BRAC University
Department of Electrical and Electronic Enginering
Introduction to PSpice

Objective

The objective of this module is to learn the fundamentals of computer aided circuit simulation using PSpice.

Introduction

SPICE is a powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behavior. This is of particular importance for integrated circuits. It was for this reason that SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies

<p style="text-align:center">**S**imulation **P**rogram for **I**ntegrated **C**ircuits **E**mphasis</p>

PSpice program is extensively used to simulate and predict the behavior of experimental circuits. Simulation is very important as it allows a potential circuit to be tested for errors before it is actually built, thus saving time and cost.

Some Facts and Rules about PSpice

- PSpice is not case sensitive. This means that names such as *Vbus*, *VBUS are vbus* are equivalent.
- All element names must be unique.
- In PSpice circuits can be analyzed in two ways
    - Text based: using "Netlist" to code up circuits
    - Graphics based: using "Schematics" to draw circuits and simulate

Large and Small Numbers in PSpice

Unfortunately, PSpice cannot recognize Greek fonts or even upper vs. lower case. Thus our usual understanding and use of the standard metric prefixes has to be modified. The metric prefix designations used in PSpice are:

| Number	Prefix	Common Name
10^{12}	T or t	tera
10^{9}	G or g	giga
10^{6}	MEG or meg	mega
10^{3}	K or k	kilo
10^{-3}	M or m	mili
10^{-6}	U or u	micro
10^{-9}	N or n	nana
10^{-12}	P or p	pico
10^{-15}	F or f	femto

An alternative to this type of notation, which is in fact, the default for PSpice output data, is "textual scientific notation." This notation is written by typing an "E" followed by a signed or unsigned integer indicating the power of ten. Some examples of this notation are shown below:

656,000 = 6.56E5

*-0.0000135 = -1.35E-5*

Part 2: Introduction to Schematic

In this part you will learn to use the PSpice circuit simulation with the schematic capture front end, Schematics. Click **Schematic** button in the MicroSim Design Manager window to get the following schematic window.
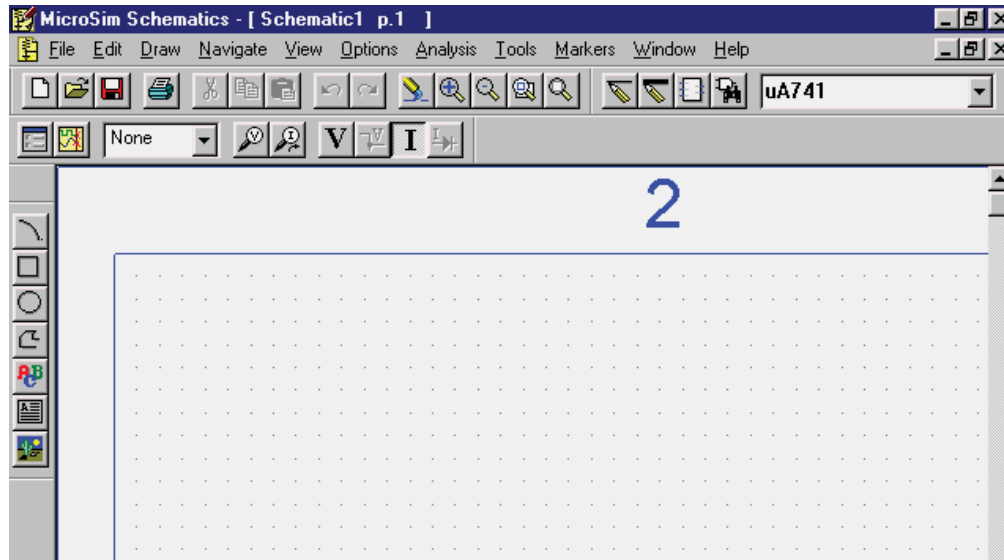
A. Drawing the Circuit

