

## **2006-1171: INCREASING PRODUCTIVITY AND AVOIDING CIRCUIT SIMULATION ERRORS IN MULTISIM**

### **John Hackworth, Old Dominion University**

John R. Hackworth is Program Director for the Electrical Engineering Technology program at Old Dominion University. He holds a B. S. Degree in Electrical Engineering Technology and a Master of Science Degree in Electrical Engineering, both from Old Dominion University. Prior to joining the Old Dominion University faculty, John had approximately 20 years of industrial experience in test engineering and plant automation. He is the principal co-author of the text Programmable Logic Controllers: Programming Methods and Applications, published by Prentice-Hall.

### **Richard Jones, Old Dominion University**

Richard Jones has been teaching at ODU since 1994. He is a retired United States Navy Submarine Service Lt. Commander with sub-specialties in Ballistic Missile, Torpedo, Sonar, and Radio systems. Richard has previously taught Mechanical Engineering Design at the United States Military Academy, West Point, N.Y., and Electrical Engineering at the United States Naval Academy, Annapolis, Md. He holds an ASEET from Cameron University, a BSEET from Oklahoma State University, and a Master of Engineering in Electronics Engineering from the Naval Postgraduate School at Monterey, California.

### **William Stanley, Old Dominion University**

William D. Stanley, Eminent Professor Emeritus, retired in December 2000 after about 35 years of service at Old Dominion University, of which he served as a department chair for 27 years. He holds a B. S. degree from the University of South Carolina and M. S. and Ph.D. degrees from North Carolina State University, all in Electrical Engineering. He is a registered Professional Engineer and author or co-author of about 20 textbooks. Since retiring, he has continued to write, teach part-time, work with the Navy College program, and has developed two asynchronous courses for that program.

# Increasing Productivity and Avoiding Circuit Simulation Errors in MultiSIM<sup>®</sup>

## Abstract

Methods for streamlining MultiSIM circuit analysis projects are shown. Methods are shown which improve analysis results, streamline analysis methods, and prevent some common errors. Several examples of MultiSIM projects are given which illustrate each of the discussed methods.

## Introduction

In recent years, MultiSIM (formerly Electronics Workbench) has proven to be an excellent learning tool for electrical engineering and electrical engineering technology students. It can be adapted to any student learning level. For example, at the “beginner” level, the student can connect individual components and instruments much as he or she would do in a conventional laboratory environment. When the simulated power switch is turned on, the instruments on the screen begin realistically functioning. The front-panel switches and pushbuttons on the instruments work much like real equipment, thus allowing the student to learn how to use the instruments while making laboratory measurements. At the advanced level, after wiring the circuit, the student can forego the instruments and power switch and go directly to more advanced analysis features such as ac steady-state analysis (frequency and phase response analysis), transient analysis, or I-V analysis.

Although students generally adapt quickly to MultiSIM to perform most of their analysis work, there are some aspects of MultiSIM that require more experience. It is the purpose of this paper to explore some of these more subtle aspects of MultiSIM. In some cases, these will be simple “shortcuts” that can be used to perform circuit analysis easier or faster, and in other cases, techniques will be demonstrated that will allow the user to avoid analysis errors or failure because the analysis was setup incorrectly, or the calculated results do not converge.

At Old Dominion University, the Electrical Engineering Technology program uses Multisim in a wide range of courses including fundamental circuit analysis, electronic devices, digital circuits, linear integrated circuits, programmable controllers, electrical machines, and power systems. The extent of use varies from simple in-class examples used to support lecture material, to full lab experiments requiring students to build, troubleshoot, and analyze circuits totally by computer simulation. Because of the wide variety of Multisim applications used in ODU’s EET program, the examples illustrated in this paper are likewise varied.

## Ideal Sources

By default, voltage sources in MultiSIM are ideal; that is, they have zero internal resistance. Generally, this functions well for most circuit analysis situations. However, consider the MultiSIM simulation of a three-phase 60 Hz delta source with wye connected load resistors, shown in Figure 1. This is a typical circuit simulation that would be assigned to students in an ac fundamentals course. Generally ac voltmeters and ac ammeters are added, which allow students to gain a better understanding of the relationships between line voltages and currents, and phase

voltages and currents. However, for this illustration, the meters are unnecessary and, for clarity, are not included. When this simulation is switched on, it will fail, giving somewhat cryptic “singular matrix” and “timestep too small” error messages (note that this type of failure is not unique to MultiSIM). The problem is that when constructing this circuit in MultiSIM, three assumptions have been made: 1) the three 208 V sources are exactly 208 volts, 2) by Kirchhoff’s law, the phasor sum of the three sources will equal zero, and 3) the sources are realistic.

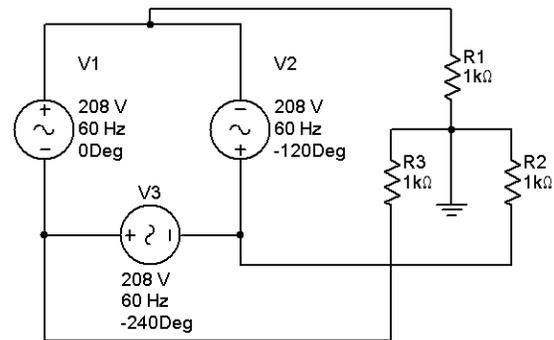


Figure 1. Delta Source Circuit.

