PSPICE 16.6 Tutorial

The outline of this tutorial is as follows:

- Introduction

- Lesson 1: Basic DC Anlysis, DC Bias

- Lesson 2: Transient Analysis Basics
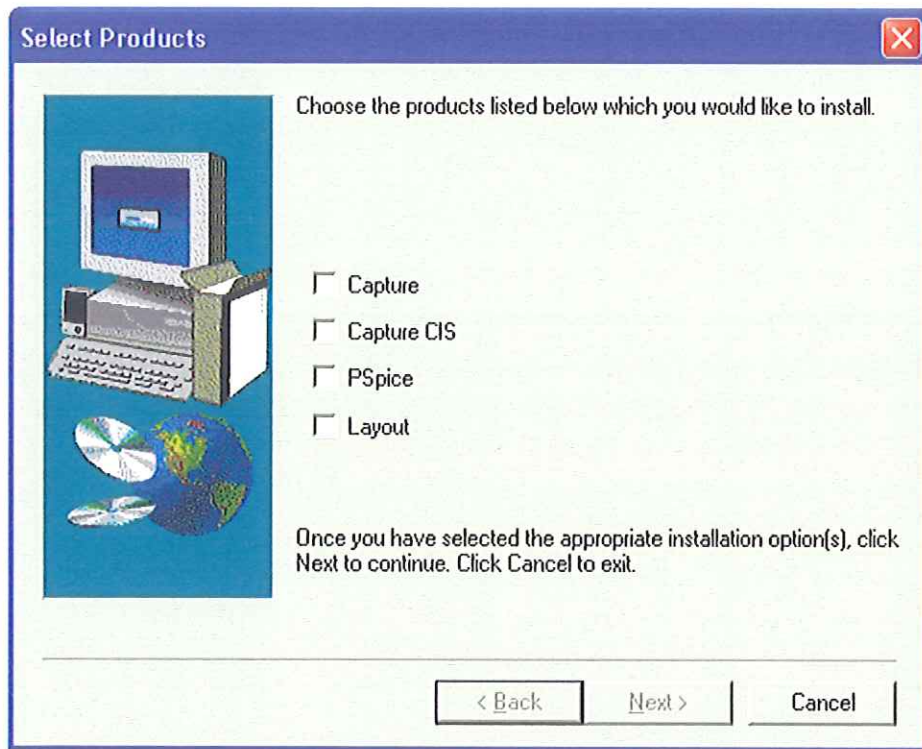
- Lesson 3: Advanced Probe Functions

-

- Lesson 5: Parametric Analysis

You can download PSPICE 16.6 from the following location:
http://www.cadence.com/products/orcad/pages/downloads.aspx#demo

Follow the instructions for downloading and installing:
OrCAD PCB Designer Lite DVD, OrCAD 16.6 Demo Software (All Products)

Select Products

Choose the products listed below which you would like to install.

☐ Capture

☐ Capture CIS

☐ PSpice

☐ Layout

Once you have selected the appropriate installation option(s), click Next to continue. Click Cancel to exit.
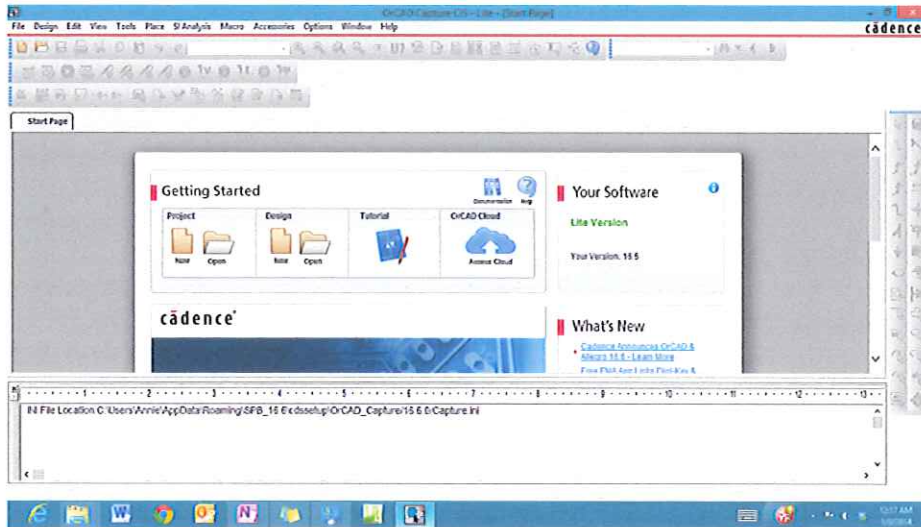
< Back   Next >   Cancel

You must install Capture and Pspice to be able to simulate the circuits shown in this tutorial. Capture is the schematic layout of the circuit which produces the netlist for Pspice to simulate and display the data. You must have both installed. Capture CIS and Layout do not have to be installed. The remainder of the steps in the installation are fairly straight-forward and need no explanation.
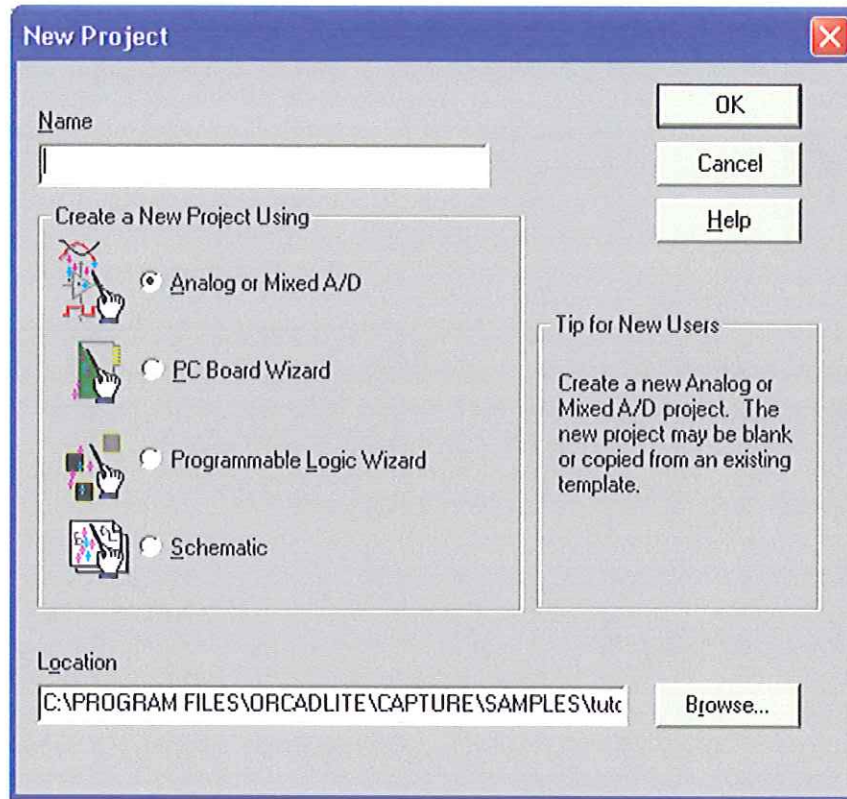
Pspice   Tutorial
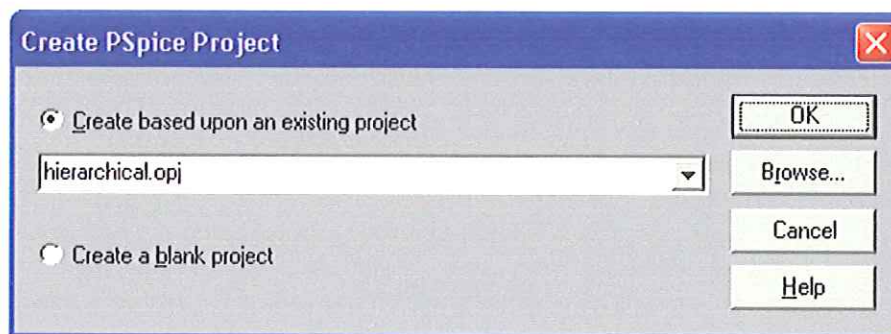## Lesson 1: The Basics of DC Analysis

