PART 1
Editing a Basic Schematic with Orcad Capture

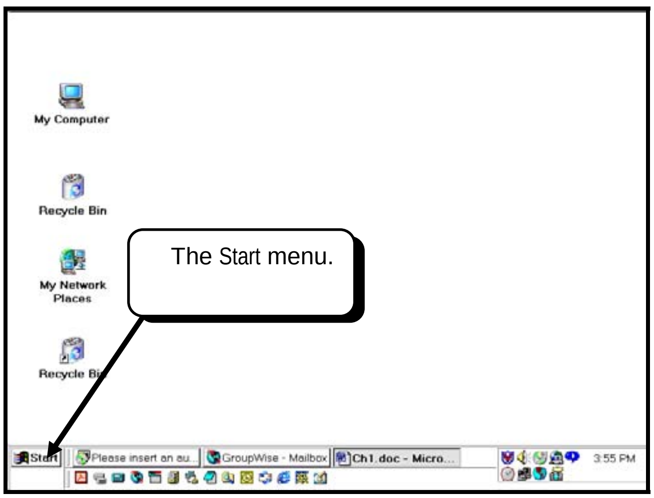
This part assumes that the Microsoft Windows operating system and the Orcad Lite Software are already installed on your computer. If the Orcad Lite software has not been installed, refer to Appendix A for installation instructions. **If you do not follow the installation instructions in Appendix A, the libraries specific to this manual will not be installed correctly.** If Windows has not been installed, refer to the Windows operating system documentation for instructions. The portions of the Orcad Lite software we will be demonstrating in this manual are:

- Capture – The schematic capture front end that has replaced MicroSim Schematics.
- PSpice – The mixed signal simulation tool. For those readers familiar with MicroSim Design Center, this tool is relatively unchanged.
- Probe – The graphical post-processor for viewing the results generated by PSpice. For those readers familiar with MicroSim Design Center, this tool is relatively unchanged as well.

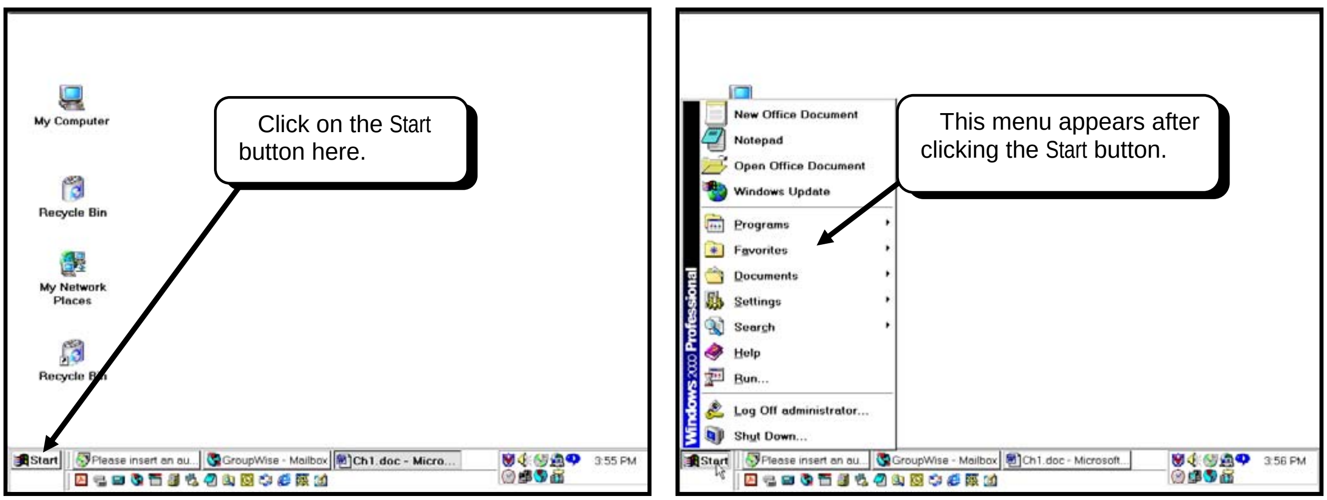
This part covers creating a circuit using Orcad Capture. Part 2 covers the basics of using Probe. The remaining parts cover the simulation of specific circuits using the suite of programs, Capture, PSpice, and Probe.

1.A. Starting Orcad Capture

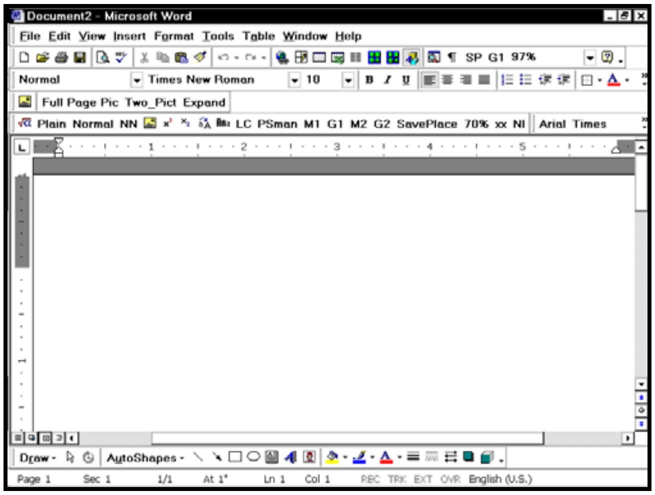
If Capture was installed properly on your computer, it can be easily started from the Windows Start menu. However, depending on how the Windows desktop is configured, the Start menu may appear differently. Usually, the Start menu is displayed at the bottom of the desktop:

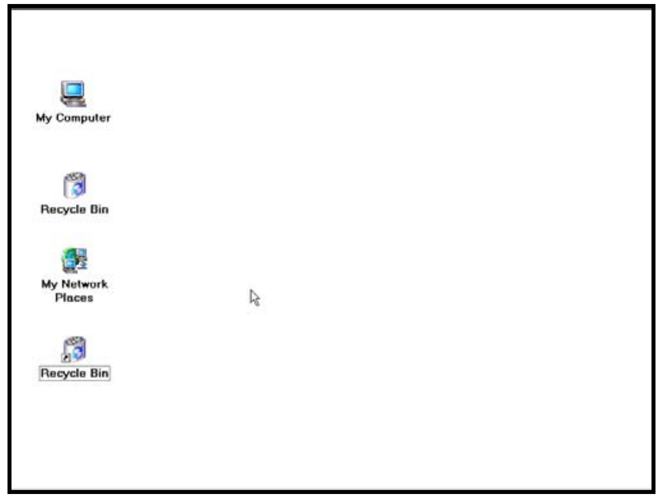


At this point, you could click the Start button  and continue:

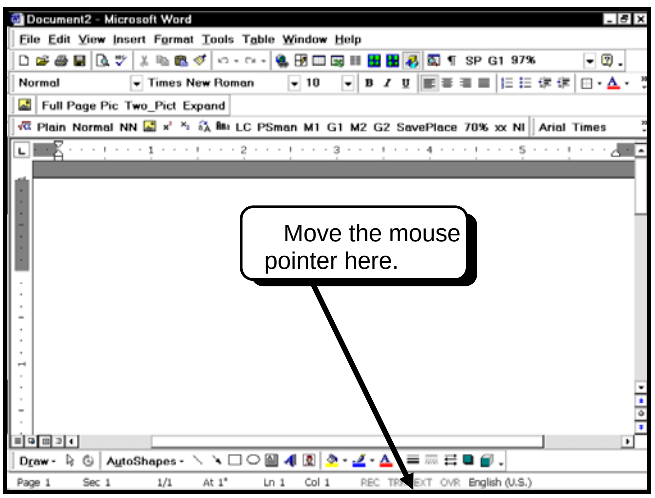


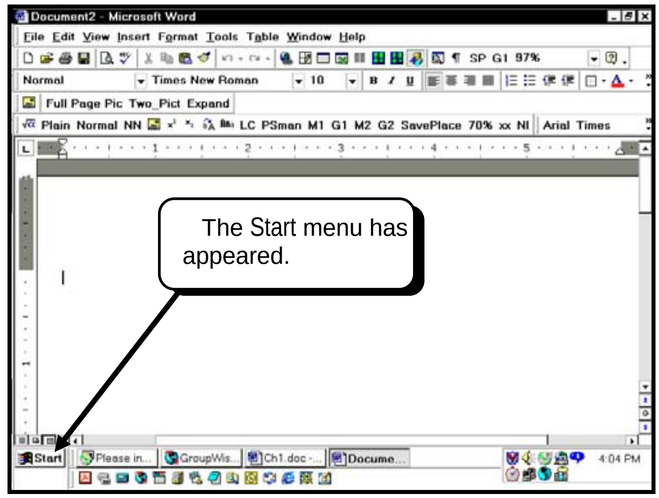
However, if the desktop appears as one of the screen captures shown below



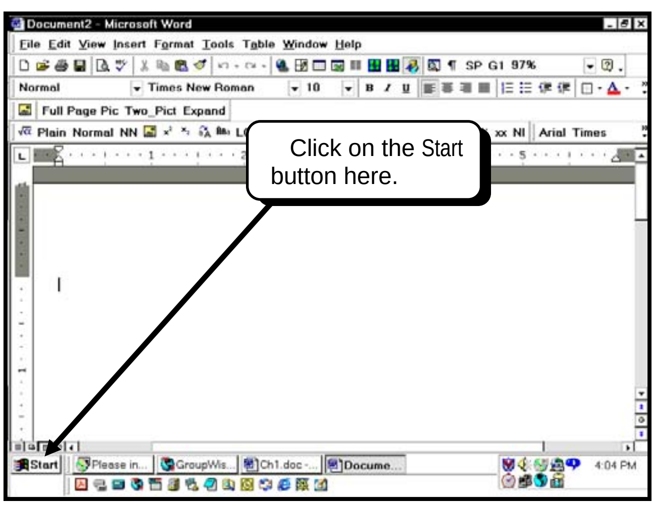


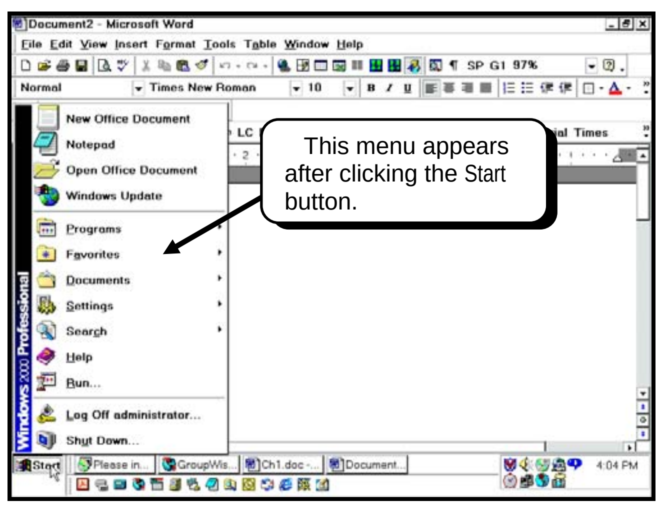
then the Start menu is hidden from view. There are two ways to make the Start menu appear. The first way is to bring the mouse pointer down to the bottom of the screen. After a moment, the Start menu should appear:



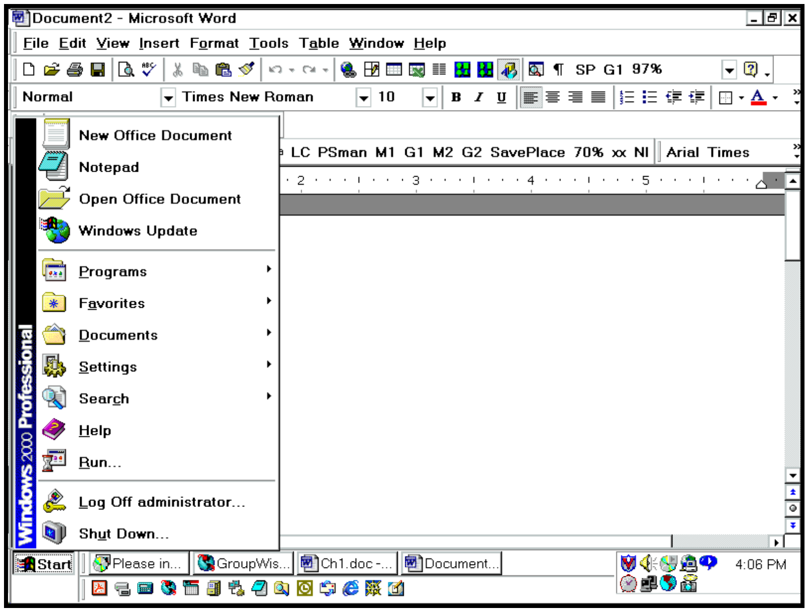


This method works if the Start menu is configured so that it is always on top and auto-hide mode is selected. At this point, you could click the Start button  and continue:

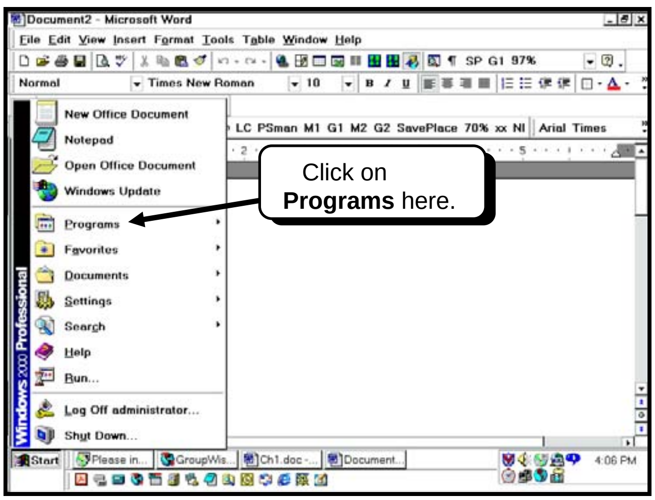


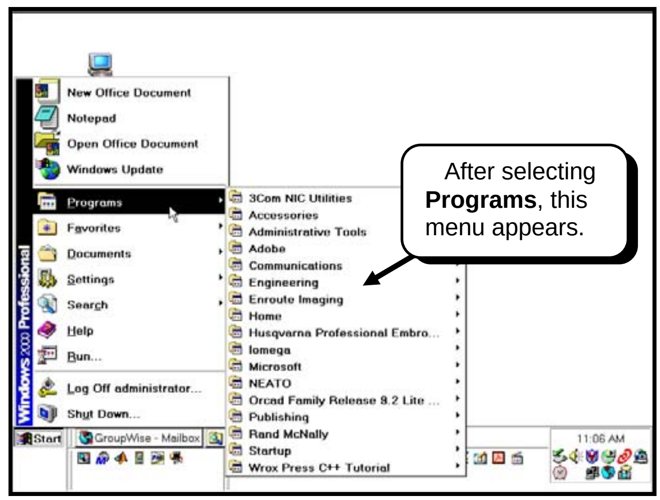


However, if you still cannot see the Start menu, you have three more options. (1) You can press the  key on the keyboard or type **CTRL-ESC**. These keys will bring the Start menu to the top and also select the Start button. (2) If you have an older keyboard and do not have the  key, you can move the mouse to the bottom of the screen and drag the Start menu up. For this example, we will press the  key:

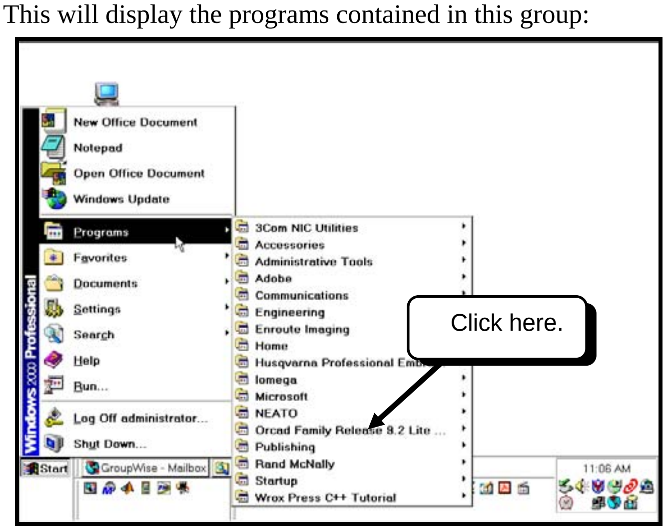


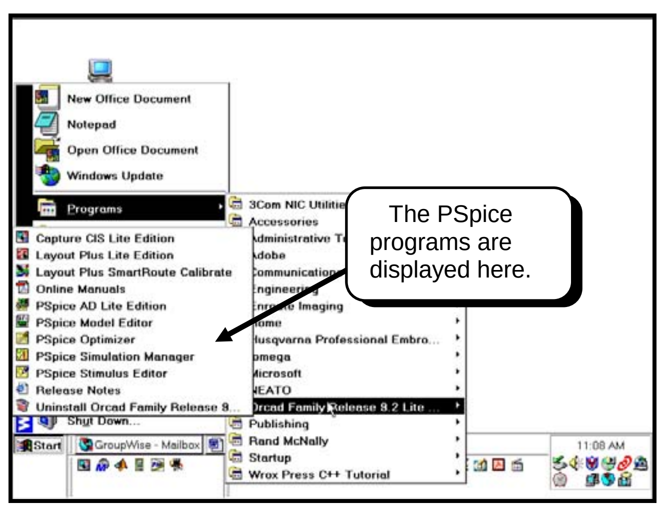
We now have the Start menu displayed. Select **Programs**:

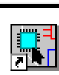


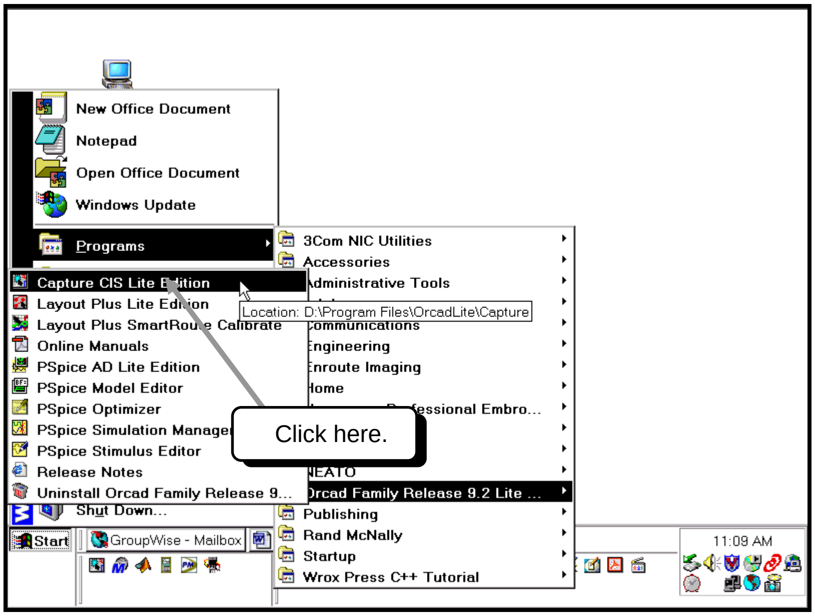


After selecting **Programs**, the programs and program groups for your computer appear. The Orcad programs are contained in the group **Orcad Family Release 9.2 Lite**. Click the ***LEFT*** mouse button on the text **Orcad Family Release 9.2 Lite**. This will display the programs contained in this group:

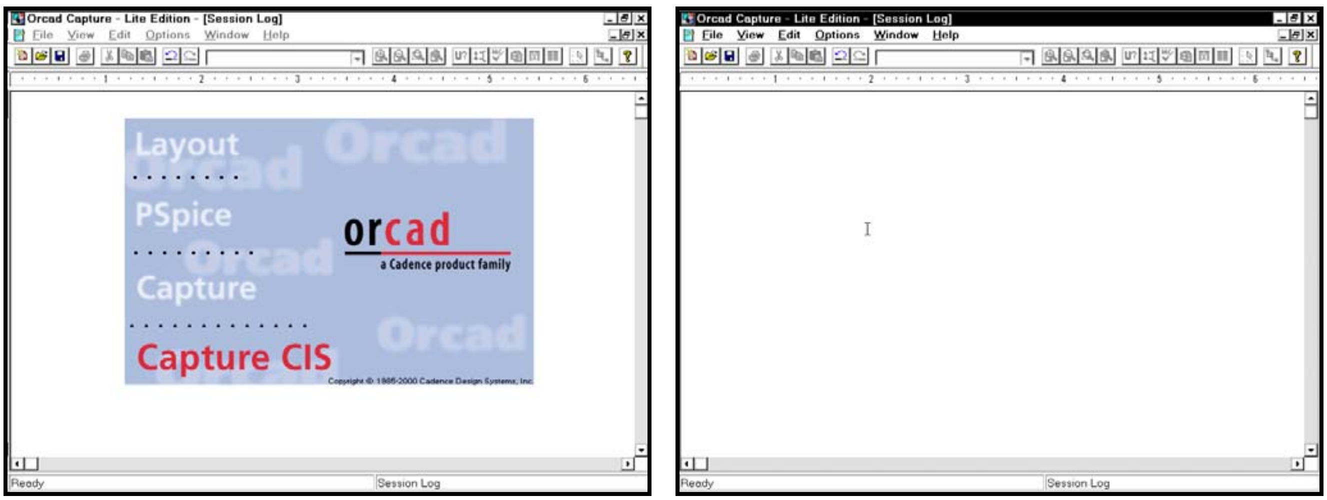




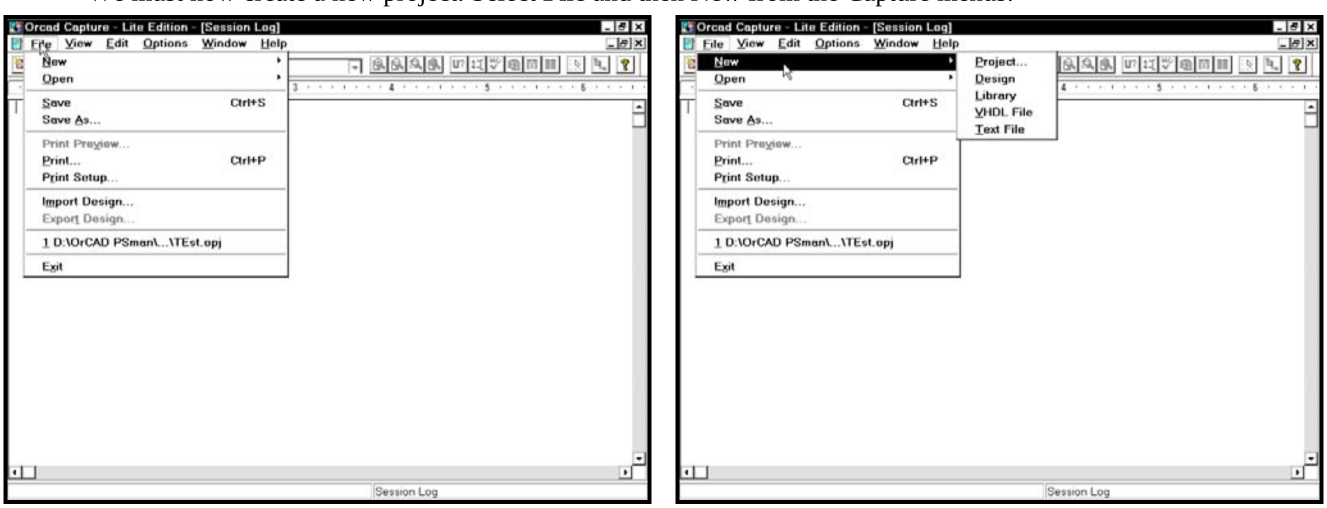
Click the ***LEFT*** mouse button on the item  **Capture CIS Lite Edition** to run the schematic capture circuit simulation package.



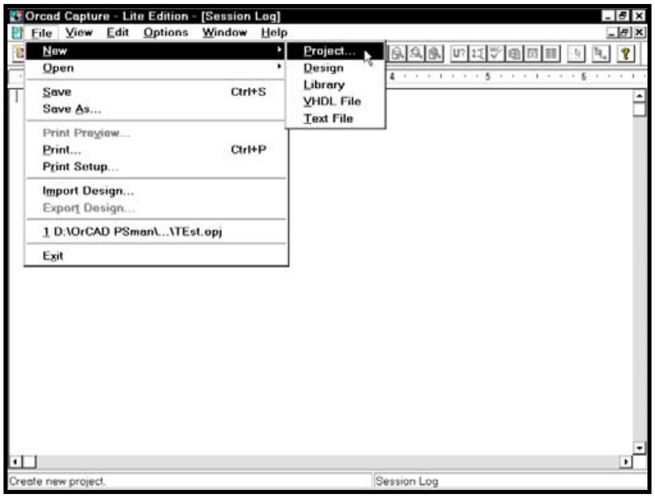
Capture should run and display an empty project:

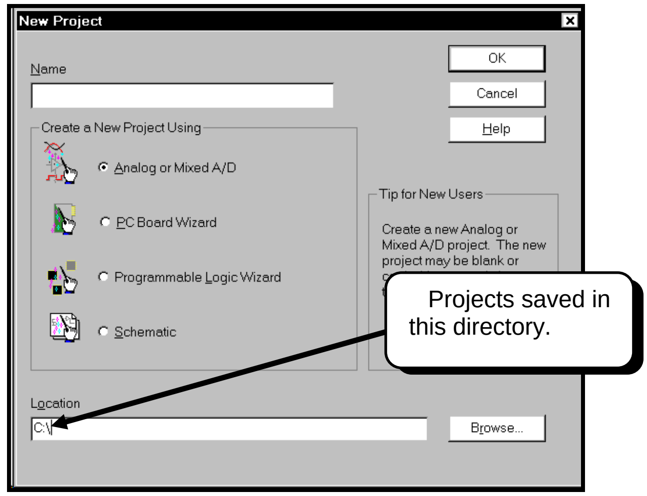


We must now create a new project. Select **File** and then **New** from the Capture menus:

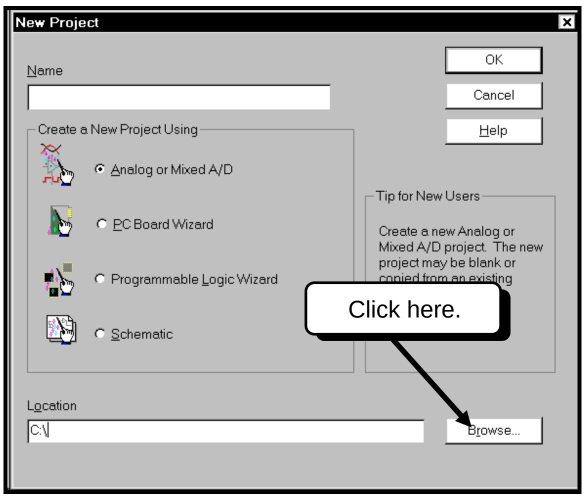


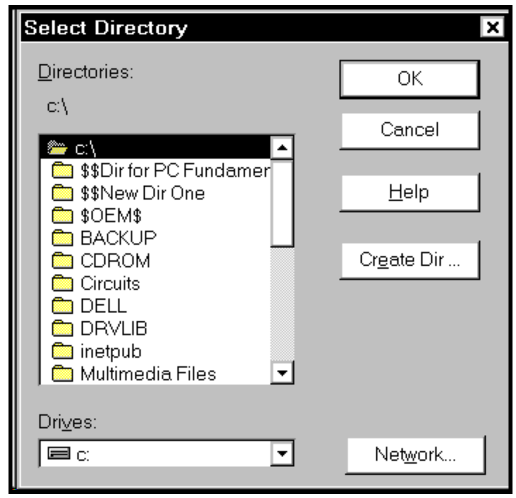
The items listed in the cascaded menu are the types of objects we can create using Capture as the front end. For this text, we are concerned only with the **Project** selection, which is used to draw schematics and simulate circuits, and the **Library** selection, which is used to create part libraries. Select **Project** to create a new project:



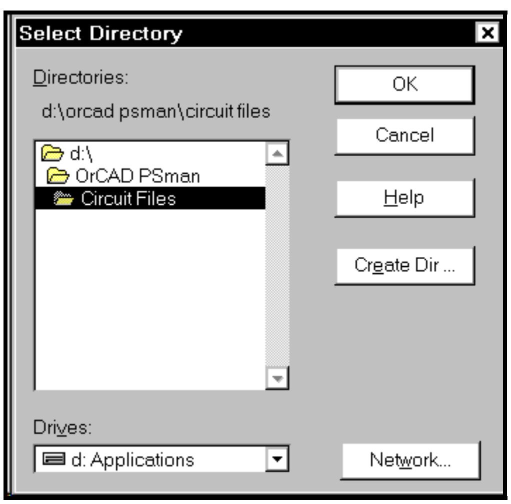


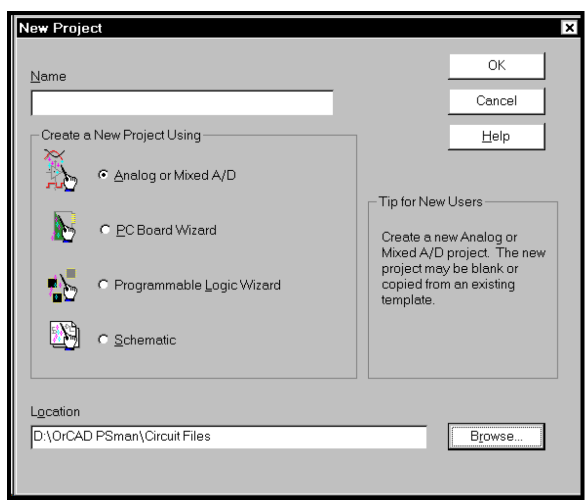
During installation, we have set the default directory for saving files to C:\. You should change this directory to one in which you would like your circuit files to be saved. For this book, I am saving all files in the directory D:\Orcad PSman\Circuit Files. Click on the Browse button to specify a different directory:



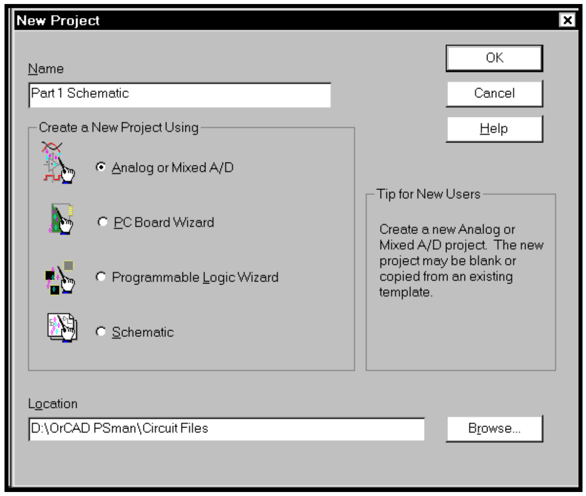


You can use this dialog box to change the current directory and to create a new directory. Select a directory in which to save your circuit files and then click the OK button:





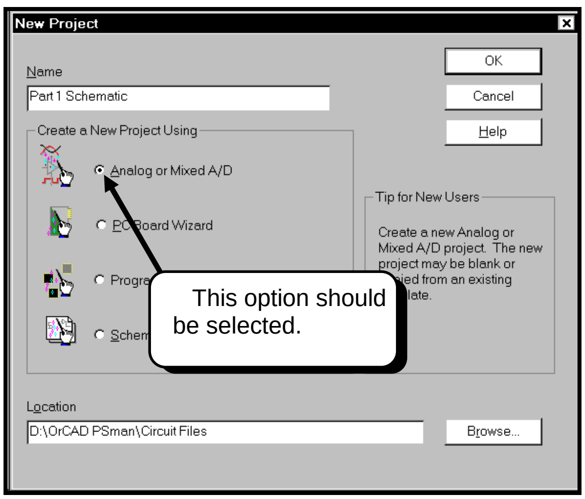
The Name field specifies the name of the project. Enter the name **`Part 1 Schematic`**:

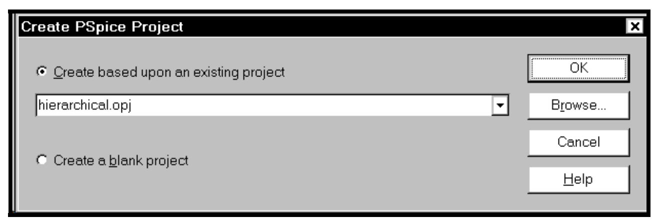


There are four types of projects you can create:

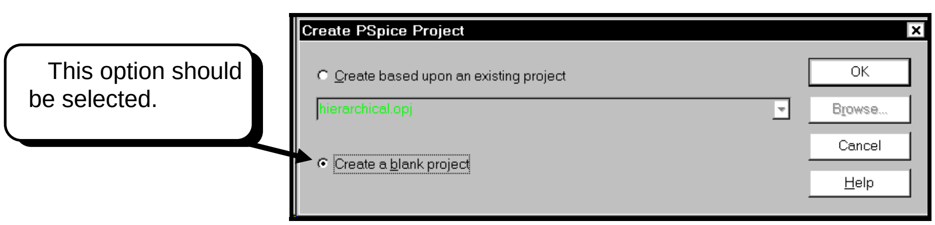
- **Analog or Mixed A/D** – Allows us to draw a circuit with Capture and then simulate the circuit with PSpice. This is the selection we will choose.
- **PC Board Wizard** – Allows us to draw a circuit with Capture and then create a PC board layout with Layout Plus.
- **Programmable Logic Wizard** – Allows us to use Capture to design a CPLD or an FPGA.
- **Schematic** – Allows us to create a schematic using Capture.

To simulate circuits, select option Analog or Mixed A/D and click the OK button:

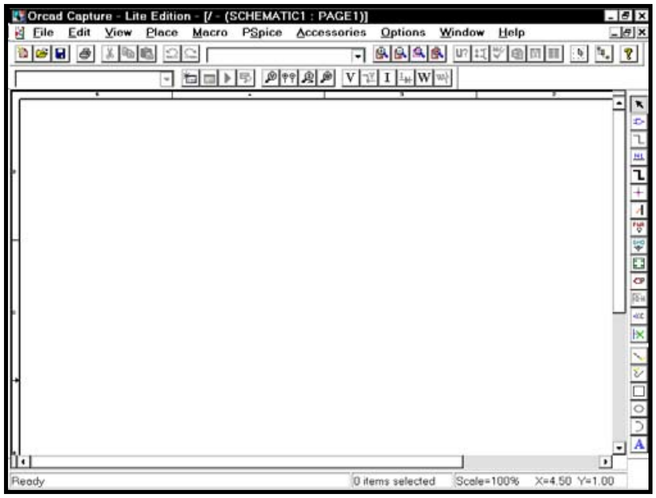




This dialog box allows us to select a project template. A hierarchical design is one with many pages, and blocks that connect one page to another. We will choose to start with a blank project:

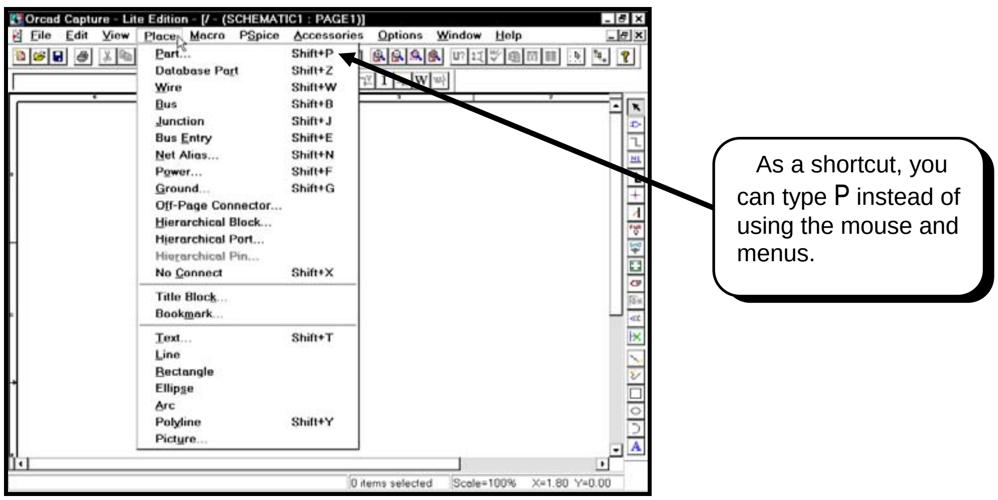


Click the OK button. You will be presented with an empty schematic page:

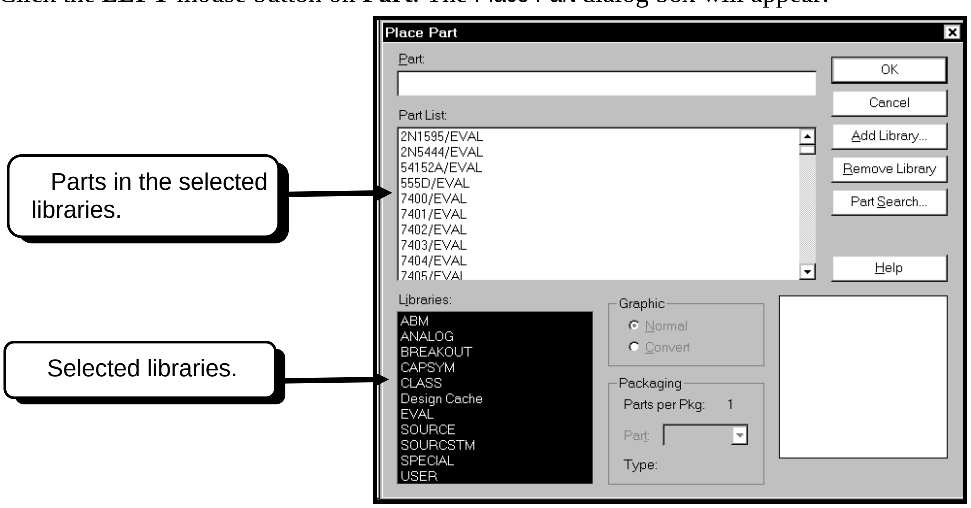


1.B. Placing Parts

We will now create a schematic. The first part we will get is an independent voltage source. All parts are retrieved using the procedure outlined below. First, click the ***LEFT*** mouse button on the **Place** menu selection:

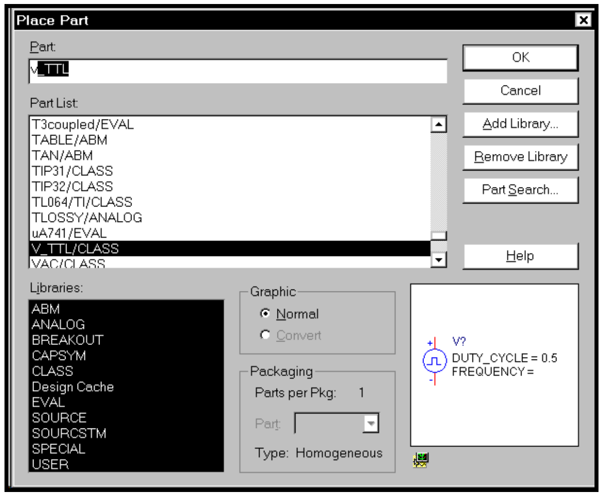


Click the ***LEFT*** mouse button on **Part**. The Place Part dialog box will appear:



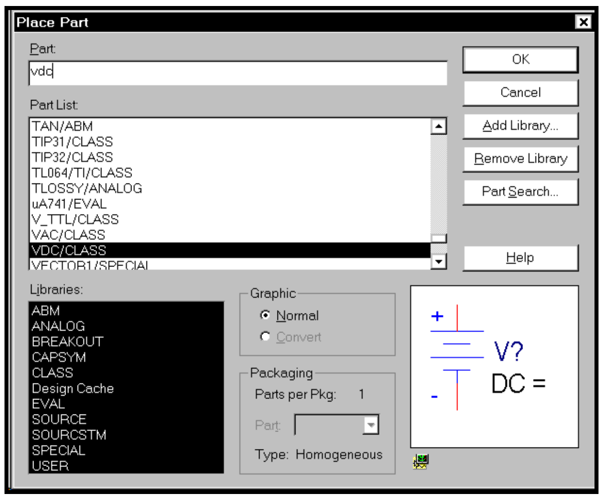
In the above dialog box, the lower left pane shows that all configured libraries are selected. The left-center pane displays the parts contained in the selected libraries. Since all libraries are selected, the left-center pane will display all parts available to us.

If you know the name of the part you want to place, you can type the part name in the box. Most independent voltage sources start with the letter v. Type the letter **v**. The left-center pane will display the parts that begin with the letter v:

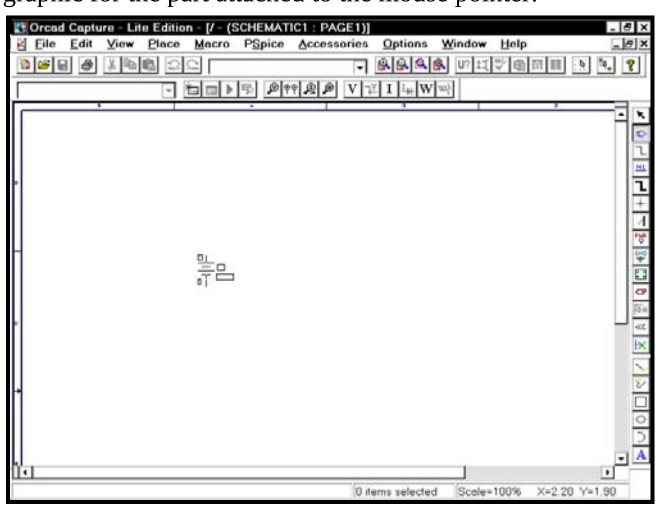


The information displayed in the highlighted line of the left-center pane is the name of the part and the library in which it is located. For example, the text V_TTL/CLASS indicates that the part name is V_TTL and the part is contained in library class.olb. The line TAN/ABM indicates that part TAN is contained in library ABM.olb.

The presently selected part is V_TTL, which was created for this textbook to generate a TTL-compatible waveform (0 to 5 V) with a specified duty cycle and frequency. We wish to select a generic DC voltage supply. Type the text **dc** to complete the typing of "vdc" and select the part:

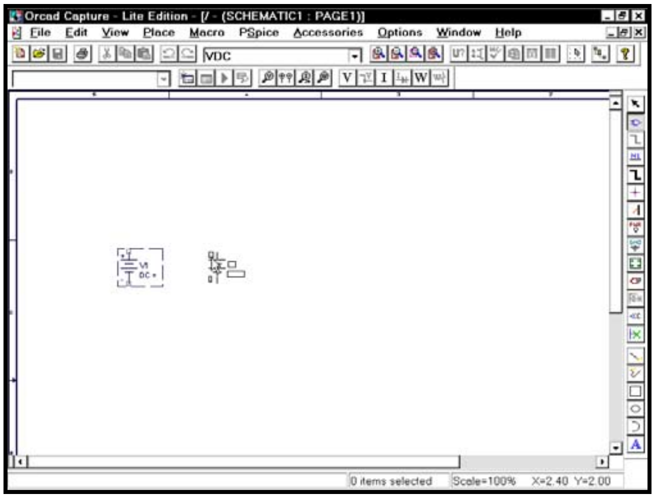


To accept the part and place it in your circuit, click the OK button. When you click the button, the program will return to the schematic with the graphic for the part attached to the mouse pointer:



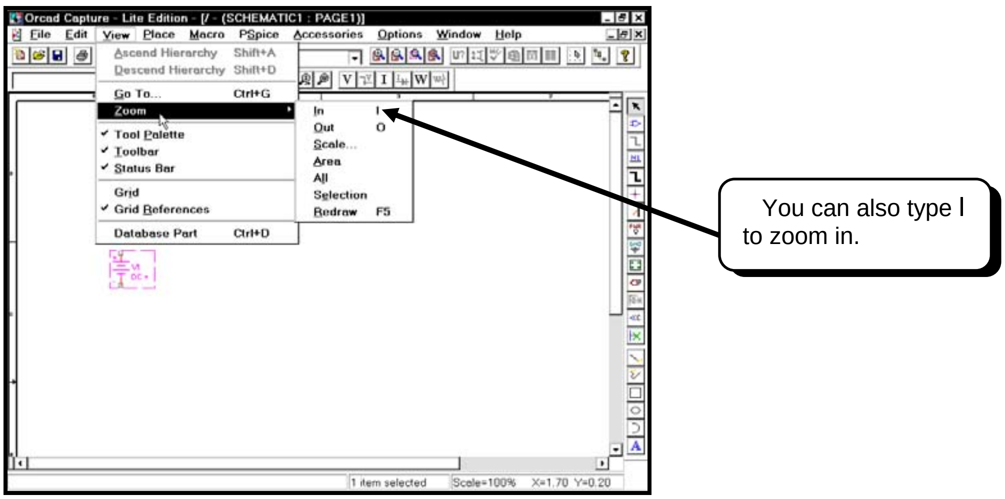
The graphic moves with the mouse. Move the graphic to the location where you want to put the voltage source. Click the ***LEFT*** mouse button **once** to place the part, and then move the mouse pointer. **Note on your screen and in the figure below**

that when you place a part, a second part appears attached to the mouse pointer. This is the auto repeat function for placing parts.

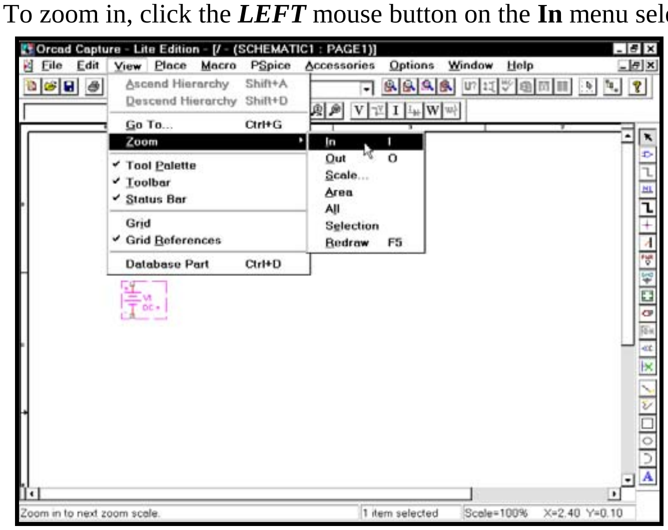


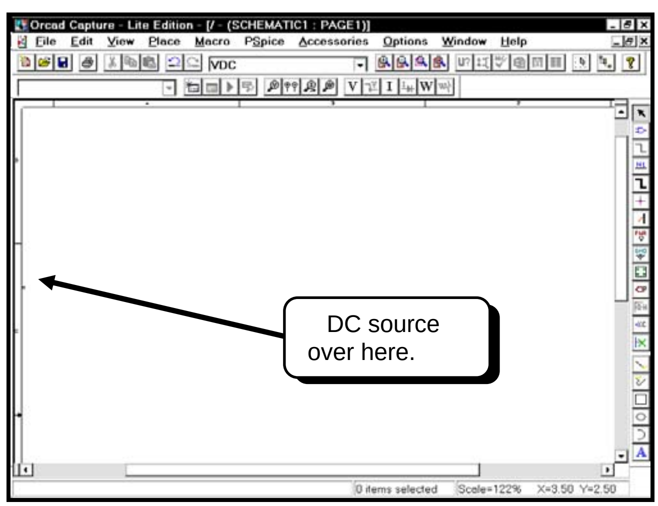
If you want to add another voltage source to your schematic, you can move the mouse to where you want to place the source and click the ***LEFT*** mouse button. We do not need another DC source at this time, so press the **ESC** key to make the second source disappear.

To make the schematic more readable, we will zoom in on the DC source. Select **View** and then **Zoom** from the Capture menus:

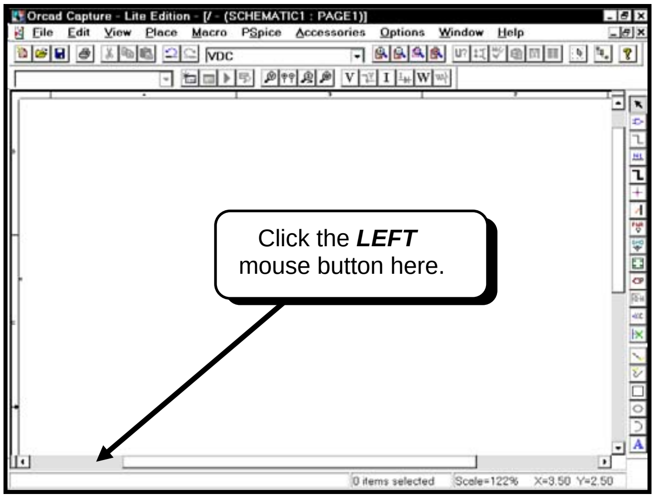


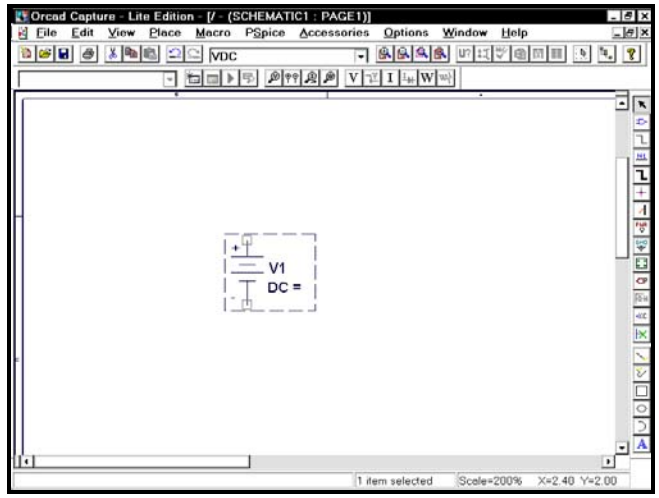
To zoom in, click the ***LEFT*** mouse button on the **In** menu selection:



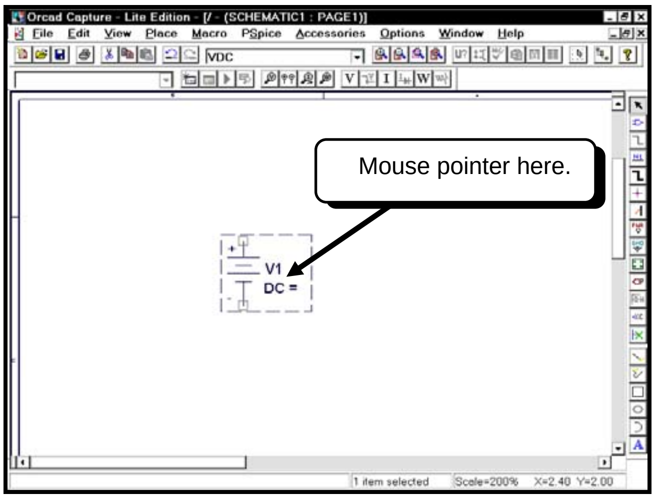


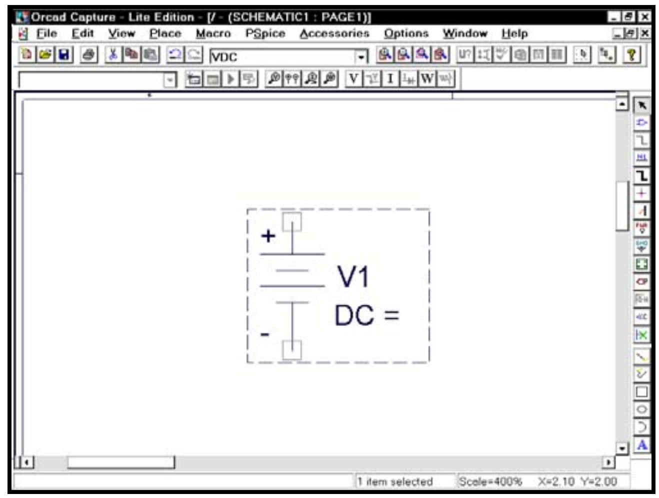
Capture zoomed in on the schematic, but the DC source is no longer visible. Click the ***LEFT*** mouse button as shown to scroll the window to the left:





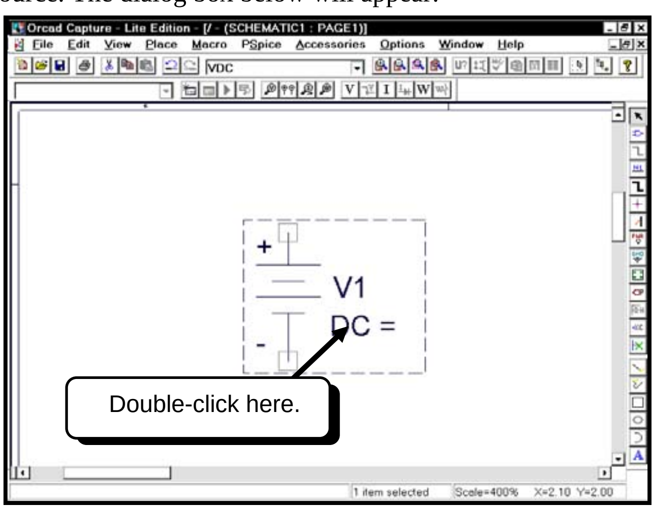
As a second example, place the mouse pointer over the DC source and then press the **I** key:

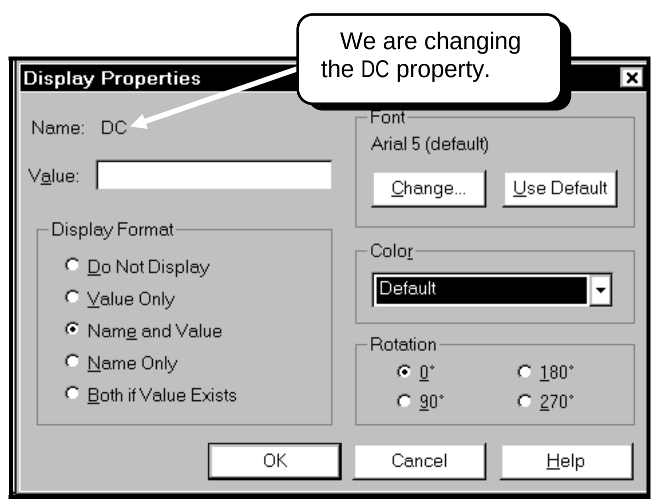




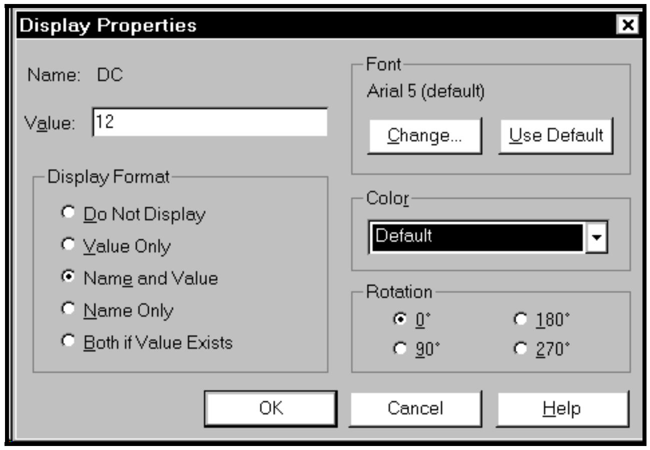
We see that the Capture zooms in around the mouse pointer. Unless you cannot remember the keyboard shortcuts, the easiest way to zoom in on an object is to place the mouse pointer at the object and then press the **I** key. (To zoom out, press the **O** key.)

We will now edit the properties of the source to make it a 12 VDC source. There are two ways to change the properties of a part. The first way we will look at is editing the individual properties that are displayed on the screen. We will first change the voltage of the DC source. To do this, double-click the ***LEFT*** mouse button on the DC= text next to the DC source. The dialog box below will appear:

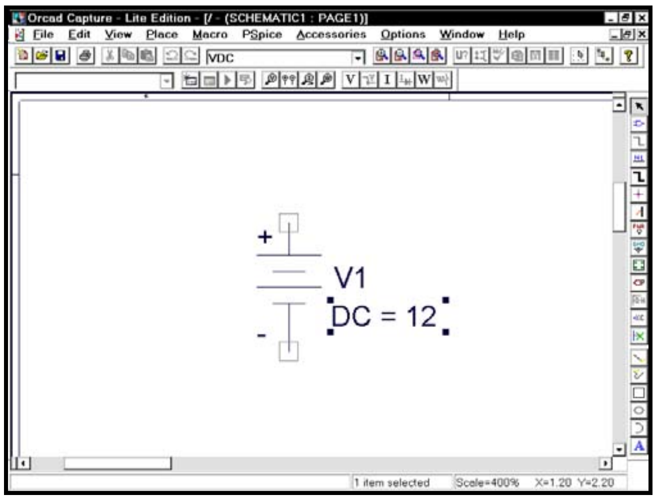




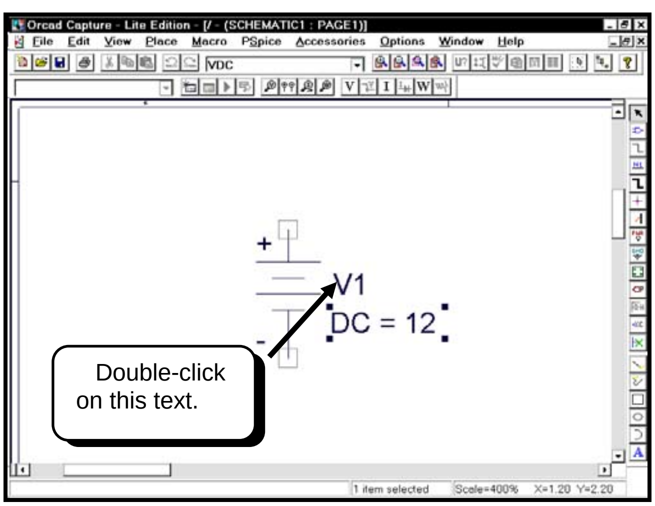
Note that the dialog box says that the property we are changing is the DC property. To change the property, type in the desired value. We will create a 12-volt source, so type in **12**:

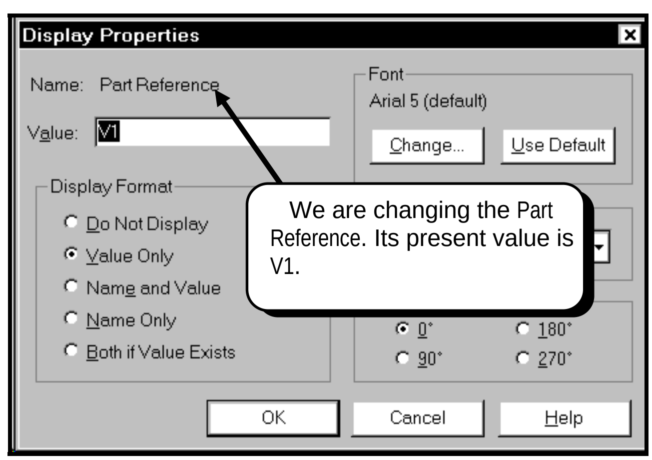


Click the OK button to change the property. The part will appear in the schematic with the line DC=12 displayed in the schematic:



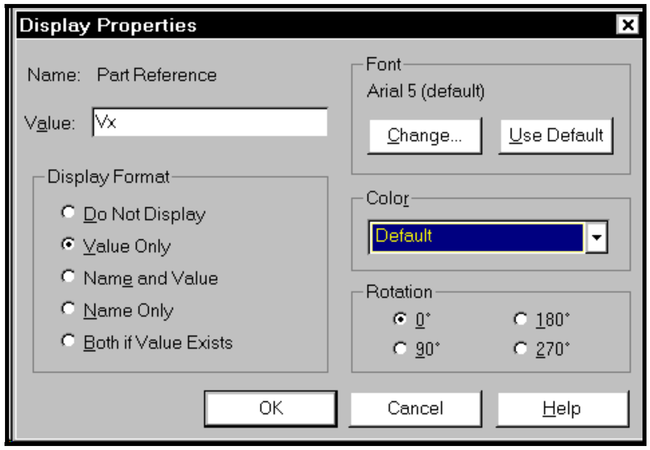
If your text appears garbled, press the F5 key to redraw the screen. To change the name of the voltage source, double-click the ***LEFT*** mouse button on the text V1.[1] The dialog box below will appear:



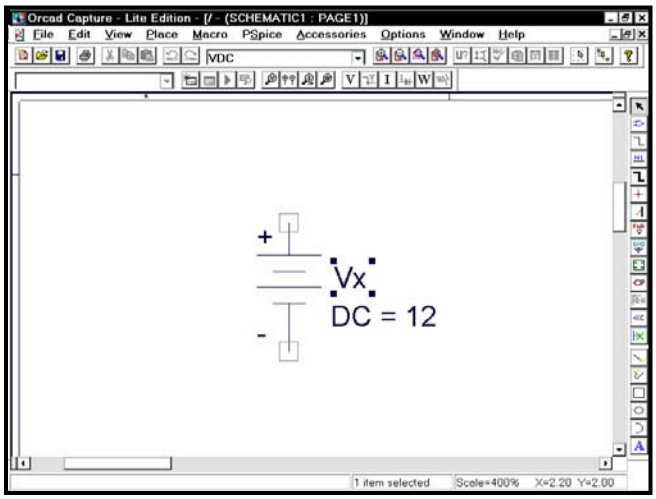


The present name of the source appears highlighted. To change it, type the desired name of the source. We will change the name of the source to Vx, so type the text **Vx**:

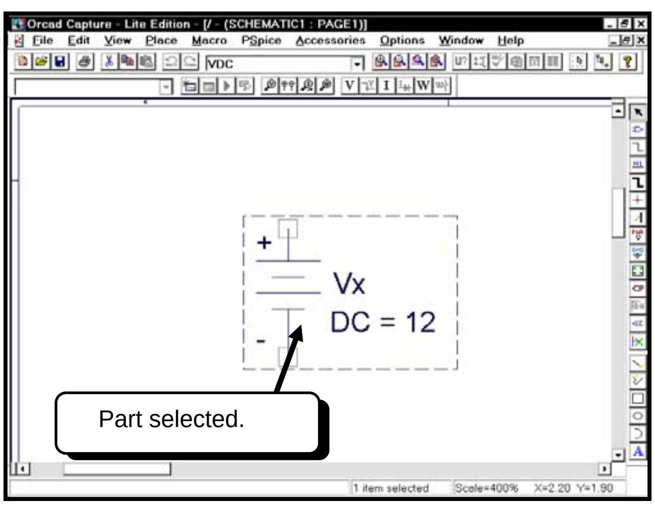
[1]The name of the source may vary depending on how many sources you have placed in your schematic.

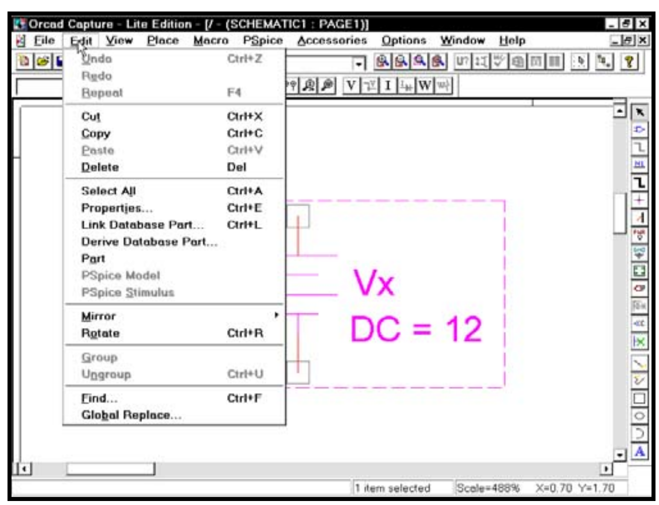


Click the OK button. The updated part will appear as shown below:



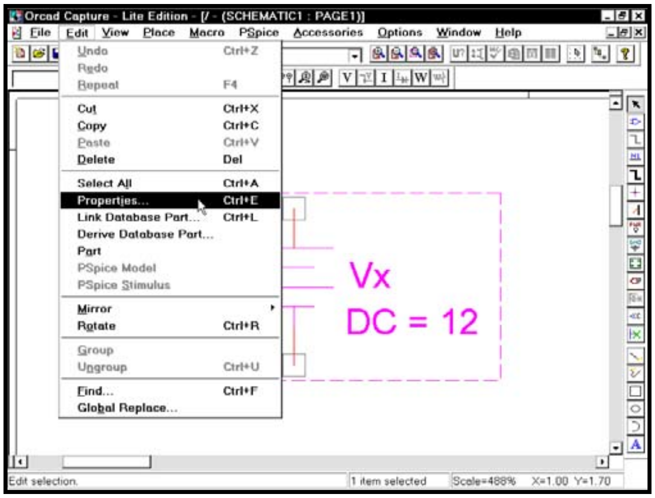
The part now has the desired properties. To illustrate the second method of changing a part's properties, we will change the source voltage to 15 volts and the name back to V1. To change the properties, click the ***LEFT*** mouse button once on the DC source graphic, . The graphic will be highlighted in pink when the part is selected. When the part is highlighted, click the ***LEFT*** mouse button on the **Edit** menu selection:

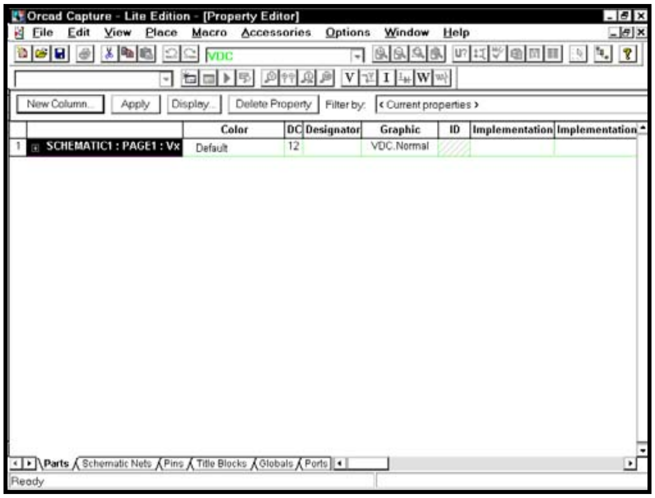




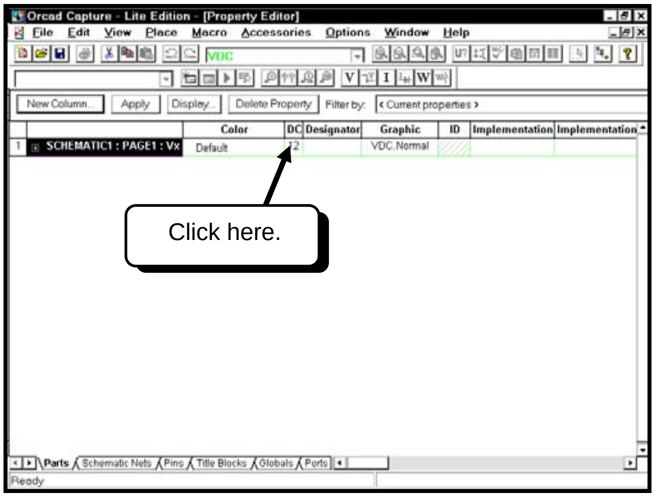
Click the ***LEFT*** mouse button on the **Properties** menu selection:[2]

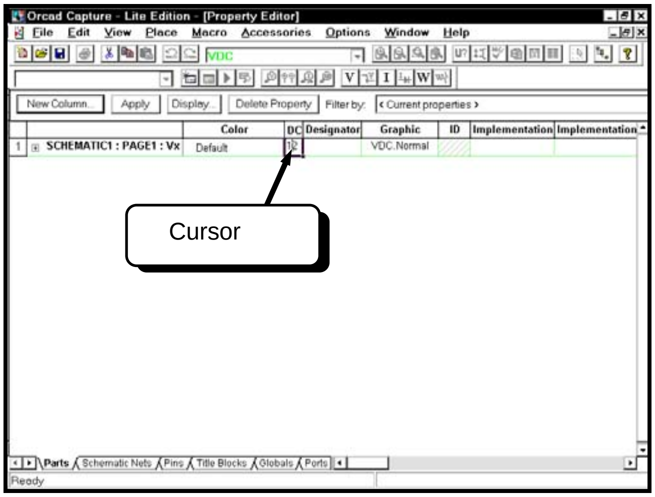
[2]This dialog box can also be obtained by double-clicking the ***LEFT*** mouse button on the DC voltage source graphic, . Note also that if the text **Properties** in the menu appears as **Properties**, you have not properly selected the graphic. Click the ***LEFT*** mouse button on the graphic again until it becomes highlighted in red. When the graphic is highlighted in red, you may edit its properties.



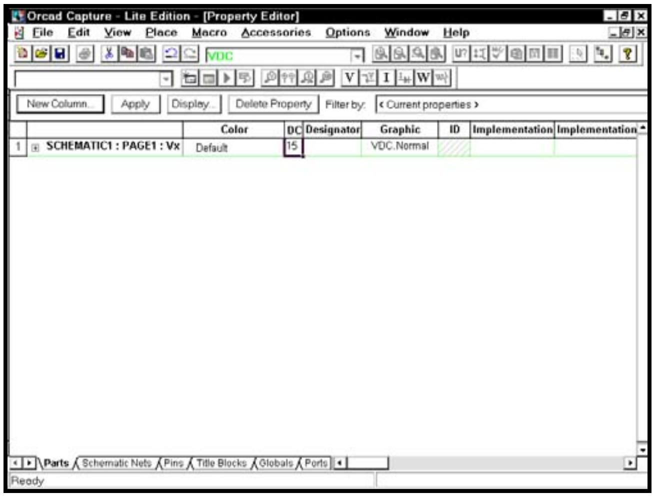


This window displays a spreadsheet of all of the part's properties.[3] Note that there are more properties than appear on the schematic. Not all properties of a part are displayed on the schematic. To change the voltage of the source, click the ***LEFT*** mouse button on the text 12 in the DC column to select the cell:



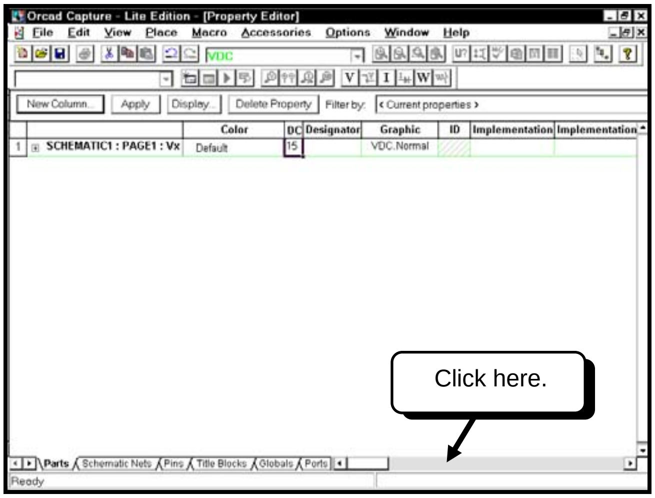


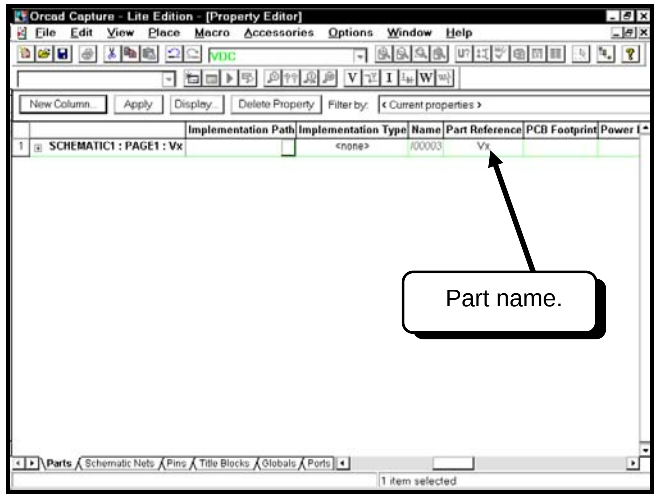
You can now edit the contents of the cell. Use the **BACKSPACE** and **DELETE** keys to clear out the cell and then enter the text **15** to make a 15-volt DC source:



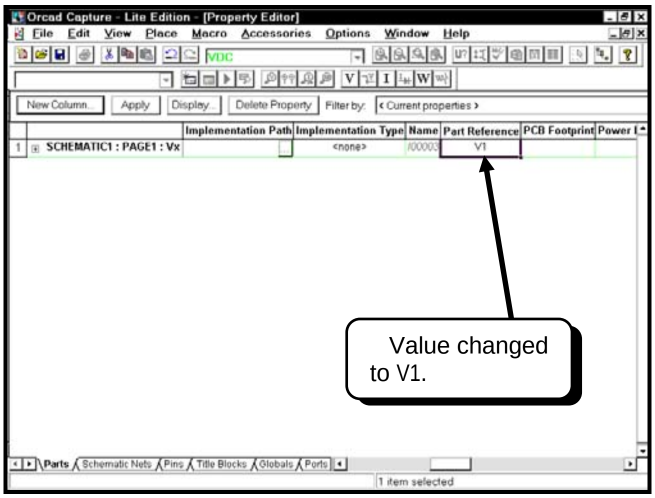
Next, we will attempt to change the name of the source. The property that specifies the name of the source is not displayed in the window. Click the ***LEFT*** mouse button as shown below to scroll the spreadsheet to the right:

[3] You can also obtain the properties spreadsheet by clicking the ***LEFT*** mouse button on a part to select it. Once the part is selected, click the ***RIGHT*** mouse button on the part and then select **Edit Properties** from the menu that appears.

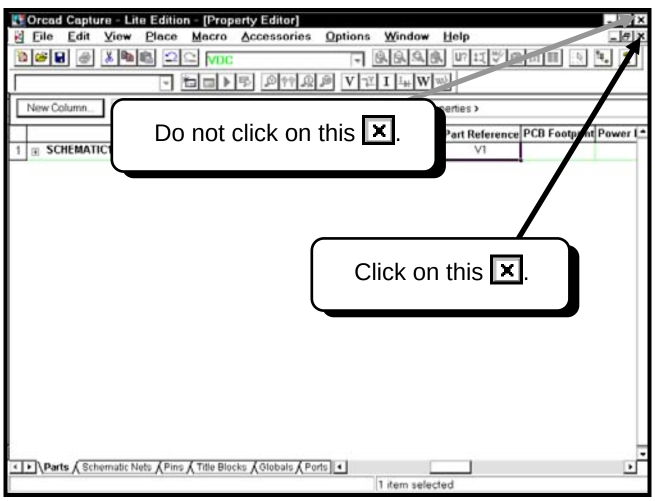


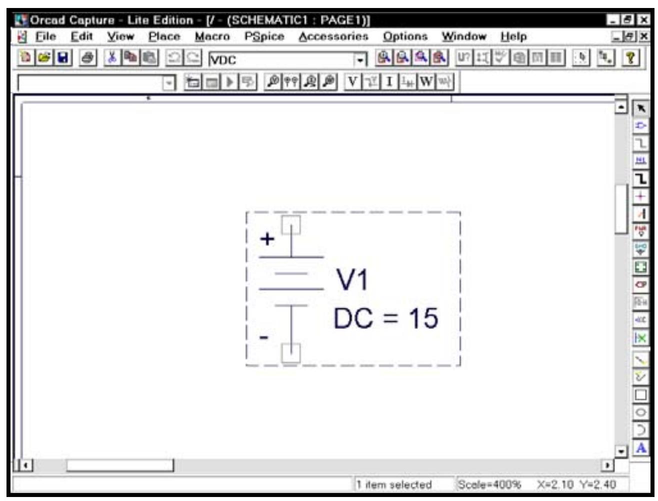


We can now see the Part Reference property. Change the value Vx to **V1**:



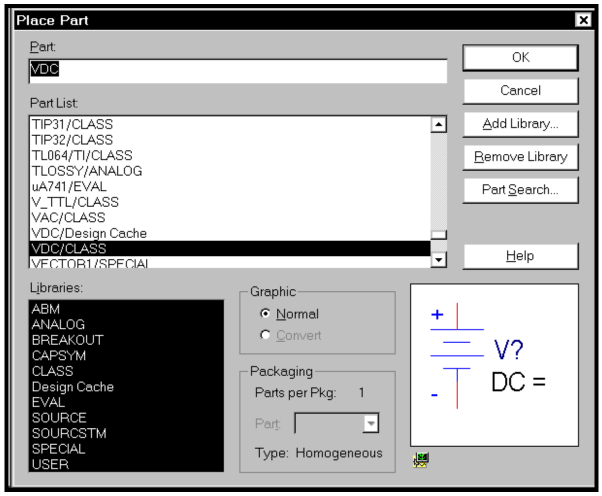
To return to the schematic, click the ***LEFT*** mouse button on the ☒ as shown below or type **CTRL-F4**:



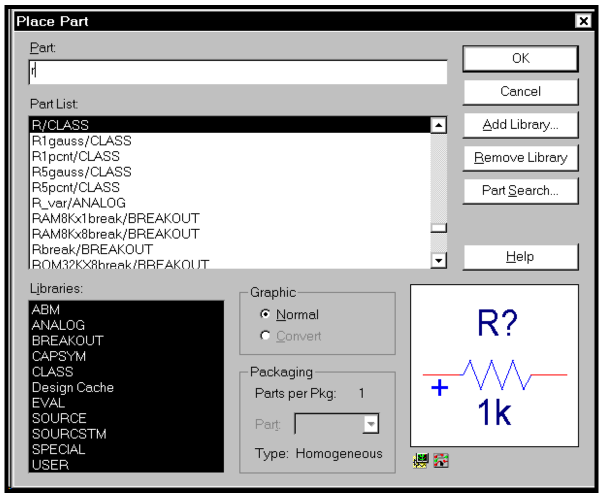


We see that both the line DC=15 and the name of the source, V1, have changed in the schematic. For practice, change the name of the source back to Vx.

