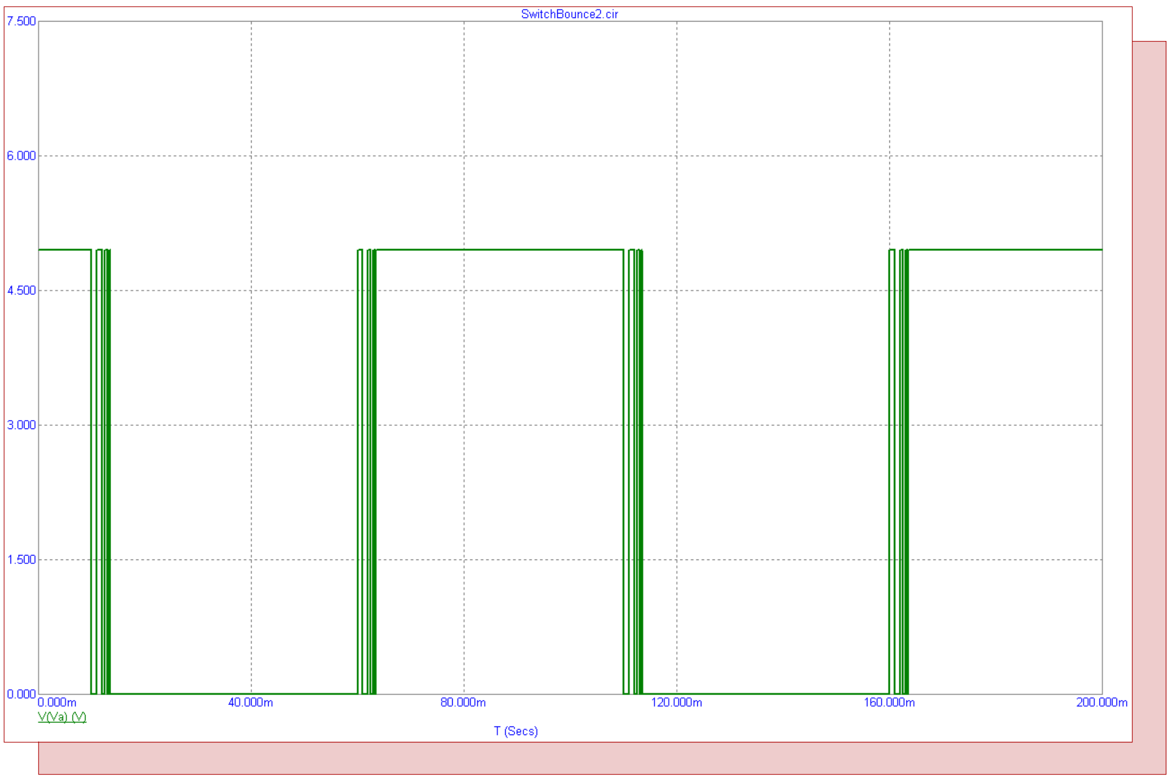
spectrum news

Applications for Micro-Cap™ Users

Summer 2008

News



Modeling Switch Bounce

Featuring:
- Plotting an AC Coupled Waveform
- Modeling Switch Bounce
- Using .Warning Statements

News In Preview

This newsletter's Q and A section describes how to create triangle, sawtooth, ramp, step and impulse input waveforms through the Pulse capability of the Voltage Source component. The Easily Overlooked Feature section describes how to set the maximum number of items that can be displayed in both the File List and the Component List.

The first article describes the use of the partial Fourier series function, FS, in plotting AC coupled waveforms in transient analysis.

The second article describes how to model the switch bounce that occurs when a switch contact does not close or open cleanly.

The third article describes how to use the .Warning statements available in Micro-Cap that provide a method where the user can create their own set of warning messages based on circuit criteria that they have specified.

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 need to use a triangle input waveform for my circuit, but I don't see one available in the Component list. How do I create one?

Answer: A triangle waveform, along with sawtooth, ramp, step, and impulse waveforms, can all be created through the Pulse capability of the Voltage Source component. The Pulse has seven parameters:

V1 - Initial voltage value
V2 - Pulsed voltage value
TD - Time delay value
TR - Rise time value
TF - Fall time value
PW - Pulse width value
PER - Period value

Triangle - For a triangle waveform, set V1 and V2 to the low and high voltages of the waveform. Set TD and PW to 0. TR and TF should be set to the rise and fall times of the triangle waveform. PER should be equal to TR + TF.

Sawtooth - For a sawtooth waveform, set V1 and V2 to the low and high voltages of the waveform. Set TD and PW to 0. TR should be set to the rise time of the sawtooth waveform. TF should be set to a small value in comparison to TR so that the falling transition is very quick but nonzero. PER should be equal to TR + TF.

Ramp - For a ramp waveform, set V1 and V2 to the initial and final voltages of the waveform. Set TD to 0. TR should be set to the simulation time that is to be used in transient analysis. Set PER to a value greater than the simulation time. The TF and PW values are not relevant to the ramp waveform.

Step - For a step waveform, set V1 and V2 to the initial and stepped voltages of the waveform. Set TD to the delay time before the step occurs. TR should be set to a small value in comparison to the simulation time so that there is a quick transition from V1 to V2. Set PW and PER to values greater than the simulation time so that the voltage stays at the V2 value for the rest of the simulation. The TF value is not relevant to the step waveform.

Impulse - For an impulse waveform, set V1 to 0. Set V2 to the impulse voltage. Set TD to the delay time before the impulse occurs. PW should be set to a value equal to 1/V2. TR and TF should be set to small values in comparison to PW so that the impulse has quick rise and fall transitions. PER should be greater than the simulation time so that the impulse only occurs once during the run.

If a current input is needed instead, the above information also applies to the Pulse capability of the Current Source component.

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 and Component List Size
There are two lists available in Micro-Cap that track recently used items. The File List under the File menu keeps track of the most recent files that were opened in Micro-Cap. This list provides an easy means of opening a recent file. The Component List under the Component menu keeps track of the most recent components that were placed in a schematic. This list provides an easy method to select a component for entry in the schematic.

The default size for each of these lists is set to ten. Both of the list sizes are user controllable though. The size of the lists can be set within the Preferences dialog box. Access the Preferences dialog box under the Options menu. In the list on the left, select General under the Options item. There are two fields available in the Options - General page called File List Size and Component List Size. These fields control the maximum number of items that can be displayed in their associated lists. The maximum value that can be set is 50 for each of these.

