Circuit Design and Printed Circuit Board Layout Using EAGLE CAD Software

Connor Boss
11/10/2014

Executive Summary:
This application note will be detailing the steps required to go from a circuit design on paper, to an electronic design of the printed circuit board (PCB) that will realize the designed circuit. Once the electronic design of the PCB is obtained, the files can be sent directly to any board manufacturer, who can then produce a physical PCB from the design files. It is most effective to have the circuit designed on paper first so all necessary calculations can be performed easily, this also helps speed up the software layout process, but it is not a necessity. Once the layout of the circuit is determined, the schematic of the desired circuit needs to be drawn in EAGLE, which is a software package for schematic and PCB design. EAGLE can then translate the schematic into a physical layout of the components with all connections established on a PCB. The user will then need to manually route all traces to satisfy the connection constraints established by the circuit schematic (there is an Autoroute function, detailed in the conclusion, but use with caution). When this has been completed, the PCB design is ready to be manufactured. One example that will be used throughout this tutorial is the design of the circuitry for The Automated Lathe Tool Condition Monitoring Project. This project is being developed for Great Lakes Controls and Engineering through Michigan State University's Capstone Design Course.

Keywords:
Project Folder
Schematic
Printed Circuit Board (PCB)
Board file
Library
EAGLE Control Panel
Trace
Trace Width
Net

The Software:
The software that will be used throughout the design process is called EAGLE. EAGLE is a software package that is widely used for taking a circuit schematic design and realizing the circuit physically on a PCB. EAGLE is available for Windows, Linux and Mac. Throughout this application note, EAGLE Version 7.1.0 for Macintosh will be shown. The software company CAD-soft produces EAGLE and offers a light version which is free to all users. The light version is available for download on their website: http://www.cadsoftusa.com/download-eagle/?language=en. The only limitation to the light version of EAGLE is the size of PCB that can be designed.

Step 1: Creating a New Project:
When EAGLE is opened, the screen in Figure 1 will appear. A project is the collection of files that make up the schematic and PCB (or board) layout. To create a new project, select the projects folder as shown in Figure 1, and navigate to the menu bar and select: **File -> New -> Project**

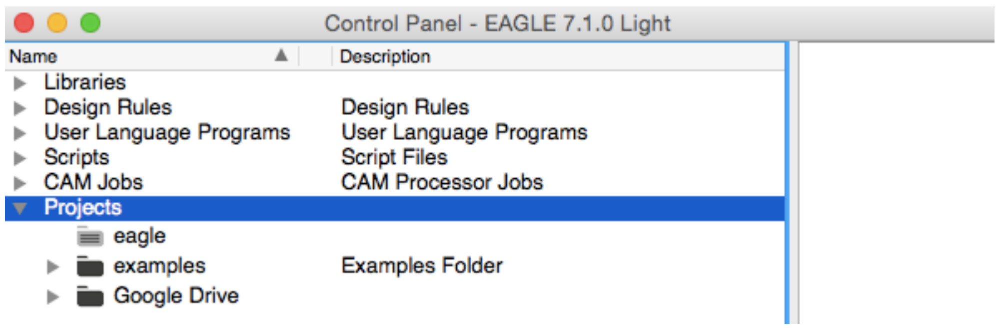

Figure 1: EAGLE Control Panel

Creating a new project will add a "**New_Project**" subfolder within the eagle folder inside the Projects folder. The project name can now be entered as the folder name, in place of "**New_Project**", as shown in Figure 2. The name of the project can be changed at any time by right clicking on the folder and selecting "Rename" from the drop down menu. The new project is automatically selected as the active project as indicated by the green dot next to the new folder name. To select a different project, double-click on the other project's folder to activate the green dot next to it.

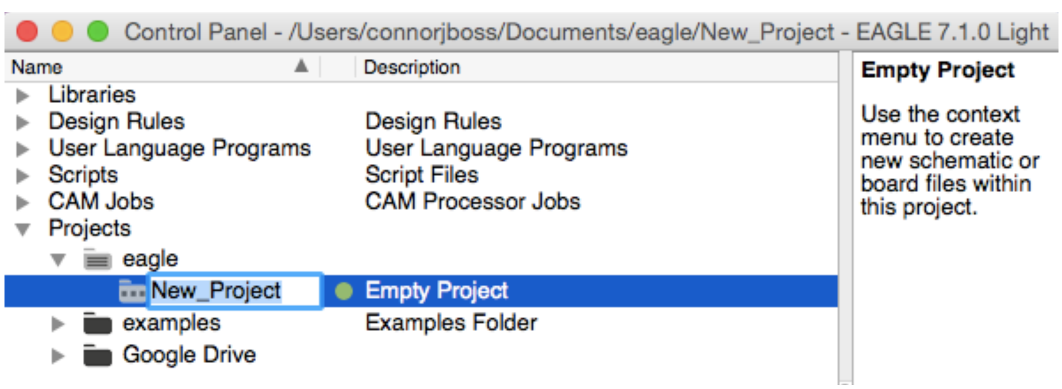

Figure 2: Naming the New Project Subfolder

Step 2: Creating a New Circuit Schematic:
Once the new project has been created, a new circuit schematic can be created inside the selected project folder. To create a new circuit schematic file, navigate to the menu bar and select: **File -> New -> Schematic.** This will bring up a blank workspace where the schematic will be created, as shown in Figure 3.

