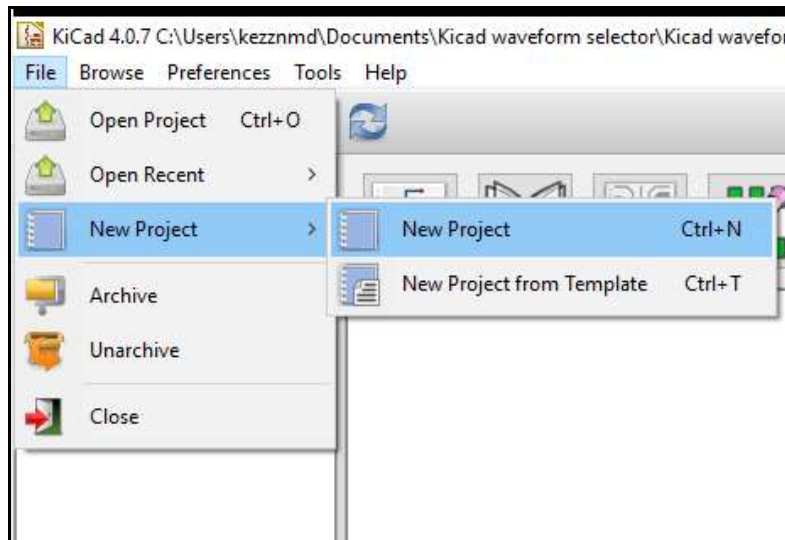


Designing Schematic Diagram using Eeschema

1. Create new Project

At the home menu of KiCAD, go to **“File > New Project > New Project”** and create your project file under new folder in any directory that you prefer.

You will notice that a project tree is automatically created at the right side of windows pane in KiCAD’s home menu.



2. Create new schematic

To create a new schematic, double click on your **“.sch”** file extension under the project tree or click on the *Eeschema application button*.



3. Edit Page Setting

Set your page setting by going to **“File > Page Settings”** and edit the details of your schematic design

4. Grid Settings

When editing schematic diagram in Eeschema, choosing the right grid setting is important to avoid your schematic diagram from being cluttered.

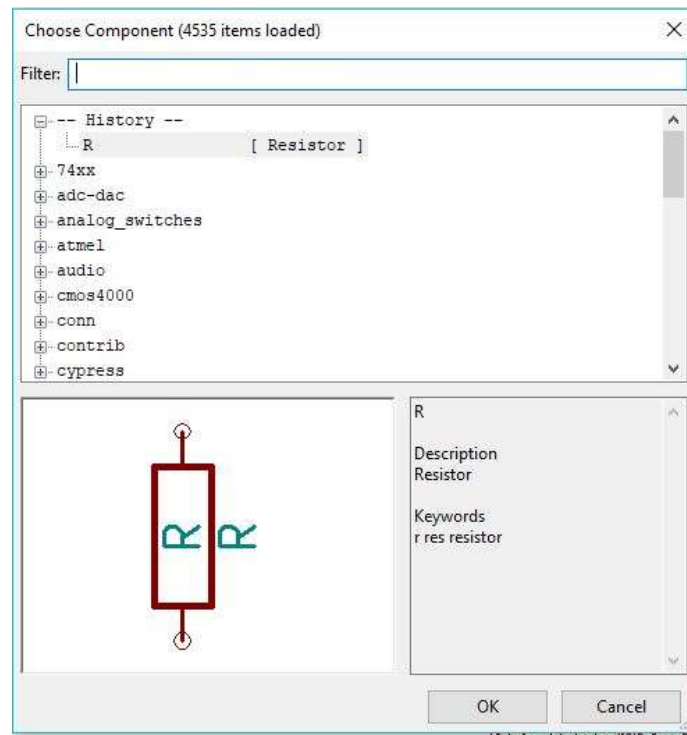
To view grid settings, **“Right-click > Grid Select”** and choose the appropriate grid view depending on your preferences.

Each ‘dots’ that appear on your screen is referred to your grid setting that you have chosen. If the ‘dots’ seems to be missing it may be because you have zoomed out too much (or your screen is dirty, wipe

5. Add Component



To start adding component click the *Place Component* and left-click anywhere on your page to place the component or go to “ **Place > Component**”



6. Edit Component's Value

Hover your mouse over the component and then “**Right-click > Edit Component > Value** “. Type in the value of your component e.g 10K for 10 000 Ω resistor.

