sammichFM Build Guide v1.0

0. Introduction

This build guide tries to explain every step of the build process so that even those people inexperienced in electronics can follow the guild closely and achieve success. However, if you are so inexperienced that you have never done any soldering at all, you can look at guides aimed at teaching you this. I recommend:

http://www.curiousinventor.com/guides

Even if you are an experienced electronics guru of many years, please read through the guide fully before beginning to solder parts. The parts list alone is not sufficient in describing everything. Pay particular attention to:

- Mounting the power socket
- Mounting the heatsinks and voltage regulators
- Soldering the male/female headers that join the two PCBs and LCD
- Soldering the LEDs

I don't assume anything is obvious; instead I expect that if there's a wrong way to do something, at least one person in a hundred will probably do it the wrong way (including myself). So please:

Check twice, solder once!

and

Check the Mount Notes for every part!

and

Leave the backing paper on the panels and use this as a mask while you paint the engraving!

and

Don't remove the clear acrylic window from the top panel!

You can build the entire sammichFM with the backing paper still on the panels, and then paint the panel engraving after you have finished. This allows you to play with your new MIDIbox FM synth while you paint the panels (a slow process).

1. Soldering Techniques

These are the techniques I use when soldering through-hole components. These could be obvious, common-sense things to do, but in case you have no idea, here they are:

The “Upside-Down PCB” Technique

Early in the build process, it's possible to insert several components which have the same height and then flip the PCB upside-down and put it flat on the work surface so that the components are held in position. Typically I do this only with the resistors and IC sockets, and only if I can guarantee that components will definitely be held flat against the PCB.

This technique sometimes fails if there is not enough weight holding the components close to the PCB.

Some components like IC sockets and switches will “snap-in” and be held in place. In these cases, make sure the part isn't moved while soldering the pins. It is better to solder one pin first and check the component is flat against the PCB before proceeding with the other pins.

The “Tacking” Technique

“Tacking” is a term used in other crafts, essentially meaning “temporarily attach”. By holding the component onto the PCB with one hand (and optionally holding the PCB as well), you can tack one or more leads/pins in place by carrying a blob of solder on the soldering iron and wiping it into the joint. ***The solder joint formed is bad and should not be left like this***, however, it should hold the component in place while making proper solder joints for the other leads/pins, and then the tacked leads/pins can be resoldered using proper soldering joint technique. For IC sockets, pins on either side should be tacked, so the part is held flat. Pads connected to the ground plane (i.e. no dark circle around the pad) are hard to solder normally and even harder to tack, so choose another pad when tacking.

Please note that good soldering joints are formed by heating the pin and the pad simultaneously and feeding solder into the point where they meet, on or near the tip of the soldering iron. ***Do not normally carry solder blobs on the soldering iron to the joint.*** Only do this for the tacked joint.

BTW... if you don't burn your finger occasionally using the Tacking Technique, you're doing it wrong :-)

Tip: Adding flux to pads will make “tacking” much easier and often the joint will be good enough and won't require resoldering. If it has a cone shape (i.e. good “tenting”), it should be good enough.

The “Add Flux To Every Pad” Technique

In case you didn't know, flux is so essential to soldering that solder has flux inside it already.

Since your kit contains a tube of **CHIPQUIK No Clean Paste Flux** and you need only a tiny bit on the surface mount ICs, there's plenty spare to use on all the other pads on the PCB. I highly recommend it for pads that are difficult to solder (ground pads, socket pads) or which you want to solder quickly to avoid overheating the component (diodes, transistors, voltage regulators, LEDs) and there's nothing wrong with using it on all pads to make soldering every joint easy and looking perfect.

Squeeze out a blob of flux paste onto a non-porous surface (e.g. blister pack plastic) and use a toothpick to smear a tiny bit on the pad. It doesn't have to be even or cover the whole pad, as it will liquify during soldering and mingle with the molten solder. You can do this before or after you insert the component.

After soldering the surface mount ICs, start using it on the resistors, so you can learn how much you need and how it affects the soldering. At the very least, use it on ground pads, as the direct connection to the ground plane will dissipate the heat from the iron making soldering difficult.

Note: be sure to clean your soldering iron tip regularly (after every 4th joint) to avoid burnt flux getting into the joints, making them look brown. This is a good thing to do even if you don't add more flux.

2. Base PCB Soldering Walkthrough

This walkthrough suggests mounting and soldering parts in the order they are presented in the Parts List.

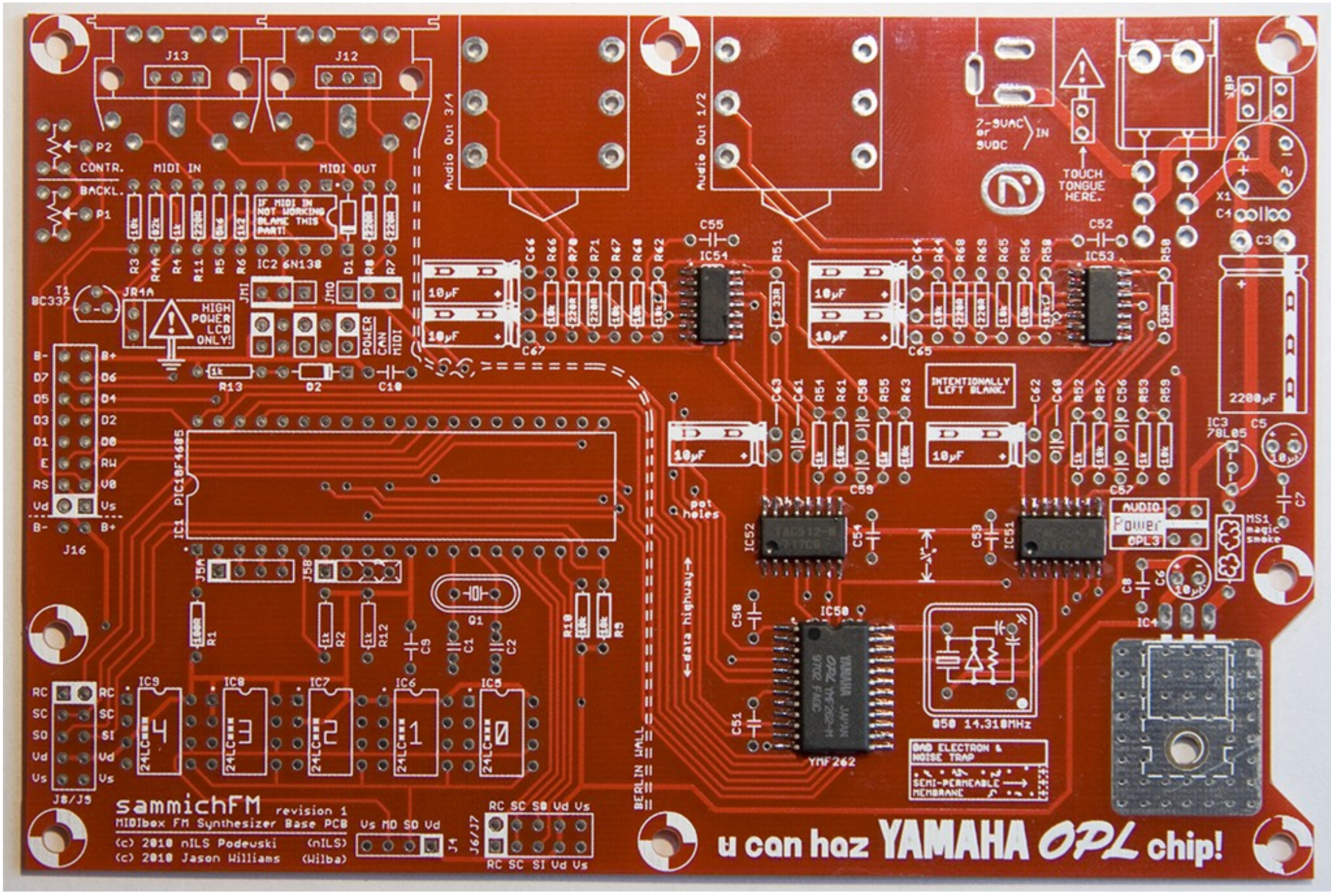
I recommend attaching the 20mm spacers to the bottom of the base PCB so the PCB can sit above the work surface. This allows parts with long leads to be inserted.

Step 0: Check if 5V and GND are shorted/connected.

In the unlikely event that your PCB has a manufacturing fault, it is a lot easier to fix if you know the PCB is at fault and not your soldering or the components.

Note: Since the 5V supply to the audio and OPL3 section is isolated by two headers, in which you use shunts to connect to the 5V supply, you need to test for possible 5V/ground shorts in two places, inside the audio/OPL3 section (e.g. at C50, C51, C52, C53, C54, C55) at and outside it (e.g. at J4:Vd/J4:Vs)

Step 1: Solder surface mount ICs.



Here is a good instructional video of soldering surface mount ICs:

http://store.curiousinventor.com/guides/Surface_Mount_Soldering/101

You don't need all the equipment they show there. I use a cheap temperature-controlled soldering iron, some flux paste applied with a toothpick, and copper braid/solder wick,

Your kit includes CHIPQUIK No Clean Paste Flux and 12” of copper braid/solder wick. USE IT!

My surface mount soldering process is fairly similar to that in the video above:

Step 1.1: Apply some flux to the pads using a toothpick.

Step 1.2: Put IC on the pads, align carefully. The flux should keep it in alignment.

Check orientation of IC with part outline on PCB, the dot on the corner of the IC should match the dot on the PCB (left of the "notch").

Step 1.3: Apply a tiny bit of solder to iron tip and touch only the blob onto the joint. Don't touch the pin with the iron tip and move it out of alignment!

Consider this joint just "tacking" the part down while you do the other pins properly. It doesn't matter if it has slightly too much solder, or has a "horn", you can fix this easily later.

Step 1.4: Check alignment again. If it requires adjustment, reheat the corner pad and move the IC while it is hot. *This is tricky to do with tweezers. Use a finger.*

Step 1.5: Repeat Step 1.4. Srsly. *Check the orientation as well! Dot to dot!*

Step 1.6: Apply more flux to the pads and pins.

Step 1.7: Solder the corner pin/pad diagonally opposite to the pin/pad already soldered.

My technique is to hold the solder vertical, touching the end to the low point of the pin (where it touches the pad), and pushing the soldering iron tip into the solder. This will instantly melt the solder and it will naturally fall into the joint. If there is too much solder in the joint, immediately remove it with solder wick before proceeding. You don't want to create solder bridges behind the pins, which might happen while soldering the next pin.

Step 1.8: Check alignment again. If it requires adjustment, reheat the pad and move the IC while it is hot. With the other corner already soldered, you can only do rotation adjustments, but that might be enough to get pins in alignment.

If you are not 100% happy with the pin alignment, desolder these two pins and start again. If you solder any more pins, desoldering becomes very difficult.

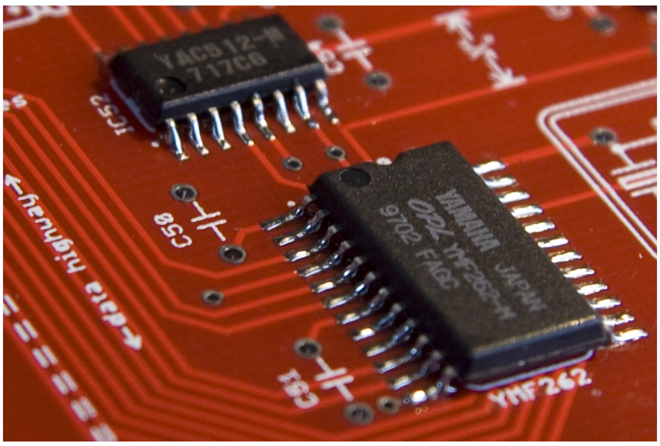
Step 1.9: Solder all the pin/pads not already soldered.

Step 1.10: Resolder the corner pin/pad you did first, it might be a poor joint, i.e. just "tacked" to hold the IC while you solder the other joints.

Step 1.11: Use solder wick to remove any excess solder or solder bridges between pins. *By "excess" I mean obvious blobs. If the solder joint is no wider than the pin/pad, it's OK. If you use just the right amount of solder per joint, you don't need to use the solder wick at all.*

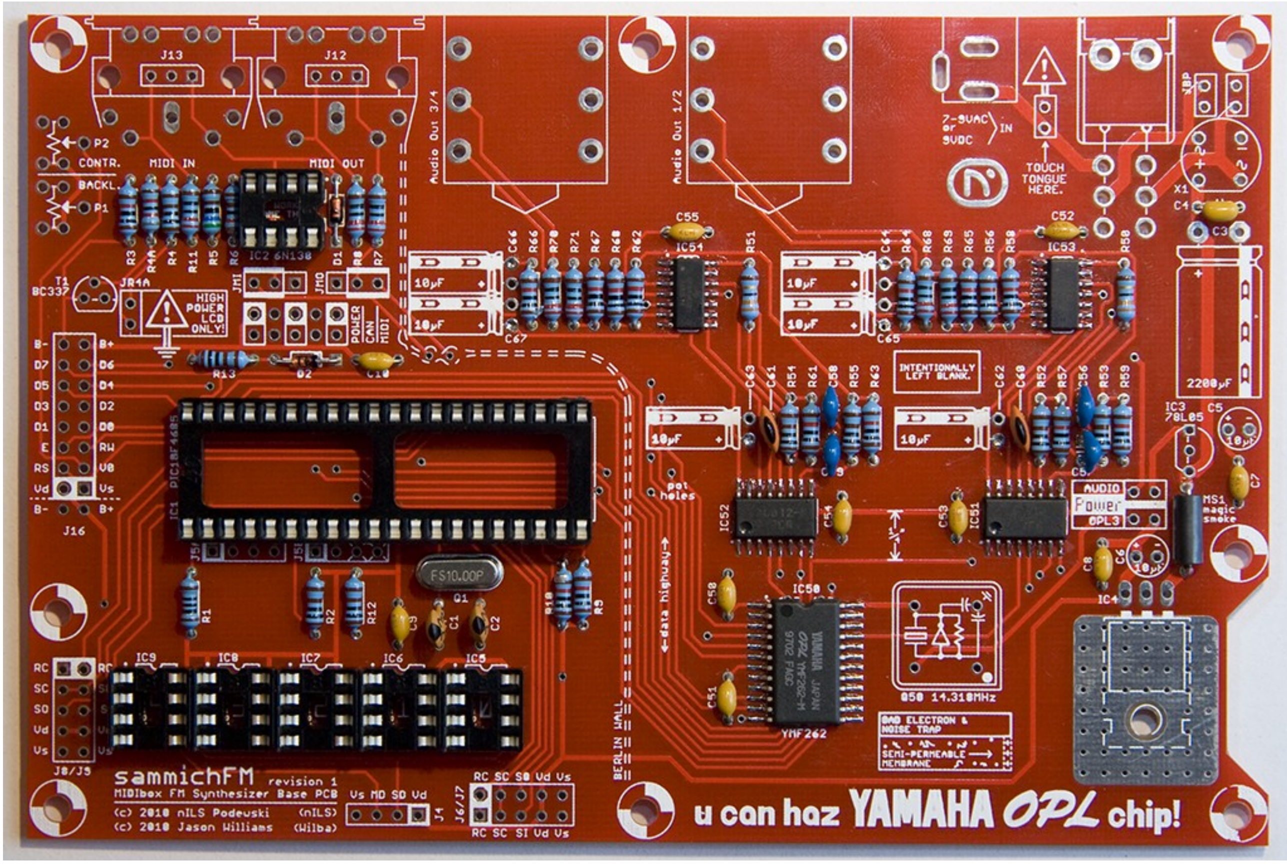
Step 1.12: Do continuity testing between all adjacent pin pairs. (Note: IC53 and IC54 each have four pairs which are supposed to be connected by tracks on top layer.)

