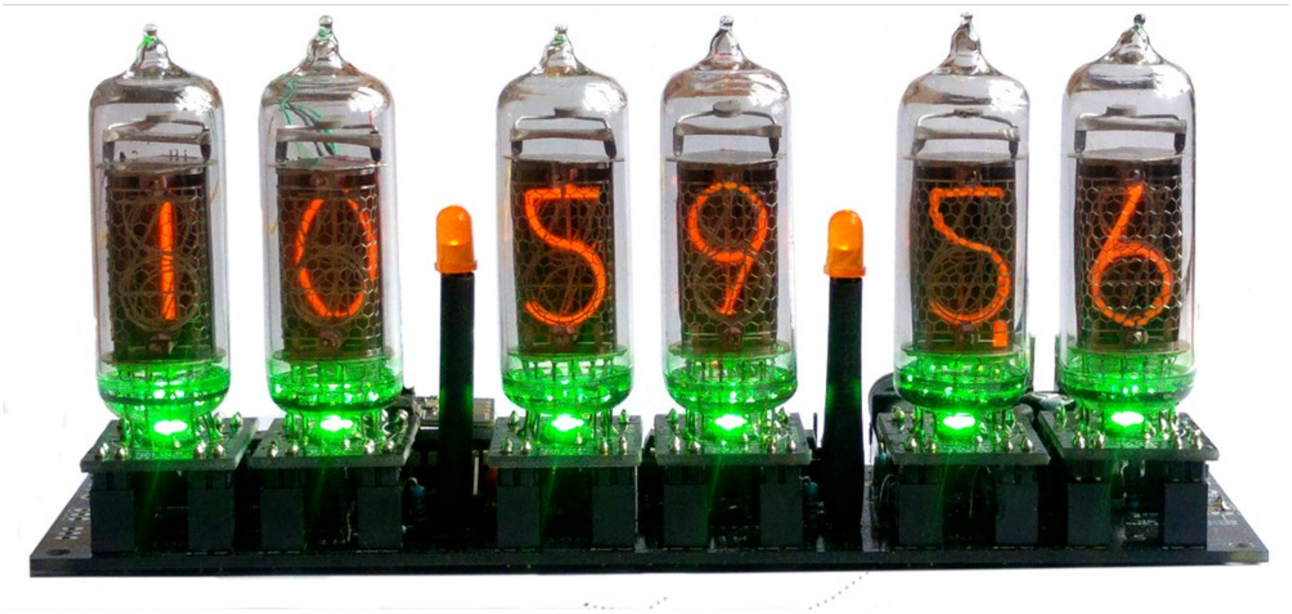
Arduino Modular IN-14 Nixie Clock Rev3

Construction Manual



What this document is about

This document is the user manual and construction manual for the Arduino Module Nixie Clock **Revision 3.** You can easily tell if you have a Revision 3 clock, because the tube holders have a 4-point mounting system (four 2x2 headers on each tube board). If you have tube holders with a single 6x2 header on it, you have a Revision 2 board.

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

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 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. Note that the startup procedure only runs for 60 seconds at a time, after that you will have to restart the clock to get the high voltage running again. This is an intended safety feature!

For a video of the startup process on a finished clock, please see:

https://youtu.be/XA3LOPLX8vI

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

Factory Reset

To reset the clock back to initial settings, hold down the button while powering on. The “tick” LED will flash 10 times to signal that the reset has been done.

Everything will be reset back to the factory default state.

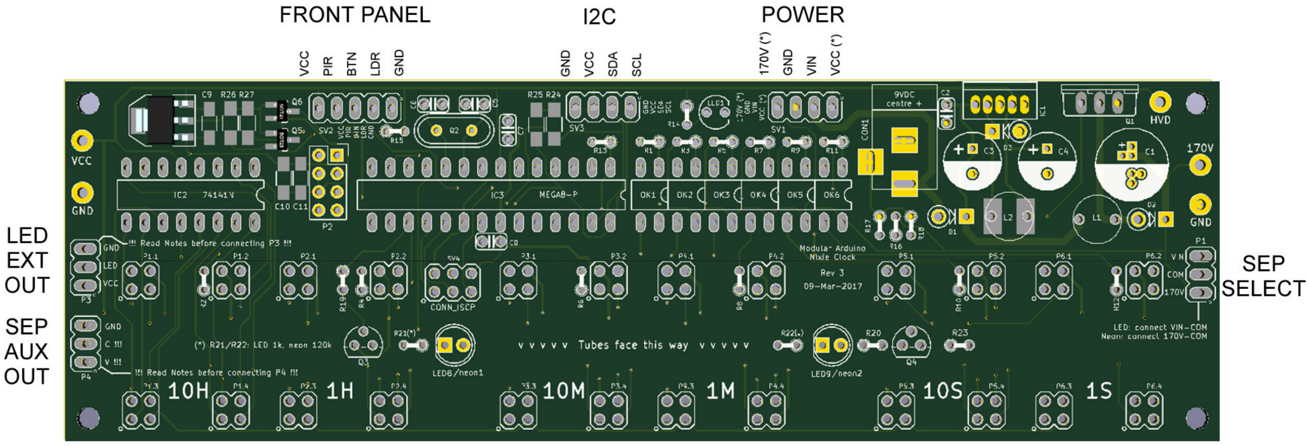
External power supply

The perfect voltage for the external power supply is between 7.5V and 12V DC. We recommend that you use a 9V DC supply.

