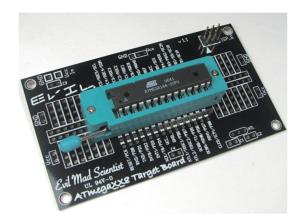
#### Introduction to Printed Circuit Board design, For robotics, using KiCAD

12/14/2019 Dennis Mangrobang







#### PCB Printed Circuit Board

#### PCBA PCB Assembly

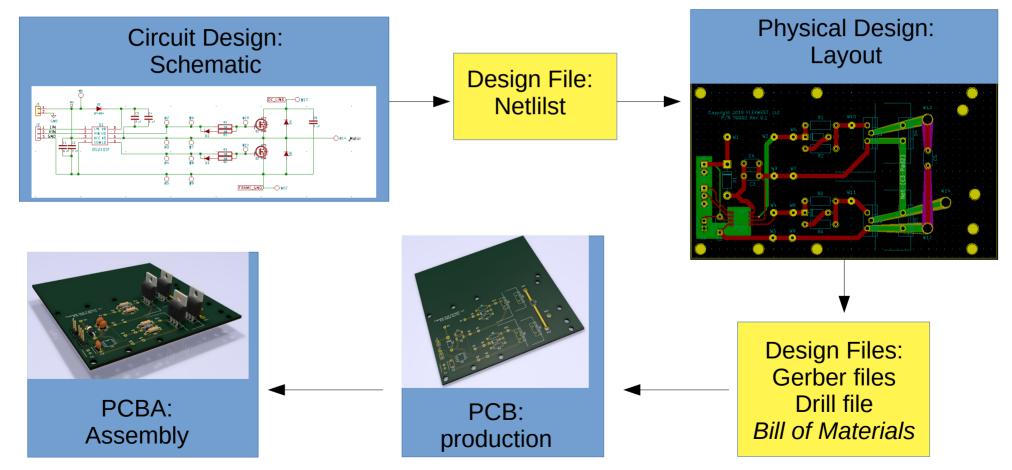




# Agenda

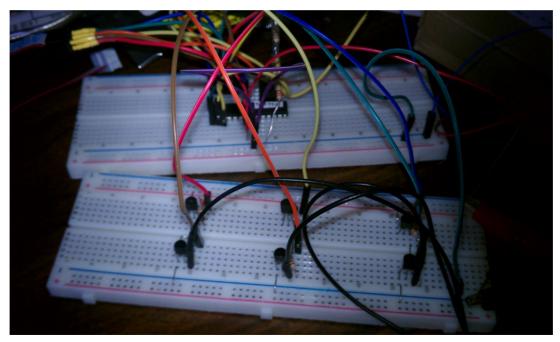
- Overview of PCB concepts
- Alternative (non-PCB) methods for building circuits
- Possible uses for PCBs in your robotics project
- Basics of PCB design using KiCad

# Typical PCB design/build workflow



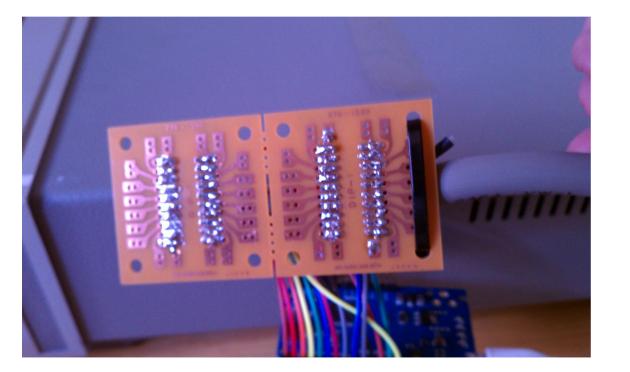
#### Alternatives to PCBs

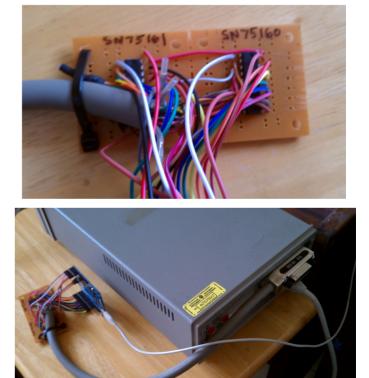
Solderless Breadboard



#### Alternatives to PCBs

Proto Board / Perf Board / Vero Board





#### Alternatives to PCBs

- "Dead bug": glue components upside down on a copper board (ground plane), and solder wires to their "legs".
- Milling a copper clad board
- Wire wrap (seldom used now)

