



Schematic Design

A very introductory introduction



What is a schematic?

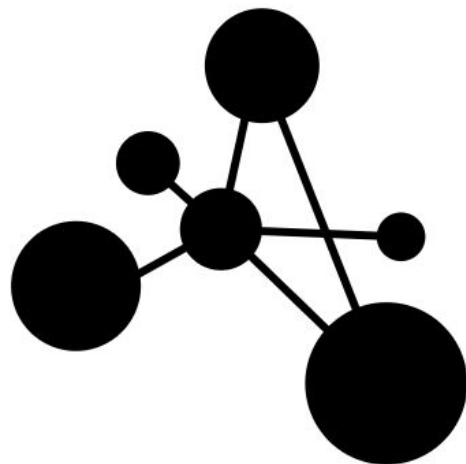
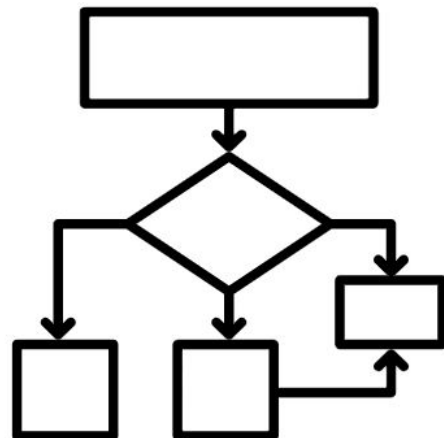
A schematic represents the **logical connections** between components

It represents **nets** connecting rather than the physical connection

Net - One connection point of a circuit

Ex) GND, 5V, 3V, a reset line

Everything in the net is connected to each other






Terminology Review

Net - A point of connection, can connect two or more pins

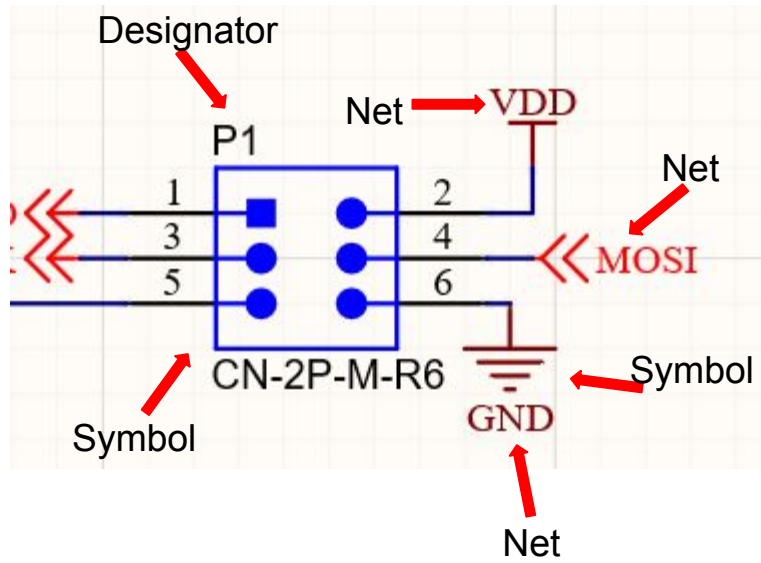
Ex) GND is a net, 5V, TX, RX, CLK

Symbol - A graphical representation of a component

Ex) 

