



# OrCAD<sup>®</sup> PSpice<sup>®</sup> with Circuit Analysis

Third  
Edition

Franz J. Monssen

FREE CD ENCLOSED!  
Book not returnable if software  
has been removed.  
PRENTICE-HALL, INC.

# **OrCAD<sup>®</sup> PSpice<sup>®</sup> with Circuit Analysis**

**Third Edition**

**Franz J. Monssen**



Prentice Hall, Upper Saddle River, New Jersey 07458

**Library of Congress Cataloging-in-Publication Data**

Monssen, Franz.

OrCAD PSpice with circuit analysis/ Franz Monssen.—3<sup>rd</sup> ed.

p. cm

Rev. ed. of: MicroSim PSpice with circuit analysis. 2<sup>nd</sup> 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

OrCAD® and PSpice® are registered trademarks of Cadence Design Systems.

Second edition, entitled *MicroSim® PSpice® with Circuit Analysis*, copyright 1998 Prentice-Hall, Inc.

First edition, entitled *PSpice® with Circuit Analysis*, copyright 1993 by Macmillan Publishing Company.

---

**Copyright © 2001 by Prentice-Hall, Inc., Upper Saddle River, New Jersey 07458.** All rights reserved. Printed in the United States of America. This publication is protected by Copyright and permission should be obtained from the publisher prior to any prohibited reproduction, storage in a retrieval system, or transmission in any form or by any means, electronic, mechanical, photocopying, recording, or likewise. For information regarding permission(s), write to: Rights and Permissions Department.

10 9 8 7 6 5 4 3



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

### 1

#### **PSPICE ANALYSIS OF DC CIRCUITS 1**

ANALYSIS OF A SERIES CIRCUIT 1  
Kirchhoff's Voltage Law for Series Circuits 1  
Ideal DC Current and Voltage Sources 1  
The Resistor 1  
Preparing the Circuit Schematic 1  
Placement of Parts 4  
Wiring the Components 4  
Labeling the Nodes 4  
Setting the Resistance Values 5  
Renaming Resistors 4  
Setting the Voltage of VDC 5  
Placing Text on the Schematic Page 5  
Obtaining the Numerical Output Data 6  
Running the Simulation 7  
Power in the Circuit 11  
Analysis of a Resistive Voltage-Divider Circuit 12  
A Current Source Applied to a Series Circuit 14  
PSPICE ANALYSIS OF A PARALLEL CIRCUIT 17  
Kirchhoff's Current Law for Parallel Circuits 17  
Equivalent Conductance/Resistance of a Parallel Circuit 18  
A Series-Parallel Circuit 19

Dependent Sources 20  
Loading Effects of Voltmeters in Circuits 25  
Low-Resistance Measurement with VOM and DMM 26  
High-Resistance Measurements with VOM and DMM 26  
PARAMETRIC ANALYSIS 27  
PROBLEMS 31

### 2

#### **DC NETWORK THEOREMS 45**

THE PRINCIPLE OF LINEARITY 45  
Demonstration of Linearity 45  
THE PRINCIPLE OF SUPERPOSITION 49  
THEVENIN'S THEOREM 52  
NORTON'S THEOREM 58  
SOURCE CONVERSION 60  
MAXIMUM POWER TRANSFER THEOREM 61  
An Ideal Voltage Source in a Circuit 61  
A Nonideal Voltage Source in a Circuit 62  
A Fallacy to Guard Against 63  
Thevenin's and Maximum Power Transfer Theorems Combined 64  
PROBLEMS 65



