

Learning EAGLE: A Printed Circuit Board Design Primer

Contents

Motivation and Acknowledgments	iii
Getting to Know EAGLE	1
Creating the Schematic	9
Laying Out the Board	40
Generating Gerber Files	67

Preface

Motivation and Acknowledgments

Electrical and electronics engineering tends to be comprised of a variety of technical tasks, routed in physical or mathematical principles. Printed circuit board (PCB) design, however, contains an artistic element. Laying out a PCB requires an intricate coupling of foundational electrical engineering and artistic prowess. Not only must the design be electrically sound, but it must also be intuitive and user-friendly. A strategically designed PCB allows for a fluid transition from electrical prototype to fully-developed product. PCBs are the core, around which all consumer electronics products are built. As the consumer electronics industry continues to grow, people around the globe are coming up with even more ways to revolutionize our interactions with technology. These individuals utilize a solid understanding of engineering, combined with their drive to make an impact on the world, to plunge into entrepreneurialism. Everyone has great ideas, but a true entrepreneur executes on those ideas. By learning how to translate your core electrical and electronics engineering knowledge into a cohesive printed circuit board design, you are taking one step closer to entrepreneurship in electronics.

Through this tutorial, you will learn the necessary skills to convert a schematic into a tangible board design. You will learn best practices in laying out components on the PCB, as well as how to generate the industry standard file set: Gerber files. After the completion of this primer, you will have the required skills to translate an idea into a fully-fledged electronic prototype, which is the basis for true product development and entrepreneurship.

I would be remiss to neglect to acknowledge the individuals who contributed to this primer. I owe a huge debt of gratitude to Patrick Cutno for providing substantial edits to the various document drafts. Thank you to Dr. Qihou Zhou and Dr. Chi-Hao Cheng for greenlighting the development of this document. I hope that it serves to spur your interest in PCB development and consumer electronics entrepreneurship.

– Tyler Stephen Maschino

Chapter 1

Getting to Know EAGLE

First, navigate to <http://www.cadsoftusa.com/download-eagle/> in your web browser to download EAGLE. This software is going to be used to help us design and lay out our printed circuit boards (PCBs). Be sure to select the correct version for your target computer. Once the software has been downloaded, follow the prompts to install it on your computer. After the installation is complete, open the software. The first screen you will be presented with in EAGLE is called the Control Panel. This is where all of your libraries, documentation, and projects will be listed.

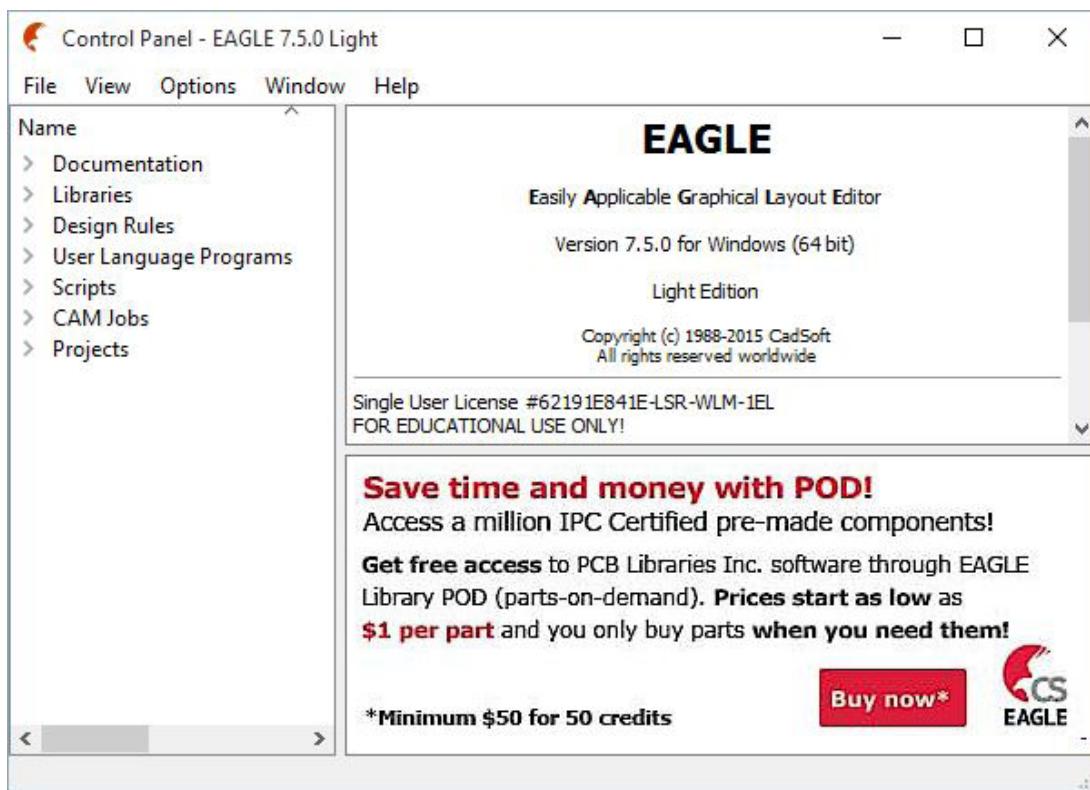


Figure 1: EAGLE Control Panel

Next, we are going to create a new project. This can be done by clicking File > New > Project in the toolbar. As soon as you click on Project, the Projects sidebar in the control panel will expand and reveal a new folder called “eagle”. This is the default folder for projects you create in EAGLE.

New project directories can be added and linked through the main toolbar by clicking Options > Directories... and then clicking on the textbox corresponding to the Projects directory. You then can browse to add whatever folder you would like to add to your filespace. You'll notice several project directories in the following image.

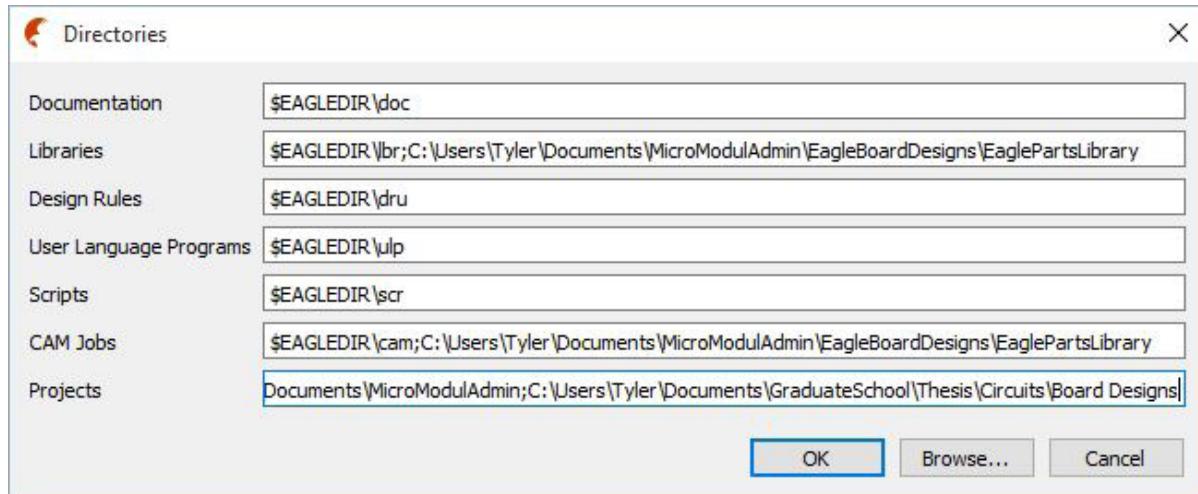


Figure 2: Selecting directories

Back in the Control Panel window, you'll notice that your new project was named New_Project, by default. Feel free to change this by right-clicking the new project and selecting "Rename".

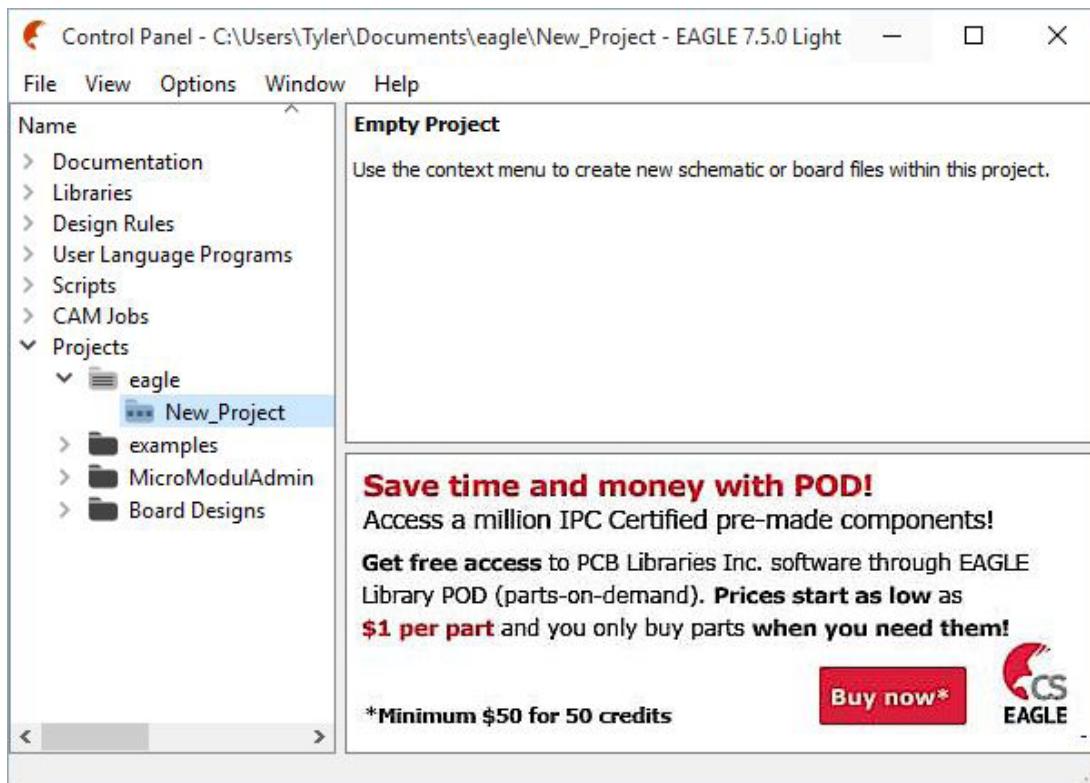


Figure 3: Creating a new project

We will be naming ours “MiamiM” because of the circuit we are going to be designing. Once you’ve renamed the project, right click on your new project. You’ll see several options here. Hover over the word “New” and click Schematic.

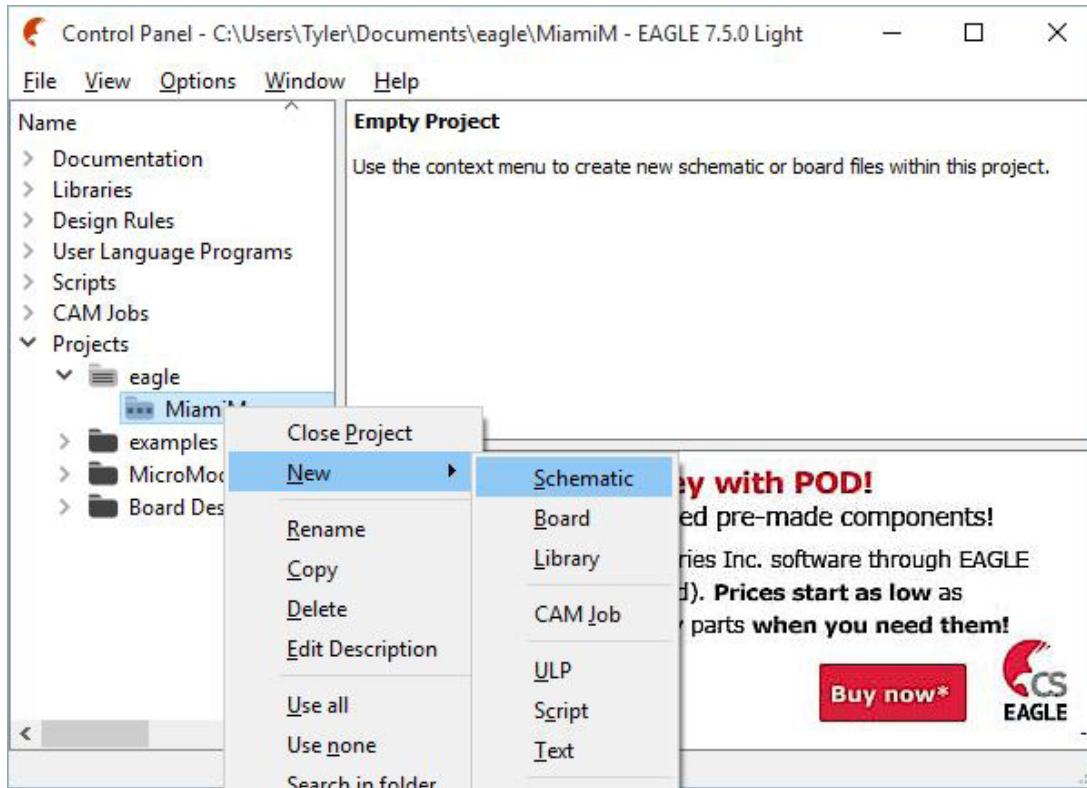


Figure 4: Creating a new schematic

As soon as you’ve clicked “Schematic”, a new window will appear. Using this window, we will lay out our entire circuit schematic prior to laying out the board. Before moving on, be sure to save this schematic. The default name is `untitled.sch`. It is good practice to name the schematic the same as the project name.

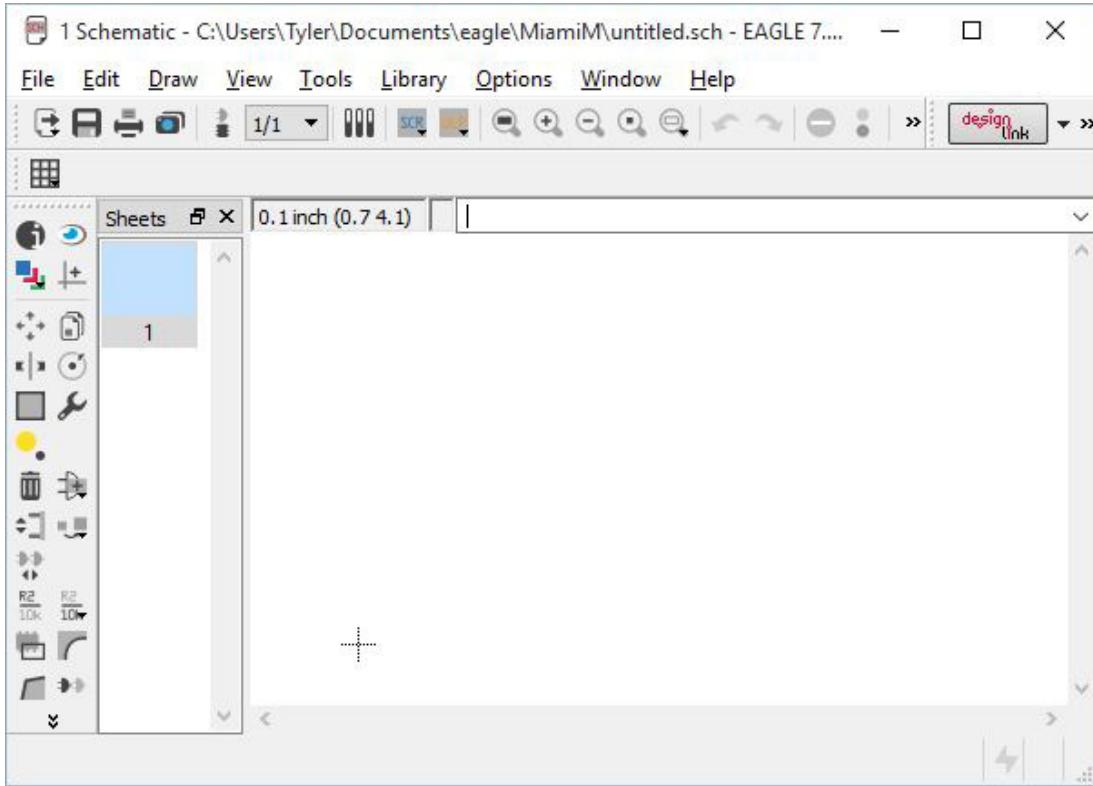


Figure 5: Blank schematic sheet

Before we wire the schematic, you will need to install a few libraries. Libraries are special EAGLE files that can be created in, or imported into, the software. These libraries can contain many different parts, in both schematic and board formats. This allows users to easily choose from a variety of ready-made parts, once they have imported the library into EAGLE. The two libraries we are going to use for this tutorial are the Maxim Integrated and SparkFun Eagle Libraries. The Maxim Integrated library can be downloaded here: <http://www.element14.com/community/servlet/JiveServlet/download/38-101091>. The SparkFun libraries can be downloaded from this link: <https://github.com/sparkfun/SparkFun-Eagle-Libraries/archive/master.zip>. Once you've downloaded and unzipped the libraries, you will need to include them in the current project. The quickest way to do this is to click on Library > Use... in the toolbar at the top of the schematic window.

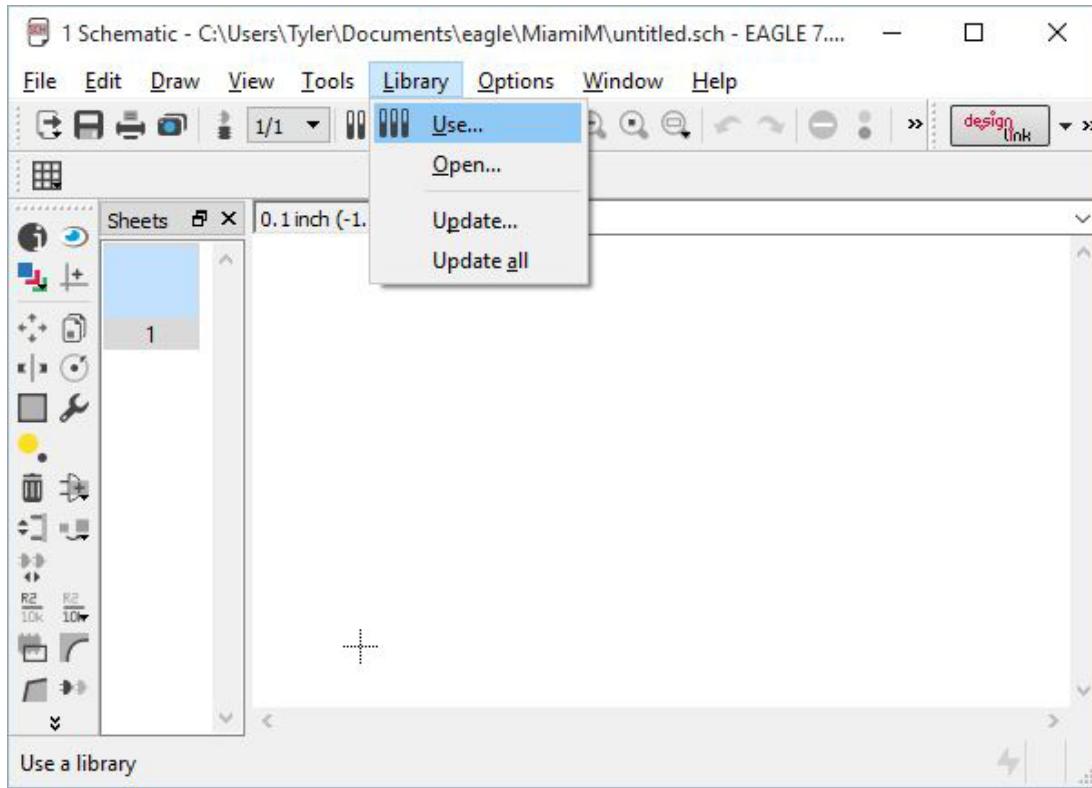


Figure 6: Using a library

This action will open a file browser, which you can use to navigate to the location of the Maxim Integrated and SparkFun Eagle Libraries. Within each extracted folder, you will find files ending with the extension “.lbr”. Select all of the files and click “Open”. Be sure to do this for both downloaded libraries.

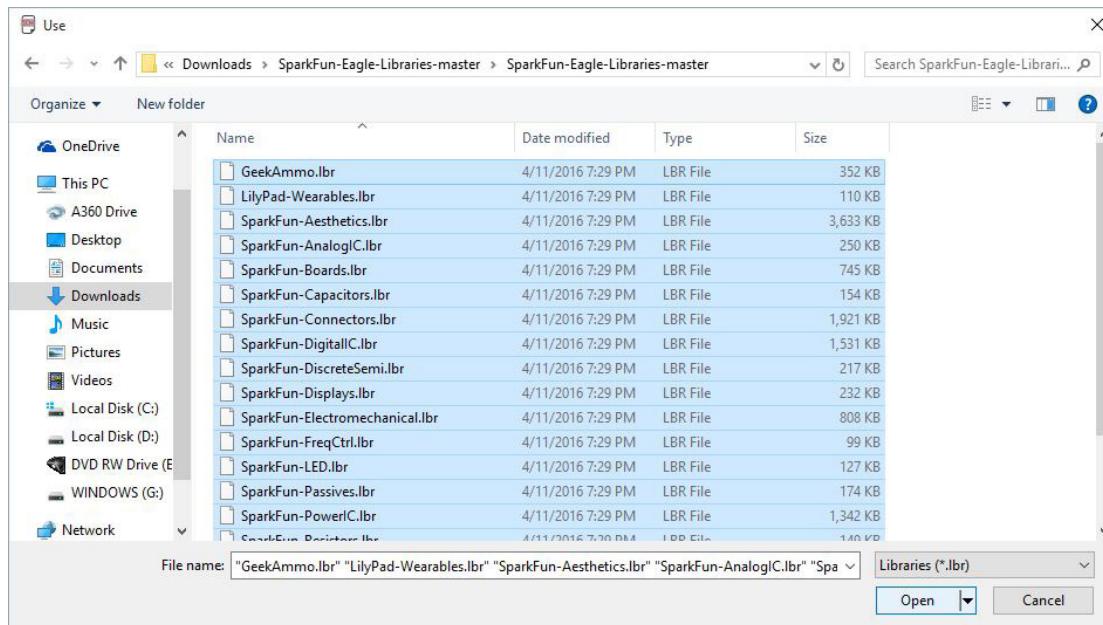


Figure 7: Opening libraries

It will look like nothing happened but, in the background, EAGLE is now using all of those libraries.

Since we have all of the libraries we need for this board design, let's take a look at the user interface in the Schematic window. You should see two toolbars; one is on the left of the window, the other is toward the top of the window. Looking at the left toolbar first, we are going to highlight a several important functions:

 **GRID** – Opens the Grid window, which allows the user to turn on and off the grid, as well as set the grid size

 **INFO** – Enables the user to click on a component in the sheet and open its Properties window

 **SHOW** – Displays objects that share the same name, but may not be visibly connected in the sheet; this is particularly useful for viewing nets

 **LAYER SETTINGS** – Opens the Display window, which allows layers to be enabled or disabled, at the user's discretion

 **MOVE** – Allows components to be moved around the sheet, when clicked

 **COPY** – Copies the selected component for placement elsewhere in the sheet; the copied component will follow the mouse around the screen until the ESC key on the keyboard is pressed

 **MIRROR** – Flips the component along its y-axis

 **ROTATE** – Rotates the component by 90 degrees on every click

 **GROUP** – Used to select multiple components at one time; be sure to select your desired tool before defining the group

 **CHANGE** – Allows the user to change a desired property with one click; first, select your change, then click on every component you'd like to change

 **INSERT** – Used in conjunction with the Copy tool and can be used between EAGLE schematics; select the group you'd like to insert, click on the Copy tool, then click on the Insert tool to insert the group



TRASH – Deletes components that are clicked on after selecting this tool



ADD – Opens the ADD window to search the included libraries and add components to the schematic



REPLACE – Quickly swap out components with similar pin diagrams; very useful when upgrading schematics with new parts



NAME – Allows the user to change the name of a component; can also be used to connect components without needing to draw connections in the schematic



VALUE – Allows the user to set the value of a component



SMASH – Used to separate the name and value graphics from the component itself; useful when rearranging components in a tight area



WIRE – Used to draw lines on the sheet; this is not the tool you are looking for if you want to electrically connect components



TEXT – Opens the Text window, which allows the user to write words or phrases and place them on the sheet



NET – Allows the user to electrically connect components



LABEL – Used to place a text label on a component; the label displays the name of the component



ERC – The Electrical Rule Check verifies that the schematic is free of common problems, such as trace overlap

There is another toolbar along the top of the schematic window. This toolbar contains the following useful tools:

-  **OPEN** – Opens a file browser to look for a file to open
-  **SAVE** – Saves the current file
-  **PRINT** – Prints the current file
-  **GENERATE/SWITCH TO BOARD** – Switches to the board layout file
-  **USE LIBRARY** – Opens a file browser to look for library files to use in the project
-  **ZOOM TO FIT** – Zooms the screen to encompass all components
-  **ZOOM IN** – Zoom in on the current center of the sheet
-  **ZOOM OUT** – Zoom out from current center of the sheet
-  **ZOOM SELECT** – Zooms in on a region; region is selected by clicking and dragging the mouse
-  **UNDO** – Undoes the previous action
-  **REDO** – Redoes the previous action

We are going to use many of these buttons intermittently throughout this tutorial.

Chapter 2

Creating the Schematic

Click on the Add tool. When you click the Add button, a new window will pop up. If you correctly added the libraries outlined in the previous chapter, you will find a series of folders labeled “SparkFun-Library_Name”. We can manually look through these folders for the parts we want, or we can use the search box in the bottom of the window.

