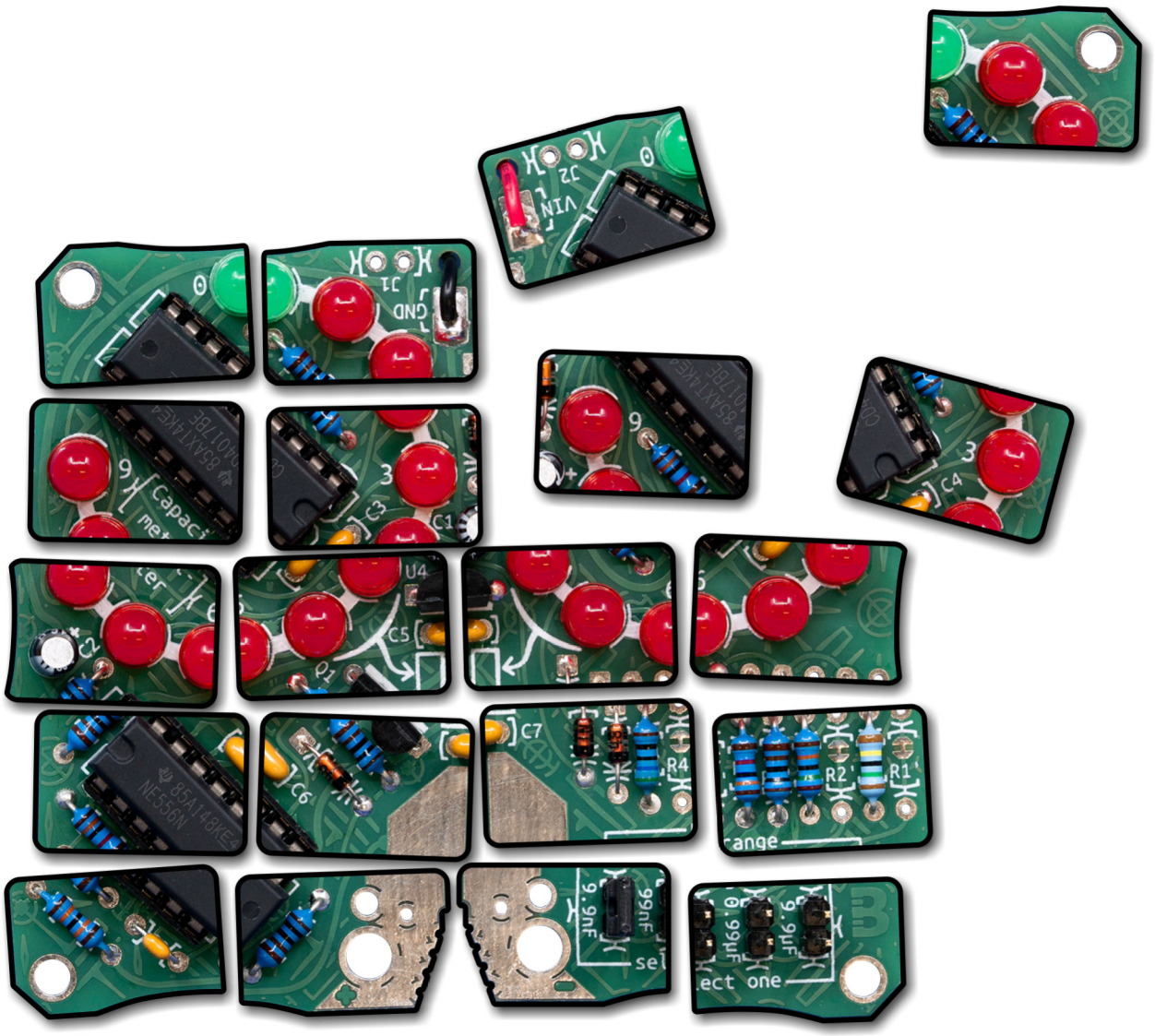
Brief Assembly Guide



Capaci-Meter

A Boldport Project by

Jez Siddons, Stephen Bernhoeft and Boldport.

The Bill of Materials (BoM)

You can use this list of parts to help you check that you've got the right part for each position.

Note that this list is not necessarily the assembly sequence, for example solder the chip sockets before inserting the chips.