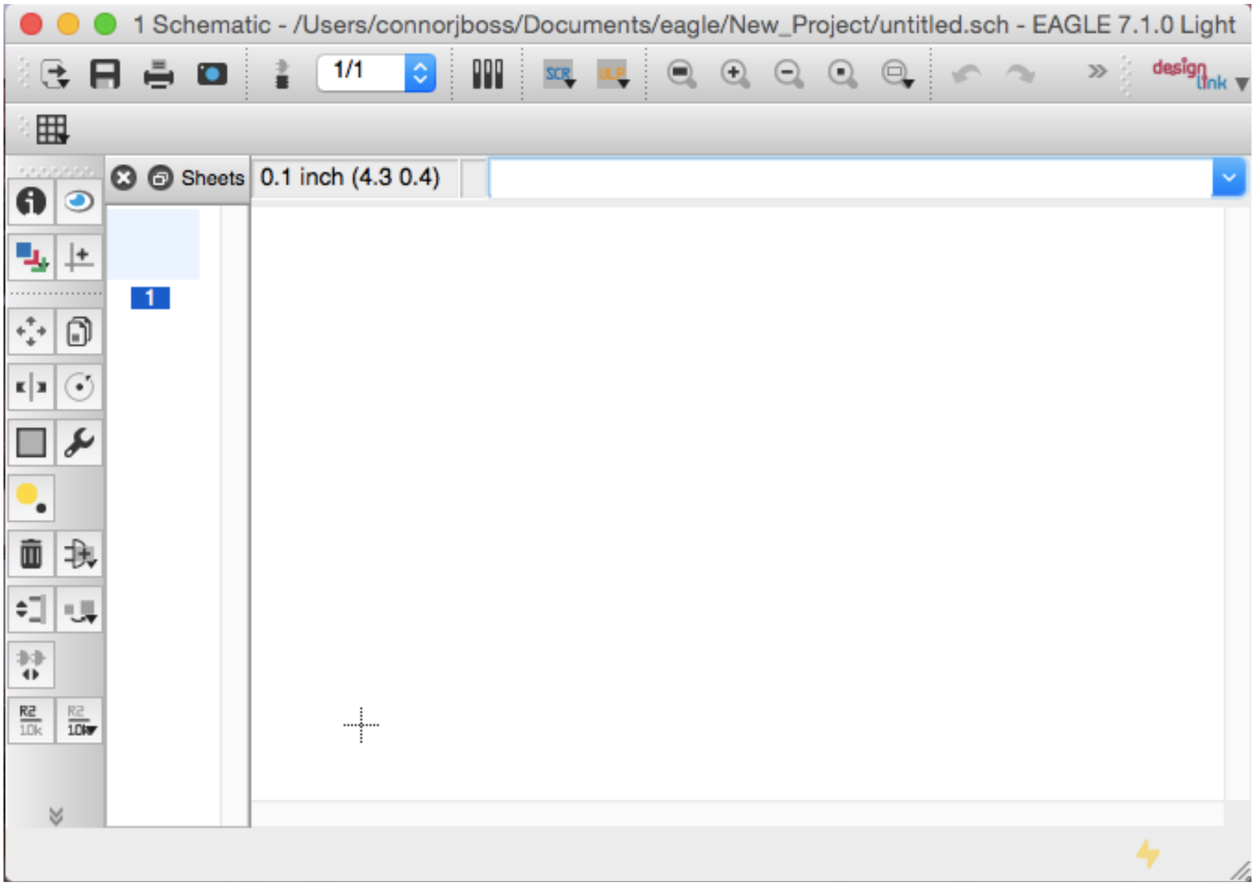

Figure 3: Schematic Editing Workspace

Step 3: Determining What Components Will Be Needed for the Circuit:
Before placing any components, it is a good idea to make sure that any specialty components needed for the schematic will be available in EAGLE. EAGLE stores all of the components that can be used in the schematic layout in Library files. The standard EAGLE Library contains most any generic component that may be required in schematic layout, however if the circuit that is being designed requires specific components, library files must be added. For the Automated Lathe Tool Condition Monitoring Project, a Teensy 3.1 micro-controller is one of the components that is necessary, but is not included in the standard EAGLE Library.

