How To Draw Schematic Diagrams

Digital Electronics Laboratory Task Force

Abstract
This application note describes the importance of schematic diagrams. Do not share this document outside UTM as it may contain copyrighted materials.

Digital Electronics Lab
Electronics and Computer Engineering Department, Universiti Teknologi Malaysia

1. Overview

A circuit diagram, or a schematic diagram, is a technical drawing of how to connect electronic components by means of standard symbols. Schematics are our map to designing, building, and troubleshooting circuits. Understanding how to read and follow schematics is an important skill for any electronics engineer.

2. Basic Components

Here are some of the standardized, basic schematic symbols for various components.

2.1 Resistors

The most fundamental of circuit components and symbols! Resistors on a schematic are usually represented by a few zig-zag lines, with two terminals extending outward. Schematics using international symbols may instead use a featureless rectangle, instead of the squiggles.

Figure 1. *Physical resistor.*

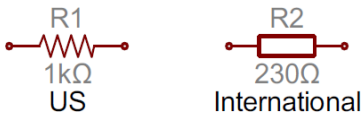

Figure 2. *Resistor schematic symbols.*

2.2 Potentiometers and Variable Resistors

Variable resistors and potentiometers each augment the standard resistor symbol with an arrow. The variable resistor remains a two-terminal device, so the arrow is just laid diagonally across the middle. A potentiometer is a three-terminal device, so the arrow becomes the third terminal (the wiper).

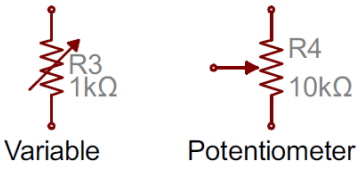

Figure 3. *Variable resistor symbols.*

2.3 Capacitors

There are two commonly used capacitor symbols. One symbol represents a polarized (usually electrolytic or tantalum) capacitor, and the other is for non-polarized caps. In each case there are two terminals, running perpendicularly into plates.

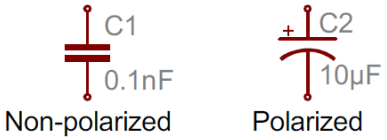

Figure 4. *Capacitors symbols.*

The symbol with one curved plate indicates that the capacitor is polarized. The curved plate represents the cathode of the capacitor, which should be at a lower voltage than the positive, anode pin. A plus sign might also be added to the positive pin of the polarized capacitor symbol.

2.4 Inductors

Inductors are usually represented by either a series of curved bumps, or loopy coils. International symbols may just define an inductor as a filled-in rectangle.

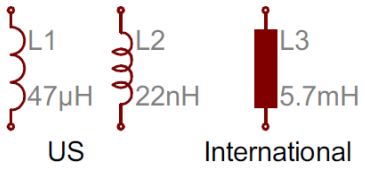

Figure 5. *Inductors symbols.*

2.5 Switches

Switches exist in many different forms. The most basic switch, a single-pole/single-throw (SPST), is two terminals

with a half-connected line representing the actuator (the part that connects the terminals together).

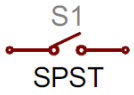

Figure 6. *Switch symbols.*

Switches with more than one throw, like the SPDT and SP3T below, add more landing spots for the the actuator.

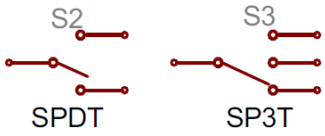

Figure 7. *SPDT and SP3T symbols.*

Switches with multiple poles, usually have multiple, alike switches with a dotted line intersecting the middle actuator.

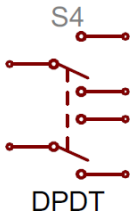

Figure 8. *DPDT symbol.*

2.6 Power Sources

Just as there are many options out there for powering your project, there are a wide variety of power source circuit symbols to help specify the power source.