However, due to small digital mathematical errors, MultiSIM cannot make the three voltages exactly 208 volts, and as a result the instantaneous sum of the voltages around the V1, V2, V3 loop at any given time will not equal to zero. Since the sources are ideal (i.e., they have zero internal impedance), MultiSIM cannot resolve the small mathematical voltage error; i.e., there is no resistive or reactive component on which to drop the error voltage. Unfortunately, the inexperienced user reads the error message “timestep too small” and wastes time increasing the timestep, to no avail. Instead the problem can be easily corrected by recognizing that the simulation needs a small resistance in the V1, V2, V3 loop that will drop the error voltage created by the calculation. In other words, the sources need to be made “realistic” by giving them internal resistance. This is done by adding a resistor in series with each source (for this simulation, one resistor located anywhere in the loop will suffice, but students find it easier to understand if a series resistance is included for every source). To prevent the resistors from complicating the measured results, their values are usually made very small, typically 1 milliohm or less.

## Special MultiSIM Sources and Controlled Function Blocks

### The Piece-wise Linear Voltage Source

Quite often it is desirable to excite a circuit to be simulated with a signal other than the standard DC or sinusoidal waveform. The Piece-wise Linear Voltage source provides the ability to create any signal which can be broken up into linear segments. The device can be programmed via a text file or, if the signal can be broken up into nine or fewer segments, the device can be opened and programmed directly. Figure 2 demonstrates how this may be achieved with the resulting waveform shown in Figure 3.

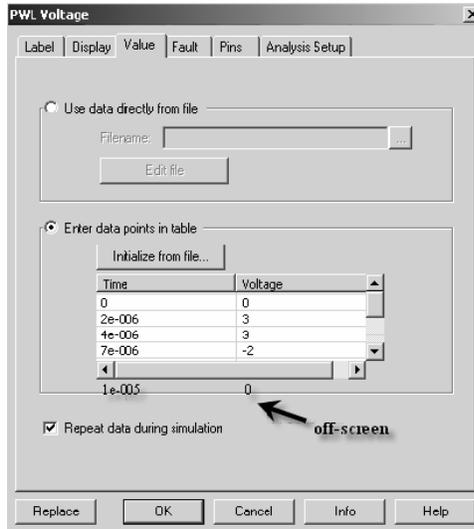


Figure 2. Piece-wise Linear Voltage Source setup window.

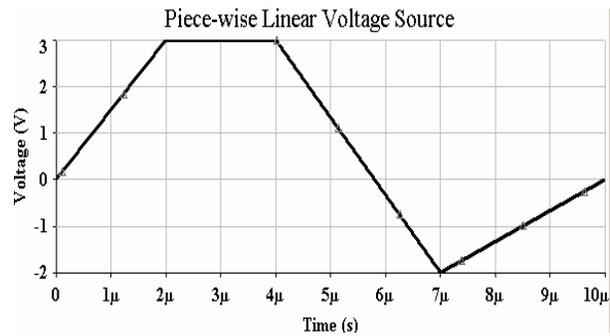


Figure 3. Voltage waveform resulting from the piece-wise linear program.

### The Current Controlled Voltage Source

This device produces a voltage waveform which is directly proportional to the current in a circuit. It can be very helpful when it is desired to plot the circuit current or when (as shown in the circuit below) the current signal is required elsewhere in the circuit. It can also be very useful when it is desired to scale the current axis in the Grapher utility. All that is required is to open the device and set the transresistance value to 1000 and then set the label on the current axis to (mA) in the Grapher window.

### Controlled Function Blocks

MultiSIM has provided a set of control blocks which can be used to manipulate signals in circuits. The major blocks available are: Dividers, Transfer Function blocks, Voltage and Current Limiters, Voltage Gain blocks, Voltage Hysteresis blocks, Voltage Limiters, Voltage Slew Rate blocks, Voltage Summers, and the two used in the example below, the Voltage Multiplier and the Voltage Integrator. These blocks can be extremely useful in the educational environment.

The example circuit shown in Figure 4 is a purely educational circuit designed to demonstrate the relationship between capacitive voltage,  $v_c(t)$ , current,  $i_c(t)$ , power,  $p_c(t)$ , and energy,  $w_c(t)$ . Besides demonstrating the circuit elements previously mentioned, it has been specifically designed to demonstrate some of the problems associated with choosing too few a number of simulation points when performing a transient analysis. The voltage waveform from the Piece-wise linear voltage source is as shown in Figure 3 above.