#### Reasons to design a PCB

- Custom expansion boards, e.g. "Hats" or "Shields" for Arduino, Raspberry Pi, etc.
  - Provide functionality not available with any board you can purchase
  - Combine functionality of several boards into one board

#### Reasons to design a PCB

- Interconnect other PCB boards, motors, etc. with a PCB, instead of wires
- The PCB can be a part of your mechanical structure
- Achieve higher reliability (compared with proto boards, breadboards, wired assemblies, etc.)

#### Reasons to design a PCB

- Create a smaller, more compact assembly or robot
- A PCB may be the only practical option (e.g. if using an IC with a very large pin count, BGA, etc.
- Make it easy or practical to share your design with others
- You need several of the same board

#### PCB Design Tutorial : David L. Jones

- I found this tutorial useful
- It was written by Dave Jones, who runs the eevblog
- http://alternatezone.com/electronics/files/ PCBDesignTutorialRevA.pdf

#### **KiCad Overview**

- Usually pronounced "KEE CAD"
- KiCad is a form of "Electronic Design Automation" (EDA) software
- Free and open source
- Runs on Linux, Microsoft Windows, OS X
- Can be used for "large" boards, many layers
- Widely used, gaining "market" share

#### **KiCad Overview**

- Usually pronounced "KEE CAD"
- KiCad is a form of "Electronic Design Automation" (EDA) software
  - Schematic capture
  - PCB layout
- KiCad alternatives
  - Eagle, Altium, Easy EDA
- Other types of EDA software
  - IC design (e.g. Verilog/VHDL), circuit simulation, etc.

#### Alternatives to KiCad

- Free and open source
- Runs on Linux, Microsoft Windows, OS X
- Can be used for "large" boards, many layers
  - No restricted versions or tiers of capability, commonly found in commercial software
- Widely used, gaining "market" share

#### Start KiCad

KiCad 5.1.5-52549c5~84~ubuntu18.04.1 /u/projects/kicadProjects/scratch1/scrat 🔶 👝 📼 🗙
File View Tools Browse Preferences Help
scratch1.pro scratch1.kicad_pct scratch1.sch Project name: ////////////////////////////////

## KiCad – Create Project

- I usually create a new folder first
- Create a new project:
  - File  $\rightarrow$  New Project
  - Select folder for the project
  - Fill in name of project

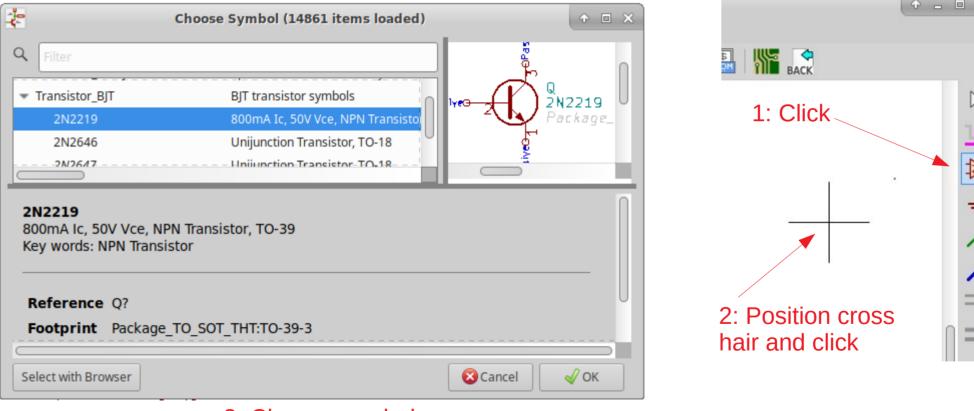
### **KiCad - Schematics**

- Express the "logical" or "theorical" design of the circuit, rather than the physical realization
- Start Schematic Editor, "Eeschema":
  - Tools  $\rightarrow$  Edit Schematic (Or click Icon)
- Schematic page settings
  - File  $\rightarrow$  Page Settings
  - Set paper, e.g. 8.5" x 11", Landscape
  - Set Date, Revision, Title, etc.

