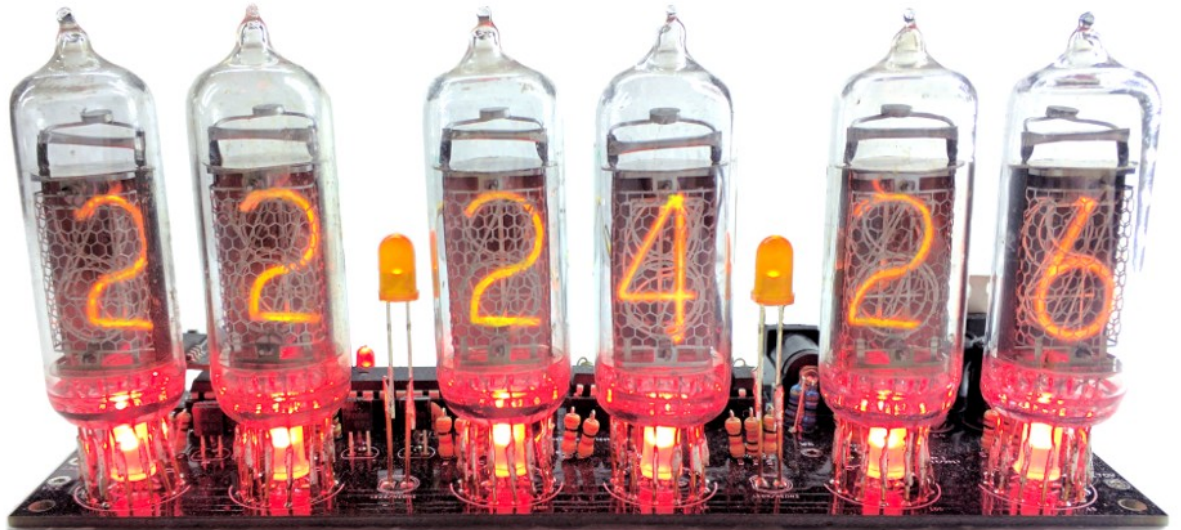


Arduino Nixie Clock **“All-In-One” Clock Rev5**

Construction Manual



Index

Table of Contents

Index.....	2
Contact Information.....	3
Software.....	3
Troubleshooting.....	3
Safety.....	3
Powering Up.....	3
Component Identification.....	4
Board Identification.....	4
Board layout.....	5
Board Connections.....	7
Schematic.....	8
Construction.....	10
Preparation:.....	10
Components:.....	10
Low Voltage Circuit:.....	10
High Voltage Circuit:.....	13
Back light LEDs.....	18
Anode Control Circuit:.....	21
Cathode Control Circuit:.....	23
Mounting the tubes.....	24
Separator LEDs/Neons.....	26
Front Panel components.....	27
WiFi module.....	28
RTC module.....	29
Troubleshooting.....	30
Programming the micro-controller.....	33
Programming with an Arduino Uno.....	33
Parts list / BOM.....	34

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 open source and is available from the location on the Manuals page:

<https://www.nixieclock.biz/Manuals.html#all-in-one>

Which leads you to the latest firmware locations, for both the clock and the WiFi module.

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.

Important: When in test mode, the HV generation will only run for 60 seconds! This is designed to avoid damage to components and avoid accidental shock, as there is no other way to see the clock is "live"!

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 Identification

There are several different versions of the board. Currently these are the versions this manual supports:

- All-In-One for IN-14 tubes
- All-In-One for IN-14 tubes “flipped”
- All-In-One for IN-8-2 tubes
- All-In-One for IN-8-2 tubes “flipped”
- All-In-One for Z570M/Z573M/Z574M tubes

The construction for each of these board is identical, apart from the final mounting of the tubes.



