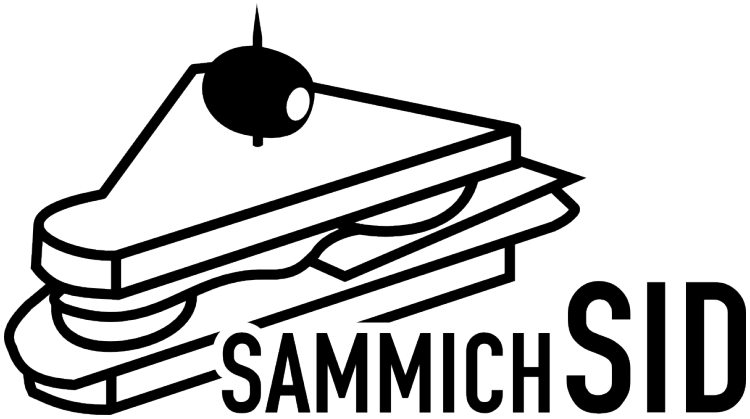


sammichSID Build Guide v1.0

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!

You can build the entire sammichSID 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 SID 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 :-)

2. Base PCB Soldering Walkthrough

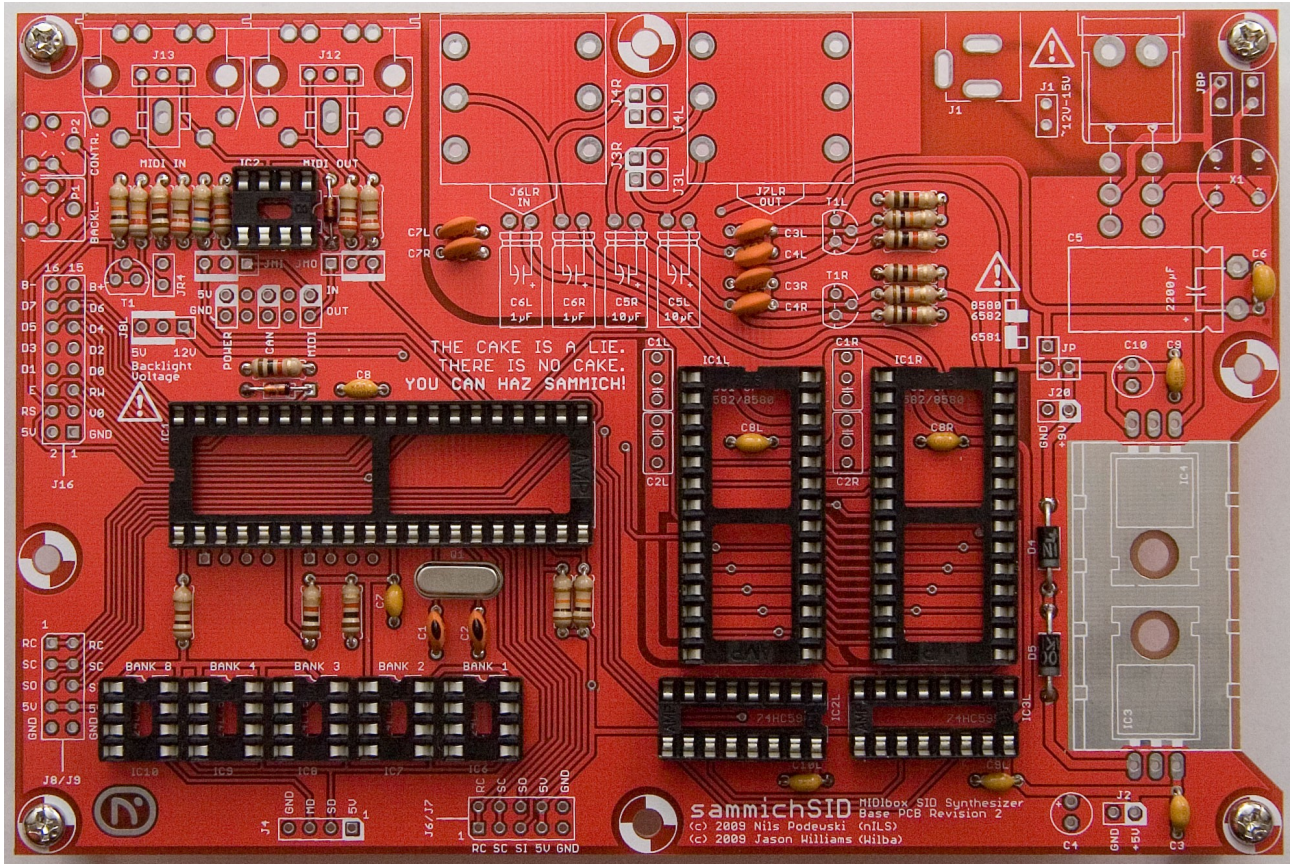
This walkthrough suggests mounting and soldering parts in the order they are presented in the Parts List.

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.

Step 1: Solder resistors, diodes, IC sockets, crystal, ceramic capacitors.

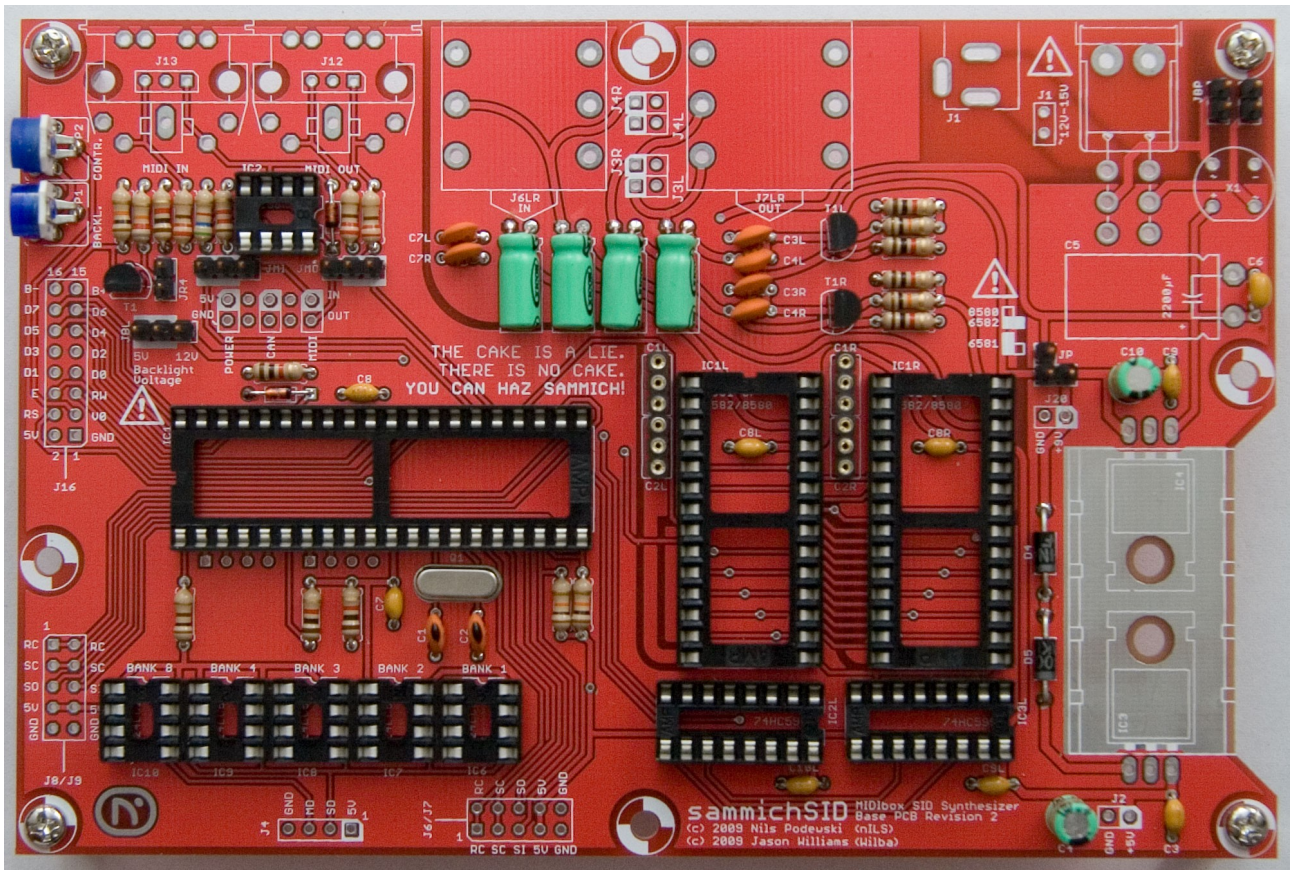


Diodes must be oriented correctly. The stripe on the diode matches the stripe on the part outline.

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

Resistors do not have a correct orientation, but look nicer if oriented consistently.

Step 2: Solder small electrolytic capacitors, machine pin strips, transistors, 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.

Do not insert capacitors into machine pin strips until you are finished all soldering, to avoid accidentally bending them.

Transistors will require 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.

Solder transistors quickly! Do not heat the joint more than 3 seconds, and allow 20 seconds to cool between soldering transistor leads.

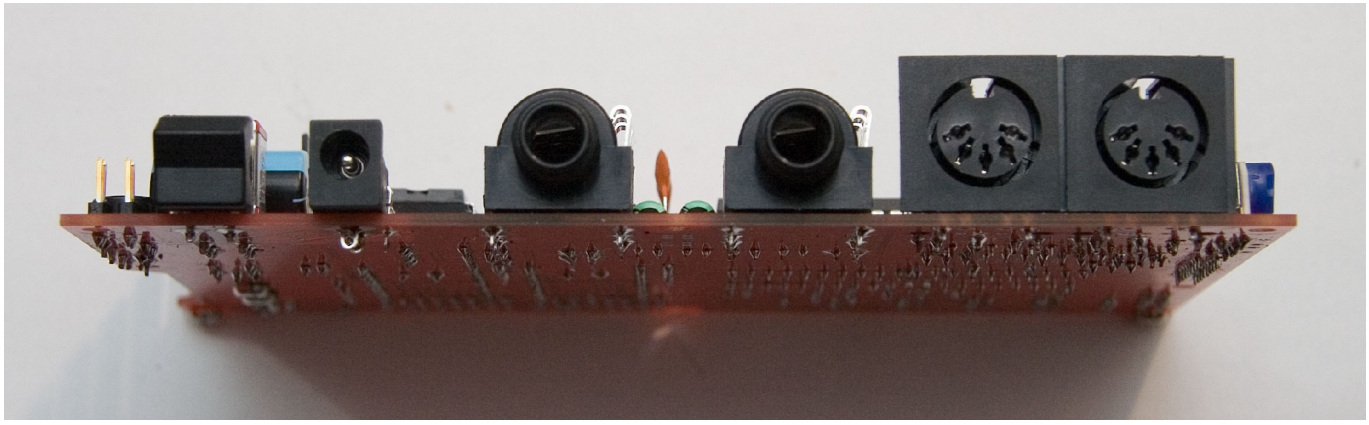
Only solder the required headers, all others are for using this PCB in other contexts.

Header **JP** can be formed easily using a shunt (jumper) to hold a single header pin to a 2-pin header. This will hold all three pins together while soldering, and ensure correct pin spacing.