2.6.1 DC or AC Voltage Sources

Most of the time when working with electronics, you'll be using constant voltage sources. We can use either of these two symbols to define whether the source is supplying direct current (DC) or alternating current (AC):

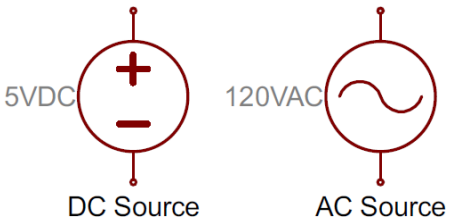

Figure 9. *Voltage source symbol.*

2.6.2 Batteries

Batteries, whether they're those cylindrical, alkaline AA's or rechargeable lithium-polymers, usually look like a pair of disproportionate, parallel lines:

Figure 10. *Battery Nodes.*

More pairs of lines usually indicates more series cells in the battery. Also, the longer line is usually used to represent the positive terminal, while the shorter line connects to the negative terminal.

Sometimes – on really busy schematics especially – you can assign special symbols to node voltages. You can connect devices to these one-terminal symbols, and it'll be tied directly to 5V, 3.3V, VCC, or GND (ground). Positive voltage nodes are usually indicated by an arrow pointing up, while ground nodes usually involve one to three flat lines (or sometimes a down-pointing arrow or triangle).

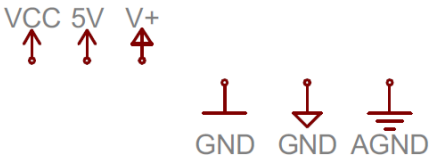

Figure 11. *Terminal symbol.*

2.7 Diodes

Basic diodes are usually represented with a triangle pressed up against a line. Diodes are also polarized, so each of the two terminals require distinguishing identifiers. The positive, anode is the terminal running into the flat edge of the triangle. The negative, cathode extends out of the line in the symbol (think of it as a - sign).

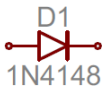

Figure 12. *Diode symbol.*

There are a all sorts of different types of diodes, each of which has a special riff on the standard diode symbol. Light-emitting diodes (LEDs) augment the diode symbol with a couple lines pointing away. Photodiodes, which generate energy from light (basically, tiny solar cells), flip the arrows around and point them toward the diode.

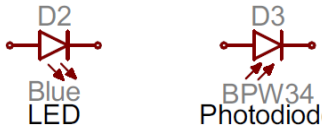

Figure 13. *LED and photodiode symbol.*

Other special types of diodes, like Schottky's or zeners, have their own symbols, with slight variations on the bar part of the symbol.

Figure 14. *Shottky and zener diode symbols.*

2.8 Transistors

Transistors, whether they're BJTs or MOSFETs, can exist in two configurations: positively doped, or negatively doped. So for each of these types of transistor, there are at least two ways to draw it.

2.8.1 Bipolar Junction Transistors (BJTs)

BJTs are three-terminal devices; they have a collector (C), emitter (E), and a base (B). There are two types of BJTs – NPNs and PNPs – and each has its own unique symbol.

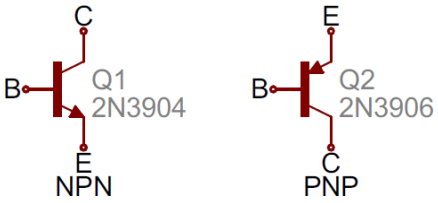

Figure 15. *Bipolar transistor symbols.*

The collector (C) and emitter (E) pins are both in-line with each other, but the emitter should always have an arrow on it. If the arrow is pointing inward, it's a PNP, and, if the arrow is pointing outward, it's an NPN. A mnemonic for remembering which is which is "NPN: not pointing in."