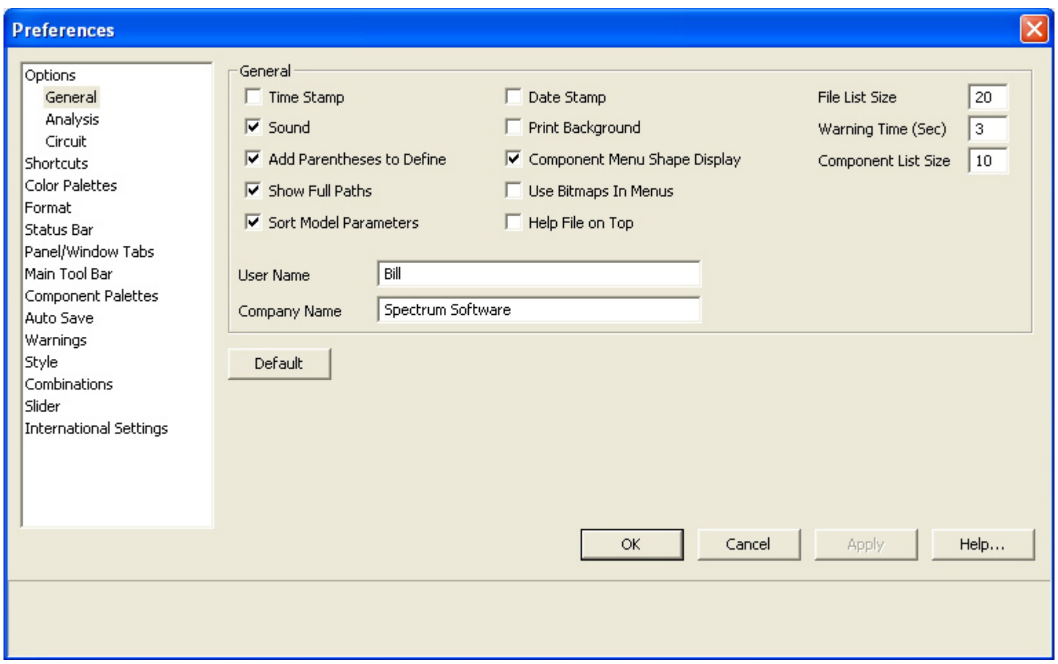


Fig. 1 - Preferences Options - General page

Plotting an AC Coupled Waveform

AC coupling of a waveform displays only the AC signal while blocking the DC portion of the waveform. In practice this is done by placing a capacitor between the measuring point and the probe. This is useful when trying to view a characteristic of the waveform such as its ripple. If the ripple is small compared to the DC value of the waveform, the ripple can be difficult to view when the plot is scaled with regards to the DC value.

In Micro-Cap, there are a few ways to offset the DC portion of the waveform in a plot. If the DC value is known, an expression such as:

V(Out) - 10

can be plotted where 10 is the DC portion of the V(Out) waveform. This cancels out the DC contribution to the waveform. However, in most cases, the DC value will not be known or may change from run to run.

Another method is to place a high pass filter on the node of interest and then plot the waveform at the output of the filter. The negatives of this method are that settling time issues and the loss of low frequencies can have a significant effect on the filtered waveform so the filter values must be chosen carefully.

The method for plotting an AC coupled waveform that will be examined in this article is to use the partial Fourier series function, FS. The FS function has the following syntax:

FS(u,[[n1],n2])

u is the waveform that is to be reconstituted from the specified range of Fourier terms. n1 and n2 are optional parameters and define the range of Fourier terms to be used in the calculation. n1 defaults to 0, and n2 defaults to the FFT Properties value of Number of Points/2. The output is the compiled sum of the Fourier terms from n1 through n2. For example, if the following expression is used:

FS(V(Out),1,10)

The Fourier series of V(Out) will first be calculated. Then the Fourier terms from the first harmonic to the tenth harmonic will be summed. The resulting sum will be the V(Out) waveform reconstituted with just the contributions from the first ten harmonics. For AC coupling, the FS operator would be specified as:

FS(V(Out),1)

In this case, the V(Out) waveform would be reconstituted using the contributions from the first harmonic to the last calculated harmonic (since n2 is unspecified), omitting only the 0'th or DC harmonic.

The FS function is an FFT function so it uses the time range and the number of points specified in the FFT page of the Analysis Properties dialog box. To get valid results from this method, the time range specified in the FFT page for the FS function to operate on should be when the circuit is running in its steady state operation. Any initial transient in the analysis needs to be excluded as that would skew the Fourier calculations used in the function.

To show how the FS operator works as an AC coupler, the example circuit below will be simulated. The circuit is a mains AC to DC rectifier. The V1 source represents the mains voltage and produces a 320V peak, 50Hz sine wave input. This sine wave is fed into a multiwinding transformer. The two output windings of the transformer are each connected to full wave bridge rectifiers to create both a positive and negative DC voltage. PI filters are used with each bridge rectifier to both decrease the ripple and reduce the harmonics of the output waveform. The AC waveforms of interest are the ripples at the output of the rectifiers and the PI filters. A pair of high pass filters have been added in the circuit at nodes VRpos and VRpos1 to provide a benchmark to the accuracy of the partial Fourier series method.

