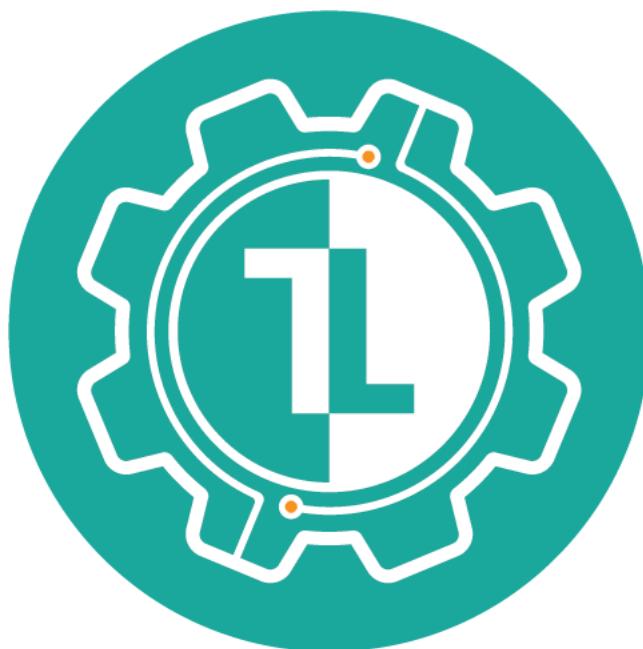


**HO CHI MINH CITY UNIVERSITY OF
TECHNOLOGY**

TICKLAB



KICAD TUTORIAL

MENTOR: BUI HUY PHUC

MENTEE: NGUYEN NGOC THANH TUAN

TICKLAB, 2019



- ## **1 Index :**
- a) ACCESS TO KICAD SOFTWARE.
 - b) INTRODUCTION ABOUT ELECTRONIC CIRCUIT.
 - c) SOME LITTLE PROBLEM WITH CIRCUIT.
 - d) SCHEMATIC AND SCHEMATIC LIBRARY.
 - e) FOOTPRINT LIBRARY AND HOW TO ADD FOOT-
PRINT TO SCHEMATIC.
 - f) PCB AND THE COMPLETELY CIRCUIT.
 - g) CONCLUSION AND SUMMARY.



2 ACCESS TO KICAD SOFTWARE

Before we learn anything about the circuit, we will research how to access kicad software. So the first question we have is what is kicad ???

KICAD is an open-source software program used to design schematic and pcb circuit. KICAD also known as a system of some independent software. There are:

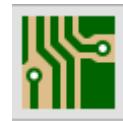
1) Schematic Layout Editor: a circuit drawing program (support to draw schematic and make schematic library.)



2) Symbol Editor: a window with some tool help make schematic library.

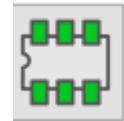


3) PCB Layout Editor: a program have tools to draw pcb circuit.





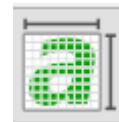
4)Footprint Editor: window have footprints to add to schematic layout, each symbol was added the footprint will be display in pcb circuit.



5)Gerber Viewer: support to observe gerber file.



6) Bitmap to Component Converter: convert Bitmap images to schematic or pcb components.

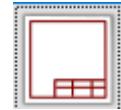


7)PCB Calculator: run components calculation, track width calculation,...





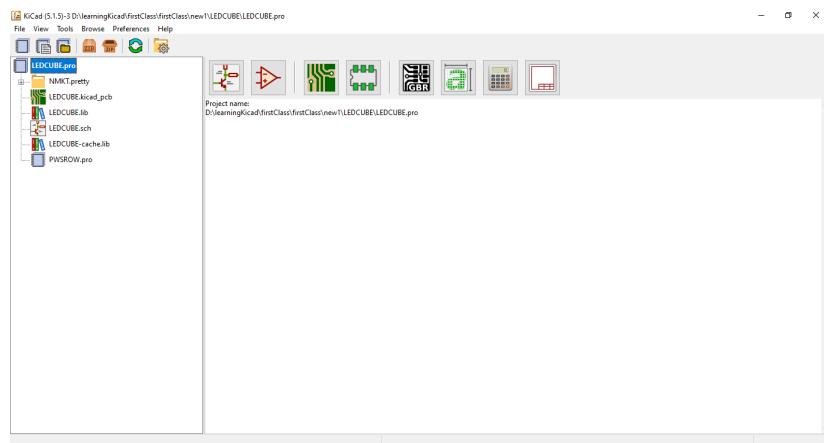
8)Page Layout Editor: Edit worksheet graphics and text.



Now we will download KICAD at website <https://kicad-pcb.org>, when we access to this page, at the center we also see the download button. After click this button, select your operating system or distribution and your version(64-bit or 32-bit). KICAD will be download into your computer. There is KICAD symbol, which display on your desktop.



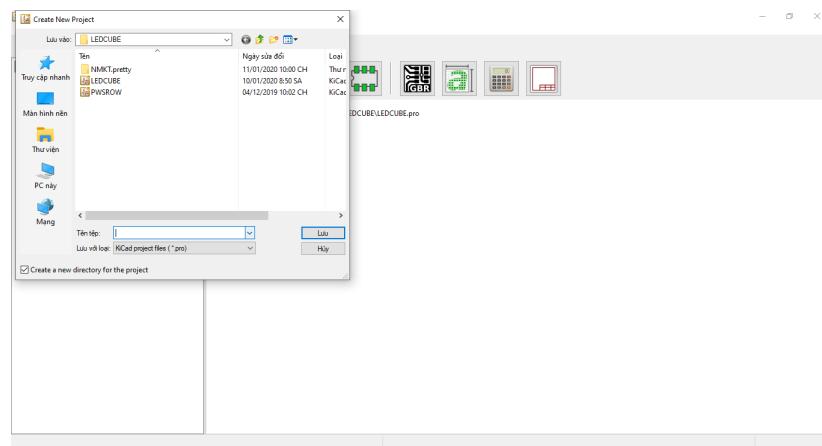
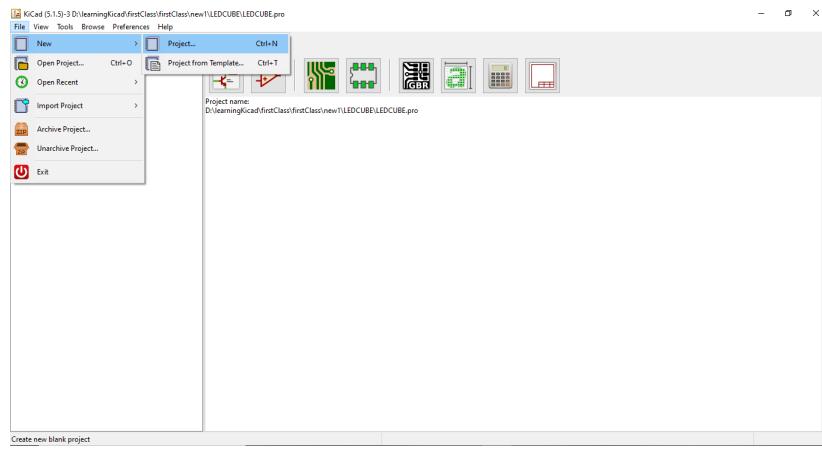
At present to start your project double click in the symbol, you will see the start-screen like there.



Begin your KICAD project, you choose File -> New-> Project(



or Ctrl + N). A folder with .pro at footer will be made, in this folder you also can make your schematic or pcb file.



And now let's begin your project with KICAD program.



3 INTRODUCTION ABOUT ELECTRONIC CIRCUIT

Firstly, we will research what is PCB? PCB stand for Printed Circuit Board, a multilayer printed circuit board and also non-conductive. All electronic components are connected together and on the circuit board and have a support bottom.

Secondly, why we use PCB circuit? Before PCB is invented, components connected by wires so the boards are very complex and non-reliable and make a complex-board like motherboard is impossible. With PCB, components is connected wirelessly and internally, thus reducing the overall circuit design complexity.

Thirdly, there are some types of PCB circuit: each types is used for another applications. There are:

- +Single-Layer PCB.
- +Double-Layer PCB.
- +Multi-Layer PCB.
- +Flexible PCB.
- +Aluminium Backed PCB.
- +Flex-Rigid PCB,etc.

Fourly, a PCB is made up of several layers of conductive and non-condutive materials arrange together to form as a board. Now , we will research elements of PCB circuit.

Substate (or PCB material): a non-conductive, rigid and com-



pact material. The most usual substatre is FR4, a fiberglass material.

A conductive-material(usually is copper):be used as a wire to connect components together.

Soldermask: this layer is above the conductive material and cover the entire circuit except the components pad.

This layer has some important advantages:

- Against the oxyation.

- Isolating between component footprints and printed circuit lines.

- Navigate small size components to the right position when soldering.

Silkscreen: this is the top layer, usually white. It is used to represent values, component positions,symbols or any shape or character that the design want to facilitate the assembly, understand the function of circuit, the meaning, the order of connector pins,...0

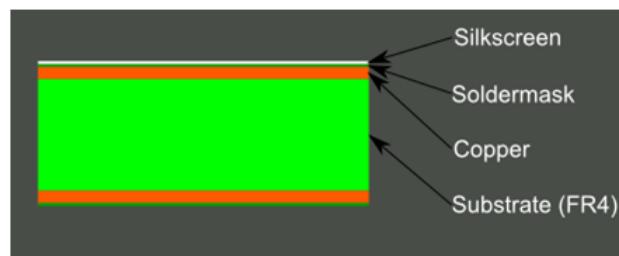
Single-Layer PCB: one side of PCB is covered by a condutive material(usually is copper). There is a layer of solder on top of the PCB to protect against oxidation. The followed layer is sikgreen, which mark all the components on the PCB.



Double-Layer PCB:As it's name, in this type of PCB, a thin



layer of conductive material such as copper is located on the top and bottom of the board. This type is more convenient, flexible, the cost is lower and reduced size of the board.



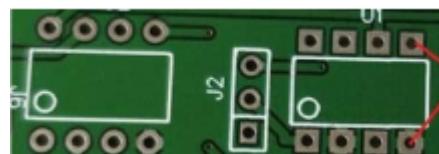
Multi-Layer PCB: this type of PCB has more than two layer, this means that it has at least three copper conductive layers. There is a glue sandwiched the insulating layers to ensure the generated heat will not damage any components.

Flexible PCB: This type of PCB uses flexible materials such as polyimide, PEEK (polyether ether ketone) or transparent conductive polyester film. Circuit boards are usually folded or twisted. The flex-rigid is a combination of flexible-layers attached rigid-layers.



If we separate PCB according to mounting system, we will have Through-Hole PCB and Surface-Mounted PCB.

Through-Hole PCB: in this type of PCB, components will be attached and soldered into the PCB by the holes, which will be drilling on the PCB.



Surface-Mounted PCB: the components in this type is very small and do not need feet to mount on board. Surface-Mounted Devive(SMD) components are mounted directly onto the surface of the board and do not requires holes in it.



Fifthly, some information about PCB material(substrate):the main element is rigid or flexible dielectric plate. This plate is used with a conductive material such as copper above.Dielectric plates are glass epoxy coatings or composite materials. Among the PCBs manufactured, the most common glass-covered material is FR4. Based on epoxy-glass compounds, FR4(Fire Retardent 4) is the most used synthetic material because it provides very good mechanical strength.





