

## Creating Custom Schematic using Schematic Library Editor

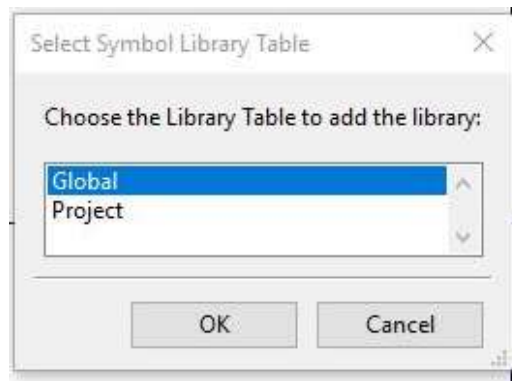
### 1. Open Schematic Library Editor

At the home page of KiCAD, double click on Schematic Library Editor or if you are currently in Eeschema, you can also find the icon to open the editor



### 2. Create a library folder

To make everything neat, you can create a folder to compile your custom schematic component inside your project file. Create a new library by going to “**File > New Library**”. Name your library and then save it into your preferred directory.



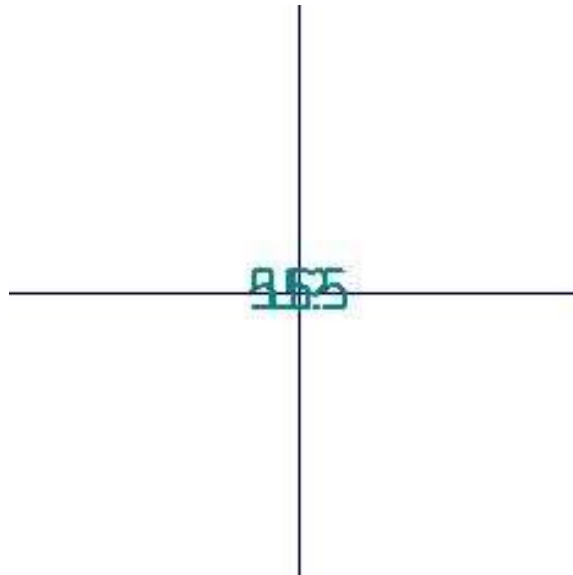
By choosing **Global**, your new library will always be visible whenever a new project is created. In contrast by choosing **Project**, your library will only be visible on your current project. Your new library should appear on the left panel list and '.lib' file will appear on the folder.

### 3. Create new component



Start by click the *add new component* icon an name your component, default reference designator ( e.g 'J' for connector , 'U' for microcontroller and etc.), and then click 'OK'

**Notice that the name of your component and reference designator will show up on your screen or you can zoom at the center of the drawing window.**

