

Sheet: input

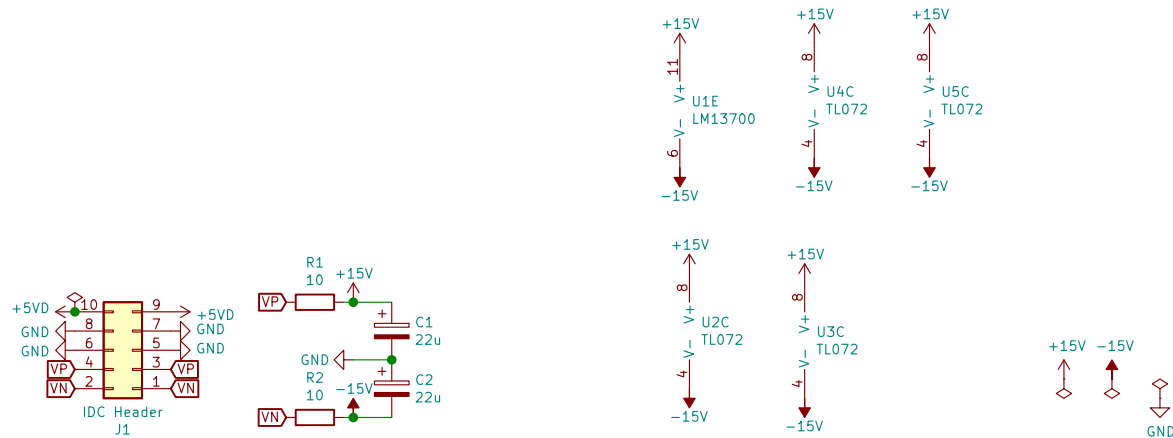
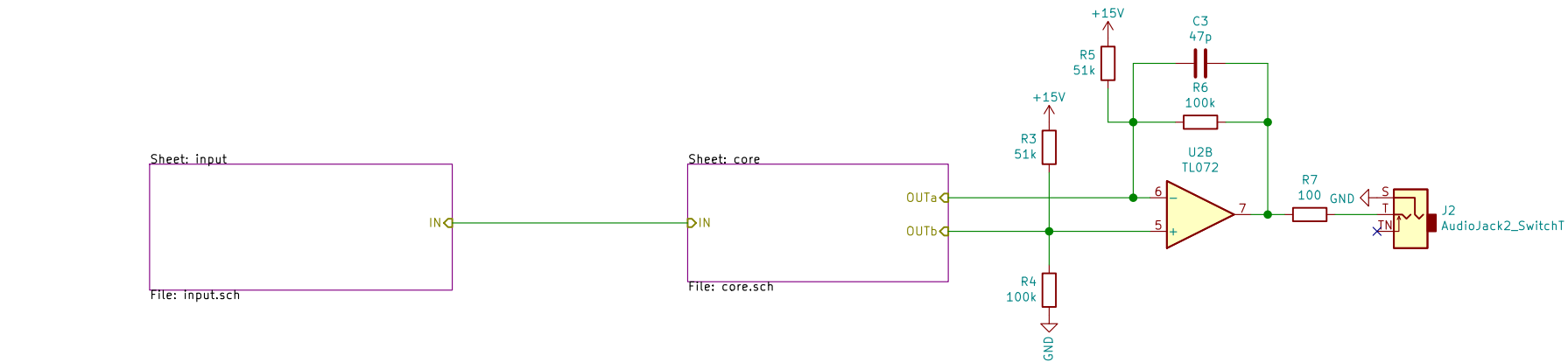
File: input.sch

