BMME 580 Microcontrollers I

Lab 1: EagleCAD Tutorials

Ross Rucho

To get credit for this laboratory exercise, and all subsequent laboratory exercises, you will need to use a template form and fill in each space as indicated. Each laboratory has a separate template form that you will be given at the beginning of each week for the new laboratory exercise. This is the template form for *Lab 1 - EagleCAD*. Except for the instances in this paragraph, wherever you see *red text in italic*, that means you need to fill in the information indicated. For example, rename this template in the format "Lab 1 – EagleCAD – *your name.doc*", then also fill in your name at the top of this page where indicated in red italic.

When your work is complete, please save it as a MS Word document and then email the completed lab form to both emails: bob@unc.edu and bob@bobslab.com and to the TA.

Generally, when you submit an assignment by email I will respond by telling you whether or not you received full credit for the assignment. You may ask for the current status of your total credit for assignments at any time.

We begin the course by learning to use the basic functions of EagleCAD. EagleCAD has been selected because it is the electronic design software of choice for many start-up and small biotechnology businesses, makers, many academic programs, and electronics design businesses. The cost of entry is very low, and there is a rich user support community. Many PCB manufacturing facilities (a.k.a. "board houses") accept original EagleCAD files for their manufacturing processes. So EagleCAD is convenient for many reasons. However, the software can be frustrating to learn at first because it has evolved from a command-line based program and it still has a lot of old-school vestiges. But once you learn it, it is a great engineering tool.

EagleCAD is available on all of the workstations in the BME Lab, but you can still download a free student version of EagleCAD. It is a limited student version, but it is very good. If you want to work on EagleCAD on your own computer the newer versions work on all platforms. See the links on the next page. Better yet, if you plan to design circuits in the future, purchase a copy.

Note that the new versions of EagleCAD are somewhat different, with different icons, and some are a bit quirky, so please expect some glitches when you do the tutorials. In some cases you'll just need to start over if it hangs up, but once you get used to it, EagleCAD is a simple and very useful application for electronic design. Please cheerfully expect a bit of frustration at first.

BMME 580 Microcontrollers I

OPTIONAL: Before starting the assigned work, I strongly recommend that you do a quick review of the basic information and knowledge that will be assumed for this course. Very good summaries can be found on sparkfun.com. Each of these will only take a few minutes to read:

If you are having anxiety about your recollection of basic electronics, you can start here: https://learn.sparkfun.com/tutorials/voltage-current-resistance-and-ohms-law (10 minutes)

https://learn.sparkfun.com/tutorials/voltage-dividers (15 minutes)

https://learn.sparkfun.com/tutorials/how-to-use-a-multimeter (25 minutes)

Then be sure to review the following since the assumption for this course is that you already know this material:

https://learn.sparkfun.com/tutorials/pcb-basics (about 6 minutes to read)

https://learn.sparkfun.com/tutorials/how-to-read-a-schematic (15 minutes)

REQUIRED: During the first three weeks of this course you need to complete the EagleCAD tutorials available from sparkfun.com. Start here:

https://www.sparkfun.com/EAGLE From this page you can select from many links.

The lab workstations have EagleCAD installed. You can download the student (freeware) version of Eagle to your computer. Either way, assignment #1 is for you to gain reliable access to EagleCAD.

https://learn.sparkfun.com/tutorials/how-to-install-and-setup-eagle

DO THE FOLLOWING TUTORIALS BY FOLLOWING THE INSTRUCTIONS THEN PASTE A SCREEN SHOT OF YOUR FINAL WORK IN THE SPACES INDICATED ON THE FOLLOWING PAGES. EACH STUDENT MUST DO THIS ASSIGNMENT INDIVIDUALLY, NOT IN A GROUP. YOU CAN ASK FOR HELP, YOU CAN GIVE ADVICE, BUT YOU MUST DO THE TUTORIALS INDIVIDUALLY.

