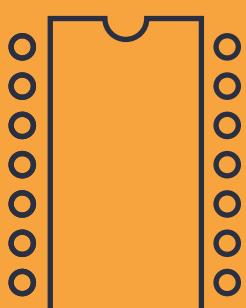
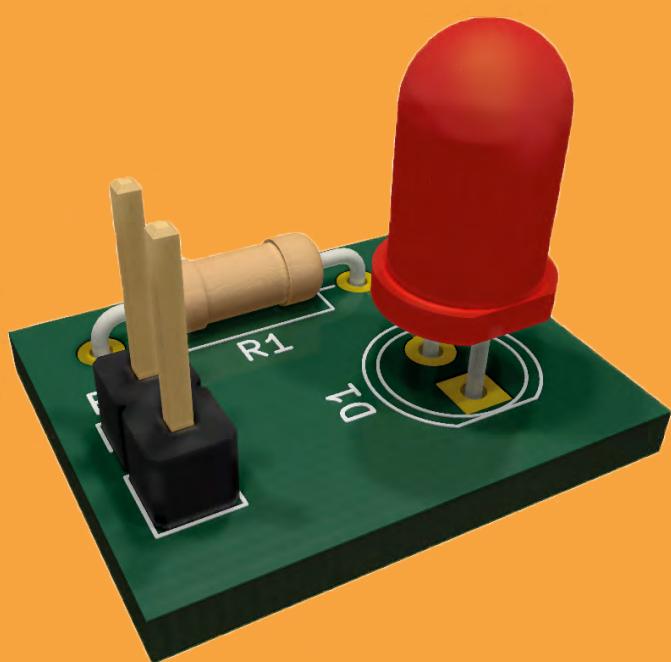


ØYVIND NYDAL DAHL

Make Your First Printed Circuit Board



 ohmify



Table of Contents

Introduction

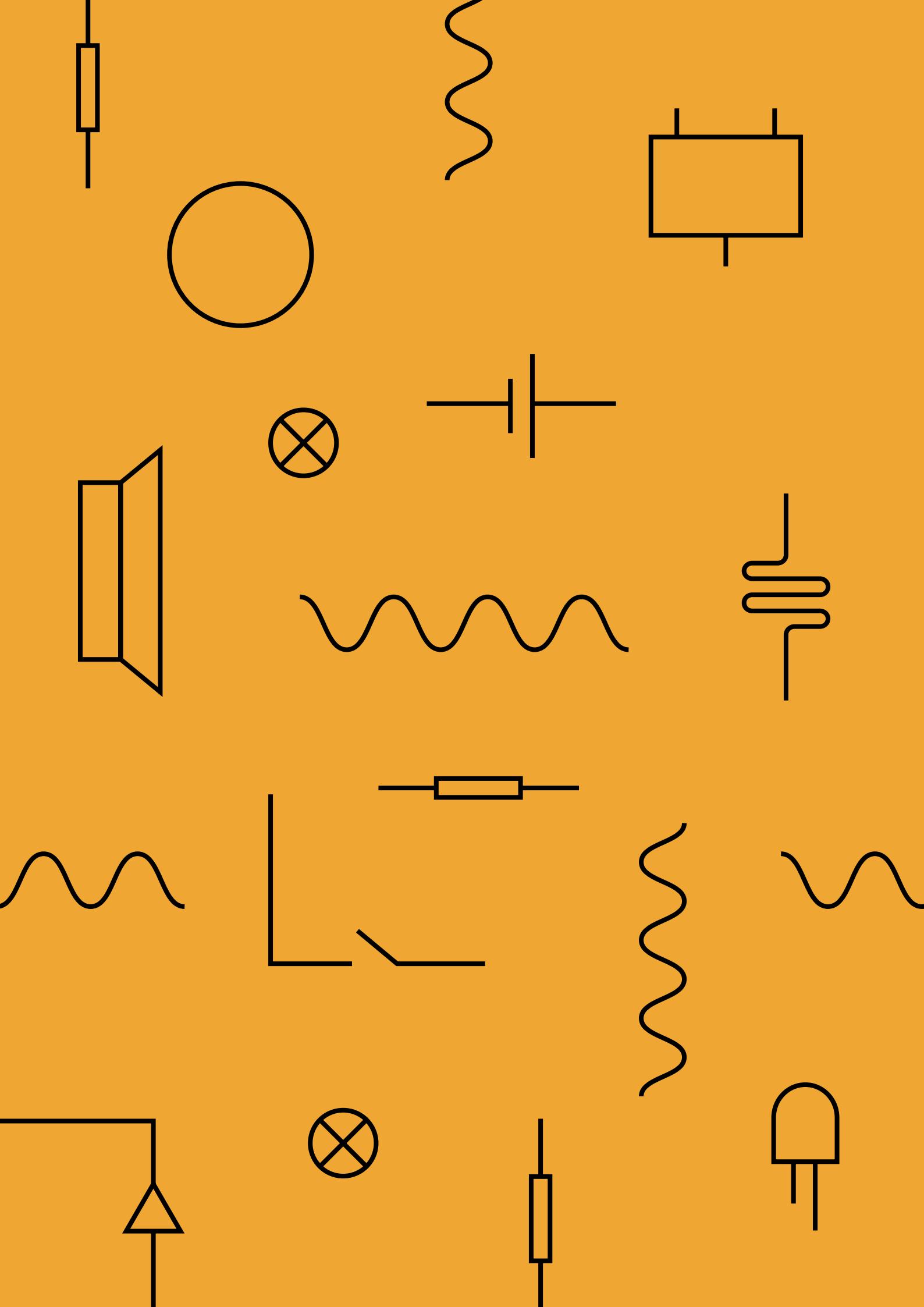
Foreword	Page 1
1. The Software: KiCad	Page 2

Drawing Your Schematics

2. Starting a New Project	Page 4
3. Adding Components	Page 8
4. Adding Wires	Page 11
5. Adding Symbol Values	Page 13
6. Annotating the Schematic	Page 14
7. Adding Footprints	Page 15
8. Running the Electrical Rules Check	Page 20
9. Schematic Design Checklist	Page 22

Designing the Board for your Circuit

10. Starting a New Board Layout	Page 24
11. Positioning the Components	Page 26
12. Drawing the Tracks	Page 28
13. Drawing Your Board Outline	Page 30
14. Running the Design Rules Check	Page 32
15. Schematic Design Checklist	Page 35
16. How to Get Your Board Made?	Page 36





Hi, I'm Oyvind!

Hey, I'm Oyvind Dahl and I run an electronics membership site for people like you to get really good at building electronics projects.

Everything is explained in simple ways and my students learn by doing - not by being swamped with complicated jargon and maths.

Take a look at the course description here

<https://ohmify.com/>

In the meantime, enjoy this tutorial on how to make your first printed circuit board!

Keep On Soldering!

Øyvind Nydal Dahl

Introduction

When I started building circuits as a teenager, I used different types of prototyping boards. I only built circuits with few components because of all the wires I had to solder here and there to get it working.

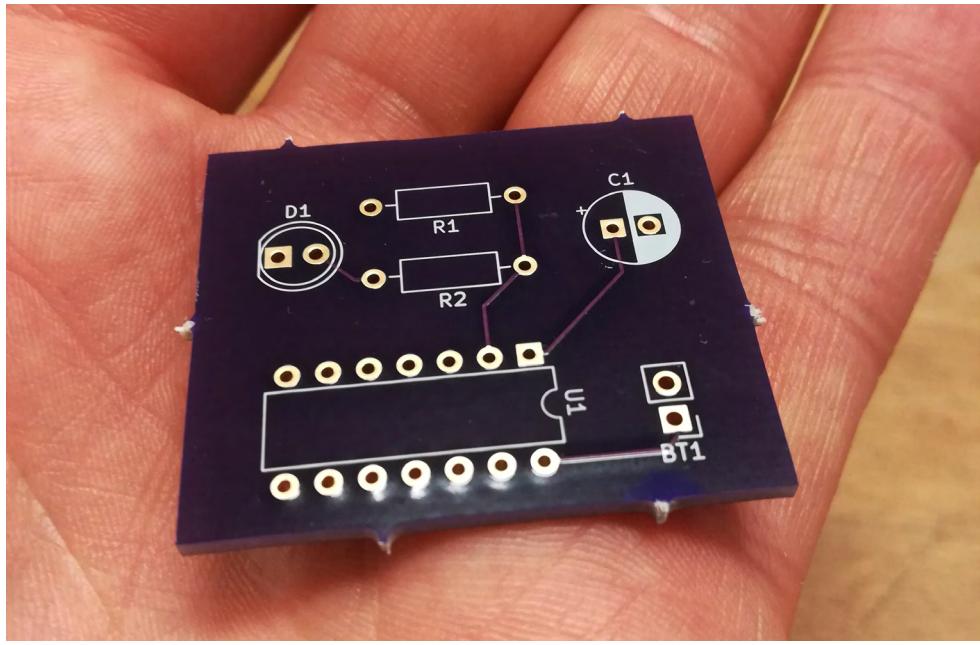
I remember finding cool projects with circuit diagrams included. Like guitar effect pedals. Or DIY weather stations. And I really wanted to build them.