To determine if a component is in the standard EAGLE library, select the

"**Add Component**" button:  and a window will appear showing all available components for the schematic design, as shown in Figure 4. A simple search can narrow down the component list to verify if the desired component is contained within the default library. For the Automated Lathe Tool Condition Monitoring Project, when searching for "Teensy 3.1" it is apparent that this component does not exist in the default library, as shown in Figure 4, the message "Sorry, no match" is relayed.

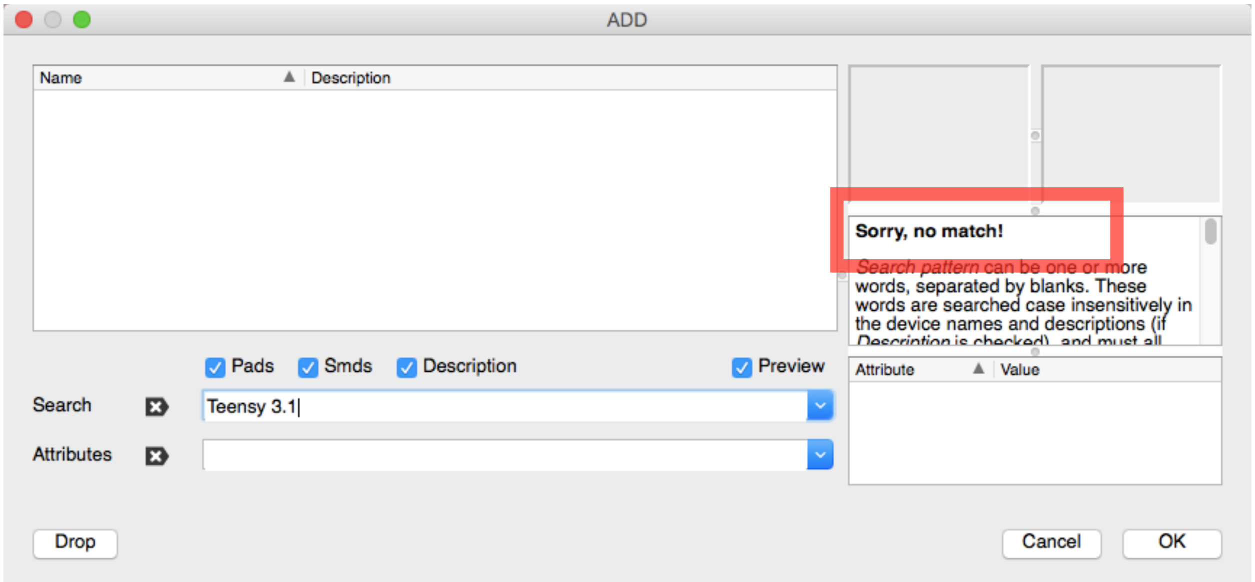

Figure 4: Add Component Window with Search Function

To correct this, a new library file can be added to the EAGLE Library folder. First, a quick internet search for "Teensy 3.1 EAGLE Library File" revealed that a library file has been created for the Teensy 3.1. This library file was then downloaded and added to the EAGLE Libraries folder. The Libraries folder is located in the EAGLE-7.1.0 folder and is named "lib". The new library files will now show up when the Libraries folder is expanded in the EAGLE Control Panel. These new files need to be added to EAGLE's operating library. To do this, right click on the desired library file and select "Use" as shown in Figure 5. The dot next to the library file will turn green, indicating that the library file has been enabled for use in the circuit design.

