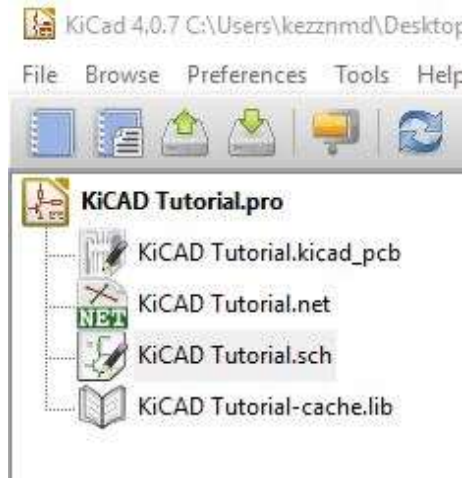


Associating Components With Footprints Using CvPCB (Eeschema)

1. Load Schematic File

KiCAD need to understand which footprint is associated to which components in your schematic diagram. The CvPCB which is under the Eeschema application will help you to do so.

First, open your schematic diagram by double-clicking your **“.sch”** file under your project tree.

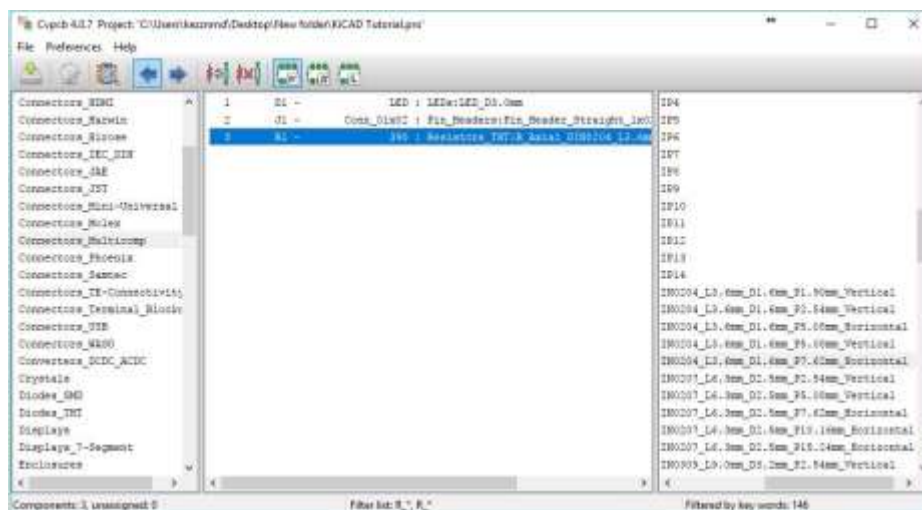


If you are currently viewing the Eeschema windows, you can run the CvPCB application by going to **“Tools > Assign Component to Footprint”** or click on the *Assign Component To Footprint* icon



2. Assign Footprint To Component

A window with a list enables you to choose the appropriate for each component. On the left side of the window is the list of components, the middle window is the list of components that you have placed on the schematic diagram while the right windows is the selection of footprint that is auto-listed by KiCAD. Double click on the list of footprint to assign it.