https://learn.sparkfun.com/tutorials/using-eagle-schematic

https://learn.sparkfun.com/tutorials/using-eagle-board-layout

https://learn.sparkfun.com/tutorials/designing-pcbs-smd-footprints

https://learn.sparkfun.com/tutorials/designing-pcbs-advanced-smd

Then, read the following tutorial. You do not have to follow the steps, just read it.

https://learn.sparkfun.com/tutorials/making-custom-footprints-in-eagle

BMME 580 Microcontrollers I

OPTIONAL: Finally, to make EagleCAD *much* more useful especially if you downloaded it to your own computer, you can download their component library files:

https://github.com/sparkfun/SparkFun-Eagle-Libraries

If you are serious about electronic design, you should take the time to download the component libraries and save them to your computer. this will save you enormous time in your designs in the future.

When you finish each tutorial, take a screen shot, paste it into this MS Word document in the spaces indicated on the following pages and save this document. Then email a copy of the completed document to the instructor AND the TA for credit once you have finished the EagleCAD tutorials.

Over the course of your professional career you will be able to find additional more advanced (and more up-to-date) EagleCAD tutorials on SparkFun and elsewhere online. I suggest you consider doing these tutorials every few years as new ones become available.

Finally, I suggest that you struggle with EagleCAD a bit before asking for help. You may contact me at any time if you need help, but preferably you can ask another student in the lab for assistance, because that results in a learning experience for both of you.

Screenshots (when you complete each tutorial) are to be pasted on the following pages.

And just for reference, the original *old-school* Sparkfun EagleCAD Tutorials are still available at:

