FLORIDA INTERNATIONAL UNIVERSITY

Department of Electrical & Computer Engineering

PSPICE TUTORIAL (BASIC)

Professor: Dr. Subbarao V. Wunnava
Teaching Assistant: Rafael Romero
COURTESY: ED LULE/ BORIS LINO/ORCAD

Updated: Spring.2006, 07 Feb, 2006

INTRODUCTION

Orcad PSpice is a simulation program that models the behavior of a particular analog or digital circuit. Used with PSpice Schematics for design entry, you can think of PSpice as a software based breadboard of your circuit in which you can test and refine your design before ever touching a piece of hardware. Essentially you have a 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.
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.

There are three versions of PSpice at the presently available at the time of this writing:

- PSpice A/D
- PSpice A/D Basics
- Pspice

PSpice A/D is the full version with no restrictions, the remaining two are fully functional but have some limitations.
Demo versions of Orcad package will let you experience all the features and functionality of the actual PSpice software. This can be obtained by going to the following website

http://www.orcad.com/download.orcaddemo.aspx

PROCEDURE FOR SIMULATION IN SCHEMATICS

Several steps are performed in the analysis of a circuit using PSpice.

1. Begin by drawing the schematic of your circuit.
2. Change any of the values of the components and sources to suit your needs.
3. Add any probe or marker.
4. Setup analysis.
5. Run simulation.

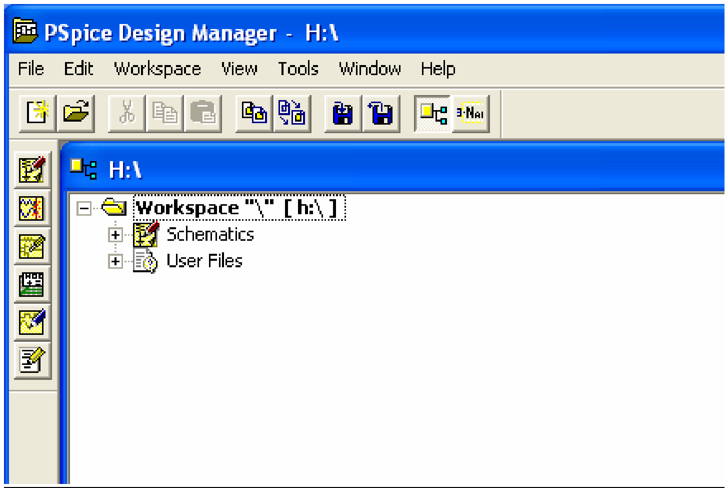
DESIGN ENTRY

1. Open Design Manager:

Begin by drawing your intended circuit into PSpice. This is done by opening up **PSpice Design Manager.** This can be found under the **Orcad** folder under Program Files or whichever path you chose to save the program under.

For Windows users the default location can be found by clicking Start->All Programs->Orcad Family Release->PSpice Design Manager.

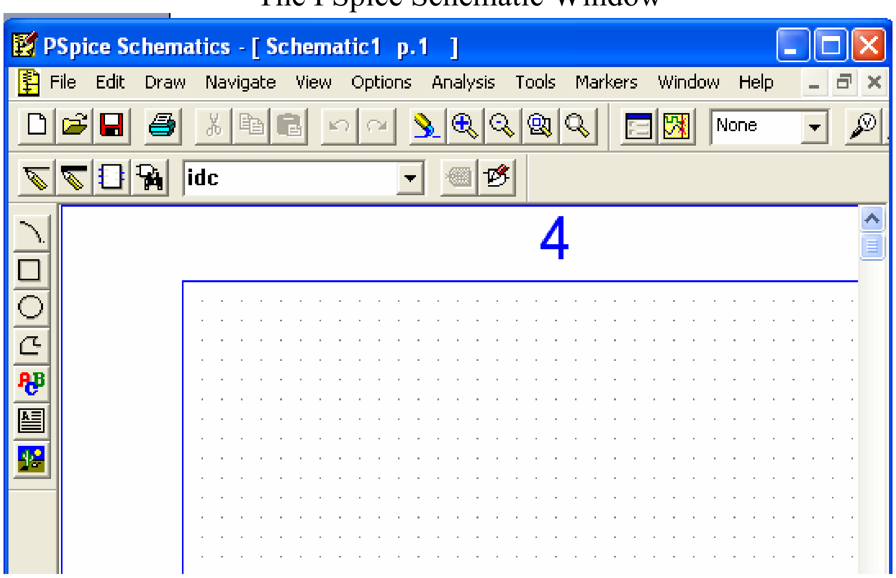
The PSpice Design Manager Window is shown below.



2. To create/open a schematic file:

Under the **Tools** toolbar, choose **Schematics.** This will open up PSpice Schematics.

The PSpice Schematic Window

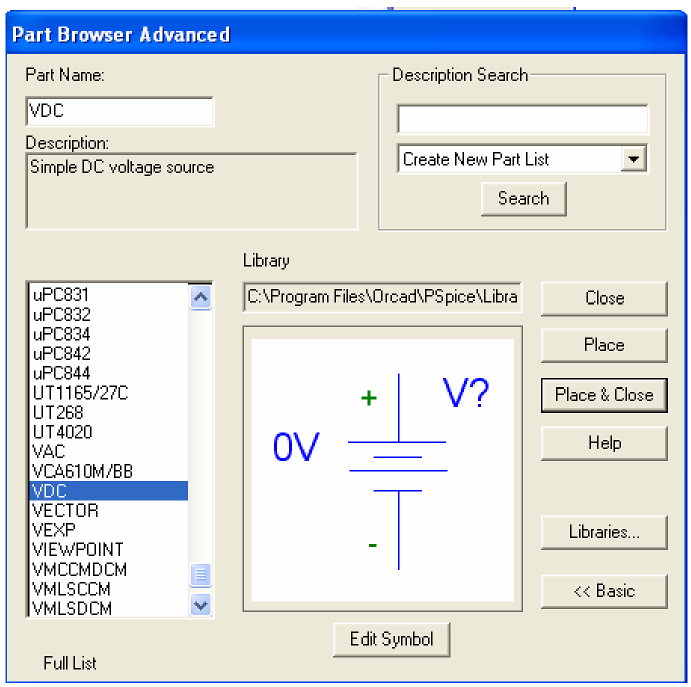


PSpice Schematics is a design entry program you need to prepare your circuit for simulation. In effect, this is the software equivalent of a breadboard. Here is where you will place all the components needed to suit your circuit, wire them together, connect voltage source(s) to power your circuit, etc. In this environment, by the click of a mouse, you can define component values, change their attributes, define waveforms and so on.

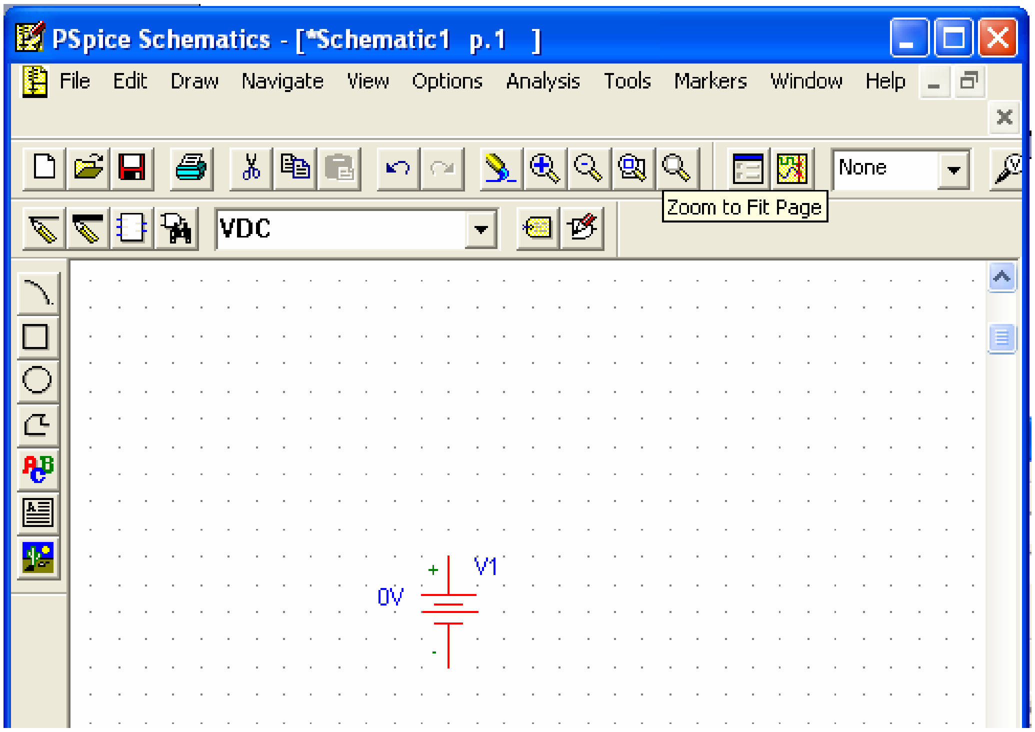
Select **File/New** or click on the **new file** icon to create a new file.
Select **File/Open** or click on the **open file** icon to open existing file.

We now continue by finding and entering all the parts you will need for your particular circuit. This is done by either selecting **Draw/Get New Part** or by clicking on the **Get New Part** icon. Shortcut keys **[Ctrl + G]** can also be used.

