Arduino Nixie Clock "Classic Revision 6"

Construction Manual
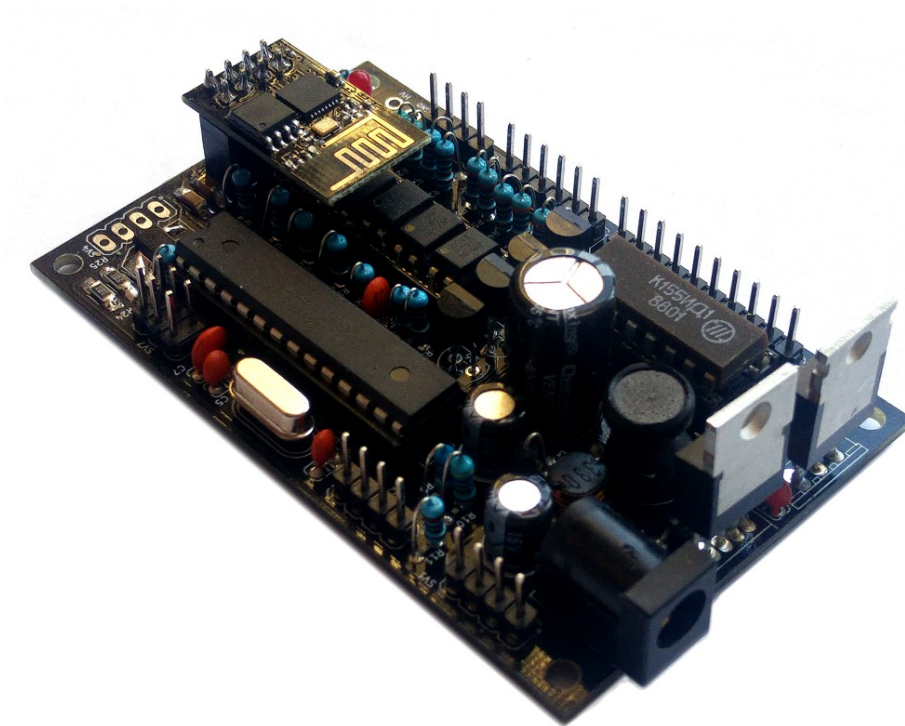
Table of Contents

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

Software

The software is available on GitHub at the address:

https://bitbucket.org/isparkes/nixiefirmwarev2

This board works with the firmware under "ClassicRev6".

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.

Time Providers

Real Time Clock (standard)

By default, the clock comes with a Real Time Clock (RTC) module which provides a battery backed time source that remembers the time even when the clock is not powered up.

WiFi Time Providers (optional)

More up to date is a WiFi real time provider, which logs into your home WiFi network and

periodically retrieves the time from Internet time sources. These are accurate to 1 second, and auto adjust for Daylight Savings Time. You configure once, and then the module remembers the configuration forever.

Additionally, the WiFi time module gives you a easy to configure interface, which you can use to set up the clock using a tablet, phone or computer.
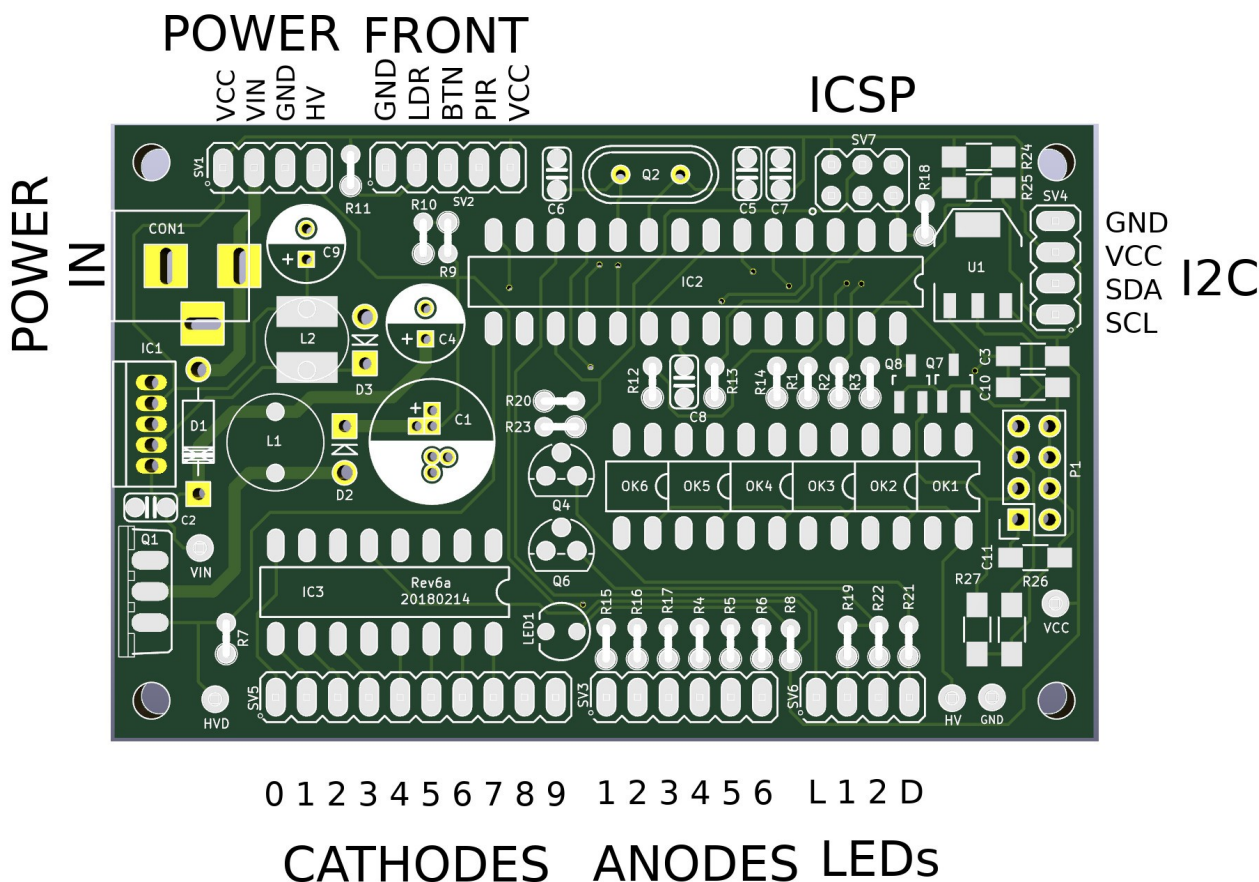
Component Identification

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	External power should be applied to the board with this connector. Any DC input source is possible, from 9V – 12V. 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 the MOSFET and voltage regulator. If you heat sink these components, they must have separate heat sinks, or be insulated from the heat sink (the metal tabs on the components are electrically connected). The absolute maximum input voltage is 16V. Any higher voltage than this will damage the board within a few seconds! The input VIN is protected against the input being connected with reversed polarity. The input current ranges from 300mA to 1A depending on the size of the tubes and the number of LEDs you are driving. **VCC:** *Output:* of regulated 5V **VIN:** *Input:* The positive side of the input supply **GND:** *Input:* The negative side of the input supply **HV:** *Output:* of high voltage for driving external neons etc.

