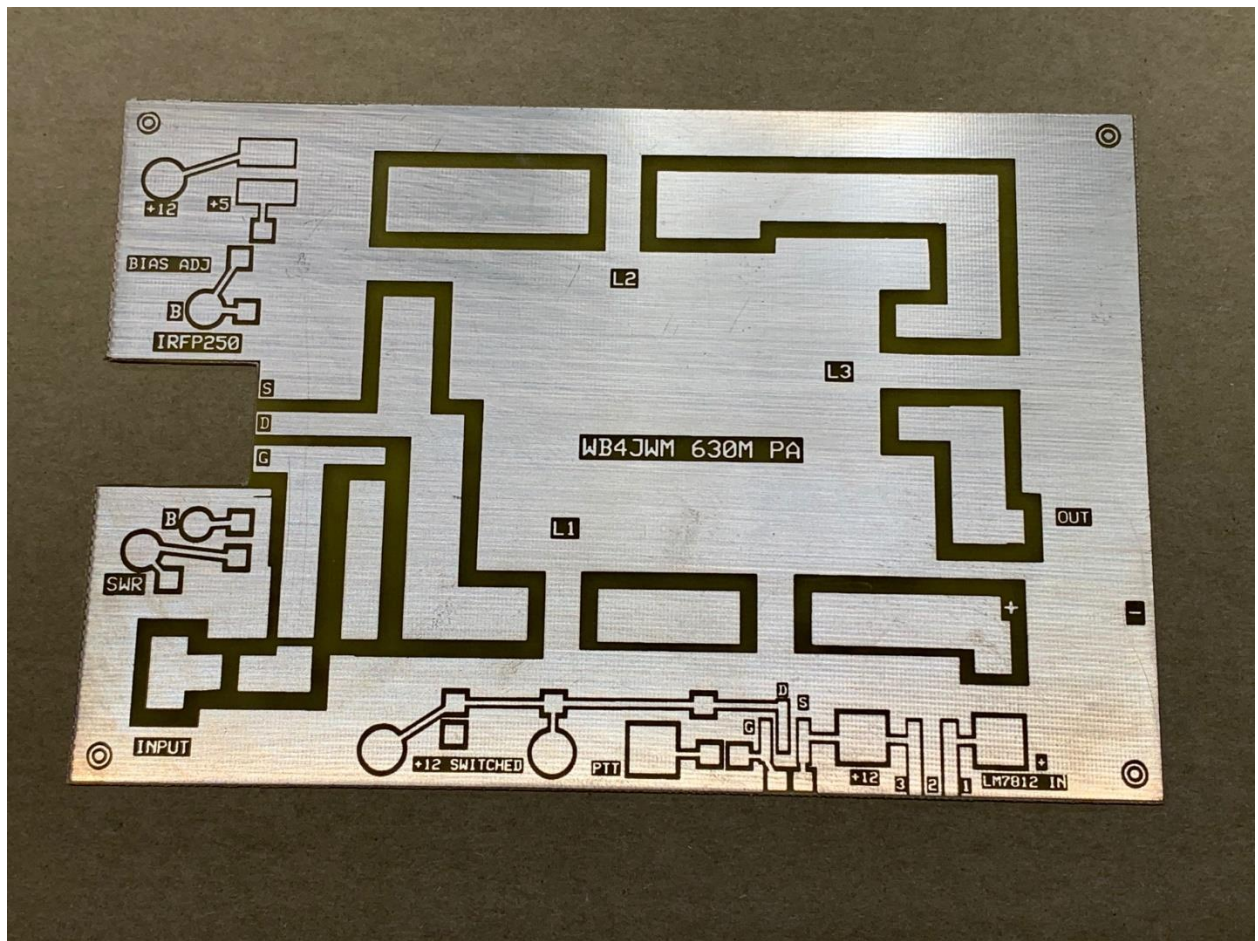


CIRCUIT BOARD FOR 630M AMPS

As of 12/21/2024

My boards are no longer available (out of stock), and I don't plan on reordering. There is a better board now available at <https://www.multus-sdr.com/> Check out there 630m 200 watt Kit, and a PCB stand along board is available. The amplifier and PCB was built using my circuit so it will work with my designs on this page, and my other amplifiers on my web site. <https://wb4jwm.com> .