2.8.2 Metal Oxide Field-Effect Transistors (MOSFETs)

Like BJTs, MOSFETs have three terminals, but this time they're named source (S), drain (D), and gate (G). And again, there are two different versions of the symbol, depending on whether you've got an n-channel or p-channel MOSFET. There are a number of commonly used symbols for each of the MOSFET types:

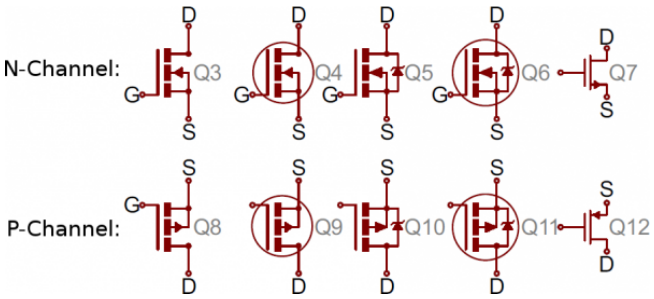

Figure 16. *MOSFET symbols.*

The arrow in the middle of the symbol (called the bulk) defines whether the MOSFET is n-channel or p-channel. If the arrow is pointing in means it's a n-channel MOSFET, and if it's pointing out it's a p-channel. Remember: "n is in" (kind of the opposite of the NPN mnemonic).

2.9 Digital Logic Gates

Standard logic functions – AND, OR, NOT, and XOR – all have unique schematic symbols:

Figure 17. *Standard logic gates.*

Adding a bubble to the output negates the function, creating NANDs, NORs, and XNORs:

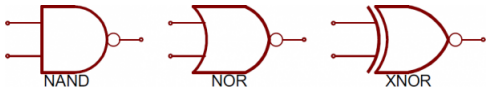

Figure 18. *Inverted logic gates.*

They may have more than two inputs, but the shapes should remain the same (well, maybe a bit bigger), and there should still only be one output.

2.10 Integrated Circuits

Integrated circuits accomplish such unique tasks, and are so numerous, that they don't really get a unique circuit symbol. Usually, an integrated circuit is represented by a rectangle, with pins extending out of the sides. Each pin should be labeled with both a number, and a function.

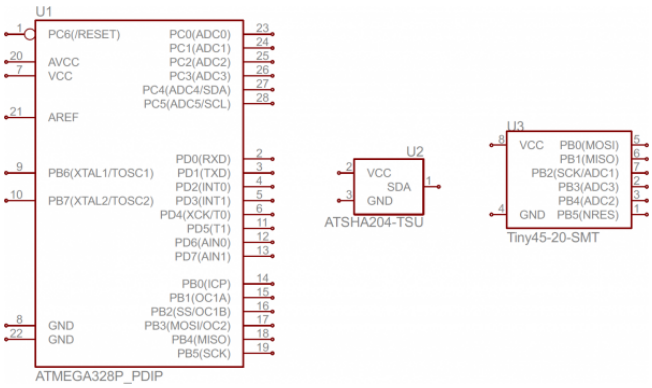

Figure 19. *ICs.*

Because ICs have such a generic circuit symbol, the names, values and labels become very important. Each IC should have a value precisely identifying the name of the chip.

3. Component Names and Values

The component symbols tell half the story, but each symbol must be given a name to complete it. Each component name on a schematic should be unique; if you have multiple resistors in a circuit, for example, they should be named R1, R2, R3, etc. [1].

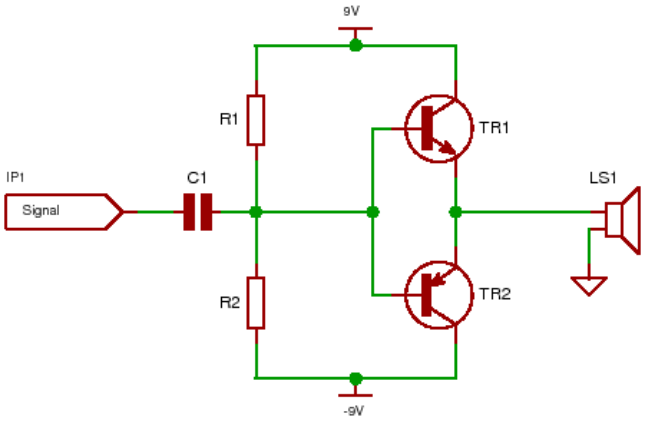

Figure 20. *Example of identifiers.*

Component names (a.k.a. *reference designator*) are usually a combination of one or two letters and a number. The letter part of the name (the *prefix*) identifies the type of component – R's for resistors, C's for capacitors, U's for integrated circuits, etc. The number uniquely identifies the component. The number is sometimes followed by a letter, indicating that components are grouped or matched with each other, e.g. R17A, R17B

