

Designing Components in Eagle

Application Note

ECE 480

Ian Meredith

Team 14

3/25/15

Executive Summary

This application note will give instructions on how to design a custom part not found in the library of the design program Eagle. Eagle is used to design printed circuit boards (PCB). The example part designed will be an MSOP-16 to DIP-20 SMT Adapter made by Proto Advantage.

Table of Contents

Introduction.....	3
Objective.....	3
Creating the Library.....	3
Building the Package	4
Creating the Symbol	6
Connecting the Component.....	6
Conclusion	7
References.....	8

Introduction

Printed Circuit Boards (PCB) provide a robust design option for implementing circuits in an application. They are the next step after prototyping the circuit using a breadboard. The PCB is more permanent and reliable than a breadboard. It is used in the large scale manufacturing as a faster and accurate method to producing circuits. The PCB is made from a non-conductive substrate with copper paths etched to form the connections between circuit elements. A PCB can have multiple layers which helps reduce the overall size of the system.

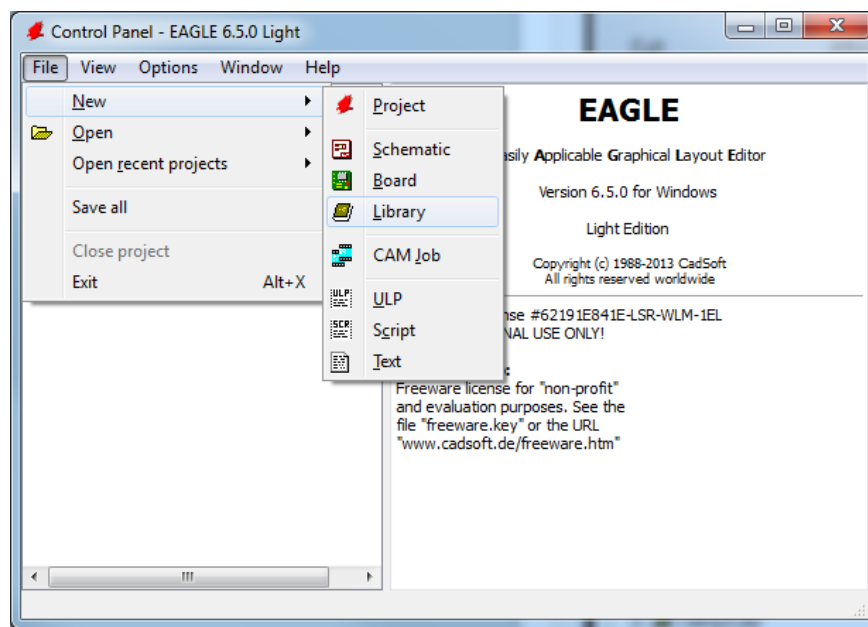
There are many software options available to design a PCB. The software used in this note is Eagle. Eagle provides both a free and a premium version which can be found on their website. The free version has a size limitation of 4 x 3.2 inches. Eagle comes with a large library of different parts that are already designed for the user to use. There are cases in which the part required is not available. When this is the case, there are two options. The first is to find a similar part to use or modify. The second option is to design the part from scratch. This second option will be discussed in this application note.

Objective

The objective of this application note is to help design a part not found in the Eagle Component library. It will describe starting the design without modifying an existing component in the library.

Creating the Library

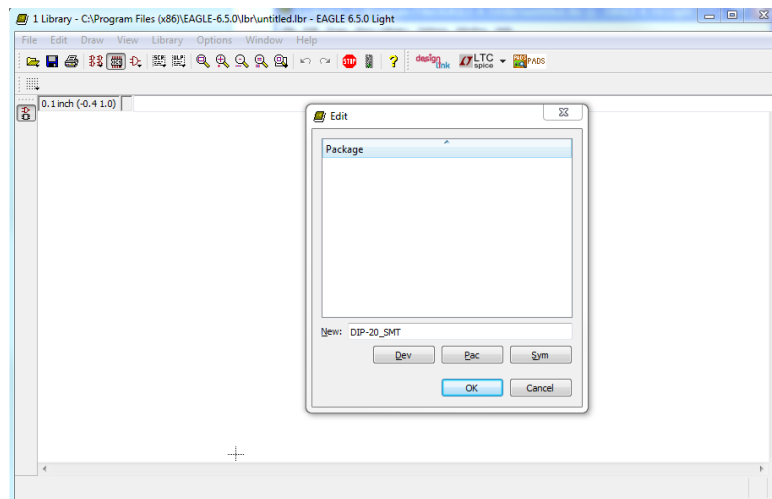
Start by opening the program Eagle. The Control Panel window will be open. Go to file>new>Library. From here the user will be able to create a new library. A library window should open up if done correctly.



