## MULTISIM DEMO 4.3: INTRODUCTION TO SIGNAL SOURCES AND THE TRANSIENT ANALYSIS

*In the previous Multisim Demo (4.2) we began to look at time-varying sources in transient analysis by using the function generator and the oscilloscope, both very useful instruments in the real world as well as in Multisim. There are other ways to generate periodic signals, and there are different ways to analyze these signals.*

Technically a lot of the simulations already discussed in this manual are transient (time-varying) analyses. Whenever you press the  button (or F5) you're running the Interactive Simulation, a transient simulation of a virtual lab where we record measurements off of the various instruments and tools we are utilizing. In some ways the Interactive Simulation is a virtual world.

We can use the oscilloscope in the Interactive Simulation in order to plot transient signals, however there is a also a special simulation devoted solely to producing time-varying plots of signals: the Transient Analysis. The Transient Analysis is run for a pre-defined time period and allows us to plot selected variables and/or combinations of variables. Just like in the DC Operating Point Analysis or the DC Sweep Analysis, the Transient Analysis allows us more control of what we measure and allows us to combine and manipulate variables. For example, we can't really *plot* power vs. time in the Interactive Simulation mode, but through multiplying a voltage variable by the appropriate current variable we can plot power vs. time in a Transient Analysis.

To demonstrate the capabilities of Transient Analysis, let's build an inverting amplifier with a clock signal for an input and then measure the input and output signals. Obtain a CLOCK_VOLTAGE source through using the component hierarchy shown in Fig. 4.3.1 below.

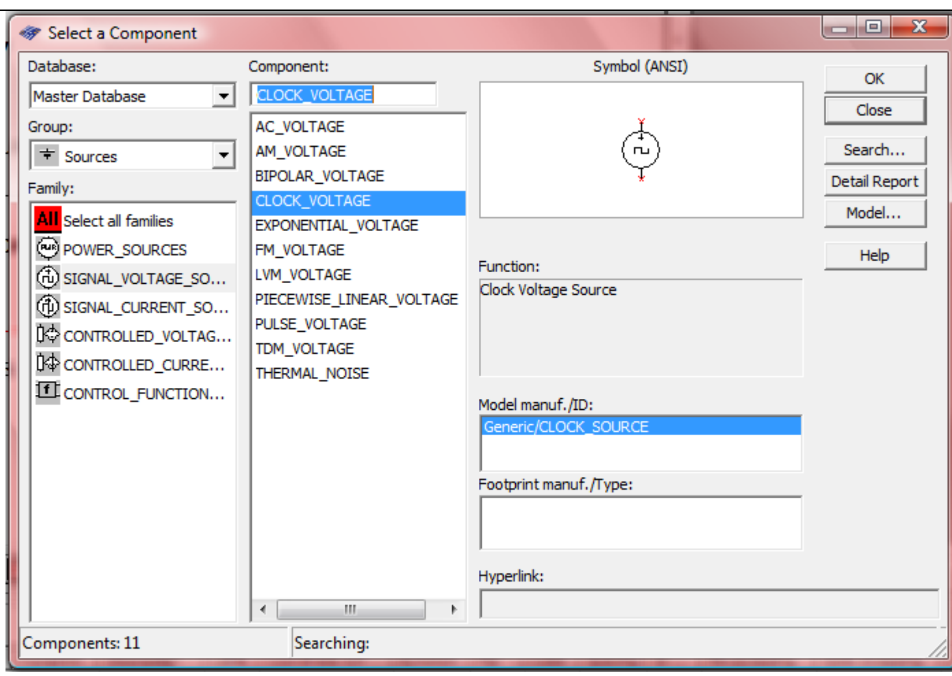

**Figure 4.3.1** Component hierarchy for the CLOCK_VOLTAGE component.

Place source onto the schematic and build the inverting amplifier shown in Fig. 4.3.2. Use the 5-terminal op amp. If you are unsure as to how to attach the power supply rails see Multisim Demo 4.2.

