

Generate PCB Layout using Pcbnew

At this point, it is crucial to update any changes that you have made into your schematic or footprint association (under Eeschema application) before proceeding to generating the PCB itself. To do so, click the *Generate Netlist* Icon on Eeschema application.

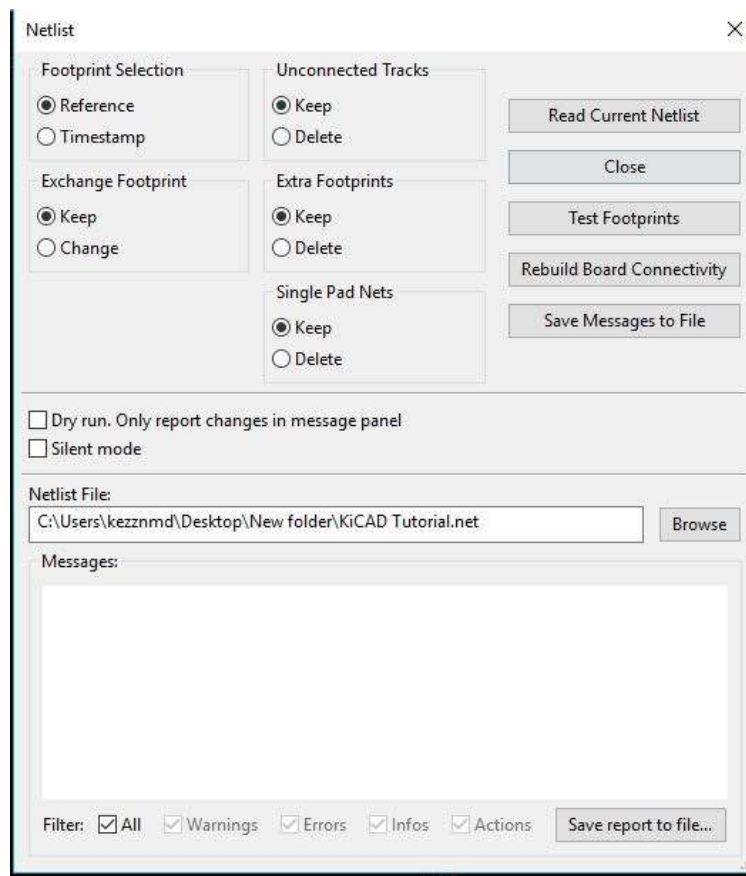
1. Load File to Pcbnew

On your KiCAD project menu, run the Pcbnew application by clicking the Pcbnew icon



2. Read Netlist

Here, KiCAD will read the Netlist that you have previously generated/updated under Eeschema. Click on the *Read Netlist icon* or go to “**Tools > Netlist**”



Click the **Browse** button and navigate to your working directories to locate your Netlist file with the **‘.net’** file extension, click **OK** and then click **‘Read Current Netlist’**. Click **Close**. Your components should now be generated.

Make sure the *Message Box* shows “Adding new components ‘..... (type of components associated).... “ as below ;

