THE DESIGN & DRAFTING of PRINTED CIRCUITS

by Darryl Lindsey

founder of the renowned MASTERS P.C. DESIGN SCHOOL

Published by

The Innovators
Bishop Graphics, Inc.
· *Westlake Village, California 91359*

Distributed by

McGraw-Hill Book Company

*New York St. Louis San Francisco Auckland Bogotá
Düsseldorf Johannesburg London Madrid Mexico
Montreal New Delhi Panama Paris São Paulo
Singapore Sydney Tokyo Toronto*

Distributed by

McGraw-Hill Book Company

New York St. Louis San Francisco Auckland Bogotá
Düsseldorf Johannesburg London Madrid Mexico
Montreal New Delhi Panama Paris São Paulo
Singapore Sydney Tokyo Toronto

The Innovators
Bishop Graphics, Inc.
5388 Sterling Center Drive
P.O. Box 5007
Westlake Village, California 91359

Printed in the United States of America

First printing, 1979

McGraw-Hill ISBN No. 0-07-005430-4

THE DESIGN & DRAFTING of PRINTED CIRCUITS

PREFACE

The *Use*, The *How* and The *WHY* of Printed Circuit Design are explained in this book. As a teaching tool, the book makes the reader think through problems by effectively explaining guiding principles, defining terms and outlining design and manufacturing processes.

Discussion of theories is in the light of the best commercial practices, making the book an accurate source of valuable information for the student as well as the industry practitioner.

The author discusses and weighs theories of design and their application in the commercial electronic field so that the book will serve as an organized technical aid. Clear and carefully selected illustrations will appeal to the many persons in industry who have had little opportunity to study P.C. Design.

The chapter on "Logic", provides a source of information that is difficult to find elsewhere. The understanding of logic as it relates to the P.C. Design greatly increases the value of the book as a text and as an important source of information.

The author's past experience and current state-of-the-art materials are presented in a manner calculated to help in the solution of future problems. To readers of this first edition it is the author's wish that this book will prove to be of lasting value.

ACKNOW-LEDGMENT

The author would like to express his thanks for all the assistance that has been provided for the various editions of this book. I am much indebted to former students, colleagues and friends for their time and efforts in helping prepare this book. I am indebted especially to the following: Don Coulthart, Gary Wilson and Don Sefervoich.

And finally to my wife, Priscilla Lindsey for her time and patience in helping me prepare the manuscript.

INTRODUCTION

Evolution of the printed circuit board does not "just happen". In fact, some rather serious thought must be given to each individual project before actual design work can begins. Once started, the design should proceed through six crucial phases. These phases are outlined in this introduction and are explained in detail within the corresponding section of this book.

Pre-design effort will ensure that the printed circuit board designer has a general knowledge of the project ahead of him, electrically and physically. Most of all, it will ensure that his ideas and those of the circuit design engineer coincide. Coordination between the circuit design engineer and the PCB designer *cannot* be overstressed. This communication link is of the utmost importance if the end product is going to meet *all* fit, form, and function requirements set forth.

Before work of any kind can be started, however, the circuit design engineer must prepare an appropriate "engineering" schematic. The engineering schematic is transferred to the PCB designer, and phase I of her/his work begins at this point. All too often, the schematic phase is not approached "head-on" and the remaining design phases suffer accordingly. To insure that you get off on the right foot, put that engineering schematic at the *top* of your "list of tools".

Printed circuitry and the printed circuit board, what are they? For years, printed circuïtry and printed circuit boards have been defined and re-defined within the electronics industry. The following are definitions that have been extracted from leading electronics dictionaries:

> "A circuit in which the interconnecting wires have been replaced by conducting strips printed, etched, etc. onto an insulating board. It may also include similarly formed components on the base board."
>
> "Also called a card, chassis or plate. An insulating board onto which a circuit has been printed."

Of course, these definitions assume that the reader has had prior experience with point-to-point wire soldering. For those unfamiliar with the process, it simply required hand-soldering insulated wires between component leads where electrical connection was required. Simple it was. However, it was also time consuming and the end product was complicated, bulky, and truly a production nightmare. Understandably, compactness, efficiency in assembly, and standardization have since become the goal of electronic industries in the United States and abroad.