4 SOME LITTLE PROBLEM WITH CIRCUIT

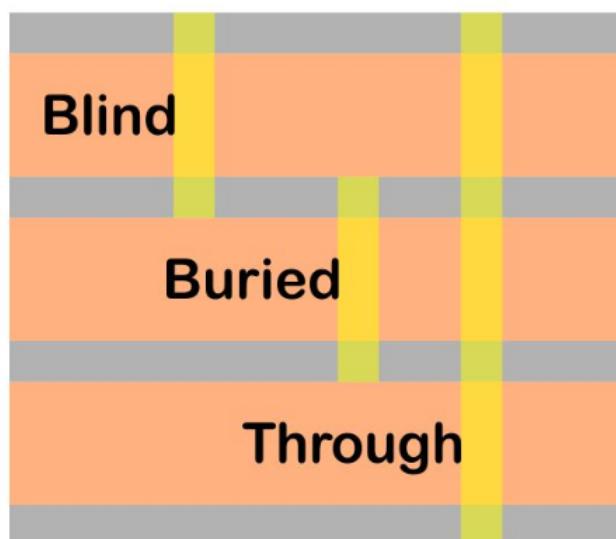
An element of PCB circuit VIA: via is an electrical connection between layers in a physical electronic circuit that goes through the plane of one or more adjacent layers.

A via consists of:
-Barrel — conductive tube filling the drilled hole
-Pad — connects each end of the barrel to the component, plane or trace
-Antipad — clearance hole between barrel and no-connect metal layer

There are three basic kinds of vias:

- +Blind vias: connect an exterior layer to an interior layer
- +Buried vias: connect two interior layers
- +Through vias: connect two exterior layers

Via Types





Thermal vias carry heat away from power devices and usually use for the GND connection. A via may be at the edge of the board so that it is cut in half when the board is separated; this is known as a castellated hole and is used for a variety of reasons, including allowing one PCB to be soldered to another in a stack.

Another application of via: connecting layers as a wire, radiate heat around.

As we know there are two kind of PCB: Through-Hole and

Surface-Mounted PCB, therefore we also have two kind of components, through-hole and smd.

*Notices when working with electronic component footprints:
+When we creating a footprint, we must open the soldermask layer to the pad about 0.15mm.

+If this pad is connected to GND which can lose heat quickly, when creating a pad, you should not connect the pad to the copper board in full contact mode (thermal relief should be used).

+Pay attention to the distance between the pads with each other to avoid welding errors.

+We must care about the size, diameter of the pad, the shape (circle, oval, square, ...).

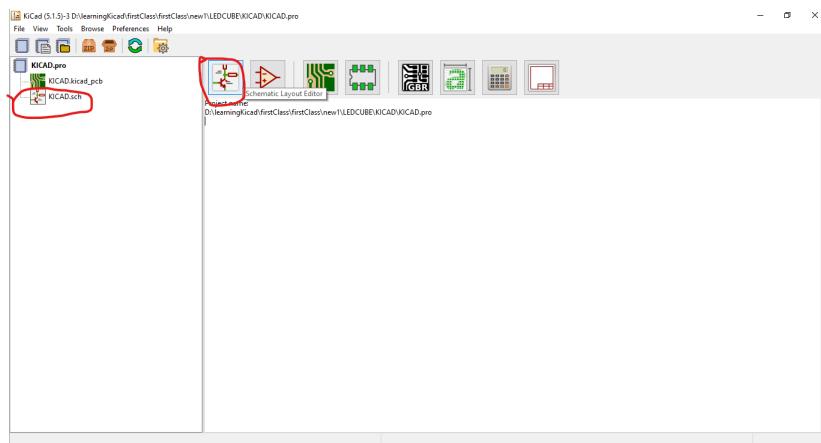
+Size of the hole: the diameter of the hole.

+Is this hole plated with tin?



5 SCHEMATIC AND SCHEMATIC LIBRARY.

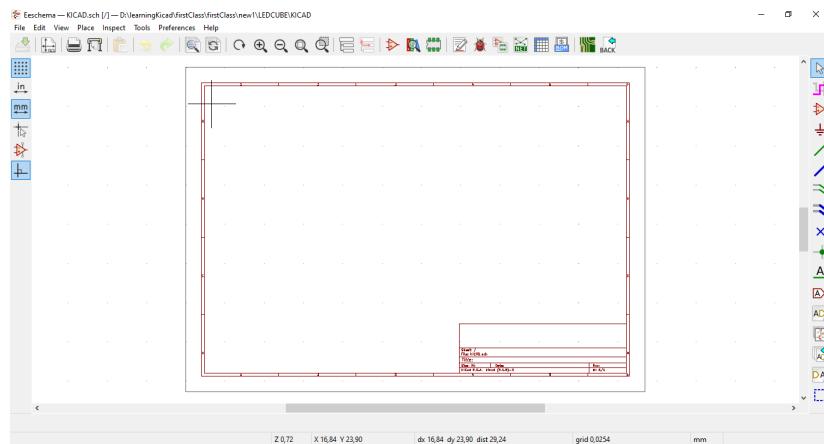
Starting draw a PCB board you must draw the schematic circuit before. To begin working with the schematic click to the fist symbol in the leftside or double-click to the file with the end is .sch in the litte window at the left.



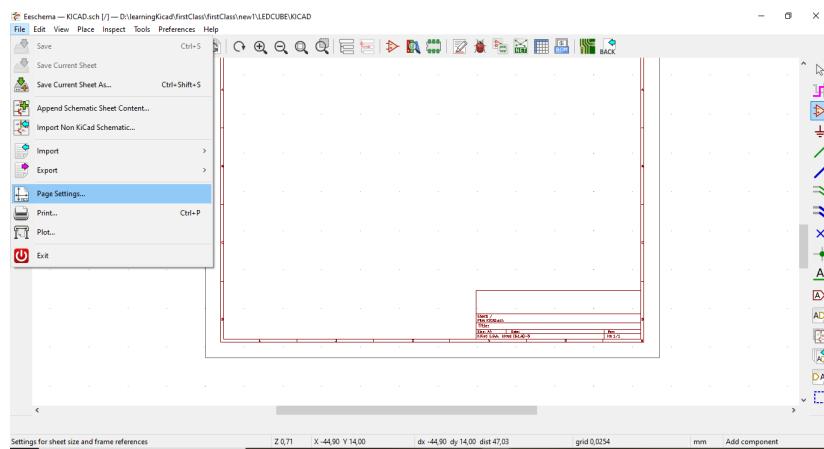
After that step you will open a new schematic window like this.



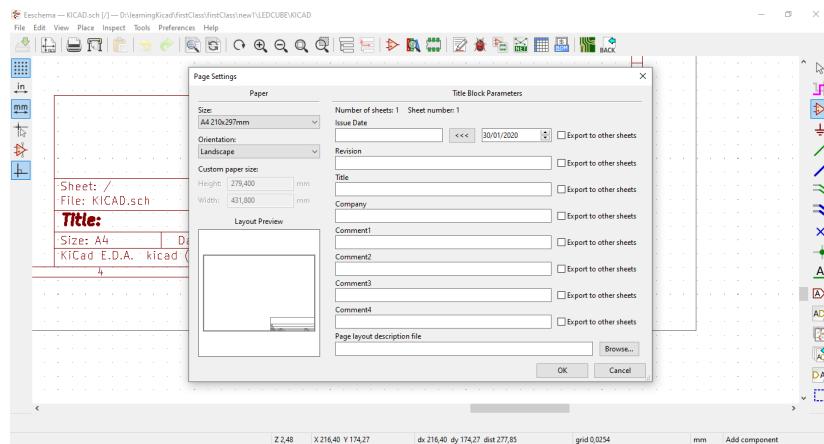
5 SCHEMATIC AND SCHEMATIC LIBRARY.



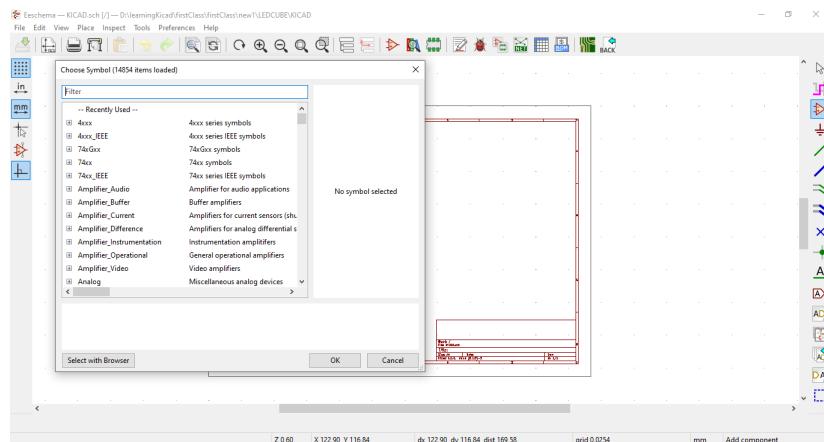
Now before working with the schematic you can edit the information at the right corner by choose File->Page Settings...



After that, you can choose a lot of properties of the page like size, orientation, title, date and time, add comments, etc.



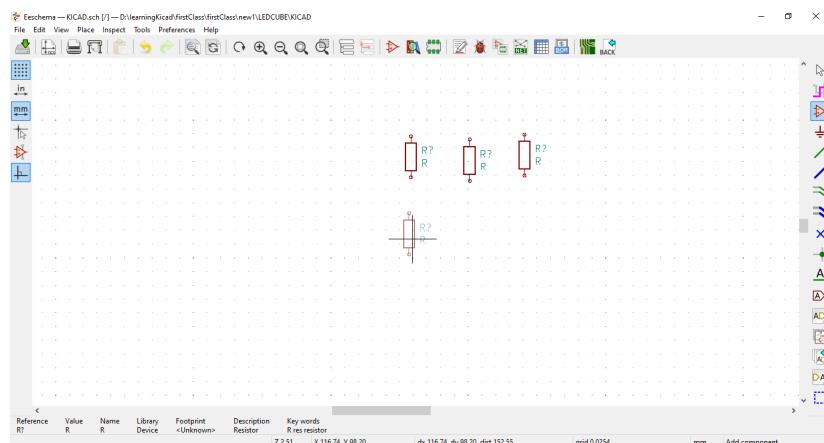
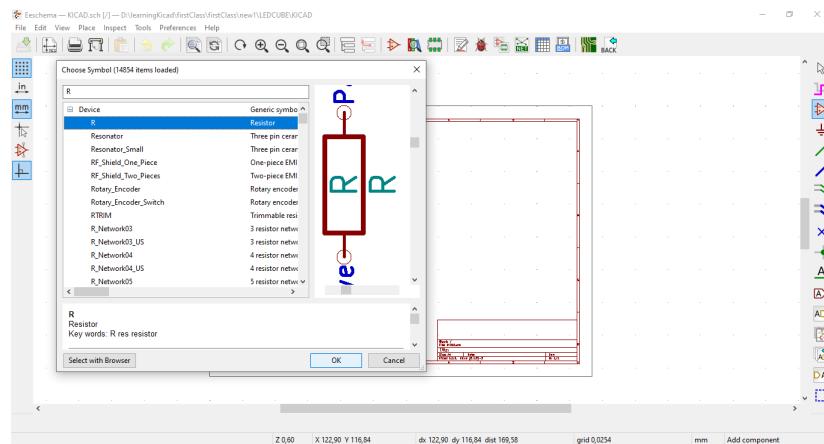
The first step to working with schematic is choose and place component symbols. To choose component symbols , click the symbol button on the right toolbar(the third button from the top).Then click on the schematic page, Choose Symbol window will appear.



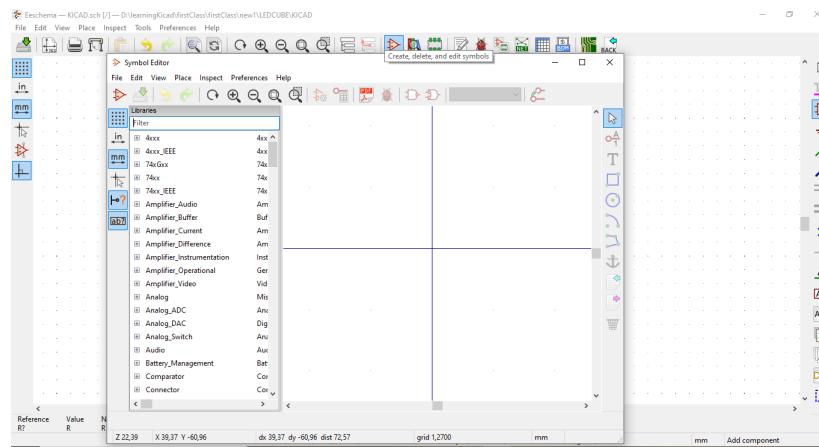
Writing the name of component that you need at the filter, choose it and click OK then put it in the position you want in the schematic.



