# Table of Contents

## Overview

---

## Solder Paste Storage

- Mini Can Fridge
- Camp Fridge

---

## DIY Solder Paste Stencils

- Create Gerber files of cream layer
- Import Cream Gerber in Viewmate
- Swell Pads
- Export
- Cut!

---

## Framed Stencils

- Framed v. Unframed
- In a Machine
- Where to Get Stencils Made

---

## Stenciling Machines

- Get the Right Stencil
- The Machine!

---

## Solder Paste Syringes

- Tips and Plunger
- 0.4mm Pitch TDFN

---

## Soldering Irons

- What We Use
- Soldering Tips

---

## Hot Air Tools

- In Practice
- We use the Hakko Hot Air station for SMT rework, especially removing small resistors/capacitor or removing/reflowing ultra-small QFN’s or UDFNS.

---

## JEDEC Trays

---

## Squeegees

- $5 Squeegees

---

## SMT Wipes

---

## Resistor Packs

---

## Tweezers

- Removing a SOIC

---

## Microscope Inspection

- USB Microscope

---

## Buy SMT Tools

---
Overview

Here are a bunch of tutorials for manufacturing small to medium scale (10 - 10,000) piece surface mount electronics.

Solder Paste Storage

If you want to do precise, fast, and fine pitch SMT assembly, you'll have to switch from 'wire' solder to 'paste' solder

Wire solder is what most people start with, it comes on a roll:

Inside the wire is a bit of flux (that's the smoke that comes off of the iron when soldering). The flux is often rosin type, no-wash type.
When doing SMT work, you can use thin wire but often times even that isn't good enough, you need to use paste! Paste comes in tubs of 1/2 - 1 lb or so. The paste has consistancy of smooth peanut butter and is made of ball of solder suspended in flux. As the paste is heated in an oven the solder melts and the flux burns away leaving a solid solder joint.

We may have a future tutorial with more details about choosing paste and solder but for now we will move on to what this tutorial is about which is how to store solder paste.

The problem with paste is that the flux can evaporate off, leaving the paste 'old' and 'dry'. It won't screenprint as well - you'll have difficulty with bridges and getting clean deposits. Paste should be kept cold, but not freezing. Kester suggests 0-10 degrees C (32-50 degrees F).

If you can get your hands on a cube fridge, we suggest using that - make sure that no food is stored in there as paste is toxic and it gets everywhere. If you don't have space for a cube fridge (like us) here are two solutions we found.

**Mini Can Fridge**

If you have a single tub of paste, you can use $20 "desktop coke can" coolers. Also mini insulin coolers might be an alternative.

There is a peltier inside, which can cool down to about 40-50 degrees F. it works best with something that has full contact with the peltier, like a big tub of paste (not a syringe).
The only annoying thing is that they have a USB port and you may not have a computer to plug it into nearby. Also, although it is USB it draws much more than the 500mA max you're supposed to pull from a port. So we suggest using an external power supply. We had a 5V 1000mA power supply kicking around so we just cut the USB cable off and soldered a female 2.1mm jack on. You can also just use one of these 5V @2A supplies and splice it directly on.

That's it! Pretty easy to use and kept our 1 lb tub cooler. We also wrapped the tub in a piece of foam to add more insulation.
Camp Fridge
Once we started having more than one tub, and also syringes for touch-up/dispensing we had to upgrade from the mini-can fridge. Usually people get a mini cube fridge but we don't have a lot of space, and also we determined that the power usage of a larger peltier-based fridge was less than the smallest cube fridge.
You can find 12V camping fridges at most appliance stores. It draws about 60W and has a temp display on the front. It tends to keep stuff inside at 45 degrees F or so which is perfect for us.

One thing to watch for with these is that they condense water during the summer months so you may want to keep a rag on the bottom to soak up water and wring it out once in a while.

**DIY Solder Paste Stencils**

If you want to make a lot of PCBs using SMT technique, its key to use reflowing instead of soldering - so that the entire board is 'soldered' at once. But to do that
you'll need to deposit paste precisely on the pads. For starting out, you can use a DIY stencil such as a laser cut kapton/mylar sheet. This is a low cost technique you can do yourself or order online!

Thanks to Ryan O'Hara at Ohararp.com for this information, he provides a stencil cutting service and is recommended!

