

BASIC PRINTED CIRCUIT
BOARD MANUFACTURE

Bob Willis
Electronic Presentation Services
2 Fourth Avenue
Chelmsford
Essex
CM1 4HA
England

Tel: (44) 01245 351502
Fax: (44) 01245 496123
Email: bob@bobwillis.co.uk
Web: www.bobwillis.co.uk

BASIC PRINTED BOARD MANUFACTURE

The production of printed circuit boards, be they single or double sided, plated through hole or multilayer, is a very complex operation. It is not possible to cover in any detail the many processes involved in board production. Each printed circuit company may have its own individual processes and techniques to produce a finished board in the most economical way. In the main each must follow the basic steps in production.

The following notes are intended to give a basic step by step guide to the main operations to produce high quality printed circuit boards. To obtain further details on individual processes a list of reference books is attached.

MULTILAYER BOARDS

Multilayer represent the most complex type of PCB. Their cost is very high owing to the complexity of the manufacturing process, the lower production yields and the difficulty in reworking most of the rejects. They find applications in professional electronics (computers, military equipment etc.) where the density of connections makes it impossible to run all conductors on two planes. Despite the increased use of high integration components (which require a large number of connections in a small space), multilayer PCBs represent only about 10% of the total area of PCBs used.

The multilayer technique makes more than two planes (normally 3-12) available to the PCB designer for running conductors and making large ground or supply areas, very useful in high speed circuitry.

To illustrate the process, a four layer board having two external layers and two internal layers, will be described. Thin copper-clad laminate is normally used from 0.2mm to 0.8mm thick to make the etched layers, being bonded together with prepreg or bonding sheet.

Multilayer printed circuits are produced by etching tracks and pads on to one or more sides of thin copper clad laminate, using dry film photoresist to image the circuit. In some countries wet resist is still used due to cost reasons, but with the reduction in tracks and gaps this practice is becoming impractical. The following are the basic steps in the manufacture cycle.

The first step is to produce the blanks from the thin copper clad laminate which will go towards the fabrication of the inner layers of circuitry. The size of the blanks will depend on the size capability of the plant equipment and the economies or production size.

Tooling holes are punched or drilled in each of the layers for use in registration or artwork and layer registration. Each of the layers are then coated with resist and imaged to produce the circuitry required. All the exposed copper areas are chemically etched and the photoresist is stripped from the remaining copper. The layers are then inspected at this stage for any defects. It is very common due to the complexity of circuits to bare board test at this stage due to the difficulty in repair at a later stage.

Each layer is now complete and ready for lamination., To obtain a satisfactory adhesion to each layer it is necessary to produce a key on the surface of the copper. This is done by chemically oxidising the surface, one of the most popular forms is a black copper oxide which is grown on the surface, this improves the adhesion of the epoxy prepreg to the circuit lines.

After oxide treatment each panel is rinsed and baked in an oven. The baking is important at this stage to remove any moisture which has been taken on during the progress steps so far. It is important to remove the moisture prior to further processing.

The next stage is to lay up the panel with the prepreg prior to lamination. The prepreg acts as the filler material which is the dielectric between the circuit layers. Prepreg is in the case of epoxy glass prepreg glass cloth impregnated with epoxy resin which is cured to the "B" stage.

With the panels and the prepreg cut to size and made up into a sandwich which is made up of all the required layers, this would consist of the following layers:

- a) A press plate, usually of hardener tool steel
- b) A panel with copper foil on one side, placed copper side down
- c) A sheet of prepreg
- d) A double sided etched panel
- e) A sheet of prepreg
- f) A panel with copper foil on one side, placed copper side up
- g) A press plate

The above package is for a standard four layer board, to produce such a circuit many board manufacturers are using a technique called mass lamination which is not covered within these notes.

To control the rate of heat transfer kraft paper is used as an insulator between outside of the stack and the press plates.

The sandwich is now inserted in a laminating press, the size and number of board packages which can be produced at one time will depend on the size of the equipment available.