Step 3: Solder DIN sockets, audio sockets, DC power socket, rocker switch, bridge rectifier, 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 3 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)

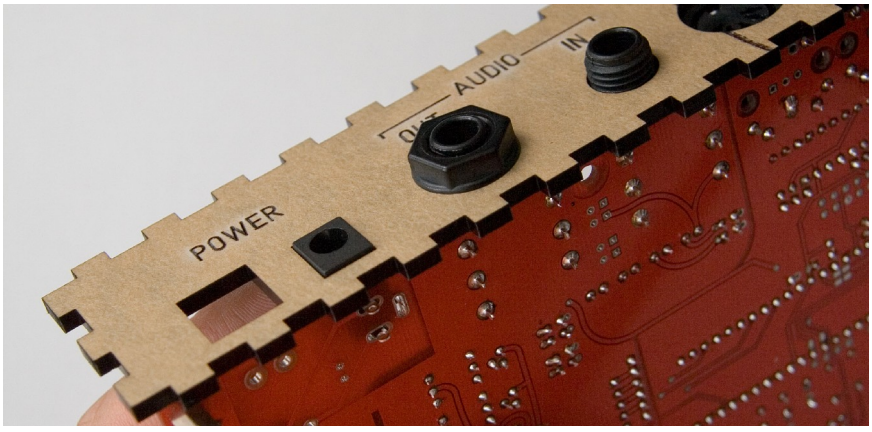


Solder only ***one pin*** of each socket to hold it in place, then you can check for correct alignment before soldering the other pins.

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 (J12, J13) 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 (J1), 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.



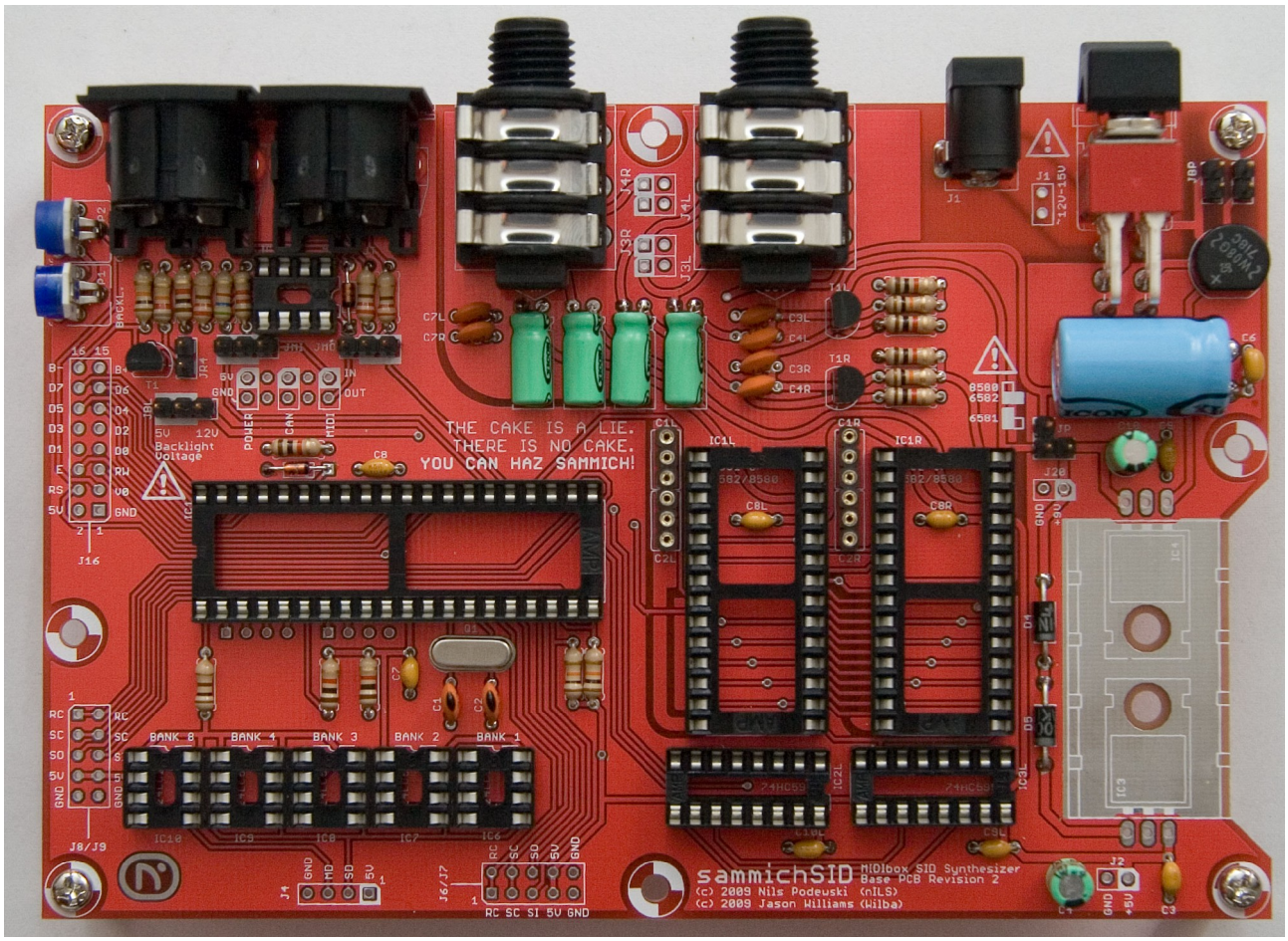
After “tacking” all the sockets, you can now check for correct alignment by placing the PCB onto the base with screws/nuts attached and adding the corner spacers (see **Case Assembly** section) and then attaching the rear panel. You should check that the power socket and audio sockets are not causing 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 big fat 2200uF electrolytic capacitor, making sure it will not get in the way of the L-shaped header **JP**

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



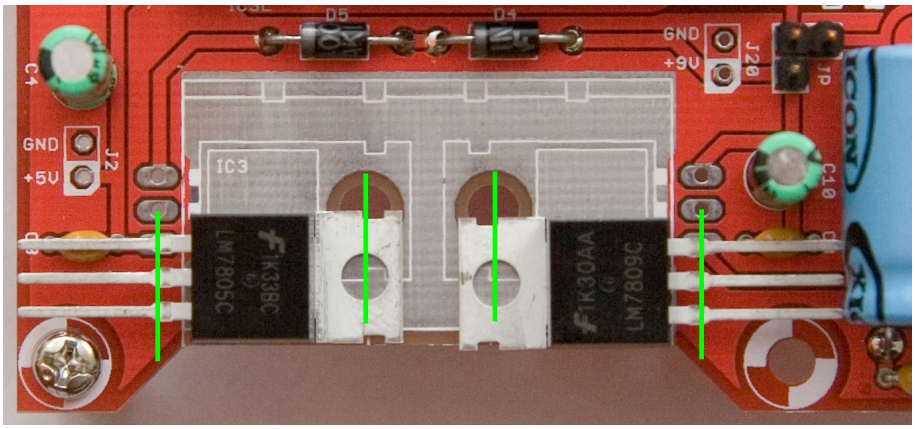
Now is a good time to check for shorts between 5V and GND before you solder the voltage regulators, so you can rule them out as a cause (removing them is painful!)

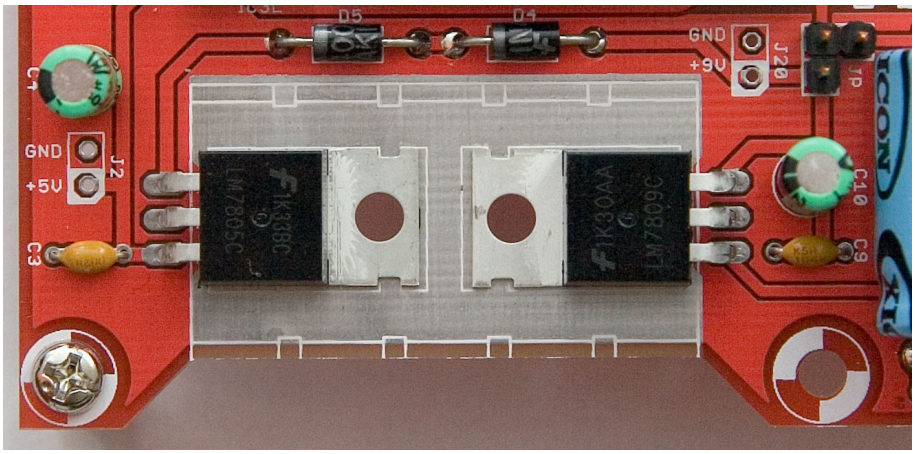
Step 4: Assemble voltage regulators and heatsinks.

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 regulators 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.

Step 4.1: Mark where to bend leads of the voltage regulators and then bend to fit. Ideally, voltage regulators 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 4.2: Practice assembling the heatsinks and voltage regulators without using heatsinking compound

Step 4.3: Apply ***non-conductive*** heatsinking compound to PCB.

Step 4.4: Place big heatsink onto PCB

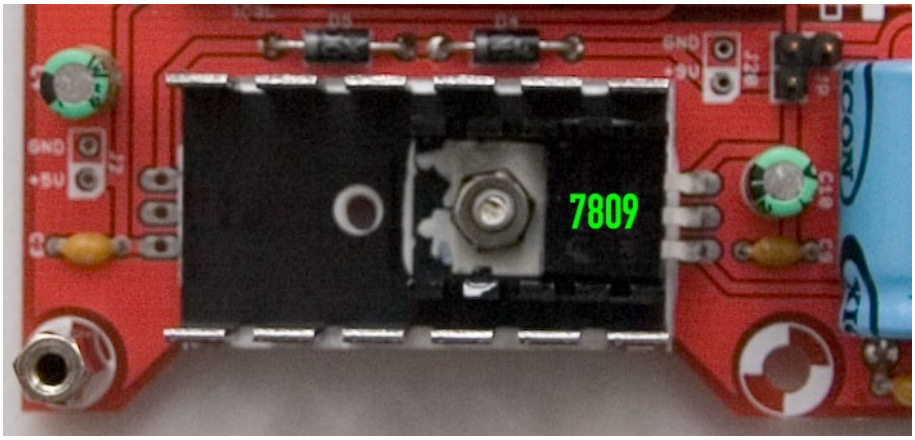
Step 4.5: Apply ***non-conductive*** heatsinking compound to bottom of small heatsink.

Step 4.6: Place small heatsink onto big heatsink, align holes. ***Small heatsink goes on side closest to rear panel!***

Step 4.7: Apply ***non-conductive*** heatsinking compound to back of 7809 voltage regulator (IC4)

Step 4.8: Place 7809 voltage regulator (IC4) onto small heatsink. ***7809 (IC4) goes on side closest to rear panel!***

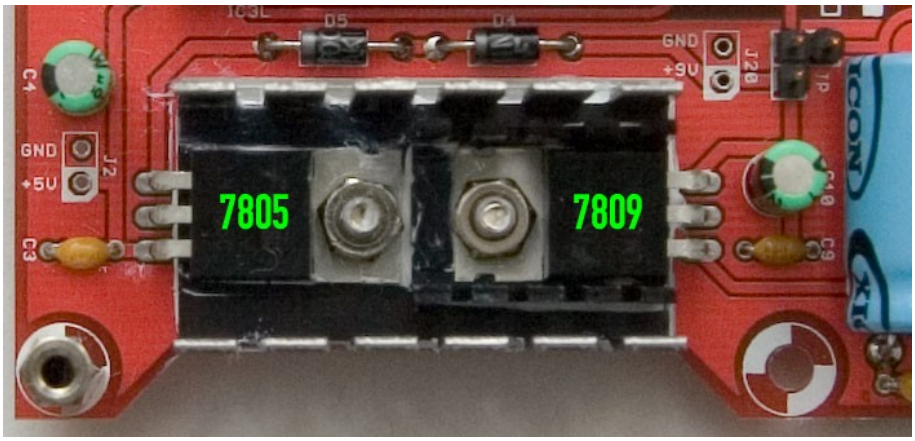
Step 4.9: Screw together 7809 and heatsinks using 9mm screw coming from the bottom side. This is easiest done by holding screw in position with a finger while attaching and tightening nut using long-nose pliers. Do not tighten firmly, turn nut until screw head turns under firm pressure from your finger.



