FUSE TTN Training Material

PRINTED CIRCUIT BOARD AND SURFACE MOUNT DESIGN

Version 7 June 1999

Contents

INTRODUCTION

A Printed Circuit Board (PCB) in its simplest form consists of a thin board of insulating material that supports the components in a circuit and conducting tracks, usually copper on one or both sides of the board material connecting the components together. Component leads are soldered to lands, which are also known as pads, (i.e.) parts of the track with space for a soldered joint between the component and the track. Lands may have holes drilled through the board to facilitate component mounting (through-hole technology) or the component may be placed on the land (surface-mount technology).

In early times this description would have been sufficient however in recent years electronic equipment for all types of applications has seen major advances in performance, size and cost. Improvements in PCB design have been one of the major contributors. PCBs now play an important role in determining the active functioning of electronic circuits and may no longer be regarded as a passive interconnection panel.

PCB CLASSIFICATION

Continual improvements in PCB technology have resulted in a host of different PCB formats to suit new component designs. It is quite common for PCB technologies to overlap making the task of defining types quite difficult. The following list attempts to clarify the situation with all PCB types falling into one of three main categories:

- Single-sided PCB
- Double-sided PCB
- Multi-layer

Single-sided

A single sided PCB has conductive tracks on one side and components on the other. Occasionally the components are located on the same side as the tracks or on both sides depending on how complex the design is. Figure 1 illustrates the most common configuration and uses through-hole technology (covered later) to demonstrate the board assembly.

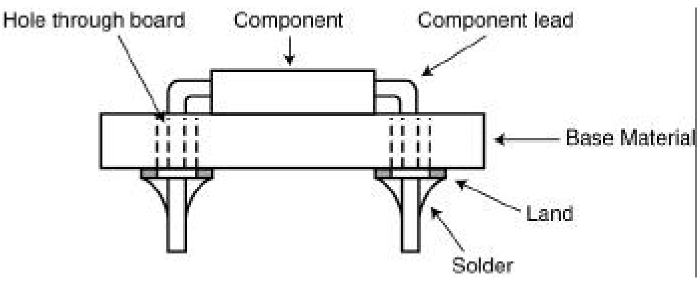


Figure 1 Single sided PCB

Double-sided

PCBs of this nature use conductive tracks on each side of the board. Once again components may be mounted on either or both sides. The combined result enables increased component densities over single sided design. Interconnection between tracks is implemented using a number of methods namely, eyelets, wire, component leads and plated through holes. The latter is by far the most common and is implemented by plating the hole-wall with a conductive material. Plated through holes which are solely used to connect tracks on opposite sides of the board are known as "
illustrates a double-sided board with tracks on both sides and a via connection between two tracks.

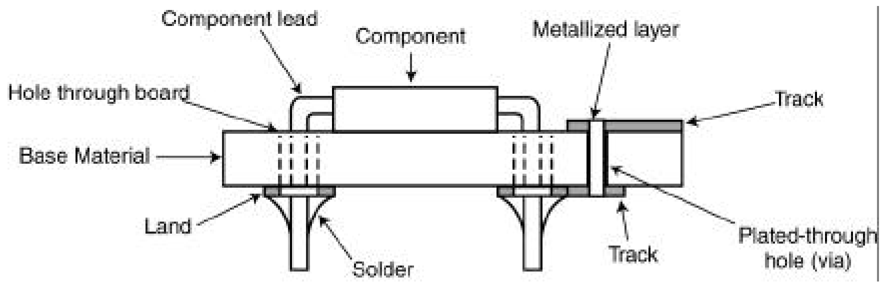


Figure 2 Double sided PCB

Multi-layer

As the name implies these boards have several layers of tracks, two of which are on the surface of the board. The remaining layers are located in the board material, which is made up of a series of laminated tracks and insulation layers. Connections between tracks are implemented using through or buried vias. A "through via" simply passes through the board while a "buried via" connects internal tracks. A multi-layer design is illustrated in figure 3.

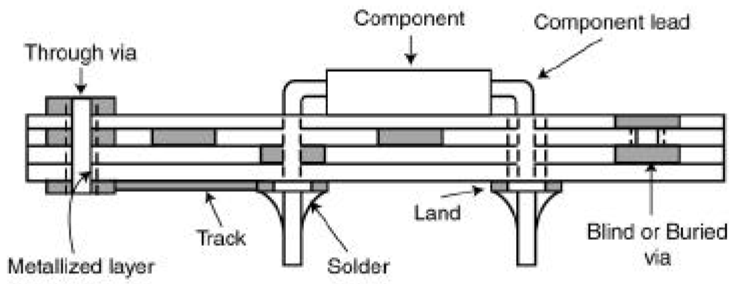


Figure 3 Multi-layer PCB

VARIATIONS ON PCB DESIGNS

The following are the most common variations to the PCB classifications outlined:

- Through-hole circuits
- Surface mounted circuits
- Hybrid circuits

Through-hole circuits

The components in this type of circuit have wire leads that are inserted through holes on the PCB, hence the term “through hole technology”. Electrical conductivity is established by soldering the wire lead to the land on either side of the board. This technology may be used on all board classifications outlined earlier and illustrated in figures 1,2 and 3. Figures 4 and 5 illustrate the difference between through hole and plated through holes mounting techniques.

Through-hole designs using single and double-sided boards have been around since the 1940s and 1950s respectively, and offer a fast and simple solution to many designs. Over the years track densities have continued to increase, today track spacing of 250µm is common. Systems of this nature are easy to build, test and service but on more complex designs the hardware is physically large and may be electrically noisy for medium to high frequency applications.

In an attempt to meet the demand for higher pin counts and reduced package size, manufacturers have produced high-density through-hole components with reduced lead pitch and innovative packaging options, some of which are covered later. Package options include “zigzag” lead out patterns and packaging where the component is essentially mounted on its side thus taking up less area (but at the expense of an increased vertical profile.) Reduced pitch devices include shrunken lead pitch (i.e. less than the conventional 2.54mm). A major problem with high density TH is that few devices are available in the higher density style packages and many advanced integrated circuits are not available at all in through hole technology (e.g. many high density FPGAs)

