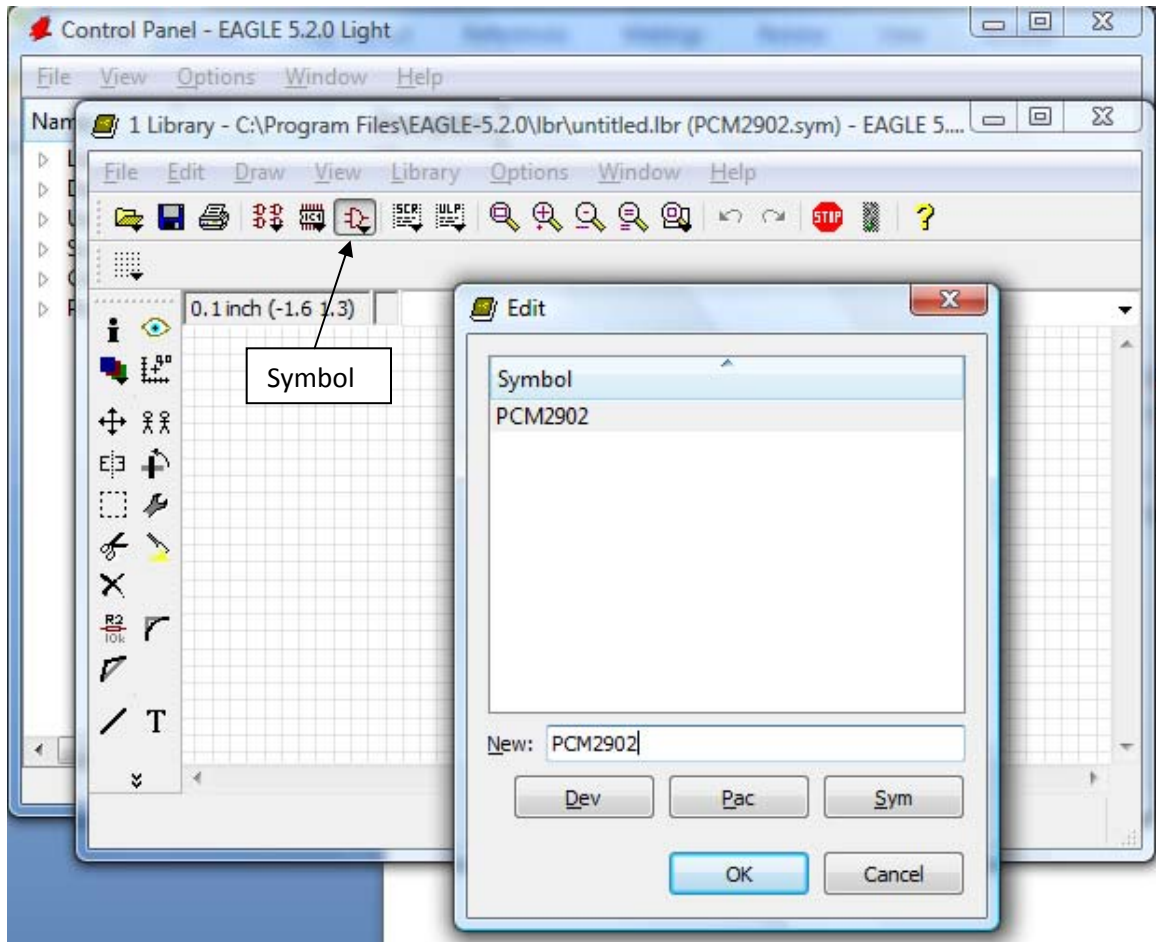


Cadsoft Eagle Tutorial: Creating New Parts

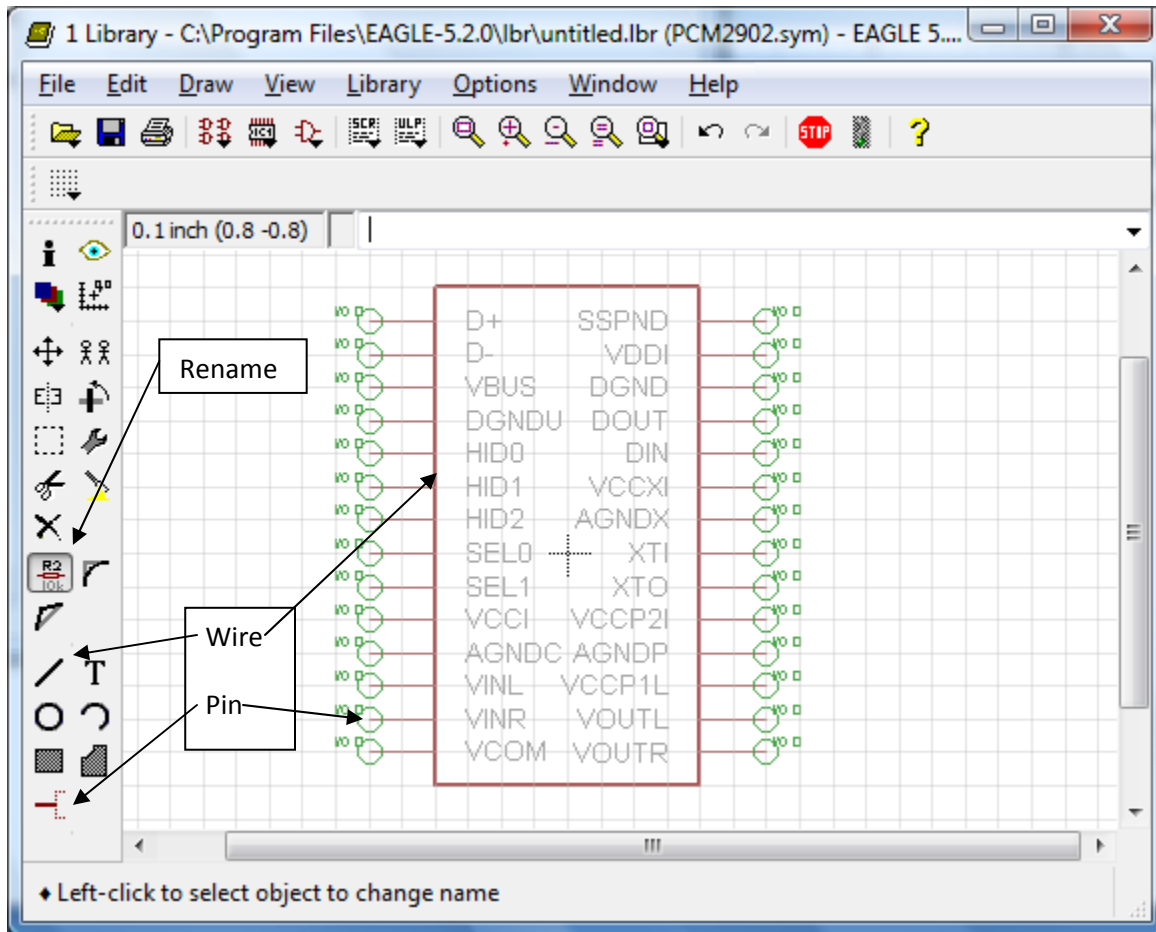
This guide will explain how to create a new part in Cadsoft's Eagle Layout Editor. This tutorial is based on v.5.2.0 but should still be valid for previous versions of the software.

To start, Open or create a new library file. [File -> Open (New) -> Library]

Click on **Symbol** (6th icon on top), enter a name in the "New" field and click **OK**.



You are then presented with a blank area to draw your new schematic symbol for the part, Use the tools on the left. The two most important tools are the **Wire** and **Pin** tools, used to draw the outline of the part and place the pins used to wire to it in the schematic, respectively. To change the pin direction, you can use the appropriate setting at the top of the toolbar. Once complete, rename the pins to something useful with the **Rename** tool. You should end up with something looking like this.

