PSPICE TUTORIAL
PART II: TRANSIENT ANALYSIS
for the
Orcad PSpice Release 9.1 Student Version

INTRODUCTION

This Tutorial will demonstrate the ability to simulate circuits having time-dependent response. We will simulate two circuits. One first-order and the other second-order.

We will first start a new Project. We will name this Transient Analysis Example.

We will design a circuit that includes a Capacitor and Resistor and examine the step response of this R-C circuit. The circuit will also include a voltage source with a *time-dependent* output. SPICE includes a powerful method for creating a time dependent source and this will become a very familiar and useful method.

Now, we will use PSpice to simulate the circuit of Figure 1, below.
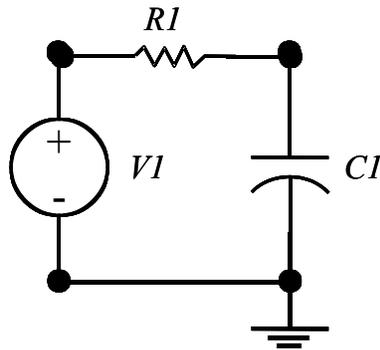

Figure 1

GETTING STARTED

We will follow the procedures we have used in the Tutorial Part 1. After installation, your Program folder should contain the PSpice software. So, select **Start > Programs > PSpice Student > Capture Student**. This will launch the Capture application. Then select **File > New > Project** to start a new project.

You should now configure the application so that an Analog or Mixed A/D (analog or mixed analog and digital design) is selected. Also, you should entire a Project Name.   Please note carefully that the Analog or Mixed A/D checkbox has been highlighted.

Now, upon confirming your selection by clicking on the OK button, another choice will appear. This prompts you as to whether you would like to create a new Project based on a previous Project. For this case, select **Create a blank project** and then press the OK button.

DESIGN

Now, proceed to draw the circuit as shown in Figure 2 with the values of R = 1kilo-Ohm and C = 3.6 Farads.

For the voltage source, select the VPWL Source. This is a "Piece-Wise Linear" Voltage Source.
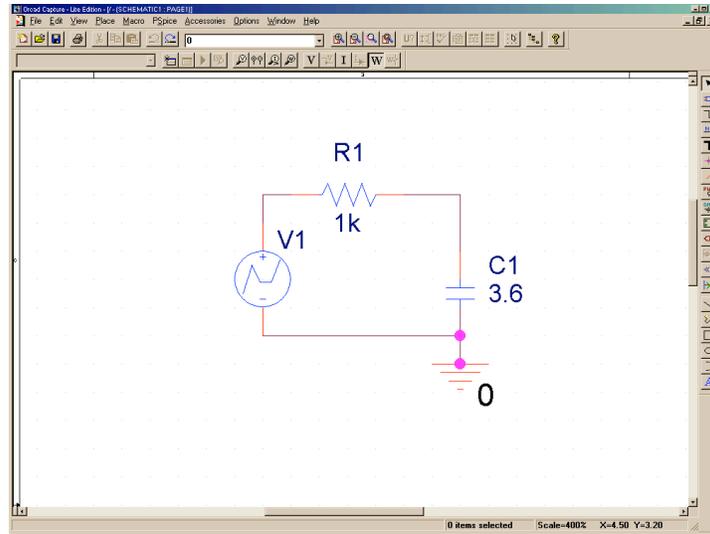
Figure 2

Now, for this system, we wish to simulate the operation of the circuit for an instance where the voltage source varies with time. First, the voltage source will have a value of zero for a period from the time origin (t = 0) to 100 seconds. Then, at 100 seconds we require that the voltage source should rise to 1V within 1 second and then remain at 1V for one hour (3600 seconds), and then return to zero within 1 second. We will allow the voltage source to remain defined until 10,000 seconds.

