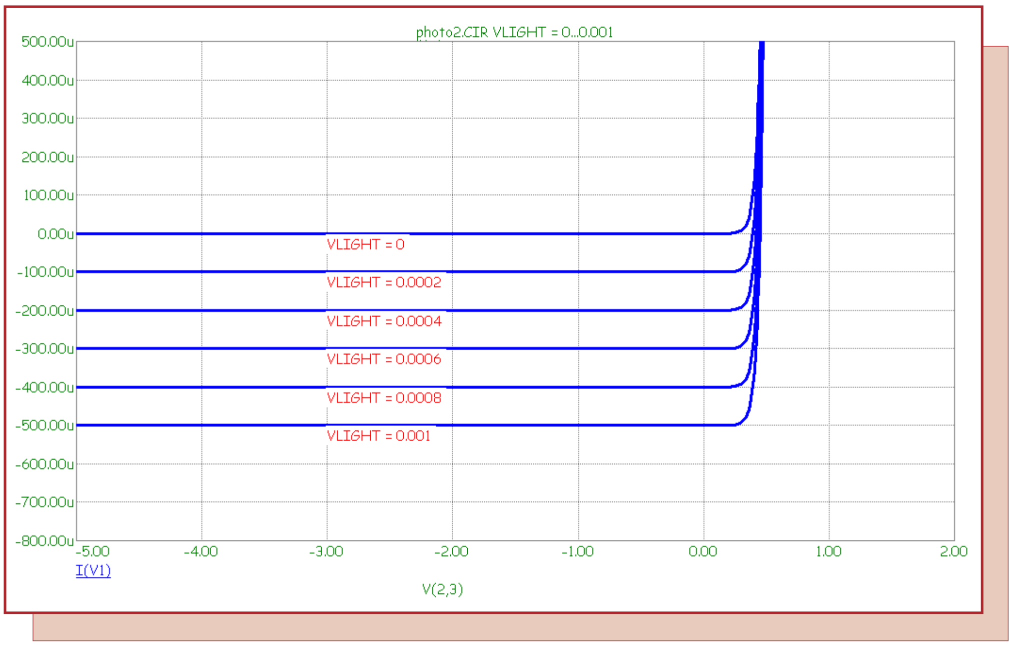
spectrum news

Applications for Micro-Cap™ Users

Summer 2004

News



Photodiode Modeling

Featuring:
- Modeling Photodiodes
- Using FFT Windows
- HSPICE Style MOSFET Binning

News In Preview

This newsletter's Q and A section describes how to enable or disable schematic displays for each component. The Easily Overlooked Features section describes the new process for updating the Micro-Cap 8 executable.

The first article describes how to model a photodiode. There are two models described. One uses the IDark parameter and the other uses the RShunt parameter.

The second article describes the use of FFT windows which create a simpler method of displaying the basic FFT waveforms.

The third article describes the use of the HSPICE style MOSFET binning which uses different model statements to adjust parameters for different length and width values.

Contents

Book Recommendations

General SPICE

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

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

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

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

MOSFET Modeling

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

VLSI Design

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

Micro-Cap - German

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

Micro-Cap - Finnish

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

Design

• *Microelectronic 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: I'm running a Dynamic DC analysis on my schematic. When I enable multiple displays such as voltage, current, and power, the schematic can start to get cluttered. Is there a way to suppress some of the display elements?

Answer: The voltage displays can not be suppressed. These are assigned to the individual nodes and can not be disabled. However, the current, power, and condition displays are all assigned to individual components and may be enabled/disabled within the Attribute dialog box. Double click on a component while in Select mode and the Attribute dialog box will be invoked. Near the top of the dialog box there is a Display section. In this section, there are Current, Power, and Condition options which determine whether the corresponding schematic display for the component will appear on the schematic when that display type is called for. These options control the display status for both Dynamic DC and Dynamic AC analyses.

There is a method available for changing the display settings of multiple components of the same type easily. For example, if you want the power display to be disabled for all batteries, double click on one of the batteries. Disable the Power option in the Display section. Hit OK. With the battery still selected, go to the Edit menu and select Change. In the Change menu, there is a command that will appear similar to:

Apply Display Properties of part_name to all part_types...

where part_name is the PART attribute of the selected component and part_types is the component type of the selected component. This command invokes a dialog box that lets you copy the settings of the component's Attribute display properties, color and font, and/or Display options to all other components of the same part type. In this case, only the Display options would need to be enabled. When OK is clicked, the Display settings of the selected battery would be copied to all other batteries in the schematic. This command does copy the settings for all six available display options to every other battery, so you do want to be careful if you have customized individual components already.

Easily Overlooked Features

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

Check for the Latest Micro-Cap 8 Updates
In Micro-Cap 8, a new method has been implemented for updating the Micro-Cap program file that is much easier than previous version updates. The first step is to make sure that you are online. You will not be able to update without an active online connection. Next, go to the Help menu within Micro-Cap and choose the Check for Updates option. The dialog box in Figure 1 will appear.

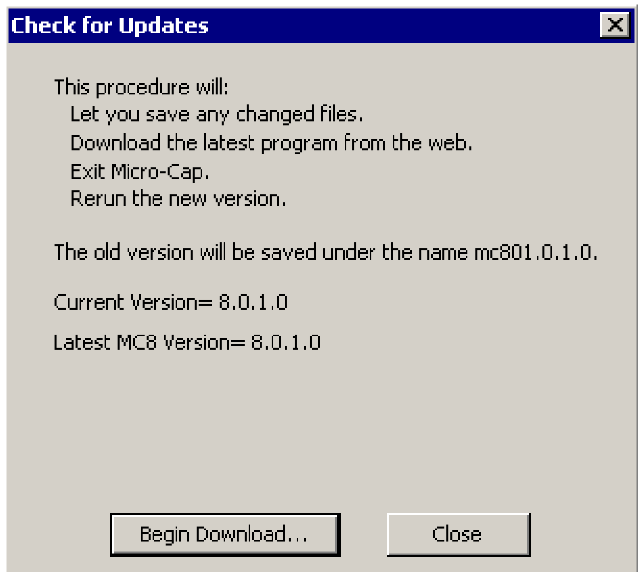


Fig. 1 - Check for Updates dialog box

The dialog box will display the current version of MC8 that you have along with the latest Micro-Cap version available. If you do not have the latest version, simply click the Begin Download button to start the update process. The update process consists of the following:

1) Prompts the user to save any loaded files.
2) Downloads the new version.
3) Exits the current version and backs up the old MC8.EXE file under a different name so the user can go back to it.
4) Starts the new version of Micro-Cap.