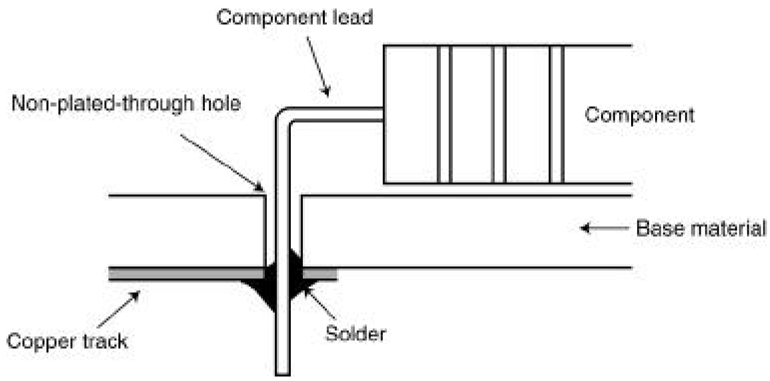


Figure 4 Through hole technology

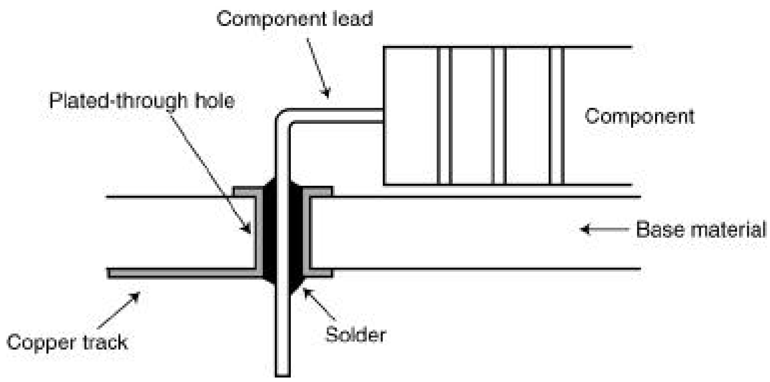


Figure 5 Plated through hole technology

Hybrid circuits

Hybrid circuits consist of passive components that are implemented using film technology placed on to a base, and discrete passive and active components are added to this film to complete the circuit. Two forms of film technology exist, namely: thick film and thin film. The former is manufactured using printing, drying and firing pasting techniques while the latter uses vapour deposition and selective etching processes.

Manufacturing techniques for this technology are well established and have been around for nearly forty years, it is however a very specialised process. The board structure is illustrated in figure 6.

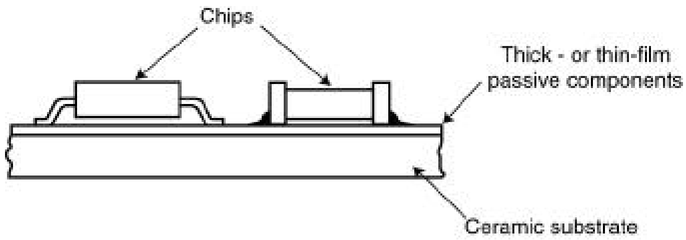


Figure 6   Hybrid Technology

Surface Mount Technology (SMT)

Surface mounted assemblies are essentially a marriage of printed and hybrid circuit methods. As with through-hole technology, discrete components are mounted onto a printed circuit board, however in this instance the lands are not drilled since the components used have no wire leads. The components are positioned on the lands and then soldered into position as illustrated in figure 7.

This technology has been around since the mid 60s and is still evolving rapidly. Initially, devices were produced that were little more than-dual-in-line (through hole) packages with the leads cropped and formed so that the component could be soldered onto the surface of the PCB. Typical lead spacing of these "conventional" SMT devices is 1.27mm, half that of through-hole types. Today, lead pitch is continually being reduced, and new package styles such as Ball-Grid-Arrays (BGA) are satisfying the most demanding applications.

SMT technology offers increased component density over TH and supports higher performance but is more difficult to manufacture and repair. Manufacturing and test are generally automated processes involving high capital and stock investment.

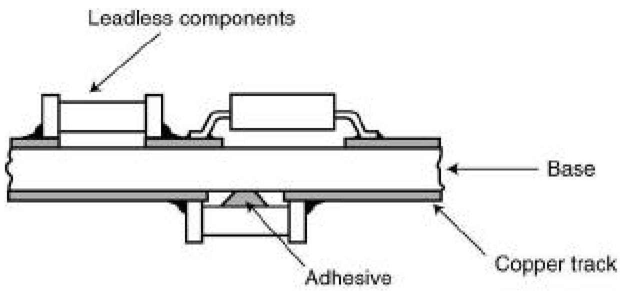


Figure 7

Packaging Technologies

Electronic equipment manufacturers are constantly striving to achieve the following:

- Ultra low cost product
- Thin, light and portable design
- High performance
- User friendliness
- Diverse functions involving a variety of semiconductor chip

These points have been the driving force behind the rapid advances in component packaging technologies that have occurred right across the board, i.e., for through hole and surface mount applications.

Through hole packaging has advanced to a level where component densities have almost been maximised with the use of pin grid array (PGA) packages.

Surface mount technology on the other hand still continues to miniaturise, to the extent where Chip Scale Packages (CSP) which are only marginally bigger than the chip itself are now common place in the more advanced circuit designs.

Figure 8 illustrates the time span over which the different packages have evolved and how the advances in technology have affected aspects such as component pin count, size and weight. Figure 9 displays some of the devices in more detail and provides an explanation for the package abbreviations commonly used.

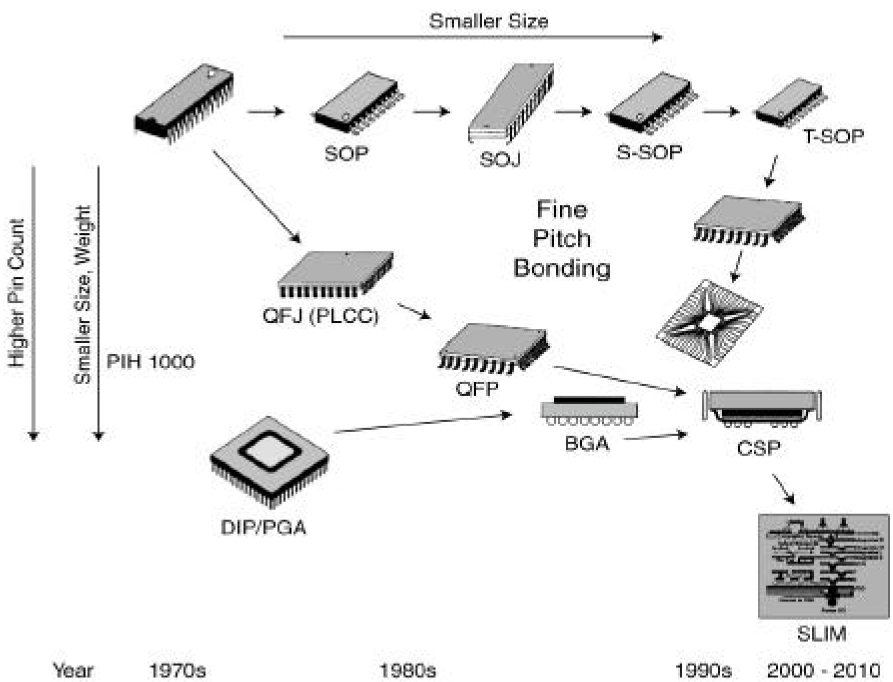
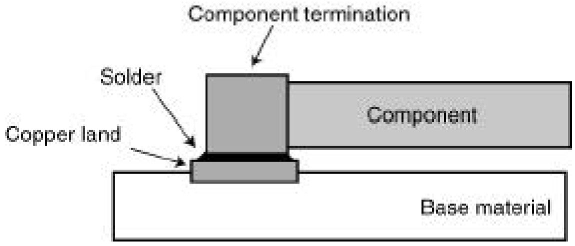
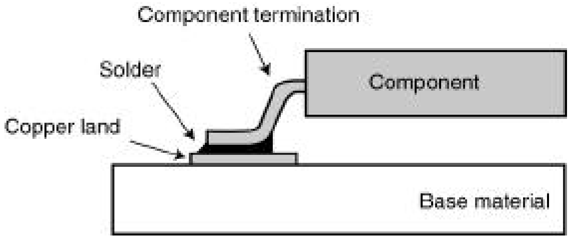


