Down East Microwave Inc. 954 Route 519, Frenchtown NJ 08825
Phone: 908-996-3584 (Voice) 908-996-3702 (Fax) http://www.downeastmicrowave.com

DEM Part Number 10368-144 PCB, BK and K
Low Power 10 GHz. Transverter PCB and board Kit

Specifications

Frequency range:	10368 MHz. = 144 MHz.
Noise Figure and Gain:	<4 dB NF, > 20 dB Gain obtainable with Complete Kit
Power Output obtainable:	>5 mW, all spurs -40dBC
TXIF Power Input to mixer:	1 mW Maximum, 10 watts maximum with Complete Kit
DC Power requirements:	Regulated 9.0 VDC min. @ 400 ma. 11-17 VDC with Complete Kit

***General Information*:**

Down East Microwave's Low power 10 GHz. transverter design is available as a circuit board, the DEM 10368-144PCB, and as a circuit board kit, the DEM10368-144BK. This PCB was a joint effort design by W1GHZ and Down East Microwave Inc. This circuit board and circuit board kit (with associated assemblies) are the exact components used in our low power DEM 10368-144LP and complete kit design, the DEM10368-144CK.

The 10368-144PCB is supplied with a schematic and component placement diagram. The PCB compliments W1GHZ's paper in the 1999 proceedings of the Eastern VHF/UHF conference and the 1999 Microwave Update proceedings to produce a 10 GHz. transverter. The circuit board alone may be assembled and used in many different configurations. It is perfect for the experimenter or anyone that has successfully assembled microwave circuits in the past. The PCB is made of 0.015" thick Teflon-glass material with a epsilon of 2.20. It has plated ½ oz. copper with plated through ground Vias, correct size MMIC mounting holes and will only require a general understanding of the circuit design and good construction practices to produce a working transverter. **It is not recommended for anyone not familiar with the techniques required for mechanical and electrical assembly of microwave circuits to purchase the 10368-144PCB!** The PCB requires mechanical support to ensure reliability against cracked components and damaged ground vias. Therefore, Down East Microwave **will not guaranty the performance** of our circuit board with your assembly configuration because results will vary depending on your components and assembly techniques. **We will not repair any transverters built from the 10368-144PCB only!**

For a higher probability of success, and 100% support of Down East Microwave Inc., we recommend the 10368-144BK. The BK includes the PCB, all components, pipe caps, connectors, required hardware, and a CNC machined circuit board mounting plate. This plate is the key to the success and reliability of the transverter. The plate is used as a fixture to attach the pipe cap filters to the circuit board. It aligns the pipe caps in the correct position and then prevents scorching of the PCB while sweating the pipe caps to the PCB. After assembly of the pipe caps, the mounting plate provides support to the circuit board to prevent cracking of components, and damaging ground vias. After assembly, the plate provides easy attachment of standard SMA connectors used for tune-up and final use. There are 2 main requirements for the completion of the kit. You will need to "Sweat" the pipe caps in place on the circuit board. This requires a propane torch. **A standard soldering iron will not work!** Second, you will need a way to detect RF power at 3cm frequencies to aid in tune-up. This can be done with either a detector diode, a microwave milliwatt power meter or a spectrum analyzer. **The filters need to be tuned after assembly!** There are 9 of them and the unit will not work with out alignment of the filters.

***Circuit Description*:**

The 10368-144 circuit board is 0.015" thick Teflon-Glass material. The material thickness is an important factor of this circuit board and was chosen for two reasons. If it were thicker, the radiation losses would be much grater and would produce a transverter that was more prone to amplifier instabilities cause by longer ground vias. The MMIC amplifiers are the basis of this design. Having added inductance on their ground leads would decrease the gain and add unstable operating conditions. The MMIC amplifiers used are already operating beyond their design frequency, therefore nothing less than optimum conditions should exist. If the circuit board was thinner, The MMIC amplifiers would operate better, but a board of this size (5.25" x 3.75" overall) would become extremely difficult to handle. It could bend, crease or tear easier thus destroying the important ground vias. This in turn decreases processing yields and costs would increase. Another factor is the width of a 50 ohm line would now be smaller than the standard pin size of a SMA connector. This would require a more critical alignment of all test points and RF connections. We also did not desire to have a line width smaller that the lead of MMIC (0.025"). With 0.015"board thickness, a 50 ohm line is 0.044" wide. Near perfect for our application and easily processed.