That is all it takes to be up and running the latest version.

Modeling Photodiodes

Photo diodes convert light energy to electrical energy. They are used in a variety of applications, including optoelectronic communications, isolation, and motion detection and control. They are particularly valuable where it is necessary to isolate the incoming signal from the circuitry that uses it.

A photo diode is constructed in the usual way, by doping semiconductor material to form a pn junction. The semiconductor materials commonly used are silicon, indium, aluminum, gallium, and arsenic. Depending upon the process and materials used, photodetectors can be optimized for any part of the spectrum from ultraviolet to infrared. The diode is packaged in such a way as to focus the incoming light through a lens, and occasionally through an optical filter, directly on the diode junction. The photons in the incident light break covalent bonds in the depletion region, producing electron-hole pairs that are swept across the junction by the electric field imposed by the external circuitry. The resulting current is proportional to the intensity of the incoming light. The result is an amplifier with an almost infinite impedance separating the input from the output. There is no Miller capacitance to degrade performance or cause cross-channel interference.

How do we model a photo diode? It is pretty simple and looks like this:

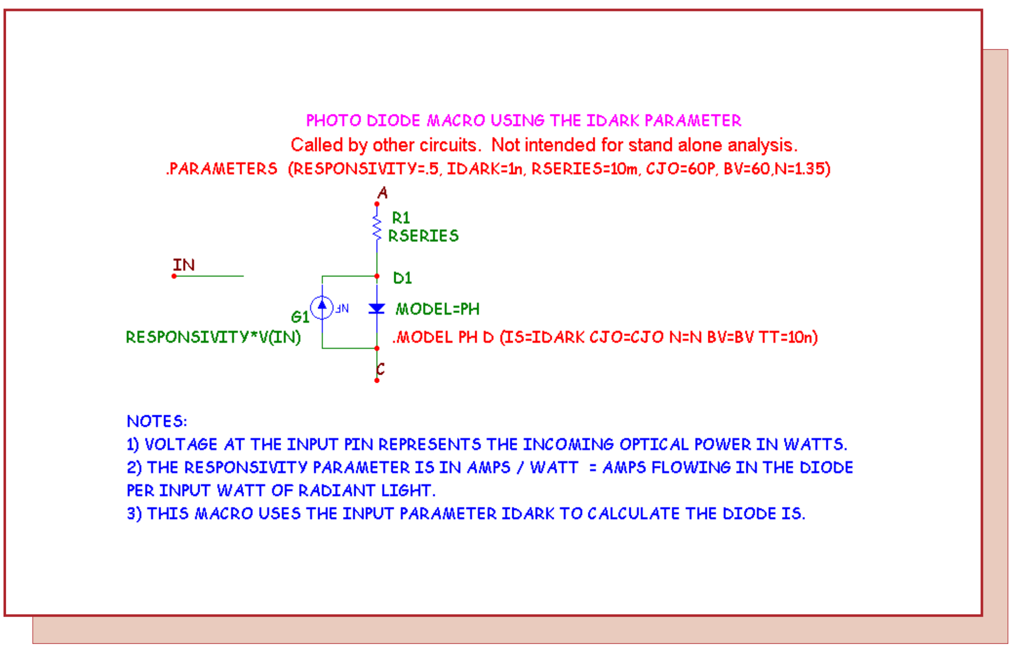


Fig. 2 - Photodiode macro using the IDark parameter

The model consists of a diode paralleled by a voltage-controlled current source, G1. The source creates the light current in the junction under control of the voltage at the IN pin. The voltage on this pin is an analog for the optical power in watts incident on the detector. The model uses six parameters:

RESPONSIVITY - This is the sensitivity expressed in units of amps per watt. Sometimes it is expressed as current for a given input power density, usually mW/cm^2. The exact units do not matter so long as the control voltage pin produces the expected current, through the relationship I = RESPONSIVITY*V(IN).

IDARK - This is the current that flows across the junction when the diode is reverse biased and there is no illumination. In this model, the diode IS value parameter is set equal to the IDARK value.

RSERIES - This is the value of a resistor in series with the diode junction.

CJO - This is the junction capacitance of the photo diode. This primarily affects the switching time of the diode.

BV - This is the junction breakdown voltage. This determines the largest usable operating voltage for the diode.

N - This is the emission coefficient. It mainly affects the slope of the dark current vs. temperature curve. The default value produces a doubling of dark current for every 8 degrees C of temperature rise.

Here is an alternative model. It uses the same parameters as the first model except that the RSHUNT resistance is used instead of IDARK to calculate the diode IS parameter. The RSHUNT resistance is the incremental resistance of the diode measured near zero bias.

