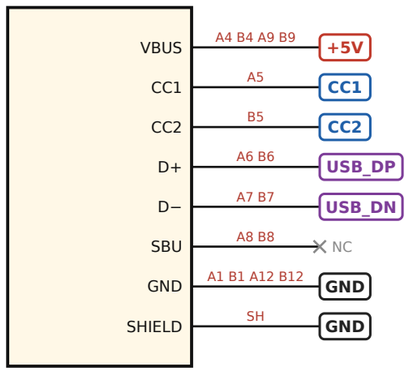
ESP32-C3 DC Power Meter — Schematic

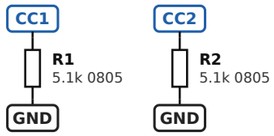
Input for AI-assisted PCB design in KiCad: every part has a reference, value and exact part number; every pin has its number and net name.

A · USB-C 5 V input + ESD

J1 USB-C receptacle
HRO TYPE-C-31-M-12

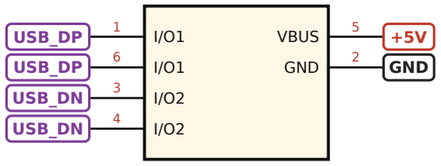


USB 2.0 device, 5 V sink



Separate 5.1k on CC1 and CC2

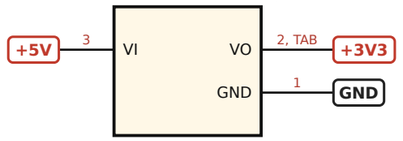
U4 USB ESD
USBLC6-2SC6 (SOT-23-6)



Place U4 next to J1

B · 3.3 V regulator

U3 3.3 V LDO
AMS1117-3.3 (SOT-223)

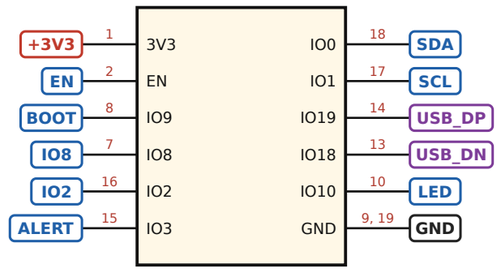




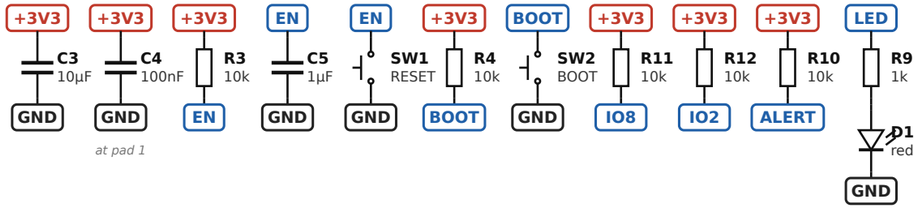
at U3 input *at U3 output*

C · ESP32-C3-WROOM-02 module

U1 ESP32-C3 module
ESP32-C3-WROOM-02 (module, not dev board)

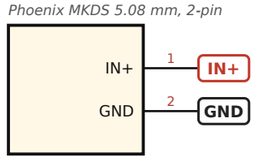


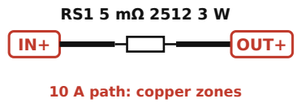
All other module pads: no connection
Antenna end at board edge, keep-out on all layers



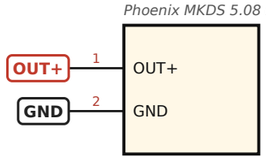
D · High-side power measurement (up to 30 V, 10 A)

