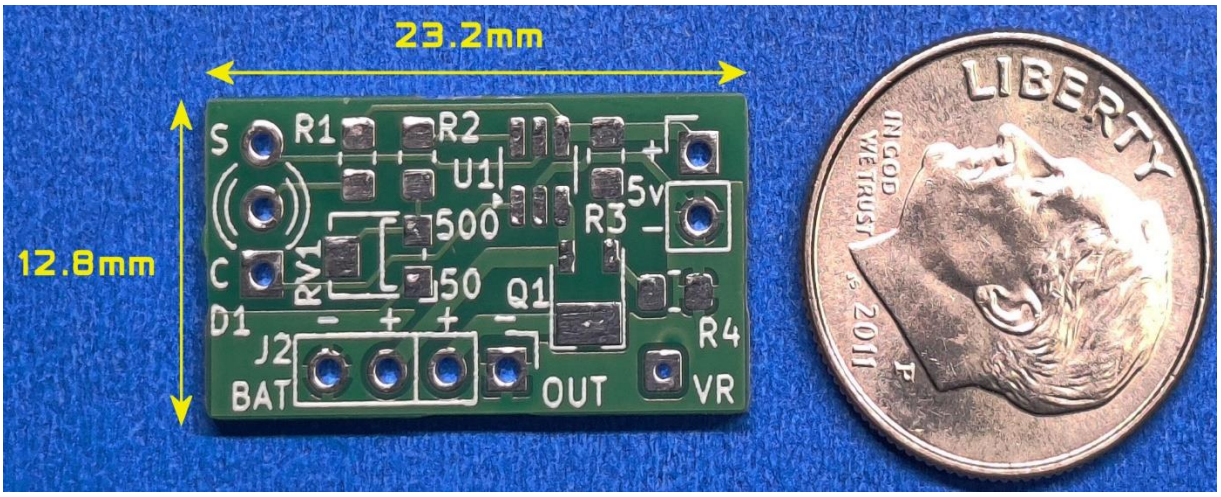
TP4057 Lipo Battery Charger Board

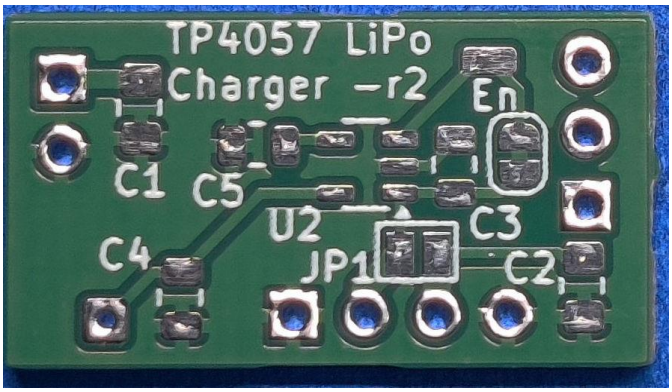
Contents

Description

Front



Back



The TP4057 Lipo Battery Charger Board is used to charge various lithium polymer or “Lipo” batteries. It supports single-cell lithium-ion or lithium polymer batteries and has an adjustable current setting from 50ma to 500 ma (RV1 can be bypassed and fixed resistor values for R2 can be used instead to set the charging current). This works great for most of the batteries needed for smaller projects easily up to 1 amp if needed. The board uses a common anode dual color LED to show the charging/standby states. The LED can either be mounted on the board or wired off board as needed. You can also use two individual LED’s if you want.

Connections to BAT IN, OUT, and 5v are at a 2.54 mm pitch supporting various headers, plugs, and sockets.

5v is used as the input voltage to charge the battery. A USB power source works well for this. I’ve made a custom 3 pin charging cable from an old USB cord to charge most of my models. Q1 is used to disconnect the output side while the battery is charging but this can be bypassed with JP1 if needed.

The back of the board has support for an optional low current voltage regulator. This fits either an SOT-23.5 or SOT-25 package of which there are a few types. Useful for driving some of the color displays that require a 3.3 voltage supply.

This board was designed to be as small as possible while providing a few options for battery charging and voltage regulation. Typically used in scale model builds, dioramas or other areas where a small compact lipo battery charger is needed.