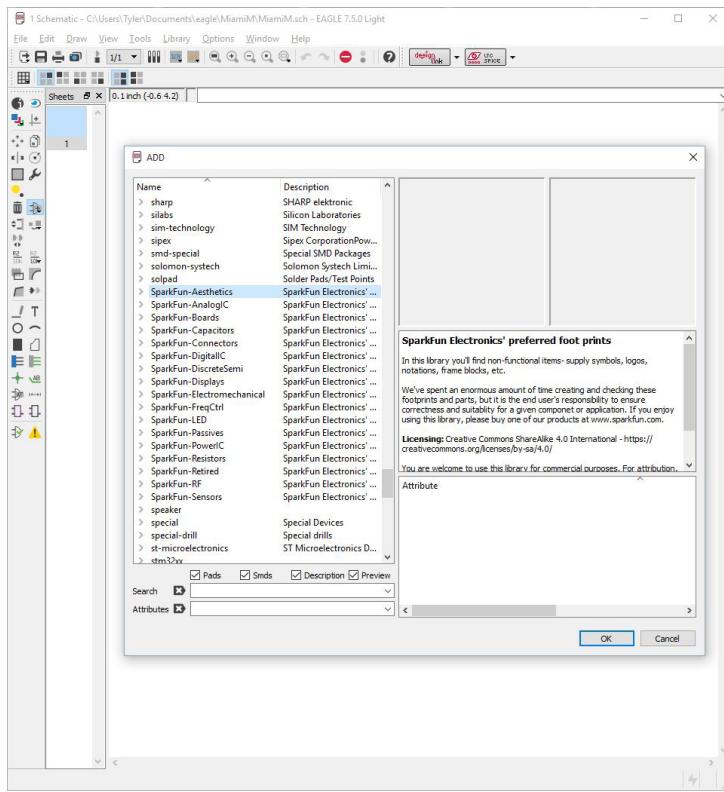


Figure 8: Opening the ADD window

The search box is the easiest way to find the parts you’re looking for, especially as you begin to add more libraries into EAGLE. Click on the search box and type “*LED*”, then press Enter on your keyboard. The “*” symbols act as wildcards, which tell the search box that you don’t care what comes before the first, or after the second, “*” symbol. By including it at the beginning and the end of the word “LED”, we are telling the system to find any parts that contain the word “LED” anywhere in their names or descriptions.

You'll notice that once you searched for the term, several folders expanded and you can see many more parts listed. Scroll down until you see the SparkFun libraries again. One of the libraries that you added is a library entirely dedicated to LEDs, aptly named "SparkFun-LED". This library will be expanded already and you will see there are subfolders containing different types of components. Click on the arrow next to the "LED" subfolder to expand it further.

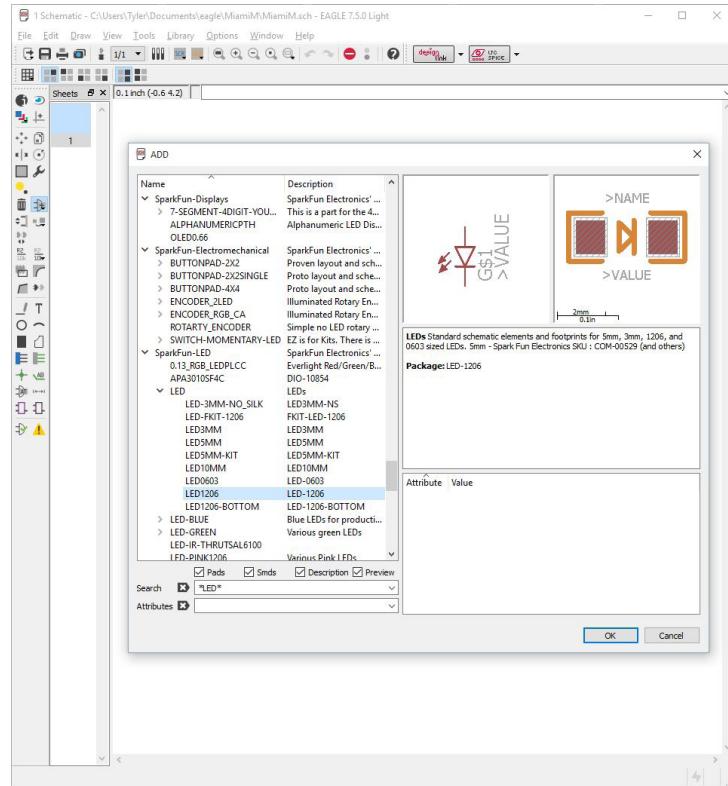


Figure 9: Searching for an LED

There are multiple components listed within the "LED" subfolder. We know they are components because they do not have the arrow next to them, which indicates a folder, and they have a PCB footprint in the upper, right-hand side of the ADD window. Toward the end of the list of components, click on the one named "LED1206". The "1206" part of the name indicates the size of the component. This is a surface-mount component, meaning that it will be soldered directly to the top or bottom of a printed circuit board (PCB), rather than having leads that go through the board to hold it in place. Surface mount devices (SMDs) are a very popular type of component because of their small footprint, but they can be much more difficult to solder by hand. The 1206 package is on the larger end of the surface mount spectrum, so an individual with decent soldering skills shouldn't have much of an issue.

After clicking on LED1206, a new picture appeared in the, formerly gray, window in the top, right corner of the ADD window. This image shows what the footprint of the LED will look like when it is placed on the board. The image directly to the left of the PCB footprint is the schematic symbol for the LED, so that's how it will look when we place this component on the page. Make sure LED1206 is still selected, then click the OK button in the bottom, right corner of the window.

The ADD window has disappeared and the LED symbol mentioned above is now following your mouse around the page. If you right-click using your mouse, you can rotate the LED. Do this until the LED is oriented with the anode (source of current) pointing toward the top of the page and the cathode (departure of current) pointing toward the bottom of the page, as shown in the image below. Another useful button, for users with a mouse wheel, is the center wheel button. If you click the mouse wheel, the LED will be mirrored along the y-axis of the page. If you click and drag, using the mouse wheel, you can move the schematic around without placing the LED. This can be very useful if you need to move to a different part of your schematic without placing the component. When you're ready, left-click on the page to place your first component.

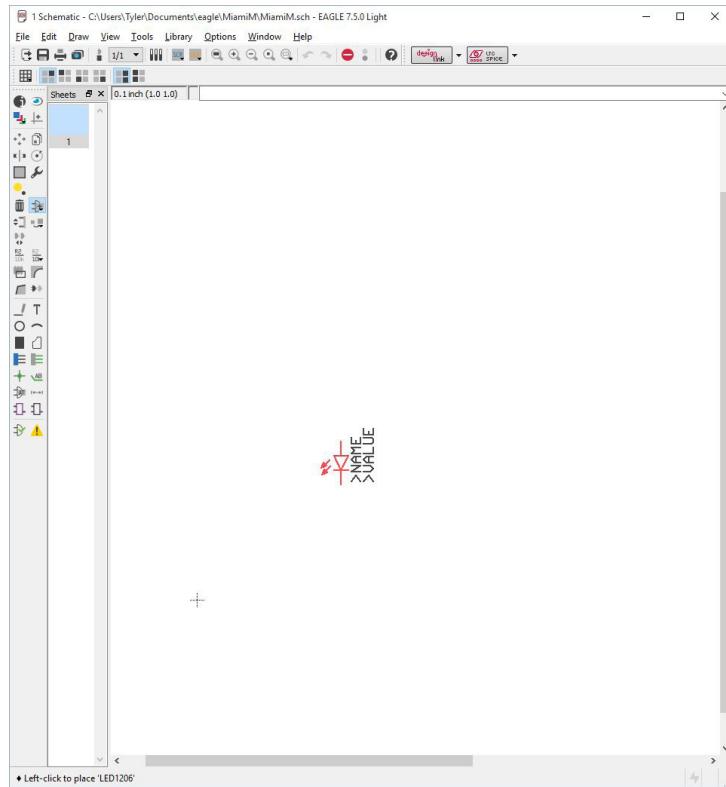


Figure 10: Placing an LED

The component will automatically be given a default name. You can adjust this name using the Name tool in the toolbar. We aren't going to worry about changing the names right now, since we are planning to add several LEDs. After the first component is placed, a new one will appear to be glued to your mouse again. EAGLE will allow you to continue placing components.

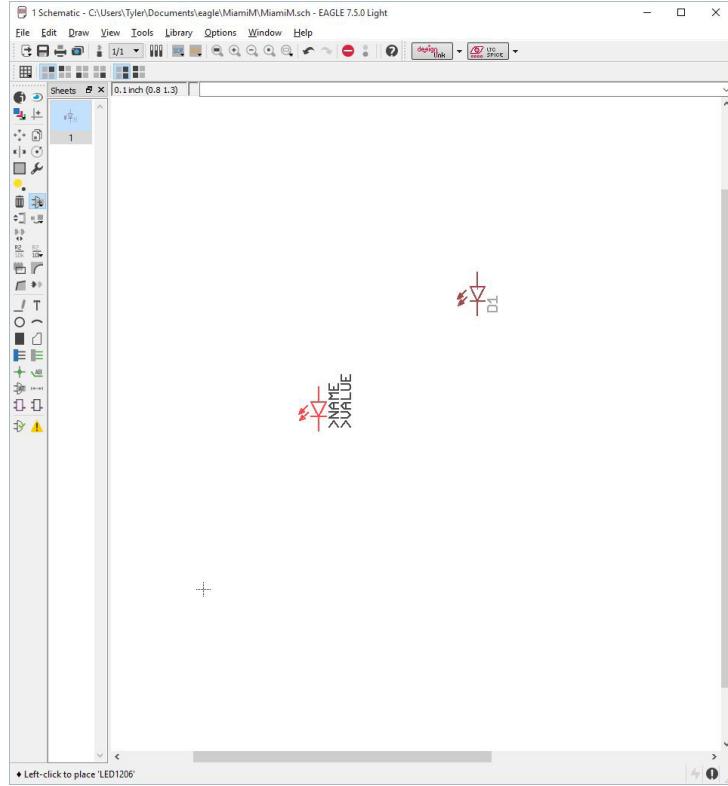


Figure 11: Placing multiple components

Add seven more LEDs next to the first LED we placed. When you've completed the row, move down a bit and add another eight LEDs in a row. Repeat this until you have up to 64 LEDs in an eight by eight grid. If you don't want to place 64 LEDs, feel free to use fewer. The integrated circuit (IC) we will be using to control the LEDs can handle up to 64 LEDs. Since we are designing an "M" in the shape of the Miami University logo, this tutorial will only show 35 LEDs. When you have finished placing the LEDs, press the ESC key on your keyboard. When you press ESC, the ADD window will reappear. Click the Cancel button to close the ADD window.

Now that we have placed several components, it is a good idea to save the project. As we progress through this tutorial, remember to save when we make additions and changes to the project.

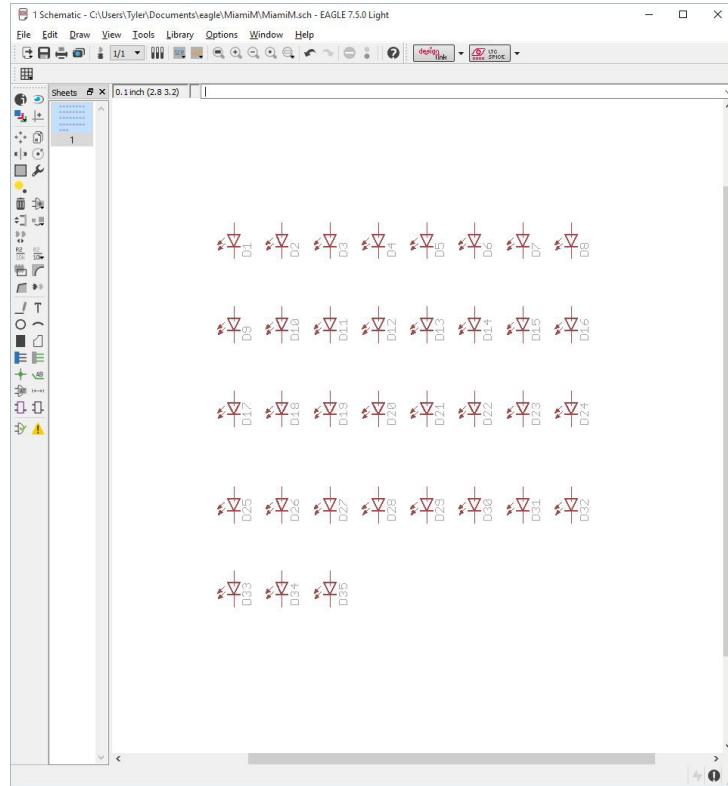


Figure 12: All LEDs placed in the schematic

Next, we are going to wire the LEDs together. All of the anodes in each column should be wired together and all of the cathodes in each row should be wired together. To connect components in EAGLE, you need to use the Net tool. Make sure that you are not using the Wire tool. This function does not connect the components together. The Wire tool should only be used for artistic purposes, as the Wire tool only draws lines and will not electrically connect components.

After clicking on the Net button, you can now left-click on the page to add a connection between components. Starting with the top-left LED (D1), click on its anode. Be sure to click as close to the end of the anode as possible. Once you click on the sheet, a bright green connection will follow your mouse around. The next time you click on the sheet, the bright green connection will become dark green, indicating the connection has been placed on the sheet. The place where you clicked then becomes the anchor point for future connections and a new bright green line will follow your mouse around from that point. When you are finished with the Net tool, click the ESC key on your keyboard to stop wiring and terminate the connection at the most recent anchor point.

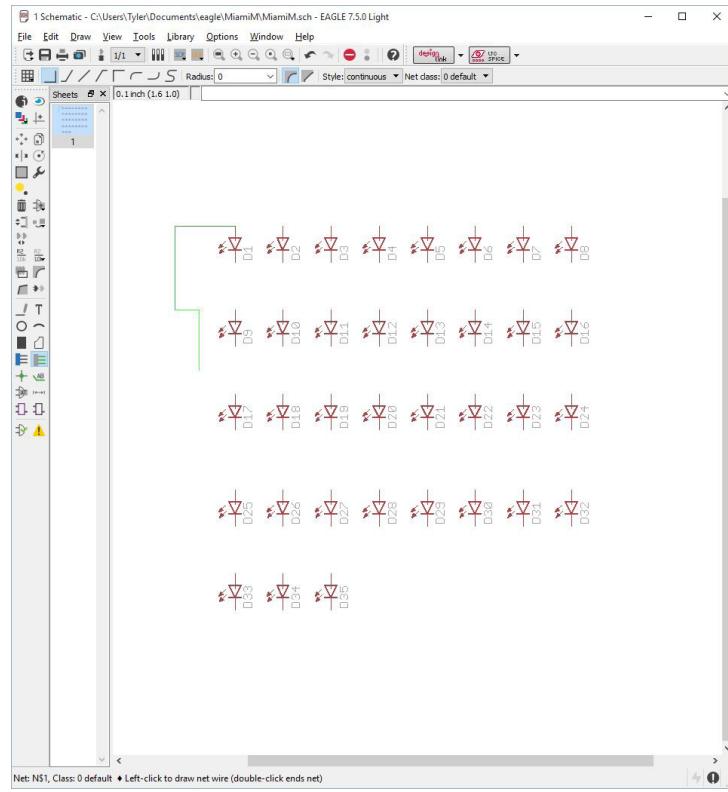


Figure 13: Connecting components with the Net tool

Using the Net tool, connect the anodes in each column and the cathodes in each row together. The first row and column are shown in the following image.

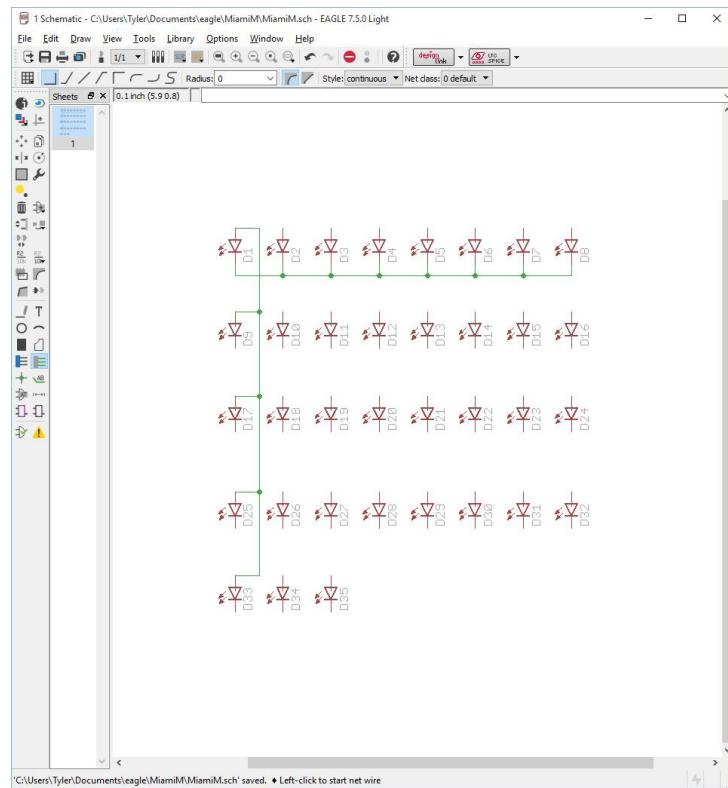


Figure 14: First row and first column connections

When we connect multiple components to a single net, a dot appears on the net. This is to indicate a connection at an intersection, rather than a non-connected overlap of nets. The image below depicts instances of connected and non-connected intersections of nets, respectively.



Figure 15: Connected (left) and non-connected (right) nets

Once you've connected the components properly, you will have a grid of LEDs that resembles the following image.

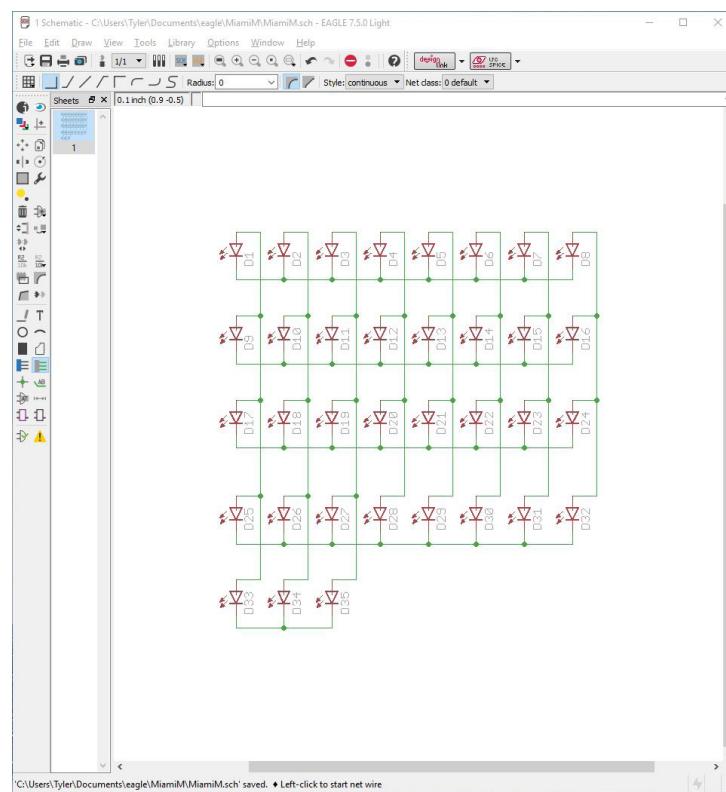


Figure 16: Final LED grid

Now that we have wired all of the LEDs properly, we are going to add the LED driver. The driver, Maxim Integrated's MAX7219, is the next component we will be incorporating into our schematic.

Click on the ADD button to open the ADD window. Type “*MAX7219*” in the Search textbox. Press the Enter key on your keyboard. You will see several options for parts. We want the surface-mount technology (SMT) part, which is “MAX7219CWG+”. When you click on the part, you can tell that it is a surface-mount package by looking at the top, right image and noting there are no drill holes in the device. If you click on SparkFun’s “MAX7219DIP” part instead, you’ll notice the package looks different; there are many circular drill holes in the image. If you want to choose the through-hole version of this IC, go ahead and click OK with this part selected. Otherwise, click on the SMD part “MAX7219CWG+” and click OK.

Having selected your part, you will see a large rectangular box with many pins following your mouse around. Click wherever you would like to place the component. When you’ve placed the part, you will see a block diagram representing the MAX7219 IC. In the center of this block, there is a small cross; this location is the origin of the chip. If you ever need to move a component after placing it, use the Move tool and click on the origin to select the part.

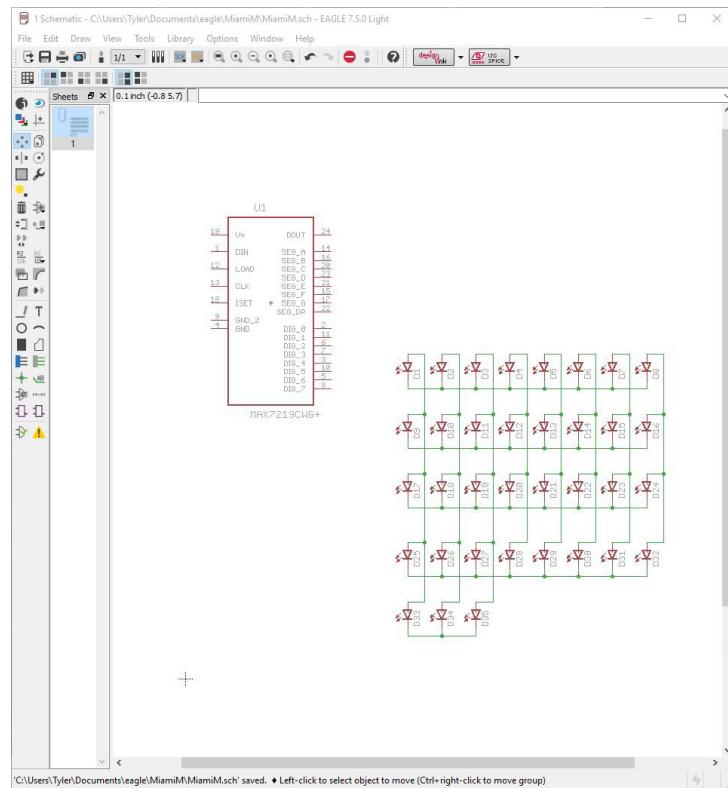


Figure 17: Placing the LED driver IC

Next, we are going to connect the V+ pin on the IC block diagram to a VCC component. Open the ADD window and search for “*VCC*”. Click on SparkFun’s VCC part and press OK.

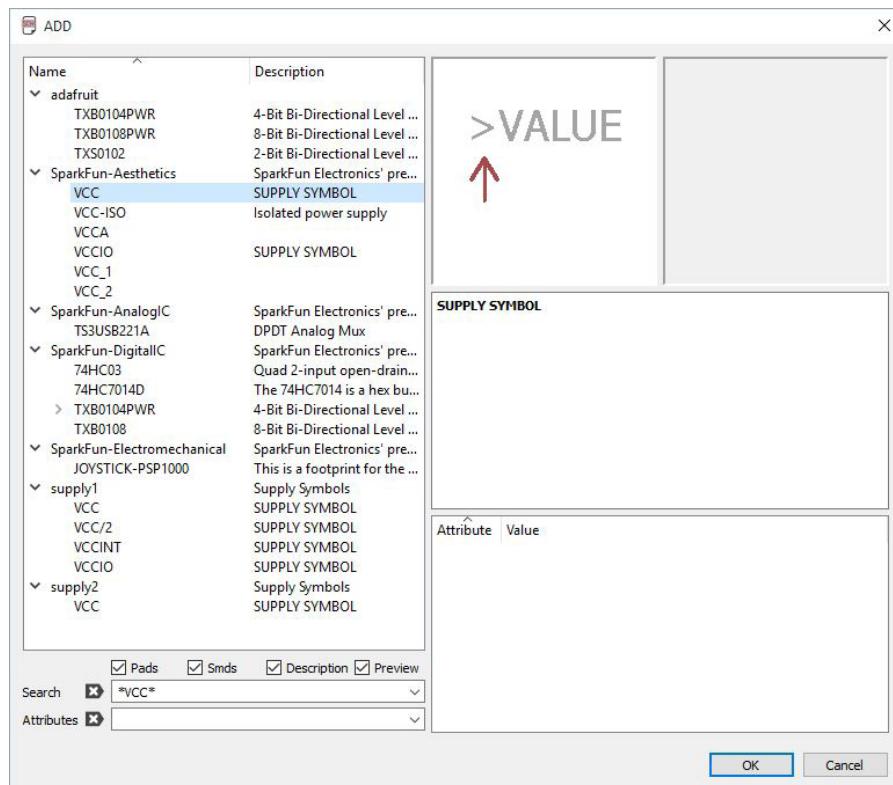


Figure 18: Adding a VCC component

Place the part near the V+ pin. Then, click on the Net button in the toolbar. Draw a net out from the V+ pin to the VCC part you just placed. It is good practice to place components, then wire them, rather than trying to place a component on an already existent net.

Now, if you rename any net in your schematic to “VCC”, using the Name tool, it will automatically be connected to all of the other nets named “VCC”. This is a very useful function to prevent designs from becoming hard to interpret because of the vast amount of connections.

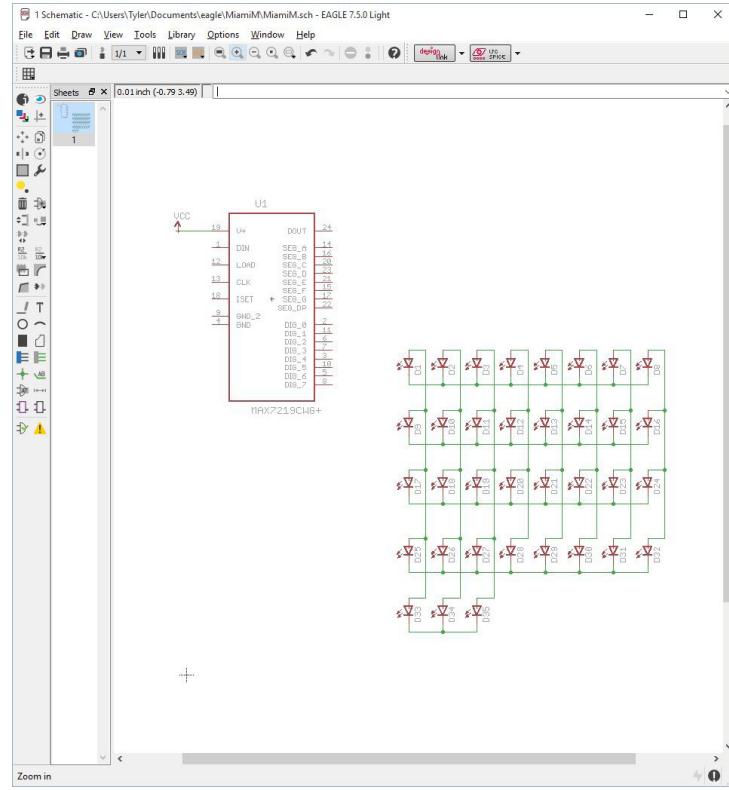


Figure 19: Connecting VCC to V+

The next component to place is GND. Click on the Add tool to bring up the ADD window. Search for “*GND*” in the search box. Use SparkFun’s GND component and place it near the MAX7219 IC’s GND pins.

