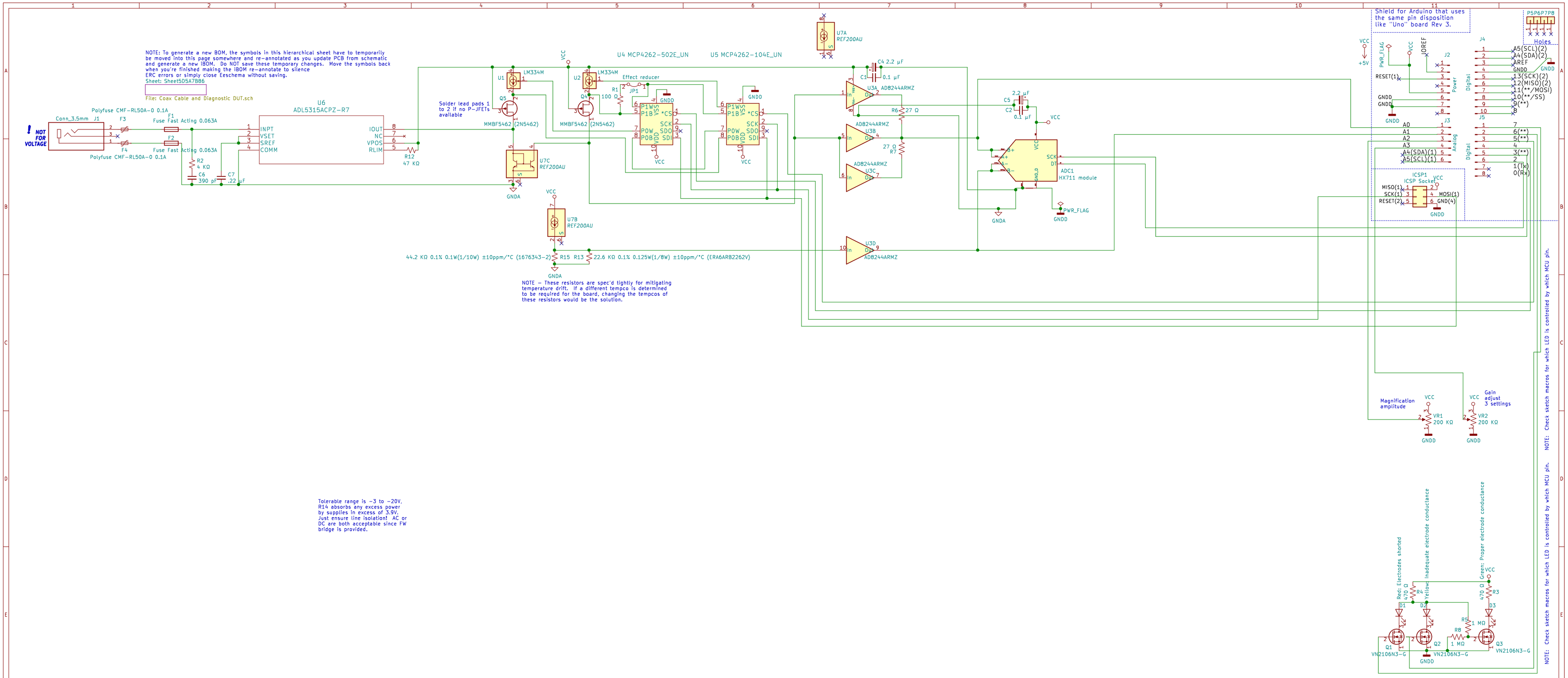


Tolerable range is -3 to -20V. R14 absorbs any excess power by supplies in excess of 3.9V. Just ensure line isolation! AC or DC are both acceptable since FW bridge is provided.

NOTE: Diagnostic jumpers must be used as JP1-2-3 or JP4 with JP5, not all jumpers simultaneously. They are primarily intended for use during the initial board building process to ensure any mistakes are caught when they would most easily be identified and corrected. JP4-5 can bypass the software-controllable analog input levels to allow the manual pots to supply them. JP1-3 is for bypass of LM334s to verify operation of the MCP4262s (which, BTW, cannot achieve high enough resistances to produce working level voltages on the legs without low R2 and DUT resistances). You'll need to insert such resistances manually...on the order of 10 KΩ to 100 KΩ with 27 KΩ suggested. For board versions without these jumpers, JP4 and 5 can be effected by disconnecting LM334 lead 2 and adding a temporary wire from the wiper of a manual pot to lead 3 of that LM334. JP1 and 2 can be effected by disconnecting LM334 lead 2 and adding a temporary wire between LM334 pads 1 and 2 (without lead 2 in circuit). In either case, LM334s may be removed completely to avoid JP cutting or to ensure lead 2 (middle lead) disconnect during diags. Lead 1 is always marked as square pad and/or silk-screened dot or mark on board.

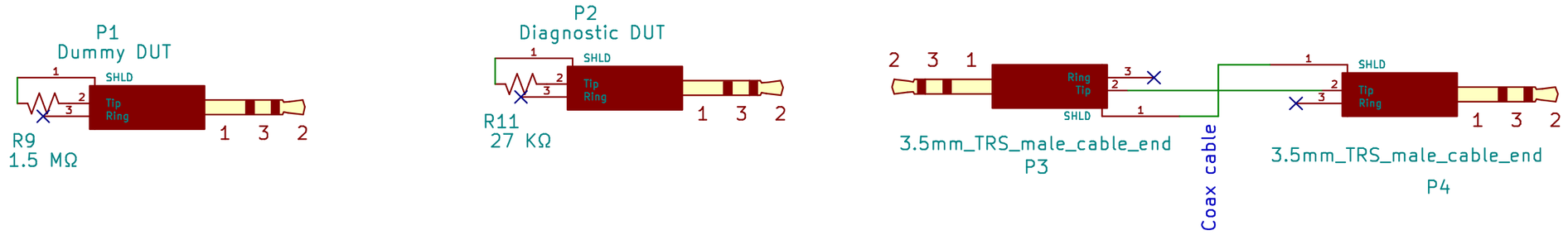
1 2

GWAAMC Training Aid Main Board for UNO with MCP4262 HX711 LM334 REF200 MMBF5462 (2N5462)

(C) Copyright Kenneth L. Anderson 2018-2019

Sheet: /		
File: uno shield.sch		
Title:		
Size: A2	Date: 05-09-2019	**Rev:**
KiCad E.D.A. kicad 5.1.2-f72e74a84ubuntu16.04.1		Id: 1/2

NOTE: These symbols are required to generate an accurate BOM. To use them in that way, copy them and paste them anywhere into the main page.
They do not have to show within any sheet boundary.
Generate the new BOM and delete the symbol copies from the main page, and main page only, when finished, always leaving the original symbols in this sheet.
If the copies are left in the main page when later updating the PCB, they will interfere with the generation of the PCB Gerber files because these have no residencies on the PCB.



Sheet: /Sheet5D5A7B86/
File: Coax Cable and Diagnostic DUT.sch

Title:

Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad 5.1.2-f72e74a84ubuntu16.04.1		Id: 2/2