This is simulated in SPICE with the PWL source. To configure it to operate properly, we use the Property Editor. To reach the Property Editor, simply double click on the Source, V1. Then scroll to the

Figure 3

To create the time dependence we require, we enter the following parameters:

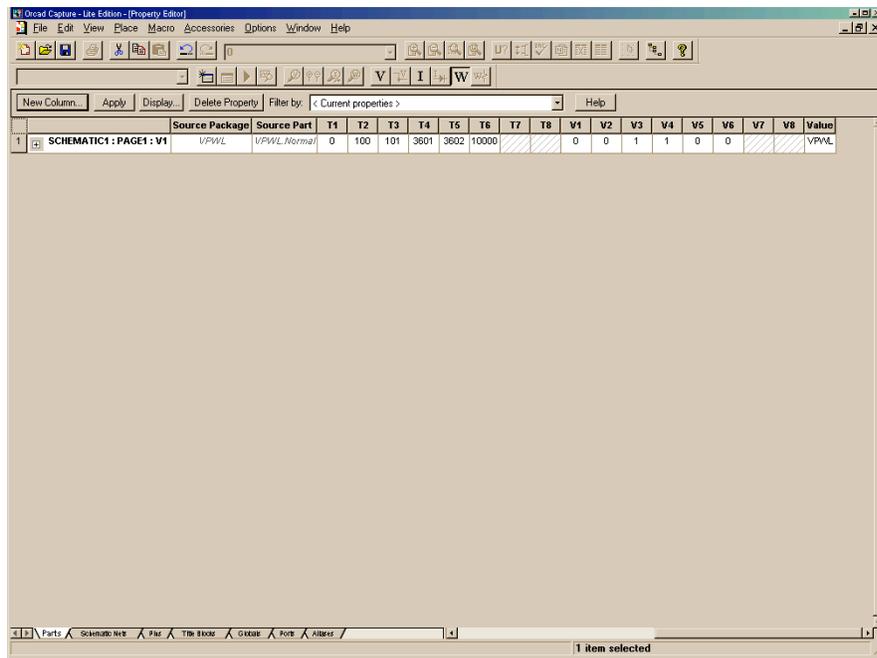| T1 = 0	V1 = 0
T2 = 100	V2 = 0
T3 = 101	V3 = 1
T4 = 3601	V4 = 1
T5 = 3602	V5 = 0
T6 = 10000	V6 = 0

Now, the Property Editor will appear as in Figure 4.

Figure 4

Note that the PSpice assumes the voltage at time zero to be the same as the voltage at -∞. Therefore, in order to create a voltage source that has a different voltage at time zero, we would need to employ the following trick. Just as an example, say that we want to create a voltage source with characteristics; 10V from 0ms to 20ms, -5V from 20ms to 40ms, and 0V thereafter. We must use the following table to create the time dependence.

T1 = 0m	V1 = 0
T2 = 0.1m	V2 = 10
T3 = 20m	V3 = 10
T4 = 20.1m	V4 = -5
T5 = 40m	V5 = -5
T6 = 40.1m	V6 = 0

Such a table, when entered into the properties of the part VPWL, will yield a source whose time varying voltages are shown in the graph below (Figure 5). In the short time intervals of 0.1ms, the voltage varies linearly with time. This can be ignored since 0.1ms is a negligible period of time.
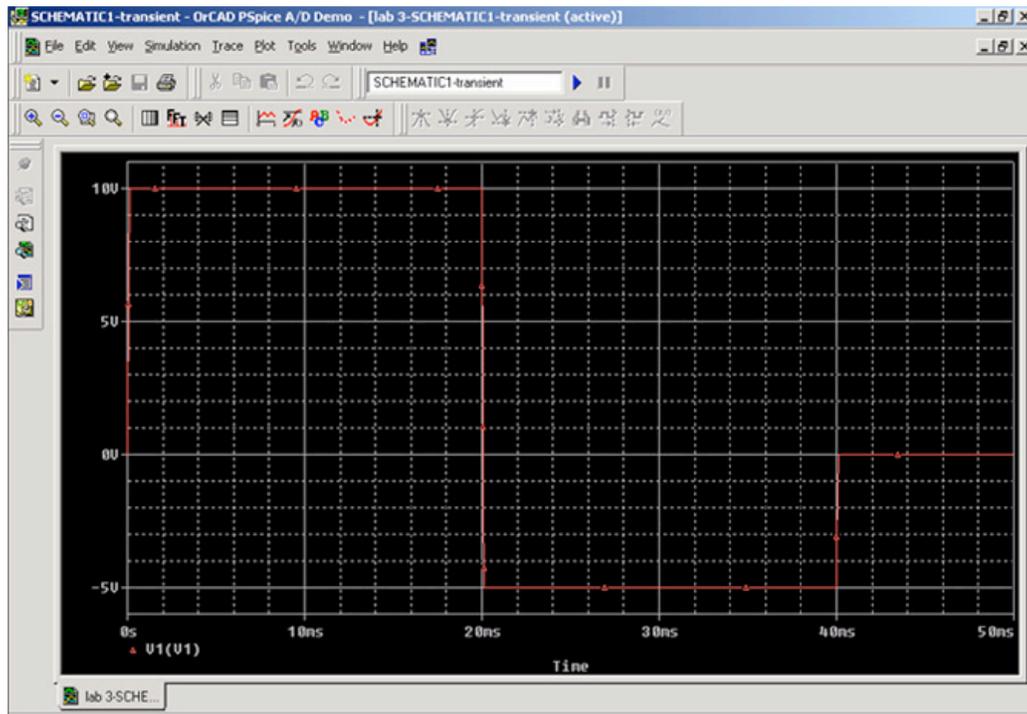

