20W Amplifier II Construction Hints – V1.5
The modern day Captain Midnight Secret Decoder Ring
(read em' now or after the parts are soldered)

ERATTA for the V1.5 board silkscreen (will be corrected on the V1.5a board)

- **R9 near Q2 should be labeled R1 (1 ohm resistor)**
- **R10 near Q3 should be labeled R2 (1 ohm resistor)**
- **L3 near the 6V Regulator should be labeled L0 (6uH inductor)**
- **R2 near the 6V Regulator should be labeled R13 (820 ohm resistor)**
- **Board ID should be labeled 20W, not 20dB**

SPLASH WORDS, CREDIT, and RATIONALLE

When it comes to Amplifiers, the Radio Amateur is responsible for making sure his amplifier operates within the guidelines established by the FCC and that it does not cause unwanted interference. This is a Broadband Amplifier and will require some sort of Low Pass Filter (LPF) on the output to attenuate harmonics. Various optional LPFs are discussed in the documentation. You may already have one you intend to use.

Amplifiers have been around a long time and continue to evolve, one design being based on another, etc. I want to give credit to some who "plowed the ground" before me: G6ALU, KE9H, WA2EBY, TF3LJ, K5OOR and above all N5OK who provided all the testing in his extensive lab. The question may come up, "Why ANOTHER Amplifier ?" To that I say, "Looks like fun, why not ?"

My idea was to provide a **good and low cost** Amplifier for the HF bands (160m to 6m) which can be driven by a SoftRock SDR or any other low power driver. Low power being defined as less than 1W. Some of the new SDR units supply less than 50mw and really need a little "boost". This Amp uses well received RD16HHF1 RF MOSFETs which are each rated at 16W output for 12.5VDC at 30+MHz. These parts are quite robust and much better than the IRF510 Switching Power Supply MOSFETs used earlier, largely because of their low cost and availability. The other advantage of the RD16HHF1 is that the mounting tab is attached to the "Drain" pin which is at ground potential …so no mica insulator is required. The Amplifier has a Signal Operated Switch (SOX) for CW

QSK. I didn't really put it there for SSB …..but who knows (still better to use PTT). Did I mention **good and low cost** ?

GENERAL

The design and components which have been selected require that the operational criteria of 1) an Input of 1W max –AND-- 2) an Output of 20W max ………whichever comes first, NOT be exceeded. To allow some adjustment, the input to the Amplifier has places for a Pi network Input Attenuator of your selection (Rx, Ry, Ry) ….3dB, 6dB, etc. Parts for a 3dB (50% power reduction) attenuator are provided in the kit. The purpose of the attenuator is to reduce input power and to provide proper 50ohm matching for the driver. This design has been extensively tested and optimized to provide good matching to 50ohms **without** the attenuator. If you elect to not use an attenuator, replace Rx with a jumper and do not install either Ry.

The Amp is designed for 13.6VDC, 50 ohm input and output, both RD16HHF1 MOSFETs with bias levels set to provide 300-500mA quiescent current on each, and it has a 3dB attenuator on the input. The result for 500mW drive (250mW at the MOSFETs) is a linear power profile (to within 0.5dBm) from 2MHz to 60MHz with 17.4W output. .....that's flat. You can push it to 20W but it's not linear there.

The major difference of the "II" vs the earlier "I" is, it uses a single homebuilt Output Transformer T3 vs the two used on many other earlier designs. This results in more efficient operation, lower IR loss, and better matching to the two RD16HHF1 MOSFETs. As a result the board is approximately ½" longer. The older 2x BN43-202 designs also ran quite hot at 20W (as do any of the designs using a single Output Transformer of #43 material like the BN43-3312). Experience has shown that the Output Transformer is one of the most critical elements of Amplifier design, especially relative to matching the MOSFET impedances. I looked around for pre-assembled units and found some of the sample parts I bought to be severely lacking in quality and highly dependent on who manufactured them (burn marks and cracks on some). Some of the permeabilities varied 25% in a single lot…..that's bad. By far, the best solution was to obtain a qty of very consistent #61 material sleeve type cable ferrites from Fair-Rite. These are mounted on a homebrew "single turn" assembly made of two 3/16" copper tubes and a piece of 1/32" copper plate (the plate is also much better than 1 oz. copper PC board). See the pictures on my kit website for more details.