Step 4.10: Apply ***non-conductive*** heatsinking compound to back of 7805 voltage regulator (IC3)

Step 4.11: Place 7805 voltage regulator (IC5) onto big heatsink.

Step 4.12: Screw together 7805 and heatsink 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. Do not tighten firmly, turn nut until screw head turns under firm pressure from your finger.



Step 4.13: While nuts are firm but not tight, you can move the heatsinks and voltage regulators to be neatly aligned. Ensure big heatsink is not touching the pads of the voltage regulators.

Step 4.14: Tighten nuts, but don't overtighten nuts :-)

Step 4.15: ***Check again that the 7809 is on the side closest to the rear panel!***

Step 4.16: Cut leads of 7809 and 7805 (on the bottom side) to be 2mm.

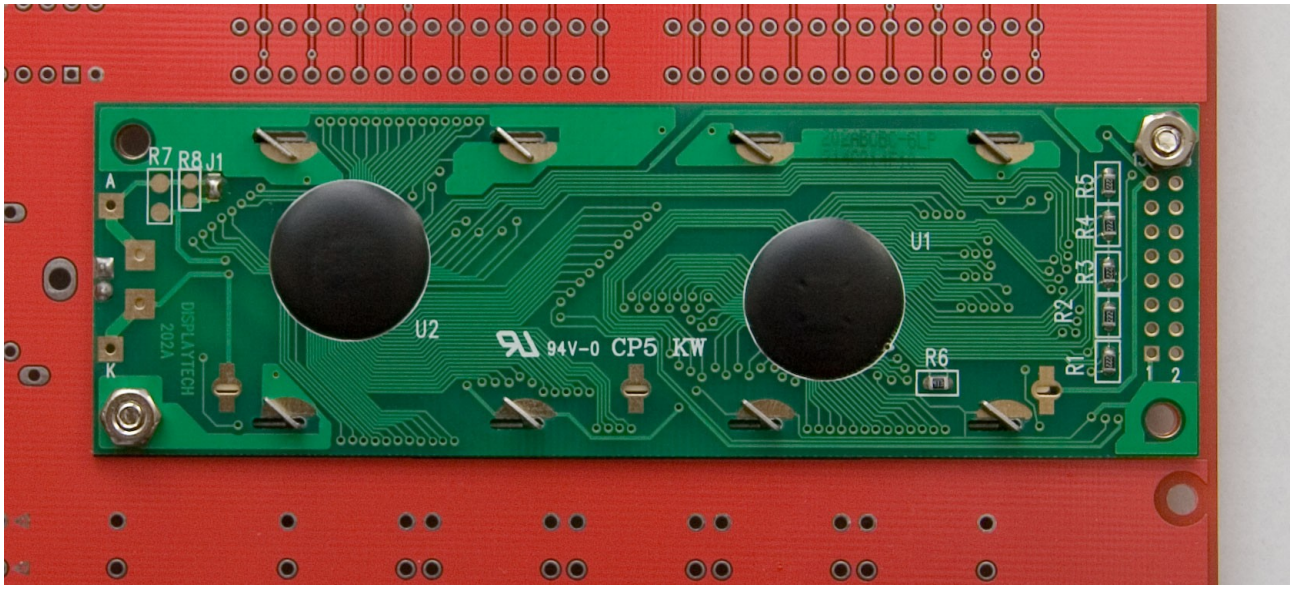
Step 4.17: Solder 7809 and 7805. Allow 20 seconds for part to cool between soldering leads.The middle pin is connected to the ground plane and will be harder to solder. Starting with a blob of solder already on the iron tip, and holding the tip at a smaller angle to the PCB (i.e. more horizontal).

Step 4.18: Clean up any excess heatsinking compound

Step 4.19: 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 5: Attach LCD to Control Surface PCB

To achieve “perfect” alignment, first attach loosely on two diagonally oppposite 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.

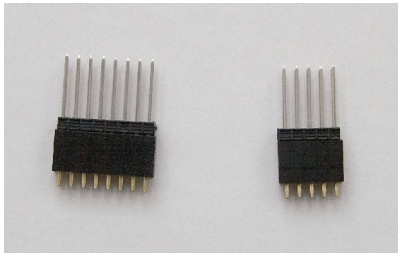
Good alignment is critical in making the control surface PCB easily “plug into” the base PCB.

Step 6: 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.

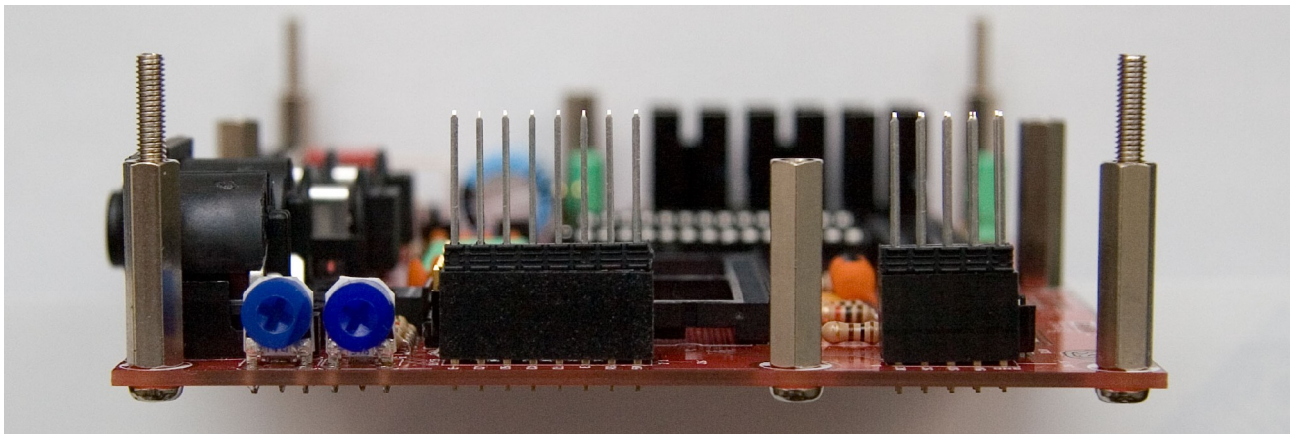
Step 6.1: Prepare the male and female headers. The shorter end of the male header goes into the female header.



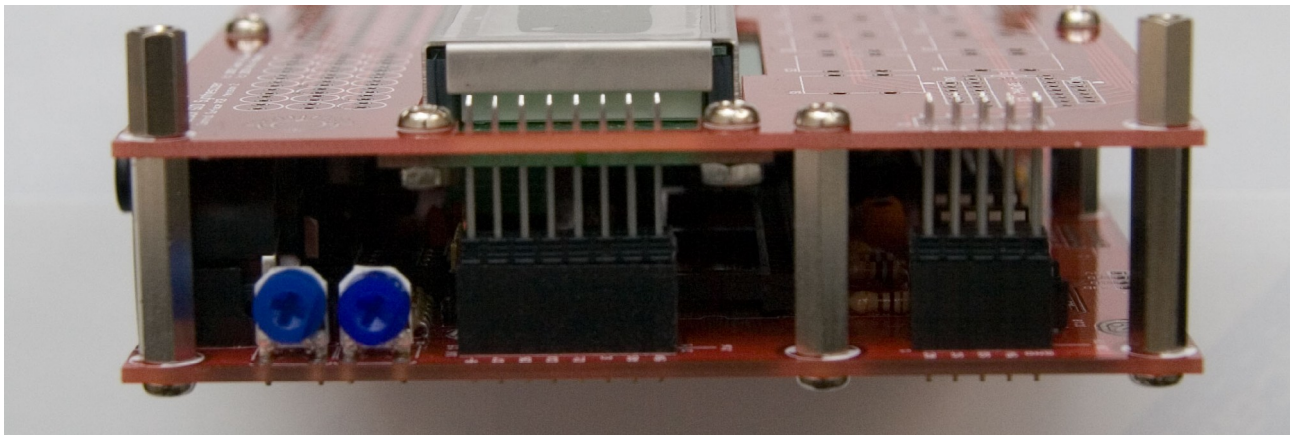
Step 6.2: Attach 32mm screws and 20mm spacers to base PCB corners

Step 6.3: Attach 9mm screws and 20mm spacers to base PCB side midpoints

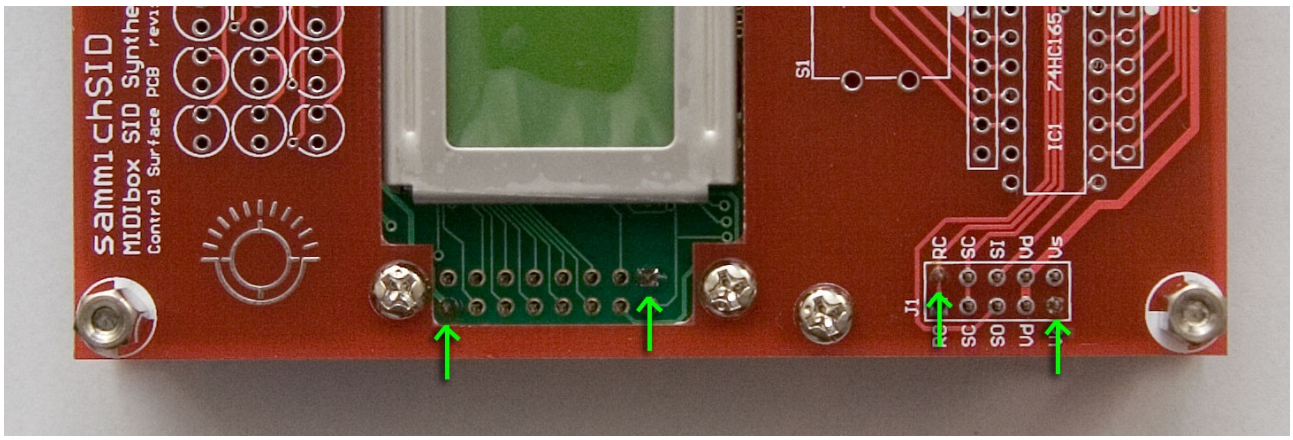
Step 6.4: Insert female header (with male headers inserted) into base PCB.



Step 6.5: ***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.

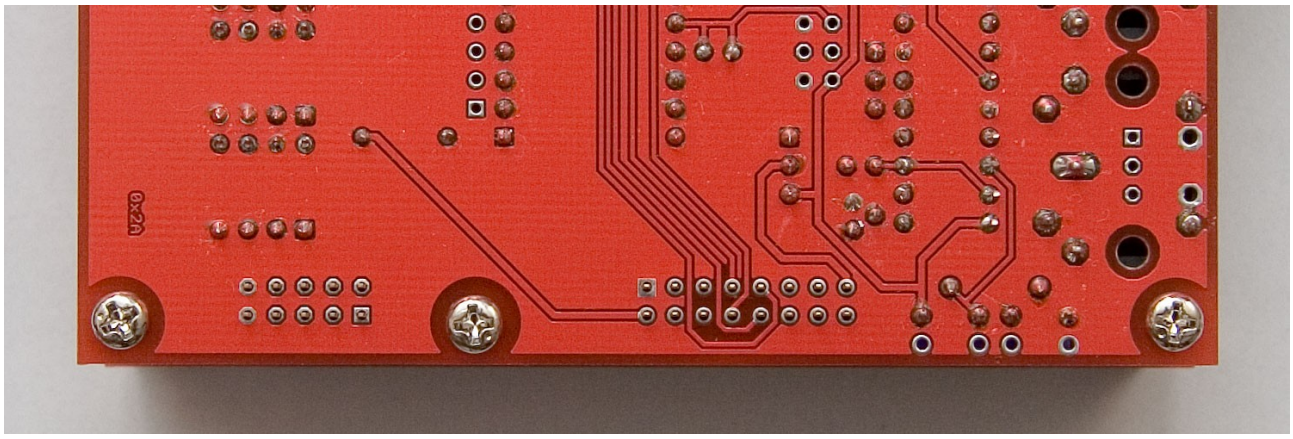


Step 6.6: Solder two diagonally opposite pins on each header on the top side, ensuring female header is flat against base PCB.