Figure 5

Now, with our circuit complete, we are ready to start a Simulation. First, we must configure the Simulation tool. Proceed to the **PSpice** menu and click on **New Simulation Profile** as in Figure 5.
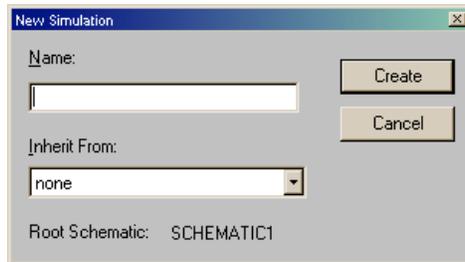
Figure 6

We have entered the name "Transient" into the **Name** field and have left the **Inherit From** entry equal to none.

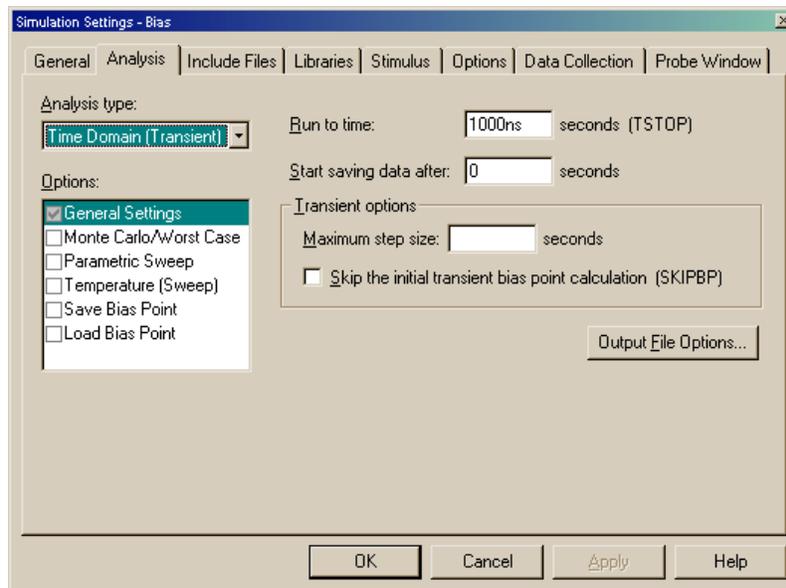So, we now click on **Create**. This will bring up the Simulation Settings-Bias dialog box as in Figure 7.

Figure 7

Now, the **Analysis type** we must select is **Time Domain (Transient).** You can reach this by scrolling through the **Analysis type** selections and clicking on **Time Domain (Transient).** We must also configure the Analysis tool to compute over the proper time interval. We enter 10000 in the **Run to time** edit box. The **Maximum step size** and **Skip the initial bias point calculation** boxes may be left in their default states. Figure 7 shows the Simulation Settings – Transient dialog as it is ready for the next steps.
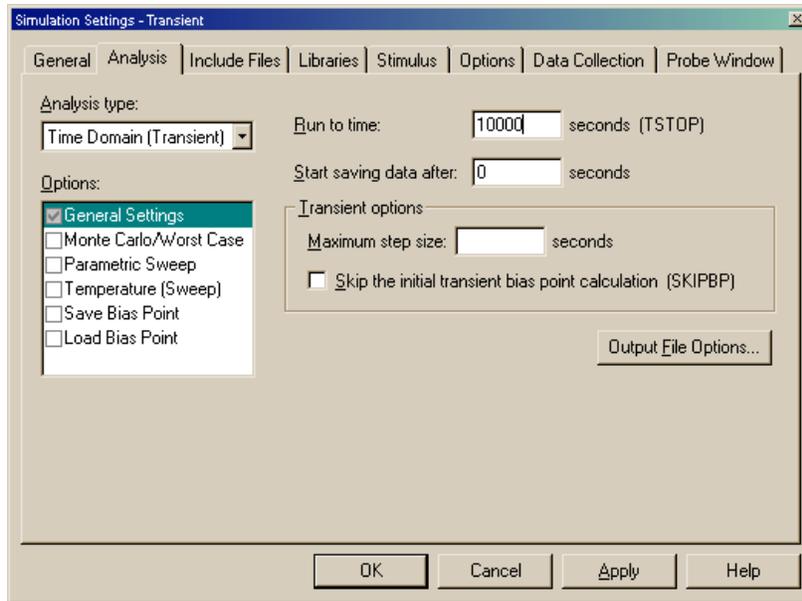
Figure 8

