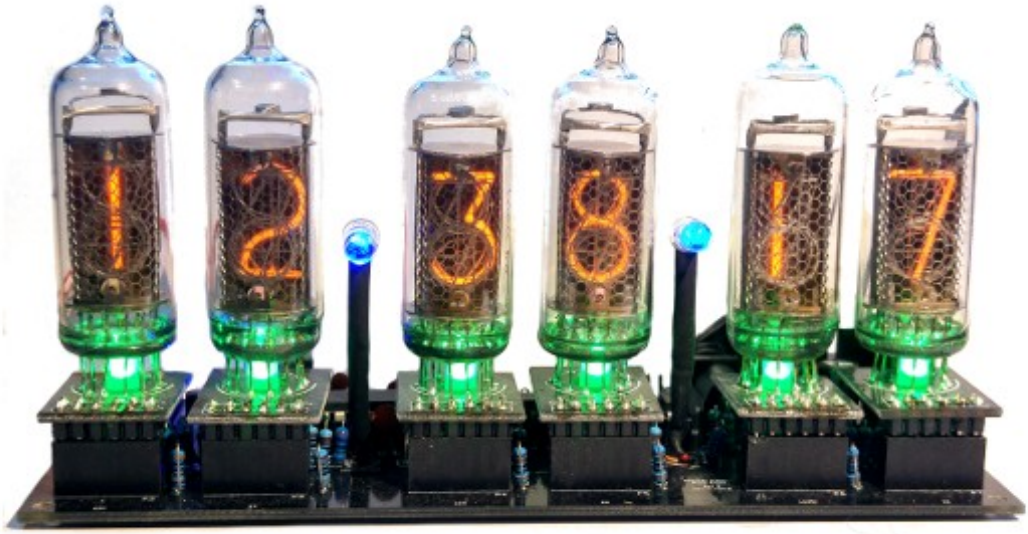
Arduino Nixie Clock
“Modular Revision 2”

Construction Manual



Contact Information

If you want to get in contact with us, please email to:

nixie@protonmail.ch

We'll usually get back to you right away. We can help you with kits or construction.

We also offer discounts for direct purchases, we save the Ebay fees, and share this with you.

http://www.open-rate.com/Store.html

Software

The software is available on GitHub at the address:

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

This board works with Release "Revision 4 boards" under the "Releases" tab.

Troubleshooting

If everything does not work as you expect, please carefully look at the tests in the construction steps, and the troubleshooting tips.

At the end of the manual, there is a troubleshooting section, which goes through some of the common problems.

Safety

The voltages produced in the High Voltage circuit can reach peaks of 400V! Take precautions not to electrocute yourself! If you are not sure what this means, please do not use this clock and return it for a full refund.

A shock from the clock high voltage circuit is at least a nasty bite. At worst it can kill you.

We decline any responsibility in the case of injury or death.

