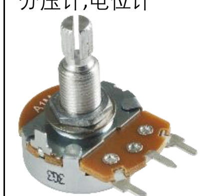
How to Read a Schematic of IC

1 IC Term

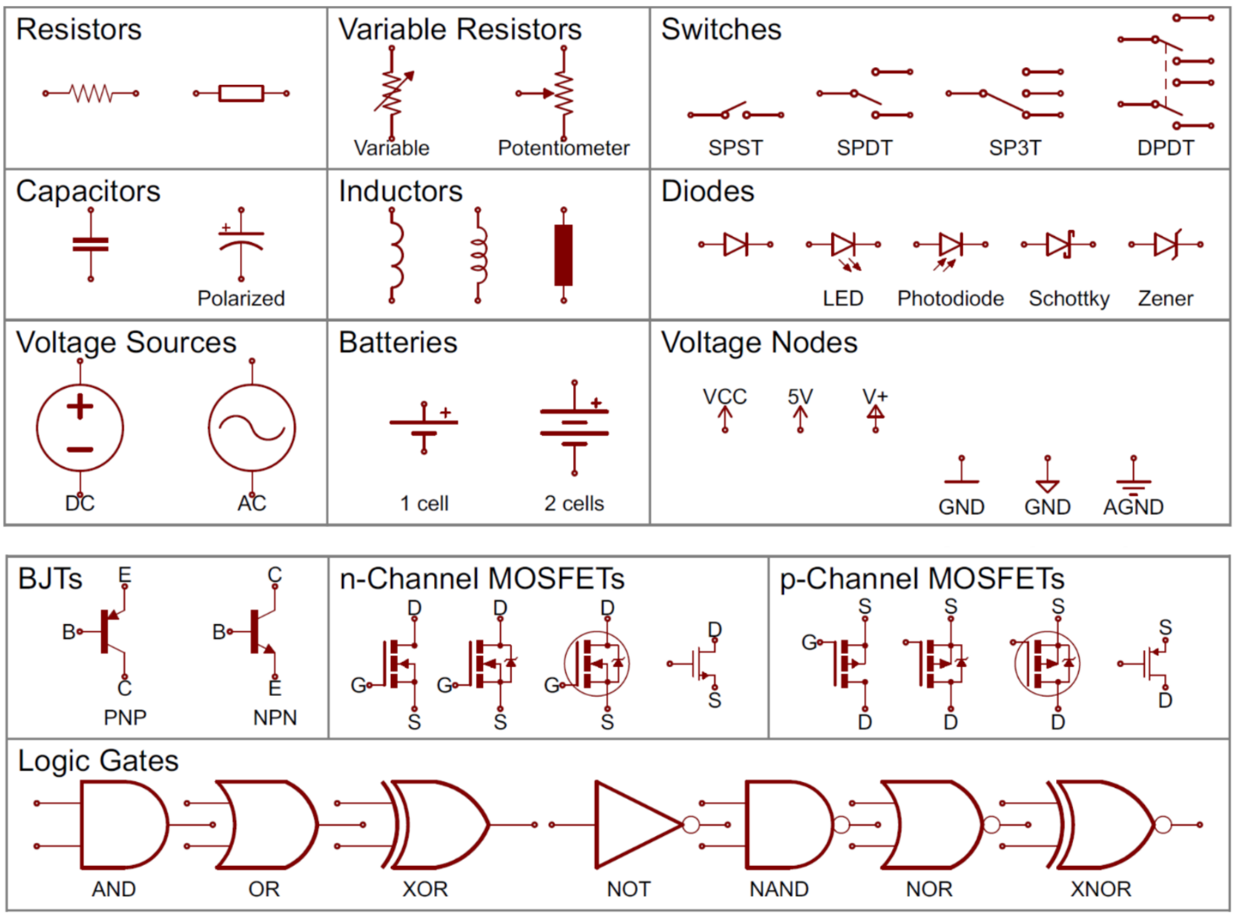
Integrated Circuit Terminology (Part 1)			
#	EN	UY	CN
1	circuit		电路
2	Integrated circuit (IC)		集成电路
3	schematic		原理图
4	two-terminal device three-terminal device		
5	Electric electronic		电气的 电子的
6	variable resistor potentiometer		可变电阻 分压计;电位计 
7	polarize		极化;两极分化
8	Cathode anode		
9	Inductors		电感
10	Transistors		三极管
11	operation amplifiers (Op Amps)		运算放大器
12	Voltage Regulators		电压调节器
13	Crystals		晶体
14	Resonators		谐振器
15	coils		线圈
16	Transformer		变压器
17	Oscillator		振荡器
18	Resonator		谐振器
19	Fuse		保险丝
20	thermistor		热敏电阻

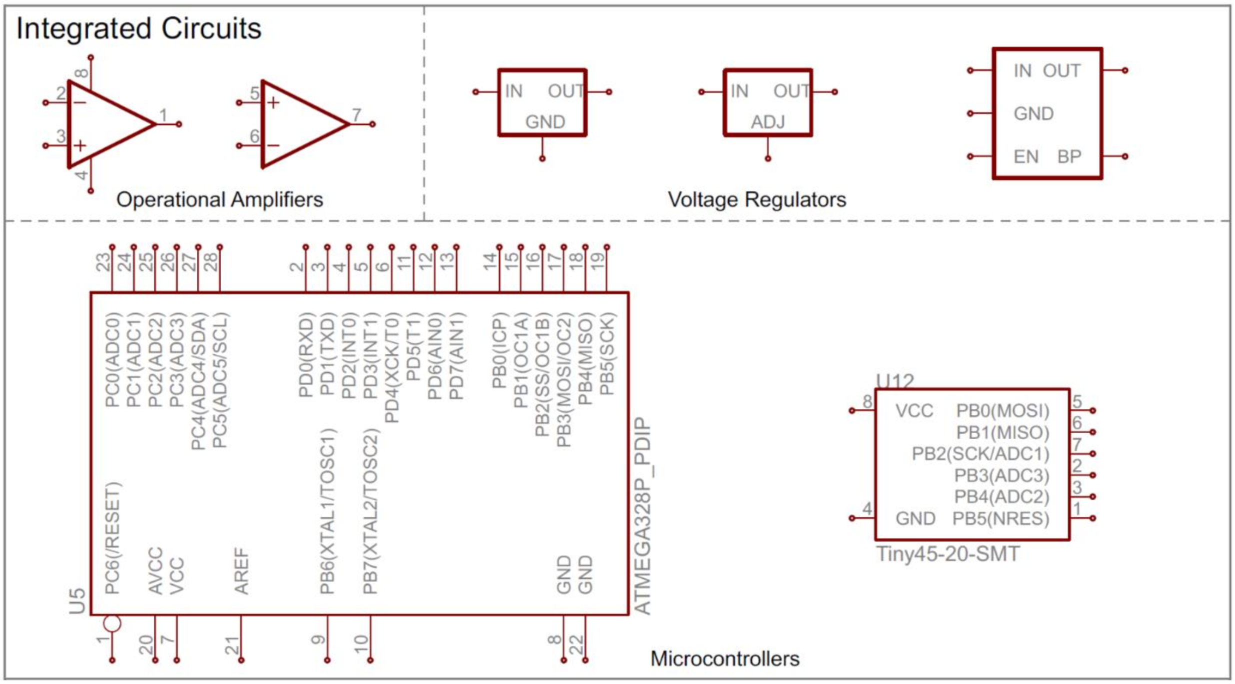
Integrated Circuit Terminology (Part 2)			
#	**EN**	**UY**	**CN**
21	Crystal		晶体
22	Designators		标志符 (Label)
23			

2 Overview

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.

