# 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 a circuit that is equivalent to a demonstration circuit that we will operate in lecture.

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.

## CIRCUIT II: TRANSIENT ANALYSIS

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

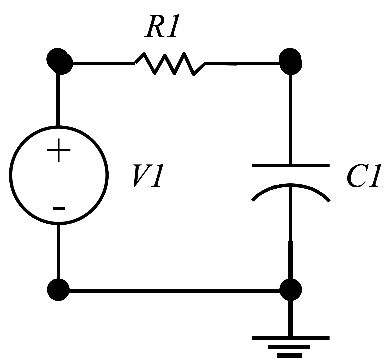


*Figure 1*

## 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.

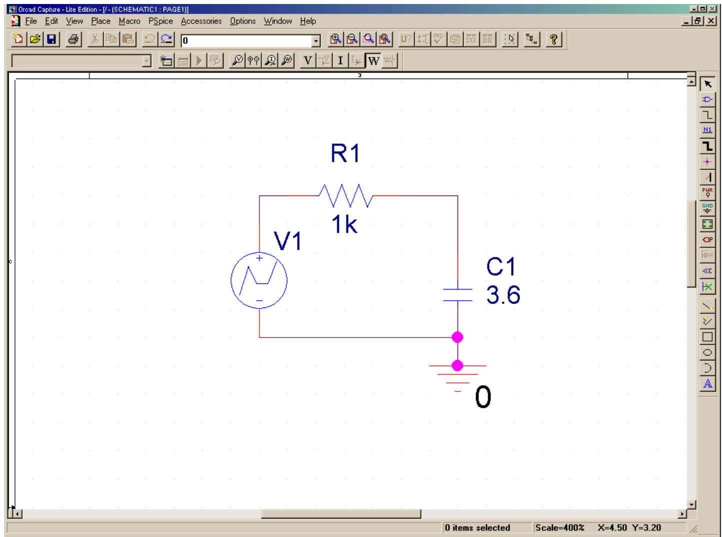


*Figure 2*

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

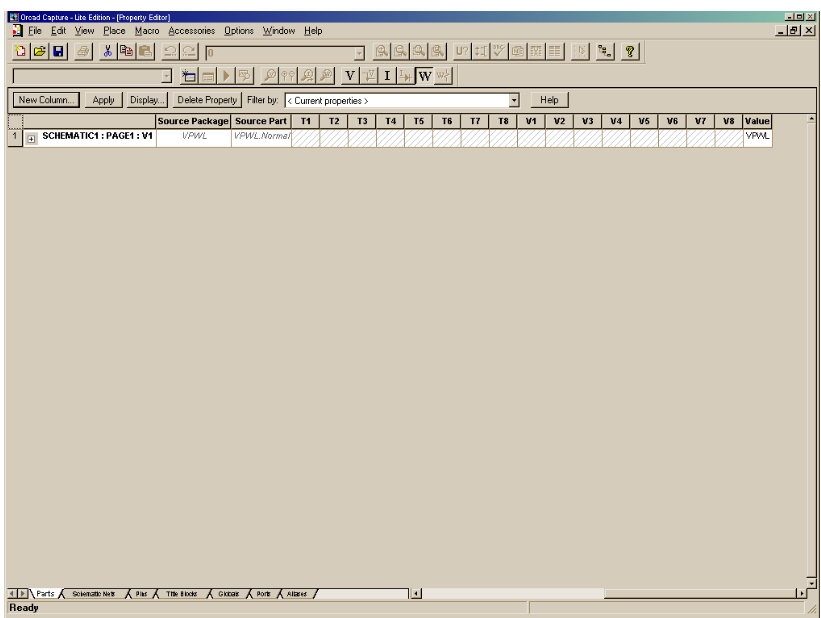


*Figure 3*

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.