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):



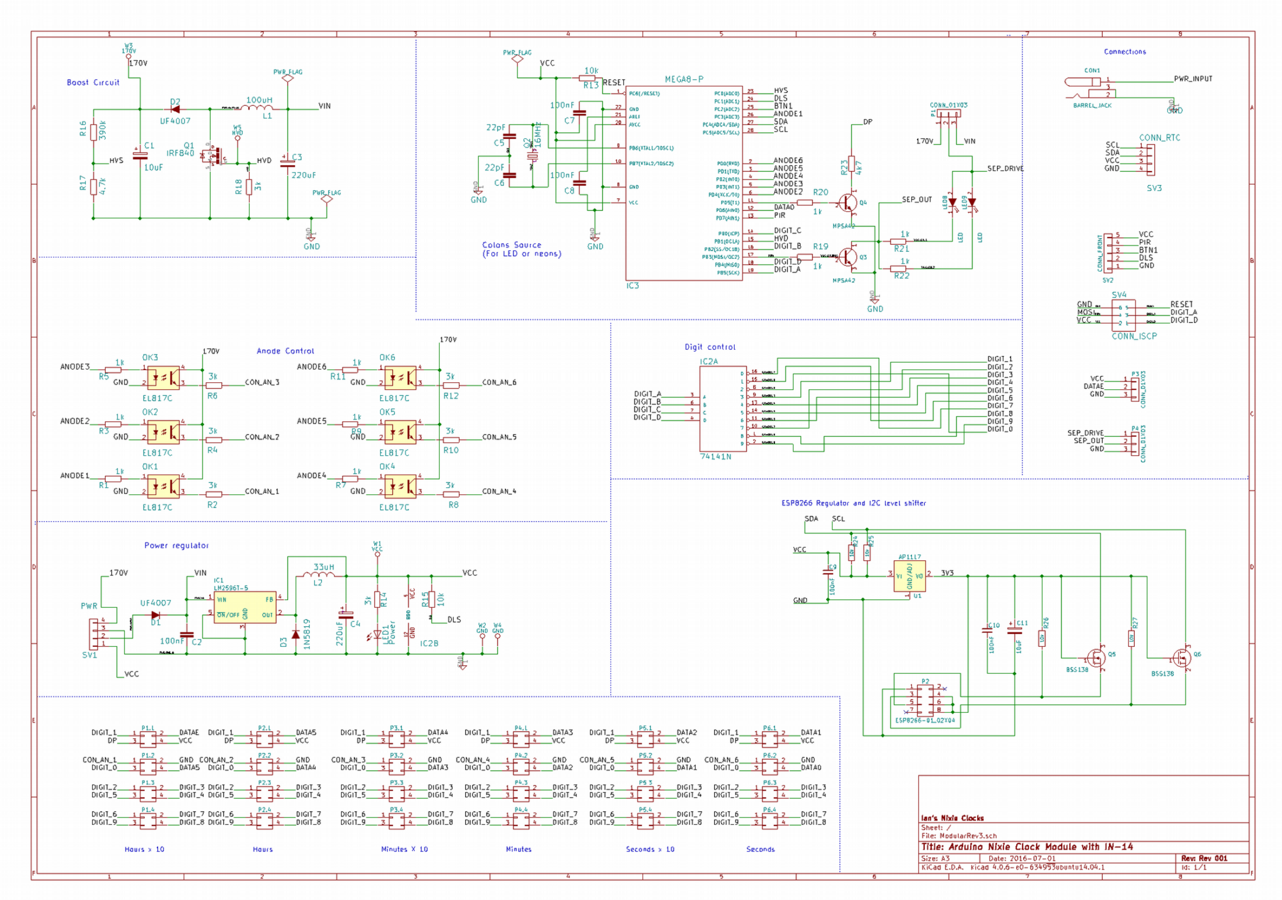
The connections are:

Connector	Description
POWER	External power can be applied to the board with this connector instead of the 5.5mm jack. Any DC input source is possible, from 7.5V – 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. The absolute maximum input voltage is 24V. Any higher voltage than this will damage the board within a few seconds! The input current ranges from 300mA to 1A depending on the size of the tubes and the number of LEDs you are driving. **170V: Output** of high voltage for driving external neons etc. **GND:** The negative side of the input supply **VIN:** The positive side of the input supply **VCC: Output** of regulated 5V, which can be used to drive auxiliary circuitry.
FRONT PANEL	These are the controls that go on the front panel: The input button and the Light Dependent Resistor to detect ambient light. **GND:** The “ground”. One lead of the button and one lead of the LDR and one lead of the button are connected to this. **LDR:** “Dimming LDR Sense”: The other lead of the LDR is connected to this **BTN:** 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 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.
I2C	Optional connection for I2C. If you want to use a traditional RTC (Real Time Clock) time provider module, connect the appropriately marked terminals on the RTC module.

