Babani Electronics Books

A Practical Introduction to Surface Mount Devices

Audio and acoustics

Circuits and projects

Data and reference

Music and MIDI

Test equipment

SW radio and communications

Bill Mooney

A PRACTICAL INTRODUCTION TO SURFACE MOUNT DEVICES

Other Titles of Interest

160	Coil Design and Construction Manual
BP121	How to Design and Make Your Own PCB s
BP285	Beginners Guide to Modern Electronic Components
BP316	Practical Electronic Design Data
BP324	The Art of Soldering
BP392	Electronic Project Building for Beginners
BP396	Electronic Hobbyists Data Book

A PRACTICAL INTRODUCTION TO SURFACE MOUNT DEVICES

by

BILL MOONEY

BERNARD BABANI (publishing) LTD
THE GRAMPIANS
SHEPHERDS BUSH ROAD
LONDON W6 7NF
ENGLAND

Please Note

First Published – October 1996

British Library Cataloguing in Publication Data

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

ISBN 0 85934 411 8

Cover designed by Gregor Arthur

Printed and bound in Great Britain by Cox & Wyman Ltd, Reading

Preface

The biggest change in electronic construction since the invention of the printed circuit board is sweeping the industry. Surface Mount Technology or SMT is now responsible for some 50% of all circuit fabrication. As conventional component insertion machinery wears out it is increasingly being replaced with Surface Mount pick and place machines. SMT gives us lower cost, better reliability and above all miniature consumer goods.

It all started with highly compact portable devices, like organisers, mobile phones and miniature amateur transceivers. SMT is now widespread and CD players, disc drives and PC mother boards are made entirely from surface mount devices. In compact Hi-Fi and many TV receivers, even the power output stages can be fully surface mount. Satellite LNBs rely on surface mount microwave stripline techniques.

SMT is not confined to mass production. Surface Mount Devices (SMDs) are easy to use for hand working. Hobbyist electronic and radio constructors, professionals building prototypes or laboratory one-offs can all use SM construction to advantage. No more drilling, sometimes hundreds of holes, for component leads. Circuit board layout is easier to design and you won't have to cope with (mirror image) tracking on the reverse side of the PCB. Soldering components in place is a much simpler task, without repeatedly turning the PCB over. No more snipping off component pigtails. As a bonus, the working circuit is neat, compact and in keeping with modern commercial practice.

This book is intended as an introduction to the complete beginner who may never have seen a chip component. It will also appeal to those with experience of conventional construction methods who want to learn new skills at an introductory level. Schools covering elementary electronics will find it useful, giving students a feel for contemporary fabrication methods. The reader will see that SMT has the depth and scope of a subject in its own right, rather than an extension of Through-Hole Technology. It is above all a practical guide to real hands-on work with SMDs. Although intended to be read, the impulsive browser like myself will also find it interesting. The difficulties associated with their tiny size are tackled head on. The reader will soon become totally comfortable in this dimension and reluctant to return to the macro world of THT.

Bill Mooney

Contents

Chapter 1

SURFACE MOUNT TECHNOLOGY

1. What is SMT?

In essence, SMT is a method of circuit fabrication where specially designed "chip" components, called Surface Mount Devices (SMDs) are soldered directly onto the Printed Circuit Board, on the same side as the copper tracks. Conventional components have long leads which pass through holes in the PCB to be soldered to the track on the opposite side. Circuit design and construction with SMDs is therefore much simpler than with through-hole components. Surface mount construction requires us to learn interesting new methods of working and novel concepts are possible such as separate circuits on each side of the PCB.

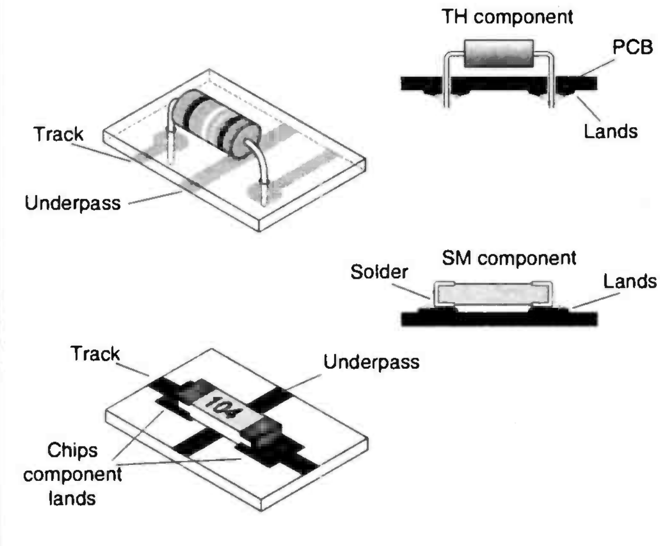


Fig. 1 Through hole and surface mount construction

SMDs are very small compared to through-hole components and SMT is therefore associated with miniature products such as mobile phones, camcorders and compact hand-held amateur radio transceivers. However SMT makes fast, low-cost circuit fabrication possible and is rapidly becoming the norm in all electronic goods; just look inside your new TV set, or CD player. But this technology need not be confined to mass production. For the small scale producer, the professional prototyper and particularly the amateur constructor there are many advantages.

2. The development of SMT

The PCB as we know it came into common usage in the 60's and is certainly the most successful method of circuit fabrication ever developed. Although SMT still makes use of a printed circuit substrate it is claimed that its introduction will have as big an impact as the development of the PCB itself. Large scale integration means that most active circuitry is disappearing into integrated circuit packages. Many connections to the IC are therefore required and it becomes increasingly impractical to drill the required number of holes, often in excess of 100 for one IC, without losing the advantages of integration. Microprocessors, memories, custom designed application specific ICs (a.s.i.c.s) and many other devices at the heart of personal organisers, portable computers and so on will therefore be made in the form of "fine pitch" SMDs with minute pin spacing. These are easily soldered to the PCB using SMT techniques. Glass fibre PCB is tough on small drill bits and their elimination represents a major cost saving apart from the obvious saving of production time. SMT is by no means limited to such large scale integration. Much of the circuitry in day-to-day use will consist of nothing more than a few ICs supported by copious numbers of discrete semiconductors and passive devices all in surface mount packages. In such cases the advantages of SMT are just as obvious; no hole drilling, fast production, better electrical performance and compact size to start with. The less obvious advantages include the cost saving on PCB material. Reduced packaging costs of the populated PCB and even the finished device will need less packaging for distribution and sale. Of course the tiny SMDs themselves will use less encapsulating material. It is with the increasing availability of familiar component types in SM format that the amateur and small scale user gets a look-in. Yes, devices we have grown to know and love like the ubiquitous 741 op amp, the BC109, the 7400 and 4000 series of digital devices, the 4k7 resistor and the 0.1µF capacitor are all available as SMDs. An increasing number of components are manufactured in SM format only. This is the case with several new discrete and small scale devices but it is especially true of the larger highly integrated circuits with fine pitch leadouts. The relative sizes of some common SMDs and THDs are shown in Figure 2. SM circuits therefore have a much reduced volume compared to TH circuits. Both area and profile are reduced, chip resistors being only about 0.6mm high and even the largest chip components rarely stand more than a few millimetres above the board.

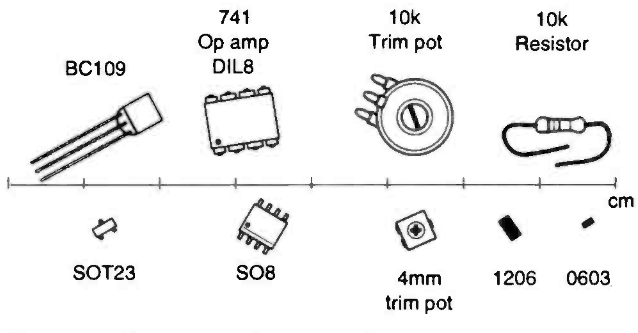


Fig.2 SMDs are much smaller than TH types - illustrations are approximately actual size

3. Industrial SMT Circuit Production

It will be useful to review how commercial circuits are put together with modern SMT production machinery as some of the techniques can be adapted for small scale use. As expected we start with the PCB. This will have been designed on a CAD system and may range in complexity from something simple like a TV remote controller or the innards of a talking doll to a computer motherboard. The more complex PCBs may well have a multilayer construction consisting of up to eight layers of tracking. Multilayering is another means of solving the interconnection problem with highly integrated circuits. Basic CAD systems for home PCs are already widely available and are in use by many hobbyists so no difficulty here. Most CAD systems contain Surface Mount pads or the facility to design them. After etching, cleaning and tinning, the PCB is coated with a solder mask. This (the green colour seen on commercial PCBs) must be photographically processed to expose only the areas where chips are to be soldered. It is a very useful addition as it reduces the chance of solder bridging between fine IC pins for example. The availability of masking film is a problem for the amateur. But thankfully it is mostly unnecessary for hand working.

A small dab of solder paste is now applied to the component lands using a precision dispenser or by screen printing. Often a spot of glue will be applied to a dummy land where the centre of the chip will rest. By the way this can make the salvaging of chip components from commercial PCBs a bit less productive as the glue is quite strong. The solder paste can have sufficient tackiness (green strength) to hold the chip in place obviating the need for an adhesive. Next the SMDs themselves are placed on the pads by a "pick and place" machine. There are many faster methods of placing SMDs on board such as chip shooters and simultaneous methods. This process is one of the marvels of SMT where chips are "picked" from supply tapes and "placed" in the exact position under computer control at impressive speeds as fast as 300,000 chips per hour. Such machines represent the "T" in SMT. For small scale production, manual "pick and place" and simple automatic machines are available. This approach might not appeal to the amateur but for schools, colleges and clubs it is not out of the question.

The populated PCB now continues on down the line and into a "reflow" oven. In a carefully controlled temperature profile the board is slowly raised to the solder paste reflow temperature and then cooled back. All the chips are thus soldered in place at once, an impressive sight. The soldermask restricts the spread of molten solder to the exposed land areas. Any misalignment of the chips is corrected as they can float on the molten solder to take up a central position by surface tension forces. The soldering process can involve very complex technology like hot inert gas, lasers and infrared. The use of solder paste is certainly an option for the amateur. Solder paste or cream is readily available and the reflow temperature is easily reached using one of the many hot air blowers on the market.

Solder pastes contain a high proportion of flux and other organic chemicals and they produce huge quantities of fumes on heating to the reflow temperature, around 180°C. Most formulations leave a lot of residue on the PCB which must be washed away with a suitable cleansing fluid. The message here for the amateur is to make use of some of the proprietary chemicals which are now available for PCB treatment. Even after normal soldering, it is good practice to clean and protect your circuit. A surface mount PCB is a work of art and a thing of beauty to us enthusiasts. This may seem a trivial point but it is worthwhile keeping aesthetics in mind at all stages with the aim of producing a good-looking piece of work. Delicate treatment of SMDs and cleaning is considered worthwhile by the mass producers, for hard economic reasons. You will find that such an approach will likewise increase your success rate.

4. The Place of SMT

Choosing a printed circuit for turning a theoretical schematic diagram into a working device may seem obvious but there are many other solutions. It will therefore be useful to consider the part played by SMT in present day circuit manufacture. The first active device was the thermionic valve and circuits in those early days were made in a style referred to as "breadboard" construction. This was a form of surface mount construction but individual devices and wiring were held in place with nuts and bolts. Since that time the "chassis" with point to point wiring underneath and valves on top, came and went. But the concept of components on one side and wiring (PCB tracks) on the other persisted. The printed circuit board with discrete through-hole components peaked and declined as most of the active circuit elements became integrated on silicon wafers. The technique of wire wrap has found its niche but this is a means of interconnection rather than circuit fabrication. With surface mount fabrication we return to the concept of having components and wiring on the same side of the substrate. Surface mount chips have had a long history in microwave construction and the concept has been ready to go for some time. Presently we have a right old mix of techniques. Among these we find Chip On Board (COB) construction in calculators and birthday cards which play a tune for you. This is an SM technique where the semiconductor die is mounted directly on the PCB by a variety of methods such as Tape Automated Bonding (TAB). This is not for the amateur constructor. As SMT developed it was common to find hybrid PCBs containing a mixture of SMDs and TH devices. About half of all main stream circuits now produced are fully surface mount and the proportion is steadily increasing. Only the capital already invested in TH insertion machinery has slowed down the introduction of SMT. SMT therefore has a very important place in circuit construction and is set to grow.

5. Amateur Construction with SMDs

Surface mount construction has many attractions for the hobbyist and its increasing use indicates that the advantages can be realised in practice. It is apparent that those constructors who try SM, soon develop the required skills and in fact strive to make it their main method of project construction. Their comments mostly refer to the simplicity of designing a PCB layout and the ease of soldering-in chip components without having to repeatedly turn the board over. From their point of view the complexity of using through-hole components seems quite pointless. Soldered-in SMDs are very secure. A certain pride is experienced in using state-of-the-art techniques and of course it is excellent self-training which could be employed in industry.

To gain wide usage by the amateur, SM must be able to compete with many very simple methods of getting a circuit up and running. In fact it has to compete with the unquestioned versatility of through-hole components. These include Veroboard and “ugly construction”. The latter is a type of point to point construction using an unetched piece of PCB as a ground plane. The leads of the TH components are soldered end to end as required and are mostly self supporting in mid air.

This is a valid approach for simple experimental circuits with a couple of active devices. Similar methods can be developed with SMDs. They can be stood on end, on their sides, mounted piggyback and soldered end to end. You can even add a couple of wires making in effect a leaded component for bridging between parts of the circuit. Do's and don'ts will become apparent to the experimenter, for example the ends of chip resistors and capacitors are easily broken off. More important some of the layers in multilayer capacitors can become disconnected causing mysterious changes in circuit performance. Therefore with this lazy approach to our chosen art a little care is needed.

There is no direct equivalent to Veroboard at this time. Although some commercial suppliers offer small test PCBs they are primarily intended for soldering practice with dummy components. But why not make a small "protoboard" PCB for yourself, possibly using some of the techniques discussed later for PCB design and construction. A good example of this is shown in Figure 3. Interwiring links are easily made with fine wire or zero ohm jumpers. The 30 awg insulated Kynar wire used for wirewrap is ideal for longer runs and zero ohm jumpers for short bridges.

At the top end of the construction stakes we are looking at a custom-made PCB for your project. Surface mount has no problem competing here. The initial design is easier and coupled with the time saved in drilling, making a little PCB for your project is a far less daunting task than a through-hole PCB. Once you are set up to produce PCBs, it is surprising how quickly a prototype circuit can be completed. After optimisation by bridging, piggybacking and any other tricks you can think of, a final show model can be run off without drilling a hole or having to face the task of mirror image tracks, endless tossing and turning during soldering and clipping of pigtails.

It would therefore appear that SM components can compete with their cousins, certainly for all low-power functions and up to moderate output powers. We would normally consider SM to be aimed at the signal/low power level. At the power end of SMT, 3 watt (SOT223) transistors have been available for some time. Above a couple of watts we are into nuts and bolts but even so 20 watt SM power devices are receiving wide acceptance. A surface mount version of the LM386 audio power amp running at 0.5 watts will provide adequate output for many small radio projects. This is the subject of an SM project described later.

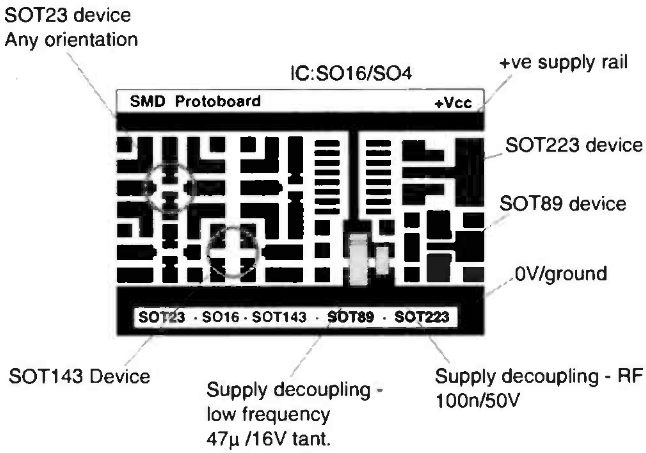


Fig.3 An SMD Protoboard (actual size)

Chapter 2

PASSIVE SURFACE MOUNT COMPONENTS

1. Chip Components

Surface mount components differ from their TH counterparts in many ways. The constructor should be aware of their likes and dislikes and treat them with due respect if they are to give their best performance. This involves the right choice of component in the first place for a particular application. In this chapter we will deal with the component classes separately, paying particular attention to their physical appearance and construction so that you will become familiar with them. This will hopefully create a desire to have a go in practice. SMDs are very much smaller than TH components. This is important in terms of electrical performance and is also reflected in the techniques for using SMDs with which we will deal later. The small size is mostly the result of using less packaging material and there is no requirement for bulky lead fixtures. There is therefore less protection between the active element of the component and the outside world. However higher quality materials are used in their manufacture and SMDs are in fact very tough with failure rates lower than conventional devices. The loss of connecting wires results in lower inductance and better RF performance and this is further aided by the shorter distances between components on the PCB. Conversely the stray capacity between components can be higher. The power rating should be kept in mind as there is less radiating surface to disperse heat. In our review we keep a close eye on the physical dimensions. This is important with chip devices as they have to fit on the PCB footprints. In general there are few problems in translating from TH to SM-based circuits. Let's now review the cast in order of appearance.

2. Chip Resistors

It is fitting that we should start our more detailed description of chip components with the basic building block of all circuits, the humble resistor. Several types of SM resistors have been developed with new additions regularly appearing. Many of these are for military/space and other specialised use and you will never see them except perhaps in surplus equipment. Down here on earth we still have choices to make between types and sizes. Fortunately there is one type which is by far the most popular and you don't need to bother with the rest until you feel the need. I will however give you a resumé of the scope of this art should you need something special. For now, the standard resistor which all us SMers have in mind is the Rectangular, size 1206, metal oxide chip.

These are essentially "thick film" leadless devices consisting of a layer of ruthenium oxide resistive material printed on a tiny slab of alumina as a substrate and protected by a glazed coating. Electrical contact is made at each end with a complex composite of platinum, palladium and silver. A nickel barrier layer is included to prevent leaching of the platinum into the solder. This effect was the reason for poor performance of early components where joints became brittle. Chip resistors are now very rugged and reliable and meet the strict solderability standards required for reflow soldering with zero errors. The oxide film is printed a little under value and then trimmed to increase the resistance to specification. This is achieved by cutting a line across it with a high powered laser or air abrasive whilst monitoring its resistance. You can do this yourself by attacking the edge with some fine emery, not recommended by the purists and it's not always reliable but it's our privilege to bend the rules if we can. The structure of a typical rectangular chip resistor is shown in Figure 4. The wrap-around terminals mean that the chips can be mounted face down.

The full range of E96 values are readily available from 1R to 10M in tolerances of 1%, 2% and 5% in this popular format. You can also get resistors below 1R, above 10M and in tighter tolerances, E192 if you need it. Chip resistors are marked with a three-digit EIA numbering system (Electronic Industries Association 1986) where the first two numbers are the value and the final number is the multiplier, i.e. the number of zeros. A convenient table of resistor codes and values is given in Appendix 2. Zero ohm jumpers, very handy in practice, have the code 000.

The selection of resistor size for hand working is more a matter of convenience than power rating. Several sizes are

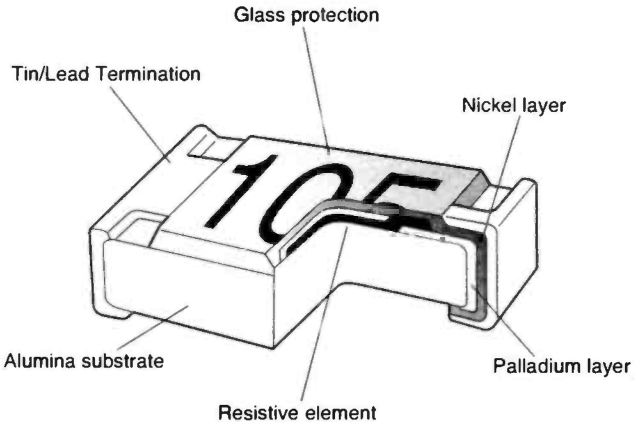


Fig.4 The structure of a thick film chip resistor

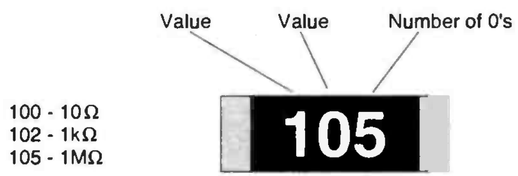


Fig.5 Chip resistor 3-digit code

manufactured, the most common being "1206" and "0805", with "0603" and "0402" coming on stream. The "0402" chips are not much bigger than a full stop and are strictly for the fanatics. The largest common size is the 1206 and must be your first choice for hand soldering. The 0805 size is the industry standard for automatic pick and place and is quite manageable for hand working. Typical dimensions for these sizes, in millimetres, are given in Table 1, but they don't relate easily to the size descriptors. This code results from the chip size in inches, because the USA and Japan are very influential in electronics. The first two digits represent the nominal length in inches for example 0.12in and the second two digits the width for example 0.06in, for the "1206" chip size. Metric based codes can also be found, particularly for devices other than resistors, so watch out for this. For example you might, very rarely, find a 1206 chip resistor described as size 3216.

Table 1
Rectangular Chip Resistor Sizes

	Power Rating		*Dimensions (mm)*	
Size	(watts @ 70°C)	*Length*	*Width*	*Thickness*
0402	0.063	1.00	0.50	0.35
0603	0.063	1.55	0.80	0.45
0805	0.100	2.00	1.25	0.55
1206	0.125	3.10	1.55	0.55

As examples of the more exotic and rare chips, take two newer types or resistor in the 0603 to 1206 size range with special properties.. These are the thin film types and the metal film types.. Thick film resistors are only good for tolerances above 1%. Thin film (nichrome) rectangular chips are high precision devices in the tolerance range 0.1% to 1%. They have good high frequency performance and a wider operating temperature tolerance. Resistor values up to 100k are made at present. The metal film types have tight resistance control with temperature stability of the order of 15ppm, compared to 200ppm for thick film types. E192 values in the range 10R to 22k are made and they have a 4-digit code, 3 value digits plus number of zeros.