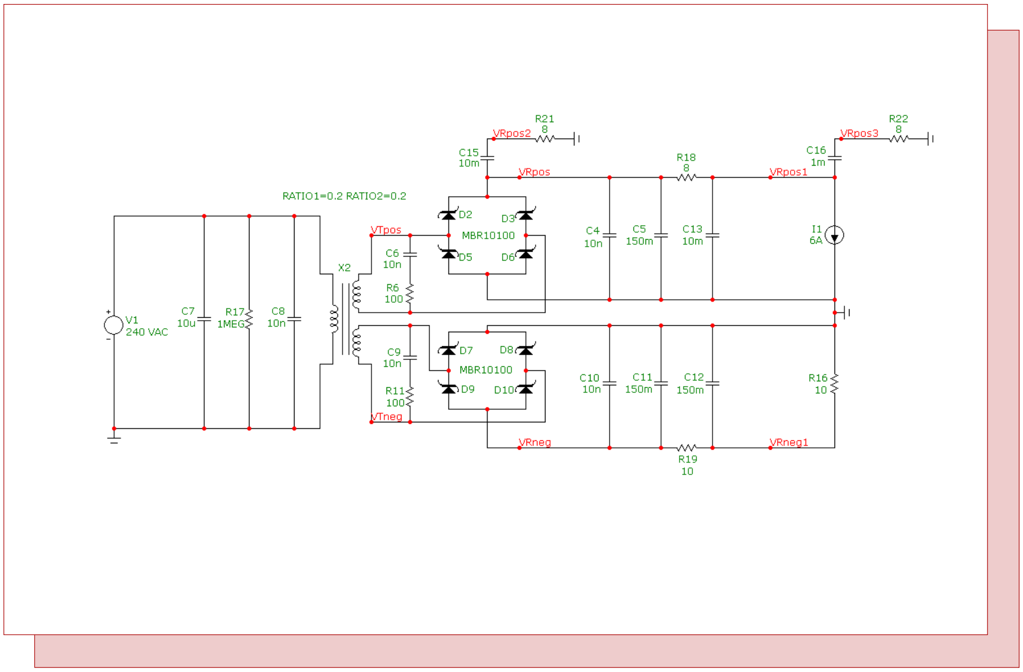


Fig. 2 - AC to DC rectifier circuit

The AC to DC rectifier will be simulated in transient analysis. The AC coupled waveforms at the bridge rectifier and the PI filter of the positive voltage rail will be plotted. The partial Fourier series expressions used to plot these two waveforms are:

FS(V(VRpos),1)
FS(V(VRpos1),1)

Both of these expressions reconstitute their associated waveforms with just the DC harmonic being omitted. The transient simulation will be run over a time range of 2 seconds. Since the FS function is an FFT function, the values in the FFT page of the Analysis Properties dialog box have been set as shown in Figure 3. The Upper Time Limit is set with the TMAX variable which uses the ending transient analysis time which in this example is 2s. The Lower Time Limit is set with the expression TMAX-50m which evaluates in this simulation to 1.95s. With these settings, the last 50ms of the simulation will be the time range that any FFT function operates on. The Number of Points field is set to 2048. This value determines the number of Fourier terms that will be calculated in the first part of the FS operation. For the FS function, the Number of Points should not be too great as the calculation time can be long for larger values. The Auto Scaling section is not relevant to the FS function.

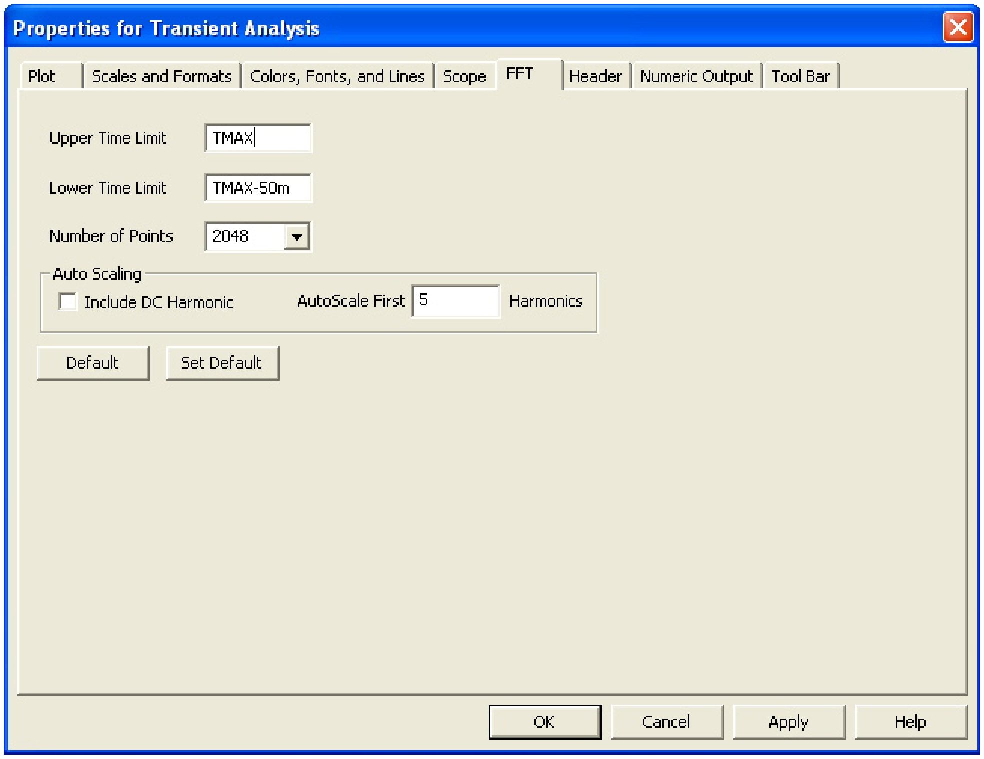


Fig. 3 - FFT page of the Properties dialog box

The resulting transient analysis is shown in Figure 4. Only the last 50ms is displayed in the plot. With the specified FFT page settings, this is the only data available for the simulation. This time window eliminates the initial transient in the circuit simulation that would have skewed the desired results for the AC coupling.

The FS function is a post processing function which means that it does not calculate and display its results until the simulation has finished. The expression will just plot a line at zero until the simulation finishes, and the function is able to produce the specified waveform.

The top plot shows the voltage at the positive rail rectifier and filter outputs. The disparity between the two DC levels along with the relatively small size of the AC portion on these waveforms make it extremely difficult to gauge the ripple characteristics. One would need to zoom in manually on one of the waveforms to see anything of interest.

The middle plot shows the ripple at the positive rail rectifier. The ripple has a triangular shape with a magnitude of approximately 165mV. The V(VRpos2) expression plots the comparable AC coupled voltage at the output of the C15-R21 high pass filter.

The bottom plot shows the ripple at the positive rail PI filter. Due to the effect of the PI filter the ripple magnitude has been greatly descreased to just a few millivolts. Note that the ripple now has much more of a sine wave shape which reduces the harmonics present in the waveform compared to the triangular shape of the prefilter ripple. The V(VRpos3) expression plots the comparable AC coupled voltage at the output of the C16-R22 high pass filter.

