

## 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.