Designator - The letter and number referencing a single part

Ex) R for resistor, C for capacitor, J for header, U for IC





Organization - Pages

Schematics can be on multiple pages (sheets) if large

Small, simple projects - one page

Title page for larger projects - define intent of board, table of contents, etc

MCU Page, Headers page

Nets must connect across pages

- Can use offsheet connectors or adjust settings

Altium defaults to creating “rooms” for each page separately





Hierarchical Design

Block diagram of all the components of the system

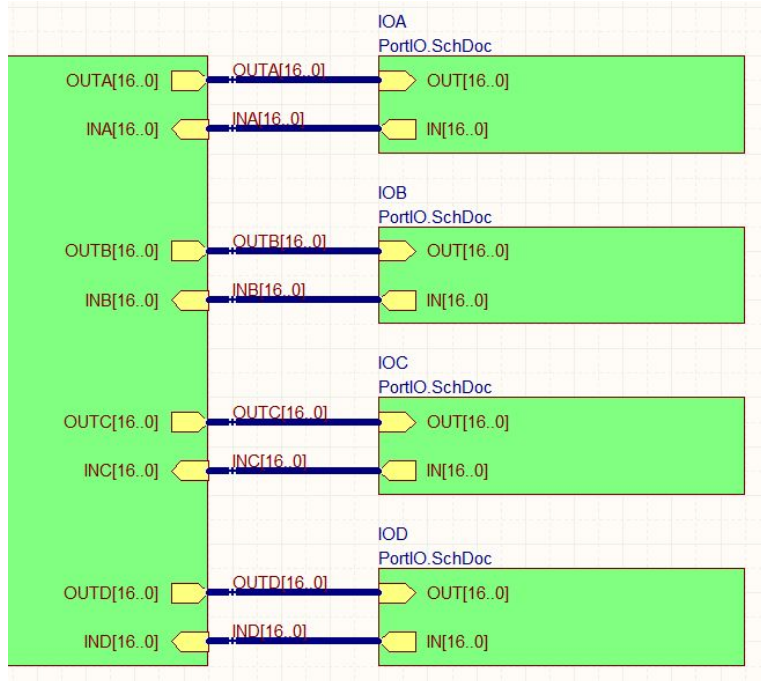
Reduces redundancy in design - 8 motor pages? One symbol, copy 8 times

Can copy layout of multiple copies as well

Helps to define cross-page connections

- Ex: what connects to 5V? Where does MCU i2c go?

Otherwise called “flat” design





Data Sheets - How to Read

Pin Descriptions

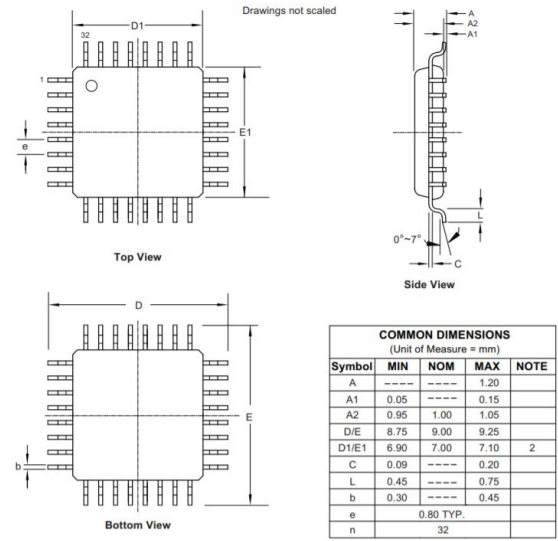
- Function associated with each pin
- Often links to longer descriptions later in datasheet

Electrical Characteristics

- Absolute max ratings
- Typical power draw, etc

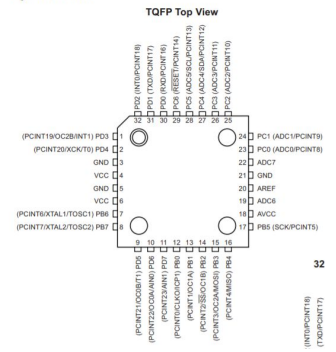
Layout

- Usually at end of sheet, used for creating CAD model of component



1. Pin Configurations

Figure 1-1. Pinout



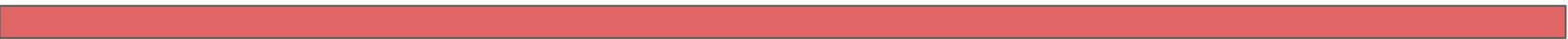
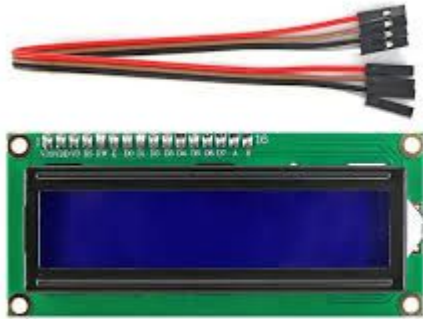


Offsheet parts/connections

Often your schematic will just have a connection to something (LCD screen, motor) instead of component

Create text box to explain what is being connected and how, part numbers appreciated

Enables next person looking at schematic to understand power constraints, communication ports, etc