https://www.sparkfun.com/tutorials/108

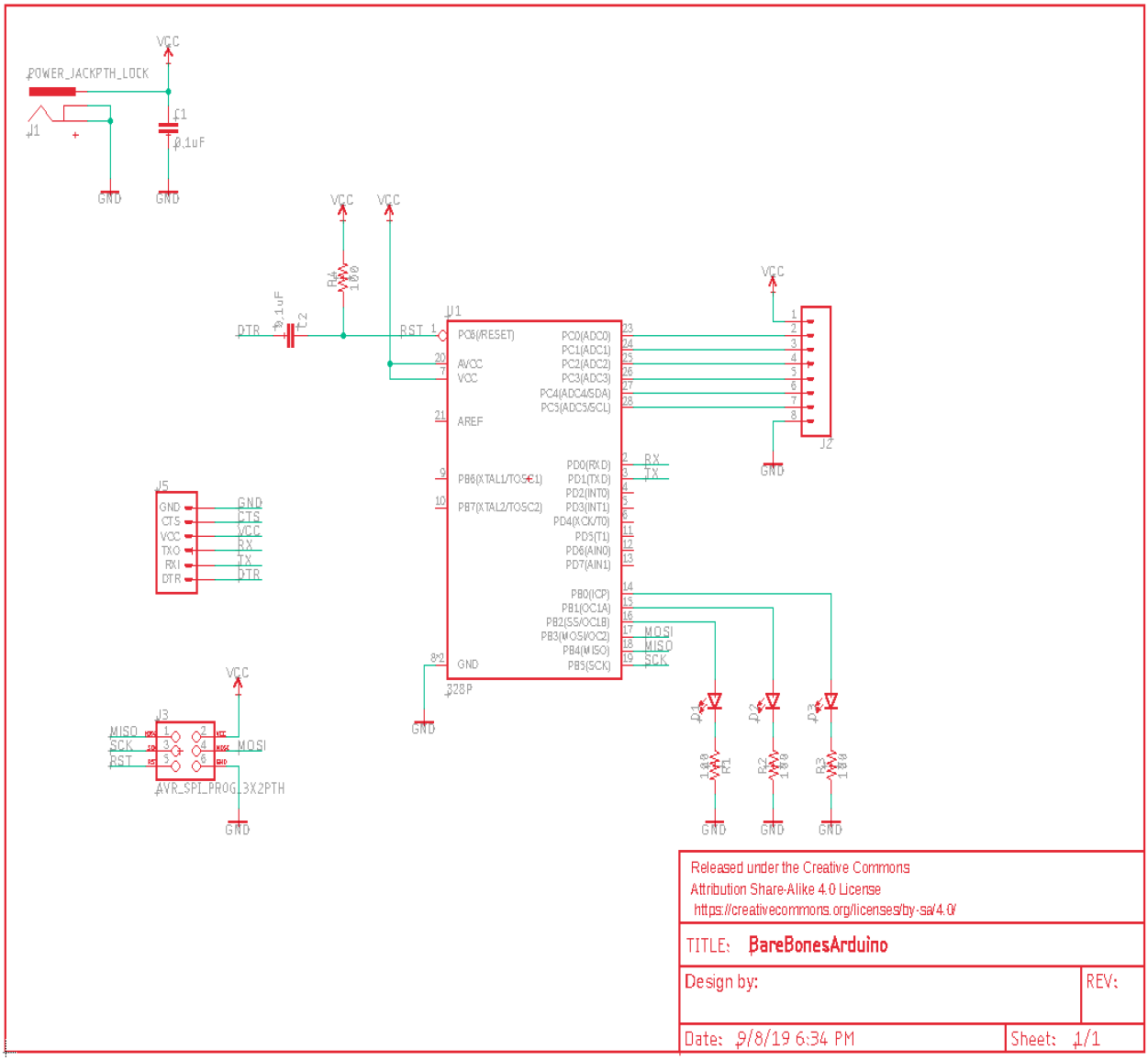
https://www.sparkfun.com/tutorials/109

https://www.sparkfun.com/tutorials/110

You are not required to do these three older tutorials for this class, but they contain some useful material. By comparing these to the assigned (new) tutorials you can see how much better the tutorials (and EagleCAD) became over the span of just a few few years.

