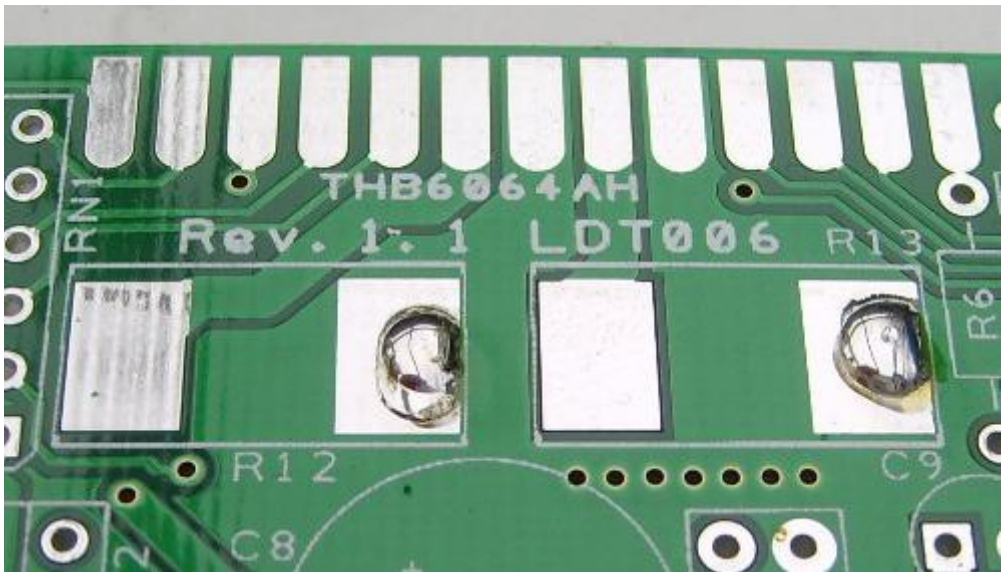
Assembly procedure for special version, Rev, 1.1

There are a few PDF's on my website: “Silkscreen” (which shows part locations and part direction of polarized components), Connections and a part list. These are needed for assembly (to locate the position of each individual part), connection and testing.

