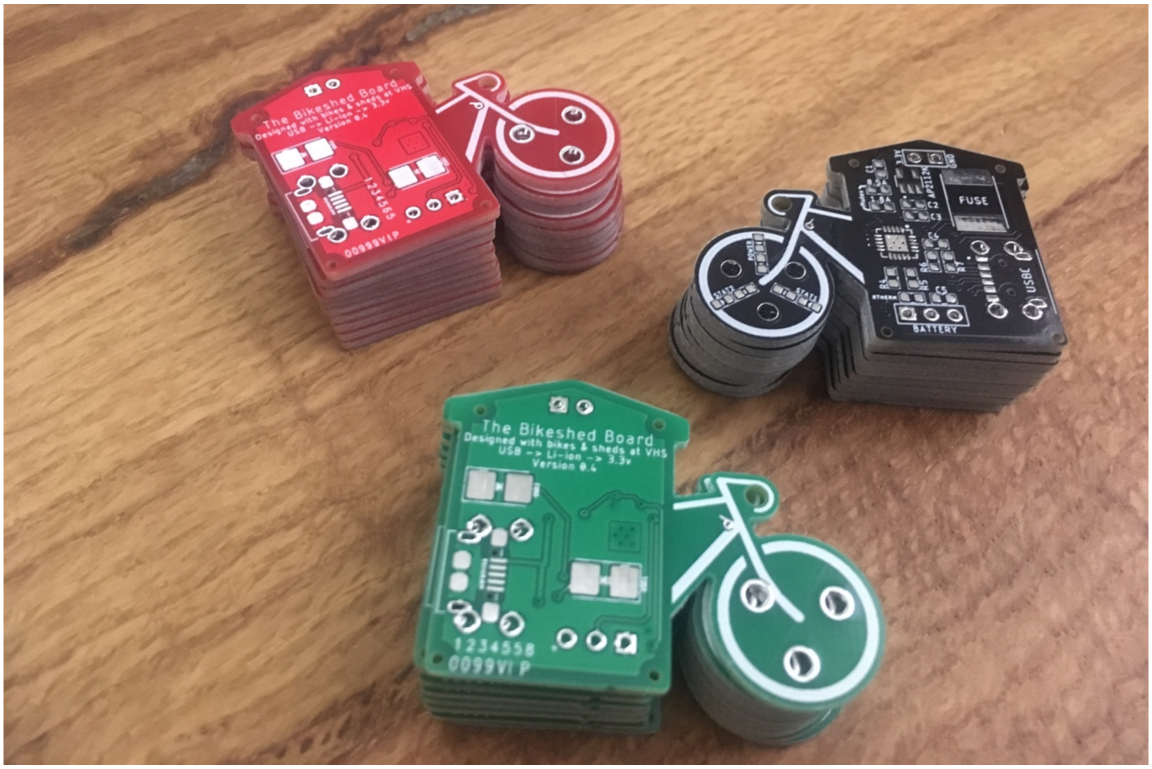
BIKESHED BOARD

Assembly Instruction Guide



What is "The Bikeshed Board"

The Bikeshed Board turns a project into a portable device with a battery. It gives you a power input through USB, it charges a Li-Ion or LiPo battery, and seamlessly switches between the two.

