

# Design a Single-layer PCB Using Eagle

---

ECE480 Senior Design Team 5

**Cheng Zhang**

**11/15/2012**

**Abstraction:** This application note is a tutorial of designing a single-layer PCB using Eagle program. The tutorial will guide the reader through building schematic, designing PCB layout and generating CAM files in the Eagle program.

**Contents**

Introduction ..... 2

Objective ..... 2

Getting started with schematic..... 2

From schematic to layout ..... 10

Preparing for fabrication with CAM..... 12

## Introduction

PCB, which is also known as printed circuit board, is very commonly used in commercially produced electronic devices. A printed circuit board usually use conductive pathways, tracks and traces etched or striped from a copper sheet covered on a non-conductive substrate. PCB is also a very reliable and a very effective way of implementing a designed circuit with a good manufacturability.

To design a PCB from a circuit schematic usually involves computer aided design software. In this application notes, Eagle program which is developed by Cadsoft will be introduced. This tutorial will walk the reader through building the schematic in Eagle and designing the layout of the PCB. Eagle, stands for easily applicable graphical layout editor, is capable of schematic capture, PCB layout, and CAM. In this tutorial, every feature will be discussed respectively.

There are a few versions of Eagle for different tasks. The one that will be used in this tutorial is a free version of Eagle. The free version of Eagle is restricted to a maximum size of 3.9 inches by 3.2 inches. It is capable of designing two-layer PCB, while this tutorial will only demonstrate the single-layer PCB layout. Even with the restrictions, the free version of Eagle is still an ideal tool of design small single-layer printed circuit board.

A free version of Eagle is available for downloading through the following link:

<http://www.cadsoftusa.com/download-eagle/?language=en>

## Objective

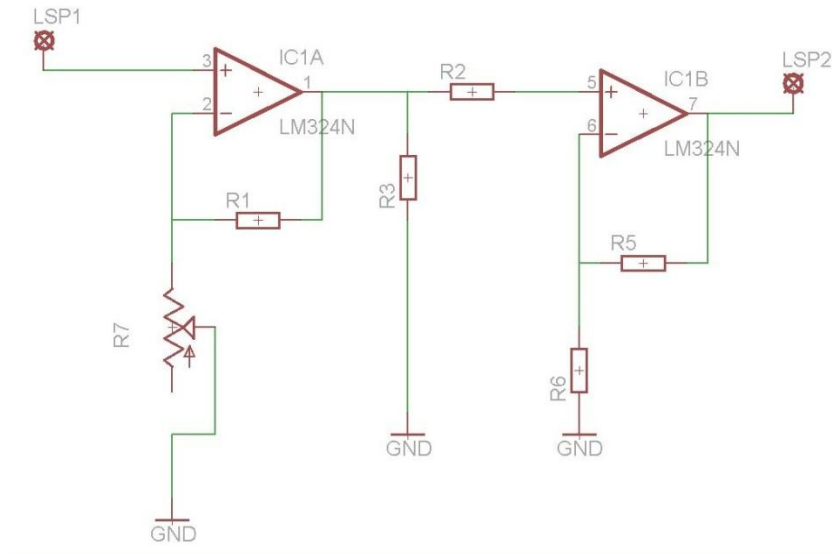
The objective of this tutorial is to help readers to design a printed circuit board from a schematic. The tutorial also includes techniques such as CAM process, which are both very useful and necessary steps for building a printed circuit board.

Key words: Single-layer PCB, layout, CAM, schematic, Eagle, Cadsoft

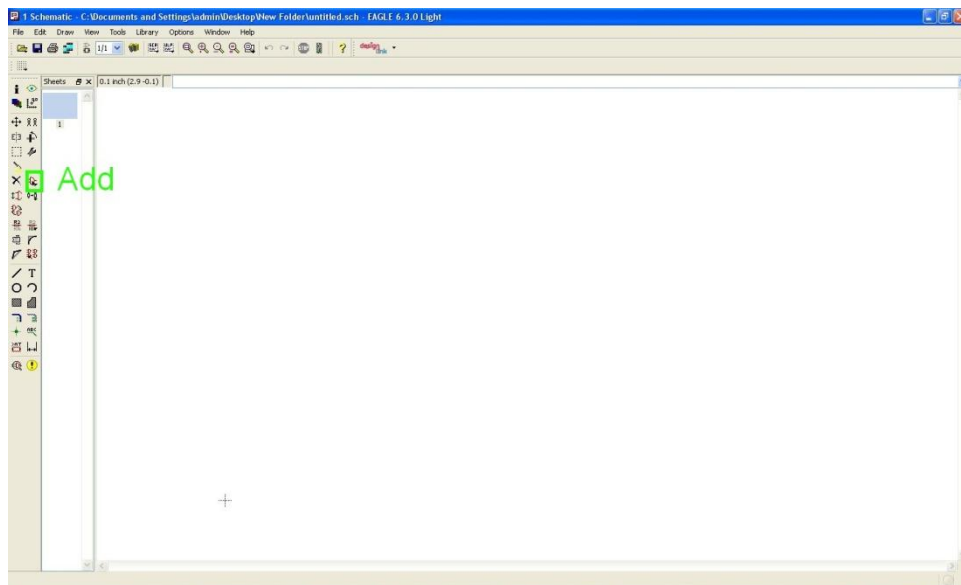
## Getting Started with Schematic

In this tutorial, the following example will be used as demonstration. This demonstration involves some representative elements to be introduced. Readers should find it helpful by follow the steps respectively.