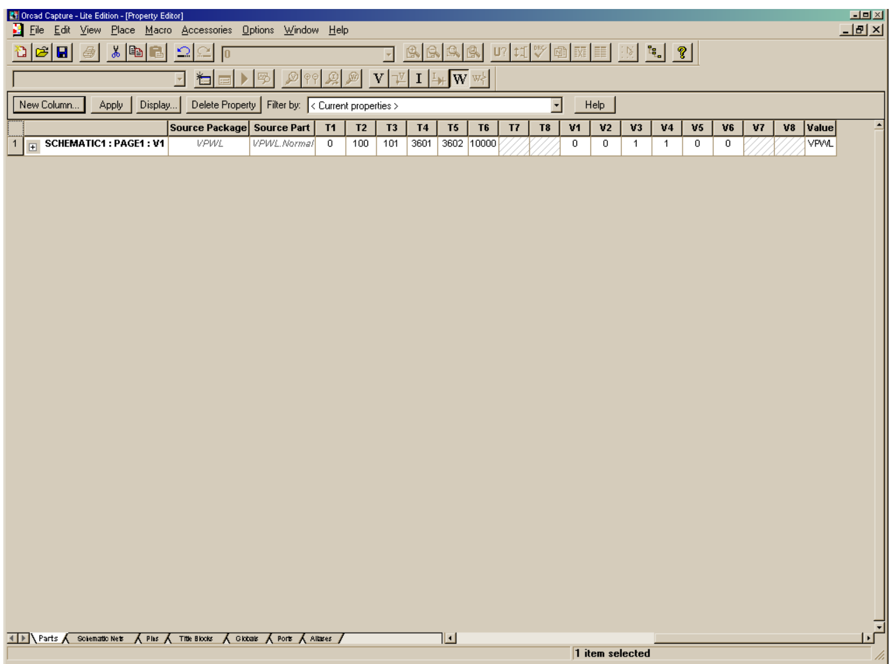


*Figure 4*

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.

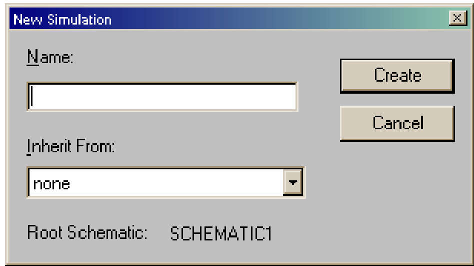


*Figure 5*

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 6.

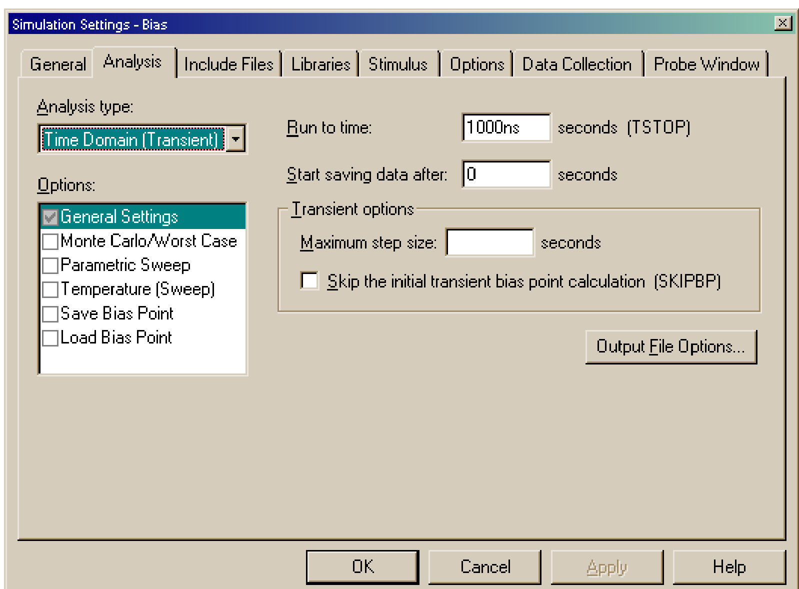


*Figure 6*

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.

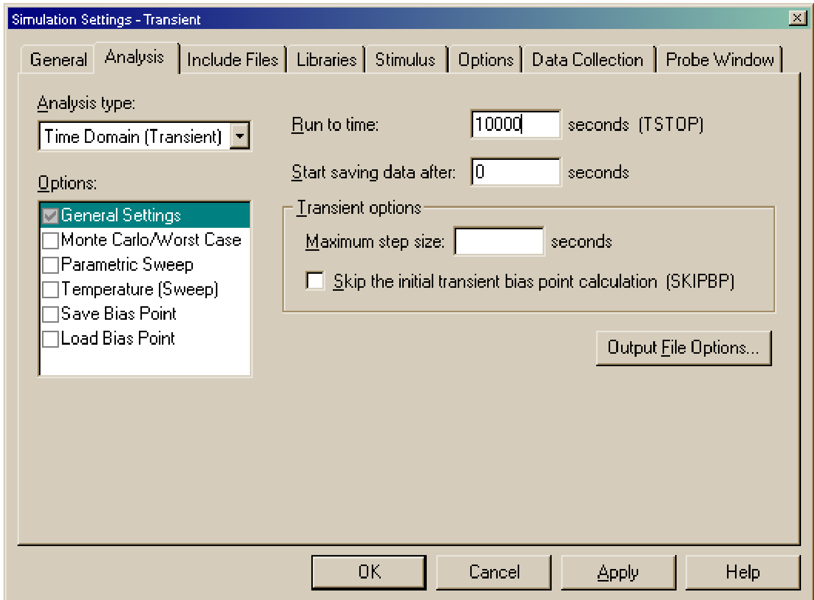


*Figure 7*

Now, start the Simulation by clicking on **PSpice > Run**. The output will appear as in Figure 8

