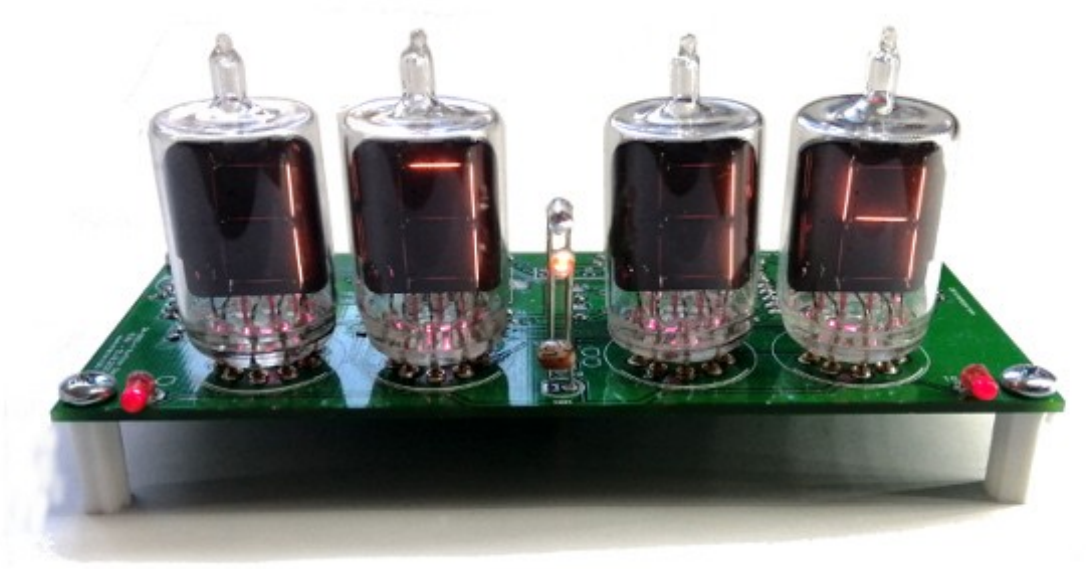
DA2000 4-Digit Numitron Clock “Revision 1.00”

Construction Manual



Index

Table of Contents

About this document

This is the construction manual for the 4-Digit Numitron Clock with DA-2000 tubes, based around the ESP8266, using an Wemos-D1 mini R2. It covers how to build the clock shown above.

If you want to have the user operating manual, please find the appropriate version matching the firmware you are using (the clock will tell you on startup, and it should be marked on the packing slip you received) at:

https://www.nixieclock.biz/Manuals.html

Look for the document called “DA2000 4-Digit Nnumitron Clock User Manual Vxx”, where “xx” is the version you are using.

Contact Information

If you want to get in contact with us, please email to:

nixie@protonmail.ch

We'll usually get back to you right away. We can help you with kits or construction.

We also offer discounts for direct purchases, we save the Ebay fees, and share this with you.

http://www.nixieclock.biz/Store.html

Troubleshooting

If everything does not work as you expect, please carefully look at the tests in the construction steps, and the troubleshooting tips.

At the end of the manual, there is a troubleshooting section, which goes through some of the common problems.

There is also a forum where many problems have already been dealt with at:

http://bit.ly/2Ec0OvW
(redirects to https://www.tubeclockdb.com/forum/wemos-nixie-clock-support-forum)

You can also find other helpful people here!