To get started, open Capture Lite Edition from wherever it is in the start menu. You should see the following screen.
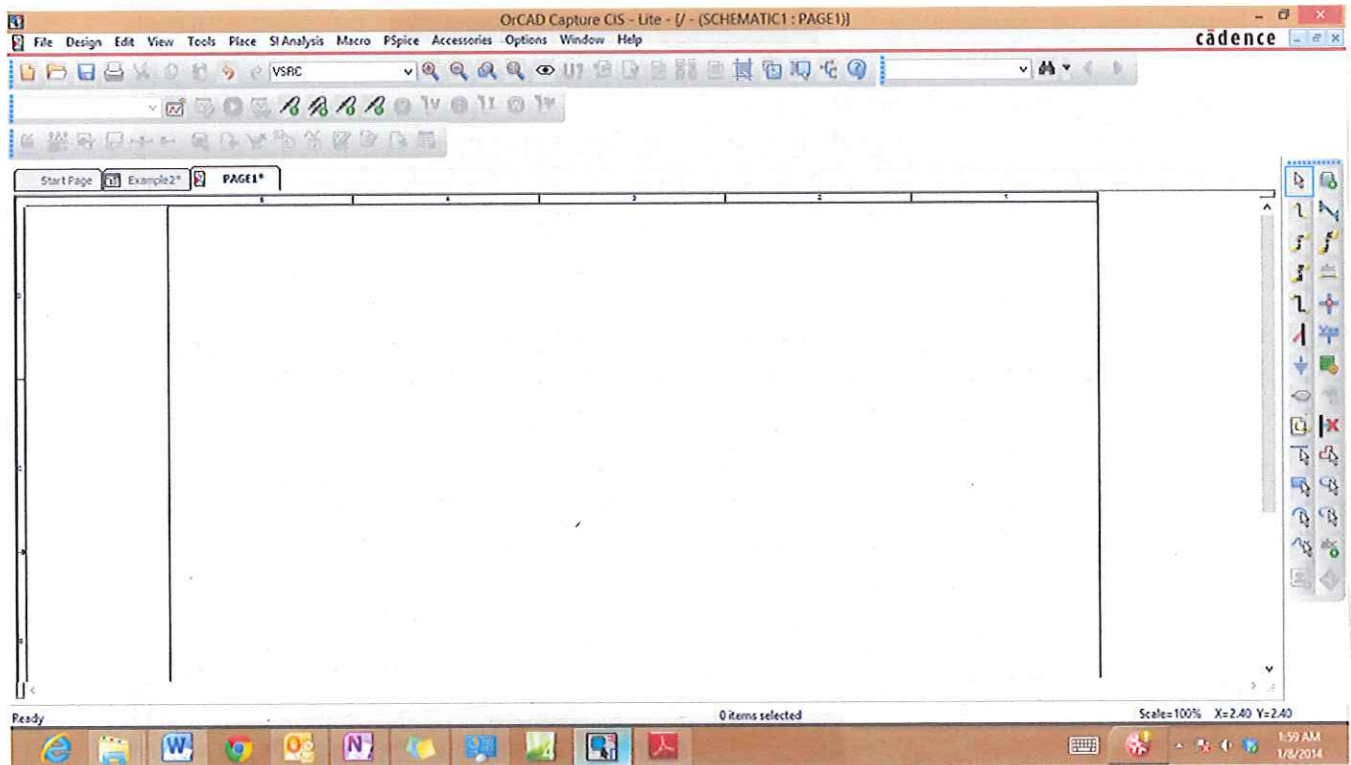

What you want to do now is to open a new project. Go to the File menu and select New and Project…. The screen below will pop-up.

New Project

Name

Create a New Project Using

- ● Analog or Mixed A/D
- ○ PC Board Wizard
- ○ Programmable Logic Wizard
- ○ Schematic

OK

Cancel

Help

Tip for New Users

Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.

Location

C:\PROGRAM FILES\ORCADLITE\CAPTURE\SAMPLES\tutc

Browse...

Unless you are just drawing a schematic for a presentation or for your reference, you want to always select Analog or Mixed A/D. Schematic only allows you to layout the circuit while Analog or Mixed A/D allows you to layout the circuit and simulate it. When you click OK after giving the project a name (for this tutorial and others, they will be called tut#cir# where the first # is the number of the tutorial in which the circuit can be found and the second # is the number of the circuit within the tutorial itself), the following screen will come up.

Create PSpice Project

- ● Create based upon an existing project
  - hierarchical.opj
- ○ Create a blank project

OK

Browse...

Cancel

Help

To make things simpler on you, from now on just select Create a blank project and then click OK. Then you should get a screen that is the one you will be working with to layout your circuits and it should look similar (the locations of the toolbars may be different due to person preferences and/or the size of the circuits used by the user).

This screen is the schematic layout that is used by Pspice.   The toolbar on the right-hand side of the screen is the one associated with building the circuits. The description of each of the significant buttons for analog simulations is shown below

This is the Place Part Button. Its details will be described later.

This is the Place Wire Button. You will use this to connect up the parts with virtual wires.
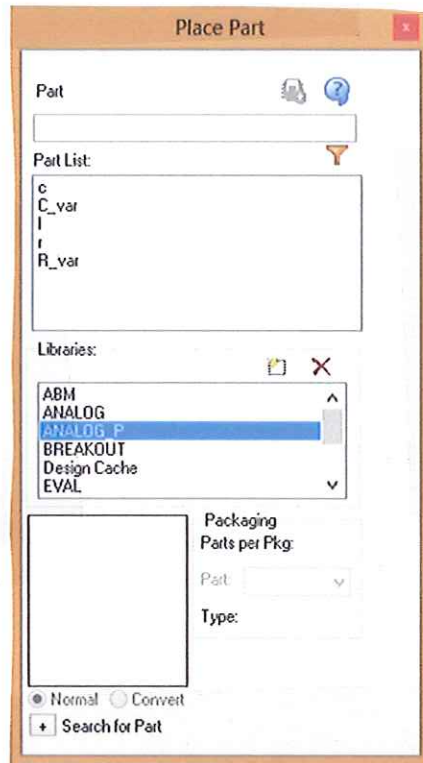
This is the Net Alias Button. It is left for a later tutorial (useful for renaming nodes when using probe)

This is the Place Ground Button. This button is very important. Forgetting to place a ground in your circuit is a common mistake among all students.
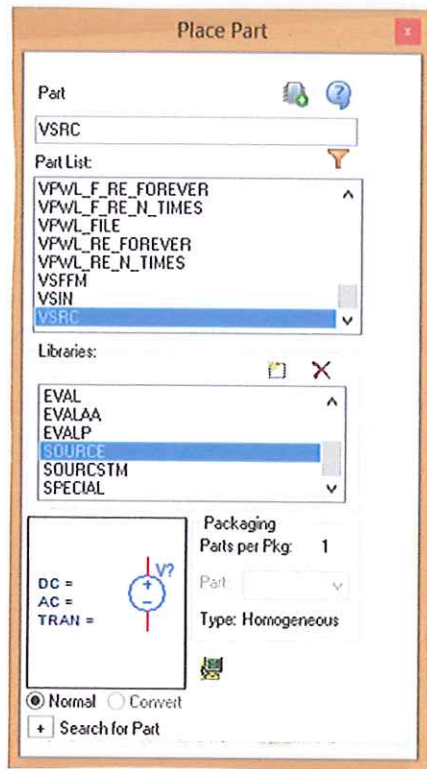
The remainder of the buttons will not be used in this tutorial. Some of them are for digital circuits and others are not relevant to our application.

Now that the basics have been covered, I will now go over a very simple voltage divider circuit. Start by clicking the Place Part Button. You should see the following
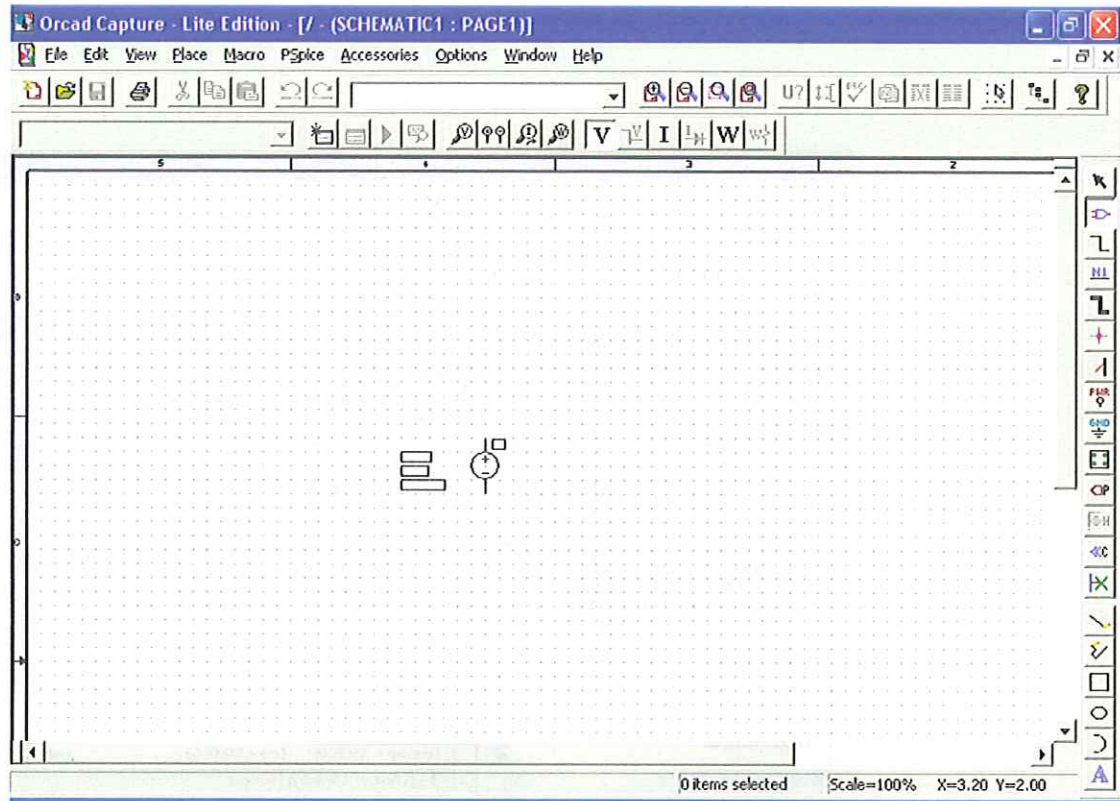
This seems confusing but it is not. Basically the Part List Window shows that you have 5 parts to choose from and which library(s) they can be found in. You have your basic 3 circuit elements (r for resistor, l for inductor, and c for capacitor) and two variable elements (discussed in lesson 6 which is pending). These circuit elements are fine and dandy but you need a source to power them.