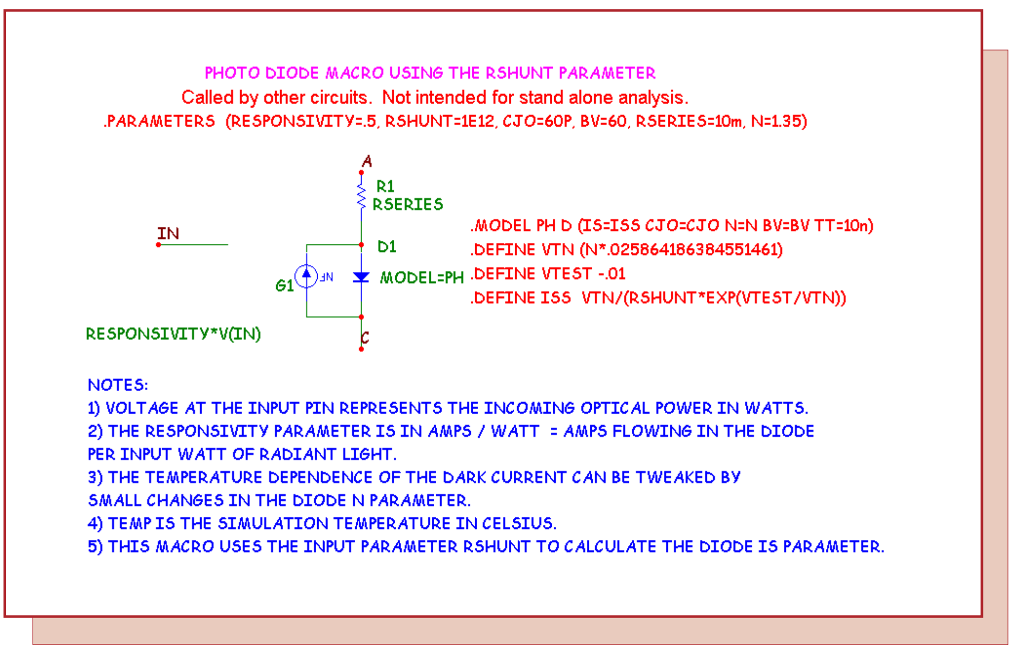


Fig. 3 - Photodiode macro using the RShunt parameter

Some manufacturers provide both RSHUNT and IDARK values. If the photo diode is to be operated in photoconductive mode, where the diode is reverse biased, then use the first model since accuracy of the IDARK value is more important.

If the photo diode is to be operated in photovoltaic mode, where the diode operates near zero bias, then use the second model since accuracy of the RSHUNT value is more important.

Here is a typical circuit operating in photoconductive mode. It shows how the photo diode can be used to amplify and isolate the input from the output.

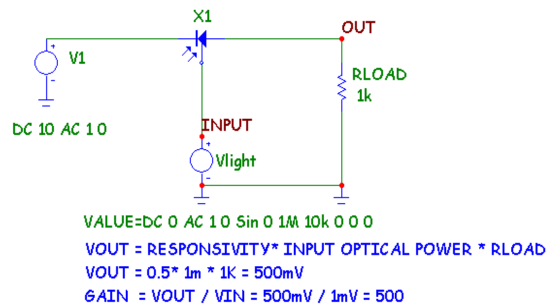


Fig. 4 - Photodiode macro used in photoconductive mode

The photo diode is reverse biased by the V1 source. The incoming light is represented by the source Vlight. Note that the wire connecting Vlight to the control pin represents a communications channel that is usually just air.

The transient analysis of the circuit showing the 1mW input optical power waveform and the output voltage waveform across the load RLOAD is displayed in Figure 5.

The AC analysis of the circuit showing the small signal magnitude and phase angle of the output voltage, again across the load RLOAD is displayed in Figure 6. Both analyses show the output small signal voltage to be:

VOUT = RESPONSIVITY* INPUT OPTICAL POWER * RLOAD
VOUT = 0.5 * 1m * 1K = 500mV
GAIN = VOUT / VIN = 500 / 1 = 500

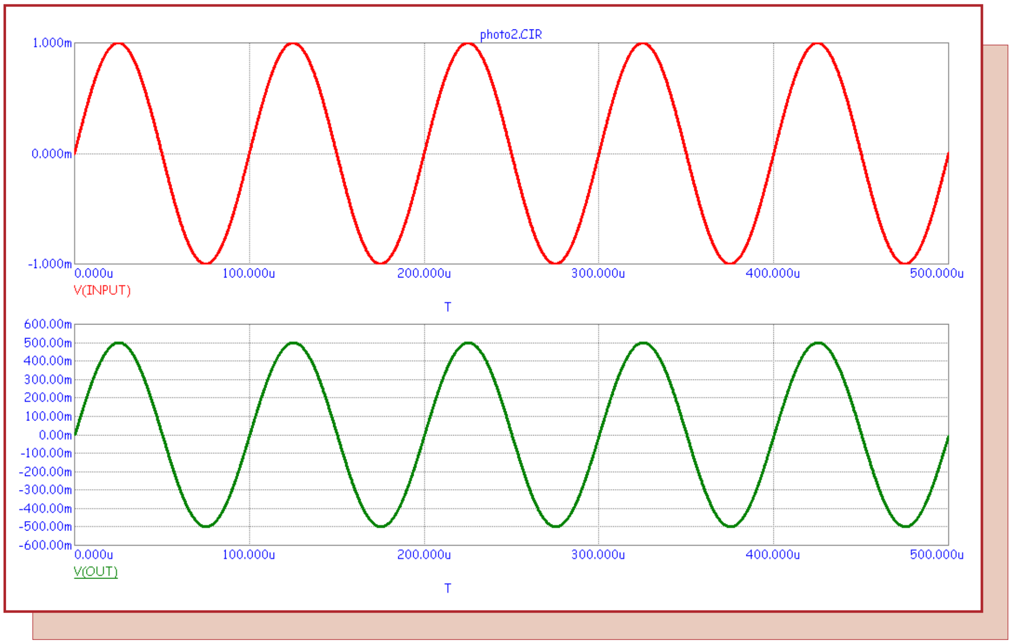


Fig. 5 - Transient analysis of the photodiode example circuit

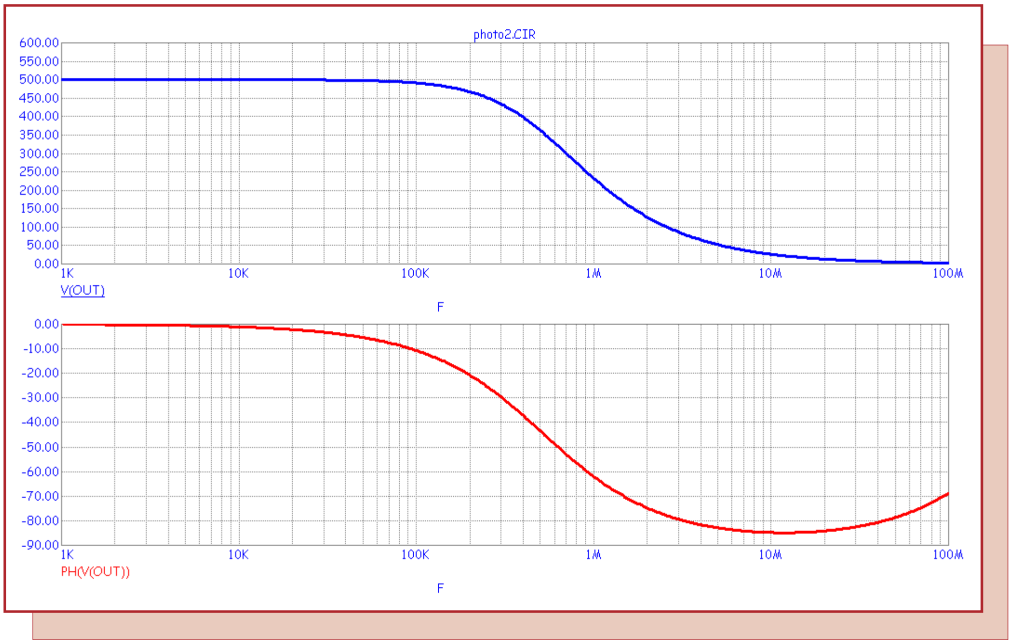


Fig. 6 - AC analysis of the photodiode example circuit

Here is the DC analysis of the circuit IV characteristics of the photo diode. This is a plot of the diode current as measured through the source V1. Each curve corresponds to a specific value of incident optical power ranging from 0 to 1mW.

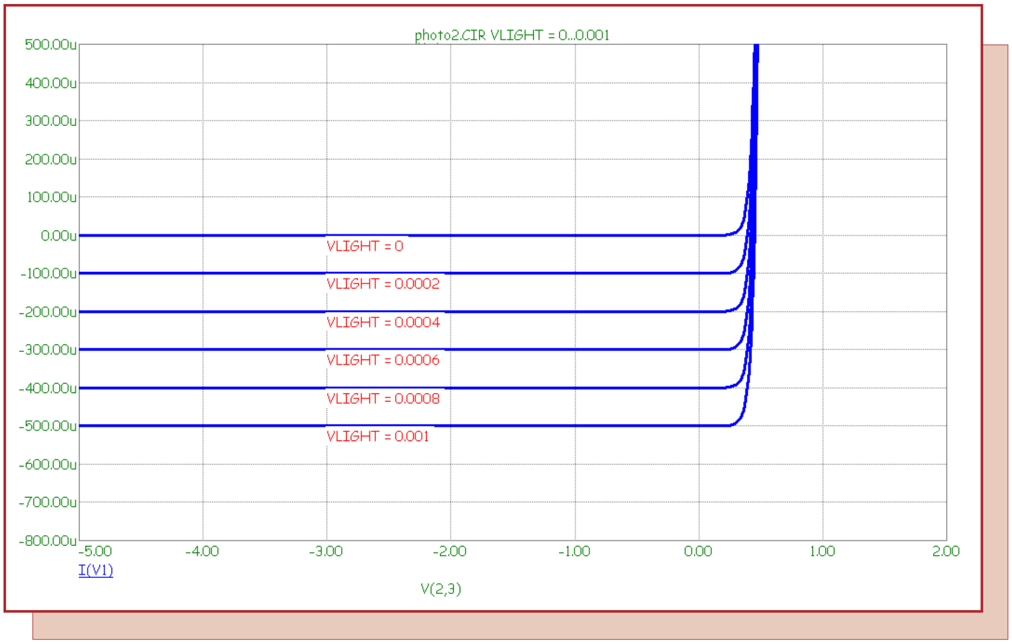


Fig. 7 - DC analysis of the photodiode example circuit

Here are some guidelines for obtaining the photo diode parameters from manufacturer specifications for the two models:

RESPONSIVITY - This parameter is usually specified but if not it can be estimated at about 0.5 to 0.6 for wavelengths of 900nm. For blue wavelengths use about 0.3. For UV wavelengths use a value of about 0.2. If you are unsure, simply use a value of 1.0 and then adjust the input voltage value to produce the expected photo current.

IDARK - The dark current is nearly always specified and is usually in the range of tens of pA for UV diodes to as much as several uA for low cost silicon types.

RSHUNT - This parameter is usually specified but if not it can be estimated at 1E7. It is usually specified in the range of 1E6 to 1E9.

RSERIES - If this parameter is not given it can be estimated at 5m. It is usually in the range of 1m to 20m.

CJO - CJO is usually given and ranges from 1E-11 to 1E-9. Use the value for 0V of reverse bias if more than one capacitance value is given. Sometimes the rise time is given instead. You can roughly estimate the CJO value from the formula CJO = TRISE / RLOAD.

BV - The junction breakdown voltage is usually specified. If not use the default value.

N - The default 1.35 N value can be adjusted to match Idark vs. temperature curves.

Using FFT Windows

The FFT Windows feature provides a simple method that can be used in order to plot the basic FFT functions. It helps to automate many of the choices required to produce a Fourier harmonic content plot. These windows can display the harmonics (in magnitude or in dB), the phase of the harmonics, and the real and imaginary components of the harmonics. In previous versions of Micro-Cap, if one of these functions was to be plotted, it had to be specified before the simulation was run. With the FFT Windows, any waveform that has been plotted in the simulation can have its harmonic contents derived after the simulation has finished. The FFT Windows are simple to use as can be seen in the following example.