Lamination requires a specific time/temperature/pressure cycle which depends upon the prepreg characteristics and upon the equipment used. In most cases this will involve a press cycle of between 30-90 minutes at approximately 170°C and a subsequent cooling period before the pressure is removed.

During lamination the heated resin softens and with pressure causes it to flow and fill all the voids between circuitry. The cycle parameters are very important to produce a board to a satisfactory standard. One problem which exists is the variation with age of the prepreg, with age its characteristics can change affecting its flow rate.

At the completion of the press cycle the board stack would be removed from the hot press and placed in another cold press for cooling under pressure. The transfer allows the best use of the resources of the hot press.

After the cold press cycle has been completed the package is removed from the press, depending on the system used for manufacture. Any tooling pins would be removed at this stage and each completed panel removed.

The epoxy flash around the board is removed to allow further processing to take place on the standard size panel.

The epoxy flash is the excess material within the prepreg which has flowed outside the edges of the panel. It is important for evidence of this flash to be visible to assure correct processing.

The panels at this stage may be baked still further to assume full cure of the prepreg and reduce instability of the basic materials. Some manufacturing companies leave boards in the cool down press and reduce the temperature over a period of time, this prevents warpage. At this stage the panels are near ready for drilling, prior to this operation. Reference marks must be exposed on the inner layers to allow registration of the panel for drilling. This requires removal of the board material down to the inner layer on chemical removal of a small strip of copper from the outside of the blank. After this stage the panel can be located and the drilling operation can proceed. Registration of the internal layers if more than two can be conducted either before or after drilling using test coupons taken from the panel or with the use of X-ray. A further method is to use an etched panel and drill it as the first off. This allows a simple evaluation of registration.

After drilling the panels are nearly ready to be treated as standard plated through hole boards, but one of the most important operations is smear removal. Smear epoxy can be placed on conductors during the drilling operation. It is most important for this to be removed to provide inter-connection during the plating operation. None or poor removal will result in loss of contact.

Smear removal is performed just before electroless copper plating of the holes. This operation is sometimes referred to as etchback, since the epoxy resin is etched back from the edge of the hole at the inner layers. Two methods are currently used, plasmer etch which requires a gas and a high voltage charge or a chemical method using permanganate. Often combination chemical processes are used to attack both glass and epoxy.

After smear removal, the process for standard P.T.H. boards is basically followed. One quality control check that is important at this stage is the effectiveness of smear removal. Test coupons may be taken after electroless plating to check the inter connections of each of the layers.

See P.T.H. notes for further process steps.

PLATED THROUGH HOLE BOARDS

The following is a basic introduction to the steps required to produce a printed circuit board. A list of books covering the manufacture and assembly can be found at the back of these notes.

The first operation in the manufacture of a printed board is the cutting to size of the copper clad laminate. This is called the process blank, which may vary in size depending on the number and finished board size. Holes are punched in each blank to enable alignment during further processing.

For the drilling operation the panels are pinned together in stacks from one to four high, depending on the panel thickness. The stacks are then mounted on to the drilling machine. A punched tape which contains the program controls the movement of the drilling head and the selection of drill size.

After drilling, the panels are deburred then they proceed to the first chemical operations. The first part of the plating process is to have copper chemically deposited in the holes this is called electroless copper, or through hole plating, the electroless copper deposit is very thin, but it serves two purposes.

1. It provides the electrical connection between both sides of the panel
2. It serves as a metal surface for subsequent electrically plated copper to be deposited

After the holes have been plated the circuit pattern is imaged onto the panels. The image defines the electrical circuit for plating and etching. The image is applied using a dry film photoresist. Both sides of the panel are coated and using the circuit artwork it is imaged. The image is obtained by laying the artwork on top of the photoresist and exposing it to ultraviolet light. The light polymerises the resist from a soft state to a hard, chemically resistant plating resist. The unexposed photoresist is washed away during the development to leave the circuit pattern.