The Part Browser Advanced window appears.



In the example above, in the **Part Name** entry box we entered a VDC component. This is just a simple DC voltage source. Next we click on **Place & Close** icon.

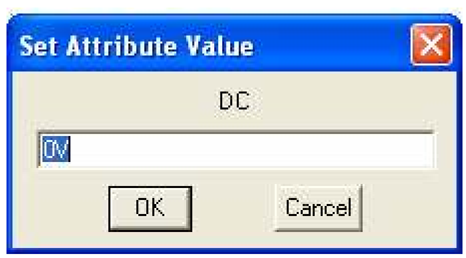


Move the component around to the correct position of your choosing. Left Click to place the component on schematic, right click to cancel placement.

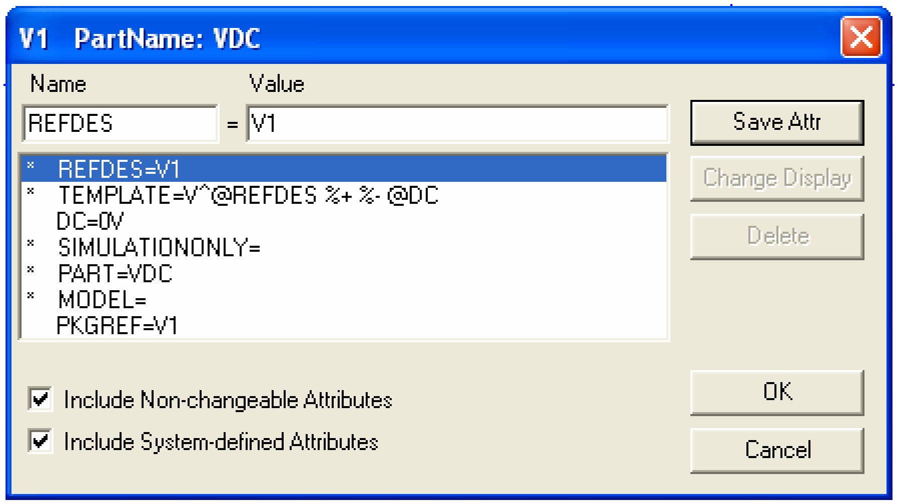
3. Changing the value of a component:

Components are created with default values. In our example our voltage source has 0 V. By double clicking on the attribute value (0 V) a **Set Attribute Value** window appears, here you can change the to the value of your choosing. Additionally you can double click on the component itself or select **Edit/Attributes** from the toolbar to change additional values corresponding to the component selected.

Set Attribute Value window



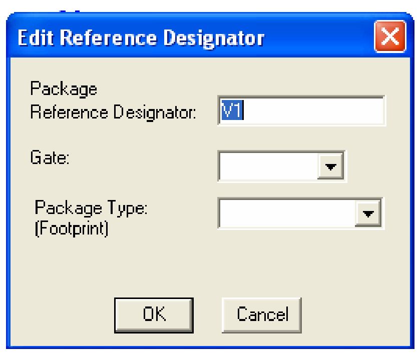
Edit Attributes window



4. Changing the reference number of a component:

Double click on the reference number of the component you want changing, in this case V1. A window labeled **Edit Reference Designator** appears. Here just enter the change you want made in terms of your reference.

Edit Reference Designator window



5. Editing the component:

Moving a component:
Click on the component once to select it. Drag the component to a new location.

Rotating a component:
Click on the component once to select it. Select **Edit/Rotate[Ctrl+R]** to rotate the component 45 degrees counter clock wise.

Flipping a component:
Click on the component once to select it. Select **Edit/Flip[Ctrl+F]** to flip the component.

Deleting a component:

Click on the component once to select it. Select **Edit/Delete** to delete the component.

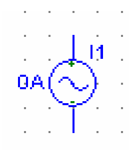
6. Wiring:

To wire a component simply select **Draw/Wire[Ctrl+W]** or click on the **draw wire** icon  to wire the components of a circuit together. Right Click on the mouse or press **[Esc]** to finish wiring. If you like to label the wire, just double click on the wire to select it. In the **Set Attribute Value** window type whatever name you want to label the wire as.

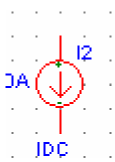
7. Adding a Current Source:

Several current sources are available, each with their own characteristics:

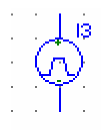
IAC
Simple current source



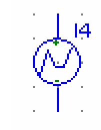
IDC
Simple DC current source



IPULSE
Pulse Current source



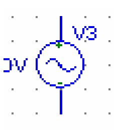
IPWL
Piecewise Linear current source



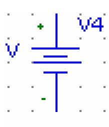
8. Adding a Voltage Source:

Several voltage sources are available, each with their own characteristics:

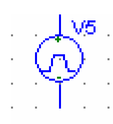
VAC
AC Voltage source



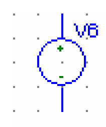
VDC
DC Voltage source



VPULSE
Voltage Pulse source



VSRC
AC or DC Voltage source



9. Adding connection bubbles:

Bubbles are components used to connect one part to another without using wire connection. Select **Draw/Get New Part** or click on the **get new part** icon. In the **Part Name** entry box type in "bubble". Place the connection bubble in the circuit.

10. Adding Probes:

Voltage and current markers are used to probe voltage or current. After simulation, PSpice automatically plots the results.

Voltage marker 
To add a voltage marker, select **Markers/Mark Voltage/Level** or click on the voltage marker icon to place it.

Current marker 
To add a current marker, select **Marker/Mark Current into Pin** or click current marker icon to place it.

Voltage Differential marker
To add a current markers, select **Marker/Mark Voltage Differential.** Two markers(+ , -) will be used, place them between the circuit.

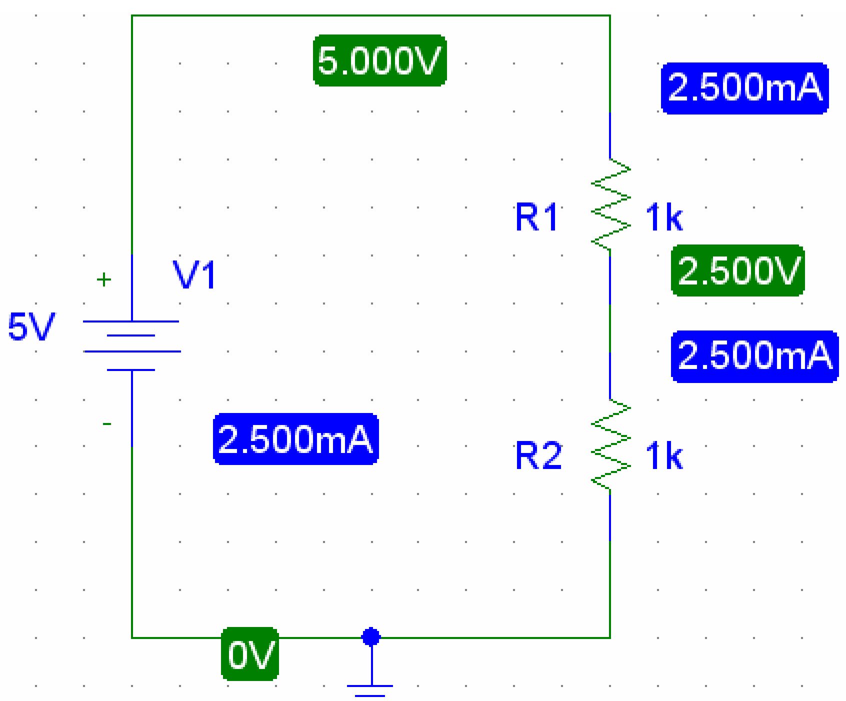
11. Enable Bias voltage display and Bias current display icon.

In the example circuit below we built a simple voltage divider. By either selecting **Analysis/Display Results on Schematic/** and checking either **Enable Voltage Display** or **Enable Current Display** or both, the resulting current and voltage in the given circuit is displayed. This can also simply be done by clicking on their respective icons.

Note: Before Voltage/Current displays can be seen, Simulation must first be run. This is covered in the Running Simulation is will be covered in the following sections.

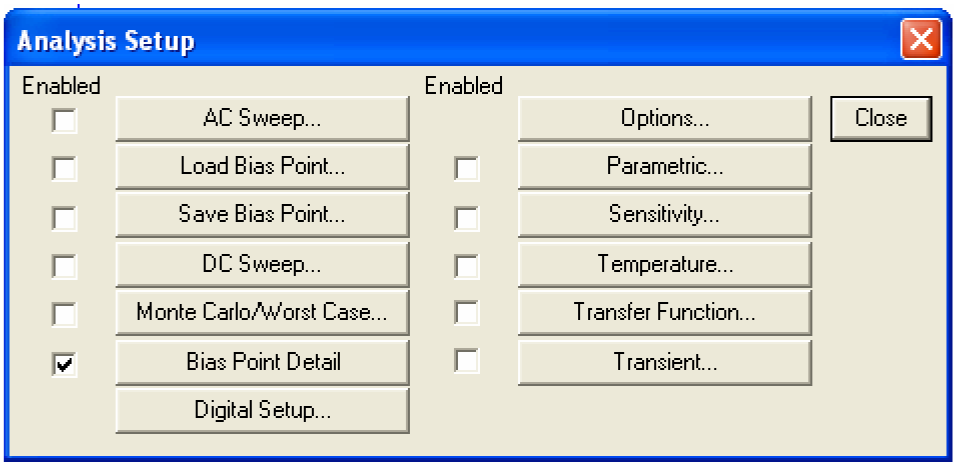
Enables Bias Voltage Display.