The fundamental schematic symbols are very basic of electronics engineer as shown below.



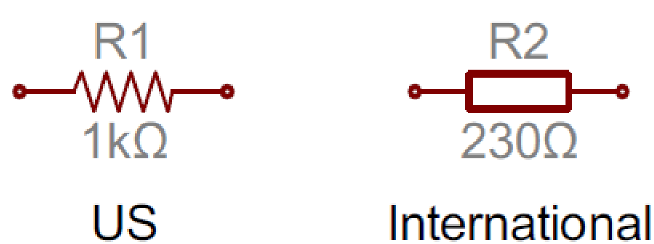


3 Schematic Symbols (Part 1)

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

3.1 Resistors

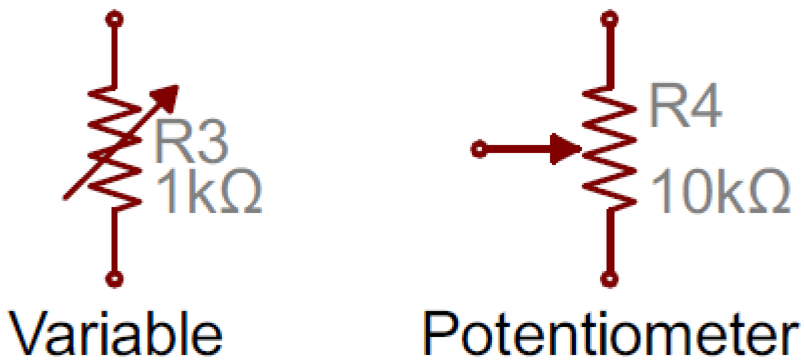
Resistor, the most fundamental of circuit components and symbols. Resistors on a schematic are usually represented by a few zig-zag [1] lines, with two terminals extending outward. Schematics using international symbols may instead use a featureless rectangle, instead of the squiggles [2].



References	
[1] zig-zag: 锯齿状的	[2] squiggle: 随意的线条或图案

3.1.1 Potentiometers and Variable Resistors

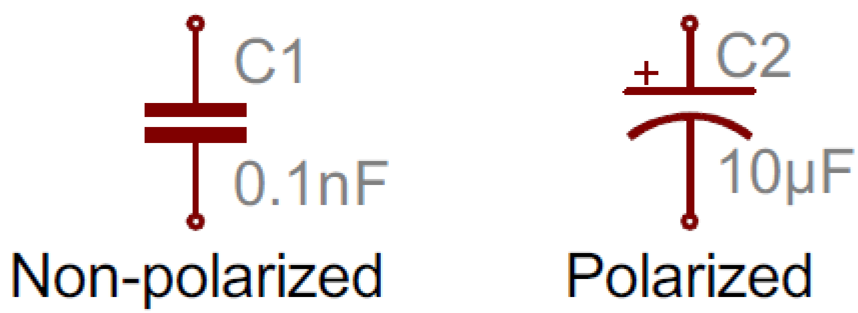
Variable resistors and **potentiometers** each augment the standard resistor symbol with an arrow. The variable resistor remains a two-terminal device, so the arrow is just laid diagonally [1] across the middle. A potentiometer is a three-terminal device, so the arrow becomes the third terminal (the wiper [2]).



Potentiometers are the most common type of variable resistors, and they have a wiper that can change the resistance value.

3.2 Capacitors

There are two commonly used capacitor symbols. One symbol represents a polarized (usually electrolytic [3] or tantalum, one of capacitor made by Chemical element Tantalum) capacitor, and the other is for non-polarized caps. In each case there are two terminals, running perpendicularly [4] into plates.

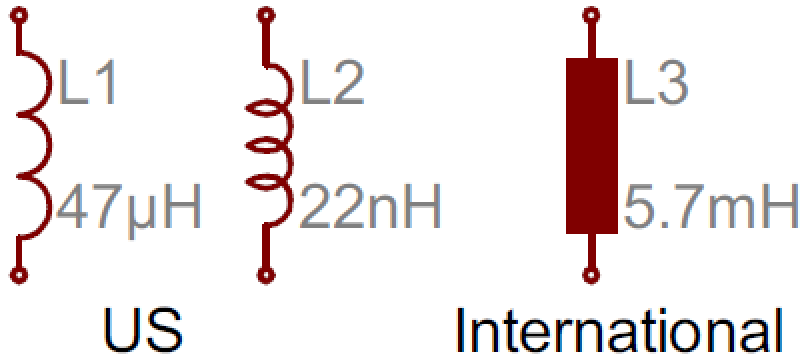


The symbol with one curved plate indicates that the capacitor is polarized. The curved plate usually represents the cathode of the capacitor, which should be at a lower voltage than the positive, anode pin. A plus sign should also be added to the positive pin of the polarized capacitor symbol.

References	
[1] diagonally: 斜对地	[3] electrolytic: 电解的
[2] wiper: 滑动片; 擦拭者	[4] perpendicularly: 垂直

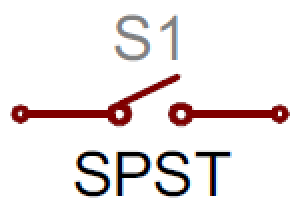
3.3 Inductors

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.

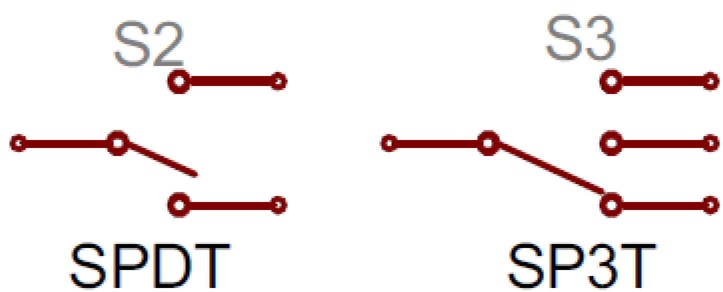


3.4 Switches

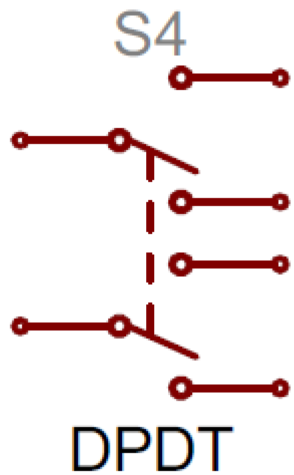
Switches exist in many different forms. The most basic switch, a single-pole/single-throw (SPST), is two terminals with a half-connected line representing the actuator (the part that connects the terminals together).