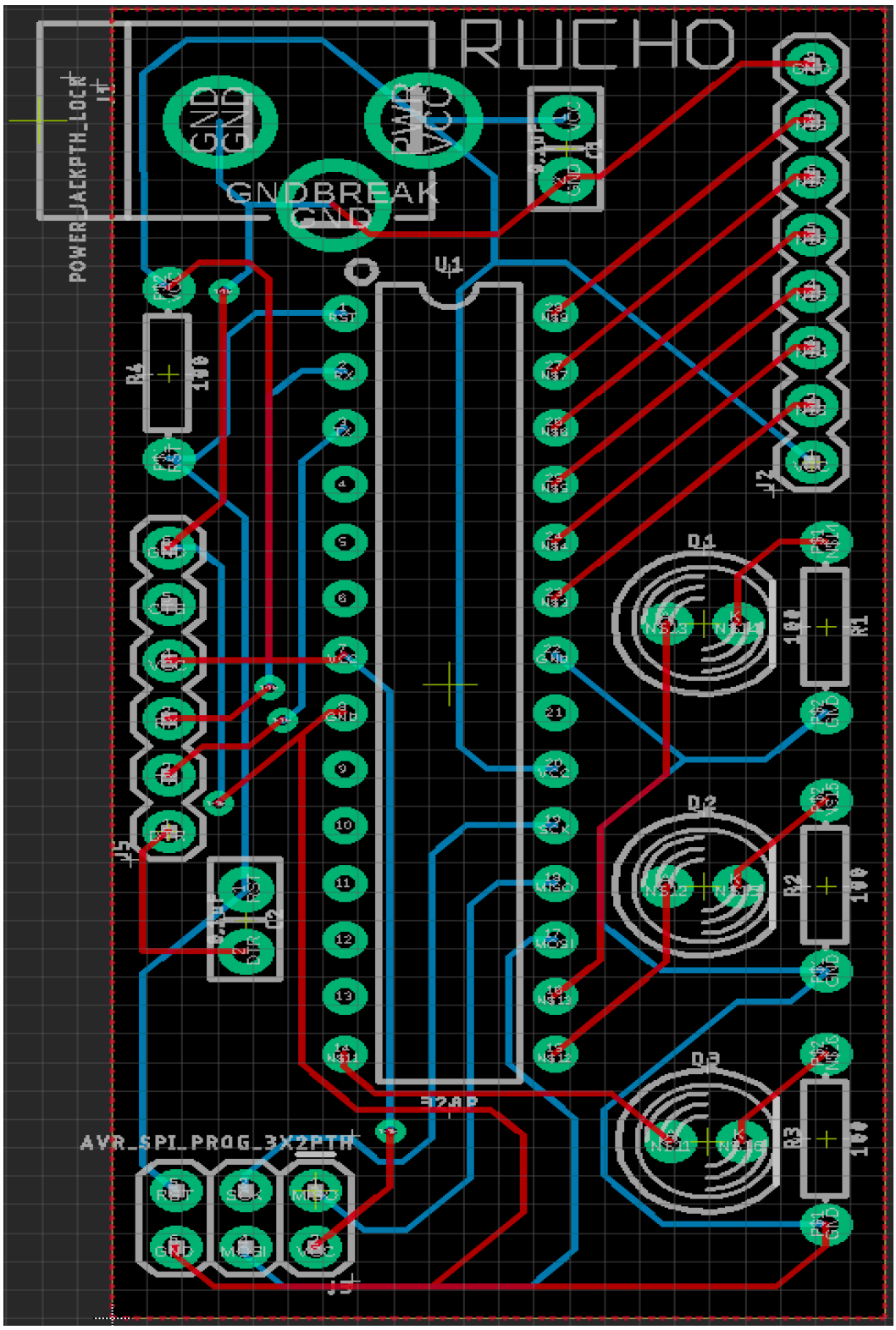
BMME 580 Microcontrollers I

EagleCAD Tutorial: Using Eagle Schematics – Your Screenshot



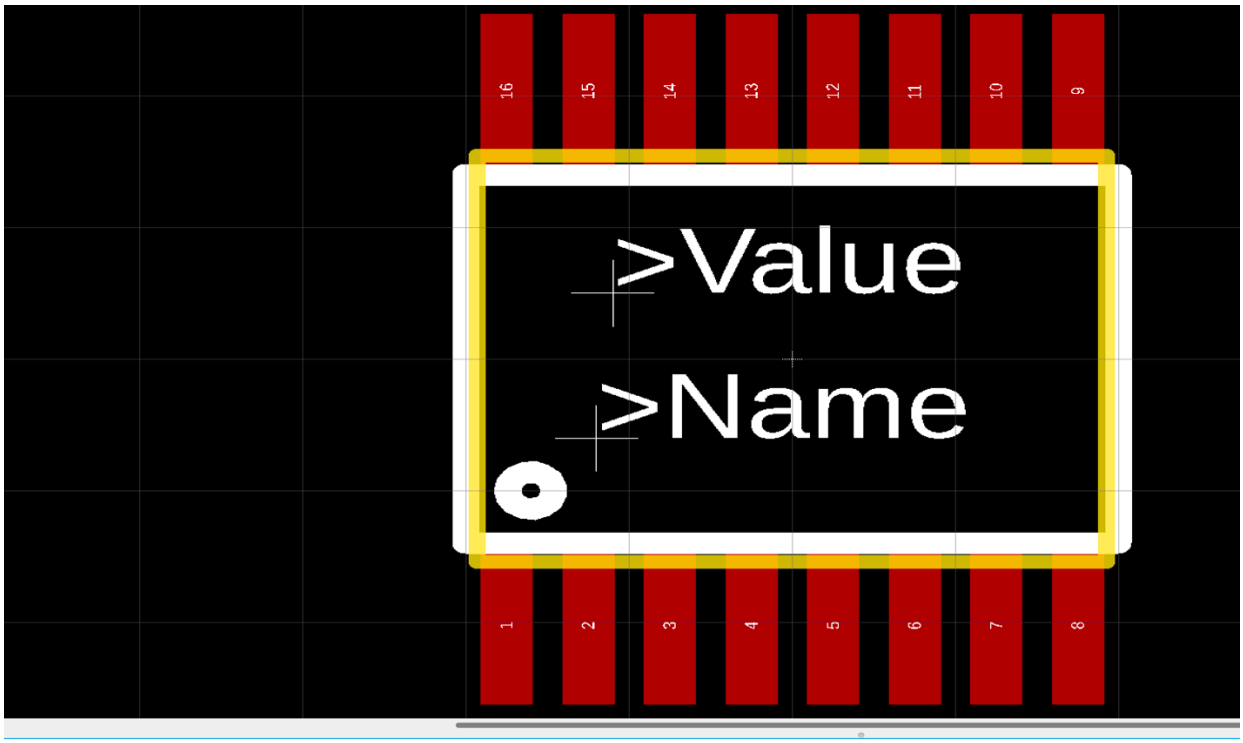
BMME 580 Microcontrollers I

