Soldering

Protective Equipment

Use protective eyewear while soldering! Position the fume extractor close to- and over your working area!

High quality, fairly expensive soldering equipment is available at Omega Verksted. Please be careful with it, and ask a board member if you are unsure about something.

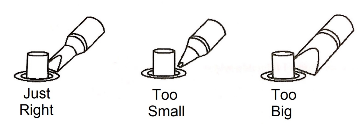
Soldering is a method for chemically joining items using heat. There are many types of soldering, but by far the most common is using a tin alloy to join copper parts. This is a staple of electronics manufacturing, and the focus of this article. The "Basics" section will give you some very condensed core insights to get started with soldering without missing important information.

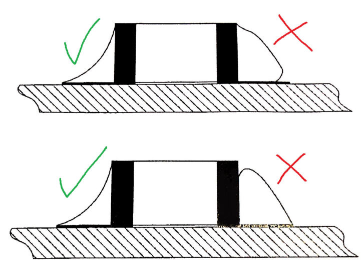
See also: Soldering Equipment

Table of Contents

Basics

- **<u>Do not use the soldering iron to exert physical force</u>**. Use pliers or other appropriate tools if you have to hold something in place or apply pressure.
- Use a soldering tip of appropriate size. If the solder joint is not heating up fast enough, usually the thing to do is to use a bigger soldering tip. A very coarse rule of thumb is that the soldering tip should be around the same size as the pad on your PCB. You can ask a board member if you can't find a bigger tip.
  - The Metcal soldering stations at Omega Verksted do not allow for adjusting the tip temperature, because they automatically use the optimal tip temperature[1] for the lead free solder stocked by Omega Verksted, about 412°C.
- There is a substance called **flux** that is a component of solder. Basically this serves to make the liquid solder flow correctly and distribute itself onto the component pins/pads. **Flux is completely essential to soldering.**
  - Flux is single use, meaning that it will only function for a single heating. When initially soldering a joint, the flux in the solder thread will be sufficient. If you have to reheat a joint, additional flux must be added externally. At Omega Verksted, you can find additional flux in the Poison Fridge.
  - When adding external flux, don't be afraid to add too much (although please don't waste excessively)
- There should be a *small* amount of tin on the soldering iron when it is in use, for any soldering operation. This will increase the contact surface area and improve thermal transmission.
- If there is oxide on the soldering iron (brownish-black stains), clean it off on the brass wool on the workstation while the iron is hot.
- In general, soldering a joint should take around 2-5 seconds.
- In general, the solder joint should have a "nice" slope. It should not be too concave or convex.
- Use a claw stand ("helping hand") to align the items you are soldering if necessary. Alternatively you can ask someone to help you hold something.
- Component legs should be cut *before* soldering.
- When finished, add some tin to the soldering iron. This will extend the lifetime of the equipment.





[1]*Not the same as solder temperature*

Desoldering

- When using a solder wick, you should add some flux to the solder joint you want to desolder.

Advanced

Although the following information could be useful for a beginner as well, you shouldn't be too concerned with it if you are just starting out. Soldering is the best way to learn soldering.

Types of Solder

There are a great many types of solder available, but at OV we usually try to adhere to lead free tin-silver solder alloys.

Leaded

- Sn60/Pb40
    - A classic old-timer
    - Easy to work with, fairly oxidation resistant
    - Low Thermal Conductivity, can be held liquid for quite a while
    - 188 °C
- Sn63/Pb37
    - Eutectic
    - Low TC
    - 183 °C

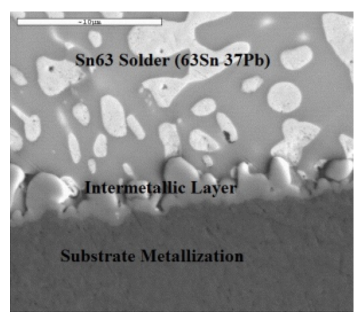
Lead Free

- Sn93
- Sn95

Weird

- Bismuth-Tin Solders
- Indium-Tin Solders
- Indium-Bismuth Solders
- Pure Tin Solders
- Silver Solder

Intermetallic Layer

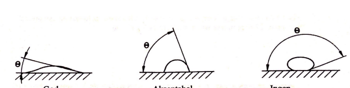


The goal of optimal soldering is to create an **intermetallic layer** (the layer of bronze between the copper and the tin) of the optimal thickness. This has been empirically determined to be **around 1m**. For most applications, this will be achieved when the solder is heated to the correct temperature, the solder flows across the copper to cover all the areas of the joint ("wetting"), and the joint is heated for around 2-5 seconds. Heating the joint for too long will result in an intermetallic that is too thick.