Various makes of cylindrical MELF SM resistors have been tagging along right from the beginning of SMT. They are simple adaptions of the leaded types with a circular metal contact at each end. There is not much standardisation of sizes, but the newer types will fit on the same footprints as the 0603 to 1206 devices and have the dubious advantage for the hobbyist of being vertically insertable. This means through a hole in the pcb with one end soldered to a track on each side. The older types have a spiral resistive element giving high inductance. Low cost is the only advantage but this is offset by poor control when hand working. Power ratings ranging from 0.125 to 1 watt are made. Some have special properties though so don't dismiss them.

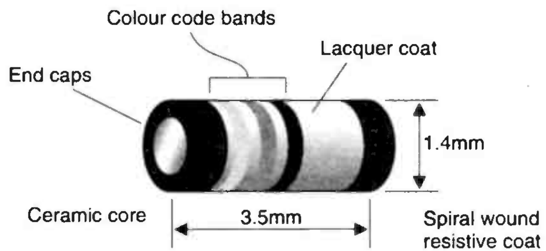


Fig.6 A cylindrical SM resistor

The power rating is a most important characteristic to keep in mind when dealing with resistors. Our chosen standard device, the 1206 chip has a maximum dissipation of 0.125W. This would be derated in the unlikely event of your circuit operating in an ambient temperature above 70°C, for example 50% derating at 90°C. A maximum continuous operating voltage of 200V is also specified by manufacturers for the 1206 chip resistor. Higher power rectangular SM resistors have been around for some time and are now becoming easier to get hold of. These come in the 1210, 2010 and 2512 size with 5% tolerance and use the standard 3-digit code. Details are given in Table 2.

At the top end of the power resistors we have wire wound SM resistors operating at 2 watts with higher short term ratings. These are suitable for current sensing and so values range from 1m (milliohm) to 10R. There is little standardisation

Table 2
Rectangular Power Resistors

Size	*Power* (watts)	*Tolerance* (%)	*Popular Values*	*Length* (mm)	*Width* (mm)
1210	0.40	5	0.25R–3M3	3.20	2.60
2010	0.50	5	0.68R–180k	5.00	2.50
2512	1.00	5	0.68R–100k	6.30	2.15

of packaging but the size descriptors S1 and S2 are typical.

Going back to low power, another useful resistive product is the resistor network. These come in every conceivable configuration starting with 4 or 8 isolated resistors or 15 resistors with one common connection. Look for the latest true SM types as the older stocks are very large and offer no space saving.

NTC and PTC thermistors mostly come in 0805 or 1206 size. They have fast response times and good reliability.

Finally we turn to trim resistors and again we find a range of devices to suit every need. The most common and lowest cost is the fully sealed type 3204. These are 4mm square and have values from 200R to 2M. The point to watch here is the mechanical life in terms of the number of cycles. Type 3204 has a life of only 20 cycles so it is most suitable for set it and leave it situations. This does not mean that the device will fall apart but its value may drift outside specification. Higher quality devices like the type 3315 series have a life of 100 cycles but they are about twice the cost. The smallest trimmers are the open frame types. These can be very compact for example 4mm, 3mm and a tiny 2mm. Again they have a specified life of 20 cycles. High quality 10-turn trimmers are also available with 200 cycle life specification.

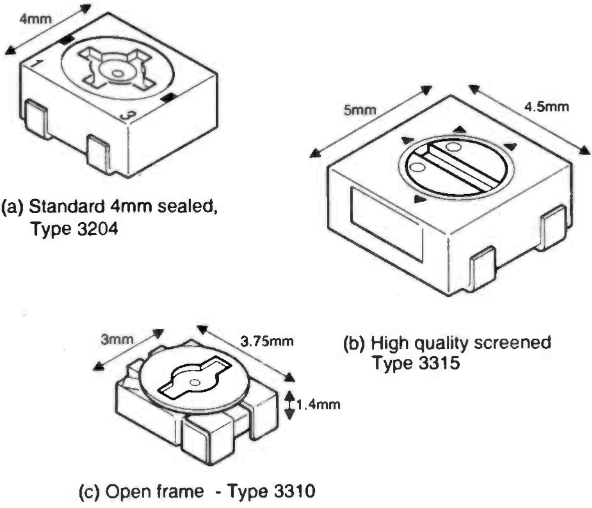


Fig.7 Surface mount trim resistors

3. Chip Capacitors

Chip capacitors come in the same sizes as chip resistors, 1206, 0850 and 0603 being commonly available with 0402s popping up when needed. The end contacts are also nickel barrier for the same reasons. Here we must regrettably part company with the resistors and take on board the peculiarities of the world of chip capacitors. The first difference is the variation of thickness with value. Again there is one type which is indigenous and a cornerstone of SM. This is the monolythic MultiLayer Ceramic Capacitor (MLCC) in 1206 or 0805 package. So let's look at this first. Its structure is shown on Figure 8.

Small wafers of ceramic dielectric material are metallised in the centre region and right to one edge for electrical contact. These wafers are then stacked with their contact edges facing alternate ends. The pile is then fired at a temperatures up to 1400°C to form a monolith. The number (2 to 50) and thickness (about 0.025mm) of the layers determine the capacitance. The

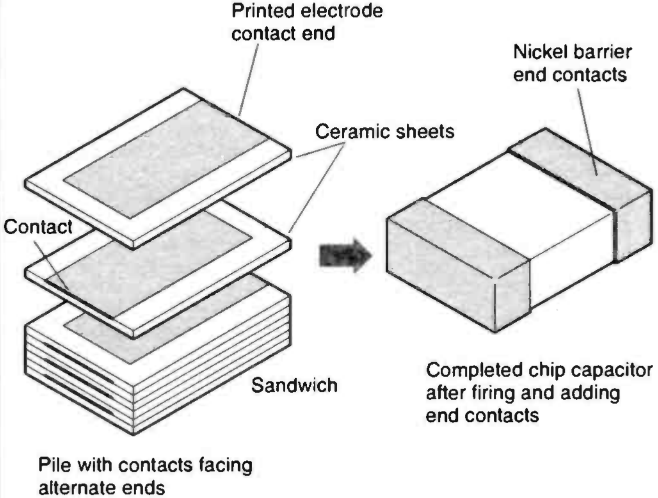


Fig.8 Structure of a multilayer chip capacitor

contacts are platinum-silver thick film composites suitable for direct soldering onto the pcb footprints. These tough little chaps are the basis of a whole range of devices from 1pF to an amazing 4.7µF in a 1206 package. They are generally not marked as the printing process could alter their parameters. The user must take care not to mix them up.

All very neat and simple you might think but the road from 1pF to 4.7µF is not a straight one. To avoid disaster you must get to grips with two vital chip capacitor parameters. The first one is the working voltage. To pack as much capacitance into these little packages the layers must be very close and this determines the working voltage. Most are 50V devices but higher capacitances in any particular size are often 16V. Conversely if you want voltages up to 500V (or 5kV) you will have to go low in capacity and big in body size, obvious really. Chip capacitor dimensions given in Table 3 cover the larger EIA sizes.

Table 3
Dimensions of large ceramic chip capacitors

Size	*Length* (mm)	*Width* (mm)	*Thickness* (Max.)
1206	3.2	1.6	1.6
1210	3.2	2.5	1.8
1812	4.5	3.2	1.8
2220	5.7	5.0	1.8
2225	5.7	6.3	1.8

The second parameter to consider is the type of dielectric. This is a subject of mind bending complexity and depth, but let's try to get a practical working understanding. The most common ceramic chip capacitors are made with one of three distinct types of dielectric which you must get to know. They are referred to as COG, X7R and Z5U. The Z5U material is alternately known as Y5V.

The COG material has the lowest dielectric constant so you tend to get low value caps in this. This ceramic material is referred to as a Class 1 dielectric and is based on rare-earth titanates which incidentally have a blue-grey colour. It is a highly stable, very low loss dielectric. COG (also called NPO) chip capacitors have a capacitance/temperature stability of only 30ppm per °C and age at only 0.00001% per decade. They are ideal for oscillators, filters and other applications where a rock steady capacitance is required. Typical COG chip caps range from 0.5pF to 1nF in our favourite 1206 50V package and are freely available. Or you can get up to 47nF in size up to 2225 working at 500V. These higher values may need a little hunting down in hobbyist circles.

Continuing now with the Class II dielectric, X7R, ceramic chip capacitors. We move down market with this material, it ages more rapidly, has a high capacitance change with temperature (+/–15%) and applied voltage (+15% to –40%) and is much more lossy than the COG material. Its dielectric constant is much higher so we can get high value capacitors in smaller packages. They are fine for decoupling, inter-stage coupling and possibly less critical filter applications. In our preferred 1206 package values range from 100pF/50V to 100nF/50V. Again you can get higher values in bigger packages up to 1.5µF/50V in 2225.

The lowest quality dielectric is Z5U/Y5V, which is brown in colour. This material will give you a 4.7µF/16V capacitor in a 1206 chip. Its tolerance is +80% to –50%. As to temperature coefficient, ageing and dielectric loss, don't ask. From a practical point of view these chips are mechanically more delicate and will fracture and crack easily with mechanical or temperature shock. Great for coupling and decoupling with the lowest readily available values being around 100nF/16V in 1206.

When large values of capacitance are required, tantalum solid electrolyte capacitors are preferred. Tantalum capacitors are more efficient at audio frequencies than aluminium electrolytic so that in many applications a 22µF tant will replace a 47µF aluminium. There is no overall standardisation of sizes but the moulded plastic case types are most popular and have lettered case sizes given in Table 4.

Table 4
Tantalum capacitor case sizes

Case	*Dimensions* (mm) *Length*	*Width*	*Height*
A	3.2	1.6	1.8
B	3.4	2.6	1.9
C	5.8	3.2	2.5
D	5.8	4.5	3.1
E	7.3	4.3	2.8

Values available range from 220μF/2.5V to 0.047μF/35V and up to 6.8μF/50V in the above case sizes. Tolerance is +/–20% in most instances. There is considerable crossover in case sizes for example a 47μF/10V can appear in case size C, D2 or D, so that you need to watch this if you have a tight spot on the PCB. In all instances it is a good idea to keep well within the rated voltage with tantalums and don't forget, never apply reverse voltage. Typical construction is shown in Figure 9. Newer types of tiny tantalum capacitors are appearing in 0805 compatible case size with values ranging from 0.47μF/16V to 2.2μF/2.5V, but they are a little rare.

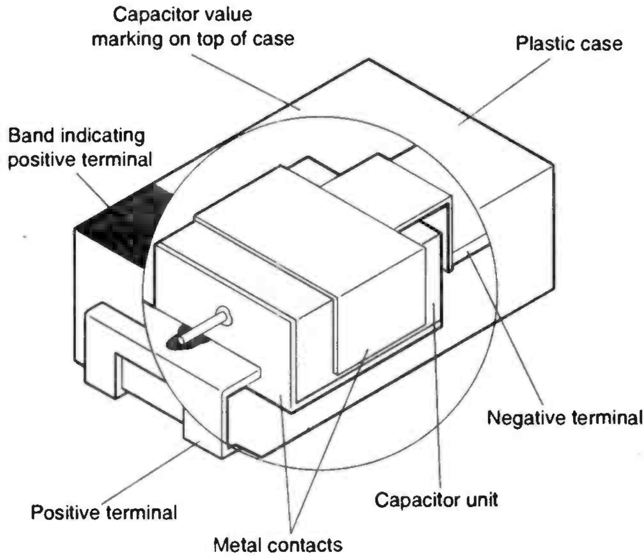


Fig.9 Tantalum capacitor construction

The aluminium electrolytic capacitor has stayed in the battle with tants and has evolved well to meet SM needs. Case sizes are far from standardised yet, but two types are prominent, *viz* vertical and horizontal mounting, Figure 10. Early

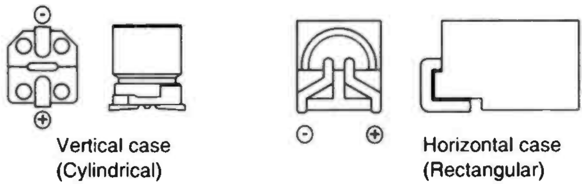


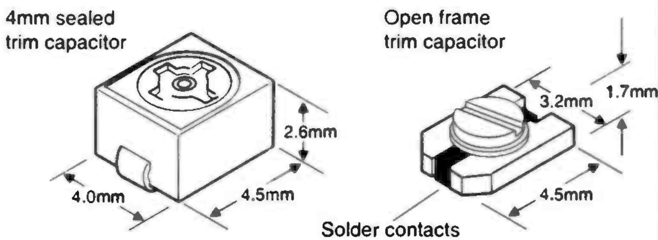
Fig. 10 Aluminium electrolytic case types

aluminium electrolytics, with liquid dielectrics, were problematic at wave soldering temperatures. Present aluminium electrolytics have the edge on tantalums for reliability and very high capacities, up to 1000μF. Very useful non-polarised versions are also available.

Newer types of capacitors are appearing in surface mount formats. These include aluminium special polymer dielectrics in the range up to 33μF with very good high frequency performance over 1MHz. Polyphenylene sulphide film (PPS) capacitors range from 100pF to 0.1μF/50V. They have excellent stability and 2% tolerance, making them suitable for filters, coupling and decoupling. Polyester film capacitors have somewhat larger formats for their capacitance, typically ranging from 0.01μF/50V in 1812 to 0.47μF/25V in 2824. These are useful up to a few MHz. Excellent RF performance is guaranteed from surface mount versions of silvered mica capacitors. The RF performance of even the best quality COG ceramic chip caps falls off in the VHF region and above 100MHz you should not take anything for granted. Above this region the resistive impedance and inductance increase dramatically. The self resonant frequency can be found from manufacturers data. For such applications the traditional porcelain chip capacitors are widely used. They are expensive but work very well indeed up to tens of GHzs. Porcelain chip caps were around before SM as we know it and the sizes adhere to different standards. Thin film, high Q, ceramic chip capacitors are manufactured in standard surface mount sizes and are designed to perform in excess of 45GHz. They are just the thing for VHF, UHF and microwave applications such as 400 to 900MHz TX/RX, and

12GHz satellite work. Values vary from 0.1pF +/–0.05pF up to sensible values. There are several makes of high Q chip capacitors, check with manufacturers' data for critical applications. A last word on oddities must include capacitor arrays in surface mount. These 2 or 4 capacitor blocks in 0805 or 1206 save mounting time and space. Values vary from about 1nF to 680nF in all dielectrics.

Finally, for RF applications, a vital component is the trim capacitor. Surface mount trim caps are made in a variety of sizes and all are easily hand solderable. The lowest cost, most common, devices are the 4mm sealed types with a ceramic element. These are conveniently colour coded for value. They are suitable for commercial wave soldering. Ultra thin, 3mm, open devices are made in a greater variety of sizes and, in my experience, seem more reliable mechanically than the 4mm sealed types. These are suitable for commercial infrared or vapour phase reflow soldering. In both formats devices are manufactured with Cmax of, from 3pF to 50pF. If you want to make something really small the 2mm ultra miniature trim caps are for you. Dimensions and construction details are shown in Figure 11. Trim caps have Q values around 400, a working voltage of 100V and temperature coefficients of 300 to 500ppm per °C.



4mm Colour codes

Brown	3pF	Green	30pF
Blue	6pF	Yellow	40pF
White	10pF	Black	50pF
Red	20pF		

Fig.11 Two types of SM trim capacitor

4. Chip Inductors

Surface mount component manufacturers certainly see the inductor as an important circuit element for there are many types available in small useful incremental values. The oldest, most common type of chip inductor is the epoxy encapsulated ferrite core wire wound device. This general purpose version comes in several variants and sizes, depending on the manufacturer. Some manufacturers are producing them in compliance with 1008, 1210,1812 and 2220 sizes now but they may be a little scarce. Typical values in whatever format range from 0.1µH (+/–20%) to 1000µH(+/–10%). Q values range from 20 to 50 and higher values can have DC resistances as high as 40 ohms. Passing a direct current through a ferrite cored inductor has no effect on its inductance until the core saturates. At this point its inductance drops like a stone, a typical maximum current would be 50mA for a 220µH chip. They are perfectly adequate for most purposes in oscillators and filters. Just as with capacitors and any other component for that matter, there is no such thing as a pure inductor. Non encapsulated chip inductors will also be found but they are less common because pick and place machines can't cope with the non smooth surface. There can be a considerable stray field from this general purpose chip inductor and magnetically shielded types perform better if needed.

The most exciting chip inductor now appearing in commercial catalogues is the multilayer monolithic type. These are fired just like the ceramic chip capacitors, on which you are

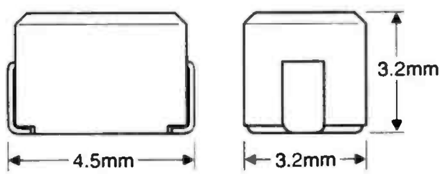


Fig. 12 Structure of a moulded chip capacitor

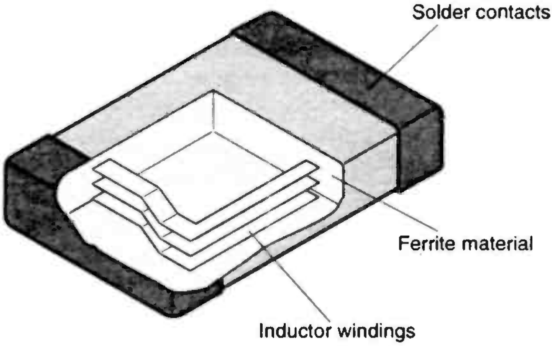


Fig. 13 Structure of a multilayer chip inductor

now an expert. These little beauties come in 0603, 0805, and 1206 case sizes. Advantages, apart from their standard sizes are: no external field, no cross coupling between inductors, high reliability and they are robust to any type of soldering. Values range from 0.047µH to 220µH. High Q multilayer inductor devices for use up to GHz go down to 2.2nH.

A more accessible range of thin film surface mount inductors for use in the 0.5GHz to 2GHz range are available in 0805 packages. These are aimed at cellphone frequencies. They have Q values around 50 and tight tolerances of better than +/–5% and are fine for manual soldering. Values range from 2.7nH to 15nH.

Variable inductors for IF transformers, RF amplifiers and oscillators have been available for some time. The most popular are the Toko 5CD types ranging from 1µH to 680µH. These have Q values of about 50 and their inductance can be varied by +/–10%. Tapped versions and secondary windings are possible. Although a huge range of configurations are made, availability in amateur circles is patchy at present. The structure is shown in Figure 14.

Let's round off the treatment of passive devices with ceramic filters and the like. Surface mount ceramic filters for

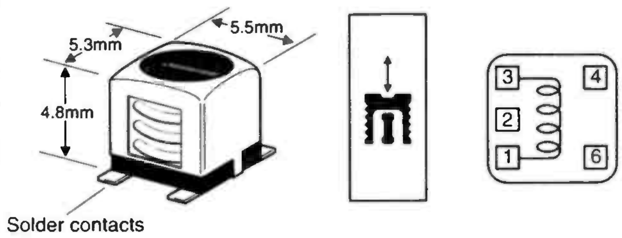


Fig.14 Miniature variable inductors

10.7MHz and 455kHz have the structure shown in Figure 15. Both are dual element types and bandwidth can be for broadcast or communications. Surface mount quartz crystals are presently made in one or two non-standard sizes. All these devices can be found in commercial products in plenty, but small quantities need to be hunted down.

5. Supporting Cast

Outside the mainstream functional components there are a host of support devices, electromechanical items, which are a great help in keeping much action on the PCB as possible. To our delight, fuses, relays, dil switches, connectors and so on are all beginning to appear in commercial SM suppliers' catalogues. There is no doubt that these will appear in the enthusiast domain in increasing numbers. Opto isolators and CMOS relays have one foot in this camp and are readily available in SM.

Putting a fuse on the PCB is not expensive but it could save money especially on the more complex PCBs. Even a fast acting fuse may not save the unfortunate load-carrying device but it may save the PCB from baking and secondary failure. Or it may save your power supply and possibly a fire. There is little standardisation in surface mountable fuses as it is an area of continuing development. Current ratings range from typically 62mA to 15A. The lower ratings are of most interest for typical gadget applications. They are available in very fast acting, fast acting or slow blow. The earlier types were cylindrical and a

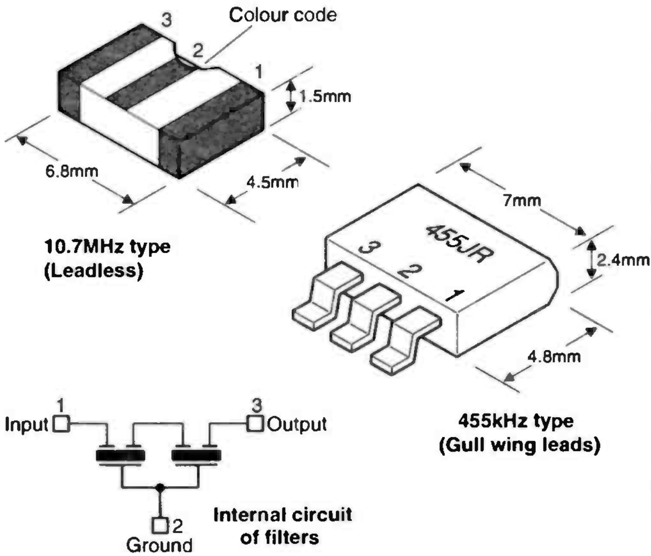


Fig. 15 Surface mount IF filters

little more difficult to handle but block types are becoming more popular and don't roll about when you are trying to solder them in place. A series of thin film fuses in 1206 size is now available from 200mA to 2A.

