Chapter 2 Getting Started

2.1 Circuit Description

The first step in the process of simulating the behavior of a circuit is to describe the components and connections of the circuit in a form that the PSpice software can understand. PSpice uses a file called a netlist as the input to be analyzed and simulated. A netlist consists of a set of single text lines that contain information about the component type, the nodes to which the component is connected, the size of the component, and possibly other relevant component values. Netlists can be entered by hand; however, the OrCAD version of PSpice has a schematic Capture program associated with it that allows the user to enter circuit descriptions in graphics form and have them subsequently converted to a netlist for analysis with PSpice. We will first show how a netlist may be generated using OrCAD Capture after which we will examine the process of generating a netlist by hand.

Example 1: Use the OrCAD Capture program to produce a netlist for the simple resistive circuit shown in Figure 2.

Solution 1: Start the OrCAD Capture program by clicking on the Start button, selecting All Programs, PSpice Student, Capture Student (symbolized in this book by Start>All Programs> PSpice Student>Capture Student). Click on File>New>Project in the OrCAD Capture window which will bring up the display shown in Figure 3.
Type in a name for the project in the area labeled Name. This name will be used to identify the project in subsequent simulation windows and data files that are generated by OrCAD or PSpice. Next, click on the radio button next to Analog or Mixed A/D causing the OrCAD Capture program to prepare a simulation file that can be used by PSpice. We use the name Resistor1 for this example. Finally, specify the location where we want the files associated with this simulation project to be placed. We may either type in the full path specification starting with the drive letter specification through all subdirectory names, or we may use the Browse feature of Windows® to specify the location. Once all of the parameters in this window have been filled in, click on the OK button. We have specified a location in the subdirectory C:\MyDocuments\PSPICE Circuits\circuits after having created it inside the C:\MyDocuments\PSPICE Circuits directory using the button labeled Create Dir... from the Select Directory window.
The Capture software will then bring up the window shown in Figure 4 asking us to choose a template around which this Capture process will function. Click on the radio button labeled Create a Blank Project, and complete the selection by choosing OK. The next screen that appears is shown in Figure 5 and contains the window in which we enter the schematic diagram along with a window that contains information about the files that are part of this Capture project. The schematic window contains a drawing grid with parts set by default to snap to the grid.

To begin entering circuit components on the schematic, click on the button, second from the top right-hand side, which contains the icon that looks like \(\begin{align*} &\end{align*}\). A window similar to the one in Figure 6 will appear asking us to choose the part that we want to place in the drawing area. Depending on the installation of OrCAD and whether or not anyone else has already used the software, there may or may not be a string of part names in the Part List area or a set of parts library names in the Libraries area. Libraries may be added to the list by clicking on the button labeled Add Library and choosing any libraries appropriate to the design. Standard libraries are located in the subdirectory c:\Program Files\OrCAD_Demo\Capture\Library\Pspice and have the file type .olb. We will use parts from the analog.olb and source.olb library files, so be sure to add these libraries at the beginning. Click on ANALOG under the Libraries area. A list of parts contained in the analog library will appear. Click on the part labeled R in the Part List. A symbol for a resistor will appear in the lower right side of the window. The default value for the resistor is set to 1 k\(\Omega\), while the default name is set to R\(\text{?}\), where the question mark is replaced by
Figure 5 Schematic entry window.

Figure 6 Parts placement dialog box.
a sequence number each time an instance of the resistor is placed in the schematic window. Numbering starts from 1 to n and may be changed after a resistor is placed in the schematic.

Click on the OK button in the dialog box. The dialog disappears, and the cursor has the diagram of a resistor tracking along its movement path. By clicking on the left mouse button you will drop an instance of the resistor onto the grid of the schematic window.

![Capture CIS - Demo Edition](image)

**Figure 7** Right-click to obtain an individual part manipulation menu.

Place a resistor in a position just above the center of the schematic window. Next, place another resistor lower and to the right of the first resistor. Hit the Esc key to end the parts placement operation. With the last resistor still selected following its placement, click Ctrl-R on the keyboard. This causes the selected part, the second resistor in this case, to rotate 90° in the schematic. A part may also be similarly manipulated by right-clicking on a part that is selected and then bringing up a menu of operations as shown in Figure 7. Rotate the R2 resistor so that it is placed vertically, and then double-click on the label indicating the default value of 1k for the resistor. A Display Properties dialog box like the one shown in Figure 8 will appear. (We may have to click Esc to de-select the entire resistor part and then click on just the characters “1k” to
bring up the properties for the resistor value.) Change the resistor size in the area labeled Value to be 9.4K as in the example of Figure 2. We may also change the name of the resistor by double-clicking on the R2 next to the resistor part. A Display Properties dialog box similar to the one in Figure 8 will be shown, the difference being that this dialog allows us to change the Part Reference, which is the name displayed on the schematic. Other parts such as voltage sources, capacitors, inductors, etc., will have adjustable properties similar to those shown in Figure 8.