## 3

### **TRANSIENTS IN RC AND RL CIRCUITS 71**

TRANSIENT RESPONSE OF AN RC CIRCUIT 71

The RC Circuit as an Integrator 77

The RC Circuit as a Differentiator 78

Pulse Train Applied to an RC Circuit 80

TRANSIENT RESPONSE OF AN RL CIRCUIT 82

The RL Circuit as an Integrator 87

The RL Circuit as a Differentiator 88

Current Pulse Applied to an RL Circuit 90

DETERMINATION OF THE AVERAGE VALUES OF CURRENT AND VOLTAGE WAVEFORMS 92

THE SWITCH IN PSPICE 94

PROBLEMS 96

## 4

### **SINUSOIDAL WAVEFORMS IN RESISTIVE CIRCUITS 107**

APPLYING A SINUSOIDAL VOLTAGE SOURCE TO A RESISTIVE CIRCUIT 107

A Problem 109

An Attempted Solution 109

A Solution: RMS 109

Circuits with Multiple Sinusoidal

Current or Voltage Sources 112

TIME DELAY AND PHASE SHIFTS IN SINE WAVES 115

The Relationship Between Time Delay and Phase shift 116

MULTIPLE-PHASE SYSTEMS 117

A Two-Phase System 117

A Three-Phase System 120

DIFFERENTIATION AND

INTEGRATION OF

SINE WAVES 124

PROBLEMS 126

## 5

### **STEADY STATE SINUSOIDAL RESPONSES OF RC, RL AND RLC CIRCUITS 139**

SINUSOIDAL RESPONSES OF CAPACITORS AND

INDUCTORS 139

Sinusoidal Voltage Applied to a

Capacitor 140

Sinusoidal Voltage Applied to an

Inductor 142

A SINUSOIDAL VOLTAGE APPLIED TO AN RC CIRCUIT 144

Phasor Analysis of Figure 5.04 146

A SINUSOIDAL VOLTAGE APPLIED

TO AN RL CIRCUIT 148

Phasor Analysis of Figure 5.05 150

A SINUSOIDAL VOLTAGE APPLIED

TO A SERIES RLC CIRCUIT 151

Phasor Analysis of Figure 5.06 153

Determining the Impedances of Figure

5.06 Using PROBE 154

Adding Phase Shift to the Input

Voltage V1 155

SINUSOIDAL VOLTAGE APPLIED

TO A PARALLEL RLC CIRCUIT 156

REDUCTION AND EQUIVALENCY

BETWEEN ELECTRICAL

CIRCUITS 158

PROBLEMS 163

## 6

### **TOTAL RESPONSE OF RC, RL AND RLC CIRCUITS WITH SINUSOIDAL SOURCES 179**

INTRODUCTION 179

The Total Response of an RC Circuit 179

Analysis of an RLC Circuit: The Overdamped Response 182

Analysis of an RLC Circuit: The Critically Damped Response 186

Analysis of an RLC Circuit: The Underdamped Response 189

RESONANCE IN ELECTRICAL CIRCUITS 191

PROBLEMS 195

## 7

### **ALTERNATING CURRENT NETWORK THEOREMS 203**

INTRODUCTION 203

THE SUPERPOSITION THEOREM APPLIED 203

Conventional Analysis of Figure 7.01 203

The Superposition Theorem Applied to Figure 7.01 204

THE THEVENIN THEOREM APPLIED 207

Conventional Analysis of Figure 7.04 207

The Thevenin Theorem Applied to Figure 7.04 208

Determining the Thevenin Voltage 208

Determining the Thevenin Impedance 209

Further Application of Thevenin's Theorem 212

Conventional Analysis 212

Determining the Thevenin Voltage 213

Determining the Thevenin

Impedance 213

THE NORTON THEOREM

APPLIED 217

Determining the Norton Current

Source 217

Determining the Norton Admittance 218

The Relationship Between Thevenin and Norton Circuits 219

APPLYING THE MAXIMUM POWER TRANSFER THEOREM 220

PROBLEMS 227

## 8

### **POWER AND ENERGY IN ALTERNATING CURRENT CIRCUITS 235**

INTRODUCTION 235

POWER AND ENERGY FOR SINGLE PASSIVE ELEMENTS 236

Power and Energy for a Resistor 236

Power and Energy for a Capacitor 238

Power and Energy for an Inductor 240

POWER AND ENERGY TO CIRCUITS 243

Power and Energy in an RC Circuit 243

Time Average Power 245

The Power Triangle 246

Power and Energy in an RLC Circuit 247

Power Factor Correction 251

Power and energy in a Circuit with a Transformer 253

IMPEDANCE MATCHING 257

POWER IN A CIRCUIT WITH SPECIFIED LOAD

CONDITIONS 260

PROBLEMS 263

## 9

**FREQUENCY RESPONSE OF RC, RL AND RLC CIRCUITS 271****TRANSIENT AND FREQUENCY RESPONSE OF AN RC CIRCUIT 271**

The Transient Response 271

The Frequency Response

(AC Analysis) 273

The Relationship Between the Transient Analysis and the AC

Analysis 275

The VSRC Voltage Source 276

The Input Impedance of an RC Circuit 278

The Single-Section RC Circuit

As a High-Pass Filter 280

Voltage Gain and Phase Response of the High-Pass Filter 283

A Two-Section High-Pass Filter 285

The RC Circuit as a Low-Pass Filter 287

The Band-Pass Filter 290

Frequency Responses of RC Circuit and RL Circuit Compared 293

**FREQUENCY (AC ANALYSIS) OF A SERIES RLC CIRCUIT 296**

Frequency Selectivity of the Series RLC Circuit 301

The Quality Factor  $Q$  303 $Q$  and the Reactive Voltages at Resonance 305Relationship between Bandwidth and  $Q$  305The Effect of Changing the Resistance  $R_1$  in Figure 9.11 305The Circuit Current  $I(V_1)$  307The Resistor Voltage  $V(1,2)$  307

The Capacitor and Inductor Voltages 308

 $Q$  (resonance) as a Function of  $R_1$  309**FREQUENCY (AC ANALYSIS) OF A PARALLEL RLC CIRCUIT 310**