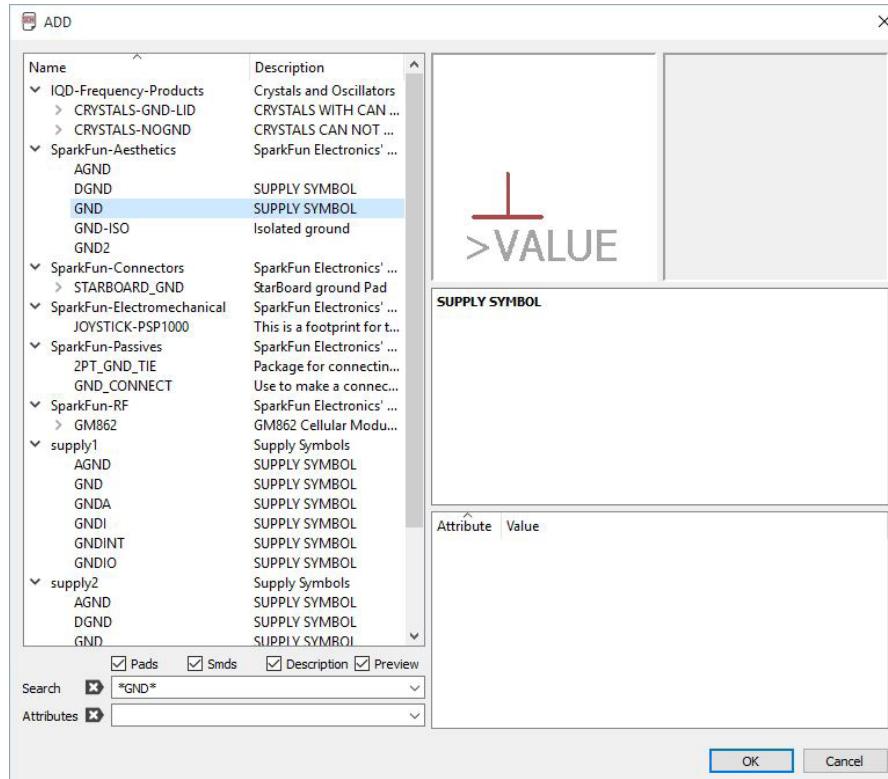


Figure 20: Searching for the GND component

Once you've placed the component, exit the ADD window and wire both GND_2 and GND to the newly placed GND component.

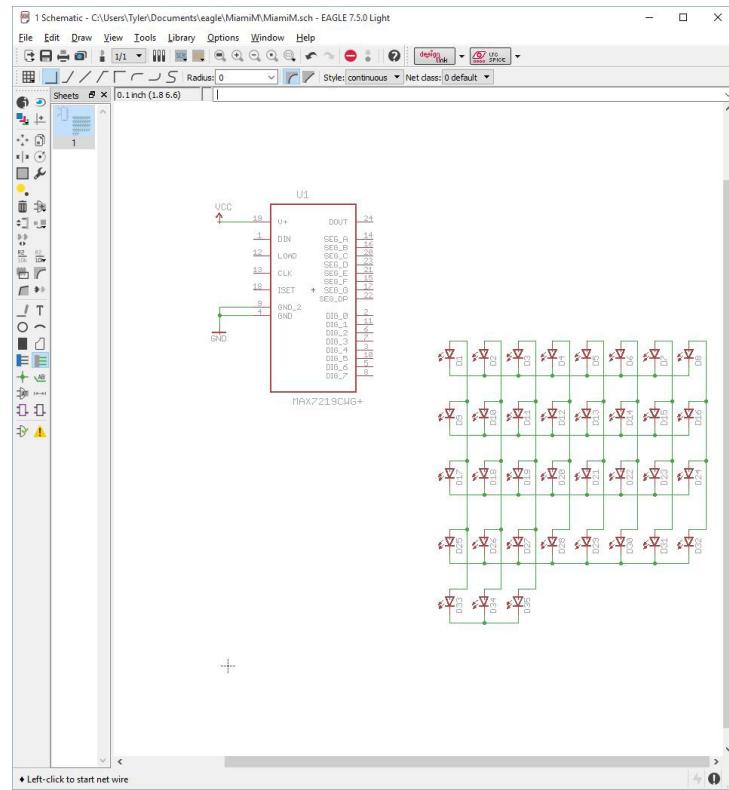


Figure 21: Connecting the GND pins to the GND component

The next components we need to add are capacitors and resistors. We will be adding two capacitors in parallel between VCC and GND, per the suggestion of the [datasheet](#).

Click on the Add tool in the toolbar and search for “*CAP*”. Scroll down through the search results until you see the “SparkFun-Passives” library. You will see a folder in this library named “CAP”. Click on the arrow next to it to expand the folder.

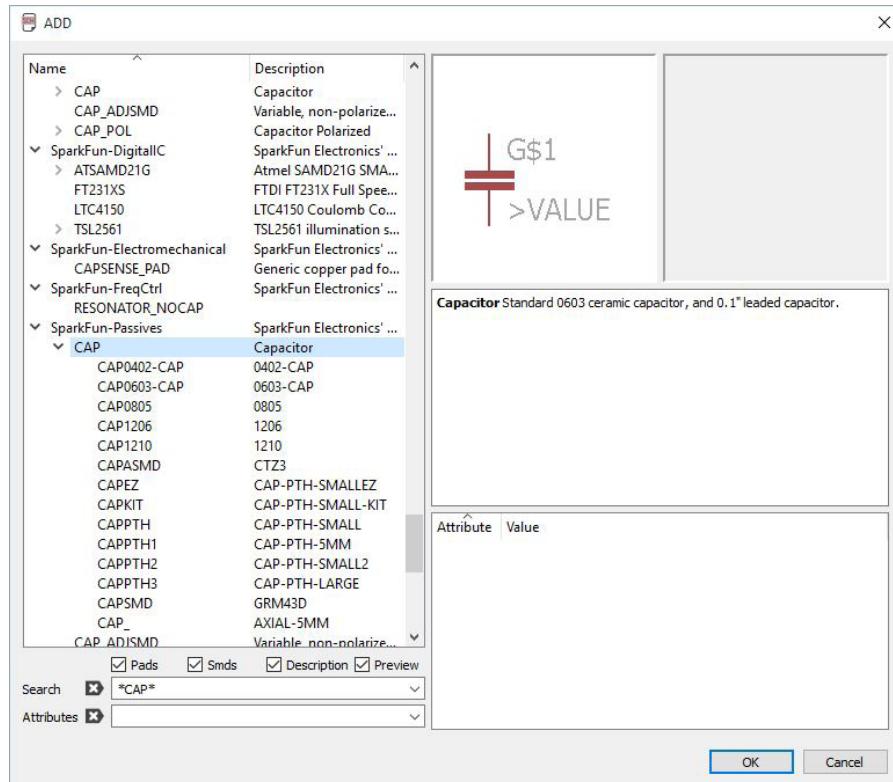


Figure 22: Searching for a capacitor

Next, click on the part named “CAP1206”. This is a capacitor with the 1206 footprint, similar to the LEDs we already added to the schematic.

Once you've selected the part, click OK and place two on your schematic in between VCC and ground. Make sure that these capacitors are placed next to one another, so they can be easily wired in parallel.

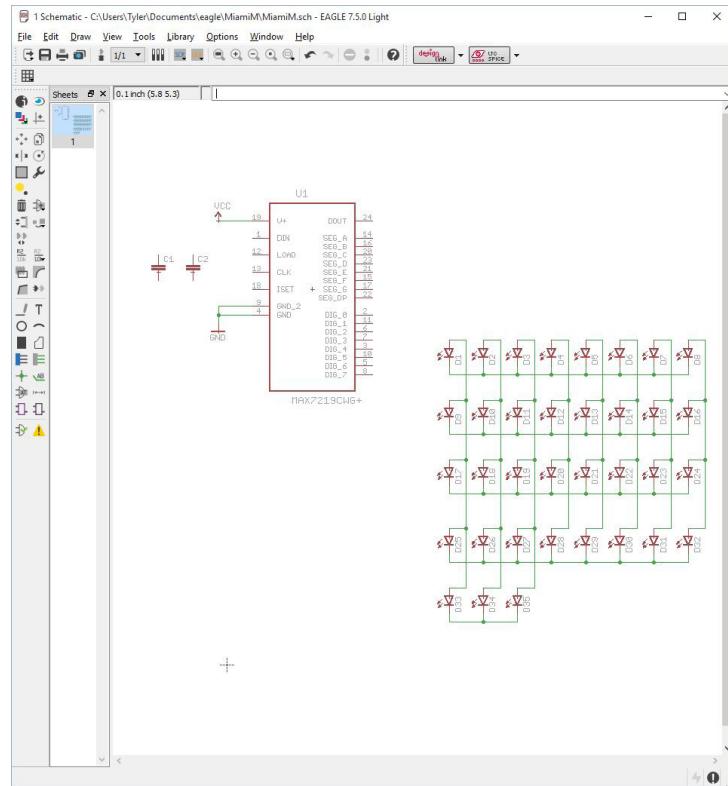


Figure 23: Placing the capacitors in parallel

Click on the Net tool and wire these capacitors, in parallel to one another, between VCC and GND.

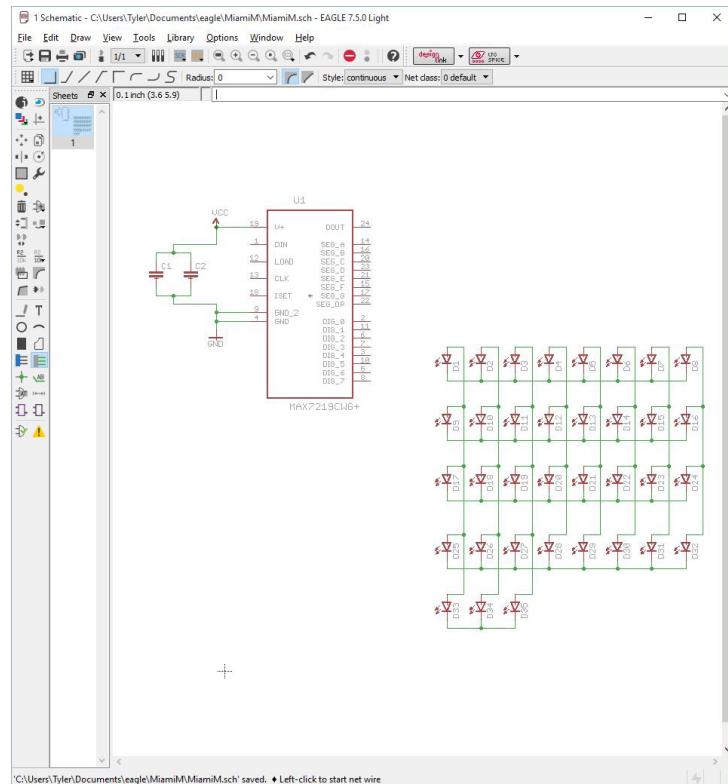


Figure 24: Connecting the parallel capacitors to VCC and GND

The next passive component we will be adding is an SMD resistor between VCC and ISET. This resistor is used to limit the current for all of the LEDs. A big reason for choosing the MAX7219 as our LED driver was that we can use only one resistor to limit the current of all 35 LEDs, rather than using a single resistor for each LED. This greatly reduces the number of parts we need to buy and solder to our board.

Click on the Add tool in the toolbar and search for “*RESISTOR1206*”. Click on the arrow next to the “RESISTOR” folder within the “SparkFun-Passives” library. You should only see one part within the folder. Click on the part and add it to the schematic. You may need to rotate it by right-clicking the mouse button, so that it is easier to wire it between VCC and ISET.

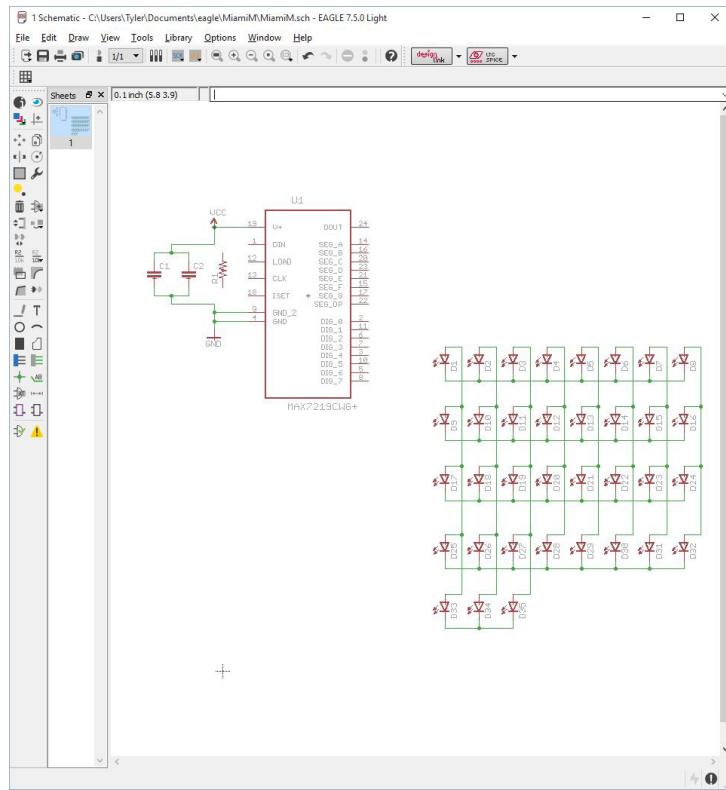


Figure 25: Placing the current limiting resistor

Using the Net tool, connect the resistor to both VCC and the ISET pin.

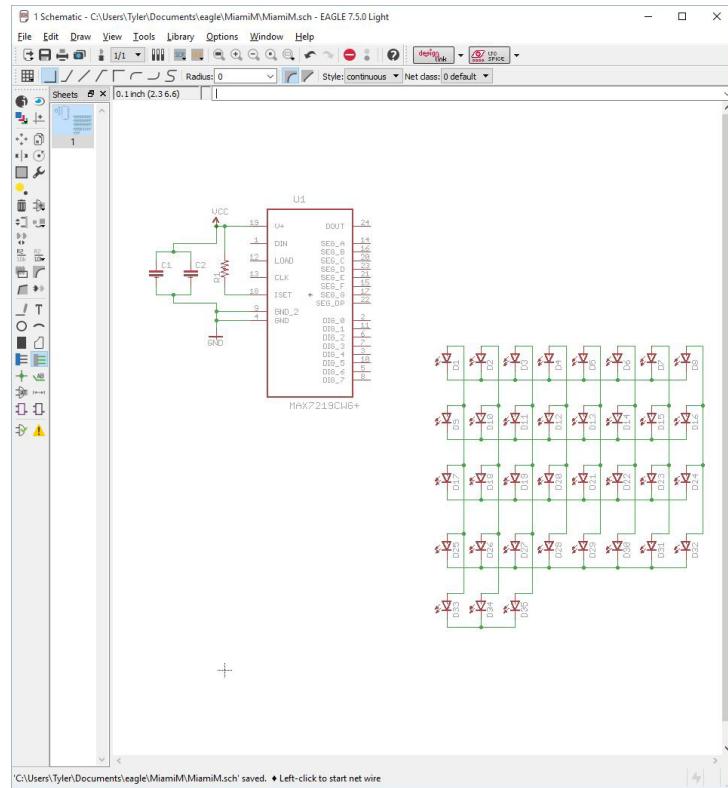


Figure 26: Connecting the current limiting resistor

The next part of this tutorial will be a bit repetitive. Using the net tool, add a connection outward from each of the remaining pins on the MAX7219 chip. Remember to click on the sheet to lay down a net and press the ESC key to exit the wiring mode.

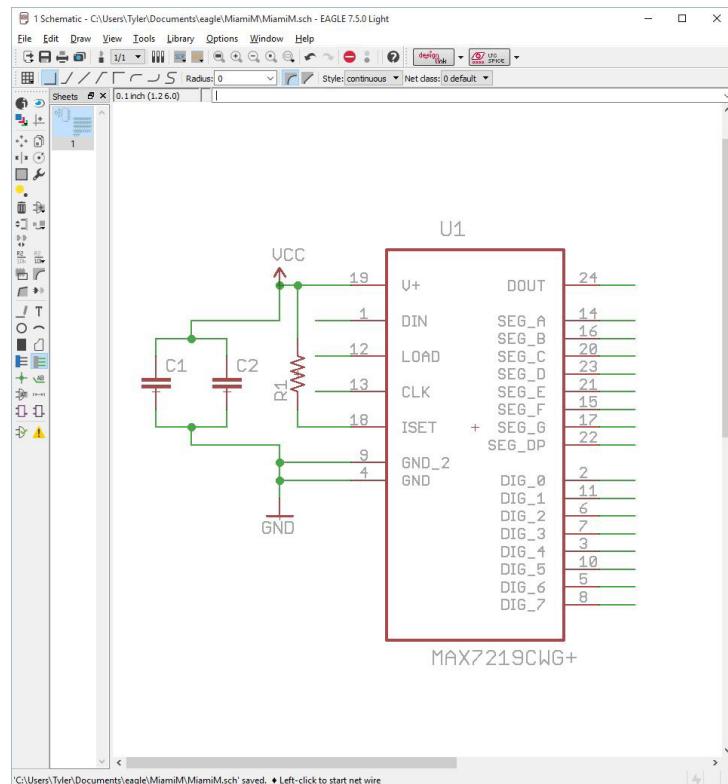


Figure 27: LED driver with all nets

After adding the necessary connections, we need to label each of the connections, so that it is easy for others to understand what each net is called. To do this, click on the Label tool in the toolbar. Next, click on one of the nets you just placed. You'll notice that a label has appeared with the letter "N" followed by a dollar sign and a number. This is the actual name of the net that you placed a few moments ago. Ignore the fact that the name makes no sense to us right now. We will be renaming these connections shortly. When you've found a good spot, click the left mouse button to place the label. Repeat this process for each of the connections.

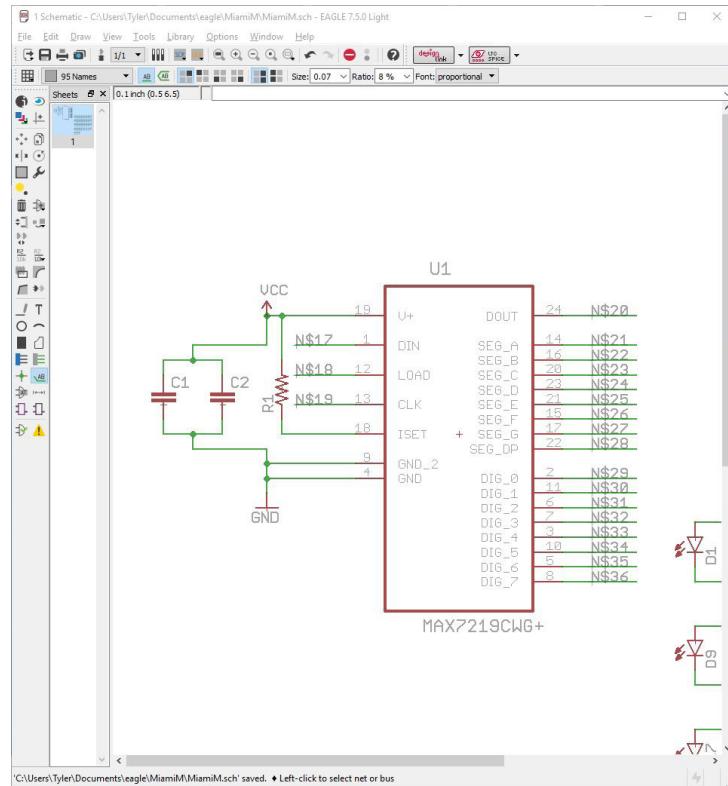


Figure 28: Labeling nets on the IC

Next, we are going to rename these nets with meaningful names. Click on the Name tool in the toolbar. Now, click on each net that you labeled in the last step. When you click on the net, the Name window will pop up.

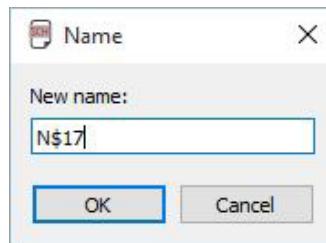


Figure 29: Renaming nets

If you click on the label, instead of the net itself, you will have to double-click to open the Name window.

Change the name of each net according to the table below.

Pin	Name	Pin	Name	Pin	Name	Pin	Name
DIN	PB3	SEG_B	SEG_B	SEG_G	SEG_G	DIG_3	DIG_3
LOAD	PB2	SEG_C	SEG_C	SEG_DP	SEG_DP	DIG_4	DIG_4
CLK	PB5	SEG_D	SEG_D	DIG_0	DIG_0	DIG_5	DIG_5
DOUT	DOUT	SEG_E	SEG_E	DIG_1	DIG_1	DIG_6	DIG_6
SEG_A	SEG_A	SEG_F	SEG_F	DIG_2	DIG_2	DIG_7	DIG_7

Your schematic should look similar to the following image.

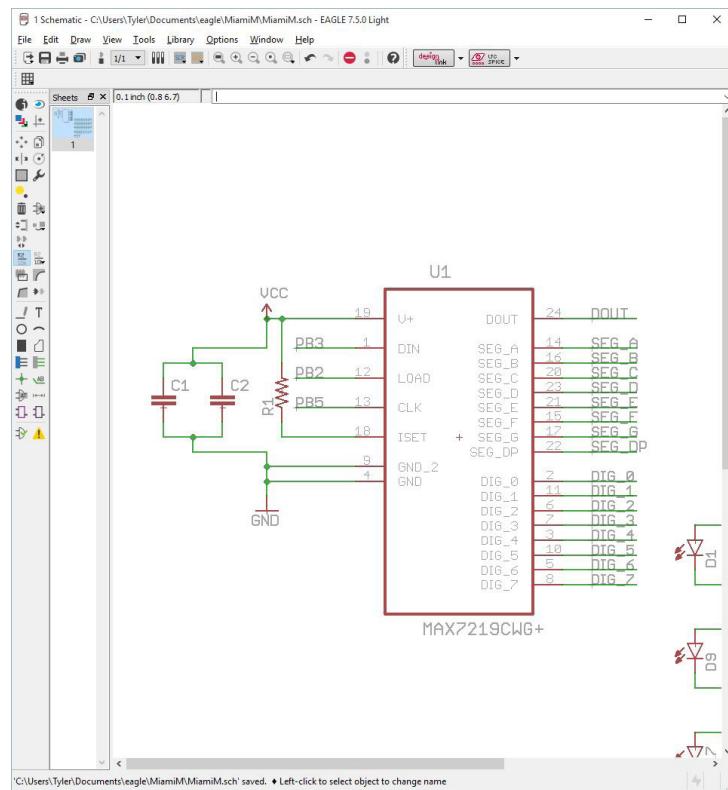


Figure 30: IC with renamed nets

Now that we know how to label nets, we need to label all of the nets in the LED grid. Because of the way these are connected, we only need labels for each row and each column. In the case of our 35 LED grid, we need 13 labels.

Label the columns first by clicking on the anode of each LED in the top row.

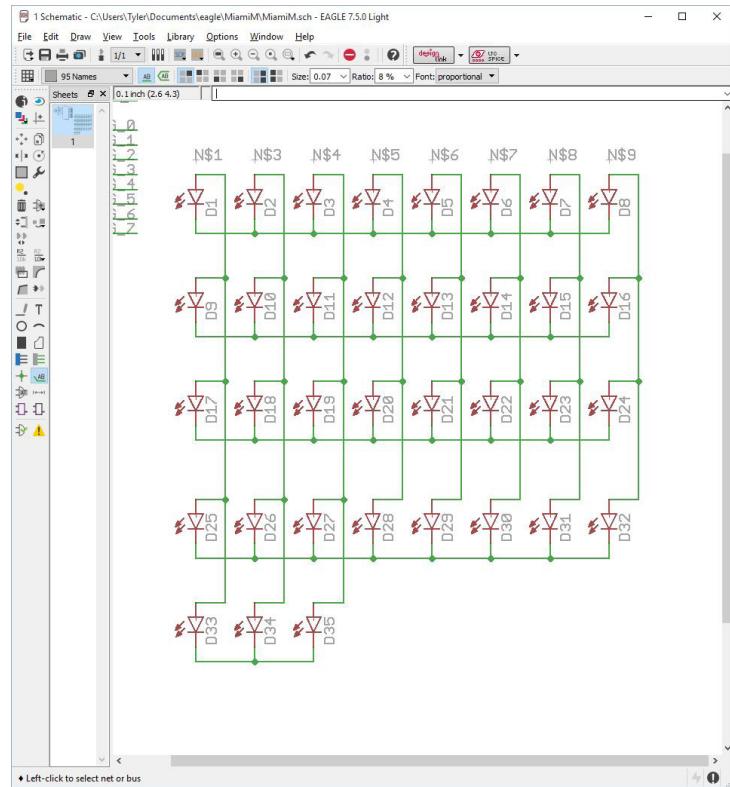


Figure 31: Labeling the columns of LEDs

After labeling the columns, label each of the rows.