---

## DISPLAYING SIMULATION RESULTS

---

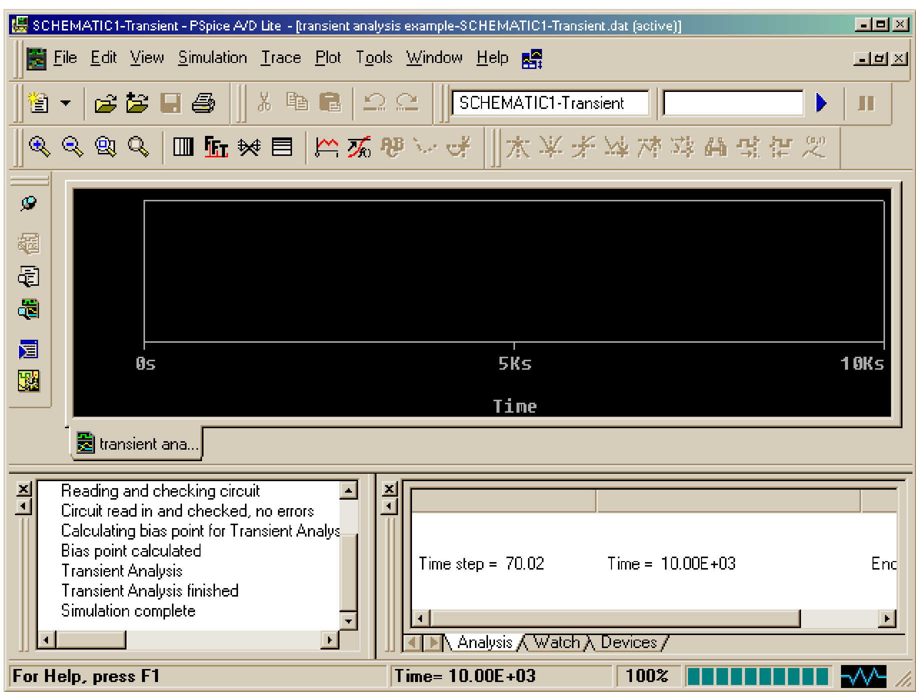


*Figure 8*

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 9. 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 10.

