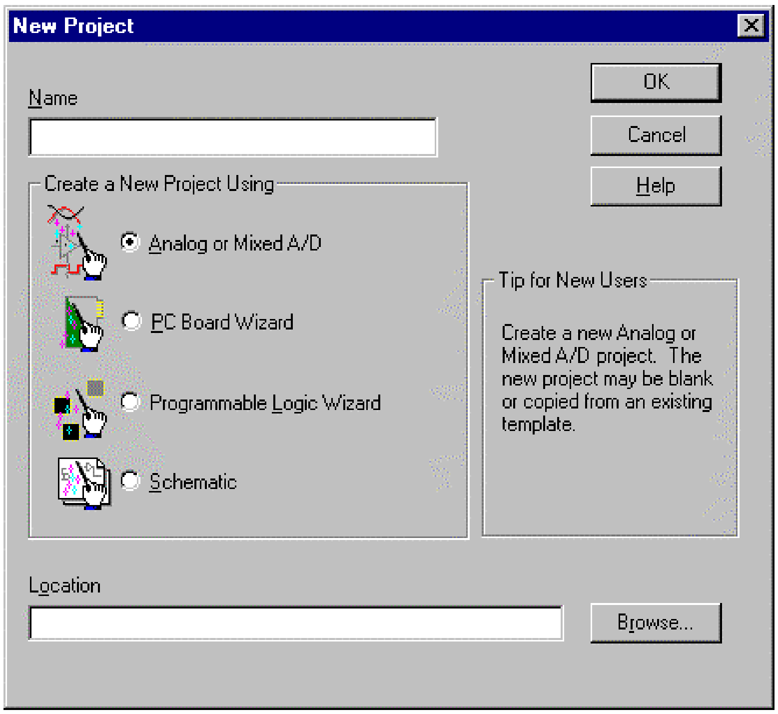
Getting Started with
Orcad Lite, Release 9.2

Professor Robert Hofinger
Purdue University - Columbus

You start a new project (program) by going to the File menu in the upper left corner, then New, and then Project.

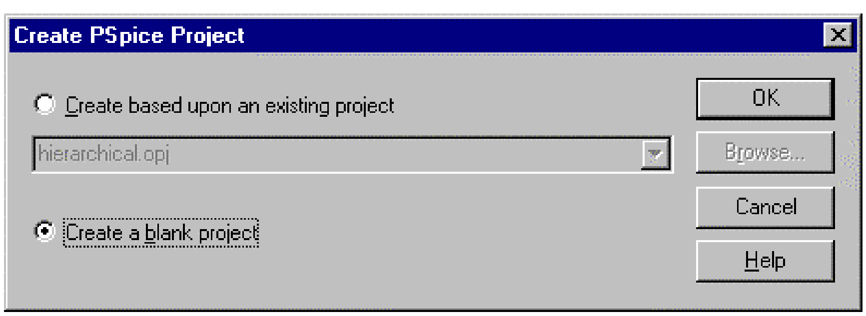
The following screen will appear:



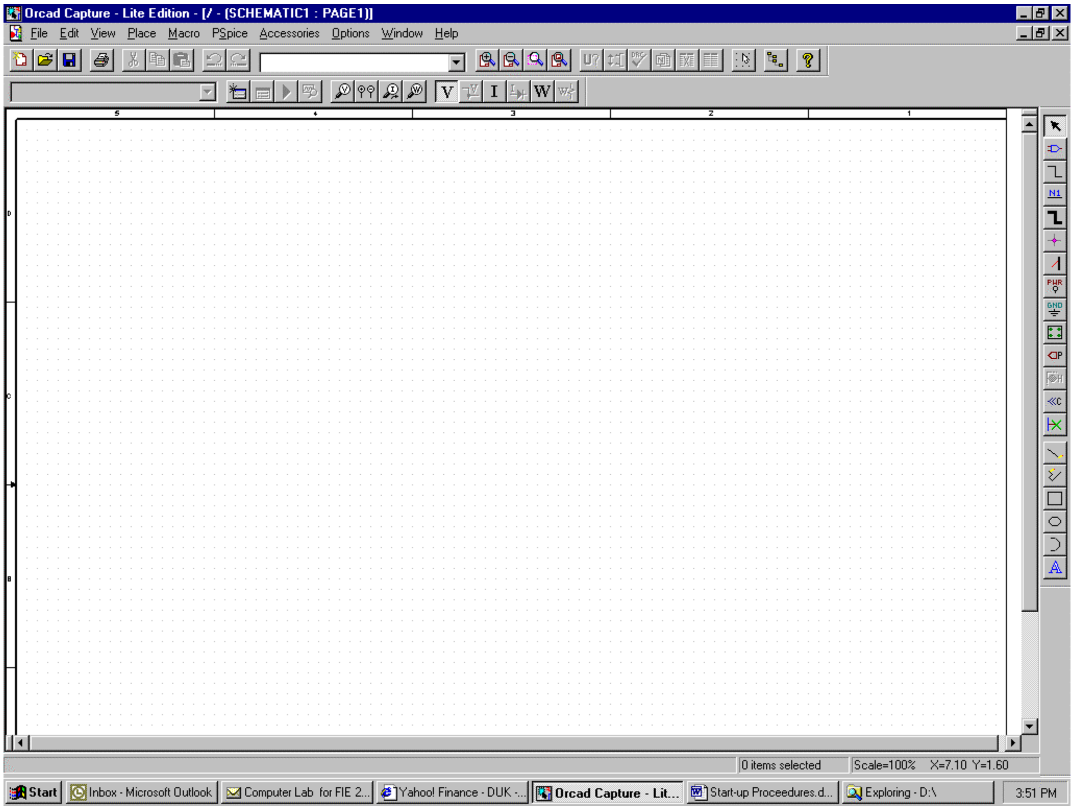
Be sure that the "Button" for the Analog or Mixed A/D selection is chosen. See above figure.

You need to fill in the top line Name with a file name and then the bottom line Location with the path name. This is the directory where you will be storing your "Project".

Now the following screen will appear. Since you are starting a new project, change the button settings as shown below. Activate the Create a blank project button.

