NTP clock

Build an NTP-based clock using a microcontroller and LCD display.

Part 2 – Builder's Guide

Bruce E. Hall, W8BH

After you've breadboarded your project, you may want to build it in a more permanent fashion. Here are a set of instructions for building the NTP Clock using a circuit board designed here at W8BH. Read on if you are interested in building this inexpensive and useful device.

Not a Kit.

Forgive me for starting these notes with a disclaimer: this is not a kit, I am not selling anything, and there is no guarantee of success or suitability for any particular purpose. Still interested?

The PCB.

The first step is to order one or more of the printed circuit boards. To order a circuit board from a manufacturer you must provide a digital set of design files, called 'gerbers'. You can obtain the gerbers on my github account.

Almost all board fabricators' websites allow gerber-file uploads. Choose the manufacturer you like. I've used OSH Park (USA, excellent quality) in the past and have been very satisfied. For this project I used JLCPCB (China, hobby-quality). Choose the default 2-layer options, get your quote, and choose whatever shipping option you want. At JLCPCB I paid $7.30 + shipping for a set of 5 boards.
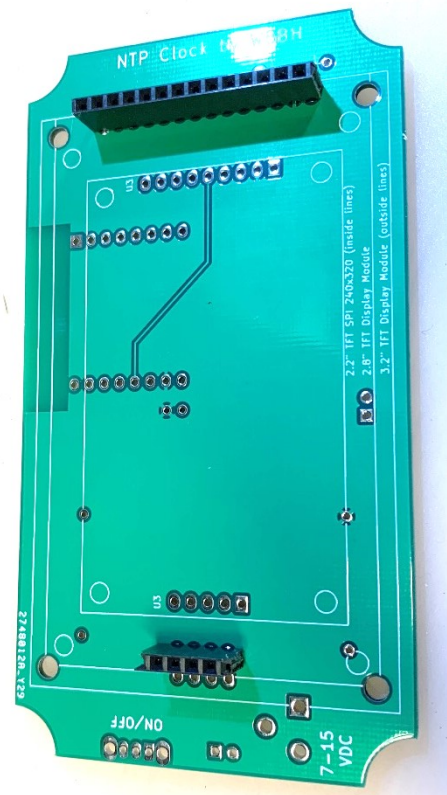
Step-by-Step building.

Before you start, make sure to obtain all the components you need. It is frustrating to get half-way through a build, only to realize that you are missing a key component!

You will need a decent soldering iron, solder, and a voltmeter. You should be familiar with soldering and have successfully kitted other projects using ICs and other through-hole components. You should have sufficient lighting, magnification, and workspace area.

As I mentioned at the start of the series, I believe in starting small. Build the smallest something that you can and test it. Don't forge ahead until you are sure everything is working. In this build we will start with the power circuit on the left-hand side of the board.   We will gradually work our way across the board, testing as we go.

Bill of Materials.

Part	Qty	DigiKey	Mouser	Prices/Other
2.2, 2.8. or 3.2" TFT Display Module, ILI9341	1	n/a	n/a	Search Ebay 2.2" Search Ebay 2.8" Search Ebay **3.2"**
Wemos D1-Mini	1	n/a	n/a	Search Ebay ($3-$6) Search Amazon
LM2596 Buck Converter Board "HW-411"	1	n/a	n/a	Search Ebay ($1-$2) Search Amazon
Barrel Jack PJ-102A	1	CP-102A-ND	490-PJ-102A	Digikey $0.64 Mouser $0.64
SPST slide switch, side mount (C&K 0S102011MA1QS1)	1	CKN9560-ND	611-OS102011MA1QS1	Mouser: n/a DigiKey: $0.39 Jameco $0.49
Shorting block, optional	1	952-2169-ND		Jameco $0.15 Digikey $0.19
0.1" female header	2	HDR100IMP40F-G-V-TH-ND		Digikey $1.09
100nF MLCC capacitor (Vishay K104K10X7RF5UL2)	1	BC5228CT-ND	594-K104K10X7RF5UL2	$0.26
Misc: 0.1" male header pins, 10K resistor, mounting hardware, 9VDC power adapter with 2.1mm DC jack,				

Next, order the parts that you need for the build, using the table above. Please note any part can go out of production, or change in specification over its lifetime. Please check the list of miscellaneous components. Do you have these already? If not, add these to your list.

