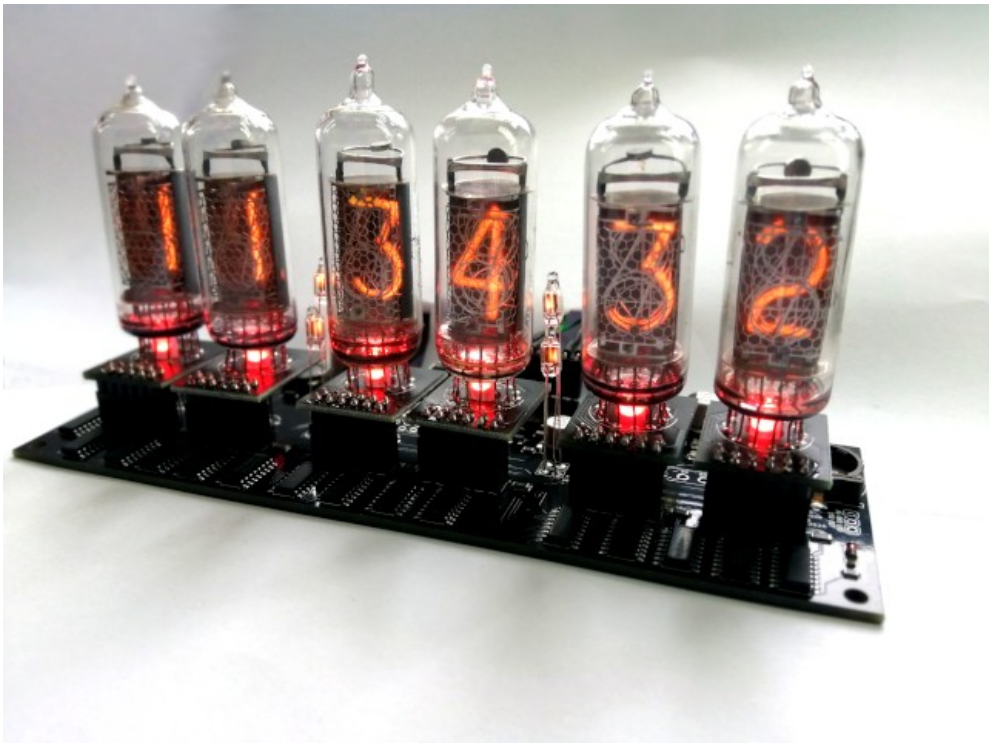
“MNC6”

Arduino Interchangeable Nixie Clock Rev1 & Rev1.2

Construction Manual



Index

Table of Contents

What this document is about

This document is the construction manual for the “MNC6” Arduino based Interchangeable Medium Tube Nixie Clock shown on the first page.

This is a brand new design, and gives you a great features in an easy to build, high quality Nixie Clock. This new design takes care all of the difficult steps of building an expandable Nixie clock, and lets you concentrate on final finishing and case design.

The User Manual for everyday use is in a different document, also available from the documentation site at

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

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.

https://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. If you can't work it out, please get in contact with us. **We guarantee that you will get going.**

Safety

The voltages produced on this board can reach peaks of 200V! Take precautions not to electrocute yourself! If you are not sure what this means, please do not use this clock and return it for a full refund.

A shock from the clock high voltage circuit is at least a nasty bite. At worst it can kill you.

We decline any responsibility in the case of injury or death. Only qualified technicians and people with an understanding of the risks involved with this unit should proceed.

Note also that you are responsible for the safety after construction. The unit should not be left in the reach of children, animals or other persons who are not suitably aware of the danger posed by this unit.

REPEAT: If you are not sure, please do not build or use the clock. We will refund you your payment, no questions asked.

External power supply

The clock requires an external power supply of between 9V and 12V DC, with a “centre positive“ 2.1mm x 5.5mm barrel jack, able to supply 1A.

It is not advised to use more than 12V. The absolute maximum permissible is 16V DC. Higher voltages than this will surely damage the clock.