Supplies you'll need:

1. A laser cutter
2. Kapton film, I like the 2 mil thick 1 ft square sheets from McMaster-Carr
3. Solder paste such as Kester No-Clean

Software you'll need:

1. PCB layout software (well, thats how I do it) - this example will use EagleCAD
2. Pentalogix ViewMate Gerber viewer software
3. PDFCreator or some other free PDF printer

Create Gerber files of cream layer

This is the PCB we'll be making a stencil for. It only has one chip but of course you can use a more complex layout.

Your PCB software should be able to create/export the Cream Layer (solder paste layer) in Gerber RS274x format. In Eagle you can make your own Job for this quite easily.
Also export the Dimension layer (PCB outline) since that will help a lot in registration.

Import Cream Gerber in Viewmate
Start up Viewmate and File>Import>Gerber one of the Gerber files generated.
You can zoom in using the Magnifying glass tool.

Swell Pads

Next we will make minor adjustments to shrink the pads a little

Select Setup>D Codes
Which will bring up a list of all the pads used. You will probably just want to select all of them.

Then select Operations > Swell.
and input somewhere around -0.002 (2 mil) to shrink all the pads by 0.002 inches in each direction.

You'll now see that your pads are thinner. This prevents bridging since the laser is not perfectly precise and tends to 'go over' the boundaries by a few mils.
Export

Now we'll export to PDF which will allow for easy importing into Corel Draw. The free version of Viewate doesn't seem to permit exporting, but you can print to PDF which is just as good.
Cut!

Import into Corel Draw and use raster not vector, to burn away the kapton film. For a 35W or 45W epilog, 30% speed and 100% power at 600 dpi made for a nice clean edge. Be sure to gently rub the stencil with water and a paper towel to get rid of the burnt kapton.

I usually use the Dimension layer info to make a jig for silkscreening by cutting out the PCB outline in a 0.062" (1/16th) clear acrylic sheet

Here is a LFCSP 16 (4mm on each side) cut out of 2 mil kapton as above.
Framed Stencils

If you want to make a lot of PCBs using SMT technique, its key to use reflowing instead of soldering - so that the entire board is 'soldered' at once. But to do that you'll need to deposit paste precisely on the pads. For starting out, you can use a DIY stencil such as a laser cut kapton/mylar sheet (low cost) or depositing the paste by hand using a syringe.

However, if you ever decide to make a few hundred boards especially those with very fine pitch type parts (say 0.4 or 0.5mm pitch) it may be time to move to a framed stencil!

Framed v. Unframed

The key benefit of framed stencils is that they are 'pre-stretched'. Especially with large PCB panels, having the stencil-board-alignment off by even a mm can cause bridges or opens. With DIY stencils, alignment is a pain and takes care by the operator for each pass.

With a framed stencil, the thin stainless steel sheet is laser cut and then stretched into a solid cast aluminum frame. Its less likely to have misalignment because the sheet can't slide around.

If you have the right equipment you can use unframed metal stencils and stretch them into a frame yourself, but unless you're a board fab house its unlikely that this is cost effective.
In a Machine

Framed stencils are used in a 'screenprinting' PCB stencil machine (we'll have another tutorial about this one). The frame is bolted in place onto a hinge so you can move the stencil up (to replace the PCB) and down (to stencil).

Since the stencil is stretched and flat, as long as it is bolted into the stenciller solidly, you will have minimal adjustment from one PCB to the next. This is what makes it ideal for multi-PCB runs. We do 10-50 PCB panels (of up to 20 pieces per panel) at a time with about 5 seconds between screenprints.

Where to Get Stencils Made

We get our framed stencils made by stencils unlimited(). Its pretty easy to make a stencil, just export the Cream Top (or Bottom) layer from your PCB layout software and upload it during your order. They will calculate the best stencil thickness (you want a thicker stencil for large-pitch parts and thinner for fine pitch so an average is taken)* and ship it the next day.

If you have a board fab house with stencil-making capabilities, you can also ask them to make you the stencil. Don't forget to have a tiled Cream gerber if you are having panels made - so if your design is tiled get them to tile the GBC file for you!
*IPC 7525* has a long detailed document on calculating stencil thickness. It should be approximately 2.64 + 0.0831 * pitch-of-component* then averaged.