When changing a Part Reference, any valid printing character may be placed in the Value field. To change the size value assigned to a part, numbers placed in the Value field must either be integers or real numbers. Integers can be either positive or negative, e.g., 14, or -642. Real numbers can be numbers containing decimal points, (e.g., 9.807), numbers containing integer exponents (e.g., 1.602E-19), or numbers containing a symbolic exponent (e.g., 0.2998G). In the latter example, the G symbolizes an exponent of $10^9$. A list of acceptable symbolic scale factors for PSpice is shown in Table 1.
Table 1 PSpice Scale Factors

<table>
<thead>
<tr>
<th>Symbolic Suffix*</th>
<th>Mnemonic</th>
<th>Exponential Form</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>femto</td>
<td>1E−15</td>
<td>10^{15}</td>
</tr>
<tr>
<td>P</td>
<td>pico</td>
<td>1E−12</td>
<td>10^{12}</td>
</tr>
<tr>
<td>N</td>
<td>nano</td>
<td>1E−9</td>
<td>10^{9}</td>
</tr>
<tr>
<td>U</td>
<td>micro</td>
<td>1E−6</td>
<td>10^{6}</td>
</tr>
<tr>
<td>M</td>
<td>milli</td>
<td>1E−3</td>
<td>10^{3}</td>
</tr>
<tr>
<td>K</td>
<td>Kilo</td>
<td>1E3</td>
<td>10^{3}</td>
</tr>
<tr>
<td>MEG</td>
<td>mega</td>
<td>1E6</td>
<td>10^{6}</td>
</tr>
<tr>
<td>G</td>
<td>giga</td>
<td>1E9</td>
<td>10^{9}</td>
</tr>
<tr>
<td>T</td>
<td>tera</td>
<td>1E12</td>
<td>10^{12}</td>
</tr>
</tbody>
</table>

* Note that in PSpice, a suffix may be either a capital or lower-case letter.

Change the value of the resistor R1 to be 4.7 KΩ, matching the value in Figure 2. Now place the DC power source in the schematic. Click on the place parts icon and choose the SOURCE library in the lower left area. Then double-click on VDC in the Part List area, and place the DC voltage source part in the left part of the schematic. Change the value of the voltage source to be 10, in the same manner that you changed the values of the resistors.

Now that we have selected all of the parts, placed them in the schematic, and set their Values and Reference Names, we are ready to wire the components together to form a circuit. From the buttons menu on the right side of the Capture screen, click on the Place Wire button and the drawing cursor will change to a cross-hair and allow us to stretch wires between component nodes. Start by clicking the cross-hair on the top node of the power source symbol. Move the cross-hair to the left node of the resistor symbol, and click again. Capture will stretch a wire between the two nodes, squaring off a 90° angle as the wire turns corners. Wire all three parts together so that the schematic looks something like the one in Figure 9.
The wire drawing capability is quite flexible, allowing users to connect wires to component nodes or to join wires to other wires. Wires may be ended at a location other than a component connection node by clicking at the desired end point and hitting the Esc key. Wires can then be joined end node to end node. Segments of wires may also be selected and stretched, moved, or deleted in the manner of a typical CAD drawing. When placing wires, we need to be careful that we do not have unexpected disconnects. Save the schematic, and then try placing, stretching, and deleting a few wires around the schematic. Afterward, get the schematic back to the form of the one shown in Figure 9 by deleting the modified wires and rewiring or by repeatedly clicking Ctrl-Z.
For PSpice to work with the circuit, we must place a special reference node called GND and referred to as symbol 0 in OrCAD Capture. Click on the GND button in the button menu on the right of the Capture screen. A Place Ground dialog box will appear. In the Libraries area on the left bottom of the dialog box, select the SOURCE library. (If the SOURCE library does not appear in this area, click on Add Library, and select the file source.olb from the ..OrCAD_Demo>Capture>Library>PSpice subdirectory.) Choose the symbol 0 as shown in Figure 11. Place the ground symbol in the schematic and connect it with a wire as shown in Figure 10. The symbol 0 is a special symbol that produces what is called the reference node for PSpice. Other ground symbols found within Capture will cause failures in the PSpice simulation and should not be used. Be sure to remember to insert the PSpice ground symbol into the circuit before simulation, for without it the circuit will not simulate. The other ground symbols in the GND parts menu are used for schematic drawing purposes only.
We have now performed all of the steps necessary to describe a simple circuit in preparation for PSpice analysis. The process we just completed is generally referred to as schematic capture. The next automated operation that will be performed is a translation of the circuit information provided in the schematic, into a machine readable description called a netlist. Later in this chapter we will analyze the netlist generated by the Capture program. We will then examine how you may directly generate the netlist and simulate the circuit without using the Capture program.

Before a simulation can be executed, we must choose the type of analysis that is to be performed by PSpice. Section 2.2 shows us how to tell PSpice the type of analysis to perform and the type of results you are interested in seeing.