# KiCad – Schematic Symbols

- Schematic symbols represent various electronic components, e.g.
  - Resistors, Capacitors, Inductors
  - Connectors, switches
  - Transistors, diodes
  - Integrated circuits, e.g. microcontrollers
- KiCad comes with a decent library of symbols
- Other symbol libraries are available from github, distributors (e.g. DigiKey) and snapeda.com
- You can create you own symbols
- In KiCad, schematic symbols ARE NOT statically bound to a physical "footprint"

#### KiCad – Place schematic symbols

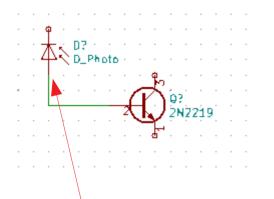


#### 3: Choose symbol

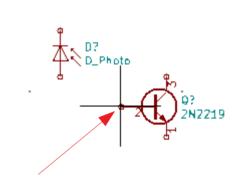
# Kicad – Set symbol values

- Some symbols will need to have properties set.
  - Resistors: Set resistance value
  - Capacitors: Set capacitance value
- Right-click on the symbol and select "Properties → Edit Value" ("v" key is shortcut)

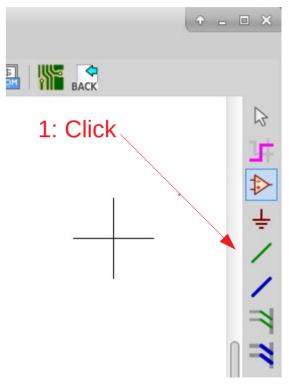
#### KiCad – Place wire connections



3: Move along desired path and click at end



2: Position cross hair and click at start position (release mouse button)



#### Annotate schematic

and the stand of the leads

	cts/kicadProjects/kicadDemo2/kicadDemo2
Annotate Schematic + • ×	
Scope:       Order:         Use the entire schematic       Sort components by X position         Use the current page only       Sort components by Y position	E   ▶ [N]   2 ★ h = 1 + 1 + 1 + 1 + 1 + 1 + 1 + 1 + 1 + 1
Options: • Keep existing annotations • Reset existing annotations • Reset, but keep order of multi-unit parts Annotation Messages: Annotation complete.	1: Click
Show: All Errors Warnings Actions Infos Save	
Clear Annotation Close Annotate 2:	3: Symbols are annotated, e.g. D3 and Q2

#### Run schematic rules checker

*	Electrical Rules Checker	• • >
ERC Options		
ERC Report:	Messages:	
Total: 3	Finished	
Warnings: 3		
Errors: 0		
Create ERC file report		
Error List:		
error) @(8.600 in, 3.100 in) Pin not connected ( error) @(8.600 in, 2.700 in) Pin not connected ( error)	(use a "no connection" flag to suppress this ): Pin 1 (Passive) of component Q2 is unconnected (use a "no connection" flag to suppress this ): Pin 3 (Passive) of component Q2 is unconnected (use a "no connection" flag to suppress this ): Pin 1 (Passive) of component D3 is unconnected	1. 5 1. 5
Delete Markers	Close	Run

