

CadSoft EAGLE PCB Design Software



Eric-John Kohler
A41441256

April 1, 2013

Abstract

Background

Whether you are just learning how to design a Printed Circuit Board (PCB), or if you are a seasoned professional, EAGLE software is one of the best and easiest PCB software tools on the market. EAGLE, which stands for Easily Applicable Graphical Layout Editor, has the ability to produce very simple PCBs as well as extremely complicated PCBs. When designing a PCB, the pre-programmed libraries within EAGLE allow for the user to utilize a variety of components. These components come in both board-mount and through-hole types. With all of the options that EAGLE has to offer, EAGLE has become one of the leading software tools in the PCB industry.

Topics Covered

After reading this document, users should be able to design a schematic, lay out a board, and then convert that file into a file type that can be used by the PCB manufacturers. This can all be done within the EAGLE software and can be completed in a short amount of time. EAGLE can be downloaded from the CadSoft website and comes in both a professional and lite version, with the lite version being free for users. In addition, this document will show you how to add libraries from outside sources that will provide the user with an extra amount of components that can be used within the software.

Keywords

Autorouter, Board, Components, Drill, EAGLE, Excellon, Gerber, Layout, Library, Routing, Schematic.

Table of Contents

Abstract.....	2
Background.....	2
Topics Covered.....	2
 Keywords.....	 2
 Introduction.....	 4
1. Download & Install.....	4
2. Initial Setup.....	4
 Schematic Design.....	 5
1. Adding a Library.....	5
2. Selecting Components.....	6
3. Connecting Components.....	7
4. Finishing Up.....	9
 Board Layout.....	 10
1. Laying out the Board.....	10
2. Routing the Board.....	11
3. Finalizing the Board.....	12
 Board Conversion.....	 13
1. Creation of the Drill Rack Files.....	13
2. Creation of the Excellon Files.....	13
3. Creation of the Gerber 274x Files.....	13
 References.....	 14

Introduction

Download and Install

To download a copy of EAGLE, go to <http://www.cadsoftusa.com/download-eagle/?!language=en>. Download the correct files for your operating system. Users can select between Windows, Linux, and Macintosh. Save the file and unzip the files to a directory that can be selected by the user. When the program is first opened, the user can either enter a purchased license or run the software as freeware. One of the main constraints is that the freeware version only allows a board to be a maximum of four inches by three inches. This document will focus on the Windows freeware version.

Initial Setup

Users will be prompted with this initial Control Panel.