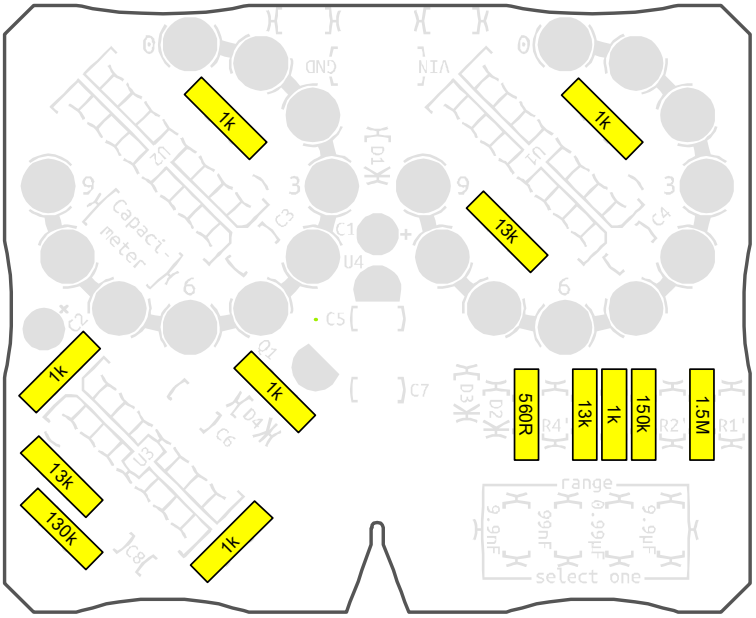
Reference	Qty	Description
Resistors		
R1	1	1.5M 1%
R2	1	150k 1%
R3, R7, R10	3	13k 1%
R3', R5, R6, R8, R9, R12	6	1k 1%
R4	1	560R 1%
R11	1	130k 1%
R1', R2', R4'	3	Not fitted (optional to adjust accuracy)
Capacitors		
C1, C2	2	10uF, 10V or higher, 20%, electrolytic
C3, C4, C5	3	100nF, 16V or higher, 20%, ceramic, Y5V/X7R
C6	1	4.7nF, 1% or 5%, ceramic, COG/NPO
C7	1	47pF, 20%, ceramic, COG/NPO
C8	1	10nF, 20%, ceramic, X7R
Semiconductors and Misc		
D1, D2, D3, D4	4	1N4148 Signal Diode
U1, U2	2	CD4017BE Decade Counter (Use socket!)
U3	1	NE556 Dual Timer (Use socket!)
U4	1	78L05 5V 100mA Regulator
Q1	1	2N7000 N-Ch MOSFET
LED1-L, LED1-R	2	Low current or high brightness green LED, 5mm.
LED2-L to LED10-L, LED2-R to LED10-R	18	Low current or high brightness red LED, 5mm.
LK1, LK2, LK3, LK4	4	Header pins (2 pins per header) for range selection jumper
JP1	1	Jumper needed for header pins (only 1 jumper needed)
J1, J2	2	Pads for aux battery or aux +5V.
J3	1	PP3 9V Battery Clip
J4	1	Connections to test capacitor
U1', U2'	2	16 pin DIL socket (7.62mm)
U3'	1	14 pin DIL socket (7.62mm)
PCB	1	

Step 1 - Resistors

There are 13 resistors to fit.

Verify the resistor value before you fit it in a particular position. Use the resistor colour codes or a multimeter to measure the resistances.

If you're using a multimeter then make sure that your fingers are not in contact with both probes at the same time.



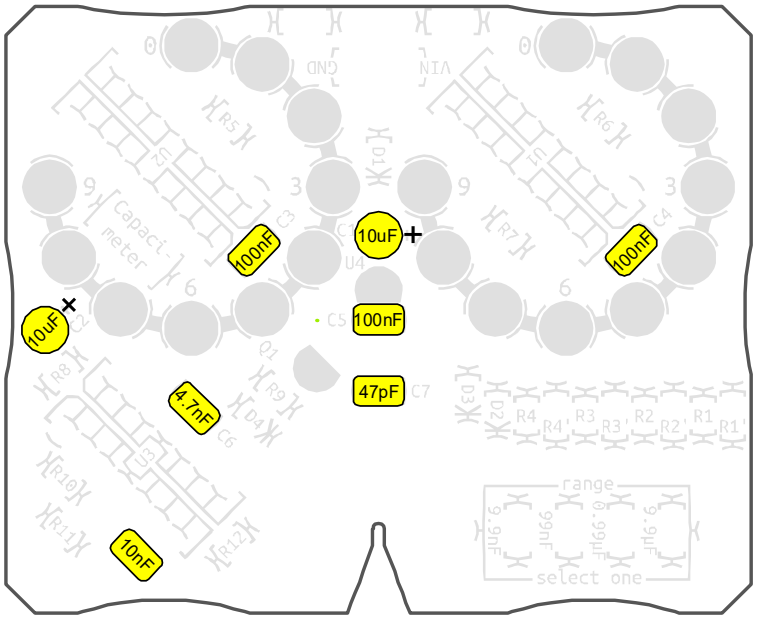
Insert the leads of each resistor into the "component side" of the PCB. That is, the face of the board that has the solid looking circles for the LEDs clearly visible.

