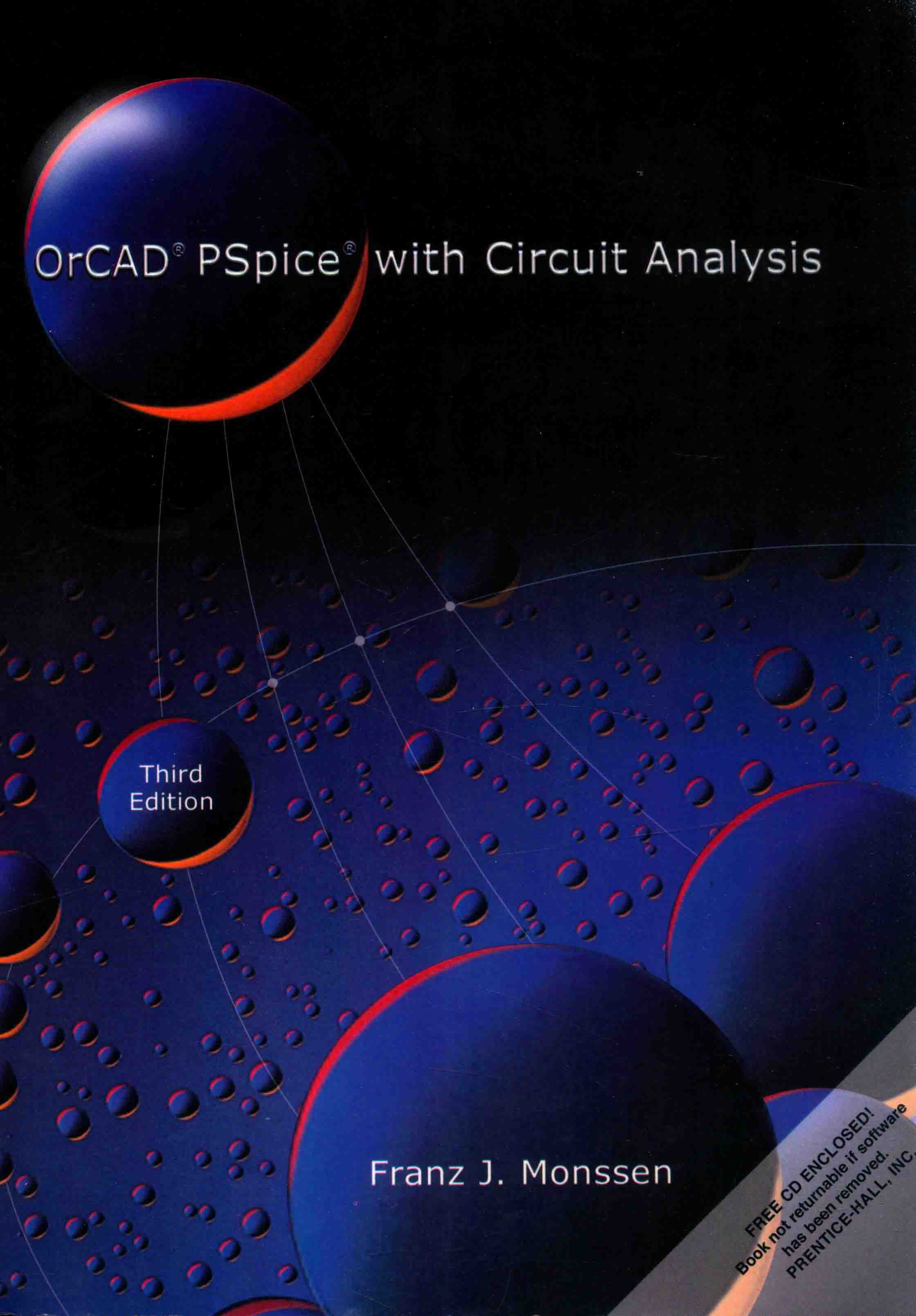

OrCAD® PSpice® with Circuit Analysis
Third Edition
Franz J. Monssen
FREE CD ENCLOSED!
Book not returnable if software has been removed.
PRENTICE-HALL, INC.

