

Designing and Creating Printed Circuits

WALTER SIKONOWIZ

TK
7868
.P7
S54



HAYDEN BOOK COMPANY, INC.
Rochelle Park, New Jersey

Library of Congress Cataloging in Publication Data

Sikonowiz, Walter.

Designing and creating printed circuits.

Bibliography: p.

Includes index.

1. Printed circuits. I. Title.

TK7868.P7S54 621.381'74 81-6524

ISBN 0-8104-0964-X AACR2

Copyright © 1981 by HAYDEN BOOK COMPANY, INC. All rights reserved.
No part of this book may be reprinted, or reproduced, or utilized in any form or by any electronic, mechanical, or other means, now known or hereafter invented, including photocopying and recording, or in any information storage and retrieval system, without permission in writing from the Publisher.

Printed in the United States of America

2 3 4 5 6 7 8 9 PRINTING

82 83 84 85 86 87 88 89 YEAR

9

Final Assembly

Component Insertion

After component-mounting holes have been drilled or punched, and the legend and solder mask have been applied to the bare board, the next step is component insertion. Either manual or automated insertion techniques may be employed, depending on the production rate and the nature of the circuit to be assembled. Manual methods require less in the way of hardware, but automated assembly offers the advantages of speed and consistency.

Manual Insertion

One of the basic tools needed for manual insertion is the mechanical lead former, which bends component leads to match board mounting-hole dimensions. Lead trimmers are necessary accessories as well, but beyond this not much more in the way of hardware is needed. The most important element of a manual insertion operation is the human being who does the work. In setting up an assembly station, the major concern is to make things as simple as possible for the worker.

Let us say that 50 components are to be inserted in a board of modest size. Our problem is to teach someone, who may not know a capacitor from a fire hydrant, how to place 50 plastic whatzits with wires sticking out in just the right places on the printed-circuit board.

The first thing to do is make a diagram showing each component in its proper place on the board. Assign a number to each component, and place that same number on the parts bin or tray where the component is stored. Not only do numbers assist in component identification, they also help to eliminate missed components, since the work flows in an orderly progression from component #1 to #50.

So far, so good. Now all the worker has to do is follow the diagram and put the components where they belong. Unfortunately, work is not going to progress very quickly because every time he inserts a component, the worker will have to refer to the chart. Even-

tually, he will become proficient because he will have memorized the chart, but this may require considerable time.

The key to efficient component insertion is repetition. When component #15 is inserted, it should be inserted into a number of boards in succession. After that, the worker can progress to component #16 and insert that into the same set of boards. In effect, the worker is not stuffing just one board at a time, but a whole set of them. He may refer to the chart when inserting a given component into the first board, but thereafter it is simple repetition. Consequently, the work flows without undue hesitation.

To facilitate the whole procedure, a turntable apparatus is commonly employed. Around the table's perimeter, the boards to be stuffed are mounted at regular intervals. The worker, seated before this turntable, simply spins it to gain access to all of the boards. Parts bins should be mounted within easy reach of the worker and restocked with components as needed.

When components are inserted, their leads will have to be bent to the proper dimensions by use of a manual lead former. To speed things up, however, component leads may be preformed by another worker using a hydraulic press especially designed for this purpose. Once all of the boards on the turntable have been stuffed, each one is flipped over so that protruding leads can be clinched.

Inspection at this point involves picking one board at random and ascertaining that each component occupies its rightful place. The intent here is to spot any consistent mistakes made during insertion. Since such mistakes would repeat from board to board, only one PCB need be checked. Further inspection is not warranted until the boards have been soldered.

A manual assembly program of the type just discussed is capable of yielding between 200 and 600 axial-lead-component insertions per hour. If a higher rate of production is needed, more assembly workers are added. When business is booming, and demand outstrips production, automated assembly techniques should be considered.

Automated Insertion

Machines for automated component insertion come in a wide variety of styles. The simplest of these do nothing more than jam a component straight down, leads first. It is up to the operator to see that a properly oriented circuit board is waiting when the component comes down. More sophisticated machines rely on a pantograph or computer to control component placement and insertion.

The relative ease or difficulty of automatic insertion depends to a large extent on the nature of the components to be inserted. Figure 9-1 shows three circuit boards, each bearing seven components. Board

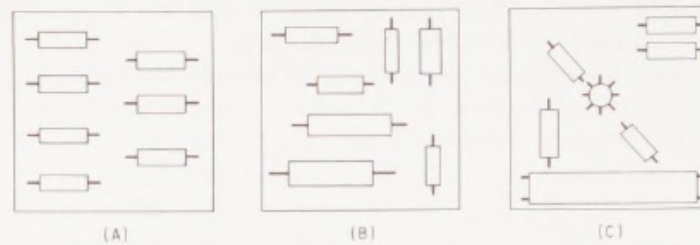


Fig. 9-1. (A) Single-axis, single-span. (B) Dual-axis, multiple-span. (C) Triple-axis, multiple-span. Difficulty of insertion increases from (A) to (C).