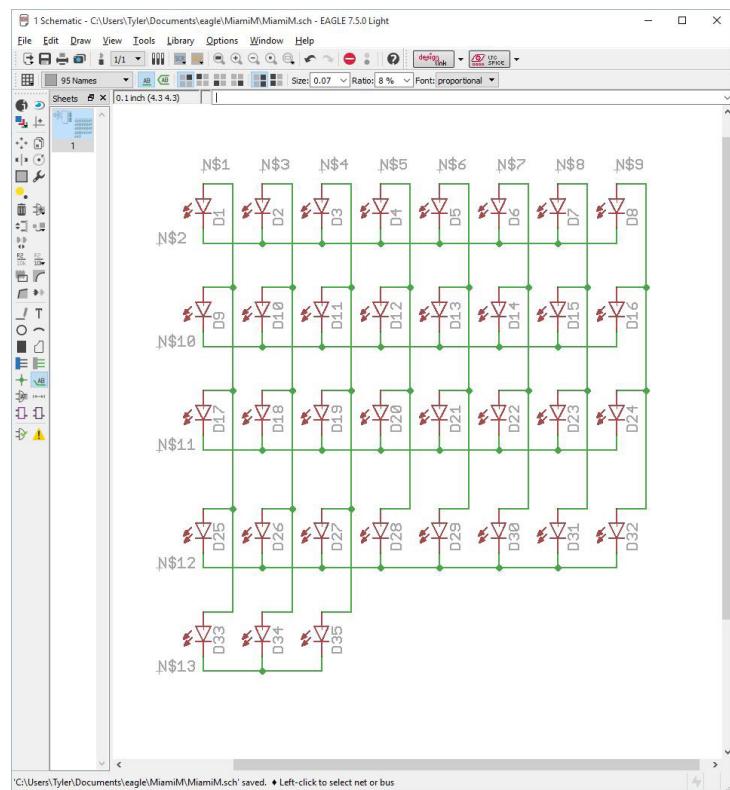


Figure 32: Labeling the rows of LEDs

Next, we need to rename all of the nets in the grid. Select the Name tool. Click on each anode in the top row and name them from left to right: SEG_DP, SEG_A, SEG_B, SEG_C, SEG_D, SEG_E, SEG_F, SEG_G. EAGLE will ask you if you want to make each connection. Ensure you are connecting the proper nets, then click Yes.

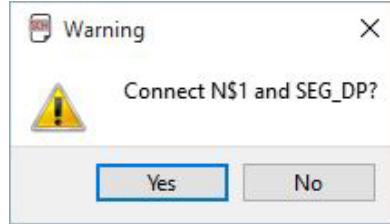


Figure 33: Confirming a connection

Now, rename the nets in the first column by clicking on the cathode for each LED in the column. From top to bottom, name them DIG_0, DIG_1, DIG_2, DIG_3, and DIG_4, respectively.

Your schematic should resemble the image below.

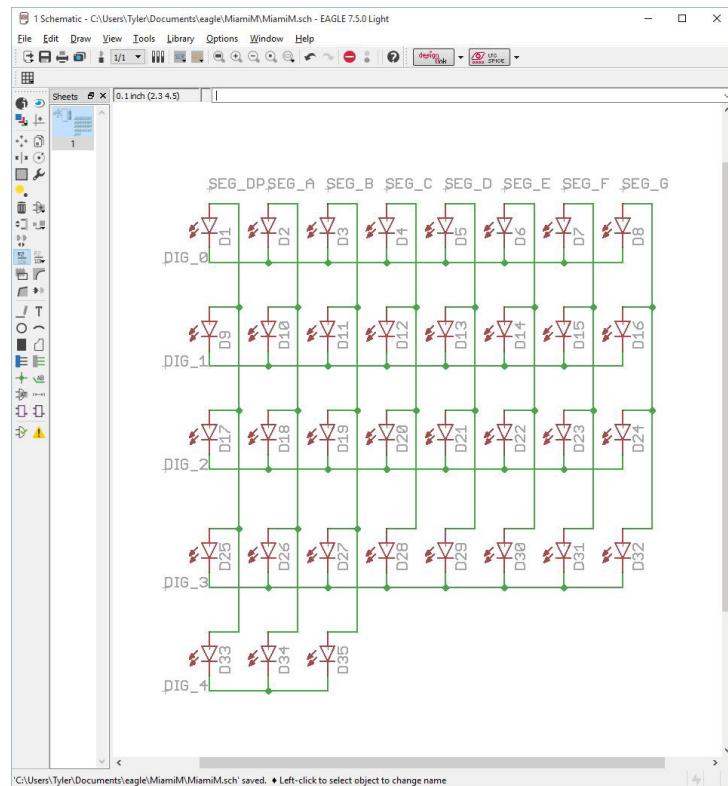


Figure 34: Renamed LED nets

Next, click on the Value tool. This tool allows us to set a value for each of the components. While this doesn't actually do anything to the board's functionality, it does help others reviewing your schematic to understand what the value of each component will be when you actually assemble the board. Click on one of the capacitors. The value window will appear. Type "10u" into the window and click OK. The value of the capacitor now appears in the schematic. Repeat this process for the second capacitor, but set this capacitor to "0.1u". Click on the resistor next and set its value to "28k".

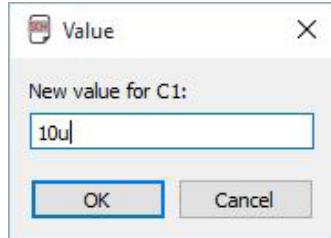


Figure 35: Changing the component value

We recommend consulting the [datasheet](#) for the MAX7219 to understand why the passive components, and their values, are necessary for the chip to function.

Your entire schematic should look very similar to the schematic shown below.

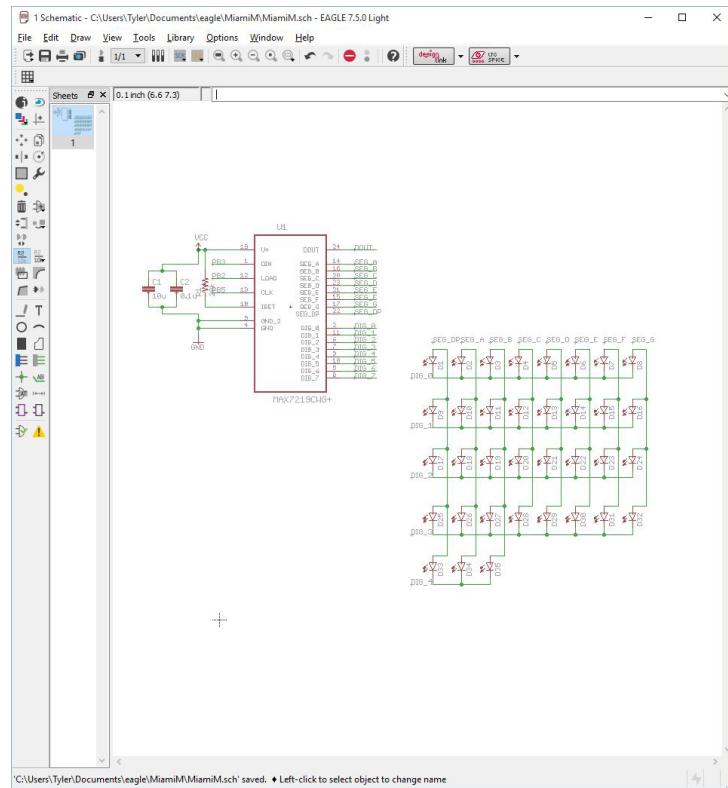


Figure 36: MAX7219 IC and LED grid

Now, we need to add a microcontroller unit (MCU) to control the LED sequence. The MCU that we will be adding is the ATmega328P. Open the ADD window and search for “*ATMEGA328*”. Select the part named “ATMEGA328P_TQFP”, within the “SparkFun-DigitalIC” library.

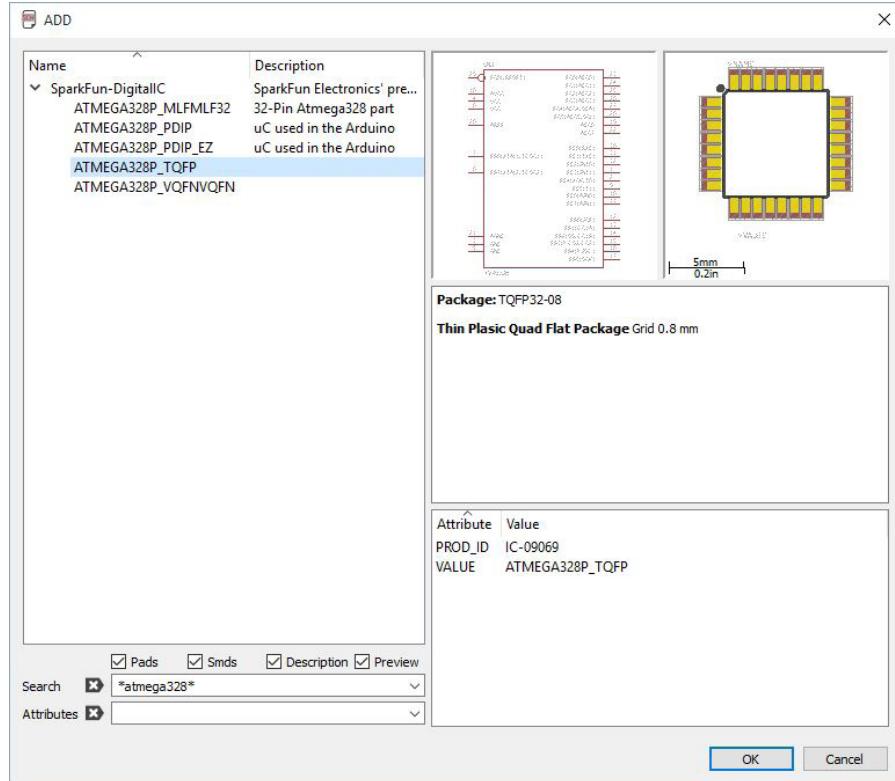


Figure 37: Searching for the MCU

TQFP stands for Thin Plastic Quad Flat Package and is a type of surface mount package for ICs. Click OK and place the IC on your schematic sheet.

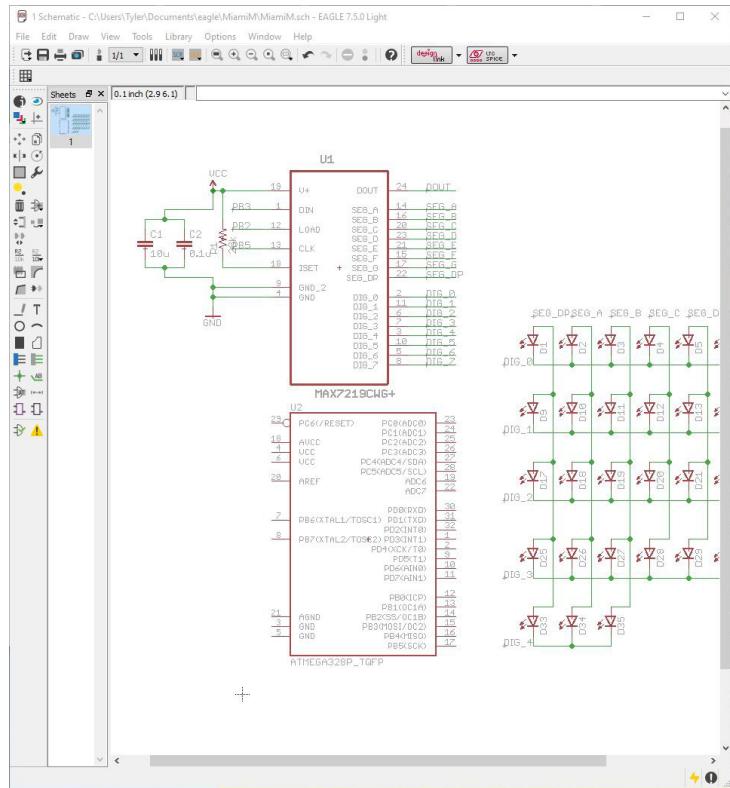


Figure 38: Placing the MCU

Once you've placed the MCU, we are going to add VCC and GND components to it, just like we did for the MAX7219 IC.

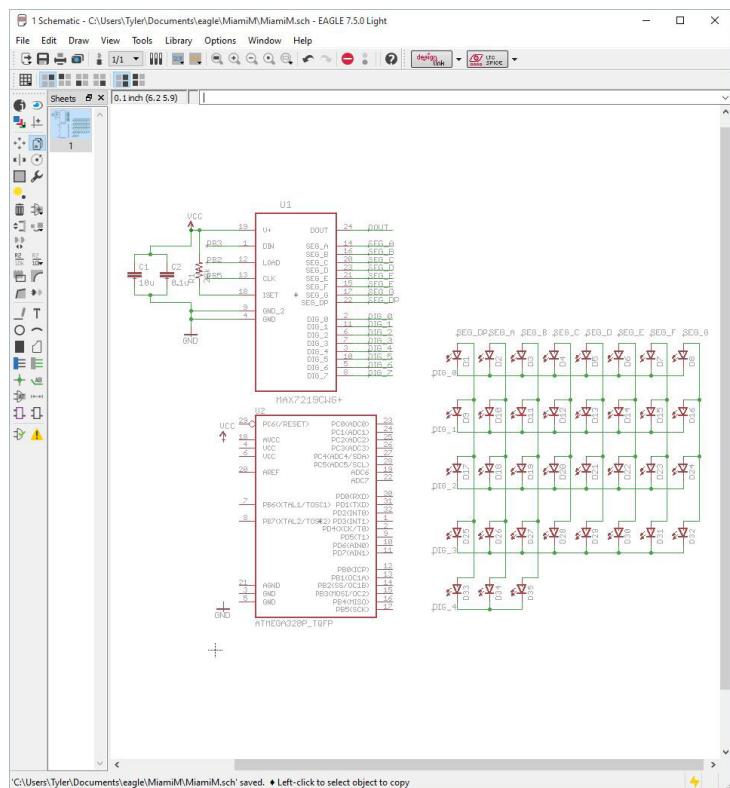


Figure 39: Placing VCC and GND components

Using the Net tool, connect the AVCC, VCC, and VCC pins to your newly placed VCC component.

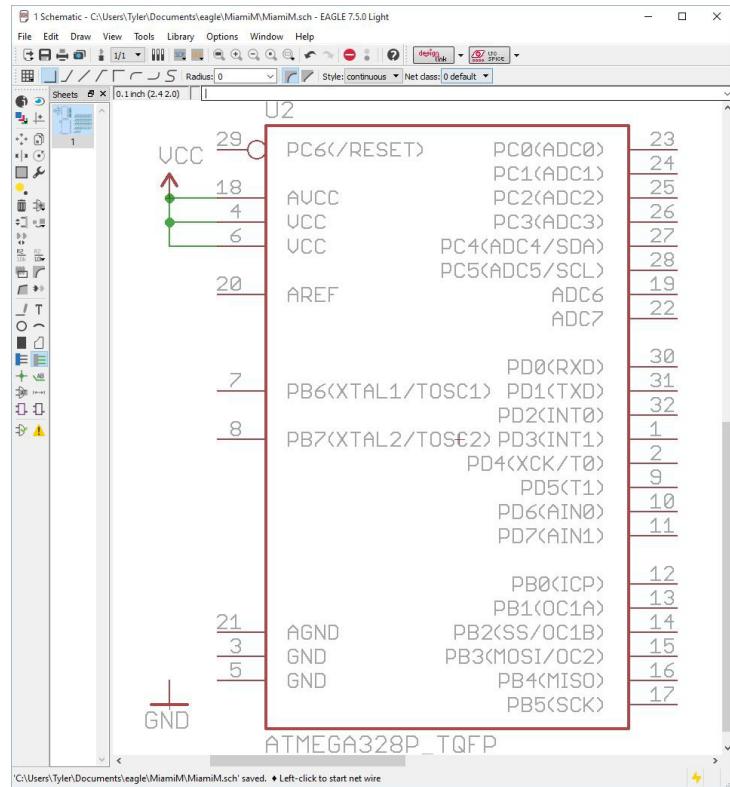


Figure 40: Connecting the VCC component to the MCU pins

Next, connect the AGND, GND, and GND pins to the GND component you placed a few moments ago.

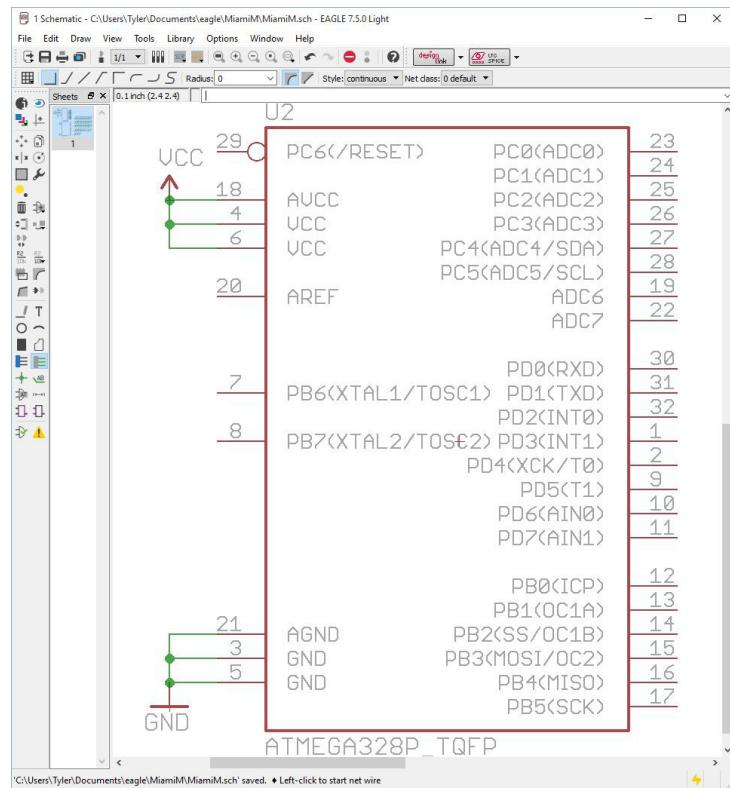


Figure 41: Connecting the GND component to the MCU pins

We're now going to add connections to each remaining pin, and label them, like we did for the MAX7219 IC.

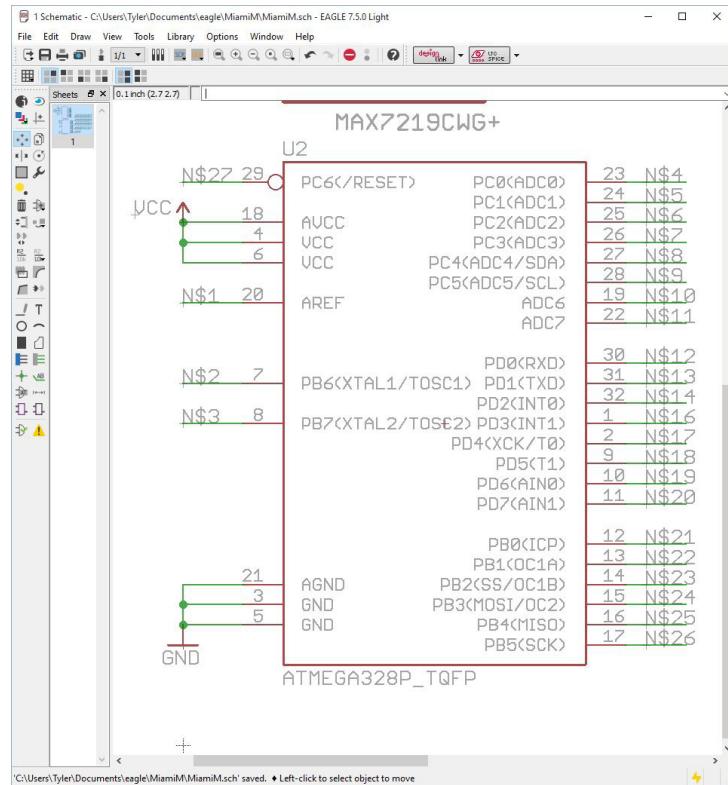


Figure 42: Labeling the remaining MCU pins

After labeling all of newly added connections, we need to rename those nets to something meaningful.

Using the following table, rename each net according to the pin names on the IC.

Pin	Name	Pin	Name
PC6(/RESET)	PC6	PD1(TXD)	PD1
AREF	AREF	PD2(INT0)	PD2
PB6(XTAL1/TOSC1)	PB6	PD3(INT1)	PD3
PB7(XTAL2/TOSC2)	PB7	PD4(XCK/T0)	PD4
PC0(ADC0)	PC0	PD5(T1)	PD5
PC1(ADC1)	PC11	PD6(AIN0)	PD6
PC2(ADC2)	PB2	PD7(AIN1)	PD7
PC3(ADC3)	PC3	PB0(ICP)	PB0
PC4(ADC4/SDA)	PC4	PB1(OC1A)	PB1
PC5(ADC5/SCL)	PC5	PB2(SS/OS1B)	PB2
ADC6	GND	PB3(MOSI/OC2)	PB3
ADC7	GND	PB4(MISO)	PB4
PD0(RXD)	PD0	PB5(SCK)	PB5

A few of these connections are already connected to something else in the schematic, so EAGLE will verify that you want to connect these to the given net. Verify the connections and click “Yes”.

Your MCU should have connections labeled just like the MCU in the following image.

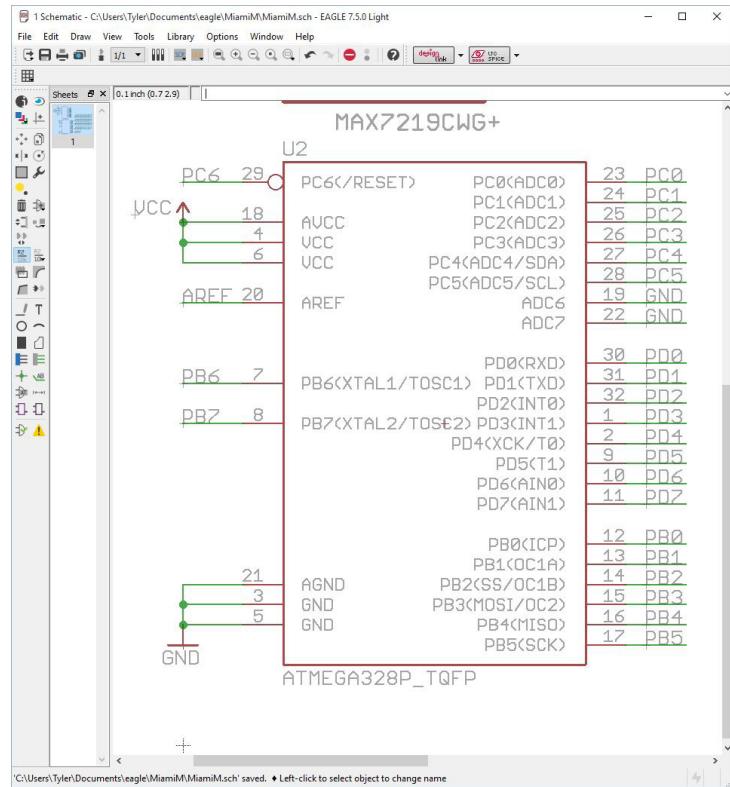


Figure 43: MCU with renamed nets

You'll notice that many of these pins are left disconnected, but we are still labeling and naming them. This is so that, if you were to add more components to this schematic, it would be fairly easy to connect them to the MCU without having to add messy wiring.

We only have two more components to add to this schematic before we generate the board file. The first is the header for programming the MCU. In-system programming (ISP) is generally done with a 6-pin or 10-pin header. For our system, we will be including a 6-pin header, which will allow this MCU to be programmed from a dedicated AVR ISP programmer or an Arduino.

To add the 6-pin header, search for “*AVR_SPI_PRG_6PTH*” within the ADD window. The component we are adding is named “AVR_SPI_PRG_6PTH”. Place the header on the schematic sheet.

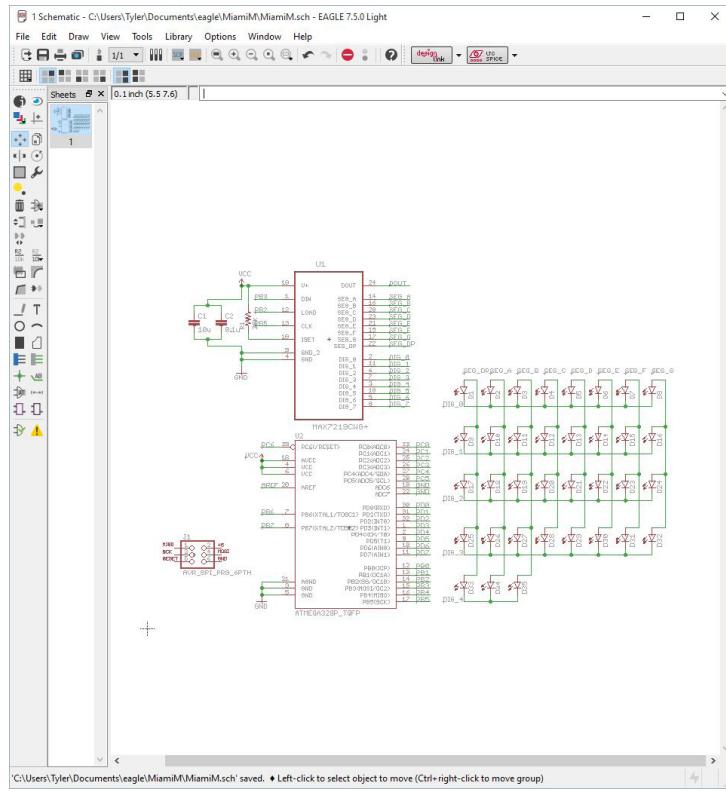


Figure 44: Placing the 6-pin header

Using your Net tool, create connections in the same manner as we have for the previous components.

Name the nets according to the table below.

Pin	Name
MISO	PB4
+5	VCC
SCK	PB5
MOSI	PB3
RESET	PC6
GND	GND

Your header should look like the header in the following image.