There are several ways of providing DC power to the MOSFETs, (#1) you can supply power through a center tap of the 1 turn primary of the Output Transformer T3, or (#2) you can provide power through a bifilar wound "phasing choke" T2 and either DC isolate T3 by (a) adding capacitors in series with T3 (will have to handle current and have low ESR), or (b) add capacitors at the center of the 1 turn primary, or (3) tie the T3 primary center tap to DC in addition to using the bifilar wound "phasing choke" T2. I decided to go with #2.

T1….increased the BN43-2402 core Input Transformer to a BN43-302 core. Never saw any indication of saturation on the BN43-2402 but went to the larger size anyway…..just in case. The RF chokes were reduced in impedance to reduce IR loss, as well as increasing power wiring size to further reduce IR loss by using a large Ferrite bead on wire.

After assembly **with the MOSFETs screwed to the heatsink** and the Bias switch closed (Class A/B), turn the bias potentiometers fully counterclockwise, connect a 50ohm load to the input connector and output connector, apply 12V to PTT which will allow the relay to be picked and turns on the 6V bias regulator. Now measure the current. There should be a small current draw. Turn one bias pot clockwise until the current increases 300-500mA, then turn the other bias pot until the current increases an additional 300-500mA. Now the bias levels are balanced and set for Class A/B operation for your Amplifier.

The design also provides Class C operation (CW only). This allows lower power consumption when you really don't need linear operation. A single jumper selectable 820 ohm resistor reduces the Gate Bias current on the MOSFETs to around 25-35mA each. You can also use the potentiometers to adjust this if you prefer.

Many MOSFET Amplifier circuits do not use a feedback circuit because this design does not need one. This circuit has the pin in hole locations to add a feedback resistor and capacitor to both MOSFETs. The resistor will probable range from 270 to 910 ohms and the capacitor around 10nF. Be sure and use a 1KV rated capacitor in the FB loop….this is needed to protect the MOSFETs. A 25 or 50V part is asking for trouble.

The compensation capacitor is made up of two 249pF Mica capacitors in series. This was done for two reasons, the distributed capacitance and lead inductance "seems" to work better than a single capacitor, Mica capacitors are very stable across a broad range of frequency and temperature ………and I happen to now have a lot of 249pF Mica capacitors. The combined 125pF provides a better response at 6m. Having the two capacitors in series allow attaching the common point to a different return point in the future if needed.

ASSEMBLY HINTS (IMPORTANT)

All the SMT components should be soldered in place **first** and the solder joints carefully inspected. **95% of all problems I've seen are defective solder joints**. Because the board does not use thermals, more heat is required for a good solder joint..

Here are some of the areas where you might have questions.

Power is connected to the board using heavy duty stranded wire so there is minimal voltage drop. Please observe polarity.

R12 is a 3.3K 1206 size SMT part which is larger than the other 0805 size SMT parts. This is done to bridge the two solder pads on the board for R12

Many of the 0805 size SMT parts have one end grounded (usually indicated by a hole in the pad to ground it to the ground plane side of the board. If you first solder the non-grounded end of the component and then the grounded end, assembly will be much easier.

Q1 and Q4 mount in the footprint with the flat side of the transistor as shown..

T1 ……is made with a BN43-302 core and 2 turns of bifilar #26 wire inserted into a 3" piece of 1/16" shrink tubing. Be sure to unkink and untwist the bifilar wire before using. Use a soldering iron to shrink the tubing before winding the 2 turns. One hairpin pass of wire through both holes is considered "1 turn". Any wire repositioning can be done with a wooden toothpick. Experimentation has shown bifilar wire is much better at the higher frequencies than twisting two wires together. Trim the shrink tube and scrape the insulation off with a single edge razor blade or X-Acto knife and tin with solder. Mounting T1 on the board is easy and the wires can go in either hole.

T1 NOTE: ….you can also experiment around with T1. Right now it has 1:1 turns (1:1 impedance) with a bifilar winding. This means for a 50 ohm input match, each load resistor (R5, R6) has to be about 27 ohms. The input impedance of the MOSFET is high, >> 27 ohms. You can make the winding trifilar and put the two output windings in series. That means it has 1:2 turns (1:4 impedance) and for a 50 ohm input match, each load resistor (R5, R6) has to be about 100 ohms. This is actually a "somewhat better" match to the MOSFET characteristics.

Be sure and install the Mode jumper to short out the 820 ohm resistor for Class A/B operation…..or you can just hardwire it and leave off R13 (820 ohm) if that's the only mode you are going to use.

There are 3 …very small… 1N4148/1N914A diodes in the SOX area. There is a Cathode mark on the diodes (need magnification to see) and a silkscreen Cathode mark on the board…..it is quite small. I will supply replacements for those that get "pinged" into the shag carpet for a SASE.