But all of them needed at least 20-30 components. Usually much more. So I didn't dare to try as I knew that I would end up with a big mess of wires, almost impossible to troubleshoot. And even if I'd be able to do it, I'd be afraid to use it in real life in case I would mess something up.

It wasn't until my early twenties that I first tried to design my own custom Printed Circuit Board (PCB). And after that, I was hooked! The number of components for a project wasn't a problem anymore. I felt I could build anything!

When you design your own custom PCB, you'll draw the connections in peace and quiet on your computer. You can double and triple check your connections easily. You even get help from the design program to check for errors.

Next, send it off to a prototype provider and get some copies of your board made for just a few bucks!



Then you'll have a board in hand that you know has all the correct connections. No wires needed. All you need to do is to solder the components to the board, just like a soldering kit, and your circuit should work.

In this book, you'll learn how to make your first PCB. I'll show you by creating a super simple project in KiCad. After you've done one project, you'll know the basics and can design more PCBs on your own.

But if you want more instructions, more projects to try, and a place to ask questions and discuss electronics projects – I also invite you to check out my electronics community Ohmify:

<https://ohmify.com/join/>

So, let's jump right into it!

1. The Software: KiCad

There are several software alternatives for designing PCBs. KiCad is a free and really good alternative. It's used extensively by both hobbyists and professionals. That's what we'll use in this book.

KiCad is available for Windows, macOS, Ubuntu, and many other platforms. Find the platform alternatives and download the installation files for KiCad here:

<http://kicad-pcb.org/download/>

Action Points:



1. Download and install KiCad.

(Note: If you get asked during installation, make sure you say Yes to installing the standard libraries).

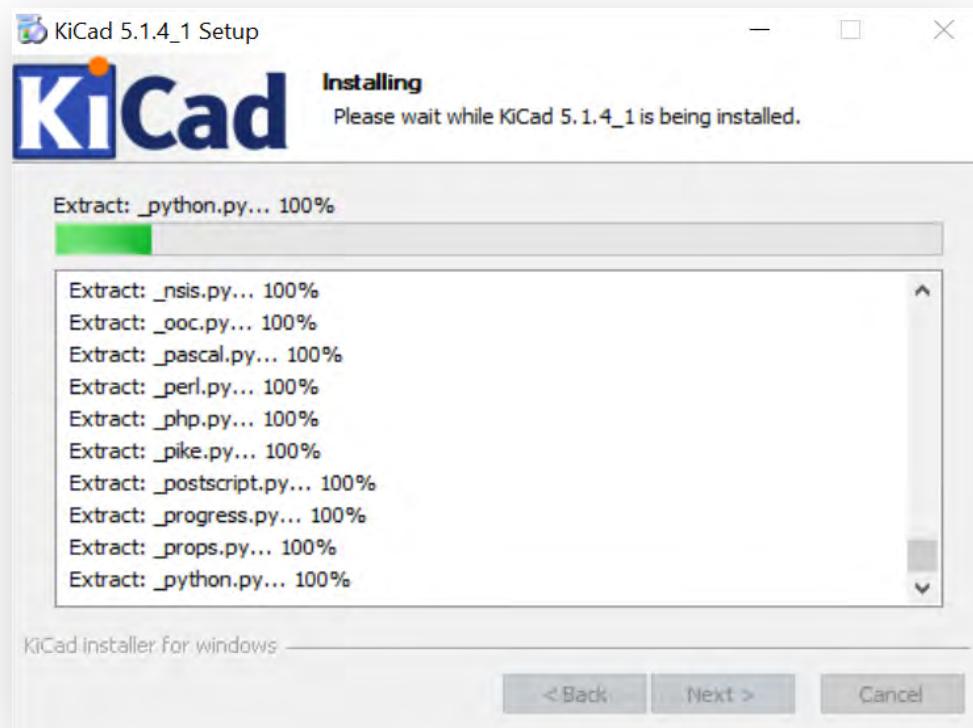
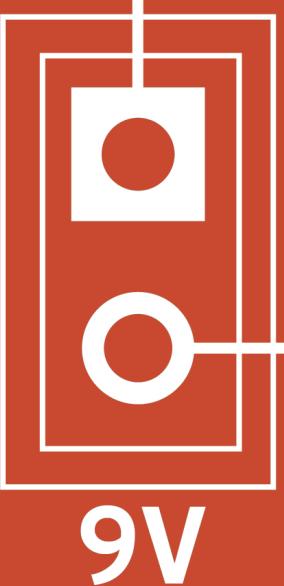
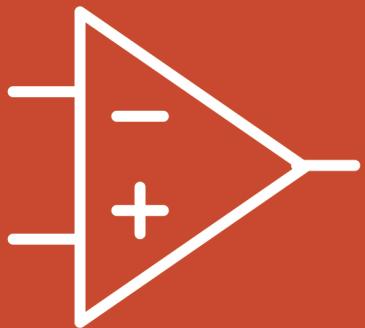
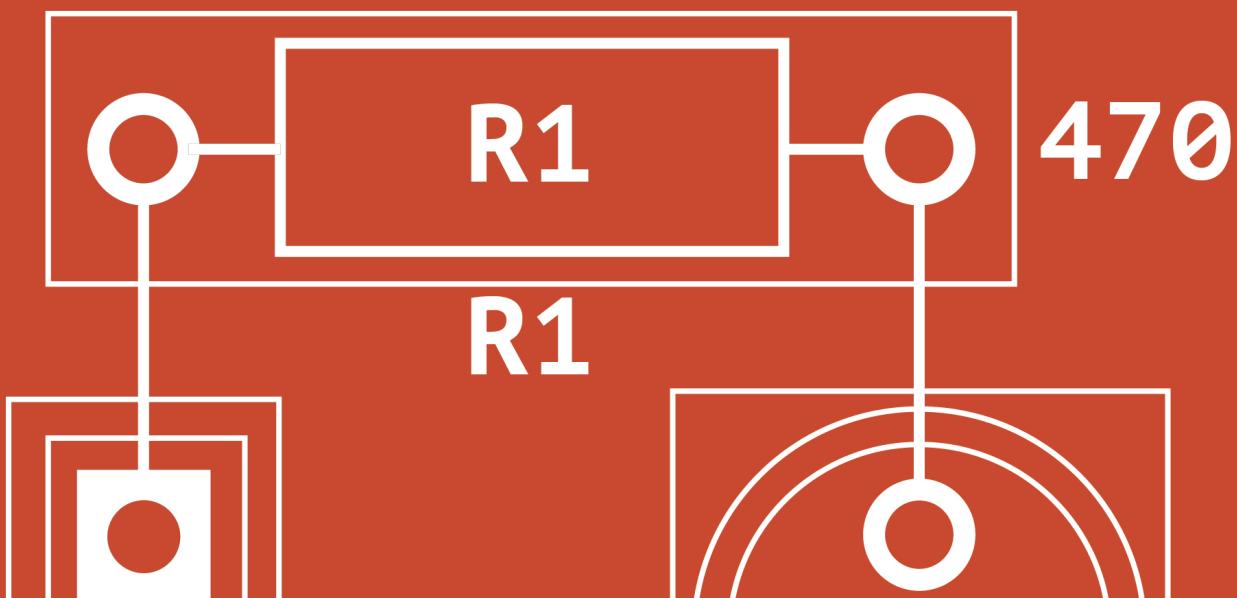


Image 1.1 Installing KICAD



PART 1

Drawing Your Schematics



2. Starting a New Project

When you open up KiCad, you'll see the window shown in Image 2.1. From here, you can open or create new projects.

To create a new project, click on the *File* menu and select *New -> Project*.

Choose where you want to save your project and type in a name for it. Then click on the *Save* button

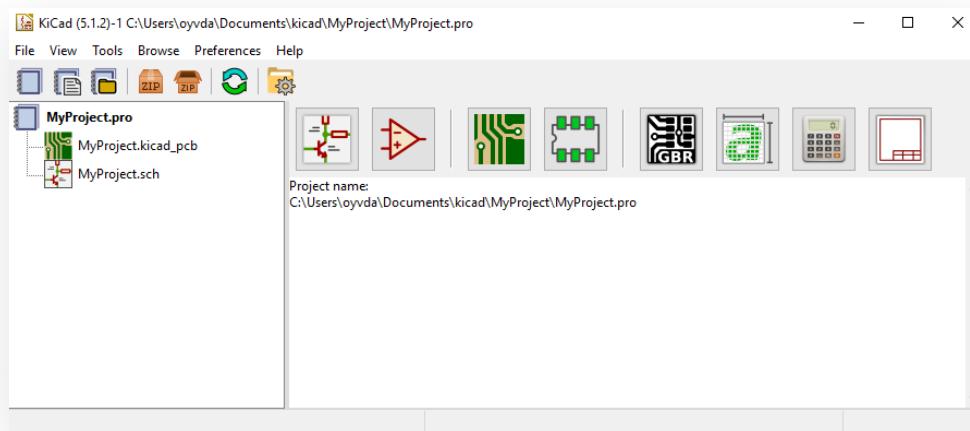


Image 2.1 The main KiCad window

Your new project will be empty. It's now your job to first create a schematic, then later draw the board layout. You create the schematic from the *Schematic Layout Editor*. Start it by double-clicking on the schematic file on the left, or by clicking on the Schematic Layout Editor button (Image 2.2).