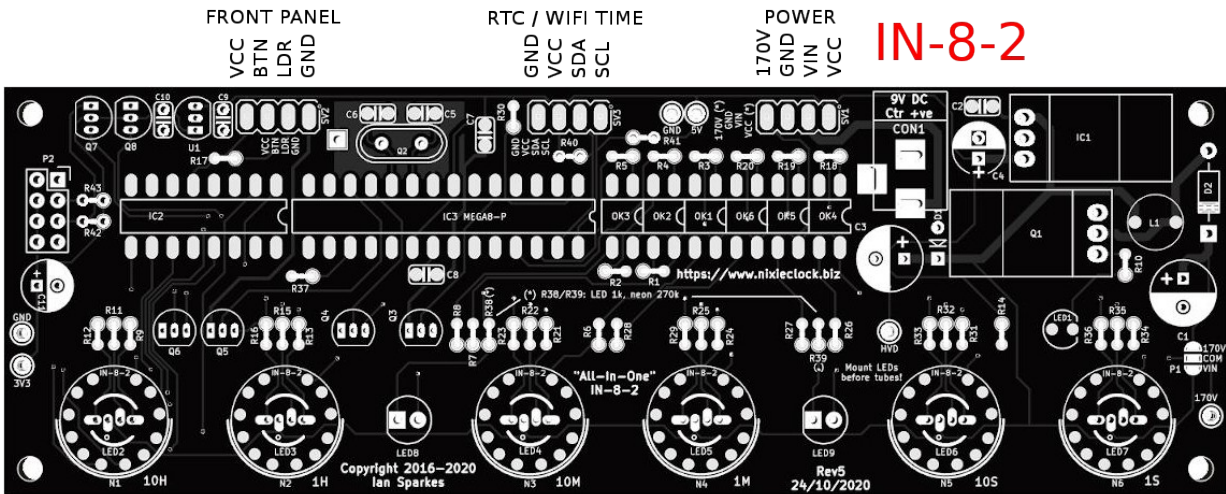
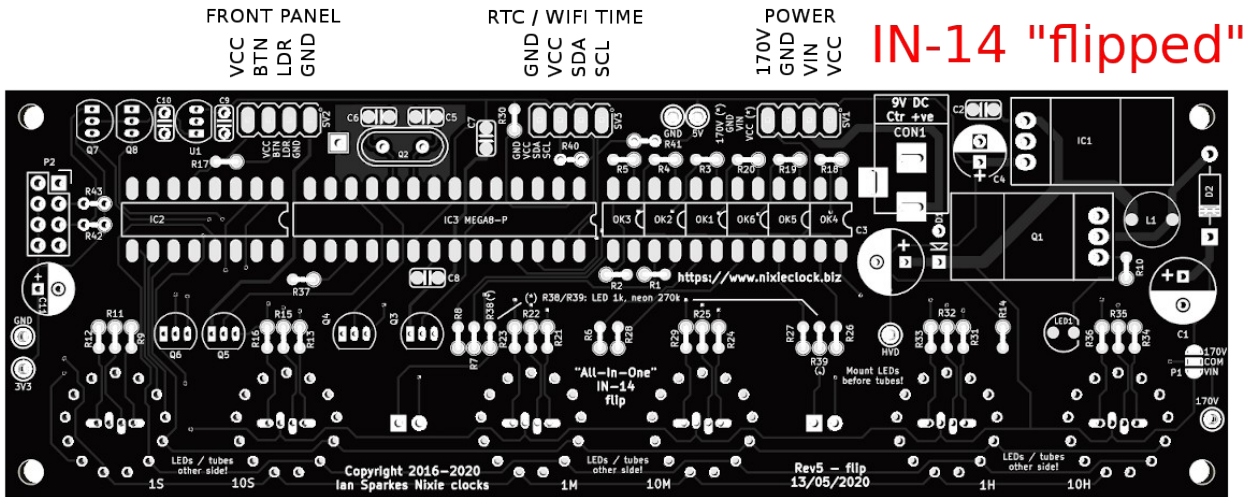
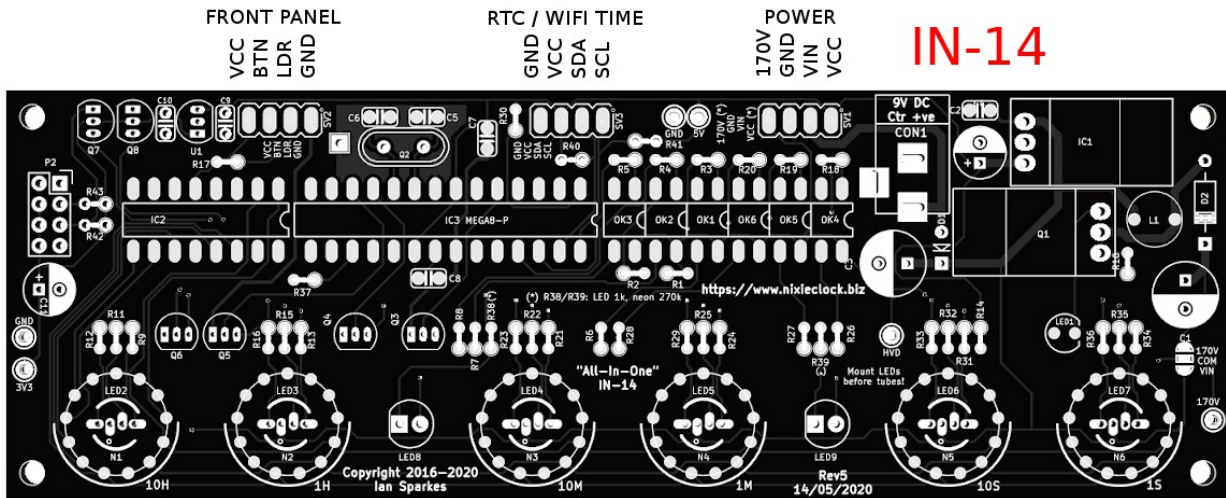
Warning!

Please follow the instructions on the board silk screen when mounting the tubes!

The silk screen tells you the side of the board to mount the tubes on, please check the silk screen carefully before starting to mount the tubes.