Board Versions

This manual covers versions

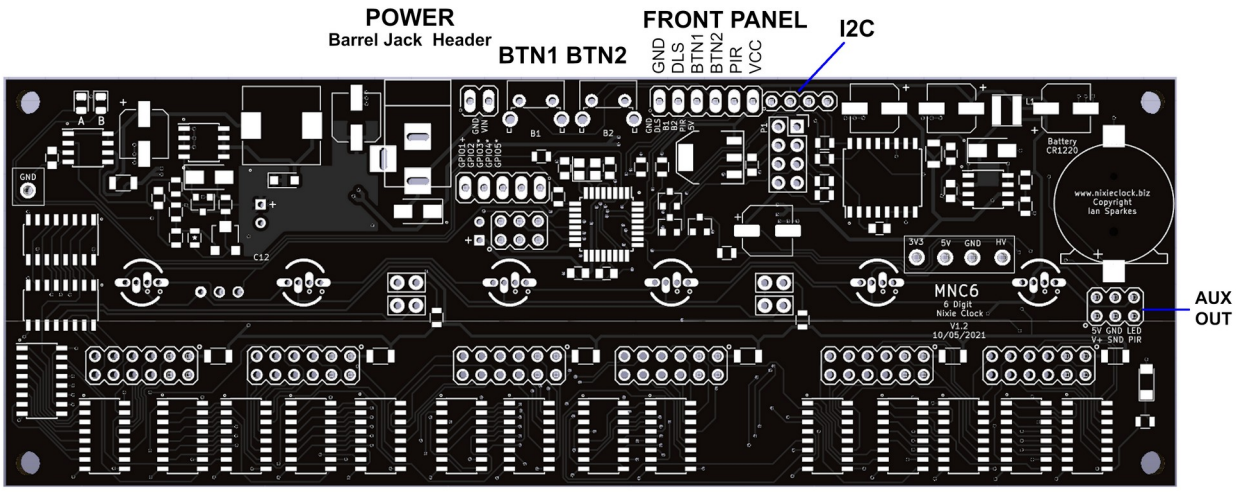
- MNC6 V1.0
- MNC6 V1.2

While there might be detail differences in the layout and silk screen of the board, you can follow the same instructions. The differences are only minor and do not change the method of construction.

Some V1.2 boards will come with the barrel jack and WiFi header already attached, this is to allow us to fully test the boards before delivery.

Board layout

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



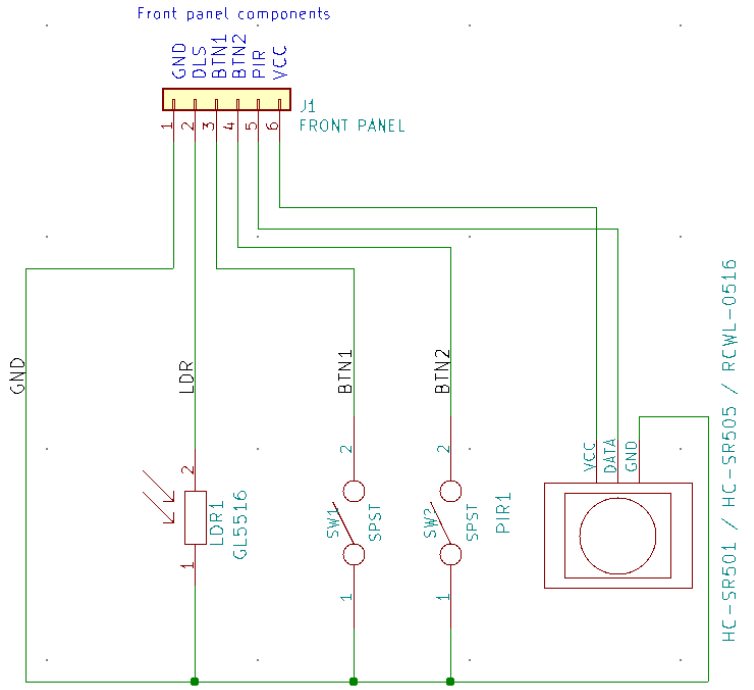
The connections are:

Connector	Description
POWER (Barrel Jack)	External power can be applied to the board with this connector using the 5.5mm jack. The absolute maximum input voltage is 16V. Any higher voltage than this will damage the board within a few seconds! The input current ranges from 300mA to 500mA depending on the tubes and the number of LEDs patterns you are using.
POWER (Header)	You can supply input power using the header instead of the barrel jack. **GND:** The negative side of the input supply **VIN:** The positive side of the input supply
BTN1	Control Button 1 – Please see the user manual for full details of what the switch does.
BTN2	Control Button 2 – Please see the user manual for full details of what the switch does.
FRONT PANEL	These are the controls that go on the front panel: The input button and the Light Dependent Resistor to detect ambient light. **GND:** The “ground”. One lead of the button and one lead of the LDR are connected to this. **DLS:** “Dimming LDR Sense”: The other lead of the LDR is connected to this **BTN1:** The other lead of the button is connected to this input **BTN2:** The other lead of the button is connected to this input **PIR:** The PIR sense lead can be connected here if you want to use the PIR sensor. You need to use a standard 5V PIR module. A 3.3V module will not work reliably. **5V:** Regulated 5V output to drive auxiliary circuitry. This output can only provide 100mA, so please do not try to run extensive circuitry from it.
I2C	Optional connection for I2C. If you want to modify the software to use external sensors or modules, this header provides a standard 5V I2C interface. module.

AUX OUT	“Auxiliary output”. The clock is capable of driving additional APA106 elements in addition to the 6 on-board individually addressable tube back lights, as well as an external sounder if you want to use an alarm. Note that the “LED Out2 and “SND” functions require a code change to implement them, the standard software does not support them. **GND:** The “ground”. One lead of the button and one lead of the LDR are connected to this. **LED:** If you want to drive additional APA106 LEDs, you can daisy chain them on this output. **SND:** This is an open collector output to drive a Piezo sounder. **V+:** Auxiliary output supplying the input voltage, used for driving the Piezo sounder **PIR:** An additional input for the PIR. **5V:** Regulated 5V output to drive auxiliary circuitry. This output can only provide 100mA, so please do not try to run extensive circuitry from it.

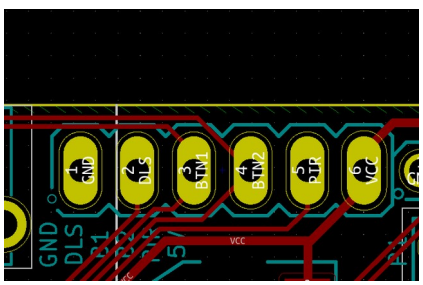
Connecting the external components

To connect the external components (Light dependent resistor, motion sensor), please refer to the image below:



The buttons BTN1 and BTN2 are in parallel with the on-board buttons. You only need to connect them if you want to have alternative buttons on the outside of your case.

Here is a detail of the edge connector:



Construction

Preparation

You should have a small tipped soldering iron, some thin (<= 1mm, ideally 0.6mm – 0.8mm) solder, and electronic side cutters.

There are no special or difficult to solder components needing assembly.

You should leave an hour for construction and testing, but usually you will be ready long before the hour is finished.

Input Power

Parts List:

CON1	Barrel Jack

The Low Voltage circuit is a buck switched mode voltage regulator. Its job is to reduce the external voltage from the power adapter down to a known and stable 5V to drive the micro-controller and the LEDs.

You can also supply power using the 2.54mm header if you prefer.

Put the connector on the board in the marked locations.

Test Step

5V power supply

Hook up the power, and check the 5V line.

Check that the voltage between the “GND” test point and the “5V” test point. It should read a voltage of between 4.9V and 5.1V.