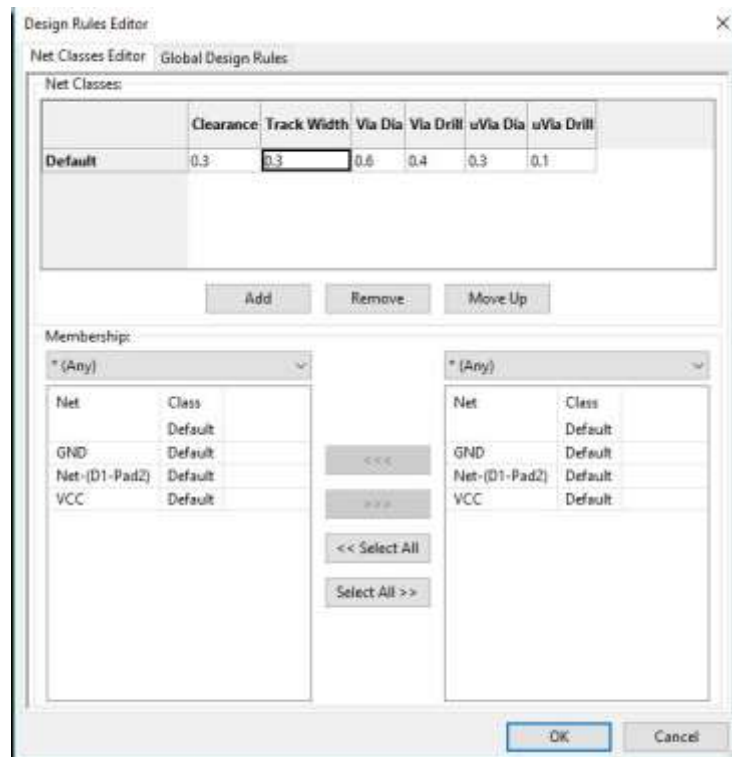


If you made any changes on the schematic (Eeschema) or footprint association (CvPCB) , update your netlist by clicking the Generate Netlist on Eeschema . This is so that KiCAD will update your components in Pcbnew by reading your Netlist file.

3. Setting Design Rules

Here, you can set your design rules such as minimum track width, clearance, via diameter etc. based on your PCB manufacturer. Go to “**Design Rules > Design Rules** “

On the *Net Classes Editor* tab, you specify the clearance, track width etc.



If you wanted to have a custom track width, you can set by click on the *Global Design Rules* tab and then specify your track width listed.

The image shows the 'Design Rules Editor' dialog box with the 'Global Design Rules' tab selected. The dialog is divided into several sections:

- Via Options:** Contains two sub-sections: 'Blind/buried Vias' with radio buttons for 'Do not allow blind/buried vias' (selected) and 'Allow blind/buried vias'; and 'Micro Vias' with radio buttons for 'Do not allow micro vias' (selected) and 'Allow micro vias'.
- Minimum Allowed Values:** A list of input fields for minimum values in mm: 'Min track width (mm): 0.2', 'Min via diameter (mm): 0.4', 'Min via drill dia (mm): 0.3', 'Min uvia diameter (mm): 0.2', and 'Min uvia drill dia (mm): 0.1'.
- Custom Via Sizes:** A table with columns 'Diameter' and 'Drill'. A note above the table states: 'Drill value: a blank or 0 => default Netclass value'. The table has 8 rows labeled 'Via 1' through 'Via 8'.
- Custom Track Widths:** A table with a single column 'Width'. It has 8 rows labeled 'Track 1' through 'Track 8'. The value '2' is entered in the 'Track 5' row.

At the bottom right of the dialog are 'OK' and 'Cancel' buttons.

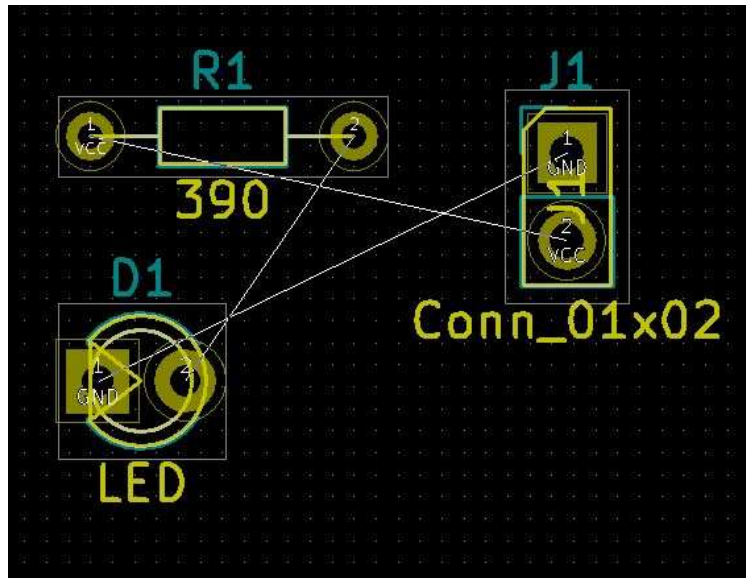
If you intended to fabricate your PCB here in The University of Nottingham, please refer to our **PCB Fabrication Request Specifications and Limitations** documentation from your moodle or sharepoint.