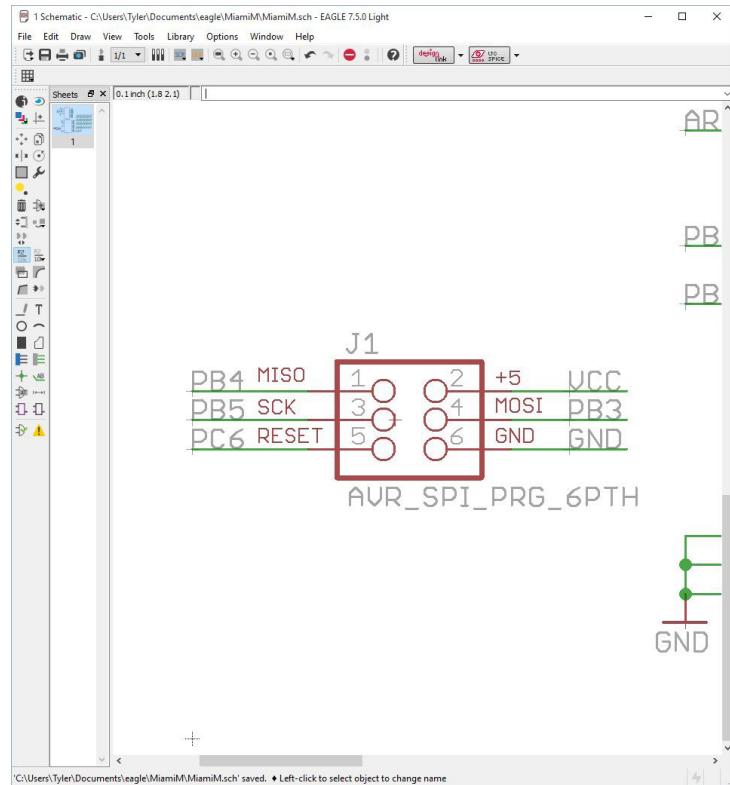


Figure 45: Labeled and renamed nets on the 6-pin header

The final component we need to add to the circuit is a power source. We opted for a screw terminal, which is a through-hole component. The screw terminal has the same package as a 2-pin header, so we will be adding a 2-pin header to the schematic. Open the ADD window and search for “*M02PTH*”. The part is contained within the “SparkFun-Connectors” library and is named “M02PTH”. Select the proper part and place it on the sheet.

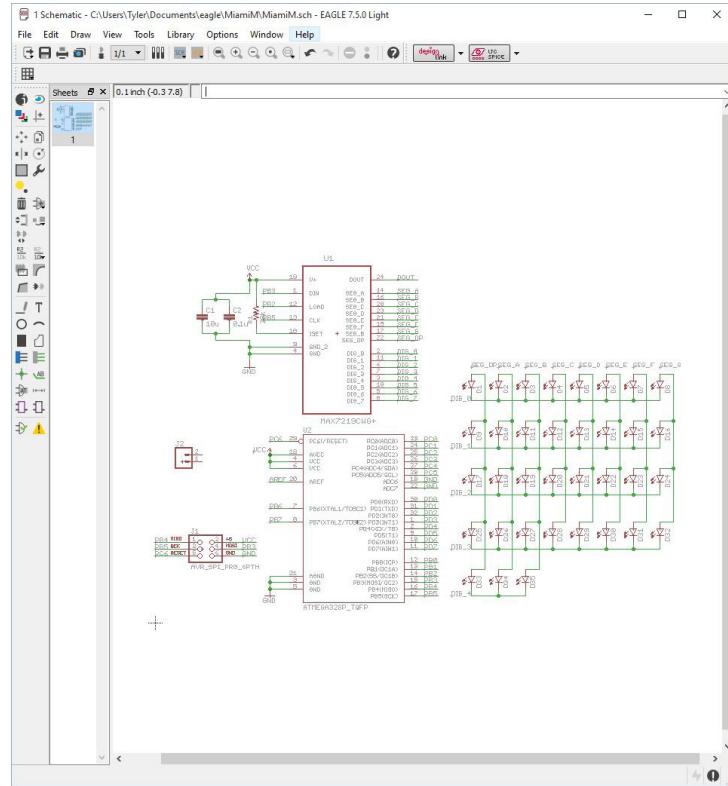


Figure 46: Placing the 2-pin header

Next, add a VCC part and a GND part to the sheet and the use the Net tool to connect each part to a pin of the header (J2). It is good practice to put VCC on top and GND on bottom, as is shown in the following image.

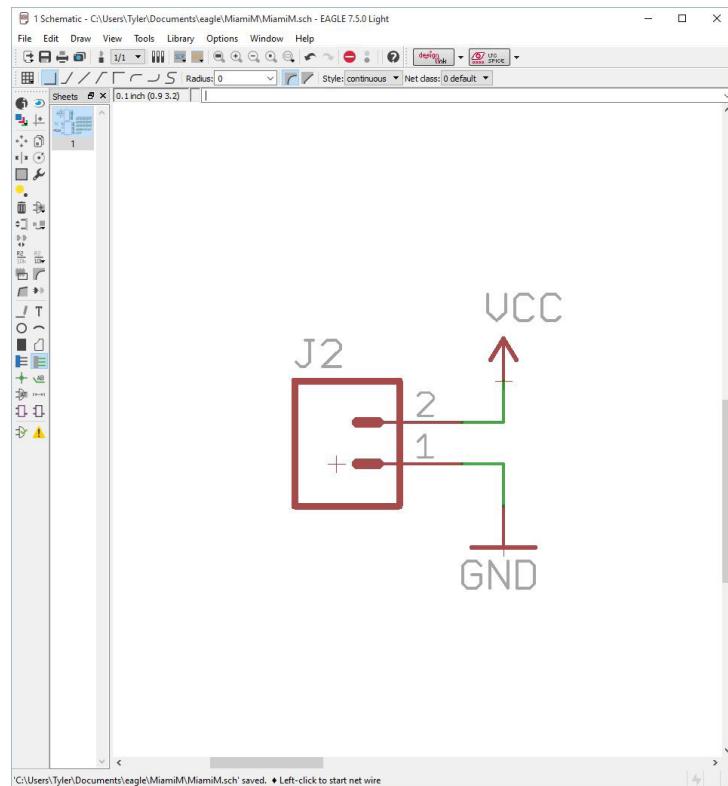


Figure 47: Connecting the 2-pin header to VCC and GND

With the header finished, we have now completed the entire schematic! It should look similar to the image below.

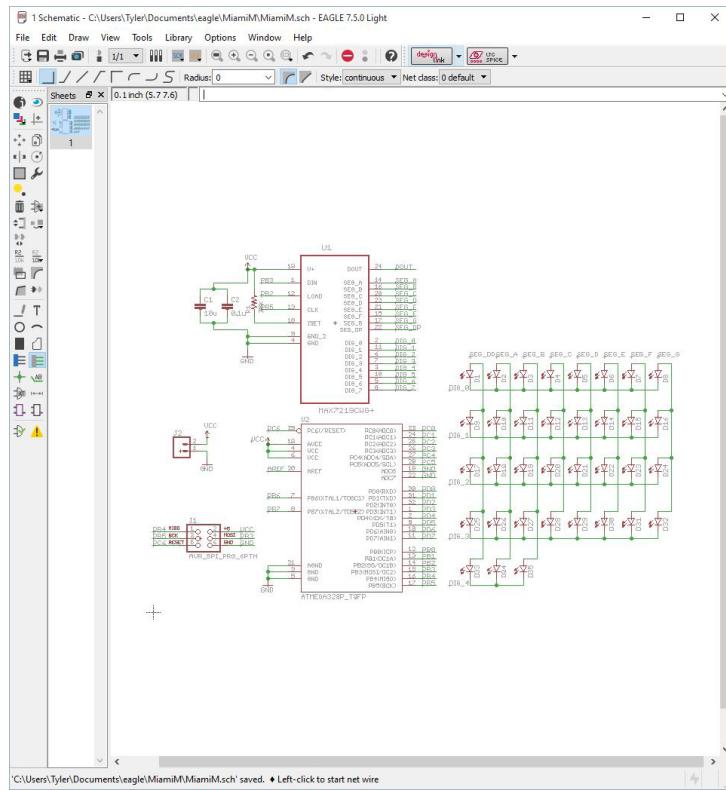


Figure 48: Overall system schematic

The final step, before we generate the board file, is the Electrical Rule Check (ERC). Click on the ERC tool. A new window named “ERC Errors” will pop up. You are going to have several warnings. This is EAGLE’s way of making sure you meant to leave so many pins on the MCU disconnected. You’ll also see an error saying the LEDs don’t have a value. This is also fine, but, if you are so inclined, you can use the Value tool to write in the color of the LED (i.e. “RED” or “BLUE”). The important thing to check in this window is that there are no errors in the Error category.

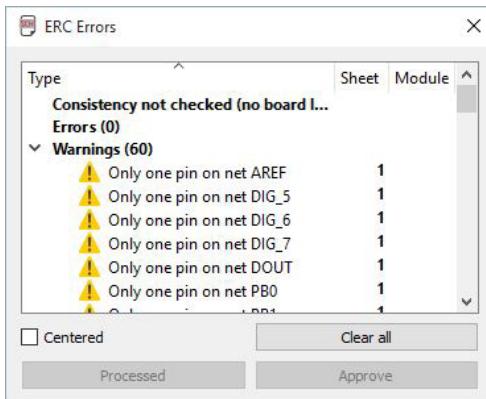


Figure 49: Viewing the ERC errors

Now that we've finished the schematic, and verified there are no errors, we can generate the board layout.

With the schematic still open, click on the “Generate/switch to board” button. After we create the board file, you can use this button to switch between the two screens quickly. When you click this button for the first time, EAGLE will notify you that you don't have a board file yet and it will ask if you'd like to create one. Click “Yes”.

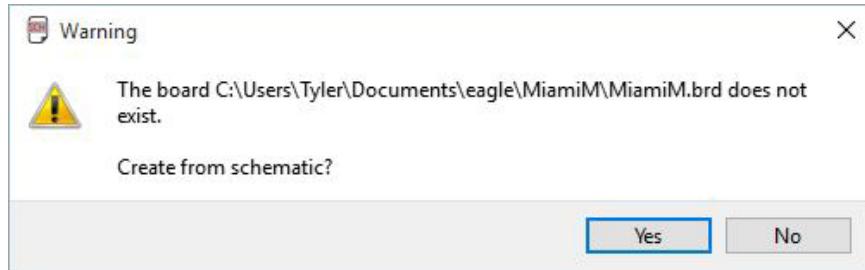


Figure 50: Generating a board file from schematic

Chapter 3

Laying Out the Board

After clicking on the “Generate/switch to board” button in the schematic, and clicking “Yes”, a new window will appear. It will be pretty small by default, so you’ll want to expand it to see the board better. There are several new buttons in the user interface, but a few will look familiar from the schematic view. The new tools we will be using in this chapter are as follows:

-  **LOCK** – When a part is left-clicked, its position is locked; the origin will become an “X” and the part can only be unlocked by holding shift and left-clicking the locked part
-  **ROUTE** – Allows users to manually route connections between components; once selected, use the top toolbar to select properties for the trace
-  **RIPUP** – Removes traces from the board and replaces them with airwires
-  **VIA** – Places a via where the user clicks; settings associated with the via are found in the top toolbar
-  **HOLE** – Used to add drill holes to the board; the size of the hole can be set using the textbox or drop down in the top toolbar
-  **RATSNEST** – When clicked, EAGLE finds the shortest distance between components and redraws the airwires
-  **AUTOROUTER** – Opens the Autorouter window for customizing and running the autorouter, a program that automatically identifies several possible board layouts and presents them to the user
-  **DESIGN RULE CHECK** – Opens the Design Rule Check window, which allows users to load design rules or check the board against the currently loaded design rules

Before we get started laying out the board, we are going to load a design rule file (.dru) into EAGLE. Design rules tell the Autorouter tool what guidelines to follow when routing the board. They also provide a rubric of sorts for the Design Rule Check tool to compare the board design against. The design rule file we will be using comes from [OSH Park](http://docs.oshpark.com/resources/oshpark-2layer.dru), and can be downloaded here: <http://docs.oshpark.com/resources/oshpark-2layer.dru>. OSH Park, a batch PCB fabrication company, aggregates orders from many different customers, then panelizes the boards, before sending them to the fabricator. Once the boards return from fabrication, they are broken up and mailed to the customer. While the lead time is longer than many other manufacturers, OSH Park is able to offer competitive pricing because of their batch PCB process. After completing this tutorial, you will be able to upload your files to their website and order your newly created board.

To load a design rule, click on Edit > Design rules... in the main toolbar. The following window will appear.

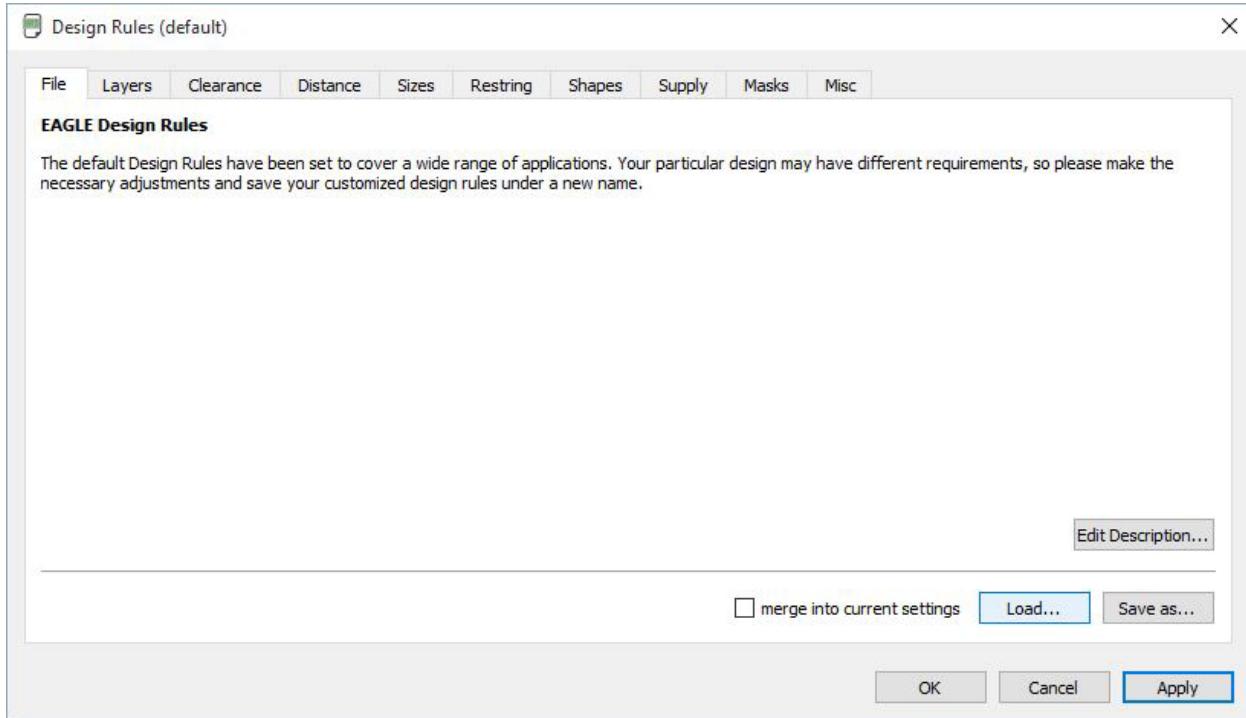


Figure 51: Loading a design rule file

Click on the “Load...” button to open the file browser. Navigate to the design rule file you downloaded from OSH Park.

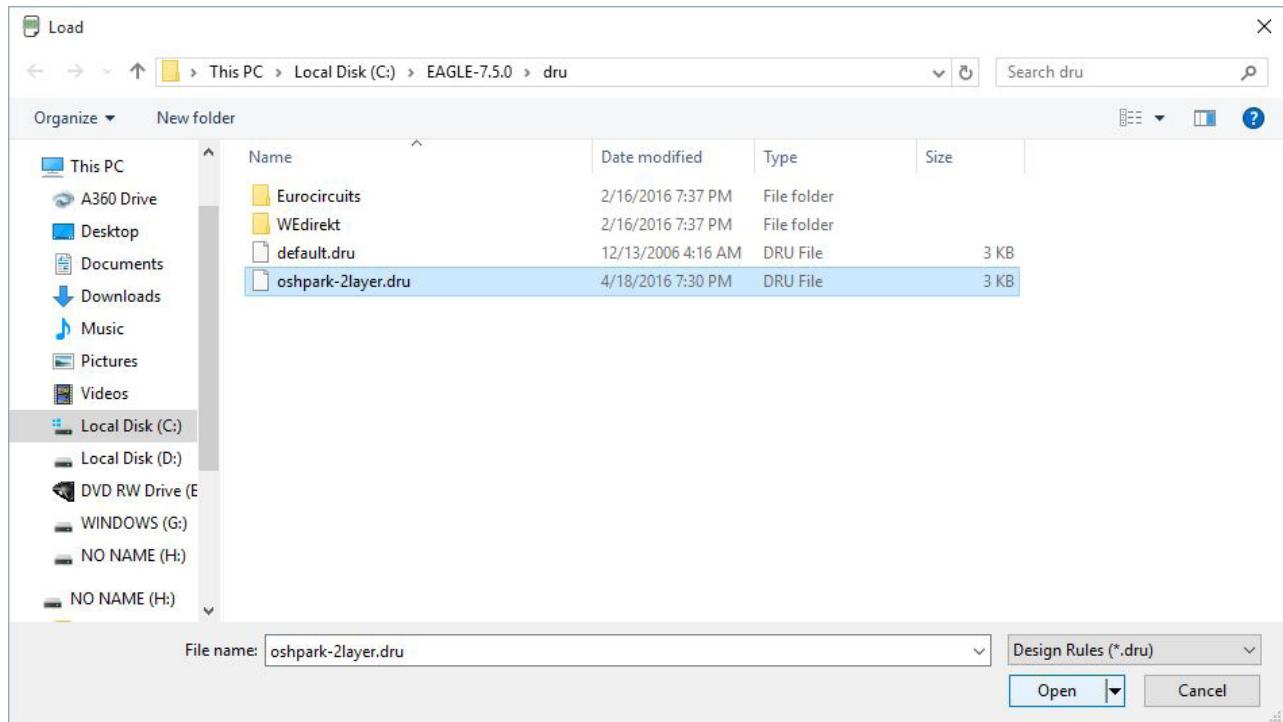


Figure 52: Opening the downloaded design rule file

Click on the design rule file and then click the “Open” button to load the design rules. The original Design Rule window will now be visible. Click the “Apply” button and then click OK to close the window. We will be running a design rule check later on in this chapter.

Next, we are going to lay out all of the components on our board.

EAGLE automatically places all of the components just outside of the board layout when a new board file is created. Using the Group and Move tools, we are going to move all of the LEDs into the gray square. The gray square you see in the center of the board layout window is the board outline. This can be resized using the Move tool, but we'll do that once we've laid everything out.

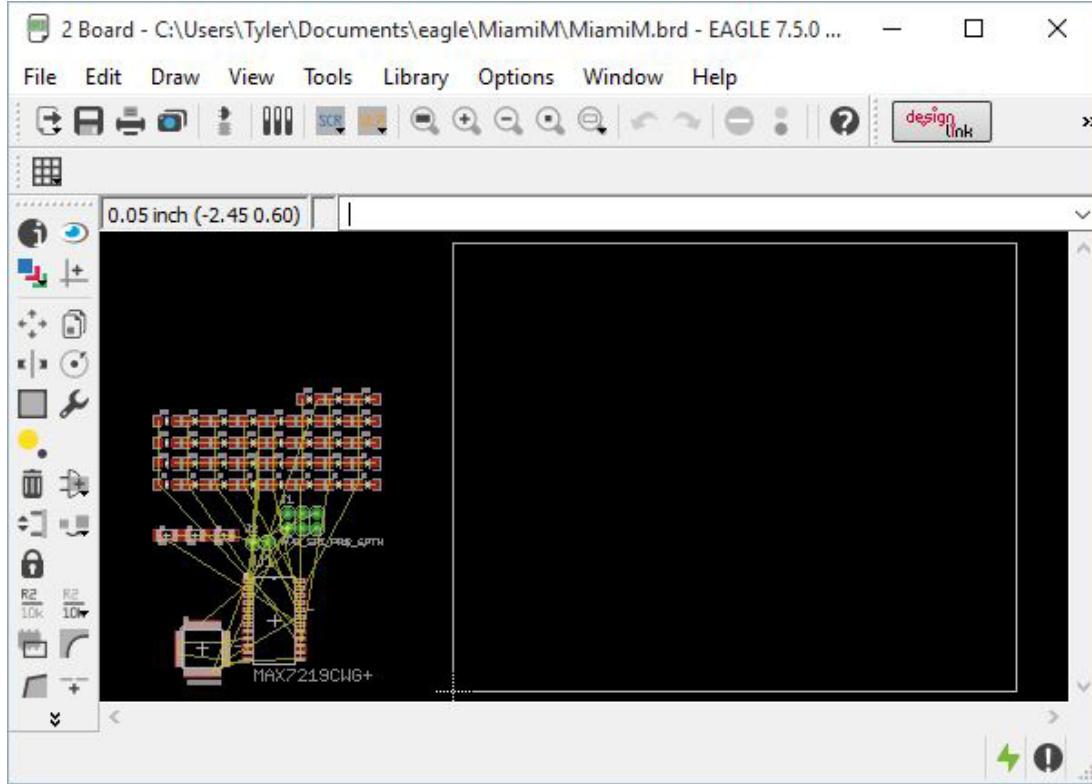


Figure 53: The automatically populated board file

First, click on the Move button. Next click on the Group button. The Group button will allow you to select multiple components at once. Click and drag in the window to define your selection area and select all of the LEDs.

Release the mouse button when you've properly defined the selection area. You'll notice that the selected components are brighter than the non-selected components. Also, it's important to note that the Move tool is now selected in the toolbar. This is because we selected it prior to using the Group tool. It is important to remember to select your tool, then use the Group tool to define a group. Now, we can right-click anywhere in the window and see an option for "Move: Group". Be aware that, wherever you right-click, that location will become the origin when you click "Move: Group". Select that option and move the group of LEDs into the square.

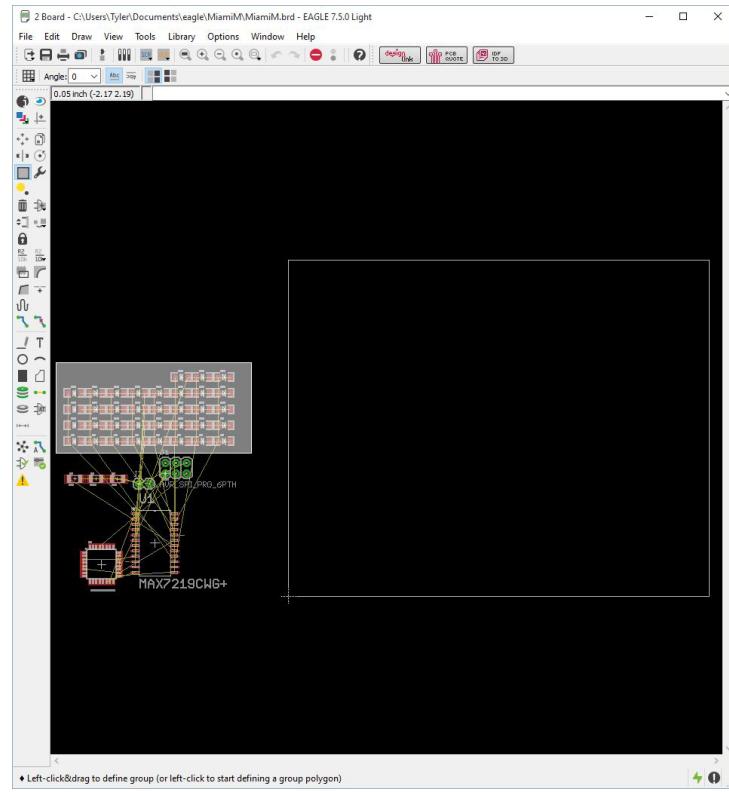


Figure 54: Selecting a group of LEDs

Now that you've moved the group, you can see there are many lines connecting the components. These are called airwires.

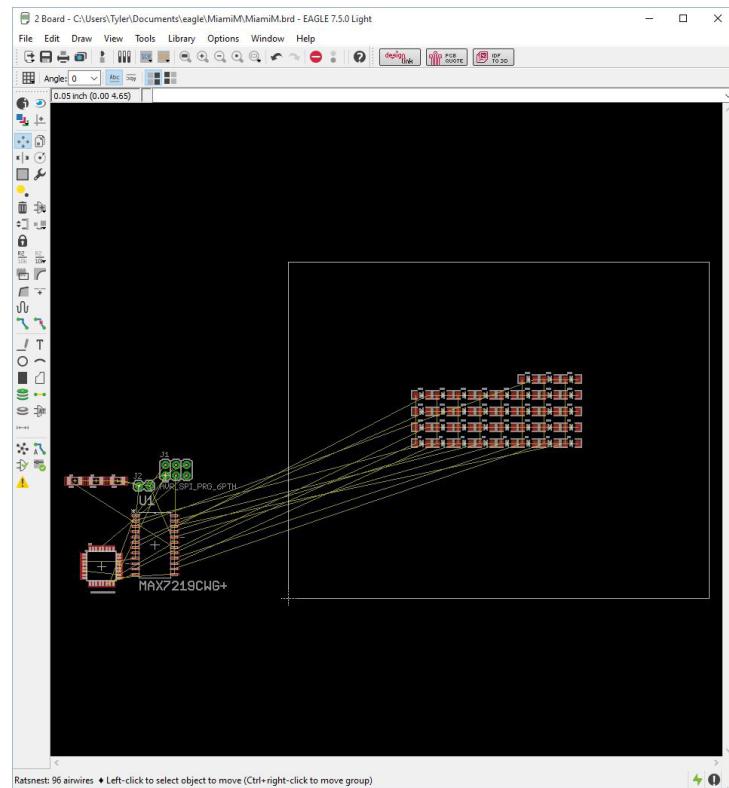


Figure 55: Moving a group of LEDs

These wires haven't been routed on the board yet, but are there to show the connections between components. If you click on the Ratsnest tool, you can find the shortest path for these airwires, which often cleans up the display quite a bit. As we move components around, be sure to intermittently use the Ratsnest tool to clean up the display.