| Connector	Description
FRONT	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 and one lead of the button are connected to this. **LDR:** *Input:* The other lead of the LDR is connected to this **BTN1:** *Input:* The other lead of the button is connected to this input **PIR:** *Input:* This input allows the connection of a PIR or microwave module to detect movement near the clock. This can be used in the firmware to set blanking to preserve tube life. **VCC:** *Output:* Regulated 5V output to drive any LEDs or lighting, as well as the PIR/microwave sensor. Note that you can also connect the LEDs to the VIN if you want to reduce the load of the regulator.
LEDs	This connector lets you drive the LEDs/neons for the separators, as well as the back light NeoPixels. Outputs 1 and 2 are open collector outputs, and can drive either LEDs or neons. **L:** *Output:* NeoPixel (WS2812B) data output. This output allows you to drive the RGB back lights. The back lights are daisy chained together. ("DATAO" on schematic) **1:** *Output:* PWM LED/Neon driver 1. ("TICK_LED1" on schematic) **2:** *Output:* PWM LED/Neon driver 2. ("TICK_LED1" on schematic) **D:** *Output:* Driver for the Decimal Points for tubes that support it. ("DP" on schematic) **Note:** Even though there are two outputs for "tick" LEDs, they will always be driven by the same signal. It is not possible to drive the outputs individually.
I2C	This is an exposed I2C bus, which can be used to connect an RTC (Real Time Clock). **SCL:** I2C Clock **SDA:** I2C Data **VCC:** 5V regulated **GND:** Ground
CATHODES	The terminals to the cathodes (individual digits "0" - "9") for each tube. Terminal "0" drives the cathode "0" and so on.
ANODES	The terminals to the anodes for each tube. The allocation of anode to tube is: 1. Seconds 2. Tens of seconds 3. Minutes 4. Tens of Minutes 5. Hours 6. Tens of hours
ESP8266 WIFI	If you have the WiFi module option, you can put an ESP8266 (ESP-01) module in this socket, which will let the clock synchronize itself using your home WiFi network, and also allow you to configure the clock via a browser, tablet or phone.

Schematic

Below is the schematic for the clock.

And for the external components, showing how they are connected.
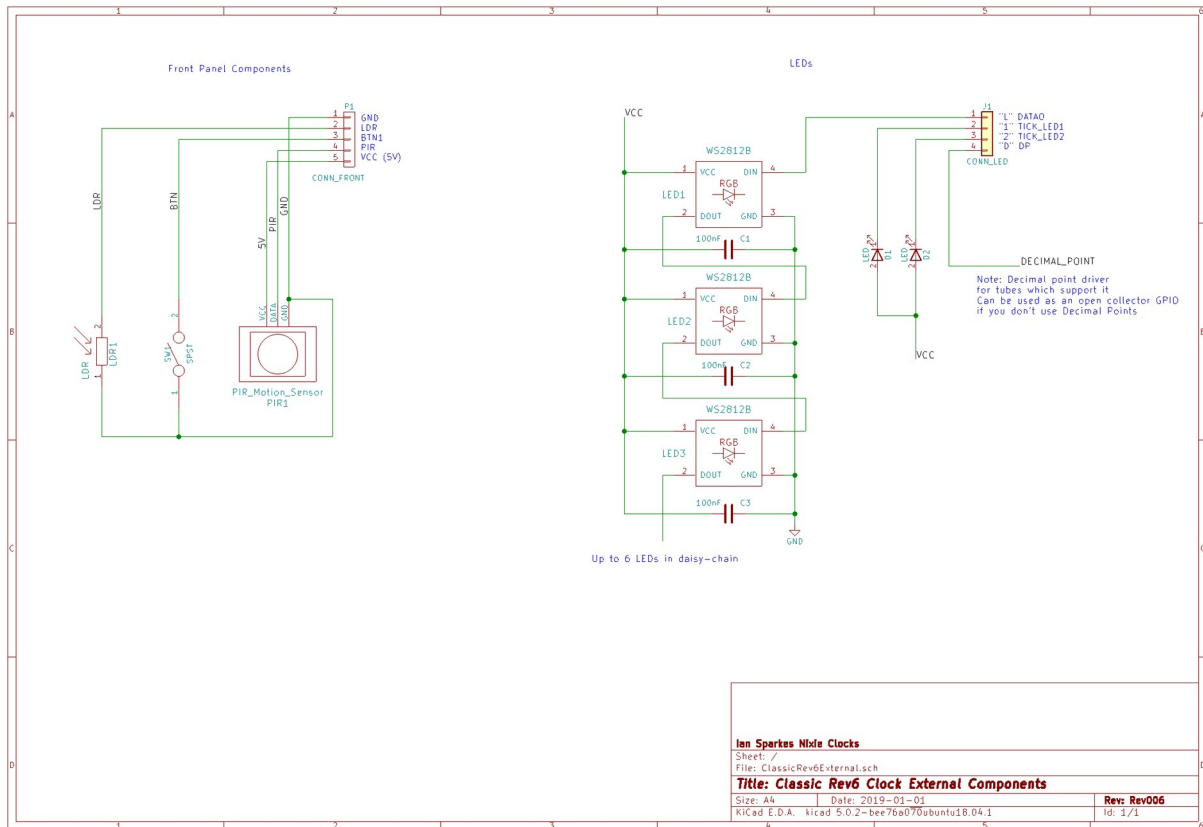
One side of the LDR and switch are connected to ground (pin 1, SV2).

The LEDs are driven from VCC or VIN, depending on the configuration. Here are some suggestions:

1 TICK LED:
Run the TICK LED directly from the board pin "1", either from VCC or VIN.

2 TICK LEDs:
Run one TICK LED directly from the board pin "1", either from VCC or VIN. Run the other TICK LED directly from the board pin "2", either from VCC or VIN.

4 TICK LEDs:
Run the TICK LEDs by putting 2 in series on each of the board pins "1" and "2". In this case you will need to run the LEDs from VIN.

RGB Back Light LEDs:
The data out drives the input of the first NeoPixel. Further NeoPixels are daisy chained up to a maximum of 6 Neo Pixels.

Decimal Point Driver:
If your tubes have a decimal point, you can use this output to drive the decimal points. It is a multiplexed output, and acts as a further cathode for the tube.

8

Construction

Preparation:

I know it is really hard, but please read through the document before you start! Many small items of information are given in the document, and it will be less confusing if you read through the document first. At very least, try to skim through the sections so that you know what is coming later.

You should have a small tipped soldering iron, some thin (<= 1mm) solder, and electronic side cutters. Lead solder is recommended, because it is easier to use, but if you have lead-free solder, that will also work.

Prepare your working space so that you are comfortable and have plenty of room to work. Keep the working area clean and tidy.

When you perform test steps, make sure that you are not going to destroy the board by following these general tips:

- Make sure that any loose wires are tied out of the way so that they don't short out on anything while you are testing.

