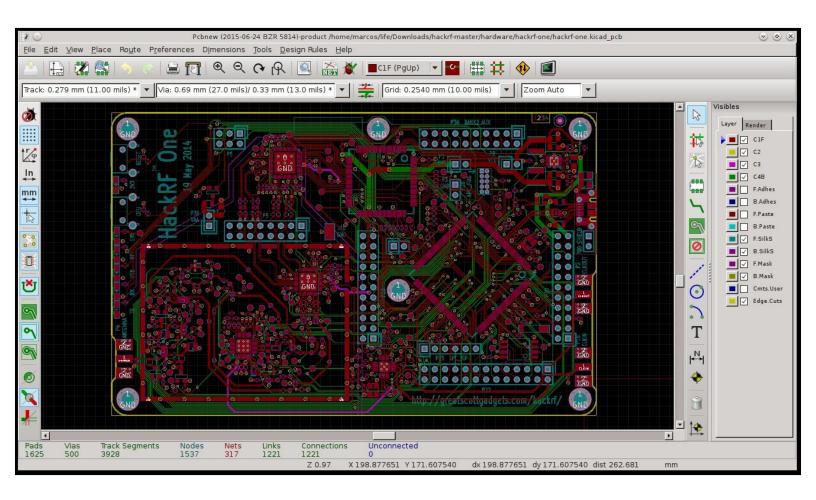


Creating PCBs In KiCAD

A guide to creating your first PCB in KiCAD, a free, open-source PCB design software.

Written By: Nathan Dillon Martin



Step 1 — Create a new Project

Coll Coll		
--	--	--

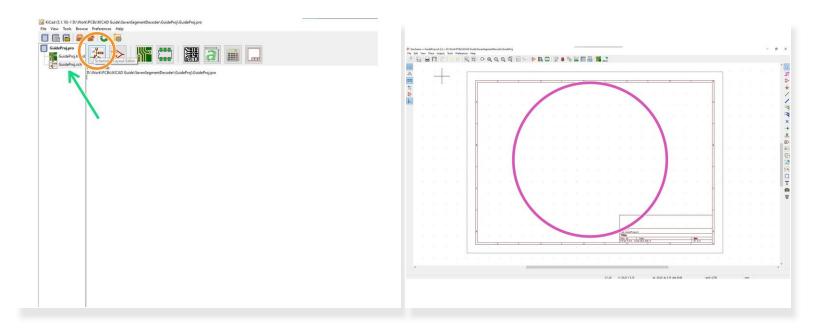
闭 Open KiCAD

- Create a project by navigating File -> New -> Project
 - Enter your project name in the following box

(i) New projects will have 2 associated files

- .sch Schematic document. Where you will build the circuit diagram (similar to circuitlab)
- .kicad_pcb PCB Layout file. This is where the physical PCB layout and routing will be stored

Step 2 — Open the Schematic Layout Editor



- Click on the schematic layout editor button
 - Alternatively, you can double click on your .sch file
- The schematic layout editor is where we will define the connections throughout the PCB
 - (i) Later on, the PCB editor will use these connections to draw guide lines for the PCB connections

Step 3 — Navigating the Schematic Layout Editor

	File Edit View Place Inspect Tools Preferences He						
	Save Chi+S) C Q Q Q 🗟 🗟 🕨 🗛 🛱 🖉 🍇 🖀 🖩 🔠	MCK.				
tertemetissient h. Diet NAKAD bei fantemetisen	Save Current Sheet						
Fire hard bell Antonin My	Save Current Sheet As Ctrl+Shift+S						
	Append Schewalic Sheet Content						
	Import Non KiCad Schematic			Page Settings		~	
	• Import			Paper	Title Block Facameters	^	
/	P Expert >				Number of sheets: 1 Sheet number: 1		
					houe Date		-
	Page Settings.			Diestainer		ort to other sheets	
x x x x x x x x x x x x x x x x x x x	Print_ Ctrl+P			concept.	Reison	ort to other sheets	
*	T-T Pist-			Custom paper size	The	A to one inces	
A				Height 279,400 mm	□top	ort to other sheets	-
	tox box			100 0000	Company	ort to other sheets	· .
				Layout Preview	Commenti	A to other sheets	
					Disp	ort to other sheets	
					Comment2	ort to other sheets	H
T					Commental	et to other sheets	
•						ort to other sheets	
					Commenta	ort to other sheets	
				her a	Page layout description file	A to other cheets	. 1
						Brevoe	
					OK	Cancel	
						_ circu	
							. 1
							. 1
							- 11