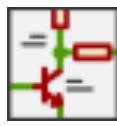
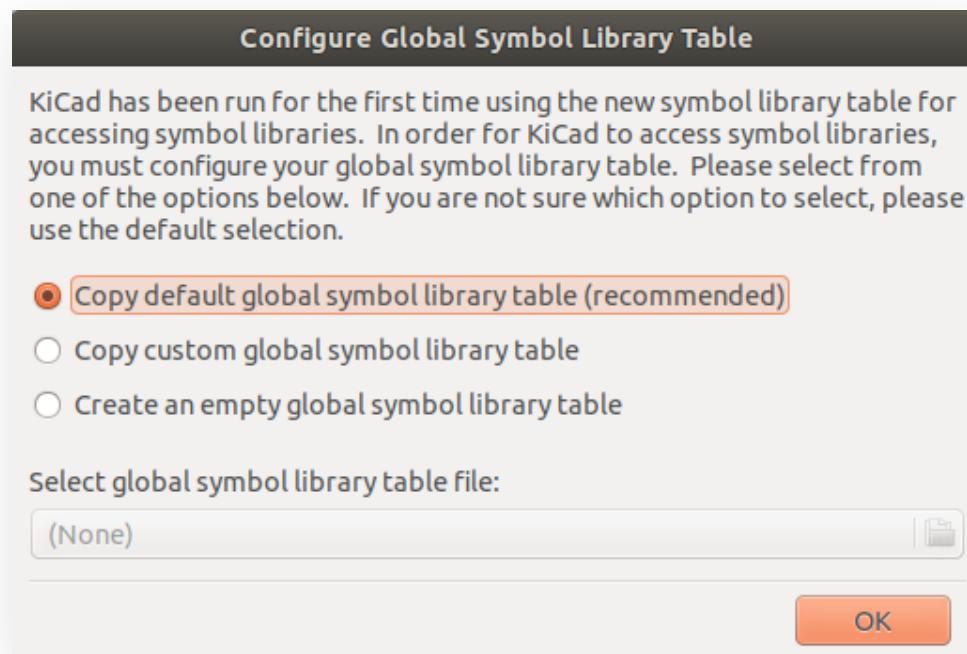


Image 2.2
Schematic Layout Editor button

When opening the editor for the first time, you might get asked to *Configure Global Symbol Library Table*. Just choose the recommended settings.



NOTE: If the recommended option is disabled, it most likely means that you installed KiCad without the default libraries. In that case, you need to download and install the libraries manually. Learn more about that here: <https://forum.kicad.info/t/library-management-in-kicad-version-5/14636>

Action Points:



1. Create a new project
2. Open the Schematic Layout Editor

3. Adding Components

Your schematic is where you draw your circuit. First, you add the components you want. Then you add wires to show how to connect your components.

To add a component, click on the *Place Symbol* button on the toolbar on the right (or press **A** on your keyboard), then click anywhere in your work area. This brings up the component chooser.



Image 3.1
Place Symbol button

From the component chooser, you can search for the component you want to add. Just type in what you're looking for. For example, if you need a resistor, type in "resistor." Look through the list of results and choose a good match*.

*What is a “Good Match”?

The most important thing is that you choose a symbol that represents the component you want to use in the best way possible. For example, when you search for “resistor,” you’ll get several options like “thermistor,” “trimmable resistor,” and such. But you also get a symbol that is just a resistor, nothing more. So that’s the best representation.

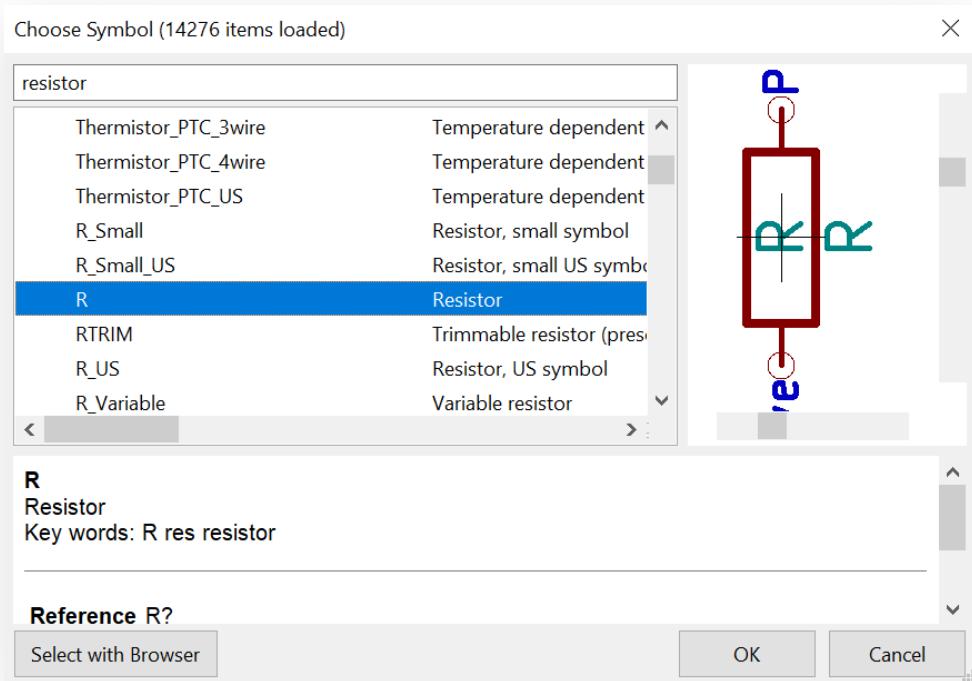


Image 3.2 The component chooser window

When you've chosen a component, place it in your schematic by clicking in your work area with your mouse. If you later want to move the component, place your mouse pointer over it and press **M** on your keyboard. To rotate the component, press **R**. You can zoom in and out by using your mouse wheel - or **F1** and **F2** on your keyboard.

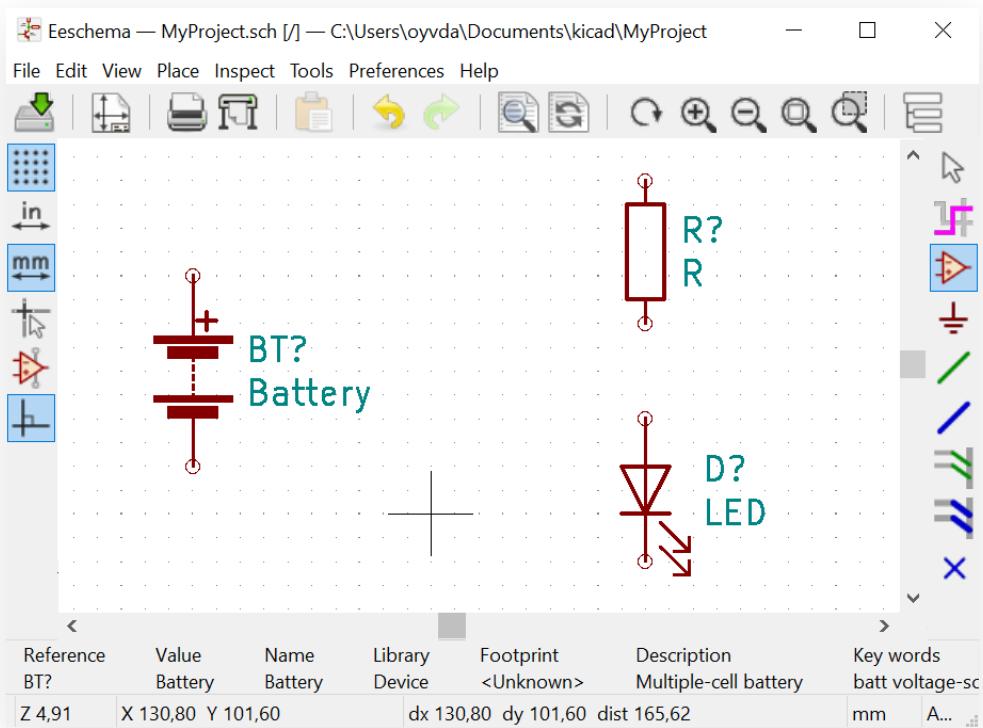


Image 3.3 A resistor, LED, and battery added to the schematic

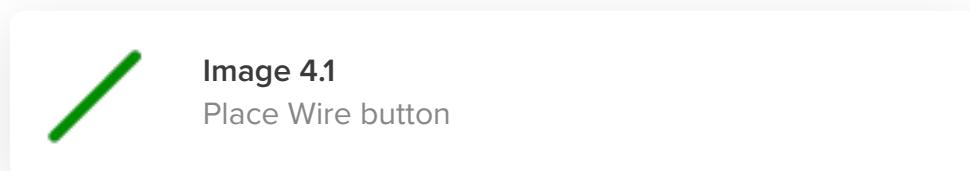
Action Points:



1. Add a resistor, an LED, and a battery to your schematic
2. Move the components so that they are positioned the same way as in Image 3.3

4. Adding Wires

Now you need to draw wires between the components to connect them. You can do this by clicking on the *Place Wire* button on the toolbar - or by pressing **W** on your keyboard.



Click on the pin of a component to start drawing a wire. Then click again on the pin where you want the wire to end.

