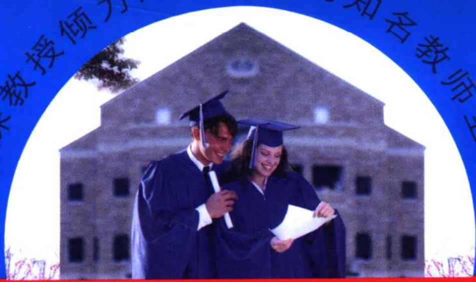
国外资深教授倾力之作　国内知名教师全力推荐



国外高校电子信息类优秀教材

PSpice入门手册

Introduction to PSpice Manual for *Electric Circuits* Using OrCad Release 9.1

（英文影印版）

James W. Nilsson　Susan A. Riedel 著

科学出版社
www.sciencep.com

国外高校电子信息类优秀教材(英文影印版)

PSpice 入门手册

Introduction to PSpice Manual

for *Electric Circuits*

Using OrCad Release 9.1

James W. Nilsson　Susan A. Riedel 著

科学出版社
北 京

内 容 简 介

本书是《电路》一书的配套手册，用于帮助读者解决书中的疑难问题和课后练习。本书支持 OrCAD 9.1 版本。

全书共分 14 章，主要介绍如何用 PSpice 绘制和仿真线性电路，如何用基本设备构造电路模型，以及如何利用计算机的输出数据验证仿真的结果。

本书可作为电子工程、计算机等专业的学生以及工程技术人员的参考书。

Introduction to PSpice Manual for Electric Circuits Using OrCad Release 9.1, by James W. Nilsson, Susan A. Riedel, Copyright ©2000

ISBN 0-13-016563-8

责任编辑：巴建芬 李 宇/封面设计：黄华斌 陈 敬/责任印制：刘秀平

科 学 出 版 社 出版

北京东黄城根北街16号

邮政编码：100717

http://www.sciencep.com

源海印刷有限责任公司 印刷

科学出版社发行 各地新华书店经销

*

2003年6月第 一 版 开本：787×1092 1/16

2003年6月第一次印刷 印张：9

印数：1—3 000 字数：200 000

定价：88.00元(含手册、光盘)

(如有印装质量问题，我社负责调换〈环伟〉)

Preface

ABOUT THIS MANUAL

Introduction to PSpice expressly supports the use of OrCAD PSpice A/D, Release 9.1 (herein after referred to as PSpice) as part of an introductory course in electric circuit analysis based on the textbook *Electric Circuits, Sixth Edition.* This supplement focuses on three things: (1) learning to draw and simulate linear circuits using PSpice, (2) constructing circuit models of basic devices such as op amps and transformers, and (3) learning to challenge computer output data as a means of reinforcing confidence in simulation. Because PSpice is designed to simulate networks containing integrated circuit devices, its range of application goes well beyond the topics covered in the textbook. Even though we do not exploit the full power of PSpice, we begin the introduction to this widely used simulation program at a level that you can use to test the computer solutions.

The use of PSpice involves learning a new technical vocabulary and a number of specialized techniques. Hence, we designed this supplement to stand on its own as an instructional unit. Our decision to separate this material from the parent textbook also is a service to you: The portable format greatly facilitates your use of the supplement at a computer terminal. In this format, the supplement adds value to your study.

You may use PSpice to solve many of the textbook's Drill Exercises and Chapter Problems. Those Chapter Problems that we think are particularly suited to PSpice simulation are marked with a  icon in the textbook. You are encourage to use PSpice to check your solutions to Chapter Problems, or to further explore the behavior of an interesting circuit.

