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	< 1.2 dB NF, > 25 dBG	LP: < 3.5 dB NF, >17 dBG
Power Out	> 2.9 Watts.	LP: >10 mw
DC Power	11-16 VDC @ 3 A	LP: < 750 mA
IF Power Input (144 MHz)	1 mW min. - 10Watt max. Adjustable	
Options:	Switchable IF frequency control	

Preliminary:

This kit and requires knowledge of assembling and testing microwave frequency circuits. This document refers to some pictures for ease of assembly. A full color PDF picture addendum document is found on line at: http://01895fa.netsolhost.com/PDF/Manuals/10368-144CK_RevG_Pics.pdf. This on-line addendum contains extra pictures and details that may help answer some assembly issues. Within the text of this document, after a description of a procedure, there may be a number in brackets e.g. **[53]**. This will refer to a picture on the addendum file on line.

The 10368-144CK and 10368 -144LPCK is a complete kit version of the 10368-144 High Power transverter and the 10368-144LP Low Power transverter **[1].** In our Complete Kit version, you receive all three circuit board kits, the 10368K (transverter PCB kit) **[2]**; the TC **[3]** and the DIGILO Synthesizer **[4]** with a prepped extruded aluminum enclosure, mounting plate and all necessary hardware and connectors to get you on the air **[5]**. When completed, you will only need to supply a TR switch, (SMA relay) **[6]** Antenna, and a 2-meter transceiver. Originally, the 10368K PC board kit was designed by W1GHZ and Down East Microwave Inc. but has seen many design revisions since the original was published by W1GHZ in the 1999 Microwave Update proceedings. 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. This latest Revision includes the latest LNA designed by W5LUA as the RX input stage in the high power version transverter only.

In this kit, 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. Along with a +9 Volt 1.5 amp regulator to supply regulated DC power to the transverter and the supporting circuit for the DIGILO that indicates the external / internal 10 MHz source activation.

The DIGILO synthesizer (the base Local Oscillator) provides a nominal +0 dBm output at 3408.000 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. After mixing, the 10368 MHz signals are tested through two separate SMA connectors, TX and RX. The TX and RX connectors are spaced correctly to allow the direct connection of common SMA relay to complete the system and allow the use of a common antenna connection.

Kit Building Success Requirements:

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 assembly types. All kits include the PCB, all of its components, pipe caps, connectors, required hardware, and a CNC machined circuit board mounting plate **[7]**. 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 or a very hot, Hot Air gun! **A standard soldering iron will not work!** Second, you will need a way to detect RF power at 3 cm (10 GHz) 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 seven of them in the low power version and nine in the 3 Watt unit. Both transverters will not work without correct alignment of the filters!

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 power level choice. If you wish to experiment with different gain stages and add-on filters and amplifiers, do so 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! (or at least close!)** If you have any questions about this policy please consult us first before proceeding with your own design or modification.

Pre-Assembly:

Basic tools that are required besides hand tools such as a #1 Philips head screw driver, cutting pliers, tweezers, and a minimum of a 25 watt solder iron with a fine wire solder are a suitable heating device to "Sweat" the pipe cap filters to the transverter's circuit board and a hot plate to warm the pallet assembly. The aluminum pallet provides excellent structural support of the PCB but will "heat sink" just about any general consumer's solder iron not allowing clean smooth solder joints on the surface mount components. Warming the pallet on a hot plate while soldering the components is the best way.

The TC board which has most of the surface mount work completed requires stuffing the board with the leaded components and general soldering. It should be assembled 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 but only should be utilized if you require more than 30 dB of system RX gain. The TC allows 1 mW to 10W common or separate IF input/output that requires the proper configuration during assembly. It's your choice.

The regulated or unregulated voltage is switched to supply any additional stages such as a LNA or power amplifier and 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 DIGILO can be experimented with before the main transverter assembly is started but is not required since they are pretested assemblies. You may set the frequency required for your transverter's conversion scheme and according to the DigiLO's document. It can then be tested for frequency, output power and spurious if desired with the correct voltage applied and coaxial connection. Do not operate it without a 50 ohm load on its output. The complete manual is found on line at our website here: **http://01895fa.netsolhost.com/PDF/Manuals/digilor18_9_1.pdf** and includes extra pictorials to illustrate that may help in assembly. Please take time to review

When configuring your system, please note that DEMI can supply a few external options for the" Low Power" version transverter. If you require a “State of the Art“ noise figure, a 3 cm PHEMT preamp designed by DEMI and W5LUA can be purchased. It is part number L3-2LNAK (board kit) or a complete kit version is L3-2LNACK. This is the same circuit design used in the High Power version transverter. If you decide to increase the output power of your Low Power version transverter, the 3-3PACK and 3-3PAP 3 watt amplifier is available if you don't have access to the surplus market of TWTs and other solid-state amplifiers. DEMI also stocks a basic sequencer, and other accessories that will enable you to complete your 10368 MHz high performance system.

Assembly:

1. 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 this microwave transverter. ------ Last chance? OK!

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 kit, 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 Low power (LP), version or the standard High Power (HP) version transverter.

2. Now, let's assemble 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) and push it through all of the filter probe holes and wiggle it around a bit to be sure it clears 14 holes total for the LP and 18 for the HP board **[8]**. Use the component placement document to identify the filter pin holes. **Do not make the holes larger or drill out!** Just be sure the pin fits in all of the holes and they are clear. Also, look very closely for shorts from the ground plane side and be sure to clear on the top side.

3. Install the PCB to the mounting plate **[9].** 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. Tighten a few

random circuit board screws around the pallet and check for alignment with a visual inspection of the clearance holes in the pallet. They are for the wire and Coax connections on the circuit board. Adjust the board to center the via pads in the pallet holes then tighten all screws. 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**[10].** The head of the screw belongs on the pallet. The screw head is on the pallet, the spacer is between the PCB and the nut. Tighten them and recheck all circuit board screws.

Then cut the excess material off the board edges with a sharp knife **[11]**. If you find yourself cutting too much of the metal of the board, the board is not centered on the pallet. You may use the pallet as a guide. It is possible that some excess solder from the foil to board attachment is in the way. If so, remove with the knife or soldering iron tip. When complete, the board edge to pallet edge will now act as the alignment if the circuit board is ever removed from the pallet. Then clean up the connector panel edge. Be careful not to cut the copper foil in installed. Again, use the pallet as a guide.