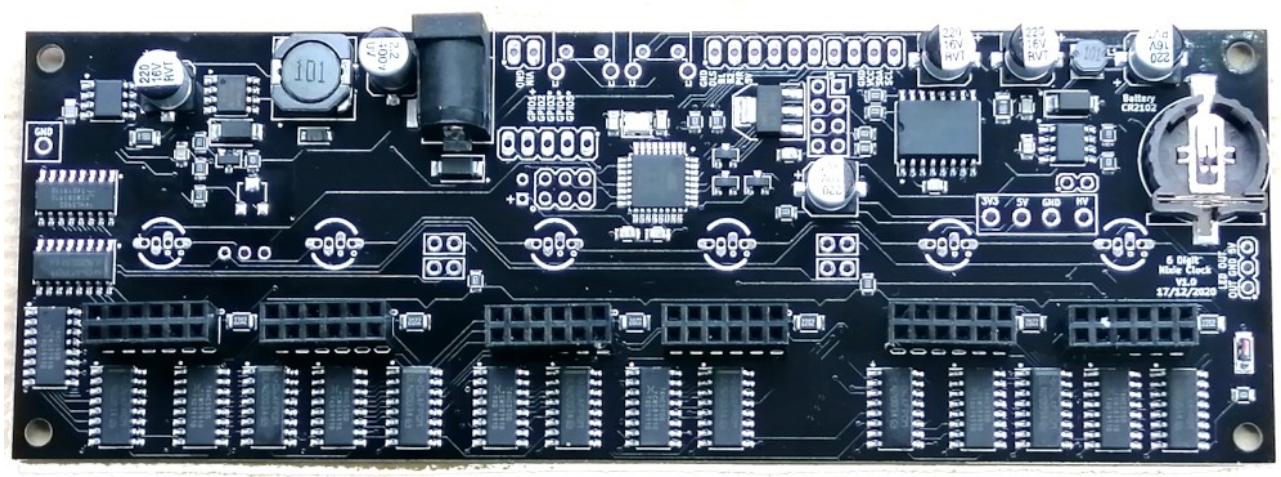
Test Step

170V power supply

Check the voltage at the 170V test point. You should read a voltage of between 165V and 175V.

No components should make a noise or get hot.

At the end of this step, your board should look like this:



Input power

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

YouTube: https://youtu.be/gHwNAqeJmmE

BitChute: https://www.bitchute.com/video/KjI5Li9KGav4/

Tube holder sockets

Parts List:

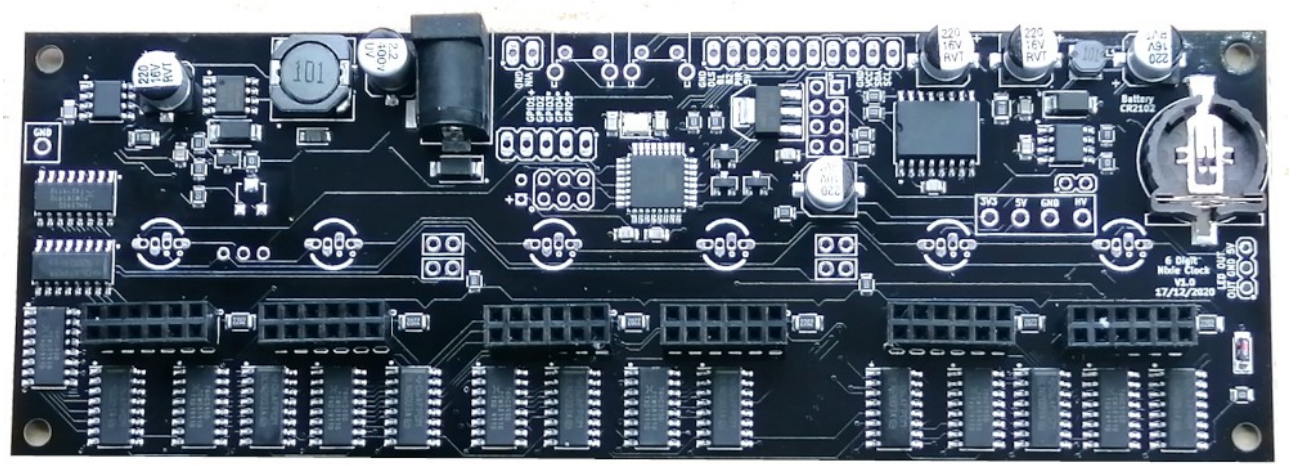
J8 - J13	2 x 6 2.54mm Female headers

We suggest you put the sockets on the board, but you can also reverse the connectors and put the header pins on the board if you prefer. We prefer to put the sockets on the board, because this means that no high voltages are exposed when the tube is unplugged.

If you decide to put the header pins on the board, please invert the components referred to in this section and the section about the tube holders.

Place the 6 sockets in the marked locations and solder them.