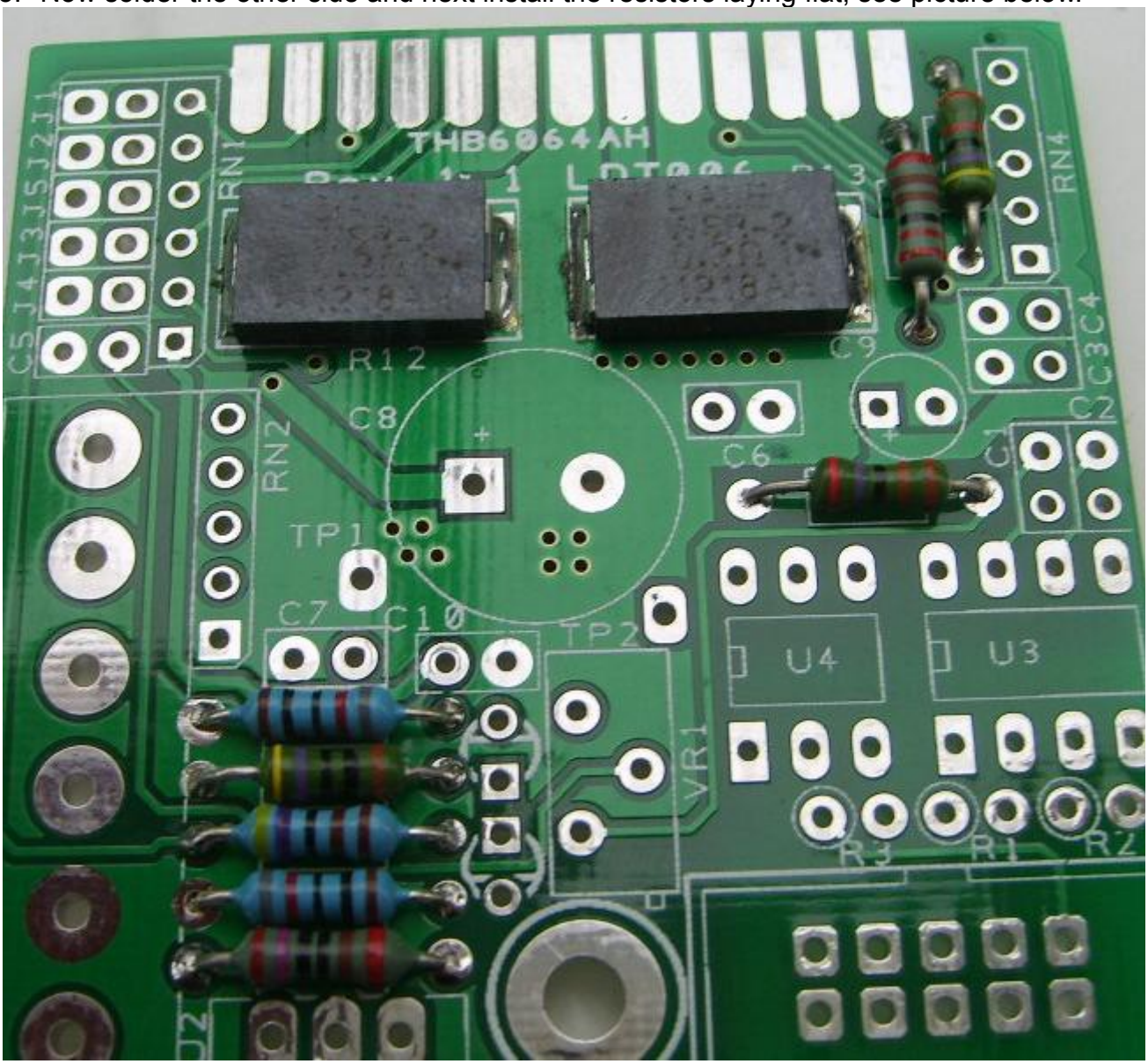
Mount and solder the components with lowest height first and work gradually to the highest ones. It's important for this version to cut all protruding wires as short as possible to prevent shorts to the heat-sink.
When mounting connectors: solder a single pin (preferably one in the center) and then reflow the soldered pin whilst pushing the connector against the PCB, this will prevent mechanical stress on the solder connections.