For a theory of operation discussion, we will use only frequencies pertaining to the standard 10368 MHz. operating frequency with a 144 MHz. IF. The circuit board design is divided into 3 basic sections: (1) The 3GHz. multiplier, (2) the 10 GHz. multiplier, and (3) the 10 GHz. RF section. The 3 GHz. multiplier circuit, which is a 3 X multiplier, was designed for a base LO frequency of 1136 MHz. at a level of +3dBm. This is available from our DEM MICRO LO or you may use your own source. A test point located on the output of 3GHz. multiplier circuit, can have a SMA connector can be attached to it. The two filters in this section are adjusted to produce a 3408 signal at a +5 dBm level. The filters will tune from 3 - 6 GHz. After adjustment, the SMA connector is removed and the signal is routed into the 10 GHz multiplier section. This is also 3X multiplier. The signal may be verified at the second test point (SMA connector) for a +7dBm level at 10224 MHz. This is the LO injection signal that enters the mixer in the third section. The 144 MHz IF frequency is injected or detected in this dual mixer combination. TX and RX functions are controlled by biasing the TX or RX stages independently. Both TX and RX have their own filtering and gain stages. The TX chain filters are adjusted first for maximum output with 1 mW of 144 MHz. IF drive signal applied to the mixer. Output power is measured on the 10368 MHz. TX output port. The RX chain is adjusted by injecting a low level 10368 MHz signal into the RX port and peaking for maximum output from the RX mixer

Other LO input, RF and IF frequency schemes may be used with the 10368-144BK if desired. Simply do the math. There are a few limiting factors. The IF will operate on any frequency up to approximately 2 GHz. and maybe higher if losses are not important. The RF filters will cover the whole 3 cm band. The first LO multiplier will tune from 3 to 6 GHz. on the output and will accept just about any frequency on the input. The second LO multiplier will tune from 8.5 GHz to 12 GHz. The signal levels are the only limiting factors for other frequency combinations, so the possibilities including high and low side injection, and odd IF frequencies are available. But! This is not recommended unless you have known signal sources and a spectrum analyzer to determine your desired outcome of the transverter.

For a more detailed circuit description, we recommend reading W1GHZ's paper in the 1999 proceedings of the Eastern VHF/UHF conference or the 1999 Microwave Update proceedings. More detailed information with pictures, is also available on our web site at www.downeastmicrowave.com.

10368-144BK COMPONENTS LIST

C1 47 ?F (0603)	C15 0.1?F	C29 1.0 ?F	R7 82 ?	R21 82?
C2 6.8 ?F (0603)	C16 0.1?F	C30 0.1?F	R8 51 ?	U1 ERA-2
C3 0.1?F	C17 1000 ?F	C31 0.1?F	R9 82 ?	U2 ERA-1
C4 1000 ?F	C18 1.0 ?F	C32 1.0 ?F	R10 51 ?	U3 ERA-1
C5 0.1?F	C19 1.0 ?F	C33 1.0 ?F	R11 51 ?	U4 ERA-1
C6 0.1?F	C20 1.0 ?F	C34 1000 ?F	R12 51 ?	U5 ERA-1
C7 1000 ?F	C21 1000 ?F	D1 MA4E2054B	R13 82 ?	U6 ERA-1
C8 6.8 ?F (0603)	C22 0.1?F	D2 MA4E2054B	R14 51 ?	U7 ERA-1
C9 6.8 ?F (0603)	C23 1.0 ?F	R1 51 ?	R15 82 ?	U8 ERA-1
C10 1.0 ?F	C24 1000 ?F	R2 82 ?	R16 82 ?	U9 MGA86576
C11 1000 ?F	C25 0.1?F	R3 82 ?	R17 51 ?	U10 ERA-1
C12 0.1?F	C26 1000 ?F	R4 51 ?	R18 51 ? (0603)	Circuit Board
C13 1.0 ?F	C27 1.0 ?F	R5 51 ?	R19 100 ?	
C14 1000 ?F	C28 1.0 ?F	R6 82 ?	R20 51 ?	

NOTE: All components are surface mount chip components. 1.0?F are 55 mil ATC. All other components are 1206 size unless noted. Any substation of values or types is at users own risk. All MMIC are biased for 9 VDC operation. For 8 VDC operation, replace all resistors with 51? (not included).

10368-144 Hardware Parts List

2 3/4" Pipe Cap	1 machined 1/4" plate	2 SMA connector	5 3-48 x 3/16" screws
7 1/2" Pipe Cap	2 4-40 x 3/8"	16 4-40 x 1/8" screws	3' # 28 Teflon wire
10 4-40 nuts	2 8-32 x 7/8" screws	18" RG-188 coax	8 4-40 x 3/8" brass screw
20 silver pins	4 1/8" x 3/16" rivets	1 4", 1/2" copper foil	2 8-32 plated nuts

Substitution of 8-32 x 7/8" stainless steel screws or 4-40 x 3/8" brass screws will alter the performance and tuning procedure of the filter circuits. The silver pins are used as the probes for the filters. They are cut to the following size. Measurement is total pin length 3/16" for all 1/2" pipe caps, 5/16" for F1 and 1/4" for F2. These lengths are established for the standard 1136 LO input to generate a 10224 MHz. LO injection to the mixers. Other frequencies may require different lengths.

