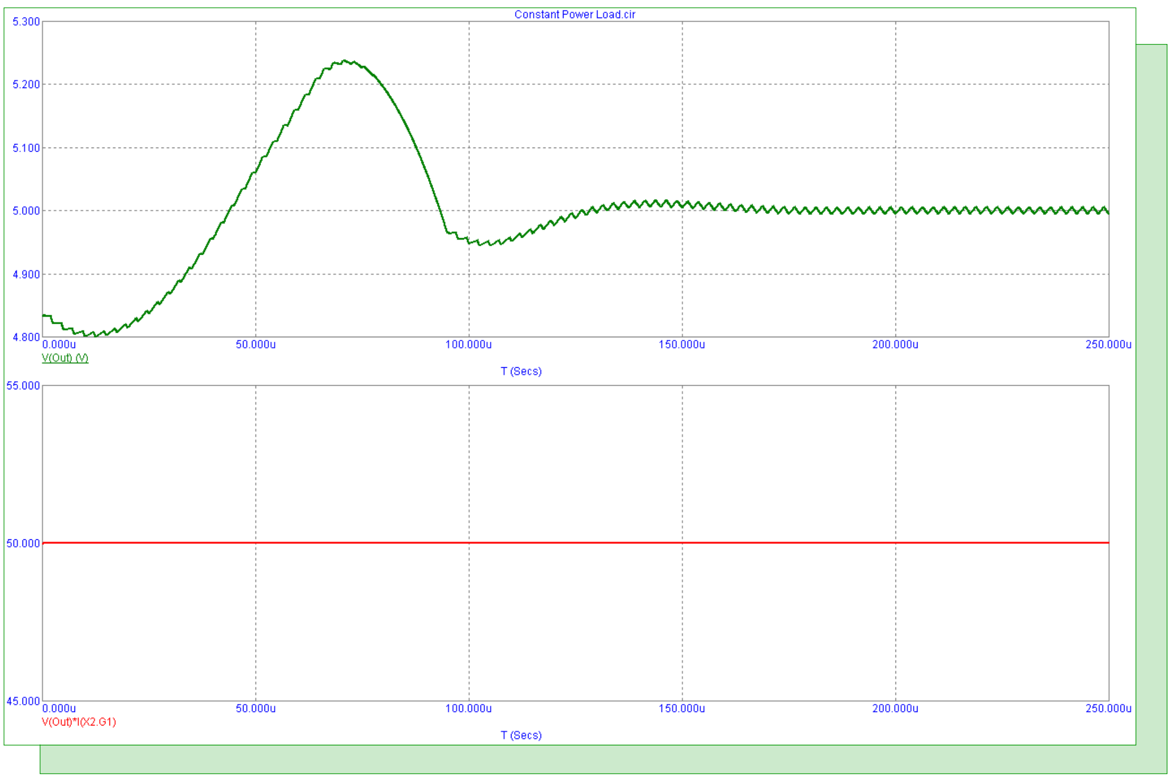
spectrum news

Applications for Micro-Cap™ Users

Spring 2008

News



Constant Power Load Macro

Featuring:

- Constant Power Load Macro
- Adding SPICE Models from Manufacturers
- Plotting Total RMS Noise Voltage

News In Preview

This newsletter's Q and A section describes how to sweep a resistance in DC analysis and how to create a voltage varying resistor. The Easily Overlooked Feature section describes the file linking capability for components that helps in accessing relevant documentation.

The first article describes a constant power load macro. This macro can be used to draw a constant power in a circuit.

The second article describes how to use the Import Wizard to import SPICE models from manufacturers or other sources.

The third article describes a technique for plotting the total RMS noise voltage of a circuit in AC 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: I would like to sweep a resistor in DC analysis so that I can plot the resistance along the X-axis versus my voltage and current plots. How do I do this?

Answer: For a DC analysis sweep you need the MODEL attribute for a resistor defined. Do the following:

1) For the resistor that you want to sweep, double click on that resistor and define the MODEL attribute. For this example, we will use the name RMod. Make sure that the model name is unique to this resistor in the schematic. Set the RESISTANCE attribute value to 1 (I'll explain why in the following step).

2) Go to DC analysis. For Variable 1, choose the appropriate resistor model in the Name attribute from the drop down list. You should see an entry such as RES RMod in the list if the model name was RMod. Set the second field so it specifies the R parameter. In the Range field, enter the range that you would like the resistance swept through. The R parameter is a multiplier and multiplies the Resistance value. That is why we set the Resistance attribute to 1 previously so that we can sweep R with the actual resistance values directly in the Range field.

Question: I need to create a resistor that varies with the voltage of a node in my circuit. How do I model this?

Answer: Simply enter the expression that you want to use into the RESISTANCE attribute of the resistor. For example, if you wanted the resistor to reference the voltage at node Mod, you would use V(Mod) in the expression. You can combine this with other operators or circuit variables to create any number of equations.

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.

File Linking Capability for Components

Each component in the Component library can have a file link associated with it. The link can be to an internet URL or to a file stored on the hard drive. The main use of this feature is to link a data sheet or an application note to a specific device although any type of web page or file can be linked to a component.

The file link for a component is defined within the Link field in the Component Editor. The figure below shows an example file link. In this case, the Peak Detector macro has its link specified as:

http://www.spectrum-soft.com/news/summer2007/peak.shtm

which points at a Spectrum newsletter article describing the operation of the macro. Note that if a file on the hard drive is specified without any path information, Micro-Cap will use the path from the Document field in the Paths dialog box which is available under the File menu.