## Design Single-Layer PCB Using Eagle

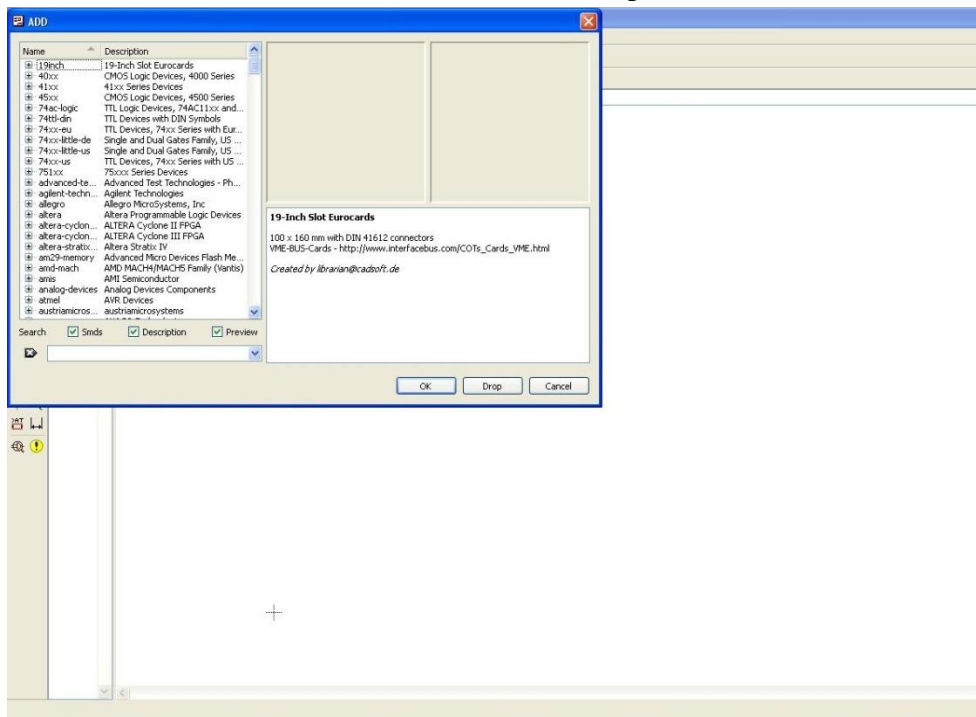


First of all, run Eagle program. The Control Panel will be displayed. On the Control Panel, select File>New>Schematic. That will open a new schematic window. On the left of the schematic window, there is a add tool:



This Add tool will add elements such as resistor, capacitor and ICs, all of which are very important in designing the PCB.

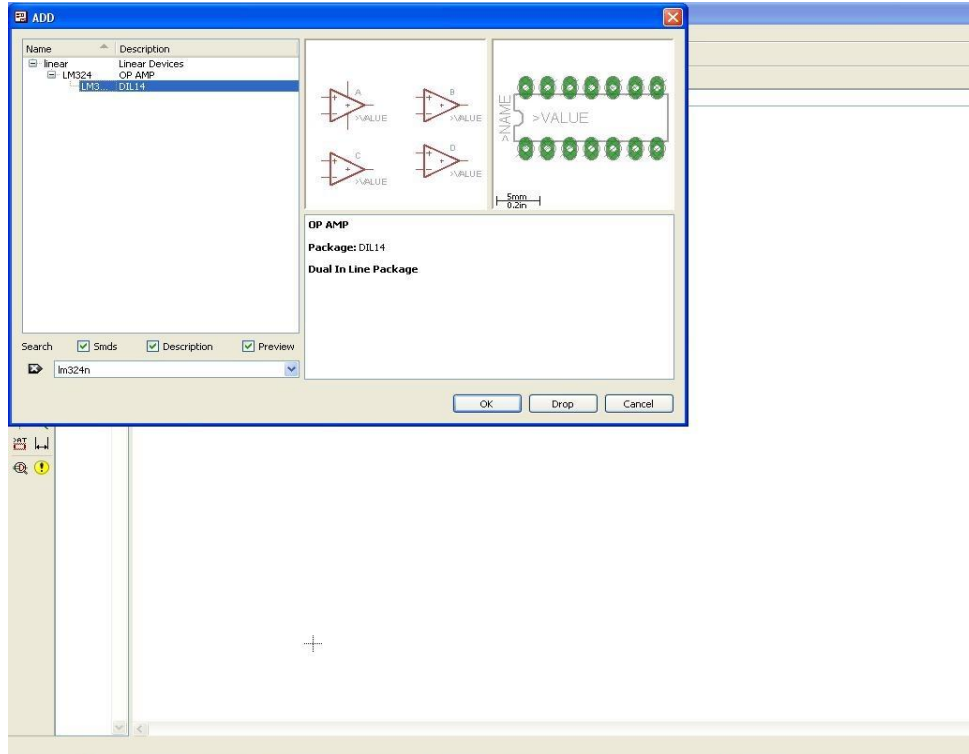
Once the add tool is select, it will open a window:



On this window, a proper element needs to be selected. The Add tool serves like a library of elements. There are a lot of elements manufactured by different companies to be chosen from. However, this library may not contain all the elements needed for project. It is possible that certain project needs to use an element that is not contained in the library. In that case, there are different element libraries online for Eagle to use. User need to find the proper library to do their project. In the demonstration, LM324N will be used to show how a chip will be added.

To find LM324N in the library, search for “LM324N”:

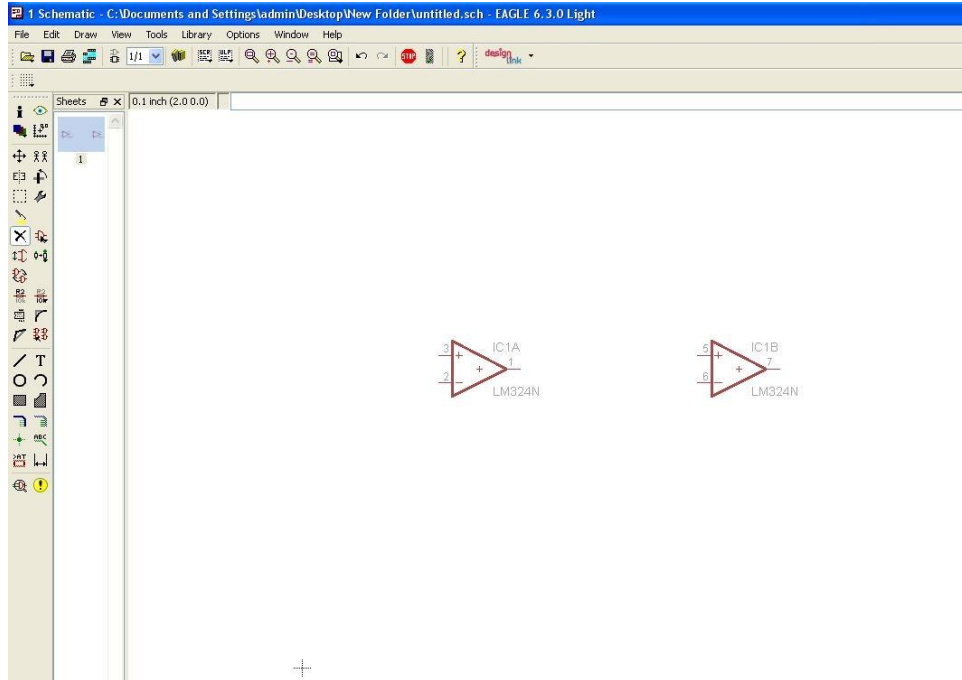
## Design Single-Layer PCB Using Eagle



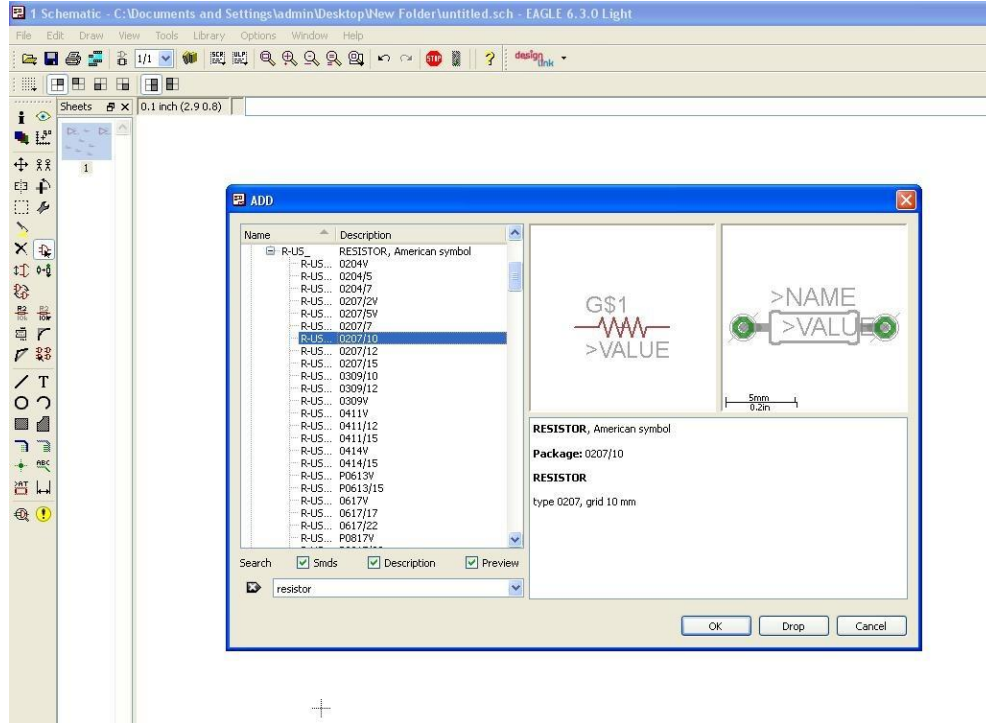
**Tips:** ICs like LM324 are usually contained in the library. However, due to different package of the ICs, the package information must be included in the key words. In this case, instead of searching LM324, searching LM324N will return the correct element.

The particular LM324N used in the demonstration has four op-amps built-in in a single chip. The chip has a DIL14 package which stands for dual in line package. This is a standard through-hole package for LM324. To add LM324 to the schematic, press OK and place two op-amps on the schematic. Eagle will automatically use two op-amps within the total of four, respectively, the first two op-amps on the chip. Now the schematic should look like:

## Design Single-Layer PCB Using Eagle

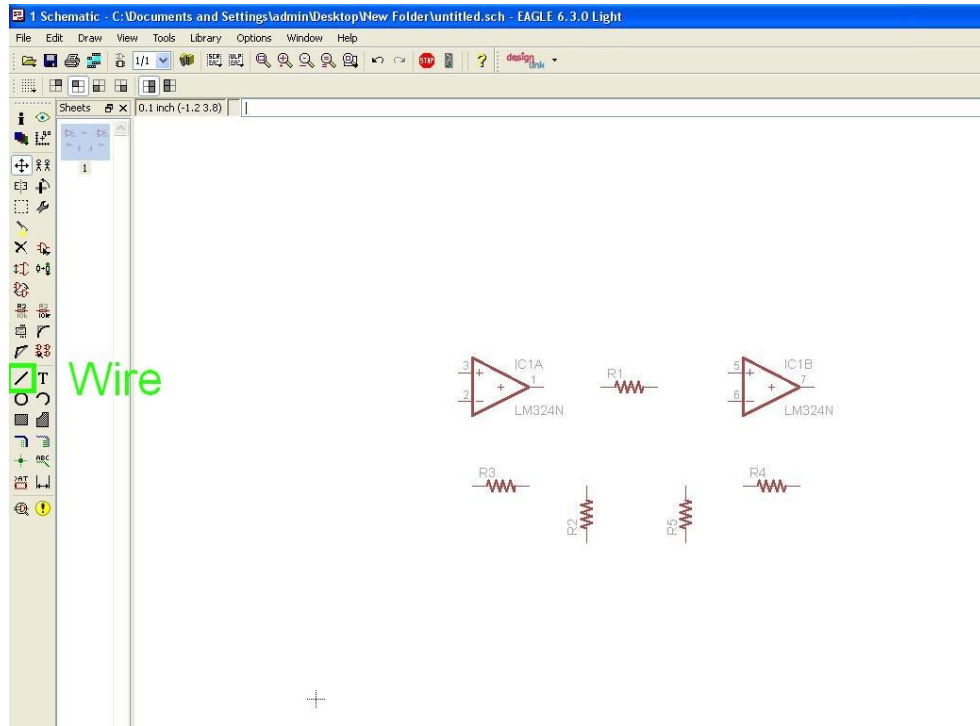


Then, resistors need to be added. Select Add tool again and search for resistor. Use the key word “resistor”. That should return a lot of results. The choice of resistor completely depends on the size of the resistor. In this demonstration, the resistor with grid of 10mm is chosen for this demo.

