LIBRARY · REFERENCE GUIDE

READING A SCHEMATIC

A schematic is the circuit as a diagram. How to read its symbols, reference designators, and nets, so any board's design opens up.

ONE THOUSAND DRONES ENGINEERING TEAM · VERIFIED 2026-07

A schematic is the circuit drawn as a diagram: symbols for the parts, lines for the connections, and labels that name them. Read it in that order and any board's design opens up.

SYMBOLS

Each part is drawn as a standard symbol: a zigzag or rectangle for a resistor, two lines for a capacitor, a triangle-and-bar for a diode, a box for an integrated circuit. The symbol shows the pins and how they connect, not what the part physically looks like.

REFERENCE DESIGNATORS

Each symbol carries a unique reference designator, `R1`, `C3`, `U2`, that ties the symbol to one real part on the board and one line on the bill of materials. Assigning them is called annotation (KiCad). Find `U2` on the schematic and you know exactly which part and which footprint it is.

NETS

A wire, or a shared label, is a net: one electrical node. Every pin on the same net is connected. Power and ground are usually drawn as named labels rather than wires, so the sheet stays readable, and two pins with the same power label are joined even without a line between them.

- KiCad. Schematic Editor documentation (symbols, annotation and reference designators, nets). docs.kicad.org

▸ FUNDAMENTALS · READING A SCHEMATIC

SCHEMATIC SYMBOLS

SYMBOL	REFDES	NAME
	R	resistor
	C	capacitor
	D	diode
	L	inductor
	U	IC / chip

Each symbol carries a reference-designator letter: R resistor, C capacitor, D diode, L inductor, U chip.

SYMBOLS, REFERENCE DESIGNATORS, AND NETS: THE THREE THINGS TO READ ON ANY SCHEMATIC.

Open the schematic for a One Thousand Drones L1.01 board and the same three things are there: symbols for each part, reference designators tying them to the BOM, and named nets for power, ground, and the USB signals.

CHECKPOINT

1. A reference designator like U2 does what?

a. Ties one symbol to one real part and its BOM line

b. Sets the part's voltage

c. Names the net it connects to

ANSWER · A

Reference designators (R1, C3, U2) uniquely identify each part; assigning them is annotation.

2. A net on a schematic is what?

a. The outline of the board

b. One electrical node that connects every pin on it

c. A single part's package

ANSWER · B

A net is one node; wires and shared labels put pins on the same net.

3. How are power and ground usually drawn?

a. As long wires across the sheet

b. They are left off the schematic

c. As named labels, so the same label means the same net

ANSWER · C

Power and ground are drawn as labels to keep the sheet readable; the same label is the same net.

- Prerequisite: resistors
- See it on a real board: the L1.01 build
- Next: reading a datasheet