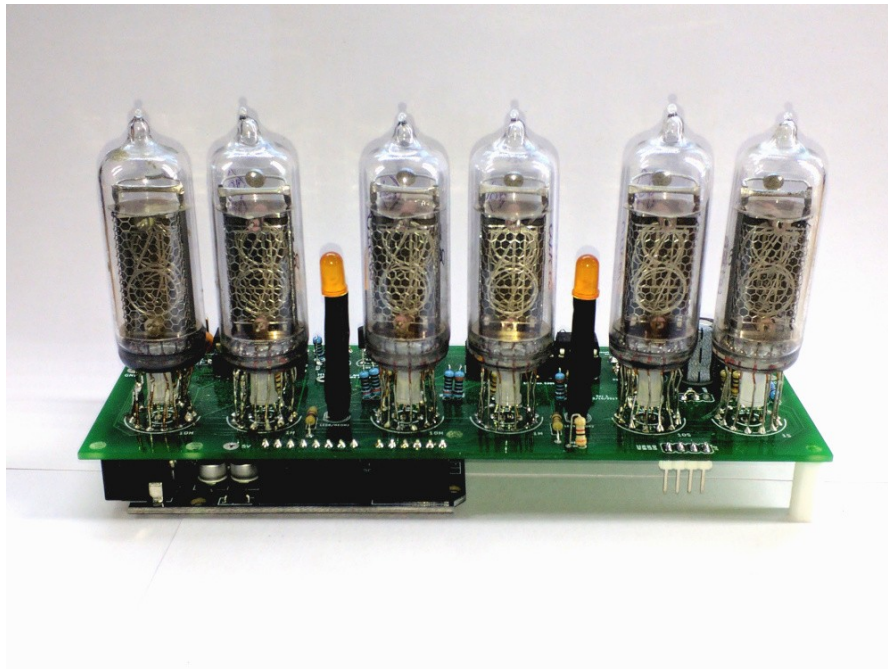


Arduino Nixie Clock

“IN-14 All-In-One” Clock Rev3

Arduino Shield

Construction Manual



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.open-rate.com/Store.html>

Software

The software is available on GitHub at the address:

<https://github.com/isparkes/ArdunixNix6/releases>

This board works with Release “[Revision 4 boards](#)” under the “Releases” tab.

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.

Safety

The voltages produced in the High Voltage circuit can reach peaks of 400V! 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.

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

Powering Up

When you power the unit up for the first time, it will go into the startup test routine. This will set the High Voltage Generator to run with some default settings which are useful for the construction of the clock. For a full description of the startup sequence, please see the User Manual.