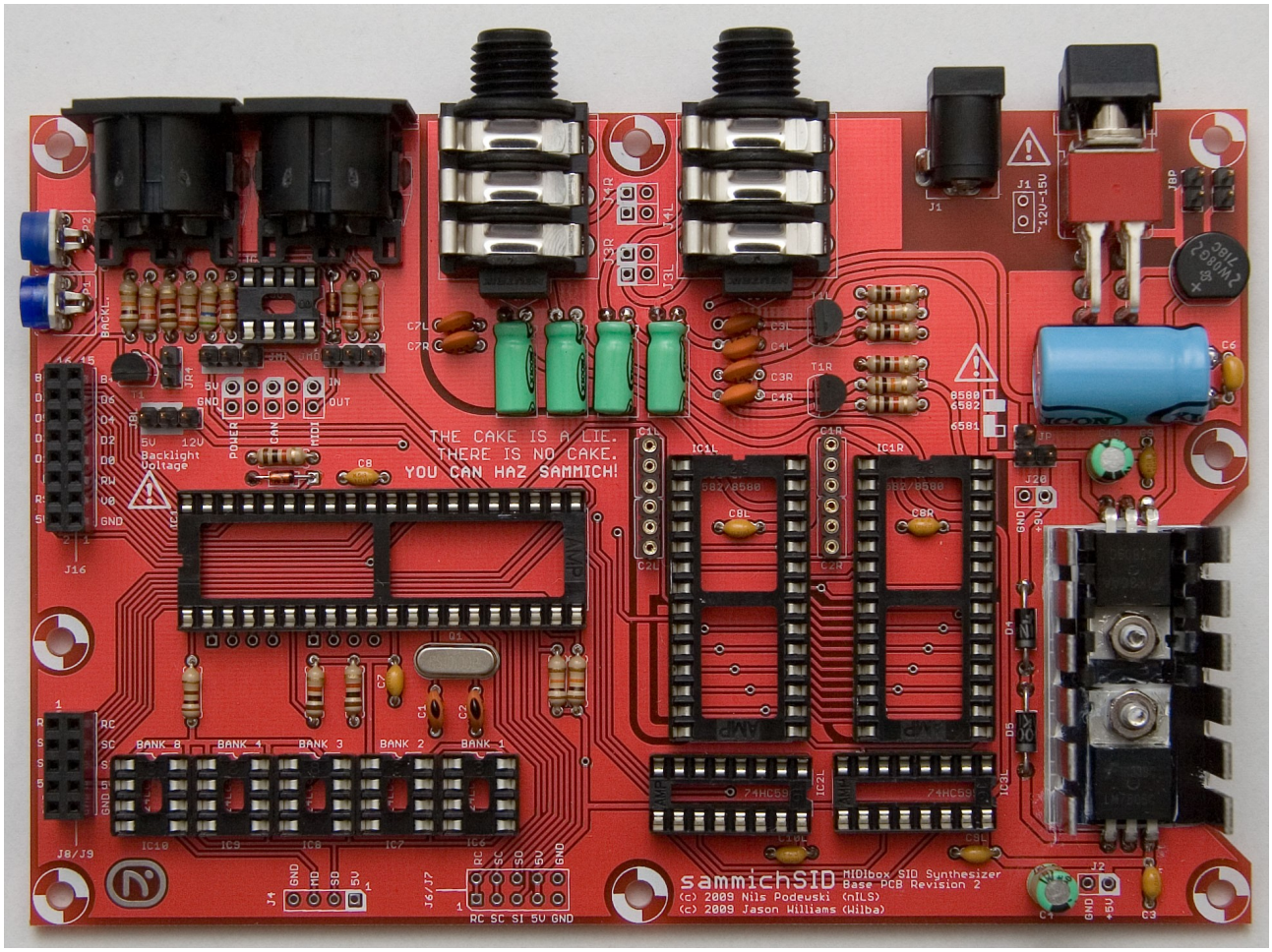


Step 6.7: Have another look from the side. Everything looks good? Female headers are still flat against base PCB? Good. ***Finish soldering the headers on the top side.***

Step 6.8: Solder all the pins of the female headers.



Step 7: Congratulate Yourself for Finishing Soldering of the sammichSID Base PCB!

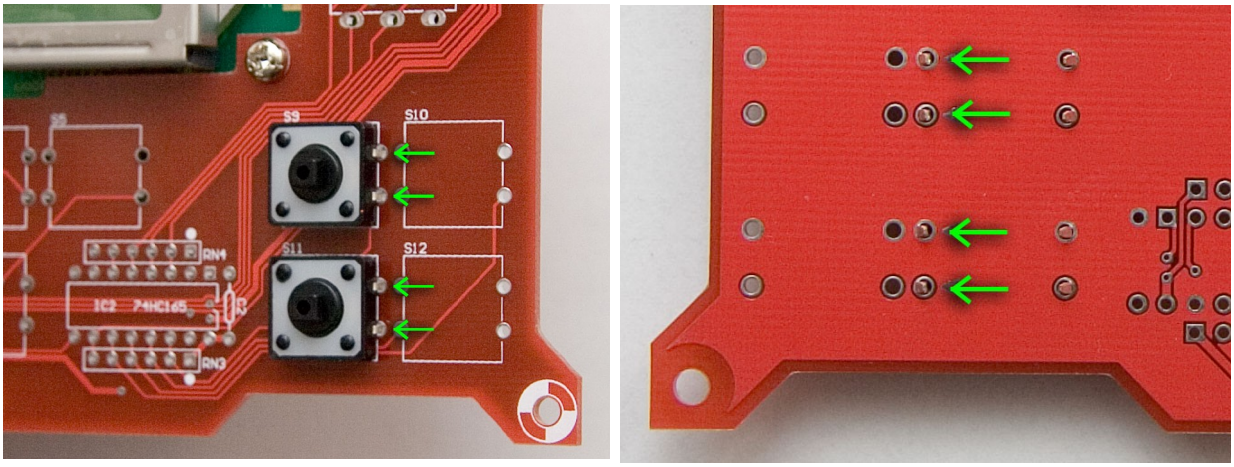


3. Control Surface PCB Soldering Walkthrough

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.

Step 1: Solder the two switches that might touch the heatsink fins



Mount these two switches. Ensure they snap-in fully into the holes and are flat against the PCB.

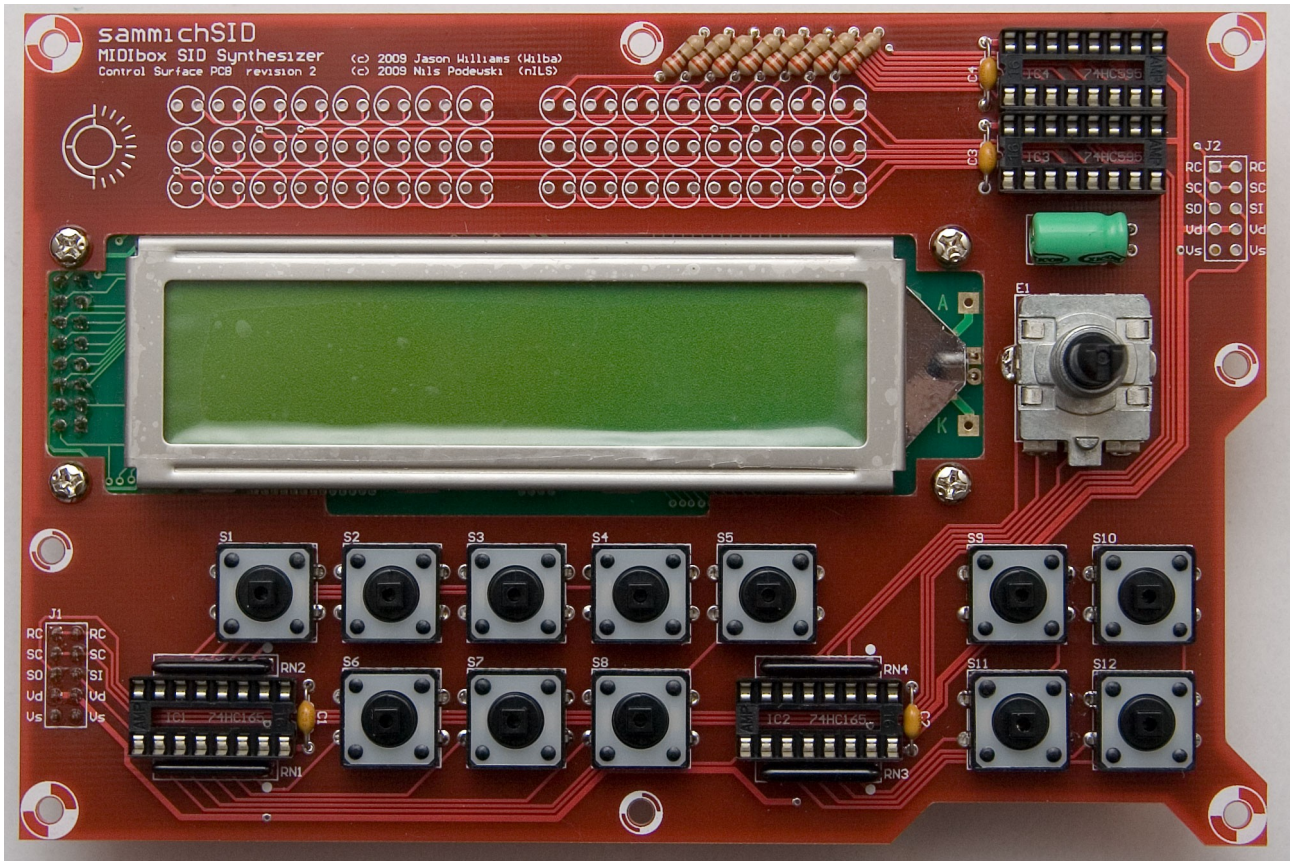
Solder the right side pads on the top side, then cut the leads close to the PCB.

Solder all leads on the bottom side, avoiding large solder joints.

Step 2: Solder remaining parts (except the LEDs)

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

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



As with the base PCB, check orientation of IC sockets, the electrolytic capacitor and the resistor networks before soldering.