5 SCHEMATIC AND SCHEMATIC LIBRARY.

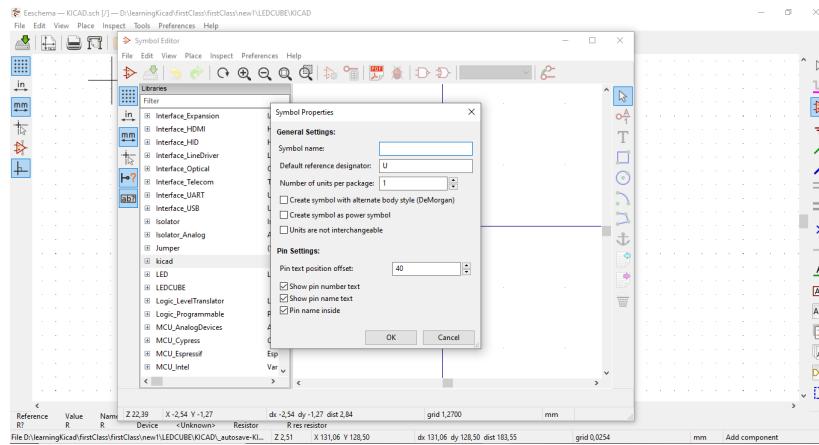


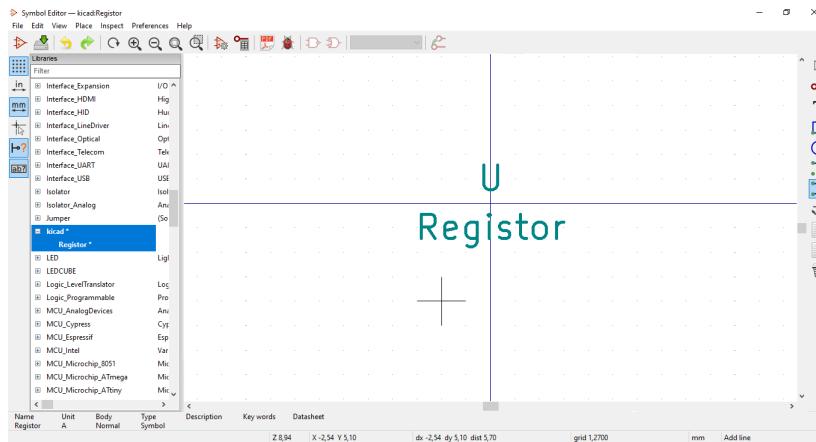
But in the symbol library, there aren't have all the components symbol (ex: UNO R3 symbols,...). In this situation, you must create new library and news ymbol by the Symbol Editor, click the Symbol Editor button in the top toolbar a new window will open.



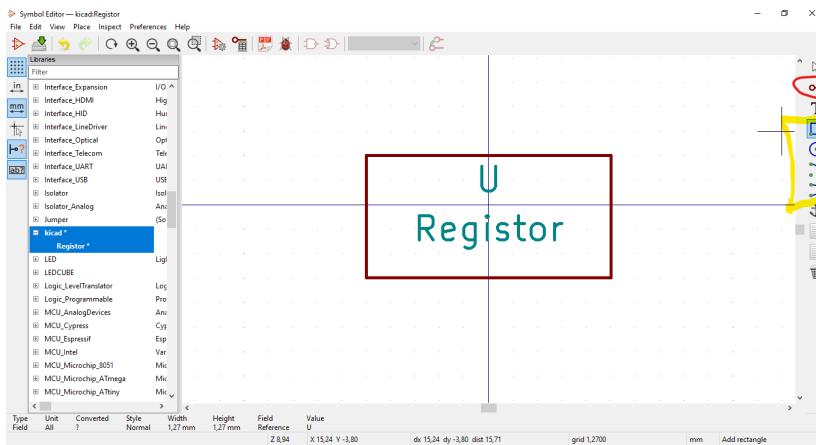
Click File->New Library to make a new library or add library if you want to add a existed one in your company to this KICAD project. If you choose to create a new one, a folder .lib will be created, and once you create a new symbol it's going to be added to this folder (note: when you create a new library, KICAD will let you choose this library can be use in all the project-Global or just in this project-Project).

Now I just create a new library name "kicad" and create a new symbol in this "kicad". A window will appear to let you choose properties of this symbol like name, number, etc.





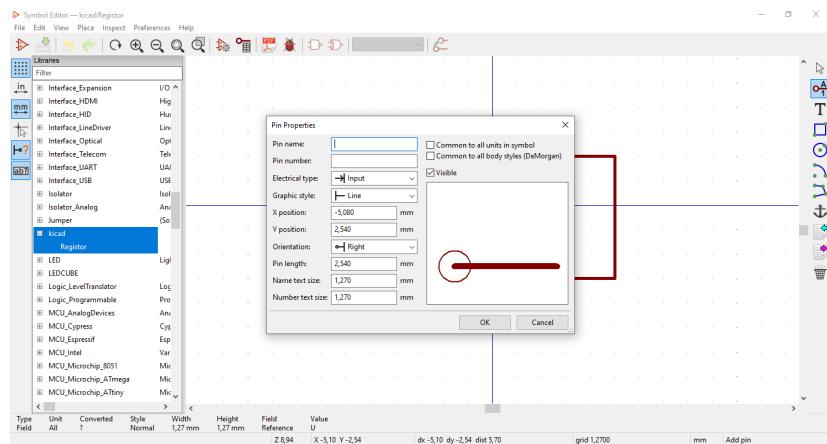
As you can see, a new symbol was made with U is the reference and Register is the value (two properties can choose when you at the symbol into the schematic). And now you just use the drawing tool that KICAD support at the right corner to make a new symbol that you want(the yellow zone). When you have finished drawing you must add pins to the symbol equal the pins that the components in the reality have.(the red zone).



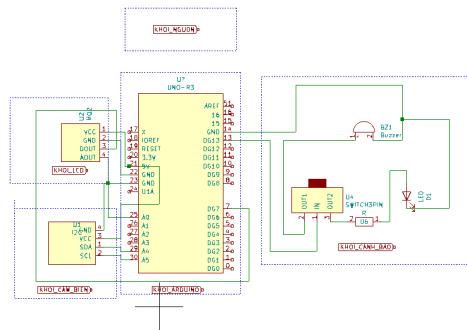
Click the Add Pins button, click to the symbol a Pin Properties will appear to let you choose the properties of this pin. Doing this step some times to finish making component symbol.



5 SCHEMATIC AND SCHEMATIC LIBRARY.

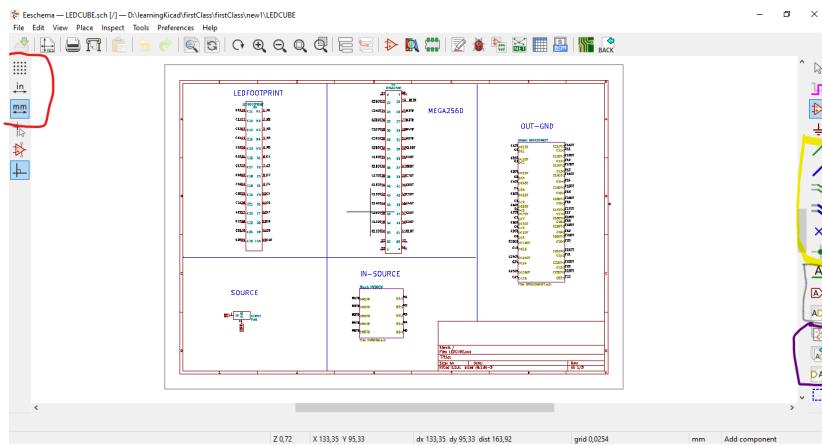


When you have enough component symbols, firstly place it in positions that convient to view, divided to some regions with each regions have the same function,...Finish this step ,you need to place wire to connect component symbols together, then you will have the complete schematic.This is an example





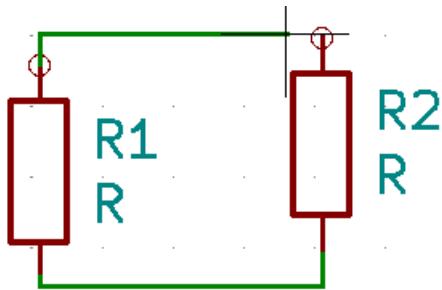
Now we will research about the schematic window. At the left corner you will see some button, the first button to choose hide or show grid to easy observe and draw schematic. Below this button, we will see two button let us can choose to use ich or mm as the main unit(Three buttons in the red zone).



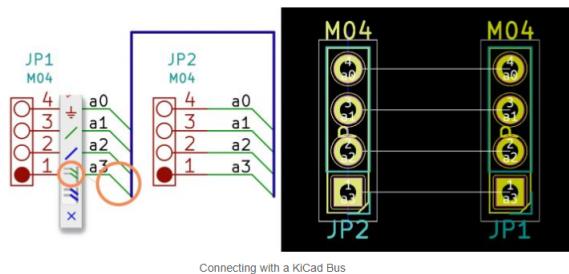
There are three toolbars on the page. Look at the one at the right corner, there is a GND symbol, that is the Place Power Port Button which let us choose Power and GND connection symbol to the circuit.

The yellow zone in the right corner is very important, it is the region that have wire and bus button. Firsly you must know what is wire and bus??

We can normally know wire as a conductor, KICAD connect components together by it and the green line symbol in the yellow zone is Wire Button, click this button. Then in schematic click to one pin, move the green line as the road you with and click one more at the pin need to be connect to the pin before. Now we have a wire connect two pins.



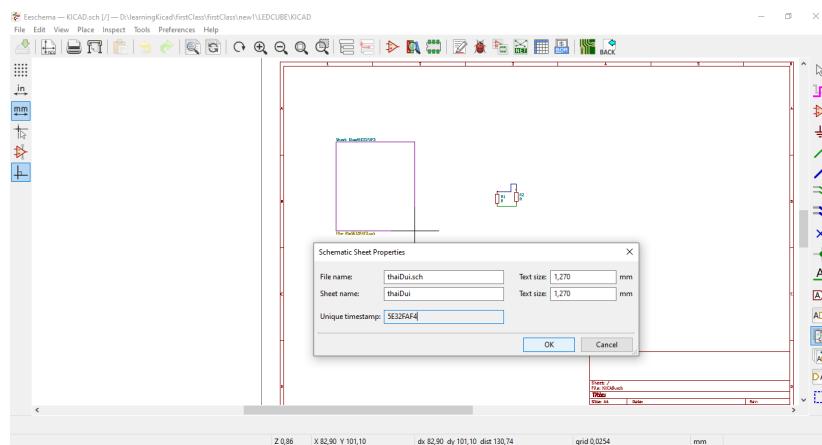
To KiCad, a bus is a collection of wires. For the most part, they are an aesthetic feature. We can connect the connectors from above together with a “bus.” However, this connection does not change anything. And you must notice that drawing a bus alone is not enough. First, you must add a “wire to bus” entry, to tell KiCad that you want the wire to be part of the bus. (So far, I have not found a way to edit the direction or length of these “entry” wires.) The blue line button is Bus button and the two green line connect together is Wire To Bus button. Bus to Bus(two blue line connect together) Button have the same function with Wire to Bus but it use to connect bus to bus. Beside, you will see a button with have blue symbol X, it is very usefull, if component pins don’t use you must place this symbol to this pin to notice that is no connection.