Board Details

- Dimensions: 23.2 x 12.8 mm
- Parts Supported: TP4057
- Battery Types: Lipo battery from 50ma up to 1 amp. Larger batteries can still be used but charging times will be longer.

Parts List

Part Reference	Quantity	Value	Description	Source Links
U1	1		TP4057	AliExpress
Q1	1		PMV30XPEAR MOSFET SOT23 SMD	Mouser Electronics https://www.mouser.com/ProductDetail/771-PMV30XPEAR
R1, R3	2	1kΩ	0805 SMD Resistor	Mouser Electronics https://www.mouser.com/c/passive-components/resistors/film-resistors/thick-film-resistors-smd/?case%20code%20-%20in=0805&instock=y
R2	1	1.6kΩ	0805 SMD Resistor	Mouser Electronics
R4	1	10kΩ	0805 SMD Resistor	Mouser Electronics
C1	1	2.2uf	0805 SMD Decoupling cap	Mouser Electronics https://www.mouser.com/c/passive-components/capacitors/ceramic-capacitors/mlccs-multilayer-ceramic-capacitors/multilayer-ceramic-capacitors-mlcc-smd-smt/?q=0805%20capacitor&capacitance=2.2%20uF%7C~10%20uF&case%20code%20-%20in=0805&termination%20style=SMD%2FSMT&instock=y
C2	1	10uf	0805 SMD Decoupling cap	Mouser Electronics
RV1	1	20kΩ	TC33X-2-203E Variable Resistor	https://www.mouser.com/ProductDetail/Bourns/TC33X-2-

Part Reference	Quantity	Value	Description	Source Links
				203E?qs=kSSonJSvPGoOcNyKSIM%252Bog%3D%3D
D1	1		LED, 3mm Common Anode - Bivar	https://www.mouser.com/ProductDetail/Bivar/3BC-3-CA-F?qs=GLEjZbDuyKP1pSWOLz6tPw%3D%3D
U2	1		Optional 3.3V LDO Regulator SOT23.5/25 MIC5219-3.3YM5-TR AP2112K-3.3TRG1 MIC5365-3.3YD5-TR	Mouser
C2, C4	2	1uF	Optional 0805 SMD Decoupling cap	Mouser
PCB	1		TP4057 Board	

Pre-Assembled Boards

If you purchased an assembled PCB your board will be assembled based on the option you selected:

- **Option 1**: Board assembly with
  - TP4057 (U1)
  - All supporting parts for TP4057 (Q1, R1-4, C1-2, RV1, D1)
    - Note: D1 will not be mounted but included with the assembled board)
  - Parts listed as OPTIONAL will NOT be loaded (U2, C2, C4).
- **Option 2**: TP4057 Board assembly with the optional regulator mounted
  - MIC5365-3.3YD5-TR (3.3v @ 150ma max)
  - C2, C4 mounted

- Pre-assembled boards will be tested prior to shipment.

Assembly Guide

Caution: Electrostatic discharge (ESD) is a sudden and momentary flow of electric current between two differently-charged objects when brought close together or when the dielectric between them breaks down, often creating a visible spark associated with the static electricity between the objects. [1]

[1] Definition provided by From Wikipedia, the free encyclopedia. For more information on ESD see https://en.wikipedia.org/wiki/Electrostatic_discharge

This type of shock can cause damage to ESD sensitive parts such as those used in this build especially U1. Proper ESD protection and soldering equipment should be used to prevent damage to parts during assembly and implementation into your project.

Assembly Planning

The smallest components are 0805 and while small can still be hand soldered with care and patience. A fine tip soldering iron is useful along with 0.015” (0.38mm) flux core solder and extra flux if needed. See the references section for a YouTube video link on assembling this and other boards.

A note on connector sockets: The BAT and 5v locations support 2.54mm pitch sockets. However, I have found that these can cause a height issue with scale models as space can be very limited. For flexibility I usually wire directly to the board or use in-line connectors to keep the board height to a minimum.

