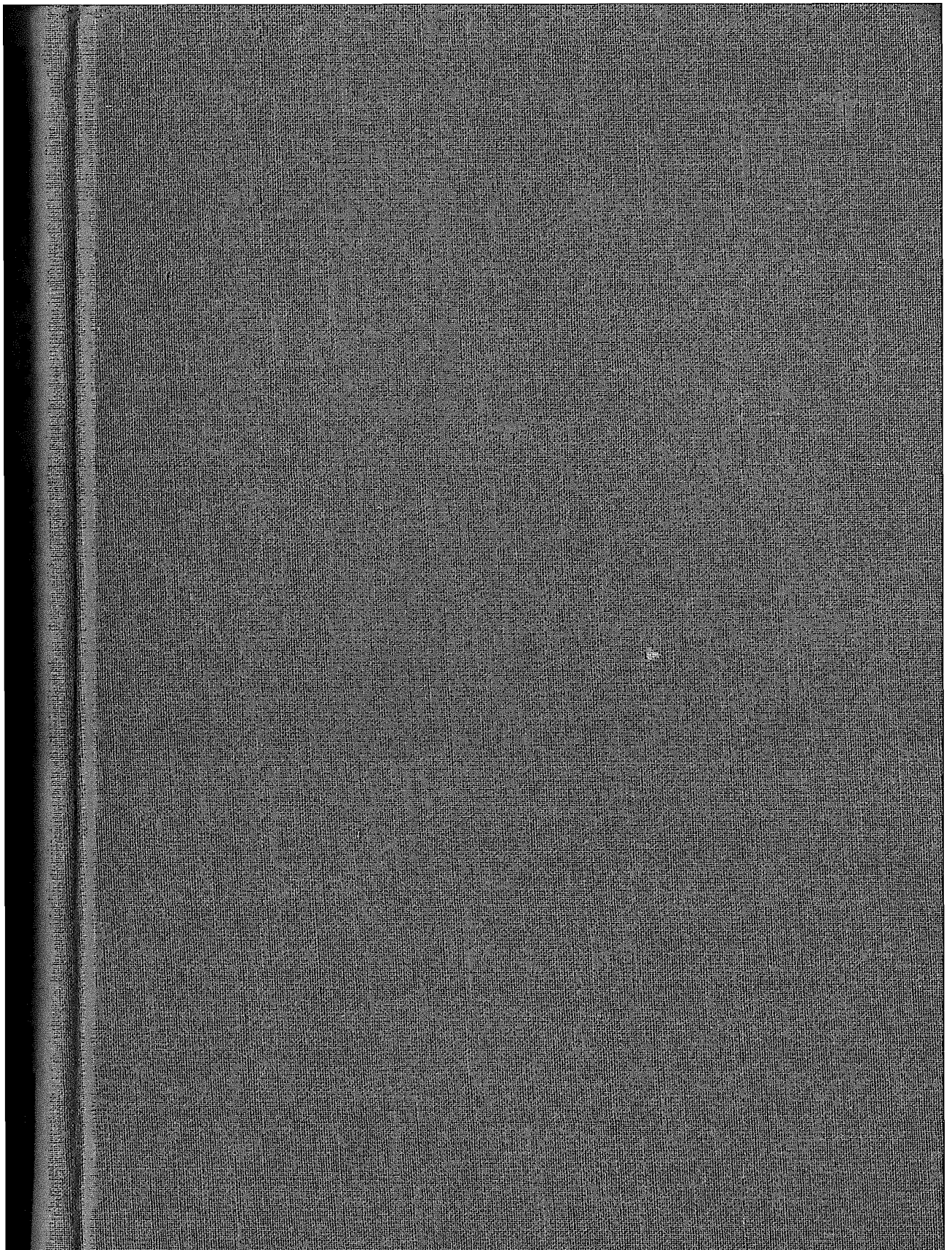
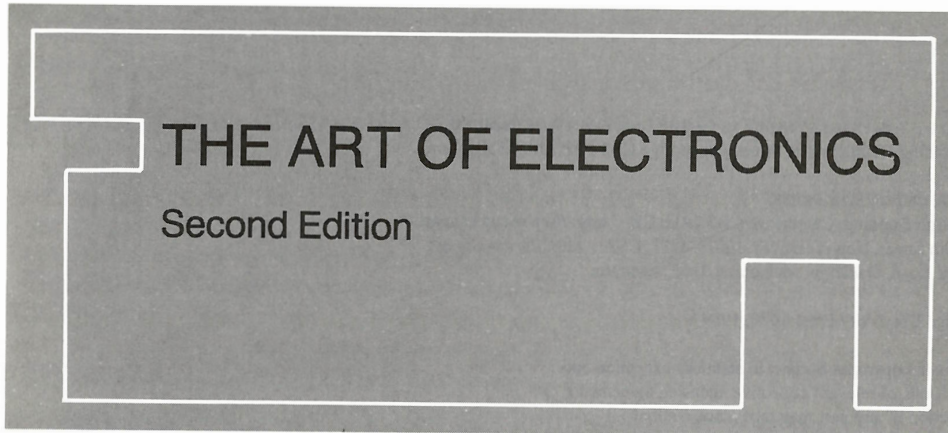




IP #67-95



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 CB2 1RP, 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

© 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)

Printed in the United States of America

Typeset in Times

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

Library of Congress Cataloguing-in-Publication Data is available

ISBN 0-521-37095-7 hardback

ELECTRONIC CONSTRUCTION TECHNIQUES

CHAPTER

12

Between completing a circuit design and testing a finished product, you've got a number of decisions to make: Is the instrument going in a benchtop case, a "relay rack" enclosure, or perhaps some sort of modular "bin" chassis? Should the circuit itself be constructed with point-to-point wiring, on a "breadboard" card, with Wire-Wrap connections, or on a printed-circuit board? Are connections to the circuit board made with solder lugs, flat ribbon-cable connectors, or edge connectors? Should individual circuit cards be housed in a card "cage," plug into a motherboard, or what? Does it pay to design a printed-circuit motherboard or use hand-wired backplane connections? Which adjustments should be on the circuit board, and which on the front (or rear) panel? Decisions like these are important for the appearance, reliability, and serviceability of the finished product, not to mention cost and ease of construction and testing. In this chapter we will try to give some information and guidance on this important subject, one that tends to be overlooked in electronics courses where circuit

work is usually done on plug-in breadboarding gadgets. We will begin with the circuit construction itself, treating interconnections, controls, and enclosures last.

Because this chapter does not deal with circuit design, it could be skipped in an abbreviated reading.

PROTOTYPING METHODS

12.01 Breadboards

The unusual name "breadboard" seems to have arisen from the early practice of building radios on handsome slabs of varnished wood, with tubes, coils, condensers, etc., and the interconnecting wires all fastened to the top side of the board. Later, radios of greater refinement and elegance (to be used by stiff-skirted ladies in the parlor) were built with holes near each component so that the wiring would be hidden from view underneath the board. The practice of testing circuits by constructing trial versions on some sort of board or jig is still called breadboarding.

Wooden breadboards are no longer used (except in the kitchen). Instead, you can get handy plastic blocks with rows of holes spaced to accommodate ICs or other components and (usually) some extra rows for distributing the power-supply voltages. These breadboards are typified by those manufactured by AP and Global Specialties, and more elaborate breadboarding boxes are made by these companies and by E&L Instruments, among others. These are intended for *testing* circuits, not for constructing permanent (even semipermanent) versions.

12.02 PC prototyping boards

To construct one-of-a-kind circuits of some permanence, probably the best approach is to use a different kind of breadboard, one of the many printed-circuit (PC) prototyping cards available with predrilled pads for ICs and other components, but with no interconnections laid out on the board itself. Instead, each component pad is connected to two or three uncommitted pads nearby, and you wire the circuit by soldering lengths of insulated wire from pad to pad. There are usually some additional lines running around the board for power-supply distribution and ground. These boards, made by companies such as Douglas Electronics, Artronics, Vector, Triad, Radio Shack, and many others, usually include a card-edge connector – gold plated “fingers” of copper board material aligned at one end of the card for easy plug-in connection to a mating PC card-edge socket.

There are several standard connector configurations, a very common one being 22 connections on each side of the card, with 0.156 inch spacing (spacings of 0.125 and 0.1 inch are also common). A connector to mate with such a card is called a 44-pin “dual-readout” PC card-edge connector. Prototyping cards are available in various sizes, accommodating from 12 to 36 or more IC packages, with some larger

computer-compatible cards available that will accept 100 or more ICs and will plug directly into small-computer mainframes. Some of these boards are single-sided, and others are double-sided with plated-through holes, a subject we will discuss shortly in connection with custom PC design. Figure 12.1 shows a small PC prototyping card (Douglas Electronics 11-DE-3) plugged into a 44-pin dual-readout socket that includes card-supporting guides (Elco 6022).

Another form of breadboard that has enjoyed considerable popularity is the so-called perfboard, a thin sheet of laminated insulating material manufactured with regularly spaced holes ($3/16$ inch is a common spacing) designed to accept little metal pins. To wire up a circuit, you shove in dozens of the little pins wherever you want, then stick in the components. You then solder wires from pin to pin to complete the circuit. Perfboard works OK, but it is awkward to use with the tight spacings of IC packages (0.1 inch between pins). Figure 12.2 shows an example.

12.03 Wire-Wrap panels

A variation on the PC breadboard is the Wire-Wrap (a registered trademark of Gardner-Denver) panel, a circuit card festooned with IC sockets (or pads), with a pin 0.3 to 0.6 inch long protruding from each socket connection (Fig. 12.3). These pins are square in cross section, typically 0.025 inch on a side, and made of a hard metal with sharp corners, plated with gold or tin. Instead of soldering to the pin, you wrap an inch of bare wire tightly around it, using an electric wire-wrapping tool (there’s an inexpensive variation known euphemistically as a “hand-operated Wire-Wrap tool”). Wire-wrapping is very fast. You just stick the stripped end of the wire into the Wire-Wrap tool, put the tool over the Wire-Wrap pin, and zip, you’re done. The standard wire used for this purpose is

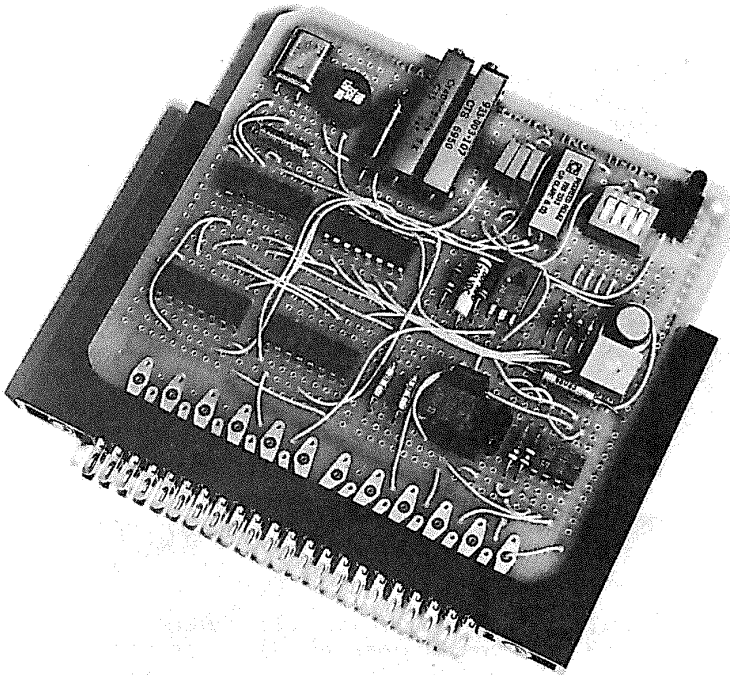


Figure 12.1. A "solder breadboard" prototyping card is useful for wiring up small circuits, especially those involving both discrete parts and ICs. This particular specimen will accommodate 12 dual in-line (DIP) packages, and it includes common lines for ground and power supply voltages. An "edge-connector" foil pattern is standard, so the card can be plugged into a card cage or supporting connector, as shown. This particular circuit uses a variety of components, including single-turn and multiturn trimmers, inductors, a crystal, a DIP switch, a miniature relay, and a logic-state indicator, in addition to both transistor and IC circuitry.

silver-plated copper wire of 26 or 30 gauge, with Kynar insulation. There are special tools available for stripping the insulation from the thin wire without nicking it. The wire is stretched tightly around the sharp corners during the wrapping process, forming a few dozen gas-tight cold welds. As a result, wire-wrapped connections are as reliable as well-soldered connections, and it is extremely easy to do them rapidly. For logic circuits, where you have few discrete components, wire-wrapping is probably the best technique for constructing one or two custom circuits of reasonable complexity.

Because Wire-Wrap panels are laid out primarily for IC packages, the technique is less convenient for linear circuits with many resistors, capacitors, etc., and the soldered prototype breadboard technique described earlier will usually be better.

