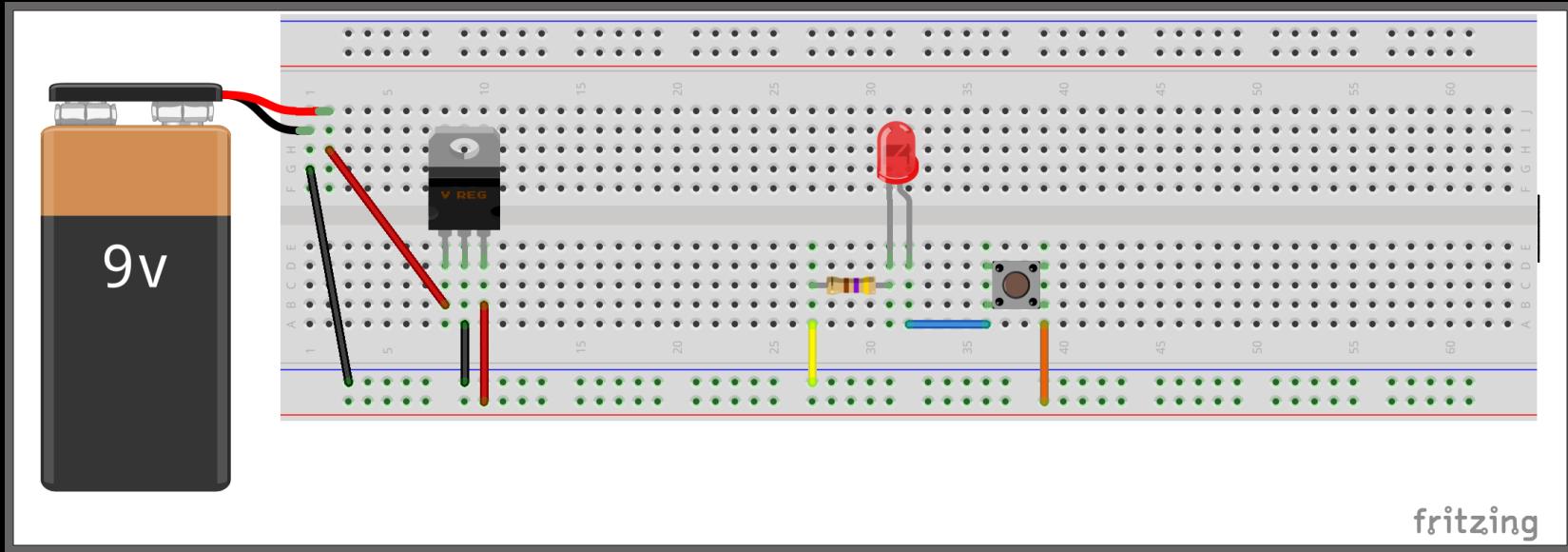


# EAGLE & co.

Gadgets S14

# Breadboards

Fast & Forgiving

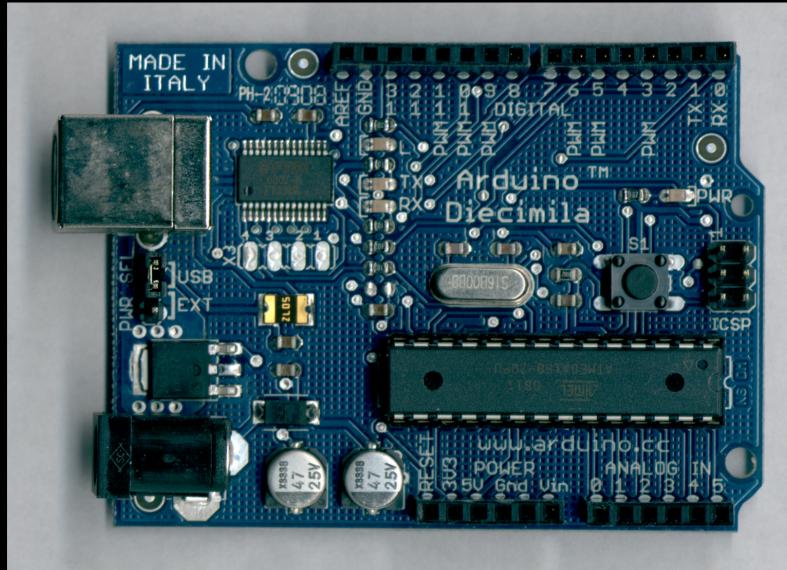


# THT vs. SMT



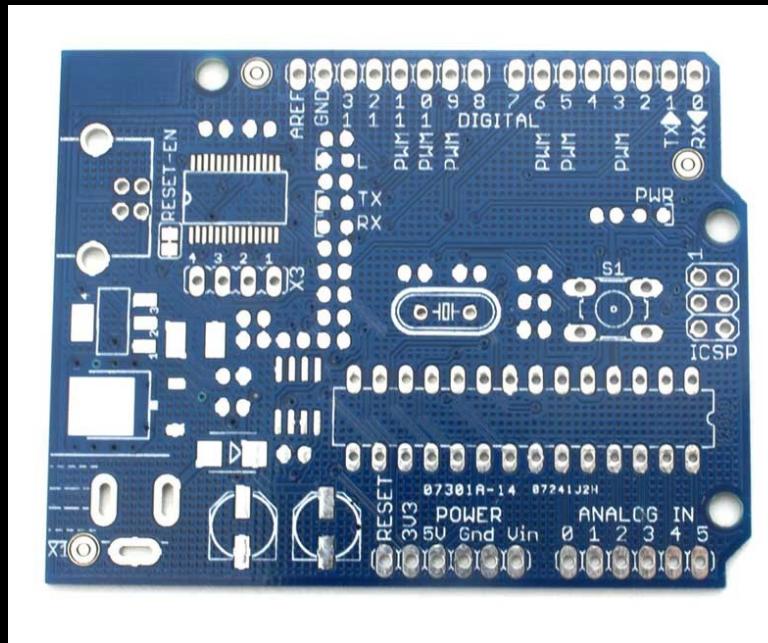
# Printed Circuit Boards (PCBs)

Cheap & Consistent



# Printed Circuit Boards (PCBs)

The bare PCB



# Schematic vs. PCB Layout

A Schematic is an abstract representation of a circuit, which can then be used to build the circuit on a breadboard, PCB, or whatever.