In contrast to modern electronic components used today, the "printed wiring board" (as it was formerly known) included designs with bulky, high-heat producing components such as vacuum tubes, large oil-filled capacitors and transformers. Miniaturization and micro-miniaturization seen in today's electronic components (such as diodes, transistors, integrated circuits, capacitors, resistors, and transformers) have played a leading role in speeding up and developing the techniques and standards employed in the production of today's printed circuit boards. This book is dedicated to *these* techniques and standards.

The author prefers to define the modern-day printed circuit board as follows:

> "The replacement of hand-soldered, point-to-point wire connections with thin lines of copper. These copper lines are affixed on one or both sides of flat, rigid, glass-epoxy insulating boards through various processes including photography and chemical etching. The board facilitates the rapid assembly of active, passive, discrete, non-discrete, and hybrid electronic components with the following result: A single compact assembly where ease of assembly, maintenance and reliability are an order of magnitude better than ever before possible."

TABLE OF CONTENTS

Chapter

1

SCHEMATICS

Understanding schematics is an essential part, and should be considered primary, in the PCB design process. These schematics are written in a language foreign to the beginner, and are sometimes confusing to the experienced PCB designer. An attempt by the beginner to read a schematic would undoubtedly end up with chaotic results. Therefore, the PCB design process must be accomplished with a complete understanding of the schematic language.

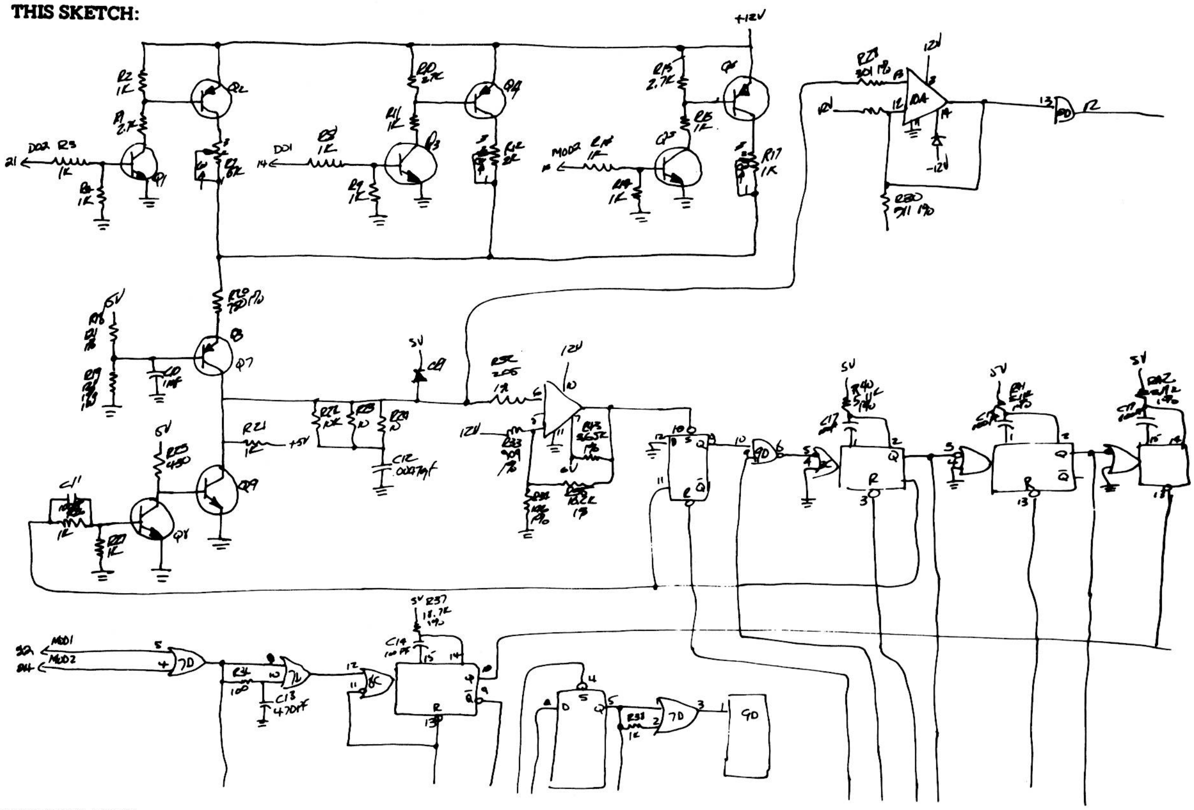
Generation of the schematic is an interactive process that starts during the conceptual design phase and continues until the design is completed. Table 1-1 lists the steps that occur during the schematic development cycle. A typical sketch and the resulting schematic are shown in Figure 1-1.

