

THE ART OF ELECTRONICS

Second Edition

Paul Horowitz HARVARD UNIVERSITY

Winfield Hill ROWLAND INSTITUTE FOR SCIENCE, CAMBRIDGE, MASSACHUSETTS



PUBLISHED BY THE PRESS SYNDICATE OF THE UNIVERSITY OF CAMBRIDGE
The Pitt Building, Trumpington Street, Cambridge, United Kingdom

CAMBRIDGE UNIVERSITY PRESS
The Edinburgh Building, Cambridge CB2 2RU, UK <http://www.cup.cam.ac.uk>
40 West 20th Street, New York, NY 10011-4211, USA <http://www.cup.org>
10 Stamford Road, Oakleigh, Melbourne 3166, Australia
Ruiz de Alarcón 13, 28014 Madrid, Spain

© Cambridge University Press 1980, 1989

This book is in copyright. Subject to statutory exception
and to the provisions of relevant collective licensing agreements,
no reproduction of any part may take place without
the written permission of Cambridge University Press.

First edition published 1980
Second edition published 1989
Reprinted 1990 (twice), 1991, 1993, 1994 (twice), 1995, 1996, 1997,
1998 (twice), 1999

Printed in the United States of America

Typeset in Times

A catalog record for this book is available from the British Library

Library of Congress Cataloging in Publication Data is available

ISBN 0 521 37095 7 hardback

TO CAROL, J.

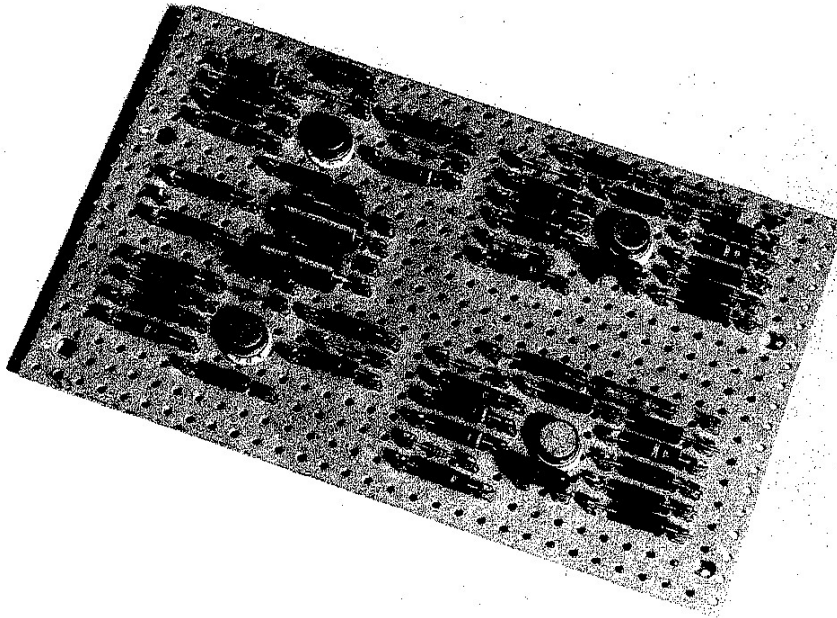


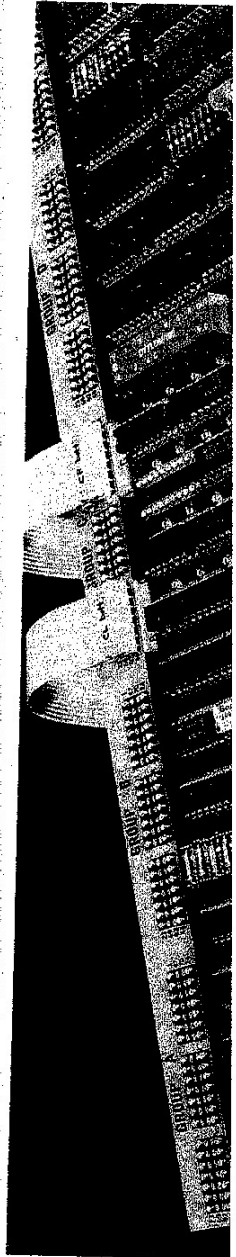
Figure 12.2. "Perfboard" can be handy for prototype circuits constructed with discrete components, although it is not particularly good for ICs. The terminals are press-fit into the holes (or flared with a special tool) and wired underneath.