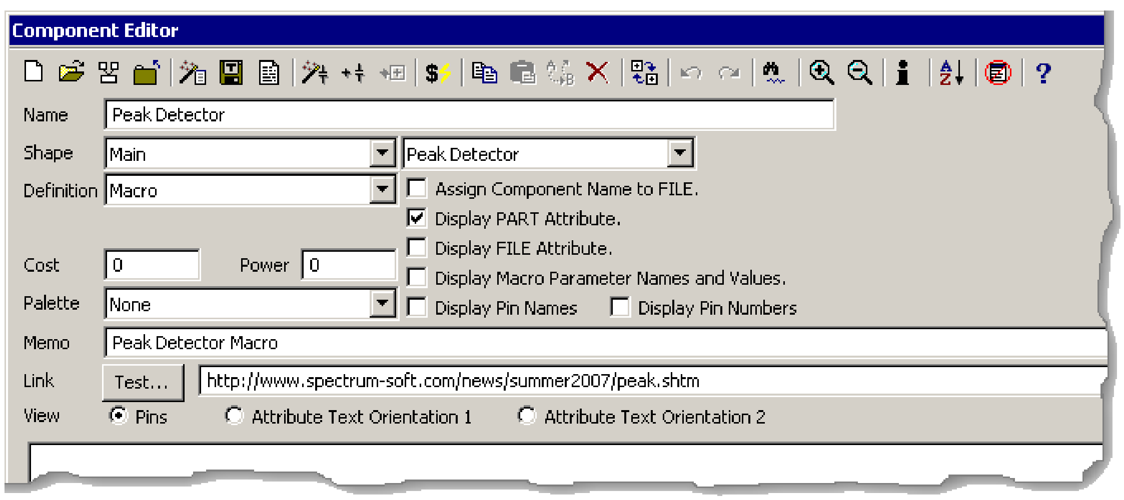


Fig. 1 - Component Editor fields

The file link is globally defined so it is available for all instances of the component. When the component is placed in a schematic, there are two methods to access the linked file. One method is to enable File Link mode which is available under the Options/Mode menu or through the icon:

Then click on the component in the schematic. The associated file will be opened. The second method is to double click on the component when in Select mode to invoke the Attribute dialog box. Below the two rows of buttons is hyperlinked text that states File Link. Clicking on this link will also open up the associated file.

If the component does not have an entry in the Link field in the Component Editor, Micro-Cap will use the default path defined in the File Link Default field in the Preferences / Options / Circuit page. The default link is initially set to perform a Google search on the part name and the phrase "data sheet".

Constant Power Load Macro

In some simulations, it can be useful to have a load that draws a constant power in the circuit. Applications that use a load such as this can be found when simulating mains, SMPS, DC/DC converters, or other power circuits. To model a constant power load, the simplest method is to use the nonlinear function current source (NFI). The obvious equation to define the NFI with would be similar to the following:

I = Power/V(I)

where I is the current produced by the source, Power is the specified constant power, and V(I) is the voltage across the current source. While this equation would work fine in certain configurations, it can often cause convergence issues should the voltage across the current source become small since an excessively large current would be generated. The solution is to adapt the above equation so that when the applied voltage across the load becomes too small, the impedance that the current source represents clips at a minimum value. The macro circuit below shows one technique for modeling the constant power load.

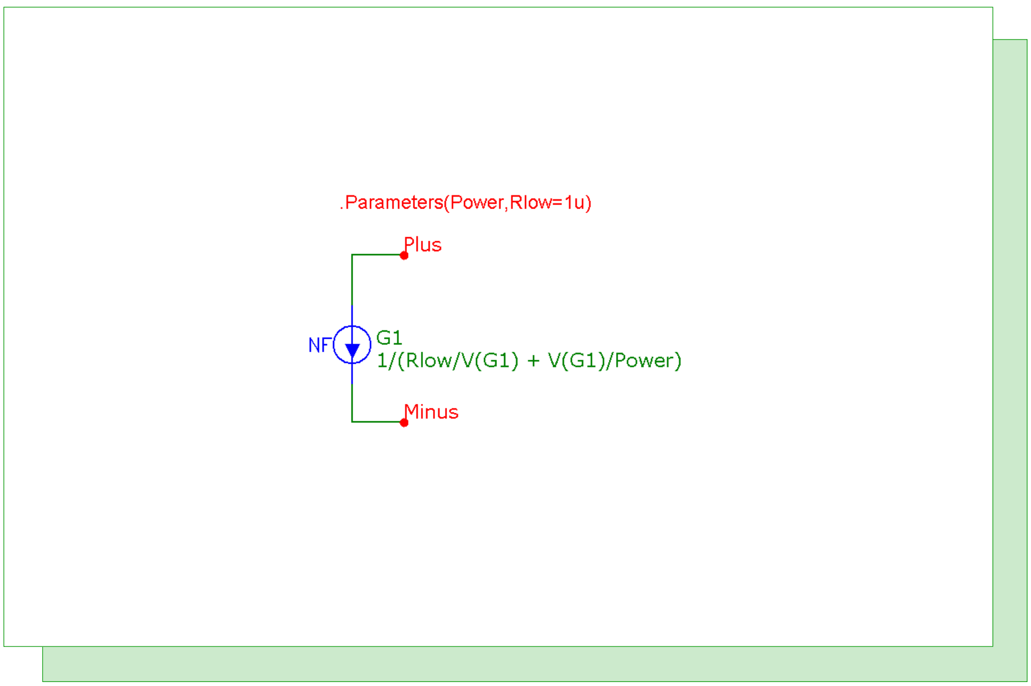


Fig. 2 - Constant Power Load macro

The macro circuit has two parameters that are passed through to it: Power and Rlow. The Power parameter defines the constant power value in Watts that the load will draw. The Rlow parameter specifies the minimum resistance that the load will clip at should the voltage across the load become too small.