**Stenciling Machines**

You can always start out your SMT designs with *DIY laser-cut stencils* but eventually you'll want to upgrade to a stenciling machine. Stencil machines hold the metal-cut stencil flat and taut so that you don't have any shifting or curling while you try to lay paste down. The good ones have a fixture so you can always get the same alignment every time - speeding up your stenciling time down to under a minute per PCB. They're a little expensive, but if you ever plan to do over 100 of a PCB they are essential!
Get the Right Stencil

First thing to note is that most machines are to be used with a certain type of stencil. Some require framed stencils. Others use foil (unframed) type. Framed stencils are bulky, and more expensive, but they are fast because you don't have to spend time stretching or loading them. If you have a lot of stencils already, of course look for a machine that matches what you've got. At ~$200 each it's a shame to reorder them!

The Machine!

The stencil machine we opted for is called an [STP-350](https://www.madell.com/products/STP-350), we picked it up from Madell (most everything else there is not suggested but this machine is fairly well made and we think worth the price) For about $1350 US.
The nice thing about this machine is that it takes framed stencils, is about $1500 after shipping and has a good fixturing area. We liked that it uses 3mm holes for alignment. Many low cost stencilers require the PCB to be flat against the bed which means you can't (easily) do double-sided boards.

We get our PCB house to place 4 x 3mm holes in each corner of our PCB panel. The first time you set it up for a PCB run, you'll need to align it by loosening the 3mm studs and placing the PCB on the fixture bed. Then you can use the knobs on the side to help align it.
It takes 5-10 minutes to set up per board but once it is set up and aligned, you can just lift the stencil, place a new one so that the corner holes snap into the fixture studs and print! We've found that the corner holes almost always line up perfectly.

Solder Paste Syringes

If you want to do small scale prototype SMT manufacture, it may be faster to just deposit paste by hand onto the PCB instead of getting a stencil made and shipped. We use this for our prototypes and its very fast once you get the hang of it. Instead of stenciling paste on, a small syringe of paste is squeezed onto each pad, sort of like a cake decorator (but with toxic metal).
After the paste is deposited on the pads, each component is handplaced with tweezers. Then the entire board is reflowed. You can use hot air but a plate or oven works best (we'll have a tutorial about that some day).

This technique works for both lead and lead-free PCB/parts/paste but of course leaded solder is easier to work with.

**Tips and Plunger**

We use LF-4300 from Amtech for our in-house lead free paste. It's not necessarily the best, just what we use so if you have a favorite brand go with that!

You'll also need a plunger and tips. You can get a huge assortment of different tips from McMaster. For every day use we like using the 20ga pink plastic tip from the shop above - best for larger parts like SOIC, passives and some larger pitch TQFP.

https://www.google.com/search?channel=nus5&client=firefox-b-1-d&q=amtech+LF-4300

For finer pitch parts, and leadless parts, we go with a luer-lock 24 gauge. It's a bit tough to push on but gives a nice fine bead.
For the plunger, we suggest shelling out for the nice plunger we have shown here, its $25 but makes hand-dispensing trivial () and you can just reuse it when the syringe is empty.

0.4mm Pitch TDFN

When hand assembling fine pitch or leadless parts, the syringe dispener makes an impossible job easy!

For this board we'll be placing a 0.4mm TDFN leadless part. Virtually impossible to do by hand! Using the fine needle, place a blog on each pad and lay a small bead on the TDFN pads. You can also place a small dot in the center of the TDFN pad - sometimes this helps with placement but it's a bit of a toss up so try both ways and see which you prefer.

Place the large parts first (so you don't bump the tiny parts as you place them).
Carefully place the fine pitch parts in the correct orientation. Be careful about placement so that the pads line up, a difficult task as you get to smaller pitches.
After you reflow, there will be tiny bridges, you can fix those with a very fine tip soldering iron and some wick.

Soldering Irons

One thing is certain, if you're doing SMT, you're doing rework. So having the right rework tools will save you tons of time, frustration and money. We try to aim for about 90% yield off the pick and place (or better) and repair the rest for a total aim of 95% or better. The first tool you'll want to invest in is a really good soldering iron.
What We Use

We originally were big fans of Weller irons and used them in school but when it came time to stock the Adafruit lab we decided to go with a Metcal MX-500 on the recommendation of a friend. It was a good recommendation! Metcals are really heavy, durable and heat up ultra fast.

They're easy to switch tips, and can power two different soldering stations (we use a hot tweezer on the 'alternate' station). The wand is light and easy to grip.