- Check the board and the workspace carefully for offcuts of component leads. Especially make sure that all the trimmed leads are well out of the way.

If you have the ESP WiFi option, you will also need some basic SMD (Surface Mount) tools and experience. Most importantly some fine solder (<= 0.6mm), some tweezers and a thin tipped soldering iron. The pads for the SMD components are designed for hand soldering, so you don't need to have an SMD oven.

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.

There are some options in this clock kit: If you use the clock with an RTC (Real Time Clock) time provider, you won't need to add the WiFi interface. These options are listed in the BOM and in the instructions.

Please see the appendix to help you identify individual components.

Low Voltage Circuit

Parts List:

L2	33uH
D1	UF4007
C2	100nF
C4	220uF 35V
C9	220uF 10V
D3	1N5819
R8	3k
LED1	LED3MM
SV1	CONN_POWER
IC1	LM2596
CON1	Barrel Jack 2.1 x 5.5

The Low Voltage circuit is a "buck" voltage regulator using an LM2596. It's 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 K155ID1.

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

Notes:

- See the section on "Component Identification" for help with identifying the components.

- If your 33uH inductor is a SMD variety, mount this first! The access to it is hard if you mount it later. Wet one pad with solder, place the inductor how you want it, then re-flow the solder. Once the component is placed, solder the other pad. Don't try to solder the other pad until you are happy with the placement! If you inductor is the traditional through hole variety, mount it as usual.

- If the 33uH inductor is a through hole type, no special instructions are necessary.

- D1 and D3 look very similar, but have different jobs to do. Be careful to get the UF4007 (D1) and 1N5819 in the right places and in the right orientation.

- D1 and D3 should be placed so that the white stripe on the body lines up with the white stripe on the board.

- C4 and C9 must go the right way round. The negative side is marked with a stripe. (See hint)

- C4 and C9 have different voltage ratings. The smaller one (C9) is the one with the lower voltage rating. The larger one (C4) is the one with the higher rating.

- The LED must go with the right polarity. The side which has the longer lead goes nearest the VCC test point. (See hint)

- Put IC1 so that the metal tab lines up with the white stripe on the board. The metal side faces to the *outside* of the board.

Test Step	Once all the components are on the board, hook up the power, and check that the power LED comes on. Check also that the voltage is 5V between the "GND" test point and the "VCC" test point and at the power connector.

Trouble shooting	If the LED does not come on, turn off immediately to avoid damage to the components. Check your soldering and the polarity of the components. If the components are in the right way, connect the power again, and check that the voltage regulator does not get hot. If it does not, measure the voltages in the low voltage circuit. Measure the voltage at the input ("Vin") and at the cathode side of D1 (nearest the capacitor C8). This should measure about 0.7V less than the input voltage.

Trouble shooting	The pad spacing on the LM2596 is quite close. Be especially sure that there are no solder bridges around this component.

If all is well, proceed to the next step. If not, check carefully the orientation of the components and the power leads. Diode D1 protects the board from having the power connected with inverted polarity.

If the LED comes on, check for a few seconds that none of the components 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"
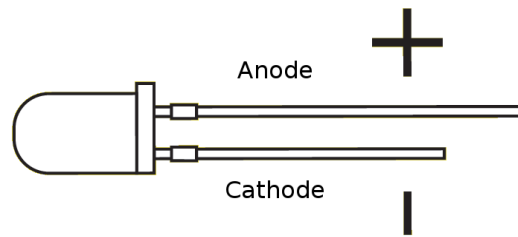
Hint: The LED orientation

The LED has one lead longer than the other, and a flat on one side. The side with the **longer** lead (the anode) goes into the hole on the board nearest the VCC test point.

The LED

11
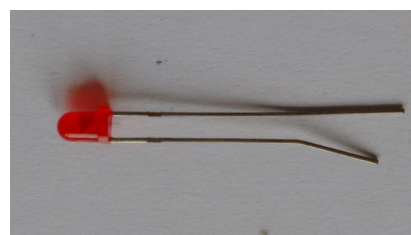
The LED should look something like this:
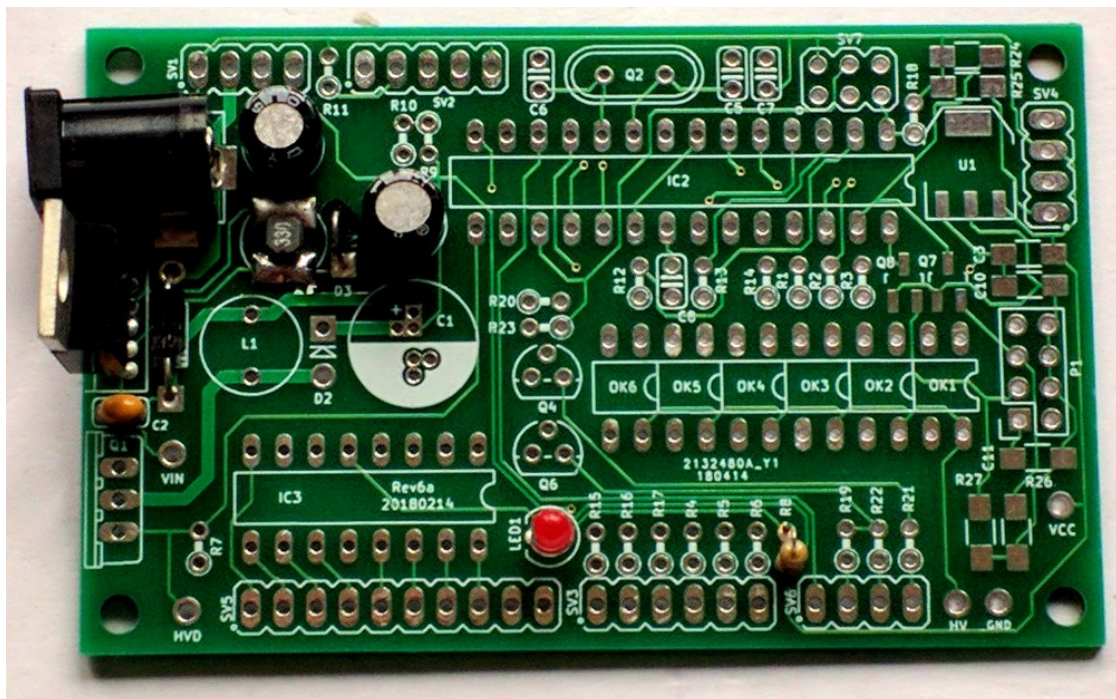

Anode +
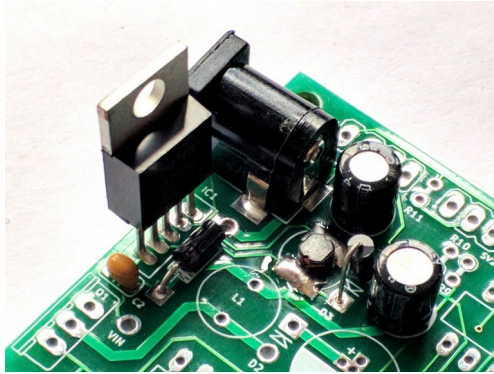
Cathode —

Hint: Mounting the 33uH inductor L2

If you have a Surface Mount 33uH inductor, you should "wet" one pad of the component footprint with solder, then use some tweezers to put L2 in place while re-flowing the solder on the pad (heating it up until it becomes liquid again). When the inductor is in place and the one pad has solidified, apply solder to the other pad, making sure that solder flows underneath the component.