With wire and bus in the yellow zone we can finished schematic

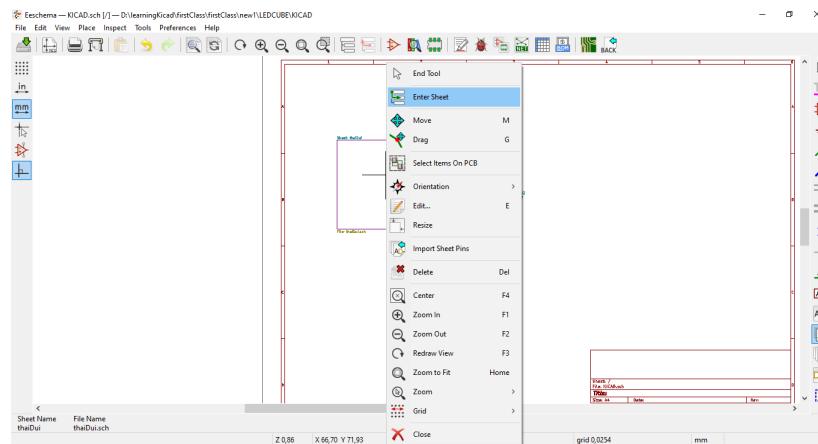


but in a large project with a huge and complex circuit we can draw it in one page. In that situation, we must use an ability of designed PCB program: multiple sheet. Each program have a method to accomplish it, in KICAD we have Hierarchical Sheet(the first button of the purple zone). Choose the Hierarchical Sheet at the right corner, drag and drop mouse at the schematic page you will achieve a zone, right click at this zone you will see Enter Sheet, choose it. A new sheet will appear this is the subsheet, and with this subsheet and mainsheet we have multiple-sheet, a ability to work with complex circuit with each sheet display a function zone or a layer of circuit.

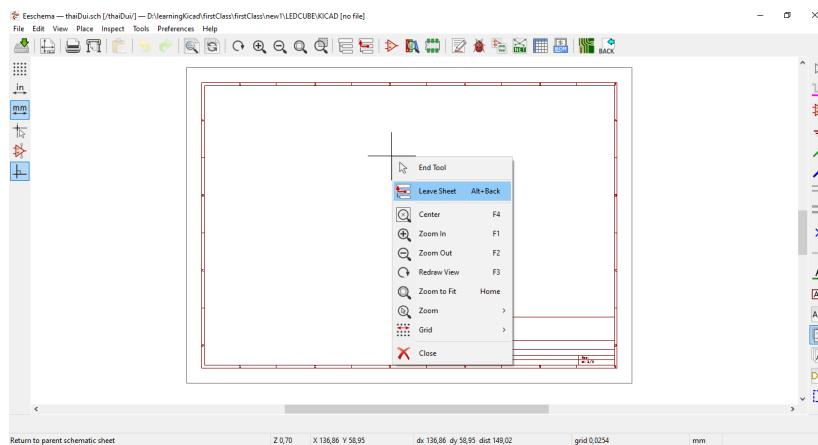




5 SCHEMATIC AND SCHEMATIC LIBRARY.



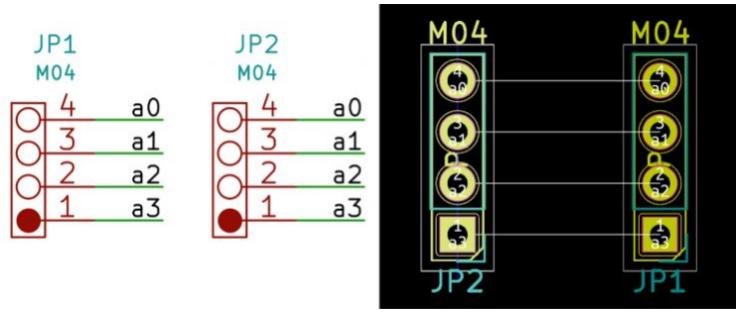
When we enter the subsheet, we can use it to place component symbols and connect its and leave to the main sheet.



But how can we connect symbols in the subsheets to the main sheets or connect among the subsheets?? The answer is the black zone: Label.



Label can be known as a wire connect component pins together, but it don't connect through line, label connect through name, pin have been added the same name was be connected together.



There are three kind of label:

+Net Label: just connect in a sheet.(the black text label).

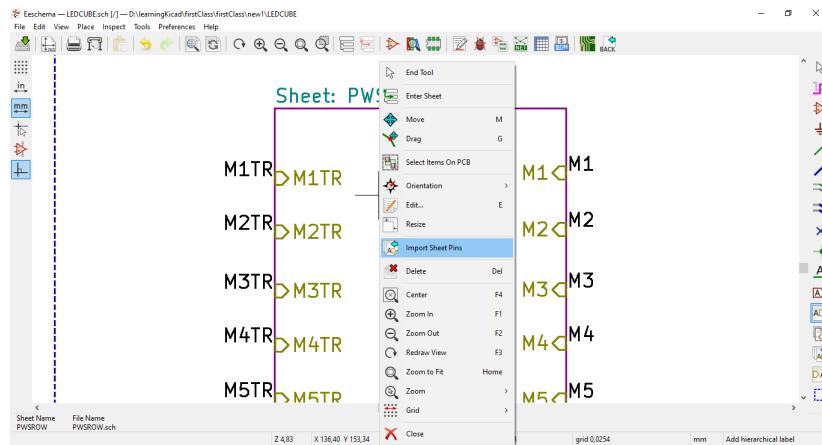
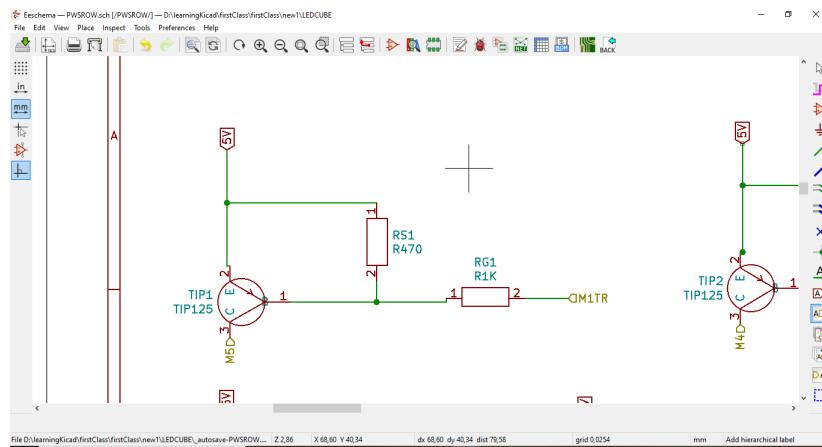
A

+Global Label: connect through all sheet.(the red text label).

+Hierachical Label: connect between two sheet mothersheet and subsheet (both main sheet and the subsheet of it).(the yellow text label).



Therefore, to connect from subsheet we just place a hierarchical label as a wire, leave subsheet and import it, then place net label to connect with other pins.



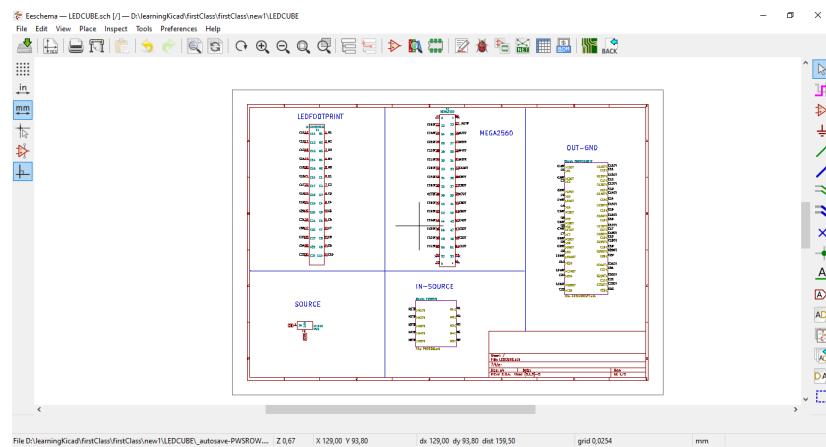
After doing all this step, you just run electrical rules check to check about mistakes is accomplishment.(Inspecet->Peform electrical rules check).

Note: before place wire and label you must annotate each symbols, you can use right click-> Properties or auto annotate by Tools-Annotate schematic symbols->Annotate. When draw schematic, should separate it by some region and note in each



5 SCHEMATIC AND SCHEMATIC LIBRARY.

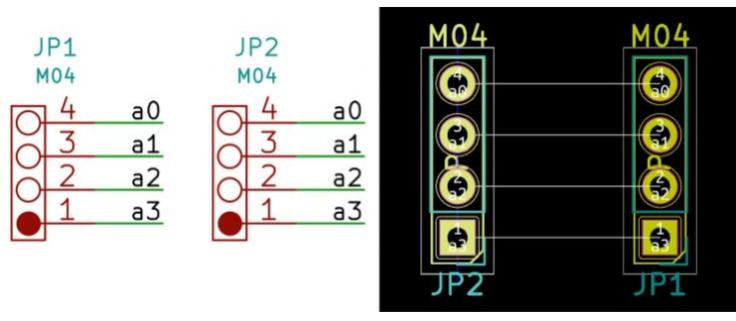
Zone.





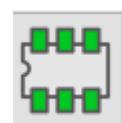
6 FOOTPRINT LIBRARY AND HOW TO ADD FOOTPRINT TO SCHEMATIC.

After finish schematic circuit, we will continue draw PCB circuit from the schematic, but symbols in schematic is just symbolize and can't use for realize components. To convert schematic to PCB, the first step is convert schematic symbols to PCB footprints. In the follow picture, in the left is schematic symbols and opposite is PCB footprints.



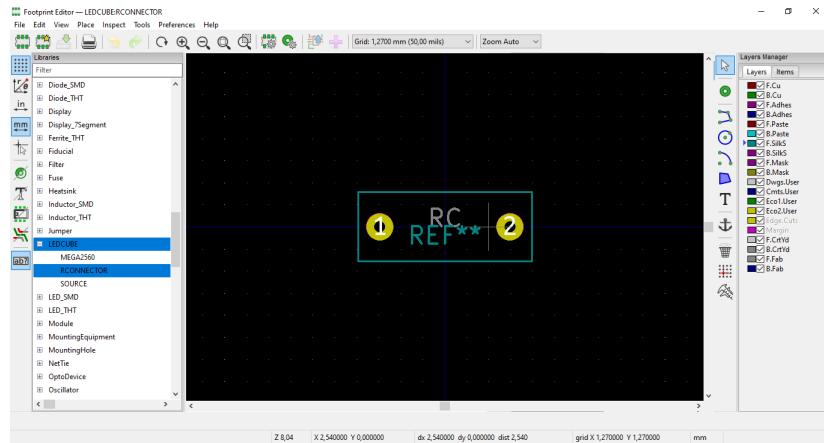
The most important thing when working with PCB footprint is you must design the footprints follow the datasheet of components. Only using information of datasheet, you can create component footprints exactly.

To work with Footprint Editor, choose the Footprint Symbol at schematic window or main project window.



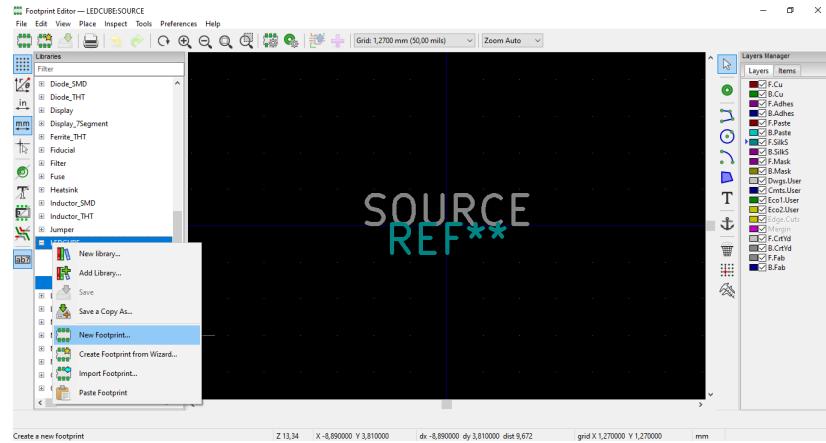


When you click to this symbol, a new window like this will open, it is the main window in whcih you can choose, create, edit, or add footprints and footprint libraries to you project.