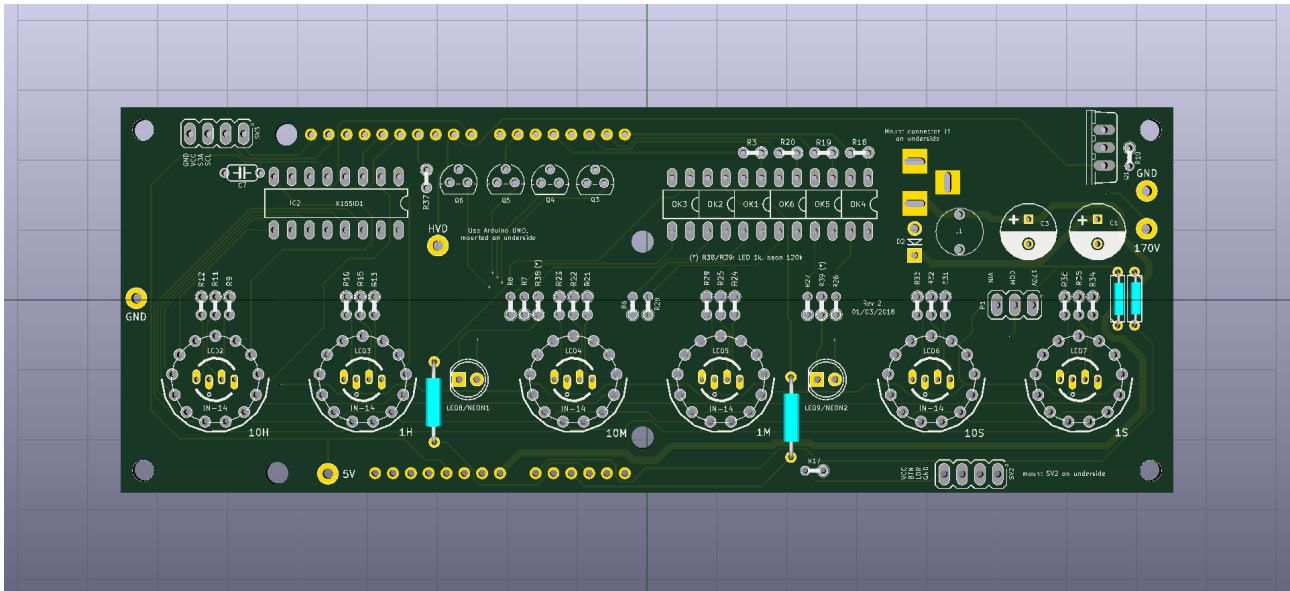
Component Identification

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

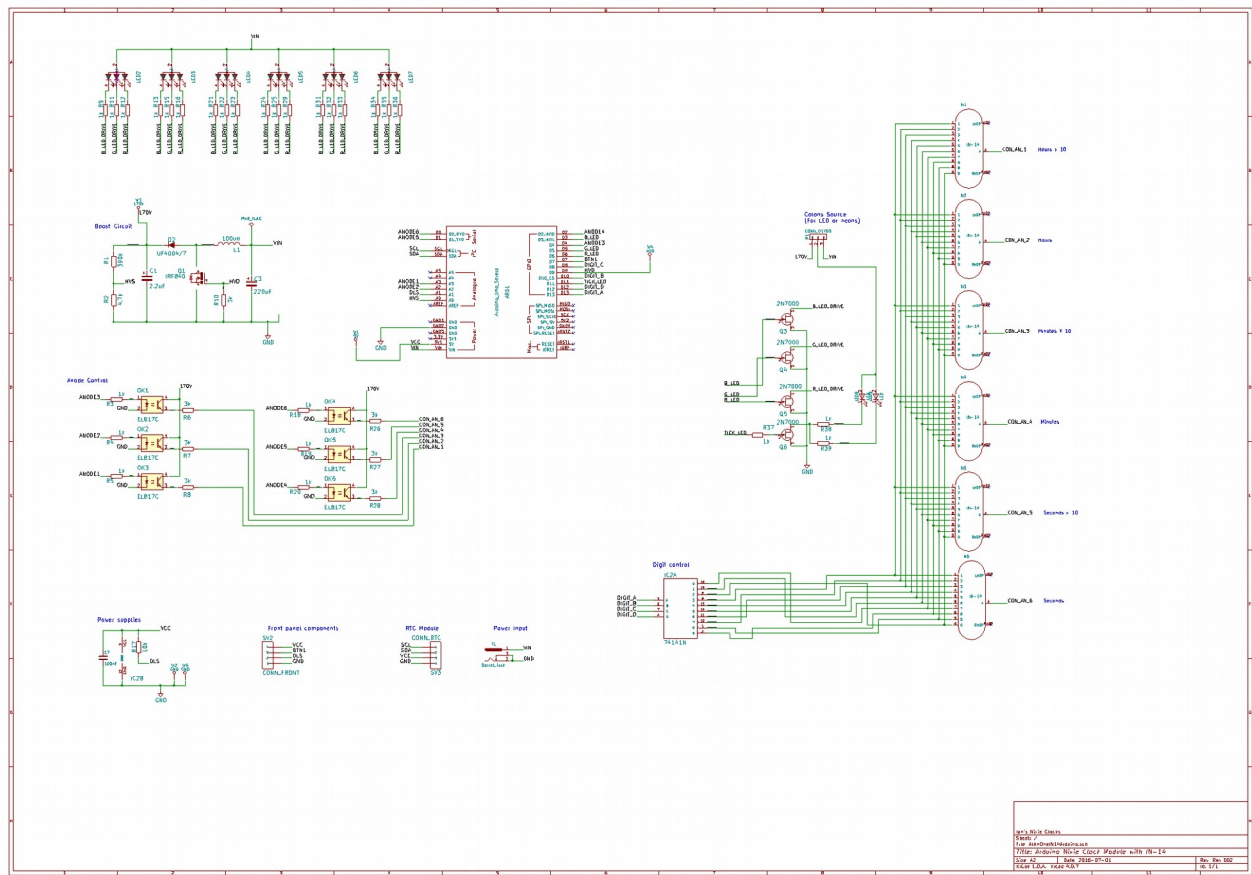


The connections are:

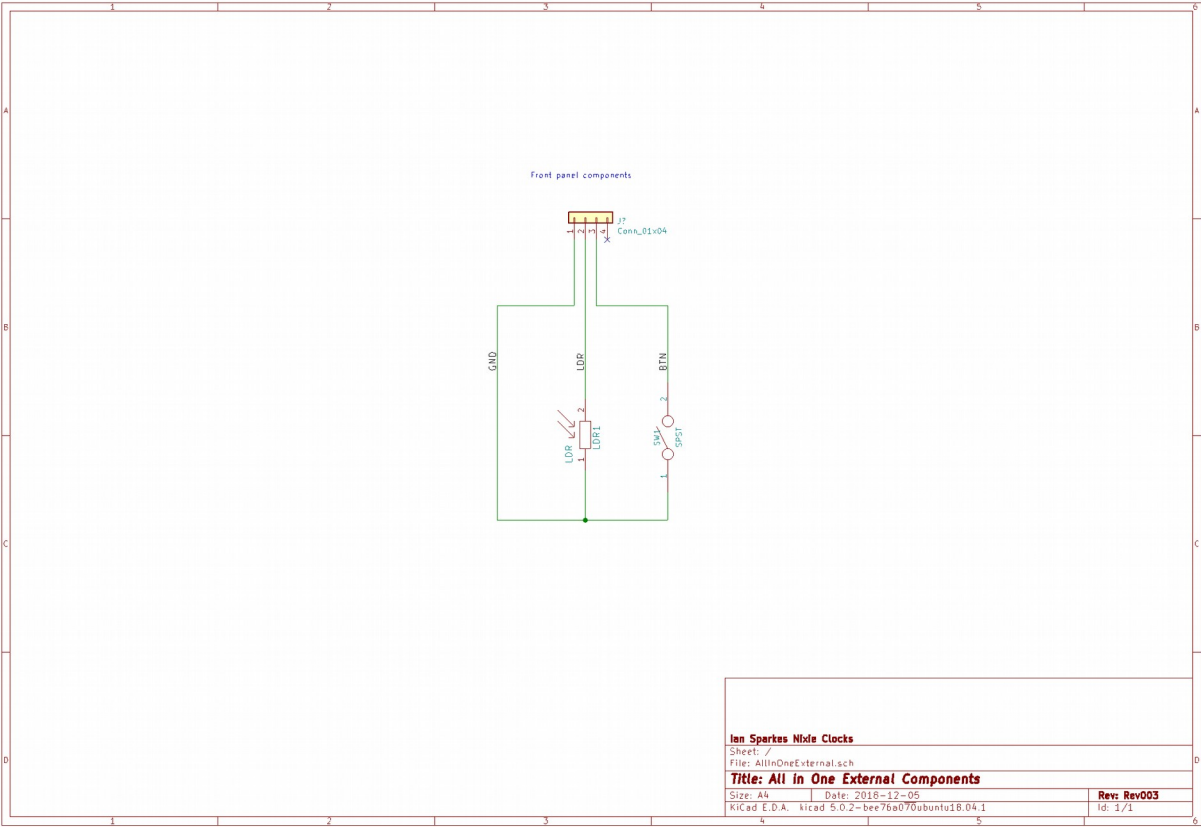
Connector	Description
POWER	<p>External power should be applied to the board with this connector. Any Arduino compatible DC input source is possible. Higher voltages may be possible, but could cause the digits to flicker if the voltage is too high, and you might have to provide a heat sink for the MOSFET.</p> <p>The absolute maximum input voltage is 16V. Any higher voltage than this will damage the board within a few seconds!</p> <p>The barrel jack connector facing towards the back of the board is a duplicate of the connector on the Arduino board. You can use either.</p> <p>The input current ranges from 300mA to 1A depending on the brightness of the tubes and the LEDs.</p>
FRONT	<p>These are the controls that go on the front panel: The input button and the Light Dependent Resistor to detect ambient light.</p> <p>GND: The “ground”. One lead of the button and one lead of the LDR and one lead of the button are connected to this.</p> <p>BTN1: The other lead of the button is connected to this input</p> <p>DLS: “Dimming LDR Sense”: The other lead of the LDR is connected to this</p> <p>VCC: Regulated 5V output to drive any LEDs or lighting. Note that you can also connect the LEDs to the VIN if you want to reduce the load of the regulator.</p>
RTC / WIFI	<p>The connection for the RTC (Real Time Clock) or WiFi time provider module. Connect this to the appropriately marked terminals on the RTC / WIFI module.</p>