The next stage is a continuous process which first electroplates copper onto the circuit pattern and through the holes to increase the thickness. The copper is then plated over with tin/lead. The second metal performs two functions:

1. It serves as an etch resist for the subsequent copper foil etching operations. Tin/lead is not the only material used, but it is the most common.
2. It can subsequently be the basis of a long term solderable finish.

After the circuit has been plated to the correct thickness, the resist must be stripped. This will expose the unwanted copper which must be etched away to leave the plated circuit.

The copper is etched by spraying the etchant on to both sides of the panel as it moves on a conveyor. The speed of the conveyor and the type of etchant used play a major role in the effect on undercut/track reduction. After etching the panels show tin/lead covered copper circuits on the base epoxy/fibreglass.

Contact finger plating is the next operation in the manufacture cycle. These are gold edge connectors which are used as contact points after board assembly. Contact fingers are rows of tabs along one or both sides of the board. Gold plate is the most popular material due to its durability and resistant to tarnish and oxidation. Plating tape is applied to mask off the edge contact from the rest of the circuit. The tin/lead is then chemically stripped in the contact areas, with gold plated to each contact as the finishing surface, a nickel coating may also be used under the gold to act as a diffusion barrier.

Reflow or tin/lead fusing is the next operation. This is conducted using infrared on a conveyor system. Tin/lead is a dull grey metal and it may be porous with a limited solderable storage life. The reflow process melts the tin/lead. This fuses the two metals to form a true alloy which is brighter, shiny and corrosion resistant. Tin/lead plating is sometimes called solder plating, because solder is an alloy of tin/lead. The actual alloy is not formed until the reflow operation.

One of the final operations is the solder mask application. This can be done in one of three ways:

An epoxy coating which is applied to each side in turn with a curing operation between each application. The epoxy ink image is applied by screen printing

Dry film resist which is vacuum laminated to the panel in the same manner as photoresist is applied, in fact both processes are very similar. Dry film after lamination is exposed to ultraviolet light through an artwork to expose limited areas of the circuit. After exposure the panel is developed and then oven cured. Dry film use has dropped considerably in the industry over the last few years in favour of liquid photo imageable materials.

Photo imageable inks are applied to the surface of the board by spray, screen or curtain coating the board, partly drying the surface and repeating the operation on the second side. They are then exposed with UV light through artwork to define the pattern.

After the application of resist the legend is applied by screen printing and a curing operation. The legend is in the form of letters and numbers which aid the assembly and identification of component positions.

The final parts of manufacture are to produce the board or boards from the process panel. Each board has its own special shape and must be routed from the panel to meet the dimensional requirements.

The manufacture of printed boards is very complex with many mechanical and chemical operations conducted on each circuit. Inspection and process control staff monitor the production cycle to maintain the controls required to produce high quality boards.

REFERENCE MATERIAL

Multilayer Printed Circuit
Electrochemical Publications 1985

J.A. Scarlett

Printed Circuit Handbook
McGraw Hill - 1988 (Third Edition)

C.F. Coombs

IPC A 600 Acceptability of Printed Circuits - 2000

SPECIAL REQUIREMENTS FOR SURFACE MOUNT CIRCUITS

The introduction of surface mount technology has had a major effect on the electronics industry with a change of materials, component type and size. It has also seen new processes which assembly engineers have had to become familiar with during the board population stage. Generally speaking the processes associated with the fabrication of printed circuit boards for surface mount technology have not changed, so the content of this section should not be read in isolation from the board manufacturing requirements.

The use of SMT has had and will continue to have a direct effect on printed board features, with reductions in track and gap size, plated through holes for both via and component mounting, this will of course effect the price and should be discussed with the manufacturer for possible alternative design considerations.

Although the basic manufacturing method for fabricating printed boards has not changed radically, the process that are used for the assembly stages have changed which may in some cases require changes to the methods of manufacture or finished quality of the basic board. The direct involvement of the printed board manufacturer, during each stage of the project will allow a better understanding of the final bare board requirement.

