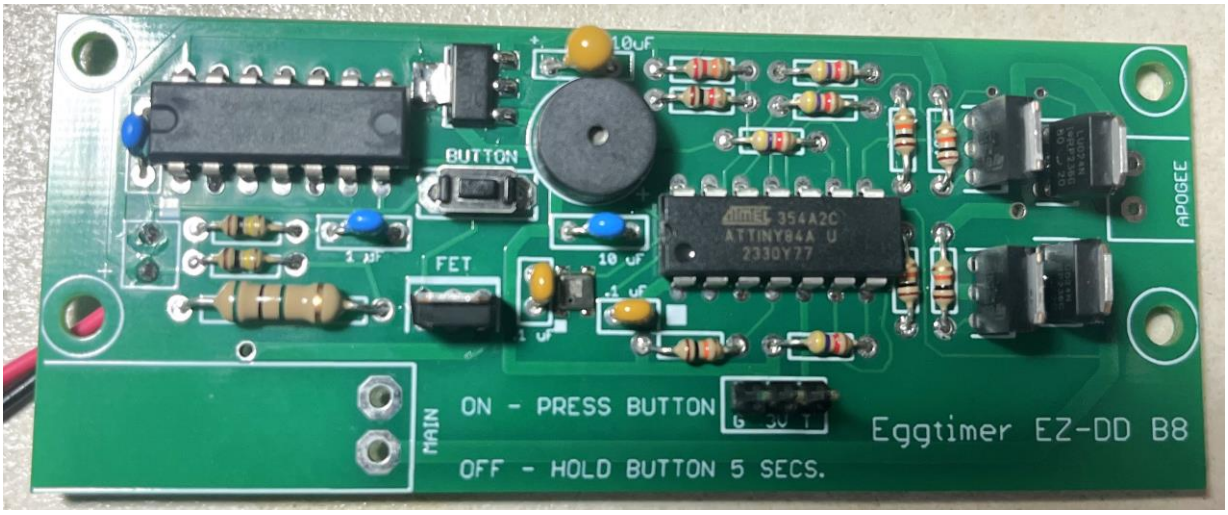
Eggtimer EZ-DD Assembly Manual

Board Rev B8



California Proposition 65 Warning

WARNING: This product contains chemicals (lead) known to the State of California to cause cancer and birth defects or reproductive harm.

This kit includes a special low-temperature ultra-fine leaded solder wire. Including the solder with the kit ensures that you will have solder that can be used to mount the surface-mount parts in the kit. Leaded solders have been used for over a century in electronic assembly, but you should take the following precautions when using it (or just about any chemical, for that matter):

- Do not eat or drink while using it
- Wash your hands after handling it
- Keep it in the protective bag when you're not using it

The MSDS can be found at

https://www.macdermidalpha.com/sites/default/files/sds_pdfs/NA/245ALLOY63_KESTER%20SDS%20GHS%20AMERICAS_ENGLISH%20%28US%29.PDF

The European Union RoHS (Restriction on Hazardous Substances) regulations exempt kits such as the EZ-DD from its regulations, because they are not for resale and since it is well known that hand soldering with non-leaded solder is much more difficult and more damaging to heat-sensitive components.

Before You Start…

• Check the parts against the Packing List in the kit, and let us know right away if anything is amiss.

• Read these directions completely before you start… that often answers any questions that you may have.

• Contact us at support@EggtimerRocketry.com if you have any questions, BEFORE you solder… that may save you a lot of trouble later on.

~~~~~~

Thanks for buying an Eggtimer EZ-DD. The EZ-DD is a dual-output deployment controller, designed especially for the EZ-DD rocket. It separates the rocket just past apogee, and deploys the main parachute at 600'. It also beeps out your apogee after every flight.

Like other Eggtimer Rocketry products, we sell it as a kit, to keep costs down and provide an outstanding value. This means that you have to do a little work, of course, but considering that most hobby rocketeers that would use our products have some degree of electronics expertise, this should not be much of an impediment. If you do not have any experience soldering kits such as the EZ-DD, we recommend that you ask around… chances are that somebody in your rocketry club would be more than happy to assist you for a small bribe (beverages work well!).

About Soldering Your EZ-DD…

Assembling your EZ-DD isn't that hard, almost all of the parts are through-hole parts and are easy to solder. We provide the solder for you, and tell you what kind of soldering iron you need to use, so you can get nice shiny solder joints. If you have never soldered before, you need to learn anyway, because if you are going to do rocketry electronics you're going to be doing some soldering. If you want to get into advanced projects like telemetry, you're probably going to be doing a lot of soldering. We recommend that you get a few small kits from Ramsey or SparkFun, put them together, and hone your skills on them first. There's a lot of fun stuff out there, so go for it!

The EZ-DD uses mostly through-hole parts, and the few surface mount (SMT) parts that we use are large by SMT standards, and are easily within the realm of being hand-solderable. In order to help make your assembly successful, we have included about 1m of very fine (.020"), very low temperature (about 180°C), no-residue solder. This is not the stuff that you get at your local hardware store… it's designed for soldering small temperature-sensitive parts without transferring much heat to the part itself.

Important note about using extra flux with this board: The solder that comes with the kit is Kester 245 63/37 .020", it uses a water-based "no-clean" flux. If you wish to use extra flux with the board, it MUST be compatible. You want a liquid (not paste) water-based no-clean flux. Kester 951 is ideal, if you can get it. Chip-Quik sells little 2ml tubes for about $2 each (unfortunately they sell them in 6-packs, you can't just get one) which works very well. If you decide to add flux, you must use only a tiny amount. A few drops will suffice for the entire board. DO NOT use Rosin Core flux (except as noted), or you will make a mess of the board and possibly damage components. We have built many kits without using any additional flux
~~~~~~

without any issues, the board is pre-tinned to make solder adhesion easier so in general you should not need to use additional flux.

For soldering components on a board like the EZ-DD, we recommend a small pencil soldering iron, about 15W. If you are only going to use it occasionally, Weller makes a decent cheap 12W iron, it's about $15. There is also a similar iron that's sold by ECG. We like those, but the copper tips seem to oxidize and corrode rather quickly compared to some more expensive irons; fortunately, the tips are replaceable and cheap. Better would be a fancier soldering pencil with iron tips; those run about $30, but they'll last forever. The best iron would be a temperature-controlled solder station, they typically start at about $50 for a cheap one and can go to a few hundred dollars if you want to get really fancy. Weller makes a good one for about $50, if you make the investment that will probably be the last soldering iron you will ever need to buy. These solder stations usually have a little well with a tip-cleaning sponge, so they end up taking less room on your workstation too. Get the smallest tip you can find, preferably a small conical tip. It should be just a bit smaller than the width of the processor's pads.

General Assembly Information

We're sure that you are ready get started, but before you do you will need to get some tools together. The tools that you will need are:

___ Low-wattage soldering iron, 15W or less, with a fine conical tip (1/32")
Or a temperature-controlled soldering station

___ Small needle-nose pliers

___ Small diagonal cutters

___ Tweezers to handle the SMT parts

___ A brass mesh tip cleaning "sponge" for cleaning the tip of your soldering iron

__ A tip tinning block

___ A 3x-5x lighted magnifier… not essential, but very helpful

___ A jeweler's loupe or small 10x magnifier, for inspecting the solder joints

___ A well-lighted place to work, preferably with a non-conductive surface,
also preferably not carpeted

___ Some PAPER masking tape (do NOT use Scotch® tape or electrical tape)

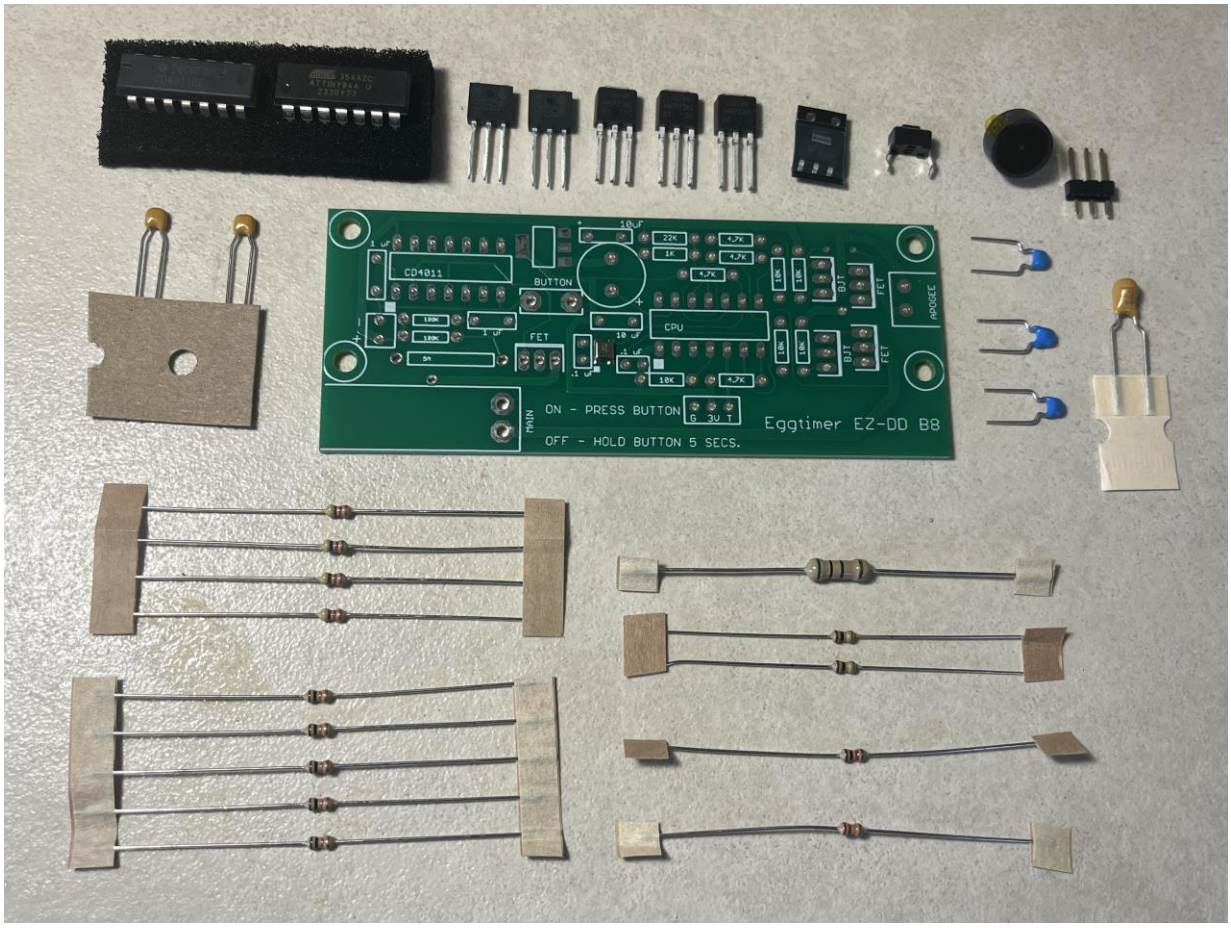
Each installation step has a check-off line, we strongly recommend that you check them off as you go, and that you perform the steps in sequence. We have listed the steps in order to make it easiest to assemble the EZ-DD, deviating from them isn't going to make your life any easier. Each step is pictured, so you can see exactly what you need to be soldering. Looking at the pictures as you go will help prevent you from soldering the wrong thing, or putting something in the wrong way.

Assembling your EZ-DD Altimeter

Step 1: Sort the Components

Before you start soldering anything, you need to lay everything out and make sure that you are familiar with all of components, and that you have everything. (Yes, we ARE human and sometimes make mistakes… if you are missing something, let us know immediately so we can send you whatever you need). You should have the following parts, check them off as you sort them…

__ PC Board with Baro Pressure Sensor
__ ATTINY84A Processor (pre-programmed)
__ CD4011N Quad-NAND Gate
__ LD1117S33 Voltage Regulator (3 pins with tab)
__ (3) IRLU024 N-FET's (3 leads, marked "LU024N")
__ (2) MJG122 BJT Transistors (3 leads, marked "MJD 122")
__ (2) .1 uF MLCC capacitors (marked "104")
__ (2) 1 uF MLCC capacitors (marked "105")
__ (1) 10 uF MLCC capacitor (marked "106")
__ (1) 10 uF tantalum capacitor, marked "106" with a "+"
__ (1) 1K ohm 1/8W resistor
(bands are brown-black-red)
__ (4) 4.7K ohm 1/8W resistors
(bands are yellow-violet-red)
__ (5) 10K ohm 1/8W resistors
(bands are brown-black-orange)
__ (1) 22K ohm 1/8W resistor
(bands are red-red-orange)
__ (2) 100k ohm 1/8W resistors
(bands are brown-black-yellow)
__ (1) 5M ohm 1/4W resistor
(bands are green-black-green)
__ (1) Buzzer, 10mm magnetic
__ (1) Button, 3mm x 6mm
__ (1) 3-pin .1" straight header
__ Solder, 63/37 no-clean (not shown)



Note that some of the components are static sensitive, so you should avoid sources of static electricity while you are handling them. We recommend that you assemble the EZ-DD on a wood or metal surface unless you are fortunate enough to have a high-temperature anti-static mat (don't buy one just to build the EZ-DD, however!) Avoid putting it on plastic surfaces that generate static, and preferably put it together in a room that's not carpeted. That being said, it's very unlikely that you will zap any of the components in the EZ-DD with static electricity, but consider yourself notified of the possibility…

Also note that some of the components are polarized, i.e. it matters which way you put them in. If you solder one of these components in backwards, the effect will range from not making any noise to nothing at all working. It is CRITICAL that you test-fit the parts before you solder, and that you make ***SURE*** that you have them pointed the right direction before soldering. Like the old adage says, "Measure twice, cut once." If you solder a part onto the board incorrectly, it can be a minor pain to remove if it only has two pins, or it can be virtually impossible for something with a lot of pins like the processor. ***The Eggtimer EZ-DD Limited Warranty does not cover incorrect assembly***, so if you mess up badly enough you may end up having to get another kit and starting over; neither of us want that.