It is possible to use discrete components with a Wire-Wrap panel. You just mount them on little "headers" that plug into IC sockets, then do the wire-wrapping as usual from the socket pins. Some Wire-Wrap panels have extra solderable pads (rather than IC sockets) available for discrete components. A very nice kind

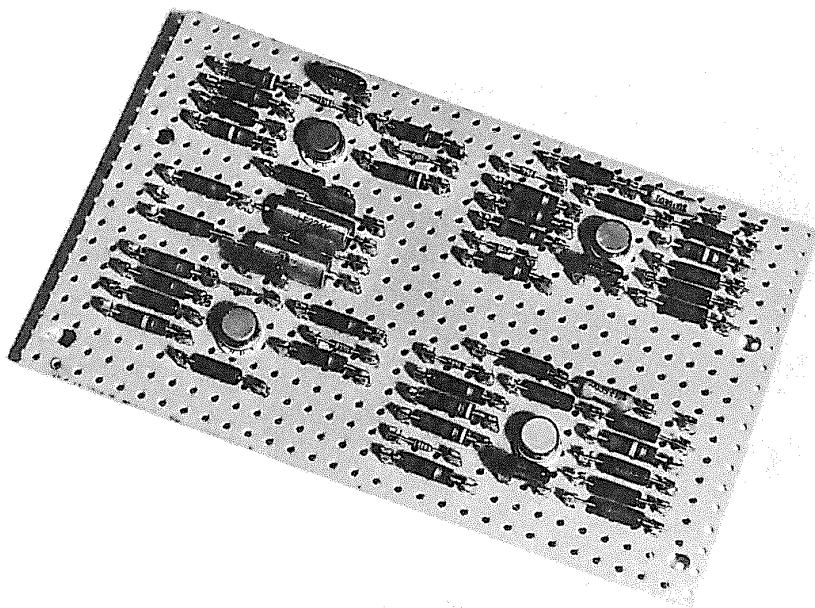


Figure 12.2. "Perfboard" can be handy for prototype circuits constructed with discrete components, although it is not particularly good for ICs. The terminals are press-fit into the holes (or flared with a special tool) and wired underneath.

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.

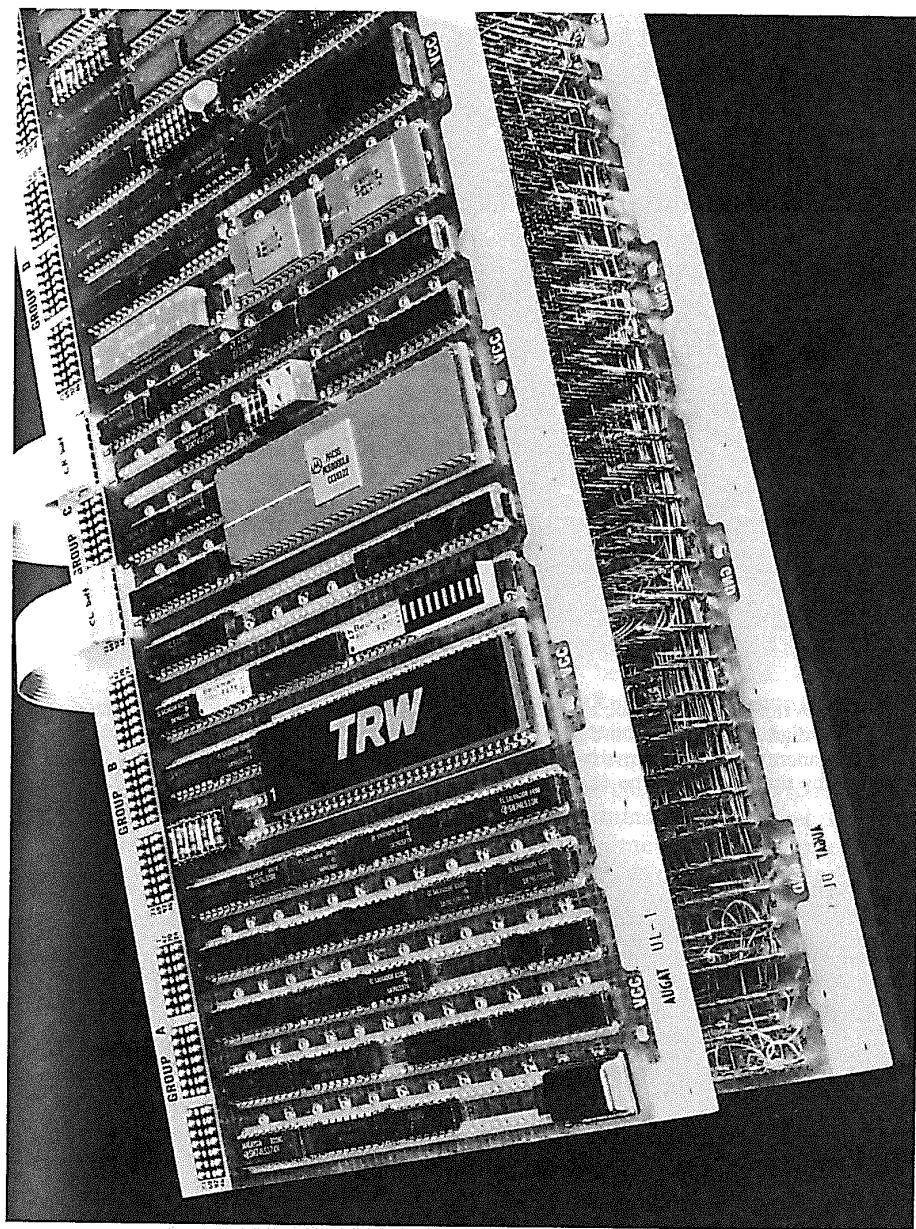


Figure 12.3. A large Wire-Wrap® panel and its underside (visible in a mirror). This microcomputer circuit was wired by machine ("CAD/CAM," see Section 12.08). Chips of different sizes, as well as component-carrying "headers," are accommodated by this general-purpose panel, which also includes an area for off-board connections via flat ribbon cable with 14-pin DIP plugs.

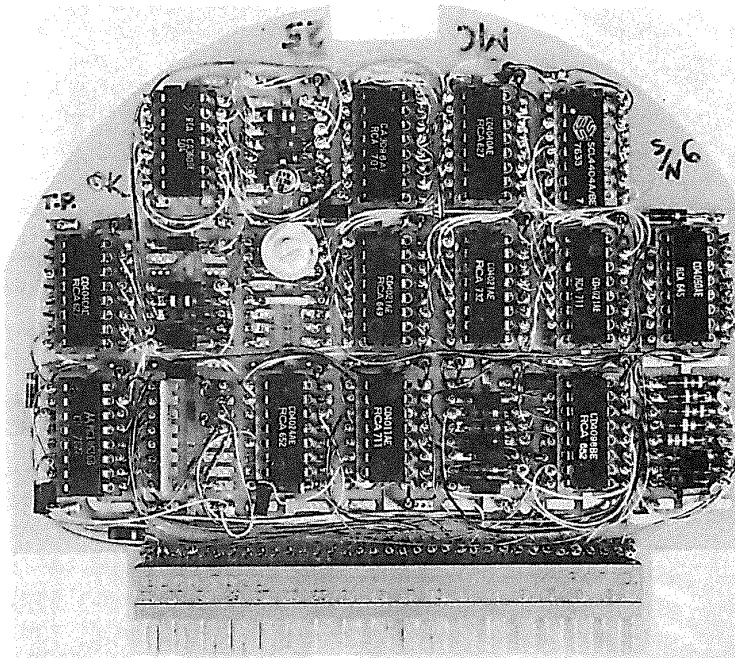


Figure 12.4. Wire-Wrap boards provide a neat and fast construction method particularly good for circuits made with digital ICs. This board uses a printed-circuit pattern to bring out the Wire-Wrap pins on the component side, an alternative to the usual underside pin configuration. Its peculiar shape is dictated by the interior of the oceanographic pressure cell into which it fits.

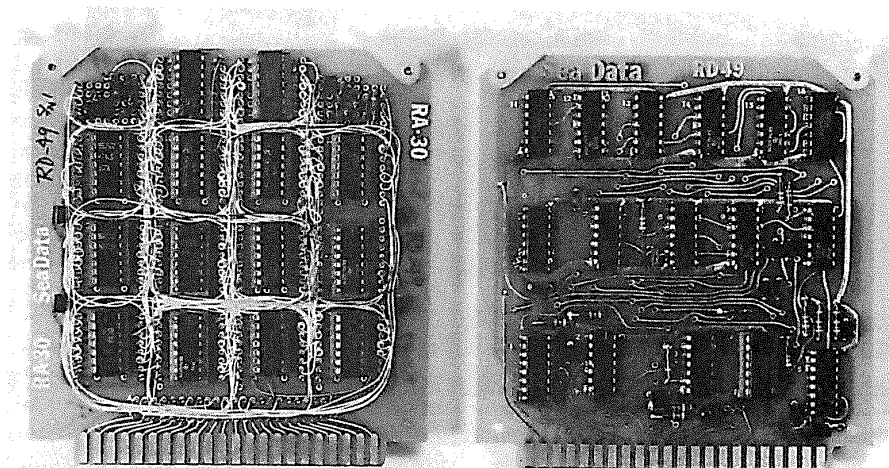


Figure 12.5. A Wire-Wrap prototype board and its printed-circuit successor. PC boards are less cluttered and far easier to fabricate in quantity. They eliminate wiring errors, too.

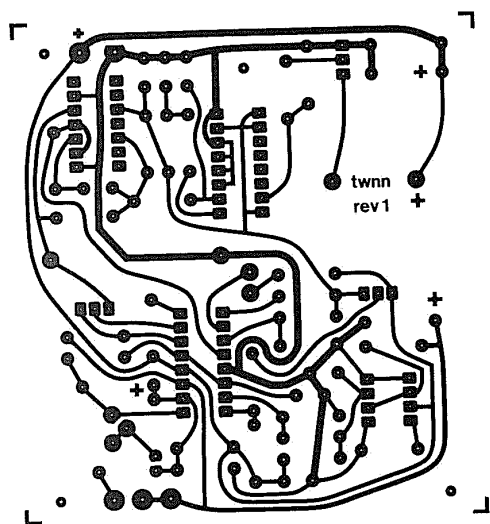


Figure 12.6. Foil pattern for a simple single-sided PC board. This "layer positive" was photographically reduced from a double-sized hand-taped Mylar pattern. The circuit board is 3.25×3.5 inches and has 4 ICs, 24 resistors, 11 capacitors, 5 diodes, 1 trimmer, and 1 piezo buzzer.

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 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 in diameter). High-density boards often use 10-10 or 8-8 rules, with which you can squeeze *two* traces between adjacent IC pads (the maximum pad diameters are then 0.050 and 0.060 inch, respectively). Occasionally you see daring boards with 0.006 inch or thinner traces; the designers of such boards are trading the increased circuit density (*three* traces between adjacent pads, 20 down the center channel!) against smaller plated-through holes and other compromises that result in poorer production yield and board robustness.

□ Connections to the board

For the majority of boards it is probably best to bring out all connections through "edge-connector" contacts, which mate directly with sockets available in a variety of contact configurations. The most commonly used spacings are 0.156 inch, 0.125 inch, and 0.100 inch between fingers. Generally you'll put an edge-connector pattern at one end of the card, bringing power-supply voltages and signals through that connector. The card is mechanically supported, and it plugs in at that end (more on that shortly).

Often you see an edge-connector pattern at the other end of the card also, used instead of a flat ribbon connector to bring some other signals off the board or to other boards. Another method for bringing out signals is to use flat ribbon cable terminated in DIP plugs; such cables plug right into IC sockets on the board. You can buy these cables prefabricated in various lengths, or you can make them yourself with a kit consisting of flat cable, unassembled DIP plugs, and a crimping tool. Ribbon cables can also connect to the board via in-line or "mass-termination" connectors, which use one or two rows of pins on 0.1 inch centers.

For simple boards the best method of connection may be to use swage-solder

terminals or PC-type barrier strips with screw terminals. Avoid the use of large pads alone for connection of external wires to PC boards.

Figure 12.7 illustrates a variety of PC board connection techniques.

□ Odds and ends

With plated-through boards, use several holes to join ground foils on opposite sides of the board. Try to avoid using multiple passes through the board to reach your destination, since plated-through connections where no component is mounted are more likely to give trouble. The layout of a double-sided board generally winds up with most tape runs going horizontally on one side, vertically on the other.

General philosophy: Use smooth curves or 45° turns, rather than right-angle turns, for hand-taped layouts. Bring lines into pads as if heading for the center of the pad, rather than coming in at an oblique angle. Don't mount heavy components on boards (a couple of ounces ought to be the limit); assume that the instrument will be dropped 6 feet onto a hard surface sometime during its life! Put polarity markings on the component side for diodes and electrolytic capacitors, and label IC numbers and pin 1 location (if there's room). It is always nice to label test points, trimmer functions (e.g., "ZERO ADJ"), inputs and outputs, indicator light functions, etc., if you have room.