Building the Component

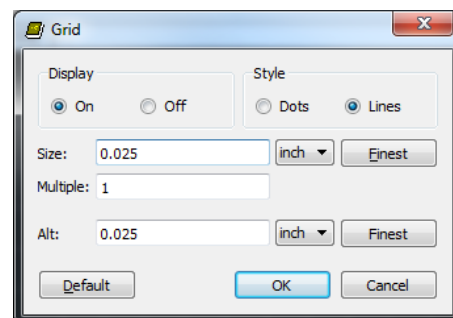
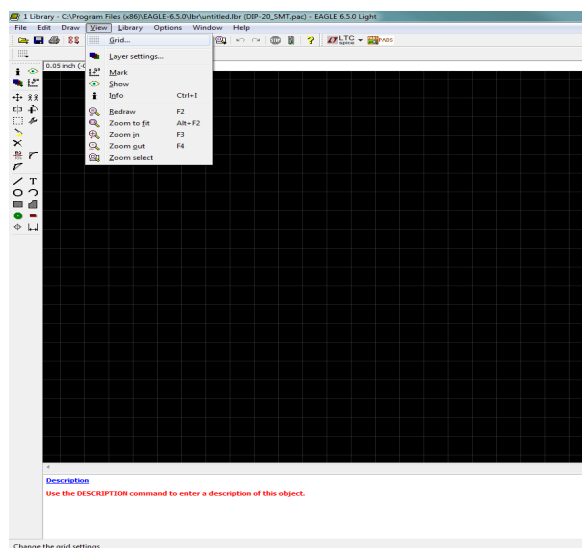
The datasheet is an important tool that will be used in the design. All of the important specifications that are needed in this design can be found on the data sheet. An alternative is to measure the physical component but this is not as time effective as using the datasheet.

Once the library window has opened, the “package” button should be clicked. An edit window will open up. This is where the name of the component being built is entered. This example will use the Proto Advantage MSOP-16 to DIP-20 SMT adapter board. Enter the package’s name and click “OK”. This will open up a grid view which will be used to design the component. The white cross marks the center at point (0, 0).

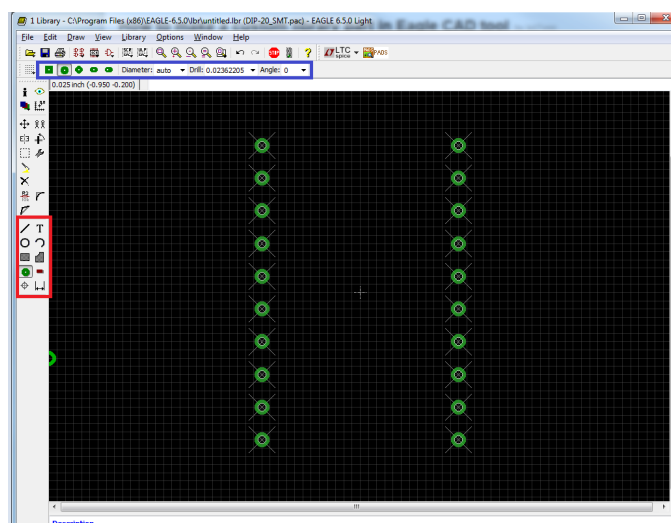


The resolution of the grid will need to be set small enough to accurately place all of the parts of the component. Using the correct resolution for the grid will make designing the component much easier. A good resolution to use is half of the smallest component found on the device. This measurement can be found on the datasheet. Changing the grid can be done in two ways. The first is to type “grid” into the command window and the second is to go to view>grid. Then type in the desired grid size in the “grid” window that opens. For the Proto Advantage MSOP-16 to DIP-20 SMT adapter board, a grid size of 0.025 inches will be used.