Component Identification

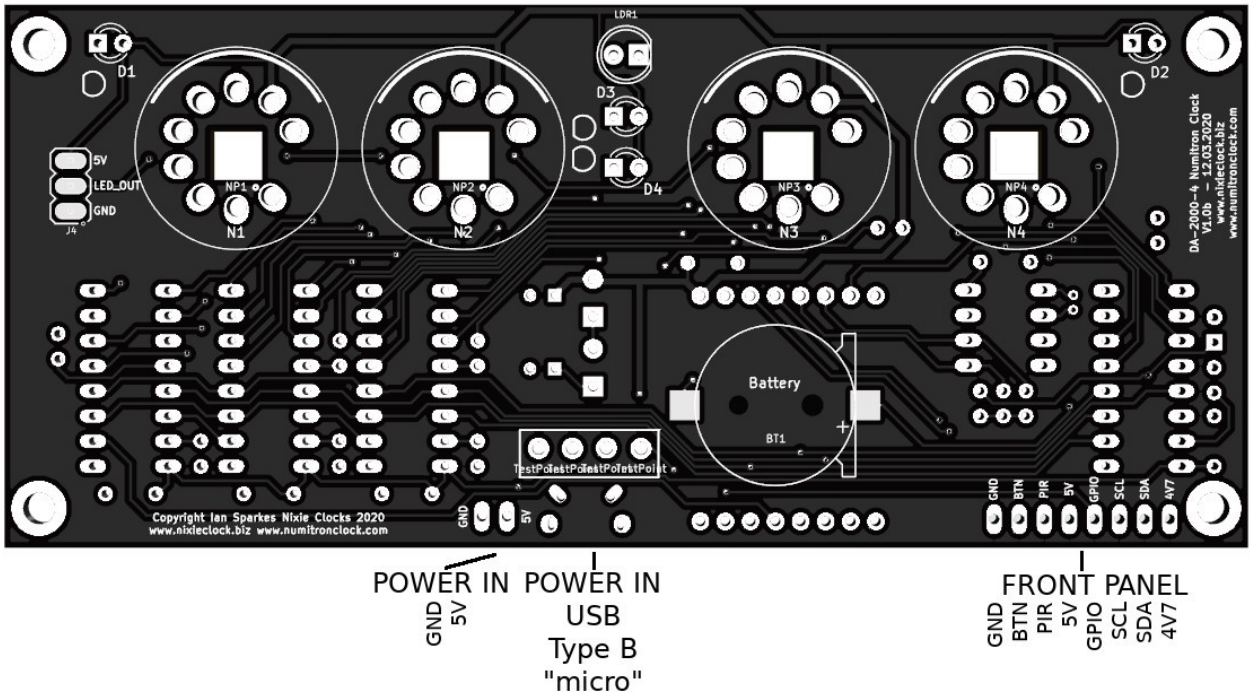
Sometimes it is hard to tell one component from another. Please see the “Component Identification” manual to help you tell one component from another. You can get this document here:

https://www.nixieclock.biz/Manuals.html

Board layout

For reference, the main board layout is as shown (viewed from the top):

DA2000 Version 1.0, 8-pin header, Pin receptacles:



The connections are:

Connector	Description
Power	External power should be applied to the board with this connector. A 5V DC input source can be used, as long as at least 1A is available from the input. The input must be regulated. **GND:** The negative side of the input supply **5V:** The positive side of the input supply, with a voltage of 5V DC.
USB Power In	A standard USB “B” charger can be used to run this board. A charger suitable for a smart phone is ideal.
Front Panel	These are the controls that go on the front panel: The input button and a PIR or microwave motion sensor can be used. **GND:** The “ground”. One lead of the button and one lead of the motion detector are connected to this. **BTN:** The other lead of the button is connected to this input **PIR:** The output pin of the PIR or microwave detector is connected to this pin. **V+:** voltage output to drive a PIR or microwave module. The voltage at this pin is user selectable to 5V or 3V3 using the solder jumper JP1. **GPIO:** A GPIO (General Purpose Input/Output) pin. Currently there is no specified use for this pin, and it is reserved for future expansion. **SCL:** The SCL line of a 3V3 compatible I2C interface. **SDA:** The SDA line of a 3V3 compatible I2C interface. **3V3:** The 3V3 line of a 3V3 compatible I2C interface. **4V7:** Stabilised 4V7 used for driving a microwave detector.

Construction

Preparation

You should have a small tipped soldering iron, some thin (< 1mm) solder, and electronic side cutters.

There are a few SMD components to install, to mount these, thinner solder (<= 0.6mm) and some electronic tweezers are recommended.

Kit Contents

When you unpack the kit, you should find the following contents as listed in the BOM (Bill of Materials). It is best to check the contents before you start, and notify me straight away if you are missing any components.

Please see the appendix to help you identify individual components.

USB connector

Parts List:

J2	USB B connector
J3	2 pin header (see note)

Note!

If you bought the board with the USB connector pre-mounted, you can skip this step!