Figure 8

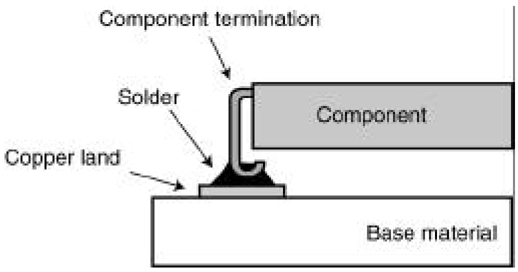
Figure 9 Component package types

The mounting technique for through hole devices has essentially remained the same. Surface mount packaging on the other hand is becoming increasingly complex and new and more complex mounting methods are an integral part of this progression.
The trend in the past has been from TH to SMT as electronic products reduced in size. Surface mount packages such as Metal Electrode Faced (MELF), SO s, QFPs, LCCs, and PLCCs are now used on a regular basis with the well established mounting techniques illustrated in figure 10.





Gull Wing



J Lead

Figure 10 Surface Mounting Techniques

The most recent trend however is from conventional SMT to BGA facilitating further reductions in physical size. BGA is seen as a high speed, high pin count package and it is generally agreed that this option is the alternative to QFP in low-pin-count consumer products and high pin-count computer products. BGAs provides a smaller foot print at a 1mm pitch than the ultimate 0.15mm pitch QFP beyond 600 pins.

The advantages of BGAs are as follows;

- Self alignment during soldering process
- Low assembly cost
- Repairable
- Reduced component cost
- Fewer production reject
- Smaller area board and yet, larger I/O pitch
- Assembled with same SMT equipment as QFP

Tape automated bonding (TAB) technology, illustrated in Figure 11, is another step up from QFP. The device is no longer enclosed in a moulded plastic body, it is covered in a thin glass wafer with it's copper leads bonded to robust polyimide tape. The manufacturer solders the inner ends of the leads to the integrated circuit and the outer ends are then soldered to the PCB during the assembly process. TAB is sometimes confused with chip on board (COB) technology that refers to the process of attaching a bare die to a board with adhesive and bonding each terminal to the board with thin wires prior to encapsulating the complete assembly. This technique is illustrated in Figure 12.

The advantages of using TAB over COB technology are as follows:

- Inner bonds to die terminals are already made
- Wires are tinned copper conductors which are easily soldered to the board
- Soldering of the outer connections to the board lands may be performed in a single operation
- Die is hermetically sealed by the glass layer
- Devices may be fully tested in tape form prior to assembly, and failed devices discarded.