Click on the Add Library button and add the library called **source**. This library includes all of the sources you will use in simulations. After you have added the library (the default path to it is Program Files/OrCad Lite/Capture/Library/Pspice if you couldn't find it), scroll down to the bottom of the list and select VSRC. Since we know that VSRC is in the source library, we can look at just the elements within the source library by clicking on it in the lower left window. Your screen should look like this:

Place Part

Part

VSRC

Part List:

VPWL_F_RE_FOREVER
VPWL_F_RE_N_TIMES
VPWL_FILE
VPWL_RE_FOREVER
VPWL_RE_N_TIMES
VSFFM
VSIN
VSRC

Libraries:

EVAL
EVALAA
EVALP
SOURCE
SOURCSTM
SPECIAL

Packaging
Parts per Pkg: 1

DC =
AC =
TRAN =

V?

Part

Type: Homogeneous
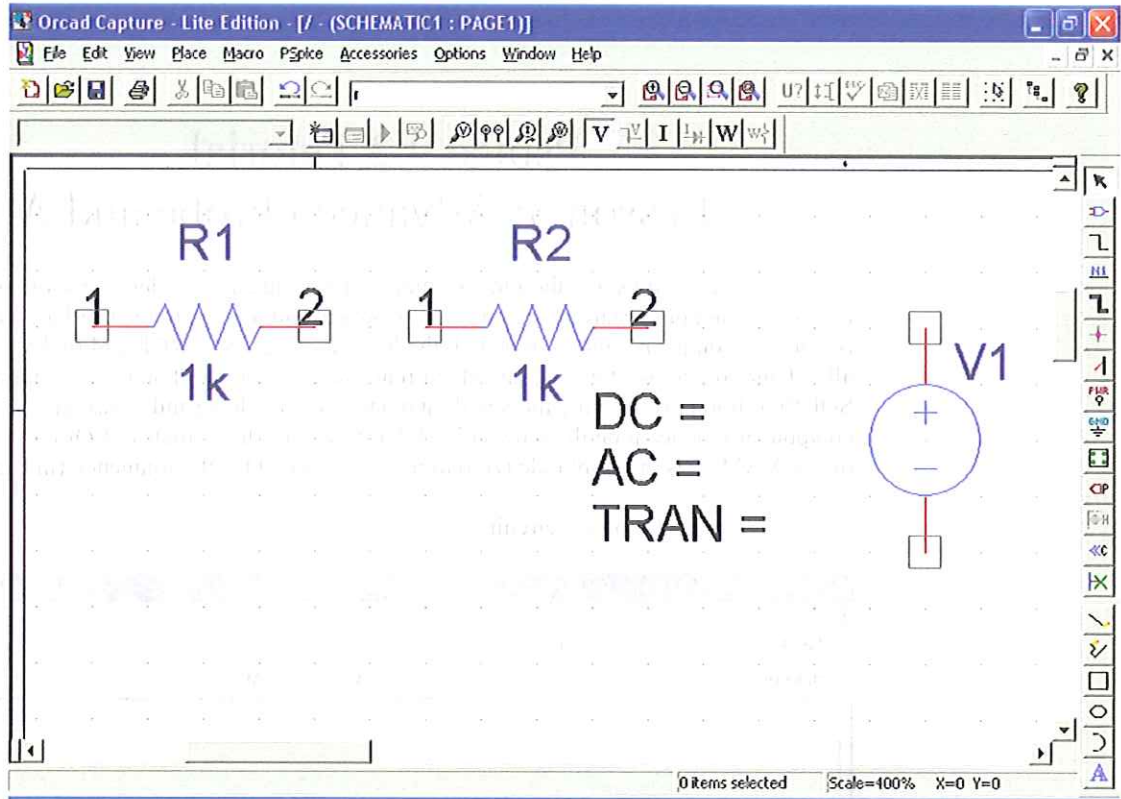
● Normal ○ Convert

+ Search for Part

You can see that VSRC looks like your standard voltage source.   When you click OK with VSRC still selected, you go back to the schematic entry screen and your mouse now has the outline of the selected component. Your screen should look similar to the one shown below.

File   Edit   View   Place   Macro   PSpice   Accessories   Options   Window   Help

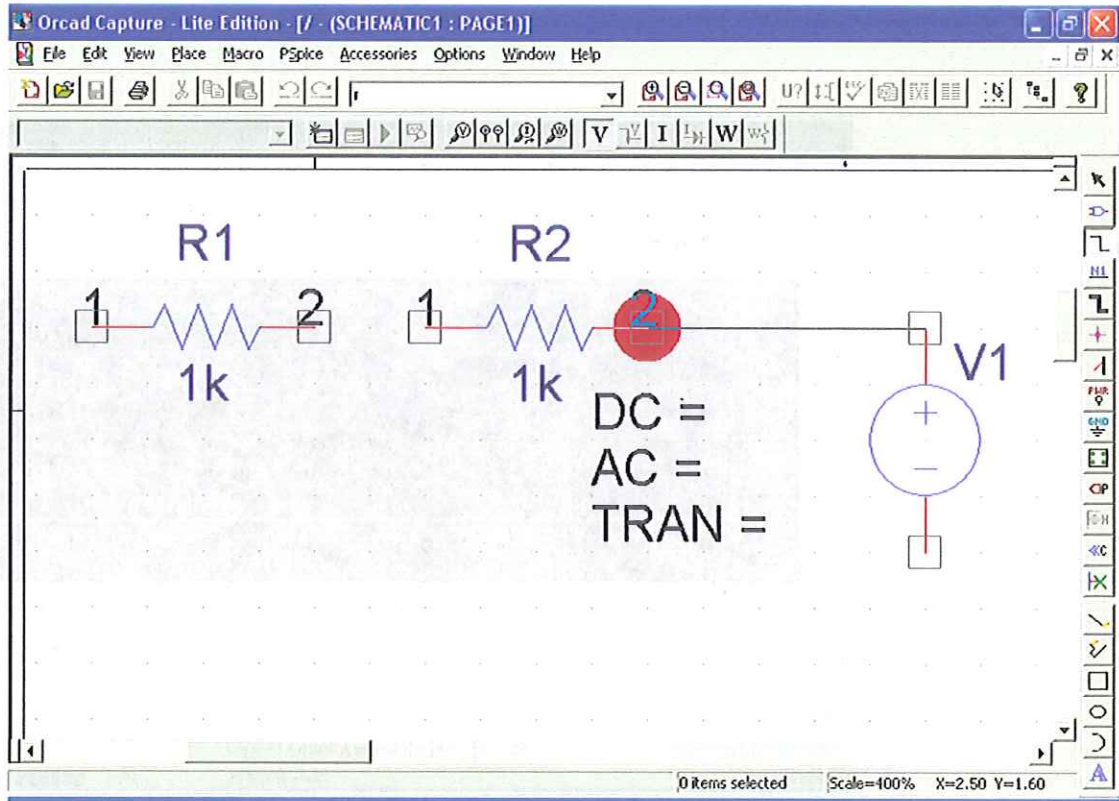0 items selected   Scale=100%   X=3.20 Y=2.00

Go ahead and place one VSRC anywhere on the schematic. You will notice that your mouse still has the outline of VSRC under it. If you had to place multiple voltage sources in the circuit, you only have to get the part once and you can place as many as you want to onto the schematic. Right-click and select End Mode. The outline of VSRC disappears and we can retrieve our second circuit element, the resistor. Remember that it is in the ANALOG_P library. Now place two resistors anywhere onto the schematic entry page.

FYI: Sometimes you have a small circuit to simulate and Pspice gives you such a large area to work with that you have a hard time seeing the circuit. Go to the View menu, select zoom, and you can zoom in and out of the schematic. The Zoom Area is the most useful because you can construct your circuit and then zoom in to print out a nice picture for your professor.