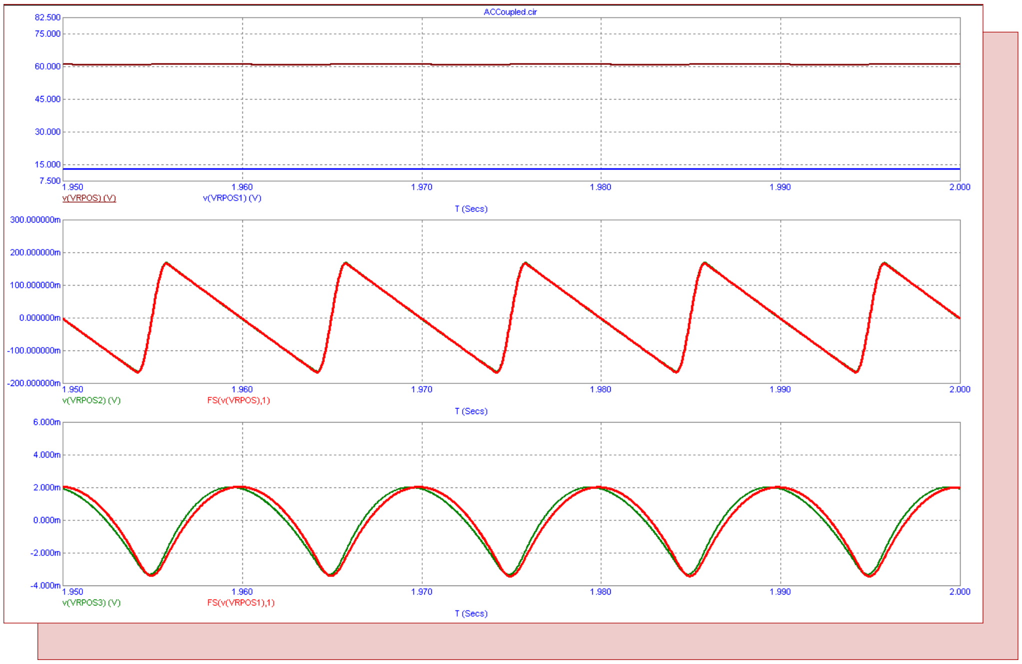


Fig. 4 - AC coupled plots displaying the ripple of the rectifier and filter outputs

Modeling Switch Bounce

Switch bounce, also referred to as contact bounce, is a common issue with many types of mechanical switches. Ideally, when the switching contact strikes another contact, it would immediately stick to that contact. However, due to the contact's momentum and elasticity, the switching contact can bounce a few times before coming to rest. This produces a pulse like effect as the switch rapidly opens and closes.

The standard switch models in Micro-Cap do not model switch bounce. These switches all have an ideal transition where they open or close cleanly. In order to model switch bounce, additional circuitry needs to be added to the switch. The schematic below displays one method of simulating switch bounce.

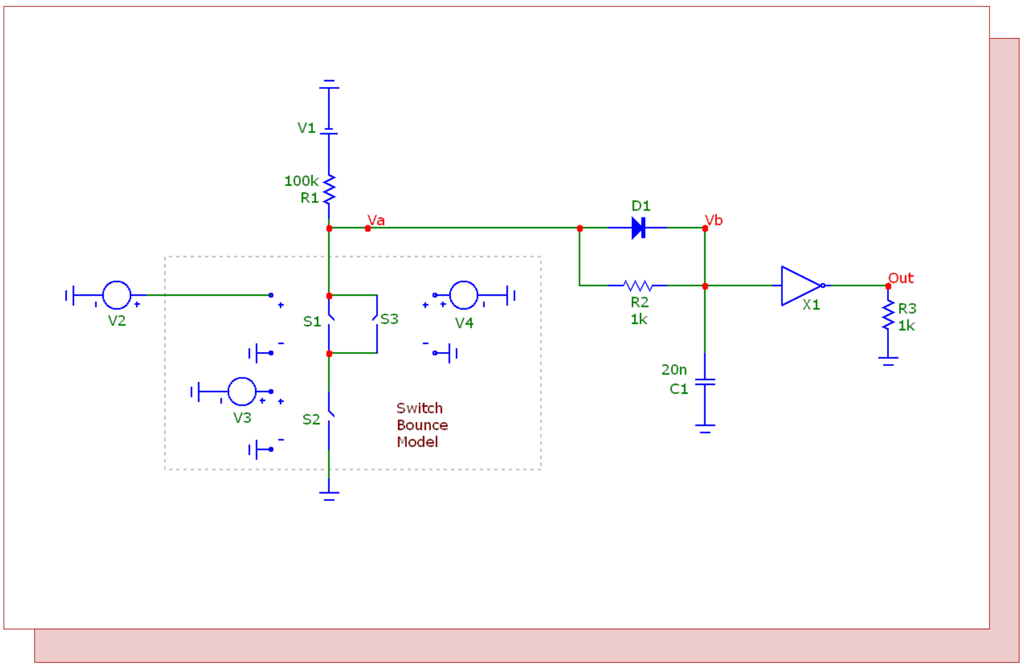


Fig. 5 - Switch bounce circuit

The switch bounce model is contained within the dashed rectangle in the schematic. It consists of three S (V-Switch) components and two voltage source components. The S1 switch is the main switch. The inputs to this switch control whether the switch model will be opened or closed. This switch is defined with the following model:

.MODEL MAIN VSWITCH (ROFF={2*Roff} RON={Ron/2} VOFF={Voff} VON={Von})

All of the parameters are defined through the symbolic variables Roff, Ron, Voff, and Von. These four symbolic variables are set through the following define statements.

.Define Von 2.5
.Define Voff 2
.Define Ron 1m
.Define Roff 10Meg

The Von and Voff variables directly define the on and off voltages of the switch. In this case, the switch will close when the input voltage is greater than 2.5V and open when the input voltage is less than 2V. In between these two input values, the switch will smoothly transition between the on and off resistances. The Ron and Roff variables define the on and off resistances of the switch bounce model and need to be scaled within the switch model statement to account for the other switch resistances in the circuit.