Switches with more than one throw, like the SPDT (single-pole/double-throw) and SP3T (single-pole/three-throw) below, add more landing spots for the actuator.



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

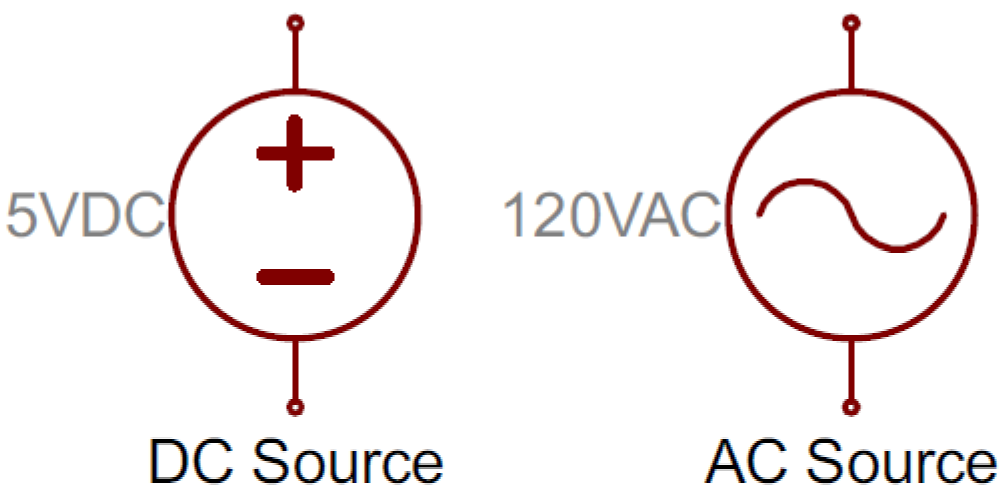


3.5 Power Sources

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.

3.5.1 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):

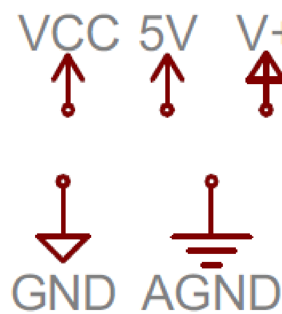


3.5.2 Batteries

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

More pairs of lines usually indicate 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.

3.5.3 Voltage Nodes

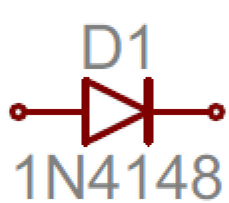


Sometimes, on really busy schematics especially, you can assign special symbols to node voltages. You can connect devices to these one-terminal symbols, and it'll be tied directly to 5V, 3.3V, VCC, or GND (ground). Positive voltage nodes are usually indicated by an arrow pointing up, while ground nodes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

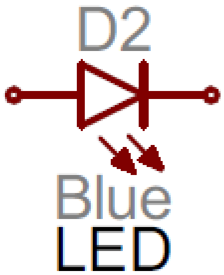
4 Schematic Symbols (Part 2)

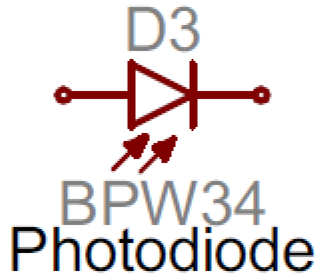
4.1 Diodes

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).

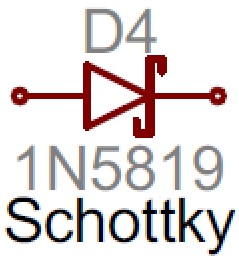


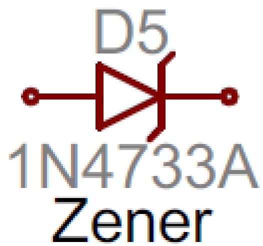
There are all sorts of different types of diodes, each of which has a special riff on the standard diode symbol. Light-emitting diodes (LEDs) augment the diode symbol with a couple lines pointing away. Photodiodes, which generate energy from light (basically, tiny solar cells), flip the arrows around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





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



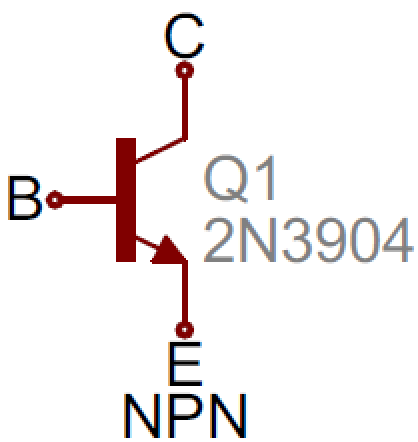


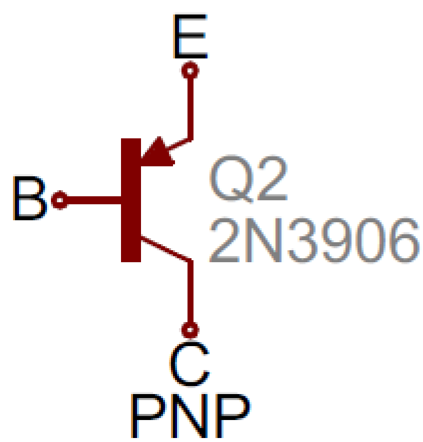
4.2 Transistors

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.

4.2.1 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.