Table 1: Relating Topics in this Supplement to Topics in the Text

MANUAL CHAPTER	TOPIC	TEXT CHAPTER
2–3	DC analysis	2–4
4	Thévenin Equivalents	4
4	Tolerance and Sensitivity	4
5	Op amps	5
6	Natural Response	7–8
7	Step response	7–8
7	Switches	7
8	Varying component values	8
9	AC Steady-state analysis	9
10	Linear Transformers	9–10
10	Ideal transformers	9–10
11	AC Power	10–11
12	Frequency response	13–15
12	Filter Design	14–15
13	Pulsed sources	16
13	Fourier series analysis	16

INTEGRATING PSPICE INTO INTRODUCTORY CIRCUITS COURSES

Although some circuits courses cover PSpice as an independent topic, many instructors prefer to integrate computer solutions with the course. To support such integration topics appear in this supplement in the same order in which they are presented in the text. Table 1 summarizes the relationship between the supplement and the textbook.

ABOUT PSPICE

SPICE is a computer-aided simulation program that enables you to design a circuit and then simulate the design on a computer. SPICE is the acronym for a Simulation Program with Integrated Circuit Emphasis. The Electronics Research Laboratory of the University of California developed SPICE and made it available to the public in 1975.

Many different software packages are available that implement SPICE on personal computers or workstations. Among them, OrCAD PSpice A/D, from OrCAD, Inc., is the most popular. PSpice's popularity can be attributed to many factors, including its user-friendly interface, extensions to SPICE that support modeling of digital circuits and much more, and its no-cost basic version. This manual focuses on how to use Release 9.1, and all the examples were produced using this release. If you are using a different version of PSpice, or another package that implements SPICE, your interaction with the software may differ from what you see in this supplement.

We limited our applications of PSpice to the types of circuit problems discussed in the textbook. Although PSpice is a general-purpose program designed for a wide range of circuit simulation—including the simulation of nonlinear circuits, transmission lines, noise and distortion, digital circuits, and mixed digital and analog circuits—here we discuss the use of PSpice only for dc analysis, transient analysis, steady-state sinusoidal (ac) analysis, and Fourier series analysis. You should refer to the OrCAD PSpice A/D User's Guide for information on all of the features of PSpice that are not discussed in this supplement.

We have included the OrCAD Evaluation CD in the back of this supplement. Insert the CD into your CD-ROM drive, and wait for the OrCAD main menu to appear after a short animation. If the main menu is not displayed after one minute, choose the Start menu and enter `D:\ORCADSTART.EXE`, where "`D`" is the letter assigned to your CD-ROM drive.

Contents

Chapter 1

A FIRST LOOK AT PSPICE

The general procedure for using PSpice consists of three basic steps. In the first step, the user describes the circuit to be simulated or analyzed by drawing it in schematic form. In the second step, the user specifies the type of analysis desired, and directs PSpice to perform that analysis. In the third and final step, the user instructs the computer to print or plot the results of the analysis. In this first look, we will carefully develop a simple example to illustrate how to perform these three steps.

1.1 DRAWING THE CIRCUIT

To begin, run the OrCAD Capture program, and select the option File/New/Project, as shown in Fig. 1. This option will invoke the New Project dialog box, as shown in Fig. 2. You should select a meaningful project name and the desired directory for your project files. You should also select the Analog or Mixed-Signal Circuit, as shown in Fig. 2, which will guide you through the rest of the process.

The next step is to load libraries of parts which will be made available to you when you are drawing the circuit schematic. The dialog window for adding and removing libraries is shown in Fig. 3. We recommend adding all of the available part libraries, to give you the widest choice of parts to use in specifying your circuit. After the libraries are added, you are back in Capture, but now have a circuit layout grid and the Capture toolbar, as shown in Fig. 4.

