



Department of Electrical & Computer Engineering

ELECTRONIC WORKBENCH MULTISIM TUTORIAL (BASIC)

Class Instructor : Dr. Subbarao V. Wunnava
Generated by : Temitope Marcus & Rafael Romero
Courtesy : Electronic Workbench / Vivek Jayaram / Jaime Montenegro
Updated : Spring.2006, 06 Mar, 2006

INTRODUCTION

Electronics Workbench's MultiSim is a circuit simulation platform, similar to other SPICE programs, that can model the behavior of a particular analog or digital circuit. In this capacity you have a software program in which you can model any conceivable circuit design, examine the corresponding circuit for values at particular components or probe the behavior of the entire circuit by performing DC, AC or transient analyses, and much more. With this tool you have access to thousands of parts and components, available at your disposal. This means that in addition to the convenience of not actually having to physically have components available, you have limitless combinations of circuit designs.

Electronics Workbench's DesignSuite software includes other programs such as Multicap, Multisim, Ultiboard and Ultiroute. In this tutorial we focus on MultiSim9, the circuit simulation platform. This is included in the DESIGNSUITE Freeware edition 9 which can be downloaded from the following website at the time of this writing:

www.electronicworkbench.com/html/proprod_dl.html

Click on the "Download Now" link and simply download the program to a temp folder. Once finished, double click on the file **dsfree9.exe** to execute it. This is a self-extracting file and the setup routines will start-up automatically. Once finished you will have successfully installed the **DesignSuite** software including MultiSim.

This tutorial is available at our website at:

<http://vlsilab.fiu.edu/projects.html>

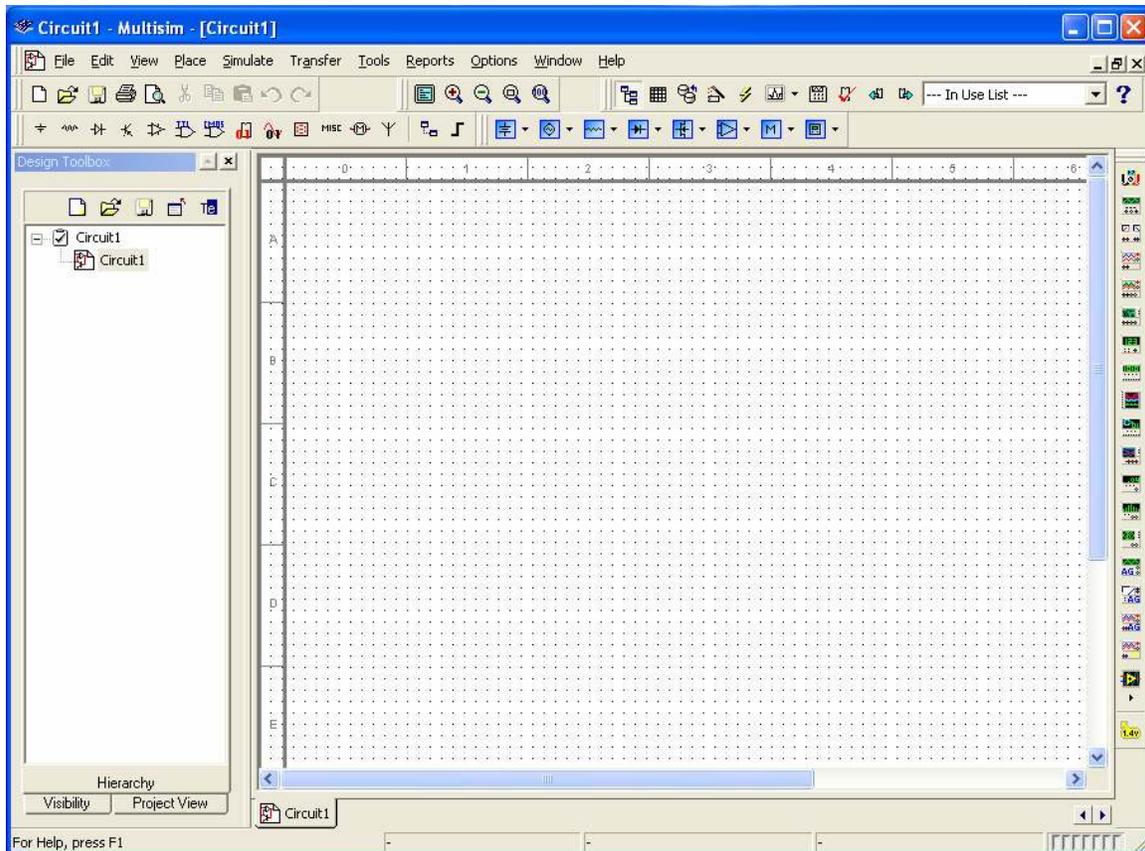
Click on the Electronic Workbench tutorial.

PROCEDURE FOR SIMULATION IN SCHEMATICS

Begin by first opening up MultiSim 9.

For Windows users the default location can be found by clicking:

Start->All Programs->Electronics Workbench->DesignSuite Freeware Edition 9->MultiSim 9.
The following program window opens up.



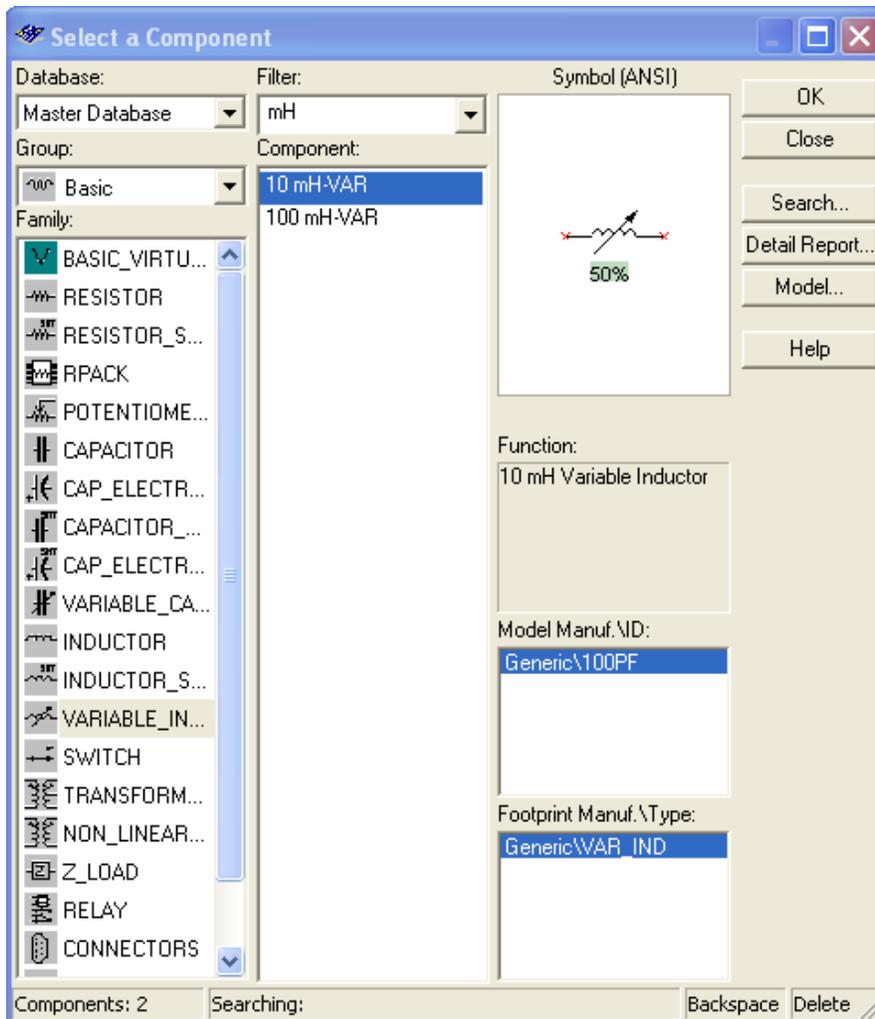
To create/open a schematic file:

As a default a blank file named “Circuit 1” is opened up on the workspace. To save a schematic under a different name simply click on **File/Save As** on the toolbar and enter the name of your choice. To open an existing file click on **File/Open** on the toolbar and select the file to open.

We’ll begin by placing parts to simulate the common voltage divider.

Placing the components:

On the toolbar select **Place/Component**, the following window will appear:



Here is where you will find all the components needed to suit your circuit. In our case we are searching for Resistors, a VDC and ground components.

In the **Group:** window select **Basic**. Next with your mouse or arrow keys select **Resistor**. On the middle part in the **Component:** window you will see a list of resistors of varying size and tolerances. For simplicity select the 1K Ω 5% resistors and click **OK**. Move the component around to the correct position of your choosing. Left Click to place the component on MultiSim window, right click to cancel placement. In MultiSim you also have the option of placing virtual components. The simple difference is that the parameters of **virtual components** can be set to any value. Placing a virtual component can be done by selecting the component from the blue icons located on the toolbar as shown below:



Getting back to our circuit, place one more 1K Ω 5%. Now we will find and place a VDC and ground components. Once again on the toolbar select **Place/Component**. In the **Group:** window select **Sources**. In the **Family:** window select **Power_Source**. In the **Component:**

window select **DC_Power**. Place it in the MultiSim window and also select the **Ground** component. You can also use the following toolbar to place non-virtual components:



Editing the component:

Once placed in the MultiSim window.

Moving a component:

Click on the component once to select it. Hold and Drag the component to a new location.

Rotating a component:

Right click on the component. Select **90°Clockwise [Ctrl+R]** to rotate the component 90 degrees clock wise or **90° CounterCW[Ctrl+Shift+R]** for counter clockwise rotation.