of Wire-Wrap board is available with the pins on the *component* side of the board (the usual procedure is to have the pins stick out the other side). Although this type of board is less dense (fewer ICs per square inch), it is easier to use with discrete components, since you can see them while wrapping, and it allows closer spacing between adjacent circuit boards, since both components and Wire-Wrap pins take space on the same side. This kind of board without sockets is actually quite convenient for construction of linear or digital circuits. Figure 12.4 shows an example. In Figure 12.5 we have compared a prototype circuit built on a Wire-Wrap panel with the final printed-circuit version used in production. Printed circuits are much easier to produce in quantity; they are superior electrically and less cluttered than Wire-Wrap panels. We will talk about PC cards next.

PRINTED CIRCUITS

12.04 PC board fabrication

The best method of constructing any electronic circuit in quantity is to use a printed circuit, a stable insulating sheet of material with thin plated copper lines bonded to the sheet forming the circuit paths. Although early printed circuits were associated with poor reliability (Remember the advertisements stressing the superior quality that only handcrafted television sets without printed circuits could provide?), the process of manufacturing board material and producing finished boards has been perfected to the point that printed-circuit boards now have very few problems. In fact, PC boards offer the most reliable fabrication technique. They are routinely used in computers, spacecraft, and military electronics where high reliability is essential.



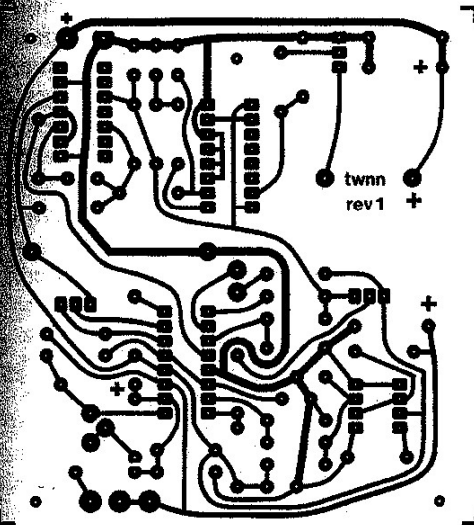


Figure 12.6. Foil pattern for a simple single-sided PC board. This "layer positive" was photographically reduced from a double-sized hand-taped Mylar pattern. The circuit board is 3.25×3.5 inches and has 4 ICs, 24 resistors, 11 capacitors, 5 diodes, 1 trimmer, and 1 piezo buzzer.

The "Mylar" or photoplot

Printed-circuit-board production begins with a set of actual-size transparencies on which an opaque pattern delineates the desired circuit traces and "pads" (Fig. 12.6). There are lots of rules and tricks in this business, but the basic idea is to figure out how to make all the interconnections the circuit demands by running lines around a board. As we will describe shortly (Section 12.08), these transparencies are usually produced directly on film by computer-driven photoplotters or laser plotters, working from a description of the circuit that you produce on a CAD (computer-aided design) system. However, for simple circuits, you may choose to do the layout manually, sticking opaque tape and patterns onto clear Mylar film. In the latter case you usually make the taped Mylar pattern twice actual

size, from which you photographically produce an actual-size transparency.

No matter what the parentage of the final 1:1 transparency, the result is a set of conductor patterns on film. For the simplest boards you may be able to make all the needed connections (perhaps aided by a few wire "jumpers") with a *single-sided* board, which has all its traces on the underside (officially called the "solder" side; the top is called the "component" side). Most often, however, you need traces on both sides of the board. Such *double-sided* boards invariably use *plated-through* holes (the holes in the board are lined with foil, connecting corresponding pads on both sides of the board). This really makes a difference when you are attempting to route traces, because you can always switch sides (using a "via" hole) when a trace runs into a dead end, whereas with a single-sided board you often get hopelessly boxed in. As an important side benefit, the use of plated-through holes ensures a superior solder joint to the component leads, because the solder wicks up through the hole.

