



# 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



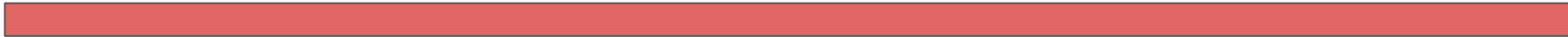
## Mouser

- 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

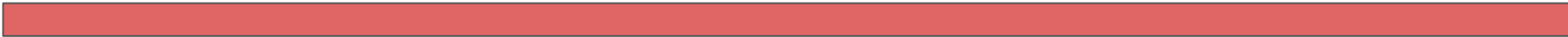
Refers to size of part

Smaller is good but difficult to solder/fix

Suggest nothing smaller than 0805 to hand solder! Start bigger for prototypes

<i>comparison</i>	Metric code	Imperial code	<i>comparison</i>
0.1x0.1 mm	0402	01005	0.01x0.01 in (10x10 mils)
	0603	0201	
	1005	0402	
	1608	0603	
1x1mm	2012	0805	0.1x0.1 in (100x100 mils)
	2520	1008	
	3216	1206	
	3225	1210	
	4516	1806	
	4532	1812	
	5025	2010	
1x1 cm	6332	2512	0.5x0.5in (500x500 mils)

Actual size





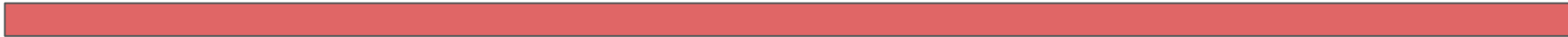
# Surface Mount Vs Thru Hole

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



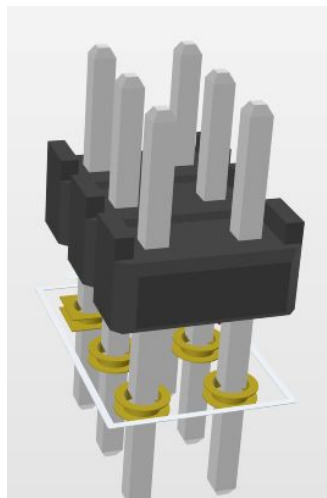
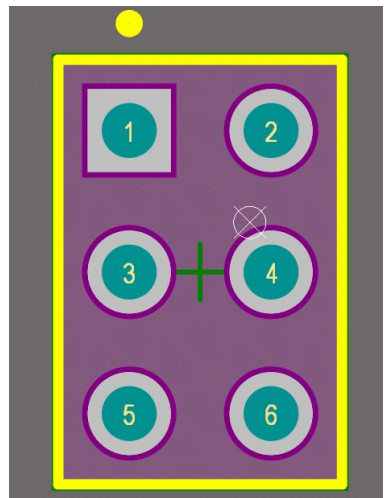
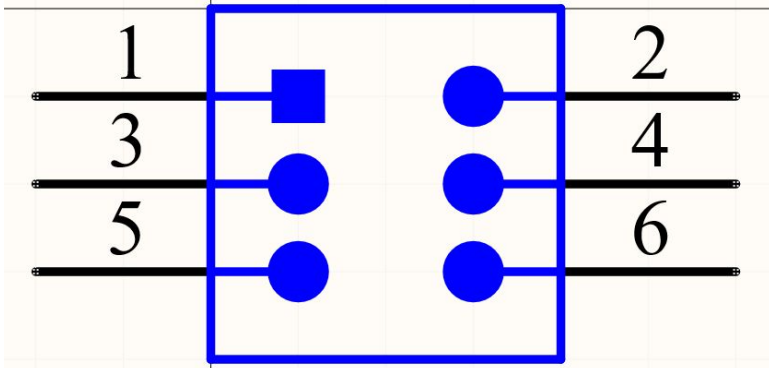


# Schematic Symbol vs Layout Footprint

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





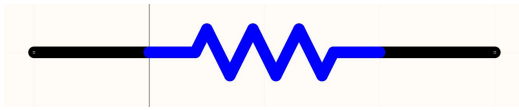
# Symbol

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

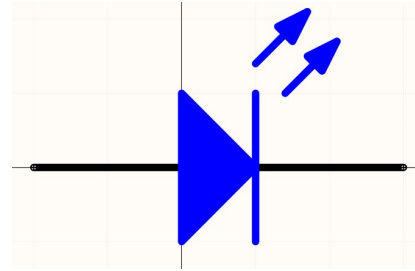
For most ICs, the symbol will just be a rectangle with labeled pins