1. SMD resistors: Put a solder blob on the rightmost pad of R13 and R12 as in the picture below.

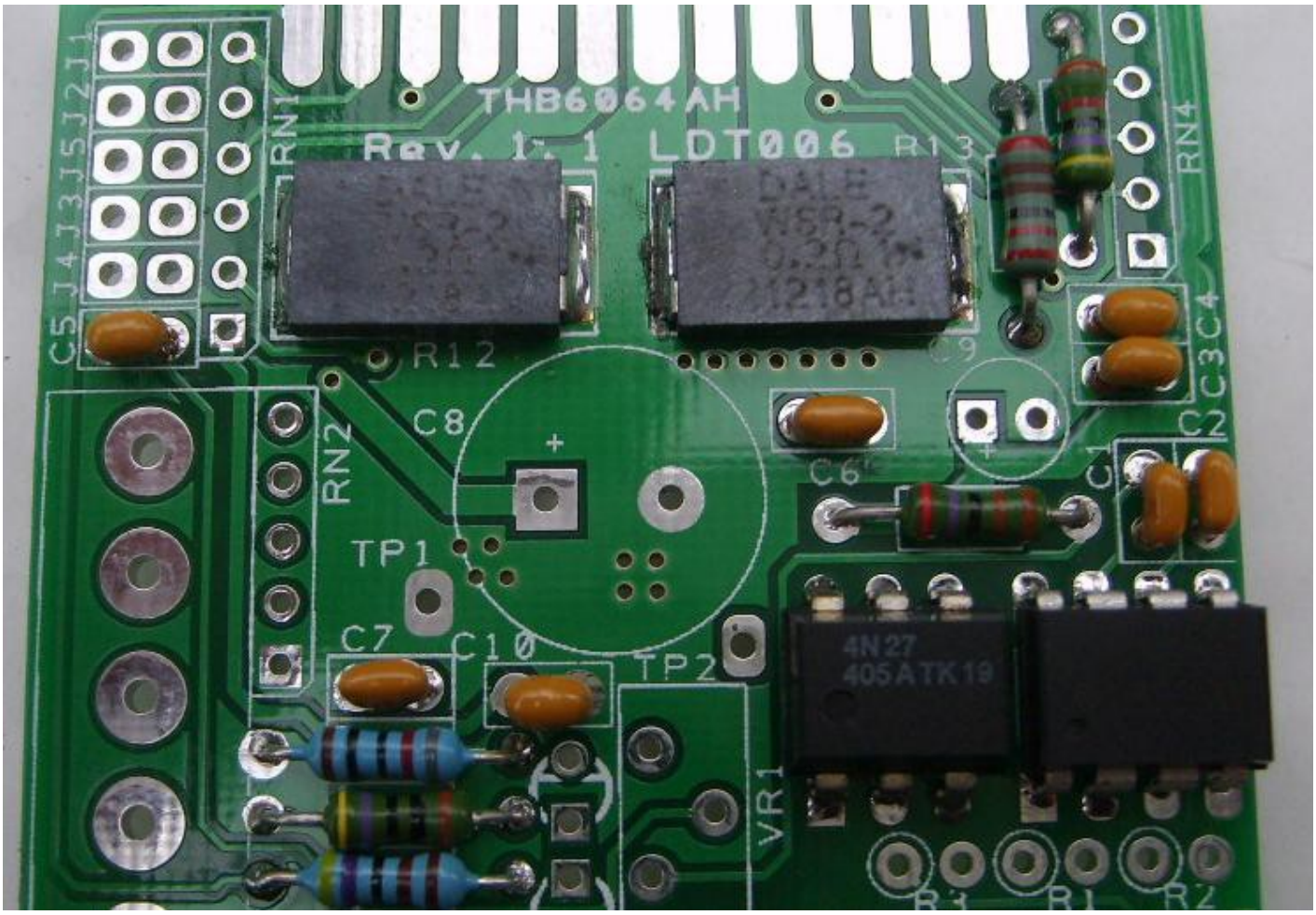


2. Place the resistor to the left of the solderblob, reheat the solder and shift the resistor to the right so that there is some space left to solder the other side.