If you have a through-hole variety, just mount it as normal.

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

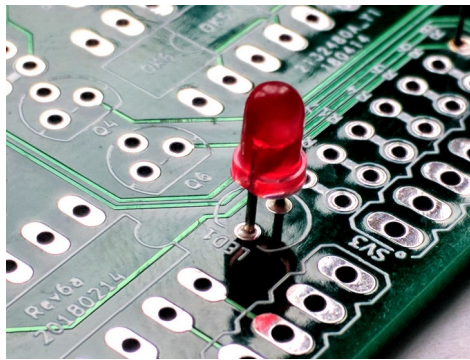*Low Voltage Circuit*

Especially note the orientation of the electrolytic capacitors (white stripe) and the diodes (white stripe on D1 and the flat on LED1):

Low Voltage Circuit Component Orientation

And the orientation of the LED:

LED Orientation

High Voltage Circuit

Parts List:

C5	22pF
C6	22pF
Q2	16MHz
C7	100nF
C8	100nF
D2	UF4007
S28	SOCKET 28
C1	2.2uF 400V
L1	100uH
R7	3k
Q1	IRF640
R9	390k
R10	4.7k
R18	10k
IC2	MEGA8-P

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

When you first start the circuit, it will be in "test mode". This activates the HV generator for 1 minute, and then shuts it down. It is long enough to test the high voltage, but protects both the user and the circuit from being left on in an unregulated state for too long.

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).

- Put Q1 so that the metal portion lines up with the white stripe on the board. The metal side faces to the outside of the board.

- D2 should be placed so that the white stripe on the body lines up with the white stripe on the board. The white stripe connection also has a square solder pad.

- Put the micro-controller socket in first. Make sure that the depression on the end of the socket lines up with the marking on the board. When you put the chip in, the chip should go in with the depression facing to the outside of the board.

Check the orientation of the components before you proceed!

Especially the orientation of R9 and R10, as well as the electrolytic capacitor C1 is important. If you switch R9 and R10, you will put 170V into the micro-

Warning!

controller, and this will destroy it.

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 top of the board (not near the 170V test point).

Warning!

Be careful, we are dealing with high voltages now!

The voltage may be <u>significantly</u> 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.

Test Step

Apply power to the board again. Listen for any stressed sounding buzzing or humming, and check that neither the 7805 nor the MOSFET get excessively hot.

Check that the power LED still lights.

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).

Warning!

Note also that the "Power" header also has high voltage exposed on it!

This is for if you want to drive neons instead of LEDs for the colons. Be careful handling the board, it is easy to touch the "Power" header by mistake. If you are sure you won't be needing it, you can snap the extra pin off and populate only the bottom 3 pins on the connector.

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.

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

The High Voltage generation works for a short time, and then stops. It might be the circuit is producing voltages which are far too high. In this case temporarily load the high voltage generator output with a tube and a series 3k resistor or just a 10k resistor (0.5W or more) and try again. (A tube is best, a resistor gets very hot and might be useless after the test).

Hint: Mounting the 28 pin socket

Mounting the 28 pin socket can be a little difficult. A good trick is to fix it in place with a small piece of tape, and the solder one leg in place. You can hold the socket firm while you "wet" the solder again, which will hold the socket firmly enough to solder the remaining pins. One leg is usually enough to hold the socket in place while you solder the others.
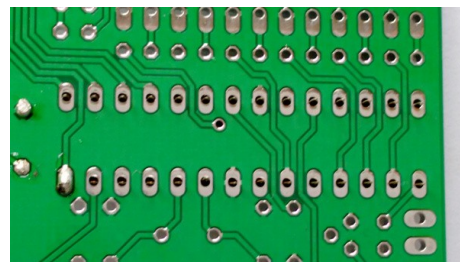
Mounting 28 pin socket

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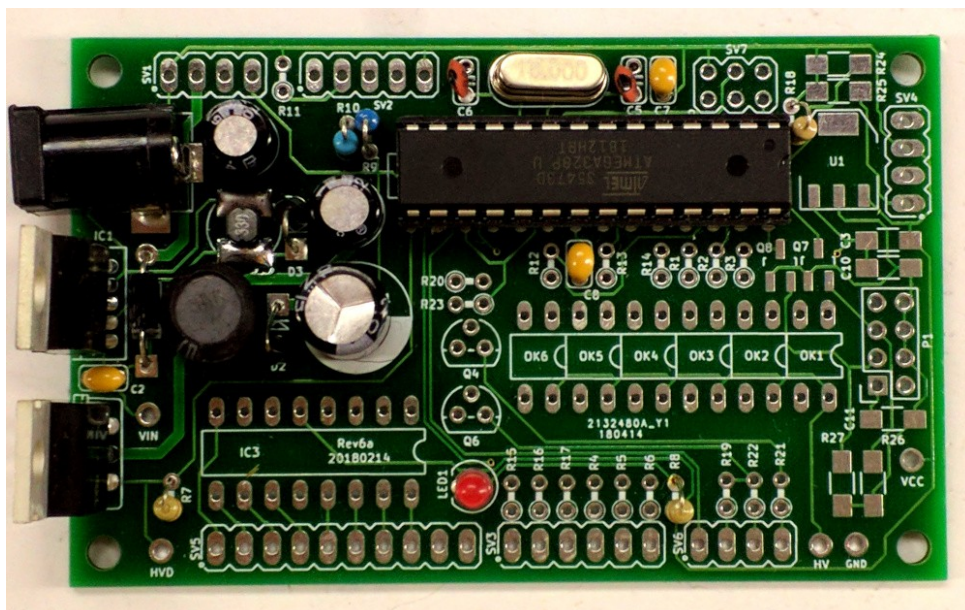
High Voltage Circuit

Anode Control Circuit

Parts List:

S24	SOCKET 24
OK1	EL817
OK2	EL817
OK3	EL817
OK4	EL817
OK5	EL817
OK6	EL817
R1	1k
R2	1k
R3	1k
R12	1k
R13	1k
R14	1k
R4	3k
R5	3k
R6	3k
R15	3k
R16	3k
R17	3k
SV3	CONN_ANODE

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.

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 micro-controller. 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.

- If you receive the connector header as a single strip, break off 6 pins for SV3.

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 is easier to solder and gives a better result.