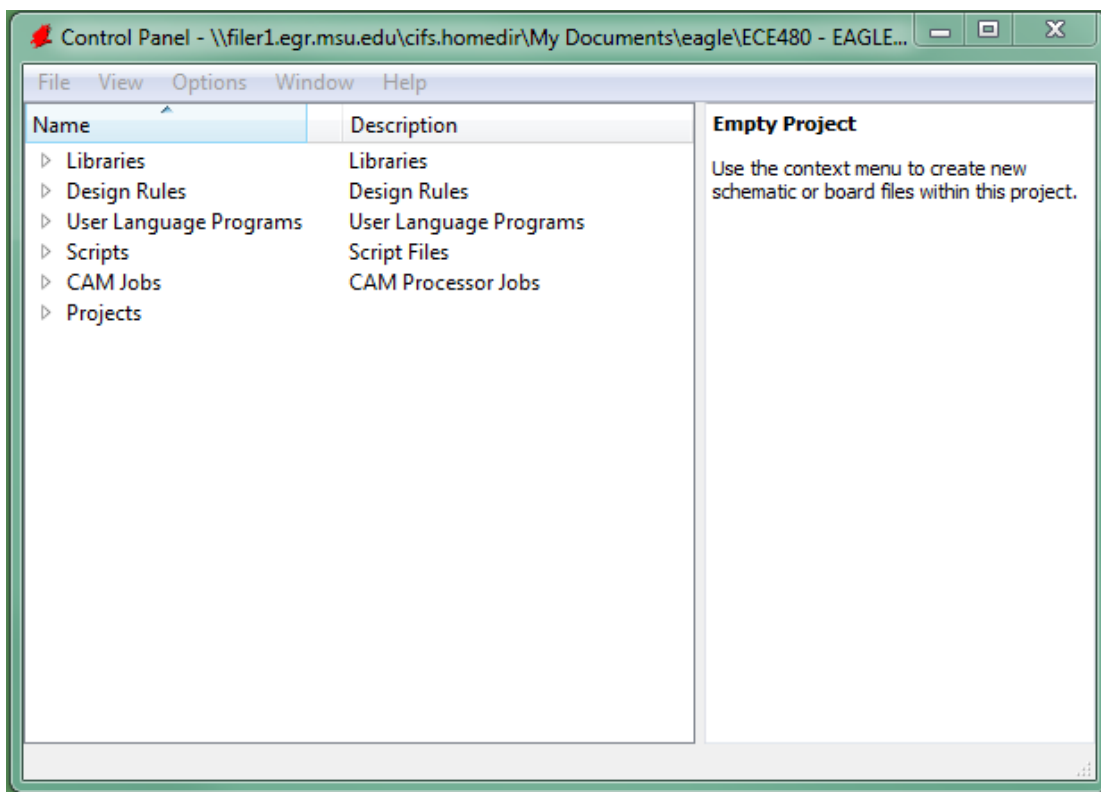


Figure 1. EAGLE Control Window

From there users should select “File” → “New” → “Project”. Once a new project is created, right click on the project title and select “New” → “Schematic”.

Schematic Design

Adding a Library

EAGLE has a variety of libraries already installed with the initial download from their website. Users need to select the libraries they want to use in their schematic design by manually adding these libraries. To do this, users need to select “Library” → “Use” on the Schematic window like so:

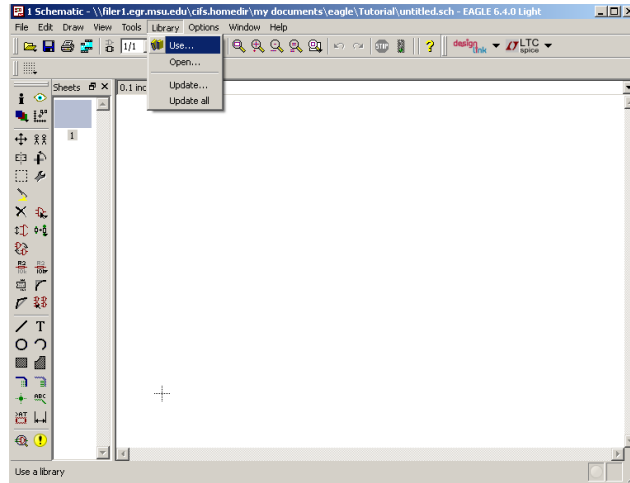


Figure 2. Library Drop-down Menu

From there, the easiest thing to do, when you are unsure of exactly what library you want to use is to select all of them. To do this, click on one of the libraries and hit “Ctrl+a”, this will select all of the libraries that are built into EAGLE. It should look something like this:

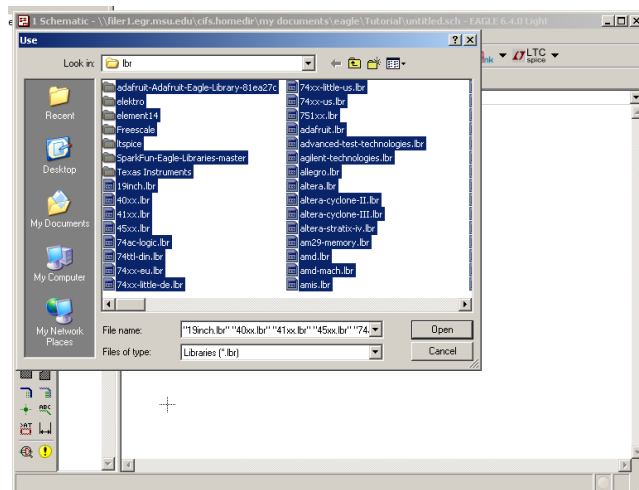


Figure 3. Selecting All Available Libraries

Once all of the libraries are selected, click “Open”, which will allow all of the available libraries to be used.

Selecting Components

Now we are going to talk about how to select a component from the libraries and adding them to the schematic.