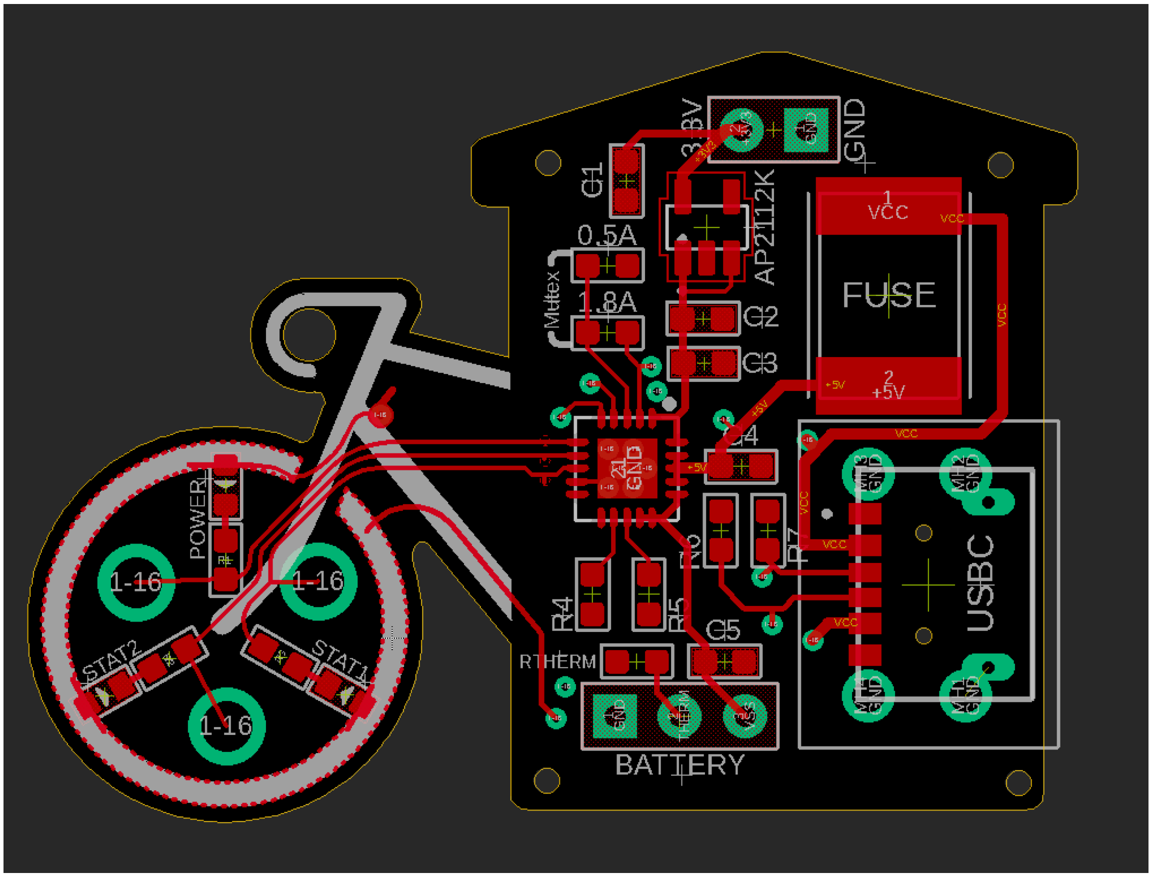
Is This Kit for Me?

This guide assumes you have a basic understanding of surface mount device soldering.

This kit will be difficult to solder with an iron; using solder paste with a stencil or syringe is recommended. Access to a reflow oven is required to reflow the solder paste.

If you have a project that needs 3.3v with less than 600mA of current, this board will be an excellent way to make it into a portable device.

GETTING STARTED



Bikeshed Board Top Layout

Apply Solder Paste

The first step to building this board is to apply the solder paste. To do so effectively, you will want to prepare the board to receive the paste by cleaning it with isopropyl alcohol (the type you can typically find at a pharmacy or in a first aid kit).

Use a clean, ideally lint-free cloth to cover the bottle of isopropyl alcohol. Reverse the bottle to slightly soak the cloth and then return upright.
Wipe the surface of the PCB to eliminate dust, oil from your skin, and any other foreign contaminates.

If you have access to a stencil for this board, align the holes of the stencil to match the pads and use masking tape to tape the stencil in place. If you don't have access to a stencil, use a solder paste syringe to apply solder paste on each pad.

If your solder paste is stored in a chilled environment, take your paste out to room temperature and allow it to warm before use.

If using a stencil, once the stencil is secured on the board, scrape paste over the board with your paste applicator to apply a thin coat of solder paste, and then return back over the stencil with a sharp angle to remove excess paste. Ideally, do each (application, and removal) in one pass each. Adding too much paste will allow it to spread under the stencil. As long as there is a thin coat on each pad, you should have enough.