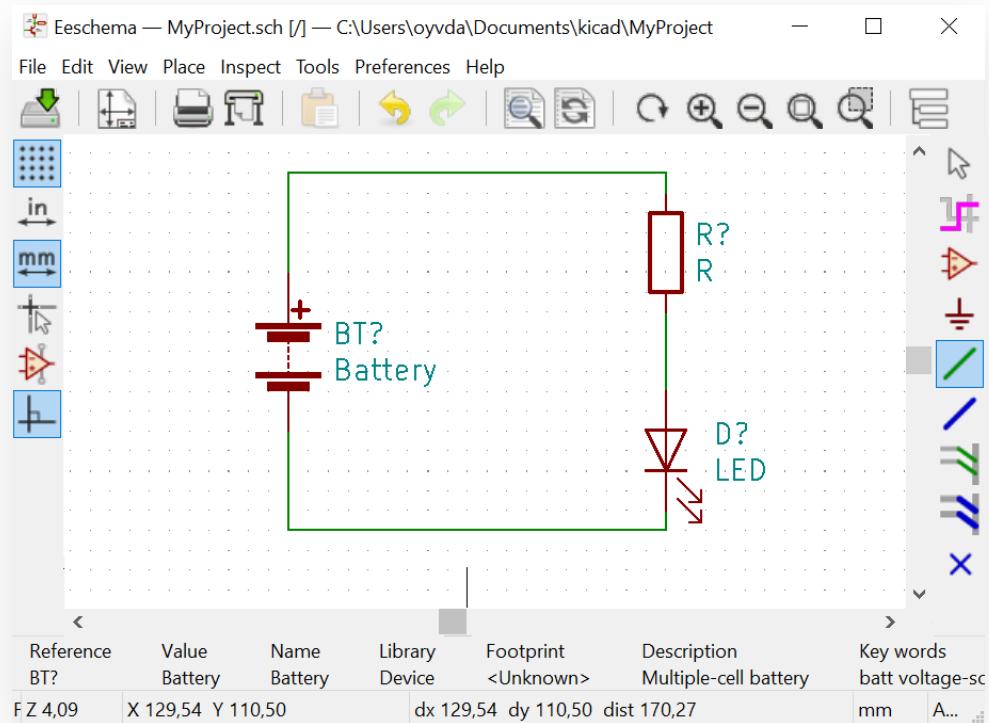


Image 4.2 The components with wires connecting them

If you want to move a component now that they are connected, you have two options: *Move* and *Drag*

If you place your mouse pointer over a component and press **M** on your keyboard, you *Move* the component without the wires. But if you press **G**, you *Drag* the component with the wires attached. If you start moving or dragging a component but then change your mind, just press the **Escape** key on your keyboard.

Action Points:



1. Draw wires between the components so that you connect them like in Image 4.2
2. Test out the two methods of moving a component (*Move* and *Drag*) after you've connected them

5. Adding Symbol Values

In most cases, a component should have a value. For example, a resistor should have a resistance value. A battery should have a voltage value, etc.

By pressing **V** on your keyboard, with your mouse pointer over the component, you can edit its value. You also find this option in the menu when right-clicking on the component.

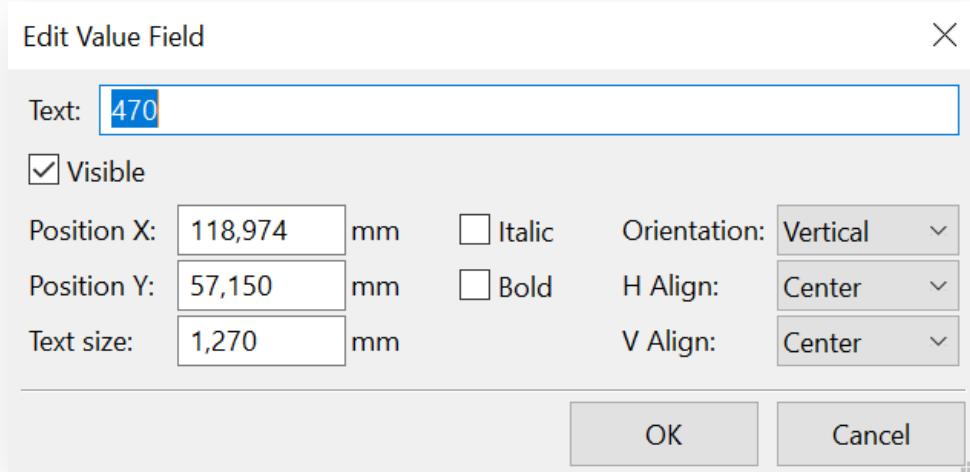


Image 5.1 The window for editing the component value

To set a resistor value to 470Ω , you'd normally write "470". And to set the battery value to 9 volts, it's normal to write "9V". For the LED you could, for example, write "2V RED" or something similar to indicate that you want to use a red LED with 2V forward voltage.

Note that the value you set for a component does nothing to your circuit. It just serves as a reminder for you, so that you'll remember what value to use when you build the circuit.

Action Points:

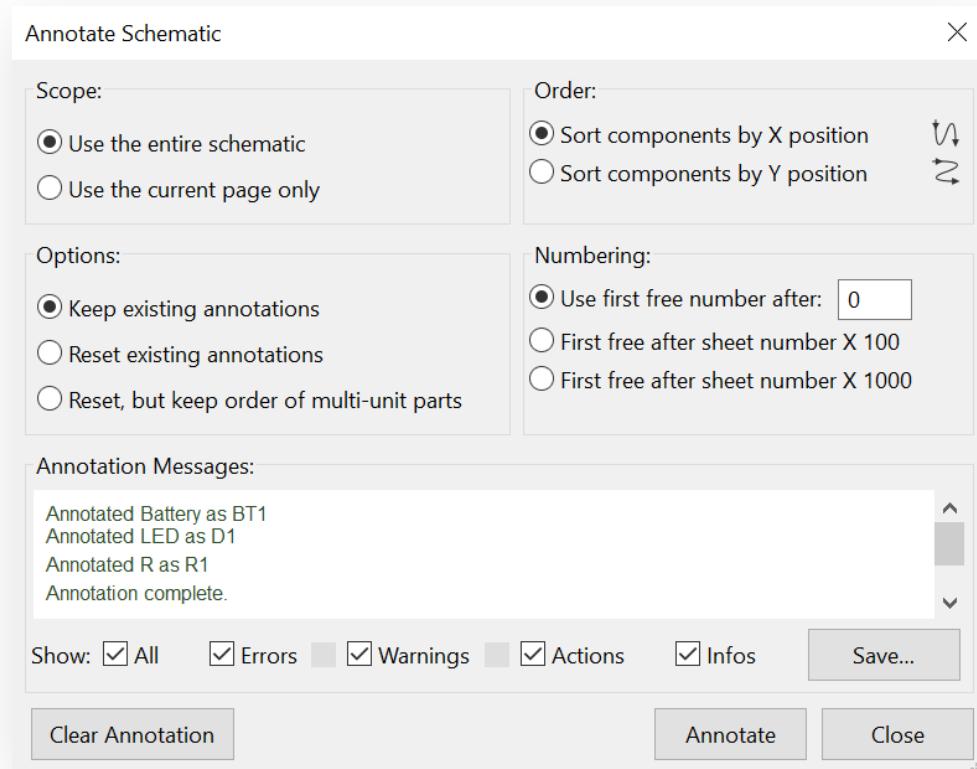


1. Add "470" as the value for the resistor
2. Add "9V" as the value for the battery
3. Add "2V RED" as the value for the LED

6. Annotating the Schematic

The next step is to *Annotate* your schematic, which means to give the components a unique identifier. For example, if you have several resistors, they will be named **R1**, **R2**, **R3**, and so on.

To do this, click on the Tools menu, then choose *Annotate Schematic*.



Leave the default options and click on the *Annotate* button. Then close the window. Now you can see that the components have numbers: **BT1**, **R1**, and **D1**.

Action Points:



1. **Annotate your schematic**

7. Adding Footprints

Before you can move on to drawing your board, there is one more thing you need to do; add *footprints* to your components.

The footprint is what will be printed onto the board for each component.

For example, for a through-hole resistor, you'll choose a footprint with two holes, so that you'll get two holes for soldering the resistor onto on your board.

Click on the *Tools* menu and choose *Assign footprints...* Sometimes it takes a bit of time to load all the libraries, so be patient.

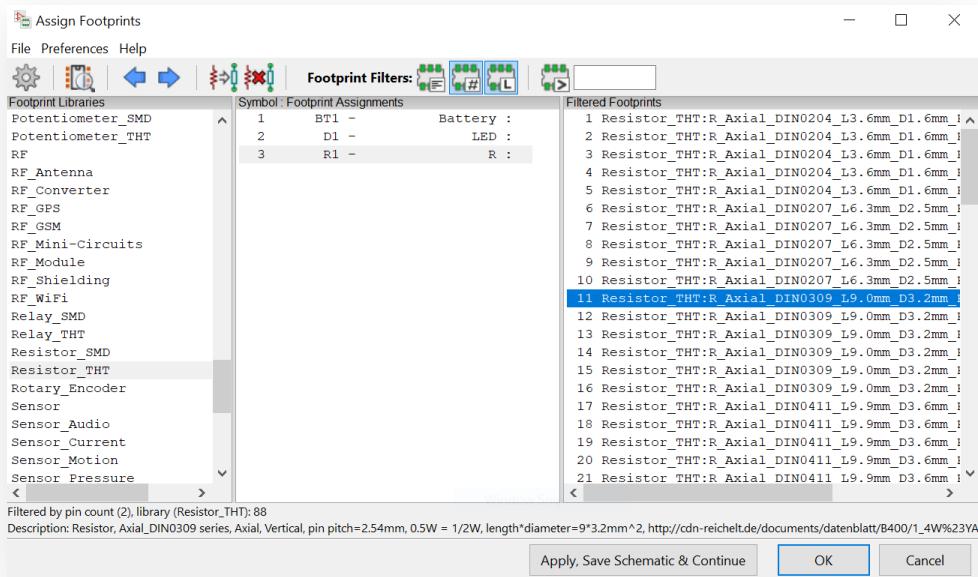


Image 7.1 The footprint selector

Image 7.1 shows the footprint selector. On the left side, you see all the footprint libraries you have available. In the middle, you have the components from your schematics. And on the right, you have a list of footprint options to choose from.