Step 0: Preparation.

o   Review the NTP clock article. Program your D1 mini with a current copy of the sketch.   Satisfy yourself that the D1 mini is programmed correctly.

o   Obtain the PCB and necessary parts as outlined above.

o   Familiarize yourself with the circuit board, noting the locations of the power circuit, microcontroller, and display. If anything is unclear, refer to the construction photos below.

o   Read through this guide, including the notes section at the end, before picking up your soldering iron.

o   Consider how you intend to power the board.   The voltage input is 7-15 VDC with a current draw of about 150 mA. I recommend a 9 VDC power adapter. The power connector should be a 2.1mm barrel jack, center pin positive. A second input is provided for battery or other external power.

o   Consider which display(s) you intend to use with this board. Three different sizes are supported.

o   Consider if you are going to enclose the board, and obtain the appropriate mounting hardware and enclosure material. The jacks and switches are placed on the left edge of the board to facilitate enclosure design, and the mounting holes are large enough (3.2 mm) to accommodate M3, M2.5 or 4-40 hardware.   If you are facile with 3-D printing this might make a good project

o   Finally, inspect your PCB under magnification for any obvious defects. If none are found, connect an ohmmeter across the pin holes for C1, as shown, which correspond to the +5V and ground planes of the board. There should be infinite resistance = no short between power and ground.
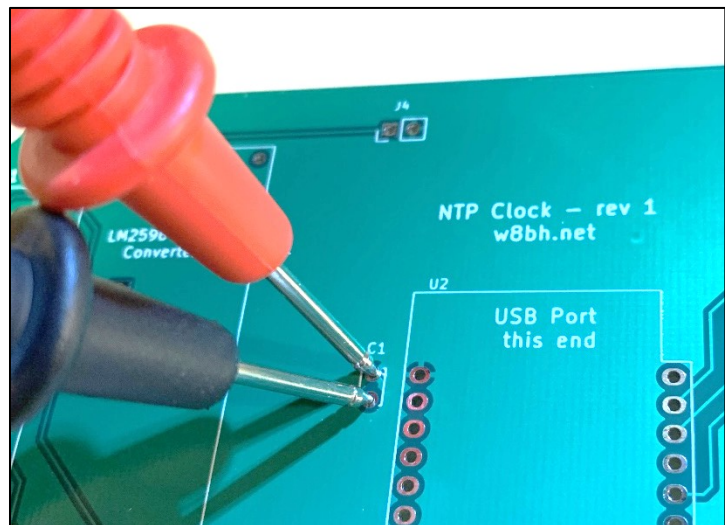
Step 1: Install the Display Headers