The S2 switch and the V3 voltage source model the switch bounce when the main switch closes. The S2 switch is defined with the following model:

.MODEL BOUNCE VSWITCH (ROFF={Roff} RON={Ron/2} VOFF=3 VON=2)

Both of the S1 and S2 switches need to be closed for the entire switch bounce model to be closed. Due to this, each of these switches needs to model half of the on resistance since they are in series. Therefore, both of these switch models have their on resistance defined as Ron/2. When the V3 voltage source at the input to the switch is below 2V, the switch is closed and when it is above 3V, the switch is open. The V3 source has its VALUE attribute set to:

PWL TRIGGER={(R(S1) < Ravg)} 0,0 .999m,0 1m,5 2m,5 2.001m,0 2.499m,0 2.5m,5 3m,5 3.001m,0 3.3m,0 3.301m,5 3.5m,5 3.501m,0

The voltage source has a piecewise linear definition. The source is dormant until the TRIGGER expression evaluates to true. To trigger this source, the resistance of the S1 switch must fall below the value of the variable Ravg where Ravg is set through the following define statement:

.Define Ravg (Roff+Ron)/2

Ravg is the average of the on and off resistances of the switch bounce model. When the resistance of S1 falls below this value, that indicates that the switch has closed. At that point, the V3 source will output the specified PWL waveform. In this case, the PWL waveform will cause the S2 switch to open and close three times over the course of the next 3.5ms which simulates the switch contact bouncing.

The S3 switch and the V4 voltage source model the switch bounce when the main switch opens. It works in a similar manner to the S2 and V3 combination. The S3 switch model is defined as:

.MODEL BOUNCE2 VSWITCH (ROFF={2*Roff} RON={Ron/2} VOFF=2 VON=3)

Both of the S1 and S3 switches need to be open for the entire switch bounce model to be open. Due to this, each of these switches needs its off resistance set to 2*Roff since they are in parallel. In addition, the on resistance of S3 also needs to be set to Ron/2 since it will be in series with the S2 switch when it is closed. The V4 voltage source is connected to the input of S3 and has its VALUE attribute defined as:

PWL TRIGGER={(R(S1) > Ravg)} 0,0 .999m,0 1m,5 2m,5 2.001m,0 2.499m,0 2.5m,5 3m,5 3.001m,0 3.3m,0 3.301m,5 3.5m,5 3.501m,0

To trigger this source, the resistance of the S1 switch must rise above the Ravg value. When the resistance of S1 rises above this value, that indicates that the switch has opened. The resulting PWL output will cause the S3 switch to close and open three times over the course of the next 3.5ms which simulates the switch contact bouncing upon opening.

This model can be easily modified. The PWL waveforms can be changed to model the appropriate switch bounce characteristics. The PWL waveform values chosen in this example were completely arbitrary. If the bounce needs to be simulated only upon either an open or close, the S2 or S3 switch can be deleted from the model. Note that the on and off resistances of the remaining switches may need to be adjusted if either the S2 or the S3 switch is deleted.

In this example circuit, the switch bounce model is connected to a 100Kohm resistor which goes to a 5V battery. The Va node at the junction of the switch and resistor is fed into a simple debouncing circuit consisting of a diode, resistor, capacitor, and inverter. The resistor and capacitor mitigate the switch bounce transitions enough so that they have no effect on the inverter. The inverter then restores the sharp transitions of the switch.

The debouncing circuit portion of the circuit is first disabled. This is done by drawing a select box around the debouncing circuitry and then choosing Disable from the Edit menu. Running transient analysis produces the simulation plot below.

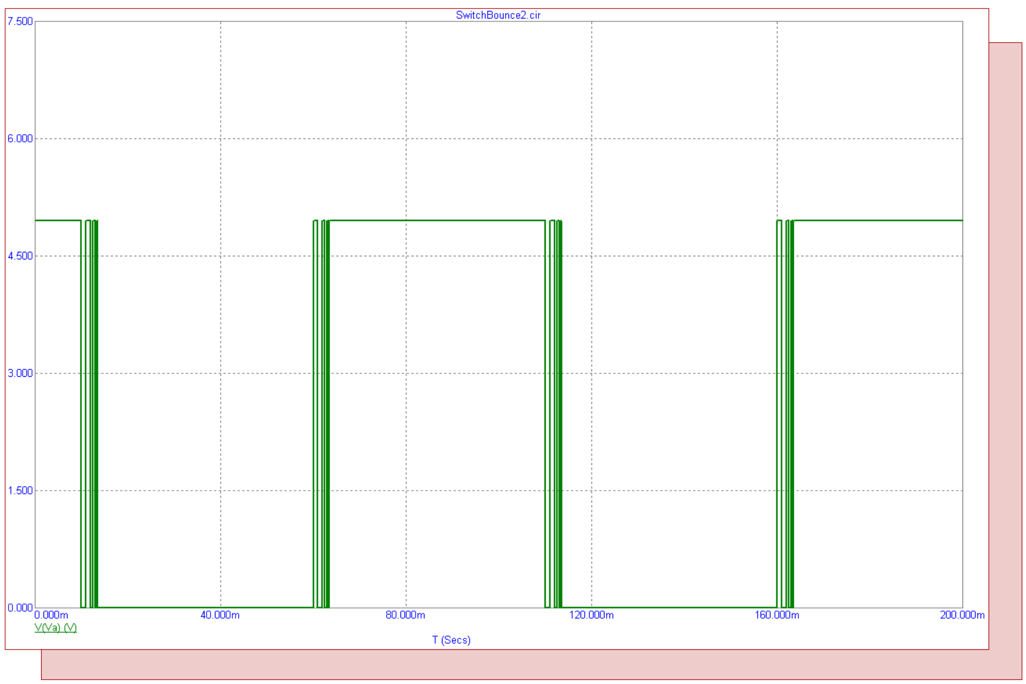


Fig. 6 - Switch bounce simulation with the debouncing circuit disabled

The V(Va) waveform clearly shows the pulse like effect produced by the switching contact bouncing upon both the open and close transitions. The bouncing transitions match precisely with the PWL waveforms that were defined within the switch bounce model.

The debouncing circuit is then enabled again by drawing a select box around the debouncing circuitry and then choosing the Enable command under the Edit menu. Running the transient analysis produces the simulation plot in Figure 7. The voltage waveform produced by the bouncing has been greatly minimized as shown in the plots for V(Va) and V(Vb). The V(Out) waveform shows that the inverter has produced the sharp, clean transition edges that one would want from the switch.

