

Assembling IN-18 Big Nixie Clock. Recommendations

It is not difficult to assemble this clock. Please have a look at circuit diagram and components layout diagram to see what and where components should be soldered in.

Here is some recommendation, which I hope will help you to assemble your clock smoothly.

- Please follow recommendation below, as these boards I would recommend to solder in right order to avoid some components soldering problem.

Assembling tube PCB:

- Please use assembling components layout diagram to solder components on one side of tube board..
- All main components including IN-18 tubes have to be inserted from the both sides using **IN18Clock Assembling tube pcb.pdf page 1** layout diagram.
- Only board interconnection pins have to be inserted from bottom side of the board. Please use **IN18Clock Assembling tube pcb.pdf page 2** layout diagram for it.
- I would recommend you to solder all resistors first, then K155ID1 chips, then all transistors and then - board interconnection pins.
- **Please note**, that to assemble this board two different type of transistors are required. As they all look the same, please carefully check marking on each transistor. You should have 5 (five) MPSA42 and 3 (three) MPSA92 transistors.
- Now it's time to put connector pins on each tube leg on all 6 IN-8 tubes. When you have done it, carefully insert each tube into the round sockets on the tube PCB. Please match the gap between tube's pins, that tube and PCB have.
*Upside down tube board, holding all tubes and PCB. *Fix it securely on your table, align tubes and solder all pins to PCB.

***We suggest you** to use foam template (can be easily cut from some appliances packaging materials) where you can insert and hold tubes securely while you need to solder it.
- The last two components on the tube board to solder are 2 neon bulbs. WE would recommend you to do it at your last assembling step, when you have checked the clock and know bulbs height position depends on your case design.

Assembling main uC PCB:

- Please use **IN18_cl_assembling_uc_pcb.pdf** assembling components layout diagram to solder components on both side of microcontroller board.
- Firstly solder all resistors laying on the side shown on the **IN18_cl_assembling_uc_pcb.pdf page1** layout diagram.
- At the current PCB release there is only one SMD component on the main board – MAX232 chip. Its size is relatively big to other SMD components, so it should not be a problem to solder it. This chip should be soldered on the bottom side of the microcontroller PCB, please use **IN18_cl_assembling_uc_pcb.pdf page2** layout diagram to locate its position. Just put in the marked position on the board, hold it by your finger and solder one contact. Have a look if its position on the board is OK, then solder other contacts.
- The rest components are easy to solder and should not be a problem. Just remember to use correct polarity on electrolytic capacitors and diodes.
- Don't forget to solder connectors and buzzer. Please note that buzzer should be installed with correct polarity too.
- **Please use only certified and regulated power adapter, as unregulated more than 15 volts could damage your clock.**
- Don't mix polarity on your 12v connection, as your clock will not work.