Board layout

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



FRONT PANEL

VCC
BTN
LDR
GND

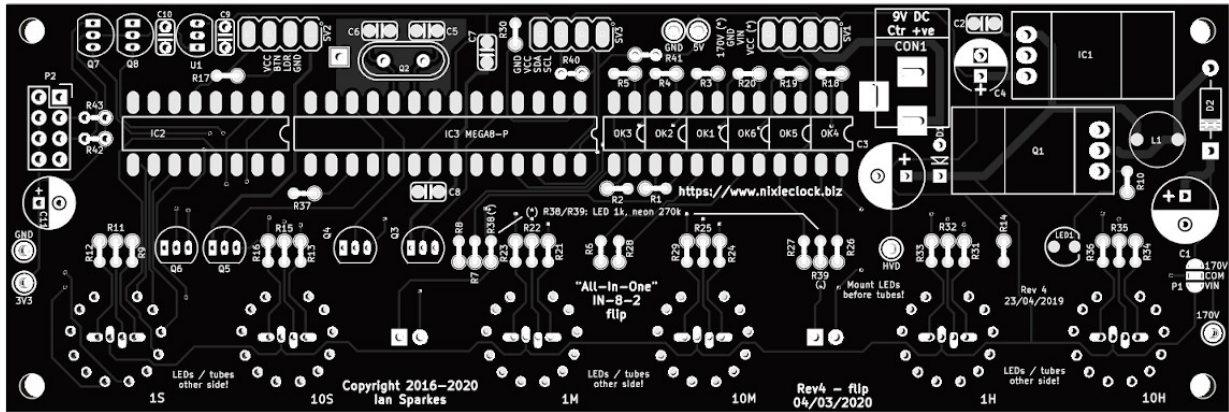
RTC / WIFI TIME

GND
VCC
SDA
SCL

POWER

170V
GND
VIN
VCC

IN-8-2 "flipped"