Using the Move tool, move the remaining components onto the board. Right now, it doesn't matter exactly where they are placed.

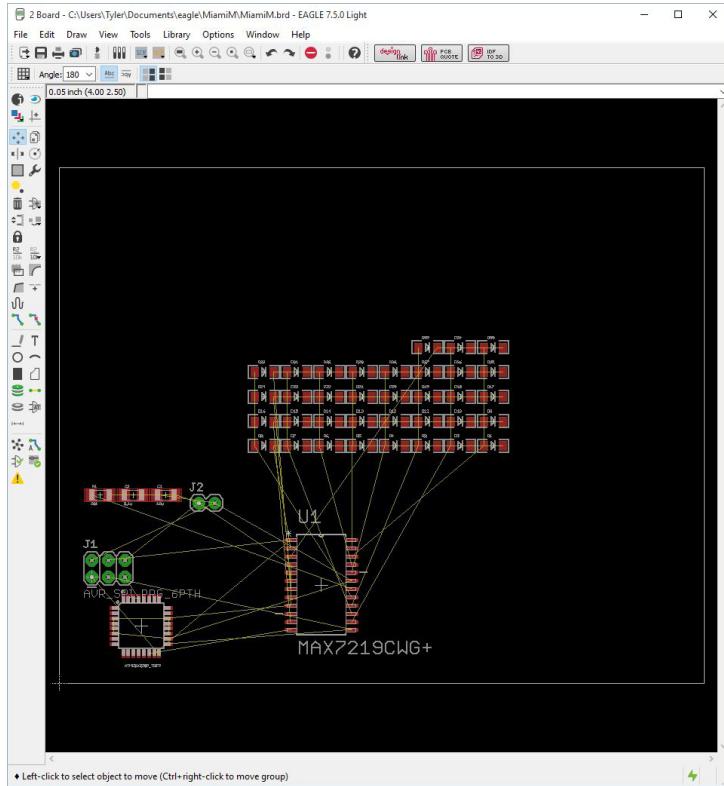


Figure 56: Placing all components on the board

When moving components, they automatically snap to points on the grid. If you'd like to move components in half-steps, hold down the ALT key on your keyboard as you move components and you will have an extra step of resolution. Additionally, by default, the grid is invisible. If you'd like to turn on the grid, click the Grid button. A new window named "Grid" will appear. By manipulating the settings in this window, you can change the grid that the components snap to.

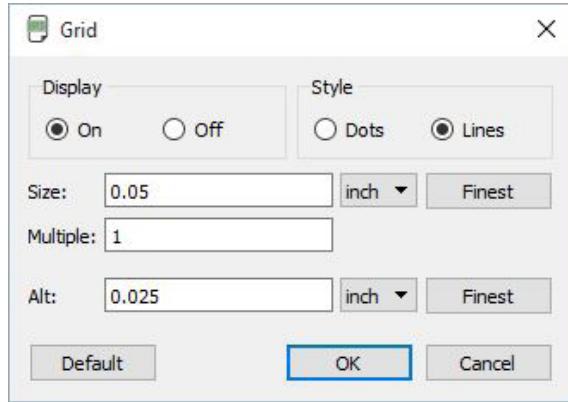


Figure 57: The Grid settings window

Now, arrange your LEDs in whatever shape you would like. In this tutorial we will be arranging them in the shape of the Miami “M” logo. Use the Move tool to accomplish this. Remember, you can rotate components with either the Rotate tool or by right-clicking when a component is being moved using the Move tool.

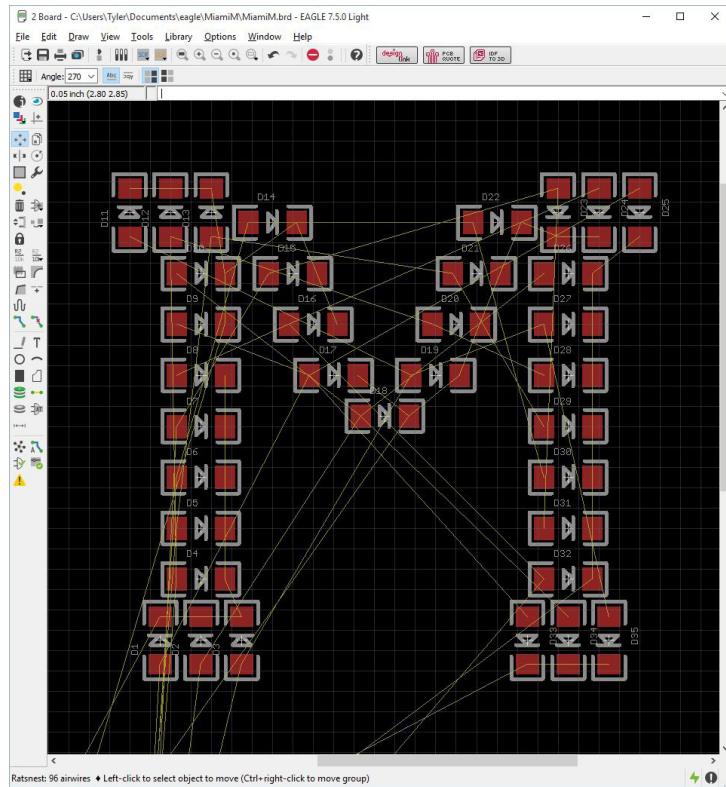


Figure 58: LEDs arranged in an “M” shape

With the LEDs arranged, we can now focus on the other components. Move the components around to minimize the length of the airwires and reduce the number of airwires overlapping. This will make the board easier to route.

It is important, when laying out a PCB, to consider the schematic you designed. For example, we know that the capacitors are wired in parallel between VCC and GND, so they should be placed in parallel on the board to reduce the length and number of traces needed.

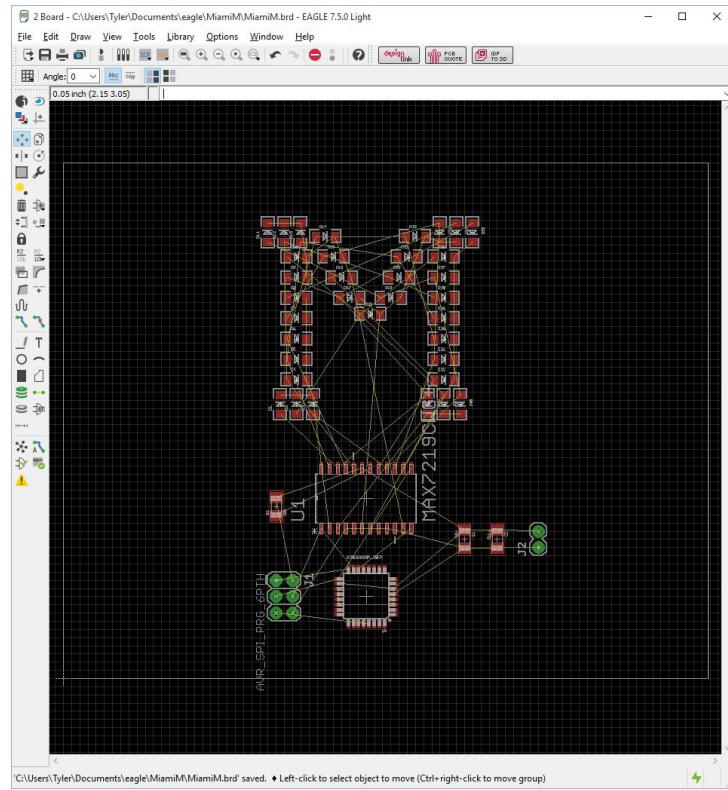


Figure 59: Initial component arrangement

Once the components have been positioned to your liking, we are going to run the Autorouter. In this tutorial, we are going to use the Autorouter tool to automatically connect the components on both sides of the board. While the Autorouter is easy to use, it doesn't always provide the most efficient connections. Sometimes, it will even say a board cannot be routed, so it's important to not rely entirely on the Autorouter. Click on the Autorouter tool. A new window will open.

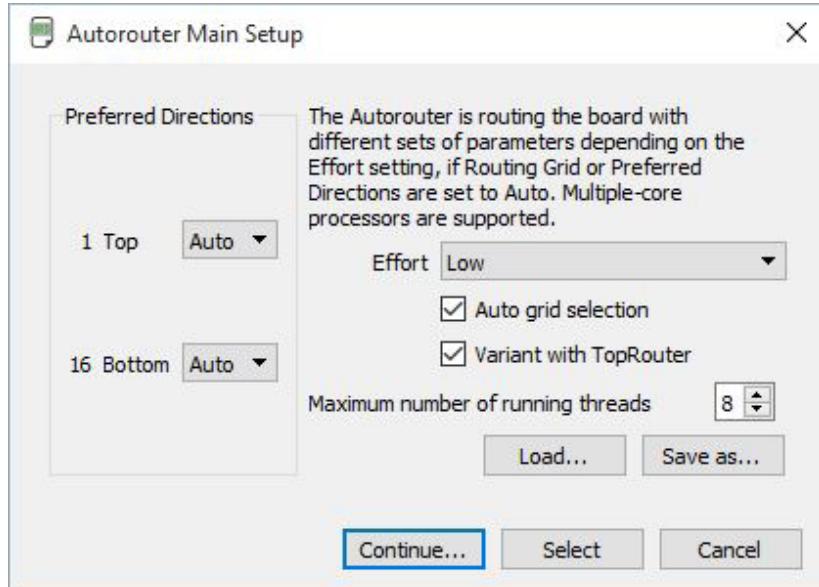


Figure 60: Adjusting the Autorouter settings

Make sure that the drop-down menus for both Top and Bottom are set to auto. This will tell the Autorouter that we have no preference whether traces are laid on the top, bottom, or both sides of the board. The next important setting to note is the Effort setting. If you have a high-performance computer, you can take this setting up a few notches. If not, especially with a complex board design, play it safe by setting the Effort to Low and reducing the maximum number of running threads. When you're ready to route the board, click "Continue...". Another new window will open. Click "Start" to begin the autorouting process.

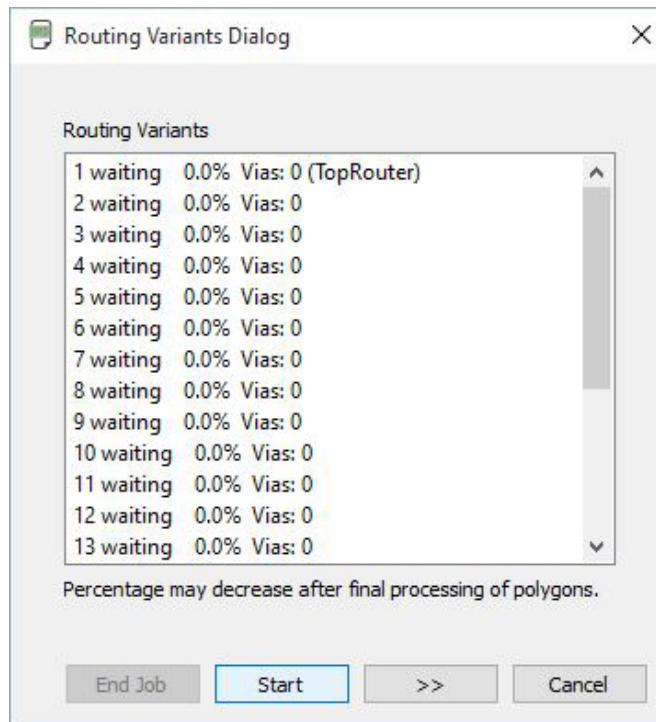


Figure 61: Preparing to start the Autorouter

As the Autorouter runs, the Routing Variants Dialog window will show rapidly fluctuating percentages. The board will also show several blue and red lines being connected and disconnected from components as the computer determines the optimal paths.

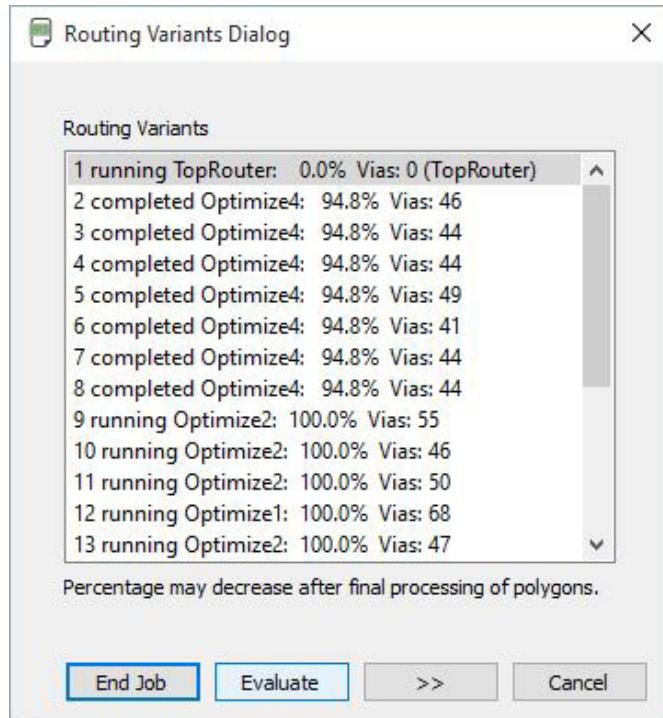


Figure 62: Running the Autorouter

When the Autorouter is finished, if the board can be routed, there are many variants with 100% connections.

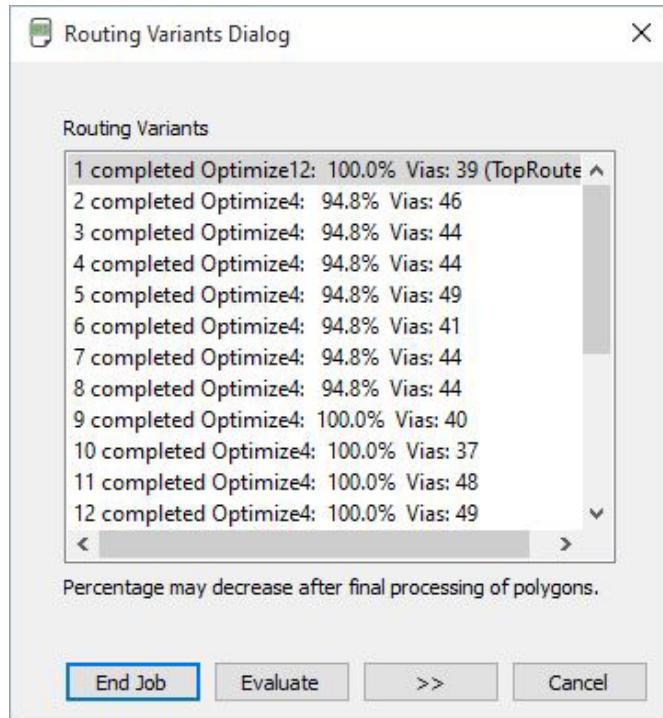


Figure 63: Finished Autorouter variants

It is important to also consider the number of vias for each Routing Variant. If you’re drilling your own board, more vias is more time consuming for the machine to drill. If you’re ordering a board from a PCB fabricator, the cost may change depending on the number of vias. Using the Routing Variants Dialog window, click through several different variants and you will see the board’s traces change. Usually, the first variant has the lowest number of vias. When you have decided on a variant, click on it and click End Job. Now, you have a fully routed PCB.

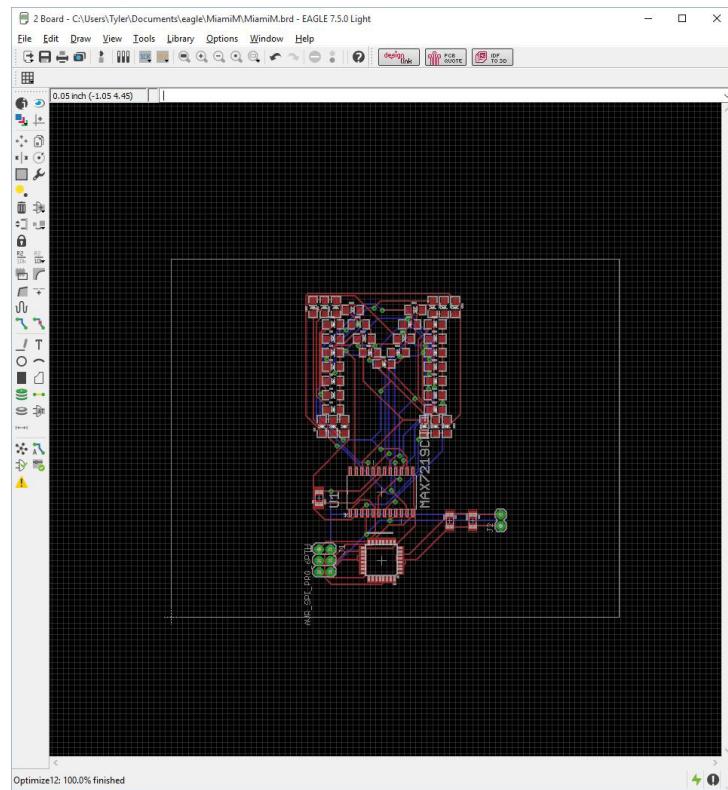


Figure 64: Routed board

Before you get too comfortable with that design, we are going to ripup all of the traces and move our components around some more. To ripup all of the traces, click on the Ripup button and then click the Group button. Select the entire board, right-click on the sheet, and select “Ripup: Group”. The board will look just like it did before we ran the Autorouter.

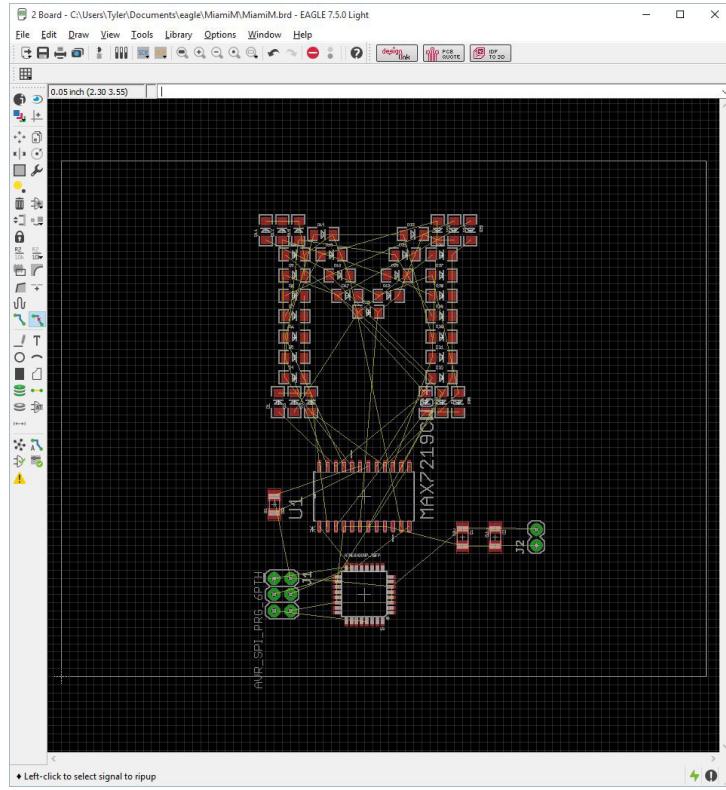


Figure 65: Board layout after using the Ripup tool to remove traces

Next, we're going to move all of the components over to the left side of the gray square and reduce the size of the board. Use the Group and Move buttons to move all of the components.

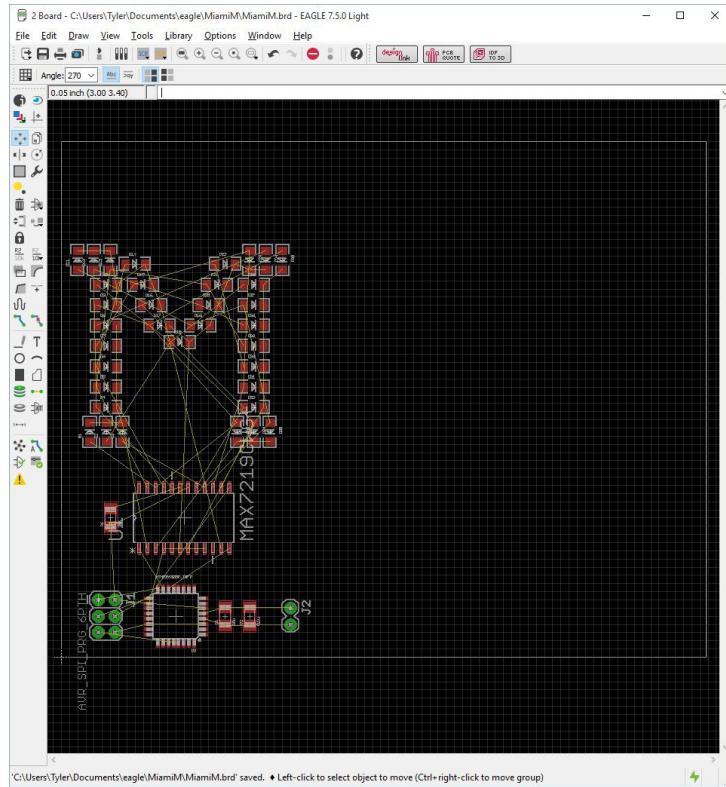


Figure 66: Moving all components to the side of the board

Using the Move tool, click on the middle of the gray line on the right side of the square. The line will now follow your mouse and you can adjust the size of the board. Do this for the right and the top of the square until the board fits nicely around the components.

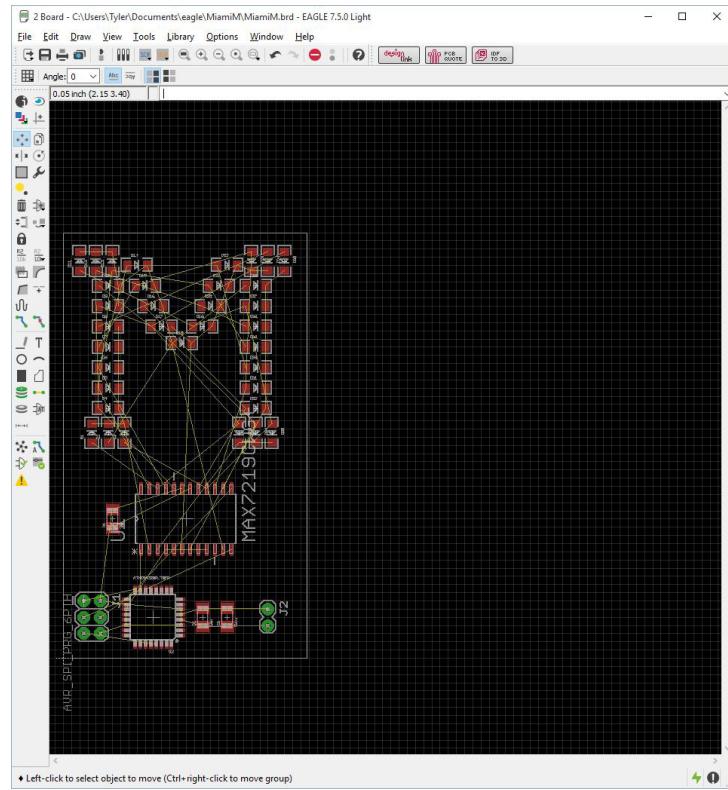


Figure 67: A PCB design with a much smaller footprint

Click on the Autorouter tool and rerun the Autorouter. More likely than not, the number of vias will increase, but the board is now much more compact, so that is a fair trade-off.

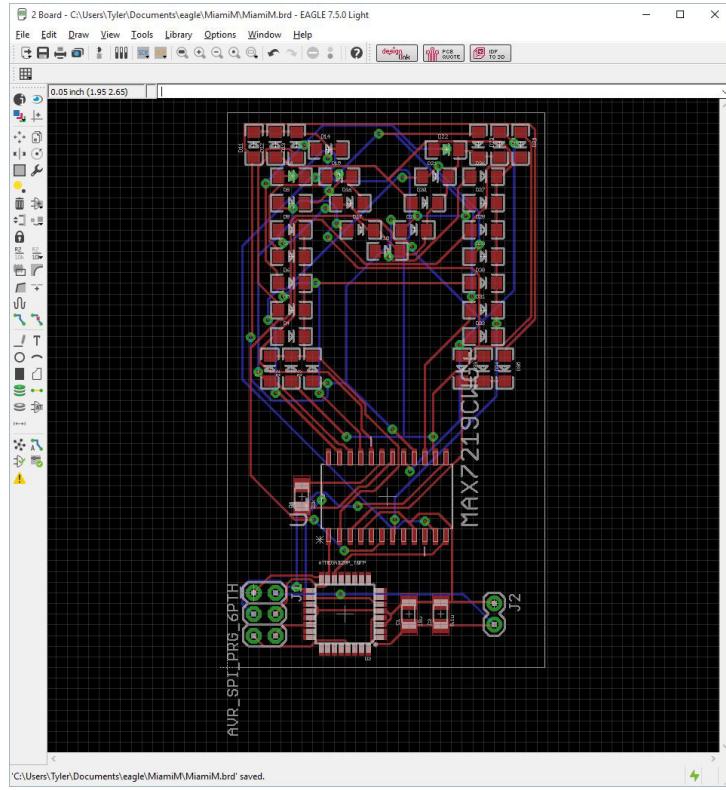


Figure 68: Completely routed board

An important step after laying out a board is to run the Design Rule Check tool. If you click on this tool, the DRC window will open. This window can be used to load new design rules or check the board with the currently loaded design rules.