The macro circuit consists of just a single NFI source. The VALUE attribute of the source has been defined as:

1/(Rlow/V(G1) + V(G1)/Power)

where V(G1) represents the voltage across the current source. At voltages near zero, the equation reduces to:

I = V(G1)/Rlow

so that the current source represents an impedance with a value of Rlow. At larger voltages, the equation reduces to:

I = Power/V(G1)

which is the basic constant power equation. In this case, the current out of the source will be adjusted depending on the voltage across the load so that a constant power is generated.

The circuit below shows the use of the constant power load macro. The circuit is a current mode buck converter that was derived by Christophe Basso. The constant power load macro is the X2 device at the Out node. The parameters for the macro were defined as:

POWER=50
RLOW=10u

The load will generate a constant 50W in its normal operating range and will clip at 10uohms when the voltage across the load becomes too small.

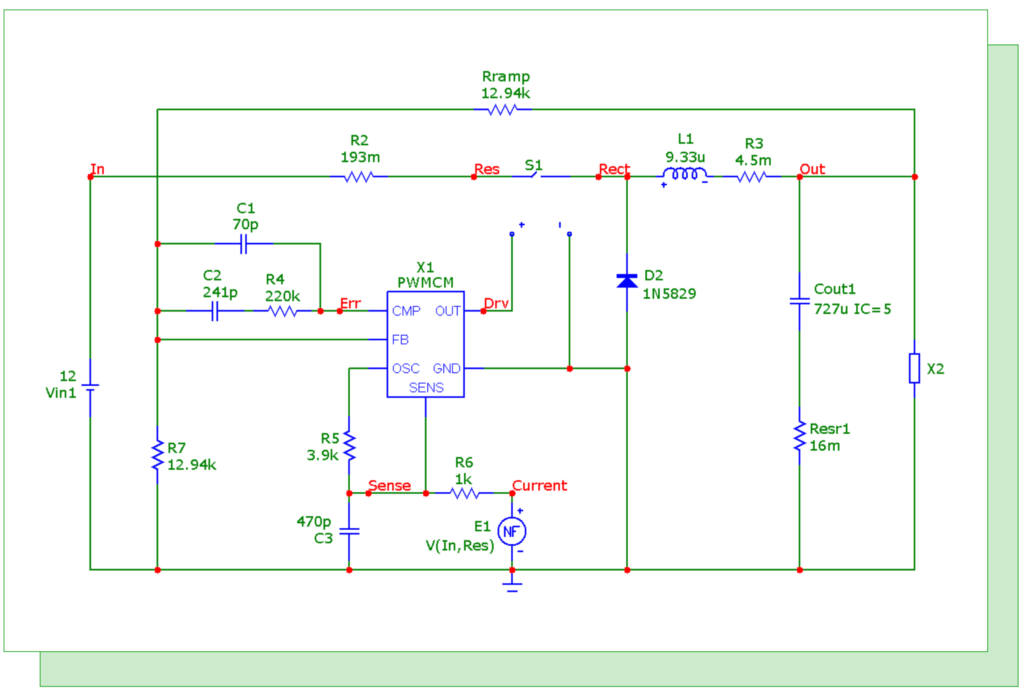


Fig. 3 - Current mode buck converter with constant power load

The transient analysis of this schematic is displayed in Figure 4. The simulation has been run for 250us. Two waveforms have been plotted.

The top waveform displays the voltage at node Out. After the initial transient, the output voltage settles down into the 5V range.

The bottom waveform displays the power dissipated at the load. The expression used to plot the power at the load is:

V(Out)*I(X2.G1)

The X2.G1 entry refers to the G1 current source within the X2 macro which is the constant power load macro. As expected, the waveform shows that a constant 50W was dissipated throughout the simulation.

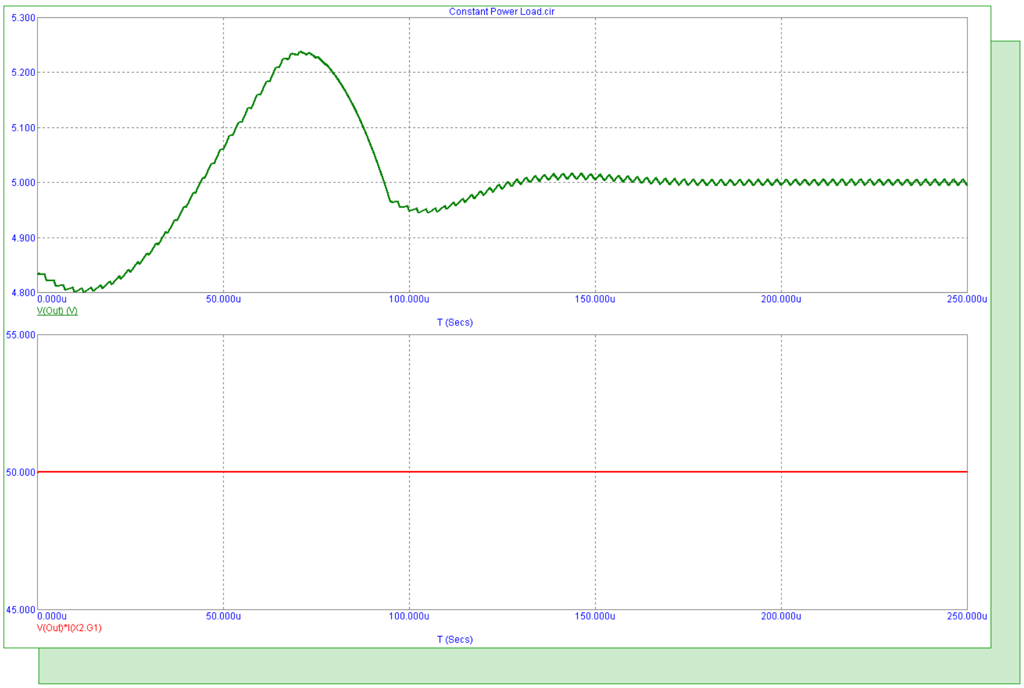


Fig. 4 - Current mode buck converter with constant power load analysis

References

1) "Modeling Constant Power Loads", Microsim Tutorial Including Application Notes and Design Ideas, Microsim, 1995.