FRONT PANEL

VCC
BTN
LDR
GND

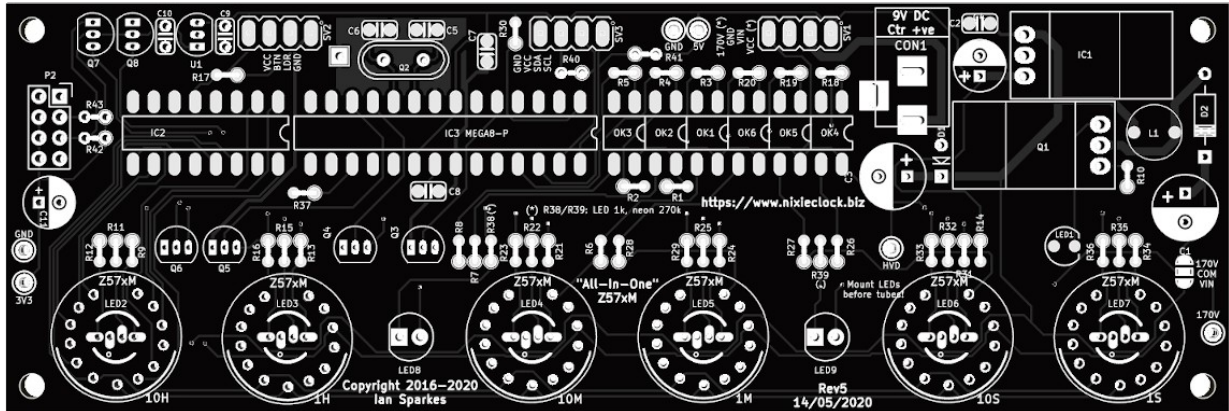
RTC / WIFI TIME

GND
VCC
SDA
SCL

POWER

170V
GND
VIN
VCC

Z57xM

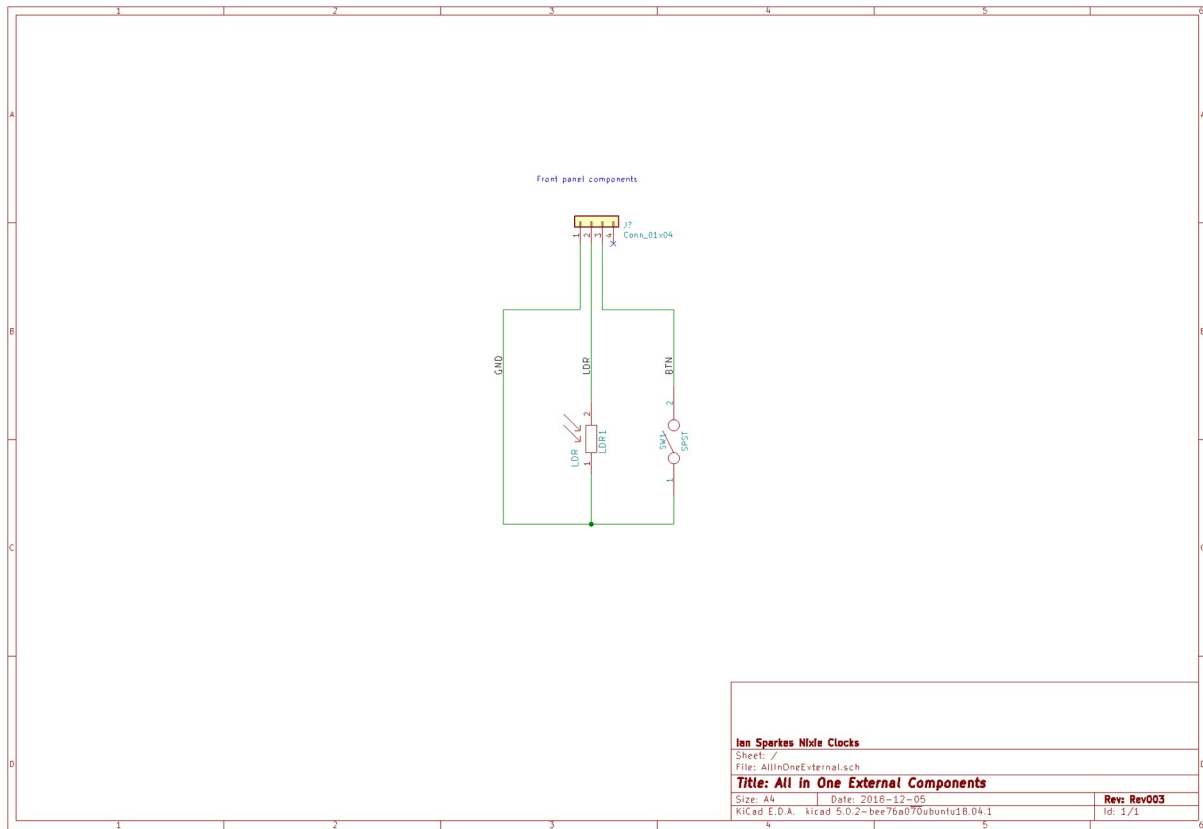


Board Connections

The connections are:

Connector	Description
POWER	<p>External power should be applied to the board with this connector. 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 MOSFET and voltage regulator.</p> <p>The absolute maximum input voltage is 24V. Any higher voltage than this will damage the board within a few seconds!</p> <p>The input VIN is protected against the input being connected reversed.</p> <p>The input current ranges from 300mA to 1A depending on the size of the tubes and the number of LEDs you are driving.</p> <p>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.</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>

And for the LDR and button:



Note that the “Front Panel” are mounted on the enclosure, not on the main board. They should be connected with some flying signal leads.

Construction

Preparation:

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

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 appendix to help you identify individual components.

Low Voltage Circuit:

Parts List:

D1	UF4007
C2	100nF
C3	220uF 35V
C4	220uF 10V
IC1	7805
R14	3k
LED1	LED3MM
SV1	CONN_POWER
CON1	BARREL JACK

The Low Voltage circuit is a very traditional voltage regulator using a linear regulator. 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. For this kit, there are detailed video construction manuals, which go through the process step by step. Please use these as a resource if you need.



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

YouTube: https://youtu.be/QY_jAK5_2jM

BitChute: <https://www.bitchute.com/video/zBpbDLuvPzT9/>



Warning!

Pictures!

For pictures of this step of the build, please see the next section.

Notes:

- See the section on “Component Identification” document for help with identifying the components. This is available from the Manuals page at the web site.
- D1 should be placed so that the white stripe on the body lines up with the white stripe on the board.
- 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 shorter lead goes into the square hole. (See hint)
- Put IC1 so that the metal tab is closest to the board when laid down. Pre-form the leads before mounting and soldering it. You have to bend the leads quite close to the body.
- The direction of the resistor does not matter. The small circle on the PCB is a suggestion where the body side of the component should go.



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 7805 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 7805 does not heat up. It should stay almost cold.

Hint: The 220 uF capacitor

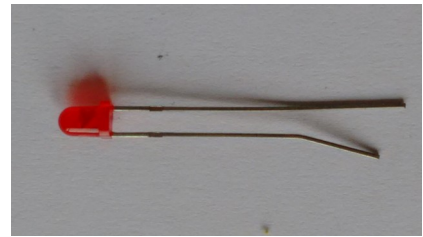
The electrolytic capacitor has a stripe on it to denote the **negative** side of the capacitor. The positive side of the capacitor (which goes into the “+” on the board) is the other one!



220uF capacitor “stripe”

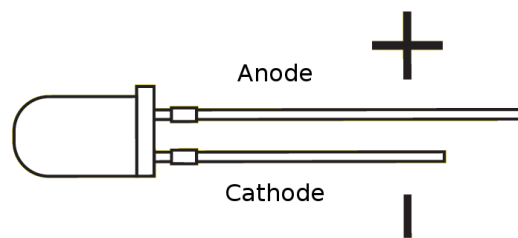
Hint: The LED orientation

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 square hole.



The LED

The LED should look something like this:



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
R10	3k
Q1	IRF640
R1	390k
R2	4.7k
R30	10k
IC3	MEGA8-P

The high voltage circuit uses the micro-controller to drive the boost circuit with a high frequency square 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.



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

YouTube: <https://youtu.be/rpw1XZ3eRtk>

BitChute: <https://www.bitchute.com/video/U1A0CqhilLpg/>

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.



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 170V 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.



