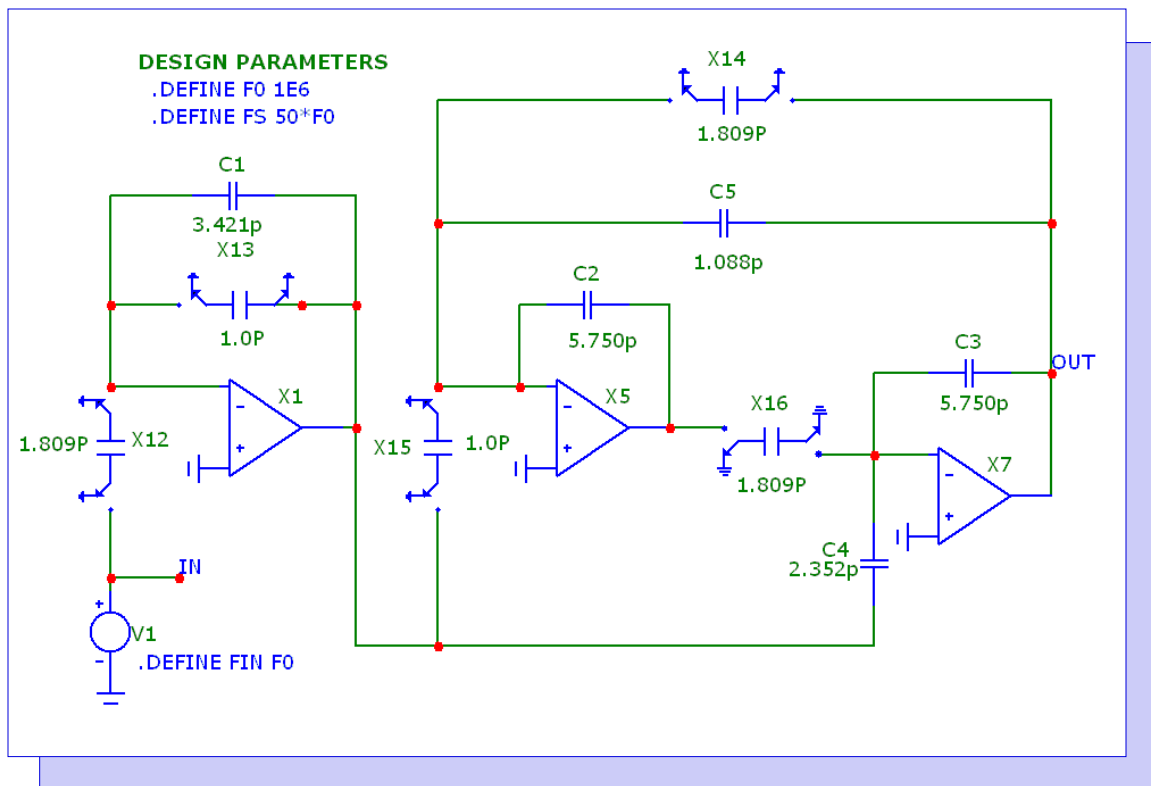


Winter 2011

News



Simulating Switched Capacitor Filters

Featuring:

- Simulating Switched Capacitor Filters
- Creating Ferrite Bead Models
- Importing PWL Data From A File

News In Preview

This newsletter's Q and A section describes how to change the attribute text color for an individual component and for multiple components at the same time. The Easily Overlooked Feature section describes some hidden commands that are available in Micro-Cap's text editor.

The first article describes how to do simulations of switched capacitor filters. A brief description of the theory is first introduced, followed by a discussion of several SC circuits and the macros provided to make them easy to use.

The second article describes how to create ferrite bead models through a lumped element circuit equivalent whose values are optimized in the Model program.

The third article describes how a file can be used to specify the data for a PWL voltage or current source.

Contents

News In Preview	2
Book Recommendations	3
Micro-Cap Questions and Answers	4
Easily Overlooked Features	5
Simulating Switched Capacitor Filters	6
Creating Ferrite Bead Models.....	14
Importing PWL Data From A File	18
Product Sheet.....	20

Book Recommendations

General SPICE

- *Computer-Aided Circuit Analysis Using SPICE*, Walter Banzhaf, Prentice Hall 1989. ISBN# 0-13-162579-9
- *Macromodeling with SPICE*, Connelly and Choi, Prentice Hall 1992. ISBN# 0-13-544941-3
- *Inside SPICE-Overcoming the Obstacles of Circuit Simulation*, Ron Kielkowski, McGraw-Hill, 1993. ISBN# 0-07-911525-X
- *The SPICE Book*, Andrei Vladimirescu, John Wiley & Sons, Inc., 1994. ISBN# 0-471-60926-9

MOSFET Modeling

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

Signal Integrity

- *Signal Integrity and Radiated Emission of High-Speed Digital Signals*, Spartaco Caniggia, Francescaromana Maradei, A John Wiley and Sons, Ltd, First Edition, 2008 ISBN# 978-0-470-51166-4

Micro-Cap - Czech

- *Resime Elektronické Obvody*, Dalibor Bielek, BEN, First Edition, 2004. ISBN# 80-7300-125-X

Micro-Cap - German

- *Simulation elektronischer Schaltungen mit MICRO-CAP*, Joachim Vester, Verlag Vieweg+Teubner, First Edition, 2010. ISBN# 978-3-8348-0402-0

Micro-Cap - Finnish

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

Design

- *High Performance Audio Power Amplifiers*, Ben Duncan, Newnes, 1996. ISBN# 0-7506-2629-1
- *Microelectronic Circuits*, Adel Sedra, Kenneth Smith, Fourth Edition, Oxford, 1998

High Power Electronics

- *Power Electronics*, Mohan, Undeland, Robbins, Second Edition, 1995. ISBN# 0-471-58408-8
- *Modern Power Electronics*, Trzynadlowski, 1998. ISBN# 0-471-15303-6

Switched-Mode Power Supply Simulation

- *SMPS Simulation with SPICE 3*, Steven M. Sandler, McGraw Hill, 1997. ISBN# 0-07-913227-8
- *Switch-Mode Power Supplies Spice Simulations and Practical Designs*, Christophe Basso, McGraw-Hill 2008. This book describes many of the SMPS models supplied with Micro-Cap.

Micro-Cap Questions and Answers

Question: How do I change the color of the attribute text for an existing component in my schematic?

Answer: In the schematic editor, change the mode to Select mode. Double click on the component whose attribute text color is to be changed. In the Attribute dialog box that comes up, click on the Font button. The Font and Color dialog box will then be invoked. This dialog box lets you change the font and color of any of the attributes for that specific component.

Note that when a new component is added to the schematic, the color of the attribute text is defined by the Default Attribute Text entry that is located within the Color/Font page of the Properties dialog box.

Question: Is there a way to change the attribute text color for multiple components at once?

Answer: The Change Attribute dialog box was designed to perform simultaneous operations such as this. The following procedure can be used:

- 1) Load the circuit file to be modified.
- 2) Click on the Edit menu. Then click on the Change submenu, and select Attributes. The Change Attribute dialog box will appear.
- 3) In the Action list, select the Change Color item.
- 4) In the Type section, select one or more of the component types that are to be changed. If you are changing all types, just click the Select All button.
- 5) In the Component section, select one or more of the components that are to be changed. If you are changing all of the components, just click the Select All button.
- 6) In the Name section, select one or more attributes that are to be changed. If all attributes are being changed, just click the Select All button.
- 7) In the Items section, select the new color through the Color button.
- 8) Click Apply.

This dialog box can also be used to add or delete attributes, change attribute values, change attribute fonts, show/hide display items, show/hide attribute names, and show/hide attribute values.

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.

Hidden Text Editor Commands

The text editor for Micro-Cap is used to edit any text files that have been created or loaded into Micro-Cap or to edit the text pages that are stored within a schematic file. The text editor has four commands that can be accessed via shortcut keys but that are not available through any menu or dialog box.

Change to Lower Case command

CTRL+L shortcut key

This command sets all selected text to their lower case characters.

Change to Upper Case command

CTRL+U shortcut key

This command sets all selected text to their upper case characters.

Delete to End of Line command

CTRL+D shortcut key

This command deletes all of the text that comes after the current cursor location. The command will only affect the text line that the cursor is positioned on.