There are several different resistors and capacitors, and some of them look alike, so make sure you get the right ones in the right place. They are marked on the boards, but once again you need to make SURE that you have them in the right place before soldering. Unsoldering parts on a small circuit board like the EZ-DD isn't a lot of fun, even if you have a vacuum desoldering tool. It's hard to get all the solder off, and easy to lift a pad off the board. Trust us, we've been there before…

Before you solder anything, make ***absolutely*** sure that you have the correct part and that it is inserted in the board correctly. The board has all of the component values, outlines, and polarities silk-screened on the top, so there shouldn't be any doubt about what goes where and how. Nevertheless, if you have any questions about the assembly procedure, do not hesitate to drop us a line at support@eggtimerrocketry.com **before** you solder the parts to the board. You may have to wait a day for the answer, but it could save you a lot of grief later on!

The Eggtimer EZ-DD Limited Warranty does not cover damage to parts while attempting to desolder them because you inserted something incorrectly. We spent a lot of time making sure that the assembly instructions were clear, but once again if you have any questions about the assembly procedures drop us a line at support@eggtimerrockety.com ***before*** you solder.

Some Assembly Hints…

For the leaded components (which is basically everything except the voltage regulator) heat up the joint for about 3-5 seconds before trying to apply the solder. The joint needs to be hot enough for the solder to flow out on its own… don't try to "melt" the solder onto the joint. You will end up with blobby "cold" solder joints if you do.

Apply enough solder to get a small fillet on the joints and for the solder to flow into the holes. You do NOT need to have large solder joints… they won't work any better, and they're more likely to be "cold" too. See the pictures… if your solder joints look like those, you're good to go.

We recommend that you orient the resistors so that the color bands are all facing the same direction. Having them face the same way makes it easier to make sure that you have the right ones in the right place. We recommend that you have the color bands read left-to right, or in the case of the ones mounted on the short plane of the board bottom-to-top. If you do that, they will match the printed legends on the PC board. See the assembly pictures for guidance.

OK, so let's get started…

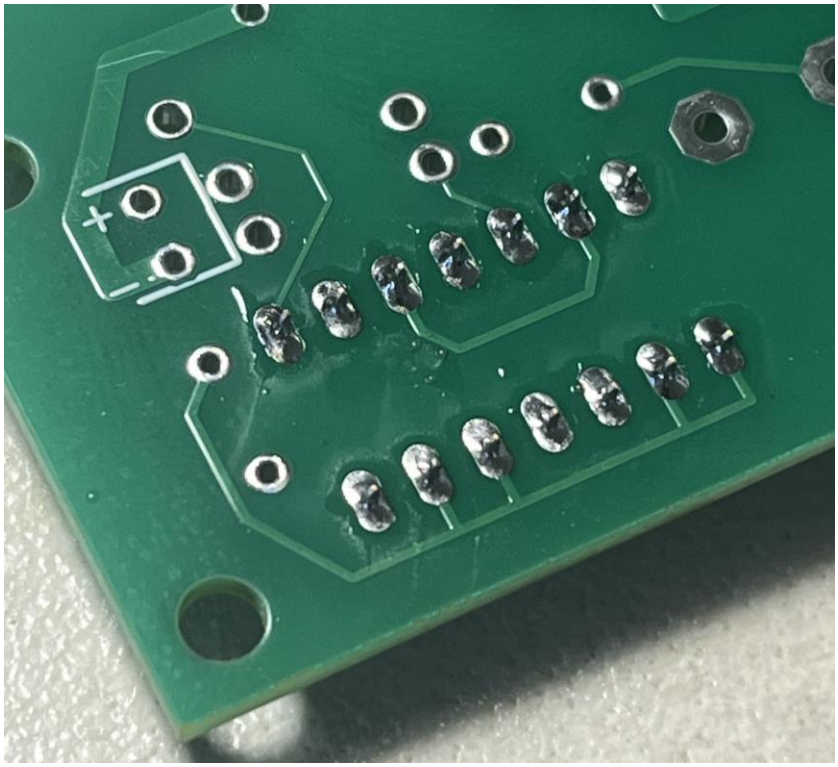
Assembling the Switch Components

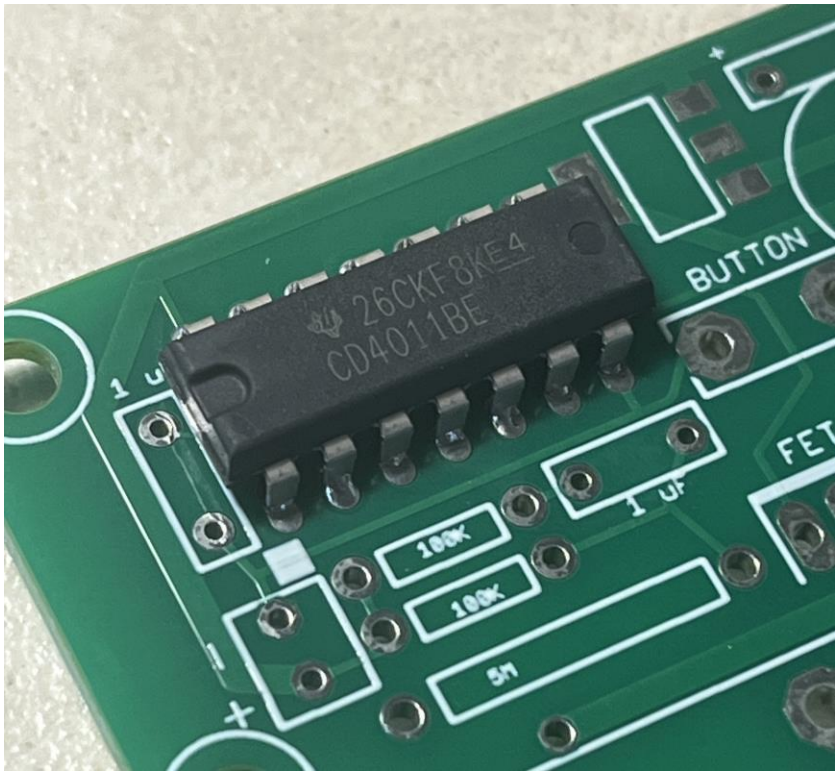
__ CD4011BE (Marked "CD4011BE")

Locate the spot for the CD4011BE, it's a 14-pin chip marked "CD4011BE". Note that there is a notch on the left side. The notch should be on the LEFT side of the pads on the board, and should match up with the square marked on the lower-left side of the pads.

Insert the CD4011BE into the holes on the board, you may have to carefully work the leads into the holes to get them all in. Hold the CD4011BE onto the board with some masking tape, then turn the board over. Solder ONE corner pin, then turn the board over and make sure that it's seated properly in the board. If it is NOT, heat up that one solder joint while gently pushing the CD4011BE chip into the board.

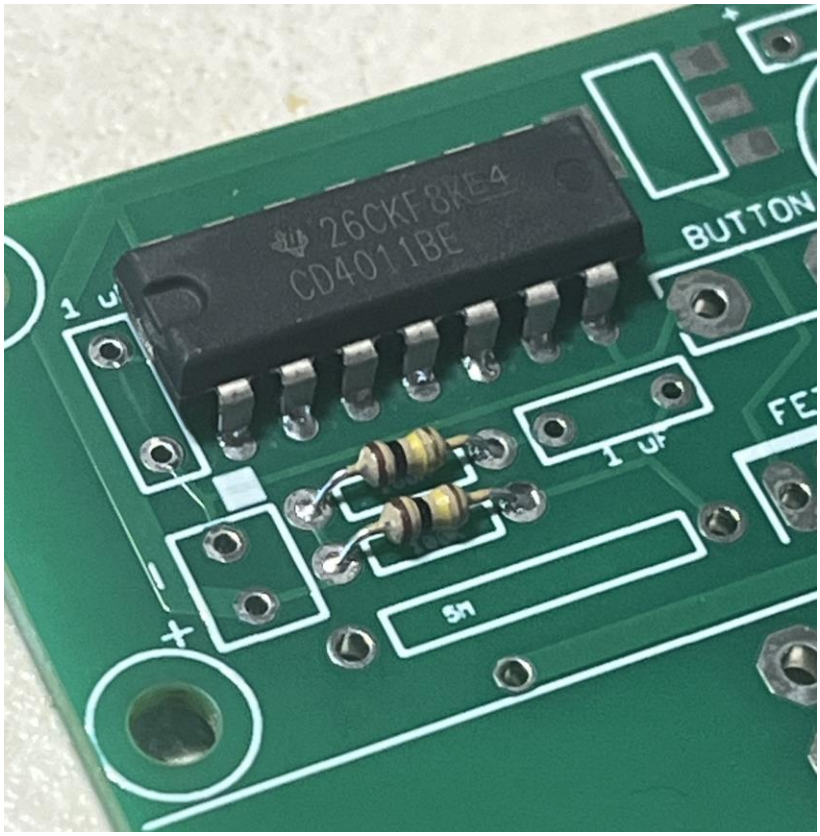
Now, solder all of the remaining leads to the pads. We recommend that you wait about 10 seconds between each lead, so you don't overheat the chip. When you're done, turn the board over and remove the tape.




26CKF8KE4
CD4011BE
BUTTON
100K
100K
5M

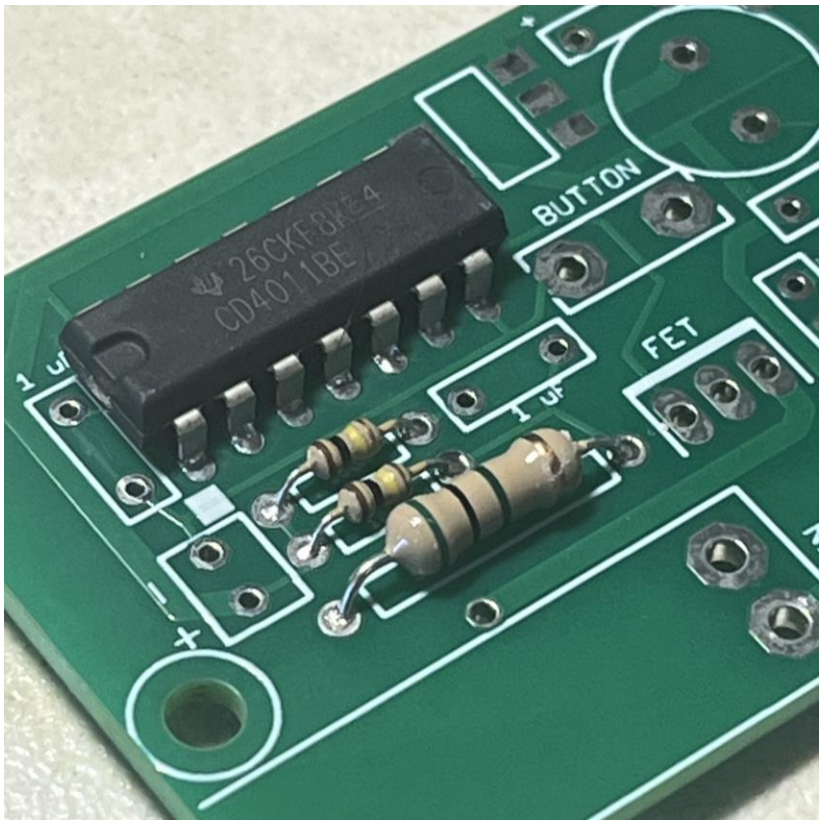
__ **100K resistors (2) (Bands are brown-black-yellow)**

Locate the two 100K resistors, they are just below the CD4011BE chip. These resistors can be identified by the color bands… brown-black-yellow. Bend the leads close to the body, then insert them into the board, into the spots marked “100K”. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



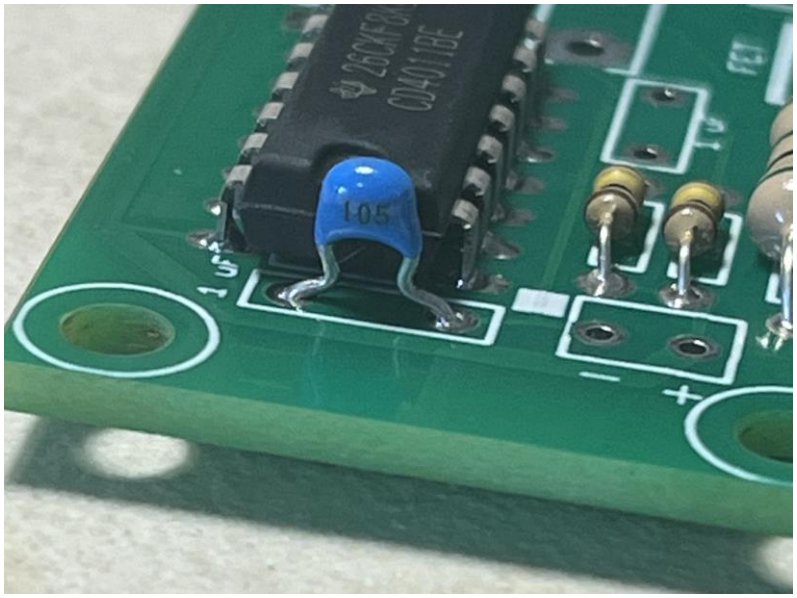
__ **5M resistor (Bands are green-black-green)**

Locate the 5M resistor, just below the 100K resistors that you just mounted. This resistor can be identified by the color bands… green-black-green, and it is much larger than the other resistors too. Bend the leads close to the body, then insert them into the board, into the space marked “5M”. Use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.