Enables Bias Current Display.



ANALYSIS SETUP

PSpice supports analyses that can simulate analog-only mixed-signal, and digital-only circuits. A few of these options can be found by selecting **Analysis/Setup** from the toolbar.



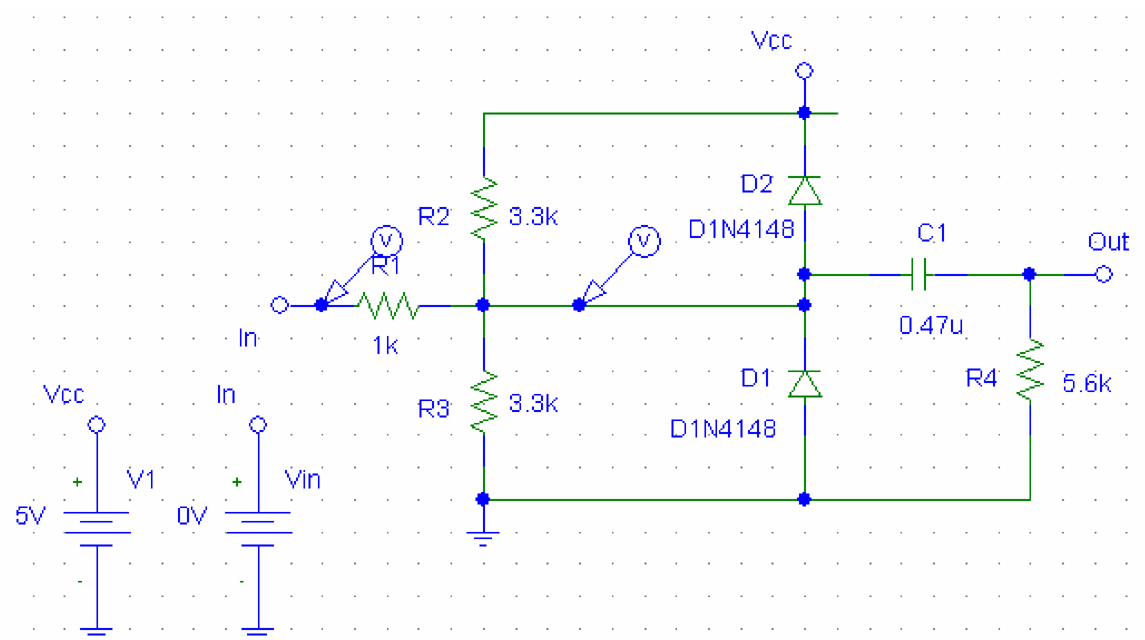
For example the DC Analysis(Bias Point Detail) calculates DC currents and voltages. This is checked as a default in PSpice. The Transient Analysis(Transient) determines the output with respect to time. DC Sweep causes a DC sweep to be performed on the circuit. This is useful in finding the transfer function of an amplifier for example. Its output is being swept with respect to the source. AC Sweep Analysis determines the output with respect to the frequency.
Lastly as a remainder before moving on to simulation, don't forget to always ground your circuit. This is key if you want your circuit to be simulated. Grounding consists of adding parts GND_EARTH or GND_ANALOG from the **Draw/Get New Part** selection toolbar.

RUNNING SIMULATION

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 PSpice simulation. Here we will perform various analyses on some sample circuits.

Open the PSpice Design Manager.
From there open PSpice Schematics.

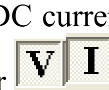
Example 1: Construct the following circuit .



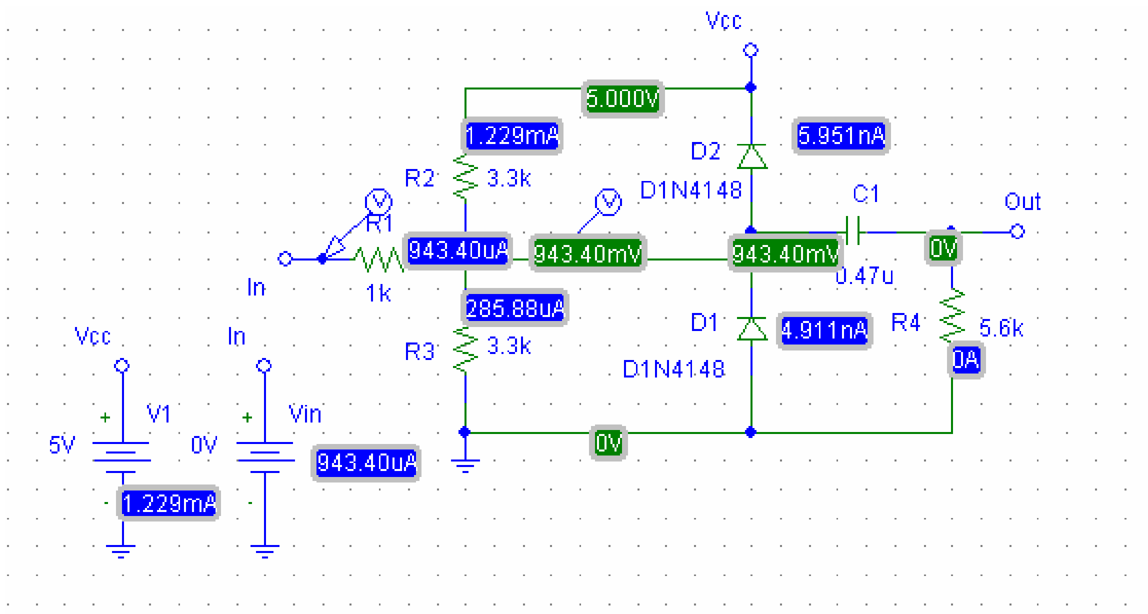
Parts lists consist of VDC, C, R, D1N4148,GND_EARTH, BUBBLE. Notice the use of the BUBBLE part, instead of connecting the corresponding DC voltage to the circuit a label of the same name is used while the VDC is placed to the side. This is intended to save space and makes the circuit look a bit cleaner.

Once the circuit schematic is completed save it by selecting **File/Save As** from the toolbar. Now we are ready to run the simulation. To do this select **Analysis /Simulate** from the toolbar.

Bias Point Analysis

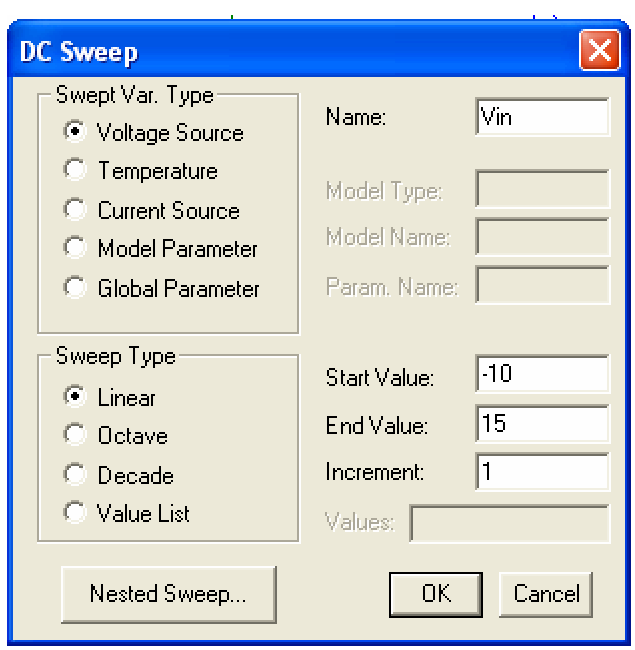
As previously stated, the DC Analysis(Bias Point Detail) of a circuit is calculated because it is set as default analysis by PSpice. The circuit now should have the DC currents and voltages displayed. If not try clicking the following icons from the tool bar 

.