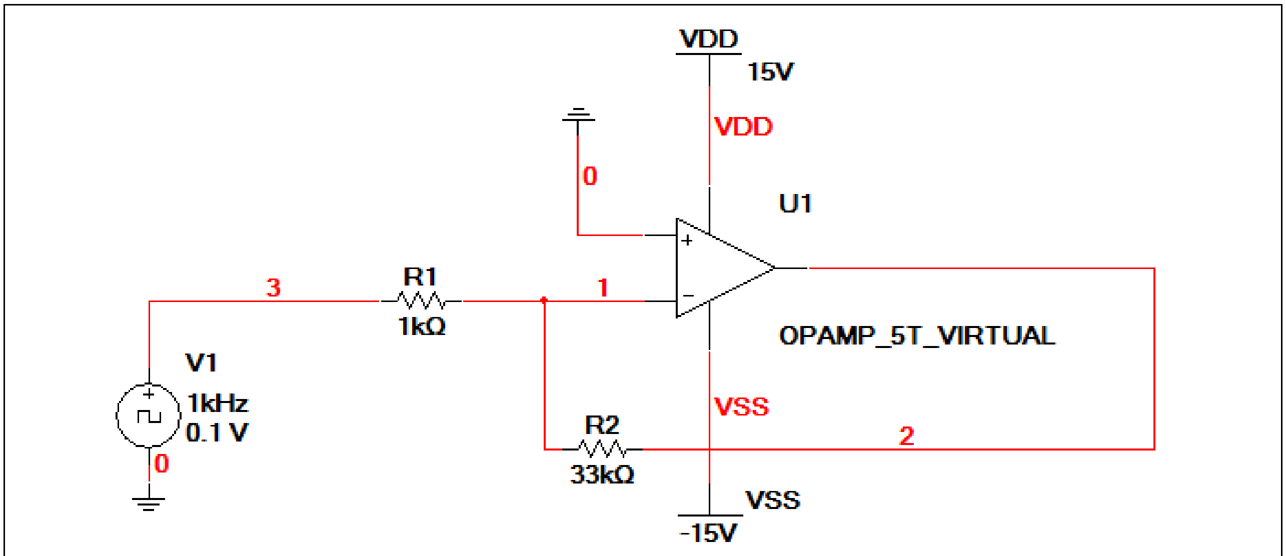


**Figure 4.3.2** Inverting amplifier .

We'll leave the Clock Voltage parameters at their default values. (1 kHz, 0 V offset, and 0.1 V pulse amplitude) Let's go and start the Transient Analysis. Go to Simulate>Analyses>Transient Analysis or use the button as shown in Fig. 4.3.3 below.