A PCB layout defines the physical properties (e.g. place on the board) of components and the traces that connect the components together.

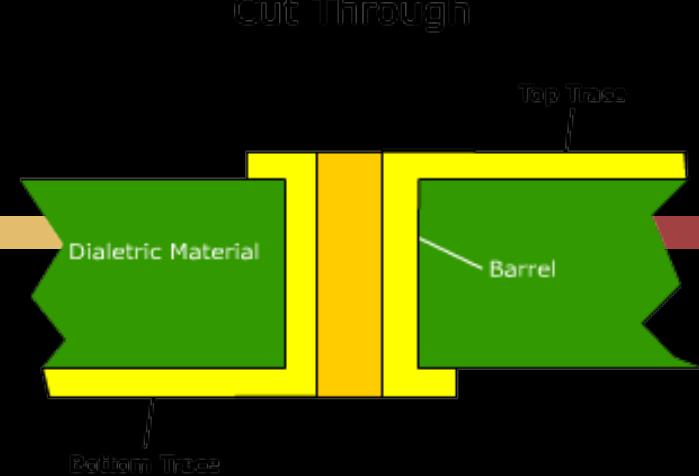
# Double Sided PCBs

The numbers of Layers a PCB has refers to the number of conductive planes available for traces.

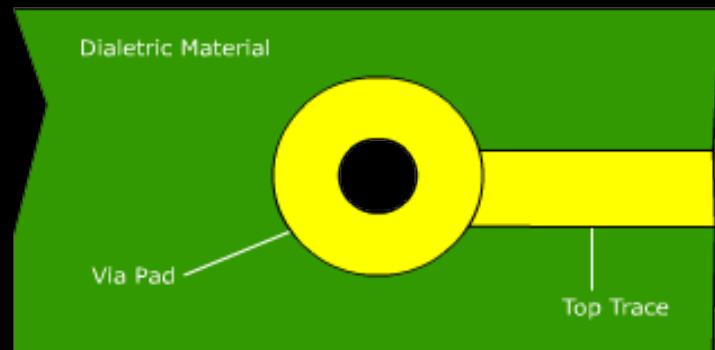
We'll be making double layer boards, which means we can have copper traces on the top and bottom of the PCB.

