20W Amplifier II Construction Hints – V1.4
The modern day Captain Midnight Secret Decoder Ring
(read em' now or after the parts are soldered)

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 ¼" 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.

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

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.

The 2 header pin location behind the power connector ……is not used, jumper them with a piece of wire.

R12 has two pin in hole pads and may be a 3.3K axial lead resistor or a 3.3K 1206 size SMT part which will bridge the two pads.

Q6…is a 2N3906 which mounts in the triangular footprint with the flat side of the transistor away from the power connector.

T1 ……is made with a BN43-302 core and 2 turns of bifilar #26 wire. One hairpin pass of wire through both holes is considered "1 turn". Be sure to unkink and untwist the bifilar wire. Any wire repositioning can be done with a wooden toothpick (prevents scratching the enamel wire coating). Experimentation has shown bifilar wire is much better at the higher frequencies than twisting two wires together. 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.

Be sure and install the Mode jumper to short out the 820 ohm resistor for Class A/B operation.

There are 3 very small 1N4148/1N914A diodes in the SOX area. There is an Anode mark on the diodes (need magnification to see) and a silkscreen Anode mark on the board…..it is quite small.

L1, L2, L3 and L6 all use the BN43-2402 core and 2 turns of #24 Teflon coated wire (white) and act as an RF choke.

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

The Ferrite Output Transformer T3 is a great design for 160m to 6m and 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. The tube ends will require some filing and clean-up with an X-acto knife, file, or sandpaper. A small round file is also handy. You can shape the copper plate with a file, round the edges, etc. The goal is to insert the copper tubes through the copper plate so the ends of the "U" are even, solder the tubes to the outside of the copper plate, and file off the excess material. Please see the pictures. When you install the ferrites, you should have at least 1/16" protruding. Strip the Teflon insulation off two pieces of #20 stranded wire, tin it, and wrap it once around each tube, and solder as shown in the picture. One end is used to attach T3 to the board and the other end is used to attach the two 249pF Mica capacitors in series. The T3 secondary is 2 turns of #20 stranded Teflon coated wire. The ends of the Teflon wire are soldered directly to the board.

T2 (on the bottom of the board) is not really a transformer, it's a "phasing choke" made of two stacked BN43-2402 cores with a single hairpin loop of two pieces of #20 enameled wire which were twisted together in bifilar fashion. This provides two 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. The common point where they are tied together becomes the point to attach to power and the other two ends attach to the two pieces of #20 stranded wire you soldered to the board and are attached to the T3 Primary. The picture on the kit website should help explain this. T2 is mounted on the bottom of the board and you need to make sure the solder joints do not come in contact with the heatsink. Since this is a choke, DC power handling should not be a problem and it is high impedance to AC waveforms.

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 both 8-32 and 6-32 holes for the MOSFETs (just turn the heatsink 180 degrees). Use the 6-32 (smaller) holes for these RD16HHF1 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 still wanting to use their own heatsink……..the first step is to drill and tap 6 holes in the heatsink. Using the Amplifier board as a template, locate the centers for the 6 holes and mark them (preferably between the heatsink fins). The 4 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 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.