Go to Brace

CTRL+] shortcut key

This command moves the text cursor to the matching brace. This command is applicable to parentheses (), curly brackets {}, and square brackets []. To use this command, place the text cursor in front of the brace whose match is to be located. Hit the CTRL+] shortcut combination on the keyboard, and the text cursor will be automatically moved to the spot right in front of the matching brace. The brace match can be made in either direction. This command is particularly useful when the braces have been nested in an expression in order to determine the priority of the calculations within that expression or to find a mismatched brace.

Simulating Switched Capacitor Filters

Switched capacitor (SC) filters have come into increasingly common usage. This article will explain a little of the theory and show how to run simulations on these circuits.

Theory

Consider the simple low pass SC filter and its equivalent analog filter circuit shown below.

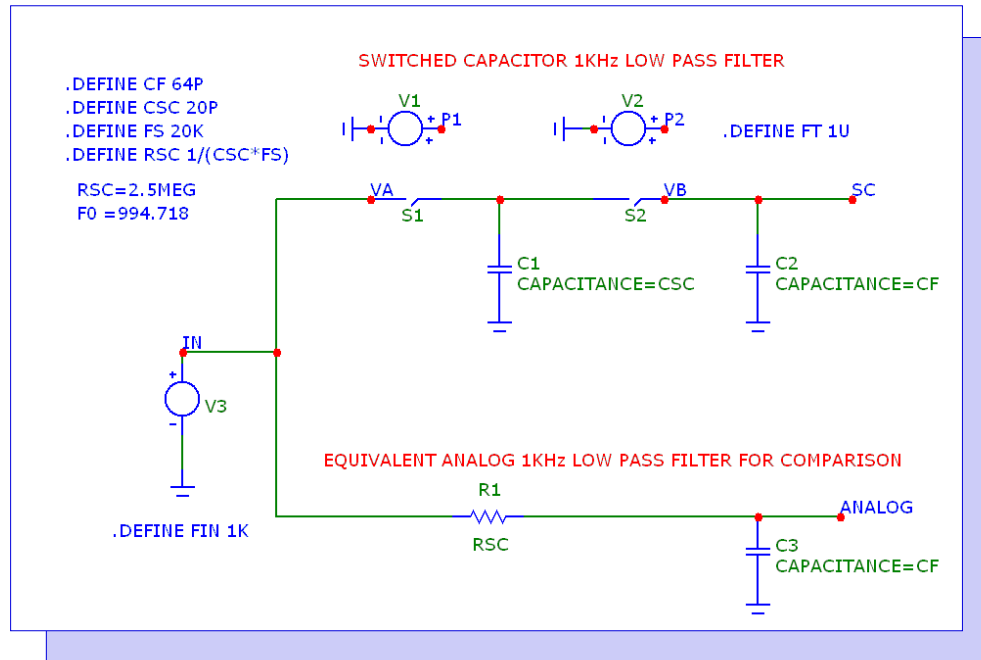


Fig. 1 - A low-pass switched capacitor filter and its analog equivalent

When S1 is closed, the capacitor C1 charges to VA. Then S1 opens and C1 holds the charge and voltage. After a short time, S2 closes and the capacitors C1 and C2 both charge to VB.

The initial charge on C1 is $Q_{\text{initial}} = C_{\text{SC}} \cdot V_A$. The final charge on C1 is $Q_{\text{final}} = C_{\text{SC}} \cdot V_B$.

The change in charge on C1 is $Q_{\text{initial}} - Q_{\text{final}}$.

$$dQ = Q_{\text{initial}} - Q_{\text{final}} = C_{\text{SC}} \cdot V_A - C_{\text{SC}} \cdot V_B = C_{\text{SC}} \cdot (V_A - V_B)$$

The current, averaged over one clock period T is $I = dQ / dT = C_{\text{SC}} \cdot (V_A - V_B) / T$

The effective resistance of the two switches and the capacitor C1 is

$$R_{\text{SC}} = V / I = (V_A - V_B) / (C_{\text{SC}} \cdot (V_A - V_B) / T) = T / C_{\text{SC}} = 1 / (C_{\text{SC}} \cdot F_{\text{S}})$$

where FS is the switching frequency of the clock. The 3dB corner of this filter is

$$F_0 = 1 / (2 \cdot \pi \cdot R_{\text{SC}} \cdot C_{\text{F}}) = 994.718$$

Two important properties of switched capacitors should be now apparent:

Switched capacitors behave like resistors: They can easily create large-valued resistances, which is otherwise very hard to do in integrated circuits. Resistances larger than $1E6$ are easy to create.

Precise control of F0: The 3dB corner frequency is controlled by the clock frequency and the CSC/CF ratio. Both the frequency and the ratio can be controlled very precisely (better than .1%). Since this is far superior to what can be achieved with the absolute value of resistors or capacitors, it is a key consideration in using SC filters.

Here is the transient analysis of the circuit and its Fourier spectrum:

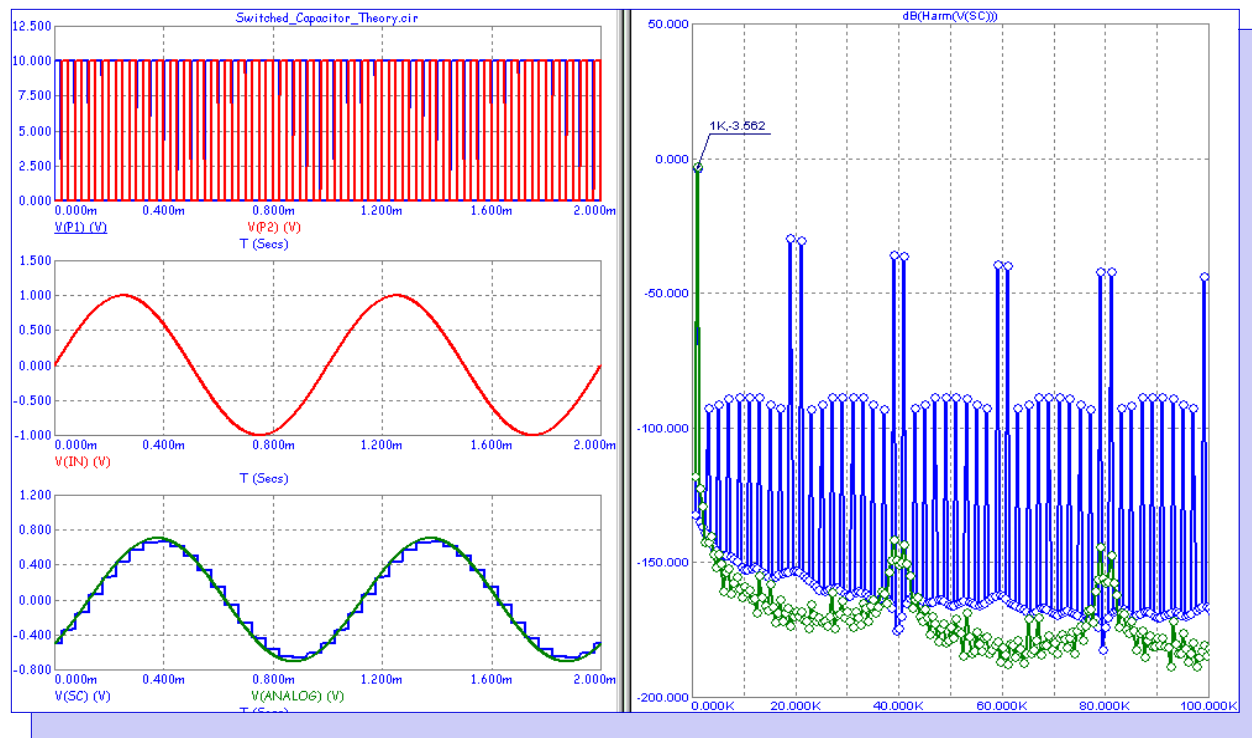


Fig. 2 - The transient analysis and its Fourier spectrum plot

Here we've run two periods of the input sine wave with $F_{IN} = F_0 = 1\text{kHz}$, the 3dB frequency of the filter. The bottom plot on the left shows the output of the SC (blue) and analog (green) filters superimposed. The spectrum plot on the right shows the dominant component at 1kHz is -3.5 dB, close to what is required at the 3dB point.