Right now we have two resistors and a voltage source. Your screen should look like this (I zoomed in so you could see and read it)
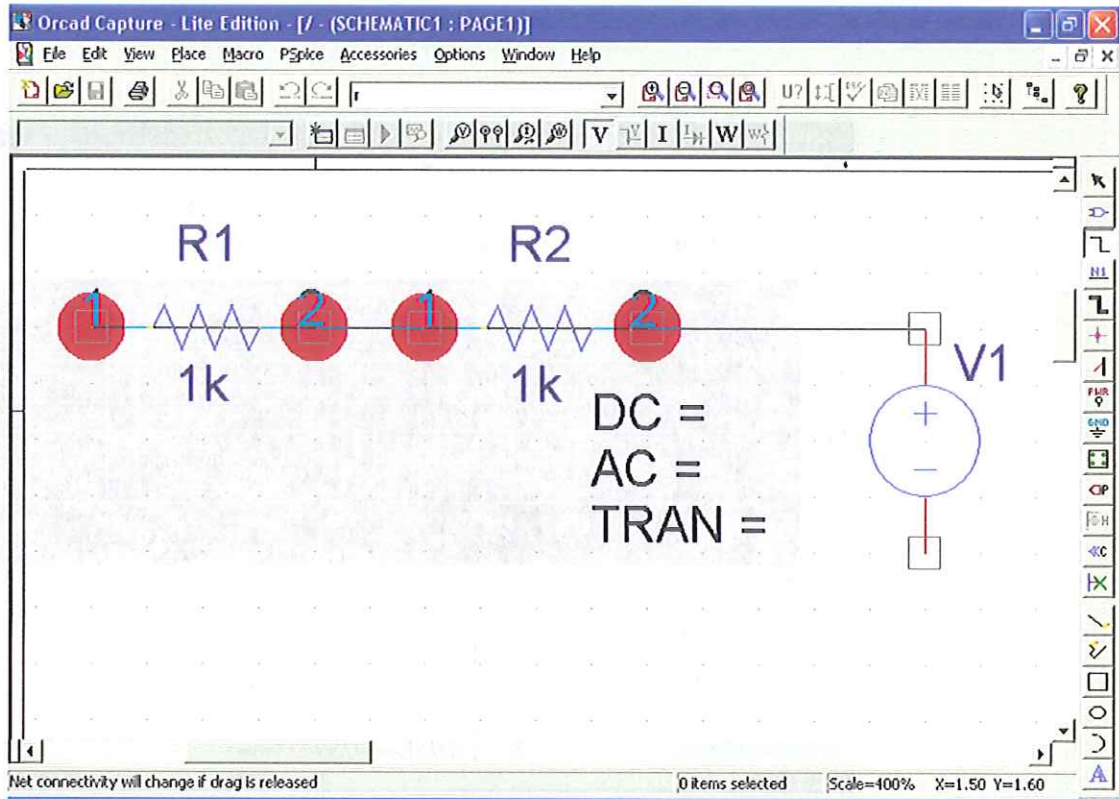
Let us work on wiring up our voltage divider circuit. Click on the Place Wire button and move your cursor over the circuit elements. The cursor has changed from an arrow to a crosshairs. Each of the circuit elements that we are using has two terminals. These are where we can connect it to other elements to make our circuit. The terminals are the gray squares. Click on the gray square on the top of V1 (the voltage source) and then move your mouse to pin number 2 of R2 or the closest terminal of your circuit. Your screen should look like:

R1

1 —/\/\/\— 2

1k

R2

1 —/\/\/\— 2

1k

DC =

AC =

TRAN =

V1

+

–

0 items selected   Scale=100%   X=2.50 Y=1.60

That giant red dot tells you that an electrical connection will be made. Go ahead and click on the red dot. The wire now connects the positive terminal of the voltage source to the resistor.
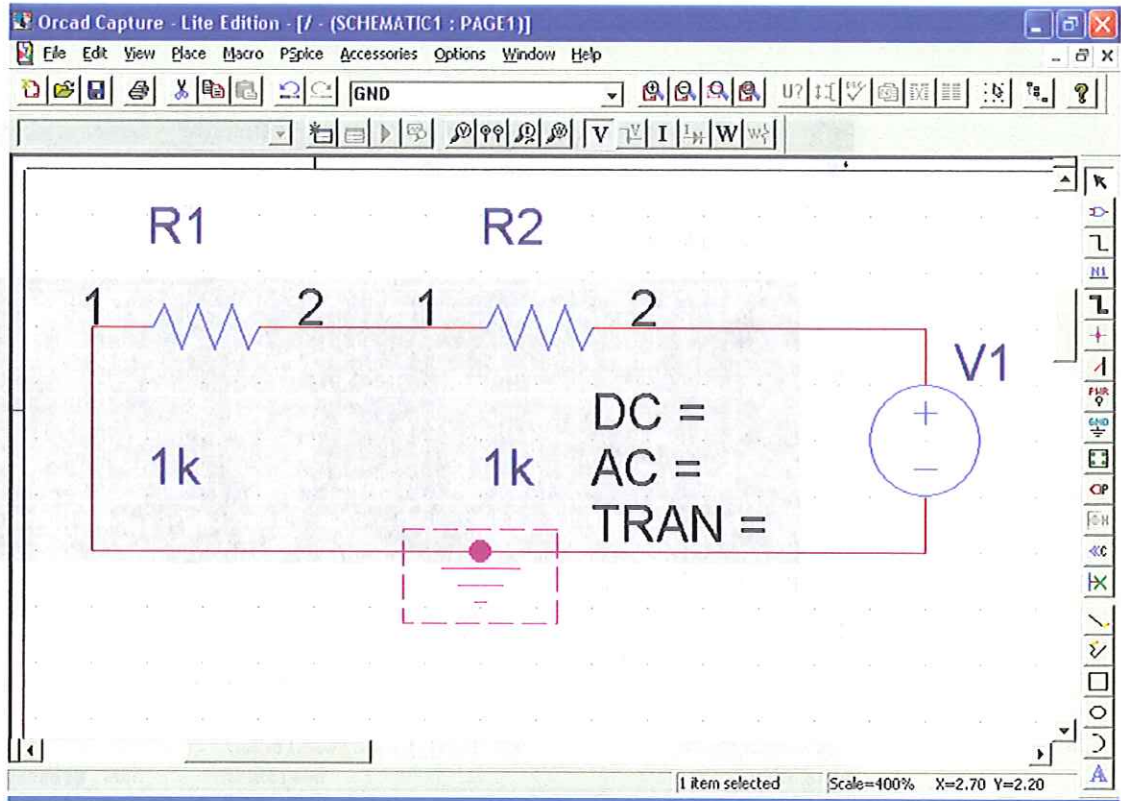
At this point in time, I want to explain a common mistake that may (and probably will) occur in your many Pspice simulations. The problem is that when you are laying out wires, you may "accidentally" come into contact with another terminal. In our example above if I continue to move the mouse (before clicking on the red dot) to pin 1 of R1 (the rightmost terminal). The screen looks like this
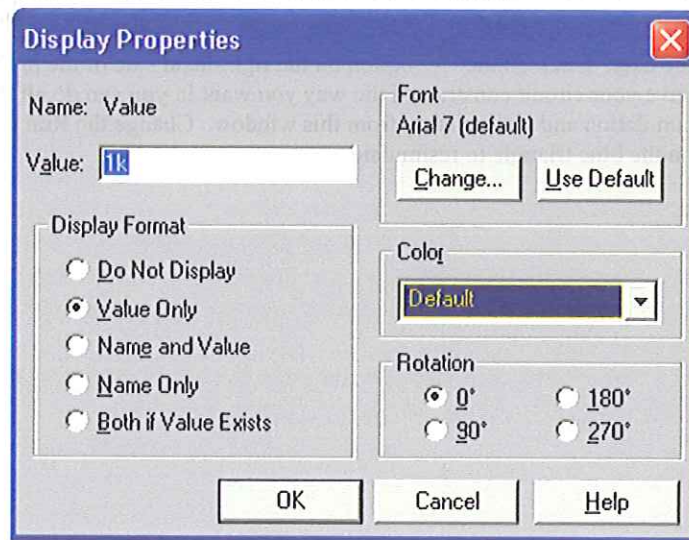
As you can see, if I were to click on terminal 1 of R1, the wire would connect all four pins of the two resistors, shorting them both out and (if this was real life) causing serious damage to the power supply. The cursor gives you a warning if a wire is going to connect to multiple terminals. You will see a yellow triangle with an exclamation point in it when this happens.

Finish wiring up the circuit by connecting terminal 2 of R1 to terminal 1 of R2 and terminal 1 of R1 to the negative terminal of V1. You have now made your first circuit. After you are done wiring, right-click and select End Wire to stop wiring. Now what you need is a ground. Forgetting a ground is a common mistake which will lead to floating node errors. To put a ground in our circuit, click on the Place Ground button. A screen comes up giving you a list of grounds to choose from. Select GND and place the terminal of it somewhere along the wire that connects the terminal 1 of R1 to the negative terminal of V1.

A purple dot should've appeared. This dot just lets you know that there are three or more elements connected to a wire. If you have a circuit with a lot of crisscrossing wires, the only ones that are connected to each other are those with dots on them. Your circuit should look something like the one shown below.

