CHAPTER 7

PRODUCTION OF PRINTED CIRCUIT BOARDS

7.1 INTRODUCTION

The completed PCBs will normally represent the greatest part of the value of the electronic system hardware. An efficient and defects free production is essential. The process is increasingly automated, but generally there still is needed some additional handwork.

7.2 PRODUCTION OF HOLE MOUNTED PCBs

In this section we shall describe the process for purely hole mounted boards, with emphasis on the automated parts. Figure 7.1 shows the process schematically.

7.2.1 Component mounting

Sequencing and mounting of axial components:
Axial components for automatic mounting are usually delivered on a tape (please refer to Section 4.9). The first step in the mounting of axial components is to make a new component tape, specifically for the most efficient mounting of a given type of PCB. In the new tape, the needed component types are placed in the order that makes the subsequent placement optimised, as illustrated in Figures 7.2 a) and b). In the sequencing machine, all needed component tapes are stored, please refer to Figure 7.3. In the example the machine cuts out one component of type 5 and places in the new tape, then one of type 14 and so on, covering all components needed for one board. Then it starts over, mounting the same series of components for the next board, etc. Figure 7.3 shows a sequencing machine, with capacity of some 80 component types, and speed up to approximately 5000 components per hour.

The new tape is placed in the axial insertion machine, see Figure 7.4, that will mount typically 3 - 5000 components per hour. The process, see Figure 7.5, includes cutting the component out of the tape, forming the leads to fit in board holes, mount, and then cut and clinch the leads to avoid that the component falls out during handling.

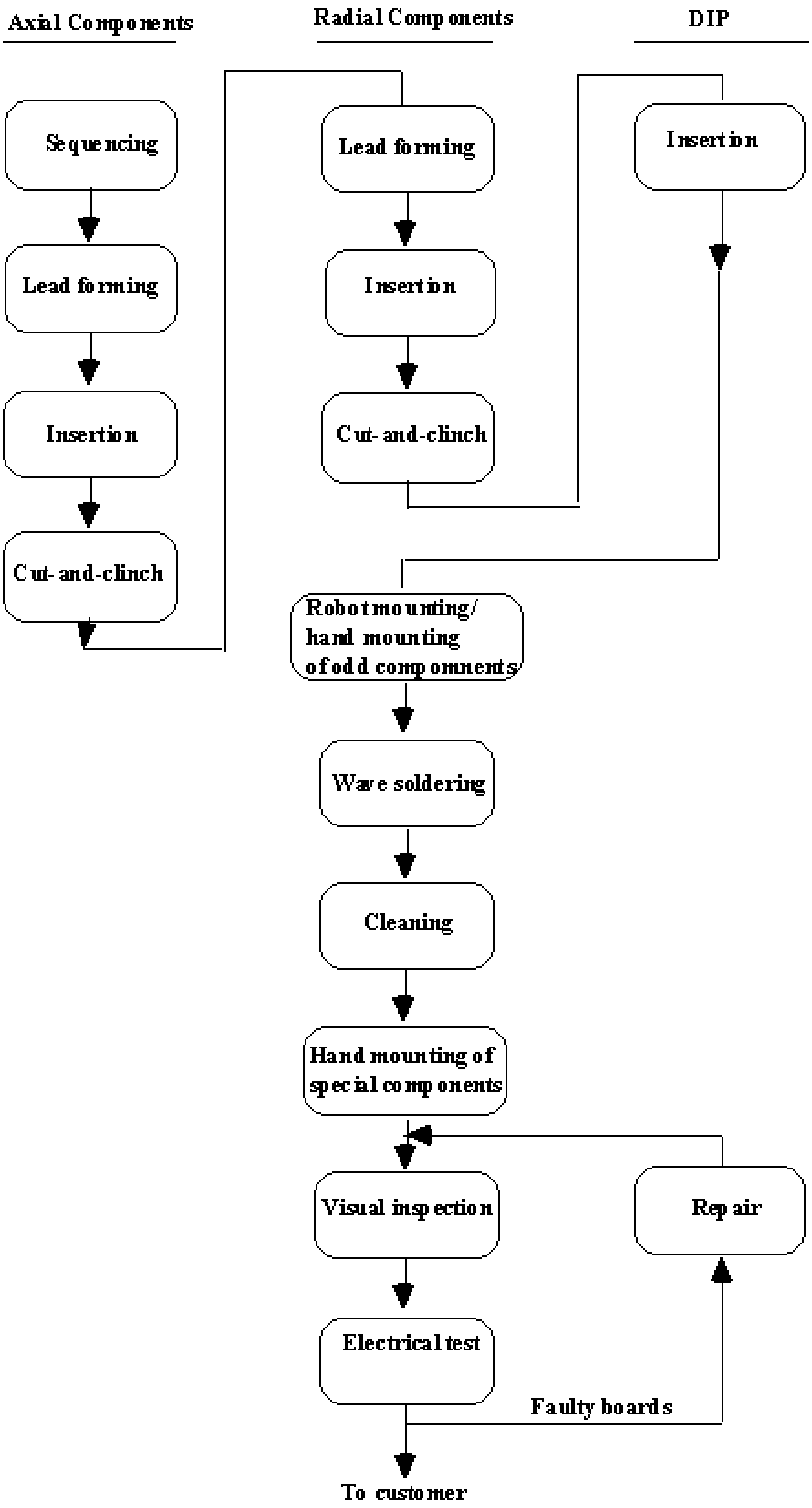

Fig. 7.1: The process for production of hole mounted PCBs.

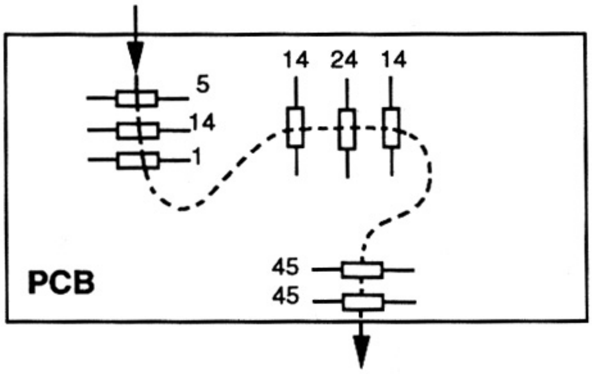

Fig. 7.2 a): Schematic example of the most efficient sequence of mounting the components of a particular PCB.

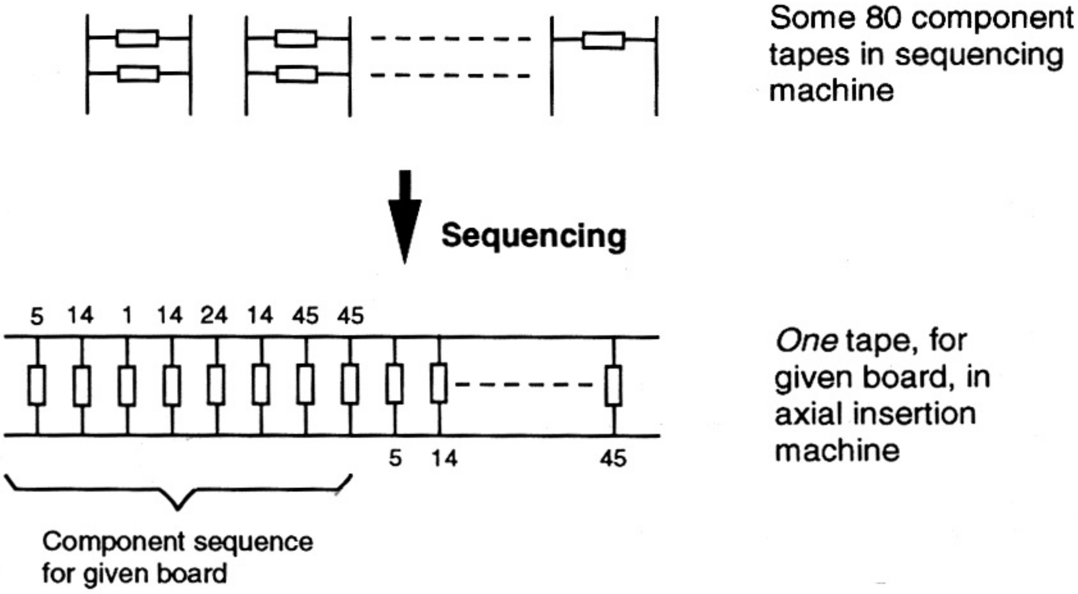

Fig. 7.2 b): The principle of sequencing.

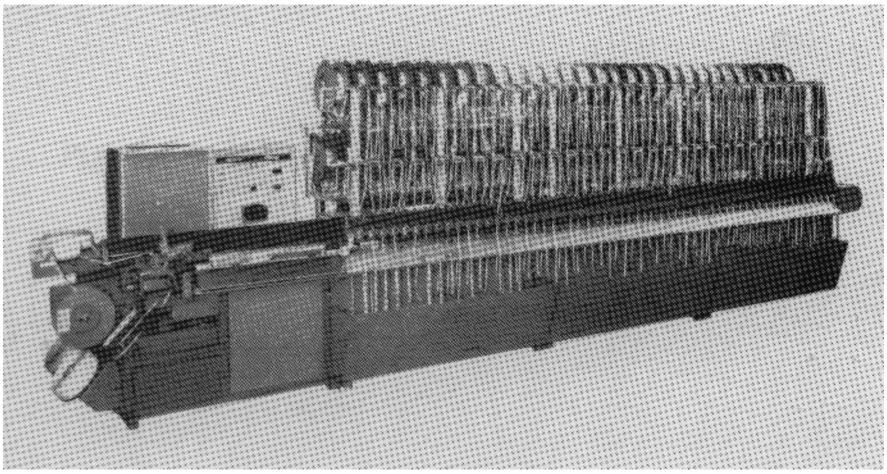

Fig. 7.3: Sequencing machine.

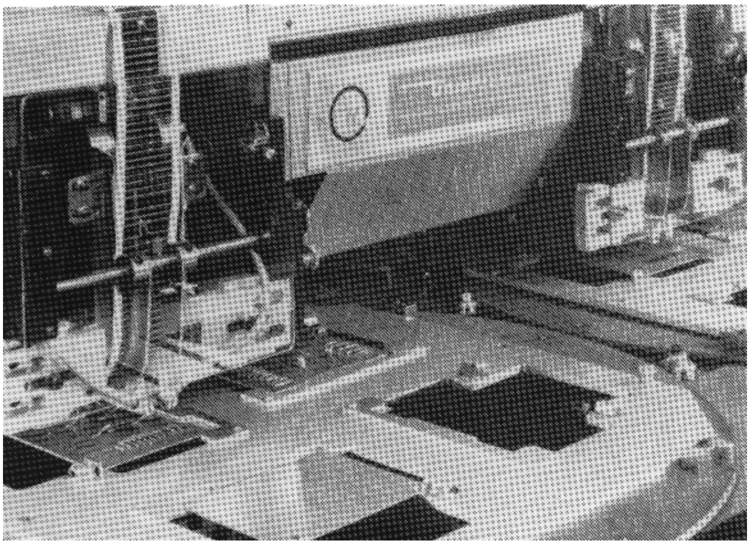

Fig. 7.4: Axial inserter with two mounting heads.

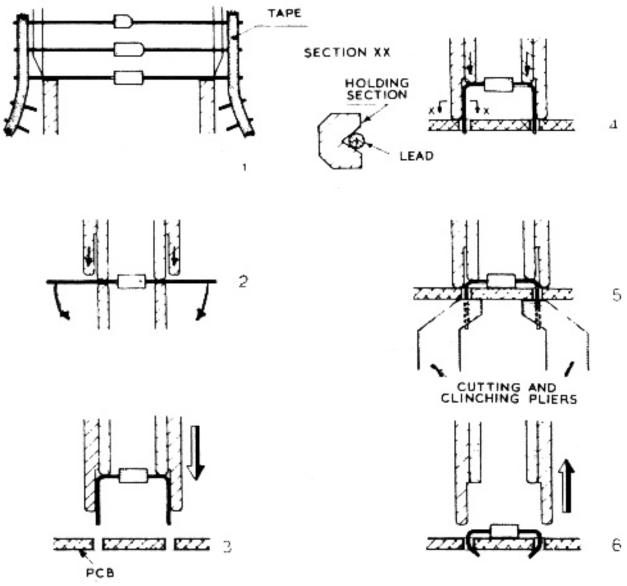

Fig. 7.5: Simplified process in the axial inserter: a): Cutting the components from the tape, b): Lead bending, c) - d): Insertion, e): Cut and clinch, f): Return to starting position.

Radial and DIP IC mounting

For radial components, sequencing is not used. The machine for radial insertion will often have stored the tapes with all components needed, and a transport mechanism ("pater noster" transport band) bringing each component from tape position to insertion position.

DIP ICs are normally delivered in plastic tubes, sticks, which are stored in a separate DIP inserter, see Figure 7.6. The holder of the tubes moves back and forth, to place the right stick in position for mounting. At the same time, the PCB is moved to get the location of the component in the right place underneath.

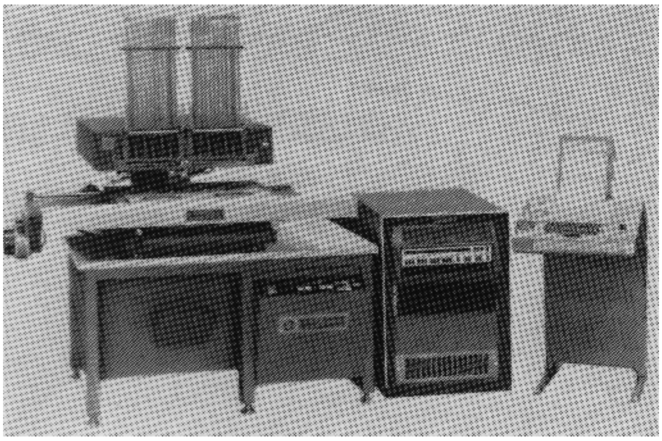

Fig. 7.6: DIP inserter.

Manual and semi-manual mounting

For odd component types and for prototype series hand mounting is still used. "Light boards" are also used, see Figure 7.7. A light board has stored a set of boxes containing the needed components for manual picking by an operator. A computer controlled lamp above the board focuses a light spot on the PCB, while the correct box of the component to be mounted is marked by the light from an LED. When the component is manually placed, the operator pushes a pedal, and the computer shows next component and place on the PCB where it should be placed. A trained operator can mount some 100 components per hour with this aid, with considerably reduced fault frequency.

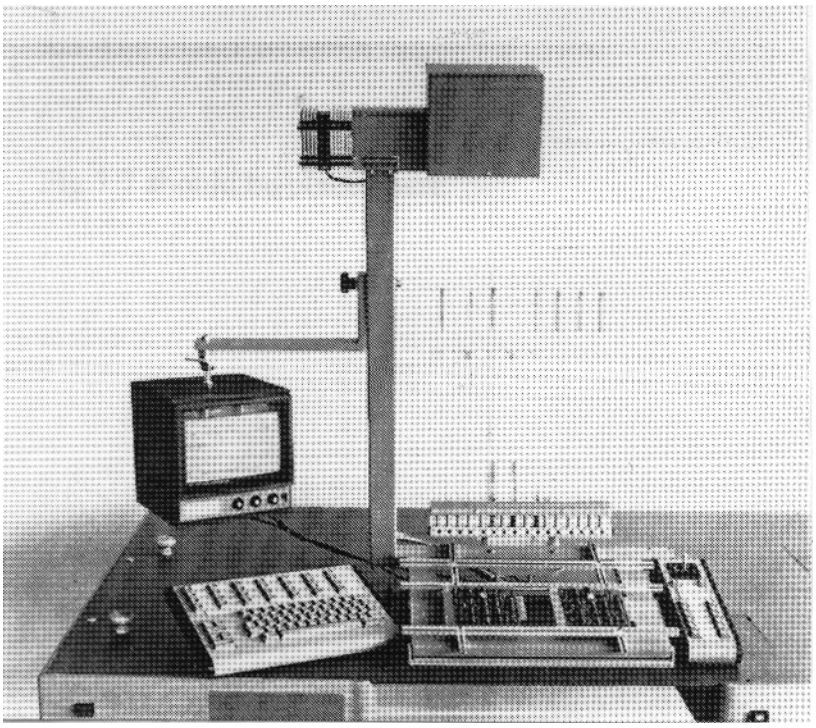

Fig. 7.7: Manual mounting board with light guide.

7.2.2 Wave soldering

Mass soldering of hole mounted board is normally done by wave soldering. Figure 7.8 shows a wave soldering machine.

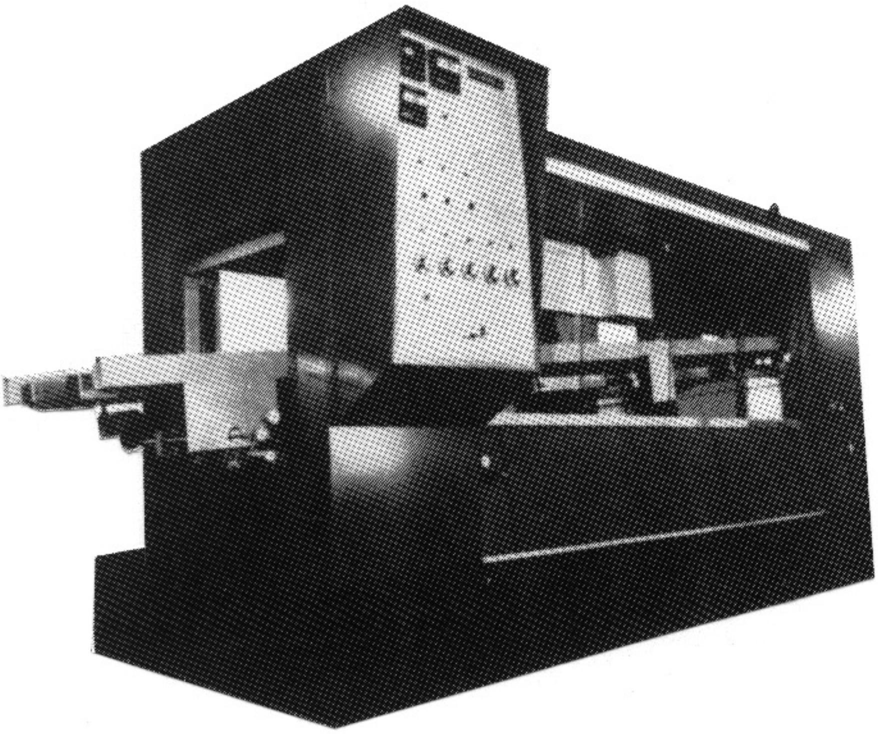

Fig. 7.8: Wave soldering machine.