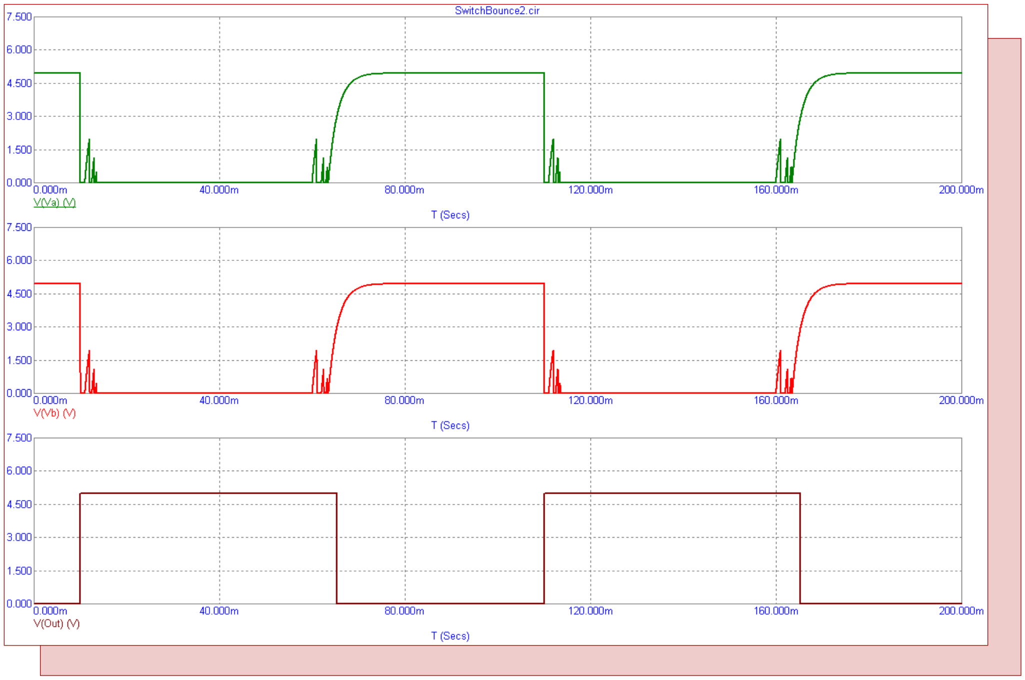


Fig. 7 - Switch bounce simulation with the debouncing circuit enabled

Using .Warning Statements

The .Warning statements available in Micro-Cap provide a method where the user can create their own set of warning messages based on circuit criteria that they have specified. This can be useful in flagging instances where the circuit has exceeded specific design parameters. The general syntax of the statement is:

.WARNING ["Title" [,]] "Message" [,] condition [,print_expr]

This command statement lets you create warning messages that will appear if the condition evaluates to True. The condition is a boolean expression like PD(R1) > 100m which would be true if the power dissipation of the resistor R1 was greater than 100mW. If the condition is true at any point during an analysis, the specified message is printed in both the schematic and the numeric output file along with the value of print_expr if it is specified. If print_expr is not specified then the analysis sweep variable (T in transient, F in AC, and DCINPUT1 in DC analysis) is printed.

The default title text for each of these messages is "Warning:". If "Title" is specified in the statement, then the specified title text will be used instead. All warning statements that share the same title text will have their messages grouped together in the schematic.

All of the commas except the one preceding print_expr may be replaced with spaces. The text "\n" forces a new line in the message text. This is not needed if the command is entered using the grid text box in the schematic.

Example #1
The example below is a schematic of a simple amplifier gain stage. Four .Warning statements have been defined for the circuit. The first warning checks the battery voltage, V2, at the positive power supply of the opamp to make sure it is less than or equal to 30 volts. It is defined as:

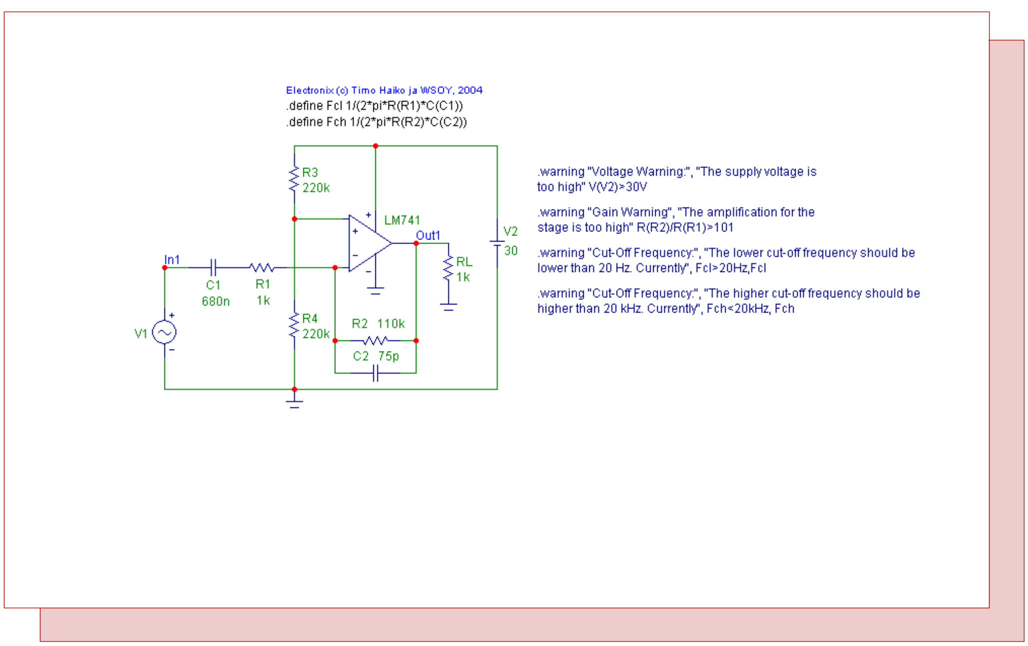


Fig. 8 - .Warning example circuit

.warning "Voltage Warning:", "The supply voltage is too high" V(V2)>30V

The next warning checks the gain of the amplifier stage by returning a warning if the ratio of the R2 and R1 resistors are greater than 101. It is defined as:

.warning "Gain Warning", "The amplification for the stage is too high" R(R2)/R(R1)>101

The last two warnings check that both the lower and higher cut-off frequencies in the amplifier design meet certain performance criteria. These warning messages reference the symbolic variables Fcl and Fch which have been set through define statements in the schematic. The Fcl variable calculates the lower cut-off frequency using the combination of the R1 resistor and the C1 capacitor. The Fch variable calculates the higher cut-off frequency using the combination of the R2 resistor and the C2 capacitor. The two warning messages are:

.warning "Cut-Off Frequency:", "The lower cut-off frequency should be lower than 20 Hz. Currently", Fcl>20Hz,Fcl

.warning "Cut-Off Frequency:", "The higher cut-off frequency should be higher than 20 kHz. Currently", Fch<20kHz, Fch

A Dynamic DC analysis is run on the schematic. With the present circuit configuration, the gain warning and the two cut-off frequency warnings have been triggered. These three warning messages appear in the schematic as shown below.