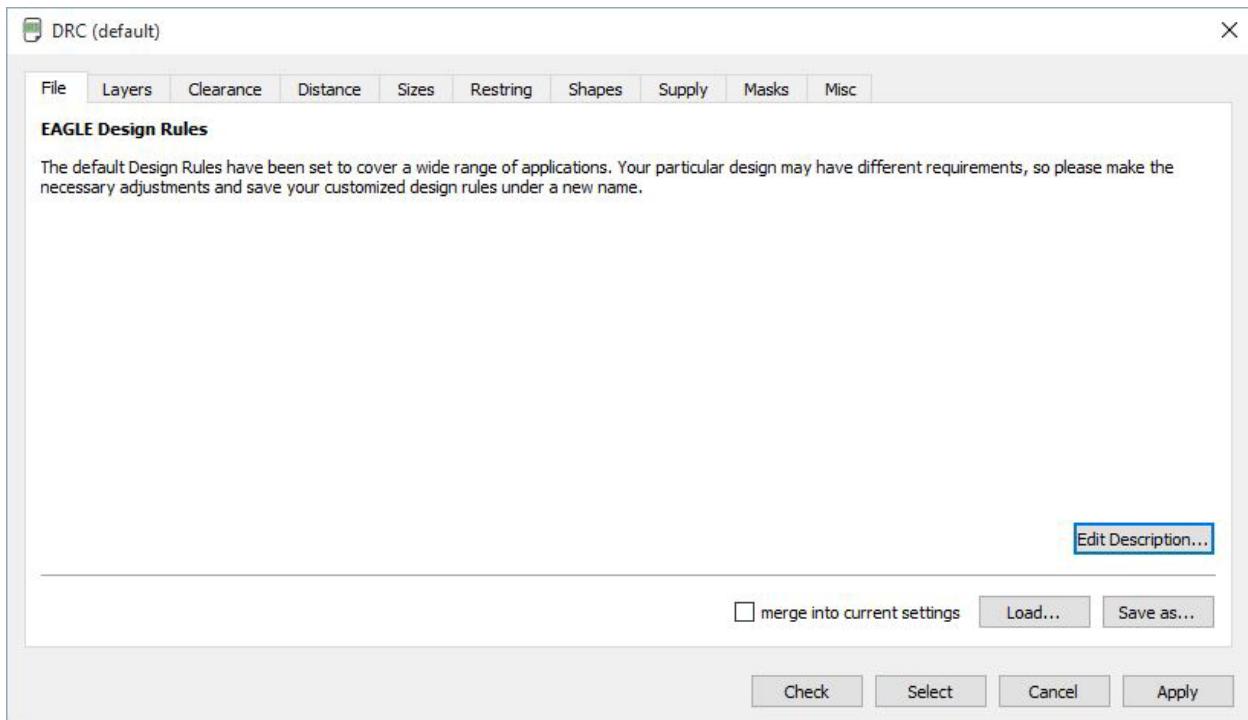


Figure 69: Design rule check

The “check” compares trace and component placement against the design rules that are loaded. The design rules contain a variety of parameters, such as trace width, minimum clearance, and layer thickness. Click the “Check” button to run the DRC. Because we routed this board with the Autorouter, EAGLE automatically tries to adhere to the loaded design rules. If there are errors, a new window will appear with the errors. If there are no errors, the bottom of the EAGLE window will display the text “DRC: No errors.”.

Now that we have laid out our board and checked it with the DRC, there are several things we can do to make it more production-worthy. You may have noticed that many of the component names are placed haphazardly across the board. There are also a variety of fonts, sizes, and orientations for the text on the board. We are now going to take several measures to address these issues.

First, we will use the Smash tool. This tool will allow us to separate the names and values of components from the components themselves. We can then resize, move, or even delete the names and values. Click on the Smash tool and then click on the origin of the 6-pin header.

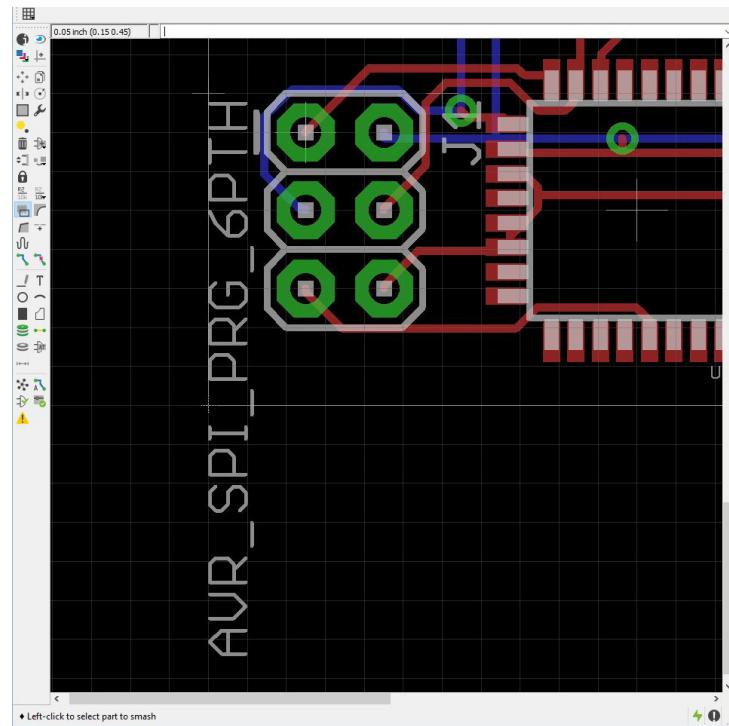


Figure 70: Using the Smash tool

We now have three separate origins: one for the part value (AVR_SPI_PRG_6PTH), another for the part attribute (J1, which stands for jumper 1), and a final one for the original component. Click the Delete tool and then delete the part value and the part attribute. You can do this easily by clicking on their origin. If you accidentally click on the part itself, EAGLE will give you a warning saying that it can't "backannotate" that operation.

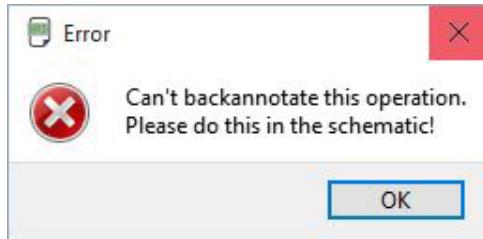


Figure 71: "Backannotate" error

Without the header text, the board is already looking quite a bit cleaner.

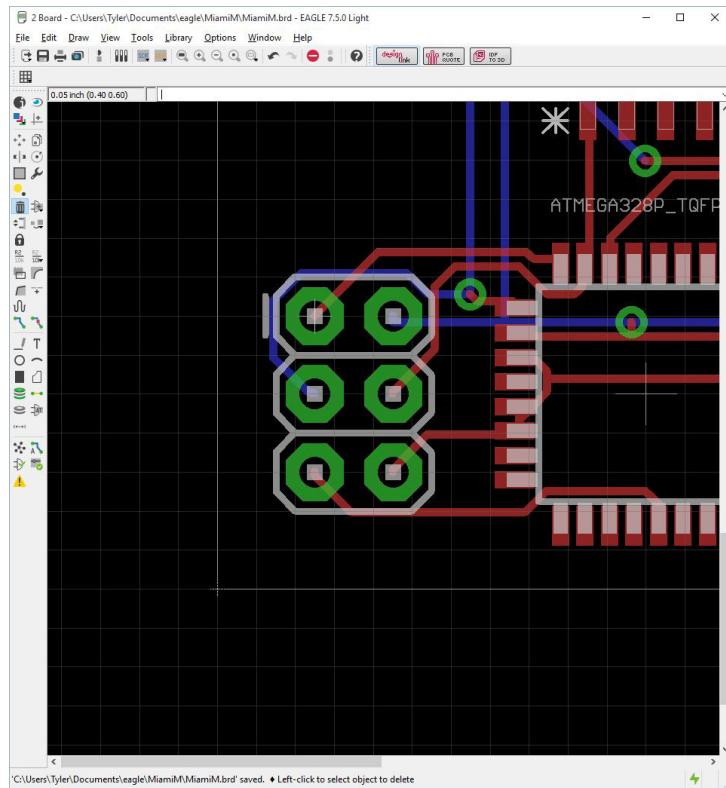


Figure 72: The 6-pin header without text

Next, we are going to add text of our own. Click on the Text tool to open the Text window. Type "ISP" into the textbox and click OK.

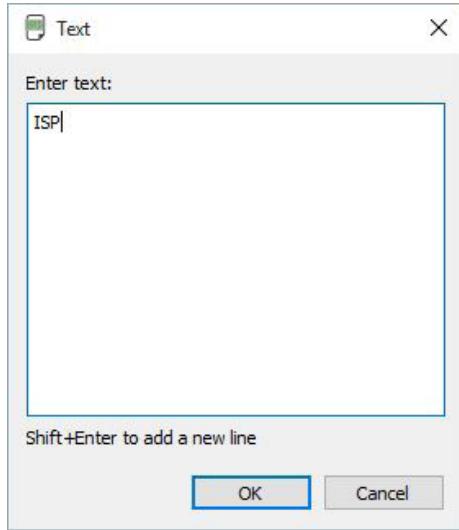


Figure 73: Adding text to the board

Before you place the text, take a look at the top toolbar. Next to the grid button, there is a drop-down menu. This lets you select the layer on which to place this text. Change the layer to “21 tPlace”. This will put the text on the top silkscreen layer, so if you order these boards from a fabricator, the text will actually be printed on top of the colored (often green) portion of the board. Place the text down near the 6-pin header. If you accidentally placed the text down on the wrong layer, you can change the layer by right-clicking the text, opening the Properties window, and changing the layer there.

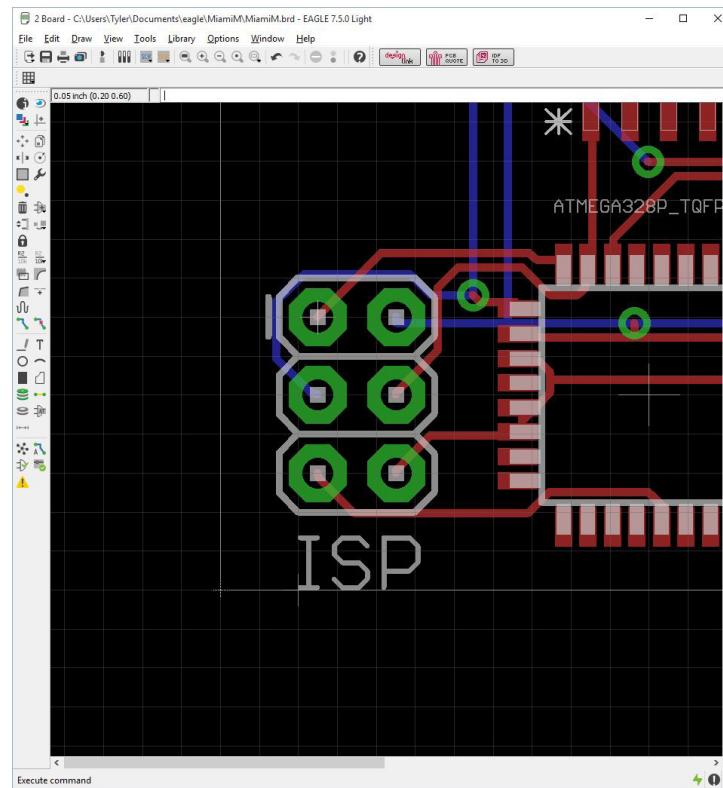


Figure 74: Placing text on the board

Now that we've placed the text, we can change the font to better fit the board. Click on the Change tool and then hover over "Font" to select "vector". If you click on the text we added, you may or may not see a change. Depending on your version of EAGLE and the User Interface settings, vector fonts may be automatically enabled because it is the de facto standard for text on PCBs. It is useful, however, to know how to change the font style.

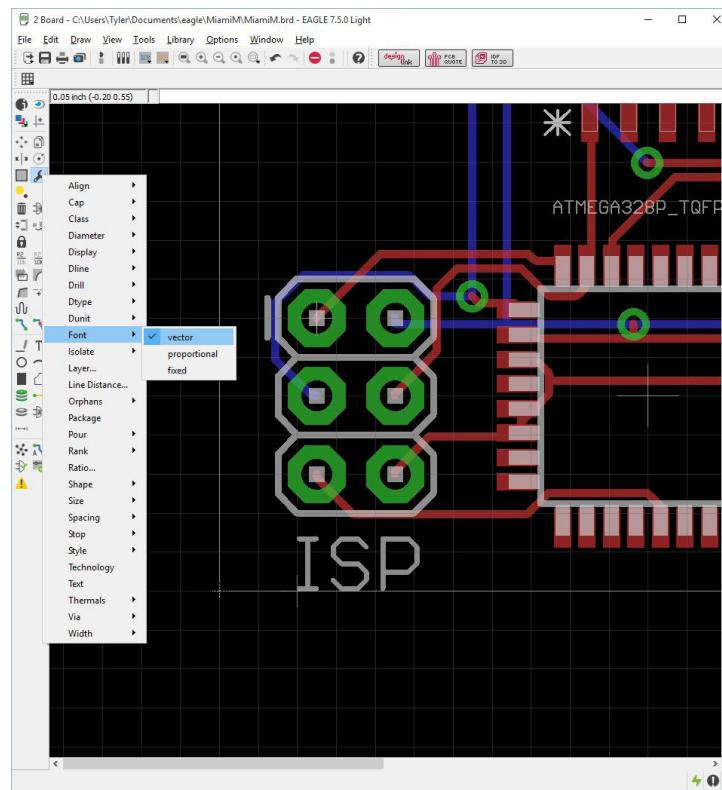


Figure 75: Changing the font

We are now going to change the ratio of the text. This ratio changes how bold the font appears. Right now, it is a rather light font, so we will be increasing the ratio. Click on the Change tool and then click “Ratio...”.

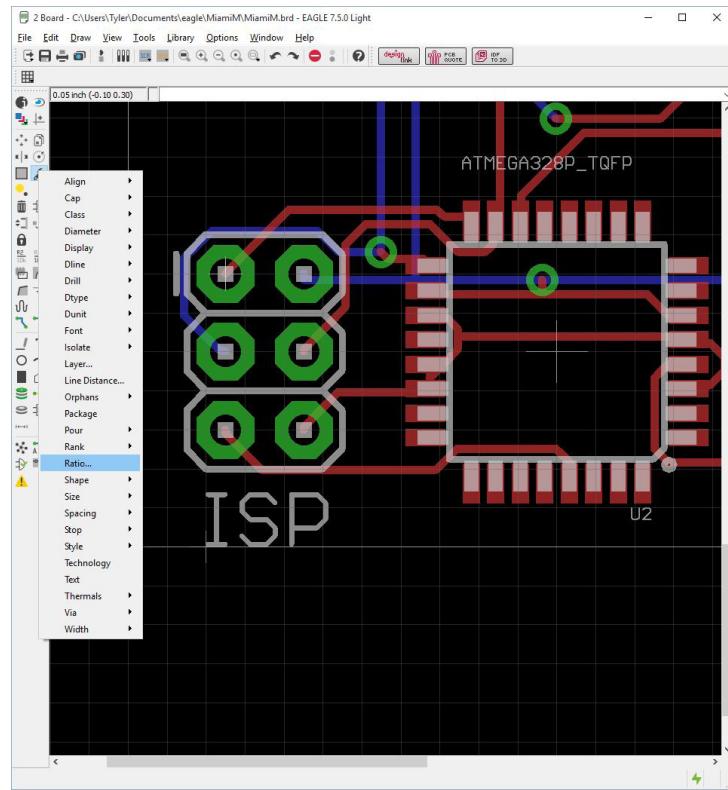


Figure 76: Changing the ratio

The Change Ratio window will appear. Type “12” into the textbox and click OK.

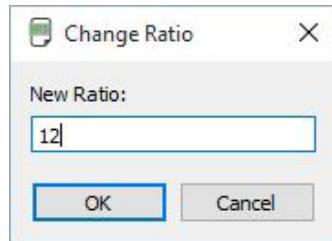


Figure 77: Setting a new ratio

Now, any text you click on will be changed to a 12% ratio. Upon clicking on the “ISP” text, you’ll notice it becomes bolder.

Next, we are going to change the size of the text because, as you can see from the above images, it's slightly overlapping with the bottom of the board outline. Click the Change tool and then hover over "Size" to select "0.05". Now, click on the origin of the "ISP" text.

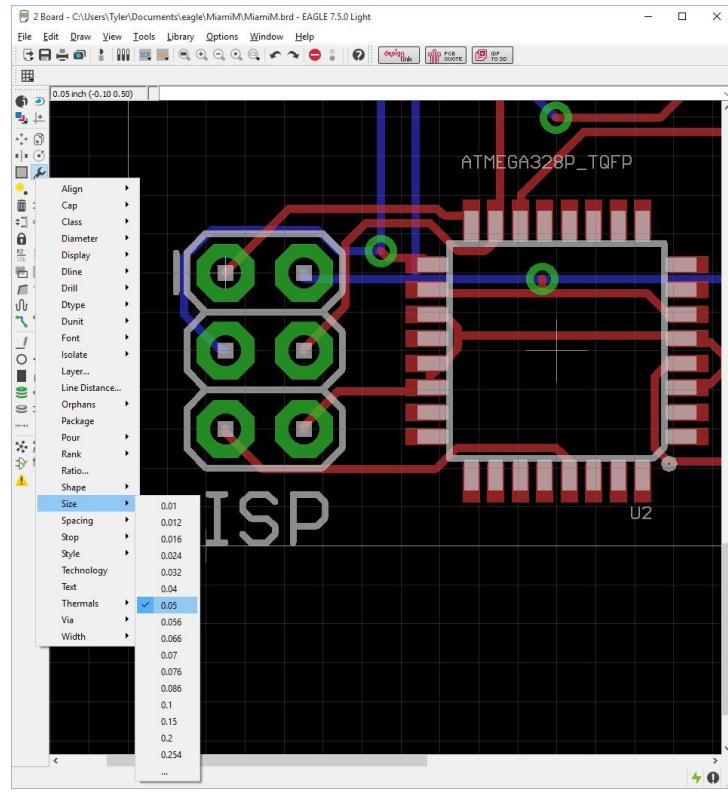


Figure 78: Changing the text size

The text size has been reduced quite a bit. Use the Move tool to place the text close to the header. You may have to use the ALT key to increase the move resolution. It is worth noting that the line you see on the side of the 6-pin header signifies the location of the first pin on the schematic. This is important because, when the board is completely assembled, you will need to know which direction to connect the programmer.

Your ISP header should resemble the header shown in the image below.

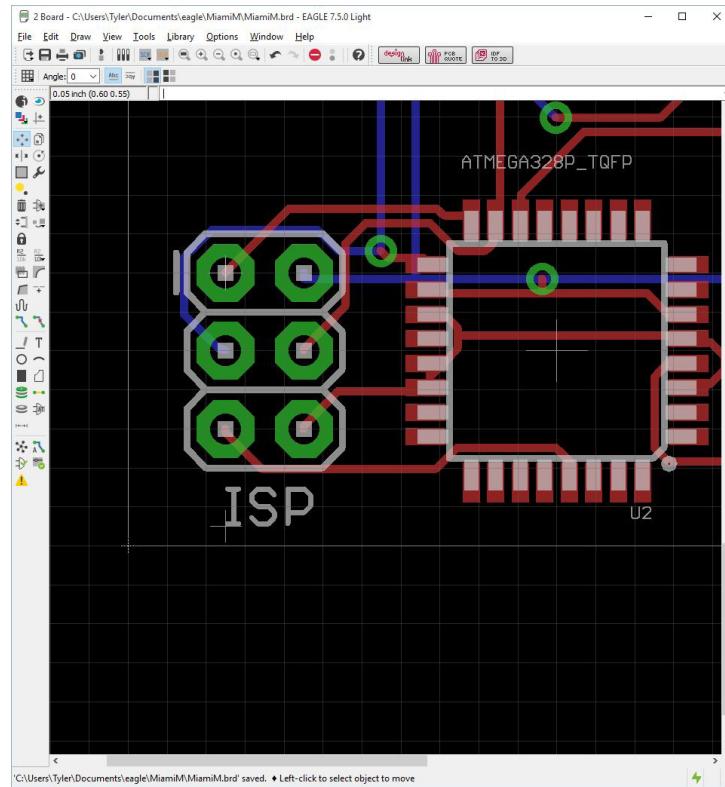


Figure 79: Finalized ISP header text

Now that we've successfully smashed, deleted, placed, resized, and moved text, we are ready to repeat that process for each of the components. Let's move on to the MCU next. Use the Smash tool to separate the text from the component and then use the Delete tool to remove it from the board. Then, add the text "MCU" near the microcontroller. Luckily, EAGLE remembered that we changed the font to vector, the size to 0.05, and the ratio to 12%, so we will not need to modify the text any further after placing it.

Your board should look similar to the following image.

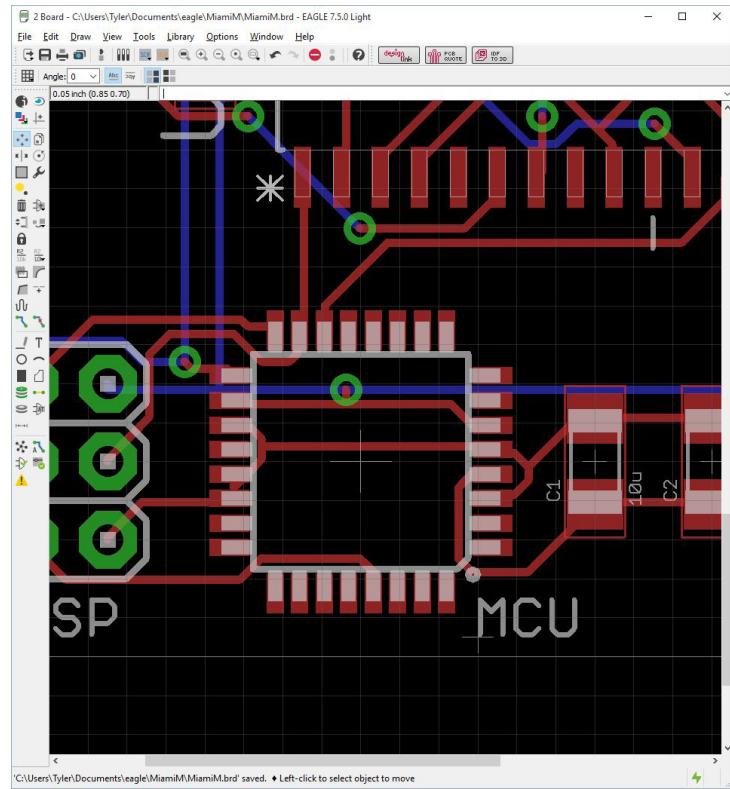


Figure 80: Finalized MCU text

Moving on to the capacitors, use the Smash tool to separate the text from the components. Before deleting anything, let's take a look at what the text actually says. On each capacitor, it indicates the name of the capacitor and the value associated with the capacitor. Since this is pertinent information, we will keep it on the board. But we do want to change the font, size, ratio, and position. Change these settings like we did for the previous two components, but make sure to set the size to "0.032". After changing the settings, position the text to your liking. If you're having trouble centering the text on the capacitor, try adjusting the grid settings using the Grid tool. The Grid settings that we used are shown in the image below.

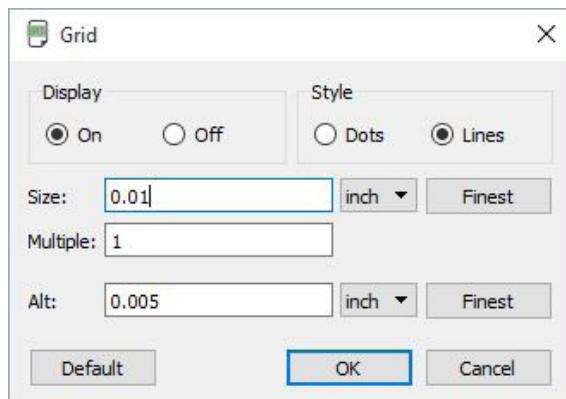


Figure 81: Adjusting the grid settings

Once you've resized and placed the text, your board will look similar to the following image.

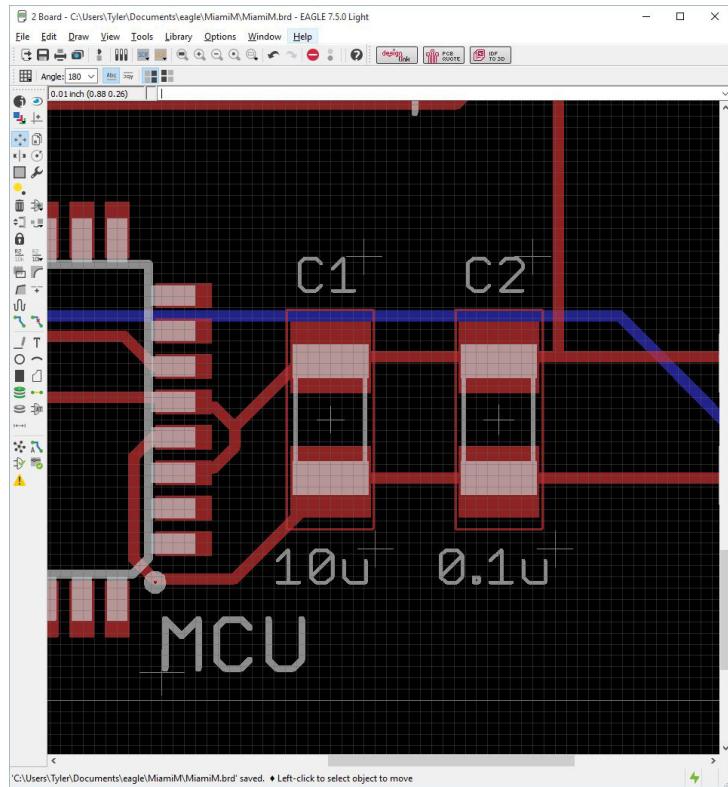


Figure 82: Finalized capacitor text

For the 2-pin jumper, we will be removing the attribute “J2” and adding text to label VCC and GND. Use the Smash tool and then delete the attribute text.