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

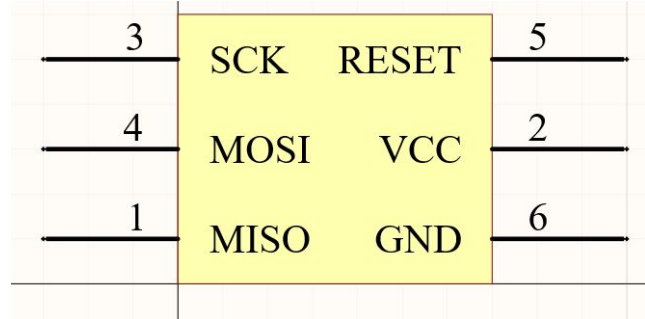
Check the datasheet for the pinout of a component



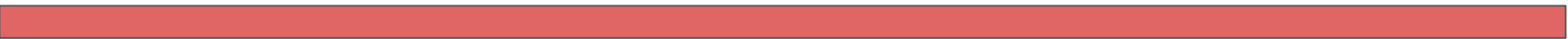
Resistor symbol (2 pins)



LED symbol (2 pins)



6 pin header, labeled pins





# Footprint - Terminology

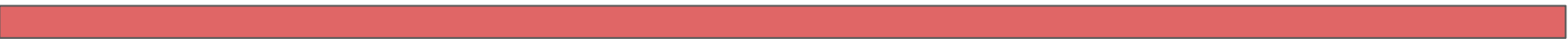
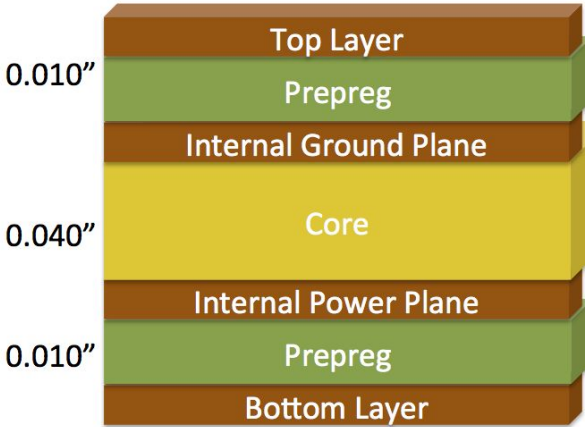
**Layers** on a PCB are just that, different physical layers of the board.

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**

2 Layers 1.6mm(0.062inch)Green for 1 Oz

Layer stack up	Layer	Thickness(mm)
	Top Solder Mask	0.01
	Top Layer	0.035
	Core	1.5
	Bottom Layer	0.035
	Bottom Solder Mask	0.01







# 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

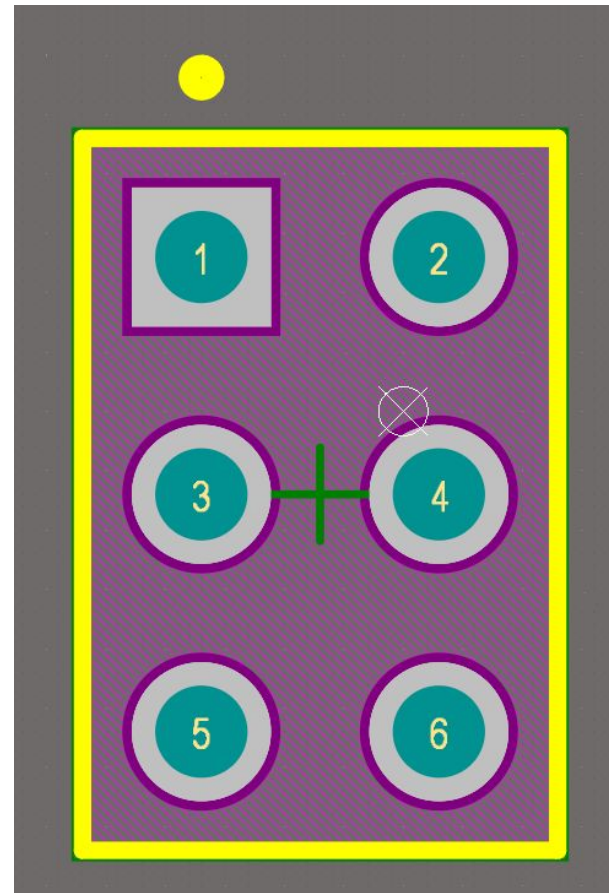
Indicate pin 1 so part is placed correctly