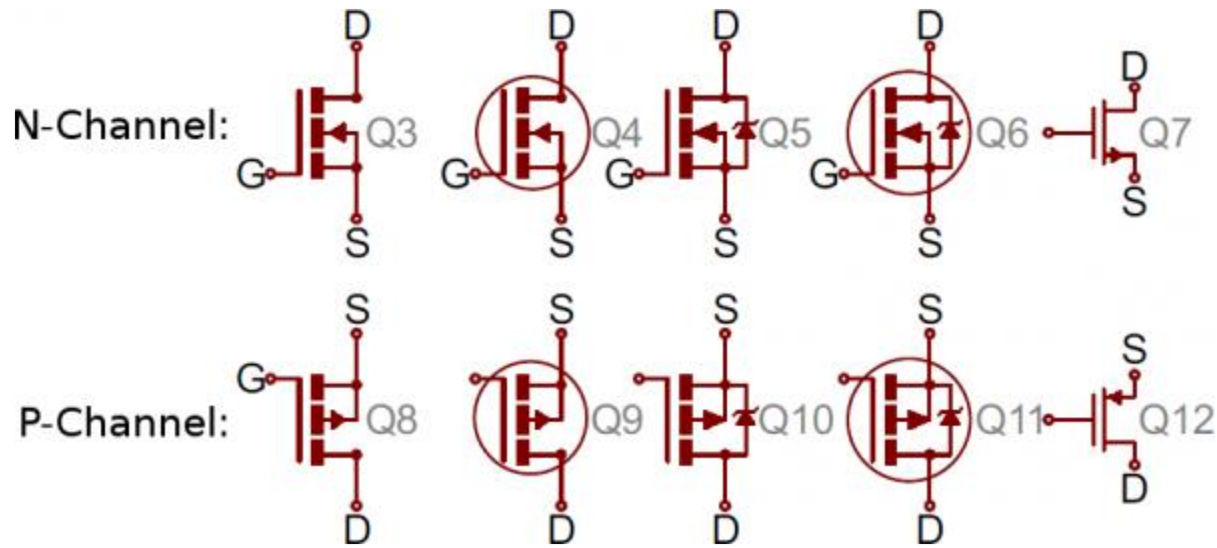




The collector (C) and emitter (E) pins are both in-line with each other, but the emitter should always have an arrow on it. If the arrow is pointing inward, it's a PNP, and, if the arrow is pointing outward, it's an NPN. A mnemonic for remembering which is which is "NPN: **n**ot **p**ointing **i**n."

4.2.2 Metal Oxide Field-Effect Transistors (MOSFETs)

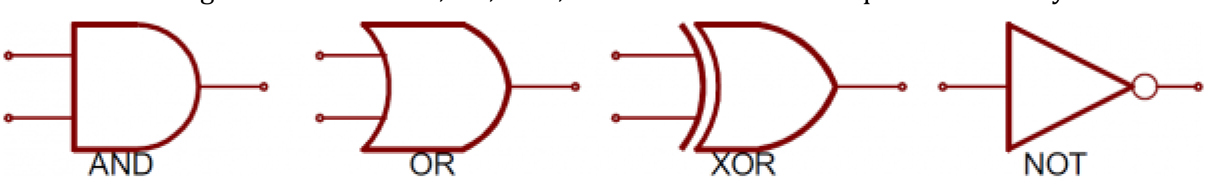
Like BJTs, MOSFETs have three terminals, but this time they're named source (S), drain (D), and gate (G). And again, there are two different versions of the symbol, depending on whether you've got an n-channel or p-channel MOSFET, separately calls NMOS, PMOS. There are a number of commonly used symbols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



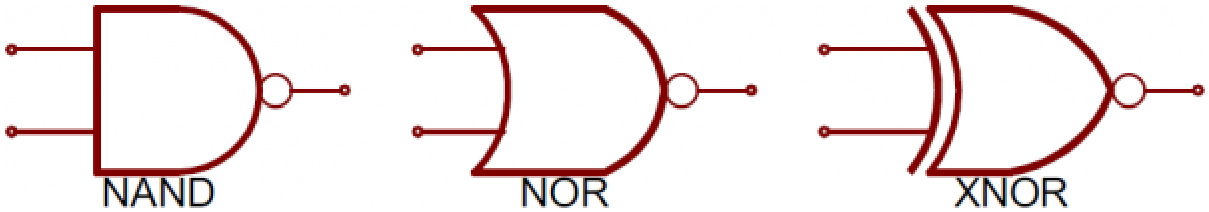
The arrow in the middle of the symbol (called the bulk) defines whether the MOSFET is n-channel or p-channel. If the arrow is pointing in means it's a n-channel MOSFET, and if it's pointing out it's a p-channel. Remember: "**n** is **in**" (kind of the opposite of the NPN mnemonic).

4.3 Digital Logic Gates

Our standard logic functions - AND, OR, NOT, and XOR -- all have unique schematic symbols:



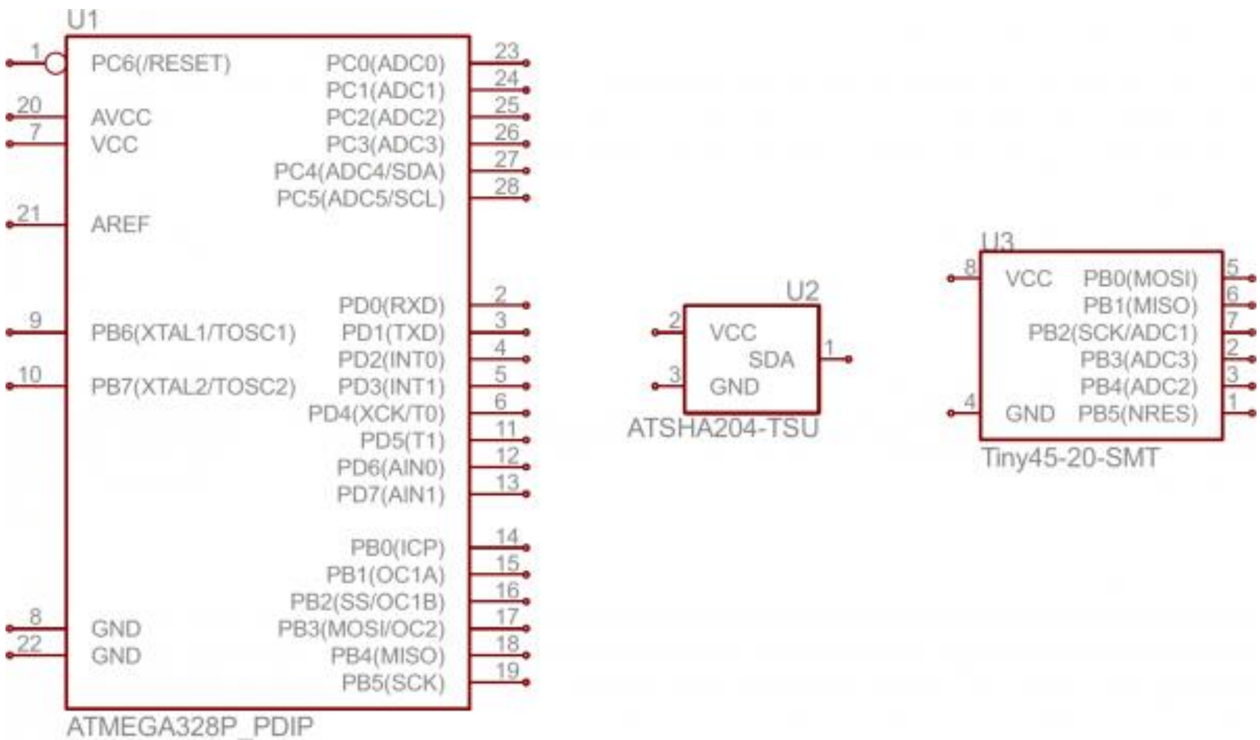
Adding a bubble to the output negates the function, creating NANDs, NORs, and XNORs:



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.