Step 3: Soldering LEDs

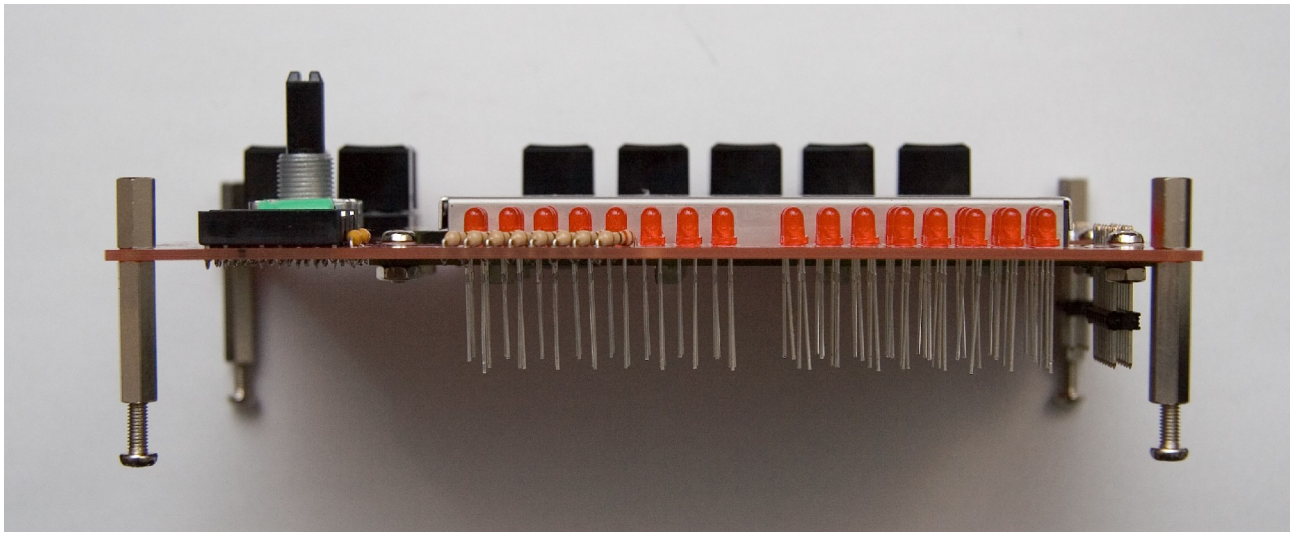
Now here comes another fun bit!

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 noticable and annoying to people with OCD (like me). Take things slowly.

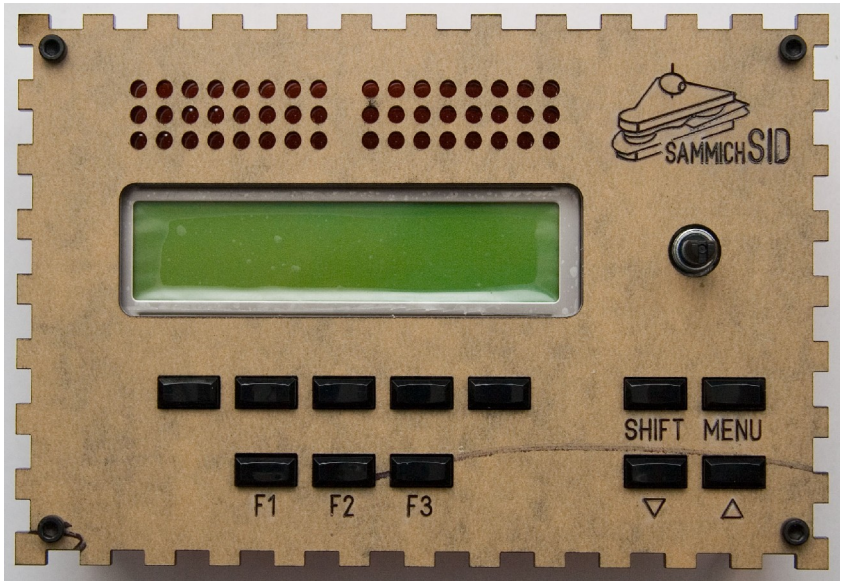
Step 3.1: Mount 20mm spacers to bottom, 10mm spacers to top, using 32mm screws. Allow enough thread in the top of the 10mm spacers so you can screw on the top panel.

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.

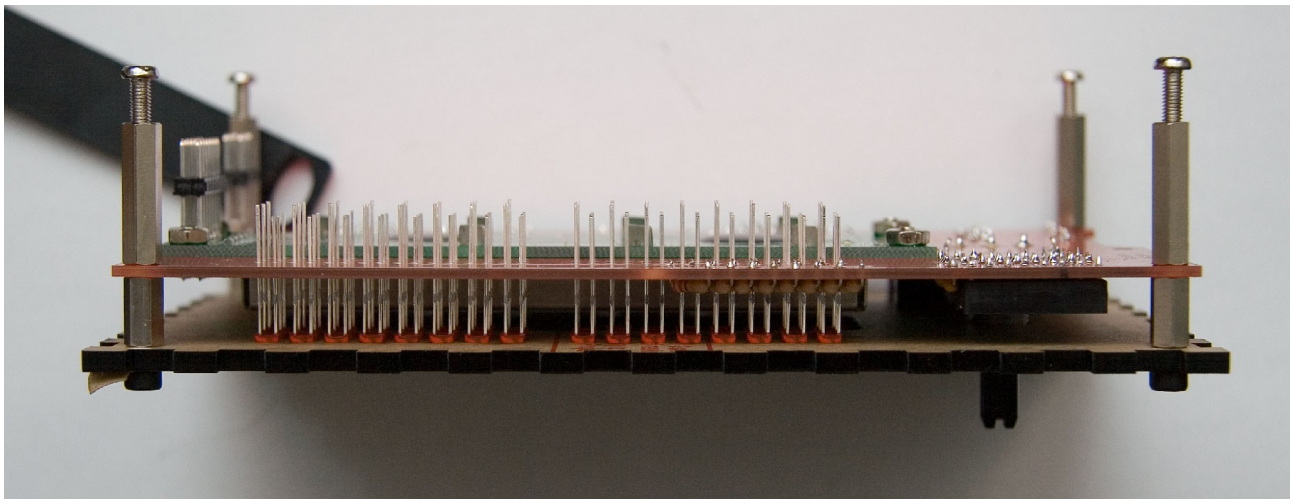
Step 3.3: Insert the LEDs and push them all the way to the PCB. Align the flat side of the LED with the flat side of the part outline. The flat side is the cathode, which is also the shorter lead. The other side is the anode which is the longer lead. So alternately, place the longer leads into the left 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.

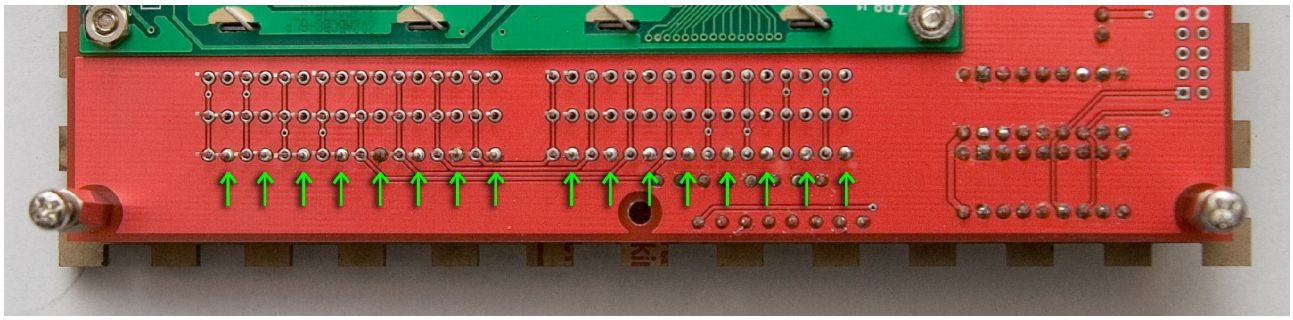


Step 3.5: Turn PCB upside down. ***Carefully and gently*** push LEDs into the holes as much as possible.



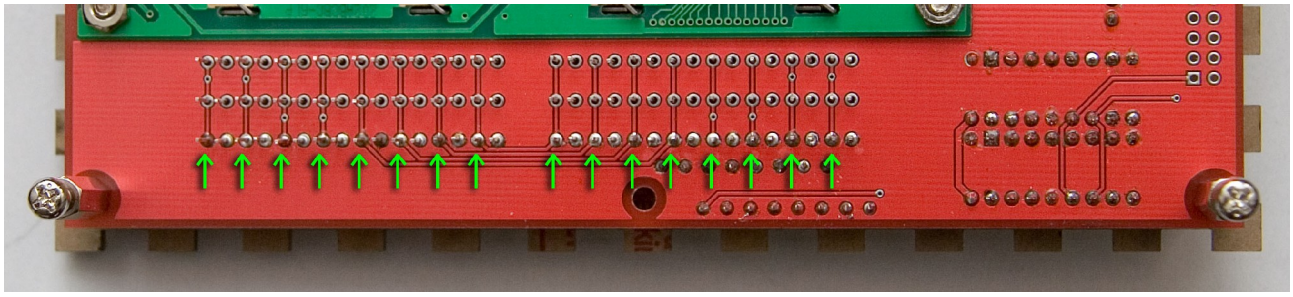
Step 3.6: Look underneath at the LEDs and check they all protrude the same amount. If any do not, it could be that a LED is stuck because the hole is slightly smaller or the LED is slightly bigger. Disassemble PCB and panel and check that LED individually. It is possible that a LED has a little bit of 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. 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!)

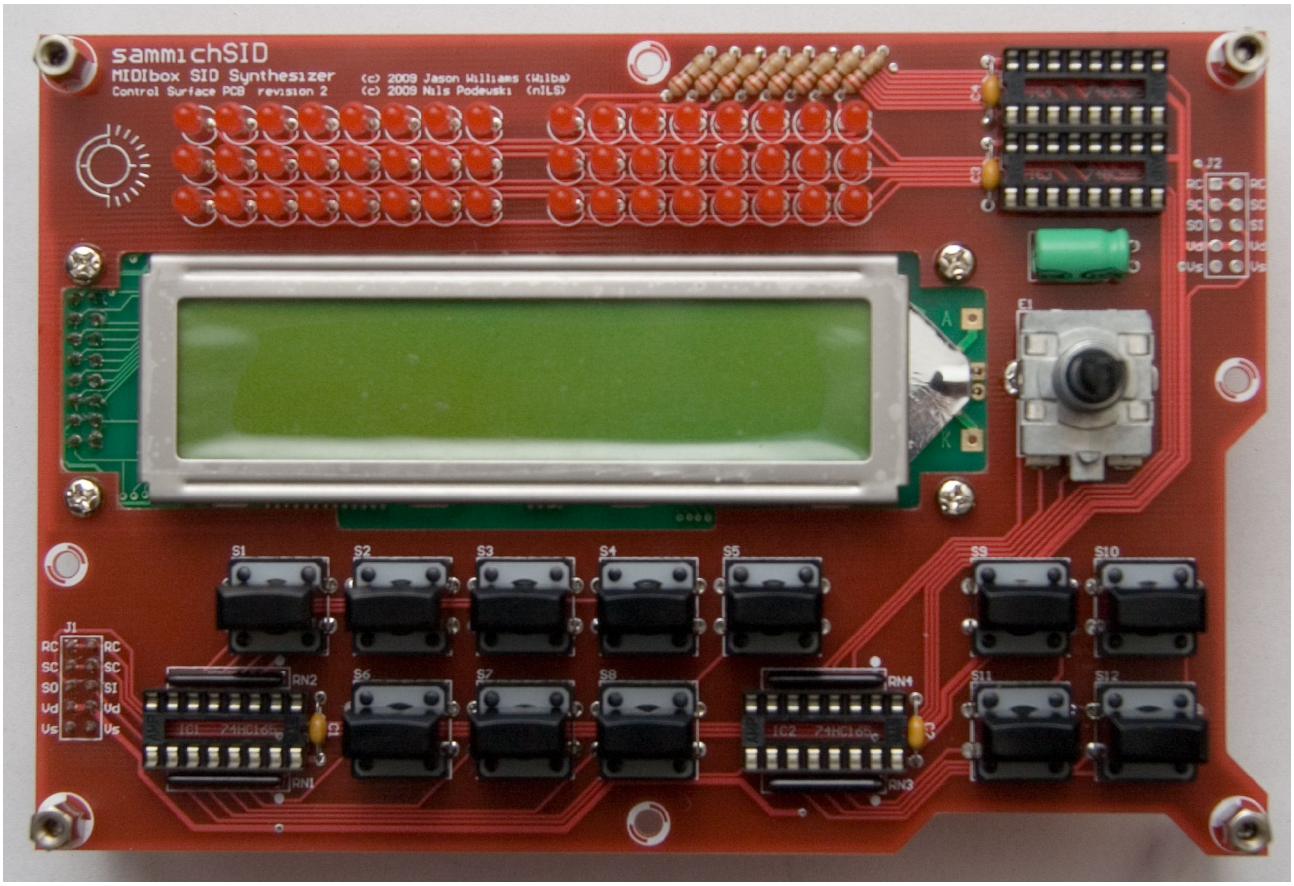
Step 3.9: Solder all the longer leads of that row of LEDs.