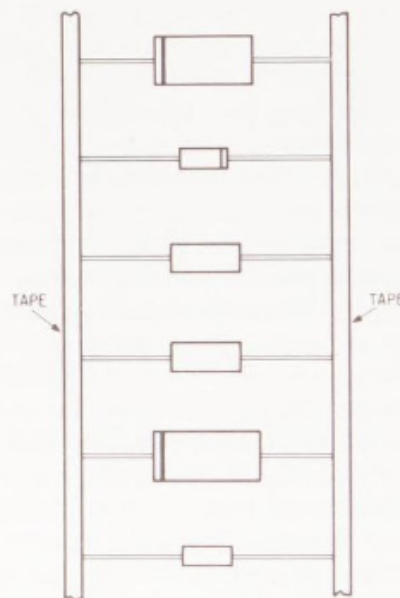


Fig. 9-2. Components taped for automatic insertion.

A is the easiest to stuff; board B is a little more of a challenge, while board C could be a real troublemaker.

Circuit board A is simple because it contains axial-lead components, all oriented in the same direction. Furthermore, all component lead spans (the distances from hole to hole) are identical.

The components on board B lie along two orthogonal axes. This means, in most cases, that after all the east-west components have been inserted, the board will have to be removed from the insertion machine, rotated 90° , and replaced to allow insertion of the north-south components. Note also that the components on board B have a variety of lead spans. This will require either a sophisticated, adjust-

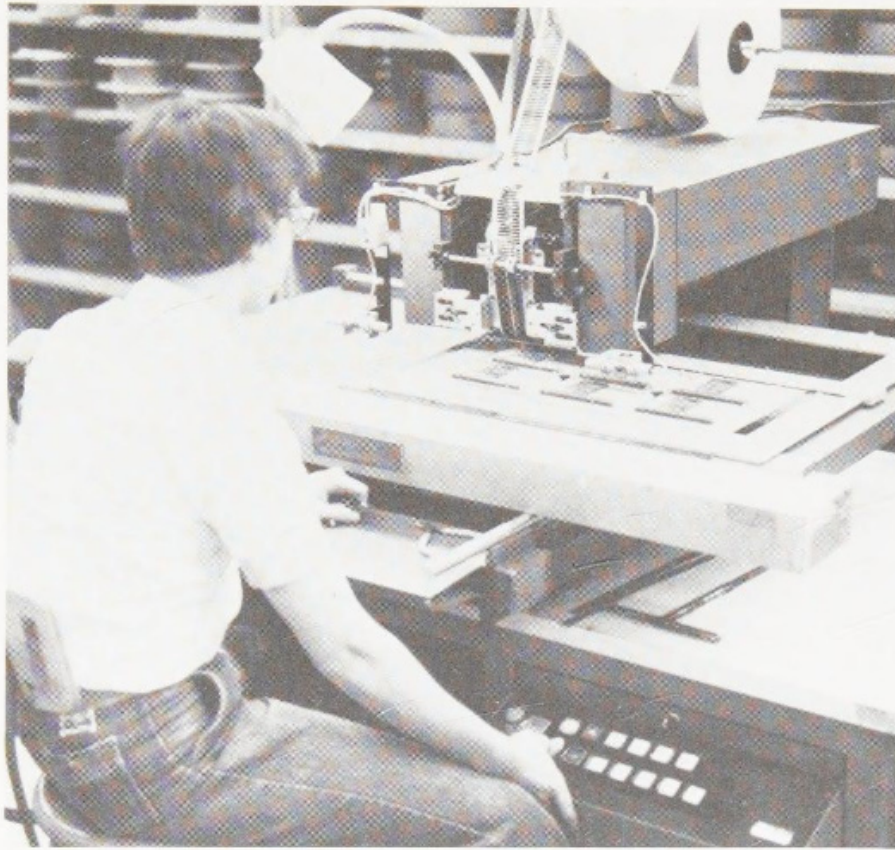


Fig. 9-3. Computer-controlled automatic insertion machine. (Courtesy Computervision Corp., Bedford, Mass.)

able insertion head or several insertion machines with fixed heads—one for each lead span.

Circuit board C introduces a third axis of component orientation. Most machines can accommodate two orthogonal axes without much trouble, but to insert components on a third axis will require manual assistance in most instances. Board C also contains components other than the simple axial-lead types present on boards A and B. The 8-lead spider-can IC and the large oblong component at the bottom of the board cannot be inserted by a mechanism designed for axial-lead components.