Schematic

Below is the schematic for the clock.



And for the LDR and button:



Construction

Preparation:

You should have a small tipped soldering iron, some thin ($< 1\text{mm}$, ideally 0.6mm) solder, and electronic side cutters.

Video Tutorials:

You can follow step by step video tutorials at the YouTube playlist here:

<https://www.youtube.com/watch?v=qJqELavPNHg&list=PLXsrUkF2v8OIJMoKY0pPJ97rVM2QEEaWL>

Components:

You should find the following contents as listed in the BOM (Bill of Materials). It is best to check the contents before you start.

Please see the “Component Identification Manual” to help you identify individual components.

Low Voltage Circuit

Parts list:

J1	Barrel Jack 5.5mm x 2.1 mm
HDR1	6 X 2.54mm header pins
HDR2	8 X 2.54mm header pins
HDR3	8 X 2.54mm header pins
HDR5	10 X 2.54mm header pins
C3	220uF

In this clock, the low voltage circuitry is provided by the Arduino board, using the on board voltage regulator. Because the side mounting power connector is not always convenient, we have provided a back mounting connector, which can be used as an alternative to the side mounting jack. You can use either, they are equivalent. There is an additional smoothing capacitor on the board to aid the Arduino’s power smoothing.

Video Tutorial: <https://youtu.be/qJqELavPNHg>

Notes:

- See the section on “Component Identification” for help with identifying the components.
- You mount the barrel jack J1 on the underside of the board.
- To make the header pins fit well, put them in the Arduino first, so that they are held in place while you solder them. It is hard to align them if you do not use the Arduino as a template!
- You can mount the capacitor C3 on the top or on the underside of the board. This component has a polarity, and you must make sure to fit it the right way.



Note!

The component numbering is not complete. For example there is no “C2”. This is not a mistake.

This board is a version of our popular “All-In-One” board, and the Arduino brings lots of components and supporting circuitry with it. Where the Arduino has the circuitry on board, we have omitted the component from the numbering.



Test Step

Once the barrel jack and the headers are on the board, put the board on the Arduino and hook up the power to the barrel jack on the PCB.

Check that the power LED on the Arduino comes on.

Check also that the voltage is 5V between the “GND” test point and the “5V” test point.

If the LED comes on, check for a few seconds that the 7805 does not heat up. It should stay almost cold.

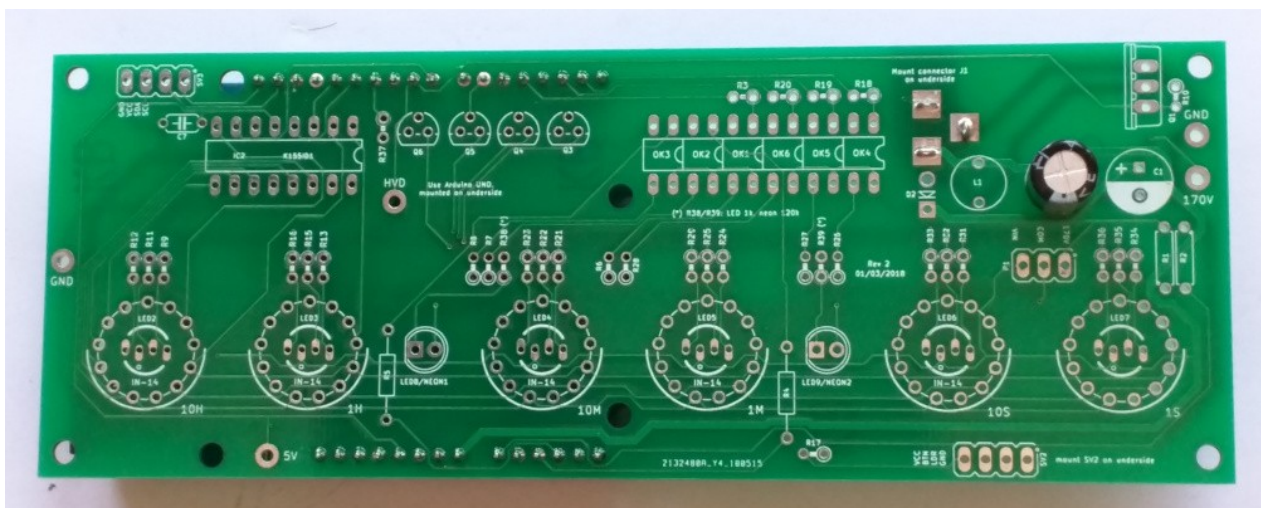
Hint: The 220 uF capacitor

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!