# Vias

A Via connects conductive layers in a PCB. Great if you can't route around other traces on the same layer: Just jump to the other side for a bit.



Top View

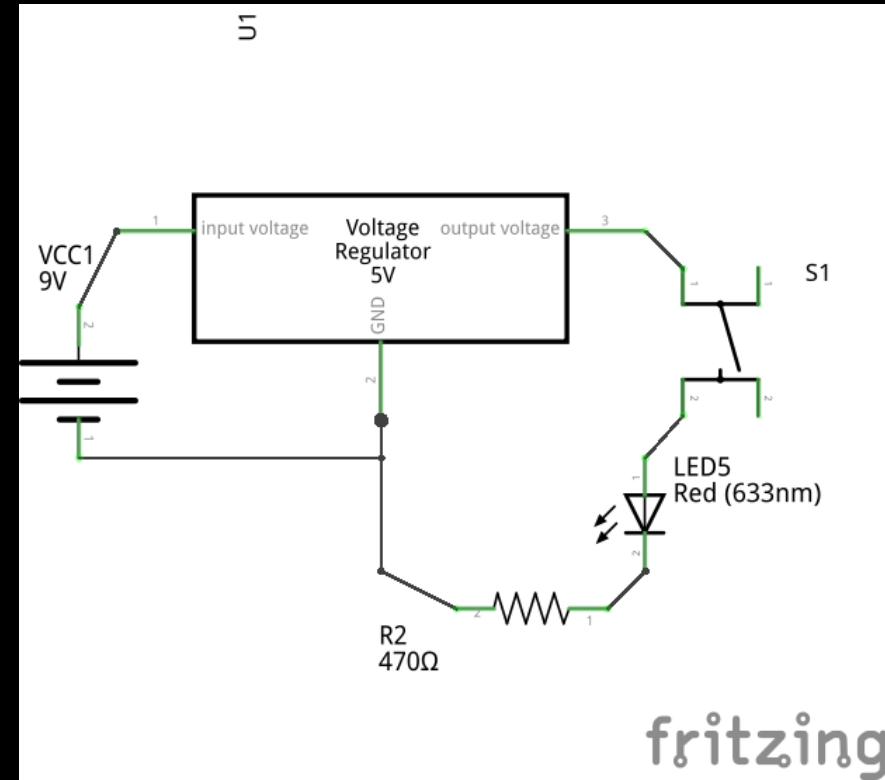


Holes for THT components are effectively vias.

# Warm up to EAGLE

Look a schematic!

Let's lay it out on a  
Graham Cracker.