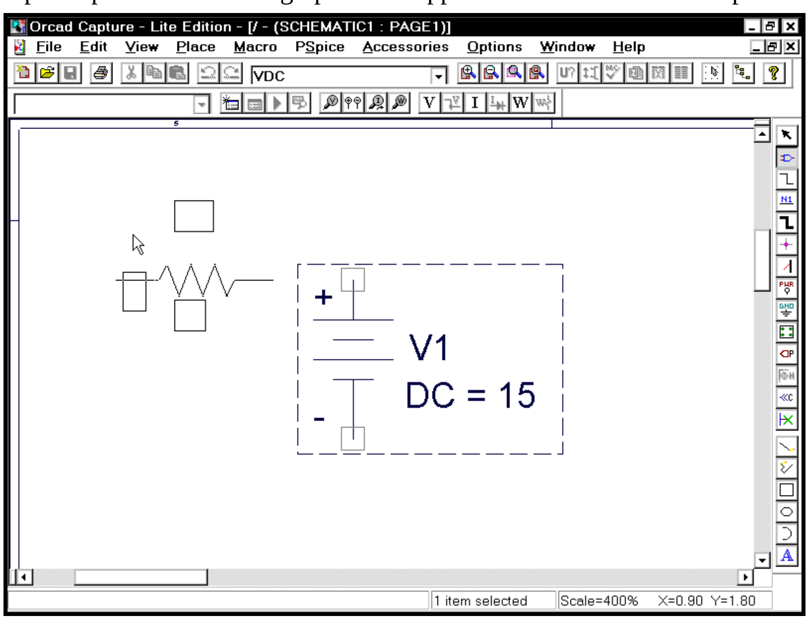
We now need to place some resistors in the circuit. The part name for a resistor is R. We could choose the **Place** and then select **Part** to get the resistor. Instead, we will type p. The Place Part dialog box will appear:



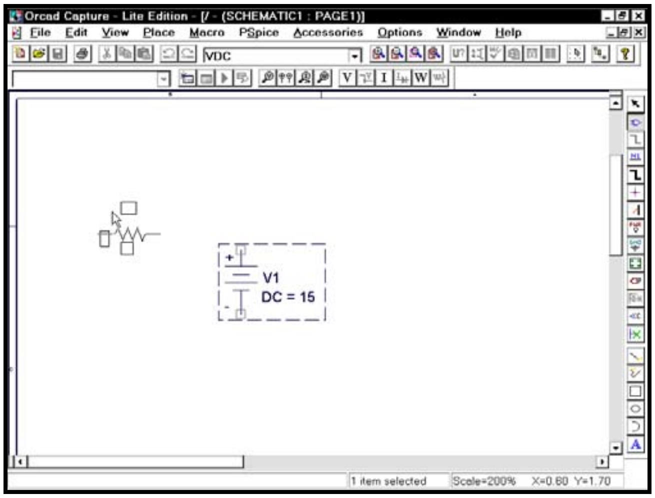
Since we know that a resistor part is called “R” we will just type in **r**:



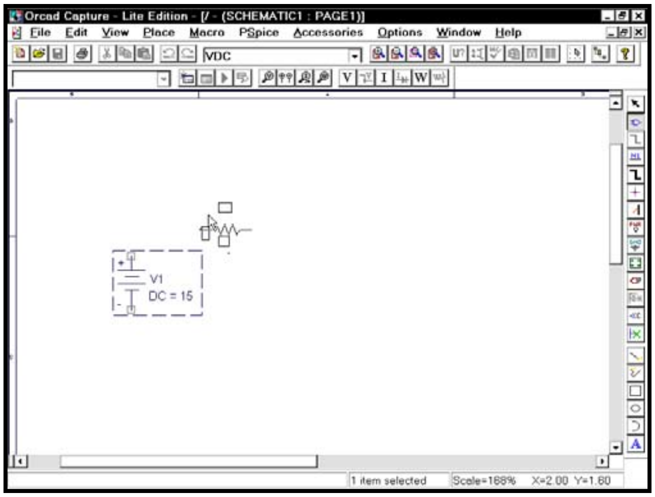
Click the OK button to accept the part. The resistor graphic will appear attached to the mouse pointer:



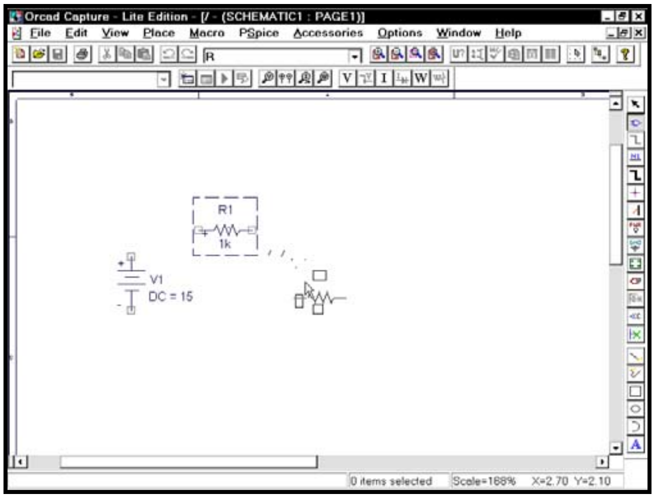
The schematic is zoomed in too much on the DC source to place any resistors. To zoom out, press the **O** key:



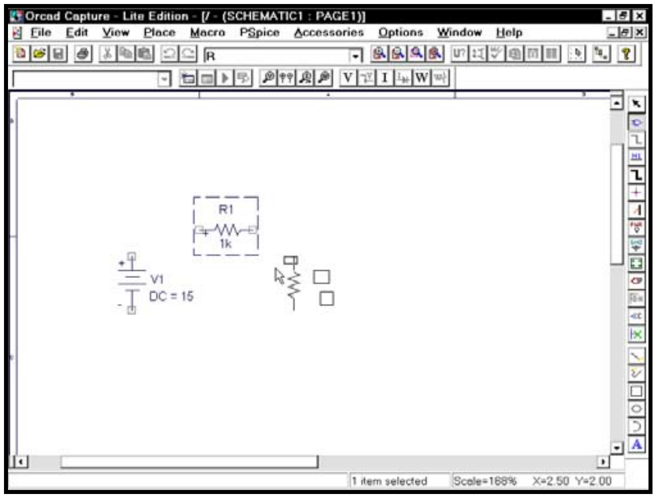
We now have a better view of the schematic page. Note that we can zoom in and out while still placing parts and that the resistor moves with the mouse. Zoom out until you see a screen close to the one shown below:



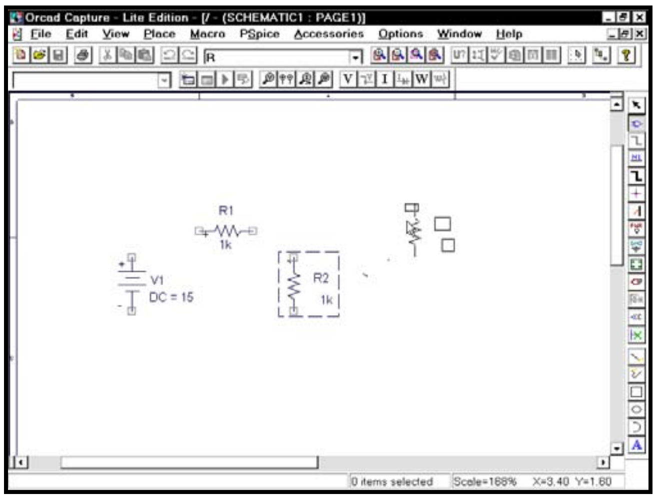
Move the resistor to where you want to place it and click the ***LEFT*** mouse button. The resistor is placed, and a second resistor appears attached to the mouse pointer:



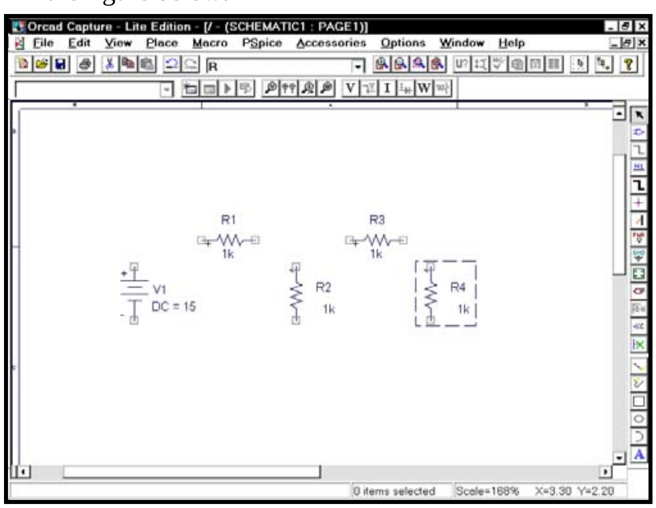
We now wish to add a second resistor to the schematic. The next resistor we want to place will be vertically oriented. Currently the resistor attached to the mouse is horizontal. To rotate the part, type R. The part will rotate while attached to the mouse. Orient the part vertically as shown in the screen below.



Move the mouse to place the part in the location shown below. To place the part click the ***LEFT*** mouse button. When you click the ***LEFT*** mouse button, the part is placed, and a third resistor appears attached to the mouse pointer:

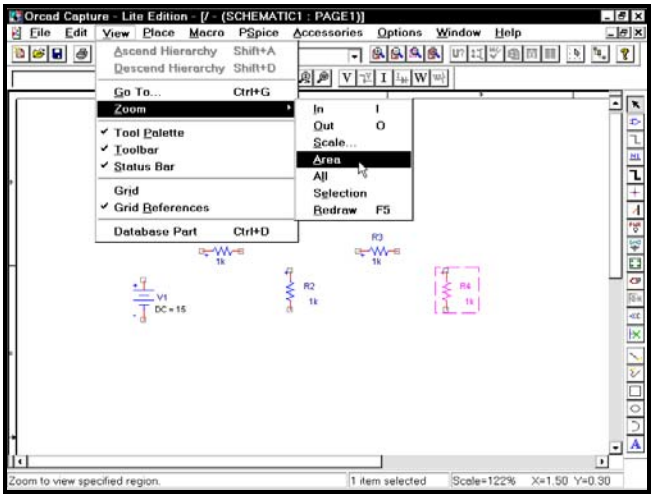


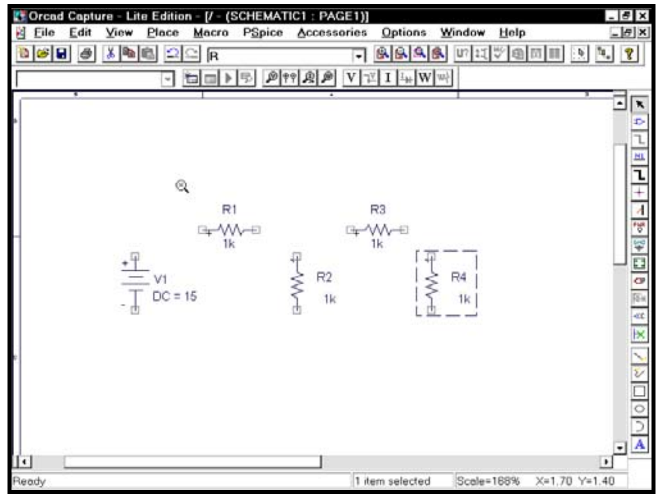
Add two more resistors as shown in the figure below:



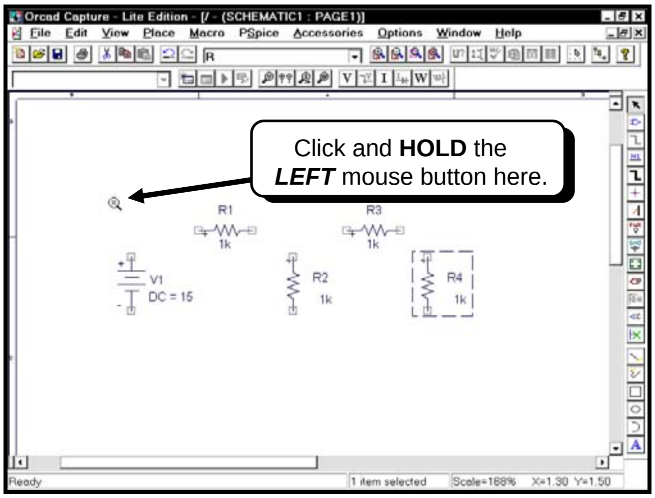
Press the R key to rotate the part when necessary and click the ***LEFT*** mouse button to place the part. When you have placed all four resistors, press the **ESC** key to stop placing resistors. When you press the **ESC** key, the resistor graphic at the mouse pointer disappears.

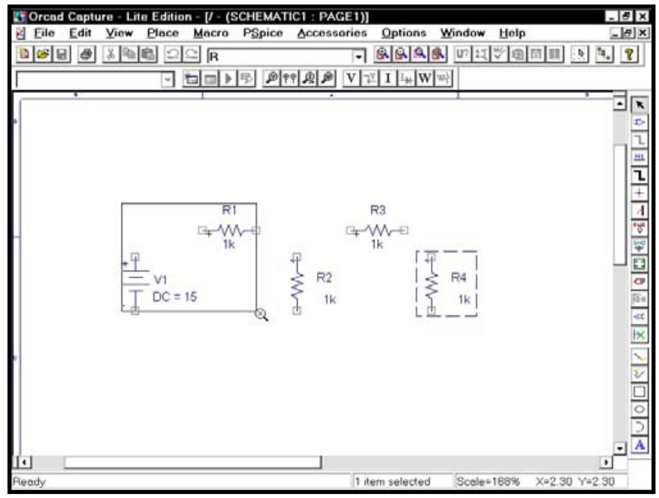
Before we continue, we would like to zoom in on the circuit and make it as large as possible while still displaying the entire circuit. Select **View**, **Zoom**, and then **Area** from the menus. The mouse pointer will be replaced by a magnifying glass:



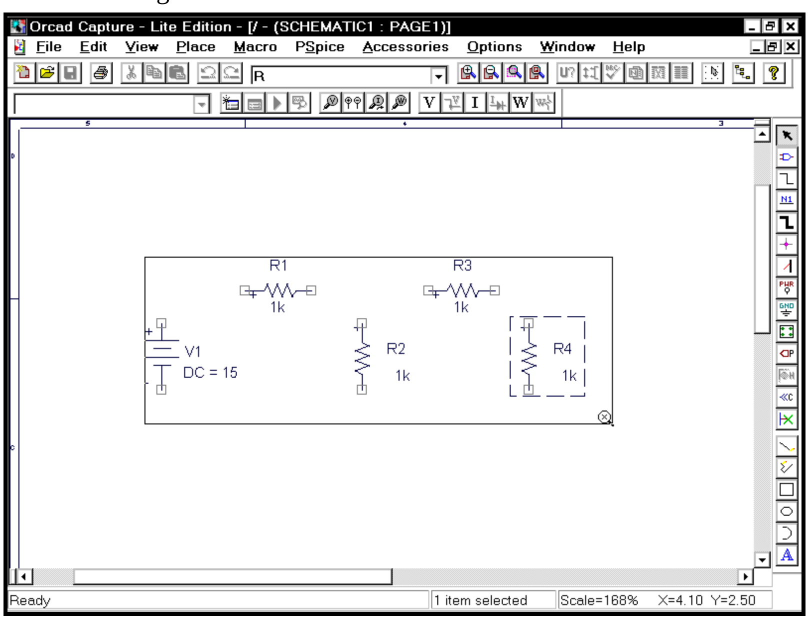


Place the mouse pointer near the upper left corner of the area in which you would like to zoom and then click and **HOLD** the ***LEFT*** mouse button. When you move the mouse away, a zoom rectangle will be shown:

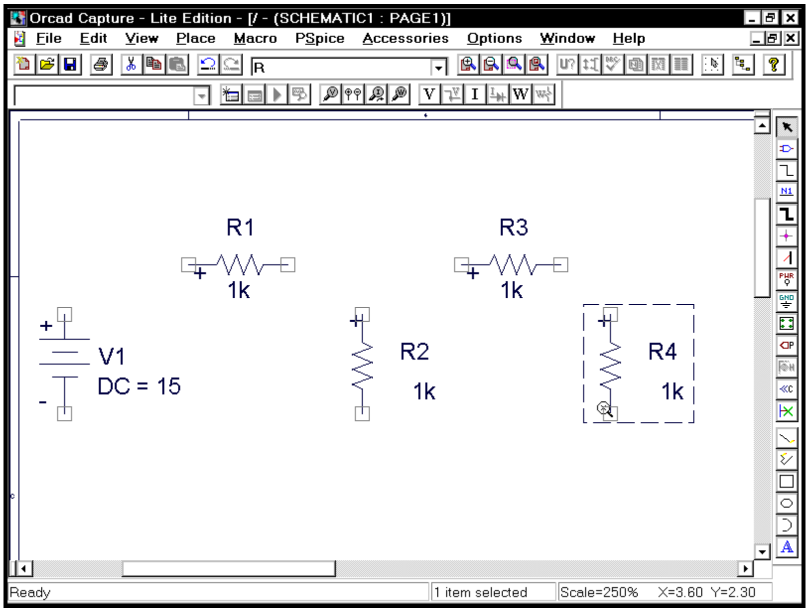




Drag the mouse to create a zoom rectangle that encloses the entire circuit:



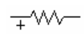
When you release the mouse button, Capture will zoom in on the area enclosed by the rectangle:



To end zoom mode, press the **ESC** key.

1.C. Correcting Mistakes

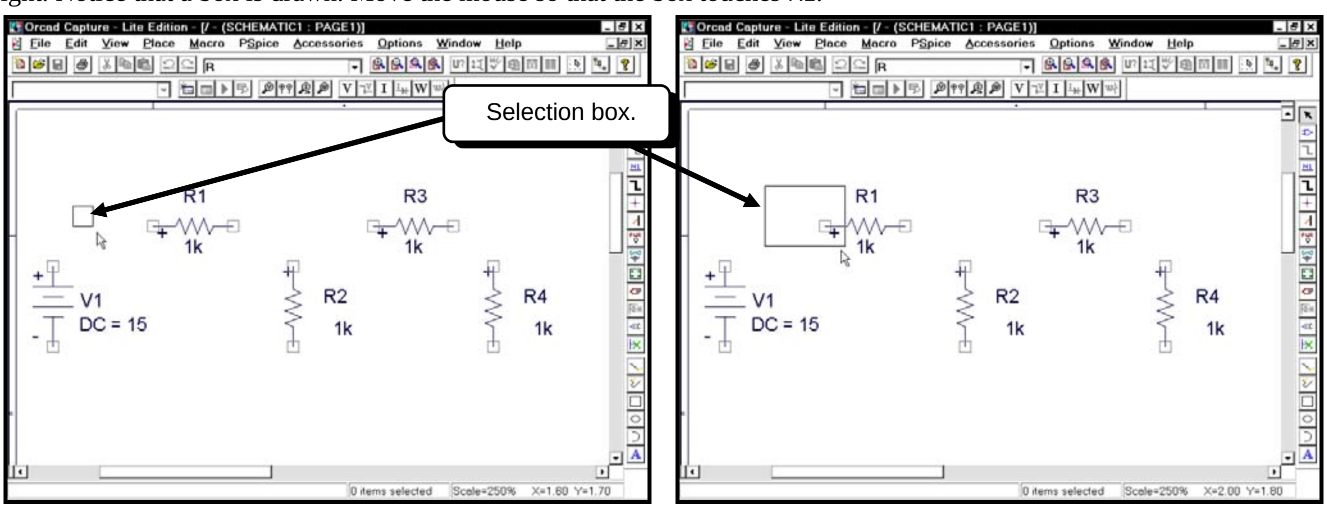
At this point you may have some mistakes in your schematic. You may have clicked the ***LEFT*** mouse button one too many times and placed too many resistors on your schematic, or you may have placed resistors too close to each other. To move parts follow the procedure below. For the moment we will assume that you wish to move a resistor.

1. Click the ***LEFT*** mouse button on the resistor graphic, , you wish to move. When the resistor graphic is highlighted in pink, it has been selected. It may take several tries to highlight the resistor.
2. When the graphic is highlighted in pink, drag the resistor graphic to the desired spot in the schematic.

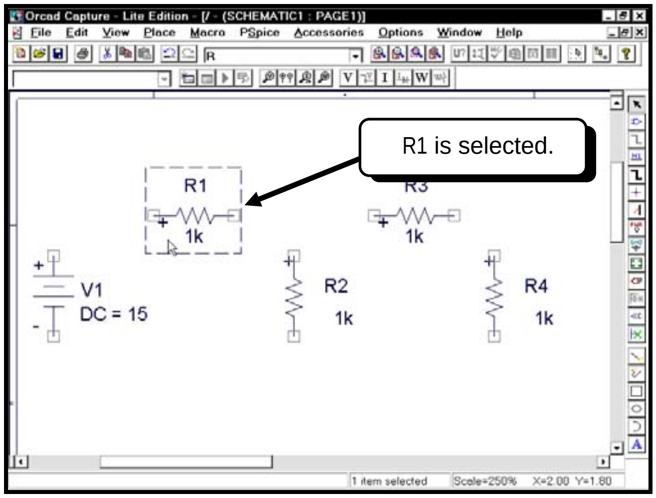
If you need to delete a part, follow Step 1 above. When the appropriate part is selected, press the DELETE key.

1.D. Changing Properties

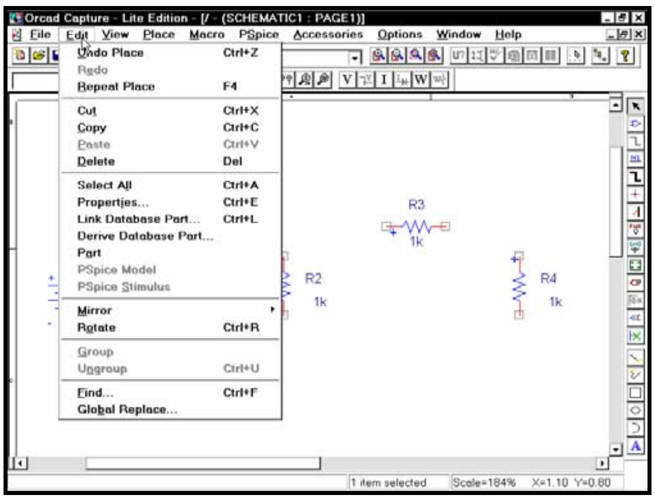
There are several ways to change the properties of parts. Some have already been illustrated previously, but we will now go over all of the different methods. The first thing we need to do is to select a part. We will select resistor R1. Place the mouse pointer to the left of and above R1. Press and hold the ***LEFT*** mouse button and move the mouse down and to the right. Notice that a box is drawn. Move the mouse so that the box touches R1:

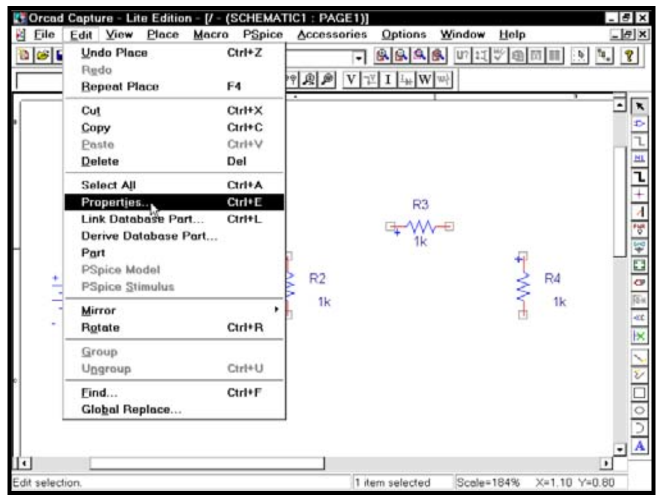


When you release the mouse button, everything touching the box will be selected. In the screen capture above, the box is touching the R1 graphic, so it will be selected:

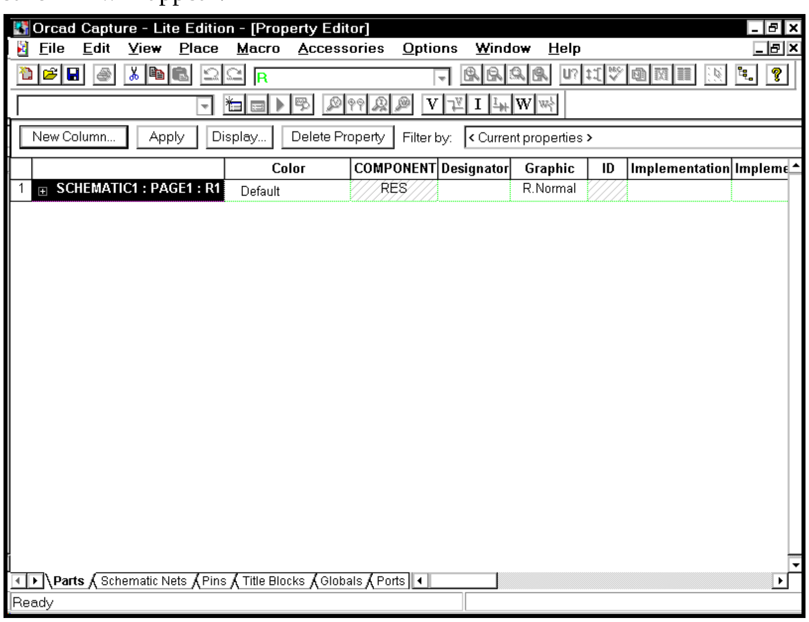


Resistor R1 is highlighted in pink and enclosed in a dashed box, indicating that it is selected. To edit all of the selected item's properties at the same time, click the ***LEFT*** mouse button on the **Edit** menu selection and then select **Properties**:

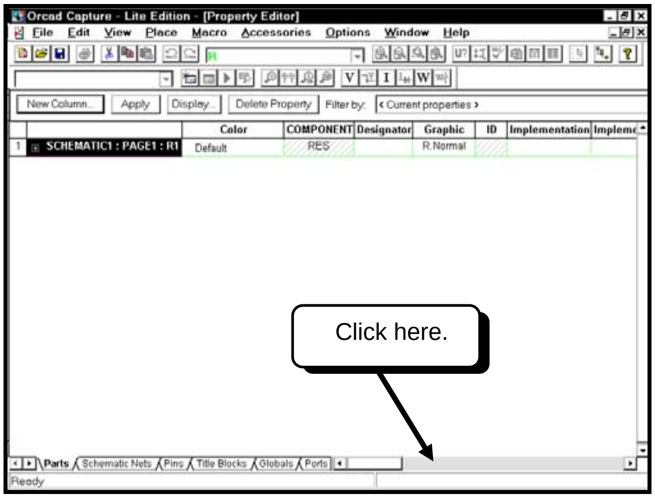


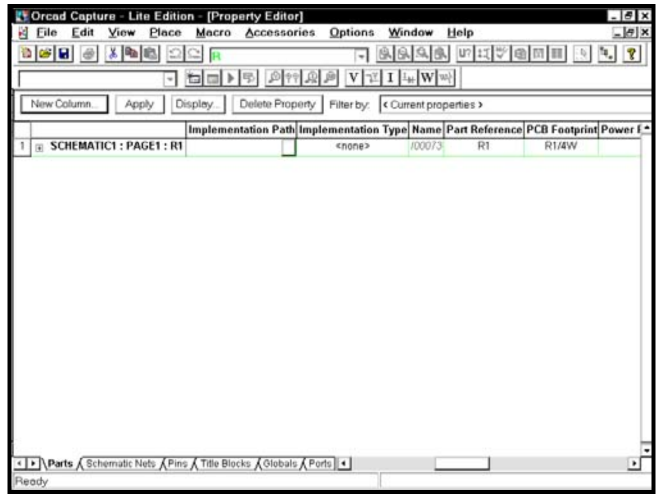


The properties spreadsheet for R1 will appear:

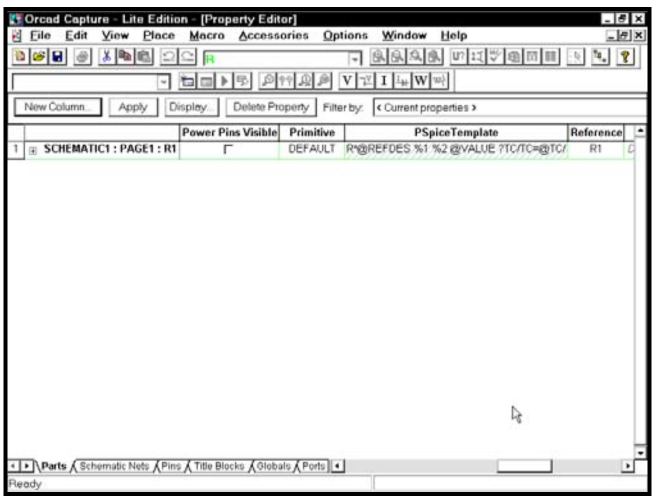


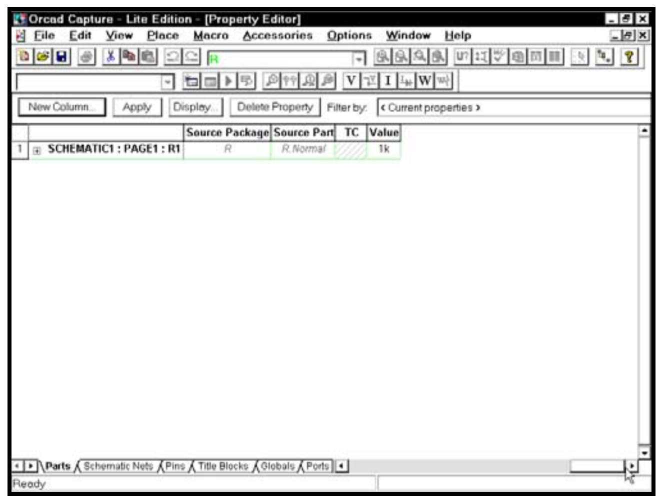
This screen shows all the properties of R1. Presently, none of the properties displayed in the window are of interest to us. Click the ***LEFT*** mouse button on the horizontal scroll bar as shown below to scroll the spreadsheet right:





The Part Reference property is the name of the resistor, R1 in this case. The PCB Footprint property is the graphic that will be used if you use this schematic to create a PC board. If we scroll the window to the right, we will see more properties. Two screen captures are shown below to display all of the properties:

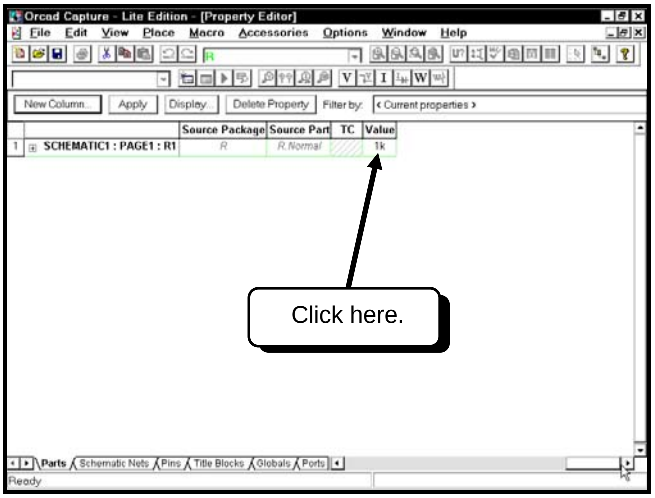


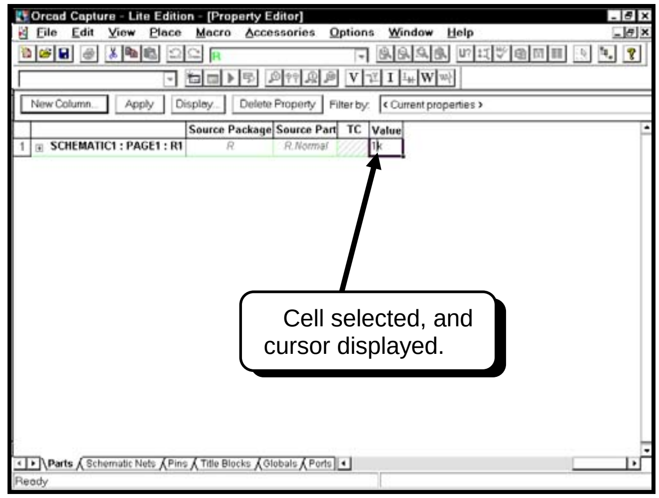


The value of the PSpice Template property generates the PSpice netlist line for the resistor when you create a netlist for the schematic. The TC property is a temperature coefficient for the resistor. Its default is zero (no temperature dependence). The property Value is the value of the resistor in ohms. The Source Library (not shown in the screen captures above) is the name of the .olb file in which the part is located.

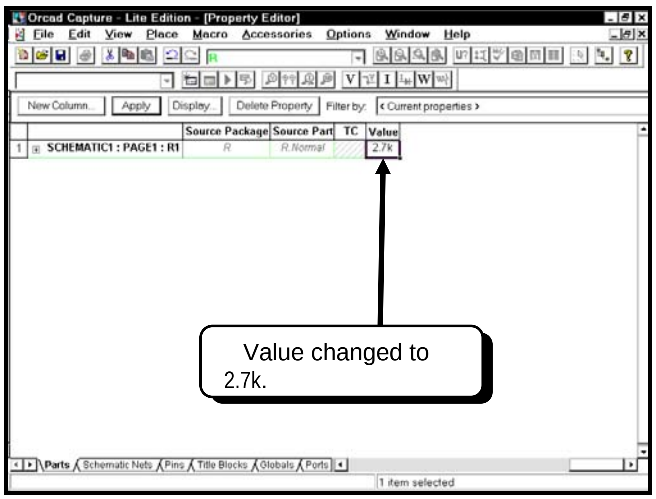
As you scroll through the properties, you will notice that two properties appear to have the same value, Reference and Part Reference. Both have a value of R1 for this part. The difference between the two properties appears when you use packages that have multiple parts, such as NAND gates and quad op-amps. For a NAND gate, the package would have a reference such as U1. Each gate within the package has its own designation, such as A, B, C, or D. An individual gate in the package would have a unique part reference, such as U1A, U1B, and so on. The Reference property is the package reference, such as U1, while the Part Reference property is the reference to an individual gate in the package, such as U1A. The Part Reference is used by PSpice and the Reference is used by the layout program.

We will now change a few of the properties. To change the resistance value, click the ***LEFT*** mouse button on the text 1k. The cell will be selected and the cursor will appear in the cell:

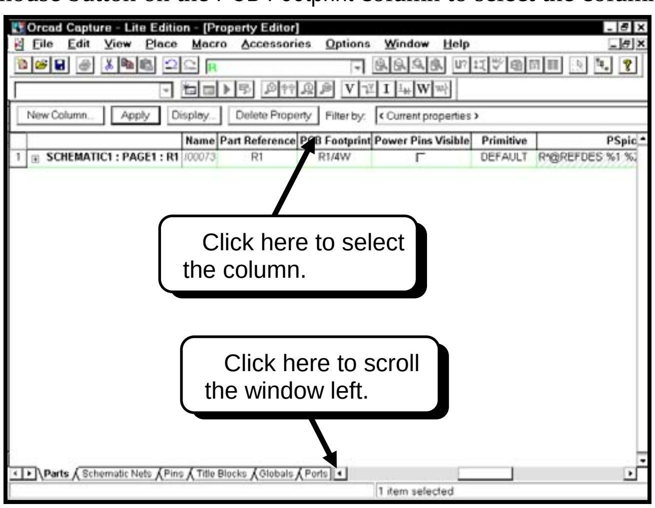


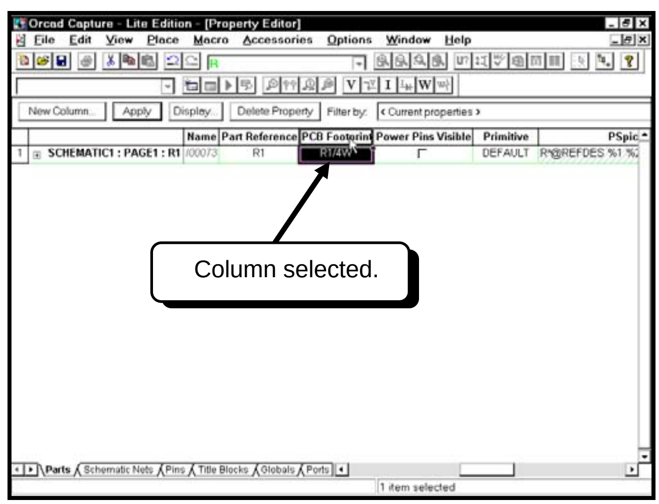


Change the value from 1k to **2.7k**. **There must not be a space between the number 2.7 and the letter "k."** After you change the value to 2.7k, click the Apply button:

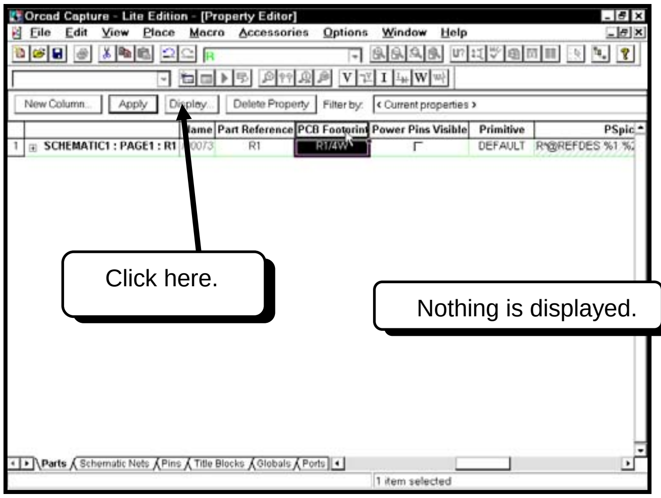


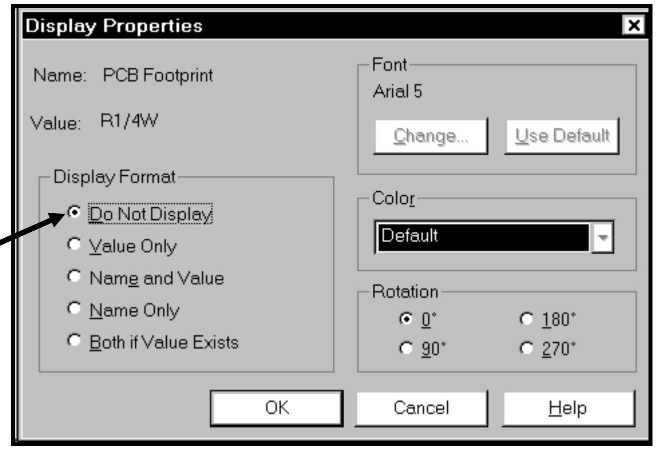
If you remember, the footprint of the resistor was not shown on the schematic. This is because the PCB Footprint property is not being displayed. All properties can either be displayed or not. We will illustrate how to display the PCB Footprint property. Scroll the properties window to the left to display the PCB Footprint property and then click the ***LEFT*** mouse button on the PCB Footprint column to select the column:



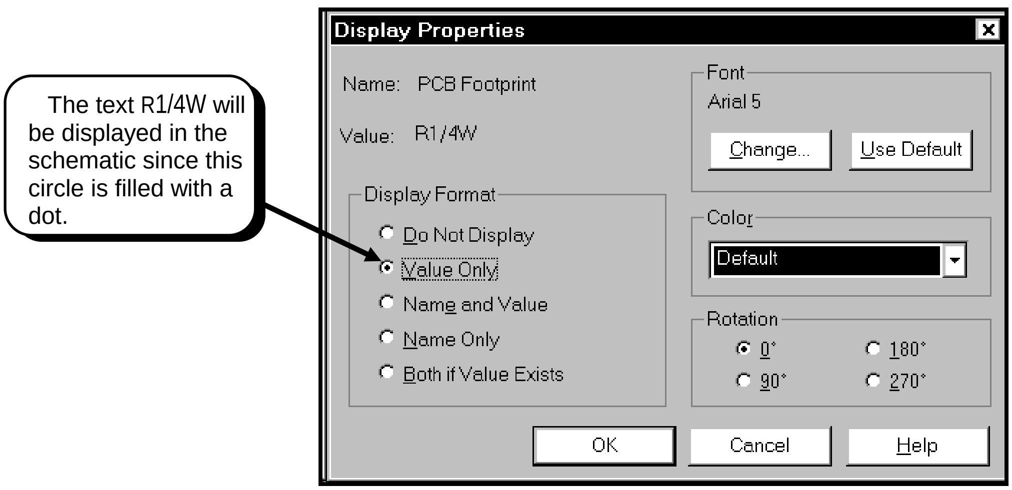


Click the ***LEFT*** mouse button on the Display button. The dialog box shown below will appear:

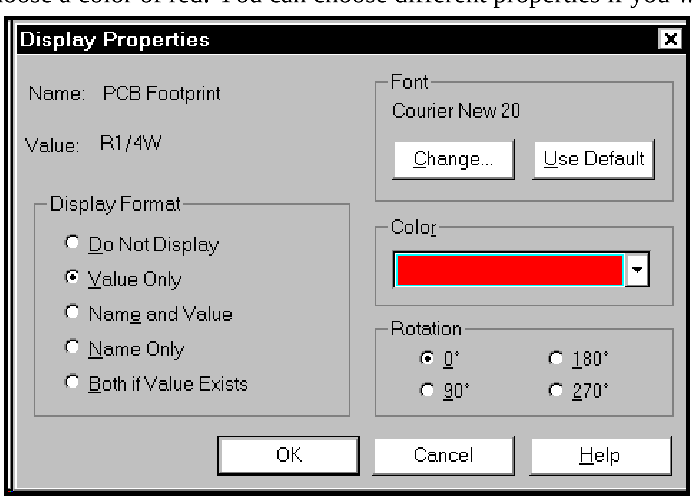




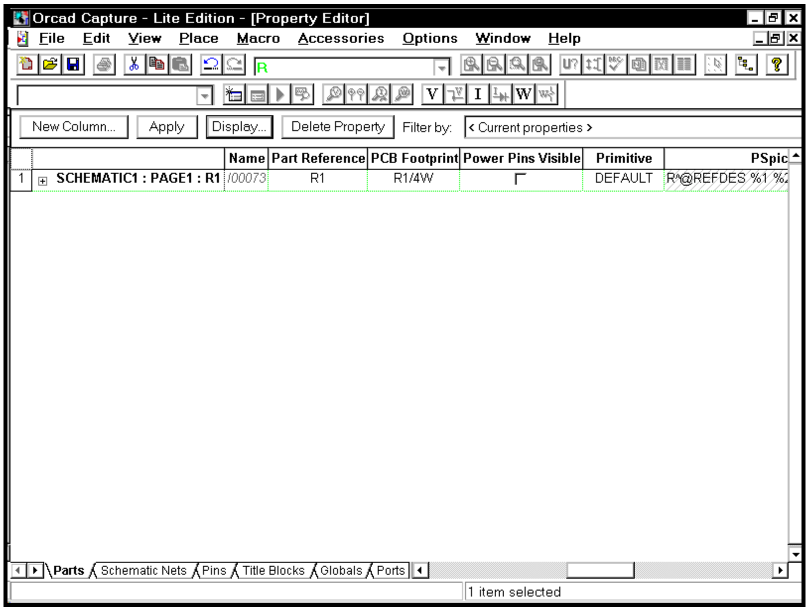
This screen indicates that neither the value R1/4W nor the name PCB Footprint will be displayed on the schematic. There are several display options. Selecting the Value Only option will display the value of the property. In this case, the text R1/4W will be displayed. Selecting Name Only will display the name of the property. In this case, the text PCB Footprint will be displayed. Selecting the option Name and Value will display the name of the property and its value. In this case the text PCB Footprint=R1/4W will be displayed. Selecting Both if Value Exists will display the property name and the value on the schematic if a value is assigned to the property. Presently, Do Not Display is selected, meaning that nothing related to this property will be displayed on the schematic. To select another display option, click the ***LEFT*** mouse button on the circle ○ next to the option you wish to select. The circle should fill with a dot ◉, indicating that the option has been selected. I will display only the value of the property:



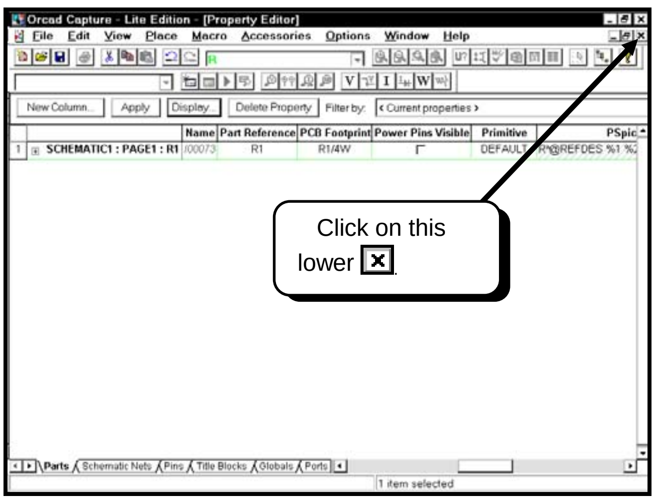
You can also change the font, font size, and the color of the font in which the property value is displayed. I will select a 20 point Courier New font and choose a color of red. You can choose different properties if you wish.

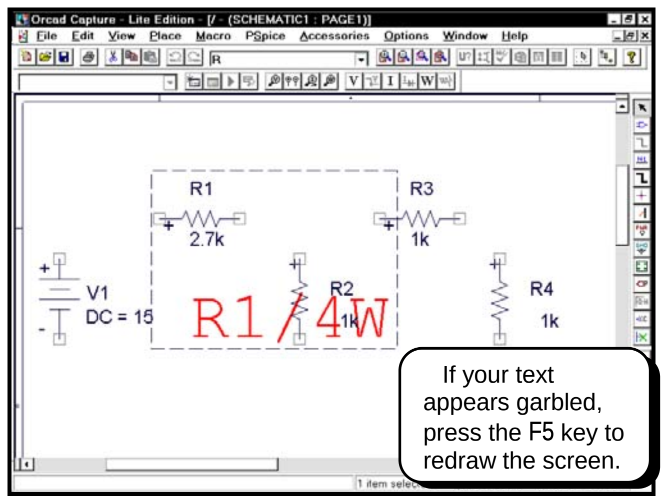


To accept the changes click the ***LEFT*** mouse button on the OK button. You will return to the properties spreadsheet:

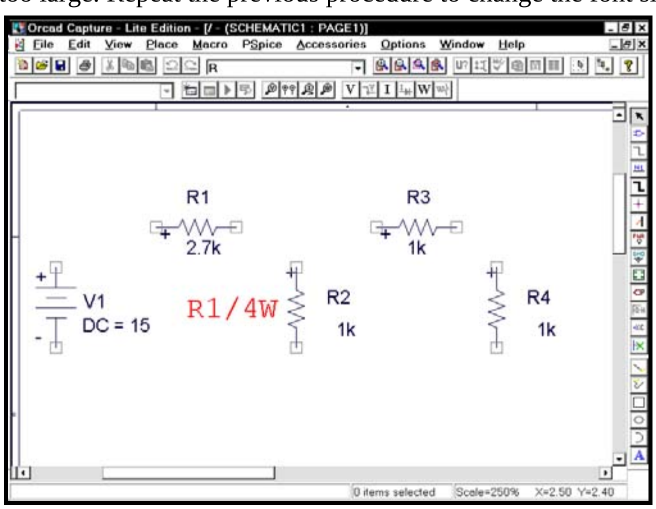


Click the ***LEFT*** mouse button on the lower ☒ in the upper right corner of the spreadsheet window to close the window as shown below. You will return to the schematic with all of the properties updated:

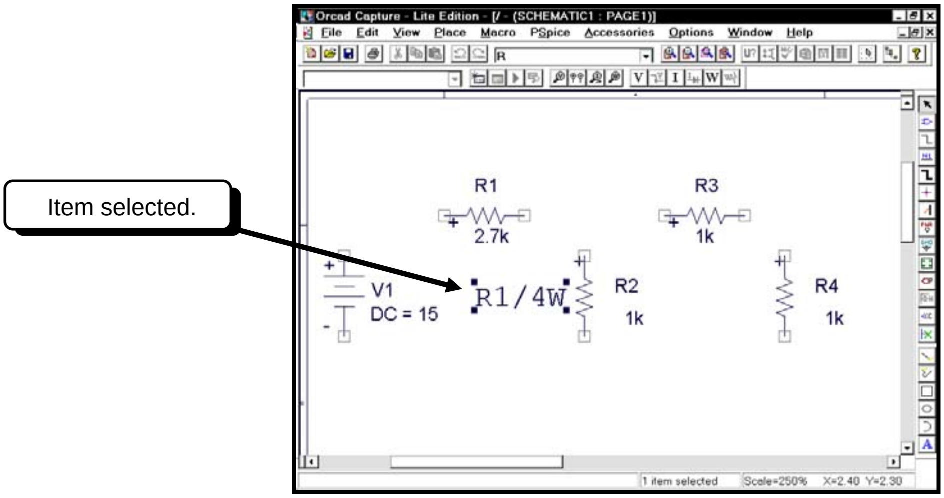




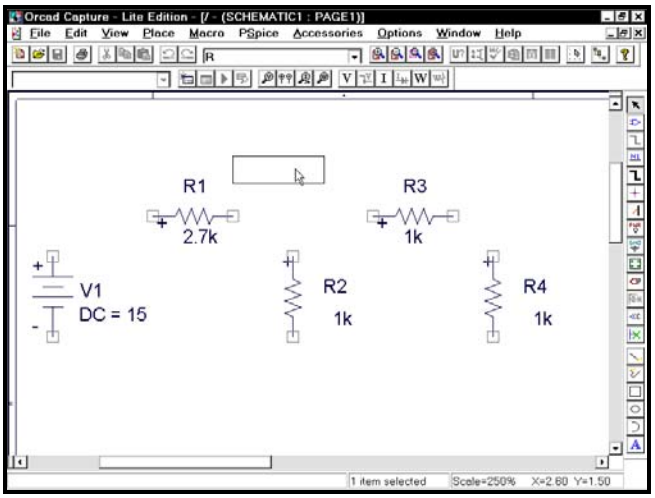
The text R1/4W is a bit too large. Repeat the previous procedure to change the font size to 9 points:



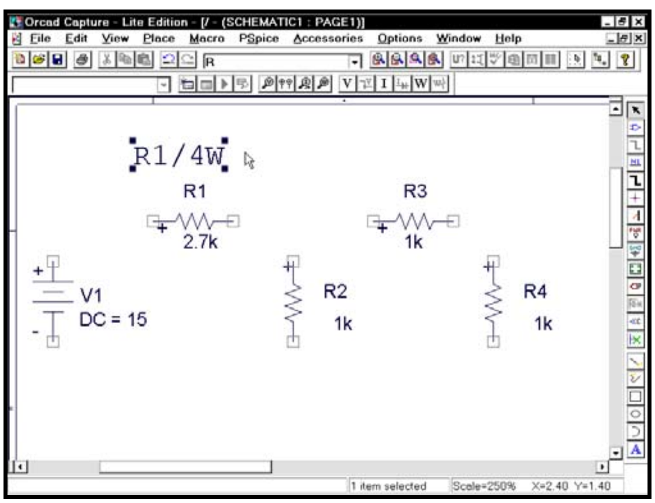
On my display, the text for R1/4W was placed in a poor location and we will move it. Click the ***LEFT*** mouse button on the text R1/4W to select it. It should turn pink and display pink handles:



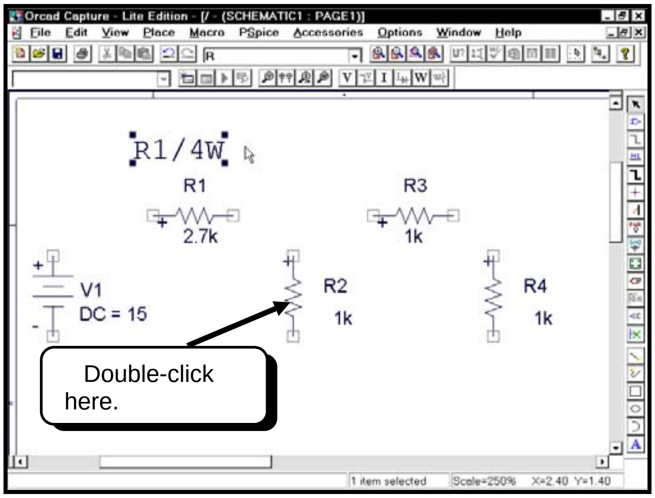
You can now drag the selected text to a new location. While dragging the text, an outline of the text will move with the mouse:

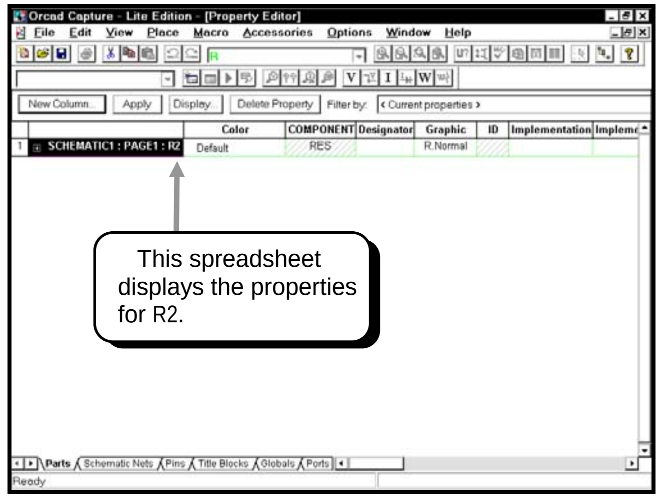


Move the outline to a convenient location and release the mouse button. The text will be moved to the new location of the box:

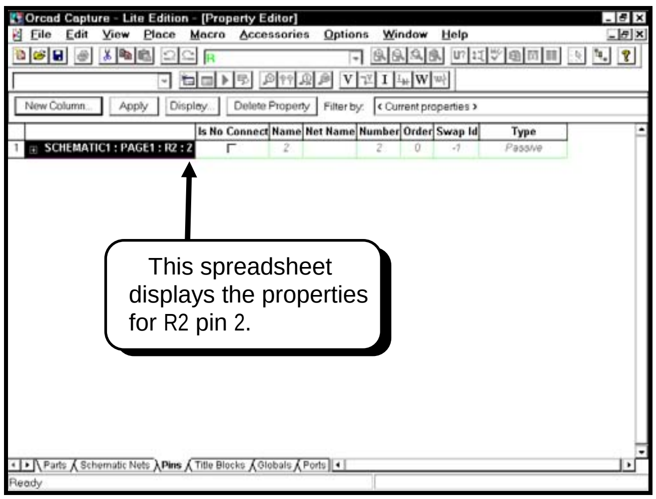


The second way to edit a part's properties is to double-click the ***LEFT*** mouse button on the graphic symbol for that part. To edit the properties of R2, double-click the ***LEFT*** mouse button on the R2 resistor graphic, . Make sure that you click on the center of the graphic. If you double-click fast enough, the properties spreadsheet will appear:

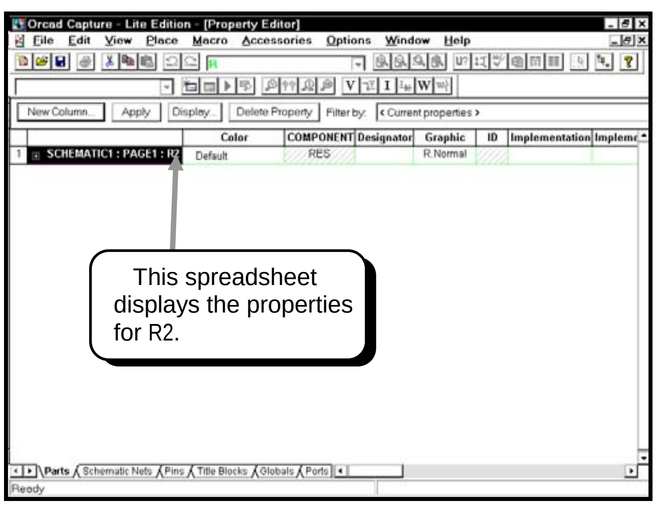




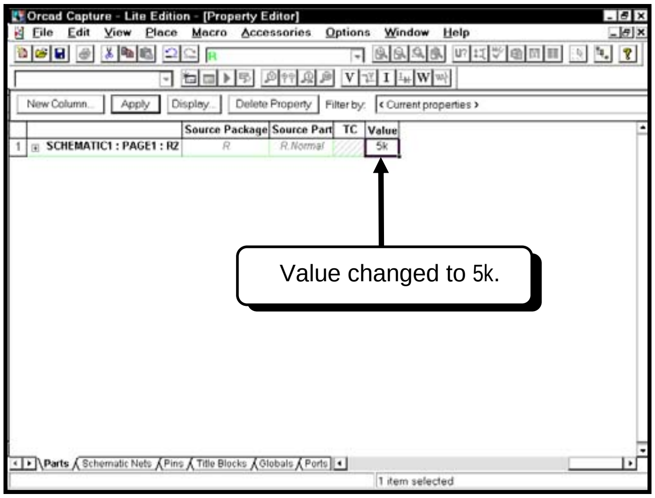
If you double-click at the wrong place, you might get the properties of another item, such as the properties of a pin as shown below:

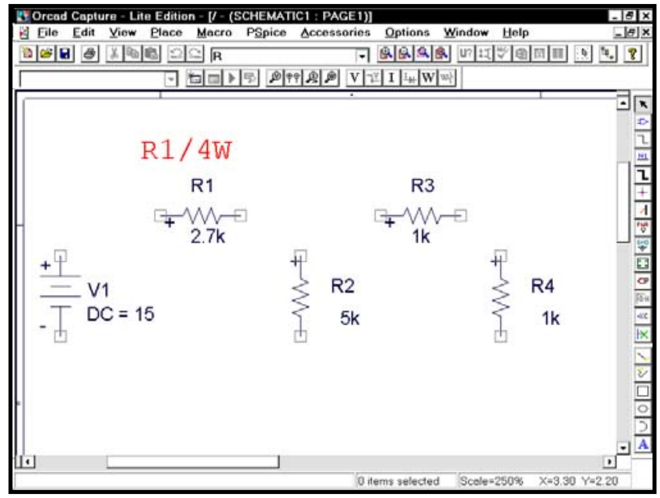


If you get the properties of the wrong item, close the spreadsheet by clicking on the lower  and try again. Keep trying until you see the spreadsheet below:

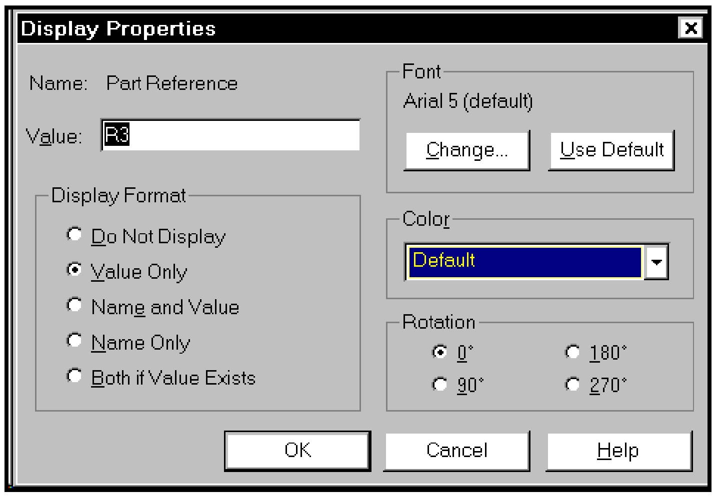


Edit the properties to change the value to **5k** and then close the spreadsheet by typing **CTRL-F4**.

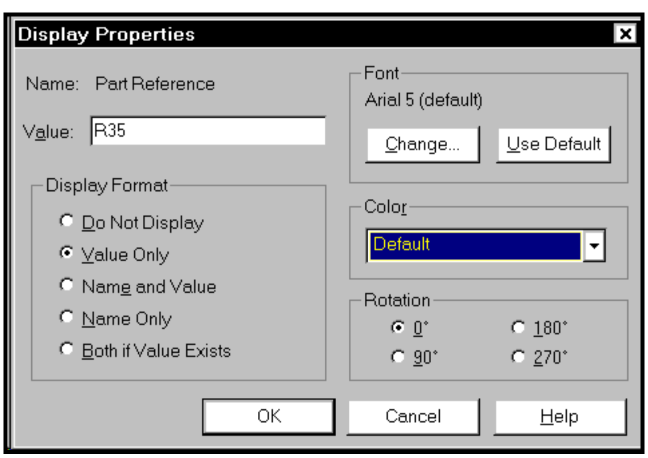


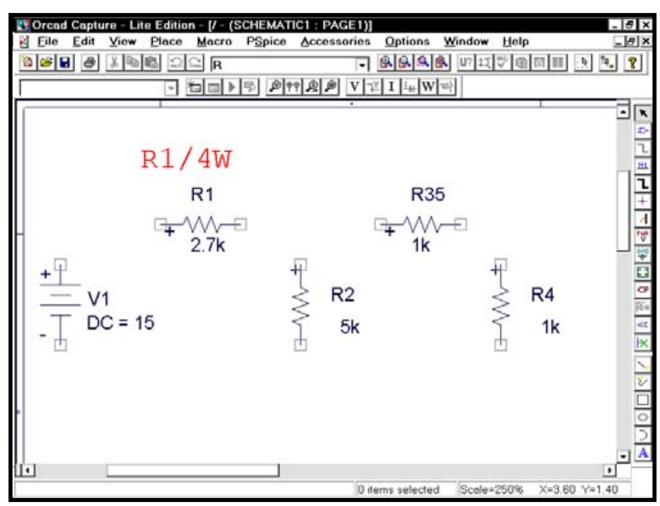


Next, we would like to change the name of R3 to R35. Double-click the ***LEFT*** mouse button on the text R3. If you double-click fast enough, the dialog box below will appear:

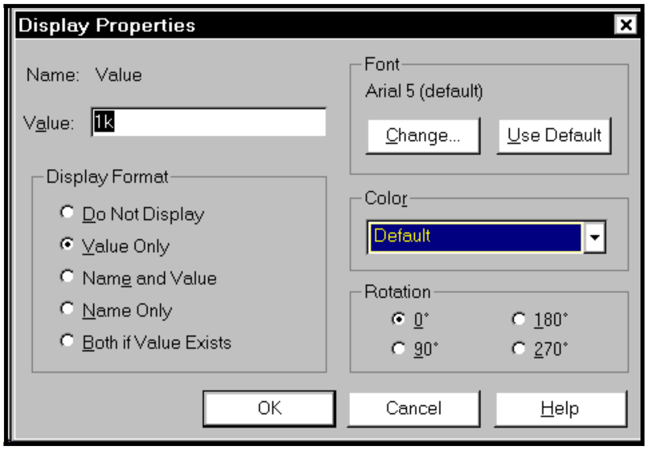


Change the name to **R35** and click the OK button. The name will be changed in the schematic.

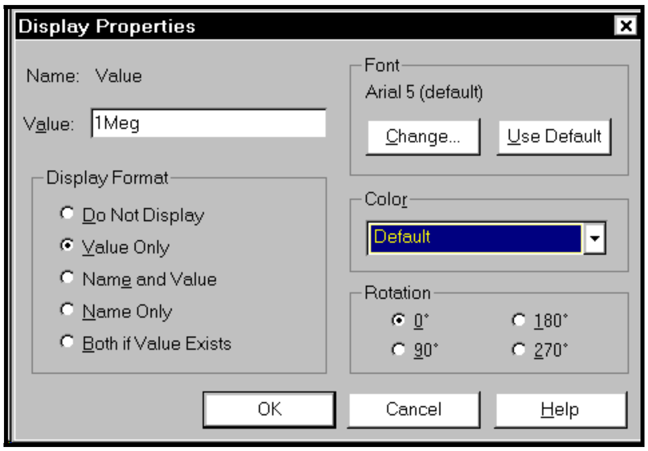




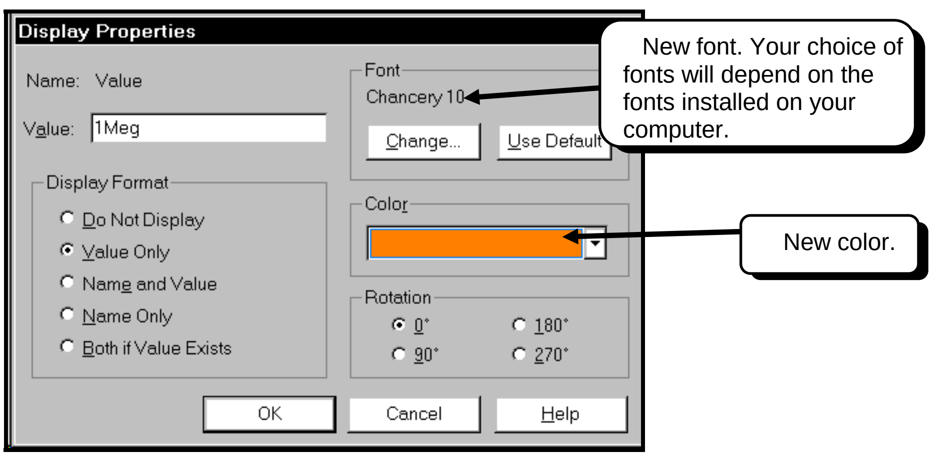
To change the value of R35, double-click on the text 1k. A dialog box will appear:



The dialog box indicates that the current value is 1k. To change the value, type in the new value, **`1Meg`**, for example:

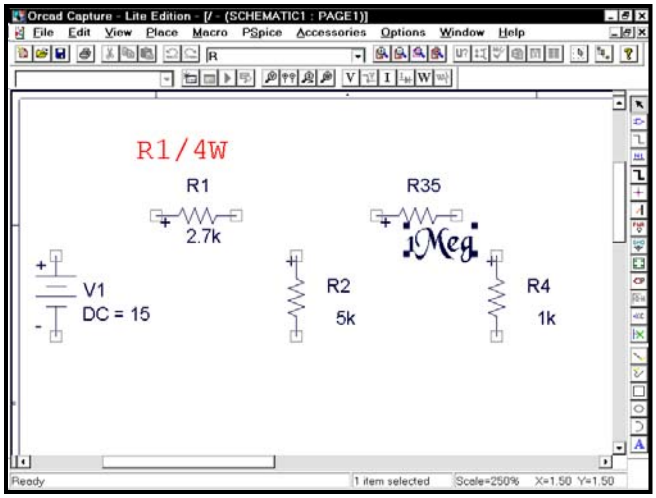


1Meg stands for 1×10^6. There must not be any spaces between the 1 and the Meg.[4] You can also change the font, font size, and font color with this dialog box:

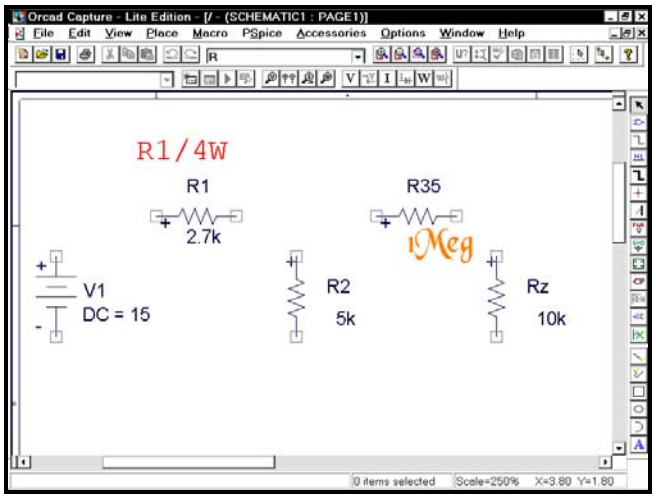


Click the OK button to accept the changes:

[4]In PSpice the multipliers m and M both refer to "milli." Thus the numbers 1m and 1M both equal 0.001.

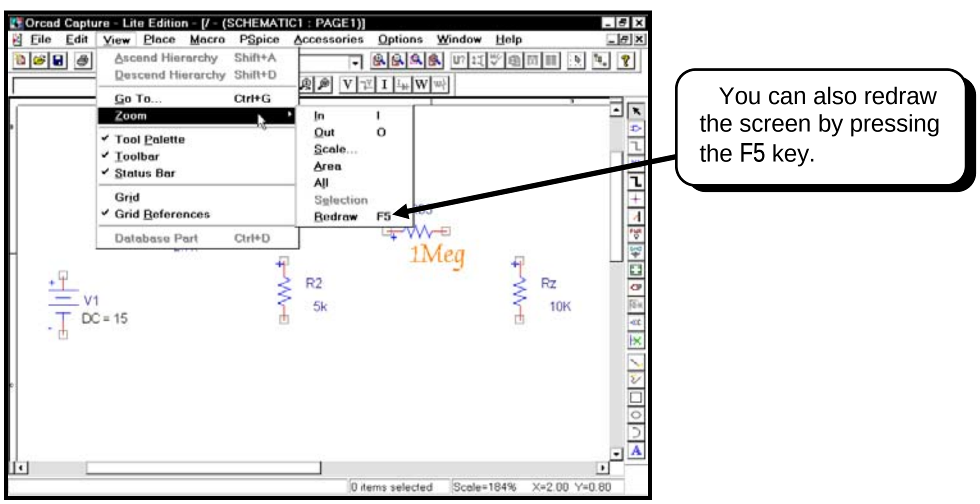


Using one of the methods given above, change the name of R4 to Rz, and the value to 10k:



We are now finished placing parts and changing properties. Using the method given on pages 24–25 to move text items, move all the necessary text to make the schematic more readable. When you are done, you should have a schematic that looks something like the previous screen.

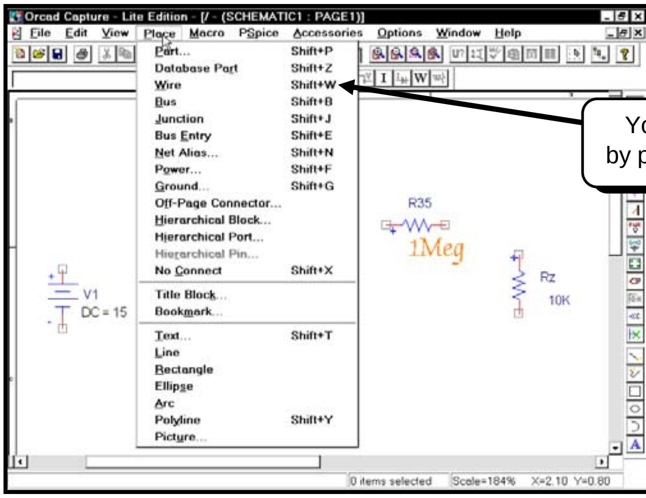
At this point your schematic may have bits of garbage floating around due to the editing. To get a fresh copy of the screen, select **View** and then **Zoom** from the menus:



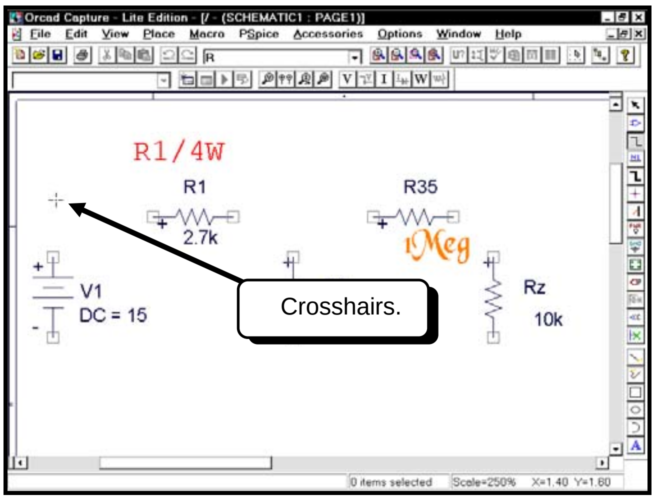
Click the ***LEFT*** mouse button on **Redraw**. The screen will be cleared and redrawn. You can also redraw the screen by pressing the F5 key.

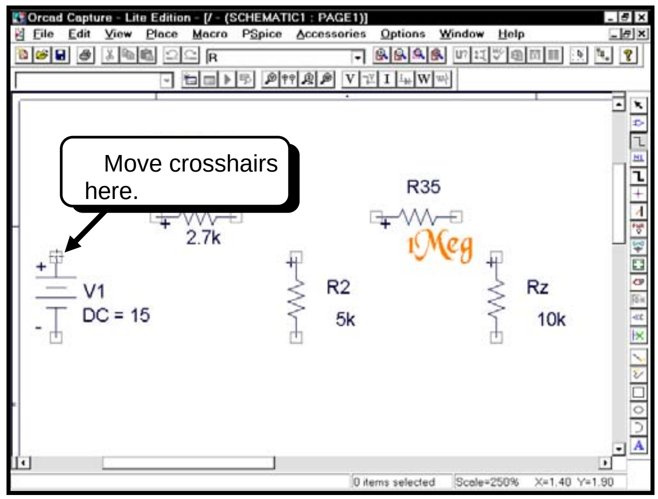
1.E. Wiring Components

We now must wire the resistors together. Click the ***LEFT*** mouse button on the **Place** menu selection:

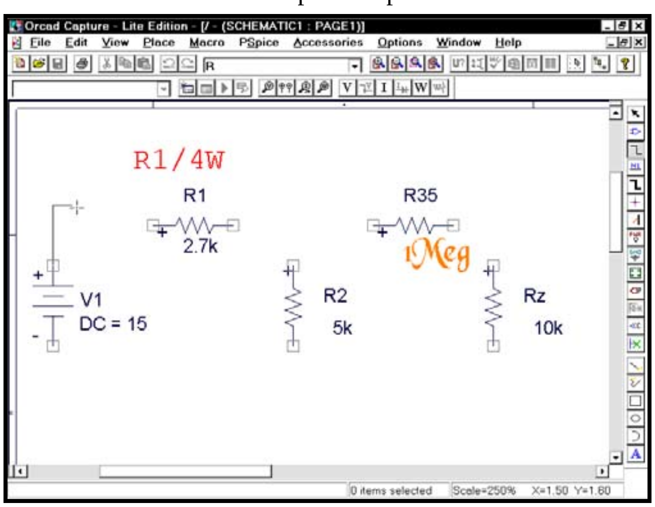


Click the ***LEFT*** mouse button on the **Wire** menu selection. Crosshairs will replace the mouse pointer. Move the crosshairs so that they point toward the top of the positive (+) terminal of the DC voltage source:

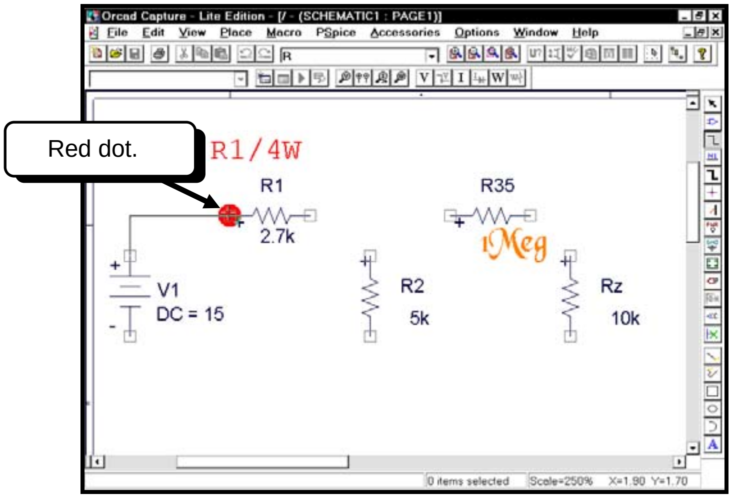




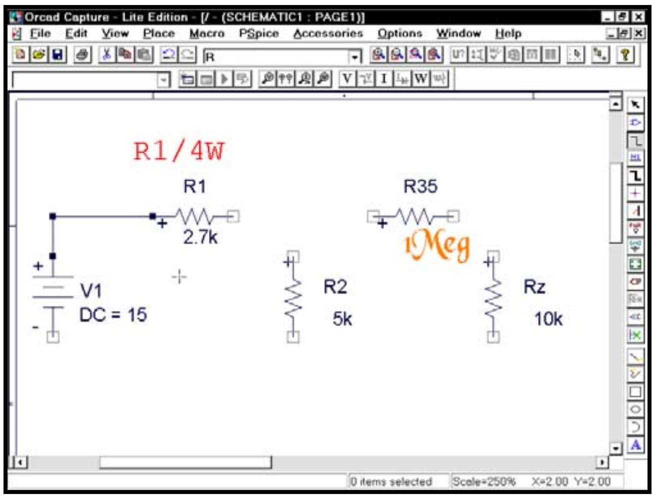
To start drawing a wire, click the ***LEFT*** mouse button on top of the positive terminal and then move the crosshairs away:



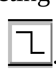
If you missed the positive terminal, press the ESC key and start over. Next, move the crosshairs to point toward the left terminal of the 2.7k resistor. A red dot should appear, indicating that you are at a pin, and will make a connection if you click the ***LEFT*** mouse button:



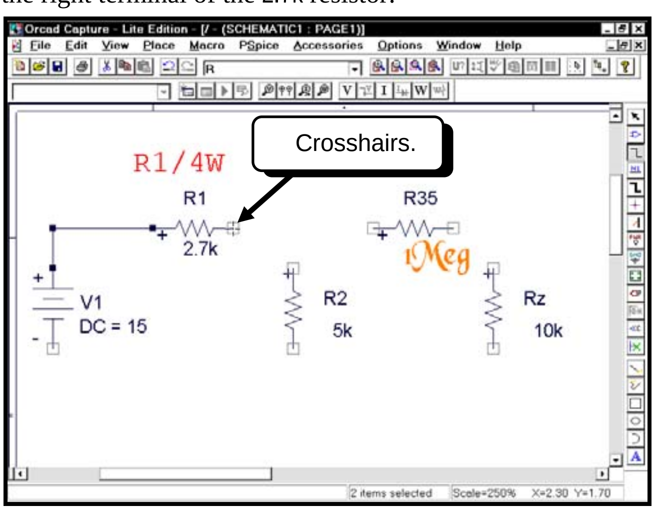
Click the ***LEFT*** mouse button to make the connection:



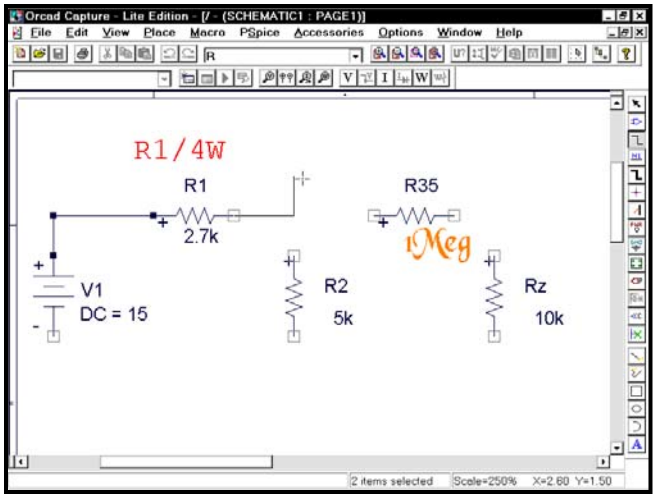
The schematic shows that the voltage source and the 2.7k resistor are now wired together. The schematic also shows that the crosshairs are still on the screen, indicating that we are still drawing wires. If your cursor is not shown as crosshairs, you can start drawing wires using three methods:

1. Select **Place** and then **Wire** from the Capture menus.
2. Press the W key. This is the keyboard shortcut for selecting **Place** and then **Wire** from the menus.
3. Click the ***LEFT*** mouse button on the draw wires icon, .

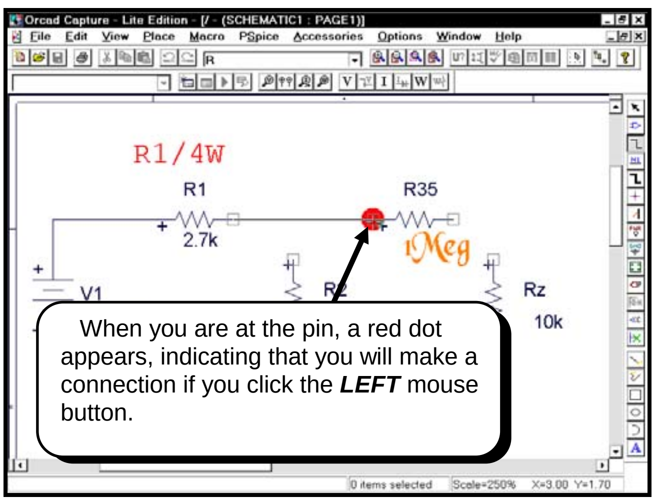
Move the crosshairs to the right terminal of the 2.7k resistor:

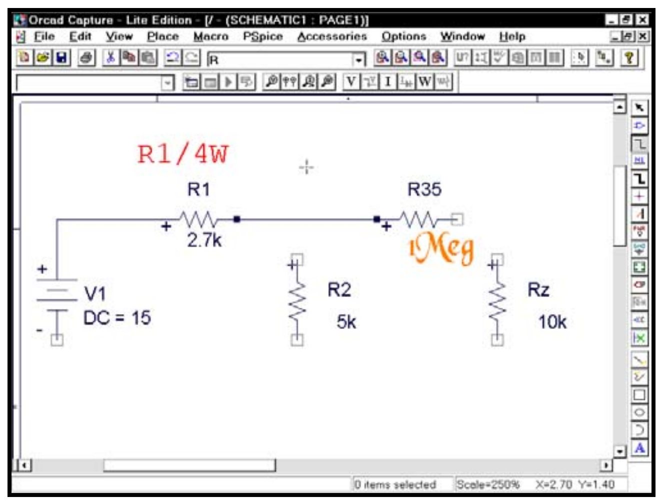


Click the ***LEFT*** mouse button and then move the mouse. You should have a wire connected to the right terminal of the 2.7k resistor:

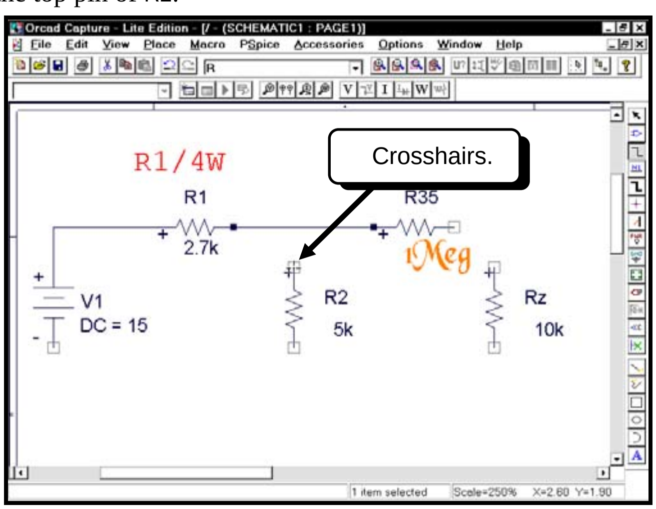


Move the crosshairs to the left terminal of R35 and click the ***LEFT*** mouse button to make the connection:

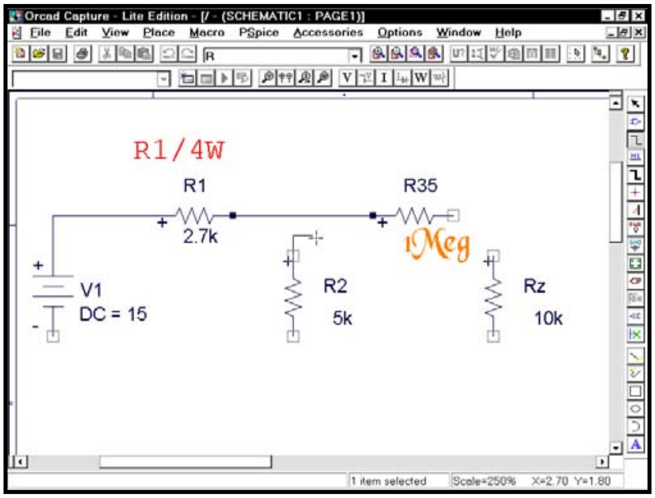




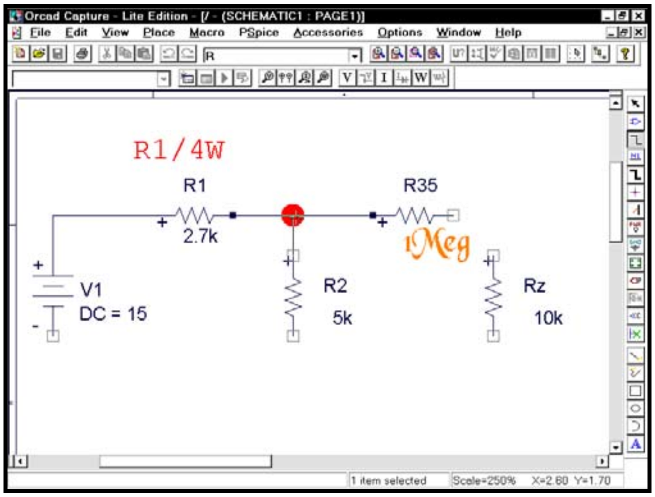
Note that the crosshairs are still displayed instead of the mouse pointer. This indicates that we can continue drawing wires. Move the crosshairs to point at the top pin of R2:



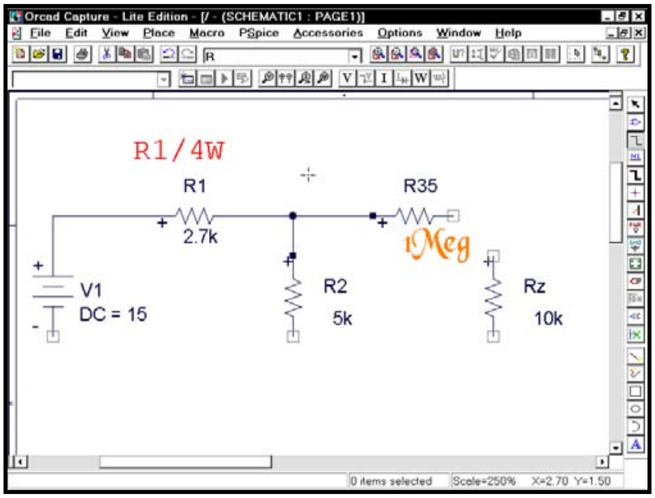
Click the ***LEFT*** mouse button to start drawing a wire and then move the crosshairs up. A wire should join with the top pin of R2:



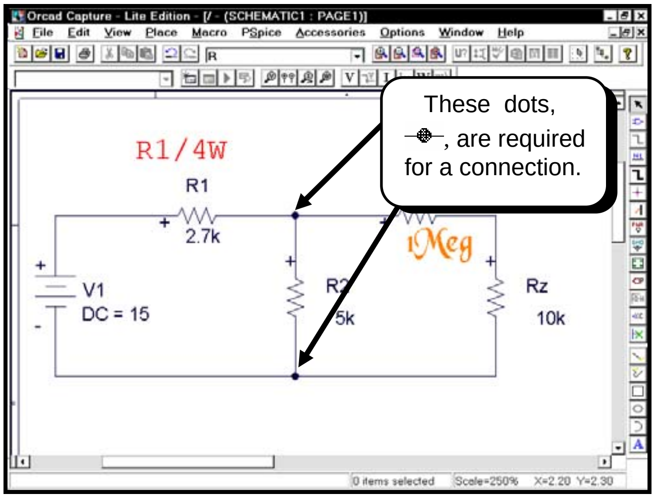
Move the crosshairs to the wire connecting R1 and R35. A red dot should appear, indicating that you will make a connection if you click the ***LEFT*** mouse button.