The prefixes of names are pretty well standardized. For some components, like resistors, the prefix is just the first letter of the component. Other name prefixes are not so literal; inductors, for example, are L's. Here's a quick table of common components and their name prefixes:

Name Identifier	Component
R	Resistors
C	Capacitors
L	Inductors
S	Switches
D	Diodes
Q	Transistors
U	Integrated Circuits
Y	Crystals and Oscillators

Although theses name are "standardized", they're not universally followed. You might see integrated circuits prefixed with IC instead of U, for example, or crystals labeled as XTAL's instead of Y's. Use your best judgment in diagnosing which part is which. The symbol should usually convey enough information.

Values help define exactly what a component is. For schematic components like resistors, capacitors, and inductors the value tells us how many ohms, farads, or henries they have. For other components, like integrated circuits, the value may just be the name of the chip. Crystals might list their oscillating frequency as their value. Basically, the value of a schematic component calls out its most important characteristic.

Some integrated circuits contain multiple copies of the same device such as the 74HC00 which contains 4 NAND gates. The different gates are identified using a letter after the IC identifier. Optionally, the pin numbers for each connection are given as well.

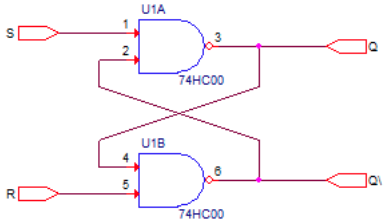

Figure 21. *Integrated circuit U1 has 4 gates, but only gates A and B are used.*

4. Connections

Understanding which components are which on a schematic is more than half the battle towards comprehending it. Now all that remains is identifying how all of the symbols are connected together.

4.1 Nets, Nodes and Labels

Schematic nets tell you how components are wired together in a circuit. Nets are represented as lines between component terminals. Sometimes (but not always) they're a unique color, like the green lines in this schematic:

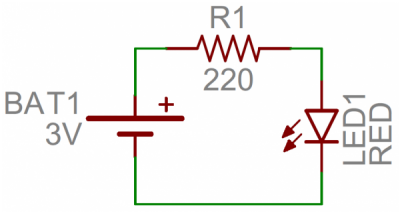

Figure 22. *Nets.*

4.2 Junctions

Circuit diagrams or schematic diagrams show electrical connections of wires or conductors by using a node as shown in the image below. A node is simply a filled circle or dot. When three or more lines touch each other or cross each other and a node is placed at the intersection, this represents the lines or wires being electrically connected at that point.

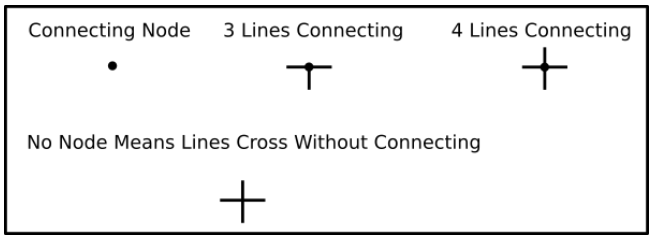

Figure 23. *Types of connections.*

If wires or lines cross each other and there is no node, as shown at the bottom of the above image, the wires are not electrically connected. In this case the wires are crossing each other without connecting, like two insulated wires placed one on top of the other.

