DEM Part Number 10368-144CK and 10368-144LPCK
10 GHz. Transverter, Low power Transverter, Kit and Complete Kit

Specifications

Frequency:	10368.000 = 144.000 standard	
Noise Figure and Gain	< 1.5 dB NF, > 17 dBG	LP: < 3.5 dB NF, > 17 dBG
Power Out	> 2.9 Watts.	LP: >10 mw
DC Power	11-16 VDC @ 2.5 A	LP: < 600 mA
IF Power Input (144 MHz)	1 mw min. - 10-Watt max. Adjustable	

Preliminary:

The 10368-144CK and 10368 -144LPCK is a complete kit version of the 10368-144 and the 10368-144LP, Low Power transverter. In our Complete Kit version, you receive all three circuit board kits, the 10368K, (transverter PCB); the TC and the A-32 Synthesizer with a prepped extruded aluminum enclosure, mounting plate, and all necessary hardware and connectors to get you on the air. When completed, you will only need to supply a TR switch, Antenna, and a 2-meter transceiver. The 10368K is a PC board kit designed by W1GHZ and Down East Microwave Inc. When W1GHZ first designed and released the details of this transverter in the 1999 Microwave Update proceedings, it was a basic PC board design with some details of how to interface it with a few circuits to produce a simple state of the art transverter. DEMI has refined the design and produced a basic PCB Kit (low power) and two Complete Kits, a low power and a 3 watt version with all of the interfacing required. It is recommended by DEMI (but not required) that you obtain the 1999 Microwave Update proceedings and read it. It will provide some technical details such as design criteria and additional interfacing not covered in this Kit assembly manual. This article also describes a power detector that is a minimum requirement for testing this transverter. We recommend that you have a Microwave milli-watt power detector of some type as a minimum for proper alignment.

In this transverter, the DEM TC is the interface circuit board that allows the use of up to a 10 watt 144 MHz transceiver. It contains TX and RX level adjustments. It provides all of the DC and RF switching functions for the transverter actuated by it's keying circuit that is either PTT-L, (to ground) or PTT-H, (+ voltage) on transmit. It also supplies all internal biasing of the transverter and external switching functions for TR switches, PAs, and LNAs. It also contains a +9 Volt 1.5 amp voltage regulator to supply the regulated DC power to the transverter.

The A-32 synthesizer (the LO) provides a nominal +3 dBm output at 1136.000 MHz. for 144 MHz IF operation. The transverter PCB accepts the LO input and multiplies it to 10224.000 MHz. A test port for tune-up and testing the LO before the signal is injected into the mixers is available on the transverter board. The 10368 MHz. signals are tested through two separate SMA connectors, TX and RX. The TX and RX connectors are spaced correctly to allow a direct connection of common SMA relay to complete the system and allow the use of a common antenna connection.

This assembly manual assumes that the complete kit version (10368-144CK or 10368-144LPCK) is being assembled but is also used for the 10368K, the PC board kit version. Use what is required from this document for the different assemblies.

All kits include 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 this 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 3 cm frequencies to aid in tune-up. This can be done with a detector diode, a microwave milli-watt power meter or a spectrum analyzer. **The filters need to be tuned after assembly!** There are nine of them in the low power version and eleven in the 3watt unit. Both transverters will not work without correct alignment of the filters!

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.

Options:

There are no assembly options for the 10368 transverters detailed in this manual. You are limited by the transverter's PCB and supplied components of what version you can assemble. The manual will describe one way of assembly that will produce a state of the art 10368 MHz. transverter of choice. If you wish to experiment with different gain stages and add-on filters and amplifiers, do it with caution. Modifications should not be attempted without test equipment that will provide accurate power levels and a spectrum analysis. DEMI will be more than happy to consult you with any of your modifications, but will only assume full responsibility for this design if you use the supplied components and assembly procedure. If you decide to stray from the original design, or assemble the 10368 PCB with your own components, you will assume the responsibility of the results. We will answer any questions you may have **but a repair and/or alignment by DEMI will only occur if the transverter kit is in stock form!** If you have any questions about this policy please consult us first before proceeding with your own design or modification.

Assembly:

The TC should be assembled first per its assembly document. Some general assembly options may be implemented to it depending on your requirements. The DEM TC has provisions for an extra RXIF gain stage if you require more than 25 dB of system RX gain. The TC allows 1 mW to 10W common or separate IF input/output that requires the proper configuration. It's your choice. The regulated or unregulated voltage is switched to supply any additional stages such as a LNA or power amplifier or the PTT circuit may be mirrored or inverted. The standard document of the TC has many more details concerning its various options of operation. Please follow it for assembly and for the completion of the complete kit version of the 10368-144 or 10368-144LP.

The A-32 should also be set up and tested before the main transverter assembly is started. Refer to its manual and select the jumpers required for the desired frequency of operation. You may temporally connect the Blue "Loc" led and the 10 MHz external source if required for testing. You may also assemble the A32 internal clock option if desired before testing

When configuring your system, please note that DEMI can supply a few external options. If you require a "State of the Art" noise figure, a 3 cm PHEMT preamplifier designed by W5LUA can be purchased. It is part number 3-2LNAHK (board kit) or a complete kit version is 3-2LNAHCK. This is the same design used in the 3watt version transverter.

If you require more output power than the 3-watt version, and have access to the surplus market of TWTs and other solid-state amplifiers, be sure of the drive level requirement of the amplifier. Some amps and TWTs may only require a maximum10 mW of drive, which can be easily

provided by the LP version of this kit. DEMI also stocks relays, a basic sequencer, and other accessories that will enable you to complete your 10368 MHz. station.

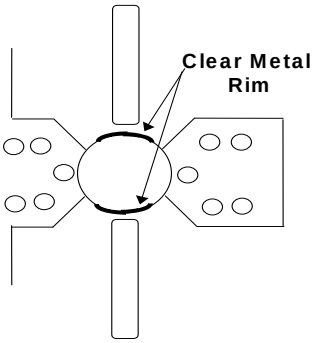
Assembly:

1. Start by inventorying 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 there too. However, remember, **No substitutions!** Review the schematic and the component placement diagram. Read all of the assembly steps, 1 - 14, identifying every component used. Some instructions will specify the LP, the low power version or the Standard 3 watt version transverter.

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 an 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 kit 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! Start by examining the circuit board for irregularities. Do not remove the copper foil that is attached to the backside of the board. This is to provide additional grounding for the 3 watt power MMIC. During the drilling and plating process, plating or debris may fill the filter probe holes and 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 for the LP and 22 for the standard unit. We have found that pushing them through from the ground planeside 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 pinhole to ground on the ground planeside of the board. 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 plated MMIC mounting holes. 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 topside ground pad!** Just touch up the imperfection with a knife when and where needed. See pictorial for clarity.



