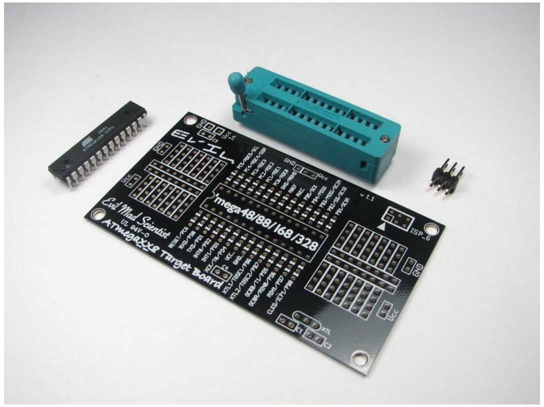
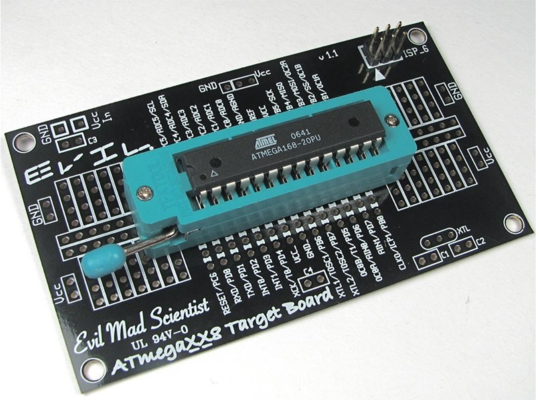
Introduction to Printed Circuit Board design,
For robotics,
using KiCAD

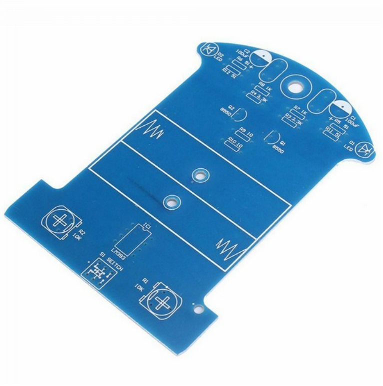
12/14/2019 Dennis Mangrobang





PCB
Printed Circuit Board

PCBA
PCB Assembly

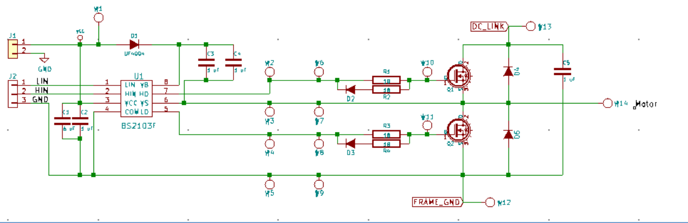


Agenda

- 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

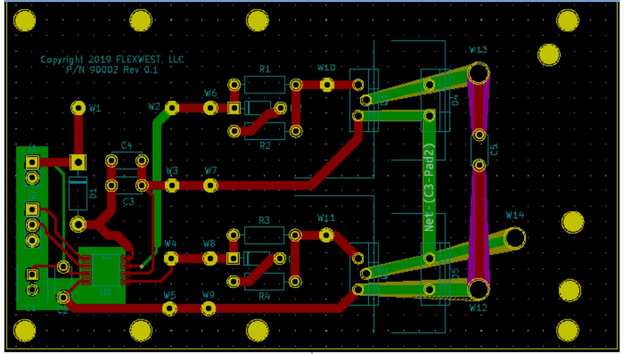
Typical PCB design/build workflow

Circuit Design: Schematic



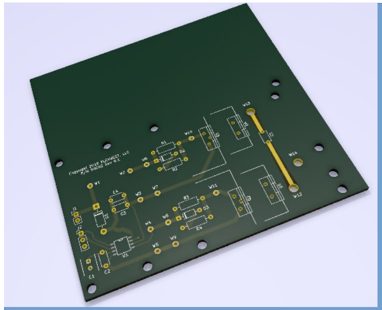
Design File: Netlilst

Physical Design: Layout

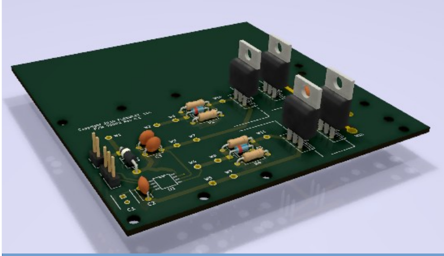


Design Files:

Gerber files
Drill file
Bill of Materials



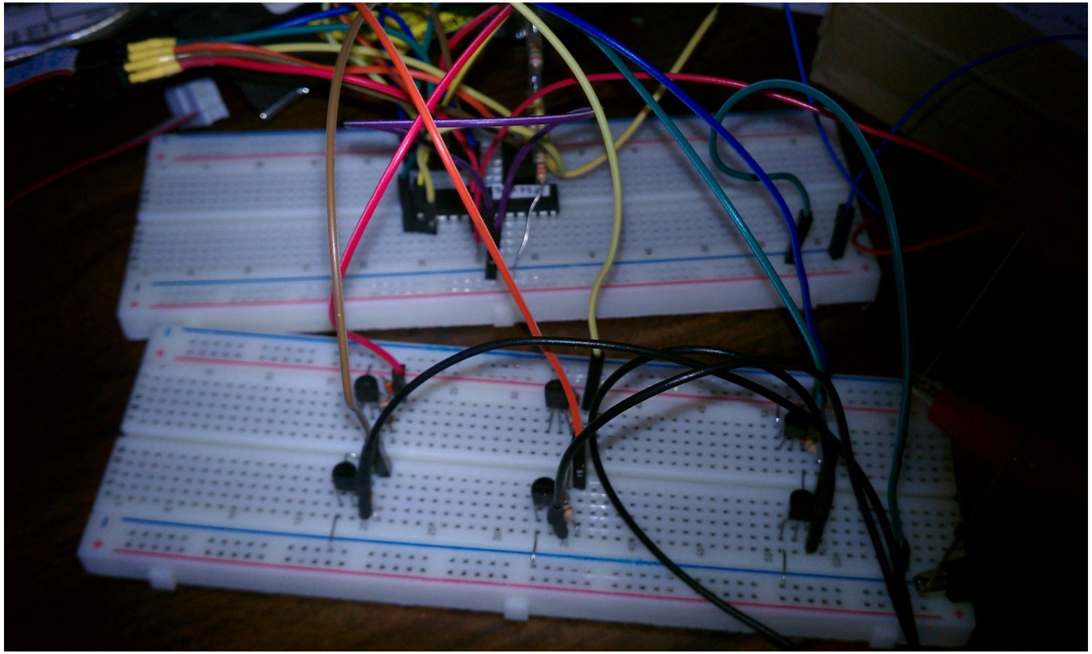
PCB: production



PCBA: Assembly

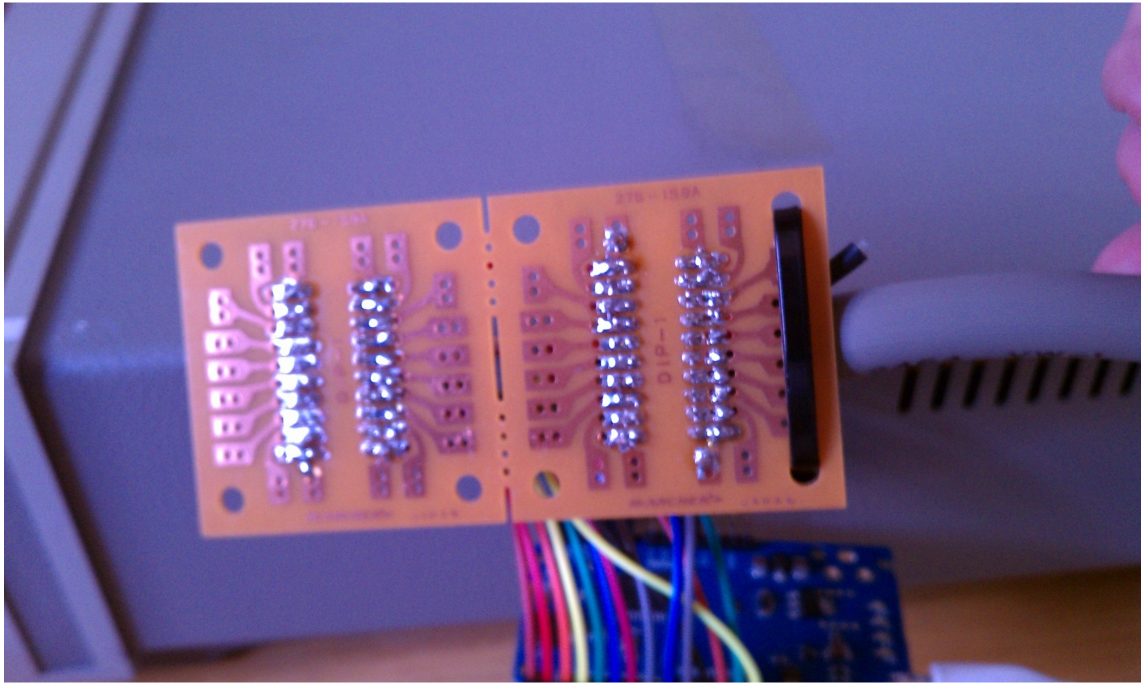
Alternatives to PCBs

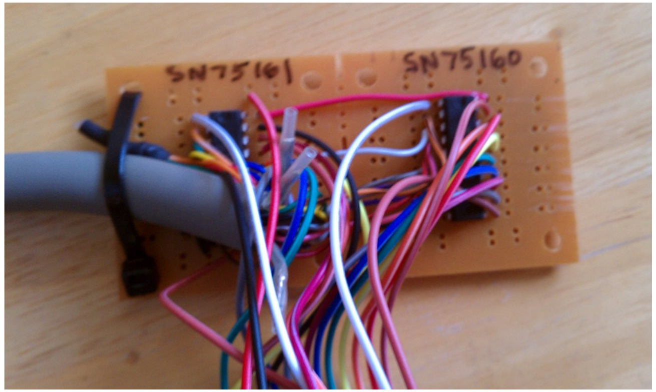
Solderless
Breadboard