Now, start the Simulation by clicking on **PSpice > Run**. The output will appear as in Figure 9.
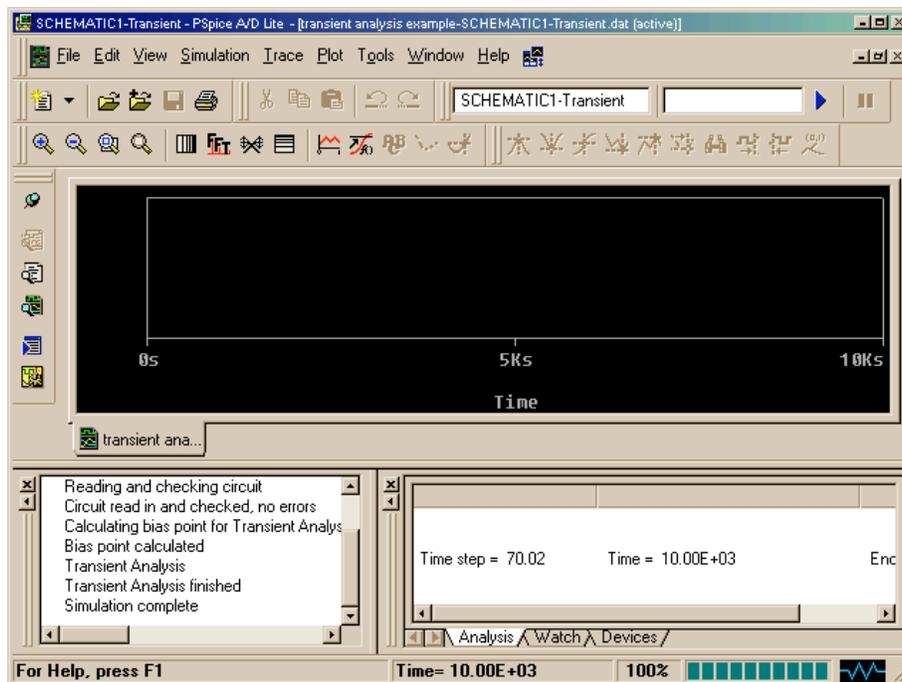

DISPLAYING SIMULATION RESULTS

Figure 9

Now, we would like to plot our results. So, first adjust the size of the Transient Analysis dialog box. This can be done by clicking and dragging this dialog box to the required size. Then, select the **Trace** menu and select **Add Trace**.