There are other frequencies as well. These frequencies, a byproduct of the switching process, are found at $n \cdot (F_S \pm F_{IN})$ with $n=1, 2, 3 \dots$. In this case, the unwanted frequencies are at 19kHz, 21kHz, 39kHz, 41kHz, and so on. If these are a problem in the design, they can easily be removed with a simple, low tolerance analog filter. It may seem odd to have to use an analog filter to 'cleanup' after a messy SC filter, but doing so is cheap and easy and still yields a filter with precision control of its band characteristics.

Switched Capacitor Integrator: In this circuit we've built a conventional analog lossy integrator and its SC equivalent. The SC integrator is built in the same way as its analog cousin but with the resistor replaced with the equivalent SC resistor. Its capacitance is set equal to $1 / (FS \cdot RVAL)$, so that its $REFF = RVAL$. This makes the integrator function in the same way as its analog counterpart.

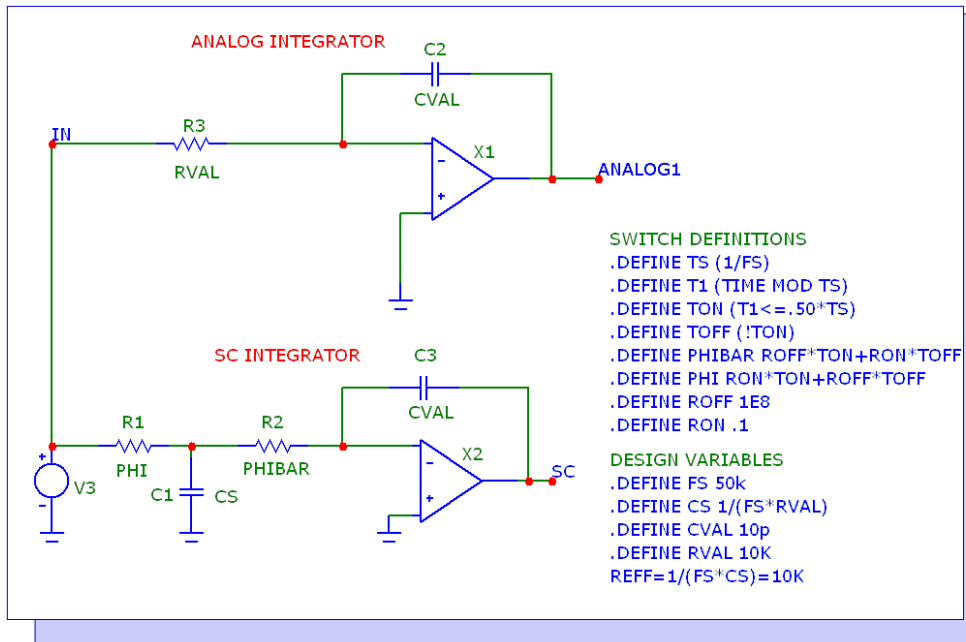


Fig. 3 - Analog and SC lossy integrators

Here we've implemented the switches using the R1 and R2 resistors whose values switch between ROFF and RON during the half periods of $1/FS$. This type of ideal switch exhibits no overlap or nonlinearities. Here is a transient analysis plot showing both analog and SC waveforms overlaid.

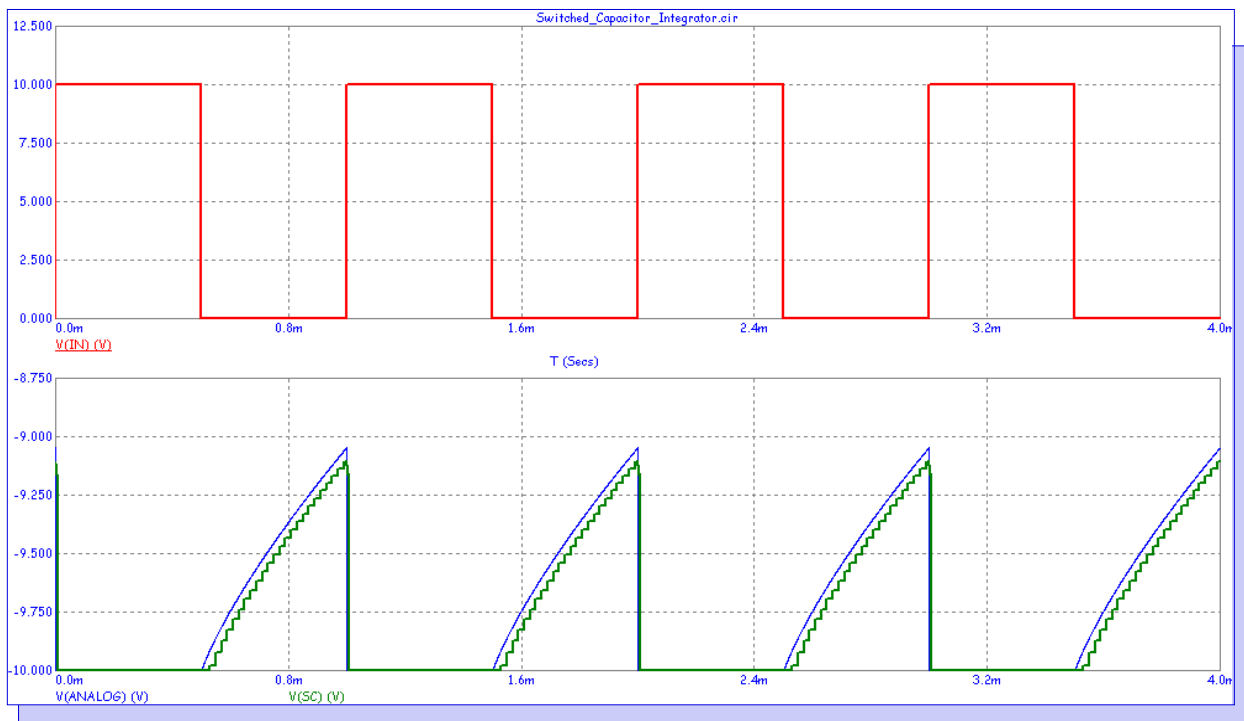


Fig. 4 - Transient analysis and spectrum plot of the lossy integrators

BP Filters: Integrators can be used to build band pass filters as in this example. The SC resistors have been implemented in this circuit as macros.

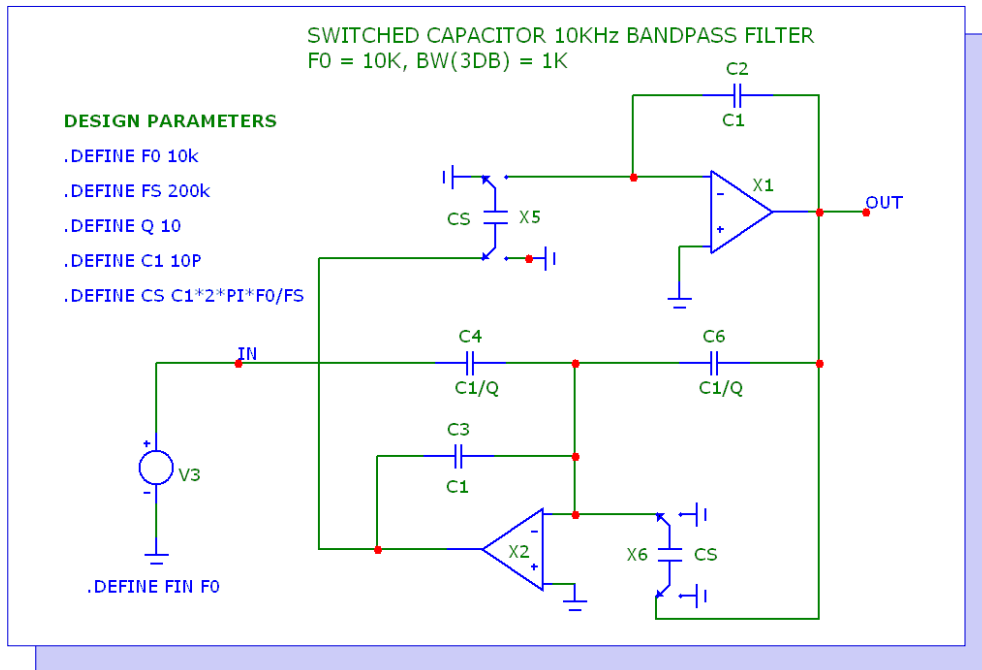


Fig. 5 - Band pass filter using SC integrators

Here is its transient analysis with an input frequency at the center of the pass band, 10kHz.