I have designed a circuit board that works for my amplifier designs, and many others using a single MOSFET switch. All components will mount on the foil side as surface mount. The pads are designed to utilize both SMD, conventional or a combination of both. The board is 4" X 6" with a cutout for the MOSFET. The board is designed to lay flat on the chassis or heatsink. I have them available for \$40.00 shipped to USA. These boards are FR 4 glass epoxy, 1/16 1 oz. copper, tin plated, and sent with a ceramic insulator for the MOSFET. Ceramic is the best for heat transfer. The boards are produced by a USA PCB company, and I'm only recovering my cost for the boards (shipping included). USA only, no international shipping. If interested email me at my QRZ.com address.

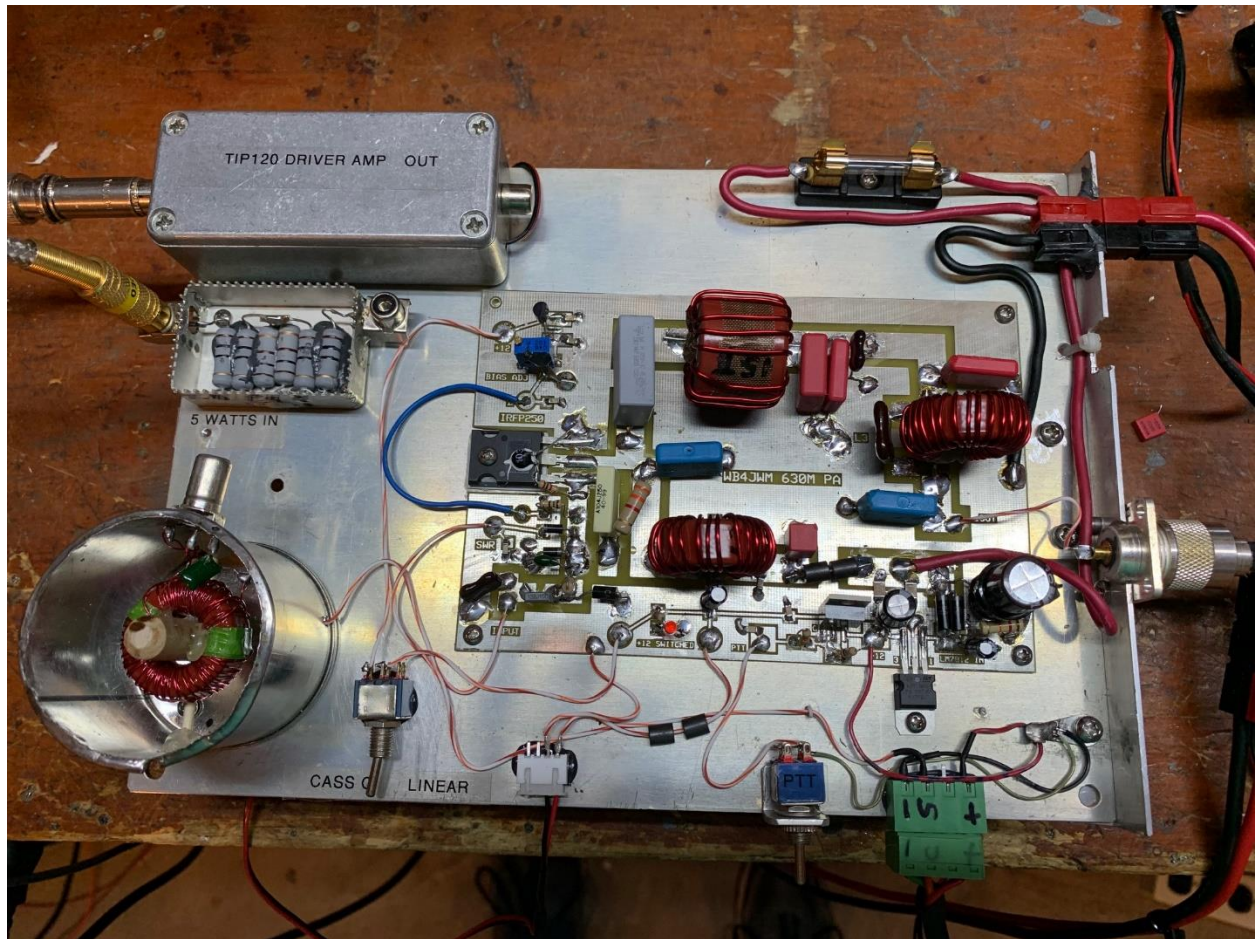


The schematic shows two main sections. The top section is a voltage divider and switching circuit. It starts with an "Input pad on circuit board" providing "+24 to +40 VDC". This input goes through a "Low voltage power switch" (represented by two diodes) and two 5V, 5Watt Zeners. The resulting "+20 to +35 VDC input" passes through a 12k resistor, a 10µF capacitor, and a .33µF capacitor to pin 1 of an LM7812 regulator. Pin 2 is grounded, and pin 3 provides a "+12V" output. This +12V line has a .33µF capacitor at its source and another .33µF capacitor before it enters the gate (G) of an IRF9540 P-Channel MOSFET. A 3k resistor connects the PTT button to the gate. The MOSFET's source (S) is grounded, and its drain (D) is connected to a "Switched +12 VDC" supply through a 3k resistor. An LED is connected between the drain and ground. The bottom section is a linear ramp generator. It takes a "Switched +12 VDC" input to pin 1 of an LM7805 or L78L05 regulator. Pin 2 is grounded, and pin 3 provides a regulated output. This output passes through a 10µF capacitor, a 200Ω resistor, and a .1µF capacitor. It then goes through a "Linear Jumper" (two open circles) and a 150Ω resistor to provide the "Bias to Mosfet Gate".