4. It is now time to prep the pipe caps. All off the pipe caps have been drilled, tapped and surfaced. Check for remaining burs inside and outside of the caps **[12]** and 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 **[13]**. The pallet will hold them in position during soldering.

5. 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 pipe caps **without the pipe cap hardware.** Include your 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 pipe caps requires a propane gas torch or a very hot "Hot Air System". 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.

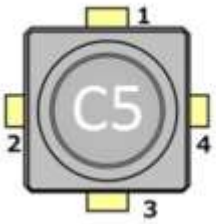
6. Starting with any pipe cap, flux the area within the pallet **[14]** and flux the open end of the cap and place it into a pallet hole **[15]**. We prefer to use "No Clean Flux" for the reason stated in step #5. Do all of the pipe caps. Now heat any cap with the torch being careful not to allow the flame to come in contact with any bare circuit board **[16]**. Remove the flame and flow the solder into the corners of the pallet hole around the pipe cap joint. Re-heat if necessary. Repeat this process on the rest of the pipe caps until finished. After a few are complete, the soldering becomes faster because the pallet is now pre-heated.

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. Allow it to cool completely before touching.

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. If so we suggest removing the PCB mounting screws and cleaning everything completely

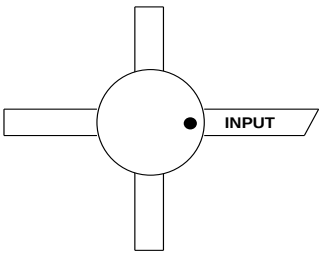
if you feel that there could be a flux invasion problem but understand it is not an easy task. Only do this if you absolutely need to. Be careful not to flex the circuit board excessively. Also, do not trim any excess solder or re-flow a filter when it is not mounted to the pallet. It will cause a misalignment and the PCB will not fit flush to the pallet when re-assembled. If you find a bad joint, re-assemble 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 on the placement. Use the component list to determine what components are installed. For the actual assembly, start by installing the MMIC's U1 - U8. 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 120 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.



Pin No.	Pin Name
1	Source
2	Drain
3	Source
4	Gate

FET



N6 MMIC and MGA86576

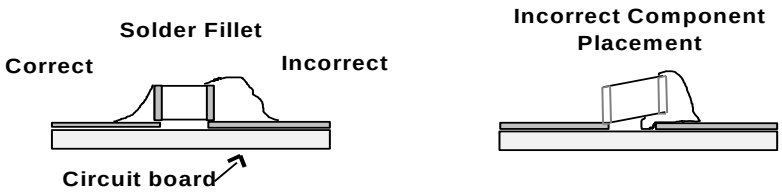
U1 is different in the low power transverter. It will solder flat to the circuit in the Low power (MGA86576) and the N6 in that position will be raised up in the HP unit. Be sure to install the MMIC's as flat and square to the pin alignment as possible **[17]**. U2-U8 may be installed the same way centered above the holes in the board but he bodies of U2- U8 will fit squarely in the PCB holes if flipped upside down. If you decide to do that, bend all four legs down before soldering. Be sure to flow the solder on all of 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 8 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.

If you have the HP version transverter, Install Q1, Q2 and VR1 paying attention to the component document alignment. Use the same technique for Q1 and Q2 as U8 **[18]**. It is important to get a good source lead to ground connection on the FET's and a ground connection

on the VR1 regulator so it may require some pallet heating. You could remove the mounting screw near VR1 before soldering the ground tab.

8. Using the pictorial below as a guide for proper SMD soldering install all of the other surface mount components in any order with the noted exceptions: **[19]**



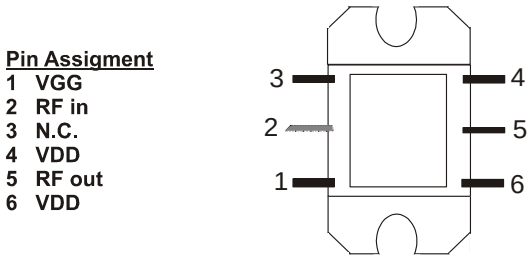
Proper SMD Assembly.

Do not install R8, R18, D1 and D3
R38 is installed in the LP version and shorted in the standard version.
R23 is installed in the standard version only after cutting the trace for installation.
Verify the correct value for R12 depending on version of transverter.
It is easier to install all of the resistors in the low power section of the transverter first.
Install C47 in the "A" position.
You will need to remove the 4-40 screws to install C16 and VR1. Replace when soldered.
R3, R8 and R9 are just jumpers
If you have the HP version transverter, do not install IC1 and the jumper.

9. Install all of the filter probes (the silver pins) after cutting to the correct size **[20]**. The length indicated is the total length including the head. Cut 10 pins at 1/4" or 0.210 - 0.230". Insert them into F1- F3, F8 and F9, then solder **[21] [22]**. Cut the rest of the needed filter pins for F4 - F7 (there are extras!) to a total length of 3/16" or approximately 0.180" - 0.190" (F1 and F7 not required for LP version). Insert pins 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.

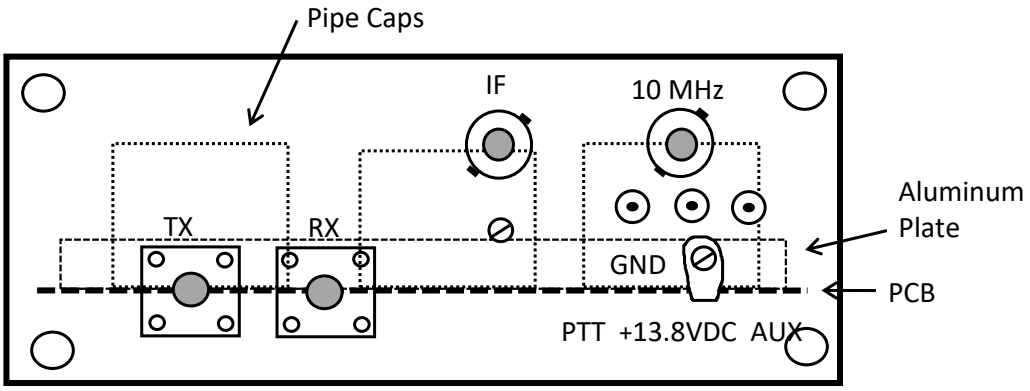
Install the 4-40 feed thru connector as shown on the bottom side pallet drawing (the filter side off the pallet). The pin will come through the board near C31. Solder it to the pad and with an ohm meter, verify it is not shorted to ground. Now, take the time to clean any excess flux off the complete assembly with any cleaner you desire. It doesn't need to be "Spotless" just not coated with excess flux.