□ Taping the Mylar

General advice: Use an illuminated "light table" with a piece of precision gridded Mylar taped to it. Don't confuse this with the inexpensive gridded plastic films that are neither accurate nor dimensionally stable; a piece of precision gridded film will set you back at least \$20. Put your clear Mylar over, and stick down the IC pads accurately on it. Use the pencil sketch for guidance while taping. Wash

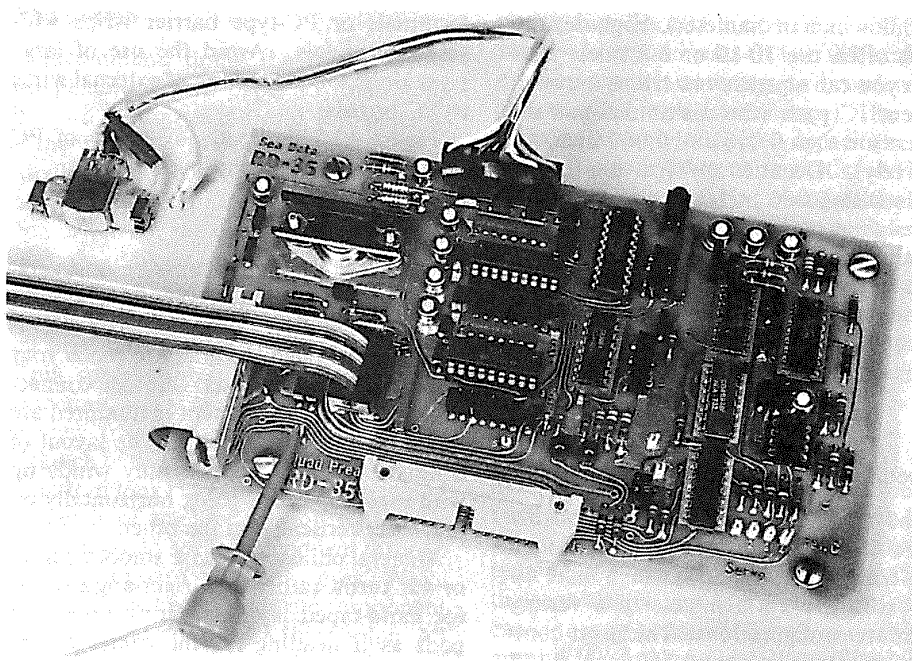


Figure 12.7. Several connection techniques are illustrated in this digital recorder printed-circuit card. The tape head connects via an in-line connector (which mates with a row of Wire-Wrap-type pins), and other signals are brought out with "mass-termination" ribbon connectors and a dual in-line ribbon connector. A test lead is shown clipped onto a "test point" terminal. This board also illustrates PC board heat sinking (upper left), a logic-state indicator (upper right), miniature single-turn trimmers, and single in-line (SIP) resistor networks.

your hands often to prevent deposition of oily film on the Mylar, and use alcohol to wipe any areas that might become oily. Use an Xacto knife with curved blade for tape and outline cutting, and learn not to cut through the Mylar. Press the tape down firmly after positioning; otherwise it will eventually curl up. Allow generous overlap where tape meets pad, etc. When laying out tape, don't hold it under tension; it will shrink and pull away from pads. Use precut bends and circles for the larger tape widths (0.062 inch or wider) when navigating tight turns. After the Mylar is completely taped, check it against the schematic by going over each connection on the diagram with a red pencil. When all seems OK, seal up

flaws on the Mylar with an indelible black felt-tip pen.

Precut PC graphics patterns are available from several manufacturers. Table 12.1 shows some recommended types. The Bishop Graphics catalog (5388 Sterling Center Drive, Westlake Village, CA 91359) includes extensive information on PC board layout and execution.

12.06 Stuffing PC boards

Your worries aren't over when you've got a finished board. You've got some decisions to make (e.g., whether or not to use IC sockets) and some important things to do (e.g., defluxing and lead trimming). Herewith, some thoughts on these subjects.

TABLE 12.1. SELECTED PC GRAPHIC PATTERNS

Pattern ^a	Bishop	Datak
Small pads (0.150" OD)	D203	JD-145
Medium pads (0.187" OD)	D104	JD-146
Large pads (0.250" OD)	D108	JD-150
Giant pads (0.300" OD)	D293	JD-343
0.150" thermal relief – pos	5272	JDS-532
0.150" thermal relief – neg	5278	–
0.187" thermal relief – pos	5232	–
0.187" thermal relief – neg	5238	–
16-pin DIP	6109	JD-64
16-pin DIP with in-betweens	6946	JD-179
20-pin DIP	6999	JD-575
20-pin DIP with in-betweens	–	JD-585
28-pin DIP	6904	JDS-398
28-pin DIP with in-betweens	–	JDS-591
TO-5 transistor	6077	–
TO-18 transistor	6274	JD-88
TO-92 transistor	–	JD-91
0.100" connector pads	5004	JD-145
0.100" edge-connector strip	6714	JD-123
0.156" edge-connector strip	6722	JD-121
0.031" black tape	201-031-11	–
0.040" black tape	201-040-11	–
0.050" black tape	201-050-11	–
0.062" black tape	201-062-11	–
0.100" black tape	201-100-11	–
0.200" black tape	201-200-11	–
0.062" universal corners	CU601	–
0.100" universal corners	CU607	–
0.200" universal corners	CU609	–

(a) for 2:1 artwork originals.

Sockets

There is great temptation to use IC sockets everywhere, for ease of troubleshooting. However, if you're not careful, the sockets may well cause more trouble than they prevent. In general, sockets are a good idea at the prototyping phase, where IC substitution may be necessary to convince you that the trouble you're having is a

design error, not a bad component. They should also be used for expensive ICs (e.g., a D/A converter, microprocessor, or the like), ICs that you're likely to want to change from time to time (e.g., a program ROM), and ICs that have a good chance of being damaged sooner or later (e.g., chips that buffer input or output signals from outside the instrument).

The problem is that a poorly designed socket may prove unreliable over extended time periods. A nonsoldered joint must have a gas-tight seal, such as that created by a mechanical metal-to-metal wiping action, with the seal then being left undisturbed. PC edge connectors, for example, used to be somewhat unreliable; with time, manufacturers learned some good tricks: bifurcated contacts (two independently sprung contacts for each finger), gold plating on the socket and on the edge fingers, and good mechanical design to ensure firm contact pressure during wiping and afterward. Joints that aren't gas-tight can be expected to fail after some time, perhaps a year or so. This sometimes happens inadvertently, e.g., by inserting a component in a PC board and then forgetting to solder it. Such connections have the maddening property of working fine at first, then becoming intermittent months or years later, owing to the formation of corrosion. A different problem can arise when heavy ICs (24 pins or more) are held in sockets. They can work their way out of the sockets after repeated vibration or shock.

We have found that the pin-and-jack type of IC socket (popularized by the Augat 5xx-AG series), although expensive compared with many other socket types, gives good reliability.

Soldering and deflusing

The usual procedure is to insert some components, turn the board over and bend the leads aside to hold the components

in place, then solder them using a thermostated soldering iron and fine solder. ICs can be inserted easily with an insertion tool (highly recommended), and it is best to use a lead bender on resistor leads, etc., in order to prevent slivers of solder being shaved off the leads during insertion. We recommend the adjustable "flip-over" board holders (marketed by OK Industries), which hold the components firmly in place with foam rubber during soldering. After soldering, the leads should be trimmed with a snipper.

Now comes a very important step: Solder flux should be removed from the board. If it isn't, the board will look just terrible in a few years, when you're not around to defend it! Rules for defluxing:

1. Do it.
2. Do it soon. The stuff gets much harder to remove with time.
3. Use a solvent such as Freon, alcohol, or some other organic solvent recommended for this purpose. Use a small brush to help dislodge stubborn globs of flux.

Commercial board manufacturers clean their boards with *vapor-phase degreasers*, in which hot vapor from a bath of boiling solvent condenses on the board (suspended above), dissolves the flux, and drips back into the bath. This method works well, especially because the distillation process continually bathes the board with clean (and hot) solvent. Because organic solvents aren't exactly good for your health, people have experimented with water-based cleaning processes. One method uses a water-based "saponifier" to emulsify the rosin so it will wash away; alternatively, you can use a water-soluble solder flux (rather than the usual flux, which is made from insoluble tree rosin). As nice as they sound, these water-based cleaning methods turn out to be tricky to do correctly; if you're not careful you can leave corrosive residue that ruins the board, in time. For the do-it-yourselfer we recommend organic

solvent cleaning; take care not to breathe the stuff or get it on your hands.

When defluxing boards, keep in mind that mere removal of the rosin residues is not enough – you have to get rid of the ionic "activators" also. Incomplete defluxing may do more harm than good, since it can release the activators from the area of the solder joints and spread them all over the board. Note also that solder flux comes in several grades of aggressiveness. We routinely use "RA" (rosin, activated), which is the most active flux for electronic soldering, because it produces good solder joints even when surface oxidation is present; RA flux residues should be thoroughly removed after soldering. The alternative, "RMA" flux (rosin, mildly activated), is less active and is often specified for government contracts or for applications where defluxing is not possible.

12.07 Some further thoughts on PC boards

The solderability of PC boards tends to decrease with time, owing to oxide formation, so it is best to stuff the components soon after the board is made. For the same reason, you should keep unstuffed boards in plastic bags, away from corrosive fumes. Good circuit boards should be made from 1/16 inch FR-4-type board material (sometimes referred to as "epoxy fiberglass") clad with 2 ounce copper. Remember that a circuit on a PC board is basically sitting on a piece of glued-together stuff; the board can absorb moisture and develop electrical leakage. Another pathology of PC board material is "hook," the variation of dielectric constant with frequency; the consequent variation of stray capacitance can make it impossible to build an amplifier with flat frequency response, for instance. Oscilloscope manufacturers are very aware of this bizarre effect.

PC runs with large currents passing through them have to be widened to prevent excessive heating and voltage drops. As a rough guide, here is a table of approximate conductor widths that give temperature rises of 10°C or 30°C for the currents listed, for 2 ounce copper PC boards. For other foil thicknesses, just scale the widths accordingly.

	0.5A	1.0A	2.0A
10°C rise	0.004"	0.008"	0.020"
30°C rise	0.002"	0.004"	0.010"
	5.0A	10.0A	20.0A
10°C rise	0.070"	0.170"	0.425"
30°C rise	0.030"	0.080"	0.200"

PC runs with high voltage need correspondingly wider spacing – a good rule is 5 volts per mil (0.001"). It is also a good idea to avoid sharp corners and points, in favor of rounded contours.

Tools

As a starting point, we lined up the most heavily used tools on our bench and came up with the following part numbers:

Long-nose pliers	Erem 11d, Utica 321-4 $\frac{1}{2}$, C.K 3772H, Xcelite 72CG
Snippers	Erem 90E, C.K 3786HF
Soldering iron	Weller WCTP-N, Ungar "Ungarmatic"
Solder	Ersin Multicore 22ga, Sn63 alloy, RA flux
IC inserter	Solder Removal 880
Lead bender	Production Devices PD801
Solvent dispenser	Menda 613
Solder sucker	Edsyn Soldapullit DS017

For the most effective desoldering of PC boards, it's worth investing in a controlled-vacuum desoldering station. These work well even on solder-filled plated-through holes, with less tendency to damage the delicate foil pads (when compared with the simple spring-powered plunger types). These desoldering stations do have a tendency to clog with solder, however. They are manufactured by several companies,

including Edsyn, OK, Pace, Ungar, and Weller. Lots of useful gimmicks for PC assembly are listed in the Contact East catalog (335 Willow Street South, N. Andover, MA 01845) and the Marshall Claude Michael catalog (9674 Telstar Avenue, El Monte, CA 91731).

12.08 Advanced techniques

Hand-drawn schematics, manually converted first to PC board layouts and then to hand-taped Mylars, are the traditional steps in PC board design technique, and they were used nearly universally through the mid-1970s. It still makes sense to do simple boards this way, particularly if you aren't trying to break new records in board component density. With a little kit of double-sized patterns, some opaque tape, a gridded table, and some clear Mylar, you're ready to go. And you don't have to buy expensive CAD software (and learn how to use it!), pay photoplotting charges, etc.

However, as soon as you strive for high-density boards, paved wall to wall with fifty or a hundred IC's, often requiring four or six layers laid out to 10-10 design rules, the honeymoon is over. Even with extraordinary effort and quadruple-sized Mylar, you can hardly achieve the precision interlayer alignment required. Furthermore, you'll need a month's vacation to recover from the concentration required to work out the routing. The first board you get back from the PC house is unlikely to be error-free, and any serious changes to the taped Mylars are sheer hell, often requiring massive areas of wiring to be pulled up and repositioned (which often introduces new errors). We aren't kidding.