Then, select a signal to be plotted. It is good practice to first plot and verify all input signals to ensure that they behave as required. So, we will highlight the Simulation Output Variable V1(R1) as shown in Figure 10. This is the voltage measured at the source, V1, at its terminal connected to R1. After highlighting this, we click **OK** and then observe the plot of Figure 11.
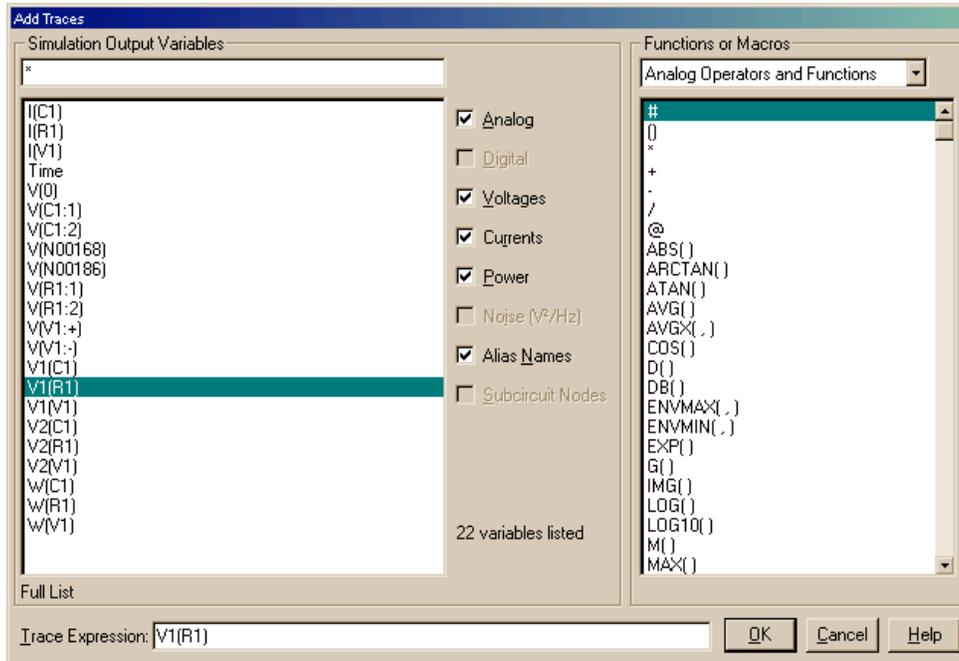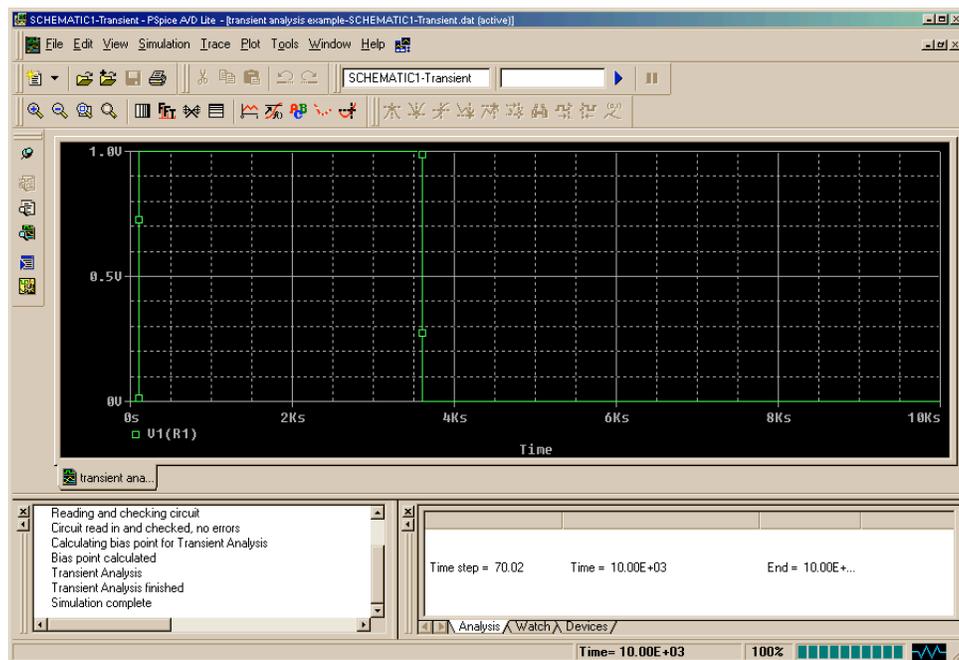

Figure 10

Figure 11

Now, upon examination of Figure 11, we see that the Piece-Wise Linear voltage source shows the required response. Its voltage relative to Ground is 0 until 100 seconds passes, and then remains at 1V for 3600 seconds, when finally it falls to and remains at 0 for 10,000 seconds.