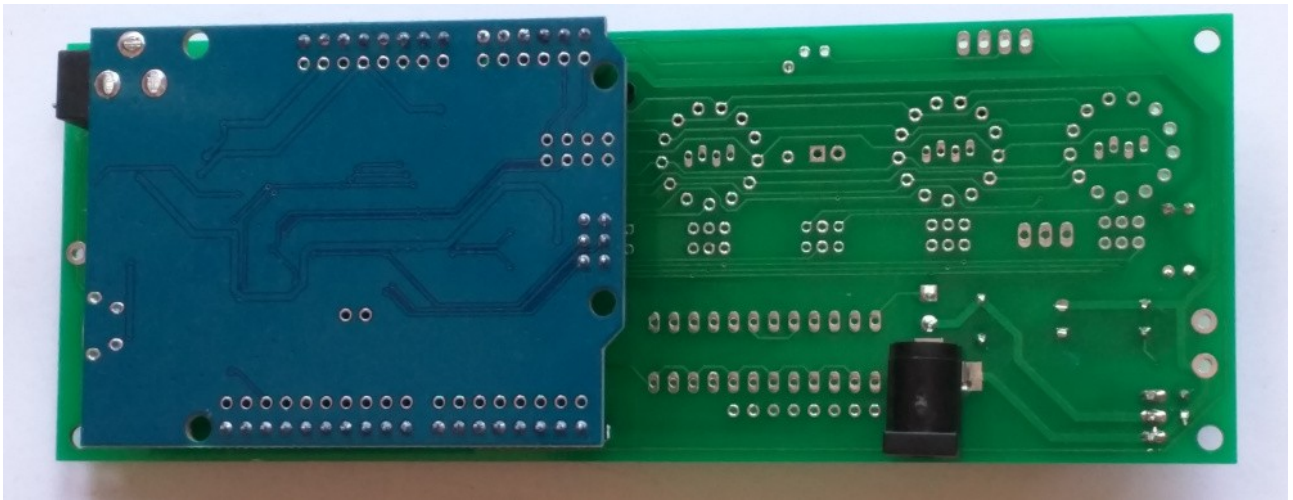


220uF capacitor “stripe”

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



Low Voltage Circuit Top Side



Low Voltage Circuit Bottom Side

High Voltage Circuit:

Parts List:

C1	2.2uF 400V
C7	100nF
D2	UF4007
L1	100uH
R10	3k
Q1	IRF840
R1	390k
R2	4.7k

The high voltage circuit uses the Arduino to drive the boost circuit with a high frequency square wave, and has a feedback loop in which the Arduino reads the voltage produced via an analogue input, and regulates the brightness of the tubes so that there is no flickering or unwanted dimming.

Video Tutorial: <https://youtu.be/-LSfZyyqdUM>

Notes:

- See the section on “Component Identification” for help with identifying the components.
- C1 must go the right way round. The negative side is marked with a stripe (see hint). It can be mounted on the top of the board or the bottom. It is important to get the polarity of the component right.
- Q1 can be mounted on the top of the board or the bottom of the board. If you mount Q1 on the top of the board, mount it so that the metal portion lines up with the white stripe on the board. If you mount it on the bottom, please make sure that the orientation is still correct.
- D2 should be placed so that the white stripe on the body lines up with the square hole on the board.



Warning!

Double check the orientation/location of the components before you proceed!

Especially check that R1 and R2 are in the right positions, and that the electrolytic capacitor C1 is the right way round. If you switch R1 and R2, you will put high voltage into the Arduino, and this will destroy it.



Note!

In order to test the High Voltage generator, you need to load the software onto the Arduino.

The Arduino drives the HV generator, and unless the code has been loaded onto the Arduino, you will not be able to test this step. Please see the section on “Programming the Arduino” near the end of this manual!

Once all the components are on the board, hook up the power. Give your work a careful check

to make sure that the orientation of the components is right. Especially check that the stripe on C1 is facing the marking on the board.



Test Step

Apply power to the board again. Listen for any stressed sounding buzzing or humming, and check that neither the Arduino nor the MOSFET get excessively hot.

Check that the power LED still lights.



Warning!

Be careful, we are dealing with high voltages now!

The voltage may be significantly higher than 170V at the moment, because the high voltage generator is powerful and the output is not loaded. Once you add a load, (by connecting the tubes), the voltage should oscillate around 170V – 190V, and might have a slight “sawtooth” appearance if you view it with an oscilloscope.



Trouble shooting

If you hear any angry sounding buzzing turn the power off immediately and check the orientation of C1! The circuit should run almost silently, with only a very faint “crackling” sound.

If you can't reach the target voltage, turn off and check the polarity of your components, especially C1. If you have an oscilloscope, you can check the voltage at the gate of the MOSFET, and it should show pulses of high frequency square wave: this is the driver waveform to the HV generator, which is being turned off and on by the voltage detection, trying to achieve the target voltage (180V default).



Test Step

Check the voltage at the 170V test point. You should read a voltage in excess of 170V.

You can also test using an old neon lamp if you have one. Temporarily connect the neon lamp between the “GND” test point and the “170V” test point with an appropriate ballast resistor (turn the power off first). Turn the power on and the neon lamp should come on.