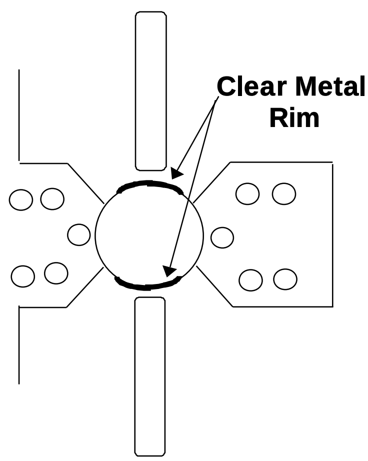
Assembly:

1. Inventory the parts list. Every part in this kit is important and should be identified. There are extra chip components packed in the vials, so no need to count them. The hardware should be sorted and identified. There are some extras. But remember, **No substitutions!** Review the schematic and the component placement diagram. Read through all of the assembly steps, 1 - 13, identifying every component used. Now is the time to get familiar with the kit and verify it is complete. This will also ensure that you have the correct tools and supplies required to complete the project. It is also time to make the last minute decision on building the kit or not. A full exchange towards a assembled version will be provided if you do not go past this step. We want

you to be on the band and operating not struggling to assemble this because you were not aware of what it takes to assemble a microwave transverter.------ Last chance?

2. OK, -----Lets build what we believe is “The most bang for a buck” 10 GHz. kit on the market! Lets start by examining the circuit board for irregularities. During the drilling and plating process, plating or debris may fill the filter probe holes an prevent the silver pins from being inserted. It is simple enough to use one of the silver pins (extras provided) an push it through all of the filter probe holes to be sure it clears 18 holes total. We have found that pushing them through from the ground plane side works best. **Do not make the holes larger**. Just be sure the pin fits in all of the holes and they are clear. Also look very closely for shorts from the pin hole to ground. After the pipe caps are soldered on, it’s to late to remove a short! Now check the TXIF and RXIF holes in the mixer (by D1 and D2) and the 1136 LO IN connection. These will be coax connections so also be sure of shorts or hole blockage with a silver pin.

3. Now inspect the MMIC mounting holes. The holes are plated. The PCB manufacturing process that provides us with the best grounding for the MMIC’s, sometimes leaves a little extra metal deposited where it will cause harm. Depending on the registration of the circuit board, the plating may creep up over the edge of the hole. If the MMIC is placed in a hole like this, it will short the input and output leads to ground. Look for a shinny rim circling the hole. If visible, (it may be hairline thin) remove with a sharp knife. Only remove the metal where the input and output leads of the MMIC’s may touch. **Do not remove the plating from the MMIC mounting hole that is connected to the top side ground pad!** Just touch up the imperfection with a knife when and where needed. See pictorial for clarity.



<u>MMIC PC Board hole with metal rim</u>

4. Install the PCB to the mounting plate. Place the ground plane side on the pallet and line up the holes. It only fits one way. Use the 4-40 x 1/8” screws but do not tighten the screws. The clearance holes in the PCB are purposely made larger so the board can move around slightly on the mounting plate to allow proper alignment of the SMA connectors and filters. Trim the Teflon insulators off of the SMA connectors. Then examine the flange of the connectors. Only be concerned with the surface that mounts to the plate. If the plating is rough or has a bur on it, use a file to remove and make smooth. Then trim the center pin length to approximately 1/16”. Remove all rough edges from the pin. Using the 3-48 screws, install the connectors. One on the RX position and the other at the 3408 test point. Now, you may find that there is excess material on the edges of the circuit board. This becomes evident when the connectors can not be mounted

flush. Cut the excess material off with a sharp knife being careful not to cut into the plated metal. You may use the pallet as a guide. Remount the SMA connectors and snug the screws in place. Now move the circuit board so that both connector pins line up with the traces. If you need to readjust the SMA connector position, do so. Having proper alignment of the connectors ensures that the filters will be correct. Tighten a few circuit board screws and check for alignment again. Repeat the process until all screws are tight and the SMA pins are properly aligned. Now install the two 4-40 x 3/8" screws and two 4-40 nuts in the clearance holes on the back corners of the circuit board. Tighten them and recheck all circuit board screws. Remove the SMA connectors, and then trim all of the excess circuit board material flush with the edge of the pallet on all for sides. This board edge while now act as the alignment if the circuit board is ever removed from the pallet.