This is the hardest part of the build. Soldering with surface mount component is quite tricky and requires a very light touch. If you are worried about it, it is possible to get the board with the SMD components pre-mounted and tested.

Spread a small amount of solder paste under the connector and in the mounting holes and put the connector in place. Turn the board over and solder the mounting holes, adding solder to fill them up and fix the connector in place. Add pressure to the connector to make sure that it is close to the board surface while you are soldering it.

Once the connector is physically connected, turn the board over and **carefully** solder the tiny wires with a heat gun or small tipped soldering iron.

Notes:

- There is an alternative power supply to the board possible via J3. If you do not want to use a USB power supply, mount J3 and use that to supply the power instead.

Test Step

Once the connector is mounted, test between the GND and 5V test points to see if there is continuity between the over rails. **If there is**, you must check for solder bridges! Do not move on until you have cleared the short circuit.

Note that solder paste is conductive even before being soldered, and therefore will seem short until you have finished soldering.

Test Step

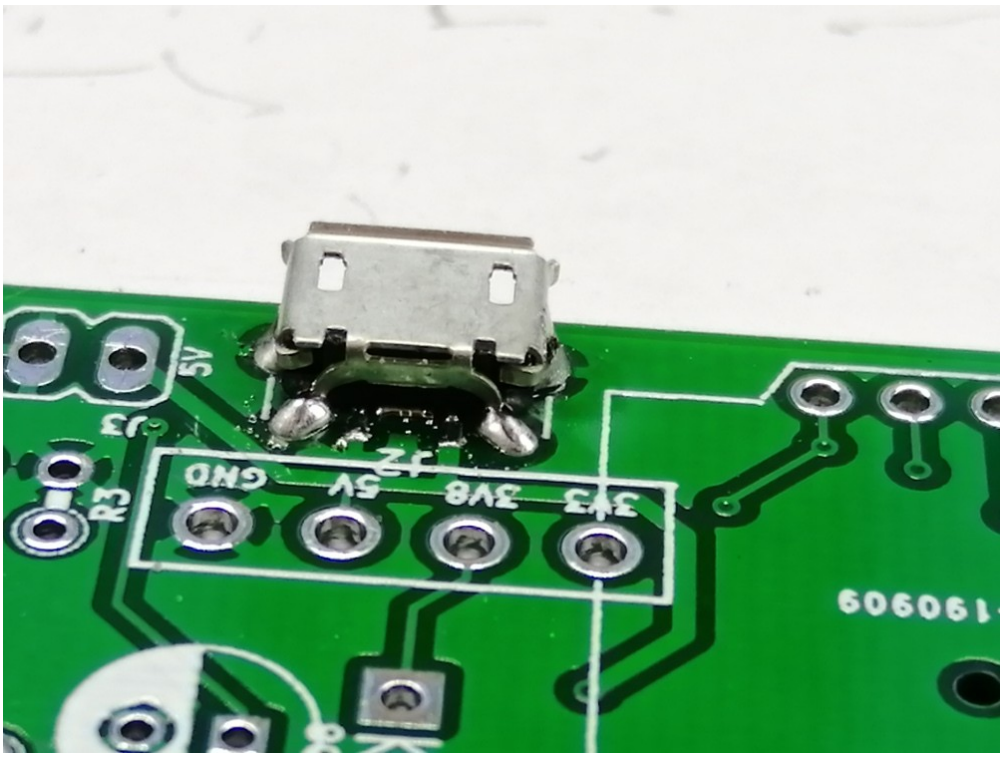
Once you have performed the continuity check, put power into the board via a USB power supply and check that you can read 5V between the GND and 5V test points. **If there is not 5V**, you must check for incomplete solder connections on the right most pin of the 5 tiny pins!