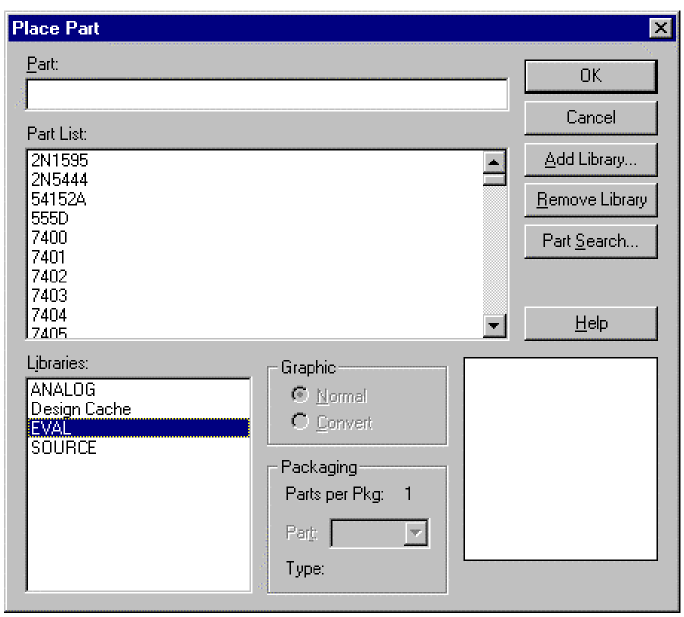


Now you should come up to a blank schematic entry screen.



You can now start adding components and symbols to your schematic, by using the Place, Part menu sequence, or the special icon (the uppermost one) on the right hand toolbar.

The following screen will appear.

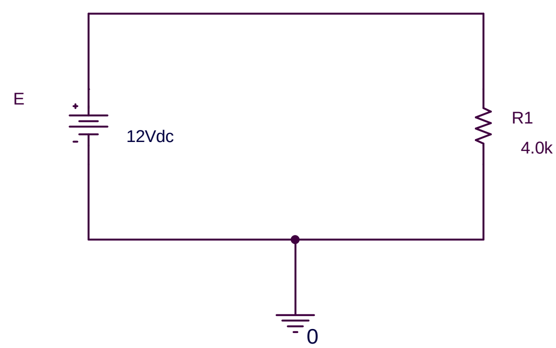


If all of the Libraries shown above do not appear on your screen, and they probably won't, go to Add Library. There you will find a list of available libraries. For this beginning tutorial, you will need the analog.olb, the eval.olb, and the source.olb libraries. Add them now.

Note: that only parts from the Libraries that are highlighted are shown in the Parts List window.

When you have found the required part, either by entering its name in the Part window or by highlighting its name in the Part List window, left-click OK and drag the part onto the schematic. Left-click to place the part on the schematic. You can continue left-clicking to place multiple copies of the same part or right click to end this selection.

Practice now by entering the schematic shown below. Change the default values and orientations as shown.



To change a value, or a reference, highlight the appropriate value (left-click) and then double left-clicking. When you have added the resistor (R), and the power supply (VDC) symbols, enter the ground symbol labeled "0", which is located in the "..../PSpice/source.olb" library, or any other ground symbol. If you use a ground symbol other than the "0" ground symbol, you need to modify it. Highlight the symbol and then double left-click it. A parts list page will appear. Change the Name of the part from GND to "0" (the number). Recall that every circuit has to have a node "0". Left-click Apply and the close the page.

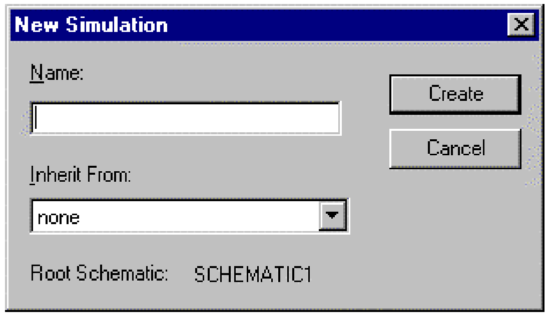
You can rotate parts by highlighting the part (left-click) and then pressing the "r" key on the keyboard. Now its time to add the connecting wires.

Again use the Place Wire menu, or the icon on the right hand side toolbar. Connecting wires requires that you drag the "cross hair" over the end of the part and left-click. This "solders" one end of the wire. Drag the wire to another connecting point and left-click again. You have now "soldered" the other end.

You are now ready to simulate your circuit.

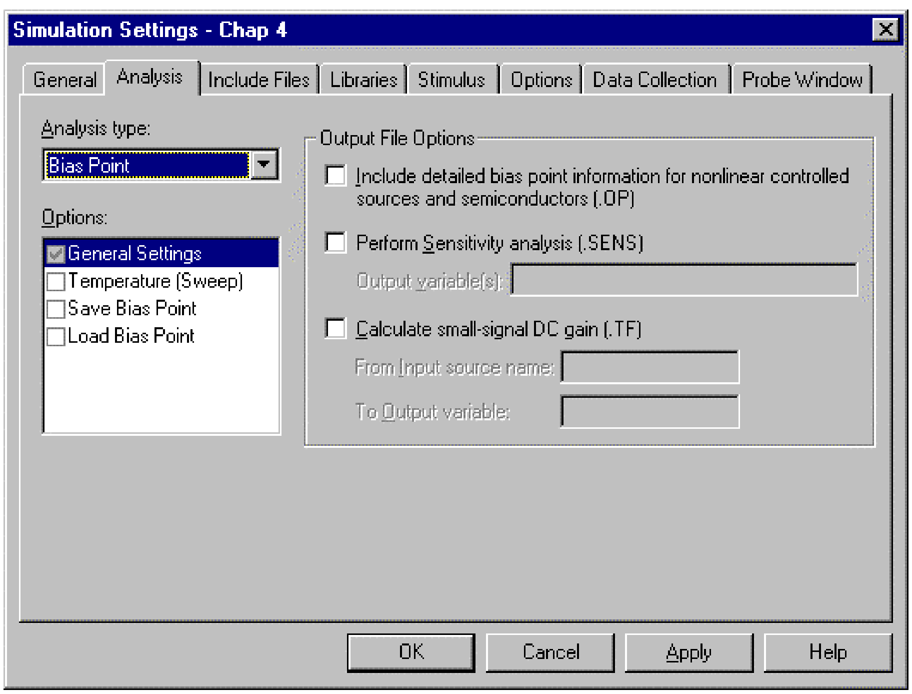
DC Bias Simulation

To start the simulation process, open the PSpice menu. The first choice available is New Simulation Profile. Left-click on it and the following window will appear.



Give the New Simulation a Name. For now use Chap 4.