OrCAD® PSpice® with Circuit Analysis

Third Edition

Franz J. Monssen

Prentice Hall

Prentice Hall, Upper Saddle River, New Jersey 07458

Library of Congress Cataloging-in-Publication Data
Monssen, Franz.
OrCAD PSpice with circuit analysis/ Franz Monssen.—3rd ed.
p. cm
Rev. ed. of: MicroSim PSpice with circuit analysis. 2nd ed. c1998.
Includes index.
ISBN 0-13-017035-6 (alk. paper)
1. Electronic circuit analysis—Data processing. 2. PSpice. I Monssen, Franz. MicroSim PSpice with circuit analysis. II. Title.

TK 454.M65 2001
621.319'2'07855369—dc21 00-061975

Vice President and Publisher: Dave Garza
Editor in Chief: Stephen Helba
Acquisitions Editor: Scott J. Sambucci
Production Editor: Rex Davidson
Design Coordinator: Karrie M. Converse-Jones
Cover Designer: Thomas Borah
Cover Photo: SuperStock
Production Manager: Pat Tonneman
Marketing Manager: Ben Leonard

10 9 8 7 6 5 4 3

Prentice Hall

ISBN 0-13-017035-6

PREFACE

Since the publication of the first edition of *PSpice with Circuit Analysis*, the change-over from the DOS to the Windows format and the proliferation of relatively cheap personal computers have created the need for this third edition of the text.

The impact of such programs as PSpice will be profound in the workplace of the present and future electrical engineer. Also, they will increasingly affect the fashion in which electrical engineering and electronics technology are taught in colleges and technical schools. The analysis of electrical and electronic circuits even of modest size involves both complex and lengthy calculations. By means of PSpice, circuit complexity is far less a hindrance to a successful analysis of electrical circuit behavior. Relatively few rules of program syntax together with a few click-and-drag operations allow the electrical engineer and the student to solve complex circuits and produce circuit schematics of professional quality.

The successful evaluation of a formula, when done by a calculator, gives the relationship between circuit variables at a particular operating point. PSpice, by contrast, allows for a global perspective of circuit behavior. The ease, compared to traditional methods, by which a frequency or transient (time) analysis can be performed well illustrates the point. Oscilloscope-like displays of circuit variables, their mathematical relationships, and concepts such as the RMS value of a signal can all be accurately and quickly displayed.

PSpice allows a shift of emphasis away from computation of circuit variables toward their interpretations. It also allows a shift away from the analysis on the component level of circuits to the analysis of systems consisting of many circuits. Traditionally, students spend considerable time analyzing circuits containing a single bipolar transistor. However, practical circuits such as multistage amplifiers, operational amplifiers, active filters, and communication circuits all contain many transistors in addition to numerous other solid-state devices. Circuits of such complexity can be analyzed with relative ease by the PSpice program.

Educators are challenged to incorporate these PSpice capabilities into the existing curriculum. With the pedagogical approach in this text, a particular circuit phenomenon is observed by means of graphical and numerical output data generated by PSpice. Having observed the phenomenon, the student is prompted to seek an explanation. At this point, mathematical concepts and formulas are introduced. The student is asked to show the correlation between data and calculation. A typical example of this approach is found in Chapter 6 of this text. There, the response of a RC circuit to a sinusoidal voltage is investigated by means of the PSpice program. Subsequent to that, the phasor method is introduced to solve for the circuit response. Finally, the results of that method and the PSpice data are compared. The structure of this book reflects this approach. Although its topical outline supports the traditional electrical engineering curriculum, a dual approach is taken. Every new circuit concept is introduced with its relevant PSpice commands. Subject matter and the PSpice program are used in a mutually supportive fashion.

This text is neither an electrical engineering textbook with the PSpice program relegated to an appendix nor is it simply a reference manual for PSpice. The underlying assumption of this book is that the combination of subject matter and the PSpice program can foster conceptual understanding and competence to advance the learning process. This text does not advocate the neglect of tradition teaching methods in favor of the use of the PSpice program; rather, it uses them in conjunction with the program.

The material in this book is divided into ten chapters, which cover the topics usually found in a one-year course in electrical circuit analysis. Chapter 1 is devoted to the analysis of dc circuits containing single and multiple independent current and voltage sources. The analysis of circuits containing dependent current and voltage sources is also included in this chapter. These devices play an important role in the modeling of many solid-state devices.

Chapter 2 introduces some fundamental network theorems as applied to dc circuits. The Superposition theorem, Thevenin's theorem, and Norton's theorems are demonstrated. Source conversions and the Maximum Power Transfer theorems conclude the chapter.