2:

cts/kicadProjects/kicadDemo2/kicadDemo2

_											_		-	_	-			_				_		_
	t		1	$\triangleright$		Q	Ę			Z		۲	\$	-	NE				_\$ <u>₿</u> 0		1	۲	в	ACK
•	•	•	•	• •			•	•	•	• •				• •	•	•	•		• •	•	•	÷	Ŧ	
																	-					÷		
•		•	-	• •					•			•					-					÷		
•			-	• •		•				• •				• •		1	: (	Cli	ick			1		
	-	•	-	• •		Î			•		•	•		• •	•	•	• •		• •	•	•		1	
						Æ	12	D3 D_F	Jho	ta i													1	
							- )					đ												
											-	_ <u>t</u> r											-	
						L					4	٢A	92	221	<u>.</u>									
											$\sim$	Y	Z٩	221	9									
						-						5												
				• •																		÷		
				• •						• •				• •			-							
•	4	-	•	• •					•		•	•	•	• •							•			
				3.	Ň	/la	rk	er	s i	n		sch	IPr	n ne	atic	· \	vill						1	
	÷																V I I				÷		1	
							τΟ	e	rrc	ors	5 (	no	τS	sn(	JŴ	ņ							B	
				he	ere	e)																		

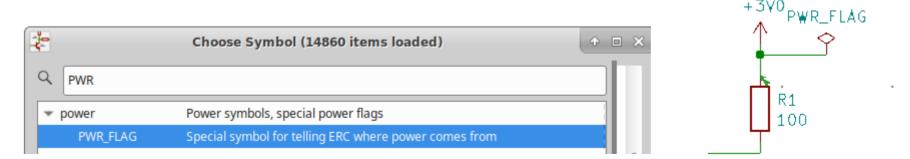
#### Associate footprints with symbols

#### 3: Optional: Right click on cts/kicadProjects/kicadDemo2/kicadDemo2 footprint to view Assign Footprints File Preferences Help 'm≓ 'm# 'mL -> Footprint Libraries Symbol · Footprint Assignments 1 Battery:BatteryHolder Bulgin BX0036 1xC LED : LED THT:LED D5.0mm 2 Battery:BatteryHolder ComfortableElectronic CH273-2 Button Switch Keyboard 2 D2 Button Switch SMD з D3 -D Photo 3 Battery:BatteryHolder Eagle 12BH611-GR Button Switch THT Λ 01 -2N2222 : Package TO SOT THT: TO-92 Inline 4 Battery:BatteryHolder Keystone 103 1x20mm Buzzer Beeper 5 02 -2N2219 : Package TO SOT THT: TO-39-3 5 Battery:BatteryHolder Keystone 104 1x23mm Calibration Scale R1 -470 : Resistor THT:R Axial DIN0309 L9.0mm D3.2mm P12.70mm Horizontal 6 Battery:BatteryHolder Keystone 105 1x2430 6 Capacitor SMD W1 -Conn 01x01 : Connector Wire:SolderWirePad 1x01 Drill0.8mm 7 Battery:BatteryHolder Keystone 106 1x20mm Capacitor Tantalum SMD W2 - MountingHole Pad : Connector Wire:SolderWirePad 1x01 Drill0.8mm 8 Battery:BatteryHolder Keystone 107 1x23mm Capacitor THT 9 Battery:BatteryHolder\_Keystone\_500 Connector 10 Battery:BatteryHolder Keystone 1042 1x18650 Connector AMASS 11 Battery:BatteryHolder Keystone 1058 1x2032 Connector Audio 12 Battery:BatteryHolder\_Keystone\_1060\_1x2032 Connector BarrelJack 13 Battery:BatteryHolder Keystone 2460 1xAA Connector Card 14 Battery:BatteryHolder Keystone 2462 2xAA Connector Coaxial 15 Battery:BatteryHolder\_Keystone\_2466\_1xAAA Connector DIN 16 Battery:BatteryHolder Keystone 2468 2xAAA Connector Dsub 17 Battery:BatteryHolder Keystone 2479 3xAAA Connector FFC-FPC 18 Battery:BatteryHolder Keystone 2993 Connector Harwin 19 Battery:BatteryHolder Keystone 3000 1x12mm Connector HDMI 20 Battery:BatteryHolder Keystone 3001 1x12mm Connector Hirose 21 Battery:BatteryHolder\_Keystone\_3002\_1x2032 Connector IDC 22 Battery:BatteryHolder Keystone 3008 1x2450 Connector JAE 23 Battery:BatteryHolder Keystone 3009 1x2450 Connector JST 24 Battery:BatteryHolder Keystone 3034 1x20mm 25 Batterv:BattervHolder LINX BAT-HLD-012-SMT Connector Molex Filtered by library (Battery), search text: 41 🖌 ОК Apply, Save Schematic & Continue Cancel