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

Powering Up

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

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

https://youtu.be/XA3LOPLX8vI

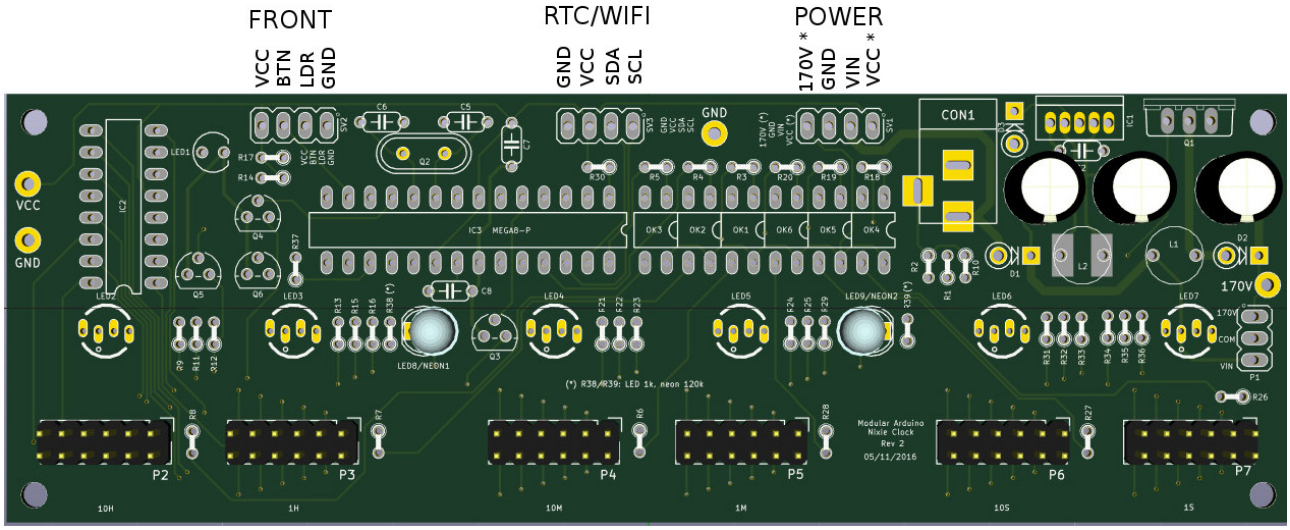
Component Identification

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

Board layout

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

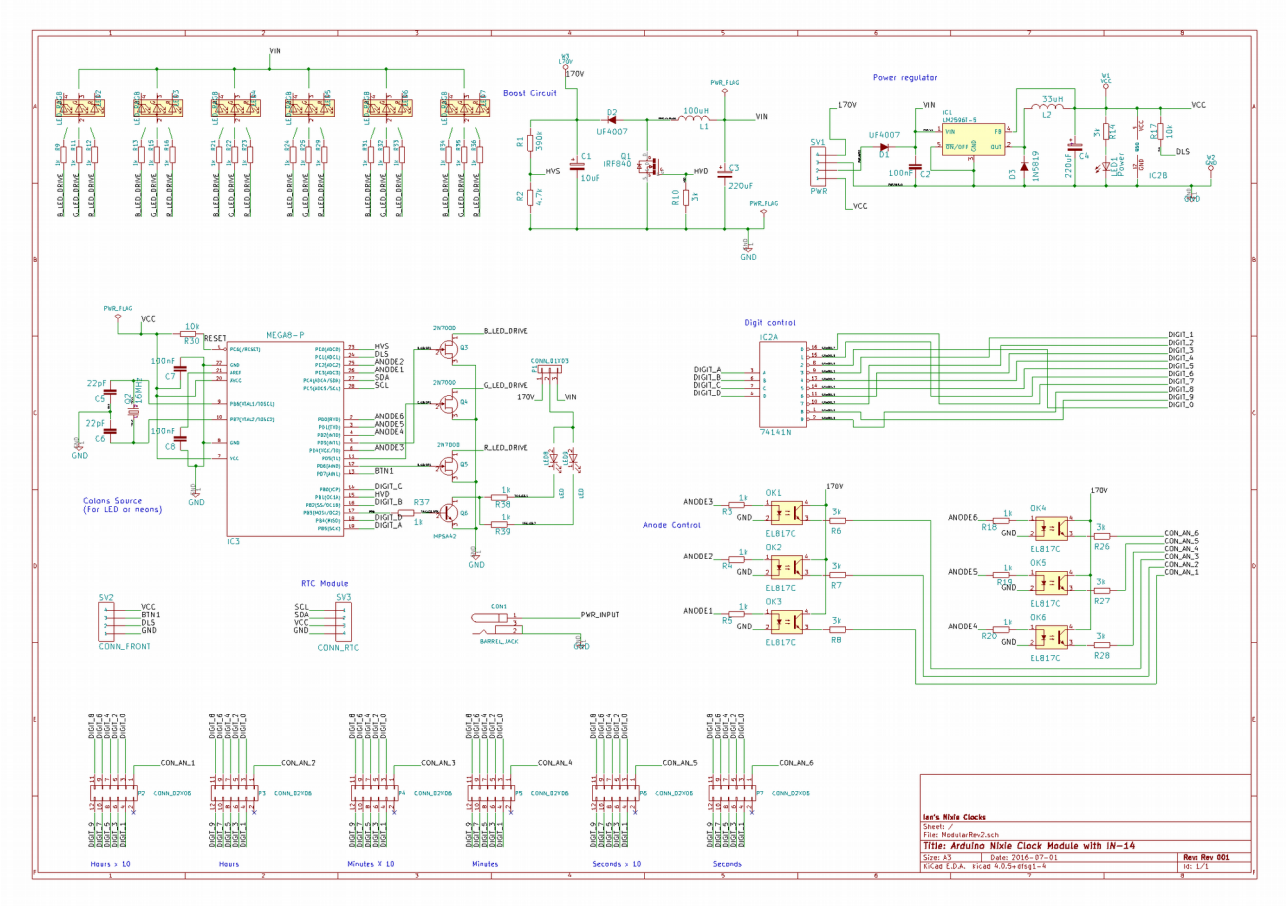


The connections are:

Connector	Description	Values
POWER	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 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 VIN is protected against the input being connected reversed. **GND:** The negative side of the input supply **VIN:** The positive side of the input supply **VCC:** Output of regulated 5V **HV OUT:** Output of high voltage for driving external neons etc.	
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. **BTN:** The other lead of the button is connected to this input **LDR:** “Dimming LDR Sense”: The other lead of the LDR is connected to this **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.	
RTC/WIFI	The connection for the RTC (Real Time Clock) or WiFi time	

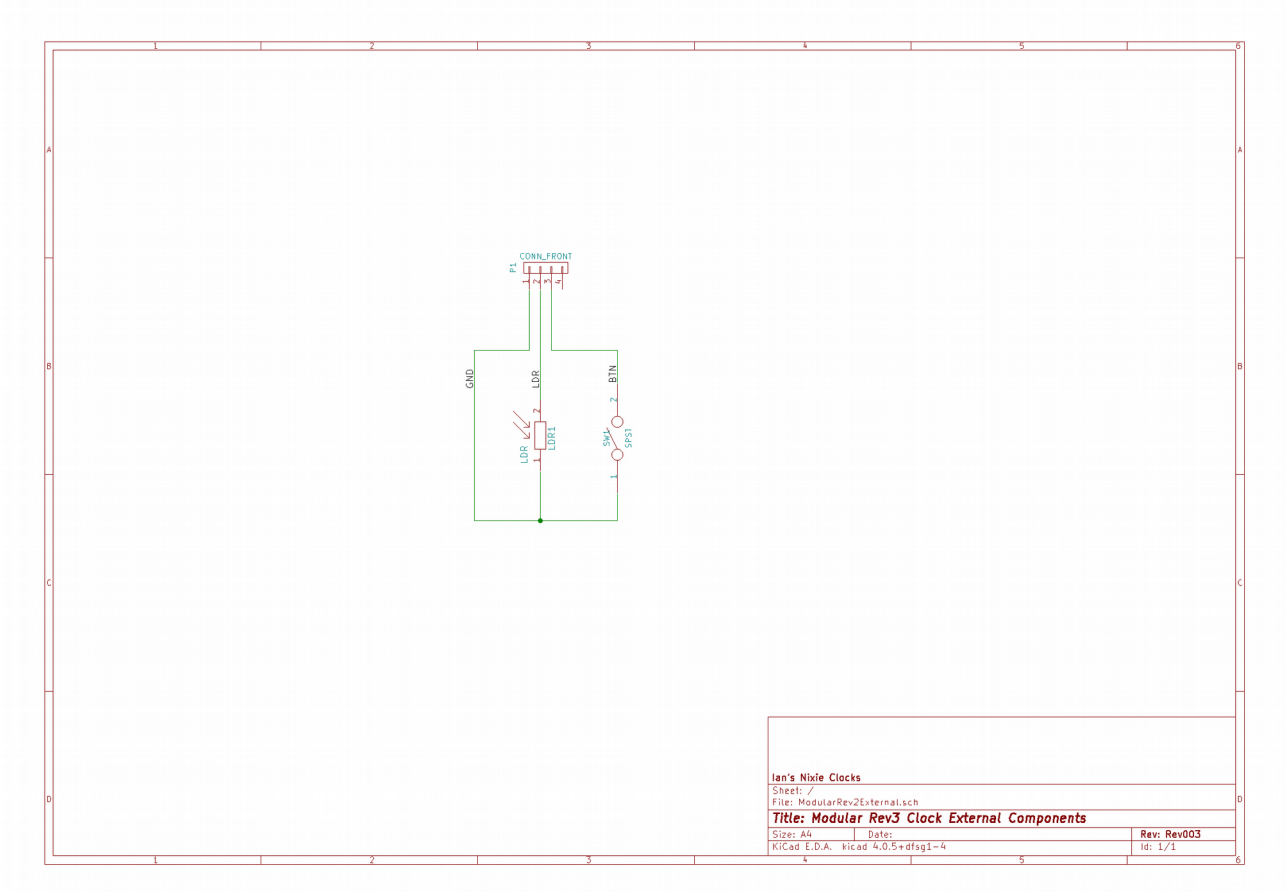
provider module. Connect this to the appropriately marked terminals on the RTC / WIFI module.

Schematic



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).

Construction

Preparation:

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

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.

Please see the appendix to help you identify individual components.

Low Voltage Circuit:

Parts List:

D1	UF4007
C2	100nF
C3	220uF
C4	220uF
IC1	LM2596
R14	3k
L2	33uH
D3	1N5819
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.

