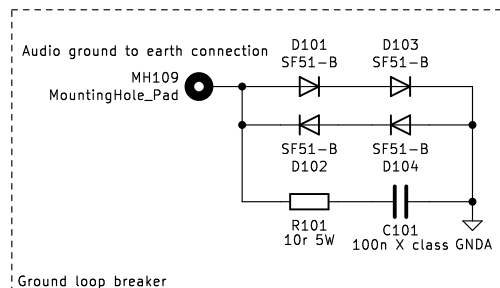
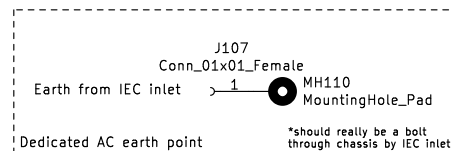
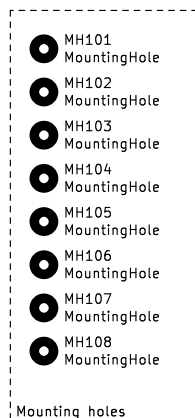
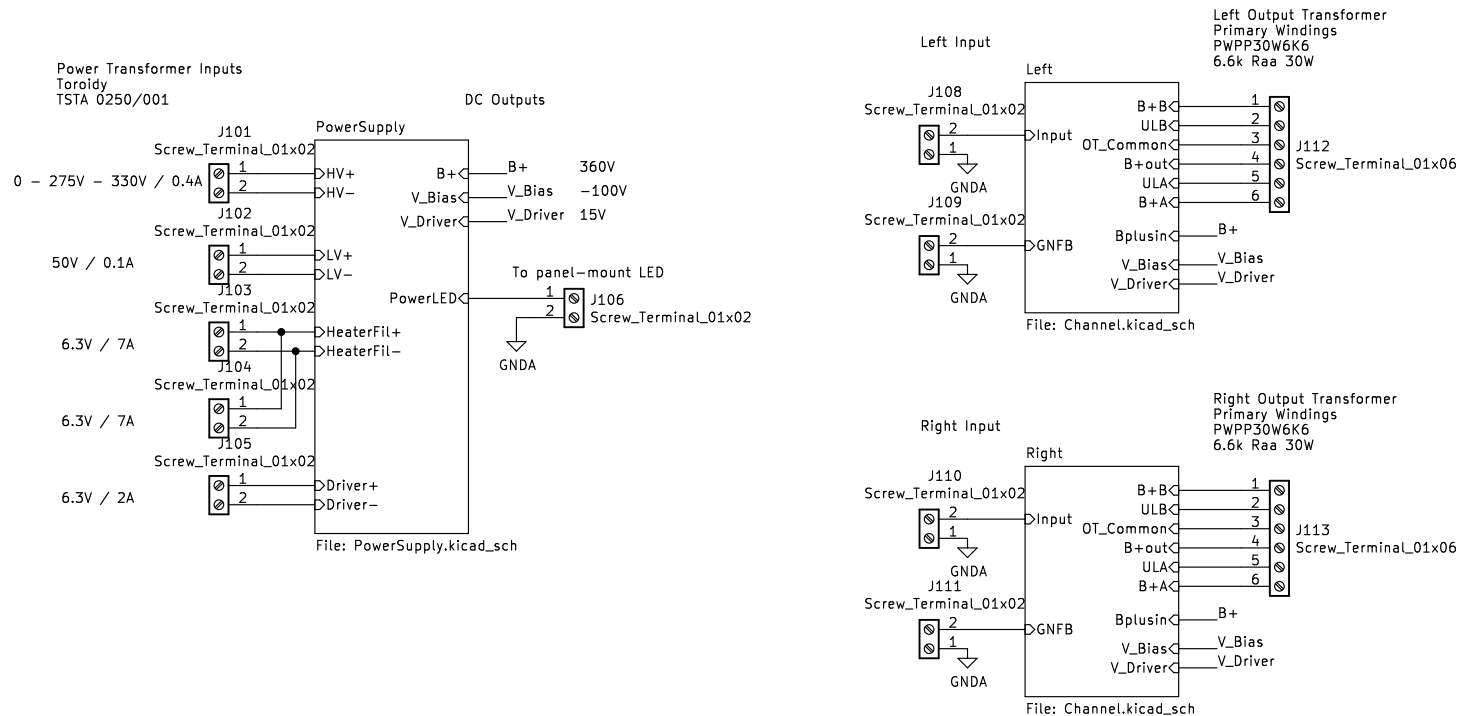