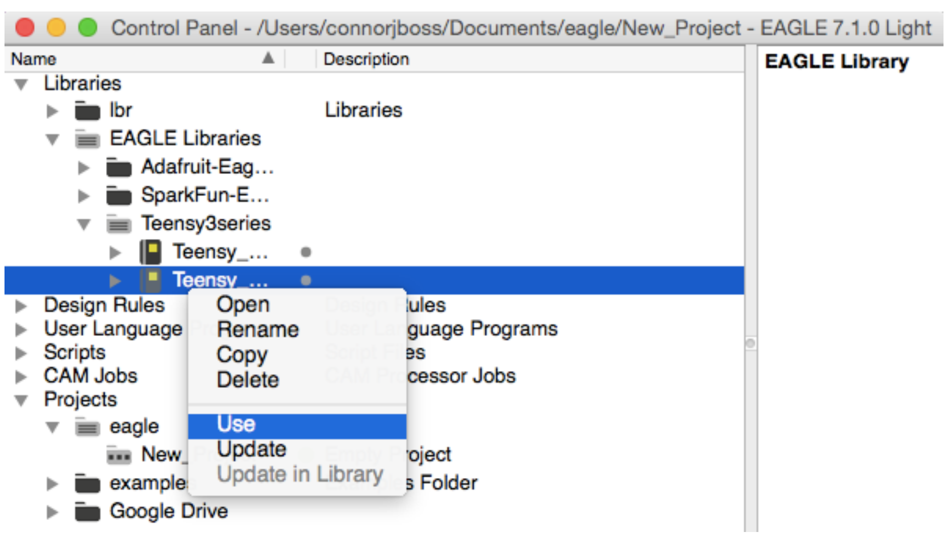

Figure 5: Selecting a Library File for Use

Step 4: Adding Components to Create A Schematic:
Once the desired component is located in the Add Component Window, it can be placed in the schematic building workspace by clicking the "Add" button in the bottom right corner of the window.

Important Note: When selecting a component in the Add Component Window, be careful to look at both the schematic symbol as well as the PCB footprint (shown on right of Figure 6) to ensure that the PCB component matches the physical component that will be used. Be careful to check through hole versus surface mount as well.

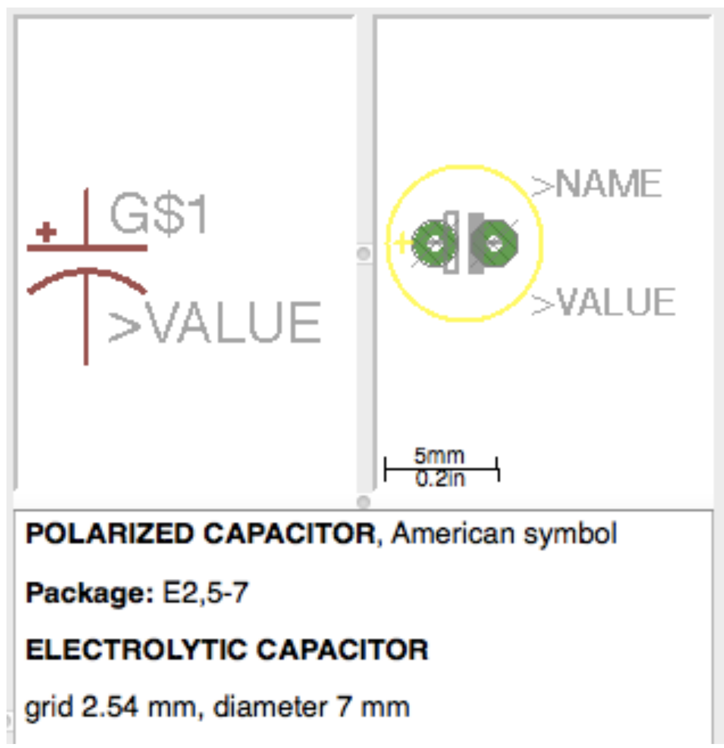

Figure 6: Schematic and Printed Circuit Board Symbols

Now that the proper component is selected it can be added to the schematic. After clicking the "Add" button, the component will follow the cursor around the screen until the mouse is clicked to place the component at that location. Once the component is placed, it will appear as in Figure 7.