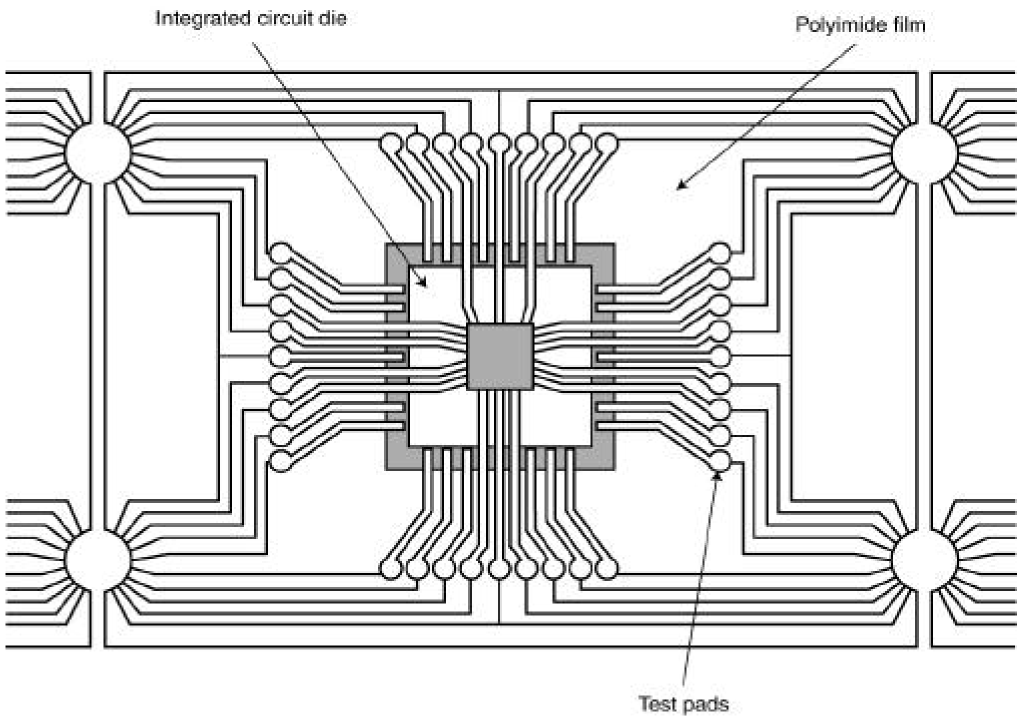


Figure 11 TAB Device

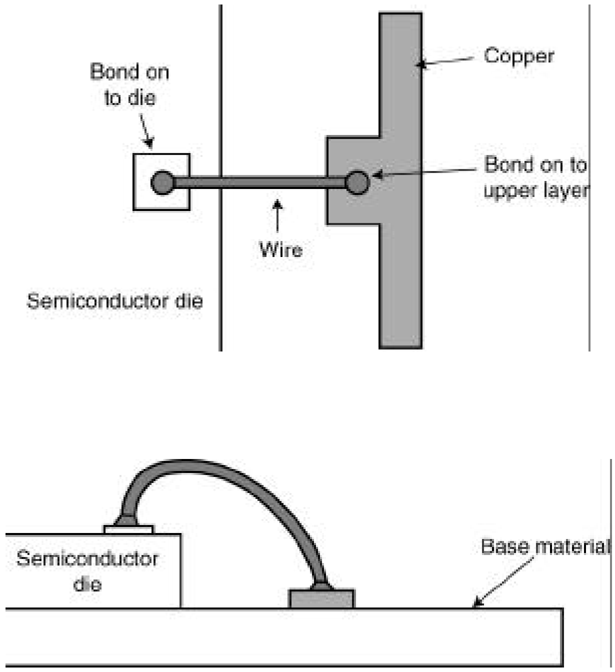


Figure 12 COB Technology

SELECTING THE CORRECT TECHNOLOGY

Selecting the correct technology is a critical part of any project and needs to be addressed throughout the product development period. In relation to this document two aspects need to be considered, namely,

- PCB classification
- Component choice.

The decision on the type of PCB to use is not an easy one, however the number of options is limited. Component choice on the other hand is more difficult since a number of different component technologies may be used on any PCB design.

In most cases there is no simple answer and compromises are invariably required. Significant time is needed to investigate the benefits and limitations of each technology. These limitations must be weighted against the requirements and constraints of the project which can generally be grouped into the following headings and are applicable to the decision making process for PCB and component choice.

• Product functionality………………..(e.g.)	Performance
((AE 372), appendix 1)	Reliability
((AE 25744), appendix 1)	
• Product physical constraints…………(e.g.)	Size
((AE) 127, appendix 1)	Appearance
((AE) 372, appendix 1)	Flexibility
((AE) 25744, appendix 1)	Operating conditions(temperature,
((AE) 25853 (2), appendix 1)	humidity, corrosion and vibration)
• Design for Testing…………………..(e.g.)	Automated/Manual
((AE) 25853 (2), appendix 1)	Location
	Training required
	Investment required
• Design for Assembly………………..(e.g.)	Manual
((AE) 25853 (2), appendix 1)	Automated TH

	Automated SMD
	Combined technologies
	Internal/External
	Training required
	Investment required
• Production ………………………………(e.g.)	Estimated production volumes
	Component availability and cost
	Quality procedures
• Cost……………………………………..(e.g.)	Design
((AE) 127, appendix 1)	Production
((AE) 372, appendix 1)	Test
	Servicing
((AE) 25853 (2), appendix 1)	Upgrades
• Competition…………………………….(e.g.)	Technology used by competitor

The AEs referred to in appendix 1 confirm that there is no hard and fast rule in determining the correct technology. It is usually safe to assume however that regardless of the other driving forces involved in determining the correct technology, product cost will always enter into the equation and usually results in compromises in other areas.

CONSTRUCTION OF A WORK PLAN

The purpose of a work plan is to ensure that aspects such as project management, product research, task allocation and time scheduling are carried out successfully with minimum risk to the project. Project plans necessitating major changes on a continuous basis throughout the project life are unlikely to succeed and those that do end up well behind schedule. There is a saying "measure twice and cut once" the same moral applies in this situation. Spend time evaluating the task in hand before implementing it.

The project plan illustrated in table 1 illustrates all the stages that need to be considered. PCB design is only one of the many tasks that have to be performed, integration into an overall plan is illustrated in AE 24640 and 25845. The PCB development process is fully expanded upon in table 2 with AEs 2064 and 25853 (1) illustrating two different design routes.

Project Management	Establish chain of responsibility. Set out reporting procedures
Project Evaluation	Project objectives and specification. It may be appropriate to carry out a feasibility study to clarify these points.
Technology investigation	A thorough examination of the different technologies available should be carried out. Get external advice if expertise not available.
Choose technologies	Choose a technology that delivers on the project requirements. Remember a compromise is nearly always required.
Training	This largely depends on the level of product support provided by the company. Determine the level of staff training required to implement design, manufacture, test and servicing of the product. It may be that all of these aspects are subcontracted resulting in a minimum of training. (See table 2 for PCB design training requirements)
Design	The design process can be performed by the FU or by a sub contractor. It may also be a combined effort. (See table 2 for a breakdown of the PCB design routes.)
Prototype Assembly	See table 2 for PCB fabrication options
Prototype Testing	The time required to perform this task is often under estimated and aspects such as set up procedures for automated testing, re-works on product hardware/software are only a few of the obstacles which may arise.
Production Batch	Often referred to as pre-production prototypes. Usually a small number of samples are manufactured to ensure that all manufacturing, quality and test procedures are in place. These samples are then used for field trials to establish if the product fully complies with the specification.
Product Documentation	May be generated throughout the design process however final modifications and correct presentation must be implemented at this stage.

Table 1 Workplan

PCB PRODUCTION					
DESIGN BY		PCB MANUFACTURE BY		POPULATE PCB	
FIRST USER	SUBCONTRACT	FIRST USER	SUBCONTRACT	FIRST USER	SUBCONTRACT
Seek expert advice Establish technology Choose CAD software (Appendix 2) Training in PCB design and related technologies Generate schematic, PCB layout and manufacturing files. (Appendix 2)	Select design facility based on previous experience, advice from other users, or reputation. Determine cost for initial contact. Establish technology by discussing project objectives, manufacturing, test and servicing requirements. Obtain quote for design ensuring all files and documentation are included for manufacture (Appendix 2) Place order for work	This process is not normally undertaken by the FU as significant investment is required. It is only viable if very large quantities of PCBs are required over a prolonged period or if the FU intends to seek contract in this area. .	Select subcontractors with quality procedures meeting your needs. Obtain quotes for PCB and choose subcontractor. (Appendix 2) Supply manufacturing files. Check that artworks are returned with PCBs.	Seek advice before purchasing and installing equipment. This is critical for automated assembly processes. Ensure that employees are trained in the appropriate assembly techniques. Adhere to component parts list when purchasing components. Only use component suppliers that can guarantee consistency in quality and supply of components. Adhere to all assembly procedures outlined in the PCB documentation.	Supply board assembly and test details to a number of companies providing this service and request a quote. (Note) It is often easier to carry out PCB testing during the assembly stage. Based on quotes select a number of companies, visit, observe facilities, quality procedures and board testing facilities. Speak to other users of the facilities and make an informed decision. (AE 24714, Appendix 2) (AE 201(1), Appendix 1)

Table 2 PCB Design Path

Minimising Risk factors

Risk may evolve for a number of reasons the most common being;

- Failure to produce an accurate product specification
- Major modifications to project specification during design process (AE 201(2), appendix2)
- Under estimating the extent of the problem in hand
- Wrong choice of software development tools
- Failure by the company to allocate the agreed resources
- Insufficient training
- Failure by the sub-contractor to meet time schedules
- Poor project management

(AE 26002 appendix 1) is an example of a situation where some of these problems were encountered.