4. Moving / Editing Components

With the same interfaces as in Eeschema, hover your mouse on the components and click '**M**' to move, '**R**' to rotate etc.

5. Hide/ Show Component's Ratsnest

As shown below, the white line that are linked between the components are called Ratsnest. This is to indicate that the components are linked as in your schematic diagram that you have made previously.



Click on the *Show/Hide Ratsnest* icon to show or hide your ratsnest

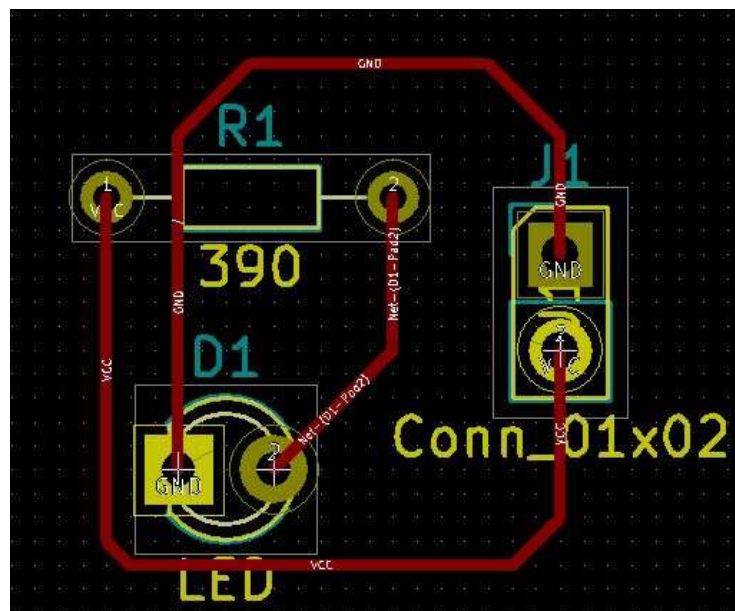


6. Add Tracks

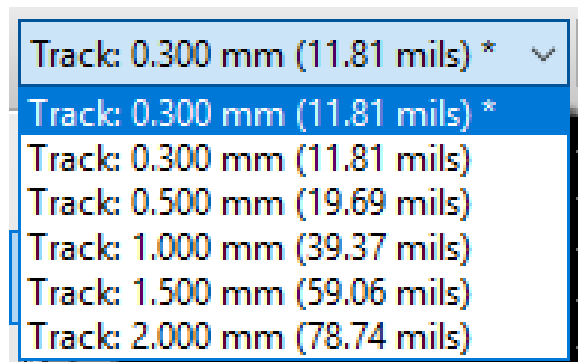
To start adding tracks, click on the *Add Tracks/Vias*




icon and then link the components. The ratsnest should disappear when a wire is link between components.

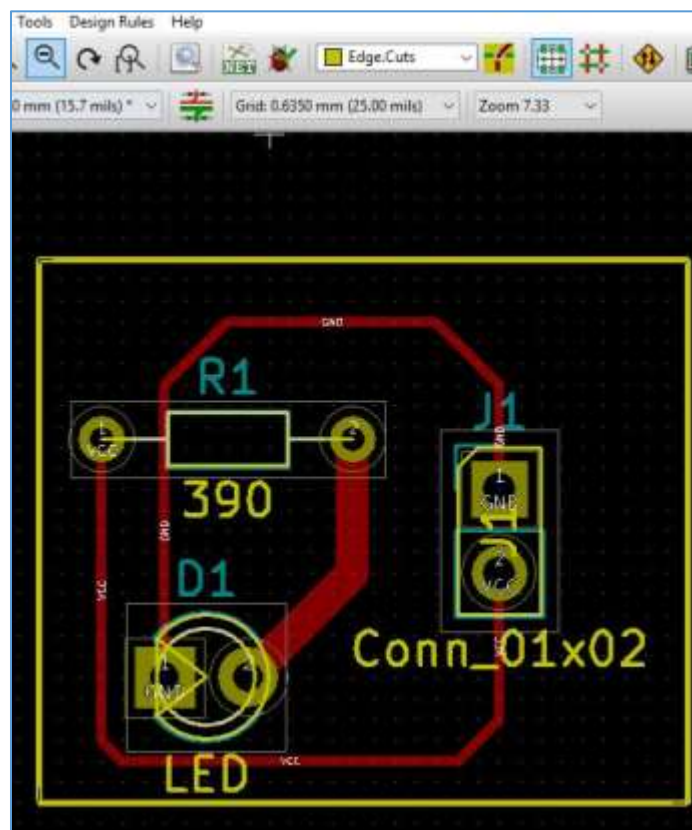


To change your track width, click on the drop down menu 'Track' and then choose your track width.