2.2 Specifying the Analysis

PSpice can perform several different types of analyses on the schematic we have drawn. For a simple resistive circuit like the one in Figure 10, the only analyses that are meaningful are analyses of the DC behavior of the circuit. PSpice will perform two different types of DC analysis: a bias point analysis and a .DC analysis. In the bias point analysis, PSpice analyzes the circuit's response to the basic DC sources operating in conjunction with the resistive elements in the circuit. The bias point analysis is also called the operating point analysis and sometimes symbolized as the .OP analysis.
To differentiate between the simulations performed, the first step is to name the simulation. Naming is done by choosing the menu option PSpice->New Simulation Profile. This selection generates the dialog box shown in Figure 12 in which we enter a name for this particular simulation run. We have named the simulation for Example 1, Simple Bias. Once we click on the Create button, a secondary dialog box appears in which we are asked to choose the analysis type for this simulation. In Figure 13 we see a Simulation Settings dialog box in which we have chosen the Analysis Type to be Bias Point. Several other settings are available in this dialog and its associated tabs. For now, leave them fixed to the default settings. One thing we may want to note for future reference is the location where the simulation output files will be placed. In the Simulation Settings dialog box under the general tab, we can see the area marked Output, and for this simulation we will note that the output files will be located at ...\OrCAD_Demo\circuits\resistor1-SCHENAMAT1C1-Simple Bias.out. We are now ready to initiate the PSpice simulation once we click OK to close this dialog box.

Simulation begins when we click on the PSpice->Run menu item in the Capture window. A display window similar to the one shown in Figure 14 appears showing the initial results of the simulation. In the lower left side of this window, we find information on the simulation process including whether or not any errors occurred during the process. To the right is an area that has multiple purposes depending on which tab is selected. For this simple simulation, the only information is contained in the Devices tab. The upper part of the window shown as dark gray will be used later to display printed and plotted results.

In the background during the simulation run, PSpice creates an output file (.OUT). It contains bias point information, model parameter values, error messages, and so on. If the simulation fails, you can view the output file to see the error messages. If the simulation completes successfully, PSpice produces a data file (.DAT). This is the file PSpice uses to display the simulation results.
Figure 13 Simulation Settings dialog box used to choose the analysis type.

Figure 14 Simulation results window.
2.3 Simulation Results

Click on the button that looks like this [button], and is the third from the top on the left-side list of the buttons in the simulation window. By clicking on this button we open the .OUT file for this simulation, which is created anew each time PSpice is run on this circuit description. The upper portion of the simulation results window will be filled with a textual description of the results of the simulation run similar to what is shown in Figure 15. The details of the netlist will be discussed further in section 2.4. For now we note that R1 is set between the node labeled N00015 and the node N00009, while R2 is set between N00009 and node 0. The voltage source is connected from node N00015 and node 0.

Scrolling down further in this part of the simulation output window, we can see the results of the nodal analysis as shown in Figure 16. Node N00009 is listed as 6.6667 volts while node N00015 is listed as 10.0000 volts. Each of these voltages is referenced to the node labeled 0. In other words, if we assume that the node labeled 0 is zero volts, then node N00009 is 6.6667 volts above zero while node N00015 is 10 volts above zero. The fact that N00015 is 10 volts above ground is established by connecting the constant voltage source V1 in the circuit. PSpice calculates the voltage for N00009 during the simulation. We can check the N00009 voltage by recognizing that the circuit is a simple resistive divider circuit. The voltage across node N00009 is calculated as

\[ V(N00009) = \frac{9400 \times 10}{4700 + 9400} = 6.6667. \]

![Figure 15 Portion of the simulation results showing the circuit netlist.](image)
In the output window, PSpice also tells us that the current delivered by the V1 voltage source is \(-7.092E-04\) amps, or \(-0.7092\) milliamps. PSpice calculates this current as

\[ I = \frac{10}{(4700+9400)} = 0.7092 \text{ mA.} \]

PSpice assumes the direction of current flow in the voltage source to be into the positive node of the source. Therefore, since the actual conventional current flow in this circuit is out of the positive node of V1, it is reported as negative.

Finally, PSpice calculates the value of the power supplied by the voltage source V1 and displays it in the bottom of the output window as 7.09 milliwatts. This number is simply the current supplied by the source multiplied by the voltage across the source. Note that PSpice has rounded the power down to three digits.

As part of the operation of displaying the output, PSpice sends simulation results information back to the Capture program to be displayed on the schematic diagram. After running PSpice the schematic Capture window should look like the one displayed in Figure 17. It may be necessary to click on the \(\textbf{V}\) button near the top-center of the schematic Capture window for the voltages to be displayed. This process is generally referred to as \textit{back-annotation} with the schematic.
diagram receiving information from the simulation program. Note that clicking on the button causes the back-annotated currents in the circuit to be displayed.

![Schematic Capture window with PSpice back-annotation.](image)

**Figure 17** Schematic Capture window with PSpice back-annotation.

When viewing results from this and any simulator, we should be sure to do some fundamental calculations of our own to insure that the simulator is providing reasonable results. Various types of errors may cause a simulation to produce results that are out of the domain of the expected solution. For example, using an incorrect abbreviation and producing a milliohm sized resistor instead of a megohm resistor will cause results that are likely to be well out of the expected range of behavior. A good designer must be able to recognize when a set of results is likely to be inside the range of correct possibilities.