MMIC PC Board hole with metal rim

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 3/16" 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 insulator off one SMA connector. Then examine the flange of the connector. Only be concerned with the surface that mounts to the plate. If the plating is rough or has a bur on it, use a small, fine file to remove and make smooth. Then trim the center pin length to approximately 1/16". Remove all rough edges from the pin. Refer to the component placement, and using the short 3-48 screws, install the connector to the pallet at the 10224 MHz. LO test point position. Now, you may find that there is excess material on the edge of the circuit board. This becomes evident when the connector cannot 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. Tighten the SMA screws in place. If you need to readjust the SMA connector position, do so. Having proper alignment of the connector and the PC board mounting screws ensures that the filters will be correct. Tighten a few circuit board screws close to the connector and check for alignment again. Remove the connector and place it in the TXRF position. Repeat the process of trimming and aligning until all board mounting screws are tight and the SMA pins are properly aligned. Now install the two 4-40 x 9/16" screws, the 1/8" spacers and two 4-40 nuts in the clearance holes on the back corners of the circuit board. The head of the screw belongs on the pallet. The spacer is between the PCB and the nut. Tighten them and recheck all circuit board screws. Remove the SMA connector and then trim all of the excess circuit board material flush with the edge of the pallet on all for sides. Try not to cut into the ground plane. When complete, the board edge will 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 off the pipe caps have been drilled and tapped. Remove all burs inside and outside of the caps and clean all excess lubricant with a cleaning solution. Then, 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. Do not attempt to "Square Off" the open ends of the caps. The mount plate will hold them in position during soldering.

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 and aligned with the un-drilled pipe caps **without the hardware.** Include you payment or payment information of $45 USD, which will also cover standard return shipping. Your pallet will be ready for assembly when returned!

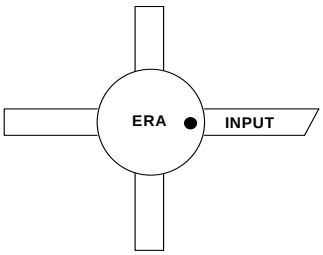
To solder the caps requires a propane gas torch. A standard solder iron will not do the job! Electronic grade flux (liquid or paste) and electronic grade solder is also required. **Do not use a Acid base flux or solder!** Even if you clean the acid flux off the outside when finished, you cannot clean 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 pallet hole at the end of the pallet. Now heat the cap with the torch being careful not to allow the flame to come in contact with the bare circuit board. Allow the solder to flow into the corners of the pallet hole around the pipe cap joint then remove the heat. Now repeat this process on the rest of the pipe caps until finished. After a few are complete, the soldering is faster because the pallet is now pre-heated.

Do not 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. Do not apply pressure to the pipe cap while cooling. It will push the PCB away from the pallet. Then 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.

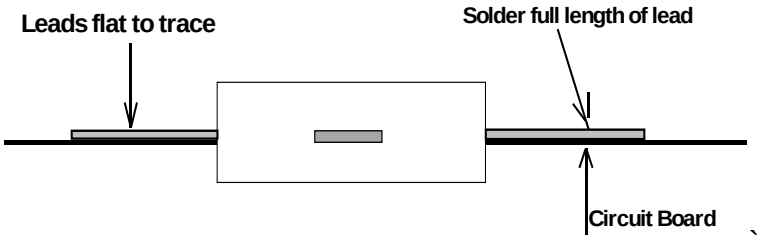
7. At the factory, we use “No-Clean” solder exclusively. It disappears with heat with minimal smoke. We never use any cleaning solution on our assembled products. The concern we have for kit builders is that after assembly of the pipe caps, excess flux and residue has migrated between the PCB and the pallet. Depending on the corrosiveness of the flux, this may be a potential problem. We suggest removing the PCB mounting 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, do not trim any excess solder or re-flow a filter when it is not mounted to the pallet. It may cause 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.

8. Referring to the component placement diagram, it depicts the standard version transverter but the LP version is indicated. Use the component list to determine what components are installed. 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 Factory. We also preheat the pallet to about 100 degrees F 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.



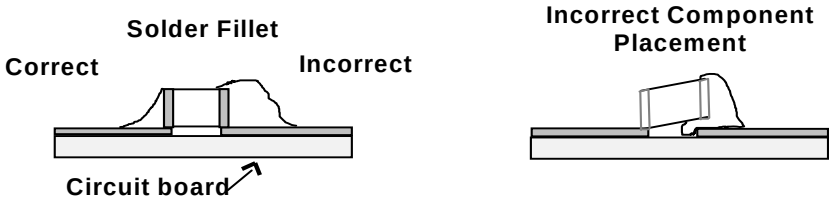
ERA MMIC and MGA86576

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 Ohmmeter 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 0.1 uF, 100 pF and most of the 1.0 pF capacitors. If you have the standard version, you should also install the 10 uF's. Be sure of their polarity!! 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.

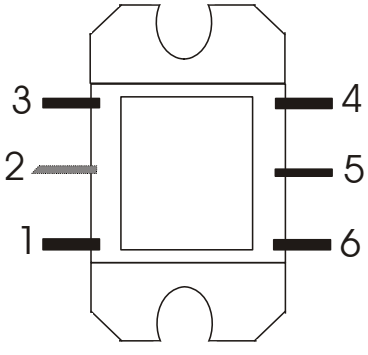


Proper SMD Assembly.

After the by-pass capacitors are installed, install the surface mount 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, C29 with R11 and R18 if it wasn't completed with the other resistors. Just **Do not install D1 and D2!**

If you have the standard version transverter, complete all of the LNA/PA section components except for IC1, Q1, R23 and the jumper. Aslo, do not install R22, R30 and IC2. You should remove the mounting screw near the VR1 position before installing. Now install Q1,then R23. Review the FMM 5061LV pictorial for pin numbers, then install IC1 after applying a light coat of thermal compound before installing with 3-48 screws. Be sure of the alignment, tighten the screws, and then solder the leads. Now Fabricate the jumper and install between both Drain pins, 4 and 6 keeping it as short as possible.

Pin Assigment
1 VGG
2 RF in
3 N.C.
4 VDD
5 RF out
6 VDD

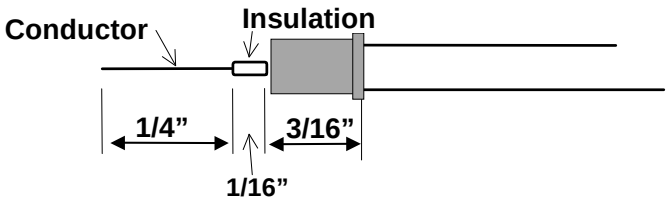


10. Install the bias wires. All wires come in through the back of the pallet through the 1/8" holes. They are soldered to the circuit board where indicated on the bottom assembly diagram. They are labeled: VRX, VTX, +9, and -5VDC if you have the standard version transverter.