2) "Switch-Mode Power Supplies: SPICE Simulations and Practical Designs", Christophe Basso, McGraw Hill, 2008.

Adding SPICE Models from Manufacturers

The websites of manufacturers are often great resources for additional SPICE models that can be used in Micro-Cap. For importing models into Micro-Cap, the Component Editor provides two wizards, the Import Wizard and the Add Part Wizard, or the user may also add the model manually. The Import Wizard provides the simplest route to importing models and will be the method desribed in this article. For information about the other two methods, reference the Working with Subcircuits chapter in the User's Guide. For this article, three models were downloaded from the internet and placed in a text file called Vendor.lib. Typically, a model file downloaded from the internet can be used directly with the Import Wizard. The Vendor.lib file was created to demonstrate the three possible outcomes that can occur when using the Import Wizard. The three models in the Vendor.lib file are:

AD827 (Analog Devices) - High speed, low power opamp
AD8145 (Analog Devices) - High speed, differential receiver
IRFE330 (International Rectifier) - 400V N-Channel MOSFET

The recommended location to place the downloaded file is the LIBRARY folder under the main Micro-Cap folder. The recommended extension of the file is .LIB. Neither of these are requirements though. Once the file is on the hard drive, the first step in importing the models is to access the Component Editor under the Windows menu. The Component Editor manages all of the circuit components used in the schematics.

In the Component Editor, select the group that you want to import these parts into. The component tree on the right hand side of the Component Editor sets the structure of the Component menu in the schematic editor. Double click on a group name to open or close the group. The Add Group icon can also be used to add a new group to the tree. Highlight the group name that the parts should be in. Then invoke the Import Wizard by clicking on the following icon in the Component Editor toolbar.

The first page in the Import Wizard is the File page which is shown in Figure 5. This page specifies the library file that contains the models to be imported. The Browse button lets you browse through the hard drive to locate the library. In addition, an option called "Copy the above file to the library directory" is available. When enabled, this option will copy the library file into the specified path. This option is useful in relocating the library file to the defined library path if it was downloaded into a different folder. Once the library file has been selected, click Next.

The next page in the Import Wizard is the Suffix page which is shown in Figure 6. This page specifies an optional suffix that can be added to all of the subcircuit names within the library file. To add a suffix, enter a string in the "Suffix to Append to SUBCKT names" field and then click the Append button. This will rewrite all of the subcircuit names in the library file to include this suffix. Note that upon clicking Append, the library file is modified and saved to the hard drive. If the wizard operation is later cancelled, the library file will still contain the modified subcircuit names. An appended suffix can be useful when trying to import a model that has the same name as one that already exists in the component library. The Import Wizard only imports models that have a name that does not exist in the component library and adding a suffix will let the wizard import the model. In most cases, no suffix is needed and this page can be ignored as it will be in this example. Click Next.