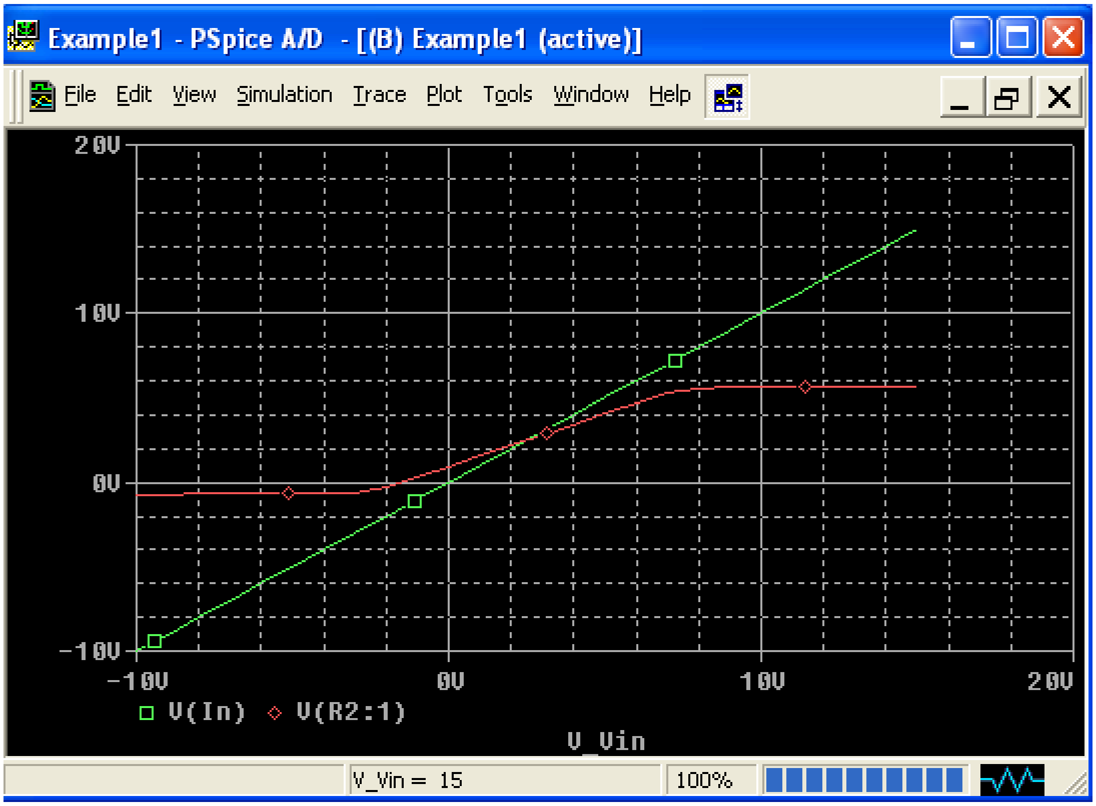


DC Sweep Analysis
A DC sweep consists of having a DC voltage of current source be “swept” over a range of values to see how the circuit behaves to the various conditions. You will need to specify the source to be swept and the Starting value, End value and increment value of the sweep measured in volts.

To set up a DC sweep analysis select **Analysis/Setup** 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 Vin and values will range from -10V to 15 V at increments of 1.



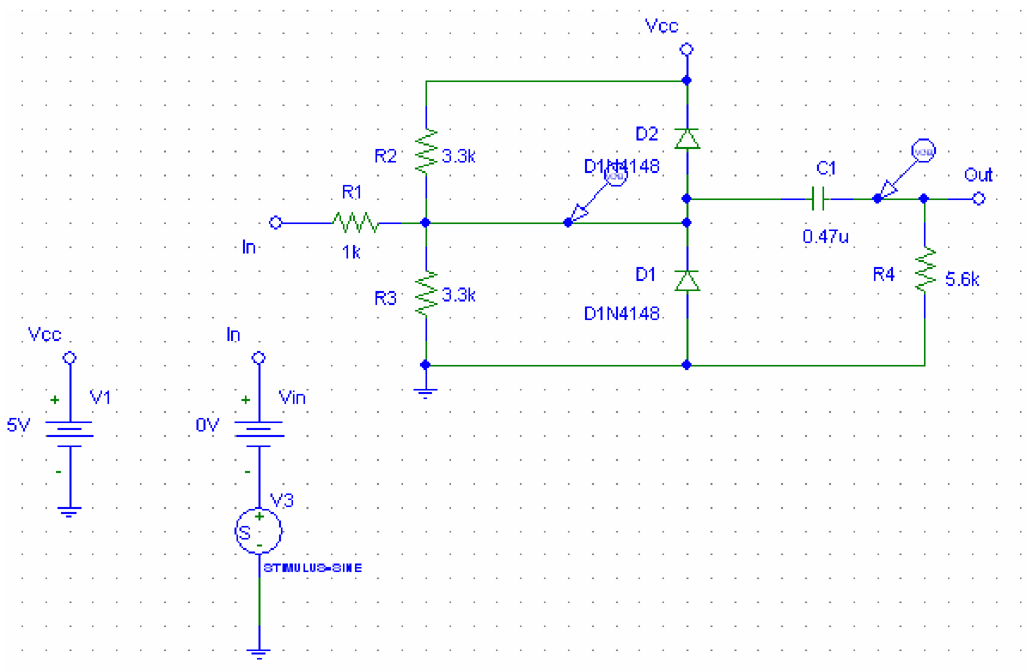
Below is the resulting DC sweep analysis of the circuit.



Transient Analysis

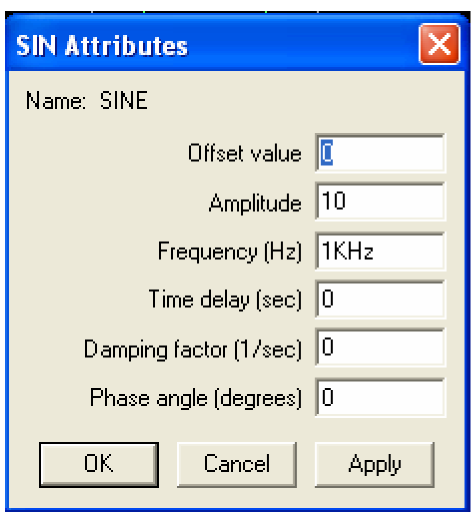
The transient analysis computes various values of a circuit in the time domain. For instance placing a probe in a sine wave of a particular circuit then running a transient analysis, the sine wave would be displayed and you would see it on an oscilloscope. Common sources that are used for transient analysis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

To perform the transient analysis on this circuit, we will need to place another component of the circuit. This component is VSTIM, and needs to be placed under the Vin DC voltage source. The circuit now appears as such.



Next Double click on the VSTIM component and when the **Atribute Value** dialog box appears, type “SINE” in it and click ok. The **New Stimulus Editor** window appears, click SIN then click OK. In the SIN Attributes dialog box, set the first 3 parameters as follows:

Offset Voltage = 0
Amplitude = 10
Frequency = 1kHz



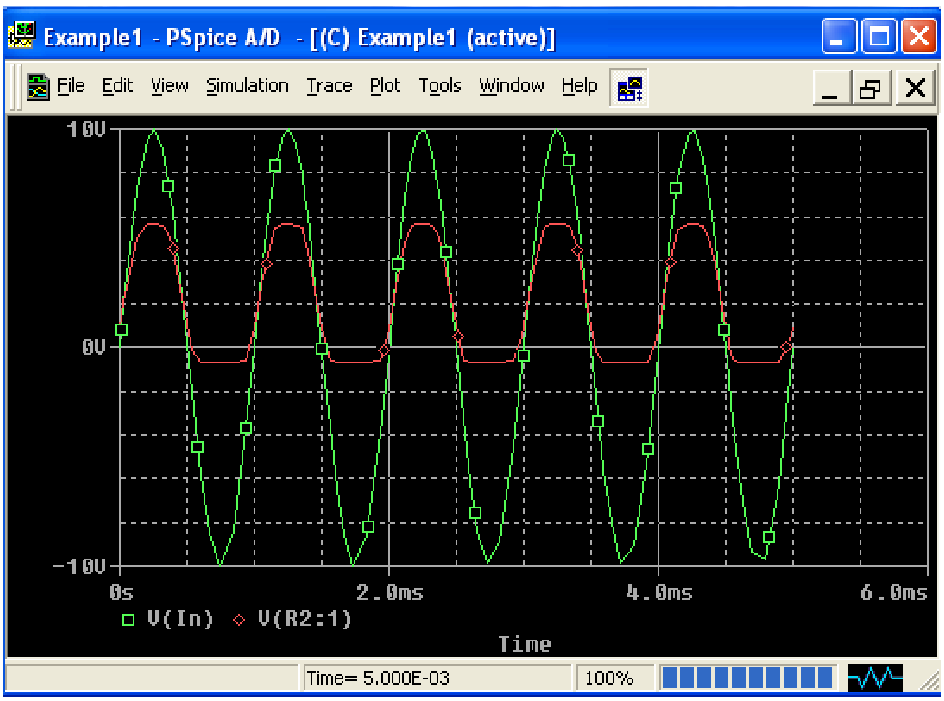
Click OK.

From the **File** menu, select **Save** to save the stimulus information.
From the **File** menu, select **Exit.**

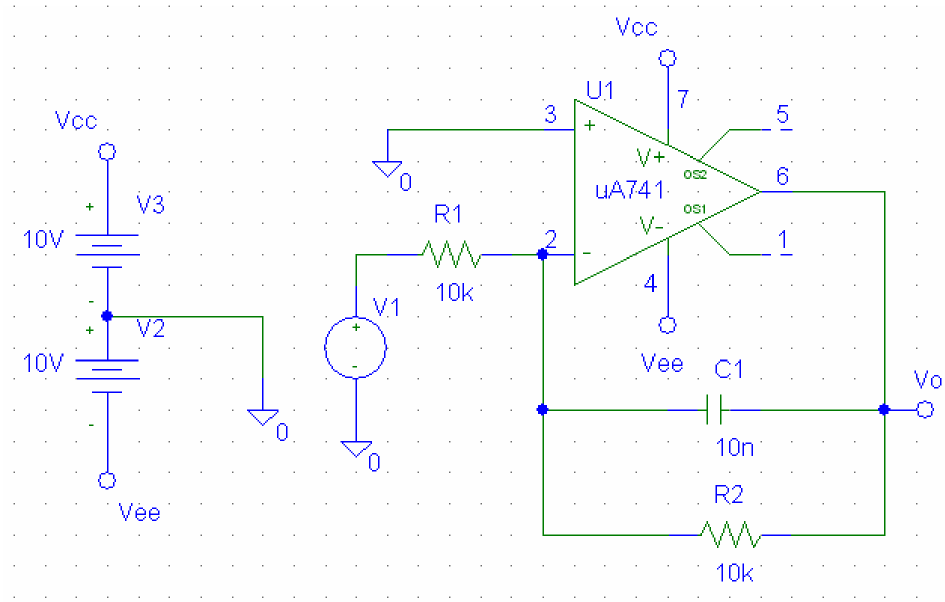
In the Schematics, from the **Analysis** menu select **Setup**.
Click Transient to display the Transient Analysis dialog box.
Set up the Transient dialog box.
Click OK
Clear the DC Sweep check box to disable the DC sweep
From the **Analysis** menu, select **Simulate**.