The only weird thing to watch for is that the temperature is set by the tip, not the station. The station only has an on-off switch. This can be good when you're using only one type of solder, say leadfree (700 degree F tips) or lead (600 degree F tips) but can be annoying if you use both - although we suppose you can just go with the hottest tips.

Metcals are a little pricey but you can pick one up used on ebay for around $200 (that's what we did).

Soldering Tips

The nicest thing about the metcal is the wide assortment of tips available. They're a snap to swap - takes only a few second of pulling the old tip out and pushing in a new one.
For general thru-hole we like the screwdriver tip STTC-137P (shown on the wand)

For soldering FPCs or reworking a long row of pins, the SMTC-061 (all the way on the right) is very handy

For fine pitch rework we use the STTC-122 (second from the right)

We also like the two long 'hard to reach areas' tips STTC-107 and STTC-140. Great for getting underneath microSD holders and miniUSB connectors.

We also got a hoof tip SMTC-1147 (second from the left) but haven't used it yet since we don't do drag soldering so much.

---

**Hot Air Tools**

Soldering irons are the primary tool used for soldering, and we use ours all the time for soldering. But for leadless parts, like QFNs, or ultra-tiny components that need rework, hot air can be superior!
The nice thing about hot air is that its just air - this means there's no physical 'tip' nudging the part to heat it up. This is great for delicate components. Another thing is that you can heat up an entire area at once, great when you want to rework a small leadless part because the pads are underneath the chip. The annoying thing is it takes a long time to heat up a board especially if there is a ground plane involved. You can try preheating the board if you have a preheater (basically a small reflow oven works fine).

We decided to go with a Hakko pencil tip hot air station. This station has some things going for it: there's an internal pump, its easy to use and it has a fine tip for small component rework. We picked it up off of ebay for about $150 which is lower than new but worth the risk of it not working out. We recommend getting a non-'chinese clone' air rework station since its common for a poorly made one to work ok at first and then die 6 months later because the pump is low quality, or the heater gets damaged. The old 'black box' Hakko's are discontinued and replaced with the new violet/yellow FX series, so it may be easier to get them off ebay for a discounted price.

However, we recently decided its not 'good enough' for us so we bought a Hakko 852 off ebay for $400. We'll post up here when it comes in about whether we like it more.
In Practice

We use the Hakko Hot Air station for SMT rework, especially removing small resistors/capacitor or removing/reflowing ultra-small QFN's or UDFNS.

This is a nice video with a lot of details on using hot air reflow. Its slow but it works!

In this video, we use it to remove some very very small 0402 resistors without damaging them so that they can be measured out of circuit:

If you use our syringe technique to lay down paste, you can even 'reflow' a small board by bathing it in hot air and slowly working from one edge of the PCB to the other. This may not work well with very thick boards with big ground planes but it is probably good enough for making breakouts and such.

JEDEC Trays

This tip is a quicky but very handy! When programming a tray into your pick and place, you will need to know the X&Y dimensions. This sounds simple but measuring them with calipers can be annoying which is when we found this really handy PDF from a JEDEC tray maker that has all the dimensions ready to go!
You can grab the PDF here, for QFP trays.

---

**Squeegees**

So you have your PCBs, cooled paste, your stencils and your stenciling machine. Only thing you need now is to apply it! This is the final tutorial in the SMT stencil series!

**$5 Squeegees**

We do manual paste deposition. We tried a whole bunch of different, expensive squeegees, but in the end we really just liked the nice rubber-gripped paint scrapers at the hardware store down the street.
We have two: one 4" and one 1" wide. We like a 1" one for mixing up paste after its taken from the fridge. This gets it warmed up and easier to work with.

We then use the paste spooned out to apply it to the larger spatula.
Pull the spatula against the stencil. Getting this right takes a lot of practice, you need to get angle, pressure, and amount of paste correct.

Sometimes we have a big stencil so we only do one side at a time.
We have a technique we use, we usually do two wipes. One is at a more acute angle, where we push the paste thru the stencil.

Then we go at a wider angle, to scrape the paste off.
That's it! It does take practice. We had to do 50 boards before we really got the hang of it.

