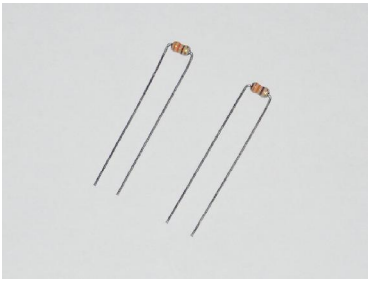
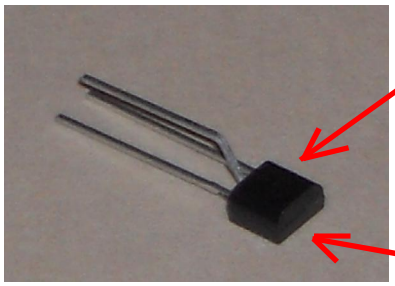
The IN12 project consists of a PCB (available from www.OshPark.com), a parts kit available from Digikey (shared cart ID #6A27796F55) and a pre-programmed microprocessor available at the link on DIYAudio.com.

The IN12 PCB uses all through hole components that are mounted on one side, with the sockets for the Nixie tubes mounted on the obverse side of the PCB.

The PCBs from OshPark are cut from a panel and may have stubs on the side where it connected to other PCBs. Lightly sand all 4 edges of the PCB to remove any burrs or sharp strands.

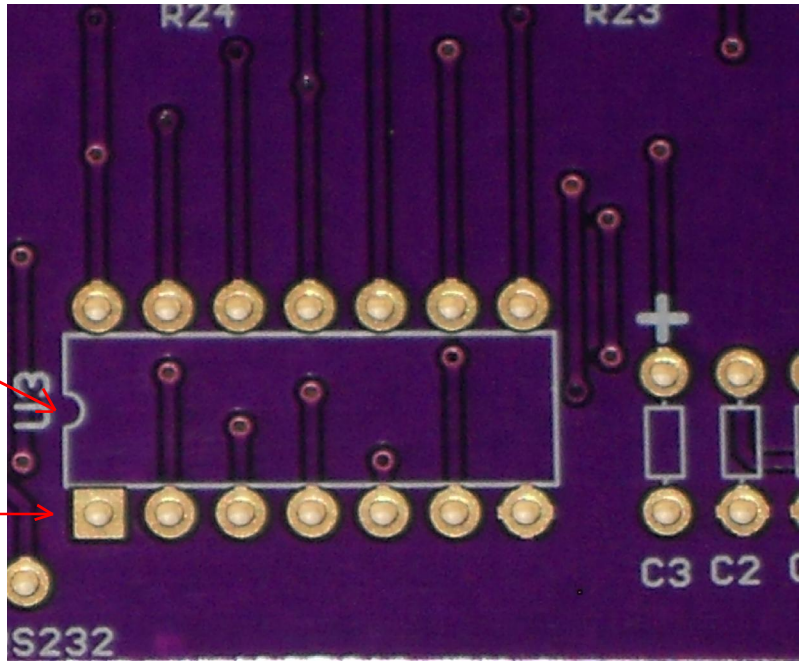
Before assembling the parts onto the PCB, take inventory and make sure all of the needed parts are present. Pre-form all of the resistors by bending the leads down in the same direction, tightly against the body of the part as shown. Pre-form the leads on all of the TO92 transistors as shown.



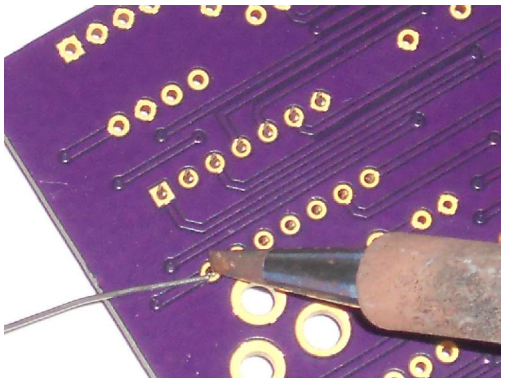


Note that the flat side of the transistor does not have any writing on it; the part # and date code are on the rounded side which is the opposite of most parts. Not also, that the "rounded" side is not completely round and it is easy to insert these parts incorrectly. It is very important that the transistors are inserted properly and pre-forming the leads will facilitate this. There are 23 N channel FETs (Q1-Q13 and QN1-QN10) and 5 P channel FETs (QP1-QP5). Where the N & P channel FETs are in close proximity, the part numbers have N and P in them to help distinguish between them. Do not mix up these parts.