Navigating the Screen

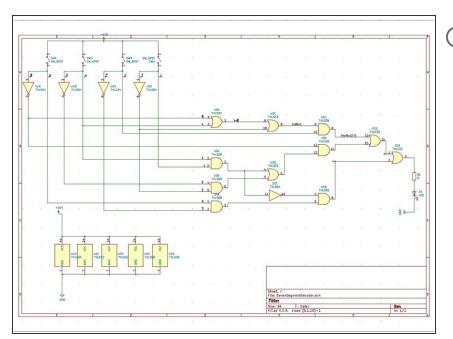
- Middle click to pan
 - Scroll wheel to zoom
- Left click to select (Hold and drag to select multiple components)
 - Left click and drag to move selected components
- Right click to get other options
- (i) If your design is large, you will want to redefine the schematic area. This step is optional, but will make your schematic look much better.
 - File -> page settings -> Select desired page size. You can also select User(custom) to use any size.

Step 4 — Placing Components

	4 H B T 🖞 🔸 🥐 🔍 6 C Q Q Q E S 🕨 🗱 🛱 🛛 🖉 🕸 🗄	E 6 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
/ Wm Solt-W	Choose Symbol (1635) items loaded)	-
1 >	in File	
Bus to Bus Daty Alte-/	P Pecently Used	
X No Connect Flag SHN-Q		U78 400
A Level Store	B 4009 Hex Buffer Inverter 1000	
Contraction Alt-Cirl-H		- D Fortball
AD Hearthical Label Shift-H	A 4012 Dual 4 inputs Nand A 4013 Dual D FileFileo. Set & reset	The same of the sa
Report Hearchical Label	4016 Quad Analog Switches 4017 Johnson Counter (10 outputs)	E Zoom in FI
DA Hearchical Pin to Sheet	4020 Binary counter 14 stages (asympton,	
Googbic Phylone Shift+1 The Googbic Part Shift+1	< >	C+ Redrov View F3 Q Zoom to Fit. Home
improvement protection of the second se	4001 Guid Nor 2 inputs Key weeds: CMIOS Nor2	Q Zoom >
		Grad and a state of the state o
	Reference U/A	X Close
	Select with Browser OK Cancel	

- Press the "Place" tab on the top ribbon to begin placing components
 - Select the type of component you want (symbol, power, wire, label)
- Clicking on the schematic will then place the component.
 - For components, power ports, and labels, this will open a selection screen to choose the component. Double click on the item that you want to place it.
 - For wires, clicking will place the first section of a wire. Clicking into open space will create a junction, and clicking on a component terminal will connect the wire.
 - (i) Right click and select "wire end" to terminate a wire without a connection.
- Right click and select "end tool" to stop placing components.

Step 5 — Build Schematic



(i) Continue placing components and wires onto the schematic until to complete your design