At the end of this step, your board should look like this:



Tube Sockets

Warning!

Do not start mounting the LEDs until you have mounted the tube holder boards.

You will need to determine the correct height of the LED using your completed tube holders.

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

YouTube: https://youtu.be/AuyDefStTdY

BitChute: https://www.bitchute.com/video/EGtqhpK7pO22/

Tube holder preparation: Tubes

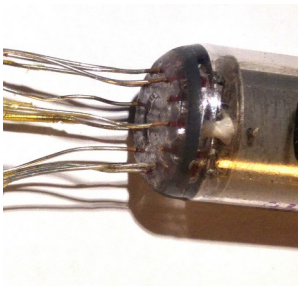
This step can be skipped if you have bought the tubes already assembled onto tube holders! Instead, go to the section “**Backlight LEDs**”!

Parts List:

6 tubes	Mid sized Nixie tubes
6 holders	Tube holders to suit

This step mounts the tubes on the tube holder boards. There are a few tricks we can use. Before you start, the tube will look like this the picture shown on the right. Don’t worry if your leads are longer or shorter than the ones shown in this picture! The beauty of the tube holders is that we are able to use tubes even with very short leads.

Carefully remove the white plastic base from the tube. Pay attention not to strain the leads too much, because the junction between the tube and the wire is one of the weak points of the tube. If you pull too hard you can easily damage the tube. If you have varnish on the tube leads, you can soften this with a hair dryer or heat gun on a low setting.



Often used tubes have the two decimal point leads removed. It is not a problem if the tube has missing decimal point wires. For many applications, these were already removed at the factory, and were never used. It is not a sign that the tubes are “faulty” or “second quality”!

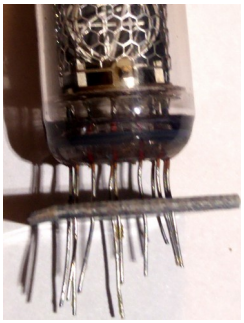
In order to mount the tube on the tube holder PCB, one useful trick is the trim the leads in a spiral, starting from the anode. This will make the leads easier to insert into the holes later. You can thread the leads into the holes one at a time.

The picture on the right shows the leads cleaned, straightened and trimmed ready for insertion into the tube holder.

Now you are ready to mount the mount the tube. The semi-circle marking on the tube holder board shows the front of the tube, closest to the 12 pin connector, and there is guide on the PCB silk screen.

Put the leads one at a time into the holes on the board, making sure to leave the holes either side of the anode empty if you have tubes without decimal points.

Once all the leads are in, push the tube down so that it is about 5mm away from the board. Check that the tube is upright and not obviously tilted to one side, or backwards or forwards. (The picture shows a tube on a bare board, for clarity).



Now solder one pin and check again that the tube is upright compared to the board. If it is correct, then you can start to solder other leads, checking at each lead you solder that the tube is still aligned correctly. Once you have soldered three or four leads, the tube will be rigid.

Repeat the same process for all the other tubes.

If you damage a tube holder board or a tube fails, spares are always available. We can also provide the PCB files if you want to make or design your own!

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

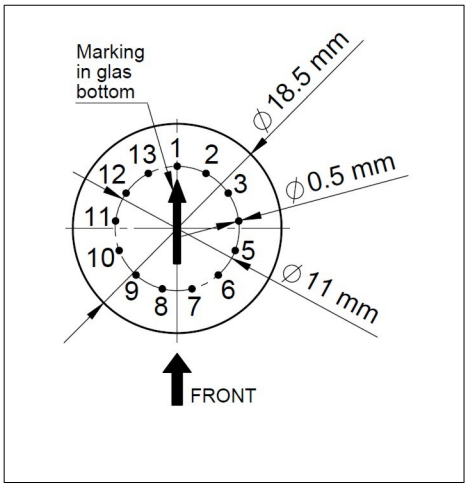
YouTube: https://youtu.be/2RLMN2giohs

BitChute: https://www.bitchute.com/video/CCM2qTKnAODy/

Some Common Tube pin outs:

IN-14. Pins 2 and 13 are decimal points, and are often removed on the tubes. The other pins are given below.