Left-click Create and the next screen will appear

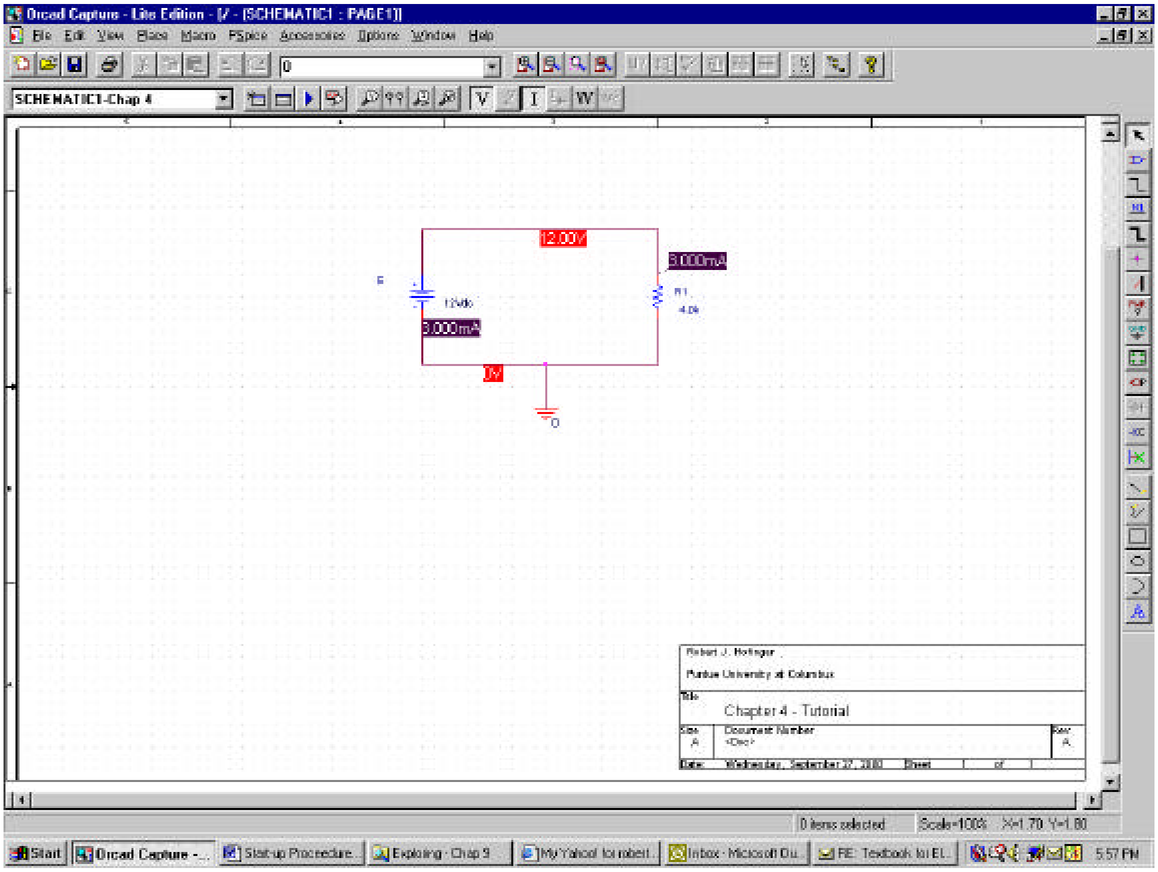


Select the Bias Point setting in the Analysis type window, and then left-click OK

Now you are ready to Run a simulation.

Go to the PSpice menu and select Run.

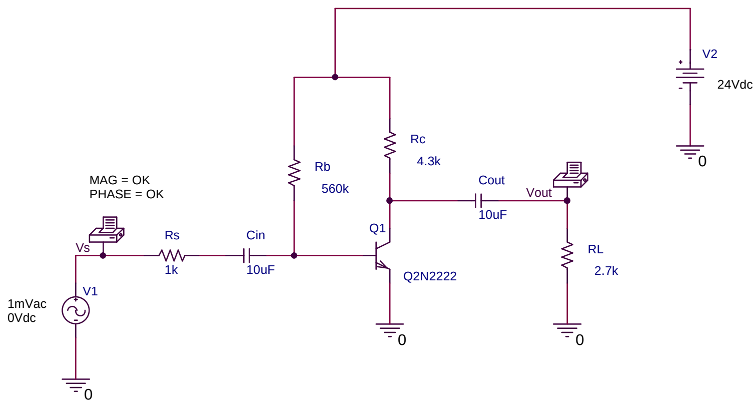
The simulation window will appear. When the simulation has completed, close this window and the schematic will appear. When the V, I, and W tool buttons are activated, the voltage, the current , and/or the power dissipated in that component results will be attached. The tool buttons along side the V, I, and W buttons allows you to alternately toggle a highlighted value OFF and ON



AC Sweep Analysis

In this example you will be looking at the ac signals and using the results to determine the ac gain of the circuit.

You will use the circuit shown in the following schematic.



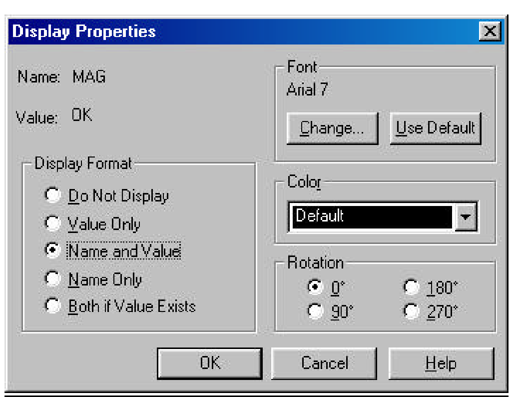
For the voltage source V1, use the VAC part. The VSIN part is used for the Transient Analysis simulations.

Note the use of a new symbol:  This is listed as VPRINT1/SPECIAL in the Part menu. This symbol is used to tell PSpice that you wish to include some characteristics of this node in your output file.

Activate the symbol (by left clicking on it) and then followed by Edit and selecting Properties. The properties screen will appear.

Under the column designated AC enter OK. Under the column designated MAG enter OK. Continue entering OK under any other characteristic that you wish to examine and record.. If you wish to display the name and/or the value of the characteristic as shown on the input node Vs, press the Display… button.