Notes:

- See the document 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)
- LED1 must go with the right polarity. The side which has the shorter lead goes nearest the FRONT connector. (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 neither the voltage regulator nor the diode nor the inductor gets 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 none of the components heat up. All components should stay almost cold.

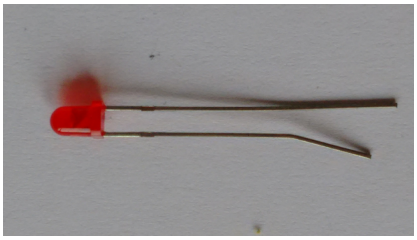
Hint: The 220 uF capacitor

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”

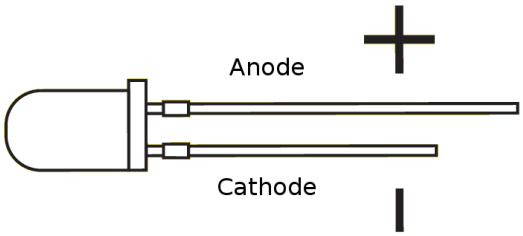
Hint: The LED orientation

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 hole on the board nearest the diode.

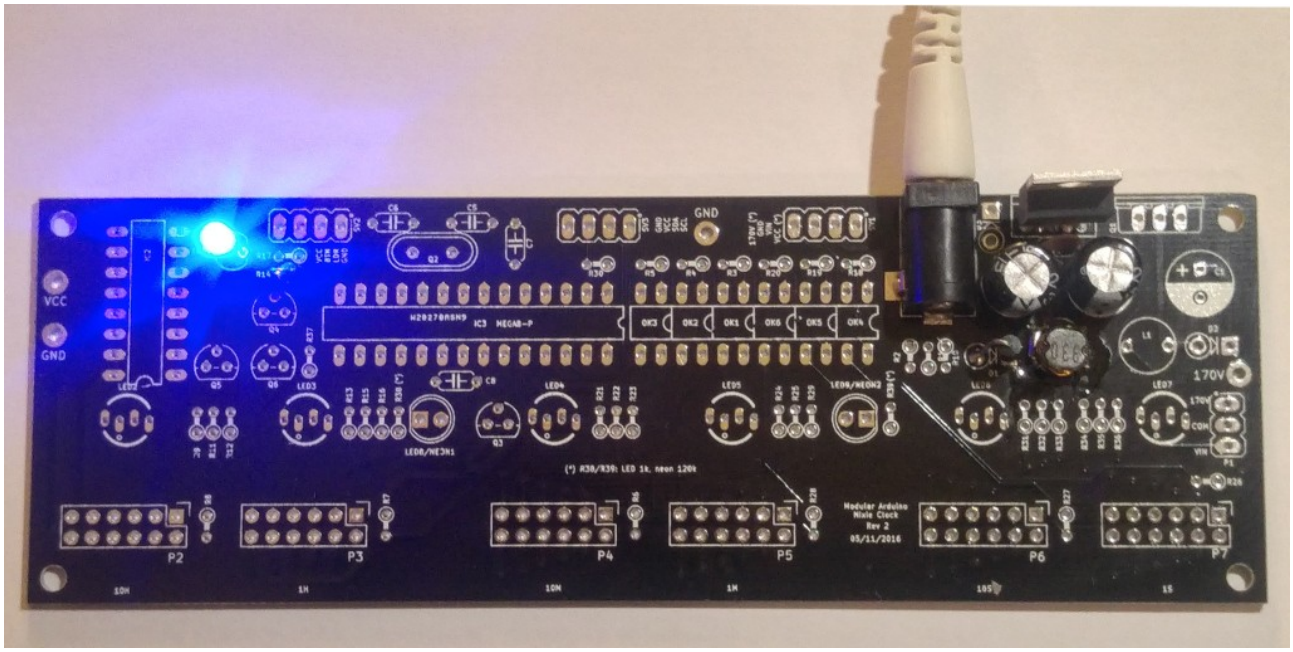


The LED

The LED should look something like this:

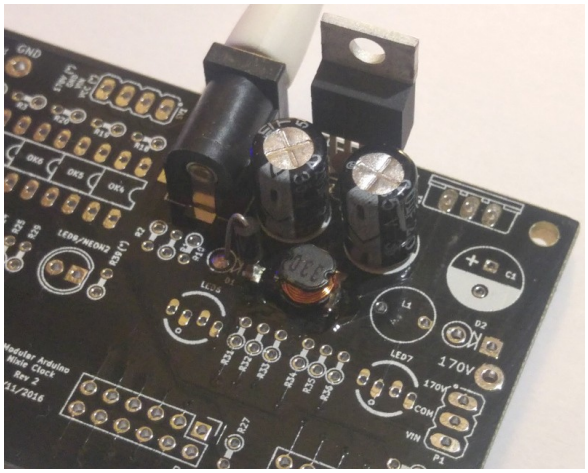


At the end of the low voltage circuit build, your board should look like this:



Low Voltage Circuit

From a different angle:



Low Voltage Circuit Component Orientation

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	IRF840
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.