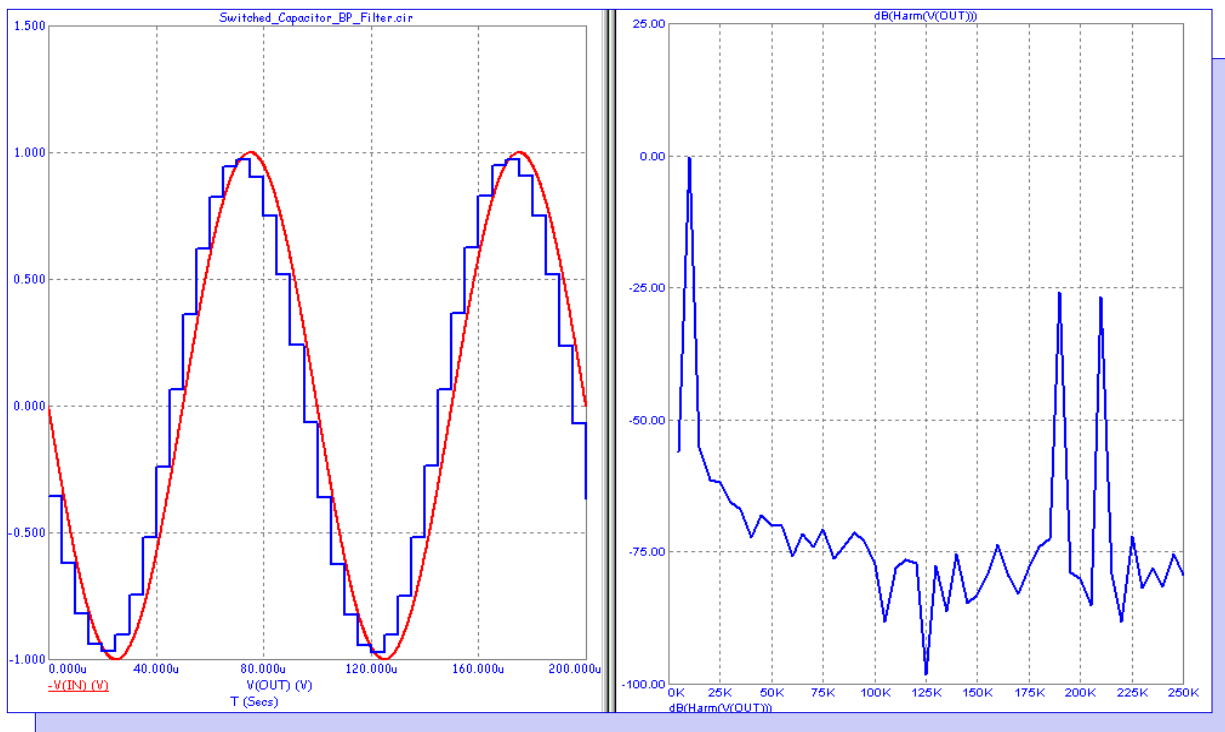


Fig. 6 - Transient analysis of the BP filter with a 10kHz sine wave

Here is its transient analysis with an input frequency at twice F0 or 20kHz.

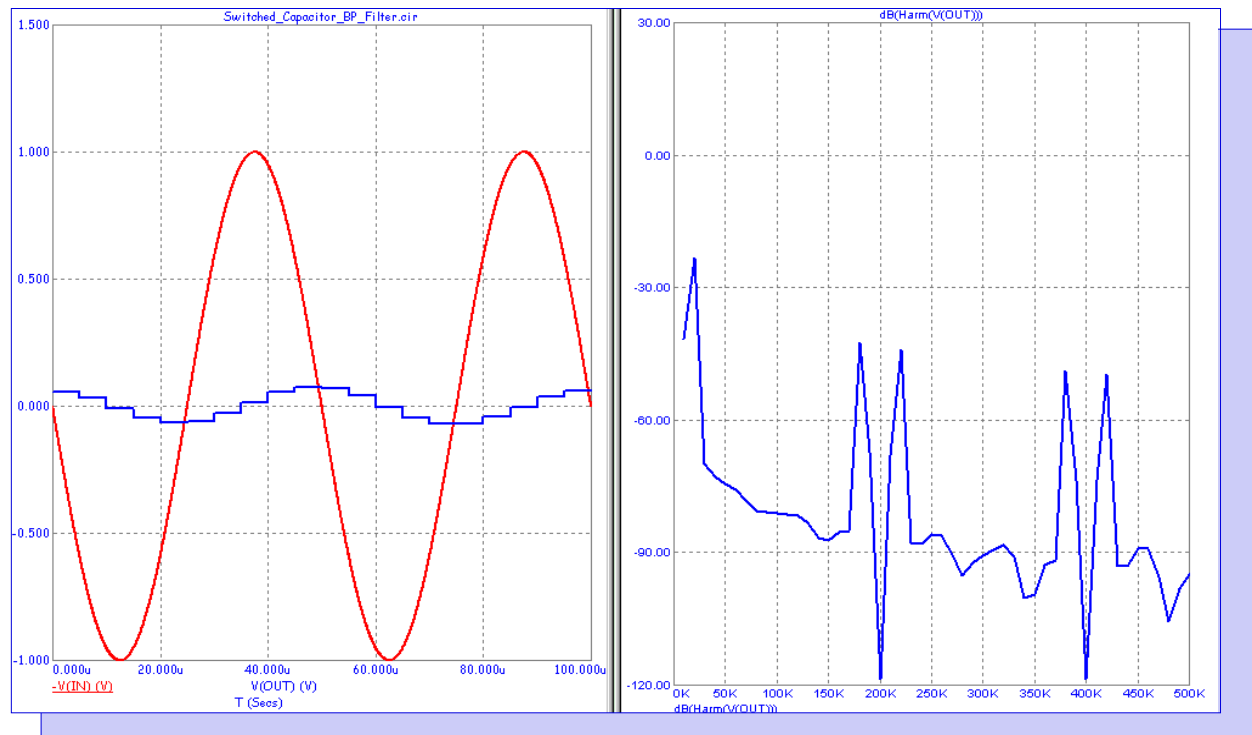


Fig. 7 - Transient analysis of the BP filter with a 20kHz sine wave

The Fourier plot shows that the gain at the input frequency has declined from 0dB (at 10kHz) to about -23db (at 20kHz), as expected.

Switched Capacitor Guidelines: Here are some important guidelines for successful simulations of switched capacitor circuits.

- **Use the switch macros:** Use the simple resistor-defined macro switches first. You can replace them later, if needed, with more realistic MOSFET switches.
- **Use the simple COMP macro in lieu of an opamp:** In these examples we've used the COMP macro where opamps are normally used. It functions as an ideal opamp with a TANH saturating characteristic. You can replace the COMPs later, once your circuit is working, with more realistic opamps.
- **Create Design Variables:** Use .Define statements for variables like FIN, the input sine wave frequency, FS, the sampling frequency, F0, the center frequency and so on.
- **Use PSS:** Use PSS to calculate the periodic steady state. This eliminates transients and makes the Fourier spectrum plots more accurate. This is more than useful, it is usually mandatory. If you do not have MC10 and thus cannot use PSS, use the Gear integration method, set the State Variables to Leave and press F2 repeatedly to get the steady state. Gear is superior to the trapezoidal method for SC circuits.
- **Keep All Source Periods Commensurable:** Make certain that FIN and FS are integer multiples. If they are not, aliasing will occur, the PSS routines will not work and you will have a mess.

• **Use variables in the analysis limits:** In the transient limits use expressions like $1/\text{FIN}$ for the Maximum Time Range and $1\text{E-}3/\text{FIN}$ for the Maximum Time Step. This insures that you are always running a multiple number of periods of the input frequency. If you want finer frequency resolution use $2/\text{FIN}$ or $4/\text{FIN}$ and so on.