There are numerous YouTube videos demonstrating how to apply solder paste. If you have questions, either ask someone who knows what they are doing, or study an instructional video for help.

Placing Components

The parts in this build are very close together. One design goal governing this board was to provide it as a physically small board. It is best to follow the build order in this guide so that it remains easy to place the last components.

Resistors

Resistors are bi-directional, which means you do not need to worry about which way they are oriented.

R1, R2, R3 – 470Ω resistors – 0603

Start by placing three 470Ω resistors in the three inner spokes of the wheel. The outer spoke pads will later be used for indicator LEDs.

R4 – 47kΩ resistor – 0603 – Charge Termination

The R4 resistor determines the charge termination point. If you choose to change this based on your specific battery, it should be set between 5kΩ and 100kΩ. To determine your specific value, use the formula:

$$I_{termination} = \frac{1000V}{R_4}$$

Where $I_{termination}$ is specified in mA and R_4 is in kΩ

R5 – 10kΩ resistor – 0603 – Fast Charge Rate

Place a 10k ohm resistor on R5 to set the maximum fast charge current. To adjust the maximum charge current, choose a resistor based on the formula:

$$I_{REG} = \frac{1000V}{R_5}$$

Where I_{REG} is specified in mA and R_5 is in kΩ

R6, R7 – 5.1kΩ resistor – 0603 – USB-C Charge Configuration

If you aren't using USB-C, leave these unpopulated.
If you are using USB-C, these tell the upstream device to set the voltage to 5V

RTHERM – 10kΩ resistor – 0603 – Battery Thermistor Override
The charge controller can adjust the charge parameters based on the internal temperature of the battery cell. If your cell has an internal thermistor, do not populate RTHERM. If you have a 2-wire battery with no temperature wire, populate this 10kΩ resistor to tell the charge controller to not use temperature control.

0.5A or 1.8A – 0Ω resistor – 0603 – Input Current Limiter
The charge controller can limit the input current to comply with USB specifications. If you are powering with USB-C or will always use a USB port that supplies higher current (such as a large phone charger), populate a 0Ω resistor on the 1.8A pads. If, however, you wish to power from legacy USB ports that limit current to 500mA, populate the 0.5A pads. **WARNING! DO NOT POPULATE BOTH 0.5A and 1.8A pads.**

Capacitors

The capacitors used in this build are bi-directional, you do not need to worry about which way they are oriented.
C1, C3 – 1 μF capacitor – 0603 – Filtering Capacitors
C2, C5 – 4.7 μF capacitor – 0603 – Filtering Capacitors
C4 – 10 μF capacitor – 0603 – Filtering Capacitors

LEDs

LEDs are polarized. Be careful to note which way they are oriented. LEDs will mark the *cathode* (negative end) with a line, dot, or some sort of indicator. Sometimes the indicator will be visible on the bottom of the component.
On a PCB pad for an LED, the arrow points at the cathode. On this board, the cathodes should be pointing at the hub of the wheel, and the anode (positive end) should be on the rim of the wheel.

POWER – Green LED – 0603 – Power Good Indicator
The power good LED will be lit any time that upstream power is provided to the Bikeshed board through the USB connectors. No LEDs will be lit when upstream power is not supplied. See

Table 1 for power indication states.

STAT1 – Yellow LED – 0603 – Status Indicator 1

The Stat1 LED is lit to indicate charge complete, or if in combination with STAT2, a fault has occurred. See

Table 1 for power indication states.

STAT2 – Blue LED – 0603 – Status Indicator 2

The Stat2 LED is lit to indicate charging is occurring, or, if in combination with STAT1, a fault has occurred. See

Table 1 for power indication states.

Table 1

State	**Stat 1**	**Stat 2**	**Power**
No Input Power	OFF	OFF	OFF
Shutdown with Power Connected	OFF	OFF	**ON**
Charging	**ON**	OFF	**ON**
Charge Complete	OFF	**ON**	**ON**
Temperature Fault / Timer Fault	**ON**	**ON**	**ON**
No Battery Detected	OFF	OFF	**ON**

Integrated Circuits

The integrated circuits have numbered pins. To help identify the correct orientation and placement, dots in silkscreen will appear both near the first pin on the chip and the first pin on the board. Match the corner of the chip with the dot to the corner on the board with a dot. Once placed, use a microscope or some form of magnification to verify placement of the component aligns with the pads on the board.