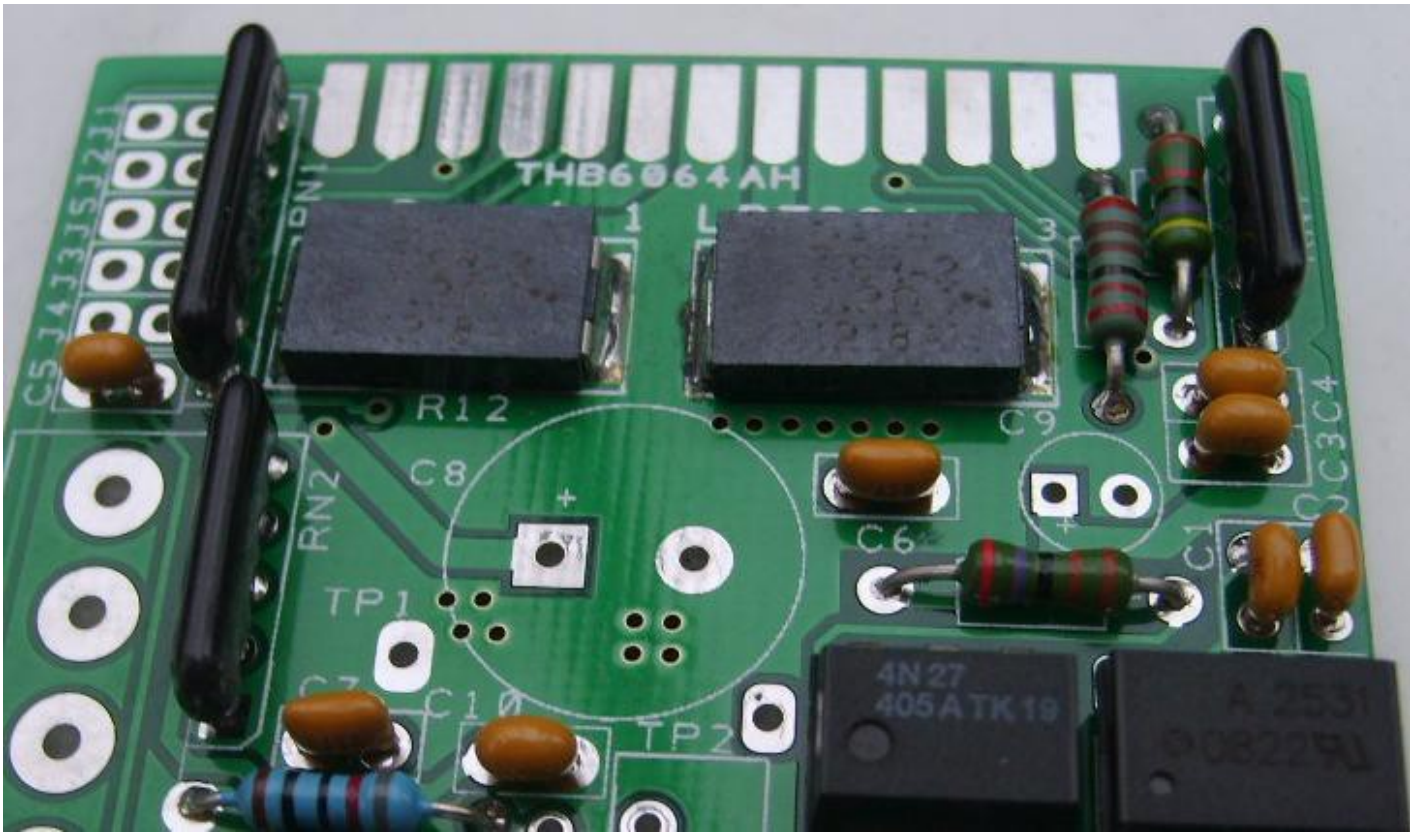
3. Now solder the other side and next install the resistors laying flat, see picture below.



