

professional tube-testing-system (c) Helmut Weigl www.roehrentest.de

Don't be afraid of SMD components

Nowadays many components are (only) manufactured as SMDs. Those tiny components have been designed for industrial production with pick-and-place machines. Unfortunately SMD components are not very practical for manual placement. For that reason I try to avoid SMDs as far as possible and prefer mixed mounting types: Use as much as possible THT (Through-Hole-Type) components and SMDs only when there are no THTs available. So there are only few SMD ICs to mount.

Many electronics amateurs are frightened of SMDs. These components appear to be too small and too complicated to solder. In practice it is not as bad as it seems. Thus don't be afraid of SMDs.

Tools

SMD components and their pin distances are smaller than conventional components. You need:

- a pair of tweezers – rugged model
- plenty of light (desk light)



- magnifier (maybe combined with magnifier lamp)
- standard soldering station
- normal wide soldering tip (better: hollow tip)
- flux (I use FL22, this does not evaporate as fast as that used in thin fluid flux pens)
- desoldering wick

Soldering (my method)

1. What does not work:

Soldering the pins one by one with a needle-shaped soldering tip. The reason is that the soldering tip hardly grabs any solder and only purely transfers heat. Moreover I just tremble around with such a tip.

2. In my opinion unsuitable:

Soldering paste and hot air. The soldering paste can hardly be applied manually (without mask). Solder bridges are the result. Solder paste residues may cause soft shorts (requires washing).

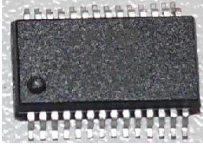
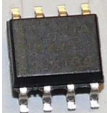


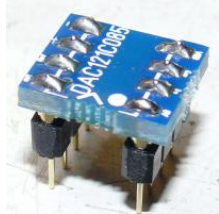


3. What works:

First solder diagonally opposite pins of the ICs. Check exact placement with a magnifier. Apply soldering paste FL-22. Wet soldering tip with solder. Draw soldering tip along the pins. Remove possible solder bridges with desoldering wick.

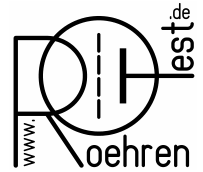
4. What works excellent:

Same as 3 but additionally use a hollow soldering tip. With this type of tip solder bridges are mostly avoided. But if there is still a solder bridge just strip of the solder from the hollow tip and pull off the solder bridge with the tip (capillary effect). Hollow soldering tips are expensive. If you have to solder SMD ICs more often you won't do without that any more.

Used SMD Components

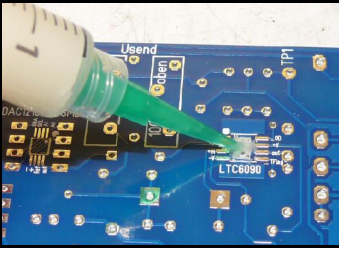
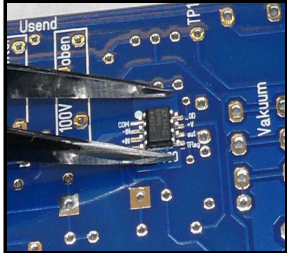
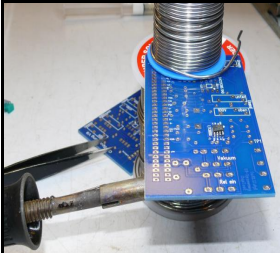
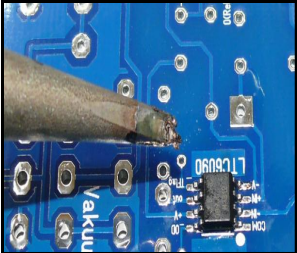
Component	Processability	Remark	Alternative
FT232RL 	more difficult small pin pitch, many pins	The provided USB adapter board PCB is most suitable for the system and has high interference immunity. Therefore I recommend to use the supplied interface board and accept the few needed SMD parts.	If you absolutely want to avoid these SMD components you can use a ready to use adapter board; but that has some drawbacks (e.g. not lead out LEDs). Due to the huge variety of offered adapter boards I cannot give any support for those. See also: www.roehrentest.de/Schnittstelle.html
ISO7421D 	easy large pin pitch		
USBLC6-2 	medium large pin pitch, but very small		
 DAC121C085	medium small pin pitch, but only 4 pins each side	IC may either be soldered directly to the PCB's bottom side or to an adapter board which is mounted at the PCB's upper side (with IC socket).	Adapter board 
LTC6090 	easy large pin pitch	Exposed (thermal) pad on bottom side must also be soldered (see below)	-
PCF8574T, PCF8574AT 	easy large pin pitch, IC is relatively large	-	THT version available (but not at Reichelt) PCF8574N, PCF8574AN The PCBs allow alternative mounting of SMD or THT versions (or SMD with an IC socket)

RoeTest - das Computer-Röhren-Messgerät -



professional tube-testing-system (c) Helmut Weigl www.roehrentest.de

The exposed (thermal) pad on the lower side of the LTC6090 is not accessible with a soldering iron from the top. I apply some FL22 solder paste to the PCB, put the IC in place, apply some solder to the soldering tip (better heat transport) and then heat the PCB from the bottom side (below the IC). This requires some heat (use wide soldering tip and optionally set soldering station to higher temperature). Heat until you see that the exposed pad has been soldered properly. Subsequently solder the IC pins.

			
Apply solder paste	Place IC	Heat from bottom side until exposed pad has been soldered properly	Solder pins