The one thing we have to go over now is how to change the values on resistors. Double-click on the 1k below R1. The following screen will appear:
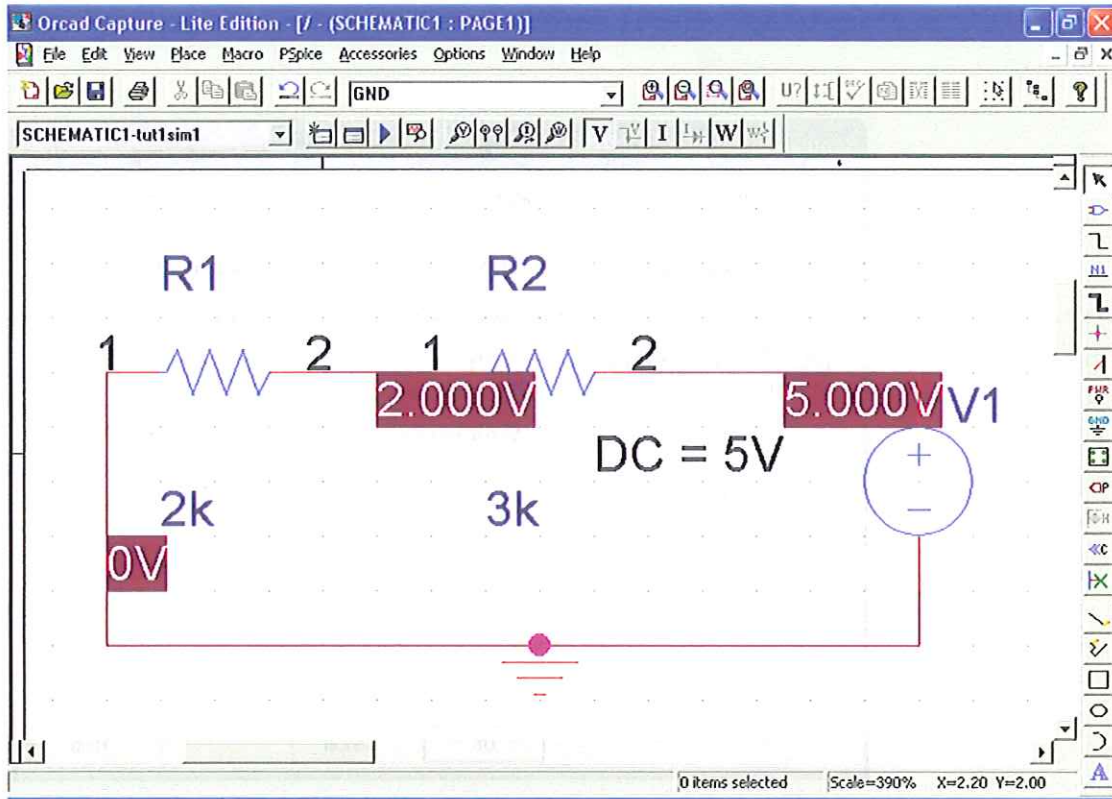

Just enter the value of the resistor in the Value box. Consult with the chart for different units of values

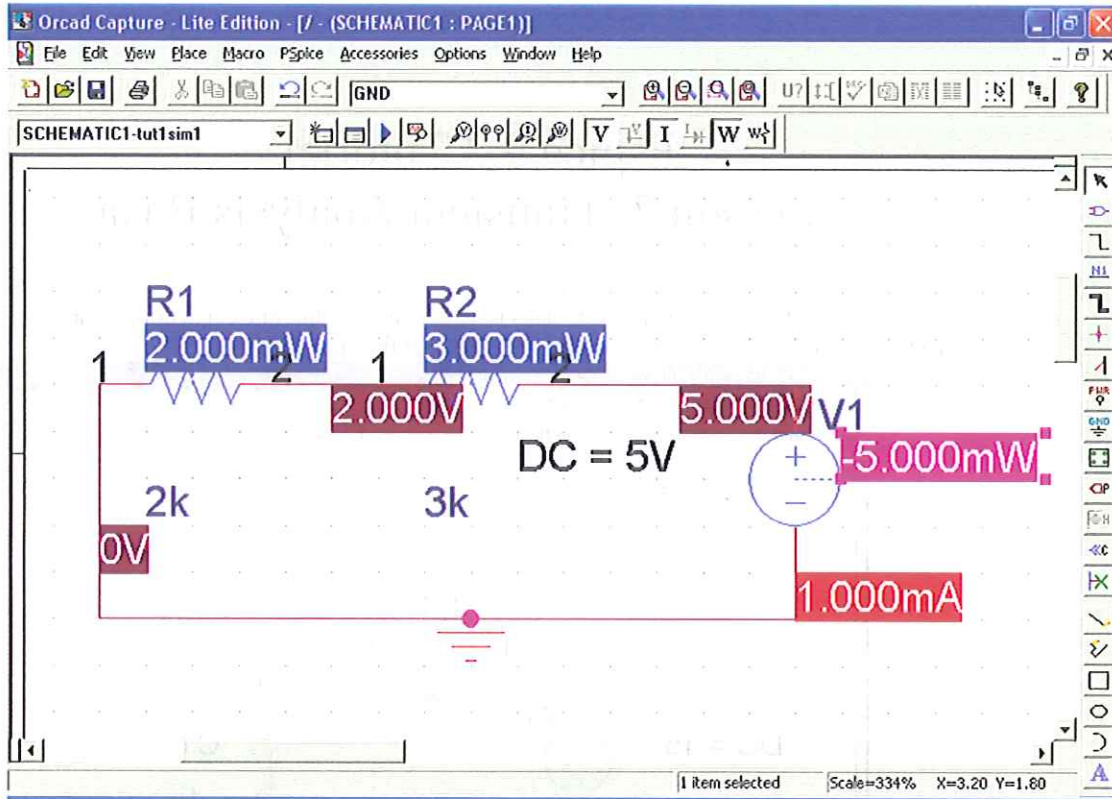| Prefix	Pspice letter(s)
Giga-	g
Mega-	Meg
Kilo-	K
Milli-	m
Micro-	u
Nano-	n
Pico-	p
Femto-	f

For example if you had a 46.7 micro-ohm resistor, you would type "46.7u" in the value box. The one you have to watch out for is mega-. If you just type in the letter m, you would put a milliohm resistor in the circuit which would greatly affect the accuracy of your circuit. These letters work on anything including voltages, currents, resistances, capacitance, inductances, and wattages.

Now, double-click on the "DC =". The same screen pops up as before. Set it to 5. Pspice will automatically recognize it as a voltage value so there is no need to put the V after it but if you want it there it doesn't hurt anything and it may look better in the circuit. Double-click on the AC= and TRAN= and set both to zero.

Now that we have assembled our circuit and made it look nice, we can actually try to simulate it now. Go to the Pspice menu and select (about the only thing you can select) New Simulation Profile. You get a window with a lot of tabs on top. Select Analysis (if it isn't selected already) and change the Analysis Type to Bias Point. (Some of the other ones will be discussed in later lessons. Now click OK. We have now told Pspice what it will do to simulate the circuit. We now have to run the simulation. ▶ is the button that runs Pspice. Click it. A new window opens up. In the lower-left corner of that window you may or may not have gotten any errors. If the last line says Simulation Complete, you had no errors. Close this window and go back to OrCad Capture. Your circuit should now have voltages on it like the circuit below.

This is fine and dandy when you want voltages displayed but what if you want power dissipated, power supplied or current. Well, you can display those on the schematic too. You will see six buttons, which look like a wheel of fortune puzzle, with the letters V, I, and W on them like [V I W]. If you click on the I button and the W button, you will display all currents and power dissipated respectively. You should now have a big mess on your hands. Some of the numbers are overlapping and some contain useless information (such as the fact that ground is at a 0V potential). Lets remove the redundant information. Click on the resistor under R1, it should be selected along with one of the current numbers and one of the wattage numbers. Since the current is the same through all three elements in this circuit, we can safely eliminate the current into the device. While the resistor is still selected, click on the button between the I and the W. The current for R1 should have disappeared. Do the same thing for the other resistor, leaving only the source current. You should have a display close to this