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

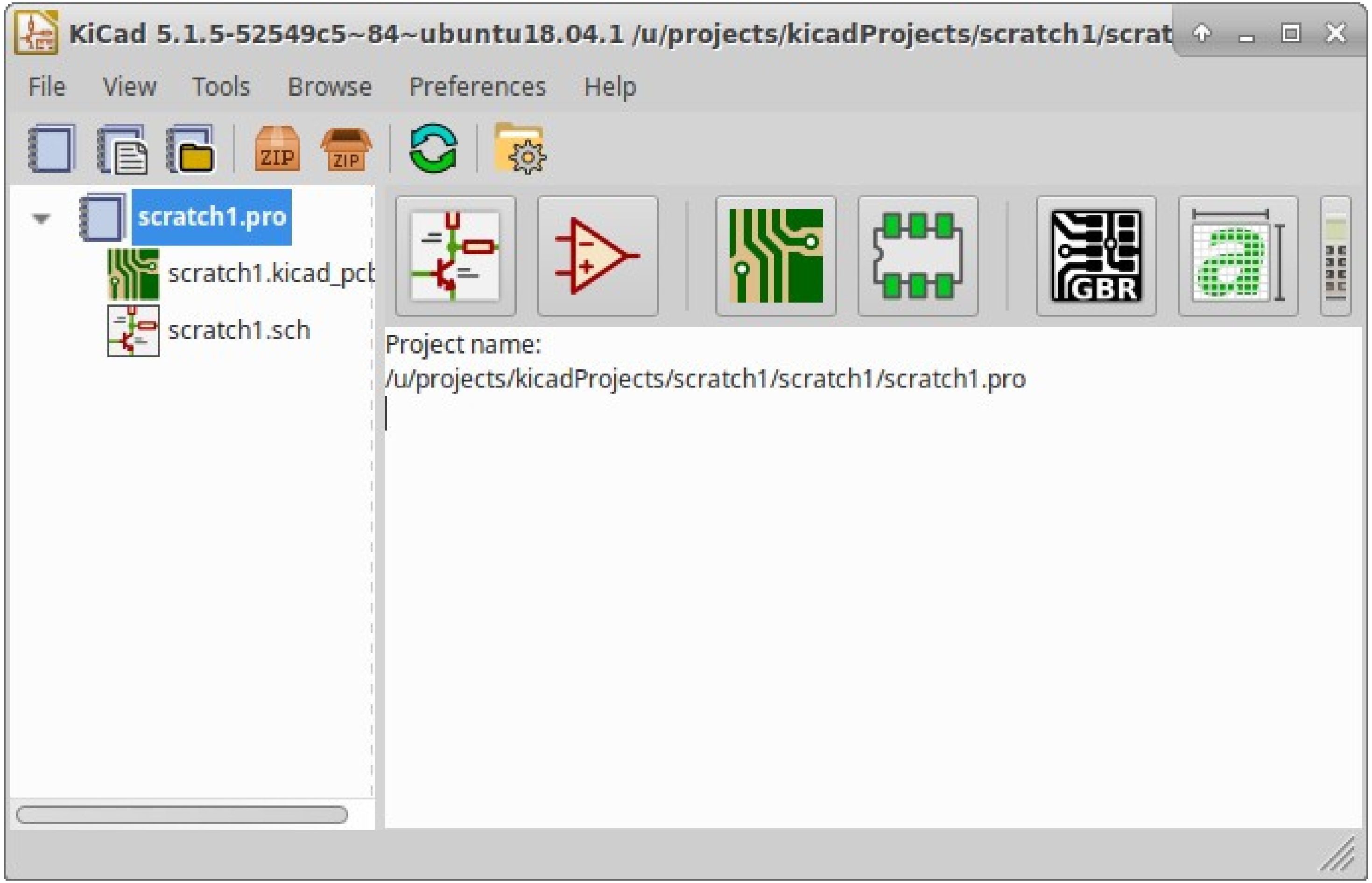
KiCad Overview

- 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
ZIP
ZIP
scratch1.pro
scratch1.kicad_pcb
scratch1.sch
GBR
Project name:
/u/projects/kicadProjects/scratch1/scratch1/scratch1.pro