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.



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.



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).



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

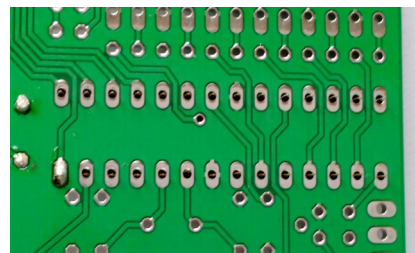
Q1 can get warm, **but should not get too hot to touch**. If it gets hot, you need to check the orientation of the components and that there are no solder bridges.

If you don't get the expected voltage reading:

- 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

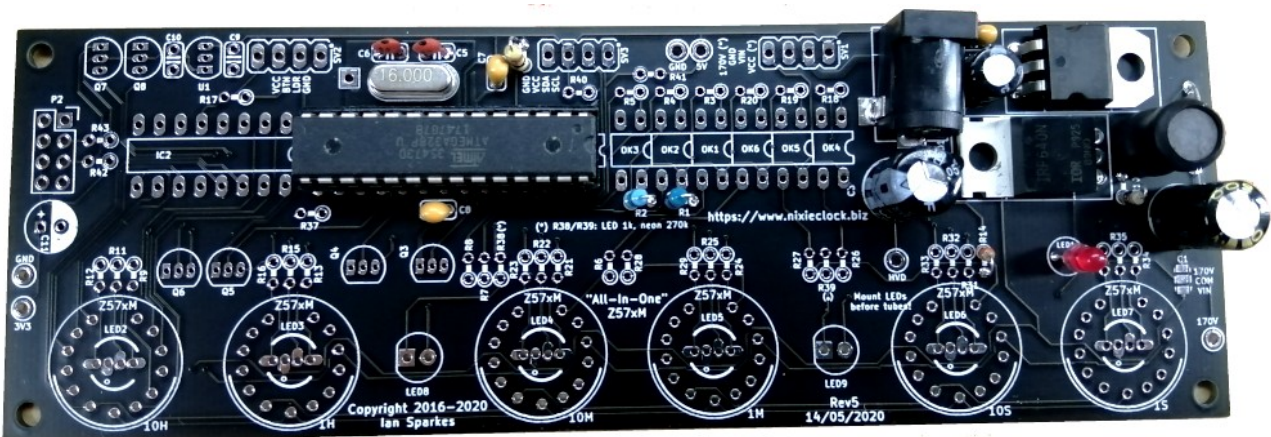
Hint: Mounting the 28 pin socket

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

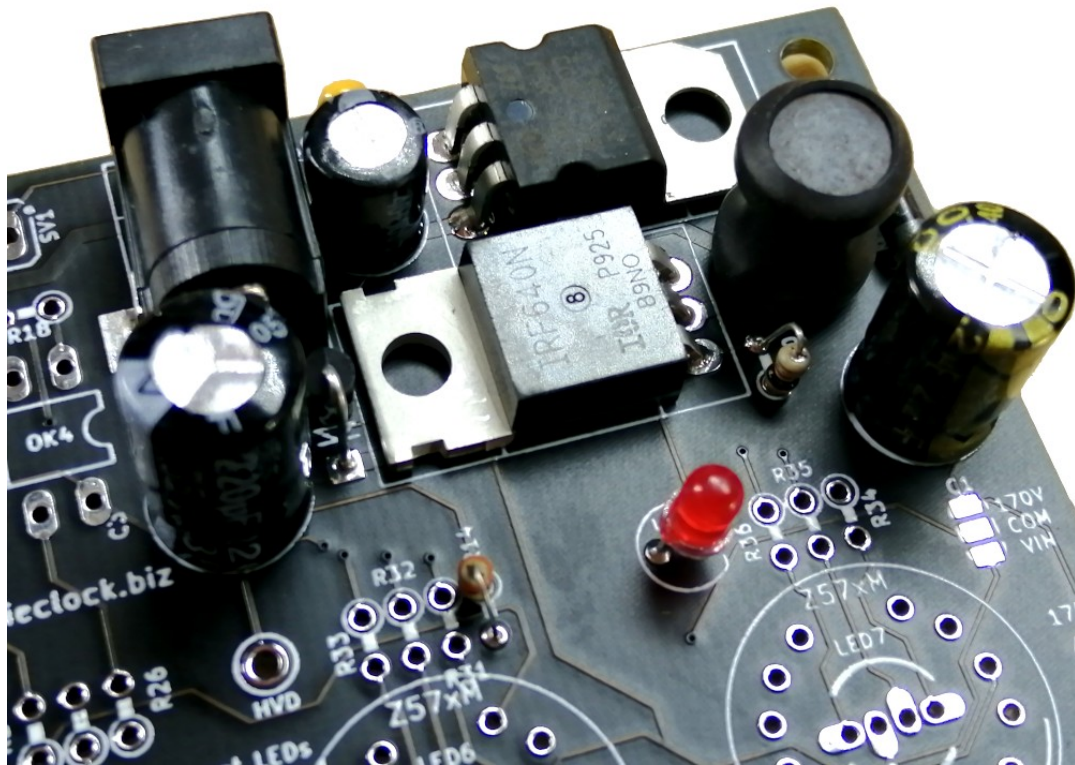
At the end of the high voltage circuit build, your board should look like this:



High Voltage Circuit

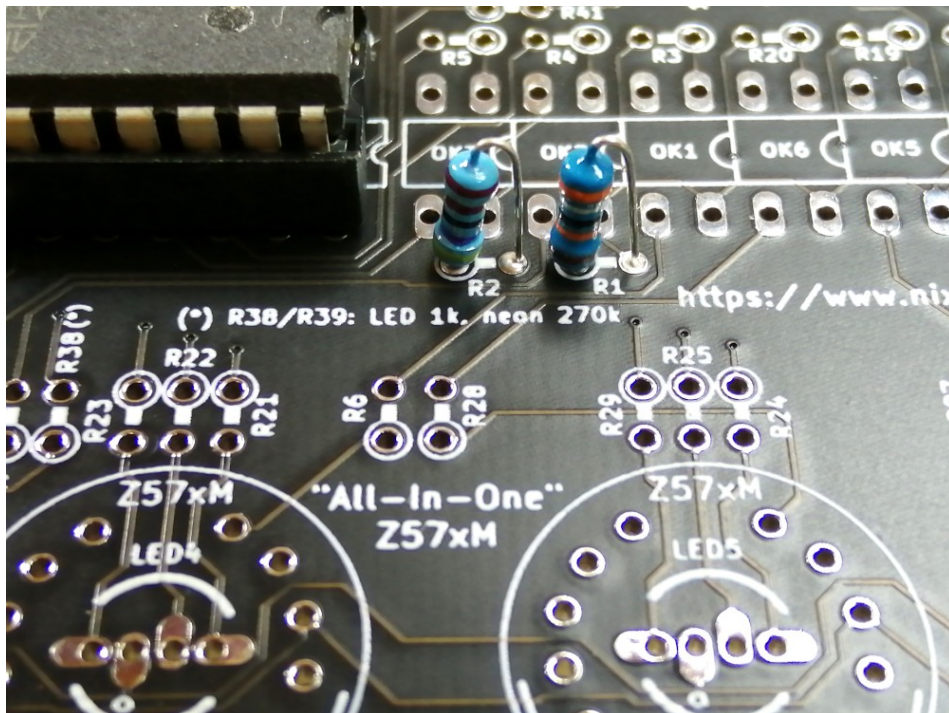
Close up of the regulator and MOSFET

High



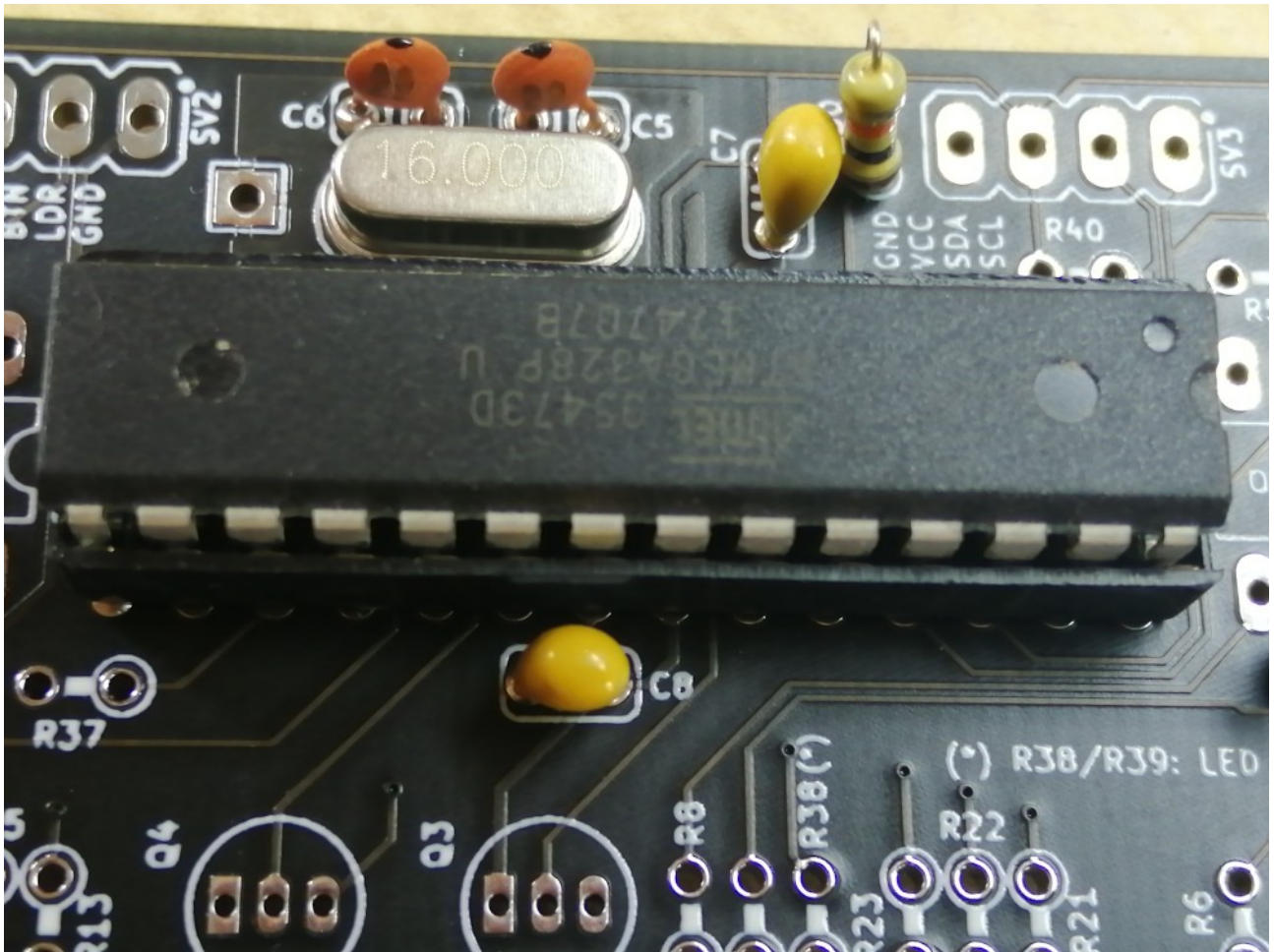
Voltage Circuit - Regulator

Close up of R1 and R2



High Voltage Circuit - R1 and R2

Close up of the controller:



High Voltage Circuit - Controller

Back light LEDs

Parts List:

R37	1k
Q3,Q4,Q5	2N7000
Q6	MPSA42
R17	10k
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.



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

YouTube: <https://youtu.be/16w6aWrF9oQ>

BitChute: <https://www.bitchute.com/video/NfANtZnCnz1e/>



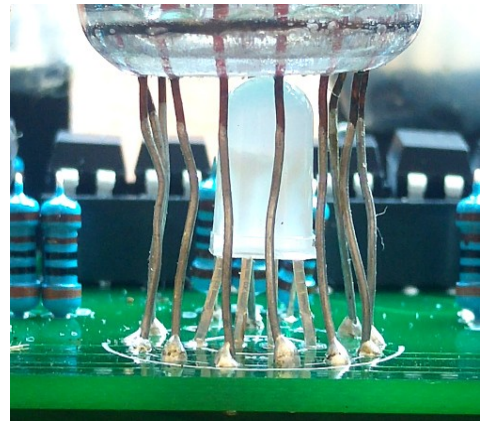
Trouble
shooting

If any of the channels does not go out completely:

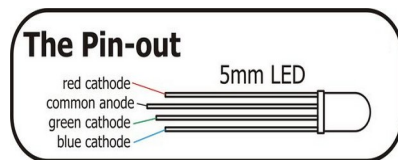
Sometimes the LEDs don't go out completely (especially the Blue channel, because the blue LED is very sensitive). If this happens, carefully clean the flux residue from around Q3-6 (of course with the power off) using a cotton bud and pure alcohol. The FETs are very sensitive to being partially turned on due to tracking over the flux. Let the board dry completely before re-testing.

Notes:

- 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 the solder jumper contacts “VIN” and “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 270k values for R38 and R39. The polarity of the neons does not matter.
- Q3 - Q6 should be orientated with the flat side as shown on the board.
- 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 “colon” LEDs need to be mounted proud of the board, on moderately rigid extension wires. It is not easy to estimate the height they should be mounted at yet (it will become clear once you mount the tubes), so for the moment, don't try to mount the separators.



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



- 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.

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.



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

YouTube: <https://youtu.be/7130LfYyFfl>

BitChute: <https://www.bitchute.com/video/k8kF2U2OQuwv/>

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

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.

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.



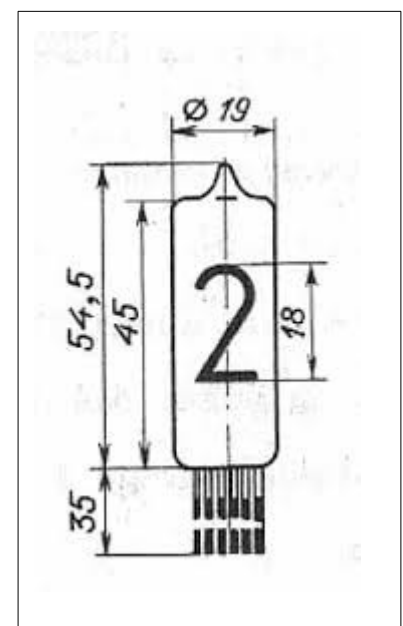
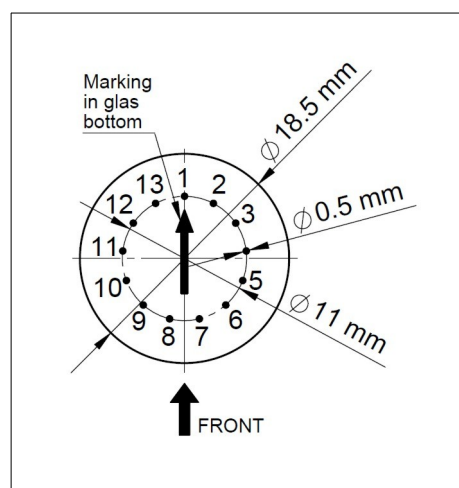
Warning!