Notes:

- See the document 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. 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.
- Mount the inductor L1 **tight** to the board. If you don't put it close enough to the board, the back of the tube holder will not sit tightly in the socket, and the tube will be tilted forwards.

Warning!

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-controller, and this will destroy it.

See the picture to help you with the orientation.

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 bottom of the board (near the 170V test point).

Warning!

Be careful, we are dealing with high voltages now!

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

Test Step

Apply power to the board again. Listen for any stressed sounding buzzing or humming, and check that neither the Voltage Rgulator 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.

Trouble shooting

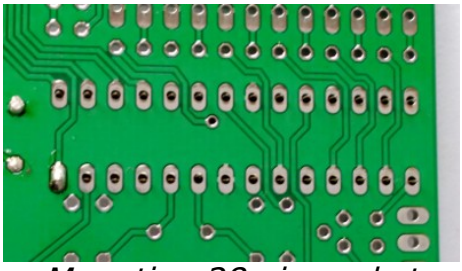
Q1 can get warm, **but should not get too hot to touch**. If it gets hot, you need to check the orientation of the components and that there are no solder bridges.

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.

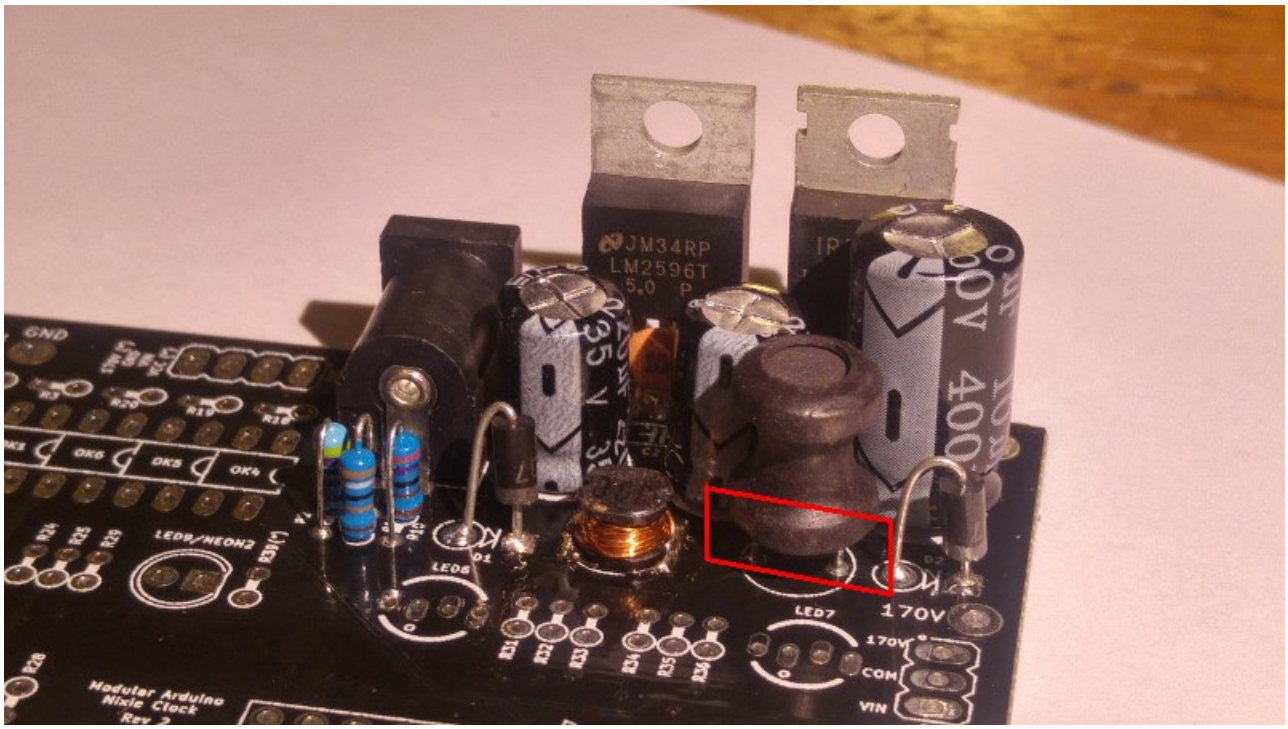


Mounting 28 pin socket

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

High Voltage Circuit

Here is a close up of the high voltage circuit. The gap marked in RED must be as small as possible, not like in the picture. The bottom of the inductor should be pushed right down onto the board