Pin	Connection
1	Anode
2	Right decimal point
3	Digit 1
4	Digit 2
5	Digit 3
6	Digit 4
7	Digit 5
8	Digit 6
9	Digit 7
10	Digit 8
11	Digit 9
12	Digit 0
13	Left decimal point



Notes:

- It is best to carefully spread the legs of the tube out. If you have long enough leads, a trick is to trim the leads to different lengths so they get shorter by 0.5mm as you go round the tube. This means that pin 3 is shorter than pin 1 by 0.5mm. Pin 4 is shorter than pin3 by 0.5mm and so on. Pin12 is shorter than pin 3 by 5mm at the end. This means that you can thread the leads into the holes more easily.
- To make the tube stand upright, solder just the first three leads at the beginning one at a time (e.g. pin 1, pin6 and pin 10). You can then easily align the tube so that it is perfectly upright by reheating only one of the pins. Once the tubes are upright and aligned, you can solder the remaining pins.
- If you don’t manage to mount the tube perfectly upright, small imperfections can be compensated for in the mounting system. Try to avoid forcing the wires on the tubes to bend slightly - tubes are robust, but they are made of glass and must be treated with care.

Tube holder preparation: Header pins

This step can be skipped if you have bought the tubes already assembled onto tube holders! Instead, go to the section “**Backlight LEDs**”!

Parts List:

J8 - J13	2 x 6 2.54mm Male header pins

We suggest you put the sockets on the board, but you can also reverse the connectors and put the header pins on the board if you prefer. We prefer to put the sockets on the board, because this means that no high voltages are exposed when the tube is unplugged.

Put the header pins in the boards, making sure that you follow the silk screen on the board. The pins go on the other side of the board to the tube.

At the end of this step, each tube board should look like this:

Tube Holder

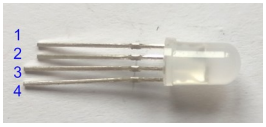
Backlight LEDs

Parts List:

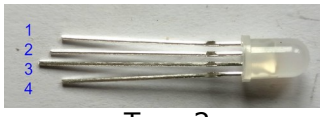
LED1 - 6	APA106

Now that you have your tubes on holders, you can put in the LEDs. The top of the dome on the LED should touch the bottom of the tube, through the hole in the tube holder. This provides support to the tube to stop it tipping backwards.

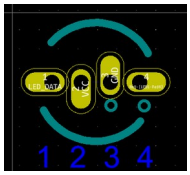
There are two versions of the APA106:



Type 1



Type 2

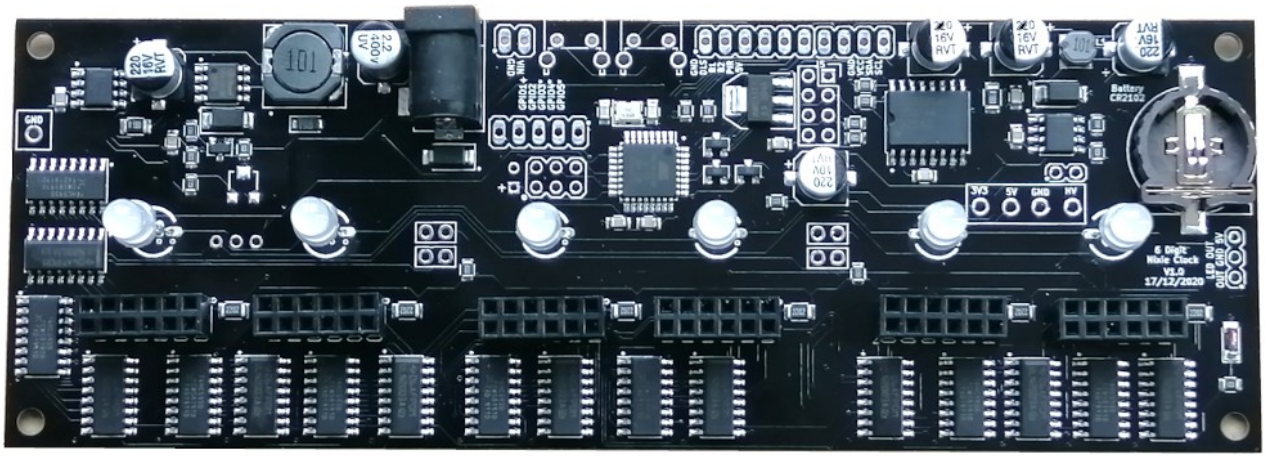


Type 1 APA106 LEDs have two leads longer that the other two. The longer leads go into the two holes with tiny round “o” markings next to them (pins 3 and 4).