The analysis in Figure 8 is a 500us transient simulation of an operational amplifier modelled at the transistor level. The input to the opamp is a 10mV, 10kHz sine wave. The opamp has been configured to have a gain of 100 although at its 10kHz operating frequency, the gain is reduced to 55. Both the input and output waveforms of the amplifier are being plotted.

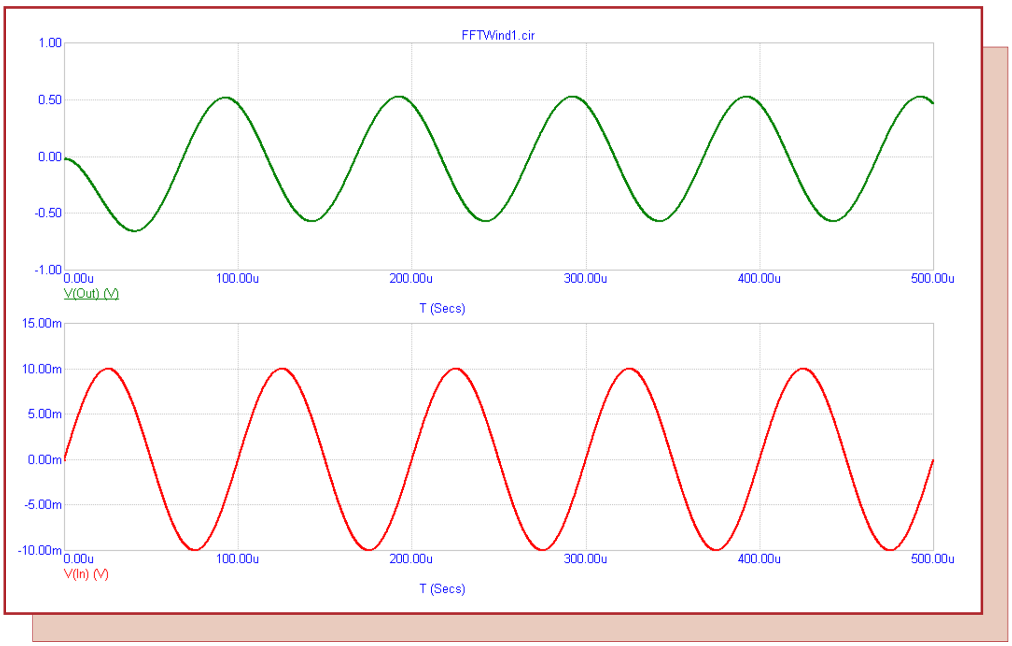


Fig. 8 - Standard Transient Analysis Simulation

In order to invoke the FFT Windows, you can either right click on the name of the waveform in the analysis plot and select Add FFT Window, or you can go to the Transient menu, click on FFT Windows, and select Add FFT Window. When either one of these actions is done, a Properties dialog box will be invoked that controls the display of an FFT Window. The Plot page of the FFT Window Properties dialog box is displayed in Figure 9. This page controls which waveforms will be plotted in the window. The default waveform is the harmonics of the selected curve which is V(Out) in this instance. If we also want to plot the phase of the harmonics in the same window, we would just need to add another expression. Click on the Add button to add a new expression. Change the What To Plot field so that Phase is selected. V(Out) will be the target expression for this function also. Next, change the Plot Group for the phase expression to 2 so it will be in a separate group from the harmonics expression. Click OK to create the FFT Window.