A conveyor transports the PCB through the machine. The boards of different sizes are placed in jigs of one standard size. First, the board will pass through a fluxing unit. Here solder flux is applied to dissolve oxides and improve wetting and solderability, please refer to Section 3.10. Most commonly, the flux is pumped up together with air, making a foam that hits the underside of the board, please refer to Figure 7.9 a). The flux contains alcohol or an other solvent to give the right viscosity. The solvent in the supply evaporates in time and is automatically replenished to keep the density of the flux bath constant, see Figure 7.9 b).

Then the board passes over a preheating zone where the board is heated by a hot metal plate underneath through radiation and convection, to approximately 100 °C. The purpose of this is:

- To activate the chemically active parts of the flux (chlorine is dissociated from the organic component of the flux)
- To evaporate the chemically inactive solvents
- To heat the board slowly to avoid thermal shock in the subsequent solder bath.

After the preheating the board goes through the solder zone, passing a wave of molten solder, see Figure 7.10. The solder is normally close to a eutectic composition of tin and lead, 63 % / 37 % by weight, as described in Section 3.10. The melting temperature is ca. 180 °C, but a typical solder bath temperature is 230 - 250 °C.

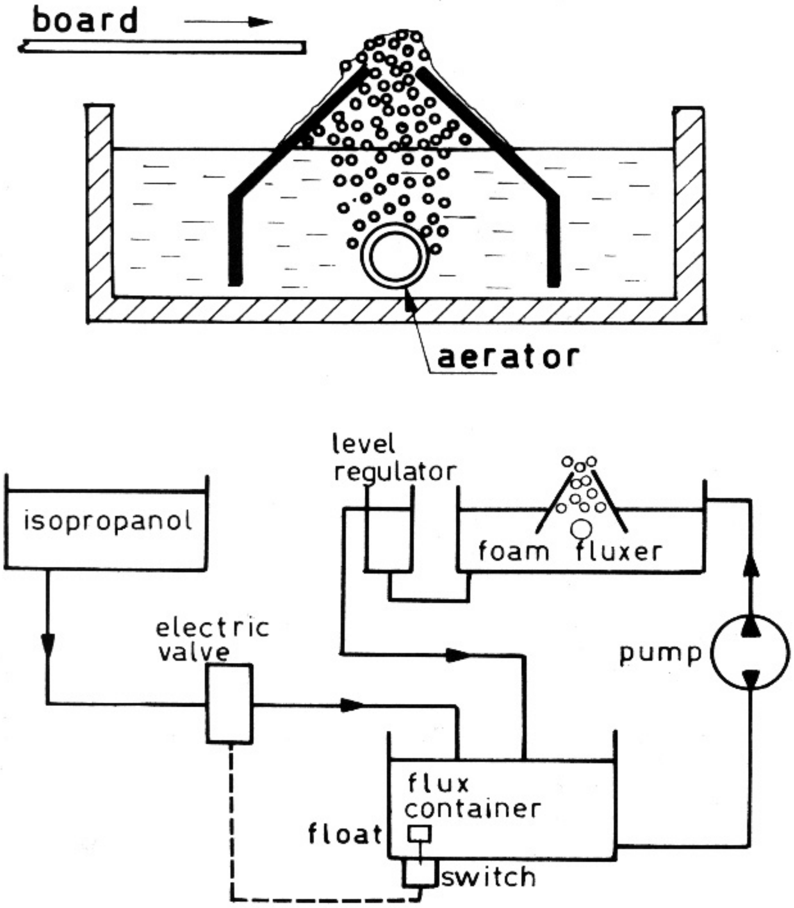

Fig. 7.9: a): Principle of foam fluxer, b): Control system for density and level of the flux bath.

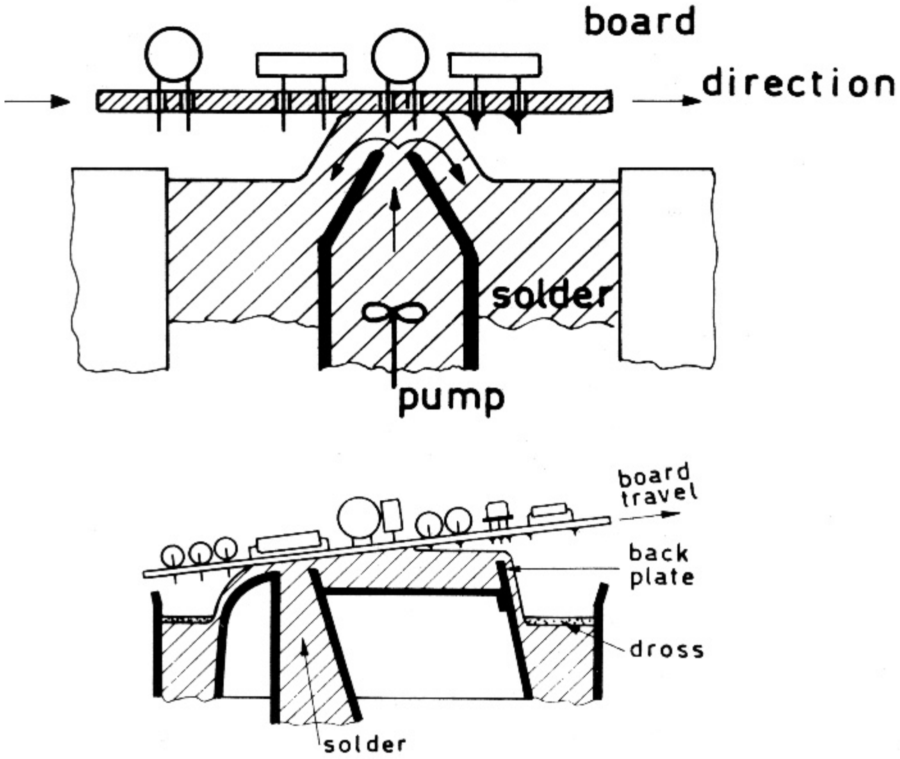

Fig. 7.10: a): Principle of wave soldering, b): The real shape of the wave.

The solder is located in a container. A pump forces it up through a slit, forming a wave at right angle to the motion of the board. The shape of the wave is important for the solder quality, and so is the speed of board motion, the contact time and the solder temperature.

The wave is adjusted such that the component lead ends and the under side of the board are wetted by the wave. The solder is also pulled up into the component holes by capillary forces.

7.2.3 Cleaning

After soldering, the board may pass directly into an in-line cleaning machine, see Figure 7.11 a), if cleaning is needed. This depends on the type (activity) of flux used and the required quality of the product for which the board is to be used. Strong fluxes leave corrosive residues that need to be cleaned, please refer to Section 3.10. Previously freon was the cleaning agent of choice for rosin flux on most products. It is now abandoned due to its harmful effect on the ozone layer of the atmosphere, after it evaporates and escapes. Instead alcohol may be used, water (with detergent), or newly developed cleaning agents. More efficient cleaning is obtained by ultrasound agitation of the cleaning bath. Vapour cleaning after immersion is also used, see Figure 7.11 b).

7.2.4 Repair

Removal of faulty components is done by firstly removing the solder. It can be done by manually heating each solder joint above melting temperature, and then sucking away the solder by a special suction tube on the solder iron, connected to a vacuum pump. After removing of the component a new part will normally be mounted and soldered in place manually.

7.2.5 ESD precautions

Precautions against electrostatic discharges (ESD) must be taken when ICs and some transistor types (MOS) are handled, please refer to Section 4.8. Today ESD protection is considered necessary throughout the production process, from test of incoming components to repair and packing of finished boards, or systems in some cases. The main precautions are:

- Conductive materials in floors
- Grounded equipment
- Grounded personnel.

An ESD protected working place is shown in Figure 7.12. It has grounded, conductive mats on the floor, on the seat of the chair and on the working bench, grounded soldering iron, etc., a grounded wrist band for the operator. The grounding is made through a resistor (typically 1 Mohm) to avoid big current spikes that may harm the operator. The operator is also grounded by wearing conductive shoes or conductive tape inside and outside the shoes. ESD protected packing of the components and board was described in Section 4.8 - 4.9.

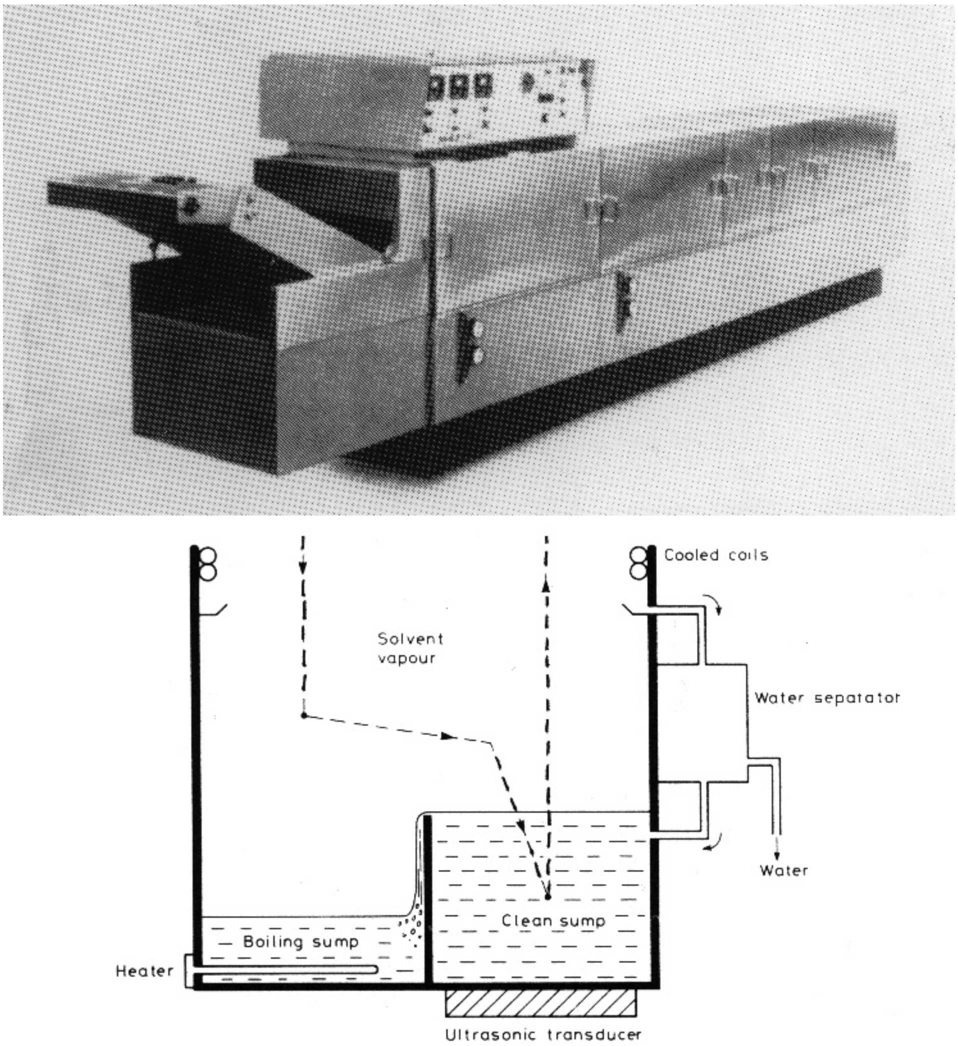

Fig. 7.11: a): Industrial in line cleaning machine, b): the principle of ultrasound and vapour cleaning.

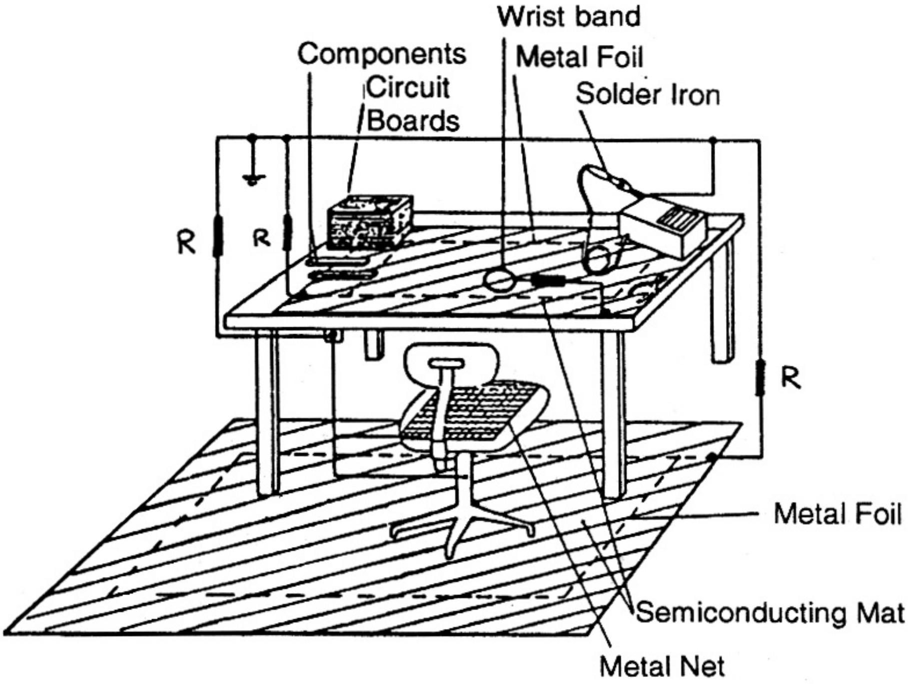

Fig. 7.12: An ESD protected working space. The resistors R normally are 100 Kohm - 1 Mohm.

7.3 PRODUCTION OF SURFACE MOUNTED PCBs

The use of surface mounted components requires a more complex production process than pure hole mounting. There are different possible alternatives of components on one or both sides of the board, please refer to Figure 6.5. Most often we have a combination of SMDs and hole mounted components. The production sequence must be carefully considered and chosen already in the design phase. Here we show the basics of the wave soldering and preceding gluing process as well as several reflow soldering processes. Then component mounting will be described. In Section 7.5 we shall return to the complete processes for different types of SMD or mixed SMD/hole mounted boards.

7.3.1 Gluing and wave soldering of SMD components

Gluing

When the SMDs are wave soldered they are hanging underneath the board and must be fastened in advance by an adhesive. There are three main methods of applying the adhesive, Figure 7.13:

- Screen printing
- Application by dispenser
- Pin transfer.

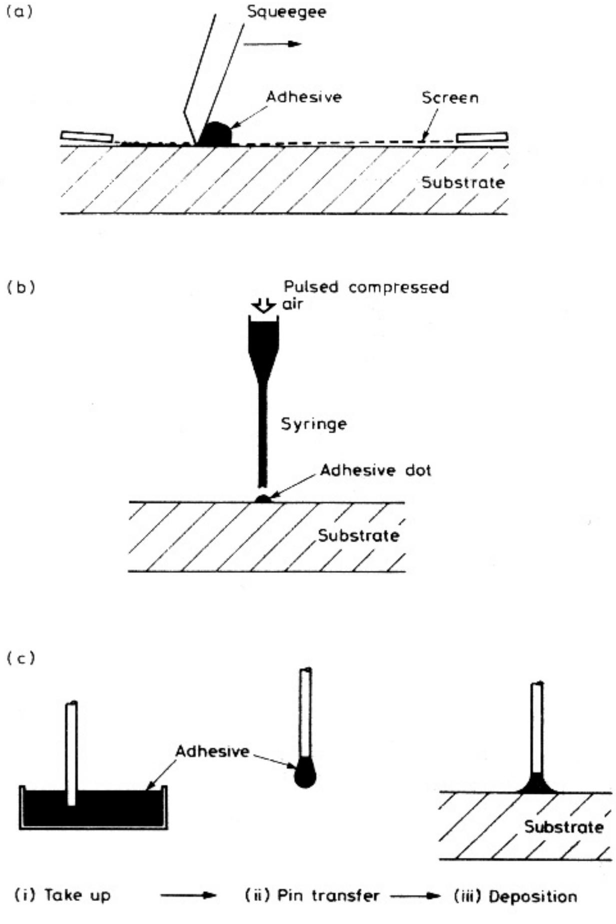

Fig. 7.13: Application of adhesive for SMD mounting by a): Screen printing, b): Dispensing and c): Pin transfer.

Screen printing of adhesive is done in a manner similar to printing of solder paste, see below. A stencil of 0.5 mm Teflon is often used. The holes may be drilled by a numerically controlled drill, with drill information for position and size of the holes from the CAD system. The size should be bigger for heavy components and components with high stand-off, needing more adhesive. This way of adhesive application is efficient, as all adhesive dots are placed simultaneously. However, the board surface must be plain, and this process must be done before any component is mounted.

The most used method is application by dispenser. The syringe may be mounted on the pick-and-place machine, and a controlled pressure is applied in the dispenser each time a glue dot is placed. The amount of glue is controlled by the time and pressure.