KiCad – Create Project

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

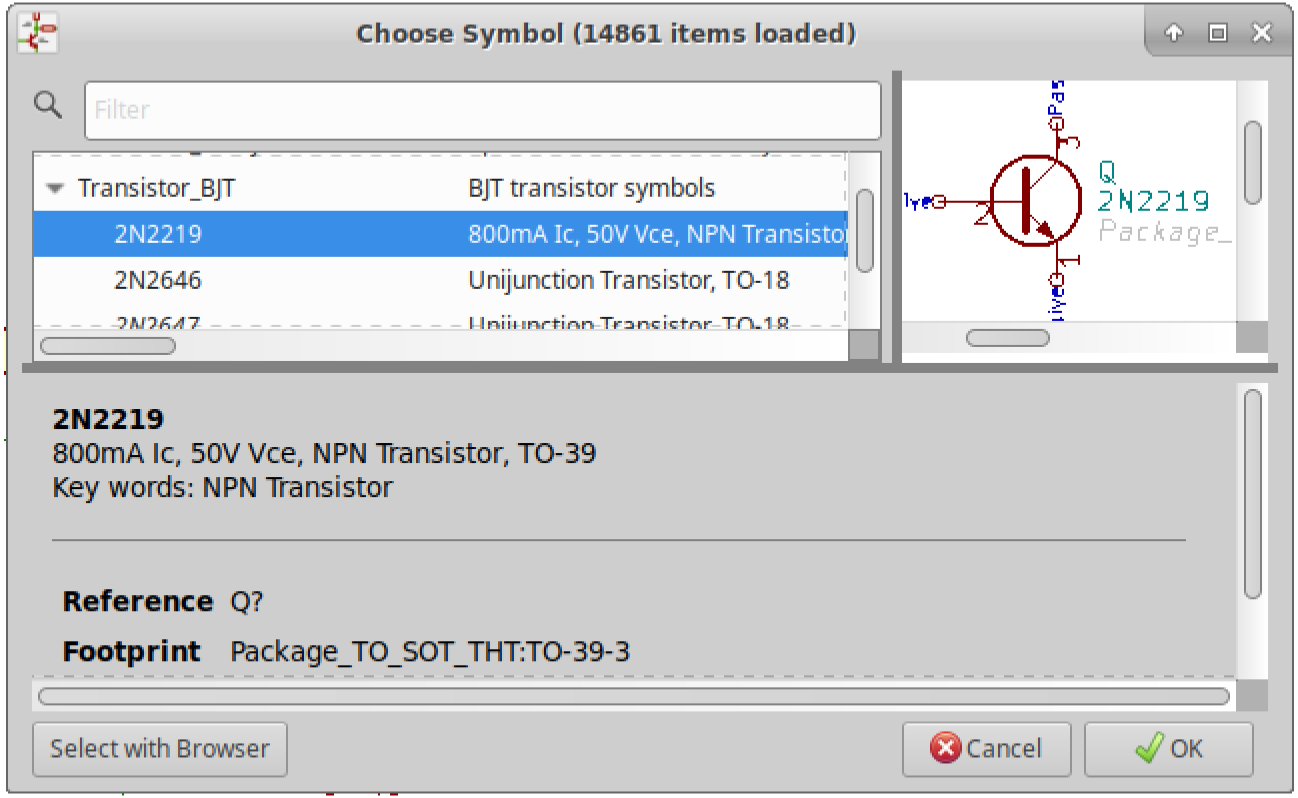
KiCad - Schematics

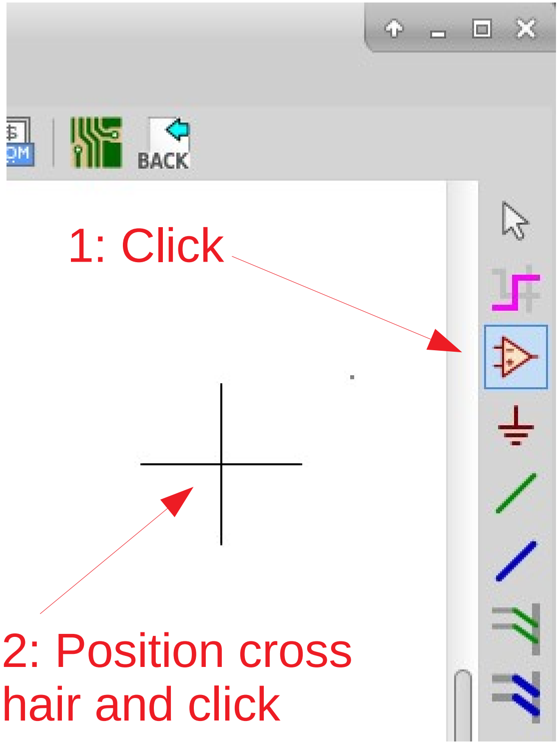
- Express the “logical” or “theorical” design of the circuit, rather than the physical realization
- Start Schematic Editor, “Eeschema”:
  - Tools→Edit Schematic   (Or click Icon)
- Schematic page settings
  - File→ Page Settings
  - Set paper, e.g. 8.5” x 11”, Landscape
  - Set Date, Revision, Title,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



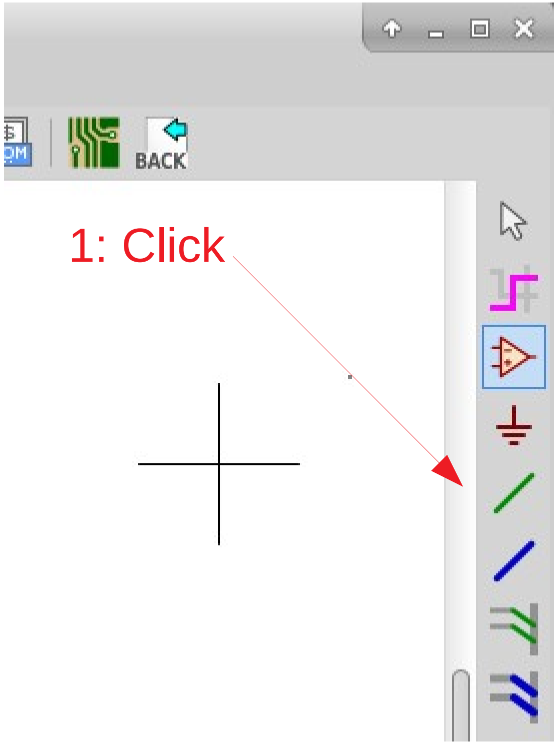


3: Choose symbol

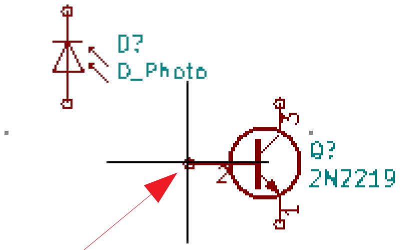
Kicad – Set symbol values

- 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)

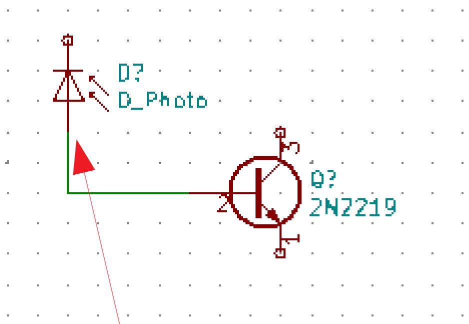
KiCad – Place wire connections



1: Click

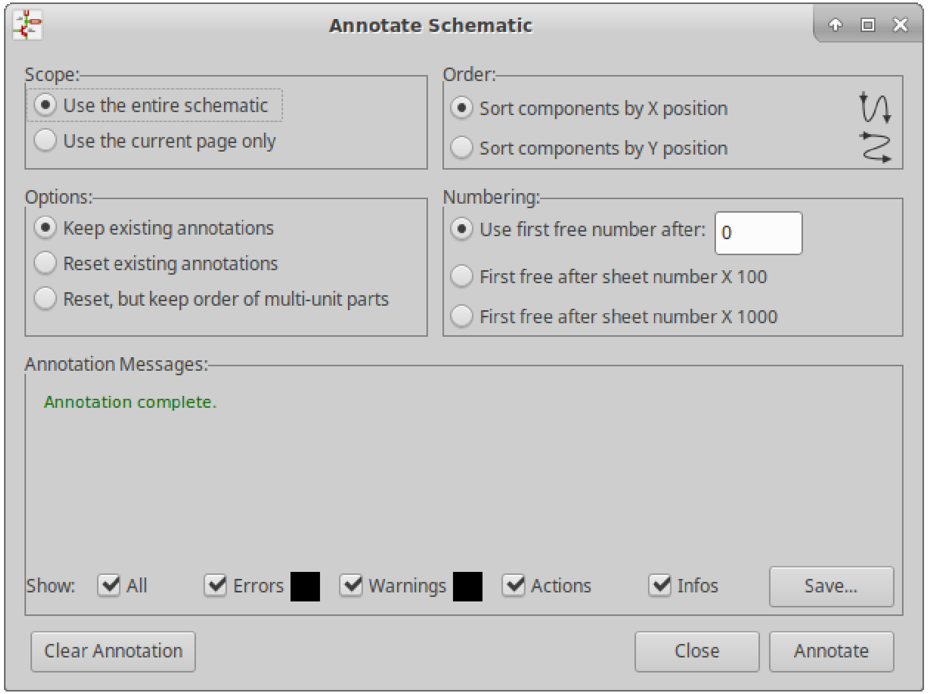


2: Position cross hair and click at start position (release mouse button)