PCB Assembly

- PCB assembly can be completed in any order.
- If using a hot plate or reflow heater I usually start with the side with the most SMD or hardest parts to hand solder and then hand solder the other side.
- If you are completely hand soldering my recommendation is to complete the top of the board by first mounting U1.
- Next mount the resistors R1-4 and RV1.
    - Note: you may want to use a fixed value resistor for R2 and if so, you won’t need to install RV1 but you will need to bypass it with a jumper wire. See the table in Setting the Charging Current for possible values.
- You can mount Q1 next or optionally bypass the need by adding a solder jumper to JP1.
- Depending on how you will place D1 you can either mount it to the PCB or connect wires for a remote placement of the LED.
- Now with the top assembled you can add C1 & C2 to the bottom.

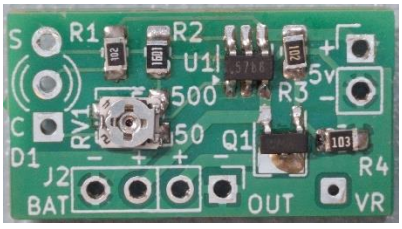
Optional Voltage regulator

- If you are using the optional LDO voltage regulator then mount U2 and C2 & C4.
- Note that C5 (470pf or some other appropriate value) may be needed for some regulators.
- The voltage output will be at the VR pad.

- If you will be using a connector for charging power and battery connections then install the 1x2 and/or 1x4 (2.54mm pitch) headers or simply wire the connections directly. You can also use a 1x3 connector for the LED.

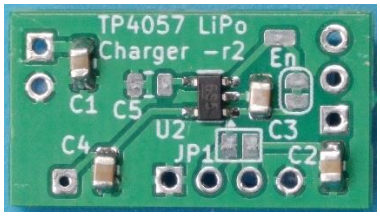
- See the Board Options/Configuration Section for more detail on bypassing Q1 or using an the enable pin for the option regulator.

Assembled Images



Top Assembly

Bottom Assembly



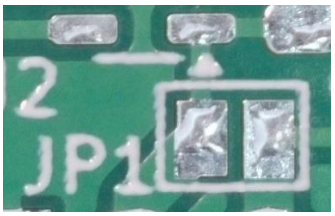
Bottom Assembly with optional voltage regulator

Board Options/Configuration

This section should be reviewed for those that are assembling the board themselves or purchased a pre-assembled version.

Cutoff Bypass (Q1)

By default, JP1 is open so that Q1 can be used to cutoff the project voltage when charging the battery. Making a solder bridge on JP1 will keep the project power on when charging. Q1 can be removed with this option.



Regulator Enable

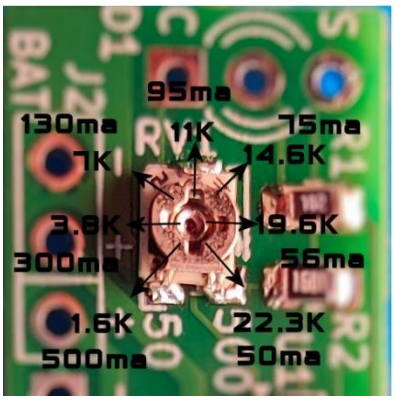
By default, the enable line for the regulator is connected to V+. To use it as a functional, enable line you can cut the connection between the EN pads. Using an Xacto knife or Dremel cutter carefully cut the trace between the two pads. Then you can use the upper EN pad to control the enable pin of the regulator. Should you decide not to use the enable at a later date then you can use a solder bridge to reconnect the previously cut EN pads.

Board Connections

Power	**5v +/-** Connect an appropriate 5 volt power source to the connector. Typically, a USB port.
Battery +/-	Connection point for the Lipo battery.
Battery Out +/-	Connection point for the project supply, 3.7v.
LED	Center is the common anode with the S side for standby (charged state) and the C side for charging. *Most sample circuits show a 1k resistor to a 5v charging source. Very few data sheets show the supported current range for CHRG/STDBY but it seems to be 5ma typical and 10ma max.*
VR	If you added a voltage regulator to the bottom side of the PCB then the VR pad will provide the output voltage.