Pin transfer consists of a pin that is first dipped into a pot of adhesive, then touching the board and leaving an adhesive dot at the point where the component will be mounted. Pin transfer is suited for very high volume production. In that case, a jig is made with a matrix of pins in the positions of all components to be glued. The jig is lowered into a big pot of glue and all adhesive dots are then deposited simultaneously.

The demands on the adhesive and the gluing process are high. The adhesive must not flow onto the solder pads, otherwise the solderability is ruined. It must wet the component well and immediately hold the component sufficiently well for the handling necessary until curing of the adhesive. It must have a long shelf life with stable viscosity, yet cure fast for a high volume production. Viscosity and rheology must be stable, assuring reproducible amounts of deposited adhesive. When repair is needed, the adhesive must soften during the repair processing (see below).

The adhesives are one- or two component heat curing epoxy, UV curing acrylic or epoxy. A problem with two component adhesives is that the curing proceeds fast when the components are mixed. Storage time for mixed adhesive is short (a couple of days). Therefore, one component adhesives are used the most.

Heat curing often takes place in an IR in-line furnace, of the type used for IR reflow soldering (see below). Typical curing conditions are 150 °C for 1 - 2 min. UV curing takes place in an in-line equipment, containing a powerful UV lamp. For UV curing to be possible, parts of the adhesive dots must "see" the UV light. The curing takes only a few seconds.

Wave soldering

Wave soldering is done in the same way as for hole components, but it is more critical. Problem areas are: not wetted surfaces, generation of solder bridges and thermal stress on the components.

The component bodies cast shadows for themselves and their neighbours, see Figure 7.14, and the simplest solder machines give unreliable contact between the solder areas and the wave. To remedy this, many machines have two waves, see Figure 7.15. The first is a turbulent wave, reaching and wetting all corners and crevasses. However, it will create solder bridges. The second, called the "lambda wave" is a gentle wave that removes the superfluous solder. In addition to the solder process conditions, also the dimensions of the solder pads are very important for defect free soldering.

There are several other principles for SMD wave soldering machines. The "jet wave" moves in the opposite direction to the board at high velocity. In some machines, a vibration is generated in the solder. Solder bridges may be blown off with a stream of hot air, "air knife", and oil can be mixed with the solder to reduce dross formation.

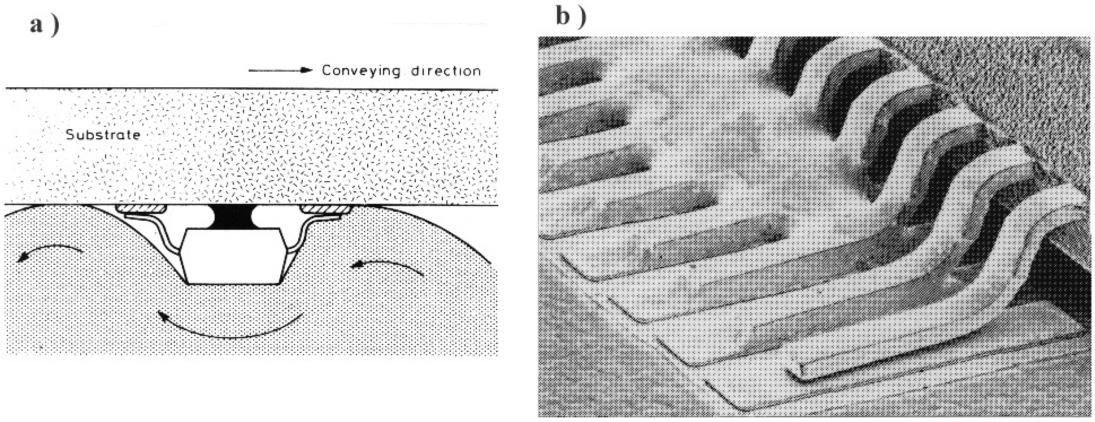

Fig. 7.14: a): Shadowing in SMD wave soldering, b): Solder bridging on fine pitch package.

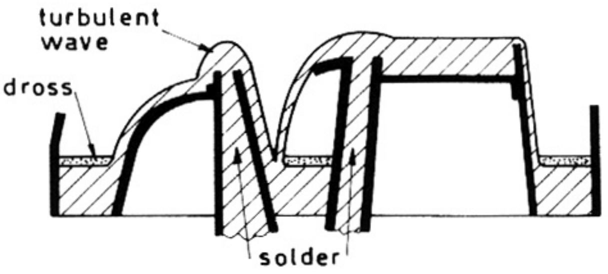

Fig. 7.15: Double wave for SMD soldering.

Figure 7.16 shows a typical temperature profile in a double wave solder process, as measured inside an SMD. The peak temperature is 230 - 250 °C, and the time during which the component is above 200 °C is 5 - 10 sec.

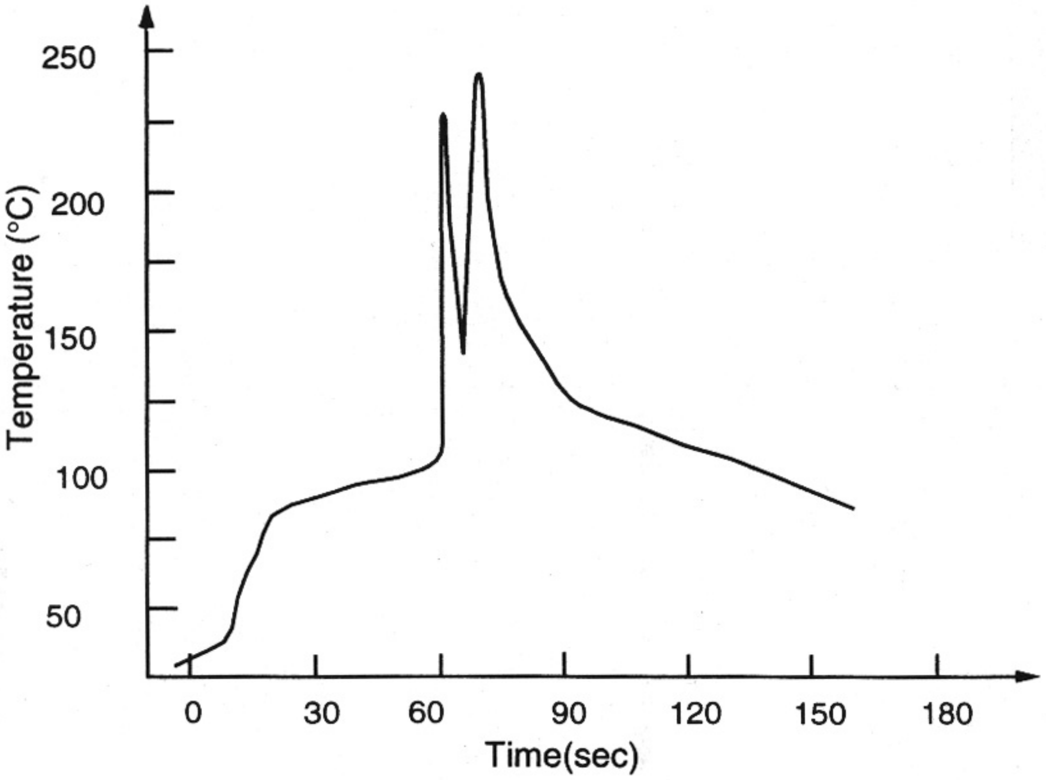

Fig. 7.16: Temperature profile during wave soldering in a double wave machine.

Limitations to wave soldering of SMDs

The shadowing effect depends on the shape of the component and its orientation with respect to the direction of motion. Not all components are suitable for wave soldering. IC packages with terminals on all four sides - flatpacks and chip carriers - should not be wave soldered. SO packages should only be mounted with their axis of symmetry along the direction of motion during soldering, or preferably not at all. High passive components should have their length axis perpendicular to the direction of motion, please refer to Figure 6.6. Other details are also given in Section 6.3.

As the SMDs are submerged in the molten solder, they are exposed to a tough thermal stress. Not all component can stand this, and the information from the component manufacturer should be consulted.

7.3.2 Solder paste deposition and reflow soldering

For reflow soldering the solder metal is first deposited on the PWB, localised to the solder lands. The solder lands should preferably be covered by a thin layer of solder metal from plating or hot air levelling in advance, to obtain good wetting please refer to Section 5.6. Then additional solder metal is applied in the form of screen printed paste. After component mounting (see below) on the solder paste the soldering is done by heating until the solder melts and wets component terminals and PCB solder lands. Various methods are used for the heating. We shall start describing the solder paste and deposition.

Solder paste

The paste, see Figure 7.17, consists of:
- Solder particles
- Flux
- Solvents.

The metal content of the paste is 85 - 90 % by weight. The alloy in the solder particles is the same as desired in the solder fillet, e.g. Sn 63 / Pb 37 % by weight. It is common to have 2 % Ag in the solder to reduce leaching of Ag from capacitor component terminals, please refer to Sections 3.10 and 4.7. The particles are sifted to give large or small average size, depending on the components to be soldered. For standard SMD ICs and passives larger than 0603 50 - 75 µm average size is used. For fine pitch components smaller particles are preferred. However, they have a larger surface-to-volume ratio and tend to oxidise more, giving poorer wetting and solder balling (see below) unless a stronger flux is used.

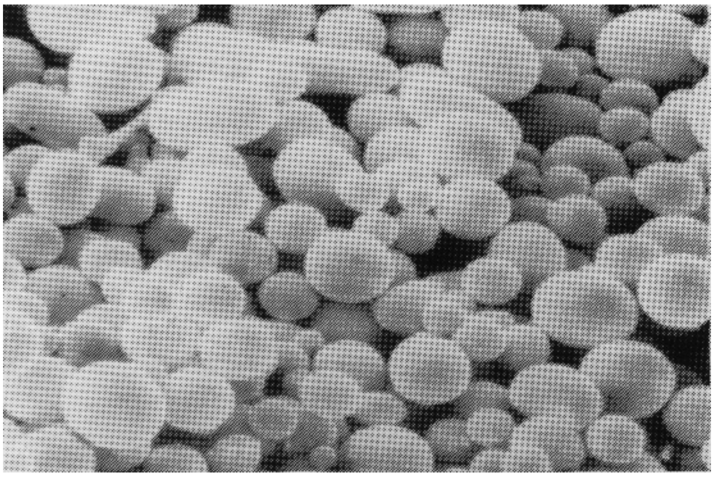

Fig. 7.17: Microphotograph of Multicore solder paste type Sn 62 RMA B 3. The designation means 62 % by weight of Sn, 35.7 % Pb, 2 %, Ag, 0.3 % Sb, RMA flux, 75 µm average particle size, 85 % metal content, viscosity 400 000 - 600 000 centipoise.

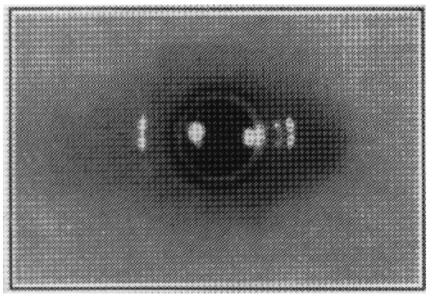

Fig. 7.18: Test of solder paste: The paste is printed through a circular opening with a diameter of 5 mm, in a 200 µm thick stencil. After reflow, the paste should melt into one body, without any particles spreading out.

The solder paste changes rheology and other properties in storage and has a limited shelf life. It may be tested by printing through a circular hole in a plate.

After reflow all solder should melt together, with no solder balls outside, see Figure 7.18.

The printing screen or stencil

The most common way of applying solder paste is by screen printing through a screen or a stencil, see Section 3.5. The screen is made of woven stainless steel filaments or polyester. The screen is covered by an organic film, with openings where the paste is to be deposited. The patterning of the organic film is done by photolithography, see Section 3.5.

Woven screens are simple and cheap, but they have some drawbacks: Due to the filaments, see Figure 7.19, and the slight deformation when the screen is pushed down by the squeegee, the definition is limited. Fine screens with thin filaments and a high mesh count must be used for high definition screens. They are easily broken, giving delay and extra cost in the production. For large areas, the screen tends to stretch unevenly during printing, also limiting the precision of paste deposition near the edges.

The volume of paste deposited depends on the thickness of the organic film, the opening area, and the fraction of the screen that is covered by the metal wire, please refer to Section 3.10. This volume is important, determining if the solder fillet will be lean or rich.

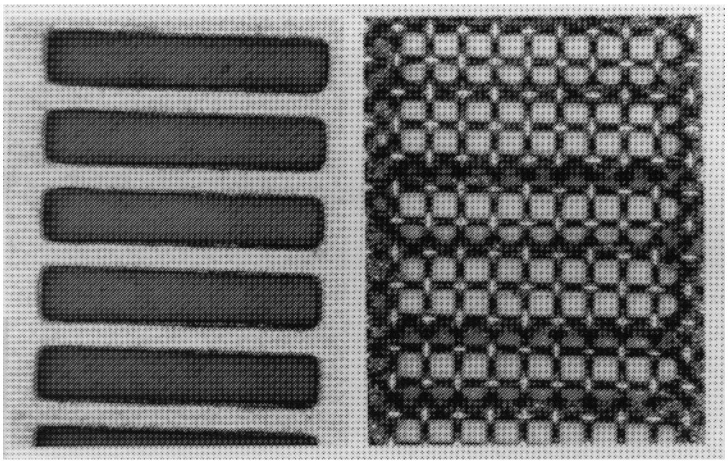

Fig. 7.19: Detail of printing screen and printing stencil with fine line printing pattern.

Use of a thin metal foil for solder stencil gives significantly better precision and reproducibility in the solder printing, and the stencil lasts many more prints. The openings are normally etched out. Typically the thickness is 150 or 200 µm. To reduce overetch the etching is done simultaneously from both sides, see Figure 7.20. The openings are the same size as the solder lands for large solder lands, somewhat smaller than the lands for fine pitch components. An alternative is to thin down the stencil over areas where there are fine pitch components, see Figure 7.21. The holes may also be drilled.

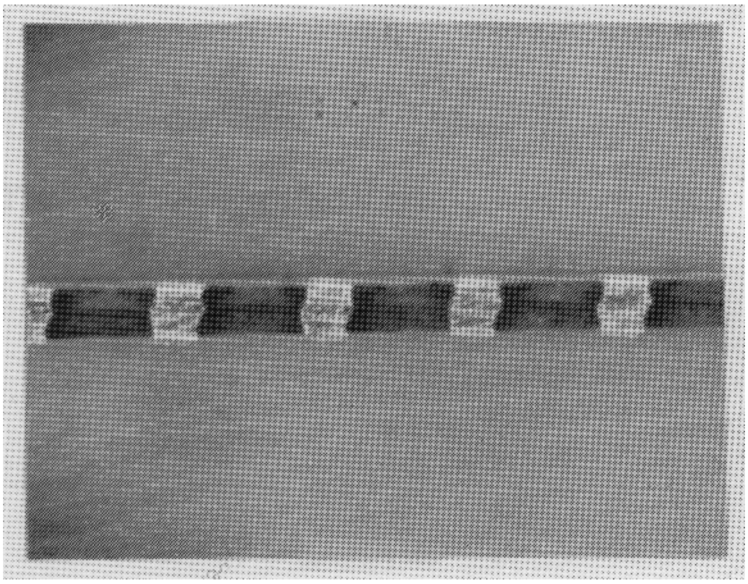

Fig. 7.20: Detail of printing stencil with fine pitch printing pattern: cross section of a stencil etched from both sides, with an acceptable, small amount of offset (40 x magnification).

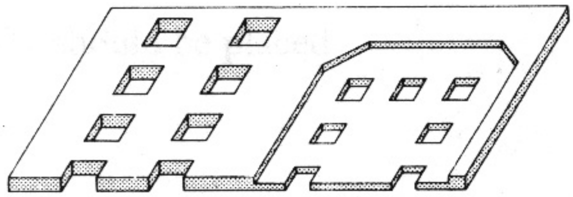

Fig. 7.21: Two steps printing stencil.

Brass, stainless steel and beryllium copper are the materials most used. The latter is best and most expensive. Alternatively the stencil may be made out of polyester (Mylar) or Teflon, see Figure 7.22, and the holes may be drilled or punched by a numerically controlled punching tool.

The printer and printing process

Figure 7.23 shows a relatively simple equipment for printing solder paste. The boards are manually placed in the printer, their position is determined by holes and guiding pins, or by the contour of the boards and guiding edges. Bigger printers are automatic, with cassette-to-cassette feeding, or in-line, so that the boards go on directly to component mounting. Advanced printers have an electronic vision system, recognising a set of fiducial marks etched in the copper. The position of each board is then adjusted very accurately relatively to the screen.

For prototypes, the solder paste is most often placed by a manual dispenser. A computer controlled solder paste dispenser may also be part of the pick-and-place machine.

After the deposition of solder paste and placement of the components, the paste should be dried, to activate the solder flux, and to evaporate solvents to make the paste more tacky. It takes place in a heating cabinet or in the entrance of an in-line solder furnace, at around 100 °C.

