"SNC46"

Arduino Interchangeable Nixie Clock Rev1

Construction Manual

IN-12

Index

Table of Contents

What this document is about

This document is the construction manual for the "SNC6" Arduino based Small-format 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 a high quality, fully featured 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.
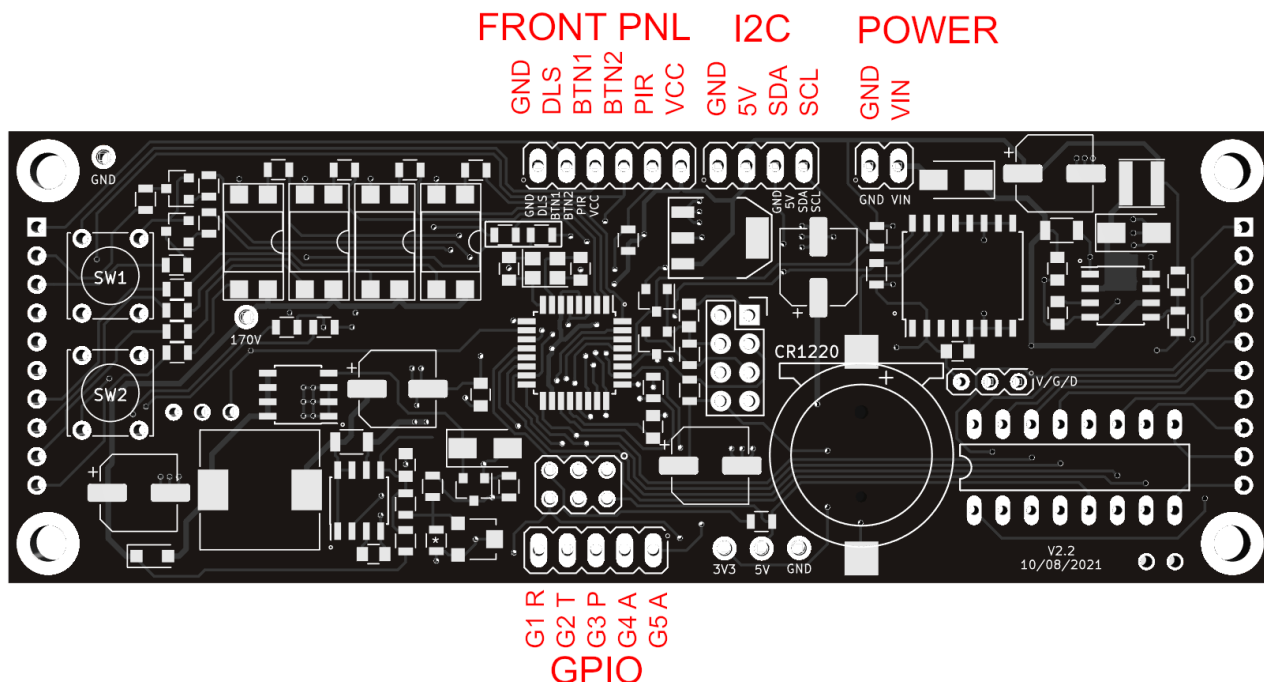
External power supply

The clock requires an external power supply of between 9V and 12V DC, 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 layout

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

The connections are:

Connector	Description
POWER (Header)	External power can be applied to the board with this connector using the two-pin header. Recommended voltage: 9V – 12V DC. 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 LED patterns you are using. **GND:** The negative side of the input supply **VIN:** The positive side of the input supply
FRONT PNL	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. **VCC:** 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.
GPIO	These are GPIOs that are for future use or for specialised versions of the board. If you have a programmer, we can supply the source code of the firmware, and you can adapt the firmware to your needs. **G1 R**: General purpose GPIO, or Serial RX **G2 T**: General purpose GPIO, or Serial TX **G3 P**: PWM capable GPIO **G4 A**: Analog capable GPIO **G5 A**: Analog capable GPIO

Before you start the construction, the main board will look like this:

Bare main board

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:

HDR 1x2	1x2 2.54mm header

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.

Put the connector on the board in the marked locations.

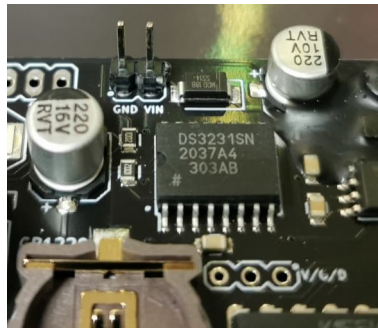|	
Test Step	**5V power supply** Hook up the power, and check the 5V line. Check that the voltage between the "GND" test point and the "5V" test point. It should read a voltage of between 4.9V and 5.1V.
Test Step	**170V power supply** Check the voltage at the 170V test point. You should read a voltage of between 165V and 175V. No components should make a noise or get hot.