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 between F3 and F6 as shown in the bottom assembly document and solder. 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 F7. Solder both wires there. Now connect the three +9 connections together as shown in the same manner using three 2.0" pieces and solder . Then install the VRX bias wire with a 6" length. This completes the LP version

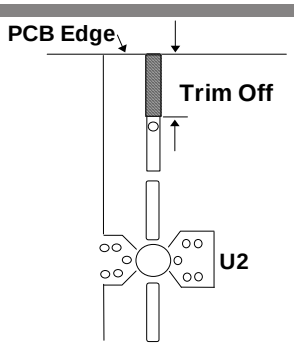
For the standard version transverter, cut and tin a 3" and a 1" #28 wire. Install both wires in the –5VDC position by IC2 then connect them to their respected –5VDC connections as shown. Then install a 10" #28 wire in the VRX and a 9" in the +9. Install the 4-40 feed thru in the +10TX hole then bend of end and solder to the pad shared with C39 on the PCB. Using 3-48 screws, install the 3-ohm, 10-watt resistor as shown. Then connect the +10TX connections with a 1" #22 wire. Connect a 9" #22 wire to the other end of the 3-ohm resistor. All wires should be routed as shown on the bottom assembly.

11. All of the coax connections need to be made next. Start by cutting three 6" pieces. The best way of making this coax assembly will result in something that looks like diagram below. Start by removing approximately ¾" of the outside jacket. Tin solder the entire exposed shield. Place the Rivet in position and solder in place. Try to keep the coax in the center of the rivet while the solder cools. Now with a #22 wire stripper, remove the shield that is below the rivet. The wire striper should score the shield if it is tinned and you will be able to "Snap" off the shield. Then with a #28 wire striper, remove the insulation off the center conductor. Leave a small amount of insulation as shown in the diagram. Tightly wrap the exposed center conductor and lightly tin the end. Be sure not to let the solder expand the diameter of the center conductor.



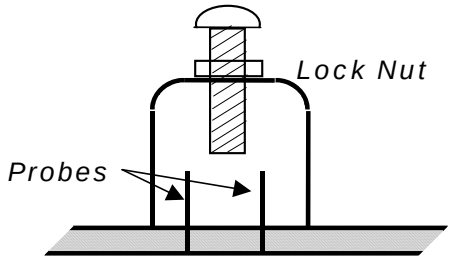
Prepped Coax details.

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 LOIN). The reason in making the center conductor so long is so you have a guide for insertion. Fit the brass rivet into the pallet hole. If it fits to tight, (solder build up), scrape or file down the rivet's high spot and try again. Notice the tapped hole located close to the rim of the rivet. Insert a 4-40 x 3/16" screw and tighten. The head of the screw should compress down on the rivet rim and hold it in place. Do not over tighten. Then do the same for the other coaxes. After all coax is installed, cut off the excess center conductor and solder it to the circuit. Then trim the excess RF line that travels to the PCB edge as shown below on the LO input circuit.

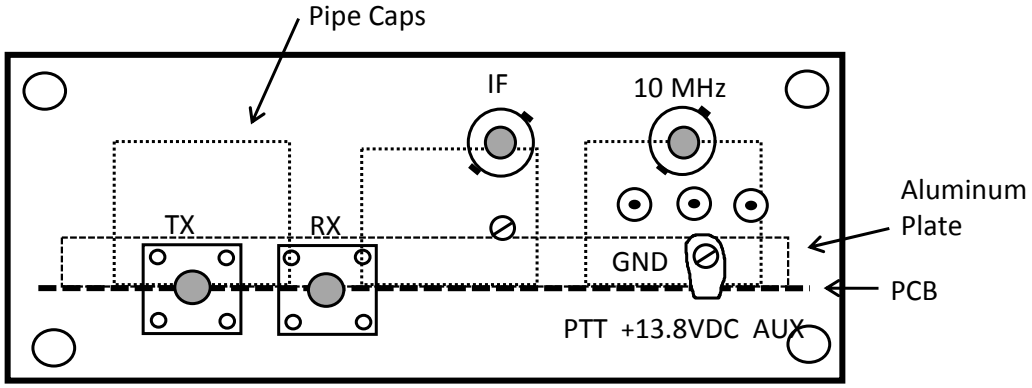


LOIN Line Trim Detail.

12. Install D1 and D2. Be very careful with the lead alignment. Then install all of the filter probes (the silver pins) after cutting to the correct size. The length indicated is the total length including the head. Cut 4 pins at 3/8" or 0.375". Insert them into F2 and F1, then solder. Cut the rest of the needed filter pins (there are extras!) to a total length of 3/16" or approximately 0.180" - 0.190". Insert them in the remaining filter holes. 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.

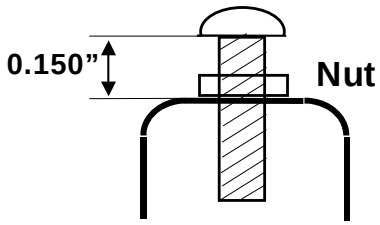


13. Install the rear panel to the PCB by first prepping two SMA connectors so that the Teflon will not interfere with the circuit board. Insert the SMA connectors through the connector panel and using the panel as a guide, trim the excess Teflon with a sharp knife flush with the panel. Make a clean, square cut. Now trim the center pins to 1/16" extending from the Teflon. Attach the SMA connectors to the panel with four of the #3-48 x 3/16" screws. Then attach the panel to the pallet with five 3-48 x 3/8" screws. Note that the ground lug is mounted under one of these screws. Look for any gaps or bulges between the pallet and the panel. Tighten the 5 screws evenly and be sure the cut center pins do not interfere with the circuit board. If needed, dis-assemble and re-trim the Teflon on the connectors or the PCB and try again to obtain a perfect fit.



Attach all of the 8-32 feed thru connectors and the required BNC connectors. If you desire a common IF, receive and transmit through a single connector, only install one BNC. If you desire separate Receive and Transmit IF connections, install both BNC connectors. Be sure to install the lock washer and the ground lug on all BNC connectors.

14. Install all of the filter hardware 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 the next pictorial. Be sure the 8-32 plated nut is also installed. It's thin for a reason. 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. Set the 4-40 brass screws and stainless locking nuts to approximately the same 0.150" height. The lock nuts should be tightened but not locked.

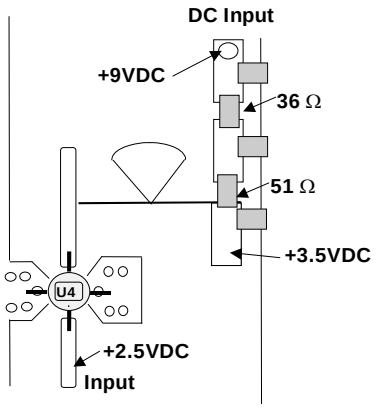


Re-inspect all solder connections. Touch up what is questionable. Now review the test procedure before proceeding. Using the assembled unit for a "Dry Run" may clarify some of the testing requirements that may have been in question when first read.