2:

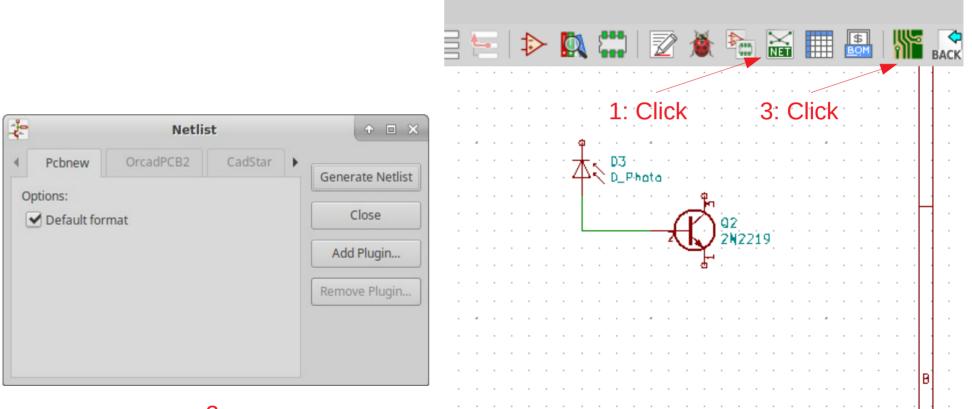
# KiCad: Power Flags

- A "Power flag" is a standard schematic symbol, used to indicate where power comes from.
- A "Power flag" should be connected to each GND and power supply rail.
- The Electrical Rules Check (ERC) will complain if these power flags are missing.



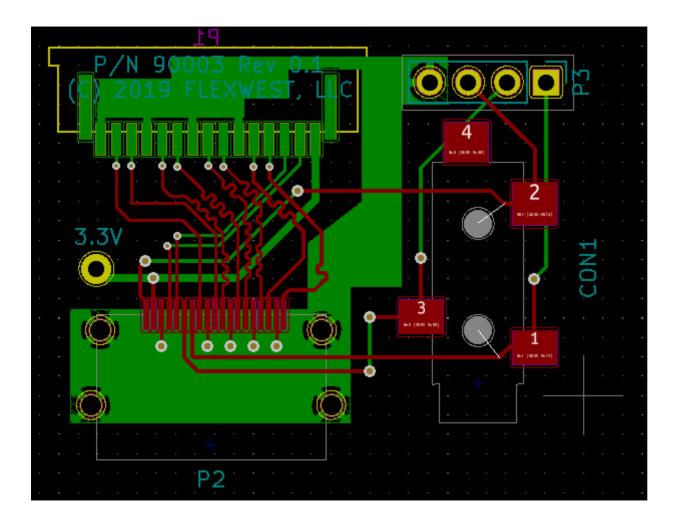
#### Generate netlist and start PCB editor

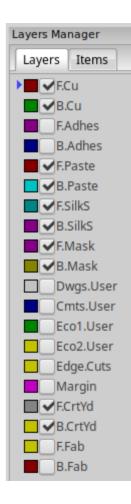
cts/kicadProjects/kicadDemo2/kicadDemo2



## Some PCB terminology

- Track or trace : connections between components
- Copper layers : 2 layer vs 4 layer PCB, etc.
- Via : connection between copper layers
- Filled zone / ground and power planes
- Solder mask : prevents solder from adhering : different colors available, determines overall color of the PCB
- Silk screen : printed text or graphics, e.g. component designations, part number, revision, etc.
- Edge cuts : PCB edges, internal slots, etc.
- Layout layers (e.g. front copper, front solder mask, etc.)