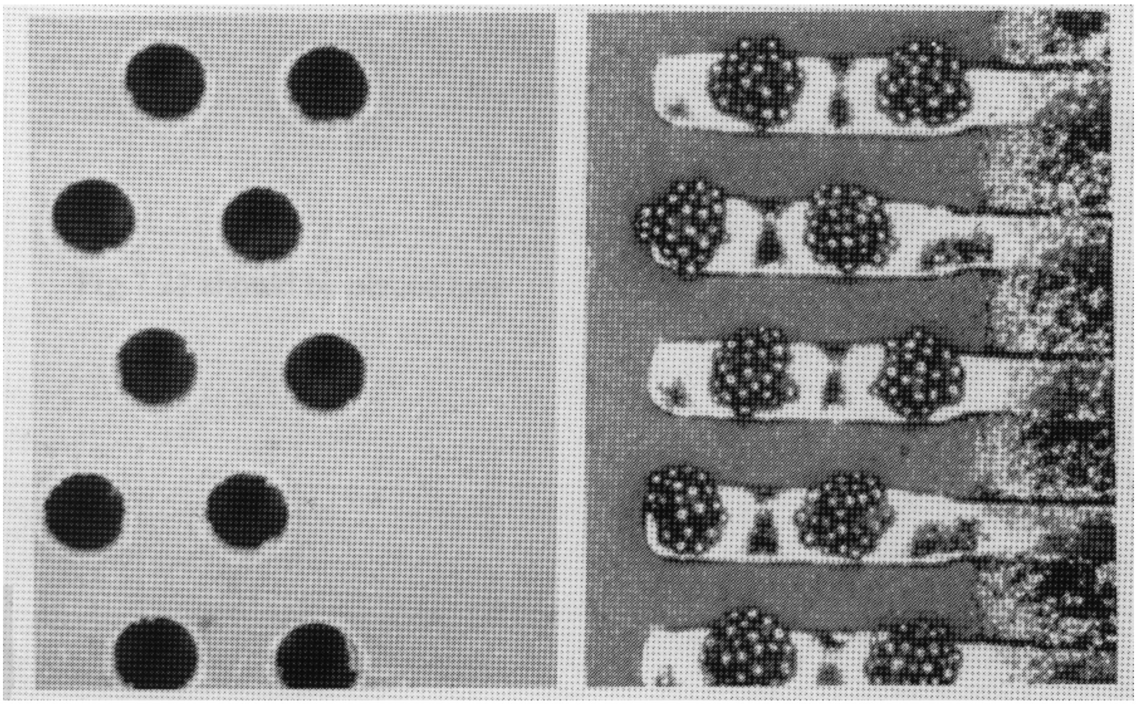

Fig. 7.22: Printing through 0.3 mm diameter holes with Mylar stencil. To obtain the correct amount of solder paste two or three small holes may be used for each solder land.

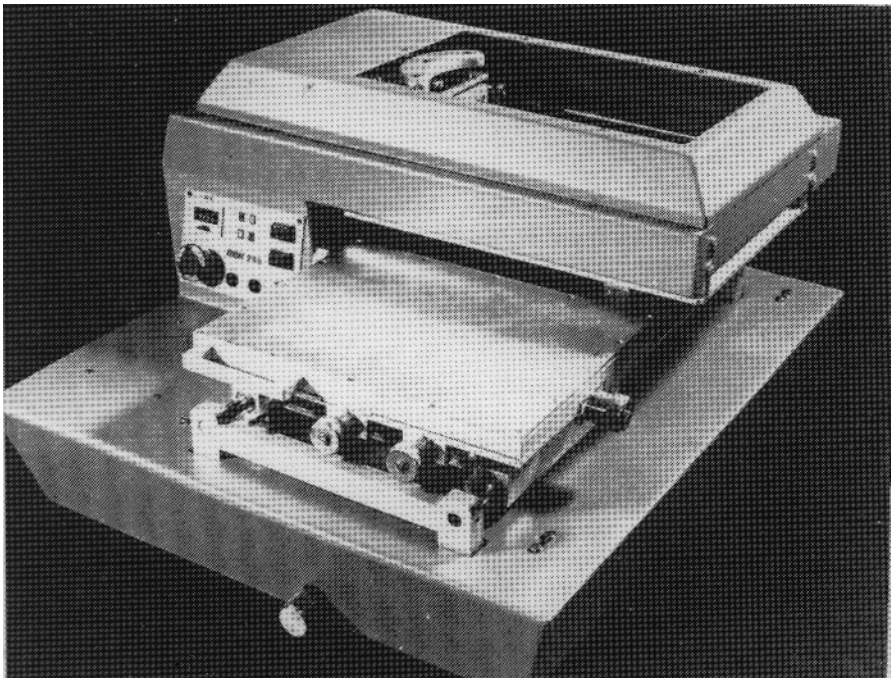

Fig 7.23 a): Screen printer.

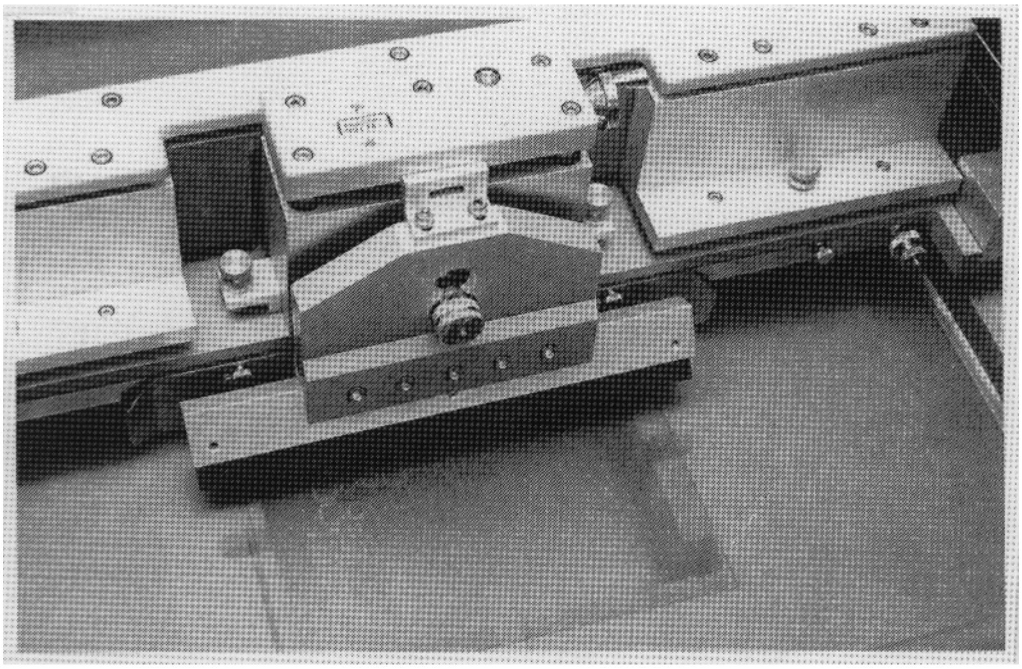

Fig. 7.23 b): The squeegee (DEK).

The IR soldering process

The most common process for reflow soldering is heating by infrared radiation (IR). It takes place in an in-line furnace, see Figure 7.24. In the furnace there are 8 - 14 heating panels, generating the radiation.

The radiation follows Planck's law [7.6], with maximum radiation intensity at a wavelength λ proportional to 1/T (Wien´s law):

$$\lambda max = k/T.$$

The amount of radiation absorbed by components and substrates depend on the emission coefficient of the materials. Originally quartz lamps were used in IR furnaces, with temperatures around 1100 °C. At this temperature electronic materials have very different emission coefficients, and were not evenly heated. The plastics in boards and component packages were often overheated. Today's furnaces use large area, low temperature, long wavelength heater panels, see Figure 7.24. This radiation also heats up the air in the furnace, and the board heating is provided by a mixture of air convection (30 - 60 %) and IR radiation. This gives a much more uniform heating. Details are discussed in [7.6].

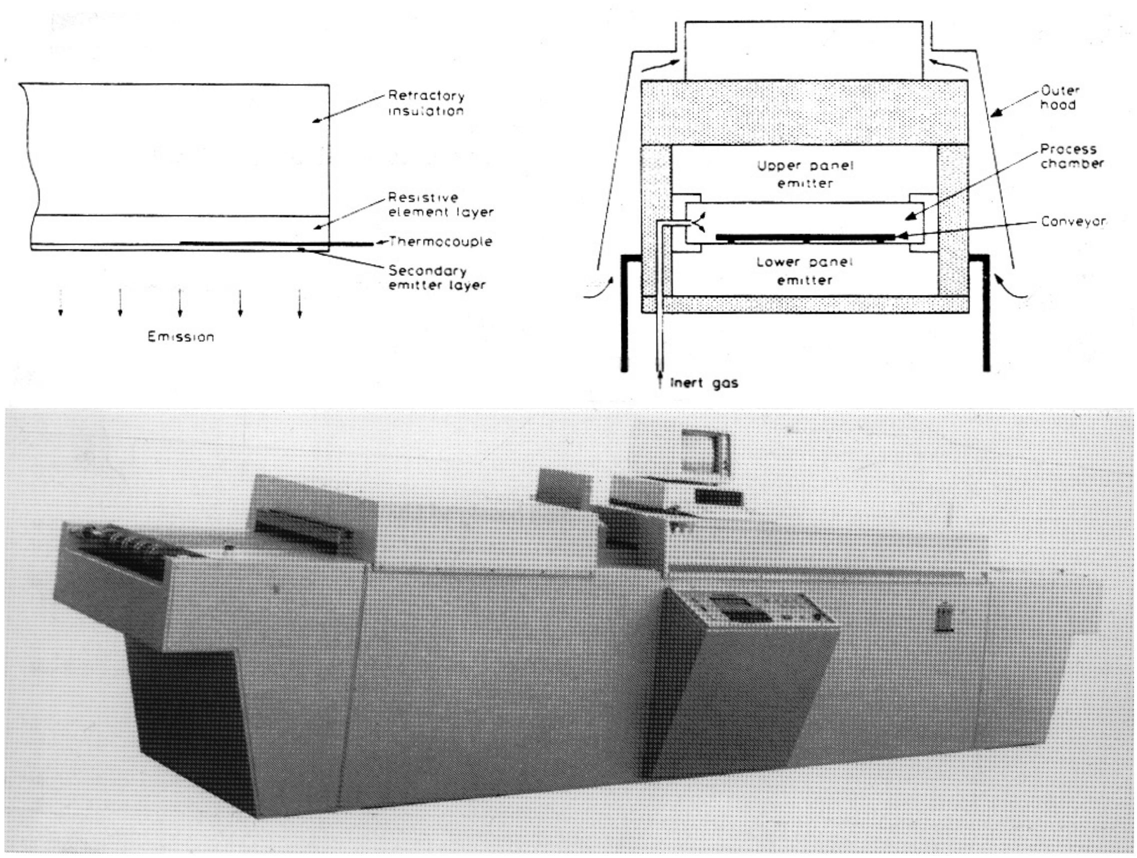

Fig. 7.24: IR furnace. a): Schematically with low temperature "area emitter". b): Industrial furnace.

Having typically 4 - 7 heater panel above as well as below the transport tunnel, the temperature profile can be adjusted to each type of board. A typical profile is shown in Figure 7.25. The first two elements give heat for the activation of the flux, and for a gradual temperature rise to avoid thermal shock. In the end, the temperature is above the solder melting point only for a short time, to reflow the solder on the board and the component terminals. The temperature profile must be re-adjusted for boards of different size and thermal mass. The temperature profiles for previously manufactured types of boards can be stored in the computer controlling the furnace.

Vapour phase soldering

Two types of vapour phase furnaces exist: in-line and batch machines. The principle of the in-line furnace is shown in Figure 7.26. The central part is a heating vessel where a fluorocarbon-based liquid is heated to boiling. Different liquids are used, with different boiling points, but the most common boiling point is 215 °C.

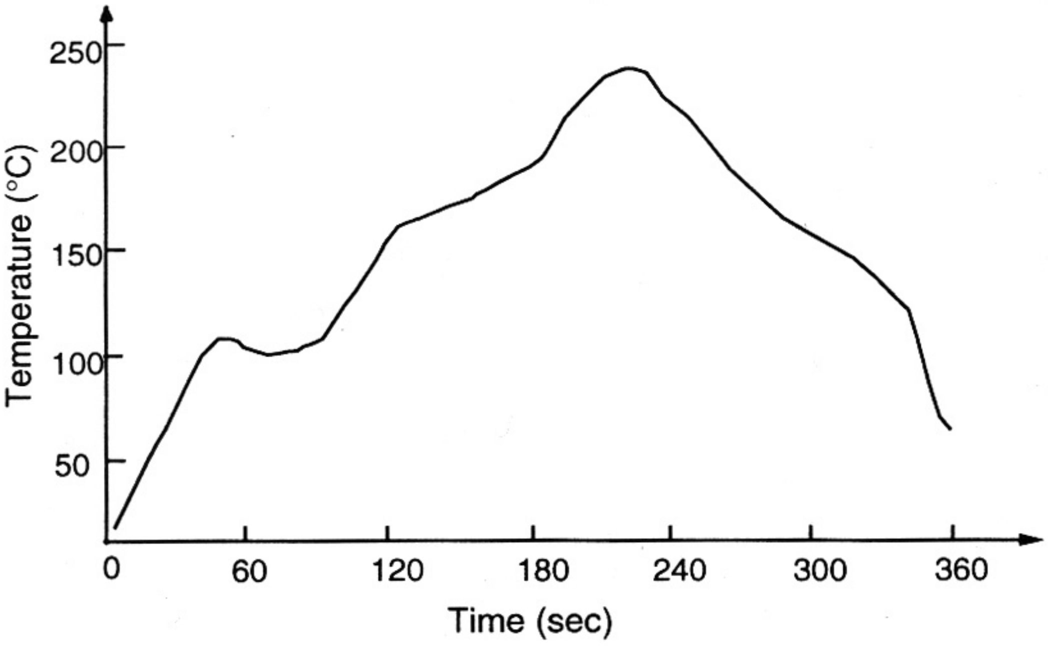

Fig. 7.25: Typical temperature profile for IR furnace.

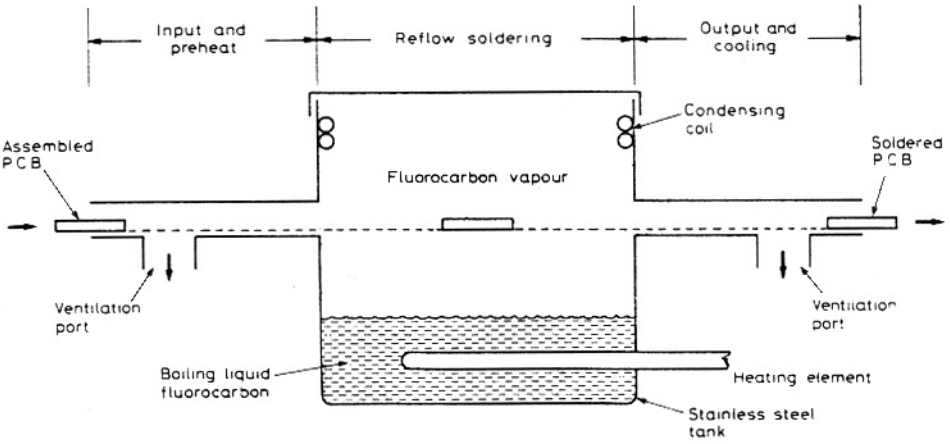

Fig. 7.26 a): Principle of in-line vapour phase soldering machine.

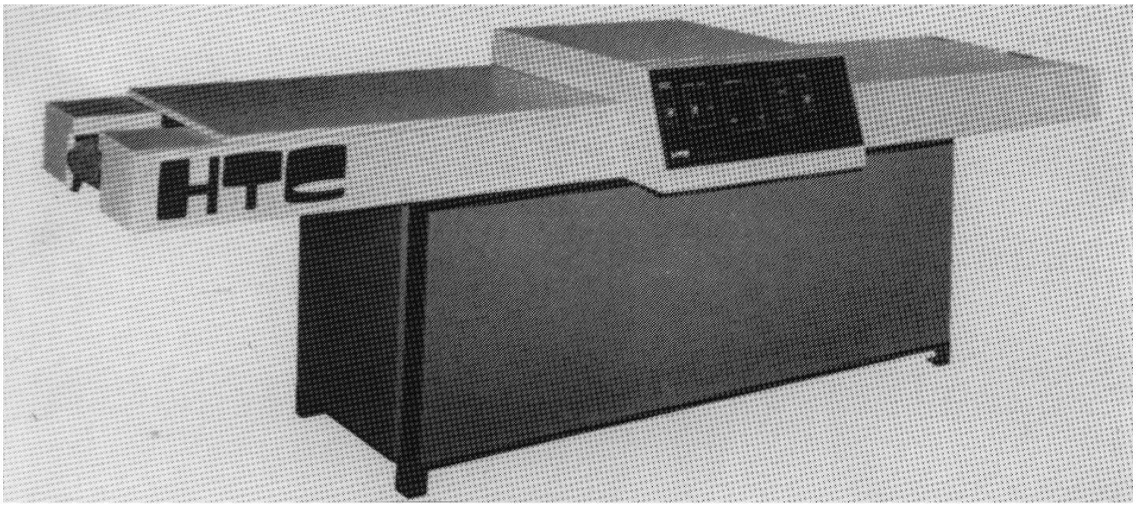

Fig. 7.26 b): Industrial in-line vapour phase soldering machine.

The boards pass through the vapour above the boiling liquid. Vapour will condense on the cold boards and transfer its heat of condensation, heating the board. After a short time, the board reaches the boiling temperature of the liquid, melting the solder. This heat transfer mechanism is extremely efficient, as discussed in Section 6.6. The heat transfer coefficient for condensing fluorocarbons is more than two orders of magnitude higher than that of air with natural convection, see Figure 7.27. A typical temperature profile, as seen by a component on the board, is shown in Figure 7.28.