CAD/CAM

The solution is CAD/CAM (computer-aided design/computer-aided manufacture). These software packages are powerful, and with the advent of desktop

workstations with multimegabyte memory, dazzling graphics, and processing speeds of tens of MIPS (million instructions per second), you don't need special hardware to run them. Some popular PC CAD systems come from Valid Logic, Mentor Graphics, and Daisy Systems. They aren't cheap, but at least they're not getting any more expensive. Here's a quick tour of board design using CAD/CAM.

- *Schematic capture.* You begin by "capturing" your schematic directly into the graphics workstation. You can draw and edit diagrams, using standard electronic symbols from a "library." Just as with word processing, you can pull up old work, extract sections you want to reuse (e.g., a DRAM memory control circuit, an active filter, etc.). With a mouse (or joystick, trackball, or graphic tablet) you can move things around, with the wiring following you around (not always totally successfully!). You name signals, give the ICs part numbers, and so on. Good CAD systems assist you by looking up the ICs and assigning pin numbers. And good CAD systems are *hierarchical*: You can, for example, do a top-level diagram in which the major subcircuits are indicated by big boxes; each box opens up to reveal its sub-boxes, eventually displaying the lowest (gate-level) description. At any stage of the design you can get handsome laser-printed schematics. See Appendix E for further thoughts on schematic diagrams.

- *Checking and simulation.* The output from the schematic capture consists of a set of drawings (Fig. 12.8) and a *netlist*, which is simply a list of every signal, telling every "node" (component pin) it is connected to.

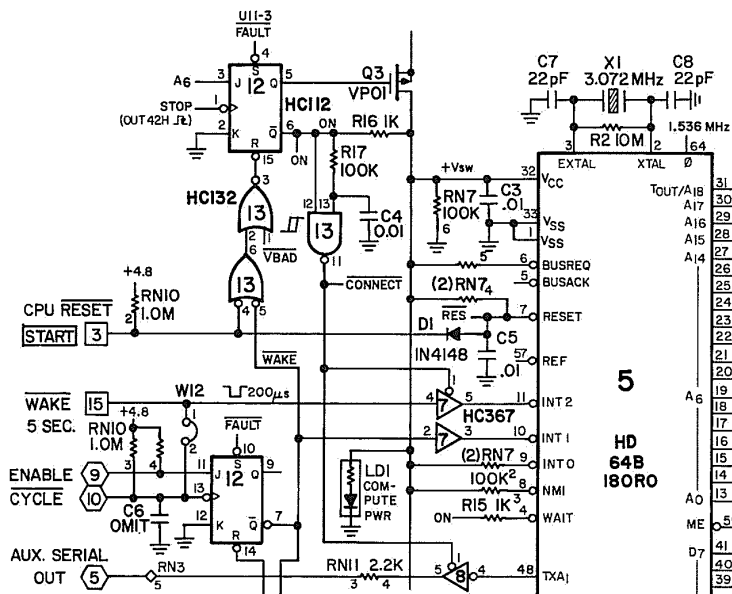
At this stage in the design you should, of course, spend plenty of time with the schematics, making sure they're what you want. A good CAD system can help you out, by flagging obvious errors, for example an output tied to ground, or to another

output. You can ask the system to simulate digital circuit operation, but you must provide it with test "vectors" that describe machine states, and you must describe the function of each chip in your circuit that is not already described in the library. Since a complex chip like a microprocessor requires lots of description, some CAD systems have a socket into which you can plug such a chip; it then uses the chip itself for hardware modeling.

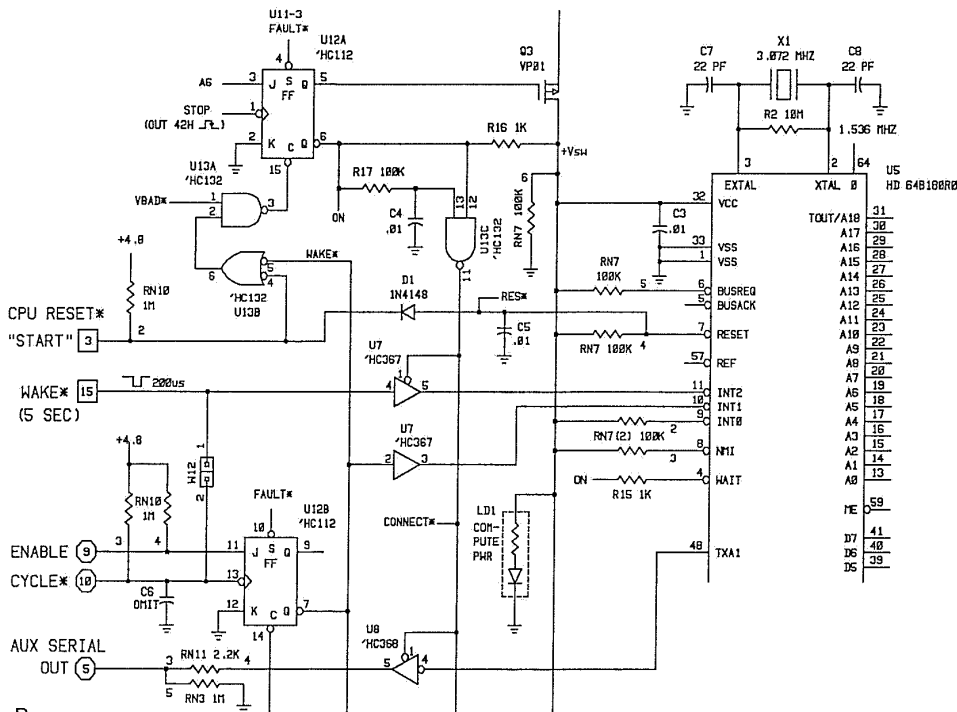
At this point, too, you can ask the CAD system to disgorge voluminous documentation showing (a) for each named signal, all the pins ("nodes") it visits and (b) for each pin of each component, all other nodes to which it is connected. Good systems will even inform you if you have violated loading rules for digital outputs, etc.

- *Placement and routing.* The next step is component placement and routing. One of the great beauties of an integrated CAD/CAM system is that it automatically extracts the netlist from its representation of the schematic – if the schematic is correct, the final PC board will be too. Although some CAD systems claim to be able to juggle the component placements into a near-ideal configuration, it's usually best to do component placement manually. As with schematic capture, this is done with mouse, joystick, or graphic tablet. You begin by setting up the board outline (it's often the same as some previous board you have designed), within which you now position the parts. Since the component library has outline and pin information, you get to play with the characteristic IC and component shapes. Good CAD systems can flag design-rule errors as you make them, so you can see if the parts are too close together, etc.

Once the parts have been placed, the wiring must be routed. At this point it is conventional to display what's called the "rat's nest," a display of the board with all the connections shown as straight lines



A



B

Figure 12.8. Portion of a complex schematic diagram from which the PC board in Figure 12.11 was produced.

A. Manually inked on a drafting table.

B. Machine-drawn, from Case Technologies schematic capture.

connecting their respective pins. It's a tangled mess. With creative use of color you can make some sense of it, for example by selectively turning on only power and ground connections.

Early CAD systems made you work hard at this point. You were forced to route every connection by hand, with just a little help from the computer. Current systems provide *autorouting*, which means what you hope it does: The software finds connection paths, taking care not to violate the design rules that specify not only line widths and clearances but also parameters such as the maximum number of "vias" (plated-through holes used only for a signal trace, not a component lead). The best CAD machines can do 100% autorouting, although the result often lacks the full elegance of hand-routed lines. For example, it may put a via daringly close to a component pad; although its placement meets all the rules, you can create a board that's more easily manufactured and soldered by shifting the via away from the pad. You usually spend a few hours cleaning up the machine's work in this way, pushing and pulling traces around a bit (Fig. 12.9).

In complex digital circuits you can often simplify the routing considerably by reassigning gates or sections within chips, or even swapping gates between IC packages. As an example, you might have a byte-wide data bus connected to an octal *D* register. Your initial assignment of bus bits to corresponding register bits might force each line to leapfrog the next, whereas by reversing the order of bits in the register the resulting routing requires no crossings. Good CAD systems can do this automatically (the library specification of each chip identifies duplicated functions), even updating ("back-annotating") the schematic via the modified netlist.

The final step is to check the proposed routing for any rule violations, and also for exact equivalence to the schematic. The

latter step is important because you may have made manual changes to the routing netlist that altered the circuit. At this point all CAD systems will produce paper pen-plots of the routing, showing any designated combination of components, wiring, labels, etc., in a dizzying array of colors.

- **Photoplot and drill tape.** If all has gone well, the last stage of the process consists of producing a machine-readable board description suitable for the PC house. For this you need two things: a tape that tells a photoplotter how to draw a precision plot of the "artwork" (the individual foil patterns for each layer, a pattern for the "solder mask," and a silkscreen pattern for printed legends) and a "drill tape," which tells each hole size and its precise position. The artwork tape is usually written in "Gerber format," named after a brand of photoplotter that exposes film by moving a large easel under a stationary projector in response to commands written on tape. (A newer form of photoplotter uses laser scanning, producing large plots in minutes rather than the hours required by the early Gerber machines.) Some PC board fabricators want you to supply the finished photoplots (Fig. 12.10), whereas others ask for the Gerber tape itself. A surprising number of board houses ignore the drill tape, preferring to figure out where the holes go by manually digitizing the photoplotted artwork (if you ask, they'll tell you that customer-supplied drill tapes sometimes have crazy commands that break their drill bits). Drill tapes, believe it or not, are not magnetic tapes; they are *paper* tapes!

Board production. In this world you don't manufacture your own PC boards; you go to a PC board house. They're all over the place, and they just love to make boards (Fig. 12.11), in exchange for money of course. Some specialize in

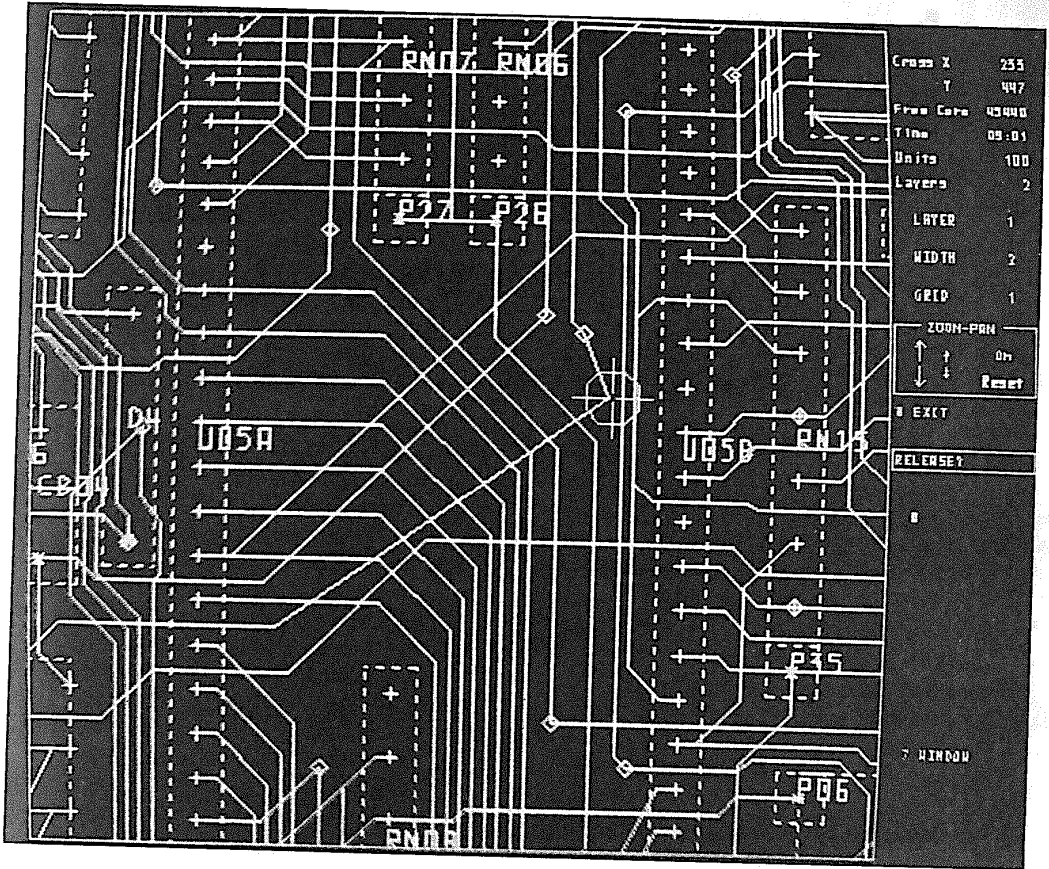


Figure 12.9. Photograph of PC board design system screen display (Racal-Redac), showing manual "cleanup" of trace routing. The cursor, steered via a "graphics tablet" stylus, shows the circuit trace that is being modified. This display is in color, with different colors and intensities designating the different layers, component outlines, legends, etc.