***DC Testing*:**

1. With an Ohmmeter, 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 a "Go" in Assembly step #8 the transverter is ready for DC and RF testing.

2. The MMICs, U1 – U10, if working correctly, will draw current. This current drain will cause a voltage drop across the resistor network. With a +9 VDC supply, the total resistance in the bias networks of all but the U9 MMIC should be approximately 80 - 90 Ohms. With + 9VDC applied to the network the voltage drop should be approximately 5.3 volts. This means U1 – U8 and U10 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 (very Rare!) 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, and 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 an 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.

3. If you have the standard version transverter, the LNA/PA section should be tested by installing and verifying the operation of IC2. Check for –5VDC (negative and it may be as low as –7 VDC) where indicated on the PCB. If OK install R22 and R30 from the pallet side of the board. After installation, preset the bias for Q1 by measuring the gate voltage at the flag by R27. Adjust R30 for –0.45VDC. Then connect a +9VDC to the VRX by R26. Measure the voltage at the flag by R24 and adjust R30 to obtain +2.5 VDC. Remove the VRX voltage when complete. Now pre-set R22 for –5VDC at the gate of IC1 measured by C37. The DC testing is complete.

LO Testing:

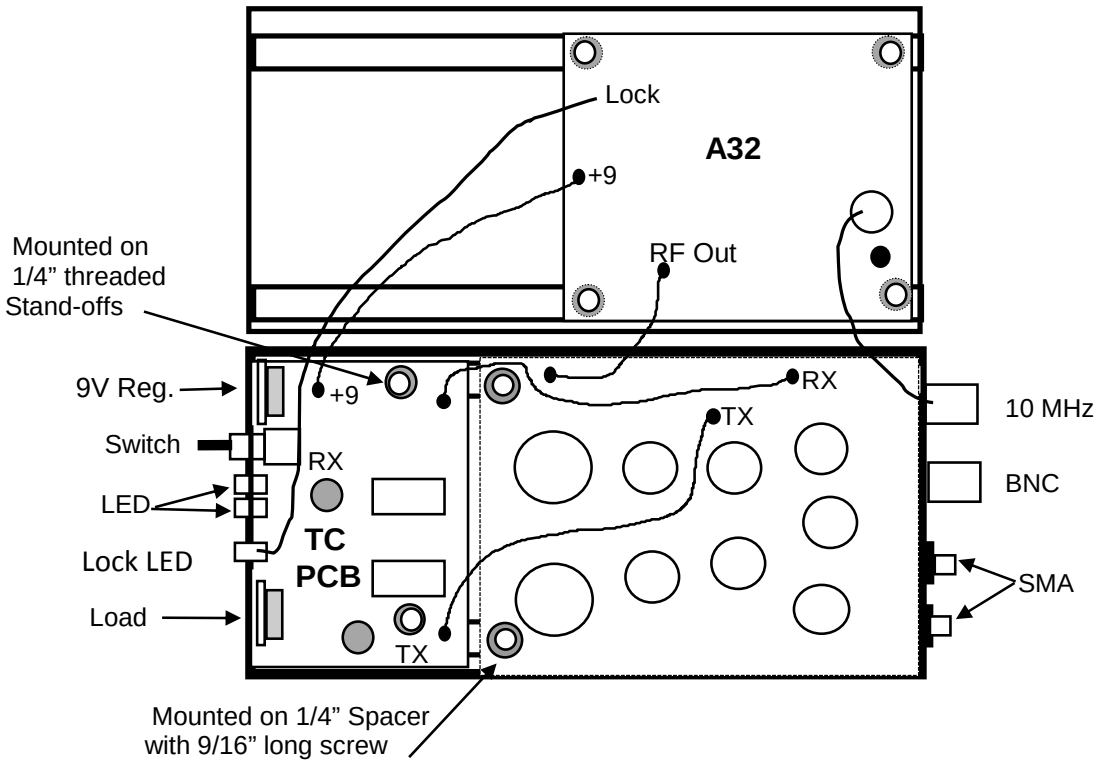
The testing procedure will describe operation for 10368 MHz. RF frequency with a 144 MHz. IF. It assumes the A-32 will be used as the 1136 MHz. oscillator. If anything other than this scheme or test procedure is used out of the 2 meter band, do not attempt without a Spectrum Analyzer.

1. Connect the trimmed 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 input with the pre-tested A-32 1136 MHz. oscillator by soldering the coax to the back of the A-32 PCB. Also connect the 10 MHz external source if required. 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 A-32 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. If using a milliwatt power meter, power should be a minimum of +3 dBm. Greater than is OK, less than 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, (The wrong starting point), Incorrect bias on U2 and U3, C9 not in the test position, Shorted C2 or just remove it from circuit, Incorrect probe length, probe missing, or probe shorted to ground. Suspect defective MMIC last if voltages are correct. 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 (C9) 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 filters. Lock the nut when peaked. Then adjust F4 for maximum. Minimum power should be +7 dBm. Again, greater than is OK, less than is not! Remove +9VDC ad place C19 in the operation position (C19) as shown on the component placement diagram, then go to step 3. If it is less than +7 dBm, 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 if voltages are correct. Always Question your construction.

3. For further testing of the complete unit, the enclosure assembly should be finished. Start by examining the two enclosure halves. Notice that in one half, the four taped holes are closer to each other than the other half. This is the transverter side. The installation of the transverter assembly is easier if the coax is detached from the MICRO-LO board first. Install the pallet assembly in the housing using the two 4-40 x 9/16" screws and 1/8 spacers. Remove the nuts and insert the pallet in the enclosure being sure not to pinch any wires. Seat the pallet in place and start the screws. Then start the two flat head panel screws. When everything lines up and you are sure about the spacers being in place, tighten all four screws. Refer to the figure below



4. Install two 4-40 standoffs with split lock washers in the two remaining holes that are in the transverter side of the enclosure. Make them snug but do not over tighten. The threads are Aluminum. Now refer to the Installation section of the TC assembly document and proceed with the installation. The only deviation from the TC instructions is to install a #22 wire on the +13.8VDC connection from the feed thru to the TC board. Be sure of your IF configuration and continue to step five below after completion.

5. After the TC installation, install the A-32 in the other housing half as shown above using four 4-40 x 3/16" screws. When both halves are aligned as in the pictorial, follow the retrofit kit instructions for the LO coax, the 10 MHz coax, the Loc LED and the DC power wire installation. If using an on board 10 MHz source, disregard the 10 MHz coax install. Re-check all connections and verify that there are no lose wires, if so identify and connect if needed. Check all mounting hardware and connectors to be sure nothing is lose. If you are assembling the standard unit, a wire from the 3-ohm, 10-watt resistor is not connected.