5. It is now time to prep the pipe caps. All 9 pipe caps have been centered and marked. The two 3/4" caps need to be drilled with a #28 drill bit and tapped with a 8-32 tap. The seven 1/2" pipe caps need to be drilled with a #43 drill bit and tapped with a 4-40 tap. Use lubricant for both steps. Then remove all burs inside and outside of the caps and clean all excess lubricant with a cleaning solution. If you do not wish to drill and tap, you may send your caps back with $15 USD (which will also cover standard shipping) for a set of prepped caps. Now with an abrasive cloth, (sandpaper, Scotch-Brite?) buff-up the open end of the pipe cap, both inside and outside. The shinier, the better the solder will flow.

6. If you do not wish to solder the pipe caps to the PCB, we can do it for you. Return the pallet with the PCB mounted to it (all 17 screws) and the 9 un-drilled pipe caps without the hard ware with $35 USD which will also cover standard return shipping. Your pallet will be ready for assembly when returned!

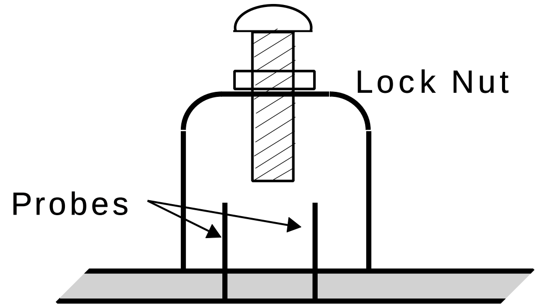
To solder the caps requires a propane torch, (a standard solder iron will not do the job!) electronic grade flux (liquid or paste) and electronic grade solder. **Do not use a Acid base flux or solder!** Even if you clean the acid flux-off of the outside when finished, you cannot clean the inside of the cap and the acid will destroy the board from the inside out.

Starting with a 1/ 2" pipe cap, flux the open end and place it into a hole at the end of the pallet. Then heat the cap with the torch, being careful not to allow the flame to come in contact with the bare circuit board, until you can flow solder into the corners of the pallet hole around the pipe cap joint. Remove the heat. Now repeat this process 8 more times until finished.

Some Tips for soldering the Pipe Caps are not to flux all of the caps at once and place them in the pallet before soldering. The flux will dry out on the last caps before soldering and cause a poor solder joint. Do not install the screws in the cap before soldering. The heated gasses inside of the cap need to escape. If you apply pressure to the pipe cap while cooling, be careful not to push the PCB away from the pallet. The excess solder will flow between the board and the pallet causing a "Bump" in the board. Although this is not a disaster, it will cause a problem when soldering the components to the circuit board.

7. At the factory, we use "No-Clean" solder exclusively. It disappears with heat without smoke. We never use any cleaning solution on our assembled products. The concern we have for kit builders is that after assembly of the pipe caps, excess flux and residue has migrated between the PCB and the pallet. Depending on the corrosiveness of the flux, this may be a potential problem. We suggest removing the 17 screws and cleaning everything completely if you feel that there could be a flux problem. Then inspect the solder joints for completeness. Be

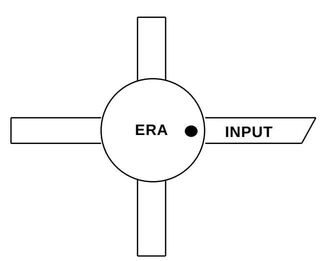
careful not to flex the circuit board excessively. Also, as ugly as it may be, do not trim any excess solder or re-flow a filter when it is not mounted to the pallet. It may case a misalignment and the PCB will not fit flush to the pallet when re-assembled. If you find a bad joint, re-assembly the PCB to the pallet being sure that the PCB is mounted flat to the pallet before tightening the screws and re-flow the pipe cap. Then, remove the PCB assembly, inspect and clean. Insert the screws and lock nuts into the pipe caps (as shown below, but do not install the probes!) to prevent any cleaning solution from running into the filter. Re-assemble the PCB to the pallet and be sure the PCB is flat on the pallet before the screws are locked down.



Pipe Cap detail

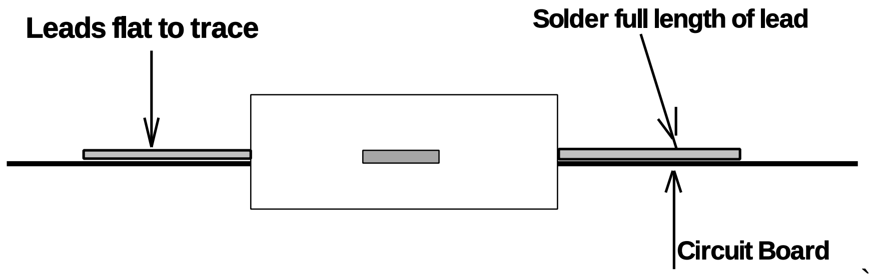
8. Referring to the component placement diagram, note that it shows all of the components that could be used on the PCB. The 10368-144BK components list has only the components required for the standard set-up.

