An Introduction to SMD

Thijs Beckers (Elektor Netherlands)

The first step is always the hardest, so we're providing a brief introduction to SMT to help you out. Here we introduce you to some of the jargon, pitfalls and packaging so you can hold your own in discussions of this technology, which has become indispensable in the production of modern electronic equipment.

If you want to work with surface-mount technology (SMT), you need to have a bit of basic knowledge. This article explains several terms, discusses some of the things to watch out for, and gives you some pointers for finding more information so you can learn to loosely mention the terms like a pro.

What does it mean?

To start off, let's take two terms that many people tend to mix up: 'surface mount device' (SMD) and 'surface mount technology' (SMT). SMT refers to the technology, which means using components (usually small) that do not have leads designed to pass through holes in the circuit board, while SMD refers to the actual component.

Another term you will see is 'SMA'. This stands for 'surface mount assembly', which indicates that a component is designed for mounting **on** the circuit board, rather than with pins that pass **through** the circuit board. The same term can also be used as an abbreviation of 'surface mount adhesive', which is the glue used to attach SMDs to the circuit board so they don't slide around during soldering.

Jargon

Nowadays SMDs are used in almost all electronic equipment. In fact, there are probably more components available now in SMD form than in 'through-hole' form (with individual pins or leads). Naturally, the reason for this is the extensive miniaturisation of electronic circuitry. SMDs are also appearing increasingly often in the DIY world.

When you use SMDs, you run into a considerable amount of insider terms. In order to get your bearings in the SMD world, you need to know these terms and understand what they mean. Most likely you have already heard the term 'ball grid array', but 'flip chip' is probably less familiar. With the latter technology, the chip is mounted with its active surface facing downward (**Figure 1**), which means that the active surface of the chip can be used directly for the connections. This makes it possible to make a large number of connections to the chip, and they have much lower inductance than with wire bonds due to the shorter distance.

Another recent development is called 'package on package' (PoP). This consists of stacking one chip on top of another one (usually discrete logic and memory), which saves space and keeps the connections short to minimise inductance problems.

When boards are assembled automatically, they must have fiducial marks. A fiducial mark (or simply 'fiducial') is a symbol that is placed on the circuit board. It can be used to determine the position of the board with high accuracy, so that the solder paste can be applied correctly or a pick-and-place machine can place the components in exactly the right positions before the board goes into the oven for soldering.
Incidentally, two standard methods have been developed for applying solder paste: silk-screening (also called screening or stencil printing) and direct printing. In the silk-screening process, a stencil is created with openings exactly aligned to the copper track layout. A rubber squeegee spreads the paste over this stencil, with the result that it ends up exactly where it should be on the circuit board. This method is feasible for series production, but producing a stencil of this sort is far too expensive for making a single PCB. The direct printing method is more suitable in the latter case. This involves using a special 'printer' (similar to an ink-jet printer) to deposit the solder paste directly on the PCB. However, these printers are rather costly.

Reflow problems

There are several common problems with soldering SMDs (including reflow soldering). One of them is called the 'tombstone effect', or 'tombstoning'. **Figure 2** shows the forces acting on an SMD component during soldering. They can cause the component to stand upright on the circuit board instead of remaining flat on the board when it is soldered. Upright SMD resistors resemble miniature tombstones, which is where the term comes from.
The component will rise up if the sum of F_1 and F_2 is less than F_3, or in mathematical terms:

$$M \cdot g \cdot [(D^2+L^2)/2] \cdot \cos(\alpha+\beta) + \gamma \cdot W \cdot \cos(\alpha/2) < \gamma \cdot D \cdot \sin(\alpha+\Phi),$$

where M is the mass of the component and g is the force of gravity.

Terminology

What does all that jargon mean?

There are several causes of tombstoning. For instance, lightweight components are more susceptible to this effect. Relatively long solder pads can also cause this undesired effect, because the portion of the pad that extends beyond the component causes an increased torque (larger value of Φ in **Figure 2**).

Tombstoning can also occur if temperature does not rise uniformly at both ends of the component. If one end is warmer than the other one, the solder will melt first at this end, leading to an undesired upright component. This problem usually does not occur in modern convection ovens, but design-related factors such as screening and cooling surfaces can lead to temperature differences.

Incorrect component placement can also lead to tombstoning, but the main cause is a temperature difference between the two ends of the component, which causes the solder to melt earlier at one end than at the other end.
'Popcorning' is another example of what can go wrong during the soldering process. This refers to a condition that can occur if moisture-sensitive components remain outside a moisture-proof package too long before they are soldered in a reflow oven. The component package can absorb moisture due to its hygroscopic properties. If such a component is heated relatively quickly, the moisture turns into steam, which may create so much internal pressure that the package will crack or burst open.

Another problem is that the component may float on the molten solder and tip over along its long axis as a result. This is particularly annoying with LEDs, since it causes the light to be emitted toward the side instead of straight up.

Standards

Since the 1st of July 2006, electronic equipment marketed inside the EU is not allowed to contain certain substances. This is stipulated by the 'Restriction of Hazardous Substances' directive, usually abbreviated as 'RoHS'. In colloquial language, this is also described by saying that the equipment and components must be 'lead-free'. The fact that a component is 'lead-free' or fulfils the RoHS standard does not necessarily mean that it is suitable for lead-free processing. It only says something about the chemical composition of the product, but not that it can withstand the relatively high temperatures used in lead-free soldering. Consider yourself warned.