o Before going any further, decide which size display(s) you might use for this clock. You should add headers for the display(s) now, because components on the other side of the PCB may make later modifications difficult.
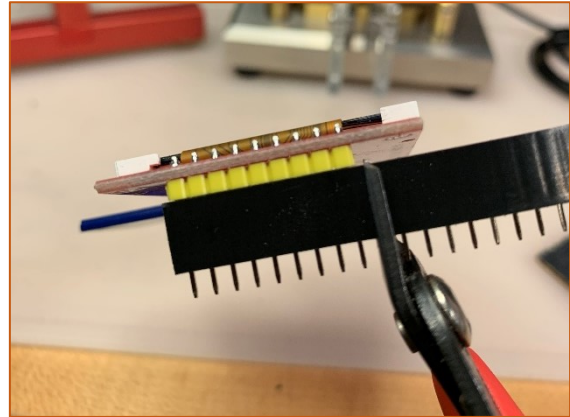
o Cut two lengths of 0.1" female header to match the display pins. There is no need to count pins. Temporarily connect a strip of female header to the display and cut at the desired spot. It is difficult to make a perfect cut between two pins using diagonal cutters, so try this instead: cut directly over the next pin, sacrificing it. The metal pin will fall out, leaving excess plastic on the side you just cut. You may file/sand this down, if you like, or leave it as is.
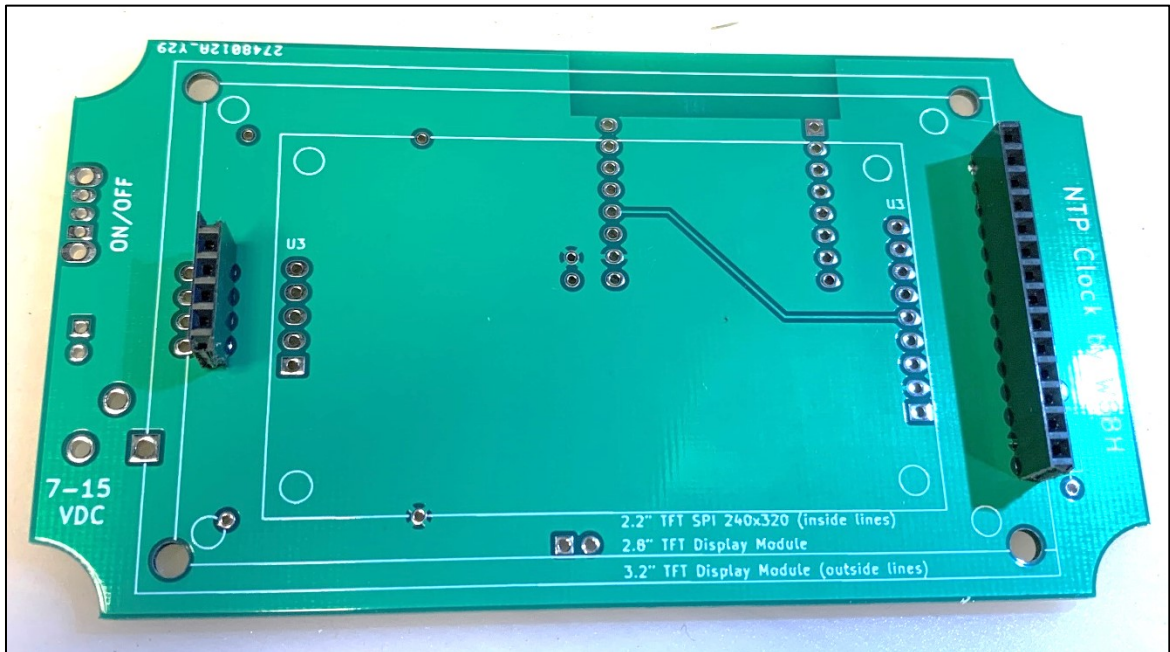
o Attach the headers on the display, which will hold them square, and then solder the headers to the _front face_ of the PCB. The above photo shows how the headers will look for a 2.8" display. Remove the display and set aside. Repeat for each size of display you intend to use.
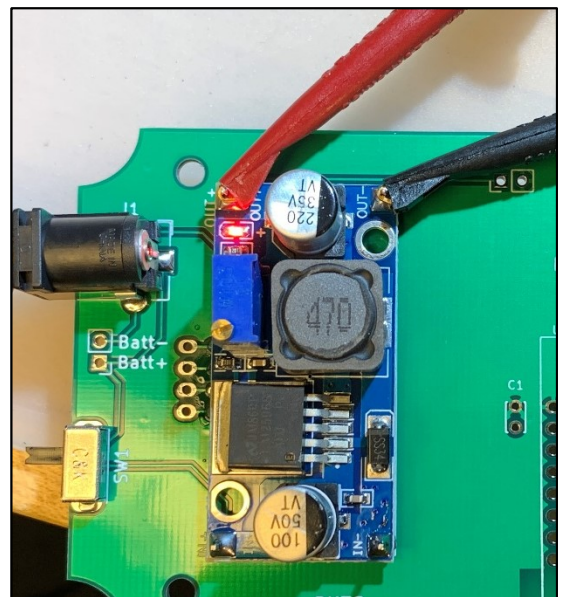
Step 2: Power Supply.

- o Orientation: Most of the parts in this project are soldered to the component face of the board – as shown here. When finished, these components will face rearward/downward. The display, mounted on the other side of the board, will face forward/upward
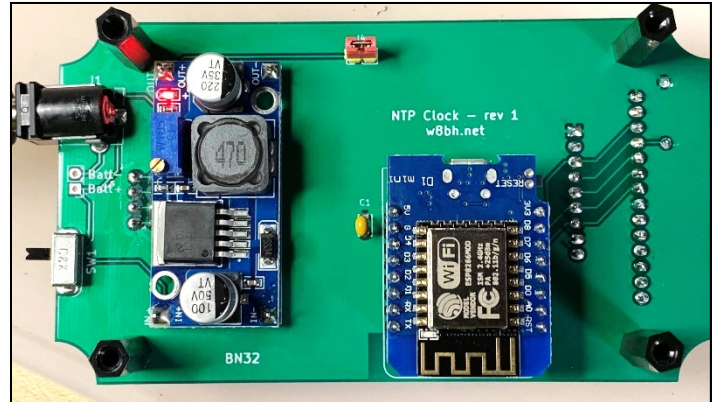
- o Solder the barrel connector and on/off switch in their correct positions (shown above) along the left side of the board. The switch is oriented such that the black-plastic slider is facing leftward, away from the board.   Put the slider down into the OFF position