The new assembly stages which board are required to pass through are screen printing of solder cream, component placement and reflow by vapour phase or infra red. The use of wave soldering techniques for mixed technology boards will also be prominent for many years as this can be a very cost effective method of board assembly. The new process stages, the component mixture and the type of working environment have affected the requirement of the printed circuit board. To provide a better understanding of the particular requirements of modern assembly methods and to provide a better understanding of the processes, which may be new to many board manufacturers, it is essential that suppliers and users have close contact during each stage of a project.

PCB SUBSTRATE MATERIAL

The selection of a substrate for surface mount applications will depend on the electrical requirements, the component type and the operating environment that the final assembly will have to withstand. Standard laminates like glass epoxy FR4 are widely used for most applications in surface mount products, it is only were the use of specialised components like leadless ceramic chip carriers (LCCC), that further consideration needs to be given to the substrate.

The use of this package in situation were the assembly is going to be subjected to either temperature or power cycling will require an alternative board material if failure of the solder joints is not to occur. The material used for LCCC packages has a low thermal coefficient of expansion (TCE), the glass-epoxy resin system has a TCE of approximately three times that of the ceramic causing a thermal expansion mismatch between the two surfaces. The temperature which the board may be subjected should be considered in detail, it is not only the operating environment which needs consideration, but the effects of power cycling , storage and transport. The mismatch will in time lead to failure of the solder joints formed between the LCCC and the base substrate.

In situations where alternative components can not be used the designer will insure an extra cost in the basic board for alternative substrates to overcome this problem. Two basic alternatives exist the use of a board with a compliant elastomeric layer, which may be bonded directly to a standard laminated like FR4, the layer forms a compliant surface between the laminate and solder joint connection, this will in certain environments absorb the stresses imposed. The second choice is the use of matched materials which will expand and contract at the same rate as the LCCC. Examples of substrates which are currently available are epoxy/polyimide kevlar, polyimide quartz and copper-invar-copper. The use of these materials should be considered in detail prior to their selection due to the cost involved and the difficulty in board fabrication.

ARTWORK AND DRAWINGS

The drawings and possibly the artwork are generally the first opportunity for a manufacturer to see the type of printed board he has been asked to produce, certain features on the circuit may change and often the tolerance requirements are different for SMT boards.

In many cases, on surface mount boards additional information and restrictions are required, which must be fully documented to prevent any confusion at a latter stage. In the case of special processes like solder levelling they should in the first case be discussed with the manufacturer and specified as a requirement for a finished coating.

The tolerance requirement on actual board size may require further documentation due to the requirements of automatic handling during assembly and in process transport. A tolerance is generally required on two opposite sides of the panel to provide parallel surface to aid smooth handling.

The artwork for the manufacture of any printed circuit board is the single most important tool to produce a quality circuit board, it is also the main area which may cause tolerance errors on the final product. To provide the optimum quality photo tooling is advisable to allow the manufacturer to produce his own master artwork directly from the customers CAD (computer aided design) system, the information can be supplied direct on magnetic tape or floppy disc. Information supplied in his format also allows the circuit to be step and repeated to provide the most economic to manufacture panel, the added benefit of this process also enables the printed board to be supplied, if required in a multi-panel to provide ease of manufacture during assembly.

The use of direct photoplotted artwork has now become a standard for new designs which require greater control of the basic tolerances. Surface mount processing has also the requirement for extra artwork, in the case of reflow where solder paste is to be applied as part of the assembly stage an artwork is required for the basic pad footprints to make the screen.

TRACK PATTERNS

To take further advantage of the reduction in size offered by surface mount technology the basic track sizes are continually under review, at present 0.006-0.008" is commonly the minimum track size requested for board tracking. As the design size decreases more considerations are required at the design stage to make manufacture of the basic board.

For close tolerance requirements the use of thinner foils, half ounce (18 microns) on the basics laminate become a necessity for fine lines, with the move to tracks less than 0.008" thinner foils, quarter ounce (9 micron) may well become common place, but at a cost.

The size of tracks and pads on surface mount boards is continuously under review in an attempt to improve the soldering yields and decrease the size of the design. The reduction of geometry on the basic board will effect its producability and should be discussed with the manufacturer as this may well effect the final price. It is often the case that savings on the product price may be obtained by early discussions with the circuit supplier.