fritzing

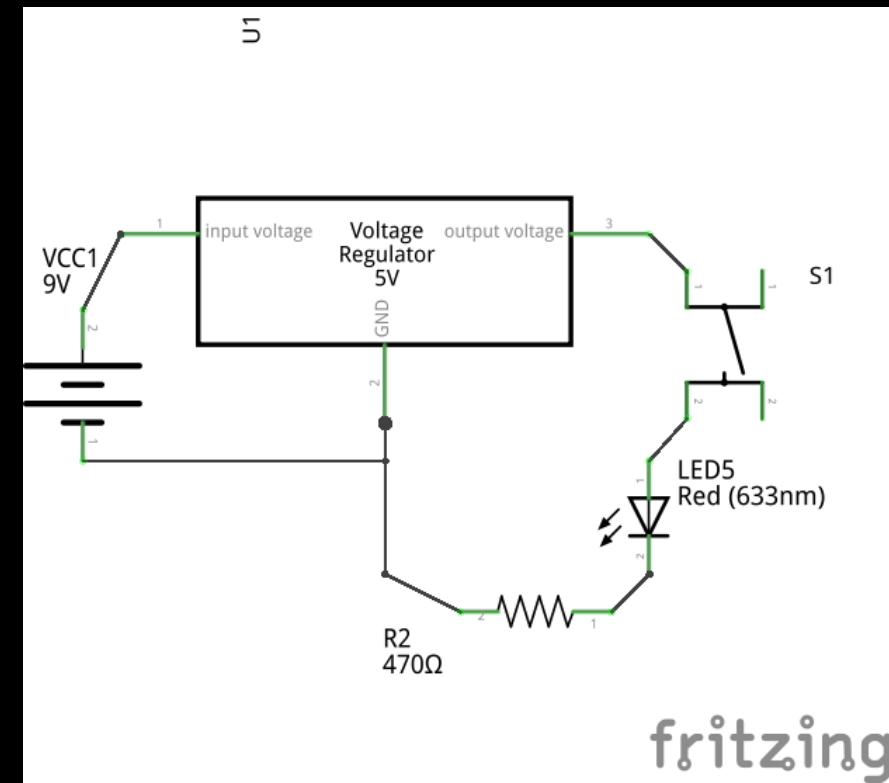
# Icing Colors

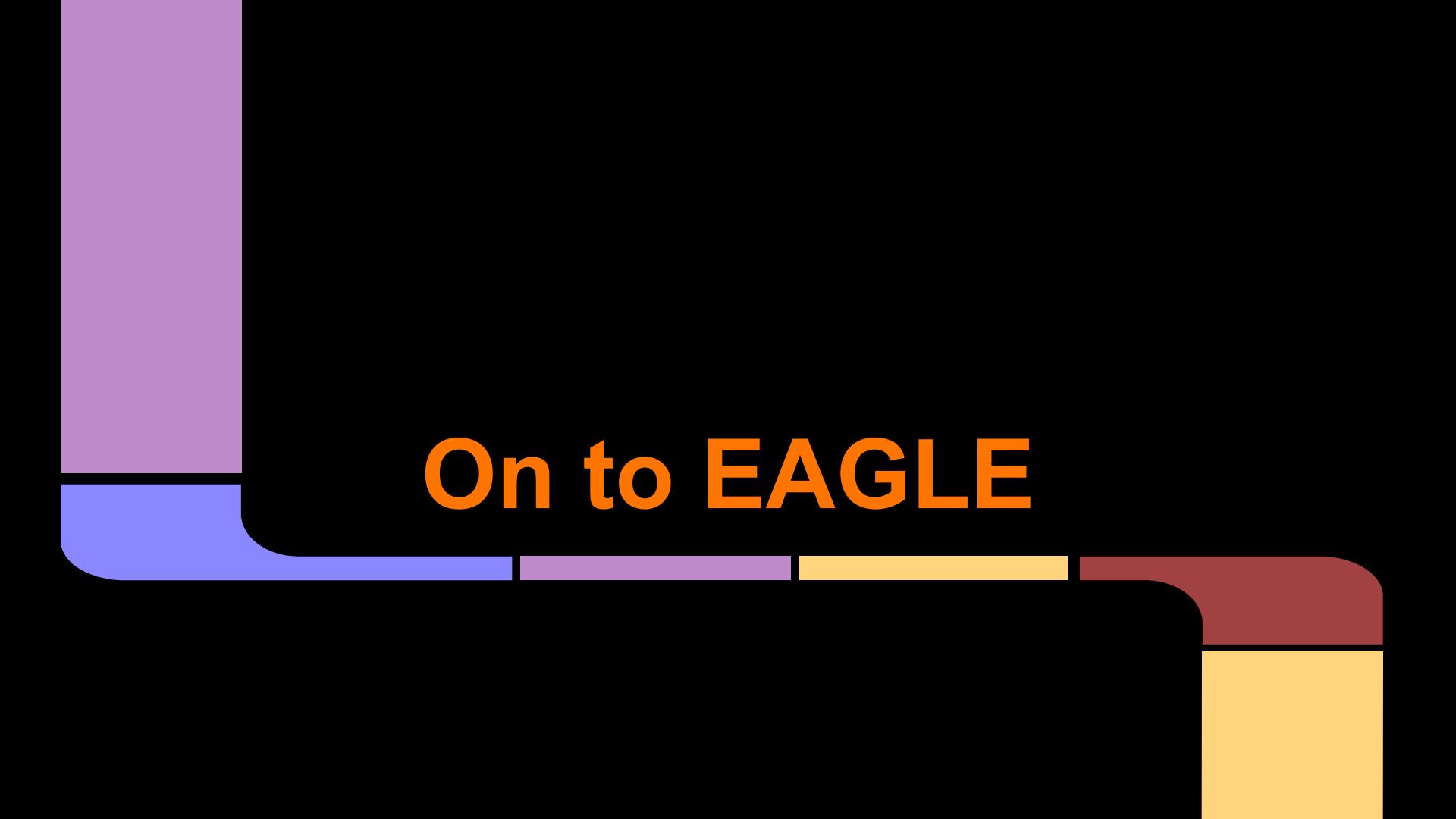
**Green:** Holes for THT components & Vias