Step 1.13 (Obsessive Completion Distinction): Do continuity testing between the top of each pin (the unsoldered part) and another pad/via on the same track, to test the solder joint is perfect.



Here is an example of the YMF262 and YAC512 soldered with "good" solder joints, IMHO. Observe how the solder fills the space behind the pin, between the pin and the pad, and a little bit stuck to the inside bend on the top of the pin. This may be more solder than what you might see in a factory assembled PCB, because they use tiny amounts of solder paste and cook it in an oven. You could remove some of this excess with solder wick, but it's not necessary.

Step 2: Solder resistors, diodes, IC sockets, ferrite bead, crystal, small capacitors.



Diodes must be oriented correctly. The stripe on the diode matches the stripe on the part outline.

IC sockets are oriented with the notch of the socket matching the notch on the part outline.

IC sockets in IC5-IC9 are best soldered together as a group so they align nicely.

Resistors do not have a correct orientation, but look nicer if oriented consistently. As you can see in the photo, I orient them so the value can be read left to right or bottom to top.

You can use the “tacking technique” (see above) for all small two-lead components, but it is more common to insert components and bend the leads outward just enough so they stay in place.

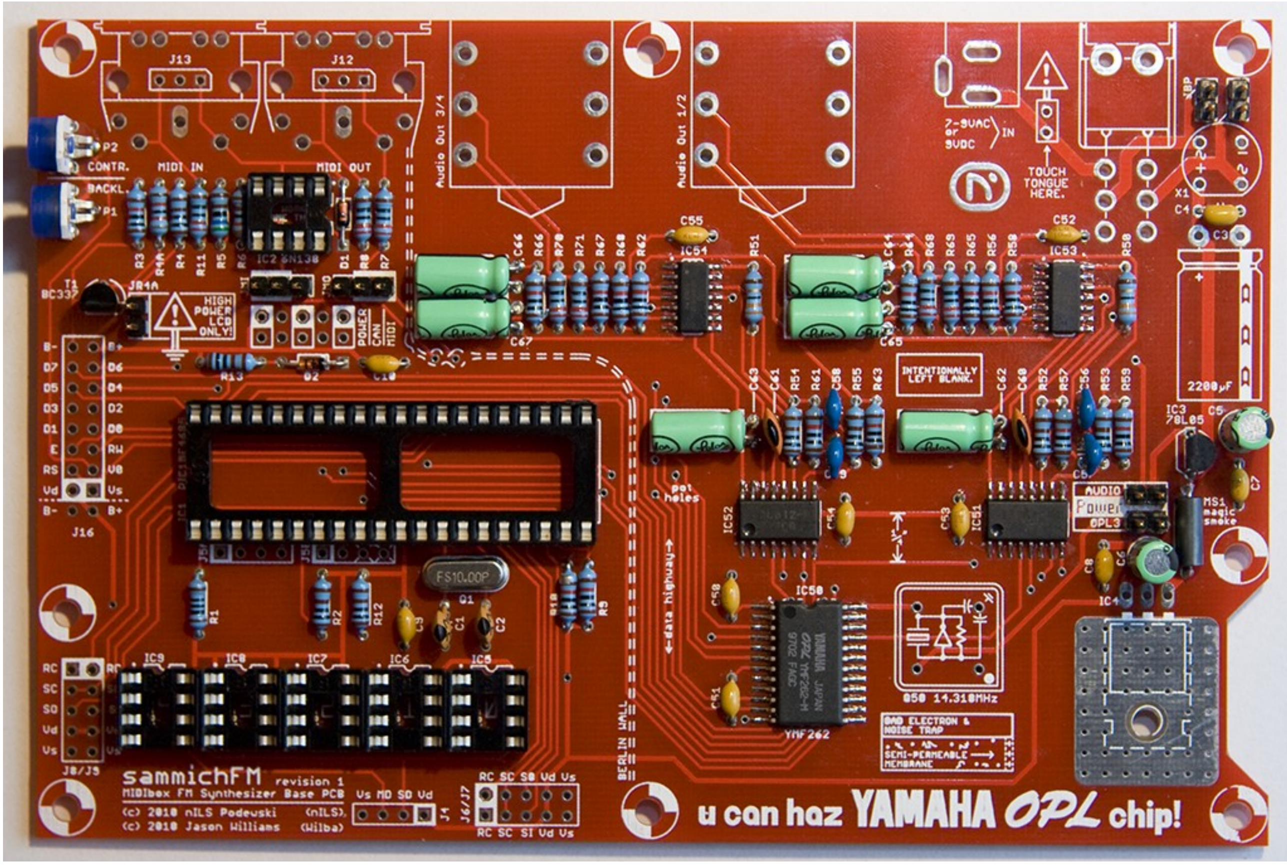
If you do the “bending leads” technique, I highly recommend cutting leads short (around 5mm) prior to soldering. This will make soldering easier, as there will be less heat dissipated through the leads, and easier to get the iron tip onto the pads.

I also recommend adding a bit of flux to ground pads (pads that are directly connected to the ground plane and are missing the dark ring around them) and starting with a blob of solder on the iron tip first, that will get the heat into the pad quickly and then feed more solder into the joint until it sucks into the hole.

Tip: Don't try to do too many components at one time!

Even if you can get all the components to stay in place while you soldered the bottom side, it's not a good idea. It's much safer, easier and probably quicker to solder all parts of one type, then trim leads before starting on another type.

Step 3: Solder small electrolytic capacitors, transistor, 78L05 voltage regulator, trimpots and header pins.



Electrolytic capacitors must be oriented correctly. The lead identified by a black stripe (with "-" inside) is the negative lead. The ***other*** lead must go in the positive pad, identified with a "+" on the part outline.

Tip: You can solder one lead of electrolytic capacitors from the top side to hold them all in place.

The transistor (T1) requires bending out of the middle lead to fit the holes. Bend it away from the flat side at 45º angle, then bend it another 45º to be parallel to other leads.

The voltage regulator (IC3) requires bending out of the outer leads to fit the holes. Bend it away at 45º angle, then bend it another 45º to be parallel to other leads.

Solder transistor/voltage regulator quickly! Do not heat the joint more than 3 seconds, and allow 20 seconds to cool between soldering transistor leads. Add flux to help solder the pads quicker.

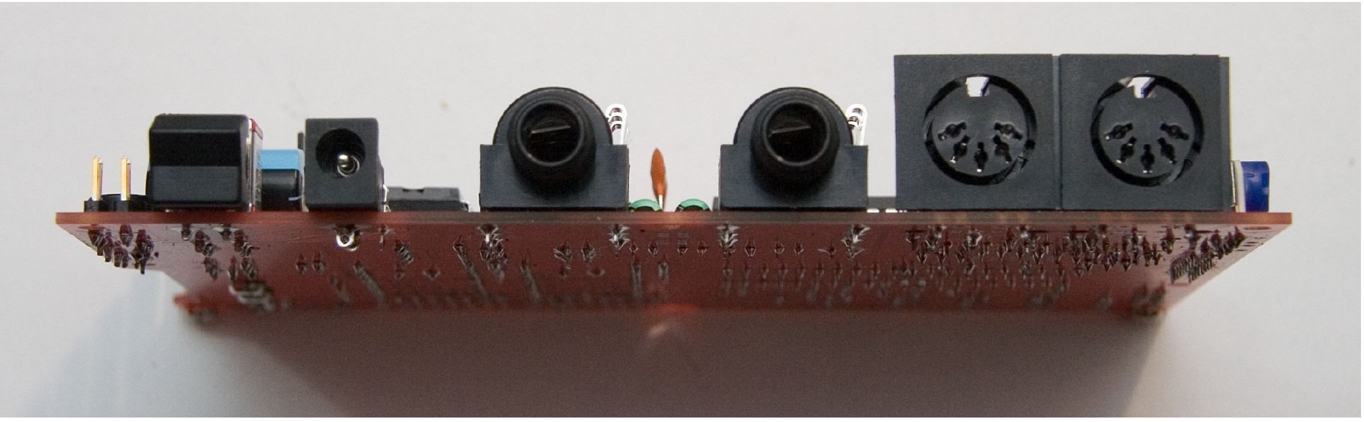
Only solder the required headers, all others are for using this PCB in other contexts.

Do not solder the big voltage regulator (IC4) now, only the little one (IC3).

Step 4: Solder DIN sockets, audio sockets, DC power socket, rocker switch, bridge rectifier, crystal oscillator, big fat 2200uF electrolytic capacitor.

NOTE: Rear panel sockets must be soldered perfectly aligned so that the rear panel will be aligned with the bottom panel. Read Step 4 instructions entirely before you start soldering.

All rear panel sockets (and switch) are soldered flat against the PCB. There should be ***no gap*** between the component and the PCB. In particular, the audio sockets and power socket (since they protrude through the rear panel) will cause rear panel misalignment if there is a gap. (By "no gap" I mean pushed into the PCB as far as possible, the audio sockets might have a ***very tiny <1mm gap*** between the plastic and the PCB)

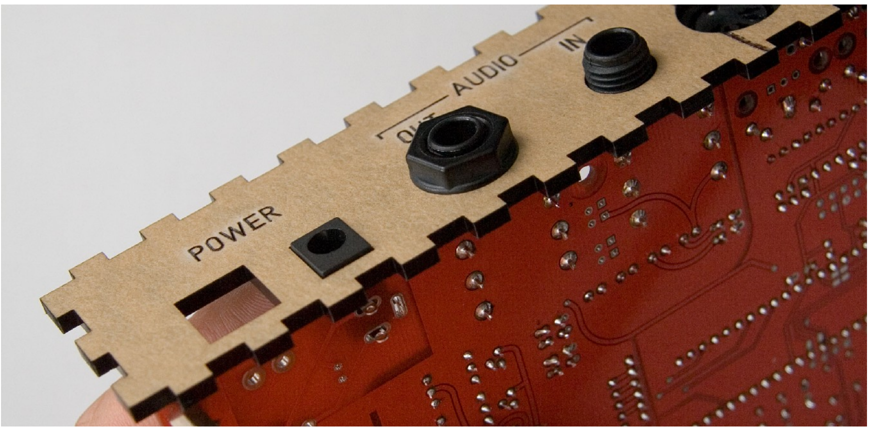


Solder only ***one pin*** of each socket to hold it in place, then you can check for correct alignment before soldering the other pins.

Use a "tacking" method – i.e. while holding the socket/PCB together with one hand, apply a blob of solder to one pin. This is a temporary joint – be sure to solder the joint properly after soldering the other pins.

Solder both DIN sockets (MIDI In, MIDI Out) at the same time to ensure they are both level with the PCB edge.

After soldering ***one pin*** of each ***audio socket***, attach the rear panel using the plastic nut on the audio socket. Tighten so it is touching the PCB edge. This helps align the DC power socket, which has a tight fit in the panel and is loose in the PCB's pad holes. It should stick out 0.5mm because when the case is fully assembled, there is a gap of 0.5mm between the PCB edge and the rear panel. You can now solder ***one pin*** of the DC power socket using the "tacking" method.