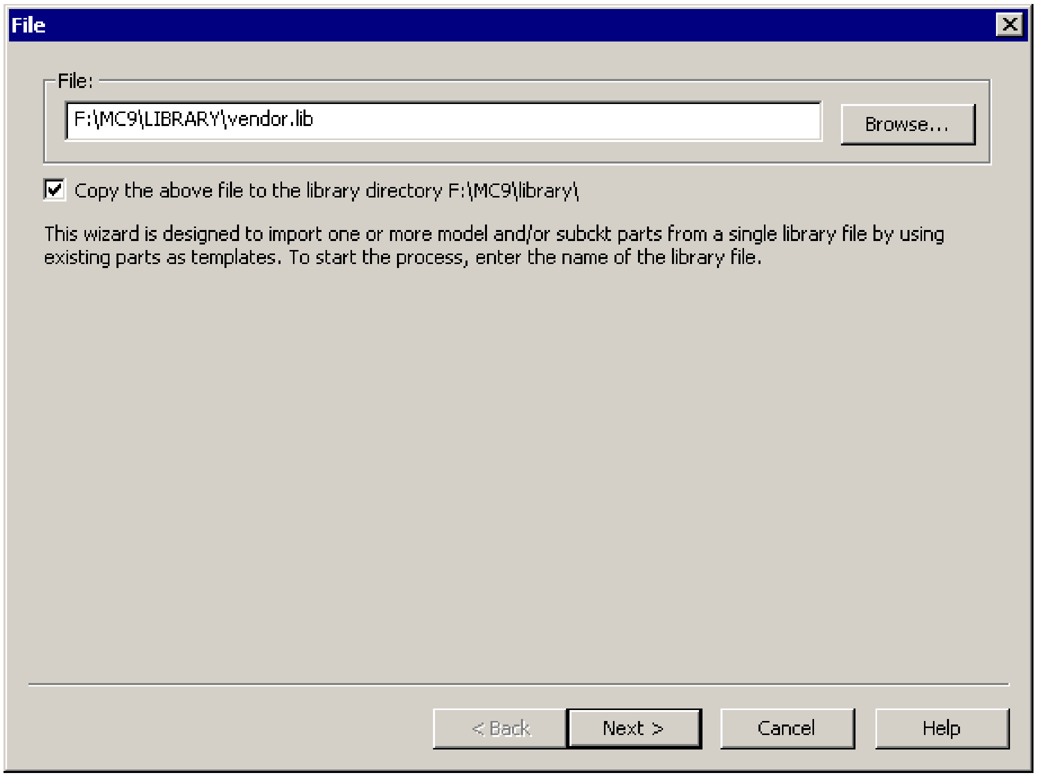


Fig. 5 - File page of the Import Wizard

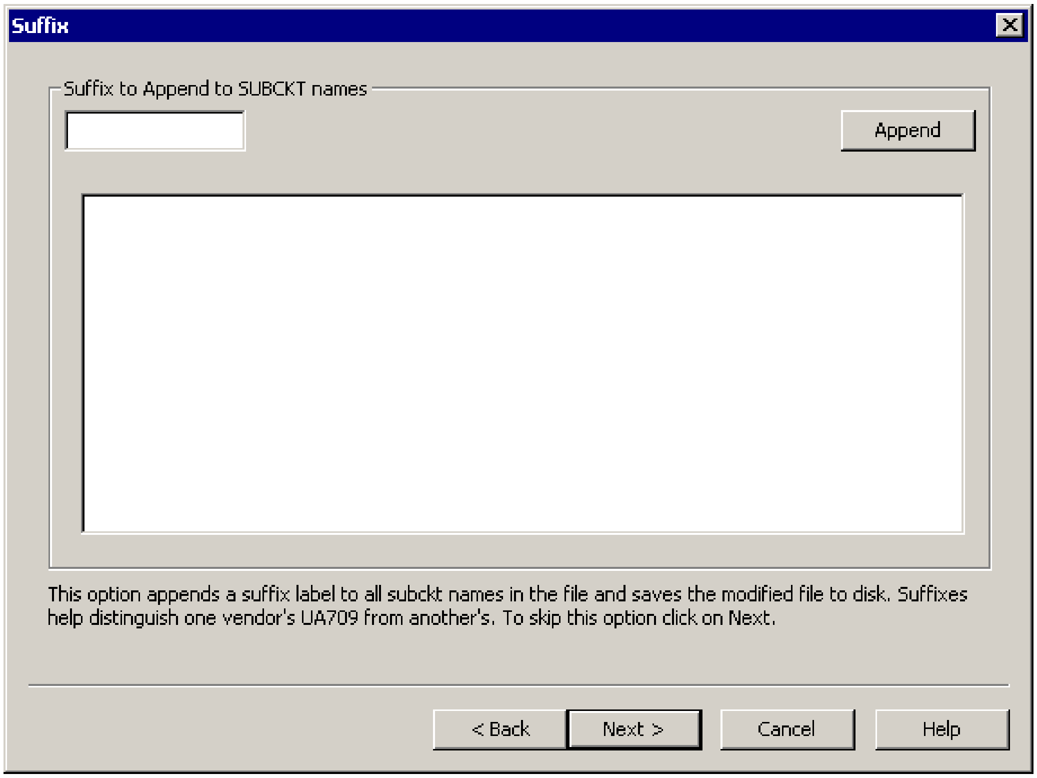


Fig. 6 - Suffix page of the Import Wizard

After the Suffix page, the wizard starts the process of importing all unique models from the library file. Micro-Cap will compare the part(s) in the library to the existing parts in the Component Editor. For subcircuits, it compares both the amount of pins defined within the model along with the names of those pins to all existing subcircuit entries. There are three possibilities that can occur with each model in the library file. If it finds a single match, it will automatically add the part using the template of that match. If it finds no matches, it will place the part in the library with a generic template along with a designation that the part needs more work. If it finds multiple matches, a list of all of the available matches is then shown, and the template that is most applicable to the specified model can then be selected by the user. In addition to the list of matching templates, a second dialog box will also be invoked that displays the netlist of the model being imported. This netlist can often be useful in deciding which of the listed templates provides the best match. In this example, multiple matches are found for the IRFE330 device. The following two dialog boxes are then displayed.

