Reflow soldering of surface mount components

1. Introduction

The soldering of surface-mount components by hand on a printed circuit board, with a little practice, is fairly straightforward. Good, fine, right angle tweezers and a hot soldering iron, along with fine cored solder are a must. There are numerous website and videos available on the web that explain the process very well.

However soldering components by hand is only possible when component pins are accessible…obvious really! In cases where the surface to be soldered is below the components, as is packages where a heatsinking or grounding pad is underneath or on the very edge of the component an oven and other reflow kit is essential. We describe here the process that we use in our group for mounting such ‘awkward’ components.

Reflow soldering is a process that uses a solder paste, a remarkably sticky mixture of powdered solder and solder flux, to temporarily attach components to their respective printed circuit pads. Following this, the assembly is subjected to heat that melts the solder, ensuring that the component is permanently connected to the pad. Reflow soldering is the most common method of attaching surface mount components to circuit boards.

The reflow process melts the solder and heats the surfaces to be joined, with little overheating or damage to the electronic components. In the conventional reflow soldering process, there are usually four stages, called ‘zones’, each having a distinct thermal profile: they are commonly called: preheat, thermal soak, reflow and cooling. The term ‘reflow’ refers to the temperature above which a solder alloy mass is certain to melt, and not merely to soften. When cooled below the reflow temperature, the solder will not flow. Warmed above the temperature again, the solder will once more flow: hence the term ‘re-flow’. In practice, the solder does not ‘flow’ again, but ensures that the granulated solder in the solder paste is at a temperature that surpasses the reflow temperature of the specific solder used.

2. The solder paste

We use a lead free soldering paste with a composition of 96,5% tin, 3% silver, 0.5% copper available in 100g cans and conforming to RoHS requirements, and containing a flux according to EN ISO 9454:2016, 1231; DIN EN 61190-1-3 / IPC J-STD-004B, REL0. It is manufactured by Felder[1] and available as part # RK-10753[2] for €19.02 + shipping from Beta e-store referenced below.

3. The oven

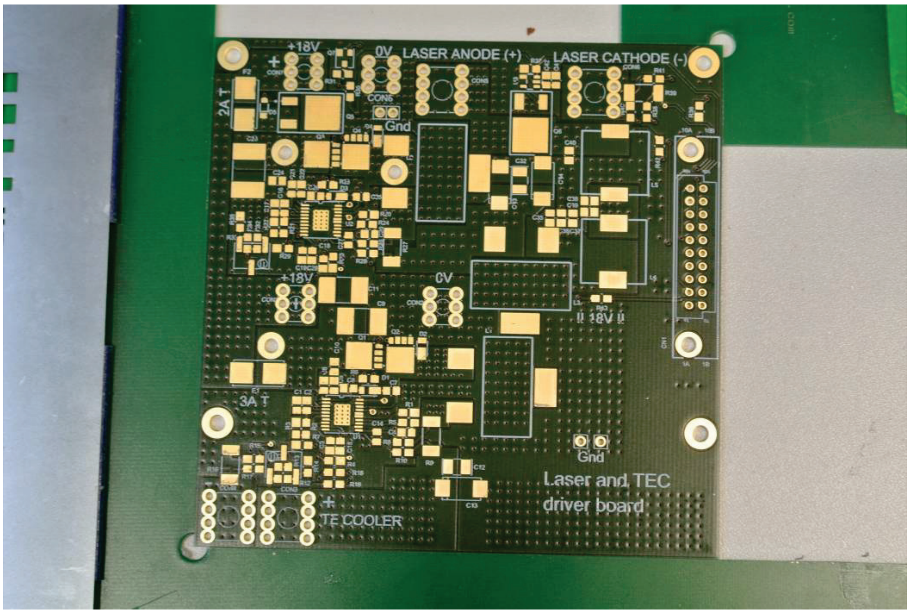
Along with many others, we use a 1.5 kW Severin TO 2034 oven, with capacity of 20 l. This can be purchased from Amazon or again from Beta e-store as part of a Reflow kit V3 basic kit[3] . The oven meets EC Directives 89/336/EEC and 73/23/EEC.