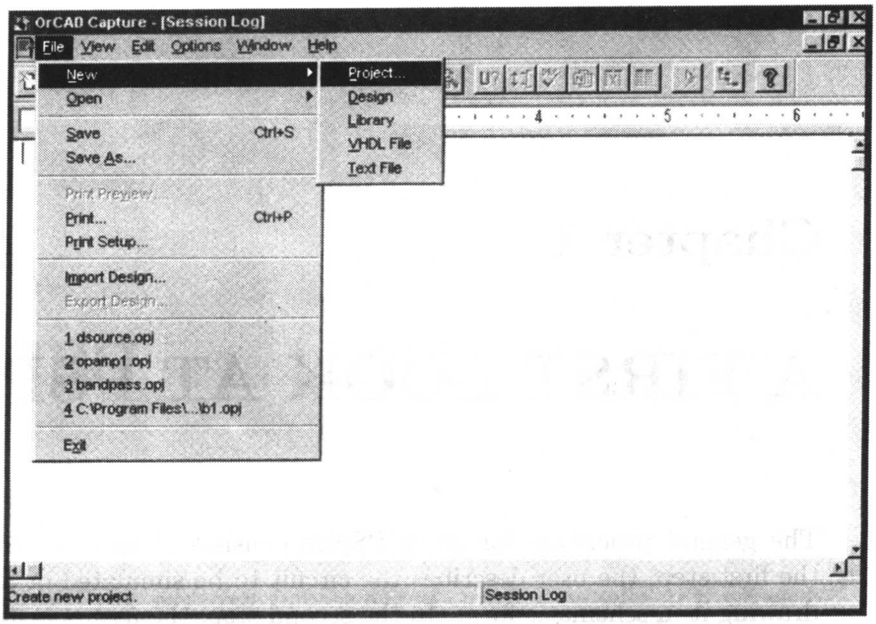


Figure 1: The initial window for OrCAD Capture.

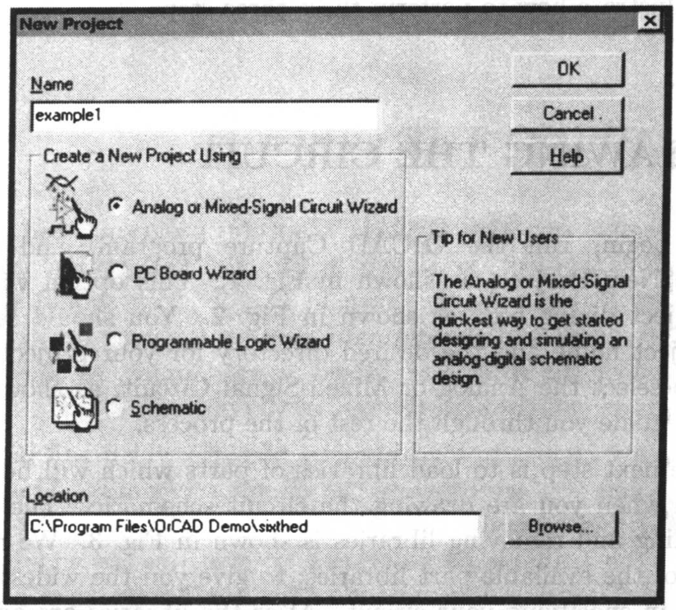


Figure 2: The New Project dialog box.

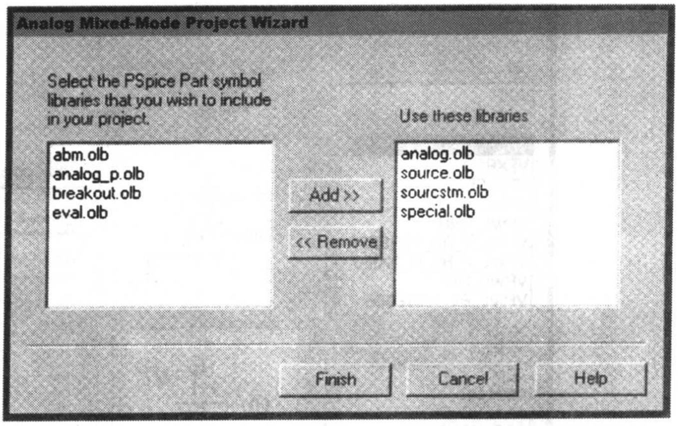


Figure 3: The dialog window for adding and removing part libraries.