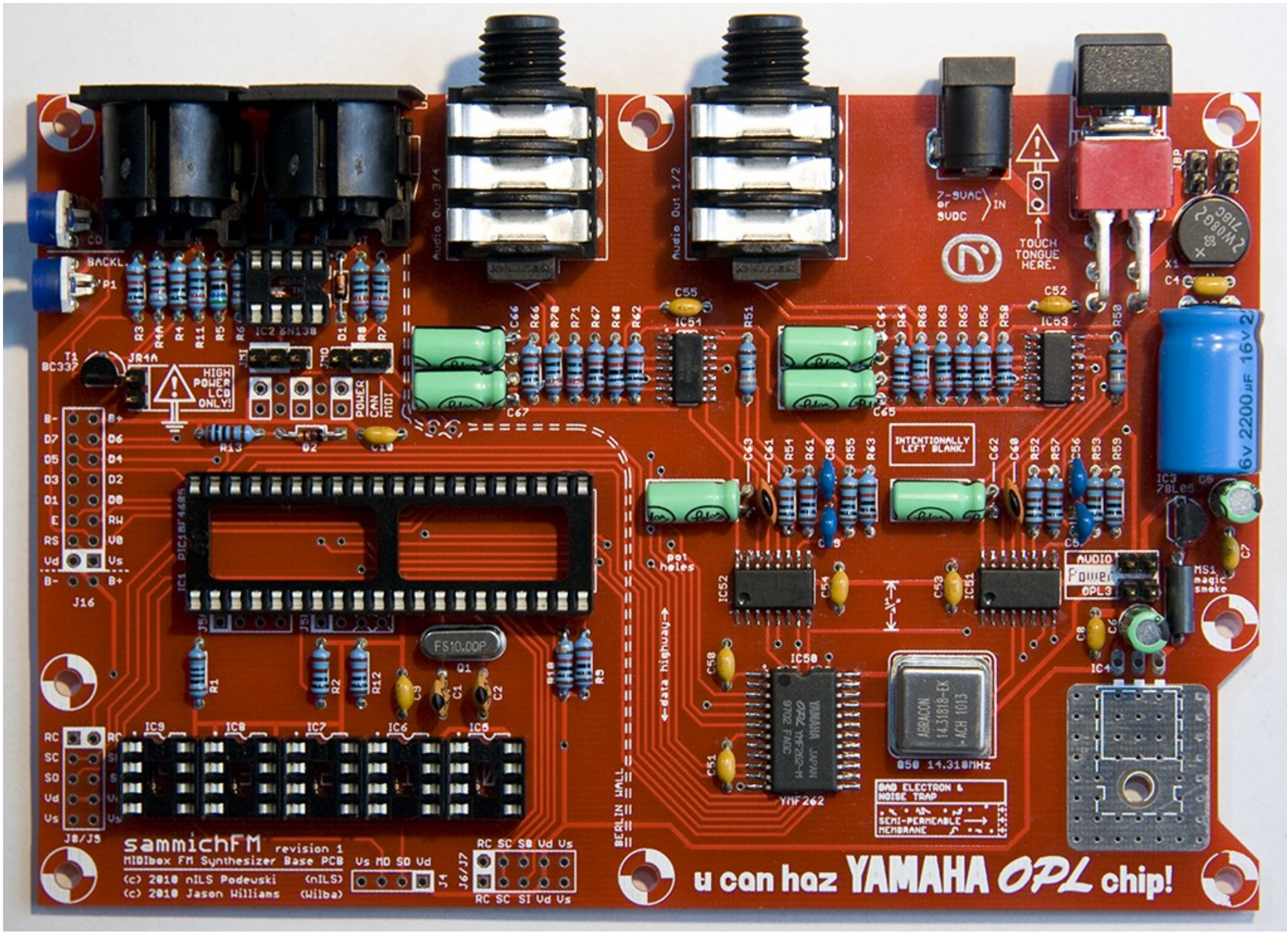
After “tacking” all the sockets, you can now check for correct alignment by placing the PCB onto the base with screws/nuts attached and adding the corner spacers (see **Case Assembly** section) and then attaching the rear panel. You should check that the power socket and audio sockets are not causing the rear panel to have a gap where it joins the bottom panel. If they do, you can heat up the “tacked” joint while you realign the socket.

Solder the other pins of each socket with proper solder joints (not “tacking”), before resoldering the “tacked” joint. Don't be shy with the solder, these joints can take a lot because they are big, and they are the ones that will receive stress from plugs going in and out.

Follow the same process for the power switch, “tack” it on, check for alignment, solder the other pins, resolder the “tacked” pin. You may need to bend the leads slightly to make it fit. Solder it as far towards the rear panel as possible (as well as centered and aligned with the outline on the PCB).

Now solder the bridge rectifier, crystal oscillator and big fat 2200uF electrolytic capacitor.

Here's what you should have when you finish. Everything soldered to the base PCB except the voltage regulator on the right and the female headers on the left.



Now is a good time to check for shorts between 5V and GND before you solder the voltage regulator, so you can rule out the voltage regulator as a cause (removing it later is painful!)

Step 5: Assemble voltage regulator and heatsinks.

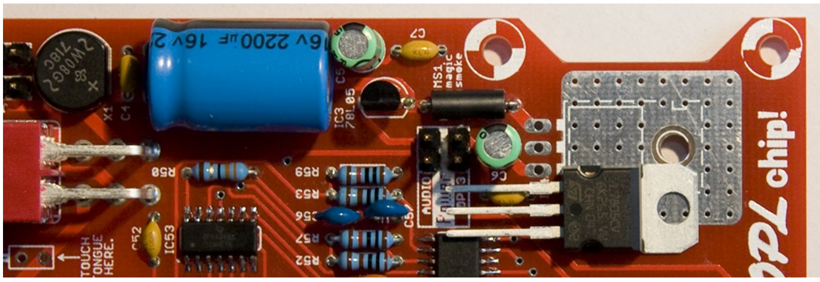
This step is fiddly and requires long-nose pliers and a philips screw driver. It is important not to overtighten the screws holding the voltage regulator to the heatsinks.

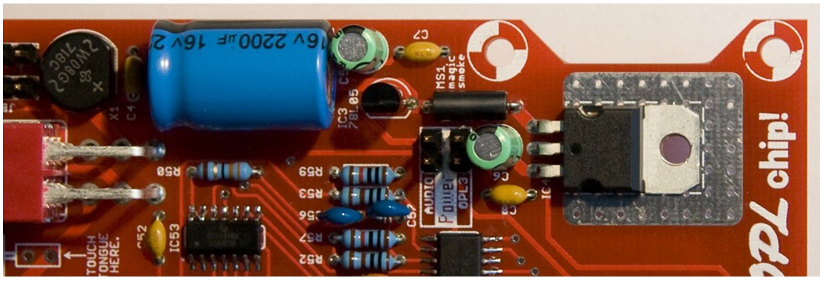
NOTE: Avoid using conductive heatsinking compound like Arctic Silver or some other fancy stuff that PC power users use between their PC's CPU and their custom CPU heatsink with all the fins and fan with LED bling etc. Conductive heatsink might cause shorts between pins of the voltage regulator, which might cause blue smoke, sparks, fried voltage regulators, fried power supplies, and a trip to the local electronics shop to buy more parts.

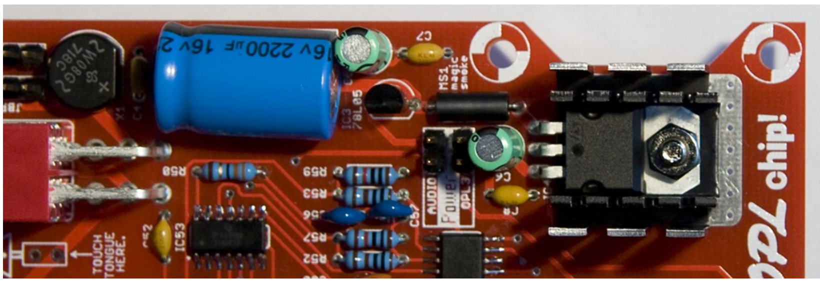
NOTE: Don't use heatsinking compound between the PCB and big heatsink. There is not much to gain from using it here and it will only make a mess due to the holes in the PCB.

Step 5.1: Mark where to bend leads of the voltage regulator and then bend to fit. Ideally, voltage regulator should have leads all bent exactly the same, so it will easily “fall into” the holes and also have the tab's hole aligned with the hole on the PCB.

Step 5.2: Practice assembling the heatsinks and voltage regulator **without using heatsinking compound!** i.e. do steps 5.3, 5.5, 5.7, 5.8, 5.9, 5.10, disassemble and repeat using heatsinking compound.







Step 5.3: Place big heatsink onto PCB

Step 5.4: Apply ***non-conductive*** heatsinking compound to bottom of small heatsink.

Step 5.5: Place small heatsink onto big heatsink, align holes.

Step 5.6: Apply ***non-conductive*** heatsinking compound to back of 7805 voltage regulator (IC4)

Step 5.7: Place 7805 voltage regulator (IC4) onto small heatsink.

Step 5.8: Screw together 7805 and heatsinks using 9mm screw coming from the bottom side. This is easiest done by holding screw in position with a finger while attaching and tightening nut using long-nose pliers, until the nut is on the screw. Thereafter, use a screwdriver on the screw while stopping the nut from turning with a finger, and then later, with pliers.

Step 5.9: While nut is firm but not tight, you can move the heatsinks and voltage regulator to be neatly aligned. Ensure big heatsink is not touching the pads of the voltage regulator.

Step 5.10: Tighten nuts, but don't overtighten nuts :-)

Step 5.11: Cut leads of 7805 (on the bottom side) to be 2mm.

Step 5.12: Solder 7805. Allow 20 seconds for part to cool between soldering leads. The middle pin is connected to the ground plane and will be harder to solder. Starting with a blob of solder already on the iron tip, and holding the tip at a smaller angle to the PCB (i.e. more horizontal).

Step 5.13: Clean up any excess heatsinking compound

Step 5.14: Check that you did not create any shorts between any of the pads you just soldered (or from using conductive heatsinking compound even though I told you five times already not to use it.)

Step 6: Attach LCD to Control Surface PCB

To achieve “perfect” alignment, first attach loosely on two diagonally opposite mount holes, then move it until the hole underneath is centered in the hole above.

You can also check if the LCD's PCB edge is parallel with the edges of the ground plane gap of the control surface's PCB (i.e. the darker red shape on the bottom which is the exact size of the LCD).

When you have all the screws tightened, check alignment again. The LCD can move on the mounts during tightening of screws.

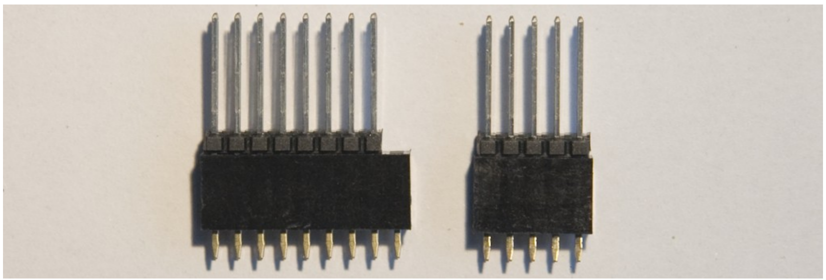
Good alignment is critical in making the control surface PCB easily “plug into” the base PCB.

Step 7: PCB Interconnection Stage

Now here comes the fun bit!

You are going to attach the two PCBs together so they're perfectly aligned while you solder the male and female headers to both PCBs.

Step 7.1: Prepare the male and female headers. The shorter end of the male header goes into the female header.



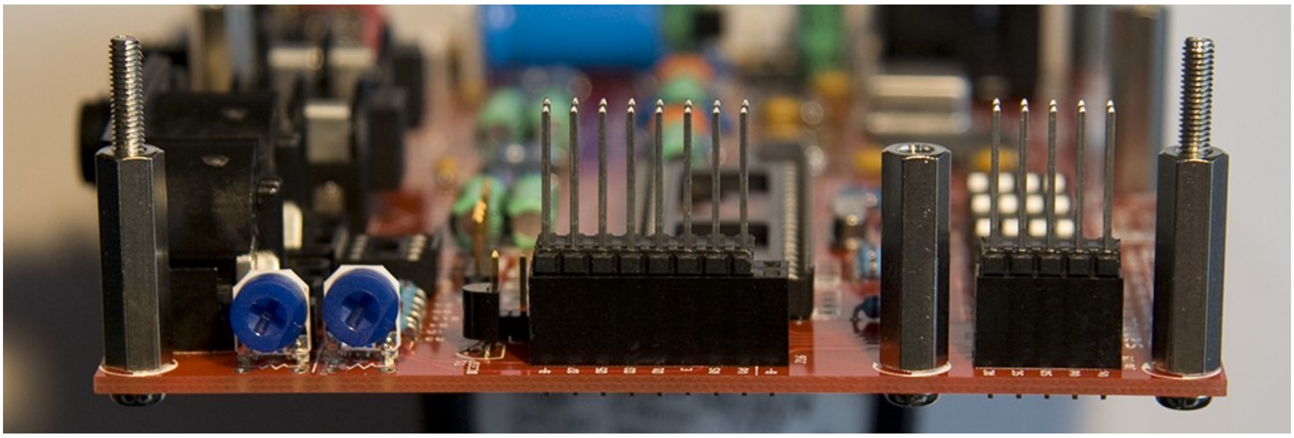
Note: depending on the LCD type, the 16 pin male header goes into the “upper” or “lower” 16 pins of the 18 pin female header. Most of the time it will be the “upper” 16 pins, as shown by the white outline on the base PCB. As far as I know, only the Optrex STEP LCDs require it in the “lower” 16 pins, because the backlight pins are at the other end of the header.

Step 7.2: Attach 32mm screws and 20mm spacers to base PCB corners

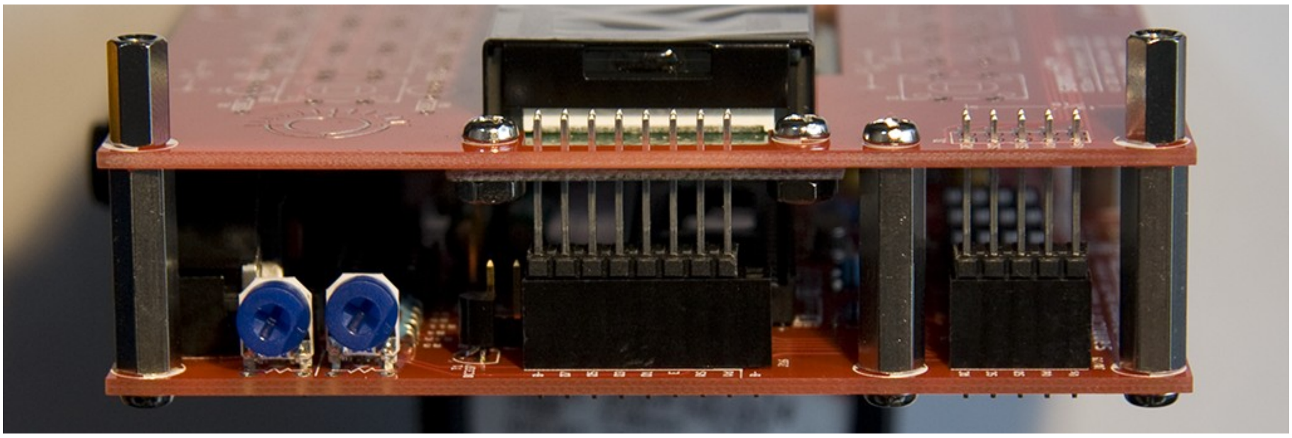
Step 7.3: Attach 9mm screws and 20mm spacers to base PCB side midpoints

Step 7.4: Insert female headers (with male headers inserted) into base PCB.

Note: as described above, for most LCDs, the male header will be in the "upper" 16 pins of the female header (i.e. on the left side, as shown in this photo). If using an Optrex STEP LCD, it will be in the "lower" 16 pins (i.e. on the right side). There's only one way your LCD can align and fit (when mounted to the control surface PCB) so don't worry too much about it unless you plan to connect the LCD while it is not mounted to the control surface PCB.