File Edit View Place Macro PSpice Accessories Options Window Help

GND

SCHEMATIC1-tut1sim1

R1
2.000mW
1   2
2.000V
2k
0V

R2
3.000mW
1   2
5.000V V1
DC = 5V   -5.000mW
3k
1.000mA

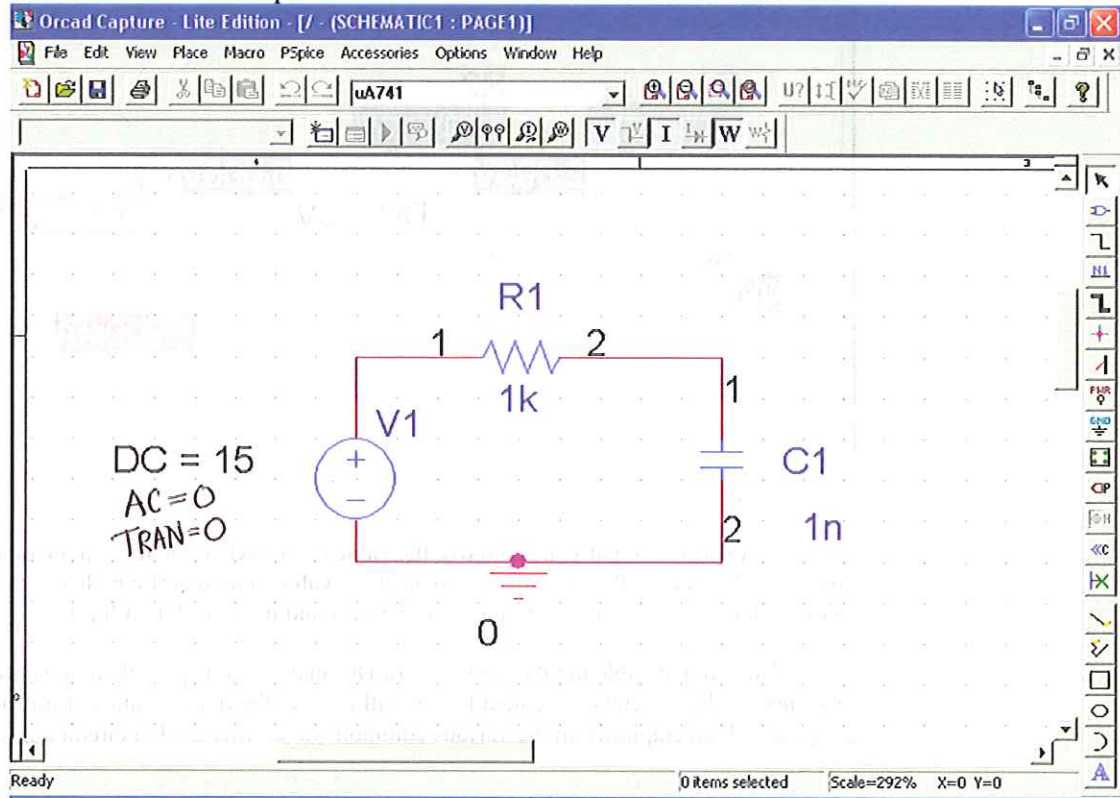1 item selected   Scale=334%   X=3.20 Y=1.80

One thing to notice is that you can move the values to anywhere on the schematic that you want. Capture will automatically connect the value to its specific node or device by a dotted line (as can be seen with the source power and its short dotted line).

You are now able to take the basics of DC analysis and apply them to simple time-independent circuits. The next lesson will go over the fundamentals of transient analysis with an emphasis on the various common waveforms used in circuit analysis.
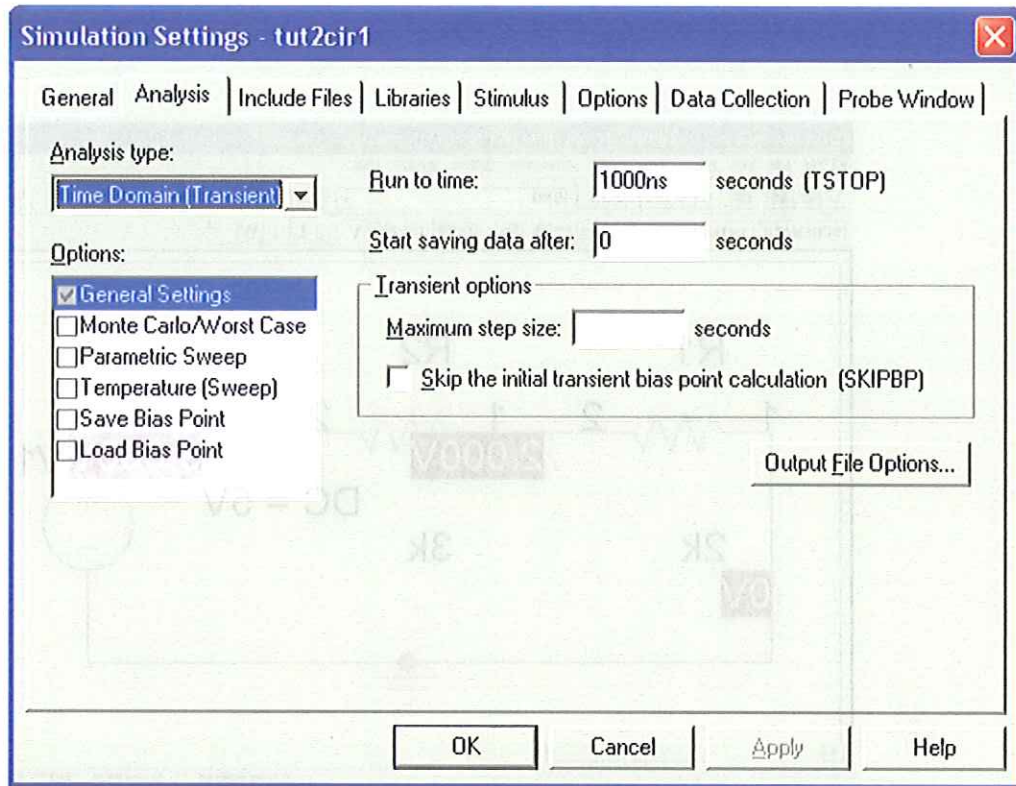
Pspice 9.2 Tutorial
Lesson 2: Transient Analysis Basics

Create a new project. This time get one resistor, one capacitor, and a voltage source. Wire them up such that it resembles the circuit below.

To get yours to look like the one in the above picture, you will have to rotate the capacitor. You can either press Ctrl+R or right-click (when placing the capacitor or when the capacitor you want to rotate is selected) and select rotate. It will only rotate 90 degrees counterclockwise. Keep in mind that some devices, such as the resistor, capacitor, and inductor, make no difference in the circuit if their pins are switched around.

Now that we have our circuit assembled, time to simulate. Create a new simulation profile and call it what ever you feel like. Since we are dealing with a charging capacitor, we need to do a Time Domain (Transient) analysis.

Simulation Settings - tut2cir1

General | Analysis | Include Files | Libraries | Stimulus | Options | Data Collection | Probe Window |

Analysis type:
Time Domain (Transient) ▼

Run to time: 1000ns   seconds (TSTOP)

Start saving data after: 0   seconds

Options:
☑ General Settings
☐ Monte Carlo/Worst Case
☐ Parametric Sweep
☐ Temperature (Sweep)
☐ Save Bias Point
☐ Load Bias Point

Transient options
Maximum step size: [   ]   seconds
☐ Skip the initial transient bias point calculation (SKIPBP)

Output File Options...
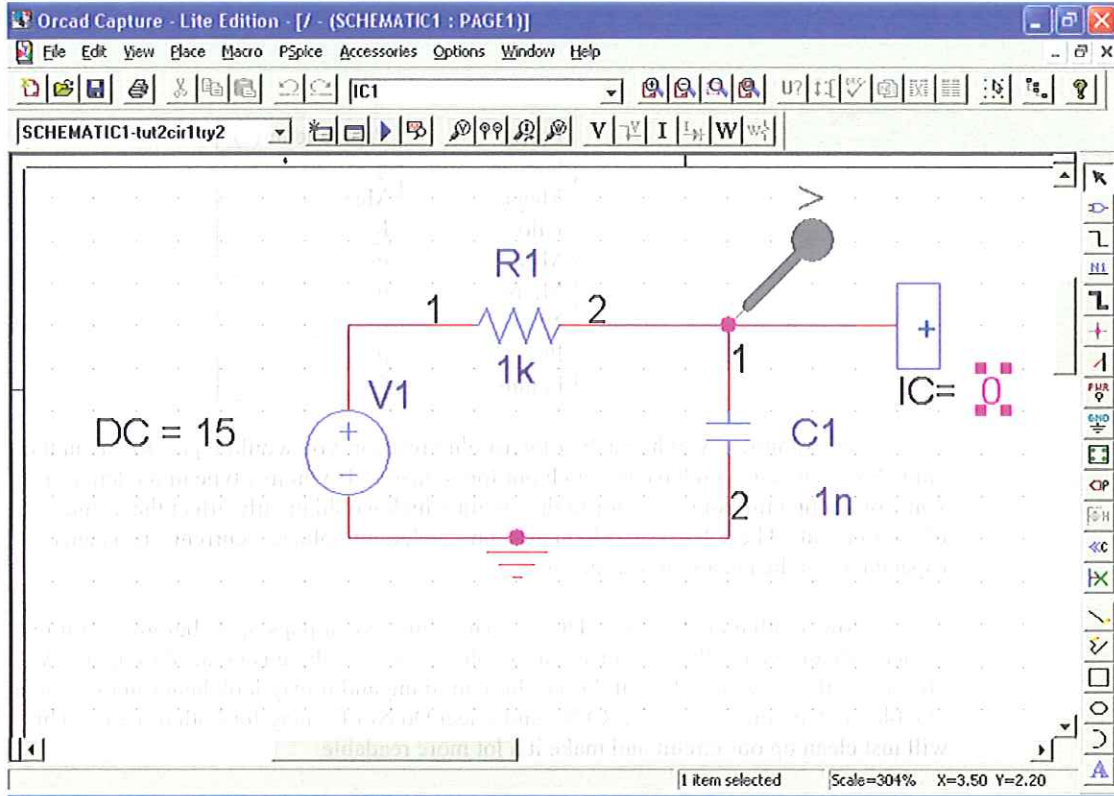
OK   Cancel   Apply   Help