If you want to watch a video of this build step, please see the associated video at:

YouTube: **https://youtu.be/M7vGTBs_UDE**

BitChute: **https://www.bitchute.com/video/4Mf7LMmTkwib/**

After soldering, the socket should look like this:



Back lights circuit

Parts List:

NP1 - NP4	WS2812B

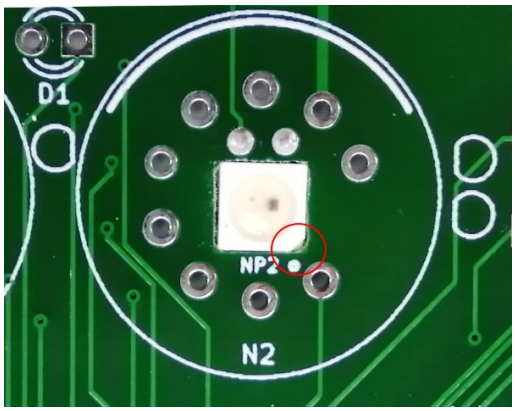
Note!

If you bought the board with the LEDs pre-mounted, you can skip this step. The soldering with surface mount components is not very difficult, but if you are worried about it, it is possible to get the board with the components pre-mounted and tested.

Mount the NeoPixel LEDs and capacitors, making sure you mount the LEDs in the right orientation.

Notes:

- The NeoPixel LEDs should be mounted from the reverse side of the board. If they are a tight fit, enlarge the holes slightly using a jeweller's square file until they fit into the holes without having to force them.
- The LEDs have to be mounted with the tiny notch on the LED matching the cut off corner of the silk screen (see the picture below – the red lines mark the notch and the cut off corner)
- The WS2812B are easily damaged by heat. Please solder them rapidly. A spare is in the parts, just in case. Make sure you don't set the iron above 290°C / 550°F!



Note!

Mount the NeoPixels one at a time, starting with NP4 and testing it after it is mounted. Temporarily attach the controller to the board and power it up. If NP4 works fine, go to NP3. Test after each NeoPixel until all are installed. The NeoPixels are daisy-chained, and any break in the chain will cause all following NeoPixels to not work as expected.

Test Step

While you are mounting the NeoPixels, it is useful to test each one as you mount it. Use the controller’s test pattern to check the functioning of each one before moving on to the next.

If you want to watch a video of this build step, please see the associated video at:

YouTube: **https://youtu.be/X6WsMHD7A2U**

BitChute: **https://www.bitchute.com/video/cPR0R1SViteN/**

After you have installed the components, your board should look like this:

Pin Receptacles

Parts List:

PR 1 - 36	1mm Pin Receptacles

We solder the pin receptacles into the PCB so we are able to plug and unplug the tubes. This step puts the receptacles into the holes on the PCB and solders them.

Warning!

Do NOT try to solder the receptacles while they are on the pins of the tube! If you do this, you will solder the receptacles to the pins, because the pins of the tube are tinned! This will cause the receptacles to become useless.

Notes:

- Apply a small amount of solder paste to the base of each receptacle from the **rear** of the board, and reflow the solder.
- It is also possible to use solder instead of solder paste, but solder paste gives a better result.
- Do not try to solder the receptacles from the top (tube) side of the board. You will almost certainly end up with solder or flux in the tube of the receptacle, and that will reduce the effectiveness.

After you have installed the components, the receptacles should look like this:

If you want to watch a video of this build step, please see the associated video at:

Bitchute: **https://www.bitchute.com/video/QwBNm5HBdpdg/**

YouTube: **https://youtu.be/kc0RcaHjnXA**

Note!

If you are **confident** in your electronic skills, you can mount the controller without using the female header strips. If you decide to do this, be aware that problem solving later might be more difficult.

If you mount the controller without the headers, make sure that the clearance between the USB connector on the Wemos board and the tracks on the PCB is sufficient!

Tube drivers

Parts List:

C1 – 4, C13	100nF 5.08mm
C14, C16	220uF 10V
U1 - U4	TLC5916N
R1 - R4	910R
DIP16	16-pin DIP sockets
D7, D8	1N4007
C16	220uF 10V

This step builds the main drivers for the tubes and the power supply smoothing for the Wemos and the drivers. The drivers are 4 identical circuits, one for each tube. The supply for the drivers is derived from the 5V supply by using a diode dropper (D7, D8) to provide an intermediate supply voltage, which means that the logic signals provided by the Wemos (3V3 outputs) are compatible to the drivers.