In Probe from the **Trace** menu select **Add.**
Select V(In) and V(Out) by clicking them in the trace list.
Click OK to display the traces.

Analysis displays as follows.

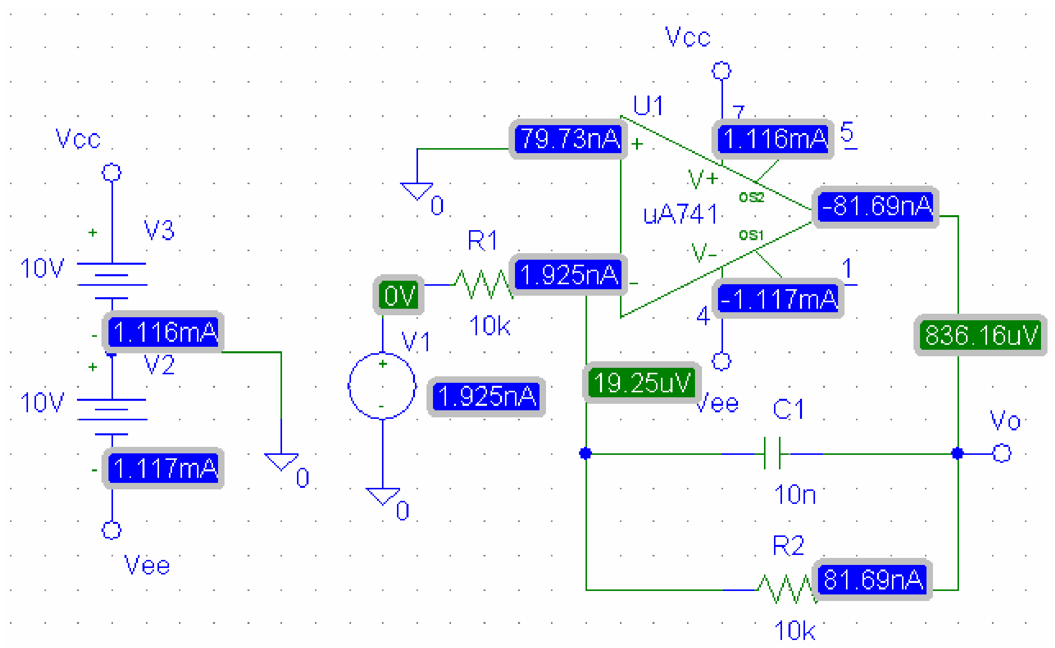


Example 2: The following is an operational amplifier based integrator. Output Vo is the time integral of input V1. A function of the integrator is that it changes a square wave into a triangular wave, and a triangular wave into a sine type of wave.



Bias Point Analysis

Once again just simulate the circuit by selecting **Analysis/Simulate** and PSpice will calculate the resulting Voltage and Current Values as default setting.



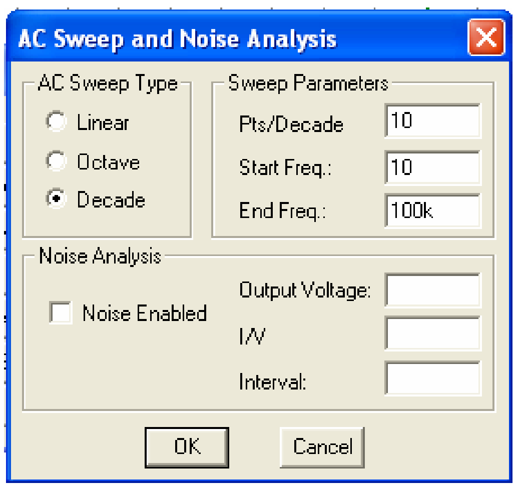
AC Sweep Analysis

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 **Analysis/Setup** and select **AC Sweep.** V1 is the sole AC source thus, the sweep analysis will be from V1.

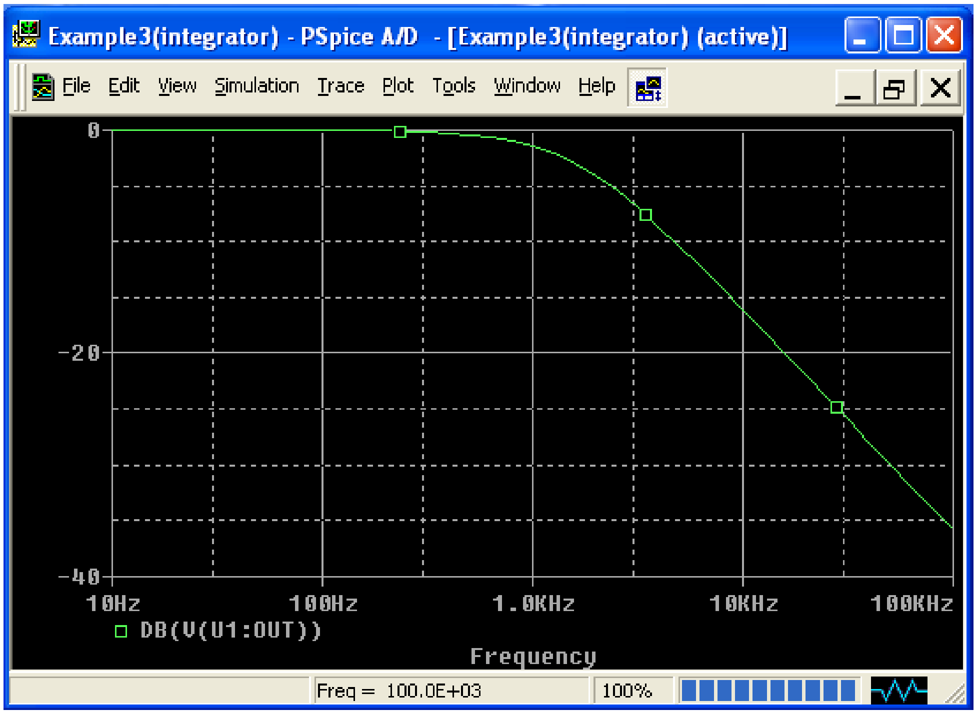
In the AC Sweep and Noise Analysis box enter the following:

Change type to Decade
Pts/Decade : 10
Start Frenquency : 10
End Frequency : 100K

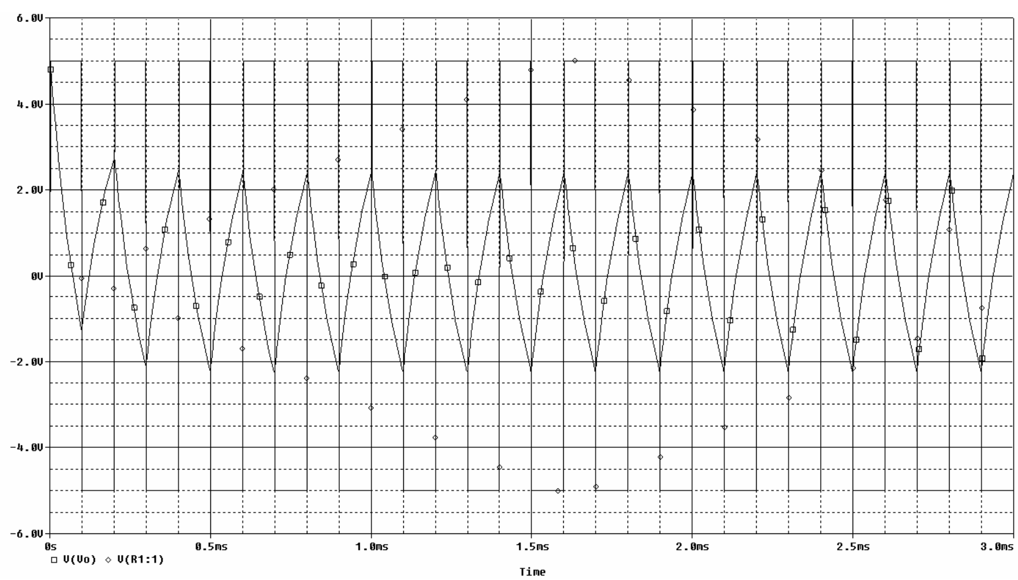


The simulation will decade form with 10 points per decade with the frequency of the AC source swept from 10Hz to 100kHz.

Simulate the circuit.
Probe window will appear.
Select **Trace/Add in** Probe.
Select DB( ) from function and then select V(U1:OUT). This will result in a decibel-frequency graph.