For complex digital circuits you often see laminated multilayer PC boards, in which interior layers (called the "core") are used to carry ground and power-supply voltages, and sometimes signal lines as well. Four-layer and six-layer boards are pretty routine these days, with occasional use of more layers (up to 40!) in desperate situations.

Manufacturing

The board material (usually 1/16 inch of so-called FR-4 board, a fire-resistant epoxy-bonded fiberglass) comes clad on both sides with copper ("2 ounce" thickness is standard; the copper is 0.0027 inch thick). The first step is to drill the holes, using a template or automated drilling machine keyed to the full-size photopositive from the photoplotter or the Mylar

pattern. The holes are then "plated through" by a tricky multistep copper plating process, creating continuous conducting paths from one side of the board to the other.

The next step is to create a tough "resist" material, adhering to both sides of the board everywhere except where the foil for the circuit is to remain. This is done by (a) coating the board with a light-sensitive film (usually a thin adhesive "dry film"), then (b) exposing the board to light with the full-size photopositive accurately sandwiched on top, and (c) chemically "developing" the film (as in conventional photography) to make the exposed areas permanent. A step analogous to photographic "fixing" then removes the unexposed film, in precisely the pattern that will ultimately become the circuit traces. Then the board, with the pattern of resist now masking the areas where the copper foil is ultimately to be removed, is immersed into a solder-plating bath. The result is to plate solder (a tin/lead alloy) everywhere that the foil pattern is to remain, including the insides of the holes.

Next the resist is removed chemically, exposing the copper that is to be removed, and the board is treated with a copper-etching compound. That leaves the desired pattern of solder-plated copper, complete with plated-through holes. At this point it is important to carry out a step known as "reflow soldering," which consists of heating the board to make the thin solder plating flow. This prevents the formation of tiny slivers of metal (from the undercutting action of the etching bath) that could otherwise cause conductive bridges. Reflow soldering also improves the solderability of the finished board; a reflow-soldered board is a delight to "stuff" with components.

The next step in board manufacture is to electroplate the edge-connector fingers with gold. The final process in board manufacture consists of applying a tough

"soldermask" coating over the entire board, covering all of the foil except for the pad areas. This greatly reduces the tendency to form "solder bridges" between closely spaced traces during subsequent soldering operations. It also makes the board moisture- and scratch-resistant. Soldermask materials can be applied by silk-screen methods ("wet mask") or by the same photographic "resist" methods used to create the foil circuit pattern ("dry mask"). You can recognize the soldermask by its dark green color and by the observed fact that it is practically impossible to remove. In industrial board manufacture, the board might then be stuffed by automatic machines, with all joints soldered in a few seconds in a "wave-soldering" machine. The alternative is to stuff and solder by hand.

There is a simpler process of board manufacture that is sometimes used, especially in small production situations or for single-sided boards, where plated-through holes aren't needed. In this method you begin by coating the board with photo-resist, then expose it through a full-size *negative* (rather than positive) of the desired pattern, i.e., a photographic film that is transparent wherever you want foil to *remain*. The resist is developed, and then the unexposed resist is dissolved away. This board then has a layer of tough resist covering the copper that is to remain, so you simply expose it directly to the etching compound (omitting the solder-plating step, above). After the superfluous copper has been etched away, the remaining resist is washed off with solvent, leaving the desired pattern in copper. At this point it is best to treat the board with an "electroless" tin-plating bath in order to cover the copper with a metal less susceptible to corrosion. As before, the edge-connector fingers will then be gold-plated. The final step in this process consists of drilling the holes by hand, using the actual conductive pattern as a guide (each "donut" pad has a small

opening in the center of the finished board).

□ 12.05 PC board

There are several ways to make a printed circuit board. In this section we will discuss the most important ones.