To do this, click on the button on the left side toolbar that looks like this:



This will bring up the libraries, which has a built in search function. For this example, we will search for a resistor and add it to our schematic. The search function will bring up all resistors that are available, select one of them and you can see what it will look like on the schematic as well as on the board. Here's what it should look like:

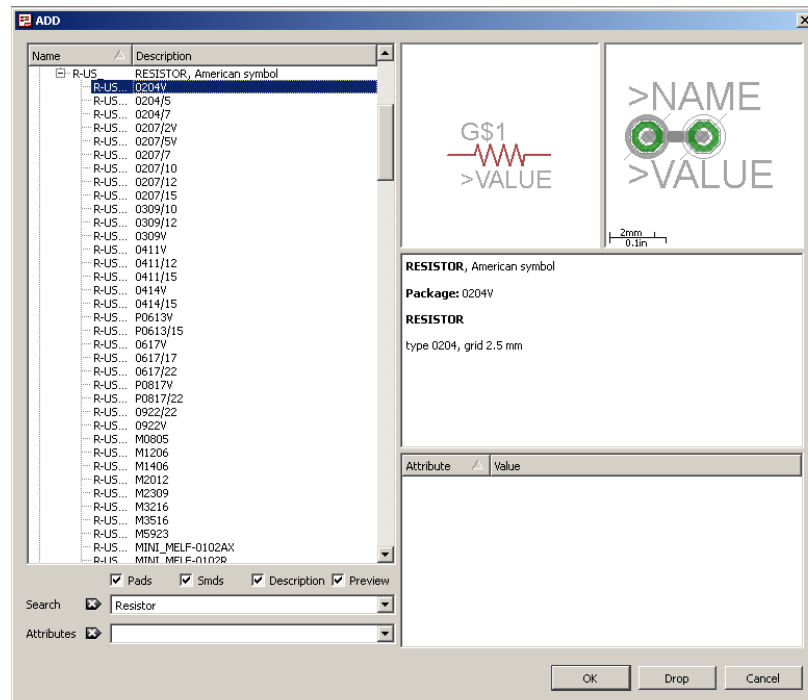


Figure 4. Library Search Results for “Resistor”

To add this component to the schematic, select “OK.”

ECE480 Application Note: EAGLE PCB Software

From there, click on the schematic to place the component on the board. If you want to add multiple of that component, continue to click on the board. Once finished, hit the “Esc” key, which will bring you back to the library. In this example, we will show you how to make a simple voltage divider. To do that, we will need to add two resistors, one battery, two grounds and an output pad to read the output voltage. Once everything is added, it should look like this:

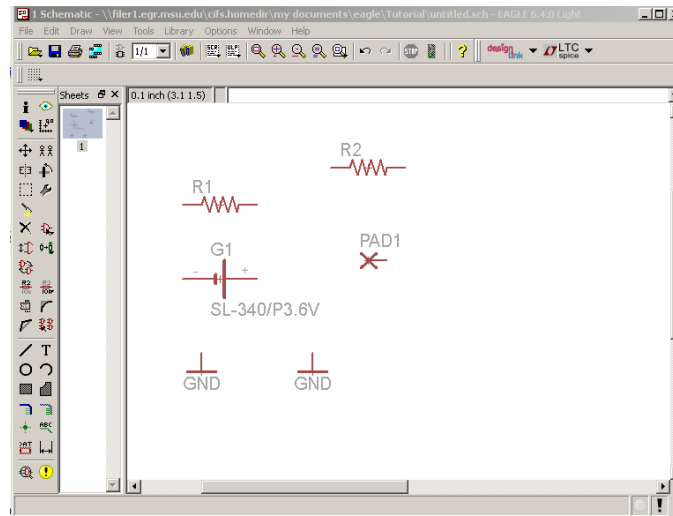


Figure 5. Schematic with Components Added

The next step is to lay out the components and wire them together.

Connecting Components

Before you start connecting components, you should learn what some of the main items on the left side toolbar mean.



From left to right, top to bottom. The buttons are “Move”, “Copy”, “Mirror”, “Rotate”, “Group”, “Change”, “Paste”, “Delete”, and “Add.”

The most useful for laying out the schematic are “Move”, “Rotate”, “Group”, and “Delete.”