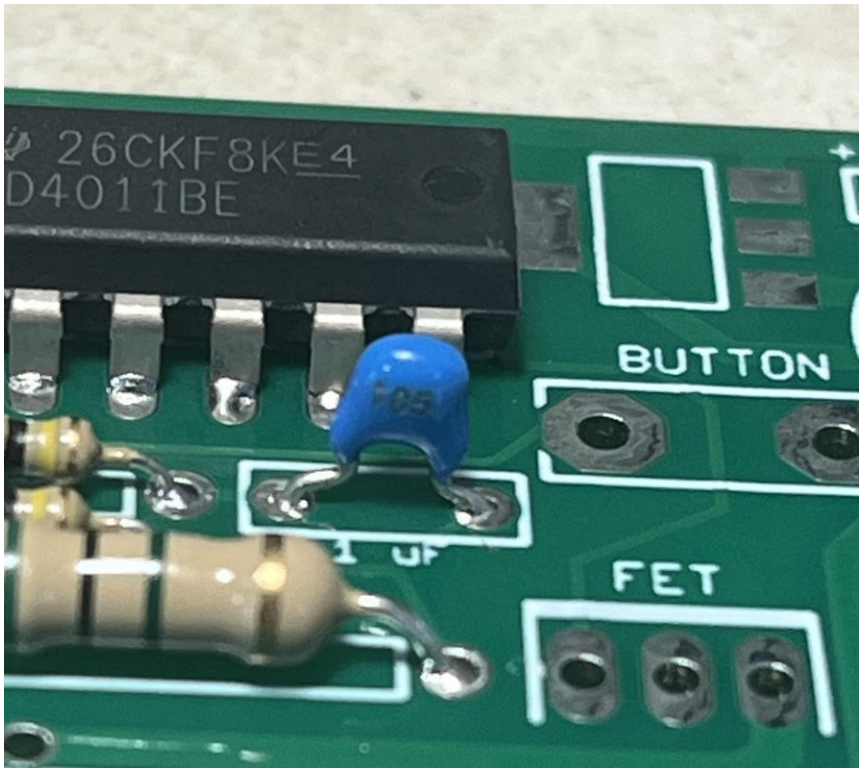


__ **1 uf capacitors (2) (Marked “105”)**

Locate one spot for the 1 uF capacitor, to the left of the CD4011BE chip. The 1 uF capacitor is marked “105”… make sure you have the right one, there are others that look very similar! Insert it into the board, and use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.

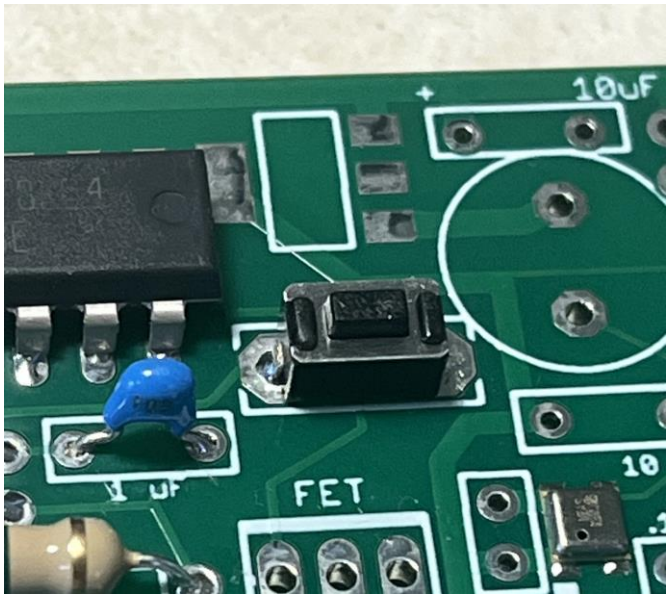


Locate the other 1 uF capacitor, just below and on the right side of the CD4011BE chip. The 1 uF capacitor is marked “105”… make sure you have the right one, there are others that look very similar! Insert it into the board, and use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



__ **Push Button**

Insert the push button into the two holes marked “BUTTON”, make sure that it’s flush with the board. Use some paper masking tape to hold it down, then turn the board over and solder both of the leads. When the solder has cooled, clip the leads close to the board.



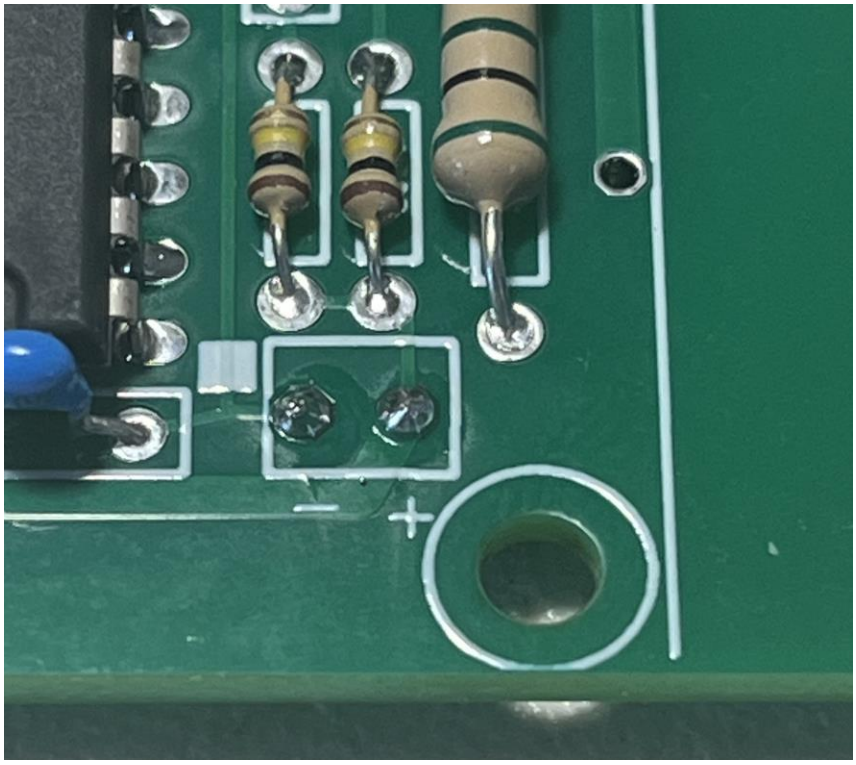
__ **9V battery connector**

Open the bag marked “AV Bay Wiring Package”, and remove the 9V battery connector. Close up the bag… you’ll use the other parts later when you finish building the EZ-DD Rocket’s AV Bay.

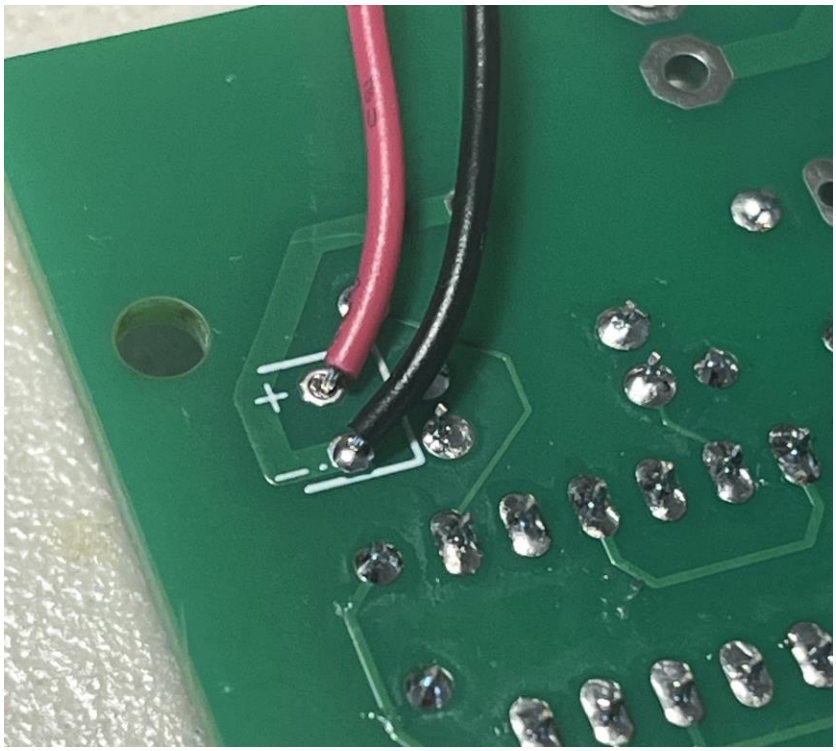
On the left side of the board you will see two pads marked “+” and “-“, these are the pads for the battery connector. Turn the board over, and you will see that the markings are also on the bottom of the board. You will be soldering the connector’s wires from the **bottom** of the board… this is so that when you mount it in the AV bay sled there won’t be any wires sticking up. (Trust us, when you mount the altimeter in the sled you’ll see what we’re talking about.)

Insert the RED wire of the battery connector into the pad marked “+”, and hold it down with some paper masking tape. Turn the board over and solder the wire to the pad, on the TOP side of the board.

Turn the board over and remove the tape. Now insert the BLACK wire of the battery connector into the pad marked “-”, and hold it down with some paper masking tape. Turn the board over and solder the wire to the pad, on the TOP side of the board.



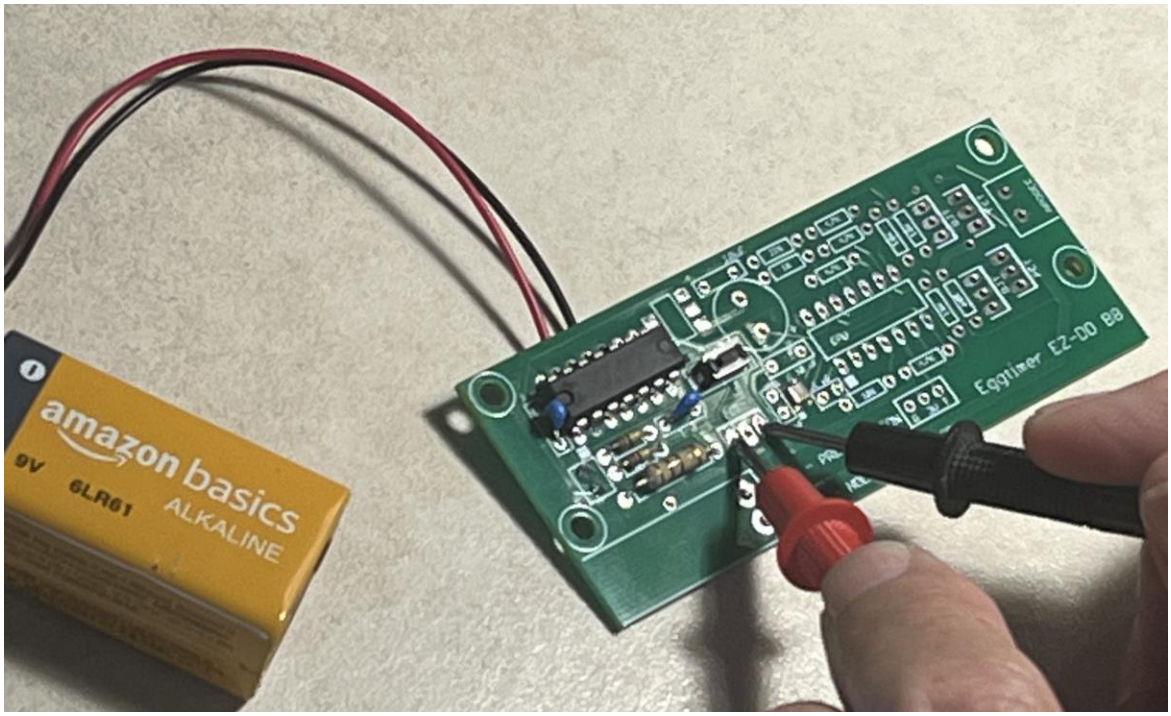
Turn the board over and remove the tape. Check to make SURE that the RED wire is on the “+” pad and the BLACK wire is on the “-“ pad. THIS IS CRITICAL… IF YOU HAVE THEM BACKWARDS YOUR BOARD WILL NOT WORK AND YOU MAY DAMAGE SOME COMPONENTS.



__ **Test the switch function**

Get a fresh 9V battery, and with a digital voltmeter (DVM) measure the voltage across its terminals. It should be around 9.0V, depending on how new it is and whether you've used it before.

Attach the 9V battery to the clip, and wait about 30 seconds (there is a "warm up" period for the switch circuitry.) Locate the empty pads for the component marked "FET", just to the right of the large 5M resistor. Using your DVM, put the RED lead on the LEFTMOST pad, and the BLACK lead on the RIGHTMOST pad. It should measure 0.0V.



Press the BUTTON and release it. Almost instantly you should see the voltage on those two FET pads jump from 0.0V to 9.0V, or whatever voltage you measured on your battery. It should stay at 9.0V… if it jumps up then goes down to 0.0V, or if it goes to 0.0V as soon as you release the button, you have something soldered incorrectly.

Now, HOLD the BUTTON for about 5 seconds or so, then release it. If you measure the voltage across the two FET pads, it should go back down to 0.0V. If it does not, you probably didn't hold the button down long enough… try it again and count slower.