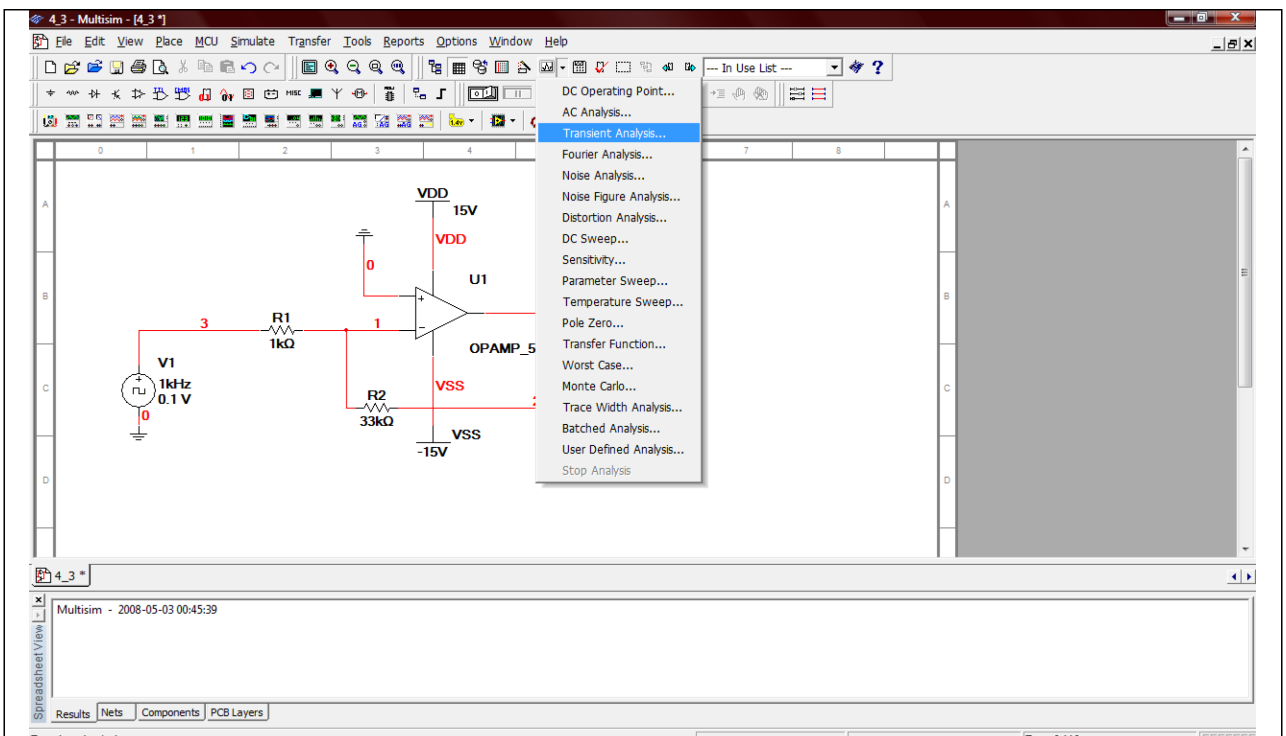


**Figure 4.3.3** Selecting the Transient Analysis.

The Transient Analysis window should appear as shown in Fig. 4.3.4 below. Under the Analysis Parameters tab:

1. Set the Start time (TSTART) to 0.1 s.
2. Set the End time (TSTOP) to 0.11 s.
3. Leave "Maximum time step settings (TMAX)" checked.
4. Ensure that "Generate time steps automatically" is selected.

Now click on the Output tab, and add V(2) and V(3), the output and input signals of the amplifier, respectively, to the selected variables for analyses field as shown in Fig. 4.3.5 at the bottom of this page.

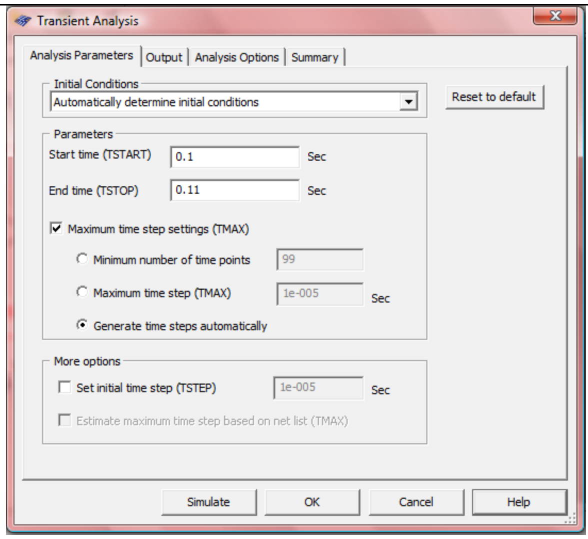


**Figure 4.3.4** Transient Analysis window showing time parameters.

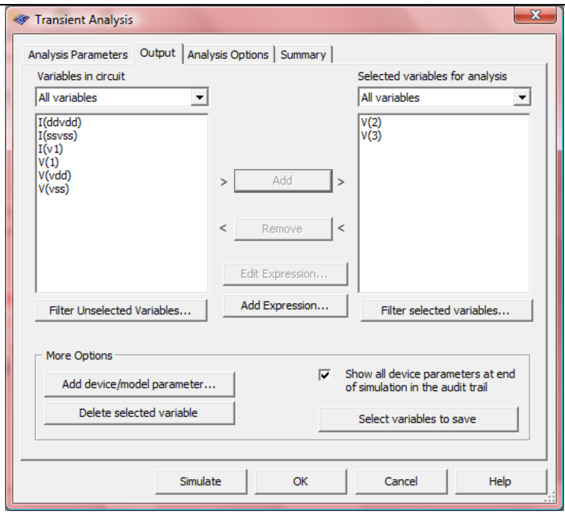


**Figure 4.3.5** Selecting the input and output signals for analysis.

 

So what we've just done is set up a simulation which will plot the input and output signal of the inverting amplifier in Fig. 4.3.2 from 0.1 s to 0.11 s. When you are ready, press Simulate. Your resulting plot should be similar to Fig. 4.3.6 below.