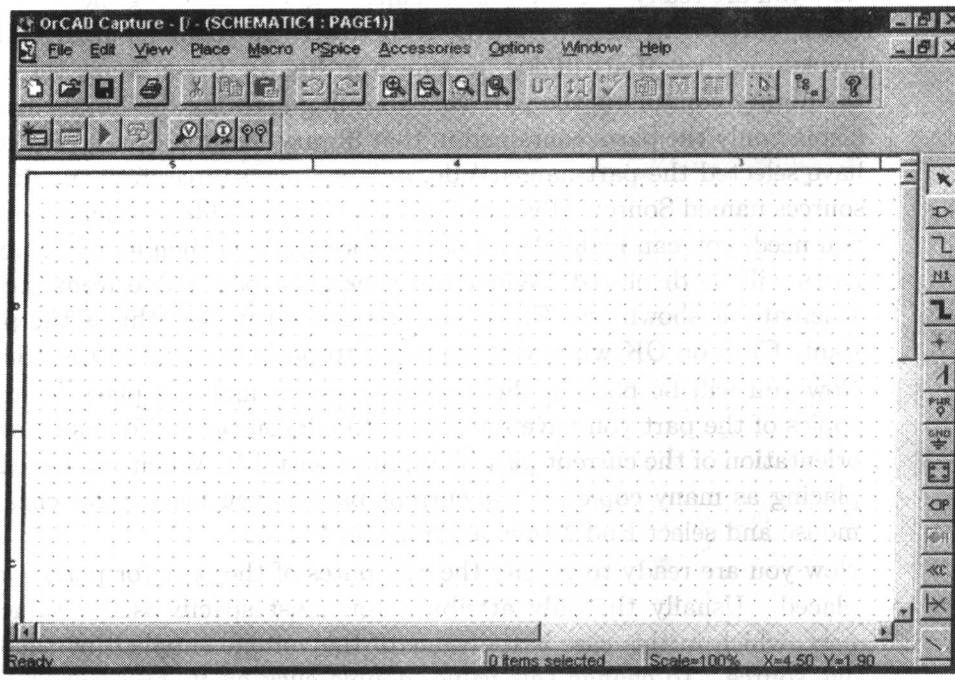


Figure 4: The circuit layout grid and Capture toolbar.

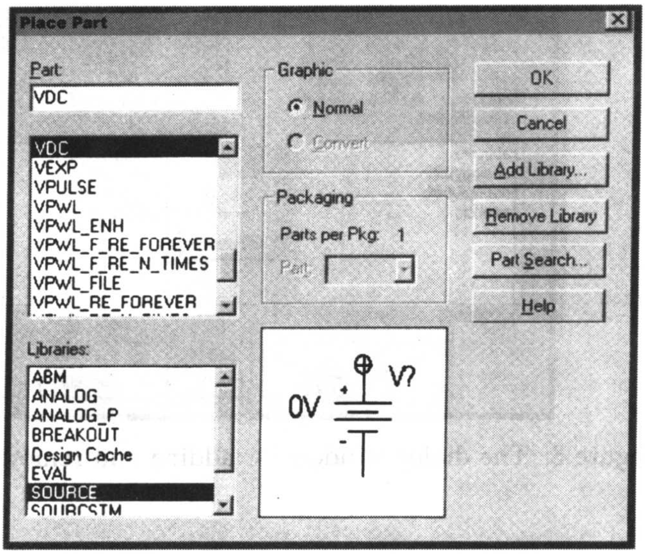


Figure 5: The Place Part dialog.