Step 6 — Assign Footprints

	Annotate Schematic		×			
	Exporting the netlist requires a completely annotated so	hematic.		Contract No.		- 0
ana — Guidefing Jash (r) — 01/06/4/FCR000CAD Guide JauendogmentDecoder/Guidefing	Scoper	Orden		2 E + + 11 M		
2 Von Poor Ingent Solo Federacco Hills 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Use the entire schematic Use the current page only	Sort components by X position Sort components by Y position	tş i	Versioner years caught caught wing	 Report topped Statistic style 	Teach lease Training and the formation of the formation
1 🧖	Options	Numbering:	4	aping_Hagnan error_[10] docad Jour		 M. 2011 M. 2011 M. 2011 Mar, 2011 Mar, 2011 Mar, 2011 Mar, 2011 Mar, 2011 M. 2011 M. 2011 Mar, 2011 Mar
	Keep existing annotations	Use first free number after: 0		et eteck every_MB		HOM HOLMOLING HUDDRING HUDRING HUDRING HUMING HUDRING HUDRING HUDRING
	Reset existing annotations	O First free after sheet number X 100		464714_387 4947		14 Mill 201 Mill 20, 100 Jun 201 Jun 2
А.	Reset, but keep order of multi-unit parts	First free after sheet number X 1000		0,00 0,00 00		14 MJ, 201 MJ, 35 MB 21 MJ, 201 MJ, 35 MB 31 MJ, 201 MJ, 35 MB 31 MJ, 201 MJ, 35 MJ, 36 MJ 31 MJ, 201 MJ, 35 MJ, 35 MJ, 31 MJ
	Annotation Messages:			unnoficipaen unnoficipaen etta		[1] MD, 2011 MD, 201. See, 2012 Status, 41, 27 and 201 MB, 201 MB, 201 MB, 201 MB, 201 MB, 201 MB, 2012 MB,
				estante estante estante estanjiti estanjiti estanjiti estanjiti		 M., 201 M., 20. Im, Restormal, G. 296, 25. Im, 20. Im, 2019; M., 201 M., 201 M., 2010, Im, Restormal, 201 206, 204, 104 M. 201 M., 2010, Restormal, 201 206, 204, 204
				CH4P_111 CH4P_111		34 582, 201 583, 30, 466, Revision A., 24, 706, 25, 566 25 582, 201 583, 30, 466, Revision AJ, 25, 706, 26, 566
						 M.S., 2011 M.S., 201, 2011 (2011), 211 (2
						21 Mit, 201 Mit, 30, 560, Responsed, M. Stan, 215, 566 (1997) 201 Mit, 201 Aug. 201 aug.
				<0xp,319 <0xp,30		32 343,255 343,25 584,25 594,25 594 33 343,255 343,25 596 31 343,255 345,25 596
				class 11 UF Mb		24 142, 201 142, 20, 5m-1 11 142, 201 143, 20, 5m-1 12 142, 201 143, 20, 5m-1
				elisar, 31, 317, 317 exitoarea, 50 exitoarea, 50		N 10.20100.01000 11.00.20100.0100000000000
				044,380 044,390		25 182, 202 183, 25, 160, (144)
				414/01,000		41 182, 2021 283, 25, 56, 56 (2011) 2014, 21, 276, 23, 56 41 182, 2021 283, 25, 76, 76 (2011) 2014, 21, 276, 23, 56, 7244
				Automa		41 145,205 145,20,00,001,001,01,200,00,000,001,000,000
				2,000		41 182, 201 182, 20, 566, 511 (1994); C. J'an, 20, 568 (1997); W. W. Wei, Nucl. Phys. Rev. Lett. 10, 1108 (1997); W. W. Wei, Nucl. Phys. Rev. Lett. 10, 1108 (1997); W. W. Wei, Nucl. Phys. Rev. Lett. 10, 1108 (1997); W. Wei, W. Wei, Nucl. Phys. Rev. Lett. 10, 1108 (1997); W. Wei, W. Wei, Nucl. Phys. Rev. Lett. 10, 1108 (1997); W. Wei, W. Wei, Wei, Wei, Wei, Wei, Wei, Wei, Wei,
8				(Junite United		41 182,202 182,25 5m,24111444,25 5m, 41 182,202 182,25 5m,24111444,25 5m,
				Charleng AND AND AND AND AND AND AND AND AND AND AND AND AND AND AND AND		31 142, 202 142, 25 Sec, 26 Lines, 26 Lines, 25 Sec, 31 142, 202 142, 26 Sec, 26 Lines, 26 Lines, 26 Sec, 26 Sec.
Like and the second sec	Show: MAIL MErrors MWarrings MA	uctions 🖂 lafes	Save	emption Models fold-blir Menugh Nate, common anoth 1460 film ony location C-Program File: 60cmLature bisadaecable.cliff, 747 are	i phonin, Mpiclan Anguateluk andakatan N.A'99066449, Kujuan Kili Bi Awini kapi Au Kria	
	Clear Annotation	Annotate	Close			
	·					