Testing:

1. Start by getting the same equipment ready that was used for the A-32 test. Preset the IF pots on the TC board by turning them counter clockwise. Connect 13.8VDC to the transverter and switch the power on. The LED should light. If you have the on board 10 MHz source, the Loc light should also come on. If not it should be off or blinking. Connect the 10 MHz external source and the Loc light will be lit. Check voltages on the TC board. There should be +9VDC on the VRX and the +9 points. There should also be +13.8VDC where indicated on the TC. Now toggle the TC board by activating the PTT circuit. The relay will click and the TX LED will light. VRX drops to zero volts and the VTX will now be +9VDC. If you have the standard version transverter, install the wire from the 3-ohm resistor to the +13TX connection on the TC board. Then verify that Pin 1 of IC1 has –5VDC then toggle the PTT and verify that there is a 4.0 - 6.0 VDC drop across the 3-ohm resistor. If not verify that that the drain voltage is connected to IC1 when in transmit. If it is low, < 7 volts, check for a short to ground of pin 4 and 6 or R22 may not be adjusted for –5VDC. If voltage is in spec the unit should now be ready for RF testing.

2. Even though the fixed attenuator has been predetermined, for initial testing, the TXIF pot should be adjusted to the maximum attenuation position. If you don't, excessive drive may produce spurious signals and make aligning the TX chain more difficult. Connect a RF power detector to the TXRF port of the transverter. Position the transverter so the filter adjustments are accessible. Key the PTT circuit and apply the 144 MHz TX signal to the transverter. 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. You may also want to adjust the TXIF pot on the TC board to verify operation. If the power doesn't change, 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 +5 dBm in the LP version with your desired IF drive level. Toggle the PTT with the IF drive off and check for oscillations. If none are detected, proceed to the RX testing. If you have a standard version transverter, finish by adjusting F10 to produce around 2 watts of output power.

If output power is less than +5 dBm, check the following in this order: Low or High voltage. Verify that the A-32 is connected to +9VDC. Check C19 to verify it is installed in the operating position (need to remove pallet to check this). IF drive power level either too low or too high could cause tuning on LO signal or wrong mixed product. Is the IF coax cable short or open? Is C28 (LP) or C38 (standard) damaged from the connector installation? Filter Probes (length, shorted, missing?). D2 blown from excessive drive. Suspect defective MMIC last if their voltage is correct. Always Question Construction!

3. 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 a 10.368 GHz. transmitter. The IF port should be connected to a 144 MHz. receiver with an "S" meter, though a low level power meter, spectrum analyzer, noise figure meter or service monitor may be used. The transmitted signal from the transverter should not be used because of the use of the same IF frequency. It would be very difficult to determine what was the desired signal versus the radiated 144 MHz signal being detected in the IF receiver, no mater what level it was. At the factory, a 10368 MHz. signal generator set for -30 dBm for the LP and –50dBm for the standard version is injected directly into the RXRF port of the transverter and is adjusted for maximum gain into a 144 MHz. receiver. Then we test and optimize it with a noise figure meter.

Your actual testing and alignment starts with generating a 10368 MHz. signal and detecting it on a 144 MHz. receiver through the RXIF connector on the transverter. Be sure that the 10368 RX port is terminated with a 50 ohm device while adjusting the filters. If connecting a signal generator directly to the RX port, do not exceed -20 dBm. A signal source higher than that will compress all of the gain stages, possibly generate harmonics, and make alignment difficult. With the DC power applied to the transverter, adjust F8 and F9 filters by turning the screws ***IN*** to maximize the IF level signal strength. Adjust 1/8 of a turn at a time while maintaining lock nut pressure. Keep adjusting until peaked. When complete, lock the nuts. If testing a standard transverter, F11 should also be adjusted and then "Tweak" R30 for the final dB or two. Further optimizing can be done with a noise figure meter.

If RX gain is less than expected, check the following in this order: Low or High voltage. Verify if the RXIF gain control is at minimum or the RXIF gain stage on the TC board, if installed, is working correctly. Verify the input signal level. If it is too high, it could cause harmonics in the Q1 stage allowing the misalignment of the filters. Is Q1 oscillating? Is the IF coax cable short or open? Is C29 (LP) or C42 (standard) damaged from the connector installation? Filter Probes (length, shorted, missing?) Is D1 blown from excessive drive during the TX test? Suspect defective MMIC last if the voltages are correct. Always Question Construction!

Completion:

When you become satisfied with the operation of the 10368-144, you may want to bundle all of the control wires and coaxes together to make a neat appearance. It is not necessary, but it will contain the wires to make closing the enclosure easier. Do not restrict the movement of the A-32 coax, DC supply wire and Loc light wire . Do a final check on the TX and RX IF levels and if you are satisfied, close the enclosure and bolt it with the four remaining flat-head screws. Be sure not to pinch the LO coax and DC wire between a filter screw and the A-32. Be sure to re-test the transverter after closing.

This completes the assembly and testing of the 10368-144. You now have enough knowledge of how this assembly works that implementing it into a complete working system should not be a technical problem. Remember that the unit is designed to operate from a +13.8VDC

source but any voltage between 11 and 16.5 VDC will work making it perfect for portable operation. You may wish to test this before going portable.

The LP versions receive conversion gain is limited and may require some additional amplification. To overcome your IF switching scheme insertion loss when interfacing the transverter with a 144 MHz. transceiver, you may want to install the MMIC to the TC board. If the transverter 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 required and/ or the use of the RX OPT may need to be omitted. If a higher power amplifier is added, consider and additional filter and/or isolator. Also consider some attenuation if using a high gain TWT amplifier for +5 dBm may be too much driving power.

The standard version only needs a SMA TR switch to make it a top-notch portable rig mated to a modest size antenna that will produce Hill top DX! Both units can be interfaced with our AOS144 RF sensed IF switch attenuator that may be used with transceivers up to 25 watts output. Higher output power transceivers are not recommended unless modified. Addition of external power amplifiers and LNA's can be accomplished with the AUX output of the transverter or by implementing a sequencing scheme. Remote location mounting is possible with this unit in is stock form. It would just need to be installed in a weatherproof enclosure.

Conclusion:

We hope you had fun with this kit and that you enjoy many hours of operation with your completed transverter. Please take time to read the papers published by W1GHZ (N1BWT) for other operation tips and suggestions including antenna designs and a similar 5760 MHz transverter design. Good luck with the DX and have fun!

NOTICE:

This equipment is to be used solely by licensed amateur radio operators within the specifications and guidelines governed by their licensed agreements.

This product should not be thrown away. Please dispose any component or equipment properly. By reusing, returning or using proper collection points which are designated for it.