Start populating the PCB by inserting each IC into the proper location on the PCB and soldering the leads from the bottom. Solder all the pins on each IC before inserting the next part. Take care that each IC is inserted properly; pin one of the IC is denoted on the PCB by a square pad where all the other pins are round. The silk screen on the PCB shows an index mark for proper orientation of the part. It's not a bad idea to use sockets for all ICs not just the µP.



Soldering is all about heat transfer. A small beveled soldering tip works best; keep the tip clean by wiping it often on a damp sponge. In order to make a clean solder joint, both the pin and PCB pad need to be heated before applying solder. Make contact with both the pin and pad with the soldering iron for 1-2 seconds, then touch the end of the solder where all three meet (pad, pin and soldering tip). The solder should flow evenly into the PCB hole and form a convcave solder joint with the pin. A proper solder joint will be smooth and shiny; if it is dull and chalky, you are applying too much heat and the joint is oxidizing. If the solder only adheres to the pad or pin and not both, you are not equally heating both elements. In either case, reflow the joint by following the above procedure.



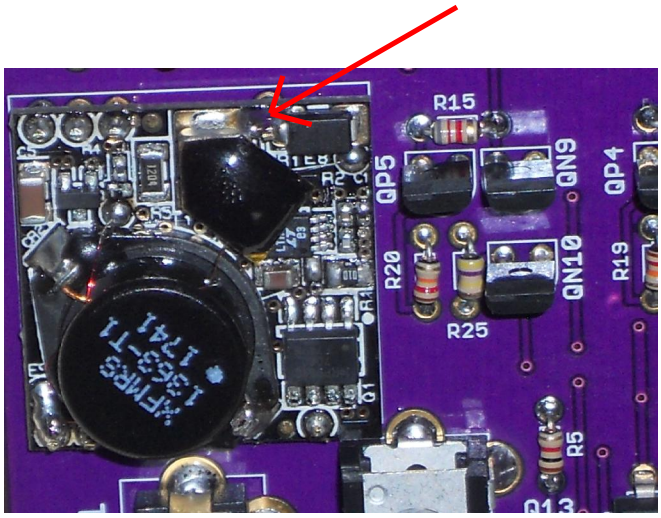
Properly soldered joints

Next, install all of the resistors, one at a time on the PCB. Insert the part and press it down so it sits flat on the PCB; bend the leads slightly outward on the other side of the PCB to hold the part in place while you solder it. Clip the leads close to the pad before installing the next part. Install diode D1 in the same manner.

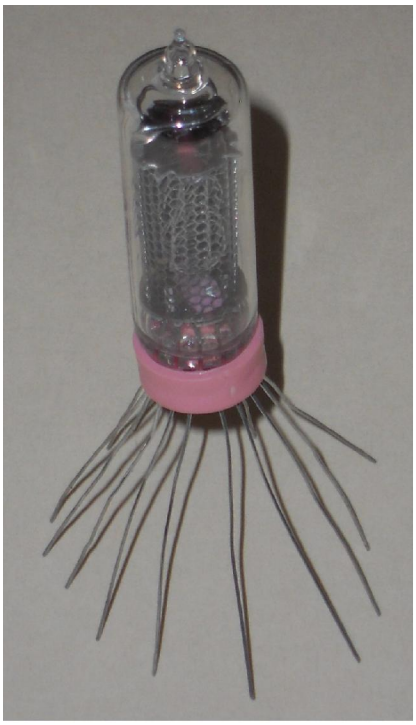
Install the transistors next in the same manner. Install crystal X1 using the same technique. Install the capacitors onto the PCB; not that the 2 tantalum caps are polarized. Install the 2 pin and 3 pin jumpers.