Chapter 3

SURFACE MOUNT SEMICONDUCTORS

1. Availability

As with the passive components the commercial producers offer a wonderful selection of semiconductor devices from the humble diode to large scale ICs. As enthusiasts our choice is limited only by our ingenuity in acquiring what we need. Practically anything available in TH is made in SM also, but it will be prudent to find a supplier before rushing ahead with a particular design. A component may be listed in a manufacturer's catalogue and may be widely used but a distributor will often need to invest in a minimum order of 100s or 1000s so that we can buy one to play with. More and more ICs are being made in SM only and this will inevitably improve availability of SM types. In this chapter we will deal with discrete and integrated devices with the emphasis on their packages so the reader can become familiar with them. More detail on electrical characteristics is always available in manufacturers' data sheets. This is particularly important when power ratings are likely to be exceeded for example when transposing output circuits from TH to SM. Only the specialist will be concerned with the small differences in electrical performance. Overall there is good agreement between lead numbering of TH and SM devices, particularly the smaller ICs.

2. Diodes Diodes Diodes

It is astounding that the simple diode still plays such an important part in electronics. Its versatility seems endless and in SM you will find PIN diodes, signal diodes, switching diodes, rectifiers, full wave rectifiers, LEDs of all colours with clear or diffuse lenses, variable capacitance diodes, zener diodes, voltage reference diodes, transient suppressors, hot carrier diodes (Schottky) and of course lasers. From a practical point of view we are saved total confusion by the fact that diodes have only two connections. But apart from a couple of standard packages there is less than perfect standardisation.

The oldest SM diodes came in one of two cylindrical MELF (Metal ELectrode Face) packages. This package is totally sealed in glass and is therefore very reliable. They are in fact made by the same route as their TH equivalents. The only drawback is the difficulty in handling both by pick and place machines and hand soldering since they tend to roll off the PCB lands. A small slot in the PCB land will help prevent rolling. The 1N4000 series of rectifier diodes come in this MELF package. Some small signal and zener diodes will also be found in the mini MELF package (Fig. 16).

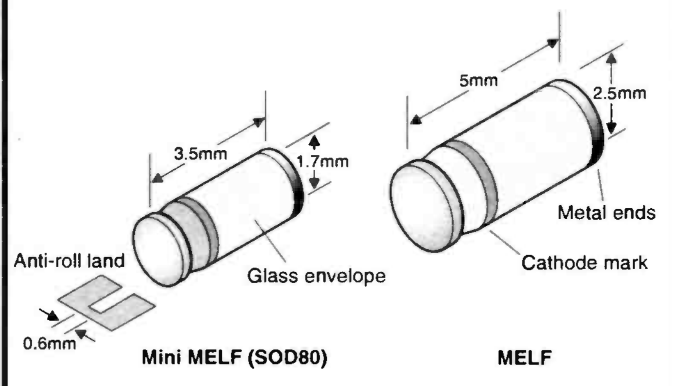


Fig.16 Structure of MELF and miniMELF diodes

Various forms of anti-roll package are now far more popular for power diodes. Half-wave and full-wave rectifiers are made in a range of surface-mount flat packages. Fast recovery and other higher performance rectifiers are also found in such packages. Three typical formats are shown in Figure 17.

By far the most popular package for all small signal diodes is the SOT23. Versions of most types of diode will be found in SOT23 including switching, PUTs, SCRs, voltage reference, varicap, photodiodes and LED's. The connections are mostly as shown in Figure 18, but all possible connections are manufactured so a quick check with a meter might save a lot of problems if you are not certain of pedigree. Dual diodes are

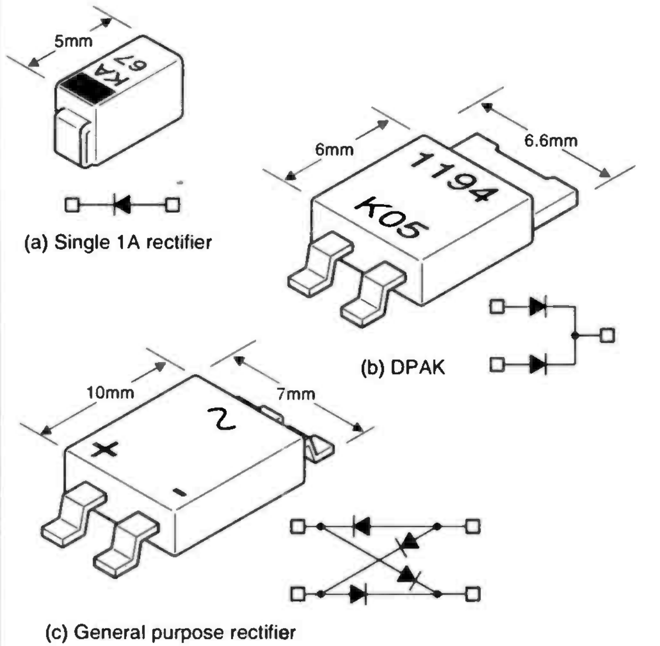


Fig.17 Typical rectifier diode packages

made in the SOT23 if one electrode is common. If two separate diodes are required the SOT143 package is used. Both these packages are ideal for handworking.

Some less common packages include six terminals for switching diode arrays. Some half-wave rectifiers come in the SOT89 package which we will cover later. A useful range of leds and photodetectors are made in the very tiny CR10 series of packages,which are stackable and measure only 3mm by 1mm.

3. Transistors

Transistors are manufactured in a beautifully simple set of packages and they have very good standardisation of pinouts. These packages are SOT23, SOT143, SOT89 and SOT223 and

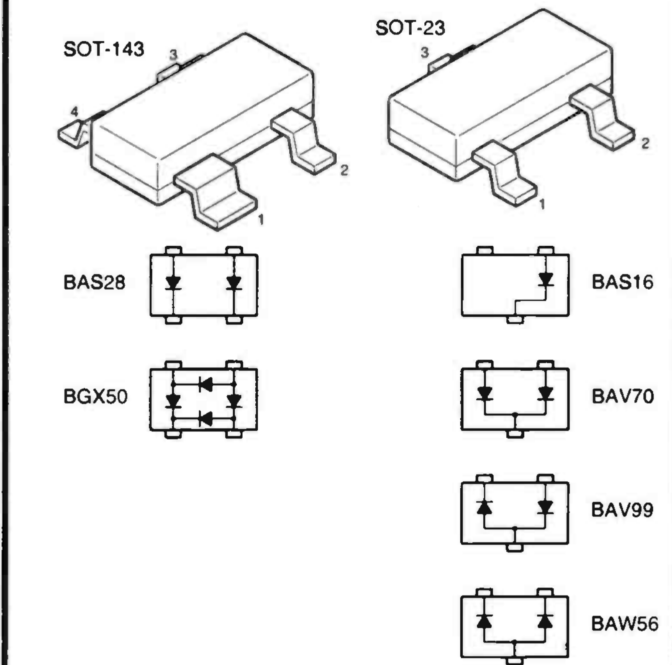


Fig. 18 Diode configurations

there will be a few oddities of course. The absolute universal outline is the SOT23. This is small but after a little practice in hand working it is quite manageable. Even for those who want to go for ultimate miniaturisation the SOT23 will not be the limiting factor. It is an excellent optimum design in all respects, electrical properties, size, and power dissipation. All the familiar small signal devices will have equivalents in SOT23 or as near as makes no difference. The SOT23 package has a power dissipation capability of up to 200mW. The essential, typical, measurements are given in Figure 19. The SOT23 package has been around for some time and when we focus in closer on it we find some slight variations in its dimensions by different manufacturers. For hand working these are all interchangeable but it explains some visual differences which you may find. The TO-236 and SC-59 are both referred to as SOT23 but their dimensions are minutely different.

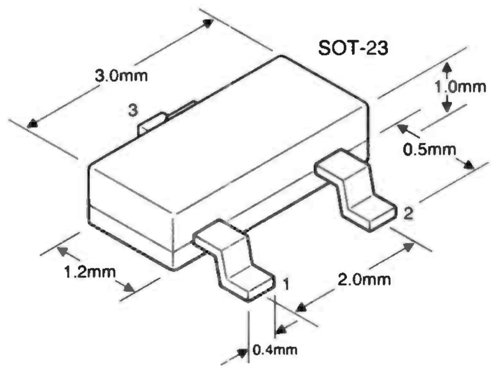


Fig. 19 The SOT23 transistor package

Common devices from the TH world are sometimes given the same or similar type numbers in SM but in most cases special type numbers are used. Most devices are marked with a manufacturer's code rather than the type number for example the BSR58 FET is marked "M6". It is a good policy to record these for reference as you come across new ones, it will help if you have a mix-up. Typical connections to some common devices found in this package are given in Figure 20. Connections cannot be taken for granted so check with manufacturers' data sheets, suppliers' notes and so on. In particular the base and emitter can be reversed, this is referred to as an "R" joggle and the type number will have an 'R' subscript. As an example the BCF32R in Figure 20 will have the code D77 as opposed to the standard BCF32, marked D7. Unfortunately different manufacturers can have different codes, those above refer to Philips

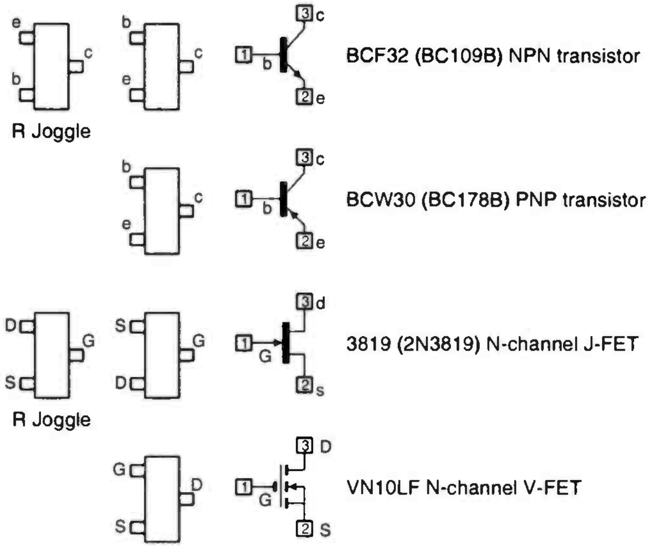


Fig.20 Some examples of SOT23 devices

devices. On the subject of pin numbering, there is some scope for con-fusion. The convention is to count anticlockwise when looking at the top of the device. Some manufacturers show a bottom view of the SOT23 package which seems pointless for an SM component.

Where a fourth output connection is needed, the SOT143 package is used. Again the Japanese SC-59 version and the TO-236/5 versions have marginal dimensional differences which are of no consequence to the hand user. Like the SOT23 variants the higher standoff versions (0.1mm) allow pcb cleaning fluids to penetrate under the casing to remove flux residues. An excellent and well known example of a SOT143 device is the dual-gate MOSFET BF998. Newer devices include the GaAs FETs type CF730 and CFY39, the CGY50 GaAs MMIC and the BCV61B/BCV62B NPN/PNP matched transistor pairs. Power dissipation up to 400mW can be handled with the SOT143 package. Dimensions are given in Figure 21.

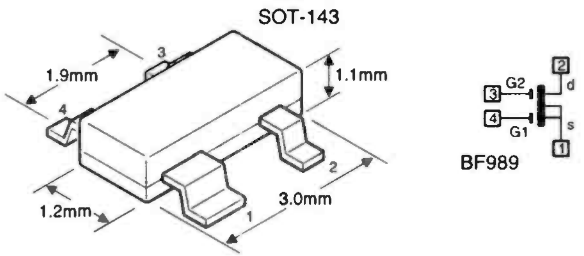


Fig.21 The SOT143 semiconductor package

Surface mount power devices start with the very well standardised SOT89 package. It will dissipate typically 1 watt but higher values have been quoted for recent devices. The heatsinking ability of the PCB, the area connected to the collector, is all important for maximising power dissipation. Dimensions for SOT89 package are less varied than the SOT23 or the SOT143. Examples of popular devices in SOT89 are the BCX54(NPN) and the BCX51(PNP) which are SM versions of the very useful BD135/BD136 complementary pair. Useful RF power amps are the BSR40, BFQ19 and BFQ17. The BST80 is an N-channel enhancement mode DMOS V-FET.

Two new power packages are now beginning to supplement the widely used SOT89 package. First, the SOT223 package fills the 1.5 to 2 watt range with bipolar power devices like FZT653(NPN) and the complementary FZT753 and the usual array of V-FETs, Darlingtons and so on. This is a very useful package and is a little more forgiving for hand working than the SOT89. The power rating assumes a 1in^2 PCB for Rth(j-amb) of 60°C/watt and a 1.5in^2 PCB will have a sinking capacity of 50°C per watt. The sight of SOT89s over-dissipating and going for a swim in solder is not unknown. The SOT223 package gives the experimenter a bit more headroom in this respect. The second package is in fact older than the SOT223 but has been adopted as the main big power package. Originally it was described as a 2 watt device but it is now fully developed and devices capable of handling 20 watts can be found. There

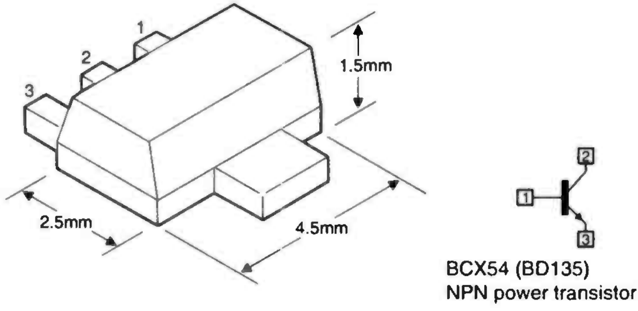


Fig.22 The SOT89 SM power package

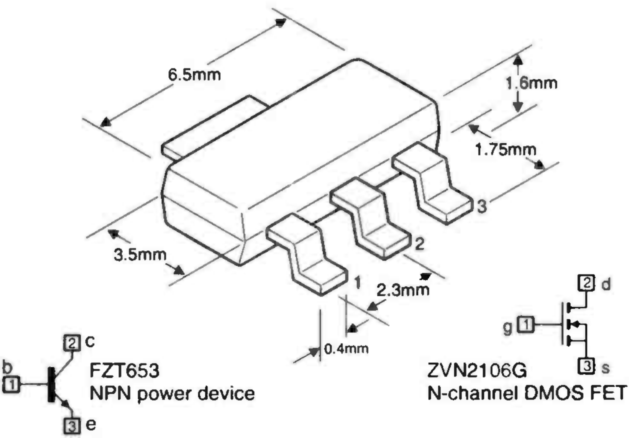


Fig.23 Details of the SOT223 package

is considerable variation in D-PAK dimensions. This lifts SM well out of the small signal domain, what more could you need. To end with, how about the PSD package, the 2SK2042 N-channel MOSFET in PSD will take 450V on the drain and dissipate 35 watts.

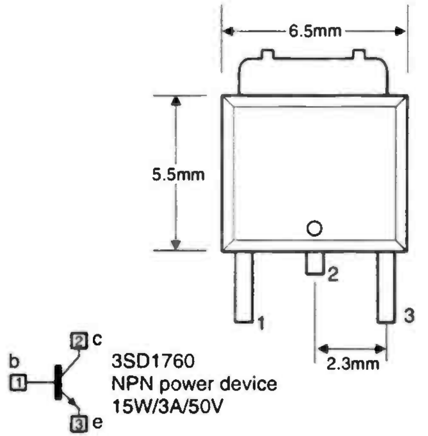


Fig.24 Details of the DPAK power package

Some new transistor outlines are worthy of mention, in particular the UMT (Ultra Mold Transistor) which is about two-thirds the size of the SOT23 package at 2mm by 1.3mm. But the ultimate is the EM3 outline measuring only 1.6mm by 0.8mm or 1.6mm by 1.6mm including the gull wing leads. A number of special-function transistor arrays in multi-lead SOT23-like packages include digital transistor arrays.

4. Integrated circuits

Surface mount IC packages have settled down to a manageable number of types. Even so the constructor will come across odd-ball packages and variants on what we can regard as the industry standards. The simplest packages are the SOIC range, Small Outline Integrated Circuits. These are made with 8 to 28 pins, depending on the complexity. The pins themselves have evolved for SM application. Three types of lead have received varying degrees of acceptance during the development of ICs generally. By far the most common lead type is the gull-wing with many years of reliable service (Fig. 25).

The gull-wing lead, bends downward from the IC body to the PCB level and then outwards forming a lap joint. It is fine

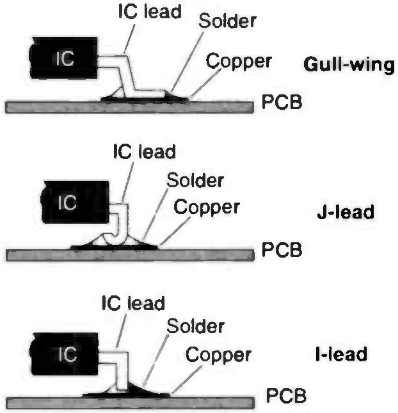


Fig.25 SMT IC leadout configurations

for handworking, with its extended solderable area. The visibility is very good so that the soldering can be readily inspected and voltage measurement is easier with a suitable probe. A disadvantage of this lead is the larger device footprint required to accommodate the extended leads. Coplanarity errors can be a problem in automatic soldering but is of little importance for hand working unless the leads are badly bent out of true. The "J" lead takes up less space but the IC packages tend to be bigger to accommodate the lead itself. Visibility and inspection are difficult. The tendency is for J leads to be used for PLCC chips but the use of finer gull wings seems to be the way things are going. The "I" lead is persisting in the race for supremacy but only just. It has good strength and is quite compact but the leads must be sheared to the same length and the bare metal of the shear must be tinned otherwise it would not meet solderability standards. The gull-wing is therefore our preferred configuration and in fact most devices used by the amateur will be of this type.

Our favourite linear and digital devices, the building blocks for small projects, are supplied in SOIC packages. This includes the standard digital devices like the 4000 and 7400

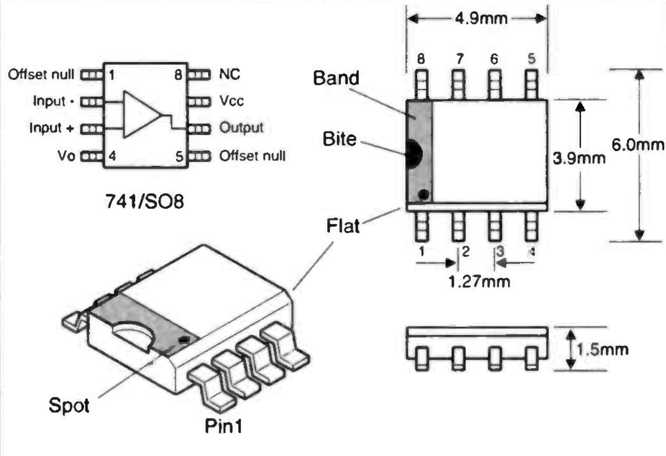
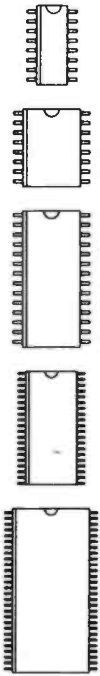


Fig.26 Surface mount version of the 741 op amp

series and a whole array of computer support devices. Also found in this package are the 555 and similar timers, the NE564D phase lock loop, all types of op amp, DAC/ADC, etc. An important variant is the SOIC Wide package where the die won't fit in the standard SOIC. This has the same 1.27mm pin spacing but a wider body and type numbers sometimes contain the /W (wide) or /L (large) subscript. The popular 741 op amp is housed in an SO8 package, Figure 26. Our first objective is to find pin 1 and there are plenty of clues. Looking down on the IC you will note one edge has a flat, putting this on the left places pin 1 on the top left. Pin 1 is sometimes indicated by a small circular indentation on the top left. The pin 1 end is very often indicated by a small semicircular bite at that end. If all this fails a white stripe may be found across the pin 1 end. It is common to find all these clues on one chip.

The smaller packages, up to SO16 can be standard or wide but the larger packages, SO20, SO24, SO28 and SO32 are all wide packages. SO32 and below have pin spacing of 1.27mm but the 40 and 56 pin VSO (very small outline) devices have a narrower pitch of 0.76 or 0.75mm and different package dimensions.



(a) Small Outline, SO16

(b) Small Outline wide, SO16W

(c) Small Outline extra wide, SO28XW

(d) Very Small Outline, VSO40

(e) Very Small Outline, VSO56

Fig.27 Some examples of surface mount IC packages

More complex, large-scale integrated devices are supplied in a variety of PLCCs, Plastic Leaded Chip Carriers. Typical devices which are well suited to this format are the AD720JP RGB to PAL encoder IC in a PLCC28 or the μPD70325L 16 bit microprocessor in a PLCC84 package. The PLCCs have 20, 28, 32, 44, 52, 68 or 84 pins and J leads which can be soldered direct or inserted into PLCC sockets. The sockets are useful where a chip may be upgraded or for burn-in tests. In these packages pin 1 is identified by a small circle as indicated in Figure 28, and the pitch is 1.27mm.

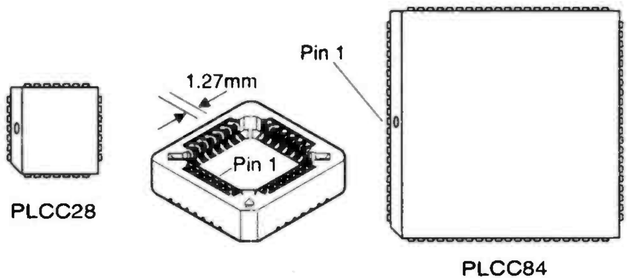


Fig.28 Plastic Leaded Chip Carrier IC and socket

The PLCC is quite bulky, particularly when a socket is used. Conversion PCBs are available for SM to TH IC sockets if a socketed device is required. Much more compact circuitry is possible with the newer Quad Flat Pack ICs. QFPs can have very fine pitch, 1.25mm down to below 0.5mm. They are not so robust in handling but are very reliable once soldered in place. Such devices have gull-wing leads and are workable by hand with a little care. Service engineers have to cope with them and reworking has proved very practical. The usage of QFP is now widespread, a typical complex device is the DP83932 network interface controller in QFP132 package or the Z84C9008VSC Z80 CPU and support circuitry in a QFP100 package.