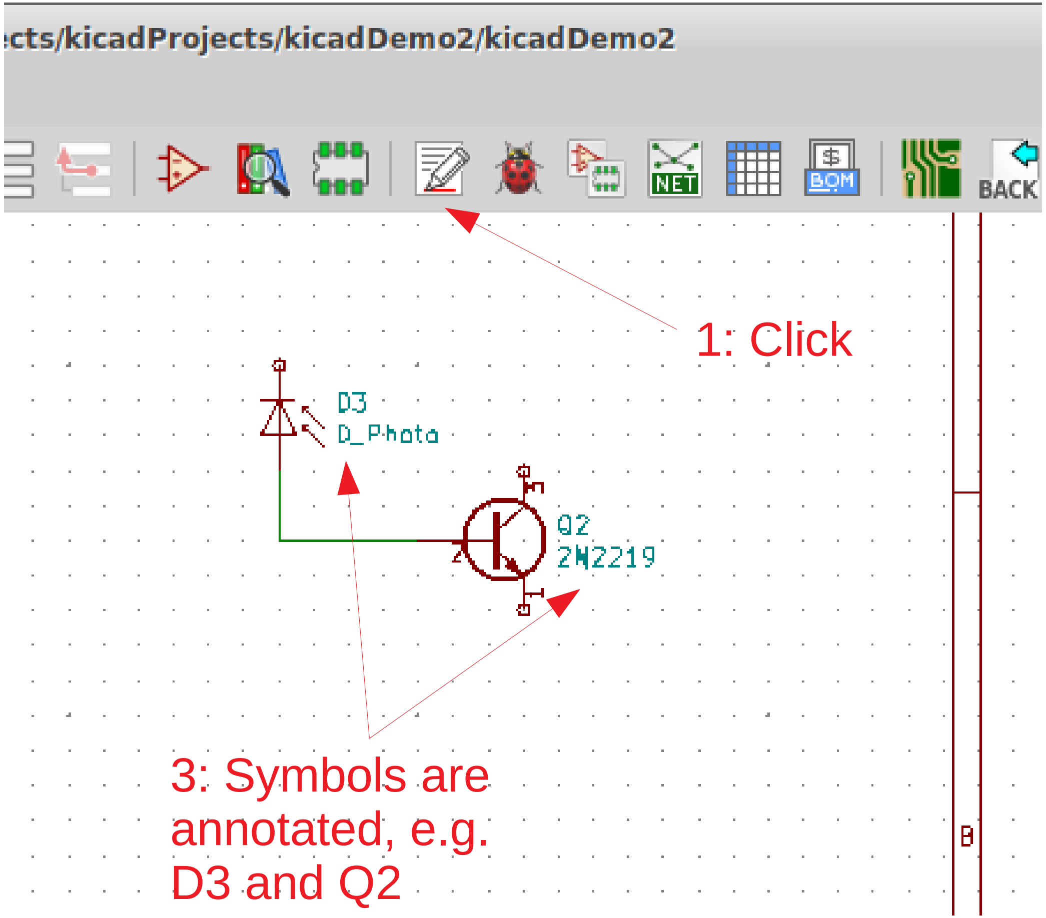


3: Move along desired path and click at end

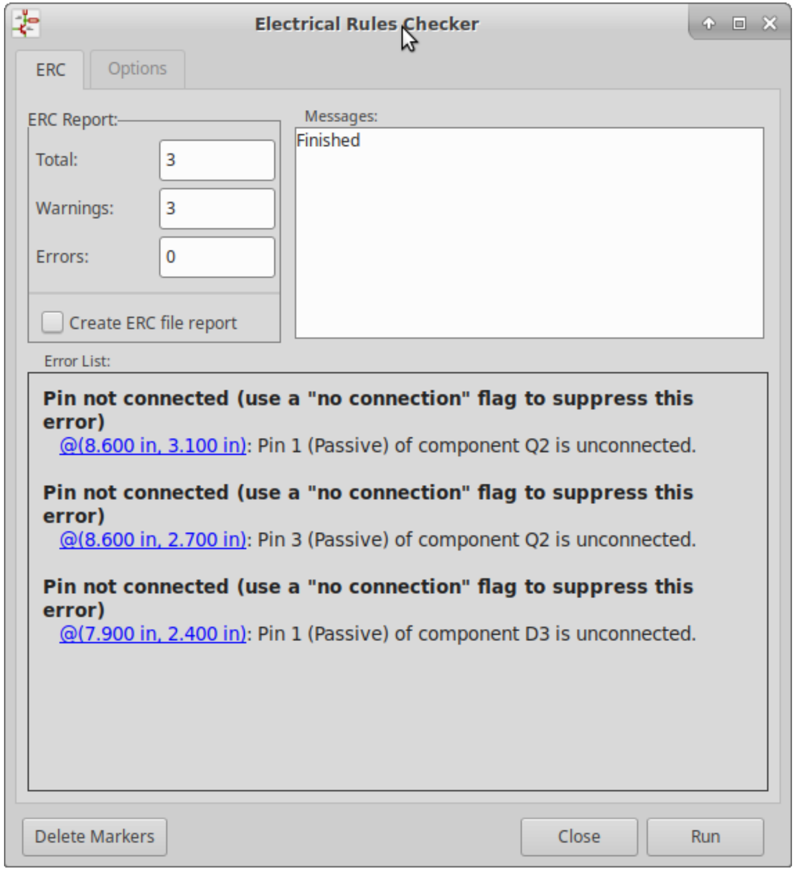
Annotate schematic



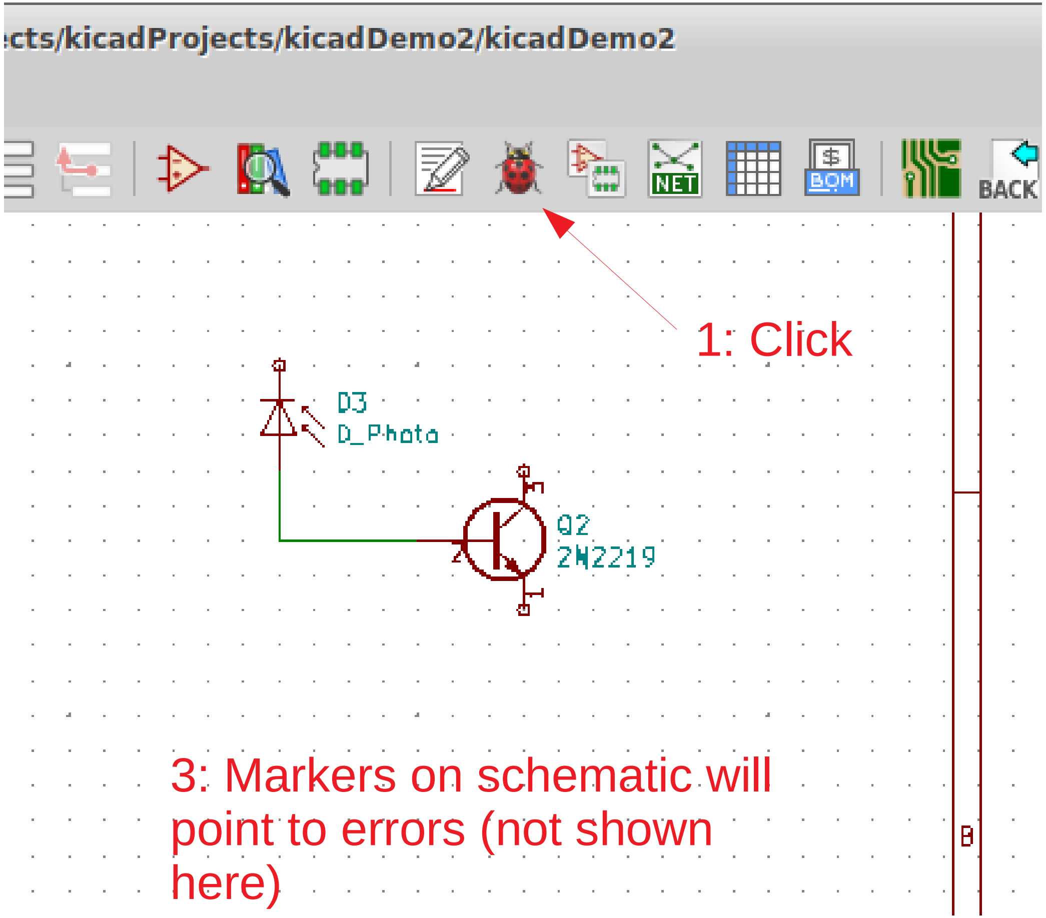