For the actual assembly, start by installing the MMIC's, U1 - U10. You will need a large solder iron to flow the solder on the ground leads. We use 40 watt irons with 700 degree tips in the Down East Factory. We also preheat the pallet to about 25 degrees F above room temperature on a hot plate. You may do the same with whatever means of pre-heating you have including the torch you used for the pipe caps. If you can still handle the pallet after pre-heating, it could be hotter! The aluminum pallet will hold the heat for a while but re-heating may be required, depending on your soldering speed, to keep the solder flowing smooth. Be sure of the MMIC's alignment. Consult the component placement diagram with every installation. The MMIC's are very difficult to remove if installed incorrectly.



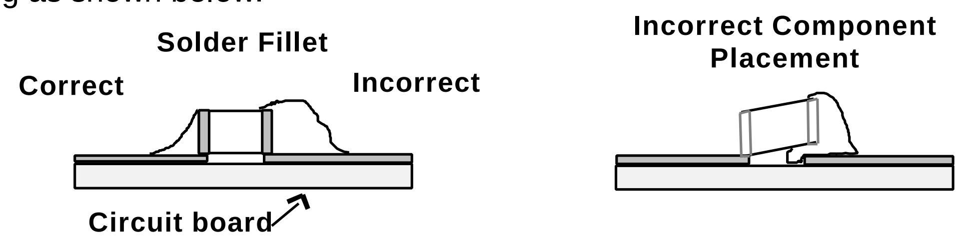
ERA MMIC and MGA86576

When installing be sure to install the MMIC's as flat as possible. The bodies, except for U-9, will fit squarely in the PCB holes. Be sure to flow the solder on the ground leads up to the body of the MMICs. These leads need to be a short as possible to ground. If you can see the ground lead, it's not soldered correctly. Use minimum amounts of solder on the input and output leads but solder them as close to the body as possible. After all, of the 10 MMICs have been installed, use an Ohm meter to check for shorts on the input and output leads to ground. If you did a good job in prepping the PCB in step #2, you should have no problems. If you do find a short, try wicking some of the solder from the shorted lead with Solder Wick? . If still shorted, try lifting the problem lead and re-wicking. Be careful not to break the lead. A close inspection should reveal a short. Use the knife and re-move, then re solder and re-test again.



ERA MMIC Installation

9. Install the DC by-pass capacitors, the 0.1uF, 1000 pF and most of the 1.0pF capacitors. They are connected from the DC circuitry to ground. Install and solder the ground side only. Do not solder the DC side of the capacitors. You may want to re-heat the pallet again to ease this installation and improve the solder flow. Be sure that the component is flat to the circuit board before soldering as shown below.



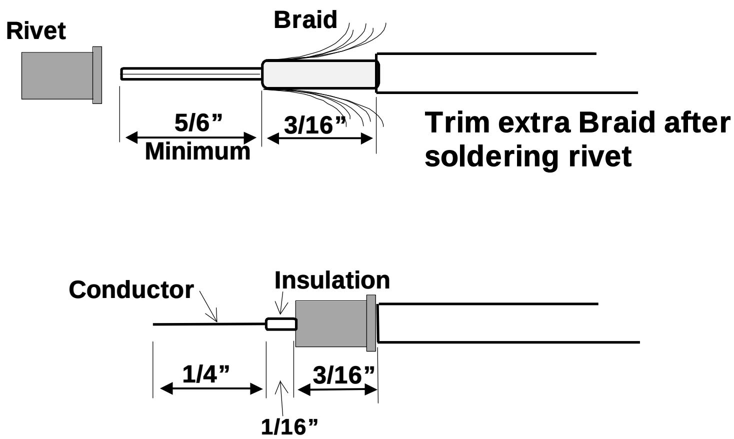
Proper SMD Assembly.