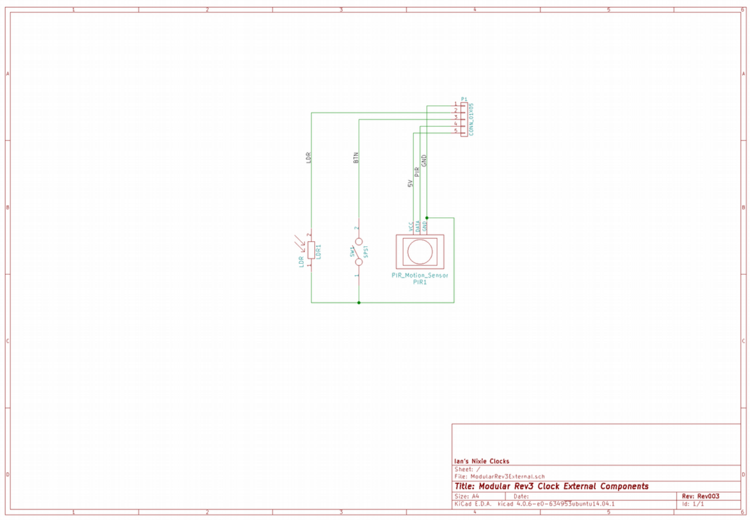
SEP SELECT	"Separator Select". This is used to select the type of separators you have between the pairs of tubes. If you want to use LEDs, connect a jumper between "VIN" and "COM". If you want to use neons, connect a jumper between "170V" and "COM". **Note:** You also have to use the appropriate resistor values for R21 and R22.
SEP AUX OUT	"Separator Auxiliary Out": This is an open collector output which you can use for driving additional separator LEDs or neons (depending on what you have selected on "SEP SELECT". You can run as many additional separator elements as you want, as long as the collector current of the MPSA42 is not exceeded (Ic maximum: 500mA). This allows you to have additional lights in your design that will work in unison with the on-board separators.
LED EXT OUT	"LED extension output". The clock is capable of driving additional WS2812B elements in addition to the 6 on-board, individually addressable tube back lights. If you want to drive additional WS2812B LEDs, you can daisy chain them on this output. Note: You will have to set the clock setting telling the clock how many WS2812B LEDs you are using. The minimum is 6, the maximum is 10.

Schematic

Below is the schematic for the clock.



And for the Front Panel components:



Construction

Preparation:

You should have a small tipped soldering iron, some thin (<= 1mm, ideally 0.6mm, especially for the SMD components) solder, and electronic side cutters.

There are a few large format Surface Mount Device (SMD) components, but this should not present any challenges to mounting by hand. Ideally, you will need some electronic tweezers, but if you don't have these, mounting by hand is not difficult.

Components:

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

Please see the separate “Component Identification” manual to help you identify individual components if you need any help.

Low Voltage Circuit:

Parts List:

L2	33uH
D1	UF4007
C2	100nF
D3	1N5819
C3	220uF
C4	220uF
IC1	LM2596
R14	3k
LED1	LED3MM
SV1	CONN_POWER
xCON1	Barrel Jack

The Low Voltage circuit is a buck switched mode voltage regulator. Its job is to reduce the external voltage from the power adapter down to a known and stable 5V to drive the micro-controller, the K155ID1 and the LEDs.

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.
- D1 and D3 should be placed so that the white stripe on the body lines up with the white stripe on the board. Don't mix up D1 and D3, they have different jobs to do!
- C3 and C4 must go the right way round. The negative side is marked with a stripe. (See hint)
- The LED must go with the right polarity. The side which has the longer lead (the anode) goes nearest the resistor R14. (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.
- L2 can be tricky to mount if it is SMD. First wet one pad and re-flow the component on one pad only. When this is done, solder the other pad, making sure that there is enough solder to connect the inductor properly.

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 centre of the board). This should measure 0.7V less than the input voltage.

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 inverted.

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

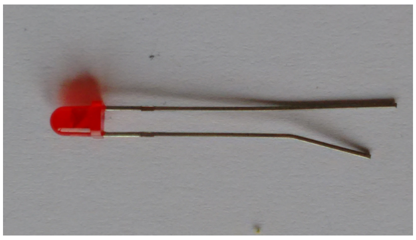
Hint: The electrolytic capacitors

> Each 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”

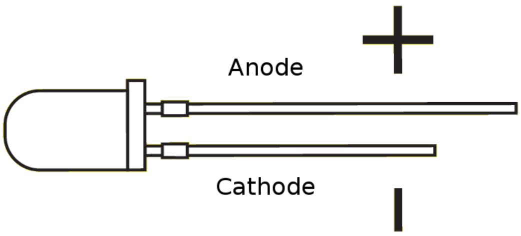
Hint: The LED orientation

> The LED has one lead longer than the other, and a flat on one side. The side with the **shorter** lead (the cathode) goes into the hole on the board nearest the diode.