Setting the Charging Current

You can get an approximate charging current by setting RV1 to one of the positions shown below. Note that the silk screen markings are reversed on the board and should be ignored. Review the charging specifications for your battery to determine the appropriate charging rate needed.



To calculate the programming resistance for the charging current use this formula:

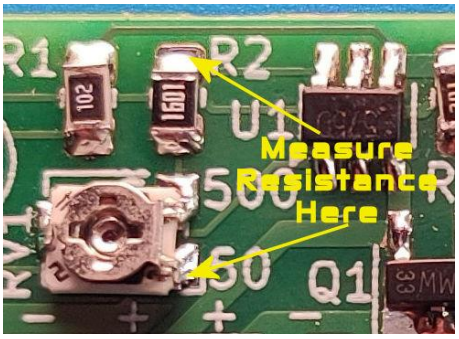
Rprog = 1000V / I BAT

Check the battery specifications for its maximum charging current. Typical capacity values are 1C or 0.5C which would be either the maximum current capacity or ½ the capacity of the battery. Some common values are shown in this table.

Some common values for Rprog (R2 + RV1)

R (Total)	I Charge BAT
1.6k	500ma
2k	400ma
3k	300ma
4.7k	225ma
10k	100ma
13.3k	75ma
20k	50ma

To set a more accurate charging current adjust RV1 for a total resistance between these two points.



Carefully adjust RV1, using a 1.6mm to 2mm screw driver blade (Typically ones used for eye glass repair will work) to set a total resistance of RV1 plus R2 as needed. Do not move RV1 past the stops at each end. You can also check the charging current using an Amp Meter on the 5v supply input.

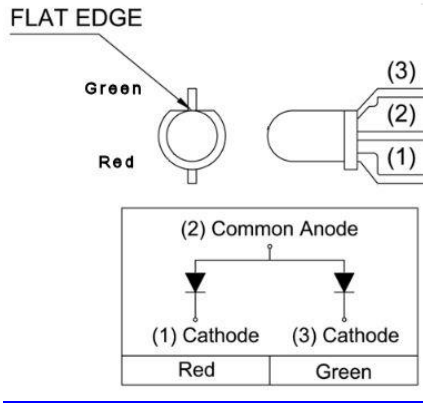
Battery Protection Circuit

This PCB does not have a battery protection circuit built in to it. Many of the lithium polymer batteries available have the protection circuit on the battery itself. Make sure that the battery you select to be used with this board has the protection circuit on the battery.

Charging LED (D1)

The charging/standby LED is not mounted for the assembled board so that it can be located or replaced with some other option. Two single LED's can be used or the dual color LED provided can be used and mounted where desired. Here is the diagram for the LED connections.

*Most sample circuits show a 1k resistor to a 5v charging source. Very few data sheets show the supported current range for CHRG/STDBY but it seems to be 5ma typical and 10ma max. This design uses a 1k*Ω *resistor as the default.*



Mounting Options

Hot glue is my go-to option for PCB mounting in models. It has great hold and sets up quickly. It can easily be removed and reapplied. Double sided tape or possibly Velcro could also be used.

References

- **Github: Development board documentation and schematics.**
  - **https://github.com/JohnnyElectronic/Dev_Boards/**
- **YouTube: Board assembly and project videos that are related to this board.**
  - **https://www.youtube.com/@Johnny_Electronic**
- **Popular Battery Charger ICs for Lithium Battery Charging and Protection**
  https://components101.com/articles/how-to-select-the-right-battery-charging-ic
- **TP4057 Top Power Data Sheet**
  http://toppwr.com/uploadfile/file/20230304/640302a47b738.pdf

Revisions

R1	First release
R2	Added variable resistor to make the charging current adjustable. Added voltage regulator option. RV1 silk screen is backwards.

Disclaimer

This information is provided “as-is” with no representation or warranty of any kind whether express or implied. However, I’ve tried to make this document (as well as the supporting videos) as useful and accurate as possible. If you find something that is incorrect or confusing, please let me know as I would like to make the correction so others will not have the same issue.

Feel free to email me for issues you may have with this board or if you need extra help with coding, programming, or just design ideas for your latest project please check out my Patreon page.

johnnyelectronic1@gmail.com