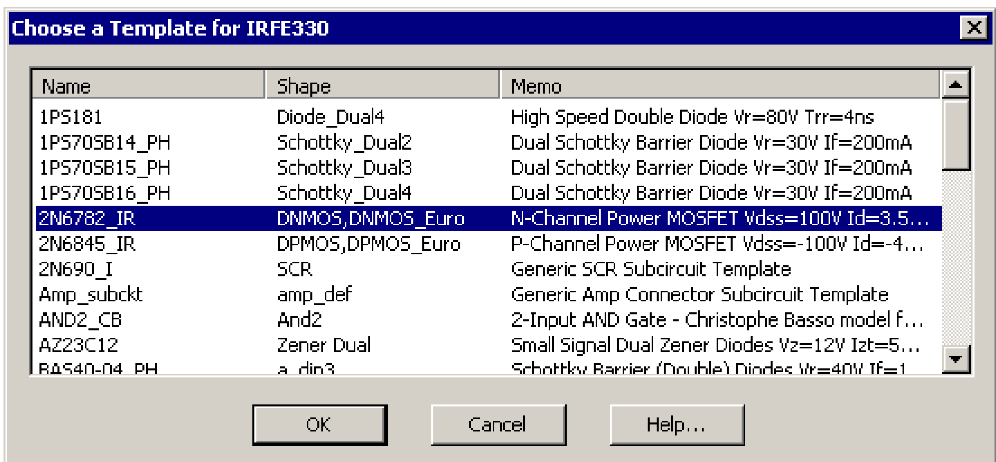


Fig. 7 - Choose a Template dialog box

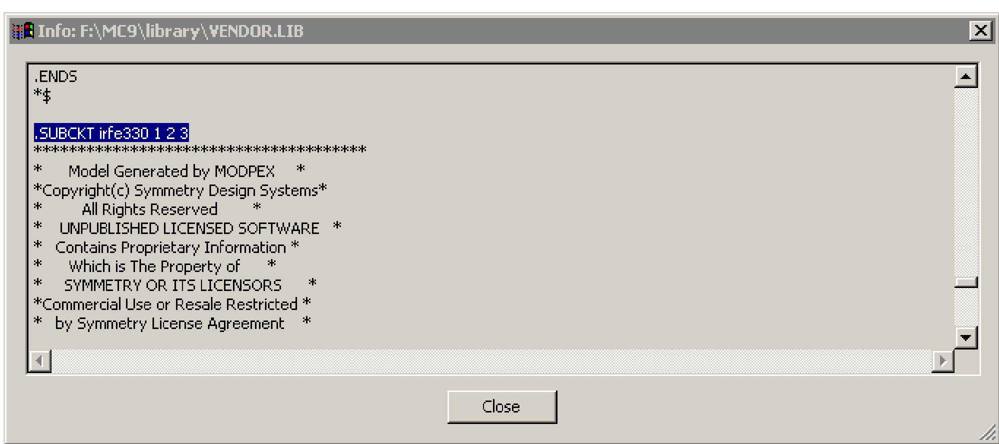


Fig. 8 - Subcircuit listing dialog box

Since the IRFE330 is an N-Channel MOSFET, the template based on the 2N6782 model which uses a DNMOS shape is selected. Clicking OK imports the IRFE330 using the same shapes and pin configuration as the 2N6782 device in the component library. Clicking Cancel will skip this component in the import process.

Once all of the models in the library file have been processed, the Import Status page below will be displayed which lists the results of the Import Wizard operation.

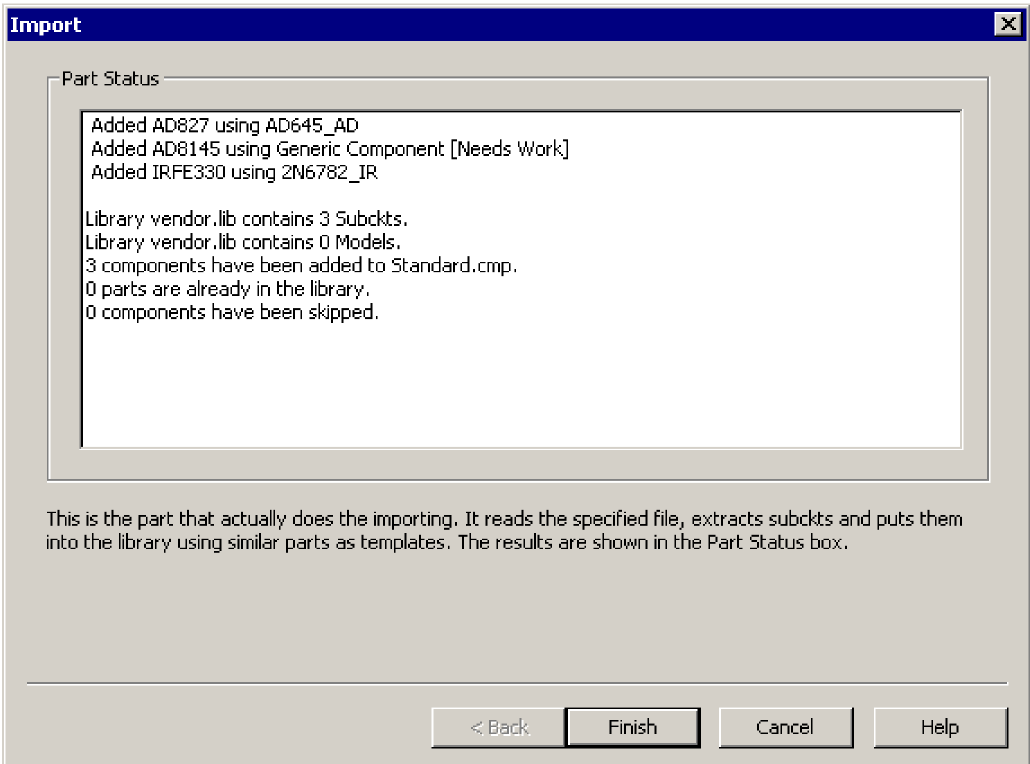


Fig. 9 - Import Status page of the Import Wizard

The top section of the status listing describes the results for each individual model in the library file. For the AD827 model, a single match was found and the part has been imported in using a template defined by the AD645_AD component. For the AD8145 model, no match was found and the part has been imported in using a generic template. Note that this part has been flagged with a Needs Work label. For the IRFE330 model, the part has been imported in using a template defined by the 2N6782_IR component which was selected in a prior step.

The bottom section of the status listing describes the total results for the library file. In this case, three subcircuit models were found in the file, and all three were imported. Click Finish to complete the import process.

The parts will now be available in the Component Editor within the group that was selected when the Import Wizard was launched. Double check the parts to make sure the shape and the pin connections are correct. Highlight a part and then click on the Info icon in the toolbar. This will display the SPICE listing for the device which usually contains comments on the pin functions. Drag on the red dots to move the pin connections around as needed. At this point, you can edit the component just like any others in the Component Editor. For this example, the AD827 and IRFE330 need no additional work. The AD8145, which states Needs Work next to the component name in the tree to the right, needs a shape assigned and pins moved to the appropriate locations. Selecting a shape such as Opamp7d and then dragging the pin connections to the ends of the appropriate leads completes the importation of the AD8145.

Plotting Total RMS Noise Voltage

Circuit designers often need to know the relationship between the signal they are working with and the noise generated by their circuit. Noise analysis is useful in making sure that the expected signal is not buried by the noise that an electrical circuit generates with its resistances and semiconductor devices. One measurement that is useful is the total RMS noise voltage which can also be used to calculate the signal to noise ratio of a circuit. Measuring the total RMS noise voltage is a simple procedure in Micro-Cap. The simple resistor divider shown below will be used to demonstrate how the theory matches the simulation results.