Monitoring the project closely maximises the possibility of detecting the above pitfalls at an early stage and enables the implementation of contingency plans.

In order to achieve this degree of monitoring, strict guidelines need to be established at the onset. These may consist of:

- A bench mark to measure progress against (usually a project plan)
- Scheduled project review meetings
- Nominated meeting attendees
- A formal process for documenting outcomes of meetings

The style and implementation of a contingency plan is largely dependent on the project shortcomings. If for example during the project planning stage, questions arise, about possible strains on company design resources, then it would be prudent to draw up a contingency plan. This plan might consist of alternative design resources such as sub-contractors who would be prepared to take on some of the work, or it could consist of provisions to employ an additional engineer to relieve the situation. Another possibility is to consider funding overtime for existing staff.

MANAGMENT of SUBCONTRACTORS

Selecting a Subcontractor

The training document "Managing Design Subcontractors in First Time Use of Microelectronics" outlines the general precautions that need to be adhered to when dealing with subcontractors. Details, which are applicable to PCB design and manufacture are catered for in this document.

Some of the more important points to remember when choosing a sub-contractor are:

- Has the sub-contractor sufficient experience in PCB design and/or manufacture?
- Do you know of any designs the subcontractor worked on, and were they successful?
- Are subcontractors manufacturing facilities geared for prototype runs or mass production runs, it may be that both types are facilitated?
- What type of quality procedures are adhered to?
- Methods of charging
- Delivery times
- Other services provided by subcontractor

Requesting a Quote for PCB Design

It is important to remember that the board designer may not be:

- Familiar with your product
- Familiar with your or your sub-contractors manufacturing and test facilities
- Familiar with the operation of the electronics circuit

The accuracy of any quote and consequently the design depend largely on the relevance and detail of the information imparted to the designer. It is therefore in the interest of the First User to have the following information at hand.

- Board material
- Schematic diagram with component package types specified

- Board details, i.e. silk screen, number of component tracks, surface mount etc.
- Information on circuit segregation
- Details of test and manufacturing requirements
- Details of board mounting requirements
- Current carrying capacity of power and signal tracks
- Identification tag details
- Copper planing requirements
- Urgency of design

<u>Requesting a Quote for PCB Manufacture</u>

It is good practice to request a quote from a number of manufacturers whose quality procedures comply with your manufacturing requirements. The information required for a quote is generally the same and makes the decision of who to proceed with much easier. The following information will normally suffice:

- Board material
- Board Reference and Issue
- Board type, plated through, surface mount, multilayer
- If multilayer, the number of layers
- Board thickness
- Finished copper weight
- Component reference and number of sides
- Solder resist and number of sides
- Edge fingers, number and gold plate thickness
- Profile, breakout required, unusual profile
- Number of drill holes and sizes
- Delivery time.

PCB manufacturers have different pricing policies for prototypes and production quantities. Standard delivery times vary, as do fees for shorter delivery time. These points are worth checking out before hand.

PCB Price Variations

The percentage variation in price are guidelines only. The number of variants are too great to cover all aspects of pricing.

Number of layers

• Single sided to double sided PTH	+30%
• Double sided to multi-layer (4 layer)	+100%
• 4 layer to 6 layer	+30%
• 6 layer to 8 layer	+20%

Number of holes

• Based on minimum of 400 holes	+5% per 100 holes thereafter

Copper Weight

• 0.5 ounce to 1 ounce	+5/10% (depending on quantities)
• 1 ounce to 2 ounce	+5/10% (depending on quantities)

Board Thickness

• 1.6mm to 2.4mm	+10%
• 1.6mm to 3.2mm	+15%

Surface Mount Pads

• Hot air level or immersion gold surface	+10%

Screen

• Solder resist	+10%
• Component Print	+10%

Charges incurred for shorter delivery times;

20 day	(standard delivery)
15 day delivery	+25% of board cost
10 day delivery	+65% of board cost
5 day delivery	+85% of board cost
3 day delivery	+125% of board cost
1 day delivery	+200% of board cost

(Information supplied by Lyncolec Ltd, UK. Tel 0161 707 8777)

An example of the typical costs for a double-sided PTH board with dimensions of 110mm*166mm, number of holes 206, number of drill changes 8, two print layers, two track layers, two masks and a ground plane are as follows;

- Tooling costs 127 Euro
- One board, standard deliver (20 days) 66 Euro
- One hundred boards, standard delivery 11 Euro each

It is important to note that the tooling costs are only incurred once for each new design. In some instances the photoplots and drill information developed as part of the tooling cost are placed on file at the PCB manufacturers. Other manufacturers keep a working copy and send the masters to the customer. This information may be used by other manufacturers in the industry to produce PCBs of a similar quality. As a result it is possible to change supplier without incurring the tooling charges again.

Check manufacturing limitations such as minimum track width, copper plate thickness, board shape and spacing between tracks.

<u>Appendix 1</u>

<u>Application Experiment 127</u>
("Technologie hybride de puissance sur SMI (IMS) pour chargeur de batteries")

The object of this experiment was to decrease the size and cost of a battery charger. Hybrid technology was earmarked as been the most suitable with some reservation expressed about unknowns such as, degree of reliability, reliable and economical production processes and the lack of expertise within the company in this area.

It would have been easier and safer at this stage to back away from the ideal solution and use a less effective technology. In this instance the company used its own resources and the expertise of subcontractors to overcome the obstacles and achieve a 10% reduction in manufacturing costs and a 42% reduction in physical size. As an added bonus system reliability was improved and employees now have the necessary design and process skills needed to enable the design and manufacture of new products using this technology.

<u>Application Experiment 201</u>
("Low Cost Parallel Port Driven Emulator with Trace Option")

(1) The following transcript illustrates the difficulty in selecting an appropriate sub-contractor and also shows how testing of PCB assemblies can be incorporated into the assembly process.