10368-144LP COMPONENTS LIST

C1 47 ρF (0603)	C15 100 ρF	C29 1.0 ρF	R7 36 Ω	R21 36 Ω
C2 6.8 ρF (0603)	C16 0.1μF (1210)	C30 100 ρF	R8 51 Ω	U1 ERA-2
C3 0.1μF (1210)	C17 100 ρF	C31 0.1μF (1210)	R9 36 Ω	U2 ERA-1
C4 100 ρF	C18 1.0 ρF	C32 1.0 ρF	R10 51 Ω	U3 N6
C5 0.1μF (1210)	C19 1.0 ρF	C33 1.0 ρF	R11 51 Ω	U4 N6
C6 100 ρF	C20 1.0 ρF	C34 0.1μF (1210)	R12 51 Ω	U5 N6
C7 0.1μF (1210)	C21 0.1μF (1210)	D1 MA4E2054B	R13 36 Ω	U6 N6
C8 6.8 ρF (0603)	C22 100 ρF	D2 MA4E2054B	R14 51 Ω	U7 N6
C9 6.8 ρF (0603)	C23 1.0 ρF	R1 51 Ω	R15 36 Ω	U8 N6
C10 1.0 ρF	C24 0.1μF (1210)	R2 82 Ω	R16 36 Ω	U9 MGA86576
C11 0.1μF (1210)	C25 100 ρF	R3 51 Ω	R17 51 Ω	U10 N6
C12 100 ρF	C26 0.1μF (1210)	R4 82 Ω	R18 51 Ω (0603)	Circuit Board
C13 1.0 ρF	C27 1.0 ρF	R5 51 Ω	R19 100 Ω	
C14 0.1μF (1210)	C28 1.0 ρF	R6 36 Ω	R20 51 Ω	

NOTE: All components are surface mount chip components. 1.0ρF is 55 mil ATC. All other components are 1206 size unless noted. Any substation of values or types is at users own risk.

10368-144LP Hardware Parts List

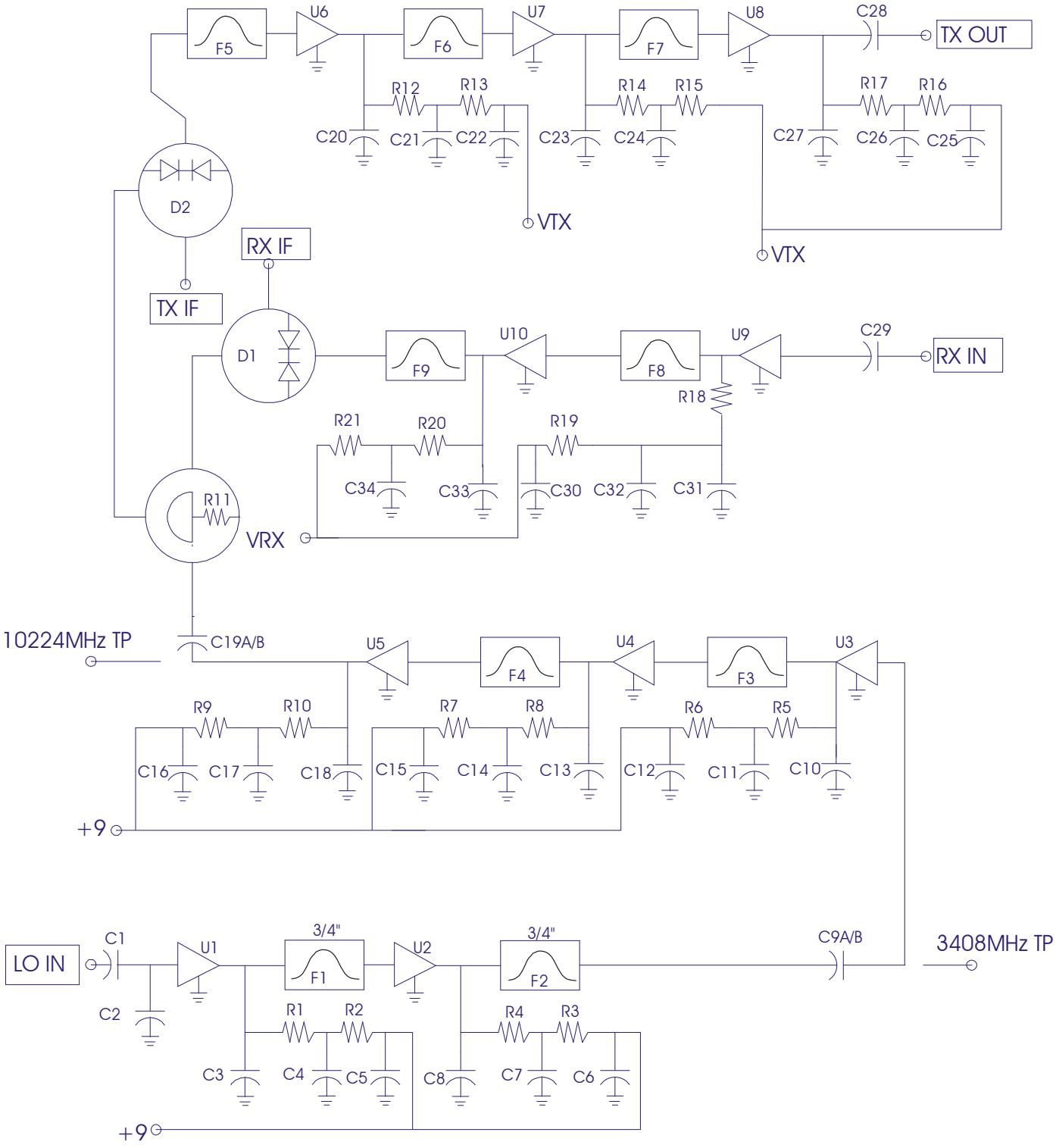
2 3/4" Pipe Cap	8 4-40 locknuts	20 silver pins	3' # 28 Teflon wire
7 1/2" Pipe Cap	2 8-32 plated nuts	36" RG-188 coax	3 SMA connector
20 4-40 x 3/16	2 8-32 x 7/8"	8 4-40 x 3/8" brass	4 1/8" x 3/16" rivets
6 3-48 x 3/16"	5 3-48 x 3/8"	1 SPDT switch	2 1/4" threaded standoff
2 4-40 x 3/8"	2- #4 split washers	1 machined 1/4" plate	2 BNC connector
2 4-40 x 9/16	2- #4 x 1/8" spacer	1 # 4 Ground Lug	2 Machined End Plates
3 4-40 x 1/4"	2 4-40 nuts	3 8-32 feed thru	8 Flat head screws
1 Hole plug			

10368-144 LNA and Power Amplifier Section (standard)