Place the components, and then temporarily tape them into place.
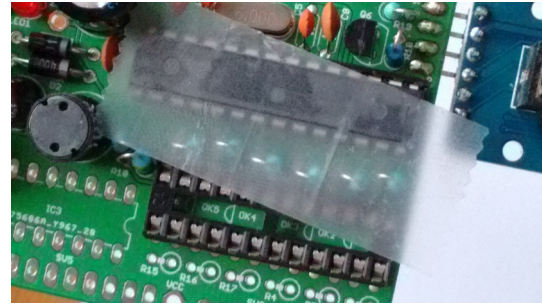
Using tape to hold resistors in place

After you have installed the Anode controls, the board should look this:
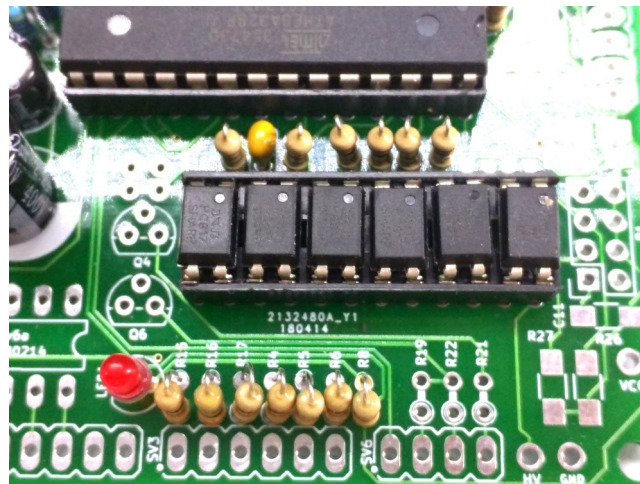
Cathode Control Circuit

Parts List:

   IC3   K155ID1
    S16   SOCKET 16
    SV5   CONN_DIGIT

This part of the circuit controls which cathode will be lit. Each time the digit to be displayed, the correct cathodes have to be set.

Notes:
- Be careful to orient the K155 correctly
- If you receive the connector header as a single strip, break off 10 pins for SV4.

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

After you have done this, the board should look like this:
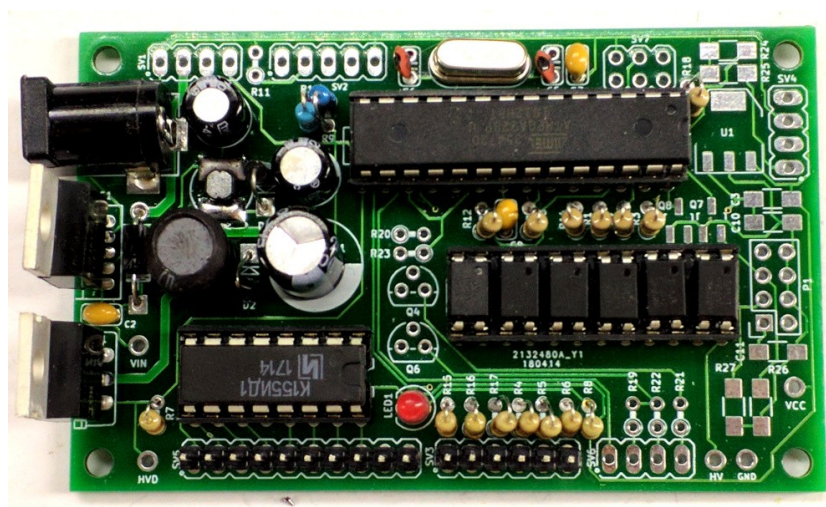
"LED" Circuit and Front Panel

Parts List:

R19	1k
R20	1k
R21	1k
R22	1k
R23	1k
Q6	MPSA42
Q4	MPSA42
TICK LED	LED 5mm (off board)
R11	10k
LDR	LDR (off board)
S1	SWITCH (off board)
LED RGB	WS2812B (off board)
SV6	CONN_LED
SV2	CONN_FRONT

This step installs the "front panel" components and the RGB LED and separator drivers. The MPSA42 device is capable of driving neon separators if you want, but will also drive LED separator(s).

Notes:

- Q4 and Q6 should be orientated with the flat side as shown on the board. Some transistors 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).

- The Separator LED must go with the right polarity. The "-" side has a flat on it, and has the shorter lead (see hint), and must be connected to pin 4 of SV6.

- The LDR is connected between pin 1 (GND) and pin 2 (DLS) of SV2.

- The switch is connected between pin 1 (GND) and pin 3 (BTN1) of SV2.

- The RGB LED is a small 5mm x 5mm package with a tiny notch in one corner. See the pin out below. You should place the small decoupling capacitor close to the WS2812B package. The device is heat sensitive! Be careful soldering it.

- 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 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.

 Test Step

Once you have populated the components, power on. The "TICK" LED(s) should flash on and off slowly. It might take a second or two to come on, but after that should "pulse" on and off.

Hint: MPSA42 mounting

To mount the MSPA42, bend the middle lead back slightly. It will then fit in the PCB without problems.
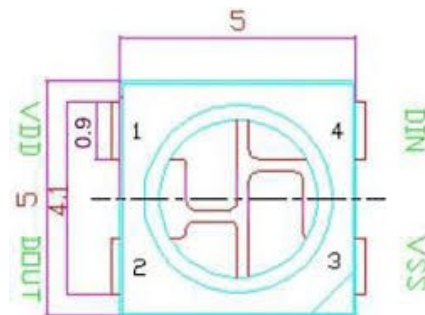
Try not the put these components too close to the board, they are easily damaged by heat. Be quick and precise when soldering them.
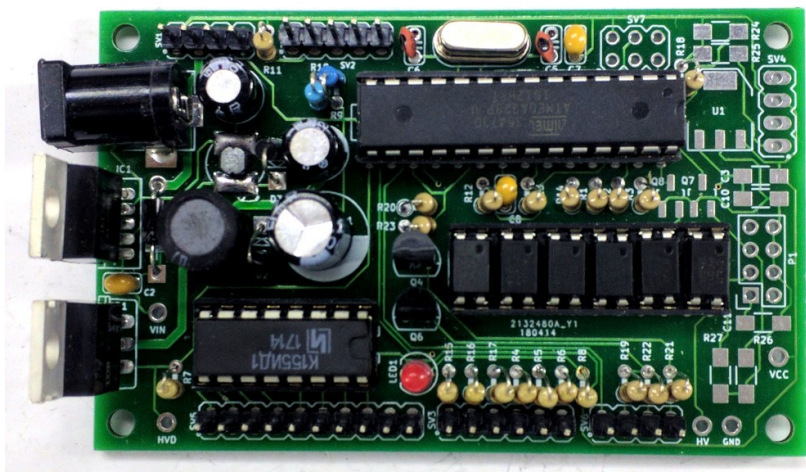

MPSA42

Hint: RGB LED pinout

The RGB LED has quite a complicated pin out. VSS (GND) is closest to the notch. VDD (5V) is opposite. The connection from the board goes into DIN, and DOUT daisy chains to further LEDs.