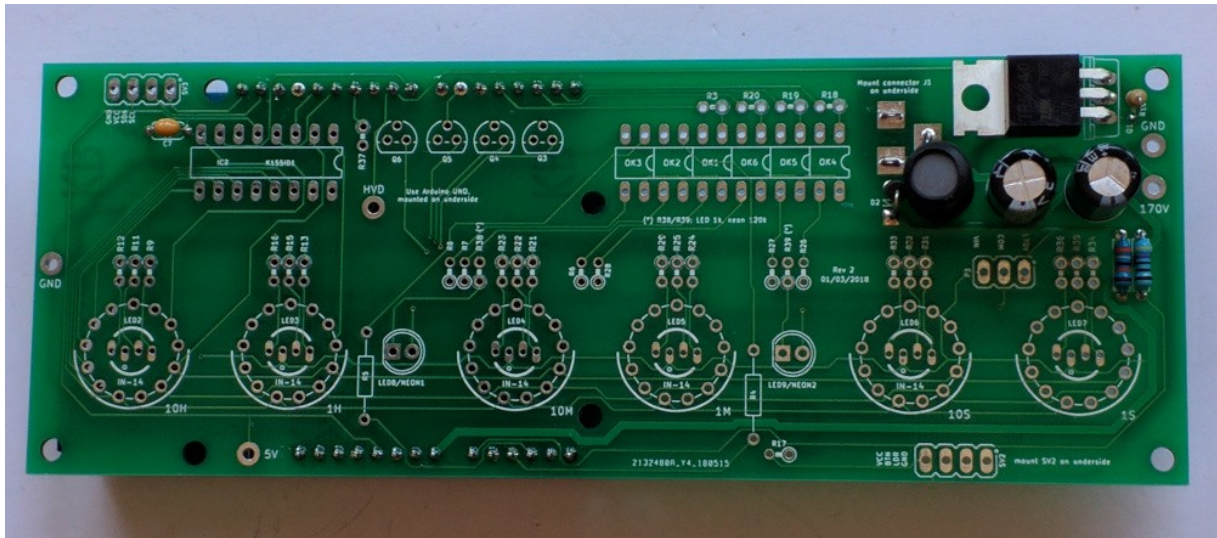
Trouble shooting

Q1 can get warm, **but should not get too hot to touch**. If it gets hot, you need to check the orientation of the components and that there are no solder bridges.

If you don't get the expected voltage reading:

- Check your soldering that there are no bridges or dry joints.
- Check that the external power supply is able to supply the power needed to achieve the high voltage: check that the VIN voltage is stable and not fluctuating.
- Temporarily connect the LDR and re-test.
- Temporarily connect the button and do a factory reset

At the end of the high voltage circuit build, your board should look like this:



High Voltage Circuit

Back light LEDs

Parts List:

Q3,Q4,Q5	2N7000
R9,R11,R12	1k
LED2	RGB Common Anode
R13,R15,R16	1k
LED3	RGB Common Anode
R21,R22,R23	1k
LED4	RGB Common Anode
R24,R25,R29	1k
LED5	RGB Common Anode
R31,R32,R33	1k
LED6	RGB Common Anode
R34,R35,R36	1k
LED7	RGB Common Anode

This step installs the RGB back light LEDs. We do this now, because the RGB LEDs are mounted under the tubes.

You should make a short test after you install each RGB LED. The startup pattern should slowly turn on and then off the colons, the Red channel of the RGB LEDs, then the Green channel and then the Blue channel. There is a gap in the sequence for the separators, which we will install later.

Video tutorial: <https://youtu.be/1Ilzu659Woc>



Warning!

The startup procedure will run over and over again until you tell the clock to move on.

You can tell the clock to move into normal mode by pressing the button when the tubes show all "8"s ("88:88:88"). If you do this by mistake and want to go back to the startup procedure, you can do a "Factory Reset". See the user manual to find out how to do this.



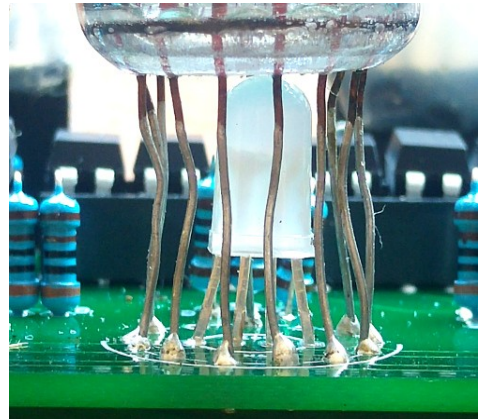
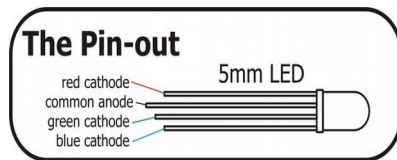
Trouble
shooting

If any of the channels does not go out completely:

Sometimes the LEDs don't go out completely (especially the Blue channel, because the blue LED is very sensitive). If this happens, carefully clean the flux residue from around Q3-5 (of course with the power off) using a cotton bud and pure alcohol. The FETs are very sensitive to being partially turned on due to tracking over the flux. Let the board dry completely before re-testing.

Notes:

- Q3 - Q5 should be orientated with the flat side as shown on the board. Some FETs come with the leads in a row rather than in a triangle. If this is the case, bend the middle lead slightly so that it fits the holes in the board (see hint).
- Mount the RGB LEDs close to the board, but be careful not to strain the LED too much. Normally it is best to mount the LEDs until the “lump” in the leads (about 3 - 4mm proud of the board). If you mount the LEDs too high, the tubes have to be mounted much too high from the board.
- The RGB LEDs have a tiny “o” on the board to indicate where the common anode goes. The anode is the **longest** lead of the RGB LED.



The RGB LEDs should be close to the board (but don't strain them)

Hint: 2N7000 mounting

To mount the 2N7000 FETs, bend the middle lead back slightly. It will then fit in the PCB without problems.



2N7000

Anode Control Circuit:

Parts List:

S24	SOCKET 24
OK1	EL817
OK2	EL817
OK3	EL817
OK4	EL817
OK5	EL817
OK6	EL817
R3	1k
R4	1k
R5	1k
R18	1k
R19	1k
R20	1k
R6	3k
R7	3k
R8	3k
R26	3k
R27	3k
R28	3k

This circuit controls passing the HV to the anodes of the tubes. The micro-controller multiplexes the anodes by turning each of them on it turn for a very short period of time. The software controls the rate of the multiplexing and the order in which the anodes are activated.