As schematic library, there are a lot of available libraries you have downloaded togethrer with KICAD software. But with each component datasheet is very different and each kind, each company's component have different datasheet(also footprint properties). Therefore, when working with Footprint Editor, most of time is create new footprints based on datasheet.

Working with Footprint Editor have the same step when work king withSymbol Editor. Firstly, you can create a new library by click File-> New Library, then choose a name for new library and confirm it is a global or project quality library. When you have created library(this is a folder with .pretty at foot), to create a footprint just right click at the name of this library and choose New Footprint.In follow example, I have just created a new library call LEDCUBE, and create a new footprint in this lib.



Now after create footprint, we wil accomplish footprint by some step, by reading datasheet we wil know infomations. The first infor we need is, this component we use is through-hole or SMD, each component have another pad.(Pad is the place that we pin of components connect to the board).When we know the first feature, we now must determine another important properties as: pin size of component, shape of pin, distance between center of two any pin, size, shape , distance from two edge(or diameter of circle shape) of components.

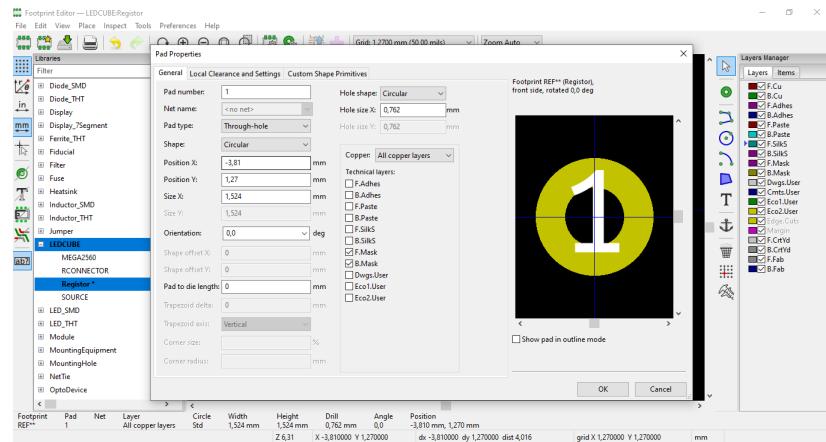
Finishing all this information, we start place pad in the sheet by choose the button Add Pad at the right toolbar.



After choose this button, when you click at the sheet a pad will be add to the sheet, but you must choose the position of pad to be suitable of the information above.We can change position by right-click at pad cand choose Properties, a new window will appear.

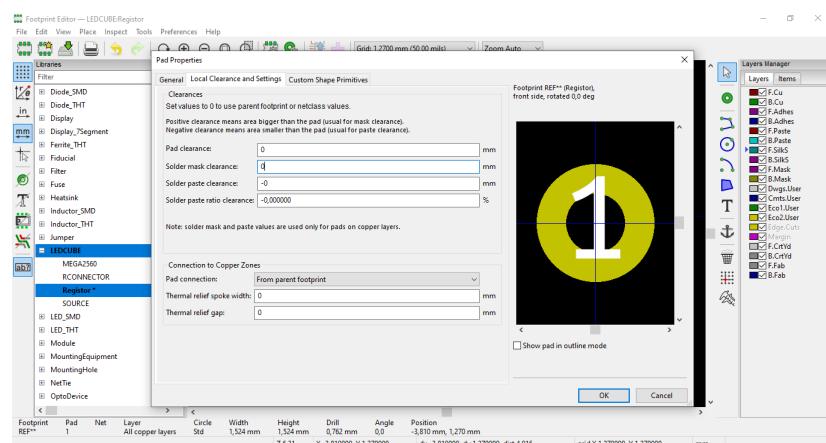


6 FOOTPRINT LIBRARY AND HOW TO ADD FOOTPRINT TO SCHEMATIC.



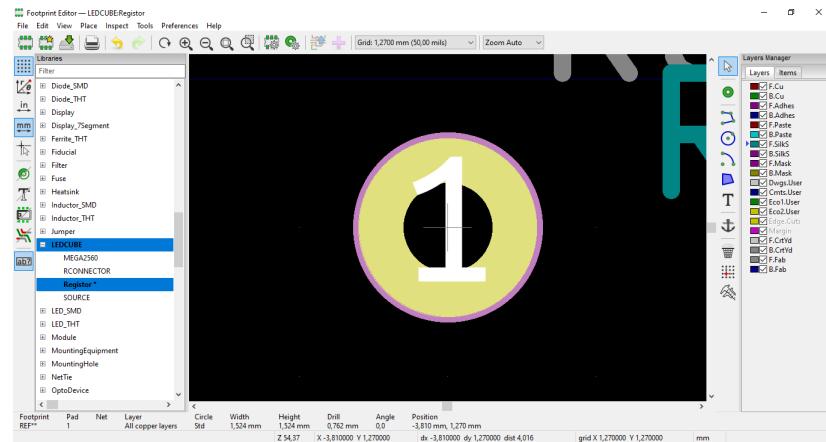
As the picture, we can use this window to determine a lot of properties of Pad such as: number, type, shape position X and Y, important is hole shape and size with through-hole pad. At the bottom-right corner we also can choose layers and technical coppers that pad will display.

Turn to the second page of this window, we can open solder mask(which is obligatory) to the board by use Soldermask Clearance.

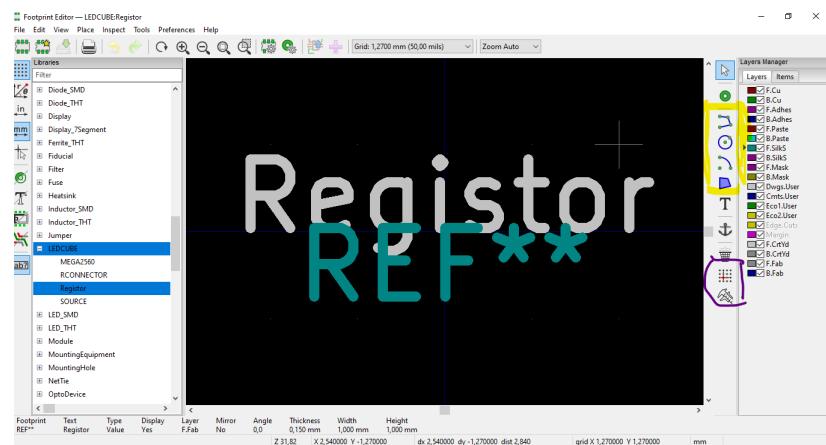




6 FOOTPRINT LIBRARY AND HOW TO ADD FOOTPRINT TO SCHEMATIC.

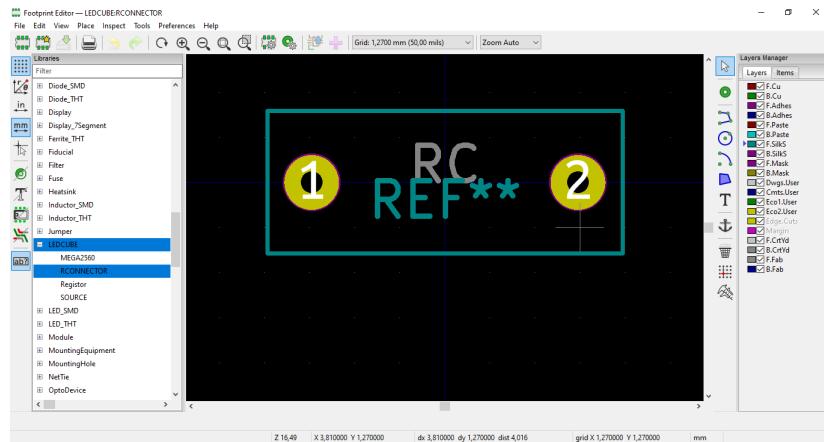


Place pad and using the drawing tool buttons of KICAD(in the yellow zone) and the support tool buttons(in the purple zone) will help you finish creating a footprint. The first button in the purple zone is Hide or Show grid, the second button will display distance as a rule with the unit you can set up by some button at the left toolbar.

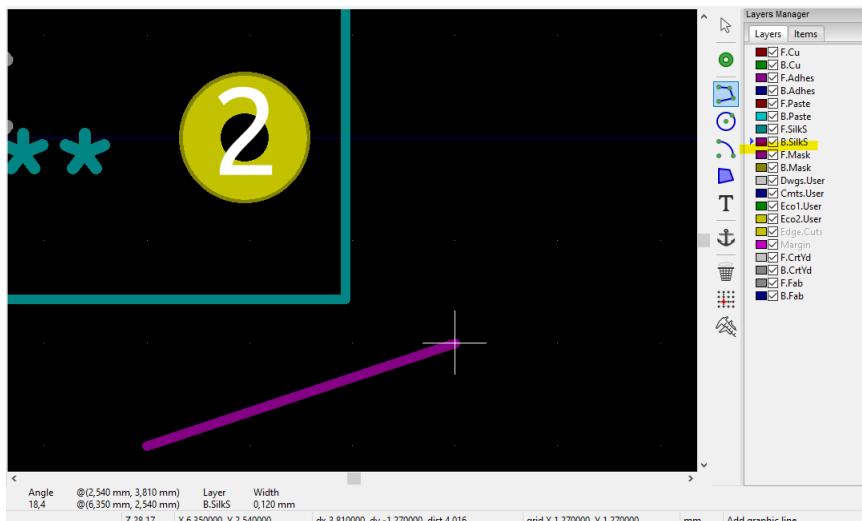




6 FOOTPRINT LIBRARY AND HOW TO ADD FOOTPRINT TO SCHEMATIC.



If you carefully observe at the right corner is Layer Manager. Is very useful to work with multiple-layer circuit, each layer have a function different and we will research later in the PCB circuit. Now we just need to know if we tick in the box behind the layer name, this layer will appear in the sheet, and if want to hide it just tick again. And if click in the space in front of the color box of layer you can turn the drawing line, text to write just in this layer.





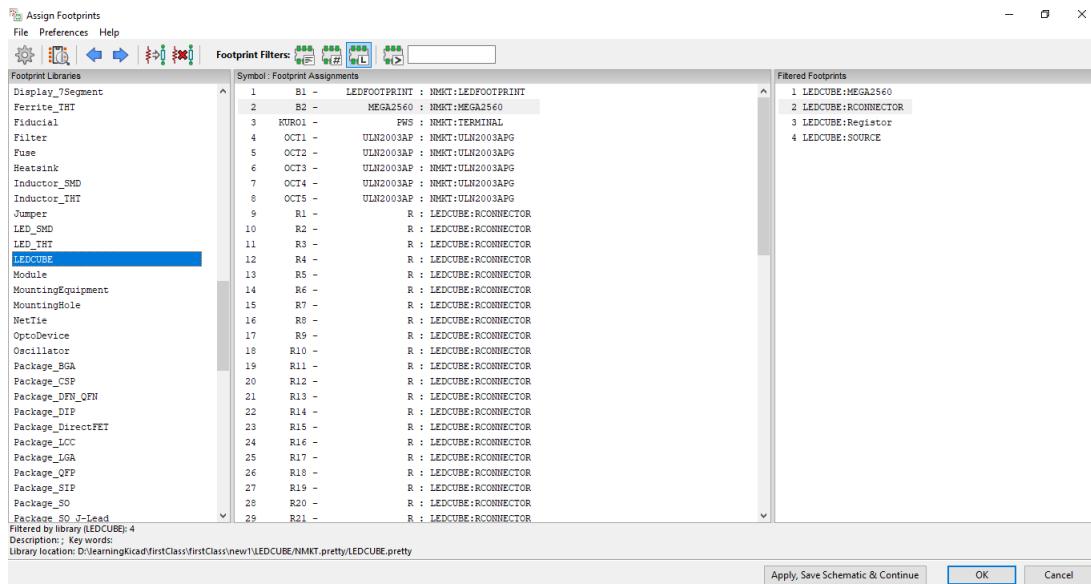
After create and choose footprint for all component symbols, to display it in the PCB Layout Editor you must add each footprint to the corresponding schematic symbol. To do this important work, click Tools->Assign Footprints.... or choose the Assign Footprint symbol(the follow picture) in the top toolbar.