4.3 Net Names

Sometimes, to make schematics more legible, we'll give a net a name and label it, rather than routing a wire all over the schematic. Nets with the same name are assumed to be connected, even though there isn't a visible wire connecting them. Names can either be written directly on top of the net, or they can be "tags", hanging off the wire.

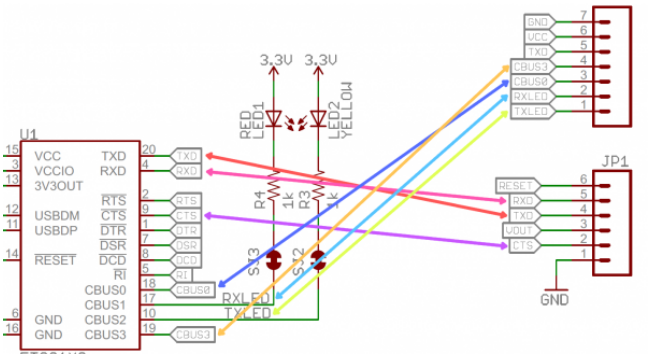

Figure 24. *Each net with the same name is connected, as in this schematic for an FT231X Breakout Board. Names and labels help keep schematics from getting too chaotic (imagine if all those nets were actually connected with wires).*

Nets are usually given a name that specifically states the purpose of signals on that wire. For example, power nets might be labeled "VCC" or "5V", while serial communication nets might be labeled "RX" or "TX".

5. Schematic Reading Tips

5.1 Identify Blocks

Truly expansive schematics should be split into functional blocks like Figure 27. There might be a section for power input and voltage regulation, or a microcontroller section, or a section devoted to connectors. Try recognizing which sections are which, and following the flow of circuit from input to output. Really good schematic designers might even lay the circuit out like a book, inputs on the left side, outputs on the right.

5.2 Recognize Voltage Nodes

Voltage nodes are single-terminal schematic components, which we can connect component terminals to in order to assign them to a specific voltage level. These are a special application of net names, meaning all terminals connected to a like-named voltage node are connected together.

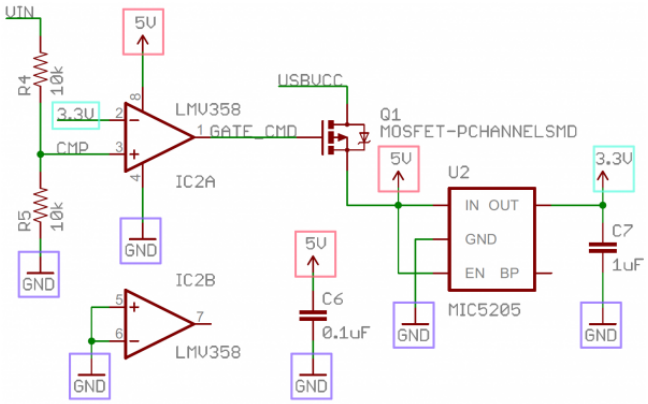

Figure 25. *Like-named voltage nodes – like GND, 5V, and 3.3V – are all connected to their counterparts, even if there aren't wires between them.*

5.3 Reference Component Datasheets

If there's something on a schematic that just doesn't make sense, try finding a datasheet for the most important component. Usually the component doing the most work on a circuit is an integrated circuit, like a microcontroller or sensor. These are usually the largest component, oft-located at the center of the schematic.

5.4 Some Circuit Diagram Rules

The following are general circuit diagram rules.

- A good schematic makes circuit functions clear. Therefore, keep functional areas distinct; don't be afraid to leave blank areas on the page, and don't try to fill the page. For example, flip-flops are usually drawn with clock and inputs on the left, set and clear on top and bottom, and outputs on the right [2].
- Component symbols in a circuit diagram are usually placed horizontally or vertically. On very rare occasions a component may be placed at 45 degrees, but only for a very good reason [2].
- Circuit diagrams are drawn as simply and neatly as possible. This means that the physical implementation of the circuit may look different to the circuit diagram, but they are electrically the same.