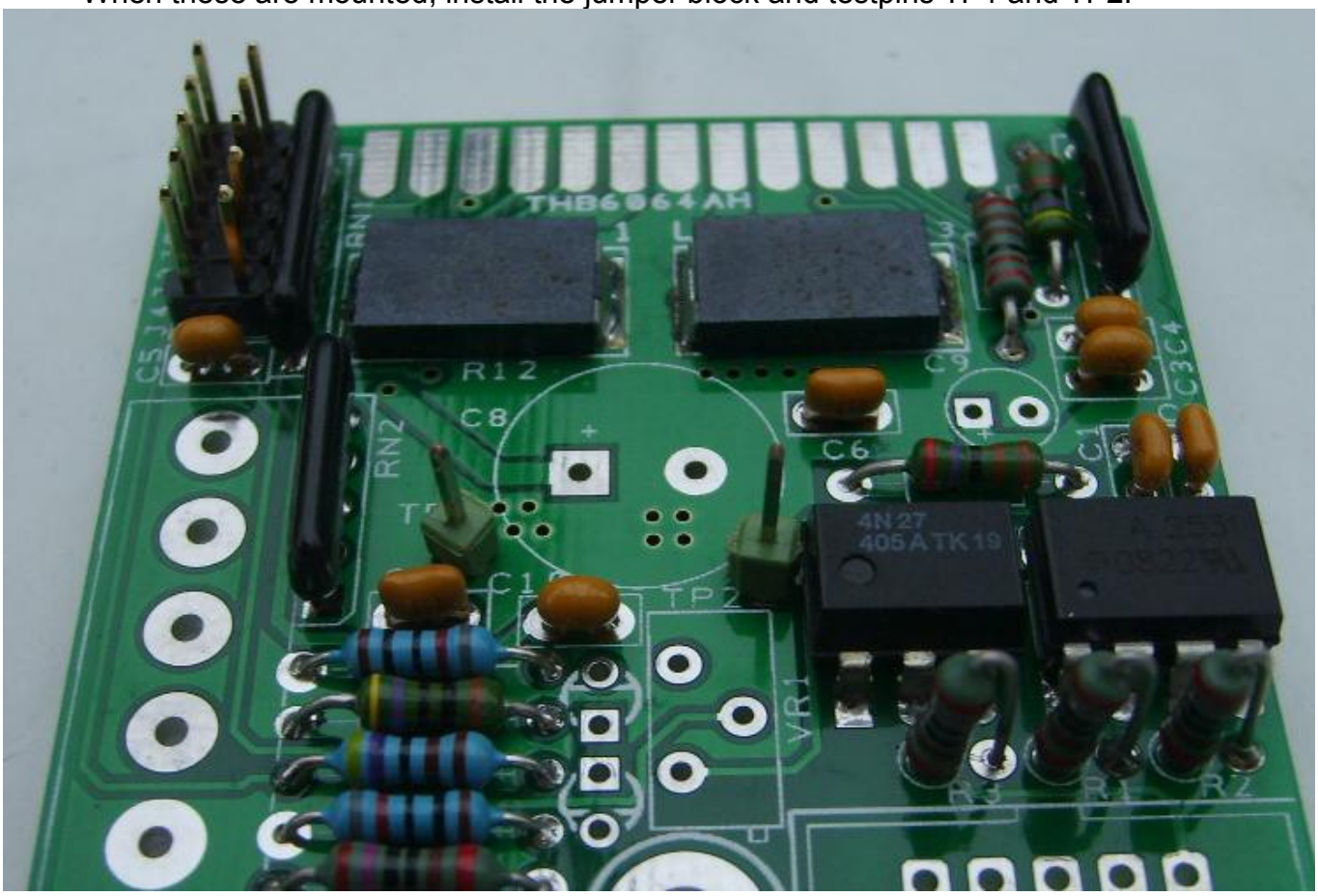
4. Now install the 6 pcs 100n (marked 104) and 2pcs 100p (marked 101) ceramic cap's, see picture below. When these are soldered mount the 2 optocoupler IC's, pay attention for direction, see picture and/or Connections.



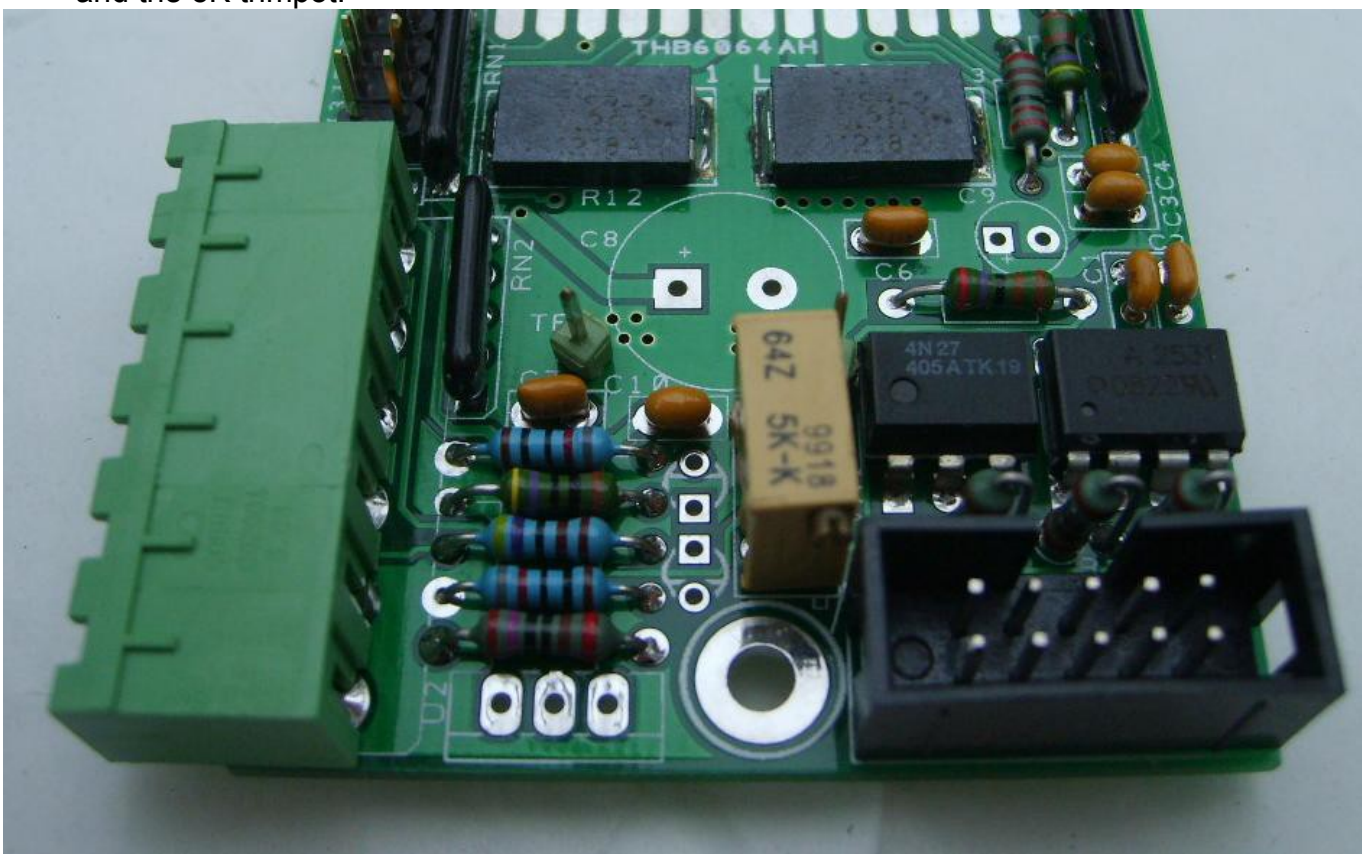
5. Next the resistor arrays, pay attention for pin1, marked with a dot on the array. The pad for pin 1 is the square one.