10. With the clean pallet assembly, clear the foil in the mounting holes of IC1. Then, review the FMM 5061LV pictorial for pin numbers and apply a very thin light of thermal compound to IC1 being careful not to smear any on the circuit board. Install with 3-48 screws. Be sure of the alignment, tighten the screws and then solder the leads. The Copper foil will stretch enough and conform to the installation. Now Fabricate the Ferrite bead jumper and install between both Drain pins (4 and 6) keeping it as short as possible as shown on the component placement document **[23]**.



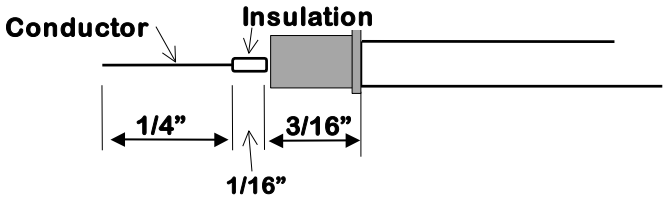
FMM 5061LV

11. Install the rear panel to the PCB by first prepping two SMA connectors so that the Teflon will not interfere with the circuit board **[24]**. 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. Look for any gaps or bulges between the pallet and the panel. If so, remove panel and re-trim excess circuit board without cutting into the foil. 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. A complete flush connection of the pallet to the panel is the most important part of this kit.



Insert all of the 8-32 feed thru caps with a #8 ground lug on each by bending them in a right angle for latter attachment. Then install two BNC connectors. Be sure to install the lock washer/ ground lug on the BNC connectors with the lugs positioned towards the RF connectors **[25]**. Then Solder the SMA center pins.

12. All of the coax connections need to be made next. Start by cutting two 7" pieces (RXIF and TXIF) and one 4" piece (LOIN). The best way of making this coax assembly will result in something that looks like the next diagram. 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 below. 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 and /or wick off **[26].**



Prepped Coax details.

Once the coax is prepped, do a trial 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 **[27]**. 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. When complete install D1 and D2 on the RXIF and TXIF. Be very careful with the lead alignment of the diodes.

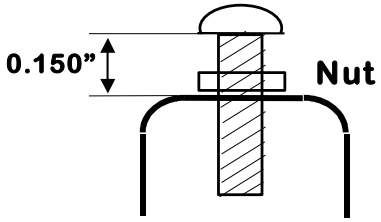
13. On the HP version only as shown on the bottom side pallet drawing, install R35, R36 and IC2 through the pallet into the board. Cut off excess leads. Then install the 3.3 ohm 10 Watt resistor with two 3-48 x 3/16" screws. Now using the supplied #26 Teflon wire, cut, strip and tin the ends of a 2.5" wires and connect it to both -5VDC points. All wires come in through the back of the pallet through the 1/8” holes **[28].** They are soldered to the circuit board where indicated on the bottom side assembly diagram. They are the via holes with an "X". They can solder to any part of the circuit board attached to the via.

The rest of the #26 wiring is labeled VRX, VTX, and +9 on both transverters. Pass one end of the wire through the pallet in all of the indicated holes shown on the bottom assembly document and solder. Cut, strip, and tin as before then insert one end of it through the hole and solder to the board. After routing the wires as shown, cut them so they extend a minimum of 3.5" past the end of the pallet.

Connect the +10TX connection from the feed thru to the10 watt resistor with a 1” #22 wire and connect a 9” #22 wire to the other end of the 3.3 ohm resistor.

Wire the three rear panel feed through as follows. The center is connected to a 11" of #22, the other two are connected to a 11" of #26. All wires should be routed as shown on the bottom assembly **[41].**

14. Install all of the filter hardware now. The screw heights may be pre-set for tuning. Examine the next pictorial. Set the 4-40 brass screws and stainless locking nuts to approximately all the same height of 0.150”. The lock nuts should be "snug" but not locked.

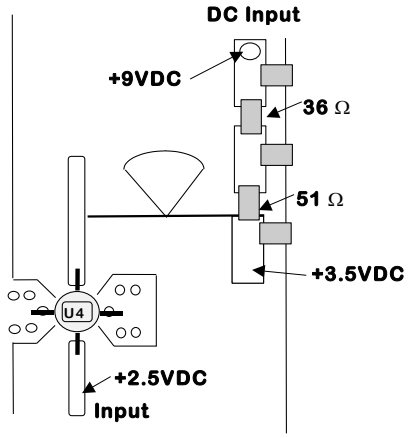


Re-inspect all solder connections. Touch up what is questionable. Now review the test procedure before proceeding. Using the assembled unit for a "Dry Run" may clarify some of the testing requirements that may have been in question when first read.

***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 – U8, 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 U8 MMIC (U1 in the low power Transverter) 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 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, U6-U8, the TX stage, U3– U5, and the RX stage to U1-U2 only. Do not apply voltage to the +VRX wire attached to LNA section in the HP version



Resistor Network with Voltage Points

U1 in the Low power transverter is different. 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 HP version transverter, the LNA/PA section should be tested by verifying the operation of IC2 first. Apply 9 VDC to the +9 and check for –5V (and it may be as low as – .7 VDC)

where indicated on the PCB. If OK, verify that the -5 V is on Pin 1 of IC1. If not adjust R35 to obtain a maximum negative voltage. If -5 V cannot be obtained, examine the circuit for defects.

Check the LNA section by verifying a negative voltage on both Q1 and Q2 gates or inputs. The voltage is limited by D2 and R4 and will not exceed -0.8VDC (or less than negative 1 volt) but could be Zero volts. Adjust R36 so the both voltages are around -0.350 VDC. If it will not adjust, seek the issue. It is either a short or open somewhere.