□ PC board layout

To make a printed circuit board, you ultimately convert a schematic diagram into a corresponding copper-foil traces on a finished board. There are two ways: (a) Working from a schematic diagram, you draw schematics, then use a computer to convert the schematic into a set of interconnect coordinates. The computer does the job, then you use a tape and preform to transfer the coordinates accurately onto the board. (b) You make the finished board by converting the schematic to a connection table. CAD (computer-aided design) software figure the trace route and place the components precisely; better still, you can use a visual schematic drafter to capture the diagrams directly into the computer station, using a graphics interface.

CAD-based circuit board design (option b, above) includes automatic documentation for design and layout. It gives you the ability to produce boards with precise traces. It is the nearly all complex board production.

the entire
il except for
reduces the
ges" between
subsequent
makes the
esistant. Sol-
plied by silk-
) or by the
ethods used
attern ("dry
e soldermask
the observed
ossible to re-
manufacture,
ffed by auto-
ts soldered in
ldering" ma-
uff and solder

of board man-
ed, especially
s or for single-
through holes
thod you be-
a photo-resist,
l-size *negative*
e desired pat-
a that is trans-
oil to *remain*.
then the unex-
ty. This board
esist covering
a, so yqu sim-
etching com-
r-plating step,
us copper has
aining resist is
aving the de-
this point it is
in "electroless"
cover the cop-
ptible to corro-
n-connector fingers
he final step in
ing the holes by
ductive pattern
ad has a small

opening in the center to aid in drilling the finished board).

12.05 PC board design

There are several important decisions you have to make during PC board design, during component "stuffing," and finally when the board is used in an instrument. In this section we will try to touch on the most important of these.

PC board layout

To make a printed-circuit board, you must ultimately convert your schematic diagram into a corresponding pattern of desired copper-foil traces that will compose the finished board. There are basically two ways: (a) Working from the manually drawn schematics, you can use pencil and paper (and lots of erasers!) to figure out a set of interconnection paths ("routes") that does the job, then manually apply opaque tape and preformed connection patterns accurately onto clear polyester film to make the finished "Mylar" masters, or (b) you can convert your hand-drawn schematic to a connection "netlist," then use a CAD (computer-aided design) program to figure the trace routing, producing a set of precision machine-drawn "photoplots" directly; better still, you can replace the manual schematic drafting with CAD-based "schematic capture," in which you draw the diagrams directly on a graphics workstation, using a graphic tablet or mouse.

CAD-based circuit drawing and layout (option b, above) has many advantages, including automatic netlist extraction, painless documentation, the ability to check for design and layout rule errors, the ability to make changes relatively easily, and the ability to produce complex multilayer boards with precise alignment of pads and traces. It is the method of choice for nearly all complex and high-density PC board production. However, we would like

to begin simply, by briefly describing manual methods of PC board layout (option a, above). Once you know how to do a PC board layout by hand, you will understand what you're trying to make with the complex computer-based CAD tools. Furthermore, for simple jobs the manual methods may be all you need, and they are cheaper (and sometimes quicker) than CAD-based methods. They are particularly appropriate for the uncrowded boards you might use in simple unsophisticated instruments, of the sort you might build in small quantities for research laboratory use. They are also well adapted to boards containing parts with unusual shapes and lead spacings. Later, in Section 12.08, we will describe the CAD methods that are mandatory for high-density digital multilayer boards intended for large production.

There are several stages along the way from a schematic diagram to a final printed circuit. Beginning with the diagram, you generally work out trial pencil sketches of component layouts and interconnections, eventually working these together into a final pencil layout drawing. From this you make the "Mylar," consisting of accurately aligned "pads" (terminal areas for component connections) and taped interconnections. Precut patterns are used for IC and transistor pads and for ribbon and edge connectors, since these have standard spacings and dimensions. The pencil sketch and Mylar are usually made double size to allow greater accuracy (and to keep your eyes from popping out!). When the Mylar (two Mylars for double-sided boards) is completed, it is photographically reduced to an actual-size negative, from which a trial board is made as described previously. You generally "stuff" the prototype board with components, turn on the power, and then hunt down the errors; this lets you correct the Mylar artwork to produce final boards. The following subsections provide some further details and hints.