Click the ***LEFT*** mouse button to make a connection and then move the mouse away. You should now have the schematic shown below:



Continue wiring until you obtain the circuit shown below:



To stop drawing wires, press the **ESC** key.

Note in the circuit above that some of the connections have a dot, . A dot indicates a connection. It is not necessary to have a dot present when a wire joins a pin. Dots are always drawn when wires meet in a "T," . If two wires cross and do not display a dot, then the wires are not connected. If two wires cross and display a dot, then the wires are connected.

1.F. Correcting Wiring Mistakes

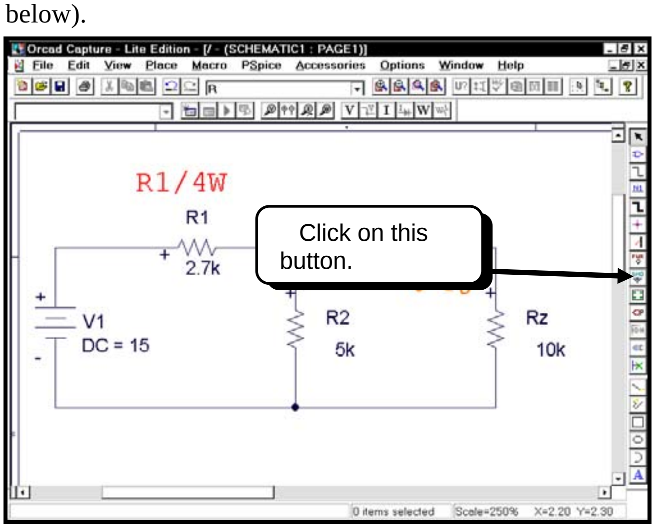
If you made a mistake drawing a wire, you can use the following procedure to delete unwanted wires, as well as components.

1. Make sure you are not in "wire" mode. If you are in "wire" mode, the mouse pointer is displayed as crosshairs. To terminate "wire" mode, press the **ESC** key.
2. Move the mouse pointer to the segment of wire or the part that you wish to remove.
3. Click the ***LEFT*** mouse button on the wire or part you wish to remove. This will select the wire or part. When the wire or part has been selected, it will turn pink.
4. Press the DELETE key to delete the selected wire or part.

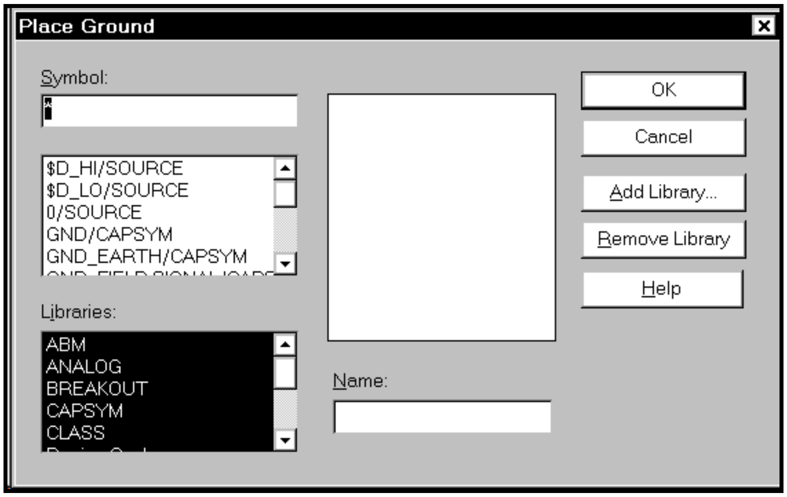
1.G. Grounding Your Circuit

To run a circuit on PSpice, you must have at least one ground connection in your circuit. To ground your circuit, you must place a symbol called "0" (the number zero). There are three ways to place ground symbols:

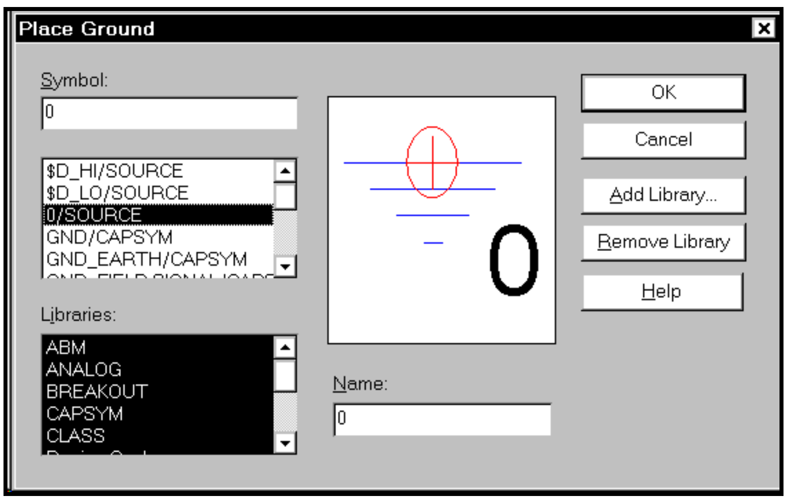
1. Select **Place** and then **Ground** from the menus.
2. Press the **G** key.
3. Click the ***LEFT*** mouse button on the ground button, GND, displayed in the toolbar on the right side of the Capture window (button shown below).



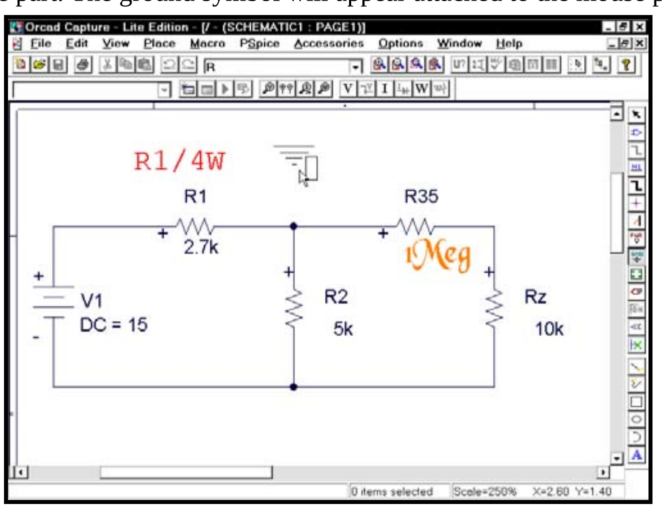
When you use any of these methods, the Place Ground dialog box will be displayed:



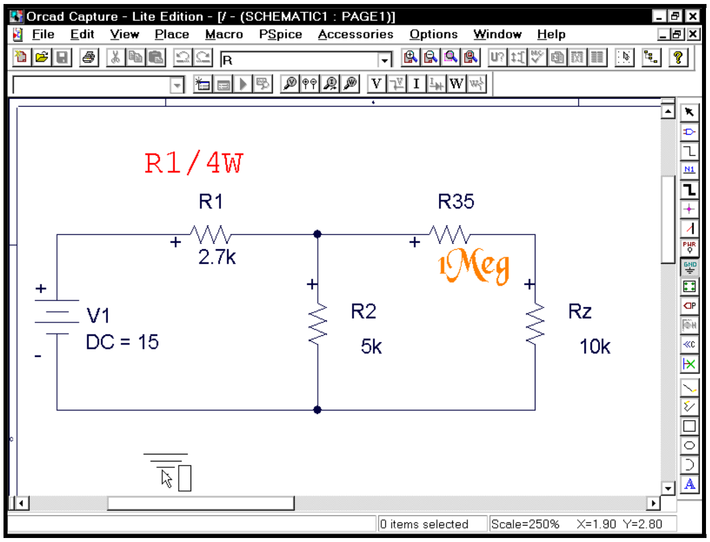
Capture has many ground symbols available for schematic drawing purposes. **Only the 0/SOURCE ground can be used for PSpice simulations.** Select the 0/SOURCE part:



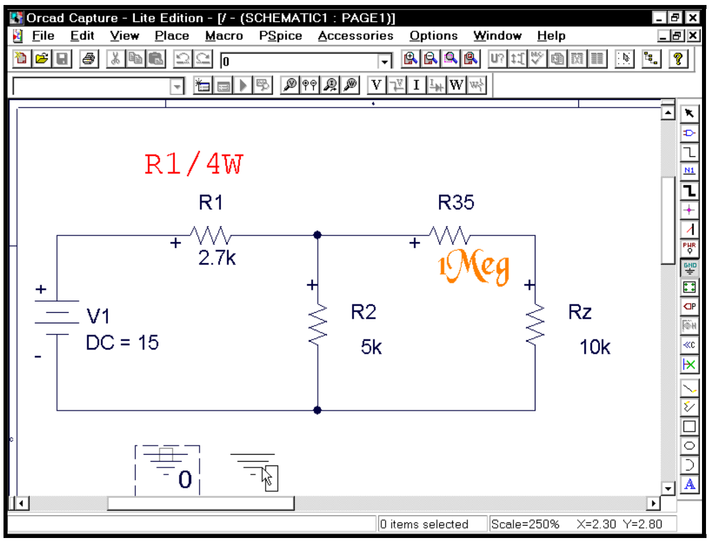
Click the OK button to accept the part. The ground symbol will appear attached to the mouse pointer:



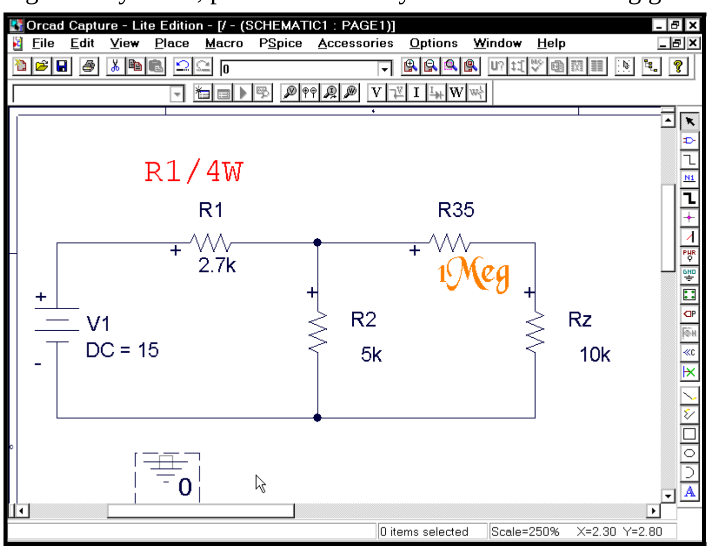
Note that the ground symbol moves with the mouse. Move the ground symbol as shown below:



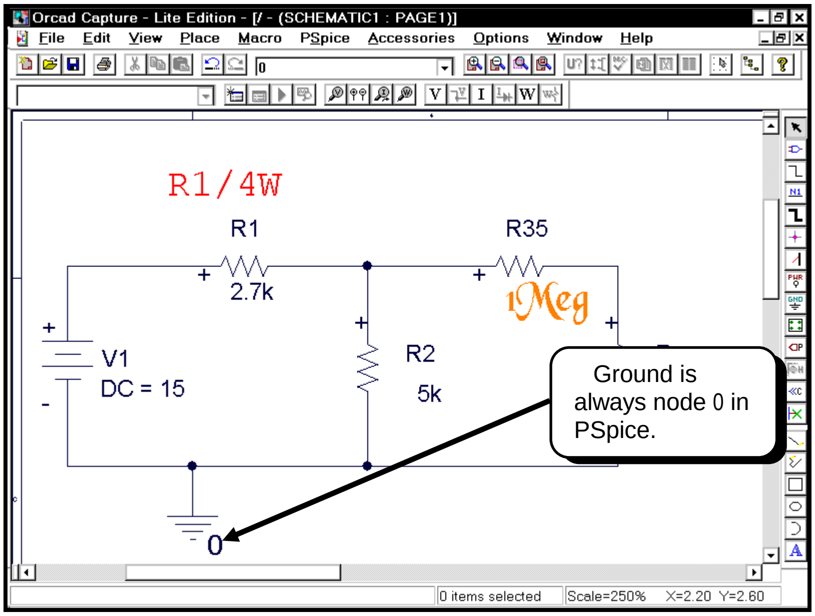
To place the ground symbol, click the ***LEFT*** mouse button. The ground symbol is placed on the schematic and, when you move the mouse, a second ground symbol appears attached to the mouse pointer:



Since we do not need any more ground symbols, press the **ESC** key to terminate drawing ground symbols.



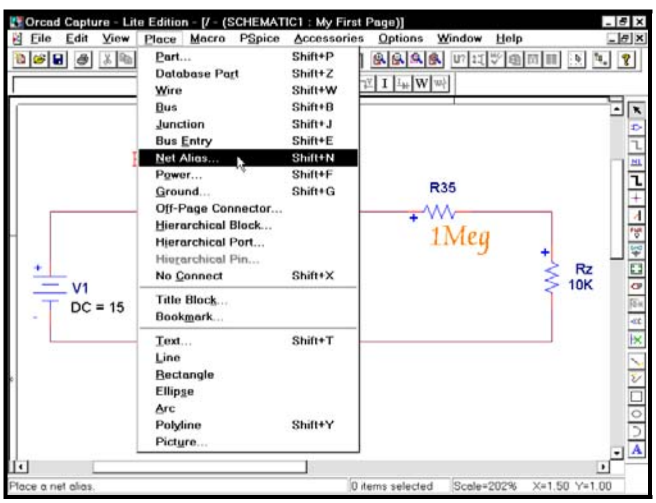
You must now connect the ground symbol to the circuit by connecting a wire from the ground to the circuit. Press the **W** key to start drawing wires and wire the circuit as shown below. To stop drawing wires, press the **ESC** key.

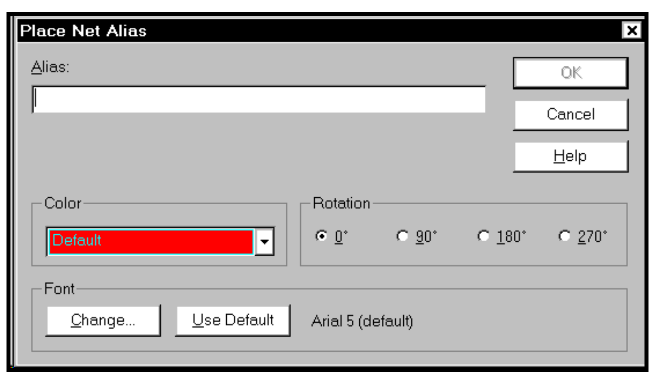


1.H. Labeling Nodes

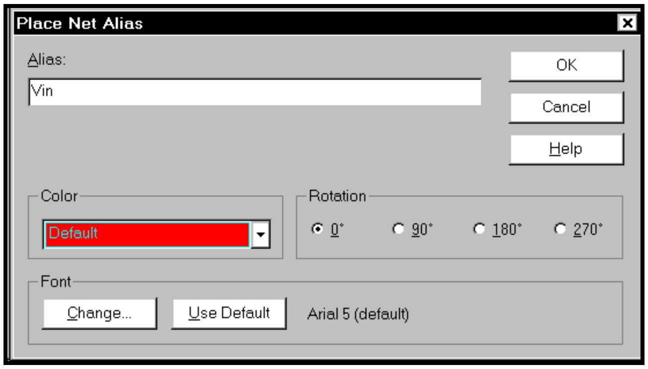
Capture arbitrarily labels nodes with names such as N01015 or N01985, and users do not know the names unless they double-click on a wire to view the name. This is normally not a problem because we are usually not concerned with probing all of the nodes of a circuit. However, there are usually a few nodes that we would like to look at, and if we do not know the names of those nodes, viewing the voltages at that nodes is more difficult. Capture provides three methods for labeling a node: placing a net alias on a wire, connecting a power connector to a node, or connecting an offpage connector to a node. All methods have the same effect of naming a node. A net alias places a text label next to a wire, while power and offpage connectors place a graphic on your schematic and then you can label the connector. Note that if you give two nodes the same name, those two nodes will be connected together. This is a convenient way of connecting parts, but if you make a mistake and accidentally give two nodes the same label, those nodes will be connected together and your circuit will not operate properly.

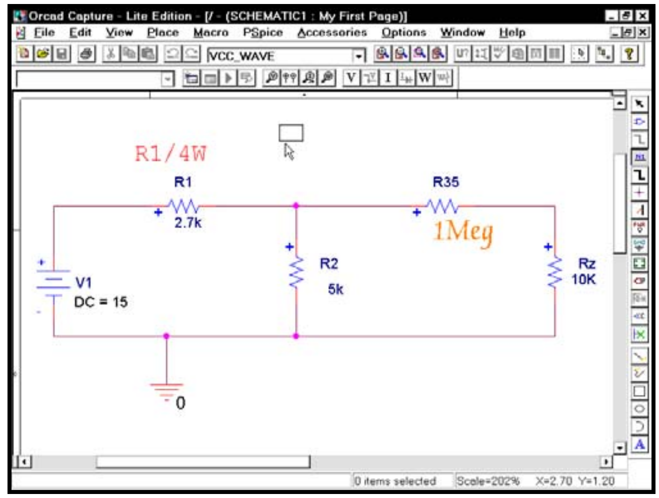
We will first label a node using a net alias. Select **Place** and then **Net Alias** from the menus:



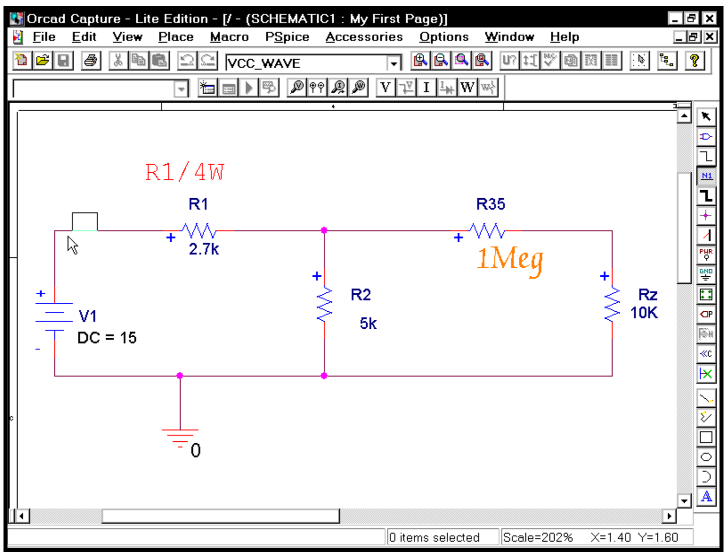


Enter a name for the alias and then click the OK button. An outline for the alias will be attached to the mouse pointer:

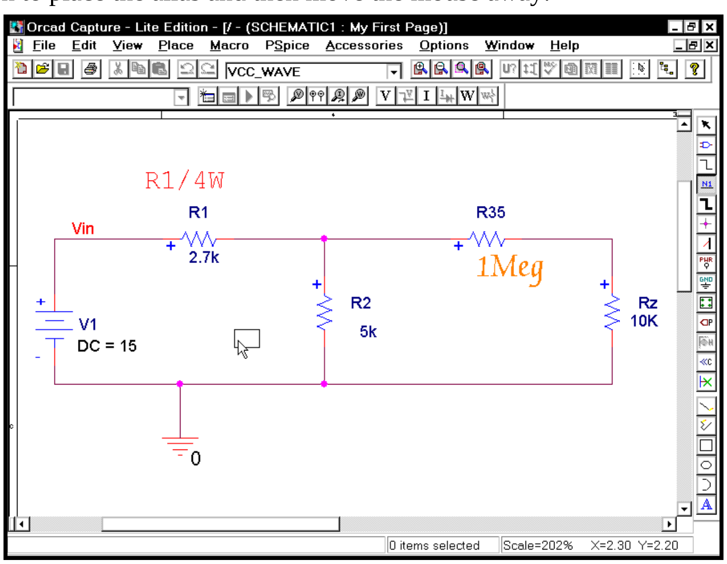




Place the outline next to the horizontal wire at the input:

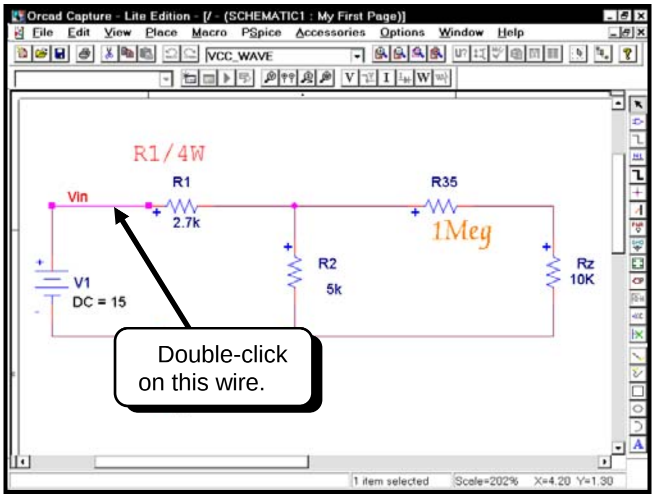


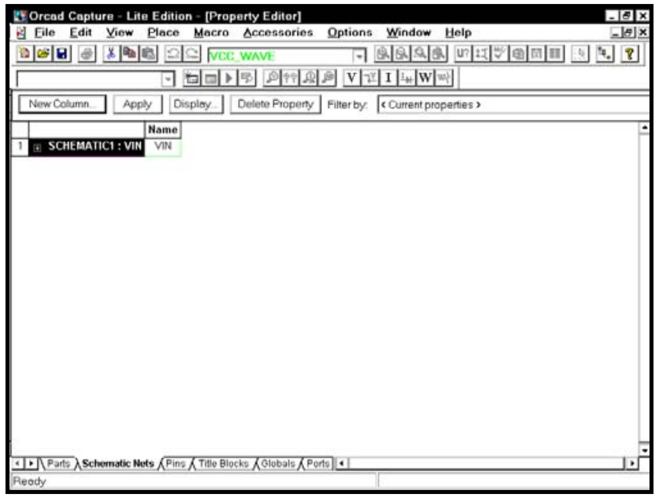
Click the ***LEFT*** mouse button to place the alias and then move the mouse away:



Press the **ESC** key to stop placing net aliases. This node is now named Vin.

If you double-click the ***LEFT*** mouse button on the wire between V1 and R1, the name of that node will be listed:



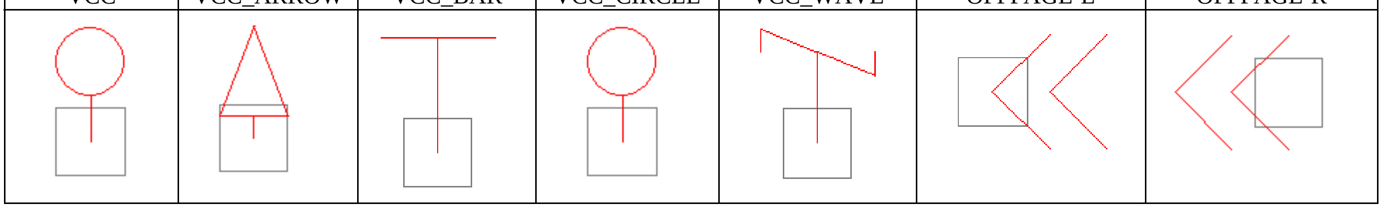


We see that the name of the wire is now Vin. Type **CTRL-F4** to close the spreadsheet.

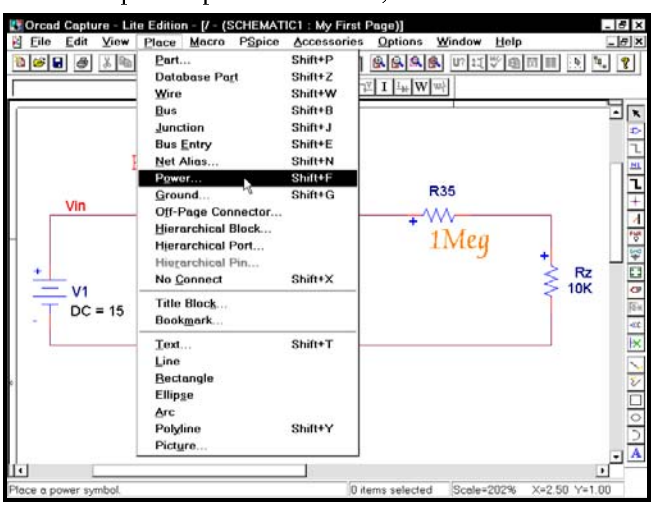
Capture provides seven different graphics symbols that we can use to label nodes. The names of the parts and their symbols are shown in **Table 1-1**.

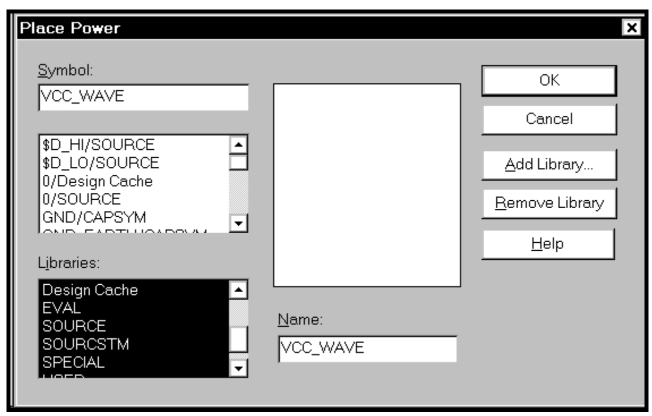
Table 1-1

VCC	VCC_ARROW	VCC_BAR	VCC_CIRCLE	VCC_WAVE	OFFPAGE-L	OFFPAGE-R

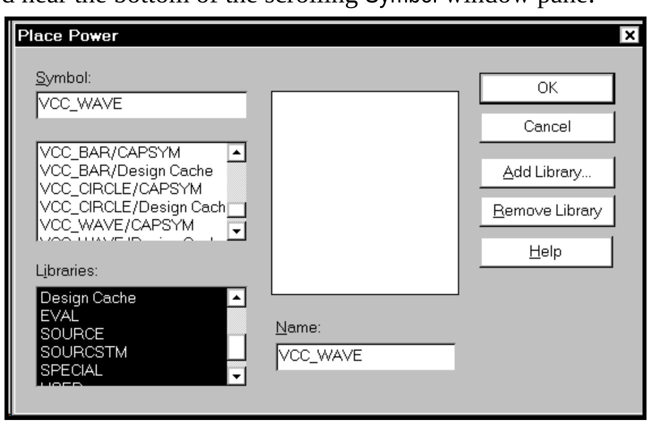


To place a power connector, select **Place** and then **Power** from the menus:

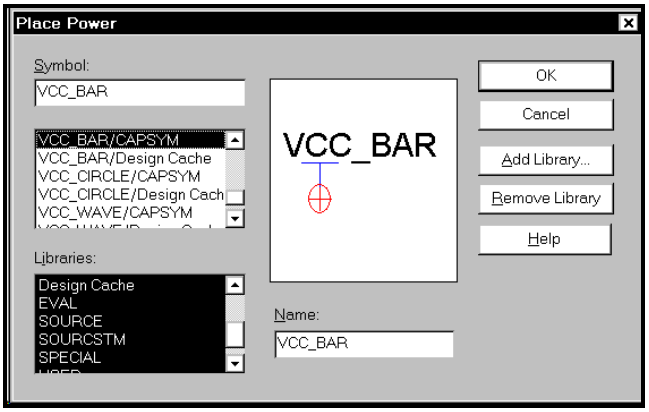


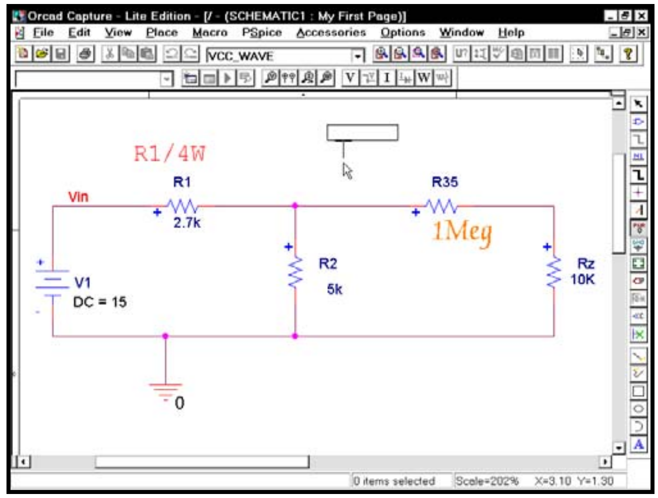


The power connectors are located near the bottom of the scrolling Symbol window pane:

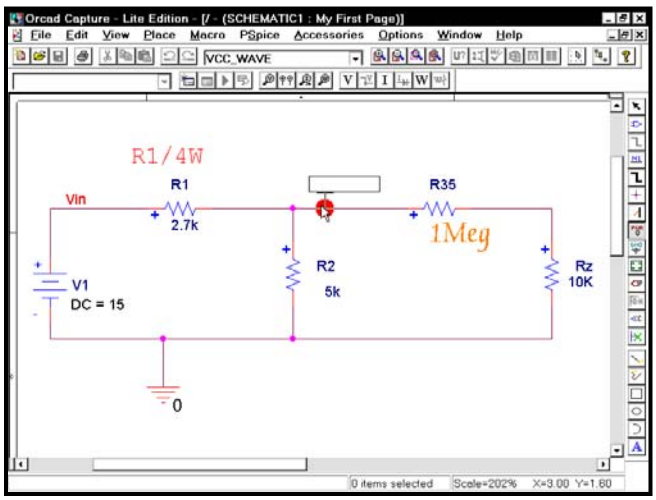


Click the ***LEFT*** mouse button on one of the power connectors, VCC_BAR for example, and then click the OK button. The graphic for the power connector will become attached to the mouse pointer.

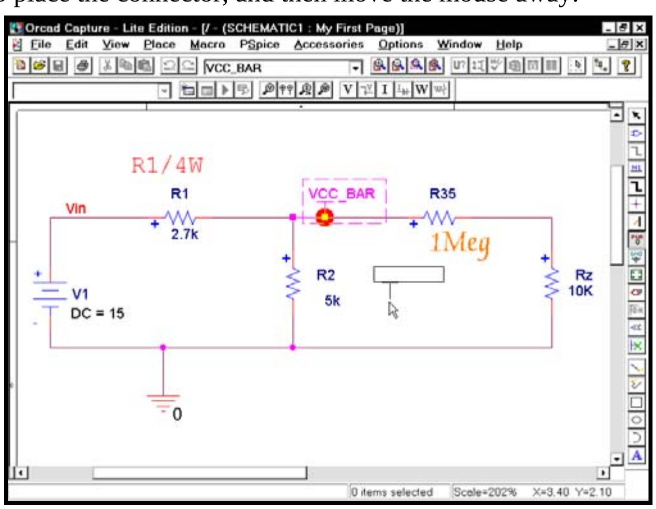




Place the power connector next to a wire at the center node. A red dot will appear where a connection will be made between the connector and the wire:

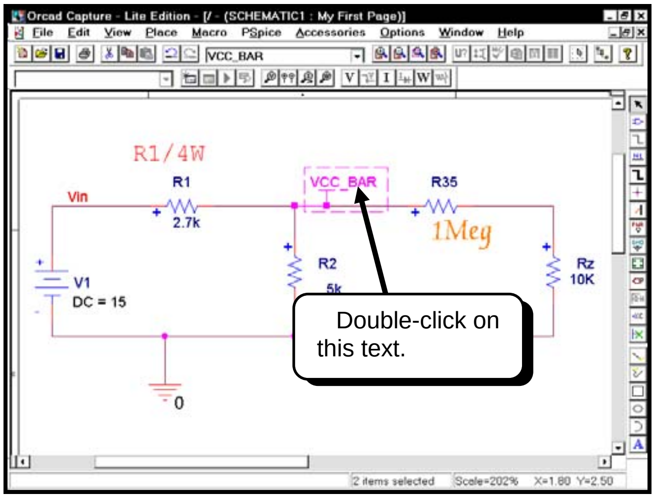


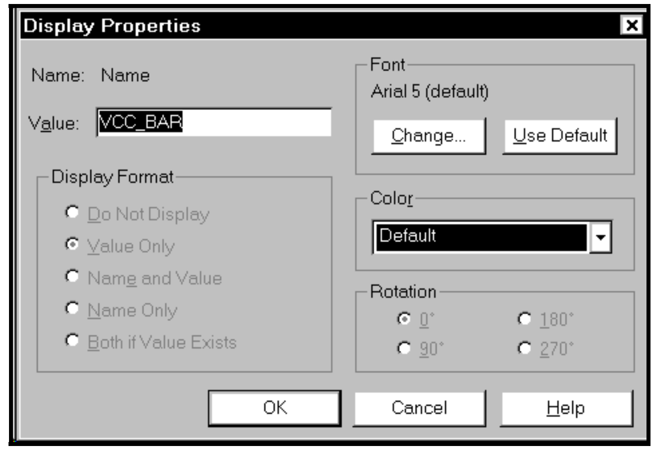
Click the ***LEFT*** mouse button to place the connector, and then move the mouse away:



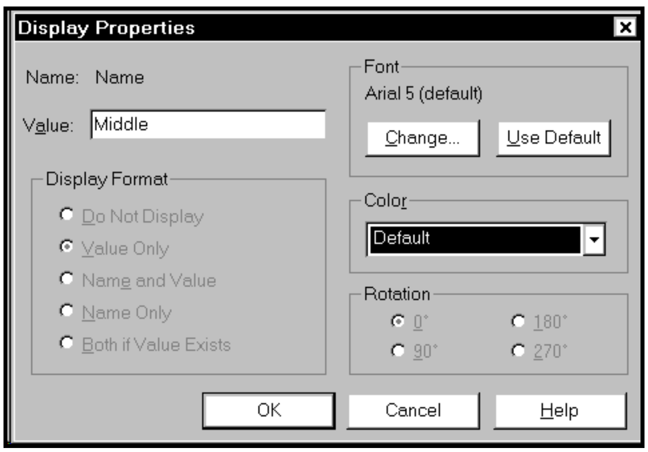
A second power connector is now connected to the mouse pointer. Press the **ESC** key to stop placing power connectors.

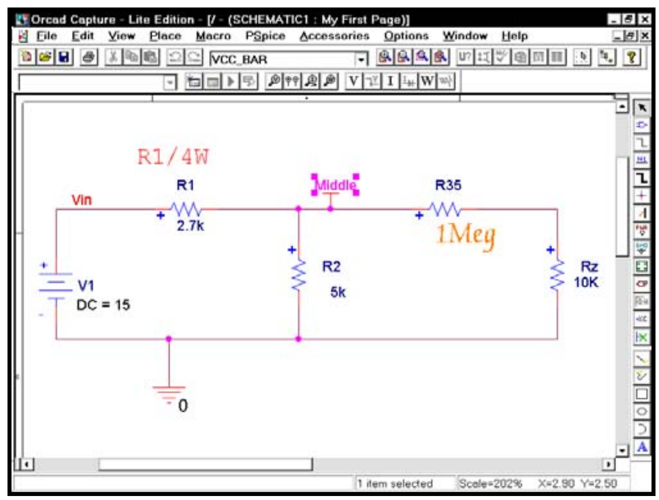
We must now change the label. Double-click on the text VCC_BAR to change it:





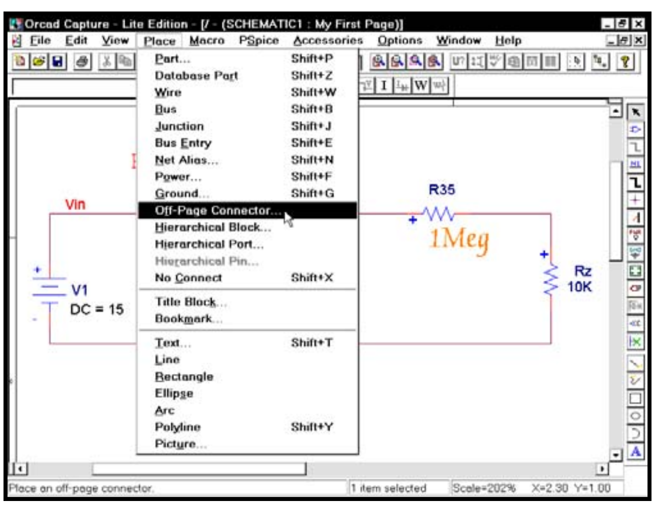
Enter a new name for the label and click the OK button:

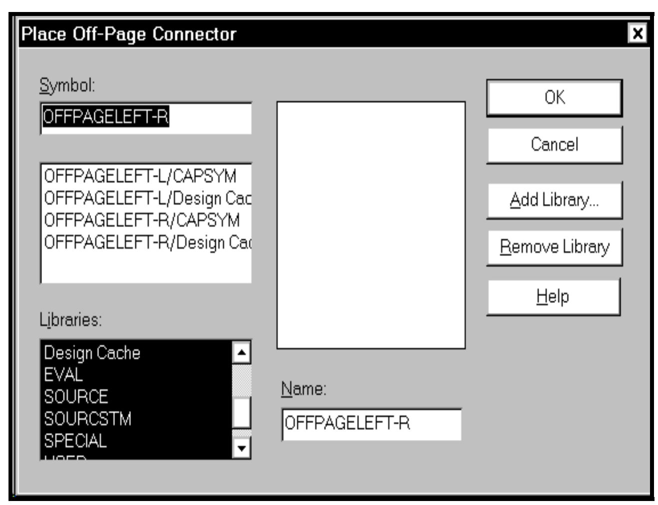




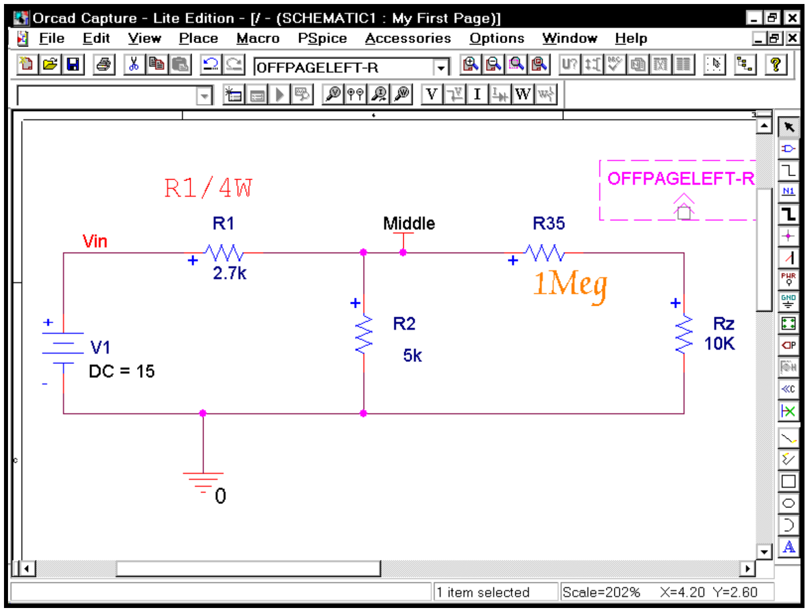
The center node is now labeled Middle.

To place an offpage connector, select **Place** and then **Off-Page Connector** from the menus:

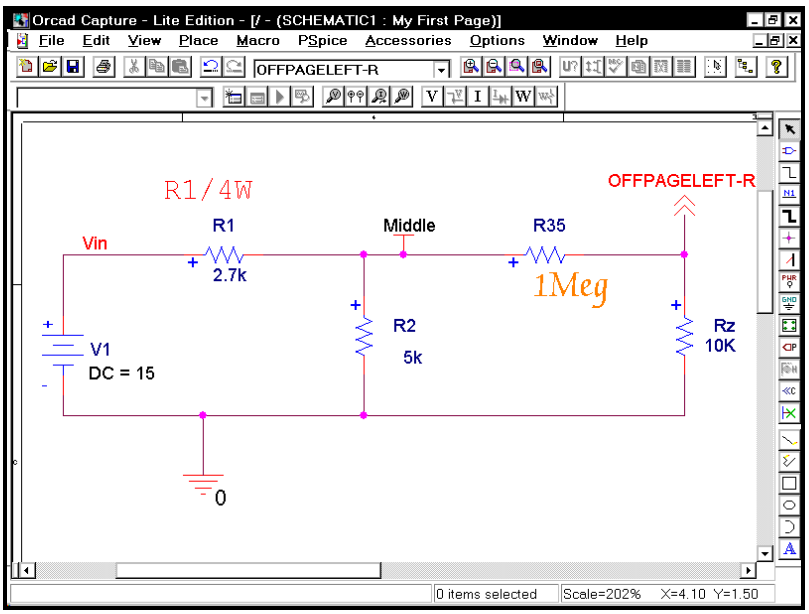




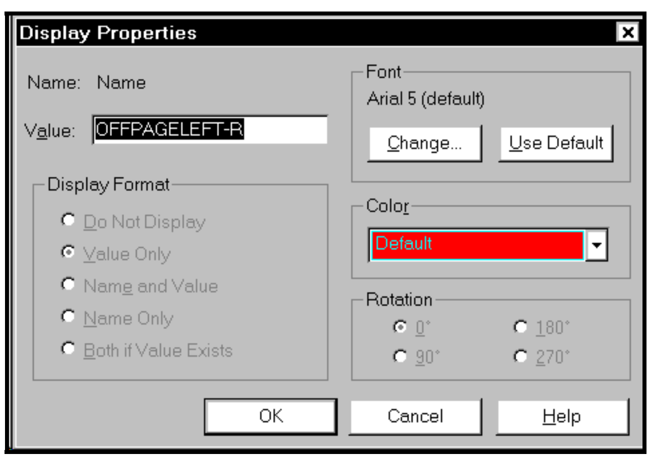
Select a connector and then place the connector as shown:



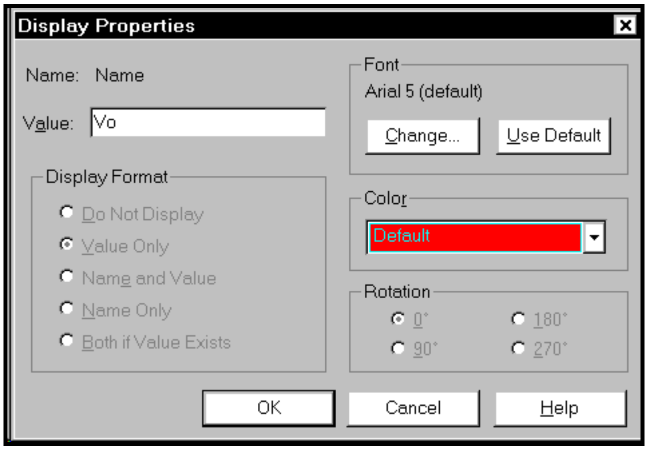
Wire the connector to your circuit:

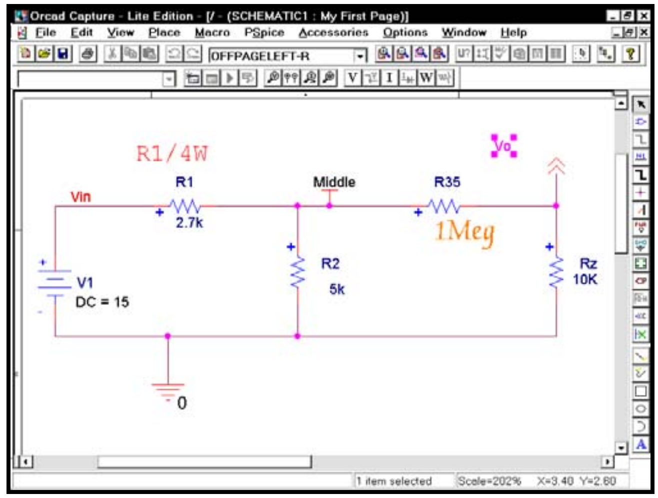


To change the label, double-click on the text OFFPAGELEFT-R:

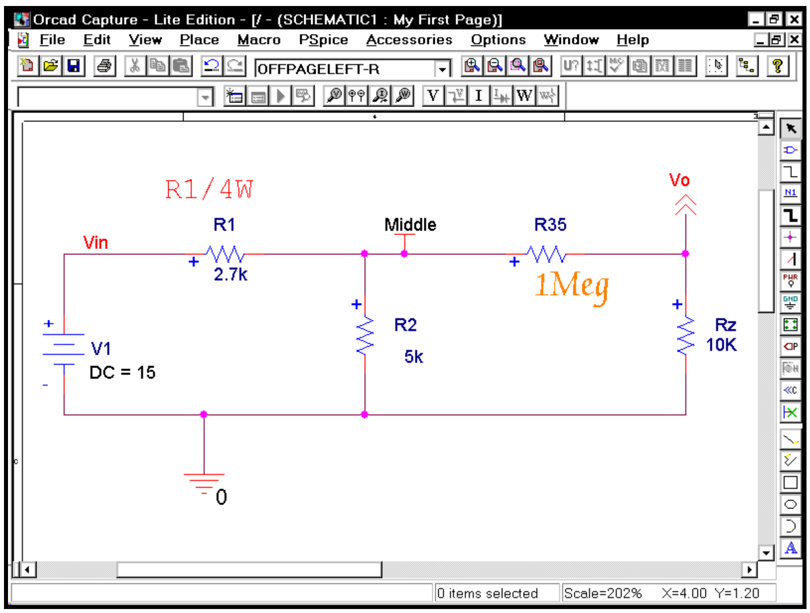


Type the text **Vo** to change the label to Vo and then click the OK button:





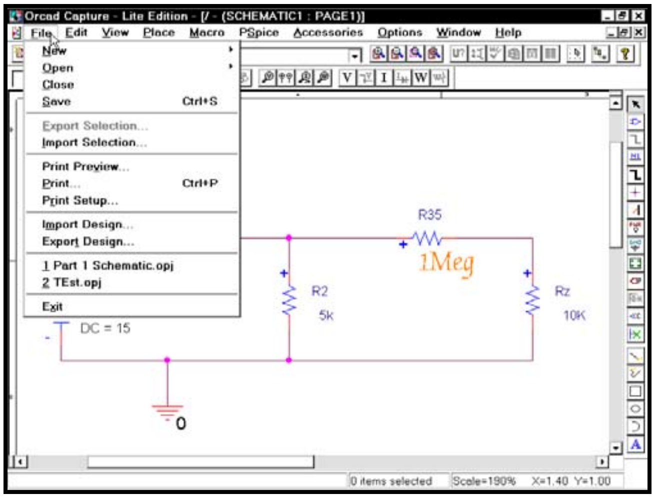
You will need to drag the text Vo closer to the offpage connector:

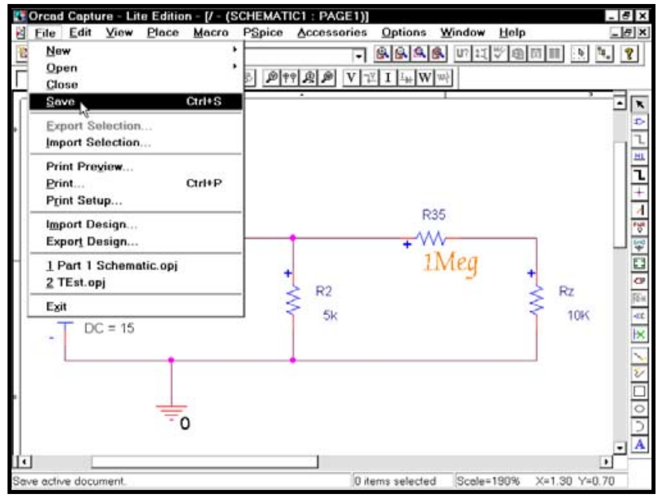


1.I. Using the Cache to Place Parts

1.J. Saving Your Schematic

Now that we have finished the schematic, we need to save it. Select **File** and then **Save** from the menus. The **File** pull-down menu will appear:





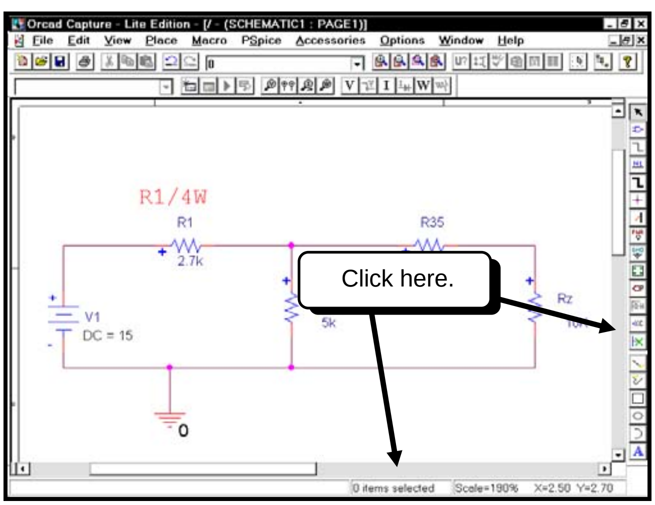
When we first created the project, we specified a name and directory location for the project, so we do not need to specify them here.

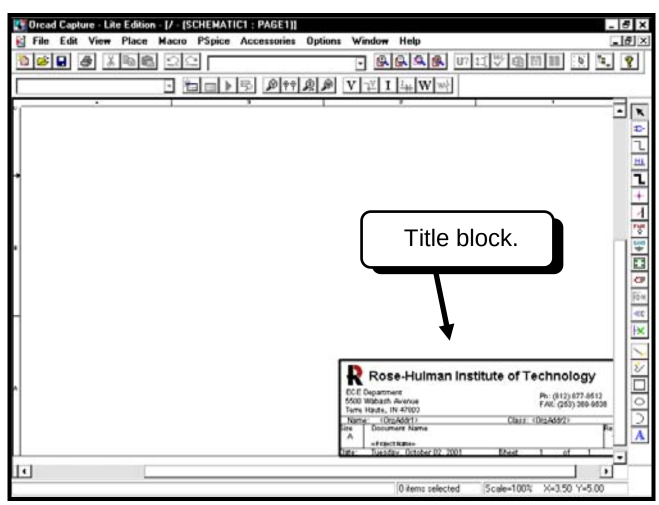
1.K. Editing the Title Block

The title block contains information about you, your company, and the schematic. We will be using the title block to identify the student, class, and university where the schematic was created. There are four ways we can modify the title block. The first way shows how to change the title block for the current schematic. This method must be repeated each time you create a new schematic. The second way shows you how to tell Capture information about yourself that is used in every schematic you create. This could be information such as your name, company, address, and phone number. The third method shows how to select a different title block from a list of available title blocks already created. The fourth method is to create a new title block and save it in a symbol library.

1.K.1. Modifying the Title Block in the Current Schematic

The first thing we want to do is place your name and class information on your schematic. In the bottom right corner of your schematic is a title block. To see this block we must scroll the view of the page to the bottom right corner. To do this, click the ***LEFT*** mouse button on the vertical and horizontal scroll bars as shown below:





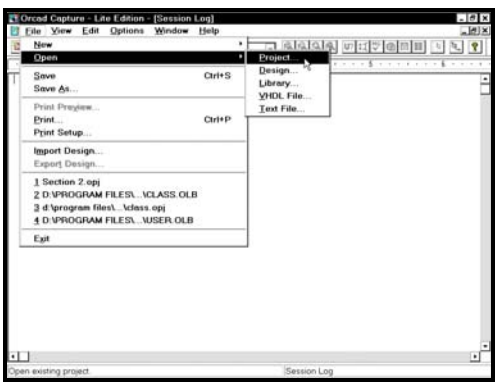
This is a title block that I created for this text and my students. In Section 1.K.4 we will show you how to customize this title block for your needs. There are three items that we can change: <OrgAddr1>, <OrgAddr2>, <Revision>, and <Project Name>. To edit an item, double-click the ***LEFT*** mouse button on the item. For example, double-click the ***LEFT*** mouse button on the text <OrgAddr1>

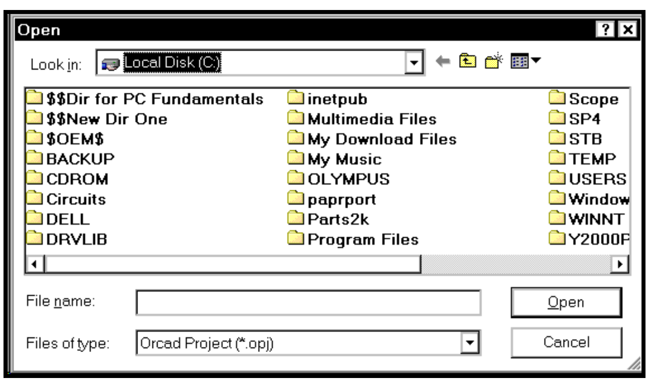
PART 2
Introduction to Probe

Probe is a program that will display the results obtained from PSpice graphically. We will be using Probe extensively throughout this manual to display the results of simulations. Various aspects of Probe are discussed in sections throughout this manual. However, if you pick specific examples you may miss those showing how to use some of the tools provided by Probe. The result may be that a section in the manual you are currently using refers to a tool in Probe that was discussed in a section that you did not cover. To avoid this problem, we will review how to use the most frequently used tools.

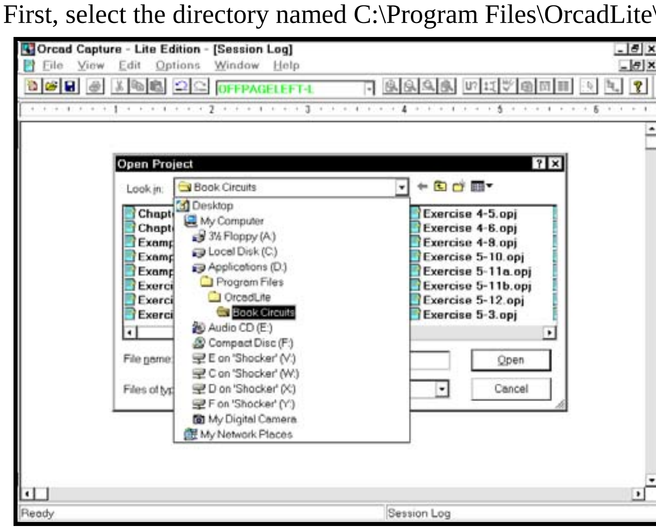
To demonstrate Probe, we will simulate a power supply circuit with a Transient Analysis. Although this may be too complicated a circuit for beginning students, the methods discussed in Probe can be used with any analysis. We are using this simulation because it provides many interesting waveforms. Open the project named Section 2.opj. This file is located on the CD-ROM that accompanies this text and may also be loaded on your hard drive. During installation of the library files, the circuit files are loaded into a subdirectory named “Book Circuits” in the Orcad Lite installation directory. If you chose the default installation options, this directory will be named C:\Program Files\Orcadlite\Book Circuits. You can use the files in this directory, or the files located on the CD-ROM. **Note: If you attempt to run this example directly from the CD-ROM, or open the file from the CD-ROM and then use the Save As command to save the file to the hard drive, you will get a write error.** To use the circuit from the CD-ROM, use the Windows Explorer to copy the file from the CD-ROM to your hard drive. Then use the Windows Explorer to change the properties of the file to not read-only. To change the properties with the Windows Explorer, run the Explorer and select the file. Next, select **File** and then **Properties** from the Explorer menus. You can then remove the checkmark from the box next to the text Read-Only. Note that the read-only property may or may not be selected when you first look at the properties and may not need to be changed. For more information on copying a single file from the CD-ROM, and then opening the file, see Section A.3 on page 621.

For this example, we will assume that you are opening the files that were copied to your hard drive during the installation. Select **File**, **Open**, and then **Project** from the Capture menu bar:

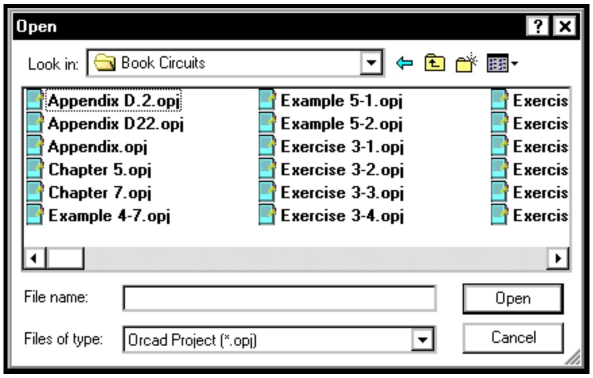




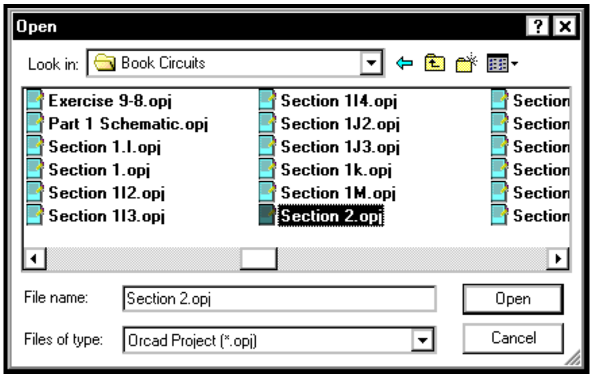
Select **Open**. We will assume that you installed Capture on drive C: in directory \Program Files\OrcadLite. (This was the default installation location.) For this installation directory the example circuits will be located in directory C:\Program Files\OrcadLite\Book Circuits. First, select the directory named C:\Program Files\OrcadLite\Book Circuits:



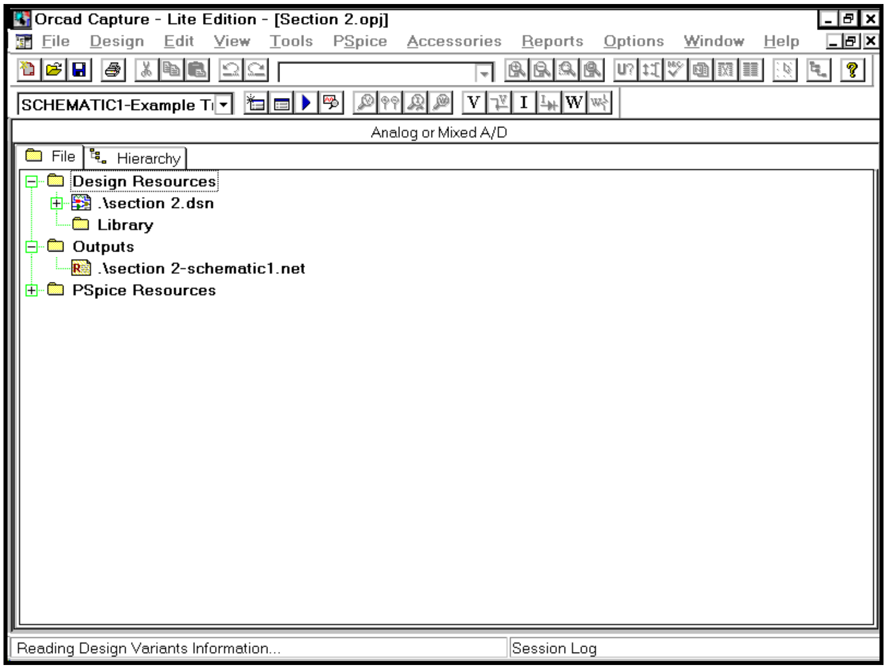
After you select the directory Book Circuits, a dialog window will display the files in that directory:

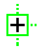


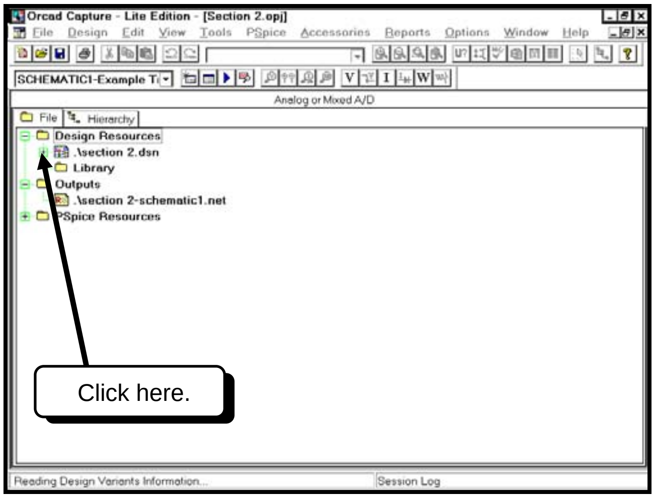
We would like to open the file named Section 2.opj so click on the text Section 2.opj as shown:

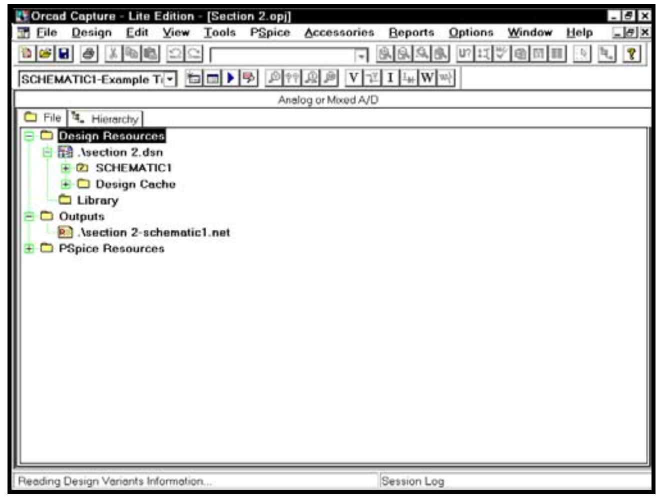


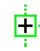
Click the Open button to open the file:

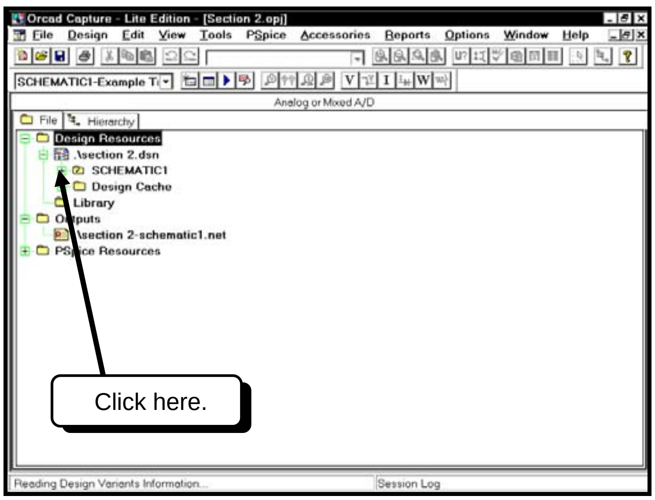


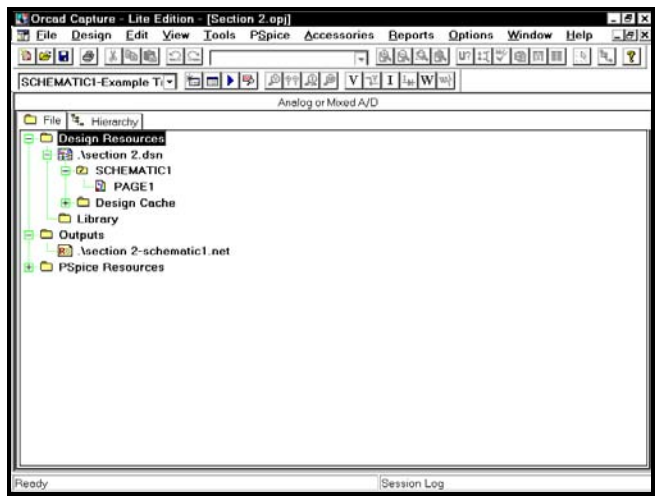
Next, we need to open the schematic page. Click the ***LEFT*** mouse button on the  as shown below to expand the tree:



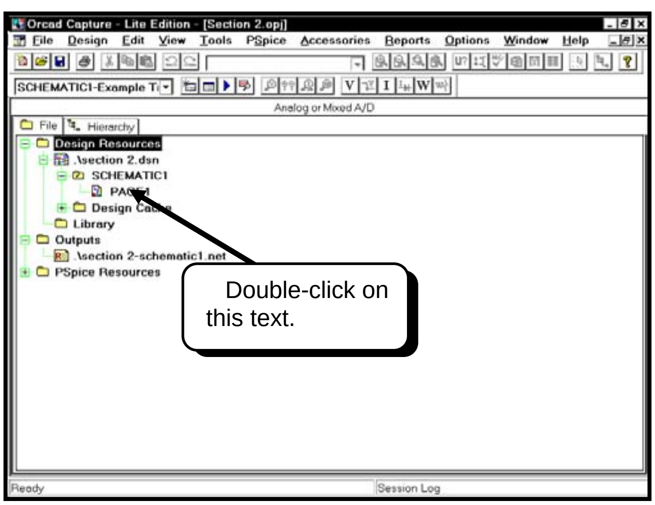


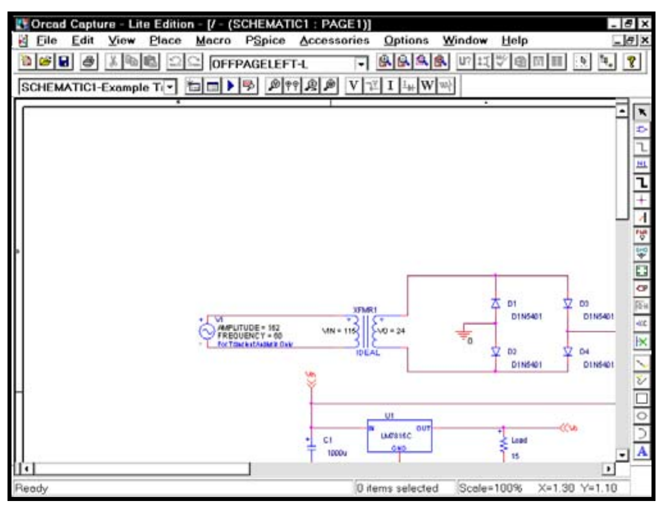
Next, we need to expand the tree to list the schematic pages of the project. Click the ***LEFT*** mouse button on the  as shown below:



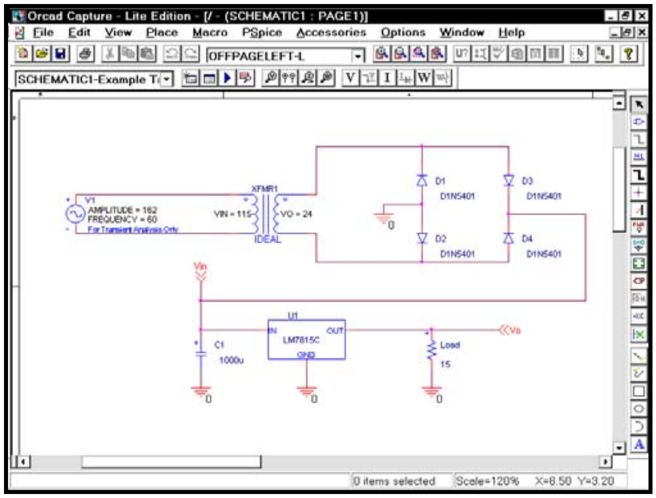


This project has one schematic page. To open the page, double-click the ***LEFT*** mouse button on the text PAGE1:



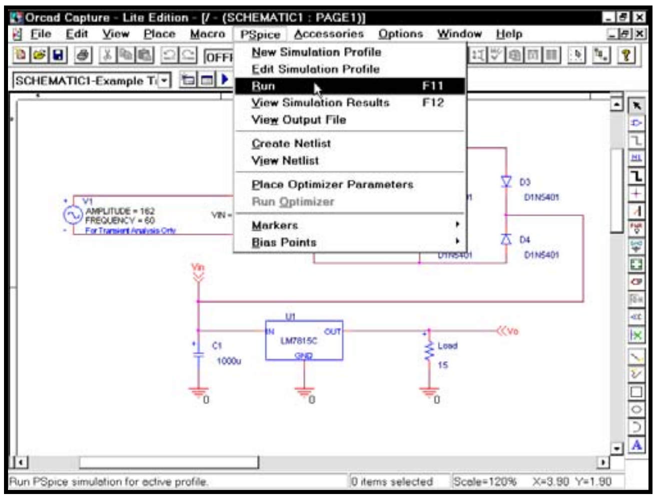


Use the scroll bars and zoom facilities to fill the screen with the circuit:

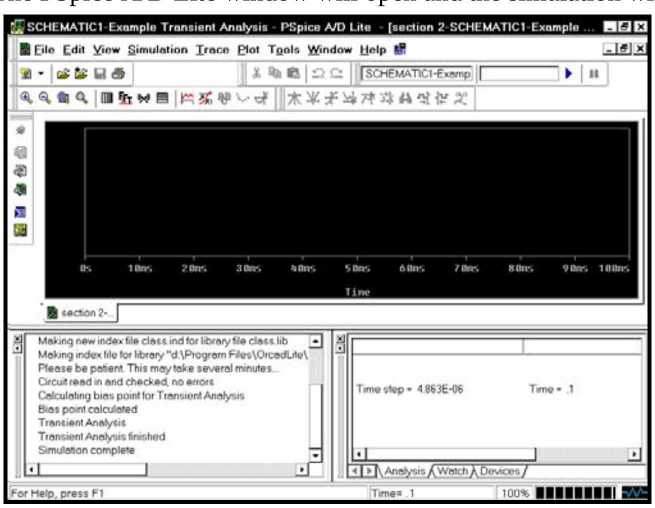


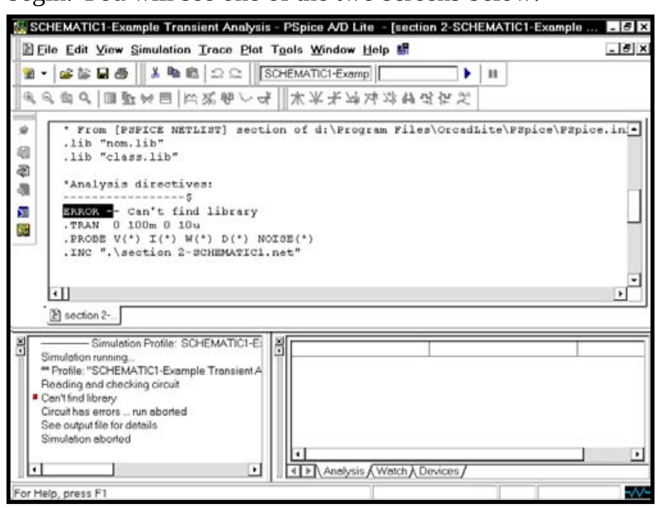
Notice that the file has two nodes labeled Vo and Vin.

To simulate the circuit, select **PSpice** and then **Run** from the Capture menus:



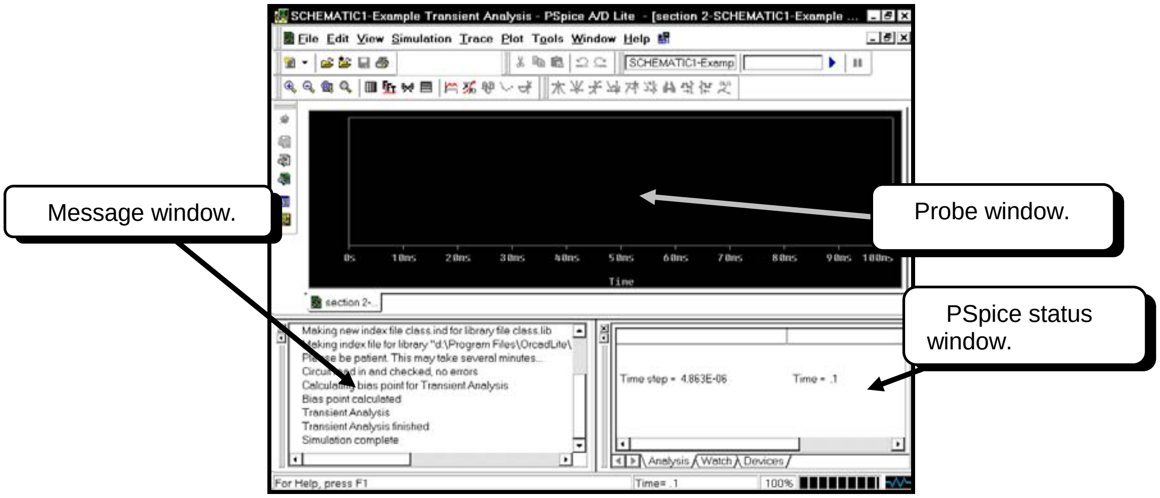
The PSpice A/D Lite window will open and the simulation will begin. You will see one of the two screens below:



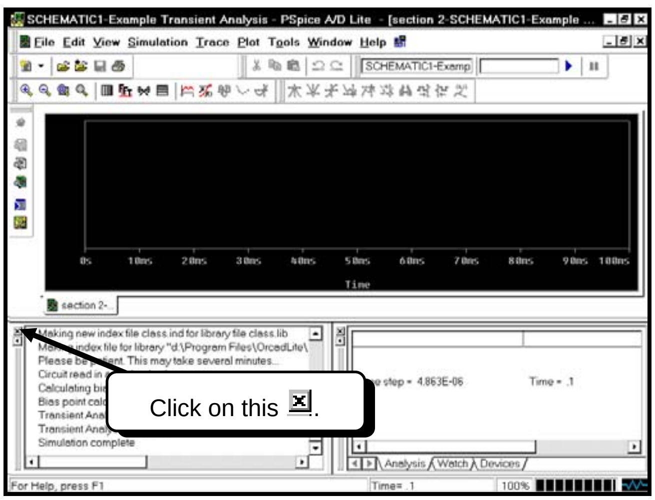


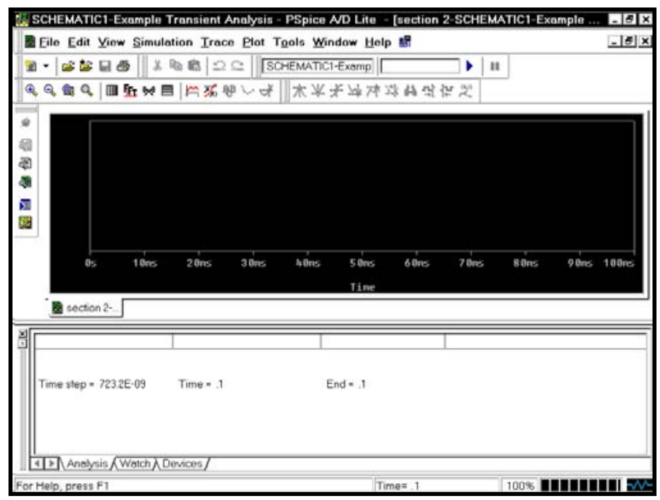
The above left screen shows that PSpice is working correctly. If your screen matches the above left screen, you can continue with the remainder of this example. The above right screen indicates that PSpice generated an error message. The error is that PSpice cannot find the library class.lib. If you see the above right screen on your computer, you may have not properly installed the libraries for this text. See Section A.2 on page 619 for the procedure for installing the libraries. After you have installed the libraries, attempt to run this example again.

We will discuss the PSpice A/D Lite window. It has three sections: a message window, the Probe display window, and the PSpice simulation status window.