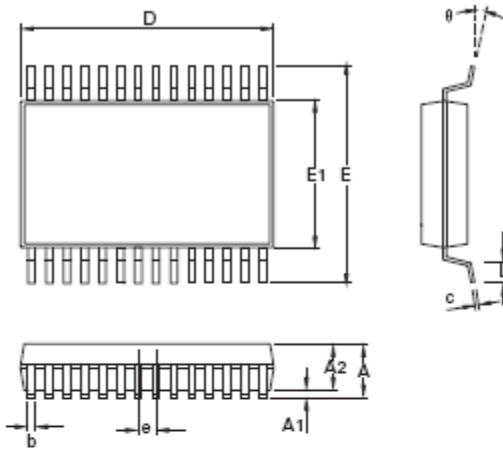


The next step is to build a footprint for the device. If your device uses a common footprint, you can open another library containing the desired footprint. Use the **Group** tool to select the entire footprint. Select the **Copy** tool and Ctrl+right click to copy the group. Open your library file again, open the **Package** box, select the appropriate package then use the **Paste** tool to paste the footprint into the window. Even if an existing footprint does not match your part, but has the same types of pads or holes that you need for your footprint, then copying that footprint as a starting point is recommended. You can always delete extra holes/pads that are not needed, or copy existing holes/pads to make additional ones that your footprint may need. This also copies the 'NAME' and 'VALUE' for the part so that you do not have to add them later on.

Alternatively, if your package uses a unique footprint, you will have to create your own footprint. Usually the dimensions will be given in the datasheet for the part.

Surface mounted, 28 pin package

Package outline



Dim.	Millimeters		Inches		Dim.	Millimeters		Inches	
	Min.	Max.	Min.	Max.		Min.	Max.	Max.	Max.
A	1.70	2.00	0.067	0.079	L	0.55	0.95	0.022	0.037
A1	0.05	0.15	0.002	0.006	e	0.65 BSC		0.026 BSC	
A2	1.65	1.85	0.065	0.073	b	0.22	0.38	0.009	0.015
D	9.90	10.50	0.390	0.413	c	0.09	0.25	0.004	0.010
E	7.40	8.20	0.291	0.323	Θ	0°	8°	0°	8°
E1	5.00	5.60	0.197	0.220	-	-	-	-	-

Note: Controlling dimensions are in millimeters. Approximate dimensions are provided in inches