Flipping a component:

Right click on the component. Select **Flip Horizontal[Alt+x]** to flip the component horizontally. Select **Flip Vertical[Alt+y]** to flip the component vertically.

Deleting a component:

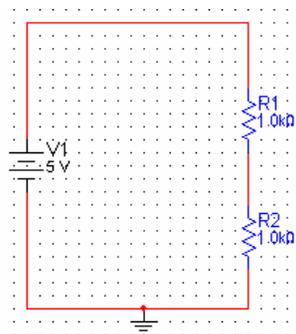
Right click on the component. Select **Delete[Delete]**.

Wiring:

From the toolbar select **Place/Wire** drag and place the wire. Alternatively, simply click on the mouse over a terminal edge of a component and hold, then just drag the wire to wherever you choose.

ANALYSIS SETUP

Our voltage divider circuit should now look like this.



Next, we perform some analysis of the circuit.

Like other circuit simulation platforms, Multisim can display the resulting voltage and current values resulting after running a simulation. Here is how:

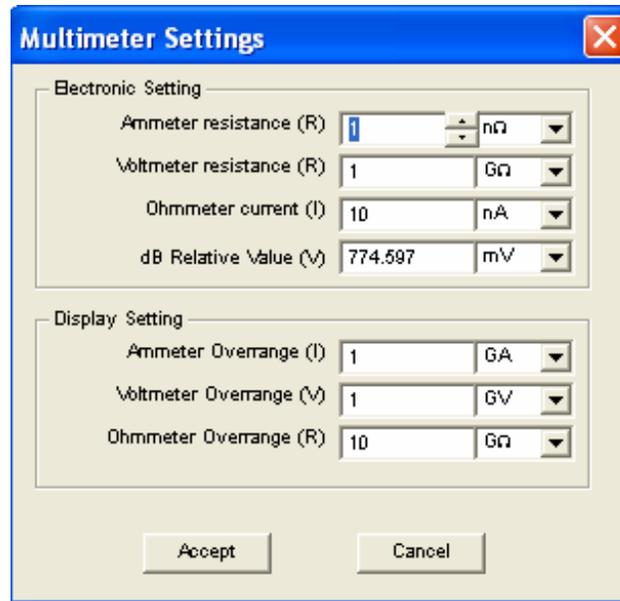
First, save your circuit by selecting **File/Save As** and entering the name of your choosing. Next, run the simulation by selecting **Simulate/Run [F5]**. Alternatively you can also run simulation by clicking of this button in the toolbar: . From the **Simulate/Instruments** select **Multimeter**. Place it next to the schematic and it to the location you want analyzing. With the multimeter you have the option of measuring current, voltage, resistance, and measuring in db.



Multisim 9 also provides other multimeter devices, and oscilloscopes tools which are exact replicas of their real world counterparts. These include Agilent function generator, multimeter, oscilloscope, and a Tektronics oscilloscope. Shown below in the Agilent multimeter:

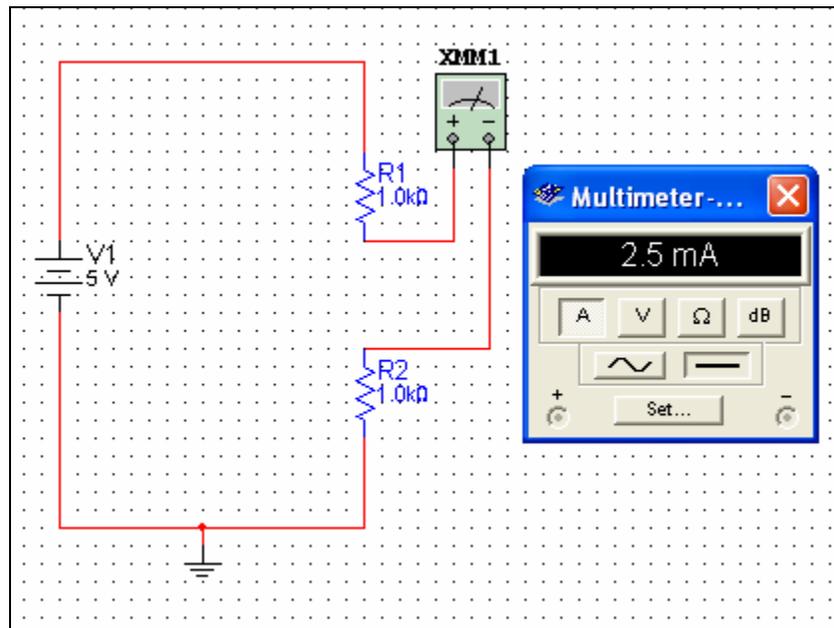


Getting back to the original multimeter,by clicking on the Set button you can also have the option of changing the multimeter settings

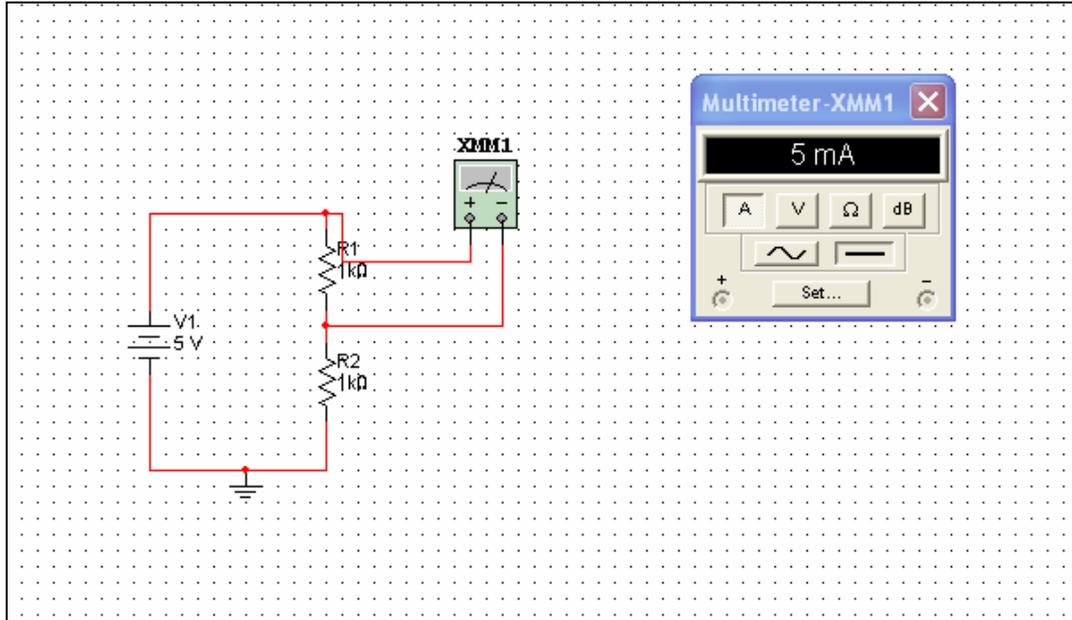


Resulting measurements:

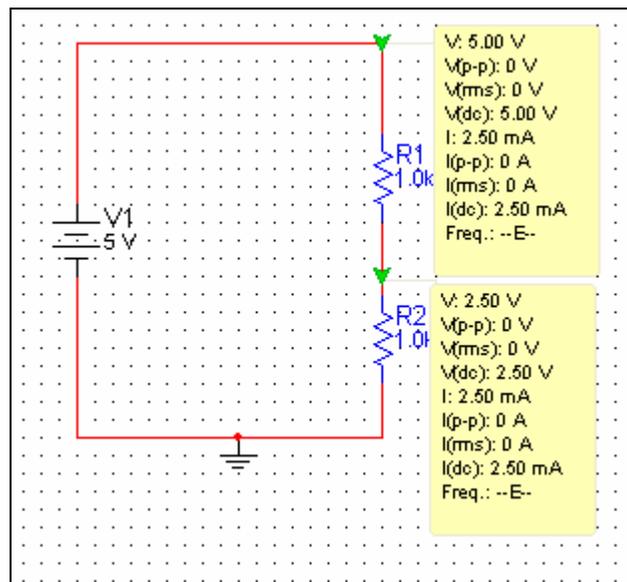
Voltage through the 1K resistor



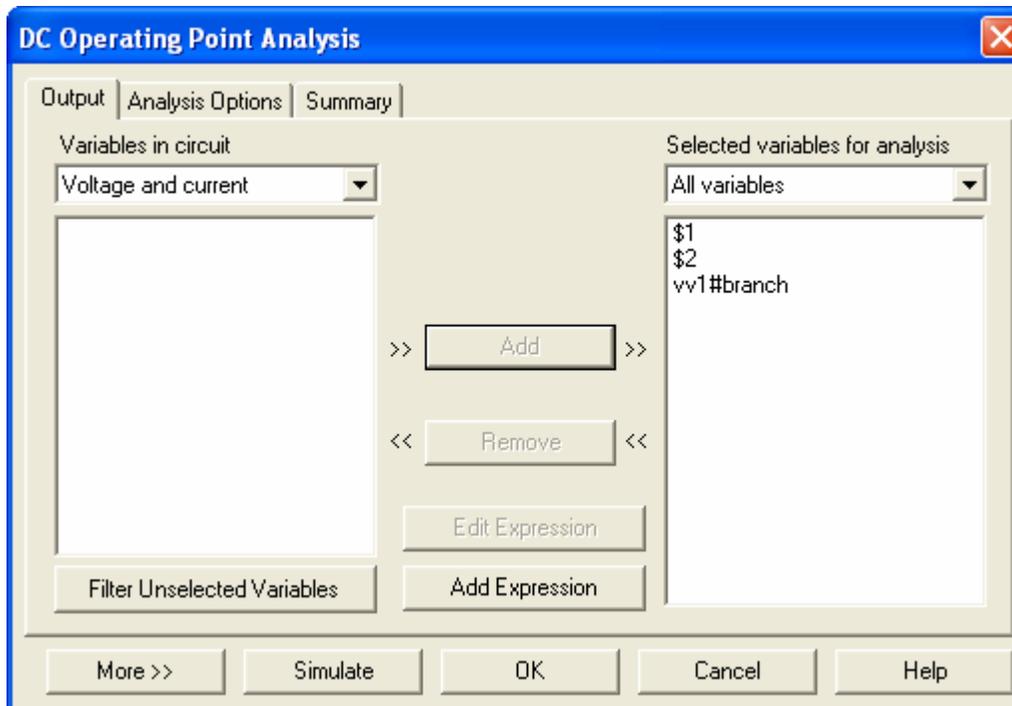
Current across the 1K resistor



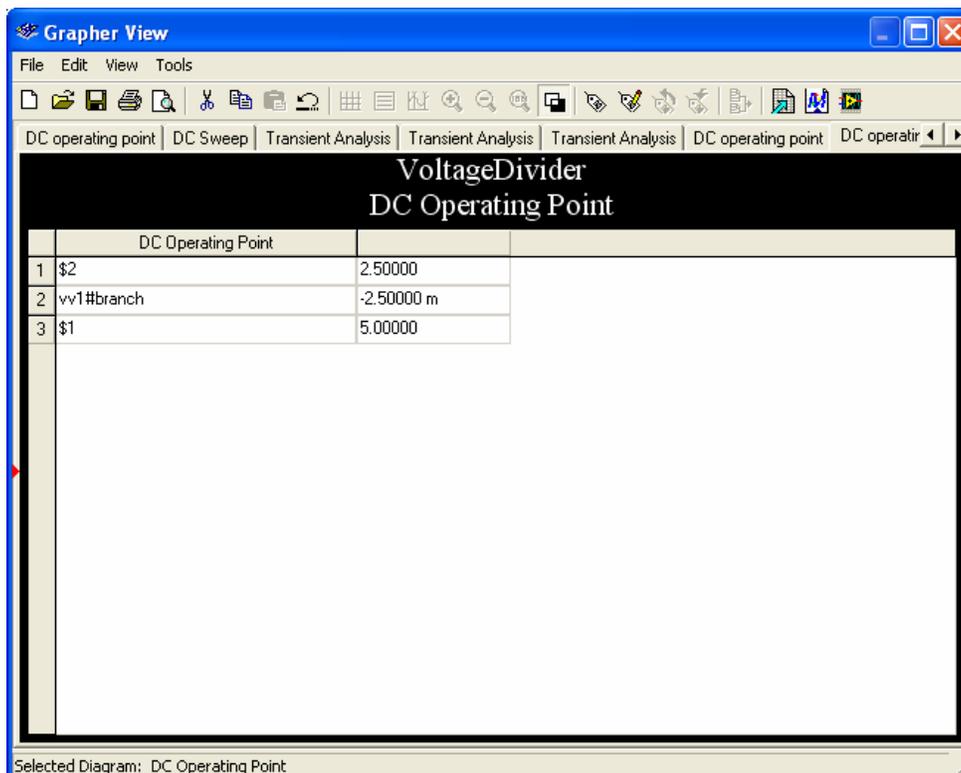
Another way of providing measurements is by using the Measurement Probe, shown as this icon on toolbar . You can either place the probes before running simulation at the places you want analyzing or just run the simulation then click on the measurement probe icon and drag the icon to wherever part of the circuit you need analyzing.



Lastly you can also view the corresponding voltage and current results by running DC Operating Point analysis. Select the **Simulate/Analyses** and select **DC operating Point**. This determines the DC operating point of a circuit and produces a detailed report of voltages and currents at each circuit node.

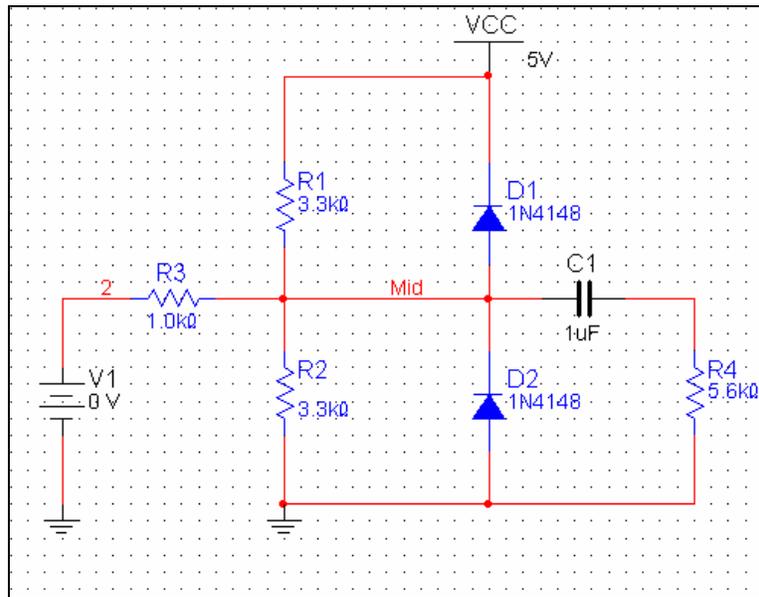


The window above will appear. Since we are interested only in the current and voltage, select those parameters in the **Variables in circuit window**. Next, add all in the first window by selecting each and adding them into the second window. Click on **Simulate** when done. The following results should be obtained.



Now that we have a basic understanding of how to assemble a circuit by finding its parts, placing them, wiring them, changing their values and/or references along with some additional options we move on to running the simulations on MultiSim. Here we will perform various analyses on some sample circuits.

Example 1: Construct the following circuit

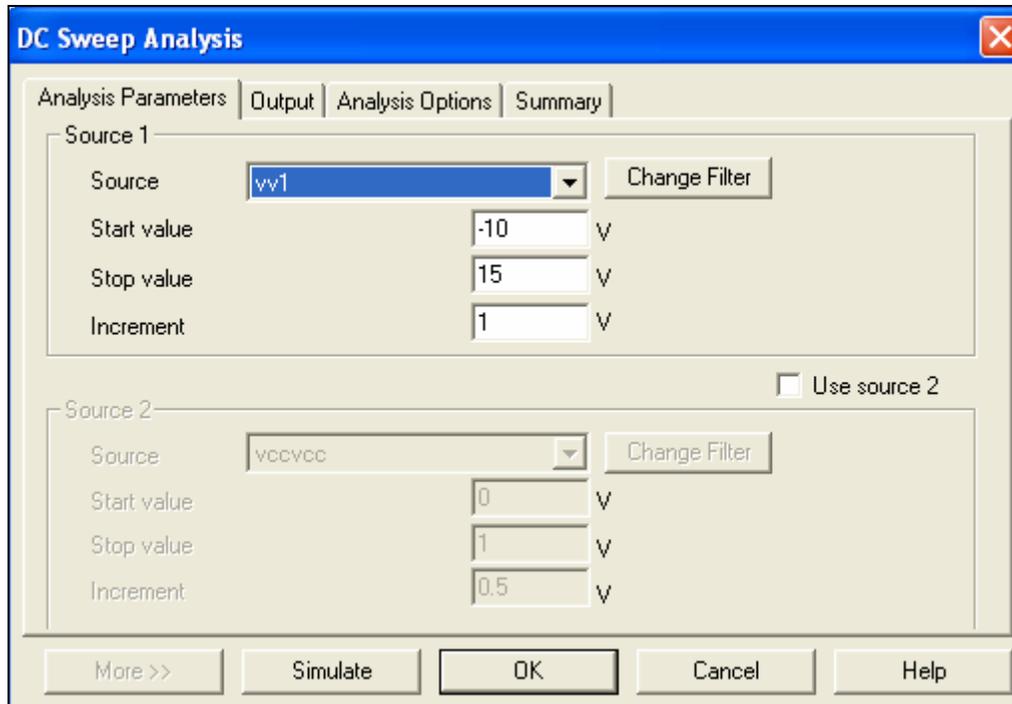


Parts lists consist of DC_POWER, C, R, 1N4148, GROUND. Once you finish the schematic, save it by selecting **File/Save As** from the toolbar, and now you are ready to run various simulations.