Notes:

- C14 and C16 must go the right way round. The negative side is marked with a stripe.
- D7 and D8 must go the right way round. The white stripe on the body must go into the hole with the square outline.
- Put the sockets in with the notch facing the notch marked on the silk screen. When you put the drivers into the sockets, align the notch on the chip with the notch on the board and the socket.
- Make even more sure that the chip goes into the sockets the right way round. The chip also has a notch, and this should match up with the notch on the socket **and** on the board.
-

Hint: The 220 uF capacitors

The electrolytic capacitor has a stripe on it to denote the **negative** side of the capacitor. The positive side of the capacitor (which goes into the “+” on the board) is the other one!

220uF capacitor “stripe”

Test Step

Once you have populated the components, power on the circuit and test the voltages at the 3V8 and 5V test points.

You should be able to read the expected voltages. It is not important if the 3V8 test point does not show exactly 3V8. It should be between 3V6 and 4V2.

Test Step

Once you have populated the components, you can attempt the first test with the tubes.

The DA-2000 tubes should fit snugly into the locations on the board.

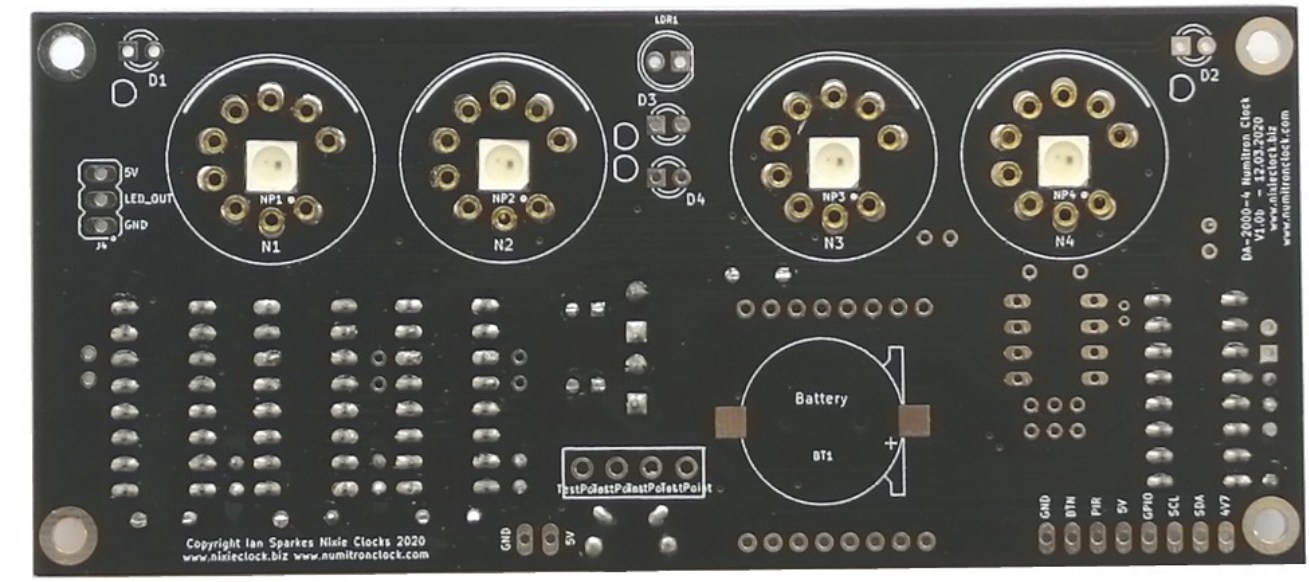
Power on the board and wait for the startup sequence to finish. After this, you should see the tubes counting from 0 to 9 repeatedly.