Video tutorial: https://youtu.be/_ojXuThD-n0

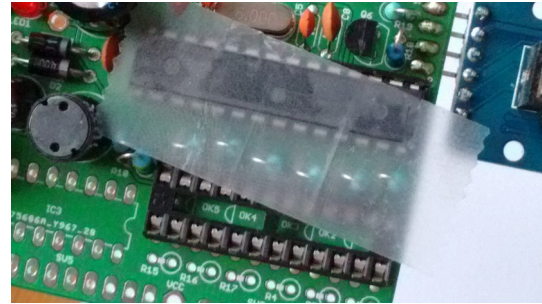
Notes:

- The Opto-isolators fit into the 24 pin socket snugly. Be careful to put them in the right way round. The dot denotes pin 1 and should be on the side closest to the edge of the board. All 6 should fit perfectly into the 24 pin socket.
- The Opto-isolators are socketed because they are sensitive to heat and are easily destroyed if you apply too much heat to them. Putting them in a socket means that we don't run the risk of destroying them while soldering.

Hint: Putting the resistors in

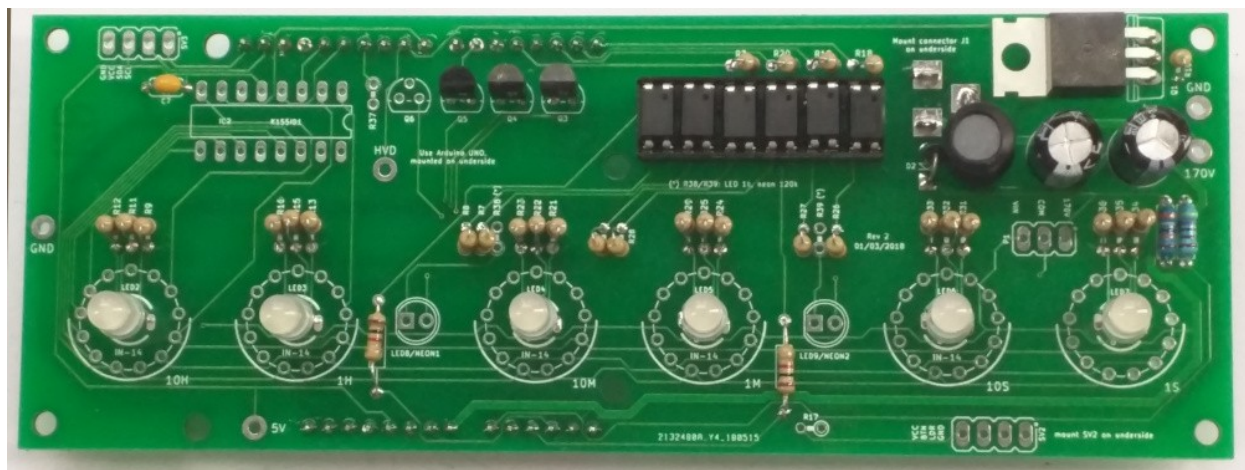
A trick that can speed assembly up is to use a piece of normal sticky tape to hold things in place while you solder them. This makes it easier to solder and gives a better result.

Place the components, and then temporarily tape them into place.



Using tape to hold resistors in place

After this step, your board should look like this:



Anode Control Circuit

Cathode Control Circuit:

Parts List:

IC2	K155ID1
S16	SOCKET 16

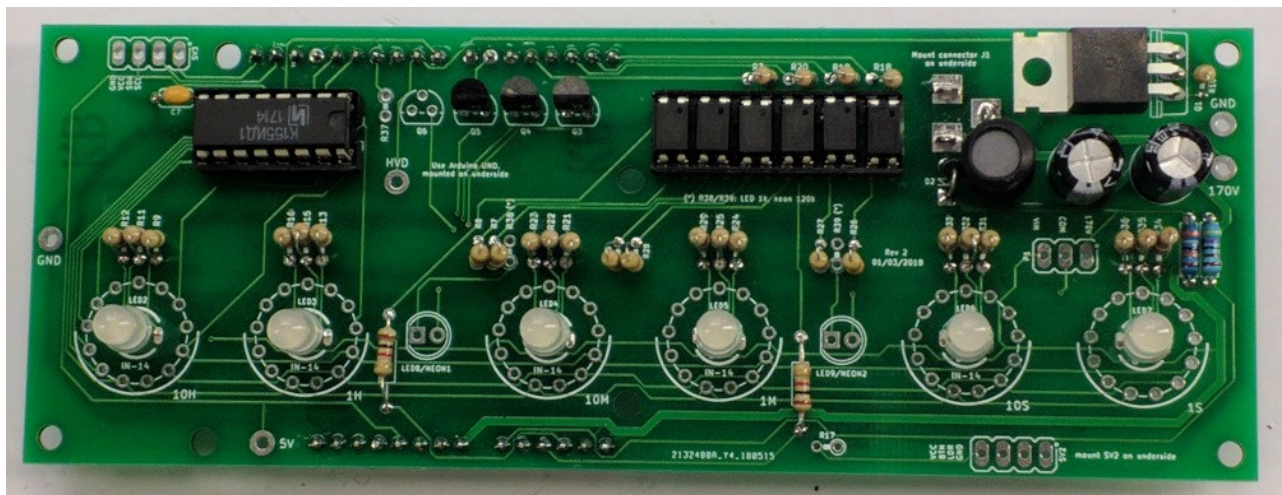
This part of the circuit controls which cathode will be lit. The cathode lit changes rapidly during normal operation, because the display is multiplexed.

Notes:

- Be careful to orient the K155ID1 correctly

Place the 16 pin socket and the connector, and then put the cathode driver chip on the board in the socket.

After this step, your board should look like this:



Cathode Control Circuit

Separator LEDs/Neons

Parts List:

Q6	MPSA42
R37	1k
R38	1k or 120k (see note)
R39	1k or 120k (see note)
LED8/NEON1	LED 5mm or neon (see note)
LED9/NEON2	LED 5mm (see note)
Neon1	Neon indicator lamp (see note)
Neon2	Neon indicator lamp (see note)
P1	Connect COM to HV or VCC

In this step, we prepare the mounting of the LEDs or neons, but we don't mount them yet. We do this after the tubes are mounted because now we can judge the height that the separators should be. If you are experienced, you can also mount the separators now, but if you are not, mount the separators as the last step.