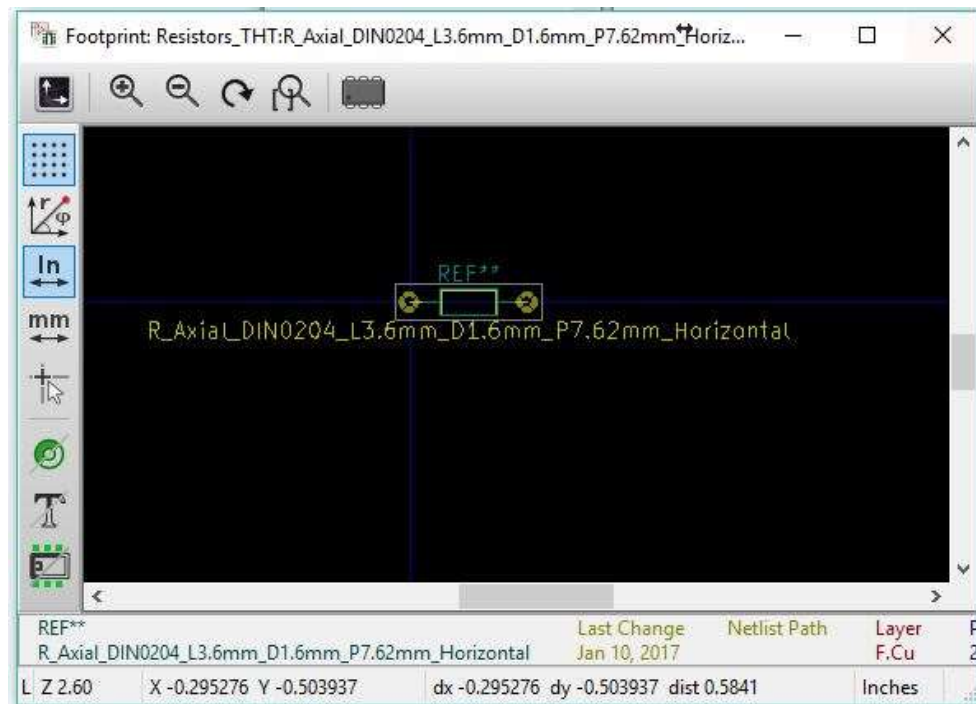
Its good to know the details of the components such as packaging type (e.g TSOP for Transistor, DIP/QFN/VQFN for Microcontroller, DIN for resistor) , pitch of footprint, orientation , as well as the type of soldering method (SMD or THT). This can be referred to the datasheet of the components under 'packaging type'.

3. View Footprint

Click on the *View Footprint* icon

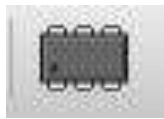


to view the actual footprint of your component. Here you can also measure the dimension for the footprint.

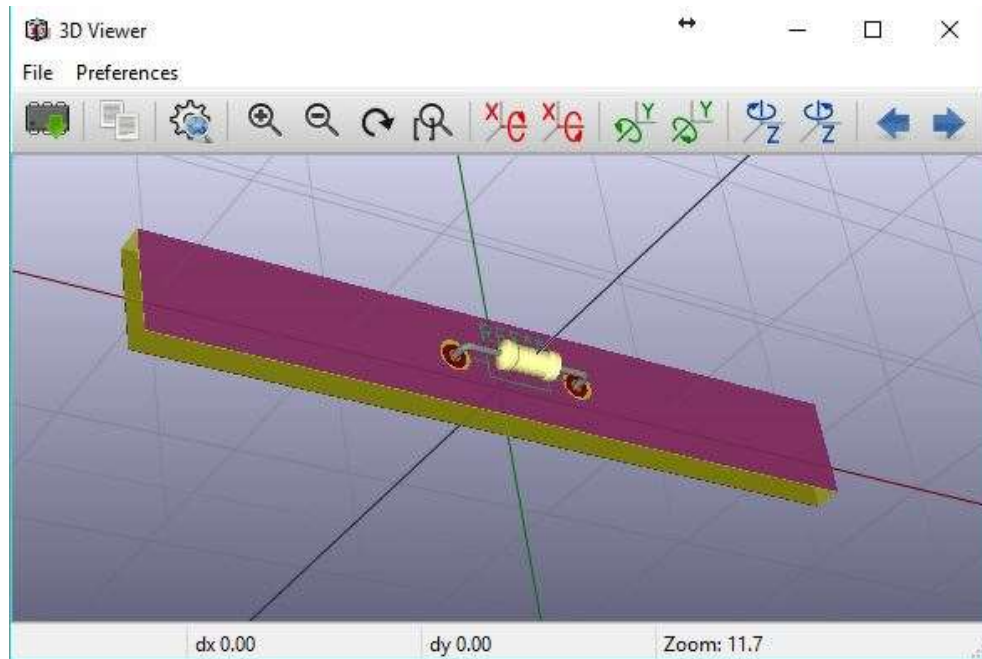


4. View component in 3D

Click the *View in 3D* icon



to view in 3D and you can orbit the component in 3D.



5. Save the Associated Footprint Selected.

Finally, you can save the selected footprints by clicking the Save icon



. Close the CvPCB

windows and click the Generate Netlist icon to update KiCAD.