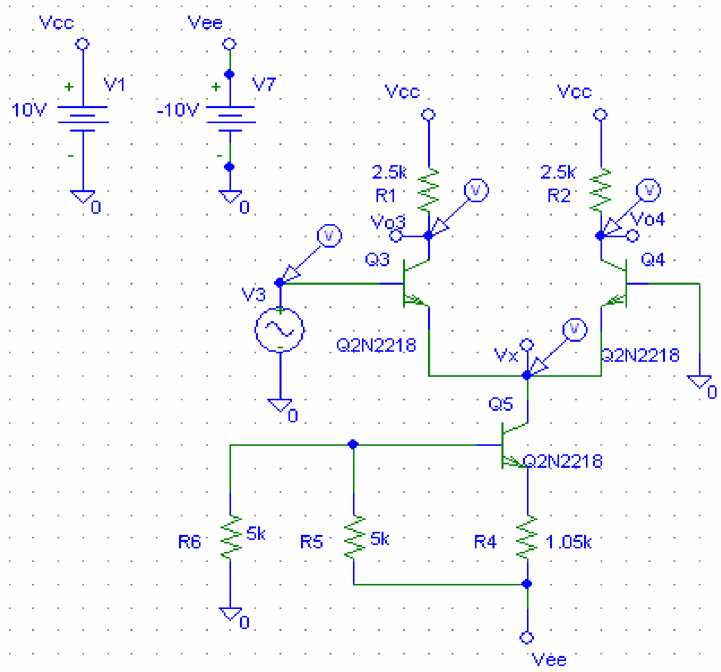
Here we see the results of the integrator when we perform a transient analysis and place probes at Vo and V1.



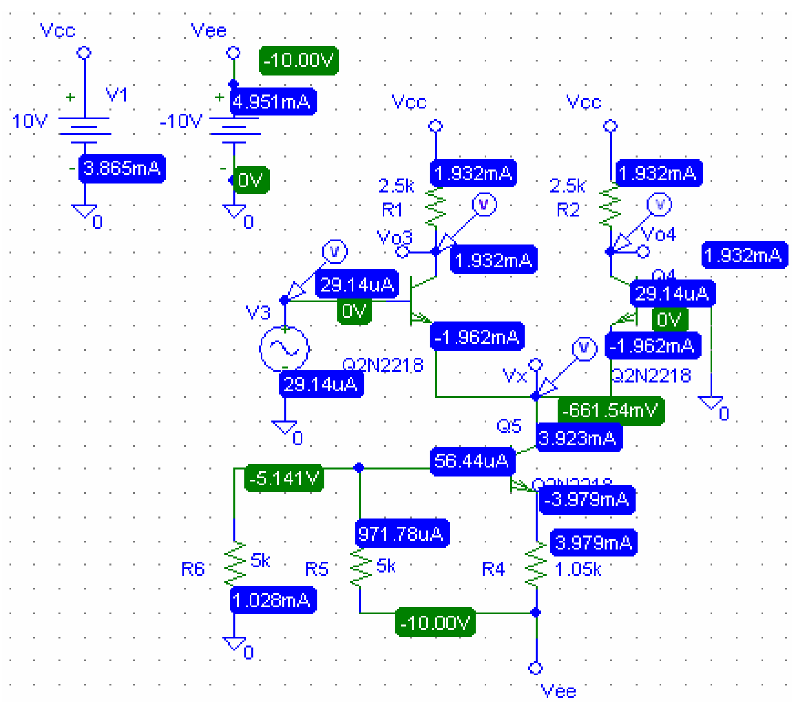
Here we can see that the square wave input what converted into a triangular wave, as expected.

Example 3:

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



Bias Point Analysis



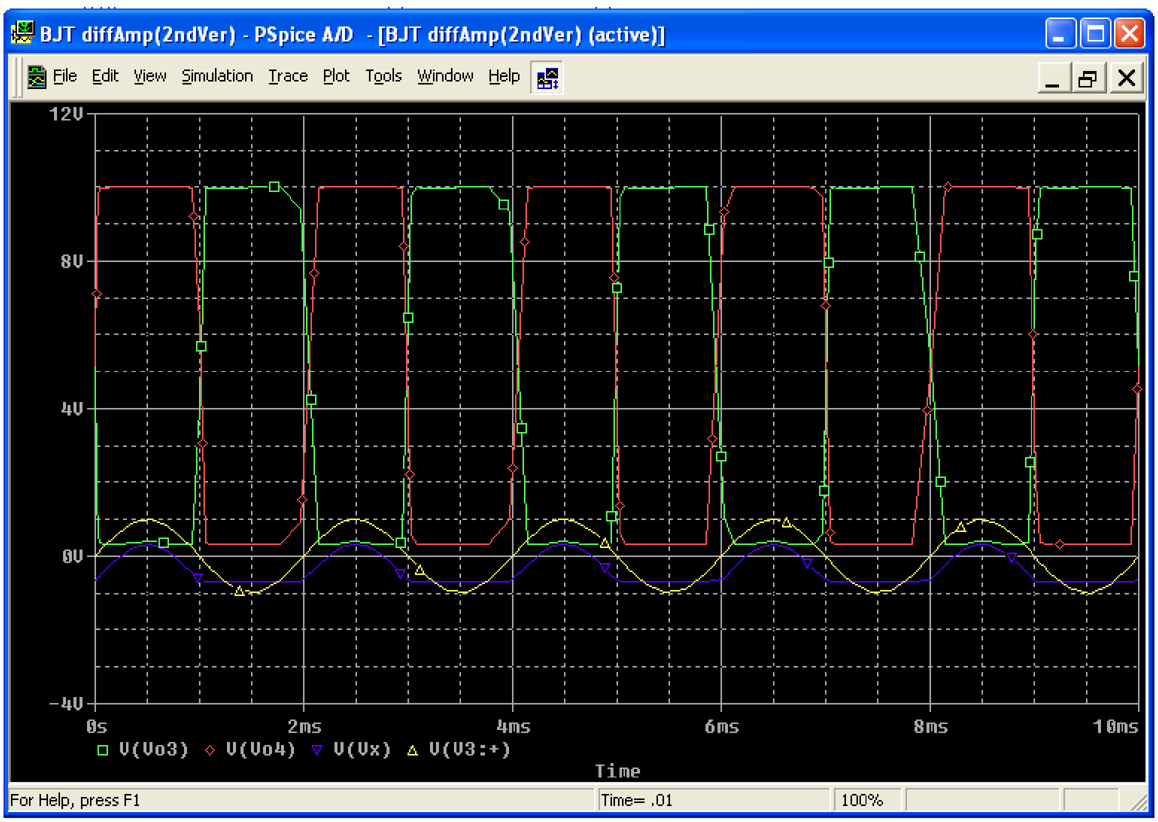
Transient Analysis

Values used for the V3 sine source:
VOFF :0 V
VAMP: 1V
FREQ: 0.5kHz

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

Placing the voltage markers at V3, Vo3, Vo4 and Vx as shown in the above circuit we perform the transient analysis with the following values.

Print Step: 0ms
Final Time: 10ms

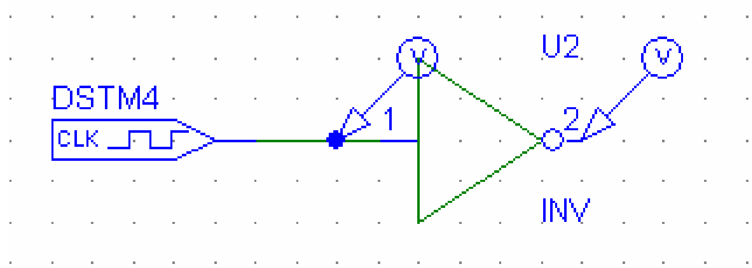


APPENDIX A: Waveforms

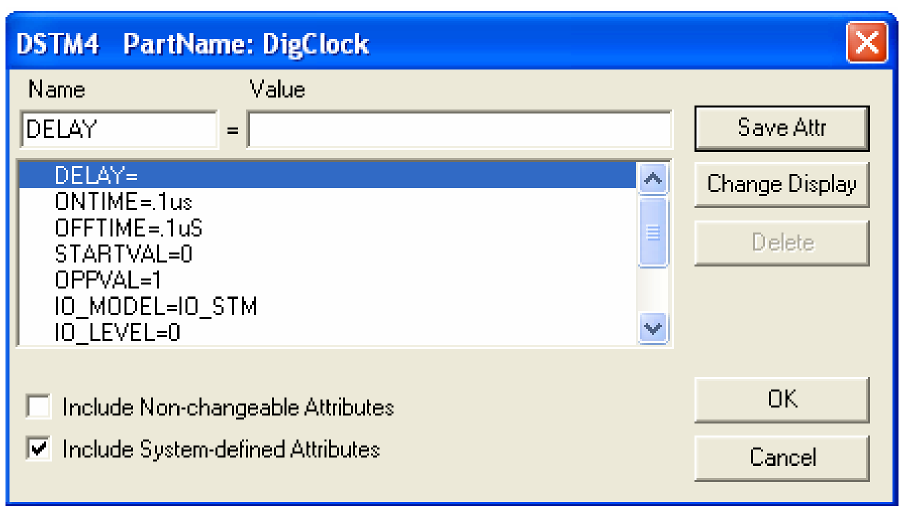
Here we provide some examples as to generating commonly used waveforms in electronics using PSpice.