A good reference source for industrial standards related to components is the Institute for Interconnecting and Packaging Electronic Circuits (IPC) – see the web link at the end of this article. The standards in the IPC-7351 to IPC-7359 series are especially important for PCB design. They provide information about suitable dimensions, shapes and tolerances of pads for SMDs, so that they provide enough surface area for soldering but not too much (which would create a risk of tombstoning).

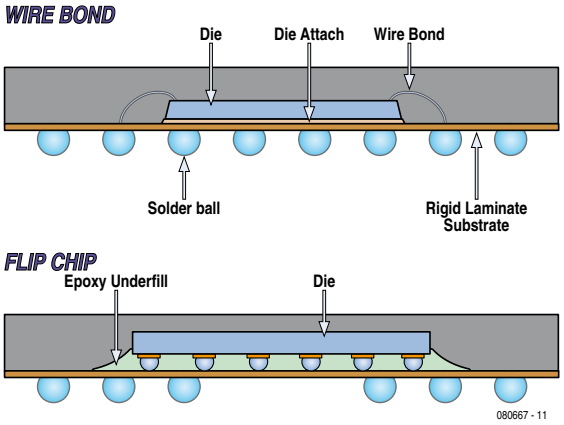


Figure 1.
Wire-bond or flip-chip? Flip-chip technology is often used in processors due to high clock rates.

Packages and packaging

We could fill dozens of pages with information about SMD packages – it's an almost endless subject. Here we recommend that you read through the overview of the most common SMD codes and pin layouts prepared by R. P. Blackwell [1].

With regard to packaging, we can keep our remarks quite short: as SMDs are usually processed by automated machines, it is essential to standardise the containers used

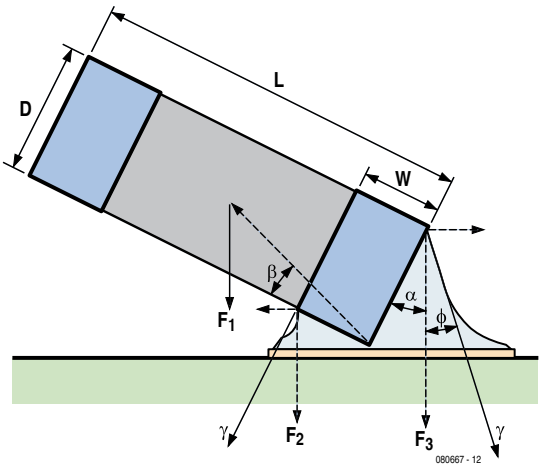


Figure 2.
The tombstone effect is caused by unbalanced forces.

to supply the components to the machines. There are four common types:

- **Tape or reel:** the components are located on a tape that is wound on a reel, just like the tape of an (old-fashioned) tape recorder.

- **Tray or pallet:** components with a small pin spacing or BGA are usually packaged in this sort of container.

- **Stick or tube:** IC with edge-mounted pins are often supplied in a plastic tube to prevent accidental bending of the pins.

- **Bulk:** a large number of the same type of component, which are not packaged in an orderly manner. Often used in the past with large quantities.

Requisites

In the old days, enthusiasts went to an electronics shop to purchase their components. As there are often a large number of package options available now for components, it is simply impossible for an average shop to keep all types of components in stock. Fortunately they can usually supply the desired version on request.

Online shops often have a larger selection, but there is a chance that they do not have the part in inventory and will have to order it from a distributor. In addition, there are usually shipping charges. The really big players, such as Farnell and Conrad Electronics, can usually deliver from stock.

Finally, you need solder paste if you want to solder PCBs with a reflow oven. There is large selection of various pastes, each with its specific properties. The one may have a higher melting temperature, while the other may have smaller solder particles, and so on. See reference [2] for more information on solder pastes.

You can also consult the web links listed below under 'Background information' to learn more about the topics discussed in this article. Once you've digested all this information, you'll be a lot more knowledgeable, and you won't be at a loss for words when the subject turns to SMT, SMD or SMA.

(080667-1)

Internet Links

[1] www.marsport.org.uk/smd/mainframe.htm

[2] www.siliconfareast.com/solder-paste.htm

Background information

www.answers.com/topic/flip-chip

www.ipc.org

www.ami.ac.uk/courses/topics/0229_place/index.html

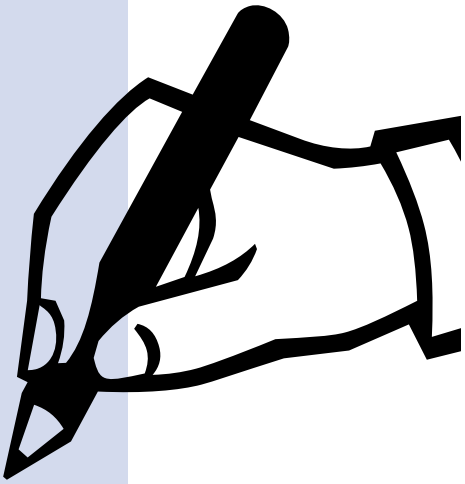
Advertisement

See your design in print!

Elektor Electronics (Publishing)

are looking for

Freelance Technical Authors/Designers



If you have

- *an innovative or otherwise original design you would like to see in print in Europe's largest magazine on practical electronics*
- *above average skills in designing electronic circuits*
- *experience in writing electronics-related software*
- *basic skills in complementing your design with an explanatory text*
- *a PC, email and Internet access for efficient communication with our in-house design staff;*

then do not hesitate to contact us for exciting opportunities in getting your designs published on a regular basis.

Elektor Electronics
Jan Buiting, Editor
P.O. Box 11, NL-6141-AV Susteren, The Netherlands, Fax: (+31) 46 4370161
Email: editor@elektor.com