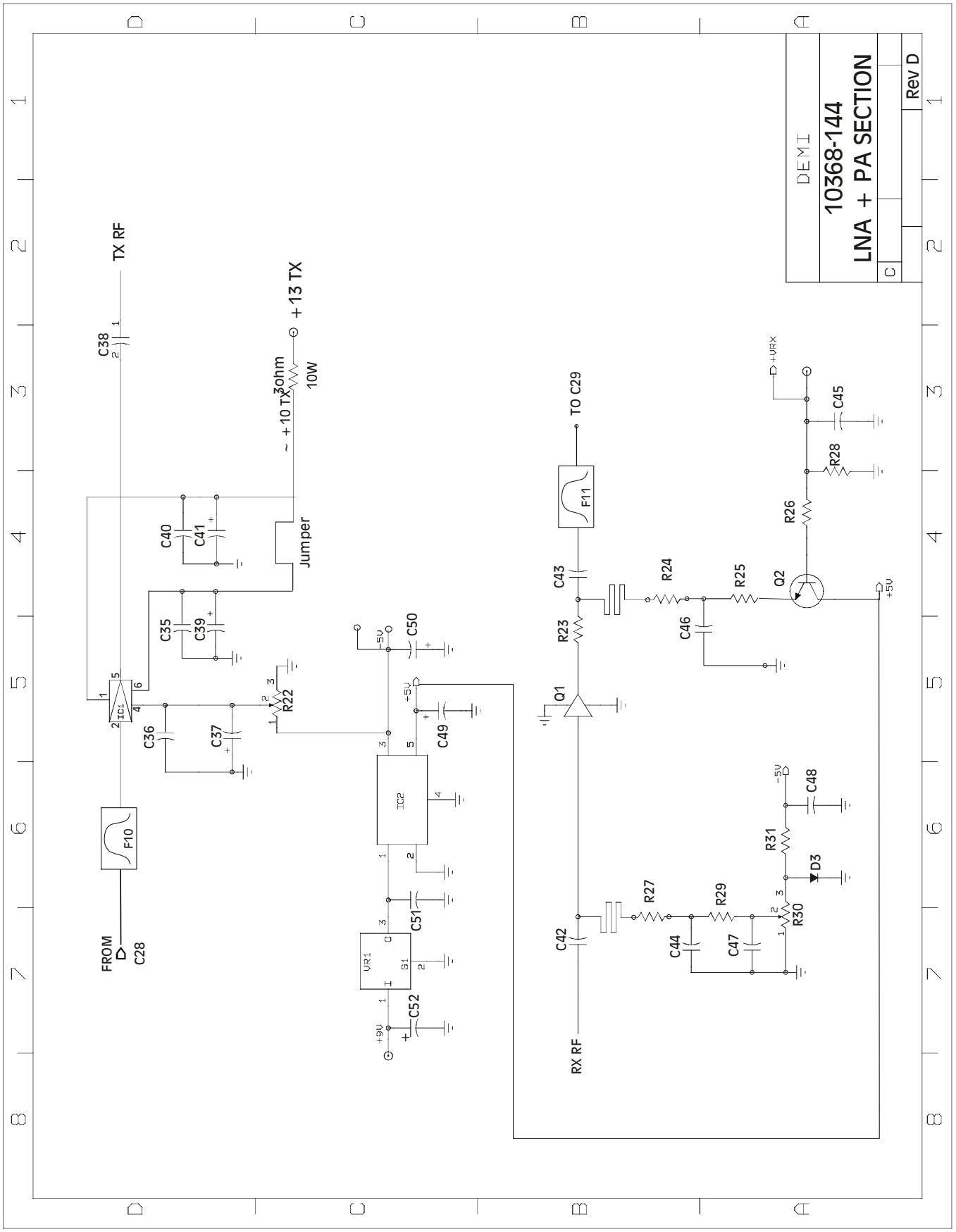
C35 100 ρF	C43 1.0 ρF ATC	C51 10.0 μF	R22 200 Ω pot	R30 1K Ω pot MT
C36 100 ρF	C44 1.0 ρF ATC	C52 10.0 μF	R23 10Ω (0603)	R31 5.1K Ω
C37 10 μF	C45 0.1 μF (1210)	D3 MMBD914	R24 51Ω (0603)	VR1 78M05
C38 1.0 ρF ATC	C46 0.1 μF (1210)	IC1 FMM5061VF	R25 100 Ω	1- 3 Ω 10W
C39 10 μF	C47 0.1 μF (1210)	IC2 NMA0505S	R26 5.1K Ω	
C40 100 ρF	C48 0.1 μF (1210)	L1 Jumper	R27 51 Ω (0603)	
C41 10 μF	C49 10.0 μF	Q1 ATF36077	R28 5.1K Ω	
C42 1.0 ρF ATC	C50 10.0 μF	Q2 MMBT3904	R29 100 Ω	

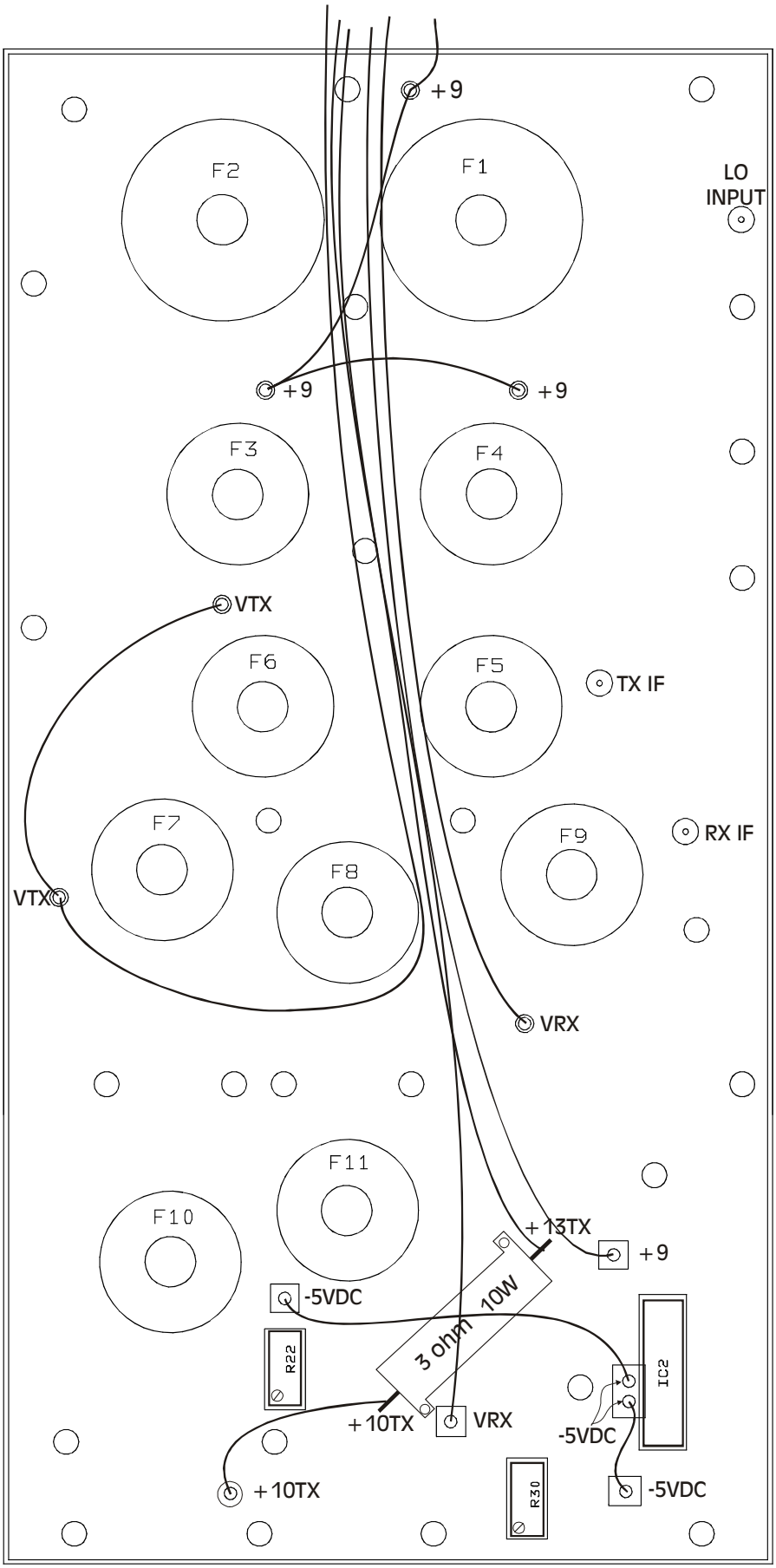
LNA and Power Amplifier Section Hardware