DC Sweep Analysis

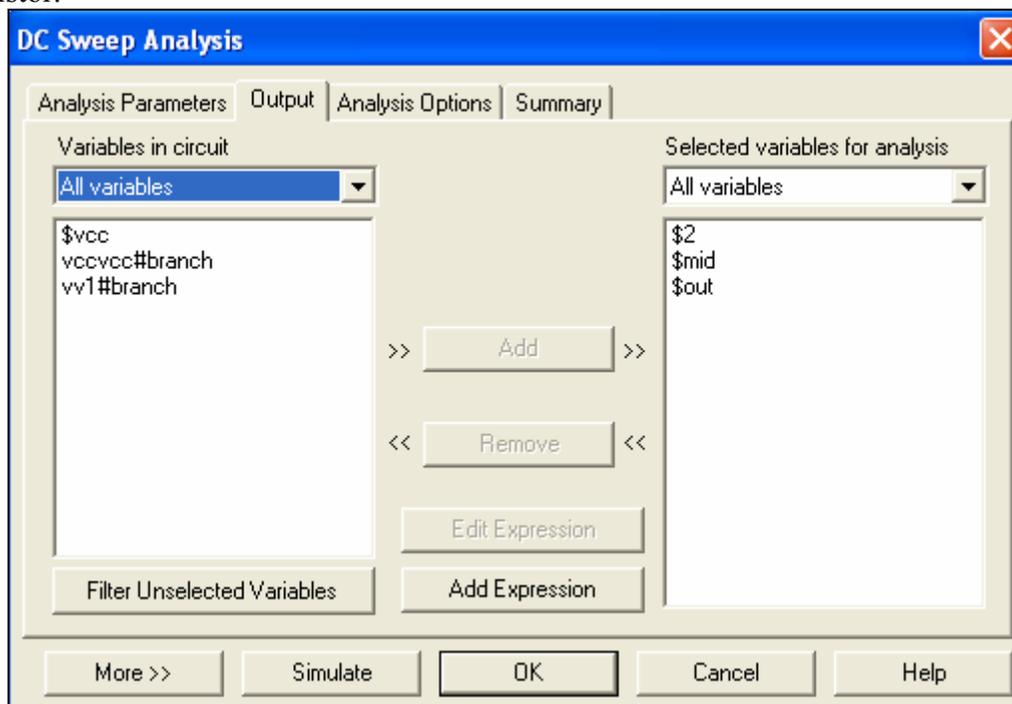
A DC sweep consists of having a DC voltage of current source be “swept” over a range of values to see how the circuit behaves to the various conditions. You will need to specify the source to be swept and the Starting value, End value and increment value of the sweep measured in volts.

To set up a DC sweep analysis select **Simulate/Analyses** and select **DC Sweep** from the toolbar. Next click on the **DC Sweep** button. Enter the values as shown in the figure below. This indicates that the DC sweep will be in reference to V1 and values will range from -10V to 15 V at increments of 1.

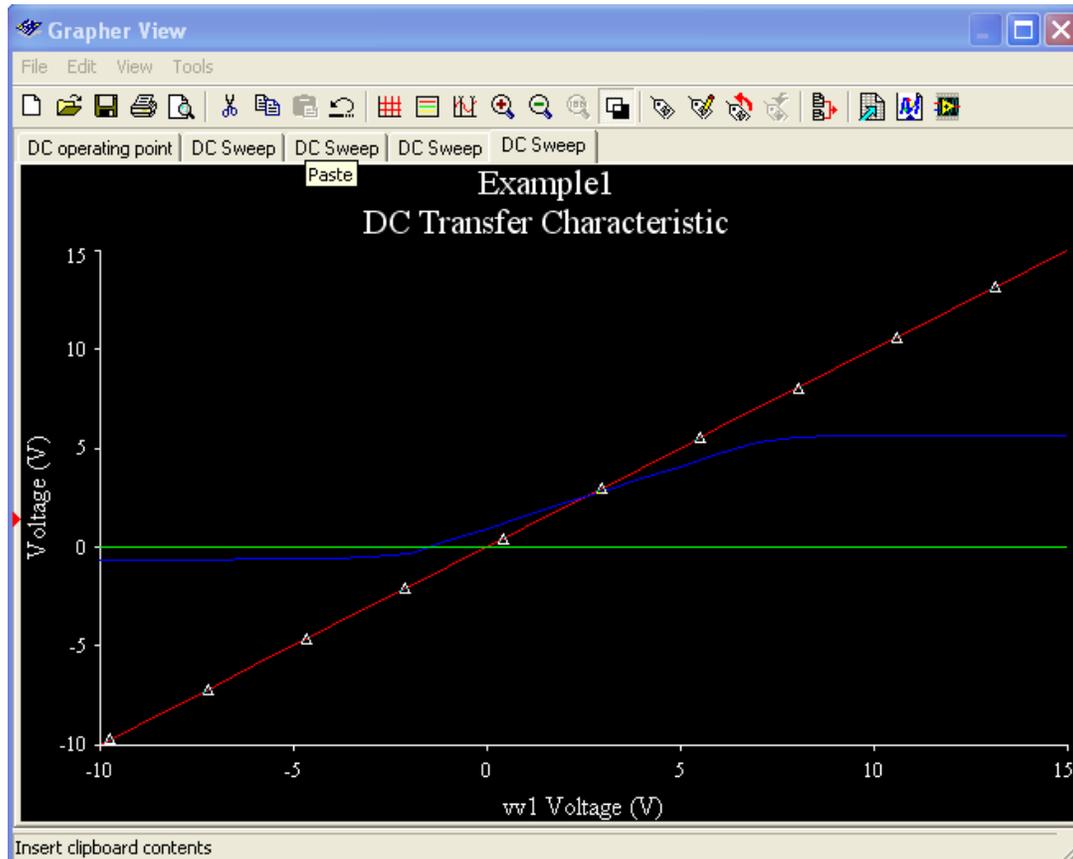


You will also need to click on the **Output** tab to select the variables of your circuit which will be shown in the output window of the analysis.

In this case the outputs displayed will be the wires labeled 2, Mid and in between the .47uF and 5.6k resistor.



Below is the resulting DC sweep analysis of the circuit.



Transient Analysis

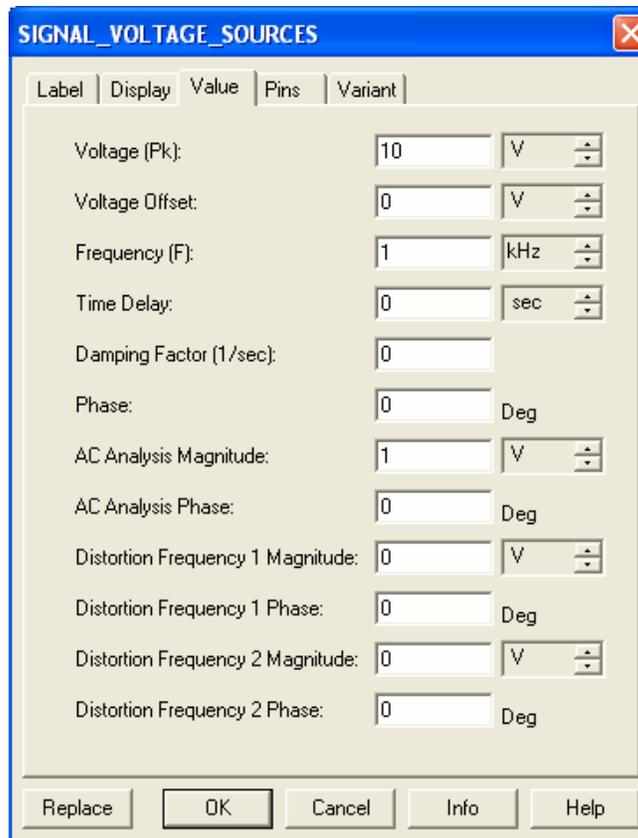
The transient analysis computes various values of a circuit in the time domain. For instance placing a probe in a sine wave of a particular circuit then running a transient analysis, the sine wave would be displayed and you would see it on an oscilloscope. Common sources that are used for transient analysis are PULSE_VOLTAGE, AC_POWER. Finally, transient analysis requires the user to enter two parameters: Print Step, Final Time. Print Step determines how many calculations Multisim must make to plot a wave form. Final time is just the time the simulation will terminate.

For this example place a stimulus (AC_Voltage) source with the following parameters.

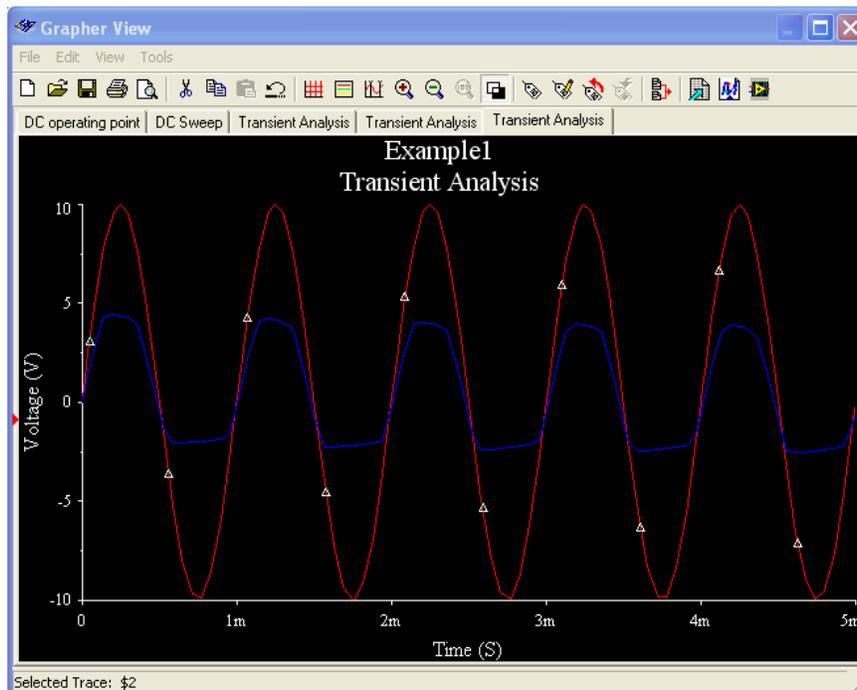
Voltage (Pk): 10V

Voltage Offset : 0V

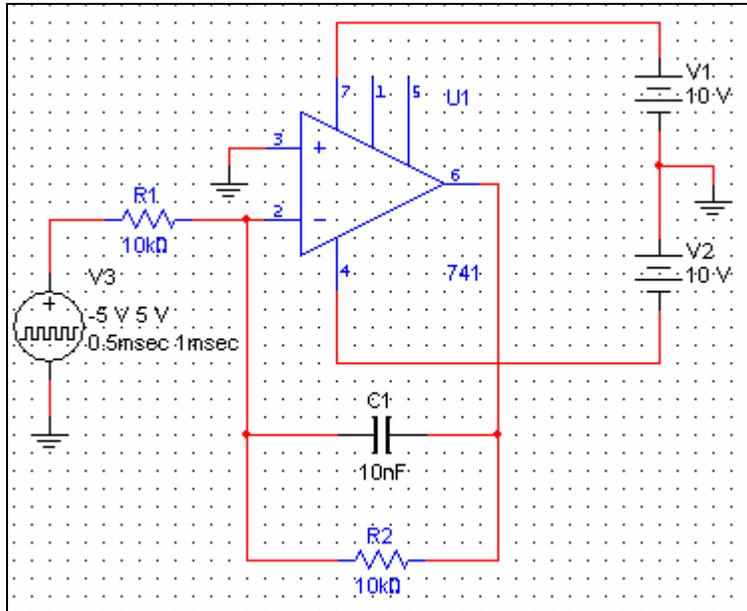
Frequency(F): 1kHz



Analysis display as follows.