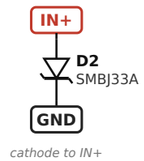
J2 IN
Phoenix MKDS 5.08 mm, 2-pin





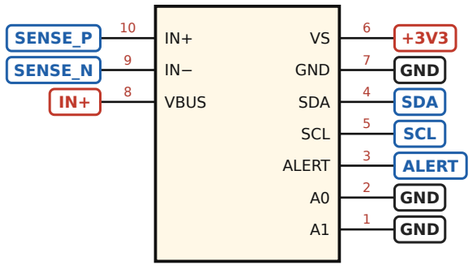
J4 OUT
Phoenix MKDS 5.08 mm, 2-pin

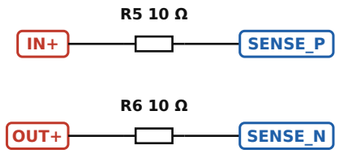


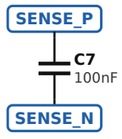


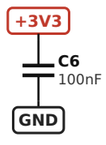
cathode to IN+

U2 Current/power monitor
INA226AIDGSR (MSOP-10), I²C addr 0x40



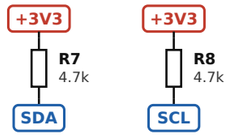




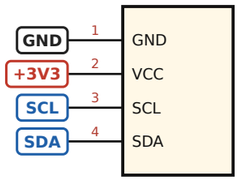


Kelvin sense: R5 and R6 connect at the RS1 pads
Route SENSE_P / SENSE_N as a pair; U2 next to RS1
Measured negative is shared with USB GND (not isolated)

E · I²C pull-ups + OLED



J3 OLED
1×4 header 2.54 mm (0.96" I²C OLED)



Check your OLED module's pin order
Firmware: Wire.begin(0, 1) // SDA=IO0, SCL=IO1

F · Current & voltage per net

Net	Max V	Max I	Copper
IN+, OUT+	30 V	10 A	zone 4.7 mm
GND J2-2→J4-2	0 V	10 A	pour
+5V	5 V	0.5 A*	0.5 mm
+3V3	3.3 V	0.35 A*	0.4 mm
SENSE_P/N	30 V	<1 mA	0.25 mm
USB_DP/DN	3.3 V	signal	0.25 mm
Other signals	3.3 V	signal	0.25 mm

Zone width: IPC-2221, 1 oz outer layer, about 20 °C rise at 10 A.
* estimated peak (ESP32-C3 Wi-Fi)
Min track / clearance 0.15 mm (MSOP-10 pad gap); min drill 0.2 mm
Mechanical: 4 × M3 holes H1–H4, no net (sheet 2)

Board	Designed by	Revision	Sheet
70 × 50 mm · 2 layers · 1 oz · 1.6 mm	**Dr. Singaravelan A**	**Rev B · Nov 2026**	**1 of 2**
Hand-solderable: 0805 passives (10/22 µF in 1206)	youtube.com/@DrSingaravelanA · AI + KiCad series, Part 2	Layout demo; not yet built	Parts list: sheet 2

ESP32-C3 DC Power Meter — Parts & layout rules

Exact part numbers and KiCad 10 footprints for every reference on sheet 1. Use only these parts; do not substitute look-alike symbols.

Parts list (36 parts: 32 electrical + 4 mounting holes)

Ref	Function / value	Exact part	KiCad footprint
U1	ESP32-C3 module	ESP32-C3-WROOM-02 (Espressif)	`RF_Module:ESP32-C3-WROOM-02`
U2	Current/power monitor	INA226AIDGSR (TI)	`Package_SO:MSOP-10_3x3mm_P0.5mm`
U3	3.3 V LDO	AMS1117-3.3	`Package_TO_SOT_SMD:SOT-223-3_TabPin2`
U4	USB ESD protection	USBLC6-2SC6 (ST)	`Package_TO_SOT_SMD:SOT-23-6`
J1	USB-C receptacle	HRO TYPE-C-31-M-12 (shield pads: SH)	`Connector_USB:USB_C_Receptacle_HRO_TYPE-C-31-M-12`
J2, J4	Screw terminal, 2-pin	Phoenix MKDS 1,5/2-5,08 or equivalent	`TerminalBlock_Phoenix:TerminalBlock_Phoenix_MKDS-1,5-2-5.08_1x02_P5.08mm_Horizontal`
J3	OLED header, 4-pin	2.54 mm pin header 1×4	`Connector_PinHeader_2.54mm:PinHeader_1x04_P2.54mm_Vertical`
RS1	5 mΩ shunt	5 mΩ ±1 %, 3 W, 2512	`Resistor_SMD:R_2512_6332Metric`
D2	TVS diode	SMBJ33A (unidirectional)	`Diode_SMD:D_SMB`
D1	LED	Red LED 0805	`LED_SMD:LED_0805_2012Metric`
SW1, SW2	Tactile switch (RESET, BOOT)	6 mm THT tactile switch	`Button_Switch_THT:SW_PUSH_6mm`
R1, R2	5.1 kΩ	0805, 1 %	`Resistor_SMD:R_0805_2012Metric`
R3, R4, R10, R11, R12	10 kΩ	0805, 1 %	`Resistor_SMD:R_0805_2012Metric`
R5, R6	10 Ω	0805, 1 %	`Resistor_SMD:R_0805_2012Metric`
R7, R8	4.7 kΩ	0805, 1 %	`Resistor_SMD:R_0805_2012Metric`
R9	1 kΩ	0805	`Resistor_SMD:R_0805_2012Metric`
C1, C3	10 µF	1206, ≥ 10 V, X5R/X7R	`Capacitor_SMD:C_1206_3216Metric`
C2	22 µF	1206, ≥ 10 V, X5R/X7R	`Capacitor_SMD:C_1206_3216Metric`
C4, C6, C7	100 nF	0805, X7R	`Capacitor_SMD:C_0805_2012Metric`
C5	1 µF	0805, X7R	`Capacitor_SMD:C_0805_2012Metric`
H1–H4 NEW	M3 mounting hole, no net	3.2 mm non-plated hole	`MountingHole:MountingHole_3.2mm_M3`