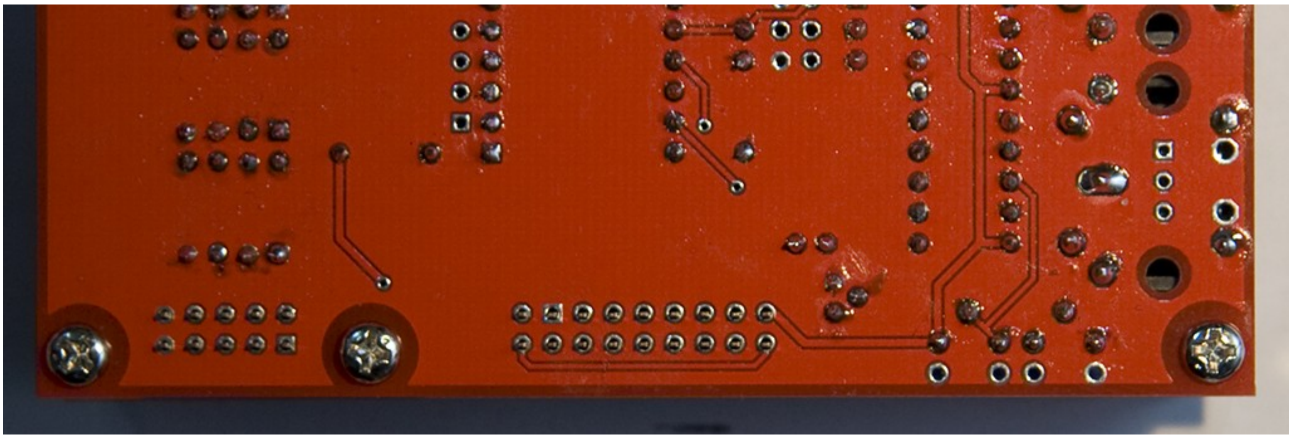
Step 7.5: ***Slowly and carefully*** place control surface PCB onto corner screws and lower down so male header pins go through holes in LCD and control surface PCB. Attach with 10mm spacers at corners and 3mm screws at side midpoints.



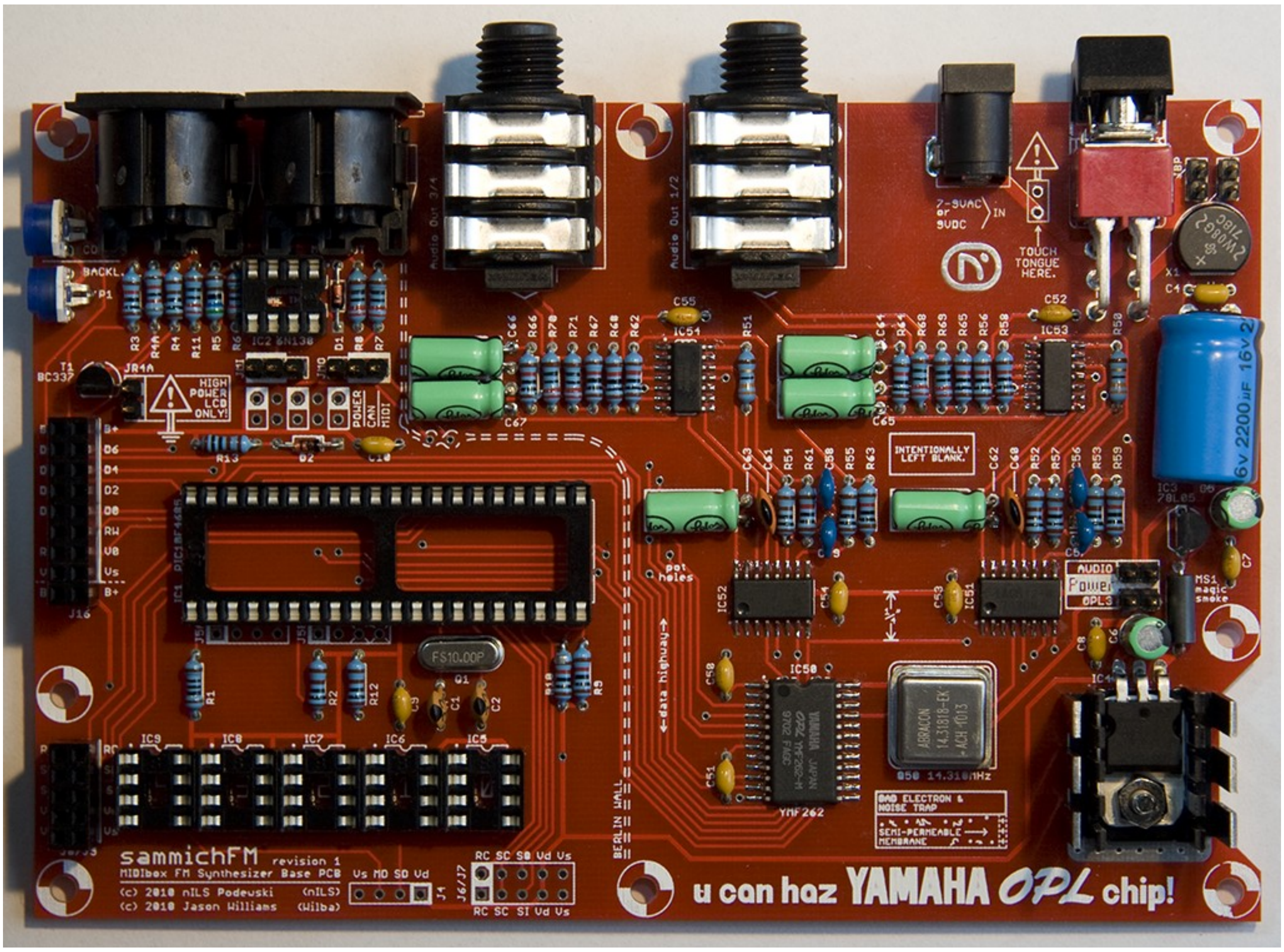
Step 7.6: Solder two diagonally opposite pins on each header on the top side, ensuring female header is flat against base PCB.

Step 7.7: Have another look from the side. Everything looks good? Female headers are still flat against base PCB? Good. ***Finish soldering the headers on the top side.***

Step 7.8: Solder all the pins of the female headers on the bottom side..



Step 8: Congratulate Yourself for Finishing Soldering of the sammichFM Base PCB!



3. Control Surface PCB Soldering Walkthrough

Step 0: Check if 5V and GND are shorted/connected.

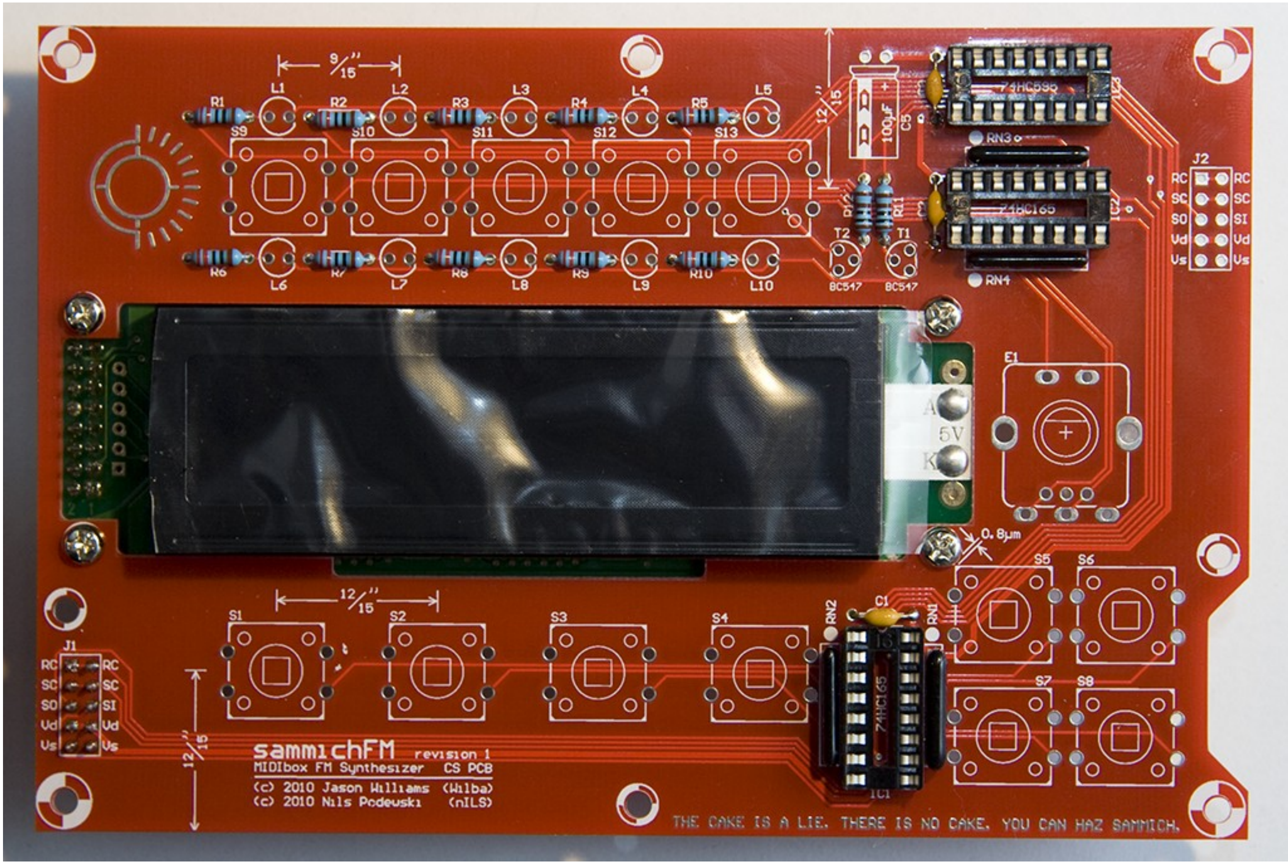
In the unlikely event that your PCB has a manufacturing fault, it is a lot easier to fix if you know the PCB is at fault and not your soldering or the components. Check between J1:Vd and J1:Vs.

Step 1: Solder the resistors, small capacitors, IC sockets, resistor networks

Insert and solder parts in the order they are presented in the Parts List.

NOTE: Pay attention to the orientation of resistor networks! Match the dot on the resistor network with the dot on the PCB.

Tip: I use a small piece of card (i.e. business card) between the resistor network and the IC socket to keep it vertical while "tacking".

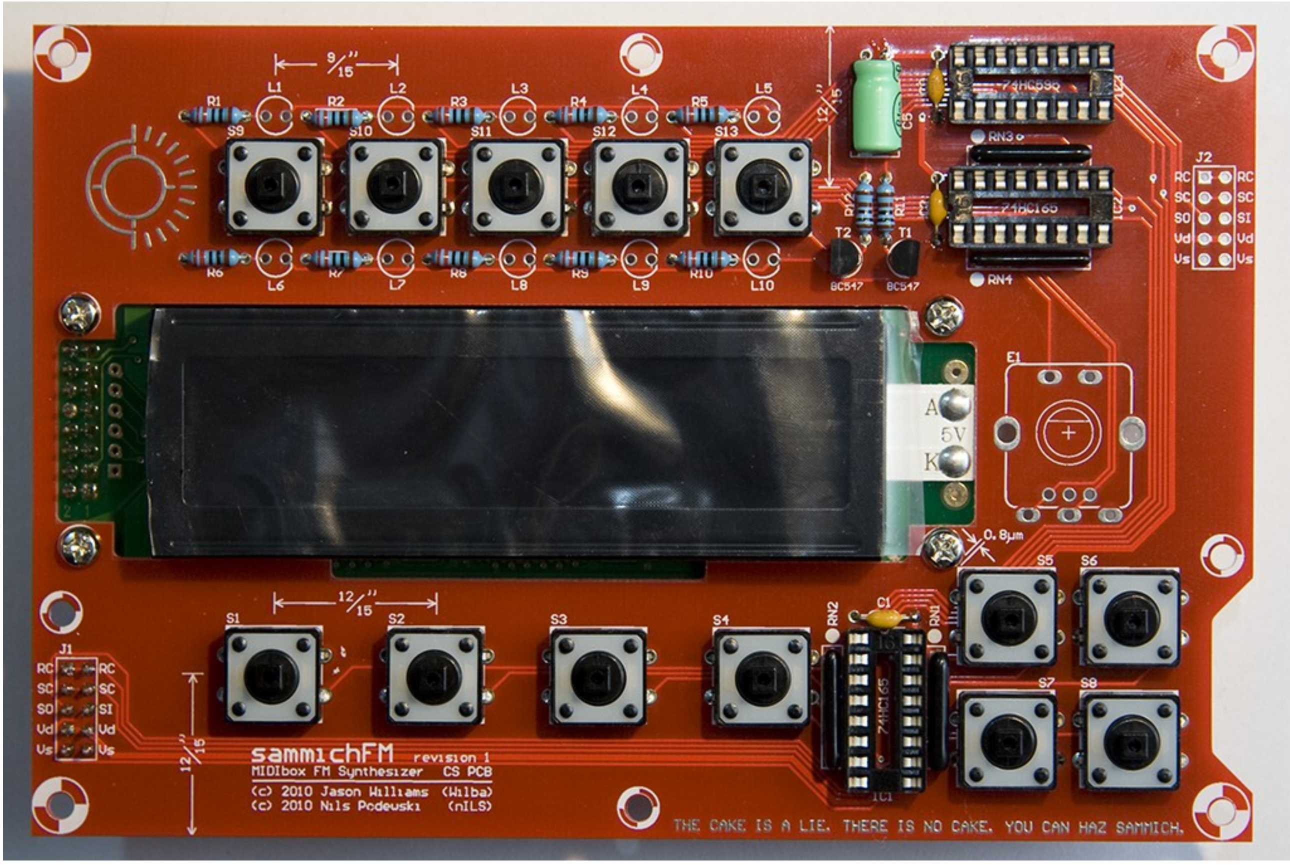


Step 2: Solder electrolytic capacitor, transistors, tactile switches

Insert and solder parts in the order they are presented in the Parts List.