4. The temperature controller

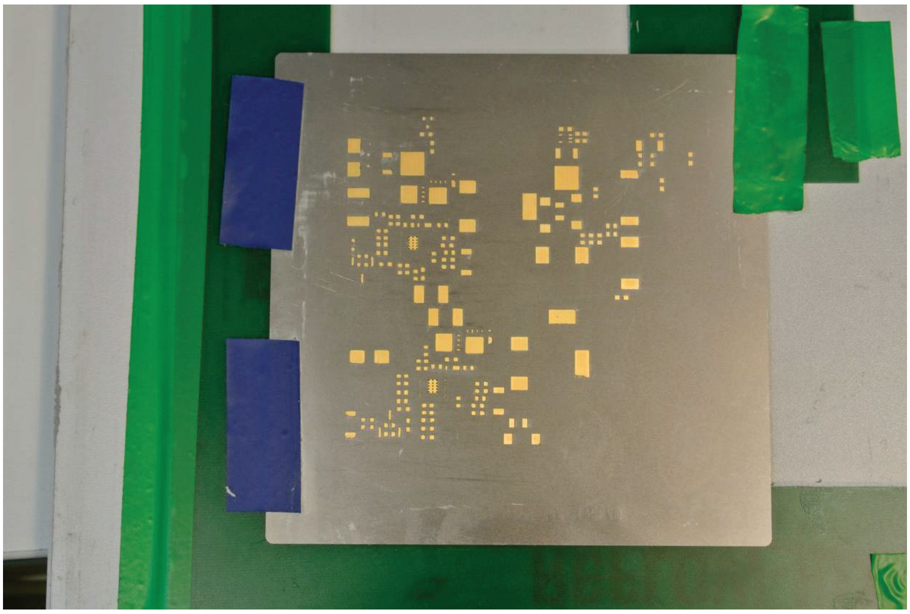
Here we use the Reflow controller V3 PRO available for €186.18 net plus shipping costs as part # RK-10579. This comes with a thermocouple temperature sensor with cable and connector. Again this is available from the Beta e-store[4] or as part of the basic kit.

5. The process

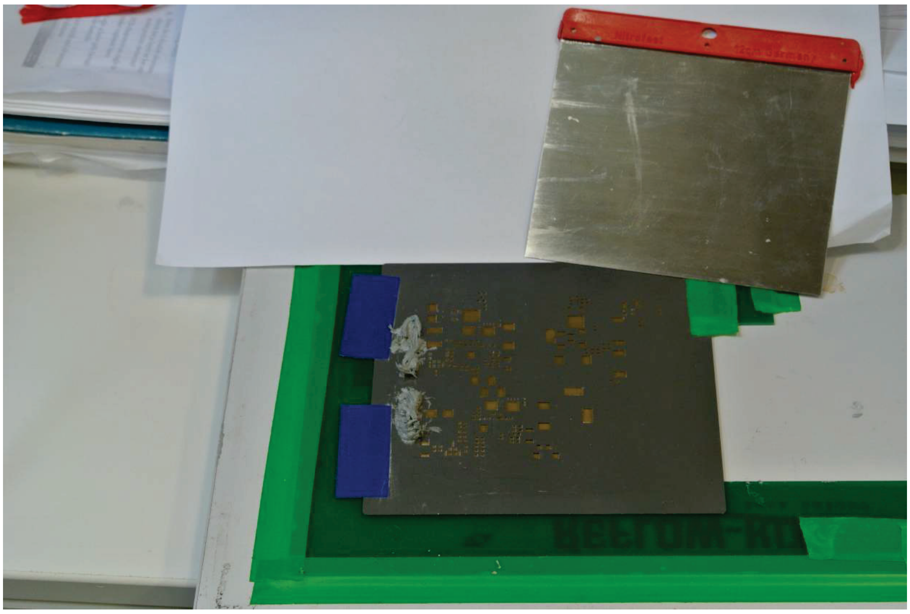
Your printed circuit board (PCB) is first held in a jig consisting of a flat support board that holds two additional pieces of FR4 material. These additional pieces are of the same thickness as your PCB but have 90 degree cut-outs, forming a jig that is securely taped down onto the main support board. These components can be obtained from Beta e-store. Your precious PCB is thus held securely in place by its corners, as shown in the image below. These additional 'corner' surfaces are at the same height as your board. If you require working on a thinner PCB, different corner pieces must be used of appropriate thickness.