Image 7.2 The Footprint Filters selector

The *pin count filter* means you will only see footprints that have the same number of pins as your component. And the *library filter* means you'll only see footprints from the library that you have selected on the left.

To see what a footprint looks like, right-click on it and select *View Footprint*.

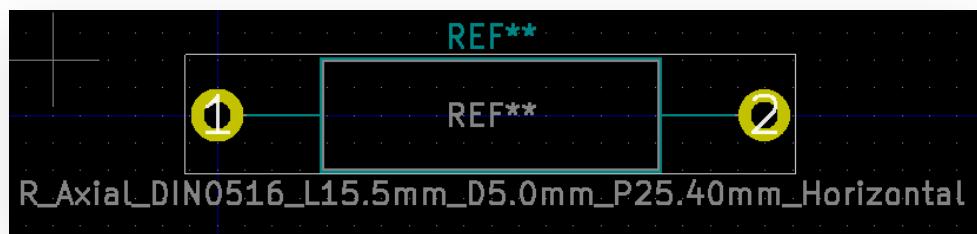


Image 7.3 An example of a footprint for a resistor. The yellow circles represent holes.

What Footprint to Choose For the Battery?

There are many ways to connect a battery. You can use a battery holder that's fixed onto the board. You can use a barrel connector. Or how about two pins? For 9V batteries though, I prefer using a battery clip. It's a clip you fasten to the top of the battery that has two wires going from it.



Image 7.4 A battery clip for connecting 9V batteries

Such a clip doesn't exist in KiCad at the time of writing. But all you need is a couple of holes where you can solder the wires from the clip. If you add a pin header with two pins, the footprint will be two holes.

One of the libraries that have many pin headers is the *Connector_PinHeader_2.54mm* library.

There are a few options here. Have a look at a couple of them by right-clicking and choosing *View Footprint*. Then click on the *3D Display* button to see what it looks like in 3D. Remember that it doesn't matter which one you choose, as long as it adds two holes to your board. Double-click a footprint to choose it.

What Footprint to Choose For the Resistor?

You'll find two resistor libraries among the standard KiCad libraries: *Resistor_THT* and *Resistor_SMD*.

THT stands for *Through-Hole Technology*. That's the type of component where you put the legs into holes on the board and solder on the other side like shown in Image 7.5. It's the classic type of component, often used by hobbyists as it's a bit simpler to solder.

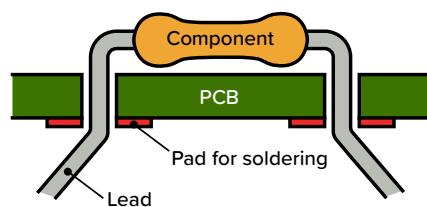


Image 7.5 Through Hole Technology (THT)

SMD stands for *Surface Mount Device*. That's the type of component that lies on top of the board. You solder them on the same side, as shown in Image 7.6. SMD is the standard component type for modern electronics. They are smaller and take up less space.

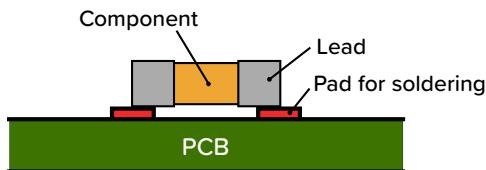


Image 7.6 Surface Mounted Device (SMD)

For this circuit, let's use Through-Hole Components, so select the *Resistor_THT* library.

There are tons of options. But note that it's not very important which option you choose here. All of them will give you two holes on the board to solder your resistor into.

But if you want the perfect fit, measure the resistor you plan to use – or look at its size specification from the store where you plan to buy it. Then choose the closest match you can find.

What footprint to choose for the LED?

The last component is the LED. Choose the *LED_THT* library to add a through-hole LED.

A typical LED is 5mm in diameter. There are many 5mm options in the library. A lot of them have the same footprint but different 3D models. Have a look at the 3D version of a couple of them to get an idea of the difference. (Right-click and select *View Footprint*. Then click on the *3D Display* button.)

Double-click to choose one. You can see my footprint selections in Image 7.7.

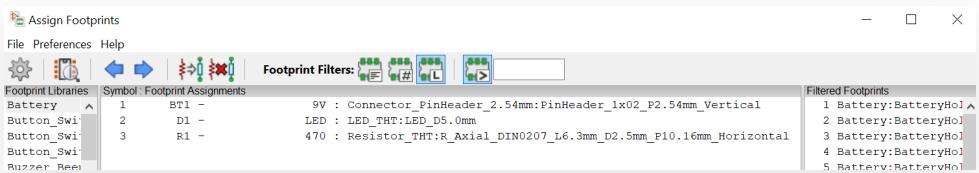


Image 7.7 My footprint selections for this circuit.

Action Points:



1. Add footprints for the resistor, the LED, and the battery. See image 7.7 for my selections

8. Running the Electrical Rules Check

It's easy to make errors, like forgetting to connect a component. Especially when you are drawing large circuits. To help you find these errors, use the tool *Electrical Rules Check (ERC)*.

Click on the *Inspect* menu and choose *Electrical Rules Check*. Or click on the Perform Electrical Rules Check button shown in Image 8.1.



Image 8.1
Perform Electrical Rules Check button

In the window that pops up, click on the *Run* button and look for error messages under *Error List*. A typical error here is if you forgot to connect the pin of a component to another component. Or if you accidentally moved a component so that it's no longer connected.

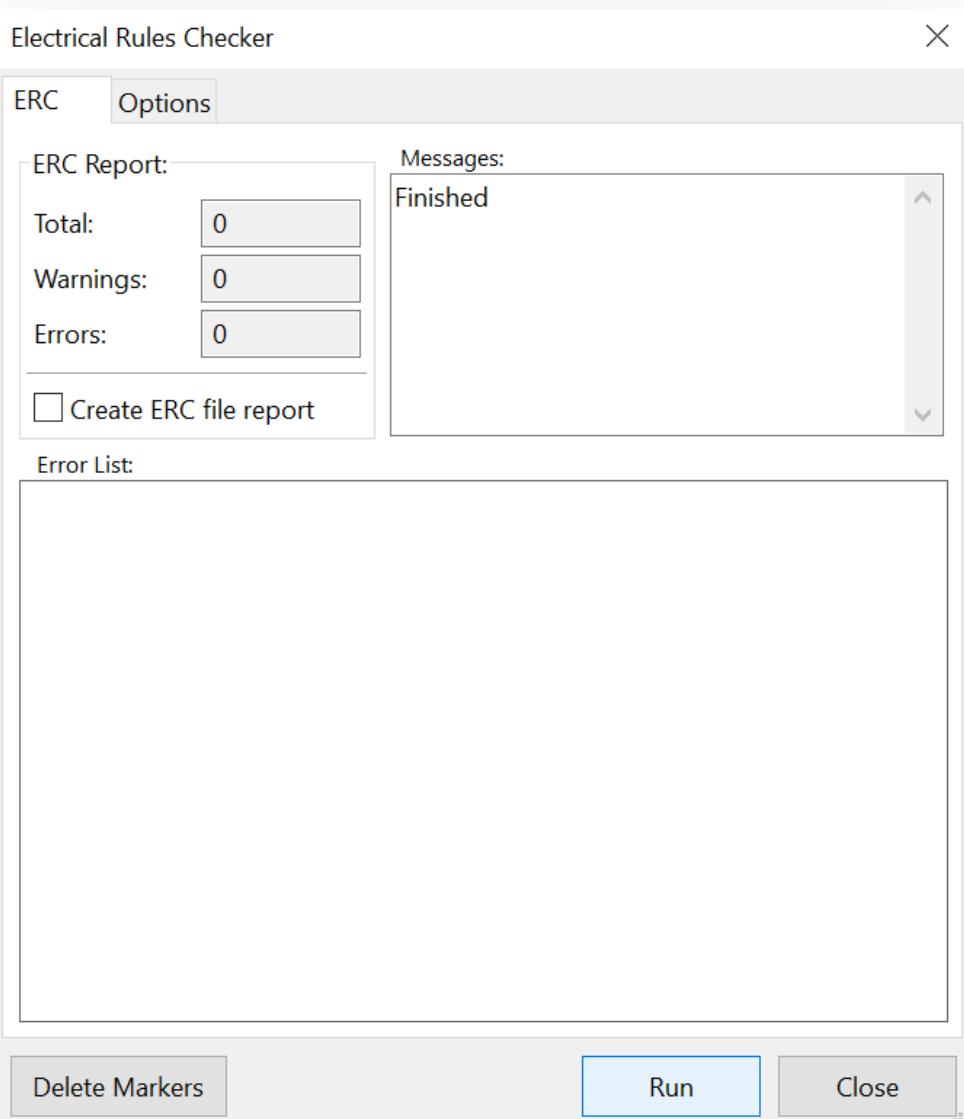


Image 8.2 The Electrical Rules Checker window

If there are no errors, you've finished the first phase of drawing the schematic. You can move on to the second phase – designing your board.