If OK, check the +5 V where indicated. It may be as high as +7.5 VDC. If OK install R8 and R18 then connect +9VDC to the VRX wire that connects to R13 and R15. Measure the voltage at the flag by R22 and adjust R36 to obtain between +2.0 and +2.2 VDC. Then verify approximately the same on the flag by R19. Remove the VRX voltage when complete. 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 DIGILO will be used as the 3408 MHz oscillator. If a 145 MHz IF is desired, select the correct DIGILO frequency. See its Chart. If anything other than a 2M IF frequency range is desired, you may select any DIGILO frequency that will be multiplied by 3 to produce the correct final LO frequency including an IF in the 70 cm band. Slight adjustment other than what is detailed in this procedure may be required and RF test equipment such as a Spectrum Analyzer may be required.

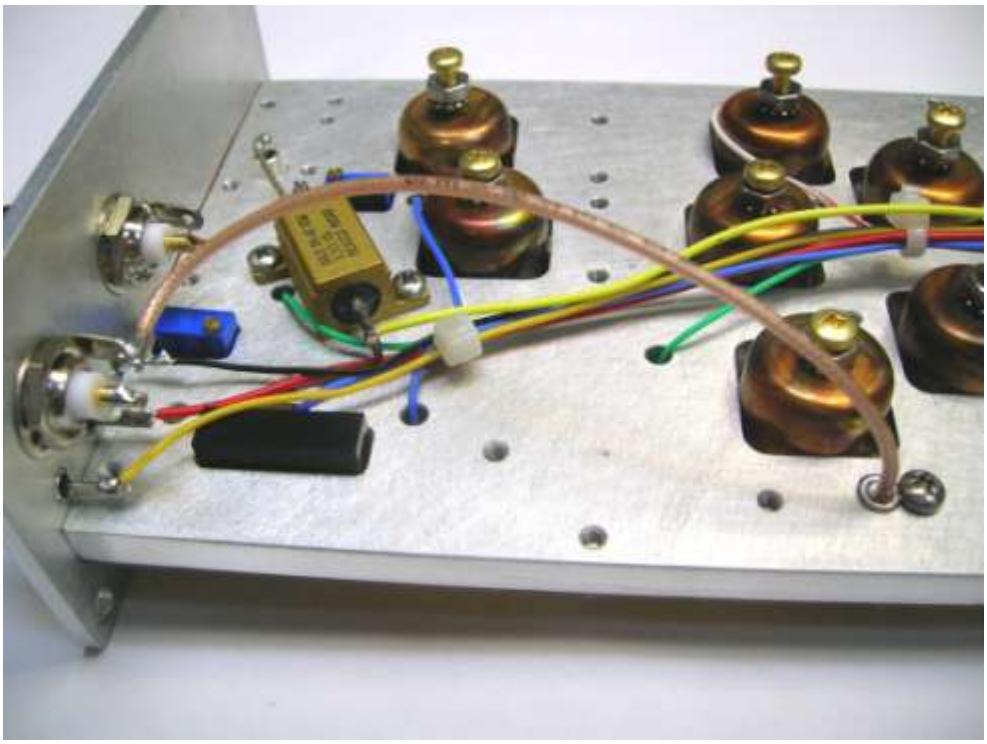
1. Connect a trimmed SMA connector to the 10224 TEST POIT. Be sure that it is a flush mount **[29]**. Verify that C47 is in the test position (C47A on the component diagram). Install the DIGILO as shown right by installing two 4-40 x 1/4" HEX standoffs and lock washers in the pallet with two 4-40 x 3/16" screws for the board. Connect the LO input to the RF OUT of the DIGILO by soldering the coax. Make this coax connection as short as possible **[40]**. The 10 MHz input is not required for testing. Set the correct jumpers (if not done previously) by shoring all 6 positions except #4 for 10368 = 144 MHz. If you desire anything different, make the selections on the DIGILO now. Make the DC connection to the DIGILO's +V with a 3" length of #26 wire **[39]**.

2. Position the transverter so the LO Multiplier filter adjustment screws are easily accessible. Connect a RF power-detecting device to the 10224 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 DIGILO and the two +9 wires on the transverter. If the screws are pre-adjusted correctly, some output power should be detected. Adjust F9 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 F8 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 between + 3 and +10 dBm. Greater than is OK, less than is not. If test is OK, remove +9VDC and go on to step 2. If the LO power is low, check the following in this order: Low DC Voltage, High DC voltage (over +10VDC). Incorrect bias on U6 - U8, or C47 not in the test position "A". Low level from DIGILO (coax short on either board). Also, revaluate the coax length between the

DIGILO and transverter. It is sensitive to length. As short as possible is required. If the LO power is still below +3dBm, a snow flaking may be required in the coax input to the transverter. A simple probing or touching will verify this. Add if required. Then consider the filters being tuned incorrectly (wrong starting point). Incorrect probe length, probe missing, or probe shorted to ground. Suspect defective MMIC last if voltages are correct. Always question your construction. When test is complete, remove the SMA test connector and reposition C47 to the "B" position.

3. Further testing of the transverter can be completed to verify tune up and operation. Both the TXIF and the RX IF cable can be temporally connected to the BNC **[43]** connectors and the tune up procedure that is described in the ***Testing:*** may be followed except the individual TXON and RXON voltages are applied as in the ***DC Testing*****:** section. For the purest, this is the best method to tune up the RX section. This will allow the ability to do any trimming or snow flaking to optimize the performance nut---



very little can be accomplished if the transverter is assembled correctly. But if you have a Noise figure meter, the Gain and Noise Figure can be optimized. Connect the IF coax as show and test and tune as specified in the test section. Then when complete, follow the next steps to complete the assembly. If you decide to test and align the TX side, do not drive the TX mixer with more than +10dBm.

4. For the complete testing of the transverter unit, the complete enclosure assembly should be finished. Start by examining the two enclosure halves. Notice that in one half, there are four taped holes. This is the transverter side. Install the microwave absorber in the bottom of the enclosure as shown away from the end with the tapped holes **[30].**
Then unbolt the DIGILO and install the pallet assembly in the housing using the two 4-40 x 9/16" screws and 1/8 spacers that are on the pallet. 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. Now start the two flat head panel screws (black in color). When everything lines up and you are sure about the spacers being in place, tighten all four screws.

5. 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 (pre assembled earlier) and proceed with the installation. Be sure of your IF configuration and continue to the next step below after completion **[31].**

6. If you are assembling the standard unit, the wire from the 3 ohm, 10-watt resistor should be connected to the +13TX via on the TC board. 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.