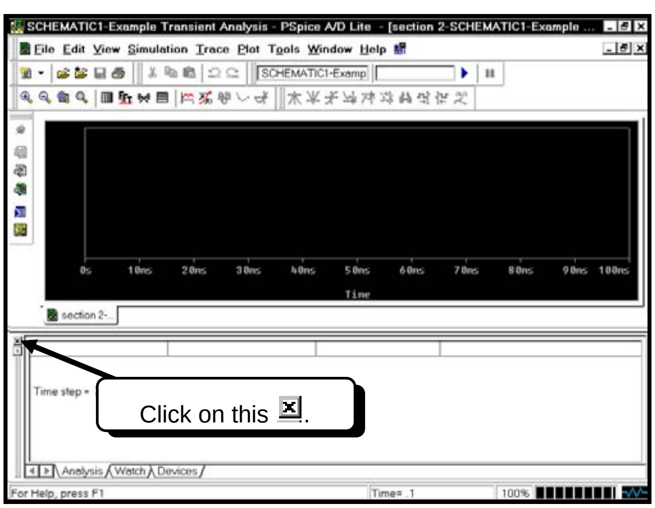


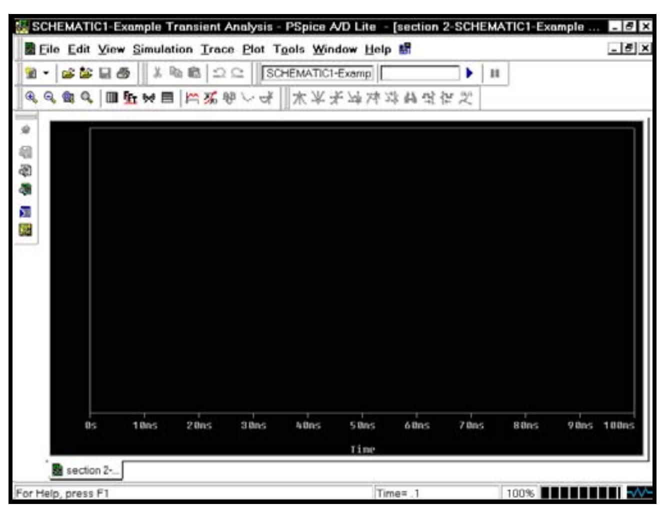
We will briefly show how to manipulate these windows. You can close either of the two bottom windows by clicking on the ☒ in the upper left corner of the window. For example, we will close the message window:





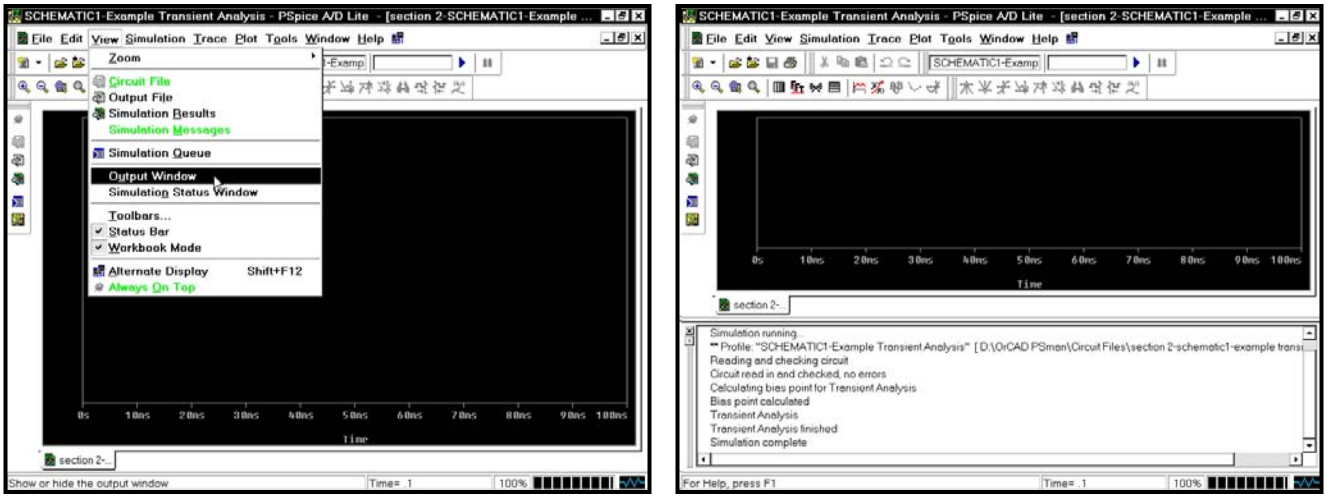
The message window is closed, and the PSpice simulation status window expands to fill the empty space. Next, we will close the simulation window:



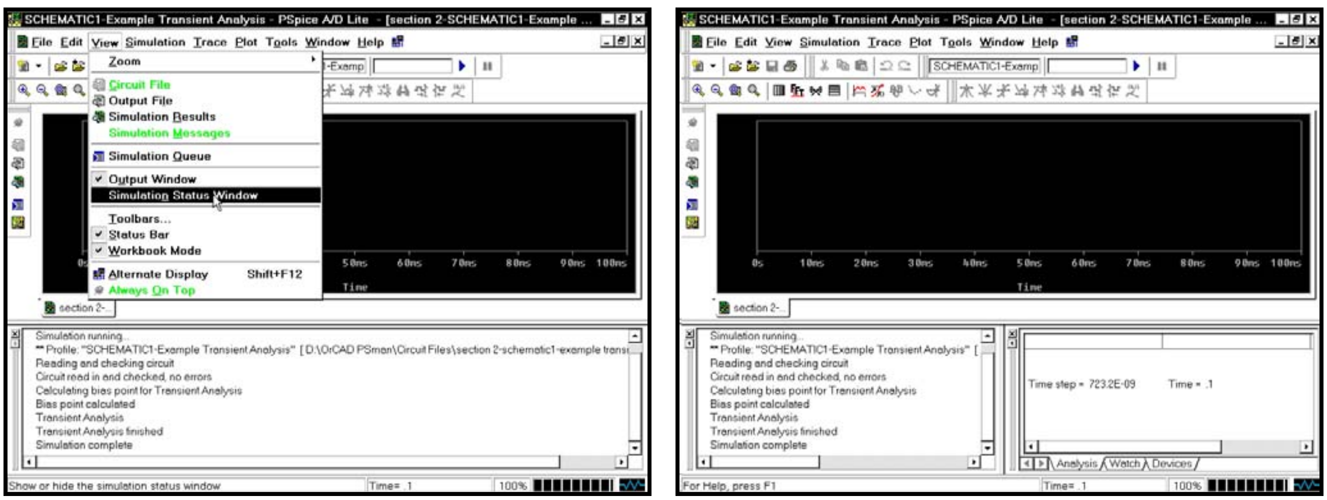


The Probe window now occupies the entire window.

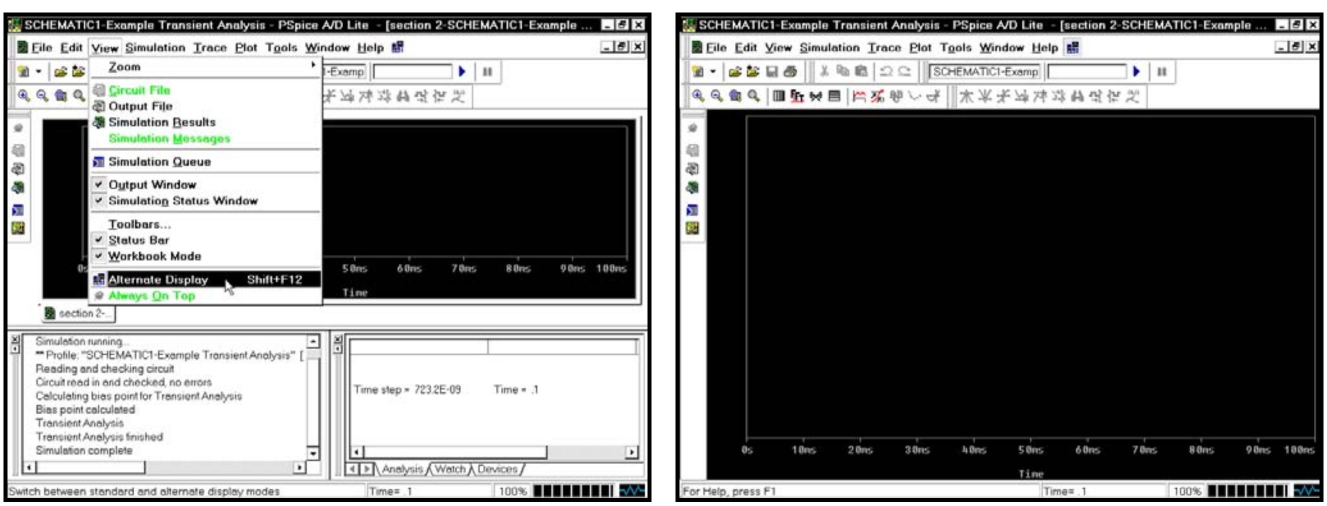
To get the message and simulation status windows back, use the menus. To display the message window, select **View** and then **Output Window** from the menus:



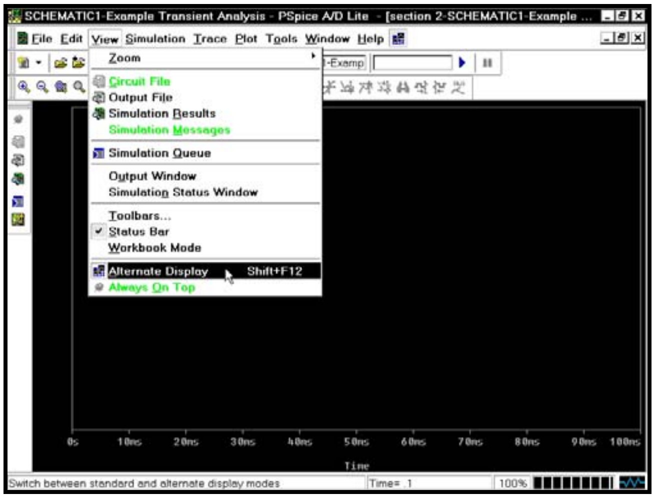
To display the simulation status window, select **View** and then **Simulation Status Window** from the menus:

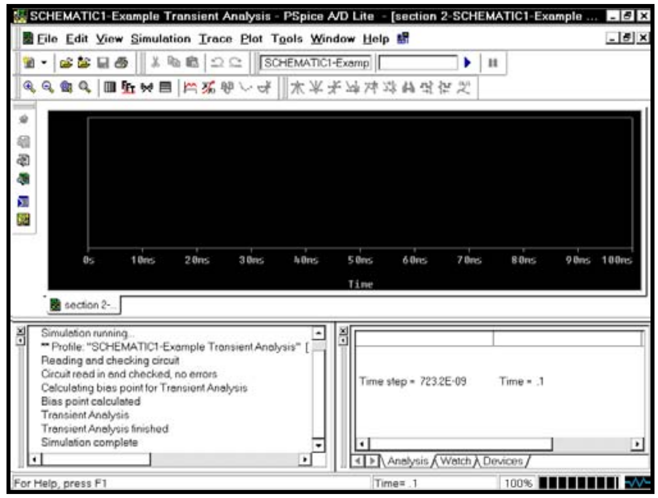


When we are running a simulation, we want the simulation status and the message windows to be visible because they give us information on problems with the simulations and indicate the progress of the simulation. When we are plotting traces, we would like Probe to use the full screen so that the traces are as large as possible. To toggle between these two conditions, select **View** and then **Alternate Display** from the menus:

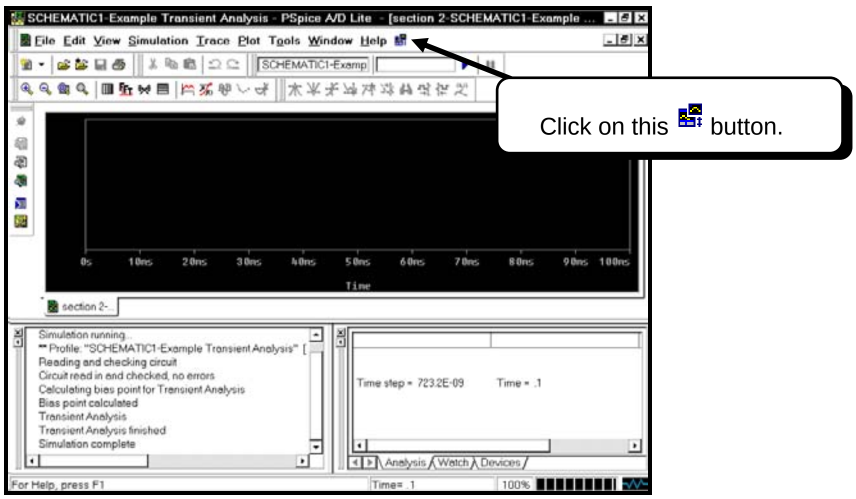


To toggle back to the original display, select **View** and then **Alternate Display** again from the menus:





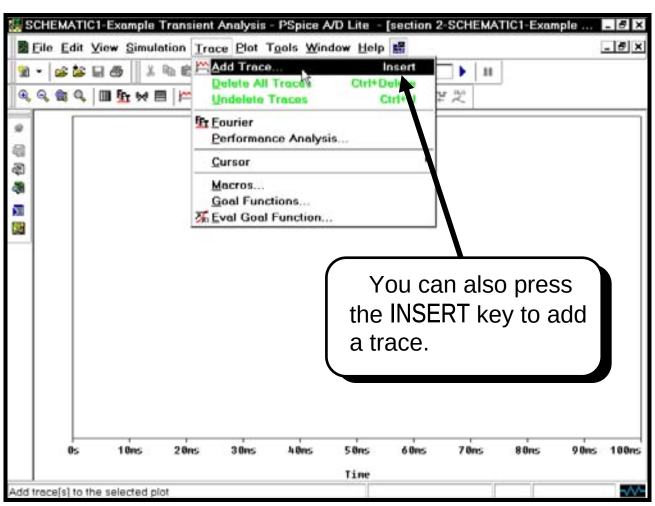
You can also toggle between the two displays by clicking on the  button as shown below:

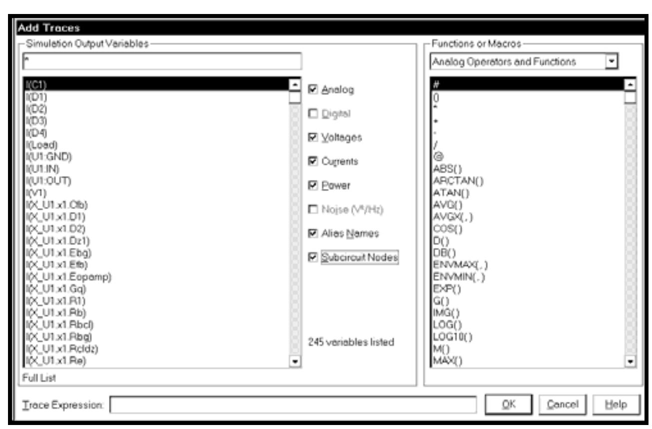


Select the display that you prefer to use. For most traces shown in this text, we will show the Probe window in full screen. Note also that your probe screen will have a black background. To make the screen captures more readable, the background color of the probe screen has been changed to white for the remainder of this book. This was accomplished by modifying the [PROBE DISPLAY COLORS] section of the pspipce.ini file located in the OrcadLite/PSpice directory.

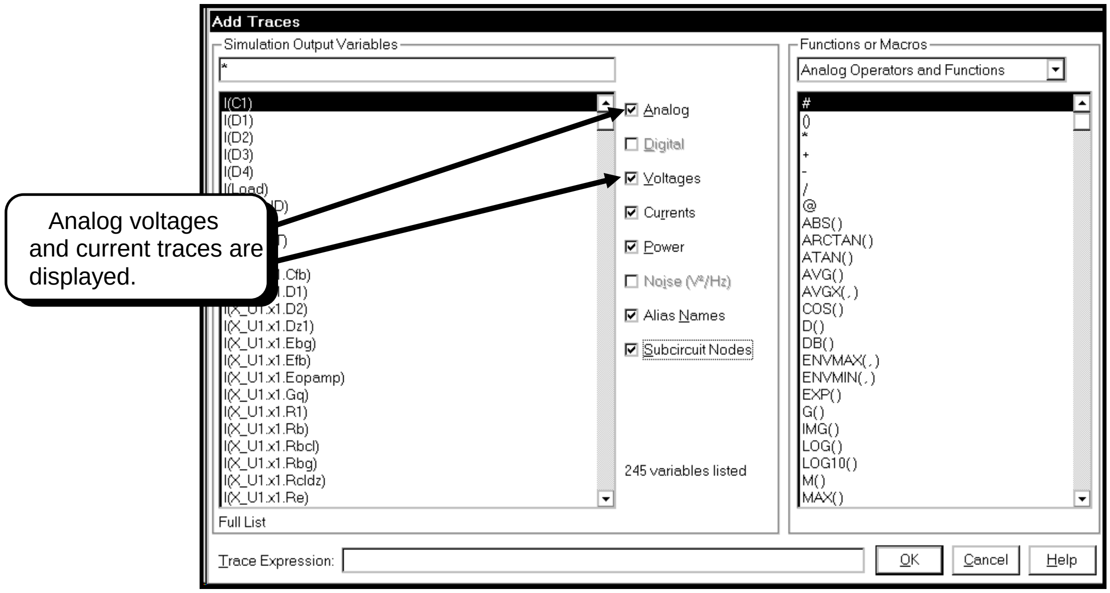
2.A. Plotting Traces

We will now add some traces. In the schematic we have labeled two nodes, Vin and Vout. Since we know the names of the nodes, we can easily plot the voltages. We will first plot Vin. To add a trace, select **Trace** and then **Add Trace** from the menu bar:



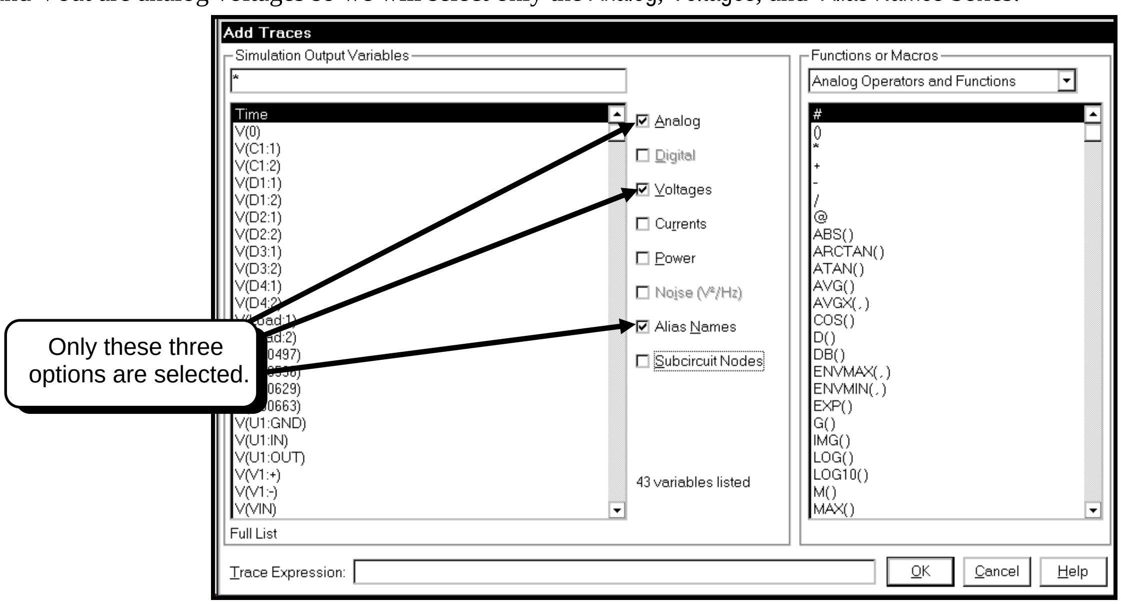


The left pane lists all of the traces we can plot. The right pane displays mathematical operations we can perform on the traces. It lists digital waveforms (we do not have any in our circuit), voltages, currents, and power. It also lists what are referred to as alias names. Aliases are different names that refer to the same thing. For example, in our circuit the cathodes of D3 and D4 are connected to the input of the voltage regulator and the capacitor. The node is also labeled as Vin. We can refer to the voltage of this node in several ways. This node could be addressed as pin 1 of D4, pin 1 of D3, pin 1 of C1, pin IN of the voltage regulator, or as Vin. Thus, to display the voltage at this node, we can use any of the aliases mentioned. Each node will have many aliases, and thus the list shown in the left pane is very large. Note that almost all trace types are displayed: Analog, Voltages, Currents, Power, Alias Names, and Subcircuit Nodes:

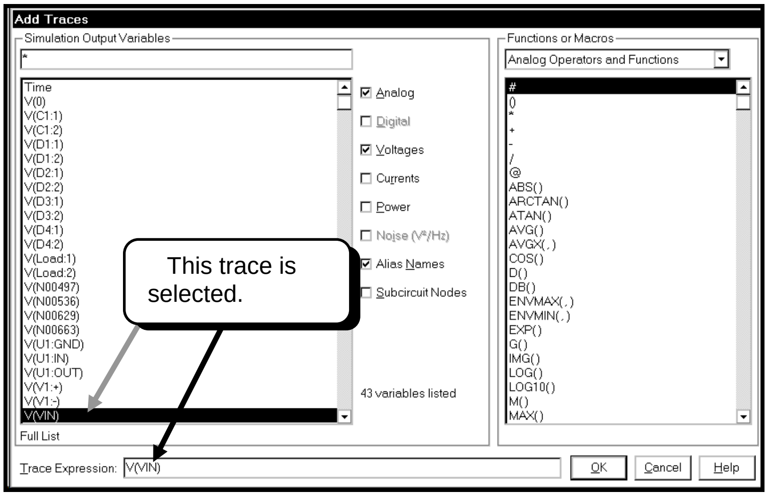


Because all of the options are selected, many names are shown in the left pane. Note that the voltage regulator is a subcircuit. A subcircuit is shown as a single block on the schematic, but it may be composed of several circuit elements within the subcircuit. The left pane is currently displaying all subcircuit nodes and the aliases for the subcircuit nodes. If you scroll through the list, you will see too many traces.

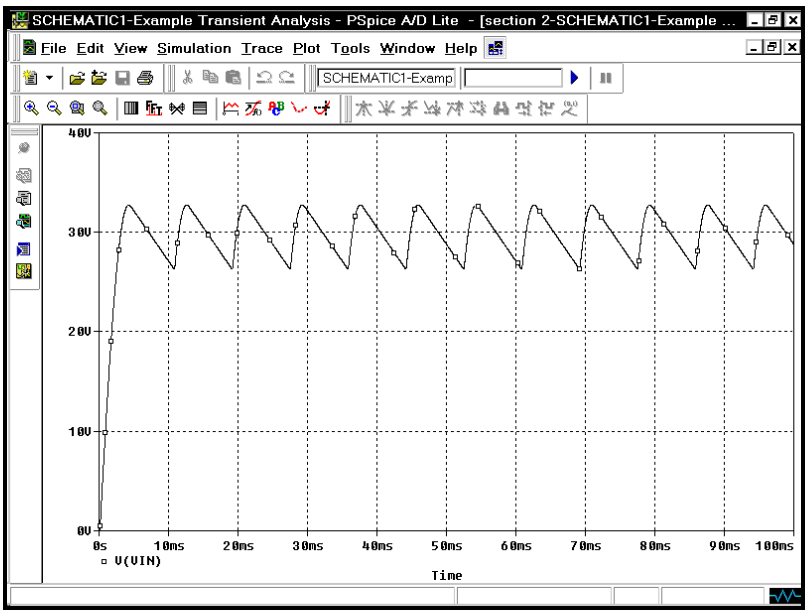
We would first like to display Vin and Vout. The list of traces is too long and the traces are not easily spotted. Vin and Vout are analog voltages so we will select only the Analog, Voltages, and Alias Names boxes:



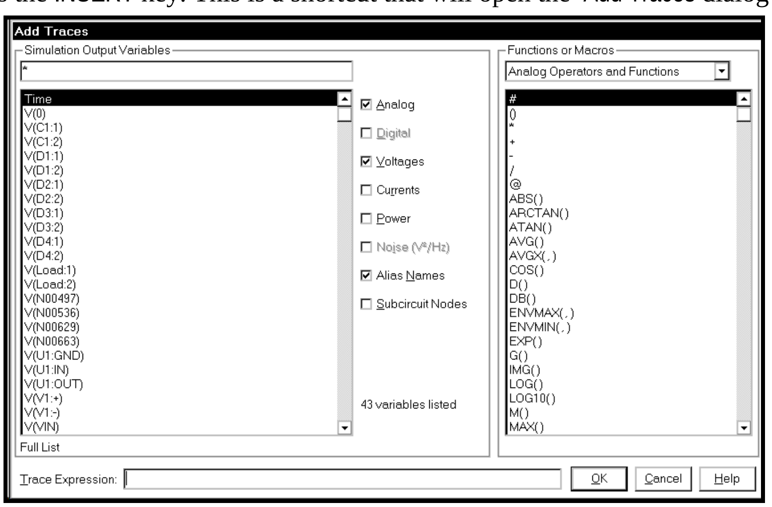
This list is now shorter and we can locate the line V(VIN) more easily. Scroll down the list until you find the trace labeled V(VIN) and then click on the text to select it. It should become highlighted:



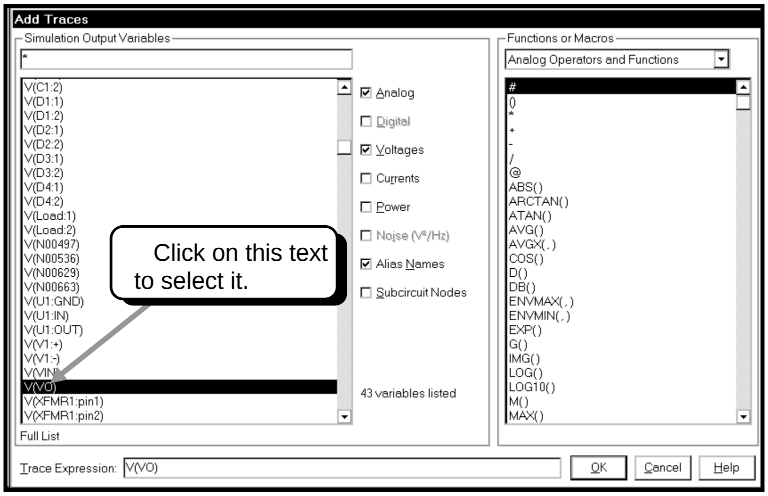
To plot the selected trace, click the OK button:



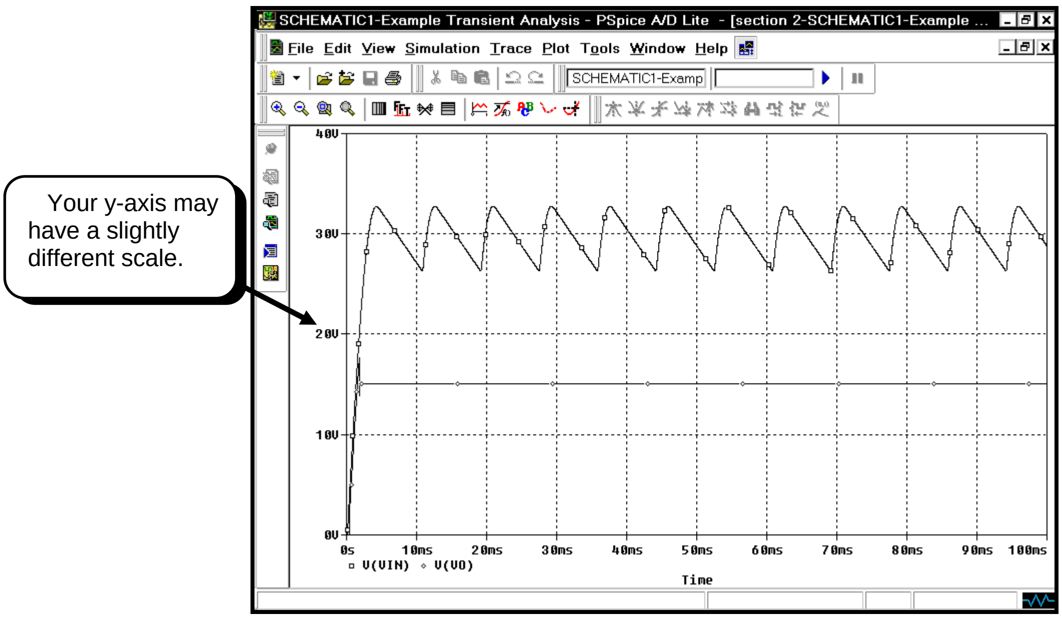
Next we will plot Vo. Press the INSERT key. This is a shortcut that will open the Add Traces dialog box:



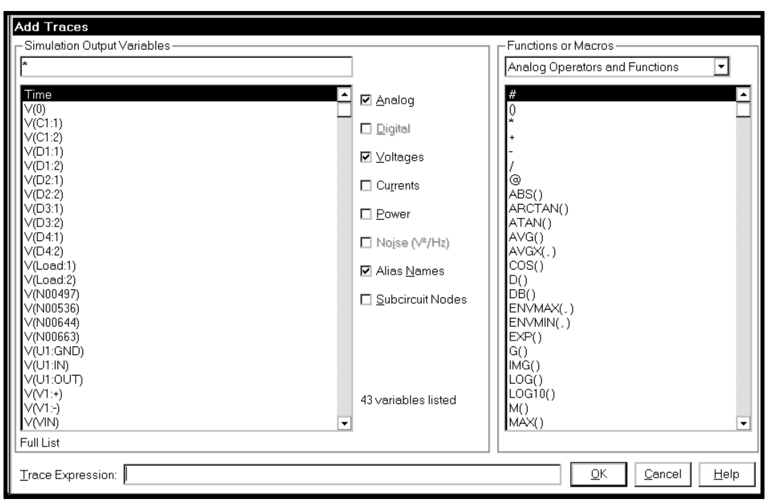
Click on the text V(VO) to select the trace:



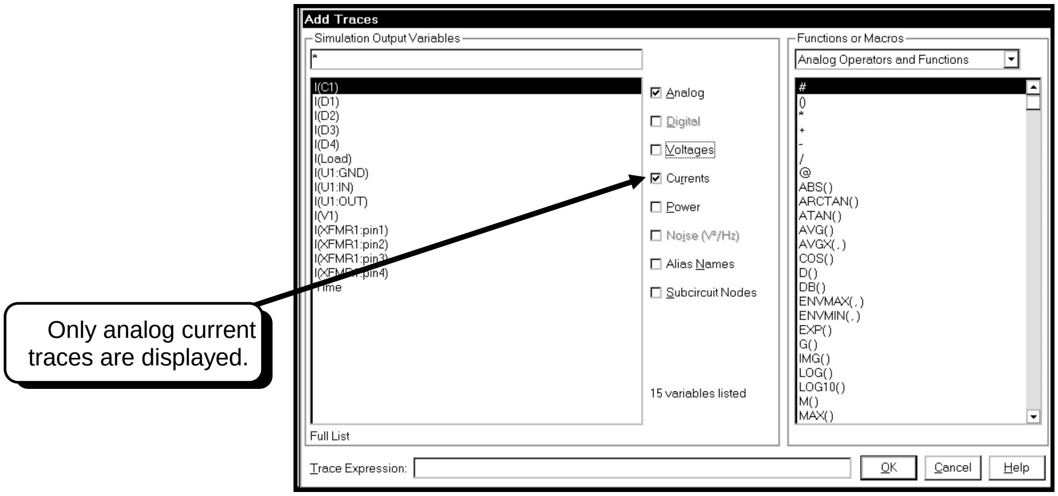
Click the OK button to plot the trace:



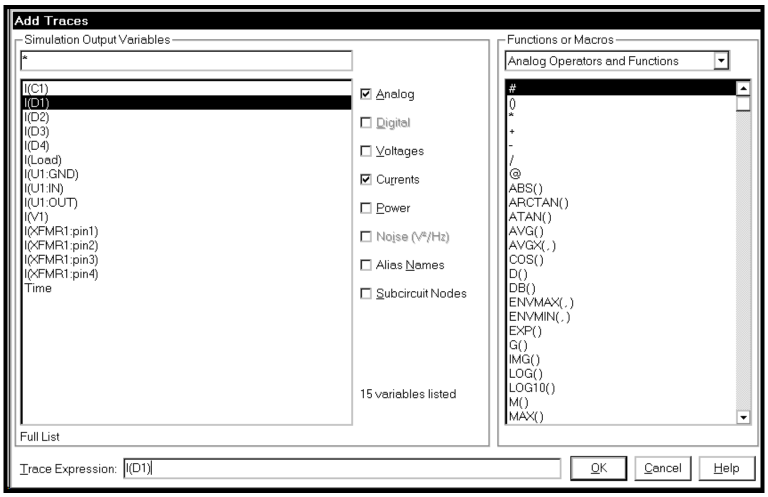
We can add many traces to the plot. Next we will display the current through D1. Press the INSERT key to obtain the Add Traces dialog box:



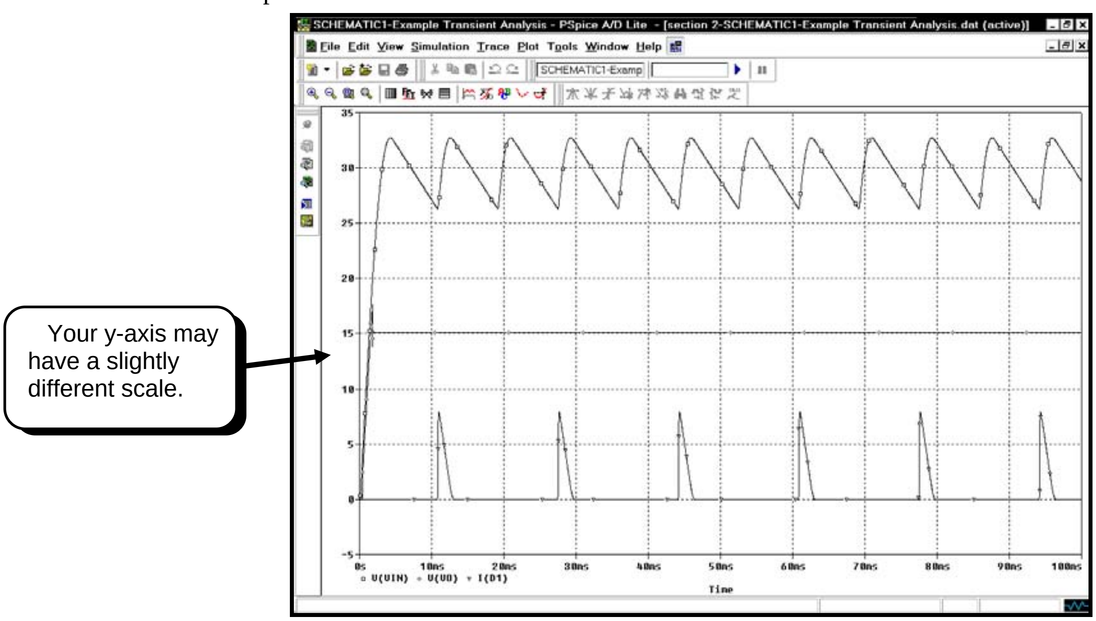
Presently the dialog box shows only analog voltages. We wish to plot a current, so specify the options as shown:



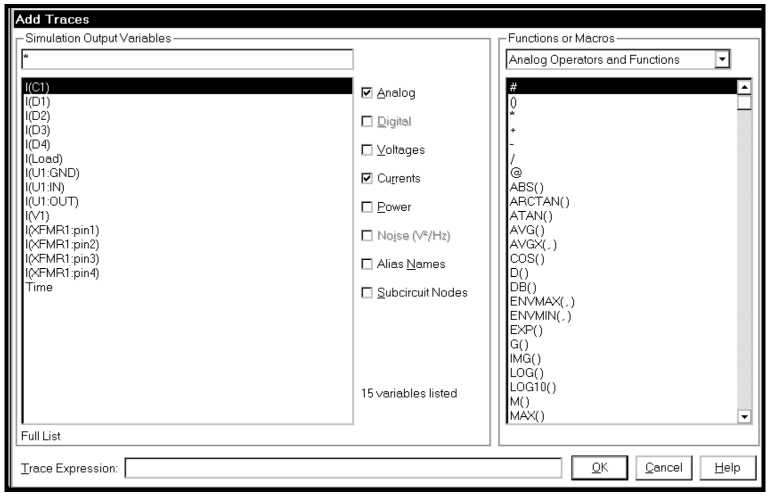
We see that only the options Analog and Currents are selected. The left pane displays only the currents. Click on the text I(D1) to select the trace:



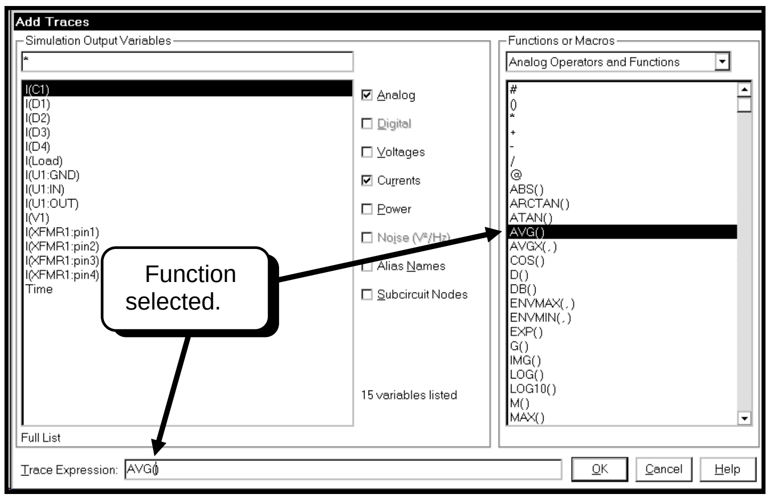
Click the OK button to plot the trace:



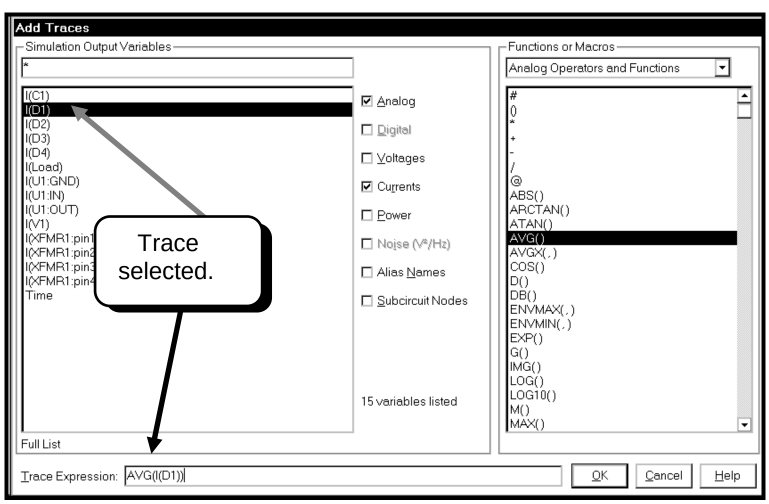
As a last example, we will show how to use one of the mathematical operations in the right pane of the Add Traces dialog box. Press the INSERT key to obtain the dialog box:



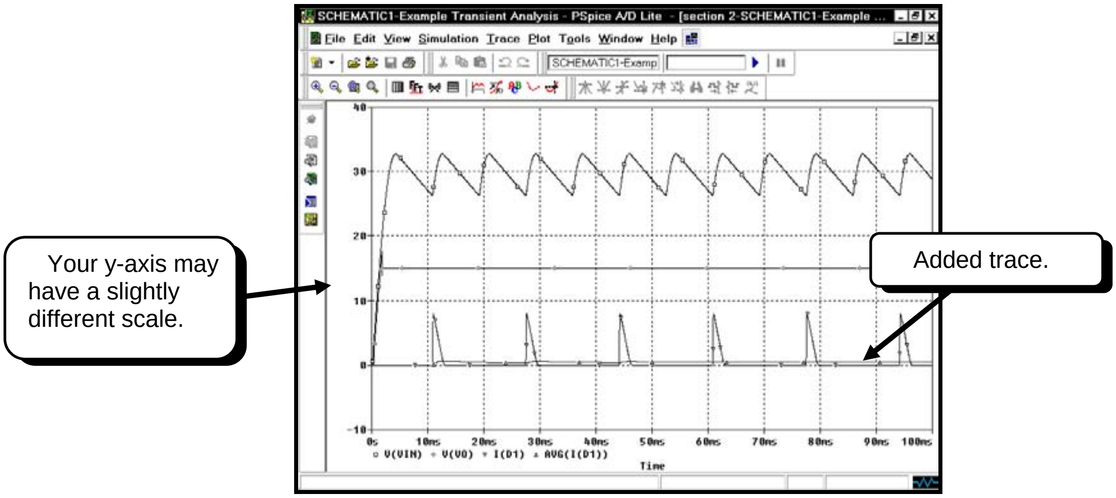
We will plot the time average current through D1. The AVG function will perform this function. Click the ***LEFT*** mouse button on the text AVG() to select the function:



Notice that the text AVG() appears in the Trace Expression text field and that the cursor is positioned between the parentheses waiting for a trace. Next, click the ***LEFT*** mouse button on the text I(D1). This will select the trace and place it within the parentheses of the AVG function:



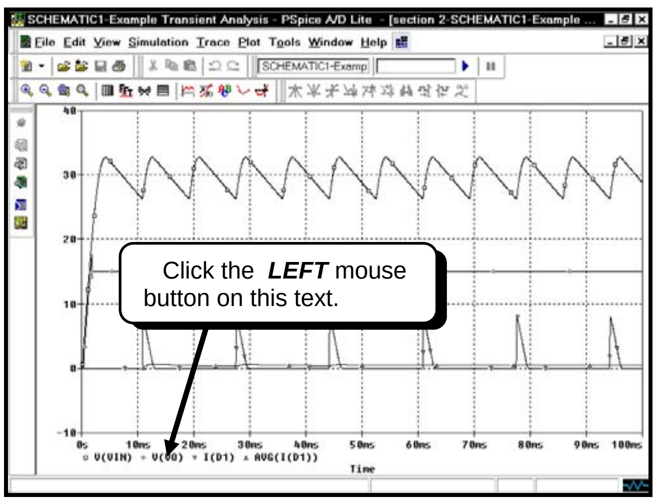
This trace is plotting the time average of the current through D1. Click the OK button to plot the trace:



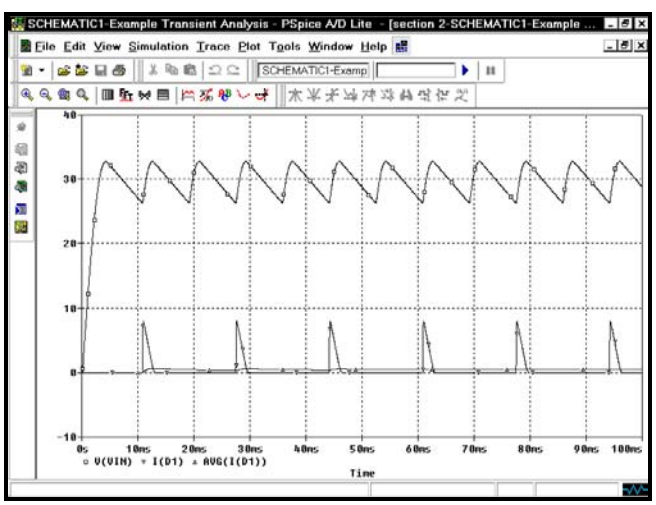
The trace starts out small but we can see the fourth trace.

2.B. Deleting Traces

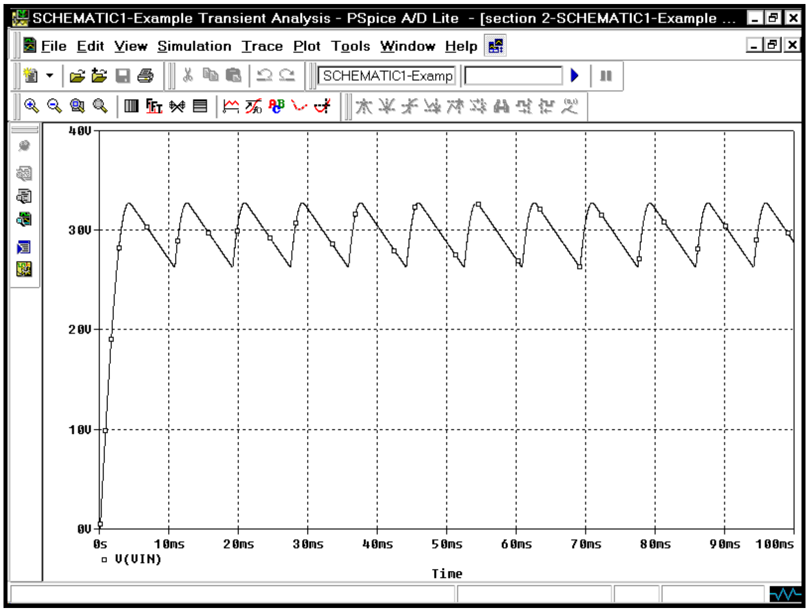
We now have a number of traces on the plot. We can remove individual traces easily. We will remove trace V(VO). Click the ***LEFT*** mouse button on the text V(VO) as shown below:



The text will become highlighted in red, indicating that it is selected. When the text V(VO) is highlighted in red, press the DELETE key. The trace will be removed:

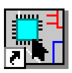


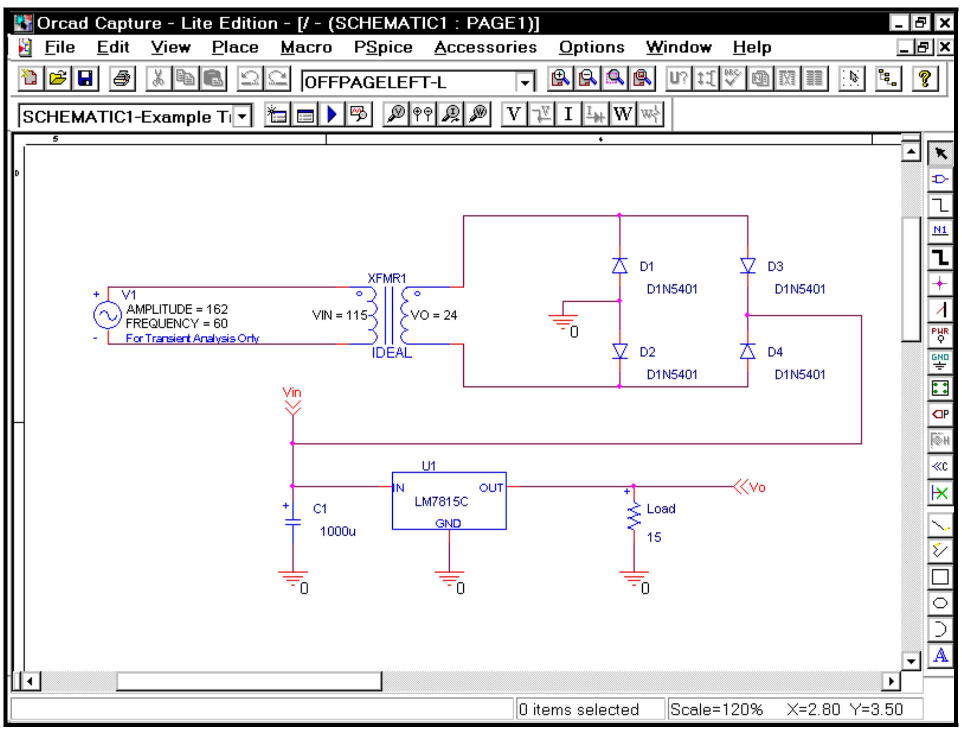
Delete all traces but V(Vin) using this method:



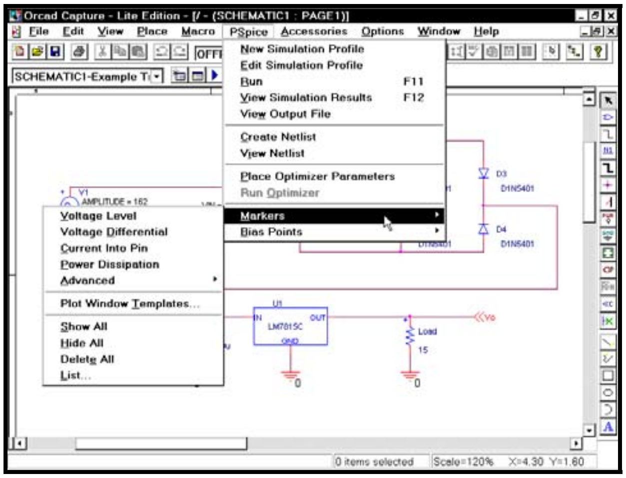
2.C. Using the Markers to Add Traces

In this example we are currently describing, both Probe and Capture are running. We would like to switch back to Capture. We can switch windows using two methods:

1. Hold down the ALT key and press the TAB key. Continue to hold down the ALT key and release the TAB key. When you do this, a window will pop up and display the icons of the programs currently running. One of the icons should be the Capture icon . Press the TAB key until the black box encloses the Capture icon. When the box encloses the Capture icon, release the ALT key. The Capture window will pop to the top.
2. You can use the Start menu to switch to the Capture window. If the Start menu is not displayed on the screen, bring the mouse to the bottom of the screen to display the Start menu, and then select the Capture button.

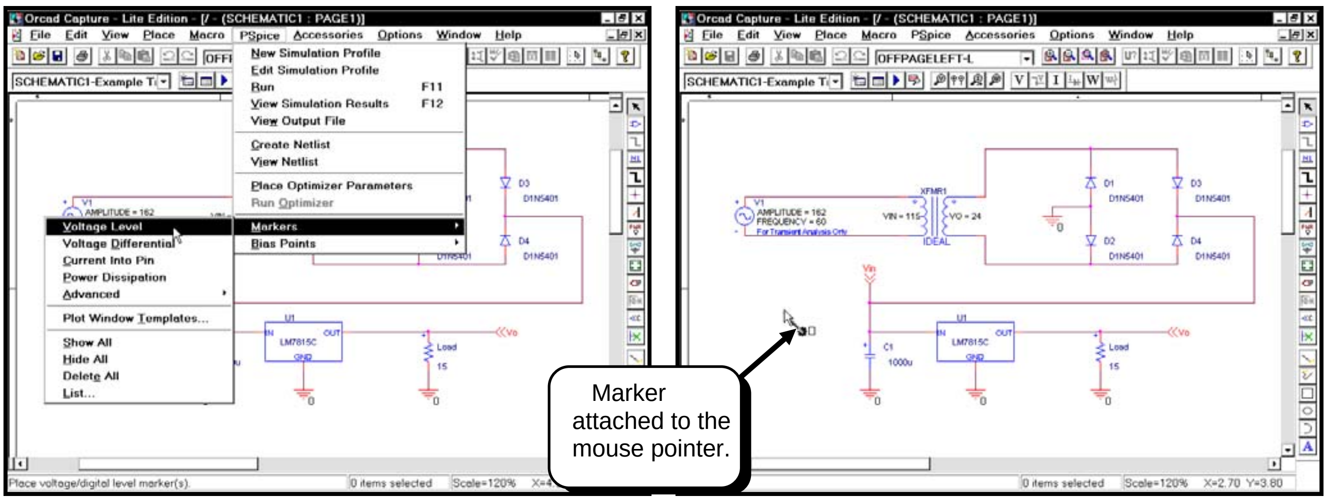


We can now use the markers to display the currents or voltages in the circuit. To obtain a voltage marker, select **PSpice** and then **Markers** from the Capture menu bar:

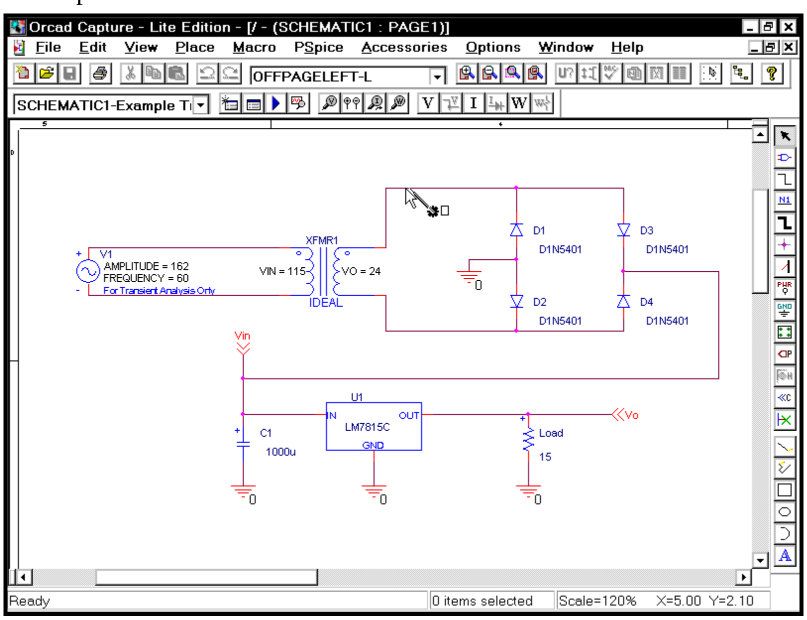


Voltage Level will display the voltage at a node relative to ground. **Voltage Differential** will display the voltage between any two points. With **Voltage Differential**, you will be required to place two markers. The first marker will designate the positive reference for the voltage difference, and the second marker will designate the negative reference for the voltage difference. **Current Into Pin** will display the current into a device. You will need to place the marker on one of the pins to the device.

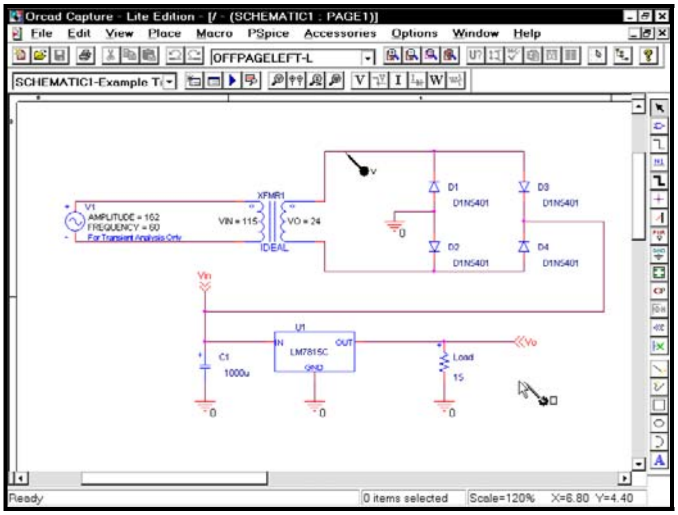
We will look at the voltage at the cathode of D1 relative to ground. Select **Voltage Level** from the menu. A marker will become attached to the mouse pointer:



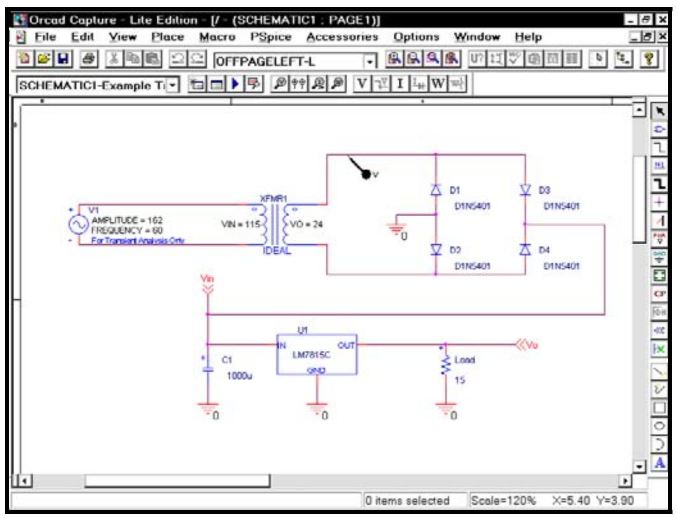
Move the mouse to position the pointer as shown:



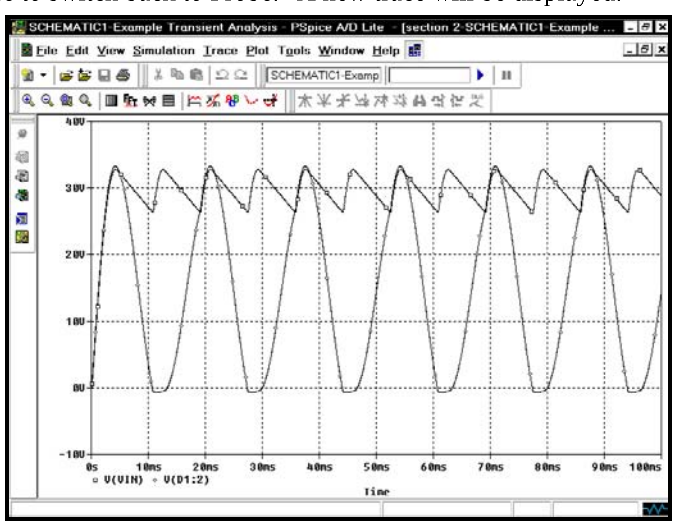
Click the ***LEFT*** mouse button to place the marker. Move the mouse away:



A marker is placed on the wire and a new marker is attached to the mouse pointer. We can place more markers if we want. Press the **ESC** key to terminate placing markers. The marker will change to the color the trace is displayed in Probe. On my computer, all Probe traces are displayed in black, so the marker is displayed in black. If your marker turns green, then the trace will be displayed in green on the Probe screen.

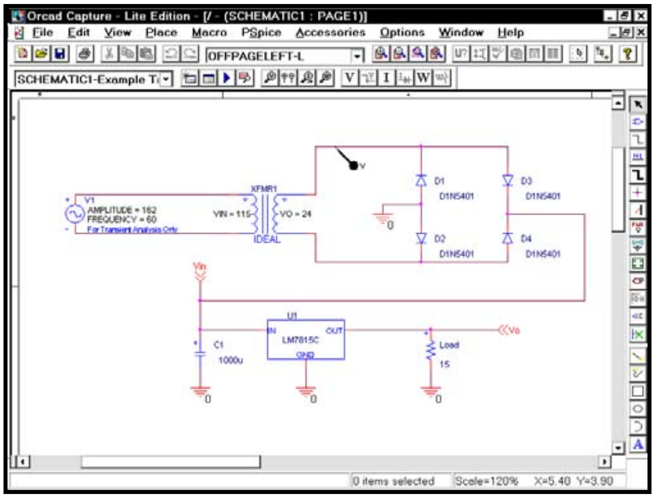


Use the ALT - TAB key sequence to switch back to Probe.* A new trace will be displayed:

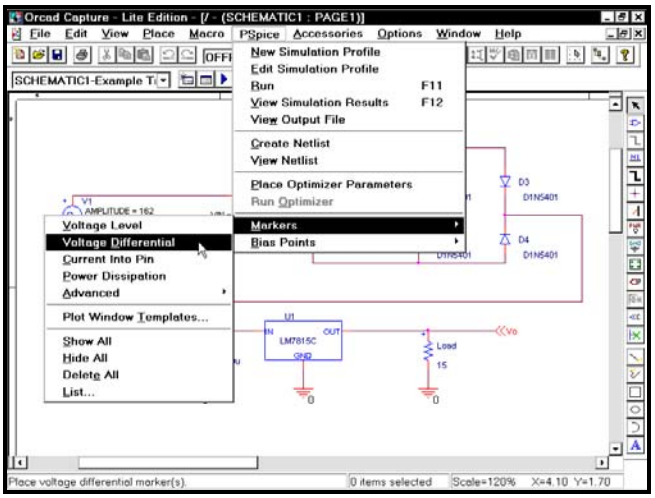


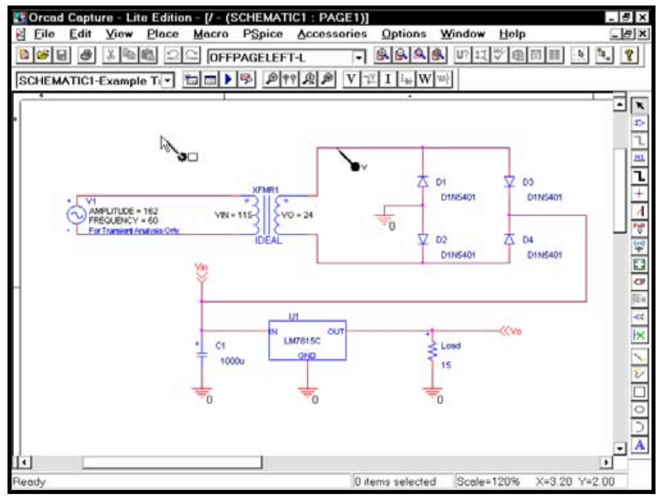
* See Page 107 for a more detailed description of how to switch between windows using this method.

We see that the voltage specified by the marker is displayed. Markers are convenient because we do not need to know the node names to plot a trace. Note that the name of this trace is V(D1:2), hardly an obvious name. Use the ALT - TAB key sequence to switch back to Capture:

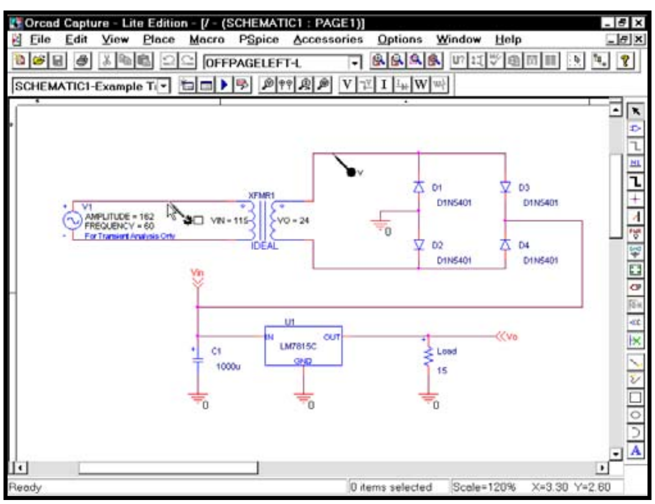


Next we will plot the voltage of source V1 using the voltage differential markers. Select **PSpice**, **Markers**, and then **Voltage Differential**. A marker will appear:

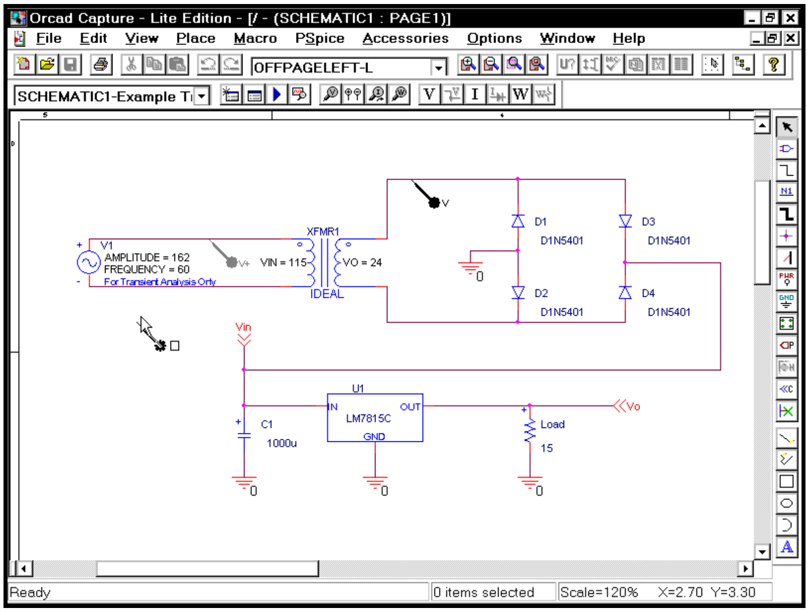




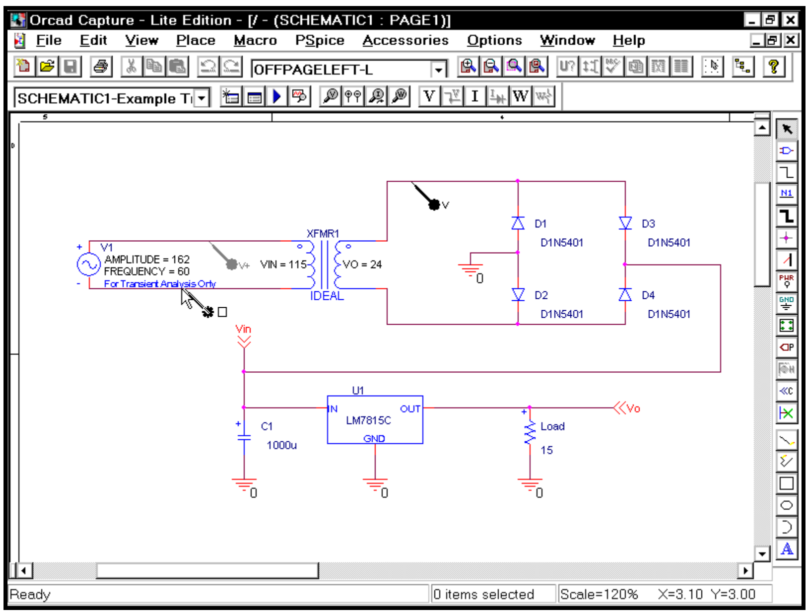
Move the mouse to position the marker as shown:



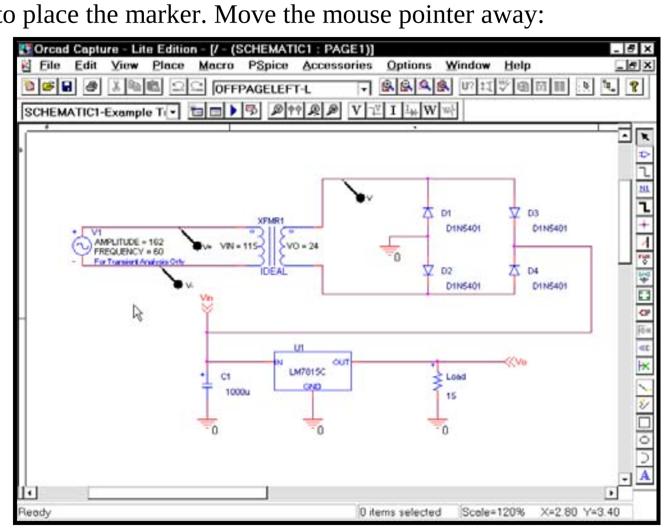
Click the ***LEFT*** mouse button to place the marker. As you move the mouse away, you will notice that a marker is placed on the wire and that a second marker is attached to the mouse pointer:



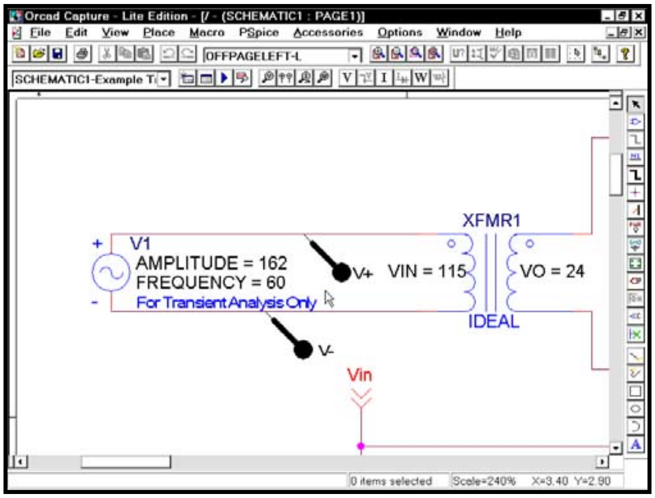
Move the mouse to position the marker as shown:



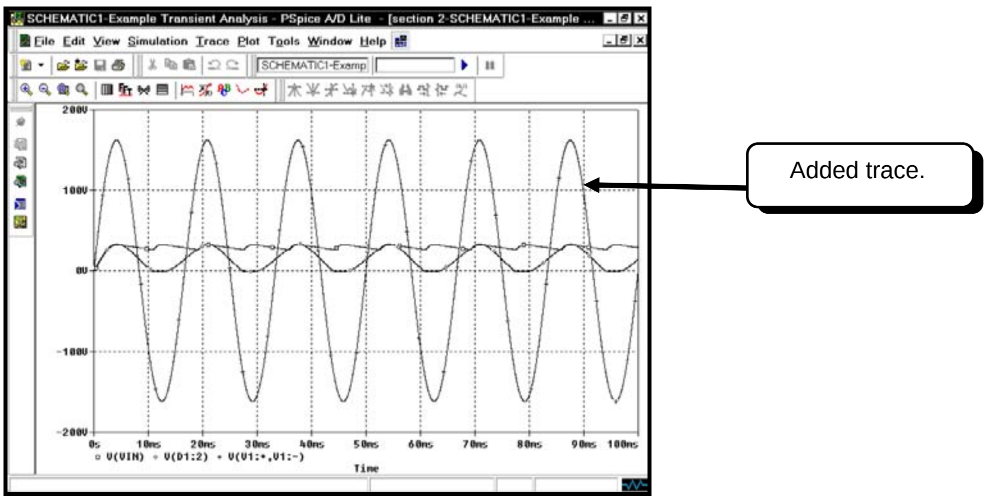
Click the ***LEFT*** mouse button to place the marker. Move the mouse pointer away:



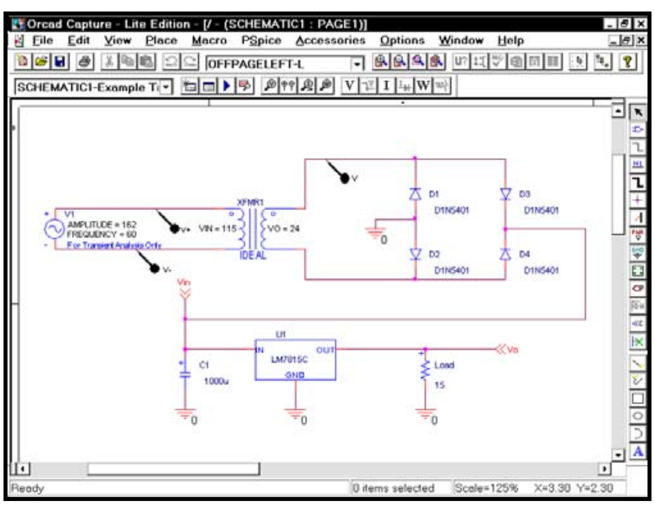
Press the **ESC** key to terminate placing markers. I will zoom in on the markers to show that one marker has a plus sign and the other has a minus sign:



These markers display the voltage difference between the two markers. To view the trace, switch to the Probe window using the ALT - TAB key sequence.* The new trace will be displayed:

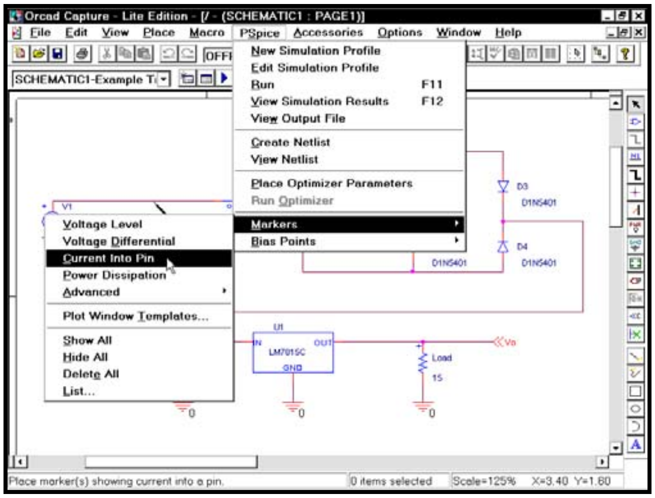


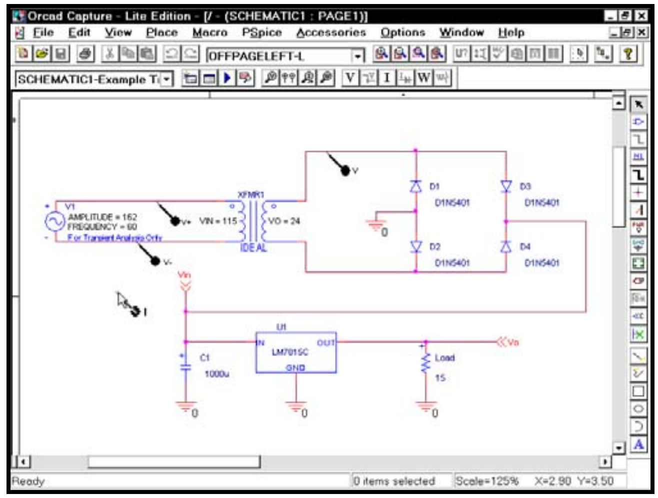
Last, we will use a current marker to display the current through an element. Use the ALT - TAB key sequence to switch back to Capture:



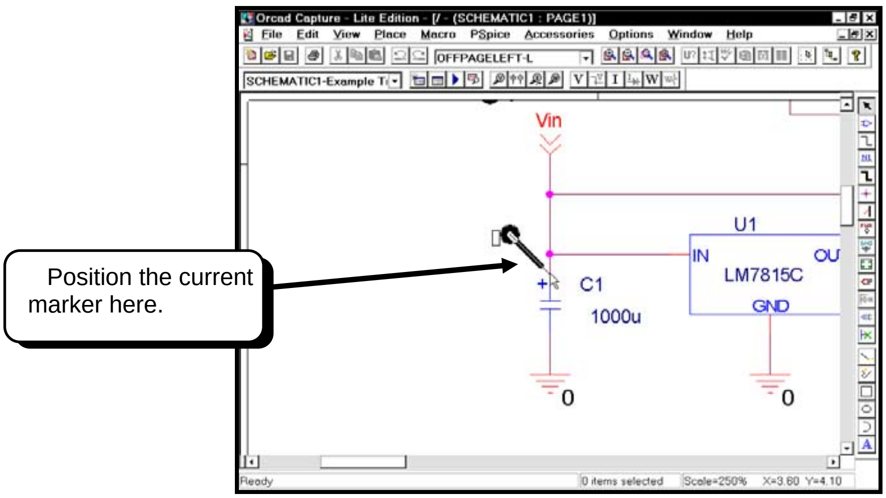
We would like to plot the capacitor current. Select **PSpice**, **Markers**, and then **Current Into Pin**. A marker with an I will become attached to the mouse pointer:

* See page 107 for a more detailed description of how to switch between windows using this method.

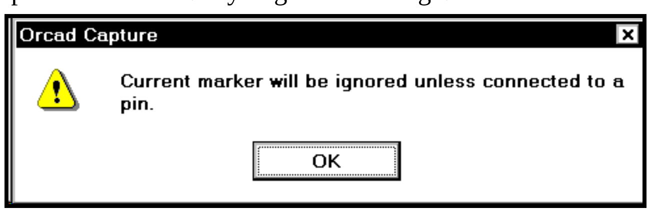




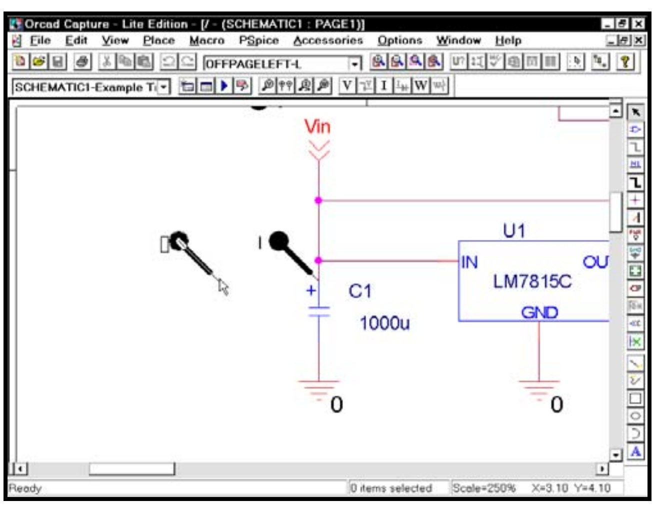
Position the marker as shown. Type **R** to rotate the marker.



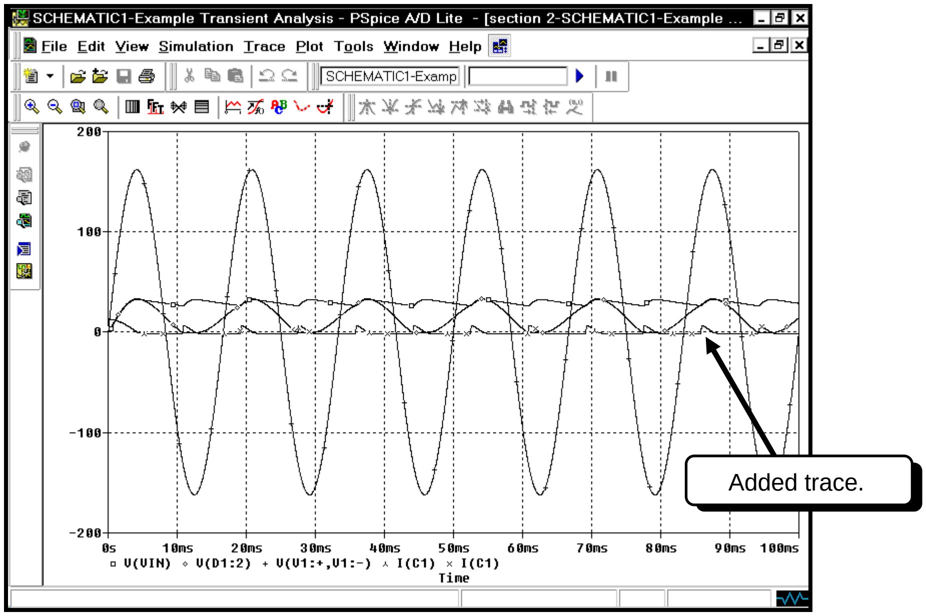
Click the ***LEFT*** mouse button to place the marker. If you get the message:



you missed the pin. Move the marker up or down one grid increment and try again. The marker must be placed at the end of a pin connected to a device. After you click the ***LEFT*** mouse button to place the marker and do not receive an error message, move the mouse pointer away:



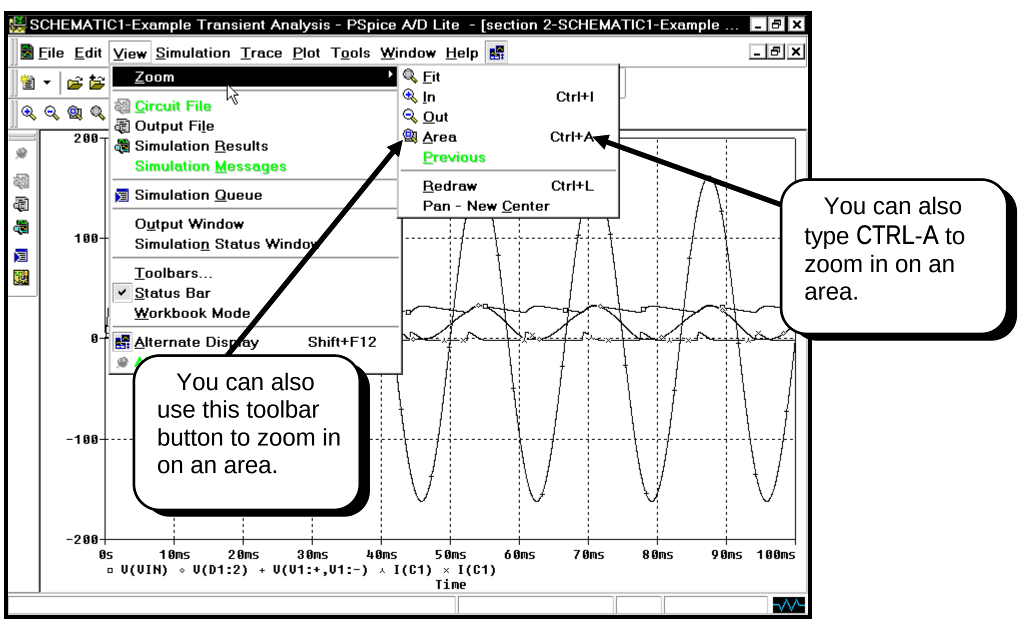
The marker is now attached to the end of the top pin of the capacitor. Press the **ESC** key to terminate placing markers. Use the ALT - TAB key sequence to switch back to Probe.* The current trace will be displayed:



The current trace is small, but its peaked shape is easy to spot.

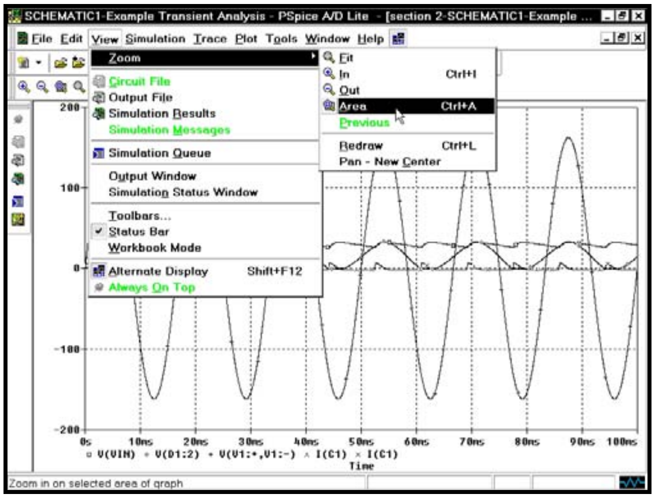
2.D. Zooming In and Out

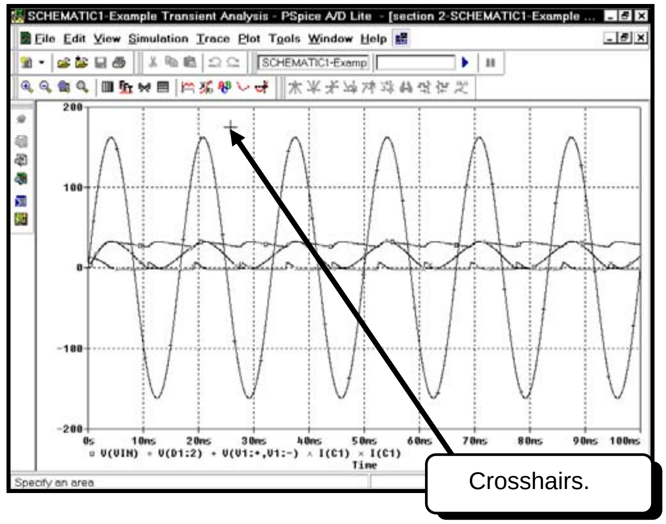
We now have a number of traces displayed. However, the current trace is small. Suppose that we would like to look a little closer at a peak in the current waveform. We can do this by using some of the zoom features provided by Probe. Select **View** and then **Zoom** from the menu bar:



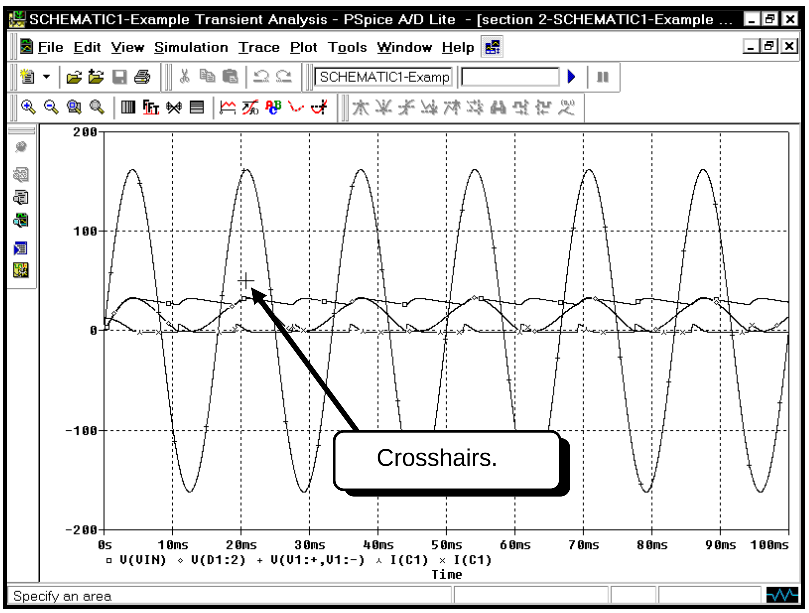
The menu lists 5 ways to zoom in and out. Select **Area**. The cursor will be replaced by crosshairs:

* See page 107 for a more detailed description of how to switch between windows using this method.