Since we do not care about any of the other options at this time, I will briefly go over the information on this screen. Run to Time tells PSpice how long to simulate the circuit for. Start saving data after is if you want PSpice to ignore start-up variations in the waveform. Maximum step size is very useful. What probe does is gets a bunch of data points, displays them, and connects the dots. You can force PSpice to put more points in to get a more accurate curve. Use the defaults for now because we can always change them later. Click OK
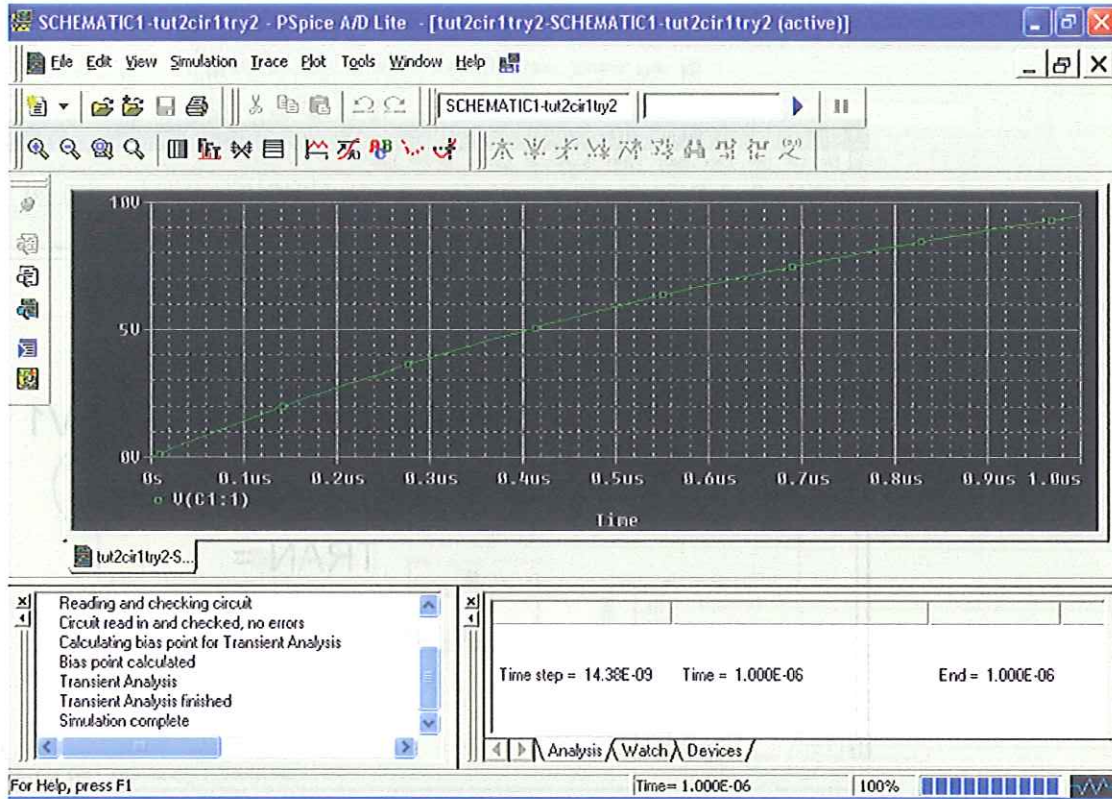
We should now tell PSpice that we want to see the voltage across the capacitor. Go to PSpice -> Markers -> Voltage Level (we would choose voltage differential but one end of the capacitor is connected to ground). You should see a thermometer with an afro under your mouse. Place one on the wire between the capacitor and the resistor. If it obstructs any of the numbers, you can rotate it like circuit elements.

Go to place part

Lets add an initial condition to the capacitor to make sure we see an exponential curve. Type in IC1 into the only box you can type into. You should get a rectangle with one legs. Rotate it once and place it to the right of the capacitor. This is the object used to set initial conditions at a point with respect to ground. There are two pieces of information displayed. One says "IC =" which is just a text label that you can change but will not affect the circuit at all. The other is the number 0 which, when double-clicked has the Name of Value and is the value of the initial condition. Your circuit should look like this.

File Edit View Place Macro PSpice Accessories Options Window Help

IC1

SCHEMATIC1-tut2cir1try2

R1

1 —WW— 2

1k

V1

DC = 15

1

C1

2   1n

IC= 0

1 item selected   Scale=304%   X=3.50 Y=2.20

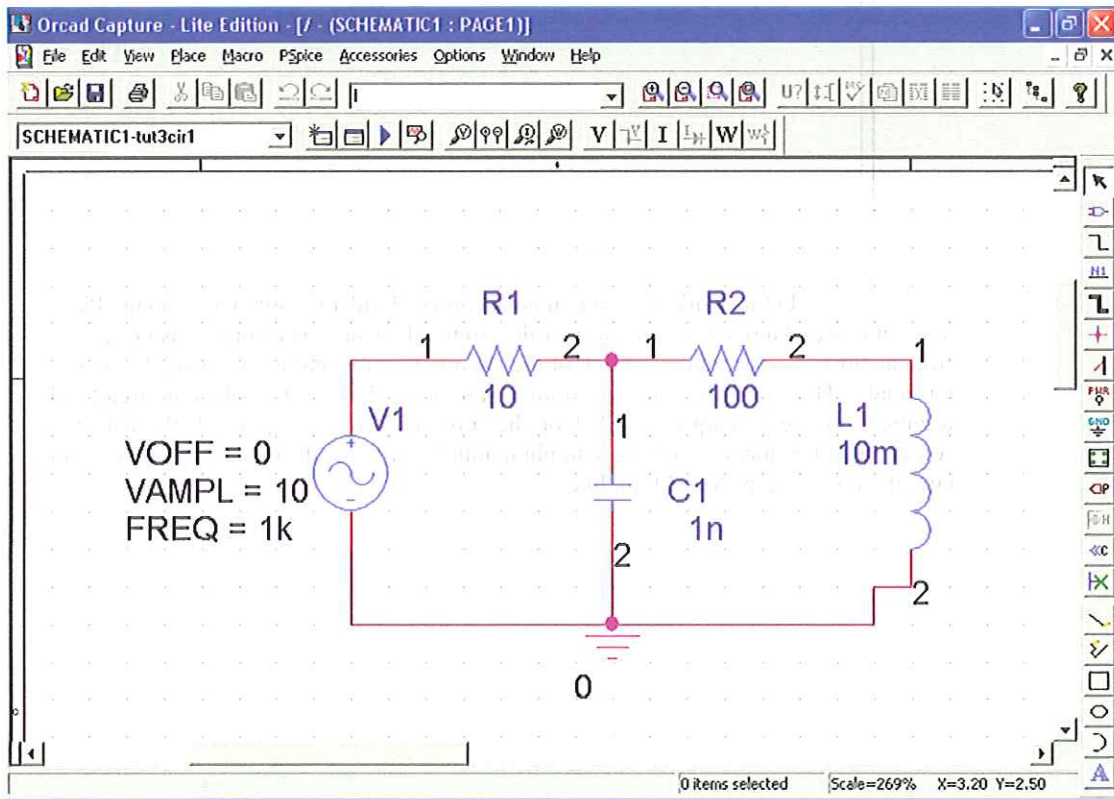Now, run the simulation (green triangle). A new window should pop up with the following graphic.

As you can see, we have not fully charged our capacitor. Go back to the schematic window and choose Edit simulation profile from the PSpice menu. Change the Run to Time to 1s, click Apply, then OK. Run the simulation again (this time it may take a bit longer). That didn't work as well. The voltage quickly jumped to 15V and stayed there for the remainder of the simulation time. Lets change it so that we can better see the data. Click on the ∧ _edit simulation_ button on the left -hand side of the probe window. Since you have your circuit constructed the way you want it, you can do all the changing of the simulation and resimulating from this window. Change the Run to time to 10us. Click on the ~~green~~ triangle to resimulate.
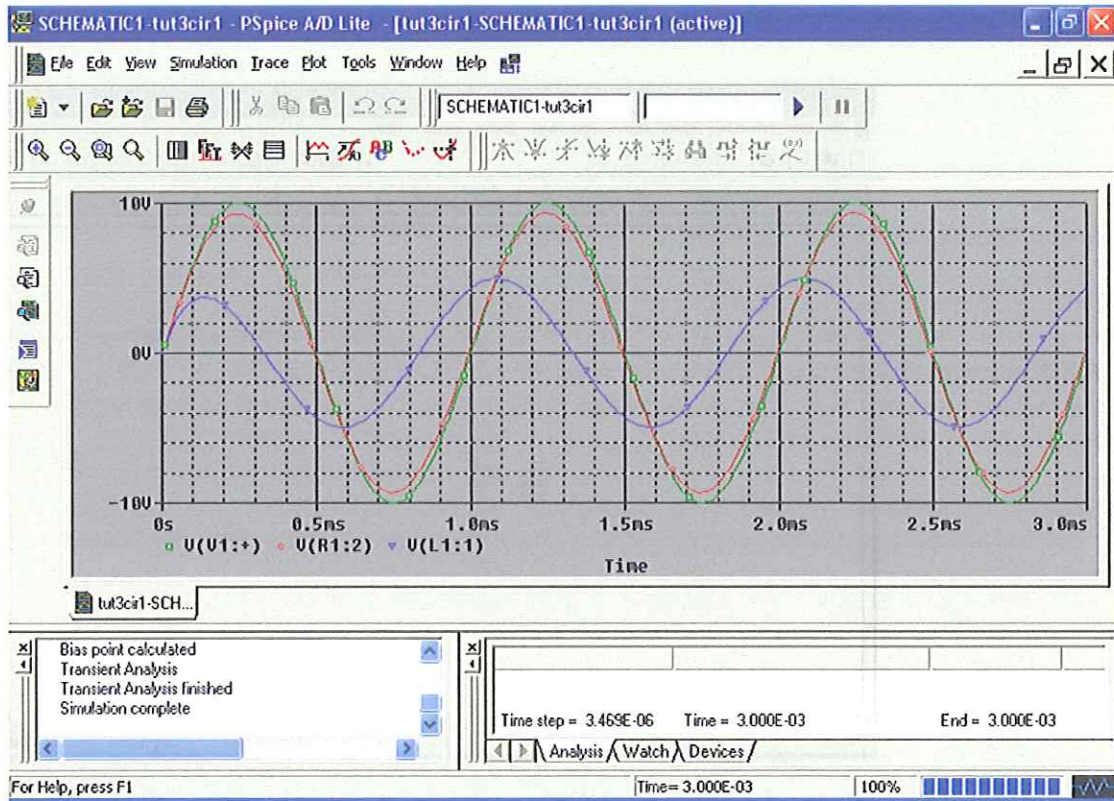
Pspice 9.2 Tutorial
Lesson 3: Advanced Probe and AC