Action Points:

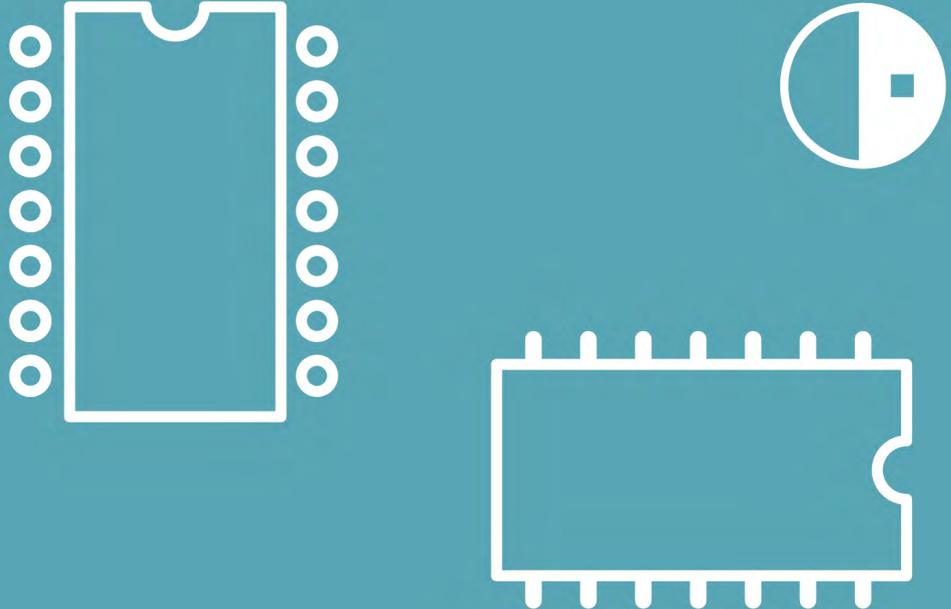


1. Run the ERC and fix any errors that appear

9. Schematic Design Checklist

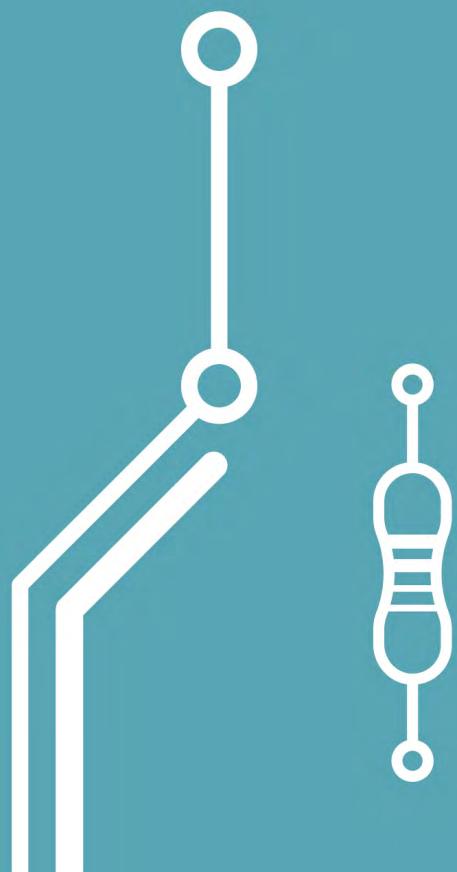
That's it for drawing the schematic. To make it easier for your next design, here's a checklist for all the action points you need to take before moving on to the board design:

- Add all components to your schematic
- Connect all components with wires
- Add values to all components
- Annotate your schematic
- Add footprints to all components
- Run the Electrical Rules Checker (ERC) and fix any errors



PART 2

Designing the Board for your Circuit



10. Starting a New Board Layout

You have defined how you want to connect the components in the schematic editor, and now it's time to put that onto your board. You need to decide where to place the components, then draw tracks to create the connections between them.

To start drawing your board layout from the Schematics Editor, click on the toolbar button that says *Run Pcbnew* to layout printed circuit board, shown in Image 10.1. Or go to the *Tools* menu and select *Open PCB Editor*. This opens up the Board Editor window.

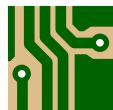


Image 10.1

The button to go from schematic design to board layout

The first thing you'll see is just an empty area. To get started, you need to retrieve information about footprints and connections from your schematic. You can do this by clicking on the toolbar button that says *Update PCB from schematic...* as shown in Image 10.2. You'll also find this option in the Tools menu.



Image 10.2

The button to update PCB from schematic

Keep the default options and click on the *Update PCB* button. This will load your footprints. Then click on the *Close* button. Click anywhere to place the components.

If you zoom in on the components, using either your mouse-wheel or F1/F2, you'll see white straight lines between the pins. These are called air-wires and tell you which pins to connect - according to your schematic.

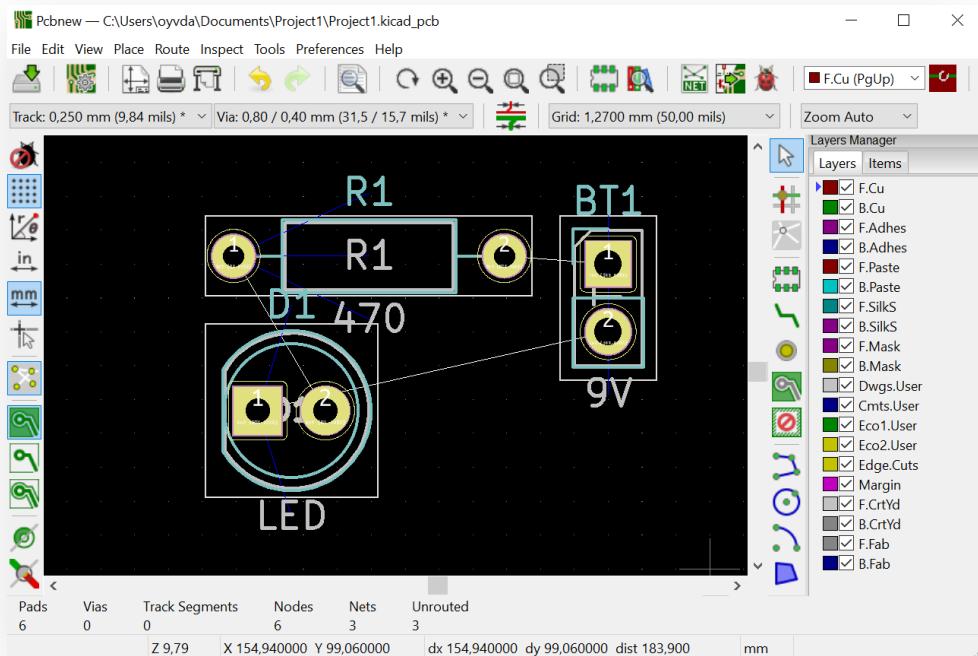


Image 10.3 The components imported from the schematic

Action Points:



1. Update your PCB from schematic, so that you see the three footprints you've chosen

11. Positioning the Components

Your next task is to position your components so that it becomes easy to connect the components according to your air-wires. If you're able to place the components in a way that no air-wires cross, that's the easiest.

I recommend positioning the components like the way you did in the schematic, if possible. It's not very important for such a simple board, but for more complicated boards it can simplify troubleshooting.

To move a component, move your mouse over it and press **M**. If you already have a group selected, the whole group will move. To deselect the group, press **Escape**. To rotate a component, press **R**. If you accidentally place your mouse over the component name, you'll move only the name. Just press **Escape** to try again.