Now, use the Text tool to add the text “VCC” and place it next to the VCC trace. If you’re not sure which is which, right-click on a trace and look at the text in the very bottom of the EAGLE window. It will say “Signal: TRACE_NAME”, where TRACE_NAME is either VCC or GND. You will also need to change the size of the text back to “0.05”.

After placing the VCC text, repeat the process with the GND text.

The jumper in your board will resemble the jumper shown in the image below.

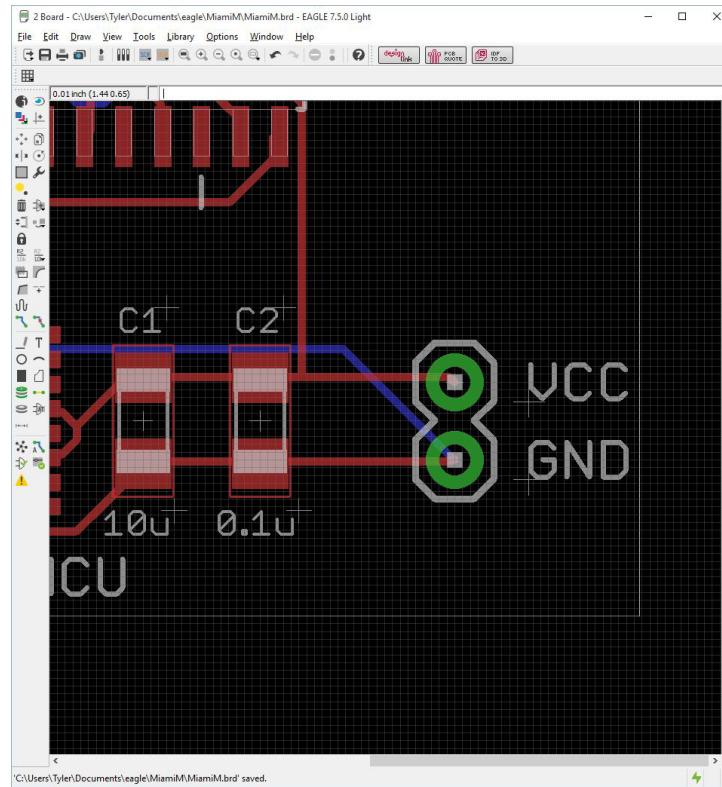


Figure 83: Finalized 2-pin header

Next, use the same technique that we used on the capacitors to change the text on the resistor.

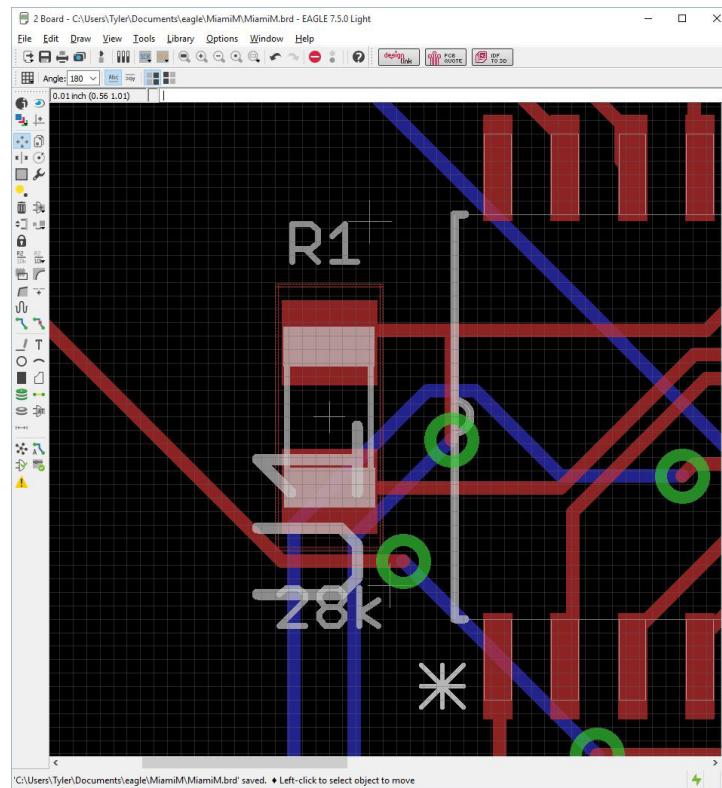


Figure 84: Finalized resistor text

After completing the resistor, let's move on to the MAX7219 chip. Smash and delete the "U1" attribute, as it's overlapping with our resistor. Then, resize the name text and move it into the center of the outline of the chip. This text is mainly to let the person assembling the board know what component is supposed to go there. Your board should now look like the board in the following image.

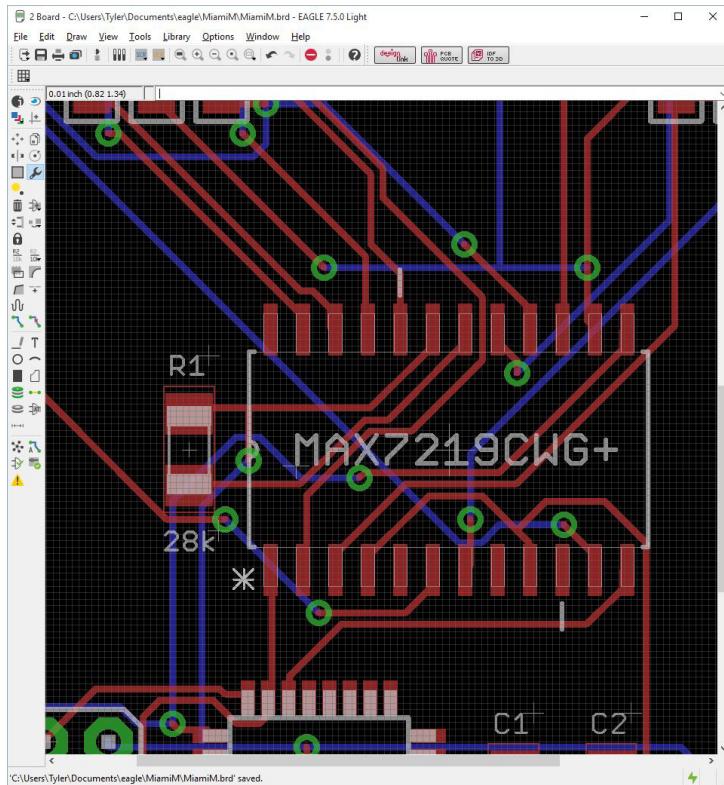


Figure 85: Finalized LED driver text

The last step in fixing the aesthetics of the components on this board is to remove the name of each diode in the "M". Because all of our LEDs are going to be the same, there isn't much of a need to list out which LED is which. Smash each of these components and delete the attribute text.

Your LED design may vary from the image below, but there should be no attribute text shown on the design.

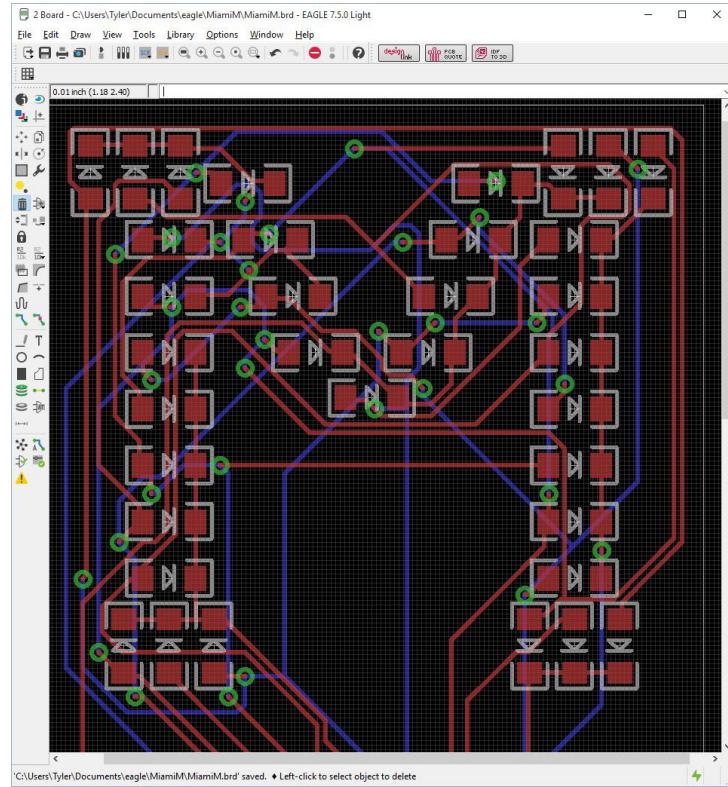


Figure 86: Removed all LED text

With all of the text for each component adjusted, your board should look similar to the following image.

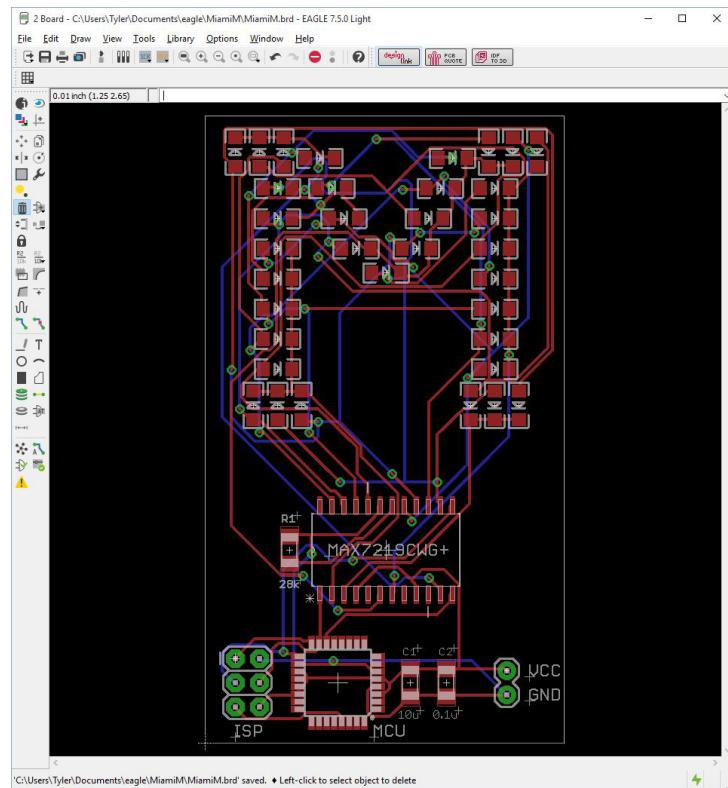


Figure 87: Board design with improved aesthetics

The final step is optional: add some text of your own to make it clear you designed this board. The text we added to our board says “Miami M”. Feel free to add text to any open location on the board.

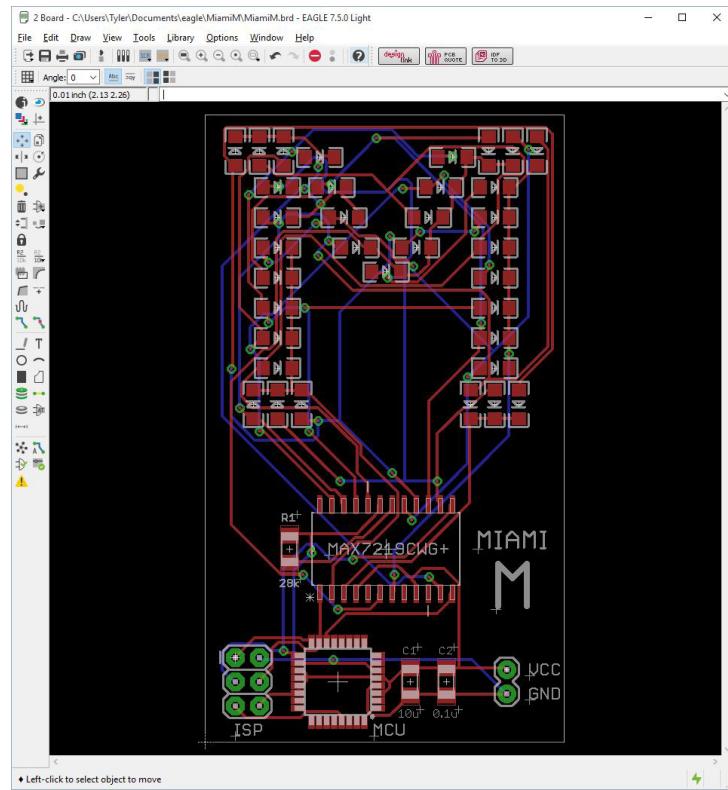


Figure 88: Final board design

You have now designed your first board in EAGLE. We are ready to convert the board files into Gerber files, which are the industry standard files for PCBs.

Chapter 4

Generating Gerber Files

Now that we have completed our board design, we need to export the design to a format that can be utilized by PCB manufacturers, such as OSH Park. This format, a collection of files, is colloquially known as “Gerber files” or merely “Gerbers”. When generating Gerbers from the board design, the user has the ability to select layers on the board to be packaged into different Gerber file extensions. These files are normally placed in a single folder, which is then compressed and uploaded to a PCB fabricator’s website. To generate Gerbers in EAGLE, we need to become acquainted with a new tool:



CAM PROCESSOR – Opens the CAM Processor, which is used to generate Gerber files

Using the CAM Processor tool, you can generate Gerber files with whatever extensions you wish. Often, PCB fabricators require a specific set of file extensions and layers to be included within each extension. Because of this requirement, EAGLE includes the ability to load in a job file, which contains ready-made layer lists for each Gerber file extension. Many fabricators are aware of this feature in EAGLE and have created custom CAM jobs that, when run, output all Gerber files in the format required for fabrication.

The prototype PCB fabricator recommended earlier in this primer, [OSH Park](#), offers a CAM job for download here: <http://docs.oshpark.com/resources/OSHPark-2layer-Eagle7.2.cam>.

Download the file to your computer. Then, click on the CAM Processor tool in EAGLE to open the CAM Processor window.

The CAM Processor window will look identical to the following image.

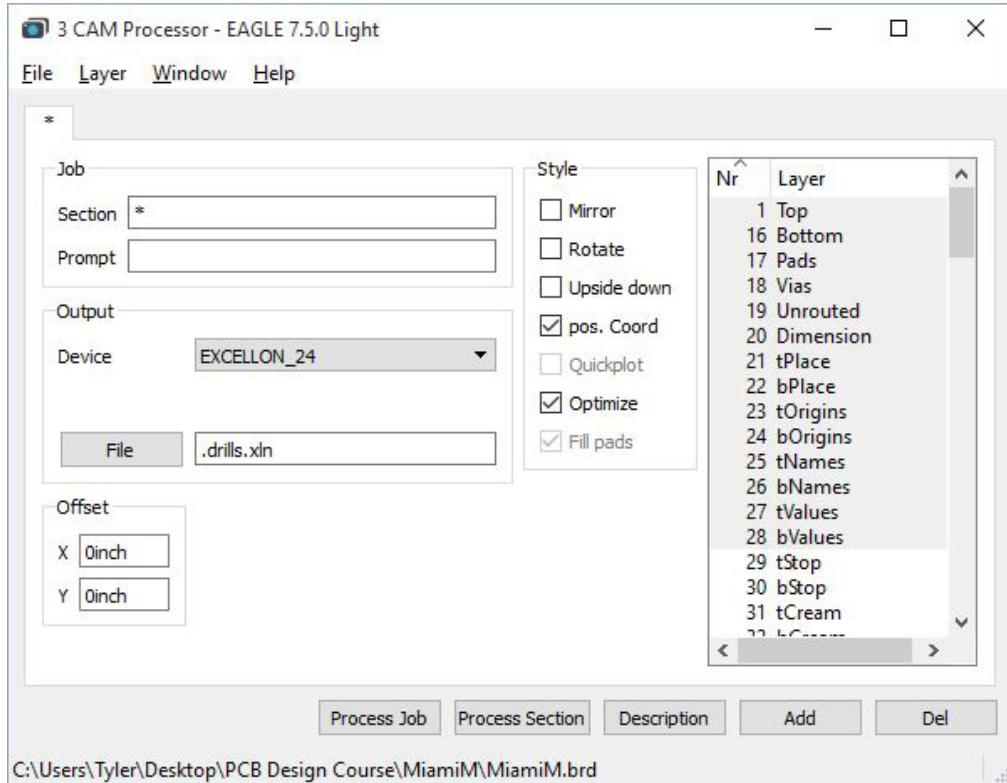


Figure 89: Cam Processor

In the CAM Processor window, click on File > Open > Job... in the main toolbar to open the file browser.

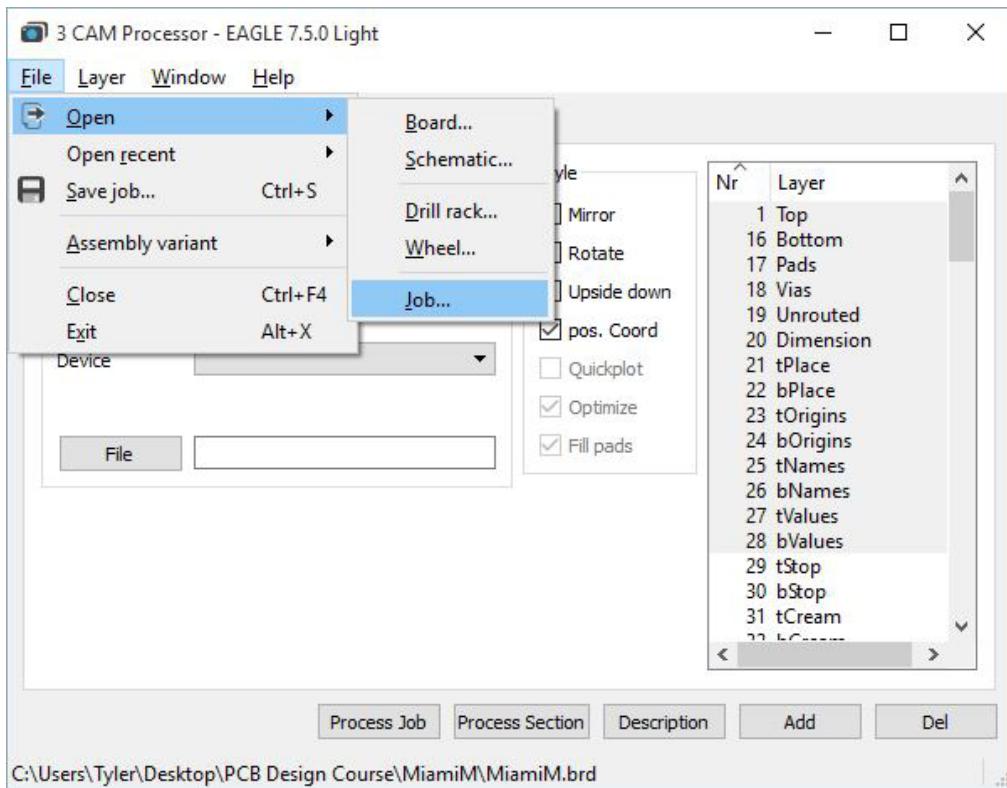


Figure 90: Opening a CAM job

Using the file browser, navigate to the location of the CAM job you downloaded from OSH Park. Select the job file and click the Open button.

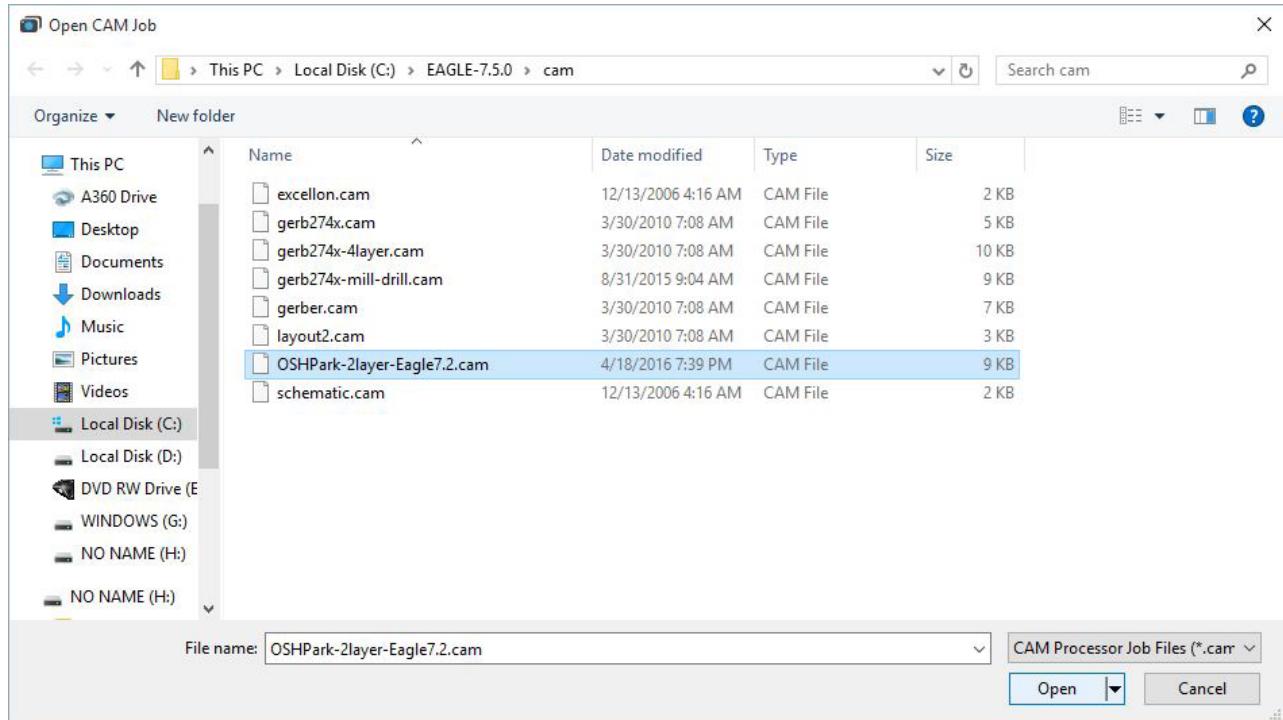


Figure 91: Selecting a CAM job

The CAM Processor window will now look exactly like the following image.

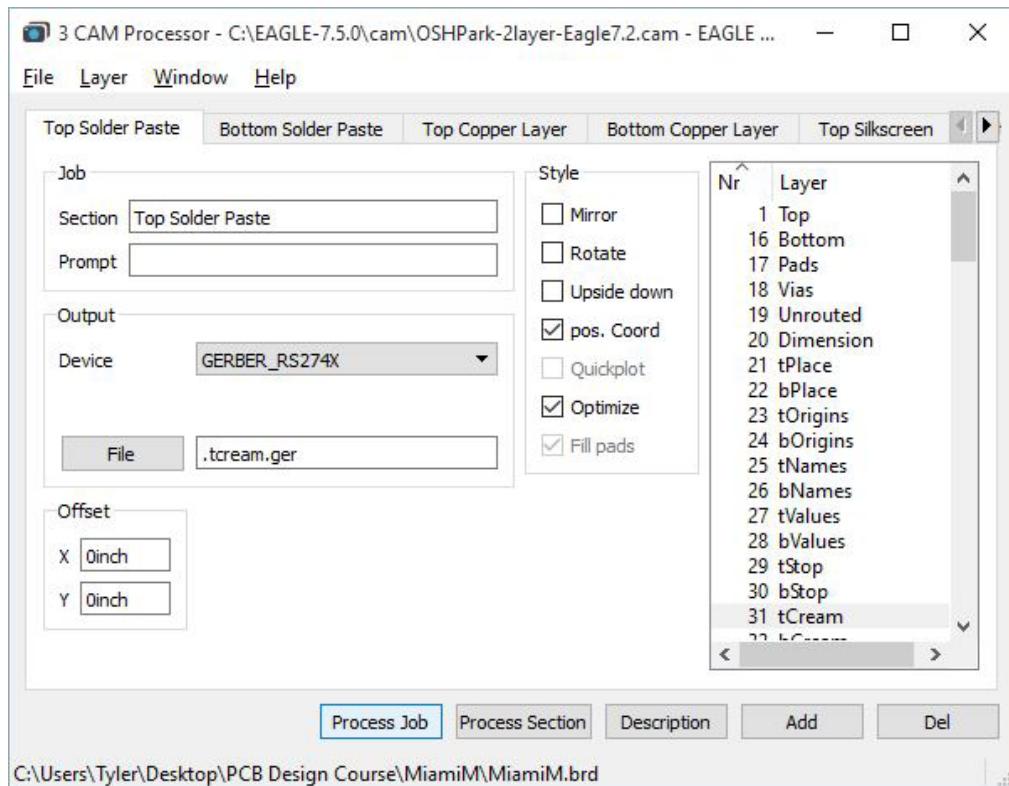


Figure 92: Processing a CAM job

The final step to generate Gerbers, according to OSH Park's specification, is to click the Process Job button. Once you click this button, the CAM job will run and several new files will be created in your project directory. Each file will have the prefix BOARD_NAME appended to it, where BOARD_NAME is the name of your .brd file. Navigate to the project directory in your file browser and create a new folder called "Gerbers". Click and drag each of the following files into the new folder:

- BOARD_NAME.bcream.ger
- BOARD_NAME.boardoutline.ger
- BOARD_NAME.bottomlayer.ger
- BOARD_NAME.bottomsilkscreen.ger
- BOARD_NAME.bottomsoldermask.ger
- BOARD_NAME.drills.xln
- BOARD_NAME.tcream.ger
- BOARD_NAME.toplayer.ger
- BOARD_NAME.topsilkscreen.ger
- BOARD_NAME.topsoldermask.ger

Finally, compress the "Gerbers" folder. The compressed folder can now be uploaded to OSH Park so that you can order your PCBs. In the event that you choose a different fabricator, be sure to look at their Gerber requirements or download and run their CAM job before uploading the files.

Congratulations on completing your first printed circuit board and generating the industry standard Gerber files. We hope that this guide has helped you add a new skill to your electrical and electronics engineering repertoire, and that you consider applying these skills and your innovative ideas to consumer electronics entrepreneurship.