The schematic diagram is composed of symbols that represent electrical and electromechanical components plus lines that represent conductors which interconnect components. The components are assigned reference designations to differentiate between similar symbols. The schematic of Figure 1-2 shows connecting lines, symbols and reference designations.

What Happens During the DESIGN CONCEPT PHASE	What Happens During the SCHEMATIC REDRAW PHASE
A. Rough schematic sketch by design engineer. B. P.C. Board in a breadboard design phase. 1. Used by engineer to layout breadboard of circuit(s) under development. 2. Usually lacks; neatness, connector pins, and final components to be used. 3. Usually is *not* under charge control, therefore must not be used by test department or by field service department. 4. Sometimes the circuit drawings are contained on many sheets of paper during the design phase. Try to visualize understanding a circuit under this constraint. 5. It's typically used for layout of the Printed Circuit Board.	A. Clean-up and redraw by draftsman or printed circuit board designer. B. P. C. Board final design. 1. Purpose for this phase: Clean up schematic, standardize on symbols, develop good flow patterns, assign connector pin numbers, condense multiple sheets into a useable document. 2. A clean schematic will assure that the PCB design phase is more successful. 3. Allows the PCB designer to get an idea of the components needed while removing unnecessary or component overage that were included in the breadboard phase. 4. Release documentation package, with schematic, to Test, Publications, Field Service Dept. for their respective uses.

TABLE 1-1 Schematic Development Cycle

THIS SKETCH:



BECOMES THIS:

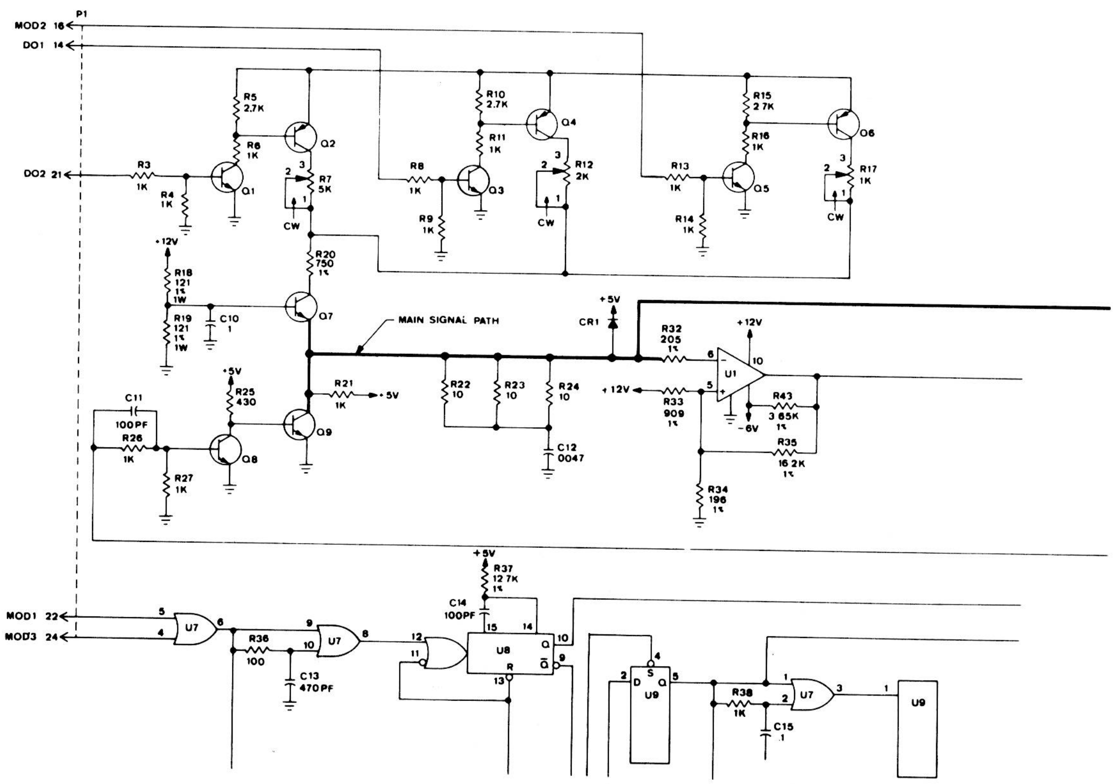


FIGURE 1-1

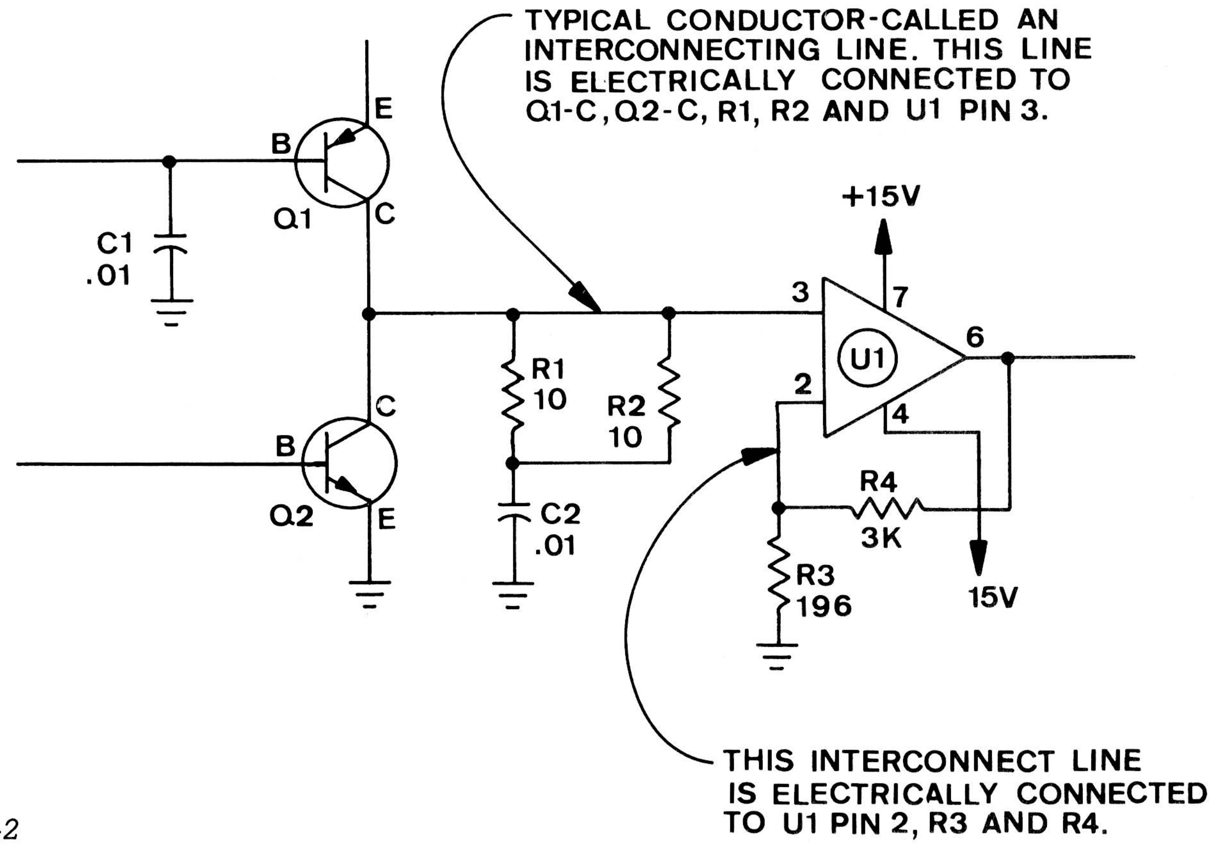


FIGURE 1-2

ABBREVIATIONS AND SYMBOLS

Reference designations are combinations of letters and numbers used to identify the components. The letters identify the type of electrical symbol and component such as: (Resistor -VVV- = R) and (Capacitor -|(= C). The number identifies the differences between electrical components of the same type: R1, R2, R3, C1, C2, C3, etc. All schematics using standard commercial practices should be drawn from *left* to *right*, input on the left side of the sheet and output on the right side. All reference designations will be numbered from left to right and top to buttom. In general, schematic diagrams should be arranged so that they can be read functionally from left to right. The overall result should be a schematic circuit layout (and reference designations) which follows the signal or transmission path from input to output.

