

Creating Custom Component using Footprint Editor

Although component's footprint can be downloaded, sometimes you need to create a custom component just for your project e.g air core inductor. For this, you can create a custom footprint as well as importing 3D object so that you can view it in 3D.

1. Open PCB Footprint Editor

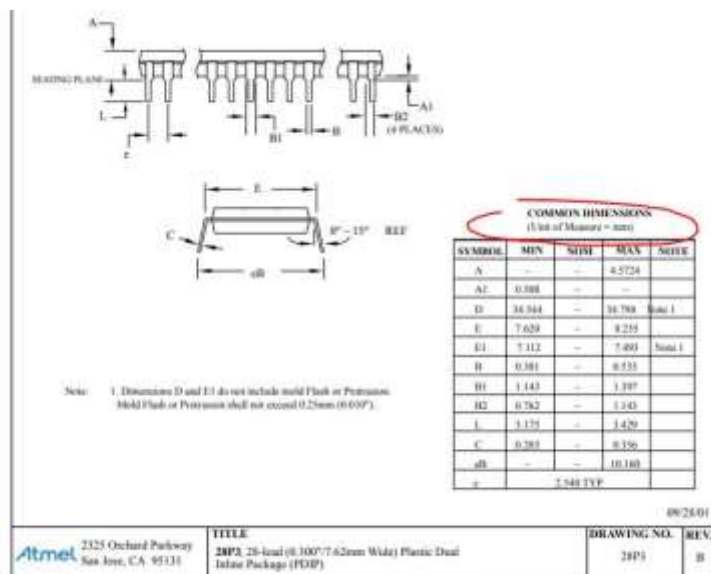
At the home page of KiCAD, double click on PCB Footprint Editor in Pcbnew, you can find the logo to open the editor.



or if you are currently

2. Set your Grid view

Depending on the size of your component, set your grid view by “**right-click > Grid Select**” and choose your preferred view. It is important to refer to the datasheet of your component as the unit will be stated there.



3. Create new component

Double click on the *new component* icon



and then name you component.

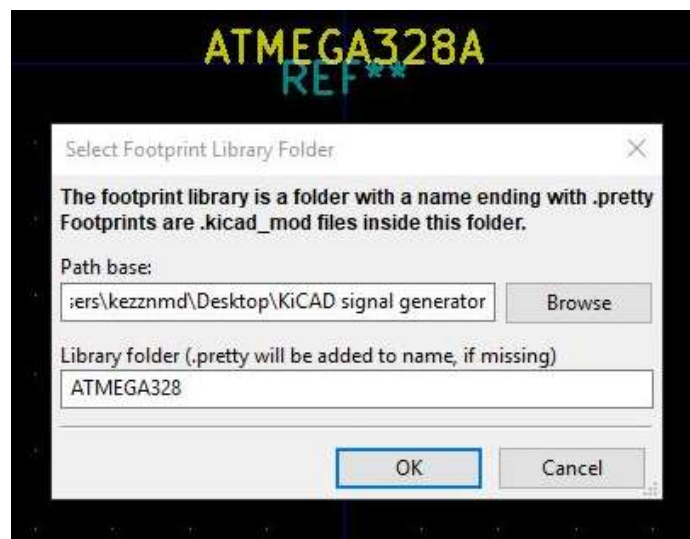
4. Create active library

Before you can start design a custom footprint, the library need to be set active or else, you cannot save the project.



If you already have a library for your project, skip this step! Else, you will clutter your project with multiple library

Click " **File > Save Foot print in New Library** to **File > Save Footprint in new library**", browse your project file, and name your library folder.



Note that a new file is created under your project file under ".pretty" extension and inside it is your component that you are trying to create.

To set your active library, go to " **Preferences > Footprint Libraries Wizard > Browse > locate your '.pretty' file that you have just created > Next > To the current project library**". You will notice that on the title menu will show that your library is active.