APK2112k – AP2112K-3.3TRG1 – SOT – 3.3V Power Regulator

This chip regulates the incoming voltage down to 3.3 volts for the system output. This chip is easy to orient because one side only has 2 pins while the other has 3.

MCP73871T-1AAI/ML – QFN-20 – Battery Charge Management Controller

Read this entire section before attempting to place this component.

This is the brains of the operation. The chip is both physically centrally located on the board and is the center of everything that is happening on the board. This board is responsible for the charge management and a host of safety features to reduce risk for charging and providing power to the regulator.

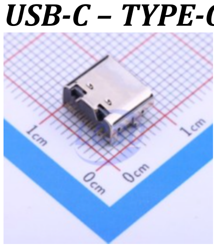
This is a static sensitive device, make sure to practice good electrostatic discharge protection practices, and use tweezers (not your hands) to handle the chip.
Before placing, make sure the pads are nicely coated with solder paste. Do not allow too much paste or bridging under the chip can occur. Neatness matters when placing this chip! If you take your time on one component on this board, this is the one to do it with.

Be sure to match the 1 pin (indicated by a dot) to the 1 pin's pad in the upper-right corner (next to C3). Try to place the component without sliding it across the pads too much.

Once placed, validate with magnification that the pins align nicely. If you are using a microscope, tilt the board slightly to see the edge of the chip and focus to account for the additional height. Check all four sides.

USB Connector

USB-C – TYPE-C-31-M-17 – USB-C Connector



If you intend to power this through USB-C, place this connector.

If you intend to add the back side Micro USB connector, omit the USB-C connector. Only install one of the two USB connectors. If adding the Micro USB connector, add after baking.

Protection

Sometimes things don't go as planned, so these components provide a few failsafe protections from some of the more common problems you can encounter.

FUSE – 0ZCF0200FF2C – 2920 – PTC Resettable Fuse

Just in case something goes wrong, this fuse is a last line of defense. Designed to allow 2A of continuous current, this fuse will trip if a spike of 4A or greater is used downstream. If tripped, it will become a very high value resistor, effectively disconnecting the entire board from your upstream power source. This should only trip in the event of a catastrophic failure.

If, however, it does trip, it is self-resetting. All you need to do is disconnect the device, wait a moment, fix whatever caused the device to trip (check for bridged connections), and then plug the device back in.

ESD1, ESD2 – SMAJ5.0 – DIOC5025X229 – ESD Diodes

These are optional protection to protect the charge configuration lines in the USB connector from unexpected spikes in power. Some cables and connectors can wear down, and if not inserted directly into the USB plug, there is a chance that the power pins can short to the CC pins.

These diodes are located on the back of the board, and unless you are set up to do SMD soldering on two sides, it is best to add these after baking. However, they are large enough to be able to be soldered by hand.

The diodes are polarized so be sure to match the arrow to point at the cathode (which will have the orientation mark). If trace the circuit, at first glance, these appear to be oriented backward, however their standard condition is to prevent power from flowing through them. If high voltage is encountered, they will allow reverse current to ground, which will aid in protecting the circuit.

Headers

2 and 3 pin headers are provided on the board with standard spacing for .1” pitch headers. However, for embedded projects it is recommend soldering directly to your project, and to solder a battery connector that mates with your selected battery. As through-hole components, these should only be added after baking.

Baking your Board

After your components are seated, do a final check to ensure nothing has moved out of place. Use magnification if available to re-check pins. Components will mostly self-align when baked, but fine pitch components, such as the charge controller require tighter tolerances. Getting the right amount of solder paste and good placement will limit failures and/or rework.

Consult the instructions for your solder oven, do not attempt to use your kitchen oven to reflow your board.