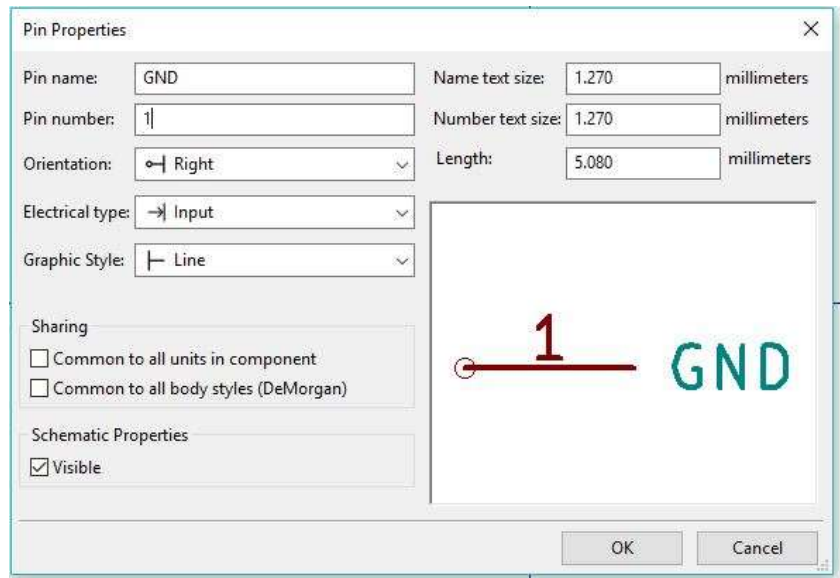


### 4. Set your Grid view

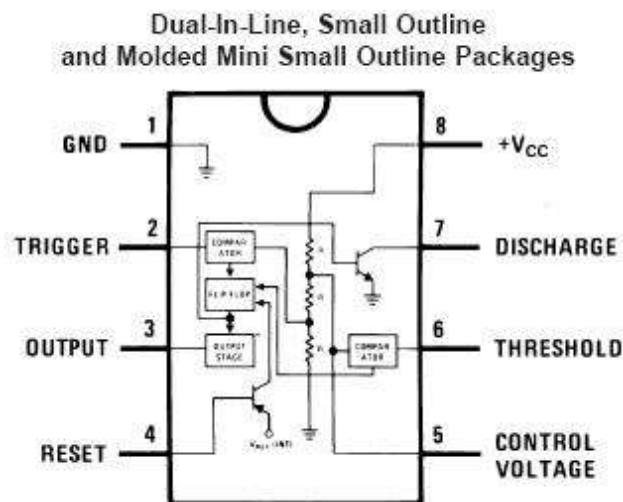
Depending on the size of your component, set your grid view by “ **Right-click > Grid Select**” and choose your preferred view

### 5. Adding pins

Click on the *add pins* icon or go to “**Place > Pin**” and start adding pin on your custom schematic by clicking anywhere on the screen. A new window name *Pin Properties* will appear, and fill the *pin name*, *pin number*, *orientation* and *etc.*



Based on the datasheet of your component, you should name the pin and designate the pin number accordingly. The name is optional but the pin number is very important as it will associate to what it will be then connected to.



## 6. Moving , Rotating Component , and editing properties

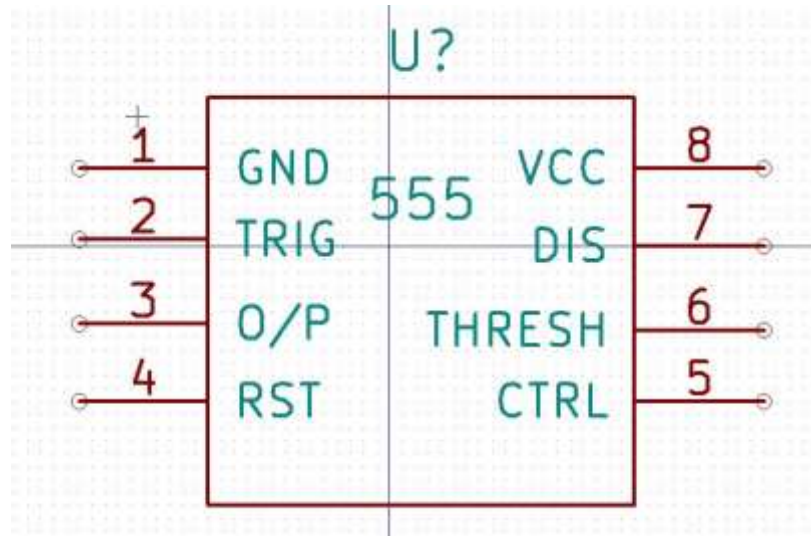
Hover your mouse on the pin, right click and then choose what command that you would like to do to it. You can also press shortcut 'M' to move, 'R' to rotate, 'E' to edit and etc.

## 7. Create an outline of the schematic

Define the outline of the component by click on the *Add Graphic Rectangle to Body* once on the screen, and then pull it to the other corner of the outline.



- . Click



## 8. Saving the component

To save, click on the **Save Current Component To New Body** icon



and and click OK.

## 9. Managing Footprint Library

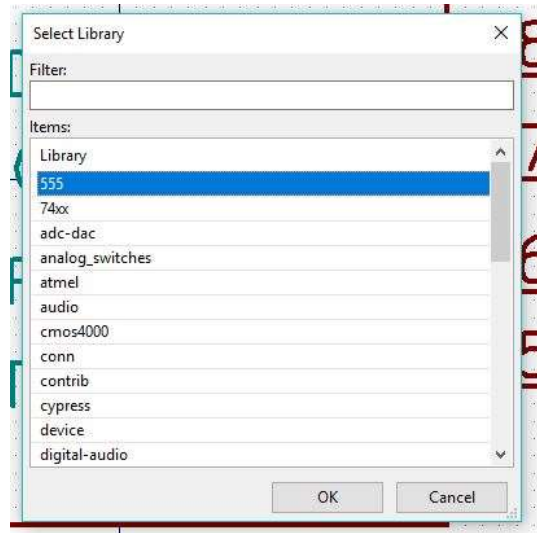
If you copy/migrate your library from other PC, you can simply copy the folder that contains the '.lib' file



Go to **“Preference > Manage Symbol Libraries > Browse Libraries > find your ‘.lib’ file > Next”**. Do the same for ‘User defined search path’. Click OK when you have done.

**Notice that your component will appear on the library file list.**

Click on the *Select Working Library* icon and find your newly defined library path.



**Notice that on the window title, the defined library will be mentioned.**

Save all your library by click on the Save icon