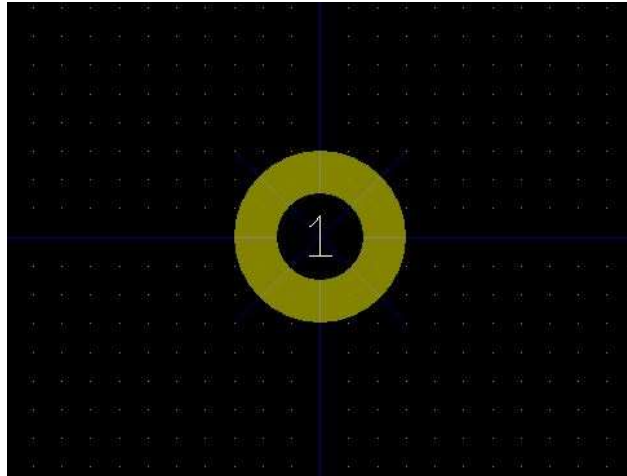
You can also choose "To global library configuration" if you wanted the same library to be visible when you are doing another project next time. To avoid hassle (in case the files are accidentally deleted or moved) you can opt to 'To the current project library' so that you will only use it only for this project

5. Create pads

Now you can start a pad where KiCAD will then associate the pad that you created with drill hole. Click on the *add pad* icon on your right

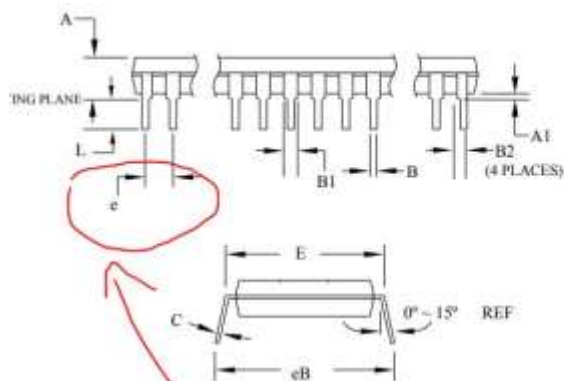


and place one at the center of the drawing.



Depending on your footprint, create another pad with the proper spacing. As for this example, you will create a pad for an ATMEGA328 chip with 28 pin. From the datasheet, it will mention the typical spacing for your reference.

Note that the spacing is 2.540 mm (100.00 mils) labelled “e” on datasheet. This is where the grid setting will help you to place the pad with the proper spacing. Refer step 2.



Note: 1. Dimensions D and E1 do not include mold flash or protrusion. Mold Flash or Protrusion shall not exceed 0.25mm (0.010”).

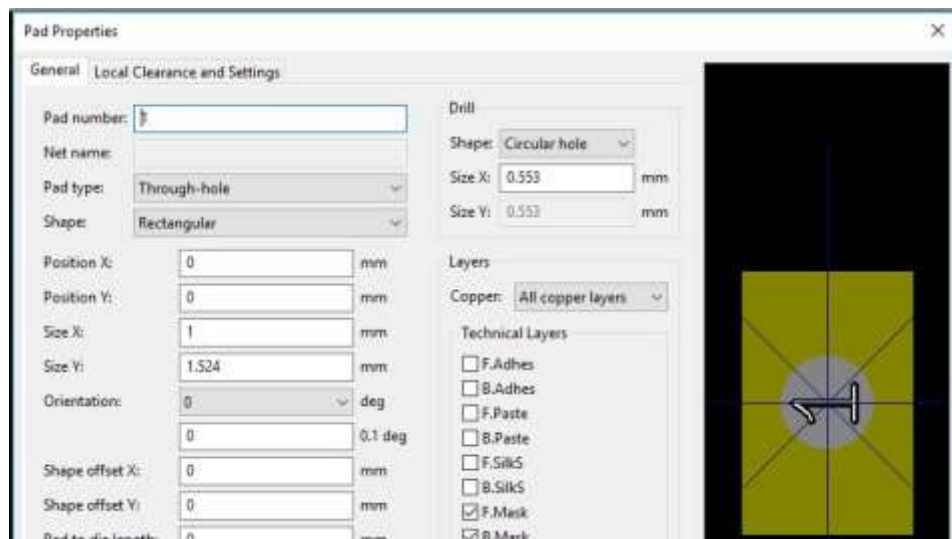
COMMON DIMENSIONS
(Unit of Measure = mm)

