

Application Note:

Creating a Model of a Component within the Eagle Cadence Software

Andrew Warner
April 5, 2013

Keywords:

Eagle	Modeling	Components	Library
Package	Symbol	Device	Schematic
Layout	PCB	Tutorial	Cadence
CAD			

Executive Summary:

This document acts as a tutorial on how to create the model of a component within the cadence program Eagle. The tutorial will walk through the basics of creating the three parts needed for a complete model: the device, the package, and the symbol. With each step in the process, an example will be accompanied creating an arbitrary switch. The document starts at opening the software, creating the three parts of the component, and ends with the completion of the component.

Introduction

In the process of creating schematics and printed circuit boards there is often the need to add a component that is not available within the current library of parts amassed for use. The component can easily be created to be used in current projects and future ones. This application note will highlight the how to create a component in the Eagle cadence software and will show a step by step pattern for create parts. While showing how to create a part, an arbitrary switch will be created to show how to use the software.

Objective

The objective of this application note is to teach how to make a component by walking through the process of creating a component within Eagle while demonstrating the process through the creation of a simple switch.

Process

There are three steps to modeling the new component once the library has been opened. These steps are creating the device, the package, then the symbol.

Opening the Eagle Library

Open the initial Eagle User Interface. Go to the folder the project is stored in or will be stored in. With your mouse, right click on the active folder and select New>Library as shown in Figure 1

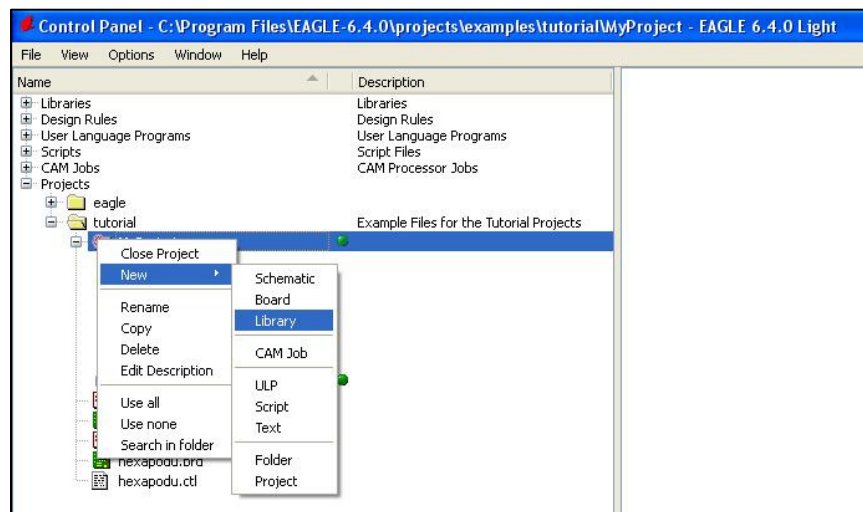


Figure 1. Opening the Eagle Library

Once the library is open it should look as seen in Figure 2.

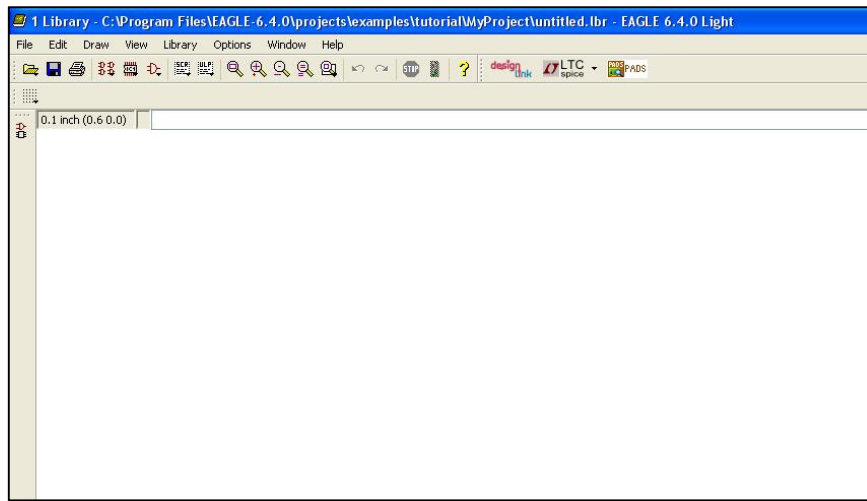


Figure 2. Library Main Screen

The next step will be to start creating the package of the new component.