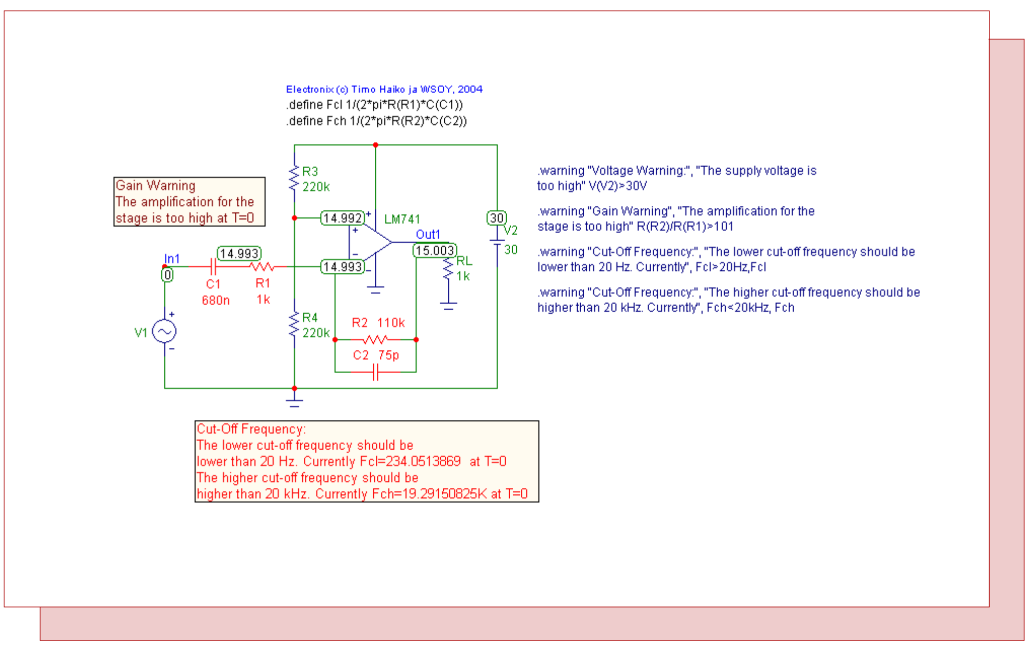


Fig. 9 - Dynamic DC analysis with R2=110k

Any components that have been referenced in the Condition expression of a warning will have their color changed to that of the warning message text. This can be seen with the R1 and R2 resistors and the C1 and C2 capacitors whose colors now match the red of the corresponding frequency warning message. The gain warning also references the R1 and R2 resistors, but in such a conflict, the color used will be from the statement that was first entered in the schematic. Note that since the two cut-off frequency warnings share the same Title that they have been grouped together in the schematic.

When the R2 resistor value is subsequently modified to 100K, the gain and the higher cut-off frequency warning conditions are no longer violated. The schematic appears as below. Just the lower cut-off frequency warning message is now displayed. Since the lower cut-off warning specifies the R1 and C1 components, these two components are highlighted in red in the schematic.

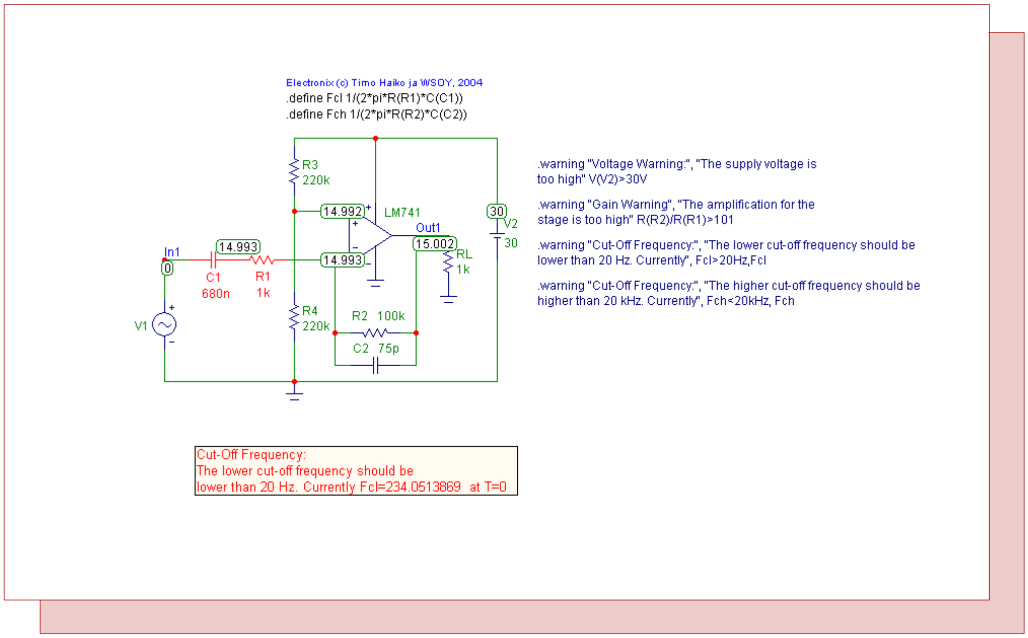


Fig. 10 - Dynamic DC analysis with R2=100k

Example #2

In the previous example, the warning statements had the individual component part names explicitly specified in the condition expression. Global part names may also be defined through the use of the @ character. All instances in the schematic that match the global part name must be valid for the specified condition expression or an error will occur. The schematic in Figure 11 uses the two following statements.