After all of the by-pass capacitors are installed, install all of the resistors. You will not need any extra heat for the rest of the soldering. The resistors share the solder pads with the by-pass capacitors. Be sure that the resistors are flat to the surface and solder one side at a time. When soldering the resistor, solder the capacitor that shares the same pad with the resistor. After the resistors are finished, install The DC blocking capacitors C1, C2, C9A, C19A, C28, and C29. Then solder R11 and R18 if it wasn't completed with the other resistors. **Do not install D1 and D2!**

10. Install the bias wires. All wires come in through the back of the pallet through the 1/8" holes. They get soldered to the circuit board where indicated on the bottom assembly diagram. They are labeled: **VTX** **VRX** **+9**

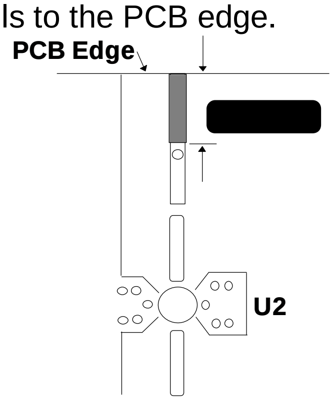
Using the #28 Teflon wire, cut, strip and tin the ends of a 2.5" length. Pass one end of the wire through the pallet in the VTX hole as shown in the bottom assembly document and solder it to the junction of R13 and C22. Cut, strip, and tin a 8" piece of wire. Insert one end of it with the lose end of the first wire through the other VTX hole by C25. Solder both wires there. Now connect the +9 together in the same manner using three 2.0" pieces and solder it by C12, C16, and C5. Now install the VRX bias wire with a 6" length to the VRX by R21

11. The coax now needs to be prepped. Cut it into 3 equal lengths (approximately 6"). The lengths given assume that you are building this transverter to be used in conjunction with the MICRO-LO and TC interface. If not, the LO IN may also be done through a SMA connector, but the TXIF and RXIF require the use of coax. If you use your own coax, be sure it is a high quality Teflon type. Nothing else will work because of the soldering technique used. Now, with whatever length and how many you have chosen, prep one end of each cable as shown in the diagram below. The idea is that the shield is soldered to the brass plated rivet and it is pressure fit into the pallet hole. Care must be taken that the shield does not short with the center conductor. It is simple to do.



Prepped Coax details.

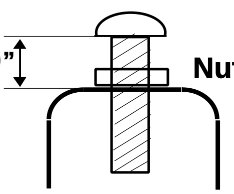
Once the coax is prepped, do a trail fit. Insert the center conductor through the hole in the pallet and then through the hole in the PCB (either RXIF, TXIF or LO IN). The reason in making the center conductor so long is so you have a guide for insertion. Press the brass rivet into the pallet hole. If the brass rivet fits loosely, remove it from the hole and squeeze it with pliers to make it out of round just a bit. If it fits to tight, (solder build up), scrape or file down the rivet's high spot and try again. We do use a bit of force to ensure a good connection to ground. Square the rivet with the hole and press in firmly with pliers. The rivet doesn't need to be inserted flush with the pallet but it needs to be a tight fit. Do the same for the other coax needed. After all coax is installed, cut off the excess center conductor and solder it to the circuit. After soldering the LO IN, also trim the excess RF line the travels to the PCB edge.



RF Line Trim Detail.

12. Install D1 and D2. Be very careful with the lead alignment. Now install all of the filter probes (the silver pins). Cut 14 pins to a total length of 3/16" or approximately 0.180" - 0.190". Insert them in the F3 through F9 filter holes, 2 each. Solder them in place using a minimum amount of solder. You do not want the solder to "Icicle down" and extend the pin length and thickness or short to the ground plane. Also, be sure that the pinhead is flat to the surface to ensure that the probe is straight. Now cut 2 pins at 1/4" or 0.250". Insert them into F2 and solder. Now insert two 5/16" or 0.310" - .0325" pins into F1 and solder.

13. Final assembly. If all of the filter hardware is not installed, install it now. The screw heights may be pre-set for tuning. Screw the 8-32's down on F1 and F2 until the bottom of the pan head is a little more than 1/8" (0.150") from the pipe cap top. Examine pictorial below. Be sure the 8-32 plated nut is also installed. It's thin for a reason. Then adjust the 4-40 brass screws and stainless nuts to approximately the same, 0.150" height. The lock nuts should be tightened but not locked.



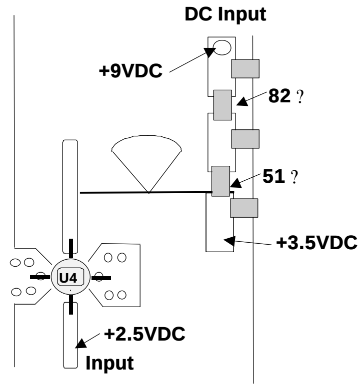
Now re-inspect all solder connections. Touch up what is questionable. Then review the test procedure before proceeding. Using the assembled unit for a "Dry Run" may clarify some of the testing requirements that may have been in question when first read.

***DC Testing*:**

1. With an Ohm meter, check all resistors and wire connections for shorts or opens. Then check the RXIF and TXIF cables for shorts. If all testing of the MMIC leads was good in Assembly step #8 the transverter is ready for DC and RF testing.