Example 2: The following is an operational amplifier based integrator. Output V_o is the time integral of input V_1 . A function of the integrator is that it changes a square wave into a triangular wave, and a triangular wave into a sine type of wave. As shown the input square wave is between -5V to 5V.



AC Analysis

The AC sweep analysis is essentially an analysis of frequency. It allows you to plot magnitude vs. frequency for inputs in your circuit. This would be a common simulation to test frequency response of an amplifier for instance

To setup an AC sweep analysis, select **Simulate/Analyses/** and select **AC Analysis**.

In the AC Analysis box enter the following:

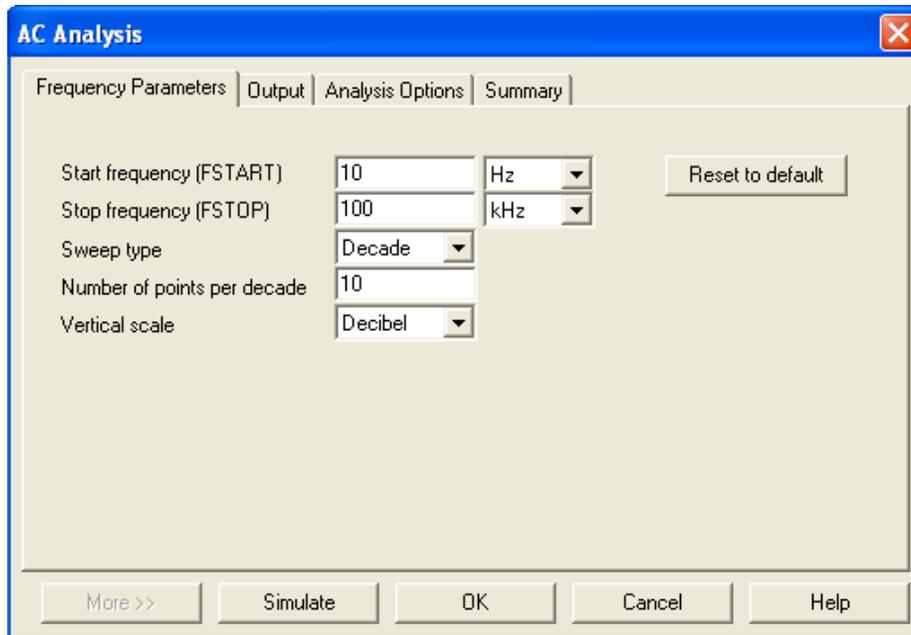
In **Sweep Type** box select **Decade**

Number of points per decade : 10

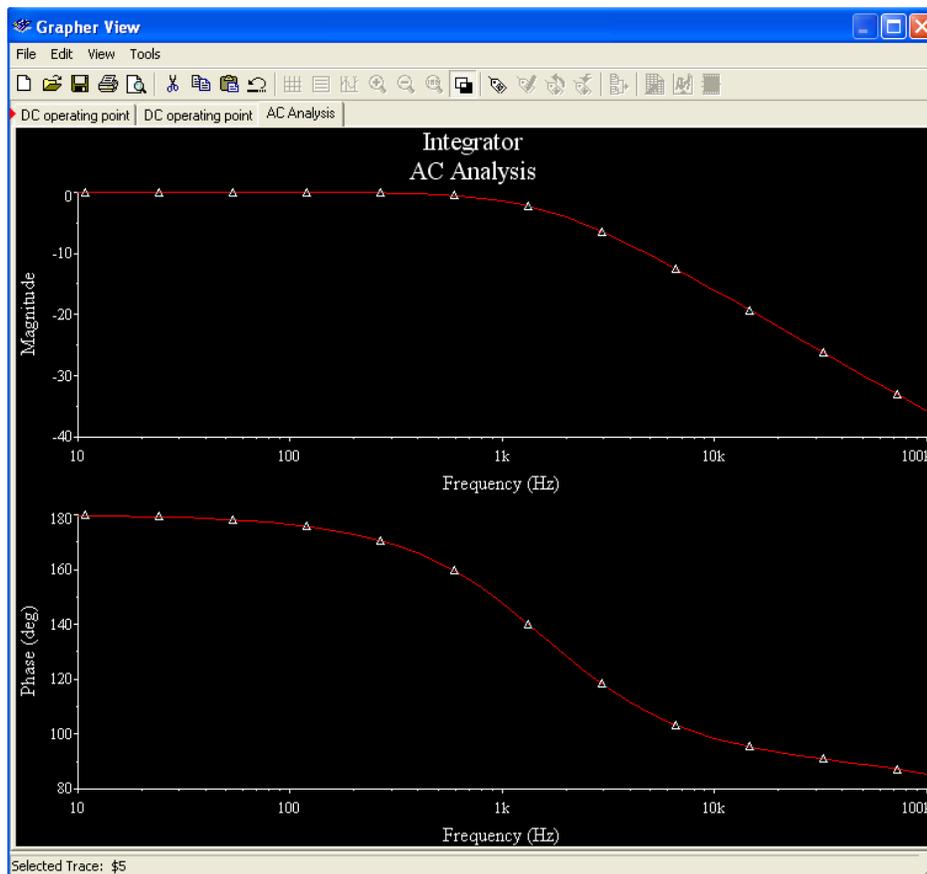
Start Frequency(FSTART) : 10

End Frequency(FSTOP) : 100K

Vertical Scale : Decibel (This will result in a decibel-frequency graph)

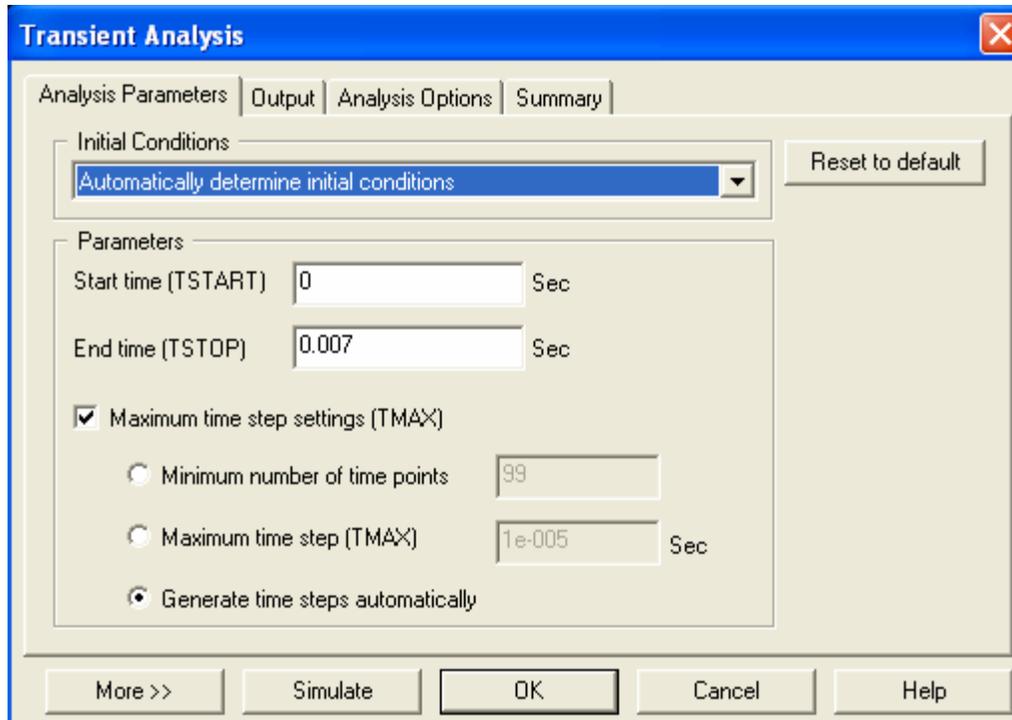


Next we need to specify the output, by clicking on the output tab. The simulation will be performed with 10 points per decade with the frequency of the AC source swept from 10Hz to 100kHz.