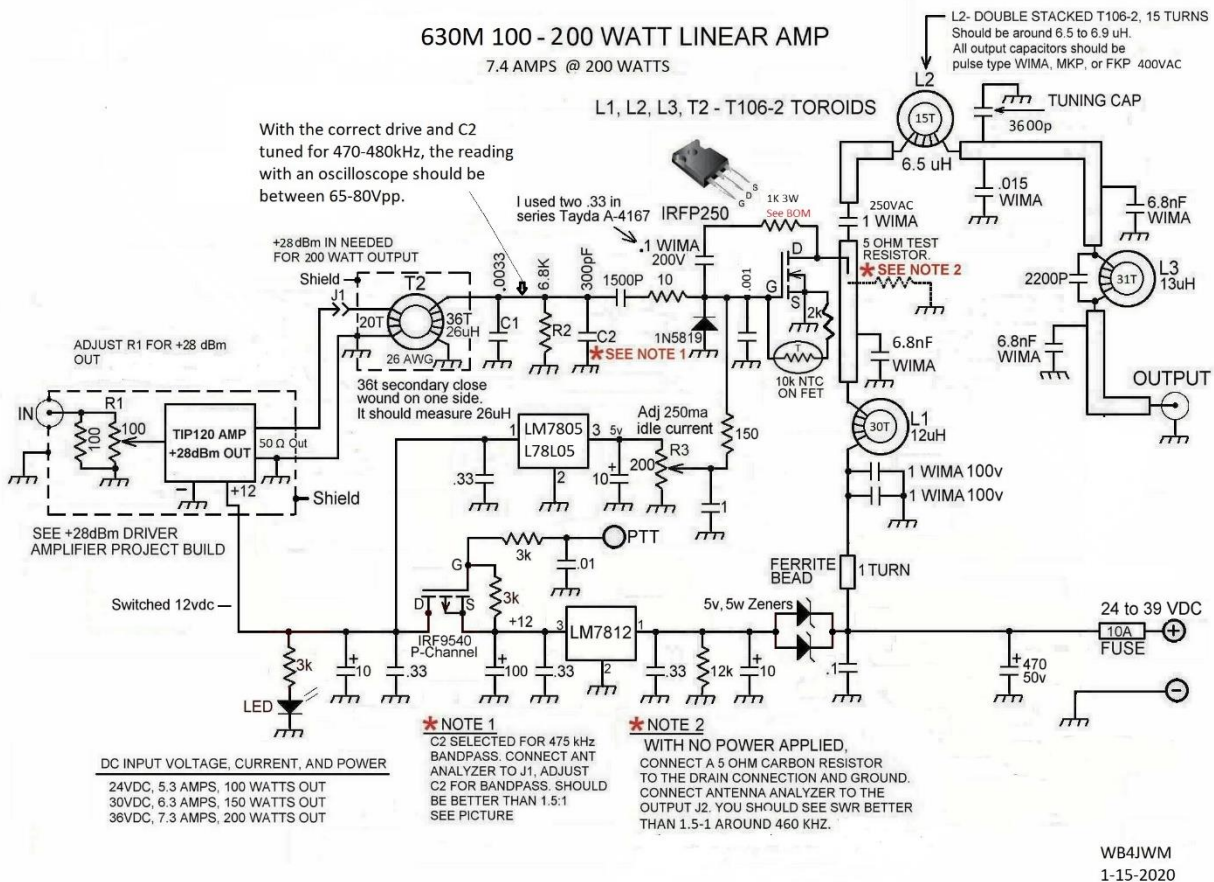
This Circuit Is designed to take care of the switching circuits for the amplifier. I now prefer leaving the Drain of the RF MOSFET hot all the time, eliminating the need for the high current switching FET or relay. This circuit will take care of the necessary auxiliary switching, plus giving a continuous +12 VDC out of the LM7812. If not running the DC input above 35 VDC, a jumper can be used in place of the two 5V 5W Zener.

The LM7812 will require a heatsink, attach to chassis. The IFR9540 and LM7805 do not require a heatsink.

I like using 1206 sized SMD components, but conventional lead components will work on the board as well.



The board is mounted on my bench test setup so every combination is added. Not all components are needed for an amplifier.



LINEAR AMPLIFIER USING THE CIRCUIT CONFIGURATION ON THE PCB. If not building linear just don't add the LM7805 bias circuit and connect the 150 ohm resistor from Gate to ground.

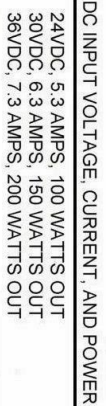
L1 wound with #18 or #16 wire

L2, L3 wound with #16

7.4 AMPS @ 200 WATTS

L1, L2, L3, T2 - T106-2 TOROIDS

With the correct drive and C2 tuned for 470-480kHz, the reading with an oscilloscope should be between 65-80Vpp.



***NOTE 1**
C2 SELECTED FOR 475 KHZ
BANDPASS. CONNECT ANT
ANALYZER TO J1. ADJUST
C2 FOR BANDPASS. SHOULD
BE BETTER THAN 1.5:1
SEE PICTURE

*** NOTE 2**
WITH NO POWER APPLIED,
CONNECT A 5 OHM CARBON RESISTOR
TO THE DRAIN CONNECTION AND GROUND.
CONNECT ANTENNA ANALYZER TO THE
OUTPUT J2. YOU SHOULD SEE SWR BETTER
THAN 1.5:1 AROUND 460 KHZ.