Creating a New Package

The package of the new component is what will be placed on the printed circuit board (PCB) and includes holes that need to be drilled, pads for soldering, and silkscreen marks to label the component. From the initial library screen in Figure 2, click the package button located on the toolbar (circled in Figure 3).

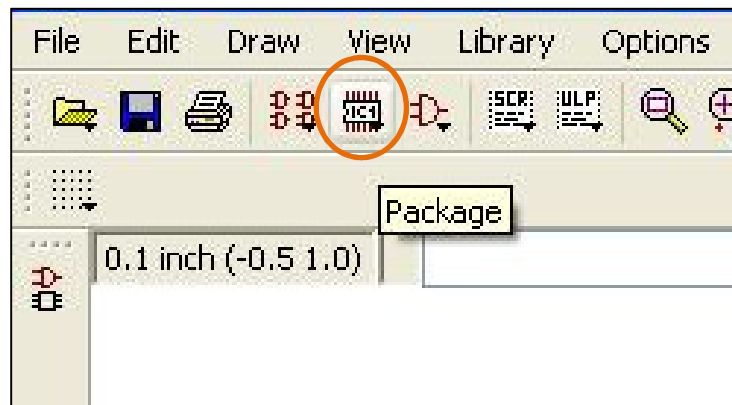


Figure 3. Select Package

Once selected, a window will open. This window determines the name of the package you will create. Since this part is assumed new, or not in any library, type in the name given to the part. Then, select "Ok." Another pop-up will appear saying, "Warning: Create new package '[insert name]'?" Select Yes, and the package making process will begin. Figure 4 shows the initial window with the example name of the arbitrary switch example while Figure 5 shows the warning. Figure 6 shows the package editing screen.

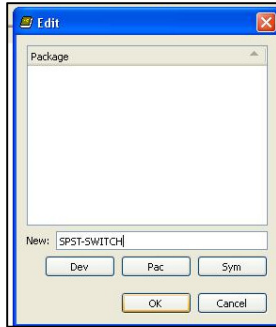


**Figure 4.
New Package
Window**



**Figure 5.
Creating New
Package Warning**

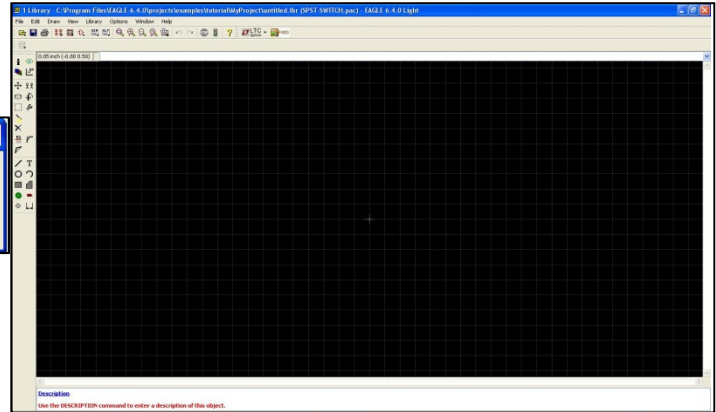


Figure 6. Package Editor

Now that the package editing page is open, the actual modeling can begin. Any order of modeling can be used, but this tutorial will work from mounting the component to outlining the component and finishing with text. A helpful tip is to set the grid to the required spacing of the part's mounting style, be it surface mount or through hole. This is done by selecting the grid button and changing the size to the necessary conditions. This can be observed in Figure 7.