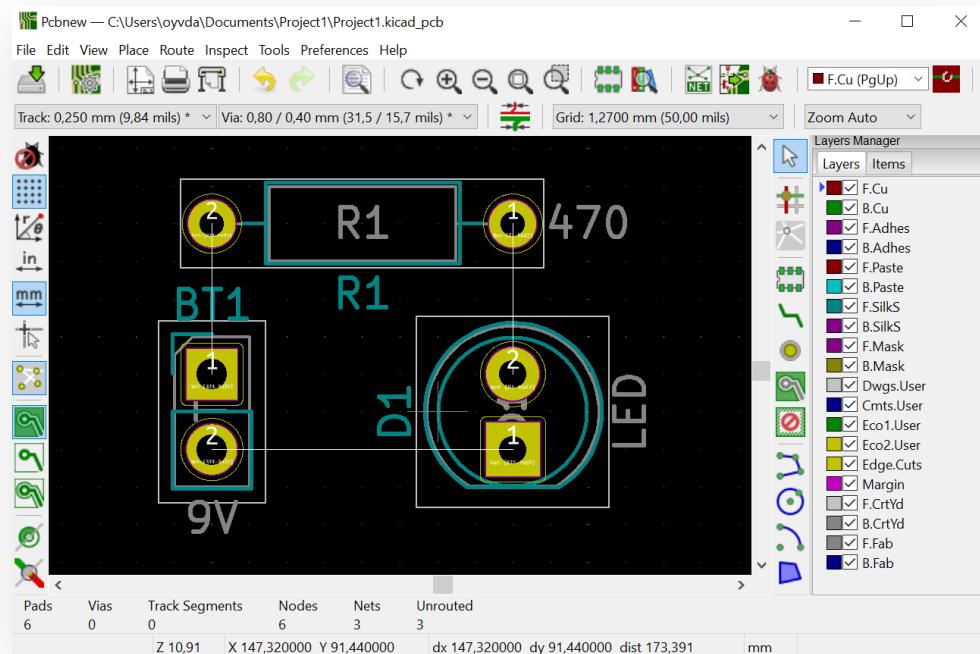


Image 11.1 The components placed so that no air-wires cross

It is possible to place components both on the front side and on the back side. To change the side of a component, right click on it and select *Properties*. Then look for the *Board Side* option to select on which side to place the component.

Action Points:



1. Place the components in an ordered way, similar to the way you placed them in the schematic diagram.

12. Drawing the Tracks

When you're happy with the placement, the next step is to replace the air-wires by drawing tracks to connect the components. The tracks you draw will become copper traces on your finished board.

Start drawing tracks by clicking on the *Route Tracks* button on the toolbar on the right (shown in Image 12.1) - or press **X** on your keyboard.



Image 12.1

The button for routing tracks

You can choose if you want to draw on the front or the back of the board, by selecting a layer in the Layer Manager.

- **F.CU is the front (or top) side.**
- **B.CU is the back (or bottom) side.**

When you use through-hole components, you can connect tracks to them on both sides of the board. But if you use surface mount components, you can only connect tracks on the side where the component is placed.

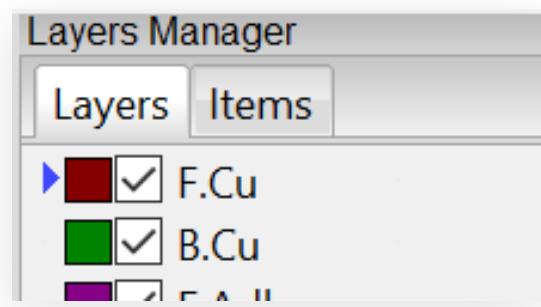


Image 12.2

The Layers Manager

Click at the beginning of an air-wire to start drawing a track. Click again where the air-wire ends to finish drawing the track.

When there are no more white lines, you're done routing the tracks.

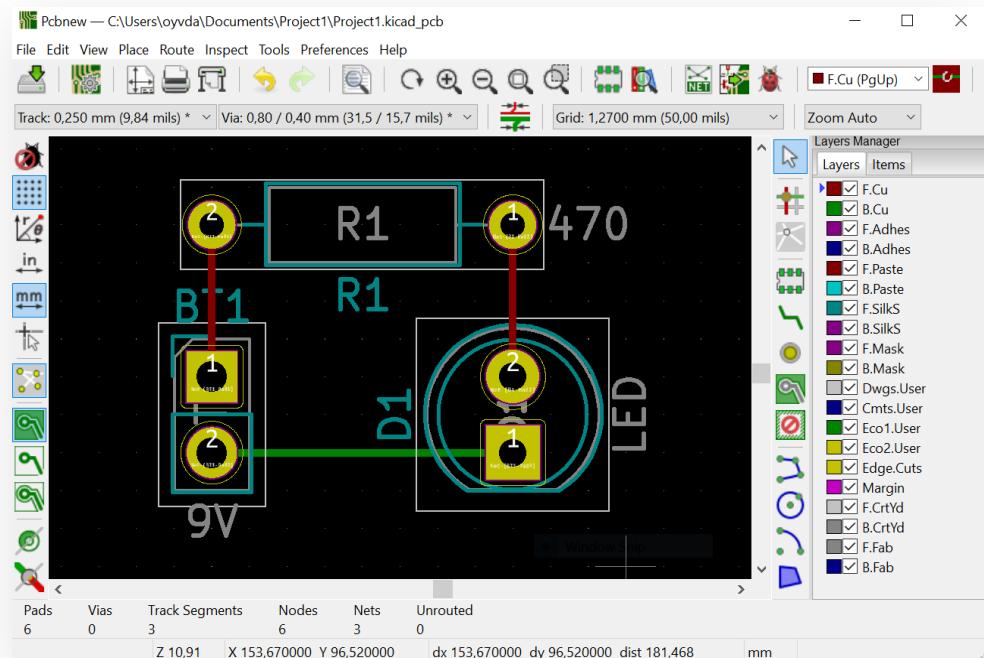


Image 12.3 The board with all its tracks routed

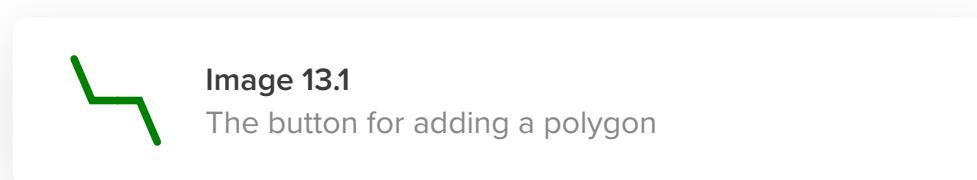
Action Points:



1. Draw all the tracks so that you have no more air-wires left

13. Drawing Your Board Outline

Next, you need to draw the outline of your board. To do this, you need to add a polygon on the **Edge.Cuts** layer. Click on the *Add graphic polygon* button on the toolbar, as shown in Image 13.1.



Select the **Edge.Cuts** layer.

Click to start drawing the polygon. Click again for each corner. To complete the polygon, click at the point where you started.

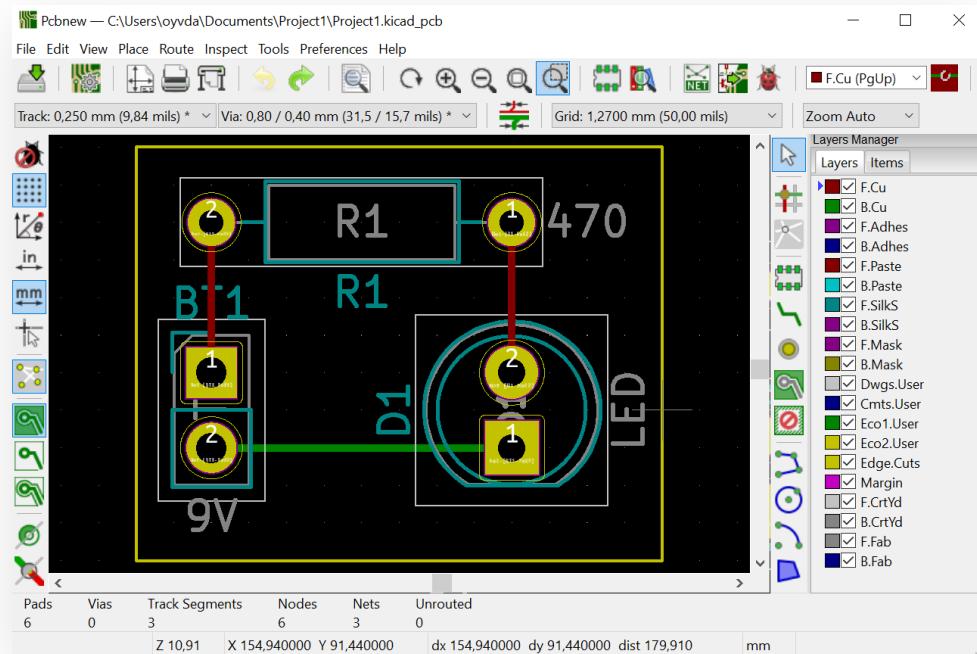


Image 13.2 The board with the board outline added

Action Points:



1. Add a polygon in the *Edge.Cuts* layer that encloses all your components and tracks

14. Running the Design Rules Check

To check if you've made any errors, you need to run the *Design Rules Checker* – also called the *DRC*. Typical errors are placing two components too close to each other, or forgetting to draw a track.



Image 14.1

The button for running the Design Rules Checker (DRC)

Click on the *Design Rules Checker* button, shown in Image 14.1, or find it under the *Inspect* menu. Leave the default options and click on *Run DRC* button.

Check both the *Problems* tab and the *Unconnected Items* tab. The *Problems* tab will list any design issues, such as a track being too close to another track. The *Unconnected Items* tab will list any air-wires that you have not connected by drawing a track.