This brings us to the question of just what can be inserted automatically. Axial-lead components obviously must be insertable, since they have been discussed for several paragraphs now. Figure 9-2 shows the standard taping method used to feed axial-lead components

to the automatic insertion head. Here components of several different kinds have been sequenced in a predetermined order by a machine designed for this purpose. Automatic insertion of taped axial-lead devices is illustrated in Fig. 9-3; note the large reel of taped components sitting above the machine. As each component is inserted, its leads are cut and clinched by a mechanism beneath the board.

Insertion heads for axial-lead devices vary in their sophistication. The simplest head inserts components on a fixed span or center distance; in other words, the distance from mounting hole to mounting hole must be constant. Such a head is useful to dispense components of one particular type, or to insert components of different kinds which happen to have the same lead span. This will not be just a happy coincidence; in order to use a simple, fixed-center-distance head for axial-lead component insertion, the designer must plan on fixed spans for all components from the outset. Machines can be adjusted to a particular span, but this span cannot be changed during the course of insertion.

Most practical designs will require more than just one lead span. In that case, several insertion machines, each adjusted to a different lead span, could be used to stuff the board. Each machine would insert only that portion of the total component count which its head could accommodate.

Some insertion heads can insert components on two different center distances. Appropriately enough, these are known as dual-center-distance (DCD) heads. Even more sophisticated insertion heads can be found on computer-controlled insertion equipment. Known as variable-center-distance (VCD) heads, these can be programmed in 0.001-in. increments to insert a wide range of axial-lead components. With both DCD and VCD heads, the center distance can be changed during the course of machine operation.

In addition to axial-lead devices, the familiar dual-in-line integrated circuit can be inserted by machines especially designed for this purpose. Instead of being taped, DIPs are dispensed from magazines. Insertion heads can accommodate dual-in-line devices with as many as 40 pins (Fig. 9-4). Note how the insertion head grips the integrated circuit along its sides. Obviously, ample space must be allowed on either side of the IC so as not to obstruct the insertion head's jaws.

Transistors can be inserted in either of two ways. First, they may be inserted radially by a mechanism that grasps the transistor's can and pushes it leads-first into the board. Or, the transistor's leads can be spread apart and taped in a manner similar to that used for axial-lead components. In either case, a special insertion head must be employed.

Sockets and jumper wires can be automatically inserted, too, provided the right machine is used. In fact, with a little ingenuity, almost

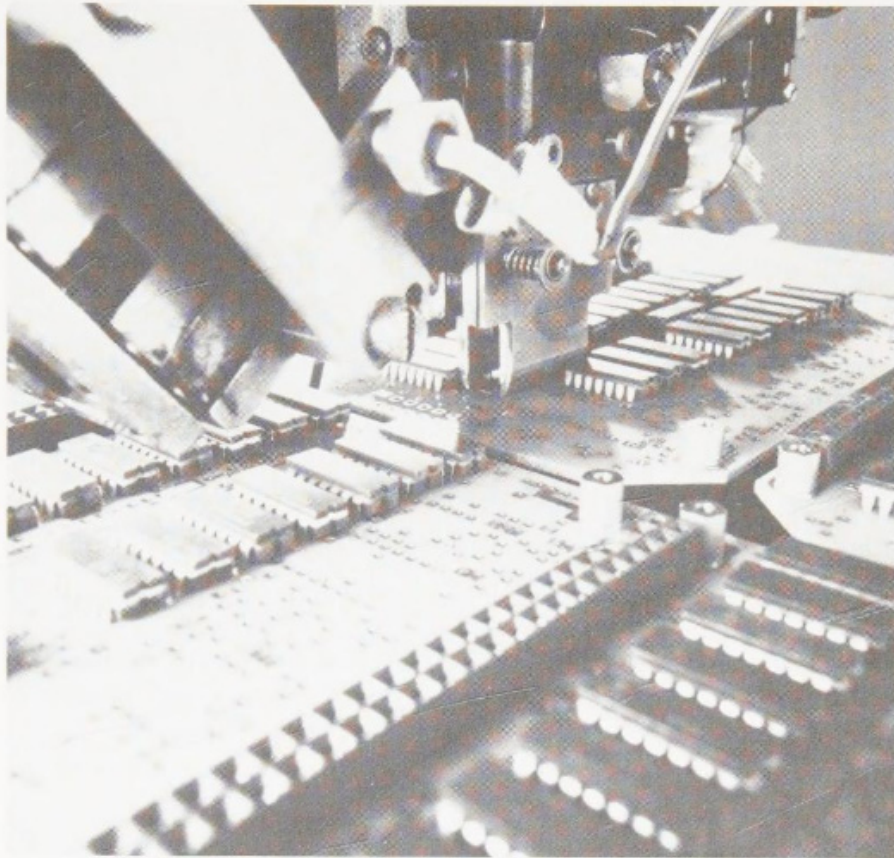


Fig. 9-4. Automatic DIP-insertion head. (Courtesy Computervision Corp., Bedford, Mass.)

anything can be accommodated by standard or modified automatic equipment. However, size and weight limitations may prove insurmountable in the case of certain components—for instance, transformers. Whatever the machines cannot insert can always be installed by hand after the bulk of the work has been done automatically.