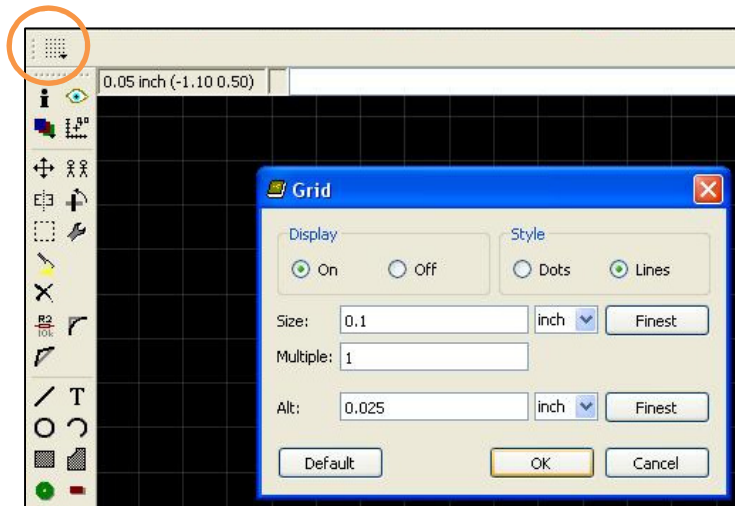


Figure 7. Grid Button and Grid Editor

Next, pads, and through holes, can be placed on the board. The following demonstrate how to place pads and vias for the switch. For the example, the pads are shown in Figure 8 and through holes in Figure 9.

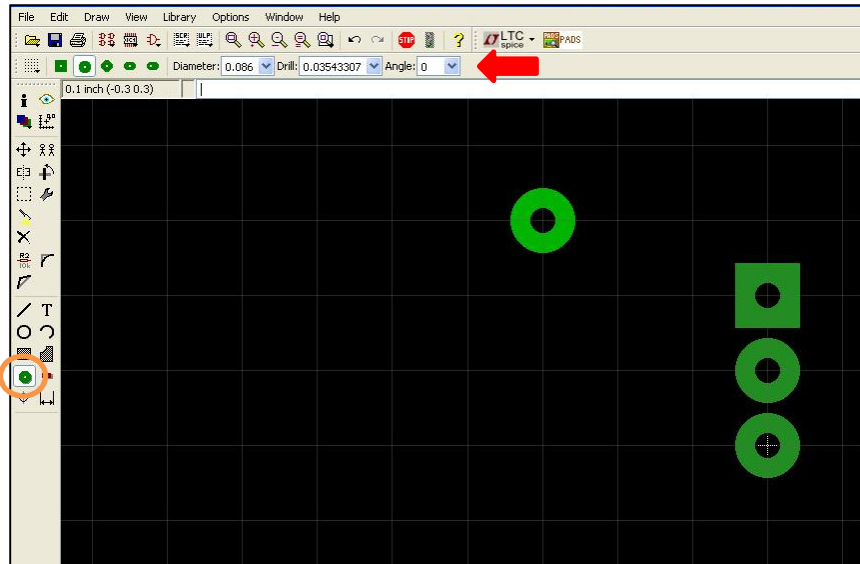


Figure 8. Placing Pads

To place a pad, select the pad button (circled in Figure 8).

The parameters of the pad (size, shape, diameter, hole size, and angle) can be customized in the toolbar along the top of the screen (shown with an arrow in Figure 8).