Now you are ready to draw your circuit. Begin by selecting the menu option Place/Part, or by clicking on the second vertical toolbar button. This will invoke the Place Part dialog, as shown in Fig. 5. If you know the name of the library containing the part you want, highlighting the library name will display only the parts contained in that library. As you can see in Fig. 5, we have selected the part named Vdc, a dc voltage source, from the library of sources named Source. If you are not sure of which library contains the part you need, you can highlight all of the libraries, and then all of the available parts will be displayed. When you click on a part name from the list its schematic is shown (see Fig. 5), so you can verify that this is the part you want. Click on OK when you are ready to place the part in the schematic.
Now you will be back in the Capture window and can place one or more copies of the part you have selected on the layout grid. You can change the orientation of the current part by typing Control-R. When you have finished placing as many copies of the current part as you need, right click on the mouse and select End Mode, as shown in Fig. 6.
Now you are ready to specify the attributes of the part (or parts) you have placed. Usually the only attribute you must specify is the value of the part, which in this case is the value of the voltage supplied by the dc voltage source. To change this value, double click on it, and you will see the Display Properties dialog for this part, shown in Fig. 7. You can see from

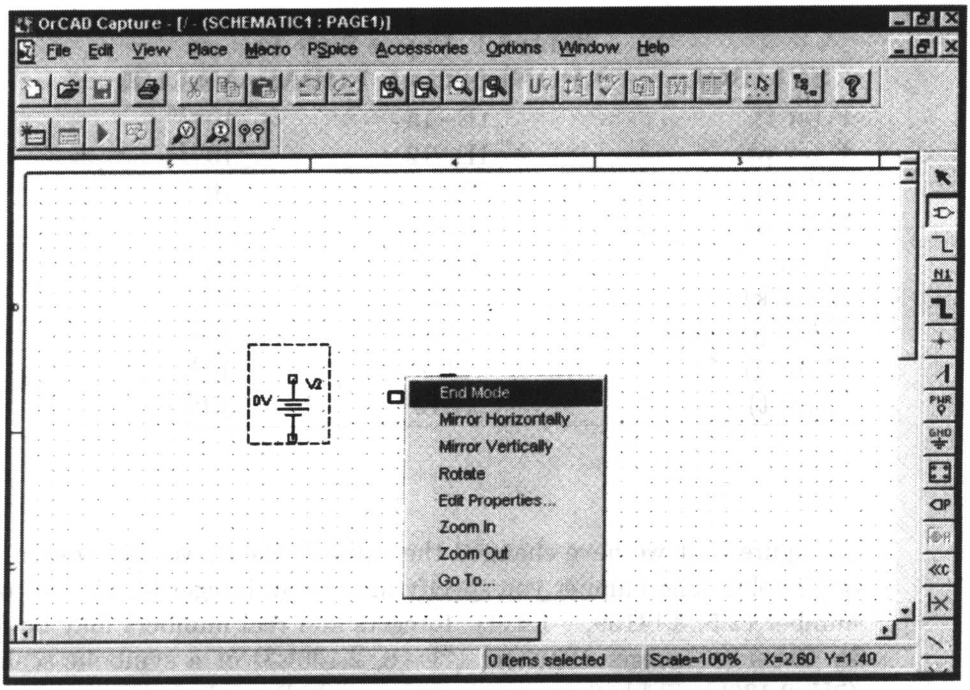


Figure 6: Place a dc voltage source on the layout grid, then right click on the mouse to finish with this part.

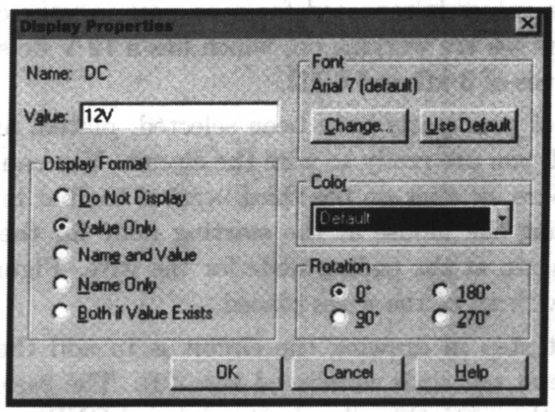


Figure 7: Using the Display Properties dialog window to set the value of the dc voltage source.