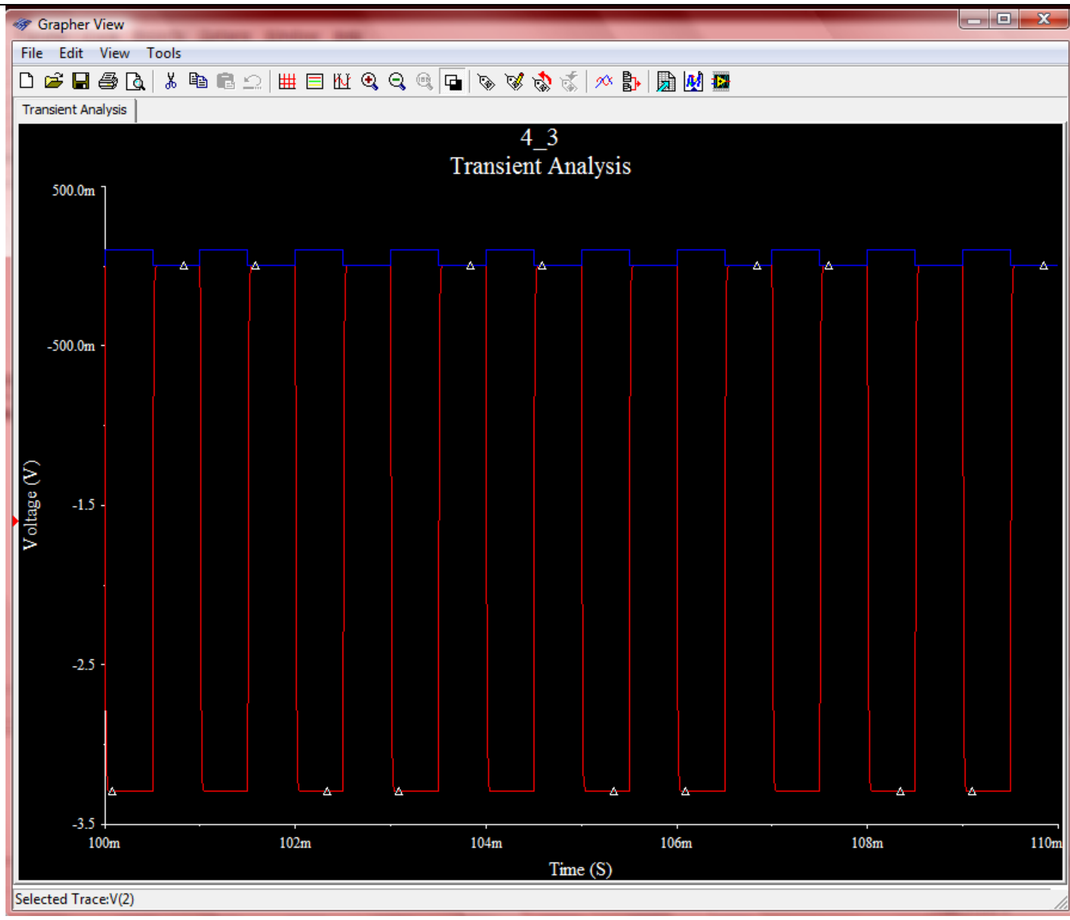


**Figure 4.3.6** Input and output plots of the inverting amplifier being fed with a 1 kHz, 0.1V amplitude square wave.

Click on the  button to bring up a legend. As expected the voltage with the much larger swing is V(2), the output of the amplifier.

Let's see what values we are dealing with. Click on the  button to bring up a pair cursors. On the left side of the screen, two almost invisible looking triangles appear. They are the cursor markers. Click there and drag the cursors into the body of the graph. Most likely you got cursor number 1, but in some instances it will give you cursor number 2.

In the cursor window, x1 refers to the x-axis location of cursor 1 and y1 refers to the y-axis location of cursor 1. Thus in Fig. 4.3.7 under the V(2) heading, x1 is telling us that the cursor1 is located at 102.233 ms and y1 is telling us that the value of V(2) at that point in time is -3.2939 V.