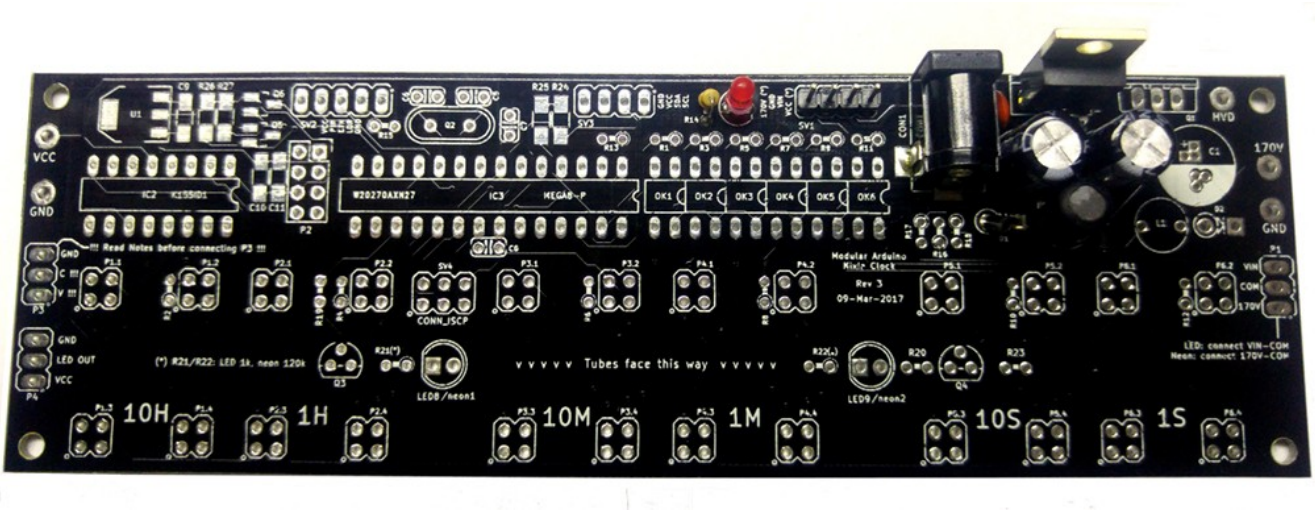


The LED

The LED should look something like this:

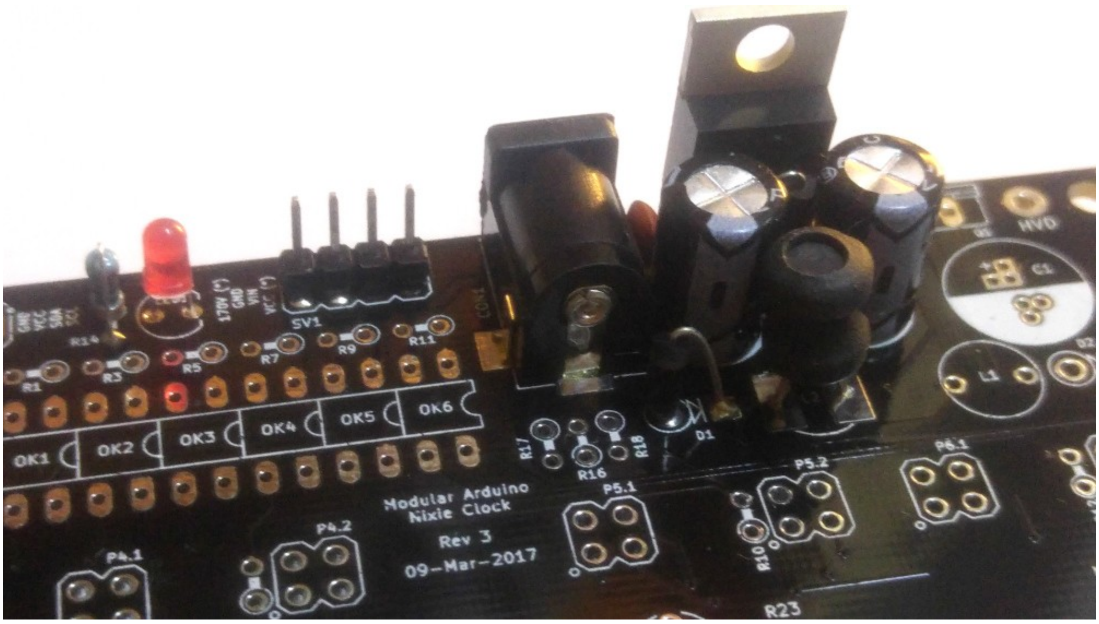


At the end of the low voltage circuit build, your board should look like this:



Low Voltage Circuit

A detail of the low voltage circuit:



Low Voltage Circuit Detail

High Voltage Circuit:

Parts List:

C5	22pF
C6	22pF
C1	2.2uF 400V
C7	100nF
C8	100nF
D2	UF4007
S28	SOCKET 28
Q2	16MHz
L1	100uH
R18	3k
Q1	IRF840
R16	390k
R17	4.7k
R13	10k
IC3	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.

For this, we have to install the controller clock circuit and the components to allow the controller to operate.

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.
- Put the micro-controller socket in first and solder it. 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 matching the silk screen on the board.

Warning!

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

Especially check that R16 and R17 are in the right positions, and that the electrolytic capacitor C1 is the right way round. If you switch R16 and R17 by mistake, you will put a high voltage into the micro-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 marking on the board.

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 “saw tooth” 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 voltage regulator nor the MOSFET get excessively hot. A gentle “crackling” sound is normal.

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).

Trouble shooting

If the high voltage stops working after a minute! This is normal, and the firmware is designed this way. We turn off the high voltage generation after 60 seconds to protect the components on the board.

Test mode is not intended for running for a long time: it runs the HV circuit with default settings and can overheat components if run for a long time!

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.

Trouble shooting

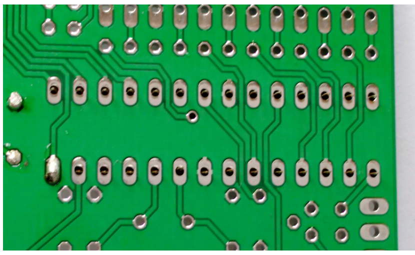
Q1 can get warm, **but should not get too hot to touch**. If it gets hot, you need to check the orientation of the components and that there are no solder bridges.

If you don't get the expected voltage reading:

- See the “Troubleshooting” section later in the document! There are lots of tips and tricks there.