4- 3-48 x 3/16"	2- 4-40 lock nuts	4- silver pins	2- 4-40 x 3/8" Brass
2 - 1/2" Pipe caps	8- 4-40 x 3/16	24" #22 Teflon wire	1- 4-40 feed-thru

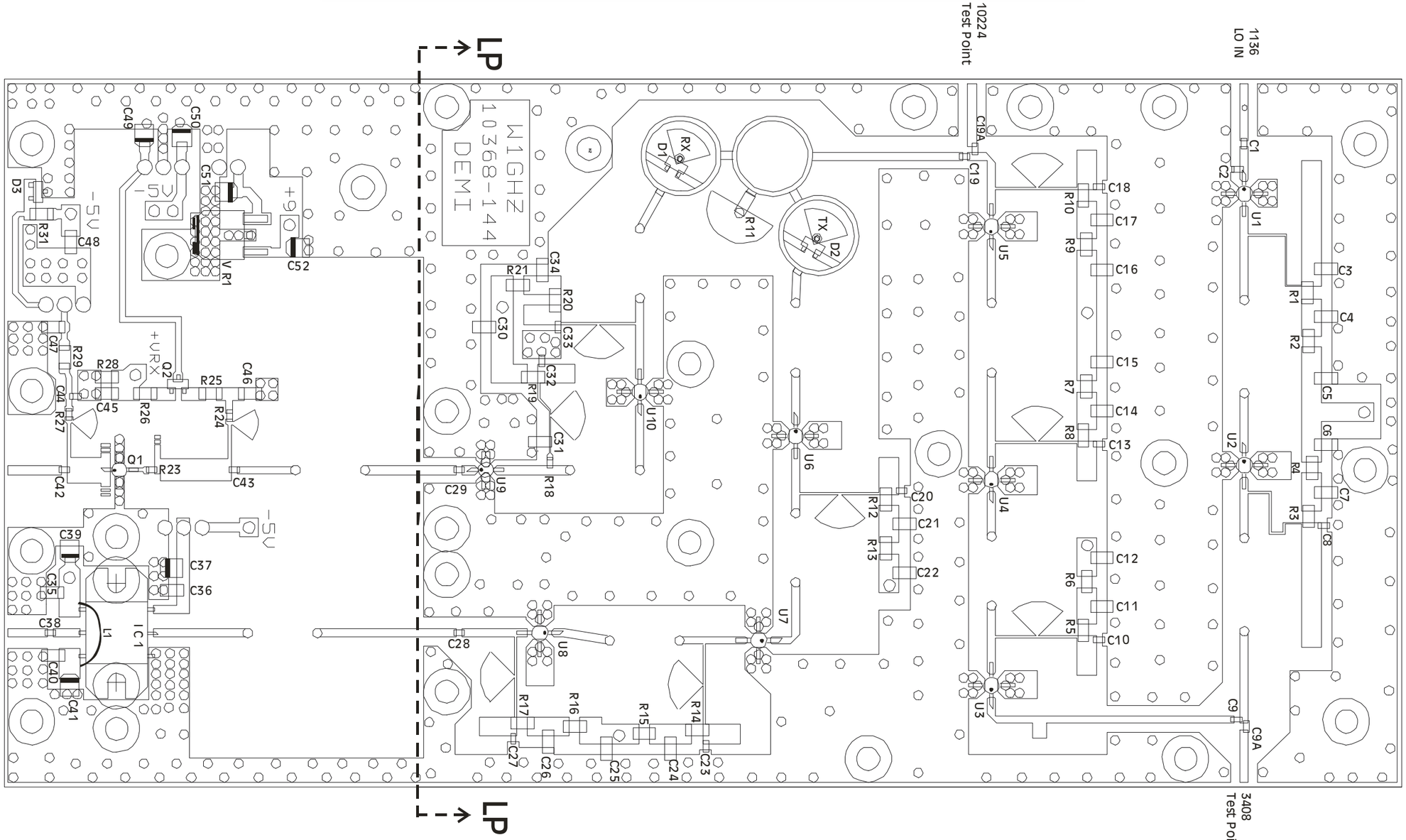


10368-144LP TRANSVERTER SCHEMATIC


DEMI
10368-144
LNA + PA SECTION
C
Rev D
TX RF
+13 TX
~ +10 TX
3ohm
10W
Jumper
C38
C40
C41
C35
C39
C36
C37
R22
IC1
F10
FROM
C28
C50
C49
IC2
C51
VR1
C52
+9V
+5V
-5V
TO C29
F11
C43
R24
R25
R23
Q1
Q2
C46
+VRX
C45
R28
R26
C48
R31
D3
R27
R29
R30
C42
C44
C47
RX RF



10368-144
BOTTOM SIDE ASSEMBLY



10368MHz Top Side Assembly Layout

NOTE - R22, R30, Ic2, 3ohm 10W and F1-11 ARE LOCATED
ON THE BOTTOM SIDE OF THE PC BOARD.