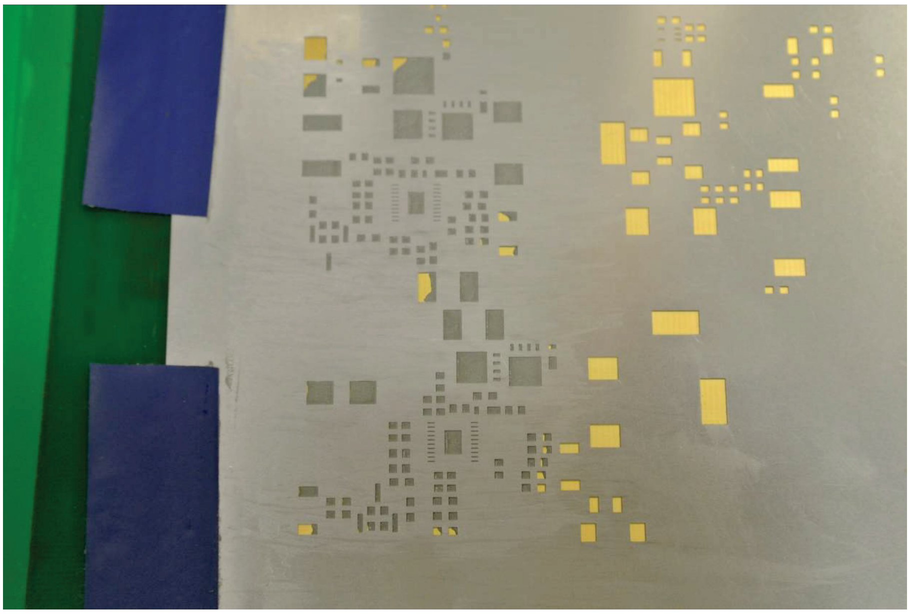
The solder mask stencil, as provided by the PCB manufacturer, is then placed on top of the PCB and carefully aligned with the component pads on your board. It is useful to securely taped down the solder mask only along one edge initially: this can act as a hinge so the PCB can easily be removed. The other edge of the solder mask is also taped down when satisfied with the alignment; the solder mask can then be taped down to the holding jig. The solder mask is designed in thickness and pad size to leave just the correct amount of paste on the pads to ensure that a good solder joint is eventually formed.



Next, spread a generous amount of solder paste onto the stencil. In the example shown below, only a limited number of components were going to be re-flowed, so the paste was concentrated just to the region where these components are to be soldered. A thin metal spreader, as supplied by Beta e-store, can be used to spread the paste through the holes. It is best to press down on the mask as well to stop any paste bleeding under the mask. This is especially important on very small pads with a fine pitch.