5.5 Tips

These tips are from [2].

- Identify parts immediately adjacent to the symbol, forming a distinct group giving symbol, label, and type or value.
- In general, signals go from left to right; don't be dogmatic about this, though, if clarity is sacrificed.
- Put positive supply voltages at the top of the page, negative at the bottom. Thus, npn transistors will usually have their emitter at the bottom, whereas pnp's will have their emitter topmost.

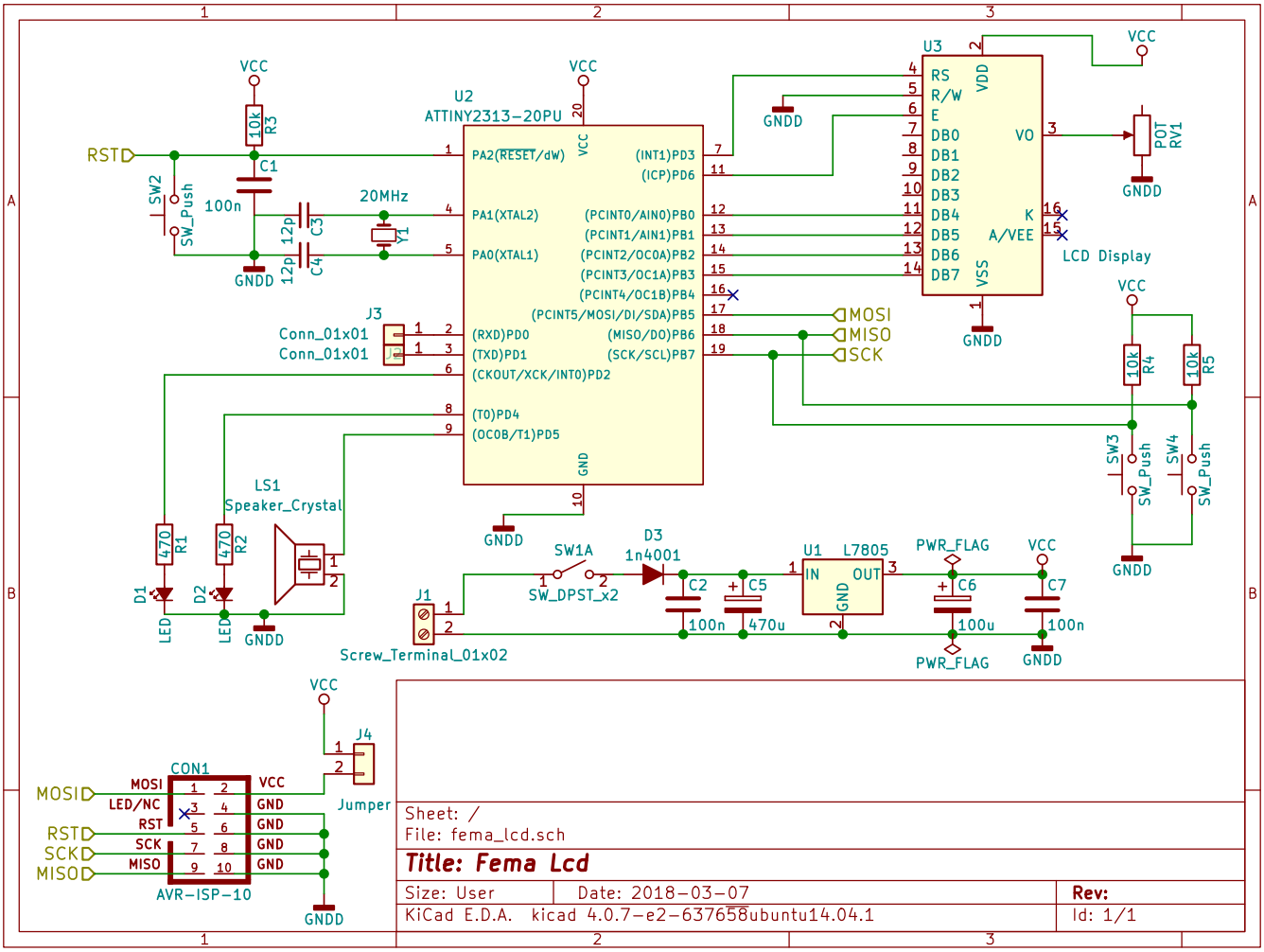

Figure 26. *A schematic drawn using open source KiCad.*

- Don't attempt to bring all wires around to the supply rails, or to a common ground wire. Instead, use the ground symbol(s) and labels like +Vcc to indicate those voltages where needed.
- It is helpful to label signals and functional blocks and show waveforms; in logic diagrams it is especially important to label signal lines, e.g., RESET' or CLK.
- It is helpful to bring leads away from components a short distance before making connections or jogs.
- Leave some space around circuit symbols; e.g., don't draw components or wires too close to an op-amp symbol. This keeps the drawing uncluttered and leaves room for labels, pin numbers, etc.
- Label all boxes that aren't obvious: comparator versus op-amp, shift register versus counter, etc. Don't be afraid to invent a new symbol.
- Use small rectangles, ovals, or circles to indicate card-edge connections, connector pins, etc. Be consistent.
- The signal path through switches should be clear. Don't force the reader to follow wires all over the page to find out how a signal is switched.
- Power-supply connections are normally assumed for op-amps and logic devices. However, show any unusual connections (e.g., an op-amp run from a single supply, where V- = ground) and the disposition of unused inputs.
- It is very helpful to include a small table of IC numbers, types, and power-supply connections (pin numbers for Vcc and ground, for instance).
- Include a title area near the bottom of the page, with name of circuit, name of instrument, by whom drawn, by whom designed or checked, date, and assembly number. Also include a revision area, with columns for revision number, date, and subject.