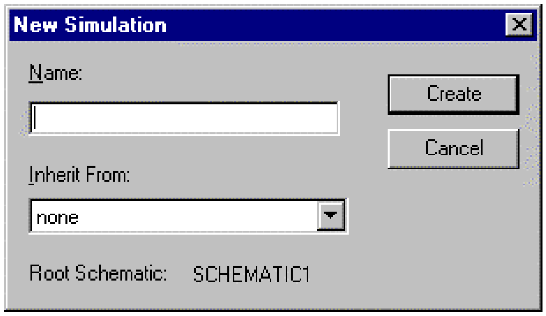
The following screen will appear:



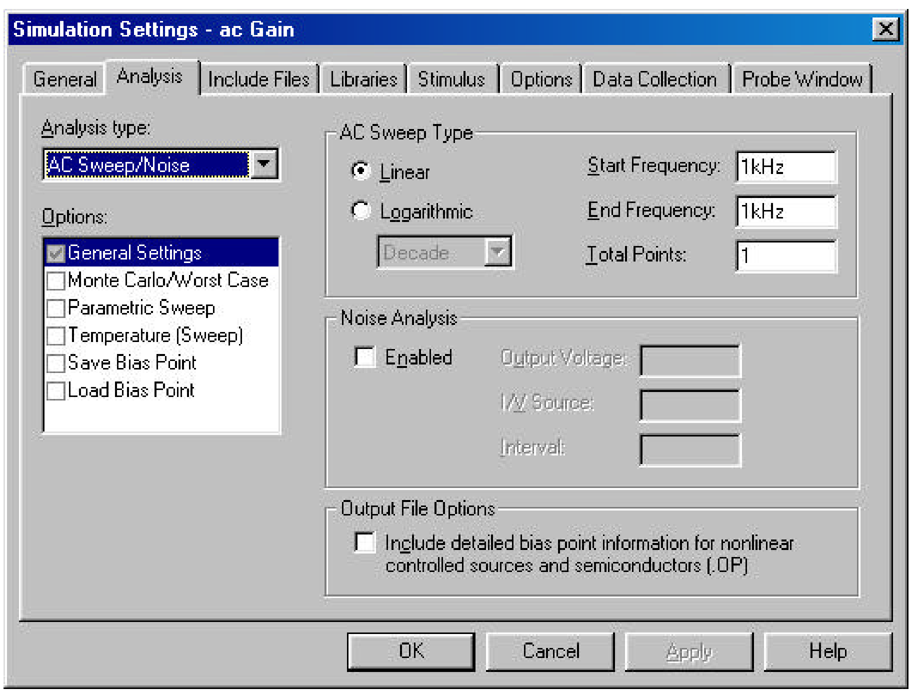
Here you can make your choices for displaying the symbol properties.

P.S. You don't have to enter OK as I have. Entering anything will activate the property.

To start the simulation process, open the PSpice menu. The first choice available is New Simulation Profile. Left-click on it and the following window will appear.



Give the New Simulation a Name. For now use   "ac Gain". Left-click Create and the next screen will appear



Choose the AC Sweep/Noise under the Analysis type option. Since in this example we will be looking at only one frequency, (i.e. 1kHz) set the AC Sweep Type as shown above.

Run the simulation and open the output file (under PSpice, and View Output File). At the end of the output file you will find the AC Analysis results, like the section shown below.

```
****     AC ANALYSIS                    TEMPERATURE =   27.000 DEG C


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


 FREQ       VM(VOUT)    VP(VOUT)
  1.000E+03   7.363E-02  -1.794E+02


**** 03/11/01 21:42:29 ************** PSpice Lite (Mar 2000) *****************


 ** Profile: "SCHEMATIC1-ac Gain"  [ A:\hmwrk14\chap_10 prob_17-schematic1-ac gain.sim ]


 ****     AC ANALYSIS                    TEMPERATURE =   27.000 DEG C


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


 FREQ       VM(VS)      VP(VS)
  1.000E+03   1.000E-03   0.000E+00
```

In this example, the frequency (FREQ), magnitude (VM) and phase (VP) are displayed for each signal node.

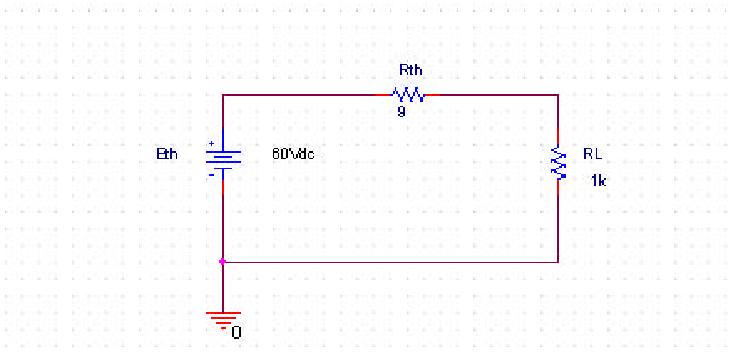
Recall that you can modify, add to or delete anything from the output file, just as you would with a text file (which it is).

Parametric DC Sweep

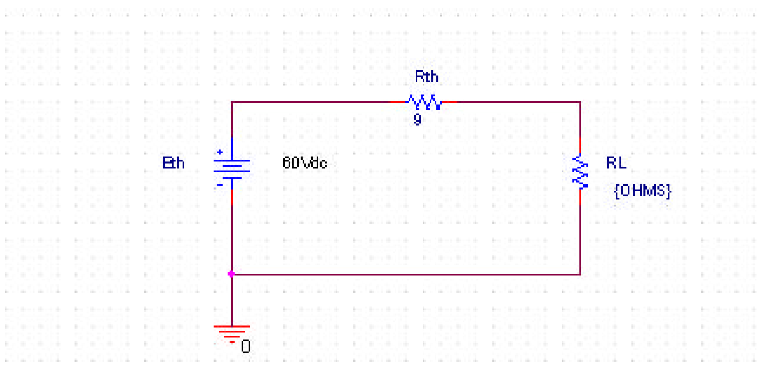
As an example of performing a parametric DC sweep, that is, varying the parameter or value of a component, lets find the power dissipated in a resistor as the value of that resistor varies and plot the results.

You will recognize that this is the Maximum Power Transfer curve.

Starting with the basic schematic:



In order to do a sweep of the values of resistor RL, you need to change its value from a fixed number to a variable text. Say we call it OHMS



Note: the text OHMS enclosed in curly brackets { }. OrCad Lite will treat this as a parameter.