As with the base PCB, check orientation the electrolytic capacitor before soldering.

NOTE: The leads of some switches (see arrows on bottom side of PCB) might touch the heatsink. You should cut only the leads indicated by the arrows as short as possible before soldering, not all of them!



Step 3: Soldering LEDs

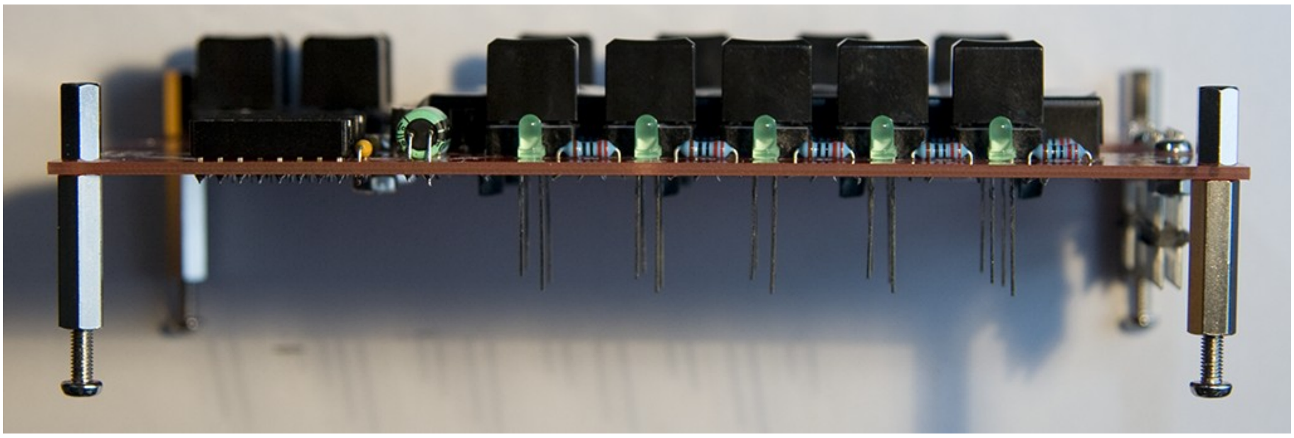
Now here comes another fun bit!

The objective here is to solder all the LEDs at ***exactly the same height*** above the PCB. If any of them are shorter, it is very noticeable, as our brains notice tiny differences in regular patterns. Take things slowly.

Step 3.1: In the corners, mount 20mm spacers to bottom, 10mm spacers to top, using 32mm screws. Allow enough thread in the top of the 10mm spacers so you can screw on the top panel (i.e. 5mm)

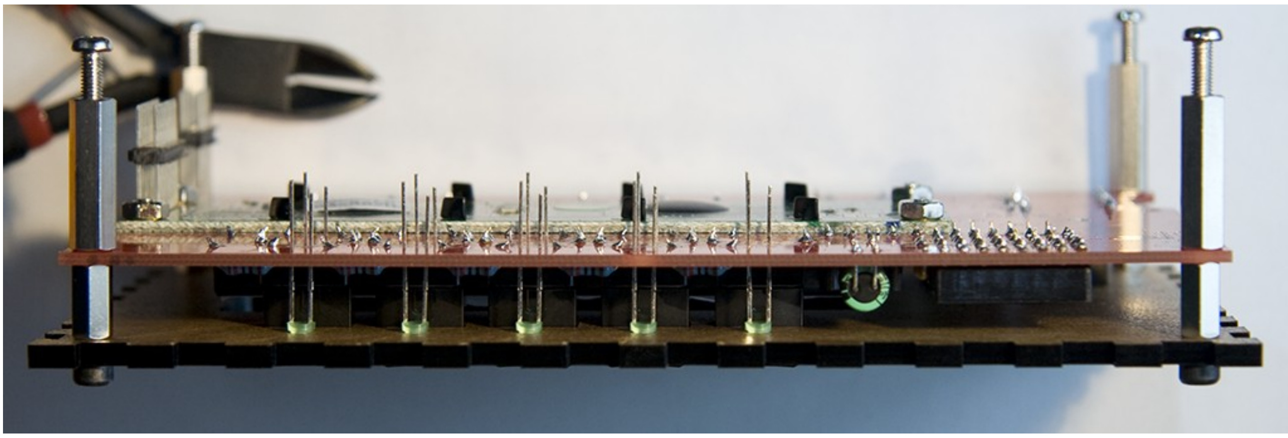
Step 3.2: Place switch caps on the switches. This helps ensure you have the panel aligned with the PCB in a later step. They require a good bit of force to snap onto the switch, so don't be concerned about this.

Step 3.3: Insert the LEDs and push them all the way to the PCB. Align the flat side of the LED with the flat side of the part outline. The flat side is the cathode, which is also the shorter lead. The other side is the anode which is the longer lead. So alternately, place the longer leads into the left hole when the PCB is facing you the right way up (i.e. with LEDs at top, buttons at bottom).



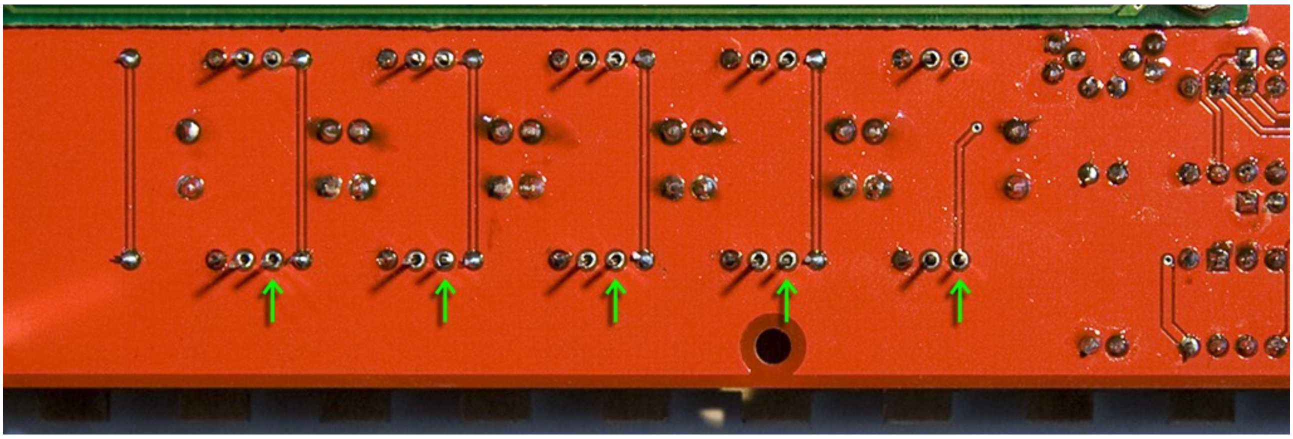
Step 3.4: Attach top panel to PCB using black screws. Check alignment by looking at the switch caps.

Step 3.5: Turn PCB upside down. ***Carefully and gently*** push LEDs into the holes as much as possible.



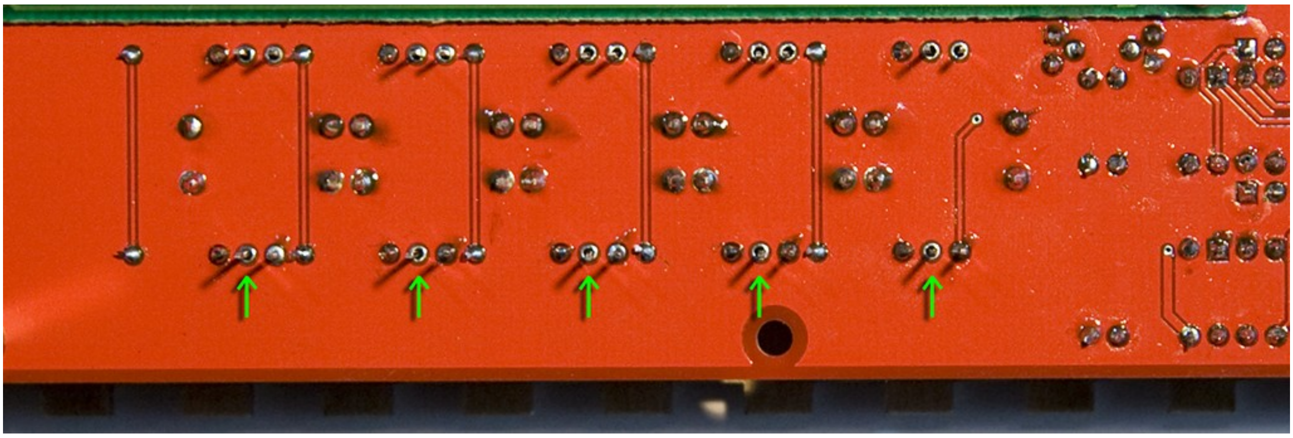
Step 3.6: Look underneath at the LEDs and check they all protrude the same amount. If any do not, it could be that a LED is stuck because the hole is slightly smaller or the LED is slightly bigger. Disassemble PCB and panel and check that LED individually. It is possible that a LED has a little bit of excess plastic from molding imperfections, which can be scraped away using a craft knife.

Step 3.7: Solder all the shorter leads of one row of LEDs. Push down on the longer lead with your middle finger while soldering, if you have one. Work quickly! No more than 3 seconds per joint! If you must resolder a joint, allow LED to cool for 20 seconds.



Step 3.8: Look underneath at the LEDs and check they all protrude the same amount. If not, heat up a joint and push down on the longer lead (there's a reason I told you to solder the shorter lead first!)

Step 3.9: Solder all the longer leads of that row of LEDs.



Step 3.10: Cut all the leads of that row of leads, as short as possible. This gives you room to solder the other row.

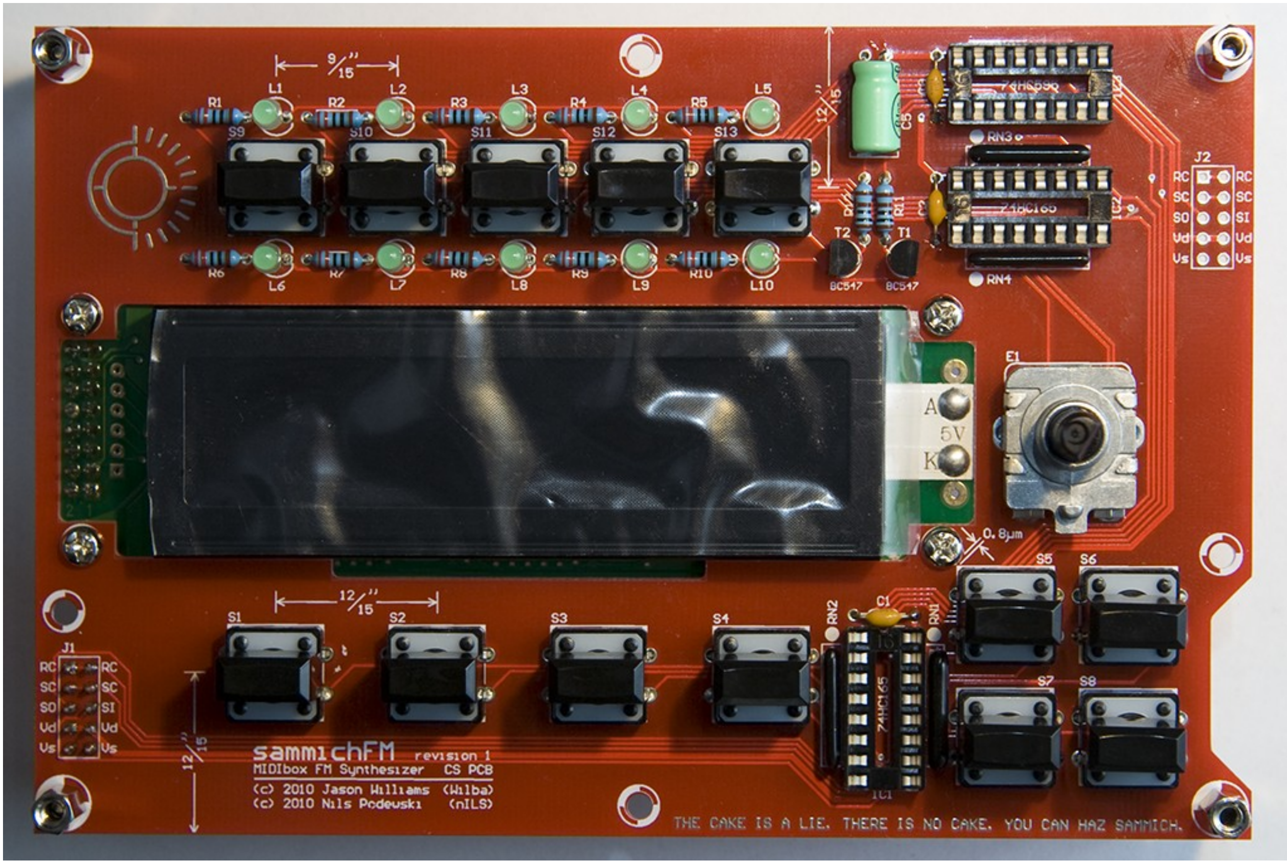
Step 3.11: Repeat steps 3.7 to 3.10 for the other row of LEDs.

Step 4: Solder rotary encoder

You solder this last so that the shaft doesn't get in the way while soldering LEDs (i.e. it can sit flat). Now you need it to sit flat while soldering the rotary encoder. To avoid bending LEDs (and if you don't have a vice), invert the spacer/screw combos so the 20mm spacer is now on the top side.

NOTE: The leads of of the rotary encoder might touch the heatsink. You should cut the leads as short as possible before soldering.

Step 5: Congratulate Yourself for Finishing Soldering of the sammichFM Control Surface PCB!



4. Jumper Configuration

MIDI Routing Jumpers

Headers **JMI** and **JMO** control whether the MIDI In and MIDI Out ports are connected to either the PIC18F4685 or the “expansion port” pads below it.

Insert a shunt in **JMI** and a shunt in **JMO** where the thick white line indicates the default (i.e. connected to the PIC).

Backlight Voltage/Current Jumpers

Header **JR4A** controls whether the backlight current is limited to 25mA (open) or approximately 250mA (closed). The header actually shorts resistor **R4A**, which is an 81K resistor in series with R4 in the traditional MIDIbox Core backlight circuit. Thus, leaving this header open will enable **R4A** and reduce the current, a shunt in this header will disable **R4A** and the current will be higher.