As noted at the outset, machines for automatic component insertion come in a wide variety of styles. The simplest machines require that the operator position a board beneath the insertion head and initiate the insertion cycle. Machines of this sort incorporate fixed-center-distance heads and are used to insert components of just one kind.

More sophisticated machines operate under the guidance of a pantograph assembly. Reliability, flexibility, and moderate cost make these machines extremely popular. Pantograph operation entails guiding a stylus over the surface of a drilled aluminum template. When the stylus drops into a particular hole of the template, the op-

erator presses a button and initiates the insertion cycle. Taking its cue from the pantograph stylus, the machine inserts a component into the location on the printed circuit that corresponds to the position of the stylus on the template. Machines of this sort are most economical to use if components have been laid out in accordance with a standard Cartesian grid. Production rates with semiautomatic equipment can be as high as 2,000 component insertions per hour, which is 5 to 10 times faster than manual insertion.

Computer-controlled machines, like the one illustrated in Fig. 9-3, are the elite class of the automatic-insertion equipment hierarchy. Since VCD heads enable equipment of this sort to accept a wide variety of component sizes and lead spans, one computer-controlled insertion machine may replace and outperform several less sophisticated pieces of equipment. Rotary work tables, which are available as options on some of these machines, enable a board to be rotated in increments of 90°. This allows the insertion of axial-lead components oriented along either the X or Y axis without manual assistance. The fastest computer-controlled machines can insert approximately 10,000 axial-lead components per hour, and dual-head designs double that figure. Programming is simple and fast, so design changes are easy to make. Furthermore, if a CAD/CAM system was used to design the printed circuit being manufactured, system software may be able to generate the necessary insertion-control tapes automatically.

Soldering

Solder provides the necessary mechanical and electrical bond between component and PCB. Joints are formed by heating the solder, the component lead, and the printed conductor above the solder's melting point. Upon cooling, solder forms a metallic bond between the component lead and the foil. Solder metal used in electronics is an alloy of tin and lead having a composition close to that of the eutectic ratio: 63 percent tin/37 percent lead.

The distinguishing characteristic of solder having the eutectic composition is that melting is well defined. The change from solid to liquid (or vice versa) takes place abruptly at one temperature, 361° F; there is no semisolid or pasty state between liquid and solid. This is a desirable property in electronic assembly because it results in strong, stress-free joints. Therefore, the alloys used in soldering will have the eutectic composition or something close to it (i.e., 60/40).

Flux Characteristics

Soldering can be done on a variety of scales. At the bottom of the ladder is the conventional soldering iron, which serves primarily for

repair work or for the soldering of sensitive components. Production-line soldering is normally done by dipping or wave techniques, both of which will be discussed later. At the moment, our chief concern is with that important preliminary to soldering: flux application.

The purposes of fluxing are myriad: Adsorbed gases and metallic oxides on the surfaces to be soldered react with the flux, which lifts these contaminants away from the surface. At the same time, the blanket of flux smothers the hot metal surfaces to be joined, thus cutting off oxygen which might reoxidize the surfaces. In addition, the flux helps promote wetting of the metallic surfaces by lowering the solder's surface tension. The ideal flux would also be completely unreactive and nonconductive after soldering.

Fluxes used in electronic work may belong to one of two general categories: the rosin-based fluxes or the water-soluble organic-acid fluxes. The former are for all intents and purposes inert after soldering. However, because there exists the possibility of metallic corrosion and degradation of the laminate's insulation resistance, especially under environmental extremes, these fluxes are customarily removed after soldering. The second class, water-soluble fluxes, is much more reactive, and flux residues must be removed immediately after soldering.

The rosin upon which the first class of fluxes is based occurs as a natural product of the pine tree. Alternatively, synthetic rosin, known as resin, may be employed as a substitute. Natural rosin is a diverse mixture of chemical compounds, and of these the most significant for soldering purposes are the organic acids—for example, pimeric and abietic. By itself, rosin is a weak flux suitable for soldering only the cleanest of surfaces. So, to the rosin base manufacturers add various activators, which boost the reactivity of the flux. The typical activator is a thermally unstable salt which breaks down at soldering temperatures to yield hydrochloric acid.