- If the intermetallic is too thin, the joint will be mechanically weak.
- If the intermetallic is too thick, the joint will become brittle.

Over time, the intermetallic will passively become thicker on its own. This can contribute to the deterioration of old electronics.

Wetting



Left to right: Decreasing quality of wetting

In soldering, **wetting** is the ability of the liquid solder to flow over and cover the desired surfaces of the components, wires and PCB pads. The degree of wetting (wettability) is determined by a force balance between adhesive and cohesive forces. Good wetting is achieved when the solder is heated to the correct temperature, there is sufficient active flux and the surfaces to be soldered are not contaminated with dirt, fat or oxides. Active flux will to a large degree remove impurities, but it doesn't have infinite capacity to clean.

Wetting is usually measured by the contact angle of the joint:

- A very shallow angle means the liquid phase has such a strong affinity for the surface and so little resistance to flow it spreads evenly.
- A shallow angle

The distinguishing appearance of wetting may be confused with large excesses of solder, for small joints this may be hard to distinguish.

- A Concave joint indicates good wetting and a contact angle well below 90 degrees, and is easily visible.
- A Convex joint with excess solder may be a well wetted joint and can be partially identified by a good surface finish and good deformation around surfaces, but this is not a good measure. Excess solder is a poor practice and should be avoided, as inspection is made more insecure.

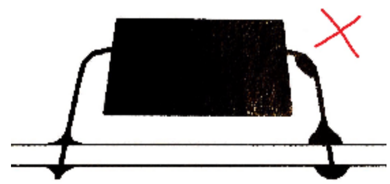
Flux

Flux (norwegian: **Fluss**) is a chemical cleaning agent, flowing agent, and/or purifying agent (Wikipedia article). One of the roles of flux is soldering is to prevent and remove copper oxides on components and PCBs. Tin based solder attaches very well to copper, but poorly to the various oxides of copper, which form quickly at soldering temperatures. By preventing the formation of metal oxides, flux enables the solder to adhere to the clean metal surface rather than forming beads, as it would on an oxidized surface. There are various types of flux:

- Resin based
- Synthetic
- Water based

After soldering is complete, there will often be some residue of burnt flux left. This may be cleaned, for example with Isopropyl-alcohol (some components may be damaged by this) or using an ultrasonic cleaner. If you are selling the electronics, or you want to make a good impression with it, you *should* clean it.

Stress relief



Many components have stress relief "built in". In Through Hole components (TH), the legs serve this function. The same goes for IC packages with for instance "seagull" legs, such as QFP packages. No-lead packages such as QFN, do not have this advantage. Putting too much solder onto the leads of a component will negatively affect the stress relief ability, and should be avoided.

Temperature

With surface mounted components, it is generally recommended that the core of the component (generally the IC) should not experience a temperature rise of over 3°C per second.

The **soldering temperature** is the target temperature of the solder during soldering. It should be 60 - 90°C higher than the **melting point of the solder**. In turn the **tip temperature** should be 72-92°C higher than the soldering temperature.

Inspection of manually soldered joints

Various errors can be detected by closely inspecting the solder joint. A microscope will come in handy.

Error	Severity	Description	Cause	Action
Cold soldering	Bad	Solder joint displays wrinkles, stripes or lines. Solder may be clearly separate from the component and/or the copper PCB.	Soldering temperature was too low. Usually thermal conduction is too poor (tip too small or not clean), or the joint was not heated for long enough.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.
Delamination	Catastrophic	PCB becomes deformed. Layers of the PCB may come apart.	PCB was overheated. Usually the joint/circuit was heated for too long.	(Irreparable)
Overheated joint	Poor	Solder has a finely grained, crystalline appearance.	Solder was overheated. Usually the joint/circuit was heated for too long, or there is a mismatch between the tip temperature and the solder used.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.
Porous solder	Poor	Solder joint has many small holes and/or grainy surface	Solder was overheated, combined with contamination. Usually the joint/circuit was heated for too long.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.
Icicles	Poor/Ok	Icicle-like structures or spikes	Solder was cooled down too fast.	Add plenty of flux and reheat for better wetting.
Solder bridges	Bad	Connections (short circuits) between pads or pins	Too much solder on the tip, often in combination with insufficient flux.	Remove solder bridges, add plenty of flux and reheat.
Dewetting	Bad	Solder has not flowed to all regions of the pad	Poor wetting, caused by contamination or insufficient flux.	Remove solder and clean. Preapply solder ("fortinn" loddestedet) before resoldering. Make sure to solder for ~2-5 seconds with an appropriate tip.
Dross	Bad	Charred areas on the surface of the solder	Contamination in solder, on components or on PCB.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.
Crack	Catastrophic	Cracks in the solder joint	Movement in the joint before solder has cooled.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.
Blobs	Bad	Blob around joint: Contact angle above 90 Degrees	Excessive solder has been applied. May if resoldered lead to a bridge, can also occlude joint making inspection of surface wetting impossible.	Remove solder, clean and resolder. Make sure to solder for ~2-5 seconds with an appropriate tip.