Switched Capacitor Macros: The switch used in this circuit is a macro comprised of the following:

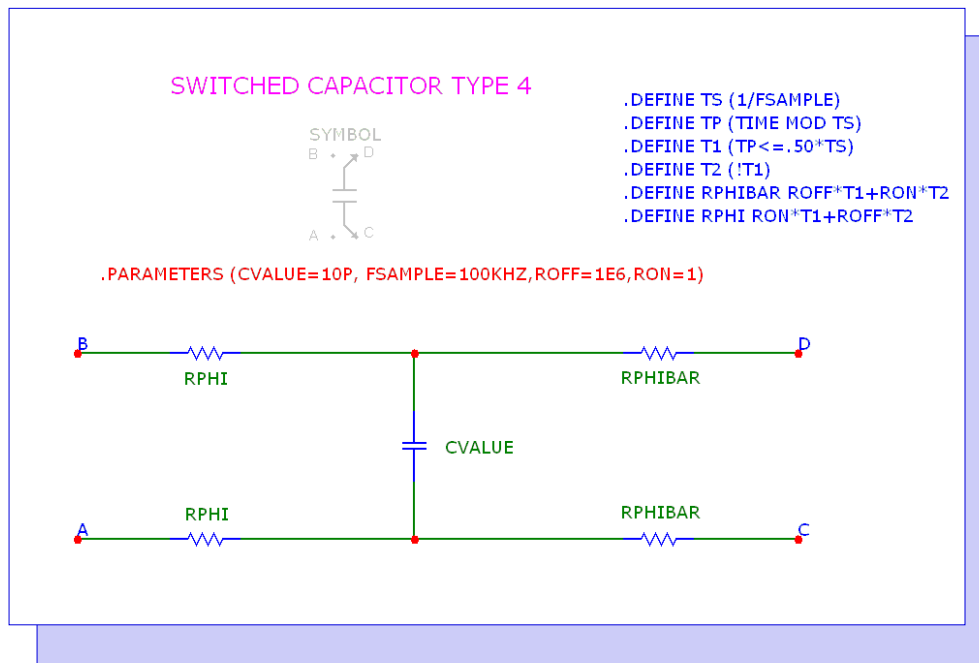


Fig. 8 - The SwitchCap4 macro

There are several types of switches used to make SC circuits. This particular one is useful when you need to switch four pins. The switching behavior is divided into two phases, each 50% of the period of the switching frequency parameter (FSAMPLE). This parameter is passed to the macro when it is placed in the circuit.

During the first half of the sample period, T1 is TRUE (1.0) and T2 is FALSE (0.0), so RPHI = RON and RPHIBAR = ROFF, connecting C1 from pin A to pin B.

During the last half of the sample period, T1 is FALSE (0.0) and T2 is TRUE (1.0), so RPHI = ROFF and RPHIBAR = RON, connecting C1 from pin C to pin D.

Of course you can build the switches in a variety of ways, from simple resistor expressions to MOS-FETS driven by two phase pulse sources. This particular implementation is simple and never overlaps as you change the switching frequency.

Here are a couple of other forms of switched capacitor macros, SwitchCap, and SwitchCap2.

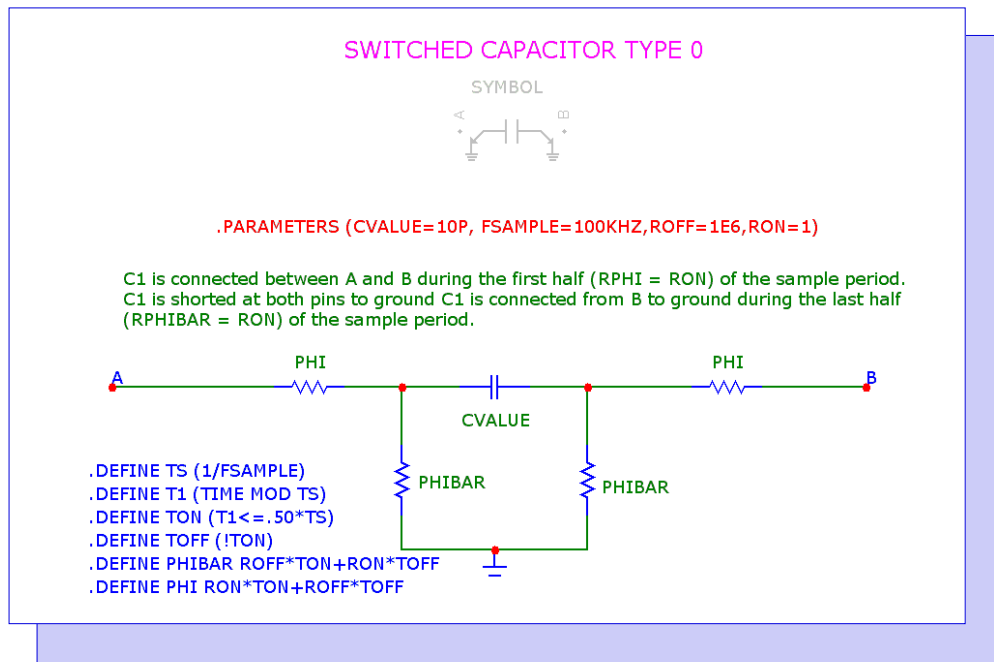


Fig. 9 - The SwitchCap macro

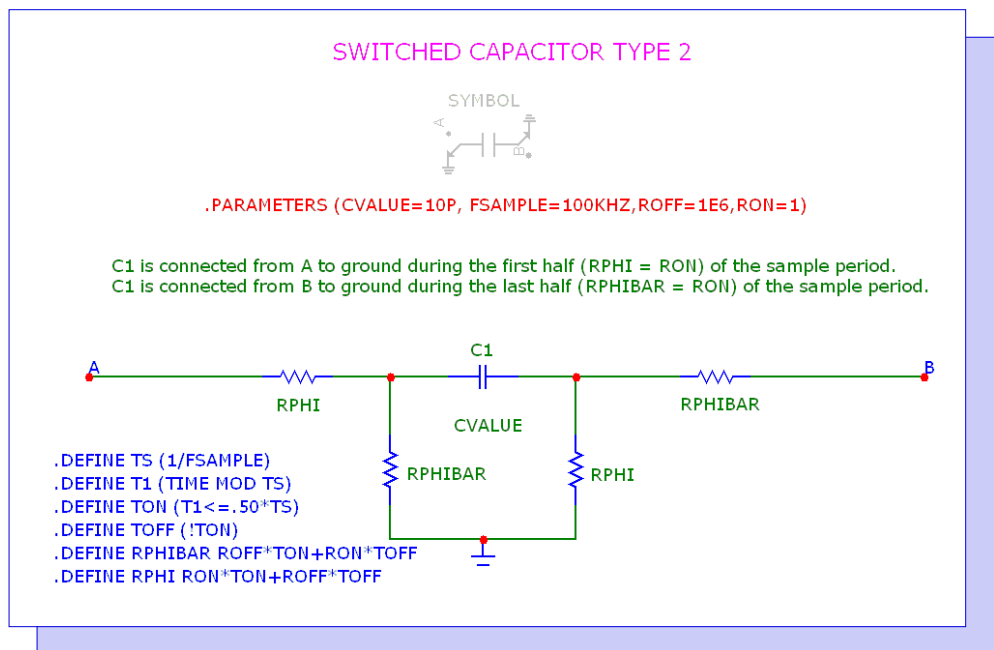


Fig. 10 - The SwitchCap2 macro

Here is a 1MHz 3rd order Elliptic filter that uses these macros.