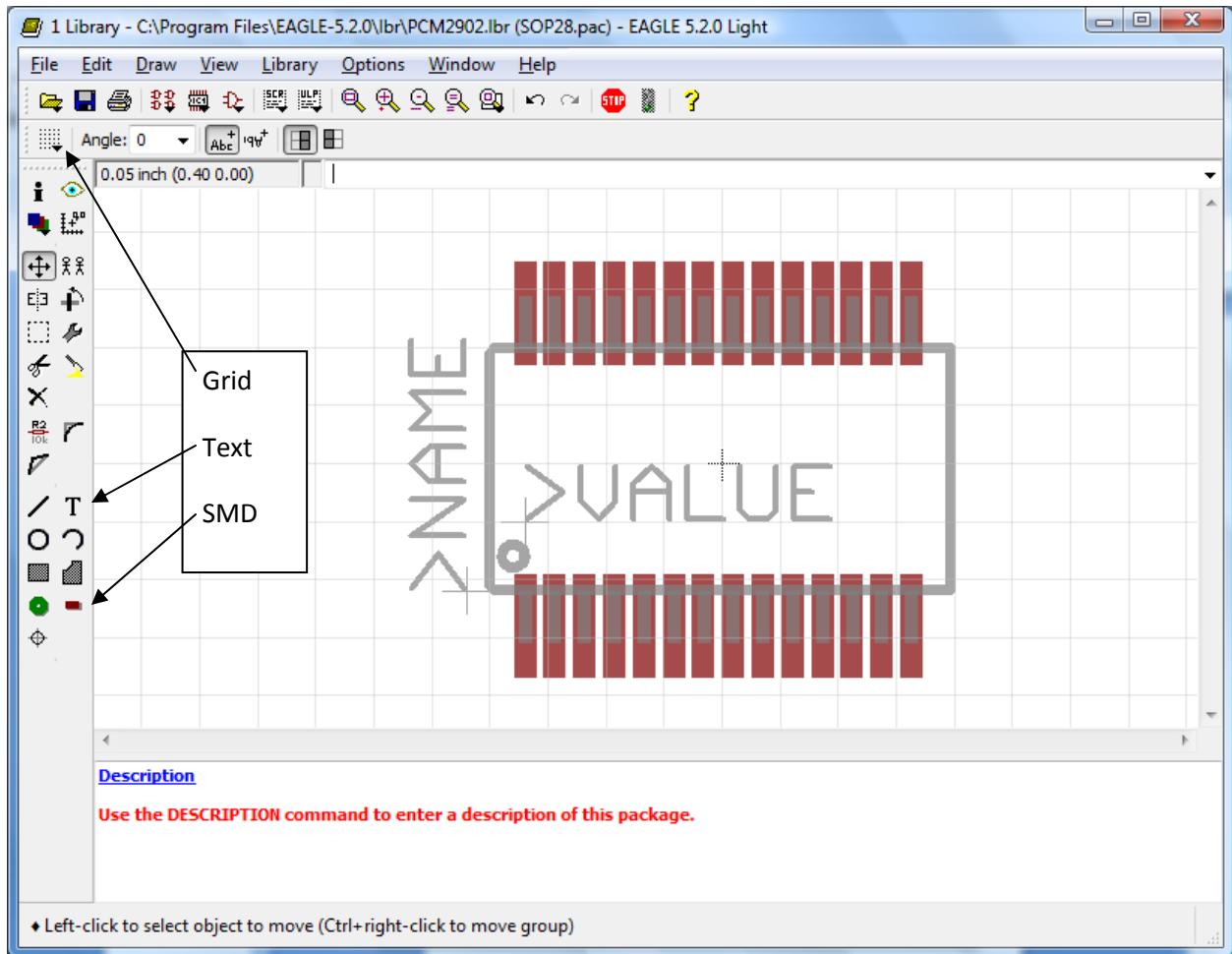
Soldering footprint



Use the **Wire** tool and either the **PAD** or **SMD** tool to create the package and footprint. Use the wire tool to outline the shape of the part and the Pad or Smd tool to make the footprint. Be sure you have all the dimensions of the part correct in your package because that is what will come back on your PCB.