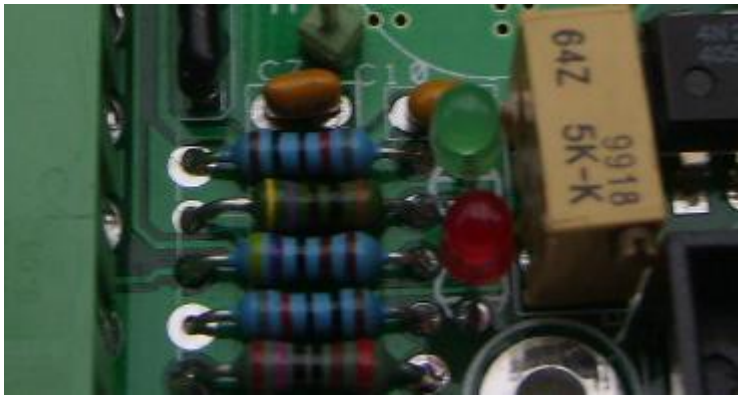
6. Next are the 5 “standing” resistors.
   When these are mounted, install the jumper block and testpins TP1 and TP2.



7. Now install the 6 pin motor connector, the flatcable header (see pictures for it's direction) and the 5K trimpot.



8. Now the 2 leds: The square pad indicates the positive (longest) pin. Let the led's drop to the tabletop so that they are mounted a bit above the PCB for better visibility.