2. The MMICs, U1 – U8, U10, if working correctly, will draw current. This current drain will cause a voltage drop across the resistor network. For a +9 VDC supply, the total resistance in the bias network should be approximately 130 to 140 Ohms. With + 9 VDC applied to the network the voltage drop should be approximately 5.3 volts. This means U1 - U9 should have +3.5 - 4.0 VDC on the output lead and +2.5 VDC on the input lead. The input lead voltage may vary up or down as much as 0.5 VDC or more. Apply +9VDC to each stage in order and check the voltages on the leads of the MMICs. If large discrepancies occur, check the bias resistors for correct value. If a MMIC does not draw current, it is dead or the resistor network is open. If it drops voltage down to less than 1 VDC, its leads are shorted. Seek the problem out and repair. Now test in the following order:

The LO stage, U1-U5. The TX stage, U6– U8. The RX stage, test U10 only.



Resistor Network with Voltage Points

To test U9, the output lead should measure between +6 and +7 VDC. If it is over 7.2 VDC, remove bias immediately and check the bias resistors for proper value. If the values are correct, R19 may need to be adjusted up in value. If it is less than +5.5 VDC, the MMIC may be oscillating which could be caused by the filter not being tuned yet. Place your finger on the input of the MMIC to verify if the voltage changes in a upward direction. If the voltage measured is less than +2VDC on the output lead, it may have a solder short under either lead. Check, and repair.

<u>RF Testing:</u>

The testing procedure will describe operation for 10368 MHz. RF frequency with a 144 MHz. IF. It also assumes the DEM MICRO LO will be used as the 1136 MHz. oscillator. If using a 1136 MHz. oscillator other than the MICRO LO, verify the frequency and level. If you are using a different scheme (frequency multiplication or IF and RF frequency), the levels required will be different, therefore you will be on your own in determining the final outcome of the transverter. It is also recommended that anything other than this scheme or test procedure described, not to be attempted with out a Spectrum Analyzer.

1. Connect a SMA connector to the 3408 TEST POIT. Be sure that it is a flush mount. Verify that C9 is in the test position. (C9A on the component diagram) Connect the LO IN with the pre-tested MICRO LO 1136 MHz. oscillator by soldering the coax to the back of the LO PCB. Position the transverter so the filter adjustment screws are easily accessible. Connect a RF power detecting device to the 3408 TEST POINT. It is preferred that this is a Milliwatt power meter but a diode detector may be used for peaking. Apply +9VDC to the MICRO LO and the +9 on the transverter. If the 8-32 screws are pre-adjusted correctly, some output power should be detected. Adjust F1 first for maximum power. Use very slight adjustments (1/2 turn maximum each way to find peak.) while maintaining lock nut pressure to ensure the contact of the screw to the pipe cap. Be careful of downward pressure with adjustment tools on the filter when adjusting. Lock the nut into position when peak is found. Then adjust F2 for maximum power using the same technique. Lock the nut when peaked. If using a detector diode you can only peak it and on to step 2 of the RF test procedure. But if using a milliwatt power meter, power should be a minimum of +3dBm. More is OK, less is not! If test is OK, remove +9VDC and go on to step 2. If power is low, check the following in this order: Low DC Voltage, High DC voltage (over +10VDC) Low level from MICRO LO (coax short on either board). Filters tuned incorrectly (wrong starting point). Incorrect bias on U2 and U3. C9 not in the test position. Shorted C2. Incorrect probe length, probe missing, or probe shorted to ground. Suspect defective MMIC last. Always Question your construction.

2. Remove SMA connector from the 3408 TEST POINT and attach it to the 10224 TEST POINT. Place C9 in the operating position (C9B) and be sure that C19 is in the test position (C19A). Position the transverter so the filter adjustment screws are easily accessible. Connect a power detector to the 10224 test point and apply +9VDC to the MICRO LO and the +9 on the transverter. Some power should be detected if the filters were pre-set correctly. Adjust F3 first for maximum (1/2 turn each way for maximum peak). Be sure to maintain lock nut pressure while adjusting. Also be careful of downward pressure with adjustment tools on the filter. Lock the nut when peaked. Then adjust F4 for maximum. Minimum power should be +7dBm. If more OK. Remove +9VDC ad place C19 in the operation position (C19B) as shown on the component placement diagram, then go to step 3. If less than +4dBm, look for problems. Incorrect voltage. C9, C19 placement. Wrong starting point for filter screws. Check bias resistors for opens or shorts. Check probes for shorts, missing, or wrong length. Suspect defective MMIC last. Always Question your construction.