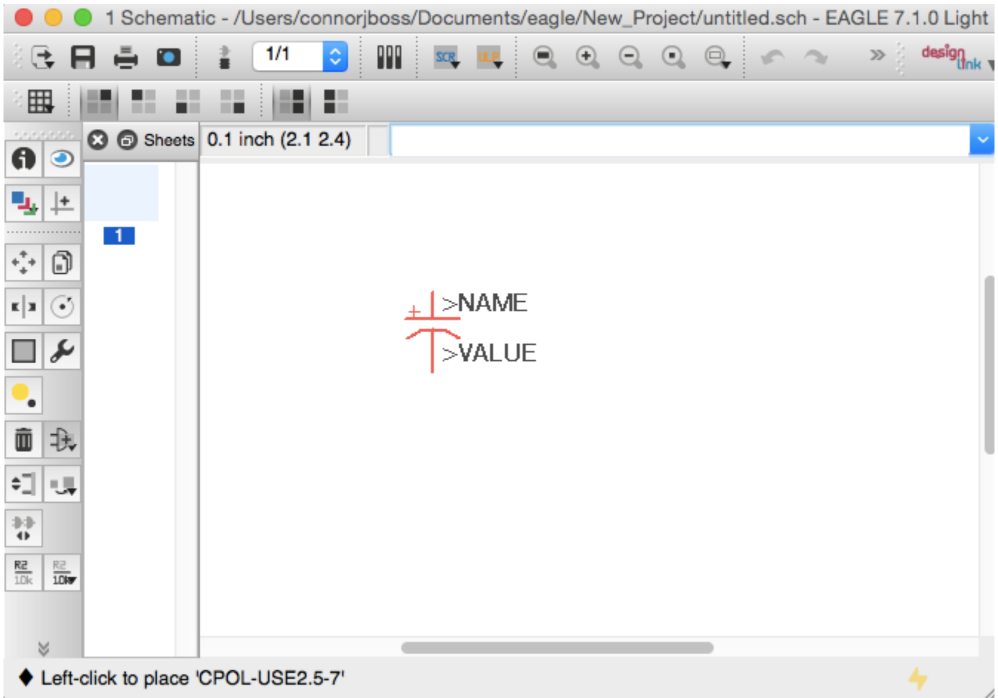

Figure 7: Component Placed in Schematic Workspace

Once a few components are placed, the nets (or the virtual wires that connect the components on the schematic) can be added by clicking on a terminal of a component and dragging to another connection point. The "Net" button, as well as nets being placed, are shown in Figure 8.

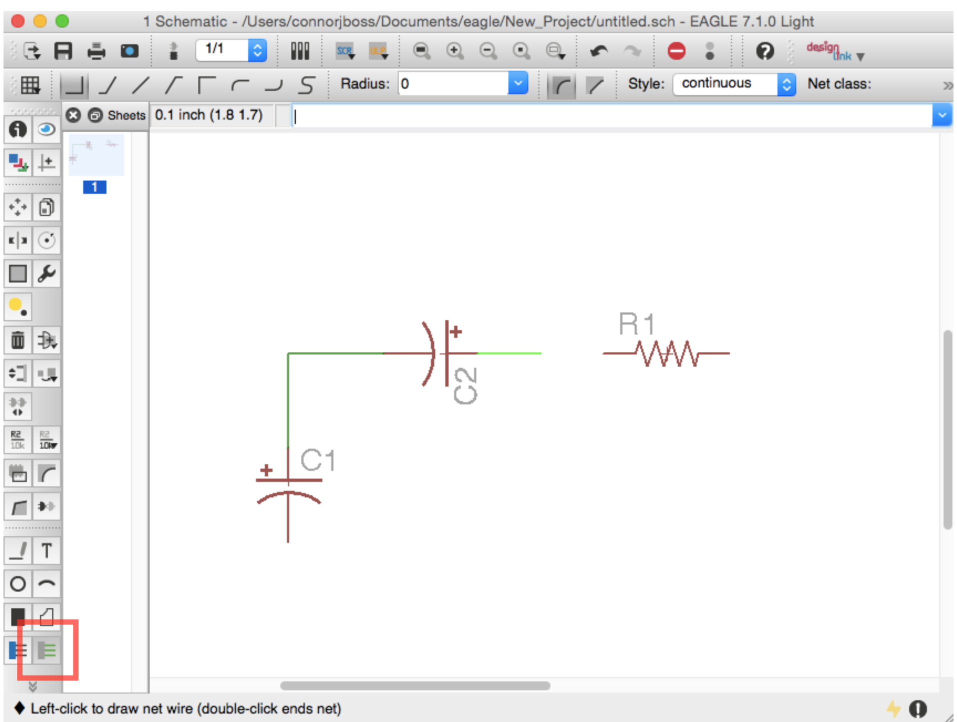

Figure 8: Components With Nets Being Placed

Step 5: Creating a Printed Circuit Board from the Schematic:
After the schematic has been created (a simple example schematic is shown in Figure 9), and double checked for errors, the schematic can be translated into a PCB. This is done by navigating to the menu bar and selecting: **File -> Switch to board.** Doing this will create a new Board file (short for printed circuit board) that will be added to the project. All components from the schematic will be added to the board, as well as yellow lines representing where connections need to be made by routing traces. Using the schematic shown in Figure 9 yields the board file shown in Figure 10.