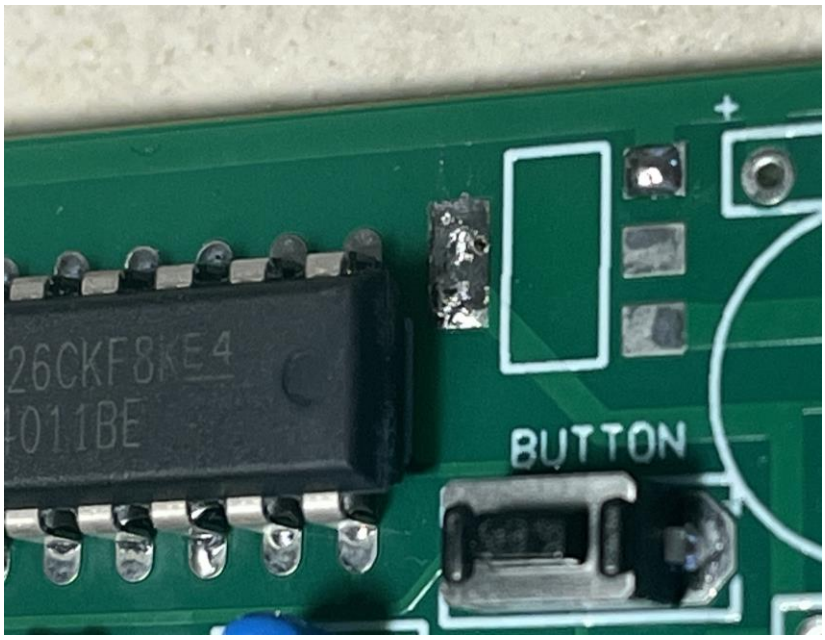
Assuming this test worked, the switch circuitry is working properly. Disconnect the 9V battery from the clip before proceeding.

Assembling the 3.3V Power Supply Components

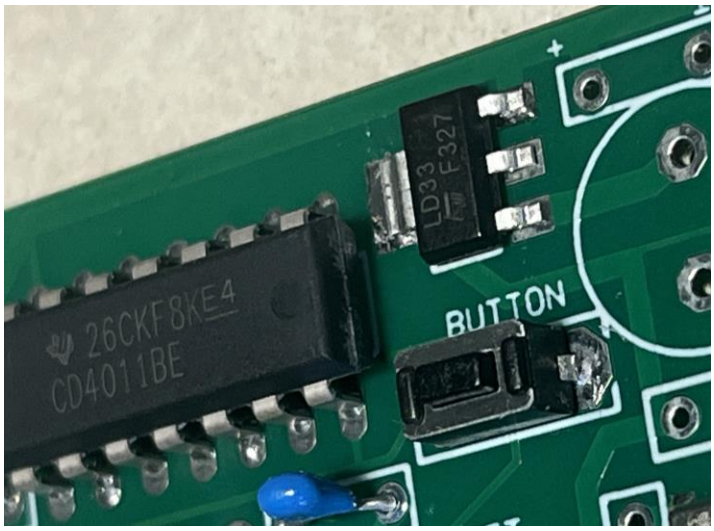
__ 3.3V regulator

Locate the spot for the 3.3V regulator. This is the only surface-mount part on the board… they don't make a through-hole equivalent. It's actually pretty easy to mount, if you follow the instructions carefully.

Lightly tin the large pad… just a swipe of solder from your iron will do. Lightly tin the topmost of the three pads on the right, with just a little bit more solder than on the large pad.

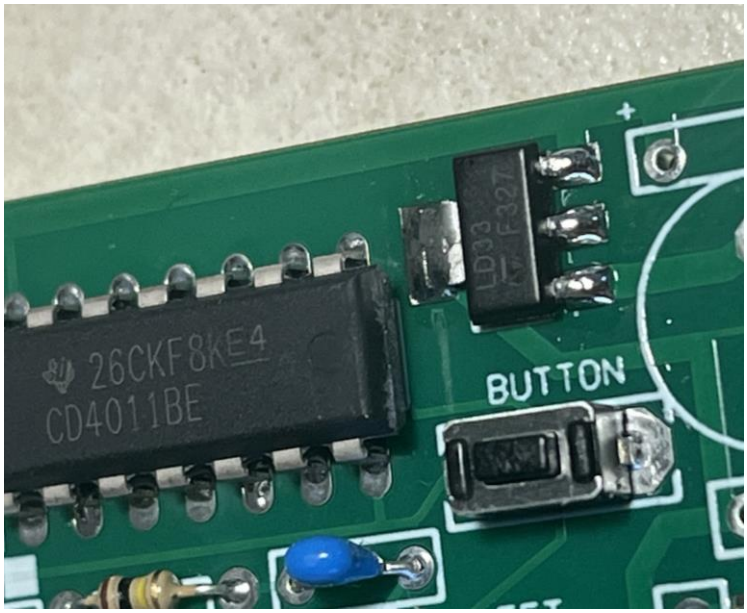


With tweezers, hold the regulator over the pads, then with your soldering iron melt the solder on the top pad. Wait a few seconds for it to completely melt, then withdraw the iron, and hold the regulator in place for another 5 seconds or so. If you let go of the tweezers it should be held in place by that solder joint. Check to see that the regulator is flat on the board… if it is not, reheat the solder joint and try again.



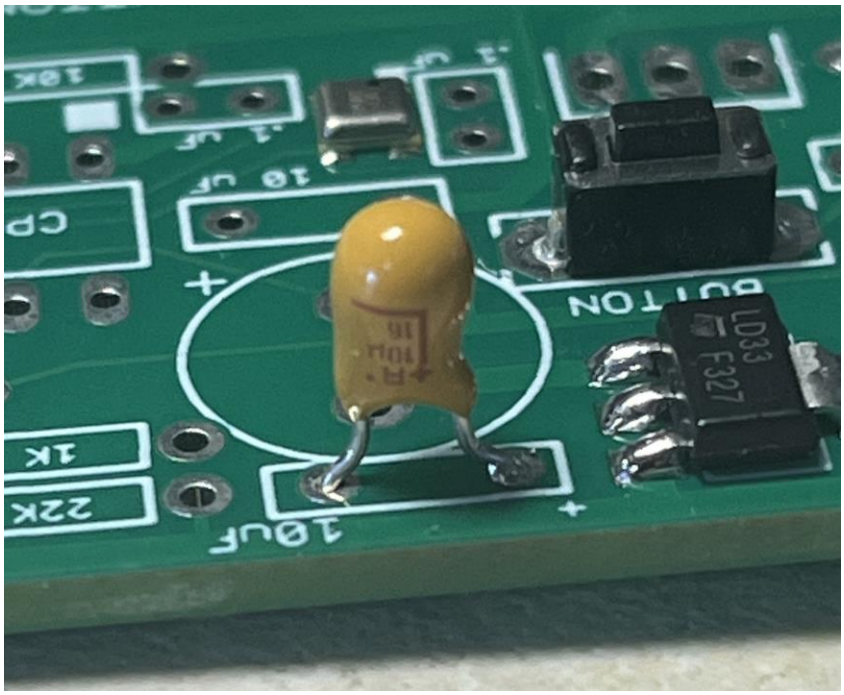
Heat up the large pad with your iron for about 5 seconds or so, then apply solder to the pad and the tab on the regulator. Hold the iron there until the solder is nice and shiny, then withdraw the iron. Let it cool for at least 10 seconds before continuing.

Now, one by one, heat up the other unsoldered pads, holding the iron for several seconds before you apply the solder so the solder will flow out. The solder joints need to be nice and shiny! You may need to go back and resolder the original solder joint if it doesn't look as good as the other ones… that's fairly normal.



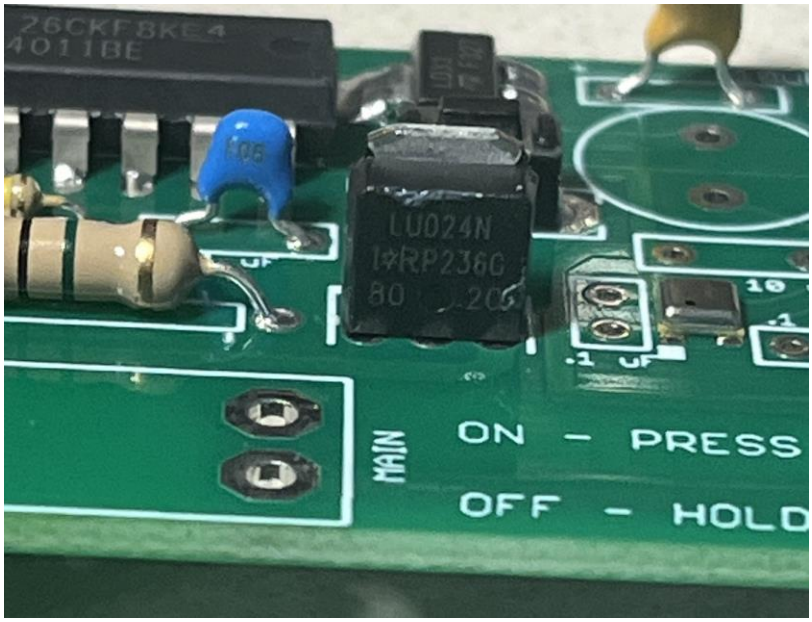
__ 10 uf Tantalum capacitor (marked "106" or 10u, with a "+" mark on it)

Locate the spot for the 10 uF Tantalum capacitor, just to the right of the regulator that you just installed. Notice that one lead is marked with a "+" marking… this MUST match the "+" mark on the board. IF YOU INSERT IT BACKWARDS, YOUR BOARD WILL NOT WORK, AND YOU MAY DAMAGE THE CAPACITOR. Insert it into the board, and use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



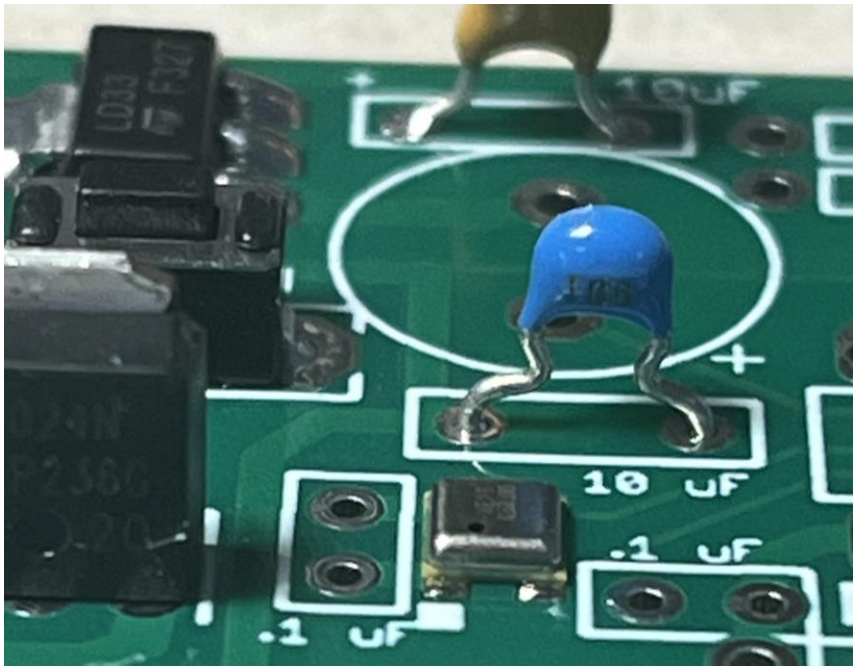
__ **Power FET (marked “LU024N”)**

Locate the spot for the switch FET on the board, it’s just to the right of the 5M resistor. Find one of the IRLU024 FET’s… they are marked “LU024N”. There are other parts that look similar, so make sure that you have the right one! Insert the FET into the board, so that the metal tab is facing the mark on the board with the “bar”. Tape it into place with some masking tape. Turn the board over, the carefully solder the leads to the pads. Note that due to the size of the leads and the pads it will take a little longer for the solder to flow out than the smaller-leaded parts. We also recommend that you let the leads cool for 10 seconds or so before going on to the next one… the tab on the FET is going to be hot to the touch, so be careful! When you’re done, clip the leads close to the board, then turn the board over and remove the tape.



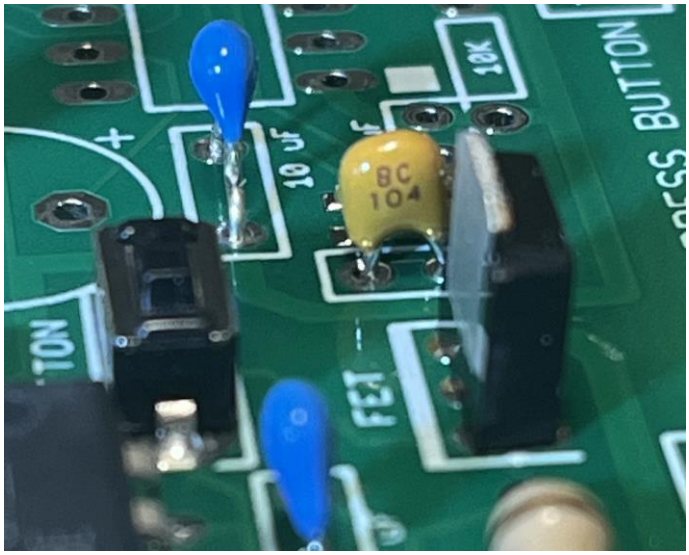
__ **10 uf MLCC capacitor (marked “106”)**

Locate the spot for 10 uF capacitor, it’s just below the circle in the middle of the board and is marked “10 uF”. Find the single 10 uF capacitor, it’s marked “106”. Note that there are other capacitors that look very similar… make sure you have the right one! Insert it into the pads, tape it down with some masking tape, then turn the board over and solder the capacitor onto the pads and clip the leads short.