As shown in Fig. 4.3.7, the cursors reveal that when the input signal V(3) is 100 mV, the output signal is -3.293 V, which corresponds to a gain of about -33, which is exactly what we should see because R1 is 1 kΩ R2 is 33 kΩ. Additionally, when the input signal is at 0 V, the output signal is at 0V as well, which is what we should expect.

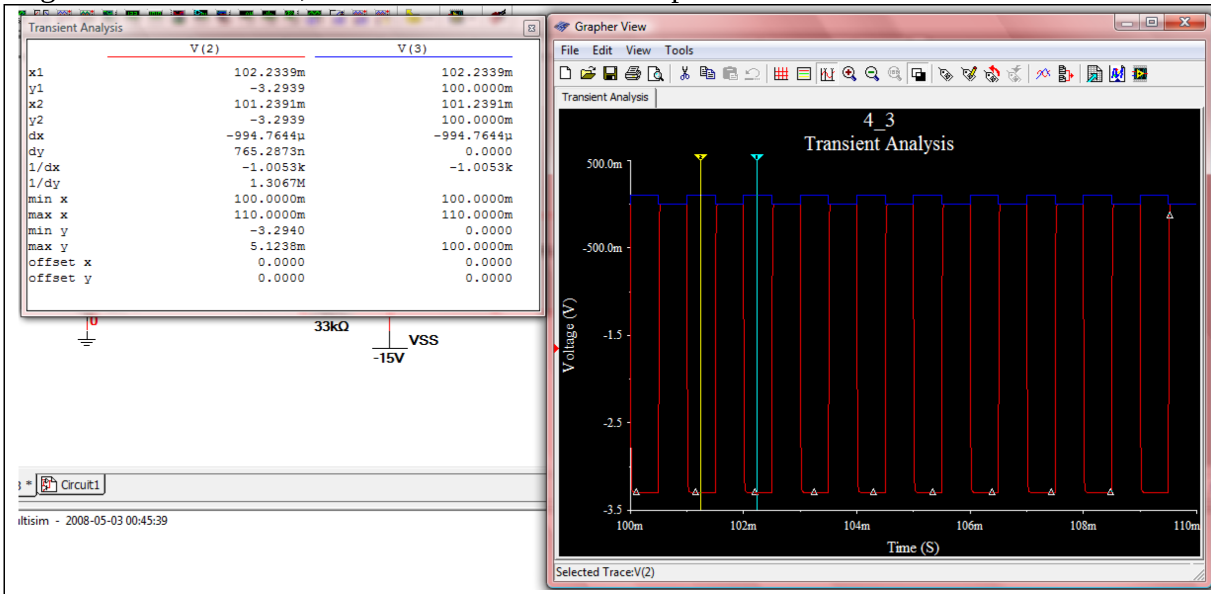

**Figure 4.3.7** Using the cursors to find quantifying values of the output signals.

Now a question which may have popped into your mind is why even bother with the 5-terminal op amp? Well the reason is that it has built in non-idealities. We will now investigate the finite time it takes the amplifier to respond to the input signal. In Fig. 4.3.7, it seems as if the output responds immediately to any change in input. If we shrink the time span of our Transient Analysis, we can zoom in around an edge of the pulse to get a better view of the response.

Exit the Grapher window and bring up the Transient Analysis window again. Leave TSTART at 0.1 s, but change TSOP to 0.101 s. Additionally, click select Maximum time step (TMAX) and set it to 1e-006 Sec, as shown in Fig. 4.3.8 below. This limits the time step size to 1 µs, and therefore ensures a good resolution. Press Simulate.