Type 2 APA106 LEDs have a package where one lead is longer than the others. This is pin 3 and goes into a hole with an “o” on it (pin 3).

The APA106 LEDs are in a “daisy-chain”. They are driven from left to right. Start with the left most LED and work to the right. You can check that the test sequence is produced correctly after each LED is soldered.

At the end of this step, your board should look like this:



Backlight LEDs

Separator neons

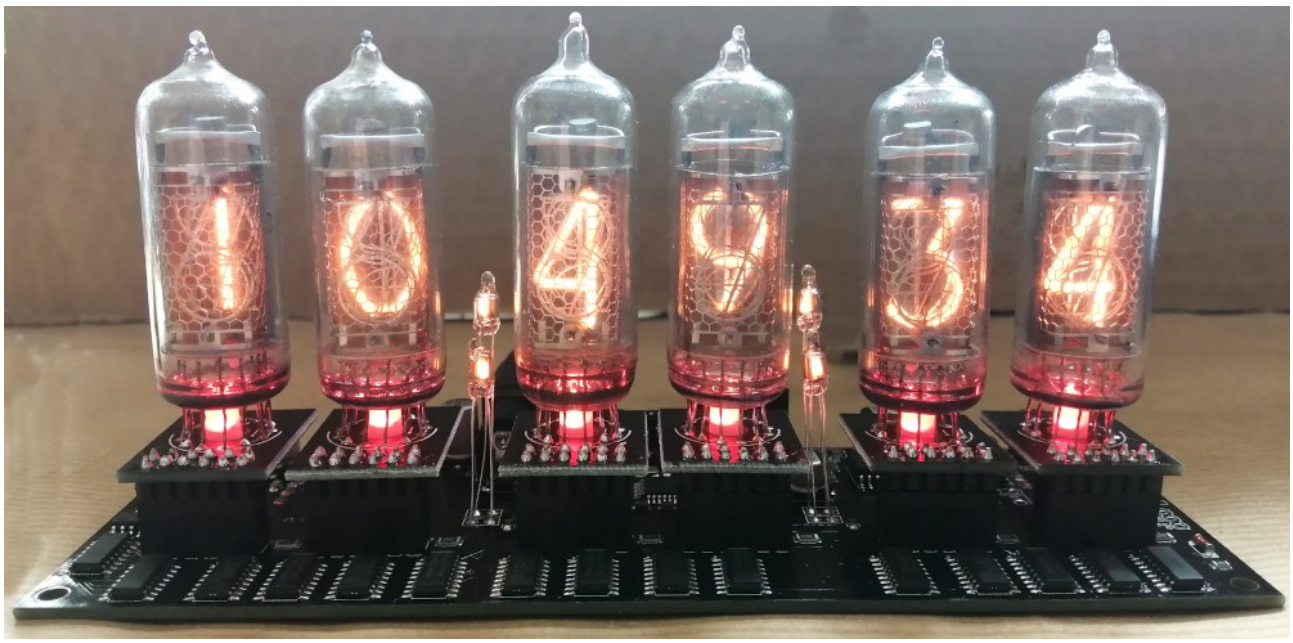
Parts List:

NE1 - 4	4mm neons

Put the 4 neons in the spaces on the board. Each pair is in series, and so will not work until both are in place. For best results, mount one higher than the other.

Put the tubes in so you can tell how the end result will look.

At the end of this step, your board should look like this:



Separators

Front Panel components

When all the components are installed, you are finished with the board.

SV2	6 pin header
BTN1 - 2	Momentary push button
LDR	LDR
PIR	PIR detector (optional)

The LDR should be mounted in such a way that the flat face of the LDR is exposed to the ambient light. This will allow it to detect the ambient light and adjust the brightness for it.