When a resistor is placed, bend both leads slightly away from each other so the resistor has a tendency to stay in the board when you flip it over. If you hold the leads "taught" as you apply the bend then it will hold better. If you apply a bend without holding the lead taught then the resistor will still be wobbly when you flip it over. When you're happy that the resistor is firm, solder just one lead on the "solder side".

Inspect the resistor to make sure it's still flush to the PCB. Then solder the other lead. When they've both been soldered you can trim the leads off with a pair of side-cutters. Repeat the process for all the resistors.

Remember:
You will have empty spaces on the board for resistors R1', R2' and R4':

Step 2 - Capacitors



There are two varieties of capacitor used in this project, Electrolytic capacitors and the Ceramic capacitors.

The Electrolytic capacitors are polarised.

The negative lead is clearly marked on the body, so the other lead is the positive that matches up to the “+” symbol on the PCB.

The electrolytic capacitors must be inserted the correct way round and are best fitted flush to the PCB.

The ceramic capacitors are non-polarised and can be inserted either way round.

Here’s a guide to the ceramic capacitor values:

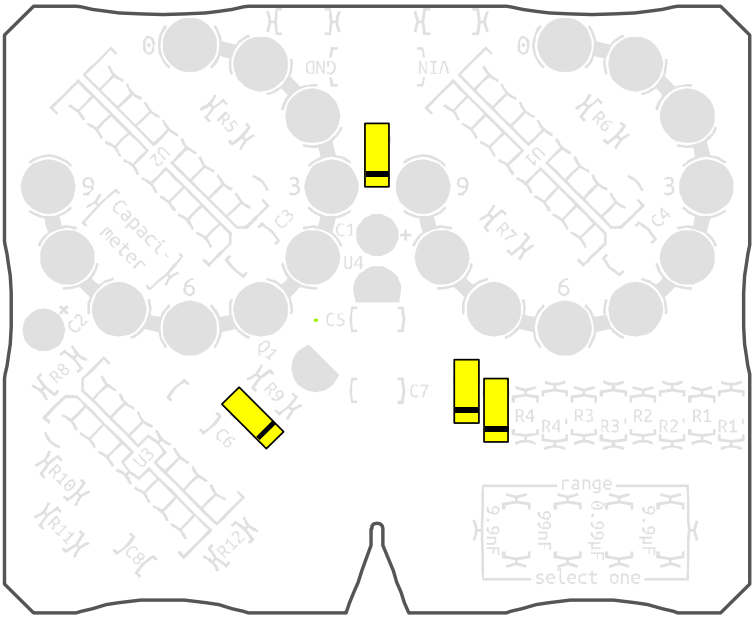
Marking	Value
104	100nF
472	4.7nF
470	47pF
103	10nF

The first and second digit of the marking represent the first two digits of the capacitor value. The third digit represents the number of zeros to add on the end. The value is then in Pico-Farads (pF). So for example, 472 is 47 plus 2 zeros, so the value is 4700pF which is the same as 4.7nF.

Step 3 - Diodes

All 4 diodes in this project are the same type (1N4148). Insert them the correct way round.