ECE480 Application Note: EAGLE PCB Software

Once everything is in place, it should look something like this:

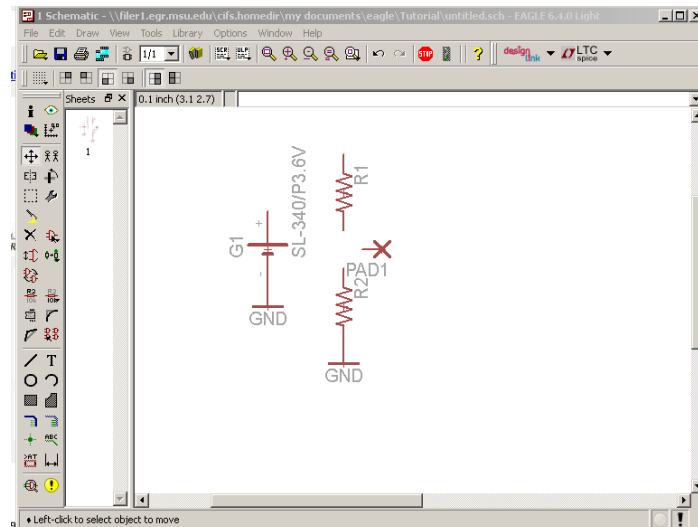


Figure 6. Schematic with Components Layed Out

The next step is to wire these components together. To do this, select the  button on the left side toolbar. To connect components, click one end of the component and then click the end of the other component. Once everything is connect, it should look like this:

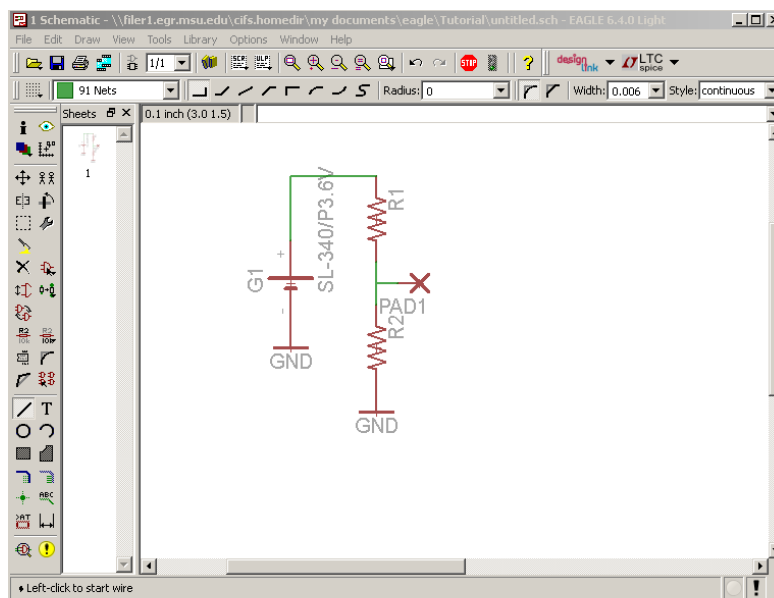


Figure 7. Schematic with Components Connected

You now have a completed schematic and you are ready to start laying out your board. Before that, we will go over a few things to make your schematic design a little more detailed.

Finishing Up

The first thing we are going to do is change the names and values of the components. To do that, we need to click on these buttons respectively:



. Change both of the resistors to 500Ω, and change the battery to 3V. Also, call the battery “Battery” and call the output pad “V_OUT.” When you are finished, it should look like this:

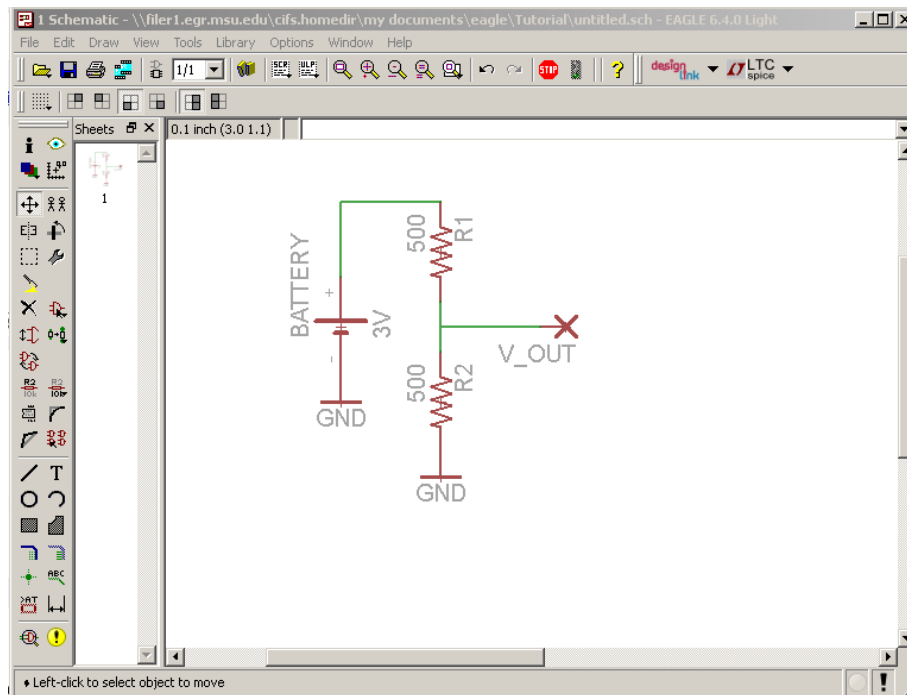


Figure 8. Schematic with Names and Values

Now we are ready to layout the board.

Board Layout

Laying out the Board

To switch to the board select “File” → “Switch to Board.” You will need to rearrange the board, similar to the way you laid out the schematic, but initially the board will look like this:

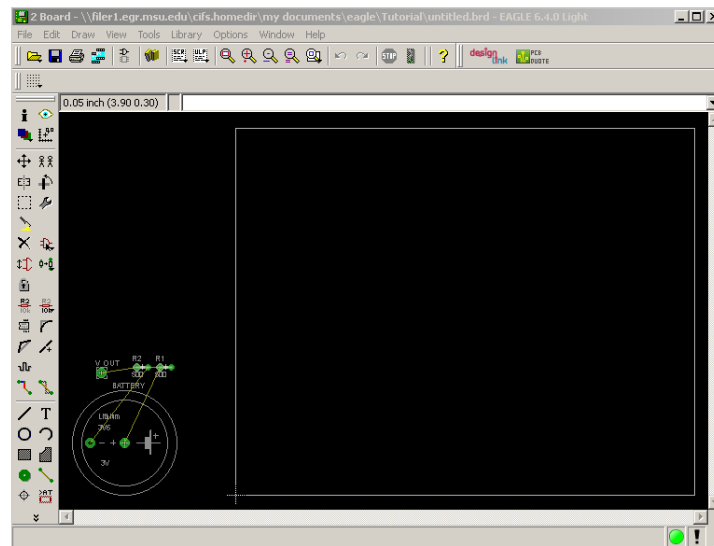


Figure 9. Board with Components to be Organized

Once laid out, your board should look like this:

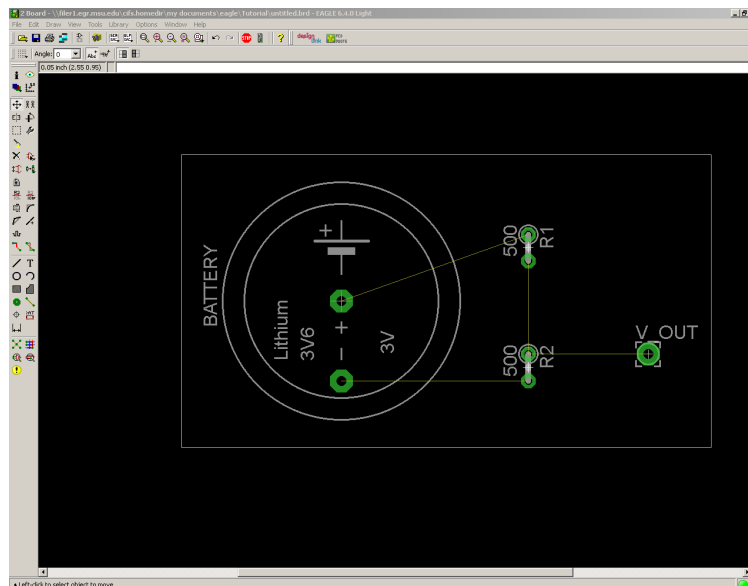
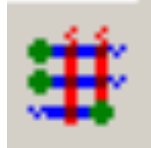


Figure 10. Board with Components Organized

To adjust the size of the board, move the edges of the board to fit the desired size. The next step is to rout the board.