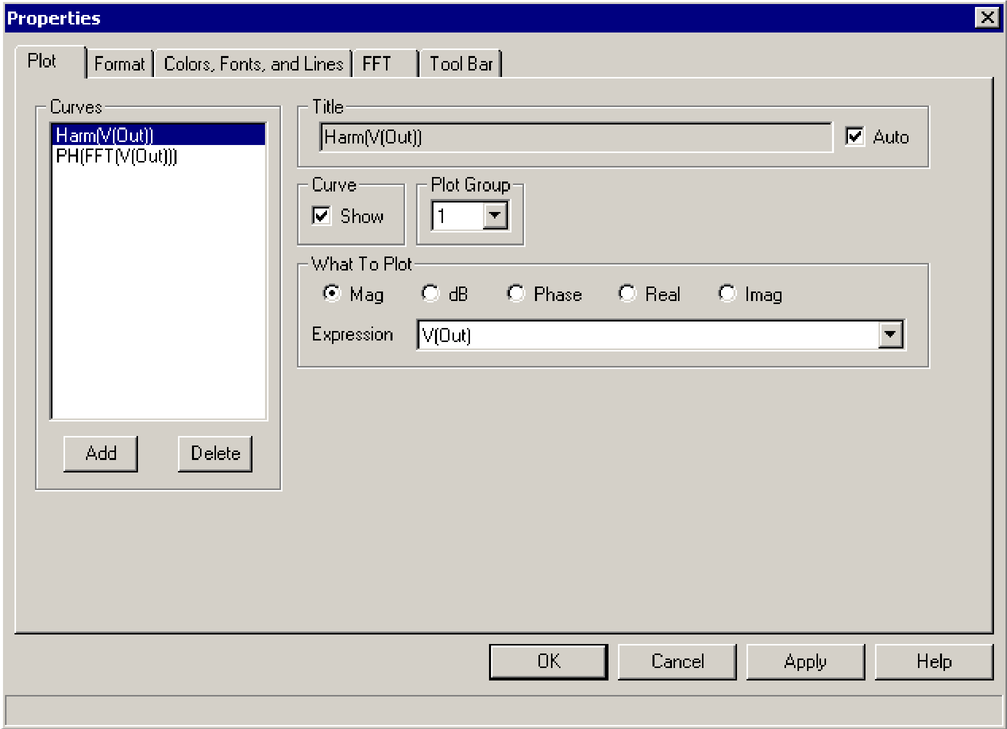


Fig. 9 - Plot Page of the FFT Window Properties dialog box

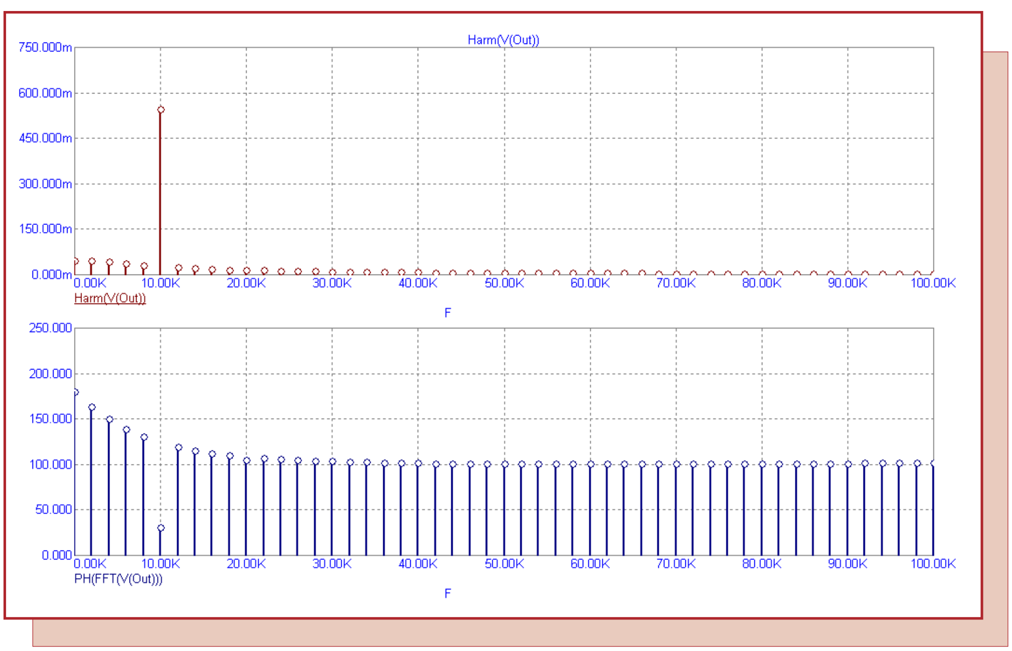


Fig. 10 - FFT Window displaying the harmonics and phase of V(Out)

The resultant plot is displayed in Figure 10. The plot displays both the harmonics and the phase of the harmonics for the V(Out) waveform. The X scale for these plots has been manually set in the Format page of the Properties dialog box to display from DC to 100kHz.

In the transient analysis plot, the output waveform doesn't settle down until the second cycle, so there is an initial transient that will have an effect on the harmonic results. This initial transient can be easily omitted from the harmonic results. Hitting F10 or double clicking in the FFT Window will invoke the Properties dialog box for that window. The time range that the harmonic functions operate in is defined in the FFT page of the Properties dialog box as shown in Figure 11.

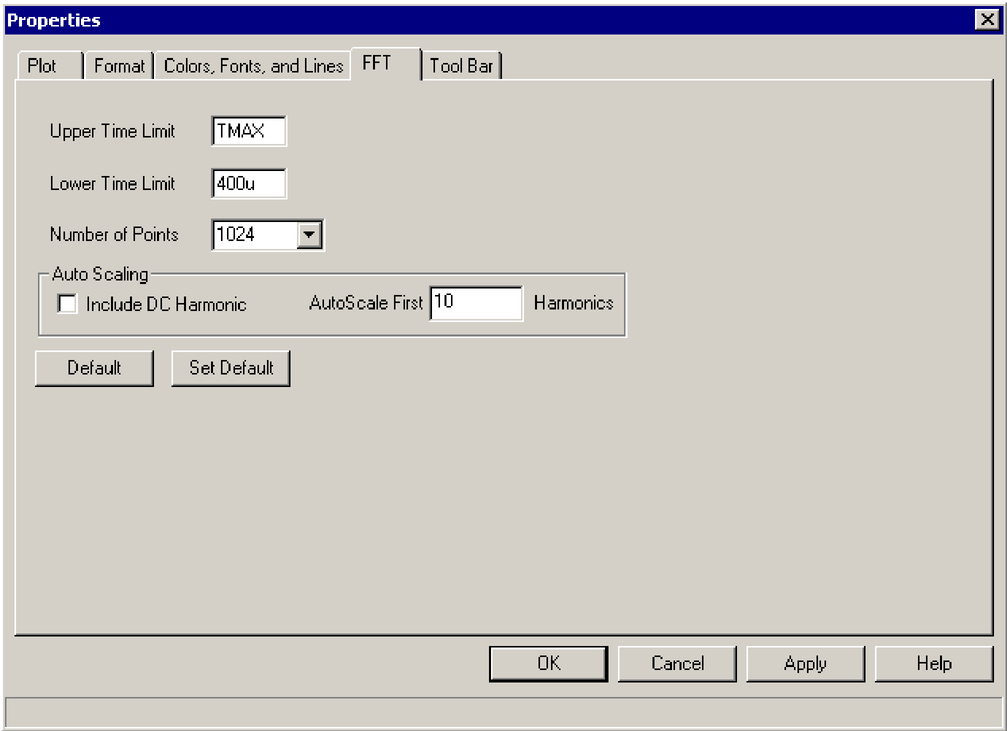


Fig. 11 - FFT Page of the FFT Window Properties dialog box

The FFT page initially has its Upper Time Limit set to TMAX and its Lower Time Limit set to TMIN. These variables take their values from the maximum and minimum times specified in the Time Range field of the Transient Analysis Limits dialog box. As initially set, the entire simulation will be included when deriving the harmonic results. To omit a startup transient, change the time window that is being used. For our analysis, setting the Lower Time Limit to 400u will create a 100us window in which one full cycle of the output waveform is analyzed. The simulation information from 0s to 400us will be ignored. The resultant display is shown in Figure 12.

