

Assembling IN-18 1digit 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 I would recommend to solder some components in the right order to avoid future soldering problem.
- This clock does have tiny smd components, its design is quite compact as well, so take extra care and make sure that close to each other pads are not short.

Assembling tube PCB:

- Please use both assembling components layout diagram to solder components on both side of tube board..
- All main components including IN-18 tubes have to be inserted from the main side using **IN18_1D_clock_tubes_pcb_a.pdf** layout diagram.
- This board have only two 4 pins connectors to be installed on the bottom side of the PCB. Please use **IN18_1D_clock_tubes_pcb_b.pdf** layout diagram for it.
- I would recommend you to solder tiny transistors first, as it will be difficult to do so, if you have already soldered IN-18 tube pins. The transistors position is clearly marked on the board, as well as in **IN18_1D_clock_tubes_pcb_a.pdf** layout diagram.
- Next step is to solder three 15K resistors - quite simple and straight forward process if you have managed to solder tiny transistors ;-).
- Then put pins on the IN-18 tube and insert tube in to the tubes PCB. Adjust tube position on the board and solder all pins.
- The last two components on the tube board I usually solder are two 4 pins connectors, which has to be inserted from the bottom side and soldered on the top PCB side. Please use **IN18_1D_clock_tubes_pcb_b.pdf** layout diagram for it.
- All transistor on the board are the same and are smd MMBT6517LT1G.
- All resistors on this board are 15K

Assembling uC PCB:

- Please use both assembling components layout diagram to solder components on both side of microcontroller board..

- Firstly I insert all resistors and cut their legs on other PCB side as close as possible to the PCB. Then I remove resistors and leave it for a while.
- Please solder PIC16F648A smd microcontroller on the bottom side of the board using **IN17clock_uC_pcb_b.pdf** layout diagram.
- Next step is to solder all components laying on the side shown on the **IN17clock_uC_pcb_a.pdf** layout diagram.
- Insert early prepared resistors and solder them from the top, as some holes are blocked by microcontroller soldered at the bottom side.
- Cut LED legs to 1cm from the body and inserted in to the board. Again, solder it from the top. Please position LED properly, so LEDs cut side is facing towards buzzer place.
- Insert the rest components, like button, trim cap, capacitors in accordance to **IN17clock_uC_pcb_a.pdf** layout diagram and solder it from the bottom board side.
- Insert from the top side and solder two 4 ways socket connectors.
- Finally insert from the bottom side and solder two 4 pins connectors.

Assembling DC-DC converted PCB:

- Please use both assembling components layout diagram to solder components on both side of microcontroller board..
- Please solder MAX1771 smd chip on the bottom side of the board using **IN17clock_DCDC_pcb_b.pdf** layout diagram.
- Nothing more should be inserted and soldered on the bottom side of DC-DC converter PCB, unless you are going to solder optional Backup battery, but you have to solder it only when clock is assembled and fully tested.
- Next step is to solder all components laying on the top side shown on the **IN17clock_DCDC_pcb_a.pdf** layout diagram.
- Next components to solder are MOSFET IRFD320 transistor and 220uH inductor.
- Two pins of MOSFET, which are connected together should be inserted in to 2 holes with thick white line close to it.
- Then solder all ceramic capacitor and resistors. Please bend resistor one wire to 180 degrees to insert it vertically to the board.
- Don't forget to solder connectors and buzzer. Please note that buzzer should be installed with correct polarity.

- Insert and solder vertically 1A fuse, shottky and 1N4001 diodes. Remember about right polarity for diodes.

Final assemble of all boards:

- As clock connector does not have any special wrong way insertion keys, please take an extra care when you connect all boards together, as wrong connection could destroy your clock electronics.
- **Please make sure that tubes back, button and power connector are all on the same vertical line virtually goes from top to bottom of the clock !!!**
- Please use only certified and regulated power adapter, as unregulated more than 15 volts could damage your clock.
- Don't mix polarity on your 9v connection, as your clock would not run