TESTING YOUR BOARD

Input Power Test

Your first test should be without a battery. Plug the USB-C connector into a known good power supply. If anything unusual happens, immediately disconnect power.

For these tests, you should apply a sacrificial load (such as resistors) across the 3.3V to GND at the top of the board. Aim to draw between 25mA and 200mA but not greater than your fast-charge current. Ideally the dummy load should be 5V tolerant in the event of a soldering failure.

Validate the following:

- **The Power LED lights up**
    - This indicates power is flowing to the charge controller
- **The bottom lead of the fuse and the square GND pin above the fuse register at approximately 5V DC**
    - This indicates power is flowing through the cable
    - This indicates the CC pins have requested the correct voltage
- **The top lead of the fuse and the square GND pin above the fuse register at approximately 5V DC**
    - This indicates power is flowing through the fuse and it is not tripped
    - This indicates the CC pins have requested the correct voltage
- **The 3.3V and the GND pins at the top of the board register at approximately 3.V DC**
    - This indicates that output power is provided by the charge controller
    - This indicates that the voltage regulator is regulating power correctly.

If any tests fail, do not continue with the next test, instead check for failed solder joints and debug the circuit. Only proceed to the next test once all tests pass.

Battery Power Test

Next, disconnect the USB-C connector and plug in a partially charged battery (less than 4V but more than 3.2V) to the battery terminals. Enclose the battery in a Li-Po safe fire-proof bag.

Validate the following:

- **No LEDs should be lit.**
    - This indicates one aspect of proper connections for the LEDs
    - If any LEDs are lit, there may be a short to +5V
- **The 3.3V and the GND pins at the top of the board register at approximately 3.V DC**
    - This indicates that battery connection to charge controller is connected.
    - This indicates that the voltage regulator is regulating power correctly.

If any tests fail, do not continue with the next test, instead check for failed solder joints and debug the circuit. Only proceed to the next test once all tests pass (including prior tests).

Full System Test

With the battery connected, plug in the USB-C cable. Enclose the battery in a Li-Po safe fire-proof bag.

Measure the +V and GND pins of the battery and note the voltage.

Validate the following:

- **The Power LED lights up**
    - This indicates power is flowing to the charge controller

The expected STAT LEDs light up. Consult

- Table 1 **for status light logic.**
    - This indicates that the battery is performing as expected.
- **The 3.3V and the GND pins at the top of the board register at approximately 3.V DC**
    - This indicates the voltage regular is still regulating power correctly.

If all tests pass, allow the board to remain energized with the battery connected for 10 to 20 minutes. Do not allow the board to be unsupervised! Monitor battery temperature to ensure it is not warm to the touch. If noticible warming occurs, disconnect the power and battery immediately.

After the 10-20 minute charge test, disconnect the USB connector and measure the +V and GND pins. Validate the voltage of the battery has increased.

BONUS FEATURES

- The wheel can be cut off and all status pins are still accessible (however, you might need a super steady hand).
- The large un-tented holes on the wheel provide electrical connections to the status pins.
- If you detach the bike portion and manage to solder to the tiny pads provided for the status pins, the detached wheel can be wired back up as a remote status indicator
- The gear shift provides +5V, just in case you need it for your indicators (or if you are doing the remote indicator hack)
- The hole near the handle bars makes a great mounting hole.

- The tiny holes near the corners of the shed work great for needle and thread for your wearable ideas.
- There are two different USB connectors available. If you are handy, you can connect both for maximum flexibility, but don't plug both in at once.
- If you intend to use a boost/buck converter, you can bridge the connection between the 1 pin and the pin above it for the APK2112K voltage regulator and omit that part. This will provide between the battery output voltage and +5V to the output 3.3V pin.
- The output pins conform to a SAO specification if pins/sockets are added. #badgelife

QUESTIONS, HELP, AND BIKESHEDS

The Bikeshed Board was made in partnership with the Vancouver Hackspace. If you have questions about your board, please feel free to strike up a conversation at https://talk.vanhack.ca

Also feel free to post a picture of your board and the creation it powers.

BACKGROUND

The Bikeshed Board is a small board designed to allow you to run low power projects with a single-cell lithium-ion or LiPo battery.