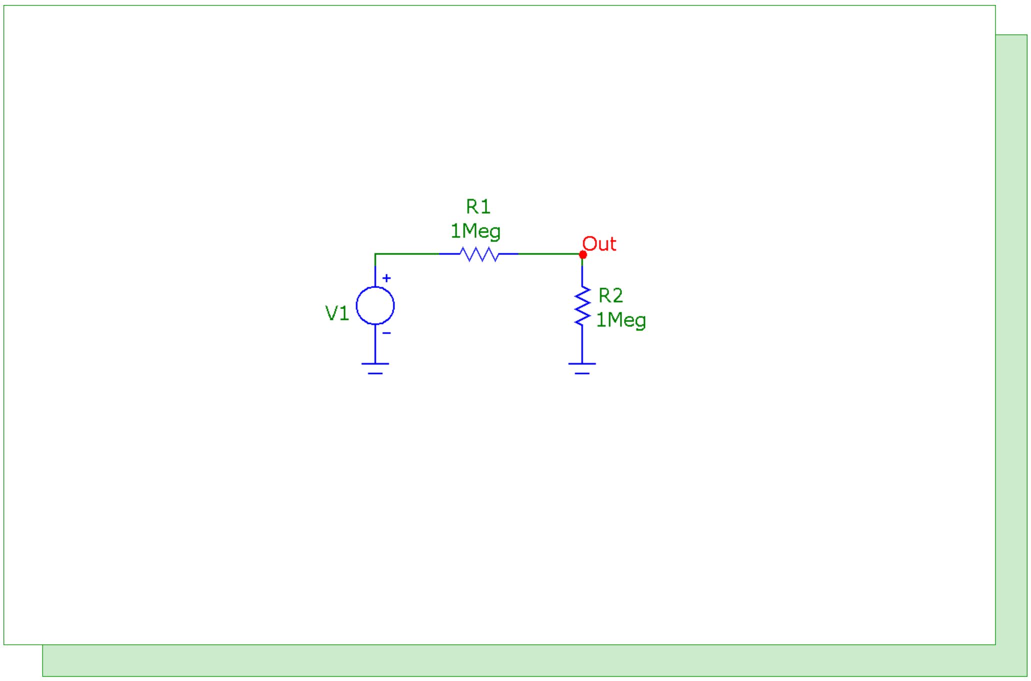


Fig. 10 - Resistor divider circuit

The noise voltage spectral density (thermal noise) of a resistance is calculated through the well known equation:

Sqrt(4*k*T*R)

where k is Boltzmann's Constant, T is the temperature in Kelvin, and R is the resistance. This produces a result in units of volts per root Hertz. To calculate the total RMS noise voltage, the frequency bandwidth needs to be taken into account. The spectral density needs to be integrated over the bandwidth of interest. Since the noise voltage spectral density of a resistance is constant across all frequencies, the integrated equation can be simplified as:

Sqrt(4*k*T*R*B)

where B is the bandwidth. For the resistor divider circuit, the theoretical noise values are calculated using the following values:

k = 1.3807e-23
T = 300.15K (27C)
R = 500k (1Meg // 1Meg)
B = 19980 (20Hz to 20kHz)

The theoretical results for the resistor divider are:

Noise Voltage Spectral Density = 91.04nV/sqrt(Hz)
Total RMS Noise Voltage = 12.87uV

In Micro-Cap, the total RMS noise voltage of a circuit can be plotted using the following expression.

Sqrt(SD(Onoise**2))

Onoise is a Micro-Cap operator which represents the noise voltage spectral density of the circuit at the specified output node. The SD function will integrate the expression with respect to frequency in AC analysis. An AC analysis is run on the resistor divider circuit from 20Hz to 20kHz. The plot is displayed below.

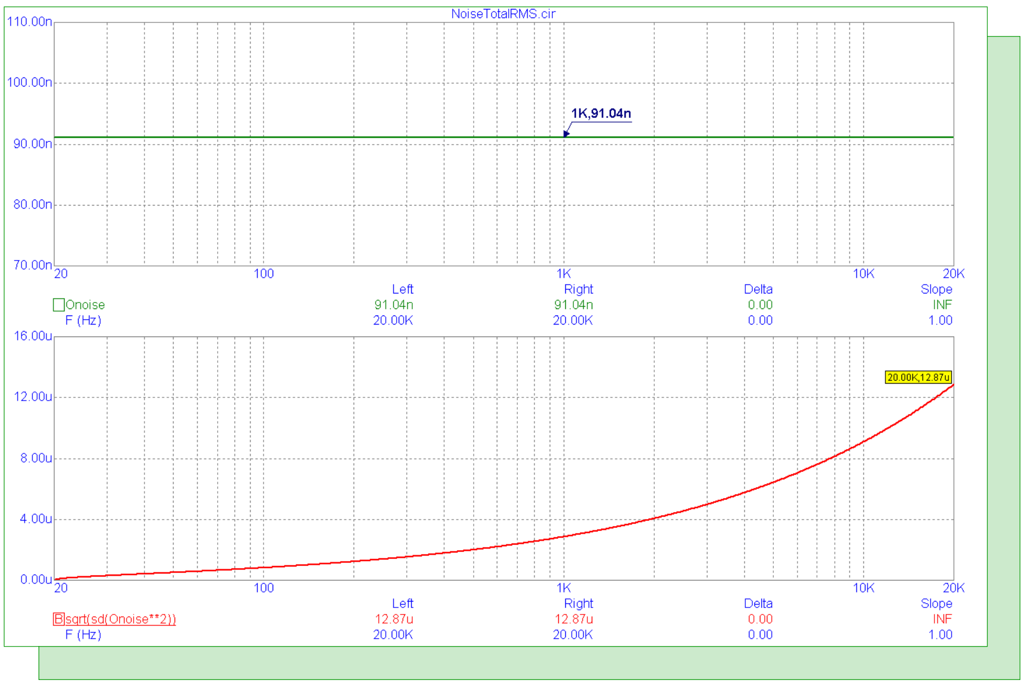


Fig. 11 - Resistor divider noise results

Both the Onoise variable and the Sqrt(SD(Onoise**2)) have been plotted. Note that since the SD function performs a running integral of the expression, only the last value in the plot is important since the value at 20kHz will be the integrated value across the entire simulated bandwidth. As can be seen in the cursor tables of the plot, the simulated values are dead on with the theoretical values.