We can now examine the Capacitor voltage. We select **Trace > Add Trace,** and select V(C1:2). This selects the terminal of Capacitor C1 connected to the Resistor R1. The resulting plot is shown in Figure 12.
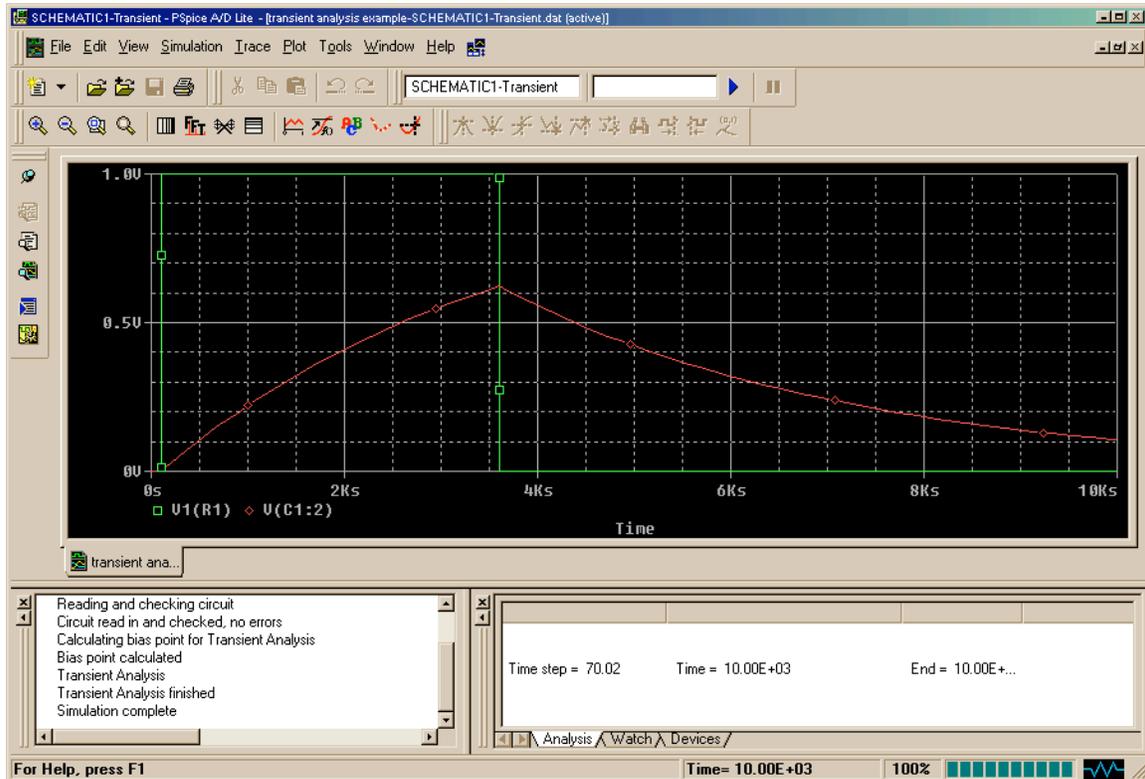

Figure 52

We see from Figure 11 that the Capacitor potential follows the expected exponential step function response. We will use this data to directly compare against the experiment we will perform in Lecture.