Now go to the Place Part menu, under the SPECIAL library, select a "part" called PARAM. If this library is not on your list, you will have to add it using the Add Library... button. Place it anywhere near your schematic. Double left-click on it, bringing up the [Property Editor]. Left-click on the New Column button give the variable a name, in this case our variable name OHMS, and a value (one that is reasonable for your design). This value will be used as the default condition if you choose to do another type of simulation on this design. Use the value 2. Apply it and then highlight this new column. Left-click Display... and then activate the button labeled Name and Value. When you close out the [Property Editor] your schematic should look like this:

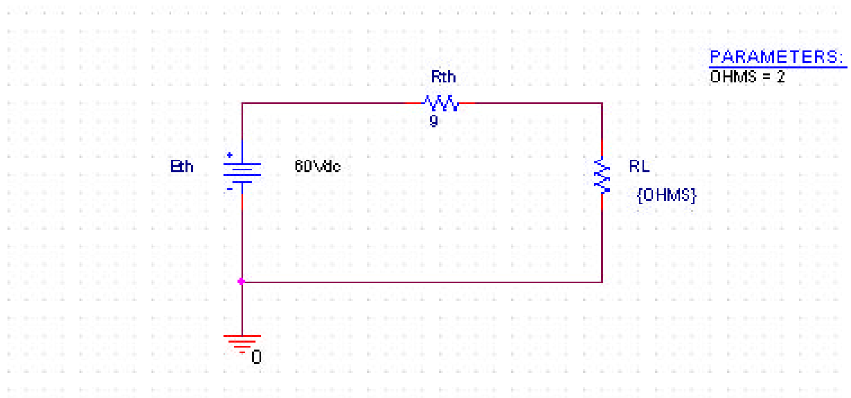
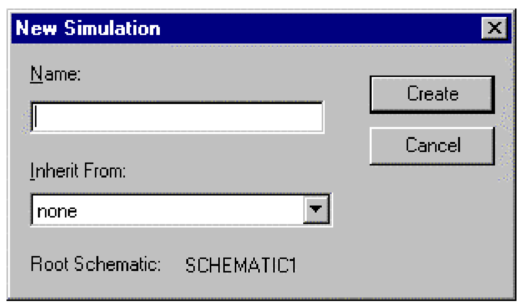


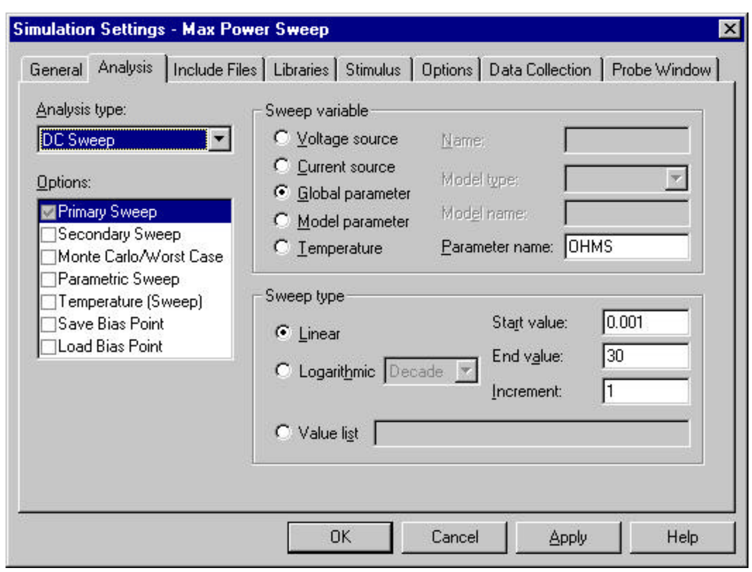
FIG. 9.120

To start the simulation process, open the PSpice menu. The first choice available is New Simulation Profile. Left-click on it and the following window will appear.



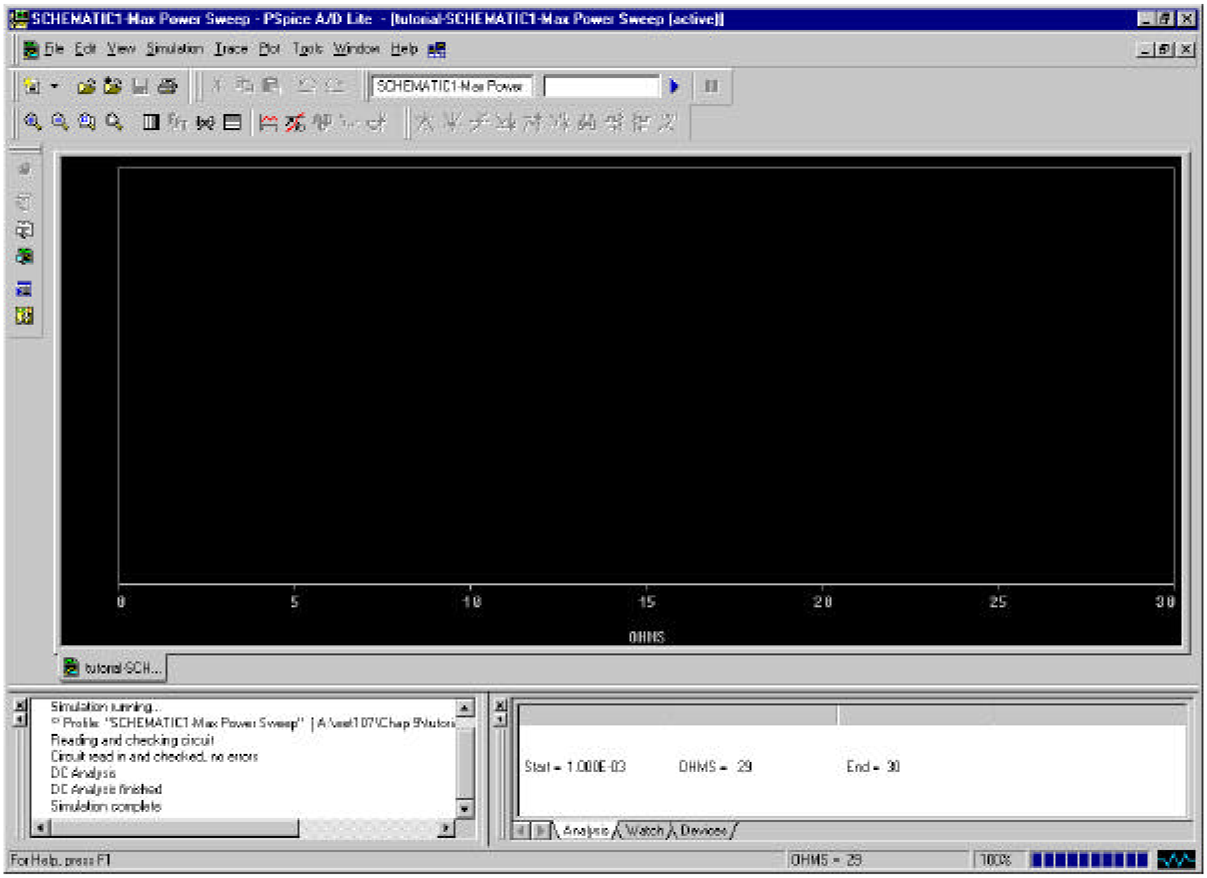
Give the New Simulation a Name. For now, use Max Power Sweep