Table 1.1: PSpice Scale Factors

SYMBOL	EXPONENTIAL FORM	VALUE
F (or f)	1E−15	10^{-15}
P (or p)	1E−12	10^{-12}
N (or n)	1E−9	10^{-9}
U (or u)	1E−6	10^{-6}
M (or m)	1E−3	10^{-3}
K (or k)	1E3	10^{3}
MEG (or meg)	1E6	10^{6}
G (or g)	1E9	10^{9}
T (or t)	1E12	10^{12}

the figure that we have changed the value of the dc voltage source to 12V, or 12 volts. The number you specify may be an integer (4, 12, −8) or a real number (2.5, 3.14159, −1.414). Integers and real numbers may be followed by either an integer exponent (7E−6, 2.136E3) or a symbolic scale factor (7U, 2.136k). Table 2 summarizes the symbolic scale factors used in PSpice and their corresponding exponential forms. Letters immediately following a number that are not scale factors are ignored, as are letters immediately following a scale factor. For example, 10, 10v, 10HZ, and 10A all represent the same number, as do 2.5m, 2.5MA, 2.5msec, and 2.5MOhms.

The process of locating a part, placing it on the layout grid, and setting the attribute values is repeated for each part in your circuit. Figure 8 shows the circuit we are working on, which has a 12 V dc source and two resistors with values of 3 kΩ and 6 kΩ.

One all of your parts have been selected, placed, and had their attributes specified, you are ready to wire the circuit. You can select the menu option Place/Wire, or click on the third vertical toolbar button. Wires are placed by clicking the mouse at the starting node for the wire, and clicking the mouse again at the ending node for the wire. Figure 9 shows the example circuit with all of the wires placed.

The last step in drawing the circuit is to add the circuit ground, which must be at the node numbered zero (0). The easiest way to do this is to click on the vertical toolbar button labeled GND, and select the part named 0/Source, as shown in Fig. 10. This is a very important step and easy to

