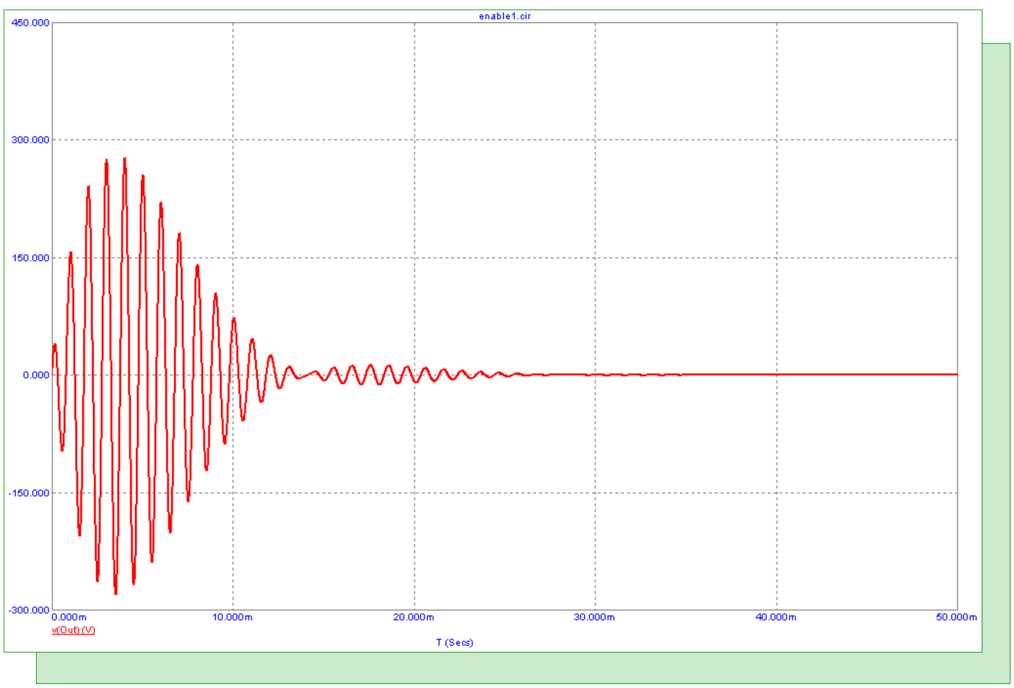


Fig. 7 - MFB impulse response analysis

Displaying Calculations in Dynamic DC

The Dynamic DC analysis calculates the DC operating point for the circuit and dynamically displays the resulting DC node voltage, device current, and device power values on the schematic. The current and power displays are only available for the analog primitive devices such as the diode or the NPN. Any components that are modeled as a subcircuit or macro will not have currents and powers available for display. Some users may want to expand on the available set of displayed values by using formula text to create their own calculations. With formula text, the existing voltage, current, and power values may be referenced to create new calculations such as device voltages, differential voltages, and subcircuit power calculations. The schematic below displays the use of formula text in making additional voltage, current, and power calculations for the circuit.

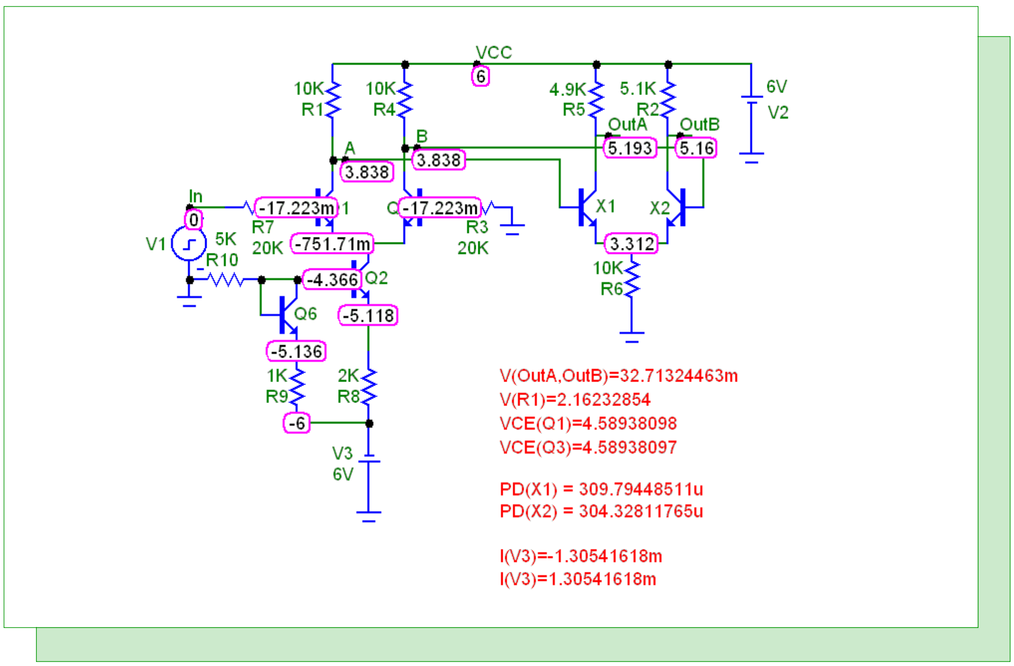


Fig. 8 - Using formula text in Dynamic DC

The circuit is a simple differential amplifier. Formula text has been entered below the circuit. There are two varieties of schematic grid text that can use formulas. The first form is as follows:

=formula

Grid text of this form behaves like a small spreadsheet. The presence of the = at the first character position tells Micro-Cap that the following text should be calculated as a formula. The four voltage calculations use this form, and the expressions used for them are shown as follows.

=V(OutA,OutB)
=V(R1)
=VCE(Q1)
=VCE(Q3)

Note that each of these expressions must be added separately through the Grid Text mode as this type of formula text can only handle a single expression. The first expression calculates the differential voltage between nodes OutA and OutB. The second expression calculates the voltage across the R1 resistor. The last two expressions calculate the voltages between the collector and emitter leads for the Q1 and Q3 NPN transistors.

The second form of formula text is as follows:

text...[formula]...text

Formula text of this type is calculated only if the Formula checkbox is enabled in the Text dialog box. The advantage to this form as opposed to the first one is that any number of formulas may be entered within the same text object. The power and current calculations in the schematic have been specified in this form. The following text was used for these calculations:

PD(X1) = [V(OutA,9)*I(R5)]
PD(X2) = [V(OutB,9)*I(R2)]

I(V3)=[I(V3)]
I(V3)=[V(R8)/R(R8)+V(R9)/R(R9)]

Since the X1 and X2 NPN transistors are actually modeled as a subcircuit, the basic power displays in Dynamic DC are not available for them. A simple calculation of the power is used instead where the voltage from the collector to the emitter is multiplied by the current through the resistor that is in series with the collector lead. The current calculations produce the same calculation using two different methods. The first I(V3) just calculates the current through the V3 source whereas the second calculation uses the voltage and resistance values of the two resistive branches that connect to the V3 source. As can be seen in the result, Kirchoff's laws are met.

One nice feature of the formula text is that the calculations are updated when the circuit is modified. The schematic below shows the updated calculations when the V2 source has been increased to 10V.

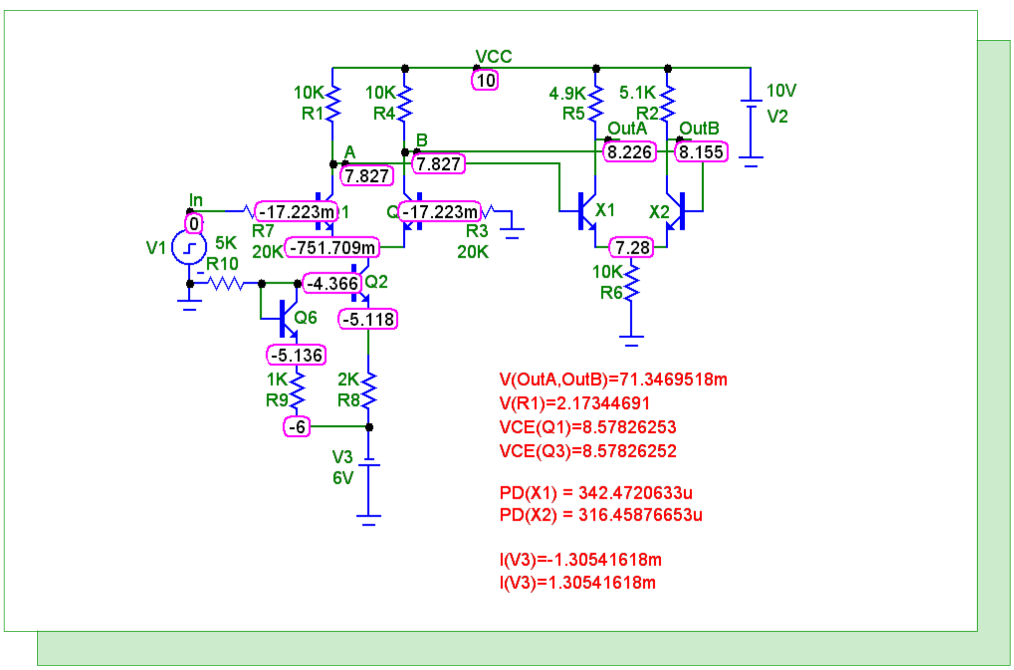


Fig. 9 - Updated formula text when V2 is 10V

Latest Version numbers

Micro-Cap 9 .... Version 9.0.6.1
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