Wemos 4-Digit Nixie Clock "Revision 1"

Construction Manual
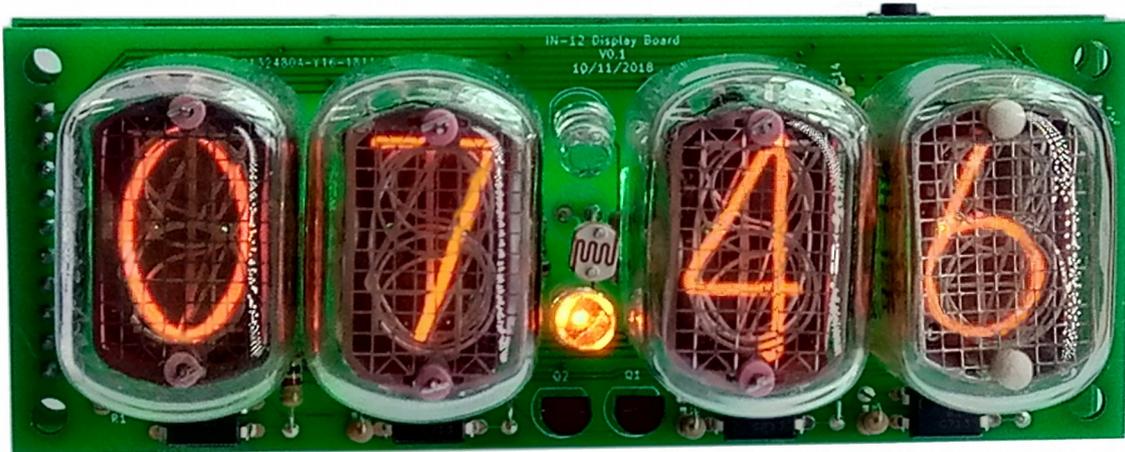
Index

Table of Contents

About this document

This is the construction manual for the 4-Digit Nixie Clock, 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 "4-Digit ESP8266 Nixie 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!

Safety

The voltages produced by this clock are generally considered safe. However, please note that voltages of up to 170V are produced, and you should take care not to short out portions of the circuit, or leave the clock unattended for long periods of time.We decline any responsibility in the case of injury, loss or death.

REPEAT: If you are not sure, please do not use the clock.

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):

The connections are:

Connector	Description
Power In	External power should be applied to the board with this connector. A 7.5V – 12V DC input source can be used, as long as at least 1A is available from the input. **GND:** The negative side of the input supply **VIN:** The positive side of the input supply, with a voltage of 7.5V - 12V DC.
Motion Detector / Button	These are the controls that go on the front panel: The input button and a PIR or microwave motion sensor can be used. **Note** that you only need to provide a button if you don't want to use the on-board one. **GND:** The "ground". One lead of the button and one lead of the motion detector are connected to this. **BTN1:** The other lead of the button is connected to this input (if you want to use an exernal button) **PIR:** The output pin of the PIR or microwave detector is connected to this pin. **VCC:** 5V output to drive a PIR or microwave module.

To make this clock work, you also need to have a display board. Currently we have the following small format display boards available:

IN-12 Display board
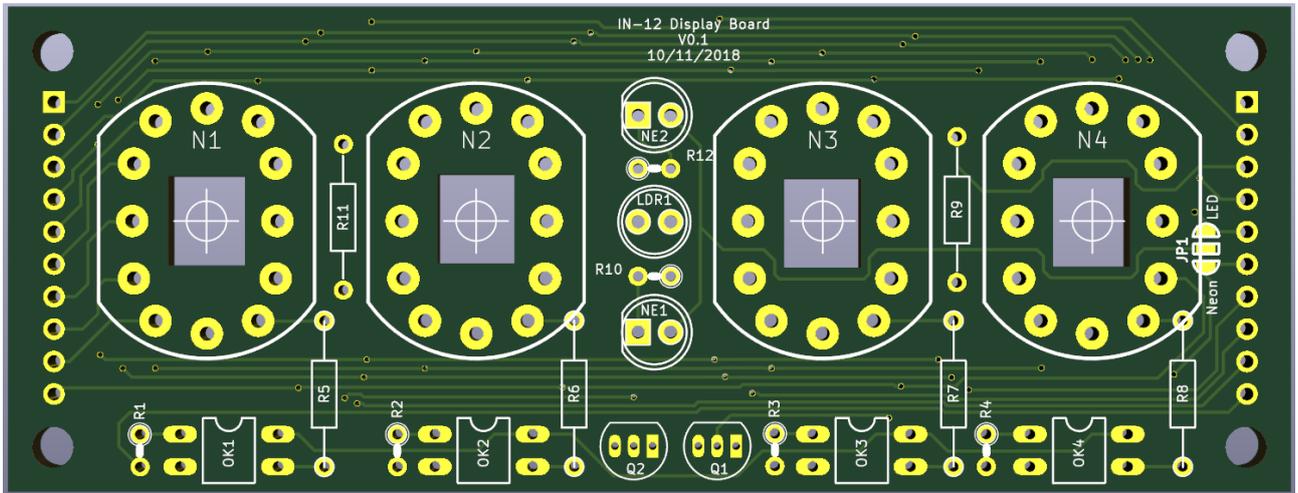
IN-2 Display board
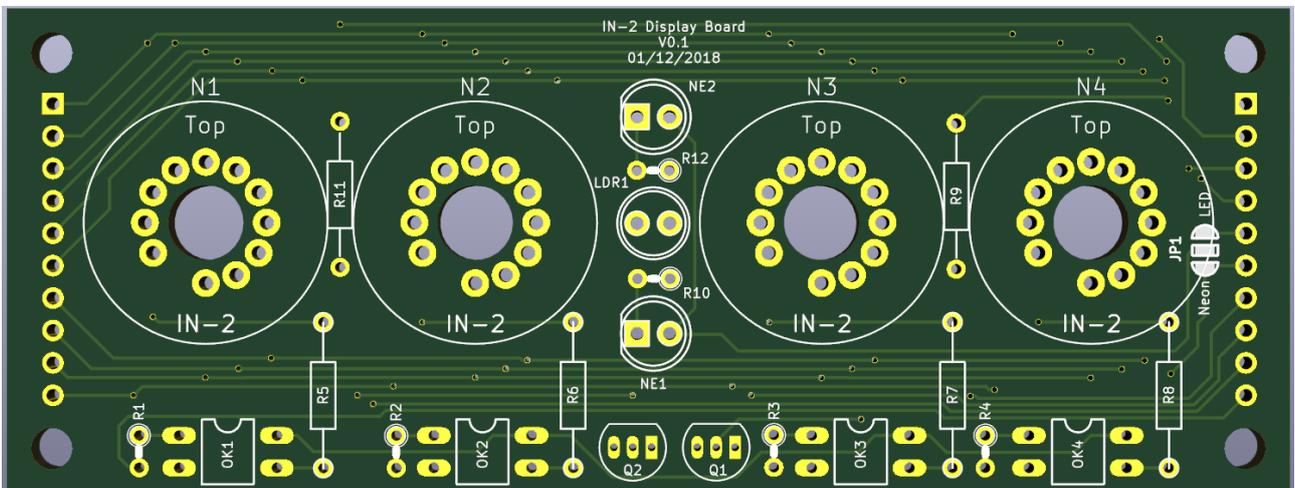
The boards do exactly the same job and are interchangeable.

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.

Back lights circuit

Parts List:

LED1 - LED4	WS2812B
C1 - C4	100nF

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

Notes:

- The LEDs go on the reverse side of the board from all other components. You can't mount them on the wrong side (they are surface mount), but please remember to turn the board over when you mount the rest of the components later

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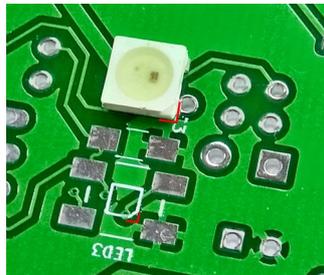
If you want to watch a video of this build step, please see the associated video at:

YouTube: https://youtu.be/Sxh1MZY2azA

BitChute: https://www.bitchute.com/video/T28KkicTEtsn/

After you have installed the components, your board should look like this:
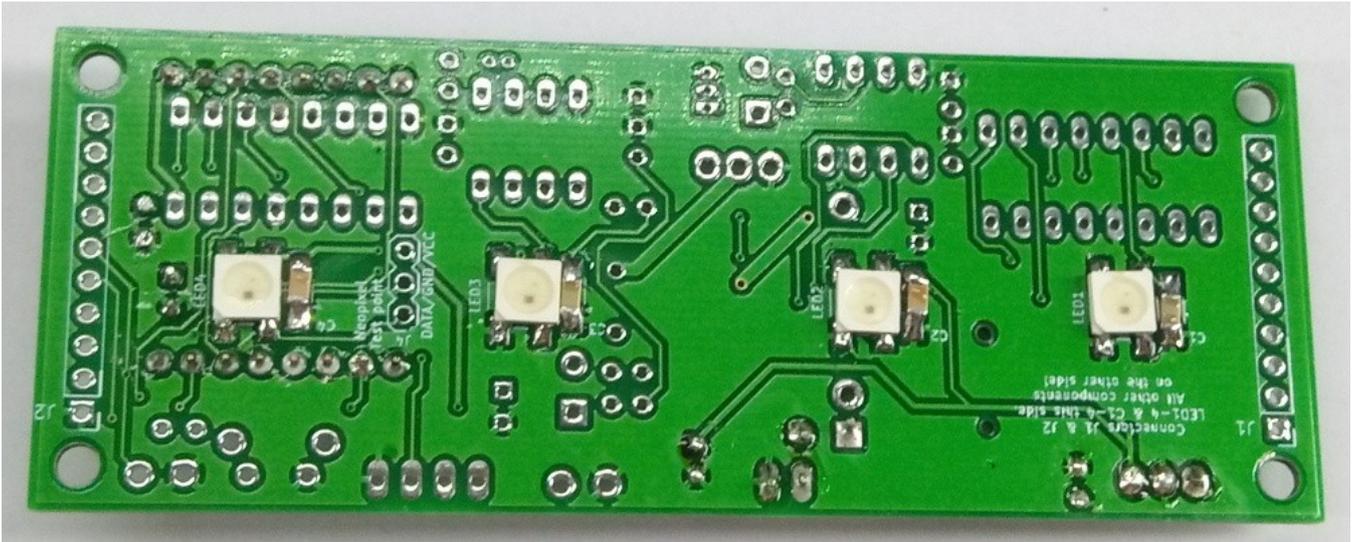
Low Voltage Power Supply

Parts List:

J3	01 x 02 male header
D1	UF4007
C8	100nF
U3	7805
C10	220uF 10V

This part of the circuit mounts the 5V regulator and checks that it provides the expected 5V output.

Put the parts on the board in the marked locations in the order they appear on the list.

Notes:

- C10 must go the right way round. The negative side is marked with a stripe. (See hint).

- D1 must go the right way round. The white stripe on the body must go into the hole with the square outline.

- Put the 7805 regulator horizontally with the metal tab facing the board. Make sure that the metal tab does not touch any of the pads for the 16 pin chip.

Test Step

Once all the components are on the board, hook up the power, and check that the 7805 does not get hot. If you have a lab bench power supply, use this for the first power up, with the current limited to 100mA.

Check also that the voltage is 5V between the "GND" test point and the "VCC" test point and at the power connector.

Hint: The 220 uF capacitor

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"