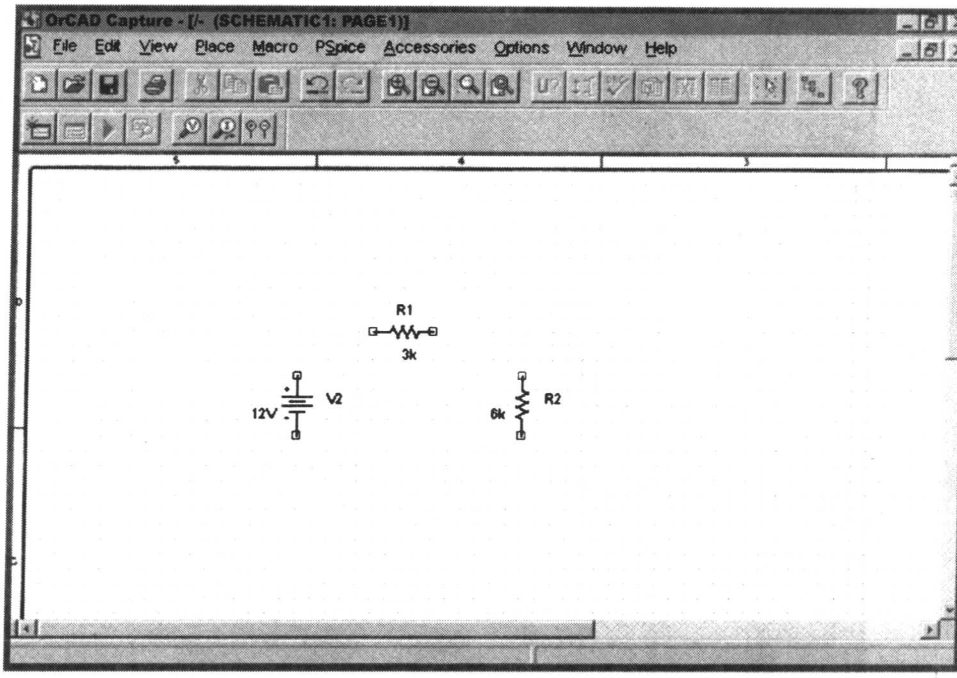


Figure 8: All of the circuit components have been selected and placed on the layout grid.

forget, but your circuit cannot be simulated by PSpice unless it contains a ground at node 0! Place the ground component on the layout grid, and wire it to your circuit. The completed circuit is shown in Fig. 11.

Now we are ready to specify the type of analysis to perform on this circuit. Before we specify the type of analysis to perform, let's briefly examine what PSpice will do with the circuit schematic we just drew. In order for PSpice to understand the circuit we have described, the schematic must be translated into a collection of statements that identify the circuit components, their attributes, and their topological connections. This collection of statements is called a netlist, and it is written to the output file before analysis begins. Sometimes it is useful to look at the netlist, especially if there is an error in the schematic. Therefore, the Appendix includes a subset of the netlist syntax, for your reference.

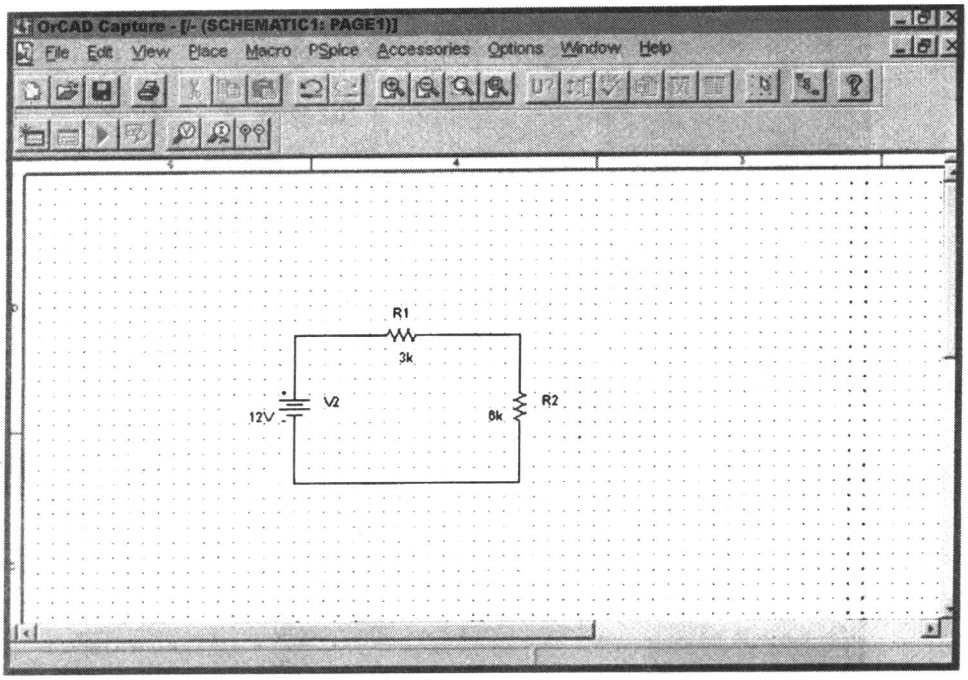


Figure 9: The circuit with all of the components wired together.