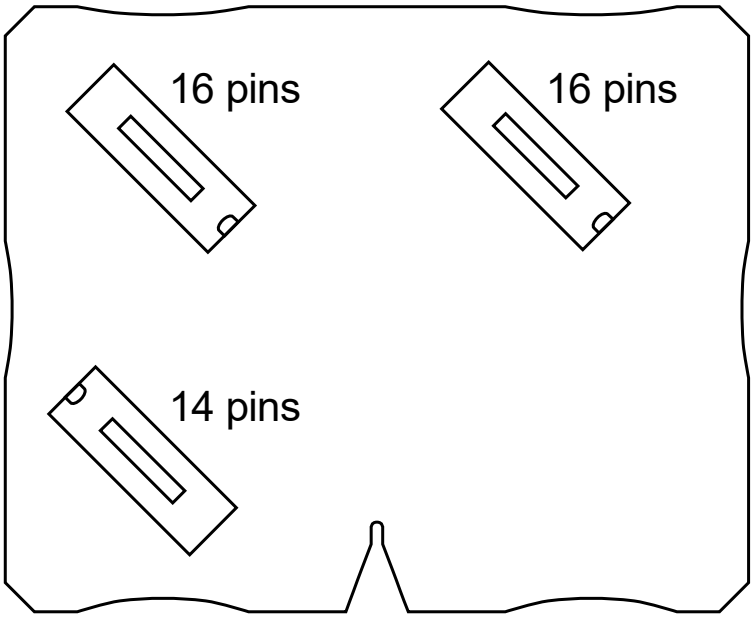
Before you attempt to insert them, bend the leads with flat-nose pliers to ensure that the bends are nicely defined and the correct spacing. Take care not to put strain on the glass or to bend too close to the glass as it will break.



Step 4 - Chip Sockets

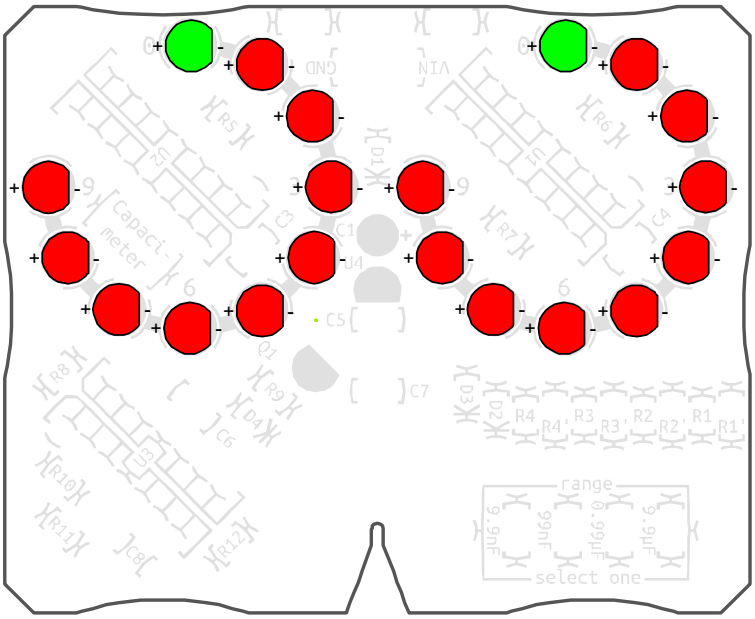
Fit the two 16-pin sockets and the one 14-pin socket in their respective positions. Do one at a time and solder just one pin, check that it's flush and then continue with the remaining pins.

Here's a diagram for the 3 sockets:



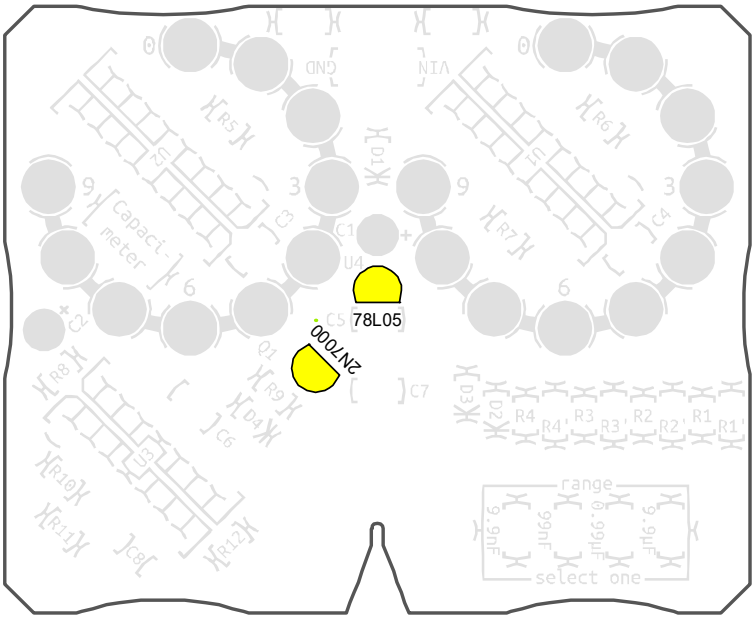
Step 5 - LEDs

All the LEDs are inserted in the same orientation, with the positive (long) lead to the left and the negative (short) lead to the right.