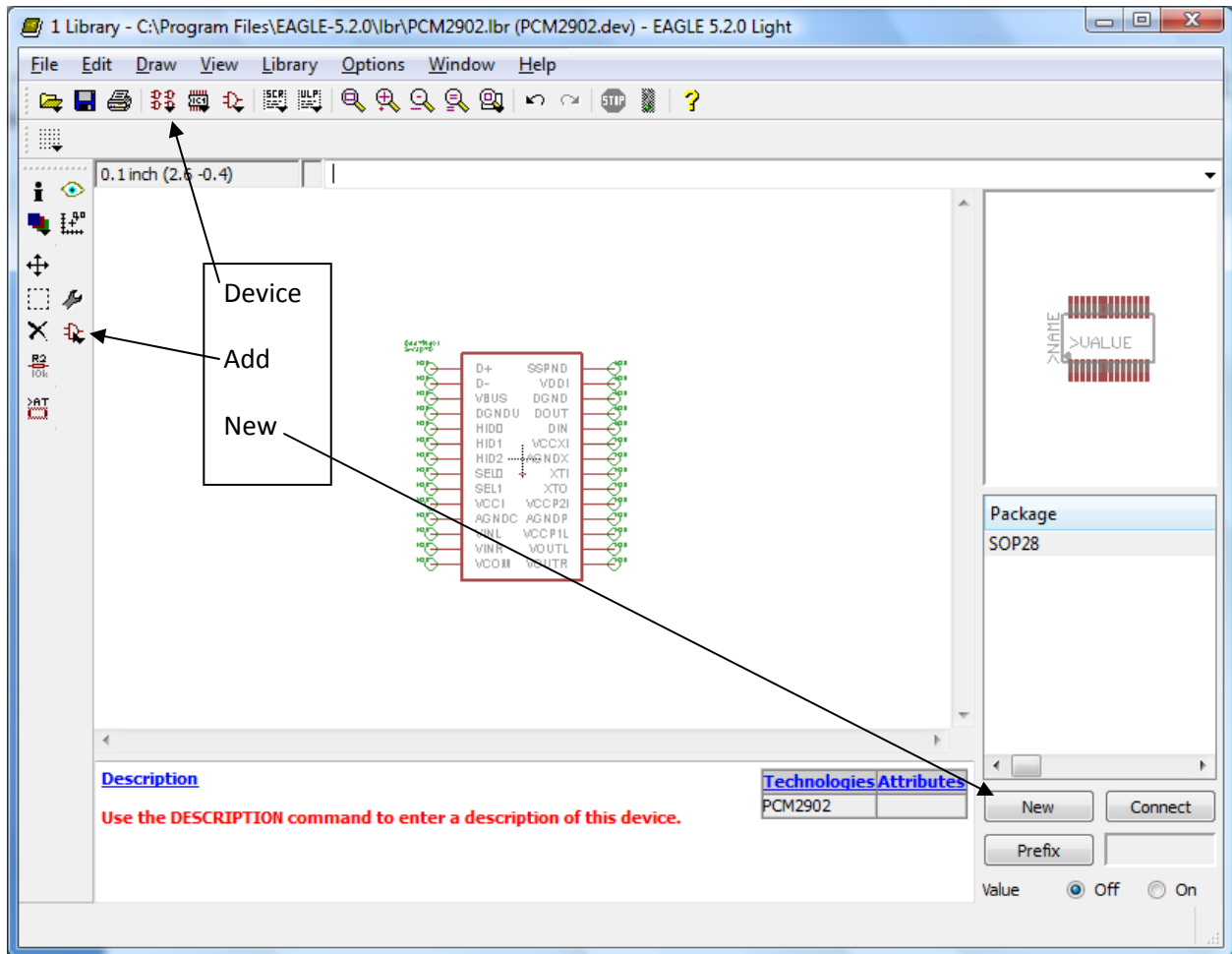
When laying out your holes/pads, choose some reference point to start from, and place everything relative to that reference point. Common choices are the center of pad/hole #1, the center of the package, etc.

Note: It may help to change the grid size for more accurate placing of the parts.



You should end up with something like this. When finished, use the **TEXT** tool to add labels to your part. One should read ">NAME" (layer *tnames*) and the other ">VALUE" (layer *tvalues*) which will be used later on. You may want to change the font to **VECTOR** as the PROPORTIONAL font may not be readable on PCBs (RBR: I have never found a case where the proportional font was not readable). Any additional text you add to the footprint should be on the tDocu layer.

Now the only thing left to do is to combine the symbol and package that you just created together. This is done in the **Device** window (located to the left of the **Package** icon). Type the name of your new device in this window and click **OK**. In the new window that pops up, click the **ADD** tool and select your device in the window, then place it on the screen. In the bottom right corner, click **NEW** and select your package. You should now see your part's symbol and package on the screen.



Next, click on the **Connect** button and you will see a listing of the Pin names and Pad names. Here, select the corresponding Pin and Pad and click **Connect**. Once again, make sure you are connecting the right Pin name with the right Pad name.

Now you are ready to use your part in your schematics.