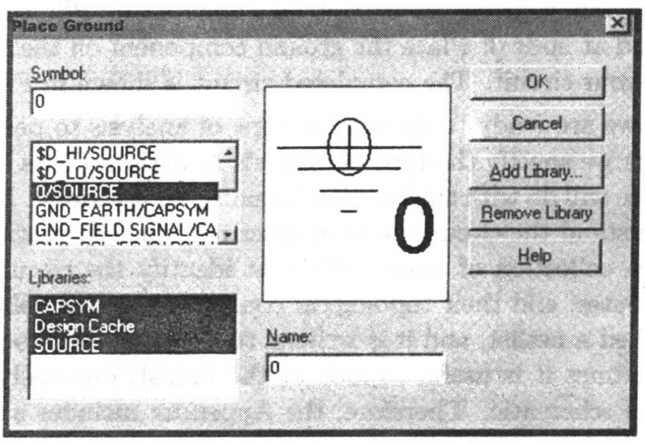


Figure 10: Selecting the ground component.

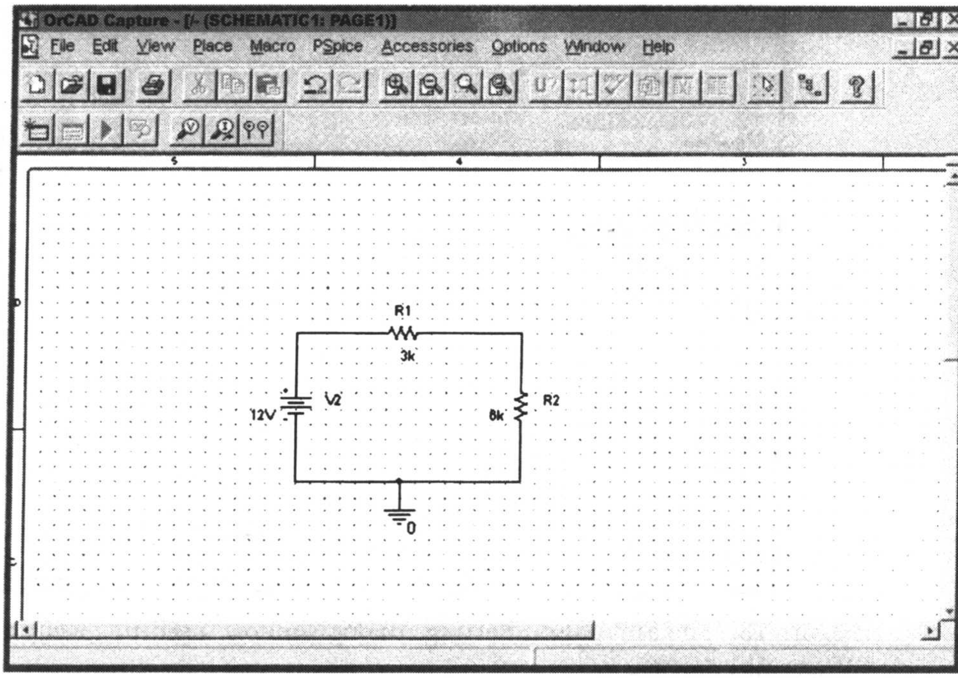


Figure 11: A completed circuit schematic.

1.2 SPECIFYING THE TYPE OF CIRCUIT ANALYSIS

You can ask PSpice to perform several different types of analysis on a circuit you have drawn. To begin, you need to select a name for the simulation. This is done by choosing the menu option PSpice/New Simulation Profile, which will generate the dialog shown in Fig. 12. We've chosen the name "bias" for this profile, as we will perform a simple dc analysis to determine

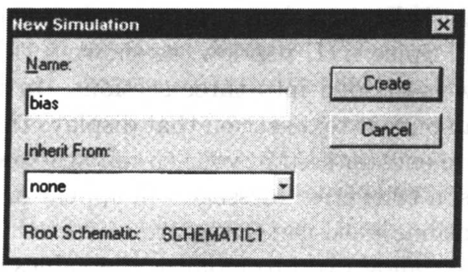


Figure 12: Creating a new simulation profile.