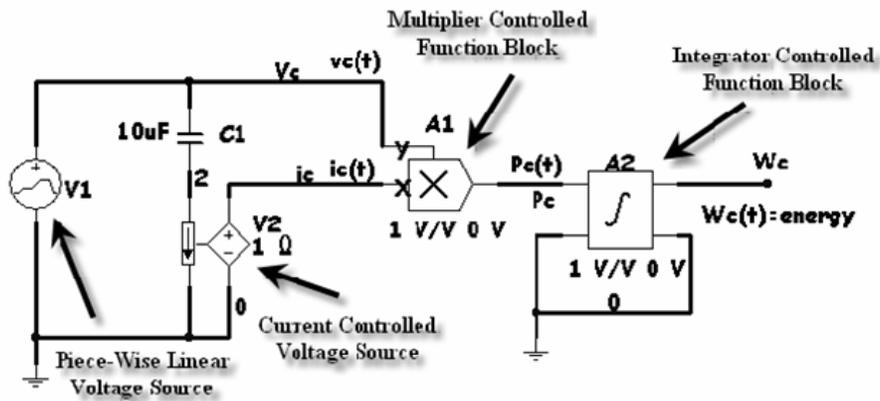


Figure 4. MultiSIM circuit to demonstrate the relationships between power and energy.

The circuit is based on the following equations:

$$p_c(t) = v_c(t)i_c(t)$$

$$w_c(t) = \int p_c(t)dt$$

The use of the Current Controlled Voltage Source is imperative in this circuit in order to provide the necessary current signal for the multiplier control block. Solution of this circuit by hand will result in the graphs shown in Figure 5.

A transient analysis will be initially performed on this circuit using the default values (Figure 6) and the resulting energy waveform will be observed.

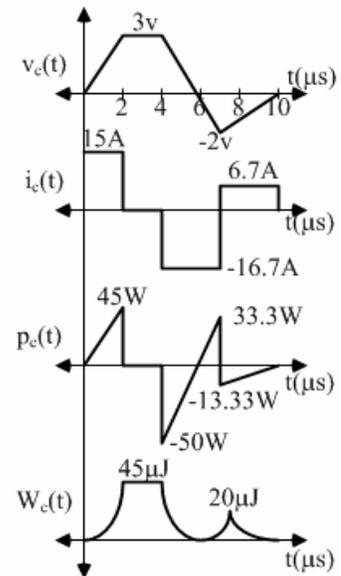


Figure 5.

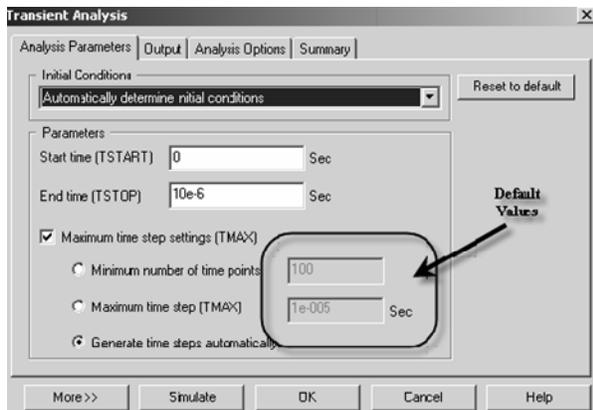


Figure 6. Transient Analysis default parameters.

When the resulting energy waveform (Figure 7) is analyzed, it is noted that there are many signs that the plot is full of errors; however, for the most part these errors are not noticeable without having first calculated the correct values. The most glaring error is that when the capacitive power is 0 W, the capacitive energy should be 0 J; however, the plot shows the value to be in the vicinity of 7  $\mu\text{J}$ . This would not be obvious to most undergraduate students unless they have seen how this problem can occur.