EagleCAD Tutorial: Using Eagle Board Layout – Your Screenshot



BMME 580 Microcontrollers I

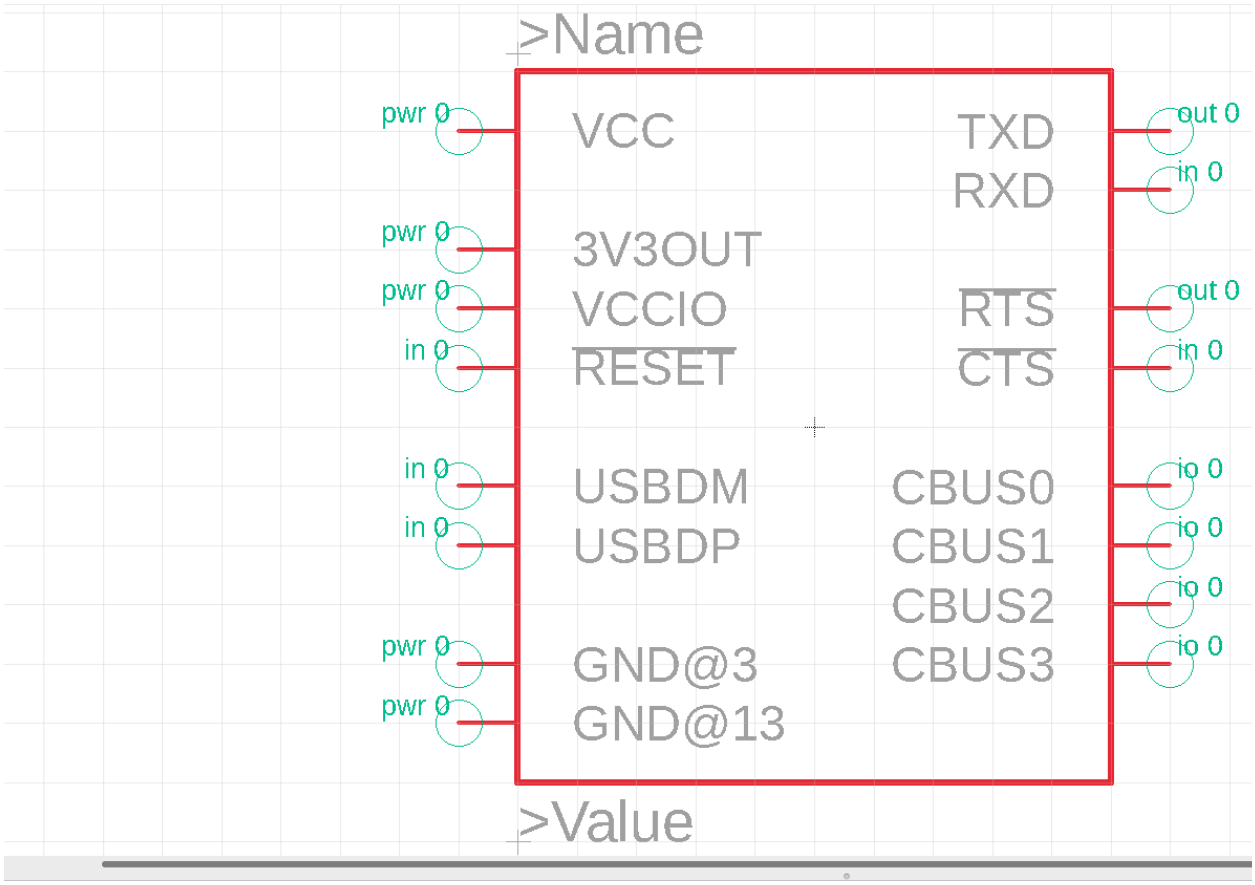
EagleCAD Tutorial: Designing PCBs SMD Footprints – Your Screenshot