Surface Mount- specific techniques (Advanced)

Reflow Soldering

The most common technique in industrial electronics manufacturing.

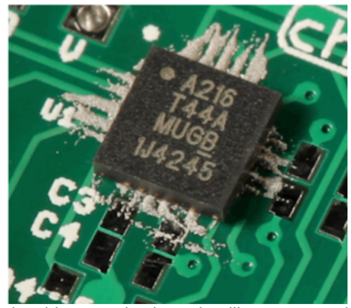
Instead of conventional solder wire, **solder paste** is used. Solder paste has the same ingredients as solder thread, but instead of being a solid wire, tiny beads of solder are suspended in a viscous flux, resulting in a gel-like substance. In addition to being solder, it also weakly glues the components to the PCB even before soldering.

Instead of using a soldering iron, the entire circuit is heated with a precise temperature profile (temperature as a function over time). Done correctly, reflow soldering generally gives better results for small or fine-pitch components unless you are extremely good at hand soldering. Either way, it also gives much better repeatability and process control, which are the main reasons it's the most common method in industrial manufacturing.

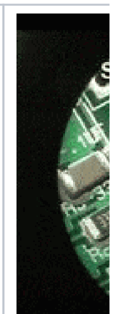
The components are free to move with the flow of the solder, and adhesive and cohesive forces (surface tension) will actually move the components towards the correct placement to some degree, although this effect has its limits. See gifs to the right.

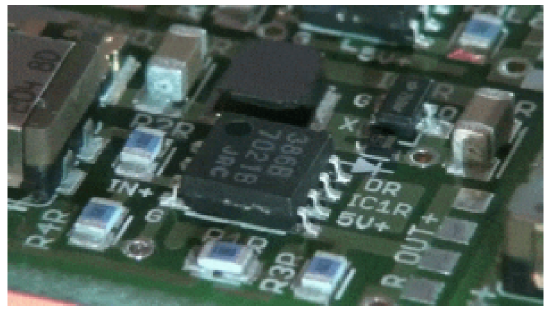
Omega Verksted has facilities for doing reflow soldering, see Reflow Oven.

(Click for satisfying gifs)



Solder paste is a bit excessive here, but illustrates a point. Notice adhesive and cohesive forces moving the component as well as the solder.





blocked URL

Different Package Types

Some packages with gull-wings like standard 50mil pitch SOIC packages have long, wide, well spaced pads and are extremely easy to solder and inspect, as they have well exposed pins with plenty of space for large pads.

Proper Stencil Use

Stencils are stiff membranes with patterned holes used for applying some liquid or emulsion to specific areas of a surface.

- Most Stencils are constructed by laser cutting holes in a thin piece (80-150um, usually 120um) of Stainless Steel, these have fairly good pad definitions, are cheap (5-20$) and easy to manufacture (<1 day) and may last up to 1000 prints. Main downside is the slightly rough final edges that can wear and dull over time, and the inner surfaces of the cut that will have a rather poor finish and cause some solder to stick to the stencil instead of the board. They are good enough for almost all applications and a good go-to for almost all stencil applications.
- Cheaper stencils made from laser cut Kapton (Weakly oriented Polyimide Film) are often available, but worse to set up, flexible and stretchy, not durable (<10 prints) and harder to rig. Not really recommended, they flop around and distort with ease.
- Volume Production Stencils are often made electrochemically with a lithography process; a thick photomask is applied to a mandrel (metallic substrate), exposed with the holes needed in the solder paste accurate to a few 10s of um and then developed, leaving tall, smooth and highly accurate pillars of cured photoresist. The plate is then put in a nickel electroplating bath, and a uniform ~100um thick layer of nickel is grown atop the mandrel. The soldermask is then dissolved and the mandrel etched away. This leaves an extremely good surface finish on all sides of the stencil, a higher strength and better lubricity, yielding a much higher duration and exceptional definition and accuracy. As you may guess, these are pricey.
- Vinyl cutters can be effectively used for home fabrication of stencils, benefits are low cost and good side surfaces but poor definition. Laser cutters can also be used but will often suffer even more. Being a cheapskate has it's downsides.