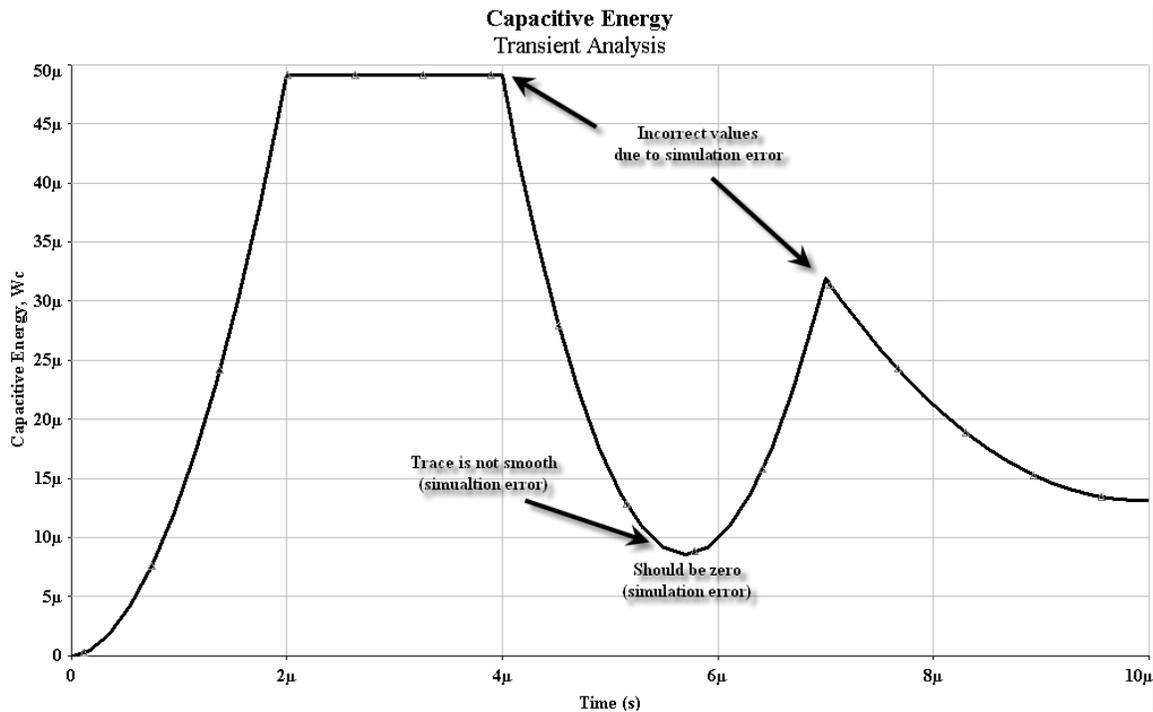


Figure 7. Transient Analysis using default simulation parameters

This problem can be corrected by either increasing the “**Minimum number of time points**” setting from 100 to 10,000 or by decreasing the “**Max time step (TMAX)**” from 1e-5 to 1e-7. The energy plot shown in Figure 8 is the result of modifying the **Minimum number of time points** parameter. It needs to be stressed to the student that the amount of change varies based on the problem and will have to be experimentally determined. Figure 8 is an example of plots produced with MultiSIM Post-Processor utility.

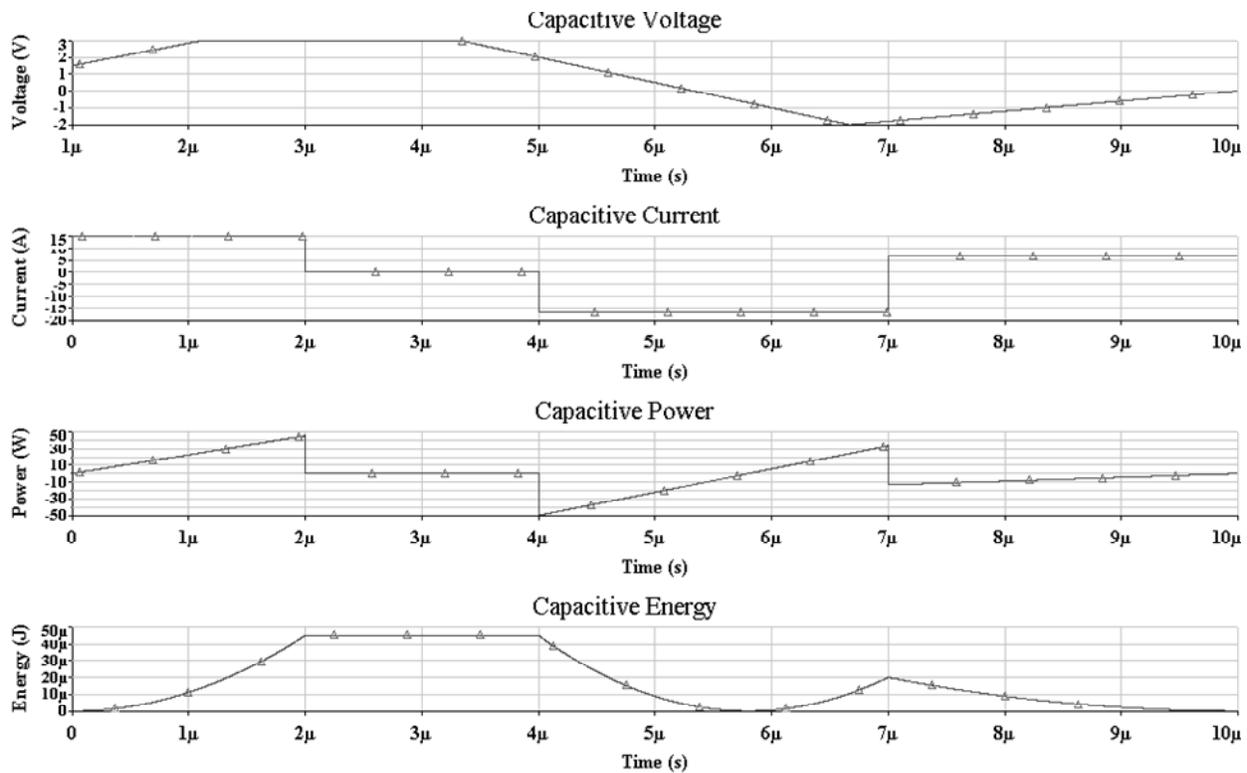


Figure 8. Final transient analysis results using the Post Processor vice the Grapher.

## Transient Analysis

Transient analysis is probably the most evasive of all the common forms of analysis with MultiSIM, as well as with other circuit analysis programs. One can perform a transient analysis and obtain results that might appear to be perfectly fine, and yet may differ substantially from the true results. The reason is that the program is performing an approximate solution of one or more continuous-time differential equations using numerical methods, and possible check procedures may not detect the errors involved.