<u>Component Definitions.</u>

Graphic symbols for electrical components are a form of shorthand and used to show the functioning of a part of a circuit. Many companies have standards of their own that are usually based on a combination of mili- tary and commercial source, Military MIL- STD-15, Industry ASA and American Standards Association Y14.5. It would be impractical to include all of the military and industry standards. Only the most commonly used symbols in schematics for printed circuit boards are described below.

CAPACITOR

Reference Designation: C **Electronic** Symbol: -||- or -|(A device consisting of two conducting surfaces separated by an insulating material or dielectric

ABBREVIATION	ITEM SYMBOL	ABBREVIATION	ITEM SYMBOL
AR	Amplifier	M	Gage or Meter
AT	Attenuator	MK	Transmitter, Microphone
B	Blower, Fan	MT	Transducer
BT	Battery	P	Connector Plug
C	Capacitor	PS	Power Supply
CB	Circuit Breaker	Q	Transistor
CP	Connector Adapter	R	Resistor—Variable or Fixed
CR	Diode	S	Switch
DC	Directional Coupler	T	Transformer
DS	Indicator—Lamp or Light	TB	Terminal Board
E	Terminal Post	TP	Test Point
F	Fuse	U	Integrated Circuit (IC)
FL	Filter	V	Vacuum Tube
G	Generator	VR	Voltage Rectifier
J	Jack-Connector Receptacle	W	Busbar or Cable
K	Relay	X	Socket, Fuse Holder or Lampholder
L	Inductor or Coil		
LS	Loudspeaker	Y	Crystal

TABLE 1-2 Abbreviations for Reference Designations

such as air, paper, mica, glass, plastic, film or oil. A capacitor stores electrical energy, blocks the flow of direct current, and permits the flow of alternating current, to a degree, dependent upon the capacitance and the frequency.

a.
b. C1
c. .)1uf
d. 25V
e. ±10%

VALUE EXPRESSED IN:

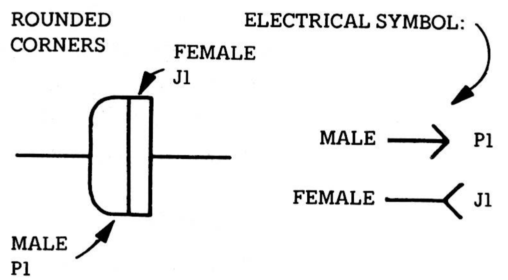
Farads (f) - most general applications (especially in P.C. boards) are in microfarads (μf) or pico forads (pf).

Tolerance: The tolerance reflects the amount that the capacitor is allowed to deviate from the value shown on the schematic. Most common tolerances for P.C. board applications are ±10% and ±20%.

Rating: Capacitors are rated by the amount of voltage that they are able to withstand safely. Capacitor physical sizes (of the same material) are controlled by their voltage ratings with most common values within a voltage rating all being the same size. Most common voltage ratings for P.C. board applications range from approximately 4 volts to several thousand volts.

Material: Capacitors used for P.C. Board mounting are most always constructed, externally, using metal, or paper. Internally, these capacitors use materials such as mica, foil, and mylar. There are various other methods of construction.

CONNECTOR PIN

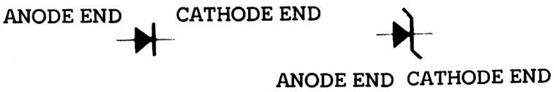


CRYSTAL

Reference Designation: Y *Electronic Symbol:* Generally employed to generate a signal with an accurate frequency output. Can also be used to convert pressure to an electrical signal, as can be done with piezo electric crystal transducer.

DIODE

Reference Designation: **CR** *Electronic Symbol:* A two-terminal device which will conduct electricity more easily in one direction than in the other.



DIODE ZENER DIODE

FUSE

Reference Designation: **F** *Electronic Symbol:* Two types of fuse symbols could be used. The first symbol represents the fuse element only. The second represents the complete fuse.

CHASSIS GROUND

Electronic Symbol: Indicates a connection made to the chassis.

COMMON GROUND

Electronic Symbol: Indicates a connection to ground.

INDUCTOR

Reference Designation: **L** *Electronic Symbol:* Used to introduce inductance into an electrical circuit. The inductor is wound into a spiral, or coil to increase its inductive intensity. It may employ an iron or air core.