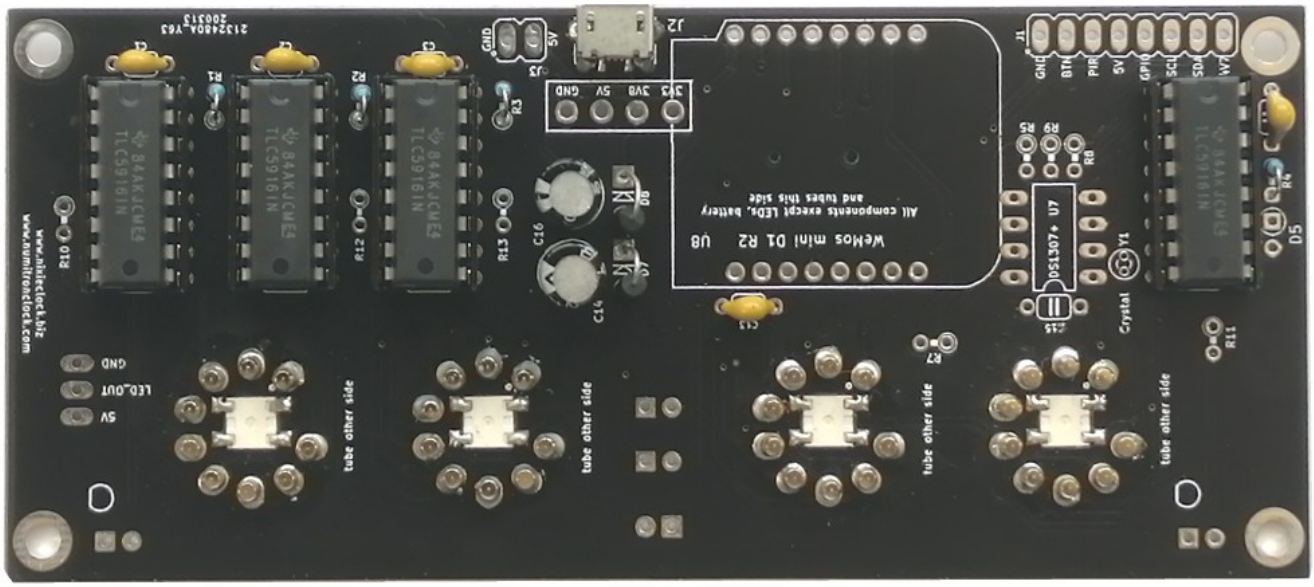
If you want to watch a video of this build step, please see the associated video at:

Bitchute: **https://www.bitchute.com/video/2PjnjXAvtNiM/**

YouTube: **https://youtu.be/o1IKIa0hg8k**

After you have mounted the components, the board should look like this:





LED Separators and indicators

There are different types of board, but each of them is identical apart from the type of tubes that are installed.

This step installs the separators, the indicator LEDs and the LDR.

Between the seconds and minutes tubes, and the minutes and hours tubes there are “separator” LEDs. There are also two “status” LEDs which can be used to convey information about the clock.

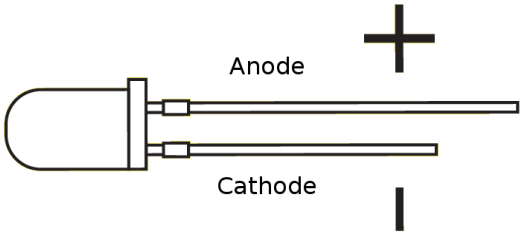
Parts List:

R10 – R13	10k
D1, D2	3mm Red LED
D3, D4	3mm Orange LED

Notes:

- The Red LEDs D1 and D2 are the indicator LEDs, and should be mounted facing forwards with a slight bend in the leads.
- The Orange LEDs are mounted with the head bent to face forwards, one above the other.
- be careful to get the LEDs the right way round. The silk screen shows you the side with the “flat” on it. (See hint below).
- Some LEDs do not have an easy to identify flat on the side of them. In this case, the **short** lead is the one that has the flat on it. (See hint below).
- Bend the head of the orange LEDs over so that they shine forwards between the tubes.

The LED should look something like this:



Test Step

Plug the display board into the main board and power up the circuit again. This time let the startup sequence complete. Please see the section on “Setting the WiFi information” for details of how to log into the clock and set it up.

You should see the tubes light up.

If you want to watch a video of this build step, please see the associated video at:

Bitchute: **https://www.bitchute.com/video/QQ3c2kdL5ok2/**

YouTube: **https://youtu.be/JavRZ9_Rw58**

LDR and front panel connector

The LDR detects the ambient light conditions and dims the display to match them. This increases the tube life and makes sure that it does not “glare” at night. The front panel connector allows you to connect a push button to the case and attach a motion detector.