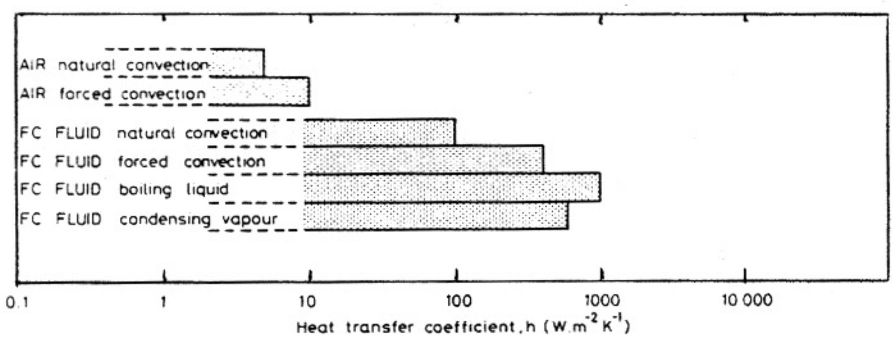

Fig. 7.27: Heat transfer coefficient for air and fluorocarbons. Boiling fluorocarbons, at the bottom, give 200 - 400 times more efficient heat transfer than air.

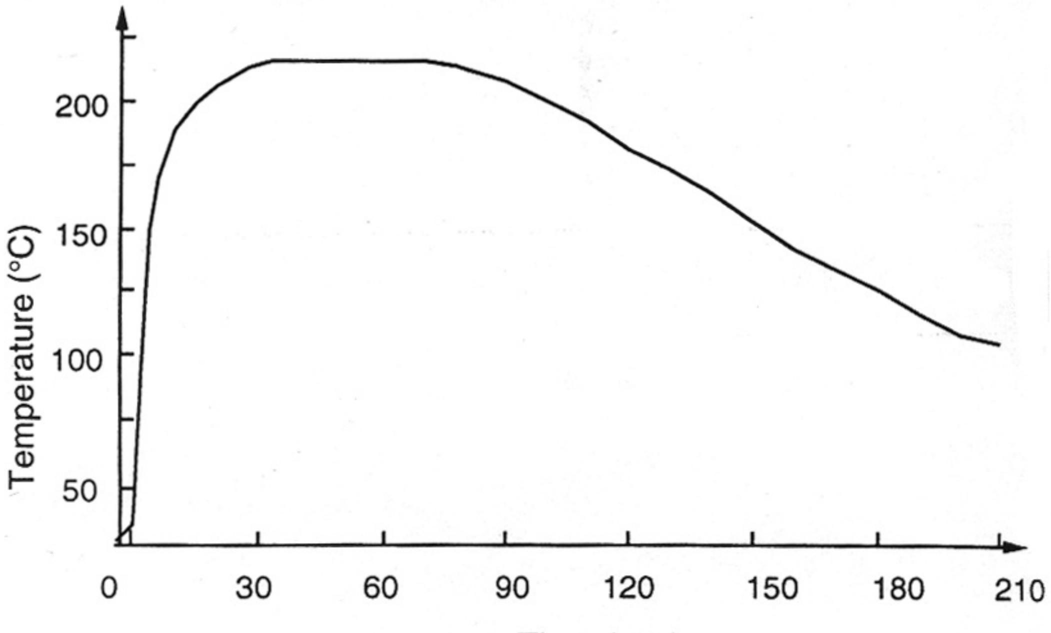

Fig. 7.28: Temperature profile through in-line vapour phase soldering machine.

The liquids used for vapour phase soldering are developed from common organic compositions by replacing hydrogen atoms by fluorine, as illustrated in Figure 7.29. They are very inert, thermally stable when they boil, and very expensive. An early liquid, FC - 70, Table 7.1, had the potential hazard that it could decompose into poisonous gasses if overheated to high temperature. Improved liquids without this problem have now replaced it.

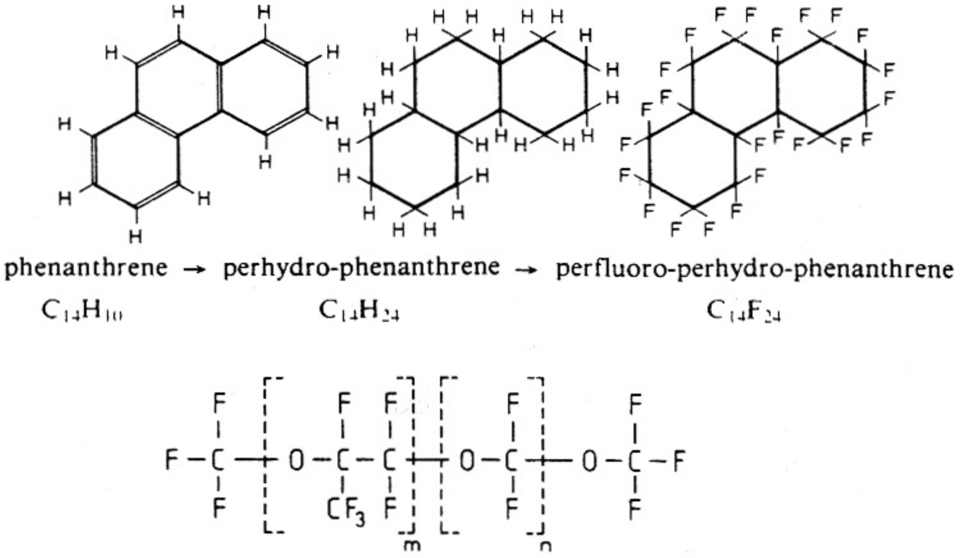

Fig. 7.29: Chemical composition of fluoro carbons for vapour phase soldering. Top: The liquid FC-5311 (3M): C14 F24 is derived from C14 H10. Bottom: The liquid LS 230 (Galden).

Table 7.1: Physical properties of some primary vapours for reflow soldering.

Property	*Units*	*R113*	*FC-70*	*FC-5311*	*LS230*
Boiling point or range	°C	47,6	215	215	230±5
Molecular weight	-	187	821	624	~650
Pour point	°C		-25	-20	-80
Density of liquid at 25°C	g cm^3	1,57	1,93	2,03	1,82
Density of saturated vapour at BP	mg cm^3	7,38	20,3	15,6	19,5
Viscosity of liquid at 25°C	cP	0,7	27	16	8
Surface tension of liquid at 25°C	mN/m	19	18	19	18
Specific heat of liquid at 25°C	J/gK	0,95	1,05	1,07	1,00
Thermal conductivity at 25°C	mW/mK	74	70	53	70
Electrical resistivity	Ohm cm		2 10^{15}	>10^{15}	10^{15}
Heat of vaporisation, at BP	J/g		67	68	63

To avoid loss of the expensive vapour outside the furnace there are cooling/condensation coils at the entrance and exit of the furnace. Still there is a certain loss, making this a more costly soldering method than IR.

The activation of the flux must be done in a separate furnace. In some large in-line machines, there is an IR preheating unit integrated with the vapour phase heating.

Another type of vapour phase furnace is the batch type, see Figure 7.30. A tray of boards is lowered into the vapour and kept there until soldering is completed. Above this "primary vapour" there is a blanket of a lighter (and cheaper) "secondary vapour" from a liquid of lower boiling point. 112-trichloro-trifluoro-ethane is used for this. It is heated by the primary vapour and boils at 48 °C. The batch machine is suitable for smaller production volumes and is less costly to use than the in-line machines.

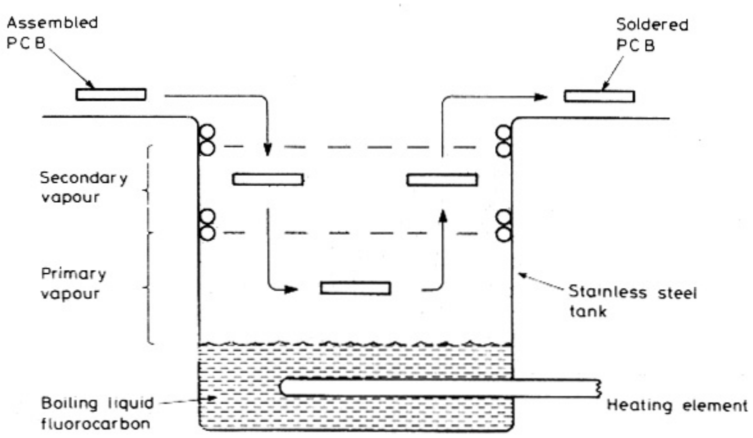

Fig. 7.30: Batch type vapour phase soldering machine.

One of the advantages of vapour phase soldering is the inert atmosphere in which the soldering takes place, giving less oxidation of the solder and the copper foil of the PCB. There is less heat damage to the polymer in the PCB, and all boards and components are sure to reach the same temperature, without special adjustment. A good discussion is given in [7.6].

<u>Other soldering processes</u>
Heating only in a stream of hot air is used, particularly in repair equipment, see Section 7.3.4 below.

Thermode soldering, also called impulse soldering or hot bar soldering, is used particularly for fine pitch flat packs, TAB circuits, etc. (Please refer to Section 3.14).

By this method the components are soldered one by one, in the same operation as they are placed on the PCB, on which solder is already deposited by printing, plating or hot air levelling. The mounting tool, see Figure 7.31, has a suction cup to hold the component, and a metal frame touching the outer parts of the leads (see also Figure 3.29). The frame is resistance heated by a high current through it, heating the leads to melt the solder while the component is pressed down. A typical temperature profile is shown in Figure 7.32.

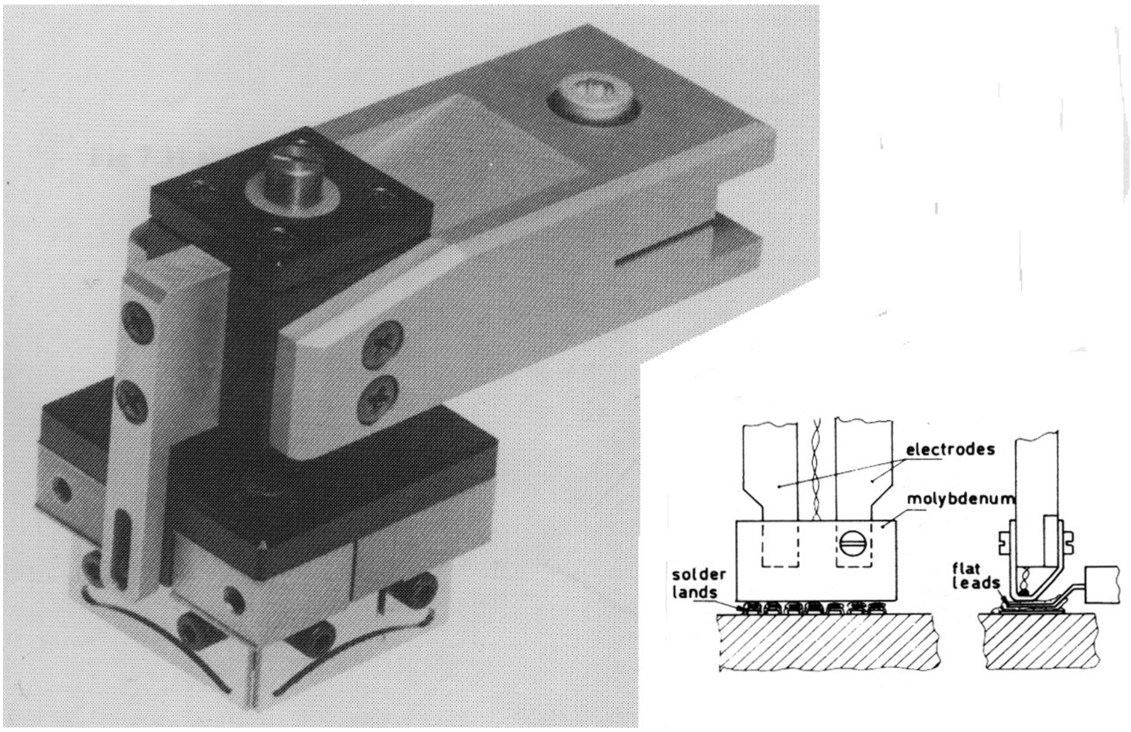

Fig. 7.31: Two types of thermodes for thermode soldering.

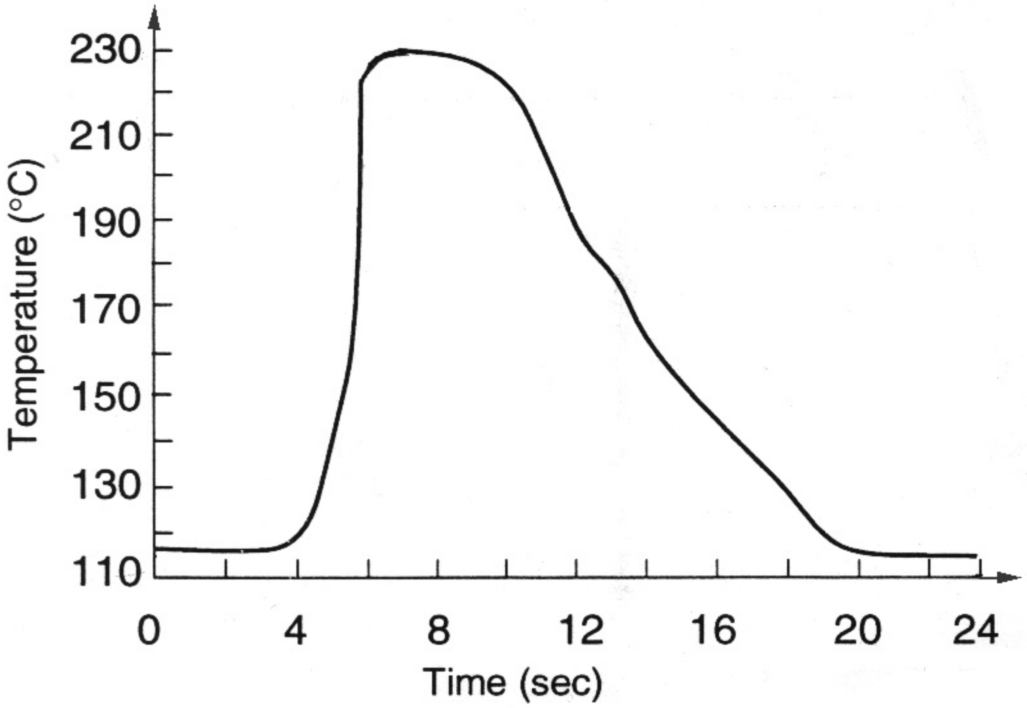

Fig. 7.32: Temperature profile for thermode soldering.

Some of the advantages of thermode soldering are:

- The component is held firmly during placement / soldering and cannot move out of position

- The leads are pressed down, and deviation from coplanarity is not critical
- Only leads and solder lands are heated, giving little thermal stress to the component body.

Disadvantages:

- The method is time consuming, as the components are soldered one by one
- One thermode is required for each component size. The thermodes are expensive and thermode change in the placement machine is time consuming.

Laser soldering uses a powerful, focused (CO_2 -) laser that can heat each solder joint by a system of movable mirrors. The mirrors are controlled by a computer, with CAD information on component location, laser energy needed to melt the solder in each joint, etc. The method is slow (maximum some 10 joints per second), the equipment is expensive, and laser soldering is not yet much used.

7.3.3 Component mounting

SMDs are usually mounted automatically, except for prototyping. Pick-and-place machines are normally used for the mounting.

The Siemens machine in Figure 7.33 illustrates the principles of a pick-and-place machine. The PWBs are fed from the magazine at the left. The boards move through the machine on a rail. They stop at two locations, underneath placement heads nos. 1 and 2. These heads are mounted on a common arm that moves synchronously in the x- and y direction. On head no. 1, there may be an adhesive dispenser (for wave solder process) or a component picking tool. On head no. 2, there will always be a component picking tool. Head no. 2 may pick and mount components, while head no 1 may put adhesive dots on the next board.

The picking tool is most often a small vacuum pipette, with computer control of the vacuum. Alternatively, there may be 2 or 4 small gripper arms, "claws", see Figure 7.34 a). Such tools can be used for coarse electrical testing of two-terminal components while they are mounted. Because of the big difference in component sizes, some machines have several picking tools that can interchange automatically during operation.

Different types of component feeders are placed in the component storage area. They may be modules for component tapes, component sticks, vibration feeders, see Figure 7.34 c), etc.

Figure 7.34 b) shows details of the component tape during mounting. The protecting cover over the components is removed over the one component in place for mounting. When the component is removed the tape moves to bring the next component into place.