At the end of this step, you should have added the power header like this:
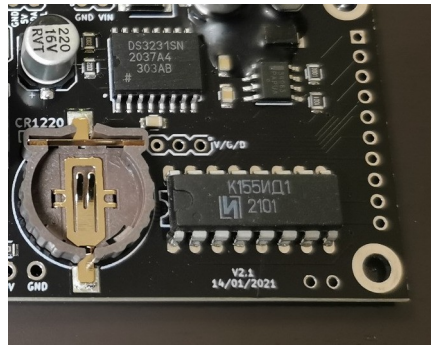

Input power header

K155ID1 display driver

Parts List:

K155ID1	K155ID1 Display driver IC
DIP-16	16 Pin DIL socket

You can fit the 16 pin display driver IC with or without the socket. The chip is robust and does not need any special precautions before soldering.

If you fit the socket, please make sure to line up the notch on the socket with the notch on the PCB. If you solder the chip directly, make sure that the notch lines up.

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

The K155 – without the optional 16-pin socket

Display board connectors

Parts List:

HDR 1x10	Header 1x10 Male
HDR 1x10	Header 1x10 Male
HDR 1x10	Header 1x10 Female
HDR 1x10	Header 1x10 Female

Warning!

Before mounting the 10 pin connectors, please check carefully the markings on the PCBs.
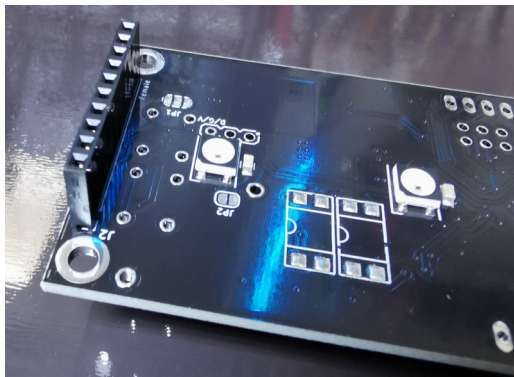
There is nothing more frustrating than finding you have mounted the connectors incorrectly and your display board is upside down or back to front!

Each PCB has the connector type ("Male" or "Female") and a silk screen box on the side you should mount the connector.
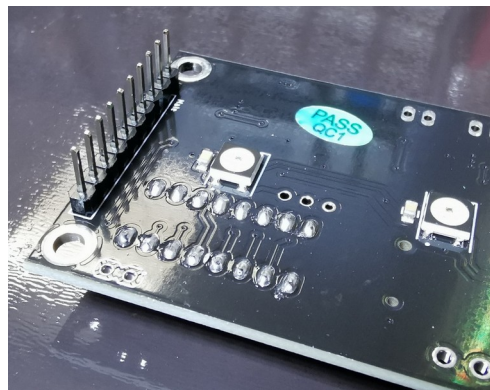
Test fit the display board before soldering to make sure that it is the right way up, with one "Male" and one "Female" connector on each PCB!

Fit the connectors to the PCBs, making sure to carefully check the silk screen for the side of the board to mount the connectors on, and the type of connector ("Male" or "Female"). At the end you should have a "sandwich" of the two boards, with the Neopixels behind the holes in the display board.

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

Female display board connector

Male display board connector

Tube pin receptacles

Parts List:

44	1mm pin receptacles

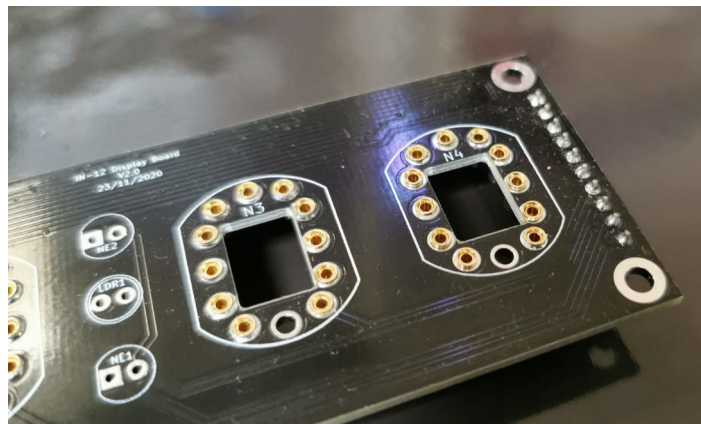|	**Do not try to mount the receptacles by using the tube pins as a guide!**
⚠️ Warning!	If you solder the receptacles while they are on the tube pins, they will likely fuse the pin to the receptacle, and that's not the idea at all. Tube pins are tinned and will easily fuse to the receptacle with even moderate heat. You will not be able to separate them any more.

Place the receptacles into the holes on the board, and tape them in place with paper (not plastic) tape. If you don't have any paper tape, you can use plastic tape, but protect it with a small rectangle of cardboard.

Solder the receptacles from behind, sparingly applying solder.

If you are a skilled solderer, you can also solder from the front, but the risk here is that you flow solder into the receptacle cup, which disables the internal spring element.