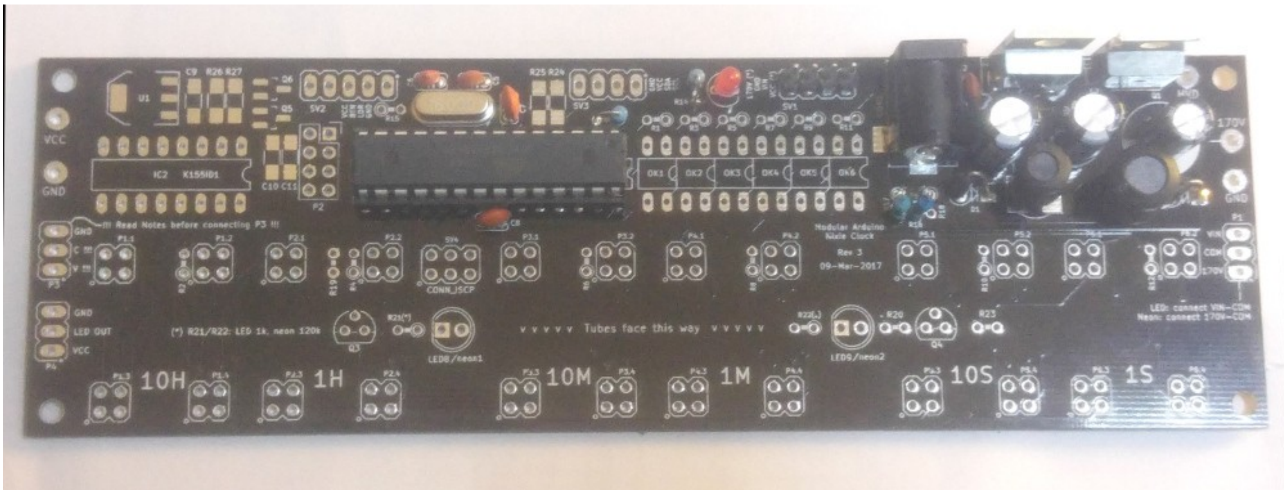
Hint: Mounting the 28 pin socket

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

Tube holder preparation: LEDs

This step can be skipped if you have bought the tubes already assembled onto tube holders! Instead, go to the section “**Tube holder preparation: Tube holder sockets on main PCB**”!

Parts List:

TH 1-6	Tube holder PCBs
LED2-7	WS2812B NeoPixels
C12-17	100nF 1206 SMD
24	2x2 Header pins
24	2x2 Header Sockets

The tube holder boards must be assembled so that they are ready for mounting onto the main board. Underneath each tube there is a WS2812B NeoPixel which needs to be mounted first, and a decoupling capacitor. We will test the LEDs in the first step, and then later we will mount the tubes.

The SMD soldering of C12-C17 and LED2-7 is quite delicate. You should use some tweezers to hold the components, and thin (<=0.6mm) solder to make life easier. It is possible to mount these components with standard tools, but it is harder.

For each of the components, put a small blob of solder on ONE pad. Then roughly place the component, and re-heat the soldered pad, sliding the component into place from the side. You may need to adjust the position or orientation after roughly mounting the component. If you need to adjust the component, re-heat the pad you soldered and adjust the component.

Only after it is correctly in place and you are happy with the component placing, solder other pads.

Warning!

Don’t solder the other pads until you are happy with the way the SMD component is installed.

Once you have soldered other pads, it becomes very hard to adjust the position of the component, and you risk damaging it through overheating if you have more than one pad soldered.

Warning!

Try not the “dwell” too long on a single pin or pad. The WS2812 devices are fragile and are sensitive to heat.

Make sure you use fine solder and a fine tipped soldering iron!

Test Step

Check each tube holder as you build it. The tube holder board should cycle through colours, the pattern repeating every 10 seconds.

Repeat this for the other 5 tube holder boards, so that you have 6 tube holder boards with the capacitor and WS2812B on them.

Warning!

Do not start mounting the tubes until you have built and tested the LEDs on the tube holder boards.

It is very hard to correct a problem on the tube holder board after you have mounted the tube.

Trouble shooting

If one or more of the LEDs does not light up correctly:

The LEDs are daisy-chained and are driven from the right hand side of the board (seconds) and are driven towards the left (hours). If one or more of the LEDs does not light up, check the right hand most LED with the problem, because it is most likely this LED that has the issue.

Tube holder preparation: Soldering the header pins

This step can be skipped if you have bought the tubes already assembled onto tube holders! Instead, go to the section "**Tube holder preparation: Tube holder sockets on main PCB**"!

Once you have the LEDs and capacitors on each tube holder board, you can solder the header pins onto the tube holder boards.

Place 4 header pins in the PCB in the 1S tube location with the long pin facing down through the main PCB (you don't need to solder them yet) so that they will support the tube holder board. You need to raise the main PCB up off your work surface so that the header pins go completely flat into the main PCB, and only the short pin of the header is left poking up.

Now place the tube holder board on the top of the 4 header pins. Solder 1 pin of each header to the tube holder board. Make sure that the headers and pins are straight, and that the tube holder board is straight as well, and then solder the rest of the pins. The tube holder board should easily come out of the socket on the main PCB, if it doesn't, it means that you have some of the pins not straight on the tube holder board.

Now separate the tube holder board from the main PCB, and prepare the other 5 tube holder boards in the same way.

Tube holder preparation: Tube holder sockets on main PCB

If you bought the tube holders ready built, you will still need to install the tube holder sockets onto the main PCB.