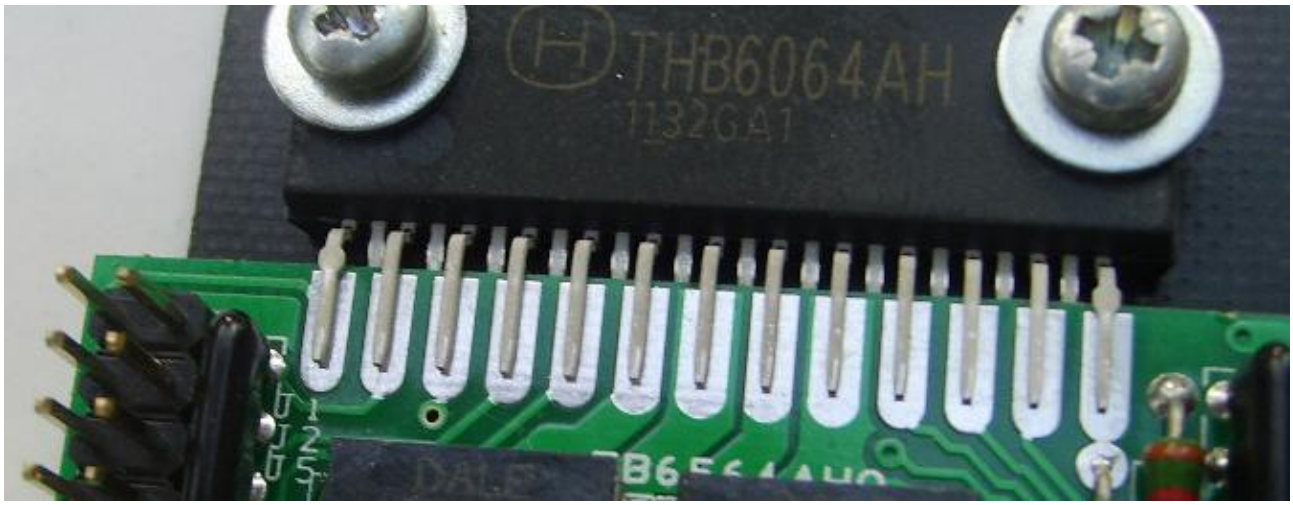


9. There are now 3 components left to install in this sequence: Small electrolytic capacitor, long wire goes in the hole marked with a “+” on the silk or the square pad.
   Voltage regulator with a small heat-sink, see silk for direction.
   Big electrolytic capacitor, long wire goes in the hole marked with a “+” on the silk or the square pad.

At this point, without the chip installed, do a visual inspection preferably with a magnifying glass. Then apply power, 12-18VDC from a small adapter or a real power supply (40VDC max) but with a 1A fuse in the +VM line. Now the 5V can be checked, the +5v led should be on and the Vref can be measured and adjusted to a low value: 0.4-0.5V. See the connections PDF for the ground testpin and Vref measuring location.