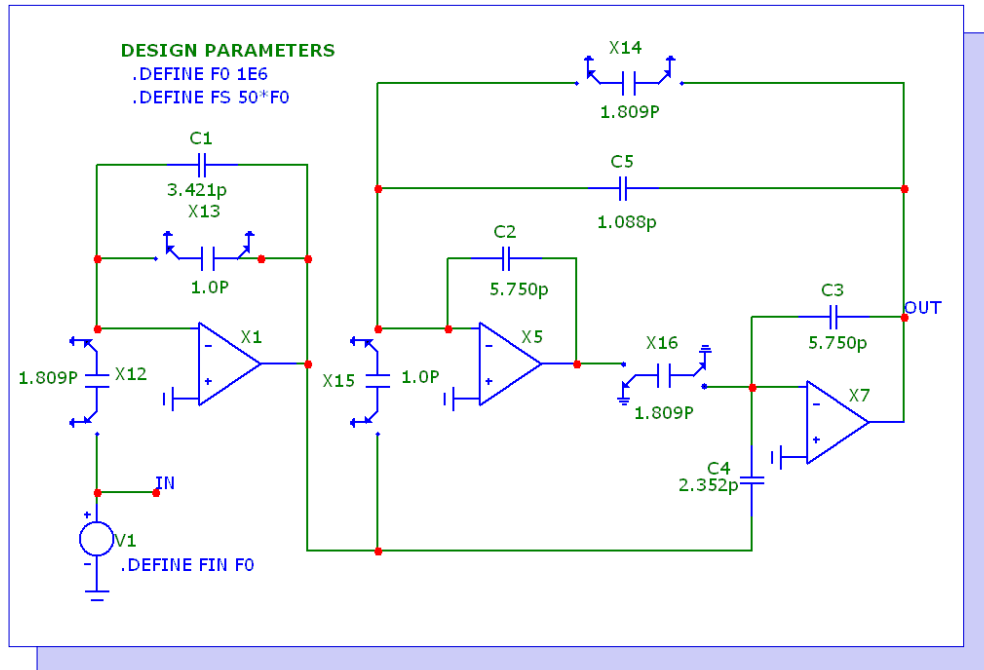


Fig. 11 - LP elliptic filter that uses the SwitchCap and SwitchCap2 macros

Here is its transient analysis and Fourier spectrum.

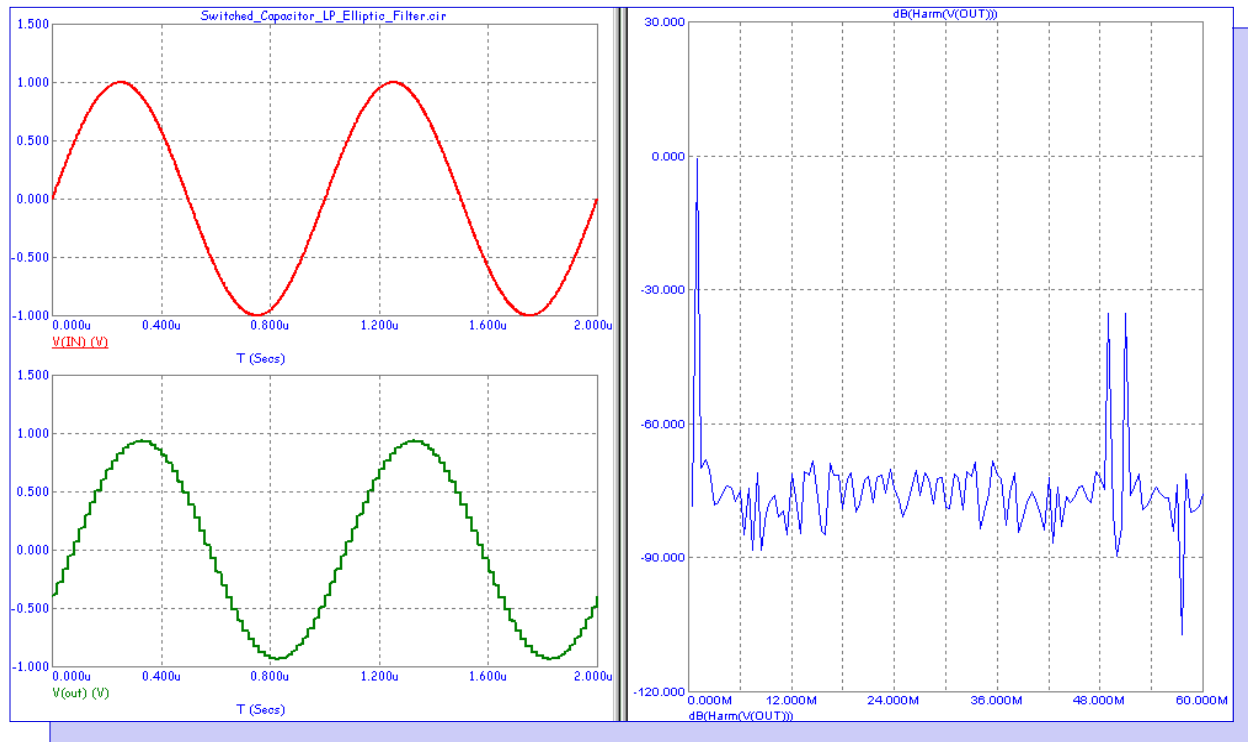


Fig. 12 - Transient analysis of the filter and its Fourier spectrum

Creating Ferrite Bead Models

A ferrite bead is a passive device used to suppress high frequency signals. It is often used to reduce electromagnetic interference (EMI) or radio frequency interference (RFI) signals within a system. A common equivalent circuit of a ferrite bead is a lumped element parallel RLC circuit. This circuit produces a reasonable match for the impedance curve of the ferrite bead. However, a more precise model for the impedance curve can be created by adding a second parallel RLC to the equivalent circuit along with a series resistance. The modified equivalent circuit is shown below.

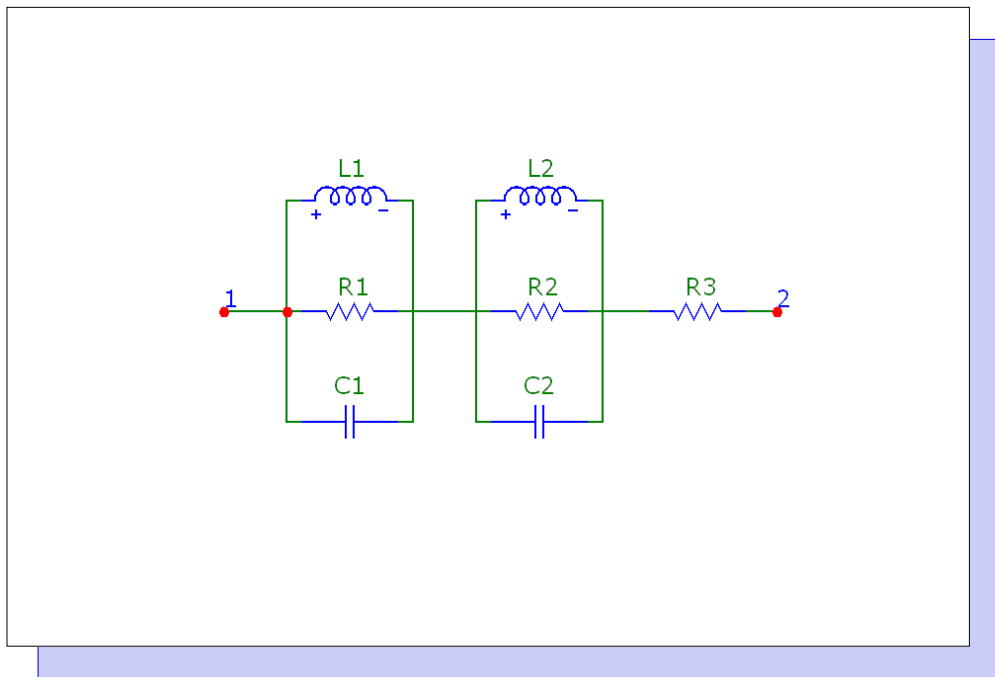


Fig. 13 - Lumped element representation of the ferrite bead

While the equivalent circuit is simple, determining the component values to use to appropriately model a specific ferrite bead can be onerous. Fortunately, the capability of the Model program to optimize any user created model is designed to handle such complexity. The first step is to create a basic template of the ferrite bead model. A subcircuit model that represents the equivalent circuit is defined as:

```
.subckt Bead 1 2
L1 1 3 1u
R1 1 3 10
C1 1 3 1p
R2 3 4 10
L2 3 4 1u
C2 3 4 1p
R3 4 2 1
.ends
```

Either a subcircuit or macro could have been used as the template. The advantage to using a SPICE subcircuit in this instance is that multiple ferrite bead models can then be stored within a single library file.