You should explore current, voltage, and power plots for this circuit. As an example of an important result is the power dissipation result of Figure 13. Here we are plotting W(C1), the power absorbed by C1. Note that for times prior to 3601 seconds, this power is a positive value. However, at times greater than 3601 seconds, we see that the power dissipation is negative, why is this?
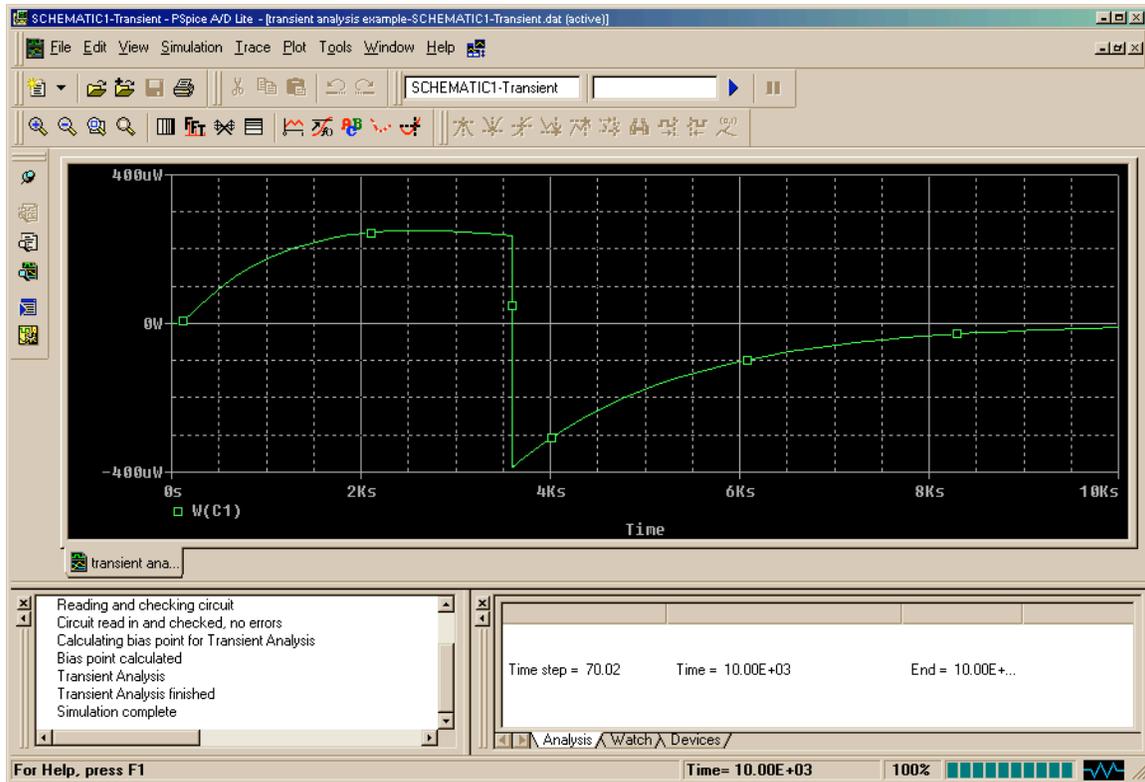
Figure 13

INITIAL CONDITIONS

In the above example, we assumed that the initial conditions were all zero. Suppose however, that the initial conditions were given to us, i.e. at time zero, say the capacitor voltage was 1V. How would this change our analysis?

In order to incorporate for the initial condition for the capacitor, click on the capacitor in your schematic and bring up its property editor. In the column labeled **IC**, enter the initial voltage of the capacitor, in this case, 1V (Figure 14).

Simulate the circuit again and note the change in the capacitor voltage this time (Figure 15). Compare the result of Figure 15 with that of Figure 12.