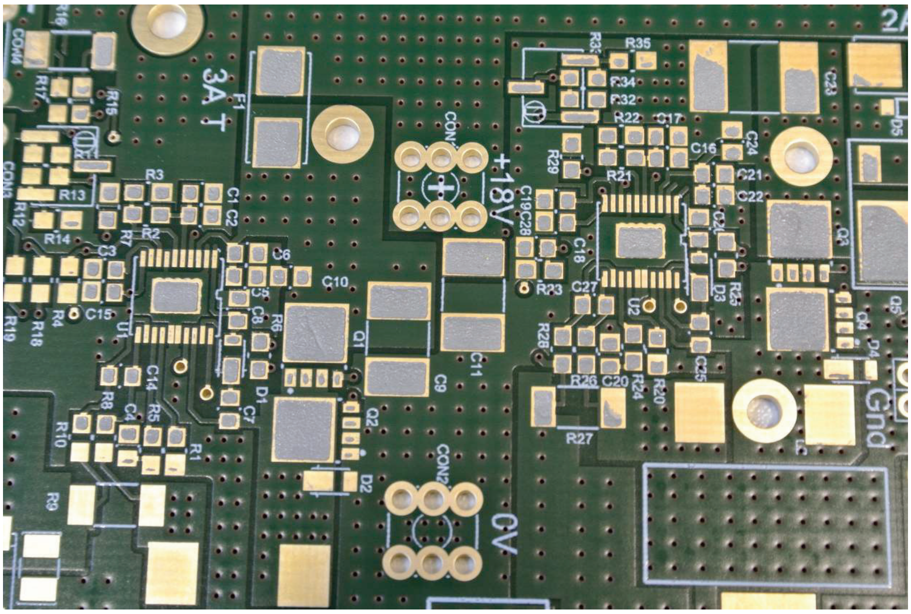


Once the solder paste is spread, the result will be as shown in the image below. The mask is then removed.

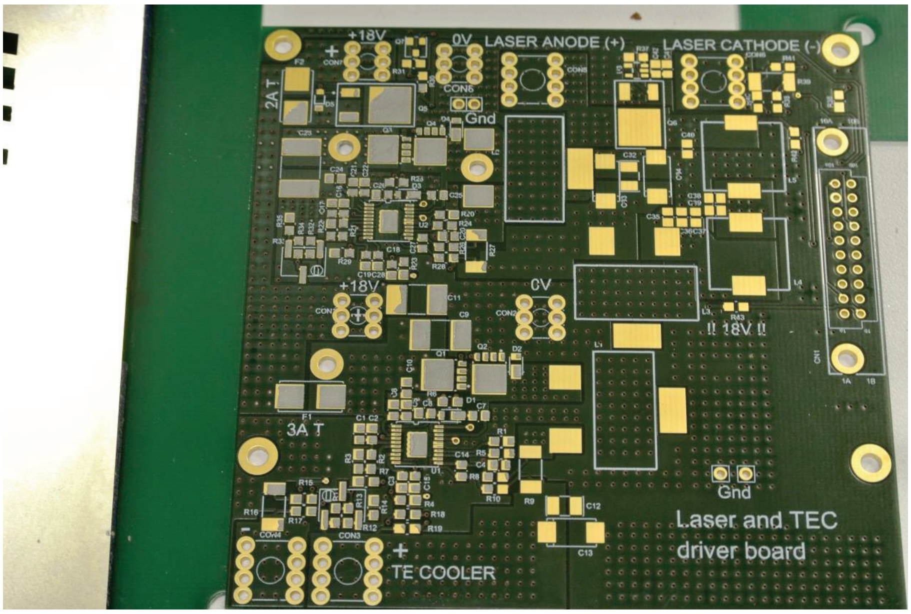


Then it is recommended that you check to see that the correct amount has been deposited on the pads. Of course, some small amounts of solder paste will inevitably remain on the solder mask. If the same mask is used on a number of PCBs with extended periods between uses, any remaining solder paste will dry out, potentially blocking mask. It is therefore a good idea to clean the mask with some PCB cleaner and a brush to clean out the holes.

The image below shows an example of this: the pads on the 20 pin devices have not been coated with any solder paste due to the old solder paste blocking the mask holes in an non-cleaned mask. The smaller the pads are, the worse this problem can be.