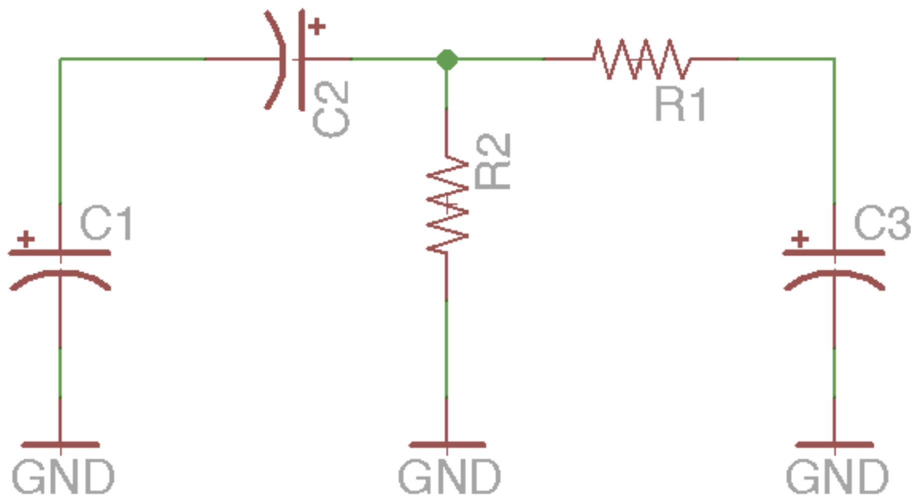

Figure 9: Simple Schematic Layout

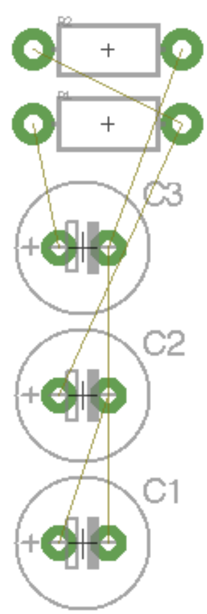

Figure 10: Board File Generated From Schematic

Once the board file is created, the components should be moved around to be placed in an efficient manor with respect to both overall size occupied, as well as having as few of the yellow connection lines crossing as possible. After all components are laid out in an acceptable fashion, traces (the copper wires on a printed circuit board that connect the individual components) need to be added. This is done by using the “Route Manually” button and works the same as adding the nets to the schematic as shown in figure 8. (There is an Autoroute function, as explained in the conclusion, but should be used with caution). Figure 11 shows the “Route Manually” button, and the trace width setting, as well as the traces that have been added to the PCB.

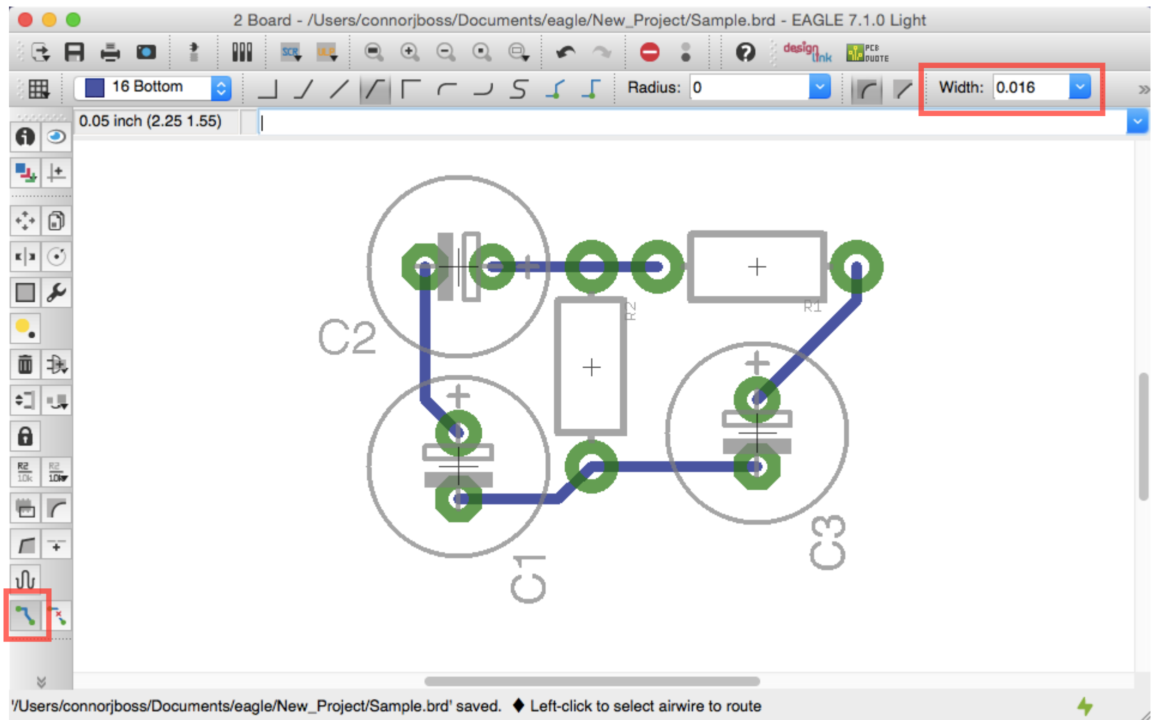

Figure 11: Fully Routed Printed Circuit Board

Now that all of the component placing and trace routing has been completed, the board file is ready to be sent to the manufacturer. Depending on the manufacturer, there may be a certain trace width that they require. The trace width sets how wide each of the copper traces are on the board. A set of typical values are available to choose from in the dropdown menu shown in the upper right corner of Figure 11.