Testing:

1. Start by getting the same equipment ready that was used for the LO testing. Preset the IF pots on the TC board by turning the TXIF counter clockwise and the RXIF clockwise. Connect 13.8VDC to the transverter and switch the power on. The green LED should light. If you connect an external 10 MHz source, the Loc light should change to Green. If not it should be Red **[32].** 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 and the +13TX will measure the transverter's input voltage.

2. Even though the fixed attenuator has been predetermined, for initial testing, the TXIF pot should be adjusted to the maximum attenuation position (full counter clockwise). 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 after verifying the correct attenuation for the version of transverter you have. Position the transverter so the filter adjustments are accessible **[33]**. 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 F4, F5, F6 and F7 if you have the standard version 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 to 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 the limit of the version transverter you have at the 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 output power is less than specified, check the following in this order: Low or High voltage. Verify that the DIGILO is connected to +9VDC. Check C47 to verify it is installed in the "B" position (need to remove pallet to check this). IF drive power level either too low or too high or wrong configuration on the TC. Is the IF coax cable short or open? Is C45 (LP) or C48 (standard) damaged from the connector installation? Filter Probes (length, shorted, missing?). D3 blown from excessive drive. Suspect defective MMIC last if their voltage is correct. Always Question Construction! **AND the 3 watt MMIC is never bad unless installed backwards!!**

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 matter 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 and is injected directly into the RXRF port of the transverter then adjusted for maximum gain into a 144 MHz receiver. Then we test and optimize it with a noise figure meter **[34].**

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 -30 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 F2 and F3 filters (and F1 if you have the standard transverter) 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. The LNA bias adjustment should not be adjusted unless you are testing the transverter with a noise figure meter and then in conjunction with additional filter tuning.

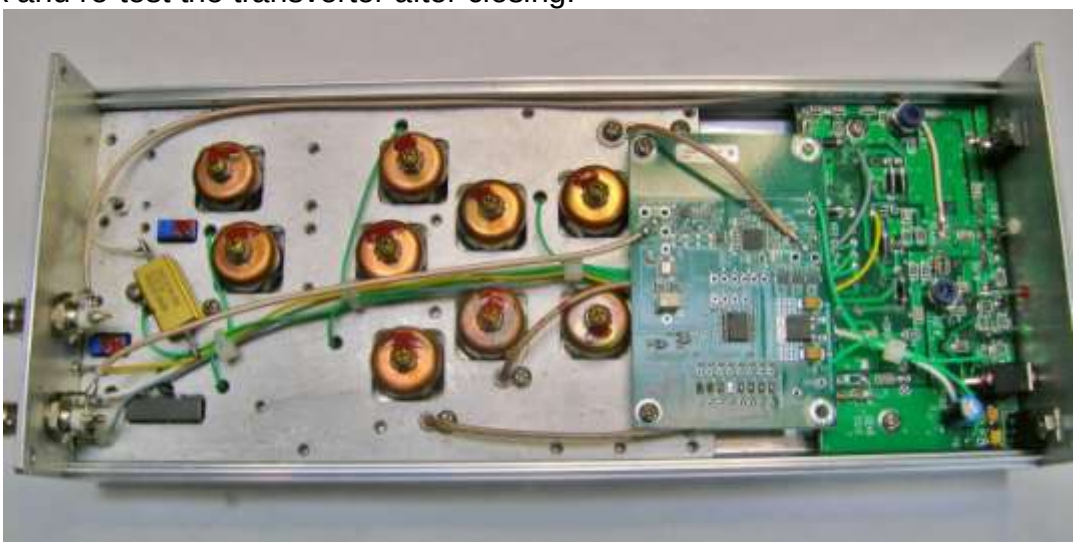
If RX gain is less than expected, check the following in this order: Low or High voltage. Verify if the RXIF gain control is at minimum or the RXIF gain stage on the TC board, if installed, is working correctly. Verify the input signal level. If it is too high, it could cause harmonics in the Q1/Q2 stage allowing the misalignment of the filters. Is Q1/Q2 oscillating? Is the IF coax cable short or open? Is C44 (LP) or C28 (standard) damaged from the connector installation? Filter Probes (length, shorted, missing?) Is D1 blown from excessive drive during the TX test while not keying the PTT line? This would also cause damage to the RXIF Gain stage but suspect defective MMIC on the Transverter board last if the voltages are correct. Always Question Construction!

Completion:

When you become satisfied with the RF operation of the 10368-144, you need to install the External 10 MHz. coax to the DIGILO and wire it's LOC detect line to the "IN" via on the TC board located next to the ON/OFF switch. Then test by connecting a external 10 MHz source and verify that the LOC led changes from RED to GREEN.

Now, you may want to bundle all of the control wires and coaxes together to make a neat appearance (*see next picture*) **[35].** It is not necessary, but it will contain the wires to make closing

the enclosure easier. 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 any wires or coax and 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 in the TC board. If the transverter is to be used in a high performance terrestrial or EME set up, a Low Noise Amplifier should be considered **[36].** 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 a additional filter and/or isolator **[37].** 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 any 2M transceiver in its standard form if it doesn't exceed 10 watts. Higher output power transceivers are not recommended unless modified or adjusted to prevent damage. 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 its 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 components or equipment properly. By reusing, returning or using proper collection points which are designated for it.

10368-144LP Transverter Board Parts List

All components are Surface Mount. All resistors are 1206 size unless indicated. All capacitors are 1206 size except for 1.0pF are 55mil ATC. All others are as indicted. Any substation of values or types is at users own risk.

C3 1.0 pF	C29 1.0 pF	C44 1.0 pF	R12 51 or 100Ω(LP)	R34 36 Ω
C5 0.1μF	C30 0.1μF	C45 1.0 pF	R14 36 Ω	R 35 100 Ω
C8 0.1μF	C32 100 pF	C46 5 pF (0805)	R20 51 Ω	R38 short or 51Ω(LP) (0805)
C9 1000 pF	C33 100 pF	C47 1.0 pF	R23 36Ω Not in LP	U1 N6 or MGA86576 (LP)
C10 1.0 pF	C34 0.1μF	D1 HSMS8202	R25 51 Ω	U2 N6
C14 1.0 pF	C37 1.0 pF	D3 HSMS8202	R26 36 Ω	U3 N6
C15 100 pF	C39 1.0 pF	R1 51 Ω	R27 75 Ω	U4 N6
C17 0.1μF	C40 0.1μF	R2 51 Ω (0805)	R29 51 Ω	U5 N6
C20 0.1μF	C41 1.0 pF	R5 36 Ω	R30 36 Ω	U6 N6
C22 100 pF	C42 0.1μF	R6 36 Ω	R31 51 Ω	U7 N6
C24 1.0 pF	C43 100 pF	R7 51 Ω	R33 51 Ω	U8 N6

10368-144 3 watt version additions.