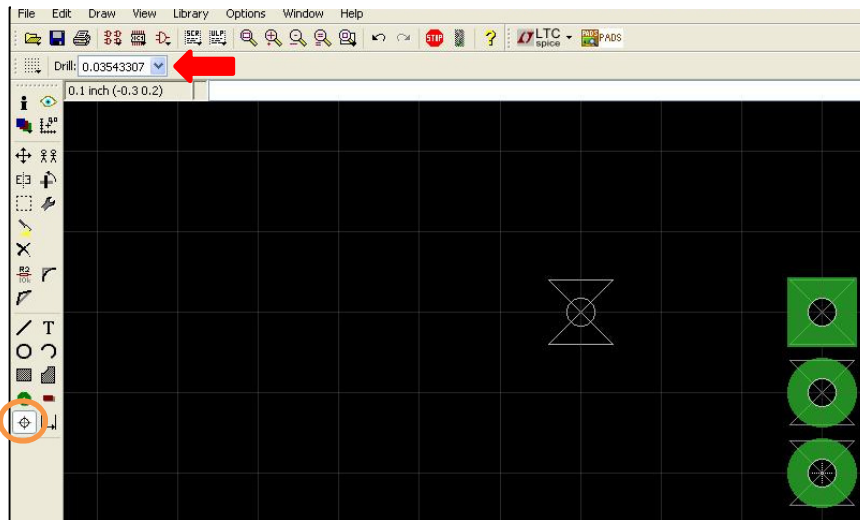


Figure 9. Placing Through Holes

Place a through hole by selecting the associated button (circled in Figure 9).

The hole size to be drilled can be set similar to the pad's hole size (seen with the arrow in Figure 9).

Now that the pads and through holes are placed, the new part's outline can be created. This is done by assembling different shapes such as lines, rectangles, circles, and polygons. The example switch needs only an outline can and the example outline can be seen in Figure 10. This was created by attaching four straight lines together to make the rectangle.

The next task is to add text. The button marked "T" allows for this. Select the text button, type in the associated text, and select "Ok." The process for creating text and inserting text can be seen in Figure 11 and Figure 12.

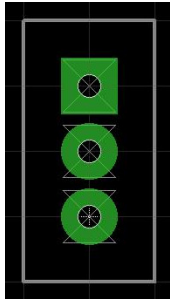


**Figure 10.
Switch
Outline**

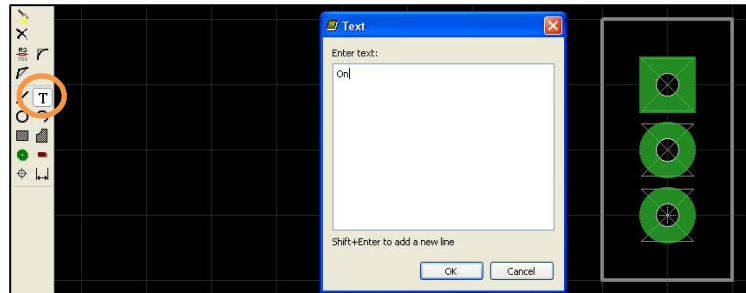


Figure 11. Inserting Text

For the switch, an “On” text needs to be added and is shown in Figure 11 along with the text button being circled. The placement of the switch’s necessary text can be seen in Figure 12.

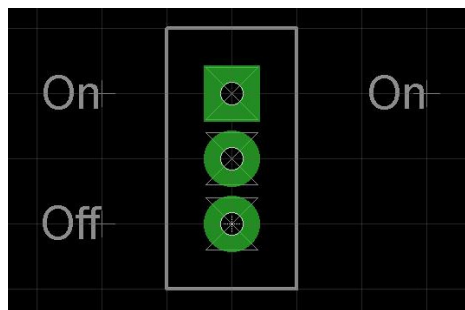


Figure 12. Completed Text

After completing the text of the package, the package could be used as a finished product, but there are several ways that would make the current model better. For instance, naming each pad that was placed can making connecting them to pins later much easier. By simply selecting the “Name” button (circled in Figure 13), clicking on the pad, and changing the name, the pad can be specifically named to whatever is necessary as shown in Figure 13.