1. Getting the Parts

- The first thing that you have to do is get some or all of the parts you need to simulate your circuit.
- This can be done by
  - Clicking on the 'get new parts' button, or
  - Pressing "Control+G", or
  - Going to "Draw" and selecting "Get New Part..."
- Once this box is open, select a part that you want in your circuit. This can be done by typing in the name or scrolling down the list until you find it
- Some common parts are:
  - r - resistor
  - GND_ANALOG or GND_EARTH -- this is very important, **you MUST have a ground in your circuit**
  - VAC and VDC – voltage sources
  - IAC and IDC – current sources
- Upon selecting your parts, click on the place button then click where you want it to be placed.
- Once you have all the parts you think you need, close that box. You can always open it again if you need more or different parts.
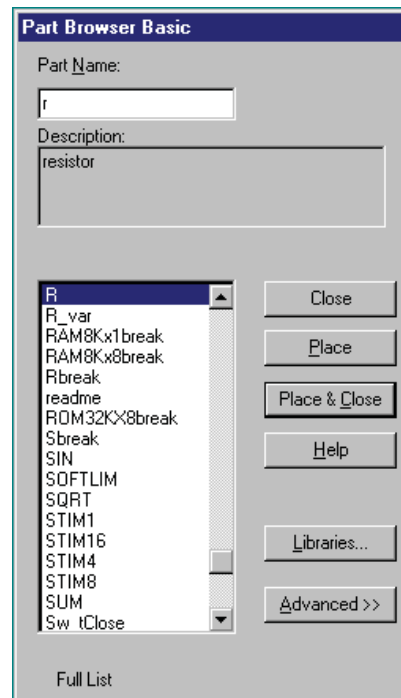

2. Placing the Parts

- You should have most of the parts that you need at this point.
- Now put them in the places that make the most sense (usually a rectangle works well for simple circuits). Just select the part and drag it where you want it.
- To rotate parts so that they will fit in your circuit nicely, click on the part and press "Ctrl+R" (or Edit "Rotate"). To flip them, press "Ctrl+F" (or Edit "Flip").
- If you have any parts left over, just select them and press "Delete".

3. Connecting the Circuit

- Now you'll have to connect the parts with wires.
- Go up to the tool bar and
  - select "Draw Wire" or
  - "Ctrl+W" or
  - go to "Draw" and select "Wire".
- With the pencil looking pointer, click on one end of a part, when you move your mouse around, you should see dotted lines appear. Attach the other end of your wire to the next part in the circuit.
- Repeat this until your circuit is completely wired.
- If you want to make a node (to make a wire go more then one place), click somewhere on the wire and then click to the part (or the other wire). Or you can go from the part to the wire.
- To get rid of the pencil, right click or press "Esc".
- If you end up with extra dots near your parts, you probably have an extra wire, select this short wire (it will turn red), then press "Delete".
- If the wire doesn't go the way you want (it doesn't look the way you want), you can make extra bends in it by clicking in different places on the way (each click will form a corner).

4. Changing the Name of the Part

- To change the name, double click on the present name (C1, or R1 or whatever your part is), then a box will pop up (Edit Reference Designator). In the top window, you can type in the name you want the part to have.
- Please note that if you double click on the part or its value, a different box will appear.
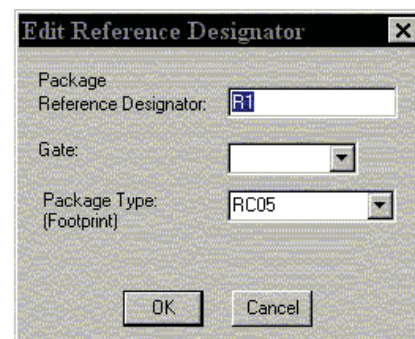
5. Changing the Value of the Part
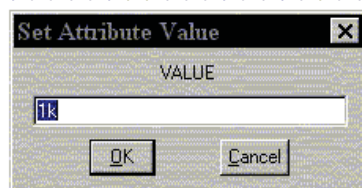
- If you only want to change the value of the part, you can double click on the present value and a box called "Set Attribute Value" will appear. Type in the new value and press OK.