Description

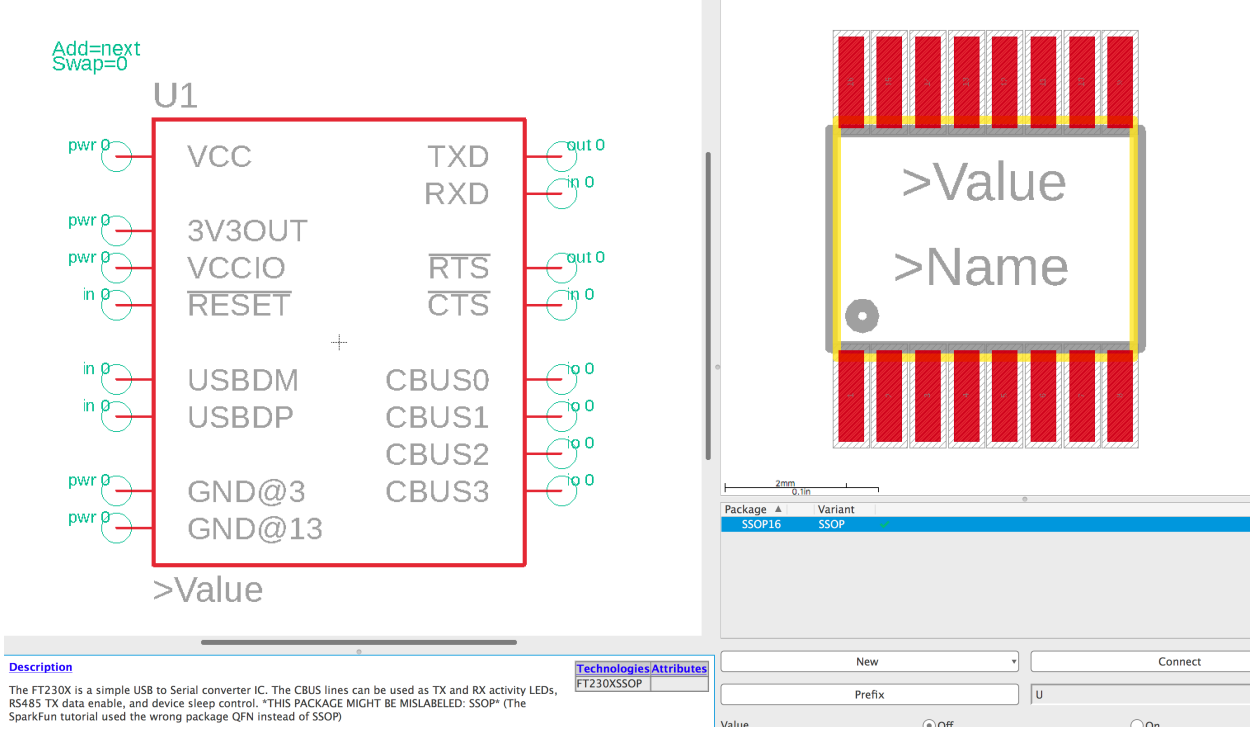
Description: This footprint is used with the FT230X.