2:



Run schematic rules checker



2:

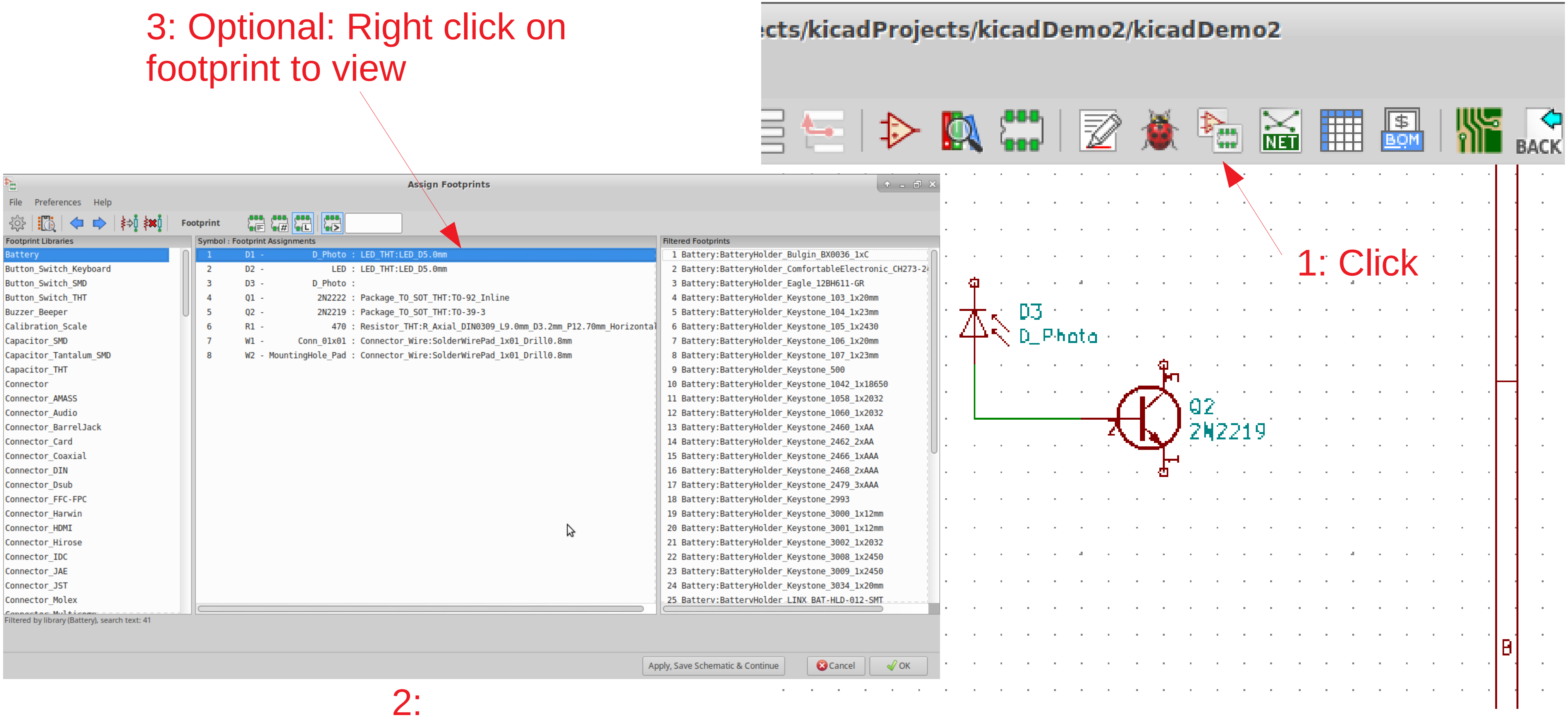


1: Click

3: Markers on schematic will point to errors (not shown here)