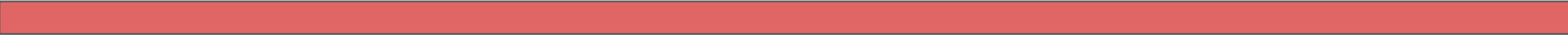
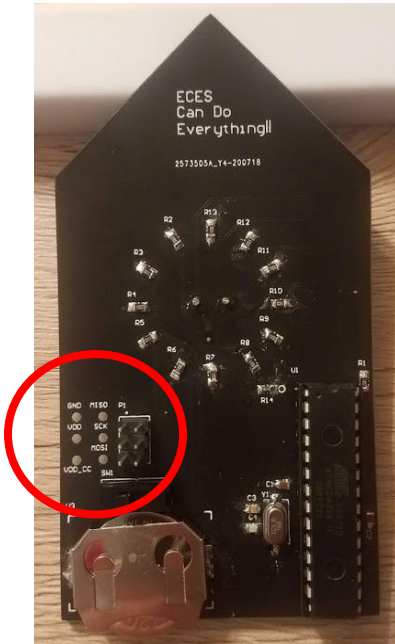
Test Points

Create designated, accessible exposed point on board connected to desired net

Very useful for debugging or prototyping

Can be surface mount or through hole

Easy to connect oscilloscope or multimeter for testing



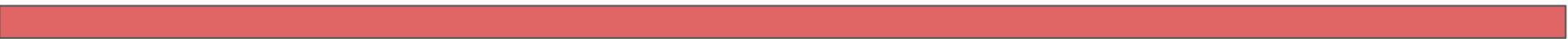
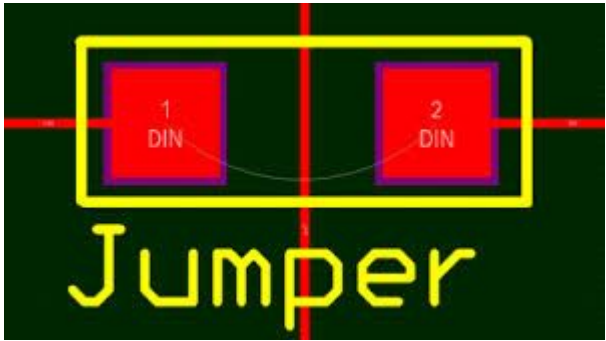


Solder Jumpers

Not sure exactly how to connect something?
Might want to easy connect or disconnect something later?

Common with TX/RX; not always clear how to connect

Exposed pads/traces will make it easier to fix up once board is manufactured





Mounting Holes

Used to attach boards to mechanical structures

Often connected to standoffs

Logically space so mechanical design is easier

Connect to ground if you want digital ground connected to chassis ground





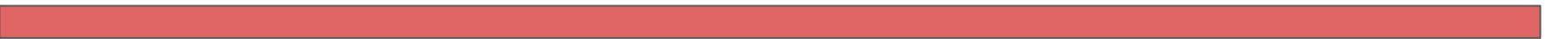
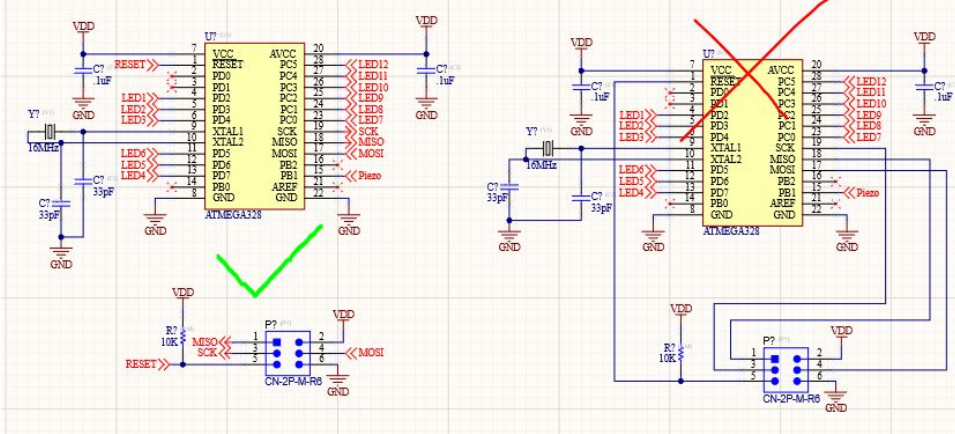
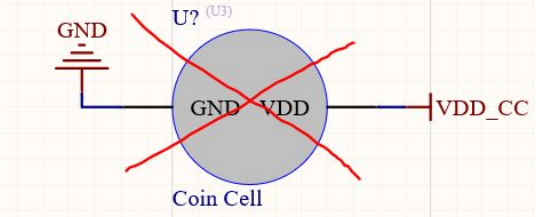
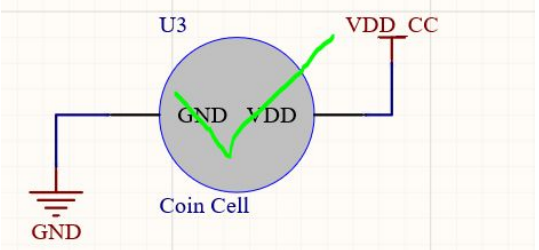
Keeping Things Legible