- o Solder the buck converter module using 4 individual male header pins: Orient these pins with the longer end "up", as shown in the photograph. Place the converter on the loose pins. Make sure that the word 'IN' on the converter matches the word 'IN' on the PCB. Solder the top-side first, then flip the board over and solder the bottom-side. Don't clip the top two header pins; we will use these to check the voltage output.
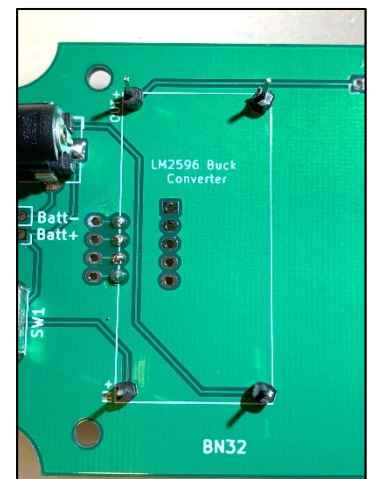
- o **Test #1:** plug your DC power supply into the barrel connector.   Briefly turn the switch up to the ON position. The LED on the buck converter should light up. If not, turn the switch off and disconnect power.   If the LED lights then the converter is getting power. Clip your voltmeter leads across its OUT- and OUT+ pins as shown.

- o Use a jeweler's screwdriver to adjust the blue square potentiometer until you get a reading of 4.9 to 5.0V. Turn the switch off and unplug your power adapter.

Step 3: Power Disconnect

○ The power disconnect is a 2-pin shorting block that allows you to temporarily disconnect the power supply from the rest of the circuit. It can be useful if you want to adjust the buck converter voltage output after the display and microcontroller have already been fitted. It is optional.

○ Cut and install a 2-pin male header at the bottom of the board where it says "5V DISCONNECT". The photo shows a yellow header in this location with a red shorting block.

○ Install a shorting jumper across the pins you just installed. IMPORTANT: the circuit board will not have power unless you install the jumper. Don't omit this jumper. (If you don't want the disconnect, solder a wire between the two jumper holes. You cannot leave these holes empty.)
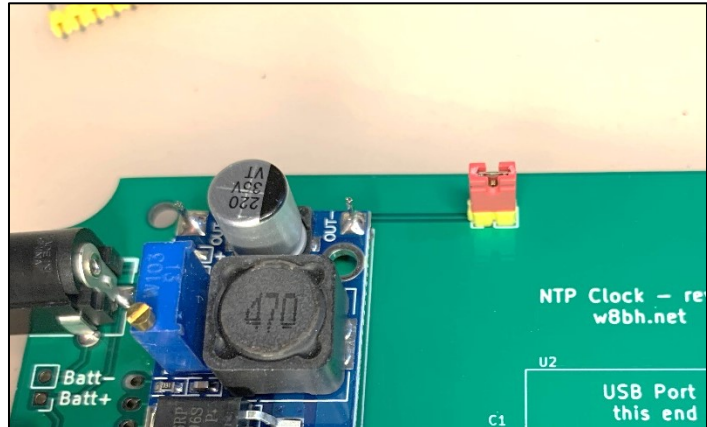
Step 4: Install Capacitor C1 and Resistor R1

○ Capacitor C1 is installed near the center of the board.

○ R1 is a 10K resistor which can be mounted either on the front or rear of the board. See photo for locations of C1 and R1. (You have not yet installed the microcontroller shown in the photo.)

○ **Test 2:** Temporarily reinstall your display, then reconnect your power adapter and briefly turn the switch ON. The display backlight should light. Does it? If not, see Troubleshooting below. Remove the display, turn the switch off and unplug your power adapter.
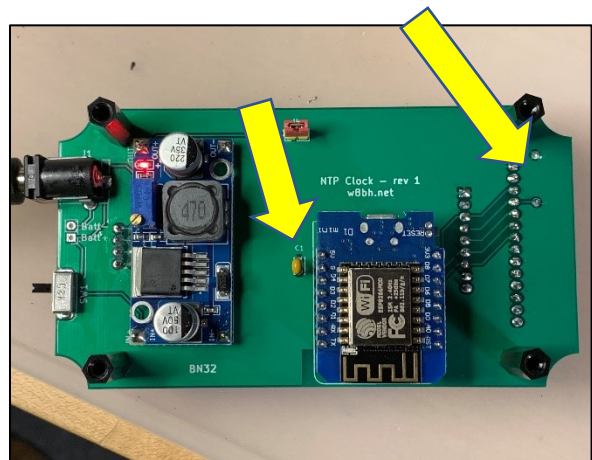
**Step 5: Install the Wemos D1 Mini microcontroller**

○ Check the proper orientation of the microcontroller, as shown in photos above. Looking at the component side of the board (with the DC jack on the left), the microcontroller is oriented with

the USB port at the top and the antenna on the bottom. The small notch in the PCB outline corresponds to the notch in the microcontroller board where its reset switch is located.