Routing the Board

This can be done easily by selecting the “Autorouter.”



The “Autorouter” button looks like this

This will prompt you with a window where you can select the Routing Grid size as well as the layers of the board. For our purposes, we will select the Top as “|” and the Bottom as “-”. Our Routing Grid will be 50. These are the default settings in EAGLE. Click “OK” and EAGLE will make the connections for our board. The top connections will be red, while the bottom connections will be blue. When finished, the board will look like this:

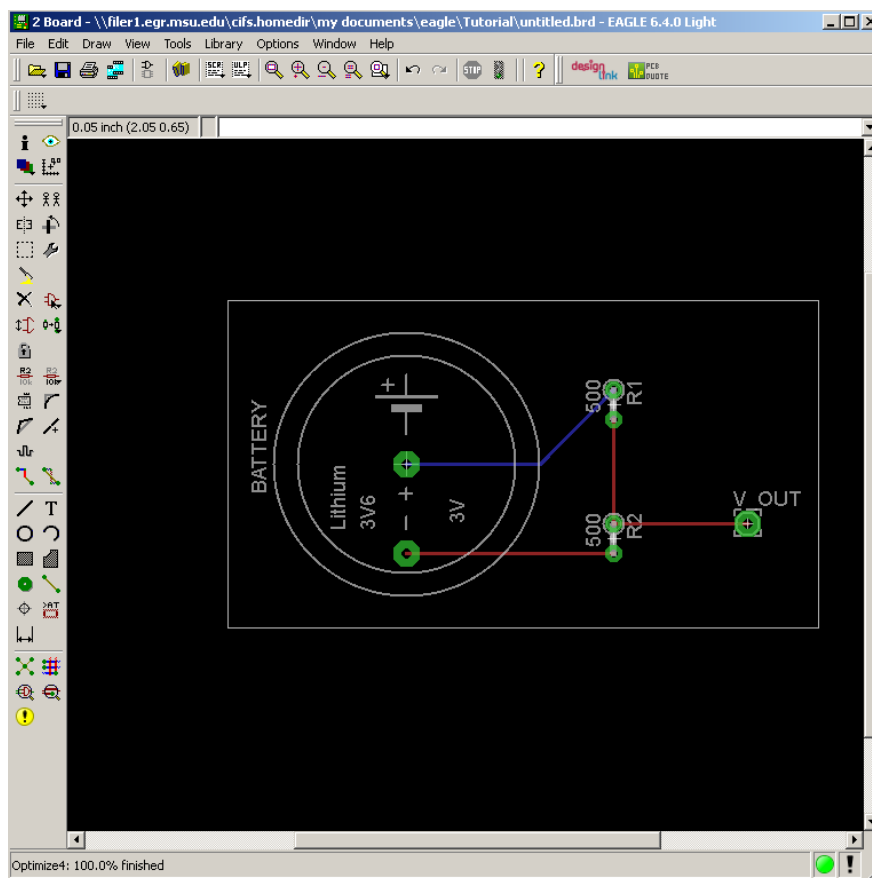


Figure 11. Board with all Routes Connected

Finalizing the Board

Often times, you will want to add a name to the board or mounting holes to the board. To add a name to the board, select the “Text” button from the left side toolbar. Type the name of your device in the box and add it to the board by clicking where you want it to appear. When you are finished, hit “Esc” twice. Next, to add holes to the board, select the “Hole” button from the left side toolbar and click on the board where you want to add them. In our example, we will add holes in each corner of the board. When that is completed, your board will be ready to convert to a file that can be used by the manufacturers and it should look like this:

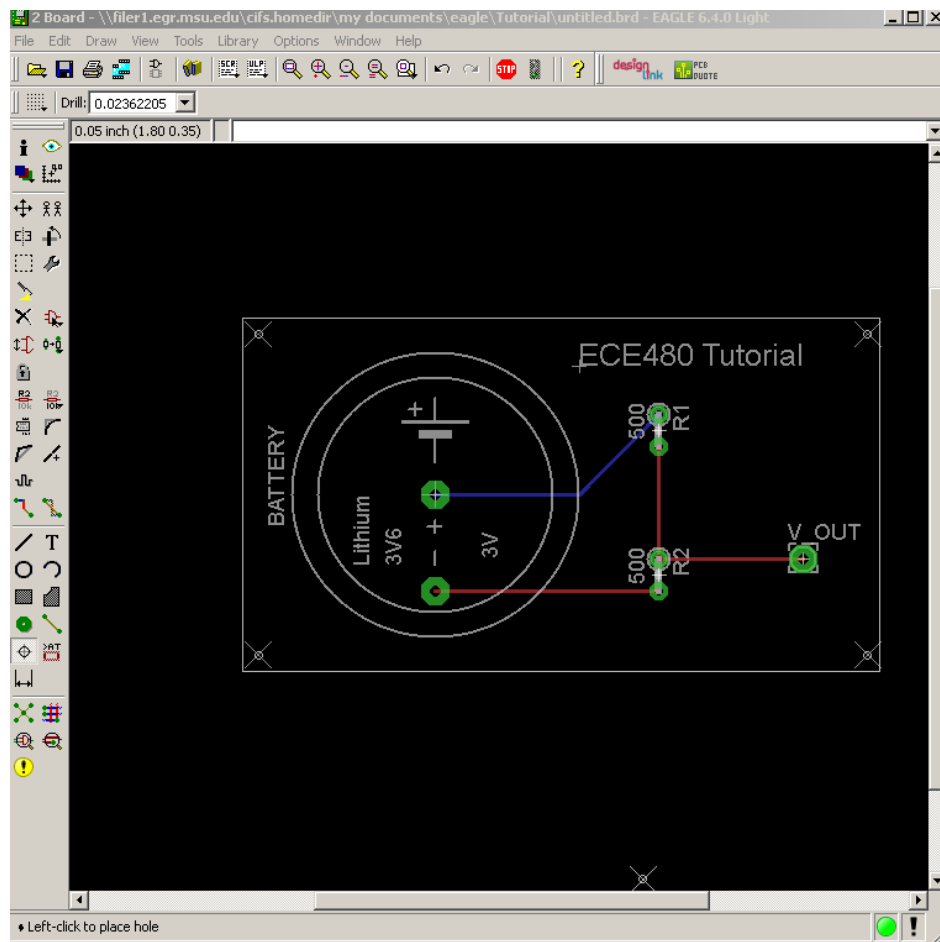


Figure 12. Finalized Board

Board Conversion

Creation of Drill Rack File

In the command bar, type “run drillcfg” and push “Enter”. EAGLE will then prompt you to select the unit for the output file. Select “inch”, then hit “OK” twice. This will then prompt you to save your file. Type in your desired file name and hit “Save.” This should be a “.drl” file.

Creation of Excellon Drill Files



The first step is to click on the “CAM” button which looks like this: That will open the “3 CAM Processor.” Select “File” → “Open” → “Job” → “excellon.cam” → “Open”. The next step is to select the file you want to convert. Click on “Rack” → “(your .drl file name)” → “Open”. We are now ready to generate the drill data and process the job. To do this click “Process Job.”

Creation of the Gerber 274x Files

Next we are going to want to make the “Gerber274” file. Click “File” → “Open” → “Job” → “gerb274x.cam” → “Open”. This will create the Component Side. Select “Process Job” and exit the “3 CAM Processor.”

You have now created the files required for the manufacturer to design your board. The files needed are:

- *.drl Drill rack data
- *.drd Excellon drill description
- *.dri Excellon drill tool description
- *.cmp Component side data
- *.sol Solder side data
- *.plc Component side silk screen data
- *.stc Component side solder stop mask data
- *.sts Solder side solder stop mask data
- *.gpi Gerber photoplotter information data

Put these files together in a *.zip file and send them to your board manufacturer. You have just completed a Printed Circuit Board.

References

CadSoft Website:

<http://www.cadsoftusa.com/?CMP=KNC-G-BRND-EC>

Tutorial 1 for Eagle: Schematic Design:

<https://www.youtube.com/watch?v=1AXwjZoyNno>

Tutorial 2 for Eagle: Printed Circuit Board Layout:

<https://www.youtube.com/watch?v=CCTs0mNXY24>

Gerber Conversion:

http://www.piclist.com/images/www/hobby_elec/e_eagle44.htm