L0 and L6 use a BN43-2402 core with 2 turns of #26 Teflon coated wire (white) and act as a 6uH RF choke.

L1, L2, and L3 are large low IR loss Ferrite Beads on wire. One end has a small amount of epoxy showing. If you mount that end "up" the Ferrite Bead will mount flush to the board. If you don't, the Ferrite Bead will "rock" …..which is OK electrically.

The PTT input header is active high 12V like you would see from a SoftRock PTT.

The Ferrite Output Transformer T3 is a new design for 160m to 6m and only runs warm at 20W which is a really good indication. It will require some assembly and I ask you to refer to the kit website for pictures which should help. The object is to insert the 2 copper tubes into the copper plate to form a "U" shape. A small round file is handy for elongating the holes if they are too close together. The goal is to insert the copper tubes through the copper plate so the loose ends of the "U" are even and protrude approximately 1/16" past the Ferrite sleeves. NOW Solder the tubes to the outside of the copper plate, and file off the excess material. You can further shape the copper plate with a file to round the edges, etc . Strip the Teflon insulation off two 2" pieces of #20 stranded wire, tin it, and wrap it once around each tube. One end will be used to attach T3 to the board and the other end is used to attach the two 249pF Mica capacitors in series. Please see the pictures….winding the secondary may be easier if the loop is positioned to the inside vs to the outside as shown in the picture. I'll do it the other way next time. The T3 secondary is 2 turns of #20 stranded Teflon coated wire. The ends of the Teflon wire are soldered directly to the board. ….MAKE SURE THE T3 COPPER PLATE AND THE T3 SECONDARY WIRES ARE NOT TOUCHING THE BOARD VIAS.

T2 is not really a transformer, it's a "phasing choke" made of a BN43-302 core with a single hairpin turn of two pieces of #20 enameled wire which were inserted into a 1-1/8 piece of 3/32" shrink tubing to make it bifilar. This provides two phased RF chokes which supply the DC voltage to the MOSFETs. Ohm out the leads to make sure you connect them in series as shown on the schematic and trim any excess shrink tubing. The common point where they are tied together becomes the point to attach to power and the other two ends attach to the MOSFETs. The picture on the kit website should help explain this. The spacing between T2 and T3 is tight….MAKE SURE THE T3 COPPER PLATE AND THE T3 SECONDARY WIRES ARE NOT TOUCHING THE OBOARD VIAS.

Once you have decided on the heatsink you will use, bend the leads on the MOSFETs to match the mounting hole in the board and align the leads with their respective holes in the board. No not apply heat conducting paste until later. After the MOSFETs are aligned and mounted to the heatsink, solder the MOSFET pins. Be sure the center ground pin on the MOSFET is well soldered on both the top and bottom of the board. As a last step before final assembly and testing apply a thin coat of the white heat conducting paste (you provide) to the back of the MOSFET surface.

I have decided that I'll provide a drilled and tapped heatsink as an **option** because not all will have access to a used heatsink or the tools to drill and tap the holes. I used a drill press and hand tapped the holes …..but I am, by no definition, a "machinist". The heatsinks are drilled for 4-40 board mounting holes and 6-32 holes for the MOSFETs. You still have to provide the heat transfer paste, mounting hardware and nylon spacers. It also means the kit has to be shipped in a USPS Priority Mail box to accommodate the heatsink.

For those wanting to use their own heatsink……..the first step is to drill and tap the holes in the heatsink. Using the Amplifier board as a template, locate the centers for the holes and mark them (preferably between the heatsink fins). The board mounting holes should be drilled and tapped for 4-40 machine screws and the 2 MOSFET mounting holes for 6-32 machine screws. It's a little tricky to miss the fins on the other side with the drill bit….so measure twice and drill once. The board coax tie points are identified and the board also has SMA coax connector footprints.

The Input/Output attachment points for the coax have one small hole for the center conductor and one large hole for the shield. The coax should be 50ohm (RG174 or other small coax) and attached perpendicular to the board. Twist the braid shield and insert it through the large hole or wrap a wire around it and solder to the braid. You can use nearby "ground holes" to fabricate a 2 turn bare wire strain relief for the coax. RG-174 is spec'd good for 40W.

C2 affects the SOX hold delay. Use whichever value works best for you ….1uF is provided ……maybe 3.3uF works best for you, maybe up to 10uF ;o) If you want to disable the Signal Operated Switch (SOX) circuitry and use PTT only, remove C1 and D2.