Optional “WiFi”

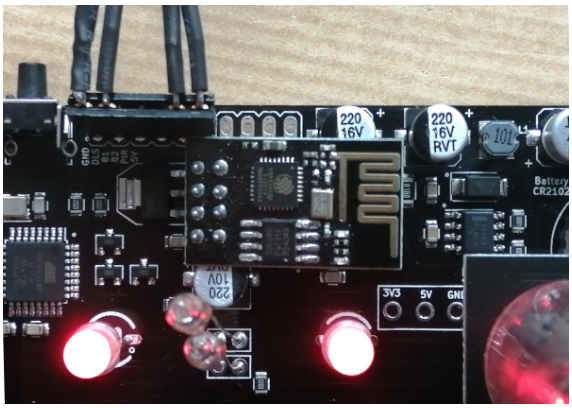
Parts List:

ESP1	ESP-01 module
P2	2x4 Female Header Socket

The board comes with a high precision, battery backed DS3231 RTC module. If you want to take advantage of the features available with the WiFi module, you can add this at an time if you did not order it with the original module.

Notes:

- The ESP8266 ESP-01 module needs to be installed with the long side over the DS3231 chip. If you install it the wrong way round, it will be destroyed. Please refer to the picture!



The ESP-01 module goes over the DS3231

Warning!

Double check the orientation/location of the WiFi module before you proceed!

If you have installed the WiFi module, make sure that it is over the DS3231! If you install it the wrong way round, you will destroy it.

The WiFi module has it's own instruction manual. Please refer to that if you have the WiFi option.

Troubleshooting

If not everything goes as you expect, please refer to the test steps during the construction and the associated troubleshooting tips. If that does not cover the problem you have, please see below. If you still can't find the answer, contact us!


Trouble shooting

The tubes flash (or blink) on and off.

This could be a symptom that the external power supply can't deliver the power needed to drive the circuit.

On start up, the High Voltage generator needs to draw significantly more power than when it is running normally, and in some cases this might overload the external power supply.

Try a different external power supply and see if the problem persists.


Trouble shooting

The tube display brightness is not constant, and appears to “pulse” rapidly.

This is a symptom that the High Voltage generator or the external power supply is overloaded.


Trouble shooting

The display is too dim.

Check if the auto-dimming is working. If the display does not change in low or high ambient light, your LDR does not appear to be working. Check the connections to the LDR.

If the LDR is correct, perform a factory reset to make sure that no strange values have been left in the EEPROM.

Check the LDR reading by pressing the button three times in quick succession when the clock is on. You should see a value between “01 00 00” and “09 99 00”. Changing the light conditions should change this value. It is normal that the value is not stable when it is in the middle of the range. We read the LDR many times a second, and it is unusual that two readings are identical.


Trouble shooting

I can see some “ghosting”.

“Ghosting” is where you can see a very faint image of another number at the same time as the one that should be shown. Some tubes are more sensitive than others, and depending on the construction and components, it might show up more.

If you see ghosting, increase the “anti-ghosting” setting, but only to the point where the ghosting is no longer visible or irritating.

The “anti-ghosting” setting decreases the overall brightness of the display slightly, and not all tubes (even of the same sort) need it, so anti-ghosting should only be used when there is a real need to use it.

Programming the micro-controller

The board comes per-programmed. You don't **need** to program it, but you might want to.

You can update the micro-controller with a newer version of the software, or even create your own software, and load it onto the chip. We have gone to a lot of trouble to make this as easy as possible.

Programming using the ICSP header

The easiest way to program the board is to use an USBASP programmer. These are easily and cheaply available from many sources. You can solder a 2x3 header into the space provided and then plug the programmer in. You will need a 10-pin to 6-pin adapter. These are usually bundled with the programmer.

If you want to get one from us, you can find this in the store here (bottom of the page):

> https://www.nixieclock.biz/StoreMNC6.html

Be sure to get the programmer the right way round! There is a small “o” on the board denoting pin 1.

The firmware is available on Git, and you can find the latest version on the manuals page, where you got this manual.

That's it!

Revisions:

V0001: 23Jan2021: Initial version
V0002: 4Aug2021: Add store links for programmer, add PIR info