__ **.1 uF capacitor (by FET) (Marked “104”)**

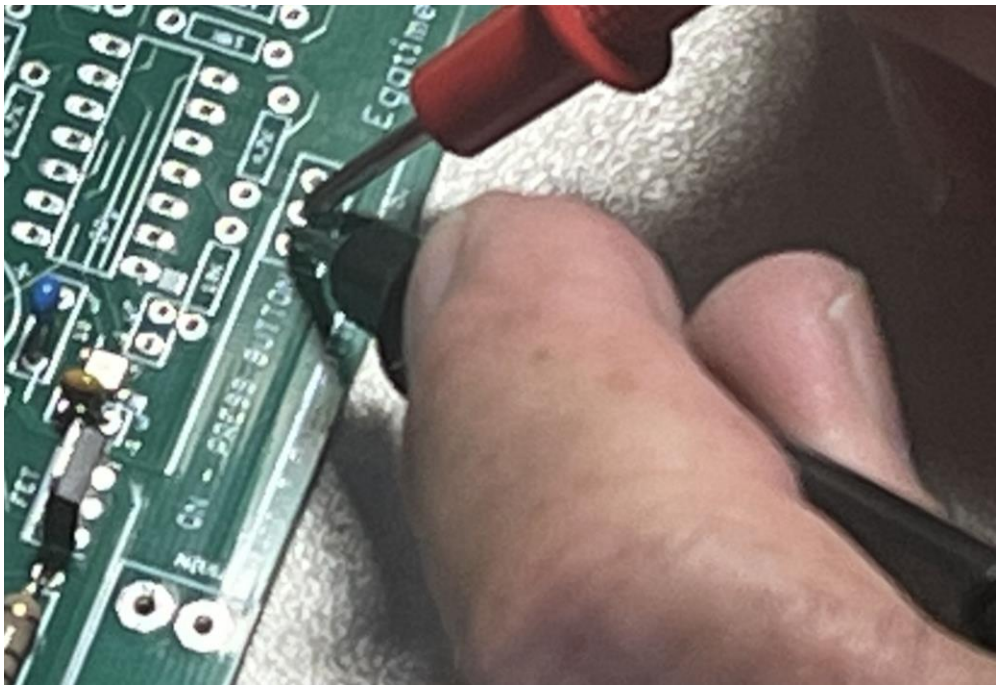
Locate the spot for the .1 uF capacitor, just to the right of the FET that you just installed. Insert it into the board, and use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



__ **Test the 3.3V output**

Connect the 9V battery to the connector, wait about 30 seconds (there is a "warm up" period for the switch circuitry), then turn the EZ-DD on by pressing the button.

Locate the spot for the 3-pin header, it's a rectangular box with three pads near the bottom-center of the board marked "G 3V T". Using a digital voltmeter (DVM), connect the BLACK (-) lead to the pad marked "G" and the RED ("+") lead to the pad marked "3V". Your DVM should register approximately 3.30V, give or take a few hundredths. If you do NOT get 3.30V, see the troubleshooting section…. and do not continue until you get 3.30V here!



Finish the Altimeter

__ Buzzer

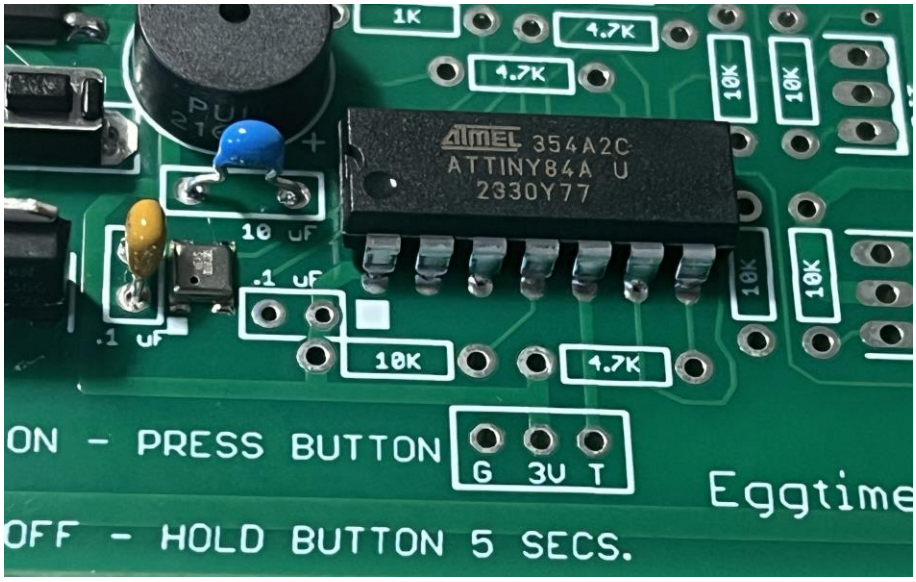
Locate the spot for the buzzer, it's a circle with a "+" marking on one side. If you look at the buzzer, you'll see that one lead is longer than the other, and there is a "+" marking on the body of the buzzer by that lead. Insert the buzzer so that the "+" lead goes into the pad on the board marked with a "+", tape it to the board with some masking tape, then turn it over and solder the pads. Clip the leads flush afterwards, then remove the tape. Also, if the buzzer has a plastic tape label covering the sound hole remove that too… you're not going to get much volume out of it if the hole is plugged!

__ CPU (Marked "ATTINY84A U")

Locate the spot for the CPU, it's a 14-pin chip marked "CPU". Note that there is a rectangular mark on the lower-left side of the pads… that is the "PIN 1" mark. On the ATTINY84AU package there will be a notch on the left side, and also an indentation near Pin 1. The notch should be on the LEFT side of the pads on the board, and the Pin 1 indentation should match up with the square marked on the lower-left side of the pads.

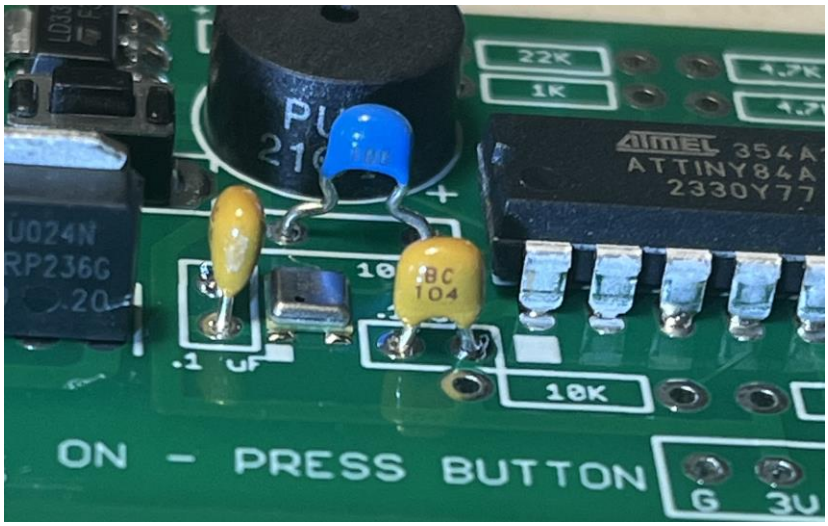
Insert the CPU into the holes on the board, you may have to carefully work the leads into the holes to get them all in. Hold the CPU onto the board with some masking tape, then turn the board over. Solder ONE corner pin, then turn the board over and make sure that it's seated properly in the board. If it is NOT, heat up that one solder joint while gently pushing the CPU chip into the board.

Now, solder all of the remaining leads to the pads. We recommend that you wait about 10 seconds between each lead, so you don't overheat the chip. When you're done, turn the board over and remove the tape.



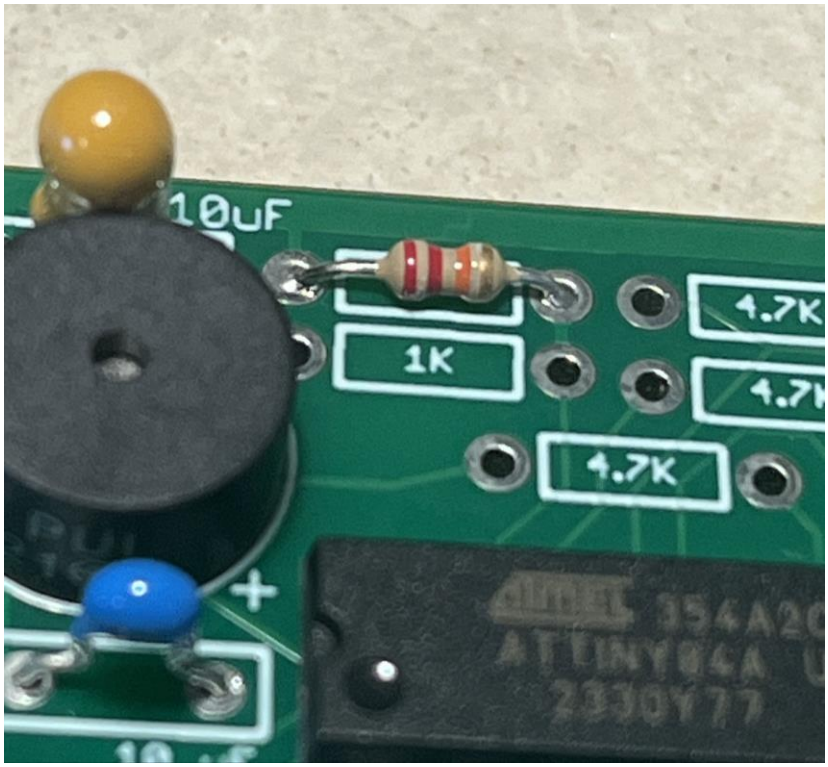
__ **.1 uF capacitor (by CPU) (Marked “104”)**

Locate the spot for the .1 uF capacitor, to the lower-left side of the CPU. Insert it into the board, and use a piece of masking tape to hold it down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



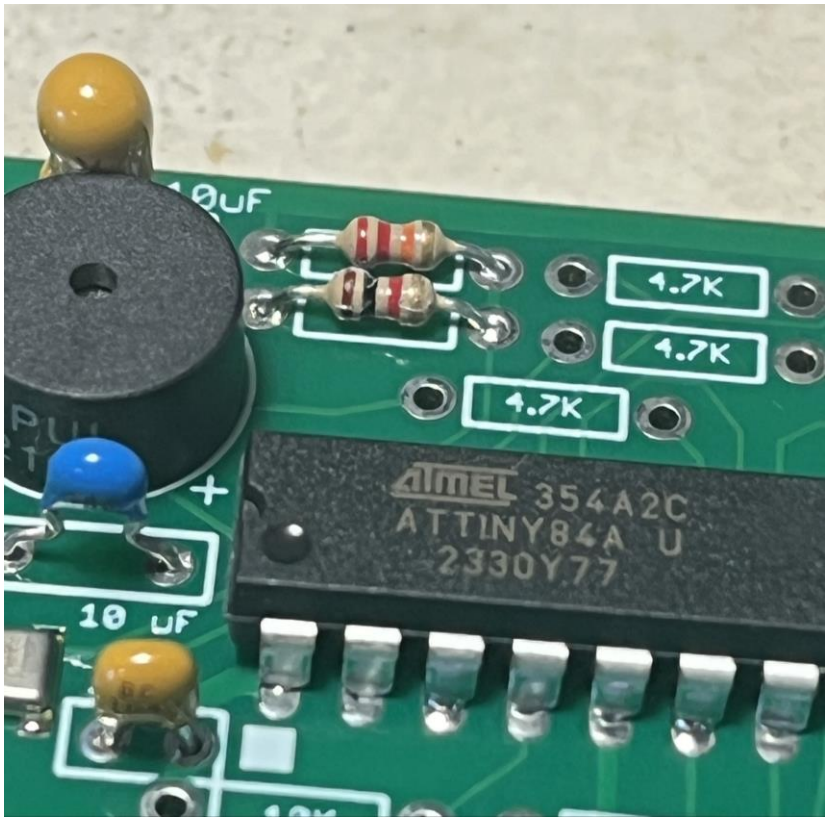
__ **22K resistor (Bands are red-red-orange)**

Locate the spot for the 22K resistor (red-red-orange) near the top-center of the board. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



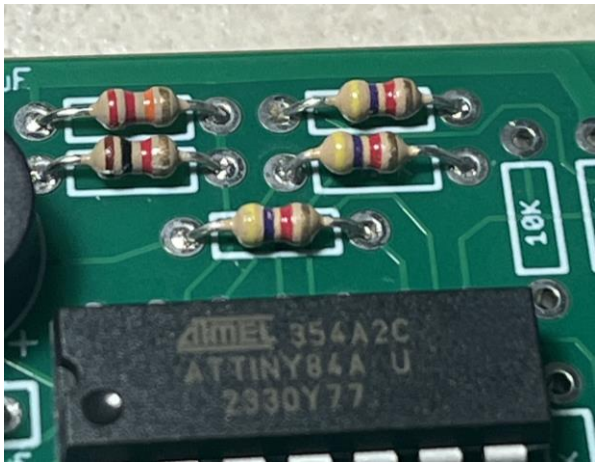
__ **1K resistor (Bands are brown-black-red)**

Locate the spot for the 1K resistor (brown-black-red) just below the 22K resistor that you just mounted. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.

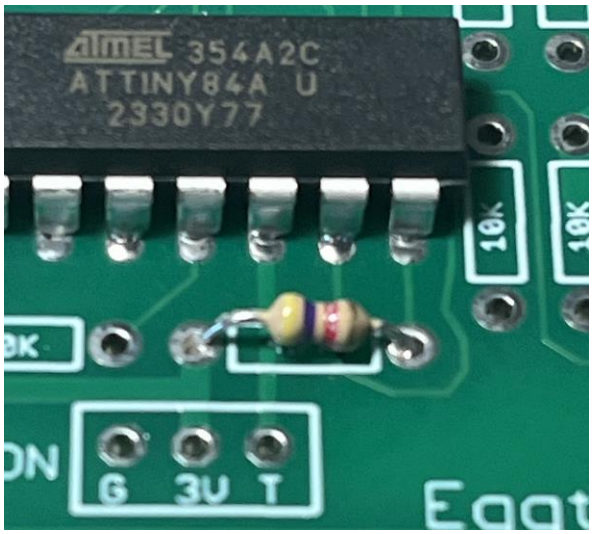


__ **4.7K resistors (4) (Bands are yellow-purple-red)**

Locate the three 4.7K resistors at the top of the board just above the processor, marked “4.7K”. These resistors can be identified by the color bands… yellow-purple-red. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.