- If you double click on the part itself, you can select VALUE and change it in this box.
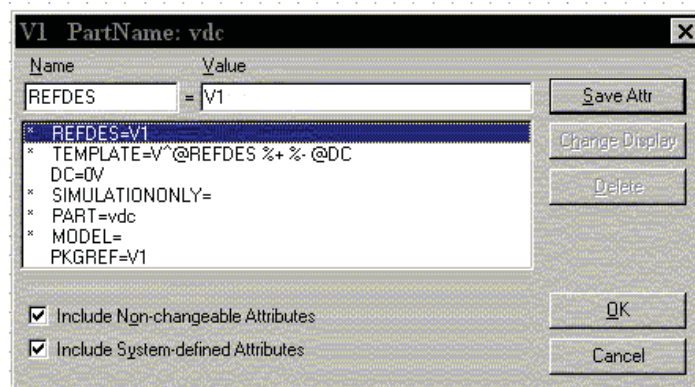
6. Making Sure You Have a GND

This is very important. You cannot do any simulation on the circuit if you don't have a ground. If you aren't sure where to put it, place it near the negative side of your voltage source.

7. **Voltage and Current Bubbles**

- These are important if you want to measure the voltage at a point or the current going through that point.
- To add voltage or current bubbles, go to the right side of the top tool bar and select "Voltage/Level Marker" (Ctrl+M) or "Current Marker" . To get either of these, go to "Markers" and either "Voltage/Level Marker" or "Current Marker".

B. Analysis

- Open the analysis menu by clicking the button. Enable the appropriate analysis options and the press close.
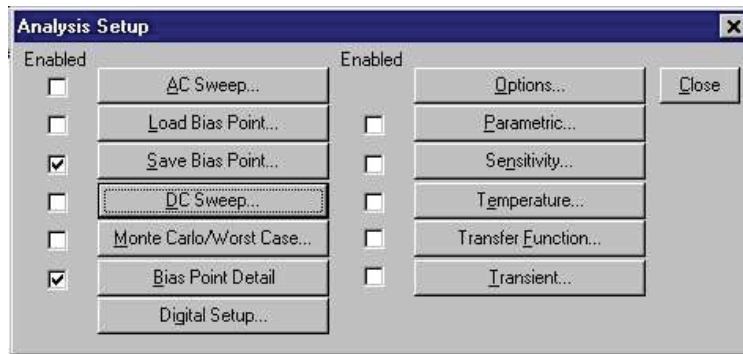
- Click on the Simulate button on the tool bar (or Analysis, Simulate, or F11).

C. Probe

1. Before you do the Probe

- You have to have your circuit properly drawn and saved.
- There must not be any floating parts on your page (i.e. unattached devices).
- You should make sure that all parts have the values that you want.
- There are no extra wires.
- It is very important that you have a ground on your circuit.
- Make sure that you have done the Analysis Setup and that only the things you want are enabled.