Install regulator U4 and secure using a 4-40 screw and nut before soldering. Install P1 and hold it flat against the PCB while you solder one of the pins. Do not solder all 3 pins yet. Turn the PCB right side up and confirm that P1 is flat and even with the edge of the PCB. With only one pin soldered, some adjustment can be made without reflowing the solder joint. If the part is not seated correctly, reheat the solder joint and correct it. When the part is fitted correctly, solder the other 2 pins. Solder P2 in the same manner.

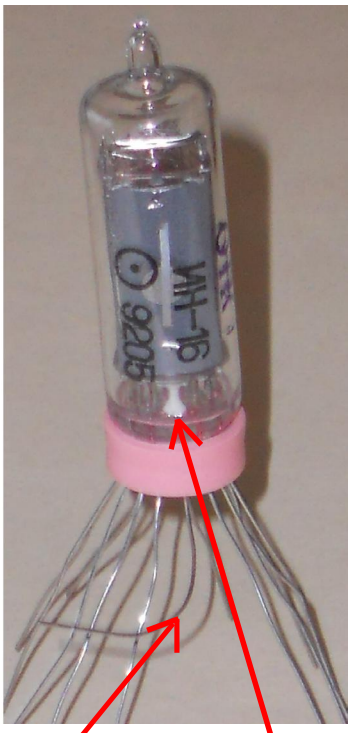
Before installing the HV PSU, it may be advantageous to clean the extra flux from the back side of the PCB using rubbing alcohol and a toothbrush. Clean and dry the PCB before installing any further components. Install the HV PSU and solder the 7 pins. Before installing the Nixie tubes or the µP, apply 12VDC to the power connector P1 (center pin is positive). Check that you have 5VDC between the output (right most) pin of U4 regulator and ground. Check that you have ~170VDC (168-172V) between the upper right most pin of the HV PSU and ground:



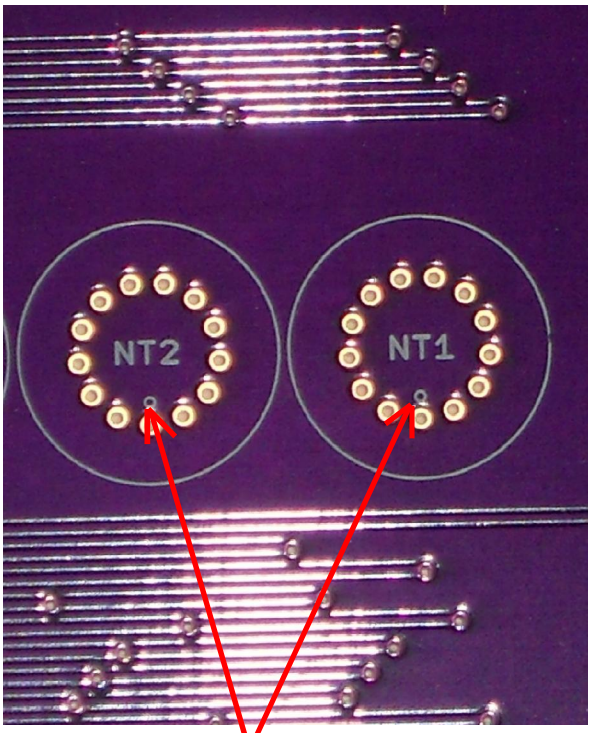
If everything measures OK, disconnect power and prepare the Nixie tubes for insertion. Separate, straighten and spread the Nixie tube leads before inserting them into the PCB as shown; this will make inserting them into the PCB easier. Ensure that Pin 1 of the Nixie tube (identified by the white insulation around the pin) is aligned with the index mark on the PCB (The displayed digits will be facing the edge of the PCB):