Note that the frequency interval that the simulation is being sampled with has changed from 2kHz to 10kHz. The frequency interval is determined by the following equation:

1/(Upper Time Limit - Lower Time Limit)

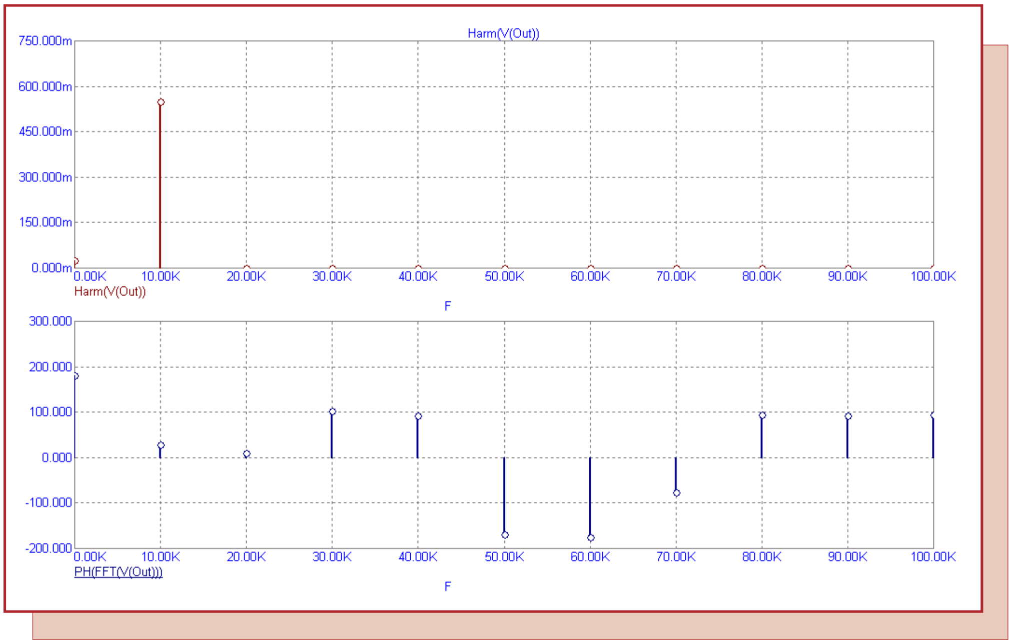


Fig. 12 - FFT Window omitting the startup transient

A right click on the waveform name in the plot will also display any FFT Windows that have already been associated with the specified waveform so that they can easily be recalled. The FFT Windows submenu under the Transient menu will show all FFT Windows associated with the analysis.

HSPICE Style MOSFET Binning

MOSFET binning is the process of adjusting model parameters for different values of drawn channel length and width. While the original Berkeley method of binning is available for all BSIM models, the HSPICE binning is only available for BSIM3 and BSIM4 models.

HSPICE style binning uses multiple model statements to model the MOSFET characteristics over a range of lengths and widths. Each model statement models a portion of the length and width ranges. There are eight MOSFET parameters that apply to HSPICE style binning. They are as follows:

LMIN - This parameter defines the minimum length that the model statement will apply to.
LMAX - This parameter defines the maximum length that the model statement will apply to.
WMIN - This parameter defines the minimum width that the model statement will apply to.
WMAX - This parameter defines the maximum width that the model statement will apply to.
LREF - This is the reference length that the model parameters are assumed to apply for.
WREF - This is the reference width that the model parameters are assumed to apply for.
BINFLAG - If this parameter is set greater than 0.9 and an LREF or WREF parameter is present in the model statement, then HSPICE style binning will be available for this model.
BINUNIT - If this parameter is set to 1, then all geometry parameters are defined in microns. For any other values, the dimensions are meters.

The LREF and WREF parameters are offset values used to interpolate a value within the boundaries created by the minimum and maximum length and width parameters. The model parameters are assumed to apply for the case Leff=LREF and Weff=WREF.

The circuit in Figure 13 provides a simple example of a MOSFET using HSPICE style binning.

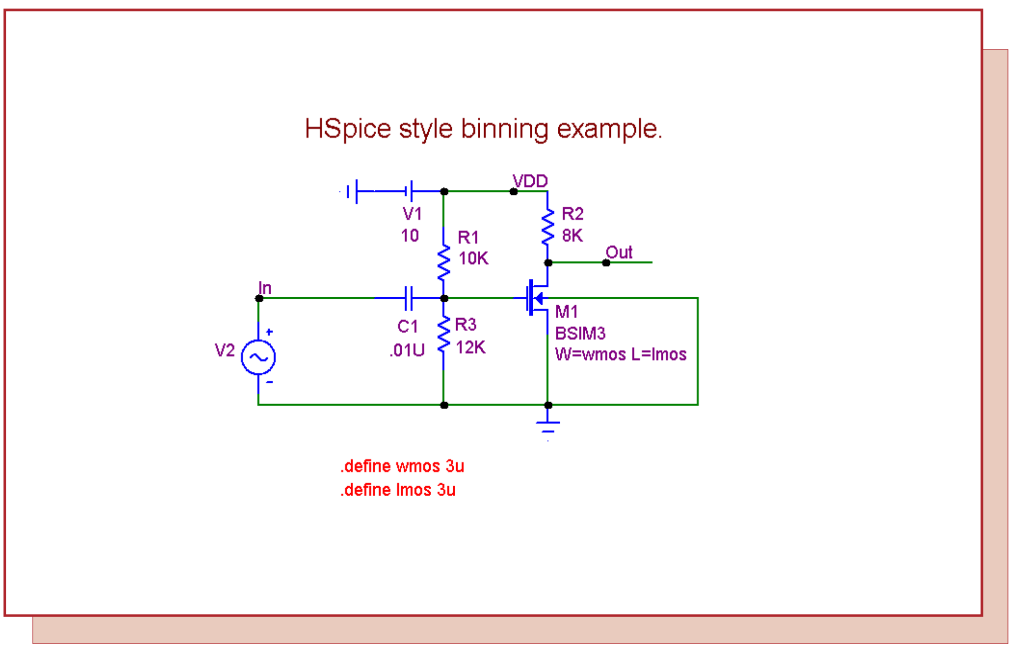


Fig. 13 - HSPICE style MOSFET binning example

In the text area of this circuit, there are three model statements as follows:

.MODEL BSIM3.1 NMOS (LEVEL=8 LMIN=1U LMAX=5U WMIN=1U WMAX=5U
+ LREF=3U WREF=3U BINFLAG=1 BINUNIT=0 VTH0=0.7 ...)