4.4 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.*



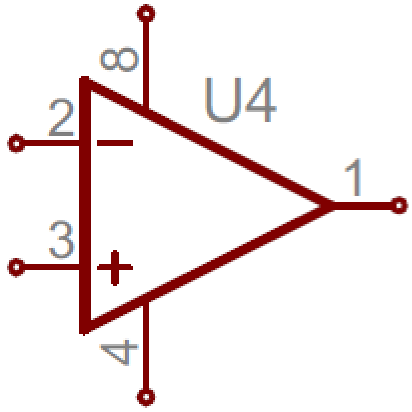
Schematic symbols for an ATmega328 microcontroller (commonly found on Arduinos), an ATSHA204 encryption IC, and an ATtiny45 MCU. As you can see, these components greatly vary in size and pin-counts.

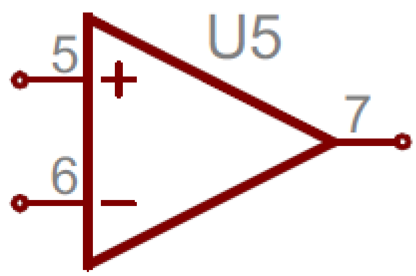
Because ICs have such a generic circuit symbol, the names, values and labels become very important. Each IC should have a value precisely identifying the name of the chip.

4.5 Unique ICs: Op Amps, Voltage Regulators

1. Op Amps (Operational Amplifier)

Some of the more common integrated circuits do get a unique circuit symbol. You'll usually see **operation amplifiers** laid out like below, with 5 total terminals: a non-inverting input (+), inverting input (-), output, and two power inputs.





Often, there will be two op amps built into one IC package requiring only one pin for power and one for ground, which is why the one on the right only has three pins.

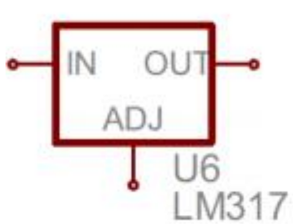
For more details about Op Amps reads:

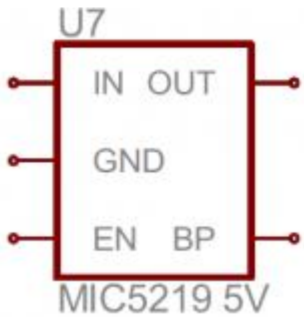
[1] https://www.allaboutcircuits.com/textbook/semiconductors/chpt-8/introduction-operational-amplifiers/

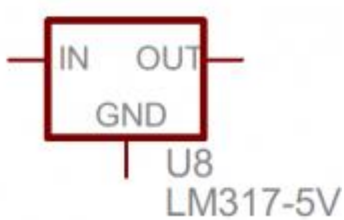
[2] https://www.electronics-tutorials.ws/opamp/opamp_1.html

2. Voltage Regulators

Simple **voltage regulators** are usually three-terminal components with input, output and ground (or adjust) pins. These usually take the shape of a rectangle with pins on the left (input), right (output) and bottom (ground/adjust).







The Voltage Regulators mission is that ensures a steady constant voltage supply through all operational conditions. It regulates voltage during power fluctuations and variations in loads. It can regulate AC as well as DC voltages.

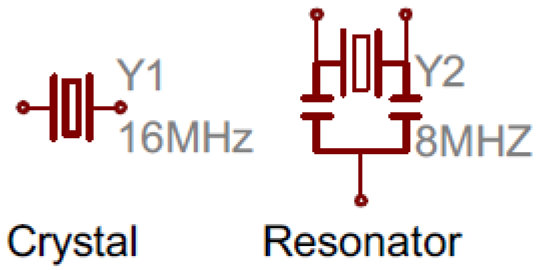
For more details about Voltage Regulators reads:

[1] https://www.digikey.com/en/maker/tutorials/2020/what-is-a-voltage-regulator

4.6 Miscellany

4.6.1 Crystals and Resonators

Crystals or resonators are usually a critical part of microcontroller circuits. They help provide a clock signal. Crystal symbols usually have two terminals, while resonators, which add two capacitors to the crystal, usually have three terminals.



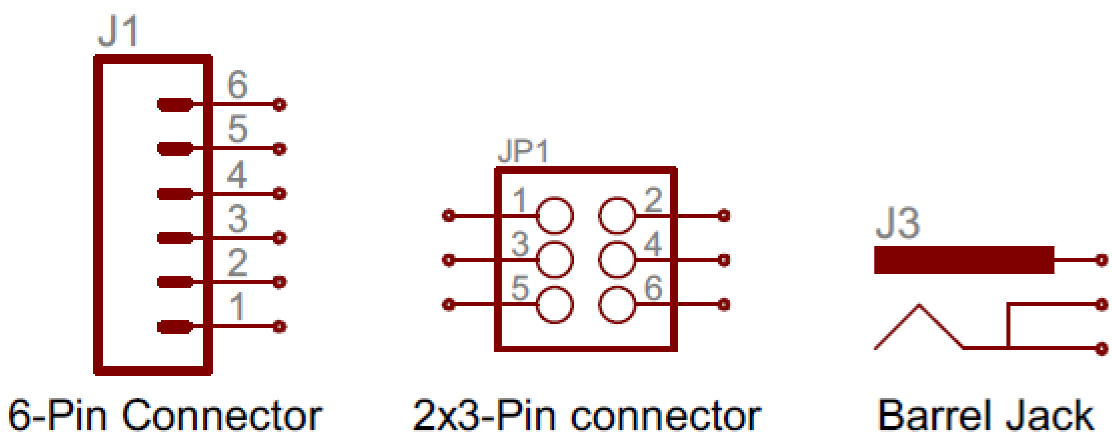
For more details about Resonators reads:

[1] https://www.genuway.com/7094.html

[2] https://www.electrical4u.com/what-is-an-oscillator/

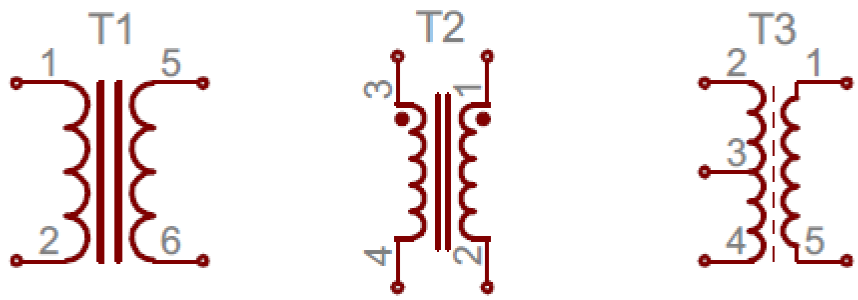
4.6.2 Headers and Connectors

Whether it's for providing power, or sending out information, connectors are a requirement on most circuits. These symbols vary depending on what the connector looks like, here's a sampling:

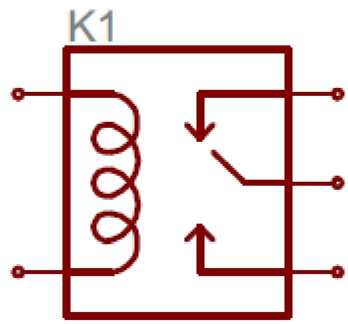


4.6.3 Motors, Transformers, Speakers, and Relays

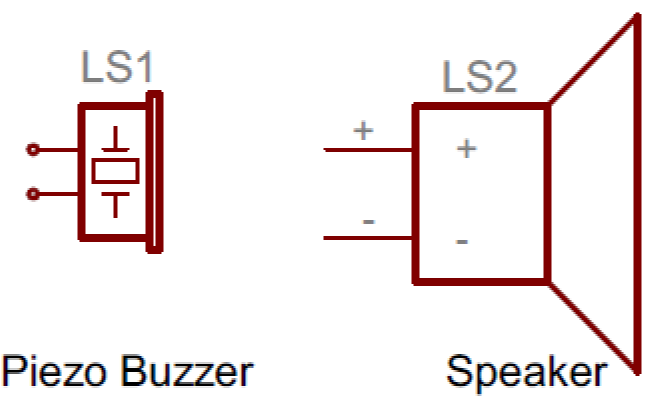
We'll lump these together, since they (mostly) all make use of coils in some way. **Transformers** (not the more-than-meets-the-eye kind) usually involve two coils, butted up against each other, with a couple lines separating them:



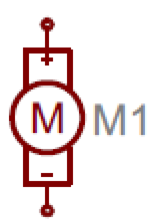
Relays usually pair a coil with a switch:



Speakers and **buzzers** usually take a form similar to their real-life counterparts:



And **motors** generally involve an encircled "M", sometimes with a bit more embellishment around the terminals:



4.6.4 Fuses and PTCs

Fuses and PTCs, devices which are generally used to limit large inrushes of current, each have their own unique symbol:

The PTC symbol is actually the generic symbol for a **thermistor**, a temperature-dependent resistor (notice the international resistor symbol in there?).

No doubt, there are many circuit symbols left off this list, but those above should have you 90% literate in schematic reading. In general, symbols should share a fair amount in common with the real-life components they model. *In addition to the symbol, each component on a schematic should have a unique name and value, which further helps to identify it.*

5 Name Designators and Values

One of the biggest keys to being schematic-literate is being able to recognize which components are which. *The component symbols tell half the story, but each symbol should be paired with both a name and value to complete it.*

5.1 Names and Values

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.

Component names are usually a combination of one or two letters and a number. The letter part of the name identifies the type of component, R's for resistors, C's for capacitors, U's for integrated circuits, etc. 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. Component names help us reference specific points in schematics.

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 (because current has already taken I [but it starts with a C electronics is a silly place]). 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

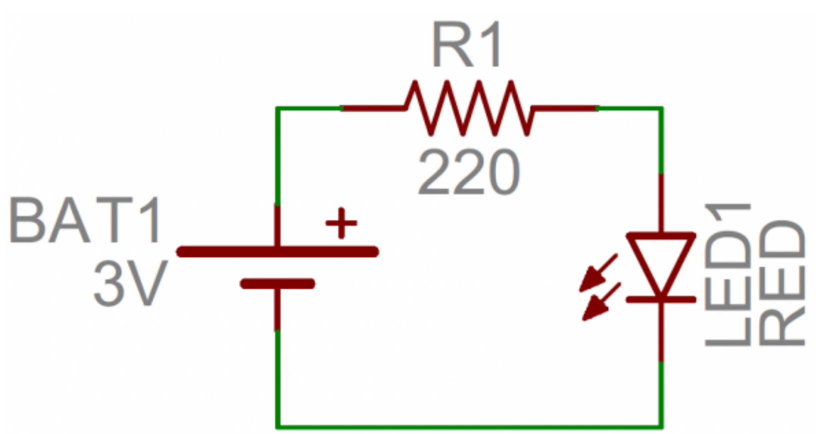
Although these are the "standardized" names for component symbols, they're not universally followed. You might see integrated circuits prefixed with IC instead of U, for example, or crystals labeled as XTAL's instead of Y's. Use your best judgment in diagnosing which part is which. The symbol should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

6 Reading Schematics

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.

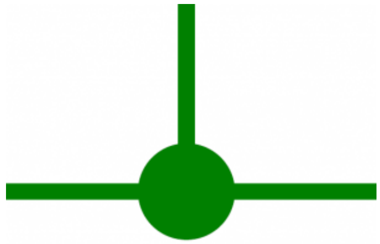
6.1 Nets, Nodes and Labels

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:

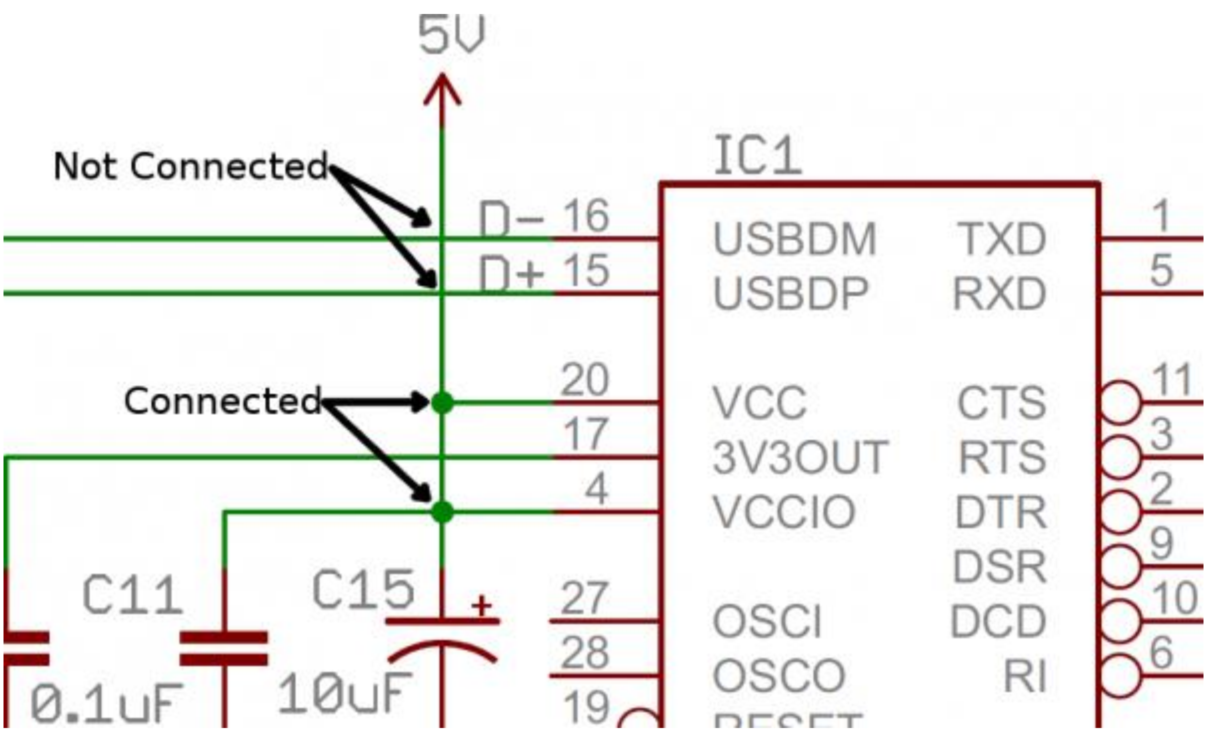


6.1.1 Junctions and Nodes

Wires can connect two terminals together, or they can connect dozens. When a wire splits into two directions, it creates a **junction**. We represent junctions on schematics with **nodes**, little dots placed at the intersection of the wires.