small-quantity, fast-turnaround prototyping, whereas others may do only production quantities. Your first task is to find out which companies do a good job. This is harder than it sounds. You can't tell much on the telephone, because it is possible to *know* how to make good boards, but still make bad ones. Here are some tricks we've tried: (a) Look at the little logos on good boards you've seen (e.g., boards you find in instruments or computers sold by

major companies, who, like you, usually buy their boards from external board houses) – companies like IBM and Apple are extremely careful about their board suppliers, (b) Ask everyone you know whom they use to make their boards, (c) Go and visit the board house; while there, look for cleanliness, high morale, native intelligence, diligence, and pride in workmanship; you might also ask to see some boards, (d) Finally, if you know

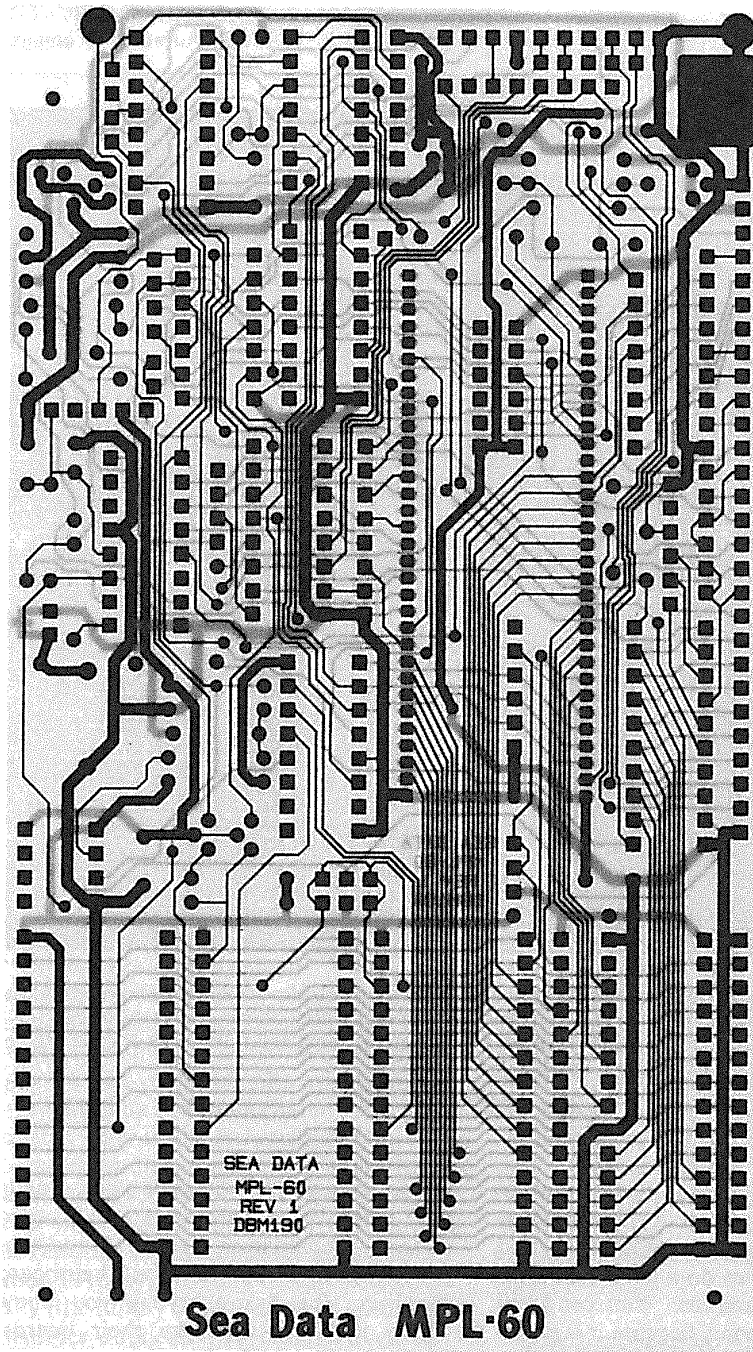


Figure 12.10. Photoplot for the double-sided board shown in Figure 12.11. Both layers are overlaid in this photograph, showing the view from the solder side.

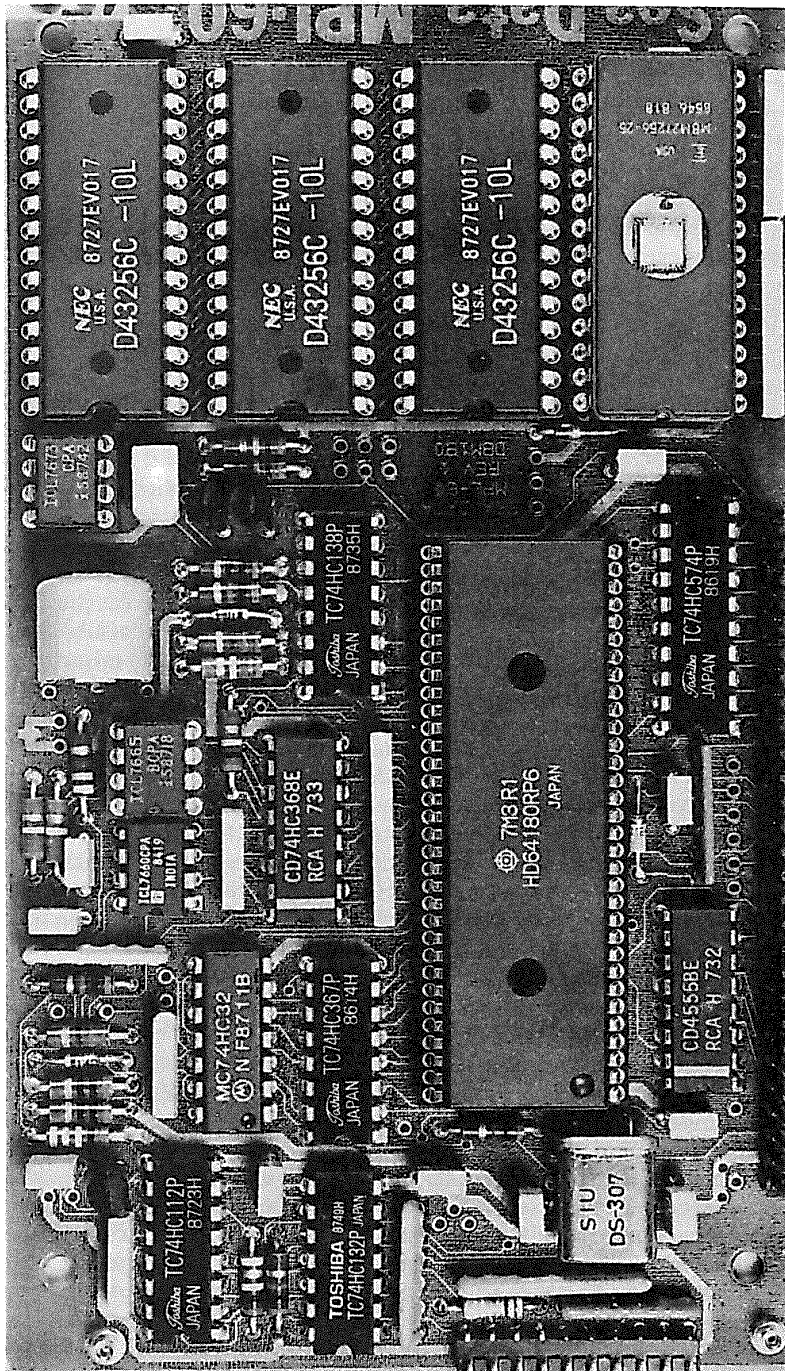


Figure 12.1.1. Finished microprocessor board, designed to double-sided 12-12 rules. Higher component density can be achieved only by going to tighter design rules, or, preferably, a multilayer board.

someone who works at a bare-board testing house, cajole him into telling you who makes good boards: Those guys really know, but they're not supposed to tell!

Next you have to get a quote for price and delivery. Be prepared to answer the following set of questions, which every board house will ask you:

1. board dimensions?
2. number of layers (sides)?
3. design rules (minimum conductor width and spacing)?
4. any gold-plated edge-connectors?
5. any fancy board cutouts or shapes?
6. solder mask? wet or dry? to cover the vias ("tented") or not?
7. board and copper thickness?
8. number of holes?
9. number of hole sizes?
10. silkscreen 2 sides, 1 side, or none?
11. any unusual options, e.g. "dry mask over bare copper"?

Having agreed on terms, you must now supply the following things:

1. actual-size layer positive transparencies (i.e., opaque = copper)
2. soldermask patterns(s)
3. silkscreen pattern(s)
4. drill drawing (hole pattern, coded by sizes)
5. mechanical drawing (precise outline, aligned to specific holes)
6. drill tape (a paper tape; often optional)
7. additional specifications and notes

The last item is important. You should strictly specify the minimum trace width (an over-etched board has thin traces), the minimum conductor spacing (under-etched board), the minimum annular ring (caused by misaligned holes or misaligned layers), the tolerances for finished hole diameters, minimum thickness of copper (and solder) plating, the maximum number of repairs permitted per board, and so on. The industry generally adheres to

some typical numbers (e.g., the IPC-600 standards), but it's best to have it clearly in writing, in case you get some boards you're unhappy with. Furthermore, your board may have unusual requirements; for example, if you've used unusually small pads for your vias, your minimum annular ring specification for vias will be less forgiving than usual.

Testing. For any new PC board design, *always get a few prototypes made first.* Stuff them with sockets, and test the dickens out of the circuit. You can expect several kinds of bugs, namely (a) your circuit had a design error, (b) the circuit was OK, but you made an error in schematic capture (which you should have caught earlier), (c) an error crept in during routing (which also should have been caught earlier), (d) the board has flaws, usually shorts or opens, due to blemishes, misalignments, or other problems during manufacture (see below), or, finally, (e) there are some bad components or solder joints.

Assuming circuit changes are needed during debugging, the usual procedure is to cut existing foil traces with a small knife, then solder in wires to make the new connections. This may be impossible in a multilayer board with buried signal layers, however. In that case you have to make your changes at the final destinations – the IC or component pins. A good way is to stack up two IC sockets, with a pin removed from the upper one.

During circuit testing, mark up your schematics with circuit fixes, for later entry into the schematic capture (or hand-drawn circuit). The second pass through the CAD system is usually easy and fast; but be *very* careful, because you probably will go directly to a production run, and any errors remaining in the production board will have to be fixed on each unit. If that becomes necessary, you use the same procedures of cutting and jumpering as above, taking care to secure the wiring

(a hot-wax gun seems to work well). These extra wire patches are sometimes called "roach wires"; the official term is "ECO," for "engineering change order."

There are plenty of things that can go wrong in board production (see the fine handbook by Coombs for gory details). With high-density boards (Fig. 12.12) even a minor blemish can create a short or open circuit somewhere. And plated-through holes have all sorts of pathologies of their own that can lead to open circuits. It's an axiom in electronics that the cost to repair a flaw multiplies at each stage of production – a bad board connection is easily fixed at the board house, fixed with greater investment of time during circuit checkout, and repaired with considerable expense once shipped into the field. So it makes good sense to have the board house do a "bare-board" test on all your production boards. This they do on a curious little setup that jabs the board with a set of pins that align with the pads (they fabricate the "bed of nails" from your drill tape, of course). Oddly enough, none of the testing houses compares the bare board with your *netlist*; instead, they just test a bunch of boards, and assume that if a number of boards are identical, they are also correct. Although sharp-eyed readers will see a flaw in the logic, they can rest assured that the system usually works anyway.

- **CAD-based Wire-Wrap.** The netlist output from the schematic-capture phase contains all that is necessary to construct the circuit. Many CAD systems offer output formats compatible with automated Wire-Wrap fabrication. You begin by choosing component placements on a standard Wire-Wrap panel (see the Augat catalog for numerous specimens), after which the CAD system constructs a special form of netlist called a "From-To" list. Each entry tells the coordinates of the two pins connected by a wire, and the wrap level

(height above board) of each end's wrap. If you send this tape to a wire-wrapping house such as DataCon, they will produce a machine-wrapped board. It typically takes a few weeks and costs about 6 cents per wire plus the price of the Wire-Wrap panel (about \$2 per IC position). As with PC boards, there can be flaws. Our experience is that you might find one or two problems per board, usually in the form of a broken wire that looks OK from the outside.

A more serious question about Wire-Wrap prototypes is "Why bother?" If your objective is to make PC boards, you're going to have to do a PC board routing eventually, so why not do the work for the prototype? Furthermore, a wire-wrapped board behaves differently than a PC board, owing to lead inductance and differences in ground-pin connection length and ground-plane inductance. The effects are most noticeable if you have fast logic such as 74F, 74AS, or 74AC(T), or memory arrays with widely bused signals. In general, a good multilayer PC board performs far better than a comparable wire-wrapped board, so you do yourself a favor by building the prototype with a PC board.

Of course, if you are building only a handful of units, you may want to avoid the one-time ("nonrecurring") expenses associated with PC board production and just build wrapped boards as the finished product. In that case the machine-wrapped boards are a rational choice. Don't overlook the alternative, however, which is the following.