Rosin fluxes are available in three different grades. The R-type fluxes are essentially rosin dissolved in an appropriate vehicle, which makes the flux fluid and easy to apply. Since no activators are present, the metals to be joined must be extremely clean. Consequently, type R fluxes see comparatively little use in the cruel world of the assembly line.

The mildly activated rosin fluxes (type RMA) consist of rosin, vehicle, and activators such as amine hydrochlorides. Since these fluxes are more reactive than the R types, they make soldering easier. The residues of RMA flux are nearly inert because of the low activator concentration. If an electronic assembly can be soldered with RMA flux, so much the better. However, a large proportion of production-line work requires a more potent flux.

Type RA activated rosin flux is the strongest of the three grades, since it has the highest concentration of activators. This makes soldering simple with surfaces of average contamination. However, since the residues of type RA rosin flux are more apt to cause eventual problems, care must be taken to remove them completely. RA fluxes find extensive application in PCB assembly.

The water-soluble organic-acid fluxes are much more corrosive than any of the rosin-based fluxes. Typical constituents of a water-soluble flux are organic acids (glutamic, stearic, phthalic, etc.), compounds with active halogen groups (aniline hydrochloride, hydrazine hydrochloride, etc.), and a vehicle such as glycol or certain other organic solvents. Water may be present as well, but it should be noted that water is never the chief solvent, since it results in a flux that spatters excessively during soldering. When water-soluble flux is used with solder-plated boards, it is advisable to choose a flux with a minimal halide content to limit the formation of lead chloride on the conductors. Residues from these fluxes should be removed from the board immediately after soldering; failure to do so invites corrosion and the formation of insoluble salts. Whereas the activators in an RMA or RA rosin-type flux are tied up in an inert residue after soldering, those in a water-soluble flux remain relatively unhindered and reactive.

Flux Application

Liquid flux can be applied by a variety of means. For low-volume soldering applications, in which boards are dipped manually into a solder pot, brushing or dipping may be suitable means for applying flux. Obviously, these techniques are fairly primitive, and they will not adapt well to automated, high-speed soldering.

For such purposes, foam fluxing and wave fluxing are the primary methods of application. The central component of any foam fluxer is a porous stone, which is submerged in a reservoir of flux. Air under moderate pressure is forced through the stone into the flux, where it creates a foam with fine bubbles. This foam rises within a tube or chimney and eventually overflows from the chimney's open top. Boards are skimmed over the top of the chimney to pick up a layer of flux that covers all areas to be soldered.

The alternative application method, wave fluxing, relies on a standing wave of flux to coat the circuit board's underside. Inside the fluxer, a pump-and-nozzle assembly creates a linear standing wave wider than the board to be fluxed. The circuit board is skimmed along so that its bottom just grazes the crest of the wave and thereby acquires the desired coat of flux.

In an automatic soldering machine, fluxing is customarily followed by preheating. The purpose here is to raise the temperature so

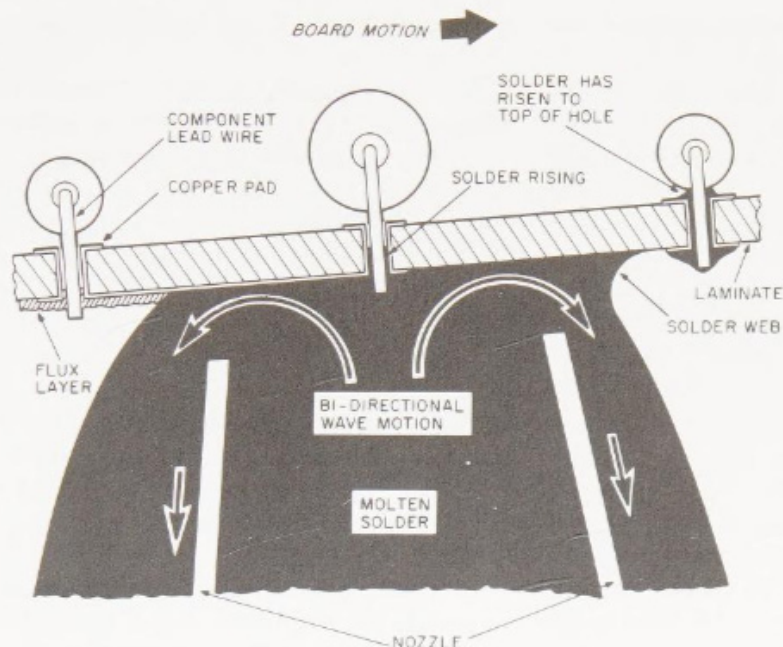


Fig. 9-5. The solder wave.

that the flux becomes reactive enough to clean the metal surfaces. While the soldering process itself will heat the flux, preheating minimizes the time the board must linger in the solder. Consequently, soldering can proceed at a faster rate in conveyorized soldering systems. When soldering is done by manual dipping, preheaters are not required. The preheat temperature is usually 190° to 200° F for rosin-base fluxes, which melt in the vicinity of 160° F.

