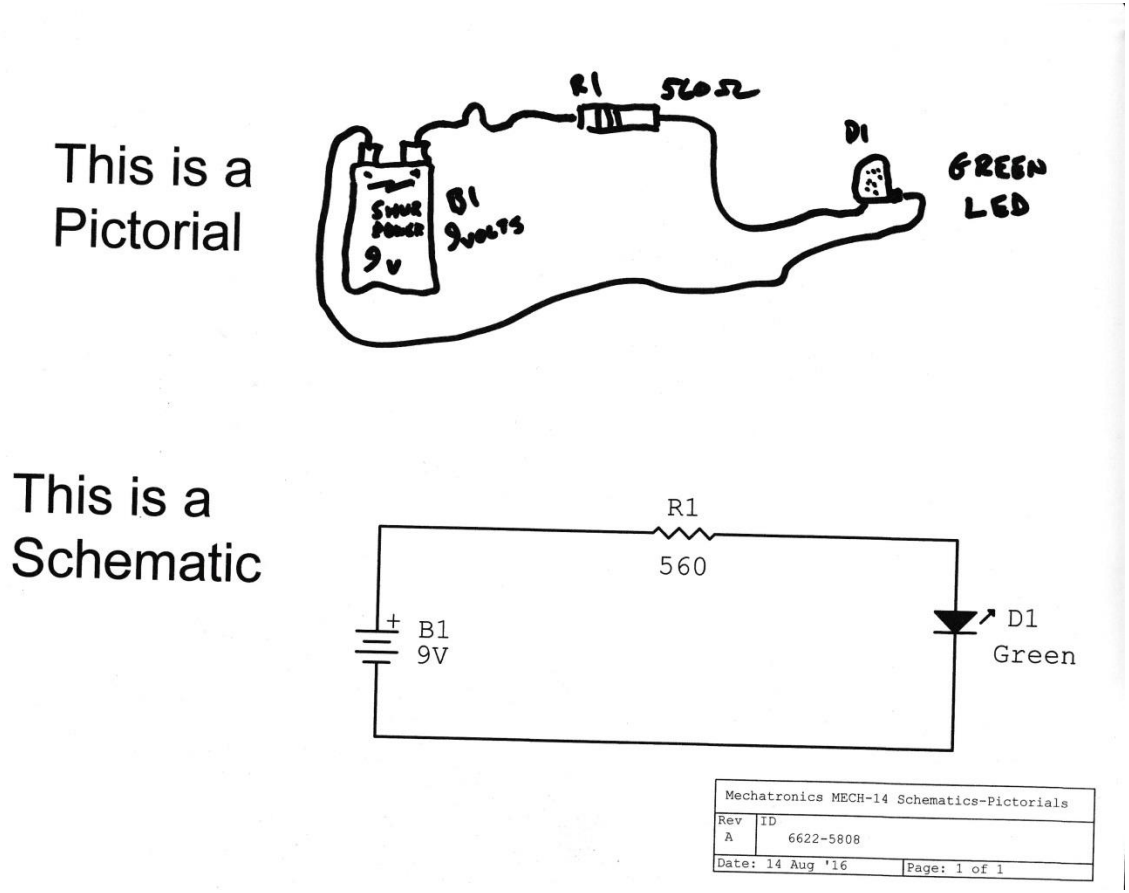
What is a schematic?

There are two ways to make a "picture" of an electronic circuit. One of them is a "pictorial" where you actually draw pictures of the parts and the wires that connect them together. The other is a "schematic" where each part is represented by a symbol that vaguely resembles the part, but is much more compact and easier to understand.

Below is the image of a very simple LED flashlight. It only has three parts; a 9 volt battery, a quarter-watt resistor, and a green light emitting diode ("LED")



While a pictorial is pretty easy for something as small as this flashlight, just think about what a pictorial would look like for something like your class computer. Or even something as relatively simple as a MECH-14 power supply.

Another good reason for using schematics instead of pictorials is that the 9 volt transistor radio battery I used for the illustration above is neatly represented by the small 4-line 9v battery symbol in the schematic. But what if I wanted to use a battery as big as a car battery or as small as a wristwatch battery? Size alone would make this a difficult to impossible task. Each and every part would have to have its own symbol and the symbols library would be huge and unwieldy. The schematic symbol is the same size for a battery the size of an Olympic swimming pool or the size of a match head.

There needs to be some sort of standard way to call out parts on a schematic. You will notice that I called the battery "B1", which is the first battery on the schematic. Likewise resistor R1 and LED (light emitting Diode) D1. Here is the standard listing for schematic symbols:

A	Antenna
B	Battery, including single cells
C	Capacitor, including variable capacitors
D	Diode, including multiple diodes in one package (bridge rectifiers), reference diode (zener), variable capacitance diode, and light emitting diode
E	(not used)
F	Fuse
G	Ground
H	(not used)
I	Lamp or Display, other than light emitting diode
J	Jack. For power connectors, the female half of a connector set as defined by the shell. For RF connectors, the female half of a connector set as defined by the center pin.
K	Relay
L	Inductor, including speaker and earphone driver
M	Meter or Motor
N, O	(not used)
P	Plug. For power connectors, the male half of a connector set as defined by the shell. For RF connectors, the male half of a connector set as defined by the center pin.
Q	Transistor, including field effect, thyristors, triac, and SCR
R	Resistor, including variable resistor
S	Switch, mechanical
T	Transformer
U	Integrated Circuit
V	Vacuum Tube
W	Wire
X	Otherwise Undefined
Y	Crystal, including crystal filter
Z	Ferrite

There are a few designers who refuse to use the standard symbols, so in some oddball schematics you may see such nonstandard terms as CR (Crystal Rectifier) instead of the standard D (Diode), or ZD for Zener Diode instead of letting the symbol for a zener diode tell you what kind of diode it is. You may also see in some British or German schematics different symbols for components, like a rectangle instead of jagged lines for a resistor.

Back in the old days (BC – Before Computers) any large company worth its salt had a HUGE room full of people sitting at large drawing tables (called drafting tables) with pen and ink, templates, white gloves (to keep from smearing the ink), ammonia (to erase ink before it completely dried) and very sour dispositions when they spilled ink on a drawing that had taken all week to produce. They were making schematic drawings of the company products, and woe be the engineer who took a first look at a drawing and said, "You know, this would work better if this resistor went to this point over here instead of where I first put it." Many a young engineer was told in no uncertain terms where he could "put that resistor".

At any rate (you knew this was headed somewhere, didn't you?) the advent of the computer relegated the pen and ink artist to a byte twiddler. In my experience, except for the preliminary SKETCHES of a design, there hasn't been a formal pen and ink schematic (or mechanical drawing, for that matter) made in the last 10 years.

Most schematic drafting packages are combined with another routine called a "layout" package. That is, once the circuit is laid out on a schematic page, the "cadastral delineator" (you KNEW a draftsperson wanted another title when they got computer-smart, didn't you?) simply pushes a button, and out the other end of the computer pops a full circuit board artwork all ready to go to the camera.