- **CAD-based Multiwire.** "Multiwire" is a trademark of the Kollmorgen Corporation, which manufactures a clever hybrid of custom-routed PC board. The basic board has pads in standard or custom groups to hold your ICs and to provide power and ground. The signal interconnections are made not with PC traces, however, but with fine insulated wire (#34 or #38) that

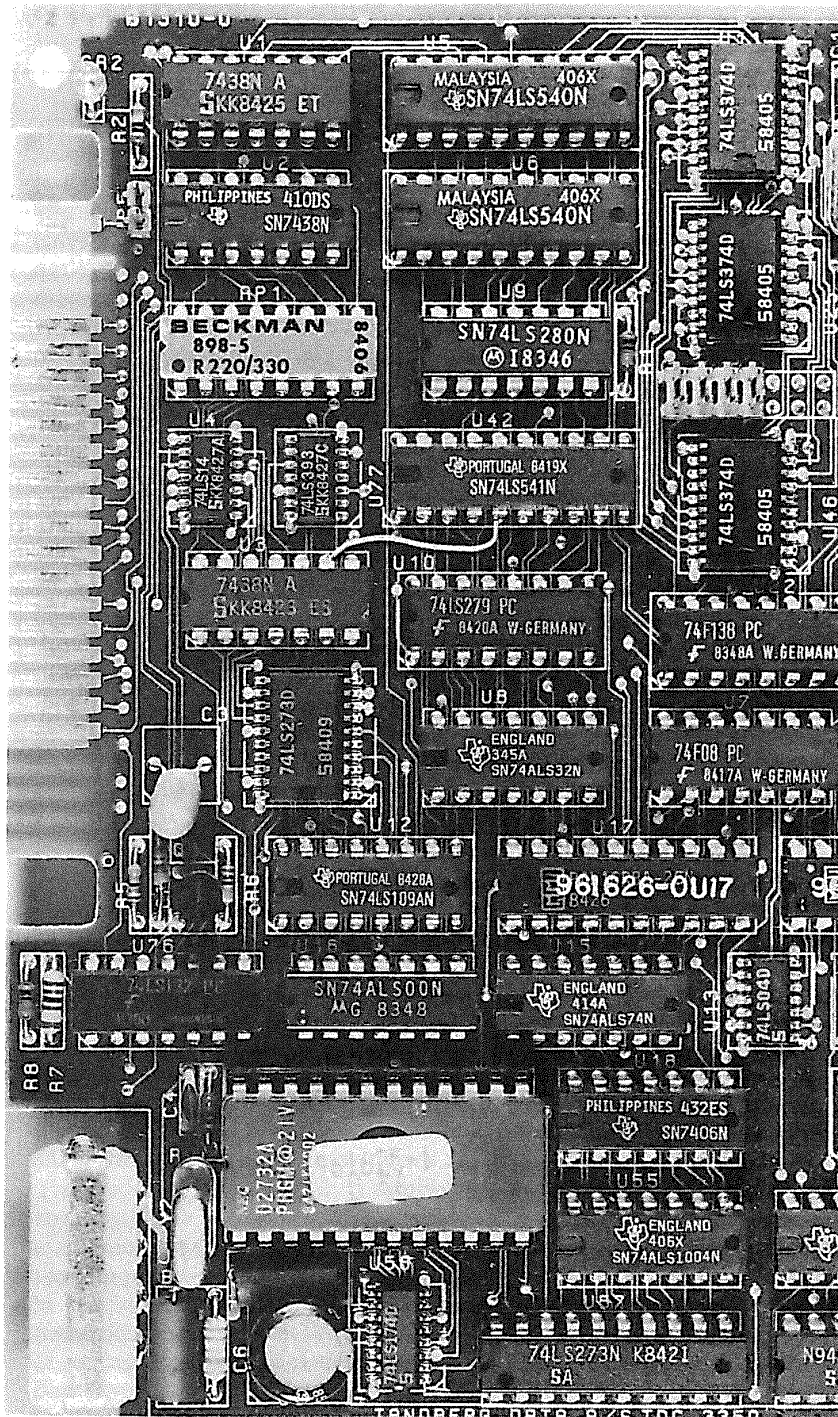


Figure 12.12. Surface-mounted devices (SMDs), with their 0.050 inch pad spacing, permit high component density, as shown in this photograph of a portion of a 4-layer board with 10-10 design rules. Both SMD and conventional through-board-mounting components are seen. The edge connector provides a scale, with its pads on 0.1 inch centers.

is machine-placed onto a sticky coating on the board. The wire ends are attached to the IC pads with a hole plating process.

Multiwire has good signal properties because of its complete groundplane, and it lets you make circuit changes in subsequent production runs relatively painlessly. Since the (insulated) wires can cross over each other, you can achieve very high component density, comparable with that of a 10-layer PC board. Kollmorgen claims that Multiwire is thus a cheaper solution than custom multilayer PC boards, particularly in small production runs and in situations where your circuit is evolving.

□ *CAD-base ASIC.* We finish our CAD narrative by pointing out that you would not dream of designing custom or semicustom (gate-array) ICs by hand (Fig. 12.13). These belong in the general class of "application-specific integrated circuits," or ASICs. Once again, CAD comes to the rescue, with capture and simulation software that gives you some assurance that your chip has a chance of working. ASICs are rational in the context of substantial production runs, and we confidently assume that the situation will only improve with time. By the time of the next revision of this book it may have become

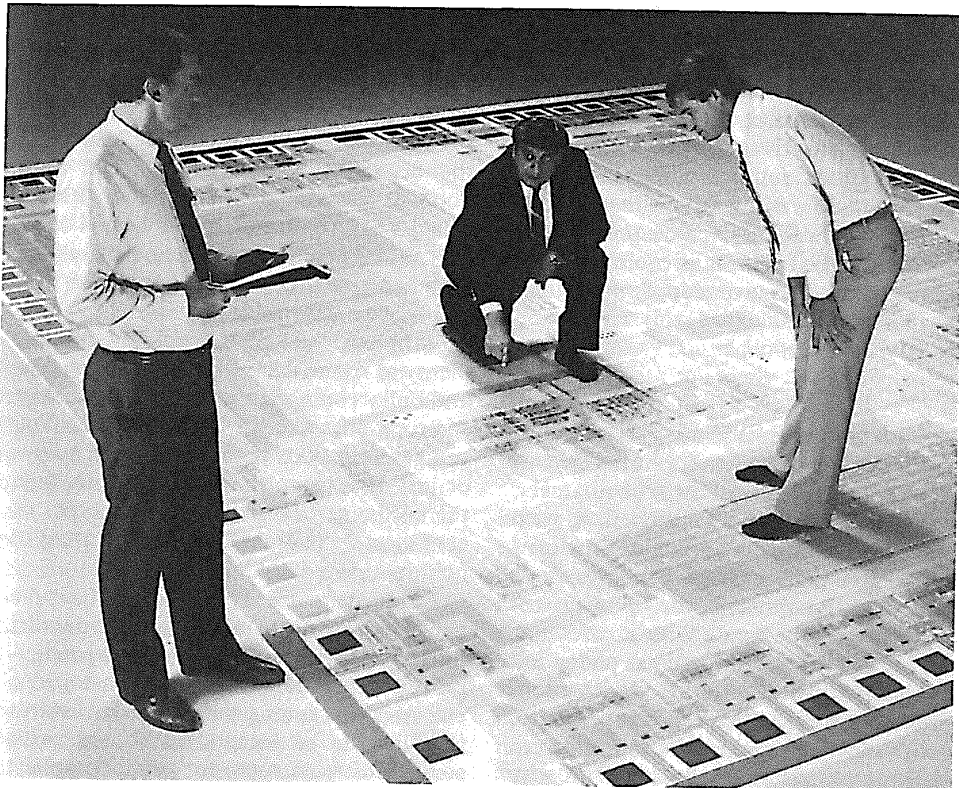


Figure 12.13. At the *chip*-design level, CAD/CAM isn't a luxury – it's a necessity. (Courtesy of Performance Semiconductor Corporation, Sunnyvale, California.)

commonplace to design custom ASICs, promptly delivered by your local silicon foundry, for just the sort of circuits for which you now design a PC board filled with standard ICs. We see no fundamental reason to prevent the ASIC solution from becoming the less expensive (and better performing) alternative.

INSTRUMENT CONSTRUCTION

12.09 Housing circuit boards in an instrument

Circuit cards, whether printed circuits, Wire-Wrap panes, or breadboarding cards, have to be mounted in some sort of enclosure and connected to power supplies, panel controls and connectors, and other circuitry. In this section we will discuss some of the popular methods of putting instruments together so that circuits are neatly mounted and accessible for testing and repair. We will begin with methods for holding the circuit cards themselves, then discuss the business of cabinets, front and rear controls, power-supply mounting, etc.

Circuit card mounting

In simple instruments you may have only a single circuit card, whether printed circuit, Wire-Wrap, breadboard card, or whatever. In that case a simple solution is to drill holes near the corners and mount the card with screws (and standoff bushings) to a flat surface, component side up. Connections can then be made with a card-edge connector socket (if the card has plated fingers), with flat cable terminated with a connector to mate with a plug on the board, or with individual soldered connections to swaged terminals on the board. With edge or ribbon connectors, the card will support the connector adequately, so no extra connector supports have to be used. Whatever

the method of connection, it is wise to arrange the wiring such that the board can be tipped upward for access to the underside so that you can make modifications or repairs.

In a system with several circuit boards, the best way to arrange things is with some sort of card "cage," a rigid assembly with guides for individual cards to slide into and aligned holes along the rear so that you can mount edge connectors to mate with the cards. There's lots of flexibility in card width, spacing, and number of cards that can fit into a card cage. A very common size accommodates cards 4.50 inches wide with 44-pin dual-readout (22 pins each side) edge connections on 0.156 inch centers. There are plenty of other formats, often with tighter pin spacing (0.1 inch is common), multiple groups of edge connectors on a single card, or the more reliable "2-part" connectors (in which one mating portion solders to the board as a component); the most popular example of the latter is the so-called VME connector, which has 64 or 96 pins. Cards can be spaced as little as 0.5 inch apart, if necessary, although 0.6 inch is a more comfortable spacing; if space is no problem, 0.75 inch spacing will allow plenty of room, even with Wire-Wrap pins and bulky components. It is best to look at some catalogs to see what's available. You can get variations with plastic card guides or just dimples in the metal sides to align the cards, and there are various types of card ejectors (attached to the PC cards) to help remove a card.

Card cages are available with simple flange mounting to a flat surface parallel to the cards, as well as in various configurations that fit nicely into rack enclosures, etc. You can even get modular enclosures that include an integral card cage, with some additional room for power supplies, panel controls, etc.

Warning: Instruments built with the circuit spread over a number of plug-in

circuit boards achieve a nice modularity and ease of repair. But this method of construction can cause difficulty in circuits with low-level signals (less than a millivolt), and in circuits with high-frequency signals (above a few megahertz). The basic problem is that you often cannot provide a sufficiently stable and low-inductance grounding system to the set of boards by way of the connectors at the ends of the boards. Combining low-level analog circuitry with digital switching waveforms is particularly dangerous. This problem is worst with a hand-wired backplane, in which the ground distribution depends on a few wires running between the connectors. Typical symptoms are low-level 60Hz or 120Hz pickup, miscellaneous "fuzz" at the level of fractions of a millivolt, and coupling of radiofrequency signals between circuits that are supposed to be isolated. If the boards have a bare groundplane and are plugged into a metal card cage, the symptoms often change as you press on the cards, altering the uncontrolled ground contact.

We've suffered with such problems on more than one occasion, and we offer the following advice. First, it may be best to avoid interconnected small boards entirely, building all the critical circuitry on one larger board with integral groundplane. On that board you may need to use coaxial lines or twisted-pairs to make connections between separated portions of the circuit. Second, if you do use interconnected boards, you will achieve better ground distribution with a PC motherboard provided with wide ground traces, rather than a hand-wired backplane. In radiofrequency systems you sometimes see springy metallic fingers used along the card guides, to provide a continuous and stable ground connection. Third, the use of coaxial line or twisted-pairs, combined with differential inputs (or "pseudo-differential" ground-sensing inputs, see Fig. 7.70), is often the best way to handle microvolt

signals, which are otherwise prone to ground-loop noise and pickup. Finally, except for the ground-sensing method just described, we suggest as many redundant ground connections as possible – multiple connections to the chassis, doubled connector pins and wires, etc. – to reduce the inductance through which ground currents must flow. Don't worry about "ground loops" in a digital or RF circuit; that's a microvolt audio circuit issue. Refer to Section 7.24 for additional ground philosophy.

Backplane connections

Card-edge connectors are available with lugs for solder connections, with Wire-Wrap pins, and with small pins for insertion into PC boards. In many cases it is best to wire up the intercard connections with point-to-point wiring between card-edge connector pins, using the edge connectors with solder lugs. A neat job requires some cabling of wire bundles, with the wires running in straight lines parallel to the card cage dimensions. In other cases it may be preferable to use Wire-Wrap connections on this backplane, especially if there are many connections between backplane pins with relatively few connections to other points in the instrument, and if there is no need for shielded-cable connections to the backplane.