When the solder paste has been applied correctly, the result will look as in the image below.



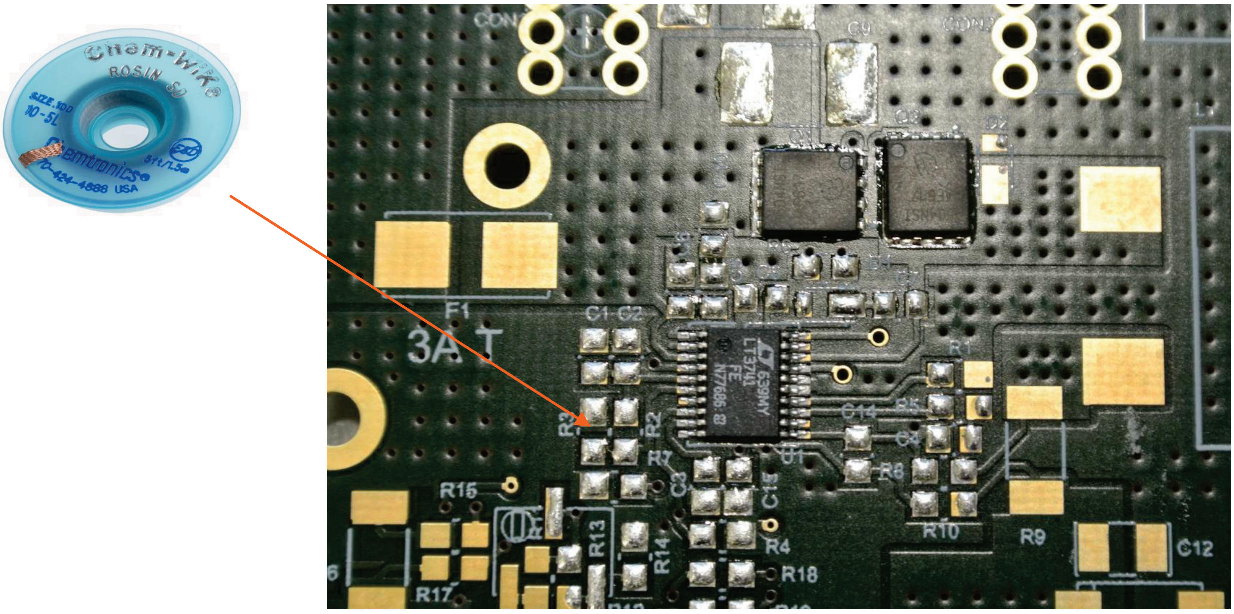
The components can now be carefully placed in position on the PCB. The paste is a little sticky and is able to hold components in place.

However, the greater the number of components, the fiddlier the process is. You may be tempted to use a pocket magnifier, like the one shown on right, to check that everything is positioned. Beware when using this: a component can easily be knocked out of place whilst concentrating on positioning another component. This is the fundamental problem of hand positioning many components on a PCB.

When satisfied all the components are in the correct position, the temperature controller's thermocouple is attached to an unused portion of the PCB with a piece of heat resistant tape (eg. Kapton).