A new window will appear, there are three region in this window, at the center are the component name, the left region are the footprint libaries name, and the right zone are the footprints name. Choose the symbol name at center, click the library have the footprint you want at the left and choose it at the right. After do this step for all component symbols click button " Apply,Save Schematic Continue" then click OK button. Now you can open PCB Layout Editor and start drawing the PCB circuit.



6 FOOTPRINT LIBRARY AND HOW TO ADD FOOTPRINT TO SCHEMATIC.



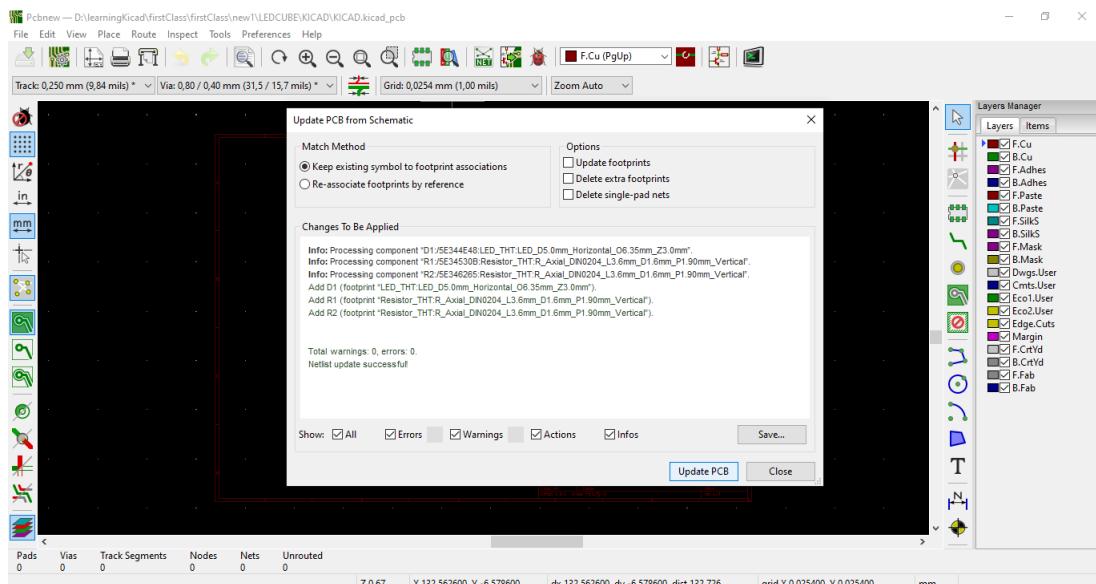


7 PCB AND THE COMPLETELY CIRCUIT.

We'll start working with PCB circuit by update footprints and nets from schematic circuit. To update, click the Update PCB from schematic button at the top toolbar.

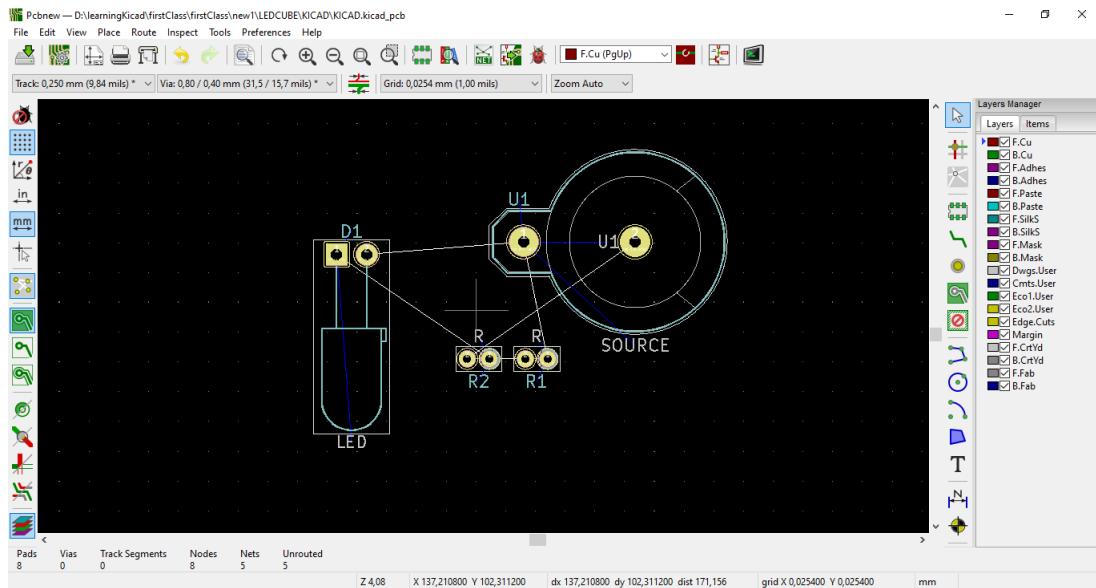


When a Update window is open, check information at the center, and click Update button, component footprints and nets are going to appear, first step to draw PCB circuit is choose the suitable position to place that block.

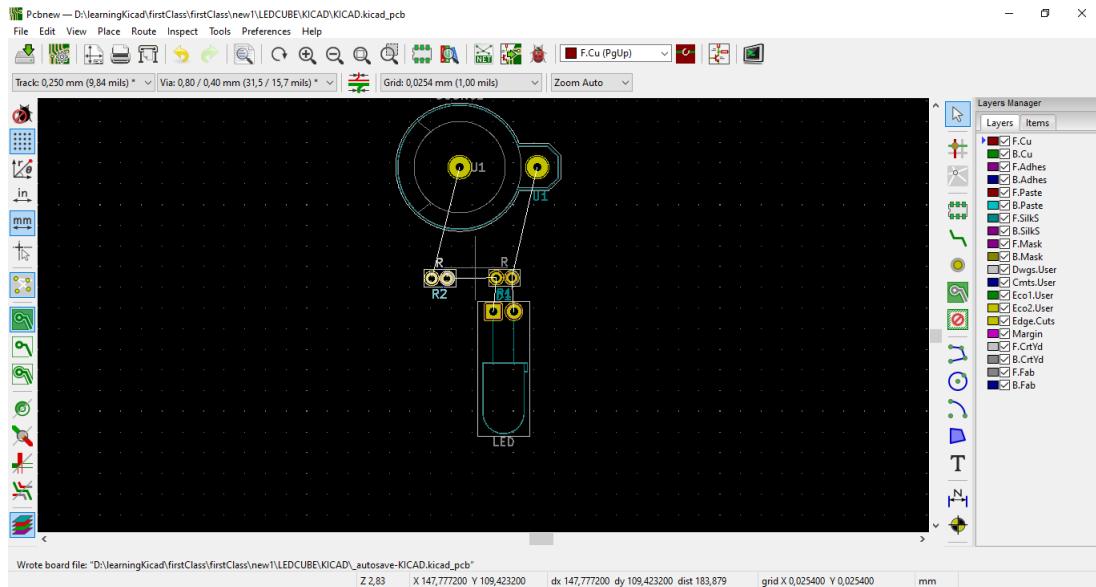




7 PCB AND THE COMPLETELY CIRCUIT.



Move each components footprints to the right position in the schematic circuit, now carefully observe you will see each pad will be connect with the other by a while thin line, this line is called net, net connect pads together(if in the schematic this pins also connect by wires or labels).





You also can hide nets by choose Hide mode by click the Hide board ratsnest button. When you place components at the right position, before drawing anything, check track width(usually is 30 mils), via size, grid size at the top toolbar. these properties is very important, it can effect directly about making stage, the operation of the accomplished circuit.

Before place any line, come with me to research about some important and common layer that we usually use at the right window that we hace mentioned in the previous part.

F.Cu: the top conductive layer(the first copper layer) that we can draw track to connect components in this layer and component always be soldered in this layer or in bottom ayer. B.Cu: the bottom conductive layer, it have all functions as the F.Cu. F.Silk and B.Silk: what is have display in two layer is decide what is wil print in the silkgreen top and bottom. F.Mask and B.Mask: if you turn off all layers and turn on one of it, you'll see only pads, yes only pads, why because two layer show the soldermask open, and soldermask open is usually use for pads only.

Egde.Cuts: this layer is very important, it is the board zone, by drawing a region by drawing tools when choose this layer, a board zone will appear and all the components, footprints, track,... out of this zone will disappear in the board. Choosing size of the board and draw limited area is the follow step.

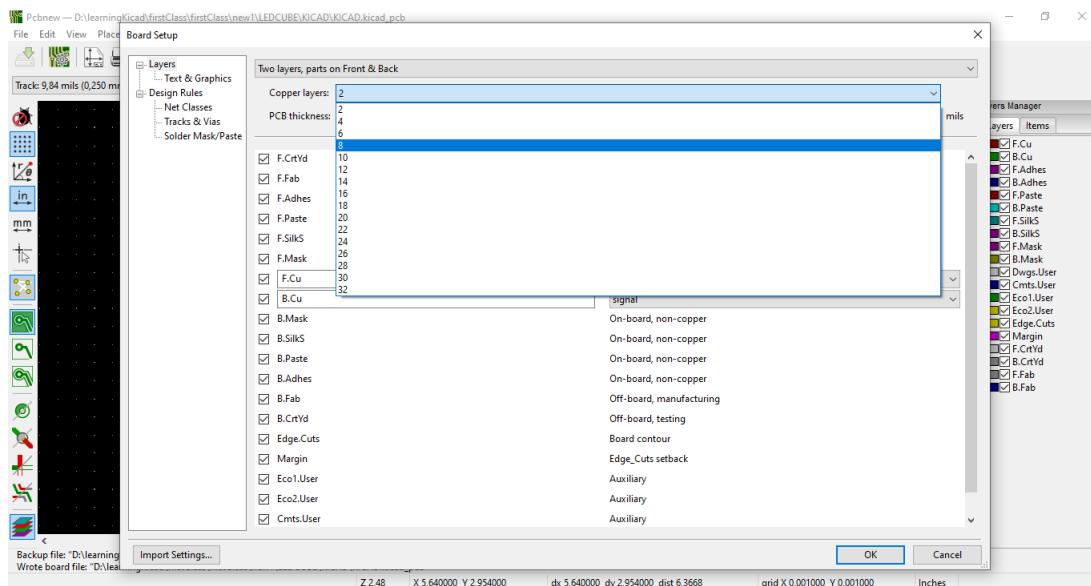
After those steps, you now can draw track by some tools at the right toolbar.(the yellow zone).



KICAD is support to work with PCB circuit from 2 to 32 layer. This mean we can open more layers than available. How we can create more layers? Choose Board Setup button(the follow picture).