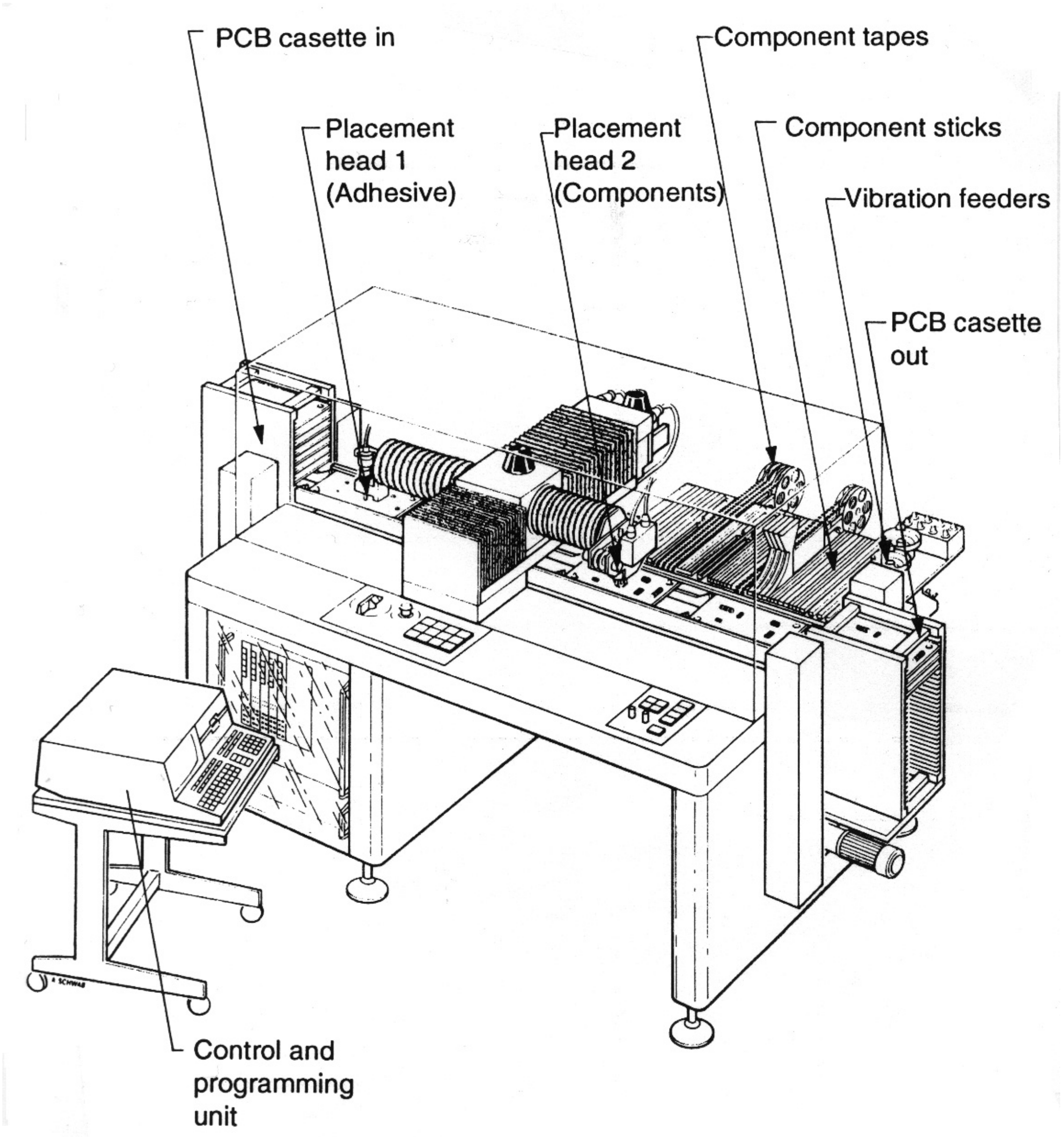

Fig. 7.33: SMD pick-and-place machine (Siemens).
The mounting head may also include an electronic vision system for very accurate placement of fine pitch components.

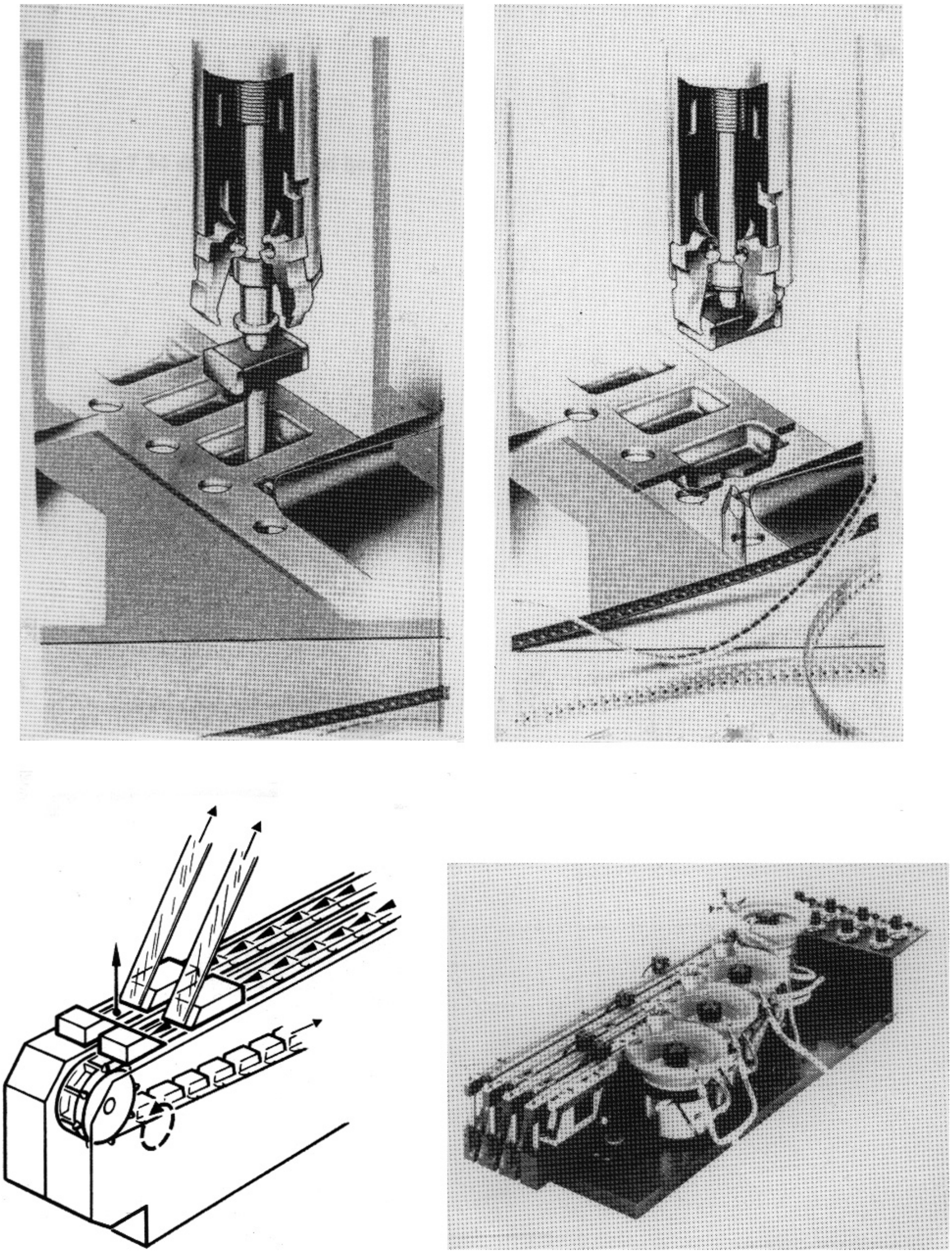

Fig. 7.34: a): Mechanical gripper in pick-and-place machine. b): Detail of the component tape when a component is in position for picking. c): Vibration feeder.

The machine is computer controlled. The information to the control system may be generated in the CAD system, with appropriate post processing. Alternatively, it may be off-line programmed. Some very high volume machines have many grippers, firmly mounted in the right positions for components to a given type of board. Then all components are mounted simultaneously. These machines are said to be hardware controlled, rather than the more common programmable, software controlled machines.

Other examples of pick-and-place machines are shown in figs. 7.35 - 7.36. Typical placement capacity for software controlled machines is 2000 - 15000 components per hour. Placement accuracy: 0.05 - 0.2 mm.

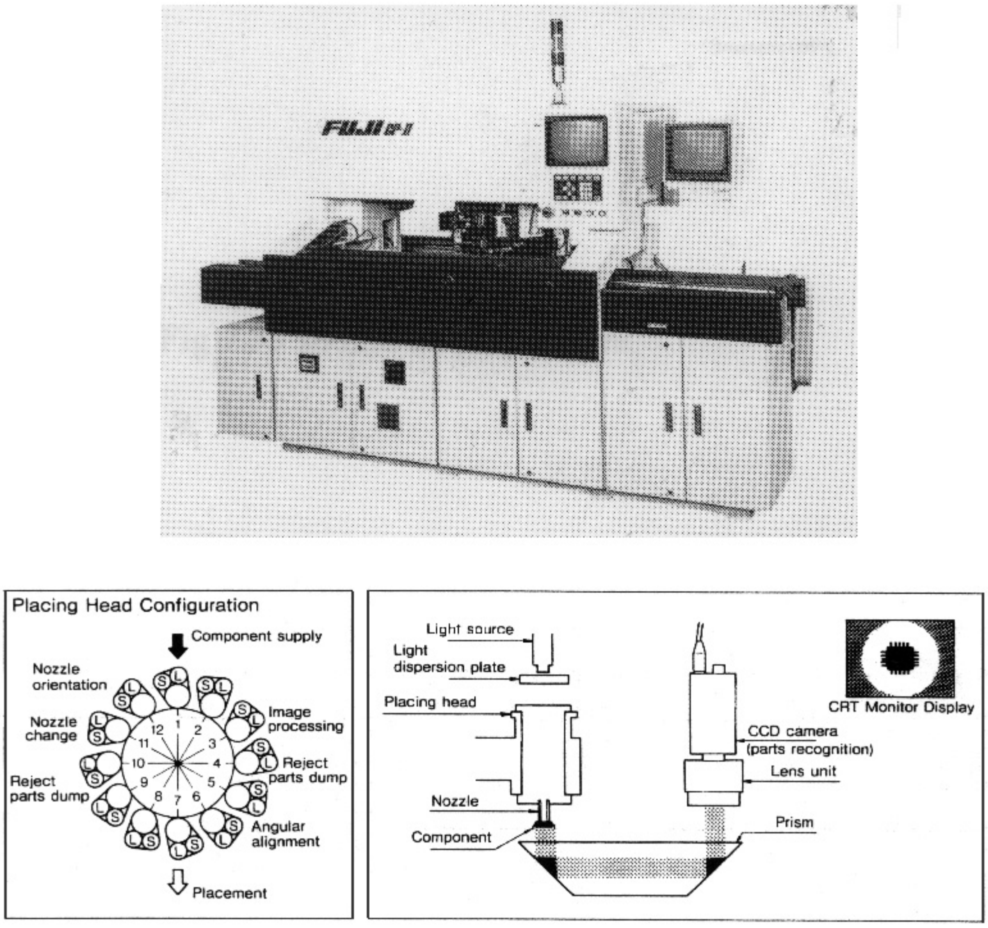

Fig. 7.35: Fuji CP-II pick-and-place machine. The machine has magazine for over 100 types of small components, nominal speed up to 15 000 components per hour, placement accuracy 0.10 mm. It has a rotating head with 12 positions, bottom figure, and two alternative tools at each position. There are components at all 12 positions at any time, with a separate operation being performed. A CCD camera shows the accurate position and orientation on a CRT screen (Fuji).

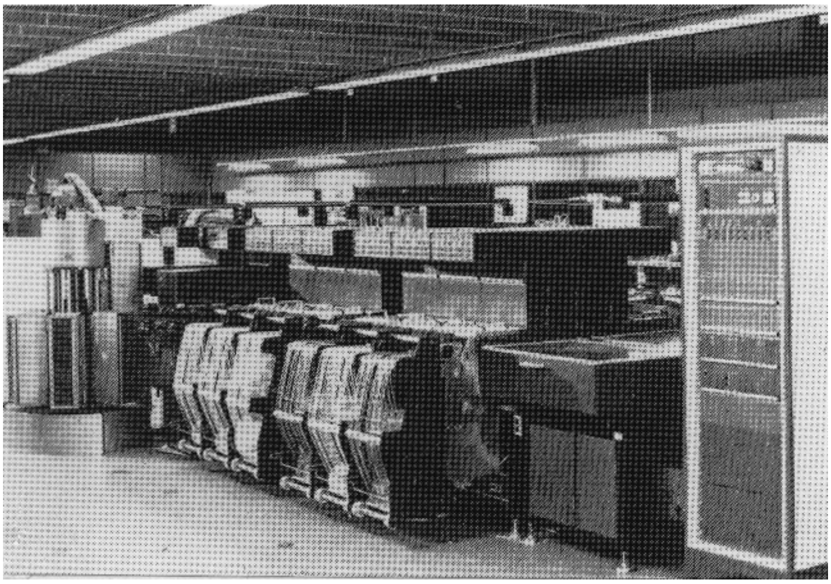

Fig. 7.36: Philips large hardware controlled pick-and-place machine.

7.3.4 Repair

Removal of soldered SMD components is done by heating to melt the solder by specially heated tweezers or "tongs", see Figure 7. 37 a), for small components. For ICs, a hot-air equipment is generally used. An early version is shown in Figure 7. 37 b). It has one air blower from underneath and one from above. The air temperature can be separately controlled for the two. Nozzles are mounted at the end of the air outlet, to limit the heated area. When the solder melts, the component is manually removed by tweezers.

More modern equipment has air nozzles that only blow the hot air at the terminals and solder lands, limiting the heated area and reducing the thermal stress on neighbouring components.

Remaining adhesive and solder must be removed and additional solder added if need be, before flux is added and a new component soldered in by a small solder iron/heating torch or with the same repair tool.

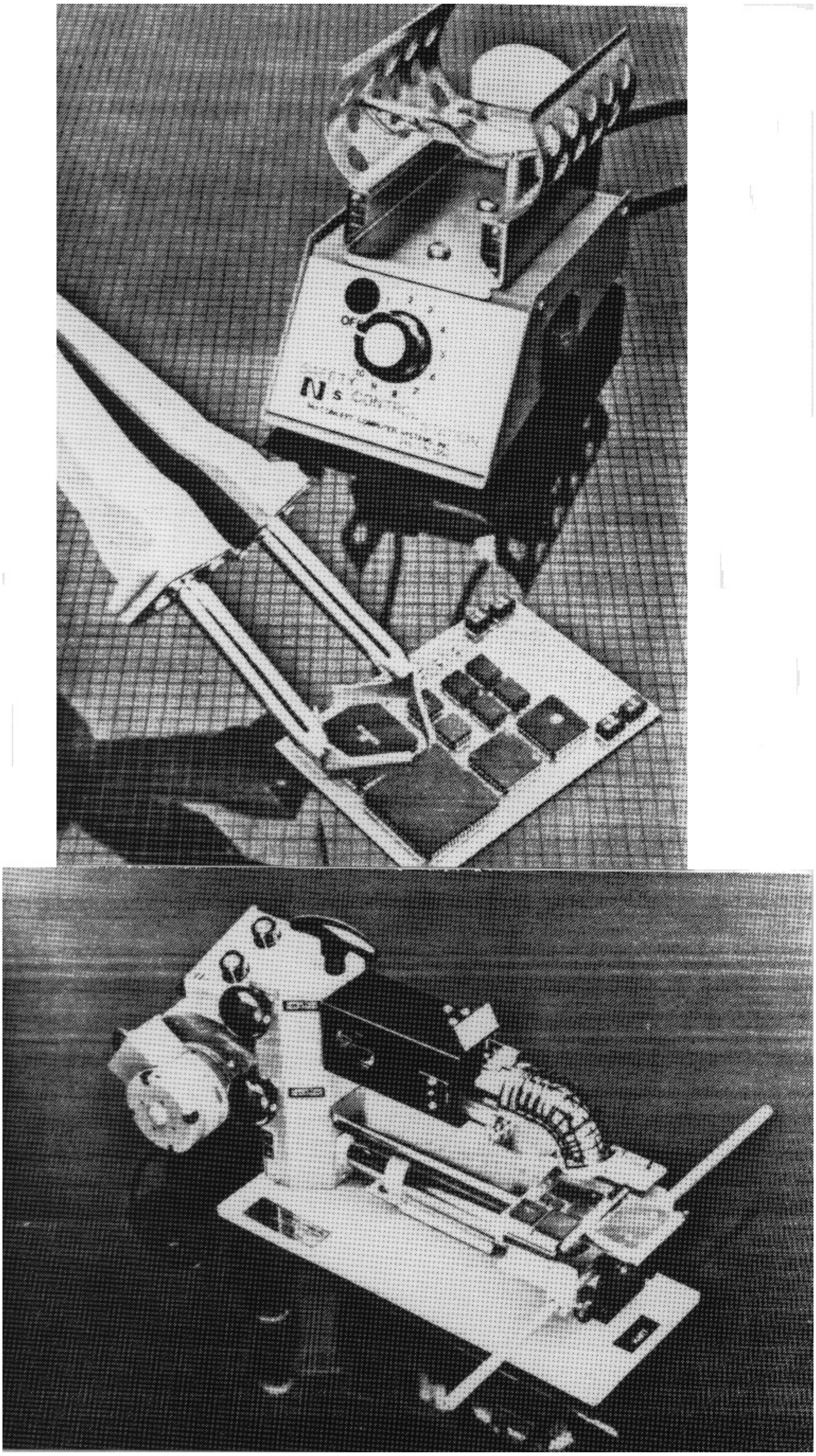

Fig. 7.37: Repair equipment for SMDs. a): Heated tweezers for de-soldering of PLCC packages. b): Hot air blower (HART).

7.3.5 Solder faults

The most common faults in wave soldering, particularly for surface mounting, are the following:

- Solder bridges, shorting terminals or neighbouring components.
  Possible reasons:
  * Incorrectly adjusted solder machine (or unsuitable machine)
  * Bad design; incorrect dimensions on solder pads, components too close, component oriented wrong way
  * Component not suited for wave soldering.

- Dry solder joints.
  Possible reasons:
  * Poor wetting or poor solderability on PCB or component
  * Shadowing
  * Contaminated solder
  * Adhesive on solder land
  * Too weak flux
  * Too low pre-heat temperature.

- Components missing.
  Possible reasons:
  * Too little adhesive
  * Adhesive too dry when component mounted
  * Adhesive too low viscosity
  * Mounting error.

The most common faults in reflow soldering are the following:

- Short circuit between components/terminals.
  Possible reason:
  * Solder balls. This means that the whole amount of solder paste on one solder land has not melted into one sole solder fillet, but some individual solder particles have spread out on the board in an uncontrolled manner, possibly causing shorts at once <u>or</u> at a later time.
  * Too much solder paste
  * Incorrect placement of board during printing.

- Visible solder balls. This is always a reason for board rejection, because of the possibility that the balls will move and cause a short circuit later.
  Possible reasons for solder balls:
  * Oxidised solder paste
  * Too rapid heating
  * Insufficient drying of solder paste before reflow soldering.

- Dry solder joints.
  Possible reasons:
  * Poor solderability
  * Not being coplanar of terminals of IC.