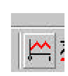
Left-click Create and the next screen will appear



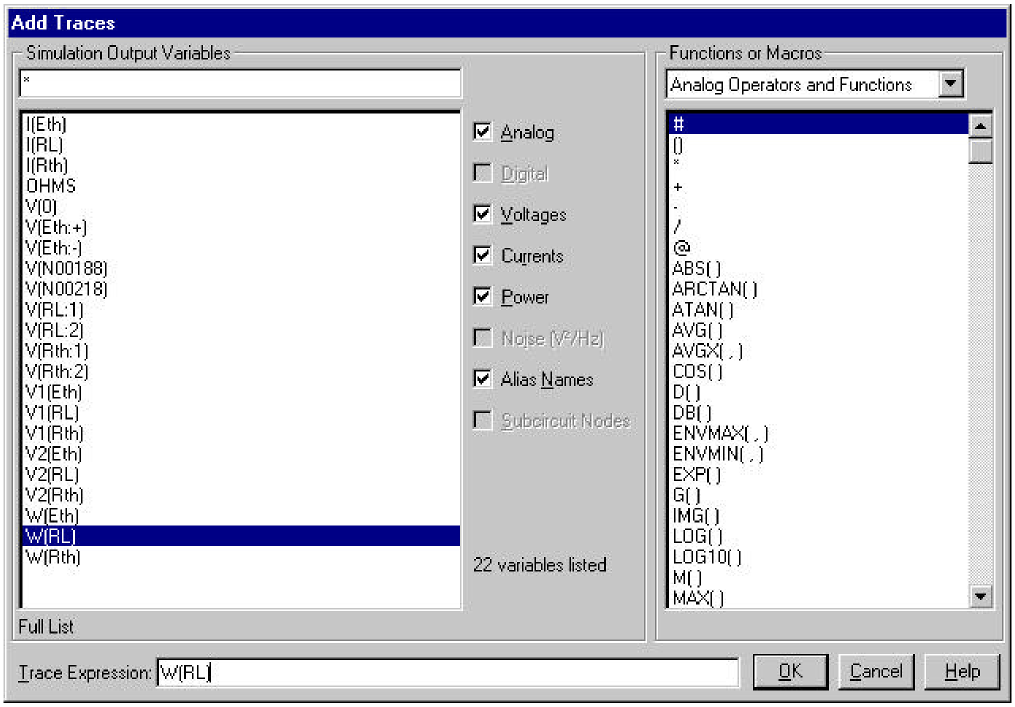
Activate the Global parameter button and add the name of your parameter that you are sweeping. In this case it is OHMS. The sweep type will be Linear. Since you will want to sweep the resistor value from 0 Ω's to 30 Ω's in steps of 1 Ω you will enter these values in the appropriate places. Note that you could not enter a Start value of 0 Ω's since this would result in a divide by zero, which is illegal. If you try it you will get an error message. Go try it, run PSpice and see the results. So you can use 0.001 Ω, a small number close enough to 0.

If everything was entered correctly, the simulation will run without any errors, and the following screen will appear:

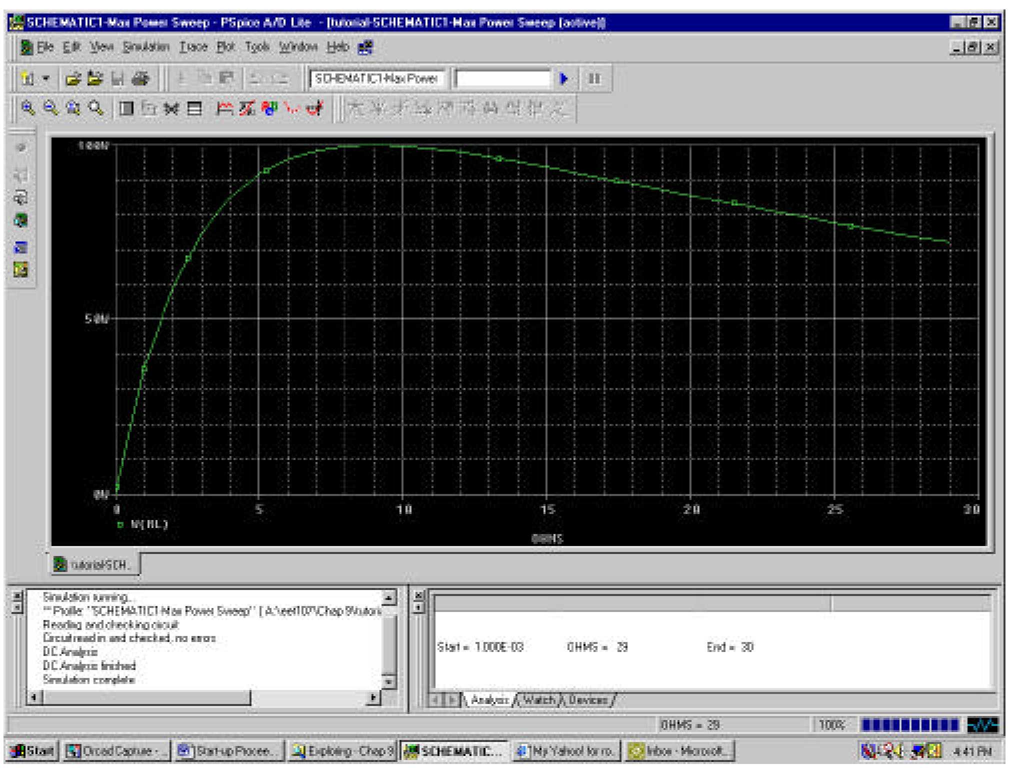


Add a trace by left-clicking on the Trace menu or on the toolbar button: 

This brings up the Add Traces menu. Since you are looking for the power dissipated in a resistor as a function of its value, highlight the W(RL).