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

Display board receptacles

Mounting the tubes into the receptacles

Parts List:

N1-N4	Nixie tubes

Press fit the tubes into the receptacles. A moderate force is needed, and you may have to straighten some of the pins on the tubes. Don't be too brutal.

It is not expected that you routinely have to remove the tubes, but if you do, you might have to "wiggle" the tubes in the receptacles to remove them.

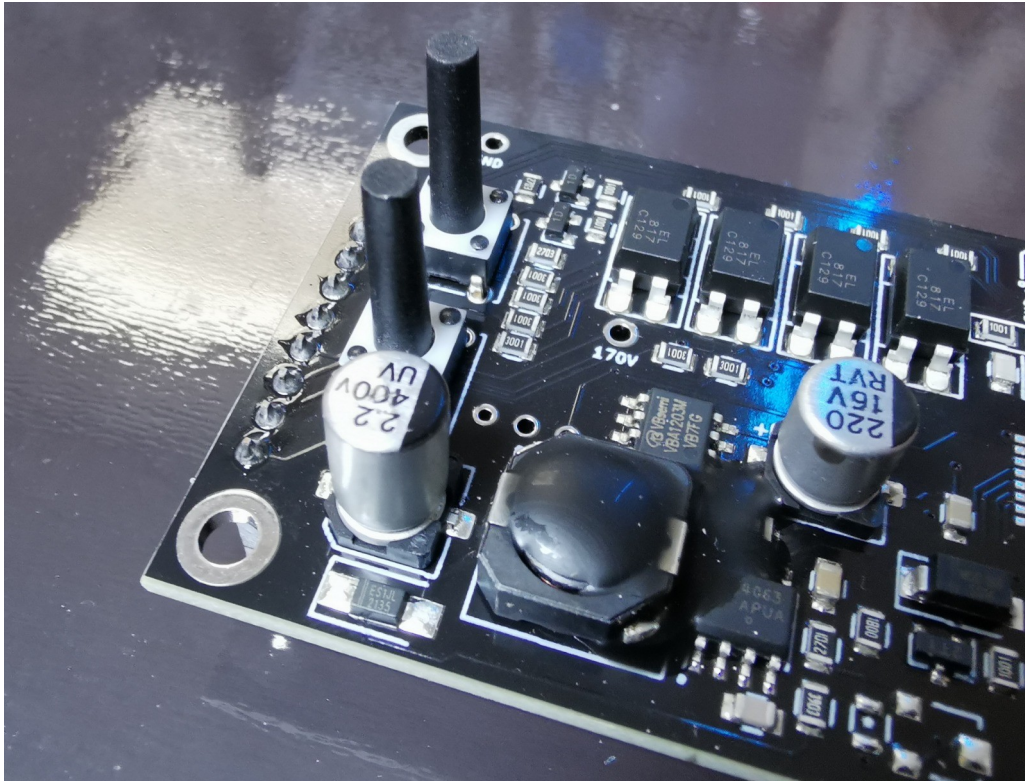*Display board with tubes in the sockets*

Switches

Parts List:

SW1, SW2	Tactile switch

Place the two switches on the PCB in the locations indicated by the silk screen.

Depending on the case you have, you might want to get switches with longer stems. If you need these, just tell us and we will put them in the package for you.

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

Switches – note that these are long stem switches

Separator neons and LDR

Parts List:

NE1, NE2	4mm neons
LDR1	GL5516 LDR

Put the 2 neons in the spaces on the display board.

Mount the LDR between them. The orientation is not important.

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

Separators and LDR

Front Panel components

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

SV2	6 pin header
PIR	PIR detector (optional)

Mount the 6-pin header. If you want to fit a motion detector module, follow the silk screen pin outs.

Additional locations are provided if you want to have switches or the LDR in an alternative location on your case.
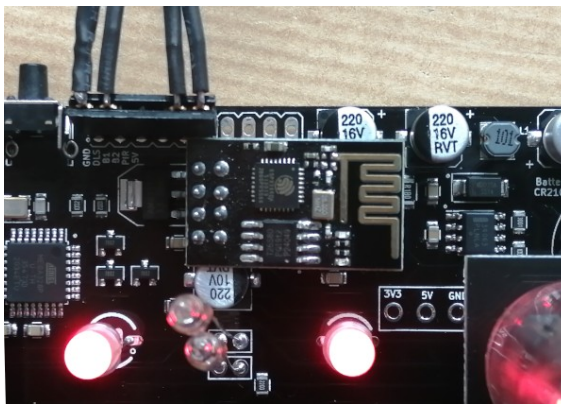
Optional "WiFi"

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

<u>The tubes flash (or blink) on and off.</u>

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

<u>The tube display brightness is not constant, and appears to "pulse" rapidly.</u>

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

Trouble shooting

<u>The display is too dim.</u>

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.

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.

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: 07Mar2021: First version
V0002: 04Jan2022: Insert pictures