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

YouTube: https://youtu.be/wdEXpCSUhHo

BitChute: https://www.bitchute.com/video/PsbLpkfRPHwP/

At the end of the low voltage circuit build, your board should look like this:
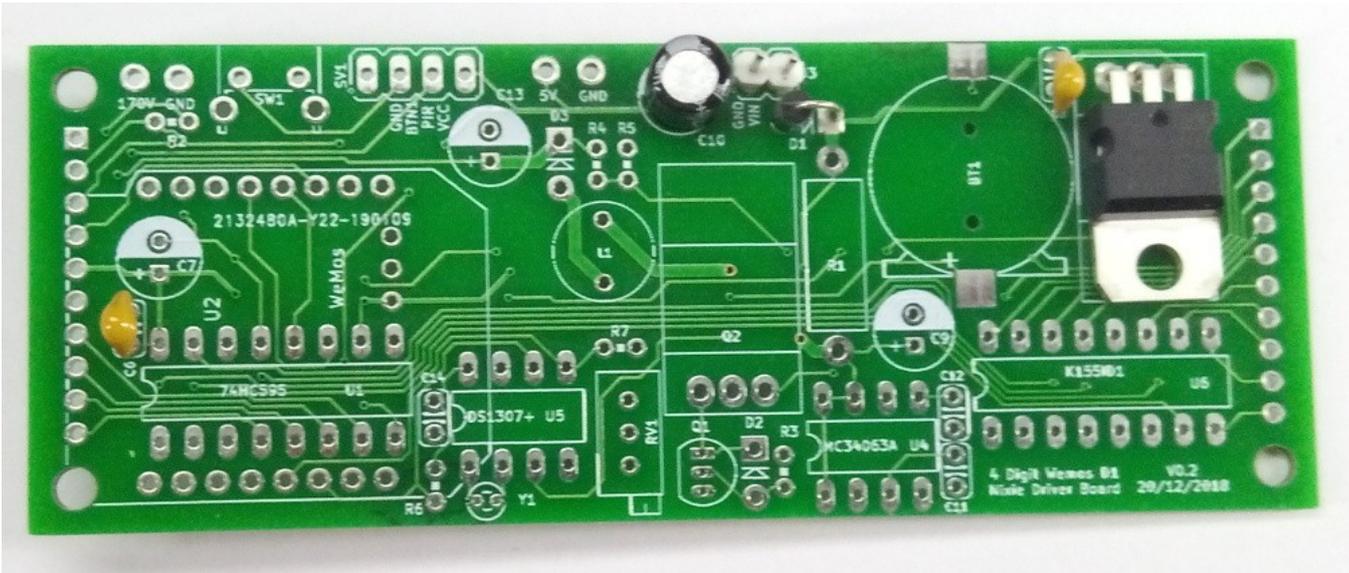
High Voltage Power Supply

Parts List:

Q1	2N3906
C13	2.2uF 400V
R4	390k
DIP8	8-pin DIP socket
U4	MC34063
R3	1k
R5	2.2k
C11	680pF
RV1	5k
C12	100nF
D2	1N5819
D3	UF4007
L1	100uH
R1	0R18 or 0R22 2W
Q2	IRF640
C9	220uF 16V

This step builds the supply for the high voltage necessary to drive the Nixie tubes. It is a boost circuit around an MC34063 chip, but with some modifications to improve the efficiency of the high step-up ratio needed to turn 9V input into 180V output (a 20x boost factor).

The exact voltage is adjustable, and you should set the voltage to 180V using the adjustable resistor RV1.

Notes:

- C13 must go the right way round. The negative side is marked with a stripe. (See hint).

- D2 and D3 must go the right way round. The white stripe on the body must go into the hole with the square outline.

- Don't mix up D2 and D3: You won't get the output voltage if you switch them. They have different tasks to do.

- Put the MOSFET regulator horizontally with the metal tab facing the board. Bend and form the leads before soldering it.

- Q1 must also be orientated as shown on the board silk screen. The flat side of the component should match to the flat side of the silk screen.

- Make sure that you mount the socket with the correct polarity. There is a notch at one end, and this should match the notch on the board.

- 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.

Test Step	Once you have populated the components, power on.
	The MOSFET Q2 should not get hot.
	Check the voltage between the test points "GND" and "170V". You should read a voltage above the input voltage, but it might not be 50V yet. We will adjust this later.

Adjustment	Adjust the voltage so that you read 170 Volts at the test point.
	To adjust to voltage, turn the small screw on the potentiometer. Monitor the voltage as you adjust, so that you arrive at about 170V.

▶	If you want to watch a video of this build step, please see the associated video at:
	YouTube: https://youtu.be/KQIg96uixaU
	BitChute: https://www.bitchute.com/video/zpzq2EF181xf/

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

Controller Circuit

Parts List:

U2	Wemos D1 R2
HDR 1	01 x 08 female header
HDR 2	01 x 08 female header
C7	220uF 10V
C6	100nF

This step puts the Wemos controller on the board and tests that it starts up.

Notes:

- Be very careful to put the Wemos on the board the right way round. You can put it in reversed, and this will destroy it!

- C7 must be installed with the white stripe on the can matching the white outline on the board.

Test Step

Once you have populated the components, power on.