Note: In order to get the signs right for the initial conditions for a capacitor or inductor, you must pay attention to the orientation of these parts. For a capacitor, the initial voltage is positive at its left-most terminal (i.e. its bottom terminal if the part is rotated once). For an inductor, the initial current enters its left-most terminal (i.e. its bottom terminal if the part is rotated once). The initial conditions for the both capacitors and inductors can be set in the property editor, in the column labeled IC.
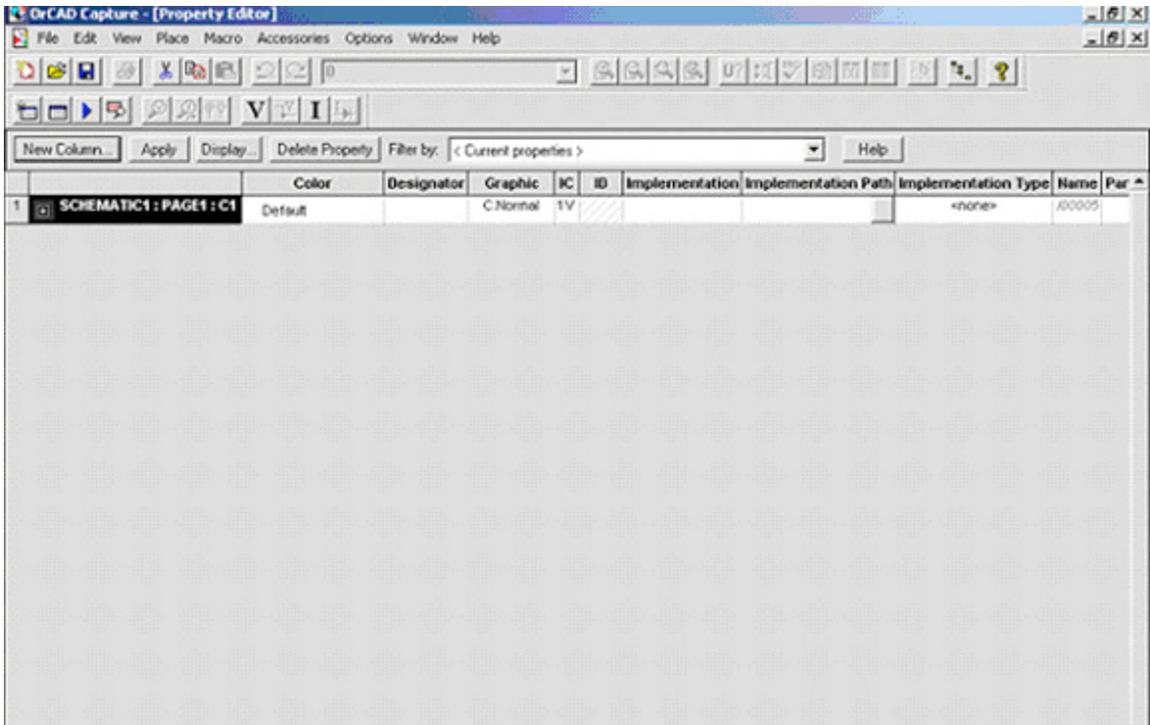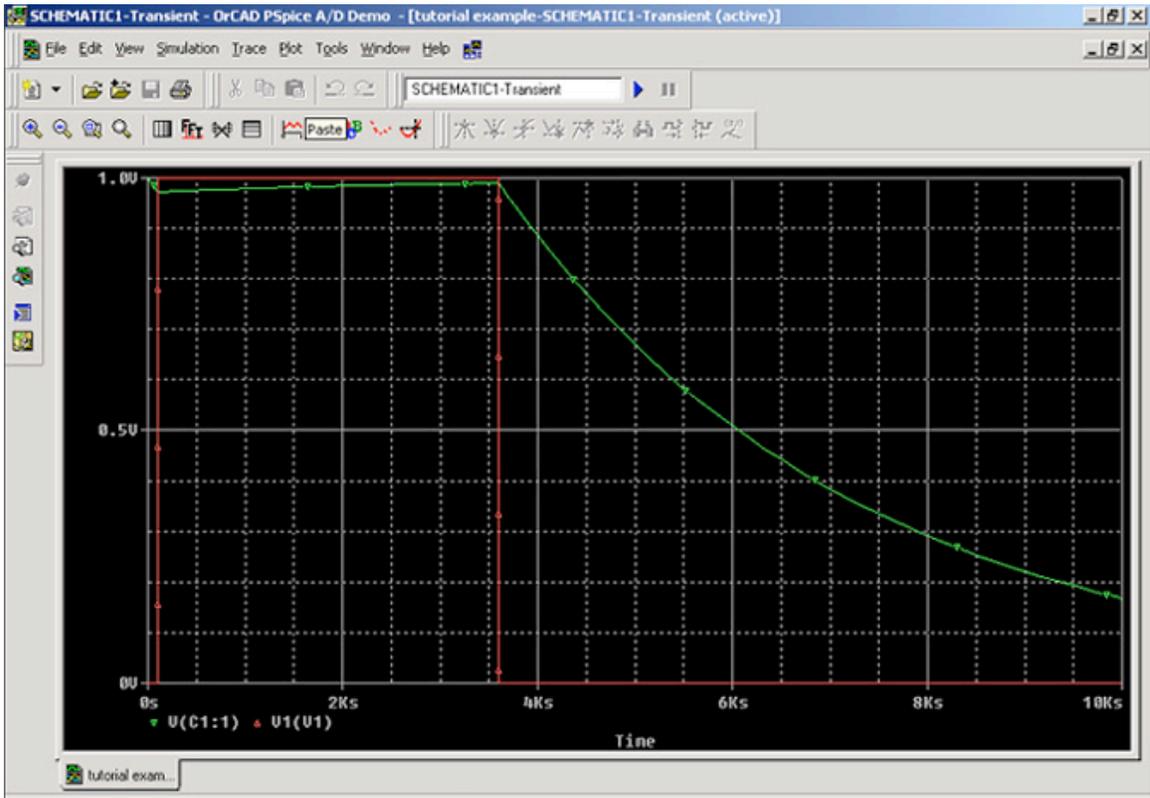
Figure14

Figure 15