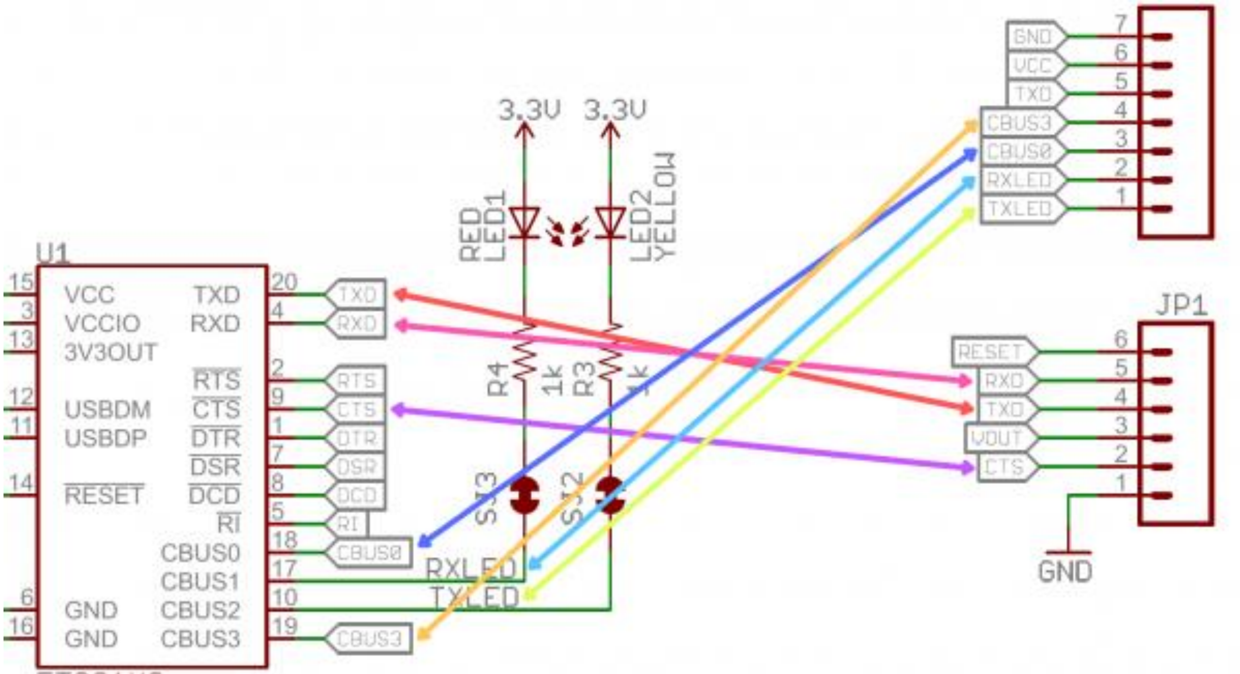


Nodes give us a way to say that "wires crossing this junction are connected". The absences of a node at a junction means two separate wires are just passing by, not forming any sort of connection. (When designing schematics, it's usually good practice to avoid these non-connected overlaps wherever possible, but sometimes it's unavoidable).



6.1.2 Net Names

<u>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.</u> Names can either be written directly on top of the net, or they can be "tags", hanging off the wire.



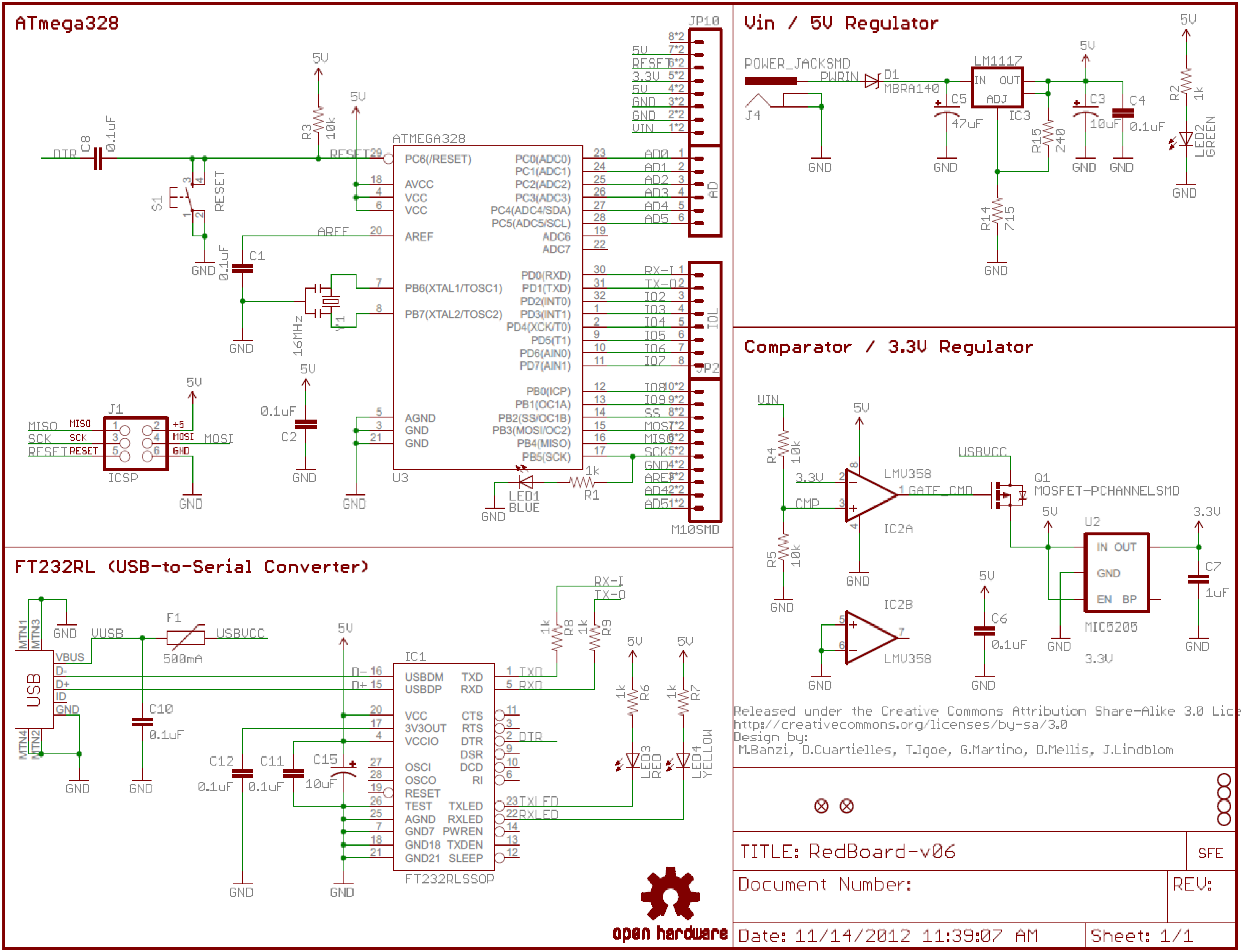
Each net with the same name is connected, as in this schematic for an FT231X Breakout Board. Names and labels help keep schematics from getting too chaotic (imagine if all those nets were actually connected with wires).