SMT is the target for continuing development and the odd new package can therefore be expected to intrude just when we get to grips with the existing standards. An example of this at the simple end is a whole set of single gate devices with type numbers in the BU4 series. The range includes NAND, AND, NOR, OR, INV, SCHMITT, and an analogue switch in a 5-lead SOT23 package. These are useful for supplementing existing logic ICs or for replacing discrete functions. Also, being very small the SOT23 gates can be dotted around the PCB just where they are needed saving the space and complexity of routing to a remote SOIC gate array package. These must be attractive to the experimenter as availability improves. In power

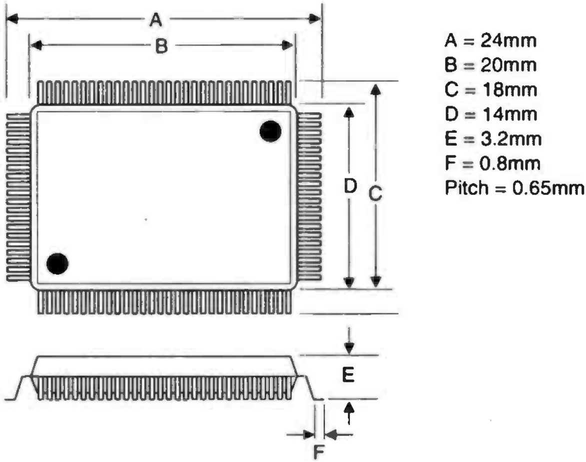


Fig.29 Quad Flat Pack, QFP package

devices there is a new DD package which seems to be mainly used for switching voltage regulators. This is a package similar to the DPAK, about 1cm square and with 3 to 8 leads.

Chapter 4

PCB LAYOUT DEVELOPMENT

1. Getting started

In this chapter we deal with techniques for development of circuit layout. To be successful at this, it is essential to have a feel for chip components. We therefore get round to meeting SMDs for real using a couple of simple projects. By now you should have a good idea of the range of SMDs available and outline descriptors like SOT23 and 1206 should be familiar to you. One or two sizes may also have imprinted themselves, in particular the 1.27mm SOIC pin spacing and the 1.9mm SOT23 base emitter spacing. Looking at pictures is all very well but the real world is a whole new ball-game and requires the acquisition of some skills in handling SMDs. At this stage I strongly recommend that you get some SMDs to play with. A couple of very simple projects will be described in just sufficient detail to get you started. Hand working methods will be covered in more detail later, when it will be better appreciated after a little practice.

The very small size and weight of SMDs result in some handling difficulties that may not be apparent from quoted dimensions and outline drawings. The first task is to come to terms with the almost microscopic size of the little beasts. If the code number on say a 1206 chip resistor cannot be read with the naked eye then some kind of visual aid will be needed to ensure that soldering is up to standard. Even those with perfect eyesight will feel the need for a small magnifier from time to time. Do not underestimate the importance of lighting conditions as we are working at the limits of human visual capability. If you doubt this try reading the code on a 1206 chip resistor first in daylight and then under a 60 watt bulb at night. In the latter condition the eyes' iris is wide open and as photographers will know this reveals all the imperfections of the lens and diminishes the depth of field. Many people with what would be regarded as quite good eyesight cannot read chip codes under artificial light. Daylight is the number one top of the shop as a light source. The optimum conditions are inside a north facing window, in daylight of course. Unfortunately most of us must work in dark winter evenings and so artificial light and optical aids are needed. The first requirement is therefore plenty of light at the work area. This is best provided as a mixture of good background light in combination with one or two spots or desk lamps. Positioning a light source can be a problem due to specular reflection. A light source off to one side is often better than if placed right in front. The most annoying reflections occur if the light forms a 45 degree angle at the working surface area in front of you.

There are many different magnifying devices which are a great help to the surface mount constructor making 1206 work easy and extend our capability to the really tiny 0603 or even 0402 scale. Some typical devices can be seen in Figure 30. The triple magnifier is particularly useful and such devices are obtainable from watchmaker and model maker suppliers. A "+4" dioptre hand lens with a handle which converts to a tripod stand is a real help for layout design at a low cost.

More complex visual aids which are handy for longer working sessions include stereo visors and Anglepoise lamp/magnifier combinations. These give a wide field of view and are actually very convenient to use. The back-light Anglepoise magnifier has the advantage of optimum lighting direction. Two examples of larger vision aids are shown in Figure 31.

Attempts to place a chip component onto its printed circuit board solder pads with unaided fingers will soon prove almost impossible. They are simply too small to get hold of with thumb and forefinger so tweezers are essential. In any case they must be held down whilst soldering. The whole chip approaches the soldering temperature and would burn the fingers. Select the tweezers well, they must be of good quality and accurate at the tip. If they have any misalignment the chips will flick out of control and the only way to find a wayward chip is with a vacuum cleaner. Try finding a black-bodied SOT23 at the bottom of a carpet pile. You will quickly find the disastrous effects of a magnetised pair of tweezers. Many chips have ferromagnetic leads (nickel content) and will fly towards a magnetic tweezers' tip and stick to it with great tenacity. Stainless steel tweezers are a must but some grades can become magnetised. A tape head or other type of demagnetiser can be

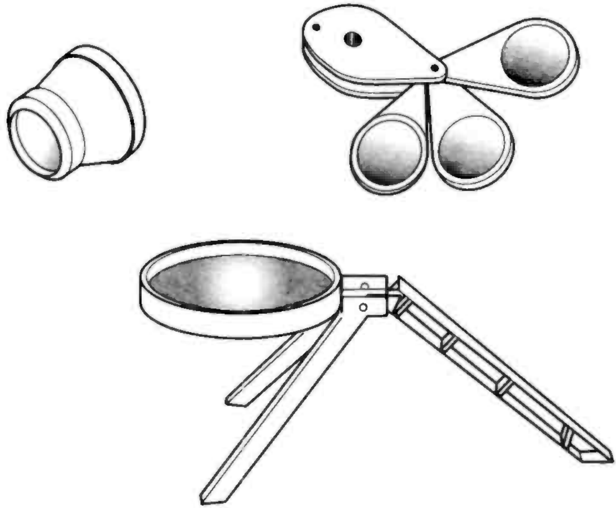
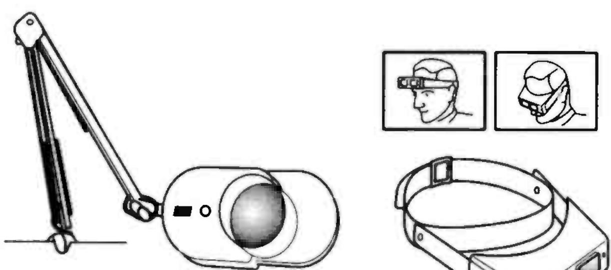


Fig.30 Selection of handy magnifiers for SM work



(a) Anglepoise light/magnifier (b) Vision visor

Fig.31 Wide field vision aids for prolonged use useful. Ceramic tipped tweezers are an excellent alternative. There are ranges of tweezers which are specially designed for SM work. These are expensive but the precise construction makes them a delight to use. The lightness of chip components gives us a further headache in that the slightest force or movement of the circuit board sets them out of alignment at best and at worst sends them flying off the board never to be seen again. A stable working environment is therefore essential. It is useful to construct a small work station about 30cm square and edged with some beading to a height of about 1.0cm. Standard or high density chipboard can be used for this and for the perfectionist an antistatic laminate surface with stud connector and four small rubber feet will complete the job.

2. Ugly construction

We are about ready for a little hands-on experience with SMDs. The first project is as simple as they come and we will use the most basic form of point to point construction. Actually this cheapskate method is very tedious and fiddly and really not SM at all. It will nonetheless help to focus the mind down to SMD level and show them at their worst, from here on it gets easier. If you find this physically impossible to make, don't give up. The proper printed circuit techniques described later are certainly a lot easier. The schematic shown in Figure 32 is for a static detector using an N-channel FET. If a statically charged object is brought near the gate of TR1 the LED changes in brightness. A 9V supply such as a PP3 battery is required. Resistor R1 protects the gate from high discharge currents. A resistor from the gate to 0V would seem normal but the circuit works well without it as a practical value would need to be of the order of 100Mohm for the small charges/high voltages we are trying to measure. You might try stringing a few 10Mohm chips together to reduce the sensitivity if required. C1 de-couples the supply and acts as an anchorage for R2 and the FET drain. The types of component specified are in no way critical and you should use any type of N-channel FET or LED that you can get hold of. The top view of the LED and TR1 shown in Figure 32 may not be correct for your chips but this is simply checked out in the usual way.

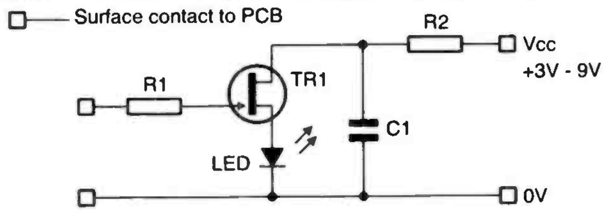


Fig.32 A static voltage detector

Component list for static detector (Figure 32)

Resistors

R1	1k5 1206 chip resistor
R2	1M0 1206 chip resistor

Capacitor

C1	100nF/50V 1206 ceramic chip capacitor

Semiconductors

TR1	BSR58 SOT23 N-channel FET.
LED	BY100R SOT23 general purpose LED

To put this lot together, select a small piece of PCB about 2cm square and make sure it is clean. A coating of flux from an aerosol will help if available. Use a standard soldering iron with the smallest tapered bit to hand. One problem with this ugly construction is the heat sinking capacity of the copper ground. A very small iron tip may not be effective and a compromise between power and fineness is called for in this case. Pick up the LED with the tweezers and solder the cathode to the ground plane, Figure 33. Yes this is a very delicate operation and now you see why good tweezers are essential.

Now solder the source of the FET to the anode of the LED, this is a little easier. Solder the 100nF capacitor, C1, to the ground and link it to TR1 drain. To the top of C1 solder R2 and to the gate of TR1 solder R1. TR1, R1 and R2 therefore hang in mid air. A couple of cm of wire connected to R1 acts as a

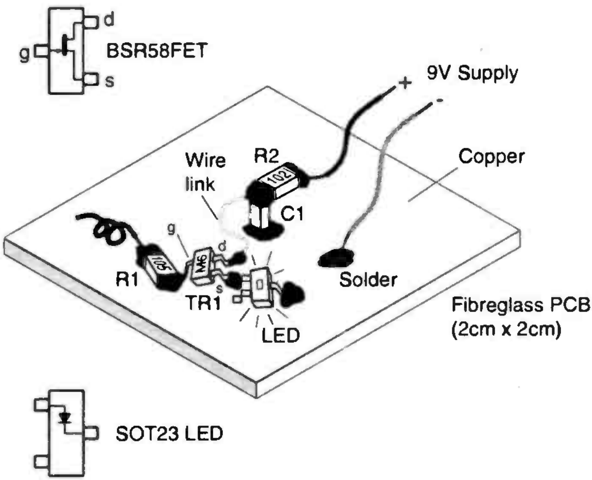


Fig.33 Ugly construction of static detector

pick-up. Now solder a short length of wire to R2 for connection to the positive supply and make a connection to the ground plane for the 0V supply. The ideal wire for the link and pickup is the fine Kynar wire used for wire wrap. The supply wires can be those of a standard PP3 connector. The current drawn from power supply should be a few milliamps if all is well.

3. Protoboard construction

To make life bearable a more effective form of construction is essential. A protoboard is the simplest viable method of circuit fabrication with SMDs. There is at present no type of stripboard which is suitable for SM use. Standard 0.1″ stripboard could be used for passive components with not a little ingenuity – anything is better than ugly construction. But real difficulties arise with active devices, SOICs with their 1.27mm

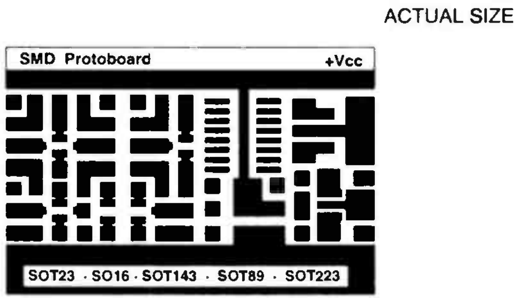


Fig.34 Track pattern for a universal SMD Protoboard

pitch means that the strip is twice as wide as we would like and impossible to use. A protoboard is therefore required and your simplest approach is to get someone to run off a couple of copies of the one shown in Figure 34. But if you wish to be self-sufficient, the simplest of PCB making equipment will suffice. Photocopying onto acetate film and making a sandwich of the three copies in register will give a dark enough photopositive to work from. The method of going from a printed design to a real live circuit is described later. The layout shown may not be optimised for your needs, and it is an easy matter to design your own protoboard to suit the type of work in hand such as digital, analogue, RF, and so on.

For a general purpose protoboard, some standard SM footprints can be obtained from a transfer sheet. A basic approach would be to set down a few SOT23 and SOIC pads along with plenty of 1206 pads. There is no spotface cutter as with strip board so gaps have to present in the design and inter-component links can be made with fine Kynar wire. Zero ohm jumpers should be used where possible. They make excellent linkers as they are neat and have very low resistance and inductance. This Kynar wire is available in several colours which can be useful for identification purposes. Quite complex circuits can be put together by using several boards for separate functions. On the other hand the unused areas of the protoboard can be cut off, making a small compact circuit if required. The protoboard design shown in Figure 34 is intended to minimise the number of links required. This was designed on a simple CAD system and the pads are somewhat larger than those from a transfer sheet so as to ease hand soldering. Referring back to Figure 3 you will notice that the SOT23 device can be connected in any orientation to accommodate joggles and other pin variations. Also, an electrode such as the collector is inevitably a nodal point where several connections are required. Typical collector connections are a load resistor and a coupling capacitor output, i.e. three connections. However several links will be required for most circuits. Let us now implement the static detector using this protoboard. We are still using the same soldering iron and solder as for TH construction. Silver solder and other refinements will be introduced later.

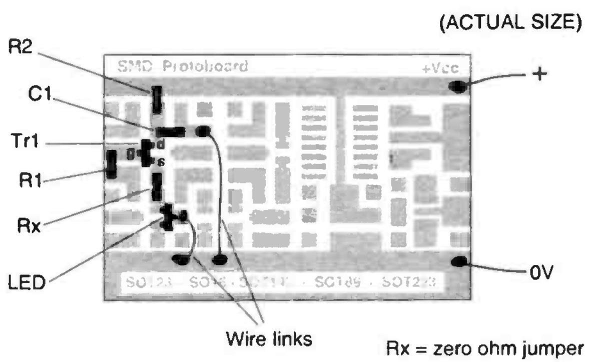


Fig.35 Static detector constructed on a Protoboard

The first job is to decide on what goes where. To do this the chips can be placed on the protoboard and just moved around with the tweezers until a suitable layout, with minimal links, is achieved. Don't spend too much time on this, it's not at all critical and it would not matter if everything was connected by links. Some Kynar or similar 30 swg tinned wire is essential as it is unlikely that all elements can be interconnected without a couple of links. Having decided on a layout it is now necessary to solder in the chips. To do this, gently press the chip down to the PCB with the tweezers tip and solder one end to anchor it. Subsequent contacts can now be made without it moving around. Start with the LED as before and work through, finally adding any wire links. One possible finished version of this circuit is shown in Figure 35, just two links are needed and an 0805 or 1206 zero ohm jumper denoted Rx.

At this point the magic of SMD should begin to dawn on you. You have just put a working circuit together without any drilling, having to turn the board over to solder, or snipping off pigtails. Including the time to get the chips out of storage, waiting for the iron heat up (starting with a ready made protoboard) this circuit can be running well within 10 minutes. The speed and simplicity of SM construction is surprising and from this point you may never look back. This exercise was aimed at getting the constructor attuned to the small size of SM chips and to some of the consequences of this. The art of SM construction does not stop here. There is much to be learned and it actually gets easier as experience is gained. It will be necessary to retrace areas like chip control and soldering in more detail later. Designing your own custom PCB is the obvious way forward.

4. Direct PCB design

The amateur is well supplied with materials for PCB design and production. Although this is primarily aimed at leaded components the principles are very similar and we have everything we need to produce excellent surface mount boards. To arrive at an etched PCB ready for population there are two main options: the layout design can be drawn "directly" on the PCB or a photopositive transparency can be used with UV sensitive PCB. Putting the design straight onto the PCB is just about practical for SM but care is needed. Both processes are summarised in Figure 36. The direct method of PCB production is the least expensive, requiring little capital equipment. The price to pay for this easy entry, apart from the fact that it is very fiddly, is that it's a once-only process. The whole design must be redrawn if a second circuit is required or if the etch goes wrong. Similarly if a modification to the design is needed you

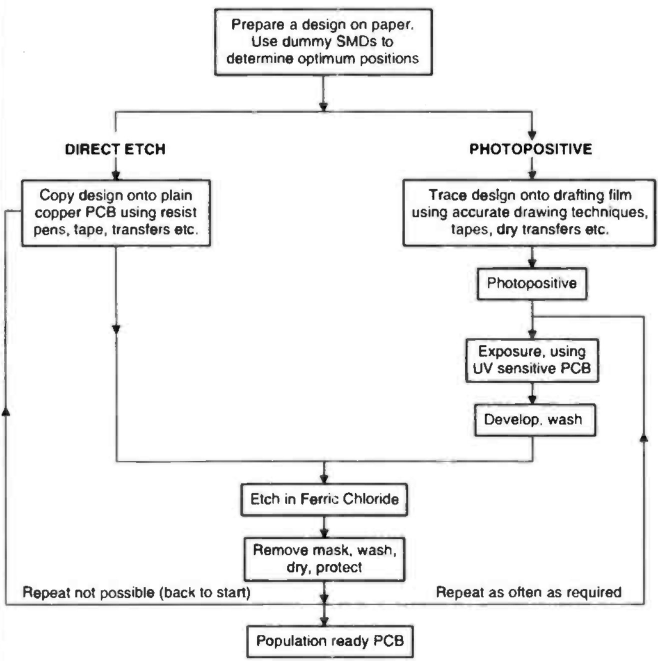
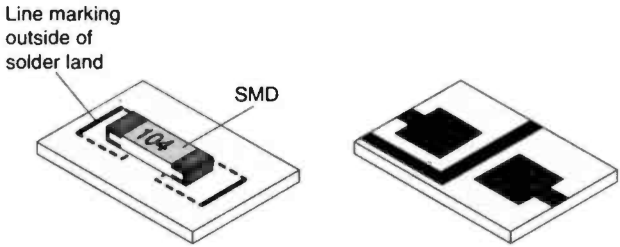


Fig.36 Options for manual production of PCBs

must go right back to the beginning and redraw it. Even so it is more practical than with a TH circuit where the holes would need to be drilled again. At the high tech end, once set up for producing positive transparencies on a computer system it is a simple matter to run-off a modification. In order to demonstrate the flexibility of SMDs and to encompass those who don't have a computer system I will start with this very basic direct method of PCB production.

The first objective is to produce a rough pencil sketch of the layout on paper where it can be knocked into shape and the optimum component positions found. A suggested approach is based on designing a new kitchen where scale models cut from paper can be used. First draw a schematic of the circuit in question. It is useful to keep a few SMDs to use as dummy components for designing layouts. Start a surface-mount junk box, it soon builds up and storage space will never be a problem. Place the chips on a piece of paper in roughly the same positions as in the schematic, but suitably scaled down. There is a tendency, at this stage to go for maximum miniaturisation. Try to avoid this until you are more experienced as it will create soldering and etching difficulties. Using a fine, 0.3mm, propelling pencil, mark a line at the ends of each chip. Remove the chips and convert each line into a rectangle forming the solder pad. This is pure guesswork (art) but it will be obvious that sufficient land must protrude out from the chip for the soldering iron to make good thermal contact. SOICs are treated similarly, just place a dot at the end of each pin, filling in as rectangles later. SOICs are tricky but it can be done. Running tracks under 1206 chip resistors and capacitors is quite practical. Running tracks between SOIC leads is really not for the beginner, especially when drawing by hand. Even with computer artwork such lines are so fine as to create problems in etching. Having created a layout on paper with pencil and rubber it must be copied onto the copper PCB. A visual aid such as a 6″ focal length stereo vision visor is very real help for all stages of manual PCB design. Hand drawing an SM track design into a copper PCB poses one main problem, that of making the track fine enough. The most basic approach is the famous Dalo pen. As supplied the tip is much too coarse and it must be pared down to a fine point, about 0.5mm, with a sharp scalpel. Simple circuits may not need a paper sketch and you could design the layout directly on the copper. The procedure for this is illustrated in Figure 37.

Place each chip on the copper surface and draw a line as on the paper layout. When the two end lines are made, remove the chip and fill out to a suitable sized solder pad with the modified Dalo pen. A transparent ruler can be used for straight lines including tracking between pads and with a little practice quite neat looking work can be produced. Fine overhead projector pens for permanent drawings on acetate film work well on copper and can also be used for this function. Etch-resistant tapes and acid-proof dry transfers are useful alternatives. The 0.031 inch tapes can be used for SOIC leads on



(a) Place a chip on the copper and mark the ends

(b) Fill out the pads and add tracks

Fig.37 Drawing a circuit design directly on the PCB

0.05in (1.27mm) pitch. Finally check the resist areas for pinholes and scratches and patch up where needed with the Dalo pen. The board can then be etched in 30% ferric chloride solution (1 part, plus 2 parts water) in the normal way at about 50°C with continuous agitation. It is then washed with water and thoroughly dried. The resist is then peeled off, removed with acetone (nail varnish remover) or otherwise. The copper should be rubbed down and polished with clean cotton-fabric, tissue paper or a proprietary polishing block. Mild abrasive cleaners for domestic use will remove the etch resist and polish the copper surface. To preserve the solderability, a coating of flux from an aerosol spray will suffice.