2. To Start the Probe

- Click on the Simulate button on the tool bar (or Analysis, Simulate, or F11).
- It will check to make sure you don't have any errors. If you do have errors, correct them.
- Then a new window will pop up. Here is where you can do your graphs.
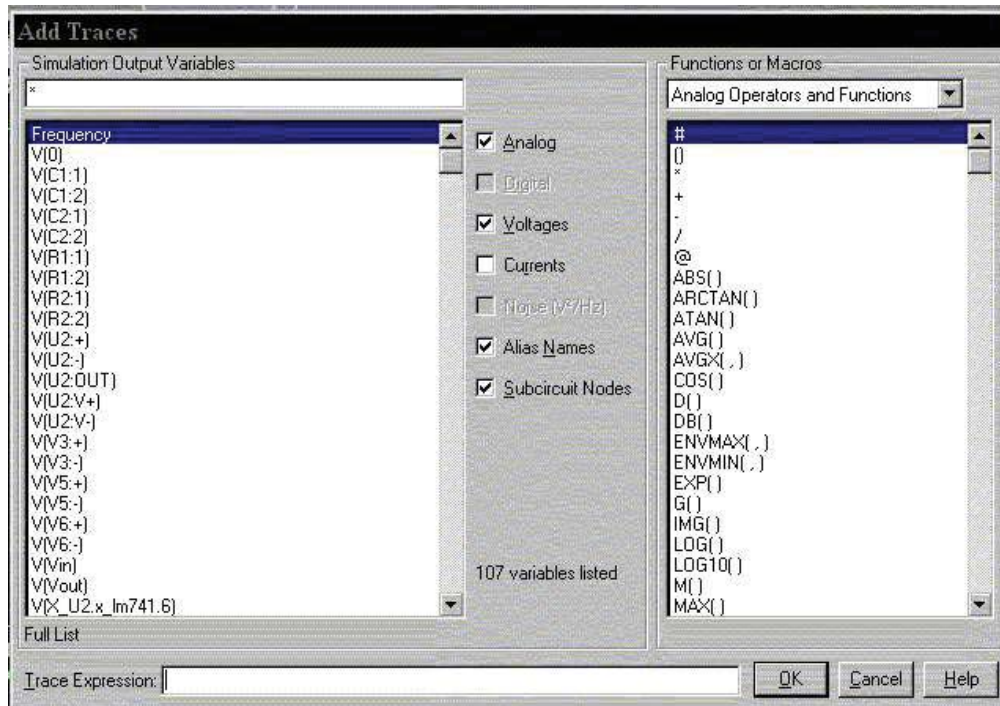
3. Graphing

- If you don't have any errors, you should get a window with a black background to pop up
- If you did have errors, go To "View - Output File" to check the errors.

4. Adding/Deleting Traces

- PSpice will automatically put some traces in.
- To change them go to Trace - Add Trace or on the toolbar. Then select all the traces you want.
- To delete traces, select them on the bottom of the graph and press "Delete".

5. Doing Math

- In Add Traces, there are functions that can be performed, these will add/subtract (or whatever you chose) the lines together.
- Select the first output then either on your keyboard or on the right side, click the function that you wish to perform.
- It is interesting to note that you can plot the phase of a value by using IP(xx), where xx is the name of the source you wish to see the phase for.

Add Traces

Simulation Output Variables

Frequency
V(0)
V(C1:1)
V(C1:2)
V(C2:1)
V(C2:2)
V(R1:1)
V(R1:2)
V(R2:1)
V(R2:2)
V(U2:+)
V(U2:-)
V(U2:OUT)
V(U2:V+)
V(U2:V-)
V(V3:+)
V(V3:-)
V(V5:+)
V(V5:-)
V(V6:+)
V(V6:-)
V(Vin)
V(Vout)
V(X_U2.x_lm741.6)

Full List

☑ Analog
☐ Digital
☑ Voltages
☐ Currents
☐ Noise (V²/Hz)
☑ Alias Names
☑ Subcircuit Nodes

107 variables listed

Functions or Macros

Analog Operators and Functions

#
()
*
+
-
/
@
ABS()
ARCTAN()
ATAN()
AVG()
AVGX(,)
COS()
D()
DB()
ENVMAX(,)
ENVMIN(,)
EXP()
G()
IMG()
LOG()
LOG10()
M()
MAX()

Trace Expression: | OK   Cancel   Help

6. **Finding Points**

- There are Cursor buttons that allow you to find the maximum or minimum or just a point on the line. These are located on the toolbar (to the right).

- Select which curve you want to look at and then select "Toggle Cursor" .

- Then you can find the max, min, the slope, or the relative max or min (is find relative max).

7. **Saving**

- To save your probe you need to go into the tools menu and click display, this will open up a menu which will allow you to name the probe file and choose where to save it. You can also open previously saved plots from here as well.

Circuit Example

- Using the steps explained above draw and simulate the following circuit. This is the same circuit that we used for our example 1.