All chip resistors are 1206 size unless indicated. All 1.0pF capacitors are ATC 55mil. 4.7μF is a tantalum and the white band indicates positive polarity. Any substation of values or types is at users own risk.

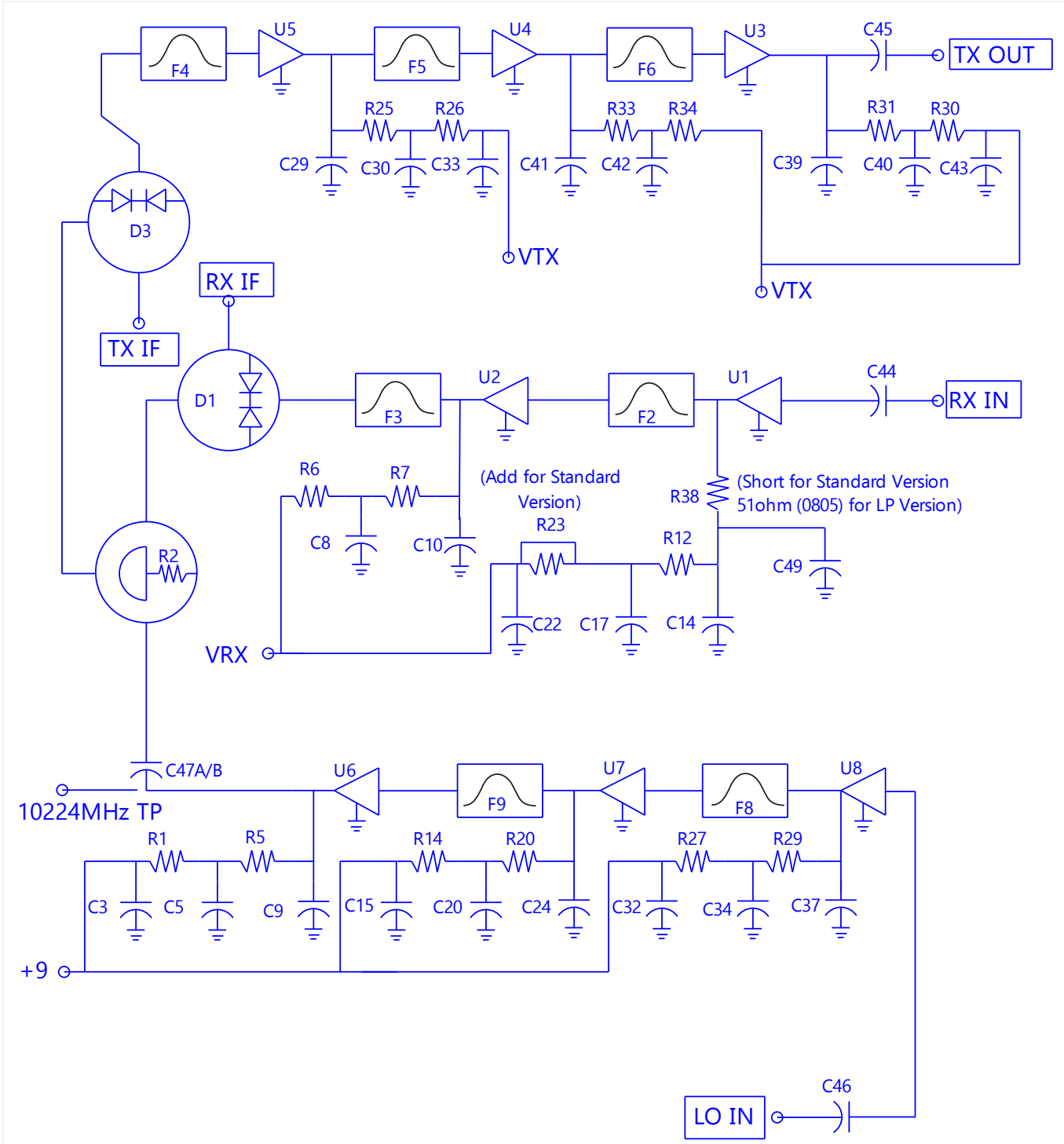
C1 4.7μF	C21 100pF(0805)	C49 0.1μF (0805)	R8 0Ω	R22 51Ω(0805)
C2 4.7μF	C23 0.1μF (0805)	C50 4.7μF	R9 0Ω	R24 15Ω(0603)
C4 4.7μF	C25 0.1μF (0805)	D2 MMBD914	R10 470Ω	R28 51Ω(0805)
C6 0.1μF(0805)	C26 1.0 pF	FB1 Ferrite Bead	R11 240Ω	R32 51Ω(0805)
C7 4.7μF	C27 1.0 pF	IC1 FMM5061VF	R13 5.1KΩ	R35 200Ω Pot
C11 100pF(0805)	C28 1.0 pF	IC2 NMA0505S	R15 5.1KΩ	R36 1KΩ Pot
C12 100pF(0805)	C31 4.7μF	Q1 CE3512K2	R16 51Ω(0805)	R37 3Ω, 10 W
C13 100pF(0805)	C35 0.1μF (0805)	Q2 CE3512K2	R17 470Ω	VR1 78M05
C16 100pF(0805)	C36 1.0 pF	Q3 MMBT3904	R18 220Ω	
C18 0.1μF (0805)	C38 0.1μF (0805)	R3 0Ω	R19 51Ω(0805)	
C19 0.1μF (0805)	C48 1.0 pF	R4 5.1K	R21 51Ω(0805)	