3. If this transverter is part of the Complete Kit version, the 10368-144C, to eliminate duplication of testing, please read and follow the Complete Kit assembly and testing instructions from here on. But If you wish, you could continue testing this assembly as if it were a independent assembly. The testing procedure of the CK version is more specific and deals with other aspects that pertain to the complete kit only. The BK testing is more general. If you are using this assembly as a independent transverter assembly please continue and follow this test procedure to the end.

For testing the 10368 MHz. TX, a low level 144 MHz. signal of about 1 mW (O dBm) is required. This may come from a signal generator or a 144 MHz. transceiver. Just do not exceed the level of 1 mW to align the TX chain while testing. When completed, you may drive the mixer with up to 10 mW safely. But, excessive harmonics will be produced and may cause problems in the alignment procedure. Connect the SMA test connector to the TX port on the transverter. Connect the TXIF coax to the 1mW 144 MHz source. Connect the power detector to the TX port. Position the transverter so the filter adjustments are accessible. Apply +9VDC to the LO chain and the VTX supply wire. Then apply the 144 MHz 1mW signal to the TX IF cable. If the adjustment screws of the filters were pre-set, monitor the power detector and adjust F5, F6 and F7 in order. Start by turning them ***IN*,** 1/8 of a turn at a time maintaining lock nut pressure. Power should be detected eventually. When power is detected, remove the 144 MHz. IF signal to verify that the detected signal vanishes. If it doesn't, you have tuned the filters for the LO frequency. If so, continue adjusting the screws 1/8 turn at a time in the same ***IN*** direction until the next power peak is detected. Verify that it is the desired signal by removing the 144 MHz IF signal. When you are sure you have the desired signal, peak all filters one at a time starting with F5. After each filter is peaked, remove the 144 MHz. IF signal and re-verify that the detected signal vanishes. Final output power should be greater than +5dBm with 0 dBm IF drive but will depend on the loss of the filters. If output power is less than +5 dBm, check the following in this order: Low or High voltage. LO multiplier and/or LO connected to +9VDC. C19 not installed in the operating position. IF drive power level (To low, or To High could cause tuning on wrong mixed product). IF cable short or open. Proper biasing of TX MMICs (check resistors). Filter Probes (length, shorted, missing). D2 blown from excessive drive. Suspect defective MMIC last. Always Question your construction.

4. To test the 10368 RX, a signal source at the desired receive frequency is required. This may be a signal generator, a harmonic from a transmitted signal source, or a on the air signal transmitted from 10.368 GHz. transmitter. The IF should be connected to a 144MHz receiver with an "S" meter, though a low level power meter, spectrum analyzer, or service monitor may be used.

Testing and alignment starts with connecting a SMA connector to the RX port on the transverter. Using whatever testing scheme you have, generate a 10368 signal and detect it on a 144 MHz. receiver through the RXIF cable. Be sure that the 10368 RX port is terminated with a 50 ohm device while adjusting the filters Do not exceed -20 dBm on the RX port from a signal source for damage may occur to the RX chain. Apply +9VDC to the LO chain and the VRX wire. Adjust F8 and F9 filters to maximize the IF level signal strength by turning the screws ***IN***. Adjust 1/8 of a turn at a time while maintaining lock nut pressure. Keep adjusting until peaked. When complete, lock the nuts. If can measure the conversion gain, it should be greater that 10 dB. It problems seem to occur, check all RX circuitry for errors or damage. The LO just tested OK!

Completion:

This completes the assembly and testing of the 10368-144BK. You now have enough knowledge of how this assembly works that implementing it into a complete working transverter system should not be a technical problem. Remember that the unit is designed to operate from a +9VDC source and if any other voltage is desired, the MMICs will need to be re-biased.

Its receive conversion gain is limited to the RX section specifications and still may require some IF amplification to over come your switching scheme when interfacing the transverter with a 144 MHz. transceiver. If the BK is to be used in a high performance terrestrial or EME set up, a Low Noise Amplifier may be desired. If so, additional filtering and isolation may be. If a higher power amplifier is added, also consider and additional filter and isolator. Also consider some attenuation if using a high gain TWT amplifier for +5dBm may be too much driving power.

As for QRP portable operation, the BK can be interfaced with our DEM TC that will control all switching functions of the transverter. You may also enclosed it in your own mechanical assembly or purchase our 10368-144 HK for a completed version that is all in one box to ease a portable operation. In it’s stock configuration, it is a reciprocal transceiver meaning it will hear as well as it can transmit.

Conclusion:

We hope you had fun with this kit and that you enjoy many hours of operation with this unit. Please take time to read the papers published by W1GHZ (N1BWT) for other operation tips and suggestions including antenna design and a 5760 MHz transverter design. The goal of this kit was to provide the radio amateur with a cost effective alternative to commercially manufactured units and a compact and more portable transverter than assembled from large sized surplus equipment.

Good luck with the DX and have fun!