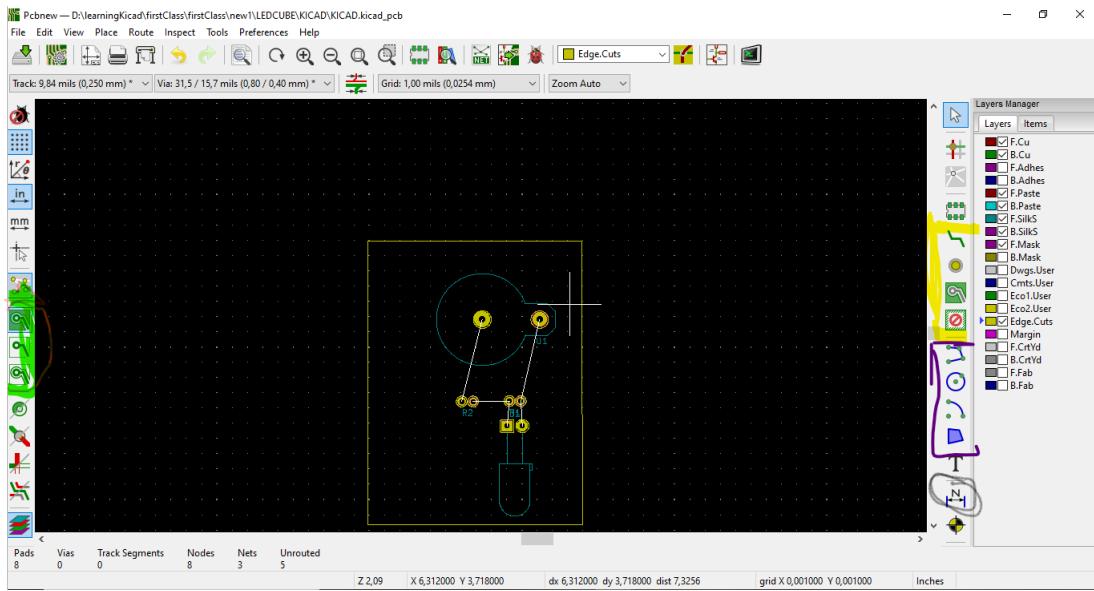
When Board Setup Window appear, choose Layer mode at the left corner, and now in this mode you can easily choose how much layers, change name, properties of each layer.



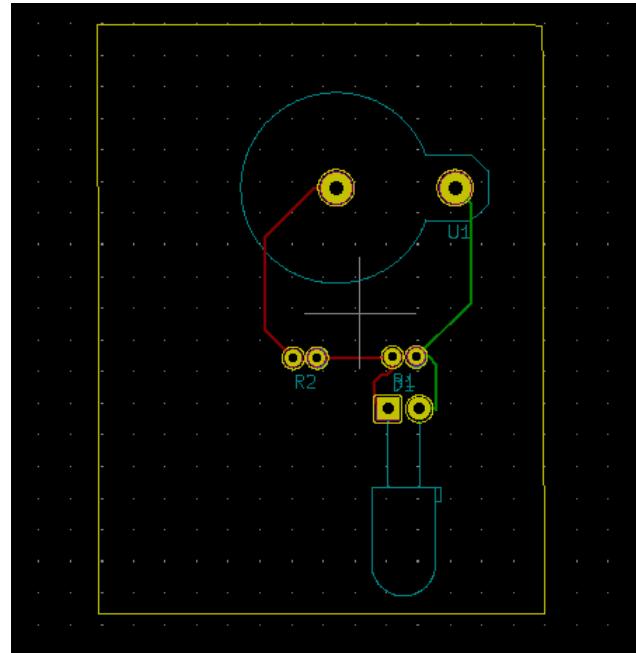
After those steps, you now can draw track by some tools at the right toolbar.(the yellow zone).



7 PCB AND THE COMPLETELY CIRCUIT.



The first button with green line will give us ability to draw track connect pads together, choose this button and click at the space in front of Layer that you want after that just connect two pads, which nets was connect before.





If you carefully observe in previous picture, track was draw by two color: red and green. Red tracks are tracks that draw in the F.Cu and green tracks are track that draw in the B.Cu, each layer have one color to distinguish with another layer. We also can choose and change layer color at the Layer Manager Bar. The second button in the yellow zone is Add Vias button, it can very helpful if you work with a complex circuit. If you remember, you see buttons at the purple zone two times ago, and of course in this editor, these buttons have the same functions., the black circle cover a button which help us to measure distance . When design anything, if you carefully use Measure Tool, it can bring a lot of advantage.

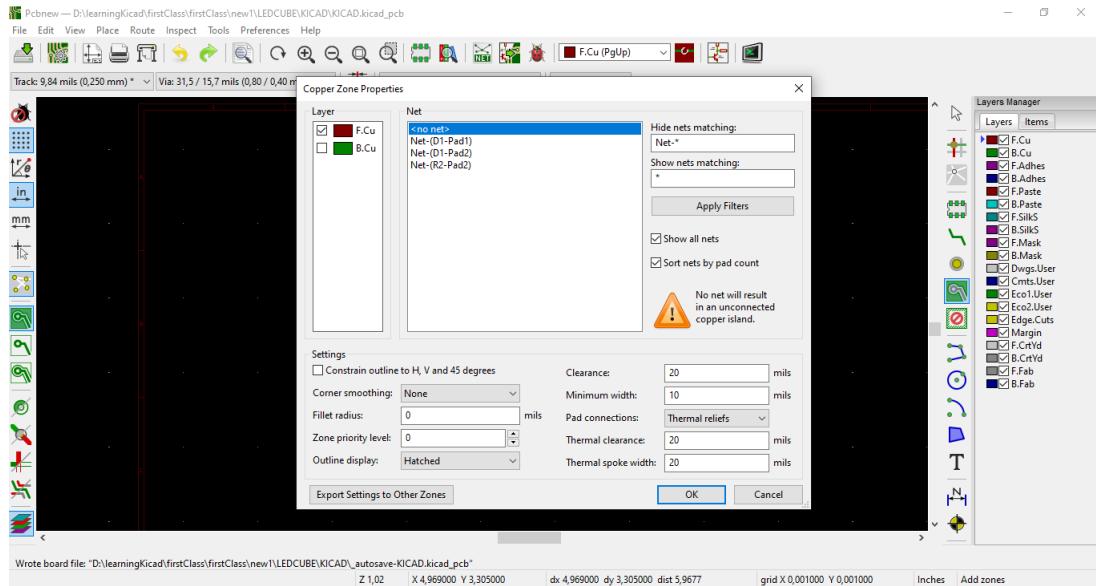
One tips to work with KICAD is, when you move mouse and put in button you want, a text block which explain application of this button. Move, read carefully and test a lot of will help you to work better with KICAD.

When you finish connect footprints, you should cover your board by copper filled zone. A copper pour or fill refers to an area on a printed circuit board where the original copper is not etched away, and remains in place, usually electrically connected to the Ground signal, producing a “Ground Plane”. This has a number of advantages, including decreasing the amount of etching fluid required during manufacturing, as well as reducing the amount of electrical noise and signal crosstalk experienced by the circuit elements.

To cover layer, firstly we must choose properties of filled zones have, at this board it choose showed filled areas in zones, one button at the left button. Then choose the button Add Filled Zones at the right corner. Then in the limited areas, draw a graphic polygon as the shape of board and limited area. Of



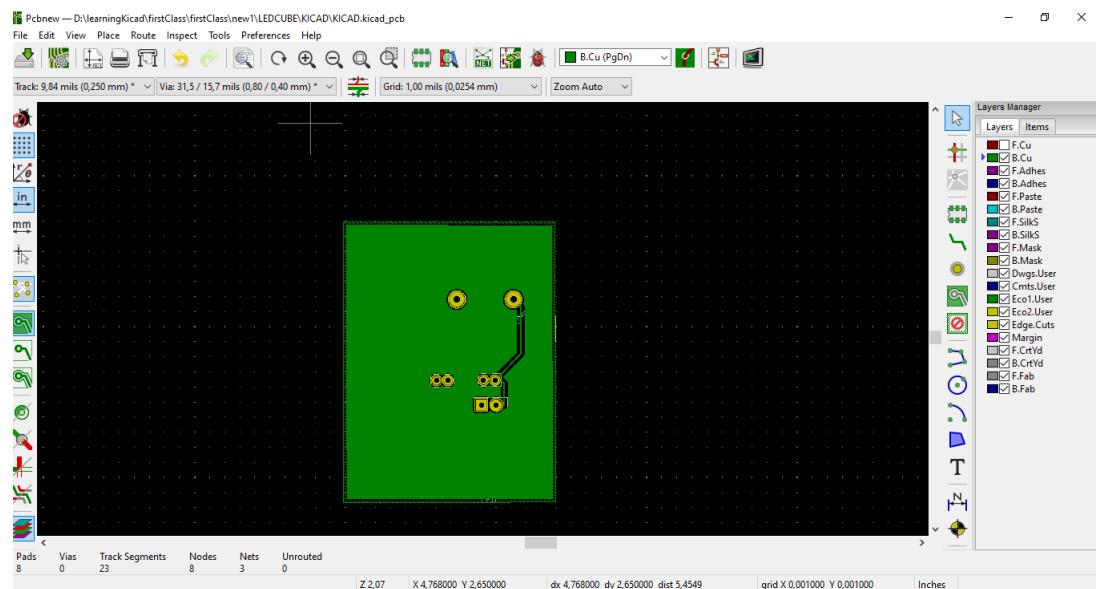
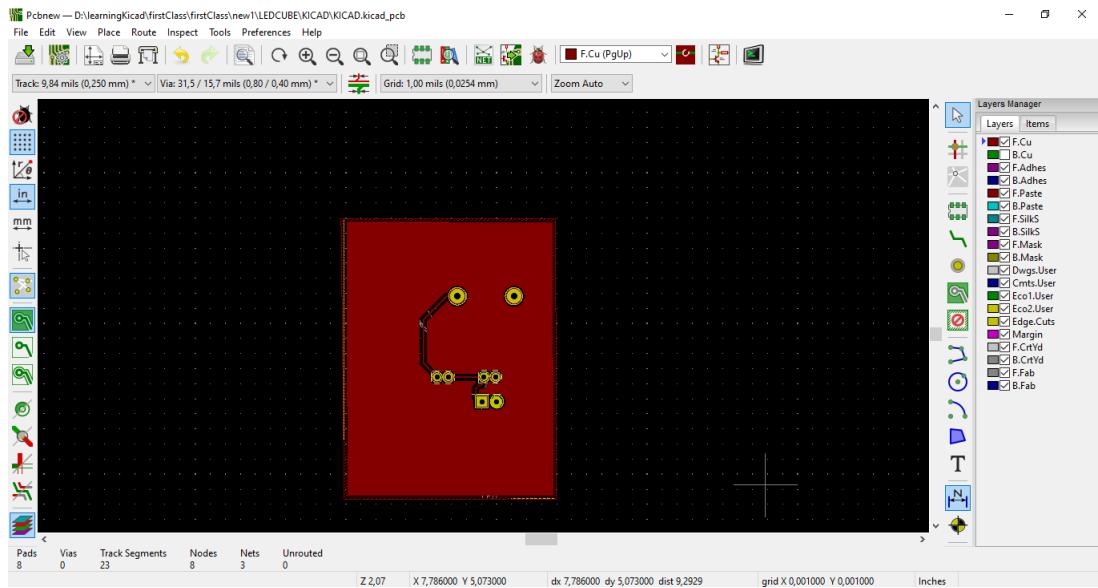
course you must choose what layer was be added filled zone(F.Cu or B.Cu,..).



Two picture following is F.Cu and B.Cu filled zone, when you have done this step your circuit is also accomplish. But you can continue doing some little step if you need such as add 3D shape to each component footprint , by doing this step you circuit can be view by 3D viewer as in realize.