RESISTOR

Reference Designation: **R** *Electronic Symbol:* A device which resists the flow of electric current in a circuit. There are two main types; fixed and variable.

a.
b. R1
c. 200 Ohms
d. ¼W
e. ±5%

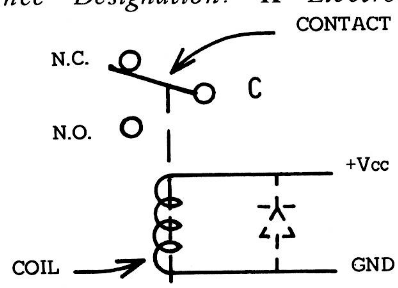
Electronic Symbol: (See Item a)
Reference Designation: R (See Item b)
Value Expressed In: Ohms - (See Item c)
Tolerance: The tolerance reflects the amount that the resistor value is allowed to deviate from that shown on the schematic. Most common tolerances used in P.C. board applications are ±.1%, ±.5%, ±1%, ±2%, ±3%, ±5% and ±10%.

Rating: Resistors are rated by the amount of wattage they are able to dissipate in a circuit. Resistor sizes (of the same material) are controlled by the wattage. The most common ratings for P.C. board applications are 1/10W, 1/8W, 1/4W, 1/2W and 1W (See Item d).

Material: Resistors are made from a variety of materials, but most commonly used on P.C. boards are carbon, wire-wound, and metal film.

RELAY

Reference Designation: **K** *Electronic Symbol:*

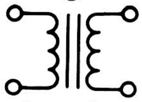


An electromechanical device with contacts that are opened and closed by a magnetic field, derived from a permanent magnet or electromagnet.

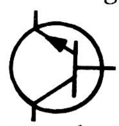
Rating: Relays are rated by (a) the amount of current that the contacts are designed to carry and i.e. (b) the voltage range at which the coil is designed to operate (i.e., 5V, 10V, 15V, and 100V).

NOTE: To protect the voltage source to the coil (especially in a transistor supply), a diode should be connected across the coil of the relay and physically located as near the coil as possible.

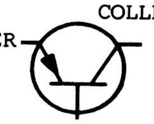
TRANSFORMER

Reference Designation: T *Electronic Symbol:*  *An electrical* device which, by electromagnetic induction, transforms electric energy from one or more circuits to one or more other circuits.

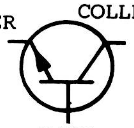
TRANSISTOR

Reference Designation: Q *Electronic Symbol:*  A semiconductor device usually made of silicon or germanium having three or more electrodes. It may be used as an amplifier, signal generator, or to perform logical functions.

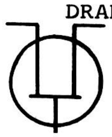
P-N-P Transistor (Positive-Negative-Positive)



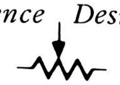
N-P-N Transistor (Negative-Positive-Negative)



FET (Field Effect Transistor)



VARIABLE RESISTOR

Reference Designation: R *Electronic Symbol:* 

ZENER DIODE

Reference Designation: VR *Electronic Symbol:*  A device that above a certain reverse voltage (the Zener value) has a sudden rise in current. If forward-biased, the diode is an ordinary rectifier.

The orientation of a symbol on a schematic drawing does not change the meaning of the symbol or its electrical function. See Fig. 1-3.

CONDUCTOR LINES

Interconnecting lines that represent the conductors on a schematic do not change weight to show differences of electrical changes. For example: voltage lines, ground lines and signal lines. However, some companies show heavier lines to emphasize the main signal path of a circuit. (See Figure 1-1)

SCHEMATIC ORGANIZATION

To avoid schematic confusion, do not spread the circuits out. Most schematics are made up of separate groups of transistors, resistors, and capacitors, etc. These groups or circuits should be layed out on the schematic drawing physically close to each other within the same group or circuit. Try not to intermix groups or circuits.

One very important rule to remember is that electrical components do not have to be shown in their physical, or actual P.C. board layout configuration.

CONDUCTOR LINES

In complex schematics, the interconnecting or conducting lines can present interpretation problems. To help eliminate the problem of hard-to-read schematics, a coding method is used. Figure 1-4 depicts use of the ground symbol to show interconnections to a common point. Parallel interconnecting conductor lines might be coded in the example of Figure 1-5.