- "Manhattan-" or tombstone-" effect: This means that some small passive components have risen on edge during soldering, see Figure 7.38. It may take place because the solder on one terminal melts before the solder on the other terminal. The molten solder pulls the component up because this minimises the surface forces in the solder.

  Possible reasons:
  - * Poor solderability on one of the solder lands
  - * Poor layout, giving more heat flow to one solder land, and the temperature reaches the melting point of the solder first on this component end (please refer to Section 6.33).

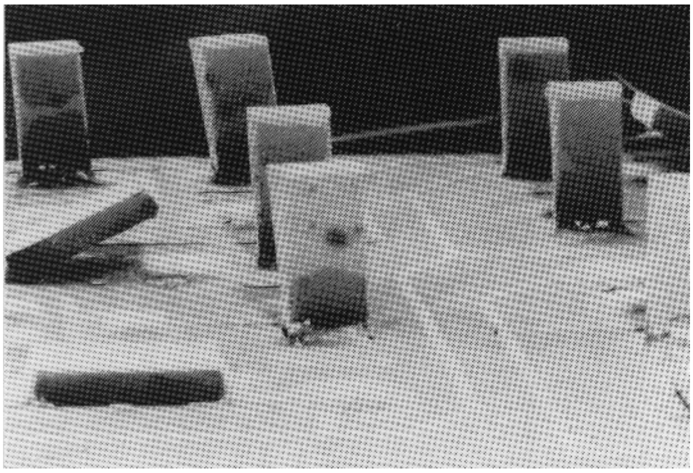

Fig. 7.38: Small SMDs standing on edge due to the "Manhattan-" or "tombstone-" effect.

7.4 ROBOT MOUNTING

Automatic component placement has been common in the electronics industry for some 15 years, based on dedicated machines, made for one type of components or component packages, as described above. Now the use of flexible, programmable robots is rapidly increasing, see Figure 7.39.

An industry robot may be defined as "a re-programmable multi-functional handling machine which, without continuous supervision, can place tools, components or materials in one or more pre-determined patterns of motion, in order to perform different tasks. It can have some amount of "intelligence" which permits it to detect and respond to its environment."

A robot may handle many types of components, and it can fetch the components from many types of feeders or magazines. It can automatically change between different types of tools, suitable for the different component types. These changes are programmed.

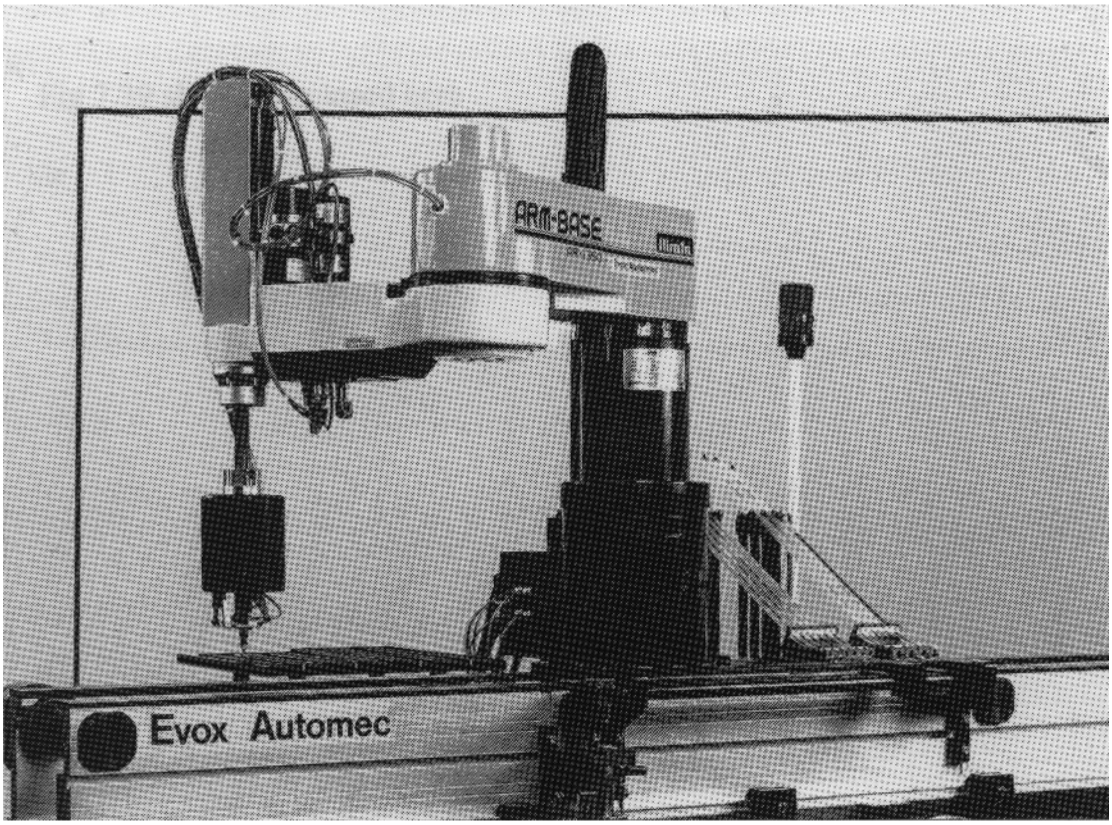

Fig. 7.39: Example of a programmable placement robot for electronics: the SCARA robot.

There are several advantages in using robots for mounting, compared to other methods:

- Flexibility, as mentioned. Once the robot has been installed and programmed, one can change between many types of boards and achieve the same productivity regardless of the number of boards of each type - down to one.
- Uniform product quality: Once the robot has been set up for a task, it will repeat it much more reliably than any human.
- Accuracy: Tolerances in placement position can be 0.02 mm or better (at reduced placement speed).
- Unattended operation is possible over long periods of time (e.g. over night).
- The robot can work in hostile and dangerous environments.
- Robots equipped with sensors can perform useful controls, and tests can be integrated into the production process.

Even if the robot gives the possibility of a high degree of flexibility, much planning is needed before it can be used for a new type of task. A suitable tool is needed for each type of component - even those with odd shapes. The feeder arrangements must be carefully planned, possibly be custom made. There are often bigger costs connected to the surrounding equipment than to the robot itself.

Although a robot will generally work much faster than humans doing manual mounting, it is much slower than a dedicated pick-and-place machine doing the same operation.

7.4.1 The components of the robot system

Normally the robot system consists of the following parts, see Figure 7.40:

- The manipulator:
  This is the mechanical part that physically performs the task.
- The "learning box":
  From it, the commands and positions are programmed.
- The control cabinet:
  The cabinet contains computer power supply and computer to control I/Os and emergency functions. It sends all command signals to the manipulator and receives signals from the learning box.

The manipulator, or robot arm, consists of joints that have linear or rotary motion. To enable the robot to place an object at an arbitrary position the arm needs at least three joints - linear or rotary. If the object also needs an arbitrary angular position, six joints are needed, of which three must be rotary.

The four main types of robot arms are the following (see Figure 7.41):

- Cartesian arm with three linear joints.
- Cylinder co-ordinates arm with two linear and one rotating joint.
- Spherical co-ordinate arm, with one linear and two rotating joints.
- "Human-like" arm with three rotating joints.

A special type of cylinder co-ordinates arm is the "SCARA" (Selective Compliance Arm Robot Assembly) with two rotating joints in the same plane, and one linear joint in the plane perpendicular to that plane, see Figure 7.39. They are much used in assembly and mounting operations, and they are fast and accurate.

Robots are driven pneumatically (by air), hydraulically (by liquid) and/or by electric motors.

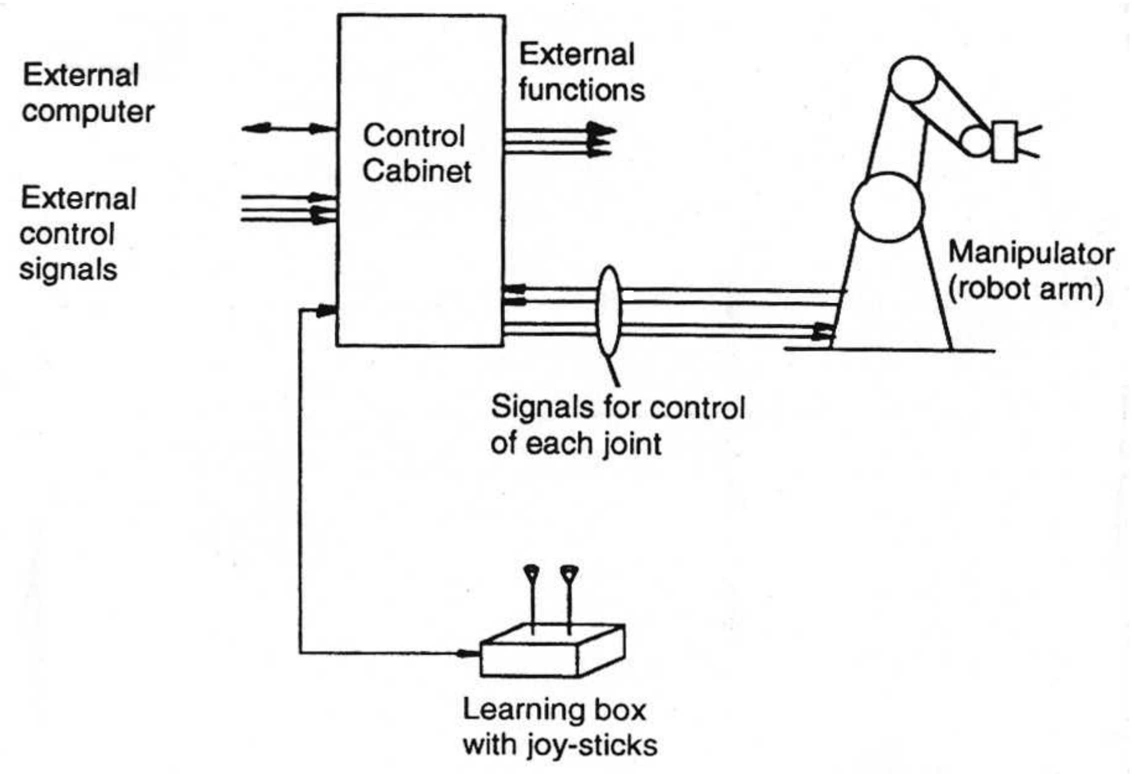

Fig. 7.40: The main components of a robot system.

7.4.2 Programming

The programming takes place in several ways:

- "Guide - and - learn":
  The robot is guided through the desired path by guiding the tool on the robot arm. The control system automatically reads the positions along the path and remembers the co-ordinates.
- "Jog - and - learn":
  The robot arm is moved along the desired path by commands from joysticks and control switches.
- "Synthetic programming":
  The co-ordinates for all the points in a program are written directly into the memory of the control system by off-line programming. When the robot is to mount components on a PCB, this can be done automatically from information generated in the CAD system that is used for lay-out of the PCB.

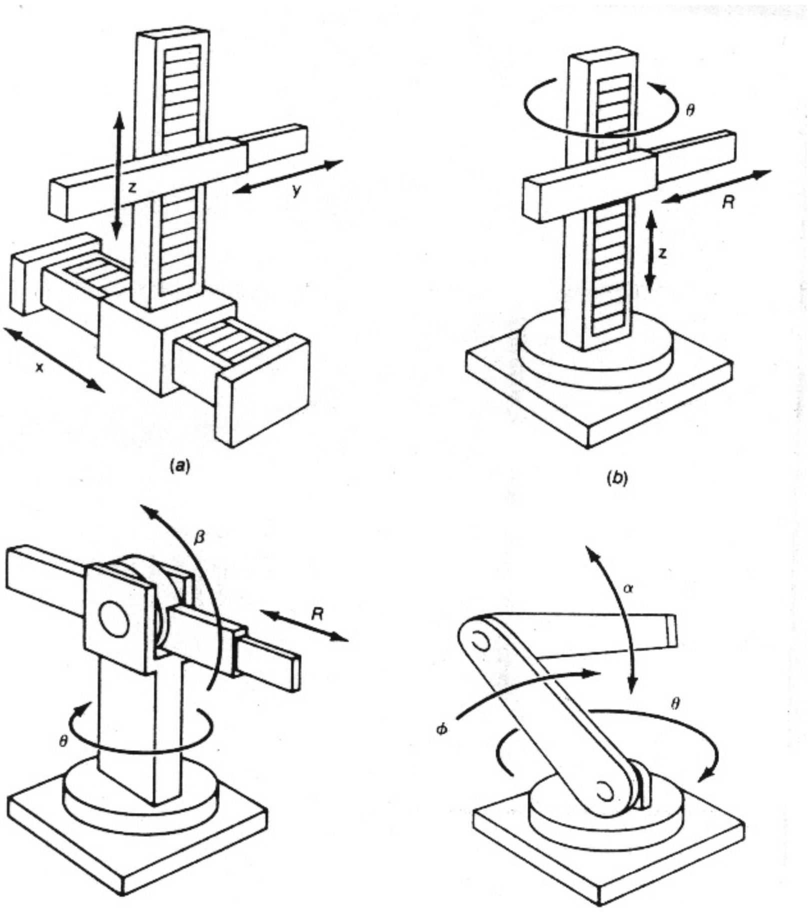

Fig. 7.41: Types of robot arms: a): Cartesian motion, b): Cylindrical, c): Spherical, d): "Human like". The SCARA robot is a special version of the cylindrical type.

7.4.3 Auxiliary equipment in the system

Auxiliary equipment is everything in the robot system that is not directly part of the manipulator and control system. It may include:

- Picking tools
- Feeders
- Sensors and actuators
- Special equipment.

The most common types of picking tools are mechanical and vacuum tools, similarly to those used in pick-and-place machines for SMD placement. The robot may also change tool automatically, or have a multi-tool head, see Figure 7.42.

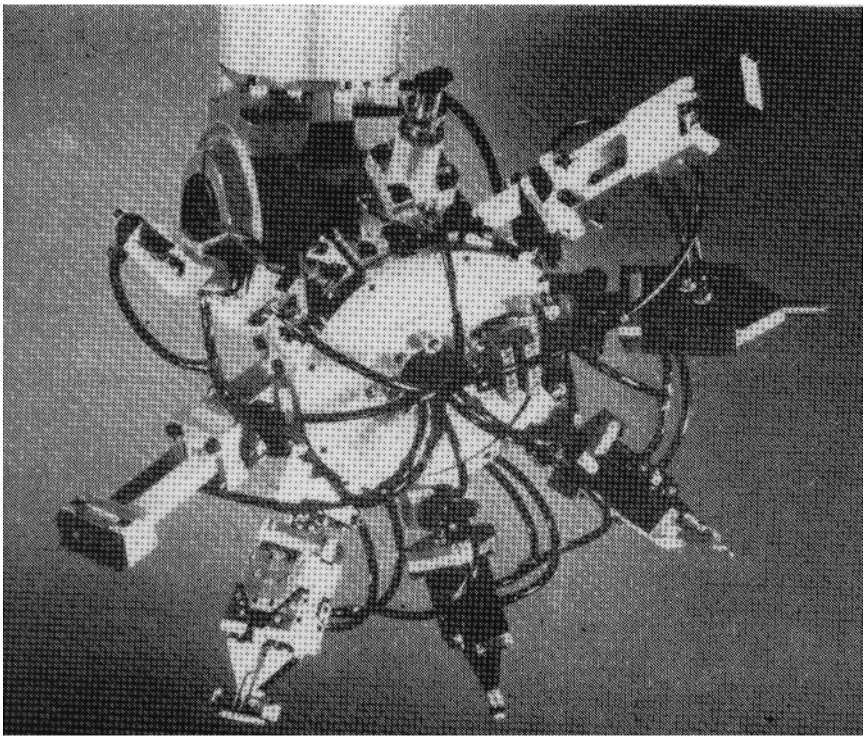

Fig. 7.42: Multi gripper head.

A simple robot cell for component placement is shown in Figure 7.43.

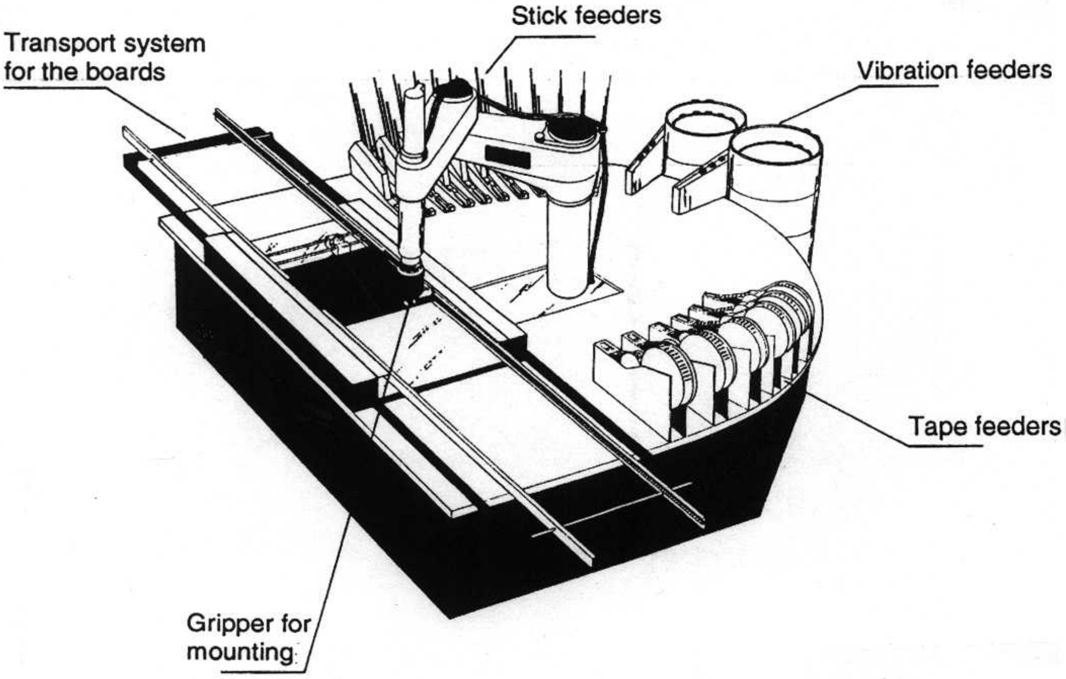

Fig. 7.43: Robot cell for electronic component placement (Adept).