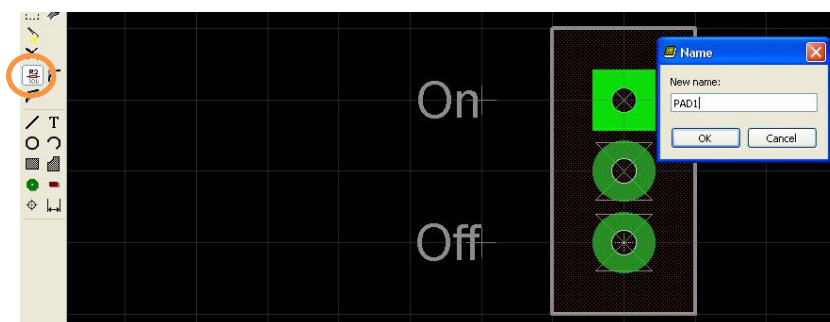


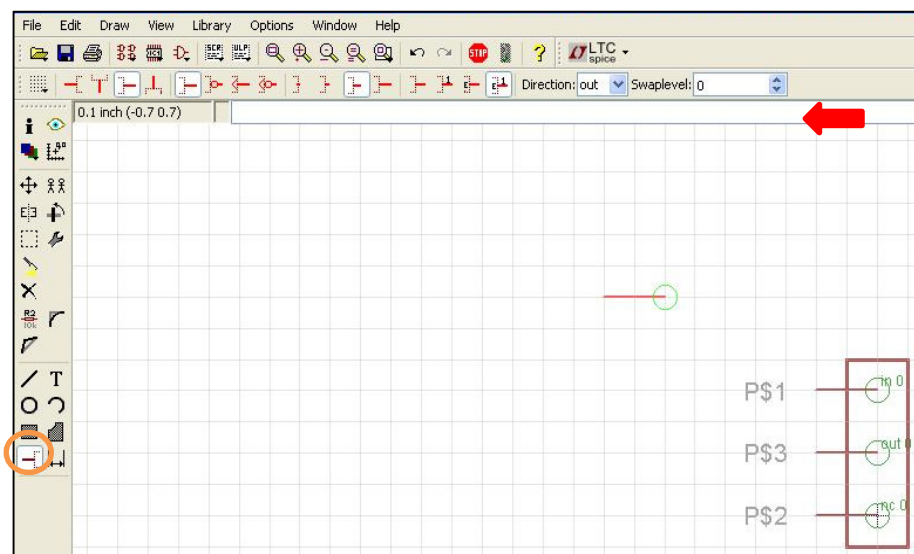
Figure 13. Naming Component Pads

Creating a New Symbol

After completing the new package, a new symbol must be created. This process is very similar to the process of creating the package, but since the symbol is only for schematic purposes, the symbol can be modeled however it is deemed necessary.

In order to open the symbol editor, go to the toolbar at the top of the screen and select the symbol button (). Once clicked, the same window that appeared for the package will appear. Type in a name for the symbol, hit “Ok”, and select “Yes” when the warning occurs (see Figure 4 and Figure 5 for more information).

Once the symbol is created, modeling it can begin. Using the same type of method for creating the package, create the model while thinking how it will be seen on a schematic. Once the initial enclosure is completed, select pins to use as connection points (the pins will be connected to the pads in the package once the device is created). The process of selecting and placing pins can be seen in



The pin button is shown circled in Figure 14. The pin parameters that can be changed (length, symbol, direction [input, output, etc], and level) with the associated toolbar indicated with an arrow within Figure 14.

Figure 14. Adding Pins to the Symbol

A note about adding pins: the green circle associated with the end of the pins is the point where any schematic connection will be made so place them accordingly.

After placing all pins, name them in the same manner as naming the pads in the package (refer to Figure 13 for help). This concludes the symbol creation.

Creating a New Device

Once the package and symbol are completed for the new component, they need to be linked together in the form of the device. By selecting the symbol button at the top of the

screen () the same window will appear as the one that resulted from selecting the package and symbol button. Follow the same practice as shown in Figure 4 and Figure 5.

The first step of making the device is to add the newly created symbol to the device. Do this by selecting the add button () finding the name of the symbol, and placing the piece in the desired location. This process can be seen in Figure 15 demonstrating the device of the simple switch.