The next step is to create a schematic that simulates an important measurement of the model. For a ferrite bead, the main specification is the impedance versus frequency characteristic. This characteristic can be easily measured in Micro-Cap through the schematic below.

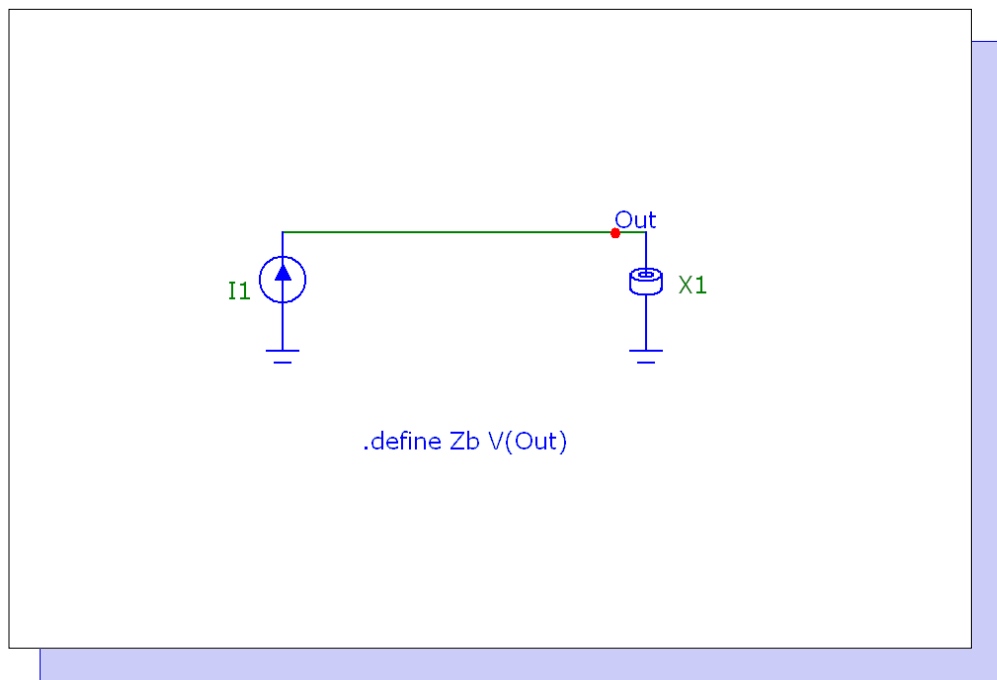


Fig. 14 - Ferrite bead impedance measurement circuit

The circuit consists of just a Current Source component and the ferrite bead subcircuit model. The Current Source component has its AC magnitude parameter set to 1A. Therefore in an AC analysis, the voltage across the ferrite bead will be equivalent to the impedance of the ferrite bead since:

$$\begin{aligned} V &= Z \cdot I \\ I &= 1A \\ V &= Z \end{aligned}$$

Also present in the schematic is the following define statement:

```
.define Zb V(Out)
```

This define variable is used as an alias. It just maps the expression V(Out) to the variable Zb, so that the AC analysis can then plot Zb versus Frequency instead of V(Out) versus Frequency. While both would produce the exact same results, this might help prevent any confusion later when the expression being plotted is used within the Model program.

Next, the analysis limits need to be setup for the AC analysis simulation. For this example, all that needs to be done is to setup the impedance versus frequency expression to be plotted. On a waveform line, the X Expression is set to F, and the Y Expression is set to Zb. The impedance of the ferrite bead model is now ready to be optimized in the Model program.

A new model file is created by selecting New from the Model menu. The New Part dialog box will be invoked automatically which prompts for the device type to specify for the first part in the model file. For a user created part, the part type selected should be User. At this point, the Analysis Type,

Circuit, Waveform, and Graph Title can be specified. The Circuit field should be set to the test circuit that was created, in this case FerriteBead.cir. Next, the Analysis Type field is specified as AC Analysis since that was the analysis type used in the test circuit. In the Waveform field, the expression F vs. Zb is selected. Finally, the Graph Title is defined as Impedance vs. Frequency. This title simply labels the plot in the Model program. The final settings for the dialog box are shown below.

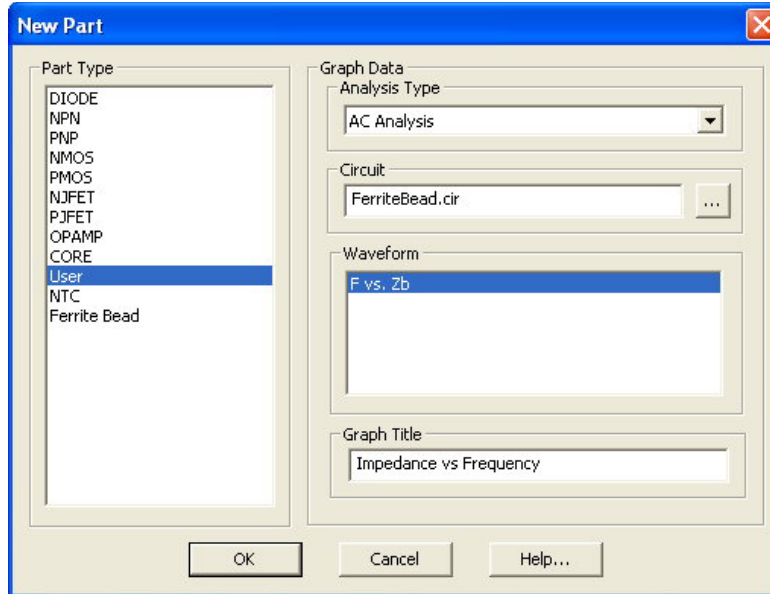


Fig. 15 - New Part dialog box settings for the ferrite bead subcircuit

All of the component values in the ferrite bead subcircuit need to be optimized. In the Parameters section of the model file, the ... buttons can be clicked to add a parameter to be optimized. In the test circuit, the ferrite bead component has the part name X1, so the following component parameters are set to be optimized: X1.R1, X1.C1, X1.L1, X1.R2, X1.C2, X1.L2, and X1.R3.

The MPZ1608S601A ferrite bead from TDK was chosen as the device to model. The input data and results for this device are shown in Figure 16. As can be seen in the figure, the optimized results produce a good match to the input data. Using the Create Model for this Part command under the Model menu, the following subcircuit model is produced.

```
.SUBCKT MPZ1608S601A 1 2
L1 1 3 1.047996547u
R1 1 3 612.3746753794
C1 1 3 798.1766699999f
R2 3 4 302.00781172
L2 3 4 2.246831281u
C2 3 4 4.317152502p
R3 4 2 5.0318715819
.ENDS MPZ1608S601A
```

The Save Part to Template Library command under the Model menu can be used to save the current configuration as a template for easy reuse in optimizing other ferrite bead models.

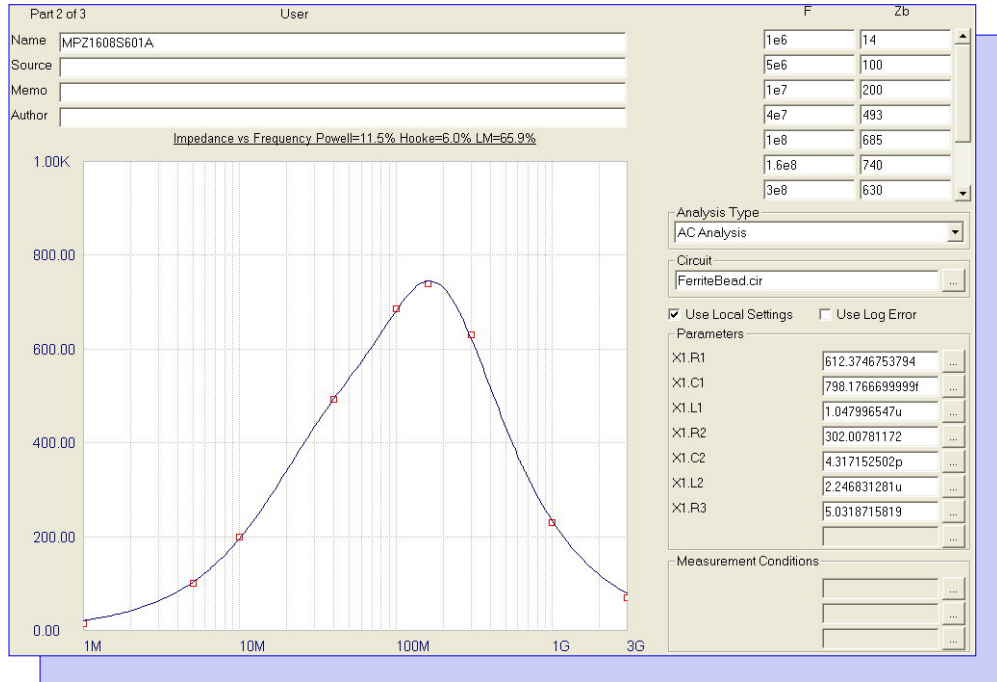


Fig. 16 - MPZ1608S601A ferrite bead optimization

TDK also has a SPICE model available for the MPZ1608S601A device. Their model uses a single parallel RLC with a series resistor. The analysis plot below compares the impedance versus frequency curves for the Micro-Cap and TDK models. The Micro-Cap model provides a much closer fit to the data sheet curve.