Conclusion:

These are the necessary tools used to go from a sketch of a schematic to a manufacturable printed circuit board. These tools are used in industry as well as hobby applications. The complexity of circuits designed using EAGLE can vary dramatically. The example above is very simple, while The Automated Lathe Tool Condition Monitoring circuitry shown in Figure 12 (Schematic) and Figure 13 (board) is more complex.

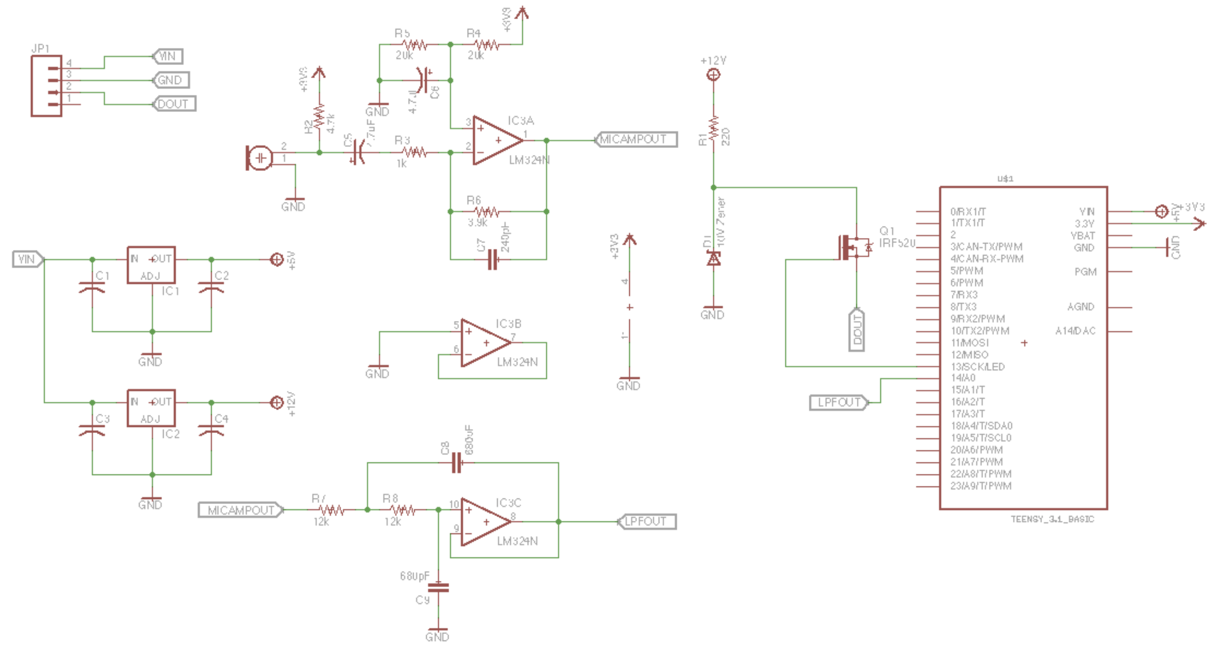

Figure 12: Automated Lathe Tool Condition Monitoring Schematic

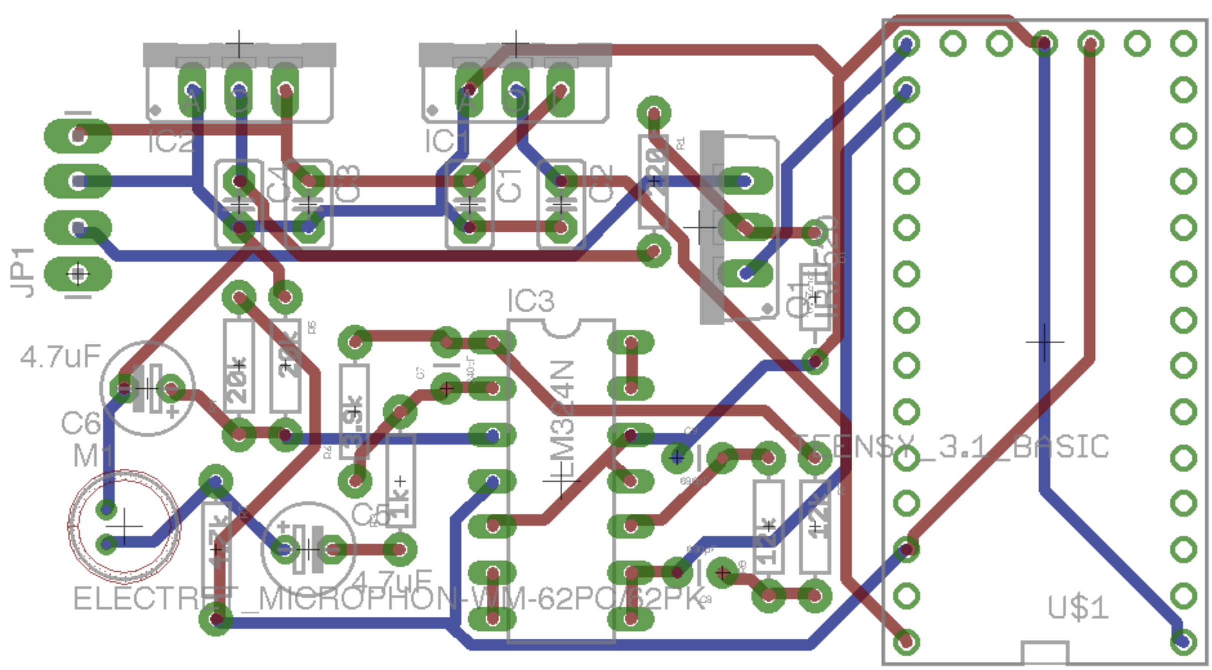

Figure 13: Automate Lathe Tool Condition Monitoring Board

The board in Figure 13 had the traces routed using the Autoroute feature in EAGLE **(Tools -> Autorouter).** It is important to triple check the board against the schematic, if there are any small errors in the schematic, it can be hard to catch if using Autoroute.