To illustrate some of the problems and the possible means for circumventing them, consider the series resonant circuit of Figure 9. The properties of this basic circuit are developed in numerous basic circuit books<sup>1,2</sup> and will not be covered here. However, some of the key properties of the circuit are summarized as follows:

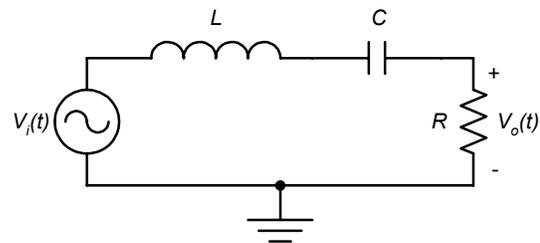


Figure 9. Series resonant circuit.

$$\text{Resonant Frequency} = f_0 = \frac{1}{2\pi\sqrt{LC}}$$

$$\text{Selectivity Factor} = Q = \frac{f_0}{B} = \frac{1}{R} \sqrt{\frac{L}{C}}$$

The quantity  $B$  is the 3-dB bandwidth.

Assume that the circuit is initially relaxed and excited at  $t = 0$  at the resonant frequency by an input sinusoid of the form

$$v_i(t) = V_i \sin 2\pi f_0 t$$

By either Laplace transform analysis or by solving the differential equation, the voltage  $v_o(t)$  across the output resistor can be determined as

$$v_o(t) = V_i \sin 2\pi f_0 t - \frac{V_i e^{-\pi B t}}{\sqrt{1 - (1/2Q)^2}} \sin 2\pi f_0 \left[ \sqrt{1 - (1/2Q)^2} \right] t$$

The first term represents the forced or steady-state response and the second term represents the natural or transient response, which vanishes with increasing time. A conservative estimate for the settling time  $T_s$  required to reach steady-state conditions is

$$T_s \approx \frac{2}{B}$$

For  $Q \geq 10$ , a very close approximation for the output voltage is

$$v_o(t) \approx V_i (1 - e^{-\pi B t}) \sin 2\pi f_0 t$$

The sinusoidal steady-state response  $v_{oss}(t)$  after the settling time is given as

$$v_{oss}(t) = V_i \sin 2\pi f_0 t$$

This simple result could have, of course, been predicted from ac steady-state conditions without all the preceding steps, but the intent was to provide a guide for the overall behavior of the response with computer simulation.

Consider next the MultiSIM simulation of the series resonant circuit shown in Figure 10 with some specific values. It can be verified that the resonant frequency is  $f_0 = 1$  kHz, the 3-dB bandwidth is  $B = 100$  Hz, and the selectivity factor is  $Q = 10$ . The element values have been represented to a higher than practical resolution so that no deviations can be attributed to element value roundoff.

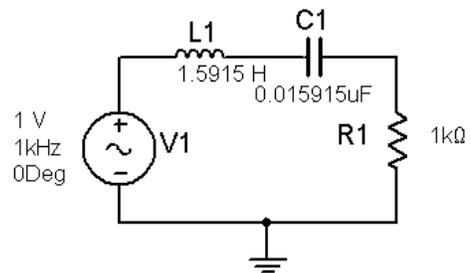


Figure 10. Series resonant circuit with center frequency of 1 kHz and  $Q = 10$ .

For transient analysis, the default initial window is shown in Figure 11. While most prone to error as will be seen shortly, it is tempting at least to obtain one run with the default condition of

**Generate time steps automatically** and that will be the case here. Although it will not be shown, the **Output** slot was set to monitor the voltage across the resistor.

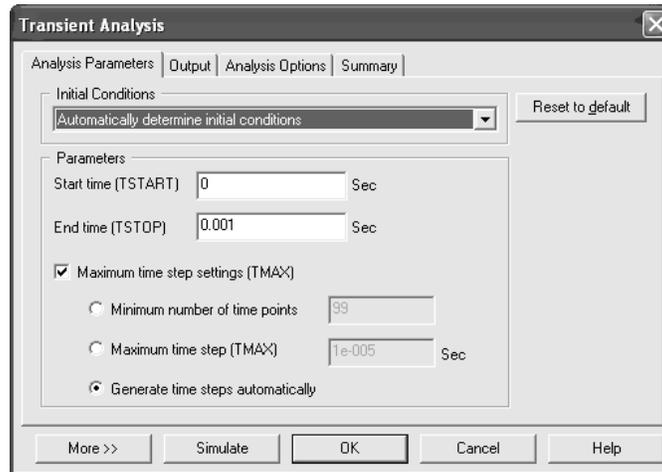


Figure 11. MultiSIM Transient Analysis window with initial default settings.

The overall response from  $t = 0$  to  $t = 2$  ms is shown in Figure 12. The envelope clearly demonstrates the exponential nature expected and steady-state conditions can be assumed by the time  $T_s \approx 2/B = 2/100 = 0.02$  s.

