Component Selection and Creation

How to pick parts and define them in CAD
Resources for Parts

Digikey
- Very complete catalogue of components
- Well defined search, easy to find components

Mouser
- Second resort for electronics parts
- Sometimes has things digikey does not

Amazon
- Great for bigger components or kits
- Specialized parts
- Low key kinda evil
Resources Cont.

Adafruit

- Existing modules, great documentation
- Also a great resource for tutorials

Sparkfun

- Good for cheap plug-and-play sensors, etc
- Also has modules
Standard Passives

Passives often denoted by code

Refers to size of part

Smaller is good but difficult to solder/fix

Suggest nothing smaller than 0805 to hand solder! Start bigger for prototypes
Surface Mount Vs Thru Hole

Surface mount components sit on board

- Take up less space
- Can run traces on other side
- Easy to get assembled by manufacturer
- Low cost for boards
- Good for passives

Thru hole has leads that pierce board

- Easier to solder by hand
- More mechanically stable
- Can ensure good/big connection
- Heat dissipation
- Good for connectors
A component in PCB design has two defining files; a symbol and a footprint.

**Symbol** - The *logical* connections of an electronic component along with a representation.

**Footprint** - a *2D and 3D* model of what the component *physically* is.
Symbol

Some components (resistors, capacitors, transistors) have existing symbols.

For most ICs, the symbol will just be a rectangle with labeled pins.

**Pin** - one input/output of a device.

Check the datasheet for the pinout of a component.
Footprint - Terminology

Layers on a PCB are just that, different physical layers of the board.

A stackup shows the critical layers of a board, the different materials that make up each

- Here are two examples, one of a 2 layer board and one of a 4 layer board
Footprint - Terminology

Different categories of layers:

- **Mechanical** - Component bodies and outlines
- **Silk Screen** - Text printed onto boards
- **Paste** - Where solder paste will go for assembly
- **Top/Bottom/Etc** - Metal layers where traces are made
- **Solder Mask** - Covers exposed copper when desired (color of board)

**PTH** - Plated through hole. Hole coated with metal to connect bottom + top

**Pad** - Chunk of copper usually for connecting component

Altium's Layer Manager
Footprint

Shows where pads and holes are

Indicate pin 1 so part is placed correctly

Give clearance - courtyard around component to not place other components too close by

Give outline

Can use wizard for standard parts

- Auto-generates nice 3D model of ICs, etc
3D body

3D representation of component

Useful for height constraints, visualizing board

Can often download models from suppliers

Ensures pins have correct spacing
Part Organization

Sometimes just keep in libraries in project

Often database of verified parts for organizations

Can link database to design software

Altium has “vault” of parts - not verified

Once you make parts you want to be consistent, not remake
In CAD, you must define your grid

Ex

- 1cm
- .1mm
- 10 mils (thousandths of an inch)

This is important for parts, schematic, and layout

Helps you make+place things with logical scale

Can switch between imperial and metric

Objects snap to grid at their origin
Walk Through Component Creation in Altium