HOLE SIZES AND VIA SIZES

The introduction of surface mount components has seen the requirement for smaller hole sizes, this has not been the case for mixed technology boards were the requirement for leaded components still limits the choice on final hole size. The actual conventional component termination does allow for hole size reduction, but as more companies are introducing automatic insertion this is the limiting factor. Due to the range of tolerances involved with boards, insertion equipment, components and the programming a average size increase of between 0.4-0.5mm to the lead size is required providing a range of finished hole sizes of 0.85-1.1mm. The pressure to reduce hole sizes is all in the area of via holes, in most cases the common sizes presently used are 0.5-0.6mm with the common requirement to reduce this figure down to 0.4mm and beyond.

The introduction to landless via hole has provided some area of improvement with the elimination of the pad area allowing more open area for track routing, this again needs consideration due to the tolerance capability of the process. The introduction of small holes does require consideration as the size and use of multilayer boards is now on the increase, the aspect ratio of the hole size to finished package thickness will have a dramatic effect on the ability to drill and plate a reliable through hole interconnection. A general guide to the aspect ratio for manufacture is a 5/6-1 ratio of hole size to package build.

SOLDER RESIST MASKS

There are three main types of solder resist commonly in use today, the first is a two part epoxy system which is screen printed directly to the circuit leaving the required exposed apertures. This process is not particularly accurate for the close tolerance requirements of surface mount designs and does lead to resist bleed on to solderable areas of the circuit. If proper clearance tolerances are provided for the process it defeats one of the prime advantages of surface mount, that of size reduction.

The process which has been shown to provide the accuracy and reproducibility is the use of photo-imagable resist systems.

Dry film resist when originally used was available in three thicknesses, 50 75 100 microns thickness each being laminated to the surface of the board as previously stated. Each of the materials were used on surface mount boards, the thinnest of these providing many advantages to the assembly operation however are rarely used today.

The use of the thinner materials has shown to provide higher soldering yields on both flow solder and reflow solder during the production cycle. The use of these resists does however put limitations on the board designer and circuit manufacturer in the materials and processes that he may specify.

The third material which is now the most popular is the liquid photopolymers which may be coated to the board in two ways, by curtain coating or open screen printing. After application the coating is dried to allow further processing to be conducted.

The next steps are the same for any photoimagable system the surface is imaged using photographic artwork and developed to relieve the solder pads. The finished coating thickness may be in the area of 25 microns this is of course dependent on the design and the plating thickness on the board surface. The manufacturing requirement should be discussed with the circuit fabricator to provide a cost effective solution which meets the mechanical, electrical and assembly requirements.

As a guide the tolerance requirements for resist systems are for screen print liquid systems 0.010" tolerance around the pad area to prevent resist encroachment. In the case of dry film or photoimagable liquid systems a 0.005" window should be left around the pad area this guide will prevent the solderable area being covered by resist.

A further point should also be made that this tolerance should also be left around test pads, in the case where thick resists are used it is possible for the test pins during test to be deflected and cause open circuits to be recorded. It is possible for this to occur during bare board or final product test.

LEGEND MARKINGS

It is still a requirement of many companies to provide markings on the board to identify selected component positions to provide component identification. This may be used during inspection, repair and test it also in many cases provides the basis for a operator assembly guide. With the general reduction in board size limited board area is now available for markings, it is not possible using existing methods of screen printing to reduce the size of lines and characters to the available area. The use of characters etched into the copper base foil during etching the circuit may in some cases provide another alternative for some applications, it should however be noted that it may effect the operation of the completed circuit.

Close co-operation with the supplier is important as the selection of resists may effect the ease of screen printing legends, the close proximity to solderable area can lead to contaminated pads. Although space is often available under certain components this in many cases leads to contaminated solder pads.