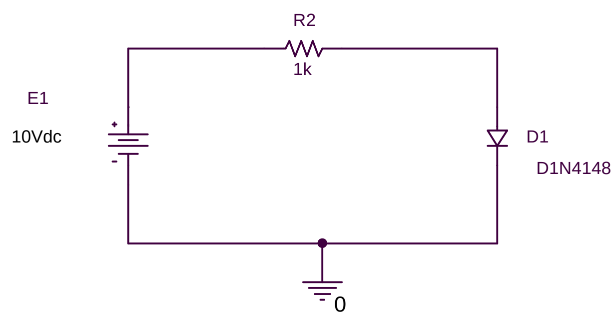
Left-clicking OK brings up the plot on the power vs. OHMS graph.



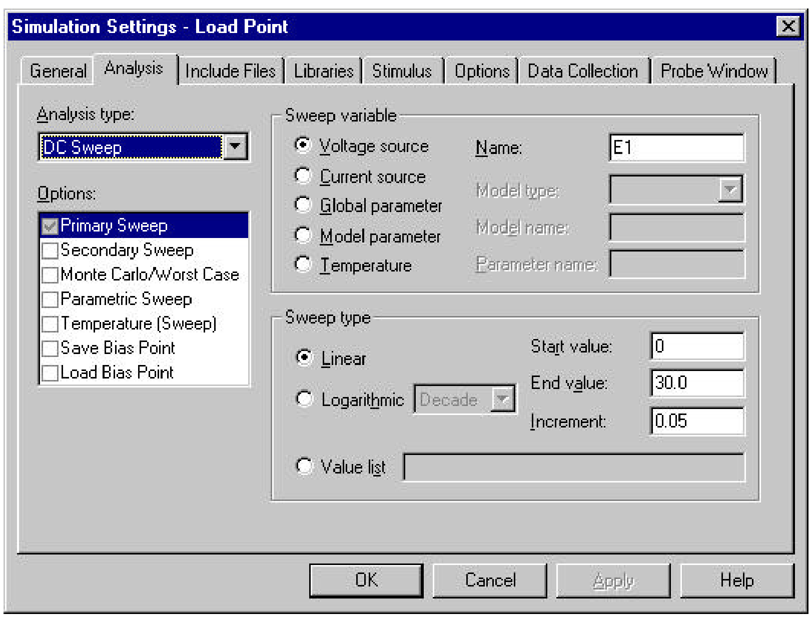
Series Diode Configuration

In this segment you will plot out the current vs. voltage characteristics of the D1N4148 diode.

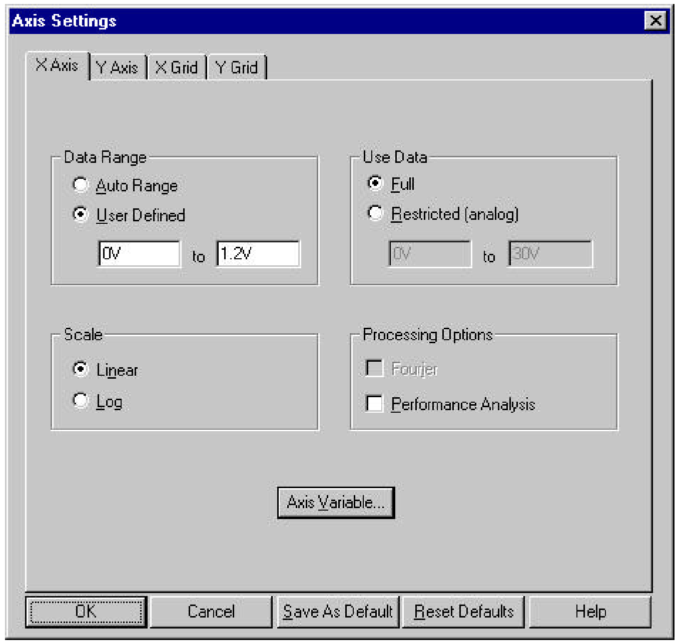
The diode network is shown below.



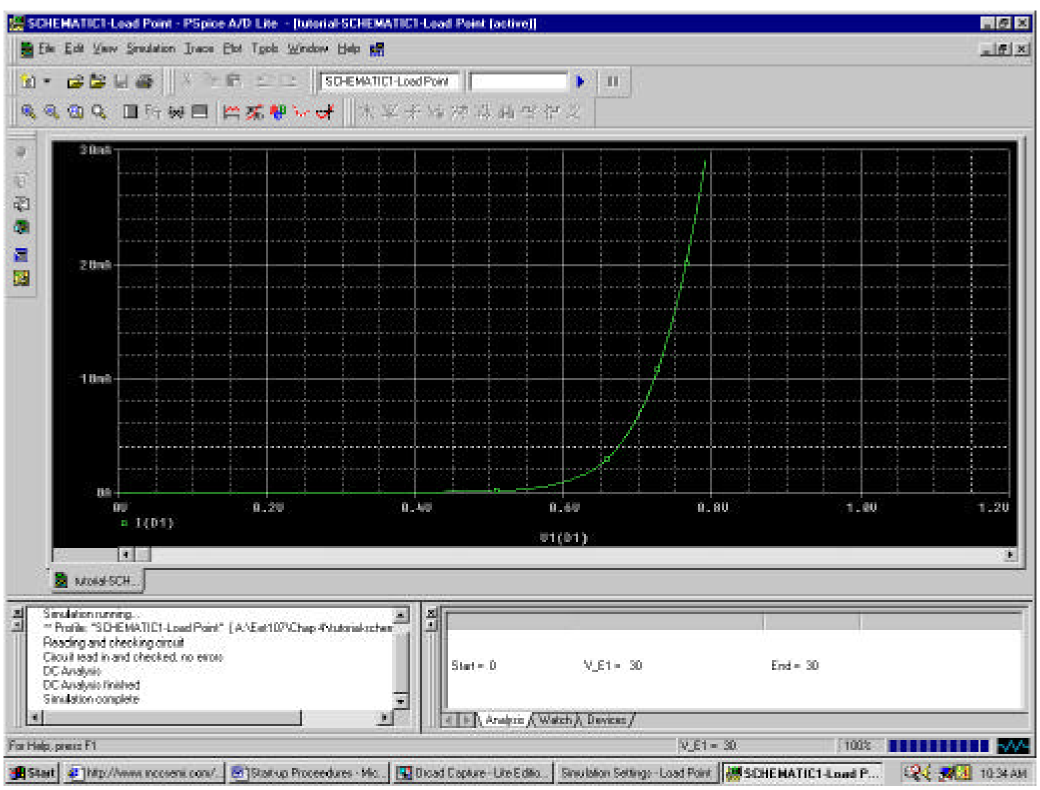
The simulation profile is set up for a DC Sweep as shown below.