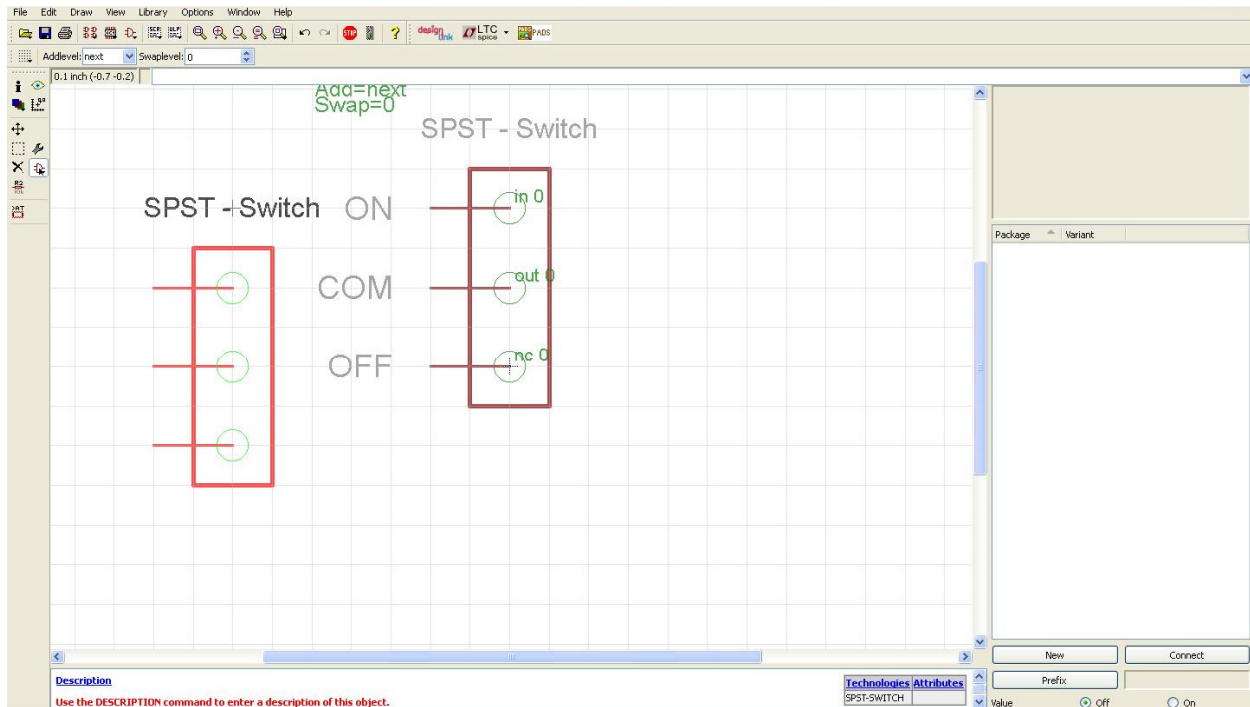


Figure 15. Placing a Symbol as the Device

Once the symbol has been placed, the pins and pads need to be connected. To do this, select the package to be associated with the symbol. Select the “New” button, find the package, and hit “Ok.” Now that the package is selected the connections can be made.

Next to the “New” button in the lower right hand corner, select the “Connect” button. This will open the window seen in Figure 16. Select the pin and pad that need to be connected and hit the “Connect” button in the lower left hand corner of the window.