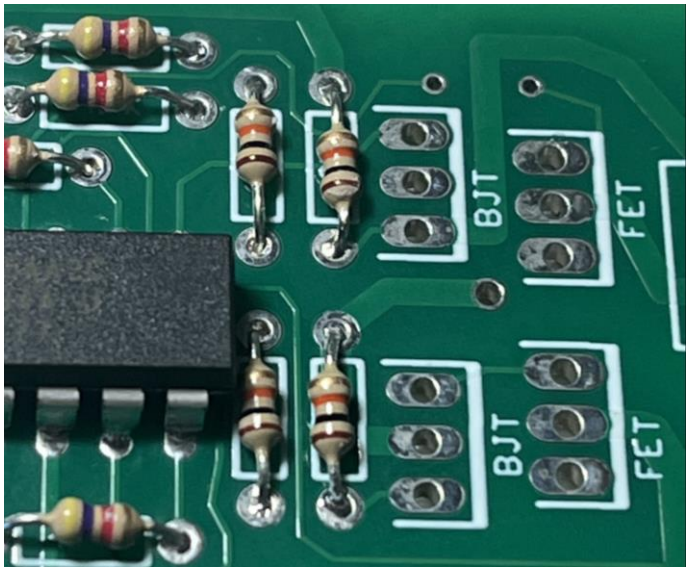


Locate the 4.7K resistor near the bottom of the processor, marked “4.7K”. This resistor can be identified by the color bands… yellow-purple-red. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.

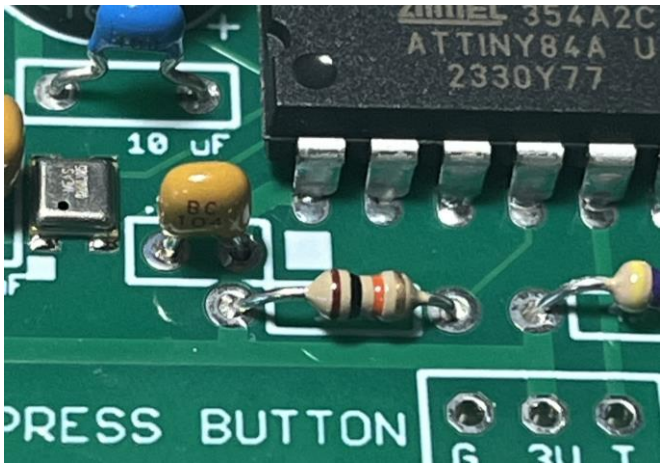


__ **10K resistors (5) (Bands are brown-black-orange)**

Locate four 10K resistors, just to the right of the CPU, marked “10K”. These resistors can be identified by the color bands… brown-black-orange. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



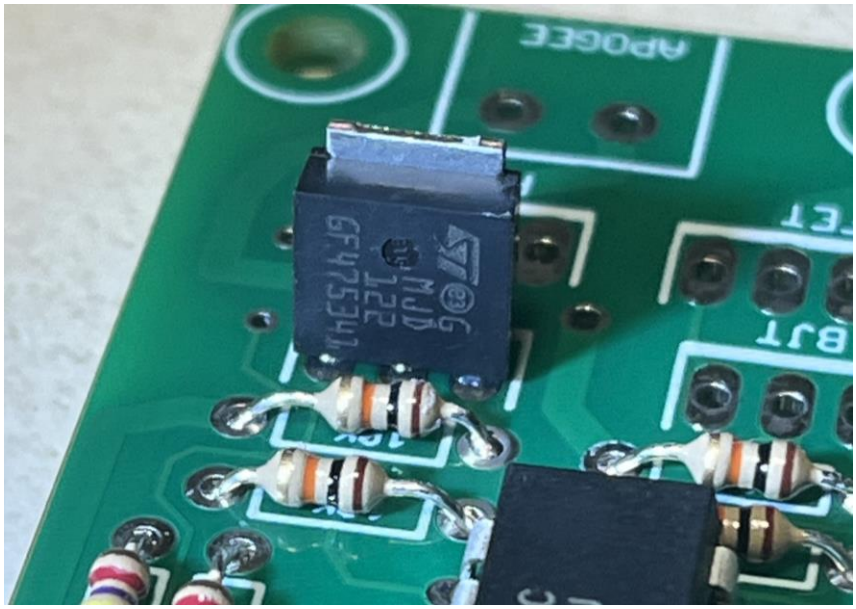
Locate the single 10K resistor next to Pin 1 of the CPU. Bend the leads close to the body, then insert them into the board. Use a piece of masking tape to hold them down, then turn the board over and solder the leads to the pads. Clip the leads off close to the board, then remove the masking tape.



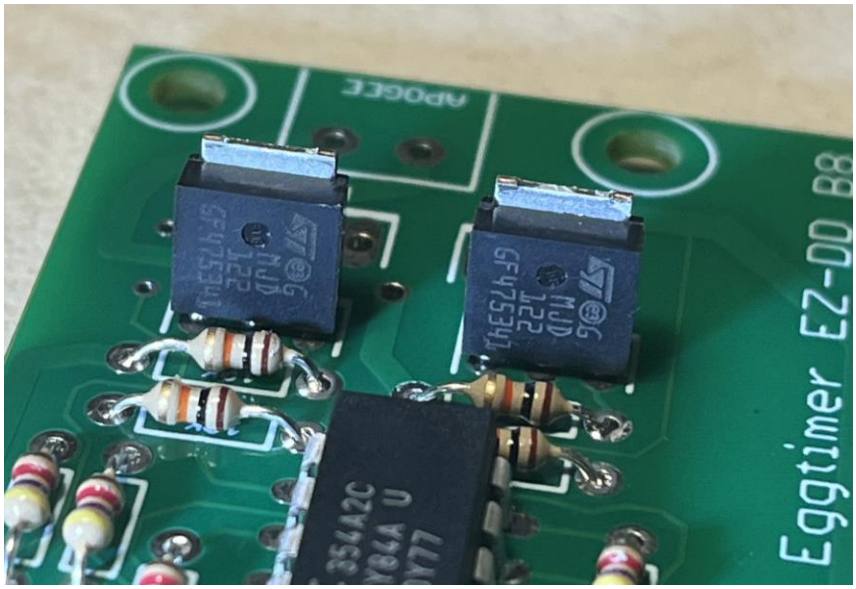
__ **BJT's (2) (Marked "MJD 122")**

Locate the two BJT transistors, marked "BJT" on the board, just to the right of the four 10K resistors that you just mounted. Get the two MJD-122 transistors, they are 3-pin parts marked "MJD 122". Note that there are other parts that look very similar, so make sure you have the right ones!

Insert one of the transistors into the pads, with the metal tab facing the right side. Note that there is a "bar" marking on the board, the bar needs to match up with the tab. Tape it into place with some masking tape. Turn the board over, the carefully solder the leads to the pads. Note that due to the size of the leads and the pads it will take a little longer for the solder to flow out than the smaller-leaded parts. We also recommend that you let the lead cool for 10 seconds or so before going on to the next one… the tab on the transistor is going to be hot to the touch, so be careful! When you're done, clip the leads close to the board, then turn the board over and remove the tape.

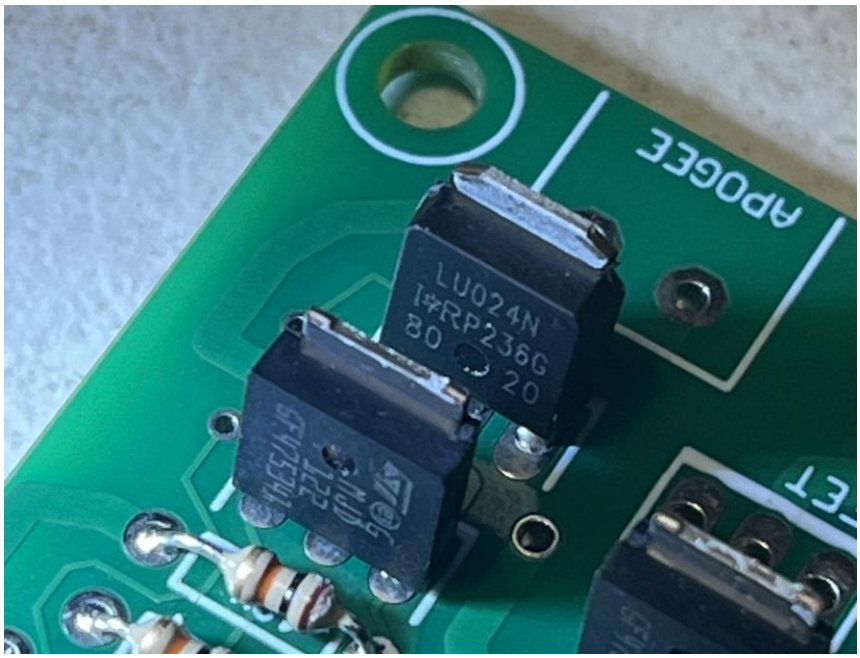


Similarly, mount the other BJT transistor.

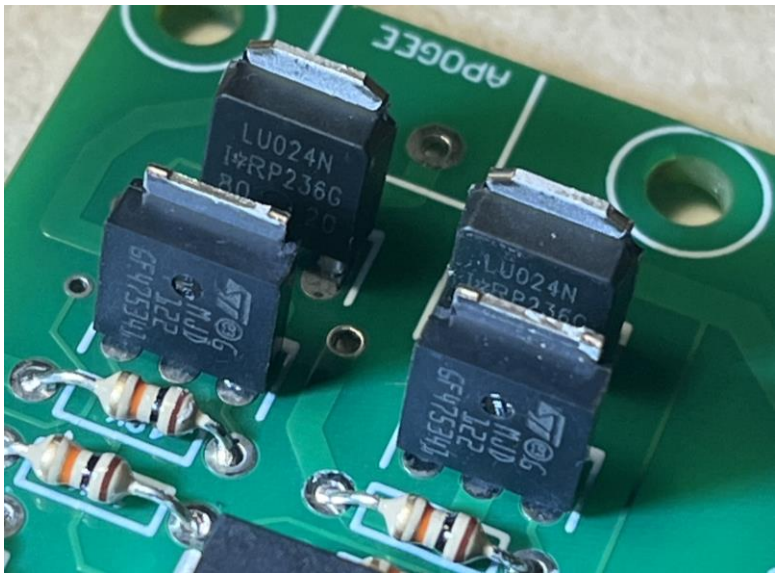


__ **FET's (2) (Marked "IRL024N")**

Locate the spot for the output FET on the board, they are just to the right of the two BJT transistors that you just mounted. Find two of the IRLU024 FET's… they are marked "LU024N". There are other parts that look similar, so make sure that you have the right one! Insert the FET into the board, so that the metal tab is facing right, and is lined up with the mark on the board with the "bar". Tape it into place with some masking tape. Turn the board over, the carefully solder the leads to the pads. Note that due to the size of the leads and the pads it will take a little longer for the solder to flow out than the smaller-leaded parts. We also recommend that you let the lead cool for 10 seconds or so before going on to the next one… the tab on the FET is going to be hot to the touch, so be careful! When you're done, clip the leads close to the board, then turn the board over and remove the tape.

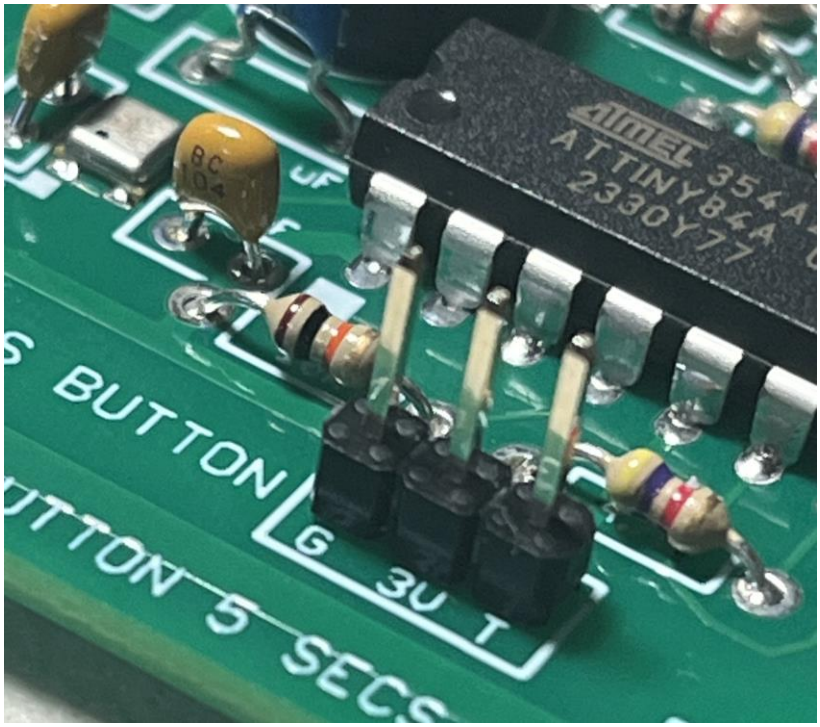


Similarly, mount the other FET.