□ Initial sketch

We recommend doing the initial layout with pencil on grid paper (5 lines/inch), with two colors to indicate foil patterns on the top and bottom (assuming it is a double-sided board). We usually use black pencil for runs on the bottom and green or red for the top (component) side. Since you're likely to do plenty of erasing, it is best to use vellum graph paper. The 0.2 inch gridding corresponds to 0.1 inch final size, the universal measure for IC pin spacings, transistor pinouts, edge connectors, etc. Your drawing should be the view from the component side; i.e., the sketch of the component-side (top-side) foil pattern looks like the final pattern, and the sketch of the bottom foil pattern is what you would see looking down through the finished board with x-ray eyes. While working on the layout, indicate component outlines with a pencil of a third color. All this work should be freehand. Don't waste time with an outline template; just use the grid lines as a guide to draw IC and component pinouts.

It is generally best to work up some trial layouts on a piece of scratch paper, particularly for sections of the circuit that may require special layout to minimize long lines or capacitive coupling. It may take some experimentation to arrive at good component arrangements. A trial layout might consist of a block of the circuit with two or three op-amps, or perhaps the input or output section of the circuit. These blocks should then be worked together onto the large gridded vellum, with adjustments being made as you go. Don't hesitate to do lots of erasing!

□ Layout dimensions and hints

Try to have all ICs pointing in the same direction, preferably in straight rows. Likewise, resistors should be in even rows, not staggered. We use 0.030 or 0.040 black tape for signal runs, with wider tape for

power supplies (0.05 or 0.062 inch) and very wide ground runs (0.1 to 0.2 inch, or even wider; it's common to broaden the ground runs with lots of tape). Be sure to include plenty of bypass capacitors, one 0.1 μ F for every two to four ICs. As you scratch your head, trying to juggle the tangled maze of interconnections, don't forget that components act as "jumpers" — they can hop over runs on the board.

Dimensions and spacings: On the actual-size PC board, we recommend holes spaced 0.4 inch for resistors ($\frac{1}{4}$ W size), with spacing of 0.1 or 0.15 inch between resistors (with 0.15 inch spacing you can get a tape run between adjacent pads). We favor CK05 and CK06 types of ceramic capacitors, with their controlled 0.2 inch lead spacing, or the "DIP" 0.3 inch types (AVX type MD01, or Kemet C630C104M5U); they can also be spaced 0.1 inch from other capacitors or resistors. Leave some room around ICs for logic clips: a minimum of 0.2 inch to the next IC pads and a minimum of 0.15 inch to the nearest resistor or capacitor pads. Leave 0.030 inch spacing between tape runs, and don't run anything closer than 0.25 inch from the edge of the board, to allow room for card lifters, guides, standoffs, etc. Avoid running lines between the 0.1 inch spaced pads of an IC, unless necessary. You can fit up to six tape runs lengthwise between the pads of a standard DIP IC pattern (they're spaced 0.3 inch).

Recent PC design practice generally favors much higher line densities (both thinner lines and closer spacing) than the values recommended above; the latter would be called "15-15" design rules, signifying 0.015 inch minimum line width with 0.015 inch minimum conductor clearance. In the PC board industry, 15-15 rules are considered quite relaxed, with 12-12 pretty standard; 12-12 rules let you run one trace between adjacent IC pads on 0.1 inch centers (check for yourself that this is permitted, providing the pads aren't larger

than 0.064 inch boards often use which you can adjacent IC parameters are the spectively). On boards with 0.1 inch the designers of increased circuit between adjacent channel!) again holes and other poorer productivity.

□ Connections to

For the majority best to bring on "edge-connectors" directly with socket of contact configuration commonly used space inch, and 0.100 inch. Generally you'll put at one end of supply voltages connector. The ported, and it p on that shortly).

Often you see at the other end instead of a flat some other signal boards. Another out signals is terminated in plug right into You can buy the various lengths, yourself with a l unassembled DI tool. Ribbon cat board via in-line connectors, which pins on 0.1 inch

For simple b connection may