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

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

Step 4: Congratulate Yourself for Finishing Soldering of the sammichSID 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 **JBL** controls whether the backlight is suppled by the regulated 5V (coming from the 7805 on the base PCB) or from the supply coming into the base PCB.

Header **JR4** controls whether the backlight current is limitied 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 reduced current, a shunt in this header will disable R4A and the.

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

If you are using a low-power backlight LCD that requires only 25mA, then you should put a shunt in **JBL** in the default “5V” position, indicated by the thick white line and ***do not*** put a shunt in **JR4**!

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

If you are using a high-power backlight LCD that requires 250mA or more, then you should put a shunt in **JBL** in the “12V” position. It is labelled “12V” but it actually will be whatever voltage is coming into the base PCB, i.e. the input to the 7809 voltage regulator. Most often this is going to be 12V-14V. Put a shunt in **JR4** to increase current. Turn brightness trimpot to approximately 2 o'clock, this gives the optimal current for a high-power backlight LCD being supplied by 12V and is considered “safe”. Experienced builders can try connecting a 2-pin SIL connector to **JBL** and measuring the actual current passing through **JBL** pins (i.e. between 12V and the backlight circuit).

SID Voltage Jumpers

The L-shaped header **JP** controls the voltage supplied to ***both*** SIDs. It will supply both SIDs with power from either the regulated 9V supply on the base PCB (i.e. directly from the 7809 voltage regulator) or from the input power supply ***which must be regulated 12V DC***.

Header **JBP** is used to short two of the diodes in the bridge rectifier. This is only for use when powering 6581 SIDs, where you want the regulated external 12V supply to fully reach the SIDs and not drop in voltage.

8580 or 6582 SIDs (marked 8580R5, 6582, 6582A)

These SIDs require a maximum of 9V. Insert a shunt in **JP** in the horizontal position and ***do not*** put shunts in **JBP**.

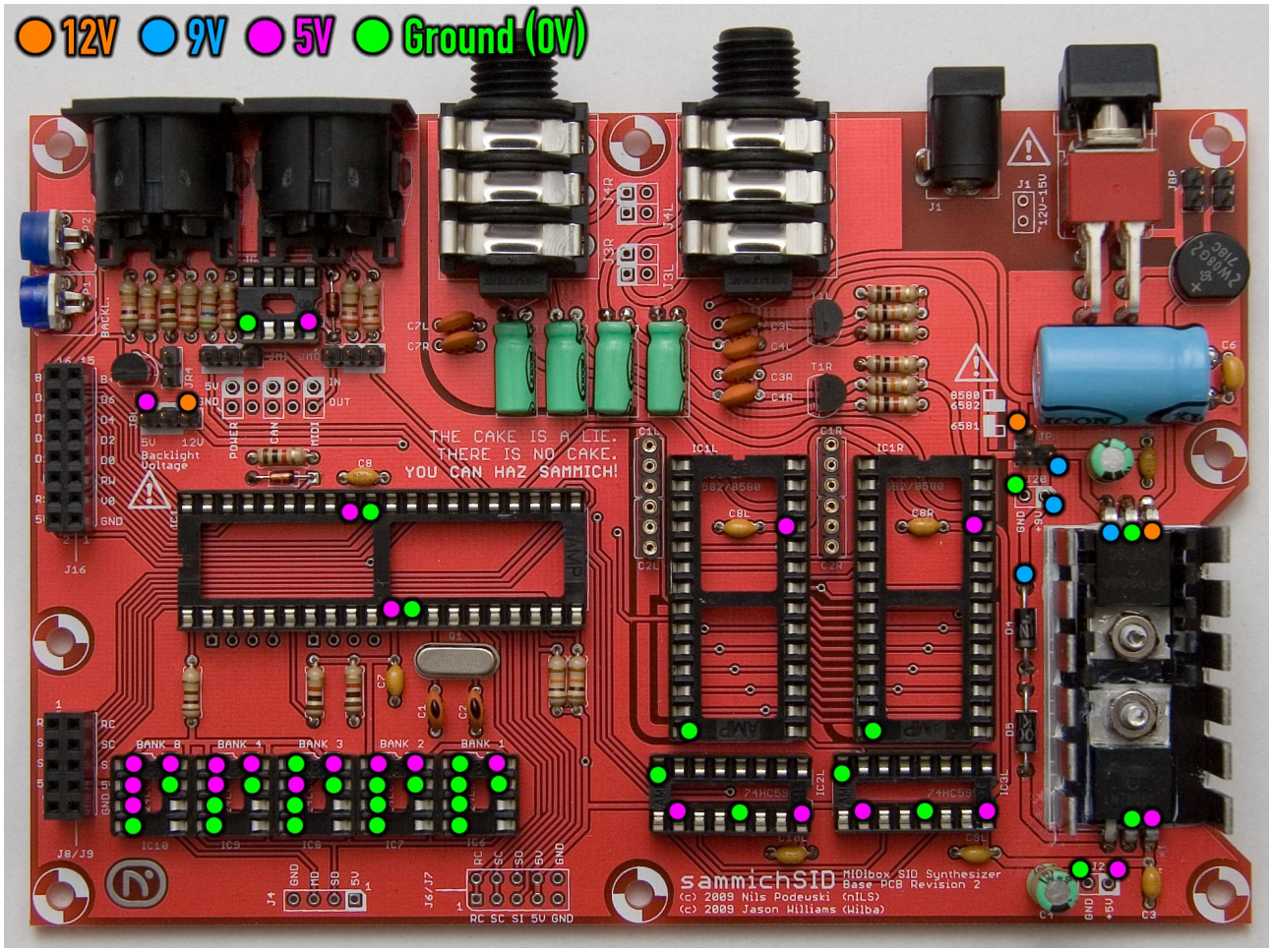
6581 SIDs

These SIDs require a maximum of 12V. Insert a shunt in **JP** in the vertical position and put two shunts in **JBP**. The regulated 12V power supply must be ***tip positive***. If it is ***tip negative***, you will not get the full 12V into the SIDs. If using 6581 SIDs and **JP** is set to 6581, ***never supply the sammichSID with an unregulated power supply!***

5. 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 J2 at lower-right corner), you can leave the black probe on **GND** at **J2** while you test 5V at other points marked with pink dots. Inversely, you can leave the red probe on **+5V** at **J2** while you test for 5V at other points marked with green dots.

Alternately, you can test voltages once at **J2**, **J20**, 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 that points labeled “12V” will only be exactly 12V when using a regulated 12V power supply. If you use an unregulated power supply, the voltage should be 10.5V or more, depending on the current load. If it drops below 10.5V with SIDs installed, then you may need a power supply that delivers more current, but not too much current, or the voltage will be higher than necessary and the heatsinks will get too hot.

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 **JBL** and **JR4** are set correctly for your LCD (See **Jumper Configuration**)

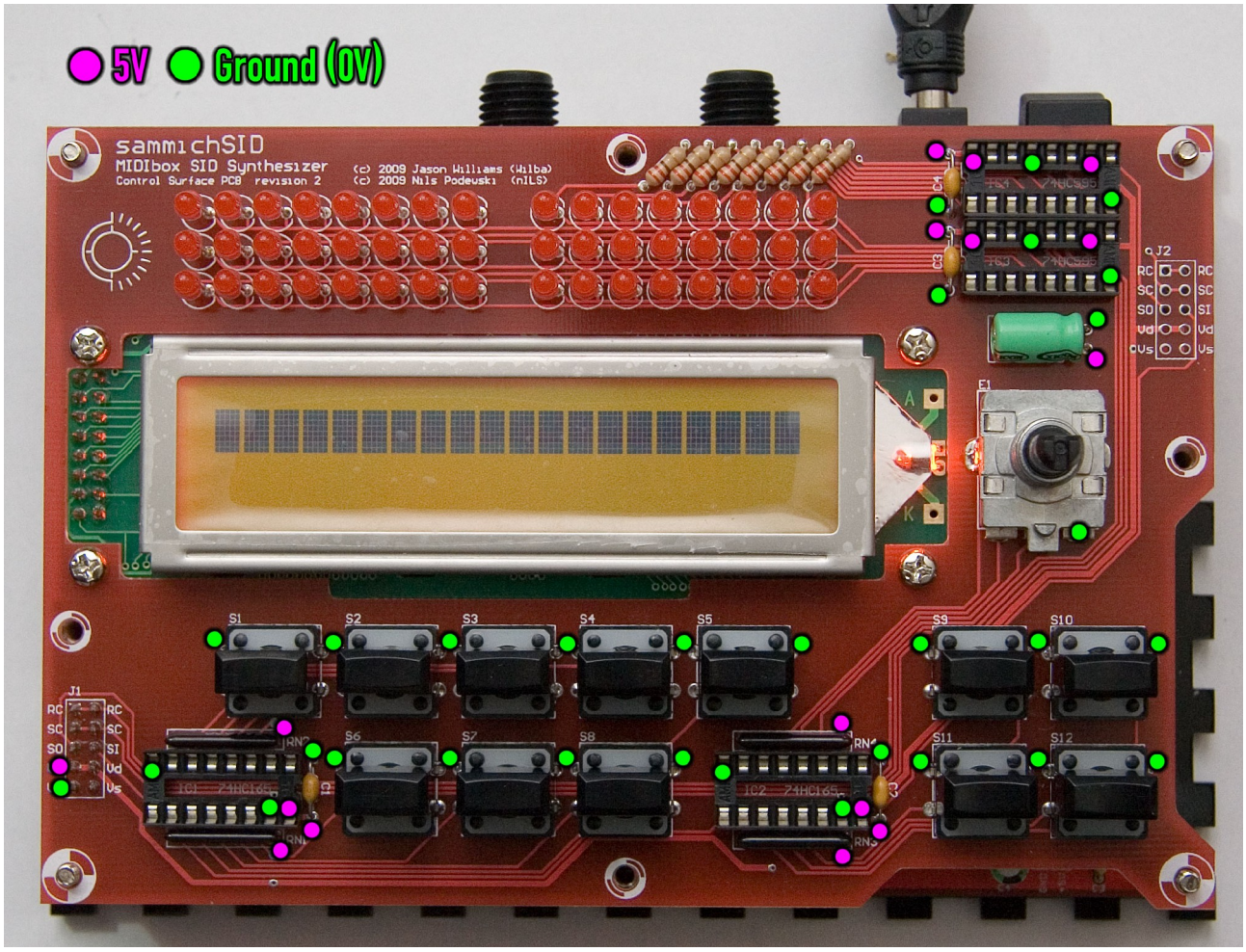
Step 2.2: Disconnect power.

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.