A further benefit of preheating is that it drives off the flux vehicle and any other moisture trapped within the board's holes. Without preheating, these liquids would rapidly vaporize during soldering to cause spattering and blowholing (the blasting away of a certain portion of the solder at a joint). Blowholes may also be the result of organic materials occluded within the plated deposit of a through-hole. Preheating will not cure this nuisance; instead, regular carbon treatments to purge the plating baths of organics are recommended.

Soldering Equipment

For modest production rates, manual soldering with a dip pot will probably handle the workload. This dip pot is nothing more than an open, heated reservoir for solder. After fluxing, the board is clamped by tongs and dipped carefully into the solder. Caution is advised lest the solder overflow onto the topside of the board. Afterward,

the board is slowly and evenly withdrawn from the pot and set aside to cool.

More substantial workloads generally utilize wave-soldering techniques. In principle, wave soldering is very much like wave fluxing, since both employ a standing wave of fluid as a means of application. Figure 9-5 illustrates the essentials of modern wave-soldering procedure; the wave itself is shown here in cross-section.

The necessary standing wave of molten solder is created by means of a pump, which forces solder through a nozzle with a relatively narrow linear orifice. The wave created as the solder exits the nozzle can be as high as 0.75 in. and as wide as 24 in. in current machines.

As Fig. 9-5 shows, the board is dragged by a conveyor so that the crest of the wave makes contact with the board's underside. As the board moves through the wave, solder wets the pads and rises in the through-holes by capillary action. At the point where the board exits the wave, a solder web is formed. If this web is too large, it may separate and leave unwanted solder on the board.

The solder web may be minimized in several ways. Inclining the conveyor slightly enlists the assistance of gravity to pull the web back into the wave. Inclines from 4° to 9° are used in practice, and they are quite effective. Another stratagem employs oil to lower the solder's surface tension which, in turn, reduces the size of the web pulled from the wave by the moving board. It also minimizes the tendency for icicles of solder to form on the joints. Oil may be mixed with solder in the pump or injected into the wave. In either case, finely divided droplets of oil become dispersed in the solder, thereby diminishing the solder's surface tension.

Soldering with oil has other beneficial side effects as well. For one thing, the blanket of oil that covers the surface of the solder limits the access of atmospheric oxygen to the solder. As a result, the formation of dross—an undesirable surface residue of lead and tin oxides, among other things—is curtailed. Since solder joints on the board also are protected by a thin film of oil, they remain bright and shiny due to the inhibition of oxidation. Since oil reduces the tendency for icicle formation, conveyor speeds may be increased, resulting in an attendant increase in productivity.

Oil is not without its disadvantages, however. The obvious one is that the oil must eventually be cleaned from the surface of the printed circuit. Another is that oil must be changed on a regular basis to prevent carbonization and contamination of the solder. Some oils are worse than others in this respect. For wave-soldering purposes, mineral oils and waxes give the best performance.

In an effort to circumvent the need for oil, some equipment manufacturers employ a supported, inclined wave which, because of the

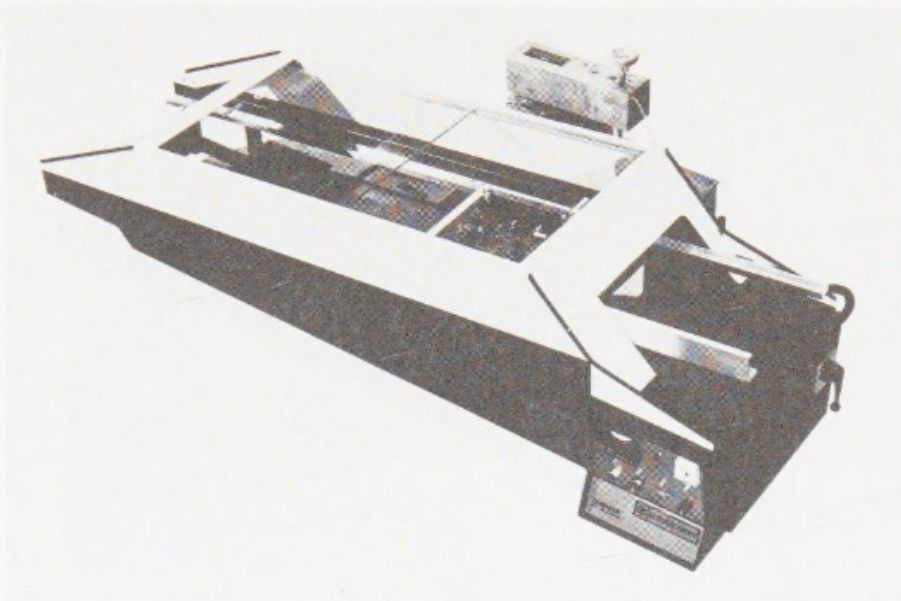


Fig. 9-6. Minipak bench-type wave-soldering system. (Courtesy Electrovert Inc., Elmsford, NY)