Notes:

- You can install either LEDs or neons as the colon indicators. There is a minor change to the components depending on which you choose.
- **For LEDs:** You need to connect P1 "VIN" to "COM" and choose 1k values for R38 and R39. You need to take care of the polarity of the LEDs, following what is marked on the board.
- **For neons:** You need to connect "170V" to "COM" and choose 120k values for R38 and R39. The polarity of the neons does not matter.
- Be careful to adequately insulate the LED leads using some heat shrink or excess plastic sleeving.
- If you are not sure about how high to mount the separators, mount these after you have finished the tube mounting. If you are ok to mount the separators now, usually the separators need to have leads that are 38mm (above board) + 5mm (through board and below board) = 43mm.

Front Panel components

When all the components are installed, you are finished with the main build of the clock. You can now add the “front panel” components. These will usually be added on the outside of the clock case.

SV2	4 pin connector
LDR	LDR
R17	10k
S1	SWITCH

The switch connects to ground when closed. It uses the internal pull-up resistor provided by the Atmega on the input pin to pull the input to VCC when the switch is not closed.

The switch is de-bounced in software, so practically any switch you want to use is suitable. A simple switch is provided in the kit, but you might want to substitute this switch with one that suits your case.

The LDR forms a potential divider with R17. One leg of the LDR must be connected to the input pin, the other leg must be connected to GND.

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.

Please see the connection diagram in the schematics to understand how to wire the front panel components.

Notes:

- You might want to mount the front panel connector SV2 on the bottom of the board.

Mounting the tubes

The tubes must be mounted carefully so that they sit neatly over the top of the RGB LEDs, and so that they are upright and parallel. It is quite tricky to mount the tubes, so take your time and read through the instructions carefully before you start.

Video tutorial: https://youtu.be/_C5GYezHrq8

It is **VERY** hard to correct mistakes made on mounting the tubes, so plan out each move you make and think it through.



Important!

Install the tubes one at a time and test after each tube is installed!

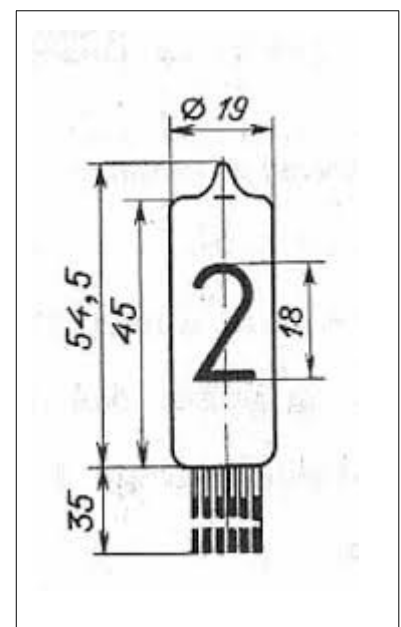
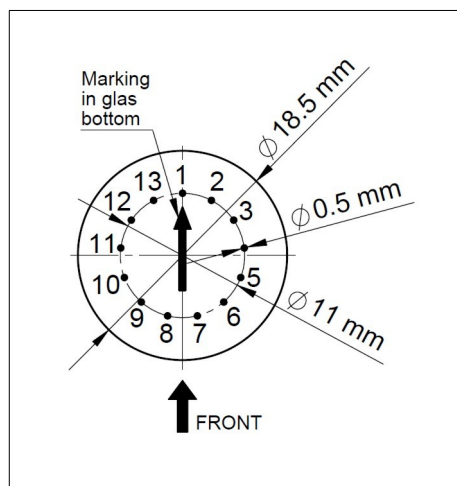
It is very hard to locate where an error is if a tube is faulty or you make a solder bridge by mistake, because all the tubes share the cathode control lines.

To help in locating a bad tube or a construction error, test the display after you install each tube, and deal with any issues that appear before moving on.

Start with the seconds tube and move to the hours tube, and let the test run for about 30 seconds each time, so each cathode is tested.

The **pin out diagram is viewed from the bottom of the tubes**. 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	Left 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	Right decimal point



Installing the tubes is quite a fiddly process, and it can take a long time and plenty of patience. Please see the hints below for some ideas how to make the process easier.

- It is best to carefully spread the legs of the tube out so that they are not all bunched up. 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 pin 3 by 0.5mm and so on. Pin 12 is shorter than pin 3 by 5mm at the end. This means that you can thread the leads into the holes more easily.
- You can remove the decimal point leads on each side of the anode (pins 2 and 13) if you want - they are not used. Remember to leave an unpopulated hole on the board on each side of the anode in this case. Be careful to trim the right leads if you want to remove them!

- Place the tube base against the top of the LED, without leaving a gap.
- To make the tube stand upright, solder just three leads at the beginning (e.g. pin 1, pin 6 and pin 10). This makes a sort of “tripod” out of the pins you have soldered and you can then easily align the tube so that it is perfectly upright by reheating only one of the pins at a time. Once the tubes are upright and aligned, you can solder the remaining pins.
- If you want to test the tube before you finally solder it, you can turn the clock on while only the first three pins are soldered, and usually all of the segments will work. Make sure you don’t short out the remaining leads.
- For last resort, final, small adjustments when all the pins are soldered, you can force the tube slightly so the leads give. It is best to align the tubes without force. IN-14 tubes are robust, but they are made of glass and must be treated with care.
- Once the tubes are mounted, you can trim the colon LEDs to match the height of the tubes.

RTC/WiFi module

Parts List:

RTC	RTC or WiFi module
SV3	CONN_RTC

The RTC module is a traditional Real Time Clock module. Unfortunately we are no longer permitted to post the battery with the kit, so you will have to get your own. The battery you need to get is a CR2032, or a LIR20132 (rechargeable).

You can mount the RTC module directly on the board, or as a separate board connected by flying leads.