# **Basic layout flow**

- Configure page settings (like was done for schematic)
- Define track and via defaults, minimum clearances
- Import "rats nest" by reading the netlist
- Draw board outline in edge layer (can do later)
- Drag footprints into (rough) position, usually by functional groups, and in ways that minimize crossing connections
- Lay out tracks (and vias)
- File  $\rightarrow$  Plot

## Track width and spacing

- Usually expressed in "mils", 1 mil = .001"
- Most PCB companies can support down to 6/6 mil width/spacing; smaller values are possible, but may incur extra charges. Generally, I try to use 10/10 mil minimum when practical.
- Width and copper thickness affect current carrying capability and resistance. Additionaly external vs internal layers affect temperature rise.
- A 6 mil external trace in 1 oz copper can support 600 mA current with a 10 deg C temperature rise. For microcontroller digital I/O with currents of 60mA, there should be no issue. For other situations, e.g. motor current, you should perform calculations. KiCad has a calculator tool to help with this an other calculations.

## Kicad: Customer Panels

- If you run Pcbnew directly (not launched from the main Kicad program), you can append
- V-score
  - A "V" routed groove through about 2/3s of the PCB, allowing it to remain together during fabrication, but possible to separate items by hand
- Tab routing
  - Routed slots, interrupted by some solid area. The solid area usually has a series of small "mouse bite" holes to make it practical to break apart after fabrication.

```
Dirty PCBs 2/4 Layer Capabilities (partial list)
                     CAPABILITY
ITEM
Material
                     FR-4 0.6mm-2.0mm 1oz copper ('standard' PCB material is 1.6mm thick)
Layer number
                     4L
Maximum size
                     600*600mm (60*60cm)
Shape
                     Almost anything! We'll send it and see if they accept it!
                     32mil (0.8mm)
Min internal slot
Min w/s
                     6/6mil(I/L) (increased from 5/5 due to poor yield)
Min silkscreen line
                     0.15mm
Min SMD width
                     8mil
Min diameter of finished hole
                              12mil
Tolerance of drill position +/- 2mil
Tolerance of finished hole size PTH +/- 3mil
Tolerance of finished hole size NPTH: +/- 2mil
PTH hole copper thickness
                              0.6~1.4mil
Max A.R of PTH
                     8:1
Surface copper thickness 1oz
V-cut/V-groove
                     80mm (8cm) minimum, 380mm maximum. Optional, extra charge
Surface treating
                     HASL (hot air surface leveling, not PB free unless specialrequest)
                     /AuSn/AgCN/Electrogilding/Ni/OSP*/G.F
```

← → ♂ ŵ	🖸 🔒 😋 https://dirtypcbs.com/store/pcbs	☑ ☆	\ □ ③	≡									
Sign In/Register	PCBs												
Search Q	Full PCB specs and info												
🖶 Shop	n Remove												
📜 Cart	File	Material	Layers	Quantity	Price	_							
	Browse No file selected.	FR4 proto 🔹	2 🗸	Protopack ±10 🔻									
≣ Categories <	Size	Size (quantity 10+)	Color	Custom silk/mask color									
₽ Printed Circuit Boards <	max 10x10 +\$5 💌	10 X 10 CM	Green	1	\$16.95								
PCB Stencils				1									
	Thickness	Coating	Copper	Stencil									
▼ SLA 3D Prints <	1.6mm -	HASL	1oz -	None		0							
4 Laser Cut Acrylic	Advanced Options												
🖋 Custom Cables 🛛 🔍													
🖌 Dirty BOM 🛛 🗸	Total				\$ 16.95								
Ø Other Services <	🗮 Add to Cart 🕂 Add more	PCBs!											

## Prepare Files for PCB Vendor

- Process may vary by vendor
- Dirty PCBs wants a zip file containing these files:
  - All the gerber files (\*.gbr)
  - The drill file (\*.drl)

## JLCPCB

- I have only used dirtypcb.com
  - They no longer provide tracking for slow/cheap shipping
- I will likely try jlcpcb.com
  - Seem to have very low prices
  - Seem to handle v-score panelization of a single design for free
  - Faster shipping with DHL combined with low board prices may result in similar cost to dirtypcb.com