4. Designing a Photopositive

Hand produced, 1:1 life sized, artwork on UV transparent, drafting film, is by far the preferred low tech approach. We will therefore look at this in more detail. It is still very much an artistic method, dimensions being obtained directly from the actual chips. We will get more quantitative later when com-puter design is considered. This hand made master artwork can be used to run off any number of copies and can be extensively modified to accommodate design updates. This is probably the lowest entry level for serious quality work. Some small investment is required but the benefits outweigh this. Producing a master starts with a paper sketch, as above, which is then used as a tracing guide. The drafting film is then fixed over the paper sketch and the final very neat artwork is produced from it. This will be of a much higher standard than the direct PCB drawing because precision drawing pens can be used and they need not be etchant proof. A drawing could be made directly on drafting film but the paper intermediate obviates corrections to the final artwork. A practical project will be used to illustrate the procedure. This is a useful little audio output stage with flexible parameters, with many uses such as a signal tracer or radio output. The circuit is shown in Figure 38.

The LM386 IC is well known and the connections to the SOIC/SO8 and 8 pin DIL/TH versions are identical. By altering the components between pins 1 and 8 the gain can be tailored to requirements. With pins 1 & 8 NC, i.e. no external components, the gain is X20. With the 1.2k and 10μF components shown the gain is X50 and if a single 10μF capacitor is used the gain increases to X200. Provision is also made for input variation such as capacitive coupling or direct coupling. The data sheet should be consulted for more information on frequency response and so on. Note, I have adopted the use of small squares for connections to the circuit to symbolise that wires will be soldered to surface pads. The components marked Rx are zero ohm jumpers and are a great help for jumping tracks and keeping the finished circuit board neat. They are also more reliable than wire links and much easier to use.

Components list for Audio Amplifier (Figure 38)

Resistors

R1	10R 1206 chip resistor
R2	1k2 1206 chip resistor or 0R for high gain option
RX	0R0 Zero ohm jumpers

Capacitors

C1, C4	100nF/50V 1206 ceramic chip cap (2 off)
C2	50nF/50V 1206 ceramic chip cap
C3, C5	47μF/16V tantalum cap (2 off)

C6	10µF/16V tantalum cap

Semiconductors

IC1	LM386 in SOIC SO8 package

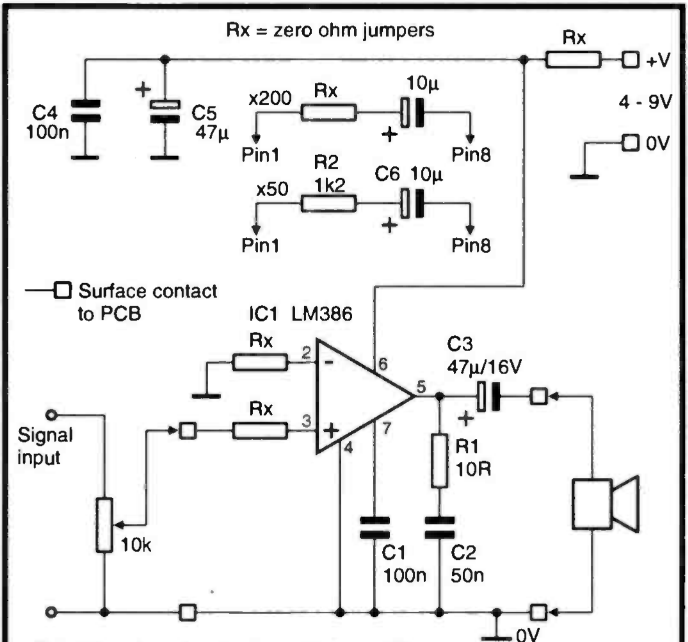
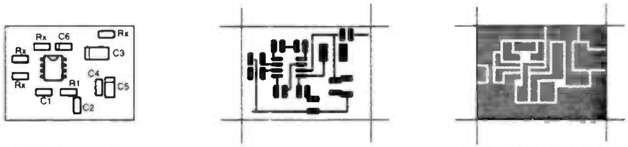


Fig.38 A universal audio amplifier

To create the pencil and paper layout design use some scrap components as dummies and lay them out as in Figure 39(a) This may take a little time as interconnecting tracks have to be kept in mind and links and under-runs kept to a minimum. A first approximation to the layout can be almost identical to the schematic diagram. This can then be stretched here and there to account for actual size of the components and to achieve an even distribution of chips on the PCB. If the components end up crowded in places, it is probably not the best layout. Don't try to make the circuit too small as plenty of pad area makes hand soldering much safer. It is good practice to make power



(a) Lay out using dummy chip components

(b) Pencil sketch land areas and link up

(c) Fill out and shade-in with pencil

Fig.39 Development of a PCB track pattern

rails as thick as possible. Even with a generous layout an SM circuit will still be very small and of course the profile is not much thicker than the PCB itself. Surface mount gives us a great volume reduction even if the area is not fully shrunk. Use a 0.3mm propelling pencil to sketch out the pads at the chip ends and SOIC leads. This is illustrated in Figure 39(b). Draw in the interconnections as fine lines. Now fill out the tracks keeping as much copper on the board as possible. A light shading with pencil helps to define the tracks. You should end up with something like Figure 39(c). Keep testing the drawing by matching up the chips and don't worry if ample use is made of the pencil eraser.

Now for the precision stage. To produce the final 1:1 positive master you will need some drafting film, which is readily available. The key to success at this stage, notwithstanding any vision aids you may choose, is a fine drawing pen. You will need a 0.25mm Rotring Isograph or similar professional grade drawing pen. A bottle of black drawing ink is also needed. This pen is expensive but it will last if maintained properly and this means cleaning it regularly. A 0.35mm pen is useful for filling in large areas and if you are pushing the barriers forward or into making bugging devices, a 0.18mm pen will make superfine masters. A scalpel and some small blades is required for corrections to the master. A Swann-Morton size 3 handle fitted with No. 15 blade is just fine. Obviously a UV light exposure unit is essential, although sunlight works to some extent. Doing a bit of DIY on the exposure unit by adding a white 60 watt bulb makes an excellent back lighting unit for making and correcting the final master. I should not have to point out that a separate mains supply is needed to operate the bulb independently of the UV source. Staring into the UV will shorten your career in SM. Finally a professional grade stencil is necessary, particularly for drawing straight lines. The type supplied by art shops with electrical layout symbols is suitable, for example Rotring Art 852.762. This has tiny pips on one side to keep it slightly off the paper and an accurate cm scale along one edge which is very useful. We are now ready to go.

Cut off a piece of drafting film, about 8cm × 6cm for the circuit in hand. Draw an accurate rectangle, defining the edge of the circuit, on the final pencil drawing such that the lines project a little at the corners to act as registration markers or fiducial lines. Place the drafting film over the drawing and fix the top edge with some Pritt adhesive. Using your 0.25mm drawing pen and stencil, trace the lines around the edges including the corner projections so that if the film is moved it can be re-registered. Use a straight edge on the stencil for all lines. A good plan is to work down the drawing, copying the horizontal lines first. Then work from one side putting in the vertical lines. Finally draw in any tricky bits freehand. You should now have an outline drawing of all the landmasses. In this final operation the gap between tracks is set and must be kept sensibly wide. A separation of 0.5mm is comfortable whereas 0.25mm is getting a bit close and 1mm looks like a barn door on this scale.

The track areas can be filled in using the larger pen to flood in wide expanses. The ink can be dried from time to time on a radiator, especially if corrections are required. It is inevitable that the odd slip will create unwanted bridges or minor changes will be needed. The drawing ink does not penetrate the drafting film, but just wets the translucent surface coating and when it is fully dried the ink is easily scraped off with the tip of a scalpel blade. This is a delicate operation but with a little practice it will be found to be a most useful facility. You should now have a hand-made photopositive master similar to Figure 40.

The most difficult step in this manual process is the drawing of footprints for SOICs, SOT23 transistors and so on. A little further investment can greatly ease this process. A4 sheets of SMT dry transfers containing all the common footprints are

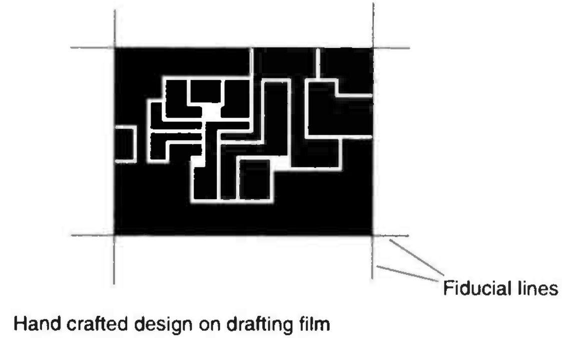


Fig.40 Photopositive for AF amp (actual size)

available. These are made for 1:1 working and can even be applied to the PCB directly. The footprints are aimed at hand working, being a little larger than would be used for machine placement and soldering. The production of a master positive is now just a matter of deciding the overall component positions, popping down the required footprint from the transfer sheet and then adding the interconnecting tracks. The transfer sheet even contains suitable tracking. In practice I find this is very useful, giving neat professional results in a much shorter time than the manual method. The one slight disadvantage is that the design does not leave much copper on the board. I therefore use the transfers in combination with the manual method. The technique is to set down the complete circuit using transfers and then fill out the tracks with the Rotring pen. Most of the pads can be expanded as they are quite small. This greatly eases manual placement and soldering of the chips. In some RF designs the more skeletal layout can be desirable but even in this case it is important to have as much ground area as possible. The ground and supply tracks can be filled out by pen, possibly the larger 0.35mm size. The transfer sheet is illustrated in Figure 41.

A photopositive made by any of these techniques is quite robust and the drawing inks so far encountered have good

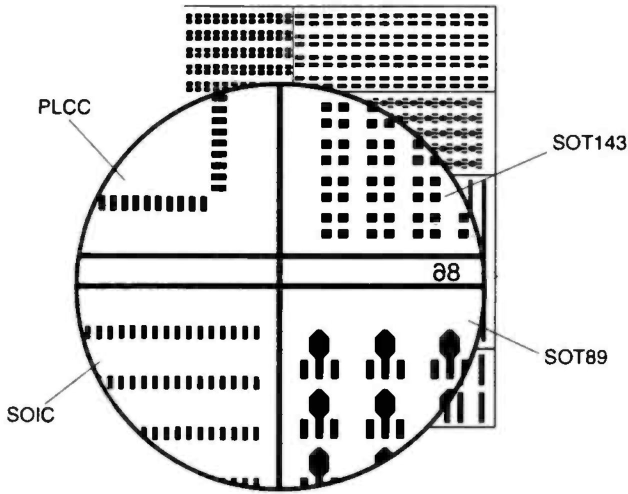


Fig.41 A4 sheet of dry transfer SMT footprints

flexibility. It is still good practice to check for pinholes and small cracks before use. This is done by back-lighting. An artwork lightbox with an A4 viewing area (230mm × 380mm) and a white diffuser is ideal for this and, indeed, for preparing the drawing in the first place. Alternately a DIY version such as mentioned previously will suffice for this purpose and the trusty 0.25mm drawing pen will do the required patching up. Manual techniques as described can lead to quite neat results.

The completed audio amplifier circuit produced from the hand crafted photopositive on drafting film is shown in Figure 42. But it is hoped that you will plough through Chapter 5 before you set to work on it.

6. Computer Based Design

Computer based PCB design systems represent the ultimate in speed and accuracy. New programs are appearing regularly and

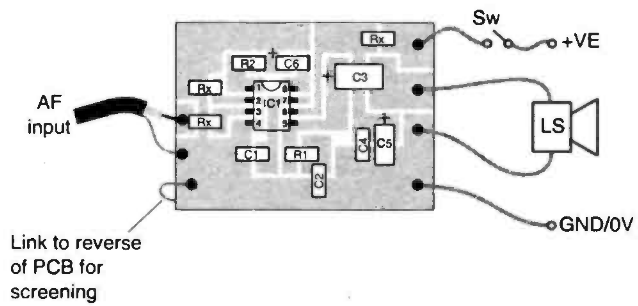


Fig.42 Audio amplifier component positions on PCB

all of them boast SMT support. Happily the prices are realistic and hardware demands modest. Even the most basic 286/VGA/9-pin matrix printer set-up will get you started. The older PCB design programs only offered limited built-in SMT facilities but they need not be dismissed. Most have the facilities to lay down rectangular pads of various sizes at the correct spacing and to link them with tracking which is the basic requirement. PCB design systems open up a world of facilities but these are better appreciated if based on some practical experience with manual methods. It is important for the beginner to relate to the physical world by printing out and matching up dummy components at regular intervals as the layout is developed. Designs are easily produced on the screen which would be impossible to use in practice. Typical problems are excessively fine lines, insufficient room to get a soldering iron tip to the pad and not enough space for component casings. More advanced users can, of course, go from schematic to PCB with fewer test printouts.

The most obvious attraction of PC based design systems is their editing power. Images can be updated instantly without loss of quality and the whole design can be repeated several times on the same printout. Extensive symbol and pad libraries, which can be added to, save time and ensure consistency. Grid-snap or freehand mode allows placement of tracks and pads down to 0.025 or even 0.002inch. The 0.05in grid-snap (half of the 0.1in standard grid) is the 1.27mm spacing required for SOICs, ideal for SMT work. The smallest areas of the layout can be magnified with a zoom facility for such accurate placement, or for routing of underpasses. Pad and track widths can be altered over a wide range. Tracks can easily be placed between SOIC leads on the screen but in the real world this pushes the etching technology to the limit and is probably best avoided. Digital system design is made easy by track repeat facilities.

A great advantage of SMT is the capability of double-sided construction leading to even more compact circuitry. Most PC systems offer this facility, but of course small projects will seldom warrant the extra complexity. Solder mask and silk screen layers are an attractive facility for small run offs.

When working with PC systems it is possible to approach a layout design with dimensions firmly in mind. This contrasts with the manual methods, especially when working at the 1:1 scale, where pad size is very much a matter of subjective judgment. Of course it is possible to operate the system in a trial and error fashion but after a little experience the advantages of an objective approach will be apparent. In previous chapters, chip dimensions were given to illustrate their small size, telling us little of the footprint requirements. In layout design, particularly with CAD systems, we are concerned with footprint areas. The establishment of pad sizes suitable for hand soldering and a knowledge of chip component area demands, leaves only track routing to worry about. Schematic capture facilities on certain systems can help here and zero ohm jumpers can reduce the number of wire links.

Chip component manufacturers recommend footprint dimensions for hand soldering, which won't let you down, but I find it better to expand these as much as possible in a particular layout. Heavy reworking can dislodge fine tracks and excessively small solder pads from the fibreglass substrate. The formulations which exist for calculating land pattern sizes are a bit optimistic and not really applicable to hand soldering. Typical chip component footprints for the more common devices are given in Appendix V. These are based on manufacturers' recommendations for wave soldering. For hand working these should be considered minimum and expanded where possible. For manual work with solder cream, use the recommended wave soldering dimensions given. When significant power dissipation is required the pad area can be an important radiating element. The copper rear side ground plane can also take some heat away. In most applications power devices can be adequately "heatsinked" with a square cm or so of copper, but if there is any chance of a meltdown, proper calculations are required. If certain circuit elements are heat sensitive, a suitable separation must be used or a separate circuit module may be needed. In any case a modular approach is very useful as small PCBs are easier to handle and updated units can be added as required.

As with manual drawing, attention to the gaps between conductors on a PCB is important and again 0.5mm is a safe rule for project work. The degree to which this can be reduced is dependent on your system for getting from on-screen to photopositive and how the UV exposure and etching is done. One possibility for determining device placement is to set a clear area, or aura, associated with each device and keep it clear of other chips. In general the component density on a board should be arrived at by considering factors like clearances required for rework and repair, cooling requirements, accessibility for soldering and inspection of joints. Some rework tools require a disappointingly large access area, at each end of chip resistors for example. It is good practice to lay out components in parallel with grid lines. Arrange for services like V+, V0, input and output to be connected to pads at the edge or ends of the PCB. CAD systems cope well with hybrid circuits containing a mix of TH components and SMDs. Plated-through holes (PTH) are easily produced using small kits made for the purpose.

To illustrate a computer based design we have another simple but useful project. This is a live wire and hash detector. I find it a very useful addition to the DIY tool box. A couple of centimetres of PCB tracking acts as an antenna. It will detect where live wires are running in a wall and will trace sources of interference. It detects the 50Hz mains as well as any hash or spikes on the line and there is usually plenty. A visual

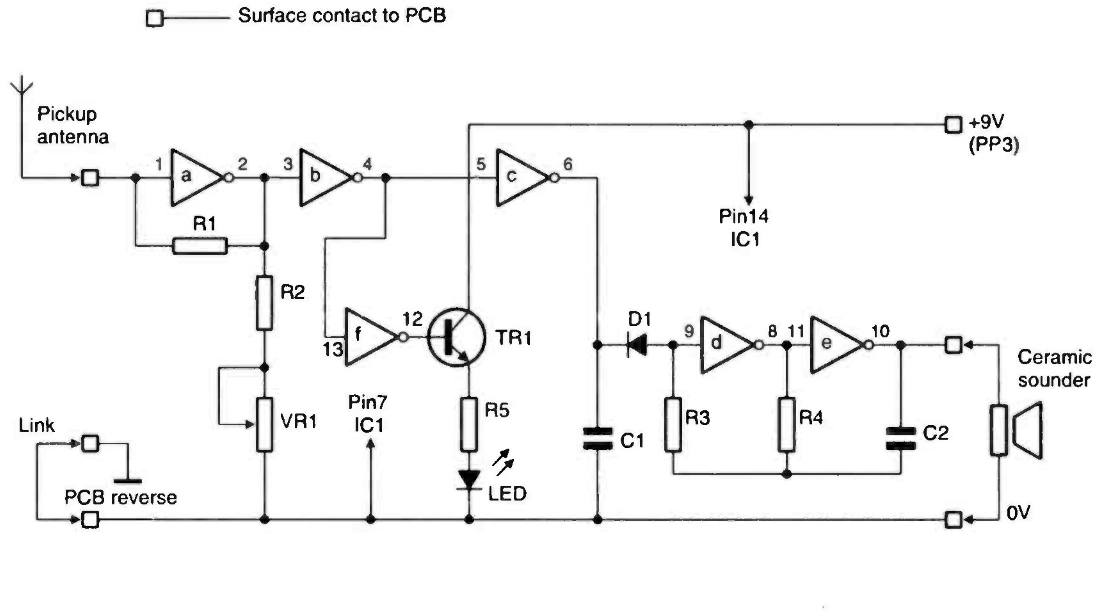


Fig.43 Live wire detector

indication is given by the LED which should be a bright one. An audio indication is also produced on a ceramic sounder. This is a high pitched tone for a strong mains signals but the switching is gradual and an indiction of the type of interference can be heard. The circuit will draw about 15mA with all guns blazing. Sensitivity is adjusted with VR1 so that the LED begins to light up when the antenna is a few inches from a live wire. The device is very sensitive and the length of the pick-up antenna can be reduced if required. The sounder leads are carrying square waves and should not be too close to the antenna.

Components list for Live Wire Detector (Figure 43)

Resistors	
R1	10M 1206 chip resistor
R2	4k7 1206 chip resistor
R3	470k 1206 chip resistor
R4	220k 1206 chip resistor
R5	470R 1206 chip resistor
Potentiometer	
VR1	50k Type 3204 preset
Capacitor	
C1	100nF/50V (X7R dielectric)1206 ceramic chip cap
C2	470pF (COG dielectric) ceramic chip cap
Semiconductors	
D1	BAS16 SOT23 diode
LED	BR100 SOT23 general purpose LED
TR1	BCX19 SOT23 general purpose NPN
IC1	4069 CMOS hex inverter in SO14

Before producing a track layout for this circuit I made up a set of 1206, SOT23 and SO14 footprints for 1:1 working and copied them onto the design as required. This was done on a standard PCB design package which had only a limited set of SM pads. For the 1206 footprints I selected a square pad, varient "C" on my system (Easy PC), and printed out a selection of pads of various widths. Matching up a 1206 chip resistor to the printout I selected pad width "12" which looked comfortable for hand soldering. The next job was to complete the footprint by adding the second pad for the other end of the chip. This was a matter of placing a second pad of the same size and fine tuning its position in freehand rather than snap mode. A couple of print-and-try sessions was all that was needed. The SOT23 footprint containing three pads was designed in the same manner but based on varient "G" which is a rectangular pad. For this project a width of "7" was selected. The SO14 IC footprint has a pin spacing of 1.27mm which is half of 0.1″ so that in snap mode "1/2" of the placement is automatically spot on. The only problem was to select a sensible size of pad with an adequate gap between the IC leads. Incidentally in designing the protoboard discussed earlier the length of the IC pads was important as sufficient area is needed for fine connecting wires to be soldered to each one. For present purposes the IC pads were made up of two varient "G" pads of width "6" placed end to end.

Having produced a set of footprints in a corner of the CAD screen it was a simple matter to block each one and repeat it in the required position. After all the footprints were placed, the tracks were added. Track widths were as wide as possible consistent with adequate gaps. A 1:1 printout was then made and dummy components laid in position. Using the block and shift facilities in freehand mode, tight spots can be relieved and a good layout produced in a few print and match up sessions. The first workable design produced by this method is a skeletal layout as shown in Figure 44(a). This is quite practical with the advantage that chip component positions are easily identified. I have used many such skeletal PCBs without difficulty, but in keeping with the philosophy of leaving as much copper on the PCB as possible the spaces can be filled up by adding tracks and pads as required. The result is shown in Figure 44(b). In this case a small residue of the original footprints has been left so that the component positions can still be found.