SOLDER FINISHES

Traditionally in the fabrication of printed circuit boards solder resists are placed over the tin/lead finish, either plated or after reflow. This has always been an area of concern to the assembler mainly from the cosmetic point of view. During the assembly operation on flow solder the tin/lead finish would reflow under the resist and cause the protective coating to crinkle, in the worst cases fall off the board surface. With the processes commonly used for surface mount assembly it is common for the hole board surface to be subjected to temperatures above the reflow point of tin/lead.

The popular alternative finish for surface mount boards is to have a selective coating of tin/lead on the plated through holes and pad areas only, this is generally applied by solder levelling, which has become an industry standard. The process provides a coating thickness which is inconsistent across the board surface, but does provide a reliable coating to provide a finish which provide a good solderable coating.

The present requirements for board finishes can be met by solder levelled finishes, but as the electronics industry move away from the common 0.050" component pitch to 0.025" the finish provided by solder levelling will not provide the ideal consistent level coating. As the size of component termination's is decreased the height of the solderable area will become more critical to the assembly operation.

During the screen printing of solder paste the deposit will depend on the height of the tinned pad areas, this in turn will effect the height of placement which may in turn lead to open circuit joints in non contact reflow methods.

Alternative coatings like the use of gold flash over a nickel coating or electroless gold flash provide advantages to the assembler of more consistent coating across the board surface. The finished provides a flat surface to screen print solder paste and place components, it provides a solderable coating which will last during storage and a coating thickness of less than 0.2microns. The thin coating of less than 0.2microns prevents any problems of brittle intermetallic being formed at the solder joint interface leading to joint failure.

A further finish which is used commonly on some products is a flux lacquer or oxide inhibitor which is applied directly to the copper surface tracking to provide a protective coating to maintain solderability. Flux lacquers are one of the cheapest coating that exist it does, however only provide a limited self life to maintain a solderable surface often less than 3 months.

REGISTRATION TOOLING HOLES

With the use of automatic component insertion equipment for conventional printed boards the most important factor is the registration of tooling holes to component holes. Generally holes up to 5-6mm are all drilled in one operation/setting of the drilling machine during the fabrication stage of the board, in this case the hole to hole tolerance is going to be good. The factors which effect the tolerances experienced during assembly are generally due to the processing steps during board fabrication and the instability of the base laminate.

The requirement for surface mount boards is different generally they are smaller than conventional boards although being manufactured in multi formats the dimensional instability will be smaller. The automatic component onserter requires good registration between the tooling holes and the copper pads, with hole drilling being one operation and photoimaging being another it is common for this to be an area of dimensional inaccuracy. The registration of a second artwork for resist application may also lead to a build up in the registration error. These examples show the possible errors which can occur which require consideration during the design to provide suitable tolerances to be included to the board layout.

For surface mount assembly there are requirements for further tooling holes too be included with in the board, these are required for registration during screen printing and component placement each must be specified for its size and be unplated. Generally the same requirements exist for tooling holes for conventional assembly during automatic insertion.

BOARD SIZE AND MULTI PANELS

The actual board size and shape are determined by the final equipment and its application, due to the considerable variety most companies are adopting standard board sizes due to the high cost of tooling for individual boards. Traditionally each board was designed as a single part with the artwork supplied to the printed circuit manufacture who would produce artwork and drilling tapes to make the board in a multiple panel. The circuit would be laid out to make the most cost effective use of his standard panel size, a common one being 18"x24".

The common approach in order that time can be saved during assembly is to take a multi panel direct from the circuit supplier with the individual board shapes routed in the panel, but still held in place by tabs of the base material. The finished board after it has been fully assembled and tested can be easily removed, this approach eliminates the handling of difficult sized boards and reduces the tooling requirements of assembly. The design of the multi panel should be discussed in detail, as the optimum layout for board manufacture may not be ideal for the assembly process.

BARE BOARD TESTING

Modern printed boards justify that circuit testing be conducted at the earliest stage in the manufacture cycle the increasing density of printed circuit boards and the move to surface mount assemblies has proved that the final test of individual boards is a necessary requirement. As the density increases the difficulty to detect failure points on the finished assembly becomes more evident, if the cause of failure is a fault on the basic board then it may be impossible to detect. It is now imperative that any defect which may be present on the circuit must be detected, the board must be confirmed to be satisfactory before further assembly processing.