- o Just as with the display, cut appropriate lengths of female header pins and fit them to the microcontroller. They should look like the photo at right.

- o Install the microcontroller on the PCB and solder.

- o **Test3:** Fit the display, apply power, and switch on. The blue microcontroller LED should flash three times. Congratulate yourself if you see a working clock! If success is not yet at hand, refer to the troubleshooting section
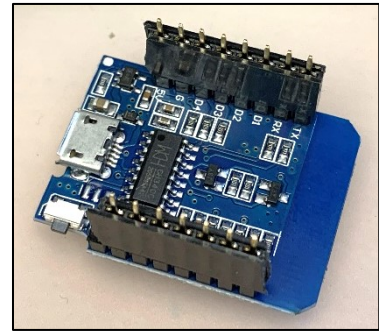
Step 6: Finishing touches

- o Clip off excess leads on both faces of the board.
- o Remove excess flux with isopropyl alcohol and a toothbrush.
- o Think about your enclosure. At a minimum, install standoffs to elevate the components above your work surface.

Additional Notes

As of this writing, there are two PCB variants for this project. The first, "BN32", is a 60x110mm board which has curved cutouts at each corner.   This style may be helpful for enclosures with faceplate mounting pillars at each corner.   The photos used in this guide show the BN32 variant.   The second PCB is "BN33", a 60x94mm board that has traditional rounded corners.   BN33 is cheaper to obtain from PCB houses, such as JLCPCB, that offer special pricing for <10cm boards. Also, check out a third board design with its own construction notes by John Price WA2FZW, which can be found on my GitHub account.

The values used for R1 and C1 are not critical. R1 functions as a logic 1 pullup resistor for the display. Any value from 1K to 50K should suffice.   C1 functions as a filter capacitor for the microcontroller, reducing noise on the power lines from the buck regulator and display.   100nF (0.1uF) is a good value, but 10nF to 1uF would suffice.

The mounting holes are 3.2mm in diameter. I like to use M2.5 mounting hardware, which provide sturdiness without being too bulky. 4-40 and M3 hardware will also work well.

For battery operation, connect your 7-15V battery pack to the holes marked "Batt+" and "Batt-". A 6V battery should also work, but will supply the circuit with less than the nominal 5V. Plugging a 2.1mm power connector into the DC jack will mechanically disconnect the battery from the circuit.

The switch is especially useful for controlling a battery powered circuit. If you plan on using external power only, the switch may be unnecessary.   You can omit the switch by soldering a jumper wire between the switch pins.

Troubleshooting

I find most troubleshooting sections lacking. They never seem to include the problem I am experiencing. If you followed the step-by-step approach above you will have a good idea of WHERE the problem is. Here are a few follow-up suggestions to isolate the problem further.

Problem	Suggestions
Test 1: The Buck Converter LED doesn't light	a) Make sure that your power adapter is in working condition and the wall plug has power! b) Use a voltmeter to test across the front and back barrel connector pins. You should see your input voltage. If not, check your adapter again. c) Turn the switch to the ON position and test the voltage across the buck converter's IN+ and IN- pins. You should see the same input voltage as in (b). d) Check the converter output voltage by putting your voltmeter across the OUT+ and OUT- pins. You should see 5V here. If you see some other voltage, try adjusting the blue potentiometer on the buck converter. If you have a normal input voltage above but no output voltage, your converter may be faulty.
Test 2: The display backlight doesn't light	a) Make sure that you have installed the shorting jumper across the J4 ("5V disconnect") pins. Unless these pins are shorted, the 5V output from your buck converter will not pass to the rest of the circuit b) Remove the display and redo Test #1. If the buck converter LED lights then you have power. Make sure it is 5.0V. c) Insert a small resistor lead or male header pin into the Vcc and GND pins of the display (these are pins 1 & 2). Connect your voltmeter to these pins - you should see 5V here. d) Now move the lead on Vcc to the display pin 8. You should see power here as well. If the power checks out, reinsert the display. The backlight should light if is there is 5V between the LED and GND pins. If not, you may have a faulty display.
Test 3: The microcontroller LED doesn't flash	a) Remove and reseat the controller board to see if it is a poor connection. b) Check all of the controller solder joints with good light and magnification. c) Carefully check the voltage across the C1 pins. Check the voltage between the controller pin marked "5V" and ground. You should see about 5V at both places. d) The sketch may not be programmed correctly. Reprogram the microcontroller.