6. 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 (**IC6**, **IC7**, **IC8**, **IC9**, **IC10**) and the SIDs (**IC1L**, **IC1R**) after fully completing both PCBs, uploading MIDIbox SID firmware and having a fully working control surface. This allows you to watch the formatting of the Banksticks by the MIDIbox SID firmware.

7. Initial Testing and Firmware Upload

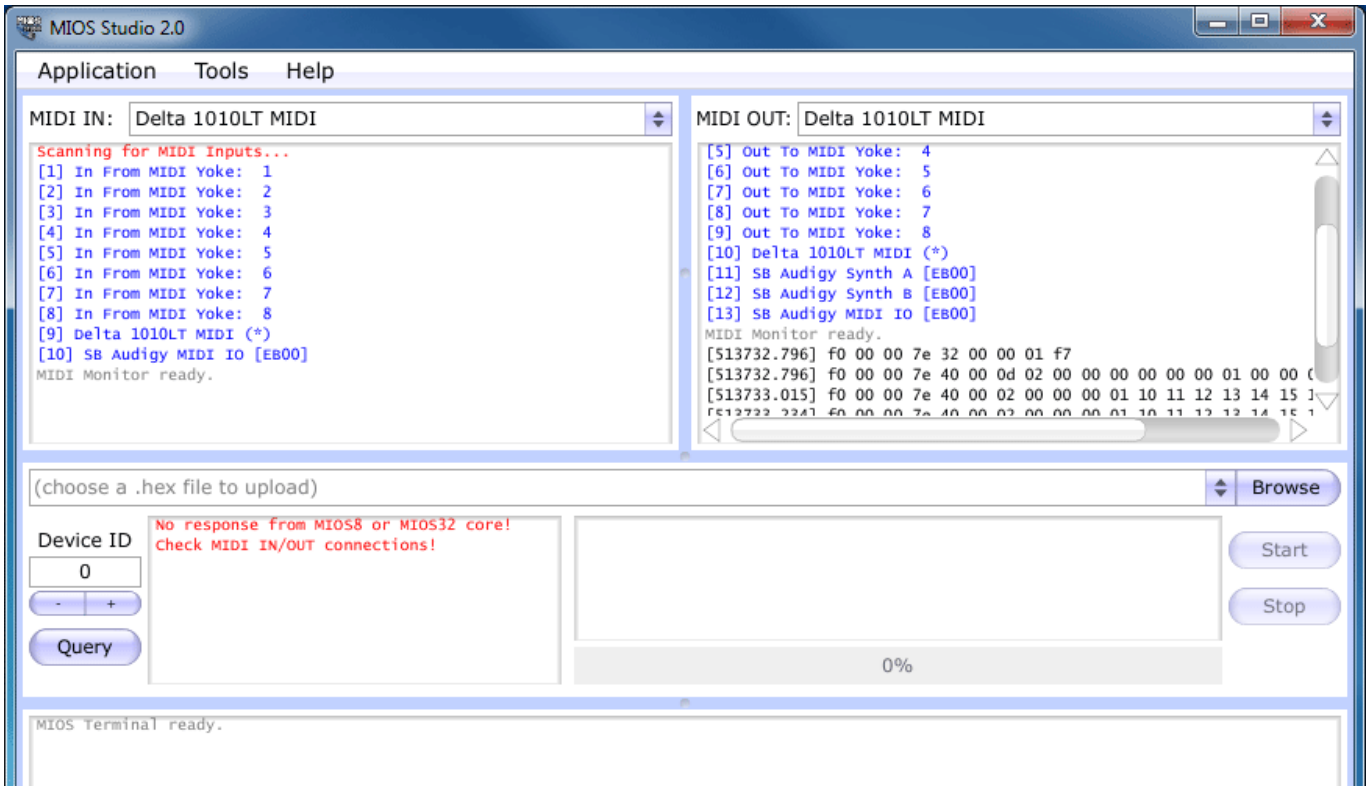
Step 1: Installing MIOS Studio 2 and Testing MIDI

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

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

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

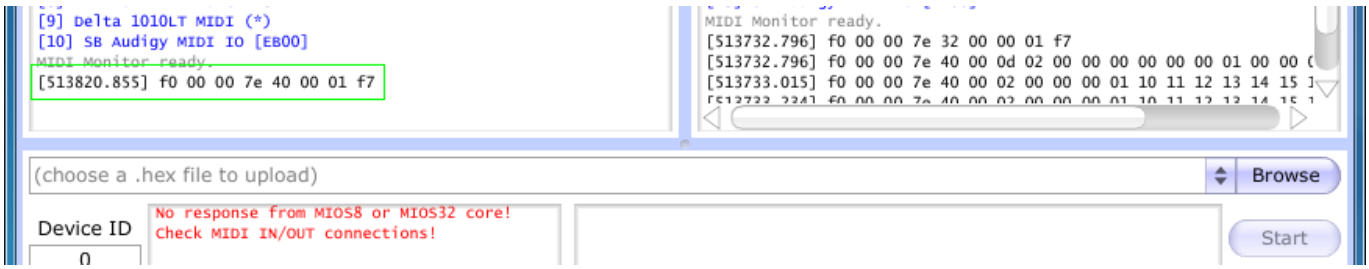
Run MIOS Studio 2. You should see a screen like this:



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

Power on the sammichSID!

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

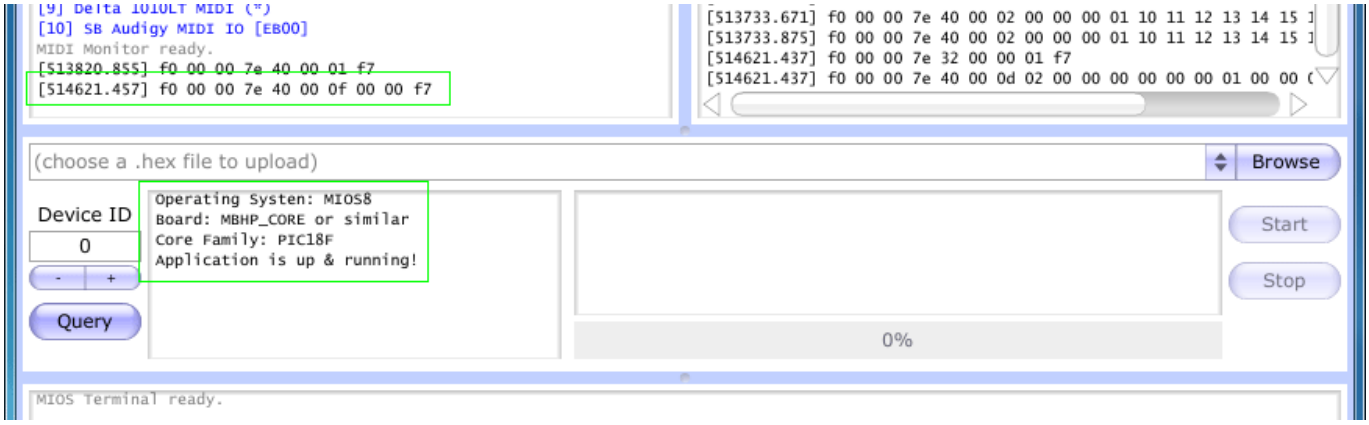


This is because the PIC18F4685 has already been burned with MIOS 1.9g. If you get this message, then you know the PIC is working and MIDI Out is working. MIOS installation can also be confirmed by the LCD showing the MIOS startup message.

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 sammichSID, 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 SID application

You can find the latest MIDIbox SID Synth V2 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_sid_v2_0_rc37.zip

Download the ZIP file and extract it.

The sammichSID firmware is in the **setup_sammich_sid.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.

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 sammichSID will reboot and show the MIDIbox SID bootscreen and main screen.

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

If everything appears to be working, then you can install the "Bankstick" ICs (**IC6**, **IC7**, **IC8**, **IC9**, **IC10**) and watch the LCD while the MIDIbox SID application formats the Banksticks.

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

The Java-based patch editor and librarian can be found here:
http://www.ucapps.de/midibox_sid_manual_ed.html

The default patches are found in the MIDIbox SID Synth release package (they are in the "presets" folder).

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

If there are no problems with formatting the Banksticks, you can then install the SIDs (see **Installing SIDs**).

8. Installing SIDs

When you install SIDs, you should first perform voltage checks to make sure you have the jumpers set correctly and are supplying the sammichSID with a suitable power supply.

Refer to **Jumper Configuration** to set the jumpers to match your SIDs.

Jumper **JP** must be set correctly to suit the SID type. Jumper **JBP** must also have shunts if using 6581.

If using 6581, make sure you are using a ***regulated 12V power supply***.

If using 8580 or 6582, a regulated 12V power supply is preferred but not essential.

Test voltage between pin 28 (top-right corner) and pin 14 (bottom-left corner) of each SID's IC socket.

For 6581, this must be ***exactly 12V***.

For 8580 or 6582, this must be ***exactly 9V***.

Power off the sammichSID!

Insert the SIDs.

Insert the correct capacitors to the left of each SID (6581 uses 470pF, 8580/6582 uses 22nF).

The 6 holes of the machine pin strips allow for two different pin spacings of capacitors (100mil/2.54mm or 200mil/5.04mm). Thus, insert the capacitors in the 2nd, 3rd, 4th and 5th holes (100mil spacing) or 1st, 3rd, 4th and 6th holes (200mil spacing). You should cut the capacitor leads to about 3mm.

Refer to **Parts List** for identification of each type of capacitor.

9. Testing SIDs

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

However, if the sammichSID 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/sammichsid

http://www.ucapps.de/

http://midibox.org/forums

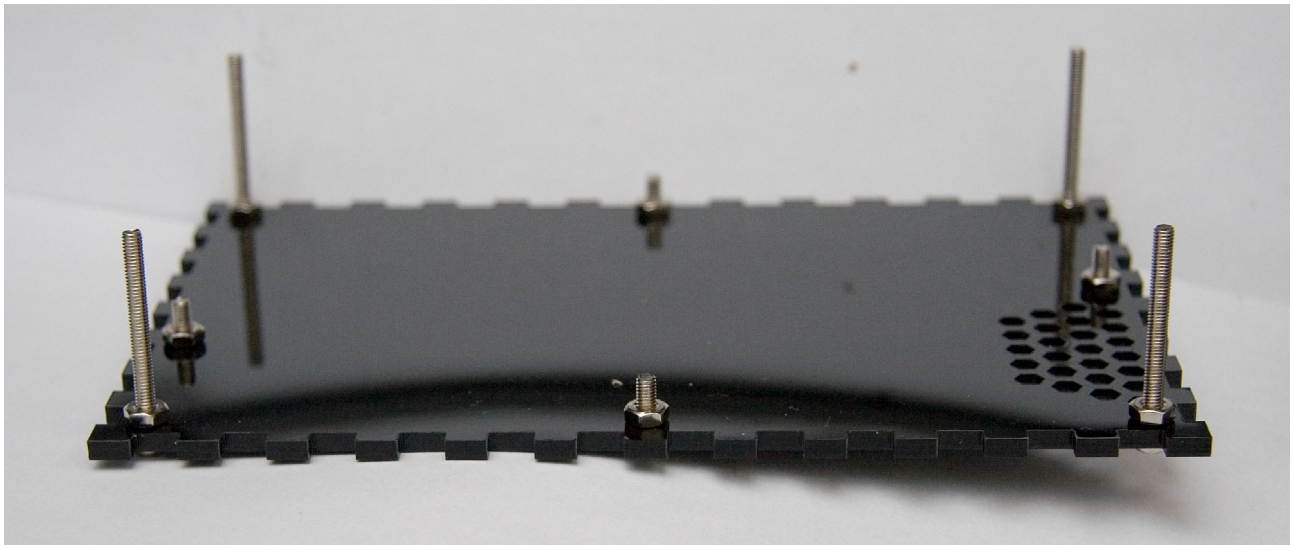
10. Using MIDIbox SID Synth V2

Read the user manual!