Now that you know the fundamentals of transient analysis, lets move on to repetitive functions. Start by creating a new project and then go to the Get Part window (by clicking on the button). Among all the libraries, select SOURCE and find VSIN (FYI, all voltage sources start with V and all current sources start with I and every source in the SOURCE library (excluding the few digital ones) has a voltage and a current component.). Place it on the schematic. It has three visible attributes. VOFF is the DC offset, VAMPL is the amplitude (zero to peak), and FREQ is the frequency (in Hertz).

Create the following circuit

Set up a transient analysis with a Run To Time of 3ms, which is 3 complete cycles of the source wave. Run the simulation. Go back to the schematic and place voltage markers on pin 1 of R1, pin 1 of R2 and pin 1 of L1. They should be colorful (red, blue, and green in some order). When you go back to probe, the three voltage waveforms should be displayed (and the color of the marker corresponds to the color of the waveform). It should appear similar to this (I changed my default colors to make it easier to see. If you want to do the same, find the pspice.ini file and change the foreground and background colors)
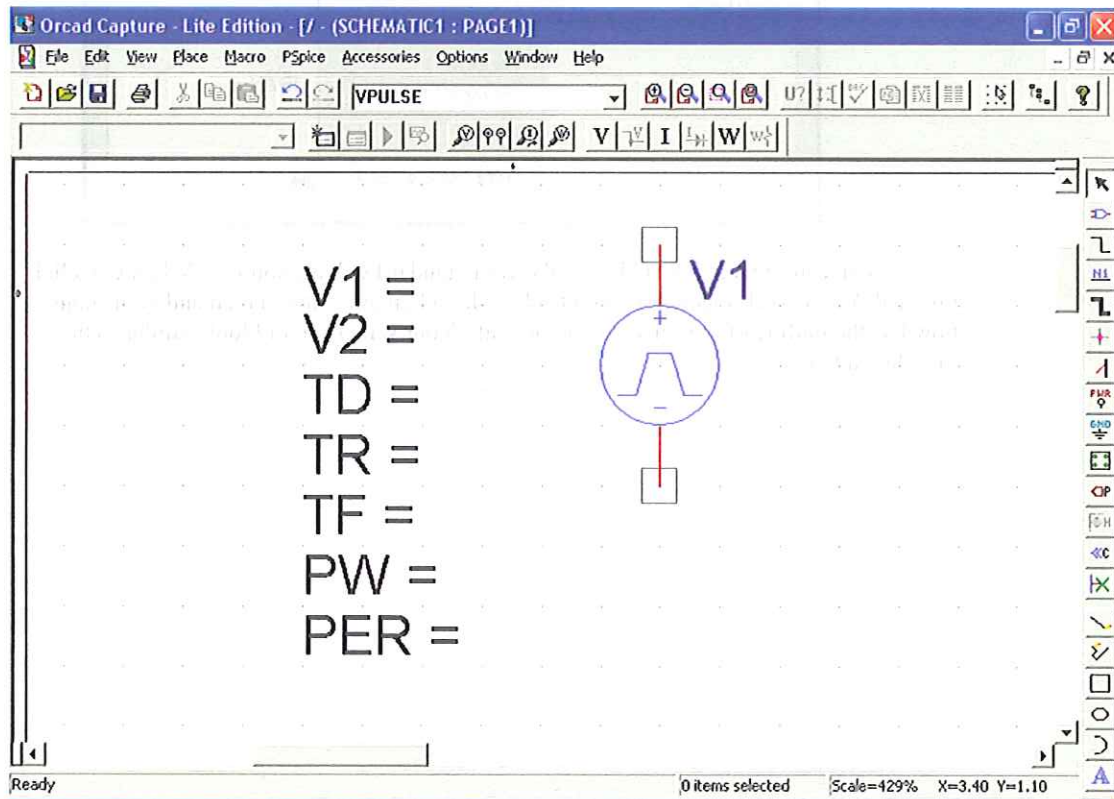
The waveforms aren't perfect. As mentioned in a previous tutorial, PSpice plots points and then connects the dots. Your waveforms may not be good representations of the data. Lets edit the simulation settings so that the waves are smoother at their peaks and troughs. Go to Edit Simulation Settings (the bottom button on the menu on the left in the probe window) and set the Maximum Step Size to 0.01m. Rerun the simulation and you should notice that your curves are much smoother.

If you wanted to measure the phase between the green and blue waveforms (the input and output respectively), you can use to cursors to do this quickly. Turn the cursors on. Both of the cursors are on the green waveform (you can tell because of the dashed box around the green symbol under the x-axis. Click on the Cursor Peak Button ⚘. The cursor is now at the first peak of the waveform. Click on it one more time so that it is now on the second peak. We need to move the other cursor to the blue waveform. RIGHT-CLICK on the little blue symbol (if you left click, you will move the cursor you moved to the peak of the green waveform). A dotted box will appear around it. This lets you know that the right-click cursor is on the blue waveform. Click on the Cursor Peak Button twice to move the right cursor along the blue waveform. To measure the phase shift, we use the formula $\Delta T * f * 360°$ (Note $f = T^{-1}$). According to the little window that shows where the cursors are, the difference in the x-coordinate between the peaks is about 160ns or 0.160ms. The frequency is 1kHz so that means our phase shift is about 57.6°.

You can put this information on the graph so that you don't have to flip between papers to find it. Click on the button that has the first three letters of the alphabet (ABC if you didn't know) which is the Text Label Button. A window should pop up that allows you to type text in it. Type in whatever kind of label you want such as "Phase Shift = 57.6", "Vc is 57.6 deg lag", or just "Phase = 57.6". Its up to you how you want to convey the information to others or even yourself. When you are done, press enter or click OK. The text you typed should appear under your mouse. You can place it wherever you want on the graph window.

Square Waves and Triangle Waves

The function generators in the lab produce three types of waveforms. You have seen how to implement the sine wave in pspice and we will now discuss the other 2 which use the same source. Get the part called VPulse. When you place it on the schematic, it looks like this.

There are seven parameters. I will discuss what each one is and then how to make the square wave and triangle wave.

Parameter	Description
V1	The initial value of the signal
V2	The pulsed value of the signal
TD	Delay Time
TR	Rise Time
TF	Fall Time
PW	Pulse Width
PER	Period

Lets say you are given a frequency F for a square and triangle wave. Here is how you would calculate all of the above parameters

Parameter	Square	Triangle
V1	Lowest Value	Lowest Value
V2	Highest Value	Highest Value
TD	Usually 0	Usually 0
TR	Very small, 1fs	$(2F)^{-1}$
TF	Very small, 1fs	$(2F)^{-1}$
PW	$(2F)^{-1}$	Very small, 1fs
PER	F^{-1}	F^{-1}

A word of warning, never make the rise time, fall time, or pulse width zero. You will get strange spikes in your waveforms. It is better to make them really really small.

Pspice 9.2 Tutorial
Lesson 5: Parametric Analysis

This is the most useful feature that PSpice has when designing. You have to be careful to follow every step or you may run into errors.

Step 1: Build your circuit and select the element you want to be changed

Step 2: Change the value of the element to a word surrounded by the squiggly brackets that look like {}.

Step 3: Place an element on the schematic called PARAM

Step 4: Double-click PARAM. A spreadsheet should come up. Create a New Column. The Name box should be the word you put in the squiggly brackets. Set the value to whatever you want. This value will only be used when not doing a parametric analysis.

Step 5: Create the simulation profile. Choose the type of simulation that you want (Transient, DC Sweep, AC Sweep, etc.) and click on the Parametric Analysis on the menu with the check-boxes. Change the sweep variable to Global Parameter and type in the word you typed inbetween the squiggly brackets. Below that is where you choose what values to give the parameter.

Step 6: Run the simulation.

As of the creation of this tutorial, it has been tested with resistor, capacitor, and inductor values and it works. To change a voltage source, you use AC Sweep or DC Sweep.