7. Set Board Edge Cuts

The Edge Cut are the outline of your PCB board that will be produced. To define your edge cuts, make sure you are working on the edge cut layer and click the *Add Graphic line or Polygon* icon  and start drawing the outline of your board.

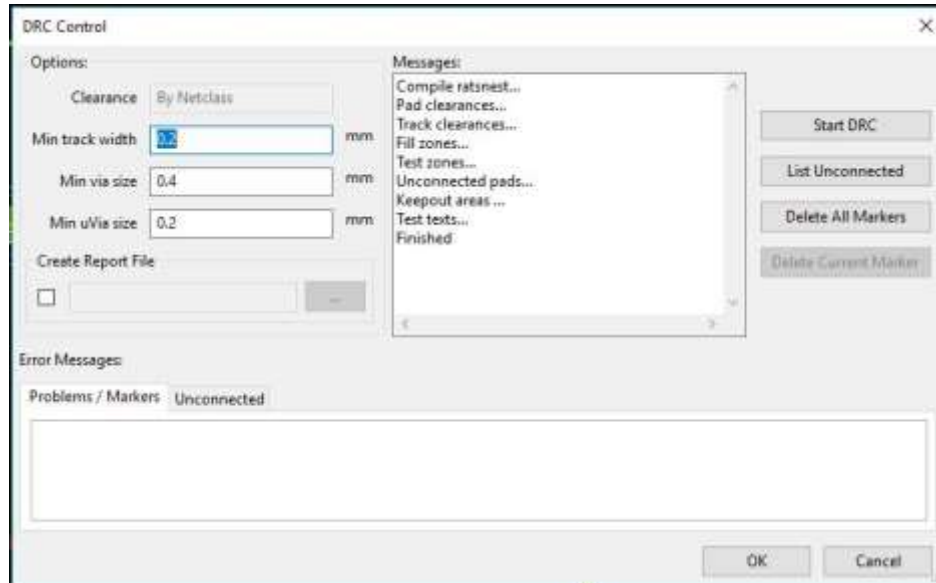


8. Perform Design Rules Check

This functions allows you to check your PCB design whether it follows the design rules that you have made or list out all the unconnected parts. To do so, click the *Design Rules Check* icon

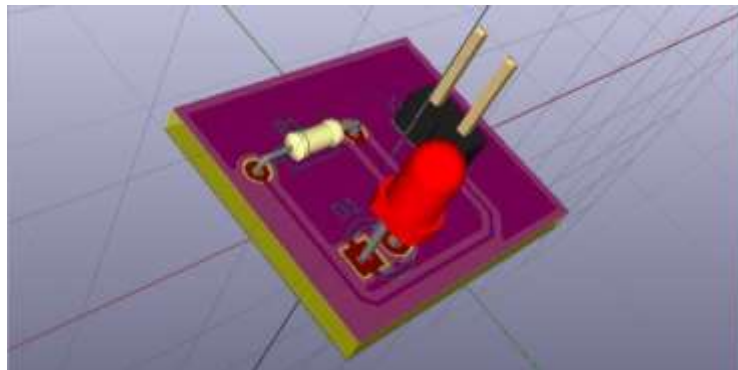


Click *Start DRC* and then make changes based on the Error Messages Listed.



9. View PCB in 3D

Go to “**View > 3D Viewer**” to view your PCB in 3D.



10. Generate Gerber Files

To send your PCB design to be fabricated, your PCB designed must be exported as Gerber file. Click “**File > Plot**” choose Gerber and browse your preferred directory to save the Gerber file.