Low-power Backlight LCD (i.e. Edge-lit)

If you are using a low-power backlight LCD that requires only 25mA, then ***do not*** put a shunt in **JR4A**!

High-power Backlight LCD (i.e. LED array)

If you are using a high-power backlight LCD that requires 250mA or more, then put a shunt in **JR4A** to increase current. Turn brightness trimpot to half-way (12 o'clock), this gives the optimal current for a high-power backlight LCD.

Audio/OPL3 Power Jumpers

The two headers to the right of **IC51** (YAC512) are for supplying 5V power to the audio and OPL3 sections of the PCB. These headers are protect the surface mount ICs until you're confident there are no problems elsewhere on the PCB, i.e. voltage tests pass, you can upload the firmware, etc.

5. Power Supply

sammichFM contains two 5V voltage regulators (7805 and 78L05) A typical LCD backlight at 50% brightness (a good setting for good contrast) draws 100mA. The rest *probably* draws less than 100mA.

The voltage regulators should receive at least 7V for good regulation. If supplied with DC power, going through the bridge rectifier will drop voltage by approx. 1.2V.

Thus *ideally*, you should provide sammichSID with *regulated* 9V DC (500mA+) supply. That would ensure that the voltage going into the voltage regulators is good (approx. 7V), keeping the heat dissipation down to absolute minimum (7V - 5V = 2V, 2V x ~200mA current = 0.4W)

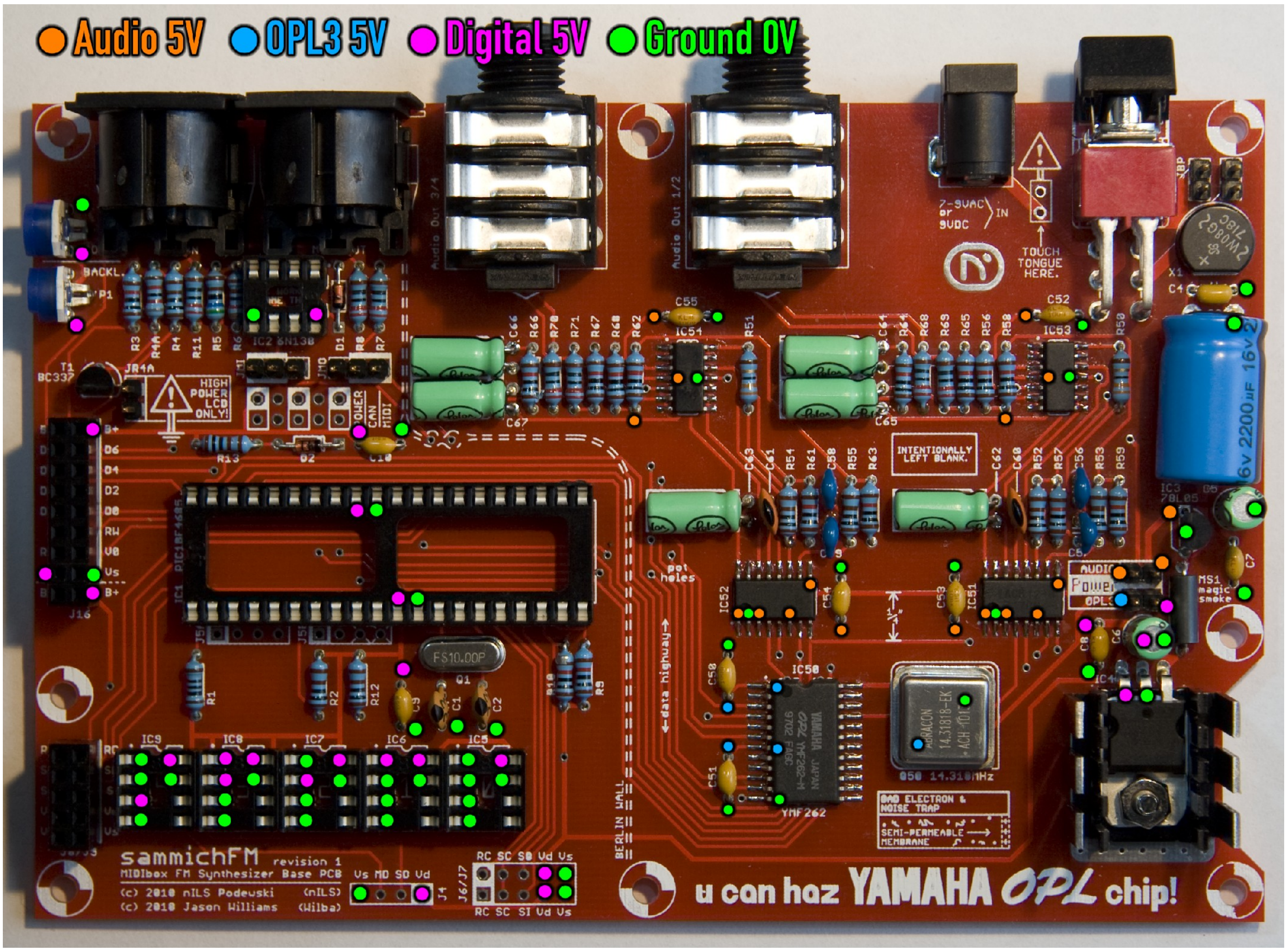
If you can't get a regulated 9V DC supply, then you can also use unregulated DC or AC, but suitable voltage/current ratings can vary. For example, a 9V AC 200mA supply would be good, but a 9V AC 500mA would be loaded so lightly that the voltage going into the regulators might be 15V or higher, which would be bad. Also, something like a 5V AC 500mA or higher *might* work, since it is loaded so lightly that the voltage would be over 7V at the voltage regulator inputs.

Avoid switchmode power supplies! They generate far too much noise in the audio. I have yet to find one that works for sammichFM. Aim to get a transformer based (linear) power supply.

Test voltage going into the voltage regulators! You should do this while sammichFM has the control surface attached. Measure between **IC3** input pin (or pins of **MS1**) and ground pin of **C7** (left side), which is accessible from the right side when PCBs are connected.

6. Voltage Tests

Before inserting any of the ICs for the first time, you should perform some basic voltage tests.Step 1: Voltage Tests on the Base PCB



Once you've tested 5V between one pair of pads (such as **J4** at bottom), you can leave the black probe on **GND** at **J4** while you test 5V at other points marked with pink dots. Inversely, you can leave the red probe on **+5V** at **J4** while you test for 5V at other points marked with green dots.

Alternately, you can test voltages once at **J4**, etc. and then ***with power unplugged***, use your multimeter's “continuity tester” (i.e. multimeter beeps when probes touch). Test continuity between one point and all other points that should be connected (i.e. all green dots).

NOTE: “Audio 5V” and “OPL3 5V” are supplied via shunts in the Audio/OPL3 Power headers, which in turn is supplied by IC3 and IC4. After putting the OPL3 shunt in, “Digital 5V” is connected to “OPL3 5V”.

Step 1.1: Test for **NO** continuity (no shorts) between “Audio 5V” and Ground (with power off)

Step 1.2: Test for **NO** continuity (no shorts) between “OPL3 5V” and Ground (with power off)

Step 1.3: Test for 5V on **IC3** pin marked (with power on)

Step 1.4: Insert both shunts in Audio/OPL3 Power header.

Step 1.4: Test 5V on pins marked with and . They may be slightly different to each other and both not exactly 5V but close (5V ± 0.05V).

Step 2: Voltage Tests on the Control Surface PCB

You can now attach the control surface PCB and perform voltage tests while it is connected.

Step 2.1: Check **JR4A** is set correctly for your LCD (See **Jumper Configuration**)

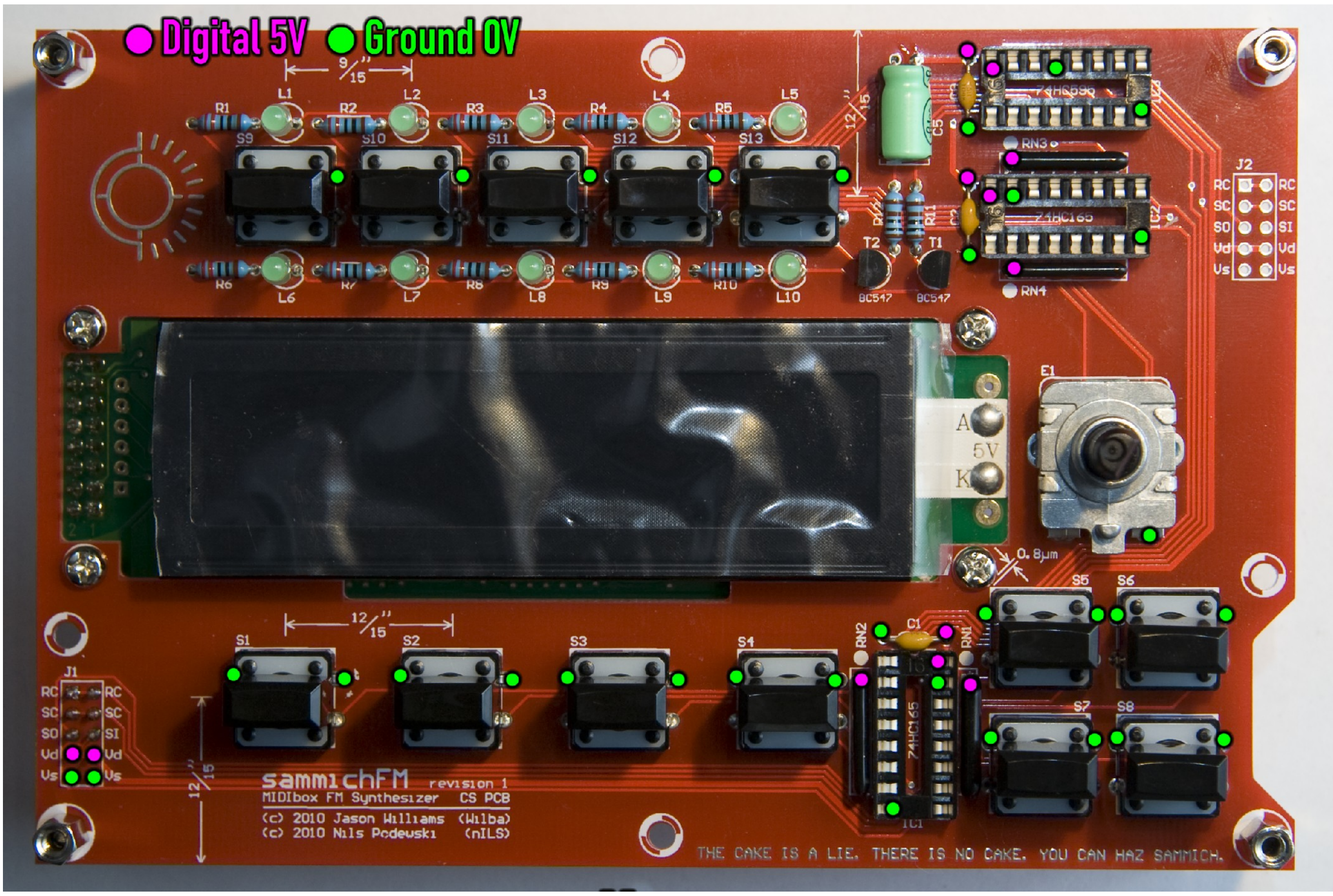
Step 2.2: Disconnect power.

Step 2.3: Assemble the case with the base PCB and control surface PCB attached (see **Case Assembly** section).

Step 2.4: Turn the contrast trimpot fully anticlockwise. Turn the brightness trimpot fully anticlockwise.

Step 2.5: Connect power and turn power on. You should see the backlight on the LCD and a row of black rectangles.

Step 2.6: Perform voltage tests on the control surface PCB.



7. Installing ICs

ICs are manufactured with the leads spread out. Before inserting into IC sockets, you will need to bend the leads so they fit. While holding the IC firmly, use your work surface to bend all leads simultaneously. Do this a little bit at a time until it looks right, then check to see if it will line up with the middle of the pins of the IC socket. It is important that all the pins are vertical and parallel, as insertion requires a good deal of force and if any pins are not aligned correctly, they might bend or break.

Perform voltage tests before installing ICs the first time.

Always install ICs with the power disconnected.

Carefully check orientation of the IC. The notch on the IC must match the notch on the white outline on the PCB (which should also match the notch on the IC socket, if you soldered that oriented correctly).

I recommend installing the "Bankstick" ICs (**IC5**, **IC6**, **IC7**, **IC8**, **IC9**) after fully completing both PCBs, uploading MIDIbox FM firmware and having a fully working control surface. This allows you to watch the formatting of the Banksticks by the MIDIbox FM firmware.