RGB LED

After this step, the board should look like this:

"Time Provider" Circuit (WiFi or RTC)

Parts List RTC module:

RTC DS3231	RTC (see note)
SV4	CONN_RTC (see note)

Parts List WiFi module:

R24	10k 1206 SMD
R25	10k 1206 SMD
R26	10k 1206 SMD
R27	10k 1206 SMD
Q7	BSS138 SOT-23 SMD
Q8	BSS138 SOT-23 SMD
C10	100nF 1206 SMD
C3	100nF 1206 SMD
C11	10uF 1206 SMD
U1	AP1117-3.3 SOT-223 SMD
P1	2x4 Female Socket

The usual time source for this board is an on-board ESP8266 WiFi module, unless you specifically want to use the battery backed RTC (Real Time Clock) module.

If you connect both, the clock will prefer to use the WiFi time provider, but if this loses connection to the network, it will fall back to the RTC module. Once the WiFi module can connect again, it will take over, and synchronise the time to the RTC module.

Notes:

- If you install the RTC instead of the the WiFi circuit, there is no reason to install the WiFi circuitry.

- Soldering the resistors and capacitors: Solder one pad and then re-flow the solder while keeping the component in place with tweezers. You may find that sliding the component onto the pad from the side helps. Once one pad has been soldered, you can solder the other pad.

- Soldering the regulator: place a tiny solder blob on the center pin and then re-flow the solder while holding the component in place. Once the pin is fixed, you can solder the other pads.

- Soldering the transistors: place a tiny solder blob on the center pin and then re-flow the solder while holding the component in place. Once the pin is fixed, you can solder the other pads.

After populating the components as above, the board should look like this:
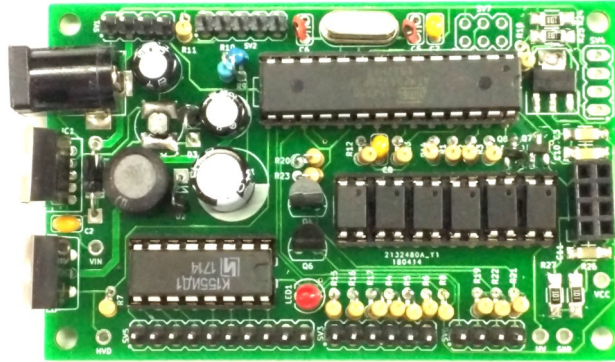
With the power off, Put the ESP8266 into the socket, with the main part of the board facing inwards over the opto-isolators.
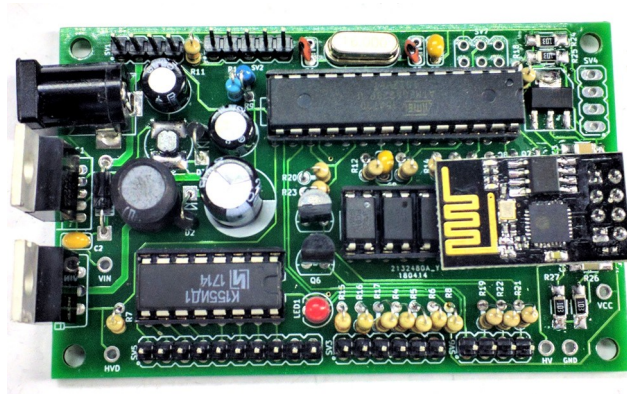

Get the ESP8266 the right way round!

The ESP8266 socket will allow you to put it in either way round. If you put it in the wrong way round you will destroy it! Check the picture below to make sure you put it in the right way round!

Warning!

The ESP8266 should go in the socket like this (so it faces inwards to the middle of the board):

Test Step

Once you have populated the components, insert the ESP module and power on. The LED on the ESP module should flash. Follow the instructions in the "WiFi Time Provider" manual to set up the module.

Once you have set the module up, it should set the time on power up and after every 60 seconds.

Hint: Trimming the extra pins on the RTC module

ONLY if you have the RTC module and 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.
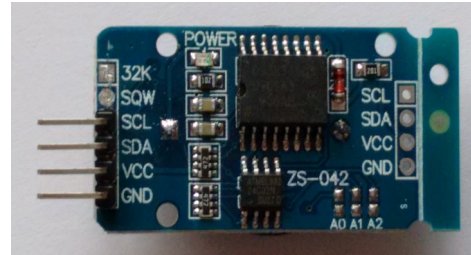
RTC Module with pins removed

Connecting the tubes

When all the components are installed, you are now ready to install the tubes. Either you can wire then by hand or you have a board to put them on. I prefer hand wiring, because I think it has a more "retro" feel.

The anodes are arranged like this:

SV3 Pin	Schematic	Description
1	Anode_1	Seconds
2	Anode_2	Seconds 10s
3	Anode_3	Minutes
4	Anode_4	Minutes 10s
5	Anode_5	Hours
6	Anode_6	Hours 10s

Anode_1 is the one next to the tiny "o" printed on the board next to the connector.

The cathodes are arranged like this:

SV5 Pin	Schematic	Description
	Cathode_0	Digit 0
	Cathode_1	Digit 1
	Cathode_2	Digit 2
	Cathode_3	Digit 3
	Cathode_4	Digit 4
	Cathode_5	Digit 5
	Cathode_6	Digit 6
	Cathode_7	Digit 7
	Cathode_8	Digit 8
	Cathode_9	Digit 9

Cathode "0" is next to the tiny "o" printed on the board next to the connector, which indicates pin 1 of the connector.

Side note: There are some tricks in the software to make the wiring and the PCB easy and logical. We use a translation table in the software, and cross some of the standard channels to make the PCB more logical. You can also see this on the "digit control" section of the schematic. This means that we are not using the 74141 "0" output to drive digit "0", but instead we are using digit "2" of the 74141 to drive digit "0".

In the code (available on GitHub), we see this:

```
// Used for special mappings of the 74141 -> digit (wiring aid)
// allows the board wiring to be much simpler<
int decodeDigit[16] = {2,3,7,6,4,5,1,0,9,8,10,10,10,10,10,10};
```

When we come in with a "0", we decode this to "2" (the 0th element of the array), and in fact, it is the "2" output that is activated, but the wiring brings the "2" output to the "0" connection.

The cathodes to the tubes (the digits for each tube) need to be wired in parallel, "daisy chaining" them, so that the run to the same digit of each tube.

The anodes run exactly one tube each.

Tube wiring

If you bought a kit without a display board, there is the task of wiring the tubes to be done. It can take a long time, but if you do it carefully, it can give a very beautiful "retro" feel.

There are a number of small tricks to make the task easier and the result better:
- Plan how long you want the lead to the board to be and how far apart the groups of digits "hours", "minutes" and "seconds" should be.
- Temporarily either glue, tape or bind together the groups of digits while you are working on them. Later you can choose to substitute the temporary binding, or keep it if you wish.
- If you use heat shrink, do not shrink it until the end of the wiring. The reason for this is that we will complete the looming at the end, and will need to disconnect the board from the loom to tidy it up. Also for the identification of errors, it is useful to be able to switch or disconnect connections.
- Cut the wires you are going to use, use different colors for each cathode or anode if possible. Keep the same color for each cathode as you proceed through the wiring. Make sure that all the wires for each task are the same length. The main wire tasks are:
  - Flying cathode connections (recommended 30cm):
    - 10 x flying cathodes from the board to the tubes
    - 6 x flying anodes from the board to the tubes
  - Intra group cathode connections (recommended 7cm):
    - 3 x 10 x cathodes
  - Inter group cathode connections (recommended 7cm):
    - 2 x 10 x cathodes
  - Flying anode connections (recommended 40cm):
    - 6 x flying anodes from the board to the tubes
- Wire anode 1 and anode 2 to the first group ("seconds") from the board to the tube. Always make the flying lead connect to tube 6 first, then tube 5. This means our connection loom will come out at tube 6.
- Wire cathode "0" and cathode "1" from the board to tube 6 and daisy chain to tube 5. Prepare the daisy chain connection to the next group but **do not strip** the wire to the next group yet.