PROBLEMS 313

## 10

**CIRCUITS WITH NONSINUSOIDAL SOURCES 327****INTRODUCTION 327****FREQUENCY COMPONENT OF A SQUARE PULSE 327**

Fourier Theorem and Series 330

An Application of the Fourier Series Theorem 331

Total Harmonic Distortion 333

The RMS Value of a Fourier Series 334

The Power Delivered by a Fourier Series 335

**FOURIER ANALYSIS OF A RESISTIVE VOLTAGE-DIVIDER CIRCUIT 337**

Square Wave with DC Voltage 339

Shifting the input Voltage Pulse

 $V_1$  in Time 341**FOURIER SERIES OF COMMON TIME FUNCTIONS 343**

The Triangular, or Saw Tooth, Wave 343

The Half Wave Rectified Sine Wave 347

The Fully Rectified Sine Wave 350

**FOURIER ANALYSIS OF THE RC LOW-PASS FILTER 353**

Rise Time of a Pulse and the Critical Frequency 356

**FOURIER ANALYSIS OF AN RC HIGH-PASS FILTER 357**

The Fall Time and the Critical Frequency 360

**FOURIER SERIES ANALYSIS OF A TUNED AMPLIFIER 360**

PROBLEMS 362

**INDEX 373**

---

## 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 ( $\Omega$ ) or kilohms ( $k\Omega$ ).

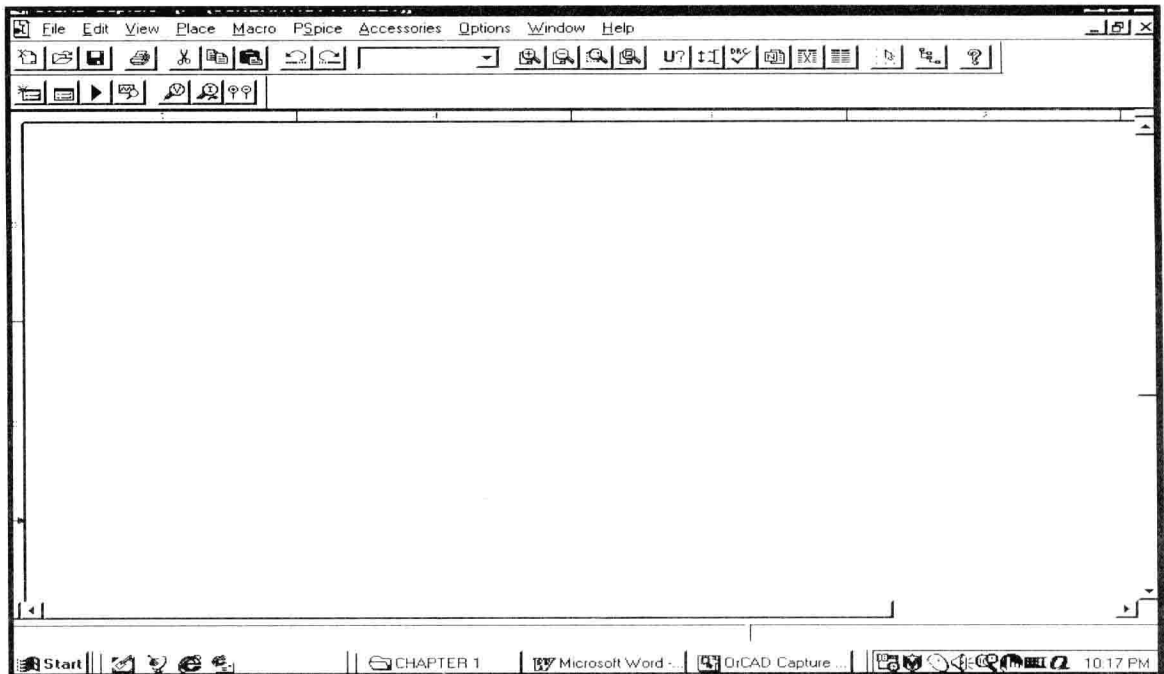
#### Preparing the Circuit Schematic

##### 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  $k\Omega$ , 4  $k\Omega$  and 5  $k\Omega$ . 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.