Associate footprints with symbols

3: Optional: Right click on footprint to view

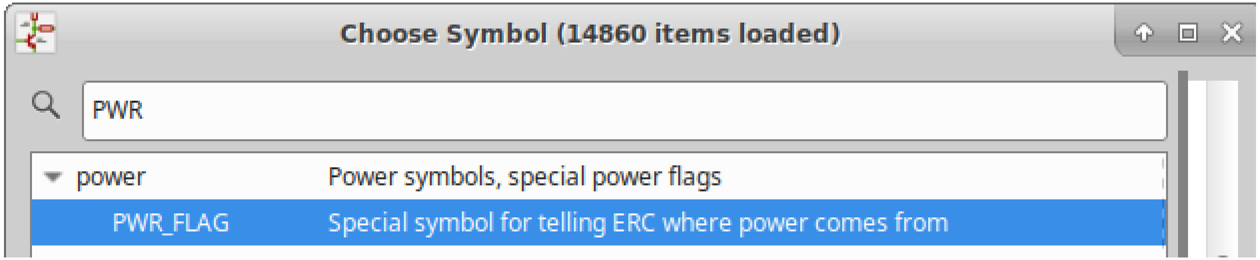


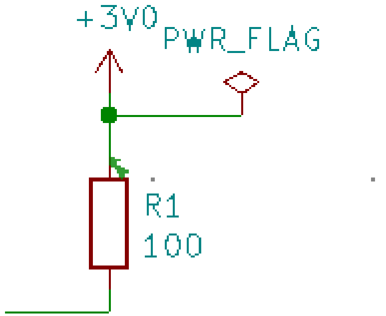
1: Click

2:

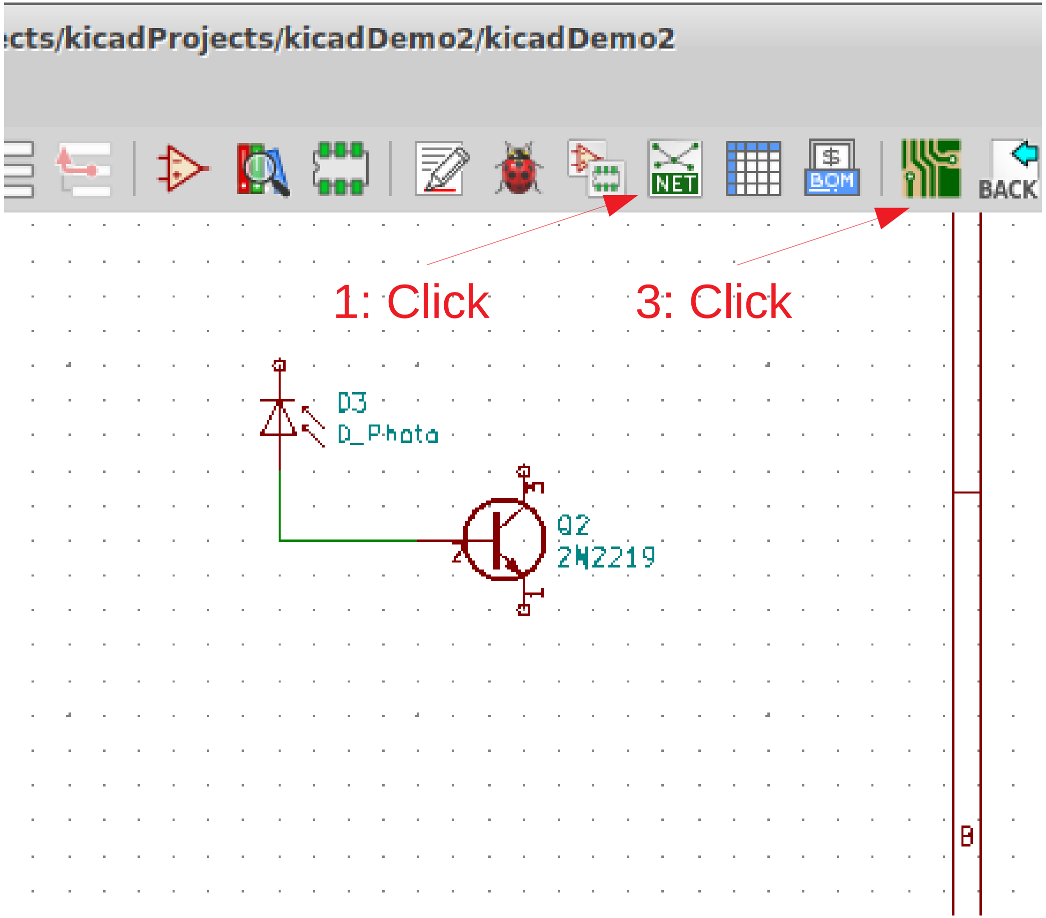
KiCad: Power Flags

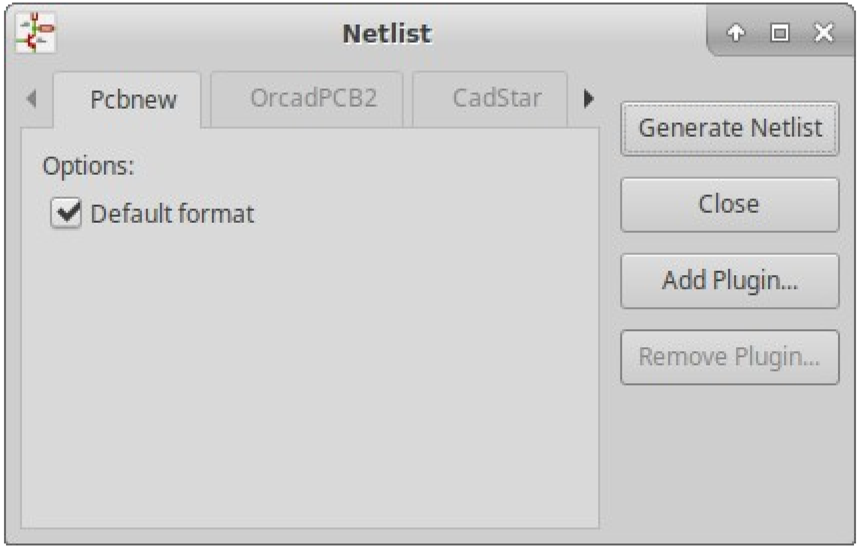
- A “Power flag” is a standard schematic symbol, used to indicate where power comes from.
- A “Power flag” should be connected to each GND and power supply rail.
- The Electrical Rules Check (ERC) will complain if these power flags are missing.