Special equipment is tailored to each task. For an example, solder equipment can be mounted on the robot arm, as well as a solder paste dispenser or an adhesive dispenser. The robot can place the component in a special cut-and-clinch tool for component terminals and pick up the component after the operation has been performed, etc.

If the robot is equipped with sensors and actuators, it can perform additional useful functions such as:

- Provide safety for the user
- Control the manipulator (position- or force sensors, etc.)
- Avoid collision
- Inspection (pattern recognition, "vision system")
- Detect and automatically react to fault situations,
- etc.

The use of sensors can improve the robot capability, but it may increase the cost of the system significantly.

Modern robot control systems can work on several tasks at the same time: control the manipulator, feeders and operator communication, react on sensor signals, etc.

7.4.4 Areas of application for robot mounting

Robots are useful, among other things, for the following tasks in electronics production:

- Component mounting
- Handling of boards/components during testing
- Automatic trimming during testing
- Screwing and gluing operations
- Soldering, etc.
- Production of special components (e.g. winding coils)
- Final assembly of board/rack/chassis.

Products should be designed for robot mounting, i.e. already in the development phase components, placement, fastening method, etc. should be chosen for an optimal use of the robot.

7.5 SEQUENCE IN THE PROCESS OF SMD- AND MIXED SMD/HOLE MOUNTED PCB's

For surface mounted boards and boards with mixed SMD/hole mounted components, several mounting and soldering processes are included. The sequence is not arbitrary, and it must be planned in the design phase. It must also be planned on what side each component is to be mounted, considering that soldering/heating processes it will be exposed to. As mentioned previously, not all components can withstand the wave soldering process, some components are not suitable for it (due to generation of solder bridges, etc.). Some components cannot stand the heat of any mass solder processes and may have to be hand mounted and hand soldered later in the process.

In Figure 6.5, the main types of boards were shown. Figure 6.5 a) showed SMDs on one side of the board only. Normally solder paste printing and reflow soldering will be used for such a board. Figure 6.5.b) showed SMDs on both sides. Two reflow processes may be used. Figure 6.5.c) showed hole components on one side, SMDs on the other. In this case, all components may be soldered simultaneously by wave soldering. In Figure 6.5.d), we had SMDs on both sides and hole components on one side. A combination of reflow soldering and wave soldering is used.

Figure 7.44 a) - d) show process sequences for various types of boards. When the board has SMDs on both sides, see Figure 7.43 b), the adhesive step may often be omitted. In an IR furnace, the heat below the board may be reduced such that the solder on the top surface melts while the solder on the bottom side remains solid.

7.6 TESTING OF PCB's

Testing takes place at various stages in the development of a product. Here we are concerned with volume testing during the production process, which normally consists of visual tests (which for very high volume production may be replaced by electronic inspection using vision systems) and electrical tests. The testing is done several times, at different level: component-, board-, module-, and system test.

7.6.1 Component testing

Previously an incoming test of components when they arrived was common. Today it is more common to purchase components on a "ship-to-stock" or "ship-to-line" basis. In this case, the component manufacturer does the testing, and test documentation can be supplied to the customer.

7.6.2 Mechanical parts for the PCB testing

The functional test and in-circuit test principles were described in Section 6.4. Here we shall only discuss the fixture and mechanical parts needed for in-circuit testing.

Most commonly, the board has wiring and components on both sides. If possible, the testing should be done on one side only. Two methods to achieve this are shown in fig .7.45. Fig .7.46 shows a "bed of nails" test fixture for the in-circuit testing. Figure 7.47 shows details of single sided and double sided test fixtures and Figure 7.48 shows test pins used.

Normally the board will be pressed against the fixture by over pressure. This must be planned during the design (position of high components, holes in the board, etc.). The double sided fixture is mechanically complicated and expensive. The test pins are pressure loaded and normally dimensioned for placement on a matrix of minimum 0.1" distances. Pins for 0.05" distances are available, but they are mechanically less robust.

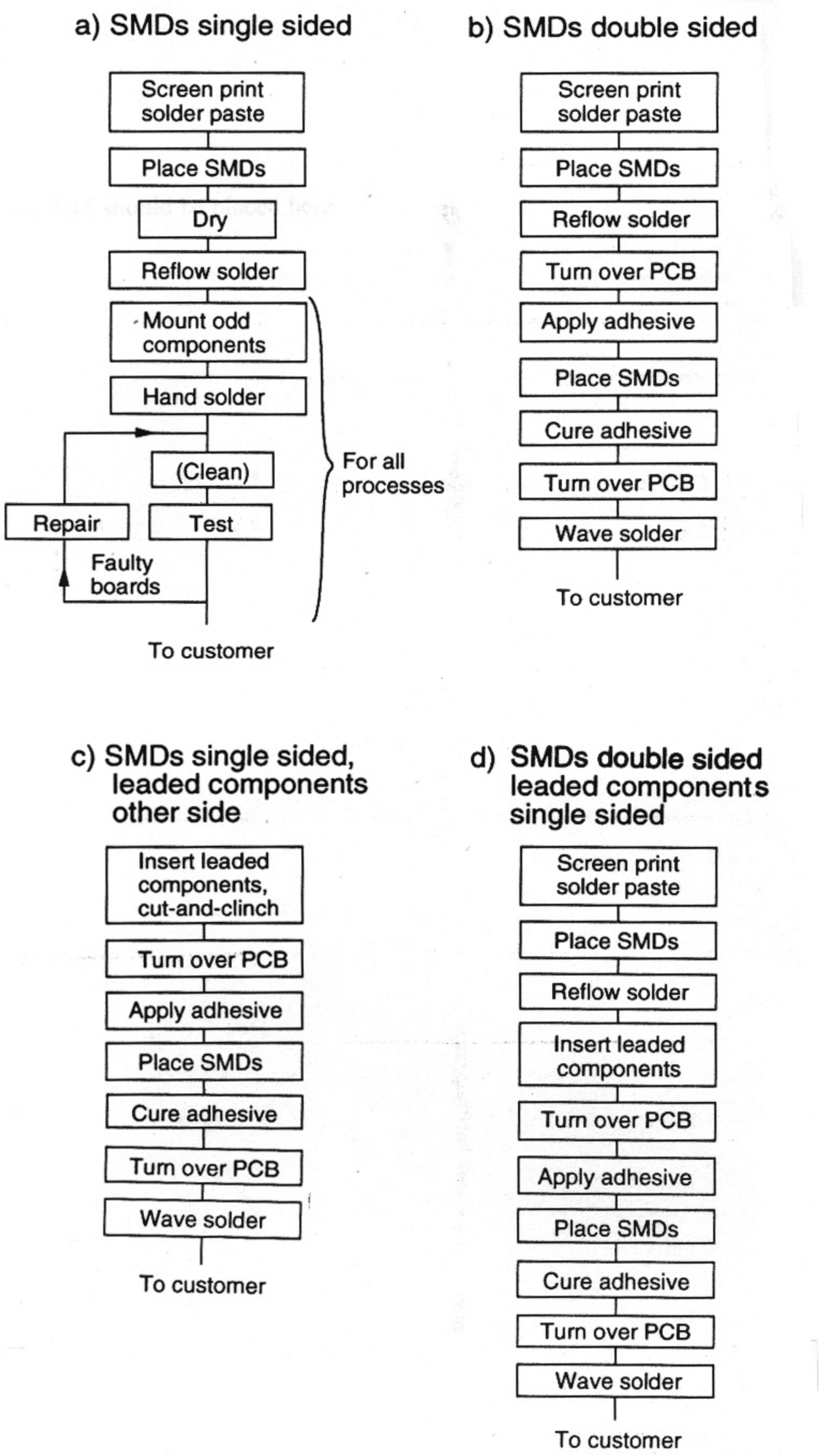

Fig. 7.44: Process sequences for boards with different types of components on the two sides. The steps marked "For all processes" on figure a) are not repeated on the other figures.

Placement of the test points is an important part of the design, please refer to Section 6.4. Figure 7.49 shows an example of a poorly designed test point, on a component lead. If the solder contact is bad the test probe will press the lead down to make contact, and the fault is not detected.

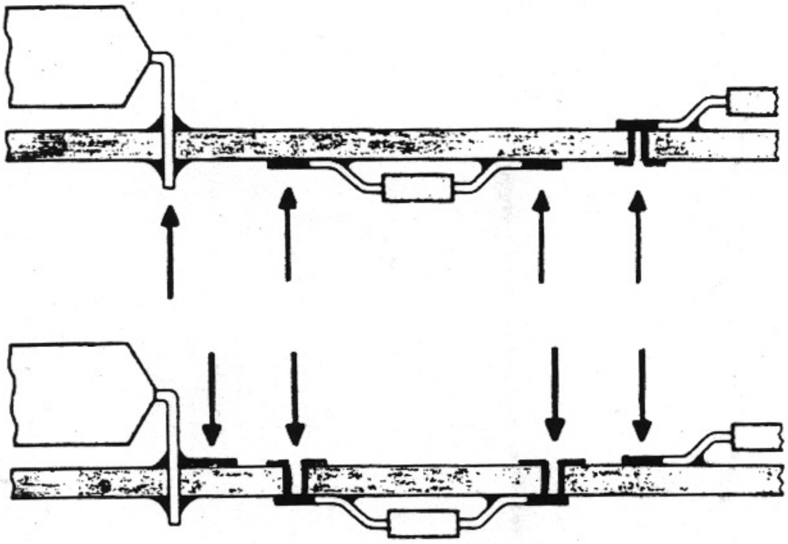

Fig. 7.45: Two methods for single sided test of a board with components on both sides.

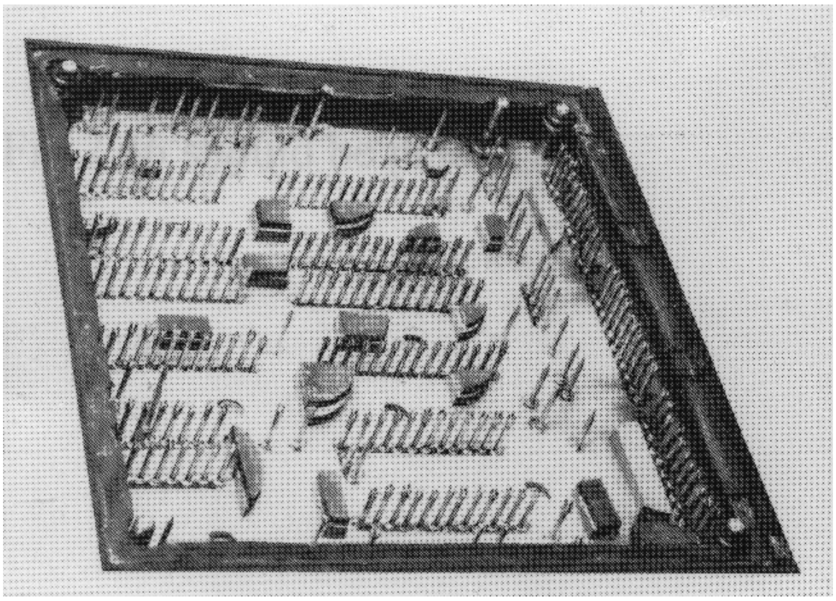

Fig. 7.46: Bed-of-nails test fixture.

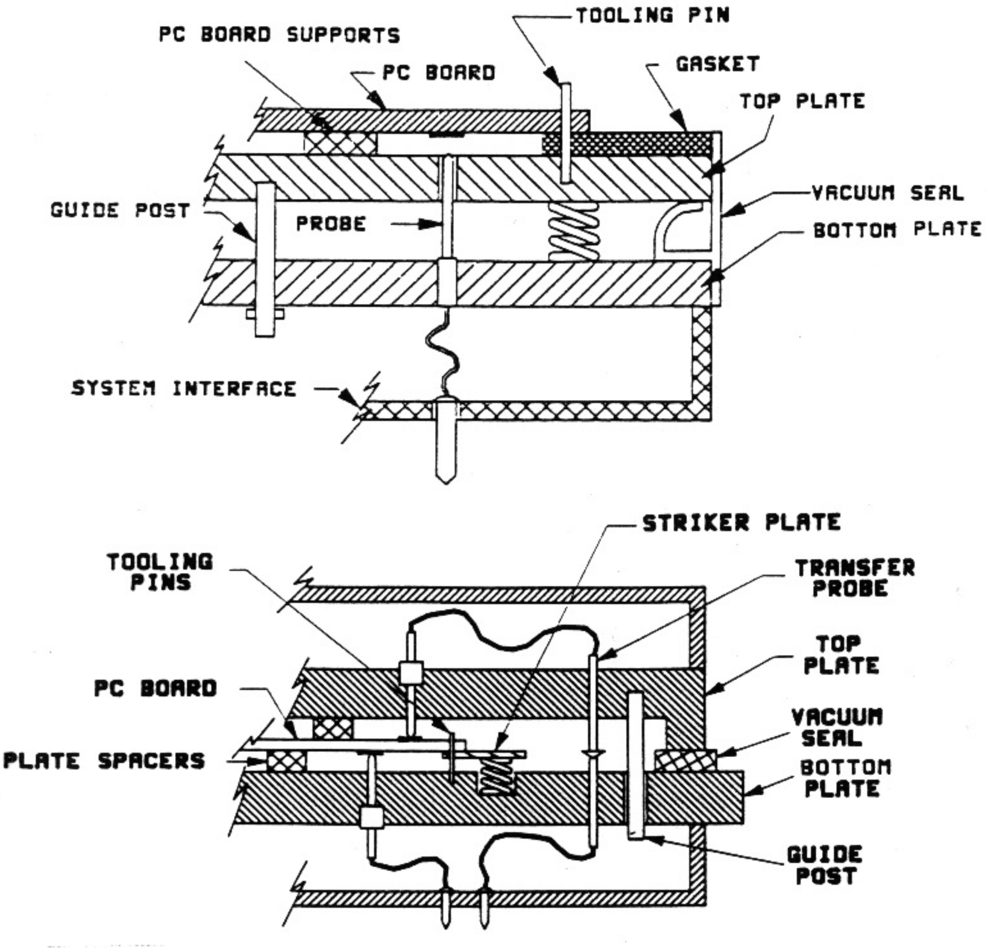

Fig. 7.47: a) Detail of single sided test fixture, b) double sided fixture.

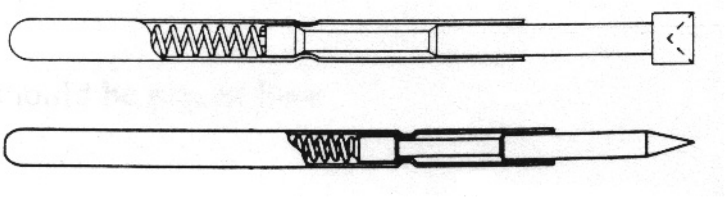

Fig. 7.48: Two types of test pins.

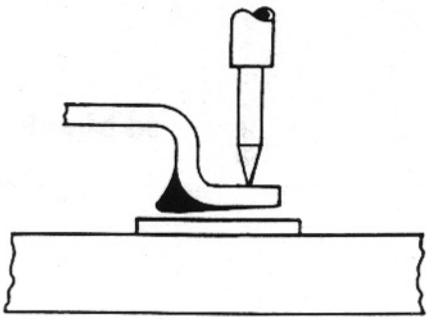

Fig. 7.49: Unacceptable testing. The test point should be on the Cu foil on the board, not on the component lead.

7.6.3 "Zero defect" philosophy

After each step in the production process, the value of the product increases. Repairs and changes may be more complicated to perform. Discarding a faulty product is more costly the later in the process it happens. Therefore, it is important to detect faults as soon as possible after they have been made and make the test strategy accordingly.

Earlier it was normal to accept a high fault rate, repair the faults and test again. Today the goal is that faults should not occur. Repairs are costly, and it is likely that other units just passing the test may fail later, in the field, where the consequences are much worse. Faults tell that there is something wrong with the process, the components, or the design. The correct procedure is to remove the cause of the fault. "Fix the process, not the product" is the principle today.

REFERENCES

[7.1] G. Leonida: "Handbook of printed circuit design, manufacture, components and assembly". (Electrochemical Publications, 1981.)

[7.2] R.J. Klein Wassink: "Soldering in Electronics" (Electrochemical Publications, 1984).

[7.3] "Robots for electronics production". EPF Workshop October 1988,
and: EPF-report R 20: "Robot systems for electronics production", February 1988.

[7.4] L. Halbo et al.: "Second generation SMT: Components, substrates, mounting and soldering". EPF-report R 29, 1989.

[7.5] C. F. Coombs: "Printed circuits handbook". (3rd Ed., McGraw Hill, 1988).

[7.6] C. Lea: "A scientific guide to surface mount technology". (Electrochemical Publications, 1988.)

[7.7] G. Ginsberg: "Surfacemount and related technologies". (Marcel Dekker, 1989.)

[7.8] ANSI/IPC-CM-770C: "Printed circuit board component mounting". (The Institute of Interconnecting and Packaging Electronic Circuits, 1987.)

[7.9] J. F. Pawling: Surface mount assemblies". (Electrochemical Publications, 1987.)

[7.10] J. Wangensteen: " Testing of surface mounted PCBs". EKF report R 43, March 1986.

[7.11] A. Gabrielsen et al.: "In-circuit testing of surface mounted PCBs". EKF report R 60, December 1987.