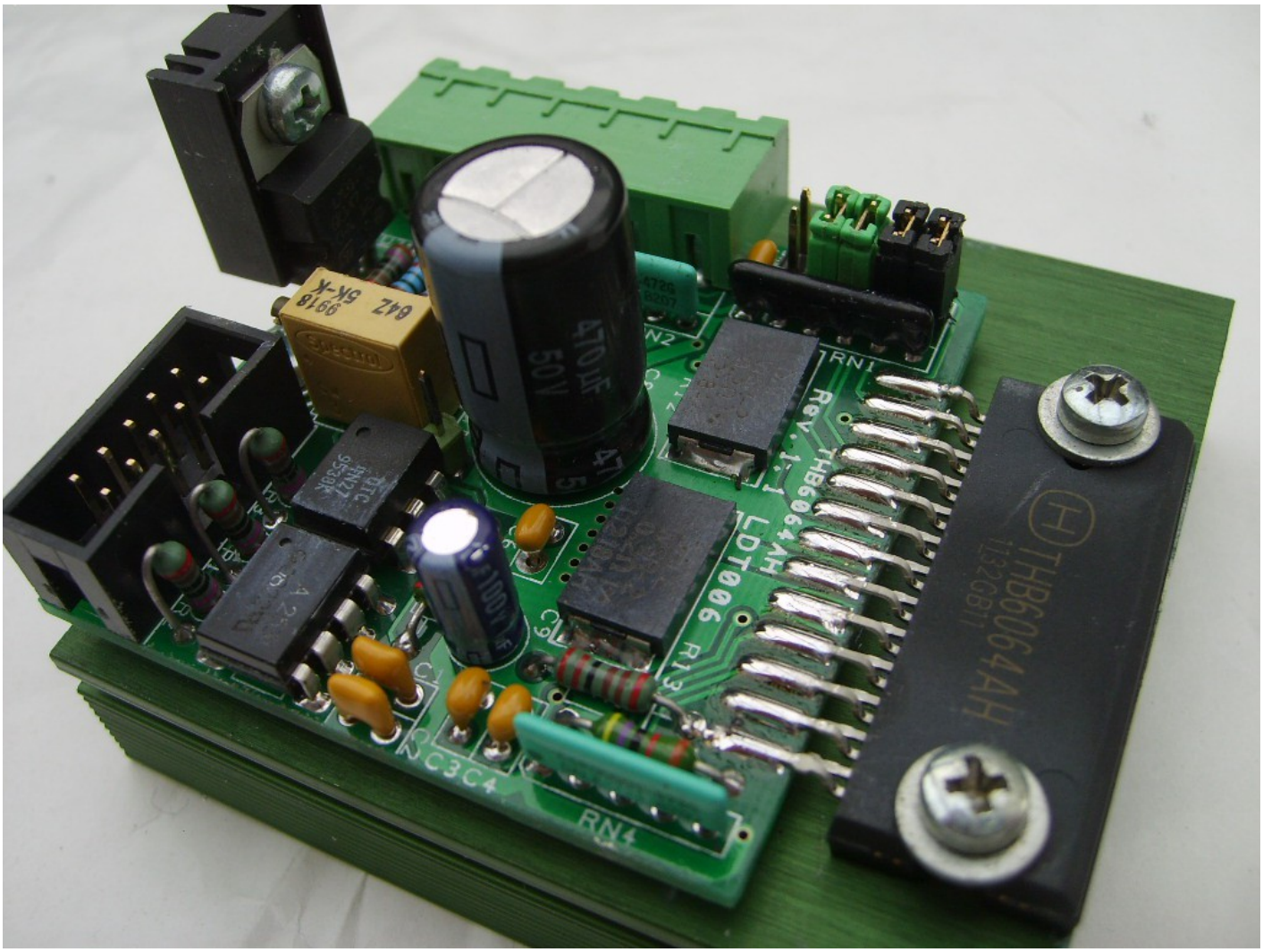
You will need a drilled heatsink or template plate (see Drill locations) for the last step, this is only needed if you want all drive's having the same mounting hole locations. I will include a template in orders for more than 1 drive kit. The template is not symmetrical, it has a marking “TOP” indicating the component side.

Mount the chip and PCB using 3 M3 screws, slide the PCB between the chip's pins, use a spacer-ring under the PCB fixation hole. Spacer height = 3mm. You can use a M3 nut which you drill through with a 3mm drill + washers, a stack of washers should work also.
I have some of these spacers in stock and will include also until EOS.
Rotate the PCB so that the pins of the chip are in the center of the pads, now solder the top row of the pins.



Remove the drive and solder the bottom row of pins.
It's best to do a second inspection for shorts and eventually some measurements on the drive power and motor connector, use a multimeter in diode setting and measure all possible combinations on the connector, none should indicate a short, it takes a bit of time but will prevent smoke or a big bang.

Prepare a heat-sink for a single or multiple drive(s), the template can also be used to mark the position of the mounting holes or to verify them if you traced them yourself.
Remount the PCB with heatsink paste between the chip and it's done, ready for testing.



There's enough space between PCB and heatsink if all pins have been cut short, you should be able to slip a PCB (1,6mm thick) everywhere between the 2.

Install jumpers for the desired settings.

Connect a motor and the power supply, not yet the signal cable. The drive is enabled when there's no current through the enable opto-coupler.
Power up and the motor should lock-up, only a tiny bit as the current is still set low,
If all seems normal you can now increase the Vref setting to approx 30% of the motor rating, why only 30%? Because the drive is in standby mode.
Be very carefull when adjusting VR1, use a matching screwdriver and avoid it to slip away, it might create a short somewhere.
The motor should now be locked, not yet as it should, remember the standby mode, thus the motor may not heat up at this stage.
Leave it on for a while and check if nothing becomes too hot.

Power down and connect the signal cable to a BOB, use whatever program to spin the motor. You can now adjust the current to the motor rating while the motor is spinning.

Revision history:
1.0 Original version

Assembly procedure for PCB rev. 1.1
Revision 1.0 11/11/2012, Copyright LDT006