.MODEL BSIM3.2 NMOS (LEVEL=8 LMIN=5U LMAX=20U WMIN=5U WMAX=20U
+ LREF=15U WREF=15U BINFLAG=1 BINUNIT=0 VTH0=0.9 ...)

.MODEL BSIM3.3 NMOS (LEVEL=8 LMIN=20U LMAX=40U WMIN=10U WMAX=40U
+ LREF=110U WREF=125U BINFLAG=1 BINUNIT=0 VTH0=1.1 ...)

Here BSIM3 is the basic model name and BSIM3.1, BSIM3.2, and BSIM3.3 are the individual binning models for the width and length values that fall between WMIN and WMAX and LMIN and LMAX. For HSPICE binning, the model names must have the format NAME.n where NAME is the model name and n is the number that specifies the individual binning model. Only n will vary between binning models. The LEVEL=8 parameter declaration defines these MOSFET models as using the BSIM3 level. Along with the binning parameters shown above, the VTH0 parameter has also been changed between each model in order to demonstrate the use of binning in the analysis.

The VALUE attribute of the MOSFET has been defined as:

W=wmos L=lmos

with the following define statements in the schematic:

.define lmos 3u
.define wmos 3u

For HSPICE style binning, the length and width must be specified in the VALUE attribute. Micro-Cap allows L and W to be specified in the model statement, but binning will always use the length and width specified in the VALUE attribute. In the schematic, the MODEL attribute of the MOSFET has been defined with the name BSIM3. Since the length and width were specified in the VALUE attribute, Micro-Cap is going to look for any possible binning models in the format BSIM3.n that match the length and width specified. If it doesn't find an appropriate binning model, Micro-Cap will then look for a model that has the name BSIM3 only. If neither of these searches are successful, an error will be returned.

For this example, an AC analysis will be run. The Stepping dialog box has been setup to step both of the define variables, wmos and lmos, simultaneously. Each of these variables is defined to step with the List method through the values: 3u, 10u, and 30u. Figure 14 displays the resulting AC analysis.

The simulation produced three runs where the length and width of the MOSFET have been defined as:

Run1: L=3u and W=3u
Run2: L=10u and W=10u
Run3: L=30u and W=30u

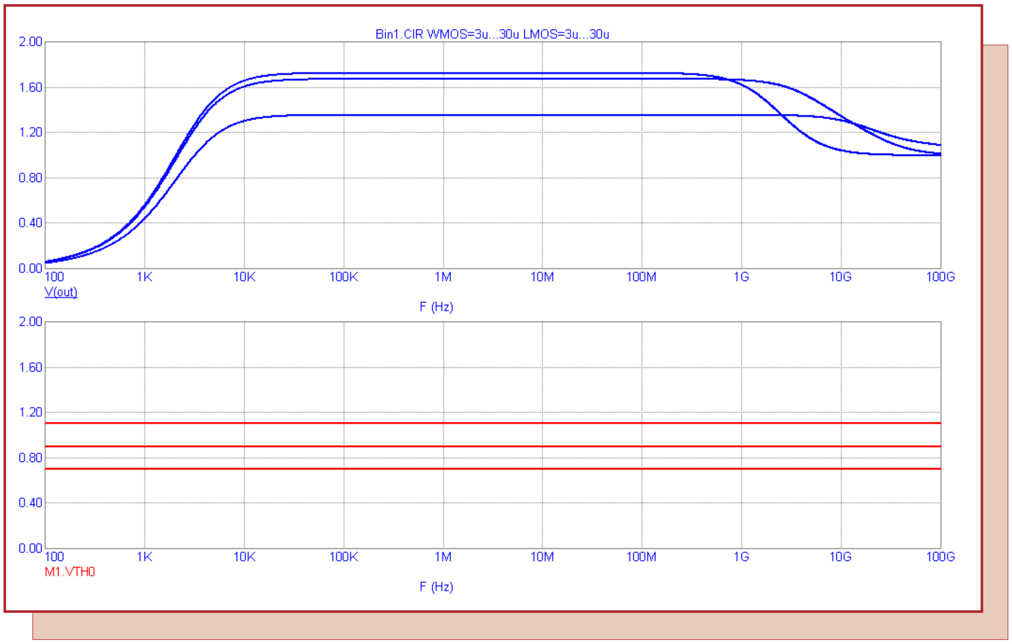


Fig. 14 - HSPICE binning AC analysis results

The MOSFET model chosen by Micro-Cap for each run is determined by the width and length boundaries of the binning models as follows:

WMIN <= W And W <= WMAX
LMIN <= L And L <= LMAX

For each run the following models were used:

Run1: BSIM3.1
Run2: BSIM3.2
Run3: BSIM3.3

In the top plot, the output of the schematic has been plotted. The bottom plot is displaying the waveform for M1.VTH0. This prints out the value of the model parameter VTH0 that the MOSFET called M1 in the schematic is using during the run. Note that the value of VTH0 has changed as expected as a new model is found for each run.

Product Sheet

Latest Version numbers

Micro-Cap 8 .... Version 8.0.2
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