The resistor divider is a simplified example to show that the mathematics for the total RMS noise voltage expression work in Micro-Cap. This same expression can be used with any type of circuit that includes shot and flicker noise as well as thermal noise. To demonstrate this, the basic audio amplifier (Ref 1) shown in Figure 12 will be simulated.

This audio amplifier is simulated from 1Hz to 100kHz. Both the Onoise and the total RMS noise voltage expressions are plotted. The results are shown in Figure 13. The total RMS noise voltage across the simulated bandwidth for this audio amplifier is 54.968uV as shown in the cursor tables.

Thanks to Sigurd Ruschkowski for helping develop this technique to work with Micro-Cap.

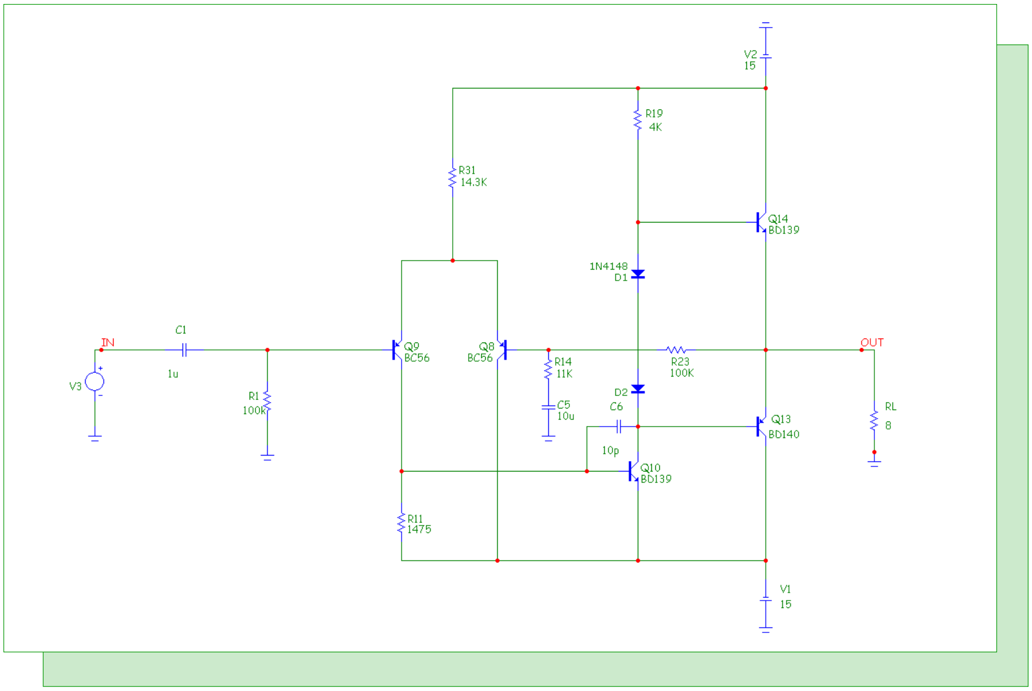


Fig. 12 - Audio amplifier circuit

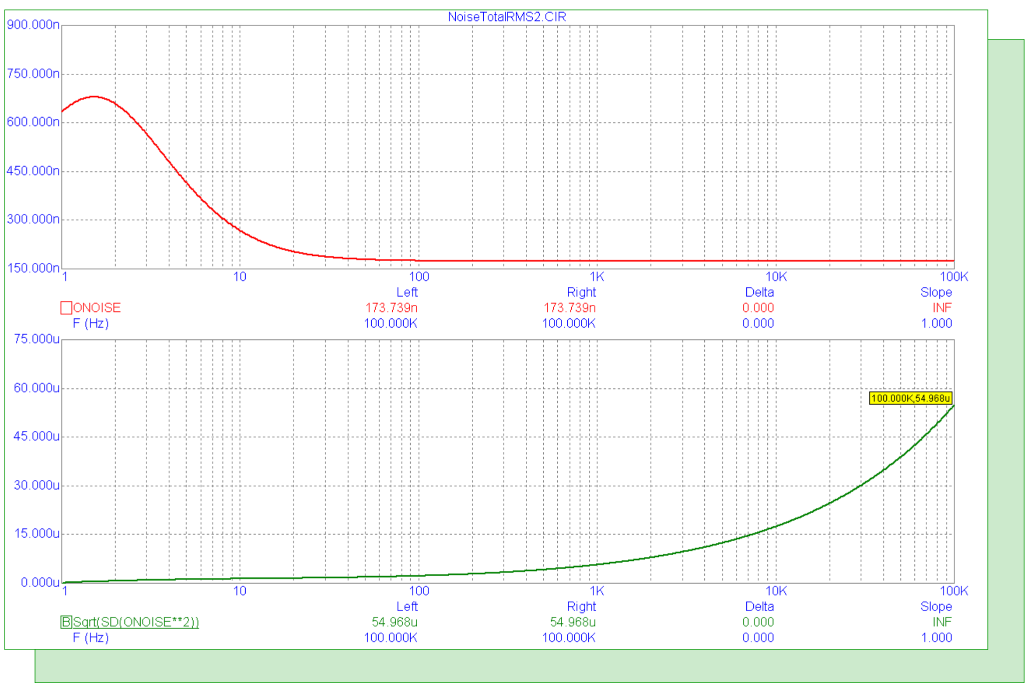


Fig. 13 - Audio amplifier noise results

References

1) "Basic Audio Amplifier" - http://www.ecircuitcenter.com/Circuits_Audio_Amp/Basic_Amplifier/Basic_Audio_Amplifier.htm

Product Sheet

Latest Version numbers

Micro-Cap 9 .... Version 9.0.4
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