Click OK to close this window and run PSpice. When the circuit is finished simulating, the Probe window will appear. At that point, you will want to change the x-axis from V_E1 to V1(D1). Pull down the Plot menu and click on the Axis Settings… option. This will bring up the following menu.

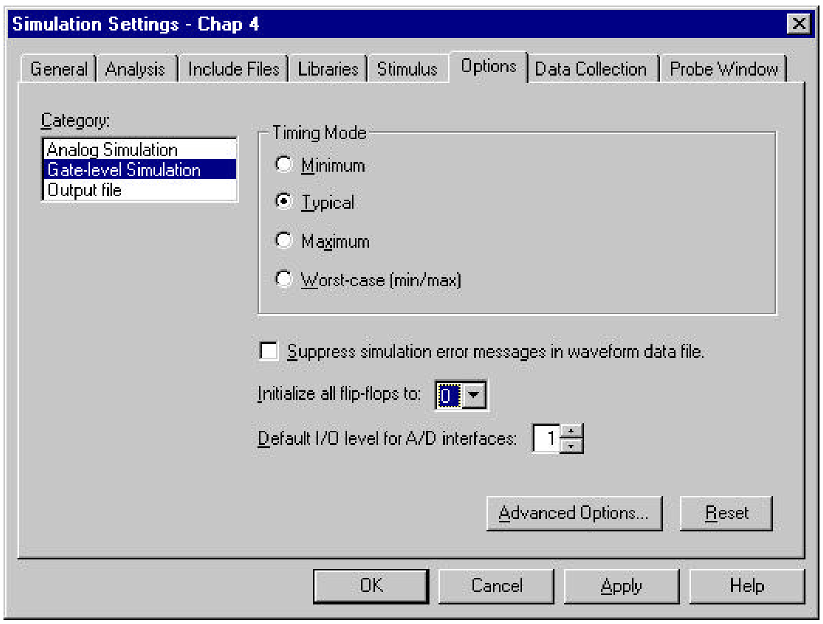


Click on the Axis Variable button and then choose V1(D1). After you have chosen the x-axis the above window will reappear. Now set the Data Range to User Defined and adjust the settings to what you prefer. In the above menu example 0V to 1.2V was selected. When you close this window, you can now select your y-axis variable. Use the I(D1) selection to plot the I_D vs V_D diode characteristics.



Digital Simulations

When simulating digital circuits that contain flip-flops or their derivatives, or circuits that contain both analog components and digital flip-flops or their derivatives, you need to initialize the data storage devices. In OrCad Lite, you do this by accessing the Options tab under the Simulation Settings menu and then highlighting the Gate-level Simulation in the Category window. An example is shown below.

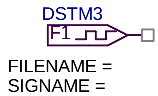


On the line labeled "Initialize all flip-flops to:" change the options from " X" (logic don't care) to either a "0" (logic Zero) or a "1" (logic ONE).

Use of Bus Wires

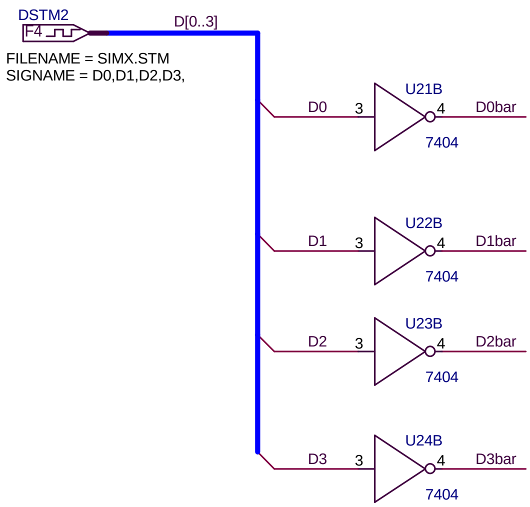
A scalar wire, the one that we have been using all along, can carry only a single signal. A bus wire can carry multiple signals. This can cut down on the number of wires on your schematic, thereby making the schematic easier to read.

We have already used a FileStim1 source to inject a single signal into our circuit.



Along with this input source we have to create a FileStim text file.

Now looking at the example block diagram below



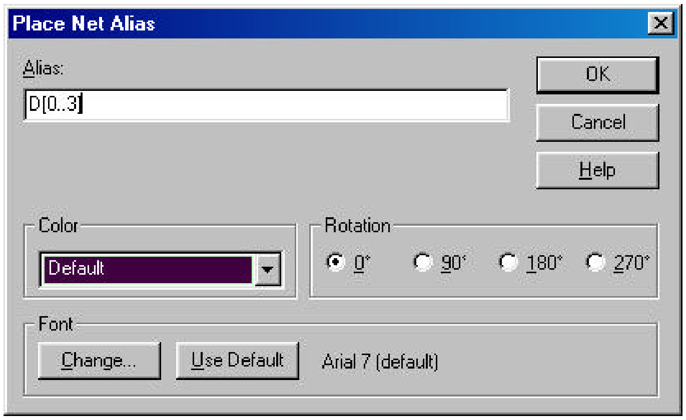
You can see that we have multiple signals coming from a single wire. This is the bus wire.

Along with that, I have used a multiple FileStim input source, in this case a FileStim4 source. The four means that it can inject 4 signals simultaneously. There are also 2 input, 8 input, 16 input, and 32 input versions in the OrCad Lite source library.

The FileStim text file is created and labeled on the FILENAME = line in the same way. But now on the SIGNAME = multiple signal name can be entered. As shown in the example above, the letter designation can be anything but the suffix following it must be a number, and <u>IN SEQUENCE</u>. Typically it is the numerical sequence of signals available.

At the signal exit points, notice the use of "BUS ENTRY" lines. A bus entry is used to tie a signal to a bus. The advantage of using bus entries instead of wires is that two bus entries can be connected at the same point on a bus without connecting the signals. If two wires are run directly to a bus at the same location, the signals are connected.

The nodes at the gates are marked with the appropriate Net Alias. Now one more thing needs to be done. The bus wire must be labeled with the names of the signals. In this case, D0 to D3. Using the Place menu and the Net Alias selection, create the label of the bus wire as shown below.



Place your Net Alias on the bus wire and your all set.