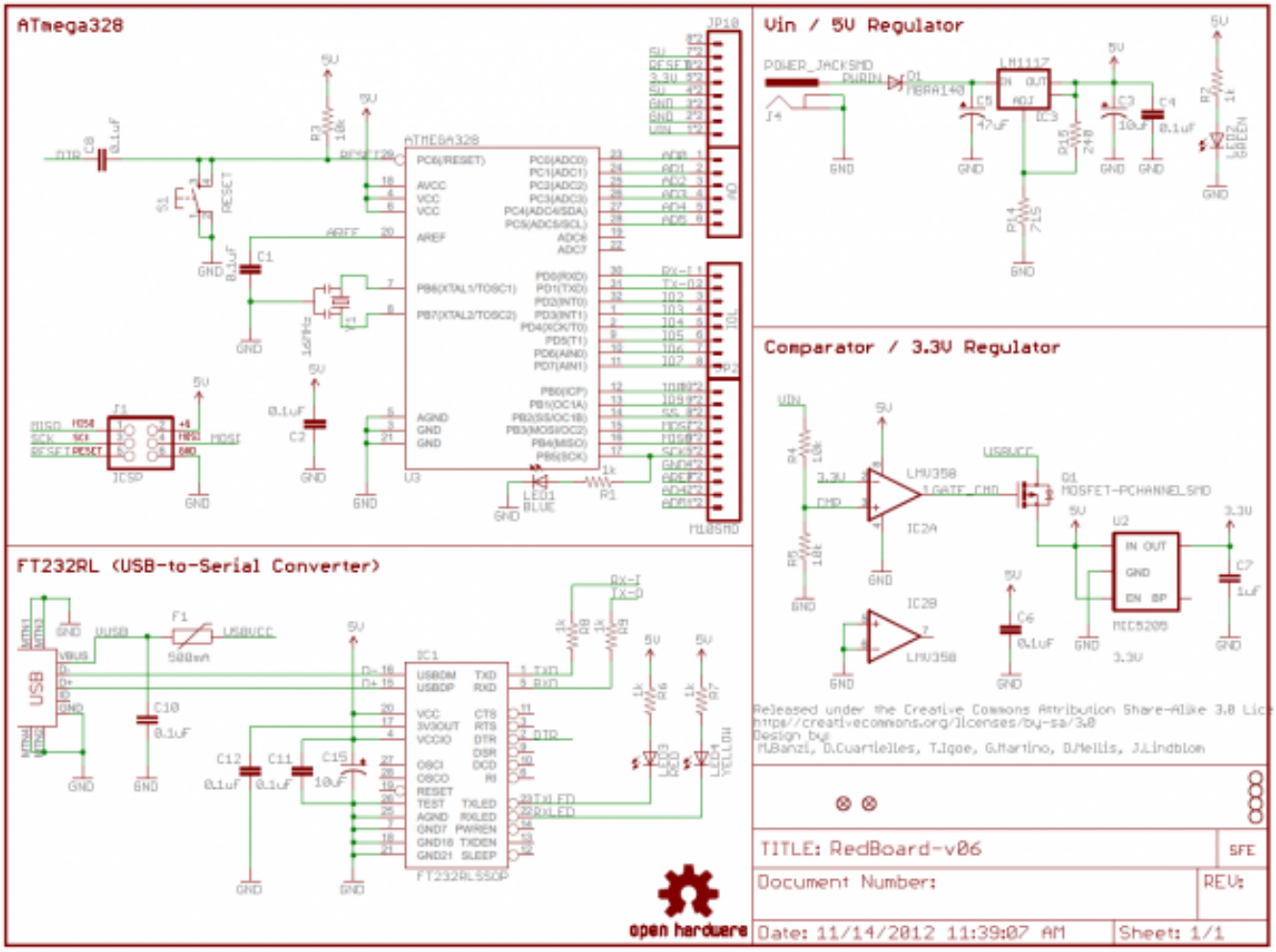

Figure 27. *A schematic diagram split into functional blocks.*

References

[1] American Society of Mechanical Engineers. *Reference Designations for Electrical and Electronics Parts and Equipment: Engineering Drawing and Related Documentation Practices: ASME Y14.8-2009 (Revision of ASME Y14.8m-1996 (R2008))*. American Society of Mechanical Engineers, 2008.

[2] Paul Horowitz and Winfield Hill. *How To Draw Schematic Diagrams*. 2nd ed. 1989. URL: http://opencircuitdesign.com/xcircuit/goodschem/goodschem.html.

[3] Munim Zabidi et al. *A Book on Digital Electronics*. Universiti Malaysia Pahang, 2019.

[4] Munim Zabidi, Izam Kamisian, and Ismahani Ismail. *The Art of Digital Design*. 2019.

[5] *EAGLE PCB Design Software*. Version 9.2. AutoDesk. 2018. URL: https://www.autodesk.com/products/eagle/free-download.

[6] *KiCad EDA*. Version 5.0.2. KiCad. 2018. URL: http://kicad-pcb.org/.

[7] Jim Lindblom. *How to Read a Schematic*. SparkFun. July 2013. URL: https://learn.sparkfun.com/tutorials/how-to-read-a-schematic.

[8] Jim Lindblom. *Using EAGLE: Schematic*. SparkFun. Sept. 2013. URL: https://learn.sparkfun.com/tutorials/using-eagle-schematic.

[9] Sam Sattel. *The Top 10 Tips to Draw Your Next Schematic Design Like a Pro*. Autodesk, 2019. URL: https://www.autodesk.com/products/eagle/blog/top-10-tips-draw-next-schematic-design-like-pro/.

[10] *How to Draw a Circuit Diagram using KiCad for Beginners*. Starting Electronics. Sept. 2017. URL: https://startingelectronics.org/beginners/draw-circuit-KiCad/.