"We subcontracted the making of prototypes to an external manufacturing company. Locating the right sub-contractor here is critical; we made one mistake by using a sub-contractor who was just moving into surface mount assembly. His price was the lowest but it was reflected in the poor quality and long delivery of the product. We have now located a small, local R&D electronic company who have been contracted to produce the first production batch. Issues we looked at included the quality/type of product they produced (use fine pitch surface mount), their willingness to accept the quality risk (they purchase all components and deliver a final tested end product to ourselves). We do a final in-house system test before shipping product out."

(2). This is an example where a project specification was changed during the early stages of the design. The changes resulted in a more competitive product but at significant expense. The decision to modify the specification turned out to be a good one but there were a lot of unexpected delays in the design process. The table outlines some of these delays and illustrates how they can accumulate throughout the design process.

(Taken from AE)

"The work plan was based on two people working in parallel on two separate boards, the emulator and the trace. However, during the early stages of the project, changing market requirements showed a growing need for a range of product options and variants, with design protection provided by special security encryption algorithms. As it was the intention to finally implement the design as an MPGA, it was not possible to design these options and variants as later projects, and it was therefore necessary to revise the project plan. Although these changes initially seemed small, in practice they greatly extended the design time, the security encryption proving particularly difficult to design”

Task	Task Description	Planned person days	Actual person days	Justification
WP1	Management	27	30	Included preparation of detailed project plan, 12 Monthly Reports and Project Reviews, 12 meetings with TTN, various justification reports, creation of Demonstrator (2 drafts), preparation and participation in Final Review.
WP2	Specification	45	60	40 days actual + 20 days additional work on the security encryption specifications.
WP3	Training	2	12	3 days training/evaluation of Xilinx FPGA tools before they were abandoned due to incompatibilty with Windows 95. 2 days training/evaluation of Altera FPGA tools. 2 days training on encryption algorithms with consultant. 5 days evaluation/self training on Protel PCB CAD tools.
WP4	Design	126	157	Additional 20 days required for design of encryption sub system. Additional 11 days due to various minor problems and overall increased complexity of design.
WP5	Evaluation.	125	177	Additional 10 days testing encryption hardware. Additional 3 days spent evaluating Altera Byteblaster interface for FPGA download. Additional 5 days needed to write own FPGA download software. Additional 20 days spent fixing timing problem relating to downloading via PC parallel port. Additional 5 days needed to rectify problems with poor quality first batch of PCBs - poor soldering, wrong components fitted etc. Additional 9 days due to various minor problems and overall increased complexity of design.
	TOTALS	325	436	

Application Experiment 372

(“Surface Mount Technology for an Industrial Computer Board”)

The main driving forces behind this project are;

- To reduce manufacturing costs
- To increase system performance
- To increase functionality
- To reduce physical volume

This company already uses through-hole technology in existing products and examined the following possibilities to achieve their project aims:

- Higher density through-hole(TH)
- Surface mount technology(SMT)
- Multi Chip-Module(MCM) (see training document on MCM technology)

While MCM technology would have achieved most of the project aims outlined, the cost of manufacturing and testing this option was not commercially viable. An SMT design was considered the best option but not without compromise. The company had to take the decision to subcontract PCB production and assembly. They also had to invest in equipment and training to enable them to service and repair their systems. It was anticipated that these resources would also be used on future SMT designs.

Application Experiment 2064
("Transfer Control Computer PCB from THT to SMT")

An example where the PCB design is the main focus of the experiment. The sub-contractor acts in an advisory and training capacity in this situation and the company designs the PCB. Sub-contractors perform testing and assembly and the company implement the modifications. This design route suited the company since they had some previous experience in PCB design but no experience or equipment for assembly and test of surface mount systems.

Benefits of technology changeover taken from AE

Using through hole PCB technology only allows:

- limited technical performance
- modern Controllers are only available for SMT technology
- reduced costs because of automatic production
- Increased reliability

Application Experiment 24640
(“GLUCO SPOT”)

Work plan illustrating the tasks necessary to help ensure successful project completion. A comparison with the work plan in AE 25845 reveals the different levels of detail that are possible. A detailed plan is more likely to reveal problems and pitfalls at an early stage.

WORKPACKAGE 1: MANAGEMENT

1. To define the specifications and to make sure that all WPs outputs will be produced within the established terms.
2. To organise and control project activities.
3. To control the state of the experiment.
4. To disseminate the Fuse action results

WORKPACKAGE 2: TRAINING

1. To acquire technological know-how in designing boards in SMT technology.
2. To acquire technological know-how in selecting SMD components and in product fabrication
3. To acquire technological know-how in development using electronics tools

WORKPACKAGE 3: DESIGN

1. To design an hand held instrument able to read the test strips
2. To define the new instrument functions and new SMT product specifications
3. To design optic block and the project schematic diagram of its validation.

WORKPACKAGE 4: PROTOTYPING

1. To get well working prototypes to check all phases of design

WORKPACKAGE 5: TESTING

1. To check the project
2. To test the optic block and analogue circuit
3. To program and test a perfectly working instrument
4. To define the test specification of the SMD prototype device, in order to certificate the new one

Application Experiment 24714
(“Environment Protection Chemical Management Unit”)

This company’s PCB are populated by a subcontractor who also carries out the appropriate hardware and software test described by the FU.

In order to implement this type of arrangement, test procedures must be largely established before a subcontractor is chosen to populate the PCB. It is then possible to determine the suitability of the subcontractor more accurately.

The test procedures outlined here are taken from the AE and presented here, in order to illustrate the range of testing which can be performed by PCB assembly services.

Initial Hardware Test

The FU will supply blank boards and components to the subcontractor. The boards will be populated in stages with basic tests to ensure integrity at each stage, e.g.

- power and ground correct,
- individual signal paths ok (e.g. each analogue signal conditioning) etc.,
- System clock path ok,
- Removing power (using the purpose built linear PSU) to test battery backup.

Software Control Test

When the board is populated, each individual block will be tested for appropriate control from the microcontroller and each test will build on the previous. This will require specific software routines to be written by Innovative which will be downloaded using a ROMulator (ROM emulator) e.g.

1. Routine to write simple text to the LCD.
2. Routine to write data and then read data from RAM and present the result on the LCD.
3. Routine to read from the keypad and acknowledge keypress on the LCD.
4. Routine to read one analogue input and present the result on the LCD.
5. Routine to read one analogue input as specified by a keypress and present the result on the LCD.
6. Routine to read a digital input as specified by a keypress and present the result on the LCD.
7. Routine to toggle an output as specified by a key press and present the result on the LCD.
8. Routine to set/read the clock and present the result on the LCD.
9. Routine to read / write from RS232.
10. Routine to set the FPGA PWM outputs (verified by an oscilloscope).
11. Routine to trigger the watchdog.
12. Removing power (using the purpose built linear PSU) to test data integrity with battery backup. This process will involve the disabling of certain features such as the watchdog.