8. Initial Testing and Firmware Upload

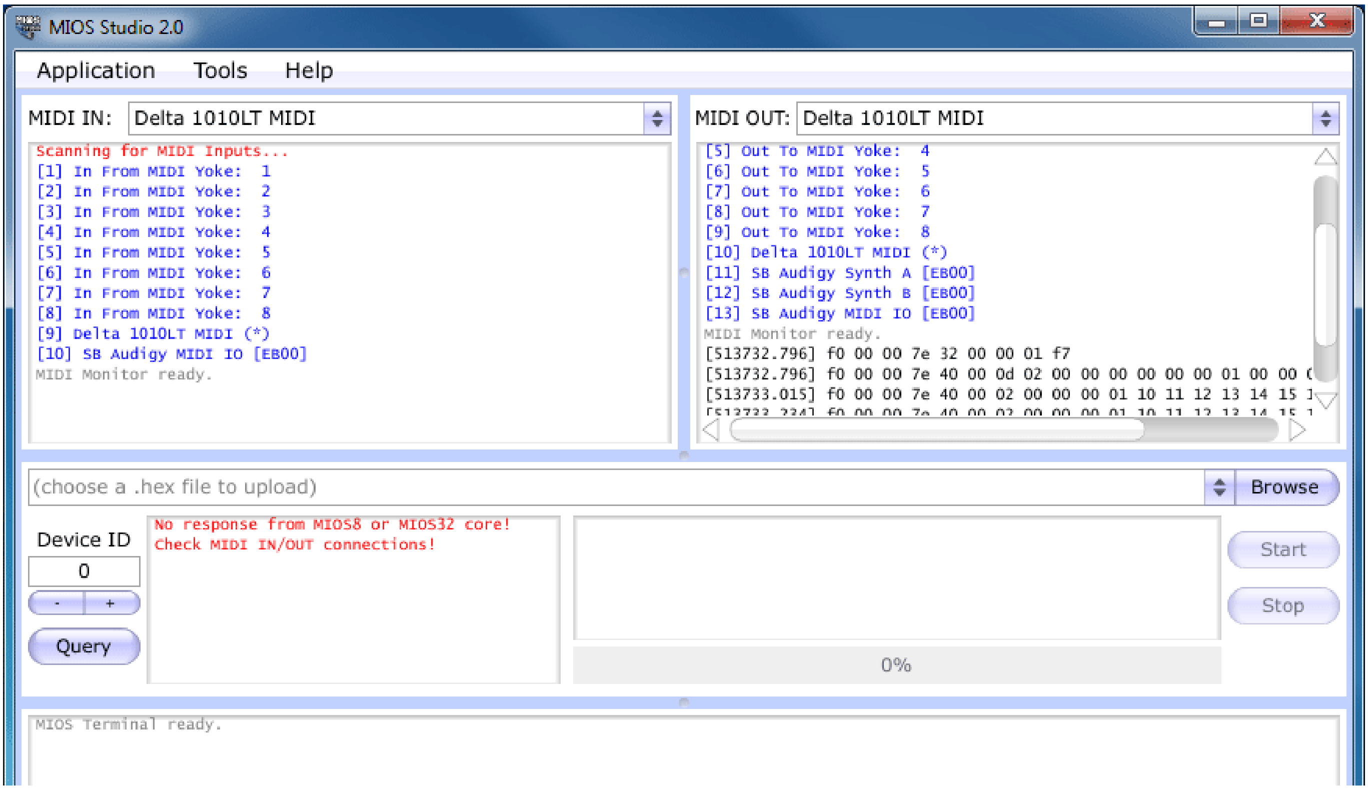
Step 1: Installing MIOS Studio 2 and Testing MIDI

Install MIOS Studio 2 from here: http://www.ucapps.de/mios_studio.html

Connect MIDI cables between sammichFM and PC. Connections should be PC MIDI Out to sammichFM MIDI In and PC MIDI In to sammichFM MIDI Out.

For the first test, leave sammichFM power off when starting MIOS Studio.

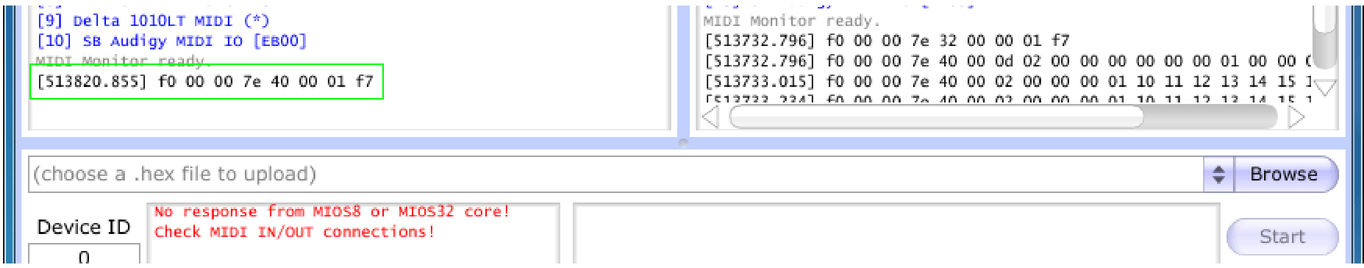
Run MIOS Studio 2. You should see a screen like this:



Set the MIDI IN and MIDI OUT combo boxes (at top) to match the MIDI interface/ports you are using to connect to sammichFM.

Power on the sammichFM!

You should see a single upload request message in the MIDI In window:

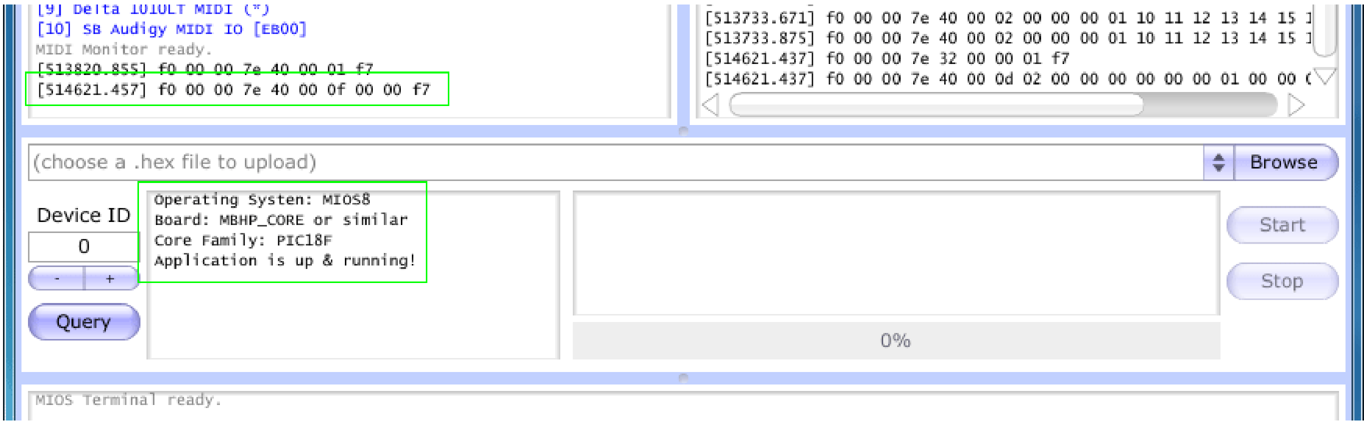


This is because the PIC18F4685 has already been burned with MIOS 1.9g. If you get this message, then you know the PIC is working and MIDI Out is working. MIOS installation can also be confirmed by the LCD showing the MIOS startup message.

If you do not receive the upload request, then check the JMO header has the shunt installed, the MIDI cables are correctly connected and match the MIDI In/Out interface/ports selections in MIOS Studio.

Refer to http://www.ucapps.de/howto_debug_midi.html for more MIDI troubleshooting advice.

Now test if the PIC is receiving MIDI In. Click on the “Query” button.



MIOS Studio will send MIDI SysEx message to the sammichFM, and expect a SysEx message response. If all is good, MIOS Studio will report that the “Application is up & running!”. This means the PIC is receiving MIDI and responding by sending MIDI, which is received by the PC.

If you do not see this message, then check the JMI header has the shunt installed, the MIDI cables are correctly connected and match the MIDI In/Out interface/ports selections in MIOS Studio.

Refer to http://www.ucapps.de/howto_debug_midi.html for more MIDI troubleshooting advice.

Step 2: Uploading the MIDIbox FM application

You can find the latest MIDIbox FM Synth V1 firmware on the ucapps.de website:

http://www.ucapps.de/mios_download.html

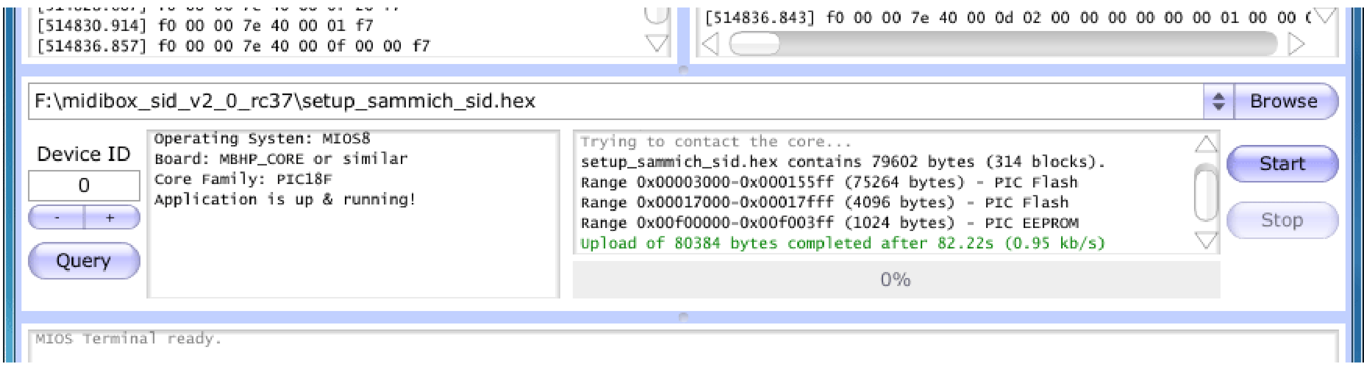
For example, the latest version at time of writing this guide is:

http://www.ucapps.de/mios/midibox_fm_v1_4a.zip

Download the ZIP file and extract it.

The sammichFM firmware is in the **setup_pic18f4685_sammich_fm.hex** file. Only upload this file; the other .hex files are for different hardware configurations.

Select the file with the “Browse” button and click the “Start” button.



(Yes, I know this screenshot shows **setup_sammich_sid.hex** but I'm too lazy to do another screenshot. **Only use setup_pic18f4685_sammich_fm.hex!**)

If there are “ignorable errors” reported (i.e. packets were retried), then try uploading again until you get no “ignorable errors”.

After the firmware has finished uploading, the sammichFM will reboot and show the MIDIbox FM bootscreen and main screen.

You can now test if the buttons and knobs are working by referring to the MIDIbox FM User Manual, which can be found here: http://www.ucapps.de/midibox_fm_manual.html

If everything appears to be working, then you can install the “Bankstick” ICs (**IC5**, **IC6**, **IC7**, **IC8**, **IC9**) and watch the LCD while the MIDIbox FM application formats the Banksticks.

If you want, you can try uploading the default patches into the Banksticks now.

The default patches are found in the MIDIbox FM Synth release package (they are in the “presets” folder).

You can try TL's “MidiBox Patch Manager”:

http://www.automatic-brain.de/midibox/

or JSynthLib:

http://www.ucapps.de/jsynthlib.html

If you have trouble with uploading patches using these patch editors, you can alternately try uploading the patches using the SysEx dumps, with a delay between each SysEx message (i.e. use MIDI-OX or equivalent MIDI utility).

9. Testing the OPL3 chipset

You can tell pretty quickly if the sammichFM is working correctly by using the MIDIbox FM application. I advise uploading the default patch set and scrolling through them, having a listen to each one.

However, if the sammichFM isn't producing any sound, or the sound is not what you expect, then you can troubleshoot what is wrong by using some test applications. The use of these test applications is explained in detail elsewhere in the MIDIbox forum, the wiki and the readme.txt files inside the test applications.

If you require help with troubleshooting, just post on the MIDIbox forum and someone will help.

The following links will help you find the other relevant information and assistance:

http://www.midibox.org/dokuwiki/sammichfm

http://www.ucapps.de/

http://midibox.org/forums

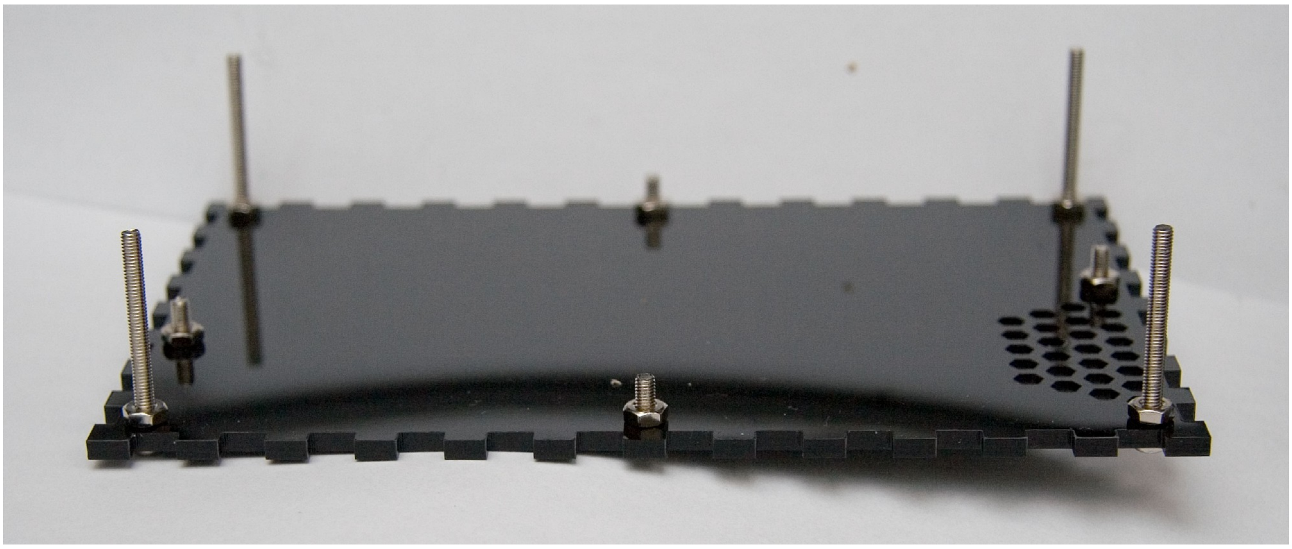
10. Using MIDIbox FM Synth

Read the user manual!

http://www.ucapps.de/midibox_fm_manual.html

There are some differences between sammichFM and the "default" MIDIbox FM synth control surface which should be covered here, but I'm sure you can work it out.

11. Case Assembly



Put the 32mm and 9mm screws into the base panel from the bottom. The 32mm screws go in the corners. Add the nuts and tighten firmly using screwdriver and long-nose pliers.

Place the base PCB onto the screw shafts. Thread the 20mm spacers onto the screw shafts. You ***should not*** need to tighten these with pliers; tightening just with fingers is sufficient.

Attach the control surface PCB, taking care to mate the headers without bending them.

Thread the 10mm spacers onto the screw shafts in the corners. You ***should not*** need to tighten these with pliers; tightening just with fingers is sufficient.

Add the 3mm screws to the midpoints of the PCB edges. (Leave these out if you think you'll often be opening up the sammichFM to show off, then you won't need a screwdriver to do this).

Add the rear panel and loosely hold in place with the plastic nuts for the audio sockets.

Add the other side panels. They should stay in position, but you can temporarily use sticky tape if you must.