10368-144LP board and enclosure hardware Parts List

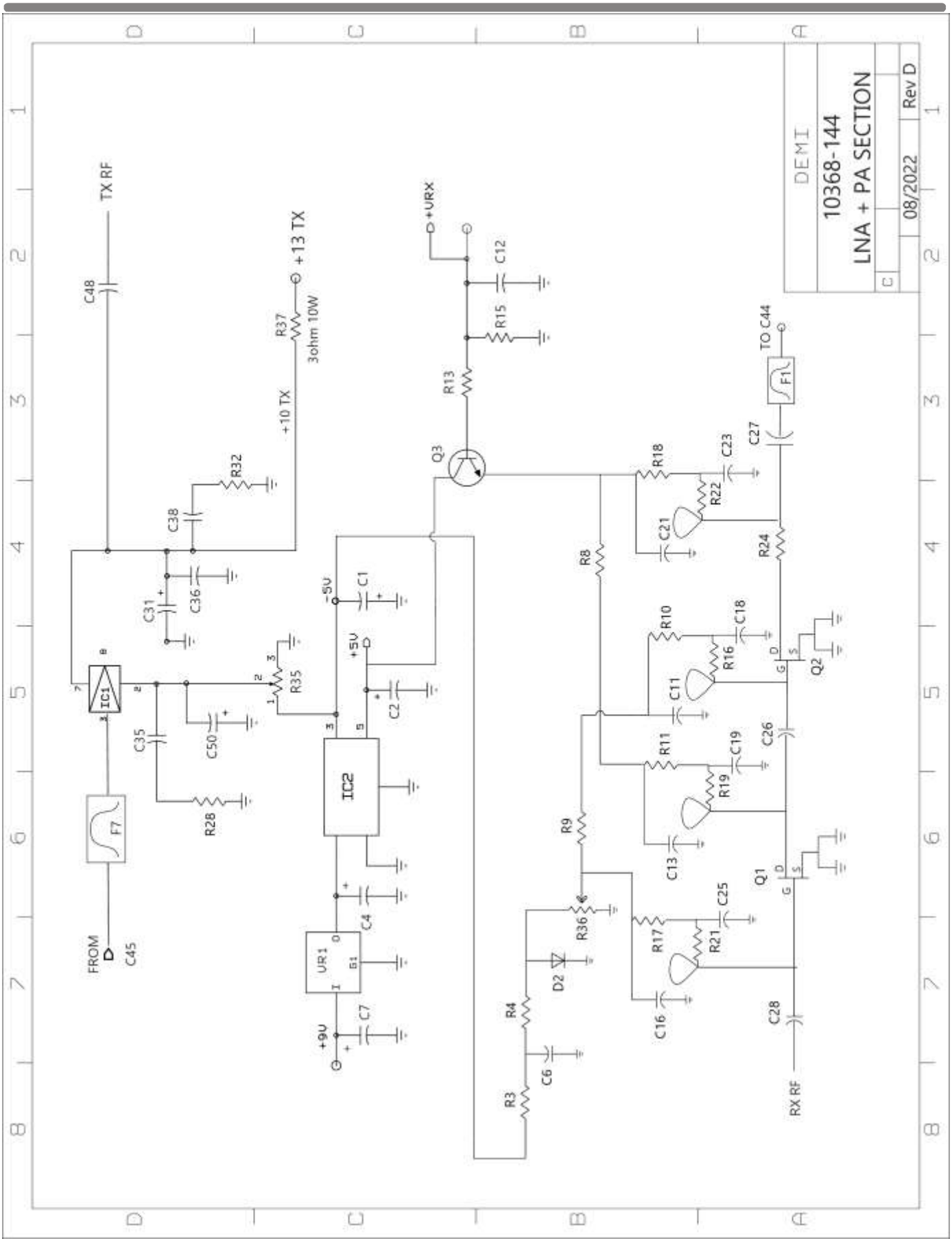
7 1/2" Pipe Cap	8 4-40 locknuts	20 silver pins	1 machined 1/4" plate
22 4-40 x 3/16	5 3-48 x 3/8"	3' RG-188 coax	3 SMA connector
6 3-48 x 3/16"	8 4-40 x 3/8" brass	3' # 28 Teflon wire	2 BNC connector
2 4-40 x 3/8"	1 # 4 Ground Lug	1 SPDT switch	2 Machined End Plates
2 4-40 x 9/16	4- #4 split washers	3 8-32 feed thru	8 Flat head screws
3 4-40 x 1/4"	2- #4 x 1/8" spacer	4 1/8" x 3/16" rivets	
1 Hole plug	2 4-40 nuts	4 1/4" threaded standoff	

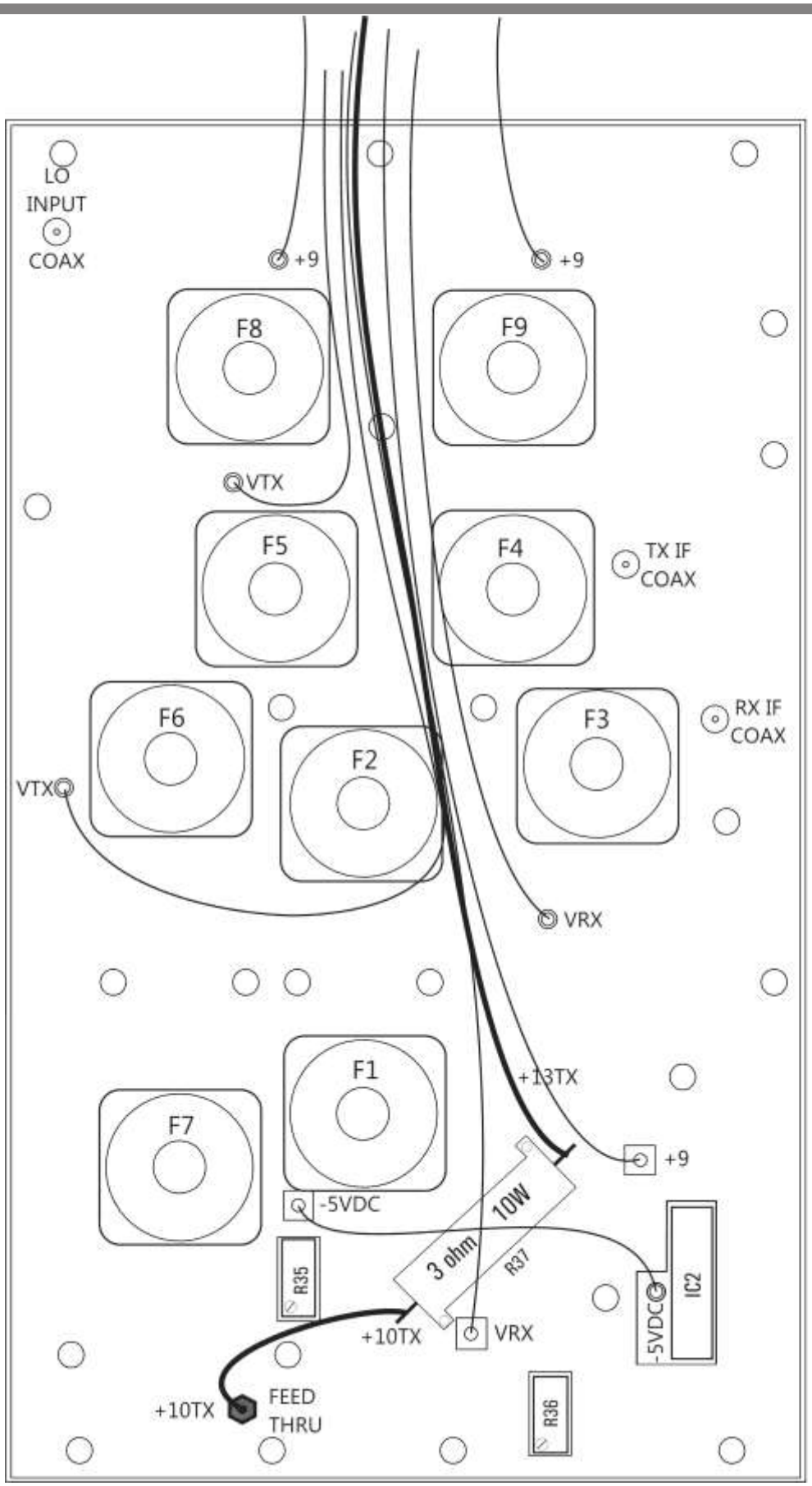
LNA and Power Amplifier Section Hardware