Notes / Questions

- Fix B+ connections wrt to the smoothing cap
- Add In net labels



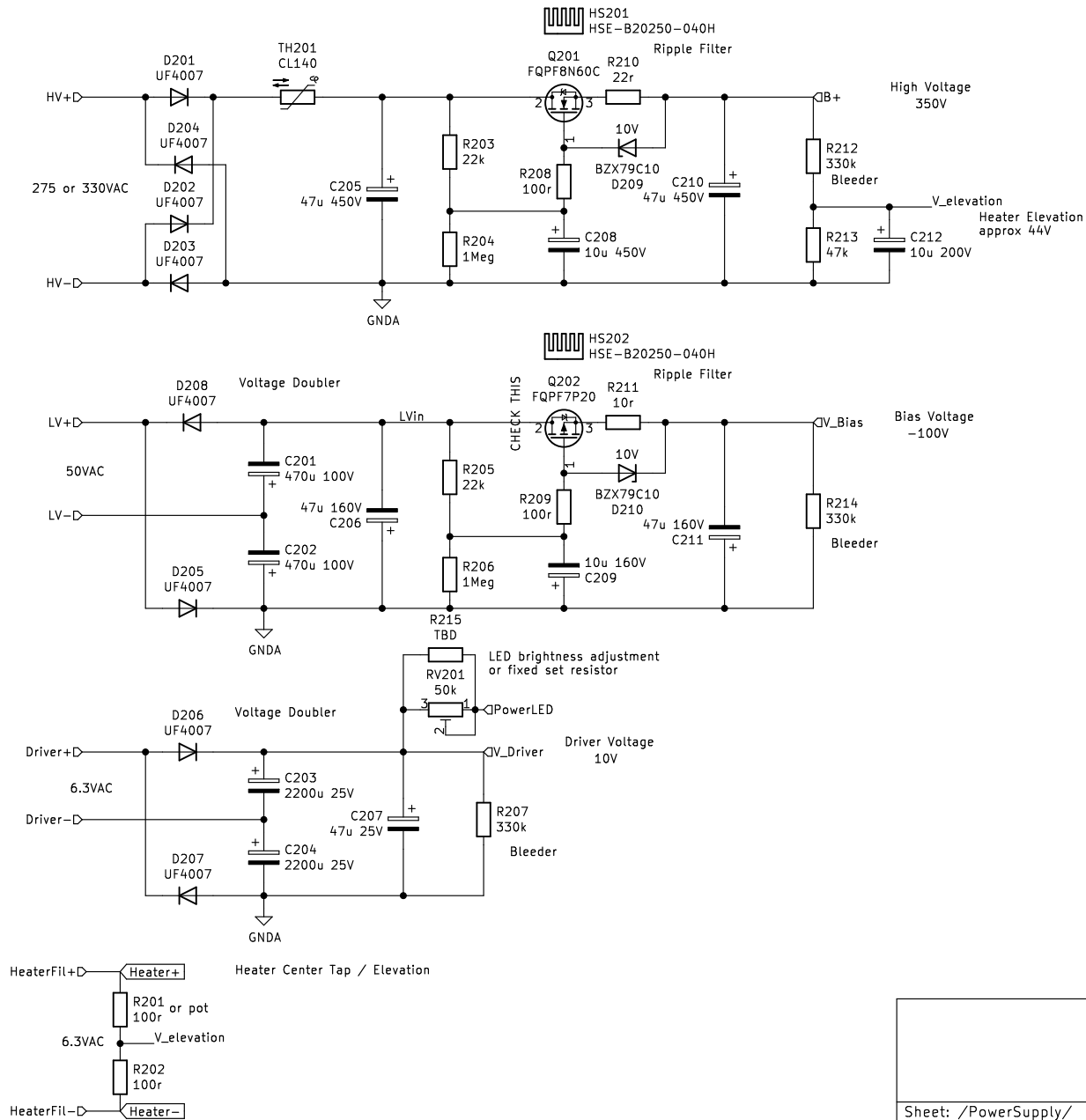
Sheet: /
File: BabyHuey.kicad_sch

Title: Baby Huey – "Engineer's Version"

Size: A4 Date: 2021-04-25

KiCad E.D.A. kicad (6.0.0-0)

Rev:
Id: 1/4



Notes

- * Both FETs are encapsulated in plastic
- no isolation pad / shoulders needed
- use heat transfer pad / paste

The regulator gives protection to inrush
 Max V across the 22r resistor is $V_Z - V_{GS} = 10 - 4 = 6$
 Max current is $6/22 = 270\text{mA}$