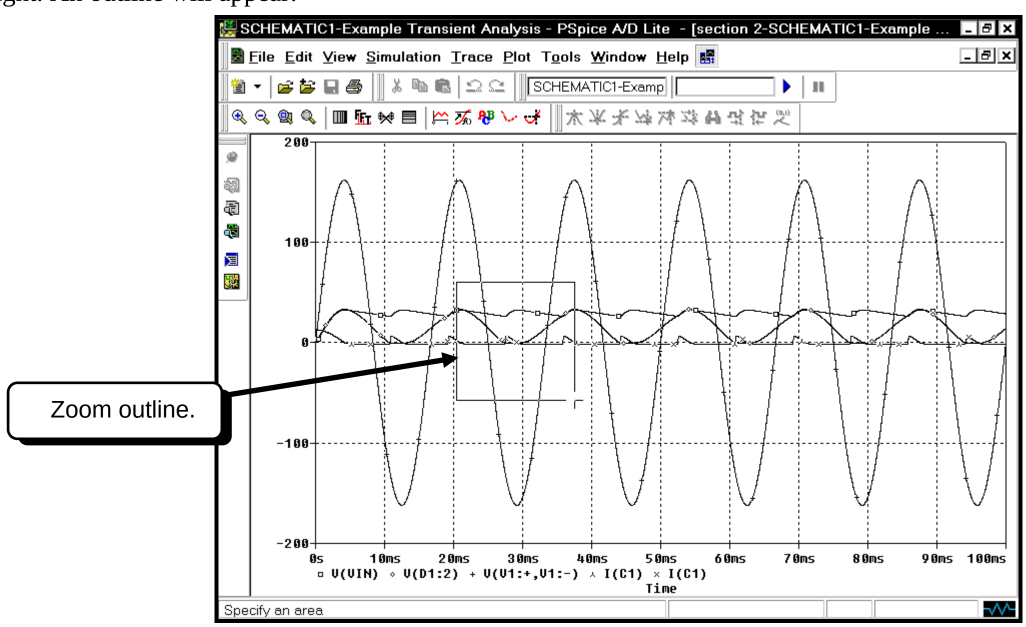




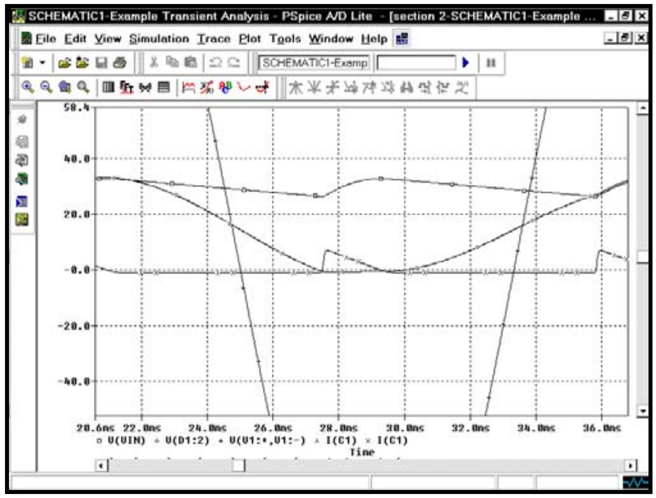
Position the crosshairs as shown below:



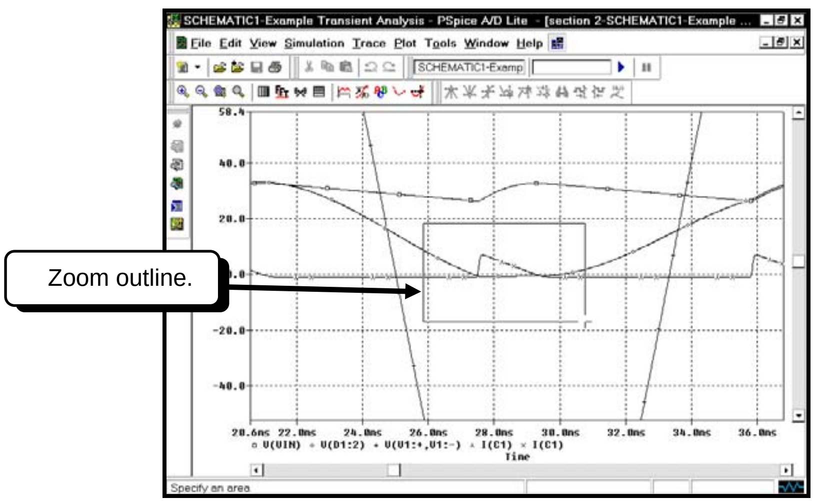
Click and **HOLD** the ***LEFT*** mouse button. While continuing to hold down the mouse button, move the mouse down and to the right. An outline will appear:



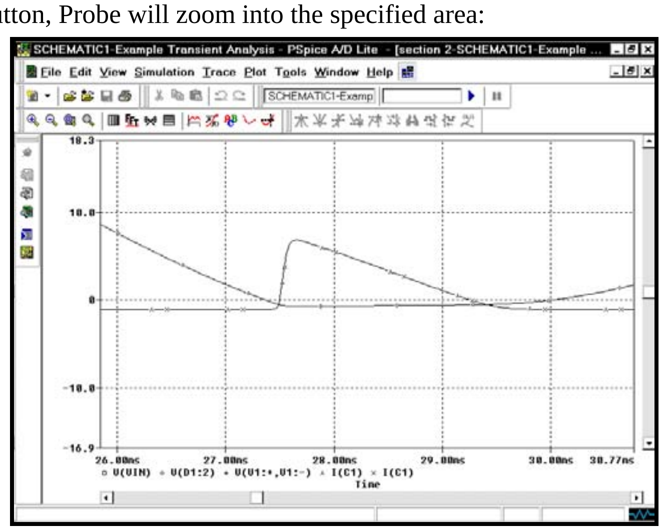
The portion of the plot inside the outline will be enlarged to fit the screen. Move the mouse to make an outline as shown above and release the mouse button. The display will zoom in on the area:



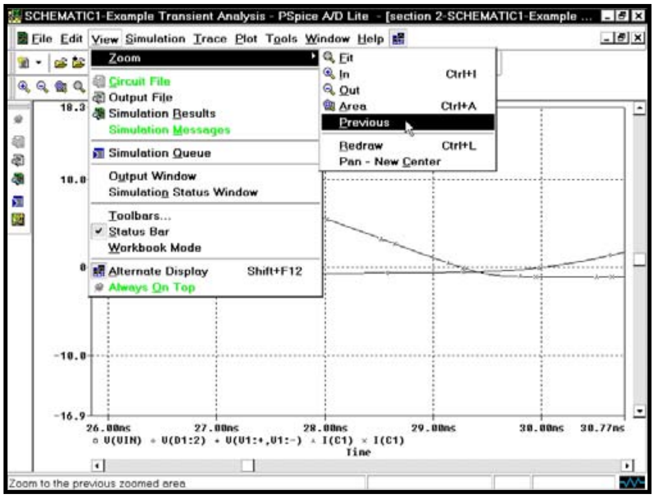
We can use the same technique to zoom in further. This time, type CTRL-A to zoom in again with the same method. Draw an outline as shown:

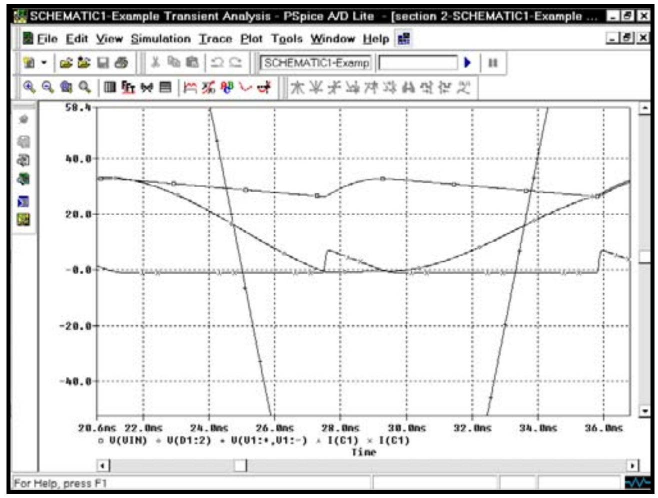


When you release the mouse button, Probe will zoom into the specified area:

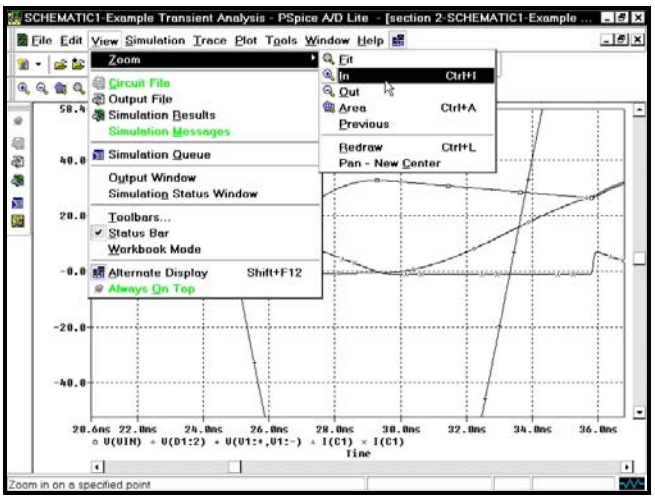


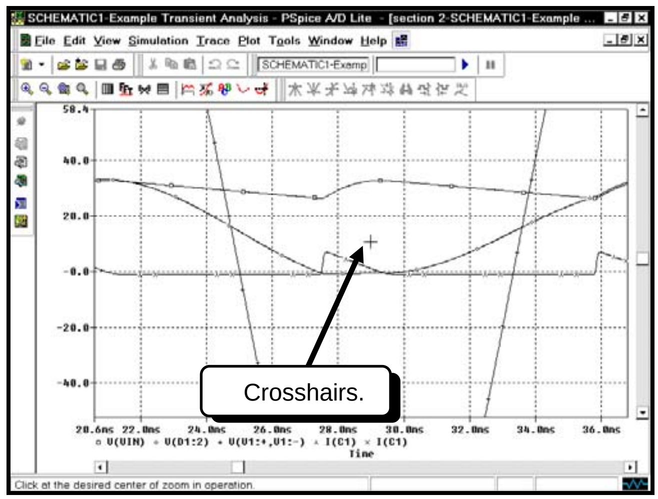
Suppose that we do not like the present view. To return to the previous view, select **View**, **Zoom**, and then **Previous**:



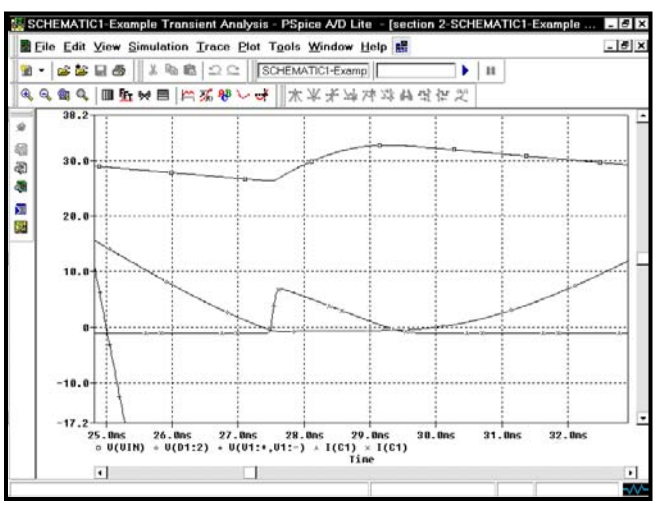


A second method for zooming in is selecting **View**, **Zoom**, and then **In**. This will zoom in around the cursor by a fixed percent. Select **View**, **Zoom**, and then **In**. The cursor will be replaced by crosshairs:

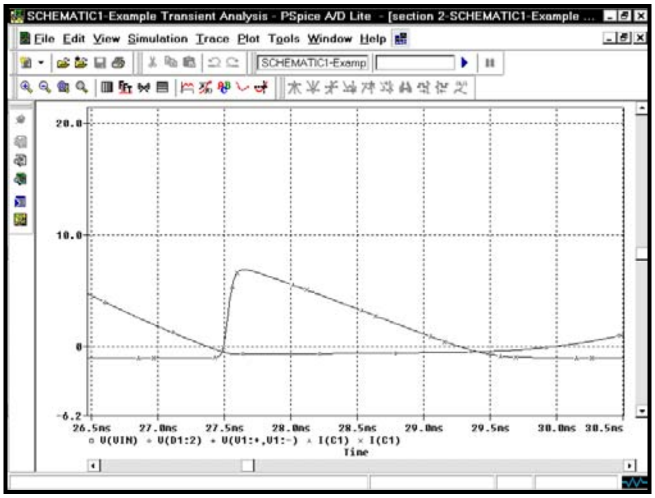




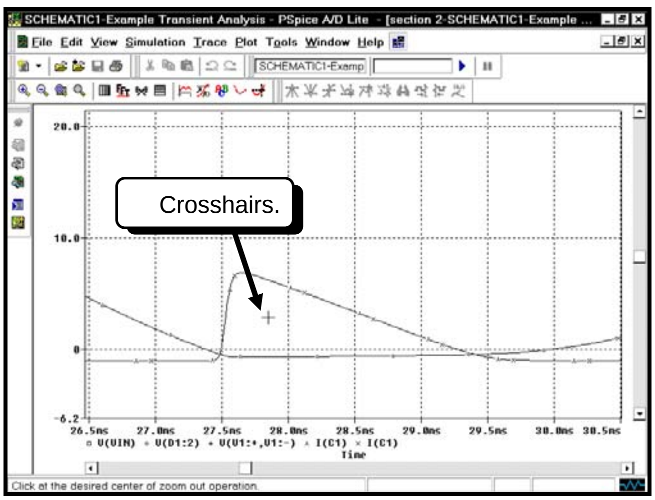
Place the crosshairs as shown above and click the ***LEFT*** mouse button. Probe will zoom in around the location of the crosshairs:

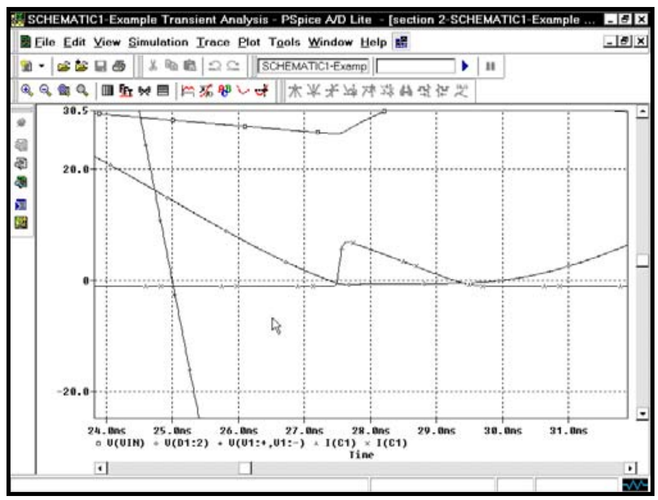


We can repeat the procedure and zoom in further. Select **View**, **Zoom**, and then **In** to obtain the crosshairs. Place the crosshairs where you would like to enlarge the plot and click the ***LEFT*** mouse button:



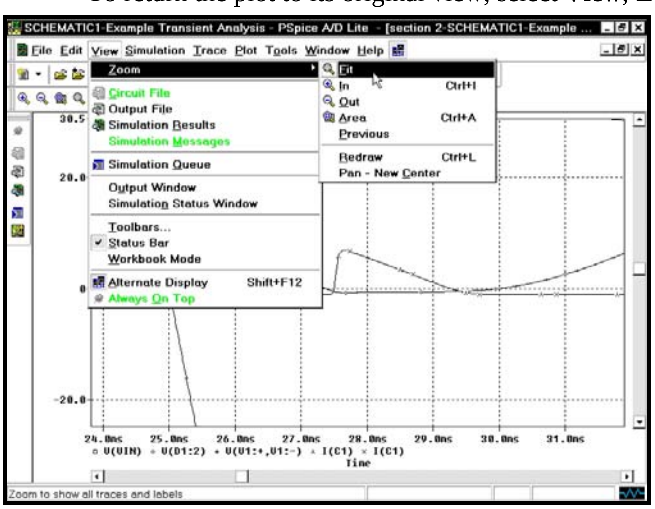
Selecting **View**, **Zoom**, and then **Out** is the opposite of selecting **View**, **Zoom**, and then **In**. Select **View**, **Zoom**, and then **Out**. Crosshairs will appear. Place the crosshairs where you would like to zoom out. When you click the ***LEFT*** mouse button, Probe will zoom out around the crosshairs:

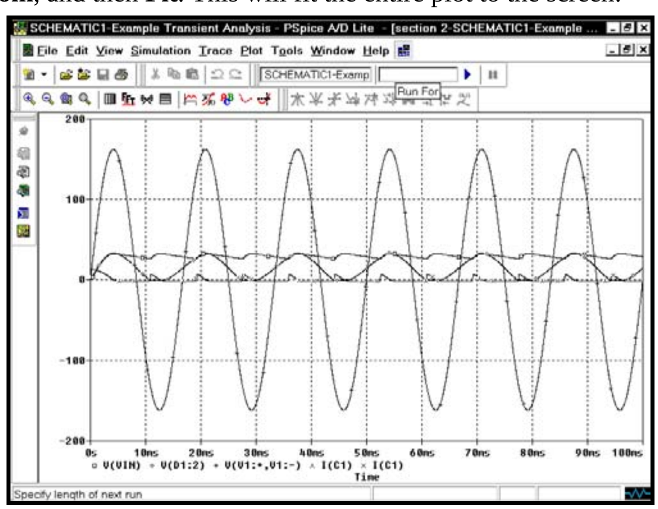




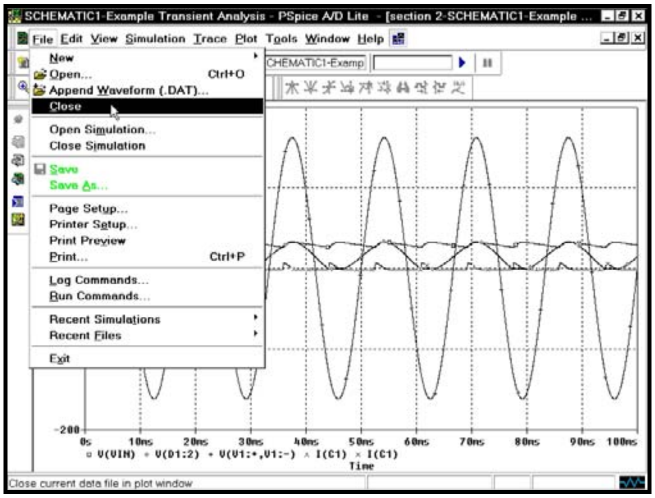
You can zoom out more if you wish.

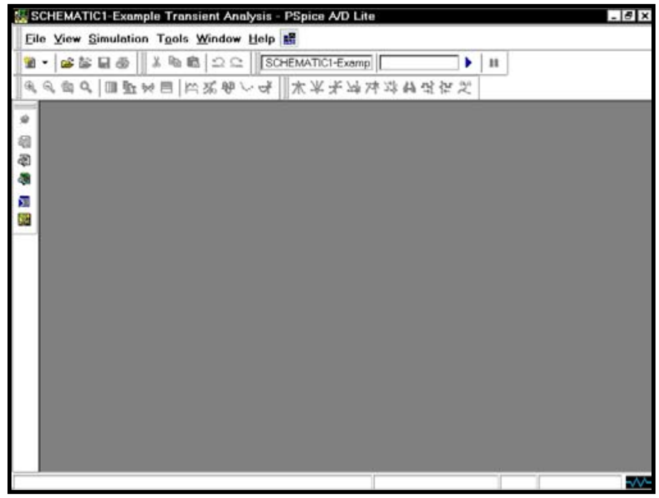
To return the plot to its original view, select **View**, **Zoom**, and then **Fit**. This will fit the entire plot to the screen:



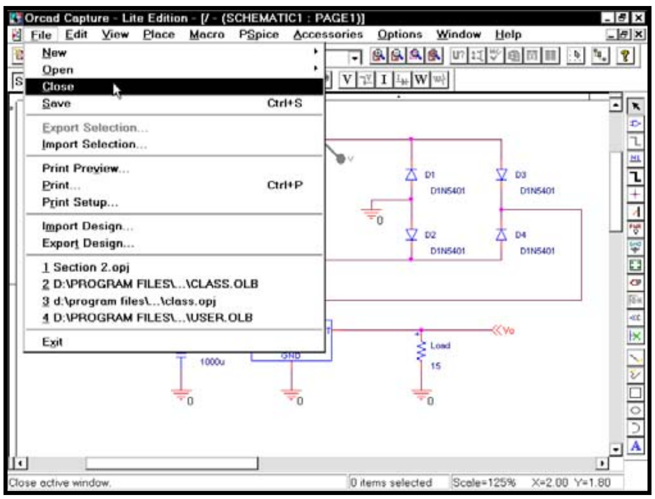


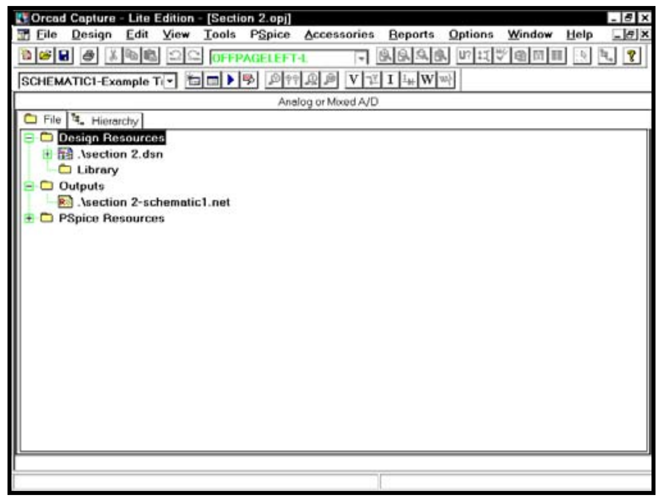
Before continuing we must close the current data file in Probe, and close the current schematic in Capture. In Probe, select **File** and then **Close** to close the data file.



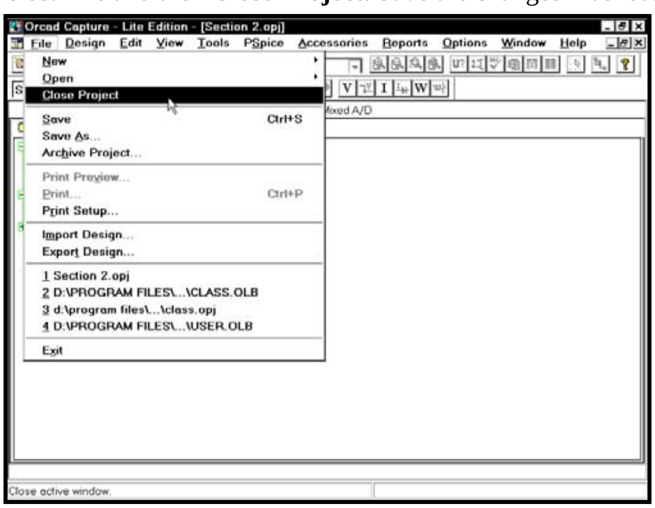


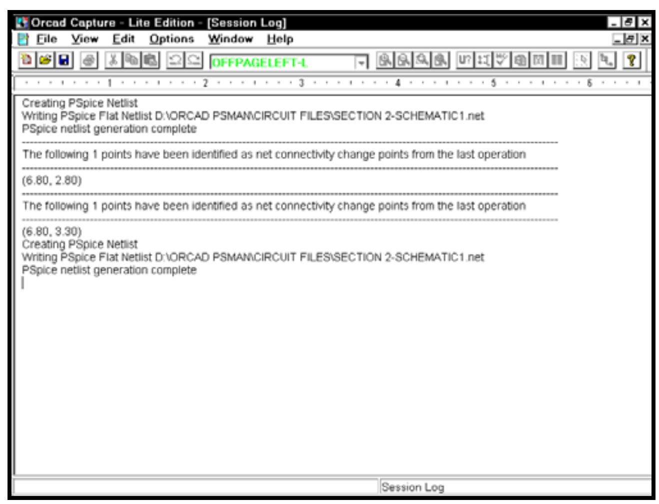
Next, select **File** and then **Close** in Capture to close the schematic. Save the changes if asked.





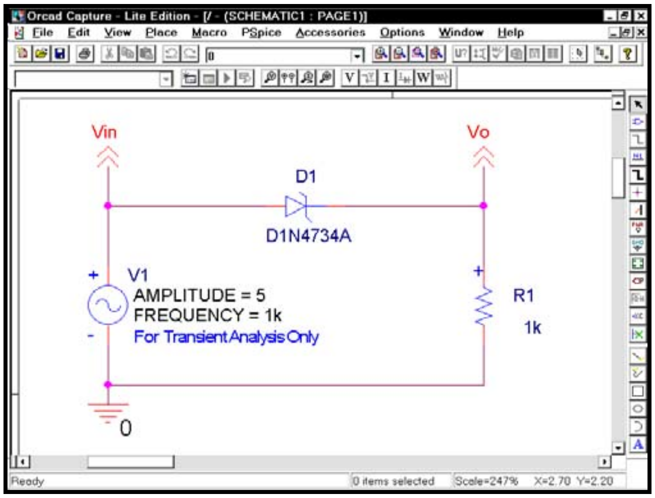
Select **File** and then **Close Project**. Save the changes if asked.



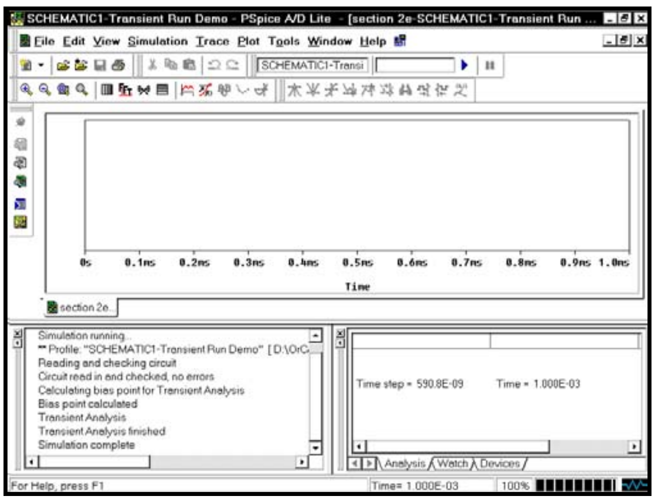


2.E. Adding a Second Y-Axis

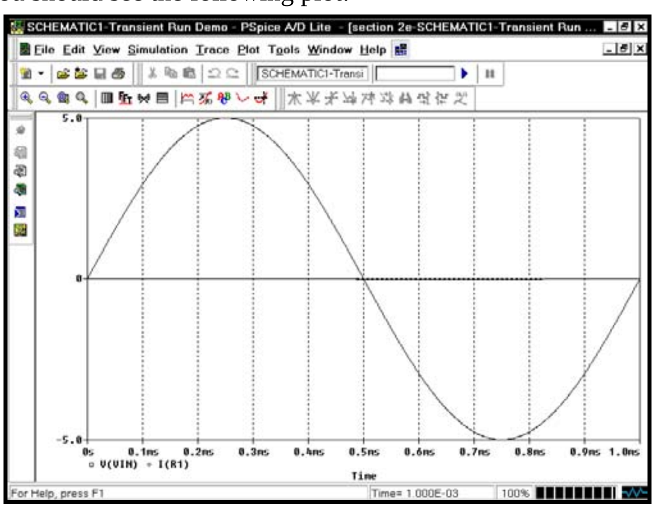
In the previous example, we had several traces on a single plot. Some of the traces became hard to see when the magnitude of the numerical values of the traces differed by large amounts. A typical example would be plotting a voltage trace and a current trace on the same plot. Typically voltage traces may be in volts and currents may be in milliamperes or microamperes. When Probe plots traces with different units on the same plot, it plots the magnitude of the numerical values on the plot. If we plot a voltage that ranges from 0 to 5 volts and currents that range from 0 to 5 mA, both traces will be plotted with a y-axis that can accommodate numerical values from 0 to 5. With this scale, the current trace will be displayed close to zero and it will be hard to see any detail. We will use the circuit below to illustrate. The name of the circuit is Section 2e.opj. Open the file from the Program Files\OrcadLite\Book Circuits directory in the same manner as we did with file Section 2.opj at the beginning of this chapter. See pages 94–95 for a detailed procedure of opening a file from this directory. The schematic is shown below:



Press the F11 key to simulate the circuit and run Probe:



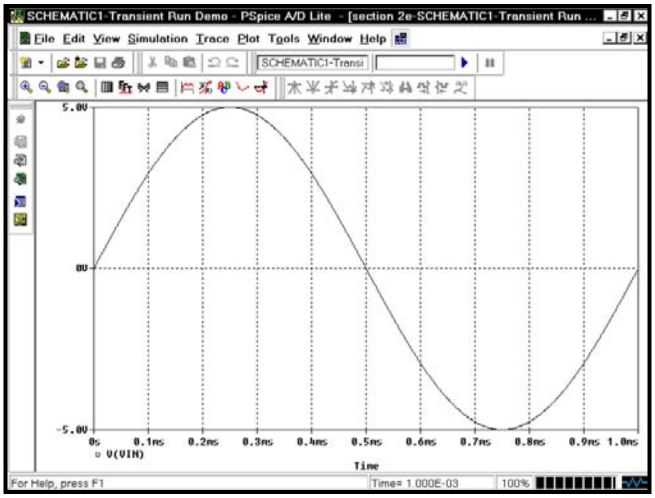
Add traces V(Vin) and I(R1). You should see the following plot:



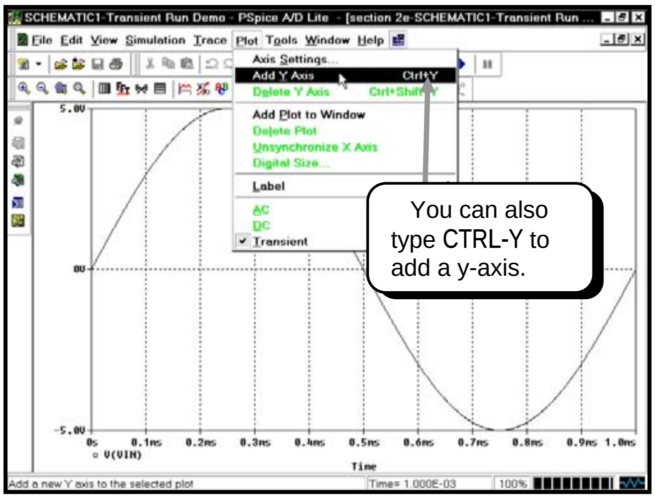
We see that the trace of I(R1) looks like it is constant at zero. This is because the numerical values of the current are 1,000 times smaller than those of the voltage. Delete the trace I(R1):*

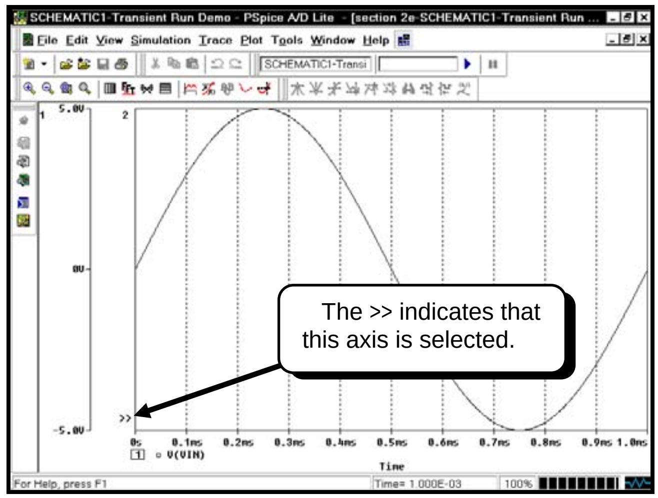
---

*See page 106 for deleting traces.

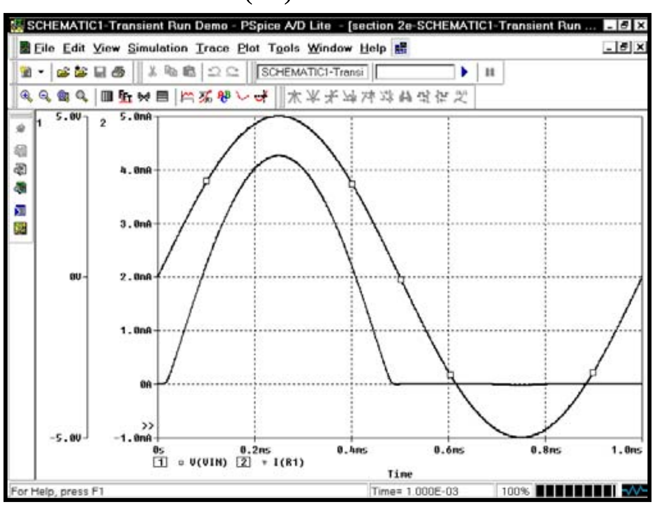


To fix this display problem, we will add a second y-axis. One y-axis will be used for the voltage trace and the second y-axis will be used for the current trace. The advantage of this arrangement is that the two y-axes can have different scales. To add another y-axis, select **Plot** and then **Add Y Axis**:

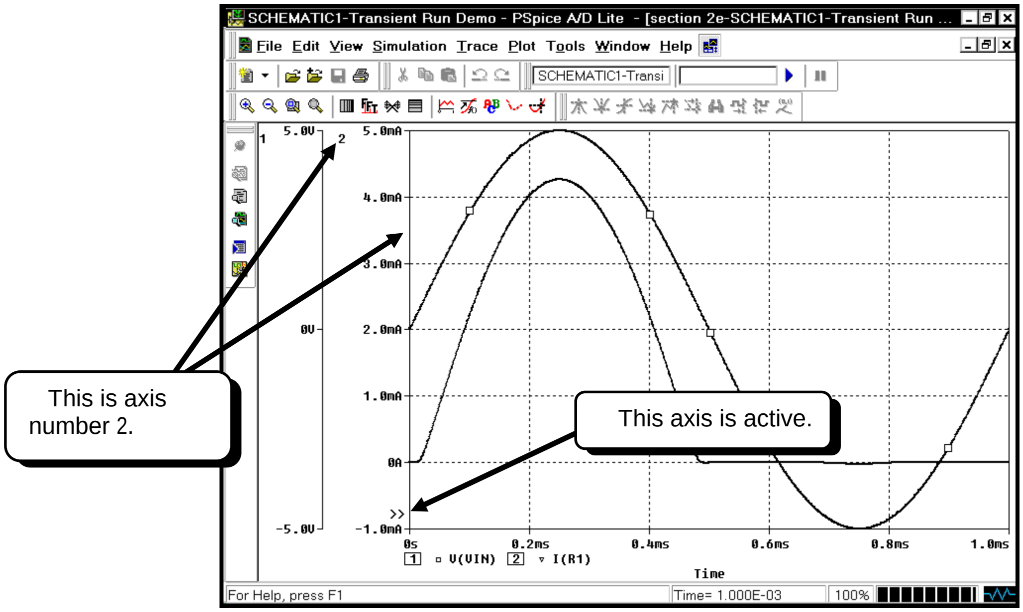




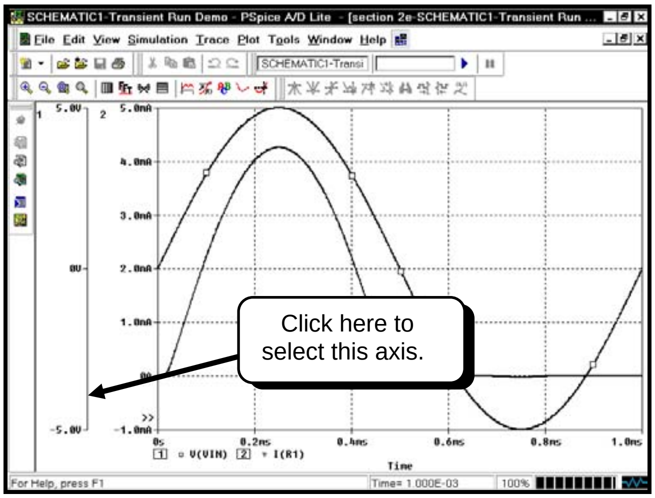
Note in the screen capture above that the axis we just added is selected. The next trace we add will be displayed using the selected axis, in this case, the new axis. Add the trace I(R1):

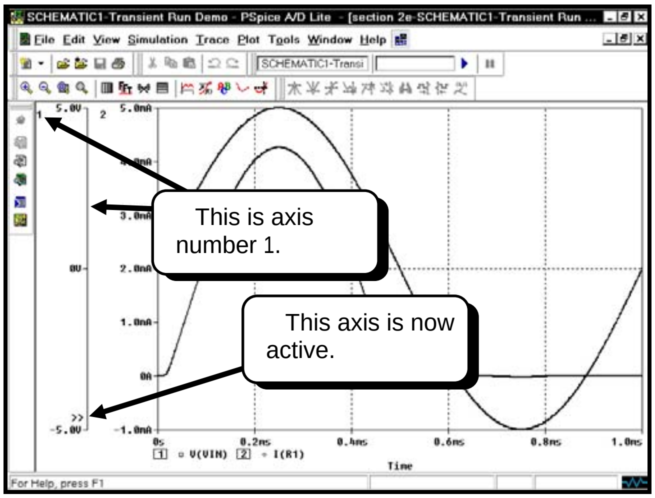


Traces that are added are placed on the selected y-axis. Below, the >> symbol indicates that the second y-axis is selected:

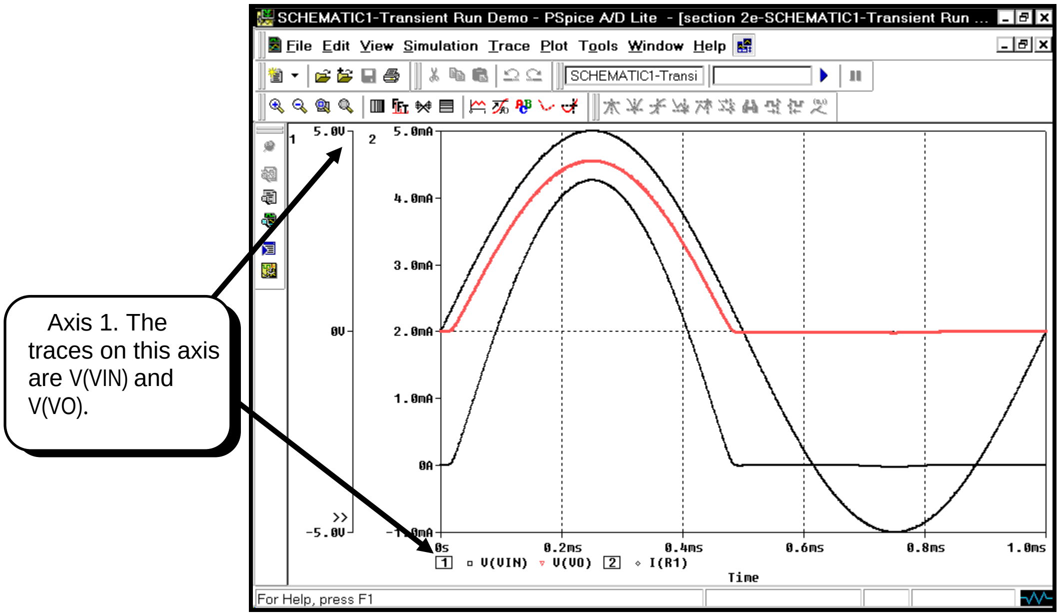


Suppose we want to plot V(Vo). This plot would fit best on the first y-axis. To select a y-axis, click the ***LEFT*** mouse button on the axis:





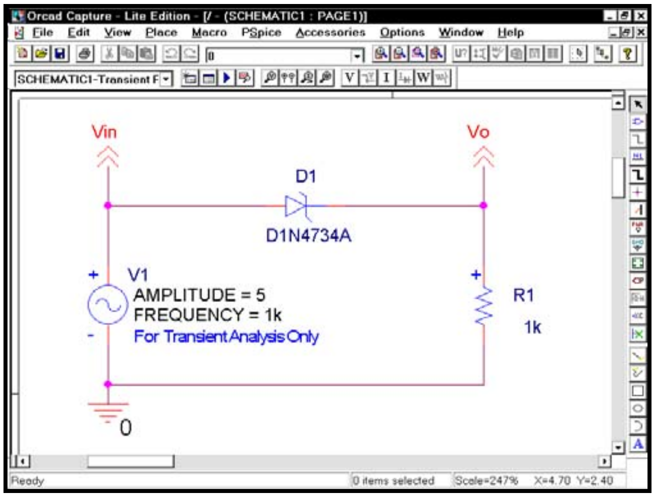
Now that the first y-axis is selected, we can add trace V(VO):



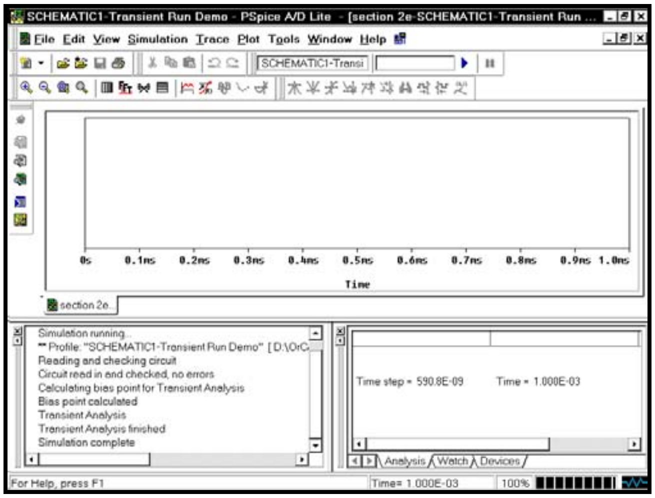
Note that trace V(VO) is placed in the list of traces for y-axis 1.

2.F. Adding Plots

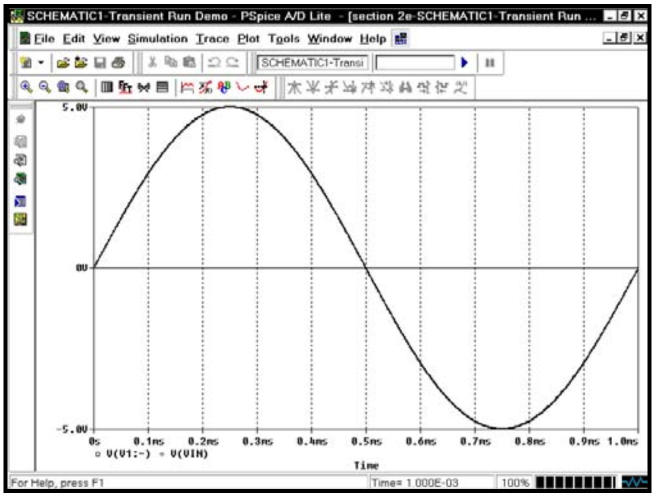
We will use the circuit of the previous example to illustrate how to display multiple plots on the same window. Switch back to Capture using the ALT - TAB key sequence.* We will start with the schematic:



Press the F11 key to simulate the circuit and run Probe:

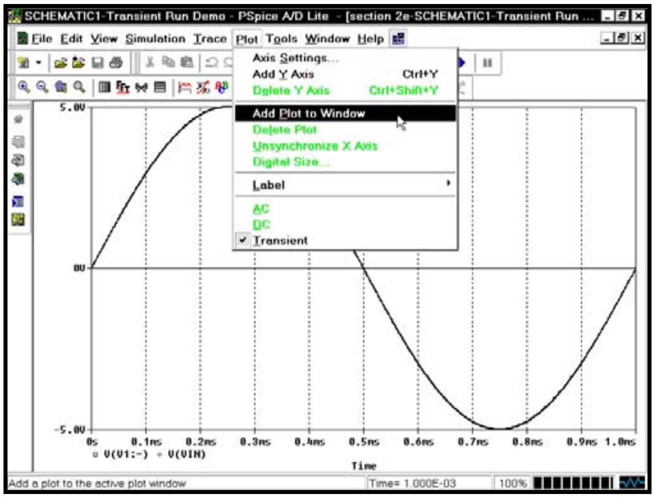


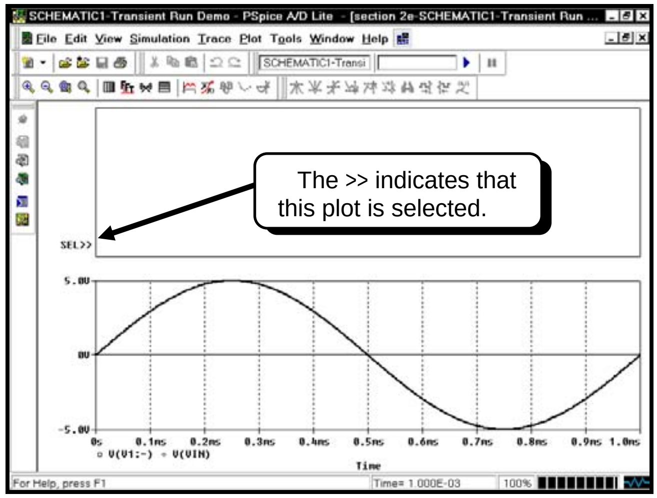
Add the trace V(Vin):



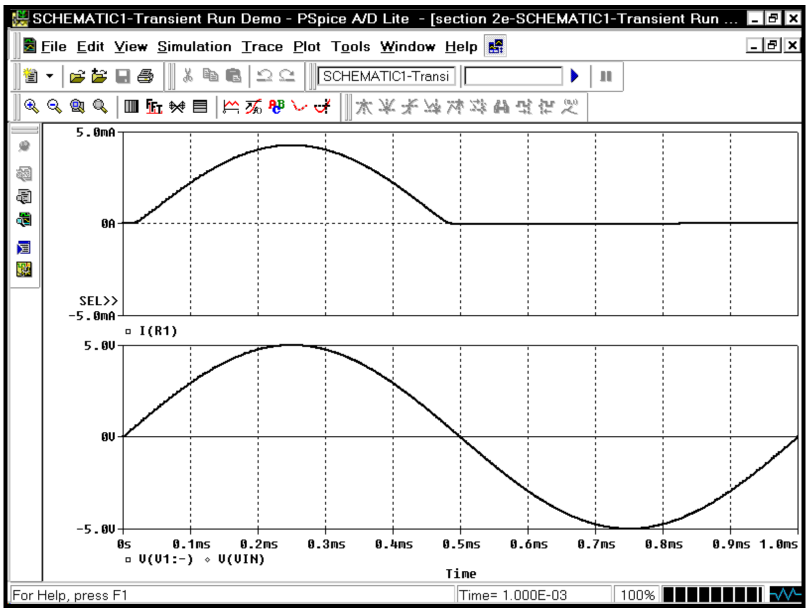
We will now create a second plot. Select **Plot** and then **Add Plot to Window**:

* See page 107 for a more detailed description of how to switch between windows using this method.

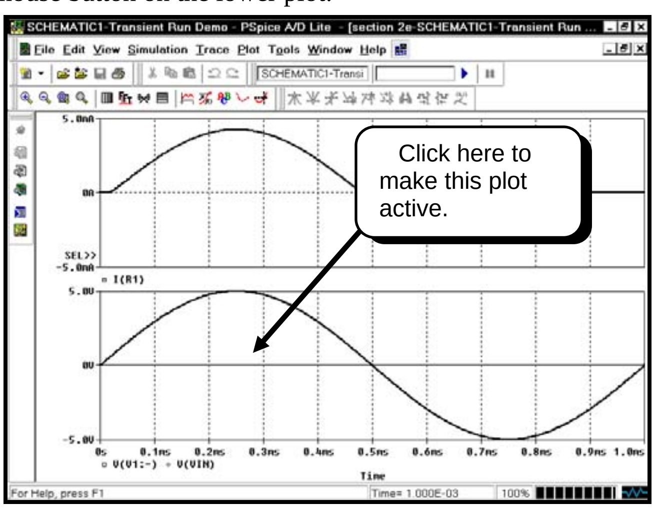


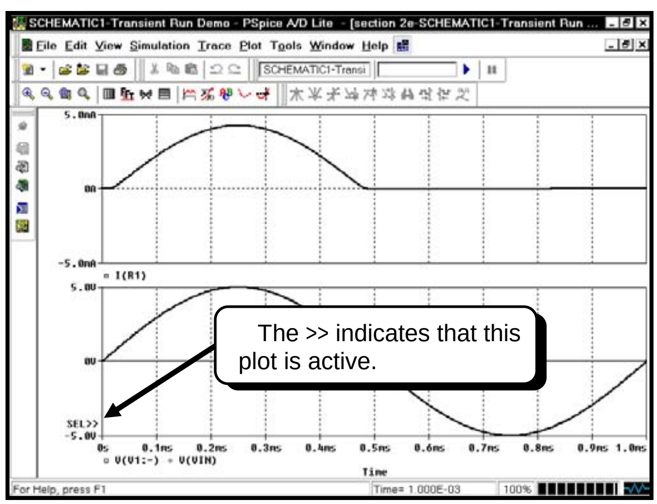


Notice that the top plot is selected. All new traces are added to the selected plot. Add the trace I(R1):

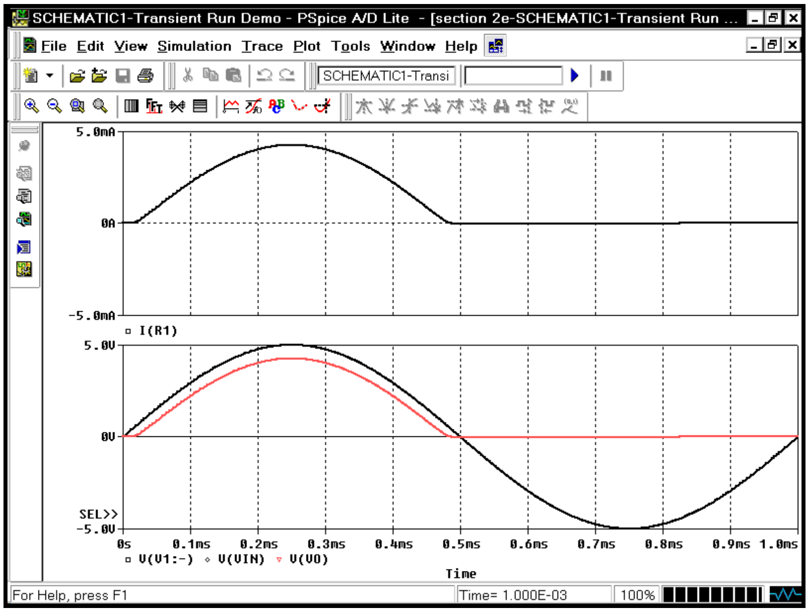


We would now like to add the trace V(Vo) to the lower plot. To select the lower plot and make it active, click the ***LEFT*** mouse button on the lower plot:

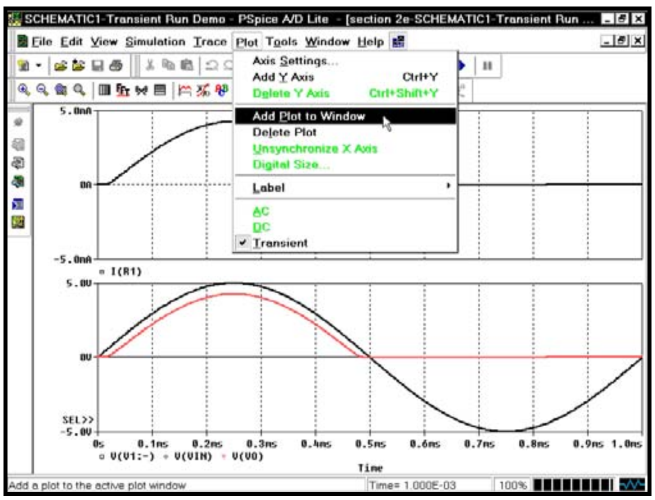


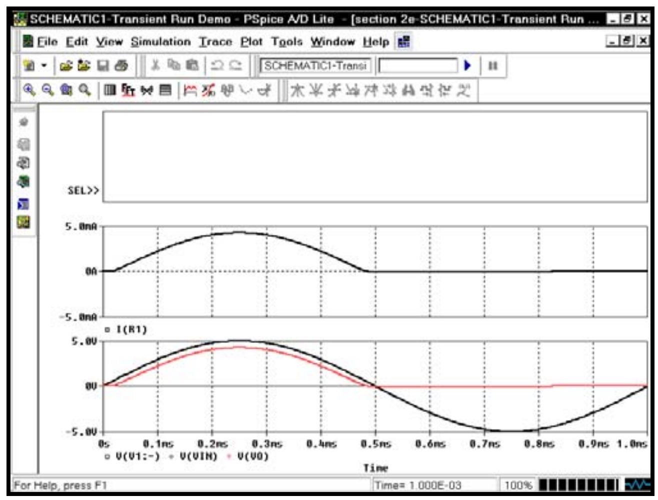


The bottom plot is now selected. Add trace V(Vo):



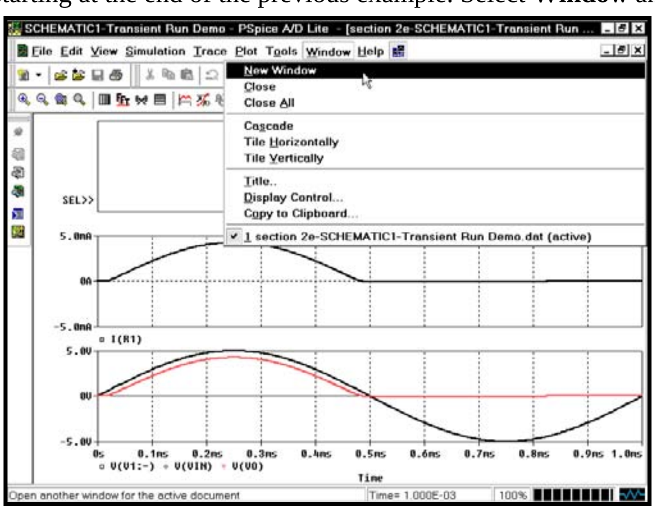
We see that the newly added trace is placed on the selected plot. We can add more plots to this page if we wish. Select **Plot** and then **Add Plot to Window**. A third plot will be added to the window:

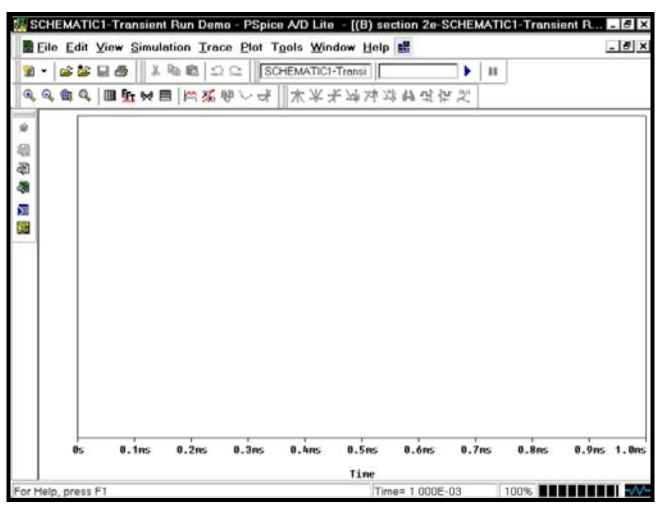




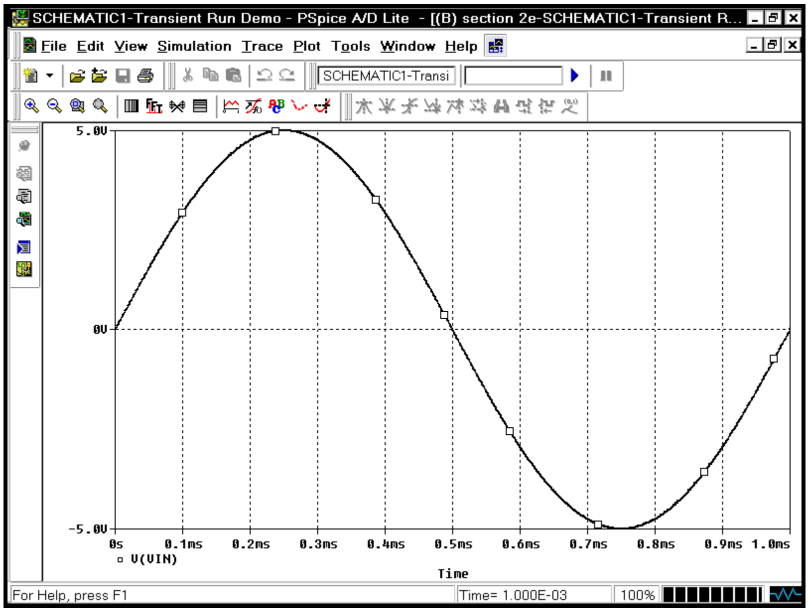
2.G. Adding a Window

Probe has the ability to display multiple windows. Each window can display different traces. We will continue now, starting at the end of the previous example. Select **Window** and then **New Window** to create a new window:

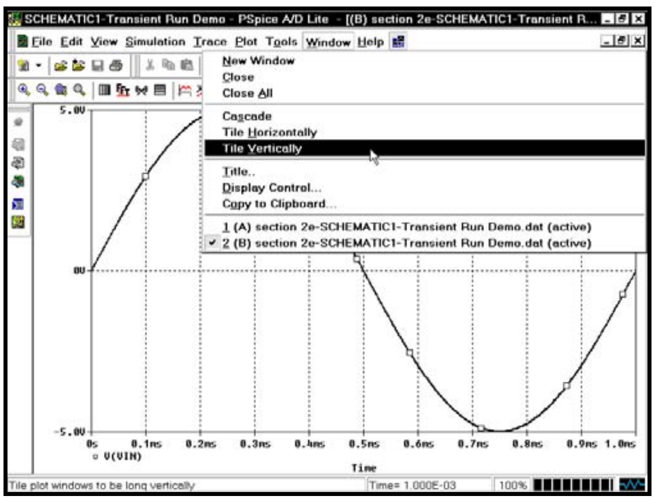


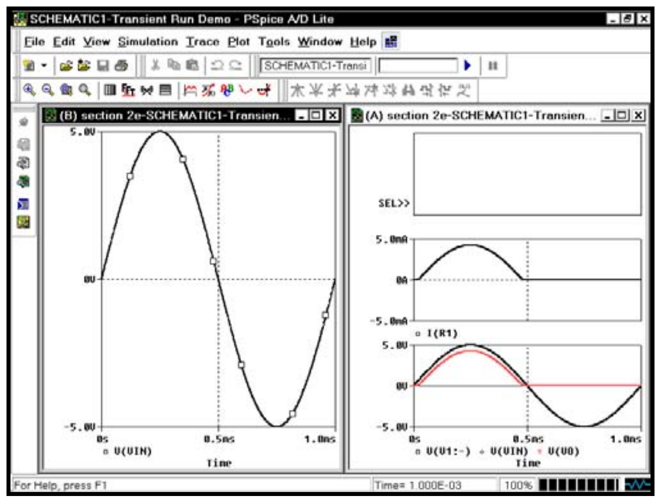


Add the trace V(Vin):

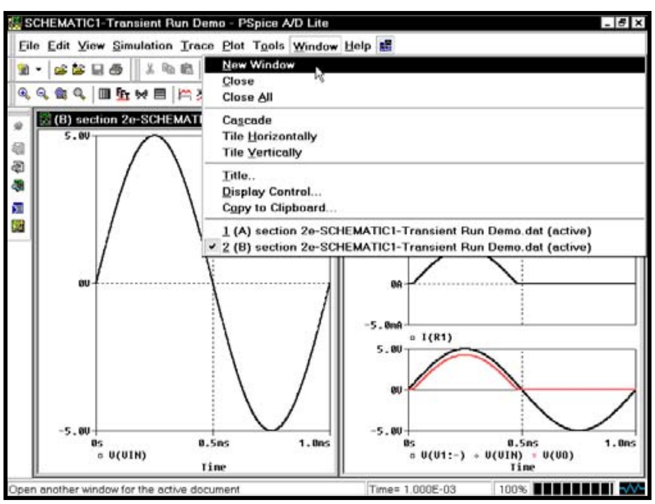


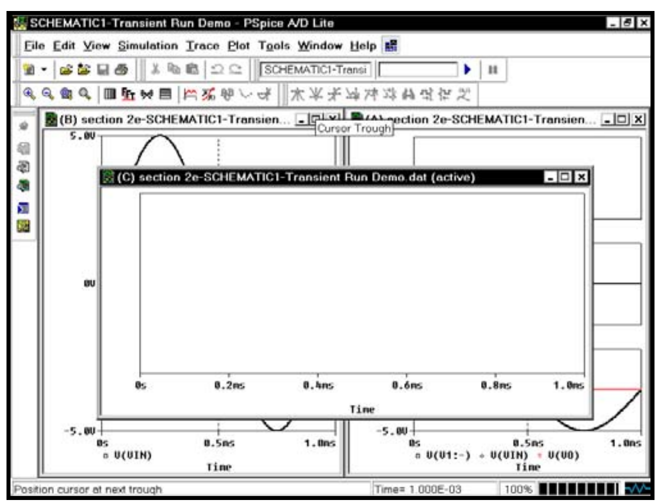
Although we can see only one window, there are two windows open. To display both windows at the same time, select **Window** and then **Tile Vertically**:



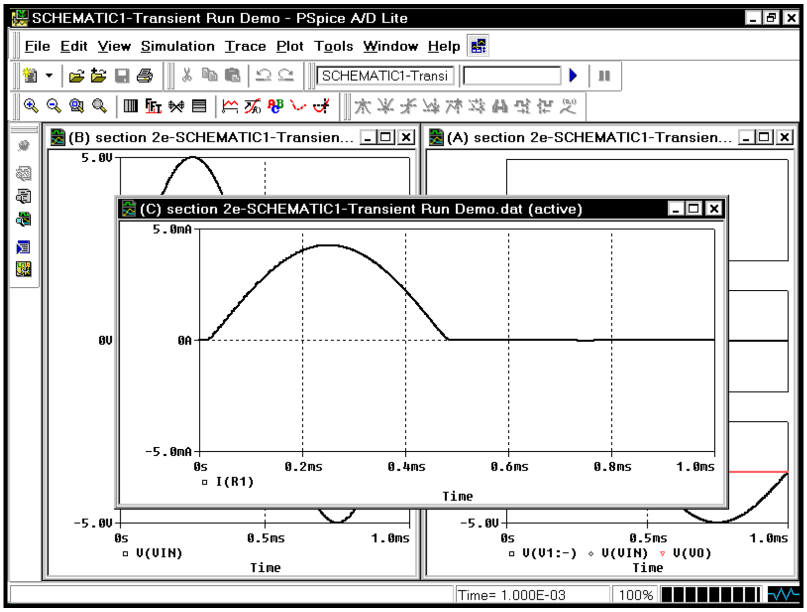


We will add a third window. Select **Window** and then **New Window**:

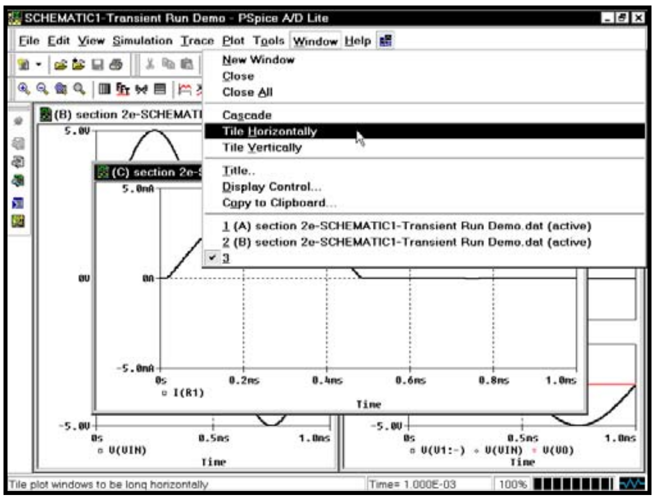


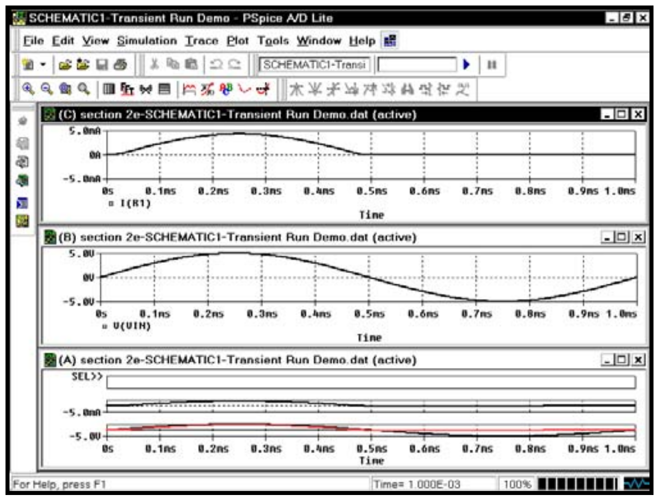


Add the trace I(R1):

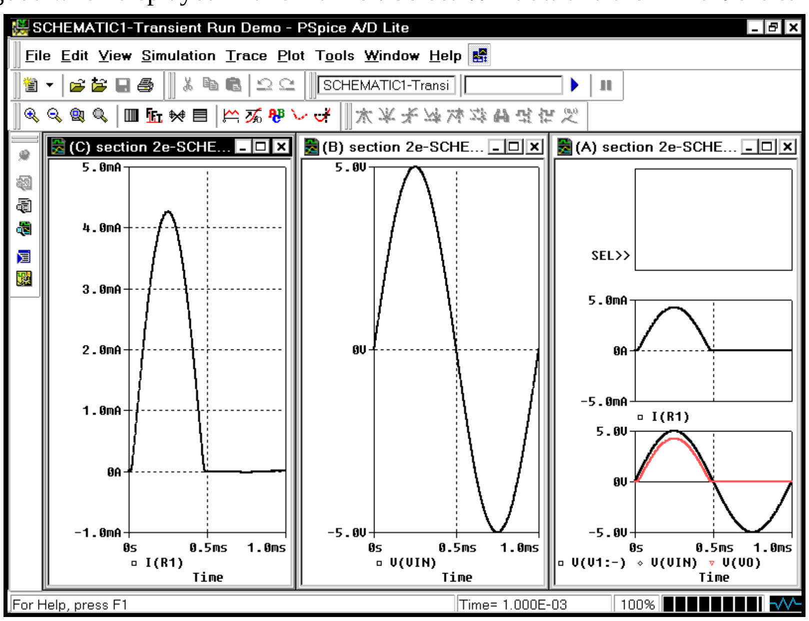


We would now like to display all windows at the same time. Select **Window** and then **Tile Horizontally**:

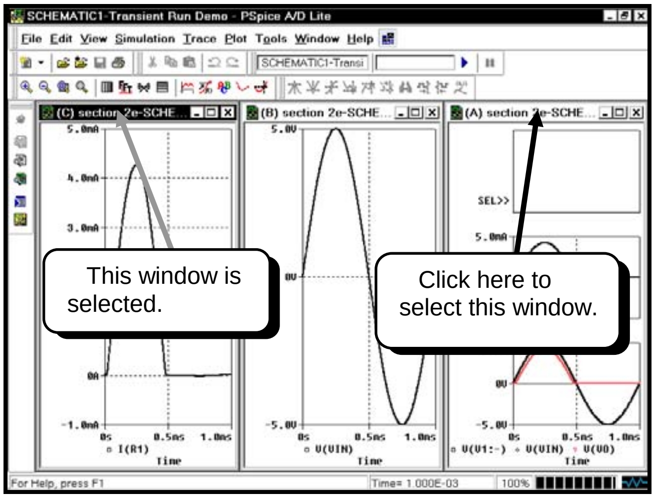


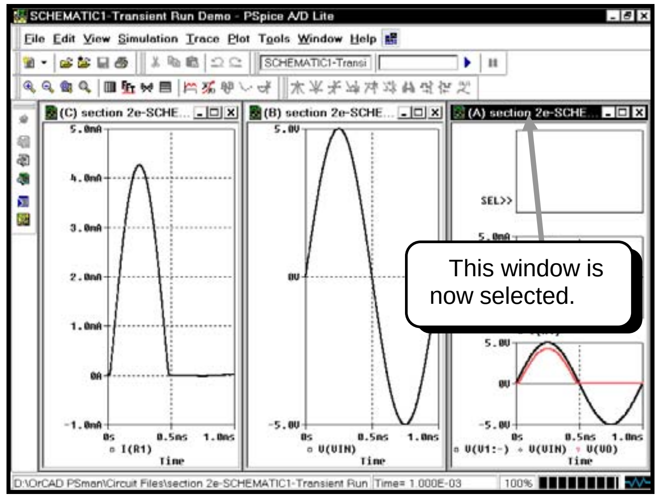


These plots do not look good when displayed in this manner. Select **Window** and then **Tile Vertically**:

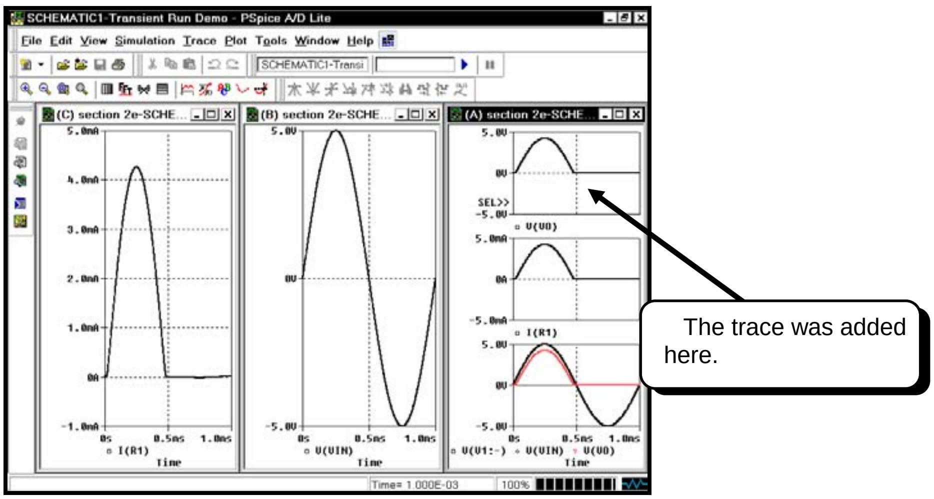


New traces are added to the selected window. To select a window, click the ***LEFT*** mouse button on the window you wish to use. For example, click the ***LEFT*** mouse button on the rightmost window. It will become selected:

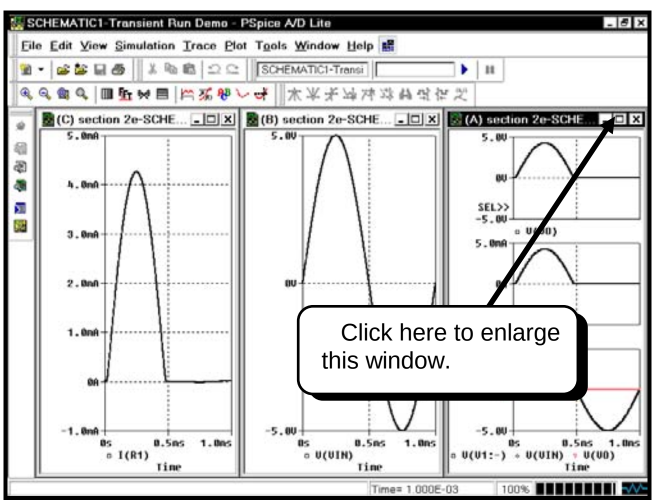


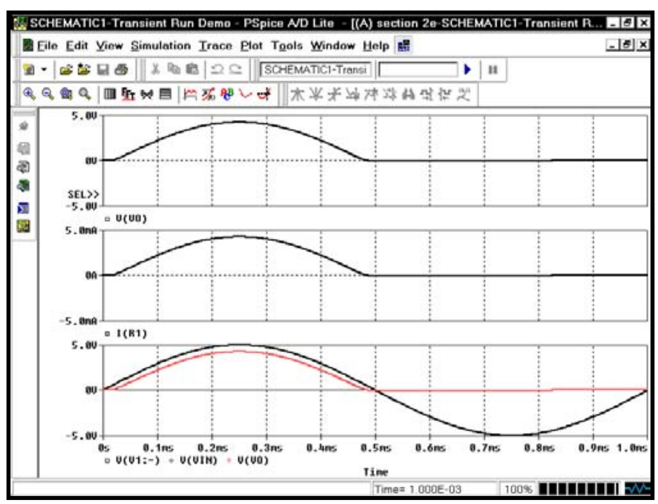


Add the trace V(Vo) to the selected window:

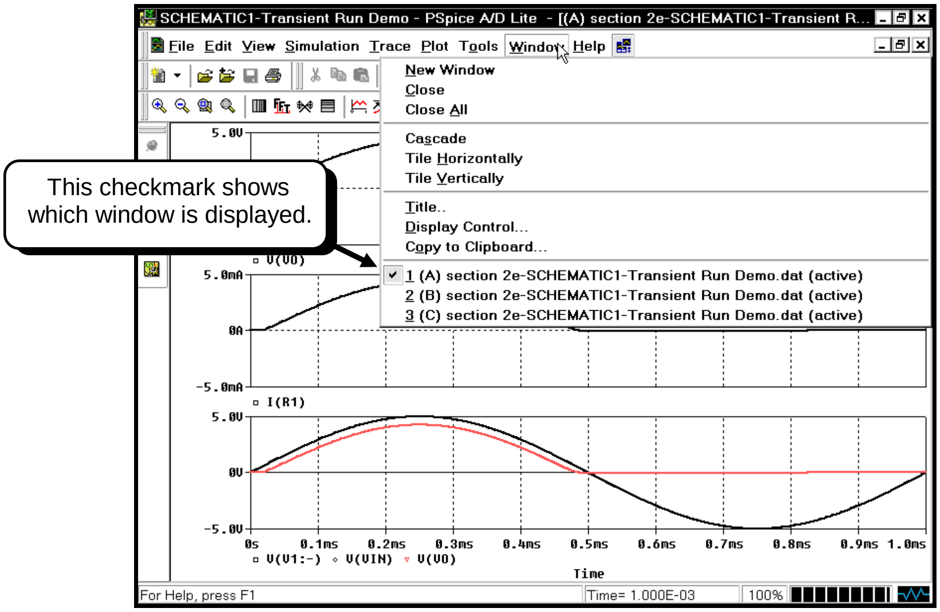


Notice that the trace is added to the active plot in the active window. To enlarge a window, click the ***LEFT*** mouse button on the maximize icon as shown below:

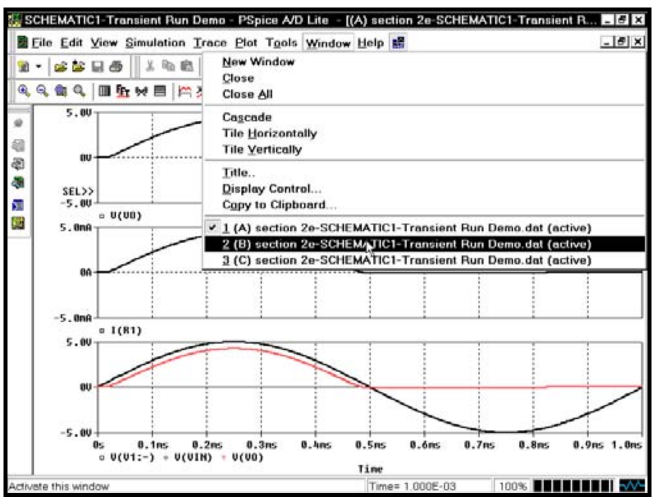


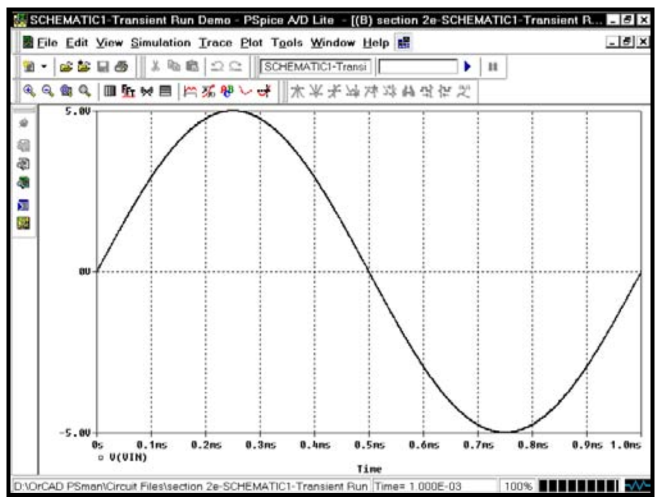


To switch between windows, we can use two methods. All of my screen captures shown recently show the Probe window occupying the entire window. In this mode we use the menus to switch between windows. Select **Window** from the Probe menus. The pull-down menu displays which window is currently selected and allows us to select a different window:

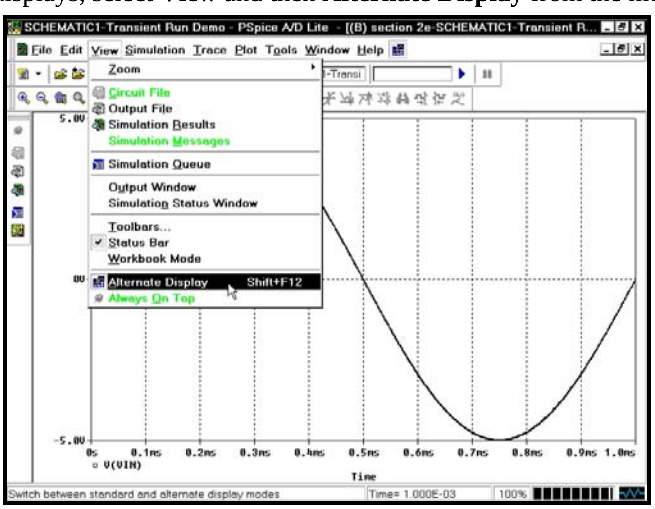


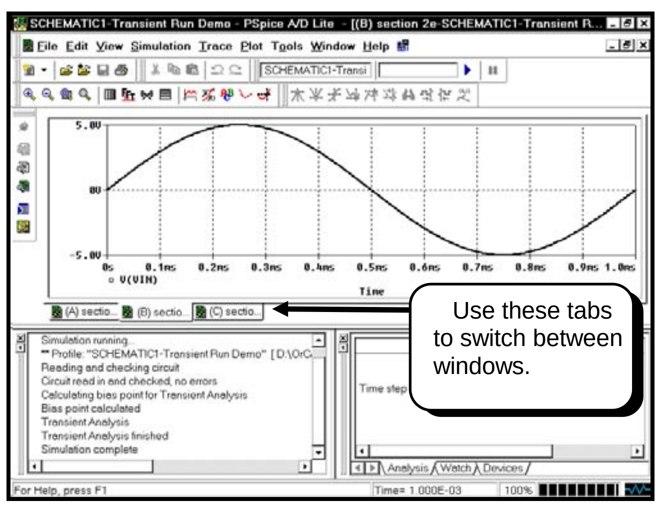
The checkmark shows that window A is selected. You can use the windows to switch to a different window. Below, we will display window B:





The second method of switching between windows requires us to use the alternate display. To toggle between displays, select **View** and then **Alternate Display** from the menus:

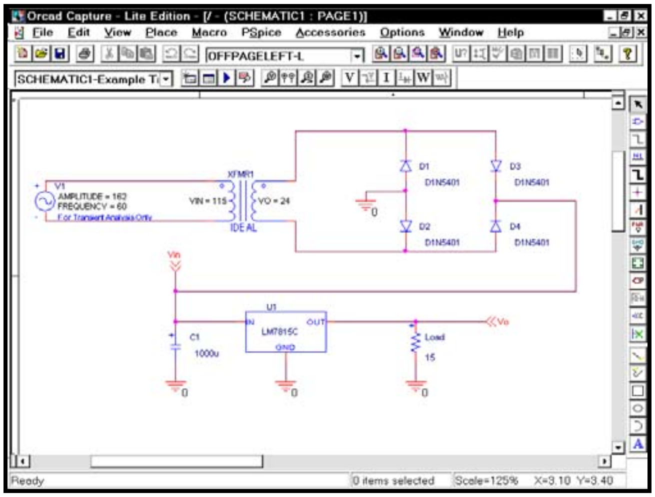




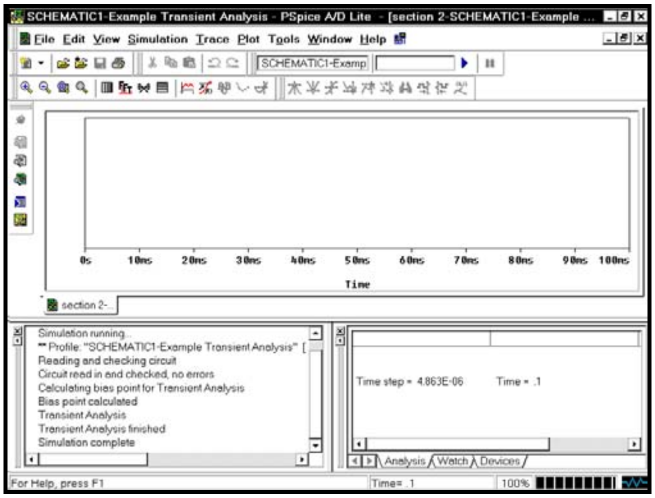
You can now use the tabs to switch between the windows. We will switch back to the previous display to show screen captures in full screen for the remainder of this text.