- You should have a result like this:

- Turn on the clock and check that you can clearly see the digits "0" and then "1" light up in sequence on the seconds tube every 10 seconds. Don't worry if there is "ghosting" on other digits at the moment – this will resolve itself as you connect more tubes and digits. (There are still a lot of floating connections at this point in the build and we are not loading the HV generator propery).
- Proceed to the other cathodes in turn, each time leaving the unstripped lead to the next group hanging.
- Try to pick an order you connect the tube cathodes in, and stick to it. This will give the wiring a "regular" feel. If there are any tangles or crossovers in the wiring, resolve them before soldering the wire in place.

- After you connected all the wires of the seconds group, you should have a result that looks like this:
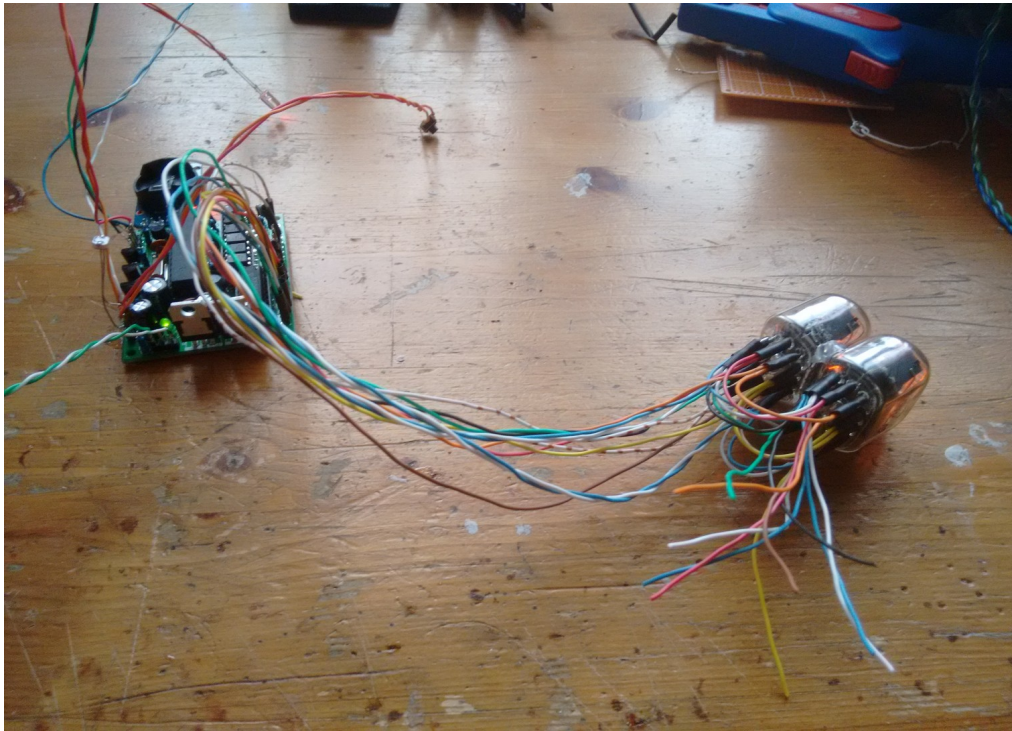

- Turn the clock on, and you should now see the seconds counting correctly.
- Complete the other groups in the same way, keeping the cathode connection colors and the order in which you connect them the same.
- At the end of the connection process, we want to "loom" the wiring. This involves binding and/or plaiting the wires to make them regular. For this you will need to disconnect the wires from the board. Make a note of the colors of each of the cathodes and anodes, because we will need to reconnect them in the same way.
- I usually plait the 6 anodes together in 3 groups two wires each, and the 10 cathodes in 5 groups of two wires. 3 plaiting and 5 plaiting can be found on YouTube. It is optional to plait the wires and simply binding them gives a good result as well.

The wiring for the tubes is this:
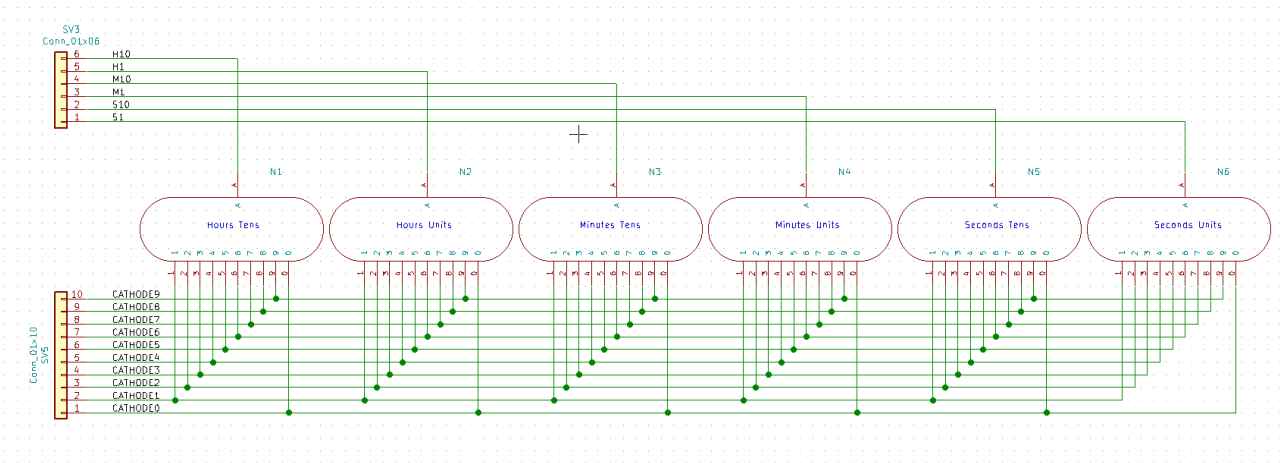
Here is a picture of a plaited loom in action. This particular version of the clock has two blue "tick" LEDs in series and three RGB LEDs (one for each digit group):
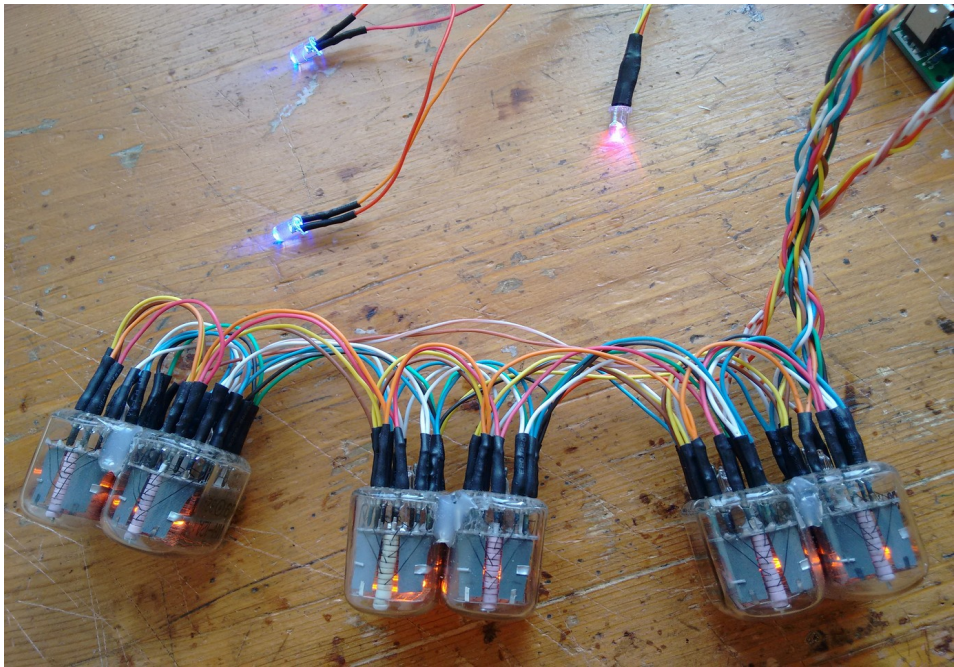
Hacks and alternatives