Parts List:

24	2x2 Header Sockets

Put the 4 2x2 header sockets on the pins of each tube holder board so that they are held straight and fixed while you solder them. For best results, note that the sockets are not identical on all four sides. Two of the sides are slightly "open", and two are "closed", meaning they reach right down to the PCB. Put a "closed" side to the front. There is no electrical difference, but it looks better!

Once you have put all 4 sockets on a tube holder, use the tube holder as a guide to mount the 4 2x2 sockets onto the main PCB correctly. Mounting them like this means that they all end up straight, and you won't have to force the tube holders in because the sockets are misaligned.

When you have finished the PCB should look like this:

Tube holder sockets mounted

Tube holder preparation: Testing the tube holder LEDs

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

It is important to test the tube holder boards to make sure that the LEDs all work correctly. It is almost impossible to adjust them once the tubes are mounted, so do not skip this step!

Place the tube holders in the sockets on the main PCB, making sure that all the tube holders are facing the right way. (There is a clear marking on the boards).

Once the tube holders are mounted, power the clock on.

Test Step

Once the six tube holder boards are mounted in their sockets, and you have double checked the orientation of the boards, hook up the power, and check that the back light LEDs all cycle through their test routine.

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

Tube holder preparation: Tubes

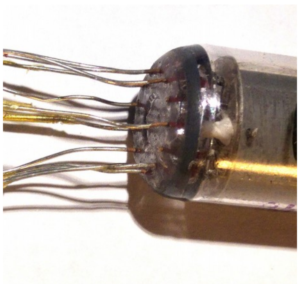
This step can be skipped if you have bought the tubes already assembled onto tube holders! Instead, go to the section “**Anode Control Circuit**”!

Parts List:

IN-14	IN-14 tubes

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

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



Often used tubes have the two decimal point leads removed. You will not be able to use the decimal points if you have tubes without the leads for the decimal points. It might be possible to add them, but it is difficult, and depends on how short they have been trimmed on your tubes.

The lead in the middle with the white coating inside the glass is the **anode**.

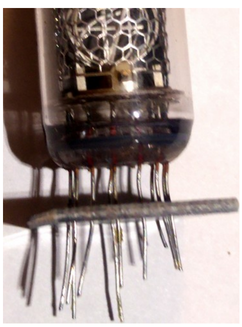
In order to mount the tube on the tube holder PCB, one useful trick is the trim the leads in a spiral, starting from the anode. This will make the leads easier to insert into the holes later. You can thread the leads into the holes one at a time.

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

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

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

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



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

Repeat the same process for all the other tubes.

When you have finished, put the tubes back in the sockets on the main PCB.

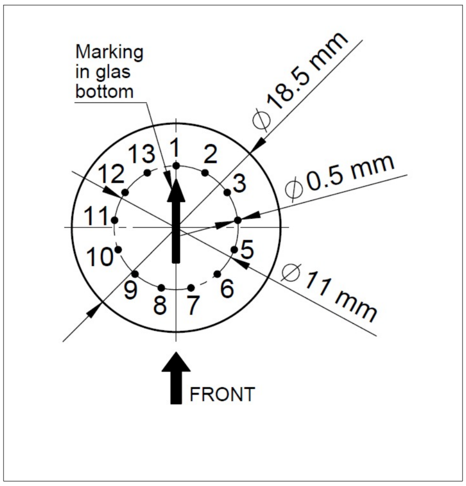
Test Step

Once the six tube are mounted on their boards, and you have double checked the orientation of the boards, hook up the power, and check that the tubes cycle through their test routine.

The digits “0” through “9” should be shown on each tube.

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



- It is best to carefully spread the legs of the tube out. If you have long enough leads, a trick is to trim the leads to different lengths so they get shorter by 0.5mm as you go round the tube. This means that pin 3 is shorter than pin 1 by 0.5mm. Pin 4 is shorter than pin3 by 0.5mm and so on. Pin12 is shorter than pin 3 by 5mm at the end. This means that you can thread the leads into the holes more easily.
- Place the tube base against the top of the LED, without leaving a gap, through the hole in the tube holder board.
- To make the tube stand upright, solder just the first three leads at the beginning one at a time (e.g. pin 1, pin6 and pin 10). You can then easily align the tube so that it is perfectly upright by reheating only one of the pins. Once the tubes are upright and aligned, you can solder the remaining pins.
- For last resort, final, small adjustments, 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.

Anode Control Circuit:

Parts List:

S24	SOCKET 24
OK1	EL817
OK2	EL817
OK3	EL817
OK4	EL817
OK5	EL817
OK6	EL817
R1	1k
R3	1k
R5	1k
R7	1k
R9	1k
R11	1k
R2	3k
R4	3k
R6	3k
R8	3k
R10	3k
R12	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.

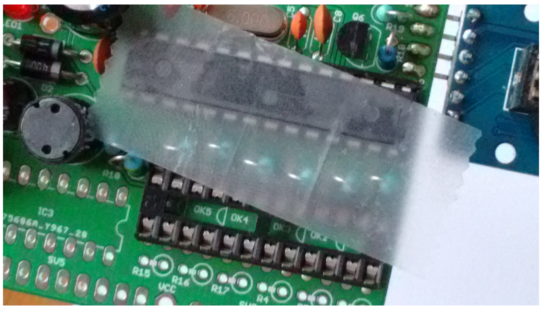
Notes:

- 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 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

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.