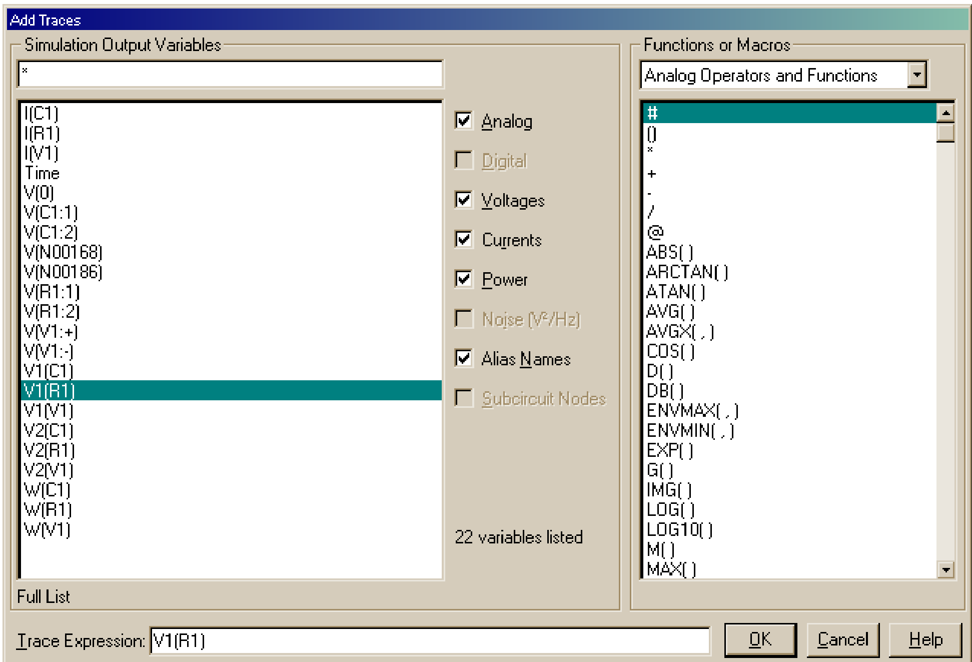


*Figure 9*

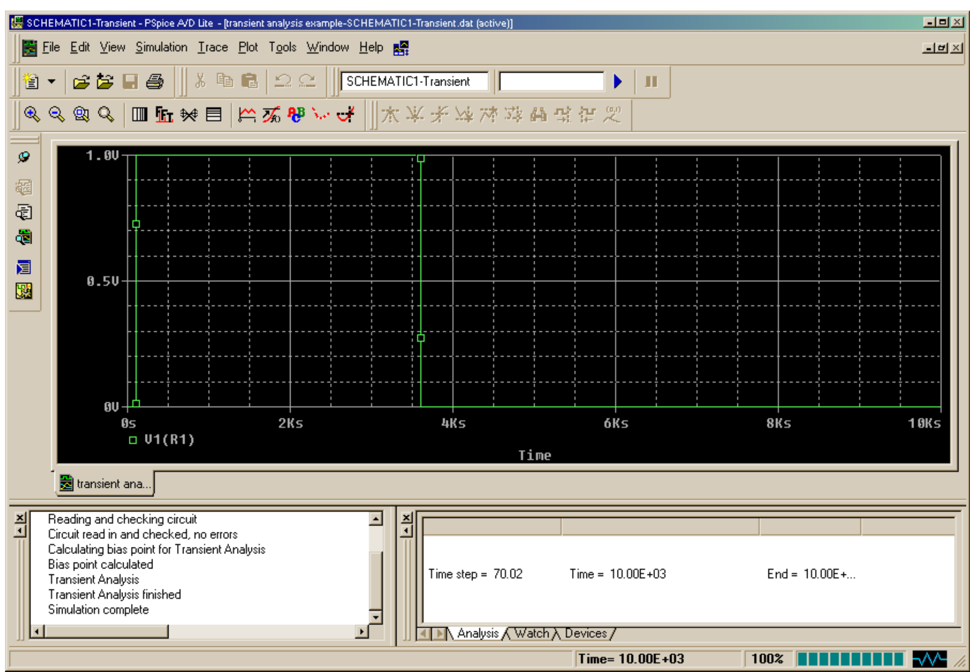


*Figure 10*

Now, upon examination of Figure 10, 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 11.