Generating a Pulse:

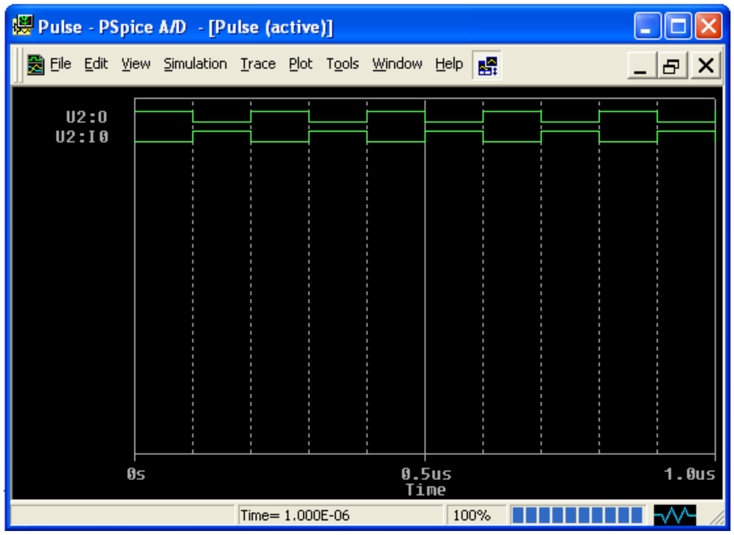
Clock Pulse forms are generated by using the Digital Clock component, which can be obtained by selecting **Draw/Get New Part** toolbar in PSpice Schematics and typing in "DigClock" in the Part Name Box. A circuit is constructed to illustrate the clock Pulse. Note an inverter part, INV, was added to the circuit and voltage markers were placed to illustrate the output of the clock pulse and its inverting form.



Double clicking on the DSMT components will bring up the DigClock window.

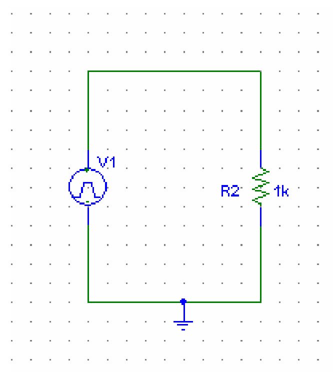


For this example we set the ONTIME to .1us and the OFFTIME to the same value. Below is resulting clock pulse.

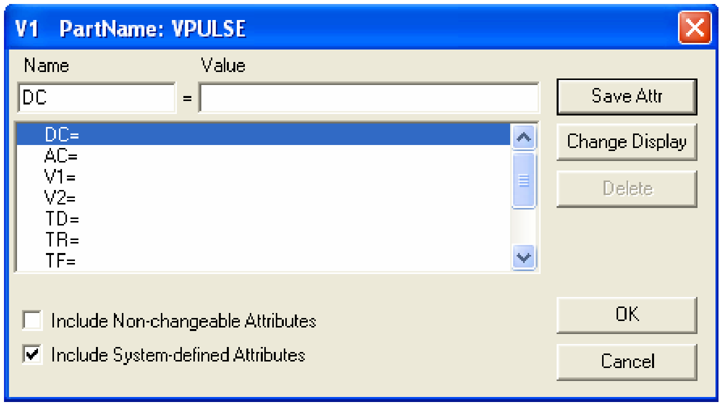


Generating a Square Wave:

Square wave forms are generated by using the **VPULSE** component, which can be obtained by selecting **Draw/Get New Part** toolbar in PSpice Schematics, then typing in "VPULSE" in the Part Name Box. A simple circuit is constructed to illustrate the resulting square wave.



Next, double click on the VPULSE component. This will bring up a window showing additional component variables that we will need to fill in order to generate our square wave.



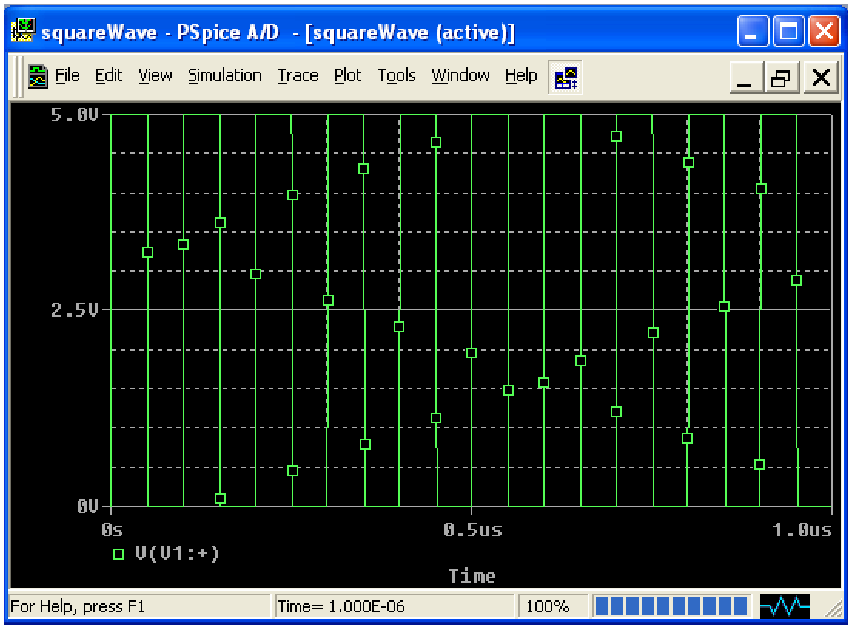
DC - DC component of the wave.
AC - AC component of the wave.
V1 - V1 is the value of the square wave when its is LOW. In our case, suppose the square wave toggles between +5V and 0V. Then the V1 component will be 0V.
V2 - V2 is the value of the square wave when it is HIGH. In our case V2 = 5V.
TD - Time Delay.
TR - Rise time of wave.
TF - Fall time of wave.
PW - Pulse width of wave.
PER – Period of the pulse.

For the purposes of our example we set it to the following values.

V1 = 0V
V2 = 5V
TD = 0V
TR = 1ns

TF = 1ns
PW = 50ns
PER = 100ns

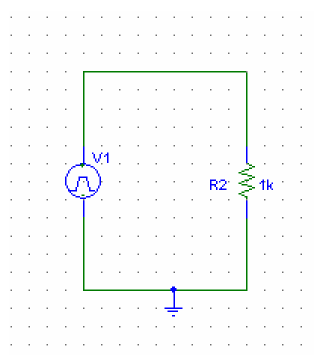
Next by placing a voltage marker at VPULSE and running a transient analysis we can observe our square wave.



Generating a Saw Tooth Wave:

Generating a Saw tooth wave in PSpice is in effect done in the same manner as the previous square wave signal with few parameter changes.

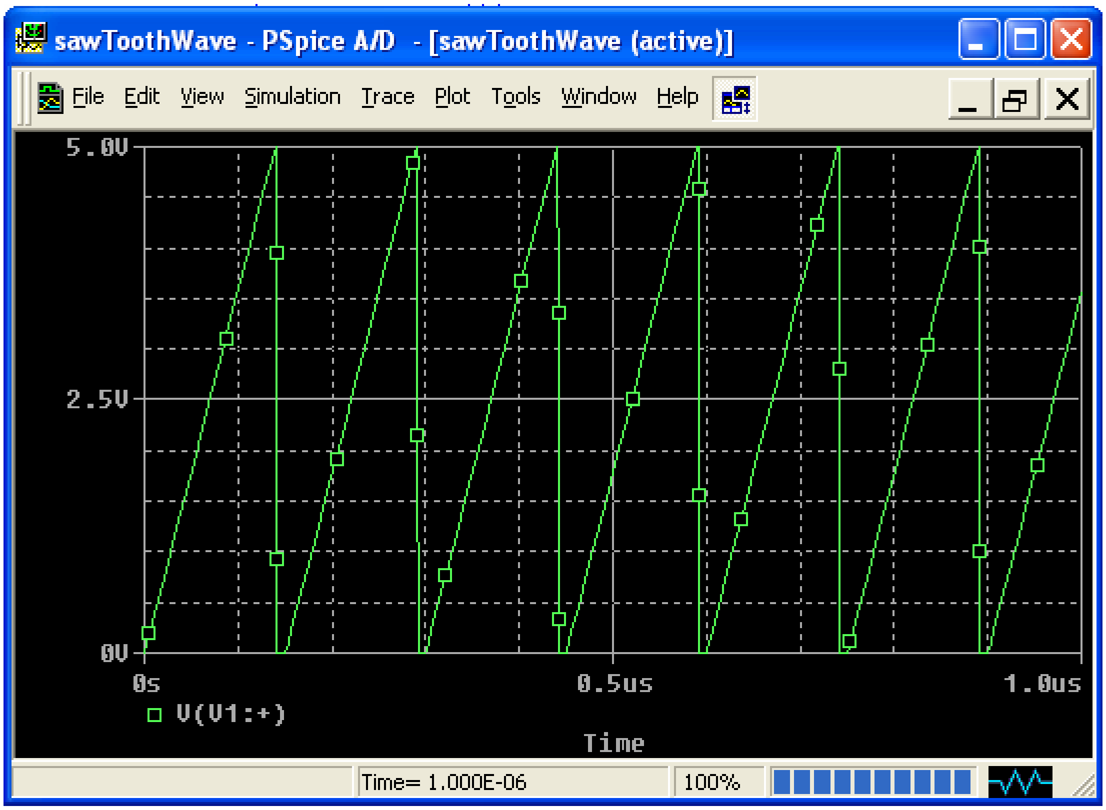
Saw tooth wave forms are generated by using the **VPULSE** component, which can be obtained by selecting **Draw/Get New Part** toolbar in PSpice Schematics,then typing in "VPULSE" in the Part Name Box. The same circuit constructed for the square wave is used to illustrate the resulting saw tooth wave.