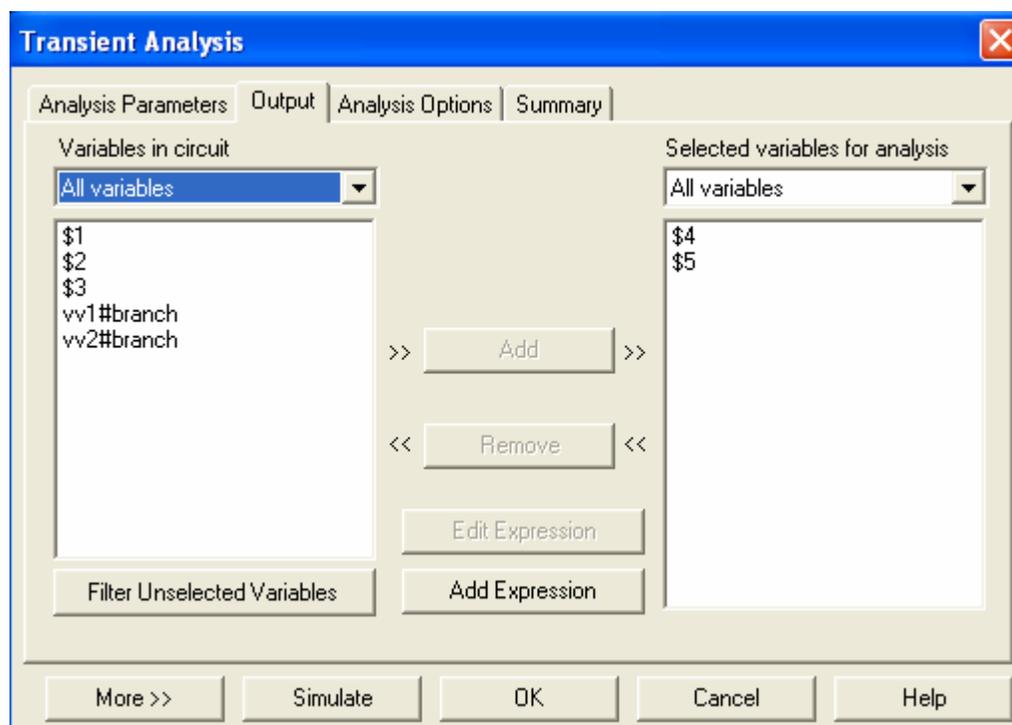


Transient Analysis

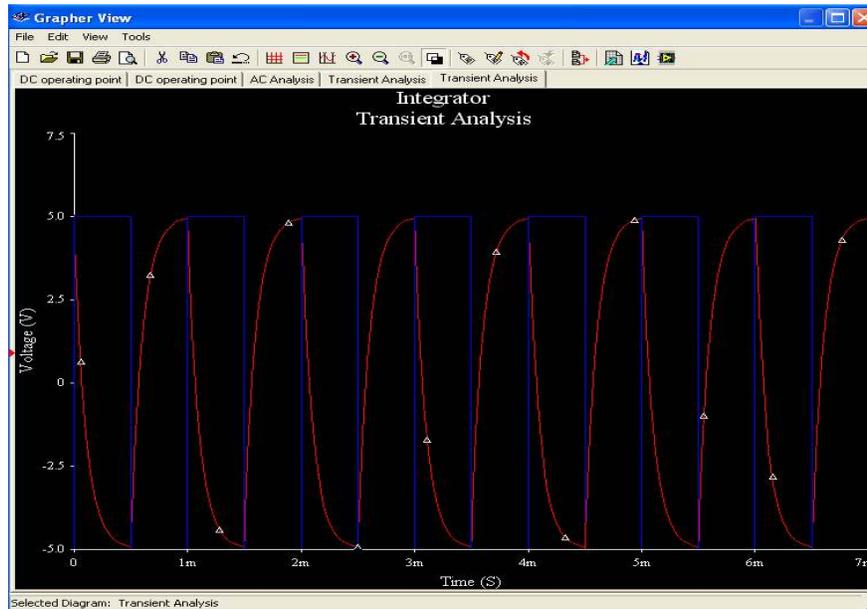
To setup an Transient analysis, select **Simulate/Analyses** and select **Transient Analysis**. V1 is the sole AC source thus, the sweep analysis will be from V1.



Click on the Output tag and select a transient of the voltage source vs output.



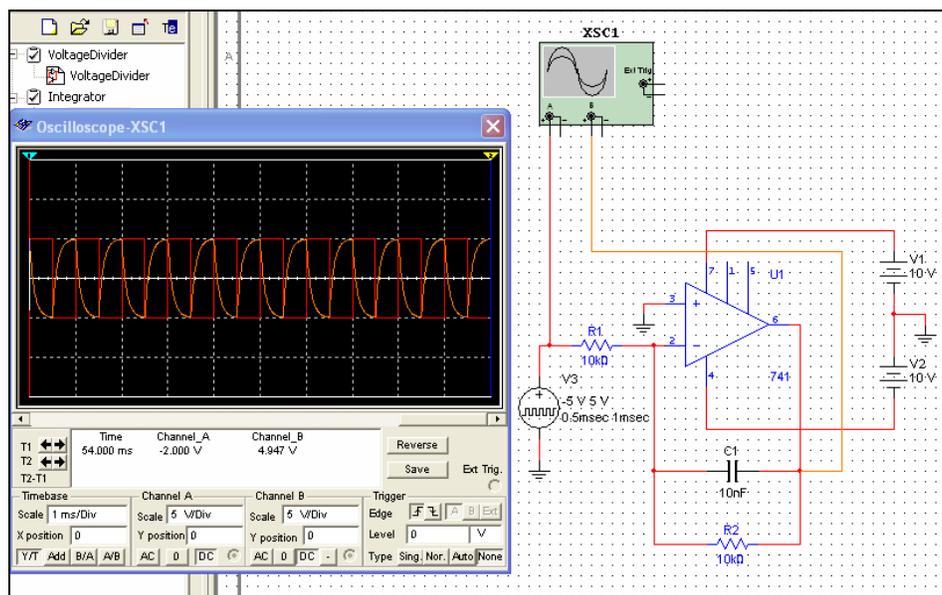
The following is the resulting transient graph.



Alternatively, Multisim has an oscilloscope components which when you run simulation lets you compare signal waveforms at particularly selected nodes.

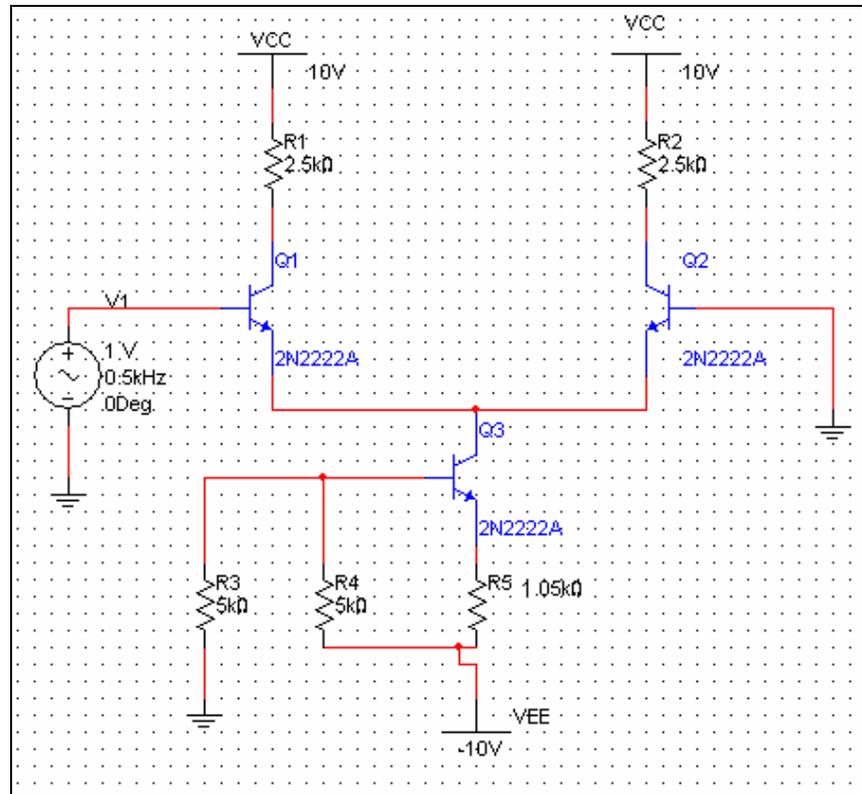
To do this select **Simulate/Instruments** and select **Oscilloscope**. Connect the inputs of the oscilloscope to the appropriate places. Next select **Simulate/Run [F5]**, and scale to get visible results.

Here is the resulting schematic of placed oscilloscope and ran analysis.



Example 3:

This example circuit consists of a common BJT differential amplifier. Here we just show the circuit's amplification of a sine wave using the transient analysis.



Transient Analysis

Select the AC_VOLTAGE component **Place/Component**. Select Sources from the Group Window. Here you will find the AC_VOLTAGE under the SIGNAL_VOLTAGE family.