Designing Components in Eagle



The component can now be designed once the grid is set. This section is how the component will be connected to the PCB. The toolbar located on the left side of the window gives the ability to add wires, polygons, holes, and pads. Each component will need a unique set of these tools. The Dip-20_SMT adapter will need 20 pads with holes drilled to allow the pins to be soldered to the PCB. While placing the pads, pay special attention to how they are placed. It is easiest to use the white cross as the center of the device. There are two methods to placing components. The first is to use the mouse to drop the components in the correct place. This is done easily if the grid is properly set up. The second is to type the coordinates into the command window after selecting the desired component. The following coordinate format must be used (x-coordinate y-coordinate). The tool shape and size are modified using the toolbar at the top of the window. Take care on which layer, top or bottom, the part is being put on. The figure below shows the tools that can be added in the red box and the toolbar used to modify the tools in the blue box.

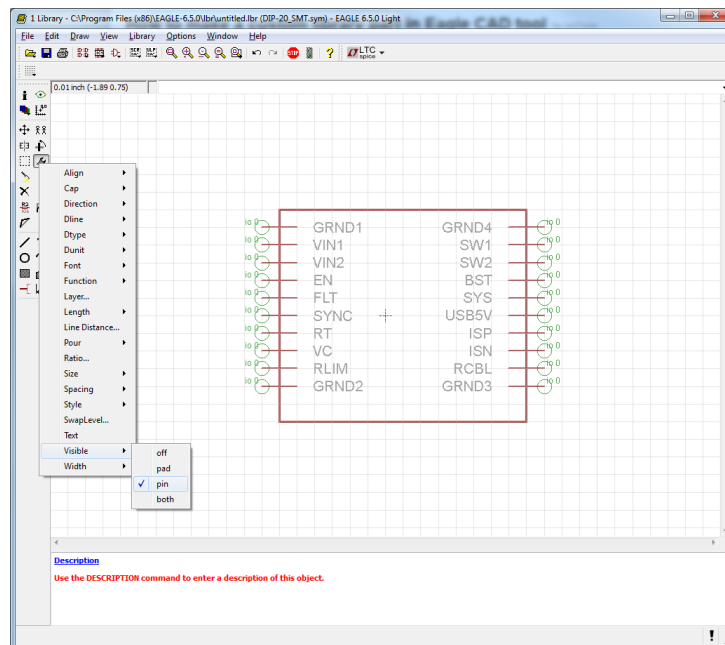


Designing Components in Eagle

The outline of the device being built will then be placed by adding a wire. The layer of the wire is 21 tPlace. This will be helpful when designing the PCB. Each pad should be named as well. This is done by typing “name” into the command window. Then click on each pad and type the correct name.

Creating the Symbol

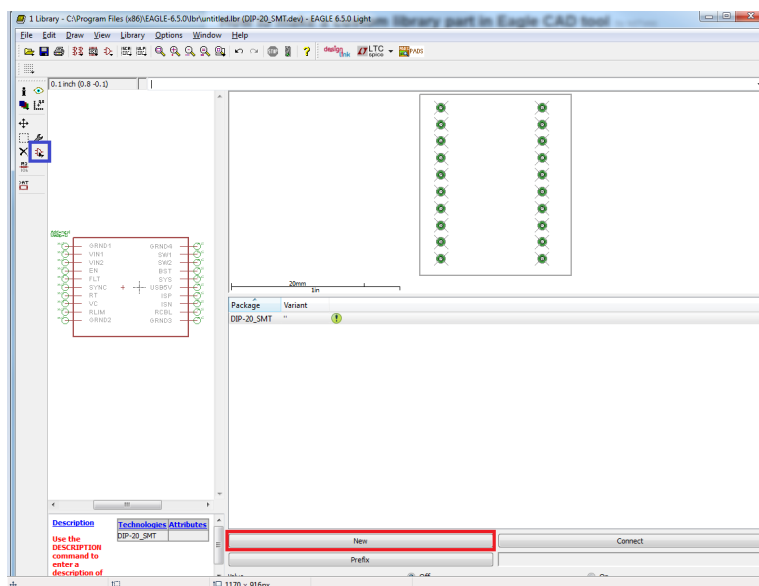
The next step is to create the symbol which will be used in the schematic when designing the PCB. The symbol is the representation of the component that was built in the previous step. Next to the “Package” button is the “Symbol” button. Clicking this will open an “edit” window. Enter the name of the component and click ok. The symbol does not need to be precisely measured as it is just a representation of the component. It should look like the part so it will be easier to add to the PCB design. Using the wire tool, create the shape of the component. The wire should drawing on the symbol layer. The pins are then added to the symbol. The pins should be named just as they were named in while building the component. Once all pins are added and named, click the change button. From here select “Visible” then “Pin”. Click on every pin. This will make the Symbol look much less cluttered when it is added to the PCB design.