Generate netlist and start PCB editor

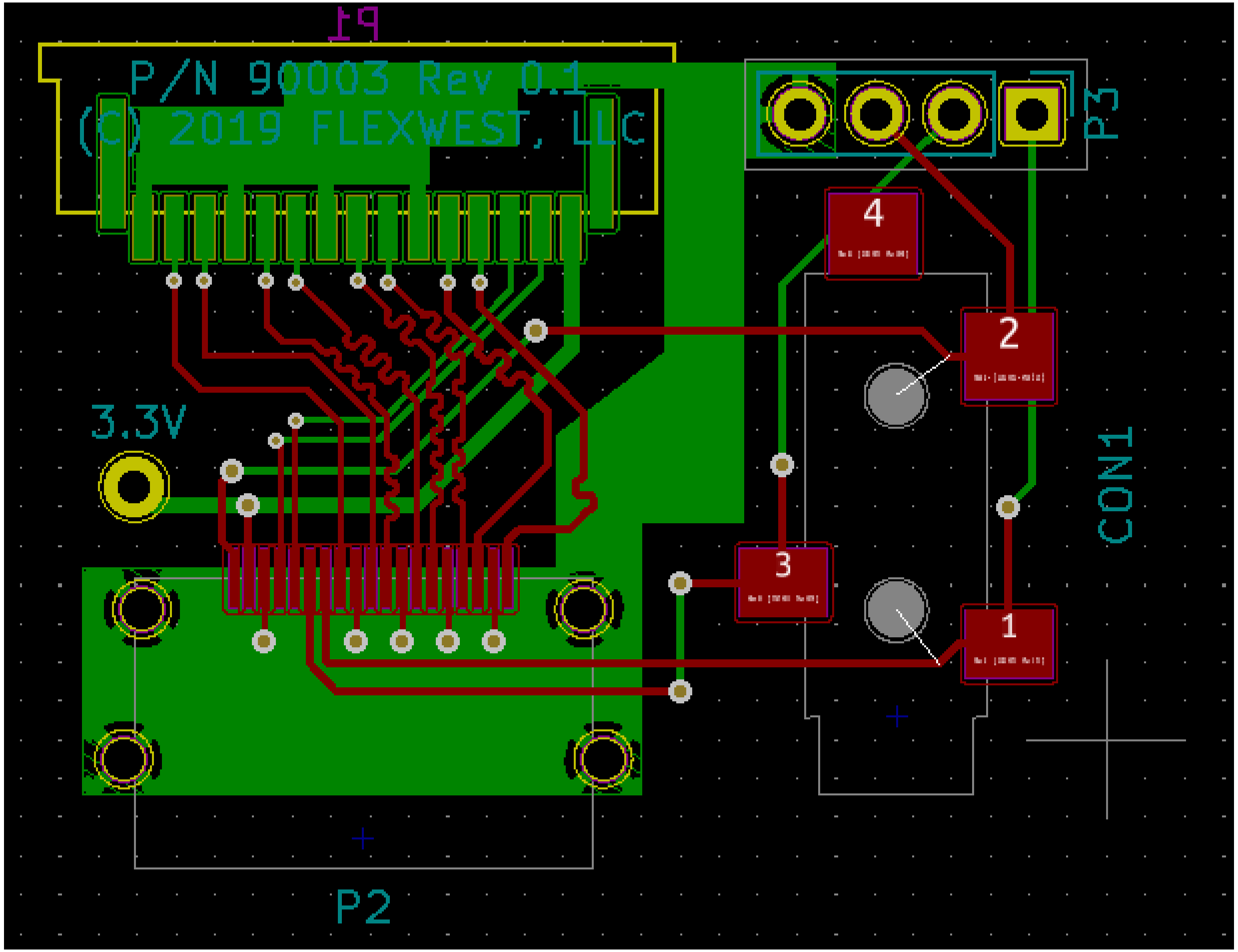


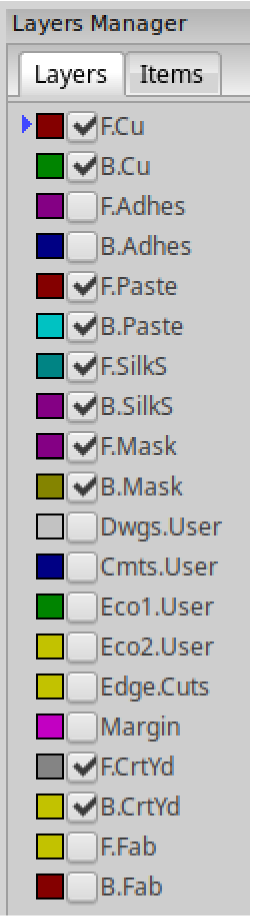


2:

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.)


P/N 90003 Rev 0.1
(C) 2019 FLEXWEST, LLC
3.3V
P3
4
2
3
1
CON1
P2


Layers Manager
Layers
Items
F.Cu
B.Cu
F.Adhes
B.Adhes
F.Paste
B.Paste
F.SilkS
B.SilkS
F.Mask
B.Mask
Dwgs.User
Cmts.User
Eco1.User
Eco2.User
Edge.Cuts
Margin
F.CrtYd
B.CrtYd
F.Fab
B.Fab

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→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)

ITEM	CAPABILITY
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!
Min internal slot	32mil (0.8mm)
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

https://dirtypcbs.com/store/pcbs

Sign In/Register

Search...

- Shop
- Cart
- Categories
- Printed Circuit Boards
- PCB Stencils
- SLA 3D Prints
- Laser Cut Acrylic
- Custom Cables
- Dirty BOM
- Other Services

PCBs

Full PCB specs and info

Remove

File	Material	Layers	Quantity	Price
Browse... No file selected.	FR4 proto	2	Protopack ±10	
Size	**Size (quantity 10+)**	**Color**	**Custom silk/mask color**	$16.95
max 10x10 +$5	10 x 10 cm	Green		
Thickness	**Coating**	**Copper**	**Stencil**	
1.6mm	HASL	1oz	None	

Advanced Options

Total	$ 16.95

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