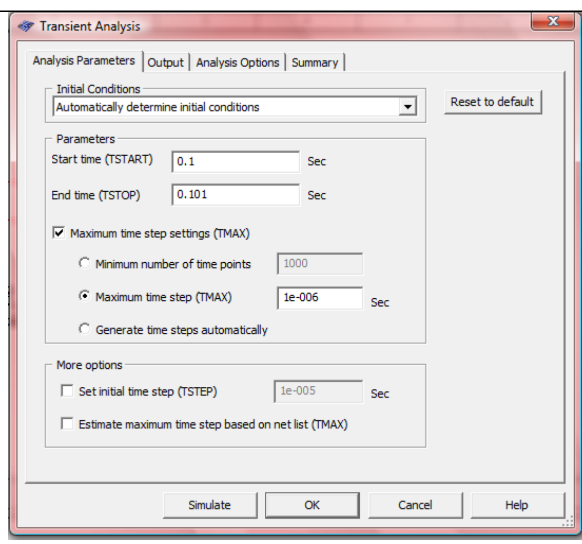

**Figure 4.3.8** Adjusting the Transient Analysis.

*Note that this simulation may take a lot longer than the first one did. It is because you are taking data at a lot more points. If you ever accidentally start a simulation that is taking way too long, you are not doomed to wait it out…Click on the down-arrow on the*  *button and go to the bottom. While the simulation is running, the* ***Stop Analysis*** *option will be available. Select it to abort the simulation. This will save you a lot of time.*

The output of the simulation should resemble that shown in Fig. 4.3.9 below. Note that the rising edge should be apparent. Using the cursors, (particularly the dx value), it seems that the output of the amplifier takes about 40.1 µs to rise from about -3.3 V to 0 V.

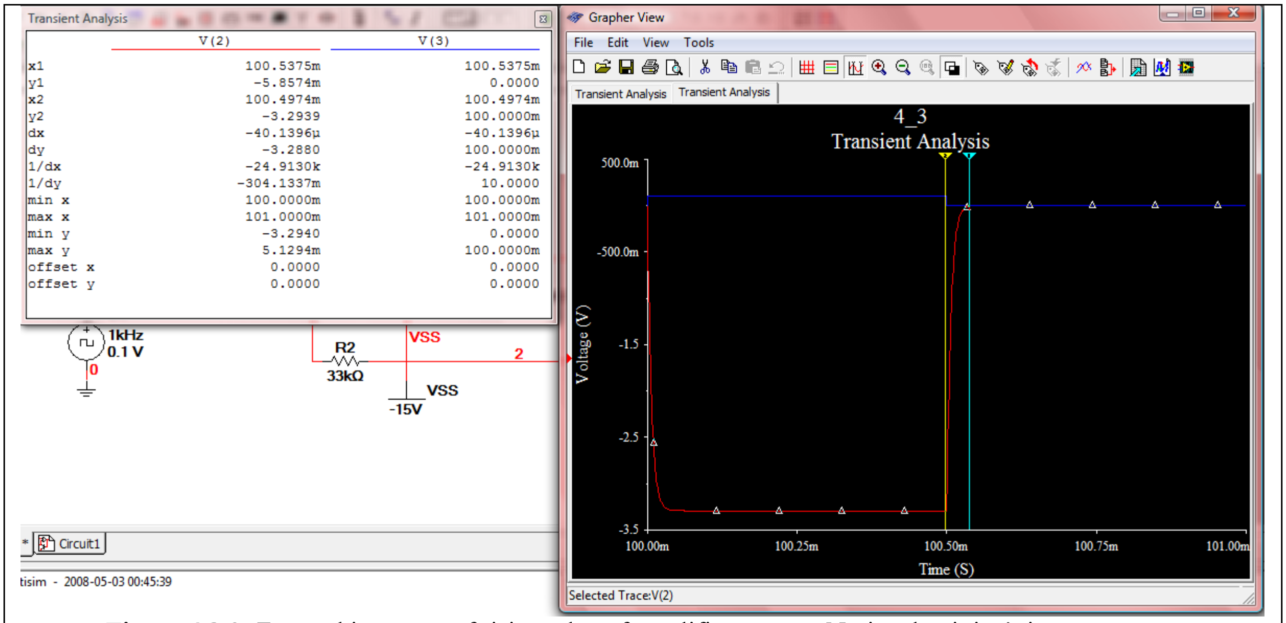


**Figure 4.3.9** Zoomed in output of rising edge of amplifier output. Notice that it isn't instantaneous.

Now let's zoom in just a little bit more. As shown in Fig. 4.3.10 on the next page, reset the TSTART to 0.01 s, adjust the TSTOP to 0.0101 s, and set TMAX to 1e-007 sec. Press Simulate. The resultant plot should resemble that in Fig. 4.3.11. What we are now looking at is the response of the inverting amplifier to a rising input. Since its an inverting amplifier the response of the output is falling.

Using the cursors we see that it takes about 37 µs to reach about -3.3 V from 0 V. (We know the voltage starts at 0 V because of the simulation starts right at the rising edge of the input.)