Chapter 3 introduces the reader to the transient phenomena encountered in RC and RL circuits. The continuity of a capacitor's voltage and an inductor's current are related to the power flow and the energy content of these elements. The concept of the time constant is introduced. The circuit response to a linear pulse train is introduced. The model of a switch as used in PSpice is applied to circuits.

Chapter 4 has as its subject the application of sinusoidal currents and voltage to resistive circuits. The latter are used so as not to introduce unnecessary complications at this time. The effects of time and phase shifts, their relation, and their effect on circuit behavior are studied. The sums of sinusoidal currents and voltages are obtained.

Chapter 5 investigates the steady-state sinusoidal response of series, parallel RC, RL, and RLC circuits. The concept of impedance is demonstrated graphically by using PROBE plots. These plots demonstrate that admittance and impedance are steady-state concepts. The equivalency between various series and parallel circuits is demonstrated.

Chapter 6 investigates the total response of series and parallel RC, RL, and RLC circuits. This response is shown to be the sum of the forced response due to the source and the transient, or natural, response of a circuit. The subject of electrical resonance investigated.

Chapter 7 extends the concepts of Chapter 2 to circuits containing resistors, capacitors, and inductors. Superposition is applied to circuits containing both ac and dc current and voltage sources. The ability of students to analyze such circuits will become increasingly important as they progress to electronic circuits containing solid state devices. It is demonstrated that Norton and Thevenin impedances are generally complex quantities. It is shown that maximum power from a source to a load will flow when the source impedance is the complex conjugate of the load impedance.

Chapter 8 shows that power and energy flow in alternating current circuits. The power and energy relations for resistors, capacitors, and inductors are investigated. The concept of apparent, real, and reactive power is introduced and obtained by PSpice. It is important to note that this chapter provides the theoretical basis upon which the electrical utility industry is built.

Chapter 9 introduces the reader to the frequency response of RC, RL, and RLC circuits. The necessary PSpice statements are introduced and applied. Both magnitude and phase plots are obtained and the impedance of circuits is plotted as a function of frequency. The concepts of the 3dB frequency are introduced and applied to the filters. The RC circuit is used both in its high-pass and low-pass configurations. The effect of multiple sections on roll-off is demonstrated. The characteristics of a band-pass filter are examined. The concept of the Q factor is covered. The relationship between transient analysis and AC (frequency) analysis is stressed.

Chapter 10 applies non-sinusoidal current and voltage sources to electrical circuits. The ability of the PSpice program to perform a Fourier analysis is demonstrated. The Fourier transforms of some standard waves are obtained. The PSpice program calculates the total harmonic distortion (THD) of a wave. The RMS value of a wave train and the power it delivers to a load are calculated. The effects of half-wave and full-wave rectification on a Fourier spectrum are investigated. Wave symmetry and the effects of time shift of a wave are explored. Finally, a square pulse is applied to a RC high-pass filter and a band-pass filter. The effects of these circuits on the Fourier transform are investigated.

Acknowledgments

At the completion of this book, it is proper to thank those who contributed to it. Even a short reflective pause makes the author aware of his indebtedness to so many. It is impossible to tell how many years ago that the seed of this book was planted. There were the dedicated teachers at the City College of New York who introduced a young and untutored mind to the exciting world of electrical engineering. There are those by whose prior intellectual efforts this author has profited. By formal lecture and informal discussions over a cup of coffee, remarks were made and insights gained that found expression in this book.

There are those with whom the author has had the privilege and pleasure of a congenial professional relationship. Among them is the late Professor Joseph Aidala, the former chairperson of the Electrical and Computer Engineering Technology Department at Queensborough Community College. Thanks are due to the present chairperson, Dr. Louis Nashelsky, whose professional competence is surpassed only by his human decency. Thanks is due to Professor Robert Boylestad, friend and colleague, for introducing me to various people at Prentice Hall who became interested in the publication of a book on PSpice.

The work not only has to be written but it also needs to be published. This obliges me to thank all those involved in that part of the process. Initial thanks must go to Scott Sambucci, the Electronics Technology acquisitions editor at Prentice Hall. A special thanks goes to Rex Davidson, who guided the editing process through its various stages. Finally, a thank you to Linda Thompson, who did the editing and added to my humility.