This track layout has to be printed out and turned into a PCB which must then be populated. The input impedance of this circuit and the static detector circuit is very high and therefore glass fibre PCB is recommended. It is assumed that double sided PCB is going to be used but the reverse side copper

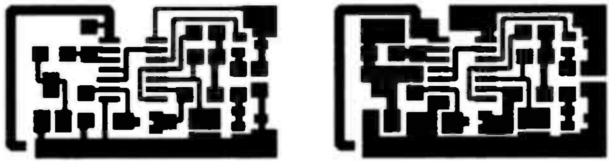


Fig.44 Skeletal (a) and filled out (b) versions of the live wire detector PCB shown life size

should be removed from the pickup antenna area. If you intend to use this track layout it is hoped that you will plough through the next chapter on working with SMD's first as it is intended to be instructive and may help to get a better result. A com-ponent overlay based on the filled in version is shown in Figure 45 for convenience. Obviously it should be housed in a plastic box.

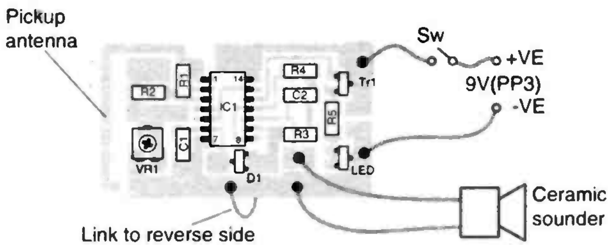


Fig.45 Live wire detector component positions

The weak link in home computer systems is the means of producing a working photopositive of the track pattern. The first degradation is the printer itself in terms of head accuracy, print density and so on. This is particularly the case with the ribbon printers. Inkjet printers give very accurate reproduction with excellent print density. This can be further improved by selecting a paper which has been designed for inkjet use. Direct printing onto drafting film with matrix or inkjet poses drying problems and the density of the image is insufficient. The simplest practical solution is to run off say three photocopies of a paper printout on clear acetate film of the type used for overhead projectors. Then cut out the images, leaving a suitable area around each. Next combine them in register using a little Pritt adhesive about 1cm away from the business area. Some form of registration aid such as a fine line around the edge of the image is a big help. Sandwiching the photocopies increases the density of the image. Rotating the paper master in the photocopier between copies will ensure that imperfections from the copier cancel out. The 1% to 2% expansion produced by many copiers is not usually a problem, but some copiers expand more than this and expansion can be different across and down the page. Copying published artwork in this fashion is also possible. The quality of matrix printers can be improved by printing out the image enlarged, ×2 or more, and then photoreducing onto film. A small improvement in final PCB track quality can also be achieved by printing out the image reversed so that the print side of the photopositive must be in direct contact with the resist on the copper.

Laser printing directly onto film can be successful. Many computer PCB design programs provide drivers for a plotter which is a very good method of getting a large image which can be reduced. It is also possible to get release paper which will transfer its laser printed or photocopied image directly onto a PCB using a hot iron. Yet another possible method is to spray a paper printout with "WD40" which makes it transparent to some extent and, with care, this can be used directly for UV exposure. Prolonged exposure is required with this technique. Remember to be careful about using aerosols in confined spaces and be aware of possible flamability dangers. Always follow manufacturers' instructions and precautions. Moving up market, the design can be stored on disc for printing by a photoplot bureau. Before continuing with the live wire detector, let's deal with some more practical matters.

Chapter 5

HAND WORKING WITH SMDs

1. Selecting the Substrate

Having produced a track pattern for your PCB, a suitable substrate must be selected. For the SMT professional this is a daunting task. Most of the problems concern mechanical properties such as thermal conductivity and thermal expansion coefficient, flexural rigidity and electrical characteristics like dielectric constant and RF losses. Such considerations lead them to choose a particular substrate from a wide range of types. Examples are fibreglass/PTFE for microwave use or alumina for military or potted engine management systems. Less accessible PCB materials include polyamide-fibreglass, polyamide-aramid, epoxy-aramid, polyamide-quartz, and various mixes of these. They have light weight, very low thermal expansion coefficients and special dielectric properties. A recent development consists of a thick aluminium base coated with an insulating layer onto which the copper is bonded. This has remarkable thermal conductivity and heatsinking properties. Inorganic, constrained core – based on copper clad Invar, compliant layer and flexible substrate will also be found. If you come across some odd looking PCB it may be one of these but it need not bother us for amateur construction. The choices for the amateur are to some extent made for him by availability, but one should be aware that this is a vast subject and if you are working with anything out of the ordinary, don't make too many assumptions.

The soldered joint, bonding the chip component to the PCB, is also its electrical contact. It is not surprising that this is a focus of attention for long term reliability of the circuit. During its lifetime the joint is subjected to mechanical stress and stress resulting from temperature cycling due to power dissipation and of course environmental temperature cycling.

SMDs are mounted very securely to the PCB. Any stresses in the board are therefore coupled into the chip. Such stress can arise from simply bending the board, for example by bolting it to an uneven surface. Also, if the PCB is tightly bolted to the equipment housing, any handling stresses will reach the PCB. Electrical power dissipation in a chip component will lead inevitably to thermal expansion. This puts stress (force) on the joint which responds with stretch or contraction which is referred to as strain. At switch-on the chip heats up faster than the surrounding PCB producing an initial strain on the joint even with well matched component and substrate. Ambient temperature increase or decrease will result in strain if the component and the PCB have different thermal coefficients of expansion. A good example of the latter would be a ceramic chip on epoxy fibreglass which would only be reliable over a limited temperature range. If the strain on the joint exceeds the elastic limit of the solder then plastic deformation will take place and after a number of cycles the joint will fail. There are many other factors which can be considered in pursuit of circuit robustness. Well that's the bad news, but don't worry, it's not so bad.

Continual cycling of the joint within its elastic limit is OK. Solders have a yield point approaching 0.5% so they will simply bend or stretch and fully recover up to this point. Luckily, chip components like SOICs and SOT23 discrete devices with gull-wing leads can also take some of the strain by slight flexing of the gull-wing. Leadless devices like chip resistors and capacitors are less forgiving, but they are so small that for most sensible temperatures the expansion is within the elastic range of the soldered joint. The compliance of the joint is maintained by using as little solder as possible consistent with making a good joint. As for mechanical stresses on the PCB it is a simple matter to arrange some compliance in the mounting yet hold it securely with plastic mounting pillars and so on. Also small PCBs will flex to a much lesser extent. I strongly suggest the use of double-sided PCB and keeping as much copper on board as possible. Most amateur projects will occupy only one side of the PCB in which case the copper screen on the unused side should be connected to the ground on the circuit side in one or more places. This can be done with Veropins or with snugly fitting wire links around the edge.

The above precautions must be pointed out for completeness so that if a critical situation arises the constructor is forearmed. It must be said that in several years of work with SMDs as the only method of circuit fabrication, I never had a circuit failure due to mechanical problems, albeit most work has been with PCBs measuring less than 100cm^2. I have also been surprised at the toughness of SMD circuits both hand-made and commercial, their indifference to abusive flexing and mechanical shock and to freezing both by nature and with aerosols. The reliability of SM circuitry is also apparent from our daily experience with consumer goods which just go on working.

The choice of easily available PCB materials is limited to SRBP in various guises or the ubiquitous fibreglass. The lowest cost materials are the SRBP paper based phenolic and epoxy reinforced composites. They have poor dimensional stability and poor flexural strength. The electrical properties are not so hot either and this is not helped by twice the water absorption of fibreglass. Double sided SRBP can be used for SM without problems but I have found that the single sided material warps badly and is probably best restricted to very small circuits. The epoxy composite material is available with a flame retardancy rating up to FR3 and is the best quality. But our old friend, woven fibreglass – epoxy laminate – is an excellent general purpose material and although it is sharply dearer than SRBP it is worth it for the peace of mind. The most popular grades are the G10 and the preferred flame-proof (FR4) material which is made to BS4584. This will run continuously at 130°C and will take whatever you throw at it for soldering. This board is 1.6mm ($\frac{1}{16}$″) in thickness with a copper content of 305g/m^2 (1oz/ft^2) and 0.0356mm (0.0014″) thick per side. Thin fibreglass PCB of 1mm thickness can be found and will save space if a very low profile circuit is needed. The flexibility of this material limits it to small projects. The double sided, standard 1.6mm, material is sometimes called 2 ounce thickness. Cutting, sanding and filing this type of PCB material produces loose glass fibre dust which has a health risk, so take appropriate care. For most purposes positive working photo-resist pre-coated boards will be used. This is covered on both sides with a peelable crepe material or black plastic film which protects it from light and scratches in storage and handling. The sensitivity of this material to UV and its development time can vary considerably with storage, especially if the temperature goes above 29°C. The protective backing is removed from one side and the master artwork is placed in direct contact for exposure. The art is placed on the PCB with the ink/image side away from the resist layer. The image therefore faces the light source, unless you are working with a reversed image to improve resolution. Drafting film is translucent and needs more exposure to the UV than clear film. You can coat your own PCB with UV sensitive photo-resist using a proprietary aerosol spray. Good clean working conditions are required and the copper surface must be well cleaned beforehand. Home treated PCB has shorter exposure and development times but it requires some practice to get an even film thickness.

After exposure to UV the positive working photo-resist which is a blue-green colour must be developed. On development the latent image initially appears as a darker colour but soon dissolves to leave the unprotected copper. Fully sealed kits are available for PCB developing and etching and these can easily be used for SM work. The most basic home brew method is described here. The developer solution is strongly alkaline and must be handled with great care as it will readily dissolve skin. If you do get a drop on your skin, wash the area under running water.

A proprietary developer will give consistent results but a simple home-made formula based on sodium hydroxide (caustic soda) works very well. This consists of about 1.2% caustic soda (weight/volume) for use with home-made spray-on resist or 1.6% for commercial pre-coated PCB. Development is at room temperature and should be complete in under 10 minutes, ideally about 5 minutes. Discard the developer after use. It could be stored for a short time in a sealed container. Plastic is better for this as it will tend to etch glass over time. Carbon dioxide from the air will slowly neutralise it during use.

The developed PCB must be well washed as any alkali carry-over will neutralise the acidic etchant, ferric chloride, and in fact it will precipitate undesirable iron compounds. A very gentle application of some abrasive cleaner will clear the mask residues from the narrow gaps between tracks and reduce the chance of copper whiskers remaining. The most convenient etchant for the copper is a solution of one part ferric chloride dissolved in two parts of water by volume. This is acidic and is corrosive to most metals including stainless steel. A plastic or glass container is essential. It will permanently stain most materials especially cotton fabrics. Etching is done at about 50°C and continuous agitation is essential – gently rocking the bath will suffice. Etching time depends on several factors but should be complete in about 20 minutes. The ferric chloride solution is usually stored for repeated use until it becomes very cloudy and obviously exhausted.

After etching with ferric chloride solution the board should be rinsed well. The mask should then be removed with stronger, 8%, caustic soda solution. The green mask should change to a more purple colour and dissolve away. If some traces remain after a few minutes the PCB can be rinsed well and then rubbed gently with an abrasive cleaner to remove it completely. Excessively strong caustic soda solution tends to make the mask less soluble.

The steps leading to the preparation of a population-ready PCB are just the same as for through-hole work. There may be a tendency to produce very fine tracks with SM and as all land areas etch sideways, in some circumstances the fine track could etch through. This can happen when there are large areas of land to remove and insufficient movement in the ferric chloride bath so that the process takes too long. Good solderability of the copper land areas is very important for SM work. For this reason some attention to the final preparation is useful.

After etching, wash off fully with clean water. If the copper is not perfectly polished then use a mild abrasive to ready it and rinse it off fully. Dry the PCB with paper tissue and then spray it with some alcohol or 1,1,1, trichloroethane PCB cleaner to remove any greasy deposits. Completely dry it in a warm place. Finally, spray with a fine coating of rework flux from an aerosol. Harden up the coating by getting the board hot on a radiator for at least half an hour. The flux is based on a rosin derivative and its high static friction coefficient helps with control of the chip components during board population. You should now have a shiny well protected PCB with excellent solderability, ready for population. Roller tinning the PCB will also give excellent solderability, although the little solder bumps at the trailing edge of the solder pads can be a bit of a problem. These could be removed with solder wick but this is not usually necessary. Air knife and other more expensive methods leave a smoother surface for SM work. Chemical tin-plating can be difficult to solder if not done correctly.

2. Working Conditions

A good working environment will go a long way to ensuring success in producing a neat looking circuit which performs its intended function reliably. You can start with just an iron and a little standard 40/60 solder but it will be useful to develop a more customised set-up. Good lighting and an array of magnifiers have been described and are a vital part of any SM set-up. Other items will be mentioned as the need arises.

SMDs are packaged mostly in reels or magazines and occasionally in bulk or loose. The 18cm diameter reels contain typically 1000 or so chip resistors, capacitors or active devices. Small quantities will be cut from the reel as required. Full or part reels appear on the surplus markets from time to time, and the purchase of a reel of common values like 1k resistors could be less expensive over time and certainly very convenient. Components on the reel are in a paper or plastic tape containing small sealed individual cells as shown in Figure 46. Left in this form they will maintain their solderability for a long time. Free components, open to the atmosphere will absorb sulphur and other compounds from the air reducing solderability. This would be a disaster for automatic production where all chips are soldered at once but for hand working where individual chips are attended to it is rarely a problem. In any case it is better not to handle chips if you can help it because salt and other constituents of perspiration slowly react with the terminations and reduce solderability. Resistors and ceramic chip capacitors tend to be in paper tapes whereas diodes and transistors tend to be in plastic tapes. In both cases the wells are covered by a continuous strip of clear plastic film. This is peeled off by the pick and place machine and as each component is exposed it is picked out with a tiny vacuum pick-up device and placed on the PCB. The orientation of discrete devices in a tape is not random but as required by the production machinery. Magazines are used for integrated circuits, they have a variety of cross-sectional shapes to suit the chip. The length of the magazine is 50cm with a flexible plug at each end but they can be cut and plugged for small numbers.

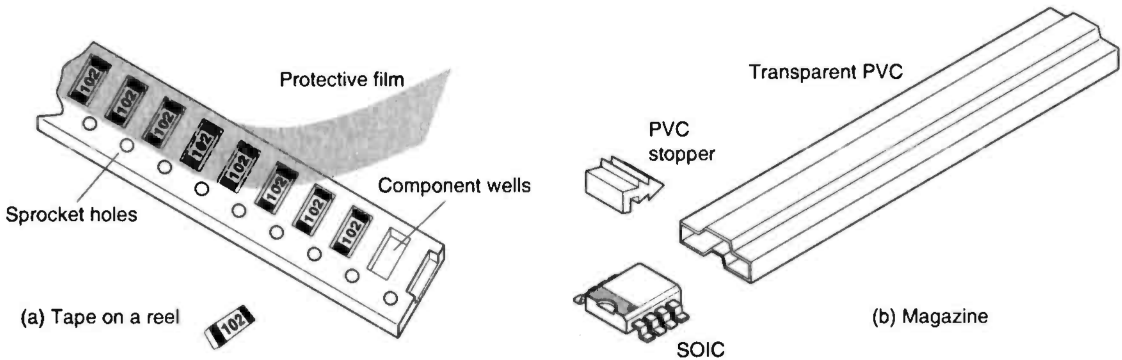


Fig.46 Tape and magazine chip packaging

SMDers have a strong interest in soldering irons and constantly search for the smallest tip on the planet. You will find that a clean, well tinned tip is more important than striving for fineness. Rule one is get some tip cleaner, such as one of those small round tins. The heat capacity is the second most important parameter. We have advocated maximising the amount of copper left on the PCB for board stability, robustness to reworking and heat dissipation from chip power devices. Copper is a marvellous heat conductor and a very fine iron tip which looks just the job can be too sluggish due to lack of heat capacity on all but the smallest pads. The tip has to be held on the job for many seconds and the whole area heats up. With a well suited iron the whole operation is fast and efficient, taking about 3 seconds. A good workhorse iron would be the small Antex 12 watt device (M240SI) fitted with a 0.5mm or 1.0mm tapered bit. The tip which narrows down to about 1mm and holds it for half an inch or so looks good but as soon as it touches the copper pad the tip cools below the solder melting point and we have to wait for heat to conduct along that narrow bit to restore the tip temperature. This type of bit is however useful for IC leads. The tip of a conical or tapered bit has a mass of hot, iron clad, copper behind it. This acts as a heat reservoir which rapidly makes up for the initial heat loss when the joint is first contacted. An important specification which makes this little iron a delight to use is a silicone-rubber insulated lead. The extra flexibility is a real help for SM work where positioning is so precise. Looking at the application of this iron tip to the solder pad under a magnifier shows that it is still just a bit big but this is offset by the good heat capacity.

A temperature controlled iron is the next step up market. I can say immediately that care is required here. It can take an age for the temperature to re-compensate after the heat has been removed from the tip by the solder pad and in practice the temperature controls' only value is in guaranteeing a maximum temperature will not be exceeded. My favourite temperature controlled iron is the Weller EC3100D fitted with an EP105 bit. This bit looks the part and performs perfectly. The iron has a small body and has a 20 watt capacity, and is controlled to within ±2°C. The temperature will need to be set to about 220°C for average work but may need to be higher for large pads. Finally, back to Antex, if you want to do some rework with their bits, a 25 watt XS240SI will be needed. Similarly Weller irons can be fitted with an excellent range of re-work bits. There are many different soldering irons around which do a fine job and some experimentation is suggested.

Solder wire for SM work needs to be fine so as not to overload the joint with solder and 26 or 28 AWG is satisfactory. Low Melting Point (LMP) silver solder wire has become standard for SM. This is a low melting point alloy of tin (Sn62%), lead (Pb36%) and silver (Ag2%). It has a solidus point of 178°C and liquidus point of 179°C. Standard 40/60 will work of course, but the LMP silver loaded alloy is a little more gentle, putting less thermal stress on the chips. Although this LMP wire is cored with a mildly active rosin I find it a little lacking in flux in all but perfect situations. An aerosol of rework flux or a syringe of flux paste should be kept within reach. A bobbin of the fine de-soldering wick, the narrow version specially made for SM, is a natural and necessary companion to this.

During the soldering operation the flux is heated to around 200°C when it produces the familiar smelling fumes. Solder cream/paste produces copious quantities of smoke also. Working with SMDs means closer proximity to the soldering operation and there is a higher chance of inhalation of fumes from the flux. Fluxes are produced from rosin acids and are therefore mildly acidic. Repeated inhalation is not advised and can produce noticeable discomfort in some individuals. It is becoming the custom to use some form of smoke absorber even for phototyping and hobbyist operations. Several makes of bench standing smoke absorbers are now available at reasonable prices. They have replaceable filters and a small mains operated fan to draw the smoke away from the work area. This is a valuable addition to the SMers kit which takes up little space and helps to make a pleasant working environment.

In producing an SMD circuit the most demanding aspect is that of controlling the chip components, holding them down and keeping them in alignment and indeed not losing them when they flick out of control. A small work station makes a useful protected area and is almost essential. This is also useful as a portable work area for use away from the workshop. The work station mentioned earlier is ideal for this purpose and I think some constructional detail would be helpful. The dimensions are given in Figure 47.

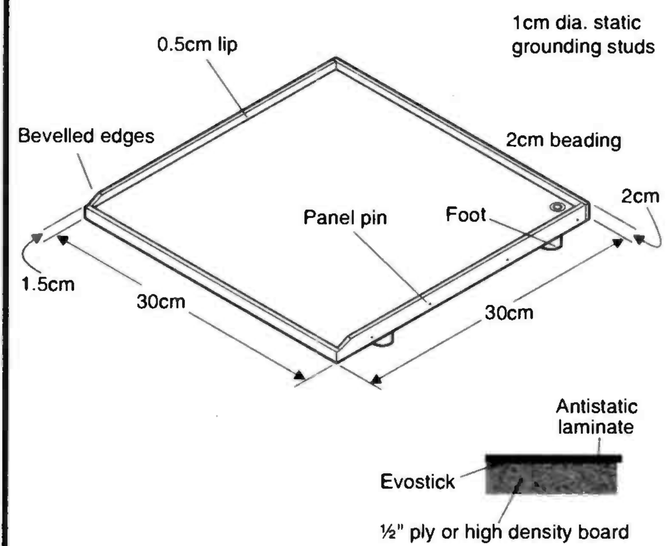


Fig.47 Construction of a small SM workstation

To construct this, prepare the baseboard from, ideally, high density chipboard or half inch plywood. Get it all squared up so that there will be no gaps at the side. Apply a layer of antistatic laminate, if you can get hold of it. I find a light green colour to be the best for SM, the grey shade is the same colour as some COG chip caps and does not improve visibility. This material has a surface resistance of 10^8 ohms per square. It is referred to as static dissipative, as it will disperse charge but will not short out a working circuit. A 10mm press stud placed at the top right hand corner will allow a ground connection and a wrist strap connection if you are working with expensive static sensitive devices. The laminate is bonded to the base board with standard contact adhesive.

Coat both surfaces and let them dry for 30 minutes. Then put them together, you've got to get this right first time because once contacted it won't move. Allow for slight misalignment and plane back to the baseboard to get a square edge. Now apply the beading which should be neatly cut and sanded. Make fancy joints if you can, but simple butt joints are adequate. Plane down the front piece to give just a small retaining wall and chamfer the sides to meet it. Long panel pins are adequate to hold the sides but chipboard screws give a more secure job if a small pilot hole is drilled first. Four rubber feet near the corners will allow the work station to sit securely on the bench without rocking. Finally paint it with clear varnish taking care not to get any on the antistatic surface. The laminate can be maintained and kept clean with proprietary antistatic cleaner.

3. Chip Control

Having produced our protected work area let's home in on the final assault on the live wire detector project in more detail. Placing a 1206 chip resistor on its solder pads will soon reveal the need for some control over the thing. The slightest movement of the PCB knocks it out of place. If you try to solder one end without holding it down it will do some strange things. As the solder wets the end of the chip it will be pulled into the centre of the solder blob or in some cases it will stand on end. This latter effect is known as "tombstoning". It has other pet names like the Stonehenge effect, the Manhattan effect and drawbridging. The effect is most prevalent in production situations using solder paste but the cause is the same. When the end of the chip is wet by the solder the surface tension is sufficient to pull it into the vertical position. The density of an SMD is much lower than lead based solder so it floats on the molten pool. Since only one end is wet and the surface tension pulls it in all directions it moves to equalise the forces by tombstoning. This is illustrated in Figure 48.