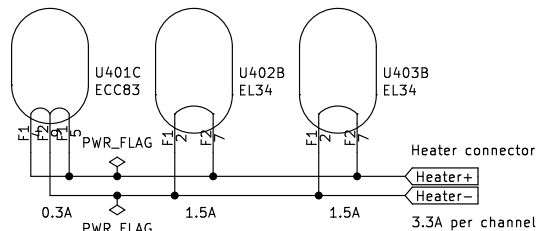
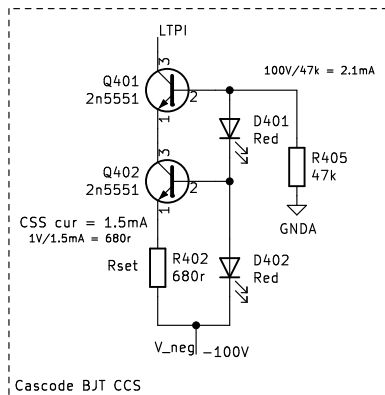
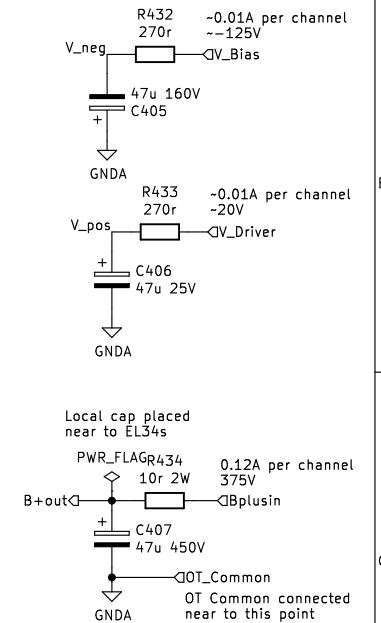
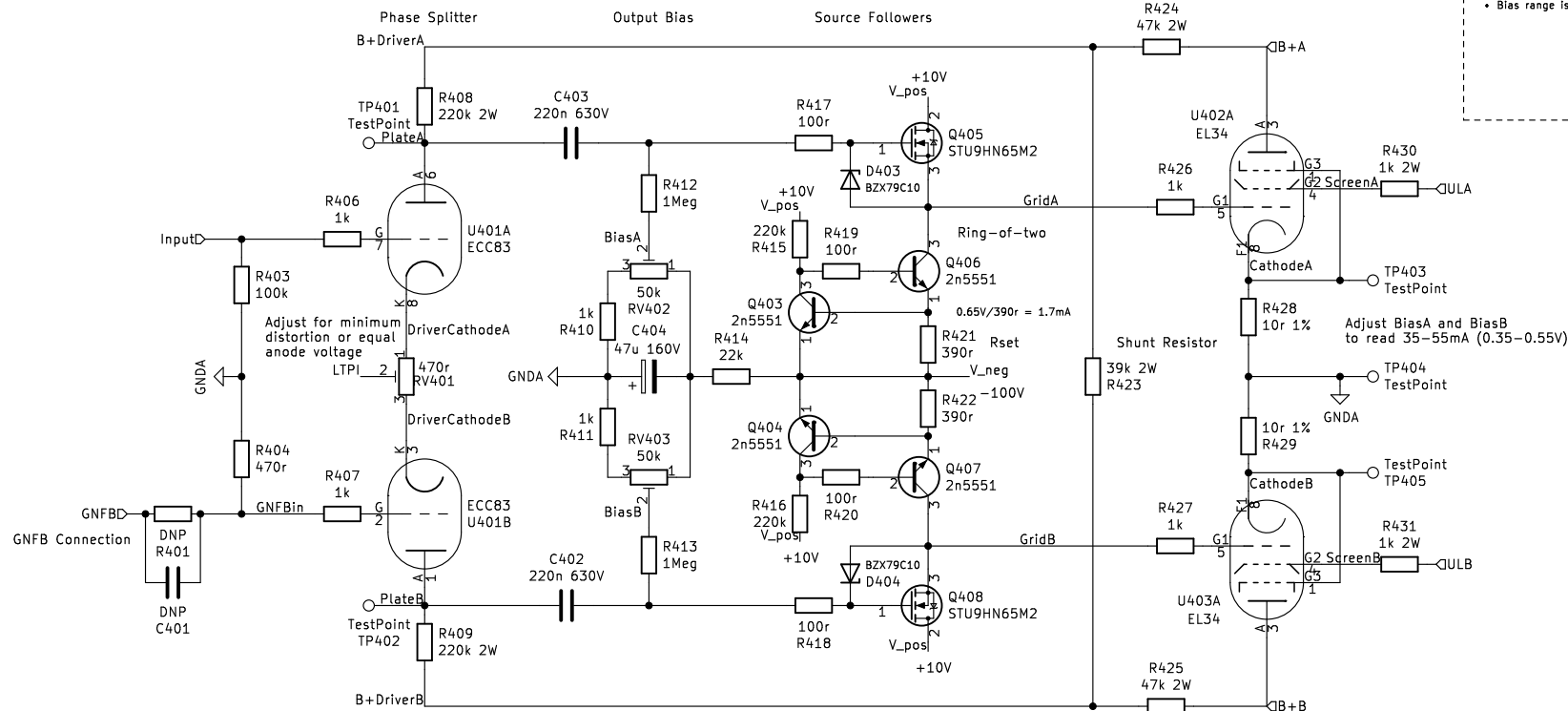
Sheet: /PowerSupply/
 File: PowerSupply.kicad_sch

Title: Baby Huey – "Engineer's Version" – Power Supply

Size: A4 Date: Rev:
 KiCad E.D.A. kicad (6.0.0-0) Id: 3/4

Notes / Questions

- What value for Shunt Resistor? 22k or 33k according to forum. 39k is too large.
- Could increase the source follower current to 2.4mA – use 270r
- However, lower starts to stress the 2N5551 dissipation limit.
- Bias range is -1V to -70V



| | |
|--|---------|
| Sheet: /Right/ | |
| File: Channel.kicad_sch | |
| Title: Baby Huey – "Engineer's Version" – Channel | |
| Size: A4 | Date: |
| KiCad E.D.A. kicad (6.0.0-0) | Rev: |
| | Id: 4/4 |