Stencils need some fixturing of some kind to be effective, and there are quite a few options.

- Steel Frame: The more pricey option (5-10$ extra), but for more complex designs often worth it. A Stainless Steel Stencil is rigidly attached to a steel tube frame, near perfectly planar and stiff enough to ring with a high note when lightly struck. Easy to mount and use, very reliable and guaranteeing a good result, this is often the best solution. The Stencil Printer rig we have here at OV is by far best suited for these kinds of stencils and well worth the extra cost.
- Frameless: This is what all kapton stencils are delivered as, and what many steel stencils are aswell. A rig of scrap PCB's, hinges and mounts made out of tape can be quite effectively utilized for fixturing larger boards, this is quite vital for smaller boards that one cannot just use 1-3 fingers to clamp down and adjust positions. If you want to be cheap of do small boars with large pitches, this is the choice for you.
- External framing system: Advanced manufacturing often needs frames to be quickly replacable, universal and cheaper on materials and labor. One can cut a series of slots around the

periphery of the frame and use pneumatically actuated hooks to secure and tighten the frame uniformly, these frames are often quite expensive but reduce individual stencil costs significantly and make interchange and positioning easier.

Temperature Profile

blocked URL

The Soldering process has a number of steps, all with their important functions.

- Preheating:
  Just heating up the board, making the flux flow and activate.
  Fairly long, relatively fast temperature gradient
- Soaking:
  Heating the solder to just under melting, cleaning surfaces with active flux
  Fairly long, shallow temperature gradient
- Reflow ramp-up
  Heating up components to somewhat over melting point
  Short, high gradient
- Reflow peak & hold
  Letting solder balls immersed in flux coalesce, wet and bind nearby metallic surfaces
  As short as possible, low gradient
- Cooling
  Constant gradient, slow enough to not cause major thermal gradients across the boards that can cause differential expansion stresses warping the board
  As slow as practical

Types of Defects

Various errors can be detected by closely inspecting the solder joint. A microscope will come in handy.

Error	Severity	Description	Cause	Action
Loose Solder Balls	Ok	Some solder paste has gone off a pad and drifted across the board.	Not a major issue if encapsulated by hard flux, but can be a shorting risk if balls are large.	Clean Board
Sintered Sponge Joint	BAD	Solder looks spongy and brittle, has almost no strength and looks like a rough sponge. Parts easily fall off and joint can be scraped off with a fingernail.	Far too long soak time, too high heat. All flux has burnt away before solder has melted, balls have stood stacked and oxidised before melting	Reduce profile time significantly, lower soaking heat.
Partially melted Balls on surface	Ok	The soldered joint seems mostly fine, but there's solder	The preface to the above sintered sponge, but not a direct issue. Marginal amounts of flux on outer balls, partial sintering has occured.	Reduce soak time
Oxidized surface	Poor	There is a strong, dull gray coating on all pads or a rainbow colored oxidation layer on exposed metal	Too high heat for too long, peak temperature is too high. Risk of burning board, delamination, frying chips	Lower soak and peak heat, reduce time
Unmelted paste	Poor	There is paste on the board that hasn't melted, there is still liquid flux in the paste	Too low heat, too short profile	Increase soak and peak temperature, profile time
Tombstoned Chips	Bad	Components have flipped vertical and stand on one pad, looks like a toombstone	Pads too wide or far apart,	Shrink footprint, use less solder paste, heat board more evenly
Head-In-Pillow	Bad	Upon closer inspection the solder has melted but not wetted the pad at all, the pad is just resting on top of the blob of solder	Too agressive heating causing one pad to heat up before the other and heat burning away enough flux that the affected pad has had more surface tension than it should	Adjust Profile to slower ramp, less soak time
Discolored Board	Ok	Copper surfaces, soldermask, chips and whatnot else are discolored, either darkened, brown	Way too high heat for far too long, peak temperature is too high. Risk of burning board, delamination, frying chips	Lower soak and peak heat, reduce time
Burnt Chips	Catastrophic	Many chips die near randomly across boards when tested. Probably comorbid with oxidised surfaces and board discoloration	Soldering profile is dangerous to part. Most IC's are surprisingly resilient and designed to be reflowed with lead-free processes at up to 270C for >30s, any oven will have to be poorly programmed for this to happen. Sentitive parts like sensors, precision voltage references or	Use recommended or tried and tested profile
Popcorning	Catastrophic	Chips may have cracks or holes in their casing, popping noises may be heard from outside the oven	Moisture has over time entered the chip, and rapid heating has caused the water to superheat to high pressure steam, cauing enough stress to crack the chip. Thin QFP's, DFN's, Large BGA's most affected. Rather uncommon for most parts and not a worry for single runs, however very significant in volume runs	Reduce parts' exposure to a humid atmosphere; use fresh components in dehumidified bags, bake parts before placement according to their MSL-rating, don't use old parts.
Oxidised pins	Bad	Part doesn't properly wet solder unless scraped or applied force to	Part has been exposed to air for too long, and tin coating on pin has corroded. Mostly a problem with grandpa's Ham radio replacement parts.	Electronic Parts have a shelf life, use new (not 10 year old and poorly stored) parts.