- In Analysis Setup window only enable "**Bias Point Detail**" option.
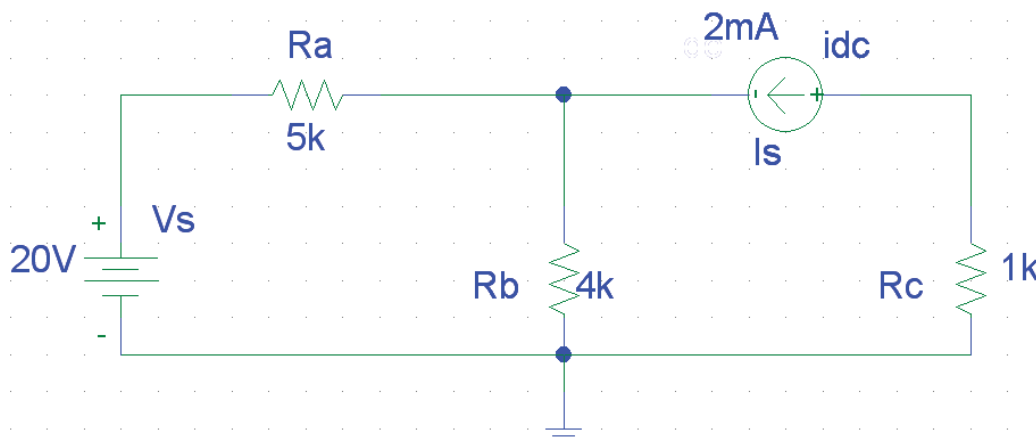
- To examine the node voltages and current through each part go to Analysis menu in the Schematic window and then Display results on Schematic. Put the check marks beside Enable, Enable Voltage Display and Enable Current Display.
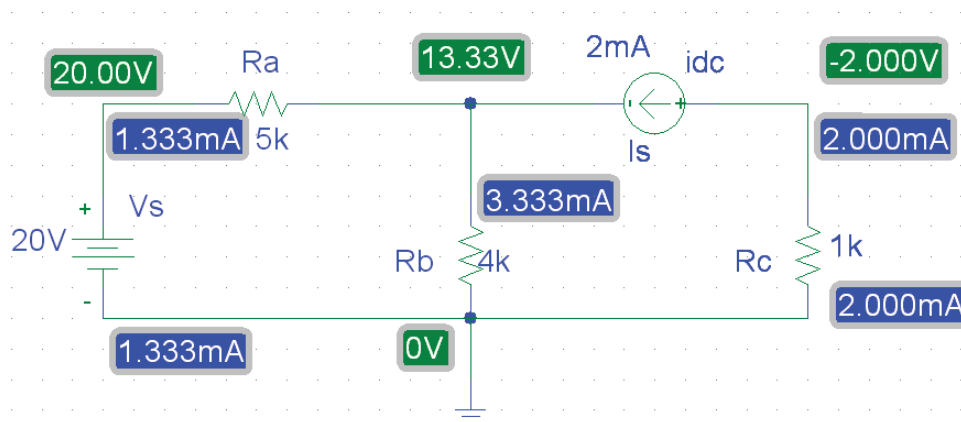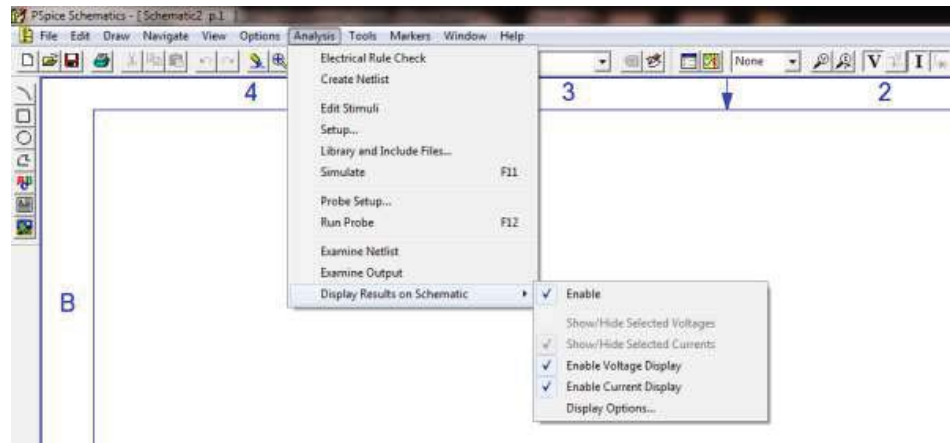