2.H. Placing Text on Probe's Screen

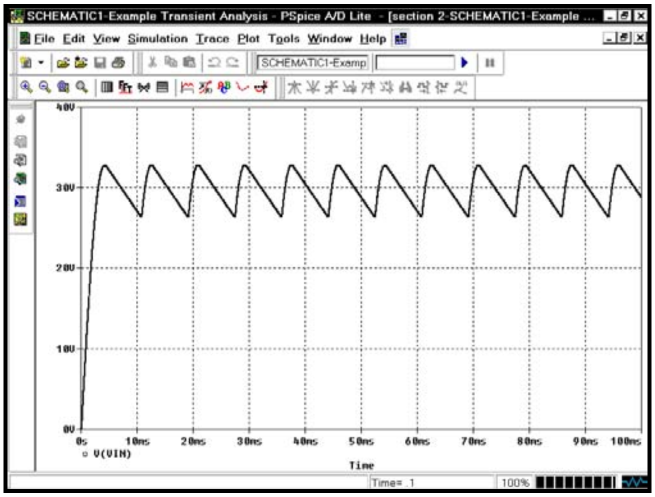
We will use the circuit named Section 2.opj. Open this file:*



Press the F11 key to simulate the circuit and then run Probe:



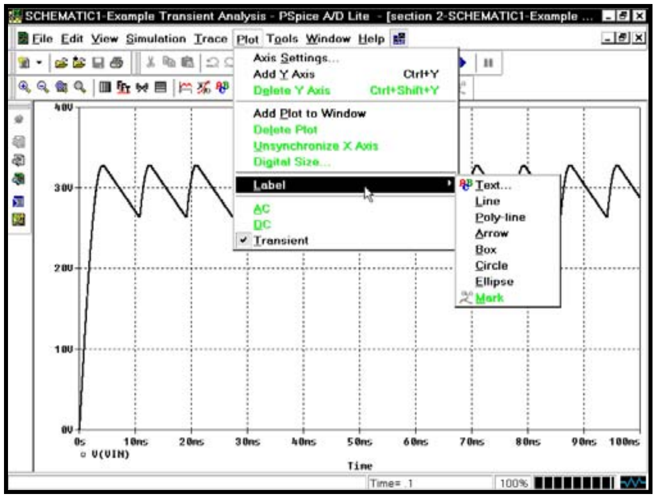
Add the trace V(Vin):



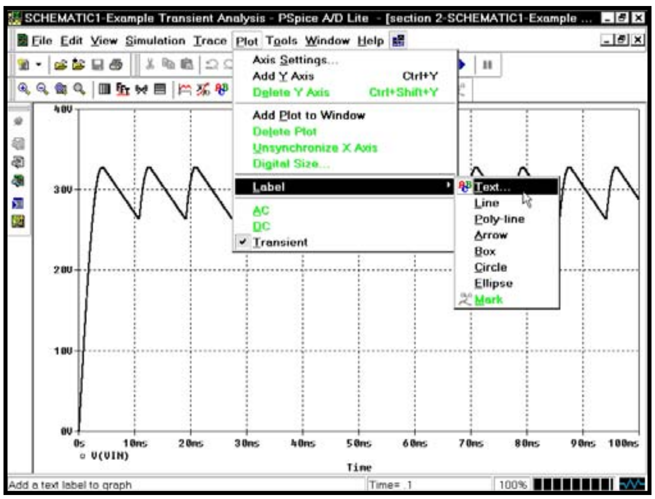
We can place text on the screen by using the menus or by using the button bar. We will first use the menus. Select **Plot** and then select **Label**:

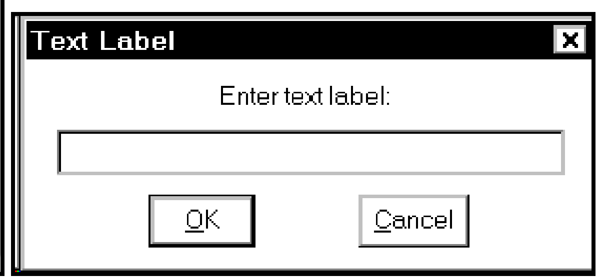
---

*We used this circuit at the beginning of the chapter. Follow the instructions on pages 94–95 to open the file. If you saved changes to file Section 2.opj earlier, your Probe screen may display some waveforms when Probe starts.

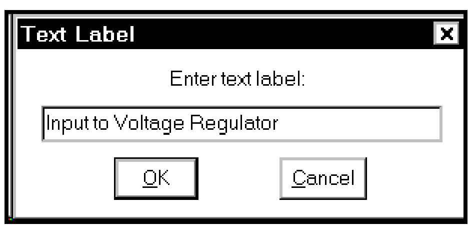


Select **Text**:

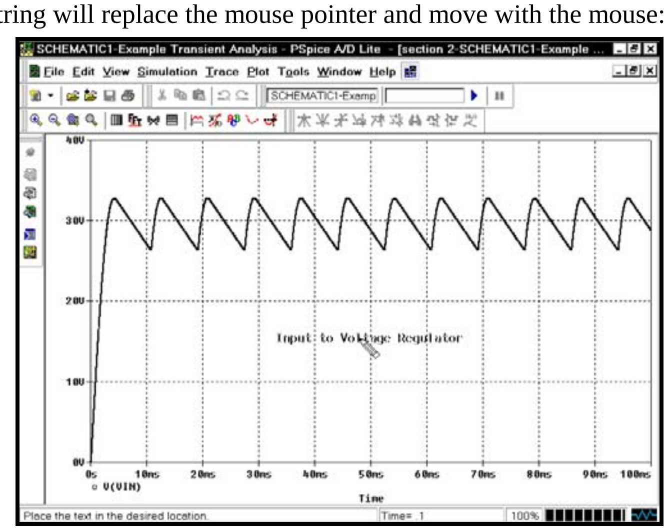




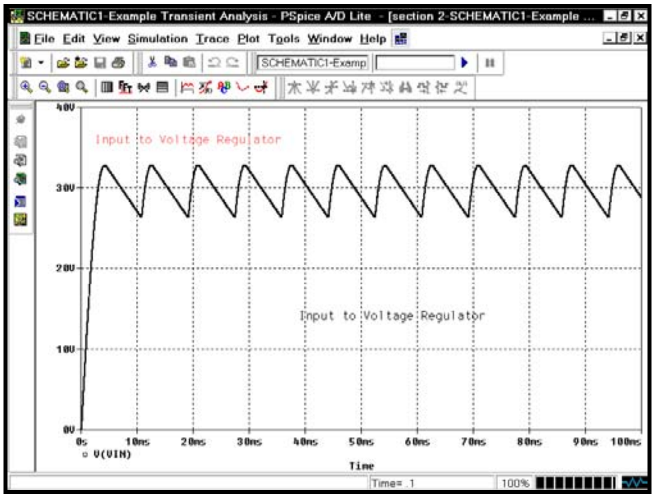
Type the text string you would like to display. Type **`Input to Voltage Regulator`**:



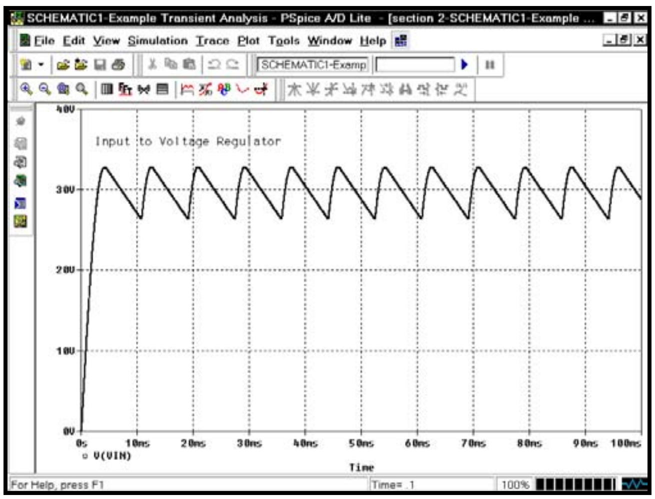
Click the OK button. The text string will replace the mouse pointer and move with the mouse:



Position the text as shown below and click the ***LEFT*** mouse button. The text will be placed and the normal mouse pointer will return:

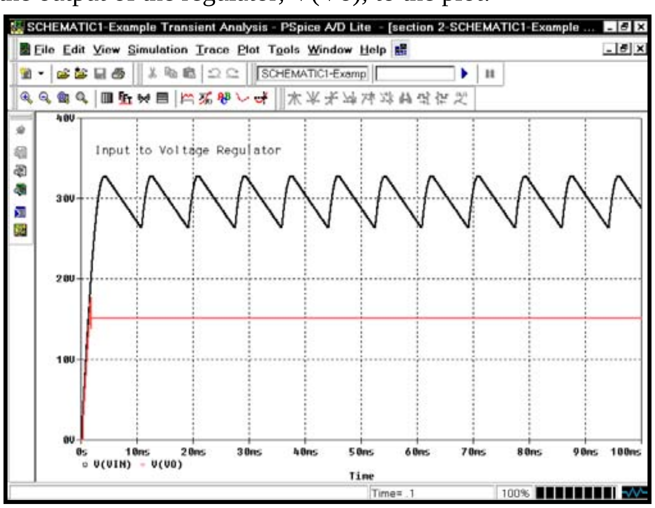


There may be fragments of text on your screen. To clear these fragments, type **CTRL-L**:

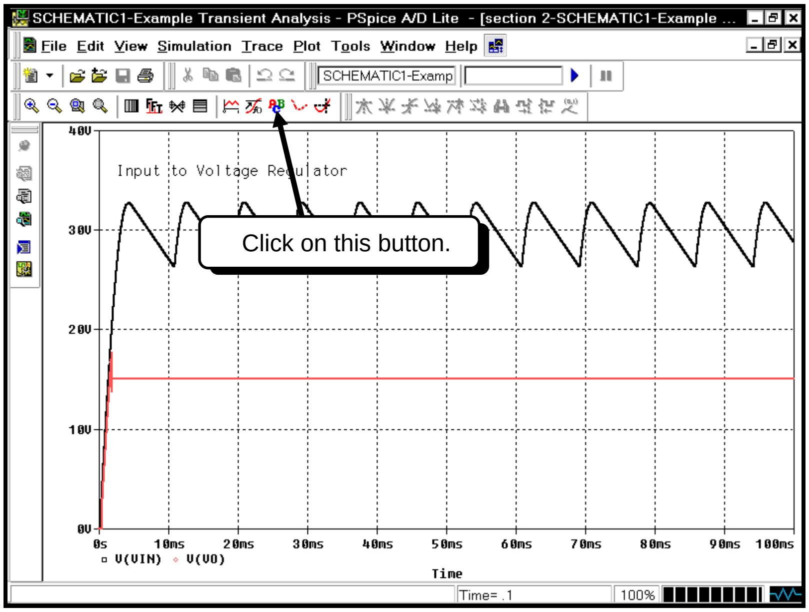


Any text now displayed on your screen is actually on your screen.

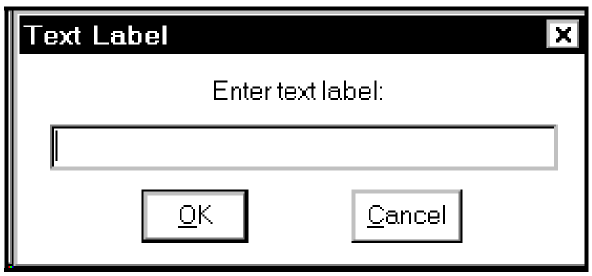
Next, add the trace for the output of the regulator, V(Vo), to the plot:



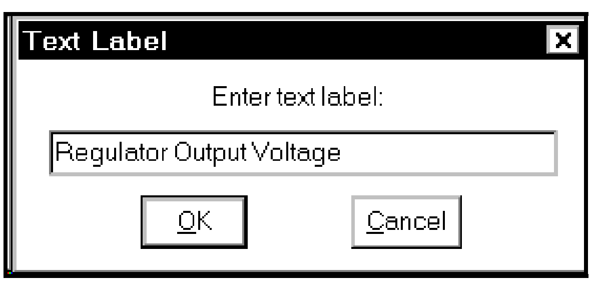
The second method of adding text is to click the ABC button  as shown below:

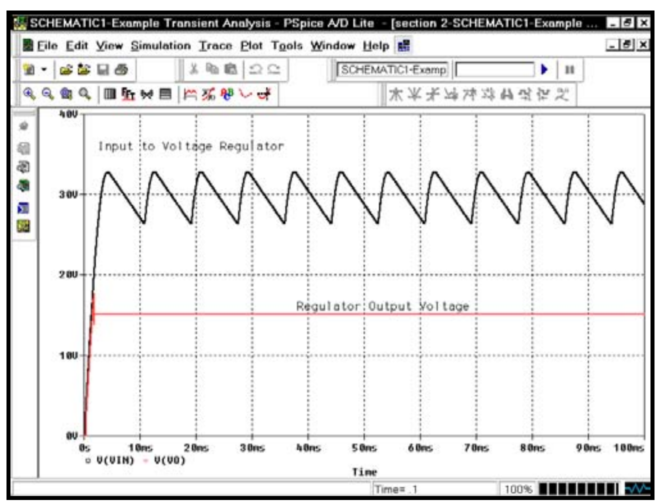


After clicking the ABC button  the Text Label dialog box will open:



Type the text **`Regulator Output Voltage`** and press the ENTER key. Position the text as shown and click the ***LEFT*** mouse button:

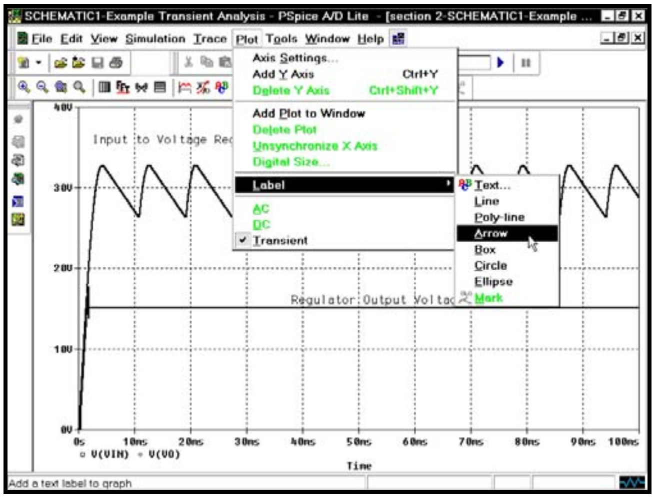


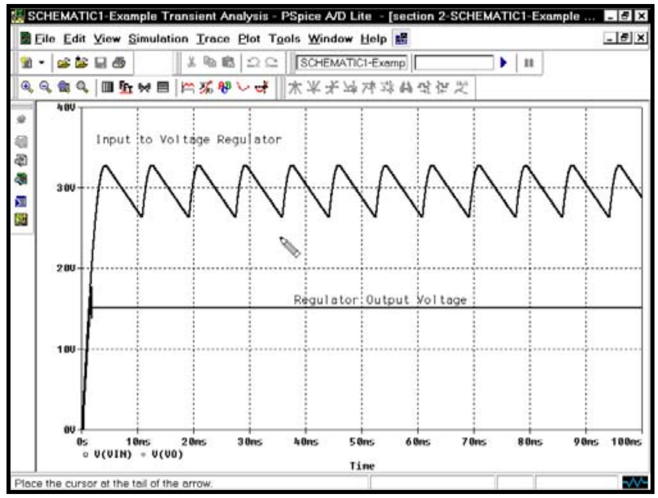


You may need to type **CTRL-L** to redraw the screen to remove text fragments.

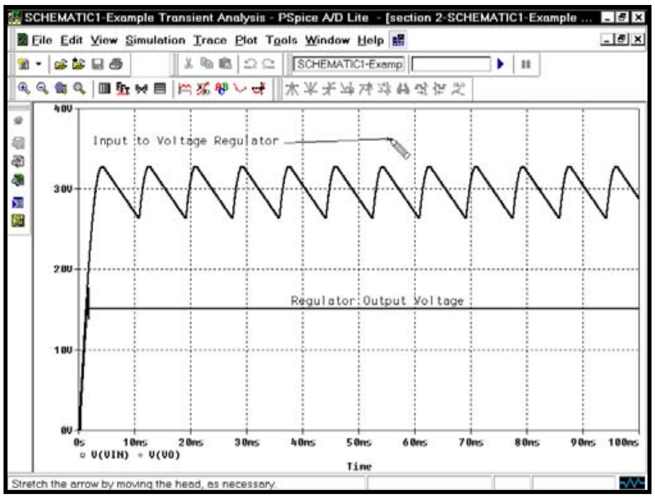
2.I. Placing Arrows on the Screen

We will now place arrows on the previous plot to point from the text to the appropriate traces. To add an arrow select **Plot**, **Label**, and then **Arrow.** The mouse pointer will be replaced by a pencil  :

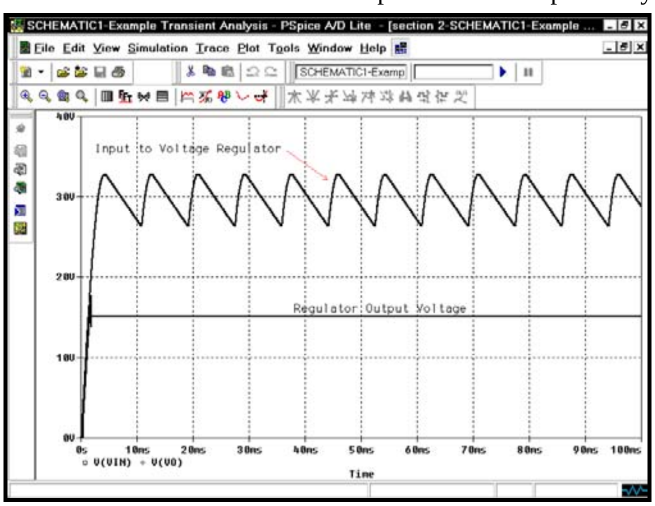




Position the point of the pencil at the place where you would like the tail of the arrow to start. In our example, we place the pencil to the right of the text Input to Voltage Regulator. Click the ***LEFT*** mouse button (do not click and hold the button). An arrow will appear on the screen and change its length as you move the mouse:

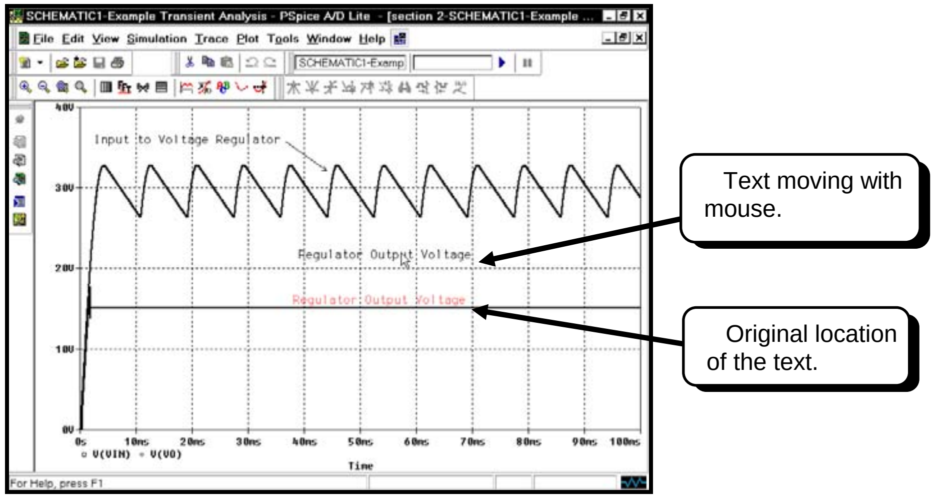


Position the head of the arrow as shown below and click the ***LEFT*** mouse button to place the arrow. When you click the ***LEFT*** mouse button the arrow will be drawn on the screen and the pencil will be replaced by the normal mouse pointer.

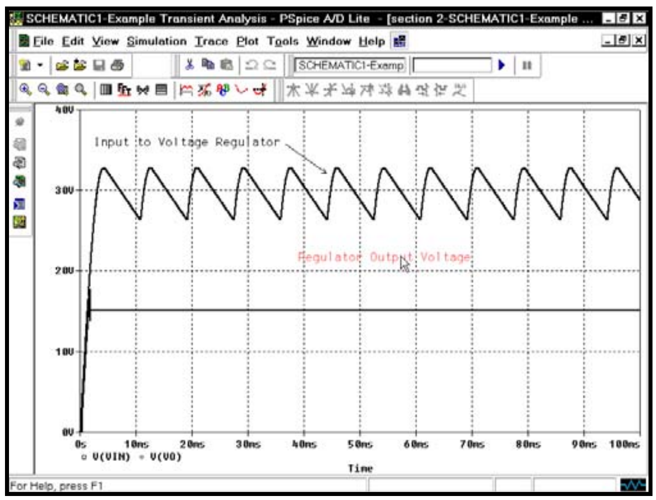


2.J. Moving Items on the Screen

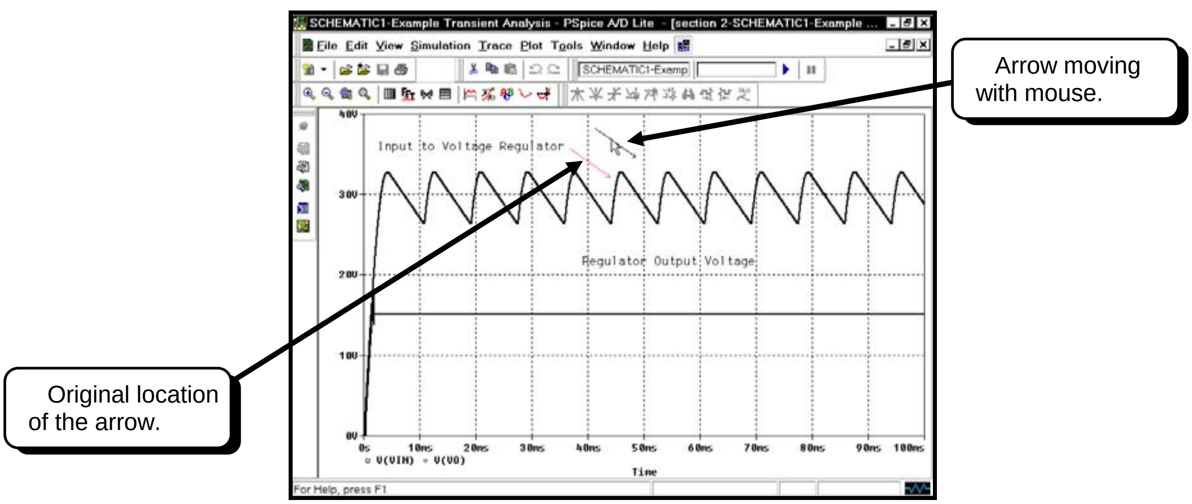
Items that we place on the Probe screen can be moved with the same techniques we use to move items in Capture. We will first move the text Regulator Output Voltage. Click the ***LEFT*** mouse button on the text Regulator Output Voltage. The text should turn red, indicating that it is selected. Next, click and drag the selected text to a new location:



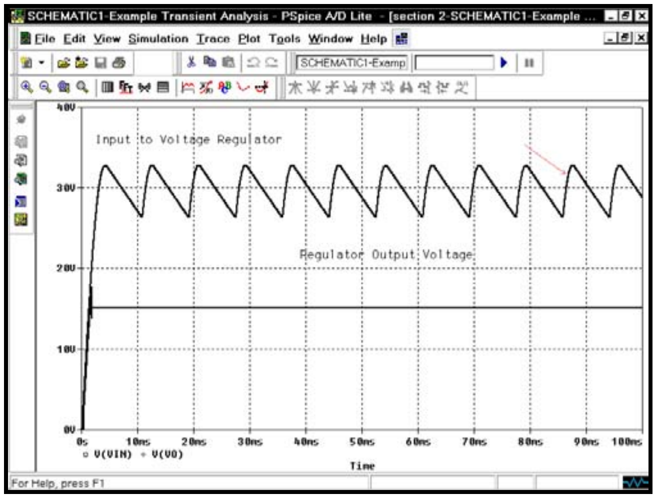
Position the text in a convenient location and release the mouse button to place the text:



We can use the same technique to move the arrow. Click the ***LEFT*** mouse button on the arrow to select the arrow. It should turn red, indicating that it is selected. Next, place the mouse pointer at the center of the selected arrow and click and drag the arrow to a new location:

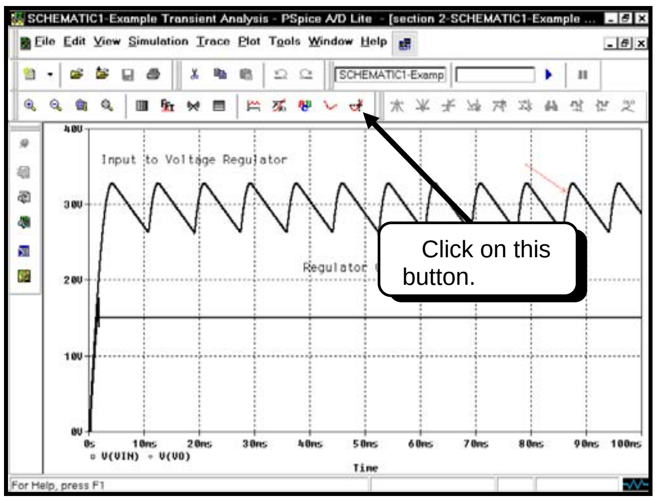


Position the arrow as shown below and release the mouse button to place the arrow:

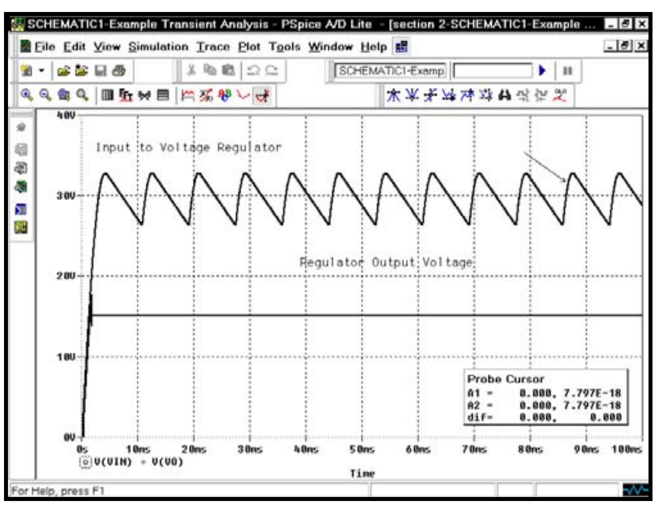


2.K. Using the Cursors

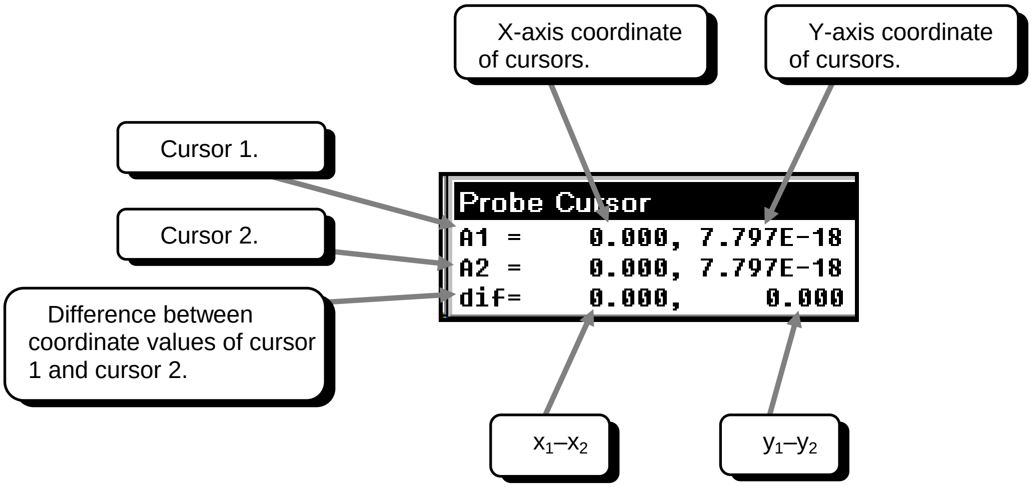
The cursors can be used to obtain numerical values from traces. To display the cursors, click the cursor button  as shown below:



After clicking the button, the cursors will be displayed:

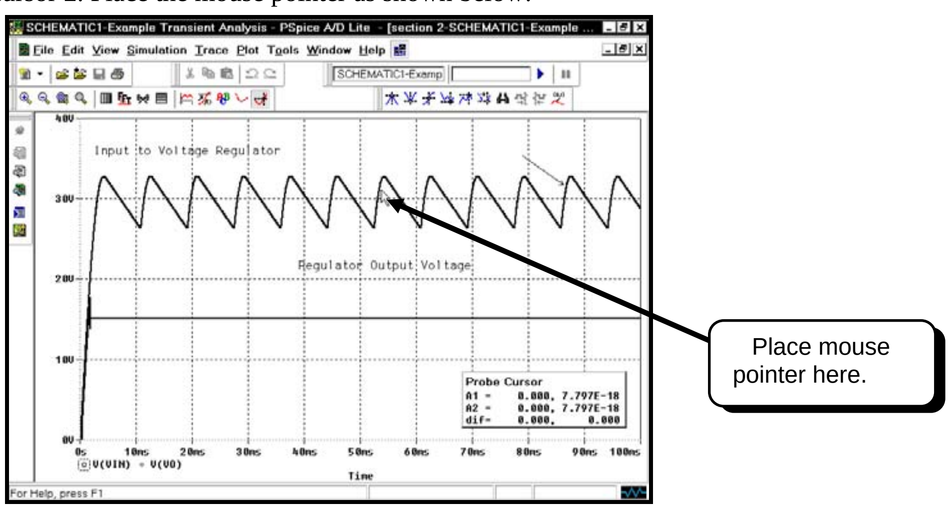


The cursors are positioned at the leftmost data point of the first trace so they cannot be easily seen. A new dialog box is displayed on the Probe screen. This dialog box displays the coordinates of each cursor and the difference between the two cursors. (There are two cursors.)

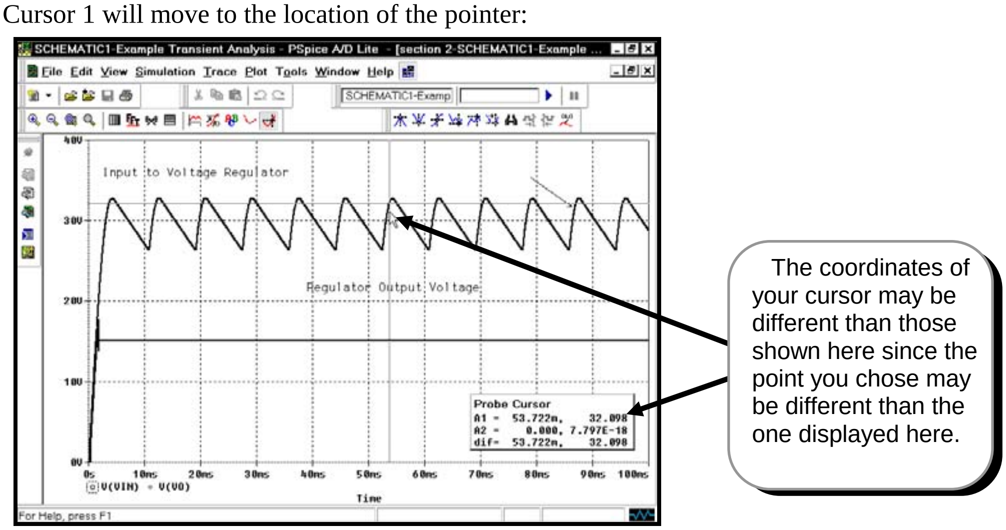


For this example and any other examples that use the cursors, the numerical values of the coordinates of your cursors may be slightly different than those shown in the examples since the points you choose may be different than those chosen in the examples.

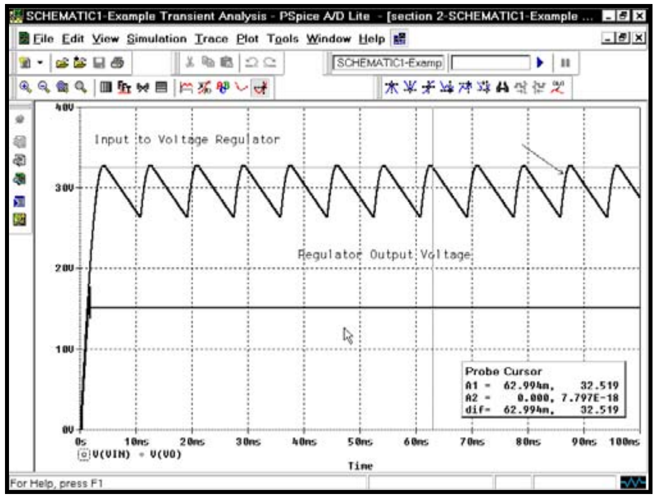
The cursors can be controlled using the mouse buttons or the keyboard. The left mouse button moves cursor 1 and the right mouse button moves cursor 2. Also, the left and right arrow keys **(**jl**)** move cursor 1, and the SHIFT key plus the left and right arrow keys **(**jl**)** move cursor 2. Place the mouse pointer as shown below:



Click the ***LEFT*** mouse button. Cursor 1 will move to the location of the pointer:

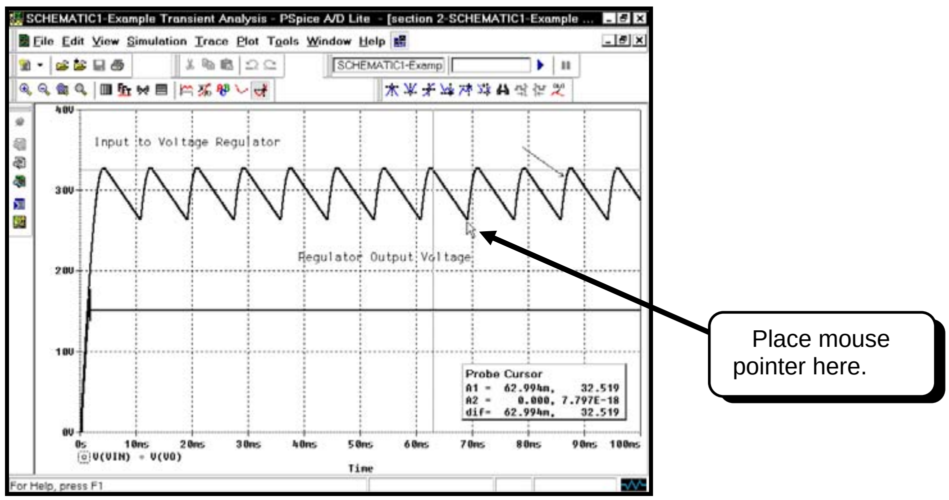


Next, press and **HOLD** the right arrow key (l). The cursor should move to the right:

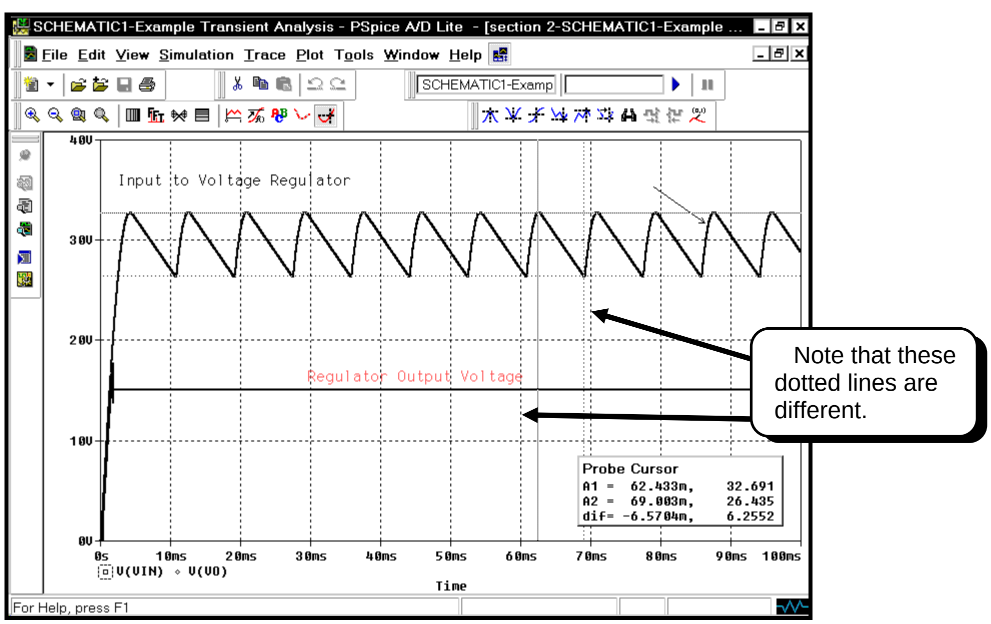


If you press the left arrow key (j), the cursor will move to the left. Note that as you move the cursor, the values in the Probe Cursor dialog box change.

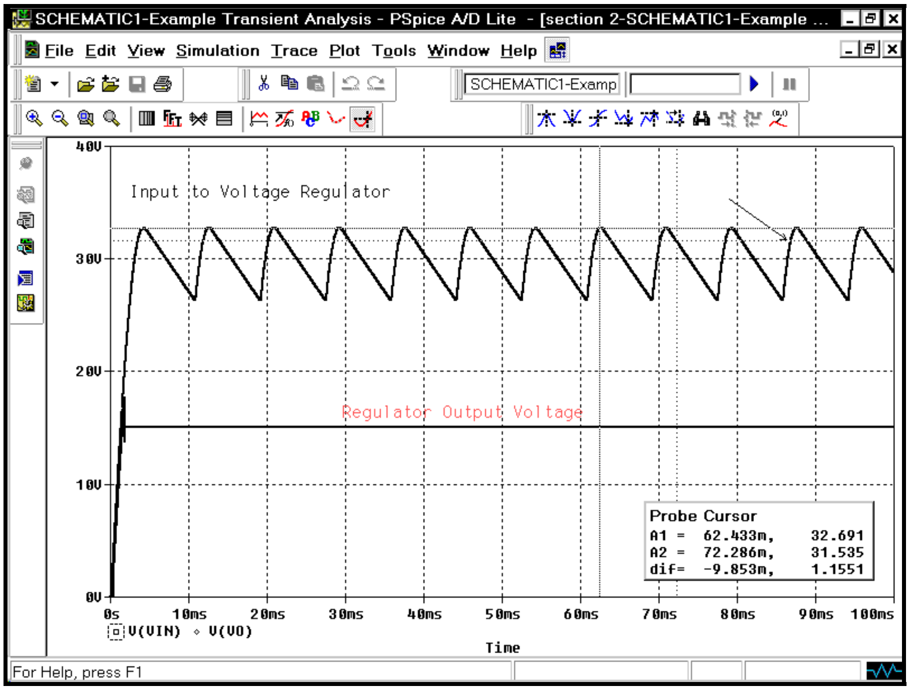
Next, we will move cursor 2. Place the mouse pointer as shown:



Click the ***RIGHT*** mouse button. Cursor 2 will move to the location of the pointer:

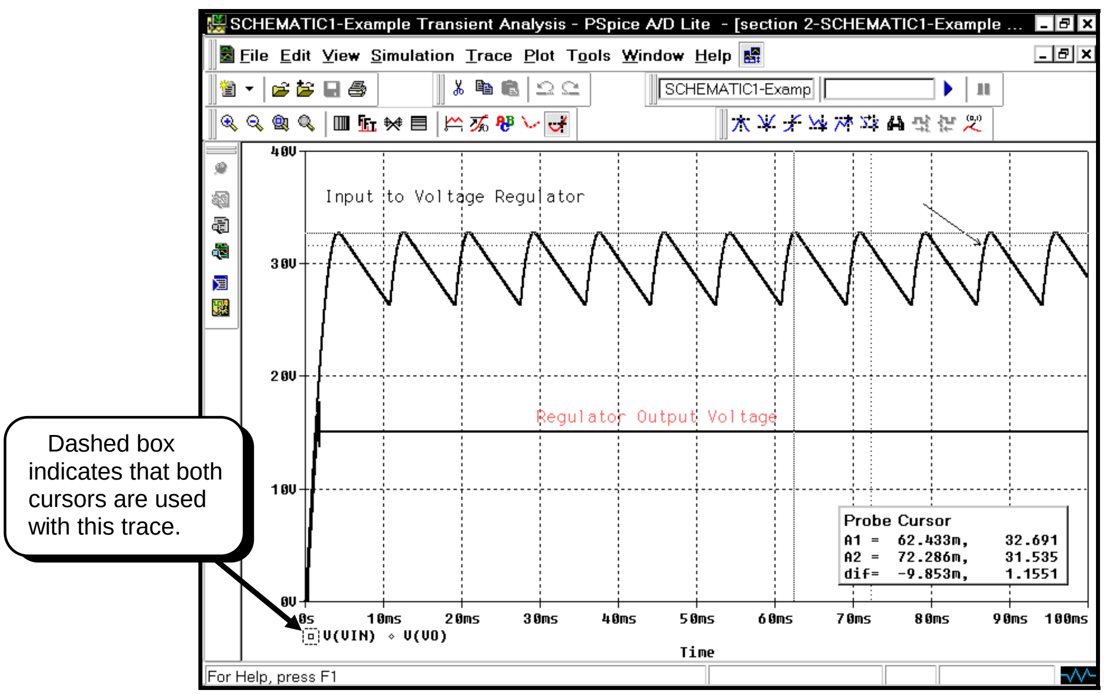


Notice that the dotted lines of the cursor are slightly different for the two cursors. Next, press and **HOLD** the SHIFT key and press and **HOLD** the right arrow key (SHIFT-l). Cursor 2 should move to the right:

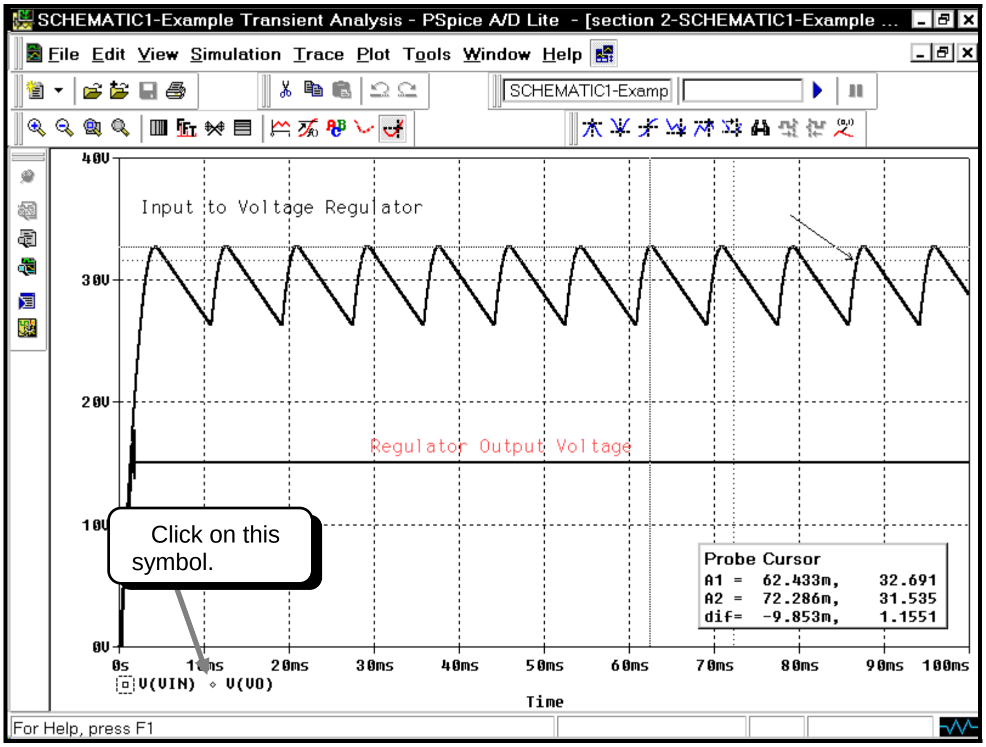


If you hold down the SHIFT key and press the left arrow key (SHIFT-j), cursor 2 will move to the left.

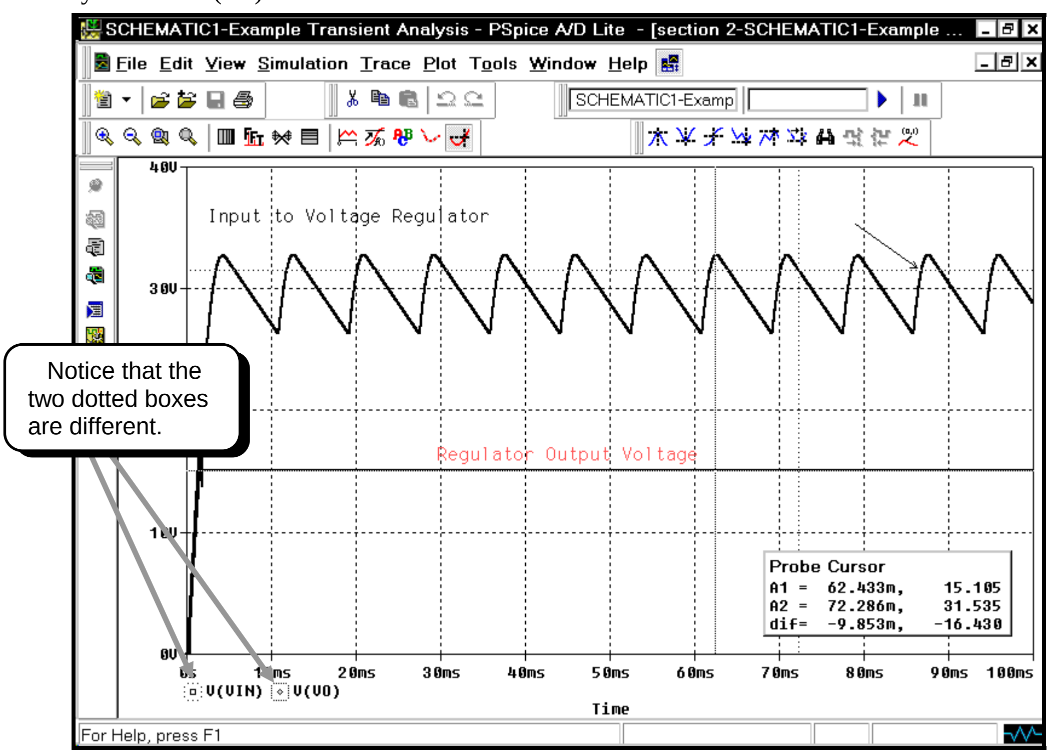
Presently, both cursors are displayed on the trace V(Vin). An indication of this is given by the dashed box around the symbol for V(Vin), as shown below:



To place cursor 1 on trace V(Vo), click the ***LEFT*** mouse button on the marker for trace V(Vo):

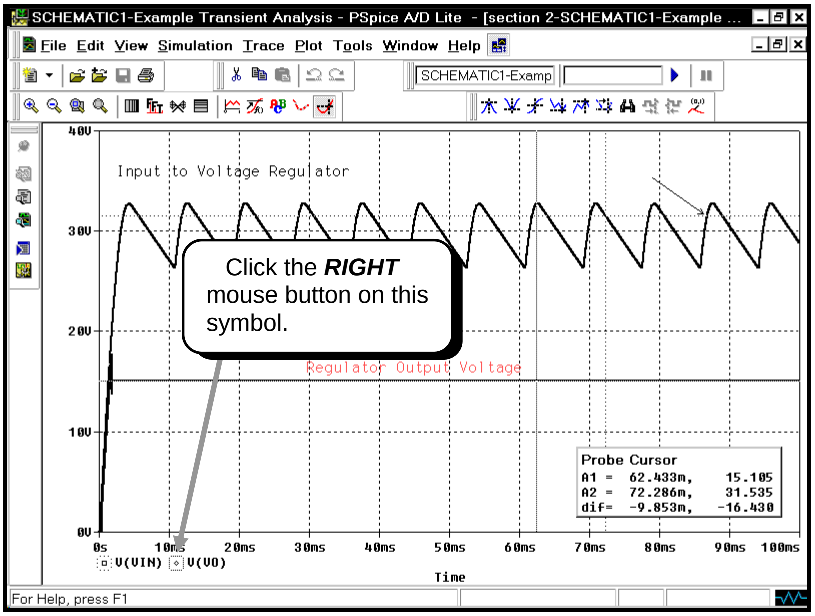


Cursor 1 will jump to trace V(Vo). Notice that a new dotted box encloses the symbol for trace V(Vin) and a different dotted box encloses the symbol for V(Vo).

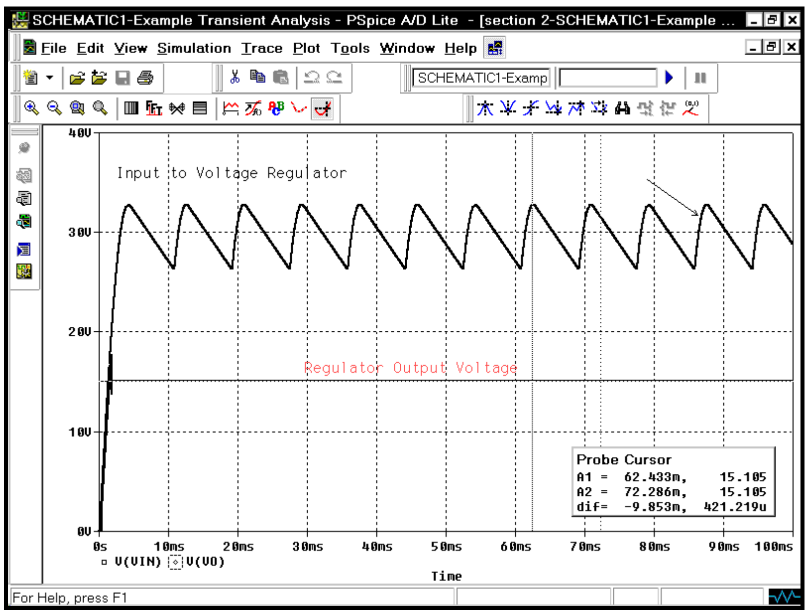


The dot patterns of the boxes match the dot patterns of the lines of two cursors. These dotted boxes indicate which cursor is attached to which trace.

To place cursor 2 on trace V(Vo), click the ***RIGHT*** mouse button on the marker for trace V(Vo):

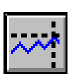


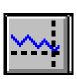
The second cursor will jump to the V(Vo) trace.

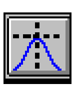


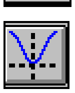
We can use the same method to move the cursors back to trace V(Vin).

The cursor that you used most recently (cursor 1 or cursor 2) is the active cursor. There are several tools in the button bar for positioning the active cursor. Some of the buttons are described below:

 - Positions the cursor to the absolute maximum of the trace.

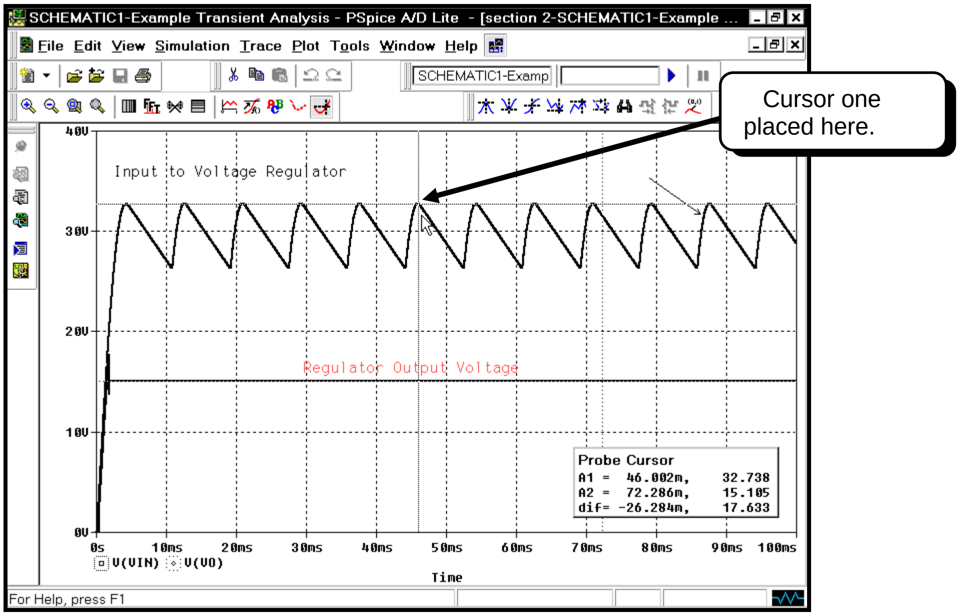
 - Positions the cursor to the absolute minimum of the trace.

 - Positions the cursor to the next local maximum.

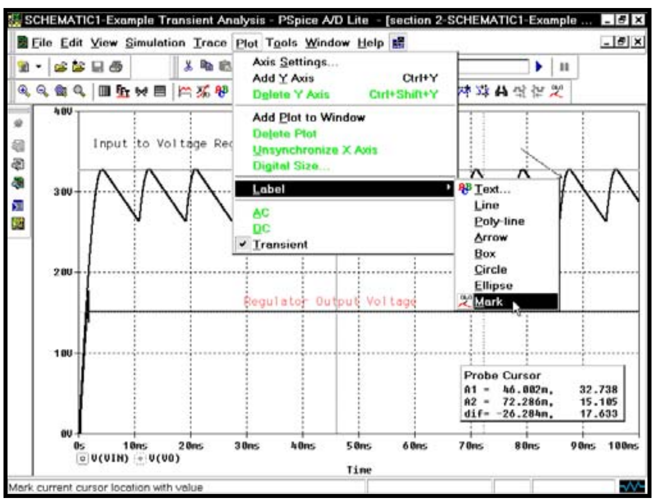
 - Positions the cursor to the next local minimum.

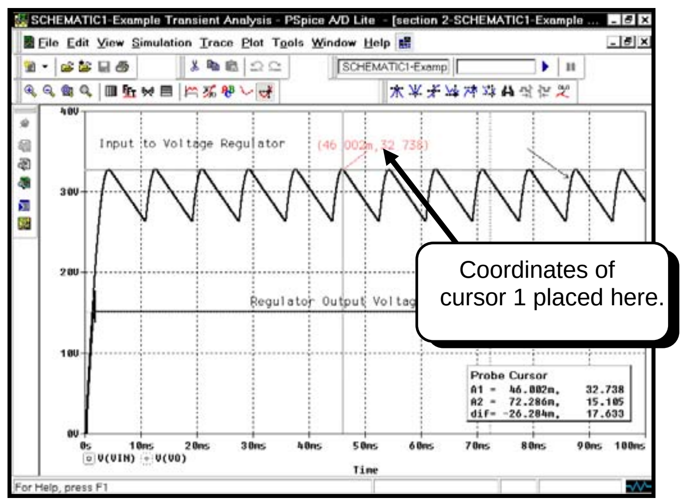
2.L. Labeling Points

The cursors are used to find numerical values of a trace. Once important points are found, the coordinates of those points can be placed on the plot. The cursor that you used most recently is the active cursor. Place cursor 1 on trace V(Vin) at the point shown:

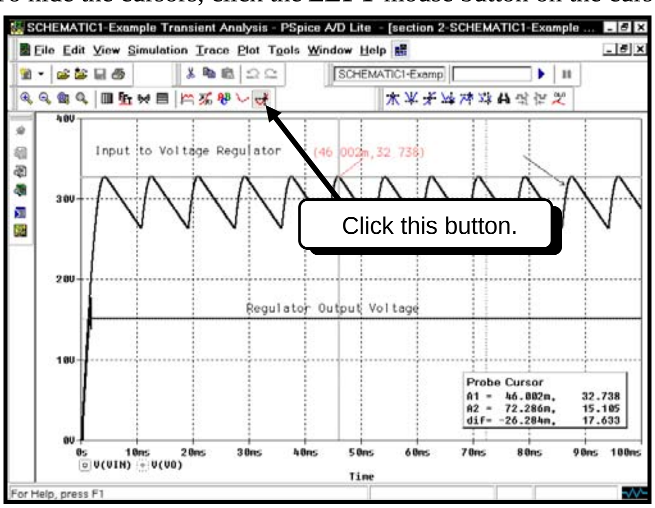


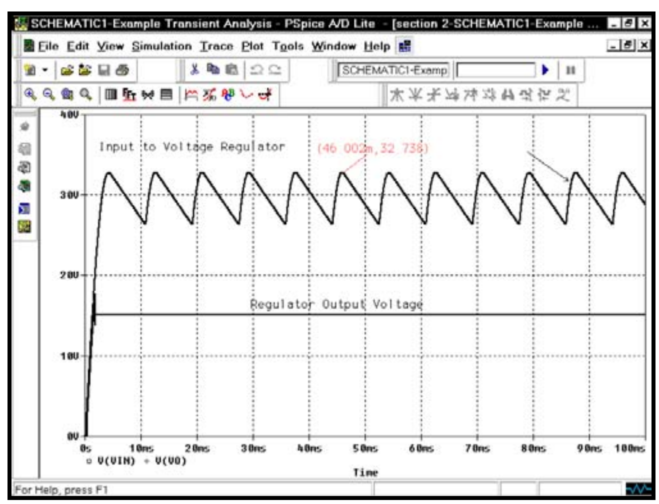
Since we just moved cursor 1, it is the active cursor. To label the coordinates of the active cursor, select **Plot**, **Label**, and then **Mark**. The coordinates will be displayed on the screen:





To hide the cursors, click the ***LEFT*** mouse button on the cursor button  :





We can now move any of the items on the screen. The steps for moving any item are the same as shown in Section 2.J.

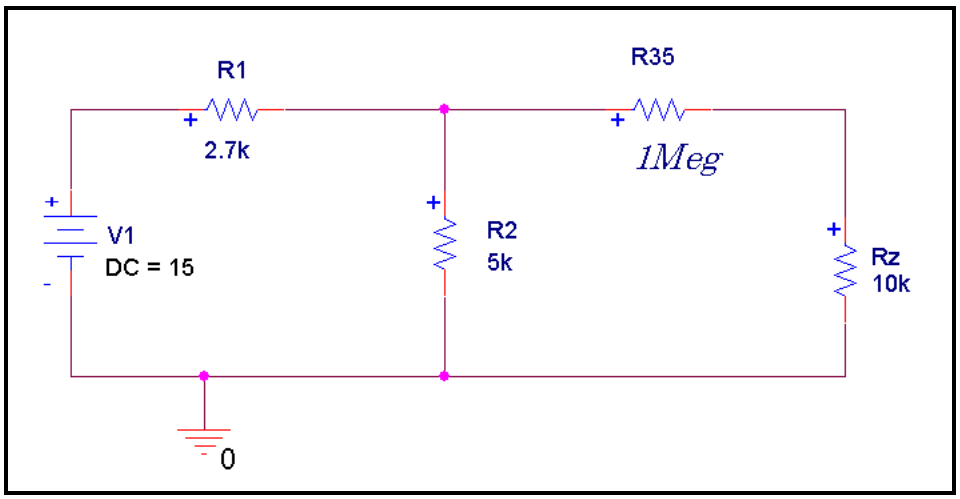
2.Q. Printing Multiple Probe Plots on a Printer Page

PART 3
DC Nodal Analysis

The node voltage analysis performed by PSpice is for DC node voltages only. This analysis solves for the DC voltage at each node of the circuit. If any AC or transient sources are present in the circuit, those sources are set to zero. Only sources with an attribute of the form **DC=*value*** are used in the analysis. If you wish to find AC node voltages, you will need to run the AC Sweep described in Part 5. The node voltage analysis assumes that all capacitors are open circuits and that all inductors are short circuits.

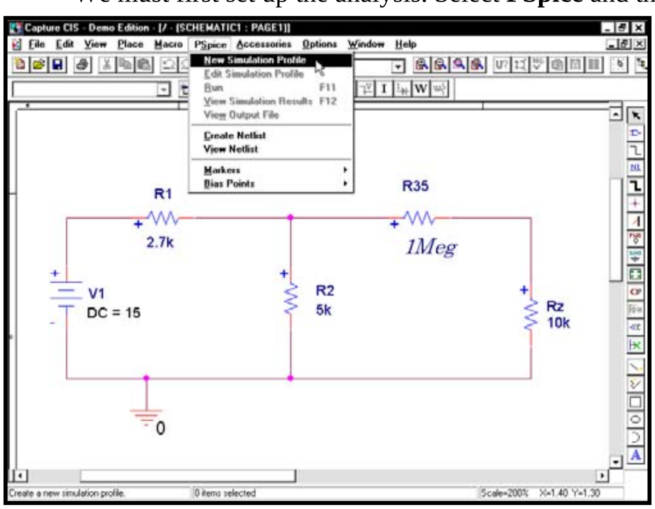
3.A. Resistive Circuit Nodal Analysis

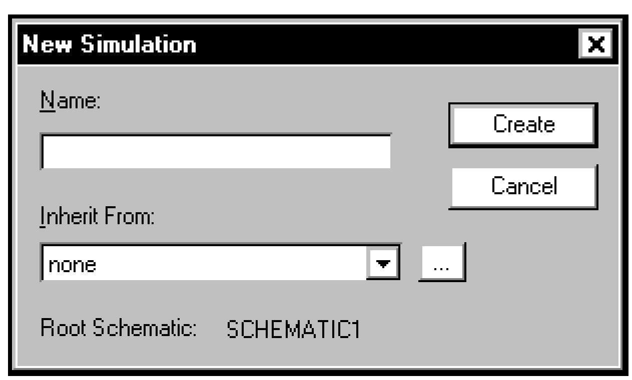
We will perform the nodal analysis on the circuit wired in Part 1 and shown on page 36. The circuit is repeated below:



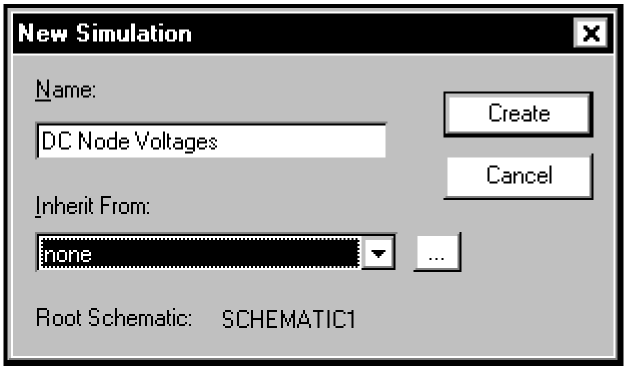
We will show several different ways of finding the node voltages and branch currents. First we will use the bias display markers provided by Capture.

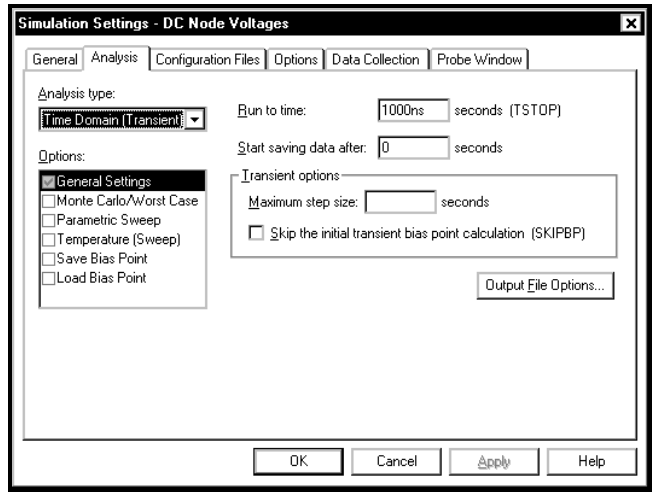
We must first set up the analysis. Select **PSpice** and then **New Simulation Profile** from the menus:



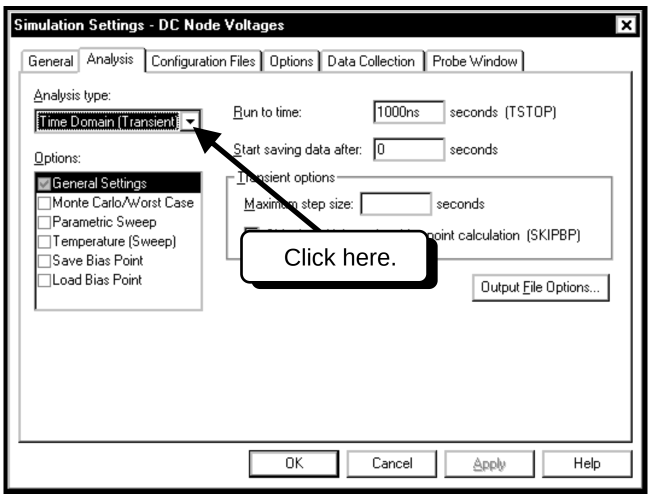


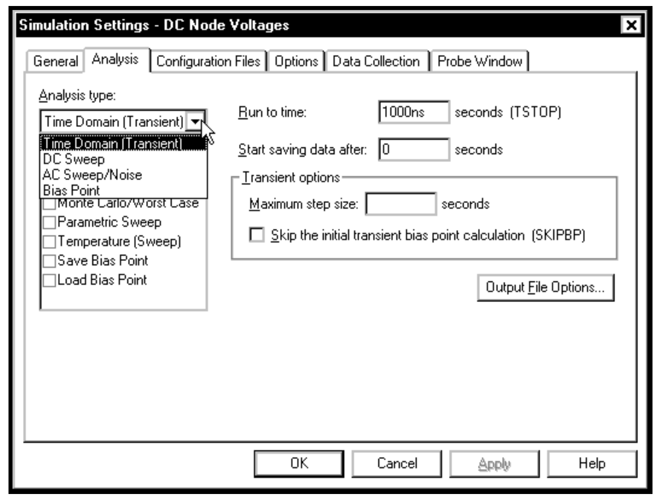
Enter a name for the simulation. I will type **`DC Node Voltages`** and click the **Create** button:



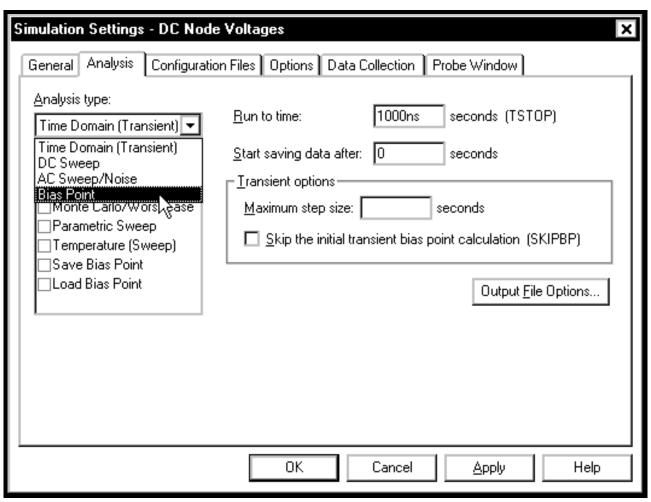


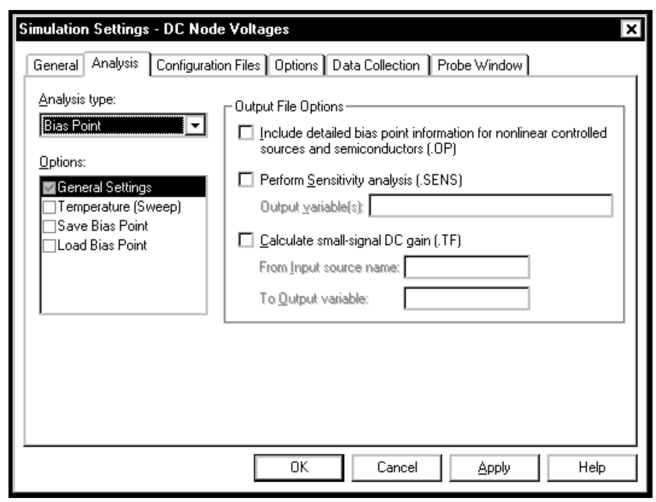
The bias point analysis is automatically run each time PSpice runs so we do not have to specify any settings. However, we will specify the simulation settings to only run a bias point calculation. Click the ***LEFT*** mouse button on the down triangle ▼ as shown to see the list of available simulations:





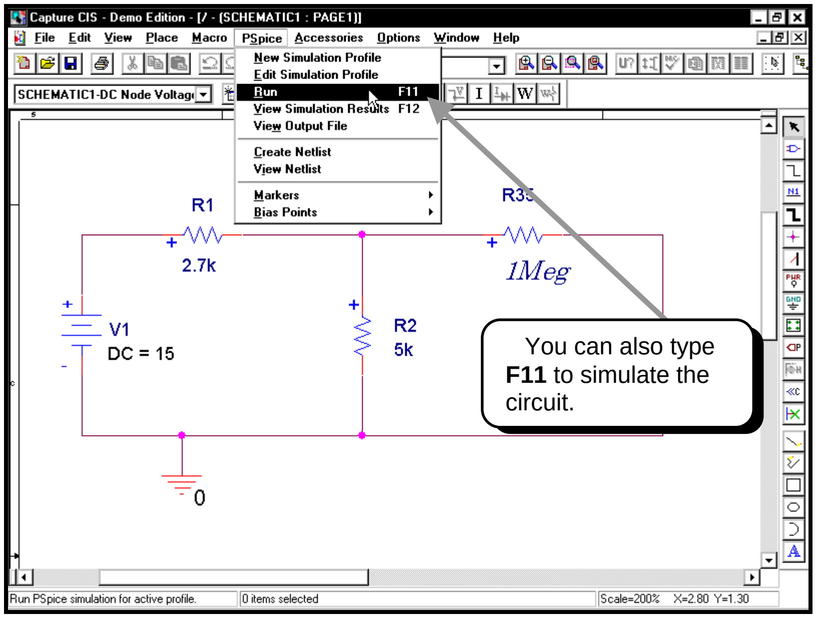
Select the **Bias Point** selection:



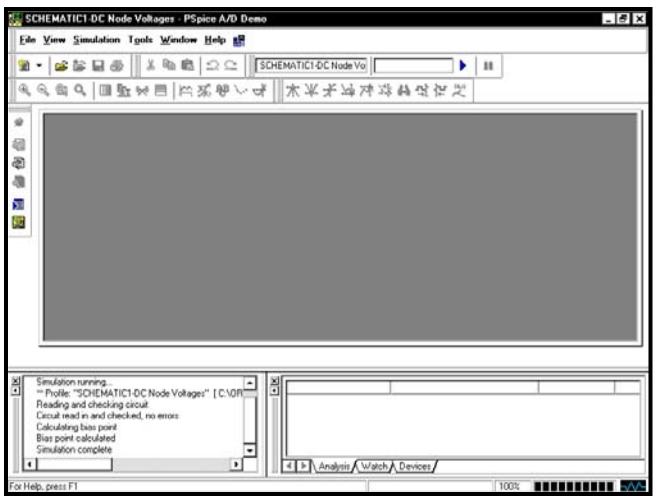


This selection specifies that only the Bias Point simulation will run. We do not need to specify any output file options because we will be displaying the results on the schematic. Click the **OK** button to save the simulation profile and return to the circuit.

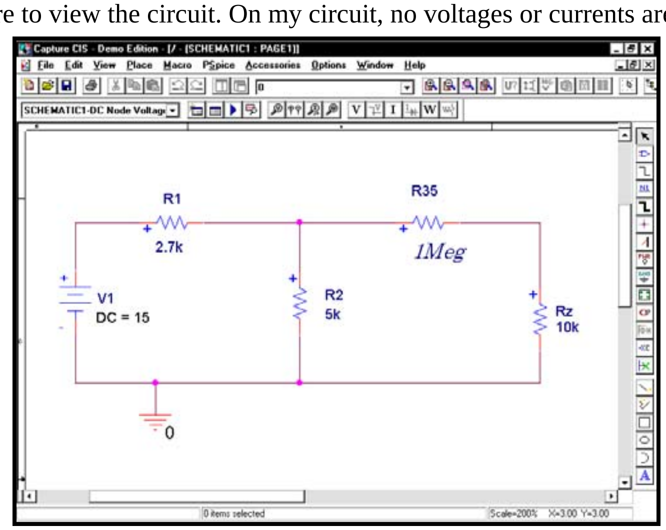
We will run the analysis to calculate all node voltages and all branch currents. Select **PSpice** and then **Run** from the Capture menus:



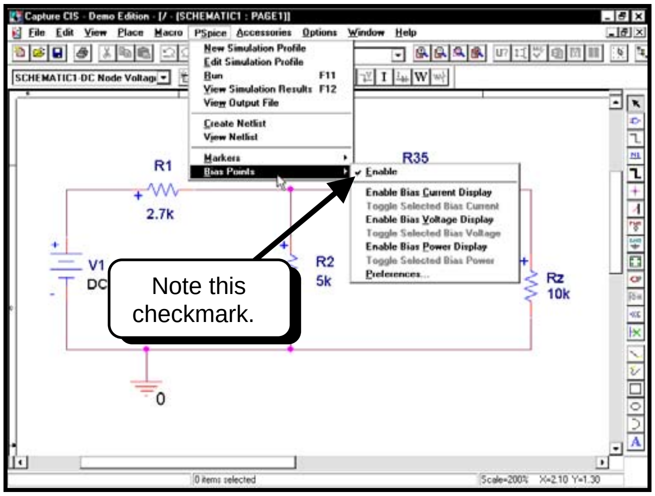
PSpice will run and the PSpice A/D Lite window will be displayed:



When the message window says **Bias point calculated**, the simulation is complete. We can now display the voltages and currents. Switch back to Capture to view the circuit. On my circuit, no voltages or currents are displayed:

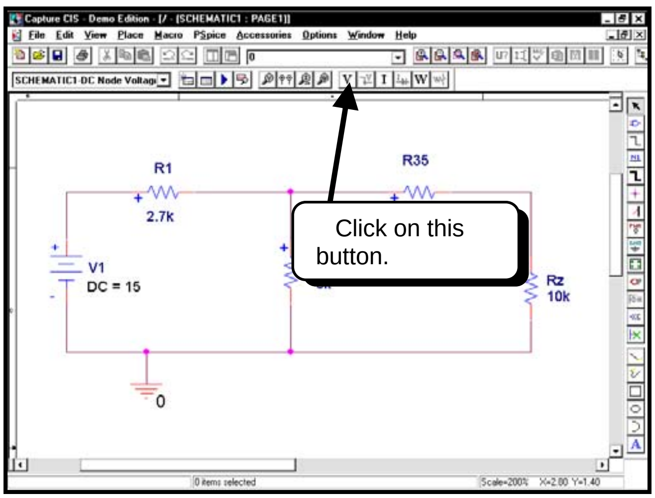


Before continuing, we will make sure that we enable the display of voltages, currents, and power on the schematic. Select **PSpice** and then **Bias Points**:

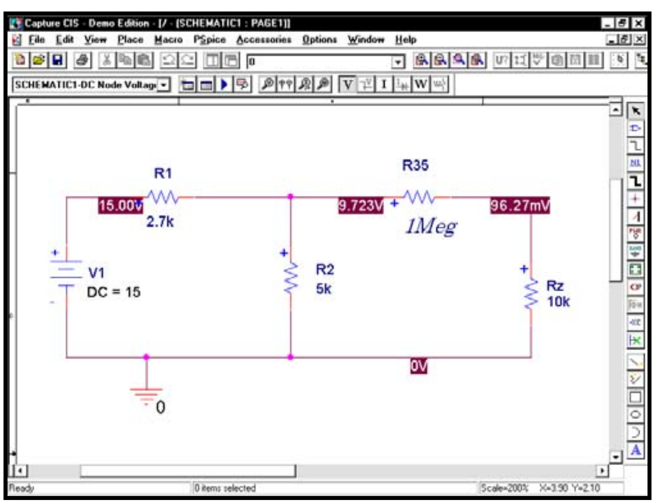


Note that there is a checkmark next to the text **Enable**. If your menu selection does not have a checkmark, as shown above, you will not be able to display voltages and currents on your schematic. **If you do not have a checkmark displayed next to menu selection Enable,** select **Enable** from the menu to toggle the checkmark on. If a checkmark is displayed, click the ***LEFT*** mouse button on the schematic somewhere to hide the menus.