7. Edit Component's Reference Field

Hover your mouse over the component and then “ **Right-click > Edit Component > Reference** “ and start type your component's reference field or number e.g R4 for 4th resistor.

This step can be skipped by using the auto annotate function. To do so, refer to step number 12.

8. Move component / Change orientation

Again, hover your mouse over the component , “ **Right-click > Orient Component**” and then choose your preferred orientation for them component.

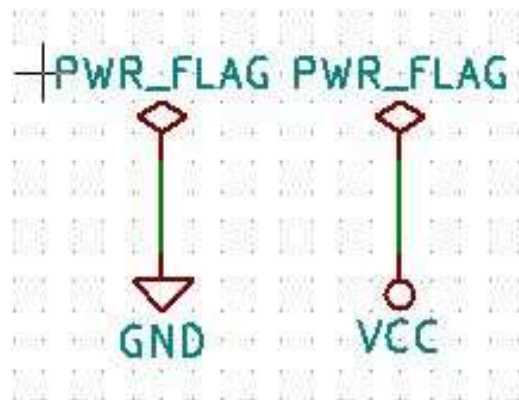
9. Place Wire

Now you can start connecting the all the component, go to **“ Place > Wire”** or click the icon .



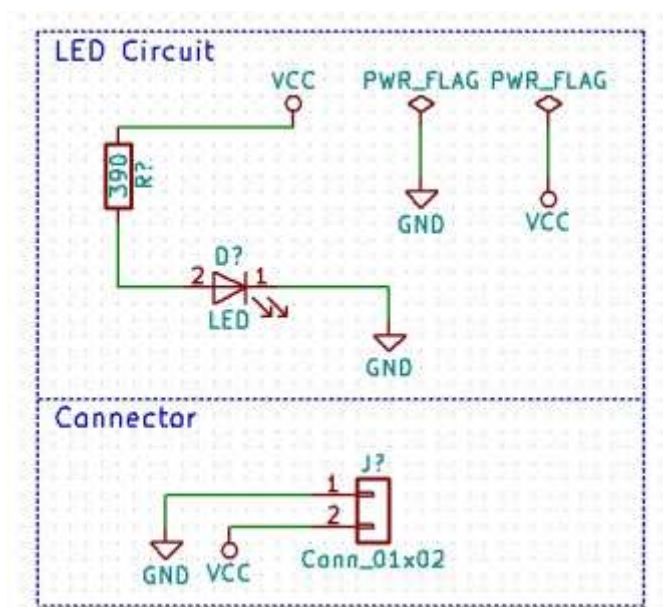
10. Adding Power Flag

It is now necessary to add a Power Flag to indicate to KiCAD that power comes in from somewhere. Place the PWR_FLAG and connect them to a GND pin and to VCC as shown below.



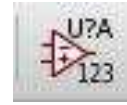
11. Place Graphic Line and Text

It's a good habit to keep you schematic neat so that you can troubleshoots your circuit with ease. Depending on your circuit configuration, you might need to place a separator so that each part of your circuit is labelled. To do so go to **“ Place > Graphic Polyline”** or **“ Place > Graphic Text”** to label your schematic.



12. Annotate Component

This step will auto annotate every component on your schematic diagram. To do so go to “ **Tools > Annotate Schematics > Annotate** ”. You can also click the Auto Annotate to do so



It's a good idea to do this step after you have completed connecting all your component. Else, you can also manually annotate each component by referring to step number 7.

13. Perform Electrical Rules Check.

This step is crucial before you can start moving on designing the actual PCB. Click the *Perform Electrical Rules Check* or go to “ **Tools > Electrical Rules Checker > Run** ”.



Refer to the error message shown on the box and make corrections on your schematics. You can also refer to KiCAD Documentation to troubleshooting the error message.

14. Generating Netlist.

Generating the Netlist enables KiCAD to understand what components and connection that you have made. This also enables you to update any changes that you have made i.e you can change or add the components during designing PCB.

Go to “ **Tools > Generate Netlist File > Generate > Save** ” and save on your working directories or click *Generate Netlist* icon



15. Saving File.

Finally! You can save your work by going to “ **File > Save Schematic Diagram** ”. Close your Eeschema windows.