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

There are probably some specifics differences between sammichSID and the “default” MIDIbox SID 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 sammichSID 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 ***should not*** need to tighten these with a hex key (Allen key); tightening just with fingers is sufficient.

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 sammichSID 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 SID synth while you paint the panels (a slow process).

The original sammichSID prototypes were painted using acrylic paint, but since then, people have used other kinds of paint. Gloss enamel paint, like what is used in model making, can be used, and might be preferable to acrylic.

I do not have a "perfect" technique yet, but you can achieve very good results by doing the following:

Find some ***good*** acrylic or enamel paint. White paint will work best, colours might appear too dark against the black background. For colours, you could try white paint and then a coloured ink glaze.

Test your technique on a small area of the rear panel, where a big mistake is less obvious and easier to fix. I suggest the "POWER" label on the rear panel. If doing this, only peel off a section of the mask where you have added paint.

Using some paint thinned with water, apply the paint into the engraving, minimizing the amount of paint is left on the mask. Wipe up excess paint with a piece of damp sponge. It is better to apply multiple coats of thinned paint (at least initially) than thick paint. Avoid the temptation to get it done quickly and slap on thick paint from the tube, at best this will require a lot of scraping of paint from the mask, at worst there won't be clean edges to the engravings, which you'll discover after peeling off the mask and making it very difficult to fix.

IMPORTANT NOTE: You do not want paint sticking between the engraving and the mask, as this will cause paint to be pulled out of the engraving when you peel off the mask. It's OK if there's a little bit of paint residue on the mask, that's unavoidable. Wiping up excess paint with a piece of damp sponge not only avoids this problem, it also makes it easier to see how much paint is actually in the engraving and how clean the edges are.

Wait until paint in the engraving has dried and apply more paint. Continue to do this until the paint in the engraving is even and solid. You are aiming for the engraving to still be concave, not totally filled. If it looks even and you're happy with it, stop adding paint.

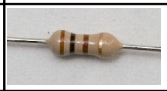
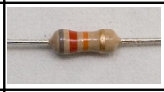
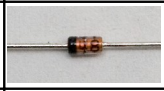
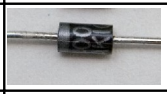
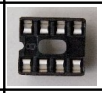
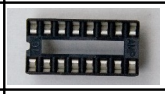
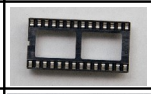
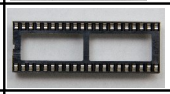
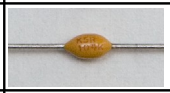
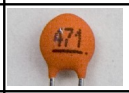
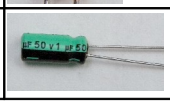
If required, use a craft knife to scrape some paint off the mask at the edge of the engraving, i.e. where it might pull paint out of the engraving when you peel off the mask. If you've wiped up excess paint as you applied each coat, you might not need to scrape off paint at all.

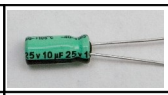
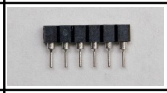
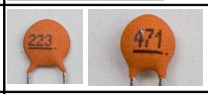
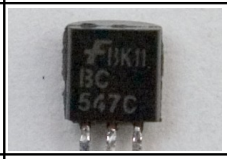
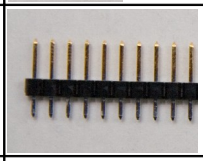
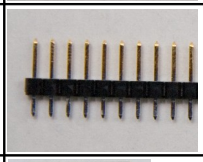
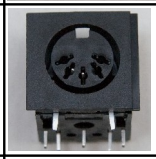
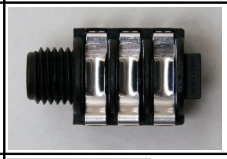
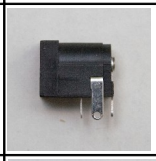
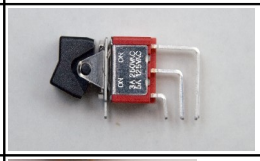
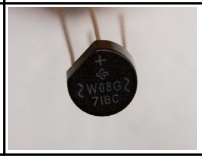
Slowly pull off the mask, checking how the mask is lifting off near the engraving.

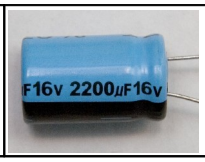
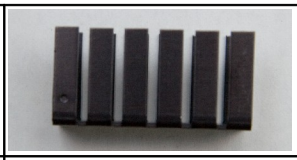
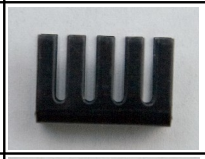
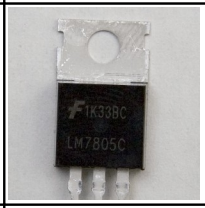
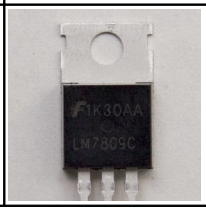
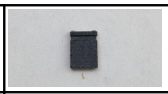
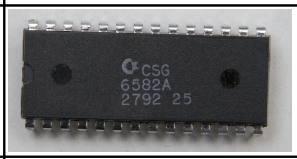
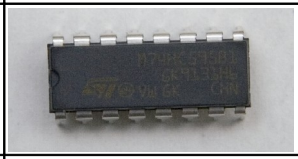
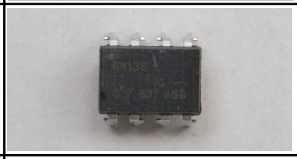
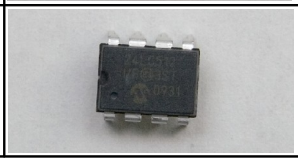
Clean up any imperfections using a craft knife. The matte black acrylic surface is quite tough and won't be scratched by scraping with the edge of a knife.

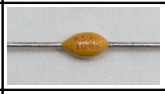
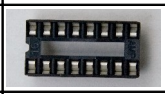
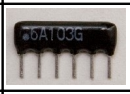
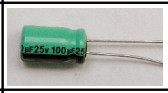
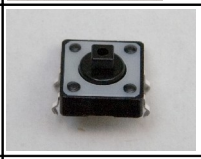
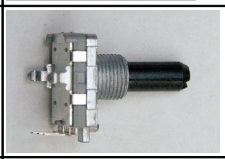
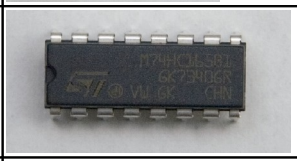
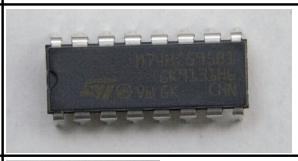
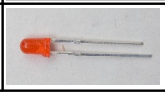
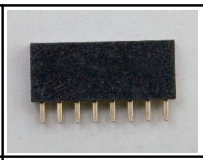
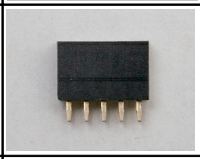
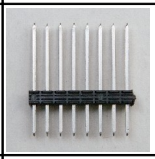
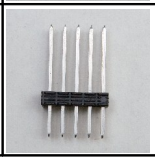
13. Parts List

Parts are presented in the preferred order of soldering.

Base PCB part name	Description		Mount Notes
R1	100 Ohm Resistor		
R7,R8,R11	220 Ohm Resistor		
R2, R4, R12, R80, R2L, R2R, R4L, R4R	1K Resistor		
R6	1K2 Resistor		
R5	5K6 Resistor		
R3, R9, R10, R3L, R3R	10K Resistor		
R4A	82K Resistor		
D1, D3	1N4148 Small Signal Diode		Align with stripe matching PCB.
D4, D5	1N4001 Power Diode		Align with stripe matching PCB.
IC2, IC6, IC7, IC8, IC9, IC10	IC Socket 8 pin		Align with notch matching PCB.
IC2L, IC3L	IC Socket 16 pin		Align with notch matching PCB.
IC1L, IC1R	IC Socket 28 pin		Align with notch matching PCB.
IC1	IC Socket 40 pin		Align with notch matching PCB.
Q1	10MHz crystal (low profile, model XT49S)		
C1, C2	33pF Ceramic Capacitor		
C3, C7, C8, C9, C8L, C8R, C9L, C10L	100nF Monolithic Capacitor (radial)		Mount flat.
C6	330n Monolithic Capacitor		
C4L, C4R	470pF Ceramic Capacitor		
C3L, C3R, C7L, C7R	1n Ceramic Capacitor		
C6L, C6R	1µF Electrolytic Capacitor		Align negative “-” (black stripe) lead with pad that IS NOT marked postitive.

C4, C10, C5L, C5R	10µF Electrolytic Capacitor		Align negative “-” (black stripe) lead with pad that IS NOT marked postitive.
C1L, C1R, C2L, C2R	machine pin strips		
C1L, C1R, C2L, C2R	22nF (6582/8580) or 470pF (6581)		Do not solder! Do not insert until after voltage checks!
T1	BC337 Transistor		Bend leads to fit PCB, align with flat side. See guide.
T1L, T1R	BC547 Transistor		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.
JP[2+1], JMI[3], JMO[3], JBL[3], JR4[2], JBP[2+2]	.100" Pin Strip Headers 40P STRT 1 ROW GOLD		For the L-shape “JP”, use a shunt to hold a 1-pin header to a 2-pin header while soldering. Don't melt the shunt by soldering too long!
J1[2], J2[2], J20[2], J3L[2], J3R[2], J4L[2], J4R[2], J4[4], J6/J7[5+5], J12, J13			These are not used in a default sammichSID construction. ***Do not solder them!***
J12, J13	DIN Jacks 5 PIN DIN PCB		
AUDIO_IN, AUDIO_OUT	Neutrik Phone Jack 3C STEREO 3-SPST NC (NMJ6HFD2)		
J1	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

C5	2200µF Electrolytic Capacitor		Align negative “-” (black stripe) lead with pad that IS NOT marked postitive.
Voltage Regulators and Heatsink Stage			
IC3, IC4	Heatsink TO-220 Dual (big one)		Use heatsinking compound! See guide.
IC3	Heatsink TO-220 (small one)		Use heatsinking compound! See guide.
IC3	5V Voltage Regulator 7805		Bend leads 90º and mount flat. Use heatsinking compound! See guide
IC4	9V Voltage Regulator 7809		Bend leads 90º and mount flat. Use heatsinking compound! See guide
Non-Soldered Components			
JBL, JMI, JMO, JP, JR4, JBP	Shunts (jumpers)		See guide for correct jumper placement! WARNING! Incorrect placement will kill SIDs or LCD!
IC1	PIC18F4685		Do not solder! Do not insert until after voltage checks!
IC1L, IC1R	6581 or 6582A/8580R5 (SID)		Do not solder! Do not insert until after voltage checks!
IC2L, IC3L	74HC595		Do not solder! Do not insert until after voltage checks!
IC2	6N138		Do not solder! Do not insert until after voltage checks!
IC6, IC7, IC8, IC9, IC10	24LC512		Do not solder! Do not insert until after voltage checks!

Control Surface PCB part name	Description		Mount Notes
R1-R8	220 Ohm Resistor		
C1, C2, C3, C4	100nF Monolithic Capacitor (radial)		Mount flat.
IC1, IC2, IC3, IC4	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.
C5	100µF Electrolytic Capacitor		Align negative “-” (black stripe) lead with pad that IS NOT marked postitive.
S1-S12	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, IC4	74HC595		Do not solder! Do not insert until after voltage checks!
LED	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 (2x8 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.