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

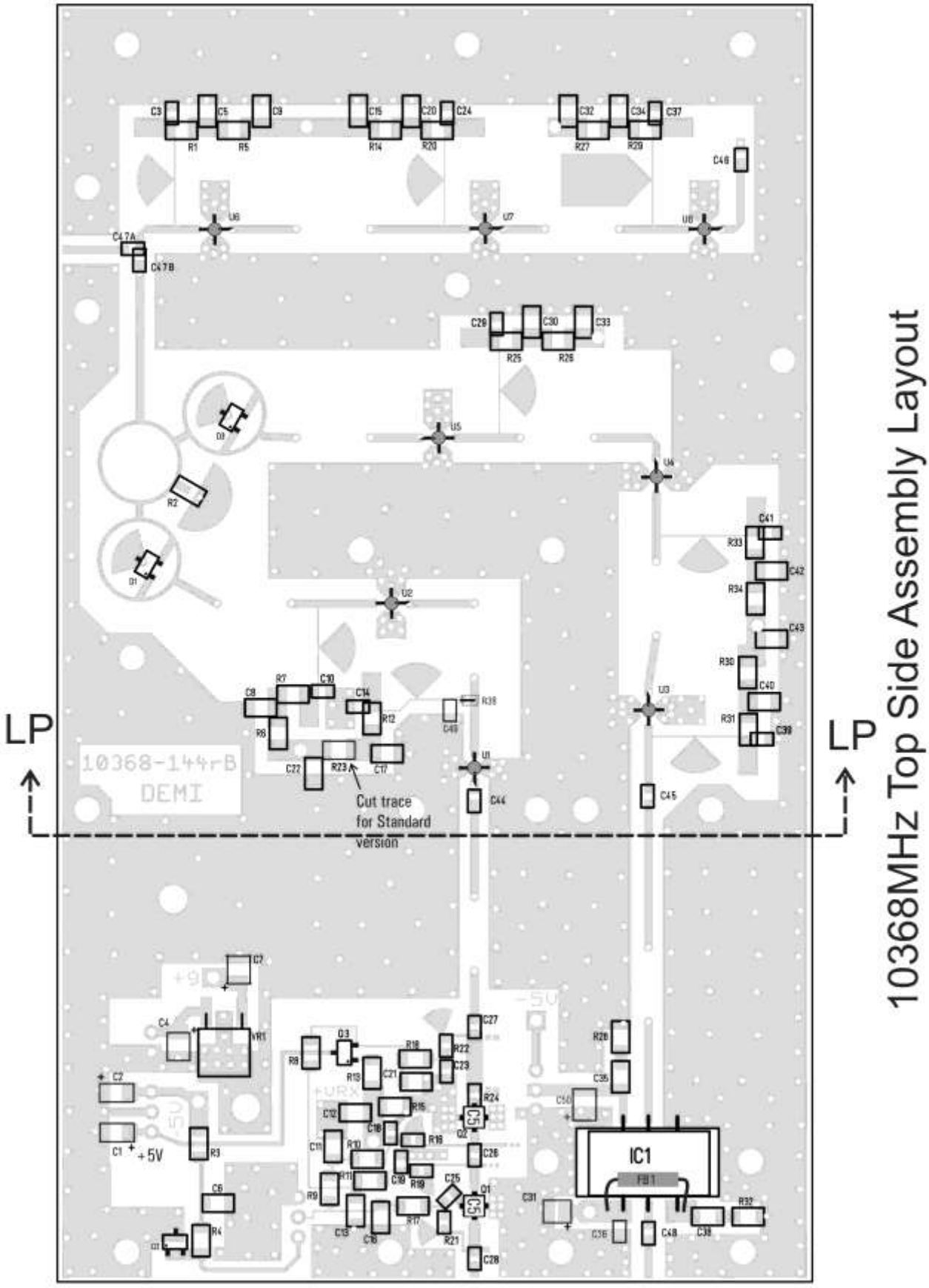


10368-144LPrC TRANSVERTER SCHEMATIC


DEMI
10368-144
LNA + PA SECTION
08/2022
Rev D
TX RF
+13 TX
R37
3ohm 10W
+10 TX
+VRX
TO C44
FROM
C45
RX RF
IC1
IC2
Q1
Q2
Q3
D2
VR1
F1
F7
+5V
-5V
+9V



10368-144 BOTTOM SIDE PALLET ASSEMBLY


LP
LP
10368-144rB
DEMI
Cut trace
for Standard
version
IC1
+5V
10368MHz Top Side Assembly Layout