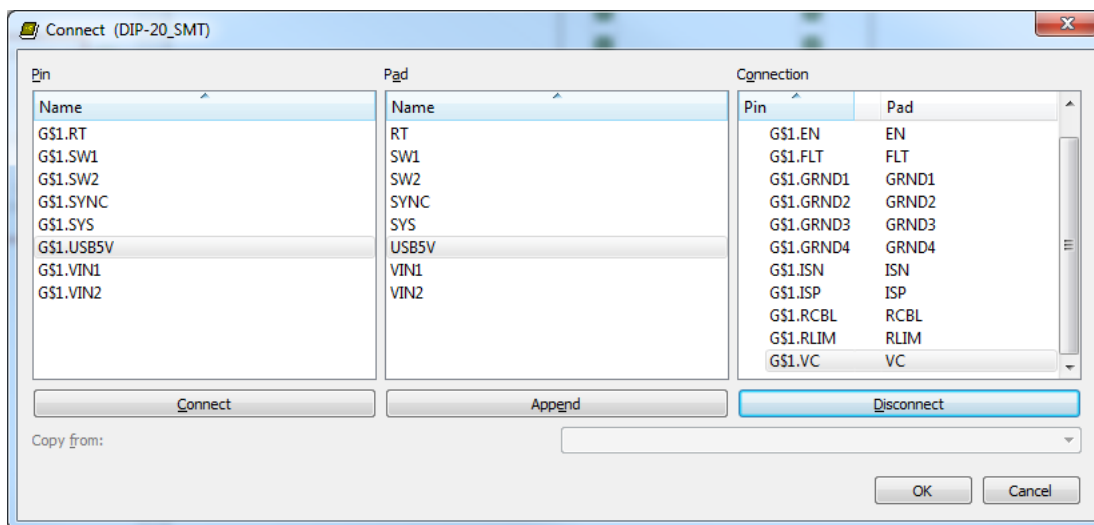
Connecting the Component

The final step is to connect the component designed to the symbol so it can be used in the PCB design. This is done by clicking “Device” which is located next to “Symbol” and “Package”. An “edit” window will open. Name the device and click ok. The figure below shows the window that will open. Click “New” which is outlined in the red box and add the package which was created. Click the “Add” button which is outlined in the blue box and add the symbol that was created.

Designing Components in Eagle



The connections will now be made. Click on the “Connect” button which is located next to the “New” button. Ensure the correct pin is in line with the correct pad. Click connect until all pins are connected to the correct pad. This should be easy if the pins and pads were named correctly.



Now the device can be saved. Save it to the correct place and the component is now complete.

Conclusion

This tutorial should have given the reader a beginner’s guide to designing a custom component in Eagle. There are more features than are described in this guide. For more information, visit the Eagle website.

References

EAGLE. (n.d.). Retrieved April 2, 2015, from <http://www.cadsoftusa.com/>

IPC0079. (2010, January 1). Retrieved April 2, 2015, from
<http://www.proto-advantage.com/store/datasheets/IPC0079.pdf>