Varying Resistance in Schematics:

1. Place a 1k ohm resistance R$_L$ and then draw the circuit diagram in **Microsim Schematic window**. Save the file.
2. First, double-click the value label of the resistor, R$_L$ that is to be varied. This will open a "Set Attribute Value" dialog box. Enter the name **{RVAR}** (including the curly braces) in place of the component value. Choose "**Get New Part**" from the menu and select the part named **param.**
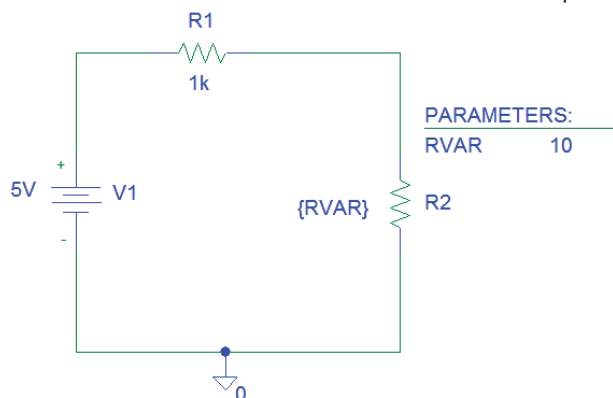
3. Place the box anywhere on the schematic page. Now double-click on the word **PARAMETERS** in the box title to bring up the parameter dialog box. Set the NAME1= **RVAR** (no curly braces) and the VALUE1= 10 (any value) to the nominal resistance value.
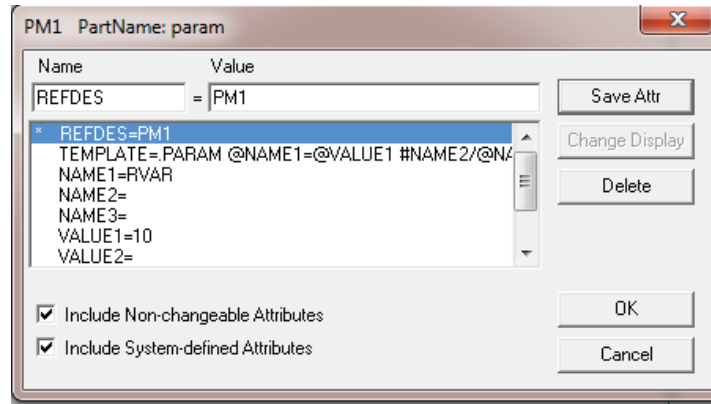

4. In the Analysis Setup dialog box, click the 'DC Sweep' button and Select 'Linear' type and 'Global Parameter' as a sweep variable. Type **RVAR** as a sweep variable 'Name' with Start value = 100, End value = 2000 and Increment = 100.
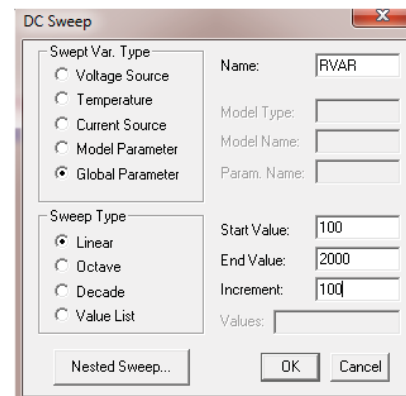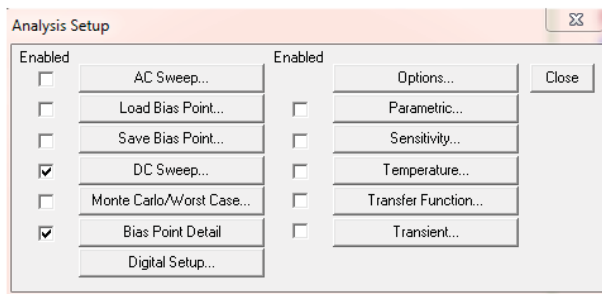
Transient Analysis Basics:

Transient Analysis is used to observe the behavior of a circuit parameter in time domain.