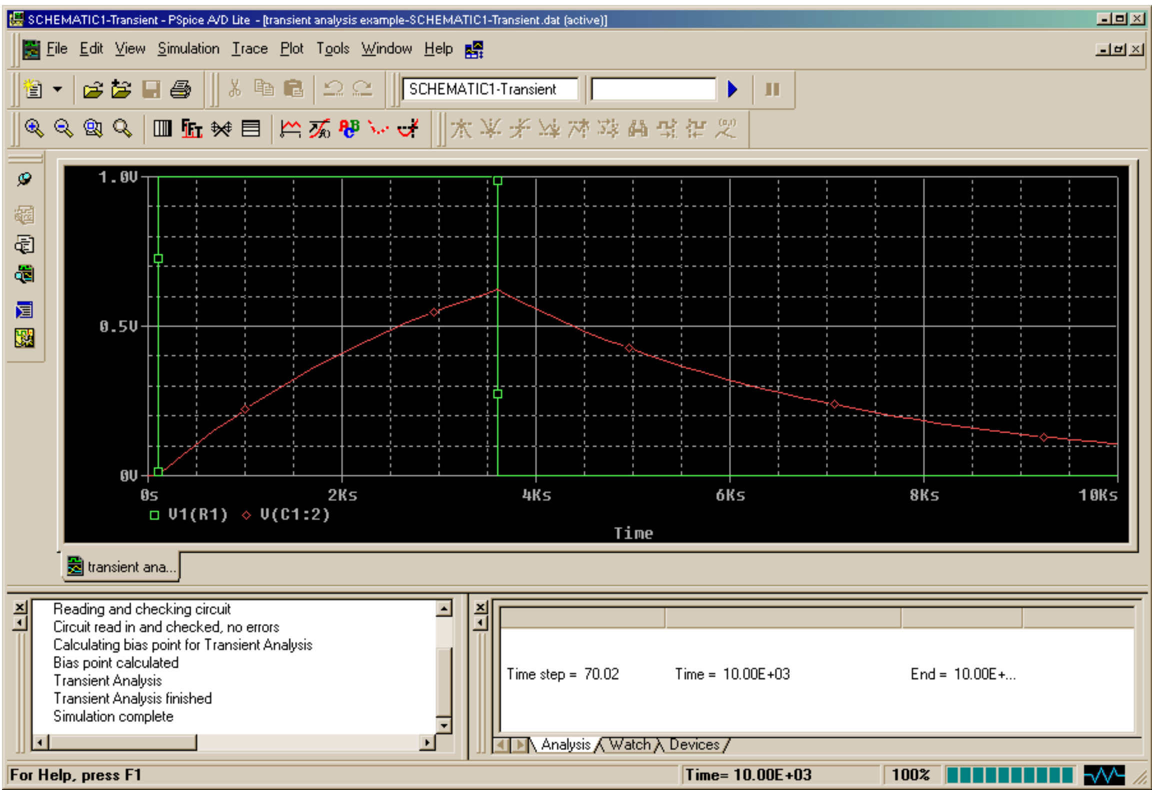


*Figure 11*

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 12. 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?

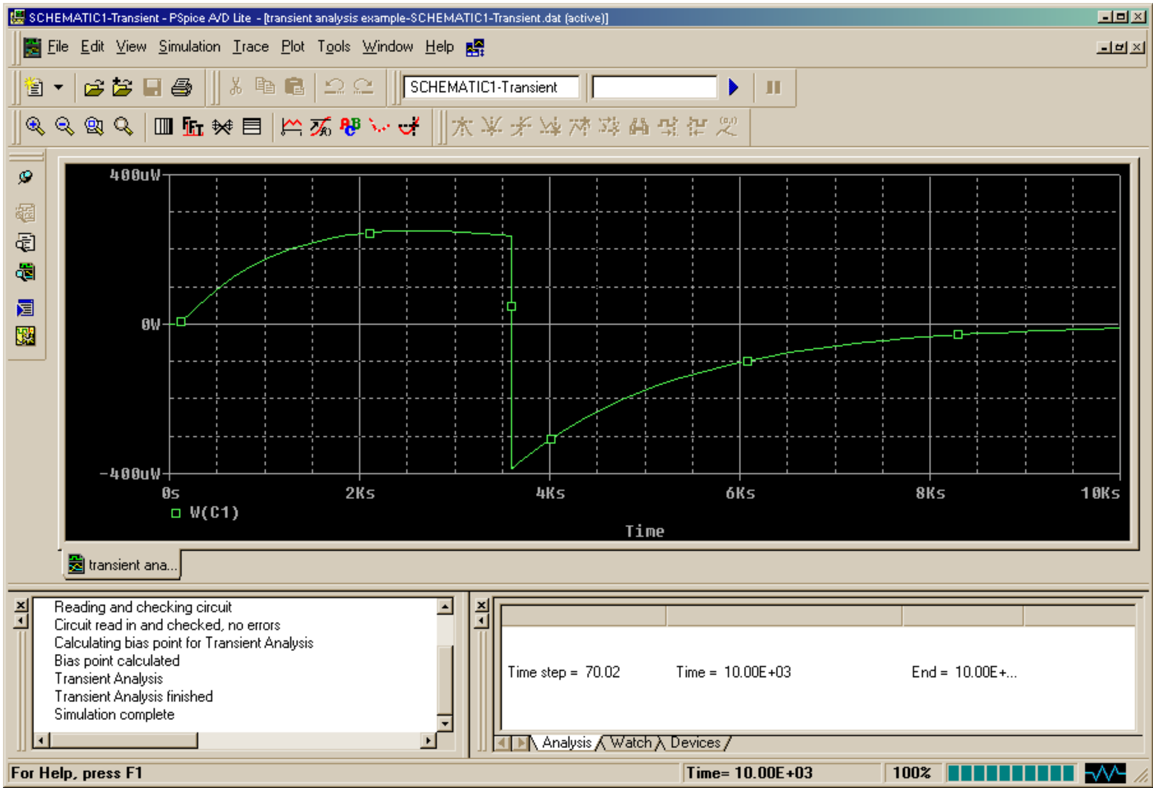
SCHEMATIC1-Transient - PSpice A/D Lite - [transient analysis example-SCHEMATIC1-Transient.dat (active)]
File Edit View Simulation Trace Plot Tools Window Help
SCHEMATIC1-Transient
400uW
0W
-400uW
0s
2Ks
4Ks
6Ks
8Ks
10Ks
W(C1)
Time
transient ana...
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analysis
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
Time step = 70.02
Time = 10.00E+03
End = 10.00E+...
Analysis
Watch
Devices
For Help, press F1
Time= 10.00E+03
100%

*Figure 12*