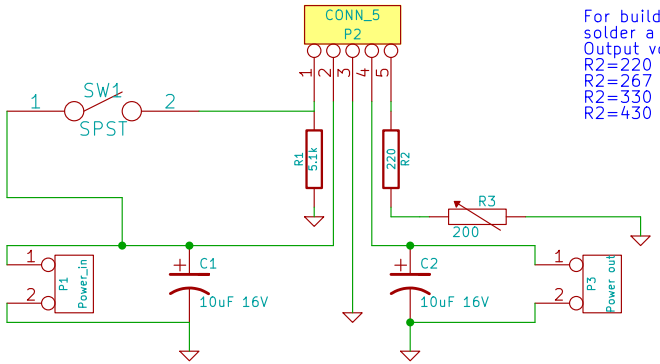

For builds that do not require variable voltage
solder a jumper wire across POT terminals (R3 in schematic).
Output voltage will be set by R2. Assuming 1% tolerance:
R2=220 ohm => Vout = 5.91-6.02
R2=267 ohm => Vout = 4.97-5.06
R2=330 ohm => Vout = 4.14-4.21
R2=430 ohm => Vout = 3.31-3.37

The Baby Potato PCB is a small backpack for, and approximately the same size as, the OKR-T6/10.
It is based on /u/slumberland 's Potato v2 schematics (recreated here) which is heavily influenced by designs/schematics from Mamu, The Goss, and others. Designed by a modder, for modders.
Documentation is intentionally sparse. While the board itself is inherently safe,
the batteries that one can attach to it are very much not. Misuse and Abuse can lead to the loss of a face.
Be careful and direct questions to
www.reddit.com/r/openpv

The Baby Potato is available for purchase at OshPark
http://oshpark.com/shared_projects/A80Nd2IR

Design files and Documentation are published on Github
https://github.com/rdasx/baby_potato

All Components on the Baby Potato board are 1206 (SMT) Footprint.
Capacitors are MLCC type at $0.40/ea and are optional but recommended. Don't be cheap.
Resistors are thick film type and $0.10/ea.
Connect your own powersupply (12V or 2 series Li-ion cells), potentiometer (optional), momentary power switch, power connector (510), and OKR-T6/10 module.

File: potato.sch Sheet: /	**Baby Potato**	Drawn By: /u/Militancy
Title: Potato v2 Schematics, recreated from /u/slumberland 's schematics		
Size: USLetter	*Date: 23 mar 2014*	*Rev:* 0.1.a
KiCad E.D.A.		Id: 1/1