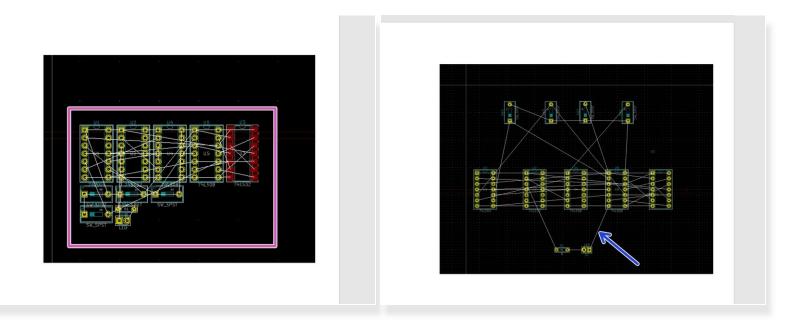
- A footprint is the shape of a PCB component. It is what is actually built into a PCB for the device to be soldered onto
 - Without the right footprint, a component won't fit onto the PCB and won't be able to work in the circuit. If you don't know what footprint to use for a component, the internet is an incredible source
- To assign footprints, click "Assign footprints to schematic symbols" on the top ribbon
 - This may bring up a popup saying "Exporting the netlist requires a completely annotated schematic". Pressing the annotate button will add annotations and continue to footprint assignment
 - Annotations are a letter and number combination that are used to define what each component on the board is and reference its position. They are also used to define which gates are being used on a multi-gate integrated circuit
- You should now be in the footprint assignment menu. This menu has three sections: on the left is the footprint categories, in the center are the components on the schematic, and on the right are a filtered set of footprints (based on what you selected on the left side)
 - To assign a footprint, select a category from the left side, click on the component you want to assign it to, and then double click on the specific footprint you want to use.
 - Repeat for each component on the schematic!

Step 7 — Send Schematic to PCB Layout Editor



- (i) Now that your schematic is built and footprints are assigned, we can design the physical PCB
- Press the "run pcbnew to layout printed circuitboard button"
 - This will open the PCB layout editor and import your schematic
 - There may be some errors here if you have not defined a footprint for every component in the schematic (missing footprints will be listed)
- To open the PCB layout editor subsequent times, click on the "Open PCB layout editor" button or double click on your .kicad_pcb file
- To reupdate the PCB with any changes made to the schematic, press the "Update PCB from schematic" button on the top ribbon

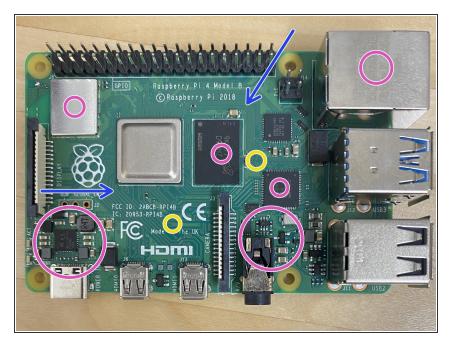
Step 8 — Layout Components



- When the PCB is generated, your components will be arranged similarly to this.
- Left click on a component and drag it to the position you want on the board
 - The white lines show each of the connections from your schematic, so they will move around as the components are arranged.

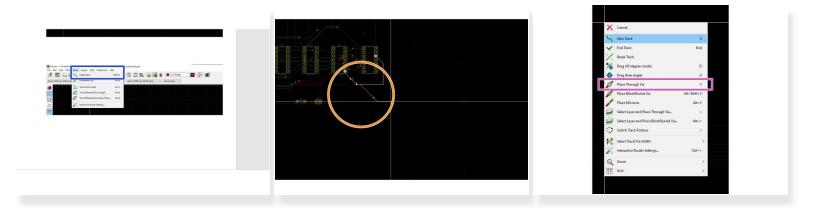
Don't worry if you don't arrange the board perfectly right away, you can always change things later