Writing on a more personal note, a special thanks must go to my dearest Daphine, who wanders with me along life's road. Thanks also to my two sons, Georg (without the final e) and Stefan, who, despite having been teenagers, turned into two wonderful adults.

CONTENTS

9

FREQUENCY RESPONSE OF RC, RL AND RLC CIRCUITS 271

10

CIRCUITS WITH NONSINUSOIDAL SOURCES 327

1

PSPICE ANALYSIS OF DC CIRCUITS

ANALYSIS OF A SERIES CIRCUIT

Kirchhoff's Voltage Law for Series Circuits

A series circuit has only one current path. If a voltage or current source is connected to a series circuit, the current is the same throughout the circuit. For such a circuit, Kirchhoff's voltage law states: "the algebraic sum of the voltages around the closed path of the circuit is equal to zero volts." It follows that the electrical power delivered by the source(s) is equal to the power consumed by the resistor(s). This is a demonstration of the law of the conservation of energy as applied to electrical circuits.

Ideal DC Current and Voltage Sources

An ideal dc voltage source maintains a constant voltage across its terminals independent of time and any external circuit elements connected to it. An ideal dc current source maintains a constant current between its terminals independent of time and any external circuit elements connected to it.

The Resistor

We begin our analysis with purely resistive circuits. A resistor dissipates electrical energy over time. The rate of energy dissipation is defined as the power to the resistor. The ratio of the voltage across and the current through a resistor is defined as the resistance of a resistor. Its units are most commonly ohms (Ω) or kilohms (kΩ).

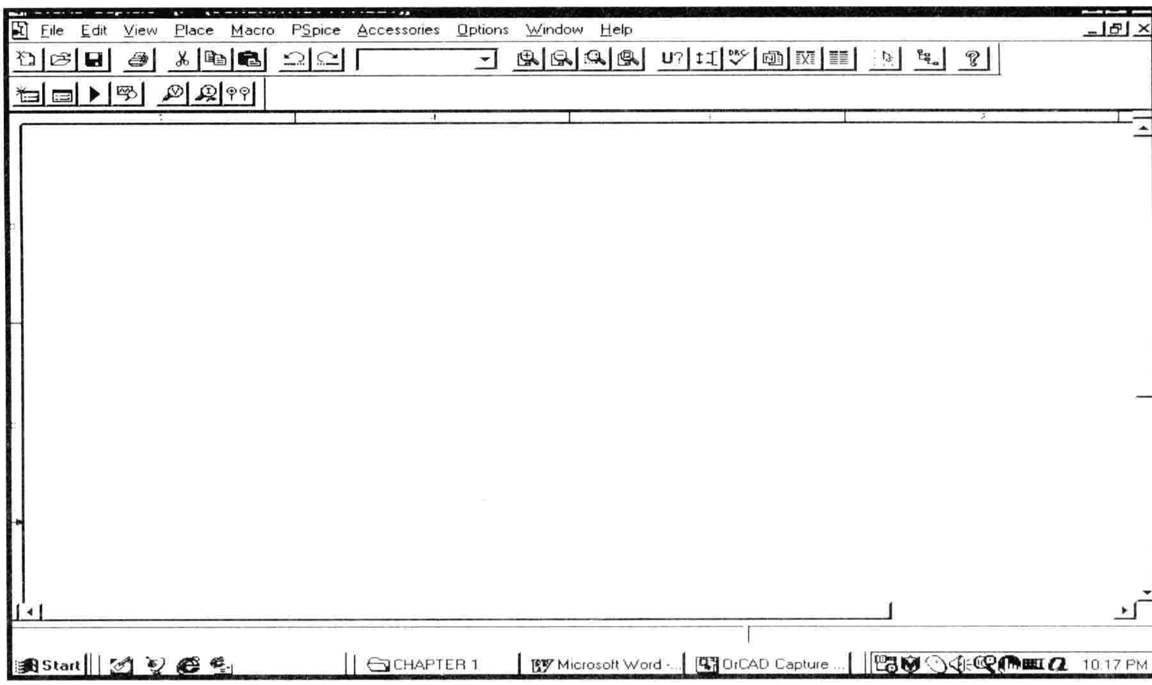
Preparing the Circuit Schematic

New file

In the analysis that follows, **OrCad's PSpice Evaluation Software Release 9.1** is used. We start our analysis with the creation of the circuit schematic. Our circuit consists of three resistors in series. Their resistances are 1 kΩ, 4 kΩ and 5 kΩ. A 20 volt dc voltage source is connected to the circuit. Do the following:

1. On the computer's **Desktop** screen, click on **Start,** move cursor right to **Program**, move right to **OrCad Demo**, move right to **Capture CIS Demo** and click on it**.** The **OrCad Capture** screen opens.
2. Click on **File**, move to **New**, move to **Project** and click**;** the **New Project** dialog box opens.
3. In the **Name** box, enter the name of circuit. The author used Figure 101.