The LEDs should light up, the one for the "seconds" digit should go green, then the 10 X seconds should turn yellow and then blue. You can turn the board off again, the test is passed.

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

YouTube: https://youtu.be/maXvCHh1Q4k

BitChute: https://www.bitchute.com/video/TKW17TousCGw/

Shift register and Cathode driver circuit

Parts List:

U1	74HC595
U6	K155ID1
DIP16	16 Pin DIP socket
DIP16	16 Pin DIP socket
01x10 HDR	10 pin header strip (*)
01x10 HDR	10 pin header strip (*)

(*) See hint

This step installs the shift register (74HC595) which controls the anodes and the cathodes, and the cathode driver (K155ID1).

Notes:

- Make sure that you mount the sockets with the correct polarity. There is a notch at one end, and this should match the notch on the board.

- Make even more sure that the chips go into the sockets the right way round. The chips also have a notch, and this should match up with the notch on the socket **and** on the board.

Hint: The header strips

If you have already built the display board, you can use this to help align the header strips by plugging them into the female connectors on the display board. If you have not yet built the display board, use the display board PCB to help you align the header strips.

Test Step

If you have already built the display board, you can now plug it in 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.

If you do not have a display board, skip this test step. We will come back to it later.

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

YouTube:

BitChute:

RTC circuit

Parts List:

R6, R7	10k
Y1	32.768kHz crystal
BT1	CR1220 battery holder
U5	DS1307+
C14	100nF
R2	10k
SW1	Tactile button

The resistor R2 and the button need to be installed.

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:

Display board

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

Parts List:

R1 – R4	1k
R5 - R8	3k
OK1 - OK4	EL817
N1 - N4	IN-2 or IN-12 tubes
R9, R11	4k7
R10, R12	1k (for LEDs)
R10, R12	220k (for neons) (option)
Q1, Q2	MPSA42
LDR1	GL5516
LED1, LED2	5mm LED
NE1, NE2	4mm x 16mm neon (option)

Notes:
- be careful not to push the tubes too hard against the PCB. This will cause the tube to crack and be destroyed.
- Set the Jumper JP1 depending on whether you are installing LEDs or neons. It's best to do this before you mount the tubes, because it can be difficult to set it once the tubes are mounted.
- Be careful to mount the display board onto the main board the right way up. The opto-isolators are at the bottom of the display board. The power connector is at the top of the main board.

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: YouTube IN-2: https://youtu.be/tcEtGR_f_00 BitChute IN-2: https://www.bitchute.com/video/vWTHm2WpYySk/

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!

Parts list / BOM

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

Designator	Quantity	Designation
L1	1	100uH
Y1	1	Watch Crystal 32.768kHz
C6,C8,C12,C14	4	100nF
C11	1	680pF
R2,R6,R7	3	10k
R3	1	1k
R4	1	390k
R5	1	2k2
U6	1	K155ID1
Q1	1	2N3906
J2,J1	2	Conn_01x10
LED1,LED2,LED3,LED4	4	WS2812B
R1	1	0R18 or 0R22 2W
D1,D3	2	UF4007
D2	1	1N5819
C4,C3,C2,C1	4	100nF
Q2	1	IRF640
U1	1	74HC595
U5	1	DS1307+
U4	1	MC34063A
U2	1	WeMos_mini
SV1	1	Conn_01x04
BT1	1	Battery_Cell
J4	1	Conn_01x03
J3	1	Conn_01x02
C13	1	2.2uF 400V
C10,C9,C7	3	220uF
RV1	1	5k POT
SW1	1	SW_DIP_x01
U3	1	L7805

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

Designator	Quantity	Designation
N1,N2,N3,N4	4	IN-2
R1,R2,R3,R4	4	1k
R10, R12 (for LED)	2	1k
R10, R12 (for neon) (option)	2	220k
OK4,OK1,OK2,OK3	4	EL817C
R11,R9	2	4.7k
J1,J2	2	Conn_01x10
Q2,Q1	2	MPSA42
NE2,NE1 (option)	2	Lamp_Neon
LED1, LED2	2	5mm LED
LDR1	1	R_PHOTO
R7,R8,R6,R5	4	3k

Revisions:

V0001: 18May2019: Initial version
V0002: 02Feb2020: Corrected R1 value in BoM, thanks Bob!