The effects of surface tension are not all bad. Surface tension forces are involved with the very important process of spreading and wetting. In reflow soldering the chips float into alignment with the solder pads as a result of surface tension. Similarly self alignment of chips on solder-masked PCBs is a surface tension effect. To avoid tombstoning and because of the

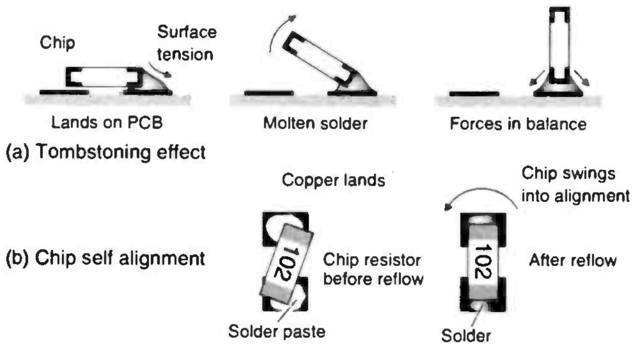


Fig.48 Surface tension effects encountered during reflow soldering

light weight of chip components it is essential to hold them in place whilst hand soldering. The simplest method is to nudge the chip into the required position with a toothpick and then pin it down with same whilst one end is soldered. The toothpick, being made of wood, does not slip on the surface of the component as the hard point of a pair of tweezers might. This is the anchor-one-end method and it works quite well but after soldering the free end it is sometimes necessary to re-solder the anchor end to get a good joint. This subjects the SMD to more thermal stress and often excessive solder loading into the joint. It is therefore not the best soldering technique in that it is difficult to solder correctly in this way. To do the job right one hand holds the chip down, one hand holds the soldering iron and another hand must hold the solder wire so that it can be applied to the joint at the same time as the iron tip. This way the whole joint is rapidly enveloped with oxide beating flux. This is the correct way to solder and requires three hands. You haven't got three hands! Putting solder on the iron and carrying it to the joint is bad because the flux vaporises in the second it takes to get there. Without a protecting coat of flux the solder arriving at the pad is already oxidising at the high temperature and becomes a poor wetter. Molecular layers of oxide already on the copper tracking and on the chip ends have no acidic flux to dislodge them. The incoming solder rests on the surfaces and a dry or unsound joint results. Therefore a device is required to hold the chip in place whilst applying the iron with one hand and the solder with the other. This requires two hands. You have got two hands. I have seen many designs for SMD assembly jigs but the one shown in Figure 49 is very satisfactory.

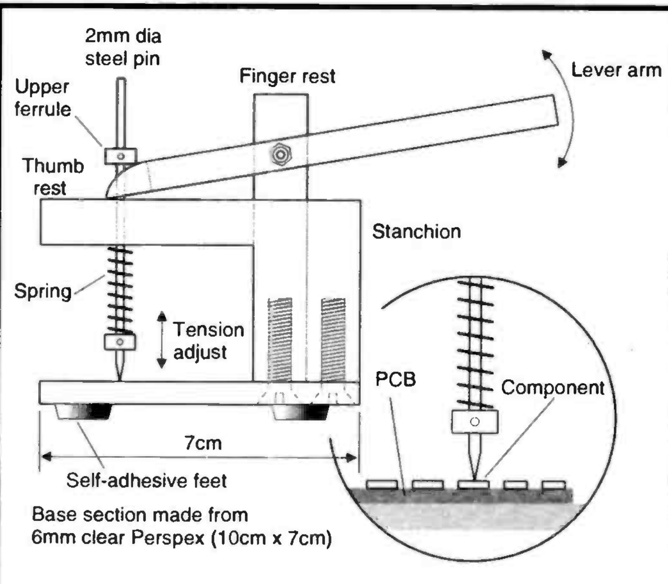


Fig.49 A well designed SMD assembly jig

This was developed for professional prototyping laboratories and educational establishments and is a real help to the amateur constructor. It will take PCBs up to several centimetres and is ideal for most SM projects. Splitting a large project into smaller functional units has many advantages. This will also help in the unlikely event that you run out of jig capacity. To use this little jig the PCB is placed in it with the target solder pads under the retaining pin. This pin is lightly spring loaded and is machined to a flattened point about 0.5mm diameter. A thumb placed on the thumb rest will steady the jig which is quite small and light, whilst the lever arm is operated with the fingers. The chip component is placed under the pin with a pair of tweezers and is held firmly in place by the pin on releasing the lever arm. The chip can be partly released to allow it to be nicely aligned using the tweezers. To lubricate the pin a non-silicone grease gives much smoother operation, particularly if Perspex is used as a construction material. The jig has other uses: it's a good way of holding a circuit whilst working on it, adding Kynar wire links, testing, and so on. I cannot recommend this little assembly jig too strongly. It makes SM work so much easier and once you have tried it you will wonder how you managed without it.

4. The Perfect Joint

Having the capability to solder, you can concentrate on making the perfect joint. The preparation of a flux coated PCB has been mentioned. A PCB prepared in this way has a high surface friction coefficient and it will be easier to control the chips even with the jig. The finer solder wires are a bit lacking in flux content and the extra flux on the PCB comes in handy. If all is clean and ready with the chip in control in the jig, apply a clean and well tinned iron tip to the solder pad, not the component. After an almost imperceptible delay of about half a second touch the pad close to the iron tip with the solder wire and remove almost instantly. Keep the iron in place and let the solder spread out wetting the chip contact area. Finally remove the iron tip. The complete operation should take less than 3 seconds. The solder loading should look like Figure 50(d). In practice it is more likely that excessive solder will be added rather than too little. Use your big magnifier to observe the action at close quarters.

Excess solder in the joint results from leaving the solder wire in contact for too long. This may seem obvious but such a small quantity of solder is required that some skill has to be developed to just touch the joint for the very short time needed. The other reason for excess solder loading is the result of poor solderability when more and more solder is pushed into the joint until finally wetting occurs. Here the need is really for more flux, but this comes with the solder and so more wire is pushed onto the job. A quick spray with rework flux from an aerosol or flux paste from a syringe will often sort matters out.

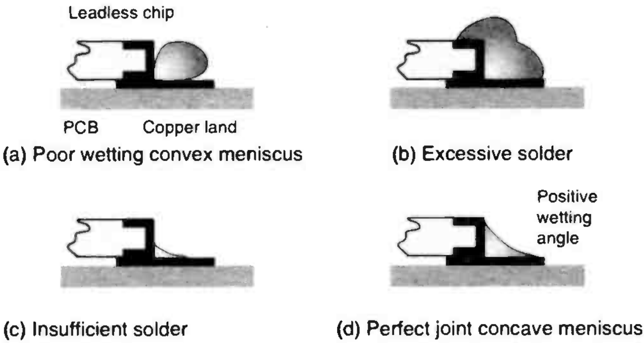


Fig.50 The perfect SM soldered joint

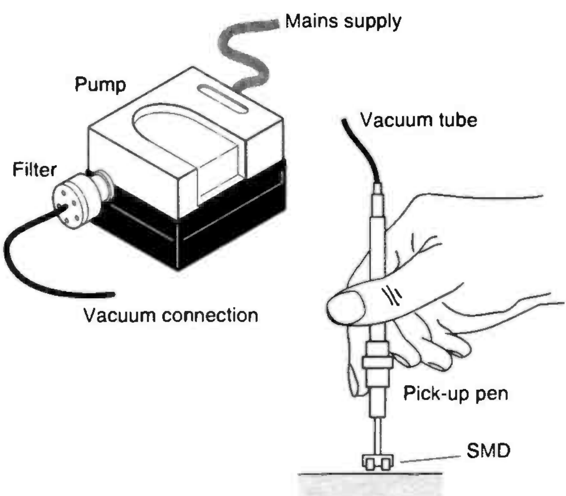


Fig.51 An effective vacuum pick-up system

Excess solder should be removed with solder wick. If you shop around you will find specially fine wick designed for SM work. When population is complete clean excessive flux, and other deposits, from the circuit with PCB cleaner and apply a protective solder-through coating. Following the advice above should lead you to a neat implementation of the audio amplifier.

5. More Gadgets

Another gadget which is popular with SMDers is the vacuum pick-up pen. Industrial automatic pick and place machines use a small vacuum nozzle, turning the vacuum on and off to pick up and release the chip. Such machines are computer controlled and the chips can be placed very accurately in preprogrammed positions. Hand operated vacuum pens are readily available. The simplest types have a manual suction bulb and they represent the lowest cost approach. The more expensive types have a vacuum pump, not unlike a fish tank air pump and these are very useful indeed. The vacuum is controlled by the forefinger on a small hole. They come into their own if several components are to be placed on a PCB ready for reflow soldering with solder paste. This device is excellent for sorting or counting of SMDs. Vacuum tweezers are better for picking up larger ICs than ordinary tweezers. Tiny rubber sucker attachments are available for this purpose. They also have uses in rework operations which will be discussed later. An example is shown in Figure 51. The in-line filter will protect the pump from flux residues which can damage the rubber in the pump.

When you are completing a project, prototyping or designing a PCB layout it is very convenient to have several common components within easy reach and possibly an SMD junk box. There is ample room on the work station to store several SMDs if suitable small containers can be found. Some ingenuity is required here and one solution comes from biochemical laboratory suppliers in the form of multi-well test chambers. These make a great bench-tidy or junk box. Typically they contain several flat bottomed wells under 1cm in diameter. This is large enough to pick chips out with a pair of non-curved tweezers. One particular sample measures 12.8cm by 8.5cm and contains 96 wells. They are very stable and won't easily topple over. In one of these you can store a full set of E12 resistors or capacitors with many wells to spare for discrete devices or junk components. Remember that chip capacitors are not marked so if you drop one into the wrong well, well! A soft sponge-rubber plug stuffed into each chamber with your tweezers will act as a good stopper. Test for a rubber that's smooth and doesn't pull the chips out with it. This type of bench tidy is illustrated in Figure 52.

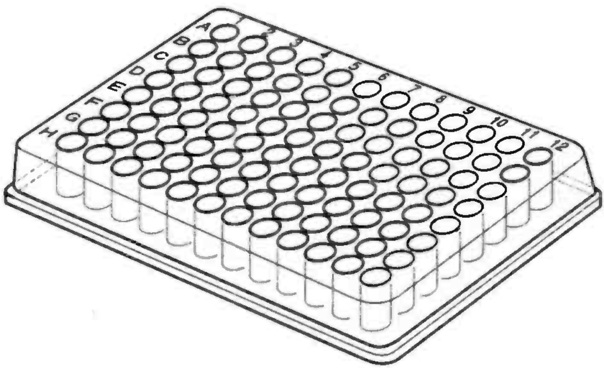


Fig.52 A useful bench-tidy for SM chips

Inter-wiring SM modules and on-PCB links can soon become unwieldy with normal hook-up wire. This type of wire requires contact pads which are out of proportion to the whole design and a waste of space, besides looking unsightly. The ideal wire to use for SM work is single core (1/0.25mm) Kynar wirewrap wire. It is solid silver plated and has excellent solderability. The thin Kynar insulation (300V) strips cleanly and easily without stretching. The small side cutters offered for SM work are very sharp and will do this for you. An important feature is the lack of melting back during soldering. A variety of colours is a great help. Where a flexible connection is required look around for some fine multistrand wire and

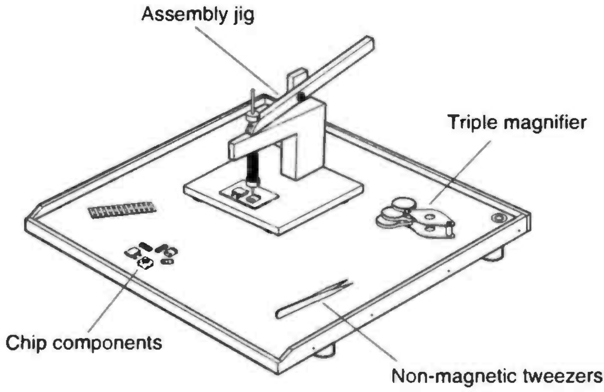


Fig.53 A workstation set-up for small SM projects

miniature SM connectors. Summarising this section, a good set-up for population and commissioning of an SM project will boast several gadgets which together make the practice a great pleasure and the results a work of art to be proud of. An example of a basic set-up is shown in Figure 53. Soldering iron, re-work bits and other support hardware can be conveniently placed close to the work station.

Chapter 6

ADDITIONAL METHODS

1. Solder Paste and Reflow

Solder paste has long been associated with surface mount construction and rework. Its use for hand working is a matter of personal preference. One application where it has a definite advantage is for board population where a solder mask is present. Here a small blob of solder paste is applied to each solder pad, about one-third its area. The chip components are then put in place and the residual tackiness of the paste keeps them there. The whole PCB is then placed in a reflow oven. The temperature is now increased to about 30°C to 40°C above the alloy melting point for less than 10 seconds. At the reflow temperature the paste decomposes and the solder coalesces, wetting the components' solder contacts and the copper lands. A great advantage of this process is that the chips can be placed on the solder pads with little accuracy. At the reflow temperature the chips float on the solder and are pulled into alignment with the printed solder mask. Hot air can be used to reflow the solder paste on an individual basis for a small prototype. It is not unknown for a small PCB to be placed under a domestic grill to reflow it, but extreme caution is required to get even heating without hotspots. Similarly hot air paint strippers are a source of suitable heat but again great care please, and no complaints if it goes wrong. This is the beginning of a new age of amateur construction and experimentation and improvision is the name of the game. Reflow ovens suitable for small scale batchwise and prototyping use are available from specialist SM suppliers.

A non-solder masked project can be reflow soldered provided the copper land area is kept small and suitably shaped. Ideally this will be a small rectangular pad with a narrow track running to it, not a vast expanse of copper as preferred for hand soldering (Figure 54).

At reflow the solder will, more or less, be contained by the pad and give good chip alignment. Of course a spot of glue, possibly on a dummy land between the pads, will hold the chip

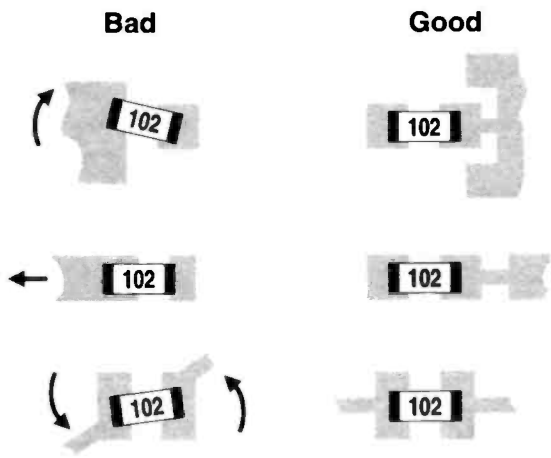


Fig.54 Non soldermasked lands for solder cream

in place whatever the pad shape. Rework of glued-in chips is not so convenient. SM adhesives of various types are available for this purpose. Hand operated dispensers which will take a syringe of solder paste or adhesive can be obtained from SM suppliers. They are easier to use than manual syringes because they have lots of mechanical gearing to make ejection of the paste easier. Solder paste for dispensing has a viscosity of 350,000 to 450,000 cP (mPa s), i.e. very thick. Population of non solder masked projects with solder paste is a viable technique for the amateur. Here reflow is easily achieved with one of the many portable butane soldering irons now sold widely, fitted with a hot air blower. The technique is to approach the solder joint slowly from several centimetres, gradually building up the temperature to the reflow point when the bright solder appears at the ends of the chip. Use a circular motion for larger chips in order to cover the area evenly. As soon as reflow starts, stop your approach and remove the heat as soon as reflow is complete. In fact the heat can be removed an instant before reflow is complete since the heat in the system will keep things going for a time. Solder paste is also useful for rework which we will cover later.

You might have guessed that there would be more than one type of solder paste. Yes, nothing is ever that simple. Overall it is dark grey in colour as you might expect. Its consistency can vary from a moderately stiff paste for dispensing through syringe tips, to a very stiff paste for silk screen or stencil application with a viscosity up to 800,000 cP. The flux is rosin based but it can be straight rosin, mildly activated, or activated. Water soluble and low residue versions are also available. The third global specification is the type of alloy. Most are lead/tin/silver but other metals can be added such as bismuth. The proportion of metals in the alloy determines the melting point. These are, in fact, eutectic compounds so that only certain ratios are possible. The metal content of a paste varies from 85% to over 90% and for dispensing, 85% total metal content is the norm.

For hand working and general purpose use you won't go far wrong with a 62/36/2 alloy. This is Tin (Sn) 62%, Lead (Pb) 36% and Silver (Ag) 2% which has a liquidus point of 179°C. A suitable flux would be a mildly activated rosin type and the formulation needs to have a low viscosity around 400,000 cP as dispensing from a syringe is the best way to apply it. Looking a little closer we find that the organics in the formulation other than the flux have an important effect on the behaviour before and during the reflow cycle. For fine pitch work a "restrictive flow" paste is recommended. Here the paste stays on the spot and does not slump near the reflow temperature, reducing the risk of bridging the leads. A normal non restrictive paste will slump and spread. For hand working it is a matter of personal choice. The solder particles in the paste are preferably spherical as this minimises the oxide content. Particle diameter varies from 45 to 75 microns, the former being more suitable for fine pitch work. Some manufacturers refer to their high quality products as solder creams.

Solder paste for commercial use can have a limited storage life, especially if stored above 10°C. Refrigerator storage is normally recommended but certain products are less critical in this respect. Remember that solder cream contains lead in a highly divided form and is therefore best considered as poisonous. Handle it with great care and if you do come into contact with it don't eat your sandwiches before washing your hands with a strong detergent.

A simple balling test gives an idea of the quality of the solder cream. To do this place a small quantity of paste, about 0.2cc, on a copper-free area of PCB – for example, the reverse side of a single sided fibreglass PCB. Hit this with a hot air blower as though you were making a solder joint and watch how it behaves. It should end up as a single bright ball of solder. Any greyness or surface features indicate high oxygen content and if lots of little solder balls remain it is not a good cream.

Two types of dispensing tips are commonly used for solder cream and adhesives – the tapered, all polypropylene type which is about 1.5″ in length and the blunt end stainless steel type Both have Leurlock fittings to standard syringes. The tapered types have lower tendency to block. A specially made dispensing gun will have a suck-back feature which helps to deliver a precise quantity of solder cream.

2. Rework and Repair

In approaching an SM circuit with a view to troubleshooting and repair we are confronted with the problem of measuring voltages, following signals, and inspecting as in a TH circuit but on a much smaller scale. Meter probes need to be smaller and there is a greater risk of accidental shorting. Joints are smaller and inadequacies in soldering are much more significant. Removal of defective components, without PCB damage, is not difficult and if tackled with appropriate equipment it is probably more convenient than removal of a TH device. Again, an advantage of SM is the fact that for simpler circuits at least, everything is laid out before you on one side of the PCB.

If you are designing the PCB it is a good idea to build in a few significant test points to avoid having to contact SOIC leads with a probe. In any case some fine probes will need to be engineered and the simplest way is to file or turn down some standard probes. For digital systems there are IC test clips but unfortunately there are several types of ICs.

Checks for mechanical failure are often the first approach to fault finding and indeed things that move, plugs and sockets, relays are often the culprit. Over dissipation, voltage breakdown and the like are design faults and more likely to appear on project circuits rather than commercial products. Component failure is associated with product age, particularly if the product is used in an unfriendly environment. High temperature and humidity, frequent power on/off cycling and operation near its design limits will increase ageing. In any case inspection followed by component replacement is the usual scenario.

The array of magnifying aids described previously will be needed for soldered joint inspection. Joint failure is likely to be the result of poor formation in the first place, but temperature cycling or direct mechanical stressing can fatigue a joint as described in Chapter 5. Adding more flux and re-soldering is the obvious action, removing excessive solder as required. Dealing with components is another matter and you may need to be inventive, particularly if not fully equipped.

Chip capacitors and resistors have very good tolerance to mechanical stress within their design limits. However, resistors can lose their ends in extreme conditions of bending. Chip capacitors of the COG type are very robust. Higher dielectric constant materials, particularly Y5V/Z5U, are quite brittle and can simply crack in a very visible way, but sometimes a crack is not so obvious. The end contacts are the weak link as far as mechanical stress is concerned and they can become wholly or partly severed. This is more difficult to detect. The reliability of tantalum capacitors has improved greatly but dead shorts or loss of performance should not be ruled out in cases of mysterious behaviour.

Passive components are the easiest to rework, but both ends of the chip must be melted at the same time before the chip can be lifted off its pads. If you are dealing with, say, a 1206 chip resistor, two soldering irons could be used. One problem here would be the presence of an adhesive in which case a slight twist in the plane of the PCB will fracture the bond. A similarly crude method is to fracture the chip with side cutters. Each half can then be conveniently removed with iron and tweezers. A more satisfactory approach is to use a hot air blower. Get ready with the tweezers, approach the chip slowly and as the solder melts pick the chip off. Next clean up the pads with solder wick, flux them, and solder in the new chip.

Alternatively put a couple of dabs of solder cream on the clean pads and reflow the new chip in place with hot air (or a soldering iron). Servicing with hot air can be problematic on compact circuits, for instance, melting the solder on adjacent chips. It may be necessary to construct a screen with a little PCB or aluminium foil. With purpose-built hot air rework tools some very fine nozzles can be used, reducing the need to screen. A vacuum pen is very handy for picking off de-soldered chips and a delight to use. Less precision is needed as it will just snap up the chip from any angle of approach. Normal tweezers have to be positioned to get the right grip and they often slip on the smooth surface of the SMD.