BMME 580 Microcontrollers I



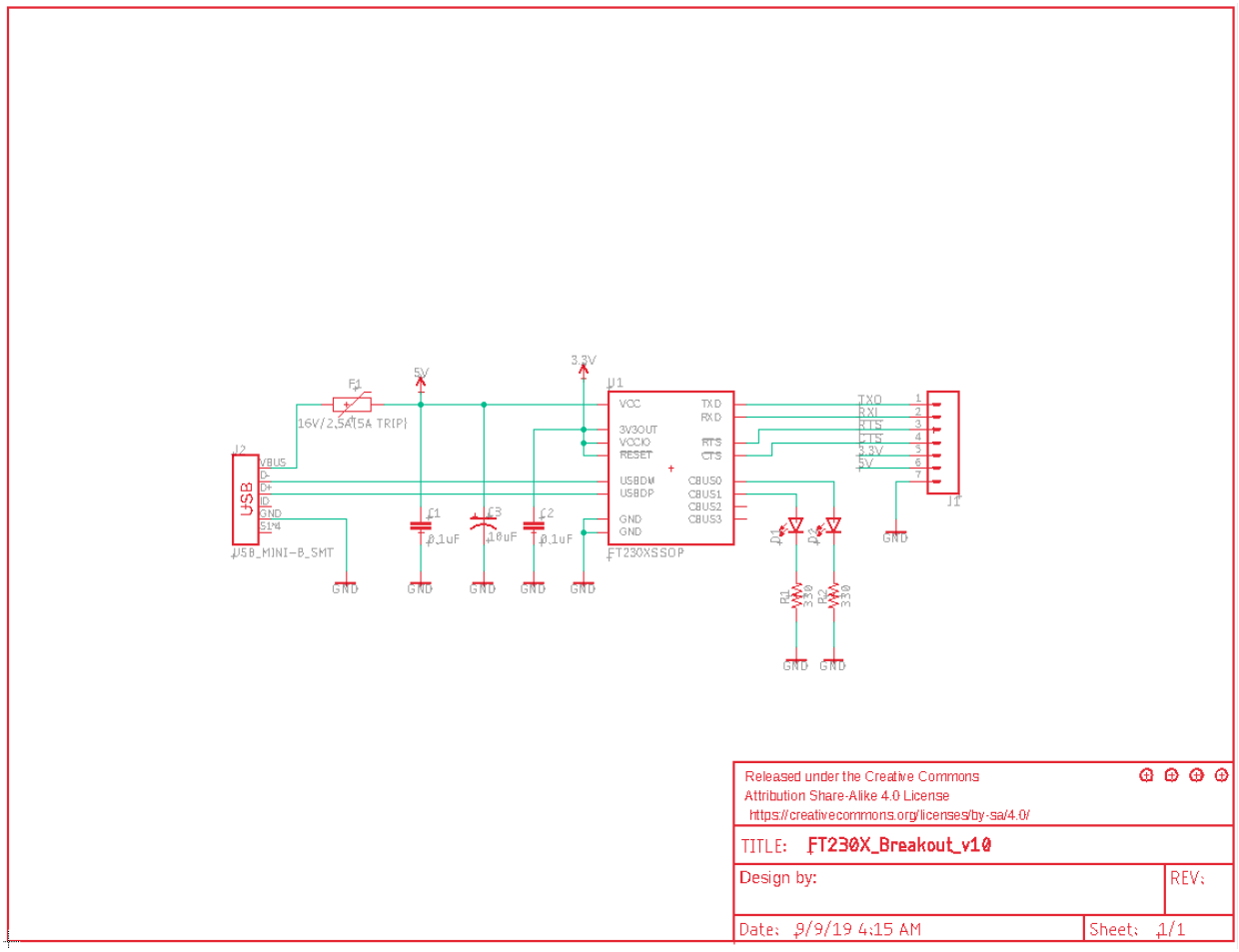
Description

Schematic component for the FT230X USB to Serial converter.