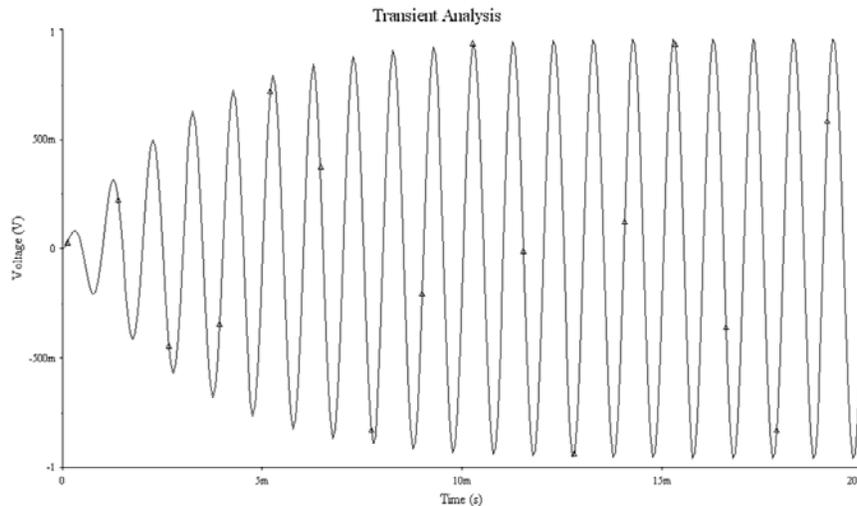


Figure 12. Output voltage from 0 to 20 ms with  $Q = 10$  and default transient settings.

An observation of the response from  $t = 0.02$  s to  $t = 0.022$  s is shown in Figure 13, corresponding to two cycles of the steady-state response. Note that the actual computation begins at  $t = 0$ , but observation does not begin until  $t = 0.02$  s. Theoretically, the steady-state output should be a sine wave having a frequency of 1 kHz and an amplitude of 1 V. The "sinusoidal" signal appears a bit jagged due to the scarcity of points, and the output amplitude is about 0.96 V, representing an error of approximately 4%. Thus, this solution, while prone to error and some misrepresentation, is at least in the correct "ball-park".

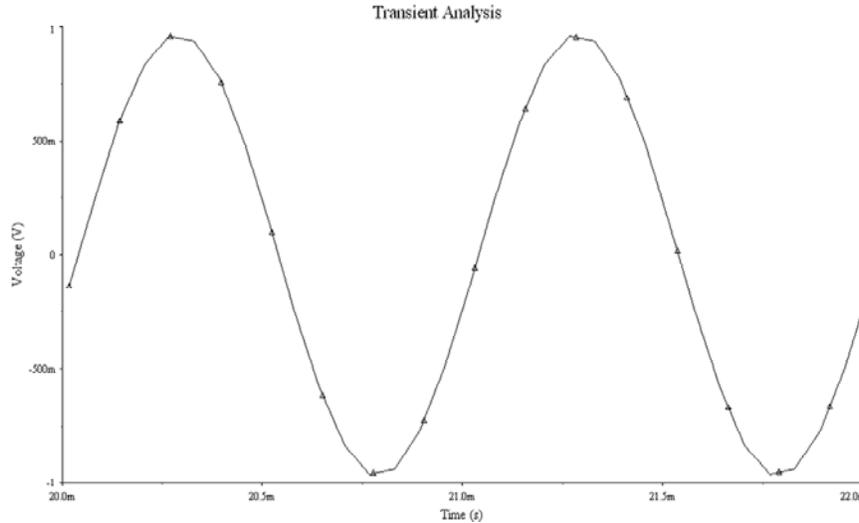


Figure 13. Steady-state output voltage with  $Q = 10$  and default transient settings.

The reader should not be misled by the title **Transient Analysis** on this graph since it arose from the MultiSIM analysis option by that name, and if the natural response vanishes by the time of observation, the result seen on the graph will be the steady-state response.

Assume next that the bandwidth is reduced to 10 Hz based on increasing the  $Q$  to 100. The resulting MultiSIM circuit is shown in Figure 14. The settling time is now increased to  $T_s \approx 2/B = 2/10 = 0.2$  s. A transient run from  $t = 0$  to  $t = 0.2$  s is shown in Figure 15 and an observation from  $t = 0.2$  s to  $t = 0.202$  s is shown in Figure 16.

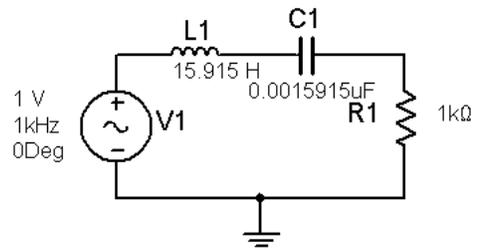


Figure 14. Series resonant circuit with center frequency of 1 kHz and  $Q = 100$ .