Several companies make a series of rework tips for fitting to their soldering irons. These are convenient to use and are almost essential for SM work. Coupled with a vacuum pick up pen these rework tips ease servicing. Two points are worthy of note however. First you need to change the tip to remove a different type of chip and a dedicated iron for your most used rework bit(s) is a practical solution. Secondly, the space to get the rework tool to both ends of the target chip is sometimes restricted. Some makes of rework bit are a little slimmer but there are price differences. Some rework tools suitable for the more popular chip components are shown in Figure 55.

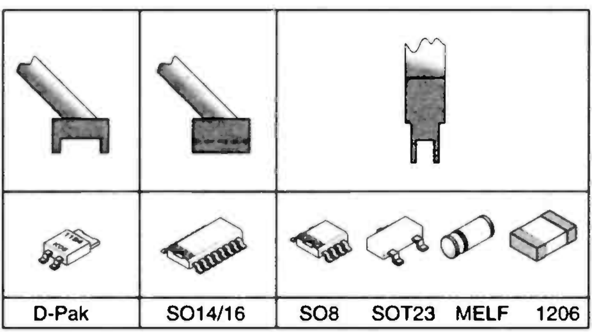


Fig.55 Soldering iron rework bits

Removal of an integrated circuit creates the biggest problem for the service engineer but again happiness increases in proportion to money spent on gadgets. Fortunately SOICs up to say 16 pins can be managed with a hot air blower. Circulate around the chip until the solder melts and then pick it off with a vacuum pen or tweezers. Clean up with solder wick and reflux as before. In this case it is probably best not to remove any solder before the chip is removed as it acts as a heat store between hot air passes. A less sophisticated method is to remove as much solder from each pin with solder wick and then bend each pin back slightly until the chip is free. We SMDers generally do not use solder suckers. The shock produced is regarded as unhealthy for our little ones.

Another low tech method consists of threading a loop of enamelled wire under one edge of the chip. This is then pulled under the pins as they are melted. Cutting off fine pitch leads with SM quality side cutters has also been advocated. The pin ends are then removed individually.

The two soldering iron method is adopted by a number of equipment manufacturers in the form of heated tweezers. These have replaceable tips suitable for a wide range of SMDs from resistors to PLCC chips. They certainly make chip removal easier and can tackle some of the larger ones. The tips melt the solder and grip the package at the same time. Some of the professional rework tools have a vacuum sucker in the centre which lifts the chip as the solder melts. This is a neat solution to the difficult problem of picking up the chip when it is enclosed on all sides with a hot rework tool. A rework tool of this type is almost essential for the very large items like the 144 pin QFP chips. Such a chip is probably more easily dealt with using a hot air rework head supplied with the higher quality rework stations. In hot gas types the heat is directed to the leads by a set of very fine tubes and as the solder melts a vacuum sucker picks up the chip. More sophisticated repair work sometimes involves heating the whole PCB up to a certain temperature approaching the reflow temperature of the solder so that when a solder joint is melted there is minimal temperature difference and less thermal shock. The point is not to attempt a repair on anything that you think may need this kid-glove treatment. Some time spent in rework practice with a scrap circuit board will be useful.

Repairs to damaged tracking is possible with adhesive copper tracking. This is aimed at the specialist and as it is very fine, considerable skill is needed.

3. PCB Protection

However pragmatic your approach to design, population and completion of an SM project, it is in your interest to work neatly and to consider the electrical and physical requirements of the chips. Protect them from moisture ingression, from dust which may be conductive and from stress forces resulting from PCB bending or excessive temperature cycling. Apart from all this, a neat layout is pleasing to the eye and adds to the satisfaction of completing a piece of artwork.

Many fluxes have a degree of acidity and after being subjected to soldering temperatures, when partial chemical breakdown and combination with the solder occurs, are conductive to some degree. It is therefore highly advisable to remove them from the PCB. Solder pastes can be of the low residue type but some can contain acid donor salts like zinc chloride and other compounds which can interfere with circuit performance. For example, the conductivity of the flux residue can vary with humidity. Whatever type of solder paste is used the circuit board should be cleaned. Proprietary cleaners are alcohol based or 1.1.1 trichloroethane based. All work excellently but the chloroethane type dries very fast. This latter type is best used in a well ventilated area.

After the circuit has completely dried a protective coating can be applied. The simplest is a clear solder-through lacquer. This is adequate for most purposes and will keep your circuit looking good. A neutral rosin based flux applied from an aerosol is not too bad as a protective coat and certainly makes rework easier. Silicone based conformal coatings are thicker and offer more protection. They can be solvent removable or solvent resistant. Before applying a conformal coat, adjustable items like trimcaps and presets may be protected with a peelable mask. This comes in the form of a latex cream which cures to a strong rubber, it can be peeled off after the conformal coating has dried. Test points can likewise be protected. It is probably not a bad policy to protect all variable devices in this way whether further adjustment is needed or not. The invading conformal coat may alter the electrical value of the device.

And finally, I have introduced many gadgets which make SM work effective but you should not be deterred by this. Very satisfactory results are possible with minimal equipment. The most important decision is to have a go.

Appendix 1

E6, E12, E24 VALUES

E6	E12	E24	$\times 10^1$	$\times 10^2$	$\times 10^3$	$\times 10^4$	$\times 10^5$	$\times 10^6$
1.0	1.0	1.0	10R	100R	1K	10K	100K	1M
		1.1	11R	110R	1K1	11K	110K	1M1
	1.2	1.2	12R	120R	1K2	12K	120K	1M2
		1.3	13R	130R	1K3	13K	130K	1M3
1.5	1.5	1.5	15R	150R	1K5	15K	150K	1M5
		1.6	16R	160R	1K6	16K	160K	1M6
	1.8	1.8	18R	180R	1K8	18K	180K	1M8
		2.0	20R	200R	2K0	20K	200K	2M0
2.2	2.2	2.2	22R	220R	2K2	22K	220K	2M2
		2.4	24R	240R	2K4	24K	240K	2M4
	2.7	2.7	27R	270R	2K7	27K	270K	2M7
		3.0	30R	300R	3K0	30K	300K	3M0
3.3	3.3	3.3	33R	330R	3K3	33K	330K	3M3
		3.6	36R	360R	3K6	36K	360K	3M6
	3.9	3.9	39R	390R	3K9	39K	390K	3M9
		4.3	43R	430R	4K3	43K	430K	4M3
4.7	4.7	4.7	47R	470R	4K7	47K	470K	4M7
		5.1	51R	510R	5K1	51K	510K	5M1
	5.6	5.6	56R	560R	5K6	56K	560K	5M6
		6.2	62R	620R	6K2	62K	620K	6M2
6.8	6.8	6.8	68R	680R	6K8	68K	680K	6M8
		7.5	75R	750R	7K5	75K	750K	7M5
	8.2	8.2	82R	820R	8K2	82K	820K	8M2
		9.1	91R	910R	9K1	91K	910K	9M1

Appendix 2

EIA CODE/ VALUES

10R–91R		100R–910R		1K–9K1		10K–91K		100K–910K		1M–10M	
E24 Value	Resistor Marking	E24 Value	Resistor Marking	E24 Value	Resistor Marking	E24 Value	Resistor Marking	E24 Value	Resistor Marking	E24 Value	Resistor Marking
0	000										
10	100	100	101	1K0	102	10K	103	100K	104	1M0	105
11	110	110	111	1K1	112	11K	113	110K	114	1M1	115
12	120	120	121	1K2	122	12K	123	120K	124	1M2	125
13	130	130	131	1K3	132	13K	133	130K	134	1M3	135
15	150	150	151	1K5	152	15K	153	150K	154	1M5	155
16	160	160	161	1K6	162	16K	163	160K	164	1M6	165
18	180	180	181	1K8	182	18K	183	180K	184	1M8	185
20	200	200	201	2K0	202	20K	203	200K	204	2M0	205
22	220	220	221	2K2	222	22K	223	220K	224	2M2	225
24	240	240	241	2K4	242	24K	243	240K	244	2M4	245
27	270	270	271	2K7	272	27K	273	270K	274	2M7	275
30	300	300	301	3K0	302	30K	303	300K	304	3M0	305
33	330	330	331	3K3	332	33K	333	330K	334	3M3	335
36	360	360	361	3K6	362	36K	363	360K	364	3M6	365
39	390	390	391	3K9	392	39K	393	390K	394	3M9	395
43	430	430	431	4K3	432	43K	433	430K	434	4M3	435
47	470	470	471	4K7	472	47K	473	470K	474	4M7	475
51	510	510	511	5K1	512	51K	513	510K	514	5M1	515
56	560	560	561	5K6	562	56K	563	560K	564	5M6	565
62	620	620	621	6K2	622	62K	623	620K	624	6M2	625
68	680	680	681	6K8	682	68K	683	680K	684	6M8	685
75	750	750	751	7K5	752	75K	753	750K	754	7M5	755
82	820	820	821	8K2	822	82K	823	820K	824	8M2	825
91	910	910	911	9K1	912	91K	913	910K	914	9M1	915
										10M	106

Chip resistor marking: The first two digits are the significant values and the last digit is the number of zeros. For example: 132 is 1300 = 1k3.

Appendix 3

CAPACITANCE CONVERSION TABLES

0.000001µF	=	0.001nF	=	1pF
0.00001µF	=	0.01nF	=	10pF
0.0001µF	=	0.1nF	=	100pF
0.001µF	=	1nF	=	1000pF
0.01µF	=	10nF	=	10000pF
0.1µF	=	100nF	=	100000pF
1µF	=	1000nF	=	1000000pF
10µF	=	10000nF	=	10000000pF
100µF	=	100000nF	=	100000000pF

Appendix 4

A SELECTION OF POPULAR SM AND TH TRANSISTOR EQUIVALENTS

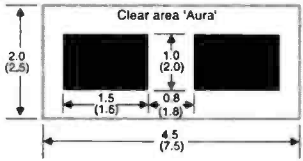
SM Type	*TH Type*	*Polarity*	*Outline*	*Marking*
BCW71	BC107A	NPN	SOT23	H2
BCW72	BC107B	NPN	SOT23	K8
BCW31	BC108A	NPN	SOT23	D1
BCW32	BC108B	NPN	SOT23	D2
BCF32	BC109B	NPN	SOT23	D7
BCF33	BC109C	NPN	SOT23	D8
BCX70H	BC107B	NPN	SOT23	AH
BCX71H	BC177B	PNP	SOT23	BH
BCW30	BC178B	PNP	SOT23	C2
BCX17	BC327	PNP	SOT23	T1
BCX19	BC337	NPN	SOT23	U1
BCW60	ZTX300	NPN	SOT23	AB
BFR92A	BFR90	NPN	SOT23	P2
BFT92	BFQ51	NPN	SOT23	W1
BFS17	BFY90	NPN	SOT23	E1
BFS19	BF184	NPN	SOT23	F2
BSR14	2N2222	NPN	SOT23	U8
BSV52	BSX20	NPN	SOT23	B2
BSR15	2N2907	NPN	SOT23	T7
BSR17A	2N3904	NPN	SOT23	U92
BSR18A	2N3906	PNP	SOT23	T9
BCV27	BC517	NPN	SOT23	FF
BCV26	BC516	PNP	SOT23	FD
BST60	BSR60	PNP	SOT89	BS1
BCX54	BD135	NPN	SOT89	BA
BVX51	BD136	PNP	SOT89	AC
BFQ19	BFR96	NPN	SOT89	FB
BSR40	BSX46	NPN	SOT89	AR1
BFQ17	BFW16A	NPN	SOT89	FA
BSR58	2N4858	N-ch	SOT23	M6
BFR30	BF254	N-ch	SOT23	M1
BFR31	BFW12	N-ch	SOT23	M2

Appendix 4 – Continued

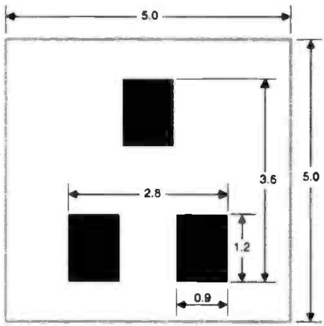
A SELECTION OF POPULAR SM AND TH TRANSISTOR EQUIVALENTS

SM Type	*TH Type*	*Polarity*	*Outline*	*Marking*
BF989	BF960	DG-FET	SOT143	MA
BF991	BF981	DG-FET	SOT143	M91
BF994S	BF964F	DG-FET	SOT23	MG
BST82	BST72A	V-FET	SOT23	O2
BST80	BST70A	V-FET	SOT89	KM

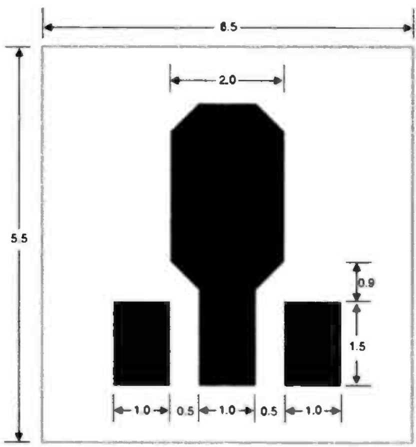
All dimensions in millimetres



Chip resistors and capacitors size 0805 and (1206)



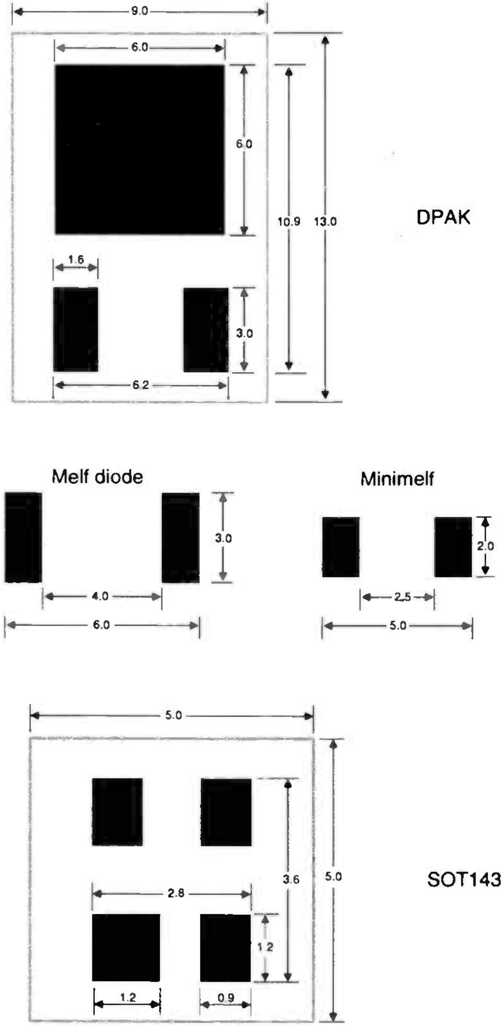
SOT23



SOT89

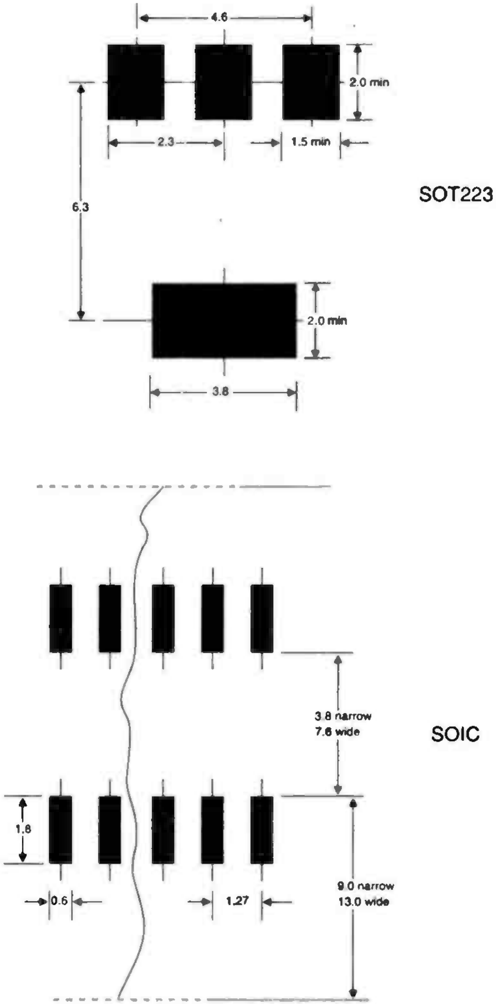
Appendix 5a **Minimum** *footprint areas for hand soldering*

All dimensions in millimetres



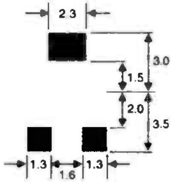
Appendix 5b **Minimum** *footprint areas for hand soldering*

All dimensions in millimetres

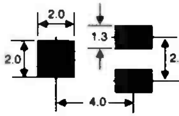


Appendix 5c **Minimum** *footprint areas for hand soldering*

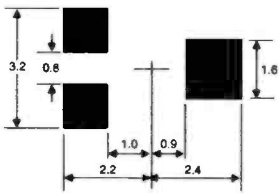
All dimensions in millimetres



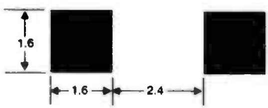
Trim Resistor Type 3204



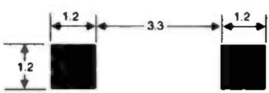
Trim Resistor Type 3315



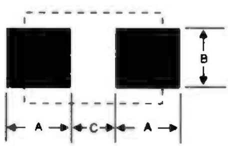
Trim Resistor Type 3130



Sealed Trim capacitor
4mm x 4.5mm sealed



Open Frame Trim capacitor
3.2mm x 4.5mm



Tantalum capacitors

Case	A	B	C
A	1.6	1.7	1.2
B	1.8	1.9	2.4
C	2.4	2.3	2.4
D	2.4	2.9	3.7

Appendix 5d **Minimum** *footprint areas for hand soldering*

1 Passive Devices

Actual size	Type	Size (mm)	Capacitors (max value)			Resistors pD(W)	Inductors Ind Range
			COG	X7R	Z5U		
	0402	1.0 x 0.5	150p	6.8n	33n	0.63	
	0603	1.6 x 0.8	330p	47n	220n	0.63	
	0805	2.0 x 1.25	1n	220n	2.2u	0.1	0.047 - 33uH
	MiniMELF	2.2 x 1.1				0.2	
	1008	2.5 x 2.0					0.01 - 100uH
	1206	3.2 x 1.6	2.7n	680n	4.7u	0.125	0.047 - 33uH
	MELF	3.6 x 1.4				0.25/0.4	
	1210	3.2 x 2.5	10n	270n	470u	0.425	0.01 - 330uH
	1812	4.5 x 3.2	18n	560n	1.0u		1.0 - 1.0mH
	2010	5.0 x 2.5				0.5	
	2220	5.7 x 5.0	39n	1.2u	2.2u		1.2 - 10mH
	2225	5.7 x 6.3	47n	1.5u	4.7u		
	2512	6.3 x 3.1				1.0	
	S1	6.3 x 3.5				1.0	
	S2	10.9 x 5.9				2.0	

2 Terminal Discrete Devices

Actual size	Size (mm)	Outline
	2.5 x 1.25	S-MINI, DSM, USM
	3.4 x 1.4	LL34, MINIMELF
	3.7 x 1.6	6OA2
	5.0 x 2.6	D1F, D0 - 214, MELF, PSM

3 Terminal Discrete Devices

Actual size	Size (mm)	Outline
	1.6 x 0.8	EM3
	2.0 x 1.25	UMT, UM5, UM6
	2.9 x 1.3	SOT23, SOT143
	2.9 x 1.5	IMD, FMT, SC-59
	4.5 x 2.5	MPT, SOT89
	6.5 x 3.5	SOT223
	6.5 x 5.5	CPT, D PAK, E PAK
	10.1 x 9.9	PSD, SQUARE PAK

Notes:
1. Package outlines **excluding** leads
2. Maximum values and ranges are approximate

Appendix 6 Device outlines - actual size

Appendix 7

APPROXIMATE RANGE OF VALUES OF 50V CHIP CAPACITORS AVAILABLE IN THE POPULAR DIELECTRICS

DIELECTRIC	CASE SIZE	RANGE	
		MINIMUM	MAXIMUM
Z5U	2220	680nF	1μF
	1812	200nF	680nF
	1210	100nF	220nF
	1206	10nF	100nF
	0805	10nF	47nF
X7R	2220	560nF	1μF
	1812	100nF	470nF
	1210	10nF	6.8nF
	1206	1pF	33nF
	0805	1pF	22nF
COG	2220	8.2nF	10nF
	1812	2.2nF	8.2nF
	1210	220pF	2.2nF
	1206	1pF	1nF
	0805	1pF	470pF

Index

Please Note

Babani Radio, Electronics and Computer books should be available from all good Booksellers, Radio Component Dealers and Mail Order Companies.

However, should you experience difficulty in obtaining any title in your area, then please write directly to the Publisher enclosing payment to cover the cost of the book plus adequate postage.

If you would like a complete catalogue of our entire range of Radio, Electronics and Computer Books then please send a Stamped Addressed Envelope to:

BERNARD BABANI (publishing) LTD
THE GRAMPIANS
SHEPHERDS BUSH ROAD
LONDON W6 7NF
ENGLAND

Babani Electronics Books

A Practical Introduction to Surface Mount Devices

This book takes you from the simplest possible starting point to a high level of competence in handworking with Surface Mount Devices (SMD's). The wider subject of SM technology is also introduced, so giving a feeling for its depth and fascination.

Surface Mount construction is easier, faster and cheaper for modern mass production, making it simpler to manufacture reliable electronic products at low prices.

Fortunately, the technology is also ideal for constructing small projects. Therefore, hobbyists, students and anyone producing one-off circuits can use SMDs to advantage. Just a few very basic skills are all you need to get started.

Subjects such as PCB design, chip control, soldering techniques and specialist tools for SM are fully explained and developed as the book progresses. Some useful constructional projects are also included.

Whilst the book is mainly intended as an introduction, it is also an invaluable reference book, and the browser should find it engrossing.

BP 411

ISBN 0-85934-411-8

00499>

9 780859 344111

£4·99