It is powered by USB-C (or MicroUSB) and provides 3.3v with up to 600mA of current to your system. You can set the maximum input current to either 0.5A (for older USB specifications) or 1.8A.

What is special about this board, however, is that it can handle a lot of conditions that other chargers can't. These are things you would expect pretty-much any embedded device to do. The design methodology that makes this possible is called PowerPath.

What can PowerPath do and why is it so useful?

PowerPath does active management of both battery charging and the output current separately. It prioritizes system power over battery charging power, and allows for a clean handoff over to battery power should the upstream source be disconnected.

If you've ever had a device that couldn't turn on until the battery was charged beyond a certain percentage, that is one symptom that the device is potentially not using PowerPath. When devices couple the battery and system power together, if the battery has a lot of demand (such as a depleted battery), there may not be enough supply current to satisfy both. In this case, the system will either not power on due to a low dropout regulator, or worse, the system will brown out when attempting to use too much power.

A device with PowerPath will, instead, allow system power to be fed immediately. Any additional power that the supply offers will be sent to the battery.
Additionally, PowerPath should provide you a seamless experience when plugging in or unplugging your device. You shouldn't need to reboot.
It works like you would expect it to work.

What about charging LiPo/Li-ion, isn't that complicated?

It is, but one component handles all of that for us. Depending on the state of the cell, a proper charger can go through several different modes of charging. Sometimes it is constant current, sometimes it is variable current. There are two settings on this board that allow you to set parameters that align to the capabilities of your battery. You have been provided some good defaults, but if you have an extra-small battery or an extra-large battery, it is recommended you adjust the configuration resistors to match.

What's with the Name?

When designing this board, a lot of suggestions were gathered from the Vancouver Hack Space circuit board design channel. Lots of great suggestions were provided (for example, this is why we have two different USB ports).

Other suggestions (such as what board color it should be) started to feel a bit like bikeshedding (https://en.wiktionary.org/wiki/bikeshedding) so it only seemed reasonable to ask for ideas on a name. The Bikeshed Board was the obvious choice when it was suggested, which contributes to the shape and theming of the board.

Those who contributed suggestions toward the design of this board are responsible for this kit being in your hands today. Without them, a vital part of this design would have not been complete. As an incomplete list, there were contributions toward component selection, electrical design, layout, feature (creep), prototyping considerations, test builds, naming, art, this document, and much more.

BILL OF MATERIALS

Qty	Value	Device	Package	Parts
3	470R	SMD-RES-470R-5%-1/10W(0603)	R0603	R1, R2, R3
1	47K	SMD-RES-47K-5%-1/10W(0603)	R0603	R4
2	10K	SMD-RES-10K-1%-1/10W(0603)	R0603	R5, RTHERM
2	5.1K	SMD-RES-5.1K-1%-1/10W(0603)	R0603	R6, R7
1	0R	SMD-RES-0R-5%-1/10W(0603)	R0603	0.5A or 1.8A
2	1uf	CERAMIC-1UF-25V-10%-X7R(0603)	C0603	C1, C3
2	4.7uf	CERAMIC-4.7UF-10V-10%-X5R(0603)	C0603	C2, C5
1	10uf	CERAMIC-10UF-10V-10%-X5R(0603)	C0603	C4
1	GREEN	SMD-LED-CLEAR-GREEN(0603)	LED-0603	POWER

1	YELLOW	SMD-LED-CLEAR-YELLOW(0603)	LED-0603	STAT1
1	BLUE	SMD-LED-CLEAR-BLUE(0603)	LED-0603	STAT2
1		AP2112K-3.3TRG1	SOT95	AP2112K
1		MCP73871_QFN20	QFN	MCP73871
1		TYPE-C-31-M-17	TYPEC31M17	USBC
1		0ZCD0200FF2C2920	2920	FUSE
2		SMAJ5.0ALFCT-NDDO-214AC,SMA	DIOC5025X229	ESD1, ESD2

A NOTE FROM THE CREATOR

Thank you for building my board, may this empower you to create great portable things!