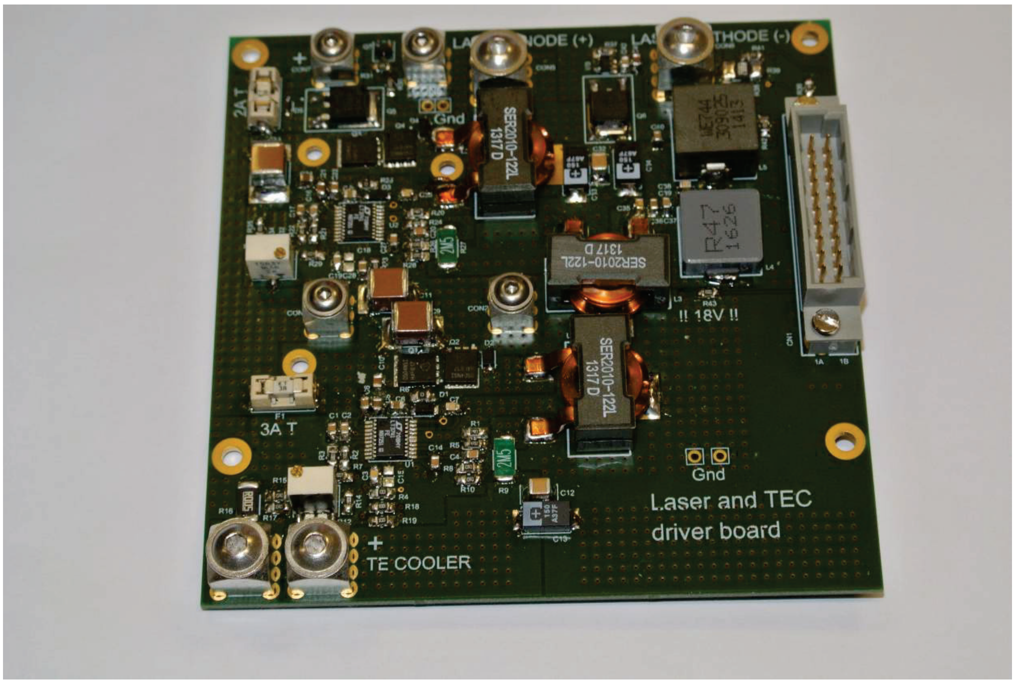
Carefully move the PCB into the oven and start the temperature controller. This will control the oven temperature to give the required reflow temperature cycle. The process takes around 20 minutes. The board will then be cool enough to take out of the oven. Obviously (!) desist from using the same oven for cooking food.

The PCB can then be checked that the process was successful. It is recommended that solder is removed from pads that were not populated. Solder wick is great for this, leaving pads ready for these additional components to be hand soldered.



As always, practice makes perfect, but the process of reflow soldering components on a printed circuit board is a fairly straightforward one.

Once any further ‘easy’ components are hand soldered, the completed board, shown below, will look almost as if it was produced with an automated machine! Well almost, that’s the good thing about poor resolution .jpg images: you will not be able to see some of the ‘blobs’