We set the values for the VPULSE to the following to obtain a saw tooth signal.

V1 = 0V
V2 = 5V
TD = 0V
TR = 140ns
TF = 1ns
PW = 1ns
PER = 150ns

Below is the resulting saw tooth wave form.

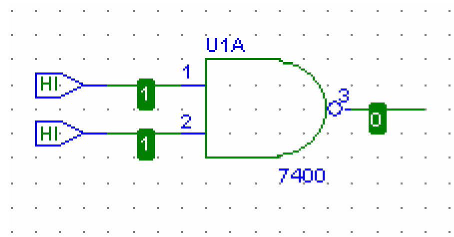


APPENDIX B: Digital Components

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

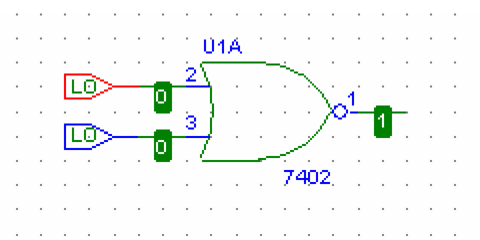
Example 1: The circuit below consists of a 2 input NAND gate. Supposing we name one input of the NAND gate X and the other input Y. Now supposing the input of X = 1(HIGH) and Y = 1(HIGH) we know the output of the NAND gate will be 0.

Select **Draw/Get New Part** toolbar in PSchematics. In the part name box enter "7400" to select a two input NAND gate. Enter "HI" or "LO" in the part name box to obtain either HIGH or LOW correspondingly. Next by running the simulation and enabling the bias voltage display, you should see the corresponding values of the gate as shown below.



The circuit below consists of a 2 input NOR gate. Supposing we name one input of the NOR gate X and the other input Y. Now supposing the input of X = 0 and Y = 0, we know the output of the NOR gate is 1.

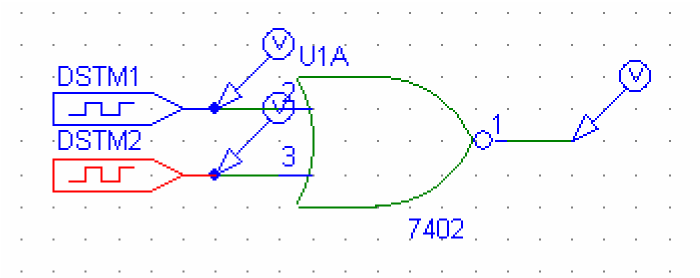
Select **Draw/Get New Part** toolbar in PSchematics. In the part name box enter "7402" to select a two input NOR gate. Enter "HI" or "LO" in the part name box to obtain either HIGH or LOW correspondingly. Next by running the simulation and enabling the bias voltage display, you should see the corresponding values of the gate as shown below.



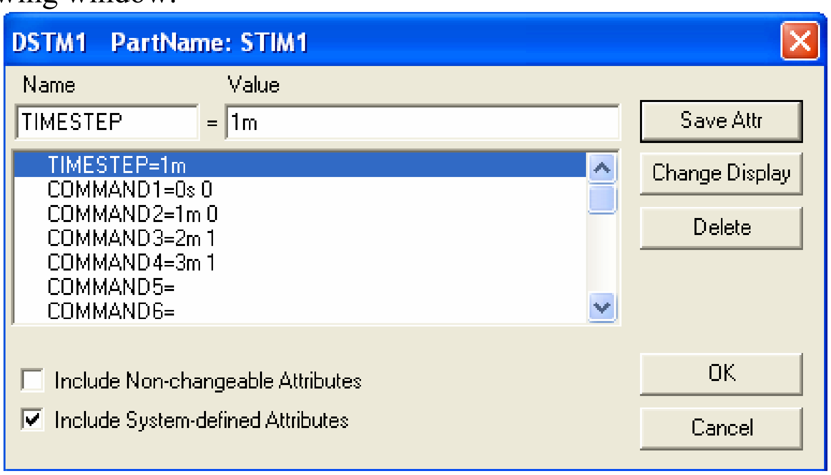
If we wanted to simulate the inputs of the gates as logic wave forms we would perform the following. As and example the circuit will demonstrate the output of a simple NOR gate with the input X and Y as follows.

X	Y	(X+Y)’
0	0	1
0	1	0
1	0	0
1	1	0

Select **Draw/Get New Part** toolbar in PSchematics. In the part name box enter “7402” to select a two input NOR gate. Next enter “STIM1” the part name box, place 2 of them in the schematics. STIM1 is a digital stimulus used to model states as time goes on.

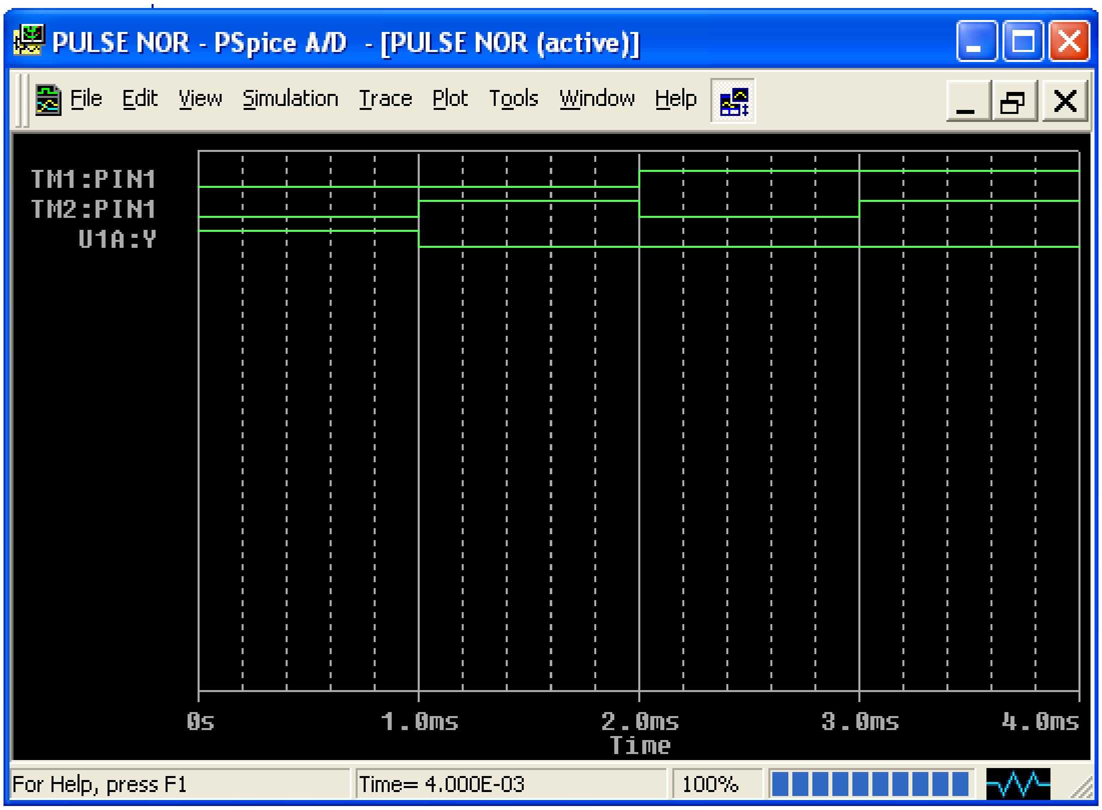


Double click on the STIM1 icon, or as it appears in the schematics as DSTIM. You will get the following window.



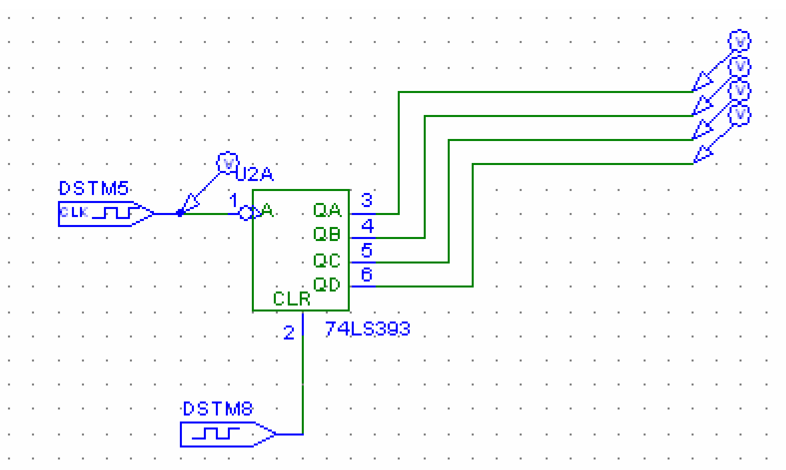
In COMMAND1 the values is 0s 0, this is just setting and input value of 0 at 0seconds, COMMAND2 is setting the value of 0 at 1ms and so on. We set up STIM2 in the same manner, but with the corresponding values.

Finally, by setting up the transient analysis with the proper Print Step and Final Time, we obtain the following result.



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

Part used in the schematic are 74LS393, a 4-bit binary counter, and DigClock for the clock pulse and STIM.



Following the transient simulation, below is the resulting graph.