SMT Wipes

Now that you've finished stenciling, you'll want to clean up! Paste can get everywhere, especially when doing a lot of boards and the paste is liquidy. We recycle our paste (putting it back into the pot) even though its suggested not to do that. (If we ever go thru a pot a day maybe we wont have to but paste is pretty expensive!) Then we wipe down the scrapers, stencil and area using SMT stencil wipes which are totally awesome and make clean up really easy. They're also good for cleaning off a board if/when you have a messy stencil paste deposit and want to redo it.
Resistor Packs

For prototyping, modding, hacking and reworking, its good to have a range of resistors in your favorite package size. We like the digikey box packs of resistors, and have a set of the 5% in 0603, 0805 and 1206. Sometimes we wish there were more values so we may get the 0805 in 1% for a few values

Either way, having a resistor set is essential!
Pick your poison:

**1206 5% resistor pack (50 of each value)**

**0805 5% resistor pack (50 of each value)**

**0603 5% resistor pack (50 of each value)**

For the 1%, they come in boxes of ranges, pick them up as necessary

**0805 1% Resistors (50 of each value):**
1.0-9.76: PHC1A-KIT-ND
10.0-97.6: PHC2A-KIT-ND
100-976: PHC3A-KIT-ND
1.00K-9.76K: PHC4A-KIT-ND
10.0K-97.6K: PHC5A-KIT-ND
100K-1.0M: PHC6A-KIT-ND

**0603 1% Resistors (50 of each value)**
1.0-7.5: PHH1-KIT-ND
10.0-97.6: PHH2-KIT-ND
100-976: PHH3-KIT-ND
1.00K-9.76K: PHH4-KIT-ND
10.0K-97.6K: PHH5-KIT-ND
100K-1M: PHH6-KIT-ND
Tweezers

If you’re doing SMT, you’re doing rework and anything that makes rework suck less is a big win. We use hot air for reworking some SMT parts but if you have leaded parts, especially big ones, hot air can take a really long time. If you want fast component removal/rewiring we suggest getting a hot tweezer jig!

The MX500 soldering station is particularly good for hot tweezing, it has two ports on the front. One connects to the iron and the other we connect to a hot tweezer. You can switch out tips just like with the soldering wand. The system is called MX-PTZ and you can check out Metcal’s page for all the various tips you can get for reworking different-sized parts.

Removing a SOIC
Our favorite use for tweezers is to remove or rework SOIC and such. These are big chips, hard to heat up evenly and quickly but the large PTTC-x06 or PTTC-x07 cartridges make em easy!

For example, I want to remove this 14-SOIC chip:

Simply wait for the tweezers to heat up, then grab both sides of the SOIC.
and voila! The chip comes off easily, cleanly and without affecting the rest of the circuit.

---

**Microscope Inspection**

When your parts get small, it can get really tough to see what's going on with them. Especially if your vision is only so-so, having some assistance with inspection is an easily solved problem. When we were at school, the lab had a really sweet stereo-microscope. You could really see quite nicely and solder while looking thru the microscope. It was wonderful but unfortunately a bit too expensive for the common user.

Thankfully, Adafruit now carries a much more affordable alternative.
USB Microscope

As electronics get smaller and smaller, you'll need a hand examining PCBs and this little USB microscope is the perfect tool. Its smaller and lighter than a large optical microscope but packs quite a bit of power in its little body. There's a 5.0 megapixel sensor inside and an optical magnifier that can adjust from 20x (for basic PCB inspection) to 220x (for detailed inspection). Eight white LEDs are angled right onto whatever you’re examining so you get enough lighting to see, and are smoothly adjustable via a dial on the side.

New! We've upgraded this microscope to 5.0 Megapixel (from 2.0M) and this one comes with two removable plastic caps to get close ups for a wider focusing range.

If you plug this into any computer, it just shows up as a standard USB camera (we used this for our weekly Ask an Engineer show) and the Windows/Mac software lets you take snapshots using the button on the side of the microscope or direct from the software (so you don't move the camera).

We tried a bunch of different USB microscopes and found this one to be the best combination of optical clarity, usability, and price. It's perfect for electronics hacking, rework, SMT (de)soldering, inspection, and soon you'll find yourself pulling it out to look and photograph all sort of cool small stuff around your lab and home.

Pick up our USB Microscope here. (http://adafruit.it/636)

We took some snaps of this 'scope while looking at a tiny 0201-populated GPS board
at 20x magnification:

and then at a raw die at 200x:

---

Buy SMT Tools

Buy SMT Tools ()