The third possibility is to use a motherboard backplane, a PC board designed just to hold the card-edge sockets. Motherboards are popular in bused systems (they are nearly universal in computers) and should be considered in any case if the instrument is intended for production in significant quantities. With double-sided motherboards, you can have the advantage of a groundplane (lower inductance and coupling of signal lines), or you can use both sides for signals if the intercard wiring is complicated. With bused systems the backplane is usually simple, with lines

connecting corresponding bus pins on all cards. In computer backplanes you sometimes see a motherboard used with Wire-Wrap pins sticking through. This is very handy if you want the motherboard to do all the bus and power-supply connections, leaving the unused pins to be connected in a custom configuration by wire-wrapped connections. Figure 12.14 shows a simple PC motherboard.

12.10 Cabinets

Depending on the intended use, an electronic instrument might be housed in a benchtop cabinet (complete with rubber feet and hinged front "bail"), in a cabinet or panel designed for mounting in a standard "relay rack" 19 inches wide (either screwed directly to the rack flanges that run vertically up from the floor or mounted

on ball-bearing rack slides for simplified access), in a modular instrument case designed to plug into slots in a larger rack-mounted "bin," "cage," or "crate" (the latter usually provide dc power connections through standardized connectors at the rear), or perhaps in some other format such as a free-standing pedestal-mounted case.

There are many cabinet configurations available in both benchtop and rack-mounting formats. Among the most popular are cabinets 17 inches wide, available in various heights (always multiples of 1.75 inch) and depths, that accept optional rack-mounting flanges or slides (a rack 19 inches wide has about 17 1/2 inches of clearance between the flanges). That way you can convert an instrument from rack-mounted to benchtop format, or vice versa, by just changing a bit of cabinet hardware.

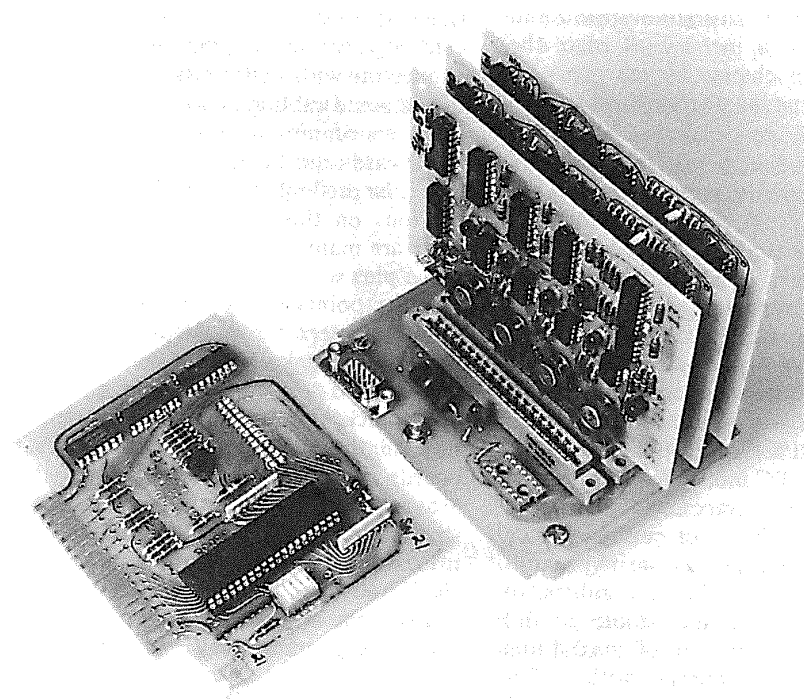


Figure 12.14. A "motherboard" provides a convenient method of interboard connection, reducing hand wiring and the possibilities for error considerably, while simultaneously providing superior electrical performance. In large systems, the motherboard and its connectors would probably be mounted rigidly at the rear of a card cage.

A point to check: Some of these convertible cabinets require removal of the outer case for rack mounting, whereas others let you keep the cabinet intact.

In the category of modular instrumentation, the NIM bin is popular in nuclear and atomic measurements, the CAMAC crate has established itself in some computer interfacing, and several manufacturers have defined modules and bins (e.g., the TM500 series from Tektronix and the EFP series of blank modules from Vector). Blank chassis are available in each of these formats, complete with rear connectors to mate with the dc power receptacle of the mother bin.

12.11 Construction hints

Rather than attempt to list the enormous variety of manufactured cabinets by name or style, we will simply offer some general comments on construction of instruments. These suggestions, together with the figures in this chapter, should help you choose electronics enclosures wisely and fill them up with circuitry in a sensible way.

In general, you use the front panel for indicators, meters, displays, keypads, etc., as well as controls and frequently used connectors. It is common to put seldom-used adjustments and connectors that don't require frequent access on the rear panel, along with large connectors, line cord, fuses, etc. (Fig. 12.15).

The traditional way to lay out a professional-looking front panel is by silkscreening the legends onto the painted or anodized aluminum surface. Although the result doesn't look bad, it tends to erode with frequent rough handling (although a clear overcoat can help considerably). If you look at a recent Fluke, Tektronix, or Hewlett-Packard instrument, you'll see the modern alternative – an adhesive Lexan panel of pleasing appearance and unbelievable toughness. These are made by silkscreening the legends on the *back* side of a matte-textured 0.010 inch Lexan film,

to which a tough adhesive is then applied. You can apply lettering and patterns in several colors, and you can specify colored plastic windows or cutouts. You can get these custom laminated panel coverings in quantity from many labeling services; you just provide them with the "artwork" (usually an actual-size positive or negative). You might prepare the artwork with dry-transfer rub-on lettering or, better yet, use some desktop publishing software and a laser printer.

Perhaps the most important thing to remember when laying out an instrument is the need for good accessibility to circuit cards and controls. It should be possible to replace any component in the instrument without great pain. This means neat cabling of wiring, so that modular units can be raised up without having to unsolder anything, and careful planning, so that circuit cards can be tested while operating in the instrument. For instance, a card cage might be mounted with the cards vertical; to get at them, you remove the top panel from the cabinet, then plug in extender cards to make the circuit cards accessible. If the cards are mounted horizontally, you might make the front panel removable, or hinged, to provide access. At all costs, fight the temptation to lay down the circuit in "layers," with circuitry nicely covering other circuitry. Figure 12.16 shows an example of neat and accessible front-panel cabling built in a cabinet with a removable front panel.

12.12 Cooling

Instruments that consume more than a few watts will usually require some sort of forced air cooling. As a rule of thumb, a small instrument running more than 10 watts, or a larger (rack-width) instrument consuming more than about 25 watts, will probably benefit from a blower. It is important to keep in mind that a box full of electronics may run at

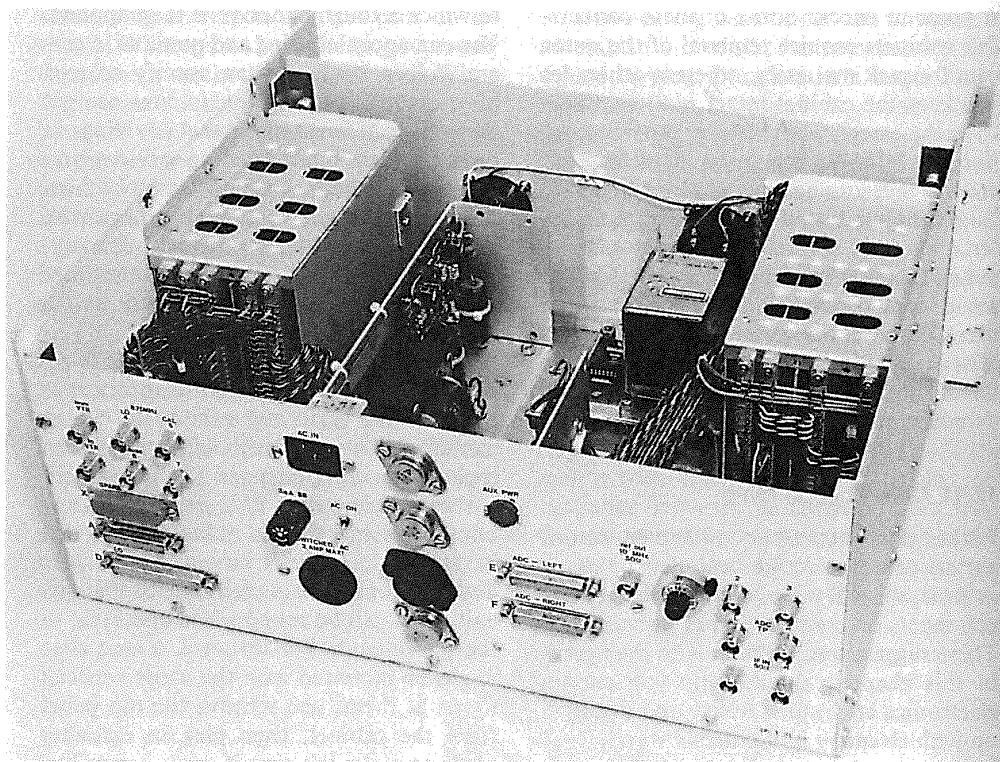


Figure 12.15. In this instrument we used the rear panel for heat-sinking power components, and for seldom-changed controls and connectors. The front panel detaches for access to the nine circuit boards housed in card cages.

a nice temperature when sitting on the bench with the top cover removed, but when installed in a rack with other heat-producing equipment (where the ambient temperature may reach 50°C), complete with its outer cover, it is likely to run very hot, leading to early failure of components and generally unsatisfactory operation.

Instruments running at moderately low power, say at the figures mentioned earlier, can often be cooled adequately with simple convective cooling. In such cases you might perforate the top and bottom covers, paying attention to the location of major heat-producing components (power resistors and transistors). It may be best to mount the high-power components on the

rear panel, using heat sinks with their fins aligned vertically (see Section 6.04). Circuit boards will also be better ventilated if mounted vertically, although heat dissipation in circuit cards is often negligible. If simple convective cooling doesn't keep things cool enough, you have to resort to a blower.

The simple "Muffin-type" venturi instrument blower, with flow rate of about 100 cubic feet per minute (CFM) assuming relatively unimpeded airflow, will adequately cool instruments running 100 watts or more. Here's the relevant formula:

$$\text{air temperature rise} (^{\circ}\text{C}) = \frac{1.6 \times P(\text{watts})}{\text{airflow}(\text{CFM})}$$

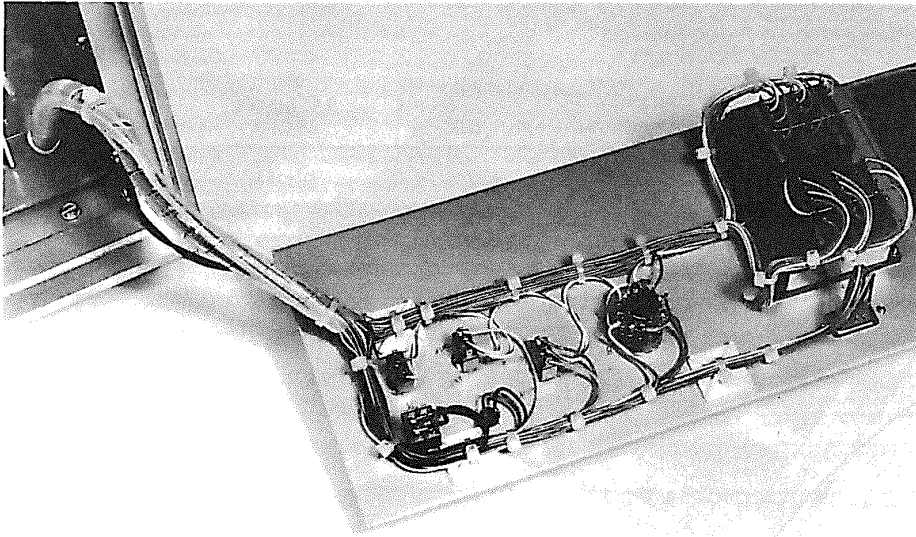


Figure 12.16. One way to ensure good accessibility to panel controls is to bring all wiring away at one end, so the panel can be hinged or otherwise detached from the instrument. In this example the panel slides into a slotted instrument case. Note the use of cable “ties” and self-adhesive supports to keep the wiring tidy.

If less airflow will suffice, a quieter version of the venturi fan is available from most manufacturers. Table 12.2 shows some part numbers. The airflow of these fans is greatly reduced when operating against high back-pressure. Figure 12.17 shows a graph.