High Voltage Circuit Detail

Tube holder sockets

Parts List:

P2 - P7	2 x 6 Female Connectors

This step mounts the tube holder connectors. We need to have these ready so that when we mount the back light LEDs later, we are able to set the right height for them.

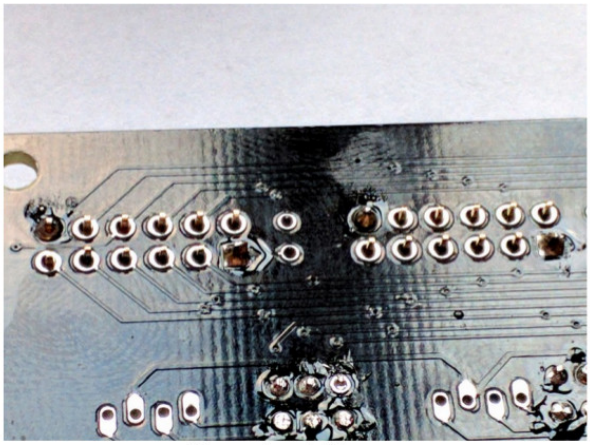
Each tube holder is held by the connector at the front, and rests on the top of the RGB back light LED at the back. This ensures that the tube is held in place, but can be adjusted a little to the left and right to make the tubes stand up parallel to one another.

The tube holder sockets can be tricky to mount, so we use some tricks to make it easier.

Place the 6 sockets on the board, and hold them in place temporarily with some tape:

Tape the sockets in place

This will then allow you to tack the sockets in place. On each socket, solder one or two pins, and then re-flow the solder, making sure that the sockets are perfectly flat on the board and upright.



Then tack them

Once you have tacked the sockets and made them flat, you can firmly solder all the rest of the pins.

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.

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.

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. Each time the digit to be displayed, the correct cathodes have to be set.

Notes:

- Be careful to orient the K155ID1 correctly

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:

After the anode and cathode circuits are installed.

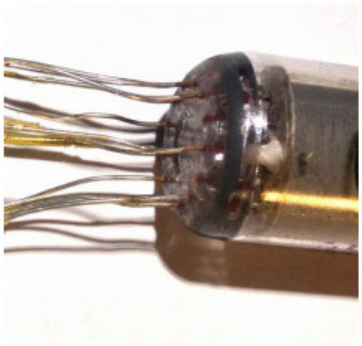
Tube Holders

Parts list:

Tube Holder	Tube holder PCB boards
Connectors	2 x 6 male connectors
IN-14	IN-14 tubes (not supplied)

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.



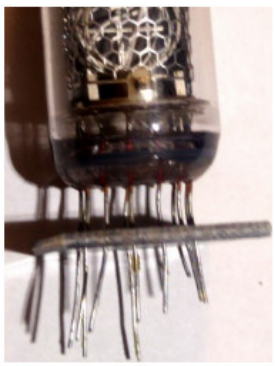
Often tubes have the two decimal point leads removed, especially if they are used tubes. Notice the two missing leads on the tube shown on the left. The lead in the middle with the white coating inside the glass is the **anode**.

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.

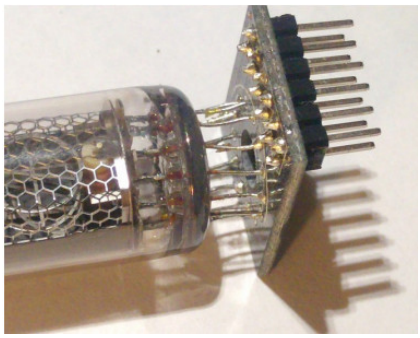
Now you are ready to mount the mount the tube. The semi-circle marking on the tube holder board shows the front of the tube, closest to the 12 pin connector.

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.

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.