Thus we can see that the virtual op amp has non-idealities built into it, which we can uncover when we know where to look. Always remember that nothing in real life is capable of truly instantaneous response.

 

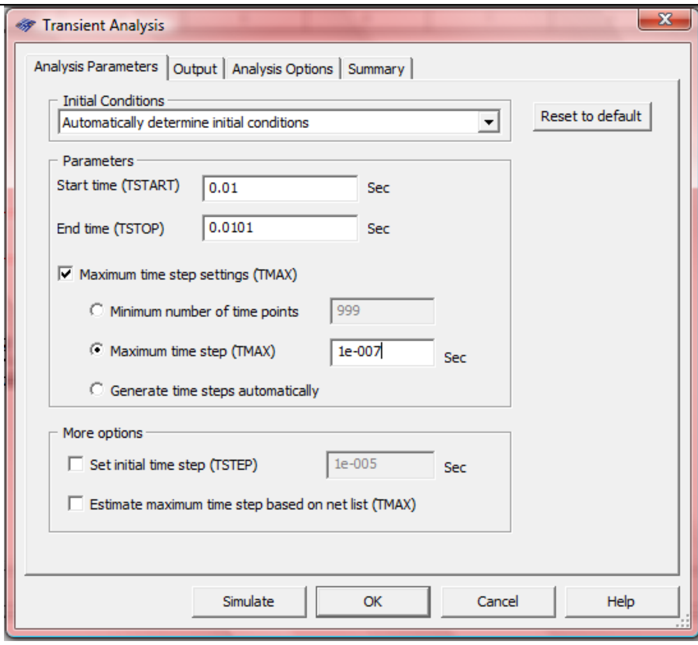


**Figure 4.3.10** Transient Analysis settings for an even closer look.

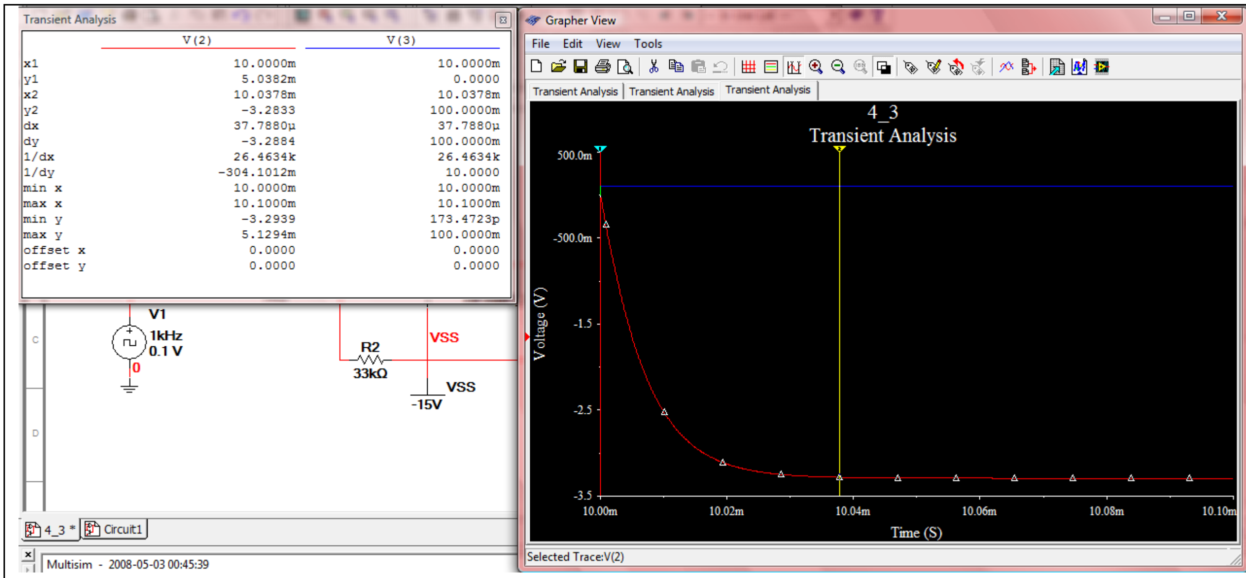


**Figure 4.3.11** Zoomed-in view of the falling response of the amplifier's output.

 