In the Analysis Setup dialog box, click the 'Transient' button and hence set the timing parameters.
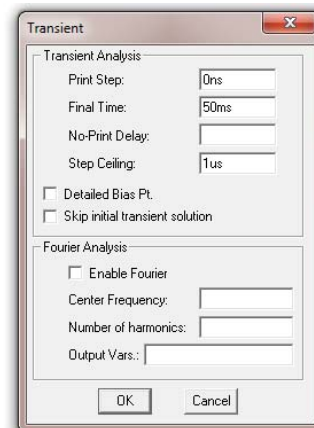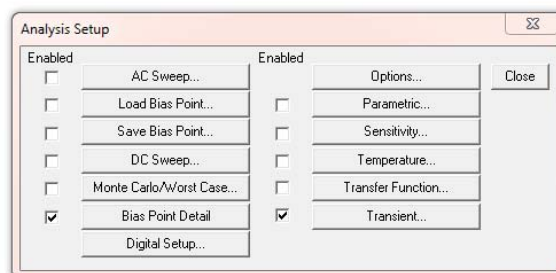
The first parameter, *print step* is the frequency with which data is saved. The actual time steps used by PSpice may be different from this.

The second parameter, *final time* is the value of time at which the simulation will be ended. Since PSpice starts at t = 0, there will be a total of 50ms time span of simulation for the circuit.

The third parameter, *print delay* is the print delay time. In some cases, we do not want to store the data for the entire time span of the simulation. Most of the time, this parameter is set to zero or not used.

The fourth parameter, *step ceiling* is the maximum time step size PSpice is allowed to take during the simulation. Since PSpice automatically adjusts its time step size during the simulation, it may increase the step size to a value greater than desirable for displaying the data. When the variables are changing rapidly, PSpice shortens the step size, and when the variables change more slowly, it increases the step size. Use of this parameter is optional.

Creating a time-varying source using VPULSE:

To create a time-varying periodic source of arbitrary waveshape **VPULSE** is used. Choose "**Get New Part**" from the menu and select the part named **VPULSE.** Now double-click on its symbol to bring up the following dialog box. Set the parameter values appropriately.
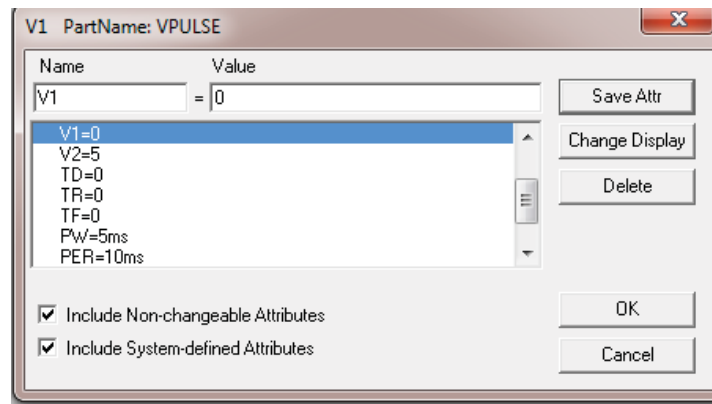
For a 100Hz square wave signal ranging from 0 to 5V with 50% duty cycle
set DC=0, AC=0, V1=0, V2=5, TD= 0, TR=0, TF=0, PW=5ms, PER=10ms.

The significance of the parameters are explained in the figure below.