SYMBOL	MIN	NOM	MAX	NOTE
A	—	—	4.5724	
A1	0.508	—	—	
D	34.544	—	34.798	Note 1
E	7.620	—	8.255	
E1	7.112	—	7.493	Note 1
B	0.381	—	0.533	
B1	1.143	—	1.397	
B2	0.762	—	1.143	
L	3.175	—	3.429	
C	0.203	—	0.356	
eB	—	—	10.160	
e	—	2.540 TYP	—	

6. Pad settings

Now, you will edit the preference for your pads. Right-click on the pad and choose “edit pad”. From the new window, set your setting such as pad type, size of the pad (size X), and drill hole size (Drill). The drill hole size can be referred from the datasheet (labelled “B”) .

Change your pad number 1 shape to square shaped so that you now which is the pin no 1 when you are trying to solder it.

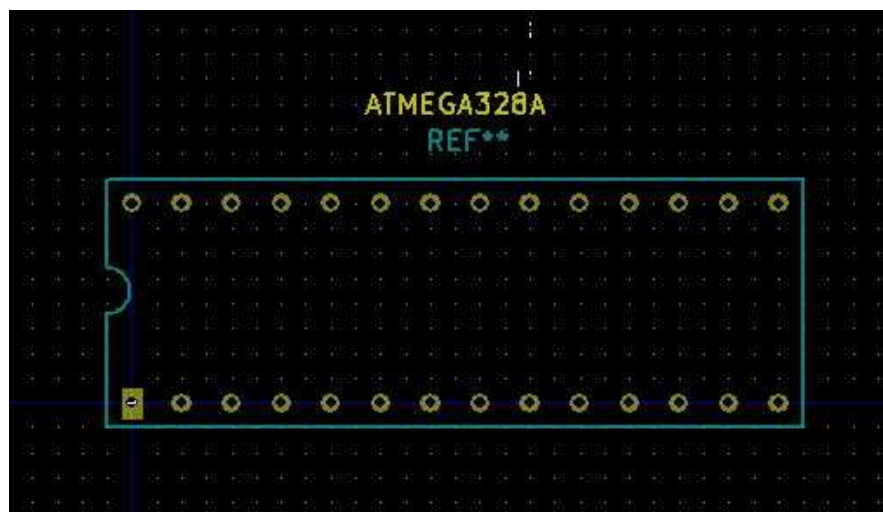


7. Create Layout

You can draw a layout by clicking on the “add graphic line” icon



and draw the outline of the component.