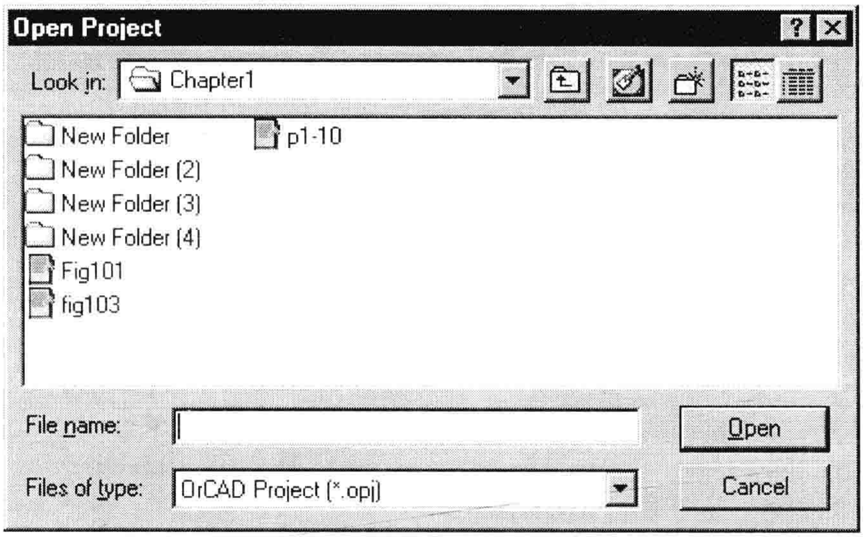
4. In the **Location** box, type information about where circuit file will reside.
5. Select **Analog or Mixed-Signal Circuit Wizard** by clicking on it.
6. Click on **OK**. The **Analog Mixed-Mode Project Wizard** dialog box opens.
7. Select analog.olb. Click on **Finish**. The **OrCad Capture [I-(Schematic1: Page 1)]** appears. If toolbar does not appear on right edge of the Schematic page, click anywhere on the screen. The screen is shown next.