Software Integration Test

When each block has been tested using purpose written routines, the final software will be downloaded and tested as far as a bench rig will allow. The bench rig will be

severely limited by the number of conditions that can be simulated (e.g. number of different analogue input voltages that can be supplied simultaneously).
If no (unexpected) problems appear when running the final software, the board will be returned to Innovative for integration testing (with whatever software Innovative consider appropriate) with the scaled down mechanical system.

Application Experiment 25744
("A Miniature Pipeline Inspection Tool Using MCM Technology)

This experiment illustrates well the choice of a particular technology based upon the physical constraints of the product. The decision to use multi chip module (MCM) technology in this application resulted in higher packing densities and enabled testing of smaller diameter pipes. It also increased system functionality and reliability and provided scope for new markets. The expected product volumes for this application are relatively small and would not initially be associated with the expenses incurred in implementing MCM technology, however the dividends paid by the new design obviously justified this decision.

Application Experiment 25845
("Inexpensive DC-High Voltage Supply For Dust Control Systems)

The work plan illustrates the tasks involved in taking a product from concept to production. The PCB design phase is nicely integrated into the plan with aspects such as training in PCB and related technologies included. It should be noted that the FU plans to consult with an adviser before and during the PCB design process. In this way potential problems can be spotted and rectified during the design.

Workplan and rationale

TASK 1: TECHNICAL MANAGEMENT

Subtask 1: Project management and dissemination

Description: Managing and directing the experiment, making monthly reports, end report and articles.

TASK 2: TRAINING

Subtask 1: Controllers

Description: Training in microcontrollers provided by subcontractor.
Training to include:

- available functions in microcontroller

- HW design issues
- SW design issues
- microcontroller selection

Subtask 2: High frequency technology

Description: Subcontractor to provide “HF technology” training to FU in the following areas.

- HF PCB design
- component selection
- EMC
- examples

Subtask 3: PCB design and transformer technology training

Description: Training in PCB design and transformer technology. One days training on the PCB design and PCB design principles. Training and consulting during the actual design.

Subtask 4: Simulation of power supplies

Description: Training in power supplies simulation.
Use of INTUSOFT simulator
Power supply simulations
Examples

TASK 3: SPECIFICATION

Subtask 1: Specification of the high voltage power supply

Description: FU will prepare the power supply specification including operational values for voltage and current, links to automation system and flashover control specification. Subcontractor consults and makes sure that the specification is realizable and that everything needed is specified.

TASK 4: DESIGN

Subtask 1: Design of the high voltage power supply

Description: Selection of components, design of high voltage connections and automation link. FU shall design jointly with subcontractor the power supply. The design phase includes:

- selection of components
- schematic entry
- simulation
- PCB design

- EMC design
- mechanical design

TASK 5: FABRICATION

Subtask 1: Fabrication of the high voltage power supply

Description: Prototype manufacturing.
PCB manufacture
PCB assembly

TASK 6: TESTING AND EVALUATION

Subtask 1: Testing of the prototype

Description: High voltage insulation testing and automation link testing indoor (laboratory).

Subtask 2: Field test

Description: High voltage insulation testing and automation link testing outdoors (on the real circumstances) with the prototype.

Application Experiment 25853
("Design of Low Emission PCB")

(1) Application Experiment 25853 "Design of Low Emission PCB" is a project which is dedicated solely to creating a new PCB design. The workplan and rationale largely consists of information on design stages, deliverables and resources involved in arriving at a completed design. The sub-contractor acts in an advisory capacity and designs the PCB in collaboration with the company. Company resources are then used to build, test and report back to the sub-contractor on the appropriate PCB modification.

(2) In this instance the experiment illustrates how the company wished to alter the product enclosure from a metallic to a plastic structure with the intention of reducing cost and improving appearance. As a result of this change, electrical emissions from the printed circuit would no longer be contained by the plastic enclosure. In order to solve this problem and retain the cost benefits of changing to a plastic enclosure, a PCB layout modification utilising existing PTH technology was adopted. Any other PCB technology would have incurred additional expense in assembly and test and would have been reflected in the product price.

Application Experiment 26002
("Modular Household Receiver")

The delays encountered in this design are largely representative of the pitfalls encountered by FU in the design of PCBs, and are as follows;

- Reluctance of staff to adopt new technology
- Insufficient training in use of CAD tools
- Insufficient knowledge / experience of the technology and the requirements for assembly and test

The company in this instant was able to overcome the problems with the aid of proper management but not without incurring a delay in the development time. A situation may arise where the delays encountered become too costly for a small company to absorb and the project has to be abandoned.

The lessons learnt in this instance are;

- The importance of proper resource allocation and the difficulties of achieving this in a small organisation.
- The importance of consulting with subcontractors throughout the project.

Appendix 2

Purchasing a PCB CAD Package

In order to evaluate if "In House Design" is a viable option, the main issues to be addressed are;

- The complexity of the PCB required for this and future designs
- The resources available
- The additional risk associated with the project success

PCB complexity may be evaluated by determining the following:

- Population density
- Number of track layers required
- Type of component mounting, through hole or surface mount, or both
- Board partitioning required, (i.e.) segregation of digital, analogue and power electronics

If after evaluating the above points and the decision are still for an "In House Design" the First User should now be in a better position to purchase a PCB CAD package.

PCB CAD software should be regarded as a tool and as with any job completion is easier with the right tools. These tools can vary in price from approximately 100ECU to 10000ECU with little or no apparent difference in performance to the inexperienced designer. Unless your PCB design requirements are of an extremely simple nature, such as a single sided board using through hole technology with 20 components or less and requires the simplest of software packages to implement. It is strongly advised that a considerable amount of time be spent investigating the software packages available before purchasing.

Demonstration software is provided for most packages and can usually be downloaded from the appropriate Internet web site. If it is not available there, then it is worth while contacting the supplier to check for availability. The demonstration software generally enables the user to complete a small design in order to evaluate the package, in some cases it is the full blown software version with the "Save " and "Load" file facility disabled.

Local Technology Transfer Nodes (TTNs) may also have CAD packages which you can evaluate on a "try before buy" basis. They will also be able to supply details on the strengths and weakness of different packages.

Every PCB CAD package should consist of the following basic features which are expanded upon in appendix 2;

- Component Libraries
- Schematic Capture
- PCB Layout
- Post-processing stage (Environment for producing manufacturing files and documentation.)