All the blower manufacturers now sell brushless dc fans, in addition to the traditional line-voltage-operated ac fans. Running the fan from low-voltage dc (usually 12V or 24V) makes it easy to control the fan speed electronically, in response to the actual temperature inside the instrument. You can either do the temperature sensing and speed-control feedback yourself (perhaps using an outboard module like the “SmartFan” controllers made by Control Resources) or buy a fan with built-in “thermal speed control” (e.g., the Rotron “ThermaPro-V” series). Either way, using a variable-speed blower makes good sense, because under most conditions the fan noise will be far less than the

worst-case (full-speed) condition, which is needed only when operating an instrument at its maximum ambient temperature.

When laying out an instrument designed for forced air cooling, try to arrange things so that the air enters the box at one end, flows around the components, and exits at the far end. In an instrument with an interior horizontal chassis partition, for example, you might punch some inlet perforations at the bottom rear, perforate the internal chassis near the front of the instrument, and mount the exhaust fan at the top rear, thus forcing the airflow to pass through all parts of the instrument. Keep in mind that a circuit board will block airflow, and plan accordingly. If there is significant impedance to the flow of air (high back-pressure), a centrifugal blower will work better than the propeller type. The blades of the latter go into “stall” when the back-pressure exceeds about 0.3 inch of water, rendering the fan totally ineffective. Finally, in any

TABLE 12.2. VENTURI FANS

Manuf	Standard 4.7" square 105-120 CFM	Quiet 4.7" square 70 CFM	Very quiet 4.7" square 50 CFM	Mini 3.1" square 35 CFM
Rotron	MU2A1	WR2H1	WR2A1	SU2A1
IMC	4715FS-12T-B50	4715FS-12T-B20	4715FS-12T-B00	3115FS-12T-B30
Pamotor	4600X	—	4800X	8500D
Torin	A30108	A30390	A30769	A30473

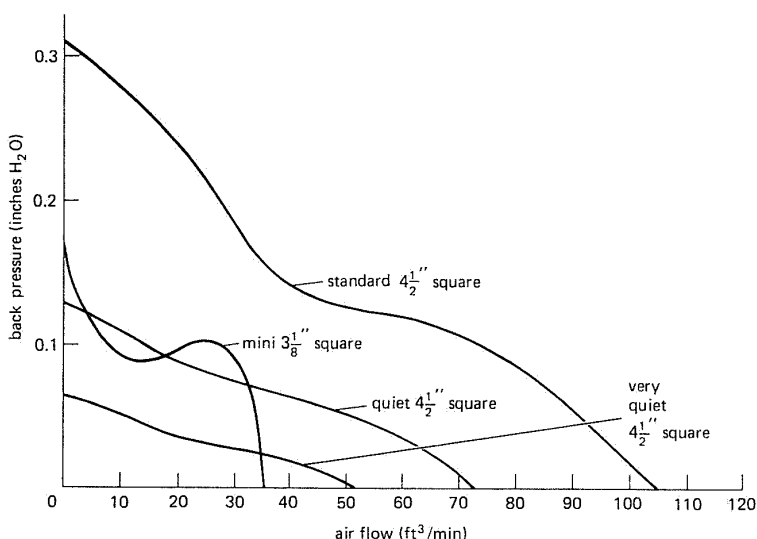


Figure 12.17. Airflow versus back-pressure for the venturi blowers listed in Table 12.2.

cooling situation it is a good idea to design conservatively; failure rates for electronic instruments rise dramatically when equipment is operated hot. Figure 12.18 illustrates good instrument design in regard to cooling and accessibility.

12.13 Some electrical hints

Unreliable components

The most unreliable components in any electronic system will be the following (worst first):

1. Connectors and cables
2. Switches
3. Potentiometers and trimmers

Keep them in mind as your brainchild proliferates in complexity.

RF line filters

As we mentioned earlier, it is a good idea to use RF filters on the ac power-line inputs. These are manufactured by a number of companies, including Corcom, Cornell-Dubilier, and Sprague. They are

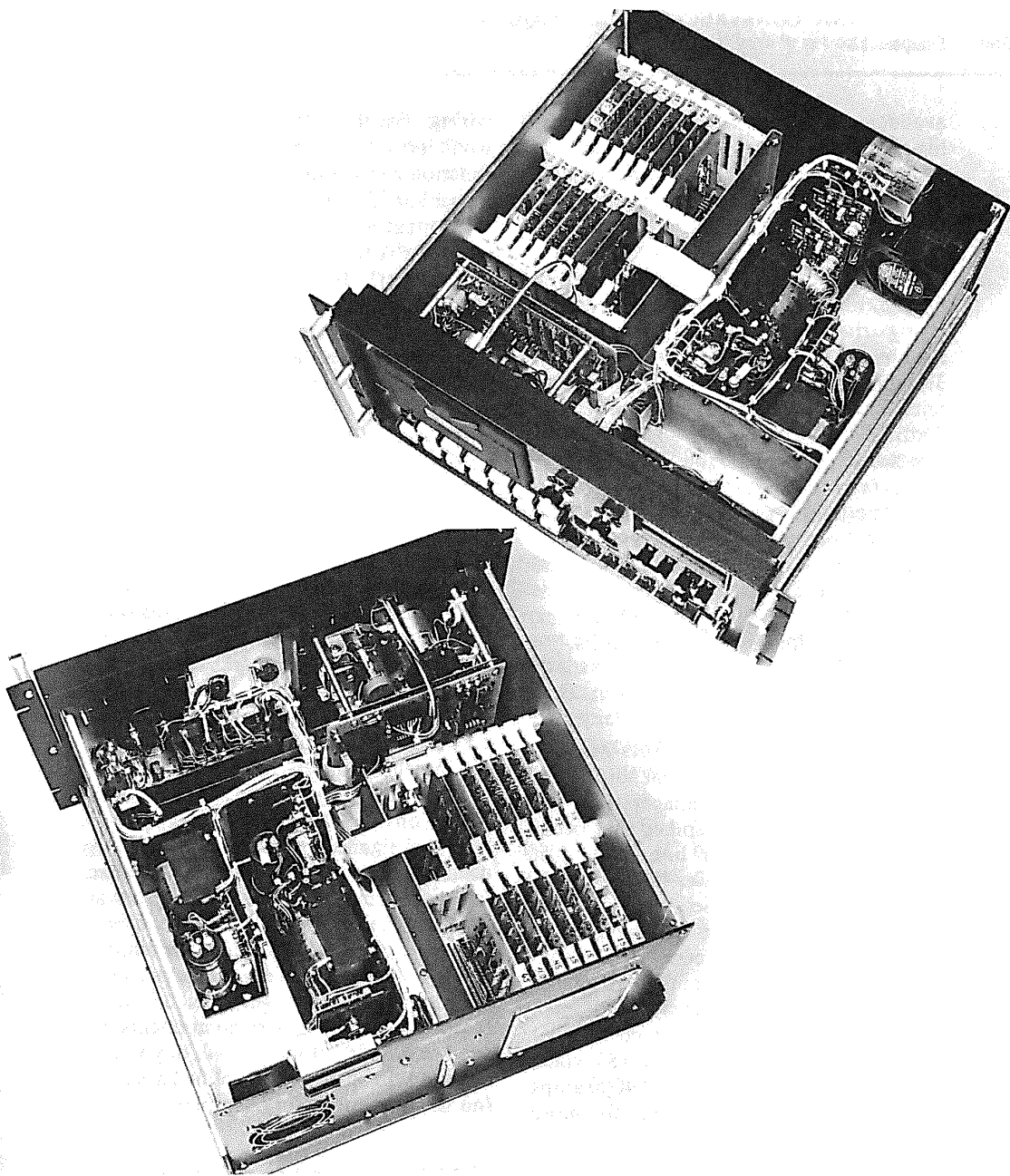


Figure 12.18. Views of a complete instrument (a digital cassette tape reader) illustrating several techniques of support and interconnection. Most of the electronics are housed in a card cage (with hand-wired backplane and mass-termination connections), and the electronics associated with the tape drive are on two boards near the drive motor (with mass-termination, in-line, and DIP plug connections). Adjustments and test points are accessible near the edges of each circuit card. Note the cooling path: Air is sucked in behind the card cage; it flows between the cards, then around the central partition and back over the power supplies before being blown out by the exhaust fan at the right rear.

859

available as simple modules with solder-lug terminals or in configurations that include an integral chassis-mounting ac line plug to mate with the standard IEC line cord. These filters provide excellent rejection of RF signals on the power line (as well as preventing their emission by the instrument itself), and they are also partially effective in reducing line transients. As an example, the Corcom 3R1 filter (rated at 3A, 115V) has 50dB rejection of RF at 200kHz and more than 70dB rejection for signals above 0.5MHz (see Section 6.11 and Table 6.3).

We favor the use of complete "power entry modules," rear-mounted units that combine 3-prong IEC power connector, RF line filter, fuse(s), line-voltage selector, and power switch. Look in the catalog pages of Corcom, Curtis, Delta, Power Dynamics, and Schaffner for representative units.

Transient suppressors

Power-line transient suppressors are also a good idea in any instrument to prevent malfunction (or even damage) from the occasional 1kV to 5kV spikes that occur on everyone's ac power line. You just put them across the power-line terminals downstream of the fuse; they act like bidirectional zeners with enormous peak current capability. They come in packages that look like disc ceramic capacitors or power diodes; the inexpensive and small GE V130LA10A, for instance, costs about a dollar, begins conducting at 185 volts, and can handle peak currents of 4000 amps (see Section 6.11 and Table 6.2 for more details).

Fusing

A line fuse is *mandatory* in every line-operated electronic instrument, without exception. As we indicated in Section 6.11, the wall socket is fused at a current designed to prevent fire hazard in the wall

wiring, typically 15 or 20 amps. That won't prevent a major disaster in a malfunctioning instrument, such as one with a power-supply capacitor failure, where the instrument may begin to draw 10 amps or so (more than 1kW heating in the power transformer). Important fact (learned the hard way by the authors): The lead from the power line goes to the innermost terminal of the fuse holder, so when you insert a new fuse, you can't get your fingers onto the "hot" terminal. Use a slow-blow fuse with rating 50%–100% greater than worst-case current drain of the instrument.

Cold-switching philosophy

Whenever possible, it is a good idea to avoid running logic signals or analog waveforms to panel controls; this is to prevent cross-coupling and signal degradation that otherwise might occur. Instead, run dc control levels to the panel switches and pots, and use on-board circuitry to handle the actual signal switching, etc. This is especially important in noisy environments, or when dealing with high-speed or low-level signals, since the dc control signals can be thoroughly bypassed, whereas fast signals cannot. For example, use select gates (multiplexers) rather than routing the logic signals through a switch, and use a voltage-controlled oscillator rather than an *RC* oscillator for panel control of frequency. The few extra components you need to do things this way will buy you increased reliability and simplified assembly (no shielded cable, for example).

12.14 Where to get components

Getting the parts you need to build some piece of electronic equipment can present some real difficulties. Most of the large electronics distributors have abandoned over-the-counter sales, making it nearly impossible for the small purchaser to go

down to the store and buy a few parts. Fortunately, the large (well-stocked) distributors will still take an order by telephone, with cash pickup at the "will call" counter. When playing this game, it is essential to know exactly what you want, by part number and manufacturer (for ICs, you may have to know the full part number, prefixes, suffixes, and all).

Many distributors are extremely hesitant to sell in small quantities, so you're often forced to buy at least five or ten of each item. Add to this the fact that a given distributor handles only a fraction of the brands you may need, and you're faced with a major chore. The consumer-oriented electronics stores (Radio Shack, etc.) will deal in small quantities and do have counter sales, but they tend to stock an extremely limited range of parts. The parts distribution system seems to be aimed at the industrial user, with his quantity ordering. Electronics manufacturers are treated well by the distributors, who pay them frequent visits, give them handsome catalogs and data books, and engage in competitive pricing.

Some special cautions are in order for buying ICs. Many kinds of ICs are manufactured without 100% testing; instead, a sample of each batch is tested, with the whole batch being rejected if the sample shows excessive failure rate. As a result, you can, on occasion, get a perfectly worthless chip straight from a reputable

manufacturer's production line. As a rough guide, you can expect up to 0.1% of new chips to be defective. That's not too serious, and in any case you can have your chips tested if a lower reject rate is necessary. Furthermore, all manufacturers test their LSI chips, and some manufacturers (AMD, for example) perform 100% testing of all their ICs.

A more serious problem arises when those rejected batches find their way into the hands of a "relabeler," a kind term for a junk peddler. Labeling machines are inexpensive; consequently, "counterfeit ICs" are all too common. Our experience has been that the large distributors (Arrow, Hamilton/Avnet, Newark, Schweber, and Wyle, to name a few) are reliable, at least for the brands for which they are authorized distributors. Surprisingly, most mail-order houses seem to ship good merchandise, often at very good prices, but there is an element of risk involved. Be suspicious of any IC without a date code. Because of the extra time and annoyance involved in finding bad ICs in a circuit, we generally recommend that you play it safe by using regular distributors for all IC buying, in spite of the generally higher prices. However, two mail-order houses that we have used successfully are Digi-Key (full-range catalog; Thief River Falls, MN) and Microprocessors Unlimited (computer chips; Beggs, OK).