8. Save your custom footprint

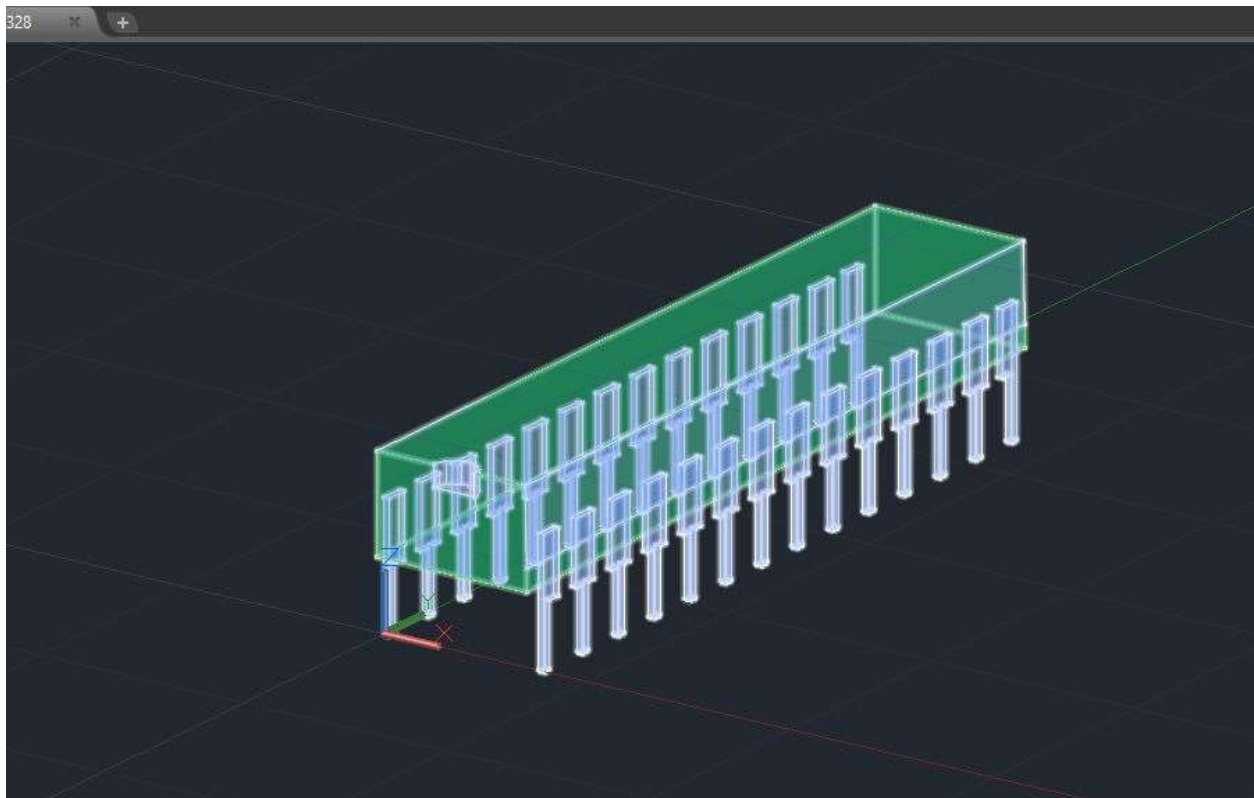
To save your footprint, go to **“File > Save Footprint in active library”** or click CTRL+S. Your custom footprint can now be use in Cvpcb (under Eeschema). Remember to update your Netlist on both Eeschema and Pcbnew.

-----Optional -----

Sometime you need to view your PCB in 3D so that each component will not overlap with each other. The custom footprint that you have created can be associated with a 3D model that you can design on your own.

9. Create your 3D component in CAD

Based on datasheet of your component, you can design your own component in 3D in any CAD software.