Parts List:

R7	10k
LDR1	GL5516
J1	8 pin header

Notes:

- Mount the LDR either so that it faces forward or upward. If you wish, you can also mount the LDR on the outside of the case using flying leads to the connector.
- Decide if you want to install the header strip, or attach the wires for the button and PIR/microwave motion detector directly to the board. If you attach the header, you might find the clearance quite tight. Note that you can also trim down the length of the header pins once they are mounted to improve the clearance.

If you want to watch a video of this build step, please see the associated video at:

YouTube:

BitChute:

RTC circuit

Parts List:

R8, R9	10k
Y1	32.768kHz crystal
BT1	CR1220 battery holder
U7	DS1307+
C15	100nF

Normally you will install the Real Time Clock (RTC) on the board as a back up option for the case that the WiFi time source is not available. For more information about this option, please see the User Manual.

Notes:

- Unfortunately, it is not allowed for us to send lithium batteries to some locations. If you do not receive one, it is because of these regulations. Please source one locally. Sorry about that.

If you want to watch a video of this build step, please see the associated video at:

YouTube:

BitChute:

PIR / Microwave motion detector

Parts List:

PIR	(optional) HC-SR505 PIR module
MW	(optional) RWCL-0516 microwave module
R5	(optional) 100R
D5	(optional) 4V7 Zener

This step attaches an optional PIR detector or microwave motion detector module. This turns the clock off when no one is there to see it, and therefor extends the tube life even more.

Warning!

The PIR input on the Wemos is only 3V3 tolerant. You can damage the Wemos if you use a PIR module or Motion Detector that has a 5V output.

Warning!

The microwave detector is very sensitive to supply voltage fluctuations. The optional Zener voltage stabiliser is only needed if you find problems with your supply. A good quality supply should not need this.

Problems will manifest as the motion detector triggering even when no motion is happening.

Notes:

- Set the solder jumper JP1 for the voltage that is needed by the module. This is either 5V or 3V3. Both the modules shown below require 5V power input but provide a 3V3 output signal.

HC-SR505 PIR module

RWCL-0516 microwave module

Setting the clock up

Once you have got this far, you have finished the construction of the clock and are ready to set the clock up.

Please see the user manual for full details about how to do this!

The manuals are at:

https://www.nixieclock.biz/Manuals.html

We hope you enjoyed building the clock, and are always interested to hear what you think!

Troubleshooting

The clock just counts but does not show the time

In this case you are probably in “First Start” mode, which is there to allow you to test all the segments in the tubes, and all the colours in the LEDs.

To exit this mode, please fit a button to the clock and press it when the clock shows “88:88”.

For more information about the meanings of the LEDs on start up, please see the User Manual.

Parts list / BOM

Here is the list of the parts needed for the main board:

Part	Qty	Value	Packed	Mounted
PCB	1	Main PCB	☐	
N1,N2,N3,N4	4	DA-2000	☐	
N1,N2,N3,N4	4	IV-9	☐	
C14,C16	2	220uF 10V	☐	
C1, C2, C3, C4, C13, C15	6	100nF 5.08mm	☐	
U8	1	Wemos D1 R2	☐	
J2	1	USB_B_Micro	☐	☐
NP1,NP2,NP3,NP4	4	WS2812B	☐	☐
LDR1	1	GL5516	☐	
D1, D2	2	LED 3.0mm Red	☐	
D3, D4	2	LED 3.0mm Clear	☐	
BT1	1	CR1220 Holder	☐	
R1,R2,R3,R4	4	910R 1/8 W	☐	
R7,R8,R9, R10, R11, R12, R13	7	10k 1/8 W	☐	
U1, U2, U3, U4	4	TLC5916	☐	
Y1	1	32.768kHz	☐	
U7	1	DS1307+	☐	
D7,D8	2	1N4007	☐	
J3	1	Header 02X01	☐	
J1	1	Header 08X01	☐	
Socket 16	4	16Pin DIP	☐	
Socket 8	1	8Pin DIP	☐	
BT1 (front panel / test)	1	Switch	☐	
Pin Receptacles (option)	36	H3161	☐	

Revisions:

V0001: 12Apr2020: Initial version
V0002: 25Aug2021: Add troubleshooting section