The markings on the board need to match up with the markings on the module. In particular, the VCC and GND need to be in the right orientation.

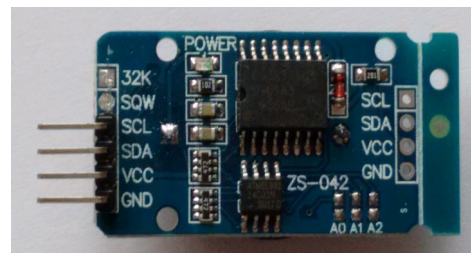
The WiFi module has it's own instruction manual. Please refer to that if you have the WiFi option.

- The RTC module has two sets of contacts on it. You can use either the side with the pins on it or wire up the other side with flying wires. If you use the side with pins, you should carefully remove the two unused pins (see hint). If you use the contacts on the module, you should leave CONN_RTC off the board.
- If you receive the connector header as a single strip, break off 4 pins for SV3.

Hint: Trimming the extra pins on the RTC module

ONLY if you want to mount the RTC module directly onto the main board (you can also do it via flying leads), trim off the pins "32K" and "SQW" using a pair of precision side cutters.

If you want to mount using flying leads, you can skip this step and use the four holes on the other side of the board.



RTC Module with pins removed

If you have the WiFi module, just wire it up as if it was the same as the RTC module.

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.

First perform a factory reset to make sure that no strange values have been left in the EEPROM.

Next, check the value of the **PWM On Time** configuration. Try increasing this until the brightness is constant, but be careful not to set the value too high. The longer the On Time, the more the MOSFET has to conduct current, and this will cause it to heat up. A good value for small tubes is 120-150, larger tubes may require 150-200.



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

The display does not come on, but I do have a high voltage.

Try pressing the button. If the display comes on, you probably have display blanking mode set. Check the configuration.

Check the orientation of the opto-couplers.

Check the LDR connection. In some cases, the dimming algorithm does not start

up as expected when no LDR is present. Shine a bright light on the LDR.
In some cases, a factory reset can help.



Trouble
shooting

One or more of the RGB LEDs will not go out completely

One of the LEDs (either one of the R, G or B channels, or the TICK LED) does not go out completely when it is configured to be dark or at the darkest point of the “pulse” flash.

The FETs which drive the LEDs are very sensitive, and can pick up the stray voltages which are carried by excess flux on the board. **Carefully** clean with a non-scratching instrument between the three pins on the FET of the affected channel. Alternatively, use a solvent to remove excess flux.



Trouble
shooting

The MOSFET gets really hot.

Try a factory reset. There is a setting about how hard the IRF740 should be driven “PWM On Time”. Perhaps the value has not been set properly. The default value should be OK most of the time, but depending on the tubes and power supply, this might need adjustment. The lower the value, the less power will be used and the less hot the MOSFET will run, but also the less power will be available to drive the tubes.

Check the power supply. If the power supply is too “strong” (too much voltage or too much current capacity), the MOSFET will have to carry high currents. Try a different power supply. 9V and 500mA is ideal.

Change the settings for the “PWM On Time”. Adjust it to be as small as possible without a loss of brightness. This also reduces the power consumption of the module: normally it should not consume more than 3W.



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 Arduino

In this kit, you will need to put the code yourself onto the Arduino. If you don't load the code onto the Arduino, the clock will not work at all. At the latest, you will discover this during the HV circuit build where the High Voltage output will only reach the input voltage instead of the required 180V.

Please follow these steps to be program the Arduino.

Installing the libraries

Before you can compile the code, you will need to install the libraries which are used by the code. If you don't do this, the clock code will not compile.

The needed libraries are:

```
#include <DS3231.h> // https://github.com/NorthernWidget/DS3231 (Wickert 1.0.2)  
#include <TimeLib.h> // http://playground.arduino.cc/code/time (Margolis 1.5.0)
```

These are standard libraries, which have not been changed, and you can install them either using the Library Manager (Sketch → Include Library → Manage Libraries) in the Arduino IDE, or by installing the reference version of the Zip files using the option in the IDE (Sketch → Include Library → Add .ZIP Library). You should find the reference libraries in the "Libraries" folder.

Compiling the code

Once you have the libraries installed, you can compile the code. You need to use the code in the folder "ArdunixFade_9_6_digit". Load this into the IDE and compile it. If it compiles, you can plug your Arduino UNO into the USB port and upload it.

Parts list / BOM

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

PCB	1
LED9,LED8 (option)	2 LED 5mm
neon1,neon2 (option)	2 neon
C1	1 2.2uF
C3	1 220uF 35V
LED2,LED7,LED6,LED5,LED4,LED3	6 LED RAGB (CA)
D2	1 UF4007
Socket 2	1 16Pin DIP
IC2	1 K155ID1
R2	1 4.7k
R1	1 390k
R7,R8,R28,R27,R26,R6,R10	7 3k
R13,R12,R9,R11,R3,R19,R18,R23,R20,R21,R22,R24, R25,R29,R31,R32,R33,R34,R35,R36,R37,R4,R5,R15,R16	25 1k
R38,R39 (option)	2 1k
R38,R39 (option)	2 120k
R17	1 10k
Socket 1	1 24Pin DIP
OK1,OK2,OK3,OK4,OK5,OK6	6 EL817
SV2,SV3	3 4 pin header
Q3,Q4,Q5	3 2N7000
HDR1	1 6 pin header
HDR2, HDR3	2 8 pin header
HDR4	1 10 pin header
Q6	1 MPSA42
C7	1 100nF
L1	1 100uH
Q1	1 IRF840
N2,N5,N4,N3,N6,N1	6 IN-14
DS3231 RTC Module (option)	1
WiFi Module (option)	1
Switch	1
LDR	1 GL5516
J1	1 Barrel Jack

Revisions:

V0001: 02Aug2018: Initial version

V0002: 26Oct2018: Update instructions about HV testing

V0003: 05Dec2018: Correct external components drawing, thanks Kent!