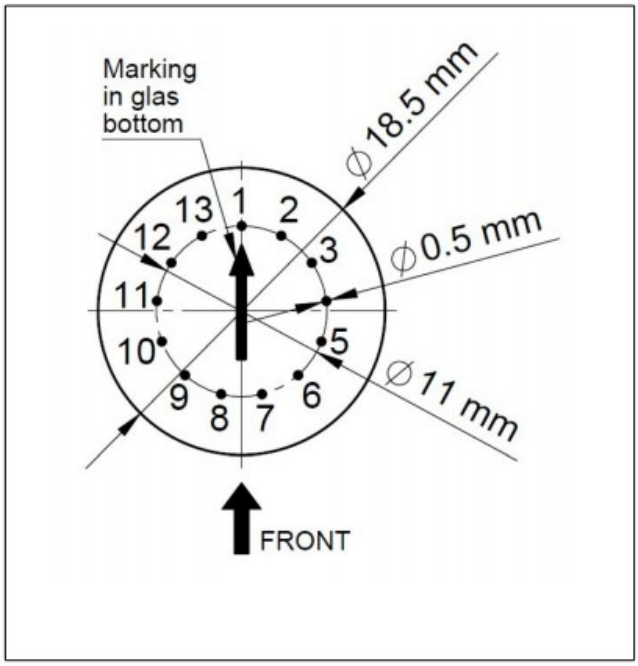


When you have finished mounting the tube, mount also the 2x6 header on the board, so at the end it looks like the picture on the right.

Repeat the same process for all the other tubes.

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 1	Anode
Pin 2	Left decimal point
Pin 3	Digit 1
Pin 4	Digit 2
Pin 5	Digit 3
Pin 6	Digit 4
Pin 7	Digit 5
Pin 8	Digit 6
Pin 9	Digit 7
Pin 10	Digit 8
Pin 11	Digit 9
Pin 12	Digit 0
Pin 13	Right 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.
- 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, through the hole in the tube holder board.
- To make the tube stand upright, solder just three leads at the beginning (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.

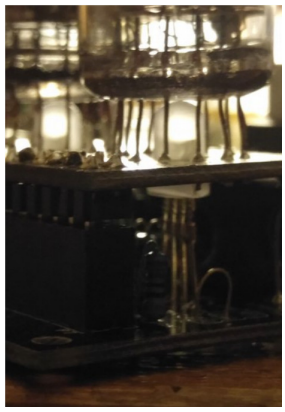
Back Light LEDs

Parts List:

Q3,Q4,Q5	2N7000
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 and checks the back light LEDs and sets their position so that they support the tubes and light them up brightly. This step needs to be done once the tube holders are ready, because the height of the LEDs must match the height of the bottom of the tubes.

You might want to test after you install each RGB LED. The startup pattern should slowly turn on and then the Red channel of the RGB LEDs, then the Green channel and then the Blue channel. Note that it can take 10 seconds for the Red channel to light up, because the startup test checks the colons first, which are not yet installed. The pattern should repeat for as long as you need.



The RGB LED needs to be installed so that it goes through the hole on the bottom of the tube holder board and touch the base of the tube. The top of the LED supports the bottom of the tube, and makes sure that the tube cannot tip backwards. Once you have set the correct height of the LED, solder it in place. The picture on the right shows how the LED supports the tube and the board, and touches the bottom of the tube to make sure that the tube is lit correctly.

Do this for all of the 6 LEDs, setting each to the right height.

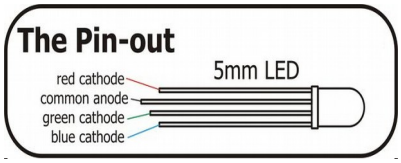
Trouble shooting

If any of the back light channels do 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.

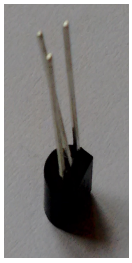
Notes:

- Q3 - Q5 should be orientated with the flat side as shown on the board. Some FETs come with the leads in a row rather than in a triangle. If this is the case, bend the middle lead slightly so that it fits the holes in the board (see hint).
- 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.



Hint: MPSA42/2N7000 mounting

To mount the MSPA 42 transistor or the 2N7000 FETs, bend the middle lead back slightly. It will then fit in the PCB without problems.

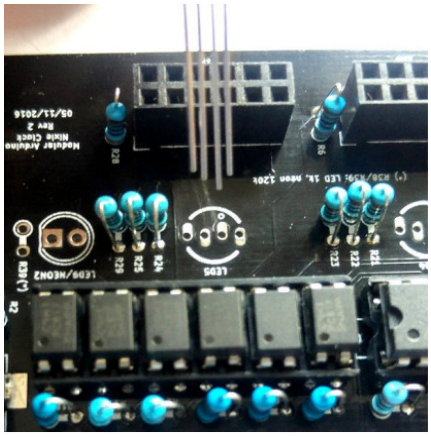


MPSA42/2N7000

Hint: Orientation of the RGB LEDs

The RGB LEDs go into the board with the longest lead in the hole marked with “o”.