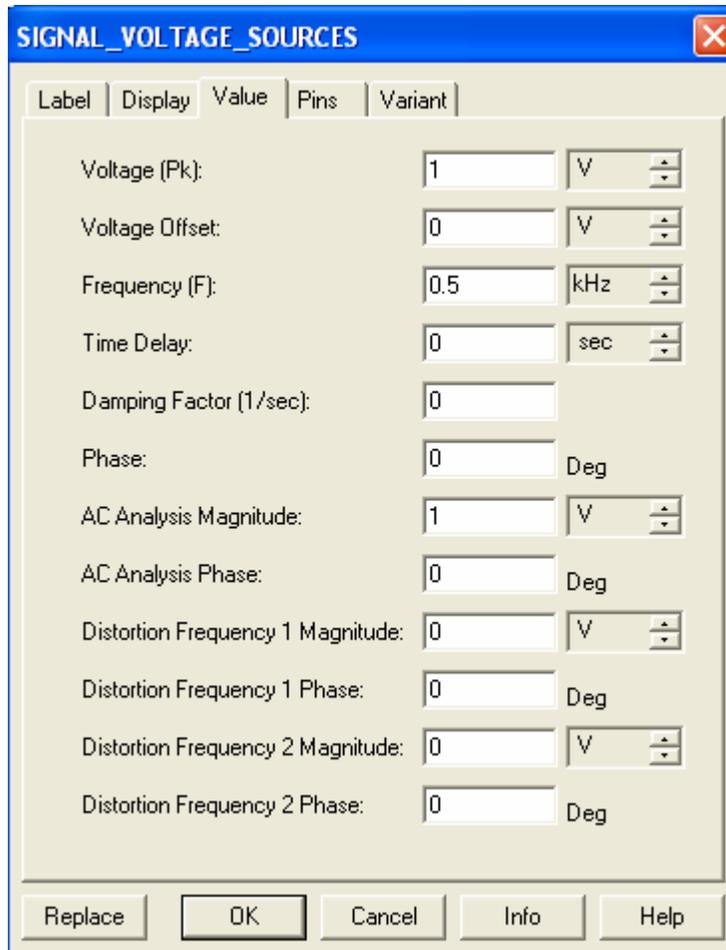
Double click on the AC_VOLTAGE component and enter in the following values:

Values used for the V1 sine source:

Voltage offset :0 V

Voltage(Pk): 1V

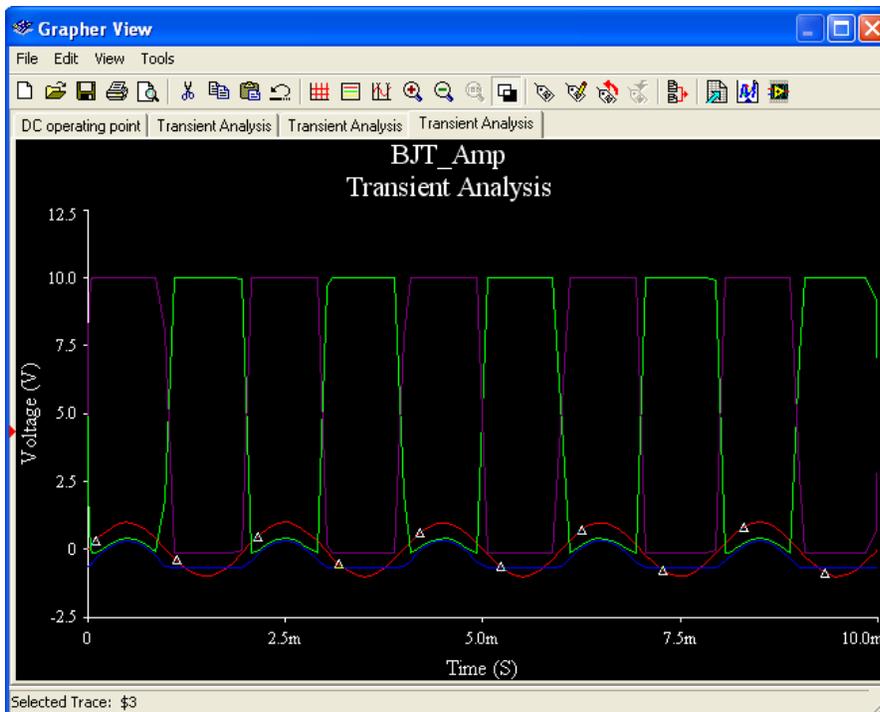
Frequency(F): 0.5kHz



To use the differential pair as a linear amplifier we apply a very small differential signal(a few millivolts).

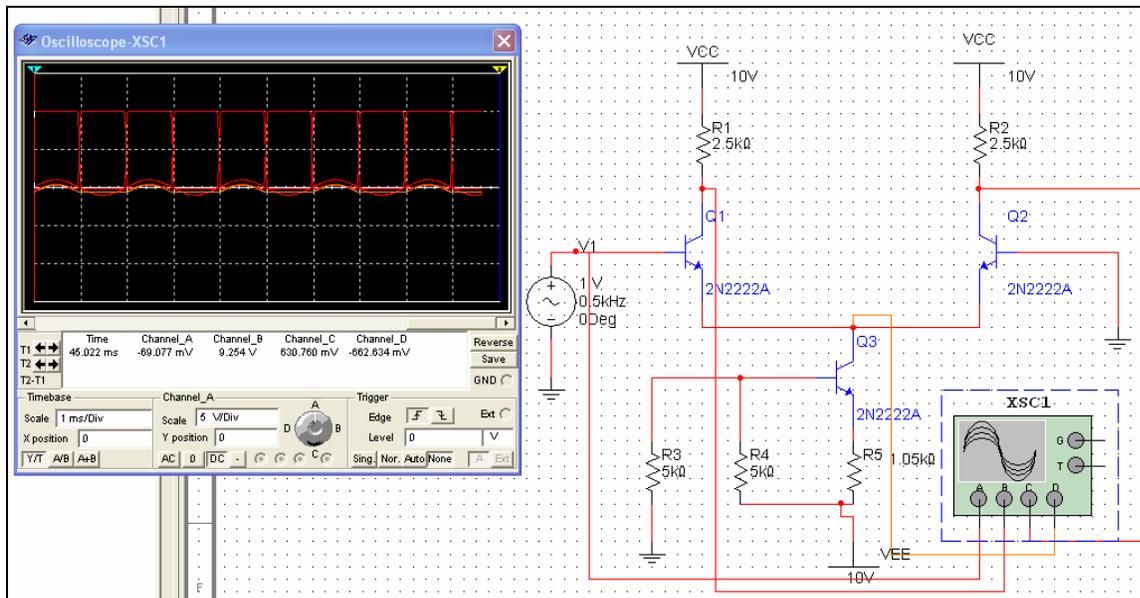
Again, to setup a Transient analysis, select **Simulate/Analyses** and select **Transient Analysis**. We chose the Start Time(TSTART) at 0 s and Stop Time(TSTOP) at 10 ms. Next, in the output tab, select the appropriate nodes from which the outputs will be seen.

The resulting analysis shows the waveforms at the voltage input, and at each of the transistors(above Q1, above Q2) and at the upper node of Q3.



Again you could also have used the built in oscilloscope in Multisim to view the results at certain points in your circuit.

Here we show the placed oscilloscope and the resulting graph when simulation is ran. Select **Simulate/Instruments** and select **Four Channel Oscilloscope**. Connect the inputs of the oscilloscope to the appropriate places. Next select **Simulate/Run [F5]**, and scale to get visible results.

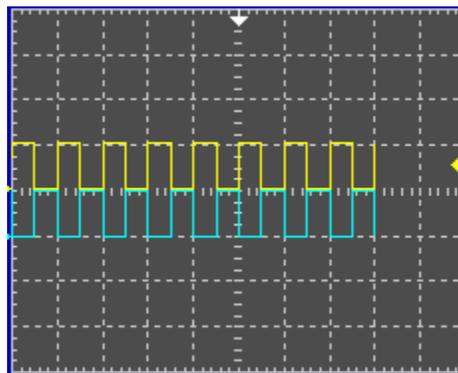
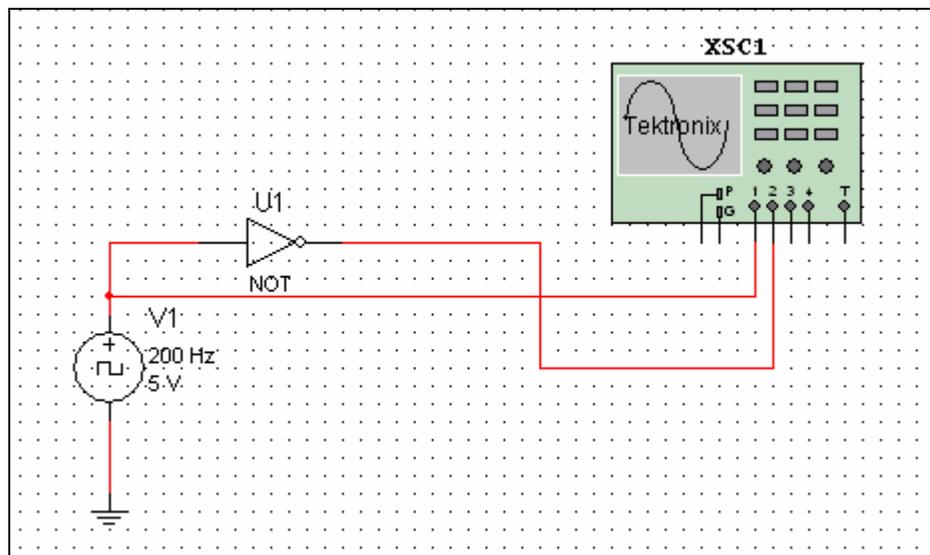


APPENDIX A: Waveforms

Here we provide some examples as to generating commonly used waveforms in electronics using MultiSim 9.