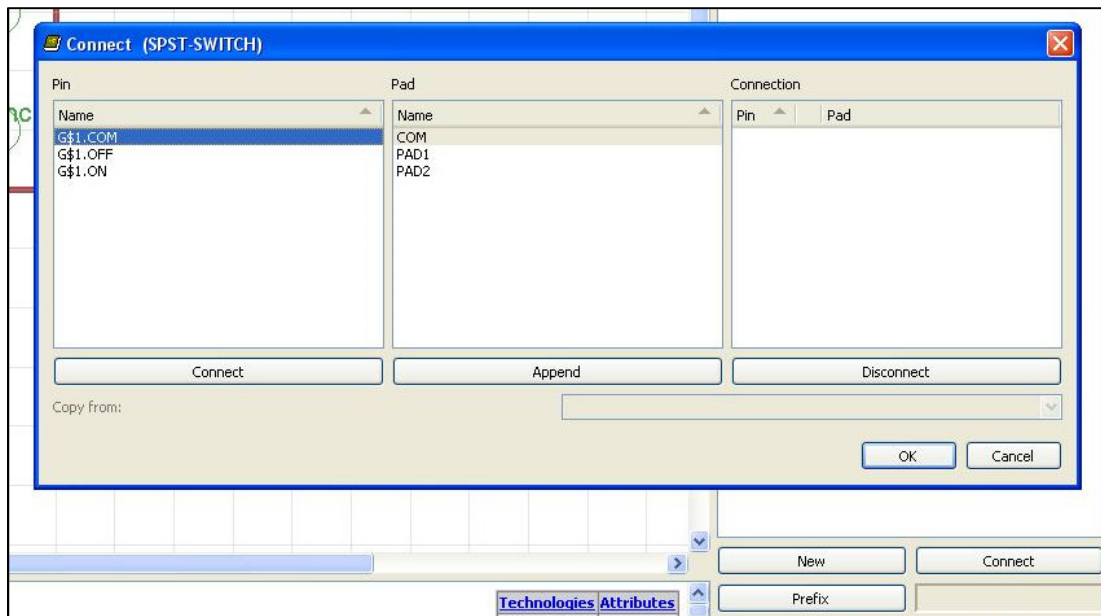


Figure 16. Connecting the Device

After making all the connections, the device is practically completed. The component will work in Eagle's Schematic Tool and PCB Tool, but other aspects of the component can still be changed. The aspects will not be covered in this tutorial.

After selecting "Save," the piece should be ready to be used. Figure 17 shows the final device of the switch that was produced during this tutorial.

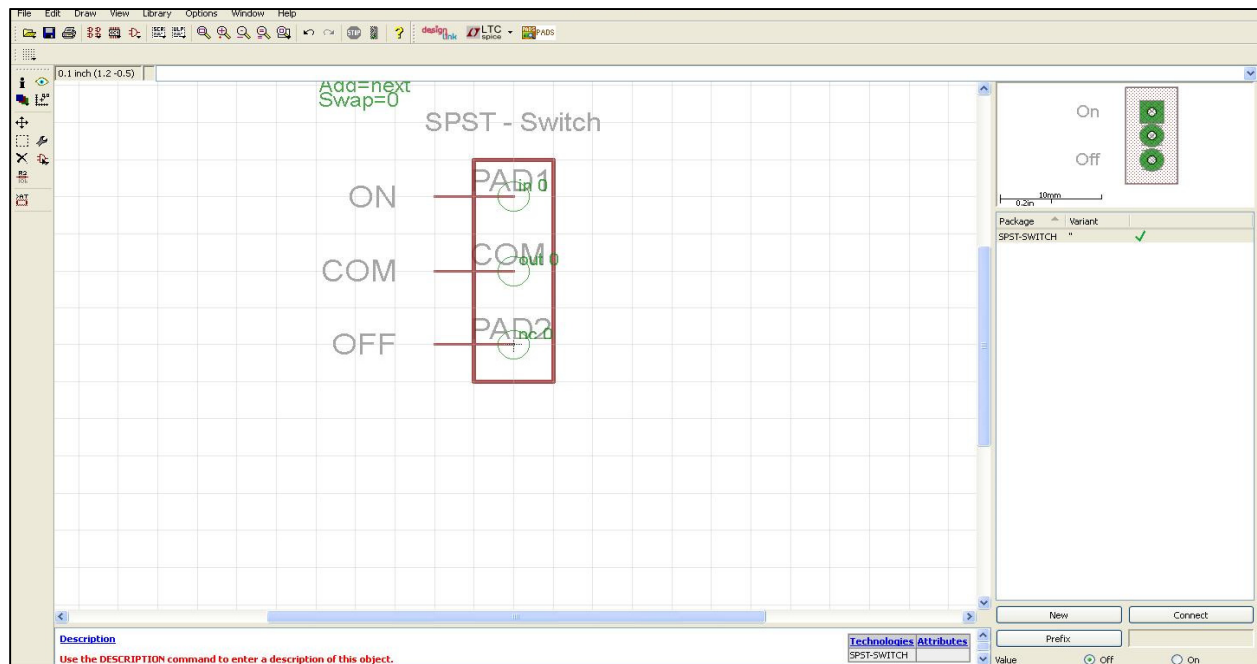


Figure 17. Completed Device

This concludes the tutorial.