Separator LEDs/neons and Decimal Point driver:

Parts List:

R19	1k
R20	1k
R21	1k or 120k (see note)
R22	1k or 120k (see note)
R23	4.7k
Q3	MPSA42
Q4	MPSA42
LED8/NEON1	LED 5mm or neon (see note)
LED9/NEON2	LED 5mm (see note)
Neon1	Neon indicator lamp
Neon2	Neon indicator lamp
P1	Connect COM to 170V or VIN

To install the separator LEDs or neons, you need to have installed the tubes so that you are able to see the height that the separators need to be.

You might have to extend the leads of the LEDs or neons to make them the right height. Use some plastic sleeving or heat shrink to make sure the leads are not able to short out.

If you are installing LEDs, make sure that you put them in the right way round, following the markings of the flat side on the PCB.

If you are using neons, the orientation does not matter.

If you are using neons, you will need to connect “170V” and “COM” on P1. If you are using LEDs, you will have to connect “VIN” and “COM”.

The decimal point driver channels the high voltage to the right pin on the tubes.

Front Panel components

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

SV2	5 pin connector
LDR	LDR
S1	SWITCH
R15	10k
PIR	PIR detector (optional)

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 you 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.

“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
Q8	BSS138
C9	100nF SMD
C10	100nF SMD
C11	10uF SMD
U1	AP1117-3.3
P1	2x4 Female Header Socket

Install one or other of these options (either RTC or WiFi). If the clock detects an RTC is installed, it will disable the WiFi interface, and all advantages of the WiFi module will be lost.

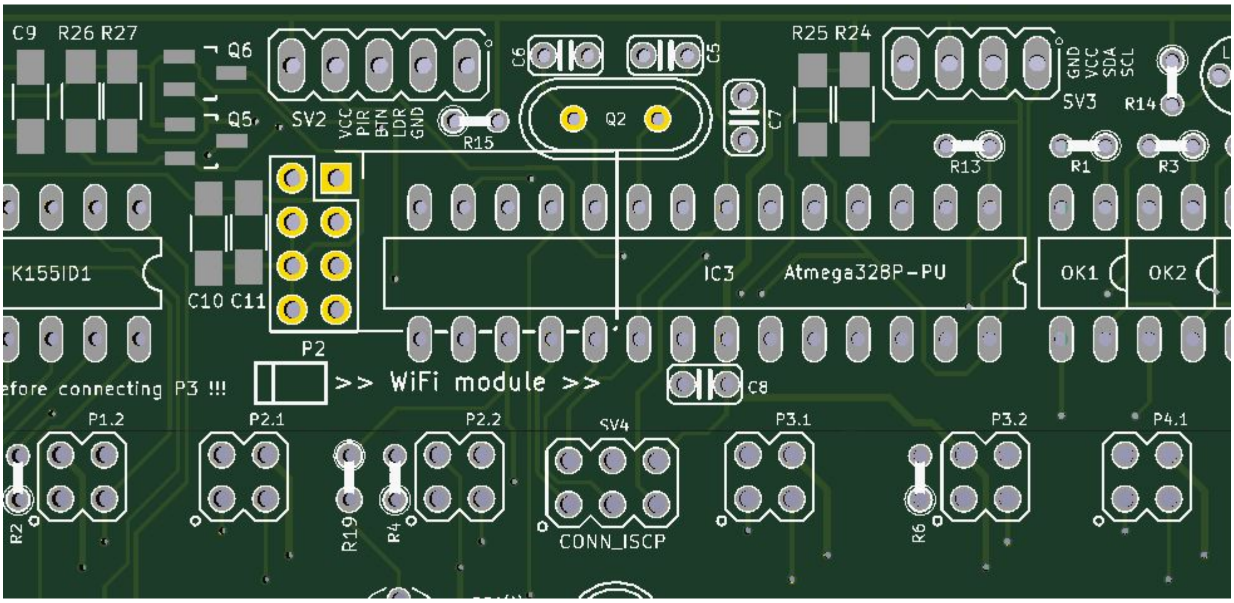
The usual time source for this board is an on-board ESP8266 WiFi module, unless you specifically want to use the battery backed RTC (Real Time Clock) module.

Notes:

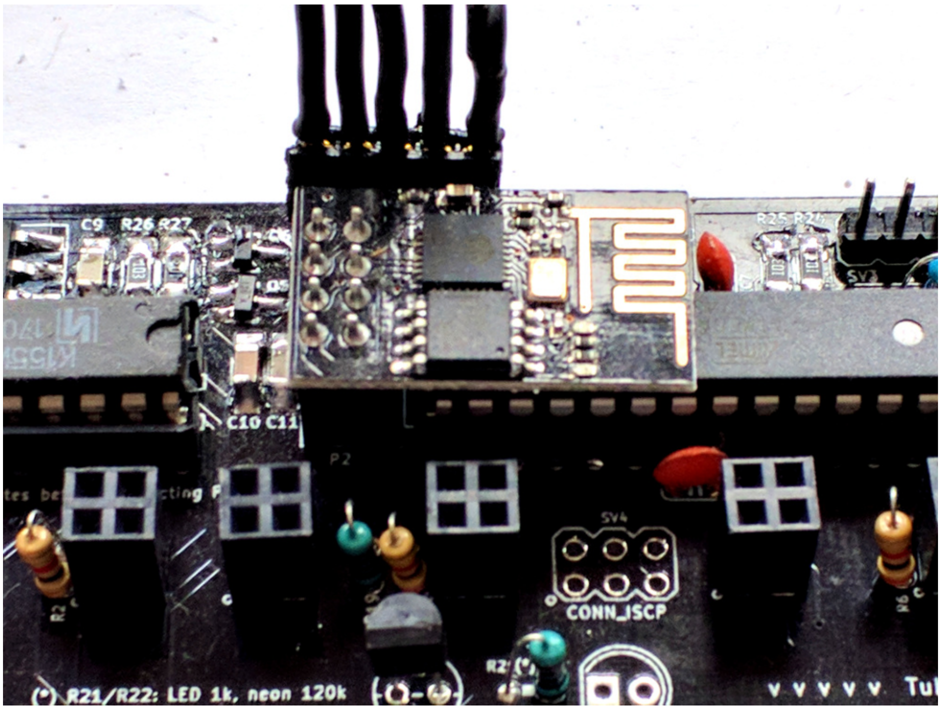
- The ESP8266 ESP-01 module needs to be installed with the long side over the micro-controller. (See picture). If you fit it so it faces over the K155ID1, you may damage the module.
- The square solder pad of P2 is pin 1. Compare this to the pin out of the ESP-01
- In general, hand soldering SMD components with a soldering iron is a bit tricky. A tip is to solder only a single pad of each component, position it with the single pad, and ONLY once it is in position, solder the remaining pads.
- You really need some small electronic tweezers to position the components. If you don’t have tweezers, a wooden toothpick or skewer can be used to hold the components in place. The hints in the rest of this section assume that you have some tweezers.
- If you install the RTC, the WiFi circuit will be disabled. You can run the clock with an RTC module simply by connecting the RTC to SV4. In this case, there is no reason to install the WiFi circuitry.
- Soldering the SMD 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 of the three pins and then re-flow

the solder while positioning the component. Once the pin is fixed, you can solder the other pads.

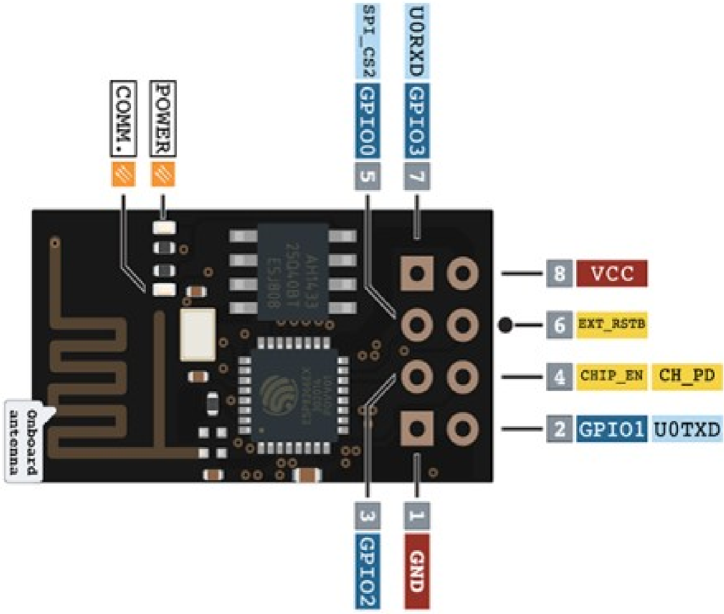
- Soldering the transistors: place a tiny solder blob on the center pin and then re-flow the solder while holding the component in place. Once the pin is fixed, you can solder the other pads.



The ESP-01 module goes over the controller



The ESP-01 module goes over the controller



ESP-01 pin out

The clock needs to know the time. To do this, an RTC or WiFi module is supplied with the kit, (depending on the option you chose). You can mount these modules 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.

Warning!

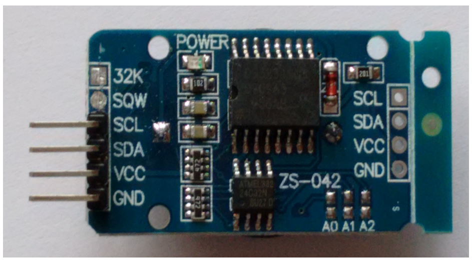
Double check the orientation/location of the WiFi module before you proceed!

If you have installed the WiFi module, make sure that it is over the controller, not over the K155ID1! If you install it the wrong way round, you will destroy it.

- 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



RTC Module with pins removed

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.

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

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

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

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.

shooting

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!

Parts list / BOM

#	Qty	Value
1	2	22pF
2a	2	LED 5mm
2b	2	NEON
3	1	2.2uF 400V
4	2	220uF 35V
5	6	WS2812B
6	1	16MHz
7	2	UF4007
8	1	1N5819
9	1	74141N/K155ID1
10	1	4.7k
11	1	390k
12	8	3k
13	25	1k
14a	2	1k for LEDs
14b	2	120k for neons
15	2	10k
16	1	MEGA8-P 328P-PU
17	6	EL817
18	2	4 pin header 0.1” pitch
19	1	5 pin header 0.1” pitch
20	2	MPSA42
21	3	100nF
22	1	LED 3mm
23	1	100uH
24	1	33uH
25	1	LM2596
26	1	IRF840
27	6	IN-14
28	1	DS3231
29	1	WiFi Provider
30	1	Barrel Jack
31	24	2x2 Female Header
31	24	2x2 Male Header

Revisions:

Rev0001: 23Jan2017: First version, derived from Rev2
Rev0002: 30Oct2017: Update BoM
Rev0003: 19Nov2017: Corrections from Derek
Rev0004: 28Nov2017: Minor corrections to build order
Rev0005: 26Apr2018: Correct labeling on the Separator circuit “HV” → “170V”