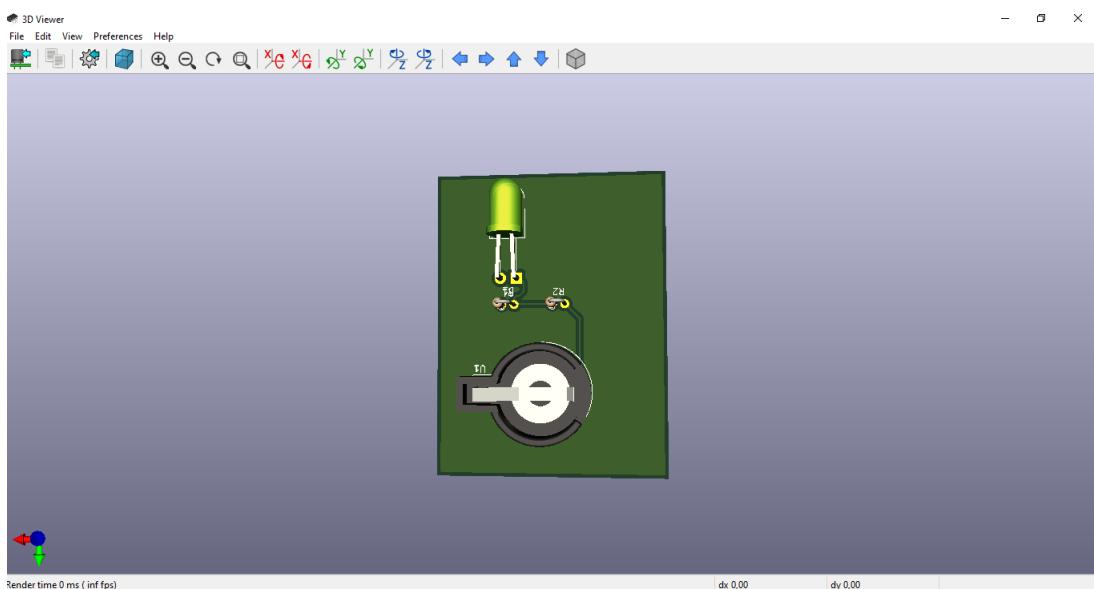
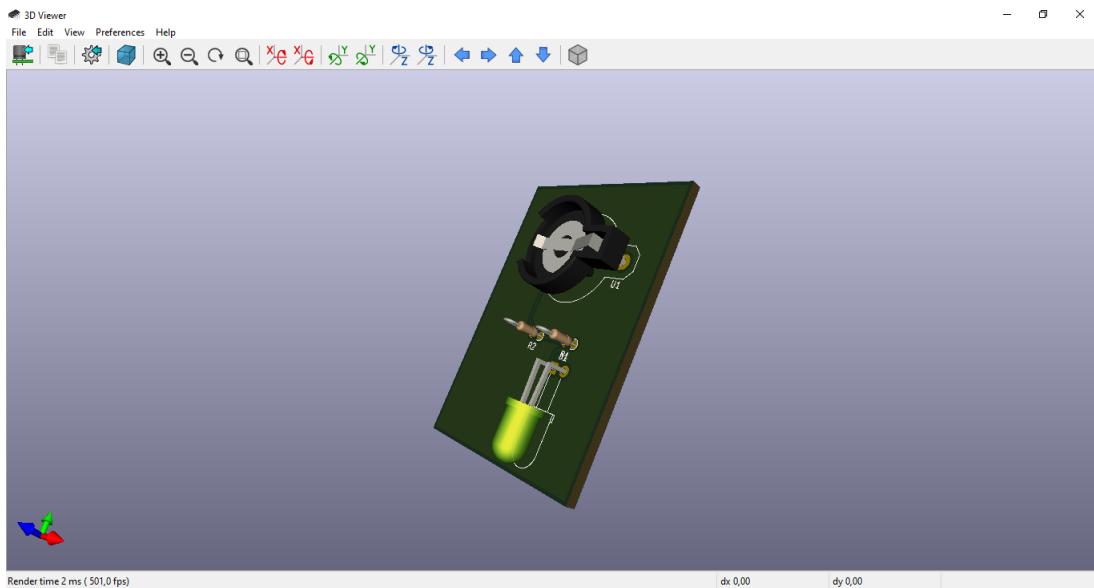
7 PCB AND THE COMPLETELY CIRCUIT.



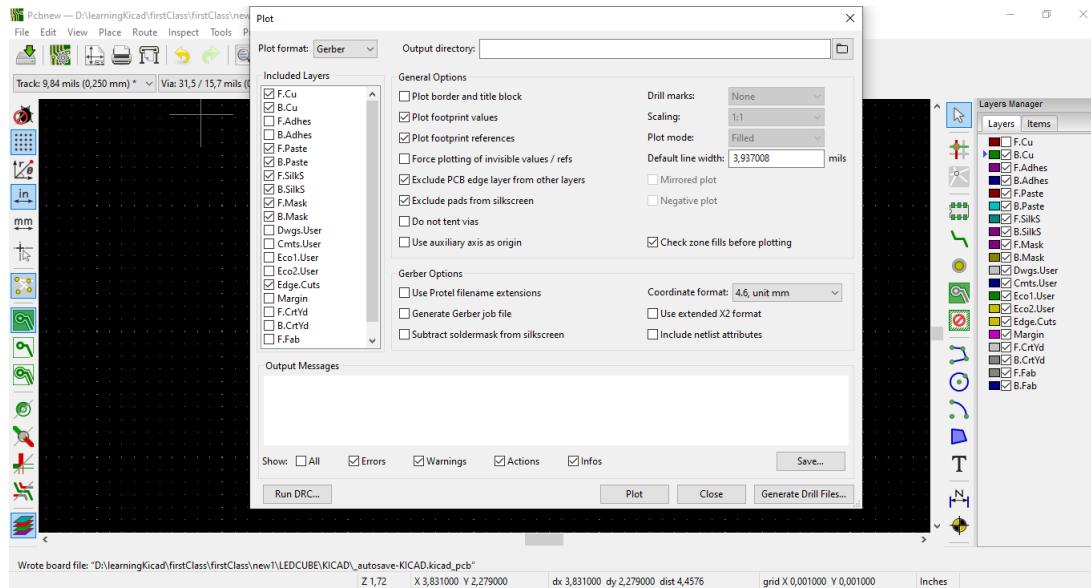
This following picture is 3D viewer mode, I have added 3D shape to footprint previous and now when I open 3D view I can see 3D component, use it carefully and you can imagine what exactly your board will be in reality.



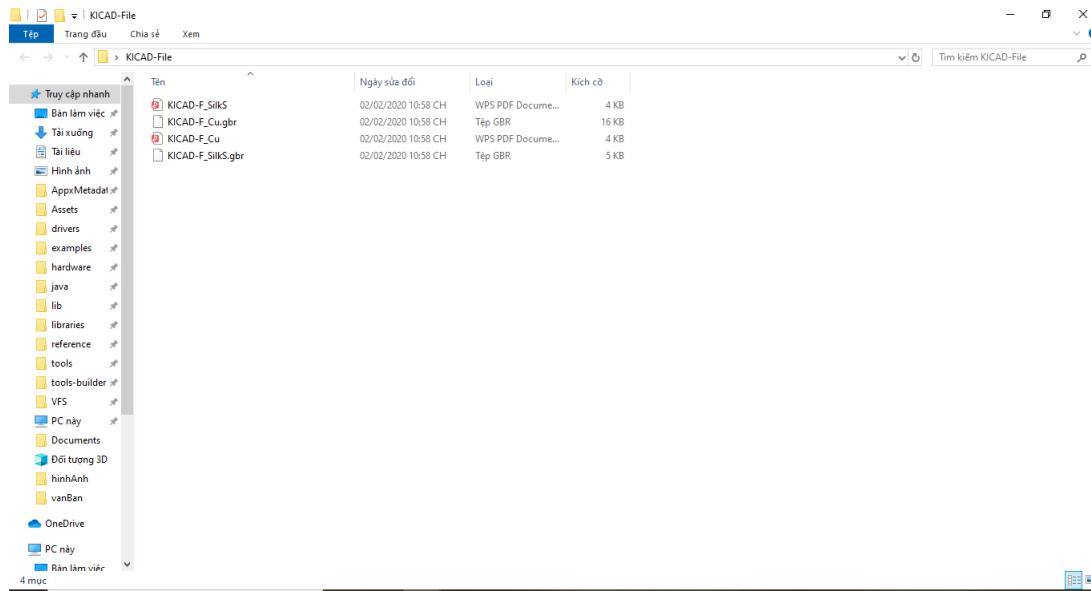
7 PCB AND THE COMPLETELY CIRCUIT.



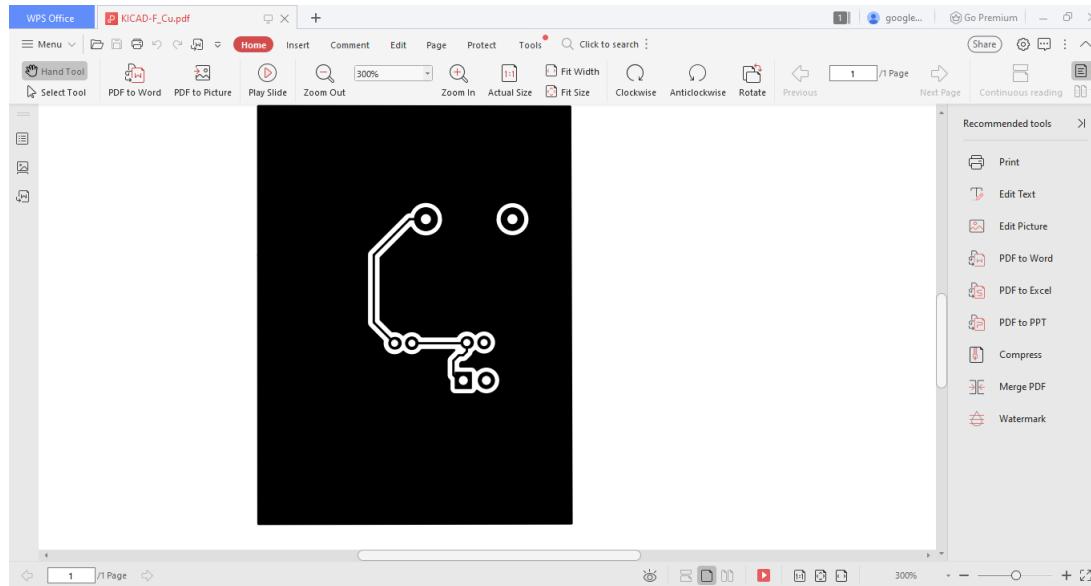
Designing stage is over and now, we come to generate gerber file to manufature stage(in industry) or pdf file if you want to make PCB handmade. Choose File->Plot and change information in Plot window you wil receive all file you want.



*Some information about Gerber: Gerber format is an ASCII vector format open for 2D binary images. This is the standard used by printed circuit software (PCB) to describe printed circuit board images, including copper layers, solder surfaces, instructions, ... During the making of the circuit board, Gerber is the standard input format for photoplotter and all other fabrication devices that need image data, such as class printers, direct photo printers or optical inspection machines. automatic learning (AOI), or to see reference data for different components. Gerber files also contain a 'stencil' layer for solder paste, and central locations of components to allow the PCB assembler to create folds, lay and mount components. The Gerber file format usually has the standard extension is .GBR.



As you can see by using Plot window I was generate 4 files, 2 files gerber and 2 files PDF of F.Cu layer and F.Silk layer. Here is F.Cu PDF file.



Yes that is a little step with PCB Editor, and now you can go to making a PCB circuit to do anything you want.



8 CONCLUSION AND SUMMARY.

Program name	Description	File extension
KiCad	Project manager	*.pro
Eeschema	Schematic editor (both schematic and component)	*.sch, *.lib, *.net
CvPcb	Footprint selector	*.net
Pcbnew	Circuit board board editor	*.kicad_pcb
GerbView	Gerber viewer	All the usual gerbers
Bitmap2Component	Convert bitmap images to components or footprints	*.lib, *.kicad_mod, *.kicad_wks
PCB Calculator	Calculator for components, track width, electrical spacing, color codes, and more...	None
PI Editor	Page layout editor	*.kicad_wks

All informations was display in six previous part, read it carefully and practice a lot, you will soon be an expert KICAD designer. But KICAD is just a step to create any machines or systems, always remember what do you want, what do you need and what do you can do now, thinking careful. The following picture is also the last picture of KICAD tutorial, it have all step to create an accomplished PCB circuit, read it and practice, wish you soon be a good designer.



8 CONCLUSION AND SUMMARY.