fluid dynamics involved, has a minimal tendency for webbing and icling. The key to oil-free soldering lies in careful tailoring of the wave's shape and velocity distribution. Both the bench-style wave-soldering machine in Fig. 9-6 and the floor-standing model in Fig. 9-7 employ specially contoured waves of the type just discussed. Except for the wave-generation mechanism, oil-free systems are structurally similar to the machines that use oil. Note the inclined conveyor, which is clearly evident in Fig. 9-7.

In the final analysis, whether a system solders with oil or without makes little difference. Both systems are capable of producing good results when the machines are properly maintained.

Solder Masks

As circuit dimensions become smaller and smaller, it becomes increasingly more difficult to avoid the problem of solder bridging between conductors of a printed circuit. If a set of closely spaced, parallel conductors goes through the wave in such a way that the conductors all run parallel to the wavefront, bridging is almost certain to occur, regardless of the type of soldering system used. Solder masking can be of benefit here, since the mask shields all areas of the circuit's underside except for the pads. Consequently, solder never touches

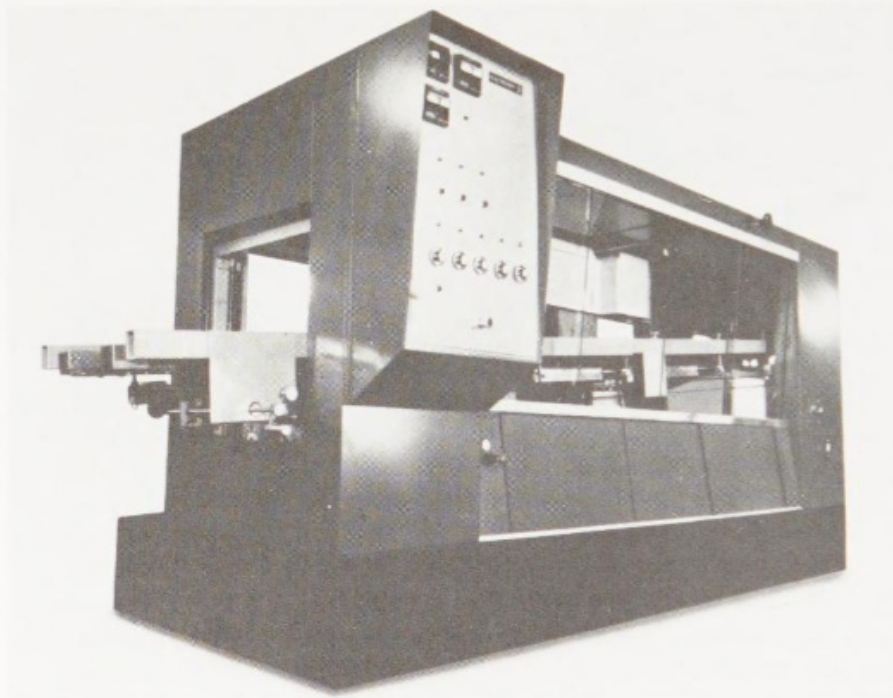


Fig. 9-7. Ultrapak floor-mounted wave-soldering system. (Courtesy Electrovert Inc., Elmsford, NY)

most of the conductors, and the incidence of bridging is reduced accordingly.

Another benefit of solder masking is reduced solder consumption, an obvious economic advantage. At the same time, less copper builds up in the solder pot because of reduced contact between conductors and solder. Since copper concentrations above 0.25 percent degrade solder-joint strength, maintaining a low copper concentration in the solder pot is a matter of practical importance.

When solder masks of the permanent type are employed, they may be left on the board's surface after soldering to protect the printed wiring. Solder masks of the temporary type may also be applied—for instance, to shield gold-plated contact fingers from solder. (Gold deposits dissolve completely in solder after less than two seconds of immersion.) Once soldering is complete, the temporary mask is simply peeled away.

Clean-Up

Flux residues remaining on the board will have to be removed quantitatively, and this can be a challenge. Chapter 10 explores this

problem in detail; suffice it to say here that the rosin fluxes may be removed by either organic solvents or water-based alkaline cleaners. The water-soluble organic-acid fluxes require an alkaline cleaner or, optionally, neutralization in ammonia water.