Step 6 - MOSFET and Voltage Regulator

Both the regulator and the MOSFET are in a 3-lead black TO92 style package.



2N7000 MOSFET

The MOSFET (Q1), is marked 2N7000 (among other digits). Insert as far down as it will comfortably go in position Q1 on the PCB.

Ensure it matches the shape on the PCB.

Solder just one lead, make sure that the device is straight and then proceed to solder the other two leads. Then trim.

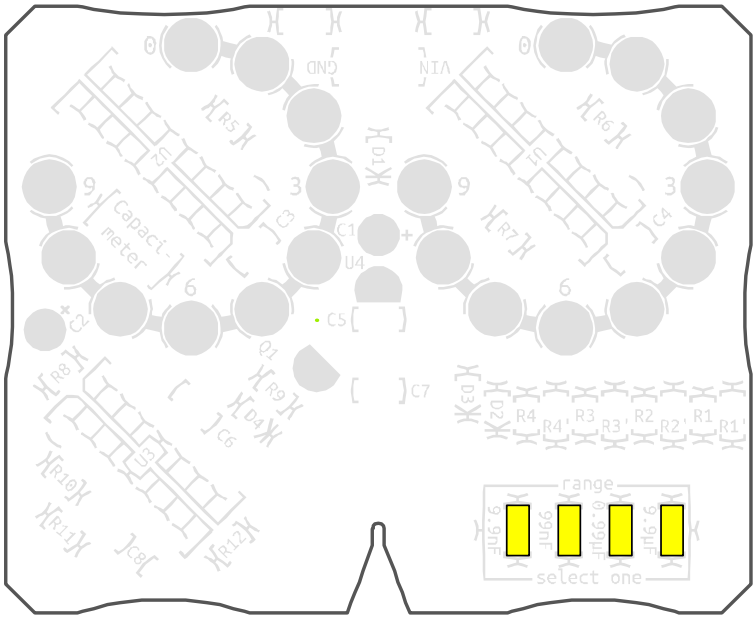
78L05 Voltage Regulator

The voltage regulator (U4) is marked 78L05. Place that device in the position of U4 as far down as it will comfortably go. Ensure that the orientation matches the shape on the PCB. Solder one pin, check the alignment and then proceed to solder the other 2 pins. Then trim.

Step 7 - 2-pin Headers

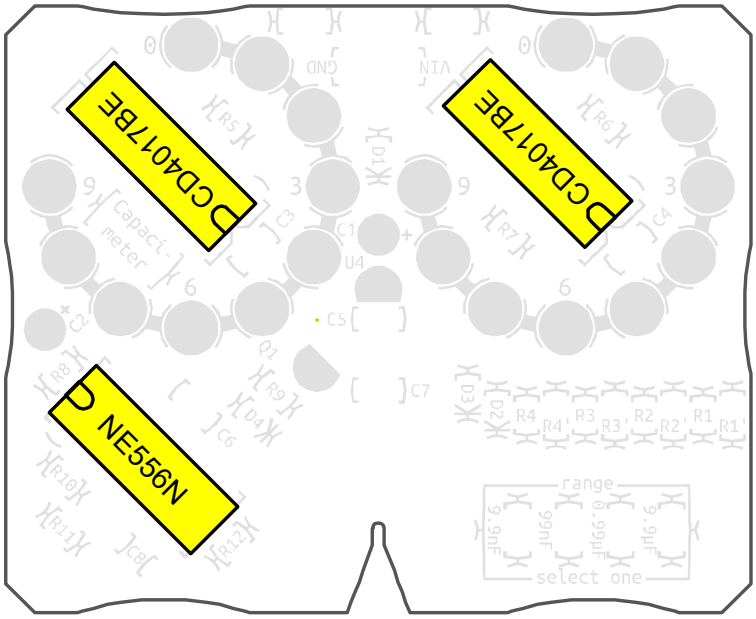
Cut the header strip into pairs and place the 4 pairs into the PCB as shown here:

For each header in turn, solder just one pin, make sure that the header is straight and flush to the PCB, then proceed to solder the other pin.



