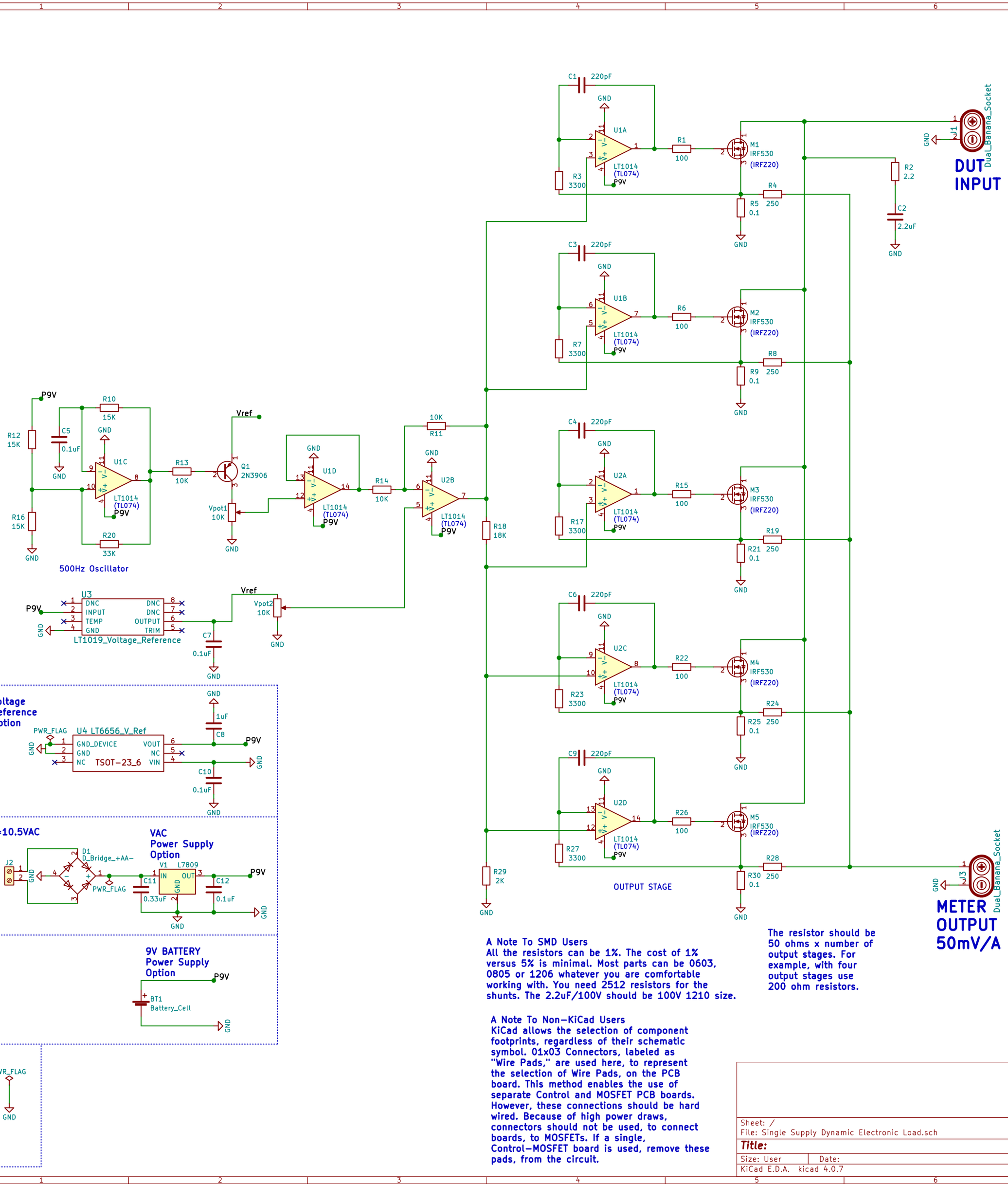
C1 220pF
U1A
LT1014 (TL074)
P9V
R1 100
M1 IRF530 (IRFZ20)
R2 2.2
C2 2.2uF
R3 3300
R4
R5 0.1
J1 Dual_Banana_Socket
DUT INPUT
C3 220pF
U1B
R6 100
M2 IRF530 (IRFZ20)
R7 3300
R8
R9 0.1
C4 220pF
U2A
R15 100
M3 IRF530 (IRFZ20)
R17 3300
R19
R21 0.1
C6 220pF
U2C
R22 100
M4 IRF530 (IRFZ20)
R23 3300
R24
R25 0.1
C9 220pF
U2D
R26 100
M5 IRF530 (IRFZ20)
R27 3300
R28
R30 0.1
R29 2K
OUTPUT STAGE
J3 Dual_Banana_Socket
METER OUTPUT 50mV/A
The resistor should be 50 ohms x number of output stages. For example, with four output stages use 200 ohm resistors.
R10 15K
R12 15K
C5 0.1uF
U1C
R13 10K
Q1 2N3906
R16 15K
R20 33K
500Hz Oscillator
Vref
Vpot1 10K
U1D
R14 10K
R11 10K
U2B
R18 18K
U3
LT1019_Voltage_Reference
Vpot2 10K
C7 0.1uF
Voltage Reference Option
PWR_FLAG
U4 LT6656_V_Ref
GND_DEVICE
TSOT-23_6
C8 1uF
C10 0.1uF
>=10.5VAC
VAC Power Supply Option
J2 Screw_Terminal_01x02
D1 D_Bridge_+AA-
V1 L7809
C11 0.33uF
C12 0.1uF
9V BATTERY Power Supply Option
BT1 Battery_Cell
A Note To SMD Users
All the resistors can be 1%. The cost of 1% versus 5% is minimal. Most parts can be 0603, 0805 or 1206 whatever you are comfortable working with. You need 2512 resistors for the shunts. The 2.2uF/100V should be 100V 1210 size.
A Note To Non-KiCad Users
KiCad allows the selection of component footprints, regardless of their schematic symbol. 01x03 Connectors, labeled as "Wire Pads," are used here, to represent the selection of Wire Pads, on the PCB board. This method enables the use of separate Control and MOSFET PCB boards. However, these connections should be hard wired. Because of high power draws, connectors should not be used, to connect boards, to MOSFETs. If a single, Control-MOSFET board is used, remove these pads, from the circuit.
Sheet: /
File: Single Supply Dynamic Electronic Load.sch
Title:
Size: User
Date:
Rev:
KiCad E.D.A. kicad 4.0.7
Id: 1/1