Board Testing

Automatic testing done on the soldered and cleaned circuit board may be of either the in-circuit or the functional type. In-circuit testing checks individual components on the PCB to see that they have been properly inserted and that they work as intended. Functional testing is much broader in scope: It checks to see whether the board as a whole performs to specification. If a fault is detected, in-circuit testing may then be needed to locate the cause of the malfunction. Consequently, some manufacturers of automatic test equipment offer machines capable of performing both in-circuit and functional analyses.

The most notable feature of in-circuit automatic test equipment (ATE) is the bed-of-nails fixture, a grid of spring-loaded contact pins. During testing, the circuit board is registered in place on the bed-of-nails and held down by a pneumatic ram or vacuum plate. Each pin of the test fixture establishes contact with at most one node of the board under test. A minicomputer, which can either drive or sense each node through appropriate interface circuitry, performs the tests and interprets the results. Programming by the user defines the test sequence and parameters. The test will attempt to verify the correct placement, orientation, and operation of each element on the circuit board.

Fixtures for functional testing consist of edgeboard connectors, IC-probe clips, or some combination of these devices. The purpose here is to tap the input and output lines of the board under test and to exercise them via a minicomputer. The user programs the system to perform relevant tests and interpret their results. In essence, automatic test equipment of this type operates by comparison, and signals from the printed circuit are checked against known-good and known-bad signals stored in memory.

For low production volumes and relatively unsophisticated circuit boards, conventional manual test methods may be satisfactory. Here, greater reliance is placed on a human inspector, who performs the test and interprets its results. With the proliferation of low-cost microcomputers, however, even the smallest of companies can afford to incorporate some degree of automation in their testing programs.

Conformal Coating

The purpose of a conformal coating is to cover a printed-circuit assembly and protect it from environmental hazards such as moisture,

dirt, and chemicals. Of these, moisture protection is probably of the greatest general significance. Ideally, a conformal coating covers not only the circuit board but the components mounted on it as well. Since conformal coatings are rarely thicker than 0.005 in., they cannot be expected to provide the same protection against physical abuse afforded by encapsulation. On the other hand, components on the circuit board remain accessible for adjustment or repair—something not possible after a board has been encapsulated.

Conformal coatings may be applied by a variety of methods including vacuum deposition, dipping, or spraying. The first of these techniques is used with coatings deposited from the vapor phase. Dipping and spraying are employed with liquid conformal coatings, which require time to cure into hard, protective films. For printed-circuit applications, the most important coatings are epoxy, parylene, acrylic, polyurethane, and silicone. Of the five, only parylene is applied by vacuum deposition; the rest are dipped or sprayed.

Parylene films are true conformal coatings, since vacuum deposition enables uniform coverage of even the most intricate surfaces. Since the coatings obtained can be as thin as 250 angstroms, they are well suited for high-density printed and hybrid circuits. Parylene is available in three grades: C, N, and D. C is the general-purpose parylene coating; N is used where very low dissipation factors are required, and parylene D is optimized for high temperatures. Common solvents as well as most acids and bases have little effect on parylene.

Acrylic conformal coatings offer the advantages of long potlife and ease of application. When repair work is needed, acrylic conformal coatings are easily removed from the board with methylene chloride or some related chlorinated hydrocarbon. This is a mixed blessing, since it means that an acrylic-coated board is likely to lose its protective coating to chlorinated organic solvents as well as others. In most applications, however, this will have little practical significance. Curing is by solvent evaporation at either room or elevated temperature.

Epoxy coatings are available as two-component systems with a limited potlife because of polymerization. Conformal coatings of the epoxy type possess excellent resistance to chemicals, humidity, and mechanical abrasion. Curing occurs in less than three hours at elevated temperature, or in about one week at room temperature. Because the epoxy resin is so impervious, it cannot be chemically stripped for repair work. Instead, it must be burned away with a soldering iron.

Polyurethane coatings are available as either single- or two-component systems. The single-component types cure in about one week at room temperature, while two-component polyurethanes require several hours at elevated temperature. Single-component types are

easier to work with and feature a longer potlife, which results in less waste. Like the epoxies, polyurethane resins offer excellent resistance to chemicals and humidity. When circuits must be repaired, polyurethane conformal coatings may be stripped chemically or burned away with a soldering iron.

Silicone resins see their greatest application as conformal coatings for circuits subjected to high temperatures. These coatings provide excellent resistance to humidity and chemicals even at temperatures as high as 200° C. Removal of a silicone coating for repair work is difficult, however, and requires that the film be cut away.

Since one of the purposes of a conformal coating is to protect the board and components from harmful chemicals, it makes no sense at all to apply the coating to a board that has not been thoroughly cleaned. In fact, putting a coating over a contaminated board can do more harm than good in some instances. Generally, all traces of flux and ionic contaminants will have to be removed, as will oils, fingerprints, and whatever else manages to get smeared on a board during assembly. This is the topic of the next chapter.