__ **3-pin header**

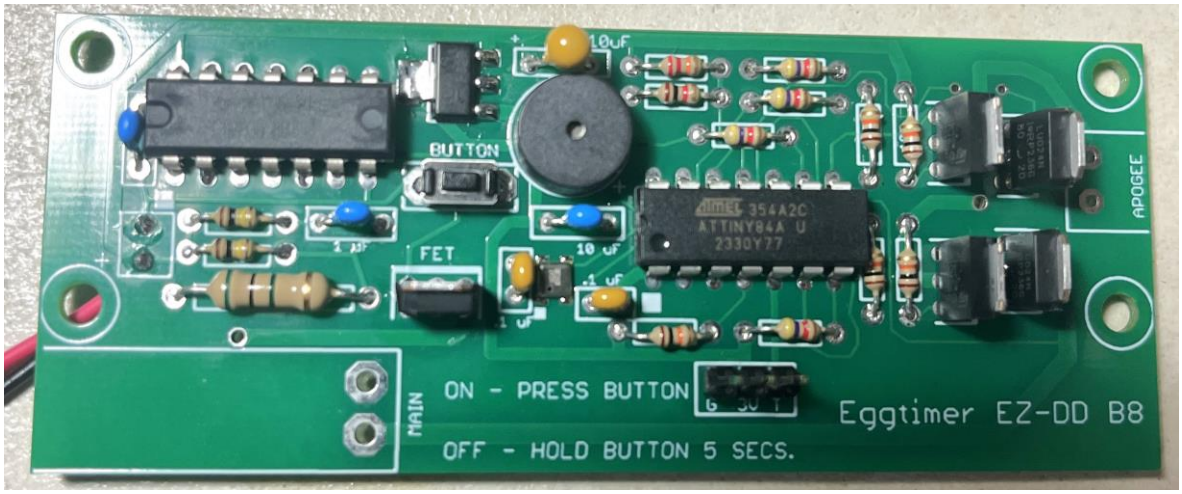
Locate the spot for the 3-pin header, it's a box below the processor marked "G 3V T". Insert the header from the top of the board with the long side facing UP… the short side should be inserted into the holes. Tape the header to the board, then turn it over and solder the pins to the pads. Turn it over and remove the tape.



__ Check Your Work

Go back and check to make sure that everything is in the right place, and all of the joints were soldered. Yes, we get support calls that end up being a missed solder joint. Plenty of them.

If you find that a part was installed incorrectly, you'll have to remove it with a solder sucker and/or desoldering wick. If you need a replacement part, send us an email, to support@EggtimerRocketry.com



__ Initial Power-on test

After you've confirmed that everything is in the right place and soldered, connect your 9V battery to the battery connector, and wait about 30 seconds (there is a "warm up" period for the power switch circuitry). Press the power button, you should almost immediately hear a long beep, followed by a number of shorter beeps. **If you DO NOT hear a beep, unplug the battery and to go the Troubleshooting section!**

The shorter beeps are the battery voltage… they work like this:

One beep – One
Two beeps – Two
….
….
….
Nine beeps – Nine
Ten beeps – Zero

The battery voltage is beeped-out in tenths of a volt. So, if you hear ten beeps, nine beeps, then one beep, your battery voltage is 091, or 9.1V.

At this point, since you don't have anything connected to the output channels, it's going to beep out "no continuity" beeps, repeating until you either power down the altimeter or fix the problem.

One beep – Apogee failing continuity test
Two beeps – Main failing continuity test

So, if you hear one beep, a short pause, two beeps, then a long pause and the whole thing repeating, this means that both your apogee and main outputs are failing the continuity test… which is to be expected at this point, because you don't have anything connected to the outputs.

At this point, you have checked the basic operation of the EZ-DD altimeter, you just need to make sure that the baro sensor and the outputs are working. Before you do that, you'll need to do a little wiring… this is actually all the wiring there is in the kit.

Installing the Output Wiring (For testing)

In order to perform an output test you need get a few pieces from the EZ-DD AV Wiring Kit bag. You're going to solder a wire to the APOGEE output, install a header on the MAIN output, and assemble the matching socket/pigtail that matches the header.

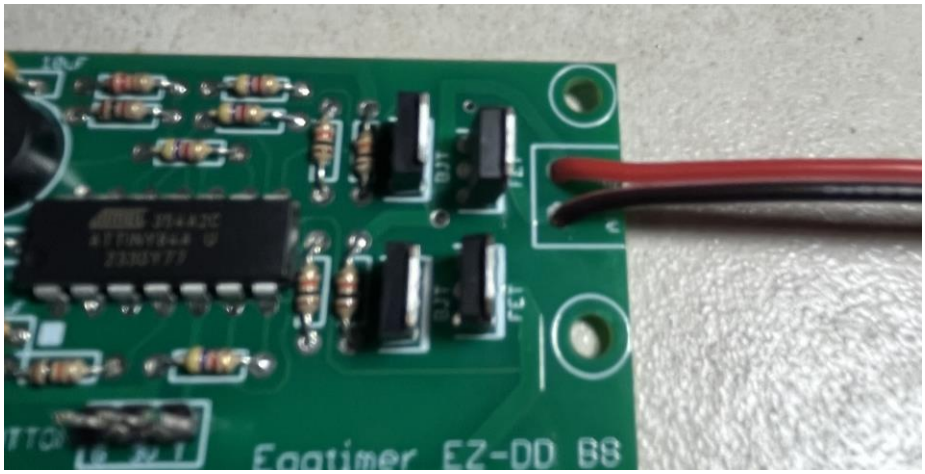
From the EZ-DD AV Bay Wiring Kit package

__ (1) 2-pin .156" locking right-angle header
__ (1) 2-pin .156" locking socket body
__ (2) Crimp pins for locking socket body
__ 10" #22 gauge stranded duplex wire

__ Install the Apogee Wires

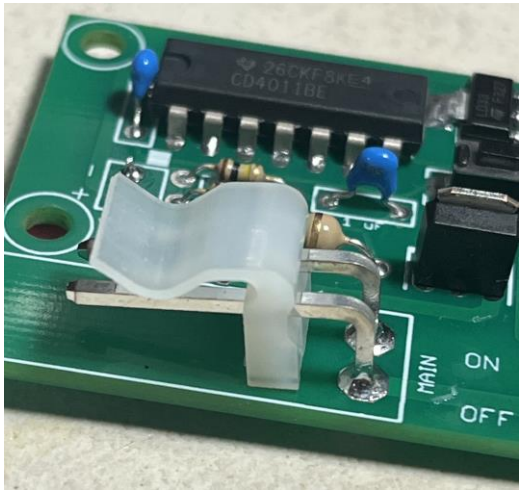
From the bag labeled "AV Bay Wiring Kit" remove the 12" of two-conductor wiring. Cut a piece 2" long, and set the rest aside. Carefully strip about 1/8" from each wire on one end, twist it tightly, then tin the ends lightly with solder. Make sure that there are no frayed ends, and that there are no blobs of solder on the wires.

Insert one end of the wires into one of the pads marked APOGEE, tape it in place with some paper masking tape, then turn the board over and solder the wire in place. Clip the wire close to the board. Similarly, insert the other wire into the other APOGEE pad and solder into place.



__ Install the Main Header

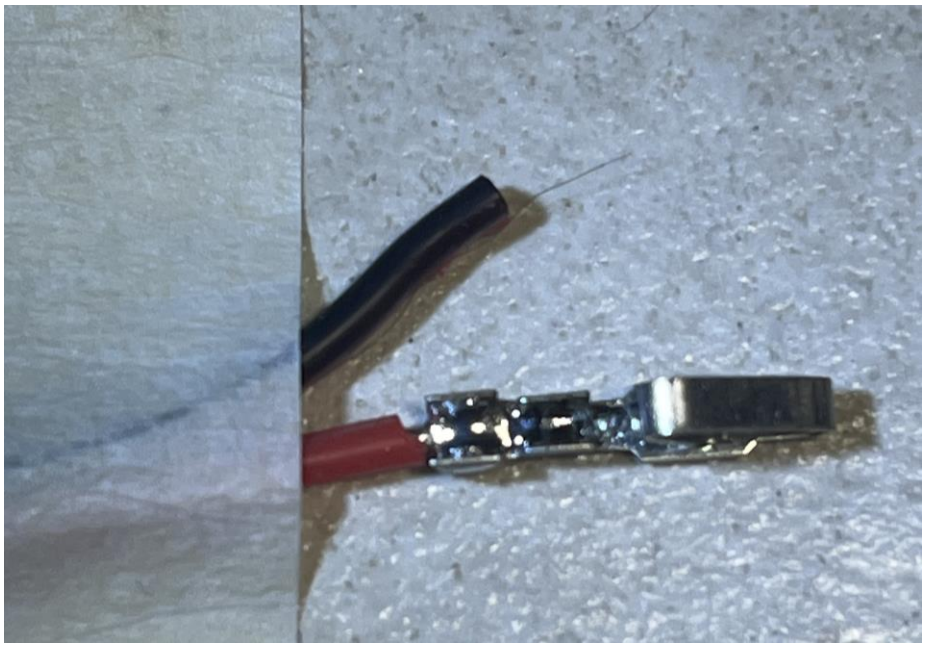
From the bag labeled "AV Bay Wiring Kit" remove the 2-pin right-angle header. Insert the short end of the header into the pads marked MAIN so that the long pins face the near edge of the board, then tape it in place with some paper masking tape. Turn the board over and solder the pins in place. Be generous with the solder, the pins and the pads are large. Turn the board over, and also solder the pins to the TOP pads of the board… this will provide extra support for the header.



__ **Prepare the Socket Plug**

From the bag labeled “AV Bay Wiring Kit” remove two of the socket pins, and the 2-pin socket body. Get the remaining piece of two-conductor wire that you set aside, and carefully strip about 1/8” from each wire on one end, twist it tightly, then tin the ends lightly with solder. Make sure that there are no frayed ends, and that there are no blobs of solder on the wires.

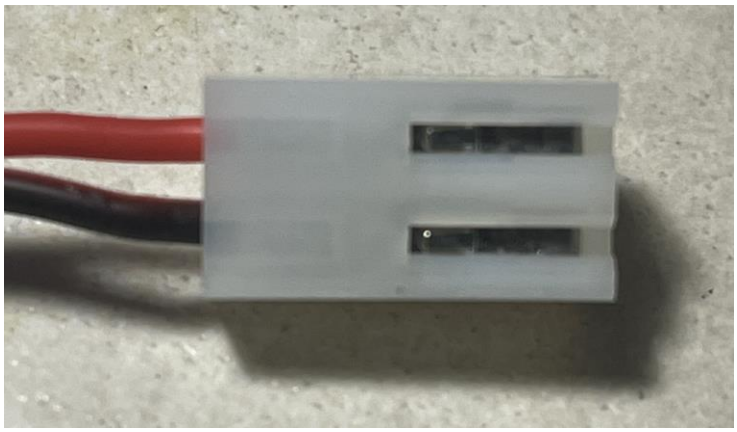
Using either a “helping hands” jig or taping them to the table, set the tinned end of one wire into the “channel” of one of the socket pins. These pins are designed to be crimped, but you’re going to be soldering them instead. With your soldering iron, heat up the tinned end of the wire in the channel, then add solder until it just fills the channel. Remove the solder and the iron, and let it cool for 10-15 seconds… the pin will be hot, so be patient.



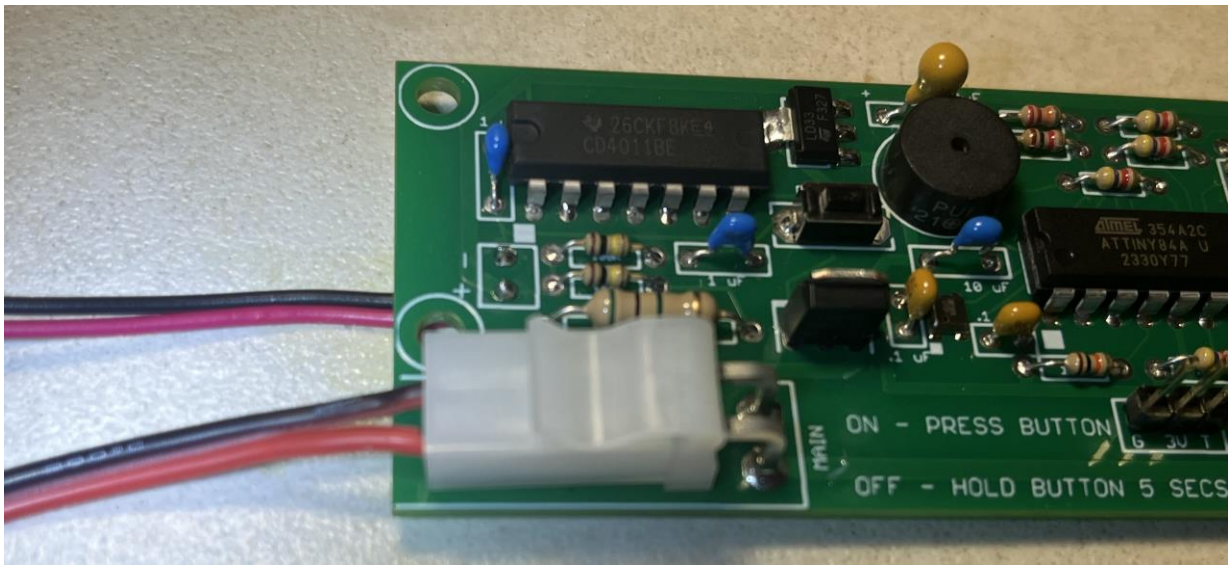
Repeat the procedure for the other wire and socket pin.

Once the socket pins have cooled, slide them into the socket body. There is a slot in one side of the socket body… that slot must line up with the “tabs” that are punched in the socket pins, opposite the solder joint. You should hear the pins “click” as you push them in, and they

should not be able to be pulled out. If they pull out, push them in farther; you may have to bend out the little tab in the pin, too.



Now, push the socket body into the header on the altimeter, it should lock in place.