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





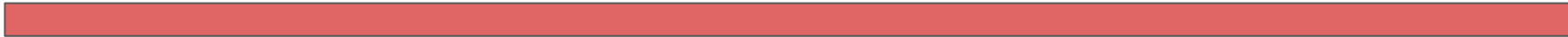
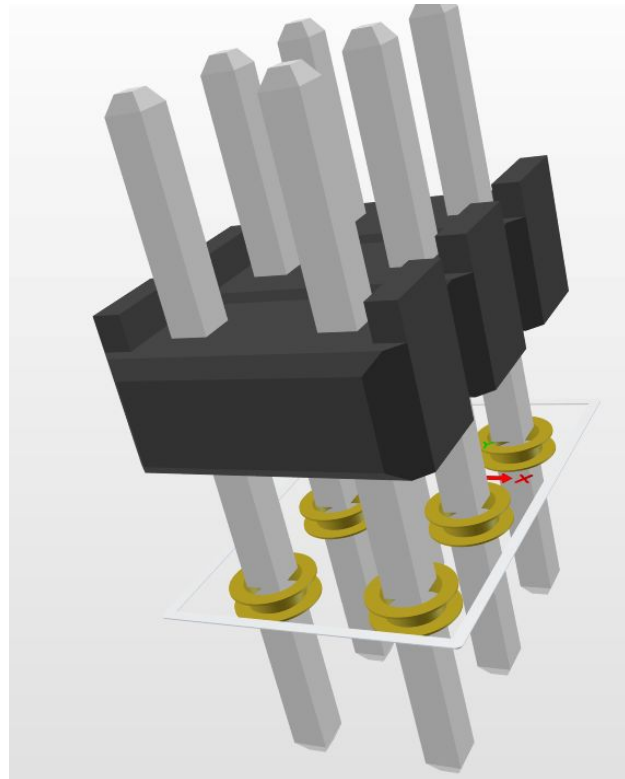
# 3D body

3D representation of component

Useful for height constraints, visualizing board

Can often download models from suppliers

Ensures pins have correct spacing





# Part Organization

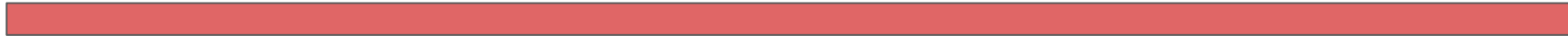
Sometimes just keep in libraries in project

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





# Grids

In CAD, you must define your grid

Ex

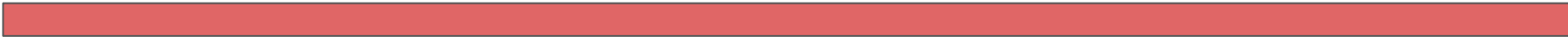
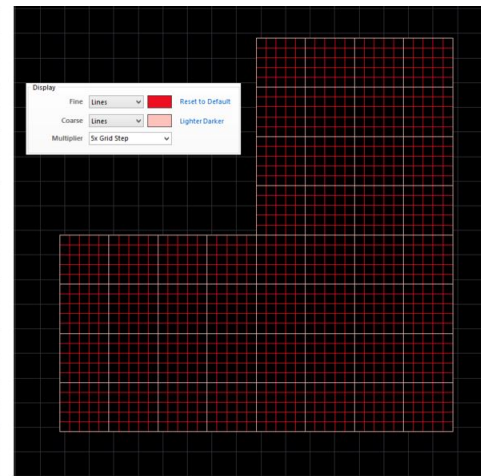
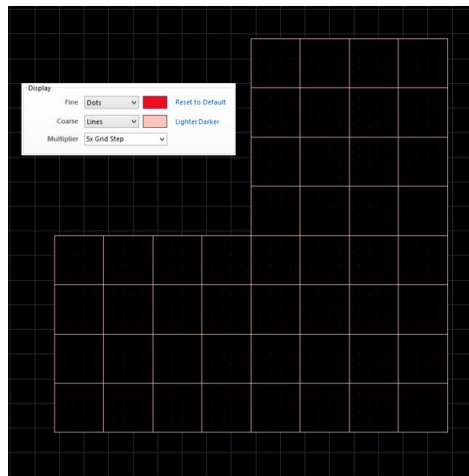
- 1cm
- .1mm
- 10 mils (thousandths of an inch)

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