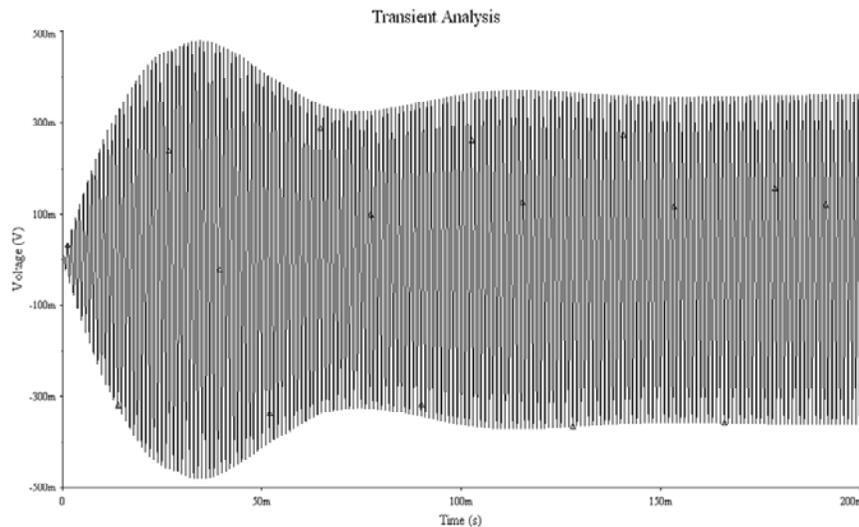


Figure 15. Incorrect output voltage from 0 to 200 ms with  $Q = 100$  and default transient settings.

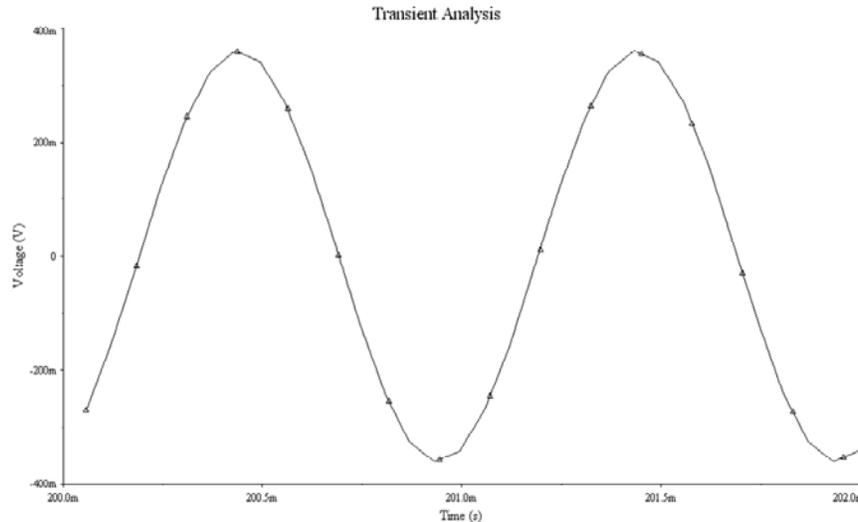


Figure 16. Incorrect steady-state output voltage with  $Q = 100$  and default transient settings.

Clearly, the results are not as they should be. The transient interval exhibits an overshoot, which should not be the case, and the steady-state peak value is less than 0.5 V. Thus, there are significant errors in both the transient interval and the steady-state interval.

To fix the problem, the analysis option on the Transient Analysis window is changed to **Maximum time step** and the value of this parameter is set to  $10^{-6}$  s = 10e-6 s. This corresponds to 100 points per cycle at the frequency of 1 kHz. The overall output voltage is now shown in Figure 17 and the steady-state voltage is shown in Figure 18. The amplitude of the steady-state output voltage is 1 V, and the shape of the transient response is correct. Moreover, the steady-state output signal clearly appears as a sine wave.

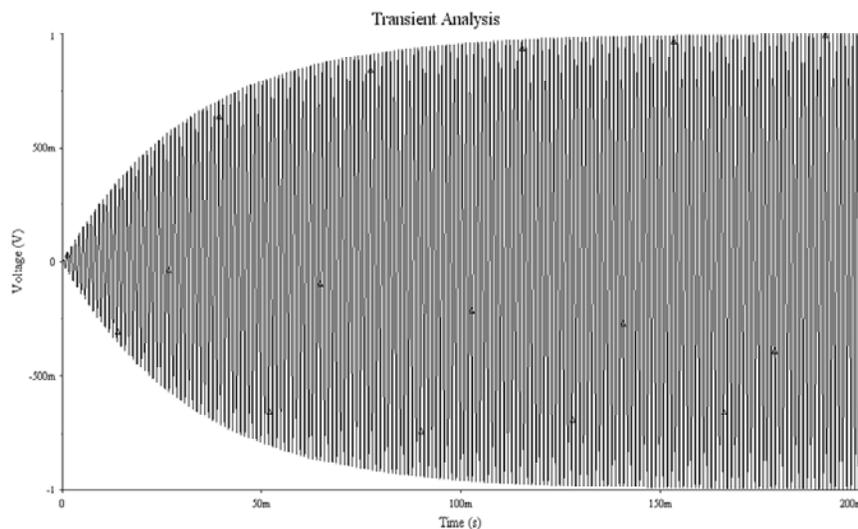


Figure 17. Correct output voltage from 0 to 200 ms with  $Q = 100$  and a maximum time step of 10 microseconds.