The LED flashes 3x but the display is blank.	a) Review the ESP8266 addendum of Part 1, and make sure the User_Setup.h file is correct (Step 3A). Pay particular attention to the pin definitions: they represent GPIO numbers, not numbers of the physical pins. b) Review Part1, and rerun the tutorial sketch for step 3/3A. c) Check the PCB traces between the microcontroller and the display for any bad solder connections or bridges.
The display shows "WiFi starting" but does not show the time.	a) In the sketch, check your WiFi name (SSID) and password, paying attention to capitalization and any special characters. b) Ensure that your WiFi system is working and in range.
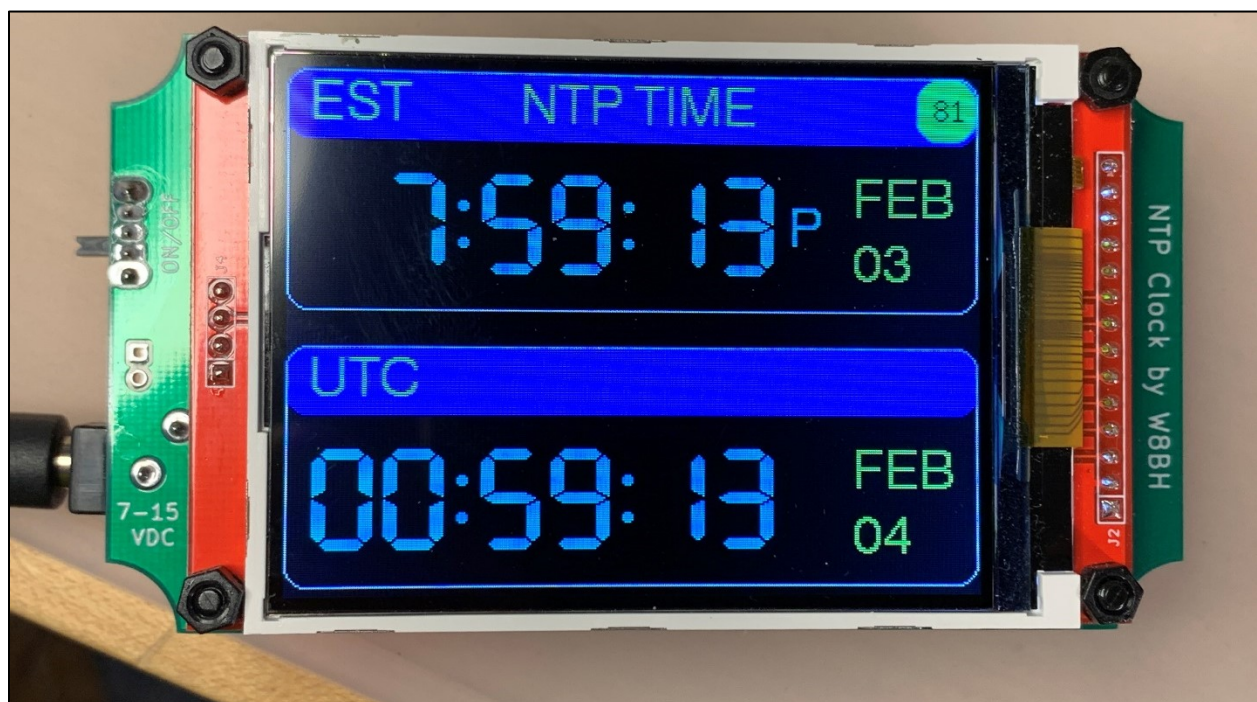
Resources.

Here are links to this project's documentation:

- Part 1: Project Description
- Part 2: Builders Guide
- Latest Source Code

- PCB "BN32" Layout & Gerbers
- PCB "BN33" Layout & Gerbers
- Schematic

I hope you have enjoyed this project as much as I have. Please send me a photo of your work!

73, Bruce.