Add the top panel, being careful not to bend the LEDs.

Add the black screws to the corners of the panel. These screws will cause the top panel to clamp the side panels and hold them firmly in position. You may not need to tighten these with a hex key (Allen key) as tightening just with fingers is sufficient. **Be careful if you use a hex key! Tight enough is good enough!**

You will notice that loosening the black screws will allow you to remove the side panels without removing the top panel, which can be useful for adjusting the LCD brightness and contrast and checking heatsink/voltage regulator temperatures with a finger.

An extra four 3mm screws are provided should you prefer to use these instead of the black screws on the top panel.

12. Painting the Panel Engraving

The panels are made from laser-cut acrylic and come with backing paper still attached and the engravings cut through the backing paper. This paper makes an ideal mask for painting the engraving.

IMPORTANT NOTE: You can build the entire sammichFM with the backing paper still on the panels, and then paint the panel engraving after you have finished. This allows you to play with your new MIDIbox FM synth while you paint the panels (a slow process).

My current technique is to apply two coats of enamel paint (i.e. hobby/model enamel paint) by filling the engraving (the "groove") completely with paint and then wiping excess away with a "squeegee" made from a piece of blister pack plastic, i.e. what they use to package things like toothbrushes.

I use Tamiya Color Enamel Paint X-2 White. It is thin but has dense (?) pigment, so you only need two coats. I suspect many other hobby/model enamel paints will give similar results. **Gloss is good! Avoid matte!**

So essentially:

Fill a small section (one label!) of engraving with enamel paint

Wipe away ***wet*** paint on the paper using the squeegee, leaving paint filling the hole.

Repeat with other labels.

Allow to dry (approx. 1 hour)

Repeat the paint filling as before, again in small sections, wiping away ***wet*** paint.

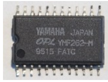
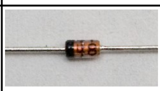
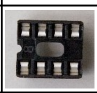
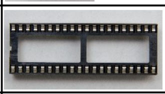
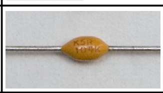
Allow to dry (approx. 8-24 hours)

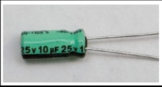
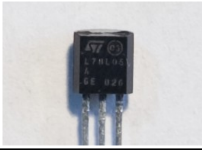
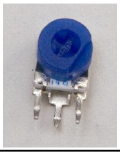
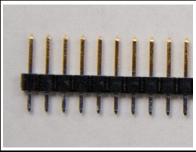
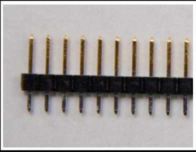
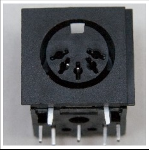
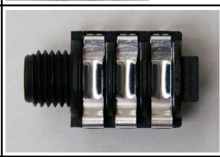
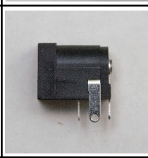
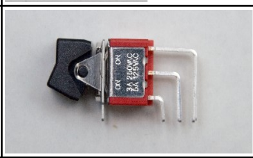
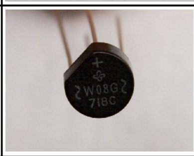
Do not peel off the backing paper until the paint is absolutely dry and hard, after 8 hours at least, preferably 24 hours! If you do it while the paint is still soft, the paper will pull it out of the engraving. I know this from trial and error. **After two coats, put it aside and forget about it for a day!**

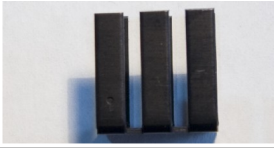
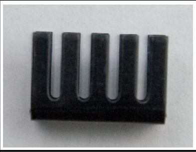
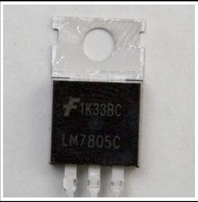
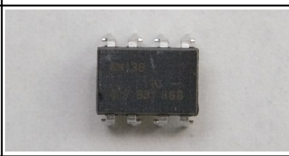
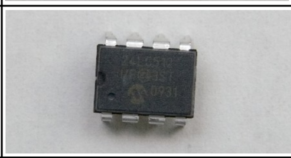
Clean up any imperfections using a craft knife. The matte black acrylic surface is quite tough and won't be scratched by scraping with the edge of a knife.

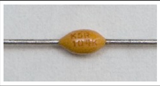
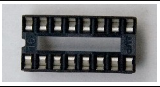
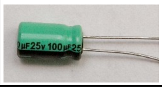
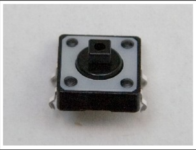
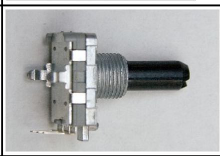
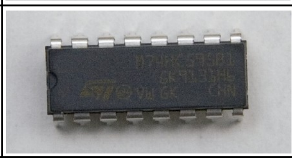
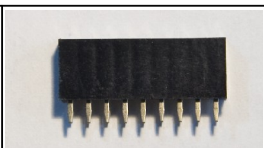
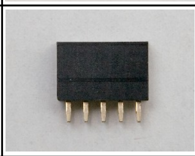
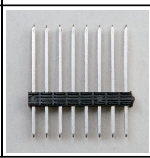
13. Parts List

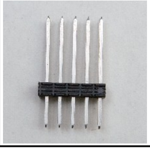
Parts are presented in the preferred order of soldering.

Base PCB part name	Description		Mount Notes
IC50	YMF262		Take off shirt while soldering this part!
IC51, IC52	YAC512		Shirt optional.
IC53, IC54	OPA4348		Shirt optional.
R50, R51	33 Ohm Resistor		
R1	100 Ohm Resistor		Very similar bands to 1K resistor!
R7, R8, R11, R68, R69, R70, R71	220 Ohm Resistor		
R2, R4, R12, R13, R52, R53, R54, R55	1K Resistor		Very similar bands to 100 Ohm resistor!
R6	1K2 Resistor		Very similar bands to 10K resistor!
R5	5K6 Resistor		
R3, R9, R10, R56, R57, R58, R59, R60, R61, R62, R63, R64, R65, R66, R67	10K Resistor		Very similar bands to 1K2 resistor!
R4A	82K Resistor		
D1, D2	1N4148 Small Signal Diode		Align with stripe matching PCB.
IC2, IC5, IC6, IC7, IC8, IC9	IC Socket 8 pin		Align with notch matching PCB.
IC1	IC Socket 40 pin		Align with notch matching PCB.
MS1	ferrite bead		Some magic smoke is emitted while soldering. Hold your breath!
Q1	10MHz crystal (low profile, model XT49S)		
C1, C2	33pF Ceramic Capacitor		
C60, C61	68pF Ceramic Capacitor		
C56, C57, C58, C59	2.7nF Capacitor		These mount very close together. Straighten the leads that will be closest together.
C7, C8, C9, C10, C50, C51, C52, C53, C54, C55	100nF Monolithic Capacitor (radial)		Mount flat.

C4	330n Monolithic Capacitor		
C5, C6, C62, C63, C64, C65, C66, C67	10µF Electrolytic Capacitor		Align negative "-" (black stripe) lead with pad that IS NOT marked postitive.
T1	BC337 Transistor		Bend leads to fit PCB, align with flat side. See guide.
IC3	78L05		Bend leads to fit PCB, align with flat side. See guide.
P1	50K trimpot		Marked "503" on back.
P2	10K trimpot		Marked "103" on back.
JMI[3], JMO[3], JR4A[2], JBP[2+2], AUDIO/OPL3 POWER	.100" Pin Strip Headers 40P STRT 1 ROW GOLD		Minimum required headers.
J4[4], J6/J7[5+5], J12, J13, "POWER/CAN/MIDI"			These are not used in a default sammichFM construction. ***Do not solder them!***
MIDI In, MIDI Out	DIN Jacks 5 PIN DIN PCB		
Audio Out 1/2, Audio Out 3/4	Neutrik Phone Jack 3C STEREO 3-SPST NC (NMJ6HFD2)		
Power Socket	DC Power Jacks PCB 2.1MM		Use rear panel mounted with audio socket nuts to check alignment. See guide.
Power Switch	C64-style DPDT rocker switch		
X1	Bridge Rectifier 1.5 Amp 800 Volt		Align with flat side
Q50	14.318MHz Crystal Oscillator		Align dot/unrounded corner with dot on PCB (lower-right corner pin).

C3	2200μF Electrolytic Capacitor		Align negative "-" (black stripe) lead with pad that IS NOT marked postitive.
Voltage Regulators and Heatsink Stage			
IC4	Heatsink TO-220 (big one)		Use heatsinking compound! See guide.
IC4	Heatsink TO-220 (small one)		Use heatsinking compound! See guide.
IC4	5V Voltage Regulator 7805		Bend leads 90º and mount flat. Use heatsinking compound! See guide
Non-Soldered Components			
JMI, JMO, JR4A, JBP, Audio/OPL3 Power	Shunts (jumpers)		See guide for correct jumper placement.
IC1	PIC18F4685		Do not solder! Do not insert until after voltage checks!
IC2	6N138		Do not solder! Do not insert until after voltage checks!
IC5, IC6, IC7, IC8, IC9	24LC512		Do not solder! Do not insert until after voltage checks!

Control Surface PCB part name	Description		Mount Notes
R1-R10	220 Ohm Resistor		
R11, R12	1K Resistor		Very similar bands to 100 Ohm resistor!
C1, C2, C3	100nF Monolithic Capacitor (radial)		Mount flat.
IC1, IC2, IC3	IC Socket 16 pin		Align with notch matching PCB.
RN1, RN2, RN3, RN4	6 Pin Common Bus Resistor Networks 6PIN 10KOhms 2%		**Align with dot matching PCB!!!!!!! !!! !!!!!!!!!!!!!!!!!!11111111111111111111**
C5	100µF Electrolytic Capacitor		Align negative “-” (black stripe) lead with pad that IS NOT marked postitive.
T1, T2	BC547 Transistor		Bend leads to fit PCB, align with flat side.
S1-S13	E-Switch TL1100F160Q Tactile Switches 12X7.3MM 160GF		Some leads might touch heatsink, marked on bottom side. See guide.
E1	16mm detented rotary encoder		Solder mounting tabs also.
IC1, IC2	74HC165		Do not solder! Do not insert until after voltage checks!
IC3	74HC595		Do not solder! Do not insert until after voltage checks!
L1-L10	LED Standard 3mm round		Mount last! Use top panel to align LEDs during soldering. See guide.
PCB Interconnection Stage			
J16 (on base)	Dual inline header, female (2x9 pins)		Solder while control surface PCB attached to base PCB. See guide.
J8/J9 (on base)	Dual inline header, female (2x5 pins)		Solder while control surface PCB attached to base PCB. See guide.
LCD (on control surface)	Dual inline header with long tail (2x8 pins)		Solder while LCD attached to control surface PCB and control surface PCB attached to base PCB. See guide.

J1 (on control surface)	Dual inline header with long tail (2x5 pins)		Solder while control surface PCB attached to base PCB. See guide.

14. Using sammichFM Base PCB Separately

The sammichFM Base PCB can be used separately in an alternative enclosure, with your own control surface board, a different LCD (i.e. 2x40 character LCD), more buttons and rotary encoders. The base PCB is essentially a Core8 module (with Banksticks) and an OPL3 module on the same PCB, with its own power supply circuit (a 7805 for the digital circuits and a 78L05 for the audio and OPL3 circuits).

You can refer to the MIDIbox FM user manual to learn how to connect DIN and DOUT modules (for your own switches, rotary encoders and LEDs). Here are some specific things you need to deal with:

14.1: Sockets

Alternate (e.g. panel mount) MIDI sockets can connect to J12 and J13, via a 3-pin male header.

Alternate (e.g. panel mount) audio sockets can connect to the pads of the PCB mount audio sockets. There is no alternate 3-pin headers (sorry, an oversight). Wiring is pretty obvious.

Input power can be supplied to the 2-pin header to the right of the DC power socket.

Alternate (e.g. panel mount) power switch can be used, and connect directly to the power switch pads. You could alternatively put the switch in series between the power socket and the 2-pin header for input power, and then bridge the middle and top pins of each 3-pin group of the power switch.

For reference, the power switch for this PCB is DPDT, the middle pin of the 3-pin group is the pole, and in the "on" position, each pole connects with the top pins (the throw). The bottom pins are not used.

14.2: LCD Header (J16)

The sammichFM base is designed so an LCD with standard pinout can "plug in" to a female header in J16. This means that even if a male header is put in J16, you cannot connect it directly with ribbon cable/IDC connectors to a male header in the bottom of the LCD, because the rows of pins will be swapped.

Here are some ways to resolve this:

1. Swap each pair of wires in a 16-wire ribbon cable, and mount male header to bottom side of LCD, male header in J16. You can do this by splitting a ribbon cable into two wire pairs at one end (say 50mm), twisting each pair, reforming into a flat ribbon using some sticky tape, and then clamping that end into an IDC connector as per usual. (You could also use female header in J16 and a male header to act as "gender bender" allowing IDC connector to plug into female header).
2. Use a right-angle male header mounted to the top side of the LCD, so the connector mounts from the side. Use male header in J16 and straight ribbon cable/IDC connectors.
3. Use a connector with crimp pins on one end of the cable, and swap each pair of wires as they go into the connector. The other end has an IDC connector. Mount male headers in J16 and the bottom side of LCD.
4. Use female header in J16, solder ribbon cable wires to short end of male header (swapping each pair of wires), and use IDC connector at other end. Mount male header to bottom side of LCD.

14.2: DIN/DOUT Header (J8/J9)

Header J8/J9 on the sammichFM PCB is identical to the J8 and J9 headers on the Core8 module, so you can connect it directly to DIN and DOUT modules with ribbon cable/IDC connectors.

14.3: MIDIbox FM Firmware

You will need to recompile the MIDIbox FM firmware with the correct settings of your hardware. The sammichFM base expects use of a PIC18F4685, and only 4-bit data to the LCD (default MIOS behaviour for PIC18F4685 anyway). Similarly, alternate PIC pins are used for the 8-bit data bus to the YMF262, this may already be default behaviour for PIC18F4685 builds of MIDIbox FM. I'm just making note of it here in case you don't get any sound out of the OPL3 module - this difference might be the cause.