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".

6.2 Schematic Reading Tips

6.2.1 Identify Blocks

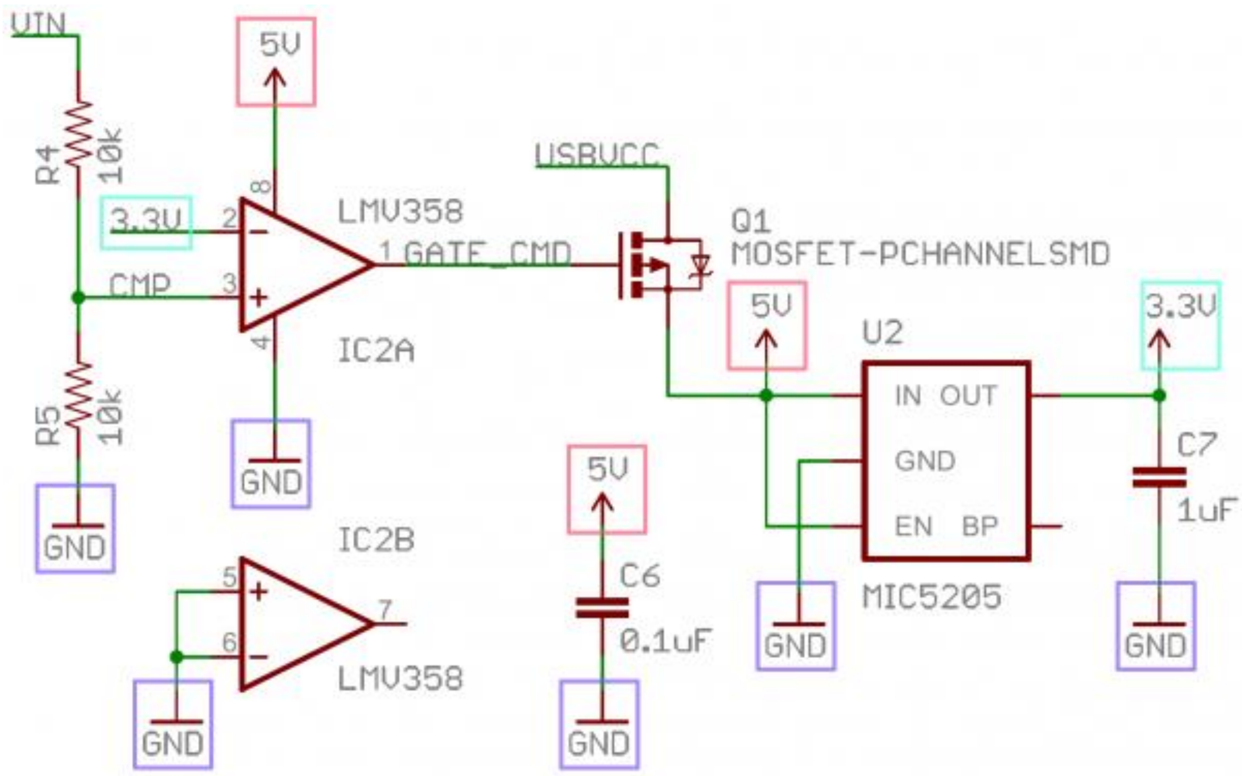
Truly expansive schematics should be split into functional blocks. There might be a section for power input and voltage regulation, or a microcontroller section, or a section devoted to connectors. Try recognizing which sections are which, and following the flow of circuit from input to output. Really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


ATmega328
ATMEGA328
U3
JP10
M10SMD
Vin / 5V Regulator
POWER_JACKSMD
PWRIN
D1
MBRA140
J4
LM1117
IN OUT
ADJ
IC3
C5 47uF
C3 10uF
C4 0.1uF
R15 240
R14 715
R2 1k
LED2 GREEN
Comparator / 3.3V Regulator
VIN
R4 10k
R5 10k
3.3V
CMP
LMV358
IC2A
IC2B
USBVCC
GATE_CMD
Q1
MOSFET-PCHANNELSMD
U2
IN OUT
GND
EN BP
MIC5205
C6 0.1uF
C7 1uF
FT232RL (USB-to-Serial Converter)
USB
VBUS
D-
D+
ID
GND
MTN1
MTN3
MTN4
MTN2
F1
500mA
VUSB
C10 0.1uF
C12 0.1uF
C11 0.1uF
C15 10uF
IC1
FT232RLSSOP
R8 1k
R9 1k
R6 1k
R7 1k
LED3 RED
LED4 YELLOW
RX-I
TX-O
DTR
open hardware
ICSP
J1
MISO
SCK
RESET
MOSI
C8 0.1uF
R3 10k
S1
C1 0.1uF
Y1
16MHz
C2 0.1uF
LED1 BLUE
R1 1k
Released under the Creative Commons Attribution Share-Alike 3.0 Lic
http://creativecommons.org/licenses/by-sa/3.0
Design by:
M.Banzi, D.Cuartielles, T.Igoe, G.Martino, D.Mellis, J.Lindblom
TITLE: RedBoard-v06
SFE
Document Number:
REV:
Date: 11/14/2012 11:39:07 AM
Sheet: 1/1

If the drawer of a schematic is really nice (like the engineer who designed this schematic for the RedBoard), they may separate sections of a schematic into logical, labeled blocks.

6.2.2 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.



Like-named voltage nodes, such as GND, 5V, and 3.3V, are all connected to their counterparts, even if there aren't wires between them.

The ground voltage node is especially useful, because so many components need a connection to ground.

6.2.3 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, often located at the center of the schematic.

THIS WHOLE DOC REFERENCED BY https://learn.sparkfun.com/tutorials/how-to-read-a-schematic.