Fig. 17 - Micro-Cap model versus TDK model

Importing PWL Data From A File

The piecewise linear capability of the Voltage Source component is useful for defining waveforms that do not adhere to any standard expressions. The PWL function uses time versus voltage data pairs in order to define the waveform of the source. Typically, the data is typed directly into the VALUE attribute of the Voltage Source component. However, if there is a lot of data to be entered, this can be time consuming. Micro-Cap has a File keyword that can be used within the PWL definition which provides a method to retrieve the time versus voltage data directly from a file. The syntax for the File keyword is as follows:

```
FILE="filename"
```

This statement can be typed directly into the PWL text field within the Attribute dialog box of the Voltage Source as shown in the figure below.

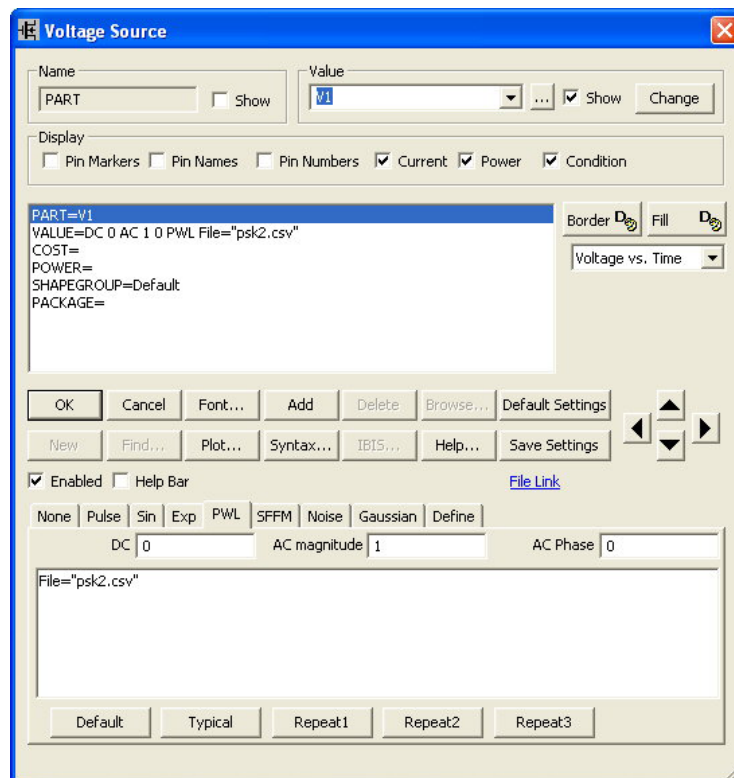


Fig. 18 - PWL file defined for a voltage source

In this example, the PWL data is stored in a file called PSK2.CSV. Since no path has been specified for the file, the PSK2.CSV file would need to be located in the Data path that is specified in Micro-Cap. The Data path can be defined by selecting the Paths command under the File menu and editing the Data field in the Paths dialog box that comes up. This path defaults to the Data folder when Micro-Cap is first installed. A path can also be specified within the File keyword definition such as:

```
FILE="C:\MYFILES\PSK2.CSV"
```

The file being referenced must contain only the time versus voltage data points. Each line in the file should have just a single data point. The PSK2.CSV file contains the following information:

```

0.0000000000000000E+000,-3.6977854932234928E-032
7.9999999999999998E-012,-8.0424771926479972E-005
1.6000000000000000E-011,-1.6084954382044749E-004
3.1999999999999999E-011,-3.2169908738079547E-004
6.3999999999999999E-011,-6.4339817268079501E-004
1.2800000000000000E-010,-1.2867963287152221E-003
...
...
...
7.9958359616359067E-005,-4.1785107382121828E-001
7.9968359616359061E-005,-3.1774869751689055E-001
7.9978359616359056E-005,-2.1744562432477649E-001
7.9988359616359051E-005,-1.1700520781574002E-001
7.9998359616359046E-005,-1.6490888309627784E-002
8.0000000000000007E-005,-1.6490888309627784E-002

```

This file contains 8158 data points which would have been taken an excessively long amount of time to enter manually. CSV files such as these can easily be created in programs such as Excel.

The resulting transient analysis is shown in the figure below. The PWL voltage source is producing 80us of a PSK modulated waveform.

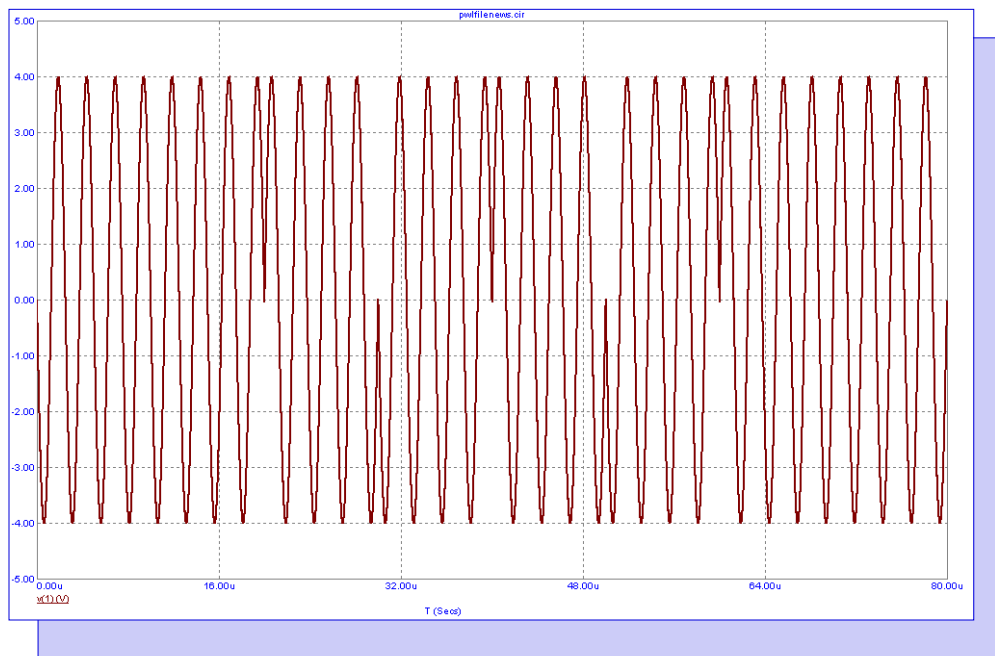


Fig. 19 - PWL file output

One other advantage to using the File keyword is that the waveform data can be modified by third party programs at any time without having to change the circuit in Micro-Cap. Simply run the transient analysis again to view how the modified waveform works with the circuit.

The PWL file feature is also available within the Current Source component.

Product Sheet

Latest Version numbers

Micro-Cap 10Version 10.0.3
Micro-Cap 9Version 9.0.8
Micro-Cap 8Version 8.1.3
Micro-Cap 7Version 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 supportsupport@spectrum-soft.com
Web Sitehttp://www.spectrum-soft.com
User Groupmicro-cap-subscribe@yahoogroups.com