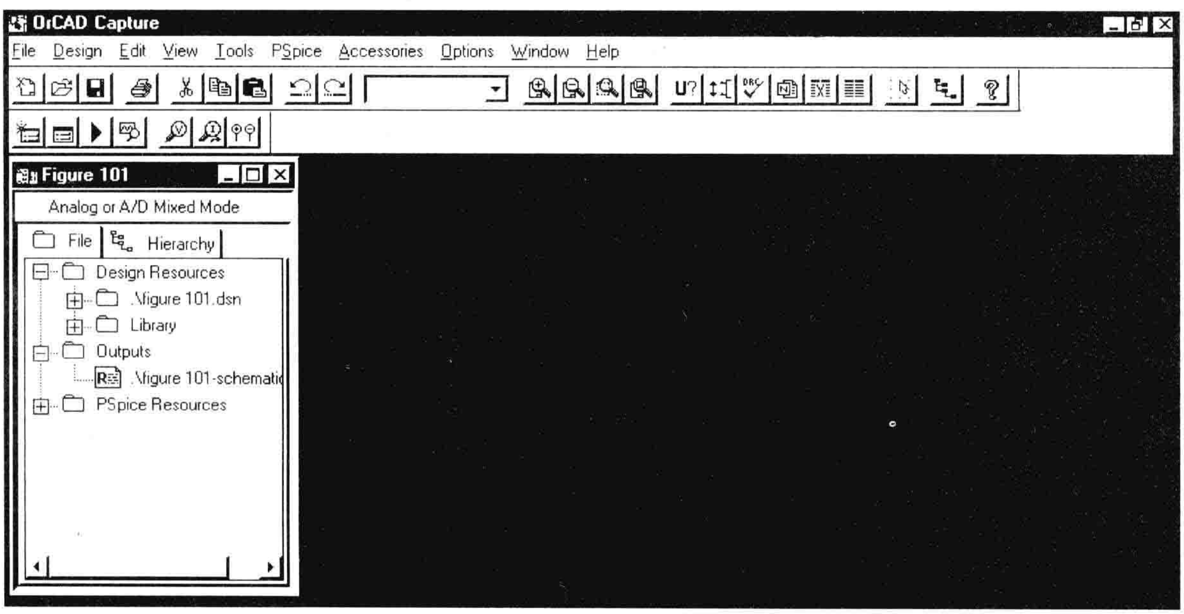
Existing file

If an existing file is to be recalled for simulation:

1. From **Desktop**, click on **Start**, drag to **OrCad Demo**, drag to **Capture CIS Demo**, and click on it. The **OrCad Capture** screen opens.
2. Click on **File**, drag to **Open**, drag to **Project** and release. The **Open Project** dialog box opens.



3. Click on the desired file and click on **Open**. The **OrCad Capture** screen opens.
4. Click on the plus sign [+] on the left side of selected file.
5. Click on the plus sign [+] on the left side of **Schematics1** folder. The **Page1** icon opens.
6. Double click on the **Page1** icon. The **OrCad Capture-[(Schematics]:Page1)]** opens. It contains the desired circuit schematic.



New File: Placement of Parts

Getting a resistor

1. Click on the icon with diode legend. It is the uppermost icon in the task bar on the right edge of the Schematic Page. The legend **Place Part** will appear briefly.
2. Click on it. The **Place Part** dialog box will appear.
3. In **Libraries** box, click on **ANALOG**.
4. In **Part** box, type **R** for resistor or scroll to **R;** click on it to select it.
5. In **Place Part** box, click on **OK**. The symbol of a resistor will appear on the Schematic Page.
6. Drag it to the desired location. Click to place it.
7. Drag to the next location to place the second resistor. Click to place it.
8. Repeat procedure for the third resistor.
9. Click the right mouse button. Select **End Mode** and click on it.
10. Deselect the third resistor by placing the pointer anywhere on the **Schematic** page and click.

Getting a voltage source

1. Click on the **Place Part** icon. The **Place Part** dialog box will appear.
2. In the **Libraries** box, scroll to and select **SOURCE**.
3. In the **Part** box, scroll to and select **VDC** by clicking on it. Click on **OK**.
4. Drag **VDC** to the desired location. Click to place it.
5. Click the right mouse button. Select **End Mode** and click on it.
6. Deselect **VDC** by placing the pointer anywhere on **Schematic** page and click on it.

Wiring the Components

1. On the right side of screen, on the tool bar, point to the second icon from the top. The legend **Place Wire** will appear briefly. Click on it.
2. Drag the cross-hair pointer to positive end of **VDC** source. Click on it
3. Move the cross-hair pointer to one terminal marker of first resistor. Click on it.
4. Move the cross-hair pointer to other terminal marker of resistor. Click on it.
5. Move onto the terminal marker of next resistor. Click on it.
6. Repeat the procedure till all components are connected.
7. Click the right mouse button. Select **End Wire** and click on it.
8. Deselect the last wire segment by clicking anywhere on **Schematic** page.

Labeling the nodes

A node is a connecting point between two or more circuit elements In our circuit, we can think of the connecting wires as elongated nodes. To identify the nodes, we shall label each of them.

1. Click on a wire to select it.
2. Point to the third icon from the top. It has the legend **N1**. Click on it.
3. The **Place Net Alias** dialog box appears.
4. In that box, type in the desired label. Click on **OK**.

5. Drag the small rectangle to selected node (wire). Click on it. The selected label will appear.
6. Click the right mouse button to select **End Mode**. Click to end labeling of the node.
7. Repeat for all other nodes.
8. Important: one node must be labeled zero! It is the ground node.

Setting the resistance values

All resistors placed on the **Schematic** page initially have a value of 1 kΩ. To change these values, if needed, proceed as follows:

1. Double click on the resistance value of a selected resistor. The **Display Properties** box appears.
2. In the **Value** box, type in the desired resistance. Click on **OK**. Click to deselect.
3. Repeat for all resistors.

