Creating a new component in EAGLE

Tim Steffes
June '10

Now that you are familiar with designing a PCB in EAGLE, you should learn one of the most powerful tools the software provides: the ability to create a new component. Can you imagine going through the design process of your schematic only to realize that you cannot find the last component you want to add and have your design consequently become useless? Thankfully you have the option to create your own custom part and add it to EAGLE's libraries!

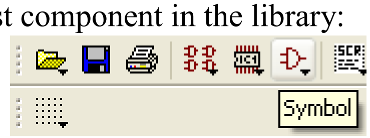
Just as with creating an Eagle project, creating a new component will require the creation of a schematic to represent connections and then a board piece for the physical layout. This guide will show you the process of creating a new component, referring to the component's datasheet for specifications. In this case, the component will be a simple dipswitch that the EAGLE library seems to lack. The datasheet can be found at http://spec.e-switch.com/W/W823121C.pdf.

Create the library

Go to the EAGLE control panel and choose File→New→Library. The base library window will open, which doesn't really do anything useful for our purposes. Start by saving the library to the EAGLE library directory, and call it "dip-switch".

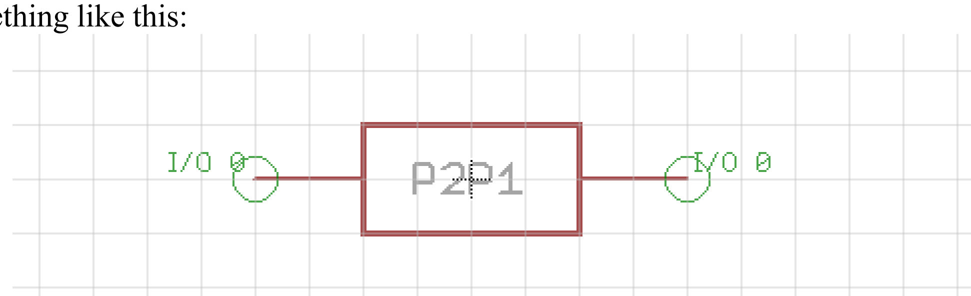
Create the schematic symbol

Now, create a symbol for your first component in the library:



Create this component symbol by its name - KAJ01LAGT. Now, simply draw a schematic symbol that you wish to use to represent the switch. This will be a very simple symbol, as it only has two connections and acts as a basic switch. You can create the schematic symbol to be a simple rectangle with a pin on either end. Start by using the Wire tool on the symbols layer and creating the box that will show up in your design schematic when you use the symbol. Keep in mind that the size you create will be the same when you use the symbol in a design, and that the middle crosshairs will represent the origin of the symbol when you need to access it in a design.

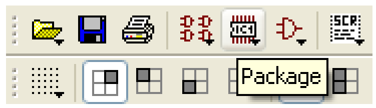
Add the two pin connections to either side of your box using the Pin tool. Remember, you can rotate the pin before placing it using by right-clicking. Then use the name tool to give each pin a name. Since this is a very simple switch, descriptive names aren't necessary. You should end up with something like this:



It is also good to allow the user of the device to change the name. To do this, add text around the box to signify where a name can be specified. Using the Text tool, write “>NAME”, and switch to the Names layer. Place the text near your box. You can do the same with a value on the Value layer, but this is not necessary for this component as it would be for something like a resistor.

Create the footprint

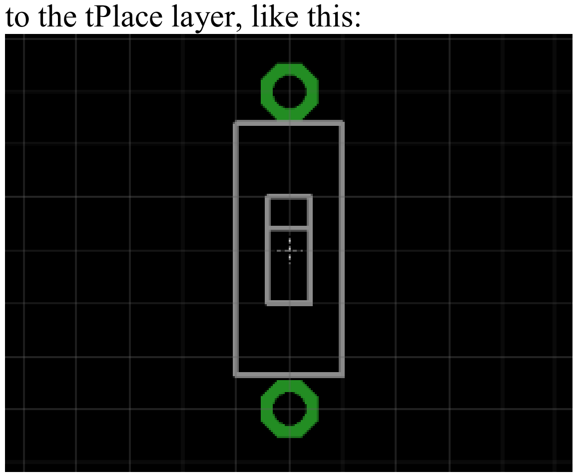
Now it is time to create the corresponding footprint. This is where the drawings in the datasheet will come in handy. We want to match the footprint as best as we can for the outline, and it is imperative that we match the soldering holes to have the correct spacing and size. Click the Package icon to create the footprint.



Give it the same name as for the symbol file. Now, we need to make the footprint to match the datasheet. Your cursor’s position is displayed in the upper-left of the sheet, which will be very helpful for this.

Remember that by holding Alt, we can get much more precision depending on the settings of the Grid tool. Since the datasheet has specifications to .001 inches, let’s change our Grid tool’s “alt” setting to match. Now go back to the schematic and start with an outline of the device. We can see that the rectangular size is .1”x.236”. So let’s make a box at all four corners of ±.05” and ±.118” using the Wire tool on the tPlace layer – the one that is used to make the silkscreen.

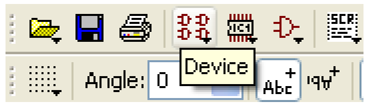
Next, we can add the through-holes or “pads”. On the upper-left of the datasheet, a recommended “PC layout” drawing is given, stating the holes should be .031” diameter and .3” apart, which makes our job very simple. Use the Pads tool and switch to an octagonal pad. Place them at (0, .15) and (0,-.15). Now you have a usable footprint. It may be helpful, though, to make it more descriptive to look at. We could add a little more graphically to show that it is a dipswitch. Let’s add more to the tPlace layer, like this:



Now there is just one thing left to do.

Create the device

We have a schematic and board symbol for our switch, but now we need to tie the two together to create the device.



Again, use the same name. This part of the design is simple – just tell the tool which pins go with which pads. Use the Add tool to add your symbol to the center of the sheet. Then hit the New button on the bottom-right and add the package that you just created. Finally, hit the Connect button. Highlight the pairs of pins and pads that you want to connect and hit the Connect button until every connection is made and hit OK. Save your device, and you are done! You can now pull up this library in the EAGLE control panel just like all the other libraries, and use your part in your design.