Remember to measure the length of the lead by placing the tube holder in the socket and making the head of the LED go right into the hole on the bottom of the tube holder, so it supports the glass base of the tube.



Separator LEDs/neons

Parts List:

R37	1k
R38	1k or 120k (see note)
R39	1k or 120k (see note)
Q6	MPSA 42
LED8/NEON1	LED 5mm or neon (see note)
LED9/NEON2	LED 5mm or neon (see note)
LED4	RGB Common Anode
R17	10k
P1	Connect COM to HV or VCC

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 “HV” and “COM” on P1. If you are using LEDs, you will have to connect “VIN” (”VCC”) and “COM”.

Time Provider

Parts list:

RTC	Real Time Clock Module
WiFi	WiFi module
SV3	CONN_RTC

You should connect the RTC module or the WiFi module, but not both. The pin out is the same for both.

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.

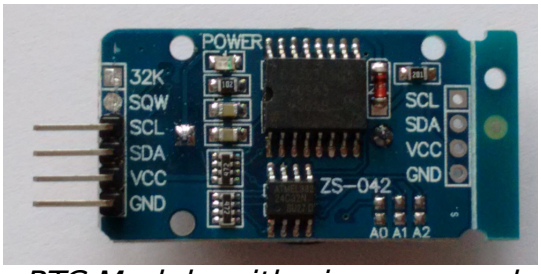
The WiFi module has it's own instruction manual. Please refer to that if you have the WiFi option.

- 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).

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

Front Panel components

Parts list:

SV2	CONN_FRONT
LDR	GL5516 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 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.

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 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

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 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

The MOSFET gets really hot.

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

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

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


Trouble shooting

I can see some “ghosting”.

“Ghosting” is where you can see a very faint image of another number at the same time as the one that should be shown. Some tubes are more sensitive than others, and depending on the construction and components, it might show up more.

If you see ghosting, increase the “anti-ghosting” setting, but only to the point where the ghosting is no longer visible or irritating.

The “anti-ghosting” setting decreases the overall brightness of the display slightly, and not all tubes (even of the same sort) need it, so anti-ghosting should only be used when there is a real need to use it.

Programming the 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

Part	Qty	Value	Packed
PCB	1		☐
TubeHolderPCB IN-14	6		☐
C5,C6	2	22pF	☐
LED9,LED8 (option)	2	LED 5mm	☐
neon1,neon2 (option)	2	neon	☐
C1	1	2.2uF 400V	☐
C3,C4	2	220uF	☐
LED2,LED7,LED6,LED5,LED4,LED3	6	LED RAGB (CA)	☐
Q2	1	16MHz	☐
D1,D2	2	UF4007	☐
D3	1	1N5819	☐
Socket 2	1	16Pin DIP	☐
IC2	1	K155ID1	☐
R2	1	4.7k	☐
R1	1	390k	☐
R14,R7,R8,R28,R27,R26,R6,R10	8	3k	☐
R13,R12,R9,R11,R3,R19,R18,R23,R20,R21,R22,R24,R2 R29,R31,R32,R33,R34,R35,R36,R37,R4,R5,R15,R16	25	1k	☐
R38,R39 (option)	2	1k	☐
R38,R39 (option)	2	120k	☐
R30,R17	2	10k	☐
Socket 3	1	28Pin DIP	☐
IC3	1	MEGA8-P 328P	☐
Socket 1	1	24Pin DIP	☐
OK1,OK2,OK3,OK4,OK5,OK6	6	EL817	☐
SV1,SV2,SV3	3	4 pin header	☐
Q3,Q4,Q5	3	2N7000	☐
Q6	1	MPSA42	☐
C7,C2,C8	3	100nF	☐
LED1	1	LED 3mm	☐
L1	1	100uH	☐
L2	1	33uH	☐
IC1	1	LM2596	☐
Q1	1	IRF840	☐
N2,N5,N4,N3,N6,N1	6	IN-14	☐
DS3231 RTC Module (option)	1		☐
WiFi Module (option)	1		☐
Switch	1		☐
LDR	1	GL5516	☐
Barrel Jack	1		☐
P2,P3,P4,P5,P6,P7	6	12 pin socket	☐
2x6 header	6	12 pin header	☐

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

Revisions:

V0001: 22Jun2017: Split out from instruction and construction manual