.warning "Resistor Failure","The maximum power value was exceeded!", PD(R@) > 250m, PD(R@)
.warning "BJT Failure","The maximum power value was exceeded!", PD(Q@) > 1, PD(Q@)

The first statement checks to see if any component whose part name starts with R has its power dissipation exceed 250mW at any point during the simulation. The second statement is similar except that it checks any component whose part name starts with Q to see if its power dissipation exceeds 1W at any point during the simulation. In this schematic, all of the transistor part names start with Q, and all of these transistors use a model statement. If one of these transistors used a subcircuit model instead, the PD operator would not be valid for that component and an error would be invoked when an analysis was entered.

A transient simulation is run for this circuit. At the end of the simulation, the schematic appears as in Figure 12. The warning messages displayed in the schematic were also written into the transient numeric output file. During the transient simulation, three resistors (RL, R13, and R14) and two transistors (Q8 and Q9) exceeded the power dissipation limits set in the warning messages. The warning messages display the excessive power dissipation values along with the time at which the violation occurred. The message will only display the first violation that occurs if there are repeat violations.

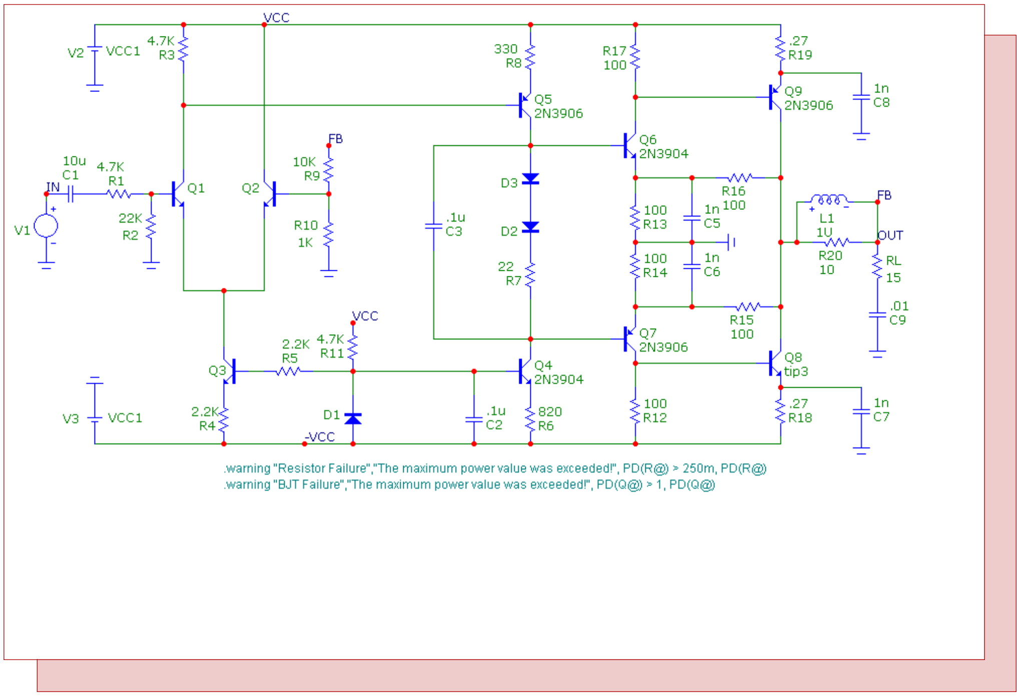


Fig. 11 - Audio amplifier schematic

The benefit of drawing the referenced components with the color of the warning message text is readily apparent in the schematic. The resistors and transistors that have triggered the warning messages are easy to pick out within the schematic, and it can really provide a visual focus on a particular section of the circuit that may need additional work.

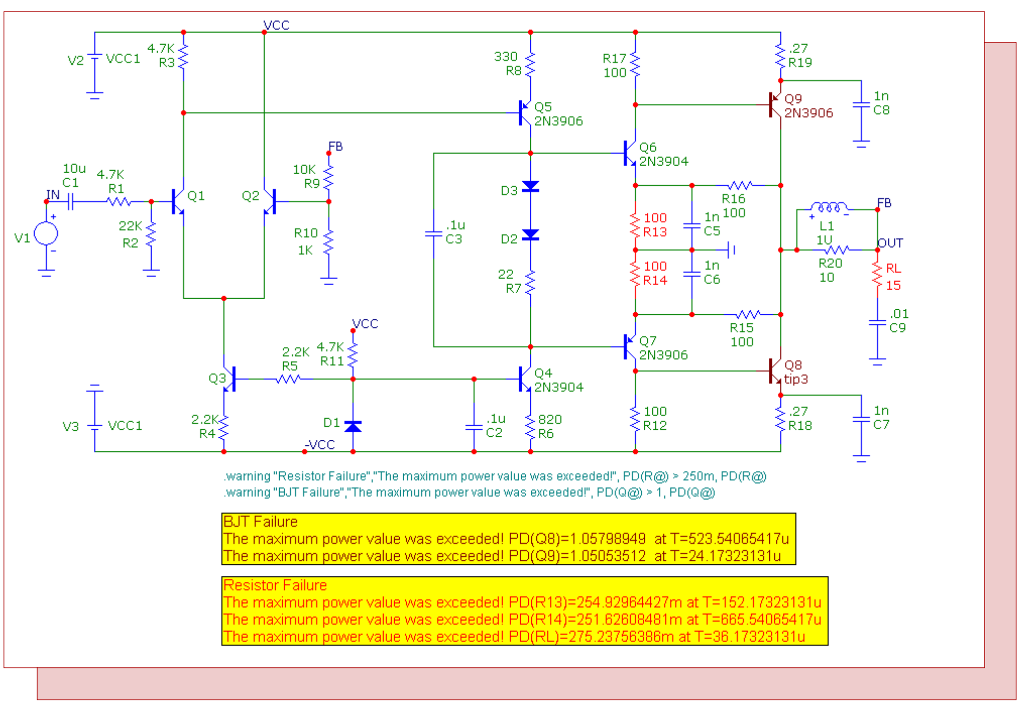


Fig. 12 - Warnings from a transient simulation

Latest Version numbers

Micro-Cap 9 ....Version 9.0.5
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