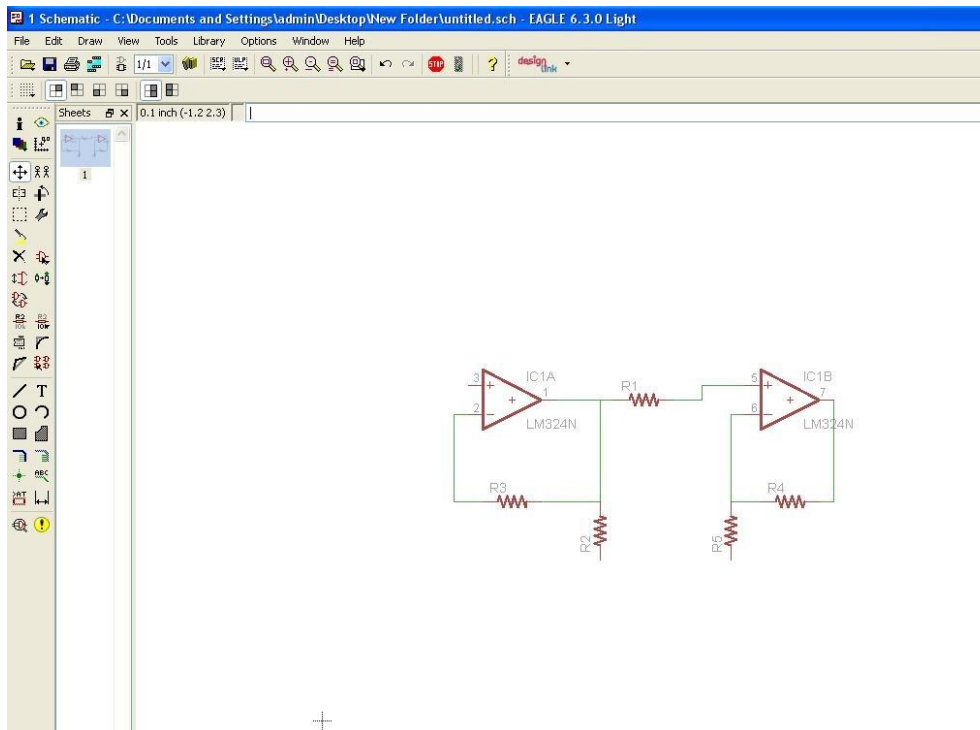


Then press OK and place five resistors on the schematic. Then wires need to be added to connect the elements. To add wires, select wire on the left of the window:

## Design Single-Layer PCB Using Eagle



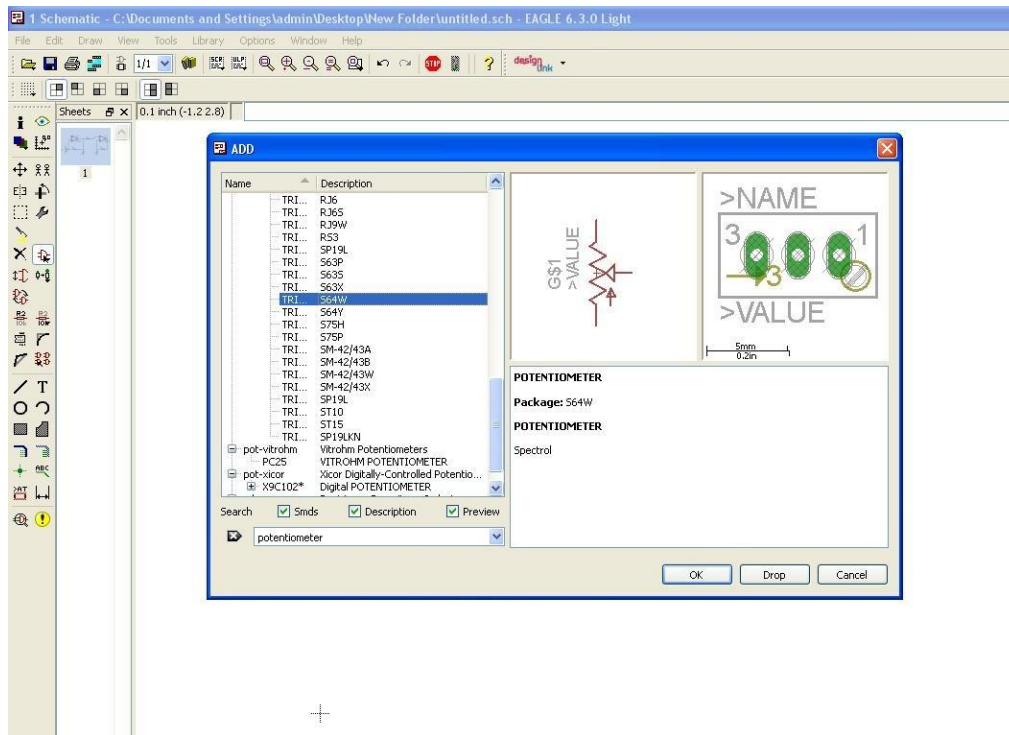
Then, click on any spot wished and that will be the start point of a wire. Click again will be place a turning point of the wire. Notice that it is not the end point. To end a segment of wire, press Esc. Then, connect the element according to the schematic:





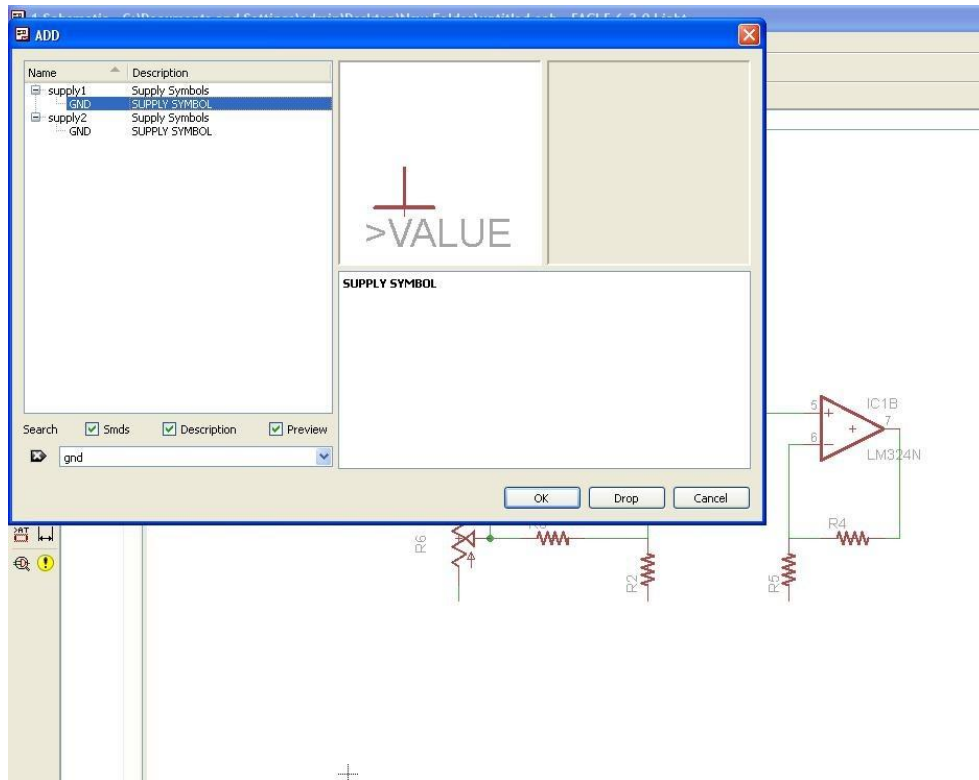
## Design Single-Layer PCB Using Eagle

To finish the schematic, a potentiometer needs to be added. To add a potentiometer takes some more efforts to find. In this demonstration, a potentiometer manufactured by Spectrol is used. This particular potentiometer has a square package and the three pins are in a line. Search for potentiometer. This particular potentiometer is in the directory of TRIM\_RE. Expand the directory and find the particular potentiometer manufactured by Spectrol:

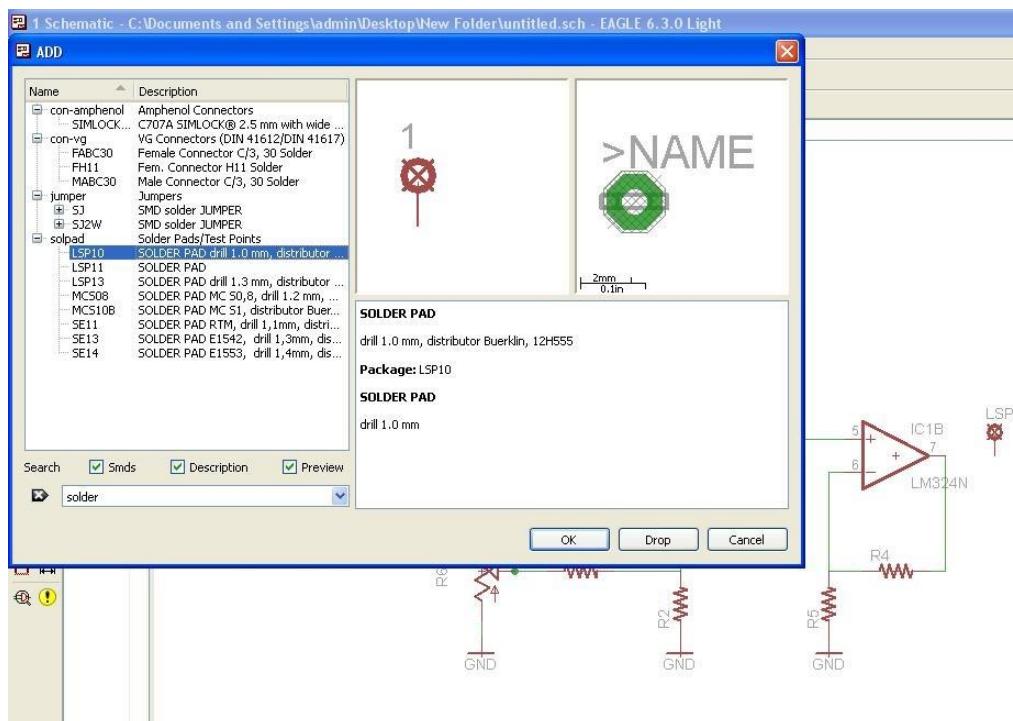


Click OK and place the potentiometer where it should be. Then connect the potentiometer to the circuit. Then, ground needs to be added to the schematic. To add ground, select add tool again and search for GND. Then choose either one from the two ground symbol available and place ground where it needs to be:

## Design Single-Layer PCB Using Eagle

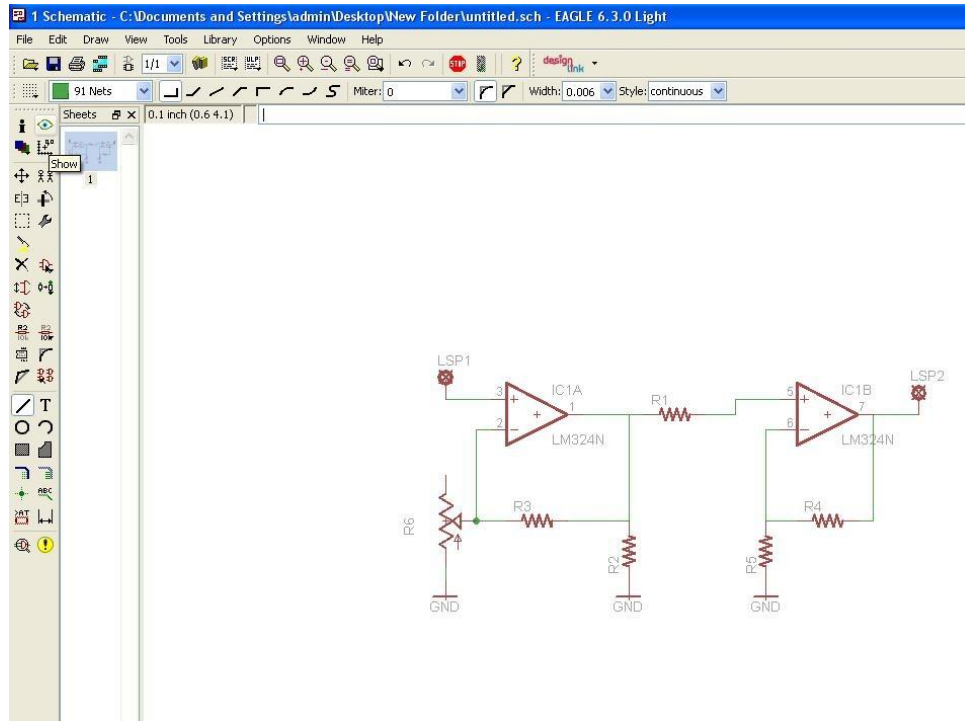


Then, connect the ground as shown in the circuit schematic. The last task is to add two soldering pad for input and output of the circuit. Open the add tool again and search for “solder”. There are several kinds of soldering pads to choose from. In this tutorial, a soldering pad of 1mm diameter will be chosen:



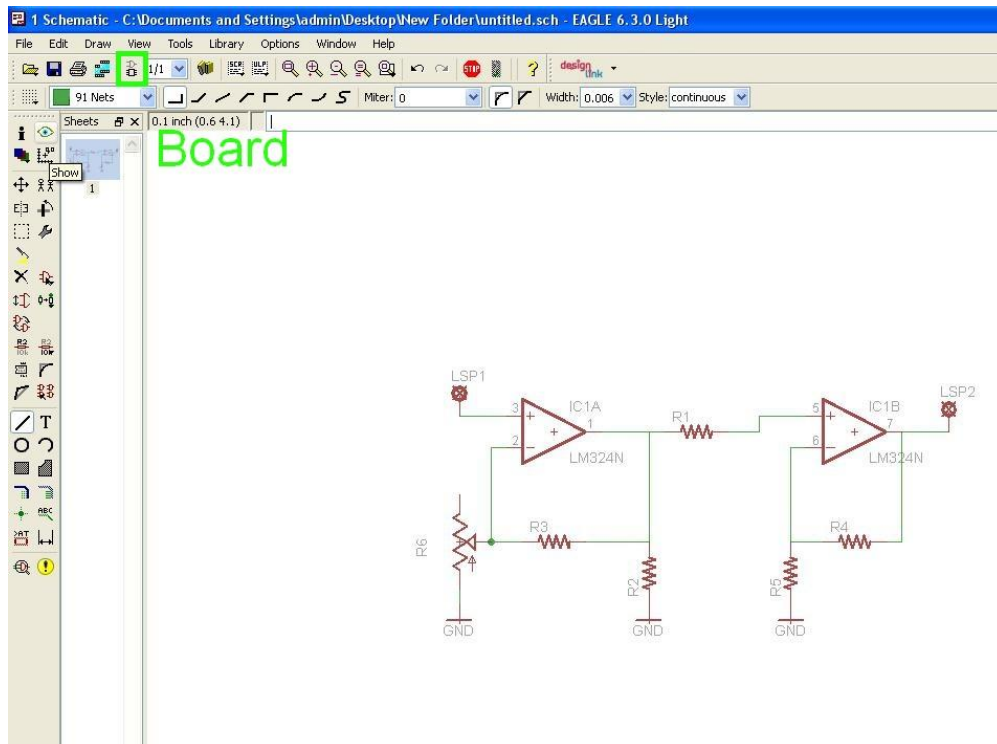
## Design Single-Layer PCB Using Eagle

Then connect the soldering pad to the circuit, the schematic of the circuit is finished then:



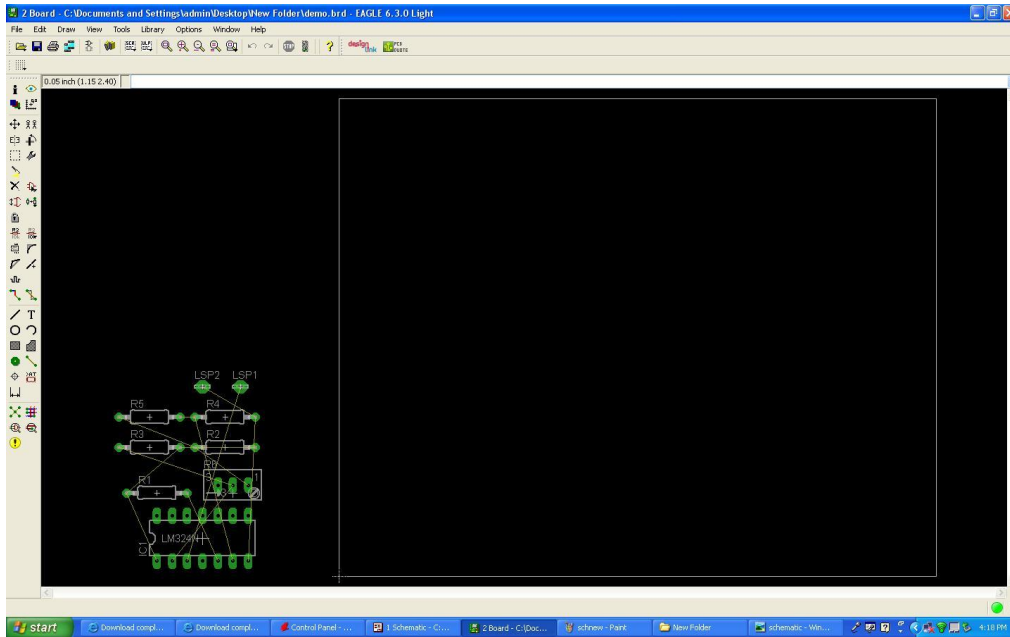
## From Schematic to Layout

After the schematic is done, press the bottom shown in the picture:

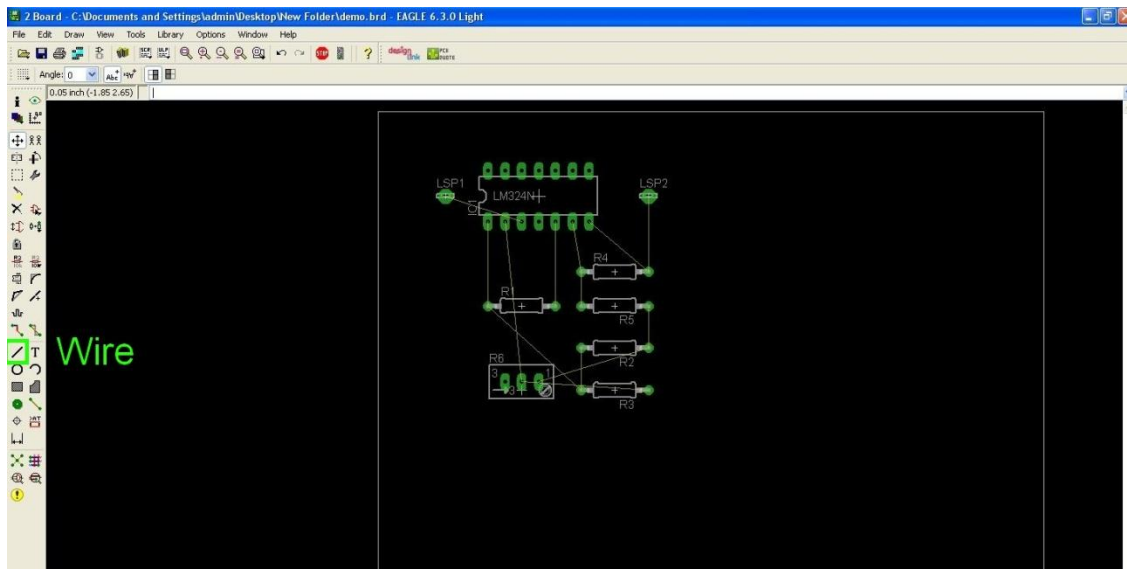


## Design Single-Layer PCB Using Eagle

This will automatically open a PCB board window and have the elements and signal wires in place:



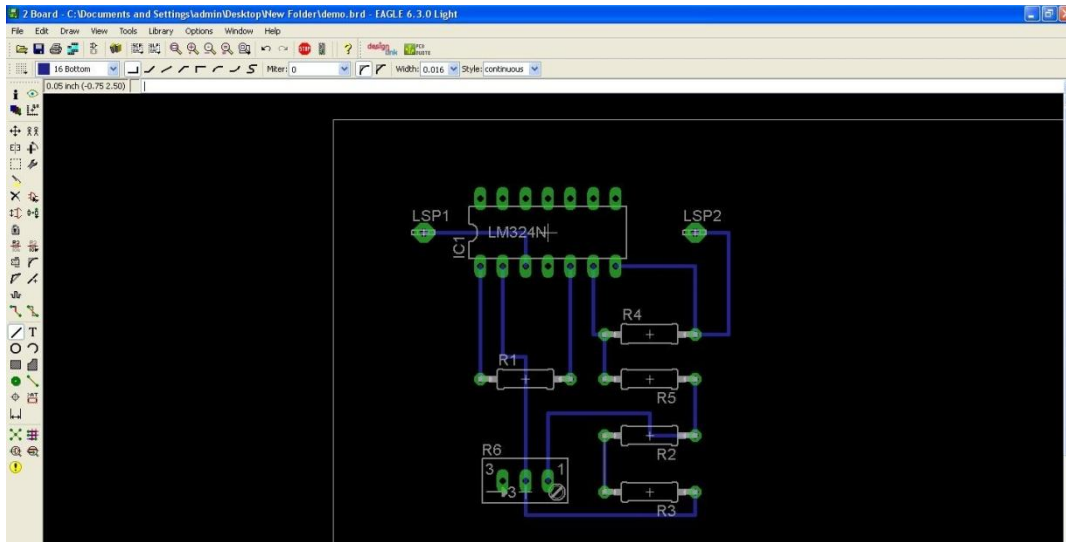
Then, move the element to the designated board area and adjust them so that traces can be run between elements without crossover:



Then click the wire bottom in the picture to wire the traces according to the signal wires on the board. Once the traces are run from a point to another, the signal wires originally on it will be automatically removed indicates that the signal has been completed.

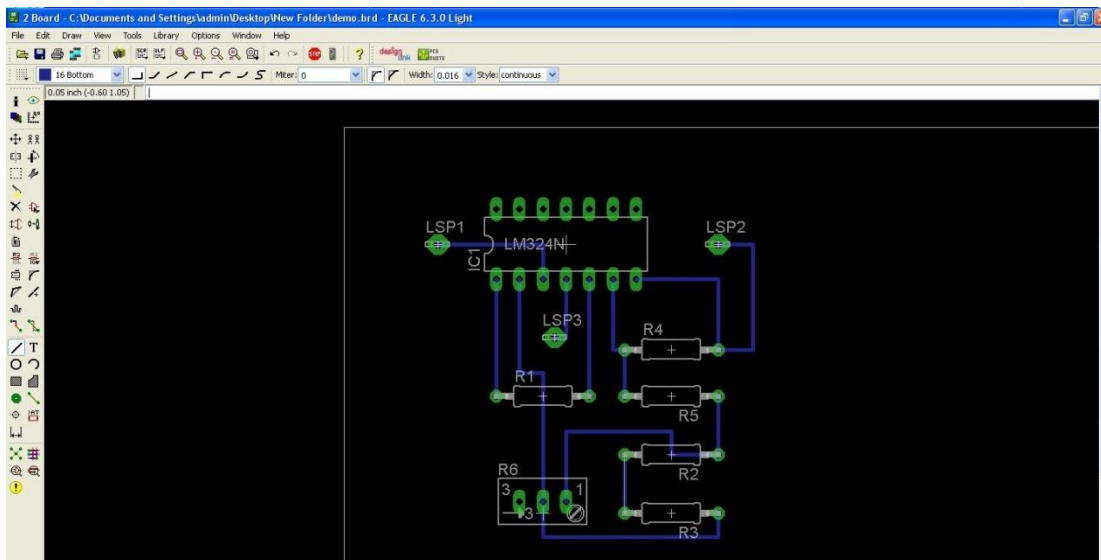
The finished traces should look something like this:

## Design Single-Layer PCB Using Eagle



There are two things to be noticed. First, in the program, because the LM324N was added without power supply pin, the power supply of the chip needs to be added afterwards. Thus, copy and paste a soldering pad as the VCC connector. Then connect the soldering pad to pin 5 of LM324N. Second, the ground was automatically connected to the ground plate. However, the purpose of this tutorial is to show how to design a single layer PCB which doesn't provide a ground plate. In this case, record the connections to be grounded and solder the connections to the copper plate and the copper plate will serve as the ground.

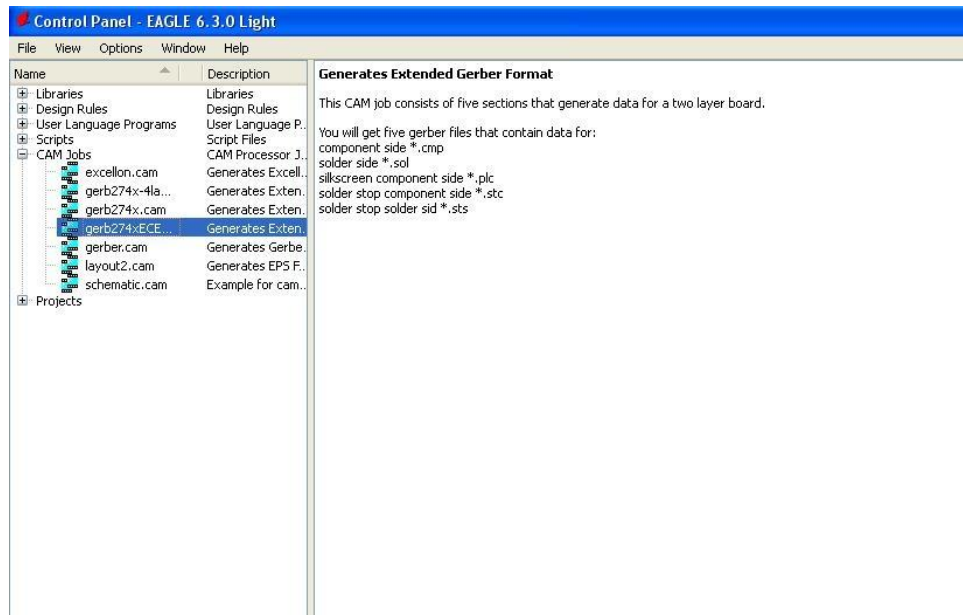
The finished PCB layout will look like:



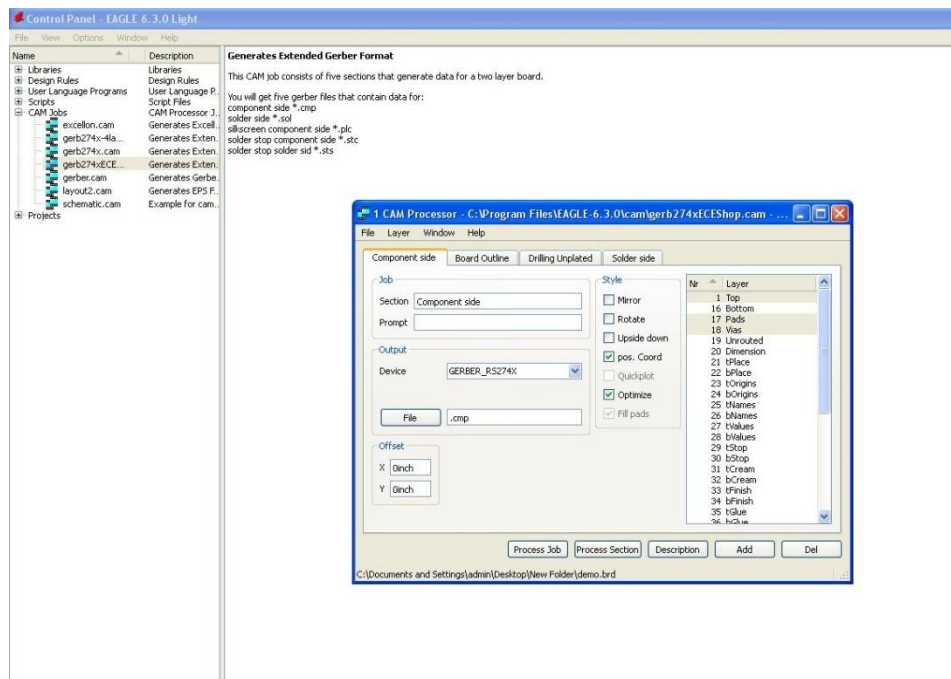
## Preparing for Fabrication with CAM

On the Control Panel, choose gerb274xECESHOP from the drop-down list of the CAM Jobs:

## Design Single-Layer PCB Using Eagle



That will open a window. Select File>Open>Board. Choose the board to be fabricated and click OK. Then on the following screen, click Process Job:



Five Gerber files will be generated in the same directory in which the board file is. They should have extensions of .bol .cmp, .gbr, .gpi and .sol. The five files are what is needed for the PCB to be fabricated.

## **Conclusion**

After read the tutorial, the readers should know how to create a schematic for the PCB to be fabricated, transform the schematic into a PCB layout and generate the CAM files necessary for fabrication. Though the demonstration and the intension of the tutorial is to build a single-layer PCB, the techniques are very similar when readers moving to multiple-layer PCBs.