Generating a Pulse:

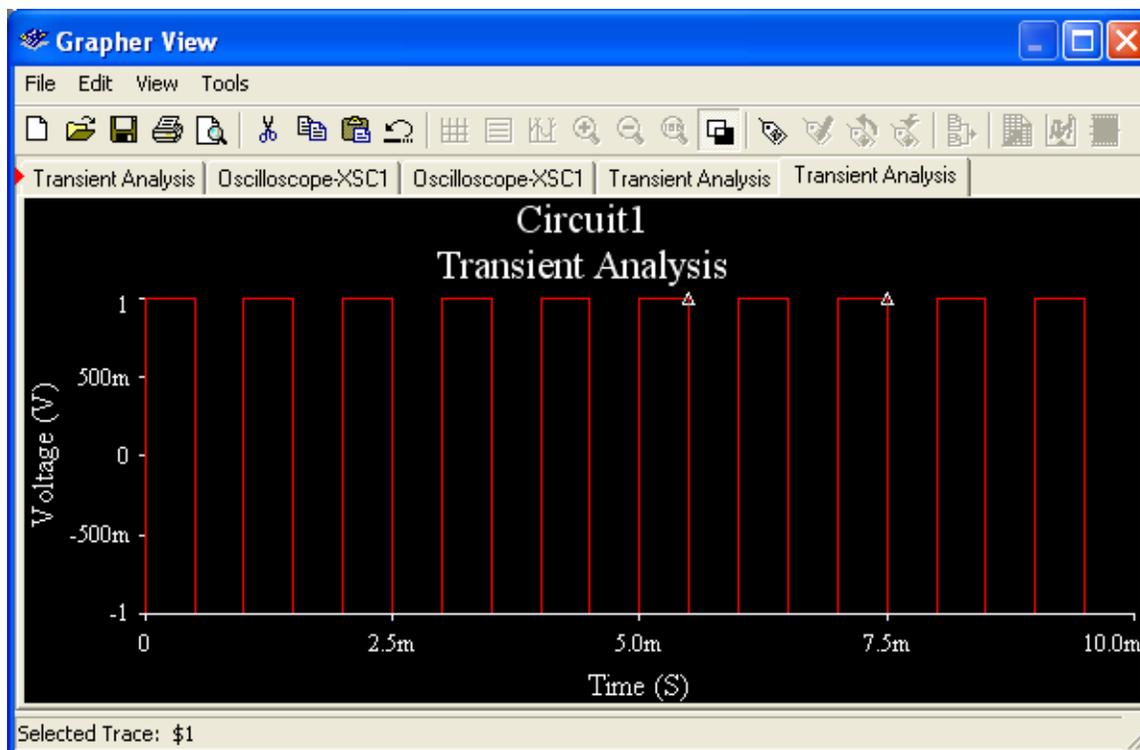
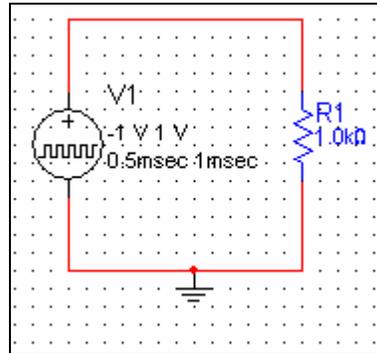
Clock Pulse forms are generated by using the Pulse_Voltage component, which can be obtained by selecting **Place/Component** and selecting **ClockVoltage** from the **Signal Voltage** family in the toolbar. A circuit is constructed to illustrate the clock Pulse. Note an inverter part, NOT, was added to the circuit so that the results will show the output of the clock pulse and its inverting form. The NOT component can be found by selecting the **Place/Component**. In the Group window select **Misc. Digital** select **TIL** and in the component window select the **NOT** component.



Alternatively you could use one of the function generators provided by Multisim.

Generating a Square Wave:

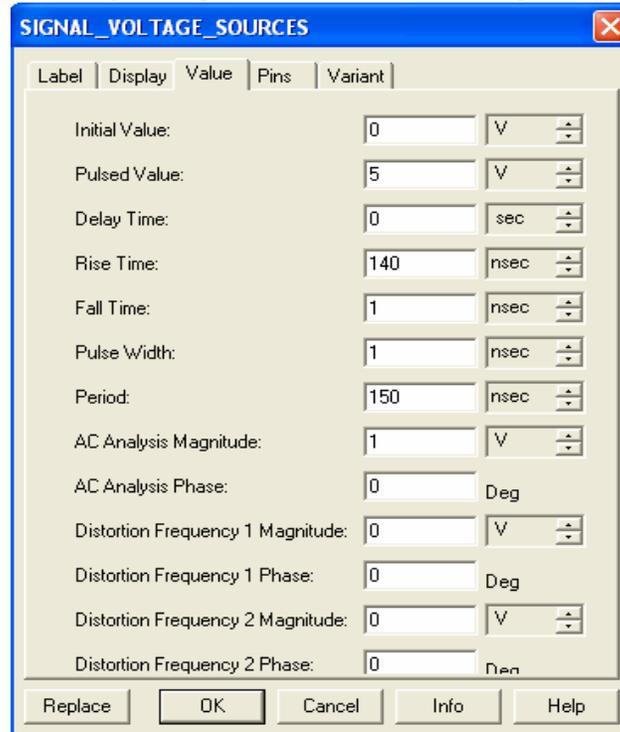
Generating this type of wave is fairly simple. Either create one with the Pulse Voltage component or by again using the function generators.



Output Display

Generating a Saw Tooth Wave

Generating a sawtooth wave is done in the same manner as generating a square wave. For this example we use the Pulse Voltage component with the following values.

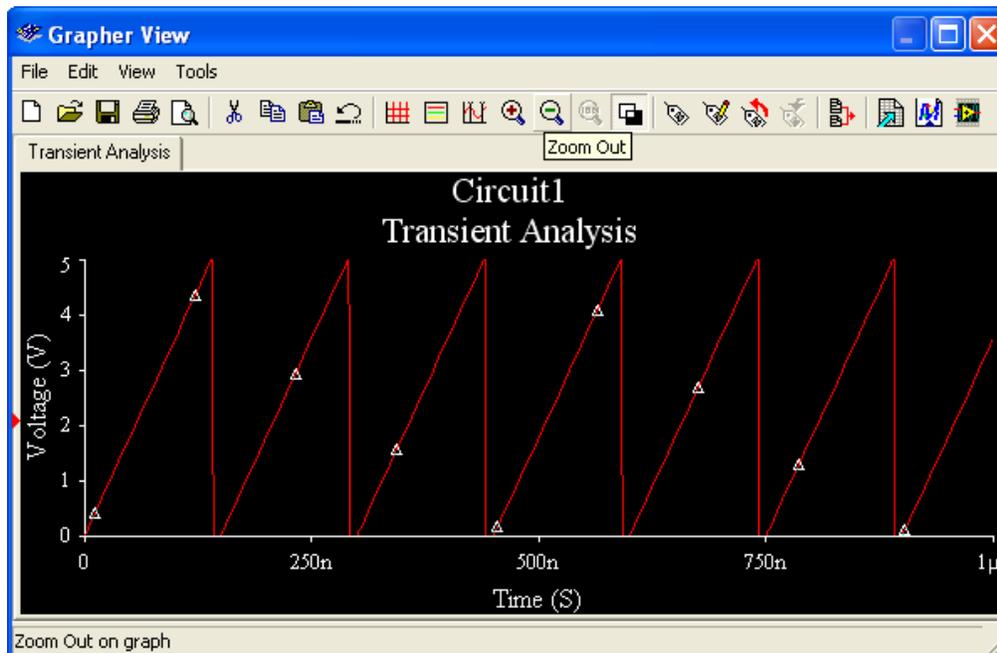


The dialog box titled "SIGNAL_VOLTAGE_SOURCES" contains the following configuration parameters:

Parameter	Value	Unit
Initial Value:	0	V
Pulsed Value:	5	V
Delay Time:	0	sec
Rise Time:	140	nsec
Fall Time:	1	nsec
Pulse Width:	1	nsec
Period:	150	nsec
AC Analysis Magnitude:	1	V
AC Analysis Phase:	0	Deg
Distortion Frequency 1 Magnitude:	0	V
Distortion Frequency 1 Phase:	0	Deg
Distortion Frequency 2 Magnitude:	0	V
Distortion Frequency 2 Phase:	0	Deg

Buttons at the bottom: Replace, OK, Cancel, Info, Help.

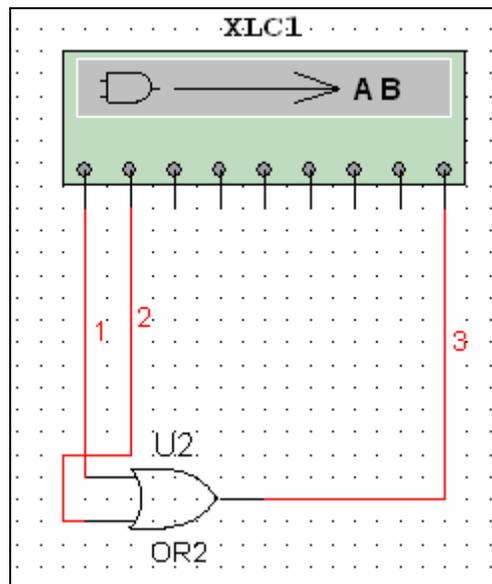
Here we see the resulting SawTooth wave.



APPENDIX B: Digital Components

Here we show a couple of simple examples using digital components using MultiSim.

Example 1: The circuit below consists of a 2 input NAND gate. Select **Simulate/Instruments** and select **Logic Converter**. This logic converter can either display the states of your circuit and it can also build a circuit according to a truth table provided by yourself.



Wire 1 and 2 are the inputs of the AND gate, while wire 3 is always going to be used for the output of a logic circuit.

Clicking on the Logic inverter and selecting  we can see the results of the two input AND gate.

The screenshot shows the 'Logic Converter-XLC1' window. It has a blue title bar and a close button. Below the title bar are eight indicator lights labeled A through H. A table displays the truth table for the circuit:

	A	B	C	D	E	F	G	H
000	0	0						0
001	0	1						1
002	1	0						1
003	1	1						1

On the right side, there is a 'Conversions' panel with several buttons:

-  \rightarrow $\overline{A|B}$
- $\overline{A|B}$ \rightarrow A|B
- $\overline{A|B}$ $\xrightarrow{\text{SIMP}}$ A|B
- A|B \rightarrow $\overline{A|B}$
- A|B \rightarrow 
- A|B \rightarrow NAND

Example 2: In this example we'll simulate a 4-bit counter using Multisim.