Some of the additional options available on the more advanced packages are:

- Simulation
- Front Annotation
- Back Annotation
- Component Auto Placer
- Component Auto Router
- Error checking
- Multilayer Board Design
- Gerber View Software
- Design rule check facility

Postprocessing facilities normally provided;

- Facility to introduce copper areas on board
- Artwork photoplots (Normally presented to PCB manufacturer in Gerber format)
- Aperture files (Required by PCB manufacturer)
- Drill files (Normally presented to PCB manufacturer in ASCII format)
- Bill of materials
- Mechanical drawings

It is advisable to check with your PCB manufacturer that the manufacturing process supports the file formats generated.

If you are unsure of the PCB complexity involved in future designs, then err on the positive side and purchase a system to deal with the worst case eventuality. Some software is supplied in a modular fashion and provides the user with the option to expand should the situation arise.

In all instances check the following software limitations and ensure that they meet with your requirements.

- Maximum definable board size
- Maximum number of components per design
- Component libraries support, (i.e.) through hole and surface mount technology.
- File output facilities supporting board manufacture

<u>Good PCB Design Practice</u>

Manufacturing is an invaluable source of information especially fabrication, assembly, test and maintenance disciplines. The approval of the relevant design issues by these disciplines will reduce the risk factors involved in the design. Some of the topics that need to be addressed are as follows:

- Technology, such as surface mount, through hole or multilayer
- Testing and fault location requirements
- Maintainable system? this may affect component location, type of components used, PCB shape, mounting hole locations and circuit density.
- Environmental condition such as vibration, shock, temperature and humidity may affect the type of board material and technology used.
- System interface requirements may affect component location, board segregation and mounting holes.
- Board marking requirements may vary for assembly, test and maintenance.

During PCB layout the circuit design should be partitioned into a number of functions and should be confined where possible to specific areas on the board. Circuit functions are usually broken down into power supply, analogue and logic circuits. These functions may be subdivided further depending on parameters such as voltage and current levels and operating frequency. The overall objective is to minimise crosstalk, simplify bare board testing and facilitate troubleshooting diagnostics. In addition the following precautions should be adhered to wherever possible.

- Keep signal and power tracks as short as possible
- Connect a ceramic capacitor of approximately 0.1 f across the Vcc supply to each logic device.
- Ensure that power and ground tracks are kept as wide as possible and run in close proximity to each other.
- Use copper planes wherever possible for power distribution purposes.
- Keep component leads short
- When routing tracks, use forty five degree rather than ninety degree bends
- Keep vias to a minimum
- Ensure that board labelling is clear and concise
- Indicate the different layers on the board design, (i.e.) component track, solder track etc.
- Include an identification label on the board.

Typical Checklist Prior to Producing PCB Manufacturing Files

- Have the guidelines for good design practice been followed?
- Does the circuit layout correspond to the schematic circuit?
- Have all the decisions made during the preliminary board design review been implemented?
- Have all board segregation requirements been carried out properly?
- Are isolation barriers sufficient between high and low voltage circuits?
- Are all track widths, spacing and pad specifications adequate?
- Are all component references, text references and board identification correct?
- Are component layouts correctly dimensioned and is component orientation correct?
- Have the number of vias and drill changes being reduced to a minimum?
- Has the board size been optimised to make the most efficient use of the manufacturing panel size?
- Have all mechanical and dimensioning restraints been implemented?
- Have all the appropriate files been generated for board manufacture?

Various Terms and Abbreviations

Printed Circuit Board Computer Aided Design Package (PCB CAD PACKAGE)

PCB CAD Package.

A software tool that enables the user to capture an electronic circuit in the form of a schematic diagram and/or PCB layout. It also generates manufacturing documentation.

Schematic Capture

PCB CAD facility which provides editing tools to create a schematic representation of the circuit. The process usually begins with the creation of schematic components. Connectivity between the components is defined using a wire route facility that ultimately defines a netlist.

PCB Layout

Tools for creation of component layout and trace routing are provided here. Normally all necessary component and netlist information is automatically transferred from the schematic stage, however PCB layout will normally support the initiation of a capture process if the schematic is not required.

Postprocessing

Supports generation of PCB manufacturing documentation, such as artwork, Numerical Control (NC) drill file, photoplotting and bill of materials.

Front Annotation

Transfer of schematic component and netlist information from schematic capture to PCB layout.

Back Annotation

Transfer of PCB component and netlist information from PCB layout to schematic capture.

Gerber View Facility

Artwork files are generated in Gerber ASCII format for PCB manufacture. The gerber view facility enables the designer to view and consequently check the artworks prior to dispatch.

Netlist

A netlist describes the interconnection of component pins

Single sided board

This type of board has copper tracks on one side only and may be used to support surface mount or through hole technology. It is normally used on very simple designs with a low component count and uncomplicated track layout.

Double sided board

This board is copper coated on both sides and supports surface mount, through hole and plated through hole technology. Boards of this nature are used for slightly more complex designs.

Multilayer

This board is the most sophisticated of the various types. It is formed by laminating several thin printed circuits together and enables the highest component densities of any printed wiring technique. It supports the same mounting technologies as double-sided designs.

Through Hole Technology (TH)

Components are mounted on the PCB with the pins protruding through to the other side. On single sided board the pins need only be soldered on one side, however on double-sided designs it becomes necessary to solder on both sides of the board.

Plated Through Holes (PTH)

This design differs from through hole technology in that all holes are plated through with copper. Soldering need only take place on one side of the board.

Surface Mount Technology(SMT)

Used extensively but not exclusively on multilayer designs this technology greatly reduces product size. SMT components are supplied with pads instead of pins and sit on the PCB rather than protrude through it. The components tend to be much smaller than their through hole counterparts

Tooling Costs

A charge incurred by the customer for each new design. Photoplots and drill manufacturing information are developed during this process.

Silk Screens

Artworks of component reference and solder resist layers

Appendix 3

Barriers perceived by companies in the first use of PCB and SMT

- Uncertainty in selection of technology
- Insufficient facilities on existing CAD packages for new designs
- Lack of knowledge on latest technologies
- Reluctance to adopt new technology
- Quality assurance issues
- Assembly and test of SMT boards.
- Difficult to evaluate the effect on existing manufacture and production techniques.
- Finance
- Lack of resources (more applicable to small companies)
- Company reluctance to take new technology on board

Appendix 4

Application Experiment extracts relevant to PCB and SMT

PCB Abstracts   SMT Abstracts

Appendix 5

Best Practice AEs.

Best Practice AEs

Appendix 6

PowerPoint presentation

See file PCBpre.ppt

(Note) Double click on icons to open.