**Red:** Copper on top

**Blue:** Copper on bottom

\* must use at least 1 via





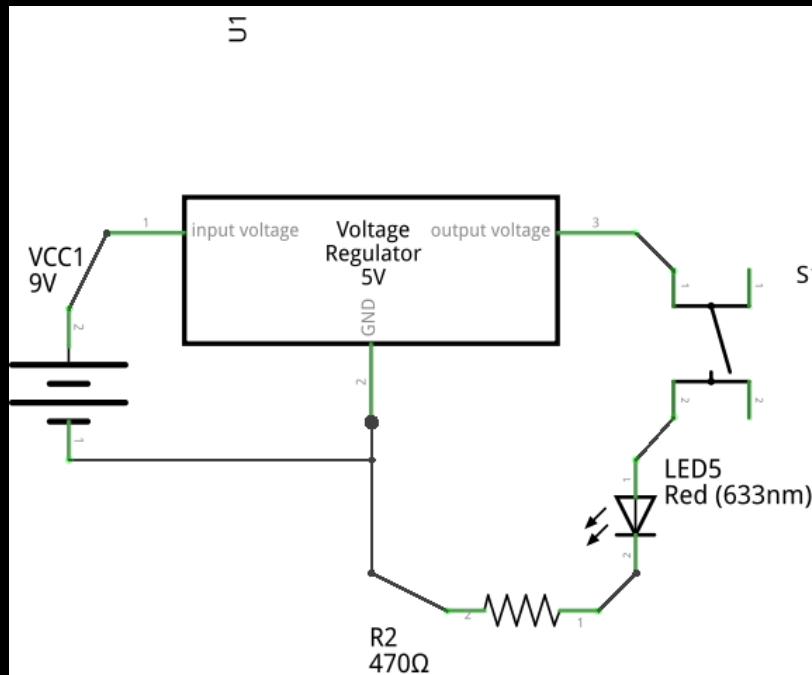
On to EAGLE

# Libraries

This is where various components are defined. We'll need to add the Sparkfun library, which has a good selection.

Get thee to Blackboard & download the zip file under “Eagle bits”. Unzip it somewhere & remember where. Don’t close the Blackboard tab just yet.

# Re-create This Schematic in Eagle



fritzing

# DRC

The DRC checks a PCB layout for various manufacturing issues, e.g.: Distance between traces, hole sizes, width of traces, etc.

n.b.: It does not yell at you for your air wires.

Since manufacturability changes between fab houses, a .drc file is used to specify what the DRC should be looking for.

Grab the file off Blackboard. Still don't close the tab.

# CAM Processor

The CAM Processor takes your nice layout and puts it into an actually usable form for the fab house.

It maps EAGLE “conceptual” layers to real layers in the PCB. e.g. Top copper traces & solder pads to a top layer of copper.

# CAM Processor (cont.)

We use a CAM job from Sparkfun, which works reasonably with fab houses like 4PCB & BatchPCB.

Go get the .cam file from Blackboard. \*Now\* you can close the Blackboard tab if you want.

# Files Generated by the CAM Job

- Top and bottom copper (.GTL, .GBL)
- Top and bottom solder mask (.GTS, .GBS)
- Top and bottom silkscreen (.GTO, .GBO)
- Drill file, 2.4 leading (.TXT)

Zip those files to ship them off to 4PCB.

# Shipping Off to 4PCB

<http://www.4pcb.com/pcb-student-discount.html>

Or ask the internet for “4PCB student discount”

“No minimum PCB quantity on ordering our famous \$33 Each 2-layer Full Spec Prototypes Special. **Type the word "Student" in the COMMENTS section when placing your \$33 Each Board.”**

# Project 2 Pitfalls

The header pins are bent 90° so your Arduino clone can fit on the breadboard...  
That means they have to stick off the side of the breadboard.

Not picking a THT component.

Did you pick the \*right\* THT component?

Not printing out a 1:1 image of your board and laying your parts on top of it.

Leaving Air Wires.

# Other Resources

Hack A Day's Guide to Preparing Layouts  
for Manufacture

<http://hackaday.com/2009/01/15/how-to-prepare-your-eagle-designs-for-manufacture/>

Sparkfun Eagle Tutorial

<https://www.sparkfun.com/tutorials/108>