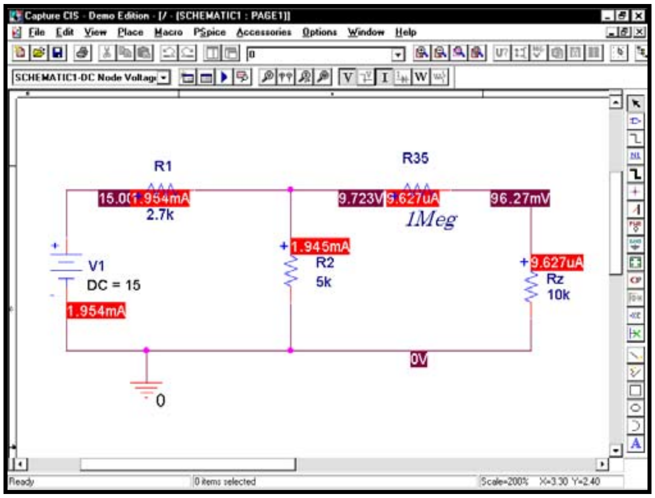
We will first display the node voltages. To toggle the voltages on or off, click the ***LEFT*** mouse button on the V button  in the Capture button bar:

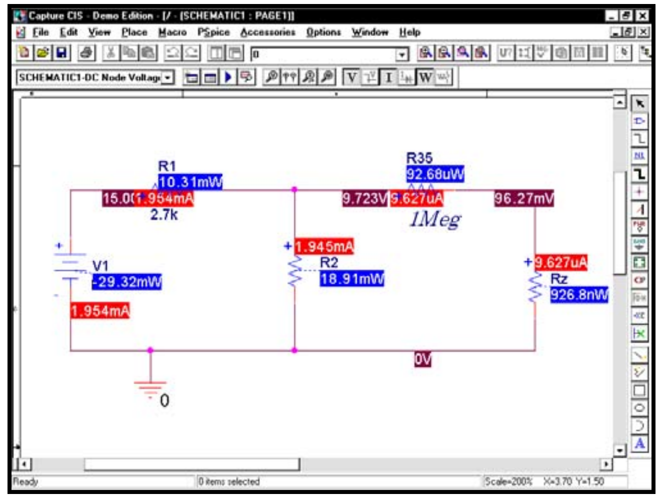


Clicking on the  button will toggle the display of node voltages on or off. Click on the  button until the node voltages are displayed:

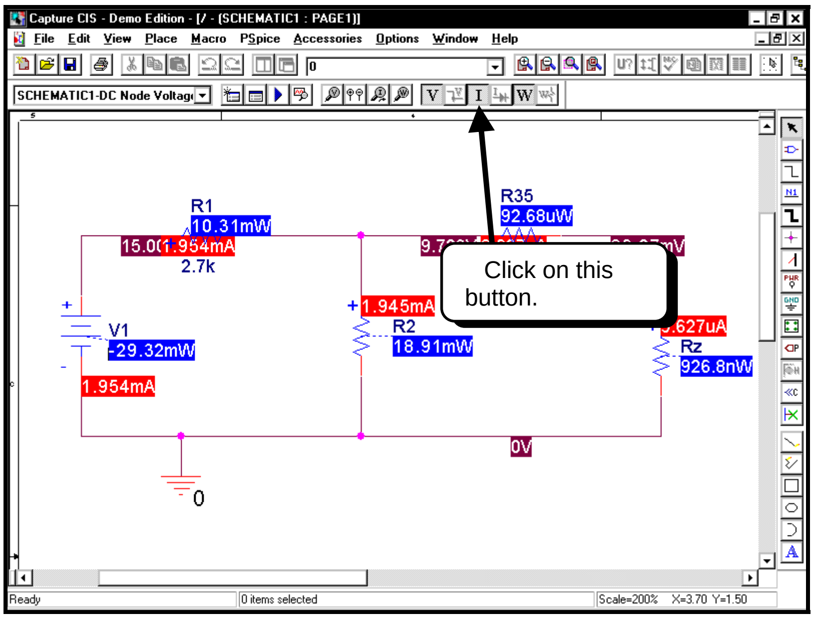


The screen capture above displays the voltages for every node. If your screen looks like one of the following:

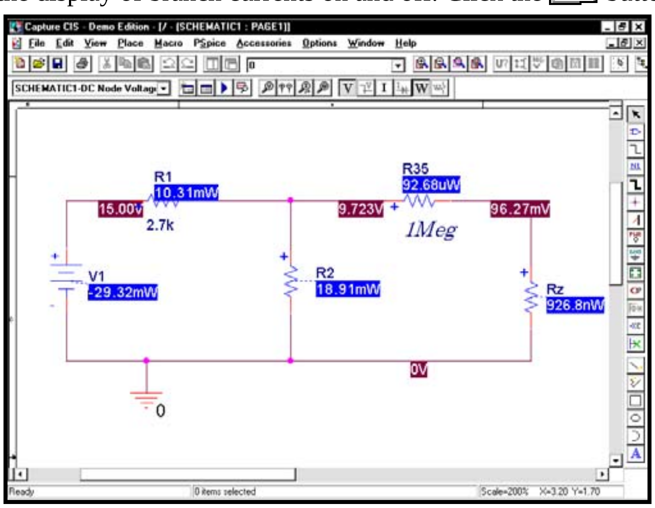




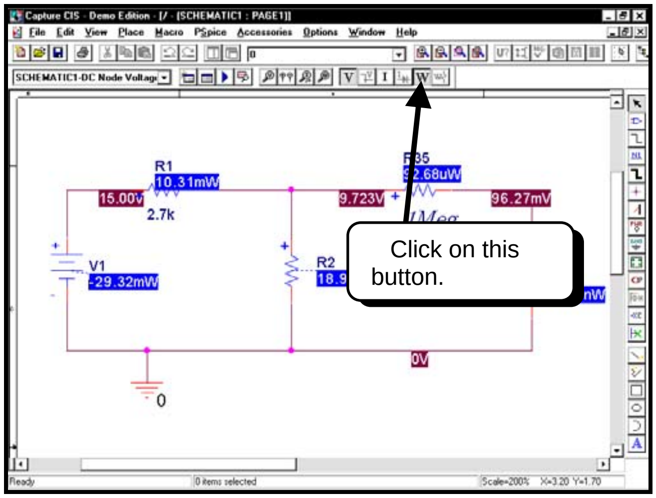
then you have node voltages, plus branch currents and/or power data displayed. To hide the branch currents, click the ***LEFT*** mouse button on the I button I as shown below:



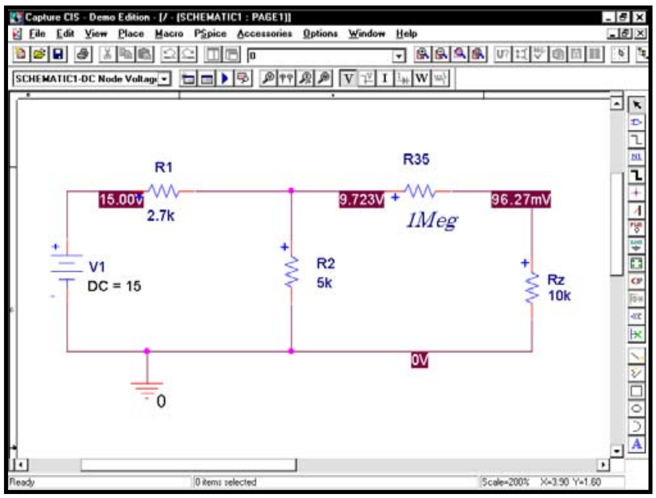
Clicking this button will toggle the display of branch currents on and off. Click the I button to hide the branch currents.



To hide the power data, click the ***LEFT*** mouse button on the W button W as shown below:

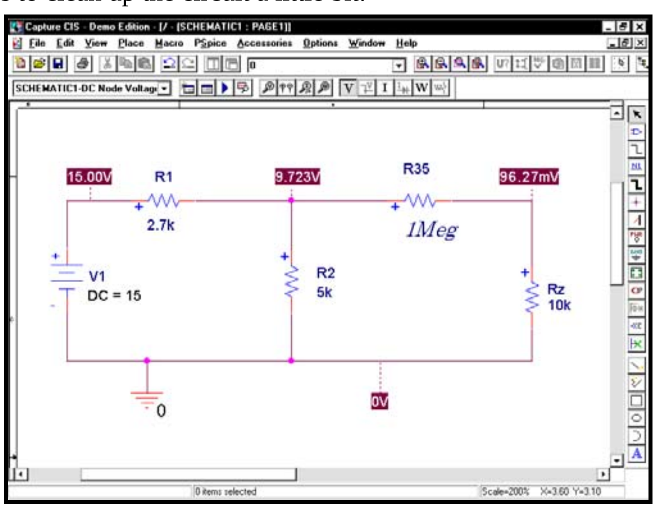


Clicking this button will toggle the display of power data on and off. Click the  button to hide the power data.



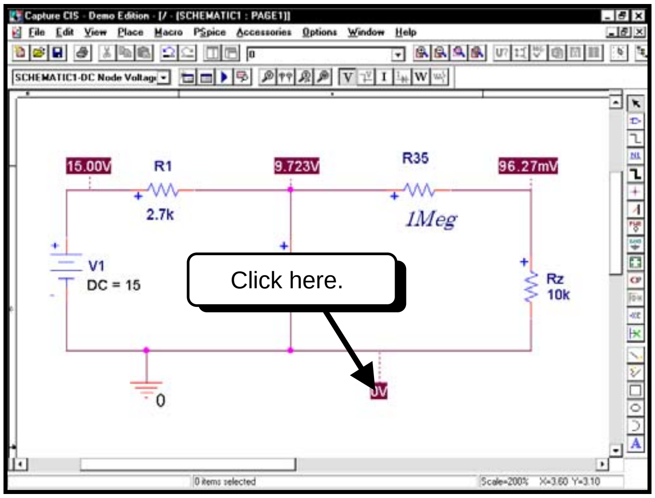
We now only have the node voltages displayed, so we can clean up the schematic a bit.

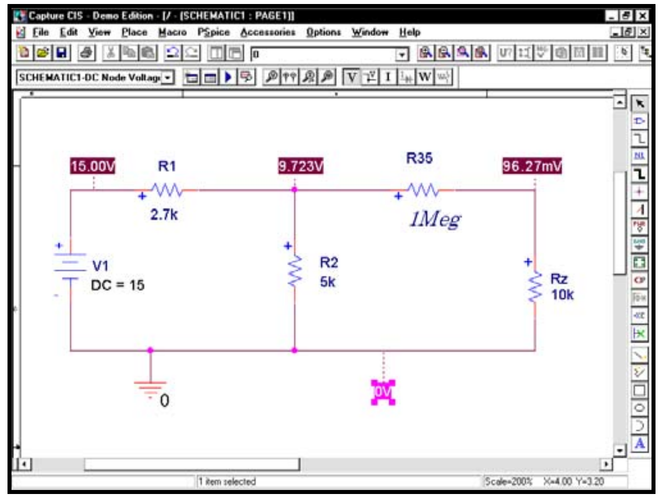
The node voltages are a little bit too close to some of the components. The numbers can be moved like any graphic on the screen. Move the numbers to clean up the circuit a little bit:

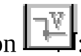


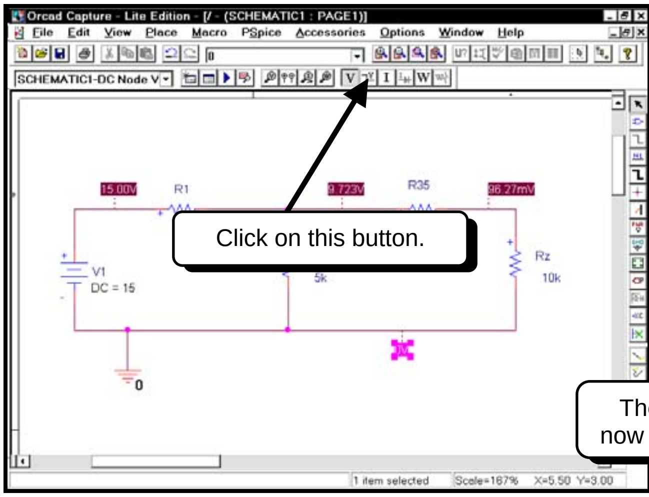
We see that dashed lines are drawn from the data to the node to which they refer. This makes it easy in complicated circuits to figure out which number belongs to which node.

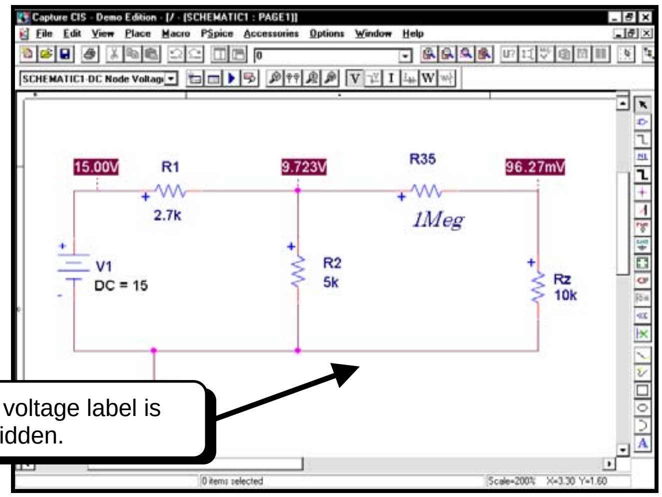
Some of the node voltages displayed are unnecessary and we would like to hide them in the schematic. For example, there is a node voltage number telling us that ground is at zero volts. This is unnecessary and we will hide it. First, click the ***LEFT*** mouse button on the text **0V** as shown. It will become selected and displayed in pink:



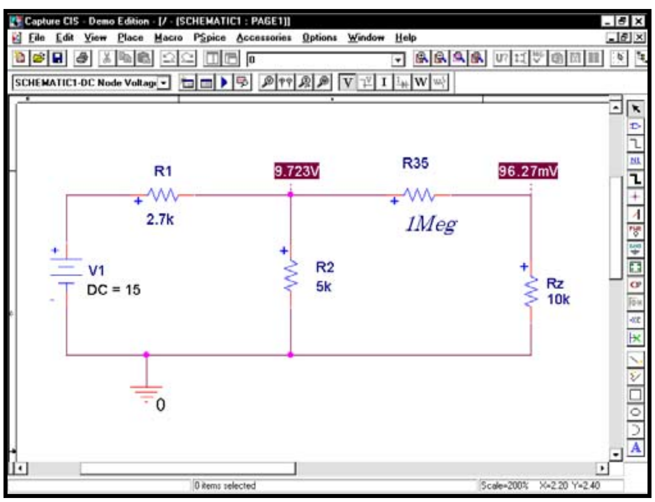


To hide the selected marker, click on the Toggle Voltages On Selected Net button :

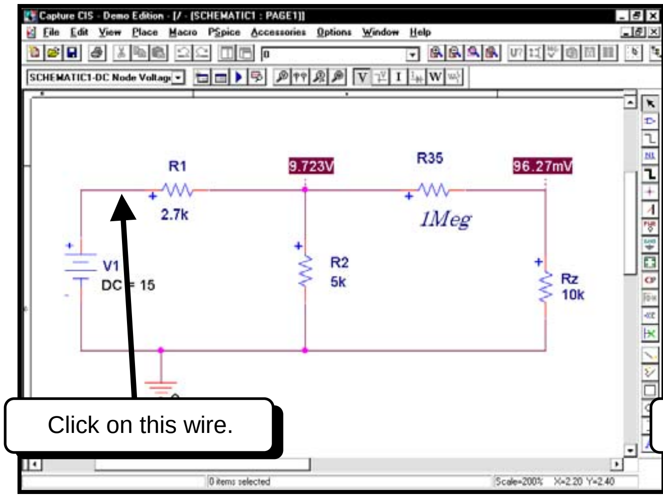


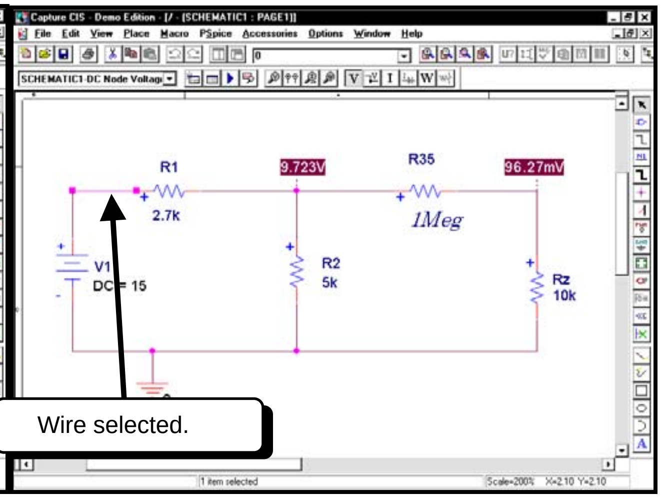


Use this technique to hide the **15.00V** number:

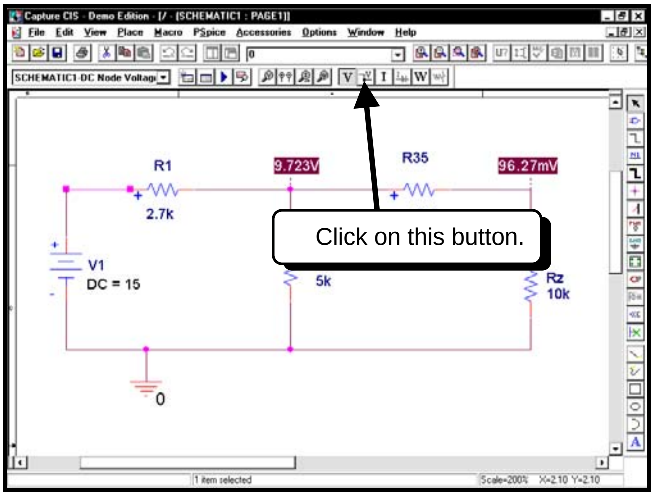


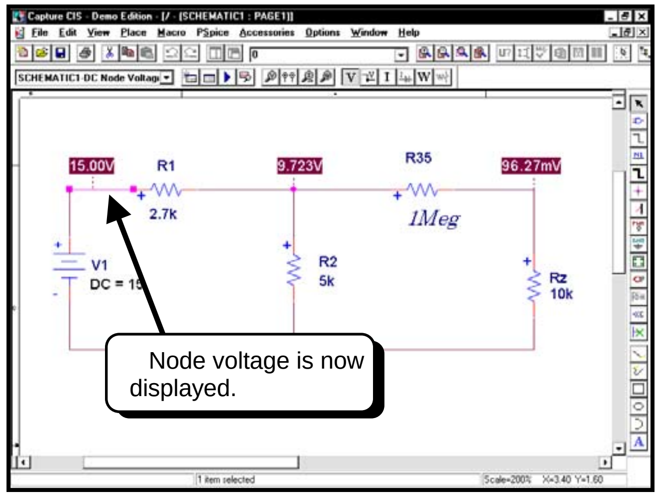
Suppose that we have hidden a node voltage and we would now like to display it. To do this, click the ***LEFT*** mouse button on the wire of which you would like to display the node voltage. The wire will turn red, indicating that it has been selected. I would like to display the voltage of the node between V1 and R1 (the one I removed previously) so I will select the wire shown below:



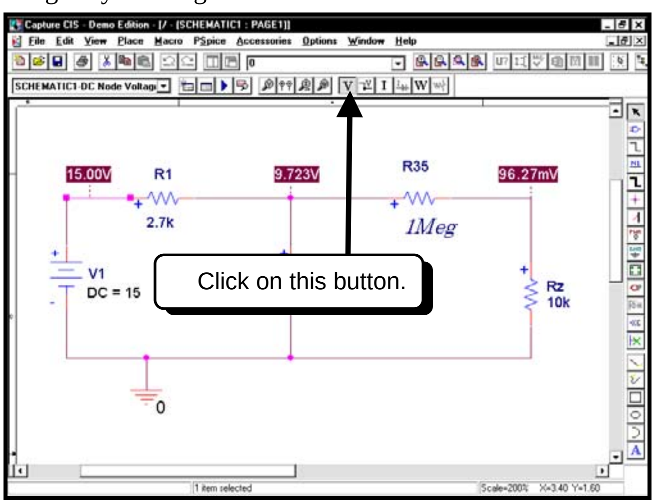


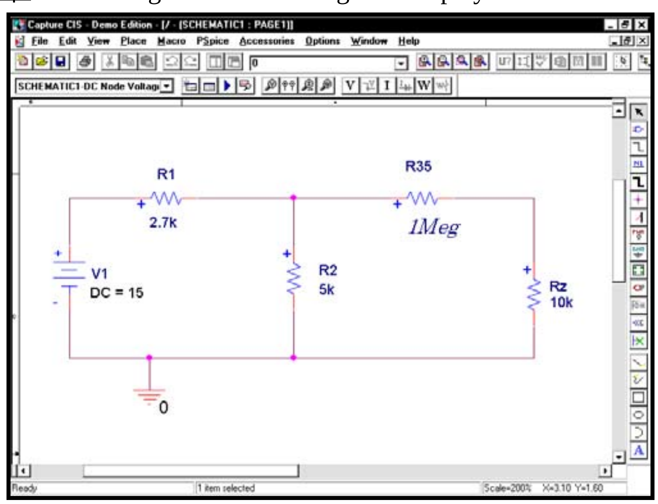
When the wire is highlighted in pink, click the ***LEFT*** mouse button on the Toggle Voltages On Selected Net button . The voltage for the node will be displayed:



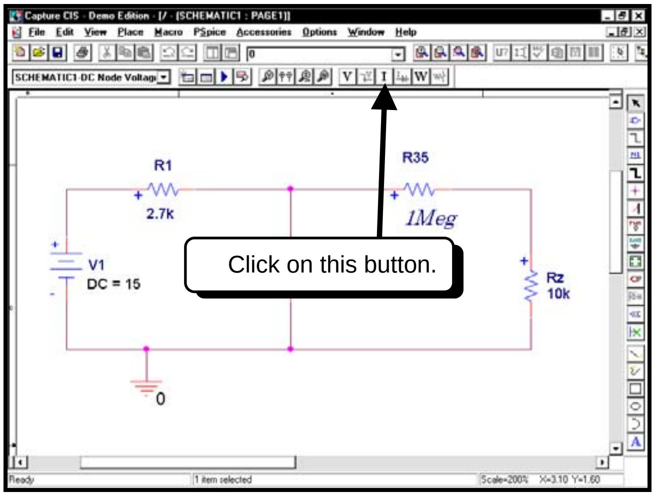


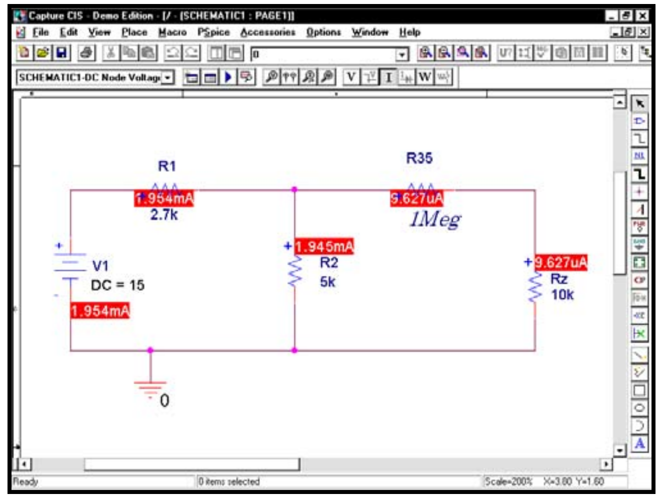
Now that we know all of the node voltages, we would like to display the branch currents. First, hide the node voltages by clicking the ***LEFT*** mouse button on the V button **V**. The voltages should no longer be displayed:



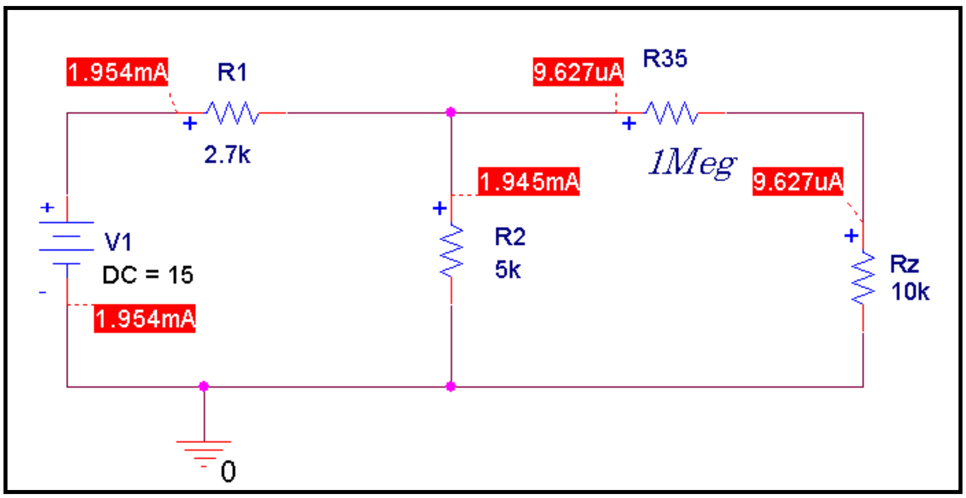


To display the branch currents, click the ***LEFT*** mouse button on the I button **I**. The branch currents will be displayed:

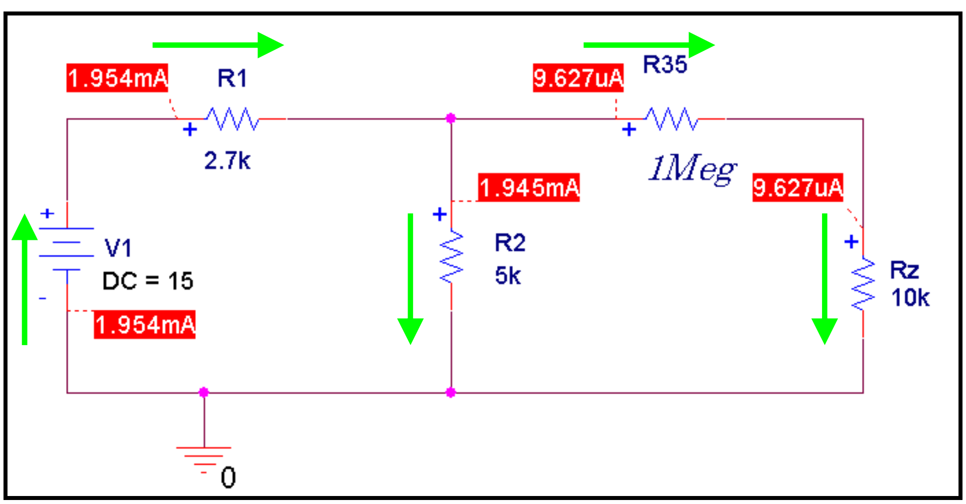




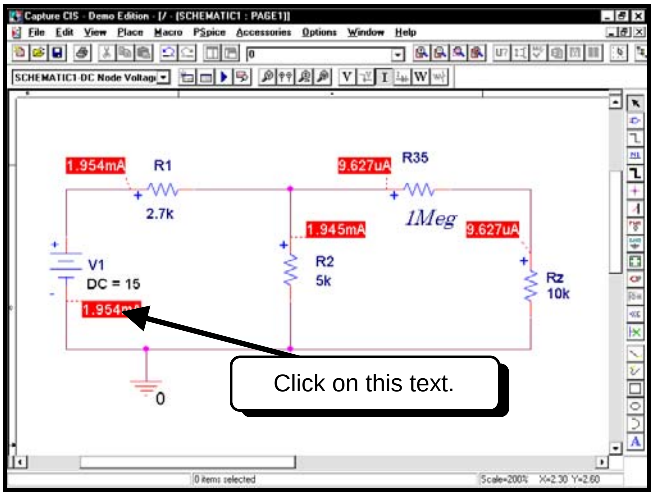
We can move the displayed numbers in the same manner as we did with the node voltages. Move the numbers so that the schematic is a little clearer:

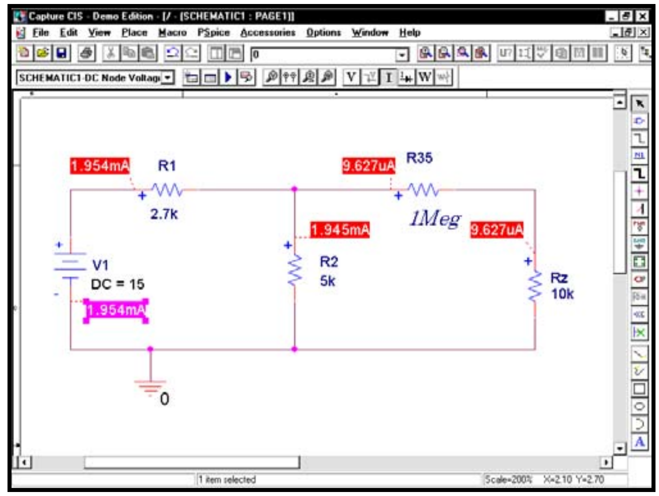


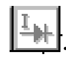
With currents we need to know which number refers to which circuit element and which direction the current is flowing. Dashed lines are shown connecting the numbers to a terminal of an element. The numbers shown are for positive currents entering the indicated terminals. For example, there is 1.954 mA entering the negative terminal of V1. This indicates that this voltage source is supplying power to the circuit. As another example, there is 1.954 mA entering the positive terminal of R1, and 9.627 μA entering the positive terminal of R35. In the drawing below, I have manually added arrows to the drawing to show the direction of the currents.

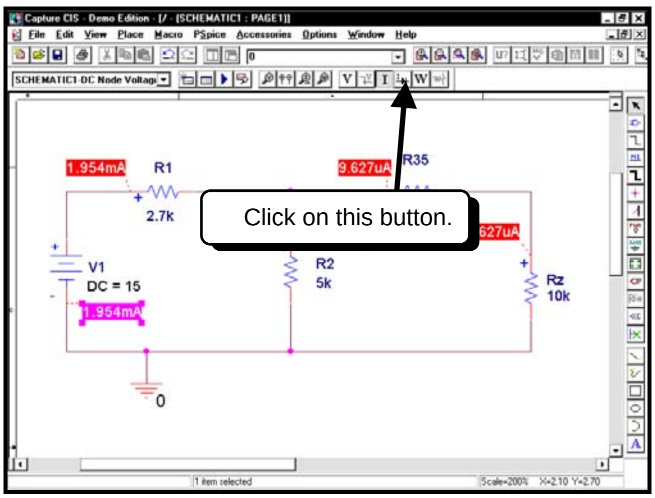


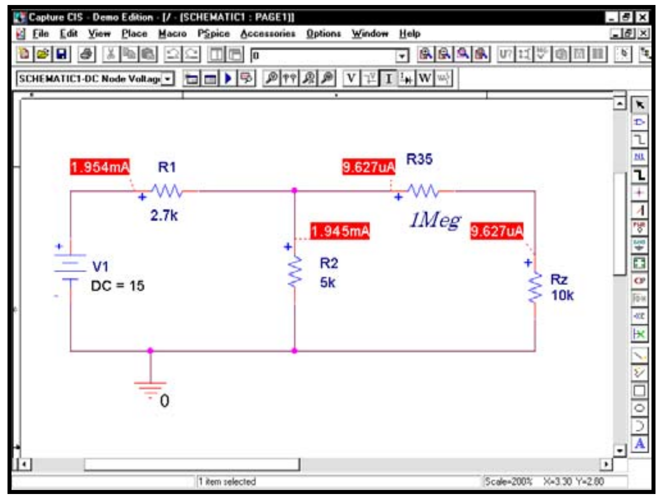
Branch currents can be hidden using a procedure similar to what we used to hide node voltages. Click the ***LEFT*** mouse button on the text **1.954mA** at the bottom of the screen. It will become highlighted in pink:



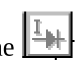


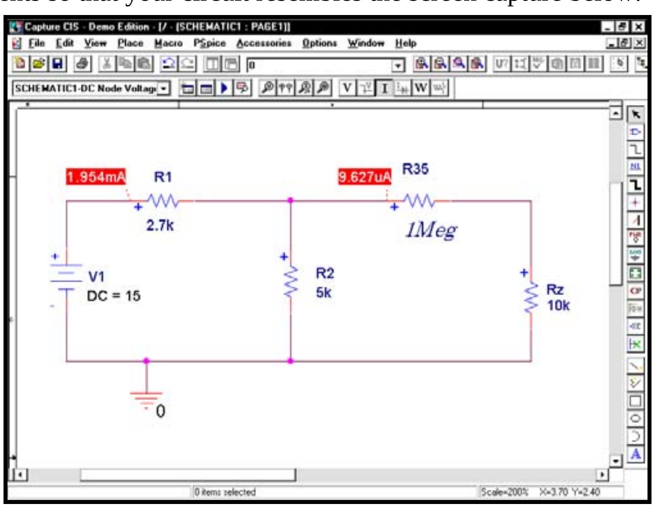
Click the ***LEFT*** mouse button on the Toggle Current On Selected Parts/Pins button . The selected current will disappear:



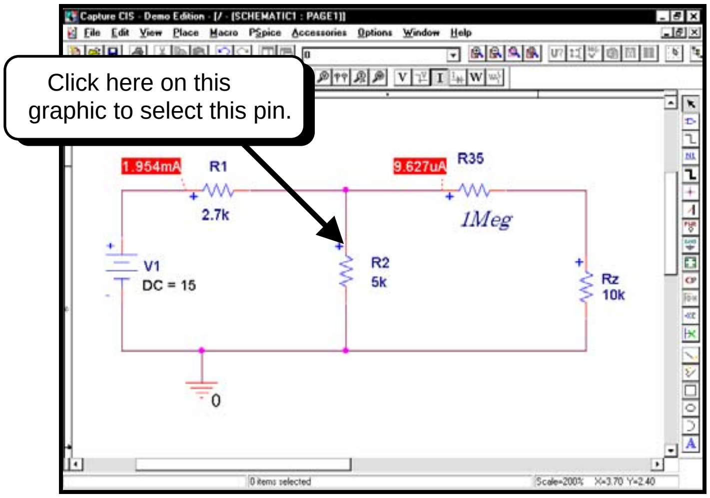


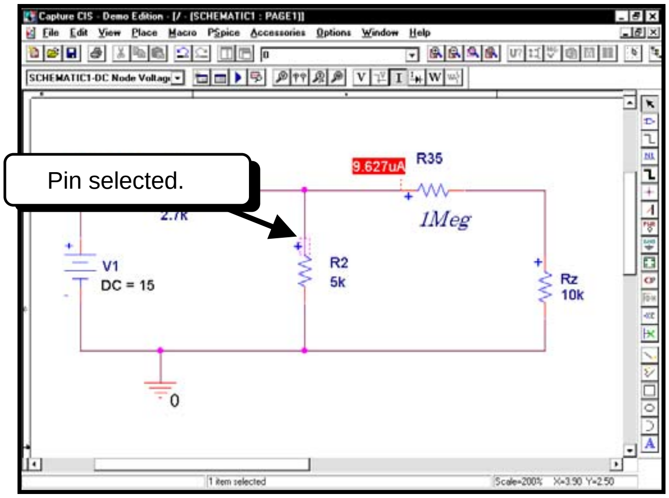
We can hide as many currents as we wish.

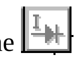
Suppose that we have hidden a number of the branch currents, and we now wish to display one of the hidden currents. Displaying hidden branch currents is a little different than displaying hidden node voltages. Node voltages are associated with wires, so, to display a node voltage, we selected a wire and then clicked the  button. Branch currents are associated with circuit elements. To display a current into an element, select the element and then click the  button. As an exercise, remove branch currents so that your circuit resembles the screen capture below:

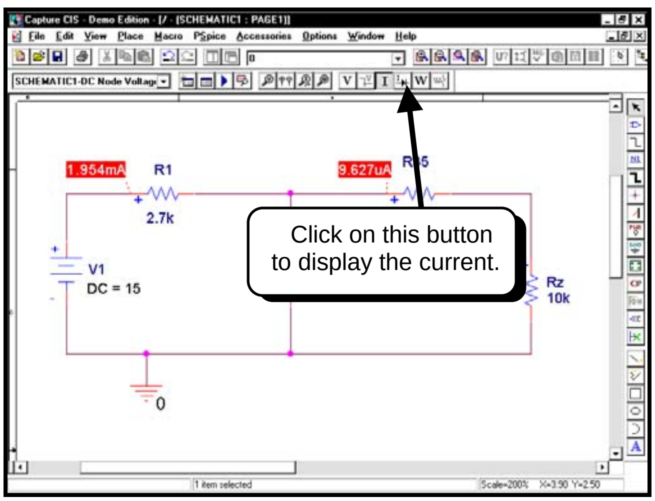


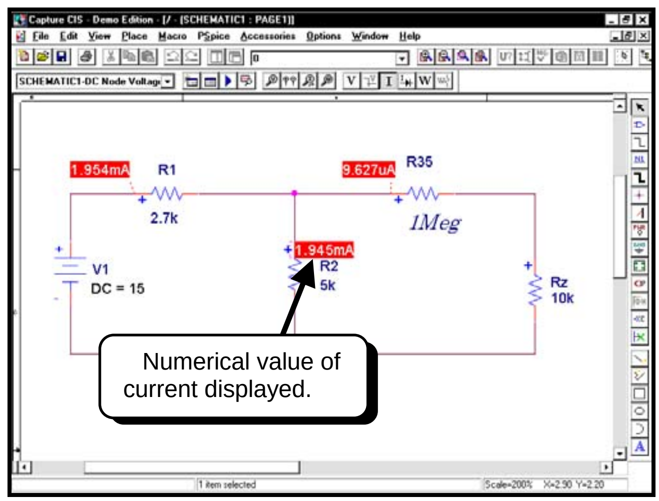
The currents through R2 and through Rz are not shown. We will display the current through R2. Click the ***LEFT*** mouse button on the positive pin of the graphic for R2 . It will become highlighted in pink, indicating that it has been selected.



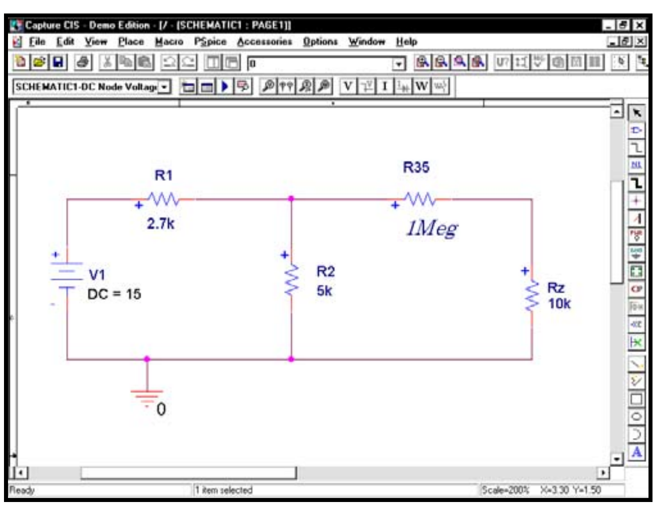


Once the pin is selected, we can display the current into the selected pin. Click the ***LEFT*** mouse button on the  button to display the current:

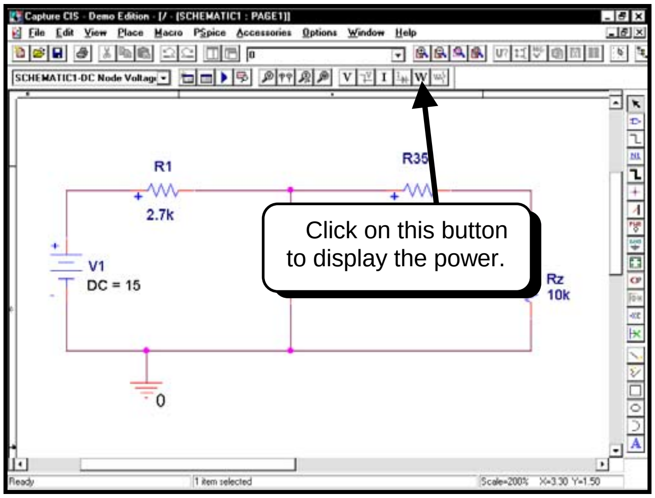


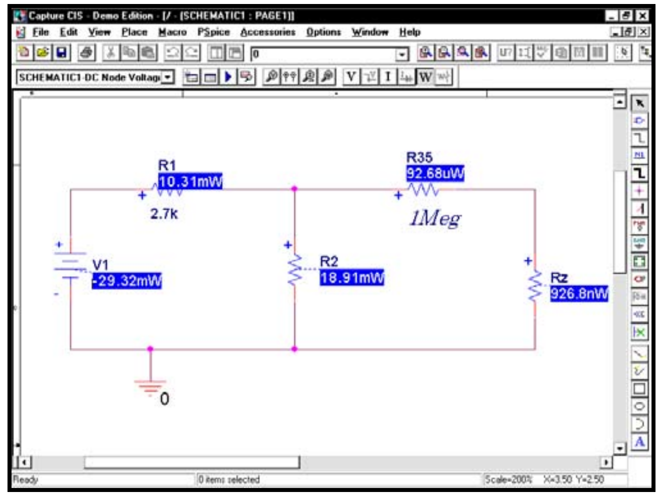


For the next example, we would like to hide all of the voltages and currents. Use the  and  buttons to hide all node voltages and branch currents:

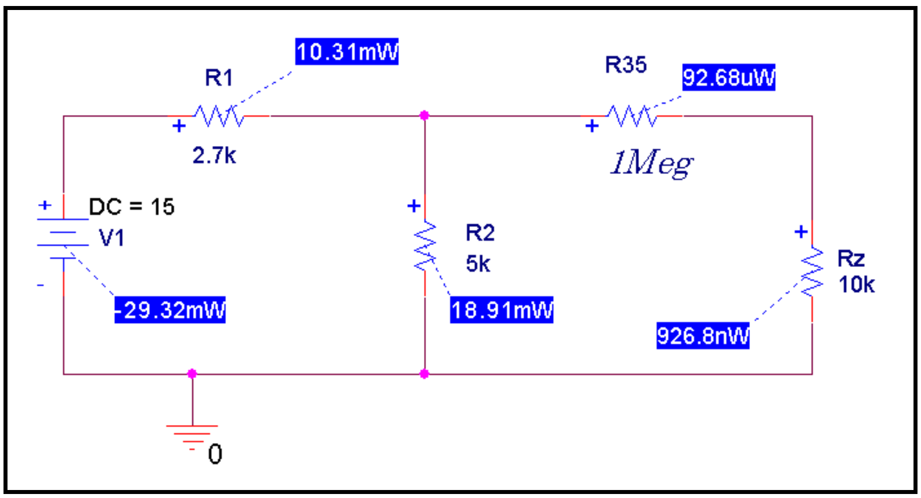


We would now like to display the power absorbed by all elements in the circuit. To display the power, click the ***LEFT*** mouse button on the **W** button:



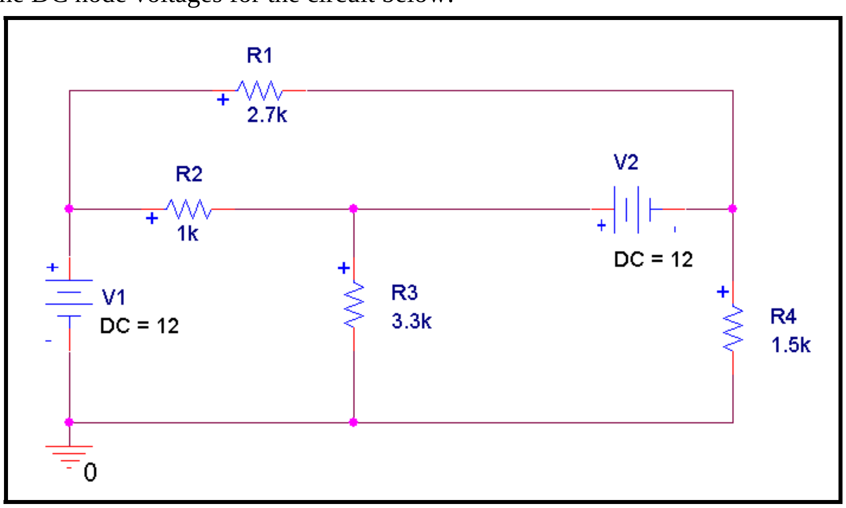


The numbers display the power absorbed by every element in the circuit. Move the numbers around to make the circuit more readable:

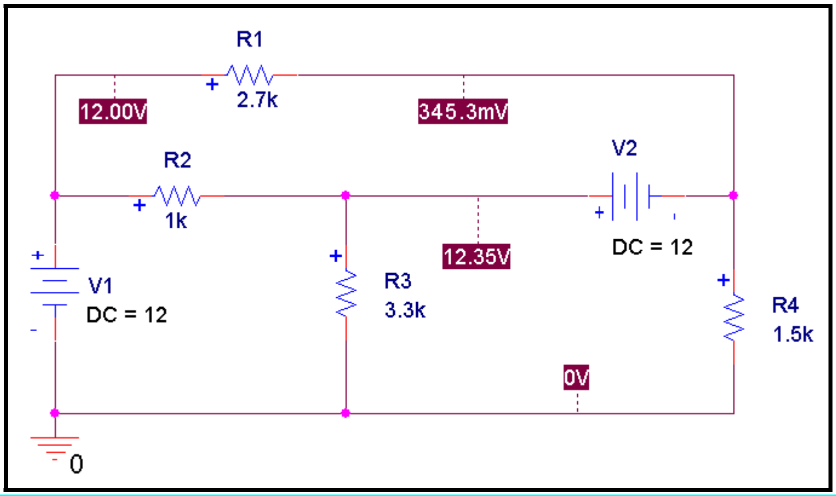


Notice that dashed lines associate a number with a circuit element. Note that the power numbers displayed for resistors are all positive. This means that those elements are absorbing that amount of power. The number displayed for the voltage source is negative, indicating that the element is sourcing power to the circuit.

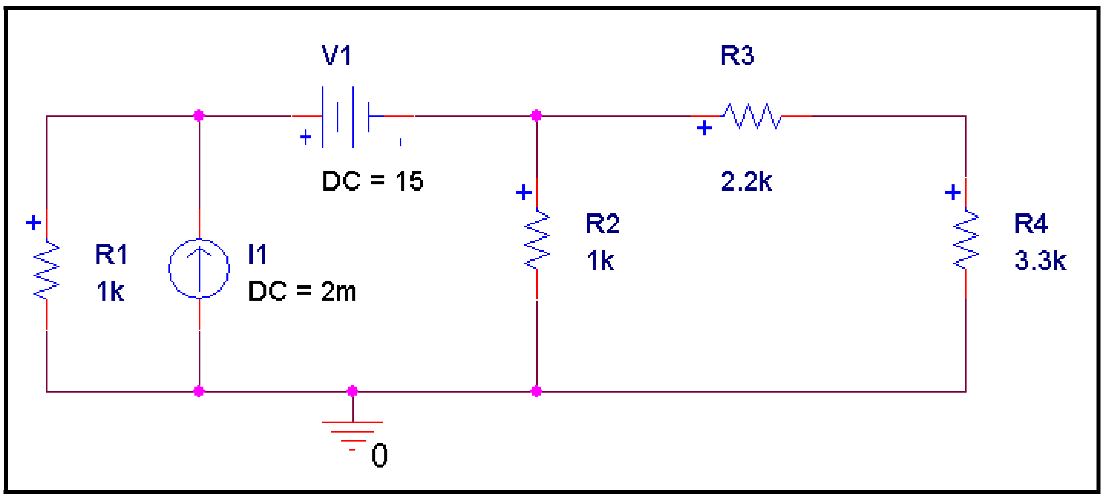
EXERCISE 3-1: Find the DC node voltages for the circuit below:



SOLUTION: Set up a bias point analysis, run the simulation, and then display the voltages on the schematic.

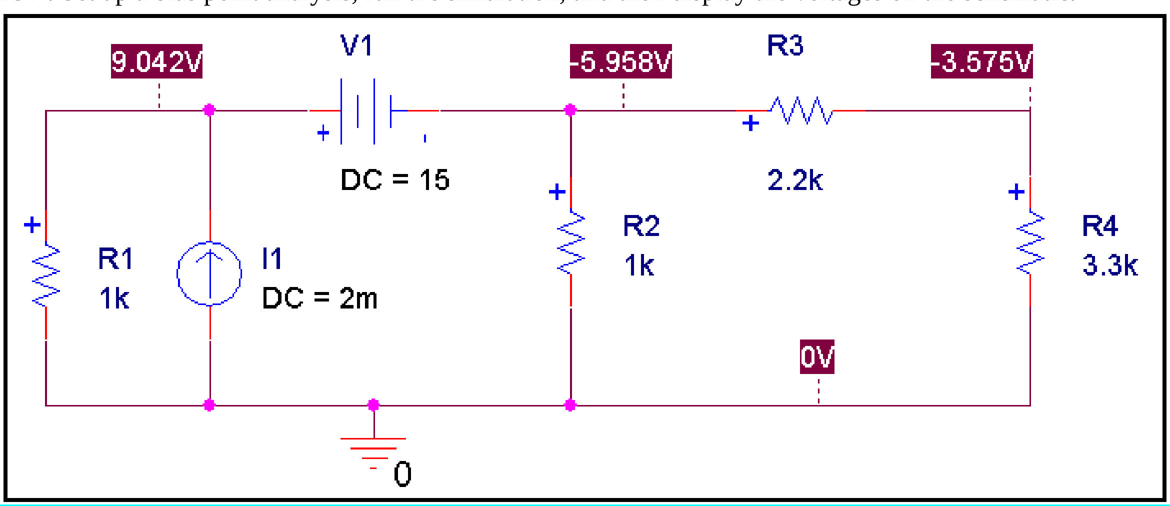


EXERCISE 3-2: Find the DC node voltages for the circuit below:



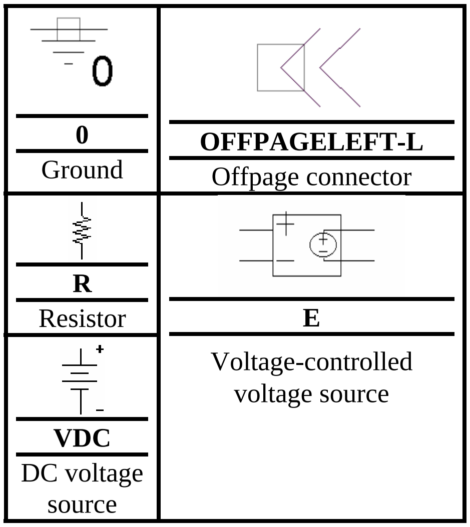
HINT: A DC current source is the part called IDC.

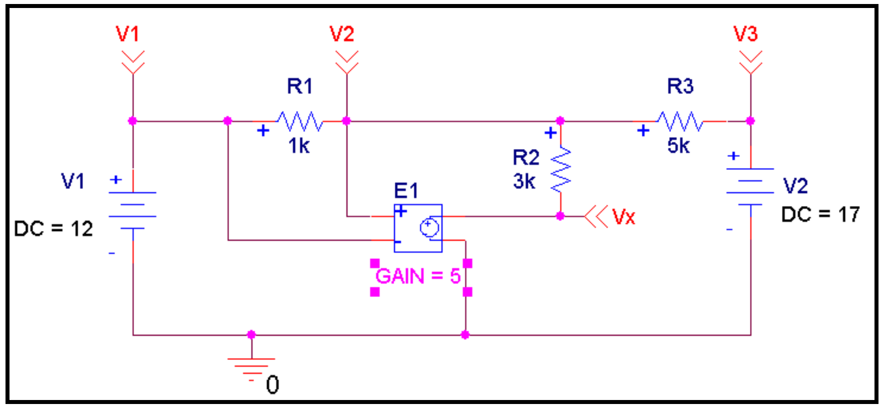
SOLUTION: Set up a bias point analysis, run the simulation, and then display the voltages on the schematic.



3.B. Nodal Analysis with Dependent Sources

To illustrate an example with dependent sources, we will perform a node voltage analysis on the circuit below. If you are unfamiliar with wiring a circuit, review Part 1. Create the circuit below.





The new part in this circuit is the voltage-controlled voltage source. The way this element is wired, the voltage at node **Vx** is 5 times the voltage across **R1**: $V_x = 5(V_2 - V_1)$. If you zoom in[1] on the voltage-controlled voltage source, you will see the graphic shown in Figure 3-1. Notice that the plus (+) and minus (–) terminals are open circuited. These connections draw no current and only sense the voltage of the nodes to which they are connected. The right half of the graphic contains the dependent source. The voltage of this source is the gain times the voltage at the sensing nodes.

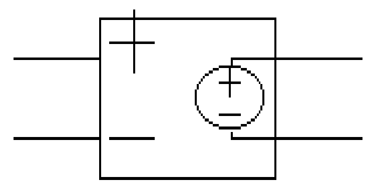
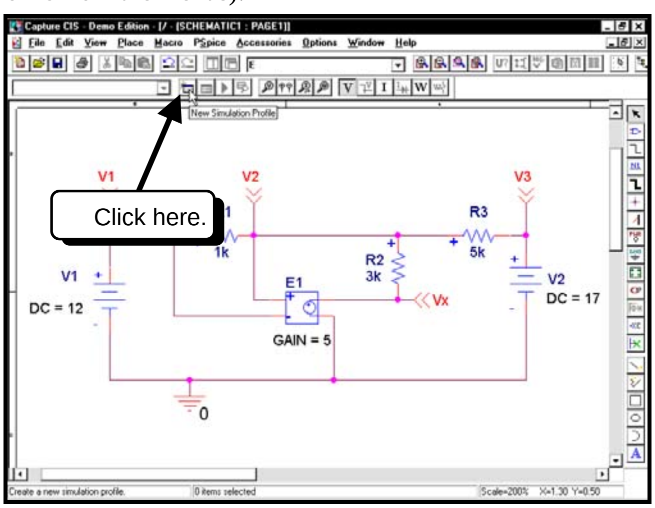
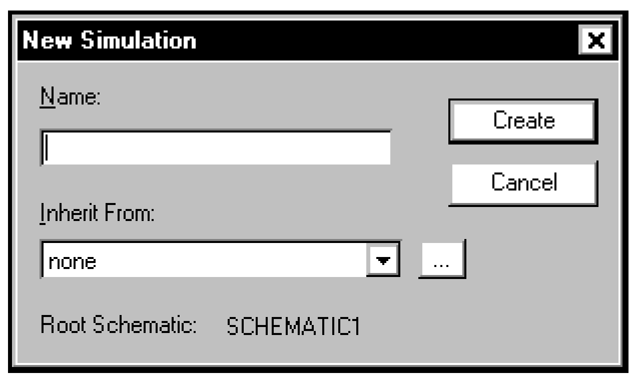


Figure 3-1: Voltage-controlled voltage source.

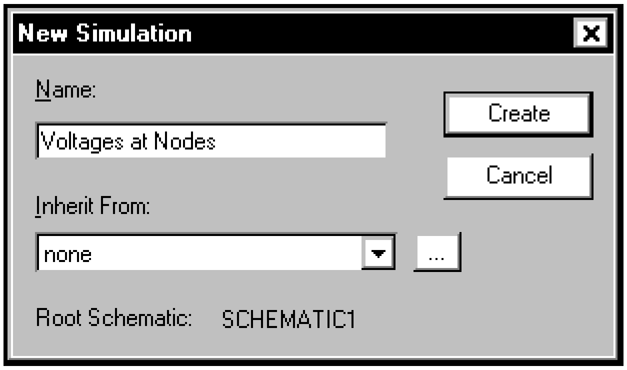
We must now set up a Bias Point simulation. Click on the **New Simulation Profile** button to create a new profile (in the last section, we selected **PSpice** and then **New Simulation Profile** from the menus):

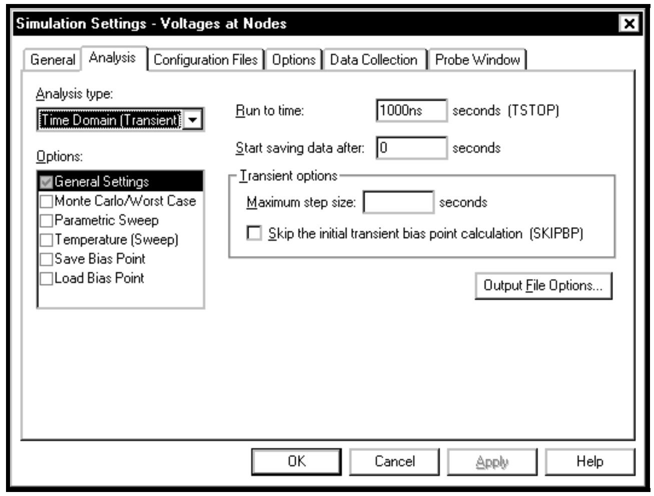




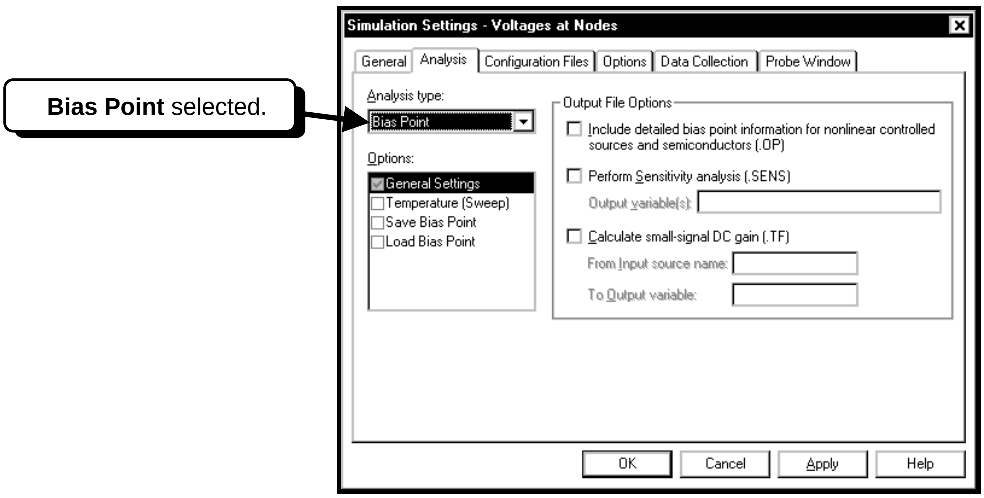
Specify a name for the simulation profile and then click the **Create** button:

[1]To zoom in on a particular spot on the screen, select **View** and then **In** from the Capture main menu. The cursor will be replaced by crosshairs (+). Move the crosshairs to the spot on the screen where you want to zoom in. Click the ***LEFT*** mouse button. Repeat the steps to make the drawing larger if necessary.



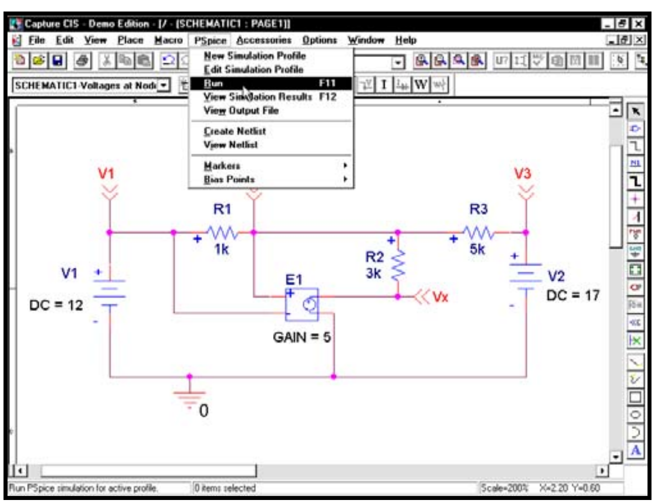


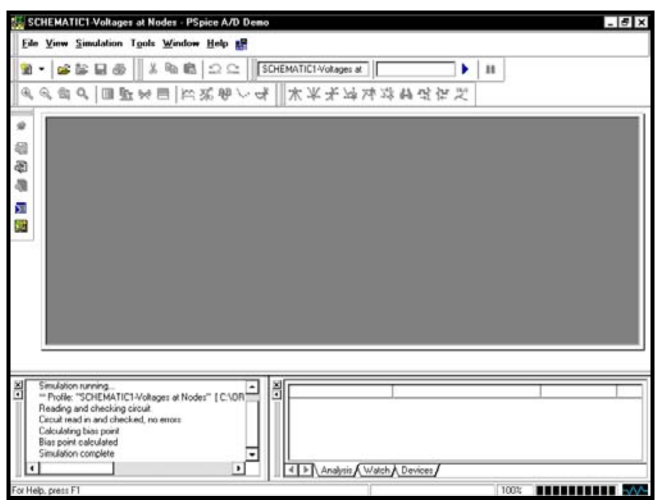
Specify a **Bias Point** ***Analysis type***:



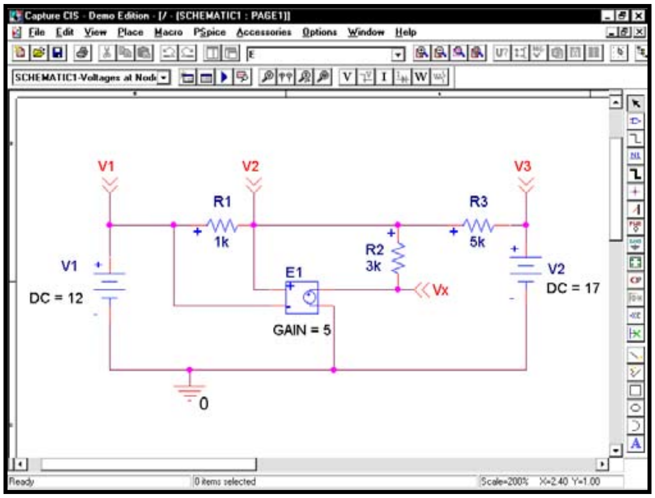
We will display the results on the schematic and we do not need to specify any ***Output File Options***. Click the ***OK*** button to save the settings.

Select **PSpice** and then **Run** to simulate the circuit:

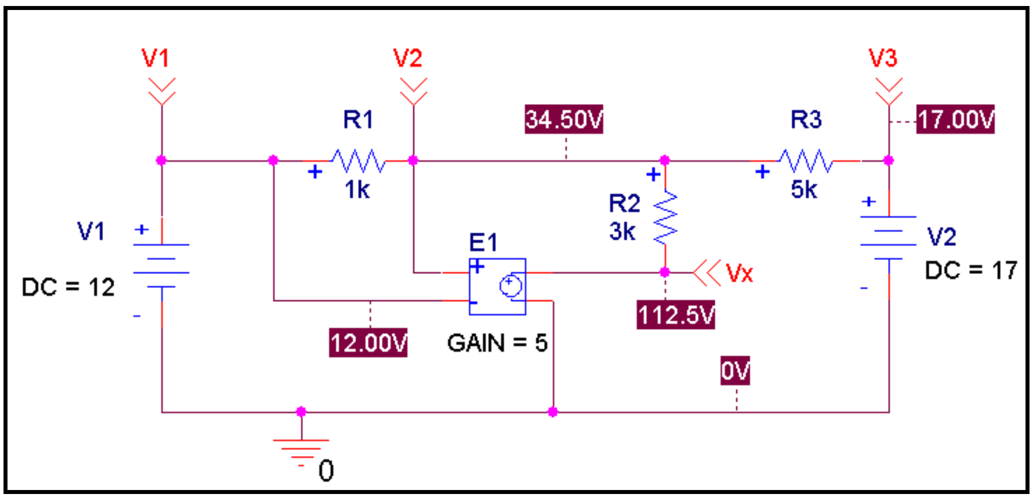




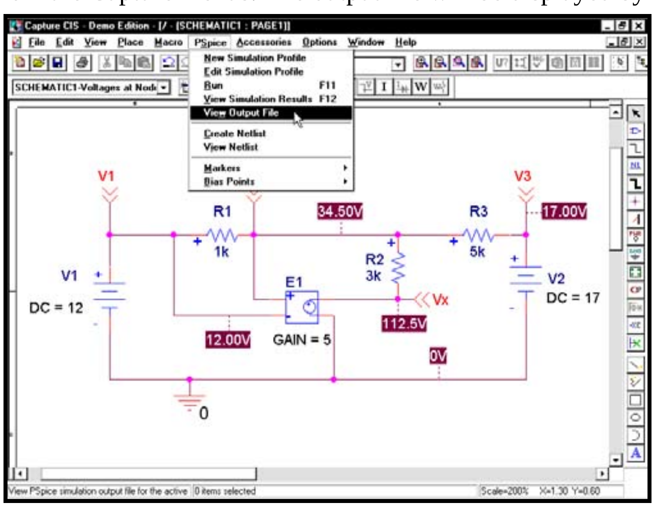
When the simulation is complete, switch to Capture to display the circuit:

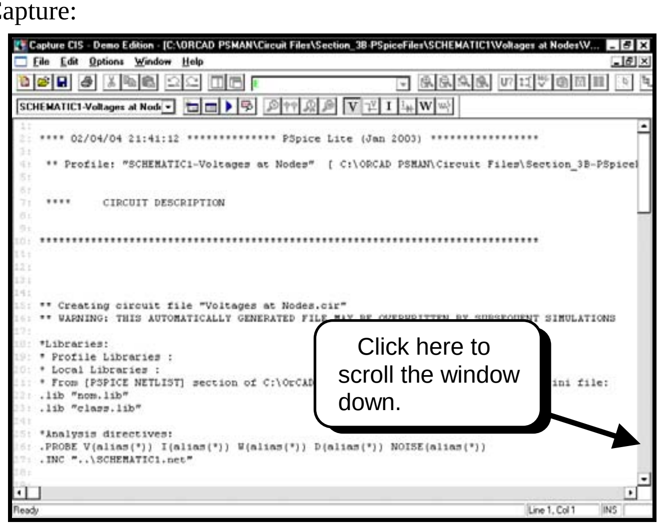


If the node voltages are not displayed, use the steps detailed in the previous section to display node voltages on your schematic:

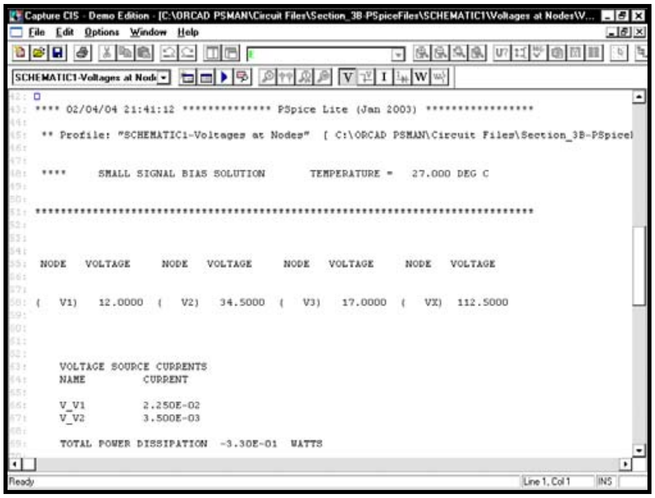


The same results are also contained in the output file. To examine the output file, select **PSpice** and then **View Output File** from the Capture menus. The output file will be displayed by Capture:

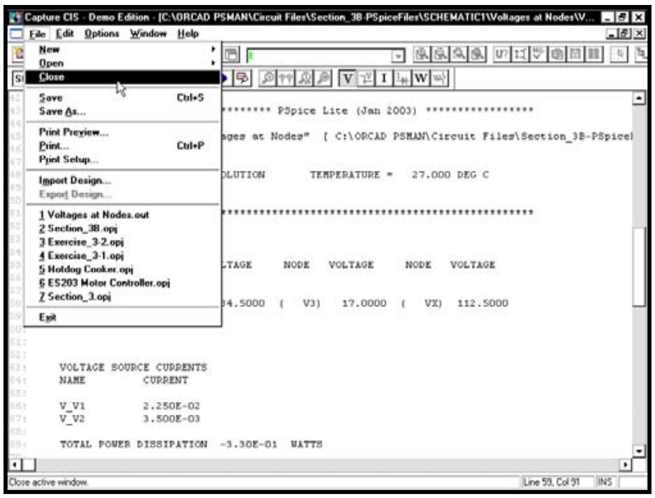


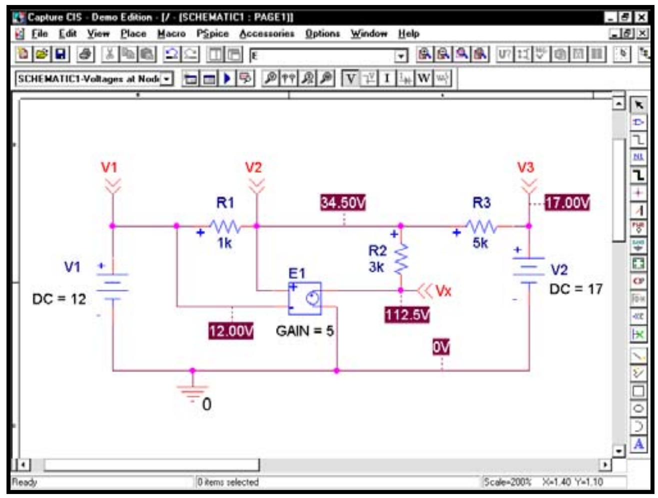


The node voltage results are contained near the bottom of the file. Click the ***LEFT*** mouse button on the vertical scroll bar until you see this text:

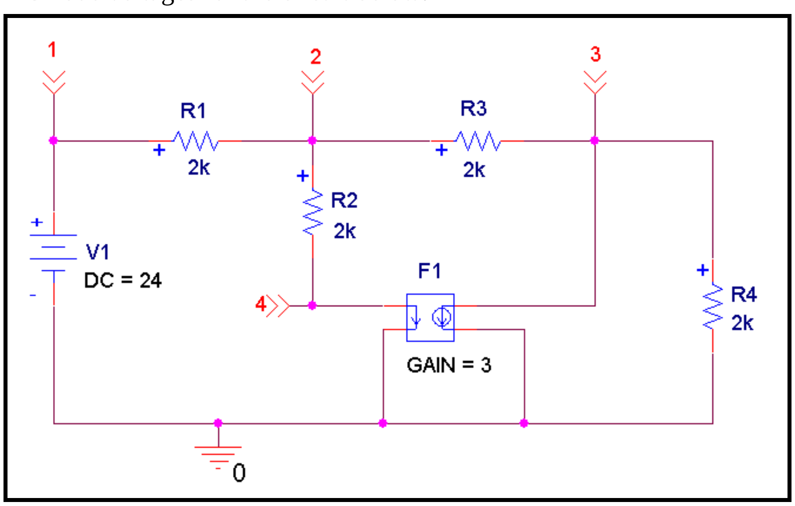


The results show the voltages at the specified nodes relative to ground, and are the same numerical results as displayed on the schematic. To close the text editor program, select **File** and then **Close** from the Capture menus. You will return to the schematic:



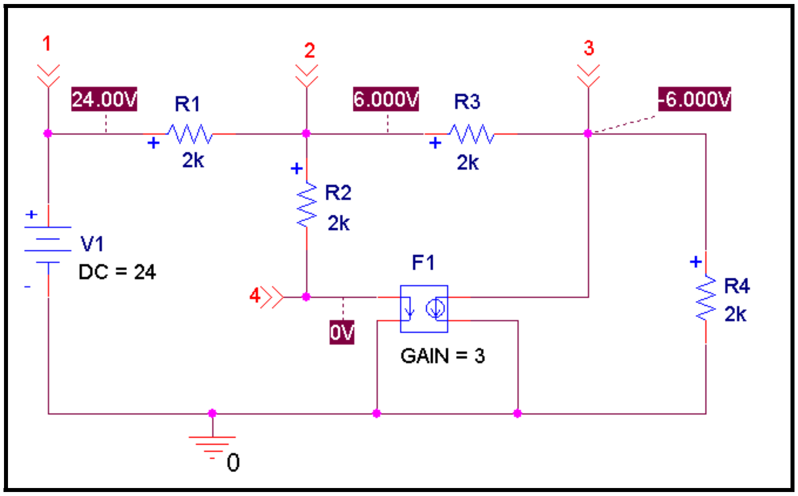


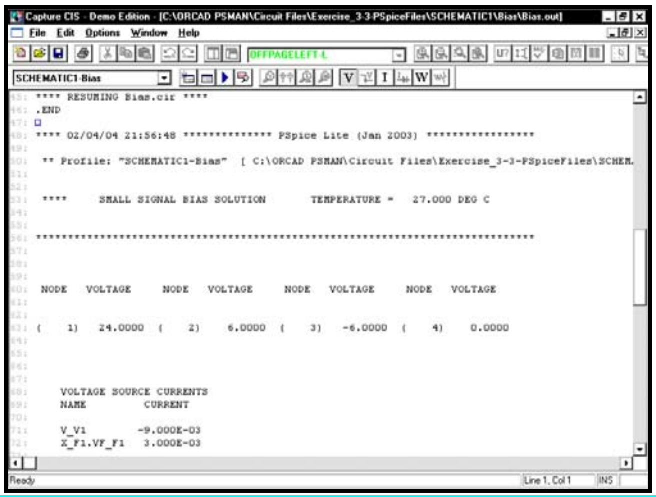
EXERCISE 3-3: Find the DC node voltages for the circuit below:



HINT: F1 is a current-controlled current source. Note that node 4 is connected to ground with a wire. Thus, the voltage of node 4 should be zero volts. Node 4 is necessary because it joins the lower terminal of R2 to the current-sensing terminal of F1.

SOLUTION: Set up a Bias Point analysis and then simulate the circuit. The results can be viewed on the schematic or in the output file:

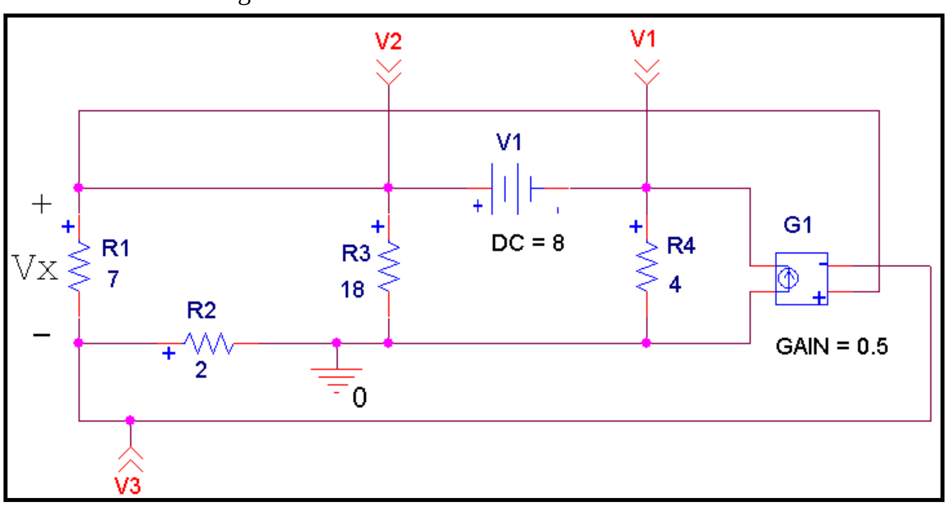




Capture CIS - Demo Edition - [C:\ORCAD PSMAN\Circuit Files\Exercise_3-3-PSpiceFiles\SCHEMATIC1\Bias\Bias.out]

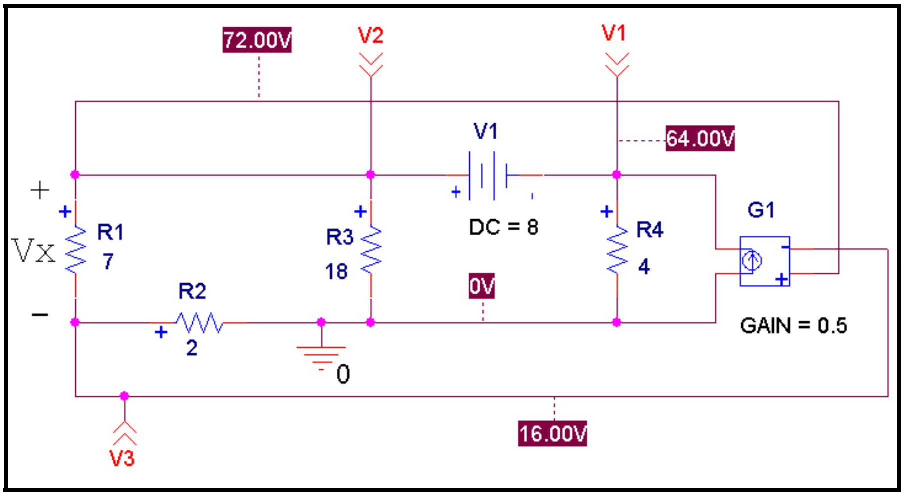
```
**** RESUMING Bias.cir ****
.END

**** 02/04/04 21:56:48 ************** PSpice Lite (Jan 2003) *****************

 ** Profile: "SCHEMATIC1-Bias"  [ C:\ORCAD PSMAN\Circuit Files\Exercise_3-3-PSpiceFiles\SCHEM.


 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C


******************************************************************************



 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE


(    1)   24.0000  (    2)    6.0000  (    3)   -6.0000  (    4)    0.0000




    VOLTAGE SOURCE CURRENTS
    NAME         CURRENT

    V_V1        -9.000E-03
    X_F1.VF_F1   3.000E-03
```

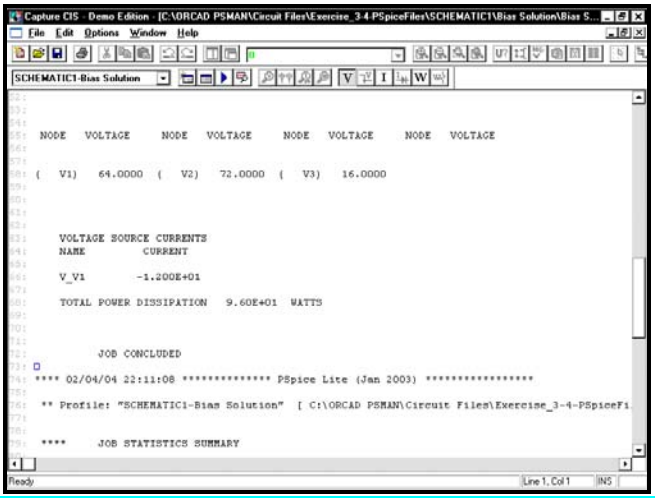
EXERCISE 3-4: Find the DC node voltages for the circuit below:



HINT: **G1** is a voltage-controlled current source. The current through **G1** is 0.5 times the voltage Vx. Note that Vx is the voltage at node V2 minus the voltage at node V3. It is not necessary to add the text Vx to your circuit.

SOLUTION: Set up a Bias Point analysis and then simulate the circuit. The results can be viewed on the schematic or in the output file:





3.C. Diode DC Current and Voltage

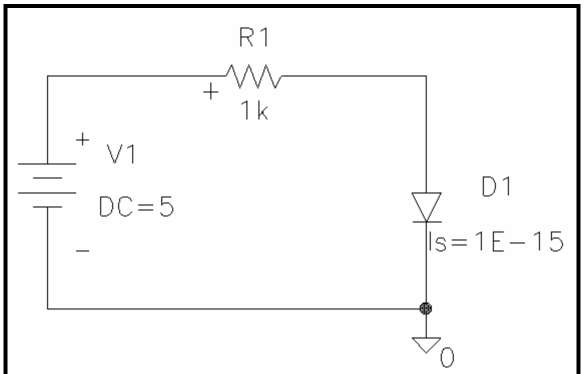
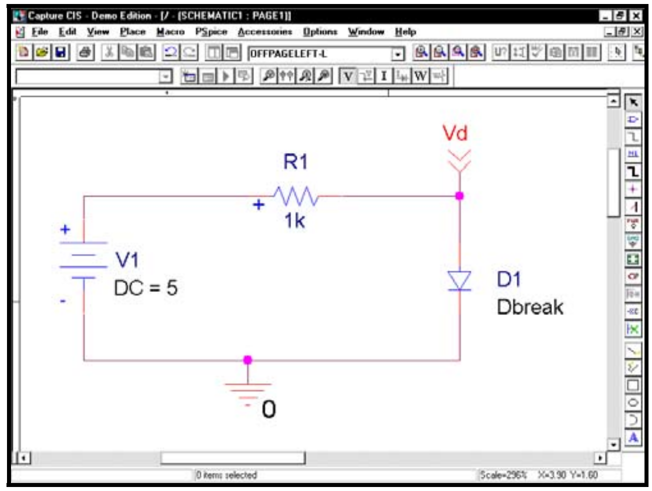


Figure 3-2

We will now use PSpice to find the diode current and voltage in the circuit of Figure 3-2. The diode current is given as $I_D = I_s[\exp(V_D/\eta V_T) - 1]$. I_S is the diode saturation current and is 10^{-15} amps for this example. V_T is the thermal voltage and is equal to 25.8 mV at room temperature. η is the emission coefficient for the diode and its default value is 1. PSpice automatically runs all simulations at room temperature by default.

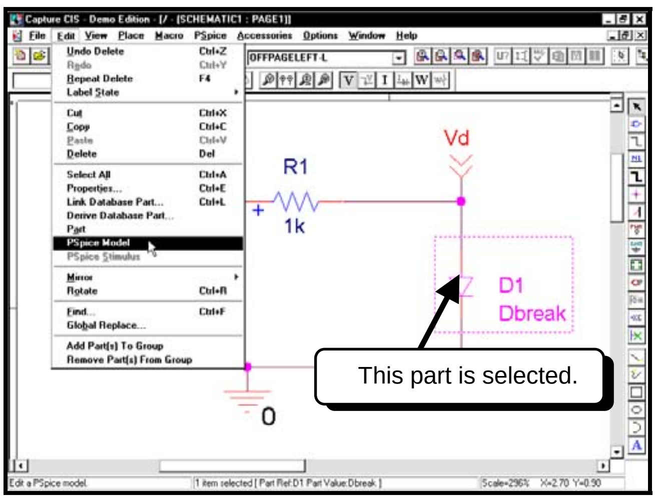
When you use a diode in a circuit, you will have to specify a model for the diode. In our case, the model will tell PSpice the value of I_S for our diode. The class libraries have a number of predefined models that are usually used in a classroom environment. However, the model for this diode is not in our libraries so we will have to define a new model for it. The part for the diode you should use is **Dbreak**. This diode is used to define your own model. Draw the circuit below:

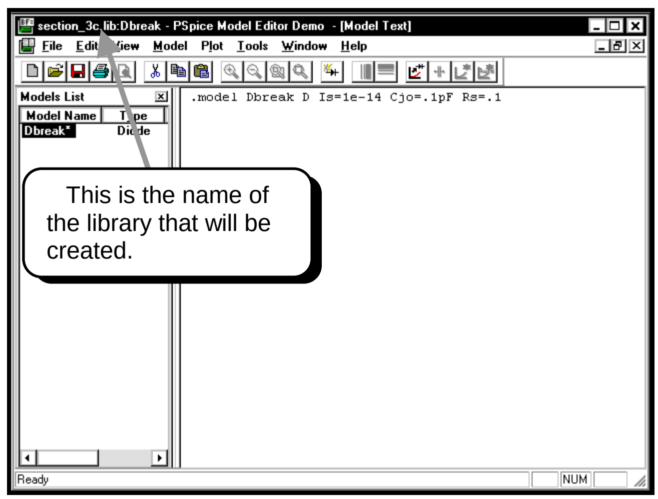
R Resistor	**0** Ground
VDC DC voltage source	**Offpage-L** Offpage connector
Dbreak Diode used to define your own model.	