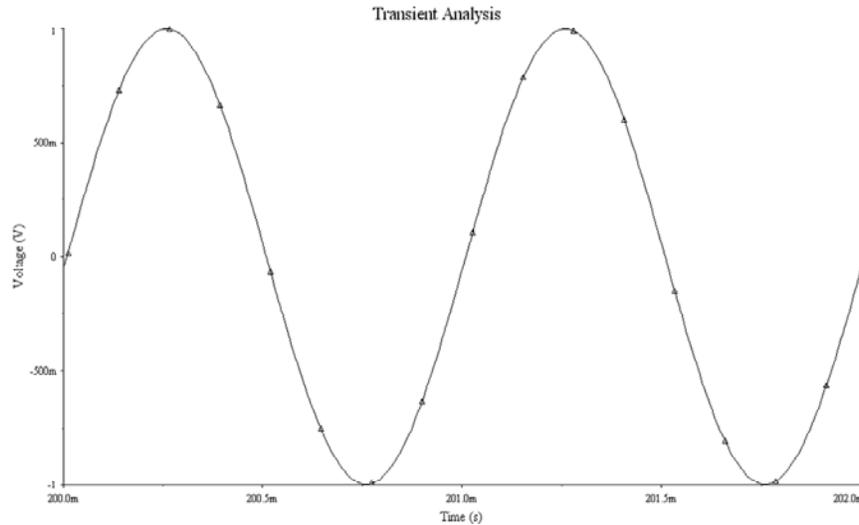


Figure 18. Correct steady-state output voltage with  $Q = 100$  and a maximum time step of 10 microseconds.

Since the correct form was known in advance, it was quite straight-forward to adjust the program parameters to obtain a good response, but how would one in general know what to do if the form of the response were not known? For example, a student might obtain the runs of Figures 15 and 16 and think that the results are correct since there is no error warning for these cases. This issue will be discussed in the section that follows.

### Some Transient Analysis Guidelines

Unfortunately, it is not possible to cover all the snafus that might occur in transient analysis, but a few general guidelines can be covered. Indeed, error messages may actually be generated in some cases, and when this occurs, it is obvious that something is wrong. The guidelines given here are those that may steer one in the correct direction even when no error message occurs. Some general guidelines follow.

- Use the default condition of **Generate time steps automatically** only as a quick check or first run and view the results with a degree of skepticism. The results should be trusted as being fully correct only after some additional analysis or with some simple circuits and excitations where certain properties of the results can be readily checked with simple analysis.
- Either of the options **Minimum number of time points** or **Maximum time step** are generally capable of providing good results *if used properly*. However, the focus here will be on the latter option and subsequent discussion will assume that option.
- If the signal consists of one or more sinusoidal functions, it is recommended that a minimum of 100 points per cycle at the highest frequency be used as a starting point. Thus, if the highest frequency is  $f$ , the **Maximum time step** should be a value no greater than  $0.01/f$  if possible.

- Decide if the natural response or the forced (steady-state) response or both are of interest and adjust the **Start time** and **End time** accordingly. For a sinusoidal excitation, a reasonably conservative estimate of the settling time is  $1/B$  for a low-pass system and  $2/B$  for a band-pass system.
- Be especially careful if the circuit has a narrow bandwidth or has a high degree of sensitivity to variations in parameter values. For certain narrow band-pass circuits, it may be necessary to transform to low-pass equivalents to obtain good results, but that topic will not be pursued in this paper.
- Finally, and this is very important, make at least two runs with different choices of the **Maximum time step** to see if it makes any difference in the results. If the results differ by more than the normal roundoff, one or both are wrong and further runs with different time steps are needed. Trust the results only after they remain the same with several different choices of the time step.

## Conclusions

Although MultiSIM is an extremely valuable circuit simulation tool, the user must exercise care in its use. The first example shown in this paper reminded the reader that in many cases, the components in computer simulations may be ideal, requiring the user to be aware of the internal characteristics of the components, and the impact this may have on the simulation.

Also, as with any engineering tool, using default simulation parameters and blindly accepting results can result in errors. It has been shown in this paper that there are several ways in which to verify the results of MultiSIM simulations. The first, and the best way to assure that results are correct, is for the user to make some “benchmark” simulation runs with conditions that give known results. Secondly, if this is not possible, make several simulation runs using different time steps (or numbers of steps) to see if each gives the same results.

At Old Dominion University, EET students using Multisim are cautioned concerning errors that can result by misusing the software. Examples similar to the ones illustrated in this paper are performed in class so that the students can have first-hand exposure to possible errors that can occur. In laboratory courses in which students are performing experiments using Multisim, introductory lab lectures warn the students of common analysis pitfalls that can cause erroneous results, and ways to avoid them. Since the ODU laboratories are equipped with digital projectors, instructors show the students sample experiments that have “gone wrong” because they contain analysis errors. Although student still often make analysis errors, by showing “bad examples”, they are now aware of the more common problems that can occur.

## **Bibliography**

1. W. D. Stanley, *Transform Circuit Analysis for Engineering and Technology*, 5<sup>th</sup> Edition, Prentice-Hall, 2002.
2. W. D. Stanley, *Network Analysis with Applications*, 4<sup>th</sup> Edition, Prentice-Hall, 2002.
3. Herniter, Marc E., *Schematic Capture with MultiSIM 7*, Prentice Hall, 2004, ISBN: 0-13-118755-4
4. *Multisim 9, Simulation & Capture*, CD-ROM, Electronics Workbench, 2005.