Renaming resistors

Should it be desired to rename the resistors, proceed as follows:

1. Double click on the label of resistor, for example, **R1**. The **Display Properties** box will appear.
2. In the **Value** box, type the desired name. Click on **OK**. The desired name will appear in place of the former name of the resistor.
3. Click anywhere on the screen to end process.

Setting the Voltage of VDC

We next set the voltage of **V1** to the desired 20 volts.

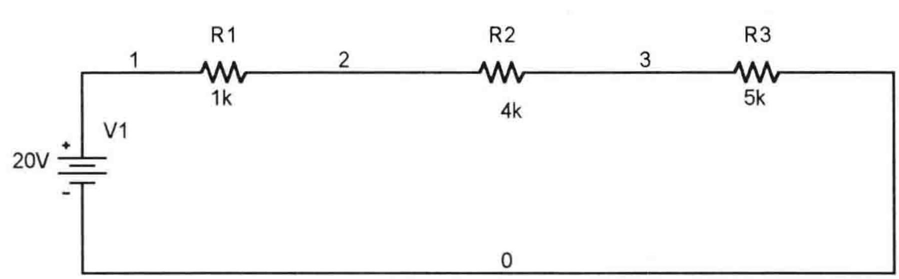
1. Double click on the voltage of V1. The **Display Properties** dialog box appears.
2. In the **Value** box, type 20V. Click on **OK**.
3. In **Schematic** page, click to deselect **V1**. Its new voltage of 20 volts will appear.

Placing Text on the Schematic page

1. Click on the **A** icon. It is at the bottom of the task bar. The **Place Text** box opens.
2. In the box, type the desired text. Click on **OK**.
3. Drag the rectangle to the desired location on **Schematic** page. Click to place.
4. Click on the right mouse button and select **End Mode**. Click on it.
5. Click in **Schematic** page to deselect text.

At the completion of this section the circuit schematic will appear as shown in Figure 1.01. We shall next prepare our circuit so that its voltages and its current can be obtained and displayed both in numerical and in graphic format.

Figure 1.01



Obtaining the Numerical Output Data

During simulation, **PSpice** generates an **Output File** which checks the correctness of the simulations and generates an error message should there be an error in the syntax of the circuit file. It prints out the value of specified circuit variables.

The circuit voltage: VPRINT1

To print the voltage of any node with respect to ground, we place **VPRINT1** at the node of interest. It is desired to obtain the printed values of the voltages at nodes 2 and 3.

1. Click on the **Place part** icon.
2. In the **Place Part, Libraries** box, select **SPECIAL**.
3. Scroll to **VRPINT1,** click on it and click on **OK**.
4. Drag **VPRINT1** to node 2 and click to place it.
5. Drag **VPRINT1** to node 3 and click to place it.
6. Click Right, select **End Mode** and click on it.
7. Deselect **VPRINT1** by clicking anywhere on the **Schematic** page.

The circuit voltage: VPRINT2

To print the voltage between any two nodes in the circuit, place **VPRINT2** at the nodes of interest. It is desired to obtain the printed value of the voltage across **R2**.

1. Click on the **Place Part** icon.
2. In **Place Part, Libraries** box, select **SPECIAL**
3. Scroll to **VPRINT2**, click on it and click on **OK**. Note: **VPRINT2** has two leads that must be wired into the circuit.
4. Drag **VPRINT2** to a place above **R2**. Click to place it and wire it into the circuit.
5. Click on **Right**, select **End Mode** and click on it.
6. Deselect **VPRINT2** by clicking anywhere on the **Schematic** page.

The circuit current: IPRINT

Since the current in this circuit is everywhere the same, the **IPRINT** device can be placed at a convenient location. It is entered into a circuit like an ammeter.

1. Select the horizontal section of node 0 and press **Del** button.
2. Click on the **Place part** icon.
3. In the **Place Part**, **Libraries** box, select **SPECIAL.**
4. Scroll to **IPRINT,** click on it and click on **OK**.
5. Drag **IPRINT** to the deleted portion of node 0. Click to place it and rewire.
6. Click on **Right**, select **End Mode** and click on it.
7. Rewire node 0.

Activating the IPRINT and VPRINT devices

The **IPRINT** and **VPRINT** devices need to be activated to obtain a printed output. Proceed as follows: Double click on **VPRINT1**. The **OrCad Capture- [Property Editor]** page opens.