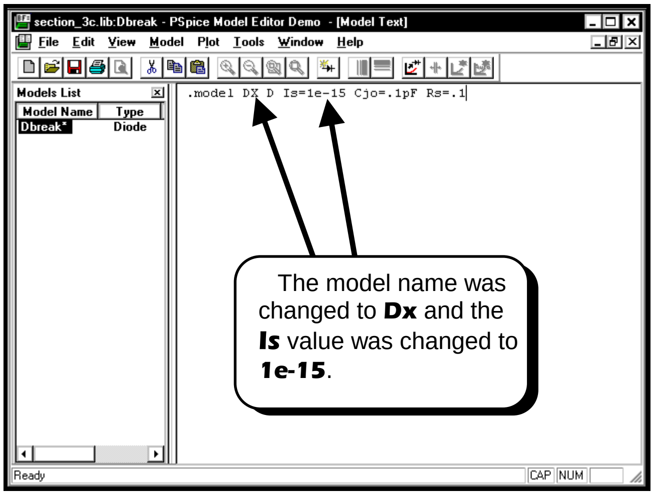
Note for this example: If you use file Section_3c.opj from the CD-ROM or from the files copied from the CD-ROM to your hard drive during installation, you may have problems with the libraries during the simulation. For this example, you should draw the circuit from scratch and not use the provided example files.

We must now define the model for the diode. Click the ***LEFT*** mouse button on the diode graphic, ─▷├─. The graphic should turn pink, indicating that it has been selected. Next, select **Edit** and then **PSpice Model** from the Capture menus:

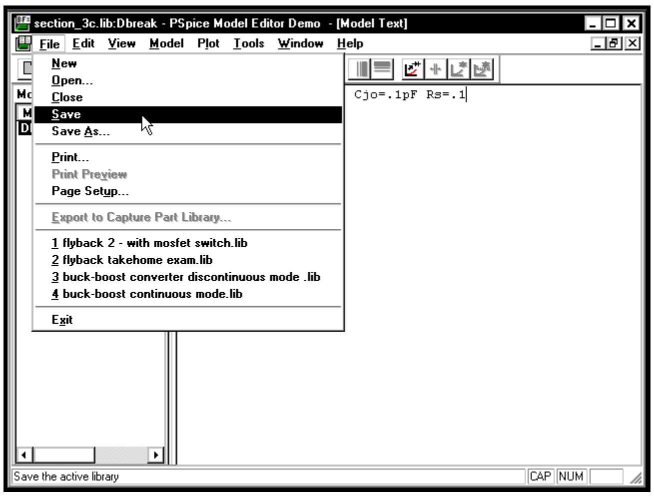


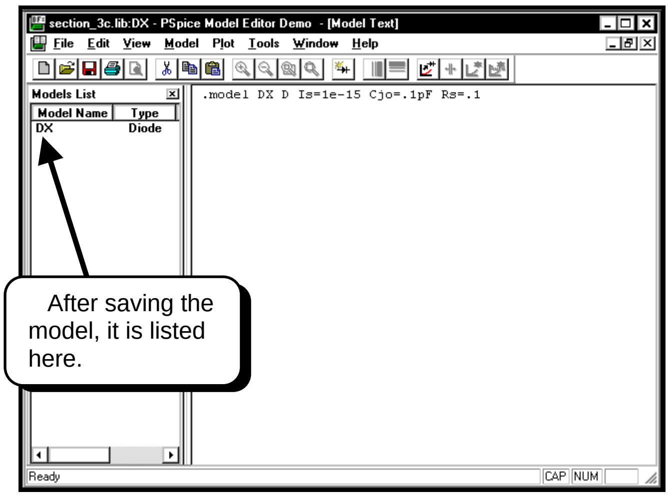


The right screen is the PSpice model editor and it tells us that the model we create will be saved in a file named **SECTION_3C.Lib**. The model for the part we selected is displayed by the model editor; in this case the model name is Dbreak. In this model, **Rs** is the series resistance of the diode and **Cjo** is the junction capacitance. The only parameter that we will change for this example is the saturation current **Is**. Note in the screen capture above that the default value of I_S is 1×10^{-14} amps. We can use the model editor to create a new model. Change the name of the model to **Dx** and change I_S to 1e-15:

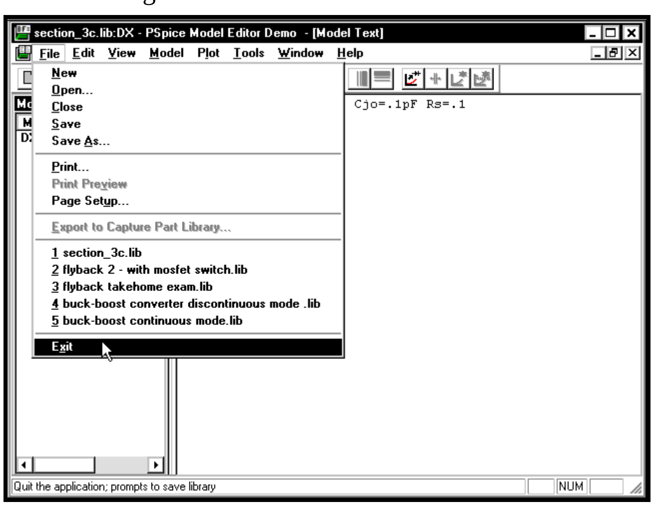


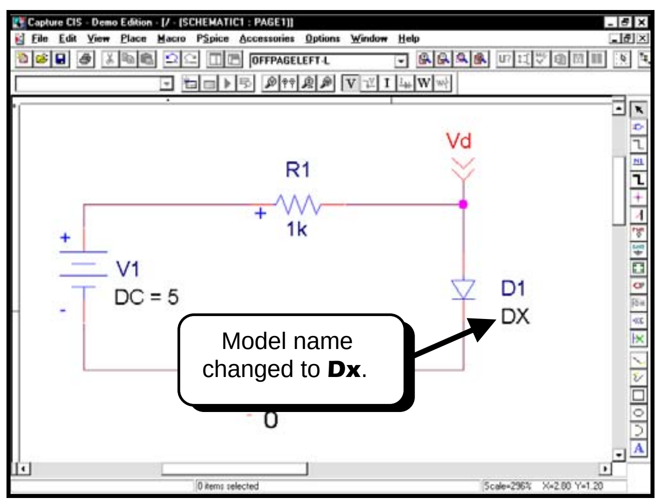
Select **File** and then **Save** to create and save the model:



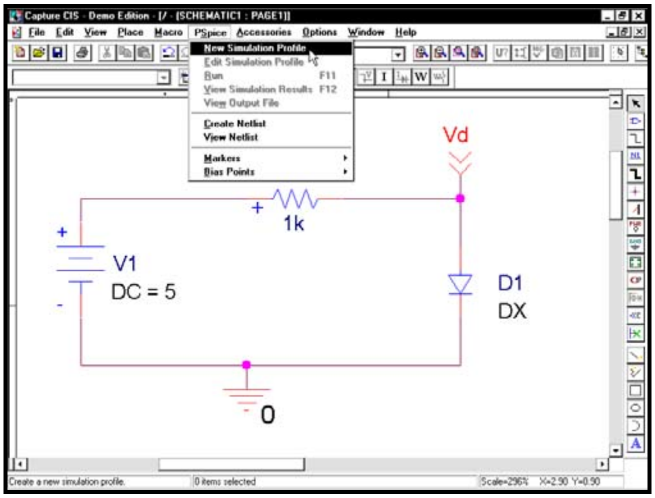


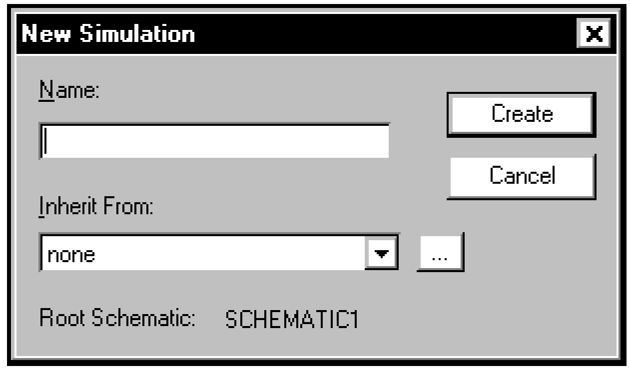
After we save the model, it is listed in the left window pane. This window pane lists all of the models contained in the library we are editing, in this case Section_3c.lib. Select **File** and then **Exit** to return to the schematic. Notice that the diode model name has changed from Dbreak to **Dx**:



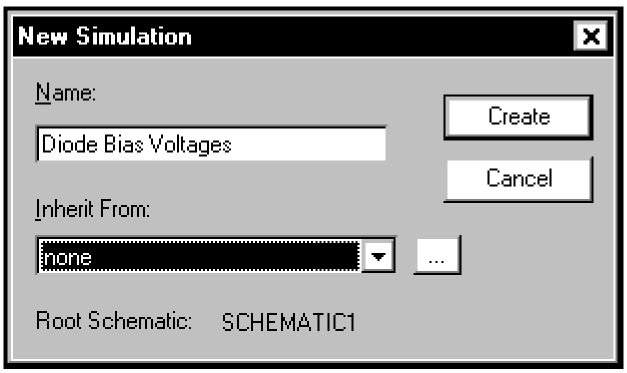


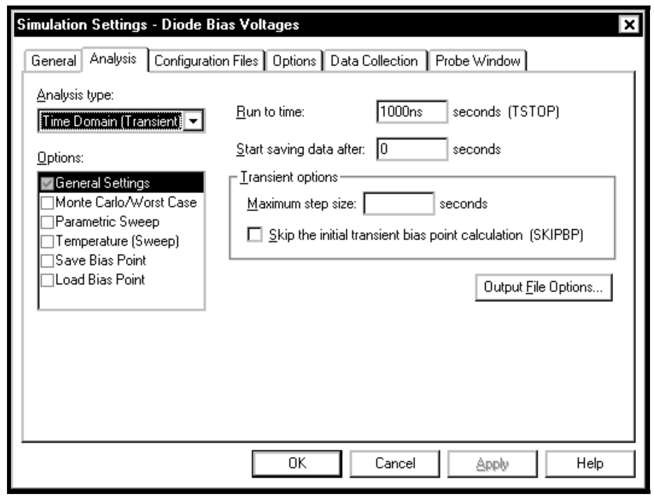
Next, we must set up the bias point simulation. Select **PSpice** and then **New Simulation Profile**:



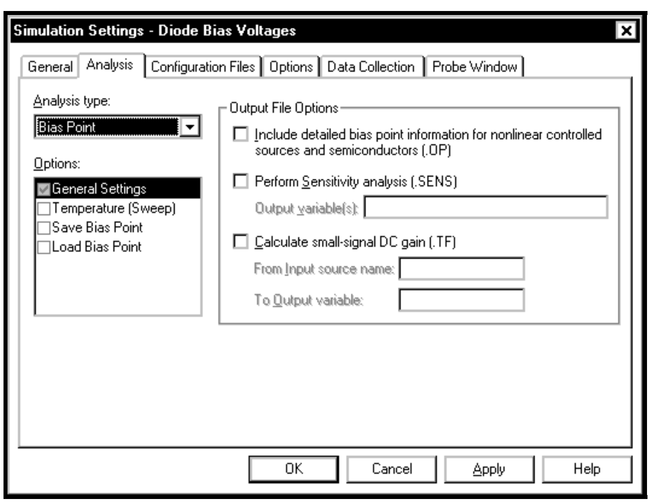


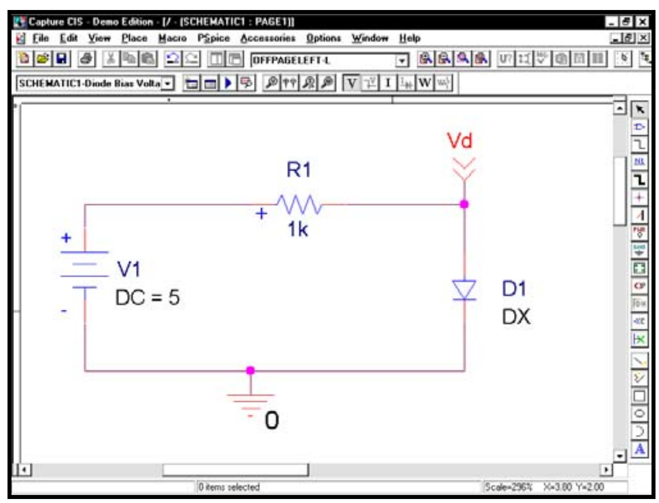
Enter a name for the profile and then click the **Create** button:



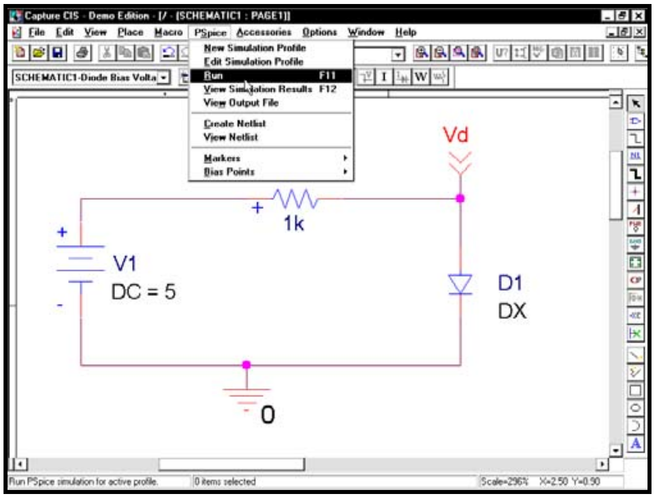


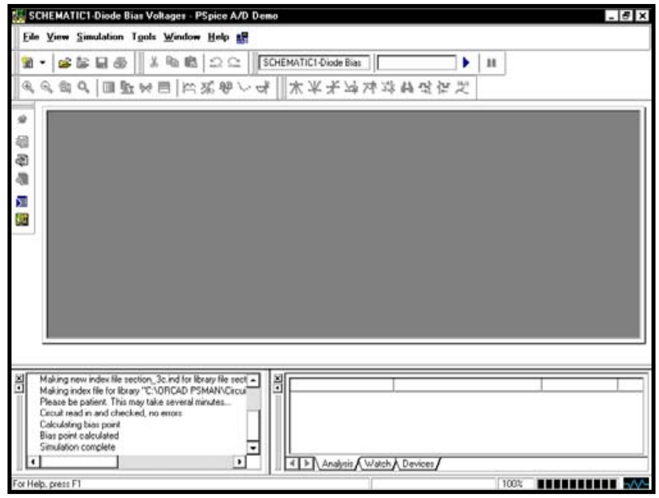
Select the **Bias Point *Analysis type*** and click the **OK** button:





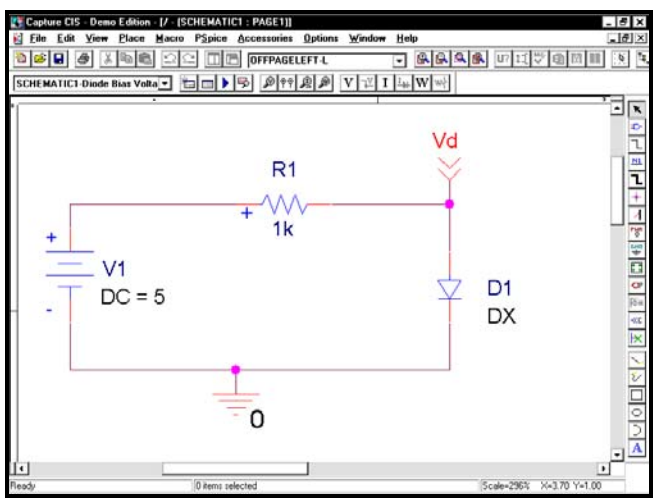
Select **PSpice** and then **Run** to simulate the circuit:



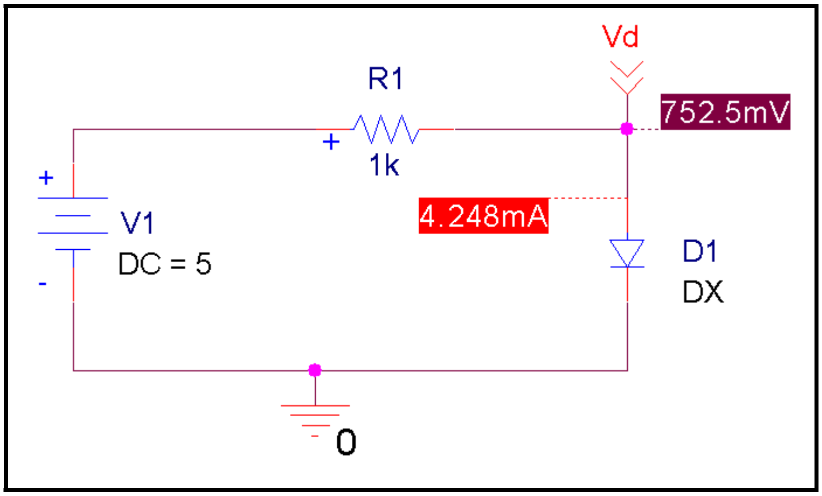


The messages indicate that an index file does not exist for the library file Section_3c.lib, and that PSpice will create the file. Library files (files with the .lib extension) are text files that contain model information. Some of the library files in the professional version of PSpice contain thousands of models. To reduce the simulation time, the libraries are compiled once into index files (files with extension .ind). During a simulation, PSpice reads the index file to get the model information it needs, not the library file. PSpice can read index files much more quickly than library files. The library file is provided because it is a text file and easy for humans to read. If you create a new library file or edit a model in an existing library file, PSpice will compile the .lib file into an .ind file. PSpice will leave the index file alone if no changes have been made to the library file. Compiling a library file into an index file can take a long time so it is done only when necessary. The warning messages are telling us that the index file for Section_3c.lib does not exist and it is creating the index file. If you run the simulation again, these warning messages will not be generated because the index file already exists. If you make changes to the diode model, you are changing the model in library Section_3c.lib. Warning messages will again be generated stating that file Section_3c.lib has changed and it must recreate the index file for that file.

In this case, the messages do not indicate a serious problem and we can view the results. Bring the Capture window to the top:

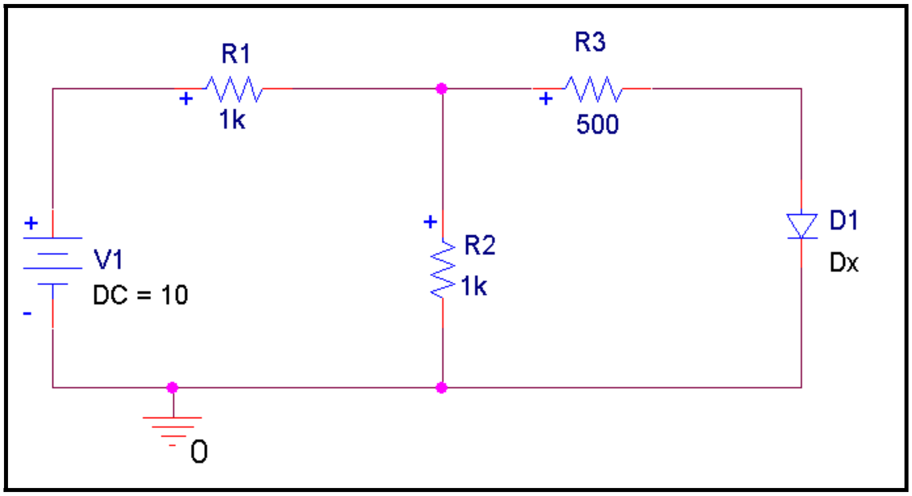


Use the techniques covered on pages 160 to **Error!Error! Bookmark not defined.** to display the node voltage at Vd and the DC current flowing into the diode:

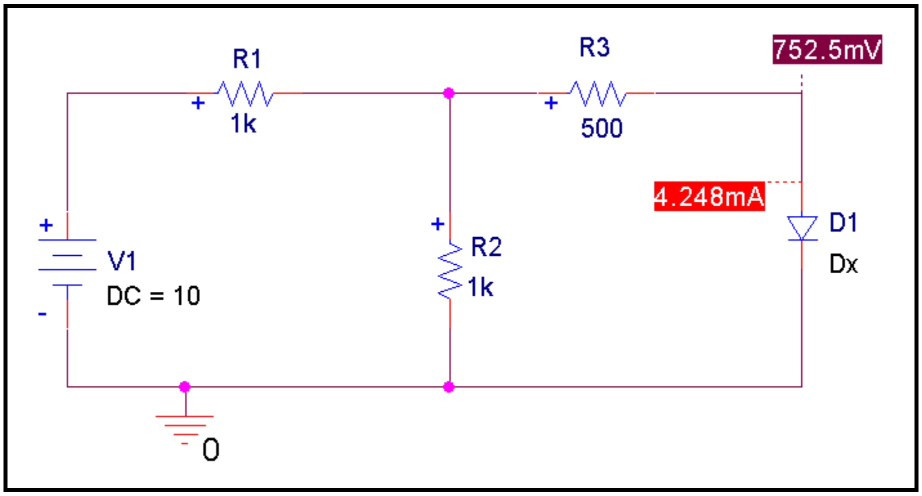


We see that the diode current is 4.248 mA, and the diode voltage is 0.7525 volts. Note that the node voltage displayed gives the voltage of the node relative to ground. Since the other side of the diode is grounded, the voltage displayed is also the diode voltage.

EXERCISE 3-5: Find the diode current and voltage in the circuit below:



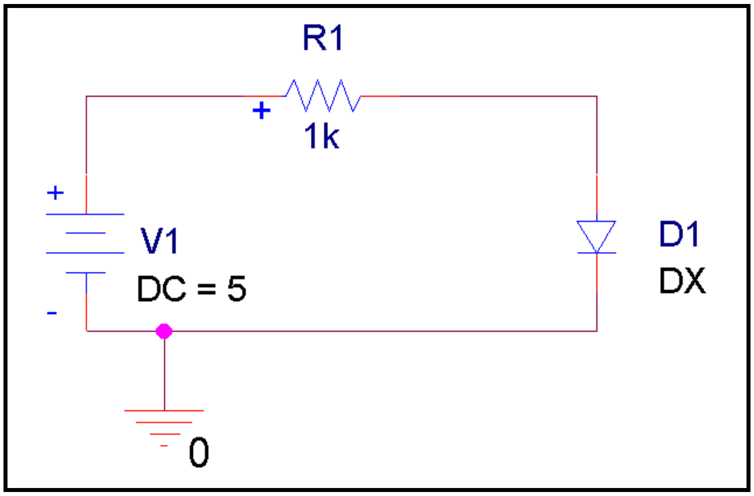
SOLUTION: Use the same diode parameters as in the previous example. Set up a Bias Point analysis and then simulate the circuit. The results can be viewed on the schematic or in the output file:



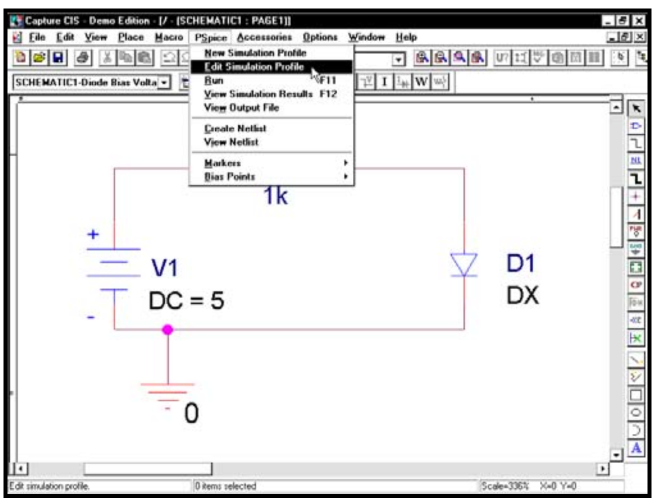
The diode voltage is 0.7525 V, and the diode current is 4.248 mA. The diode voltage and current of this circuit are the same as the diode voltage and current of the previous example. This result should be expected since the Thevenin equivalent of V1, R1, R2, and R3 in this exercise is exactly the same as the circuit of the previous example.

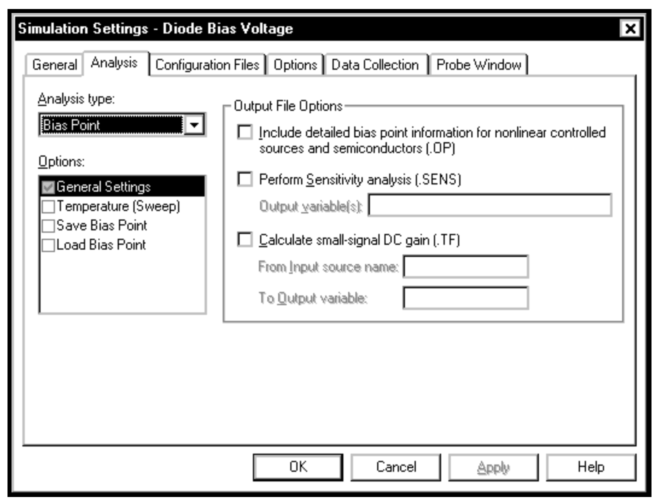
3.C.1. Changing the Temperature of the Simulation

In the last simulation we found the diode voltage and current at the default temperature of 25 °C. Suppose we want to simulate the circuit at a different temperature? This can easily be done by selecting the temperature option in the simulation profile. We will continue with the circuit of the previous simulation:

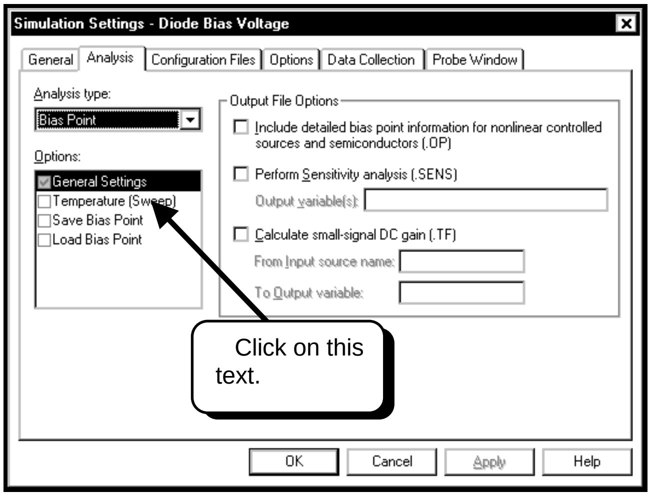


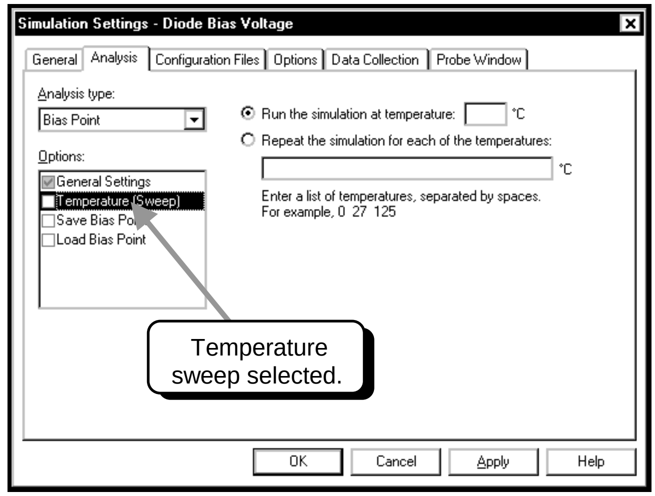
We now wish to edit the simulation profile. Since we are using a previously created profile, we need to open the profile rather than create a new profile. Select **PSpice** and then **Edit Simulation Profile** from the Capture menus to edit the existing profile:



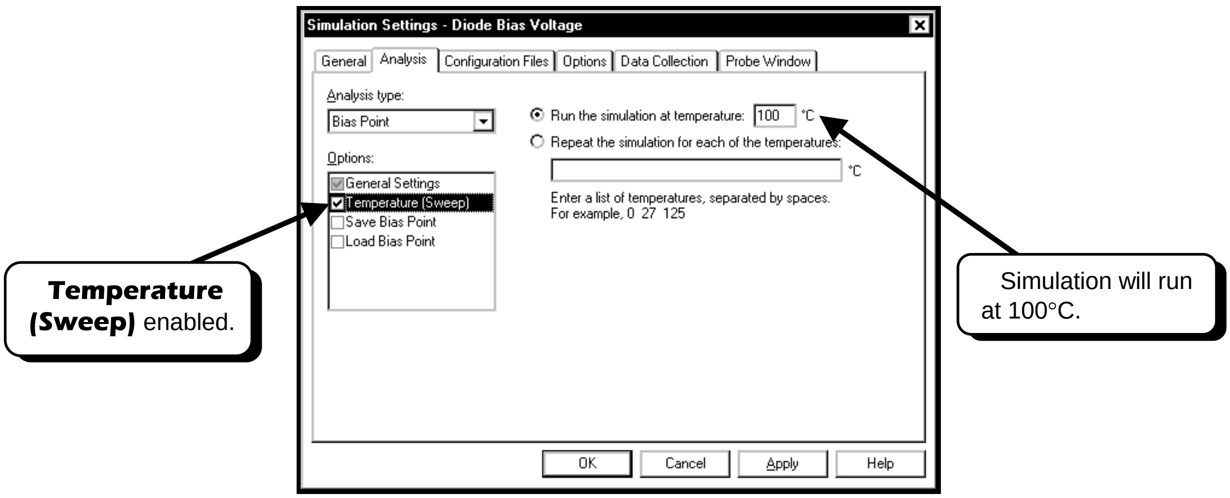


The **Bias Point** simulation is already selected from the previous example. Here we need to specify the temperature of the simulation. By default, all simulations are run at 25°C. To specify a temperature other than the default, select the **Temperature (Sweep)** option:

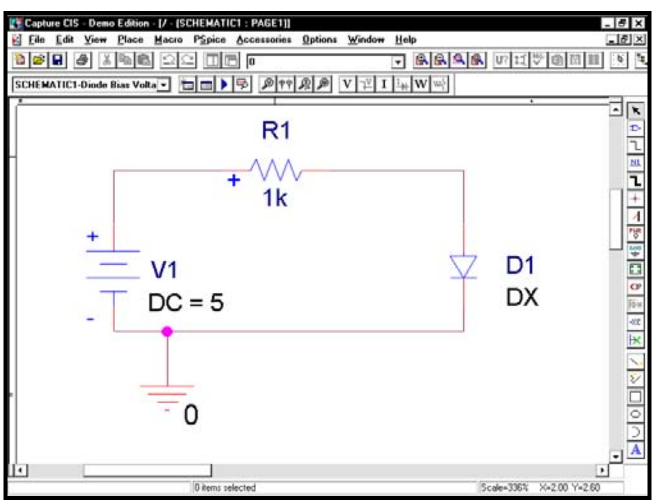




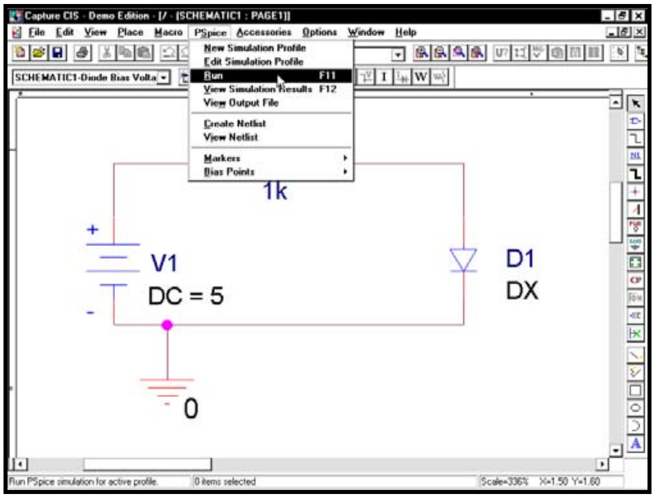
The **Temperature (Sweep)** allows us to specify a single temperature or a list of temperatures. If you specify a list of temperatures, the simulation will be run several times, once for each temperature you place in the list. We only need to run the simulation once, so fill in the dialog box as shown:

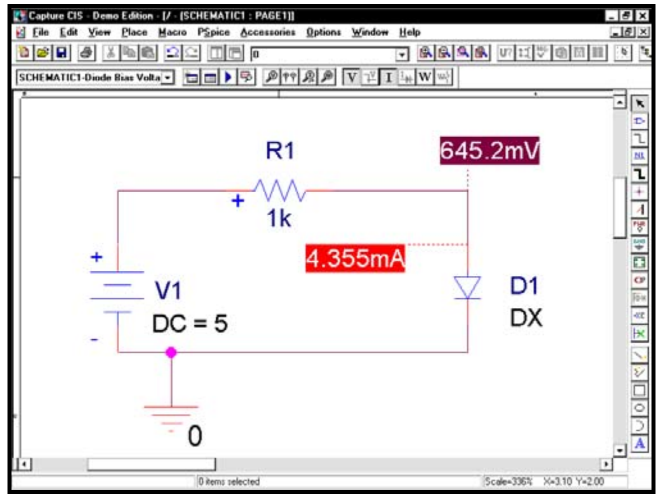


We have specified the simulation to run at 100°C. The simulation of the previous simulation did not specify a temperature and the simulation ran at the default temperature of 25°C. Notice in the screen capture above that there is a checkmark ☑ in the box next to the **Temperature (Sweep)** option. The temperature sweep will not run if this box is not checked. The option above specifies that the **Bias Point** simulation will run at 100°C. Click the **OK** button to return to OrCAD Capture.



Select **PSpice** and then **Run** to run the simulation and then display the diode current and voltage on the schematic:

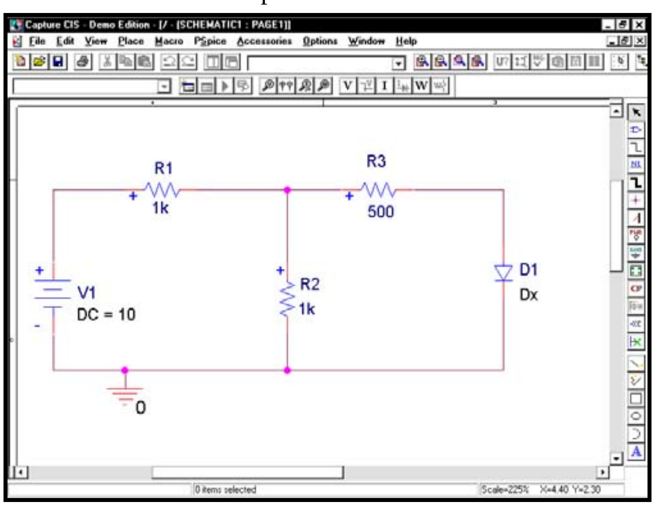


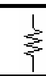
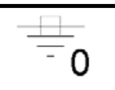
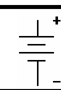
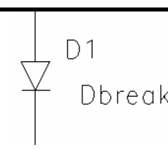


3.D. Finding the Thevenin and Norton Equivalents of a Circuit

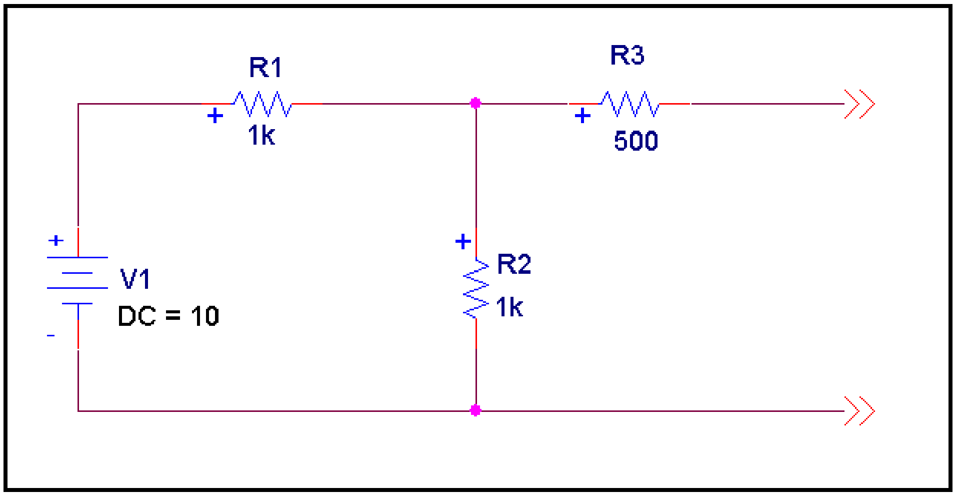
Capture and PSpice can be used to easily calculate the Norton and Thevenin equivalents of a circuit. The method we will use is the same as if we were going to find the equivalent circuits in the lab. We will make two measurements, the open circuit voltage and the short circuit current. The Thevenin resistance is then the open circuit voltage divided by the short circuit current. This will require us to create two circuits, one to find the open circuit voltage, and the second to find the short circuit current. In this example, we will find the Norton and Thevenin equivalent circuits for a DC circuit. This same procedure can be used to find the equivalent circuits of an AC circuit (a circuit with capacitors or inductors). However, instead of finding the open circuit voltage and short circuit current using the DC Nodal Analysis, we would need to use the AC analysis.

For this example, we will find the Thevenin and Norton equivalent circuits for the circuit attached to the diode in EXERCISE 3-5. The circuit is repeated below:

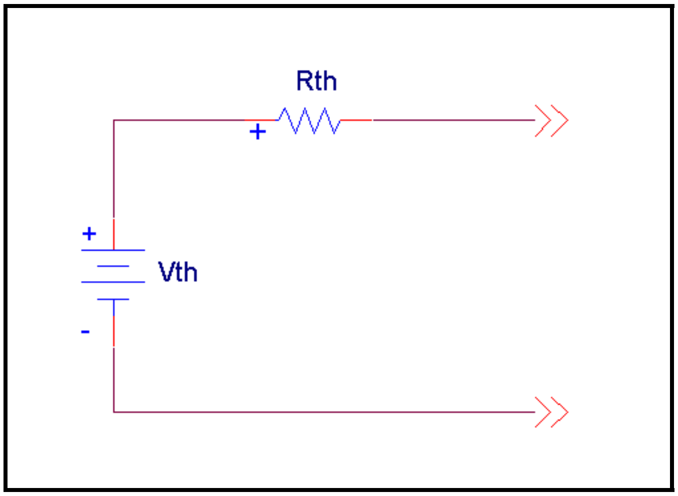


 **R** Resistor	 **0** Ground
 **VDC** DC voltage source	 **Dbreak** Diode used to define your own model.

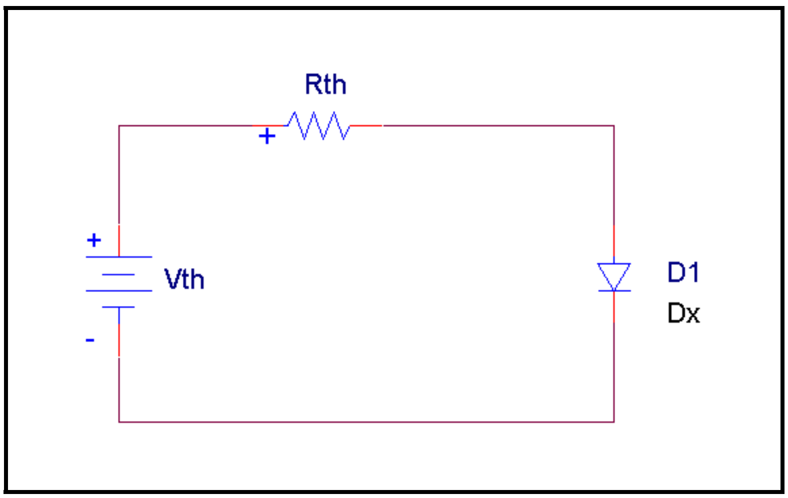
This circuit is difficult because it contains a nonlinear element (the diode) and a complex linear circuit. If we could replace V1, R1, R2, and R3 by a simpler circuit, the analysis of the nonlinear element would be much easier. To simplify the analysis of the diode, we will find the Thevenin and Norton equivalent circuits of the circuit connected to the diode; that is, we will find the Thevenin and Norton equivalents of the circuit below:



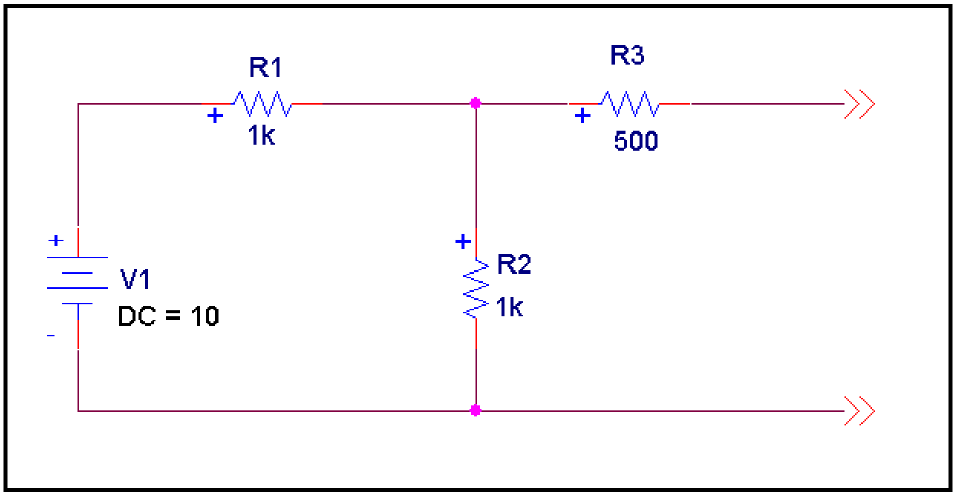
We will convert this circuit into the Thevenin equivalent:



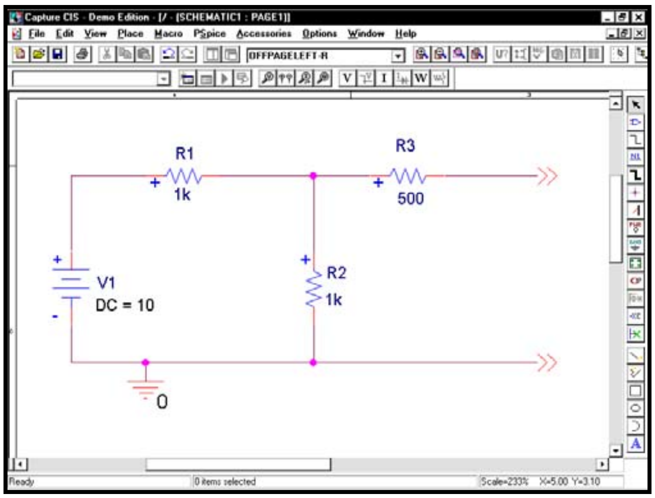
Once we find numerical values for Vth and Rth the entire circuit of EXERCISE 3-5 reduces to:



For determining the diode voltage and current, this circuit is much easier to work with than the original. This example is concerned with finding the numerical values of the equivalent circuit. The analysis of the circuit above was covered in Section 3.C. We will now find the Thevenin and Norton equivalent circuits of the circuit shown below:

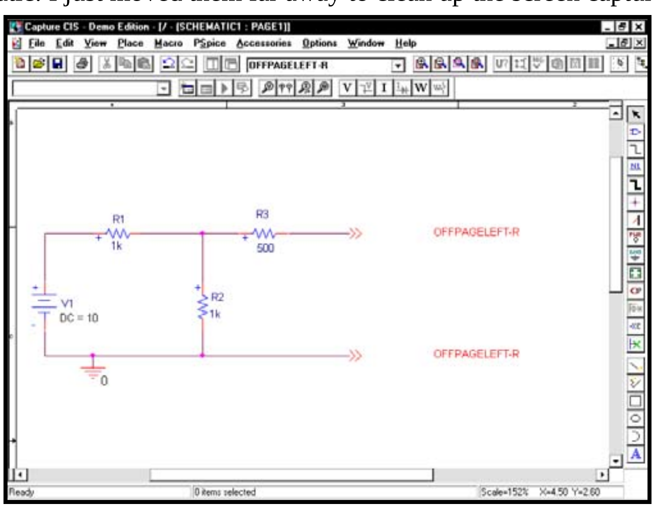


We will first find the open circuit voltage. This is just the voltage across the two terminals in the circuit shown above. First we must add a ground to the circuit. This is necessary because PSpice requires all circuits to have a ground reference:

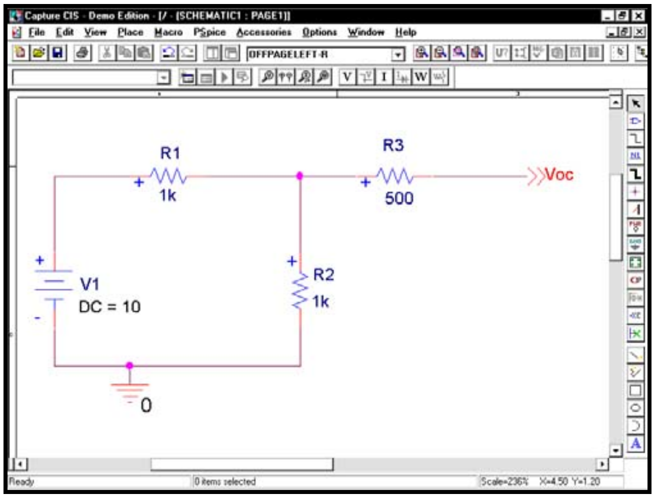


The lower terminal is now at ground potential, zero volts.

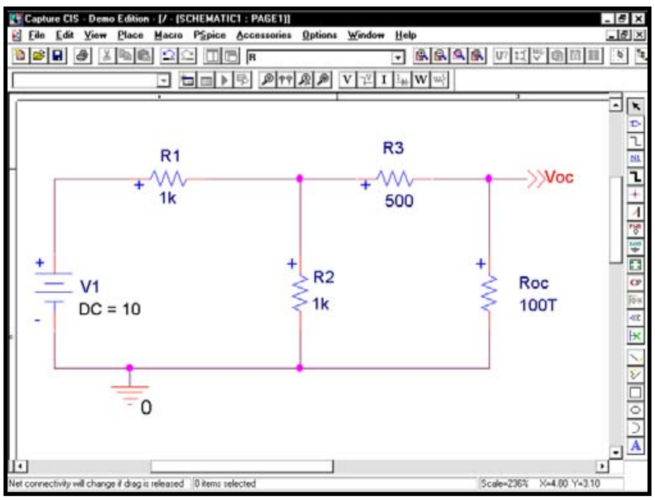
There are several errors in the circuit above. One is that there are two offpage connectors in the circuit and neither of them have labels. All offpage connectors must be labeled or errors will be generated and the simulation will not run. There are actually labels on the schematic: I just moved them far away to clean up the screen capture:



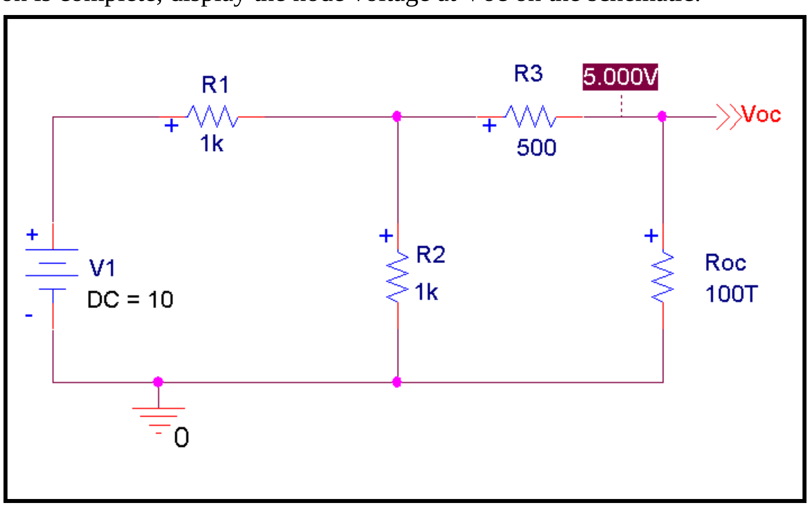
I will delete the lower offpage connector, and rename and move the upper offpage label:



A second error is that there is only one element connected to the upper right node (now labeled **Voc**). That is, nothing is connected to the right terminal of the 500 Ω resistor. PSpice requires that all nodes have at least two elements connected to them. To fix this problem, we must add another element to the circuit that does not affect the operation of the circuit. To simulate an open circuit, I will add a resistor of value 100T. The suffix T in PSpice is a multiplier with a value of 10^{12}. Thus, a resistor with the value 100T will have a resistance of $100{\times}10^{12}$ Ω. This value is significantly larger than all other resistors in the circuit and is an open circuit for all practical purposes. Thus, we will simulate the circuit below:

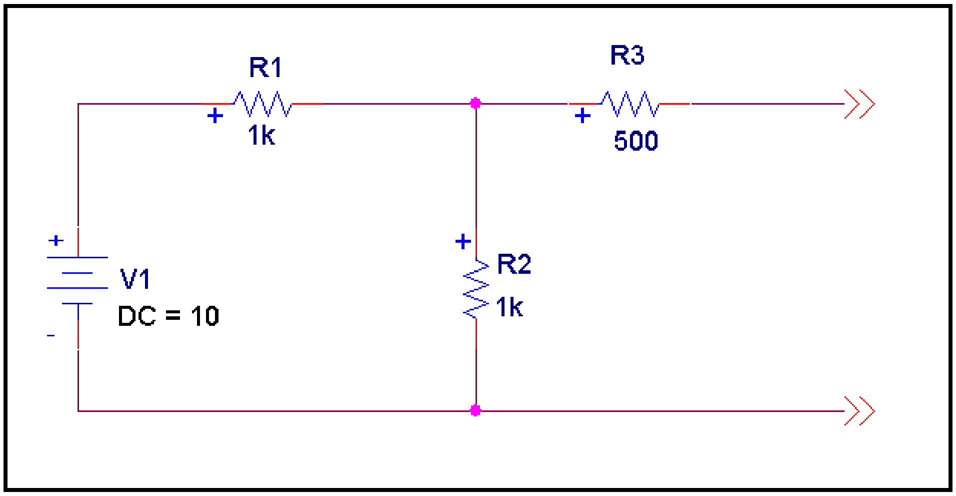


Set up the DC Bias simulation (select **PSpice** and then **New Simulation Profile**) and then run PSpice (**PSpice** and then **Run**). When the simulation is complete, display the node voltage at Voc on the schematic:

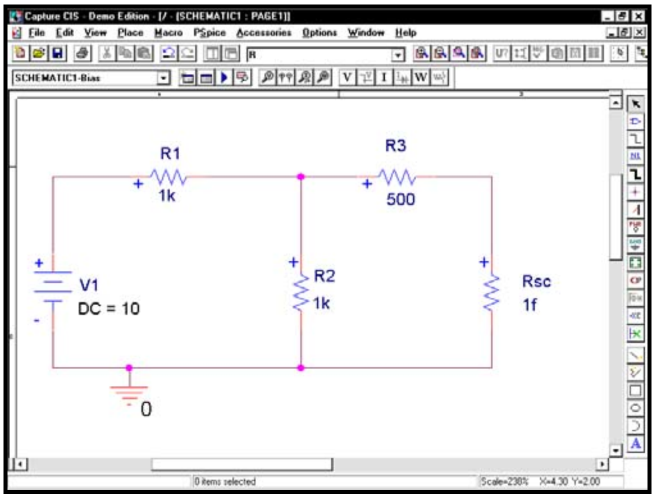


The open circuit voltage is **5.000** volts.

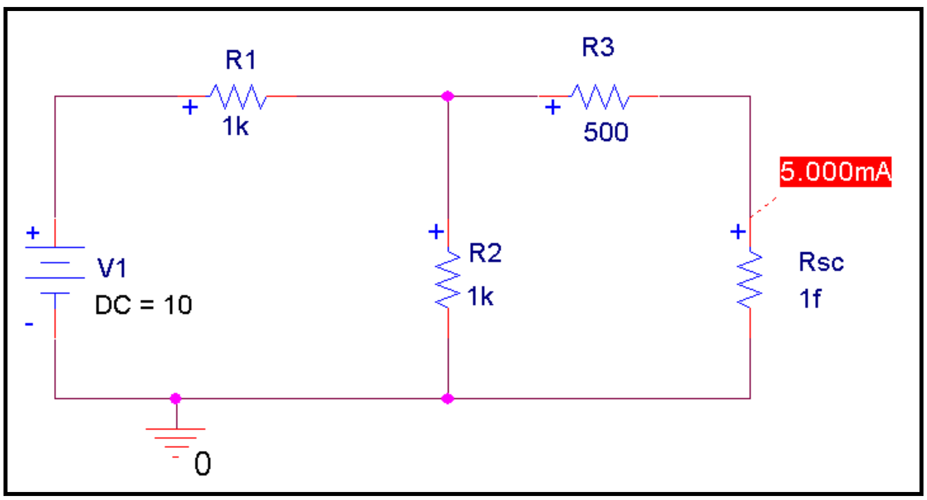
Next we must find the short circuit current. We will start with the original circuit as shown below:



We can short the two terminals together by placing a very small resistance between the two terminals. The current through this resistance will be equal to the short circuit current. We will use a resistance of 1 fΩ or 1×10^{-15} ohms. For all practical purposes this is zero resistance and a short. Modify the circuit as shown:



Simulate the circuit and display the current through **Rsc** on the screen::

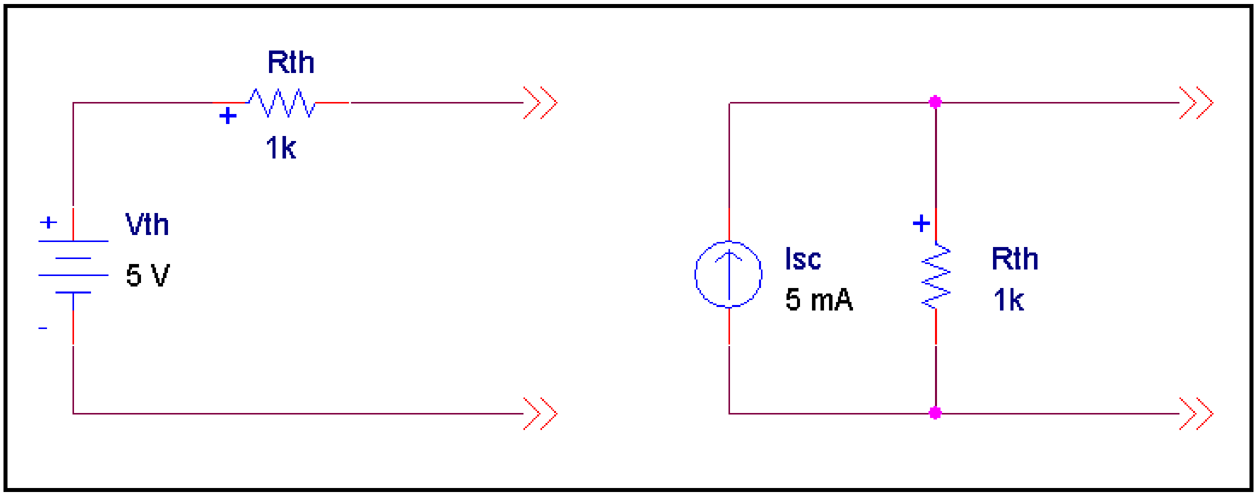


We see that the short circuit current is **5.000** mA.

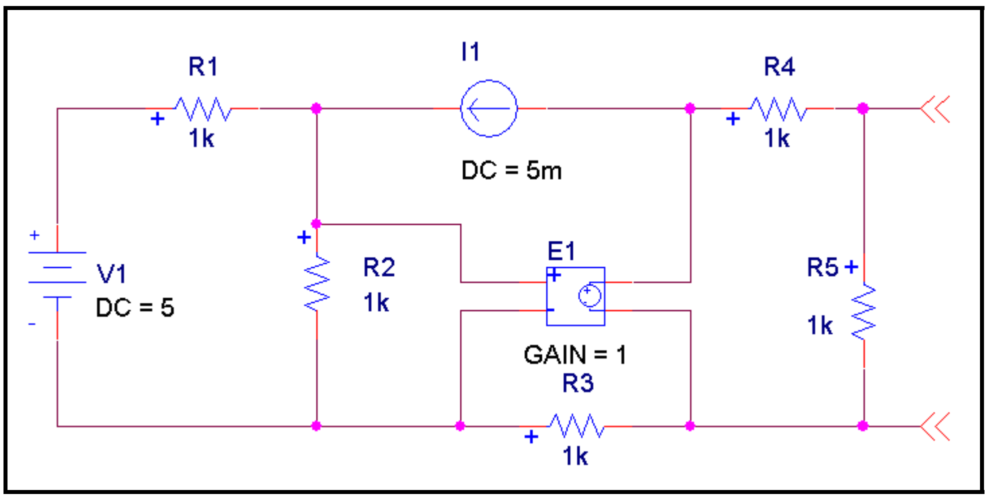
We can now find the Thevenin resistance by dividing the open circuit voltage by the short circuit current:

$$R_{th} = \frac{Voc}{Isc} = \frac{5.000\ \text{V}}{5.00\ \text{mA}} = 1000\ \Omega$$

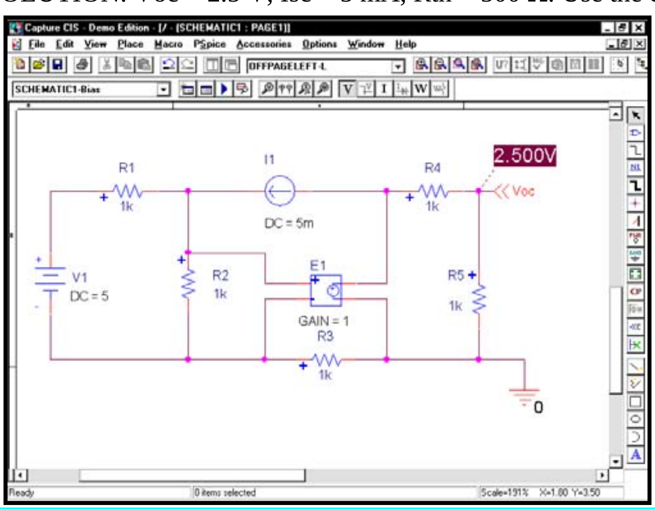
Our Thevenin and Norton equivalent circuits are shown below:

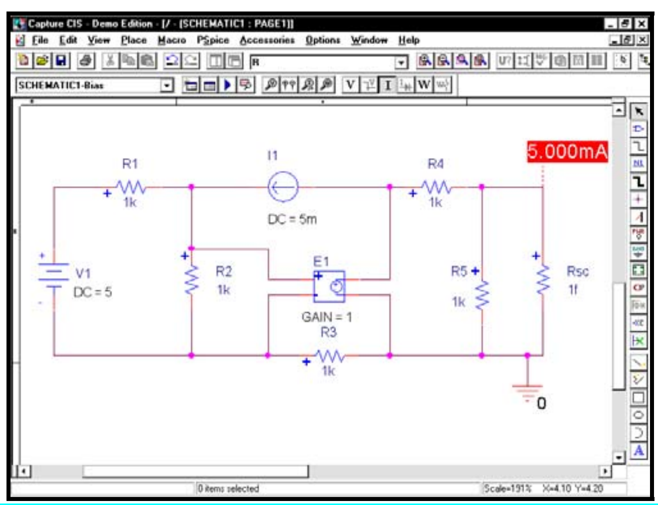


EXERCISE 3-6: Find the Thevenin and Norton equivalent circuits for the circuit below:



SOLUTION: Voc = 2.5 V, Isc = 5 mA, Rth = 500 Ω. Use the circuits below:

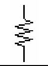
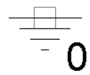
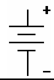
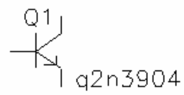


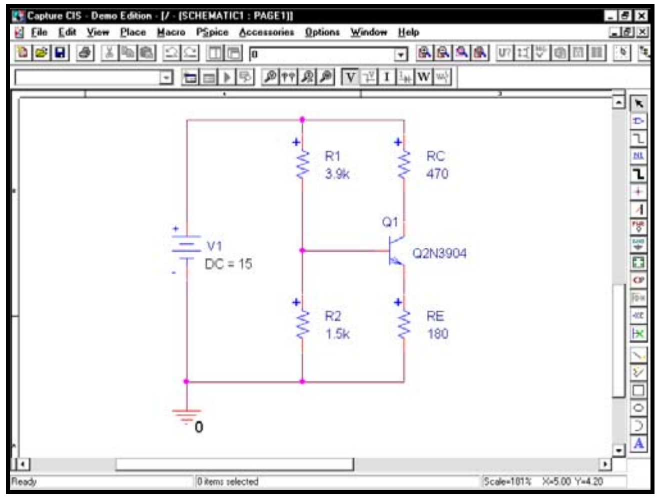


3.E. Transistor Bias Point Detail

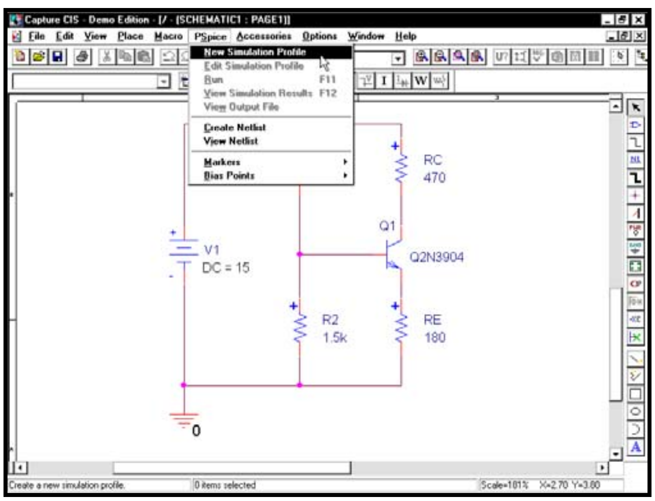
One of the first things you should do when you are simulating an amplifier circuit is to check the transistor operating point. If the transistor bias is incorrect, none of the other analyses will be valid. If another analysis does not make sense, check the operating point. When PSpice finds the bias point, it assumes that all capacitors are open circuits and that all inductors are short circuits.

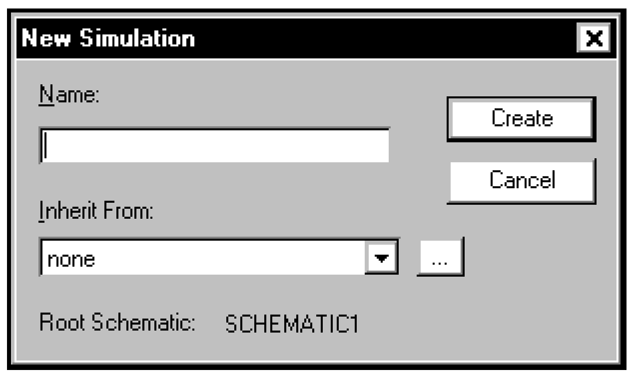
For a BJT, the Bias Point Detail gives the collector current, the collector-emitter voltage, and some small-signal parameters for the BJT at the bias point. For a jFET, the Bias Point Detail gives the drain current, the drain-source voltage, and some small-signal model parameters at the bias point. The results of the Bias Point Detail are contained in the output file. We will illustrate the Bias Point Detail analysis with the circuit below:

 **R** Resistor	 **0** Ground
 **VDC** DC voltage source	 **Q2N3904** NPN small-signal BJT

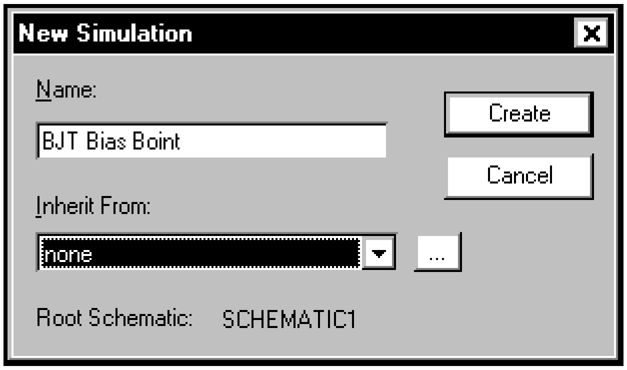


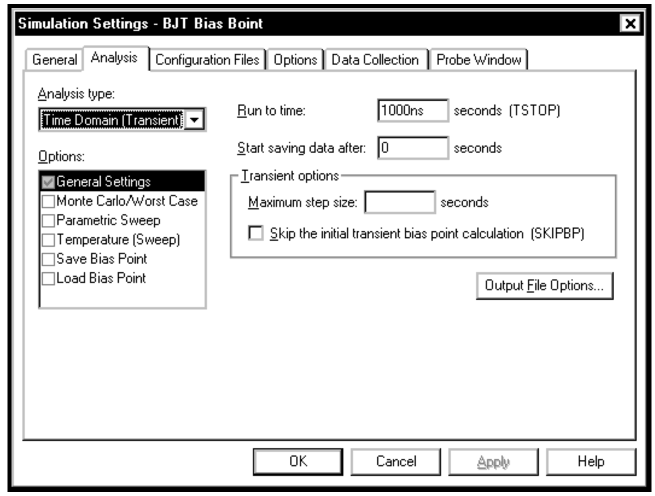
We must first set up the Bias Point simulation. Select **PSpice** and then **New Simulation Profile** from the menus:



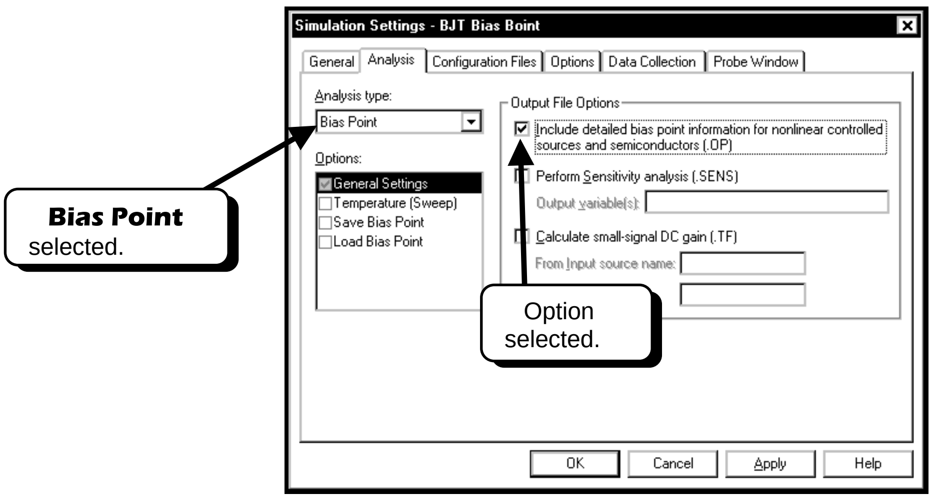


Specify a name for the new profile and click the **Create** button:

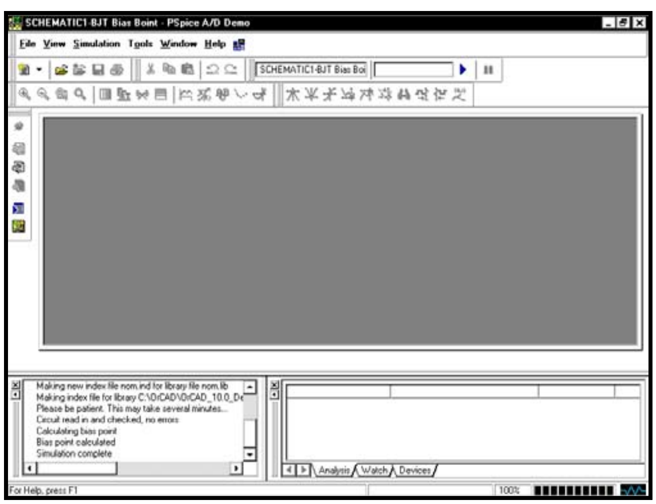




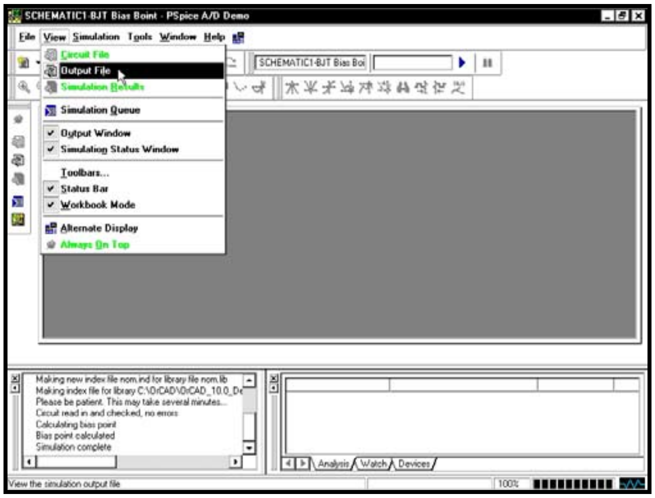
Specify the **Bias Point Analysis type** and select the option to include detailed bias point information for nonlinear controlled sources such as BJTs.

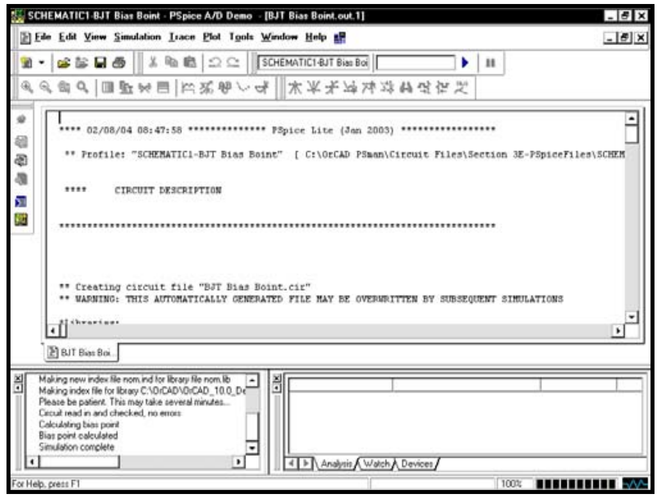


Click the **OK** button to accept the settings. We can now run the simulation. Select **PSpice** and then **Run** from the Capture menus or press the F11 key. PSpice will run:

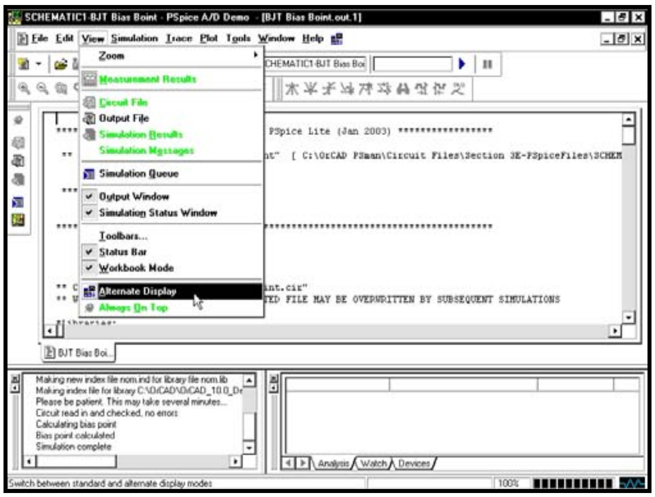


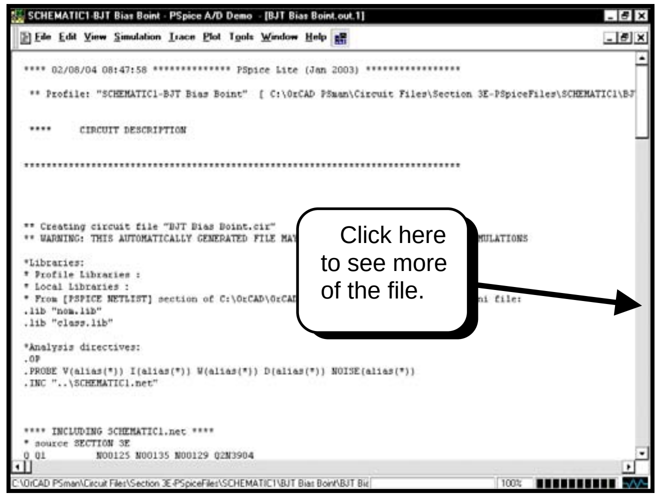
The results of the analysis are contained in the output file. Select **View** and then **Output File**:



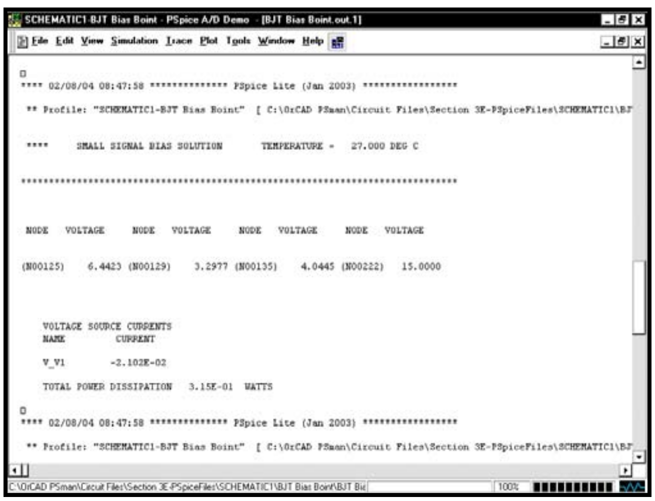


To view the text file in a full window, select **View** and then **Alternate Display**:





To see the results, click the ***LEFT*** mouse button on the vertical scroll bar to scroll the window through the remainder of the file. You will first see the node voltages:

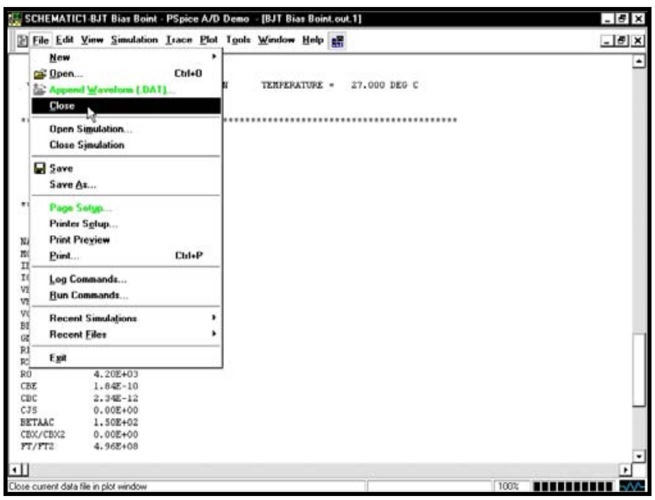


Since we did not place any BUBBLEs or name any wires, the node names are a bit cryptic to us. If we were interested in the node voltages, we could have named some of the wires or placed a few BUBBLEs at the nodes in question.

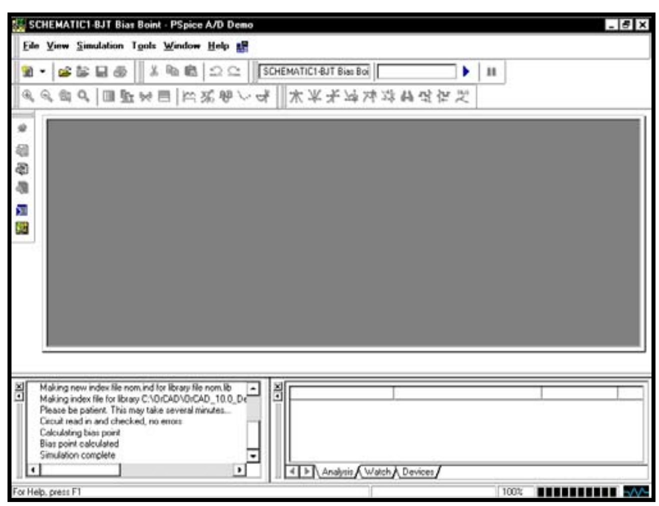
Further down in the file we see the **OPERATING POINT INFORMATION**:


```
**** BIPOLAR JUNCTION TRANSISTORS


NAME         Q_Q1
MODEL        Q2N3904
IB           1.13E-04
IC           1.82E-02
VBE          7.47E-01
VBC         -2.40E+00
VCE          3.14E+00
BETADC       1.61E+02
GM           5.82E-01
RPI          2.57E+02
RX           1.00E+01
RO           4.20E+03
CBE          1.84E-10
CBC          2.34E-12
CJS          0.00E+00
BETAAC       1.50E+02
CBX/CBX2     0.00E+00
FT/FT2       4.96E+08
```

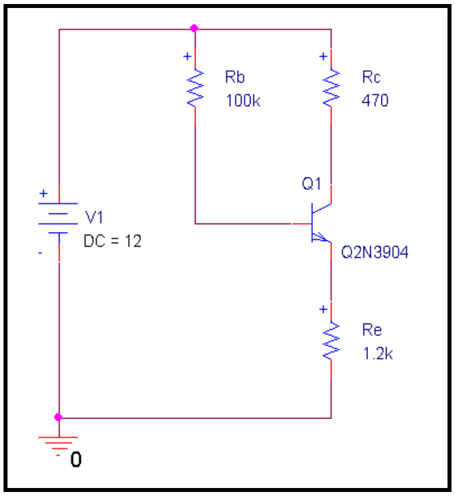
These results show several parameters for the BJT. In particular I_C is 18.2 mA and V_{CE} is 3.14 volts. Had there been more than one BJT in the circuit, the operating point information would be displayed for all BJTs. Similar information is shown for MOSFET, jFETs, and other three-terminal devices. To close the Text Editor program select **File** and then **Close** from the menus:



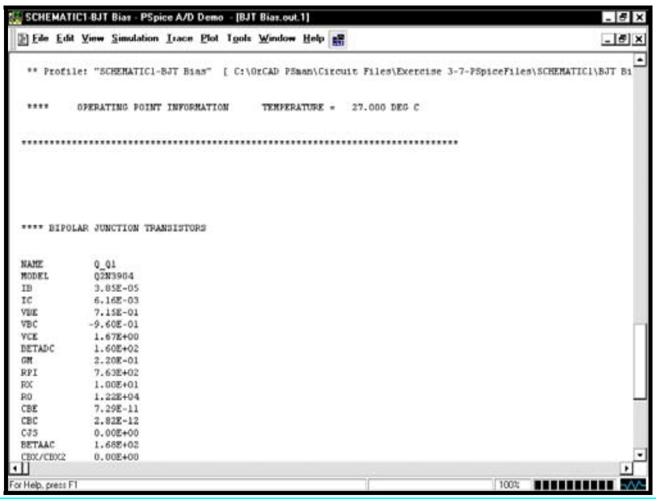
To switch back to the three-window display, select **View** and then **Alternate Display** from the menus:



EXERCISE 3-7: Find the bias point for the transistor circuit:



SOLUTION: The results of the Bias Point analysis are contained in the output file:



```
**** BIPOLAR JUNCTION TRANSISTORS


NAME         Q_Q1
MODEL        Q2N3904
IB           3.85E-05
IC           6.16E-03
VBE          7.15E-01
VBC         -9.60E-01
VCE          1.67E+00
BETADC       1.60E+02
GM           2.20E-01
RPI          7.63E+02
RX           1.00E+01
RO           1.22E+04
CBE          7.29E-11
CBC          2.82E-12
CJS          0.00E+00
BETAAC       1.68E+02
CBX/CBX2     0.00E+00
```

3.F. Summary

- The DC Nodal Analysis finds the DC voltage at every node in the circuit. The voltages are relative to ground.
- The results are given in the output file. Use **Examine Output** from the **PSpice** menu to view the results.
- All capacitors are replaced by open circuits.
- All inductors are replaced by short circuits.
- All AC and time-varying sources are set to zero (IAC, VAC, Vsin, Isin, Vpulse, etc.).
- Node voltages and element currents can be displayed on the schematic using the V and I buttons V I.