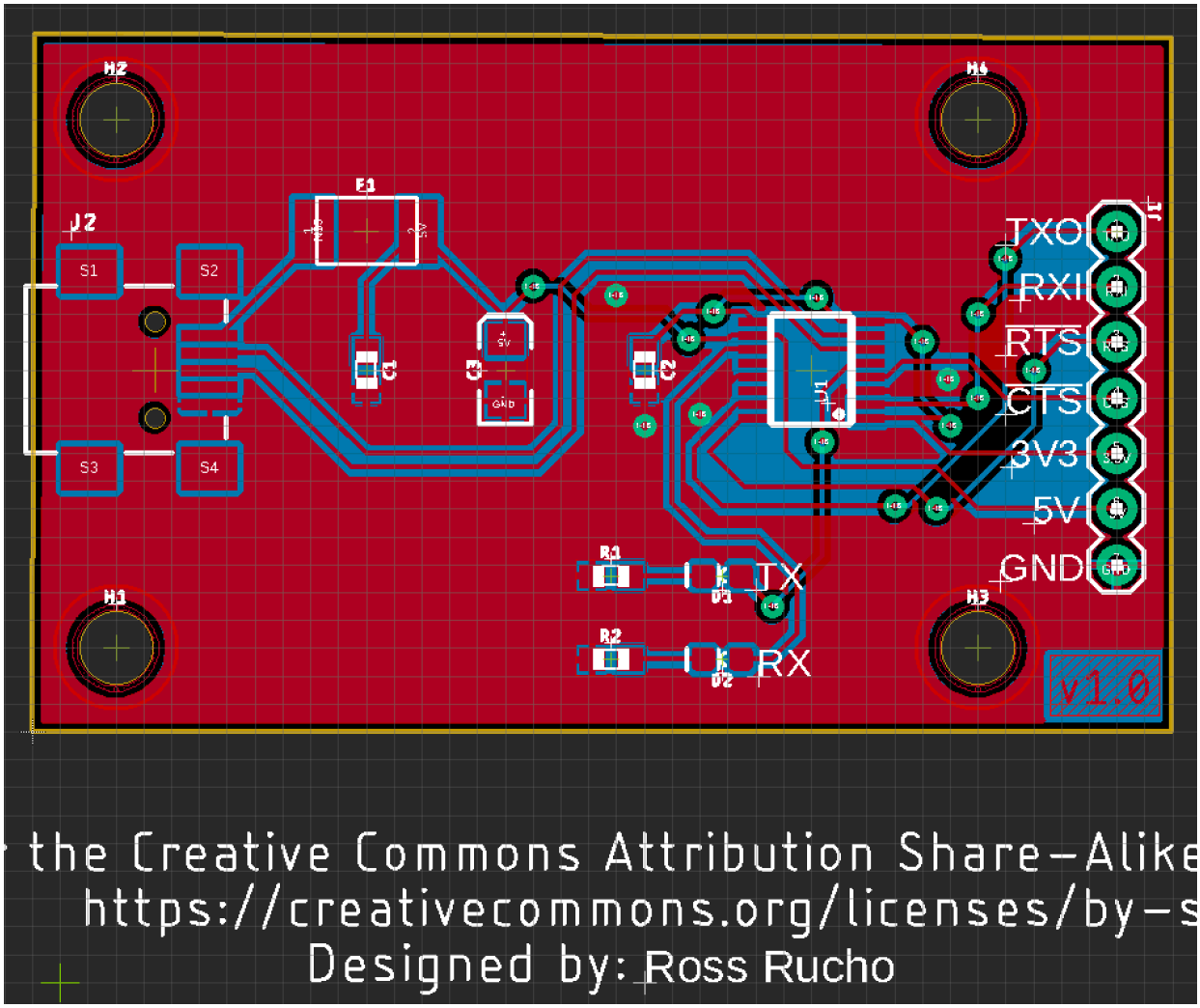


BMME 580 Microcontrollers I

EagleCAD Tutorial: Designing PCBs Advanced SMD – Your Screenshot



BMME 580 Microcontrollers I



BMME 580 Microcontrollers I

OPTIONAL: When finished with the tutorials, do not forget to follow the links on https://github.com/sparkfun/SparkFun-Eagle-Libraries to build up a very useful set of component library files for eagle.

MY ADVICE: whenever you need to use a new component, build a new component file for yourself on EAGLE. If you do this regularly, pretty soon you will end up with a large library of components that you use regularly.