There are a number of options which can be built into the board easily.

Using Neons instead of LEDs for the "Tick" LED.

This is a very simple modification. Q6 is a high voltage MPSA42. The MPSA42 transistor has a compatible pin out to the FET (drain = collector, gate = base, source = emitter), but can withstand 300V.

This means that you can connect a neon indicator (with a ballast resistor) instead of LED(s).
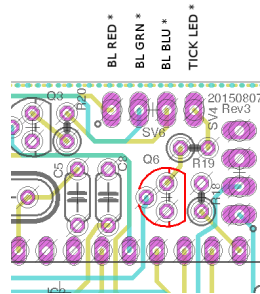
This gives a more "retro" feel to the result.

For each neon, place a 270k resistor in series, and drive from the HV output on the power connector on the board. Here is an example with a 270k resistor and two INS-1 neons:

Driving 2 neons from the "Tick" circuit

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. 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

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.

shooting   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.

No high voltage, but nothing gets really hot. LDR attached

Check that the LDR is connected between GND and the correct pin, and the connections have not been reversed with VCC (also on the same connector) and the BTN pin. One symptom is when you press the button the digits light up brightly.

Trouble shooting

The MOSFET gets really hot.

Try a factory reset. There is a setting about how hard the IRF640 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

The MOSFET gets really hot.

Try a factory reset. There is a setting about how hard the IRF640 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

Trouble shooting	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 micro-controller comes preprogrammed. 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 with an Arduino Uno

We supply the 328P micro-controller chips with a standard Arduino boot loader, so you don't need to have a special programmer in order to update the software, a standard Arduino UNO is enough.

To program the 328P, simply remove it from the clock board, and place it in the Arduino UNO. Then you will be able to program the controller as you would any other Arduino UNO, simply upload the software onto the controller. Put the 328P back into the clock board and you are done.

You can also program the 328P micro-controller with a programmer, but you will lose the possibility to program in the Arduino UNO, unless you remember to burn the boot loader again.

That's it!

Programming with ICSP (In Circuit Serial Programming)

As of Rev4, there is also an ICSP header socket provided on the board. This means that you can program the micro-controller without even removing it from the board. For this however, you do need to have a suitable programmer. If you intend to use ICSP, you need to populate the header.

Programming this way is extremely simple. Plug the 6 pin ISCP programmer cable into the header (pin 1 is nearest the micro-controller) and upload the program is you would with any other ICSP target.

Parts list / BOM

Here is the list of the parts needed:

Low Voltage Circuit

PCB	PCB	☐
D1	UF4007	☐
C2	100nF	☐
C4	220uF	☐
IC1	LM2596T-5	☐
C9	220uF	☐
R8	3k	☐
LED1	LED3MM	☐
SV1	CONN_PWR	☐
CON1	Barrel Jack	☐
D3	1N5819	☐
L2	33uH	☐

High Voltage Circuit

C5	22pF	☐
C6	22pF	☐
C1	2.2uF 400V	☐
C7	100nF	☐
C8	100nF	☐
D2	UF4007	☐
S28	SOCKET 28	☐
Q2	16MHz	☐
L1	100uH	☐
R7	3k	☐
Q1	IRF640	☐
R9	390k	☐
R10	4.7k	☐
R18	10k	☐
IC2	MEGA328-P	☐

Front Panel

R11	10k	☐
LDR	LDR	☐
S1	SWITCH	☐
LED RGB	LED RGB Anode	☐
TICK LED	LED 5mm	☐
SV2	CONN_FRONT	☐

RTC (Option)

SV4	CONN_RTC	☐
RTC	RTC	☐
BAT1	CR2032	☐

Anode Control Circuit

S24	SOCKET 24	☐
OK1	EL817	☐
OK2	EL817	☐
OK3	EL817	☐
OK4	EL817	☐
OK5	EL817	☐
OK6	EL817	☐
R1	1k	☐
R2	1k	☐
R3	1k	☐
R12	1k	☐
R13	1k	☐
R14	1k	☐
R4	3k	☐
R5	3k	☐
R6	3k	☐
R15	3k	☐
R16	3k	☐
R17	3k	☐
SV3	CONN_ANODE	☐

Cathode Control Circuit

IC3	K155ID1	☐
S16	SOCKET 16	☐
SV5	CONN_DIGIT	☐

LEDs

R19	1k	☐
R23	1k	☐
R20	1k	☐
R21	1k	☐
R22	1k	☐
Q6	MPSA42	☐
Q3	2N7000	☐
Q4	2N7000	☐
Q5	2N7000	☐
SV6	CONN_LED	☐

WiFi Interface (SMD)

R24	10k	☐
R25	10k	☐
R26	10k	☐
R27	10k	☐
Q7	BSS138	☐
Q8	BSS138	☐
C10	100nF	☐
C3	100nF	☐
C11	10uF	☐
U1	AP1117-3.3	☐
P1	CONN_ESP	☐

Note: If you find 3k resistors hard to find, 2k7 will do just as well.

Revisions:

V0001: 23Jun2017: Split Instruction and construction manual
V0002: 31Dec2018: Correct note about installing RTC and WiFi at the same time
V0003: 23Jan2019: Correct board legend
V0004: 02May2019: Correct External schematic
V0005: 23Apr2020: Correct External Wiring on P28, thanks MikeS
V0006: 30Apr2020: Clarify C4 and C9 voltage ratings
V0007: 19Jun2020: Correct MOSFET naming
V0008: 31Dec2020: Correct anode naming