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.

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.

Separator LEDs/Neons

Parts List:

R38	1k or 270k (see note)
R39	1k or 270k (see note)
LED8	LED 5mm (see note)
LED9	LED 5mm (see note)
Neon1	Neon indicator lamp(see note)
Neon2	Neon indicator lamp
P1	Connect COM to HV or VCC

Now that we have the tubes mounted, we can mount the separators. We do this after the tubes are mounted because now we can judge the height that the separators should be.

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 270k 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.
- Don't use a pluggable “jumper” header for setting P1: Solder a wire between the pins you want to jump directly. If you use a header, someone later will want to change it.



Trouble shooting

The separators do not light up.

Make sure that you set the jumper P1 according to the separators you are using. Set to “VIN” for LEDs, or “170V” for neons.

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
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.

WiFi module

Parts List:

R40	10k
R41	10k
R42	10k
R43	10k
C9	100nF
C10	100nF
C11	220uF 10V
Q7	2N7000
Q8	2N7000
U1	78L33
P2	2x4 Female Header
WiFi	Preprogrammed ESP-8266

If you have the WiFi module, you need to install the level shifter components on the board. This provides an interface between the 5V Atmel controller and the 3V3 ESP circuitry.

Notes:

- Be careful of the orientation of the polarity of the components Q7, Q8, U1 and C11
- The ESP 8266 must be mounted so that the body of the board sits over the K155ID1, **NOT** pointing outwards



Warning!

The ESP 8266 must point inwards to the board. It must not point outwards over the edge of the board.

If you mount the ESP 8266 incorrectly, you will destroy it! The ESP 8266 does not have any key to make sure it cannot be mounted incorrectly, so you have to take care.

RTC module

Parts List:

RTC	RTC
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.

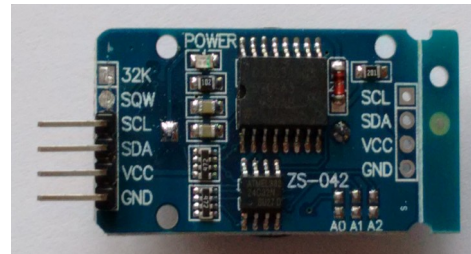
The WiFi module has it's own instruction manual. Please refer to that if you have the WiFi option.

- 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

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.



Trouble
shooting

The clock sometimes just stops showing the digits.

In especially noisy environments with clocks that have no grounding, external interference can cause the controller to lock up. In this case, please either put the clock in a grounded enclosure or select a higher quality power supply. (Poor quality power supplies don't provide any path to ground).

In extreme cases, there is a grounding point close to the crystal and you can ground the clock circuit.

Programming the micro-controller

The micro-controller comes preprogrammed. You don't **need** to program it, but you might want to, especially if you want to change the code.

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!

One last thing: The controller used in the IN-14 All-In-One clock Rev is completely compatible with the Classic Nixie Clock module Rev2, Rev3 and Rev4. They can be interchanged without danger.

:)

Parts list / BOM

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

#	ID	Qty	Value
1	C5,C6	2	22pF
2	LED9,LED8	2	LED 5mm
3	C1	1	2.2uF
4	C3,C4	2	220uF
5	LED2,LED7,LED6,LED5,LED4,LED3	6	LED RAGB (common anode)
6	Q2	1	16MHz
7	D1,D2	2	UF4007
8	IC2	1	74141N
9	R2	1	4.7k
10	R1	1	390k
11	R14,R7,R8,R28,R27,R26,R6,R10	8	3k
12	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	27	1k
13	R38,R39	2	1k for LEDs, 270k for neons
14	R30,R17	2	10k
15	IC3	1	MEGA8-P 328P
16	OK1,OK2,OK3,OK4,OK5,OK6	6	EL817
17	SV1,SV2,SV3	3	4 pin header
18	Q3,Q4,Q5	3	2N7000
19	Q6	1	MPSA42
20	C7,C2,C8	3	100nF
21	LED1	1	LED 3mm
22	L1	1	100uH
23	IC1	1	7805
24	Q1	1	IRF640
25	N2,N5,N4,N3,N6,N1	6	IN-14
26	CON_1	1	Barrel Jack

Parts needed for the optional WiFi module:

#	ID	Qty	Value
1	R40, 41, 42, 43	4	10k
2	C9, C10	2	100nF
3	C11	1	220uF 10V
4	Q7, Q8	2	2N7000
5	U1	1	AP1117-3V3
6	P2	1	2x4 Female Header

Revisions:

V0001: 12Jan2018: New manual for Revision 3

V0002: 05Dec2018: Correct External Components drawing, thanks Kent!

V0003: 14May2020: Correct Wifi BoM, and separators section.