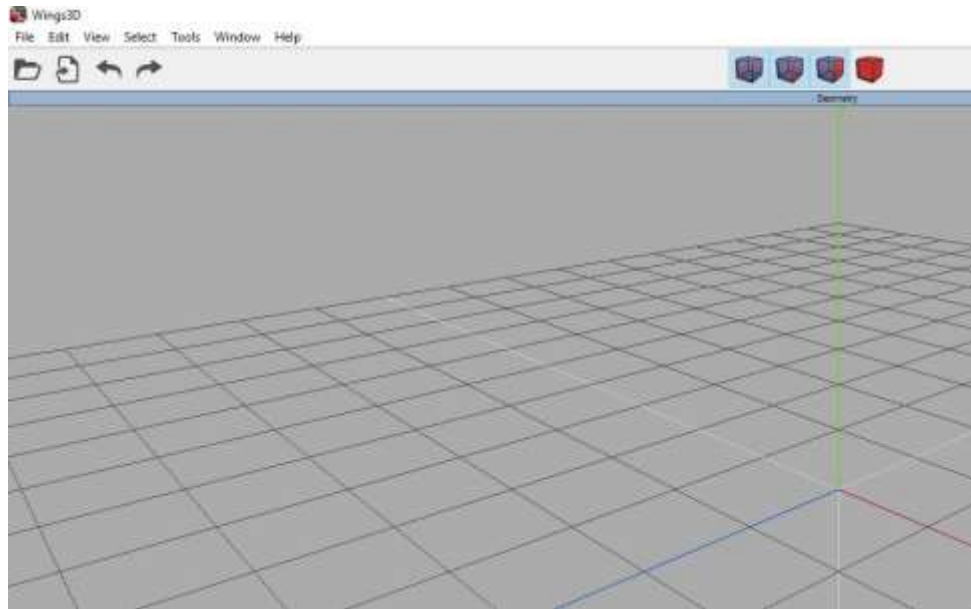


Note that KiCAD can only view a 3D model in '.wrl' file the next step will cover '.wrl' file conversion.

10. Convert your STL to WRL

KiCAD 3D viewer can only read a 3D model in '.wrl' file. You can export a 3D model from CAD software to '.stl' file and then use Wings3D to convert it to '.wrl'. http://www.wings3d.com/?page_id=84

From Wings3D import you '.stl' model. Click **"File > Import > Stereolithography .stl"** and locate your model.



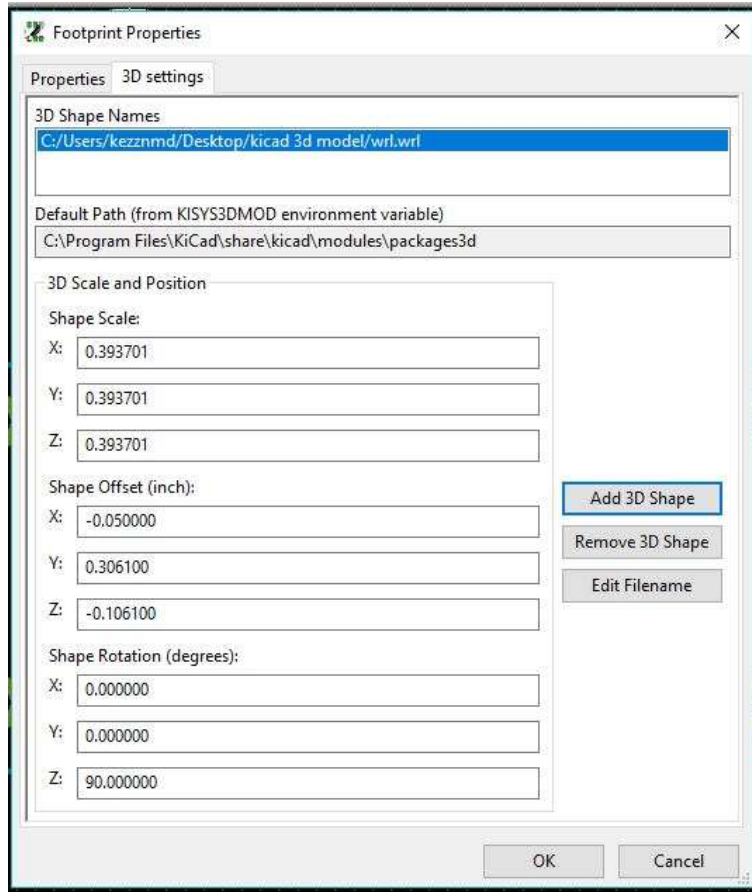
Export it to '.wrl' by going to **"File > Export > VRML 2.0 (.wrl)"** and then save it to your project file.

You can also set the color of your component and the orientation from Wings3D to make it more appealing

11. Associating your 3D model with KiCAD

From *Footprint Editor* you can associate your model by clicking on the *Footprint Properties* icon. Click **Add 3D shape** and then browse your model.





If your design is in mm, convert it to inch by adding 0.393701 at *Shape Scale* . Set the shape offset and shape orientation (this is based on where do you put the XYZ home UCS on your CAD software). Click 'OK' to proceed.

12. View in 3D

Click "View > 3D Viewer" or ALT+3.