AI output (KiCad 10) — files and run order

1. **ESP32C3_DC_Power_Meter.net** — exact references, values, footprints, net names and pin numbers above; NC pins get no net. Import: File → Import → Netlist, link by reference.
2. **1_Board.py** — 70 × 50 mm outline, 2 layers / 1.6 mm, design rules, net classes from sheet 1 table F.
3. **2_Place.py** — places all 36 footprints by the rules on the right, nudges small parts off any overlap, prints a self-check. *Engineer reviews the placement here.*
4. **3_Copper.py** — IN+/OUT+ zones, GND pour, Kelvin tracks, bottom silkscreen, zone fill, self-check. All other routing: Freerouting.

Scripts: find footprints by scanning `board.GetFootprints()`; check that no pad lies on any script-drawn track before drawing it; end with a self-check report. Ctrl+S before each script; restart the PCB Editor before Freerouting and after any script error.

Layout rules

Board	70 × 50 mm rectangle, 2 layers, 1 oz copper, 1.6 mm FR-4
Mounting	H1–H4: M3 holes, centres 3.5 mm from each corner; no parts within 3.5 mm of a hole centre
Edges	J1 (USB-C) on the left short edge, mouth at the edge; J2 IN and J4 OUT on the right short edge, wire entries facing out; U1 antenna end on the bottom edge
Module	Position U1 by its body outline (F.Fab), not its courtyard — the courtyard includes the antenna keep-out. Use the footprint's own keep-out: no copper, parts or tracks on any layer
Power path	J2 → RS1 → J4 short; RS1 pad 1 (IN+) towards J2. IN+ / OUT+ as top-layer zones ≥ 4.7 mm wide, each covering only the outer half of its RS1 pad; GND pour on the bottom; J2-2 / J4-2 solid to the pour (no thermal spokes)
Sensing	Kelvin: R5 / R6 pad 1 faces RS1; tracks start at the inner half of the RS1 pads and run as a pair; U2 next to RS1 with pins 6–10 facing it; C7 between R5/R6 and U2
Decoupling	C4 on the bottom side, directly under U1 pad 1; C3 beside pad 1; C6 at U2 VS; C1 / C2 at U3
Support parts	R3 / C5 (EN), R11 (IO8), R4 (IO9) in a column beside the module pads, above the keep-out; SW1 / SW2 side by side along the top edge; D1 + R9 visible, not under the OLED
USB	U4 next to J1; J1 D+ / D− in line with U1 pads 13 / 14; USB_DP / USB_DN short and together
Net classes	Sheet 1 table F. 10 A is carried by zones and the pour, so IN+, OUT+, GND, +3V3 use 0.4 mm tracks (fit the 0.5 mm-pitch MSOP-10 pins); +5V 0.5 mm; SENSE, USB, others 0.25 mm
Design rules	Min track and clearance 0.15 mm; min drill 0.2 mm (check your fab's limits)
Assembly	All parts on the top side except C4 (bottom); hand soldering
Silkscreen	Bottom side: name, “AI + KiCad”, Rev B (and the YouTube URL only if it fits); off pads, holes and the antenna area; part references on the Fab layer

Before building hardware: confirm D2's clamping voltage at your expected surge current is below the INA226 VBUS absolute maximum (datasheets), check every footprint name in your KiCad library, and check the pin order and mounting height of your OLED module.