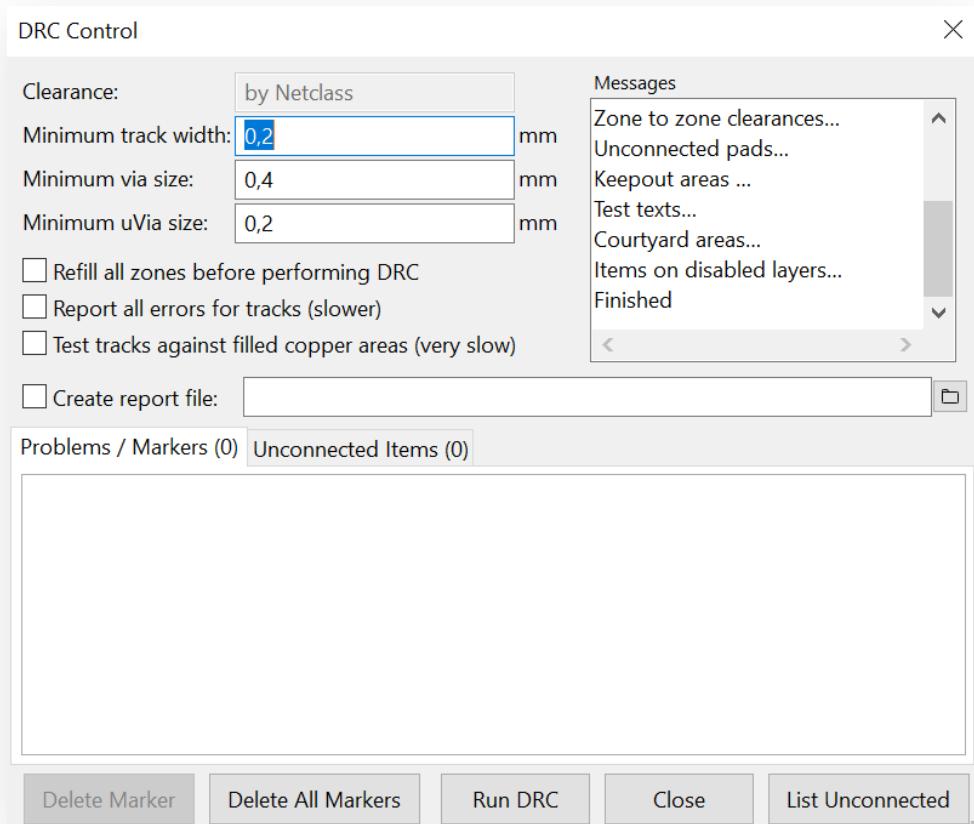


Image 14.2 The Design Rules Checker window without any errors

If you see any items on any of these tabs, make sure you fix them before moving on. If you have a lot of errors, do not despair. Often one little mistake can result in lots of messages. Just focus on one message and try to figure out what it says. Then fix it and run the DRC again.

When you have zero errors listed here, you are done. Your board is finished! Click on the *View* menu and select *3D Viewer* to see your finished board.

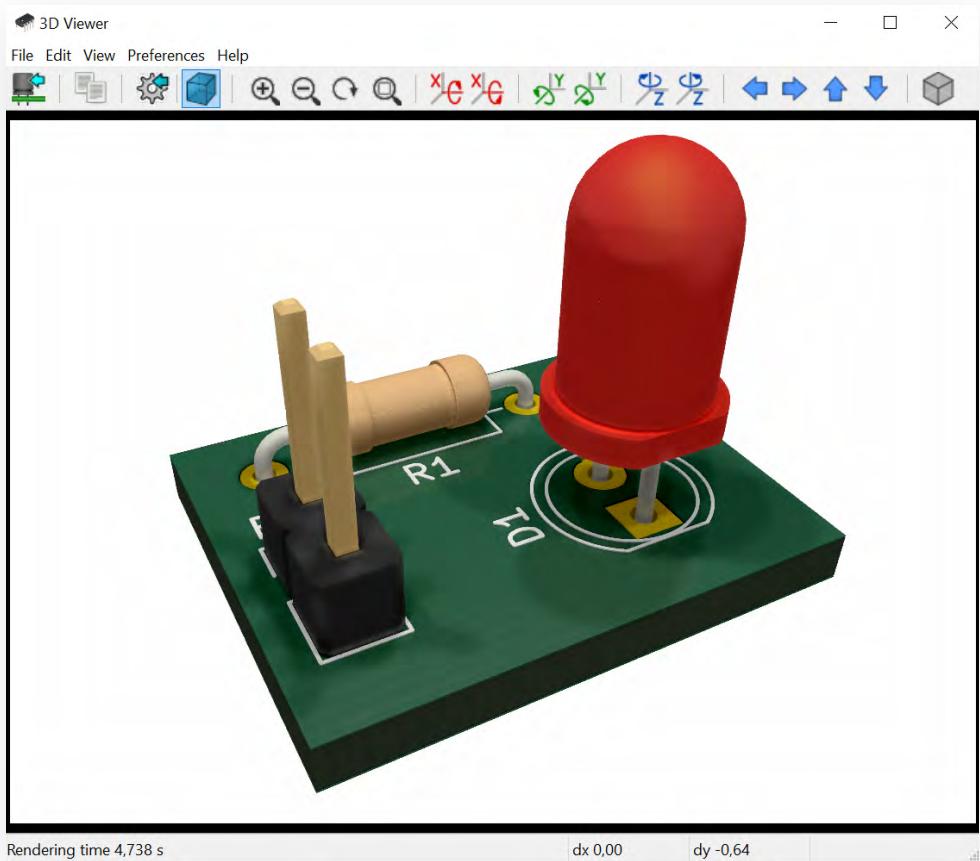


Image 14.3 3D view of the finished board

Action Points:



1. Run the Design Rules Checker (DRC) and fix any errors that you might have
2. Check your design in the 3D Viewer

15. Schematic Design Checklist

That's it for drawing the board layout. To make it easier for your next design, here's a checklist for all the action points you need to take to complete the board design:

- Open the Board Layout Editor and run Update PCB from schematics
- Place your components in an orderly way
- Draw all the tracks so that you have no more air-wires left
- Create your board edges by adding a polygon in the Edge. Cuts layer that encloses all your components and tracks
- Run the Design Rules Checker (DRC) and fix any errors that you might have
- Check your design in the 3D Viewer

16. How to Get Your Board Made?

You're now ready to get your board made. This is something that many think is hard and expensive. But that couldn't be farther from the truth!

These days, PCB manufacturing is available at low cost to anyone who wants to get their circuit boards made. You can find companies that will make a few copies of your boards for only \$5-10 (USD).

And it's become so easy to order that even the most computer-challenged among us can do it.

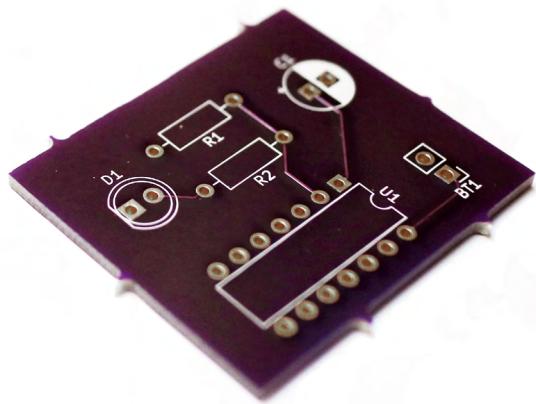


Image 16.1 A blinking LED board from one of the courses at: <https://ohmify.com>

The Easiest Way

The US manufacturer OSH Park allows you to drag and drop your KiCad PCB file onto their website. That's it! They process the file in real-time and show you the details of your board so that you can double-check for errors.

Check the file, make the payment, then all you need to do is to sit back and let them make the board for you. The whole process takes less than 5 minutes.

I think we'll see more manufacturers offering this in the future.

The Traditional Way

The traditional way of submitting your board design to a manufacturer is via Gerber and Drill files. This is a format that all the usable software alternatives for designing PCBs can output, and that all PCB manufacturers can read.

From the PCB Editor in KiCad you can create Gerber files by going to the File menu and selecting Plot... Click on the Plot button to create the Gerber files. To create Drill files, click on Generate Drill Files... to open the Drill File dialog. Then click the button Generate Drill File.

There are a lot of options when generating these files. Often the PCB manufacturer you want to use will have instructions on what options to choose.

List of PCB Manufacturers

Here's a list of a few places you can get your board made:

OSH Park (US)

<https://oshpark.com/>

Eurocircuits (EU)

<https://www.eurocircuits.com/>

Seeed Studio Fusion Service (China)

https://www.seeedstudio.com/fusion_pcb.html

PCBWay (China)

<https://www.pcbway.com/>

jlcpcb.com (China)

<https://jlcpcb.com/>

allpcb.com (China)

<https://www.allpcb.com/>

There are many more. Just search for “PCB Prototypes” or similar on your favorite search engine.

Learn to Build Your Ideas With Electronics

Congratulations, you've made it to the end!

If you're new to electronics, you probably did not understand the details of how to do everything in the tutorial, and that's okay. The fact that you read all the way to the end means you have the interest needed to get really good at electronics.

I'm constantly creating new learning material to help people build their own ideas with electronics. Whether you want to build an automatic cat door for your house or the next must-have gadget for the world, there are some steps you need to take to build up the skills and knowledge necessary to build your own inventions with electronics.

To help people from all backgrounds learn these steps, I've created Ohmify. It's an online electronics builders club with lots of courses and resources to learn electronics such as:

- Lots of courses on skills such as soldering, Arduino programming, basic electronics and more.
- Step-by-step instructions for cool projects like a robot, music player, phone charger and the Atari Punk Console
- A community forum for discussing ideas, asking questions and getting help to do your projects.

Learn more about Ohmify here:

<https://ohmify.com/join/>