Metallurgy, Advanced Inspection

Eutectic Alloys, Liquidus, Solidus and Glass Transition

Some packa

Material defects, failures over time

The metallurgy of soldering processes is somewhat complicated and unbearably dry, the open literature is usually poorly explained, case specific and

Type	Description	Causes	Caused By
Cracks			Clean Board
Voids			
Thick Intermetallic Layer			
Short, stiff lead			
Die CTE (Coefficient of Thermal Expansion) Mismatch	In Heating-Cooling cycles, large shear forces can occur		
Lead-Board CTE mismatch			
Poor Homogeneity (multi-alloy solder)			

Advanced Inspection

Filtered/Automated Optical Inspection

> With a high resolution camera a series of macro shots of the board are stitched together into a whole image, this is then filtered to enhance shiny solder edges and hide all but component and pad outlines. The specified Solder Paste Layer (.gtp/.gbp) and Solder Mask Layer (.gts/.gbs) are combined and occluded by defined component bodies on an assembly layer, giving a reference for where there should be visible solder and where there shouldn't.
>
> The image is then at each joint compared with the reference "image", and from noise and light diffusion due to surface roughess and oxidation, the extra volume of excess solder and the lack of a toe and sides due to insufficient solder classical machine vision techniques like edge detection, blob detection and area estimation, surface finish by sharpness of reflection via deconvolution and inverse distortion of reflections of the light source can all be applied to find almost all solder joint parameters. If any joint of issue is found, the board and location(s) are tagged and the board is sent to rework.

X-Ray

> Many modern packages have as a technique to increase pin count density gone from having pins around the periphery of the package to having pins, balls or pads in arrays covering most or all of the underside of the package. As a results densities of >1000pins per cm^2 may be achieved, but the pins can no longer be seen. We cannot use direct optical techniques for these components, though we may be able to see the outer rim of balls underneath the soldered part, observe all balls being deformed equally and check the part for flatness to estimate whether the soldering has worked or not. This is however just an estimate. If we want to inspect the joints properly, we have to see through the part:
>
> X-ray sources between ~10-60kV can be used for advanced board inspection as these will have attenuation lengths of some mm and as such mostly pass through the copper plates of the component leadframe and the copper layers of the board, but at the same time be attenuated enough by thick layers of tin. As these X-rays have wavelengths in the 10s of pm- region they can't experience refraction in any normal material; no lenses can be constructed for them. Reflective optics can be constructed but are insanely expensive, and so most modern x-ray-sources are simply point sources. To get as good a focus as possible "microfocus"-tubes are used, these have emitting zones less than 30um in diameter, often at the cost of beam current and hence luminosity, tube life (melting anode becoming deformed) or immense cost (see Excelium's Gallium Microjet E-Beam- x-ray source or Synchrotrons). Occasionally benchtop medical autopsy machines like the Faxitron FX-50 with quite decent specs appear on eBay for a decent price.
>
> For board inspection we want to resolve at least ~3x the width of our smallest traces, so with a geometric magnification of 2 we at least want a 60um source.
>
> X-ray inspection can reveal cracks inside joints, voiding, partial joints, deformations in joints, pad delaminations, as well as board features like copper shapes and bond wires in packages and is as a result an extremely powerful technique for both inspection and reverse engineering.