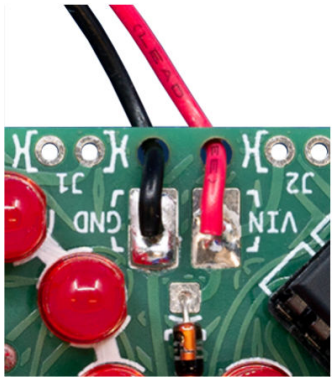
Step 8 - Chips

Insert the 3 chips in the orientation shown here.



Step 9 - Battery Connections

Thread the battery wires through two of the PCB holes and solder the wires into the pads as shown here.

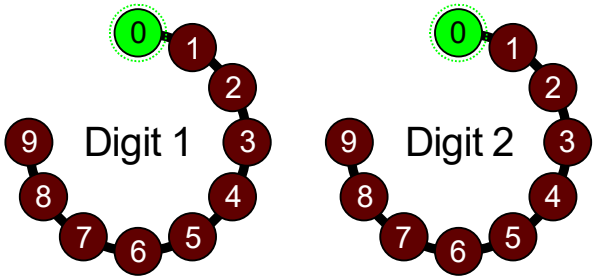


Step 10 - Testing

Before testing, it's worth having a double check that the whole board looks ok and that there are no shorted joints. Check also that the chips are inserted correctly.

Place the jumper "shorting link" across one of the range selection headers.

Now attach a 9V battery to the battery clip. You should see the green LEDs light, possibly in addition to a red LED on the right hand circle of LEDs.



If you don't see any LEDs lit then check the temperature of the regulator U4 with your finger, it should be cool. Also check the temperatures of the chips with your finger, they should be cool too.

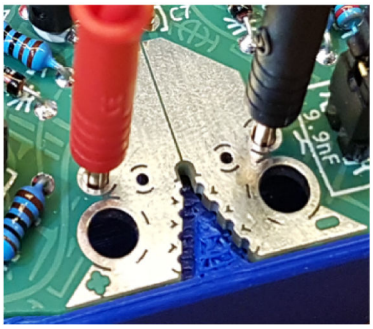
If any of the devices are warm/hot then disconnect the battery and check your solder joints and parts orientation.

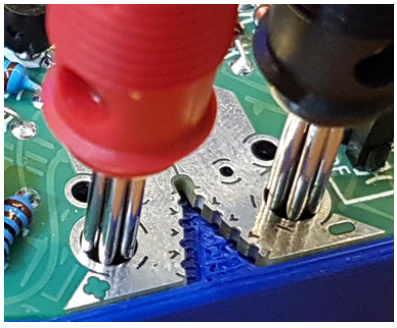
Assuming that the board works ok at this stage, find a capacitor that you'd like to test. Ideally it needs to be one that has a value that sits neatly within one of the 4 measurement ranges:

Range	Minimum	Maximum
1	0.2nF (200pF)	9.9nF
2	2nF	99nF
3	0.02µF (20nF)	0.99µF (990nF)
4	0.2µF (200nF)	9.9µF

For example, if you have a 4.7nF capacitor then the best range to select would be range 1. Of course, if you don't know what the value of the capacitor you're testing is then start with the top range (9.9µF) and work downwards. It doesn't matter if you start on a low range and work upwards, no harm will result from over-range conditions. The display will show "99" if it is over-range (or perhaps just at the top of the current range).

Apply your test capacitor across the two test pads, or use test leads if you wish. The holes in the test pads are suitable for 2mm and 4mm banana plugs:





You should then be able to measure the capacitance value on the display. If not, check the selected range and double-check your PCB joints.

Finally

I recommend that you read the “How it Works” document which gives much more detailed information on the circuit design and operation. It will also help you if you need to make any adjustments or if you have any problems.

I am happy to help with any questions regarding this project. The best way to contact me is via the Capaci-Meter project channel on the Boldport Club Server on Discord.

You can find more information on the Boldport website: boldport.com/capaci

Additionally, all documents (including the latest issue of this one) should be available on the Boldport Github page.

I hope you enjoy building and using this project as much as I have.

Jez Siddons

E&OE