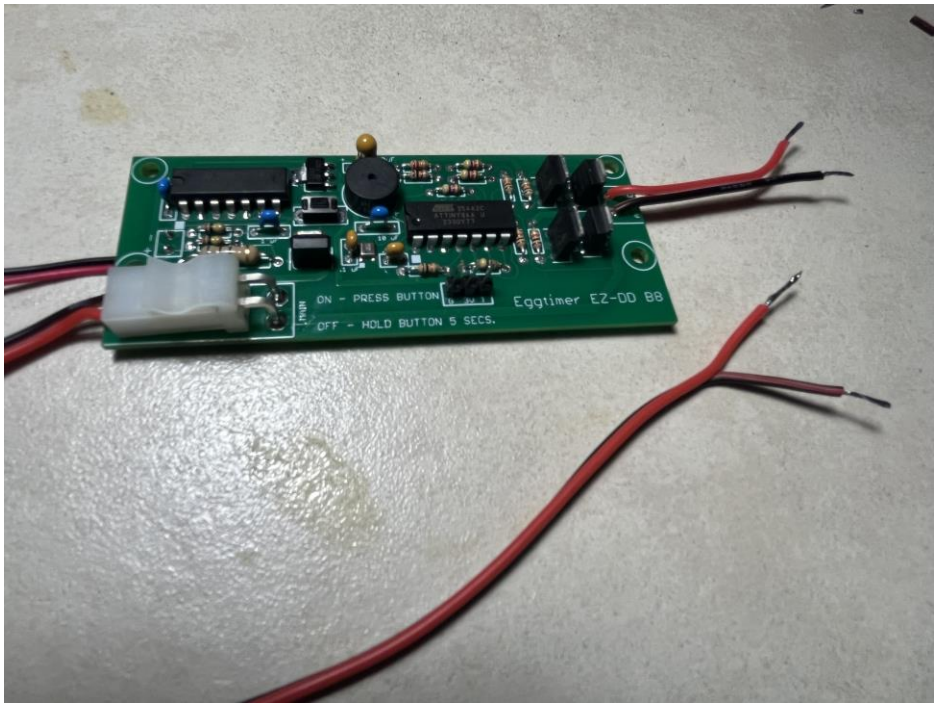


__ **Tin the Pigtail Ends**

You will now have wires connected to both the APOGEE and MAIN outputs of the altimeter, with the insulation still intact.

On the APOGEE cable, carefully strip about 1/8” from the wire on one end, twist it tightly, then tin the ends lightly with solder. Make sure that there are no frayed ends, and that there are no blobs of solder on the wires. Repeat for the other wire on the cable.

On the MAIN cable (connected to the board with the right-angle connector), carefully strip about 1/8” from the wire on one end, twist it tightly, then tin the ends lightly with solder. Make sure that there are no frayed ends, and that there are no blobs of solder on the wires. Repeat for the other wire on the cable.



Performing a Vacuum Test

The best way to check the full functionality of your EZ-DD altimeter is with a vacuum test. As your rocket ascends, the air pressure decreases, until you hit apogee, then the pressure increases as you descend. You can simulate this with a vacuum cleaner by holding the hose over the pressure sensor and drawing a vacuum, keeping it constant for a bit so the vacuum stabilizes (this is what would happen around apogee), then turning off the vacuum to simulate a descent.

Note that this is NOT a good simulation for an actual flight, because the vacuum will be drawn and released much more rapidly than it would be in a real flight. However, it works just fine for functional deployment testing, which is what we're aiming for.

You will need:

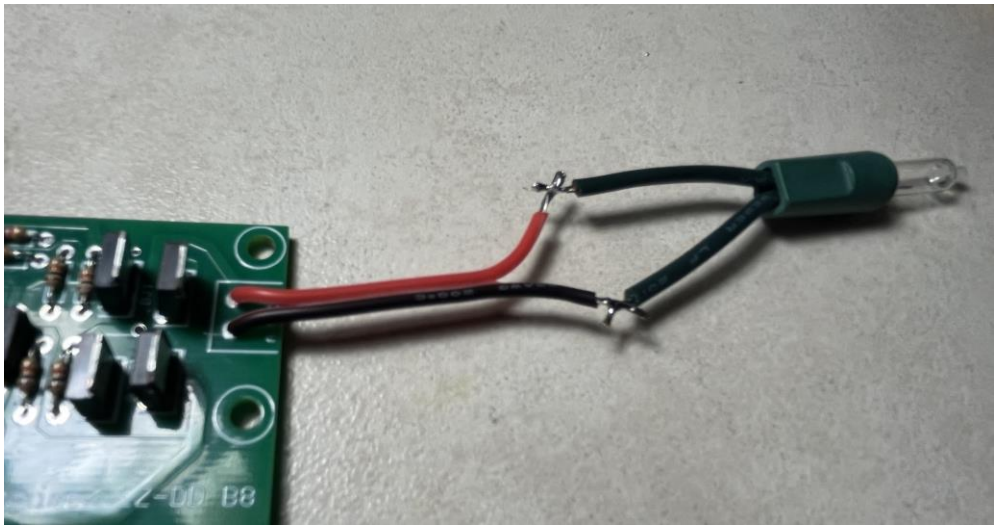
- Your EZ-DD altimeter with a 9V battery, with the preliminary tests completed
- Some light bulbs on the outputs… we like miniature incandescent Christmas tree lights (DO NOT use LED's… they won't work properly)
- A vacuum cleaner with a PLASTIC hose (DO NOT USE A METAL HOSE!!!)

Basically, you're going to hook everything up, turn it on, wait for it to be ready, then draw a vacuum with the hose until the Apogee channel "fires". After you turn off the vacuum (to simulate the rocket coming down) the main channel will "fire".

You will need to temporarily connect the light bulbs to the altimeter's outputs. We recommend that you use miniature Christmas tree bulbs for this (NOT LED's!), you will need to cut a couple from the string and strip/tin the ends, then tack-solder them to the APOGEE and MAIN pigtails.

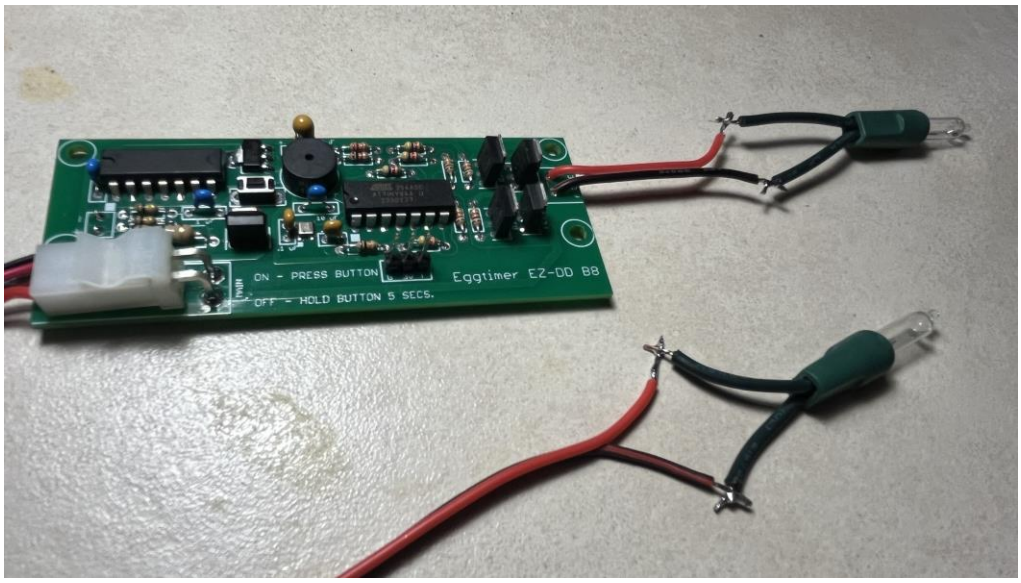
__ **Connect an APOGEE output load**

Tack-solder a Christmas tree light bulb (or equivalent load) to the pigtail for the APOGEE output. You may want to put some masking tape over the joints temporarily to prevent them from shorting.



__ **Connect a MAIN output load**

Tack-solder a Christmas tree light bulb (or equivalent load) to the pigtail for the MAIN output pigtail, the one that's plugged into the MAIN header.. You may want to put some masking tape over the joints temporarily to prevent them from shorting.



__ **Perform the Test**

Now that you have some output loads that you can verify visually, and which should pass the continuity test, you're ready to test.

Since a video is worth 1,000 words, we have a YouTube video that shows you how to do this… it's actually pretty easy.

https://www.youtube.com/watch?v=YhpOUyR3KXg

It's important that you hold the vacuum for awhile until the APOGEE output fires… it could take 10 seconds or so, depending on how steady your vacuum is. The mach-safe logic in the EZ-DD altimeter expects the pressure to stabilize, such as it would around apogee, so if your vacuum isn't steady you may have a hard time getting an APOGEE event. If you can't get the APOGEE to fire, just turn off the vacuum… that should make it light.

Note that in a vacuum test it is NOT unusual for both lights to appear to go off simultaneously if you force the APOGEE output to "fire" by shutting off the vacuum. This is because the vacuum is dropping so rapidly that your MAIN altitude (600') is reached almost as soon as you turn off the vacuum. This does NOT mean that there is a problem, and it will NOT occur in a real flight.

Note that in the video we soldered the pigtails to the terminal blocks first; you may find that to be a little easier than tack-soldering the light bulbs to the pigtails, but if you do that you're going to have to unsolder the pigtails from the terminal blocks before you mount the altimeter to the sled. That's why we have you tack-solder the light bulbs directly to the pigtails… it's easier to remove them than to unsolder from the terminal blocks and clean up the excess solder.

Once you perform the vacuum test and both lights have lit successfully, you're done with the altimeter… time to go onto building the AV bay, mounting the altimeter, and finishing up the wiring. See the Eggtimer EZ-DD AV Bay Assembly Guide.

If you CAN NOT get the outputs to fire, send us an email at support@EggtimerRocketry.com.

Troubleshooting

If your EZ-DD doesn't work after assembly and testing, take a deep breath, get out a beverage to clear you mind, and start troubleshooting…

Check Your Solder Joints

The very first thing you should do is to check out all of the solder joints under a lighted magnifier. The most common reason for things not working are solder bridges, i.e. putting too much solder on the pads and shorting two adjacent pads together. You can also get into problems by bridging pads with "vias" on the board, the smaller holes that don't have any components soldered to them. Many of the pads are very small, so it doesn't take much solder to get a nice "tented" solder joint. If you get a solder bridge, heat it up and use a solder wick or a vacuum bulb to remove the excess; afterwards, we recommend resoldering the joints. Note: NEVER use "canned air" or compressed air to "blow away" excess solder. The resulting splatter will almost always cause more damage than the original solder bridge, and if you get solder splatter under the baro chip there's no easy way to fix it.

Another thing to look out for is "cold" solder joints, they look dull and blobby compared to a nice shiny "tented" solder joint. If you have a cold solder joint, it won't conduct well; at the low power that the EZ-DD uses this could easily keep things from working. If you have a cold solder joint, heat it up and put just a little bit of solder on it, the main idea is to get a little more flux on the joint. If there's too much solder, use a fine solder wick or a vacuum bulb to remove the excess, then heat it up and resolder the joint.

Check Your Component Polarity

Most of the components aren't polarized, with some notable exceptions. The outline of the parts is silk-screened on the board, so you should be able to see readily if you have a component soldered in backwards.

If you inserted a component incorrectly, you will have to carefully unsolder it, clear any solder residue from the holes, and resolder it. If you find that a component was soldered incorrectly, you will have to use a vacuum bulb or vacuum desoldering tool to unsolder it. We cannot stress enough that you need to check the orientation of the parts ***before*** you solder them. The Eggtimer EZ-DD Limited Warranty does not cover damage to a component while attempting to unsolder it, so make take your time and make sure you get it right before you solder.

Check Your Battery & Connector

Make sure that you are using one of the recommended batteries to test with, we recommend using a NEW 9V battery, type 6LR61 (those have six "AAAA" cells welded together, the other kind have six "stacked" cells pressed together.) Make sure that you have the polarity correct: The RED wires must go to the "+" side and the BLACK wires must go to the "-" side.

If It Still Doesn't Work…

There is, of course, always an outside chance that you have a bad component. Each component is tested at the factory, but it is always possible that something may be wrong; there may be a bridge on the PC board itself, etc. If you have gone through all of the troubleshooting steps and the board still doesn't work, let us know at support@eggtimerrocketry.com . A high-resolution picture (5 megapixel or better) of both sides of your circuit board and a description of the problem would be very helpful…

Troubleshooting Tips (in approximate order of likelihood)

No 9V output when the Button is Pressed

- CD4011BE chip backwards
- CD4011 BE missing a solder joing
- Wrong capacitors or resistors in place (check the bands/markings)
- Bad solder joint on the capacitors/resistors
- Bad solder joint on the button
- 9V battery clip soldered backwards

No 3.3V output when the Button is Pressed

- Bad solder joint on the regulator
- 10 uF tantalum capacitor mounted incorrectly (does the "+" marking match the board?)
- 10 uf "106" capacitor mounted incorrectly
- Wrong part in the "FET" position (it should be marked "LU024N")
- FET mounted incorrectly (does the metal tab match the "bar" marking?)

No Beeps When the Button is Pressed

- Check the polarity of the processor
- Check the polarity of the buzzer
- Bad solder joint on the processor pads
- Incorrect battery polarity, or bad solder joint on battery connector pads
- Bad solder joint on the capacitors
- Incorrect resistor/capacitor mounted (check the markings!)
- Bad solder joint on the resistors
- Bad solder joint on the buzzer
- Tantalum capacitor backwards (does the "+" symbol match the board?)

Light doesn't come on when performing deployment test

- Inadequate vacuum… hold the vacuum hose close to the board, for at least 10 seconds
- Bad solder joint on the BJT or FET transistors
- BJT or FET transistors backwards
- BJT transistor in the wrong place (they are marked "MJG 122")
- FET transistor in the wrong place (they are marked "LU024N")
- Bad connection between your EZ-DD and the lights (check your wiring/connectors)
- Bad light bulb and/or socket (try another one… there's plenty of them on a string!)