Step 9 — PCB Fundamentals



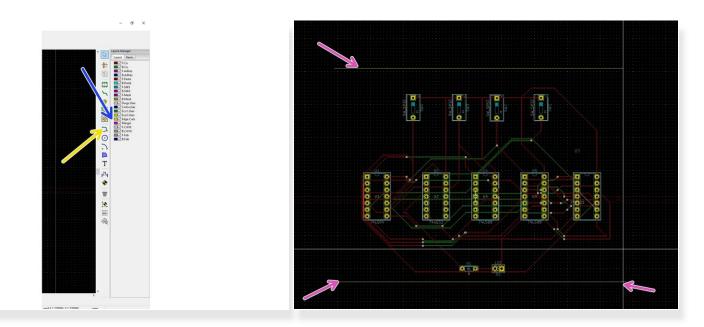
- Here's a quick rundown of the Terminology we will use when laying out a PCB
- Component/ Device: Anything that goes onto a PCB. Could be a Resistor, Integrated Circuit, Pin Connection, or many other things
- Trace: A Section of the PCB with copper that connects components
- Layer: A sheet of copper containing traces. Basic PCBs will have 2 layers, while more complex PCBs typically have anywhere from 3 to 8 layers.
- Via: Small hole in the board that passes electricity across PCB layers. This allows for traces to "cross" and more complex designs to be made.
- Gerber Files: Files used to manufacture PCBs. They consist of many different "layers" that define what operations should be done on the PCB.

Step 10 — Route Wires



- To route traces, select Route -> Single Track.
- The next place that you click will become the start of a trace. Moving the mouse will cause the software to find the fastest available route from the starting component to the ending component.
 - Clicking in empty space will create a point that the trace must pass through
- Eventually, there will be no more room on the front of the board. Right click and select "Place Through Via."
 - The next place that you click will create a via and turn the trace green. This is a trace routing on the other side of the PCB!

Step 11 — Define Edges



(i) Now we need to define the edges of the PCB

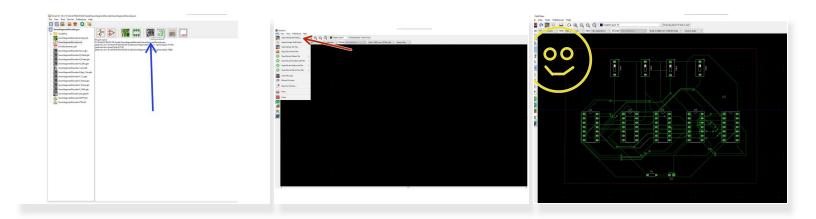
- On the right side of the screen, select the "Edge.cuts" layer
- Once this layer is selected, select the "add graphic lines" button
- Click and draw to define the footprint of the PCB

Step 12 — Export PCB

Edit View Place Route Inspect Tools Preferences Help	Plot	×			1
Gene Place Roote Implement See Preferences Here Copy Ab Cele Shifts Implement Implement Implement Copy Ab Cele Shifts Implement Implement Implement a Cele Shifts Implement Implement Implement a Cele Shifts Implement Implement Implement a Shifts Implement Implement Implement a Shifts Implement Implement Implement Shifts Implement Implement Implement Implement Shifts Implement Implement Implement Implement	Part General General Control C		Output folder: Dull File Format Dull File Format Dull File Format Dull File Format Dull Auf DPI in single file Out Holes Dull Mode Bil Ein aude command recommended) Dies alternate dill mode Geber X2 (experimental) Map File Format D+PGL Bil Format	Dill Origin © Abrolate © Auxiliary axis Dull Units Millimeters © Inches Zeros Format © Suppress Italing zeros © Suppress Italing zeros © Keep zeros	Hole Counts Plated packs B2 Non-plated pack 0 Through vias 39 Micro vias 0 Burled vias 0
Print Ctrl+P Plat Archive Footprints	F.Critid Formid Formid Forb Scritid Forb Subtract soldermask from sillscreen Subtract soldermask from sillscreen	⊠Use extended X2 format ⊠Include netifist attributes	○ Gerber ○ DXF ○ SVG ○ PDF	Precision: 2:4	
	Stow All Stors Wanings Stations E	Stre.	Messages Generate Report File		Generate Drill File Close Generate Map

- Starting in the PCB Layout editor, enter the plotting menu
 - File -> Plot
- Make sure that the following layers are selected: Top Copper (F.Cu)+ Soldermask (F.Mask) + Silkscreen (F.SilkS) Bottom Copper (B.Cu) + Soldermask (B.Mask) + Silkscreen (B.SilkS) Board outline (Edge.Cuts)
 - (i) These are the most basic required layers for the PCB. For more complex boards, more layers may be used
- Generate the Drill Files (Default settings are fine)
- Press the Plot button to generate the gerber file!

Step 13 — Using the Gerber File Viewer



- Click on the gerber viewer from the main page
- Select file -> open Gerber plot files
- Select all of the gerber files that were just generated
- Admire your beautiful new PCB!