Autorouter Function:
Again, the schematic needs to be checked to make sure that all of the net connections are correct because it can be hard to catch on the printed circuit board when Autorouting. Once the Autorouter function has been selected, a window will open where specifications for the Autorouting process can be entered, as shown in Figure 14.

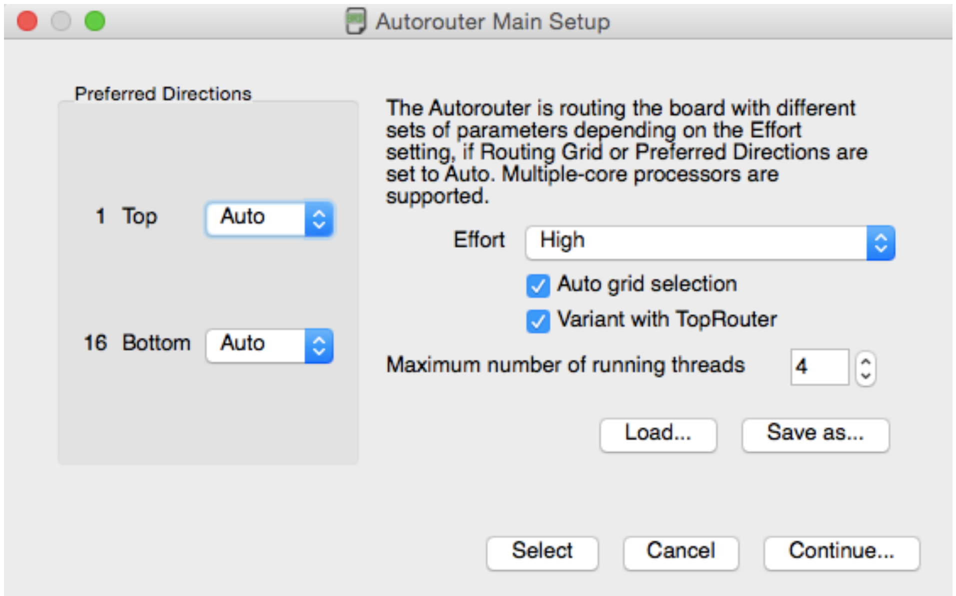

Figure 14: Autorouter Main Setup Menu

Once the effort has been set and the preferred trace layers have been selected, click continue.

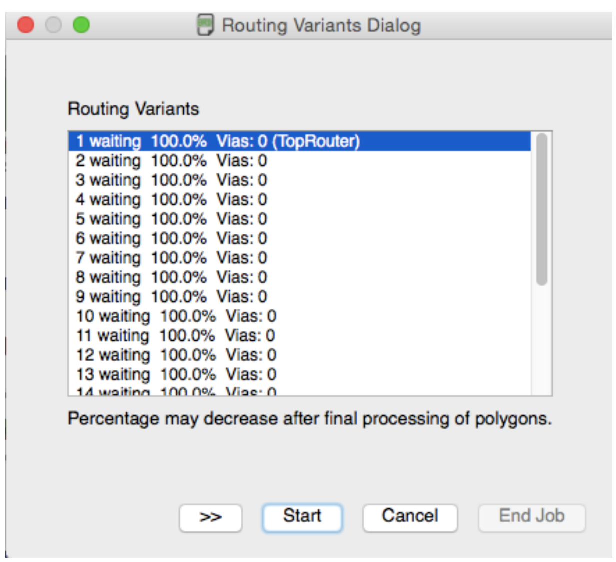

Figure 15: Routing Variants Dialog

Now click start, and the traces will be Autorouted. Double check Autoroute accuracy.

