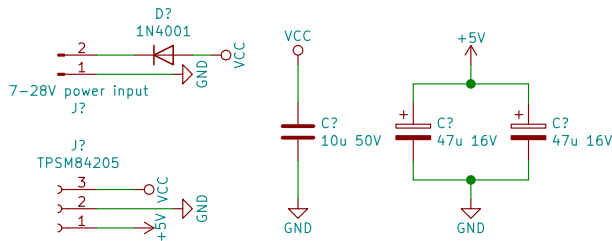
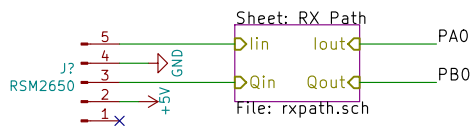


NOTE  
this schematic only shows what approximately was done on the  
hand-soldered prototype board.

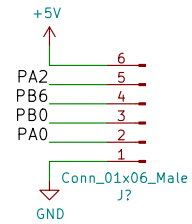
Power supply 7-28V input 5V output



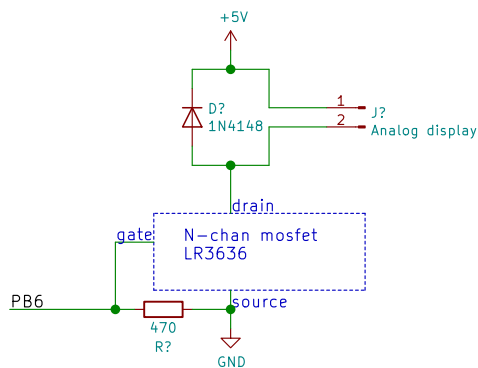
2 channel analog frontend



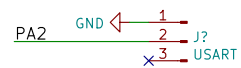
Connections to 1bitsy module



PWM output



Debug header

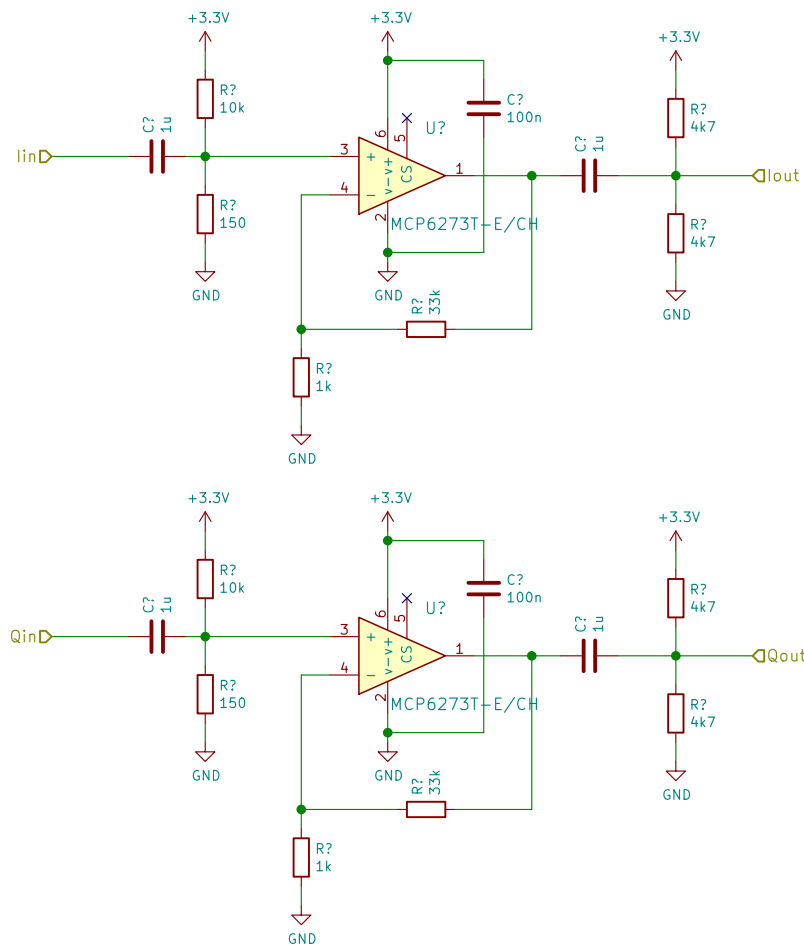


Sheet: /  
File: wurstradar.sch

**Title:**

Size: A4 Date:  
KiCad E.D.A. kicad 5.1.4

**Rev:**  
Id: 1/2



Sheet: /RX Path/  
File: rxpath.sch

**Title:**

Size: A4

Date:

KiCad E.D.A. kicad 5.1.4

**Rev:**

Id: 2/2