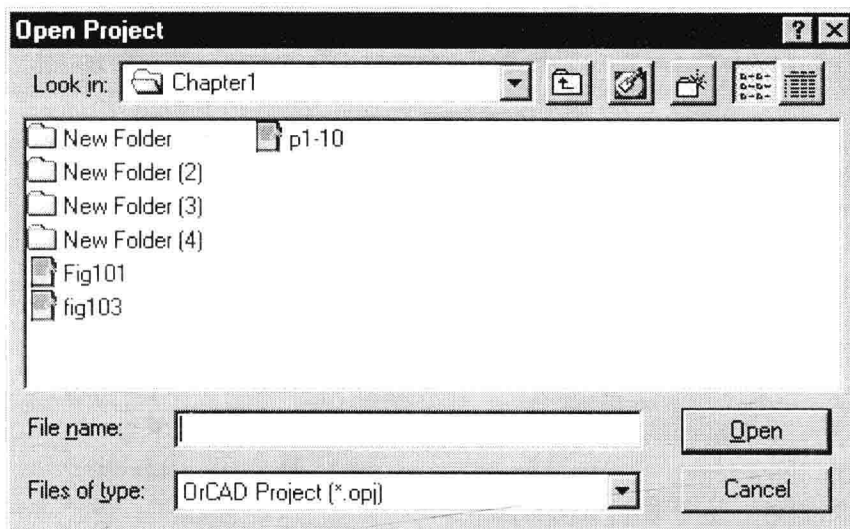
4. In the **Location** box, type information about where circuit file will reside.
5. Select **Analog or Mixed-Signal Circuit Wizard** by clicking on it.
6. Click on **OK**. The **Analog Mixed-Mode Project Wizard** dialog box opens.
7. Select analog.olb. Click on **Finish**. The **OrCad Capture [1-(Schematic1: 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.



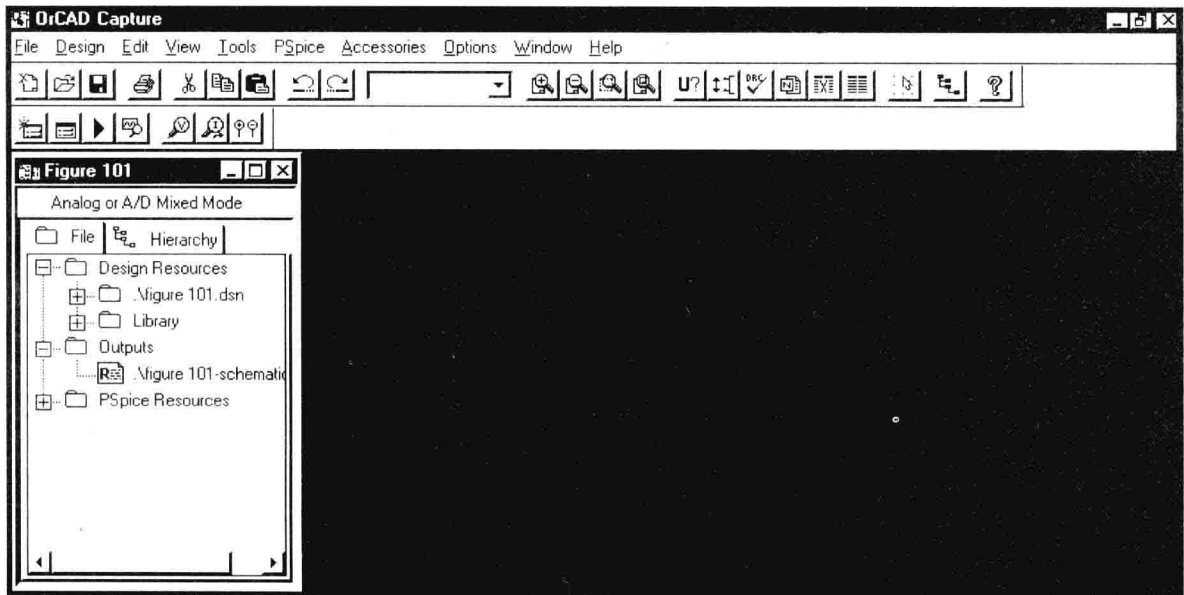
### Existing file

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 k $\Omega$ . 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.

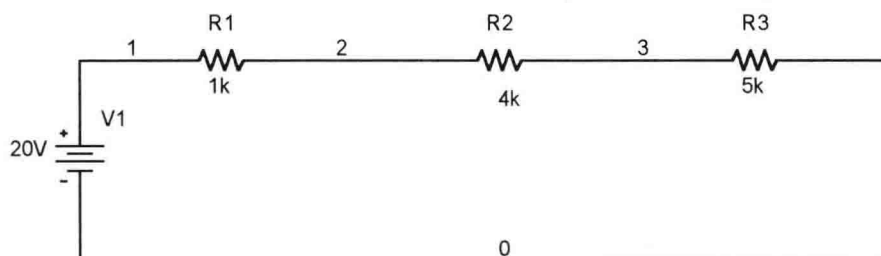
1. Double click on the voltage of **V1**. The **Display Properties** dialog box appears.
2. In the **Value** box, type 20V. Click on **OK**.
3. In **Schematic** page, click to deselect **V1**. Its new voltage of 20 volts will appear.

### 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 **VPRINT1**, 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.