Sheet: core

File: core.sch

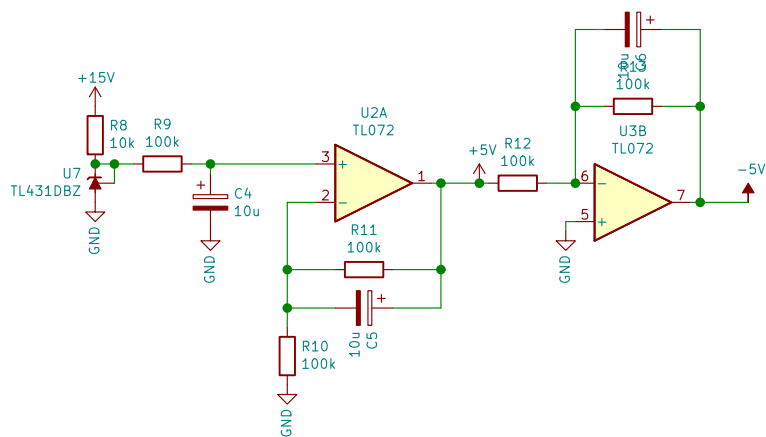
Sheet: Voltage_Reference

File: Voltage_Reference.sch



- H1 MountingHole
- H2 MountingHole
- H3 MountingHole
- H4 MountingHole

License CC BY 4.0 – Attribution 4.0 International	
(description) schema for pcb	
Sheet: / File: main.sch	
Title: (title)	
Size: A4	Date:
KiCad E.D.A. eeschema 5.1.9+dfsg1-1	Rev: R01 Id: 1/4



Sheet: /Voltage_Reference/
File: Voltage_Reference.sch

Title:

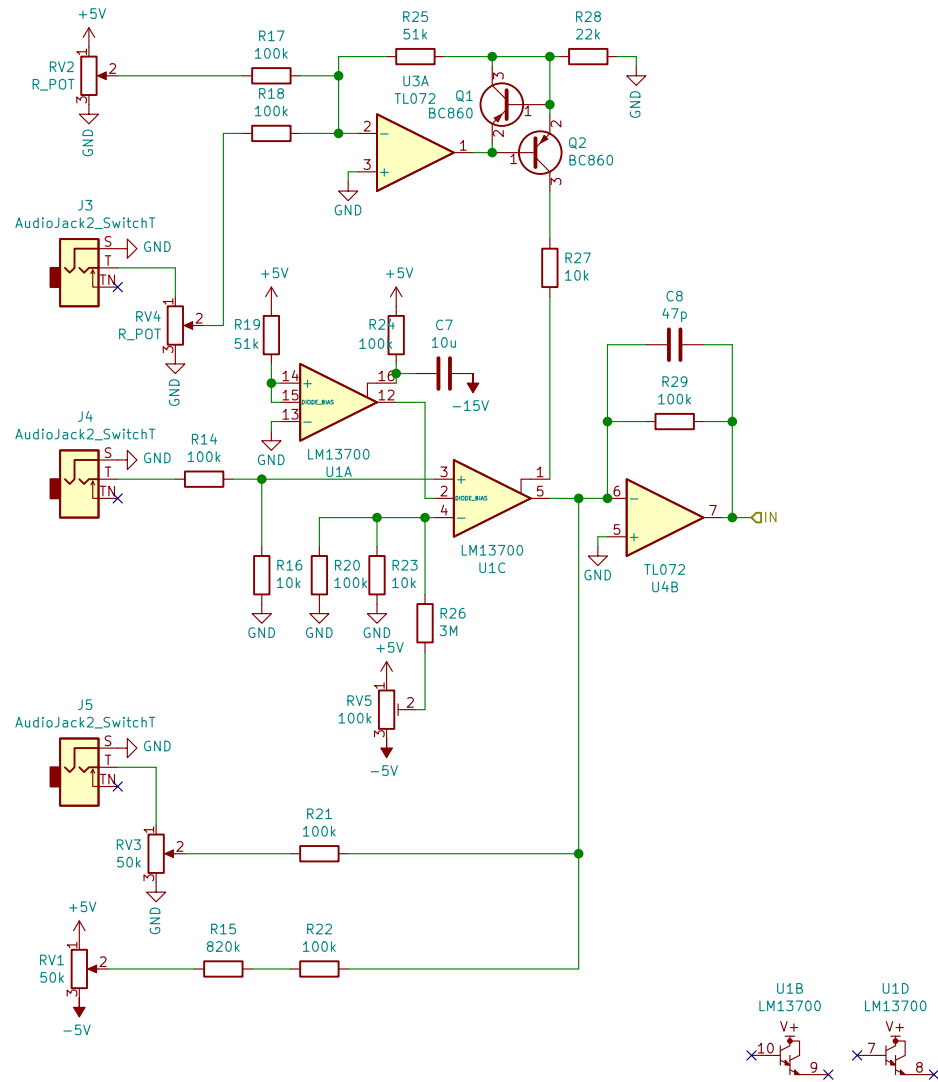
Size: A4

Date:

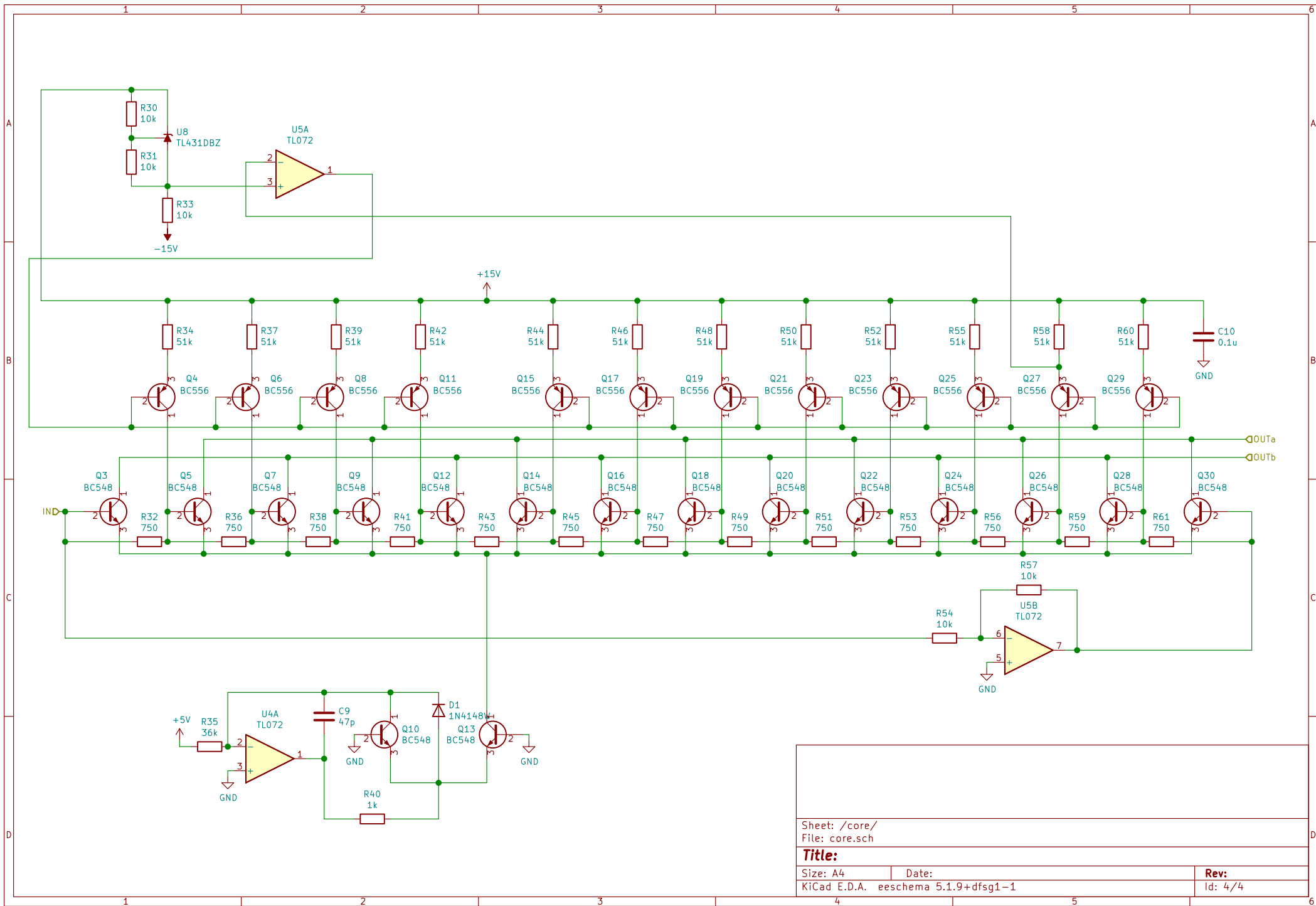
KiCad E.D.A. eeschema 5.1.9+dfsg1-1

Rev:

Id: 2/4



Sheet: /input/ File: input.sch		
Title:		
Size: A4	Date:	Rev:
KiCad E.D.A. eeschema 5.1.9+dfsg1-1		Id: 3/4



Sheet: /core/
File: core.sch

Title:

Size: A4

Date:

KiCad E.D.A. eeschema 5.1.9+dfsg1-1

Rev:

Id: 4/4