In the case of specifying bare board test it is important that it be discussed in detail so that the individual requirements are understood, 100% bare board testing does not necessarily mean that full circuit tracking have been 100% tested.

TEXT BOOKS AVAILABLE ON SOLDERING AND PCB MANUFACTURE

- Soldering in Electronics By Klein Wassink** ISBN 0 901150 14 2
Electrochemical Publications Ltd
8 Barns Street
Ayr
Scotland
KA7 1XA
- Solders and Soldering By Howard Manko** ISBN 0 07039897 6
McGraw Hill Book Co
- Printed Circuits Handbook By Clyde Coombs** ISBN 0 07 012609 7
McGraw Hill Book Co
- Scientific Guide to Surface Mount Technology By Colin Lea** ISBN 0 901150 22 3
Electrochemical Publications Ltd
8 Barns Street
Ayr
Scotland
KA7 1XA
- Flexible Circuits-Design and Applications By Steve Gurley** ISBN 0 8247 7215 6
Marcel Dekker Inc
New York
- Handbook of Surface Mount Technology By Stephen Hinc** ISBN 0 582 005175
Longman Scientific Technical
Longman House
Burnt Mill
Harlow
Essex
- Multilayer Printed Circuit Handbook By John Scarlet** ISBN 0 0 901150 15 0
Electrochemical Publications Ltd
8 Barns Street
Ayr
Scotland
KA7 1XA
- Quality Assessment of Printed Circuit Boards By Preben Lund** ISBN 0 9601748 4 2
Bishop Graphics Inc
5388 Sterling Center Drive
PO BOX 5007
Westlake Village
California 91359 5007

**Cleaning and Contamination of Electronics
Components and Assemblies by Brian N Ellis**
Electrochemical Publications Ltd
8 Barns Street
Ayr
Scotland
KA7 1XA

ISBN 0 901150 20 7

Solder Paste in Electronics Packaging by Jennie S Hwang
Van Nostrand Reinhold
Chapman and Hall
11 New Fetter Lane
London
EC4P 4EE

ISBN 0442 20754 9

Surface Mount Technology by Ray P Prasad
Van Nostrand Reinhold
Chapman and Hall
11 New Fetter Lane
London
EC4P 4EE

ISBN 0 442 20527 9

**Microelectronics Packaging Handbook
by Rae Tummala and Eugene J Rymaszewski**
Van Nostrand Reinhold
Chapman and Hall
11 New Fetter Lane
London
EC4P 4EE

ISBN 0 442 20578

**Handbook of Printed Circuit Design, Manufacture,
Components and Assembly by G Leonida**
Electrochemical Publications Ltd
8 Barns Street
Ayr
Scotland
KA7 1XA

ISBN 0 901150 09 6

**Thick Film Hybrids Manufacture and Design
by Malcolm R Haskard**
Prentice Hall
66 Wood Lane End
Hemel Hempstead
Hertfordshire
HP2 4RG

ISBN 0 7248 1194X

Handbook of Machine Soldering by Ralph W Woodgate

ISBN 0 471 85779 3

John Wiley & Sons
Baffins Lane
Chichester
West Sussex
PO19 1UD

**End Users Guide to Innovative Flexible Circuit Packaging
by Jay Miniet**

ISBN 0 8247 7584 8

Marcel Dekker Inc
New York

Polymer Thick Film Technology A Review by P L Kirby

Patinel Publications
England

**Productivity & Quality Improvements
A Practical Guide to SPC by John L Hradesky**

ISBN 0 07 030499 8

McGraw Hill Book Co
Shppenangers Road
Maidenhead
Berkshire
SL6 5BR

Computer Integrated Testing by Alan Buckroyd

ISBN 0 632 02042 3

Blackwell Scientific Publications
8 Johns Road
London
WC1N 2ES