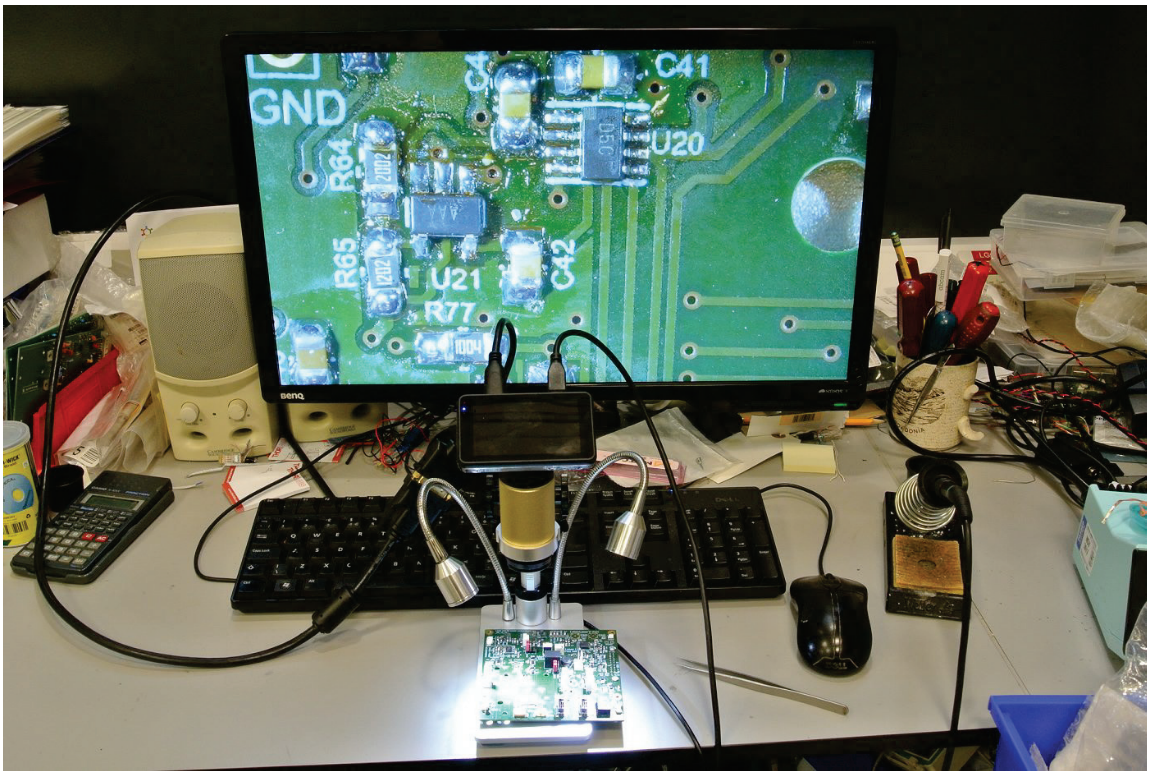
6. Hand soldering

For completeness, we present details of essential additional components required for hand soldering components.

1. Soldering iron (digital or analog); we use a Weller WSD 81, an 80 W temperature-controlled.
2. Solder (recommend water soluble, rosin core acceptable).
3. De-soldering braid. This can be obtained in small packs as in the image on the top left of the previous page, though larger packs are usually more cost-effecive. We use 2 mm wide braid Chemtronics SW14035, obtainable from RS, order number 508-6352.
5. Tweezers are essential for handling small surface-mount components. Tweezers with a bent, fine tip are preferred, though user comfort is also an important consideration.

Of course, the most important item in soldering kit, particularly when using components smaller than the ‘easy’ 0805’-sized components is a good pair of eyes! The trouble is that the more experienced you become, the more the eyes degrade. Glasses are useful, pocket magnifiers are more useful, large magnifiers with built-in illumination are really useful, but there comes a point where video magnifying technology becomes necessary. Long working distance video magnifiers are often quite expensive, but fortunately, not always. Thee days, high resolution CMOS cameras are ubiquitous and computer monitors are everywhere.

We use an Andonstar ADSM 201 USB microscope[5] which has its own small screen but also has a HDMI output which with a suitable cable can be plugged into the DVI socket in the back of most monitors giving adjustable magnification. It comes with a small stage with LED illumination.



We have not experienced any reliability issues when making boards this way and have not had any failures when the boards operated in normal, laboratory environments. For extended temperature work, then a more professional approach is advised.

Now that we know how to do it, you can too! All good things come in small packages!….However, ball-grid array chips are something else….and we have tried our rough approach on these devices.

References

1 https://www.felder.de/Electronic_industry/kategorie/smd-solder-pastes-and-fluxes.html

2 https://www.beta-estore.com/rkuk/order_product_details.html?p=793 ISO-Cream 'Clear'

3 https://www.beta-estore.com/rkuk/order_product_details.html?p=740

4 https://www.beta-estore.com/rkuk/order_product_details.html?p=613

5 Elektor microscope details: https://www.elektor.com/usb-digital-microscope-adsm201?mc_cid=5bd3eca79c&mc_eid=98dda63eb5

Acknowledgements

This note was prepared by R.G. Newman and B. Vojnovic in early 2018. Thanks to I.D.C. Tullis for useful comments and for 'forcing' us more than a decade earlier to adopt regularly, rather than occasionally, the use of surface mount techniques.

We acknowledge the financial support of Cancer Research UK.