Name nets logically

Use nets instead of trying to directly wire everything

If multiple i2c, serial, etc, be specific

Power Up, Gnd Down



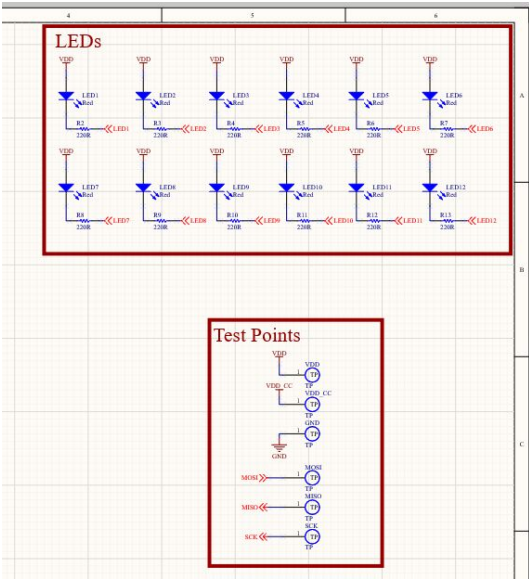
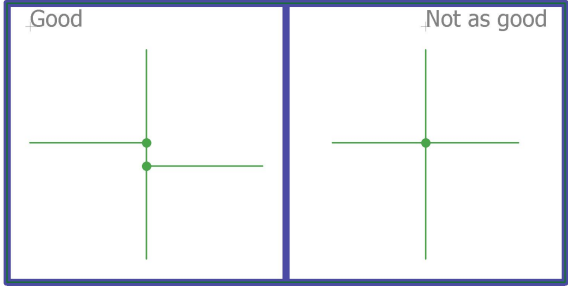


Legibility Cont.

When overlapping lines, clearly show what is connected and what is not

I like to put boxes around related groups of items to delineate

- Allows you to keep things organized without using excessive pages





Decoupling Capacitors

Caps on sensitive voltage lines (DC)

If any EMI, can have spikes on voltage lines

If MCU, IC doesn't have clean power line, can be messy

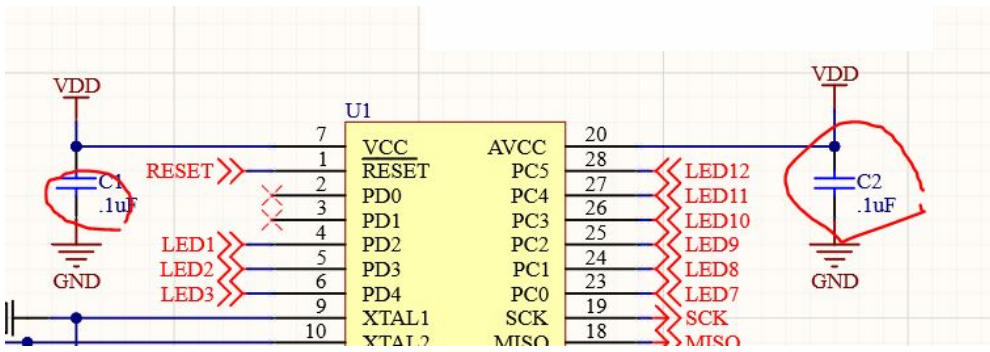
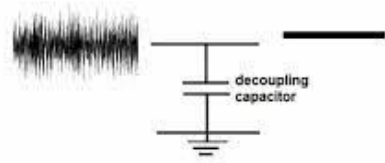
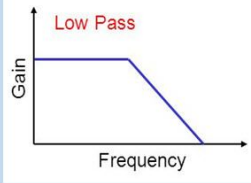
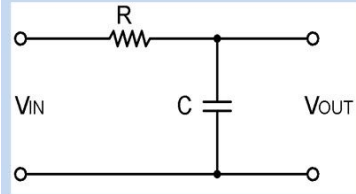
Especially important for analog signals!!! (ADC)

Put one on each voltage input

Put .1uF cap

Passive Low Pass RC Filters

ELECTRONICS HUB





Walk Through Clock Tower Schematic