Separate and straighten pins



Pin 1 White Insulation



Index Marks

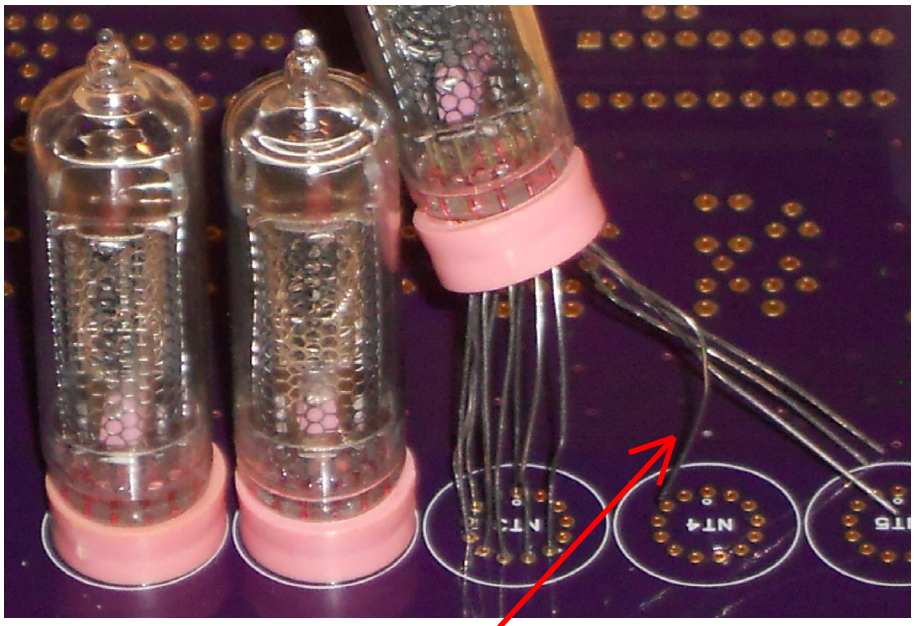
Note: The Nixe tubes are installed on the OPPOSITE side of the PCB from the other components.

Start by inserting the lead for pin 1 into the PCB; push the lead ~¼" through the other side and bend it at a right angle to prevent the lead from pulling out. Carefully bend each lead in sequence to create a "loop" with the end of the lead close to the PCB pad it will be inserted into. Use a tweezer to push the lead through the PCB far enough so the lead is again straight:

When all of the leads have been inserted into the PCB straighten the pins and start to work the tube back and forth while pushing down to push the leads all the way through. The plastic base can be used to straighten the leads by pushing it away from the tube and towards the PCB.

When the tube is completed seated against the PCB, ensure that the display elements are facing forward and the tube is perpendicular to the PCB. Solder two opposing pins and adjust the tube if necessary by heating each solder joint, one at a time, to allow the tube to move.

Solder the remaining leads of the tube before moving on to the next one.



Bend each lead before inserting into the PCB

When everything is soldered in place and verified as correct, install the µP into the socket. Install the shorting jumper across the test pins adjacent to the µP. With the test jumper in place, the µP will light each digit (0-9) of each display tube in sequence for 3 seconds without multiplexing; if there is an error or a bad element (tube, transistor, IC), it will be easier to trouble shoot as each element in the path will be on in a static condition rather than switching at a high speed. If an error is detected, remove the jumper to hold the display in that condition and trouble shoot the signal path with a volt meter. Q1-Q10 are the cathode transistors and coorespond to the display digits 1-9 & 0; for the digit to be illuminated, the gate of the transistor should be at 5VDC and the drain at ground. For each display tube to be active, transistor QP1-QP5 will be switched on cooresponding to Nixie Tube 1-5 (NT1- NT5). The drain of the transistor should be at 170VDC; the drains are connected to R11-R15. After a problem has been corrected, the test jumper can be reinstalled and counting will resume.

After all of the digits and tubes have been verified. Remove power and reinstall the jumper at the correct position on J1: TTL if connected directly to the Arduino output pins or RS232 if connected to a RR tach or an Arduino with an RS232 shield.

If connected to a RR tach, a standard 3.5mm stereo audio cable (male-male) can be used between the tach and the Nixie tube display. If connected to an Arduino application, the data (either TTL or RS232) will enter the Nixie PCB on the “Ring” connection of the 3.5mm cable (Tip/Ring/Sleeve) with the “Sleeve” connection being ground.